

KISSSOFT
Calculation programs for machine design

KISSSOFT RELEASE 03/2017 USER MANUAL

Issue 1.5

Copyright Notice

© 2017 KISSsoft AG
Rosengartenstrasse 4
CH-8608 Bubikon Schweiz

All rights reserved
This documentation may not be copied without the express written approval of KISSsoft AG.

Table of Contents

I General I-41

1 Installing KISSsoft I-42

1.1 Basic installation I-43
1.2 Downloading a license file I-44
1.3 Licensing I-45
1.3.1 Test version I-45
1.3.2 Student version I-45
1.3.3 Single user version with dongle (protection key) I-45
1.3.4 Single user version with license code I-46
1.3.5 Network version with dongle (protection key) I-46
1.3.6 Network version with the license code I-47

2 Setting Up KISSsoft I-48

2.1 Directory structure I-49
2.2 Language settings I-50
2.3 System of units I-51
2.4 Defining your own default files I-52
2.5 Rights I-53
2.6 Global settings - KISS.ini I-54
2.6.1 Definitions in [PATH] I-54
2.6.2 Definitions in [SETUP] I-55
2.6.3 Definitions in [REPORT] I-56
2.6.4 Definitions in [GRAPHICS] I-57
2.6.5 Definitions in [LICENSE] I-57
2.6.6 Definitions in [CADEXPORT] I-58
2.6.7 Definitions in [INTERFACES] I-58
2.6.8 Definitions in [SOLIDEDGE] I-59
2.6.9 Definitions in [SOLIDWORKS] I-59
2.6.10 Definitions in [INVENTOR] I-59

2.6.11 Definitions in [CATIA] I-60
2.6.12 Definitions in [PROENGINEER] I-60
2.6.13 Definitions in [HICAD] I-61
2.7 User-defined settings I-62
2.7.1 Configuration tool I-62
2.8 Rules I-66

3 Starting KISSsoft I-69
3.1 Initial parameters I-70
3.2 Disconnect license from the network I-71

4 Elements of the KISSsoft User Interface I-72
4.1 Menus, context menus and the Tool Bar I-73
4.2 Docking window I-75
4.2.1 The module tree I-75
4.2.2 The project tree I-76
4.2.3 The Results window I-76
4.2.4 The Messages window I-76
4.2.5 The info window I-76
4.2.6 Manual and Search I-77
4.3 Graphics window I-78
4.3.1 Tool bar and context menu I-79
4.3.2 Comment field I-81
4.3.3 Context menu I-81
4.3.4 Properties I-81
4.3.5 Toothing I-83
4.4 Main input area I-85
4.4.1 Report Viewer I-85
4.4.2 Helptext Viewer I-86
4.5 Tooltips and status bar I-87

5 KISSsoft Calculation Modules I-88
5.1 Standard and special tabs I-89

5.2 Input elements I-90
5.2.1 Value input fields I-90
5.2.2 Formula entry and angle input I-90
5.2.3 Unit switch I-91
5.2.4 Tables I-91
5.3 Calculating and generating a report I-92
5.4 Messages I-93

6 Project Management I-94
6.1 Creating, opening and closing projects I-95
6.2 Adding and deleting files I-96
6.3 The active working project I-97
6.4 Storage locations I-98
6.5 Project properties I-99

7 Dynamic user interface I-100
7.1 Modified tabs and dialogs supplied with the system I-101
7.2 Adding additional tabs and dialogs I-102
7.3 Formatting I-104
7.3.1 Elements I-104
7.3.2 Columns I-105
7.3.3 Groups I-105
7.3.4 Tabs I-106
7.3.5 Comments I-106
7.3.6 Special elements I-106

8 Results and Reports I-110
8.1 Results of a calculation I-111
8.1.1 Add your own texts in the results window I-111
8.2 Calculation reports I-112
8.3 Drawing data I-113
8.4 Report settings I-114
8.4.1 General I-114

8.4.2 Page layout I-114
8.4.3 Header and footer I-114
8.4.4 Start and end block I-115
8.5 Report templates I-117
8.5.1 Storage locations and descriptions I-117
8.5.2 Scope of a report I-118
8.5.3 Formatting I-118

9 Database Tool and External Tables I-128

9.1 Viewing database entries I-130
9.2 Managing database entries I-133
9.2.1 Generating a database entry I-133
9.2.2 Deleting a database entry I-134
9.2.3 Restoring a database entry I-134
9.3 Import and export data from the database tool I-135
9.4 External tables I-136
9.4.1 Functions tables I-137
9.4.2 Range tables I-139
9.4.3 List tables I-140
9.4.4 List of key words used I-142
9.5 Description of database tables I-144
9.5.1 Center distance tolerances I-144
9.5.2 Machining allowance for cylindrical gear I-144
9.5.3 Reference profiles I-144
9.5.4 Compression springs standard I-144
9.5.5 Selection of hobbing cutters I-145
9.5.6 Basic material, glued and soldered joints I-145
9.5.7 Manufacturing process for bevel and hypoid gears I-145
9.5.8 V-belt Standard I-145
9.5.9 Spline Standard I-146
9.5.10 Chain profiles ISO606 ..... I-146
9.5.11 Adhesives I-146
9.5.12 Modifications I-146

9.5.13 Load spectra I-146
9.5.14 Solders I-148
9.5.15 Surface roughness of shafts and shaft-hub connections I-148
9.5.16 Key standard I-148
9.5.17 Polygon standard I-148
9.5.18 Woodruff Key standard I-148
9.5.19 Bolts/pins I-148
9.5.20 Lubricants I-148
9.5.21 Bolts: Tightening factor I-150
9.5.22 Bolts: Bore I-150
9.5.23 Bolts: Strength class I-151
9.5.24 Bolts: Nuts strength class I-151
9.5.25 Bolts: Coefficients of friction classes I-151
9.5.26 Bolts: Thread type I-152
9.5.27 Bolts: Nuts I-152
9.5.28 Bolts: Type I-152
9.5.29 Bolts: flat washer I-152
9.5.30 Selection of pinion type cutters I-152
9.5.31 Disc spring standard I-153
9.5.32 Tolerances standard I-153
9.5.33 Beam profiles I-153
9.5.34 Multi-Spline standard I-153
9.5.35 Materials I-153
9.5.36 Material of gears I-157
9.5.37 Rolling bearing I-160
9.5.38 Rolling bearing tolerance I-168
9.5.39 Rolling bearing Tolerance classes I-168
9.5.40 Tooth thickness tolerances I-168
9.5.41 Toothed belt standard I-169

10 Description of the Public Interface I-171

10.1 Interfaces between calculation programs and CAD - Overview I-172
10.1.1 Efficient interfaces I-172

10.1.2 Open interfaces concept in KISSsoft I-173
10.2 Defining input and output I-175
10.2.1 Preamble I-175
10.2.2 Requirements placed on the third party program I-176
10.2.3 Used files I-176
10.2.4 Service life of files I-177
10.2.5 Explicitly reading (importing) and generating data I-177
10.3 Example: Interference fit calculation I-178
10.4 Geometry data I-180
10.5 COM interface I-181
10.5.1 Registering the server I-181
10.5.2 Server functionality I-181
10.5.3 Example of a call from Excel I-184

11 3D interfaces I-190
11.1 Overview of the available CAD interfaces and their functionality I-191
11.2 Generation of 3D gears I-192
11.3 Generating 3D shafts I-194
11.4 Viewer with neutral format interface I-196
11.4.1 Parasolid Export of 3D Shafts I-196
11.4.2 Face gear - 3D geometry I-197
11.4.3 Bevel gear - generating a 3D model I-198
11.4.4 Worm wheel - generating a 3D model I-199
11.4.5 General information about 3D modeling in Parasolid I-200
11.5 3D interface to SolidWorks I-201
11.5.1 Gear teeth in the case of an existing basic solid I-201
11.5.2 Integrating the KISSsoft Add-in (menu options in CAD) I-203
11.5.3 Add-in functions (calls) I-206
11.6 3D interface to Solid Edge I-209
11.6.1 Change to the parameter for generation I-209
11.6.2 Gear teeth in the case of an existing basic solid I-210
11.6.3 Integrating the KISSsoft Add-in (menu options in CAD) I-211
11.6.4 Add-in functions (calls) I-215

11.6.5 Opening the calculation file for the created gear I-216
11.6.6 Simplified view of the gears I-216
11.7 3D interface to Autodesk Inventor I-217
11.7.1 Gear teeth when existing shaft data is present I-217
11.7.2 Add-in (menu options in CAD) I-218
11.7.3 Add-in functions (calls) I-221
11.7.4 Opening the calculation file for the created gear I-222
11.8 3D interface to Unigraphics NX I-223
11.8.1 Add-in (menu options in CAD) I-225
11.8.2 Running KISSsoft via the Add-in I-228
11.9 3D interface to Creo Parametric (ProEngineer) I-237
11.9.1 Integrating the KISSsoft Add-in I-240
11.9.2 Modifying the selected 3D model I-244
11.9.3 Cutting teeth on an existing shaft I-245
11.9.4 Modifying the teeth on an existing shaft I-247
11.9.5 Changing base settings in the interface I-248
11.9.6 Save the files to the PTC Windchill working directory I-249
11.10 3D interface to CATIA I-250
11.10.1 Registering the interface I-250

12 Answers to Frequently Asked Questions I-253
12.1 Changing the output of angles in reports I-254
12.2 Input materials for gear calculations in the database I-255
12.3 How can I test the software? I-256
12.4 What licenses are available? I-257
12.5 Add your own texts in the results window I-258
12.6 Restoring the previous stages of the calculation I-259

II Toothing II-260

13 Introduction II-261

14 Cylindrical Gears II-262

14.1 Basic data II-264
14.1.1 Hand of gear for gear teeth II-264
14.1.2 Normal module II-264
14.1.3 Pressure angle at normal section II-265
14.1.4 Helix angle at reference circle II-265
14.1.5 Center distance II-265
14.1.6 Number of teeth II-266
14.1.7 Facewidth II-266
14.1.8 Profile shift coefficient II-267
14.1.9 Quality II-270
14.1.10 Geometry details II-273
14.1.11 Materials and lubrication II-274
14.2 Rating II-281
14.2.1 Calculation methods II-281
14.2.2 Service life II-289
14.2.3 Reliability II-289
14.2.4 Application factor II-291
14.2.5 Power, torque and speed II-292
14.2.6 Strength details II-292
14.2.7 Strength details (AGMA) II-307
14.2.8 Define load spectrum II-308
14.2.9 Calculate scuffing II-311
14.2.10 Calculating the internal temperature and the flash temperature II-312
14.3 Factors II-313
14.3.1 Transverse coefficient II-313
14.3.2 Dynamic factor II-313
14.3.3 Load distribution coefficient II-314
14.3.4 Alternating bending factor II-315
14.3.5 Face load factor II-318
14.3.6 Taking into account shaft bending (face load factor and contact analysis) II-340
14.3.7 Z-Y coefficients and the technology factor II-342

14.3.8 General calculation procedure for KHbeta as specified in ISO 6336-1, Annex E. II-345
14.4 Reference profile II-346
14.4.1 Configuration II-346
14.4.2 Pre-machining and grinding allowance II-353
14.4.3 Tip alteration II-354
14.5 Final machining II-356
14.6 Tolerances II-357
14.6.1 Tooth thickness tolerance II-357
14.6.2 Tip diameter allowances II-359
14.6.3 Root diameter allowances II-359
14.6.4 Center distance tolerances II-360
14.6.5 Settings II-360
14.7 Modifications II-361
14.7.1 Type of modification II-361
14.7.2 Underlying principles of calculation II-362
14.7.3 Profile modifications II-364
14.7.4 Tooth trace modifications II-372
14.7.5 Sizing modifications II-378
14.7.6 Notes about profile modification II-382
14.7.7 Using diamond dressing wheels and grinding worms II-382
14.8 Tooth form II-387
14.8.1 Context menu II-388
14.8.2 Operations II-389
14.9 Tooth flank fracture II-409
14.10 Contact analysis II-411
14.10.1 Theory of Contact Analysis II-413
14.10.2 Discretized model II-416
14.10.3 Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis II-416
14.10.4 Reduced stiffness on the side edges II-418
14.10.5 Coupling the individual slices II-419
14.10.6 Contact analysis model for planetary systems II-419

14.10.7 Base pressure angle of contact analysis II-420
14.10.8 Wear iteration II-421
14.10.9 Contact analysis with load spectra II-422
14.11 Gear pump II-423
14.12 Operating backlash II-426
14.12.1 Temperatures II-428
14.12.2 Relative water absorption during swelling II-428
14.12.3 Coefficient of thermal expansion for housing II-428
14.12.4 Taking into account the bending of the shafts and width modifications II-429
14.12.5 Tooth deformation II-429
14.13 Master gear II-430
14.14 AGMA 925 ..... II-432
14.15 Root stress FEM II-434
14.16 Rough sizing II-435
14.17 Fine sizing II-439
14.17.1 Necessary entries in the input window II-440
14.17.2 Conditions I II-440
14.17.3 Conditions II II-442
14.17.4 Conditions II II-442
14.17.5 Results II-446
14.17.6 Graphics II-448
14.17.7 Geometry fine sizing for 3 gears II-449
14.17.8 Additional strength calculation of all variants II-449
14.18 Modifications sizing II-450
14.18.1 Conditions I/II II-450
14.18.2 Results II-451
14.18.3 Graphics II-452
14.18.4 Report II-452
14.19 Measurement grid II-453
14.20 Settings II-456
14.20.1 General II-456
14.20.2 Plastic II-459

14.20.3 Planets II-463
14.20.4 Sizings II-464
14.20.5 Calculations II-466
14.20.6 Required safeties II-471
14.20.7 Contact analysis/face load factor II-472
14.20.8 Summary II-475
14.20.9 Diagrams II-475
14.20.10 Generation of 3D model II-476
14.21 Tooth thickness II-479

15 Bevel and Hypoid gears II-480

15.1 Underlying principles of calculation II-481
15.1.1 General II-481
15.1.2 Overview of the bevel gear manufacturing process and the terminology used in it II-481
15.1.3 Calculation according to Klingelnberg, Gleason and Oerlikon II-482
15.2 Basic data II-484
15.2.1 Mean normal module II-484
15.2.2 Reference diameter gear 2 ..... II-484
15.2.3 Pressure angle at normal section II-485
15.2.4 Pressure angle driving/driven flank: Hypoid gears II-485
15.2.5 Spiral and helix angle II-486
15.2.6 Shaft angle II-487
15.2.7 Hypoid offset II-488
15.2.8 Number of teeth II-488
15.2.9 Facewidth II-489
15.2.10 Profile shift coefficient II-489
15.2.11 Tooth thickness modification factor II-489
15.2.12 Quality II-490
15.2.13 Addendum angle and root angle II-491
15.2.14 Angle modifications II-492
15.2.15 Geometry details II-493
15.2.16 Manufacturing process II-493

15.3 Type II-495
15.3.1 Converting or inputting Gleason toothing data II-497
15.4 Manufacture II-500
15.4.1 Cutter radius II-500
15.4.2 Number of blade groups II-500
15.5 Rating II-501
15.5.1 Methods used for strength calculation II-501
15.5.2 Required service life II-504
15.5.3 Power, torque and speed II-505
15.5.4 Strength details II-505
15.5.5 Application factor II-507
15.6 Reference profile II-509
15.6.1 Default values for tip clearance II-509
15.6.2 Default values for addendum coefficients II-509
15.7 Contact analysis II-510
15.8 Modifications II-511
15.9 Rough sizing II-515
15.9.1 Facewidth ratio II-515
15.9.2 Module ratio II-516
15.10 Fine sizing II-517
15.10.1 Required entries in the standard tabs II-517
15.10.2 Conditions I II-518
15.10.3 Conditions II II-519
15.10.4 Conditions III II-520
15.10.5 Results II-525
15.10.6 Graphics II-526
15.11 Notes on calculations in accordance with the Klingelnberg standard II-527
15.11.1 Bevel gears with cyclo-palloid® gear teeth II-527
15.11.2 Hypoid gears with cyclo-palloid gear teeth II-527
15.11.3 Normal module ranges for Klingelnberg machines (cyclo-palloid) II-528
15.11.4 Bevel gears with palloid toothing II-529

15.11.5 Definitions and dimensions of standard cutters for palloid toothing II-530
15.11.6 Minimum safeties II-531
15.11.7 Surface roughness at tooth root II-531
15.11.8 Accuracy grade bevel gears II-531
15.11.9 Characteristic number II-531
15.12 Settings II-533
15.12.1 General Information II-533
15.12.2 Calculations II-533
15.12.3 Differential gears II-533
15.12.4 Helpful information about the Generation of 3D model tab II-534
15.12.5 Factors II-534
15.12.6 Contact analysis II-537
15.12.7 Display II-537

16 Face gears II-539

16.1 Underlying principles of calculation II-540
16.2 Basic data II-543
16.2.1 Normal module II-543
16.2.2 Pressure angle at normal section II-545
16.2.3 Helix angle at reference circle II-545
16.2.4 Axial offset II-546
16.2.5 Profile shift coefficient (on the pinion) II-546
16.2.6 Quality II-547
16.2.7 Geometry details II-548
16.2.8 Materials and lubrication II-549
16.3 Rating II-550
16.3.1 Methods used for strength calculation II-550
16.3.2 Service life II-552
16.3.3 Power, torque and speed II-552
16.3.4 Application factor II-552
16.4 Factors II-554
16.4.1 Face load factor II-554

16.5 Modifications II-555
16.5.1 Addendum reduction II-555
16.5.2 Type of modification II-555
16.6 Settings II-556
16.6.1 General II-556
16.6.2 Sizings II-557
16.7 Notes on face gear calculation II-558
16.7.1 Dimensioning II-558
16.7.2 Pinion - Face gear with Z1 > Z2 ..... II-559

17 Worms with enveloping worm wheels II-560
17.1 Underlying principles of calculation II-561
17.2 Basic data II-563
17.2.1 Axial/transverse module II-563
17.2.2 Pressure angle at normal section II-563
17.2.3 Lead angle at reference circle II-563
17.2.4 Center distance II-564
17.2.5 Number of teeth II-564
17.2.6 Facewidth II-565
17.2.7 Profile shift coefficient II-565
17.2.8 Tooth thickness modification factor II-566
17.2.9 Quality II-566
17.2.10 Geometry details II-567
17.2.11 Materials and lubrication II-568
17.3 Rating II-570
17.3.1 Methods used for strength calculation II-570
17.3.2 Service life II-571
17.3.3 Application factor II-571
17.3.4 Permissible decrease in quality II-571
17.3.5 Power, torque and speed II-572
17.3.6 Strength details II-572
17.4 Tolerances II-576
17.5 Settings II-577

17.5.1 General II-577
17.5.2 Reference gearing II-578
17.5.3 Calculations II-579
17.5.4 Required safeties II-580

18 Crossed Helical Gears and Precision Mechanics Worms II-582
18.1 Underlying principles of calculation II-583
18.2 Basic data II-584
18.2.1 Normal module II-584
18.2.2 Pressure angle at normal section II-584
18.2.3 Helix angle reference circle gear 1 ..... II-584
18.2.4 Center distance II-585
18.2.5 Facewidth II-585
18.2.6 Profile shift coefficient II-585
18.2.7 Quality II-585
18.2.8 Define details of geometry II-586
18.2.9 Material and lubrication II-587
18.3 Rating II-588
18.3.1 Methods used for strength calculation II-588
18.3.2 Service life II-593
18.3.3 Application factor II-594
18.3.4 Power, torque and speed II-594
18.3.5 Strength details II-595
18.4 Settings II-602
18.5 Notes II-603
18.5.1 Checking the contact pattern II-603

19 Beveloid Gears II-604
19.1 Underlying principles of calculation II-605
19.2 Basic data II-606
19.2.1 Normal module II-606
19.2.2 Pressure angle at normal section II-606
19.2.3 Helix angle II-606

19.2.4 shaft angle II-606
19.2.5 Number of teeth II-607
19.2.6 Width II-607
19.2.7 Cone angle II-607
19.2.8 Profile shift coefficient (center) II-607
19.2.9 Quality II-607
19.2.10 Material and lubrication II-607
19.3 Reference profile II-608
19.4 Modifications II-609
19.5 Factors II-610
19.6 Dimensioning II-611
19.7 Manufacturing Data and Working Data II-612

20 Non-Circular Gears II-613
20.1 Input data II-614
20.1.1 Geometry II-614
20.1.2 Tolerances II-617
20.1.3 Reference profile II-618
20.2 How to use KISSsoft II-619
20.2.1 Angle error II-619
20.2.2 Checking the meshing II-619
20.2.3 Improve tooth form II-620
20.2.4 Accuracy of the tooth form II-620
20.2.5 Export individual teeth II-621
20.2.6 Report II-622
20.2.7 Temporary files II-622

21 Report Menu II-624
21.1 Drawing data II-625
21.2 Manufacturing tolerances II-626
21.3 Summary II-627
21.4 Service life II-628
21.5 Sizing of torque II-629

21.6 Proposal for the hardening depth EHT II-630

22 Graphics Menu II-631

22.1 AGMA 925 ..... II-635
22.1.1 Lubricant film thickness and specific oil film thickness II-635
22.2 2D geometry II-636
22.2.1 Gear tooth forms II-636
22.2.2 Gear tool II-637
22.2.3 Manufacturing a gear II-637
22.2.4 Meshing II-637
22.2.5 Profile and tooth trace diagram II-638
22.2.6 Flank curvature radii II-642
22.2.7 Angle of flank normal II-643
22.2.8 Drawing II-643
22.2.9 Assembly II-643
22.2.10 Manufacturing drawing II-643
22.3 3D geometry II-646
22.3.1 Tooth system II-647
22.3.2 Tooth form II-647
22.4 Evaluation II-648
22.4.1 Specific sliding II-648
22.4.2 Contact temperature II-648
22.4.3 Flash temperature II-648
22.4.4 Hardening depth II-648
22.4.5 Theoretical contact stiffness II-649
22.4.6 S-N curve (Woehler lines) for material II-650
22.4.7 Safety factor curves II-650
22.4.8 Oil viscosity, depending on temperature II-650
22.4.9 Gaping II-650
22.4.10 face load distribution II-650
22.4.11 Tooth flank fracture II-651
22.4.12 Sliding velocity (face gear) II-651
22.4.13 Contact line (face gear) II-651

22.4.14 Stress curve (face gear) II-651
22.4.15 Scuffing and sliding velocity (face gear) II-651
22.5 Contact analysis II-653
22.5.1 Axis alignment II-653
22.5.2 Transmission error II-653
22.5.3 Transmission error acceleration II-654
22.5.4 Amplitude of transmission error II-655
22.5.5 Contact lines on tooth flank II-655
22.5.6 Normal force curve II-655
22.5.7 Normal force distribution II-656
22.5.8 Torque curve II-656
22.5.9 Single contact stiffness II-656
22.5.10 Stiffness curve II-656
22.5.11 Amplitude of contact stiffness II-657
22.5.12 Bearing force curve and direction of the bearing forces II-657
22.5.13 Kinematics II-658
22.5.14 Specific sliding II-658
22.5.15 Power loss II-658
22.5.16 Heat development II-658
22.5.17 Stress curve II-658
22.5.18 Flash temperature II-659
22.5.19 Micropitting (frosting) II-659
22.5.20 Wear II-661
22.6 Gear pump II-664
22.7 3D export II-665
22.8 Settings II-666
22.9 Graphics list II-667

23 Answers to Frequently Asked Questions II-668

23.1 Answers concerning geometry calculation II-669
23.1.1 Precision mechanics II-669
23.1.2 Deep toothing or cylindrical gears with a high transverse contact ratio II-669

23.1.3 Pairing an external gear to an inside gear that has a slightly different number of teeth II-670
23.1.4 Undercut or insufficient effective involute II-670
23.1.5 Tooth thickness at tip II-671
23.1.6 Special toothing II-671
23.1.7 Calculating cylindrical gears manufactured using tools specified in DIN 3972 ..... II-671
23.1.8 Composites as defined in DIN 58405 ..... II-672
23.1.9 Automatic change of reference profiles II-673
23.1.10 Non-identical (mirrored symmetry) tooth flanks II-673
23.1.11 Internal toothing - differences in the reference profile if you select different configurations II-674
23.1.12 Effect of profile modifications II-675
23.1.13 Number of teeth with common multiples II-676
23.1.14 Allowances for racks II-677
23.2 Answers to questions about strength calculation II-678
23.2.1 Differences between different gear calculation programs II-678
23.2.2 Difference between cylindrical gear calculation following ISO 6336 or DIN 3990 ..... II-678
23.2.3 Calculation using Methods B or C (DIN 3990, 3991) II-679
23.2.4 Required safeties for cylindrical gear units II-679
23.2.5 Insufficient scuffing safety II-680
23.2.6 Work hardening factor (strengthening an unhardened gear) II-681
23.2.7 Defining the load stage scuffing (oil specification) II-681
23.2.8 The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error. II-681
23.2.9 Load spectrum with changing torque II-682
23.2.10 Strength calculation with several geometries on one gear II-684
23.2.11 Bevel gears: – Determine permitted overloads II-685
23.2.12 Take shot peening data into account in calculating the strength of toothed gears II-686
23.2.13 Calculation according to AGMA 421.06 (High Speed Gears) .. II-687

23.2.14 Comparison of an FEM calculation with crossed axis helical gear wheel calculation II-688
23.2.15 Estimating the strength of asymmetrical spur gear toothings.... II-688
23.2.16 Determine the equivalent torque (for load spectra)..... II-689
23.2.17 Check changes in safeties if the center distance changes..... II-689
23.2.18 Warning: "Notch parameter qs …. outside RANGE (1.0 to 8.0) ..." II-690
23.3 Abbreviations used in gear calculation..... II-691

III Shafts and Bearings III-698

24 Defining Shafts III-699
24.1 Input window III-702
24.1.1 Shaft editor III-702
24.1.2 Elements tree III-703
24.1.3 Elements list III-704
24.1.4 Elements editor.....III-705
24.2 Element overview III-706
24.2.1 The Shaft element.....III-706
24.2.2 Outer contour.....III-711
24.2.3 Inner contour III-719
24.2.4 Forces III-719
24.2.5 Bearing III-725
24.2.6 Connection elements III-728
24.2.7 Cross sections.....III-730
24.3 Basic data.....III-731
24.3.1 Position of shaft axis in space III-731
24.3.2 Number of eigenfrequencies III-732
24.3.3 Number of buckling cases III-732
24.3.4 Speed III-732
24.3.5 Direction of rotation III-733
24.3.6 Reference temperature.....III-733
24.3.7 Temperature of housing.....III-734

24.3.8 Lubricant temperature III-735
24.3.9 Load spectra III-735
24.3.10 Gears III-736
24.3.11 Rolling bearings III-736
24.3.12 Tolerance field III-737
24.3.13 Modified service life according ISO 281 ..... III-740
24.3.14 Consider weight III-740
24.3.15 Consider gyroscopic effect III-740
24.3.16 Housing material III-740
24.3.17 Lubrication III-740
24.3.18 Impurity III-741
24.4 Module specific settings III-742
24.4.1 Non-linear shaft III-742
24.4.2 Take into account deformation due to shearing and shear correction coefficient III-742
24.4.3 Activating offset of load application center point III-743
24.4.4 Using the 2013 solver III-743
24.4.5 Saving temporary results in CSV format with .tmp file extension. III-743
24.4.6 Standard radius at shoulders III-744
24.4.7 Node density III-744
24.4.8 Iterative calculation of load distribution III-745
24.4.9 Input different load cycles for bending and torsion (for finite life calculations) III-745
24.4.10 Save user-defined rolling bearing in calculation file III-745
24.4.11 Entering an impurity separately in each bearing III-746
24.4.12 Axial clearance III-746
24.4.13 Failure probability III-746
24.4.14 Required service life III-746
24.4.15 Maximum service life coefficient III-747
24.4.16 Displaying critical bearings III-747
24.4.17 Surface roughness of housing III-747
24.4.18 Calculation method for friction III-748

24.4.19 Oil level III-748
24.4.20 Type of oil lubrication III-748
24.4.21 Moment of friction, seals III-748
24.4.22 Bearing manufacturer III-749
24.4.23 Show coordinate system III-749
24.4.24 Show automatic dimensioning III-749
24.4.25 Equivalent stress for sizings III-749
24.4.26 Maximum deflection for sizings III-749

25 Calculating Shafts III-750

25.1 Deflection and Bearing Forces, Distribution and Force of Torque III-751
25.1.1 Calculating force on bearings with a contact angle III-754
25.2 Eigenfrequencies III-756
25.2.1 Bending critical speed III-756
25.2.2 Torsion critical speed III-756
25.3 Buckling III-757
25.4 Rough sizing of shafts III-758
25.5 Strength III-759
25.5.1 Calculation method III-760
25.5.2 Calculation type III-765
25.5.3 Service life III-767
25.5.4 Strength parameters in accordance with Hänchen and Decker III-767
25.5.5 Strength parameters in accordance with FKM III-768
25.5.6 Strength parameters according to DIN III-771
25.5.7 Strength parameters according to AGMA III-772
25.5.8 Stress III-774
25.5.9 Stress ratio III-774
25.5.10 Load factor for static analysis III-775
25.5.11 Load factor endurance calculation III-775
25.5.12 Cross sections III-776
25.5.13 Sizing III-778
25.5.14 Cross section types III-778
25.5.15 General entries III-784

25.5.16 Thermally safe operating speed III-784
25.6 Tooth trace modification III-786
25.7 Campbell diagram III-789
25.8 Forced response III-791
25.8.1 Calculation procedure III-791
25.8.2 Results III-791

26 General Bearing Calculation III-792
26.1 Classification of bearings III-793
26.1.1 Properties III-793

27 Rolling bearings III-795
27.1 Selecting the type of rolling bearing III-796
27.1.1 Properties of the most important bearing types III-796
27.1.2 Comparing types III-798
27.1.3 Hybrid bearings III-801
27.2 Load capacity of rolling bearings III-802
27.2.1 Dynamic load capacity III-802
27.2.2 Permissible static stress III-802
27.2.3 Rolling bearing calculation with inner geometry III-803
27.3 Thermally safe operating speed III-804
27.3.1 Thermal reference speed III-804
27.3.2 Process for calculating thermally permitted operating speed (DIN 732-2) III-806
27.4 Moment of friction III-809
27.4.1 Calculation according to SKF Catalog 1994 ..... III-809
27.4.2 Calculation according to SKF Catalog 2013 ..... III-811
27.4.3 Calculation according to Schaeffler 2014 (INA, FAG) III-813
27.5 Maximum Speeds III-815
27.6 Service life III-816
27.6.1 Modified service life calculation in accordance with the Supplement to DIN ISO 281 (2007) III-816
27.6.2 Service life calculation with load spectra III-817

27.7 Failure probability III-819
27.8 Bearings with radial and/or axial force III-819
27.9 Calculating axial forces on bearings in face-to-face or back-to-back arrangements III-820
27.10 Oil level and lubrication type III-822

28 Rolling Bearings (Internal Geometry) III-823
28.1 Bearing data tab III-824
28.1.1 File interface III-824
28.1.2 Bearing data III-826
28.2 Load tab III-830
28.2.1 Rating III-830
28.2.2 Modified service life calculation according to ISO 281 ..... III-830
28.3 Tab Elastic Rings III-832
28.3.1 Basic data III-832
28.3.2 Details III-834
28.4 Graphics III-836
28.5 Fine sizing III-837

29 Hydrodynamic Plain Radial Bearings III-839
29.1 Calculation methods III-840
29.2 Module-specific entries III-841
29.3 Thermal expansion coefficients III-842
29.4 Average surface pressure III-843
29.5 Geometries according to DIN 31657 ..... III-844
29.6 Lubrication arrangement III-846
29.7 Heat transfer coefficient III-848
29.8 Heat transfer surface III-849
29.9 Oil temperatures III-849
29.10 Mixture factor III-850
29.11 Sizing the bearing clearance III-850
29.12 Sommerfeld number III-851
29.13 Bearing width III-851

29.14 permissible lubricant film thickness III-852

30 Hydrodynamic Plain Thrust Bearings III-853

30.1 Calculation III-856
30.2 Sizings III-857
30.3 Calculation of volume-specific heat III-858
30.4 Limiting values in the calculation III-859

31 Answers to Frequently Asked Questions III-860

31.1 Intersecting notch effects III-861
31.2 Notch effects on hollow shafts III-862
31.2.1 Notches on the outer contour III-862
31.2.2 Notches on the inner contour III-862
31.3 Fatigue Limits for New Materials III-863
31.4 Taking double helical gearing into account in the shaft calculation III-864

IV Connections IV-865

32 Cylindrical Interference Fits IV-866

32.1 Inputting Tolerances IV-869
32.2 Coefficient of friction IV-870
32.3 Variable external diameter of the hub IV-872
32.4 Convert external pressure with multiple interference fit IV-873
32.5 Materials IV-874
32.6 Settings IV-875
32.7 Sizings IV-877

33 Conical Interference Fits IV-878

33.1 Calculation IV-880
33.2 Application factor IV-882
33.3 Axial spanning with nut IV-883
33.4 Variable external diameter of the hub IV-885

33.5 Conicity IV-886
33.6 Materials IV-887
33.7 Settings IV-888
33.8 Sizings IV-889

34 Clamped connections IV-890
34.1 Calculations IV-891
34.2 Sizings IV-892
34.3 Settings IV-892
34.4 Materials IV-893

35 Key IV-894
35.1 Main screen IV-895
35.1.1 Additional inputs for DIN 6892 Method B IV-896
35.2 Application factor IV-899
35.3 Load factor IV-901
35.4 Own inputs IV-902
35.5 Permissible pressure IV-903
35.6 Materials IV-904
35.7 Settings IV-905
35.8 Sizings IV-906

36 Straight-sided spline IV-907
36.1 Standard profiles IV-908
36.2 Application factor IV-909
36.3 Torque curve/Frequency of change of load direction IV-910
36.4 Occurring contact stress IV-911
36.5 Length factor IV-912
36.6 Share factor IV-913
36.7 Permissible pressure IV-914
36.8 Materials IV-915
36.9 Settings IV-916
36.10 Sizings IV-917

37 Splines (Strength) IV-918

37.1 Standard profiles IV-919
37.2 Application factor IV-920
37.3 Torque curve/Frequency of change of load direction IV-921
37.4 Occurring contact stress IV-922
37.5 Length factor IV-923
37.6 Share factor IV-924
37.7 Permissible pressure IV-925
37.8 Materials IV-926
37.9 Settings IV-927
37.10 Sizings IV-928

38 Splines (Geometry and Strength) IV-929

38.1 Underlying principles of calculation IV-930
38.1.1 General IV-930
38.1.2 Calculation of spline connections as described in DIN 5480 with diameter centering IV-930
38.1.3 Calculating spline connections according to DIN 5480 with diameter centering IV-931
38.2 Basic data IV-933
38.2.1 Geometry standards IV-933
38.2.2 Normal module IV-934
38.2.3 Pressure angle at normal section an IV-934
38.2.4 Number of teeth IV-934
38.2.5 Profile shift coefficient IV-934
38.2.6 Quality IV-935
38.2.7 Niemann geometry data IV-936
38.2.8 Geometry details IV-936
38.2.9 Define details of strength IV-936
38.2.10 Materials IV-943
38.3 Tolerances IV-944
38.3.1 Tooth thickness tolerance IV-944
38.3.2 Effective/Actual IV-945

38.3.3 Shaft/hub: diameter of ball/pin....IV-945
38.4 Templates....IV-946
38.5 Tooth form....IV-948

39 Polygons....IV-949
39.1 Standard profiles....IV-950
39.2 Application factor....IV-951
39.3 Torque curve/Frequency of change of load direction....IV-952
39.4 Occurring contact stress....IV-953
39.5 Permissible pressure....IV-955
39.6 Materials....IV-957
39.7 Settings....IV-958
39.8 Sizings....IV-959
39.9 Graphics....IV-960

40 Woodruff Keys....IV-961
40.1 Standard profiles....IV-962
40.2 Application factor....IV-964
40.3 Torque curve/Frequency of change of load direction....IV-965
40.4 Occurring contact stress....IV-966
40.5 Length factor....IV-967
40.6 Share factor....IV-968
40.7 Permissible pressure....IV-969
40.8 Materials....IV-970
40.9 Settings....IV-971
40.10 Sizings....IV-972

41 Bolts and Pins....IV-973
41.1 Influence factors....IV-975
41.2 Materials....IV-976
41.3 Settings....IV-977
41.4 Permitted values....IV-978
41.5 Sizings....IV-979

42 Bolts IV-980
42.1 Special features in KISSsoft IV-982
42.2 Inputs for Basic data IV-983
42.2.1 Operating data IV-983
42.2.2 Bolt data IV-995
42.2.3 Type of bolted joint IV-998
42.2.4 Washer IV-1000
42.2.5 Extension sleeves without external forces IV-1000
42.2.6 Tightening technique IV-1001
42.3 Data input for clamped parts IV-1003
42.3.1 Geometry of clamped parts (connecting solids) IV-1003
42.3.2 Distances for eccentric clamping/load IV-1006
42.3.3 Load application IV-1006
42.4 Constraints data IV-1007
42.4.1 Technical explanations IV-1009
42.4.2 Coefficient of friction IV-1010
42.4.3 Angle of rotation-controlled tightening IV-1011
42.5 Stripping strength IV-1012
42.6 Settings IV-1014

43 Welded Joints IV-1018
43.1 Welded joints IV-1019
43.2 Seam length IV-1021
43.3 Welded seam equivalent stress IV-1022
43.4 Weld seam boundary stress IV-1023
43.5 Part safety coefficient IV-1024
43.6 Boundary safety coefficient IV-1025
43.7 Materials IV-1026

44 Glued and Soldered Joints IV-1027
44.1 Basic materials IV-1029
44.2 Settings IV-1030
44.3 Sizings IV-1031

44.4 Bracket connection IV-1032
44.5 Shaft connections IV-1033

45 Retaining Rings (Self-Locking Rings, Seeger Rings) IV-1034
45.1 Basic data IV-1035
45.2 Automatic calculation of load factor q IV-1037
45.3 Automatic calculation of the dishing angle ψ IV-1038
45.4 Module specific settings IV-1039

46 Answers to Frequently Asked Questions IV-1040
46.1 Adding new bolt types to the database IV-1041
46.1.1 Extending an existing bolt series IV-1041
46.1.2 Create a new bolt type IV-1043

V Springs V-1044

47 Compression Springs V-1045
47.1 Strength values V-1046
47.2 Shear stress values V-1046
47.3 Bearings coefficient V-1047
47.4 Materials V-1047
47.5 Tolerances V-1048
47.6 Relaxation V-1049
47.7 Drawing data V-1050
47.8 Sizings V-1050

48 Tension Springs V-1051
48.1 Strength values V-1052
48.2 Shear stress values V-1052
48.3 Manufacturing type V-1053
48.4 Eyes screen V-1053
48.5 Materials V-1055

48.6 Settings V-1056
48.7 Tolerances V-1056
48.8 Relaxation V-1057
48.9 Drawing data V-1058
48.10 Sizings V-1059

49 Leg Springs V-1060
49.1 Strength values V-1061
49.2 Bending stress values V-1062
49.3 Spring design V-1062
49.4 Assumptions made for the calculation V-1063
49.5 Materials V-1063
49.6 Tolerances V-1064
49.7 Drawing data V-1064
49.8 Sizings V-1065

50 Disc Springs V-1066
50.1 Strength values V-1067
50.2 Stress values V-1067
50.3 Materials V-1068
50.4 Calculate number V-1069
50.5 Limit dimensions V-1070

51 Torsion-Bar Springs V-1071
51.1 Head forms V-1072
51.2 Strength values V-1073
51.3 Shear stress V-1073
51.4 Limiting values V-1073
51.5 Sizings V-1074

VI Belts and chain drives VI-1075

52 V-Belts VI-1076
52.1 V-belts data VI-1077
52.2 V-belt standards VI-1077
52.3 Configuring tensioning pulleys VI-1078
52.4 Application factor f1 ..... VI-1078
52.5 Center distance VI-1078
52.6 Belt length VI-1079
52.7 Effective number of V-belts VI-1079
52.8 Tensioning pulley diameter VI-1079
52.9 Position of tensioning pulley (x/y) VI-1080
52.10 Inspecting V-belts VI-1081

53 Toothed Belts VI-1082
53.1 Technical notes (toothed belts) VI-1083
53.2 Toothed belt standard VI-1084
53.3 Possible sizings/suggestions VI-1085
53.4 Configuring tensioning pulleys VI-1085
53.5 Application factor and summand for operational behavior VI-1086
53.6 Center distance VI-1086
53.7 Belt length and number of teeth on belt VI-1087
53.8 Effective belt width VI-1087
53.9 Number of teeth in tensioning pulley VI-1087
53.10 Position of the tensioning pulley x/y VI-1089

54 Chain Drives VI-1090
54.1 Sizings VI-1090
54.2 Tensioning pulleys VI-1091
54.3 Standard VI-1091
54.4 Chain type VI-1091
54.5 Number of strands VI-1091
54.6 Application factor VI-1092
54.7 Speed/number of teeth/ratio VI-1092
54.8 Configuration VI-1092

54.9 Center distance VI-1093
54.10 Polygon effect VI-1093
54.11 Number of links VI-1093
54.12 Sprocket geometry VI-1095

VII Automotive **VII-1096**

55 Synchronization VII-1096
55.1 Geometry VII-1097
55.2 Operating data VII-1098

56 Friction clutches VII-1098
56.1 Calculation VII-1101
56.2 Definition of spring forces VII-1104
56.3 Definition coefficients of sliding friction and velocities VII-1105
56.4 Graphics VII-1106
56.5 Settings VII-1107

VIII Various **VIII-1108**

57 Tolerance Calculation VIII-1109

58 Strength verification with local stresses VIII-1110
58.1 General VIII-1111
58.1.1 Functionality of the software VIII-1111
58.1.2 Areas of application for the FKM guideline VIII-1111
58.2 Background VIII-1113
58.2.1 The FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile VIII-1113
58.2.2 Usefulness of the service life calculation VIII-1113
58.3 Implementation in KISSsoft VIII-1117
58.3.1 Main screen VIII-1117

58.3.2 Load cases VIII-1119
58.3.3 S-N curve (Woehler lines) VIII-1119
58.3.4 Number of load cycles VIII-1119
58.3.5 Temperature VIII-1120
58.3.6 Temperature duration VIII-1120
58.3.7 Protective layer thickness, aluminum, section 4.3.4, Figure 4.3.4 VIII-1120
58.3.8 Stress ratios VIII-1120
58.3.9 Spectra VIII-1122
58.3.10 Surface factor KV, section 4.3.3, Table 4.3.7 ..... VIII-1122
58.4 Materials VIII-1123
58.4.1 Surface roughness VIII-1123
58.5 Settings VIII-1124
58.5.1 General settings VIII-1124
58.5.2 Required safeties VIII-1128
58.6 Estimation of the fatigue strength for surface treated parts (section 5.5) ... VIII-1130

59 Hertzian Pressure VIII-1131
59.1 Settings VIII-1134

60 Hardness Conversion VIII-1135

61 Linear drive train VIII-1136
61.1 Calculation VIII-1139
61.2 Sizings VIII-1144
61.3 Settings VIII-1144
61.4 Materials VIII-1145

62 Deformation of the Gear Body VIII-1147
62.1 Calculation procedure VIII-1148
62.2 Results VIII-1150

63 Plastics Manager VIII-1151
63.1 Gear test results VIII-1153
63.1.1 Case I. Test results measured on identical test gears VIII-1153
63.1.2 Case II. Test results available from gears with different gear geometry VIII-1154
63.2 Additional settings in the "Test data" tab VIII-1155
63.3 Settings in the Module specific settings VIII-1157
63.4 Extrapolation of the calculated permissible root and/or flank stresses VIII-1158
63.5 Other calculation options VIII-1159
63.6 Writing material data to the KISSsoft database VIII-1160
63.7 Graphics VIII-1161

IX KISSsys IX-1162

64 KISSsys: Calculation Systems IX-1163
64.1 General IX-1164
64.1.1 Structure of KISSsys IX-1164
64.1.2 Ways in which KISSsys can be used IX-1164
64.1.3 The user interface IX-1165
64.2 Creating Models in KISSsys IX-1170
64.2.1 Classic method IX-1171
64.2.2 Element Assistant IX-1172
64.2.3 System Assistant IX-1173
64.2.4 Setup with icon IX-1173
64.2.5 Creating and modifying tables IX-1174
64.2.6 Adding variables to tables IX-1175
64.2.7 Individual names for elements IX-1177
64.3 Extended functionality for developers IX-1178
64.3.1 Properties dialog IX-1178
64.3.2 Table view IX-1179
64.4 The existing elements IX-1181
64.4.1 Variables IX-1181

64.4.2 Calculation elements IX-1182
64.4.3 Elements for shafts IX-1184
64.4.4 Connection elements IX-1185
64.4.5 Displaying elements in 3D graphics IX-1186
64.4.6 System settings IX-1187
64.5 Programming in the Interpreter IX-1188
64.5.1 Expressions in variables IX-1188
64.5.2 Functions IX-1189
64.5.3 Important service functions IX-1191
64.5.4 Variable dialogs IX-1192
64.5.5 Defining 2D graphics IX-1200
64.6 Specific functionalities IX-1203
64.6.1 Load spectrum calculation IX-1203
64.6.2 Efficiency Calculation IX-1204
64.6.3 Taking into account housing deformation in static KISSsys calculations IX-1205
64.6.4 Modal analysis of shaft systems IX-1208
64.6.5 Campbell diagram for shaft systems IX-1209
64.6.6 Analysis of unbalance response of shaft systems IX-1210

X Bibliography and Index X-1213

65 Bibliography X-1214

XI Index XI-1221

Part I
General

Chapter 1
Installing KISSsoft

1.1 Basic installation

After you have inserted the KISSsoft CD in the appropriate disk drive, the setup program starts automatically. If it does not, you can run the setup.exe file directly in the CD root directory by double-clicking on it.

The setup program guides you through the installation process step by step. All you need to do is select an installation folder and the required language for the installation. If you change the default installation folder, it is advisable to include the version descriptor as part of the directory name of the other installation folder (e.g. C:/Programs/KISSsoft xx-20xx).

At the end of the installation we recommend that you install the latest Service Pack (patch). Download the latest patch http://www.kisssoft.ch/patches.php from our website. You can choose between an installation program (*.exe) and zipped files (*.zip). The installation program automatically copies the necessary files after you specify which installation folder it is to use. However, not all companies permit exe files to be downloaded. In this case, you must unpack the ZIP file and manually copy the files it contains into your installation folder. Any files that are already present must be overwritten by the ones contained in the patch.

After you have installed KISSsoft you need to license (see page I-45) it. If KISSsoft is not licensed, it will only run as a demo version.

NOTE:

If you are installing KISSsoft on a server, we recommend that you perform the installation from a client (workstation computer). Consequently, all necessary directory entries will automatically be added to the KISS.ini (see page I-54) file correctly. Otherwise, you will have to change these directory entries from the local drive name (e.g. C:/...) to the appropriate share name in the network, later, manually, using an editor.

1.2 Downloading a license file

1. Go to our website, www.KISSsoft.ch, and click on the Service/Support page link on the left. There, you will find a link to the "customer zone". Click on the link. You will see the Customer Zone web page. In that page, on the top right-hand side, enter your license number in the License Number field, and click on "Open".
2. A login window will open, in which you enter your license number, and also your download password, again. If you do not have this password, please get in touch with your commercial contact representative or contact KISSsoft directly by email on info@KISSsoft.CH or phone number +41 55 254 20 53.
3. You are now in your personal download area. Save the lizenzxxxx.lic file in the license directory of your KISSsoft installation.

NOTE:

It may be that your personal download area contains license files for different versions of KISSsoft. Please make sure you select the correct license file for the system version you have just installed.

1.3 Licensing

After you have completed the KISSsoft installation (see page I-43), you must license the software either by downloading a license file or activating the program's license. Please read the relevant section for your license type.

1.3.1 Test version

1. If you start KISSsoft from the client (workstation computer), the user account for the test version will become active.
2. Select `License tool` in the `Extras` menu and click on the Activate license tab.
3. **Activating the license online:** If your computer has Internet access, and you have received an online code from us, enter this code under the `Release Test or Student version` option and then click on `Activate license`.
4. **Activating the license directly:** Under the `Activate test version by phone` option you see a question code. Call the telephone number you see there, and tell us this code. We will then give you the appropriate answer code. Input this in the corresponding field, and then click the `Activate license tab`.

1.3.2 Student version

1. Copy your license file (you will usually be given this by your high school) to your License directory (see page I-55).
2. Select `License tool` in the `Extras` menu, and then click on the Activate license tab.
3. Input your online code (which you will also be given by your technical university) under the `Activate test or student version` option and click on `Activate license tab`.

1.3.3 Single user version with dongle (protection key)

1. Copy your license file (see page I-44) to your license directory (see page I-55).
2. Now, simply plug in the dongle supplied with the system.

NOTE

The single user version of KISSsoft can also be installed on a central server. Local clients (workstation computers) can then run the software directly from this server. Please note here that the dongle must always be plugged into each particular client.

1.3.4 Single user version with license code

1. Start KISSsoft from the client (workstation computer) for which the software is to be licensed.
2. Select `License tool` in the `Extras` menu, and then click on the Activate license tab.
3. Enter your contact data under the `Request license file` option and click on `Send` to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
4. You will receive an email as soon as we have created your license file.
5. Download your License file (see page I-44) and copy it to your License directory (see page I-55).

1.3.5 Network version with dongle (protection key)

For the network version with dongle a server program has to be installed in addition to the licensing of the KISSsoft installation.

1.3.5.1 Installation on the server

1. Copy the KISSsoft dongle/MxNet installation directory onto a server.
2. Start MxNet32 on the server. You will see a dongle icon in the task bar.
3. Double-click this icon to start the user interface.
4. Now enter `Application`: KISSsoft and any file with the file extension *.mx as the `server file`. The clients must have both read and write access to this file. Now click `New Entry` to add this entry.
5. Then click the `Active Users` button to check who is using KISSsoft. You can also reactivate a license that has already been used.

1.3.5.2 Licensing the KISSsoft system.

1. Copy your license file (see page I-44) to your license directory (see page I-55).
2. Complete the necessary details in the "ServerFile: serverfilepath" line after the checksum line in the license file. The "serverfilepath" is the path to the server file that is defined in the server program.

NOTE

The KISSsoft installation will also run if the client is not connected to the network and if the dongle is inserted in the client instead of in the server. You can also "check out" the license if you remove the dongle.

1.3.6 Network version with the license code

1. Start KISSsoft from a client (workstation computer).
2. Select `License tool` in the `Extras` menu and go to the General tab.
3. Select an access directory on a server. Please note: If you change this, you will need a new license.
4. Go to the Activate license tab.
5. Enter your contact data under the `Request license file` option and click on `Send` to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
6. You will receive an email as soon as we have created your license file.
7. Download your License file (see page I-44) and copy it to your License directory (see page I-54).

Chapter 2
Setting Up KISSsoft

2.1 Directory structure

If there are several users, it is advisable to store shared data (**databases**, user-defined **report templates** and **standard files**) on one server. This ensures that, if there are changes and upgrades, all users will be able to work with one uniform set of data. To set this up, move the KDB, EXT and TEMPLATE directories onto a server that can be accessed by all users, and then tailor the corresponding variables, KDBDIR, EXTDIR and TEMPLATEDIR, in the KISS.ini (see page I-54) file.

In contrast, the temporary directories should be defined locally on the workstations for several users. Otherwise, the interim results of individual users might overwrite each other. For each installation, KISSsoft uses the temporary user directory in accordance with the operating system. The CADDIR and TEMPDIR variables can, however, be tailored in the KISS.ini (see page I-54) file.

If you want to open or save a calculation file or report, KISSsoft displays your own personal **user directory** as the first choice storage location. This property saves you frequent searches in the directories on your system. You can define this user directory via the USERDIR variable in the KISS.ini (see page I-54) file. The user directory will be ignored if you have selected an Active working project (see page I-97). In this case, KISSsoft offers you the project directory as the first choice storage location.

2.2 Language settings

KISSsoft is available in seven languages: English, German, French, Italian, Spanish, Russian and Portuguese. When you select a language, the program differentiates between the language used for the user interface and the language used for the reports. It is therefore possible to operate KISSsoft in one language and to simultaneously display reports in a different language. Messages will be displayed either in the same language as the user interface or as the reports.

For global language settings, you need to edit the KISS.ini file (see page I-55). Additionally, you can also quickly toggle between languages in the program by selecting `Extras > Language`, and then the required language. The user can change the language used for reports by selecting `Report > Settings`.

2.3 System of units

KISSsoft recognizes two unit systems: the metric system and the US Customary Units system. For global language settings, you need to edit the KISS.ini file (see page I-55). You can also quickly toggle between systems of units in the program by selecting `Extras > System of units`. In addition to changing the system of units, it is possible to switch the unit used for a particular value input field (see page I-90).

2.4 Defining your own default files

Anyone who frequently carries out the same, or at least similar, calculations has to repeatedly enter the same values into selection lists and value input fields. Thanks to default files, KISSsoft makes your work considerably easier here. For each calculation module, there is an internal default setting for all values. If, however, you have defined your own default file, this default file will be used when you open a calculation module or load a new file.

To define a default file, you open a new file in the corresponding calculation module and enter your default settings. Click `File > Save as template` to transfer your values to the template file. All template files will be saved in the directory that has been defined as TEMPLATEDIR (see page I-54).

Default files can also be defined as project-specific files. To define special standards for a project (see page I-94), select this project in the project tree (see page I-76) and open its properties by selecting `Project > Properties`. There, select `Use own templates for this project` and specify a directory for the default files. To define the default files you must select this project as the Active working project (see page I-97).

2.5 Rights

You can restrict the rights for selected areas of KISSsoft for some users.

Right	Implementation
Changes to the general settings	Write protect the KISS.ini (see page I-54) file
Changes or additions in the databases	Write protect databases (files of the type *.kdb) as well as the directories DAT and EXT/DAT (but write rights for KDBDIR (see page I-54) should be retained)
Changes to the report templates	Write protect RPT, EXT/RPT and EXT/RPU directories
Changes to the template files	Write protect the TEMPLATE directory

2.6 Global settings - KISS.ini

Global settings for KISSsoft are defined in the KISS.ini file, which is located directly in the installation folder. Most of these settings can also be defined directly in the software and are then saved to the KISS.ini file.

2.6.1 Definitions in [PATH]

Variable name	Explanation	Note
KISSDIR=<INIDIR>	The KISSsoft installation folder is generally defined with the INIDIR variable.	
HELPDIR	Directory for user manual and help figures	
DATADIR	Directory for files of the type *.dat	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the DAT subdirectory in the EXTDIR.
RPTDIR	Directory for report templates (*.rpt)	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the RPT subdirectory in the EXTDIR (described below).
USERDIR	Default directory for opening and saving	
CADDIR	Default directory for CAD export	Should be located locally on a workstation. %TEMP% sets the temporary directory to the operating system default.
TMPDIR	Directory for temporary files	Should be located locally on a workstation. %TEMP% sets the temporary directory to the operating system default.
KDBDIR	Directory for KISSsoft's databases (*.kdb)	If several users are using the system, we recommend you store the databases on one server to ensure data uniformity if there are changes and upgrades.
EXTDIR	Directory for user-defined report templates and additional DAT files	If there are several users, it is advisable to store this directory on one server.
TEMPLATEDIR	Directory for template files (STANDARD.*).	If there are several users, it is advisable to store this directory on one server.
LICDIR	Directory for the license files	You can install this directory on one server so that all users can access the new license files.

Table 2.1: Table of variables used in the PATH environment

NOTE

You should have write permission for the directories set in these variables: TMPDIR, CADDIR, USRDIR and KDBDIR.

Depending on the configuration, you do not have write permission in the operating system directory: C:\Program Files\<KISSsoft Directory Name> or C:\Program Files\<KISSsoft Name directory>. Any files you create are then diverted to the operating system's internal directories. Here, please select directories with write permission.

The TMPDIR, CADDIR, USERDIR and EXTDIR directories can also be defined in the "Directories" tab, in the "Program settings" dialog (Extras->Settings). You can also use JAVADIR to define the path to the java.exe here. You need this file if you want to use Code_Aster (FEM) to calculate the deformation of planet carriers due to torsion.

2.6.2 Definitions in [SETUP]

Variable name	Explanation	Values
USCUSTOMARYUNITS	Sets the system of units	0: metric, 1: US customary units
MATERIALSSTANDARD	Specifies the standard in which the materials are defined (configuration tool)	0: DIN, 1: BS, 2: AISI, 3: UNI, 4: AFNOR, 5: JIS, 6: CN
REPORTLANGUAGE	Sets the language in which reports are displayed	0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian, 6: Portuguese, 11: English with US Customary Units
SHOWCALCTIME	Shows the calculation time	0: No, 1: Yes
SHOWPROGRESSBAR	Shows the progress bar for time-intensive calculations	0: No, 1: Yes
DISPLAYLANGUAGE	Sets the language in which the user interface is displayed	0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian, 6: Portuguese.
DISPLAYFONTSIZE	Sets the font size in KISSsoft (FONT)	0: System size, otherwise the direct font size
MESSAGESINREPORTLANGUAGE	Sets the language in which messages are displayed	0: as interface, 1: as reports
MESSAGESSHOWSTATE	Defines which messages are to appear as a message box.	0: all, 1: Information only in message window, 2: Informa-

		tion and warnings only in message window
EDITOR	Path to the external editor	
USEEXTERNALEDITOR	Defines whether the external editor is to be used.	0: No, 1: Yes
DATEFORMAT	Date format, e.g. DD.MM.YYYY	
TIMEFORMAT	Time format, e.g. hh.mm.ss	
ENABLENETWORKING	Defines whether the network/Internet may be accessed (for example, to display innovations).	0: No, 1: Yes
CHECKFORUPDATES	Defines whether the system is to search for updates when the program starts.	0: No, 1: Yes
USETEMPORARYDATABASE	Defines whether the databases are to be copied to a temporary directory when the program starts	0: No, 1: Yes
RECENTFILESCOUNT	Number of most recently used files in the File menu	
FORCEEXCLUSIVEOPEN	Defines whether the files can only be opened exclusively.	0: No, 1: Yes
CALCONOPEN	Defines whether calculations are to be performed immediately on a file when it is loaded	0: No, 1: Yes, 2: no if KISSsoft. is started from KISSsys, otherwise yes
CALCINTERFACEOUT	Defines whether temporary reports for manufacturing data are to be written during the calculation	0: No, 1: Yes
ENABLEUSERSETTINGS	Defines whether the settings in kiss.ini can be overwritten by local settings.	0: No, 1: Yes
USEFILEEXPLORER	Defines whether the Explorer is to appear in the "View" menu list. This process will slow down KISSsoft considerably.	0: No, 1: Yes

Table 2.2: Table of variables used in the SETUP environment

2.6.3 Definitions in [REPORT]

Variable name	Explanation
SIZE	Number 0÷9 that specifies the length of the report

INCLUDEWARNINGS	0/1: Warnings are contained in the report
FONTSIZE	Number for the font size in the report
PAPERFORMAT	Paper format: A3, A4, A5, Letter, Legal
PAPERORIENTATION	0/1: Portrait/Landscape
PAPERMARGINLEFT	Distance from the left-hand page margin [mm]
PAPERMARGINRIGHT	Distance from the right-hand page margin [mm]
PAPERMARGINTOP	Distance from the top page margin [mm]
PAPERMARGINBOTTOM	Distance from the bottom page margin [mm]
COMPARE	0/1: Adds date/time to the report in comparison mode
SAVEFORMAT	0÷4: RTF, PDF, DOC, DOCX, TXT
LOGO	Picture file displayed in the header and footer
HEADER	Definition of the header
USEHEADERFORALLPAGES	0/1: header only on first page/on all pages
FOOTER	Definition of the footer
USEFOOTERFORALLPAGES	0/1: footer only on first page/on all pages

Table 2.3: Table of variables used in the REPORT environment

2.6.4 Definitions in [GRAPHICS]

Variable name	Explanation
BACKGROUND	0: black, 15: white (for more information, see Graphics > Settings)

Table 2.3b: Table of variables used in the GRAPHICS environment

2.6.5 Definitions in [LICENSE]

Variable name	Explanation
LOGGING	Number to activate the logging of license usage 0: no log file 1: Log in, Log out, No license, Used and Missing rights 2: Log in, Log out, No license

	3: Log in, Log out, No license, Missing rights In network versions, the user's uptime is also displayed in minutes when they log out.
LICENSELOGFILE	*.log file for generating reports of license usage
TIMEOUT	Duration until an unused floating license is activated on the network again [min]

Table 2.4: Table of variables used in the LICENSE environment

2.6.6 Definitions in [CADEXPORT]

Variable name	Explanation
USEDXFHEADER	0/1: DXF header will be used for DXF export
DXFVERSION	0/1: Version 12/15
INPUTLAYER	Name of the layer for import
OUTPUTLAYER	Name of the layer for export
DXFPOLYLINE	0/1/2: Uses polygonal course, lines or points for the export

Table 2.5: Table of variables used in the CADEXPORT environment

2.6.7 Definitions in [INTERFACES]

Variable name	Explanation
DEFAULT	Name of the CAD system:
	Solid Edge
	SolidWorks
	Inventor
	CATIA
	Creo
	HiCAD
GEAREXPORT3D	Displays the CAD system name in lists (see DEFAULT)
SYMMETRIC	0/1: Full tooth space/half tooth space mirrored (symmetrical) (default = 0)
SAVEFILENAME	0/1: Saves the entire file contents/Saves only the file name and the

	path (Default = 1)

Table 2.6: Table of variables used in the INTERFACES environment

2.6.8 Definitions in [SOLIDEDGE]

Variable name	Explanation
LIBRARY	Interface dll (kSoftSolidEdge.dll) directory
SIMPLIFIEDPRESENTATION	0/1: Set the variable to 1 to also generate a simplified gear
SMARTPATTERN	0/1: Fastpattern/Smartpattern
APPROXIMATION	1/2/3/4: Polygonal course (supported)/Arcs of circles (supported)/Quadratic splines (supported)/Cubic splines (default)
USERPARTTEMPLATE	Template file directory (e.g. C:\Template\metric.par) or just the template file name (e.g. metric.par) - the path is then taken from the settings in Solid Edge)

Table 2.7: Table of variables used in the SOLIDEDGE environment

2.6.9 Definitions in [SOLIDWORKS]

Variable name	Explanation
LIBRARY	Interface dll (kSoftSolidWorks.dll) directory
SIMPLIFIEDPRESENTATIONNAME	Setting this variable generates a simplified gear with this name
APPROXIMATION	1/2/3/4: Polygonal course (supported)/Arcs of circles (supported)/Quadratic splines (supported)/Cubic splines (default)

Table 2.8: Table of variables used in the SOLIDWORKS environment

2.6.10 Definitions in [INVENTOR]

Variable name	Explanation
LIBRARY	Interface dll (kSoftInventor.dll) directory
APPROXIMATION	1/2/3/4: Polygonal course (supported)/Arcs of circles (supported)/Quadratic splines (supported)/Cubic splines (default)

Table 2.9: Table of variables used in the INVENTOR environment

2.6.11 Definitions in [CATIA]

Variable name	Explanation
LIBRARY	Interface dll (kSoftCatia.dll) directory
LIBRARYSWMS	Interface manufacturer's *.dll file directory
LANGUAGEFILE	Interface manufacturer's *.ini file directory
DEBUG	Interface manufacturer's variable
DEBUGPATH	Interface manufacturer's variable
HELPFILE	Interface manufacturer's variable
LASTSETTING_CONSTRUCTION	Interface manufacturer's variable
LASTSETTING_GEARNAME	Interface manufacturer's variable
LASTSET-TING_PRODUCTIONINFO	Interface manufacturer's variable
LASTSETTING_CALCINFO	Interface manufacturer's variable
LASTSETTING_FLAGINFO	Interface manufacturer's variable
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/Arcs of circles (not supported)/Quadratic splines (default)/Cubic splines (not supported)

Table 2.10: Table of variables used in the CATIA environment

2.6.12 Definitions in [PROENGINEER]

The ProEngineer interface has an individual subsection/menu for each version (for example, Wildfire 5, 32bit). However, the definitions in "kiss.ini" are the same in every 3D interface to Creo Parametric (ProEngineer) chapter.

Variable name	Explanation

LIBRARY	Interface dll directory (kSoftProEngineer.dll)
INTERFACECOMMAND	Directory containing the interface manufacturer's *.exe files
USCUSTOMARYUNITS	0/1: System of units used in the metric or US Customary Units model
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/arcs of circles (default)/quadratic splines (not supported)/cubic splines (not supported)

Table 2.11: Table of variables used in the PROENGINEER environment

2.6.13 Definitions in [HICAD]

Variable name	Explanation
LIBRARY	Interface dll directory (kSoftHiCAD.dll)
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/arcs of circles (default)/quadratic splines (not supported)/cubic splines (not supported)

Table 2.14: Table of variables used in the HICAD environment

2.7 User-defined settings

User-defined settings can be reset via `Extras > Configuration tool`.

2.7.1 Configuration tool

In the `General` tab, you can select the older version's "kdb" database directory (prior to 03-2017. After that, it is called "udb".) by selecting the `Update database` option. Click "Run" to transfer the data records you defined yourself in the older version to the current version, to ensure these records are available in the current version.

Click `Update external data` to select the "ext" directory of the older version. This then automatically copies the "dat", "rpt" and "rpu" subdirectories to the current release.

Click `Update settings` to transfer your personal settings from the previous version to the current release.

Select `Connect file extensions` to link all the KISSsoft files with the current version so that you can double-click on any particular file to open it in the current release.

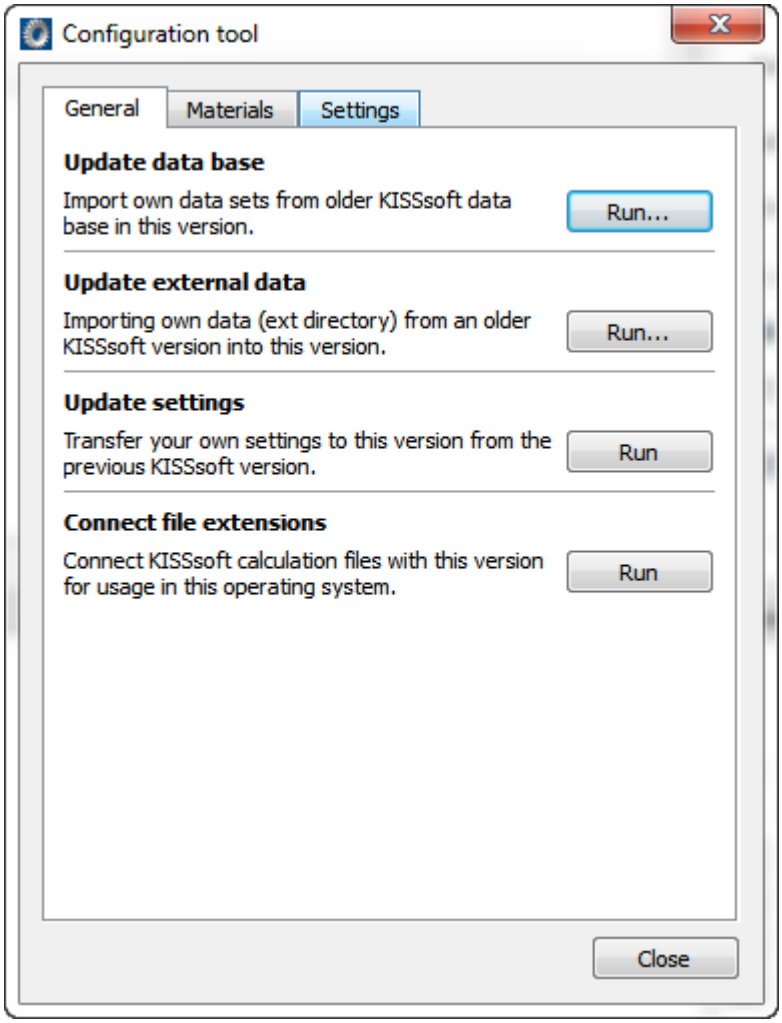


Figure 2.1: `General` tab in the `Configuration tool` window

In the Materials tab, you can specify the standard with which the material descriptions in the database are to comply.

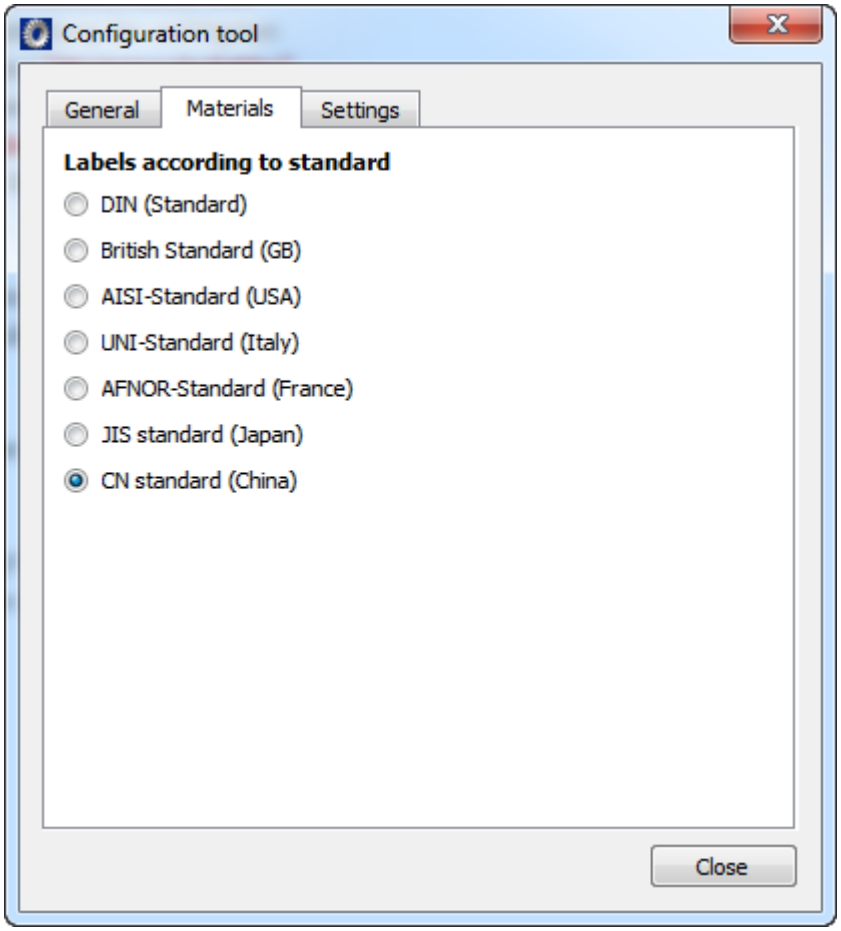


Figure 2.2: Materials tab in the Configuration tool window

In the `Settings` tab, you can delete the user-defined settings (divided into groups). This reloads the default values.

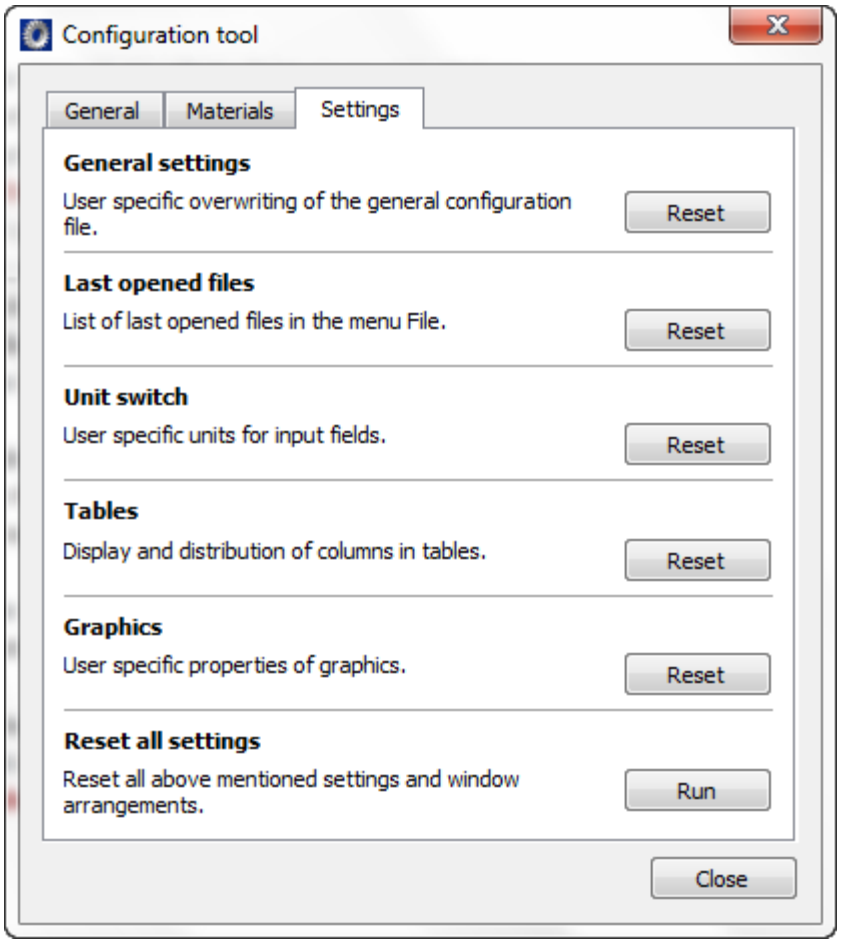


Figure 2.3: `Settings` tab in the `Configuration tool` window

2.8 Rules

Rules are used to ensure that in-house guidelines for the ranges of validity of parameters are applied and adhered to. This typically concerns the maximum and minimum limits of input values, calculated values and the relationships between these values i.e. length-width relationships, length-diameter relationships or even the relationship between the module and the center distance.

These rules are defined by being stored in a <module>.rls file, where <module> stands for the calculation module's in-house label, e.g. Z012 for cylindrical gear pairs.

These rules are subdivided into those that must be fulfilled before the calculation is performed and those that must be checked afterwards. If a rule is infringed, the appropriate messages can be displayed. In the case of rules that must be checked before the calculation, variables can also be set to constant or calculated values.

The following statements are possible in this situation:

precalc: marks the beginning of the rules that must be checked before a calculation is performed.

postcalc: marks the beginning of the rules that must be checked after a calculation.

assert(<Condition>): The <condition> is ensured. In this case, the <condition> usually represents a comparison in which both the right-hand and left-hand side of the comparison can also be calculated.

action msg <Message>: If the <condition> of the previous assert has not been fulfilled, the <message> is output. Here the <message> can include variables, in the same way as report templates.

action set <Assignment>: If the <condition> of the previous assert has not been fulfilled, the <assignment> is performed. The assigned value can be a constant or be calculated from variables, in the same way as for the report templates.

An assignment is only really useful in the precalc section because changing the contents of variables after the calculation merely leads to inconsistent results and has no other effects.

Here is an example file for a helical gear calculation:

```
precalc

assert (ZR[0].x.nul < 1)

  action msg "Profile shift for Gear 1 too big, is {ZR[0].x.nul}, maximum 1.
It has been reset to 1."
```

```
  action set ZR[0].x.nul = 1


assert (ZR[1].x.nul < 1)

  action msg "Profile shift for Gear 2 too big, is {ZR[1].x.nul}, maximum 1.
It has been reset to 1."

  action set ZR[1].x.nul = 1

postcalc

assert ((ZP[0].a/ZS.Geo.mn) < 200)
```

action msg "Center distance too big for module (a={ZP[0].a}, mn={ZS.Geo.mn}, a/mn={ZP[0].a/ZS.Geo.mn})."

Explanations:

The "precalc" statement open the section of the rules that must be executed before the calculation.

The first "assert" statement checks whether the nominal profile shift of gear 1 is less than 1.0.

If this "assert" is not fulfilled, the "action msg" statement outputs the message that the profile shift is too big, displays the current value and tells you that the profile shift has been set to 1.0.

The "action set" then sets the profile shift to 1.0.

The second "assert" statement checks the same values for gear 2.

The "postcalc" statement signifies the end of the set of rules to be executed before the calculation and opens the section of the rules to be checked after the calculation. The example shows a definition of an "assert" statement. This checks the ratio from the center distance to the module. If the rule is infringed, the "action msg" statement triggers a message. However, there is no point in converting one of these two values after the calculation and this is why the "action set" statement is not present here.

The "General – Results and Reports – Report templates – Formatting – Calculation variables" section details which operators and functions can be used in the formulae.

The file containing the rules is stored in the template directory (TEMPLATEDIR, usually the "template" subdirectory, see the "Directory structure" section in the "Setting Up KISSsoft" section). As the template directory can also be project-specific, you can also define project-specific rules.

Chapter 3
Starting KISSsoft

3.1 Initial parameters

You can run KISSsoft from the input prompt with the following initial parameters:

Parameter	Description
INI=directory	The KISS.ini (see page I-54) file will be loaded from the specified location. You can transfer a file name including its directory path, or only a directory name.
START=module	The specified calculation module will be started. The module descriptor is, for example, M040 for bolt calculation or Z012 for cylindrical gear pair calculation.
LOAD=file name	The calculation module belonging to the file is started and the file is loaded. If the supplied file name does not include a path, the system looks for the file in the User directory (see page I-54).
LANGUAGE=number	KISSsoft starts with the language specified for the interface and reports. (0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian, 6: Portuguese, 11: English with US Customary Units)
DEBUG=filename	A log file containing debug information will be written. It can be very helpful for error-tracking. It is advisable to define the file name with a complete path, so that you can find the log file easily later.
File name	The calculation module belonging to the file is started and the file is loaded. This also provides a way to associate KISSsoft with the appropriate filename extensions in Windows.

3.2 Disconnect license from the network

If KISSsoft has not been properly shut down, it may be possible that users remain registered, in the case of a network version. This may lead to licenses being blocked even though some users are no longer working with KISSsoft. You can disconnect a license from the network by selecting the required license (the user and start time are also specified) under `Extras > License tool` in the `Network` tab, which deletes the appropriate cookie file and activates the blocked license on the network again.

Unused licenses will be activated after a certain time, as soon as the next user logs on. This time-span can be predefined via the TIMEOUT (see page I-57)variable in the kiss.ini (see page I-54) file.

NOTE

A user who has been disconnected from KISSsoft can no longer carry out calculations in the current session. The user must restart KISSsoft. However, data backups can still be carried out.

Chapter 4
User Interface

KISSsoft is a Windows-compliant software application. Regular Windows users will recognize the elements of the user interface, such as the menus and context menus, docking window, dialogs, tooltips and status bar, from other applications. Because the internationally valid Windows Style Guides are applied during development, Windows users will quickly become familiar with how to use KISSsoft.

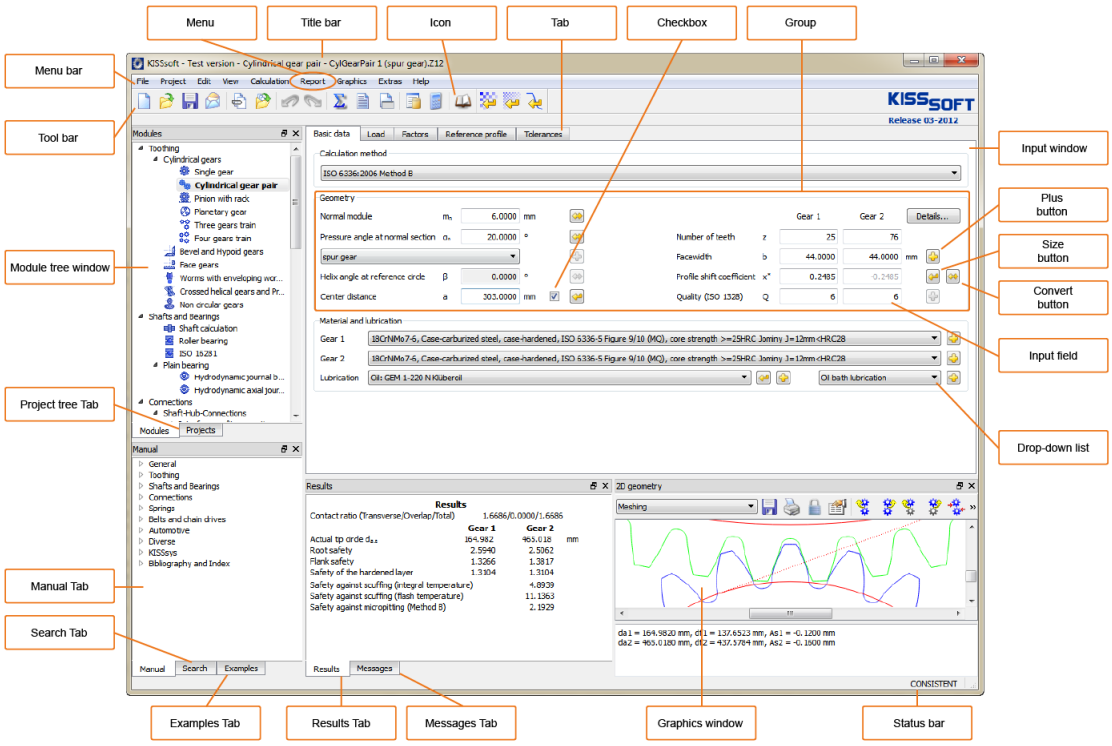


Figure 4.1: KISSsoft's user interface

4.1 Menus, context menus and the Tool Bar

In the `File` main menu, you can open and save calculation files, and send them as e-mail attachments, restore previous calculation stages, view file properties and close KISSsoft. Click `File > Save as template` to retain user-defined default values (standard files (see page I-52)).

You can use the KISSsoft Project Management (see page I-94) functionality from both the `Project` main menu and the Project tree (see page I-76). You can open, close and activate projects, insert files into a project, or delete them, and also view project properties.

Each individual docking window (see page I-75) in the user interface can be hidden or displayed in the `View` main menu. If you are in the report or helptext viewer, select `View > Input window` to return to the calculation module input dialog.

In the `Calculation` main menu, you can run the current calculation (see page I-88), add more calculations to the calculation module as default or special tabs and call subcalculations as dialogs. Select `Calculation > Settings` to change the module-specific settings.

In the `Report` main menu you will find actions for producing and opening a report. The system always generates a report for the current calculation. Click `Report > Drawing data` to display Drawing data (on page I-113) for the element currently selected in the Report Viewer (see page I-85). Click `Report > Settings` to change the report's font size, page margins and scope. The actions for saving, sending and printing are only active if a report is open.

You can open and close the Graphics (see page I-78) window of a calculation module in the `Graphics` main menu. Select `3D export` to access KISSsoft's CAD interfaces. Select `Graphics > Settings` to select the CAD system into which you want to export the selected element.

In the `Extras` menu you will find the license tool, the configuration tool and the database tool. In this main menu, you can start the Windows calculator and change the language (see page I-50) and system of units (see page I-51). In `Extras > Settings`, you can change general program settings such as the formats for time and date values.

In accordance with Windows conventions, at the end of the menu bar you will find the `Help` icon, which you can use to navigate in the KISSsoft manual. In `Help > Info` you will find information on the program version and on the support provided by KISSsoft.

In addition to the main menu, KISSsoft uses **context menus** in many locations. Context menus give you access to actions for a particular area or element of the

software. Normally, you click the right-hand mouse button to display context menus.

The **Tool bar** gives you faster access to actions from the menus that are used particularly frequently. You should also note the tool tips which display information about the actions in the tool bar as well as other descriptions in the status bar (see page I-87).

NOTE

The `Calculation`, `Report` and `Graphics` main menus are only active if a calculation module is open. The actions available in these menus may vary depending on the current calculation module.

4.2 Docking window

Beside the menu bar, tool bar and status bar, the docking windows are important elements in the KISSsoft user interface. Docking windows are windows that, can either be moved freely on the desktop, like a dialog, or can be docked onto the pages of the program, in any arrangement that suits you. Several docking windows can be placed on top of each other and be represented as tabs.

You can unlock a docking window by double-clicking in its title bar. You move a docking window by clicking with the left-hand mouse button in the title bar and moving the mouse with the key held down. If you move the mouse close to the edge of the main window, a new position for the docking window will be displayed. Release the mouse button to position the docking window. Docking windows can be displayed and hidden via the `View (see page I-73)` menu.

4.2.1 The module tree

The module tree shows all KISSsoft calculation modules in an easy to understand and logically structured list. Any calculation modules for which you have not purchased a license are grayed out. To open a module, double-click on it with the left mouse button. The current calculation module will be shown in bold.

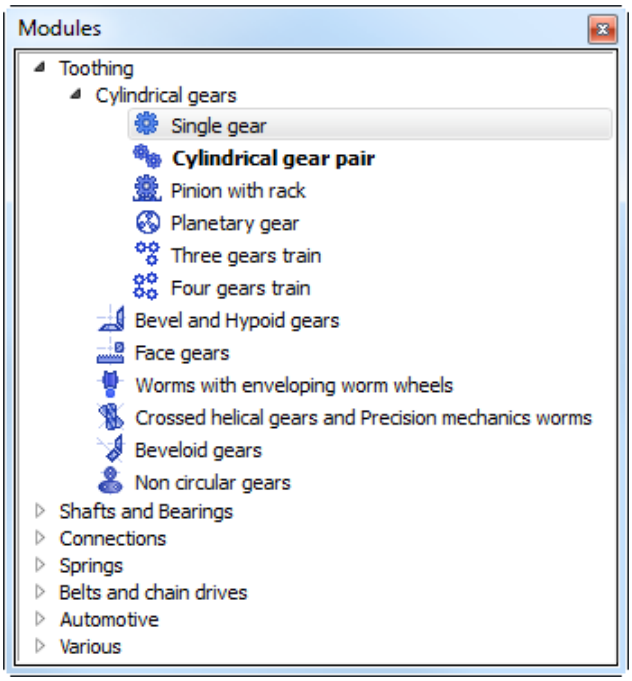


Figure 4.2: KISSsoft calculation modules

4.2.2 The project tree

The project tree gives you an overview of the open projects, and the files belonging to these projects, and highlights the active working project (see page I-97) in bold. You use the project management (see page I-94) functions via the `Project` menu or from a context menu (see page I-73).

4.2.3 The Results window

The KISSsoft results window displays the results of the last calculation.

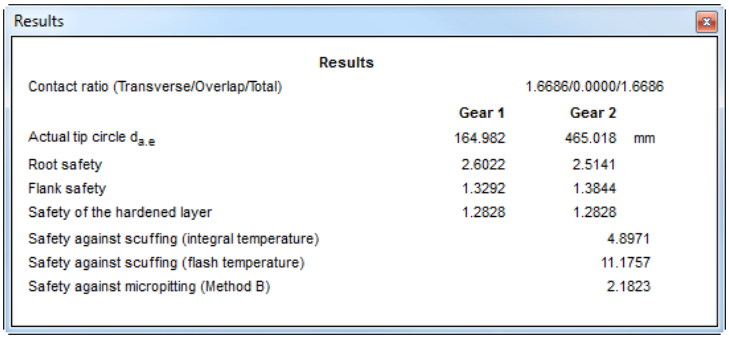


Figure 4.3: The KISSsoft results window

4.2.4 The Messages window

The messages window displays all information messages, warnings and errors. Generally, all additional messages are not only displayed, but also appear in a message box. You can change the way information and warnings are displayed in a message box by selecting `Extras > Settings`.

4.2.5 The info window

The Info window displays information that is displayed when you click on an Info (see page I-90) button in the calculation module. You zoom and print the information via a context menu (see page I-73).

4.2.6 Manual and Search

The manual's Table of Contents and search function are also available as docking windows. When you select an entry, by double-clicking on it, the Helptext viewer (see page I-86) opens and the relevant section in the manual is displayed.

4.3 Graphics window

In KISSsoft, you can open as many graphics windows as you need at the same time and arrange them in the same way as the other docking windows (see page I-75). This means you can see all the graphics and diagrams you require for your calculations at a glance. To make working with graphics more effective, you can use the tool bar (see page I-79), the Comment field, the context menu (see page I-81) and the Properties (see page I-81).

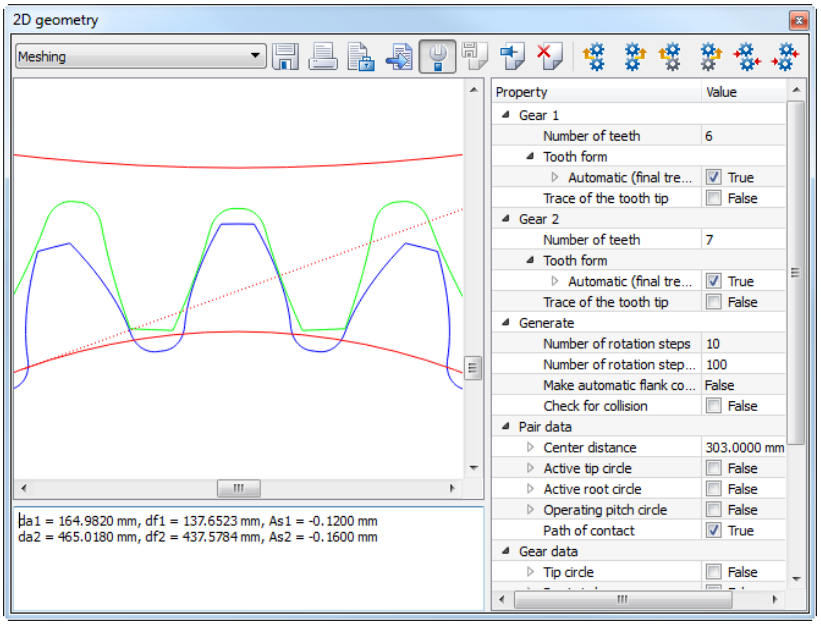


Figure 4.4: Components of the graphics window

4.3.1 Tool bar and context menu

Use the selection list in the tool bar to switch from one graphic to another in a group. You will also see various icons for saving, printing and locking a graphic, as well as functions for highlighting and graying out its properties.



Save graphics as

This stores the graphics as DXF, IGES or other image or text formats under the name you enter here.

Saving diagrams in a DXF file usually creates a conflict between the diagram axis units and the unit used in the DXF file. For this reason, when you save a diagram, the program opens a dialog in which you can specify the drawing area to which the diagram is to be projected in the file.



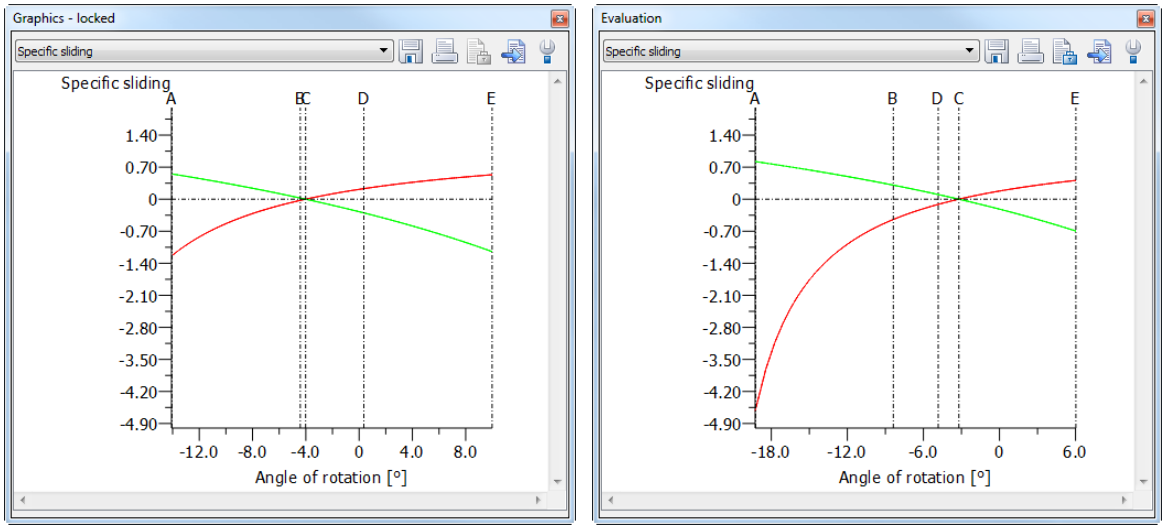
Print

Prints the current section of the graphic. The information underneath the graphics is defined in the graph?.rpt report templates (see Report templates (on page I-117)).



Lock

This is useful for comparing two calculation results. In this way, you can, for example, generate a `Specific sliding` graphic for a toothing scenario, lock this graphic and then, after having changed the gear parameters, open a new graphics window that shows the new calculation results. The locked window will no longer be updated.



(a) Locked window (b) Window with new calculation results

Figure 4.5: Locking graphics windows

When you lock a graphics window, a dialog will open, in which you can enter a title for the window, which will make it easier for you when you are comparing different graphics.

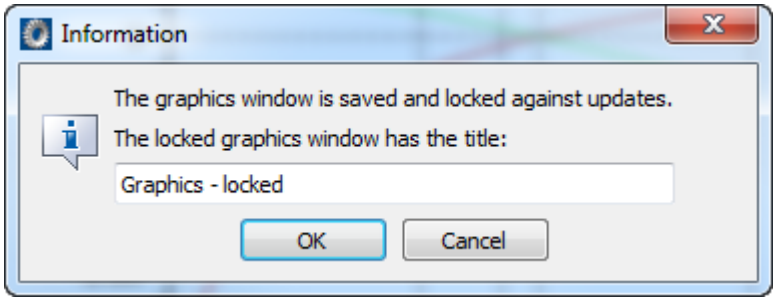


Figure 4.6: Dialog window for inputting the window title

Properties

This opens a list with the Properties (see page I-81) of the current graphic in the same window.

4.3.2 Comment field

In the Comment information is displayed about the graphic. You can change the Comment to suit your needs and it is included in the print output.

4.3.3 Context menu

Here, use the left-hand mouse button to select, move, zoom and measure elements in a graphic. You can permanently select which action is to be performed in the context menu. You can access this more quickly by using these combinations: Move: Shift, Zoom: Ctrl and Measure: You can select multiple separate items by pressing the Alt key while pressing the left-hand mouse button.

Other actions in the context menu are: Zoom In (plus), Zoom out (minus) and Fit window (Pos1 or Home). Use the direction keys to move the current section of the graphic.

4.3.4 Properties

In Properties, you can display or hide elements in a graphic and change its colors and line styles. You can make different modifications, depending on the graphic: for diagrams and such like, you can modify the value ranges and units to match the axes, or, for a meshing, you can change the center distance.

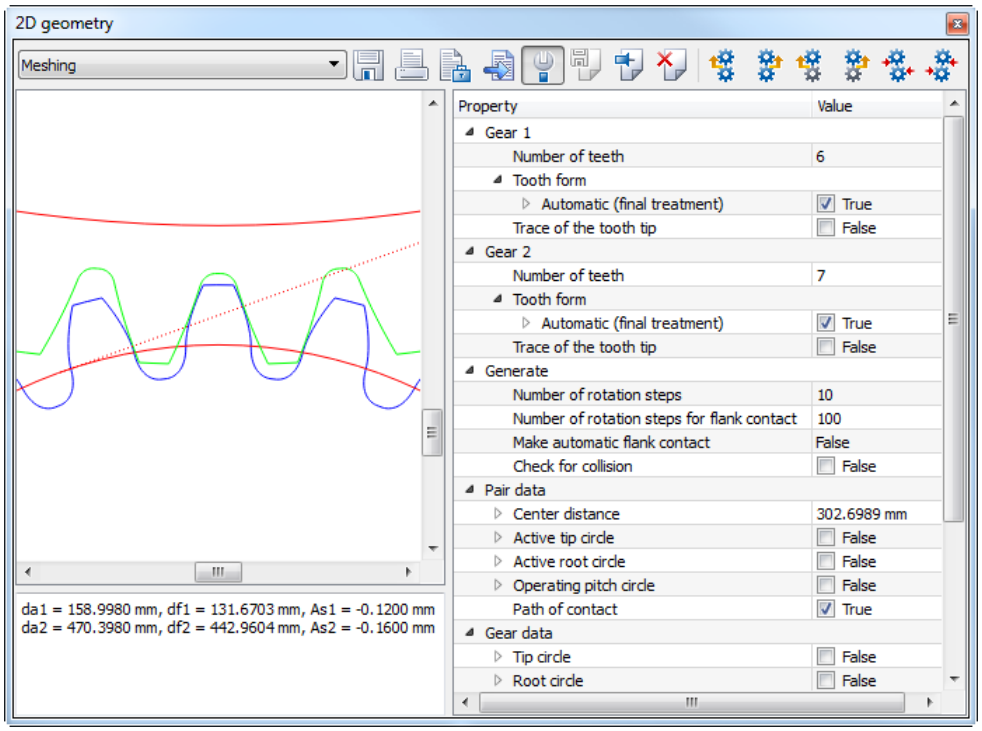


Figure 4.7: Graphic properties

If the properties are displayed, you will see three other icons in the tool bar. You use them to store curves in a graphic as text, or in the graphic itself.



Save curve as text

Stores the coordinates of the curve selected in Properties in a text file. This makes it easy to transfer curves to, for example, an Excel file.



Save curve

Stores the curve selected in Properties in the graphic. This function is ideal for comparing the graphical outputs of a calculation while you change its parameters.



Delete memory

Deletes the curve from the memory.

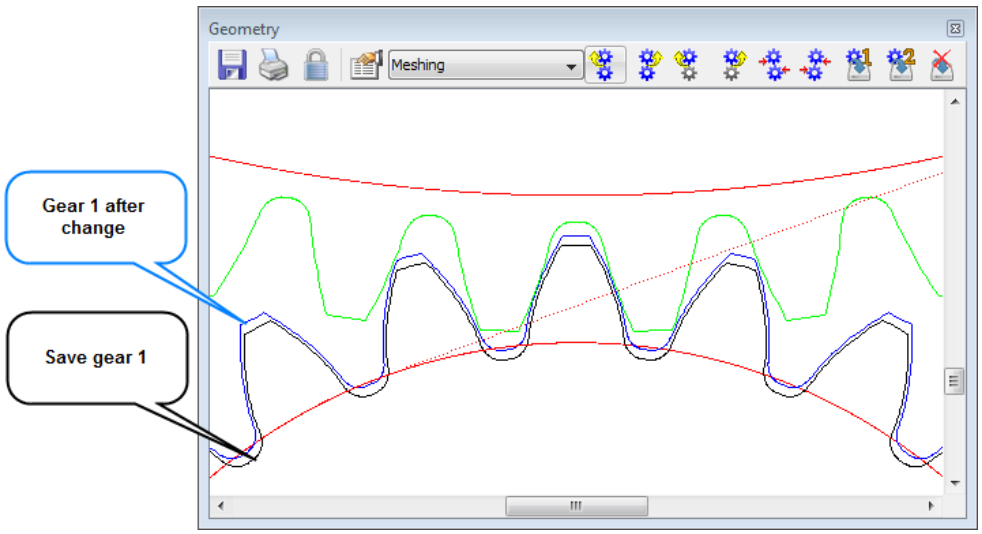


Figure 4.8: Graphics with saved and different curves

4.3.5 Toothing

If you select Toothing, additional icons are displayed for generating the gear pair and creating the flanks when you open the `Geometry` graphics window.



Rotate to the left

Rotates the gear pair to the left.

Key combination: Ctrl + left direction key



Rotate to the right

Rotates the gear pair to the right.

Key combination: Ctrl + right direction key



Rotate independently to the left

One gear remains static while the other is rotated to the left. The profiles overlap.

Key combination: Alt + left direction key



Rotate independently to the right

One gear remains static while the other is rotated to the right. The profiles overlap.

Key combination: Alt + right direction key



Make flank contact left

The gears are rotated until the flanks of both gears touch on the left.



Make flank contact right

The gears are rotated until the flanks of both gears touch on the right.

NOTE:
Hold down a rotate button to rotate the gears continuously (movie).

NOTE:
Click Properties (see page I-81) to specify the number of rotation steps for the rotation. The number of rotation steps here refers to the pitch.

4.4 Main input area

The main input area shows a calculation module's input window. In addition, it is used to display the internal report viewer or the internal help viewer.

4.4.1 Report Viewer

When you generate a report in KISSsoft, the report viewer in the main input area will open, the entries in the `Report` menu will be activated and the report viewer tool bar will be displayed. The report viewer is a text editor that supports the usual functions for saving and printing a text file. In KISSsoft, you can save reports in Rich Text Format (*.rtf), in portable document format (*.pdf), in Microsoft Word format (*.doc) or as ANSII text (*.txt).

The report viewer's other functions are Undo/Redo, Copy, Cut and Paste, and Search with the usual shortcuts. You can zoom in on the view and edit the report later on by changing the font size, bold, italics and underlining style. To change the general appearance of the report, select `Report > Settings`.

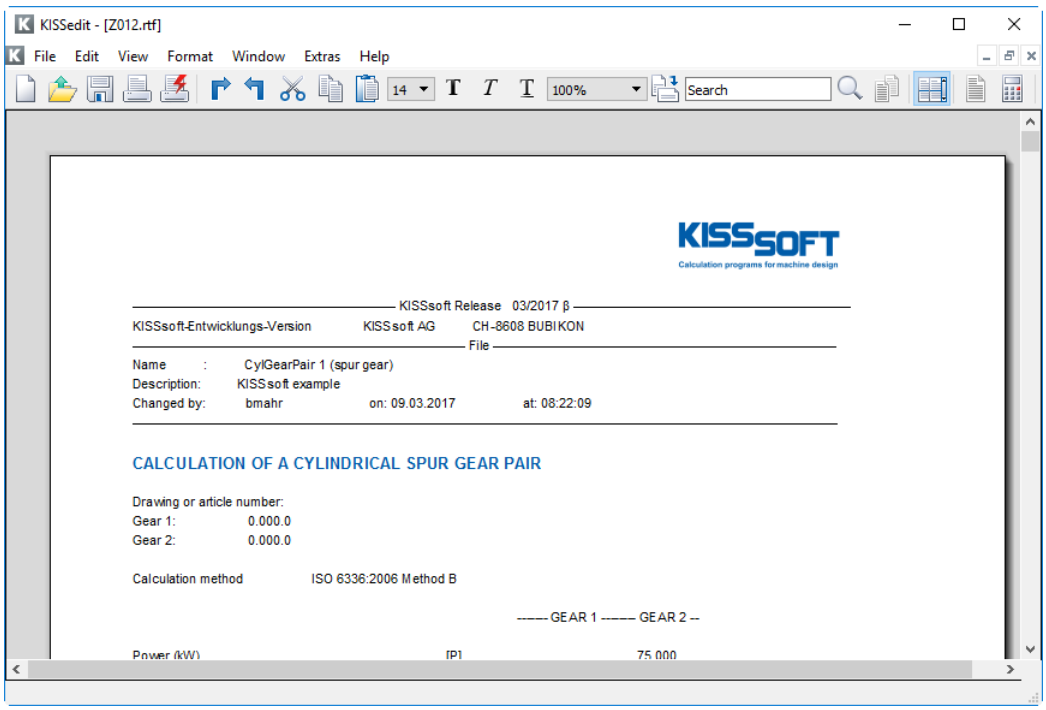


Figure 4.9: The KISSsoft report viewer

4.4.2 Helptext Viewer

The KISSsoft manual is displayed in the Helptext viewer in HTML format. To open the manual, select something in the Table of Contents or the Search function. If you press function key `F1`, the system displays more information on the location in KISSsoft at which the cursor is currently is located.

4.5 Tooltips and status bar

Whenever it is useful, tool tips are provided in KISSsoft, to give you additional information about program elements. Tooltips appear automatically if you slowly move the mouse over a program element.

If you position the mouse over a particular menu option, the system will display detailed information on all actions available in that menu, in the left-hand area of the status bar. If the mouse is positioned over a selection list, the currently selected list entry will be displayed in the status bar. This is especially helpful if the display is restricted by the width of the selection list.

In the right-hand area of the status bar, the system will display the current status of the calculation. The flag is set to CONSISTENT if the results are current. INCONSISTENT shows that a new calculation needs to be carried out.

Chapter 5
KISSsoft Calculation Modules

5.1 Standard and special tabs

The input window for most calculation modules is subdivided into different tabs. This ensures that inputs are separated logically. For more complex calculations such as for a cylindrical gear pair, the system does not automatically display all existing tabs. When you open a new calculation, you only see the tabs that contain the absolutely necessary inputs (e.g., for a cylindrical gear pair this would be the Basic data, Reference profile and Tolerances tabs). In the `Calculation` menu you can add more tabs if needed (e.g., for a cylindrical gear pair you would need the "Modifications" and "Correction of the gears" tabs).

KISSsoft calculation modules use two types of tabs: Standard tabs and special tabs, as shown in Figure 1.1.

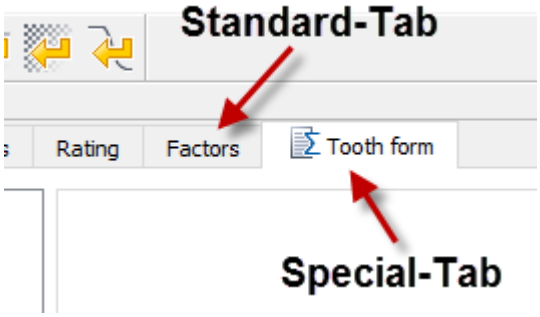


Figure 5.1: Standard and special tabs

If a standard tab (e.g. Basic data) is active when the calculation is run, then the standard calculation will be executed and the results of this standard calculation will be displayed in the Results window (see page I-76). When a report is generated, the default report is created.

Special tabs are marked with the  icon. If a special tab is active when the calculation is run, then a special calculation will be executed in addition to the standard calculation, (e.g., for a cylindrical gear pair the calculation of the meshing line under load). The results of this additional calculation will then be displayed in the Results window, and when you generate reports you will get a report containing the results the additional calculation.

5.2 Input elements

All KISSsoft calculation modules use the same input elements for input. These input elements are described in more detail in the sections that follow.

5.2.1 Value input fields

In general, a value input field always includes the label of the variable, a formula character, the edit field and a unit. If the edit field is grayed out, this variable cannot be predefined. Instead it will be determined during calculation. One or more of the following buttons can follow a value input field:



You can retain a value by selecting the Check button



You can set a radio button to specify which values in a group should be calculated and which should be retained



Click the sizing button to calculate the value using calculation methods



Click the convert button to calculate the value using conversion formulae



Click the plus button to display additional data for a value



Click the Info button to display information in the Info window (see page I-76)

5.2.2 Formula entry and angle input

In some cases it is advisable to use a small auxiliary calculation to determine a value. Click the right-hand mouse button in the edit field of a value input field (see page I-90) to open a formula editor. In it, you can enter a formula, which must be one of the four basic calculation types: +, -, * and /. Additionally, you can use all the functions that are supported by the report generator (→ see Table on page I-123). Confirm the formula by pressing Enter. The system will evaluate the formula. The formula itself will be lost: if you return to the formula entry dialog, the calculated value will be shown there instead of the formula.

In value input fields (see page I-90) that display an angle, a dialog in which you can input degrees, minutes and seconds will be displayed instead of the formula editor.

5.2.3 Unit switch

In KISSsoft, you can switch all the units in the value input fields (see page I-90) and in the tables (see page I-90). To do so, click on a unit with the right-hand mouse button. A context menu will open, offering all possible units for the value. If you select a different unit from the one that is currently in use, KISSsoft converts the current value in the value input field into the new unit.

To switch between metric and US customary units globally, select `Extras > System of units`.

5.2.4 Tables

In some modules, data will be displayed or entered in a table. You select a row by double-clicking, just like when you select a field for input. For tables, additional information is often displayed in a tooltip (see page I-87). In general, the following buttons come after tables so that you can input data:



Click the Add button to add a row into the table.



Click the Remove button to delete the selected row from the table



Click the Clear button to delete all entries in the table

5.3 Calculating and generating a report

Click on `Calculation` > Click `Run` to perform the current calculation. In addition, the tool bar and the `F5` function key give you quick, convenient access to this action. Here, please note that a calculation module can have other special calculations in addition to the standard calculation. These special calculations are only executed if the appropriate special tab (see page I-89) is active.

Select `Report > Generate` to generate a report about the current calculation. Also note the differentiation here between the default report and the reports about the special calculations in the special tabs (see page I-89).

The status of a calculation is consistent if it could be performed without error. As soon as you change data in the input window, the calculation becomes inconsistent, which means that the results of the calculation in the Results window and the graphics no longer match with the data in the interface. The current status of the calculation is displayed in the status bar (see page I-87).

5.4 Messages

A calculation sends different types of messages to the input window: information, warnings and errors. Information and warnings should always be taken note of to ensure accurate results. If an error has occurred, the calculation is interrupted.

Normally, all the messages are displayed in a message box and in the Messages window (see page I-76). You can change the way information and warnings are displayed in a message box by selecting `Extras > Settings`.

Chapter 6
Project Management

KISSsoft contains its own project management system, which you can use to organize your calculation files and external files. The most important area in the project management system is the KISSsoft project tree (see page I-76). In it, you can see which projects are currently opened or active, and you can see all the information about the files belonging to the individual projects.

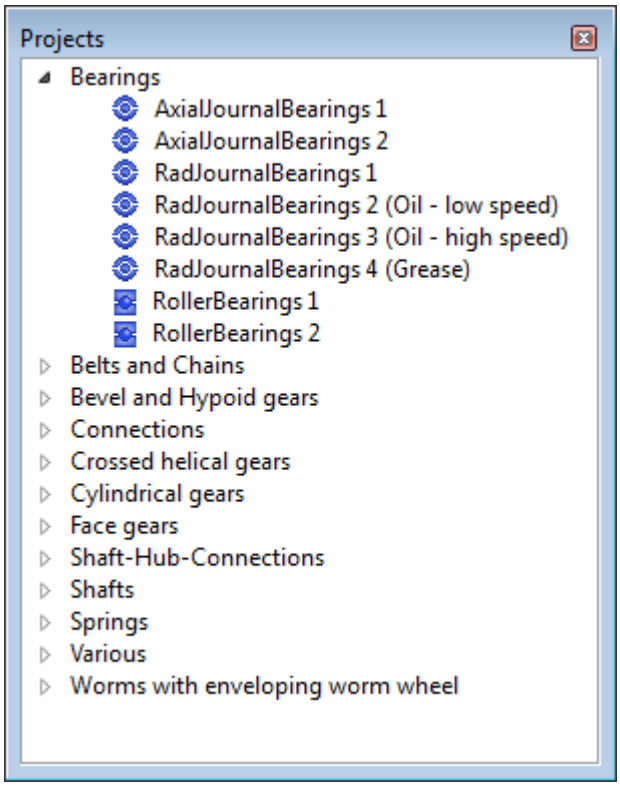


Figure 6.1: The KISSsoft project tree

6.1 Creating, opening and closing projects

Click on `Project > New...` to create a new project. A dialog opens in which you enter the name of the project, the project directory, descriptions and comments, and also the directory for the templates (see page I-52) that are to be used. The newly created project is inserted into the project tree and defined as the Active working project (see page I-97).

When you open an existing project (click on `Project > Open...`) this is also inserted into the project tree and defined as the Active working project (see page I-97).

You close a project by selecting it and then clicking `Project > Close`. You will also find this action in the project tree's context menu (see page I-73). The project will still be retained, and you can open it again at any time.

6.2 Adding and deleting files

Files can be added and deleted either via the Project properties (see page I-99) or the context menu (see page I-73). Not only can you insert calculation files from KISSsoft into a project, but also any external files.

6.3 The active working project

The project tree shows all opened projects, and it is not absolutely necessary to define an active working project. If you have defined an active working project, it is highlighted in bold. Click on `Project > Set as working project` to enable a project. Alternatively, you can do this by selecting a context menu option. If you select `Project > Work without project`, this deactivates the active working project.

The current calculation file does not necessarily have to belong to the active working project.

6.4 Storage locations

Files belonging to a project do not necessarily have to be stored in that project's directory. Consequently, files can also belong to several projects simultaneously. However, if you have defined an active working project (see page I-97), KISSsoft will prompt you with its project directory as the first choice storage location whenever you want to open or save a calculation file or a report. If you are working without a project, the system will display your personal user directory (see page I-54) as a default storage location, which you can change.

6.5 Project properties

Click on `Project > Properties` to display the project properties of the selected project. Alternatively, you can display this in the project tree's context menu (see page I-73).

Chapter 7
Dynamic User Interface

The KISSsoft interface is defined by its editable text files (descriptive data). The elements it contains are fixed components of the software. However, any user can decide how these elements are divided up and arranged. Frequently used entries can be given priority in the tabs and dialog and less commonly used entries can be either hidden or write-protected. KISSsoft can therefore easily be adapted to suit the requirements of individual users.

7.1 Modified tabs and dialogs supplied with the system

The description files for the tabs and dialogs supplied with the system are stored in the kui (kisssoft user interface) directory. These files should never, under any circumstances, be modified by the user. This is because interface upgrades, which are supplied with a patch, always overwrite any user modifications. To modify the interface to suit your own requirements, copy the corresponding description file to the ext/kui directory and change it there. KISSsoft evaluates the files in this directory first. The description files are assigned to the corresponding calculation module by file name and file extension *.kui. This is why the file name must not be changed.

7.2 Adding additional tabs and dialogs

The description files for additional tabs and dialogs are stored in the ext/dui (dynamic user interface) directory. KISSsoft evaluates the files in this directory every time a module is started. You can give these files any name you want, although the file extension must always be *.dui.

The <module> tag tells KISSsoft which calculation module the description file was defined for. This entry is mandatory for tabs. The titles of the tabs or dialogs are defined by the <title> tab. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

Use the <before> tag to define the position of the additional tabs. If you do not see the <before> tag, the additional tab is placed after the standard system tabs. An additional tab can also be used to replace a standard tab. To exclude a tab, set the <exclude> tag.

Example of an additional tab:

```
<KISSsoft filetype="userinterface">
<module>Z012</module>
<title>My own title</title>
<before>Tolerances</before>
<exclude>Z012_BasicData.kui</exclude>
<element>a</element>
<element>Q</element>

</KISSsoft>
```

An additional tab is always displayed. Set the <permanent> tag to define that the tab can be enabled via the "Calculations" menu.

```
<permanent>false</permanent>
```

Additional tabs always work in the same way as the standard tabs supplied with the system. Insert the <calculation>, <report> and <results> tags to represent the behavior of a special tab. The <calculation> tag executes a COM function. All the functions that are available via the COM interface are also available here. The name of the corresponding template is set for the report and the results (see Results and Reports (on page I-110) section in the manual).

Use the <setup> tag to give additional dialogs a COM function. This function is executed when the dialog starts.

Examples of additional description files can be requested from KISSsoft AG.

7.3 Formatting

7.3.1 Elements

Set the `<element>ElementName</element>` tag to add an element. The elements in the description file appear in the same sequence as they appear in the interface.

The following element types are available:

Value input fields	For entering whole number values or floating values
Selection lists (drop-down lists)	For selecting list entries, database entries, materials, lubricants or load spectra
Checkboxes	For enabling/disabling calculation options
Titles and texts	For structuring the interface

The following attributes can be set for an element:

de, en, fr, it, es, pt, ru, all	Overwrites the element's label. Use this option to create company-specific or regional glossaries. The attribute can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).
nolnput	Set this attribute to true to write-protect the associated element. Use this option to predefine values (see Defining your own default files in the manual) and prevent other users from changing them.
unit	The set unit is then used as the default in the interface.
index	Elements with multiple entries are reduced to a fixed index.

```
<element attribute1="wert" attribu-
te2="wert">Name</element>
```

Lists of available elements can be requested from KISSsoft AG requested.

7.3.2 Columns

Set the <column> tag to add a column. The columns in the description file appear in the same sequence as they appear in the interface. You will not usually need more than two columns.

Example of a two-column layout:

```
<column>

<element>Element1</element>

<element>Element2</element>

</column>

<column>

<element>Element3</element>

<element>Element4</element>

</column>
```

7.3.3 Groups

Set the <group> tag to add a group. The groups in the description file appear in the same sequence as they appear in the interface. Groups can also contain columns. Groups cannot be nested.

Set the <title> tag to define a group's title. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

Example of a Group:

```
<group>

<TITLE>145</TITLE>
```

```
<element>Element1</element>

<element>Element2</element>

</group>
```

7.3.4 Tabs

Dialogs can also contain tabs. Set the <tab> tag to add a tab. The tabs in the description file appear in the same sequence as they appear in the dialog. Each tab includes elements that are arranged in groups or columns. Sub-tabs are not supported in the tabs in a calculation module.

Set the <title> tag to define the title of a tab. The tag can contain either an actual text or the ID (number) of a text from the KISSsoft Glossary (wpoolUi_.txt).

7.3.5 Comments

Comments in a description file are a useful way of explaining how the file is structured. Comments start with //.

```
<title>32</title>        // Basic data
```

7.3.6 Special elements

7.3.6.1 Separator

A (horizontal) separator can be added like this

<line></line>

7.3.6.2 Label

A fixed, static text can be defined as follows

<text column="3" alignment="left">My sample text</text>

This line will place the "My sample text" text in the third column (column="3"). The text can be justified to the left (alignment="left"), to the right (alignment="right") or centered (alignment="center"). If you want to use a text from the KISSsoft Glossary (wpoolUi_.txt), enter its ID (number) directly, for example

<text column="3" alignment="left">63</text>

The "column" and "alignment" attributes are not required entry fields, and can be omitted as required. If you do not define these attributes, the following default values will be used.

Attribute	**Default value**
column	1
alignment	Left

You can also define multiple text values, but if you do, you must also define the corresponding columns, for example

<text column="3|5" alignment="left|right">My sample text|Another text</text>

7.3.6.3 User-defined variables

You can use the <var> element to define your own variables. The general definition of the <var> element is

<var type="double" default="15.2" label="My var label" formula="β" unit="DEGREE" save="true">MyDoubleVar</var>

The properties of the attributes of the <var> element are listed below

Attributes	Meaning	Default value	Mandatory field
type	"double", "integer" or "string" defines the corresponding variable type	-	Yes
default	The default value for this element, if the variable has not yet been defined (e.g. when the file is loaded for the first time)	-	No
label	Variable label	No text	No
formula	Variable formula (according to HTML)	No formula	No
unit	Variable unit (e.g. "MICROMETER", "MILLIMETER", "METER", "NEWTON", "SECOND" etc.)	No unit	No
save	If "true" the variable is saved along with the file.	"false"	No

Every user-defined variable can be used as follows in the report (rpt):

Type	Variable name	Name of the variable in the report
double	my_double_var	UserDefined |0|my_double_var
integer	my_integer_var	UserDefined |1|my_integer_var
string	my_string_var	UserDefined |2|my_string_var

For example, an "integer" variable called "NumTeeth" is defined like this in the corresponding kui file:

<var type="integer" default="29">NumTeeth</var>

and is used like this in the report:
1Number of teeth Own definition %i {UserDefined|1|NumTeeth}

Chapter 8
Results and Reports

8.1 Results of a calculation

KISSsoft displays the results of a calculation in the Results (see page I-76) window. If no results are displayed, an error has occurred during the calculation. In this case, you will be alerted to the error by a system message in a message box. An indicator in the status bar (see page I-87) shows whether the results are consistent, i.e. whether the results match up with the data in the user interface.

8.1.1 Add your own texts in the results window

To enable this, define a new file in the KISSsoft installation folder in "…\ext\.rpt\". This file must then be named using this convention: "Module name + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values then appear at the end of the "Results" window.

The syntax corresponds exactly to the entries for the report templates.

8.2 Calculation reports

Select `Report > Generate` to generate reports about your calculations. In addition, the tool bar and the `F6` function key give you quick, convenient access to this action. The report contents depend on which tab (see page I-89) is currently active. The Length (see page I-118) and Appearance (see page I-118) of standard reports can be influenced by user-defined report templates (see page I-117).

A calculation module can contain further reports which you can access via the `Report` menu.

Reports are usually displayed in the KISSsoft Report Viewer (see page I-85).
Important: The report is not saved when you return from the report viewer to the input window. To make it permanently available, you must save it with a new name!

NOTE

In general, a report should only be created if the calculation is consistent (see page I-92). If this is not the case, you can still generate the report, but the status of the calculation will then be noted in the report.

NOTE

When you generate a report, the system generates an RTF file with the module's label as its file name. The file is saved in the directory defined as the TEMPDIR (see page I-54) in the KISS.ini (see page I-54) file.

8.3 Drawing data

Depending on which calculation module you are using, click `Report > Drawing data` to create a report that can be used to output drawings.

8.4 Report settings

Under `Report > Settings`, you can tailor the automatic generation of reports. All the settings can also be defined globally in the KISS.ini (see page I-56) file.

8.4.1 General

Here you define the scope of the report (see page I-118) and whether warnings from the calculation are to be included in it. Further options are the font size and language, along with the standard format used to save the report.

The report can be viewed in two different modes: "overwrite" or "compare".

If a report is generated, and a previous report is still open, the data will be updated. The cursor in the editor will remain in the same line as it was left. This feature will help you analyze specific values using different inputs.

In the report settings, change the report mode to "compare" if you need to compare two or more reports at a time. This mode can only by set if you are using KISSedit as the editor. You can also synchronize the reports and scroll through them all at the same time.

You can also set these report settings directly in the KISS.ini file.

8.4.2 Page layout

Here you can define the paper size and the page margins used to create reports automatically.

8.4.3 Header and footer

In KISSsoft, reports are usually generated with headers and footers. You can define your own header and footer lines. There are a number of placeholders available for this.

Placeholder	**Explanation**
%logo	Picture file
%date	Dated
%time	Time
%pn	Number of pages
%pc	Number of pages

%t	Tab

The %logo placeholder uses the selected graphics file to integrate a user-defined logo (company label). The date and time are output in accordance with the details specified under `Extras > Settings`.

8.4.4 Start and end block

Reports in KISSsoft are usually generated with a start and an end block. You can define these start and end blocks yourself. The start and end blocks are defined in template files which are stored in the `rpt` directory in the installation folder.

Language	**Start block file**	**End block file**
German	kissd.rpt	kissfd.rpt
English	kisse.rpt	kissfe.rpt
French	kissf.rpt	kissff.rpt
Italian	kissi.rpt	kissfi.rpt
Spanish	kisss.rpt	kissfs.rpt
Russian	kissr.rpt	kissfr.rpt
Portuguese	kissp.rpt	kissfp.rpt

Commands that can be used in these templates and what they mean:

Command	**Explanation**
DATE	Date (set your own output format under "Extras/Settings")
TIME	Time (set your own output format under "Extras/Settings")
PROJECT	Project name
PROJECTDESCRIPTION	Description of the project
FILENAME/DESCRIPTION	File name
FILENAME.EXT	File name with extension (e.g. "Example1.Z12")
FILEPATH	Path with file name (e.g. "C:\Temp\GearPair.Z12")
DESCRIPTION	Description of the file
COMMENT	Comment for the file
CUSTOMER	Customer name as defined in the project

USER	User name (Windows user name)
RELEASE	Version number (e.g. "04-2010")
COMPANY	Company name (as defined in the license file)
NLINES	Number of lines in the report
IMPERIALUNITS	Whether US customary units are specified for IF statements
METRICUNITS	Whether metric units are specified for IF statements
PROJECTUSED	Whether projects are used for IF statements

8.5 Report templates

For each calculation module, KISSsoft provides report templates to define the form and content of the reports. You can use these supplied templates as the basis for generating user-defined templates to produce reports that meet your requirements. However, you must ensure the Formatting (see page I-118) and Storage locations (see page I-117) remain the same.

8.5.1 Storage locations and descriptions

The report templates supplied by KISSsoft are stored in the directory that has been set as RPTDIR (see page I-54) in the KISS.ini (see page I-54) file. If RPTDIR (see page I-54) was not defined in KISS.ini (see page I-54), you will find the templates in the installation folder under rpt. It is essential that user-defined report templates are stored in the RPT subdirectory, in the EXTDIR (see page I-54) directory. This is the only way to prevent your templates from being overwritten if a patch is installed. When the system generates a report, it uses the user-defined template from the EXTDIR directory, if present. Otherwise it uses the template from the RPTDIR to create the report.

The descriptions of the report templates have the structure *MMMMlsz.rpt*, which consists of the following:

MMMM	Module descriptor	e.g. *M040*
l	historical	always = *l*
s	Language of the report	*s = d, e, f, i, s or a*
z	historical	always = *0*
.rpt	File type	

EXAMPLES

Bolt calculation:	
M040LD0.RPT	Bolt calculation, German printout
M040USER.RPT	Default printout via the interface, results in the M040USER.OUT file
Cylindrical gear calculation:	
Z012LD0.RPT	Cylindrical gear pair, German printout
Z012USER.RPT	Default printout via the interface, results in the Z012USER.OUT file
Z10GEAR1.RPT	Output via interface, contains only data
	for gear 1, results in file Z10GEAR1.OUT

Z10GEAR2.RPT	Output via interface, contains only data
	for gear 2, results in file Z10GEAR2.OUT
Z011LD0.RPT	Single gear, German printout
Z013LD0.RPT	Rack, German printout
Z014LD0.RPT	Planetary gear, German printout
Z015LD0.RPT	3 gears, German printout
Z016LD0.RPT	4 gears, German printout
Spring calculation	
F10SPRING.RPT	Default printout for drawing data results in the F10SPRING.OUT file
English printout:	
M040LE0.RPT	Bolt calculation, English printout
American printout:	
M040LA0.RPT	Bolt calculation, American printout

8.5.2 Scope of a report

The scope, or length, of a report can be preset on a scale of 1 to 9 in the `Report > Settings` menu. 9 will produce a complete report, and 1 will produce a short report. In the report template, you see a number between 1 and 9 at the beginning of every row. This number acts together with the setting described above to determine whether or not the row is to be read.

Example: If you entered 5 (medium) as the report length, all the lines in the report template that start with 1, 2, 3, 4 or 5 are read. Rows with 6, 7, 8 and 9 will be not read.

8.5.3 Formatting

Both the report template and the report created from this are text files that are created with the Microsoft Windows font. You should always edit text in MS Windows, otherwise accented characters such as ä, ö, ü, as well as some special characters, may be represented incorrectly.

The following statements and key words are defined in the report format:

- Texts that are to be output
- Comments that are not to be output

- Descriptions and formatting of calculation variables
- Limited branchings (*IF ELSE END*)
- Repetitions (*FOR* loops)

8.5.3.1 Text formatting features

In general, reports in KISSsoft are created in RTF format. RTF can handle the following text formats:

Description	Start	End
Underline	<UL>	</UL>
Cross out	<STRIKE>	</STRIKE>
Grease	<BF>	</BF>
Italic	<IT>	</IT>
Superscript	<SUPER>	</SUPER>
Subscript	_	
Font size	<FONTSIZE=xx>	
Enlarge font size	<INCFONTSIZE>	</INCFONTSIZE>
Reduce font size	<DECFONTSIZE>	</DECFONTSIZE>
Page break	<NEWPAGE>	
Line break	 	
Text color red	<RED>	<BLACK>
Text color green	<GREEN>	<BLACK>
Text color blue	<BLUE>	<BLACK>
Blank space	<SPACE>	
Insert figure	<IMAGE=name,WIDTH=xx,HEIGHT=yy>	
Insert image	<INCLUDEGRAPHIC=name,WIDTH=xx,HEIGHT=yy>	
Adding a report template	<EXECUTE=name.rpt>	

8.5.3.2 Comments

Comment lines begin with //. Comments are ignored when a report is created

EXAMPLE

```
// I changed the report template here on 12.13.95, hm
External diameter mm : %10.2f {sheave[0].da}
```

In this case, only the second line will be output.

8.5.3.3 Calculation variables

You cannot define your own variables (apart from the number variables used for FOR loops (see section "FOR loop" on page I-125), which you (as the user) specify, and which can output a value.

Placeholder

Use placeholders to specify the file type and formatting for a variable:

- *%i* stands for a whole number
- *%f* stands for a floating point number
- *%*v_1*.*v_2*f* stands for a formatted floating point number with v_1 places in total (including sign operator and decimal point) and v_2 decimal places
- *%s* stands for a left-justified character string (text)
- *%ns* stands for a right-justified character string in an *n*-character-long field (*n* is a whole number).

The data types must match the definition in the program. The value is returned in exactly the place where the placeholder is positioned. The syntax of the formatting corresponds to the C/*C++* standard.

EXAMPLES

- *%10.2f* returns a right-justified 10-digit floating point number, with 2 decimal places.
- *%i* returns an unformatted whole number exactly in this location.
- *%30s* stands for a right-justified character string in a field that is 30 characters long (if the number 30 is omitted, the characters will be left justified).

COUNTER-EXAMPLES

- *%8.2i* is an invalid formatting because a whole number has no decimal places.
- *%10f2* outputs a right-justified 10-digit floating point number. However, the 2 decimal places are ignored and output as text 2. The default setting is to output floating point numbers to 6 decimal places.

Variables

The variable to be displayed must stand after the placeholder in the same row. The variable is identified by being enclosed in curly brackets. If these brackets are left out, the variable name will appear as normal text.

Important: It is essential that the number of placeholders exactly matches the number of pairs of brackets {}.

EXAMPLE

%f {sheave[0].d} returns the value of the variable *sheave[0].d* in the location *%f* as a floating point number with 6 decimal places.

Basic calculation types - output of changed variables

You can output changed variables in the report. They can be multiplied or divided with a coefficient. You can also add or subtract a number. This functionality is also available in the arguments used in the *IF* or *FOR* statements (see below).

Value of the variable multiplied	*%3.2f*	*{Var*2.0}*
Value of the variable divided	*%3.2f*	*{Var/2.0}*
Value of the variable added	*%3.2f*	*{Var+1.0}*
Value of the variable subtracted	*%3.2f*	*{Var-2}*

The two Degree and Gear functions are also available for converting variables to degrees or radians:

Angle %3.2f {grad(angle)}

Variables can also be directly linked with each other, e.g. in the form *{sheave[0].d-sheave[1].d}*. More than two numbers can be linked. Numbers that have sign operators must be enclosed in brackets, for example *{ZR[0].NL*(1e-6)}*.

The available functions are listed in Table 8.2.

Function	**Meaning**
sin(angle)	sine of angle in the radian measure
cos(angle)	cosine of angle in the radian measure
tan(angle)	tangent of angle in the radian measure
asin(val)	arcsine of val, returns radian measure
acos(val)	arccosine of val, returns radian measure
atan(val)	arctangent of val, returns radian measure

abs(val)	|val|
exp(val)	e^{val}
log(val)	Return value x in e^x = val
log10(val)	Return value x in 10^x = val
sqr(val)	Return value val^2
sqrt(val)	Return value $\sqrt{val}$
int(val)	Whole number of val
pow(x;y)	Return value x^y
sgn(val)	Return value $\begin{cases} 1 & wenn\ val > 0 \\ 0 & wenn\ val = 0 \\ -1 & wenn\ val < 0 \end{cases}$
sgn2(val)	Return value $\begin{cases} 1 & wenn\ val \geq 0 \\ 0 & wenn\ val < 0 \end{cases}$
grad(angle)	Converting from the radian measure to degrees
rad(angle)	Converting from degrees to radian measure
mm_in(val)	Return value val/25.4
celsius_f(val)	Return value $\frac{9}{5}$val + 32
min(v_1; ...; v_5)	The return value is the minimum of v_1,...,v_5
max(v_1; ...; v_5)	The return value is the maximum of v_1,...,v_5
and(v_1; v_2)	binary and function
or(v_1; v_2)	binary or function
xor(v_1; v_2)	binary exclusive or function
AND(v_1; ...; v_5)	logical and function
OR(v_1; ..., v_5)	logical or function
NOT(val)	Return value $\begin{cases} 0 & wenn\ val \neq 0 \\ 1 & wenn\ val = 0 \end{cases}$
LESS(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 < v_2 \\ 0 & wenn\ v_1 \geq v_2 \end{cases}$
EQUAL(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 = v_2 \\ 0 & wenn\ v_1 \neq v_2 \end{cases}$
GREATER(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 > v_2 \\ 0 & wenn\ v_1 \leq v_2 \end{cases}$
ROUND(x;n)	Rounds off x to n places

strlen(str)	Length of character string
strcmp(str1;str2)	Compare character string Return value: 1 if str1 = str2 0 otherwise

Table 8.2: Functions available for calculations in the report

8.5.3.4 Condition query IF ELSE END

The condition query or branching enables you to only output certain values and texts if a particular condition has been fulfilled. The following conditions are supported:

Combination of characters	Meaning
==	equal
>=	greater than or equal
<=	less than or equal
! =	unequal
<	smaller
>	larger

This condition is entered as follows:

IF (condition) {Var}
  Case 1
ELSE
  Case 2
END;

EXAMPLE

IF (%i==0) {Zst.kXmnFlag}
  Addendum modified no
ELSE
  Addendum modified yes
END;

If the *Zst.kXmnFlag* variable is equal to 0, then output the first text, otherwise output the second text. There can be any number of rows between *IF, ELSE* and *END*. For each branching opened with *IF* you must use *END*; to close it again (do not forget the semicolon after *END*). The key word *ELSE* is optional. It reverses the condition. Branchings can be nested within each other up to a depth of 9.

EXAMPLE OF A SIMPLE BRANCHING

IF (%i==1) {ZP[0].Fuss.ZFFmeth}
Calculation of tooth form coefficients according to method: B
END;

If the variable *ZP[0].Fuss.ZFFmeth* is equal to 1, then output the first text, otherwise it is not output.

EXAMPLE OF ENCAPSULATED BRANCHINGS

IF (%f<=2.7) {z092k.vp}				
	Lubrication regularly by hand			(Text1)
ELSE				
	IF (%f<12) {z092k.vp}			
		Lubrication with drop dispenser (2 to 6 drops per minute)		(Text 2)
	ELSE			
		IF (%f<34) {z092k.vp}		
			Lubrication with oil bath lubrication	(Text 3)
		ELSE		
			Lubrication with circulation system lubrication	(Text4)
		END;		
	END;			
END;				

If variable *z092k.vp* is smaller than or equal to 2.7, text 1 is output. Otherwise you are prompted to confirm that *z092k.vp* is smaller than 12. If yes, text 2 is output. Otherwise you are prompted to confirm that *z092k.vp* is smaller than 34. If yes, text 3 is output, otherwise text 4 is output.

8.5.3.5 FOR loop

In KISSsoft you can also use *FOR* loops in the report generator. Within a FOR loop a numerical variable will be incremented (or decremented). You can use constructs that are nested down to 10 levels.

This loop is specified as follows:

FOR varname=%i TO %i BY %i DO {startwert}{endwert} {schritt}
// access variable with #varname or $varname

...
END FOR;

- Instead of *%i* or *%f* you can also use fixed figures (static *FOR* loop):

  FOR varname=0 TO 10 BY 1 DO
  ...
  END FOR;

- or a mixture:

  FOR varname=5 TO %i BY -1 DO {endwert}
  ...
  END FOR;

- Each *FOR* loop must end with the statement *END FOR;* (including semicolon). Each defined numerical variable (*varname*) within the loop can be addressed with the statement *#varname*.
- The increment can also be selected as a negative value (for example -1). However, it must never be 0. You must always specify the intervals.
- The *#varname* statement can be used for defining a variable. For example:
- Number of teeth: *%3.2f {ZR[#varname].z}*
- The *$varname* statement can be used for outputting the variable value as a letter. The value 0 corresponds to A, 1 corresponds to B etc. For example:

  FOR quer=0 TO 3 BY 1 DO
  Cross section *$quer-$quer : %8.2f {Qu[#quer].sStatisch}*
  E*ND FOR;*

EXAMPLE OF A SIMPLE LOOP

FOR i=0 TO 10 BY 1 DO
Run number #i $i
END FOR;

Results in the following output:

Run number 0 A
Run number 1 B
Run number 2 C
Run number 3 D
Run number 4 E
Run number 5 F
Run number 6 G

Run number 7 H
Run number 8 I
Run number 9 j
Run number 10 K

The numerical variable can be used anywhere within the loop, even for arrays.

Chapter 9
Database Tool and External Tables

In addition to the unique data you input for calculations, you may also encounter recurring data such as a material's characteristics. KISSsoft will store these characteristics in databases. You view and change them with the `database tool`, whose use will be explained in the following sections. Tables form the elements of the databases and are contained in your program package as editable ASCII files. The External tables (on page I-136) section deals with the setting up and handling of external tables (also called "look-up tables).

In KISSsoft there are four databases:

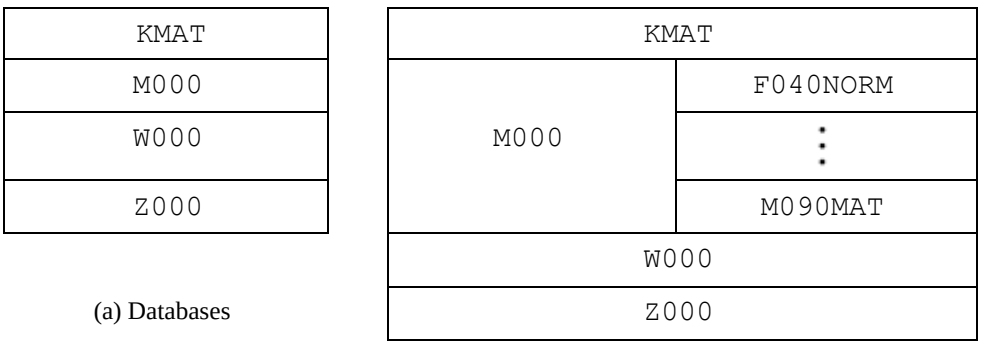
`KMAT` - Materials

`M000` - Shaft-Hub Connection and Bolts

`W000` - Shafts and Bearings

`Z000` - Gears

In Figure 9.3 you can see an example of the `M000` database which shows how data is organized in KISSsoft. As shown there, the `F040NORM` and `M090MAT` tables belong to the group of shaft-hub connections.



(a) Databases

(b) tables

Figure 9.4: How data is organized in KISSsoft

Up to now, the following tables have been created in the databases: Center distance tolerances, Reference profiles, Bore standard, Thread type bolt, Production process for hypoid bevel gears, Manufacturing process for bevel gears, V-belt standard, Spline standard, Chain type DIN 8154, Chain type DIN 8187, Chain type DIN

8188, Glue materials, Load spectra, Soldering materials, Key standard, Polygon standard, Woodruff key standard, Lubricants, Bolt type, Washer standard, Multi-spline standard, Rolling bearing, Materials for glued and soldered joint, Material, Tooth thickness tolerances, Toothed belt standard.

9.1 Viewing database entries

Open the database in the `Extras Database tool` menu option, as shown in (Figure 9.5, ①). A dialog window appears with the question whether you want to open the database with write authorization (②). If you press `Yes`, you can edit the database entries, otherwise they are write protected. If you choose `No`, the actual database tool window (③) will start in read-only mode. There, you can select a table from a list that is assigned to a particular database. The row of a table contains the values that set the parameters for the database entry. The columns contain the parameters for the database entries, i.e. values for the yield point of different materials. This section (see section "Managing database entries" on page I-133) describes how to edit database entries. You can also display table entries by selecting a row in the database tool window and then confirming this by clicking `Display` (④). The `Display entry` window opens with a structured display of the value amount from a table row (⑤).

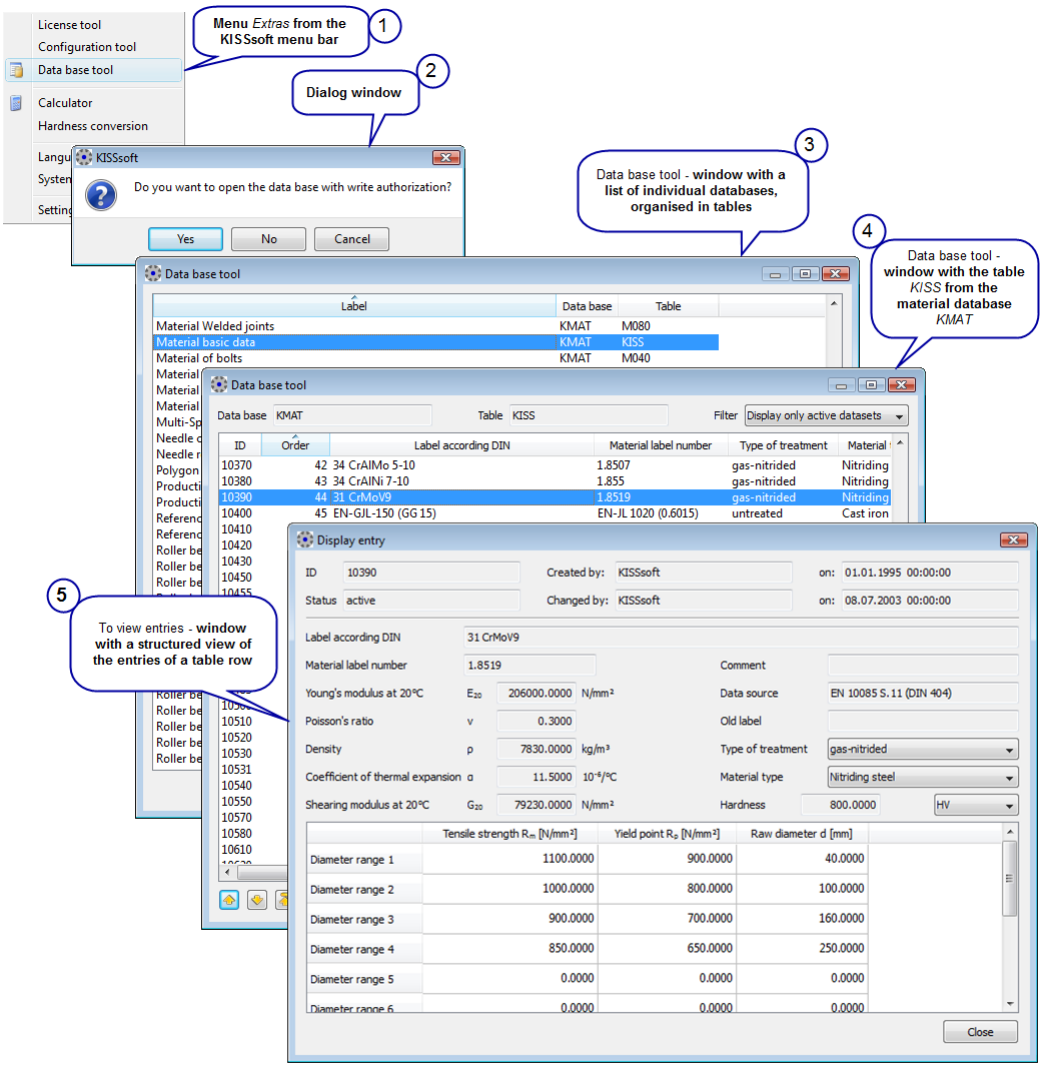


Figure 9.5: Accessing database entries

NOTE:

With the KISSsoft database tool you can change the databases and expand them with your own entries. The data stored in the databases are in a sense "sensitive", so that incorrectly entered values can have consequences that are initially imperceptible, yet eventually far-reaching and serious. For this reason, when you open the database you are asked whether the access should have write authorization. If you answer this question in the negative, you can view the data in the tables but not change it.

If you want to make absolutely sure that the databases remain unchanged, you can write protect their corresponding files (*.kdb). Any attempt to open a table with write authorization results in an error message and the table will normally be opened in write protected mode. To change the write protection attribute of a file, right-click on the file in Windows® Explorer, and then click on `Properties`. Click in the `Properties` dialog field, on the `General` tab, and then click the `Write Protected` selection box. If you want to make changes to a write protected file, you must first deselect the selection box `Write Protected` or else save the file with a different name.

9.2 Managing database entries

If you want to change one of your own entries in a table in the database, you must work in write authorization mode. To do this, go to the (→ sh. Figure 9.5 on page I-130) dialog window and click on `Yes`. In the list that you see next, (③) select the required table by double-clicking on the appropriate row or single-clicking on the `Edit`button at the bottom right of the window, after you have selected the row. The database tool window now shows a list of the table entries (④) and a row of new buttons is displayed on the bottom left in the window:

 Moves the selected item up one row

 Moves the selected item down one row

 Moves the selected item(s) to the start of the list

 Moves the selected item(s) to the end of the list

 Adds a new item to the list

 Moves the selected item into the list of hidden data records

With the `Filter` drop-down menu on the top right of the window you can select between displaying active data records, hidden data records, or both. Active data records can be used within the calculation modules, hidden ones cannot.

9.2.1 Generating a database entry

If you click on the  button without having selected a row, the `Display entry` window (⑤) opens and the input fields in it are empty. Only the `Name` field contains the entry `_NEW`, which normally identifies the new table entry. After you have transferred the necessary data, confirm your entries by clicking on `OK` and then `Save`in the database tool window. The new entry is assigned an identification number (`ID`) ≥ 20000 and is then transferred into the list of active data records. Use the `Edit` button to change entries with an ID of ≥ 20000.

If you click on the  button after having selected a row, the `Display entry`window opens and contains predefined values in the input fields according to the table entry. The suffix `_NEW` will automatically be attached to the name, in order to differentiate it from the original data record. In all remaining steps, you then proceed as described above.

Example: Generating a database entry

Let's assume you want to add a new spring material to the `KMAT.F000` table. In accordance with the described procedure, you would select the `F000` table from the `KMAT` database, click on the  button in it, to add a new entry/new row to the table, and then transfer the new data into the input fields in the `Display entry`

window. However, as only a few parameters can be freely selected there, the next question is, where can the other values such as the yield point and Young's modulus be changed? The reply is: In the base material input fields, i.e. in table KMAT.KISS. You must always specify a base material before you can introduce a new spring material. If this is not present, you have to define it in the KMAT.KISS table first of all and finally complete the missing entries in KMAT.F000.

NOTE

All material-specific tables such as KMAT.F000 or KMAT.Z080 - with the exception of KMAT.KLUB - have a checkbox beside the Base material drop-down menu. If you have marked the checkbox, you have the option of selecting an alternative base material in the associated drop-down list-menu. If the checkbox is empty, access to the menu of the base materials is blocked. This option helps to protect against unwanted changes during the assignment of the base material.

9.2.2 Deleting a database entry

Data records in KISSsoft will never be deleted. It is only possible to move entries with an ID ≥ 20000 into the table of hidden data records. Select the appropriate entry with a single mouse click in the window and then click the button. The selected row will be copied into the range that contains the hidden data records and removed from the list of active data records. To access the table of inactive data records, select the Display only hidden data records option in the Filter drop-down list menu in the top right of the database tool window.

9.2.3 Restoring a database entry

In the table of hidden data records, select the appropriate row with a single mouse click and then click on the button. The entry will be copied into the table of active data records and then removed from the range of inactive data records.

9.3 Import and export data from the database tool

The datasets from every table in the database can be exported to a file or imported from a file. These data records can be imported or exported as a single data record or imported from a list of data records.

To import a list of data records, you will need to save it in a file, preferably an Excel spreadsheet with the extension "*.csv". The inputs in the spreadsheet columns should correspond to the database table columns.

The lists saved as text files with the "*.txt" extension can also be interpreted by the software. The list inputs should be separated by a "comma" or a "semicolon". Which separator is used depends on your operating system configuration.

-  Import a data record from a file with the extension "*.kds".
-  Export the selected data record to a file with the extension "*.kds".
-  Import a list of data records from a file with the extension "*.csv".

Important notes:

1. Only "user data records" (ID >= 20000) >can be exported or imported.
2. An existing "user data record" can be overwritten if you are processing individual data records.
3. The names of the columns in the "*.kds" files is case sensitive and must exactly match the names in the database tool. You could export a data record to verify the column names.
4. A new ID will automatically be assigned to every dataset if an entire list is imported or exported.

9.4 External tables

KISSsoft uses external tables, also called look-up tables, to handle larger data volumes. It is the task of external tables to assign one or more output values to one or more input values (see Figure 9.6).



Figure 9.6: Principle of functionality of external tables

The output data that is assigned to the input data are contained in the table.

The external tables are stored in the /<*KISSsoft installation folder*>/`dat` directory. If a new table name is entered into a database, a file with the same name and the file extension `.dat` must also be created manually.

Because tables are located externally, KISSsoft can only determine how many of them there are during program execution. The user directly benefits from the fact that they can generate their own files with data tables in a similar way to the files supplied by KISSsoft. The tables are readable ASCII files and consequently can be edited and expanded by the user. It would, for example, be possible to use an internal standard as an alternative to the ISO base tolerances.

Figure 9.7 shows the three table types used by KISSsoft in one diagram:

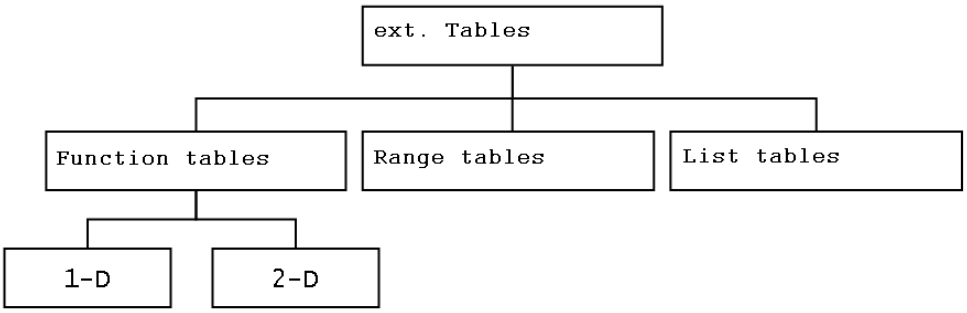


Figure 9.7: Types of external tables

A table always has the following structure, no matter what type it is:

```
:TABLE  <Type> <variable or ID>
  <Table header >
DATA
```

<Data>
END

The :TABLE command marks the external table as an external table. You must use one of the following designations for the <Type> argument:

FUNCTION	Functions tables
RANGE	Range tables
LIST	List tables

NOTE

Blanks in tables can be marked with *, – or blank spaces. Note here that no space characters can be used if they are followed by more values. KISSsoft interprets blank space as value separators.

The structure of the table header and the body data, which is dependent on the type, is described with example applications in the following sections.

9.4.1 Functions tables

Functions tables are tables that expect one or two input values (1D or 2D table) and which return exactly one corresponding value.

EXAMPLE 1D TABLE

The angle coefficient (factor) is determined on the basis of a specified (angle). For example: if the input value angle = 45 supplies an output value of factor = 0.35.

-- table type: Functions table; output variable: factor					
:TABLE FUNCTION factor					
-- INPUT X angle defines the input parameter angle;					
-- interim values will be interpolated linearly					
	INPUT X angle TREAT LINEAR				
-- data: 1. line: input values), 2nd line: output values					
DATA					
	0	30	60	90	. . .
	0.1	0.25	.45	.078	. . .

END

INPUT is a key word, i.e. a word that is reserved by the Tables Interpreter, and is followed by an argument X, which assigns a dimension to the angle input parameter. The key word TREAT, with associated LINEAR argument, specifies that interim values are to be interpolated linearly. The output value factor will determined using the value of the angle variable. The first row of data in the 1D table (between DATA and END) corresponds to the input value angle, and the second row corresponds to the output value. The data in a 1D table is therefore always a (2 × n) matrix, i.e. both rows must contain the same number of values.

EXAMPLE OF A 2D TABLE

The nominal power is defined on the basis of the speed and the sheave diameter. For example: if the input values diameter = 60 and speed = 60 supply an output value power = 8.6.

-- table type: Functions table; output variable: power						
:TABLE FUNCTION power						
-- INPUT X diameter defines the input parameter diameter;						
-- INPUT Y speed defines the input parameter speed;						
-- interim values will be interpolated linearly in both dimensions						
	INPUT X angle TREAT LINEAR					
	INPUT Y Speed TREAT LINEAR					
-- data: (→ sh. Figure (see section "Example: Interference fit calculation" on page I-178))						
DATA						
		50	100	200	300	. . .
	50	4	7	12	25	. . .
	75	12	25	30	35	. . .
	. . .	. . .	. . .	. . .	. . .	. . .
END						

Here, the variable power is defined with the input variables INPUT X and INPUT Y. Interim values running down the columns (Y) should be interpolated linearly. The same applies across the rows (X). The first row in the table corresponds to the values of the INPUT X entry variables. The first column

corresponds to the values of the `INPUT Y` entry variables. The values placed at the points where the entry values intersect are values which correspond to the output variables (see Figure 9.8).

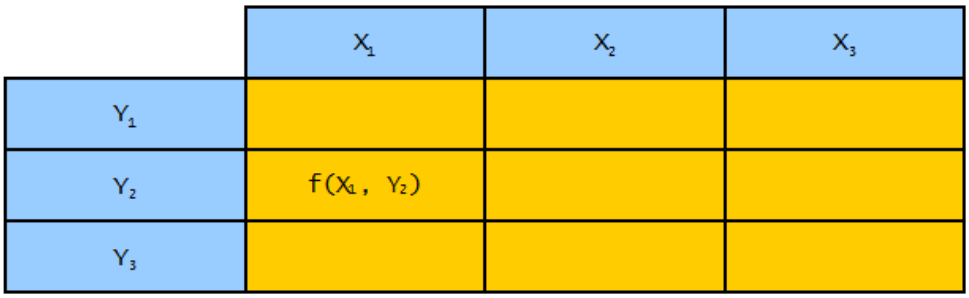


Figure 9.8: Data schema of 2D tables

Please note that in this way it would be possible to define an inverse table. Assuming that, in your `XY` belt catalog, the table that shows the power output contains the speed in the first row, and the diameter in the first column, then there is no need for you to turn your table upside down. Instead, simply change the assignment in the table header (i.e. replace `X` with `Y`).

9.4.2 Range tables

Range tables check whether a given value is moving within a defined range.

EXAMPLE

```
-- table type: range table; Name of the table: 'A'
:TABLE RANGE 'A'r
-- INPUT X drehzahl defines the drehzahl (speed) input parameter;
-- interim values will be interpolated logarithmically.
-- INPUT Y power defines the input parameter power
    INPUT X drehzahl TREAT LOG
    INPUT Y power
-- data: 1. line: INPUT X, 2nd line: INPUT Y upper limit
 -- 3rd line: INPUT Y lower limit

DATA
```

		200	300	500	1000	4000
	LOWER	1.5	2.0	3.0	10	20
	UPPER	10	15	20	15	40
END						

The two input variables are `drehzahl` (speed) and `leistung` (power). The output value represents the decision on whether the power in dependency with the speed is moving within a defined range and does not have to be declared. Interim values of the speed will be interpolated logarithmically. The first row of the body data corresponds to values of the `drehzahl` (speed) variable. The other rows correspond to values of the `leistung` (power) variable with `LOWER` as the lower, and `UPPER` as the upper, limit. The input value of `leistung` (power) will be compared with these limits and a report sent to the program stating whether the `leistung` is located below, within, or above, the given range `A`.

9.4.3 List tables

In list tables containing at least one input value, several output values will be defined. If more than one input value is entered, the sequence of the input values is important. The reading direction goes from left to right and the first input value defines the range of the next input value, which in turn defines that of the next one, etc. up to the last. All input values apart from the last one must correspond with the entries in the body data (`TREAT DIRECT`, list of used key words (see page I-142).

EXAMPLE 1

The following three input values are assumed:
`g.d = 2.0; g.P = 0.8; s.l = 6`
In accordance with the code shown below, the output values would then be:
`s.l = 7; s.k = 3; s.k = 4.5.`

-- table type: List table; output variable `s.norm`	
`:TABLE LIST s.norm`	
-- INPUT g.d defines the input parameter `g.d`;	
-- INPUT g.P defines the input parameter `g.P`;	
	INPUT g.d
	INPUT g.P
-- IN_OUT s.l defines s.l as phase variable	
-- TREAT NEXT_BIGGER specified how interim values are handled	

	IN_OUT s.l	TREAT NEXT_BIGGER
-- OUTPUT s.k, s.dk declares s.k and s.dk as output variables		
	OUTPUT s.k,s.dk	
-- data: A (N × Nin) matrix		

DATA					
	2.0	0.4	0		
	2.0	0.8	5	3	4.5
	2.0	0.8	7	3	4.5 - relevant data row
	2.0	0.8	10	3	4.8
END					

In contrast to functions tables, `s.norm` in the first row of the code specifies the name of the external table, and not the output variable. `IN_OUT s.l` declares a variable `s.l`, which is used both as an input and output variable (phase variable). `TREAT` functions again as a key word for processing the interim values: `NEXT_BIGGER` shows that input values are to be evaluated it they are not present in the corresponding column of the body data. In the example, the input value `s.l = 6` lies between the values 5 and 7 and, in accordance with `NEXT_BIGGER`, will be promoted to the next bigger value. `OUTPUT s.k, s.dk` declares not only `s.l.` but also the output variables `s.k` and `s.dk`. The number of the columns of the body data must at least correspond to the number of input variables and at most correspond to the number of input variables + output variables, in this case: $3 < N_{in} > 5$.

EXAMPLE 2

To determine the diverse measurements of a bolt, two input values will be used: the bolt type, here represented by the `typ` variable and the length of the bolt, given by `l`.

:TABLE LIST schraube.geometrie		
	INPUT type	
	INPUT L	TREAT NEXT_SMALLER

	OUTPUT M, dw, (s), e, bez, vorrat

DATA								
	. . .							
	12x2.5	20	12	14.57	23.78	5.75	ID 1	1
	12x2.5	25	12	15.78	24.88	5.75	ID 2	1
	. . .							
END								

This table is called `bolts.geometry`. The sequence in the table header defines that within the columns, the first column therefore corresponds to the `typ` variable, the second to the `l` variable, etc. The `typ` and `l` variables will be used as inputs, where the value for the `typ` variable must be present in the list. If an interim value is given for the `l` variable, the row with the next smaller value will be interpreted as the result. Blanks are not permitted, i.e. in this type table values must always be present. It may happen that individual variables are shown in brackets in the output definition. This has the effect that the corresponding column is ignored, i.e. this variable will not be specified.

NOTE

Commented-out output definitions cannot be changed by the user.

9.4.4 List of key words used

`--`	Everything in a row coming after this comment character will be ignored by the Interpreter.
`DATA`	Below this is located the data matrix.
`END`	Ends the input area of the external table.
`INPUT [<dim>] <var>`	Input variable, with definition of the dimension if required.
`IN_OUT <var 1>[, <var 2>, ...]`	List tables: Phase variables
`LOWER`	Range tables: Lower limiting value.
`OUTPUT <var 1>[, <var 2>, ...]`	Output value(s)

:TABLE <*Type*>	Defines the type of the external table.
TREAT DIRECT	Interim values: none permitted. Input values in the corresponding column/row must match up with those of the body data.
TREAT NEXT_SMALLER	Interim values: The next smallest value is assigned.
TREAT NEXT_BIGGER	Interim values: The next highest value is assigned.
TREAT LINEAR	Interim values: Linear interpolation.
TREAT LOG	Interim values: Logarithmic interpolation.
UPPER	Range tables: Upper limiting value.

9.5 Description of database tables

The individual database tables have very different structures. The next section describes these database tables and their specific fields.

The **Label** field is displayed in every table, and is only described here. You must enter a unique name for the data record in this field. This name is then used to select the data records in the program.

Note: Fields in which file names are to be entered have an auto-fill function. To perform this, the software searches in the ..\dat and ..\ext\dat folders, and also in the current project directory.

9.5.1 Center distance tolerances

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). The tables used for center distance tolerances begin with K10-???.dat. The center distance tolerances specified in ISO 286 are imported directly from the program code and not from a file.

9.5.2 Machining allowance for cylindrical gear

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). The tables for the cylindrical gear machining allowance begin with ZADDT-???.dat.

9.5.3 Reference profiles

You input reference profile data directly in the database. However, each individual value depends on the other.

- **Description in accordance with ISO: the standard on which this is based**
- **Comment: Text field for your own use**
- **Data source: Text field for your own use**
- **Definable reference profile data: Dedendum coefficient h^*_{fP}, root radius coefficient ρ^*_{fP}, addendum coefficient h^*_{aP}, tip radius factor ρ^*_{aP}, topping, protuberance height coefficient h^*_{prP}, protuberance angle α_{prP}, tip form height coefficient h^*_{FaP}, ramp angle α_{KP}**

9.5.4 Compression springs standard

You can store data from geometry standards for compression springs.

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). The tables for compression spring standards begin with f010-??.dat.
- **Tolerance:** tolerance data for the geometry standard

9.5.5 Selection of hobbing cutters

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). The table for cutter data according to DIN 3972 is called Z000-BP.dat.

9.5.6 Basic material, glued and soldered joints

- **Tensile strength R_m:** [N/mm^2] Data about the tensile strength of the material is required to calculate glued and soldered joints.

9.5.7 Manufacturing process for bevel and hypoid gears

These values are only necessary for calculations using the Klingelnberg method. They correspond to tables for machine types that use the Klingelnberg in-house standard.

- Values that must be defined: **Machine type, Cutter radius cutter tip r_0** [mm]**, No. of cutter blade groups z_0, Maximum machining distance MD_{max}** [mm]**, Minimum normal module $m_{n,min}$** [mm]**, Maximum normal module $m_{n,max}$** [mm]

9.5.8 V-belt Standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for the V-belt standard begin with Z090-???.dat.
- **Calculation method:**
  - 1) Narrow V-belts (Fenner)
  - 2) Narrow V-belts/Force belts
  - 3) Conti belts

- More definitions: **Maximum belt speed v_{max}** [m/s]**, Elasticity E** [N]**, Weight per length q** [kg/m]**, Coefficient of friction** μ_r

9.5.9 Spline Standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for spline standard norms begin with M02C-???.dat.
- **Calculation method:** the appropriate calculation method is selected for each spline.

9.5.10 Chain profiles ISO606

- Values to be defined for this table: **Type, Pitch p** [mm]**, Number of strands** n_s**, Maximum diameter of roller** d_1 [mm]**, Maximum bearing pin body diameter** d_2 [mm]**, Minimum width between inner plates** b_1 [mm]**, Maximum width over inner link** b_2 [mm]**, Total width** b_{tot} [mm]**, Maximum inner plates depth** h_2 [mm]**, Ratio** t_H/t_S

9.5.11 Adhesives

- **Comment:** Text field for your own use
- Definable values: **Minimum** and **Maximum shear strength** $\tau_{B,min}$ **,** $\tau_{B,max}$ **[N/mm**^{2}].

9.5.12 Modifications

The different modifications applied to gears are defined as database classes. If a data record is hidden in the database, it will no longer appear in the modifications selection list.

Although you can add new data records to the database, these will not be visible in the calculation module.

9.5.13 Load spectra

All inputs (frequency, power, speed) must be defined in factors. The power and speed are given as factors of the nominal power. In the calculations, the factor for torque (load factor/speed factor) is used for forces and moments. You can either import load spectra from a file or enter them directly. If you input this data directly, the number of load cases is defined by the number of lines you enter.

- **Input:** Specify whether the factors are for power or torque. This also applies if the load spectrum is imported from a file.
- **Link with file:** This option is displayed if the load spectrum is set to "Own input". If the option is displayed, load spectrum values can be imported from a selected file. The import can be performed in two ways: If the flag is not set, the imported load spectrum values can be modified. If the flag is set, the load spectrum values are automatically overwritten by the values in the selected file, and cannot be modified.
- **Own input of load spectra:** You can input the load spectrum directly, or import it from a file.
- **File name:** Click the  button to select a file from the directories. The file containing the load spectrum must be a text file (*.dat). You will find a sample load spectrum file (called "Example_DutyCycle.dat") in the "dat" directory. You should store load spectra you define yourself in the "EXT/dat" directory to ensure they are always available even after a version upgrade.

```
0.1  1    1
0.1  0.9  1
0.1  0.8  1
0.1  0.7  1
0.1  0.6  1
0.1  0.5  1
0.1  0.4  1
0.1  0.3  1
0.1  0.2  1
0.1  0.1  1
```

Example of a file used to input a load spectrum

- **Frequency:** H_0 ... H_{19}, the sum of these frequencies must be 1.
- **Load factor (torque factor):** P_0 ... $_{19}$ $0<P_n<,\infty$.
- **Speed factor:** N_0 ... N_{19}, $0<N_n<\infty$.

Load bins with negative power or speed are interpreted as a change from driving to driven. This does not affect the root and flank strengths, however it does change the scuffing safety and safety against micropitting.

9.5.14 Solders

- Definable values: **Minimum** and **Maximum shear strength** $\tau_{B,min}$, $\tau_{B,max}$ [N/mm^2].

9.5.15 Surface roughness of shafts and shaft-hub connections

You can also input your own data for shaft-hub connections in the "Own input" field. The Rz values for the shafts are taken from the list.

- **Comment:** Text field for your own use
- Definable values: **Average roughness height R_z** [µm] and **Roughness average value R_a** [µm].

9.5.16 Key standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for key standards begin with M02A-???.dat.

9.5.17 Polygon standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for polygon standards begin with M02D-???.dat.

9.5.18 Woodruff Key standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for polygon standards begin with M02E-???.dat.

9.5.19 Bolts/pins

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for bolt/pin standards begin with M03A-???.dat.

9.5.20 Lubricants

- **Comment, description as specified in ISO, data source:** Text fields for your own use
- **Additive for rolling bearings:**

- **without additives:** lubricants without additives, or with those additives whose effectiveness in rolling bearings has not been tested.
- **with additives:** lubricants, whose effectiveness has been tested in rolling bearings

- **Oil/ Grease:** specify whether the lubricant is an oil or a grease.
- **Kinematic viscosity at 40°C and at 100°C** ν_{40},ν_{100} **:** [mm^2/s]
- **Lubricant base:** Selection options:
  - Mineral oil
  - Synthetic oil based on polyglycol
  - Synthetic oil based on polyether
  - Synthetic oil based on polyalphaolefin
  - Synthetic oil based on ester

  Polyalphaolefin: similar to mineral oil, easily mixable with mineral oil, some approved for use with foodstuffs.

  Esters: some approved for use with foodstuffs, some biodegradable.

- **Test procedure scuffing:** Selection options:
  - No information about scuffing
  - FZG Test A/8.3/90; ISO 14635-1 (normal)
  - FZG Test A/16.6/90
  - FZG Test A/16.6/120
  - FZG Test A/16.6/140
  - FZG Test A10/16.6R/120; ISO 14635-2
  - Input of scuffing temperature
  - FZG test A10/16.6R/90
  - FZG test S-A10/16.6R/90
- **Failure load stage scuffing FZG test:** Input load stage scuffing as specified in the FZG test. These values are required for gear calculations.

  1= weakest stage; 12=best stage

  Good gear lubricants all have load stage scuffing 12.

- **Scuffing temperature** θ_s**:** You can also enter the scuffing temperature for the scuffing test procedure.
- **Micropitting process:** Selection options

- No information available for micropitting load stage
- C-GF/8.3/90 (FZG)

- **Load stage micropitting test:** The best achievable load stage is 10.
- **Density** ρ**:** [kg/dm^3]
- **Cone penetration at 25°C (grease)** P_e**:** [0.1mm] This value is only required to calculate grease lubricated plain bearings.
- **Soap proportion (Grease)** c_s**:** [Vol%] This value is only required to calculate grease lubricated plain bearings.
- **k-coefficient, s-coefficient (compression viscosity) k, s:** Coefficient used to calculate compression viscosity (AGMA 925):

$$\alpha = k \cdot \eta_M^S$$

$$\eta_P = \eta_{atm} \cdot e^{\alpha_p}$$

If you do not know these values, you can input 0 and then the values are taken from the standard (AGMA 925-A03, Table 2).

- **Lower/ Upper limit working temperature** θ_{min}**,** θ_{max} **:** [°C]

9.5.21 Bolts: Tightening factor

- **Calculation method:** Select VDI 2230(1990) and VDI 2230(2015)
- **Minimum tightening factor** α_{Amin}**:** Minimum value, can be defined per tightening technique.
- **Medium tightening factor** α_{Amid}**:** Medium value, can be defined per tightening technique.
- **Maximum tightening factor** α_{Amax}**:** Maximum value, can be defined per tightening technique.

9.5.22 Bolts: Bore

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for bores begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

9.5.23 Bolts: Strength class

- **Comment:** Text field for your own use
- Definable values: **Yield point R_p** [N/mm²], **Tensile strength R_m** [N/mm²], **Shearing strength ratio τ_b/R_m[-]**

NOTE:

The yield point and tensile strength for the lower diameter limit for strength classes 8.8 and SAE J429 Grades 2 and 5 are always displayed in the database. If the diameter is greater than the diameter limit, this is corrected in the program.

The shearing strength ratios were taken from the Table 5.5/2 in VDI 2230 (2015) according to the strength classes. Undefined SAE classes are set to the ratio 0.6.

9.5.24 Bolts: Nuts strength class

- **Comment:** Text field for your own use
- Definable values: **Vickers hardness for standard thread for diameter d <= 16 mm** [HV], **Vickers hardness for standard thread for diameter d > 16 mm** [HV], **Vickers hardness for fine thread for diameter d <= 16 mm** [HV], **Vickers hardness for fine thread for diameter d > 16 mm** [HV]

NOTE:

The minimum value specified in the DIN EN ISO 898-2:2012 standard is assumed in the strength classes. However, as this value depends on the diameter, in some strength classes, both values are defined in the database. In the program, you can then specify which diameter the value is to be used for. The same applies to differentiating between fine or standard threads. The program handles the value that is being used differently, according to the selection in the interface.

9.5.25 Bolts: Coefficients of friction classes

- **Comment:** Text field for your own use
- Definable values: **Minimum coefficient of friction μ_{min}** [-] and **maximum coefficient of friction μ_{max}** [-]

NOTE:

The minimum and maximum coefficients of friction are always displayed in the database for the coefficient of friction classes. The values are used to size the coefficient of friction for bolts.

The predefined values are taken from Table A5 in VDI 2230, Sheet 1 [75].

9.5.26 Bolts: Thread type

- **Name:** Text field for your own use
- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for threads begin with M04-???.dat.
- **Coefficient used to calculate the flank diameter/core diameter**
- **Flank angle α:** [°]

9.5.27 Bolts: Nuts

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for nuts begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

9.5.28 Bolts: Type

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for bolt types begin with M04-???.dat.
- **Name:** Text field for your own use
- **Thread type screw:** selection list to show which thread type this bolt has.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

9.5.29 Bolts: flat washer

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for flat washers begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

9.5.30 Selection of pinion type cutters

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for pinion type cutters begin with Z000-Cutter-?.dat.

9.5.31 Disc spring standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for disc springs begin with F040-?.dat.

9.5.32 Tolerances standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for tolerances begin with K10-???.dat.

The tolerances according to DIN EN ISO 286 have been programmed directly into KISSsoft. For tolerance classes H, h, JS and js, the tolerance has been extended up to the nominal length 10000 mm (according to the standard up to 3150 mm). The values were determined by extrapolation.

9.5.33 Beam profiles

- **Drawing file:** image displayed on screen when a shaft is calculated.
- Values for profiles: **Height h** [mm], **Width b** [mm], **Cross section A** [cm^2], **Moments of inertia of plane area on x/z axis I_x/ I_z** [cm^4], **Moment of inertia of torsion I_t** [cm^4], **Section modulus relating to x /z axis W_x/ W_z** [cm^3], **Moment of resistance in torsion W_t** [cm^3]

9.5.34 Multi-Spline standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for multi-spline profiles begin with M02b-???.dat.

9.5.35 Materials

The materials consist of a database table **Basic data Materials** and the particular table for the modules. The Basic data table lists the general material data. As the materials can then be transferred to the individual module tables, you therefore only need to define the basis data once. Module-specific data is then defined in the module tables.

In module-specific tables, you must always select one **Base material**.

9.5.35.1 Basic data: Materials

- **Label according to DIN, BS, AISI, UNI, AFNOR, JIS, CN, Old label, Material number, Origin of data, Comment:** Text fields for your own use
- **Young's modulus at 20°C E_{20}:** [N/mm^2]
- **Poisson's ratio ν:** [-]
- **Density ρ:** [kg/dm^3]
- **Coefficient of thermal expansion α:** [$10^{-6}/K$]
- **Shearing modulus at 20°C G_{20}:** [N/mm^2]
- **Type of treatment:** In this list, you can select the type of treatment.
- **Material type:** In this list, you can select the material type.
- **Hardness value:** This value is purely for information purposes and has a negligible effect on the calculation.
- **Hardness unit:** You can select this in this list.
- **Core hardness value:** this value is used, for example, for the strength calculation according to AGMA 6101-E08/AGMA 6001-E08. However, for other calculations, the value is purely for information purposes and has only a minor influence on the calculation. For case-hardened case hardening steels, the value for the tensile strength Rm was taken from the DIN EN 10084 standard for a diameter between 16 and 40 mm, and was then calculated with the hardness conversion of the Brinell hardness HBW.
- **Core hardness unit:** You can select this in this list. You can select Brinell hardness or Vickers hardness as input values.
- **Tensile strength R_m:** [N/mm^2] A max. of 10 different diameter ranges can be defined
- **Yield point R_p:** [N/mm^2] A max. of 10 different diameter ranges can be defined
- **Raw diameter d:** [mm] A max. of 10 difference diameter ranges can be defined

9.5.35.2 Material Spring calculation

This table applies to Compression (F010), Tension (F020) and Leg springs (F030):

- **Permissible shear stress:** the database entries refer to external tables (see External tables (on page I-136)). Tables for springs begin with F01-???.dat. In this file, you can view or define the permissible shear stress, the values for the

Goodman diagram and the values for the relaxation diagram. If the curves of the relaxation diagram are only defined with 2 points, you must set the values for tau3 and rel3 to 0 so KISSsoft can recognize them.

- **Comment:** Text field for your own use
- **Minimum** and **Maximum wire diameter d_{min}, d_{max}** [mm]
- **Shearing modulus depending on temperature α_G:** [1/K]
- **Use:** selection list with the cold and thermo-formed variants

9.5.35.3 Material of plain bearings

- **Comment:** Text field for your own use

9.5.35.4 Material of enveloping worm wheels

The table applies to worm wheels (Z080):

- **Comment:** Text field for your own use
- **Material characteristics:** Selection list (such as CuSn bronze/CuAl bronze/GJS40/GC25/PA-12)
- **Mineral oil coefficient W_{MLOel}:** Material/lubrication coefficient for mineral oil
- **Coefficient for polyglycol (DIN)/ (ISO) W_{MLGDIN}/ W_{MLGISO}:** Material/lubrication coefficient for polyglycol
- **Coefficient for polyalphaolefin W_{MLA}:** Material/lubrication coefficient for polyalphaolefin
- **Material coefficient Y_W:** (see DIN 3996, Table 5)
- **Pitting strength σ_{HlimT}:** [N/mm^2] (We recommend you use reduced values as specified in ISO 14521)
- **Shear fatigue strength τ_{FlimT}:** [N/mm^2]
- **Reduced shear fatigue strength $\tau_{FlimTred}$:** [N/mm^2] (If no slight deformation is permitted, you must include reduced strength values in the calculation.)

9.5.35.5 Material Interference fit

- **Comment:** Text field for your own use

9.5.35.6 Material of bolts

The table applies to the bolt module (M040)

- **Comment:** Text field for your own use
- **Permissible pressure p_G:** [N/mm^2] (Data should be entered as specified in VDI 2230)
- **Shear strength τ_B:** [N/mm^2]

9.5.35.7 Material Welded joints

- **Comment:** Text field for your own use

9.5.35.8 Material Disc spring calculation

The table applies to disc springs (F040):

- **Comment:** Text field for your own use
- **Goodman diagram:** the database entries refer to external tables (see External tables (on page I-136)). Tables for the Goodman diagram begin with F04-???.dat.
- **Young's modulus depending on temperature α_E:** [1/K]

9.5.35.9 Material of shaft-hub connection

- **Comment:** Text field for your own use

9.5.35.10 Housing material

- **Comment:** Text field for your own use

9.5.35.11 Material Shaft calculation

The table applies to shafts (w010):

- Values for calculating strength according to Hänchen:
  - **Bending fatigue limit σ_{bW}:** [N/mm^2]
- Values for strength calculation according to DIN 743:

- **Reference diameter d_b** [mm], **Tensile strength R_m** [N/mm^2], **Yield point R_p** [N/mm^2], **Bending fatigue limit σ_{bW}** [N/mm^2], **Tension/Compression fatigue limit σ_{zdW}** [N/mm^2], **Torsion fatigue limit τ_{tW}** [N/mm^2]
- **Experimental data:** the database entries refer to external tables (see External tables (on page I-136)). Tables for the experimental Haigh diagram begin with W01-???.dat.
- **CrNiMo case hardening steel:** When determining the size coefficient K1 for case hardening steel, you need to specify whether the steel is CrNiMo case hardening steel. The materials 20NiCrMo13-4, 17NiCrMo6-4, 17NiCrMoS6-4, 18NiCr5-4, 17CrNi6-6, 18CrNiMo7-6, 14NiCrMo13-4, 22CrMoS3-5 and 15NiCr13 belong to this group. According to DIN 10084, there is no significant drop in the tensile strength of these materials. These materials also belong to this group according to Prof. Linke.

☼ Values for strength calculation according to FKM:

- **Tensile strength for reference diameter $R_{m,N}$** [N/mm^2], **Yield point for reference diameter $R_{e,N}$** [N/mm^2], **Effective reference diameter for deff,N,p $d_{eff,N,p}$** [mm], **Effective reference diameter for Rm,N $d_{eff,N,m}$** [mm], **Constant used to calculate Kd (flow) $a_{d,p}$, Constant used to calculate Kd (fracture) $a_{d,m}$, Tension/compression fatigue limit for reference diameter $\sigma_{W,zd,N}$** [N/mm^2], **Fatigue limit for bending for reference diameter $\sigma_{W,b,N}$** [N/mm^2], **Shear stress fatigue limit for reference diameter $\tau_{W,s,N}$** [N/mm^2], **Torsion fatigue limit for reference diameter $\tau_{W,t,N}$** [N/mm^2]
- **Elongation at break A:** [%]
- **FKM Group:** Selection list showing the material group to which the entry belongs.

9.5.36 Material of gears

☼ **Comment:** Text field for your own use

- **File for hardness curve:** the database entries refer to external tables (see External tables (on page I-136)). Tables for the hardness curve begin with Z22-???.dat. Measured hardness value of the material shown as a graphic in module Z22. Does not influence the calculation. Here is an example of how to create this type of hardness curve in an external table.

```
-- ----------------------------------------------------------------------
-- File   = ~modname: z22-100.dat~
-- Verlauf der Haerte des Werkstoffs/Curve of the material hardness
-- Werkstoff/Material: GGG50 180HB
--
-- (c) KISSsoft AG, CH-8608 Bubikon
--
-- 1.0     14.10.1995  em        Inital Version
--
-- INPUT  nummer      Punktnummer/ number of the point
-- OUTPUT Tiefe, HaerteHV Tiefe(mm), HV-Wert/Depth(mm), HV-Value
-- ----------------------------------------------------------------------

:TABLE LIST haerteHV
  INPUT nummer
  OUTPUT Tiefe, HaerteHV
DATA
1     0.0   890
2     0.1   523
3     0.2   470
4     0.3   295
5     0.4   290
6     0.5   207
7     0.6   207
8     0.7   207
9     1.0   207
10    1.5   207
END
```

Figure 9.08: Example of a hardness curve definition (Z22-100.dat)

- **Woehler line file:** the database entries refer to external tables (see External tables (on page I-136)). Tables for S-N curves (Woehler lines) begin with *.dat.
  You must input a file name for plastics here. The file contains material data - S-N curves (Woehler lines), Young's modulus, etc. - used in the calculation.
  For metallic materials, you can also input a file name here. The file contains the S-N curves (Woehler lines) for bending strength and for Hertzian pressure that are used in the calculation, if the `Calculate with own Woehler line` flag is set.

```
-- Data for significant no. of cycles (edge points for interpolation of Woehler line)
-- Important: This list must contain ALL cycle values of FlankSigH and FootSigF in numerical order!
:TABLE FUNCTION EdgeCycle
      INPUT X number TREAT NEXT_BIGGER
DATA
      1      2      3      4      5      6      7
    1E3    1E5    2E6    3E6    5E7   1E10   1E99
END

-- Data for Hertzian pressure sigH
:TABLE FUNCTION FlankSigH
      INPUT X Cycles TREAT LOG
DATA
      0    1E3    1E5   3E6    5E7   1E10   1E99
   2700   2700   1920   1485   1200   1104   1104
END


-- Data for fatigue strength tooth root sigF
:TABLE FUNCTION FootSigF
      INPUT X Cycles TREAT LOG
DATA
      0    1E3    1E5    2E6    1E10   1E99
   1080   1080    423    230    195    195
END
```

Figure 9.09: Example of a file with S-N curves (Woehler lines) for a metallic material

- **Endurance limit root (ISO, DIN/ AGMA 2101) σ_{Flim}/s_{at}, Endurance limit flank (ISO, DIN AGMA 2101) σ_{Hlim}/s_{ac}:** [N/mm^2] Endurance limit values specified in DIN 3990 or ISO 6336 Part 5.
- **Endurance limit root (AGMA 2001) s_{at},Endurance limit flank s_{ac} (AGMA 2001):** [lbf/in^2] Strength calculation based on AGMA 2001.
- **Mean peak-to-valley roughness root/flank R_{zF}/ R_{zH}:** [µm]
- **Thermal contact coefficient B_M:** [N/mm/s$^{0.5}$/K] This coefficient is needed to calculate the flash factor. You will find more information about this in DIN 3990, Part 4, equations 3.11, 4.17, 4.18, 4.19. For the most commonly used materials, it is 13.795.

9.5.37 Rolling bearing

Rolling bearing tables are sub-divided into 2 different tabs:

- Basic data tab
- Internal geometry tab

9.5.37.1 Rolling bearing basic data

- **Bearing label:** The codes for the bearing series are as specified in DIN 623 Part 1. SKF bearings have a * in front of the bearing designations, if an Explorer bearing (high performance bearing) is involved.
- **Comment:** more detailed description of the bearing (e.g. SKF-Explorer, X-Life, etc.)
- Main dimensions of the bearing: **Inside diameter d** [mm], **External diameter D** [mm], **Bearing width b** [mm], **Corner radius** $\mathbf{r}_{smin}$ [mm]
- **Dynamic load number C:** [kN], **Dynamic radial load rating** $\mathbf{C}_r$: [kN] (if radial and axial values are predefined, e.g. cross roller bearing)
- **Static load number** $\mathbf{C}_0$**:** [kN], **Static radial load rating** $\mathbf{C}_{0r}$: [kN] (if radial and axial values are predefined, e.g. cross roller bearing)
- **Dynamic axial load rating** $\mathbf{C}_a$: [kN] (if present, e.g. cross roller bearing)
- **Static axial load rating** $\mathbf{C}_{0a}$: [kN] (if present, e.g. cross roller bearing)
- **Hybrid bearing:** [Yes/No] Yes: if hybrid bearing (ceramic rolling body). No: if not a hybrid bearing.
- Coefficients **X1, Y1, X2, Y2, e, e0, X01, Y01, X02, Y02**

	Defining individual factors:
X1,Y1,e:	Coefficients in formula *P = X1*Fr + Y1*Fa* for *Fa/Fr <= e*
X2,Y2:	Coefficients in formula *P = X2*Fr + Y2*Fa* for *Fa/Fr > e*
X01,Y01,e0:	Coefficients in formula *P0 = X01*Fr + Y01*Fa* for *Fa/Fr <= e0*
X02,Y02:	Coefficients in formula *P0 = X02*Fr + Y02*Fa* for *Fa/Fr > e0*
X1,Y1,X2,Y2,e:	For some bearings, these values are not imported from the database. Instead they are imported from the files, depending on the axial force.

Ball bearing: dependent on *f0*Fa/C0*

- at bearing clearance C0: Data is imported from file W05-100A.dat (single row bearing) and W05-100B.dat (double row bearing)
- at bearing clearance C3: Data is imported from file W05-101A.dat (single row bearing) and W05-101B.dat (double row bearing)
- at bearing clearance C4: Data is imported from file W05-102A.dat (single row bearing) and W05-102B.dat (double row bearing)
- with no clearance: Data is imported from file W05-104.dat

Angular contact ball bearing: depending on *f0*Fa/C0* and contact angle

- at contact angle 5°: Data is imported from file W05-100A.dat (single row bearing) and W05-100B.dat (double row bearing)
- at contact angle 10°: Data is imported from file W05-101A.dat (single row bearing) and W05-101B.dat (double row bearing)
- at contact angle 15°: Data is imported from file W05-102A.dat (single row bearing) and W05-102B.dat (double row bearing)
- at contact angle 20°< α< 45°: Data is imported from file W05-103A.dat (single row bearing) and W05-103B.dat (double row bearing)

NOTE:

Bearing clearance class C2 is handled in the same way as C0. The factors were taken from the "Die Wälzlagerpraxis" document [39].

- **Speed limit using grease lubrication n_{Gmax}:** [1/min] SKF bearings only have one speed limit specified. No difference is made between grease and oil lubrication.
- **Speed limit using oil lubrication n_{Omax}:** [1/min] SKF bearings only have one speed limit specified. No difference is made between grease and oil lubrication.
- **Weight m:** [kg]
- **Contact angle α_0:** [°] Input the contact angle for shaft bearings, taper bearings, etc.,
  for four-point contact bearings: If you input 0°, this is set to 35°
  for axial spherical roller bearings: If you input 0° this is set to 50°.
- **Permitted axial force F^*_{azul}:** [-] Input the permitted axial force in % of Fr. The permitted axial force is not checked if you input 0.
- **Maximum set angle α:** [min] If you input 0, the angle adjustability (i.e. a comparison of the permitted angular deviation of the shaft with the effective angular deviation in the bearing) is not checked.

- **Thermal reference speed $n_{\theta r}$**: [1/min]
- currently not evaluated in KISSsoft: **Availability** (0=in stock; 1=not in stock), **price** [in local currency]
- **Addition A-E:** You can input additional data for specific types in these fields. (see table: Use of additions A-E.)
- **radial** and **axial spring stiffness c_r ,c_a**: [N/µm]
- **Spring stiffness for bending c_{rot}:** [Nm/°] Input spring stiffness for bending.
- **Coefficient f_0:** Required to determine x and y (for example, for deep groove ball bearings) as these values depend on the coefficient *f0*Fa/C0*.
- **Minimum load P/C:** Minimum load P/C (P: average dynamic equivalent load: C: dynamic load rating) is usually:
  - 0.01 for ball bearings with a cage
  - 0.02 for roller bearings with a cage, 0.04 for pure roller bearings with a cage

  If you input 0 in the database, these values are used automatically in the calculation.

  This entries only apply to radial load. The minimum axial force is calculated directly in the software.

- **Fatigue load limit C_u:** Coefficient for calculating the modified service life

Type	Addition A	Addition B	Addition C	Addition D	Addition E
Angular contact ball bearing (single row)		Displacement a (mm) (*2)			
Angular contact ball bearing (paired)		Displacement a (mm) (*2)			
Cylindrical thrust roller bearing		Coefficient A (*1)		Max. axial force (kN)	
Taper roller bearing (single row)		Width B (mm)	Displacement a (mm) (*2)	Dimension C (mm)	
Taper roller bearing (paired, O)		Dimension T (mm) (*1)		Dimension C (mm) (*1)	
Taper roller bearing (paired, X)		Dimension 2B (mm)		Dimension 2T (mm)	
Barrel-shaped and spherical roller	Calculation coefficient k1 (SKF: CARB,	Calculation coefficient k2 (SKF: CARB,			

bearings	Internal geometry)	Internal geometry)			
Spherical thrust roller bearing	Dimension d1 (mm)	Dimension T2 (mm)	Dimension D1 (mm)	Dimension T1 (mm)	Coefficient A (*1)

Table 9.4: Use of additions A-E

Descriptions given in additional data conform with those in the INA/FAG catalog 2014.

(*1) Values are only used for SKF bearings, as specified in the SKF catalog 2013.

(*2) Values for the mass a for FAG bearings have been provided by the manufacturer up to the center point. In KISSsoft, half the bearing width was then added to this value for the database (this may result in values that vary slightly from those in the bearing catalog). The value at the bearing center was used in this calculation. This corresponds to the data we received from the manufacturer.

9.5.37.2 Rolling bearing Internal geometry

Description of the **Internal geometry** tab:

Inner geometry data is not yet available for every bearing type.

The **Material ID** is present in every table in which you must select a material for the balls. However, this is not yet taken into account.

List of bearings whose internal geometry is taken into account.

You need the details documented below in order to calculate inner geometry.

For rolling bearings, a user-defined roller profile can be supplied using a ".dat" file. The third column is the user-defined roller profiles, given relative to the roller diameters in non-dimensional form. (Acceptable range of values, from 0 to 0.5). The second column is the position on the rollers of the profile defined above, and is given relative to the roller length in a non-dimensional form. The origin is the roller center, so the acceptable range here is from -0.5 to +0.5. The first column is the data index, and is not used in the calculation.

- Deep groove ball bearing (single row), four-point contact bearing: **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

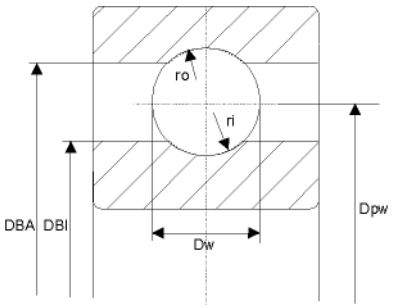


Figure 9.10: Dimensions of the deep groove ball bearing

- Angular contact ball bearing (single row): **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm], **Minimum initial tension v_{min}** [µm], **Maximum initial tension v_{max}** [µm], **Minimum pretension force F_{vmin}** [N], **Maximum pretension force F_{vmax}** [N]

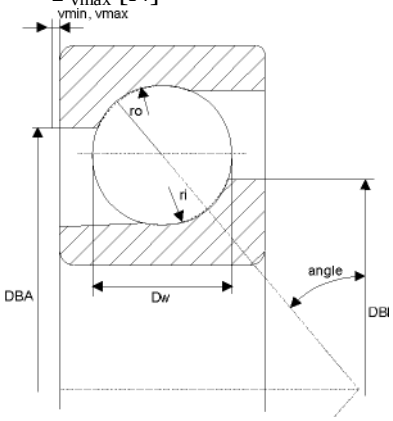


Figure 9.11 Mass of the angular contact ball bearing

- Cylindrical roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating**

bearing v_l [mm], **Axial displacement possibility locating bearing v_f** [mm]

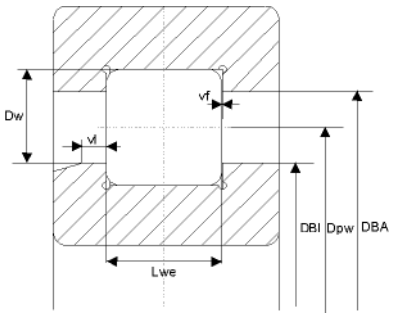


Figure 9.12: Mass of the cylindrical roller bearing

- Cylindrical roller bearing (double row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Row distance a** [mm]

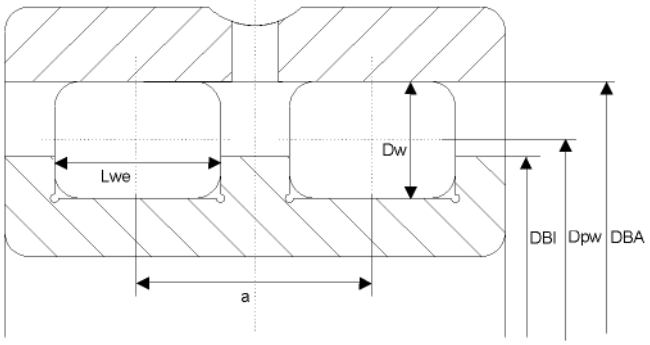


Figure 9.13: Mass of the double row cylindrical roller bearing

- Taper roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

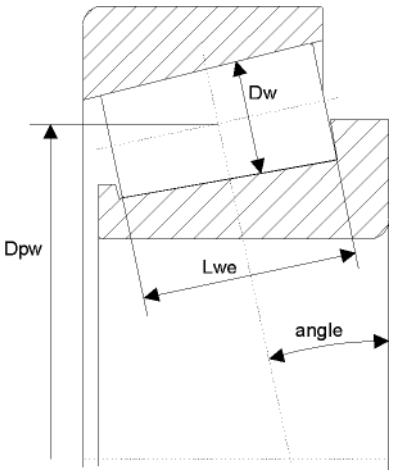


Figure 9.14: Mass of the taper roller bearing

- Double row self-aligning roller bearings: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

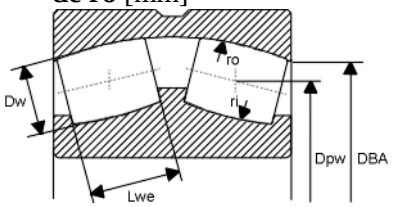


Figure 9.15: Mass of the double row self-aligning roller bearing

- Needle roller bearing, Needle cage: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating bearing v_l** [mm]

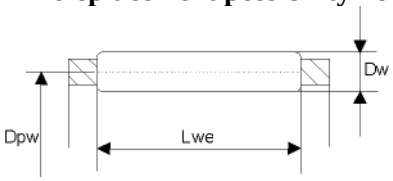


Figure 9.16: Mass of the needle roller bearing/needle cage

- Deep groove thrust ball bearing: **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

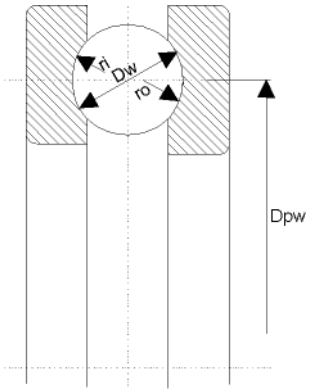


Figure 9.17: Dimensions of the deep groove thrust ball bearing

Cylindrical roller thrust bearing: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

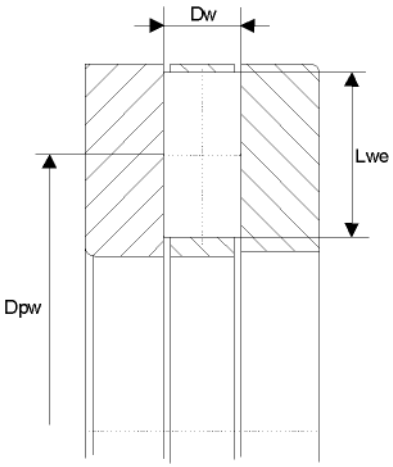


Figure 9.18: Mass of the cylindrical thrust roller bearing

- Axial spherical roller bearings: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Distance L_{WC}** [mm], **Radius of curvature, inside r_i** [mm], **Radius of curvature, roller R_p** [mm], **Radius of curvature, outside r_o** [mm]

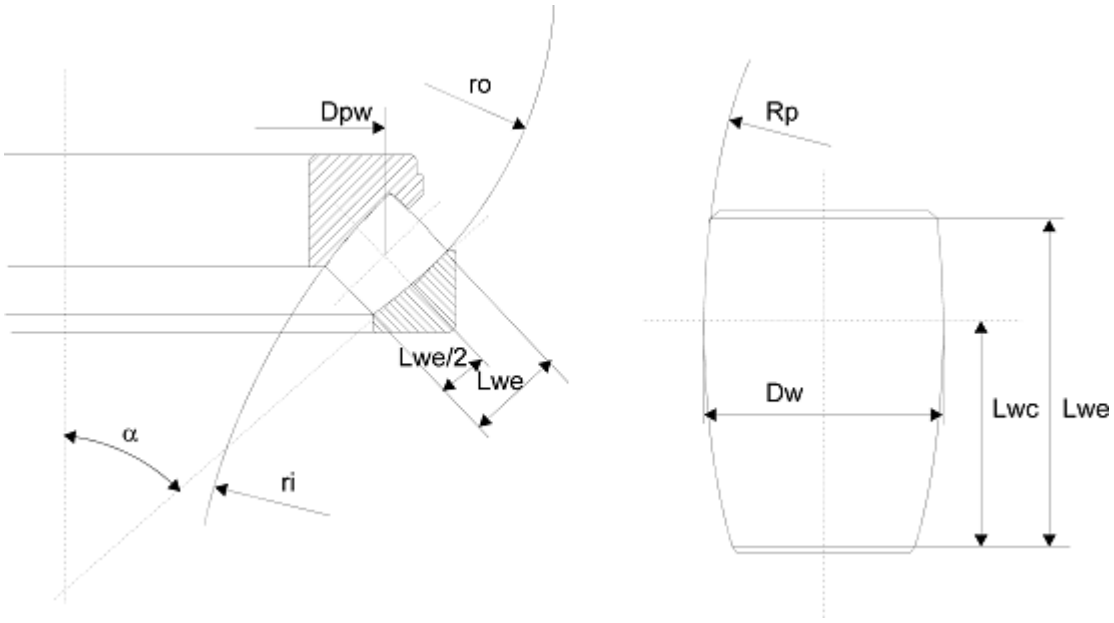


Figure 9.19: Mass of axial spherical roller bearings

9.5.38 Rolling bearing tolerance

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for rolling bearings begin with W05-??-??.dat.

9.5.39 Rolling bearing Tolerance classes

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for rolling bearing tolerance classes begin with W05-???.dat.

9.5.40 Tooth thickness tolerances

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for tooth thickness tolerances begin with Z01-???.dat or Z9-???.dat.

- **Interpret as:**
  - Tooth thickness allowance: the data is interpreted as a tooth thickness allowance.
  - Base tangent length allowances: the data is interpreted as the base tangent length allowances (or normal play).

9.5.41 Toothed belt standard

- **File name:** the database entries refer to external tables (see External tables (on page I-136)). Tables for the toothed belt standard begin with Z091-???.dat.
- **Calculation method:**
  - 1)"normal" toothed belts (RPP)
  - 2) GT types (PolyChain)
  - 3) AT types (Brecoflex)
  - 4) PG types (PowerGrip)

  Differences:

  - Special calculation for toothed belts with integrated steel rope (Method 3)
  - Calculating the operating factor: the special factor for the speed increasing ratio is added (Method 1, 2, 4) or multiplied (Method 3)
  - Additional performance table for higher performance at greater conversions (Method 2)
- **Calculation method for belt pretension:**
  - 1) in % of the circumferential force; Slack = 1/50 of the tension length
  - 2) in % of the maximum permitted circumferential force; Slack = 1/50 of the tension length
  - 3) in % of (operating factor*performance(W)/circumferential speed reference circle (m/s)) (according to DAYCO RPP Panther) Slack = 1/64 of the tension length
  - 4) in % of the circumferential force ; Slack = 1/64 of the tension length
- **Nominal range for power table b:** [mm] belt width, which corresponds to the performance data stored in the file (see file name).
- **Coefficient for belt pre-tension f:** 0 ... 1.0 (% factor for calculating the coefficient for belt pre-tension)

- **Maximum belt speed v_{max}:** [m/s]
- **Summand for operation F_s:** No influence
- **Pitch p:** [mm] pitch of the toothed belt
- **Elasticity E:** [N] Elasticity = force that doubles the length of a belt (with nominal width). If you do not know this value, enter 0 as the threshold value (in this case the elasticity is ignored when the bending test is performed).
- **Strain ε:** [%] Strain along the total length of the belt
- **Weight per length q:** [kg/m/mm] per meter length and millimeter width

File contents:	
List of suggested standard tooth numbers for toothed lock washers	*:TABLE LIST z.RadZahne*
List of suggested belt standard tooth numbers	*:TABLE LIST z.NormZahne*
Minimum number of teeth, depending on the speed (small disc)	*:TABLE FUNCTION z091k.factorINCR*
Correction factor for powering up, depending on the ratio (this is added to the operating factor)	*:TABLE FUNCTION z091k.factorINCR*
Transmissible power depending on the number of teeth (small disc) and speed (small disc)	*:TABLE FUNCTION z091k.powerNr*
Correction factor for the number of contacting teeth (small disc)	*:TABLE FUNCTION z091k.factorCorrEZ*
Correction factor for belt length	*:TABLE FUNCTION z091k.factorLength*
Correction factor for belt width	*:TABLE FUNCTION belt.bth*
Correction factor for belt width (same values as shown in the table above)	*:TABLE FUNCTION belt.beff*
Disc width depending on belt width	*:TABLE FUNCTION z091k.ScheibenBreite*
Belt type sizing: minimum transmissible power (lower limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWlower*
Belt type sizing: maximum transmissible power (upper limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWupper*

Chapter 10
Description of the Public Interface

10.1 Interfaces between calculation programs and CAD - Overview

The closest contact point of calculation programs within a CIM concept is the one with the drawing program (CAD). KISSsoft's public data interface can be freely formatted, enabling very powerful communication with third party programs.

All input and output data can be exported in ASCII format. The scope and format of this data is freely definable. For this, each calculation module contains a special, editable report file. The MMMMUSER.RPT[1] files are used as a template for this data transfer. The default setting is that these files are empty. If you want to output data over the interface, you first have to expand the templates. External programs can, in addition, transfer input data (also in ASCII format) to calculation modules. This data is read automatically during start-up, and then displayed on the screen.

10.1.1 Efficient interfaces

An automated data transfer between calculation and CAD should only be established if the benefits are considerably larger than the effort required. For example, an interface between a bolt calculation program and CAD is only of secondary importance since the information to be transferred (for example that, due to the calculation, an M10 bolt has to be selected) is too limited and could be transferred much faster "by hand". If, however, a standard parts library with bolts is available, the bidirectional link between the three components (calculation program, standard parts library and CAD) can prove very efficient.

The following efficient interfaces are available (but this list can be extended):

- General

  It should be possible to start the calculation programs from the CAD environment (for example by pressing a function key). This enables you to carry out a short calculation while you are drawing, transfer the results and then continue drawing.

- Shafts and bearing calculation

  – Output of a contour from the CAD (i.e. a shaft from detailed or drawing with combined elements) and reading it into the calculation program. (Problem: in many CAD programs it is unfortunately rather difficult to define the contour to be exported.)

[1] *The character string MMMM in a file name is a placeholder for the module to which the file refers. Example: M040USER.RPT*

- Output of a shaft that has been optimized in the calculation program (including rolling bearings etc.), reading it, and importing it into CAD as drawing information.
- Transfer of bending and similar data into the CAD.
- Rolling bearings and plain bearings are calculated, and then the contour is transferred to the CAD system. (Frequently, the CAD system already contains information on rolling bearings, so that only the bearing label is of interest.)

☼ Gear calculation

- Calculation of fabrication data in the program and transfer of the required values to the CAD as text. This is a very important function, since the recording of the data is very prone to errors, with potentially serious consequences.

  Calculation of the exact tooth form in Print Preview and transfer to the CAD system. (Although this results in very pretty drawings, it usually does not supply any necessary information, except if the data undergoes further processing, i.e. via transfer onto a wire electro-discharge machine.)

- Transfer of the schematic axial section or the Print Preview of the gears to the CAD system (but can also be achieved just as quickly in CAD, manually).

☼ Machine elements

Transfer the contour of calculated machine elements to the CAD such as bolts, V-belt sheaves etc. (Frequently, the CAD station already contains appropriate, preprogrammed information, so that only the parts definition is of interest).

☼ Shaft-hub connection

The sizing or proofing of connections should be implemented directly in a CAD system, so that known data from the CAD can be transferred into the calculation and the results of the calculation can in turn be returned to the CAD system.

10.1.2 Open interfaces concept in KISSsoft

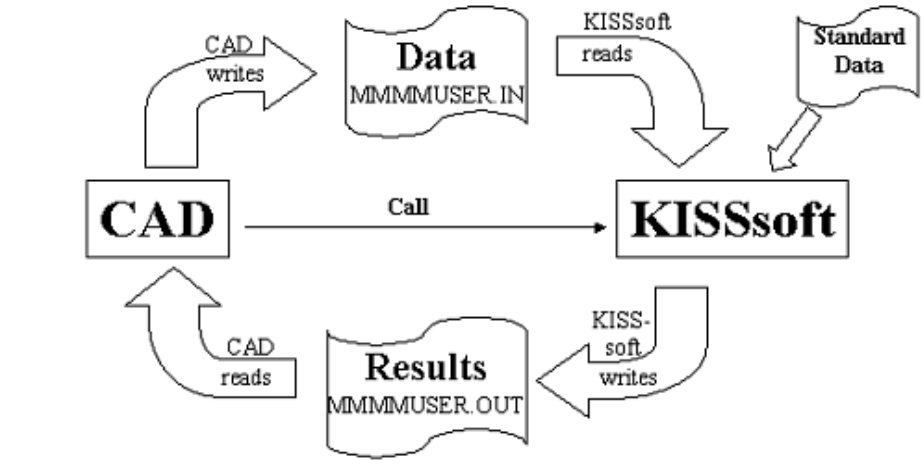
The KISSsoft interfaces concept has a simple, yet very flexible structure.

It should be possible to integrate calculation programs into all kinds of CAD systems as simply as possible, and use them in different environments (operating systems such as MS Windows or UNIX).

The interface mechanism between the CAD system and KISSsoft is based on a text data record (ASCII file), and an ID is transferred together with the numerical value for all transfer data (see Figure in the example (see section "Example: Interference fit calculation" on page I-178)). This data record can be of variable length, but only the values that are known in the CAD system will be transferred. This depends on the CAD system and the currently active drawing.

KISSsoft will test the data record transferred by the third party program, to ensure it is complete and consistent, and, if necessary, you will be promoted to input additional data in the KISSsoft input system. KISSsoft will then carry out the calculation and write the output data that the CAD system requires to a second text data record. It then returns this data record to the CAD system. By using the report generator you can select any format for the output file, i.e. KISSsoft adapts itself to the third party program. The CAD can now read the data required by the situation and selectively process them.

This concept results in simple interface forms, consequently enabling even non-specialists to write applications quickly.



CAD:

Generates data from drawing
Writes input file
Calls KISSsoft
Waits until calculation is done
(ie existing output file)
Reads results
Compiles new data in drawing

KISSsoft

Reads standard data
Reads input data, deletes file
User carries out calculation
Writes results in file

10.2 Defining input and output

10.2.1 Preamble

- In this description the KISSsoft program is always taken as a reference, i.e. an input file for KISSsoft becomes an output file for the third party program and vice versa.
- For automatic data exchange with other programs you will require files with the name *MMMMUSER.RPT*. You can adapt these files to your own requirements. However, if you have purchased KISSsoft interfaces, you should act with caution, since these files are also required for these interfaces.

File name	Storage location	Description
MMMMUSER.IN	<CADDIR> *)	Input file for KISSsoft (is written by the third party program) User's temporary input file (= will be deleted when imported into KISSsoft)
MMMMUSER.OUT	<CADDIR>	KISSsoft output file (will be written by KISSsoft and read by the third party program). Temporary (= should be deleted by the third party program)
MMMMUSER.RPT	<KISSDIR>	Defines the output format (similar to report), can be permanent/optional (= will usually be created once and is retained)
Z10Gear1.RPT *Z10Gear2.RPT* *Z10Gear3.RPT* *Z10Gear4.RPT*	<KISSDIR>	Defines the output format for the manufacturing data in the case of cylindrical gears (see below). Corresponds to MMMMUSER.rpt for this special case.
Z10Gear1.OUT ⋮	<KISSDIR>	Output file of the toothing stamp for cylindrical gears.
Z70Gear1.RPT *Z70Gear2.RPT*	<KISSDIR>	Defines the output format for bevel gears.
Z17Gear1.RPT *Z17Gear2.RPT*	<KISSDIR>	Defines the output format for crossed helical gears.
Z80Gear1.RPT *Z80Gear2.RPT*	<KISSDIR>	Defines the output format for worm gears.
Z9aGear1.RPT *Z9aGear2.RPT*	<KISSDIR>	Defines the output format for spline connections.
Z??Gear1.OUT *Z??Gear2.OUT*	<CADDIR>	Toothing stamp, similar to definition files.

*) If you specify the complete file name including the directory, it can also be read from any location.

10.2.2 Requirements placed on the third party program

To successfully operate KISSsoft within a third party program, the following minimum requirements must be met. The third party program must

1. have a query mechanism (i.e. macro language) in order to provide information, e.g. input data,
2. be able to write and read ASCII files,
3. be able to start a program.

10.2.3 Used files

10.2.3.1 Input file

An input file with the name *MMMMUSER.IN* will be used. It has the same structure and the same function as the saved calculations, except for its temporary status. The values will be assigned to the KISSsoft variable names with =. A separate row will be used for each variable.

EXAMPLE

VERSION=2.5;
m02Aw.dWa=30;
m02Aw.lW=20;
m02An.lN=25;

The input file will be read after the default values are predefined (see page I-52), i.e. the values of the temporary input file will overwrite the values set by the default.

Note: Temporary input files are used for frequently changing variables such as geometry and/or performance data: data which typically changes from calculation to calculation. It would also be possible to write this data into the template files, since they represent normal input variables. This would, however, mean that the program generating these files had to interpret the data that has already been written, i.e. had to accept permanent constraints, to enable it to completely define the standard and to reset it again at the end.

10.2.3.2 Output file

To return the data that is relevant for the KISSsoft calling program, the specified output file *MMMMUSER.OUT* will be generated immediately after a calculation.

The scope and the format of the output file will be defined in a report template called *MMMMUSER.RPT*.

This means that KISSsoft can fully adapt itself to the syntax of a third party program. The range of commands and the syntax of the report generator are described in the Reports (see section "Report templates" on page I-117) section. To help you, example report files are supplied.

10.2.4 Service life of files

The input file *MMMMUSER.IN* is generated by the third party program and, after having been read, will be deleted by KISSsoft. The *MMMMUSER.OUT* output file will be deleted when KISSsoft starts, and written again after a calculation.

10.2.5 Explicitly reading (importing) and generating data

In addition to the automatic definition described above, you can click the `File > Interface > Import data` menu to read (import) data specifically via or click `File > Interface > Export data`. You are therefore completely free to choose the point in time and thus use it for many varied tasks, i.e. the generation of an order form etc.

10.3 Example: Interference fit calculation

The following example of the Interference fit assembly calculation is used to illustrate the way that the KISSsoft interfaces concept works, in more detail.

For the interference fit assembly between the gear rim and the cylindrical gear hub, you need to find the one tolerance pairing that meets the following constraints:

Permanent torque MD = 88000 NM

The tolerance pairing involves a system of the standard drill hole *(H)* .

Safety	against sliding > 1.4
	against fracture of the wheel or hub > 1.5
	against fracture of the gear rim > 1.5
	against the yield point of the hub > 1.1
	against the yield point of the gear rim > 1.1

Procedure:

The necessary information for the geometry will be extracted direct from the drawing, with a suitable CAD routine, and converted into the interfaces format defined by KISSsoft:

```
m01allg.df=640
m01n.da=800
m01w.di=242
m01allg.l=200
```

Content of the *M010USER.INI* file

Then, start the KISSsoft module. It accepts the geometry data and displays it in the main screen.

In the main screen, enter any parameters that are still missing, the torque, and the materials, and then start the calculation. In KISSsoft, you can also size the tolerance pairing. Here, you are prompted to select the suitable tolerance combinations from a list. The system then performs the calculation with your final selection.

After you have finished the calculation, the results file is automatically converted to a format that can be read via the CAD macro. The format of this result file is defined via the templates file, *M010USER.RPT*:

```
[SHAFT]
ntol_max = %f{m01w.tol.max}
```

```
ntol_min = %f{m01w.tol.max}
ntol_bez = %s{m01w.tol.bez}

[HUB]
ntol_max = %f{m01n.tol.max}
ntol_min = %f{m01n.tol.max}
ntol_bez = %s{m01n.tol.bez}
```

Content of the *M010USER.RPT* file

The result has then following appearance:

```
[SHAFT]
wtol_max = 390.000000
wtol_min = 340.000000
wtol_bez = s6

[HUB]
ntol_max = 50.000000
ntol_min = 0.000000
ntol_bez = H6
```

Content of the *M010USER.OUT* file

Via the macro, this data will now be attached directly to the appropriate dimension in CAD.

Summary:

The individual tasks are therefore split up: Each side of the interface will perform only the tasks it is best suited to. The CAD administers the geometry and passes this information on to the calculation program, which knows how to process the data, and which, in turn, will return the result to the CAD.

The CAD system and calculation program can be used efficiently, in combination, with the defined interface.

10.4 Geometry data

KISSsoft contains different interfaces for transmitting geometry data (contours, drawings):

- DXF format (recommended for communication with most CAD systems)
- IGES format (with which tooth forms can be exported as splines)
- BMP format (Windows bitmap)
- JPG/JPEG format (pixel image)
- PNG (Portable Network Graphic) format

10.5 COM interface

KISSsoft includes the option of remote control via a COM interface. It can easily be accessed from Visual Basic or Excel.

10.5.1 Registering the server

Now register the KISSsoft COM server on your local computer.

There are two different ways of doing this:

- Right-click at the Windows prompt and select "As administrator" from the context menu. Then, go to where the ".../bin32" sub-directory is stored and run the "KISSsoftCOM_Register.bat" file.
- To do this, enter these two command lines in a Windows input prompt, in the KISSsoft installation bin32 directory:

```
KISSsoftCOM.exe /regserver
regsvr32 KISSsoftCOMPS.dll
```

You will need administrator rights to register the program.

10.5.2 Server functionality

The server provides a number of functions that you can use to start a calculation module, read or set values, and perform a calculation.

- `GetModule([in] BSTR module, [in] VARIANT_BOOL interactive)` starts a calculation module from the module descriptor (e.g. Z012 or W010). "interactive" defines whether the calculation module is to be generated with a graphical user interface.
- `Calculate()` performs the main calculation for the active module.
- `CalculateRetVal([out, retval] VARIANT_BOOL* isOk)` runs the main calculation for the active module, and returns a value to tell you whether the calculation is OK.
- `SetVar([in] BSTR name, [in] BSTR value)` is a function with which you can set variables to a required value. This data is transferred as text. You will find the variable names in the report templates, but there is no guarantee that all these variables will remain the same in the future.

- `GetVar([in] BSTR name, [out, retval] BSTR* value)` returns a variable from KISSsoft as text.
- `ShowInterface([in] VARIANT_BOOL wait)` displays the graphical user interface. Use the "wait" parameter to specify whether the function is to wait until the dialog is closed.
- `IsActiveInterface([out, retval] VARIANT_BOOL* isActive)` shows whether a KISSsoft dialog is active.
- `IsActive([out, retval] VARIANT_BOOL* isActive)` shows whether a module has been loaded.
- `ReleaseModule()` releases the loaded module again. You must always release a module again, to reduce the load on the server.
- `LoadFile([in] BSTR filename)` loads the specified file.
- `SaveFile([in] BSTR filename)` saves the calculation in the specified file.
- `CheckLicense ([in] name BSTR, [out, retval] VARIANT_BOOL* isOk)` shows whether the license is valid.
- `GetININame([OUT, retval] BSTR* name)` supplies the name of the loaded INI file.
- `GetVersionFromFile([in] BSTR filename, [out, retval] BSTR* version)` supplies the version number of the KISSsoft module in the calculation file (e.g. 2.6). (The version number depends on which module is being used)
- `GetModulFromFile([in] BSTR filename, [out, retval] BSTR* name)` supplies the KISSsoft module name in the calculation file (e.g. M040).
- `GetKsoftVersionFromFile([in] BSTR filename, [out, retval] BSTR* kSoftVersion)` supplies the KISSsoft version number (e.g. 03-2011), given in the calculation file.
- `GetKsoftVersion([out, retval] BSTR* kSoftVersion)` supplies the KISSsoft version (e.g. 03-2011), that is registered and was started via the COM interface.
- `GetDBName([in] BSTR db_name, [in] BSTR table, [in] SHORT flag, [in] LONG ID, [in] LONG order, [out,retval] BSTR *name)` Use the "flag" parameter to specify whether the ID (flag = 0) or the result (flag = 1) is to be used as the input. The output is then either the order and the name of the entry or the ID (in material

database BEZ_DIN). If there is an error, no message is displayed. A "False" is returned for the function.

- `GetDBValue([in] BSTR db_name, [in] BSTR table, [in] LONG ID, [in] BSTR fieldname, [out,retval] BSTR *name)` supplies the value present in this database field. If there is an error, no message is displayed. A "False" is returned for the function.
- `GetKsoftVersionSettings([out, retval] BSTR* kSoftVersionSettings)` supplies the KISSsoft version of the temporary settings folder in which the personal settings are stored (e.g. 03-2014).
- `SetSilentMode([in] VARIANT_BOOL silent)` defines whether messages are to be hidden or not, so that calculations can be performed without you having to confirm system prompts.
- `Report([in] LONG show)` writes the report. You can specify whether or not this report is to be displayed. The report is created in the Temp directory in the "KISS_?" subfolder.
- `ReportWithParameters([in] BSTR infile, [in] BSTR outfile, [in] LONG show, [in] LONG type)` creates the report using the specified report template (infile) in the predefined place with the predefined name (outfile) and supplies the file type. You can enter file names either with or without the path. When you enter the report template ("infile") you should also input the file extension (e.g."Z012ld0.rpt"). If you do not enter a path for this file the program will search the default directory (see also Reports) for the file. You must also enter the file extension for an output file. If you do not specify the path, the file is saved to the Temp directory with the name "KISS_?". The file extension should match the specified type. Use the "show" parameter to define whether the report is to be displayed (or not). Use the "art" (meaning "type") parameter to define the output format. (art=0 → rtf format with *1 ; art=1 → rtf format without *1; art=2 → html format with *1 ; art=10 → txt format without *1 ; art=20 → txt format in Unicode without *1)
  - *1 = takes into account the data level
  - Examples of possible combinations: With default report templates → RTF format: ReportWithParameters("C:\Program Files (x86)\KISSsoft 03-2014\rpt\Z070ld0.rpt","C:\Temp\Z070ld0.rtf" ,1,0), HTM format: Call ksoft.ReportWithParameters("Z070ld0.rpt", "C:\Temp\Z070ld0.htm", 1, 2) ; with drawing stamp report template → TXT format: ReportWithParameters("Z10GEAR1d.rpt","C:\Temp\Z010GEAR1d.txt" ,1,10)
- `Message([out] VARIANT *strings, [out] VARIANT *types:, [out] LONG *numElem)` returns the messages from the last

calculation in the first parameter, as an array containing strings. The second parameter contains the particular message type (error, warning, info). The number of existing messages is shown in numElem.

- `CallFunc([in] BSTR name)` Use these functions to run special calculations. A more detailed list of the available calculations is available on request.
- `CallFuncNParam([in] Variant paramArray)` Use these functions to run special calculations. A more detailed list of the available calculations is available on request.

10.5.3 Example of a call from Excel

The best way to describe the functionality is to use an example. To use KISSsoft from Excel, you must first activate the KISSsoftCOM type library in the Visual Basic Editor in `Tools>References`.

The first example shows how to use a single gear calculation to define the tip and root circles of a gear:

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As CKISSsoft

Dim da As String

Dim df As String


' get KISSsoft instance

set ksoft = New CKISSsoft


' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", False)


' set values

Call ksoft.SetVar("ZR[0].z", "20")

Call ksoft.SetVar("ZS.Geo.mn", "5.0")

Call ksoft.SetVar("ZR[0].x.nul", "0.5")


' Calculate
```

```
Call ksoft.Calculate

' get values
da = ksoft.GetVar("ZR[0].da.nul")
df = ksoft.GetVar("ZR[0].df.nul")

' release module
Call ksoft.ReleaseModule

' release server
Set ksoft = Nothing

End Sub
```

The second example shows how to display the KISSsoft user interface:

```
Public Sub ExampleKISSsoftCOM()
Dim ksoft As CKISSsoft
Dim da As String
Dim df As String

' get KISSsoft instance
Set ksoft = New CKISSsoft

' get KISSsoft module for single gear
Call ksoft.GetModule("Z011", True)

' show interface
Call ksoft.ShowInterface(True)
```

```
' get values

da = ksoft.GetVar("ZR[0].da.nul")

df = ksoft.GetVar("ZR[0].df.nul")


Call ksoft.ReleaseModule


Set ksoft = Nothing


End Sub
```

The same example with "later binding" (the exact property or method is not determined until runtime, which enables you to compile the Visual Basic client without having to know the exact function of the call):

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As Object

Dim da As String

Dim df As String


' get KISSsoft Object

Set ksoft = CreateObject("KISSsoftCOM.KISSsoft")


' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", True)


' show interface

Call ksoft.ShowInterface(True)


' get values

da = ksoft.GetVar("ZR[0].da.nul")
```

```
df = ksoft.GetVar("ZR[0].df.nul")


Call ksoft.ReleaseModule


Set ksoft = Nothing


End Sub
```

The fourth example shows a contact analysis that was run using the caControll.dat control file (you will find an example file in the dat folder) and they way messages were processed after the calculation:

```
Public Sub ExampleKISSsoftCOM()

On Error GoTo ExitOnErr

Dim ksoft As CKISSsoft


' get KISSsoft instance

Set ksoft = New CKISSsoft


' get KISSsoft module for gear pair

Call ksoft.GetModule("Z012", True)


' load File - change this to fit to a real file on your machine

Call ksoft.LoadFile("C:\yourPathHere\ExCOM3.z12")


' calculate

Call ksoft.Calculate


Dim ioData(0 To 2) as String

' Which calculation to start

ioData (0) = "CalculatePathOfContactForPairKS"
```

```
' controlling file
ioData (1) = "C:\ yourPathHere\caControl.dat"
' Path for results
ioData (2) = "C:\ yourPathHere\prot"

' calculate contact analysis
Call ksoft.CallFuncNParam(ioData)

' Check for messages
Dim mess As Variant
Dim types As Variant
Dim numElem As Long
Dim typesElem As Long
Dim typesElemStr As String

Call ksoft.Message(mess, types, numElem)

If (numElem > 0) Then
    Dim msg As String
    For i = 0 To numElem - 1
        msg = CStr(mess(i))
        typesElemStr = CStr(types(i))
        typesElem = CLng(types(i))
        If (typesElem = 0) Then
            Call MsgBox(msg, vbInformation)
        ElseIf (typesElem = 1) Then
            Call MsgBox(msg, vbExclamation)
        Else
            Call MsgBox(msg, vbCritical)
```

```
        End If
    Next
End If

' close ksoft
Call ksoft.ReleaseModule

' no problems, so exit
Exit Sub
ExitOnErr:
    MsgBox ("error occurred when calling KISSsoft.")

End Sub
```

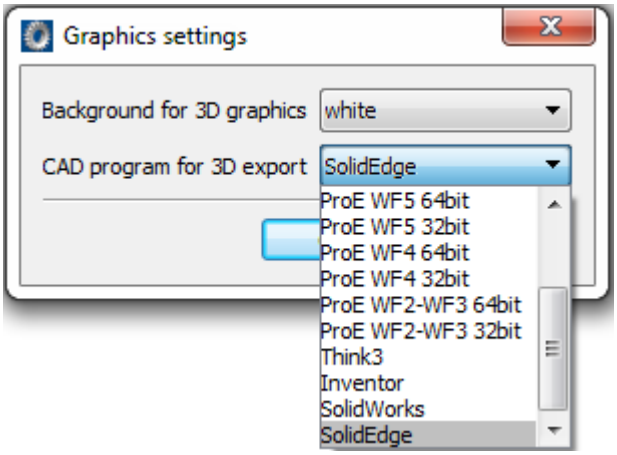
Chapter 11
3D Interfaces

11.1 Overview of the available CAD interfaces and their functionality

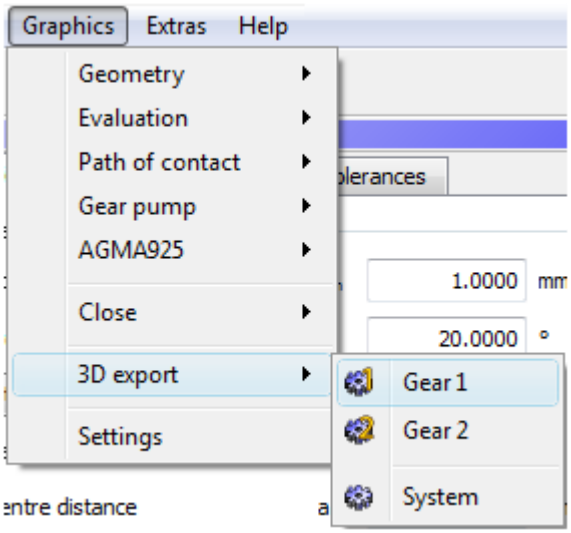
Feature/CAD	SolidWorks	Autodesk Authorized Developer Autodesk Inventor	SIEMENS Solution Partner PLM SolidEdge: Voyager Partner	SIEMENS Solution Partner PLM UGS NX: Status-Gold	PTC PartnerAdvantage SILVER ProEngineer, Creo Parametric	CATIA Catia	HiCAD	Parasolid / Neutral-formatschnittstelle (step)
Version	2014 - 2017	2014 - 2017	ST6, ST7, ST8, ST9	NX8 – NX11	Wildfire 5, Creo 1, Creo 2, Creo 3	V5 (R16-R22)	2010 - 2015	
Stirnräder Gerad/ Schrägverzahnt	✓	✓	✓	✓	✓	✓	✓	✓
Stirnräder Innen/ Aussenverzahnt	✓	✓	✓	✓	✓	✓	✓	✓
Schnecken/ Schraubräder	✓	✓	✓	✓	✓	✓	✓	✓
Zahnstange Gerad/ Schrägverzahnt	✓	✓	✓	✓	✗	✗	✓	✓
Geradverzahnte Kegelräder	✓	✓	✓	✓	✓	✓	✓	✓
Schrägverzahnte Kegelräder	✗	✗	✗	✗	✗	✗	✗	✓
Bogenverzahnte Kegelräder	✗	✗	✗	✗	✗	✗	✗	✓
Kronenräder	✗	✗	✗	✗	✗	✗	✗	✓
Kerbverzahnung (Welle-Nabe)	✓	✓	✓	✓	✓	✓	✓	✓
Verzahnung auf vorhandene Welle	✓	✓	✓	✓	✓	✓	✗	✗
Wellen	✓	✓	✓	✓	✗	✗	✗	✓
CAD Add-In Menü	✓	✓	✓	Nur über Menü	✗	Auswahl-menü	✗	✗
Herstelldaten	✓	✓	✓	✓	✓	✓	✗	✗
64 bit Version	✓	✓	✓	✓	✓	✗	✓	✗

11.2 Generation of 3D gears

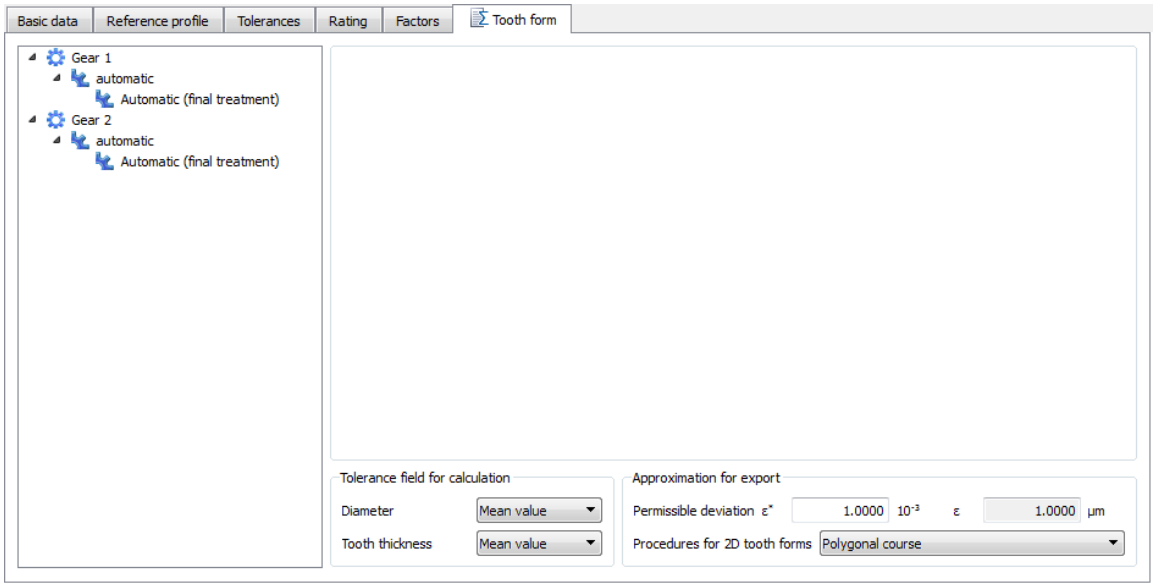
You must first perform a gear calculation to ensure that the results are consistent. Select `Graphics > Settings` to choose the CAD system to which you want to export the selected element.



Then, in the `Graphics > 3D export` menu option, you can select which individual gear and the configuration (only possible as individual gears) you want to generate.



In the case of Unigraphics NX, generation is only possible if you have started KISSsoft from the NX add-in menu, then run the gear calculation and pressed the required generation button. In the case of Creo Parametric (ProEngineer) and CATIA, you must run the CAD interface so that you can start the gear generation process from KISSsoft. In the SolidWorks, Solid Edge and Inventor CAD systems, press a generation button to start the CAD process, if it is not already open.

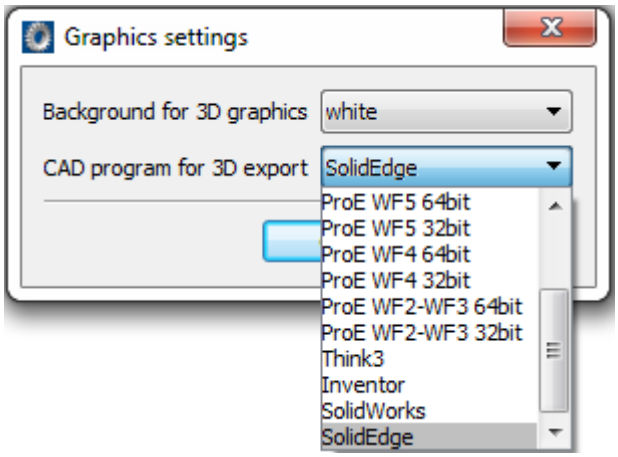


The default setting runs the gear generation process with a permitted tooth form deviation (tolerance band) of 1 μm. If this tolerance is too large, you can open the `Tooth form` tab to change the tolerance. Once this is changed, you must click `Calculate` again (Tooth form tab active), to transfer the inputs and recalculate the tooth form.
Changing the generation type in the Tooth form tab (polylines, circular pitch approximation, arc estimation, splines) only affects the 2D display. In NX, SolidWorks and Solid Edge, the part is generated with splines. In Inventor, Creo Parametric (ProEngineer) and CATIA, it is created with arcs of circles. SolidWorks and Solid Edge also support other generation types, which you can change by entering a value in the additional `APPROXIMATION=1` parameter, in the `kiss.ini` (see page I-59) file, in the relevant CAD system.
In the case of the gears, the transverse section of the tooth space is usually cut out from a cylinder, and then duplicated as a pattern. For worms with a helix angle of > 50° and number of teeth < 4, the tooth space is cut out in the axial section and then duplicated.

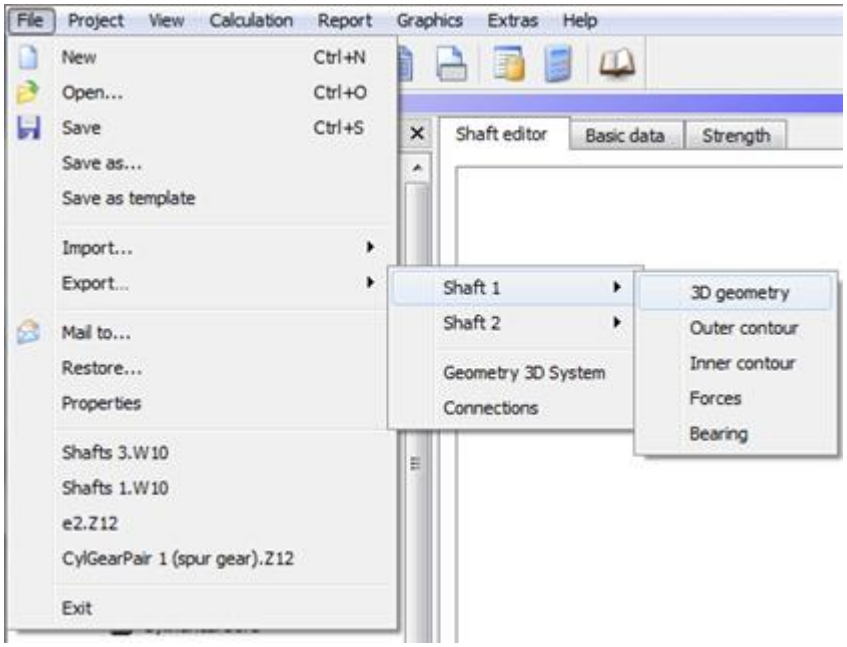
11.3 Generating 3D shafts

Until now it has only been possible to generate shafts in 3D in the SolidWorks, Solid Edge, Autodesk Inventor and NX CAD systems.

First, a shaft analysis must be performed to ensure the results are consistent. Select `Graphics > Settings` to choose the CAD system to which you want to export the selected element.



Then click `Graphics > 3D Export` to select the shaft(s) and configuration (if you want to generate more than one shaft) that you require. In a configuration, each shaft is created individually, in sequence, in its own part.

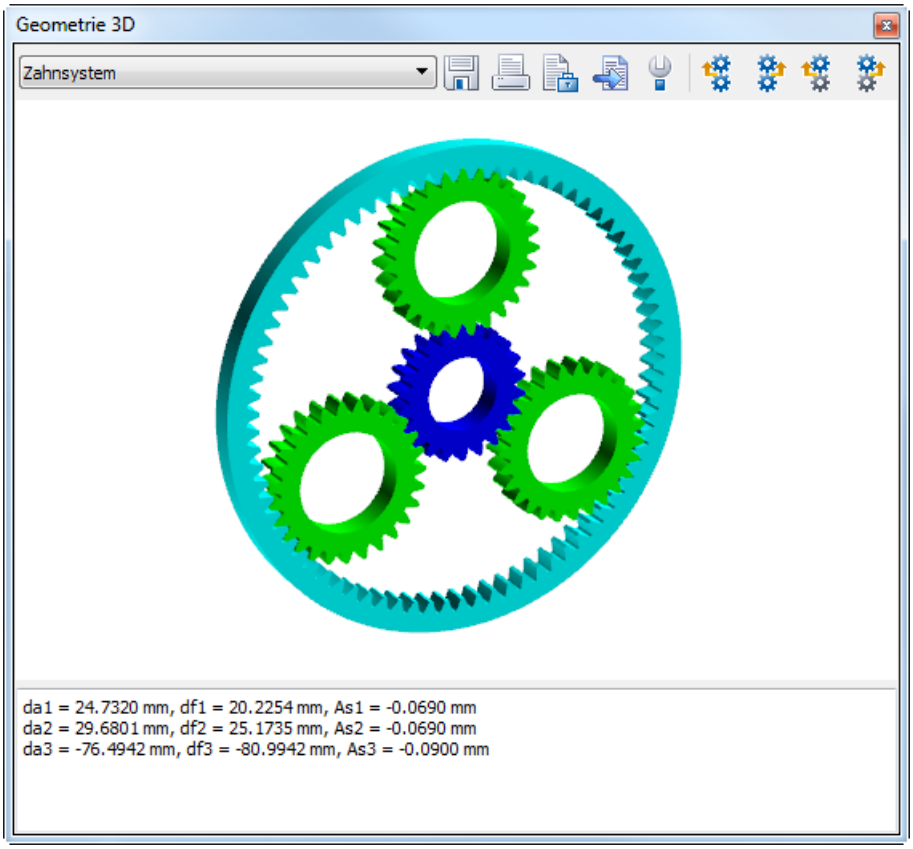


You can therefore create a 3D shaft in the CAD system at the click of a button, using the data from a KISSsoft shaft analysis.

11.4 Viewer with neutral format interface

KISSsoft provides a 3D viewer for displaying individual gears or a gear system. The viewer is activated from the Graphics > 3D Geometry menu.

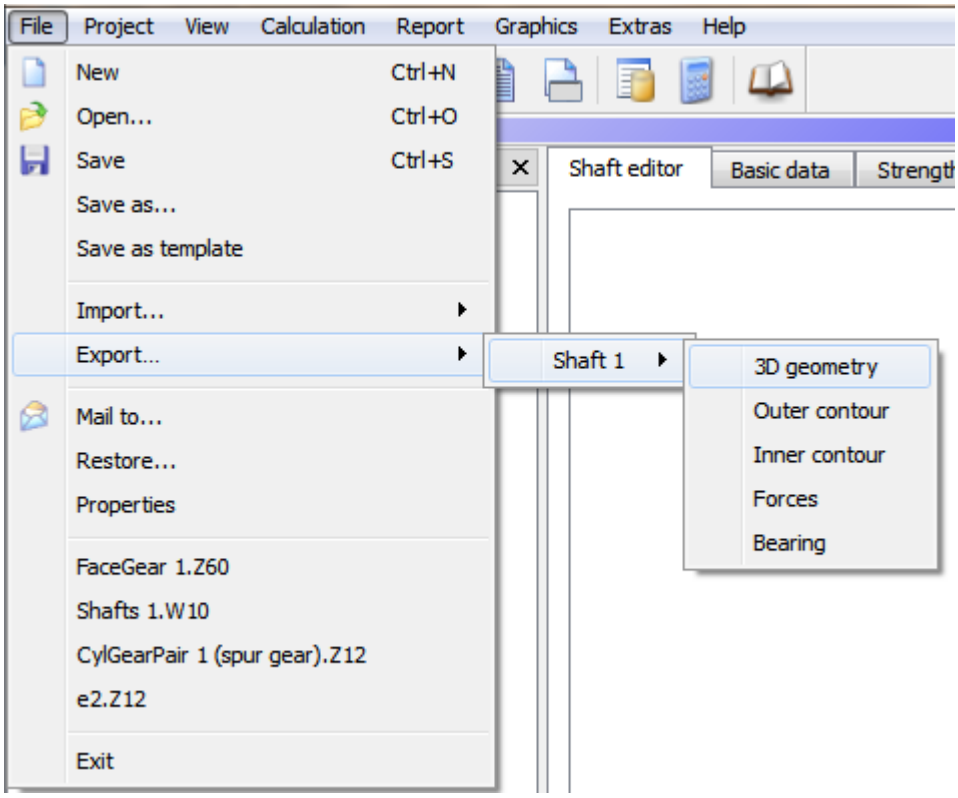
In the 3D viewer, you can export the solid model in STEP and Parasolid formats (text and binary). Supported gears (see page I-191) and for operating the viewer (see page II-646). You can change the settings by selecting Calculation > Settings > 3D Generation.



11.4.1 Parasolid Export of 3D Shafts

Parasolid can be used to generate the solid model of the shaft. The available formats for export are STEP, Parasolid text (X_T) and binary (X_B).

Select `File > Export > Shaft > 3D Geometry` to generate the model. If the calculation model contains a number of shafts, you can export them by selecting `File > Export > Geometry 3D System`.



11.4.2 Face gear - 3D geometry

The 3D model of a face gear is generated by simulating the cutting process, and has no limitations affecting the helix angle, shaft angle and offset. The reference coordinates of the model are defined in accordance with Roth [79], and the corresponding positions of pinion and gear are defined by equations (1) and (2).

$$L_S = \overline{O_S Q_S} = \frac{r_{t2} + \left(r_{tS} + x_S \cdot m_n\right) \cdot \cos\theta}{\sin\theta} \tag{1}$$

$$L_2 = \overline{O_0 Q_2} = \frac{r_{tS} + x_S \cdot m_n + r_{t2}\cos\theta}{\sin\theta} \tag{2}$$

Here, r_{tS} is the pinion reference radius and x_S is the pinion profile shift coefficient. Here, r_{tS} is calculated from the number of teeth of the pinion type cutter.

The shaft angle and radial offset (θ and a) are defined in `Geometry > Details…..`

The face gear model is generated by simulating the cutting process, and the tooth flank is approximated as a spline surface.

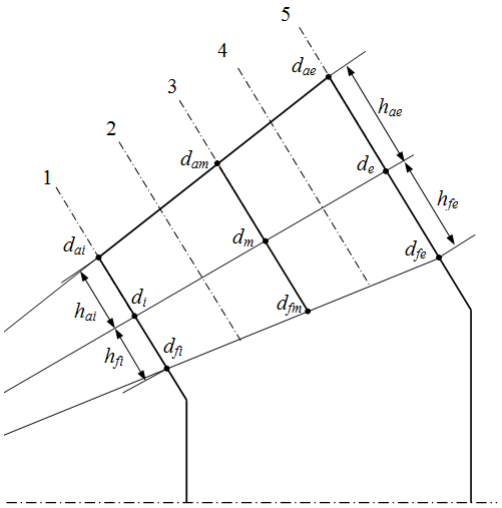
The manufacturing process is based on the Parasolid core, where the quality of the model depends on the settings made in Parasolid modeling (see Calculation > Settings > Parasolid).

NOTE:

The strength calculation is performed with the assumption that the shaft angle is 90° and the radial offset is 0. The shaft angle and radial offset are only used for 3D model generation, so the strength calculation result may not be valid.

11.4.3 Bevel gear - generating a 3D model

The 3D geometry model for straight, helical and spiral bevel gears is defined according to ISO 23509 and the tooth form is calculated for several sections along the facewidth. The tooth form is placed across the planar involutes of the virtual spur gear, at ninety degrees. Then, the tooth flank surface is generated by sweeping the tooth forms of the sections (Figure 10.1). The tooth forms in the individual sections are transformed by the angle φ_β into the relevant position. The angle of each section φ_β is calculated both for the generating and face milling processes by using the auxiliary angles φ and η. Therefore, the final tooth form along the facewidth is an extended epicycloid (face hobbing) or circular (generating) form, as shown in Figure 10.2.



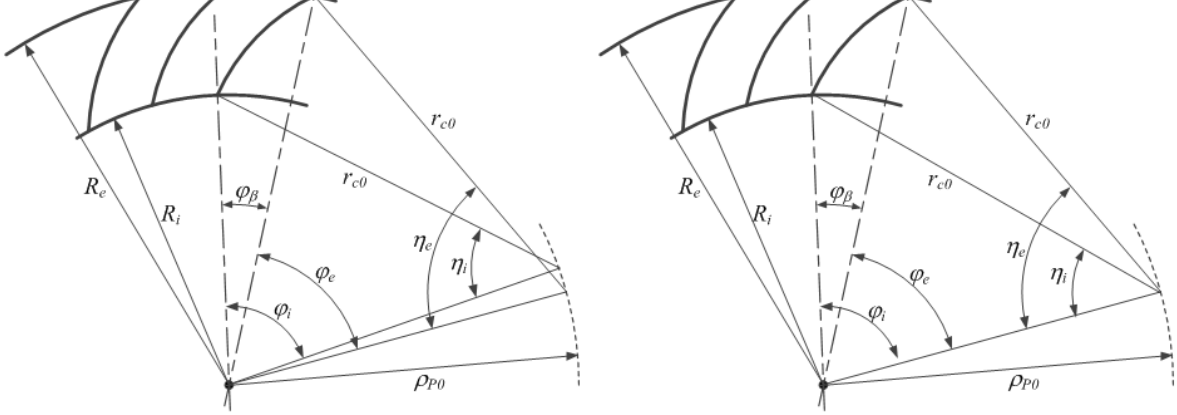


Figure 11.2 Transformation angle of generating (left) and face milling (right) processes

Machine tool manufacturers (such as Klingelnberg and Gleason) also have their own processes for generating tooth forms, that differ slightly from the procedures mentioned above. The tooth form is called an octoid and may differ slightly from our tooth form. However, we have ascertained that the difference between the tooth forms is much less than the tolerance range, and will not give rise to any problems in practical use.

11.4.4 Worm wheel - generating a 3D model

The 3D model of the enveloping worm wheel is generated by simulating the actual cutting process. The tooth forms at several sections along the facewidth are calculated, and the tooth flank is approximated as a spline surface. The model is generated using the best possible tool to manufacture the worm. Theoretically, the tool generates the worm, in regard to arc of circle, pressure angle, and tooth form. However, if the tool itself was manufactured to these specifications, it would no longer be usable after resharpening, because it would be smaller than the worm. The tools used to manufacture worm wheels are therefore slightly larger than the worm they are to create so that they can be resharpened several times, as required [91]. To generate the model using the larger tool, you can set the oversize factor in the module-specific Settings window.

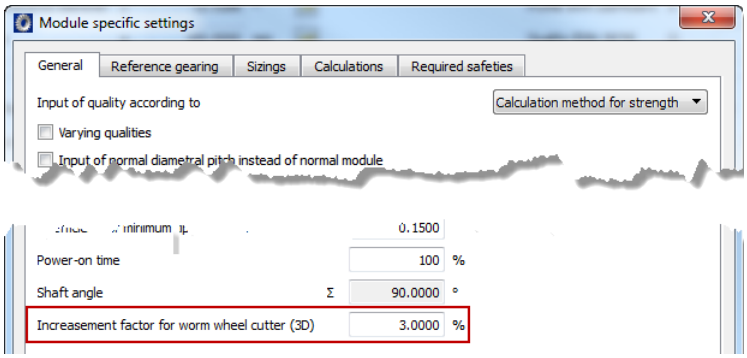


Figure 11.3 Oversize factor for worm wheel cutter

In this case, the tool will have a larger tooth thickness, and therefore generate a smaller tooth thickness on the gear. The cutting distance between the tool and the

gear will then be changed accordingly, to ensure a consistent result for the root and tip diameters on the gear.

11.4.5 General information about 3D modeling in Parasolid

If the model could not be generated correctly, you can improve it by modifying the Parasolid settings (see `Calculation > Settings > 3D Generation`) or, if gears are involved, by reducing the permitted deviation (`Tooth form > Approximation for export > Permissible deviation` tab).

11.5 3D interface to SolidWorks

Manufacturer: KISSsoft AG

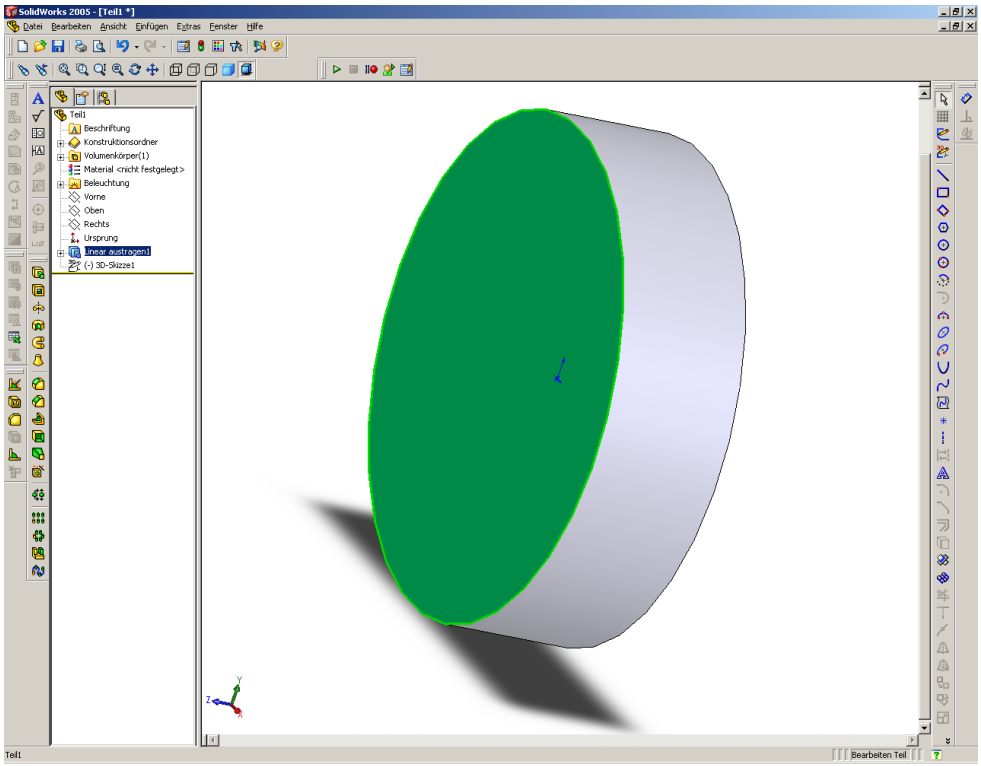
The interface between SolidWorks and KISSsoft creates the direct integration into the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within SolidWorks. Cylindrical or bevel gears calculated in KISSsoft can be generated directly in SolidWorks as a 3D part (see page I-192) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-194) directly in SolidWorks. From within KISSsoft, you can start SolidWorks with one click on a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts. Furthermore, you have the option of adding toothing to existing shafts (see page I-201) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see page I-206) as a text field. The gear manufacturing data is attached to the relevant cutout (tooth space).

11.5.1 Gear teeth in the case of an existing basic solid

11.5.1.1 Procedure for toothing creation

1. Select the required area in CAD

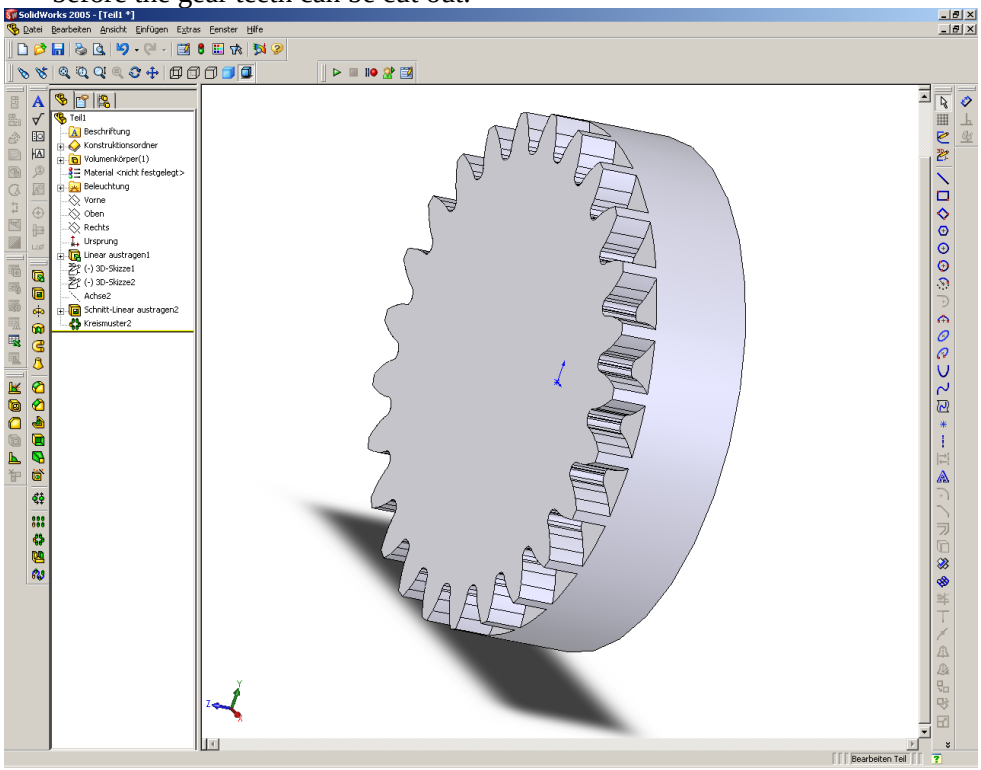
2. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.



Prerequisites:

- The diameter of the cylinder must already have the correct external diameter of the toothing before the generation starts.

- In the case of internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be performed for inside and outside cylindrical gears with spur and helical teeth.

11.5.2 Integrating the KISSsoft Add-in (menu options in CAD)

The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in SolidWorks, you must register the add-in manually.

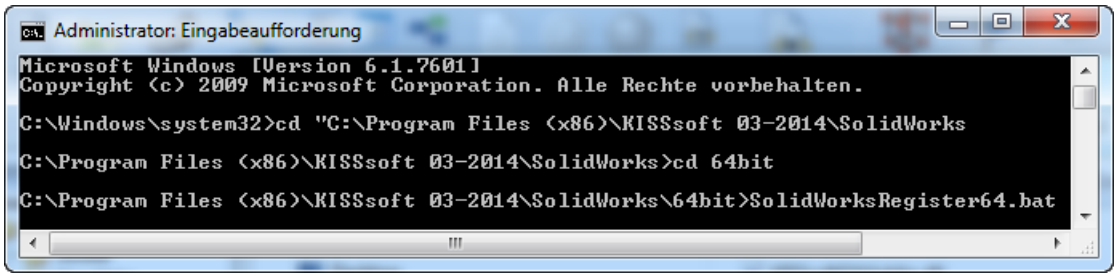
This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (`SolidWorks` folder). Confirm by pressing `Enter`.

3. Run the registration file. Confirm by pressing `Enter`.



If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

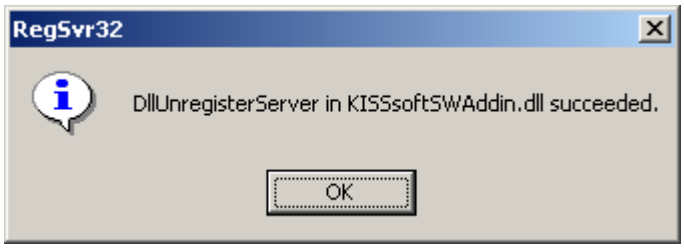
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

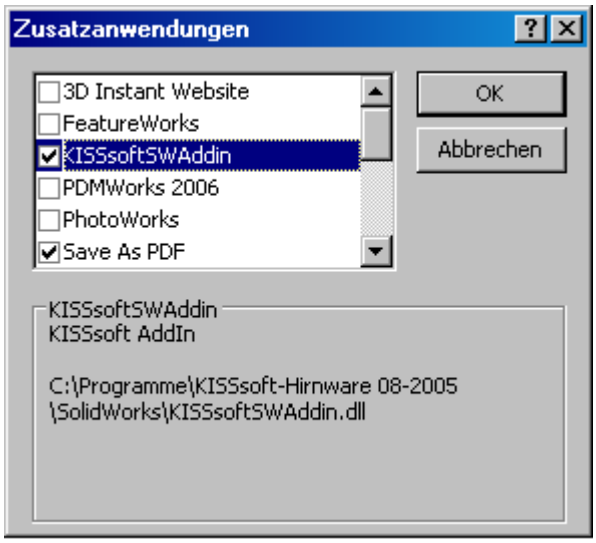
If the KISSsoft add-in has been registered successfully, you see this message.



To remove the registration, double-click on the `SolidWorksUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



If the add-in does not appear directly in SolidWorks, select the `Extras > Add-ins` menu option to open this window.



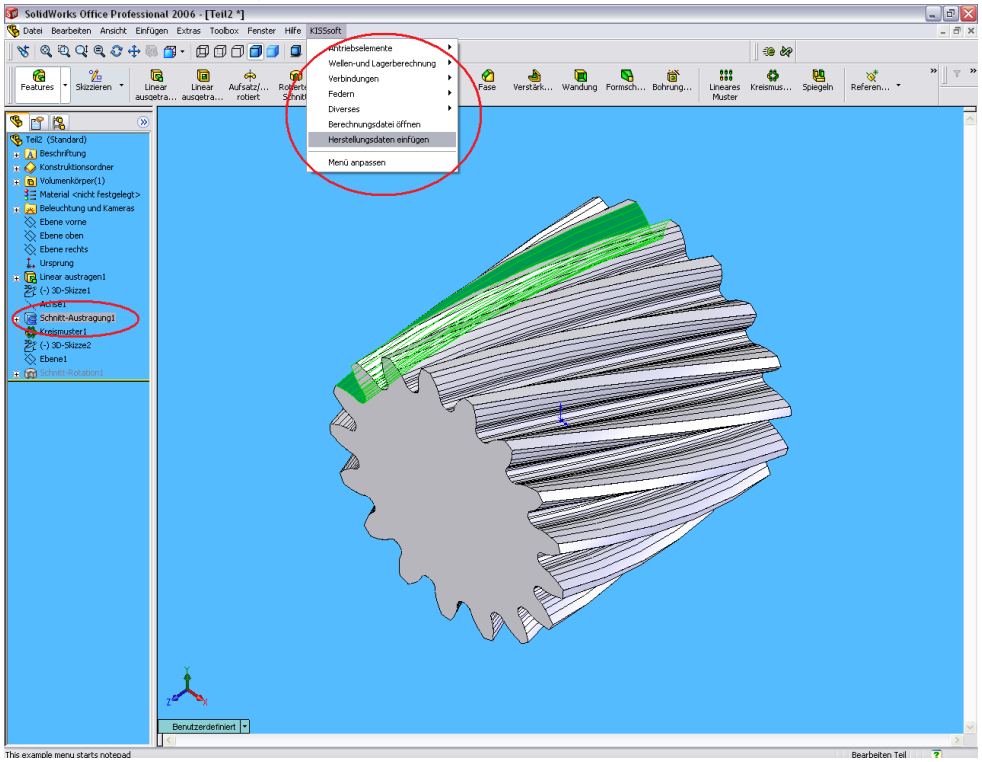
Here, select `KISSsoftSWAddin` and then click `OK` to confirm.

This integrates the KISSsoft menu options in SolidWorks. The menu remains present, even after a restart, and only needs to be linked once.

The KISSsoft add-in menu options are available in seven languages (English, French, German, Italian, Portuguese, Spanish and Russian). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

11.5.3 Add-in functions (calls)

11.5.3.1 Running KISSsoft via the Add-in



Select the `KISSsoft` menu option to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information previously given about gear generation (see page I-192).

11.5.3.2 Adding manufacturing data

The `Add manufacturing data` menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's `Cutout`.
2. Select the `Add manufacturing data` menu option.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.5.3.3 Opening the calculation file for the created gear

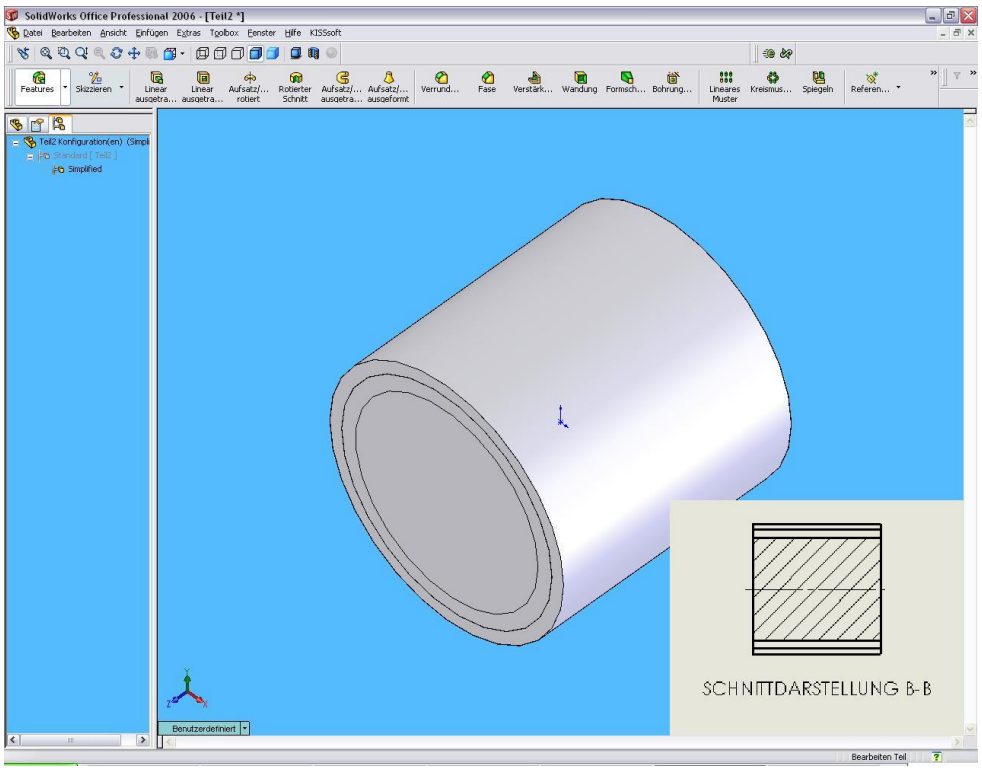
The `Open calculation` file menu option only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select a tooth's `Cutout`.
2. Select the `Open calculation file` menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.

11.5.3.4 Simplified view of the gears

You have the option to draw the gear in two different views. With the simplified view you can create a section display view of the gear in the drawing in which only the edge contours and the reference circle of the gear are shown. At the moment, the simplified view is only available for external teeth. In the default setting, the simplified view will not be carried out.



To obtain a simplified display, open the `KISS.ini` file in the KISSsoft installation folder and change this entry:

SIMPLIFIEDPRESENTATIONNAME=`Name`

The name given in the `kiss.ini` file is also the name of the view.

11.6 3D interface to Solid Edge

Manufacturer: KISSsoft AG

The interface between Solid Edge and KISSsoft creates the direct integration into the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within Solid Edge. Cylindrical or bevel gears calculated in KISSsoft can then be generated directly in Solid Edge as a 3D part (see page I-192) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-194) directly in Solid Edge. From within KISSsoft, you can start Solid Edge with one click on a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts. Furthermore, you have the option of adding toothing to existing shafts (see page I-210) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see section "Adding manufacturing data" on page I-215) on the drawing, as a text field. The gear manufacturing data is attached to the relevant cutout (tooth space).

NOTE:

The standard template file (e.g. metric.prt) is used to generate gears. To use a user-specific template file, either define a variable called `USERPARTTEMPLATE` in the kiss.ini file, in the `[Solid Edge]` section, or overwrite the default template file and copy it into the user-specific template files folder on the default path. If a user-specific path has been specified for templates in Solid Edge, this path is used. Otherwise, the default path for template files is used (e.g. `...\Solid Edge ST9\Template`).

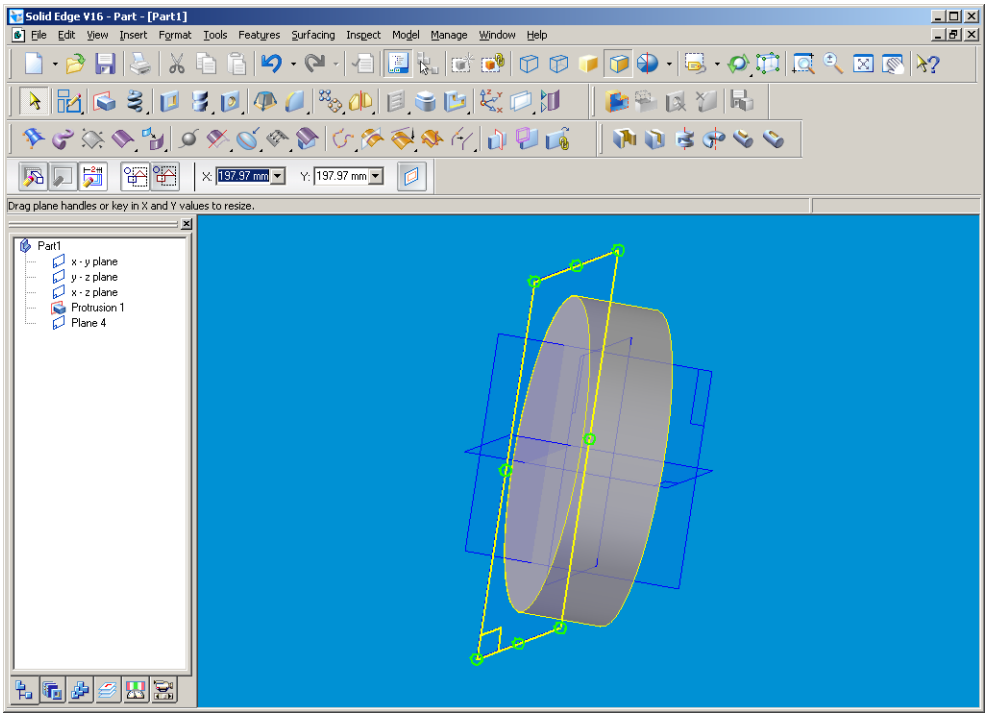
11.6.1 Change to the parameter for generation

When copying the tooth space (pattern) in Solid Edge, you can switch between two settings. The possible modes are: SmartPattern and FastPattern. In the case of SmartPattern, a more precise generation of the tooth form is carried out, but it takes a long time and the file containing the gear data will be very large. FastPattern uses a less precise method, but this ensures quick construction and a smaller generation file. Until now, SmartPattern has always been used for gear generation, since otherwise the gears cannot be created or represented correctly. In the `kiss.ini (see page I-59)` file in the KISSsoft installation folder you can set `SMARTPATTERN=0`, which executes the copying of the tooth space in FastPattern mode.

11.6.2 Gear teeth in the case of an existing basic solid

11.6.2.1 Procedure for toothing creation

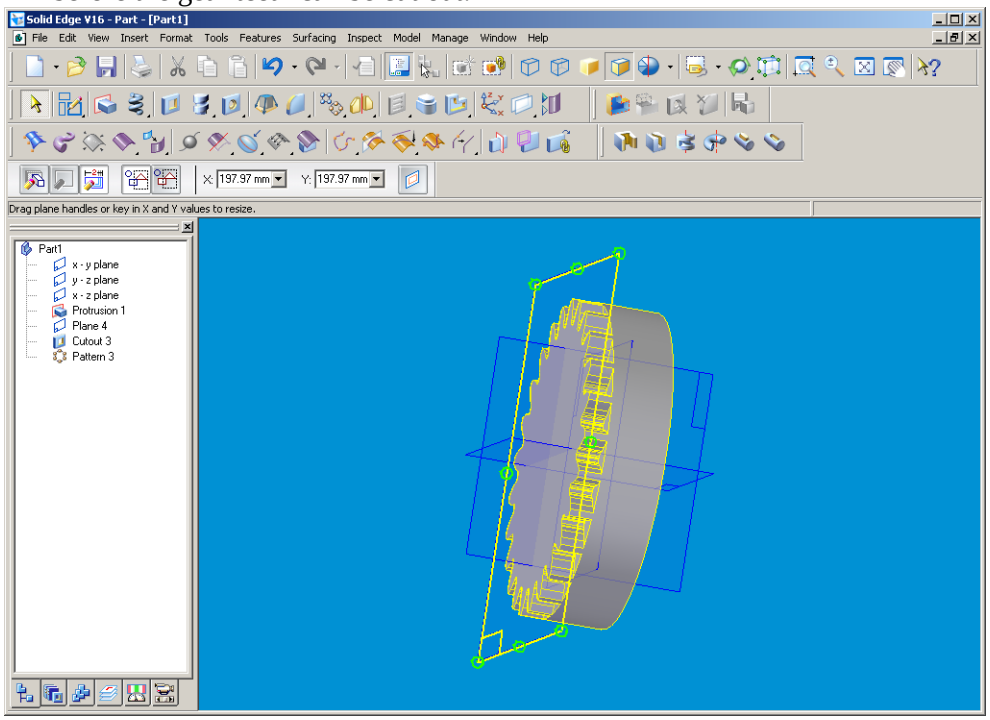
1. In Solid Edge, draw a surface in the required area where the gear teeth should be cut out.
2. Select the level
3. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.



Prerequisites:

- The cylinder diameter must already have the same external diameter as the gear toothing before the generation process starts.

- In the case of internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be carried out in the case of inside and outside cylindrical gears with spur and with helical teeth.

11.6.3 Integrating the KISSsoft Add-in (menu options in CAD)

The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in Solid Edge, you must register the add-in manually.

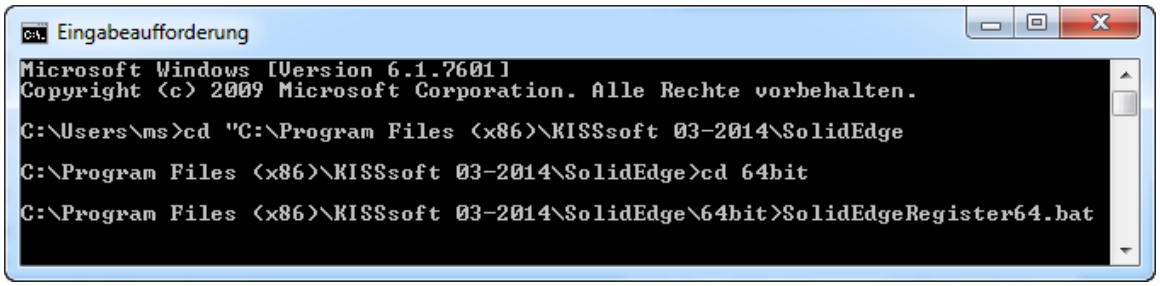
This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (`Solid Edge` folder). Confirm by pressing `Enter`.

3. Run the registration file. Confirm by pressing `Enter`.



If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

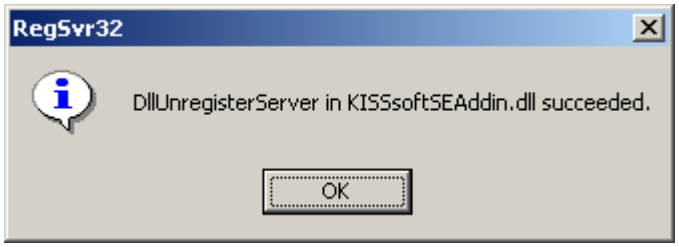
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

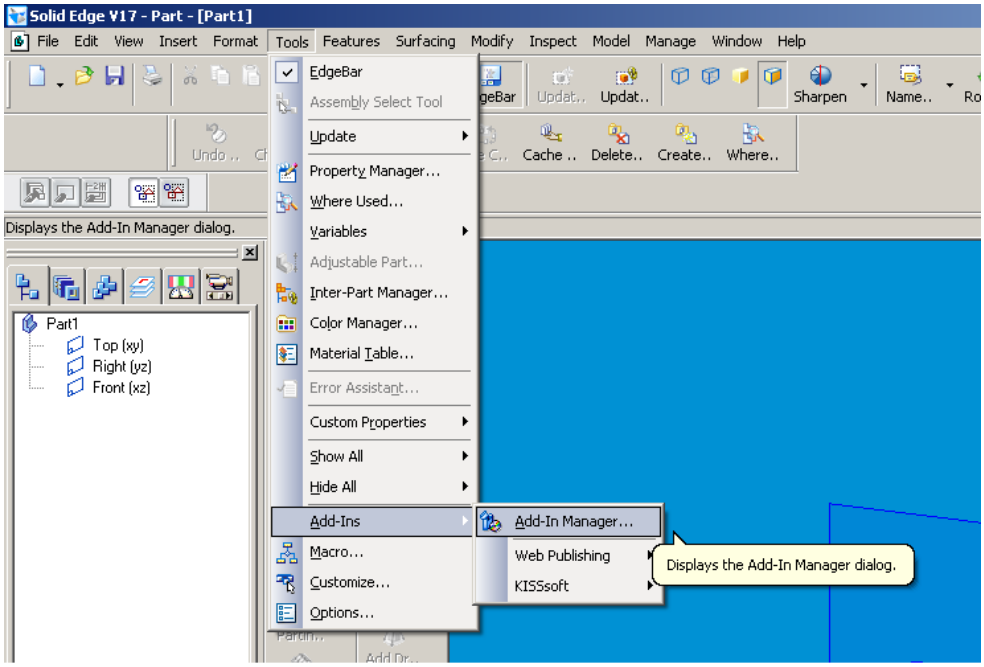
If this message is displayed, the add-in has been successfully registered.

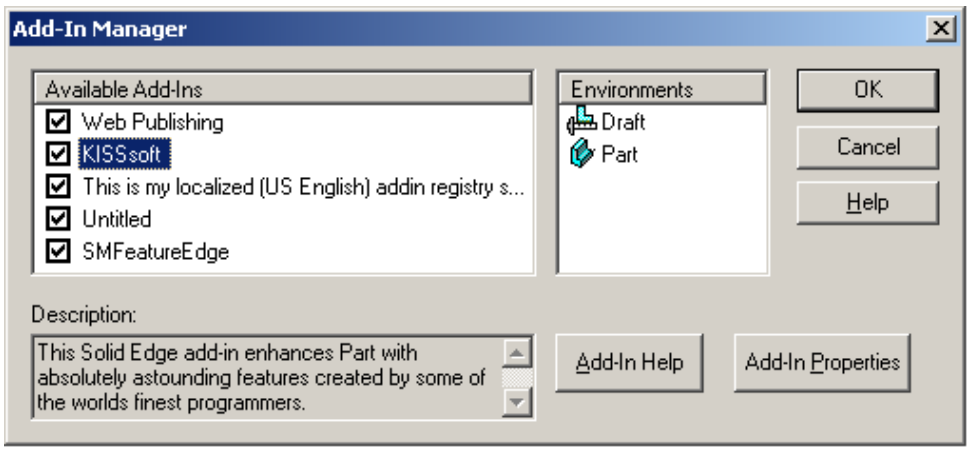


To remove the add-in registration, double-click on the `SolidEdgeUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process completes successfully.

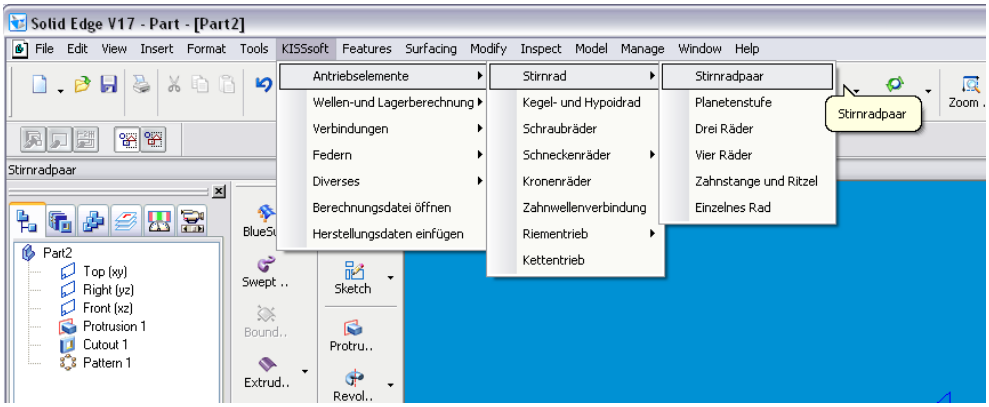


Under `Tools/Add-Ins` you can select `Add-In-Manager` where you can activate/deactivate the add-in.





You will see the KISSsoft add-in in the main menu. This integrates the KISSsoft menu options in Solid Edge where they remain present, even after a restart.



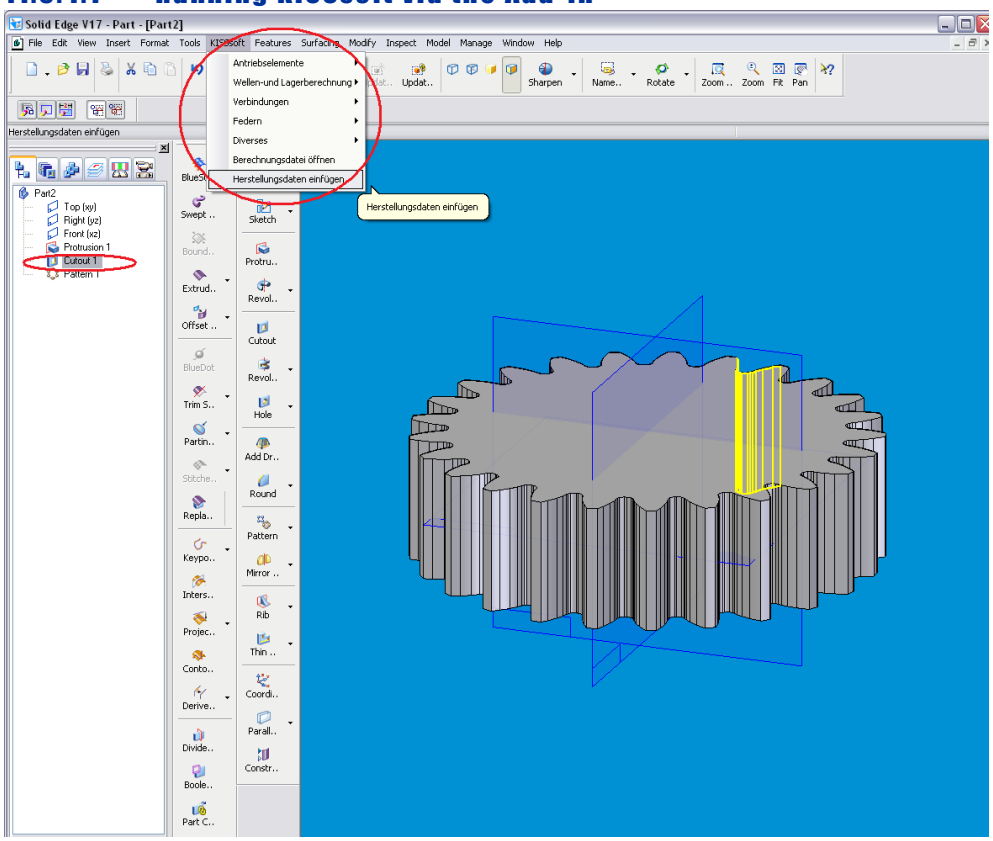
The KISSsoft add-in menu options are available in seven languages (English, French, German, Italian, Portuguese, Spanish and Russian). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

NOTE:

If the selected language uses Unicode fonts (e.g. Cyrillic for Russian), the Localization must be set to this language (a country with this language) in the operating system.

11.6.4 Add-in functions (calls)

11.6.4.1 Running KISSsoft via the Add-in



Select the `KISSsoft` menu option to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information previously given about gear generation (see page I-192).

11.6.4.2 Adding manufacturing data

The `Add manufacturing data` menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's `Cutout`.
2. Select the `Add manufacturing data` menu option.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.6.5 Opening the calculation file for the created gear

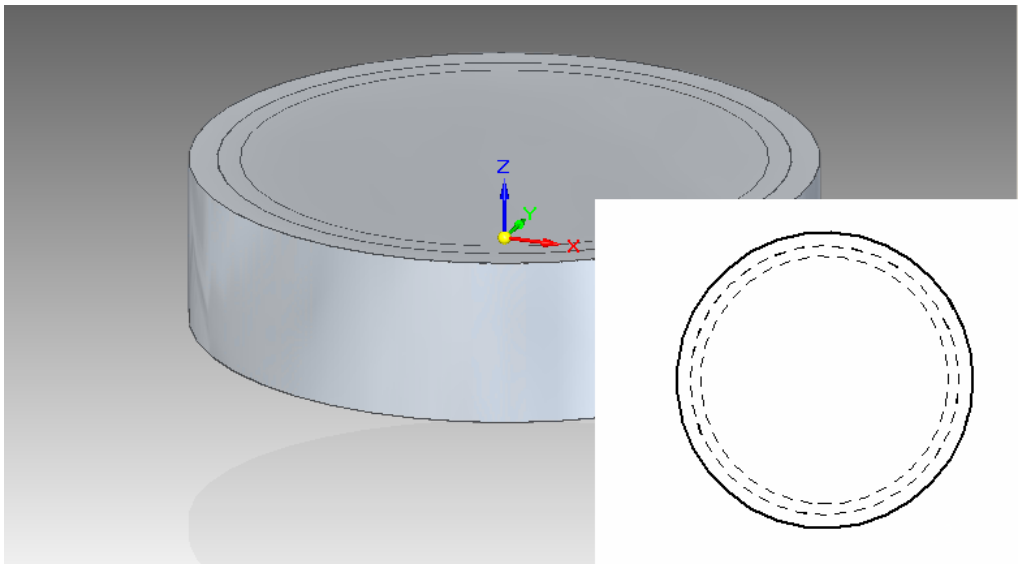
The `Open calculation` file menu option only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select a tooth's `Cutout`.
2. Select the `Open calculation file` menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.

11.6.6 Simplified view of the gears

You have the option to draw the gear in two different views. With the simplified view you can create a section display view of the gear in the drawing in which only the edge contours and the reference circle of the gear are shown. At the moment, the simplified view is only available for external teeth. In the default setting, the simplified view will not be carried out.



To obtain a simplified display, open the `KISS.ini` file in the KISSsoft installation folder and change this entry:

SIMPLIFIEDPRESENTATION=1

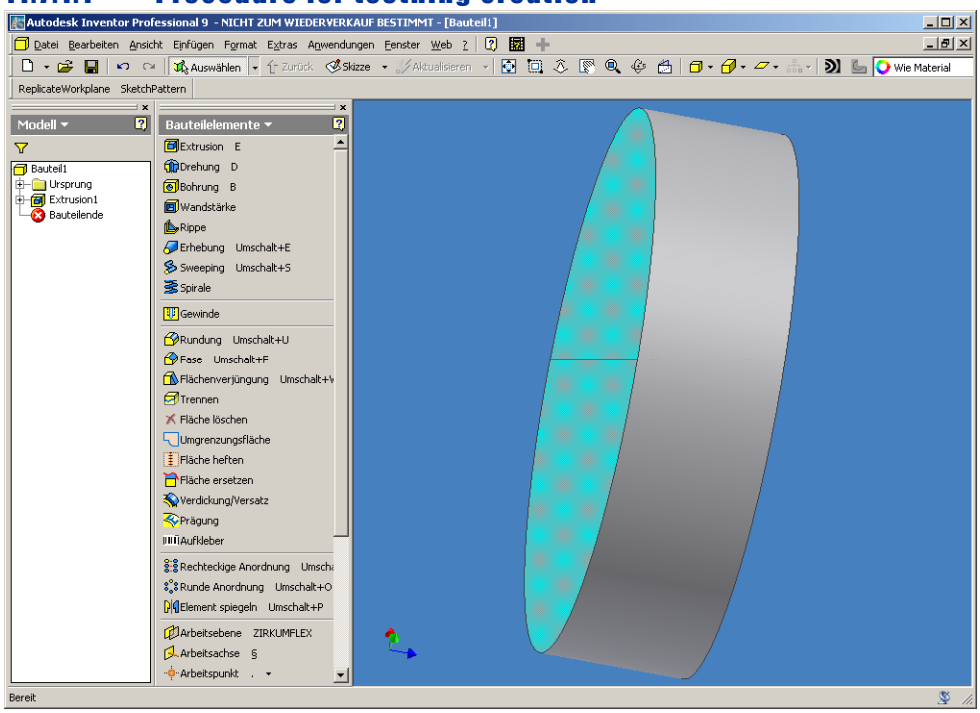
11.7 3D interface to Autodesk Inventor

Manufacturer: KISSsoft AG

The interface between Inventor and KISSsoft creates the direct integration with the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within Inventor. Cylindrical or bevel gears calculated in KISSsoft can then be generated directly in Inventor as a 3D part (see page I-192) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-194) directly in Inventor. From within KISSsoft, you can start Inventor with one click on a button. The system opens a new part, and generates the appropriate part. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts. Furthermore, you have the option of adding toothing to existing shafts (see page I-217) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see section "Adding manufacturing data" on page I-221) on the drawing, as a table. The gear manufacturing data is attached to the relevant cutout (tooth space).

11.7.1 Gear teeth when existing shaft data is present

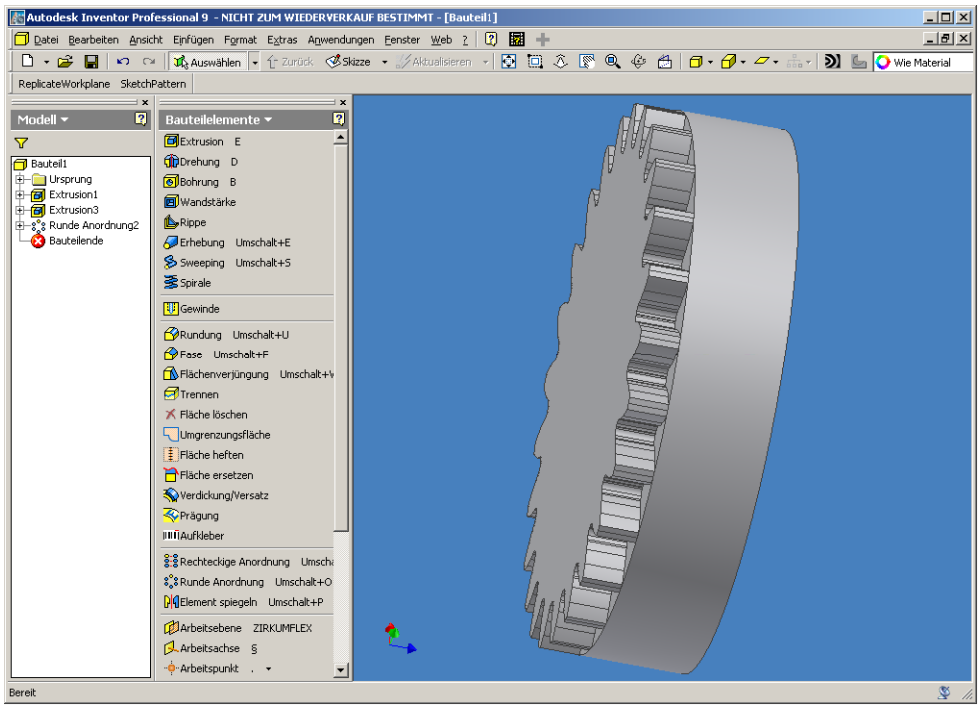
11.7.1.1 Procedure for toothing creation



1. Select the required area
2. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.

Prerequisites:

- The cylinder diameter must already have the same external diameter as the gear toothing before the generation process starts.
- In the case of internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be performed for inside and outside cylindrical gears with spur and helical teeth.

11.7.2 Add-in (menu options in CAD)

11.7.2.1 Integrating the KISSsoft Add-in

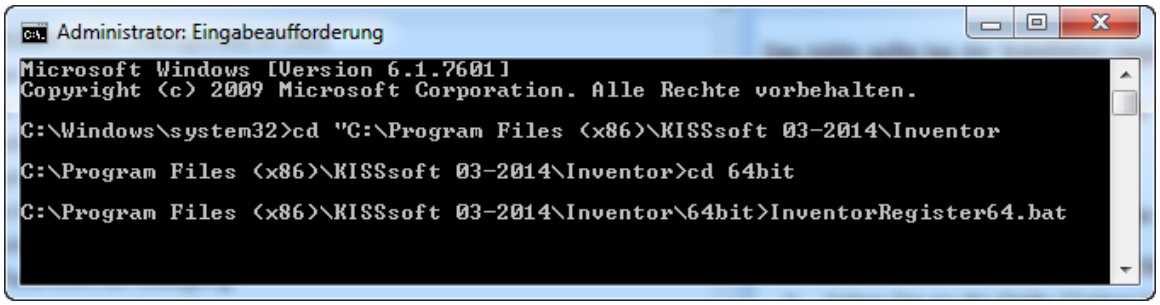
The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in Inventor, you must register the add-in manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (`Inventor` folder). Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.



If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

If this message is displayed, the add-in has been successfully registered.

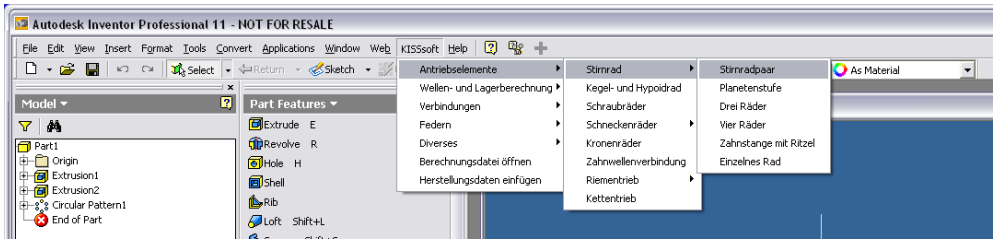


If you no longer want the Inventor add-in to be registered, double-click on the `InventorUnRegister.bat` file in the KISSsoft installation folder. If the add-in has been registered successfully, you see this message.



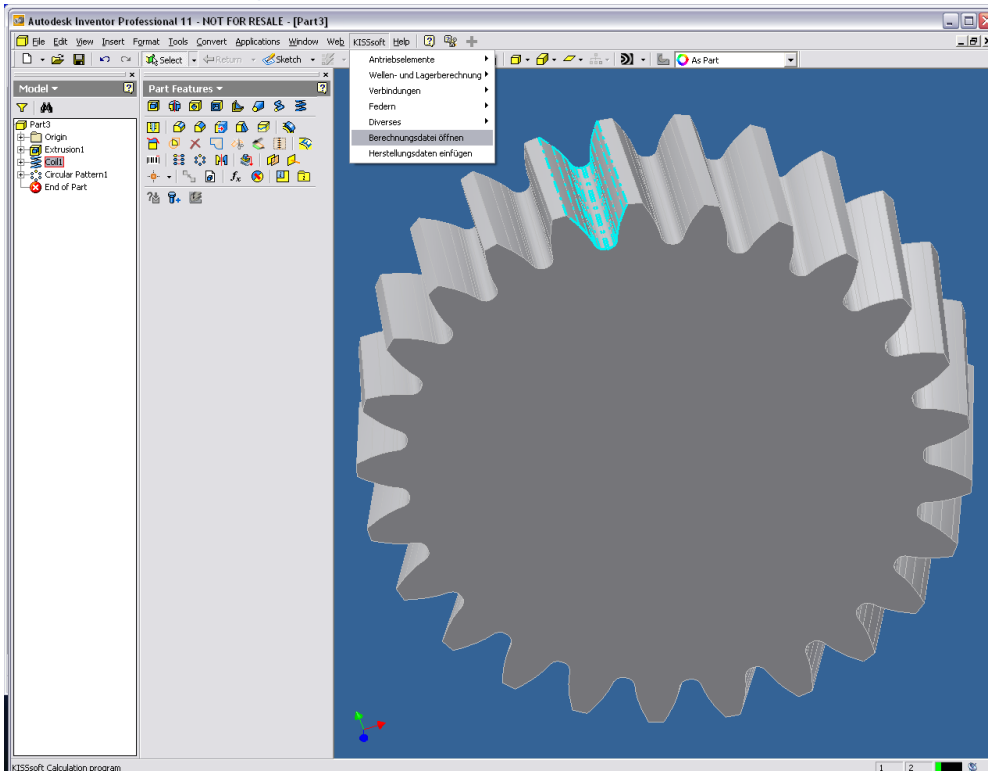
The KISSsoft add-in menu options are available in seven languages (English, French, German, Italian, Portuguese, Spanish and Russian). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

This integrates the KISSsoft menu options in Inventor. The menu remains present, even after a restart, and does not need to be linked.



11.7.3 Add-in functions (calls)

11.7.3.1 Running KISSsoft via the Add-in



Select the `KISSsoft` menu option to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information previously given about gear generation (see page I-192).

11.7.3.2 Adding manufacturing data

The `Add manufacturing data` menu option only works in the Part view. Procedure for adding a gear stamp to a drawing:

1. Open the part and select a tooth's `Cutout`.
2. Select the `Add manufacturing data` menu option.

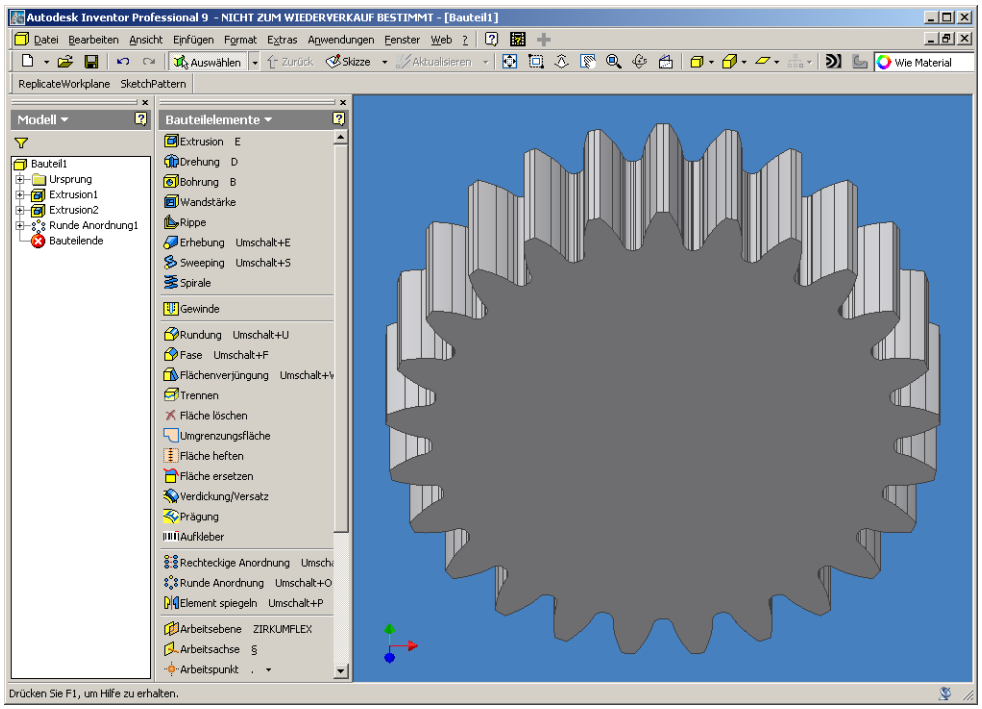
This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

11.7.4 Opening the calculation file for the created gear

The `Open calculation` file menu option only works in the Part view. Procedure for opening a calculation file:

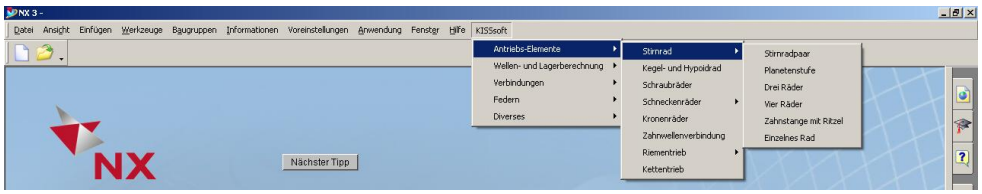
1. Open the part and select a tooth's `Cutout`.
2. Select the `Open calculation file` menu option.

This starts KISSsoft in each particular calculation module and opens the calculation file.



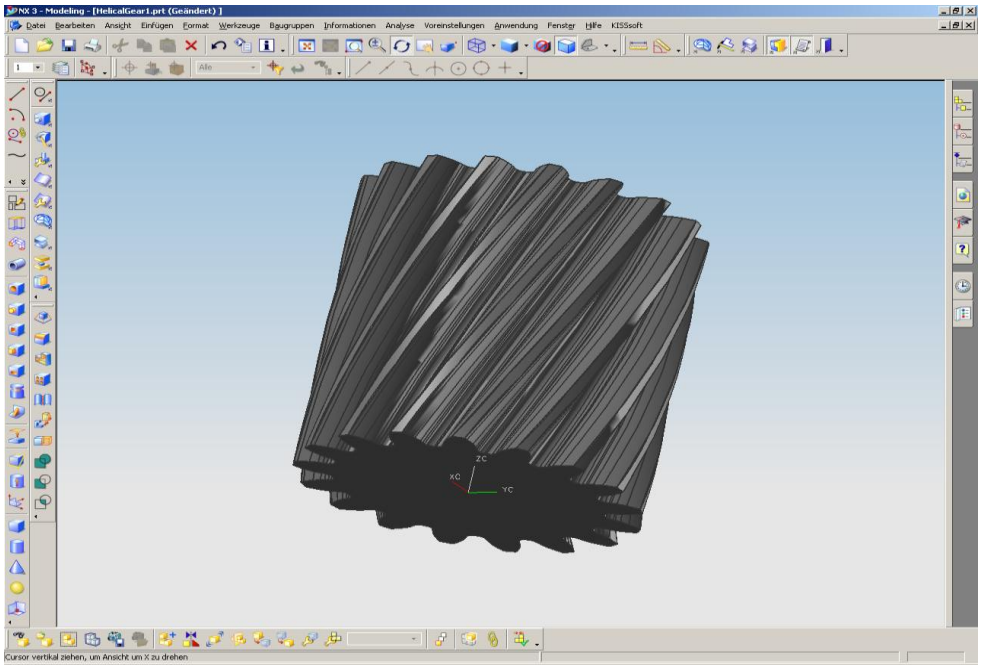
11.8 3D interface to Unigraphics NX

Manufacturer: KISSsoft AG



The interface between NX and KISSsoft is created by direct integration in the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within NX. Cylindrical or bevel gears calculated in KISSsoft can then be generated directly in in NX as a 3D part (see page I-192) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-194) directly in NX. You can create cylindrical gears with straight or helical teeth, which are outside or inside, worms, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.

If you create a new part, the `New` dialog opens first. In it, you can enter the name of the file in which the part should be generated. When you use Teamcenter, its dialog is displayed automatically so you can also generate or save the part in the Teamcenter environment. Furthermore, you have the option of adding toothing to existing shafts (see section "Gear teeth when existing shaft data is present" on page I-229) at a later point in time. In addition, gear manufacturing data in the 2D range can be automatically inserted as a table on the drawing, with the interface. The gear manufacturing data is attached to the relevant cutout (tooth space).



11.8.1 Add-in (menu options in CAD)

11.8.1.1 Integrating the KISSsoft Add-in

First, copy the supplied folder, e.g. `NX9`, with its `startup` sub-folder, to a location that can be accessed by users at any time.

The definitions of the KISSsoft add-in menu options are located in the "kSoftNX_e.men" file. This file has different name endings to indicate which language has been selected. For example, the `_e` in this file name represents `English`. Also used: _d: for `German`; _f: for `French`; _i: for `Italian`; _s: for `Spanish`; _r: for `Russian`; _p: for `Portuguese`; The file in the required language can be copied to the `startup` folder to ensure the KISSsoft menu is displayed in the selected language.

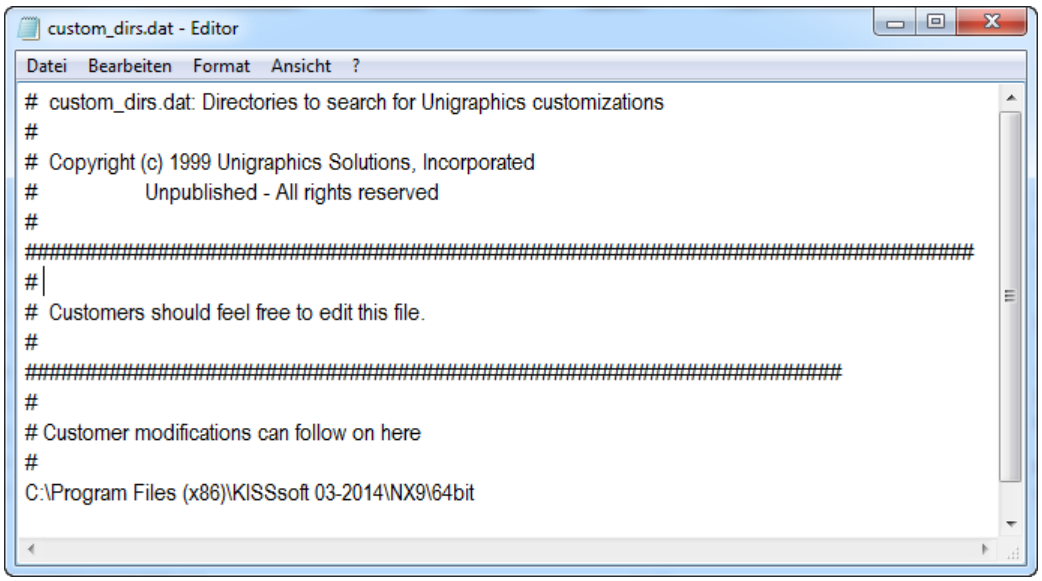
NOTE:

If the selected language uses Unicode fonts (e.g. Cyrillic for Russian), the Localization must be set to this language (a country with this language) in the operating system.

From NX9 KISSsoft is also available as a ribbon menu. By default, the English menu with "_e" is integrated in the "startup" sub-folder. If you want to change the language in which the menu is displayed, delete all the files in the startup sub-folder that have "_e" in their name. The "... \\NX9 \\64bit" sub-folder contains a sub-folder for every available language (e.g. "kSoftNXRibbon_d" for German). You can copy the entire contents of the folder that has the language you require into the "startup" sub-folder. The menu will then be displayed in this language.

The `kSoftNX9.dll` file (for example), which contains the links and commands for the menu options, is also stored in this folder.

You must enter the path of the previously copied folder, for example, `NX9/64bit`, in the "UGII\menus\custom_dirs.dat" file in the UGS directory, so that the UGS system can tell where the files it is to use are stored.



The KISSsoftCOM server should be registered as part of the installation process. However, if this did not happen, and the KISSsoft interface does not work, you must register KISSsoftCOM manually.

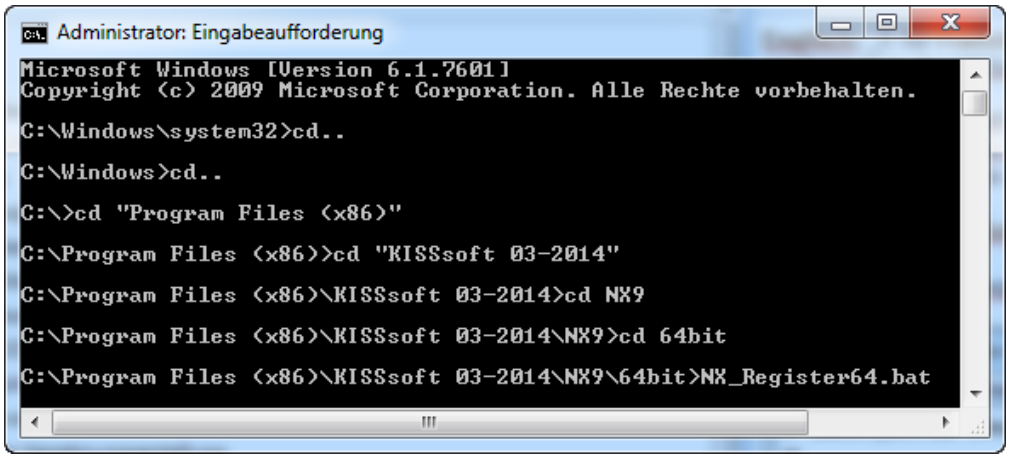
This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (the folder, e.g. `NX9/64bit`). Confirm by pressing `Enter`.

3. Run the registration file. Confirm by pressing `Enter`.



If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

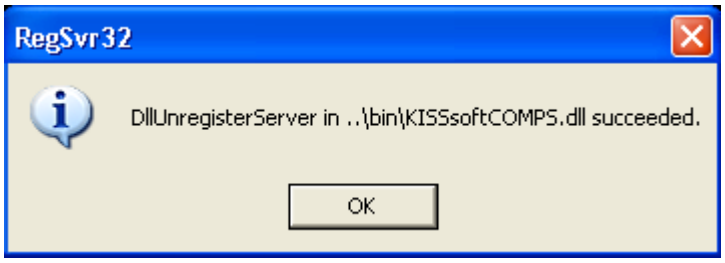
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the KISSsoft add-in has been registered successfully.



To remove the registration, double-click on the `NXUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



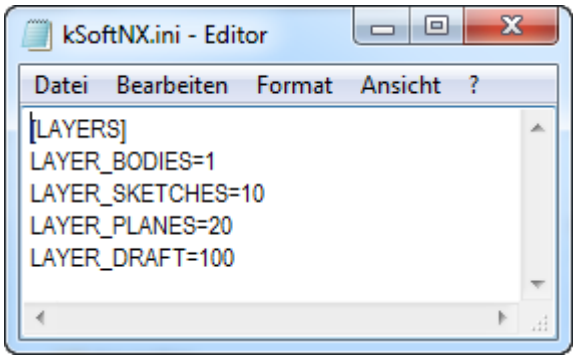
To ensure the KISSsoft icons are displayed next to the menu options, you must also set a system variable with the path, to tell the program where the KISSsoft icons can be found.

Set system variable and use it as a value for the path, for example:

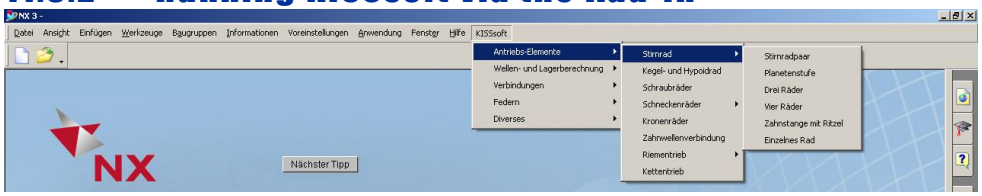
KSOFT_ICONS

C:\Program Files(x86)\KISSsoft 03-2014\bin32\icons

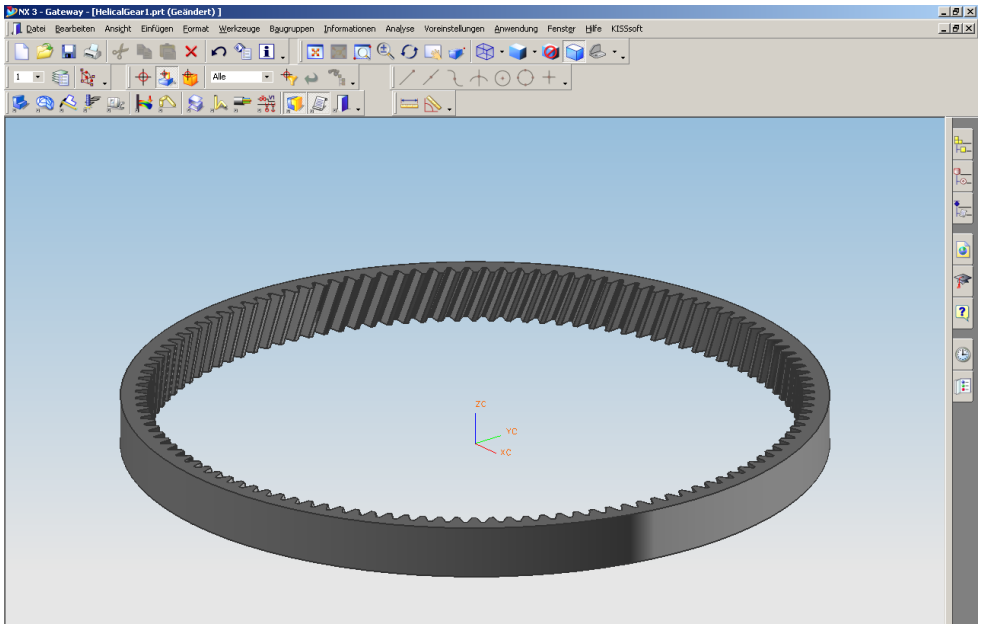
The `startup` folder also contains a `kSoftNX.ini` file, in which a part's layers, sketches, planes, and drafts, can be changed.



11.8.2 Running KISSsoft via the Add-in



Select the KISSsoft menu option to open all KISSsoft calculation modules directly. By doing this you can perform NX8 calculations (for example) in KISSsoft quickly and easily during the design process. The NX8 (etc.) menu options are inactive while KISSsoft is open. In order to reactivate the CAD program, you must close KISSsoft.



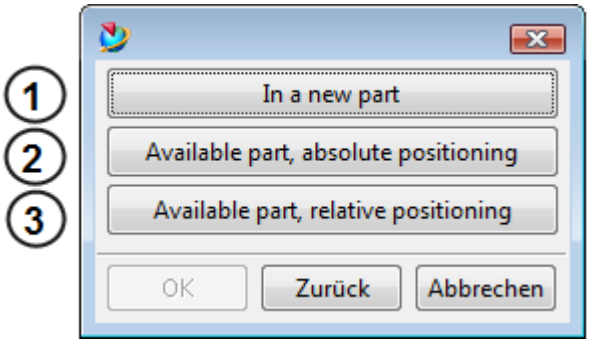
11.8.2.1 Gear teeth when existing shaft data is present

Prerequisites:

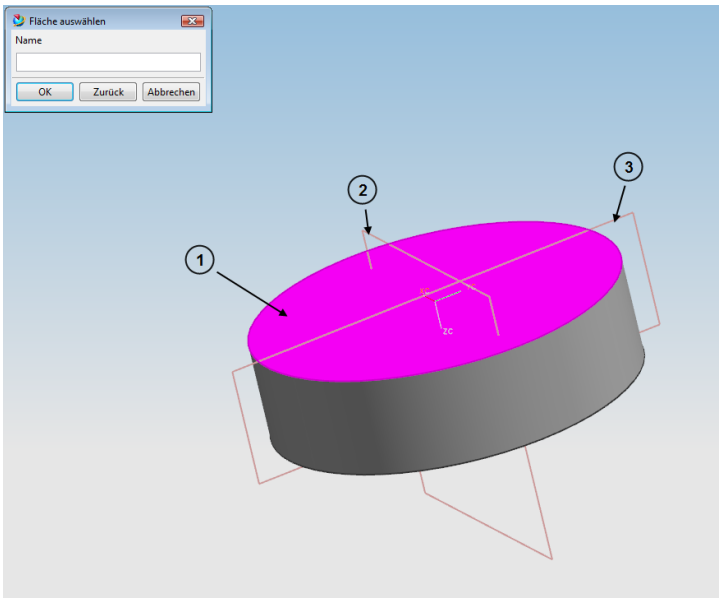
- The cylinder diameter must already have the same external diameter as the gear toothing before the generation process starts.
- In the case of internal toothing, a hollow cylinder must already be modeled before the gear teeth can be cut out.

For example, go to the KISSsoft menu in NX8 and select the cylindrical gear pair calculation. The procedure for generating the gear (see page I-192) is identical to the procedure for creating a new one.

If a part is already opened in NX (whatever version) when you return, the following list is displayed:



1. A new part opens, and the complete gear is generated.
2. If you select `Available part, absolute positioning`, only one side surface needs to be selected, on which the gear teeth are to be cut. For the generation, fixed levels will be generated, on which the gear teeth will be positioned.
3. If you select `Available part, relative positioning`, you can select a side surface and two levels (which will intersect the side surface). Consequently, the toothing can be positioned at relative planes (DATUM PLANE) and is not dependent on the absolute zero point. This positioning is mainly required in the case of the methodical operational behavior defined in the Master Model concept (Teamcenter).



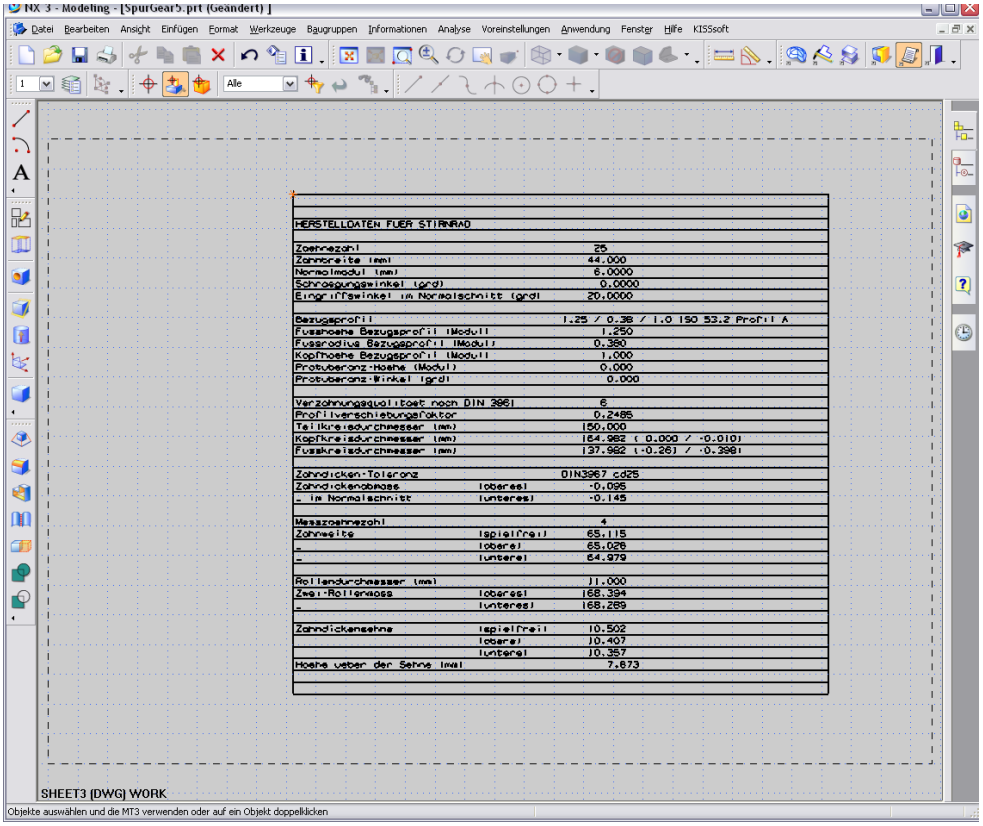
The generation of toothing on existing cylinders is performed on both inside and outside cylindrical gears with straight or helical toothing.

11.8.2.2 Adding manufacturing data to the drawing

You can use the `Add manufacturing data` menu option to insert a gear stamp of the current gear in a drawing.

- Teamcenter: If you are working in accordance with the `Master Model concept`, the features of the master part are displayed automatically in the non-master drawing when you call up `Add manufacturing data`.

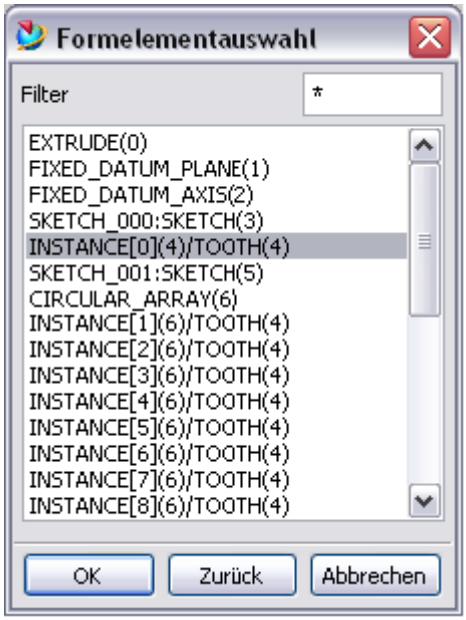
After you select this menu option, the following screen is displayed:



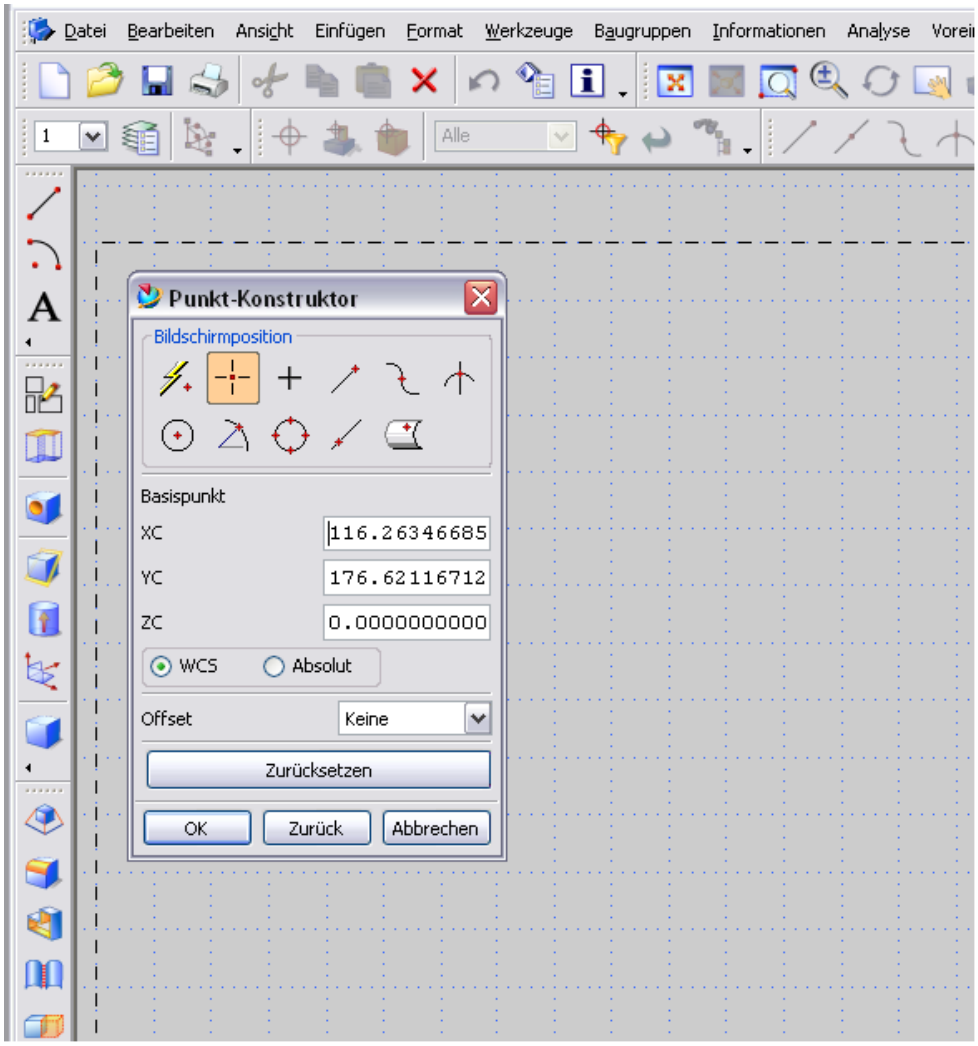
In this screen, select the following:

- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)

- Cylindrical helical gears/worms/bevel spur gears: TOOTH

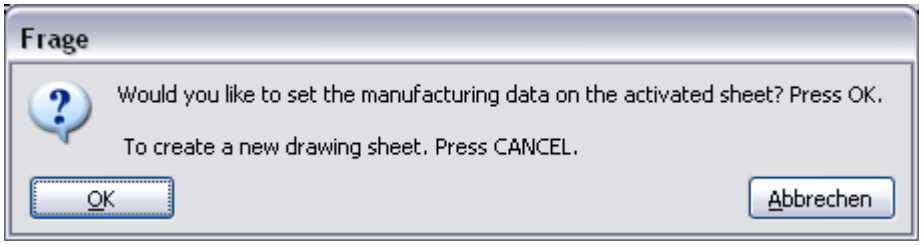


If you press OK, a new drawing will open and the following window appears:

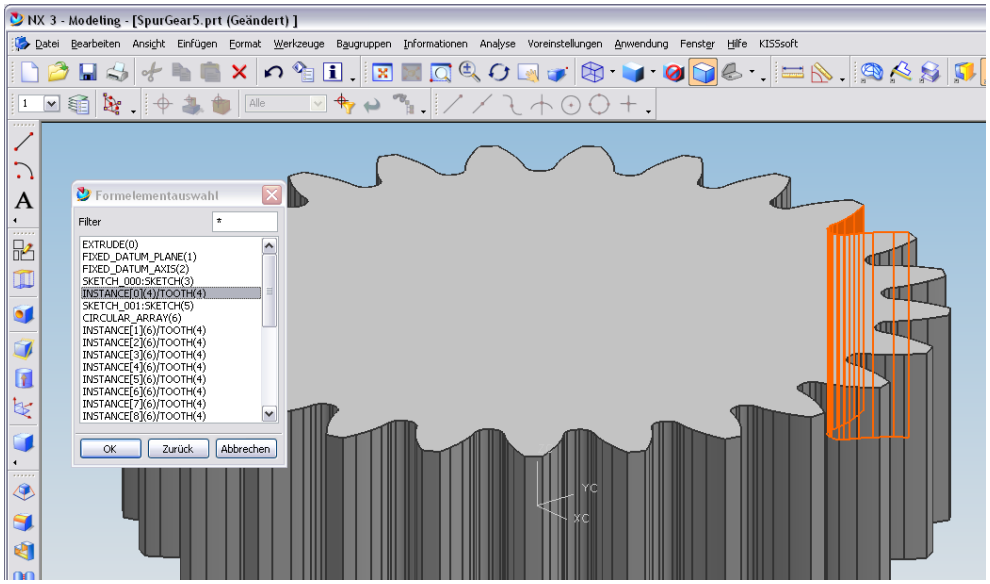


With one mouse click you can position the manufacturing data on the drawing. The mouse click will position the upper left corner of the table.

If you want to insert the data into an already existing drawing sheet, you must select the tooth space in the Drawing view if the required drawing sheet is opened. You will then see the screen in which you can select the tooth space, and are then prompted to decide if it should be inserted into the current drawing sheet.

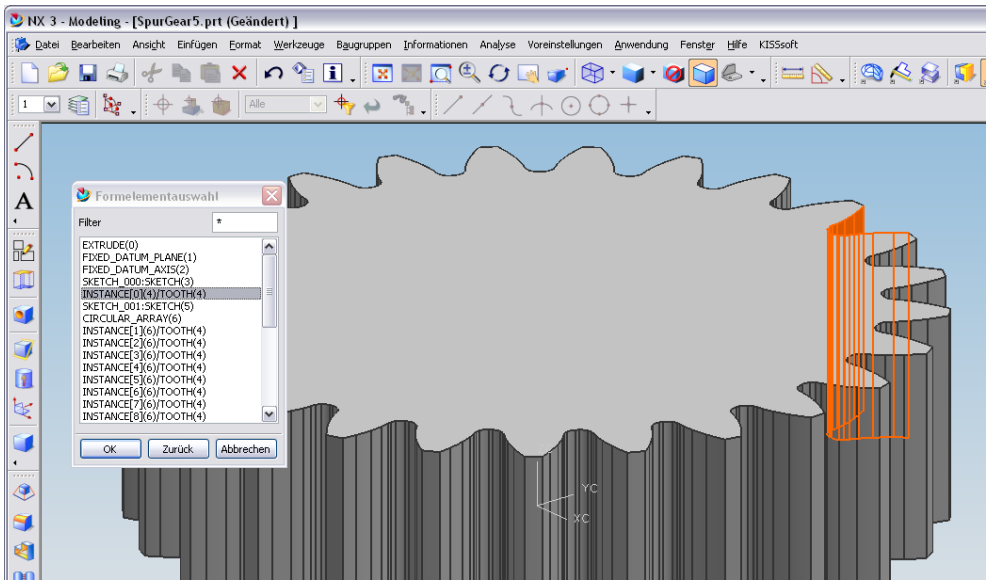


If you click OK you can use the mouse to position the current manufacturing data on the drawing. Click CANCEL to open a new drawing sheet in which you then can insert the manufacturing data.



11.8.2.3 Opening the calculation file

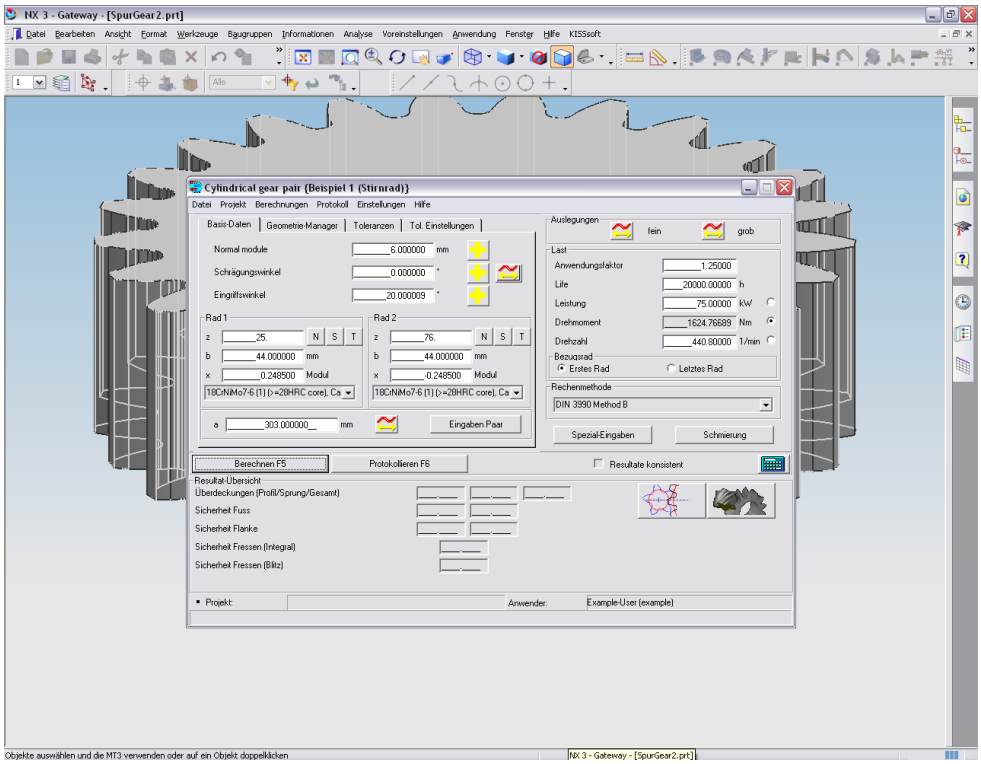
Select the `Calculation file` menu option to start KISSsoft and load calculation file for the toothing whose information is saved directly on the toothing Feature (tooth space). After you select this menu option, the following screen appears:



In this screen, select the following:

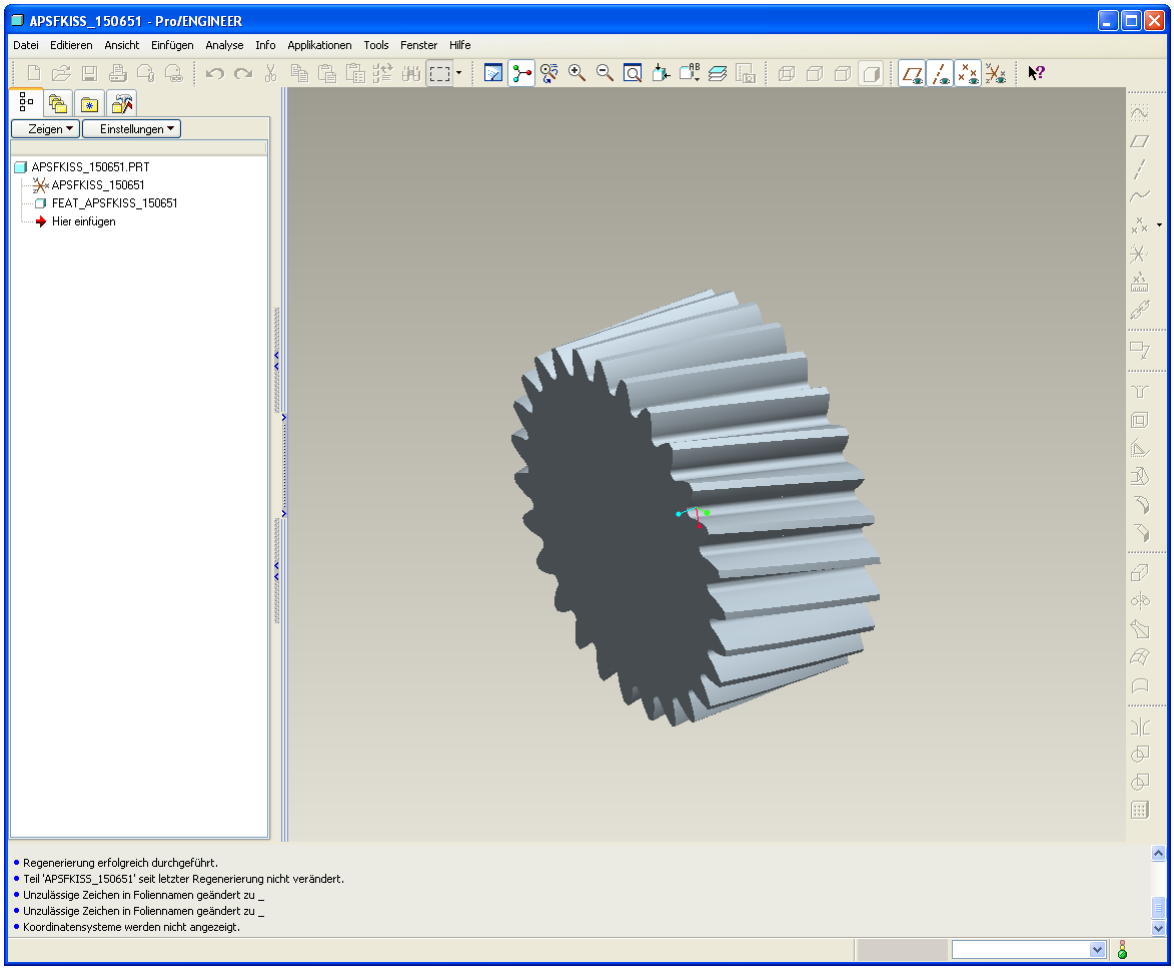
- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)
- Cylindrical helical gears/worms/bevel spur gears: TOOTH

If you then click OK, KISSsoft opens in the corresponding module with the toothing's calculation file loaded.

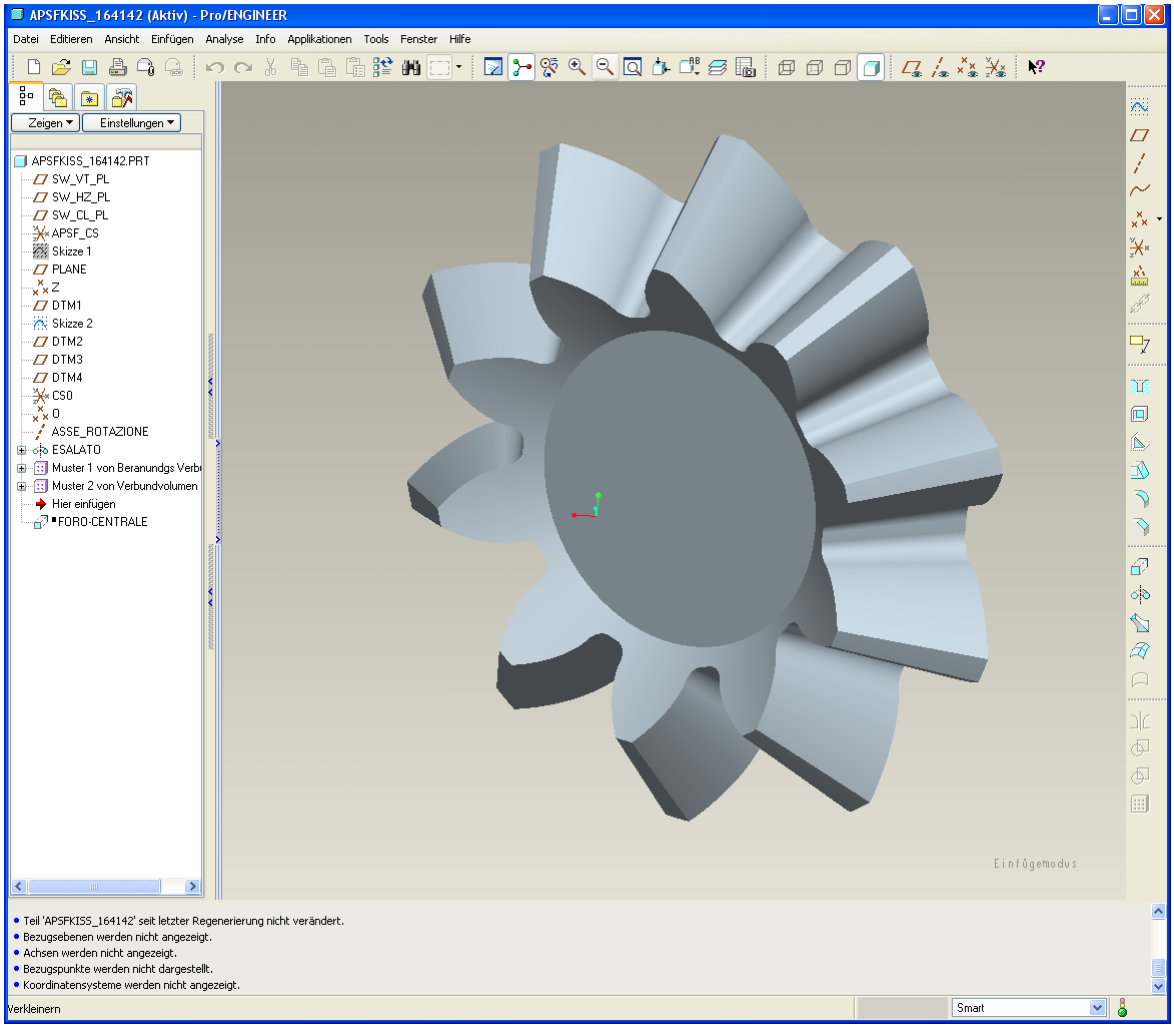


11.9 3D interface to Creo Parametric (ProEngineer)

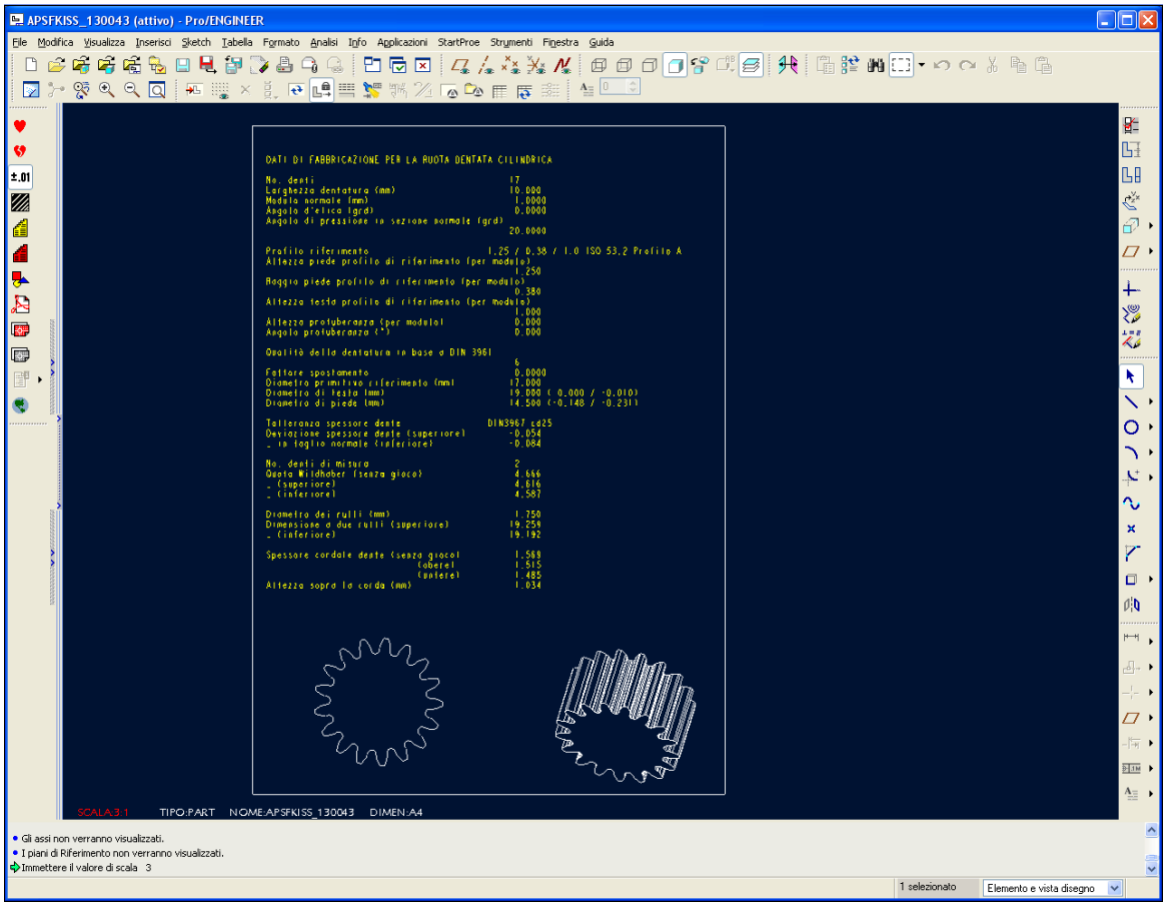
Manufacturer: Applisoft Europe (Italy)



The cylindrical gears or bevel gears calculated in KISSsoft can then be generated directly in Creo Parametric (ProEngineer) as a 3D part (see page I-192) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.



In addition to the part, the system opens a drawing into which the gear manufacturing data are inserted in a table. Open the CAD system before you start generating a part in the 3D interface to Creo Parametric (ProEngineer).



In the interface to ProEngineer, you can enter additional variables in the files for the particular gear (e.g. Z10GEAR1CAD.rpt) in the `CAD` directory. These additional variables will later be defined as parameters and saved in ProEngineer.
The parameters used for the generating process are already defined in ProEngineer and can no longer be used. Predefined parameters:

- pz, z, b, da, d, df, di, elica, USUnit

If you want to create a model of a part in US customary units (not metric), go to the `kiss.ini (see page I-60)` file and set the `USCUSTOMARYUNITS` parameter to 1.

You can also change an existing toothing without actually affecting the part (see Modifying a selected 3D model (see section "Modifying the selected 3D model" on page I-244)).

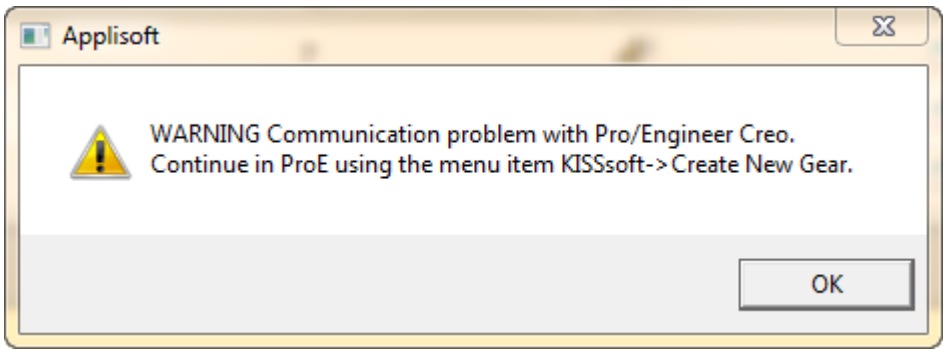
You can also cut a toothing on an existing shaft (see Cut intermeshing on an existing shaft (see section "Cutting teeth on an existing shaft" on page I-245)).

A new dialog opens as soon as you start the generating process:



here you can select what you require by clicking `Generate gear in new file` to generate the gear in a new part file.

If communications cannot be set up with ProEngineer, because the PRO_COMM_MSG.exe file is blocked, for example by a firewall or anti-virus program, the following message is displayed to tell you what to do, so you can still generate a gear:



NOTE:

You can either set your anti-virus program to permit the **pro_comm_msg.exe** and **apsfkissvb.exe** processes to run, or generate the gear directly in the KISSsoft menu in ProEngineer.

If you can also specify if you want to prevent the selection menu or message from being displayed (see Changing base settings in the interface (on page I-248)).

11.9.1 Integrating the KISSsoft Add-in

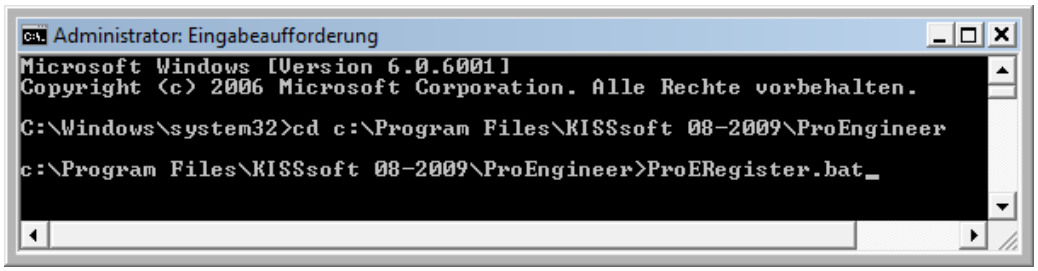
The KISSsoftCOM server should be registered as part of the installation process. However, if this did not happen, and the KISSsoft interface does not work, you must register KISSsoftCOM manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (`ProEngineer` folder). Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

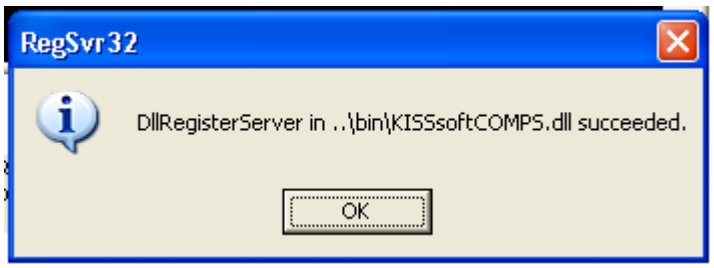


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

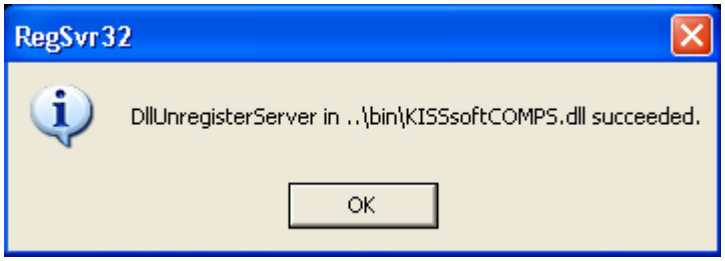
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the KISSsoftCom server has been registered successfully.

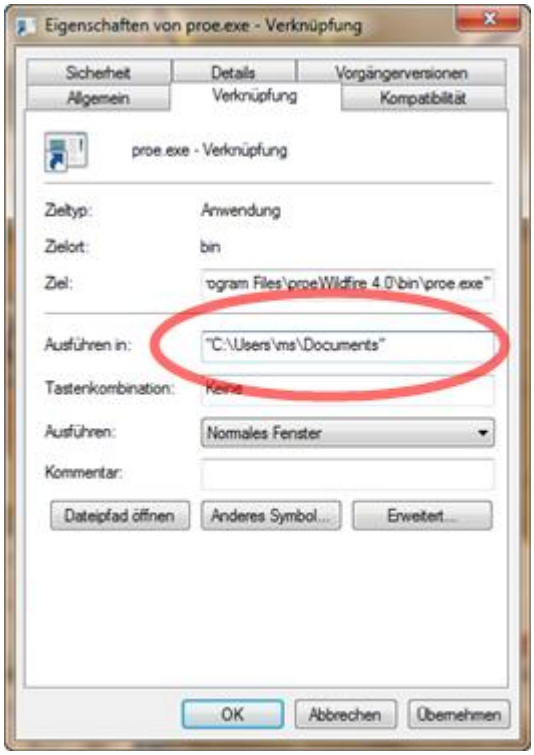


To remove the registration, double-click on the `ProEUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



Use one of the three following methods to ensure the KISSsoft menu is present every time you start ProEngineer/Creo:

1. You can copy the **Protk_EditGear_Creo1_64bit.dat** file (depending on the version of ProEngineer/Creo) to this Creo sub-directory: **.../Common Files/.../text/** (for ProEngineer, copy it into the **.../text/** sub-directory). Then rename the file to **Protk.dat**.
   Using this method, you can change your ProEngineer start directory to ensure that the KISSsoft menu always starts at the same time.
   If a different **Protk.dat** file is already present, you can add lines from the **Protk_EditGear_Creo1_64bit.dat** file to the **Protk.dat** file.
2. Copy the **Protk_EditGear_Creo1_64bit.dat** file into the ProEngineer Start working directory. Rename the file to **Protk.dat**.



This method requires you to copy the **Protk.dat** file to the Start directory (you will find the path displayed under Properties).

3. Then enter the following line in your **config.pro** file (in ProEngineer). (This is where you define your own path):
   protkdat C:\Program Files (x86)\KISSsoft 03-2014\ProEngineer\Protk_EditGear_Creo1_64bit.dat
   Select the path to the file that contains the name of your version of ProEngineer.
   This method saves you having to copy or rename any files.

Description of the content of the Protk.dat file:

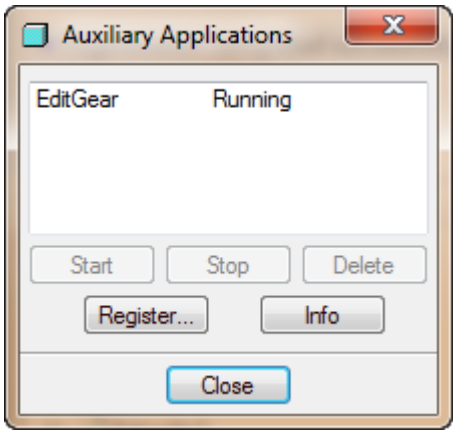
```
NAME EditGear

EXEC_PATH C:\Program Files (x86)\KISSsoft 03-
2014\ProEngineer\EditGear\bin_nt\EditGear_64bit_Creo1.dll

TEXT_PATH C:\Program Files (x86)\KISSsoft 03-
2014\ProEngineer\EditGear\text.GB

STARTUP DLL

ALLOW_STOP TRUE

UNICODE_ENCODING FALSE
```

END

EXEC_PATH and TEXT_PATH must be the absolute path of the installation.

STARTUP DLL and UNICODE_ENCODING FALSE are predefined (do not change them).

ALLOW_STOP TRUE Use this to stop the ProEngineer program (Tools->Auxiliary Application->Stop).



You can delete this line in the **Protk.dat** file to prevent users from stopping the interface.

NAME EditGear and END must be present, although you can change the **EditGear** name if required.

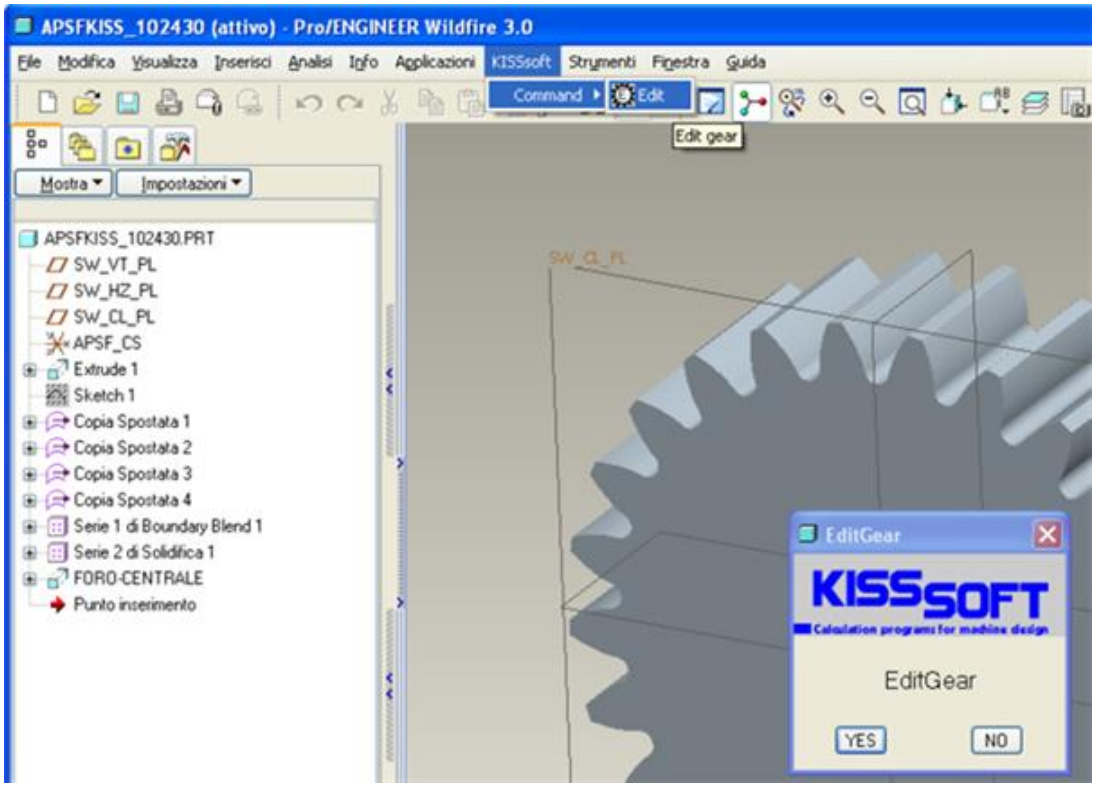
11.9.2 Modifying the selected 3D model

Every time you export a tooth form from KISSsoft, the model is generated in a new part in ProEngineer.

To modify an existing model:

1. Import the model you want to modify into ProEngineer (or use the current part)
2. Go to the KISSsoft menu and select Edit, and then click Yes (to import the existing toothing)

3. Then select `Open calculation file`. KISSsoft is then loaded with the corresponding toothing data.



KISSsoft can then regenerate the modified toothing, and therefore adapt the existing toothing.

11.9.3 Cutting teeth on an existing shaft

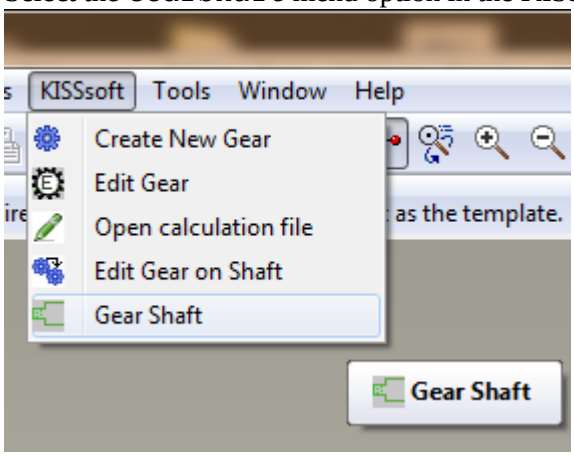
The following menu is displayed if you activate the KISSsoft 3D export:



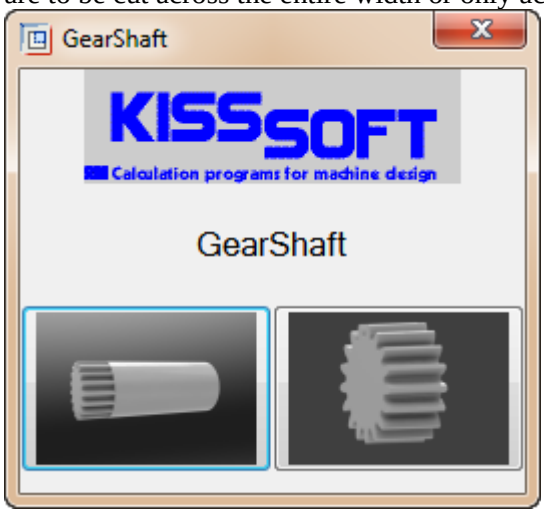
To modify an existing model:

1. Select `Generate gear on shaft`

2. In ProEngineer, open the shaft on which you want to cut the gear teeth.
3. Set a new system of coordinates to describe the point at which the gear teeth are to be cut. You can use this coordinate system if you want the gear teeth to be cut from the point of origin.
4. Select the `GearShaft` menu option in the KISSsoft menu in ProEngineer.



5. This opens another menu in which you can specify whether the gear teeth are to be cut across the entire width or only across part of the shaft.

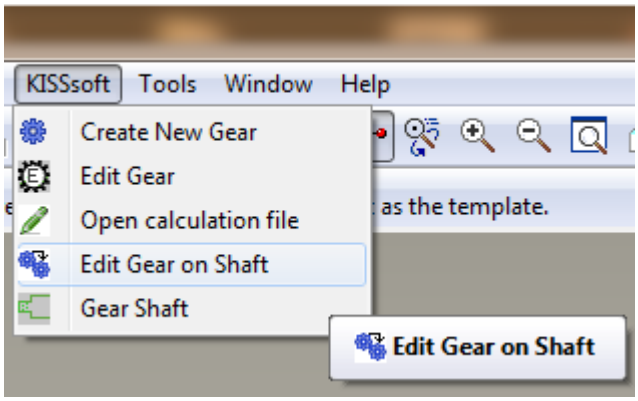


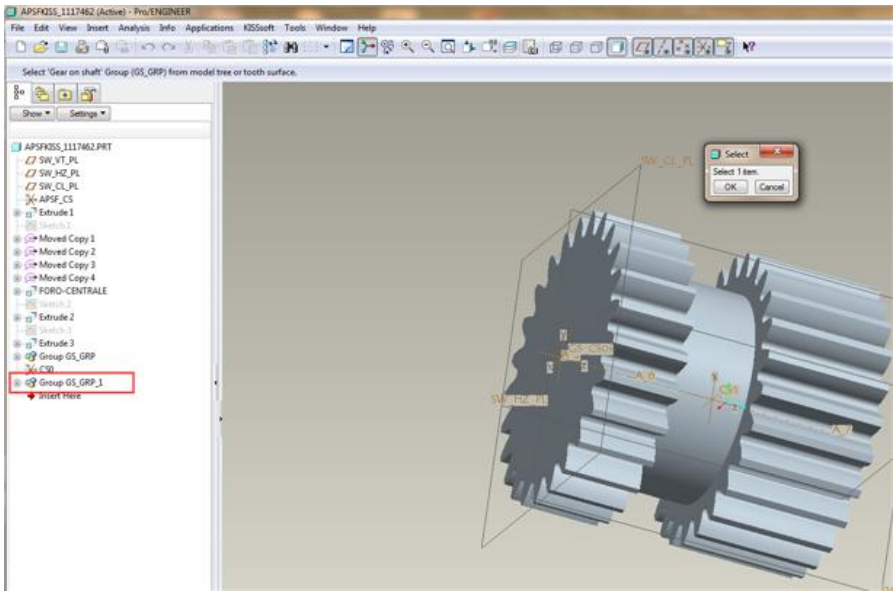
6. After you select the option you require, you can then select the coordinate system into which the toothing is to be inserted. (the co-ordinate system you select must have a Z-axis that is equal to the shaft axis).
7. The gear teeth are then cut on the shaft.

11.9.4 Modifying the teeth on an existing shaft

Follow these steps to modify the teeth of a gear on an existing shaft in a model that has already been generated (gear generated using the KISSsoft interface):

1. Import the model you want to modify into Creo Parametric (ProEngineer), or use the current part.
2. Select the `Edit Gear on Shaft` menu option to select the toothing you want to modify. The system then opens KISSsoft directly with the data that was saved for the toothing element when it was generated.

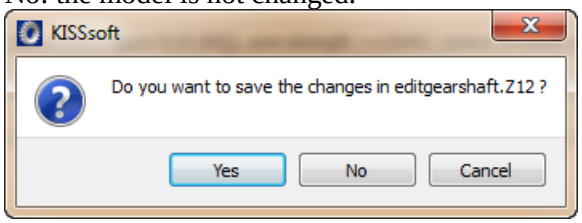




3. You can then modify or recalculate the toothing in KISSsoft, and restart the 3D export for corresponding toothing.
   Then, click on the cross at the top right to close the KISSsoft window.



You then see this prompt to save or not save the temporary calculation file.
Yes: the model is modified
No: the model is not changed.



4. The system now modifies the model if you confirm the prompt.

11.9.5 Changing base settings in the interface

You can set up your interface in a number of different ways. For example, you can tailor it by setting environment variables:

KISS_PROE_INTERFACE_NO_MENU = YES

This is designed for users who do not have a connection to Creo Parametric (Pro-Engineer) (via PRO_COMM_MSG.exe).

If you set this environment variable to "Yes", the interface will no longer attempt to use this process to manage the connection. You will also no longer see a warning that this connection is not possible.

KISS_PROE_INTERFACE_NO_MENU = NO

If you set this environment variable to "No", a warning is displayed if no direct connection to ProEngineer can be created.

The warning message describes what you must do to generate the gear even though the connection to ProEngineer is not present.

KISS_PROE_INTERFACE_CLASSIC = YES

The extra dialog in which you can select either "Generate gear in a new file" or "Generate gear on shaft" is not now displayed.

KISS_PROE_INTERFACE__CLASSIC = NO

A dialog is displayed in which you can select either "Generate gear in a new file" or "Generate gear on shaft".



If no environment variables are set, both these values are set to **NO**.

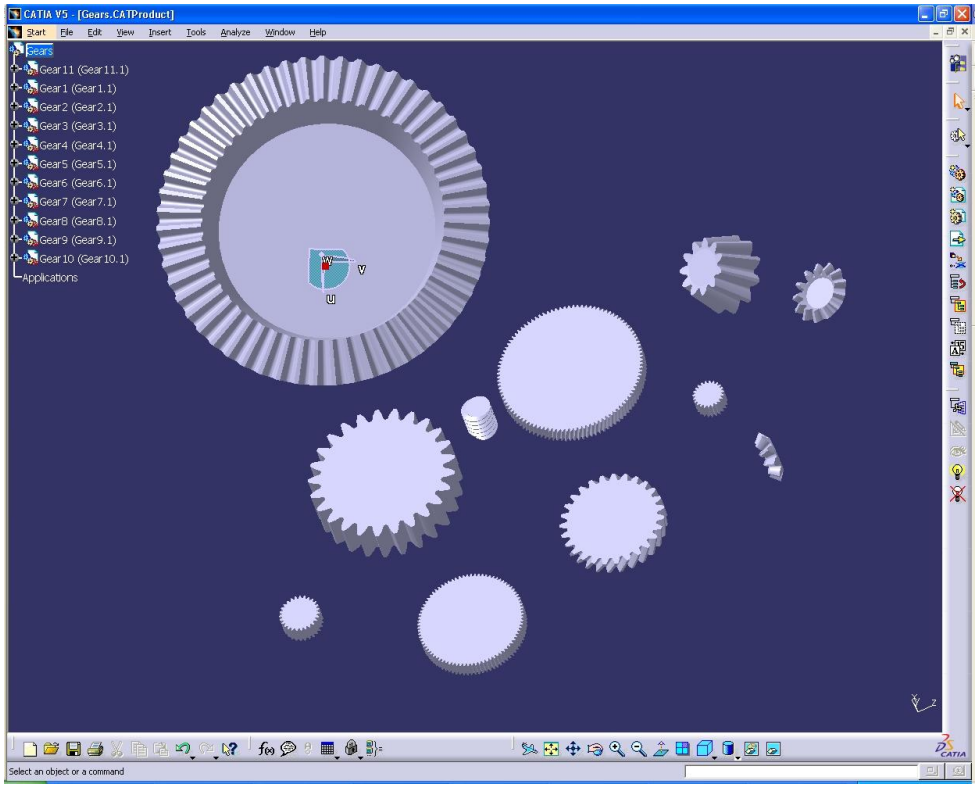
11.9.6 Save the files to the PTC Windchill working directory

To save the generated interface files to the PTC Windchill working directory in the PLM system, change the **WORK_IN_CURRENT_FOLDER** parameter setting in the **SETUP.txt** file from **NO** to **YES**. The **SETUP.txt** file is located in the '...\ProEngineer\EditGear\SETUP.txt' sub-folder, in the KISSsoft installation.

11.10 3D interface to CATIA

Manufacturer: SWMS (DE)

Face or bevel gears calculated in KISSsoft can be generated directly in CATIA as a 3D part (see page I-192) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.
Furthermore, you have the option of adding toothing to existing shafts at a later point in time.
You will find a more detailed description of the interface in a *.PDF file in the `CATIA` folder, in the KISSsoft installation folder.



11.10.1 Registering the interface

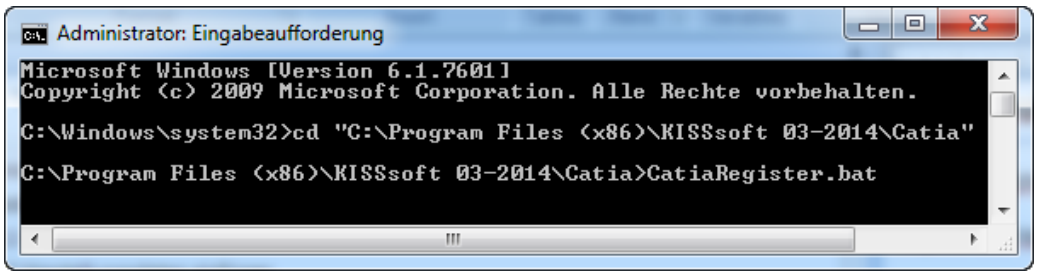
The CATIA interface should be registered as part of the installation process. If this did not happen, you must register the interface manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you can only perform the registration if you have administrator rights, you can only do this here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to where the registration file is to be executed (`Catia` folder). Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

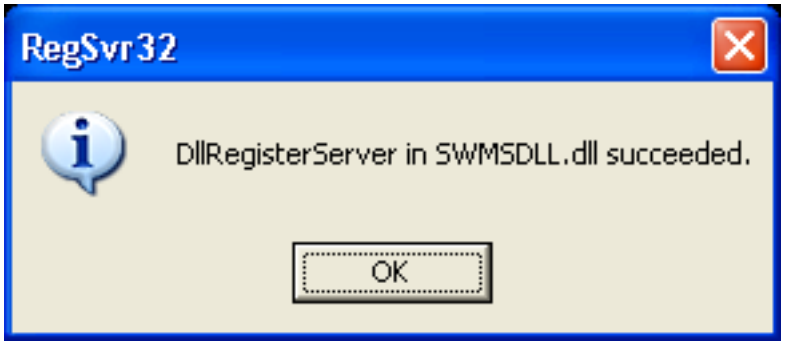


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the registration was successful.



To remove the registration, double-click on the `CatiaUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



Chapter 12
Answers to Frequently Asked Questions

12.1 Changing the output of angles in reports

Can you output angles (in calculations) in the KISSsoft angle report as degree values as well as decimal numbers?

Current form: ##.#### °

Required form: ## ° ## ’ ## ’’

To do this, modify the report template (*.rpt) accordingly. Refer to the Report templates (see page I-117) notes in the manual. The calculation is then performed in the report.

A helix angle is used to show this method:

Current form as a decimal:

Helix angle (grd) %11.4f{Grad(ZS.Geo.beta)}=>

Afterwards, the required form:

Helix angle (grd) %i° %i' %i" {Grad(ZS.Geo.beta)} {(Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60} {((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60-int((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60))*60}

12.2 Input materials for gear calculations in the database

When comparing the materials used for gears in a particular company, it became evident that not all the required materials were present in the database provided by KISSsoft.

In particular, the following key values, necessary for gear calculation, are missing. These include σ_{Flim}/S_{at}, σ_{Hlim}/S_{ac}, R_{zF}, R_{zH}, $B_{M.}$

When you redefine materials and their properties, you must compare them with similar materials in our materials database.

First of all, define the basic data for a material in the database. Then, define the gear-specific data for this base material.

Then, calculate the values of $\sigma_{Flim/Sat}$, $\sigma_{Hlim/Sac}$ depending on the hardness values, as described in ISO 6336-5.

To do this, you can use either the relevant material diagram, the conversion function for Own input for materials (see page II-274) or formulae from ISO. The S_{at}, S_{ac} values are converted on the basis of σ_{Flim}, σ_{Hlim}.

If you do not know the thermal contact coefficient BM, simply leave this entry blank so that the default values are used in the calculation.

For medium total heights, specify average values with R_{zF} 10µm and R_{zH} 3µm. You will find more detailed information in ISO 6336-2.

You will find more information about the influence of medium total heights in point 2 of our article at http://www.kisssoft.ch / deutsch / downloads / doku_artikelISO6336Neuheiten.pdf.

12.3 How can I test the software?

A demo version (see page I-43) of the software is available. Although the demo version does not have an expiration date, its functionality is limited so that, for example, you cannot change and store material data. The demo version is designed to give you an initial impression of the software. For a detailed trial, request a test version (see page I-45). The test version runs for 30 days, is free of charge and is the same as the full version (without third party programs).

12.4 What licenses are available?

Individual user licenses and floating licenses are available for both KISSsoft and KISSsys. A floating license enables the software to be used at more than one workplace.

However, floating licenses are not available for some of the third party products, for example, some CAD interfaces.

12.5 Add your own texts in the results window

To do this, define a new file in the KISSsoft installation folder in "…\ext\rpt\". This file must then be named using this convention: "Modulname + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values then appear at the end of the "Results" window.

12.6 Restoring the previous stages of the calculation

(Select File > Restore... (acts like the Undo function) to retrieve an earlier state of the current calculation file. For this reason, every calculation run stores the current state as a restore point. The list of restore points is deleted when you open a different file.

Part II
General

Chapter 13
Introduction

KISSsoft provides calculation modules for different toothing types, ranging from cylindrical gears in different configurations to bevel gears and face gears to worm wheels. The input windows for the different gear calculations are very similar. There are also calculation options for multiple modules. The table below shows you all the input windows in the individual calculation modules.

Input window	Section											
Basic data	14.2	is supported by all calculation modules										
Rating	14.3											
Factors	14.4											
Reference profile	14.5											
Tolerances	14.6											
Modifications	14.7											
Tooth form	14.8											
Tooth flank fracture	14.9		✔	✔				✔				
Contact analysis	14.10		✔	✔	✔	✔	✔	✔				
Operating backlash	14.12		✔	✔	✔	✔	✔			✔	✔	
Master gear	14.13	✔	✔	✔	✔	✔					✔	
AGMA 925	14.14		✔	✔	✔	✔						

Table 13.1

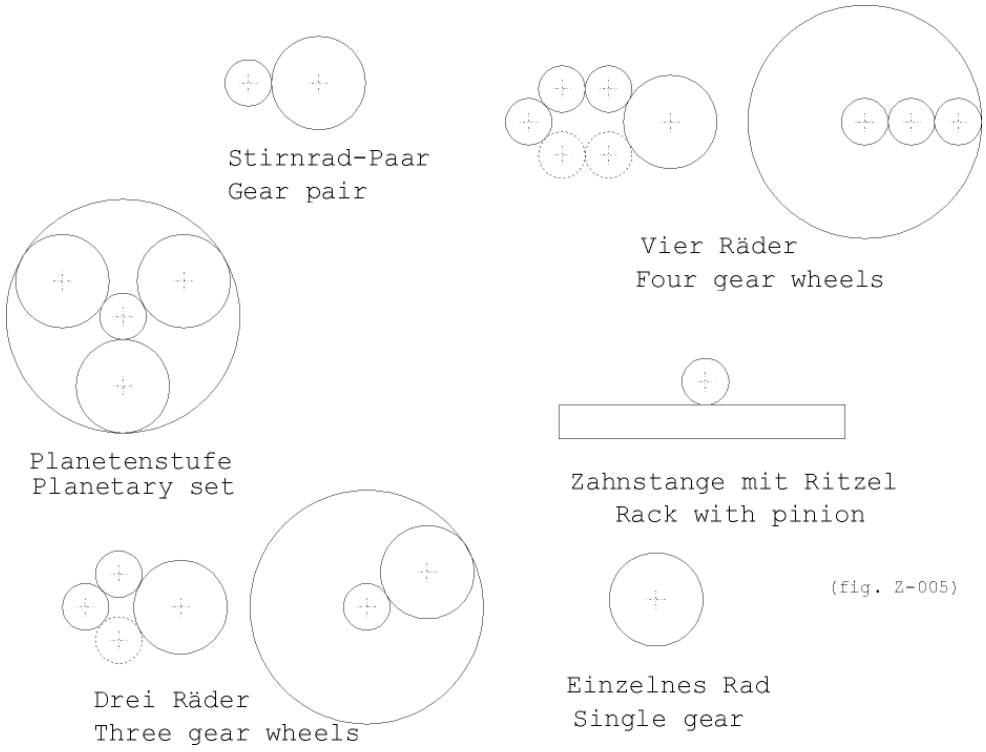
- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear, - Three gears, - Four gears, - Bevel and Hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears and precision mechanics worms, - Splines (Geometry and Strength)

Chapter 14
Cylindrical Gears

You can use KISSsoft's cylindrical gear calculation software to calculate a range of different configurations.

- The single gear calculation has been developed to calculate the geometry and test dimensions of individual gears
- The cylindrical gear pair is the most important configuration for geometry and strength. You can also use it for additional calculations and several individual calculations at the same time.
- The planetary gear software checks the practical aspects of the configuration and monitors both pairs of gears while they are being sized. The Fine Sizing function enables you to optimize the center distance quickly and efficiently. You can usually input your own values here. However, you must take into consideration that, as torque cannot be applied to the planet, it is not possible to perform a strength analysis on a Wolfrom drive or on a Ravigneaux gear set.
- The configurations for three and four gears enable you to calculate a gear wheel chain, in which torque is applied only to the first and last gear.
- The calculation used for a rack and pinion only includes one rack in the geometry calculation and one cylindrical gear with a large number of teeth for the strength calculation.

As the input screens for the different configurations are very similar, they are described together in the sections below.


Stirnrad-Paar
Gear pair
Vier Räder
Four gear wheels
Planetenstufe
Planetary set
Zahnstange mit Ritzel
Rack with pinion
Drei Räder
Three gear wheels
Einzelnes Rad
Single gear

(fig. Z-005)

14.1 Basic data

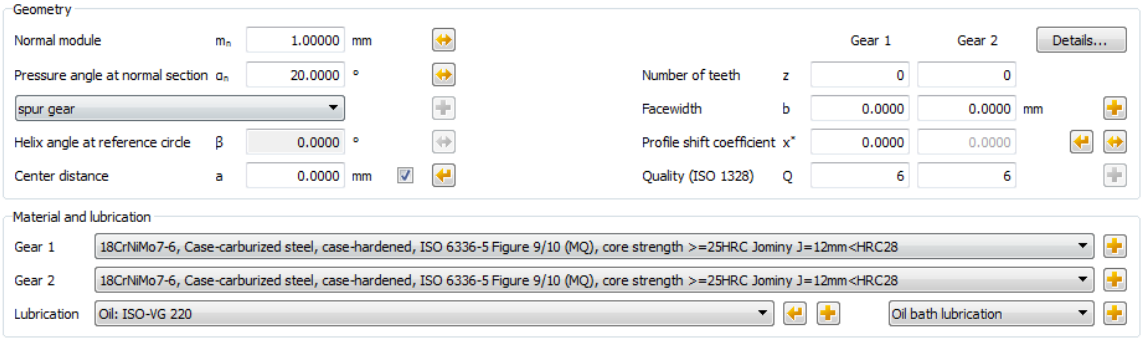


Figure 14.1: `Basic data` input window for cylindrical gear pair

The `Basic data` input window is one of the standard tabs (see page I-89) and is subdivided into the two groups `Geometry`, `Material and Lubrication`.

14.1.1 Hand of gear for gear teeth

Hand of gear for gear teeth (→ see Figure on page II-265) defines the direction of the axial forces. Helical gear teeth usually produce less noise than spur gear teeth, but it generates an additional bending moment and an axial force. A gear with continuous double helical teeth consists of two halves of a helical gear with a different hand of gear. Although it does not generate any axial forces, it must be possible to adjust the gear along its axis and it is more difficult to manufacture. In a herringbone gear (with continuous double helical teeth), click the button to set the gap width b_n.

14.1.2 Normal module

Enter the normal module. The normal module defines the size of the teeth. A standard series is for example defined in DIN 780 or ISO 54. However, if you know the pitch, transverse module or diametral pitch, instead of the normal module, click the button to open a dialog window, in which the conversion will be performed. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

14.1.3 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard toothings, the pressure angle is $\alpha_n = 20^o$. Smaller pressure angles can be used for larger numbers of teeth to achieve higher contact ratios and insensitivity to changes in center distance. Larger pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

14.1.4 Helix angle at reference circle

Enter the helix angle in [°]. Click the  button in the `Convert helix angle` window to calculate this angle from other values such as, for example, the overlap ratio and axial force.

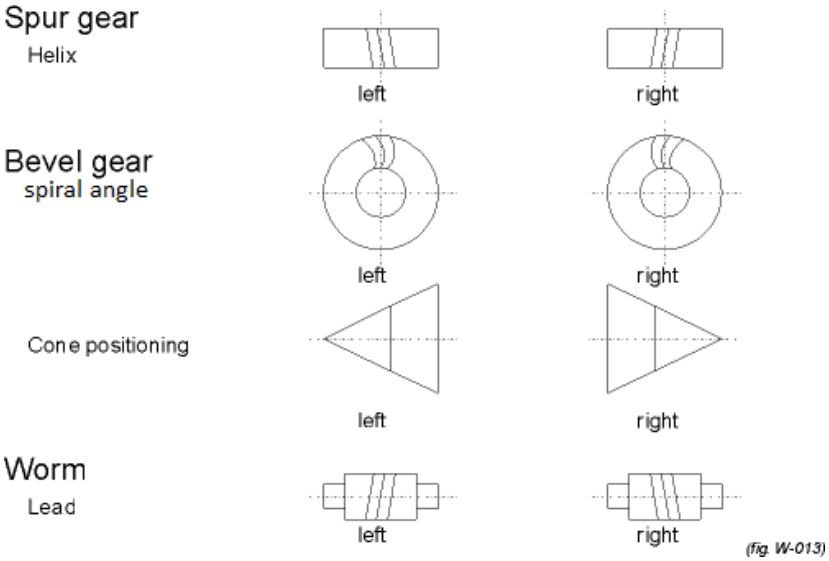


Figure 14.2: Helix angle at reference circle.

14.1.5 Center distance

As stated in ISO 21771, the center distance for external and internal teeth is positive for two external gears and negative for an external gear paired with an internal gear. For internal toothing, the number of teeth on the internal gear and the axis center distance are always negative.

If you select the checkbox to the right of the axis center distance unit, the value used in the calculation will remain constant. Otherwise, the center distance will be calculated from the profile shift total.

Click the button to select one of the following sizing options:

- Fixed sum of profile shift coefficients. The center distance is calculated on the basis of a predefined profile shift sum. Click the button to display a suggested value for the profile shift sum (as defined in DIN 3992). The sum of profile shift influences the profile shift coefficients of both gears as well as the operating pitch circle and the operating pressure angle.
- Fixed profile shift coefficient Gear 1 (or 2), balance specific sliding. Optimize axis center distance with respect to balanced sliding: For a specified profile shift of a (selectable) gear, this option calculates the axis center distance in such a way as to balance gear pair specific sliding (for cylindrical gears). If the `Own input` menu option is not selected from the Own input drop-down list in the `Reference Profile` input window, this calculation is performed with automatic tip alteration as specified in DIN 3960. You can also enter your own tip alteration value in the `Basic data` input window by clicking the `Details...` button. In the `Define geometry details window`, select the checkbox next to the `Tip alteration` input field.

14.1.6 Number of teeth

The number of teeth is, by default, a whole number. You can also enter the number of teeth as an amount with values after the decimal place (see section "Input of number of teeth with decimal places" on page II-458). For internal toothed gears, you must enter the number of teeth as a negative value as stated in ISO 21771. For a pinion-ring internal gear gear pair, the center distance must also be entered as a negative value (e.g. $z_1 = 20$, $z_2 = -35$, $a = -7.5$, $m_n = 1$).

The minimum number of teeth is limited by geometric errors such as undercut or tooth thickness at the tip. For spur gears without profile shift there is for example undercut if there are fewer than 17 teeth.

14.1.7 Facewidth

Normally the facewidth shouldn't be greater than 10 to 20 times the normal module, or also not greater than the reference circle of the pinion. The contact pattern deteriorates if the facewidth is too large. Click the button to the right of the facewidth input field to enter the axial offset b_v (see also Figure 14.3). The axial offset reduces the effective width for the strength calculation. The common width

is used to calculate the pressure. A certain amount of overhang is taken into account for the Tooth root strength. The selected pinion width is often somewhat greater than the gear width.

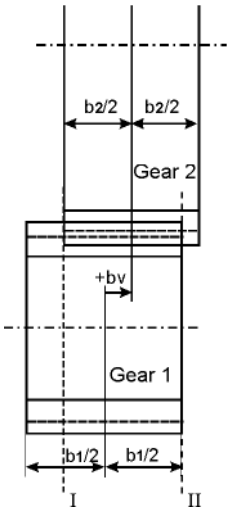


Figure 14.3: Axial offset b_v

In double helical gears[2] you must specify the total width of the gear teeth (i.e. the width of both halves together with the gap). To enter the width of the gap b_n, click the  button on the right of the hand of gear drop-down list for the toothing.

14.1.8 Profile shift coefficient

Preliminary note: If the profile shift sum has not yet been specified, click the Sizing button (), to the right of the center distance (see page II-265) input field, to display a suggested value for the distance in the `Sizing center distance` window. The suggested value is based on DIN 3992 recommendations for well balanced toothing (Area P4/P5). You will find more information about this in DIN 3992 or in Niemann [64], Fig. 22.1/6.

The tool can be adjusted for manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and

[2] *Double helical gears are gears that consist of two gear halves; the first half is angled to the left and the second half is angled to the right.*

undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

The distribution of the total profile shift affects the tooth thickness, sliding movements and strength values. It can be performed according to a range of different criteria. To achieve this, use the various sizing options provided by clicking the  button in the `Sizing of profile shift coefficient` window.

- For optimum specific sliding
  The value suggested here shows the profile shift for a cylindrical gear pair that has been balanced for a specific sliding between the pinion and the gear. When more than two gears are involved, the profile shift coefficient is set to the smallest value that corresponds to the specific sliding movement at the root.
- For minimum sliding velocity
  The minimum sliding velocity at the tip of the two gears is often used for speed increasing ratios. In a cylindrical gear pair, this means both gears have the same sliding velocity and that the access and recess length of the path of contact are also the same.
- For maximum root safety
  The profile shift coefficient is defined iteratively for the interval x^*_{min}, x^*_{max}.
- For maximum flank safety
  The profile shift coefficient is defined iteratively for the range x^*_{min}, x^*_{max}.
- For maximum scuffing safety
  The profile shift coefficient is defined iteratively for the range x^*_{min}, x^*_{max}.
- For gear 1 without undercut and point at tip (min)
  The minimum value of the profile shift coefficient for gear 1 is calculated from the undercut boundary of gear 1 and the minimum top land of gear 2.
- For gear 1 without undercut and point at tip (max).
  The maximum value of the profile shift coefficient for gear 1 is calculated from the minimum top land of gear 1 and the undercut boundaries of gear 2.
- For undercut boundary per gear.
  The proposed value only refers to the selected gear. No check is performed to see whether the resulting profile shift is also permitted for the other gear in the pair. For more information, please refer to the explanations above.
- For minimum top land per gear.
  The proposed value only refers to the selected gear. No check is performed to see whether the resulting profile shift is also permitted for the other gear in the pair. You can specify the minimum thickness of the top land in `Calculation > Settings > General > Coefficient for minimum tooth thickness at the tip`. For more information, please refer to

the explanations above.

NOTE:

The sizing button is deactivated if the 'Do not change tip circle in case of profile shift' or 'Do not change root circle in case of profile shift' option is activated.

Click the  button, and KISSsoft will determine whether the profile shift coefficient is taken from measured data or from values given in drawings.

The following options are available here:

- **Base tangent length**
  Here you must enter the base tangent length (span) and the number of teeth spanned. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.
- **Measurement over two balls**
  To do this, enter this dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins, see Measurement over pins.
- **Measurement over 2 pins**
  To do this, enter this dimension and the diameter of the pin. For helical gear teeth and gears with an odd number of teeth, you must also enter a minimum span, so the measurement can be performed. The measurement over pins cannot be measured in internal helix gears.
- **Measurements over 3 pins**
  Here, enter the measurement over pins and the pin diameter. For helical gear teeth and gears with an odd number of teeth, this is equivalent to the measurement over 2 rollers. You cannot use this option for internal and helical gear teeth, or gears with an even number of teeth.
- **Tip circle**
  This is a rather imprecise calculation because the tip diameter does not always depend solely on the profile shift.
- **Tooth thickness at reference circle**
  Here, you specify the tooth thickness. You can also enter the arc length or chordal length, and whether the value is in transverse or normal section.

NOTE

If one of the two profile shift values appears in gray, this means it will be calculated by calculated. This is what happens when you activate the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for one of the other gears. calculated.

14.1.9 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The accuracy grade specified in ISO 1328 is approximately the same as in DIN 3961 or BS 436/2.

The accuracy grades that can be achieved are displayed in the Quality values (see Table "Quality" on page IV-935) table.

Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 14.1: Accuracy grades for different manufacturing processes

Conversion of accuracy grades according to AGMA:
When converting accuracy grades as defined in AGMA 2015-1-A01, Annex B.2, the total of the accuracy grade figures in version 2015 (comparable with ISO) and version 2000 equals 17.

Quality as specified in ISO 1328 and AGMA: 2015	**Q. according to AGMA 2000**
1	16
2	15
3	14
4	13
5	12
6	11
7	10
8	9
9	8
10	7
11	6

Table 14.2: Accuracy grade values in different standards

If you want to define different tolerances, click `Calculation->Settings>General` and set the `Varying qualities` flag.

This activates the Plus button next to Quality in the main screen. Click the Plus button to open a new window, in which you can enter the tolerances you require.

You can input the tolerances in standard-specific tabs. The changes in the window are then applied to all the gears in the calculation module.

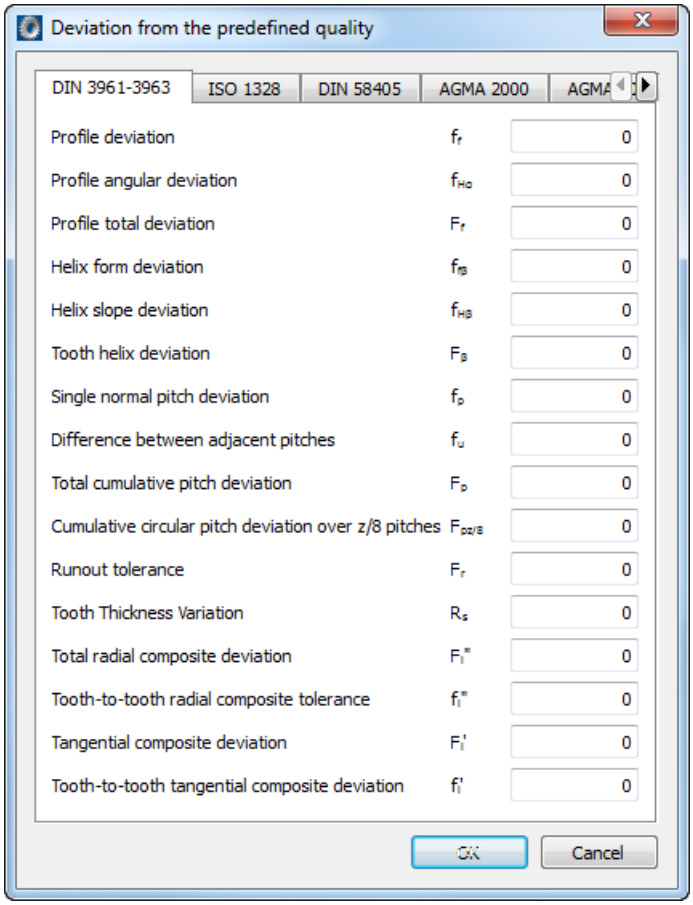


Table 14.3: Input window for different tolerances

This is the table in which you input any deviation from the base accuracy grade (specified in the "Basic data" tab). Example: The base accuracy grade of gear 1 is 6. If you then input +2 for the runout, the runout will be calculated with an accuracy grade of 8.

In every case, only tabs (standards) that are possible for the calculation module are displayed.

The user entries remain in this window as long as you continue using the same calculation module. You can therefore import a different file, and set the flag.

The same entries will still appear in the window next to the Plus button. You only need to input the data again if you change calculation module.

14.1.10 Geometry details

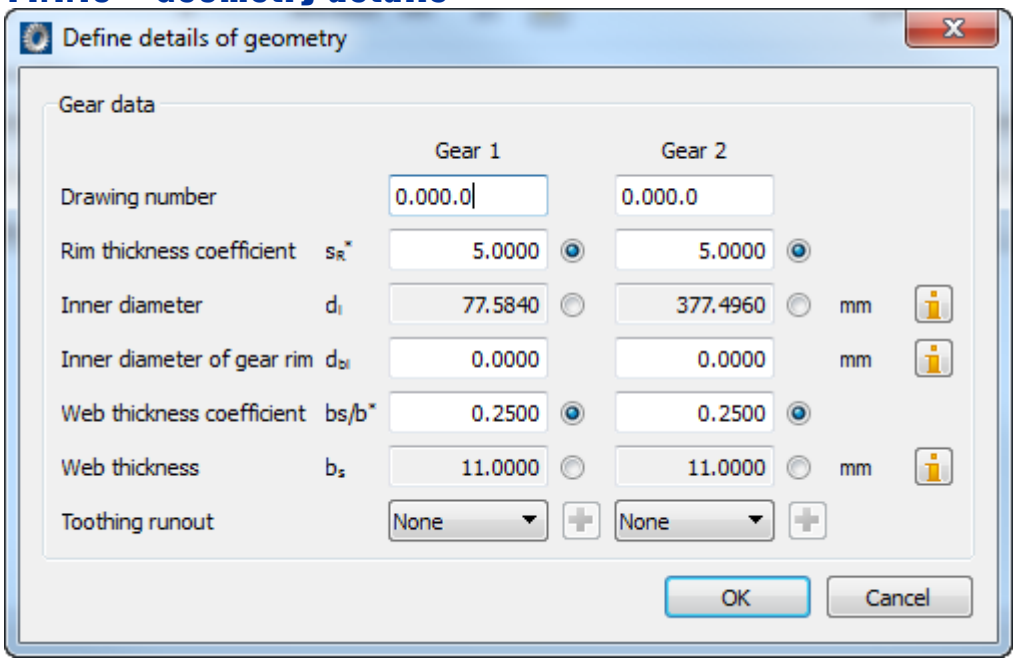


Figure 14.4: `Define details of geometry` input window for a cylindrical gear pair

To open the `Define geometry details` window, click the `Details...` button in the upper right-hand part of the Geometry area. Here you can change the values for:

- Drawing number
- Rim thickness coefficient S_R^*
- Inner diameter d_i
- Inside diameter of rim d_{bi}
- Web thickness factor b_s/b^*
- Web thickness b_s

The drawing number is only used for documentation purposes. You can enter any text here.

The inside diameter is needed to calculate the mass moment of inertia. For solid wheels, enter 0, for external wheels with a web, enter the corresponding diameter d_i as shown in Figure 14.4. For internal wheels, enter the external diameter of the

gear rim. The inside diameter can either be defined by entering d_i or the rim thickness coefficient S_R*.

According to ISO or AGMA, the gear rim thickness s_r, defined by the inner diameter of rim d_{bi}, affects the strength. If no gear rim thickness is present, you can enter a value of 0 for d_{bi}. In this case, the gear rim thickness s_r will be determined from the diameter d_i. If a diameter for gear rim d_{bi} has been entered, the effective gear rim thickness S_r is derived from $(d_f - d_{bi})/2$. The gear rim thickness S_r will be output in the report. Where thin gear rims are used, this factor can greatly influence the calculation of safety factors. For thin gear rims, this value can also be calculated in accordance with VDI 2737 (see page II-470).

Web thickness factor: If the inside diameter is <> 0, the value input for the web thickness (b_s or b_s/b) is taken into account. If $b_s/b = 1.0$, this means no web is present. In this case, the gear body coefficient CR is 1.0. The ratio b/b_s can vary between 0.2 and 1.2. In this case, CR is then < 1 (if $b/b_s < 1$) or > 1 (if $b/b_s > 1$). The coefficient CR is then used to calculate the tooth contact stiffness ($c\gamma\alpha$).

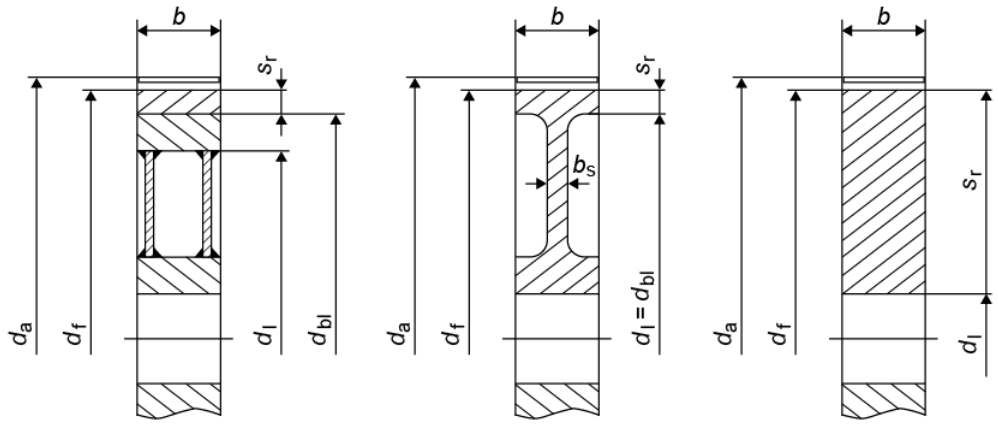


Figure 14.5: Dimensioning the diameter.

14.1.11 Materials and lubrication

14.1.11.1 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own input` from the list or enter the material in the database (see section "External tables" on page I-136) first. Click the  button next to the materials drop-down

list to open the `Define material, Gear 1(2)` window, in which you can select the material you require from the database list of available materials. Select the `Own input` option to enter specific material characteristics. This option corresponds to the `Create a new entry` window in the database tool.

Strength calculation with normal gear materials:
The cylindrical gear strength calculation formulae defined in ISO 6336, DIN 3990 or AGMA 2001 only involve specific (most commonly used) materials and treatment methods: These are:

- Through hardening steel
- Case hardening steel
- Nitriding steel
- Structural steel
- Grey cast iron with spheroidal graphite
- Cast iron with flake graphite

Strength calculation with unusual gear materials (not taken into account in standards):

- Stainless steel
- Free cutting steel
- Aluminum and bronze alloys

KISSsoft handles these materials in the same way as heat treatable steels. This affects a range of less important values that are used to calculate the permitted tooth root and flank strength: factors Y_{NT}, Y_{drelT}, Y_{RrelT}, Y_X, Z_{NT}. The endurance limit values σFlim and σHlim must either be measured or already be known. The S-N curve (Woehler lines) must be defined and used to achieve more accurate calculations.

- Sinter
  According to information from the company MIBA (A), sinter has similar properties to GC. For this reason, all the factors specified in DIN or ISO, which depend on the material type, are determined for sinter according to all the formulae that are applicable for GC.

Plastics

The strength of plastic gears can be calculated either according to Niemann VDI 2545 or VDI 2736. The material properties (Young's modulus etc.) and the permitted tooth root and flank stresses are greatly affected by the temperature and the type of lubrication. This is why calculating the characteristics for plastic gears requires so much time, effort and experience, especially if only very little material data is available. VDI guideline 2736 lists the tooth root and flank strengths for a number of basic materials:

- Tooth root strength: POM, PA 12, PA66, PET, PE, laminates
- Flank strength: PA 12, PA6, PA66, PBT, laminates
- Tensile fatigue strength for worms: POM, PA46, PA66, PEEK

Materials manufacturers also provide gear data that can be used to calculate the strength of plastic gears. If requested, KISSsoft can also provide the relevant material files.

KISSsoft users can also add their own material data to the plastics database. The corresponding DAT file contains specific data for each material. The user can then edit the DAT files to calculate plastic gears using the values for their own materials.

As defining the permitted root and flank limiting values takes so much time and effort, and because these values are often not present, KISSsoft can also perform the calculation using a limited data volume (for example, static calculations).
As additional information, the name of the plastic includes an overview of the data that is available for calculating plastic gears.

The data used to calculate plastic gears is available in this format: **[S B Fog Wd]**.

Abbreviations used here:

S - the ultimate or yield material strength of the material is provided for calculating static root strength

B - S-N curves (Woehler lines) are provided for calculating the root endurance limit

F - S-N curves (Woehler lines) for all lubricant types are provided for calculating the tooth flank endurance limit

Fo - S-N curve (Woehler lines) for oil lubrication are provided for calculating the tooth flank endurance limit

Fg - S-N curves (Woehler lines) for grease lubrication are provided for calculating the tooth flank endurance limit

Fd - S-N curves (Woehler lines) for a dry run are provided for calculating the tooth flank endurance limit

Fog - S-N curves (Woehler lines) for oil and grease lubrication are provided for calculating the tooth flank

W - Wear coefficients for all types of lubrication are provided for calculating wear

Wo - Wear coefficients for oil lubrication are provided for calculating wear

Wg - Wear coefficients for grease lubrication are provided for calculating wear

Wd - Wear coefficients for a dry run are provided for calculating wear

WORM - S-N curves (Woehler lines) are provided for calculating the root endurance limit for crossed helical gears

NOTE:

When a calculation method according to Niemann or VDI is selected, the tooth root, tooth root and wear are calculated automatically, if the data for the calculation is present. If no data is present for one or more of these methods, only the calculations for which data is available are actually performed.

Converting hardness to endurance limit values σHlim, σFlim

When you enter data for your own material, the hardness can be taken for conversion into the endurance limit values σHlim, σFlim. To open the conversion dialog, click the appropriate conversion button next to the input fields for the endurance limit values σHlim, σFlim. The data is converted in accordance with the ISO 6336-5:2003 formula described in section 5.

(The data for forged steels is used for "unalloyed/through hardened" and "alloyed/through hardened" heat treatable steels.)

σHlim, σFlim=A*x+B

x: Hardness value in the unit used in the table (depending on the HV or HBW material type)

A,B: Factors for the particular material type and processing. (from Table 1, ISO 6336-5)

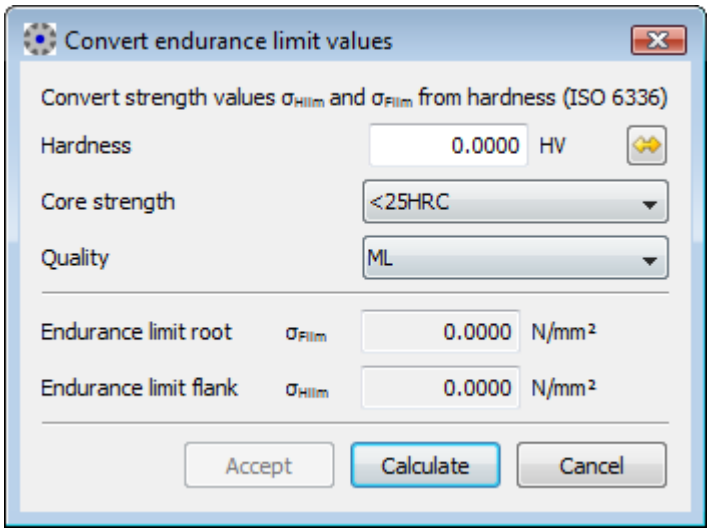


Figure 14.6: `Convert endurance limit values` dialog window

Values for σHlim and σFlim that are required for the conversion specified in ISO 6336-5 are displayed directly in the material screen under "Own input" if these values are possible with the specified hardness and material type.

In the next conversion dialog, click on another conversion button next to the hardness input field to start converting the hardness value. In the case of materials that are not alloys you can calculate the hardness from the tensile strength value or other hardness values.

Shot peened gear

"`Shot peened`" only affects the alternating bending factor Y_M if this is calculated according to ISO 6336-1, Appendix B. This information is primarily required for documentation purposes, so that this data can be added to drawings. It is well known that shot peening improves the root safety factor. Until now, the standards for gear strength have not included any suitable data. However, if measurements have shown the extent to which σFlim is increased by shot peening, this effect can be taken into account either with the technology factor Y_T or by increasing σFlim.

14.1.11.2 Calculation of the wear coefficient kw for steel

According to Niemann [65], Table 21.6/5, and Plewe's dissertation (Plewe, H-J: "Untersuchung über den Abriebverschleiss von geschmierten, langsam laufenden

Zahnrädern" (Abrasive wear and endurance calculation for lubricated, low-speed gears), Technical University of Munich, 1980) which calculates an approximate guide value for coefficient of wear kw. kw depends on the size of the lubricant gap in the operating pitch circle hc. The function defined by Plewe, kw = f(hmin), is valid for standard mineral oil without high pressure additives.

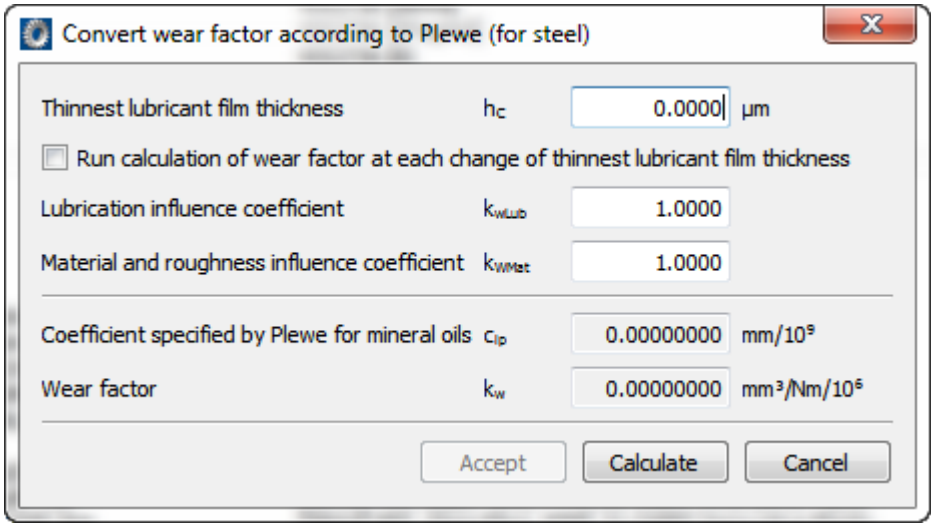


Figure 14.7: Input window for wear coefficient

You should take care when using this guide value, because the existing information is far from complete. In particular, very little is known about the influence of surface roughness and the influence of lubricant additives. You should take careful measurements to check the wear coefficient to ensure reliable results from the calculations.

Influence coefficient of lubricant: As stated in [65], adding suitable additives to a lubricant can significantly reduce the amount of wear. The influence coefficient of the lubricant can therefore lie in a range between 0.01 and 1.

Influence coefficient of material: Plewe took measurements from various different material pairings: Gear made of through hardening steel paired with a hard or soft counter gear, gear pairs made of case hardening steel, and gear pairs made of nitriding steel. The kw as defined by Plewe was then determined for these combinations. The influence coefficient (if known) can be used for other combinations. For more information see [65].

14.1.11.3 Lubrication

Select the lubricant from a list. If you select `Own input`, click the button to specify your own lubricant.

If you see the note (with kw info) after the lubricant description, this means a lubrication influence coefficient k_{wlub} is present for this lubricant. This coefficient can then be used to determine the wear coefficient k_w more accurately.

You can select oil bath or oil spray lubrication, or grease lubrication, or none at all (dry run). You can select dry run only when using a calculation method for plastics.

Click the button to the right of the lubrication type drop-down list to open the `Define temperatures` window (see Figure 14.13)

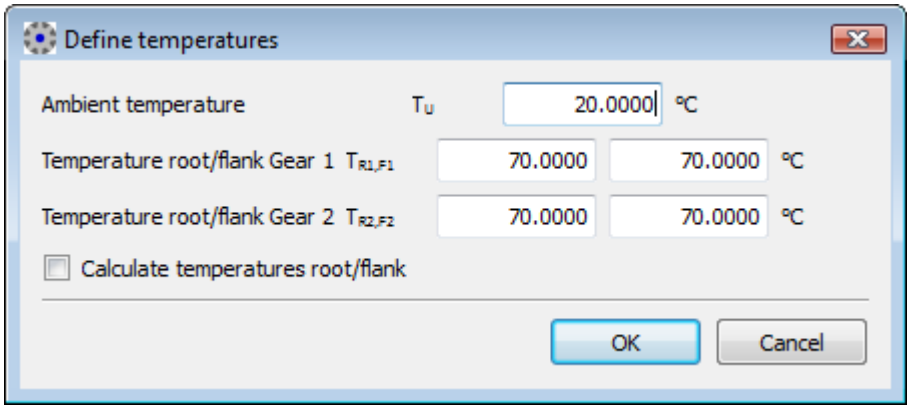


Figure 14.8: `Define temperatures` for `dry run` dialog window

Here, you can either specify your own lubricant temperature or enter the root and flank temperatures for a dry run in the case of plastics. Usually, these temperatures will be calculated for plastics. However, you can also switch off the calculation and define your own temperatures.

12. Calculating the required amount of lubricating oil

When the spray lubrication method is used, the required amount of lubricating oil is calculated as specified by Schlecht [97]. This assumes a difference of 10°C between the temperature of the oil at the inlet and outlet. The specific heat capacity cp (Ws/(KG*K)) and the specific weight at operating temperature are defined as specified by Niemann [64].

14.2 Rating

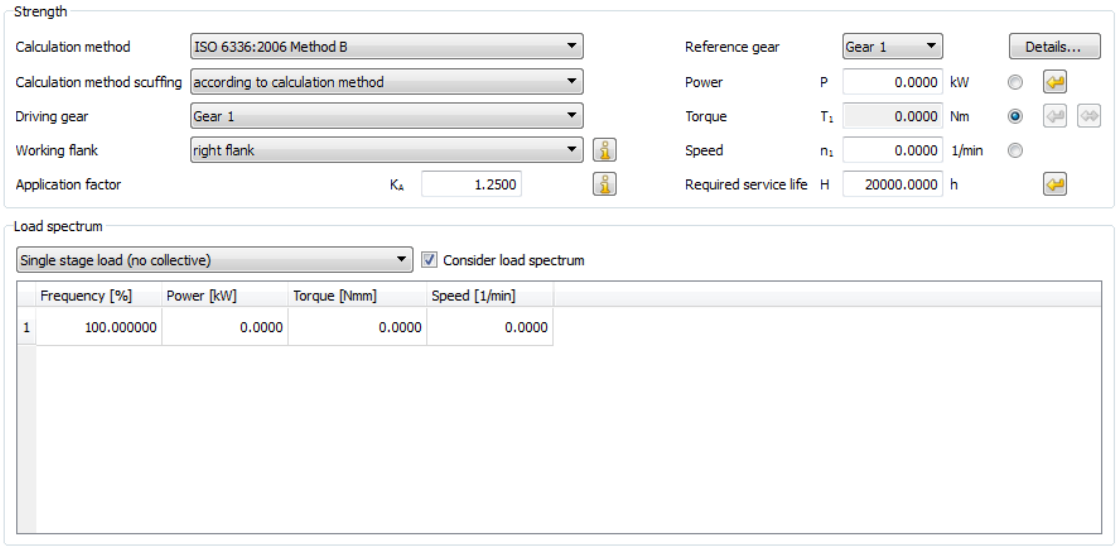


Figure 14.9: `Rating` input window for cylindrical gear pair

The `Rating` input window is one of the standard (see page I-89) tabs and is subdivided into 2 areas: `Strength` and `Load spectrum`.

14.2.1 Calculation methods

In the drop-down list, you can select the following calculation methods:

1. **Geometry calculation only.** If the Rating module is not selected in the Calculation menu, only the geometry is calculated.
2. **Static calculation**. Unlike DIN 743 which, for example, has a specific method for static shaft calculations, ISO 6336 does not have its own calculation method for static calculation. In a static calculation, the nominal stress is usually compared with the permitted material parameters (yield point and/or tensile strength). This runs a static calculation of cylindrical gears in KISSsoft where the nominal stress in the tooth root (calculated by tooth form factor Y_F) is compared with the yield point and tensile strength. See Static calculation (on page II-285).
3. **ISO 6336:2006 Method B** (Calculation of load capacity of spur and helical gears). Method B is used for this calculation.
4. DIN 3990, Method B (Calculation of load capacity of cylindrical gears). This calculation is also performed using Method B. However, either Me-

thod B or Method C can be used to calculate the tooth form factor (We recommend Method C for internal meshings. Otherwise, use Method B).

5. **DIN 3990, Method B** (YF Method C).
   (See DIN 3990, Method B)
6. DIN 3990, Part 41 (Vehicle Transmission), Method B (Load capacity calculation for vehicle transmissions). Method B is used for this calculation. You must enter two application factors (see page II-291) to represent load spectra accurately.
7. **AGMA 2001-B88**. (See AGMA 2001-C95)
8. **AGMA 2001-C95**. This edition of the AGMA 2001-C95 American national standard replaces AGMA 2001-B88. The previous version of the AGMA standard has been retained because many companies still use old versions of the guidelines. In fact, there are very few differences between the old edition, B88, and the new edition, C95. However, the new edition does include the service factor calculation.

   The standard is implemented in its entirety, and the dynamic factor and the face load factor are calculated according to AGMA recommendations. The geometry factors (for tooth root and flank) are calculated entirely according to ANSI/AGMA 908-B89.

   The following values are also output in addition to all the relevant intermediate results:
   Pitting Resistance Power Rating, Contact Load Factor, Bending Strength Power Rating, Unit Load for Bending Strength, Service Factor.

   This calculation can also be used for every other cylindrical gear configuration (including planetary stages). However, it is remarkable that the AGMA standard does not permit tooth root strength to be calculated directly in internal gear pairs. In this case the calculation must be performed using the graphical (see page II-307) method.
9. **AGMA 2001-D04**.
   Most recent version of AGMA 2001. Differs only slightly from the previous version, C95.
10. **AGMA 2101-D04**. (Metric Edition)
    Equivalent to AGMA 2001-D04, but all values in SI units.
11. **Special AGMA standards: 6004-F88, AGMA 6014-A06**
    Special standards used in the USA to calculate the strength of open gear rims. These calculation methods are based on the AGMA 2001 or 2101 basic standards. However, some factors have been specifically defined for

special applications.
AGMA 6014 replaces the old AGMA 6004, but both methods are still available, because AGMA 6004 is still requested separately.

12. **AGMA 6011-I03: For turbo drives (High Speed Helical Gear Units) and API 613**
    The AGMA 6011 standard is a special edition for high speed drives and is less complex than AGMA 2001 (or the metric AGMA 2101) base standards. In this case, less complex means that some data is already predefined. For example, AGMA 2001 has the options "Open gearing", "Commercial gear unit" and "Precision gear unit" for defining the face load factor, whereas AGMA 6011 has "Precision gear unit" as a predefined requirement. AGMA 6011 also provides information to help you select the application factor KA for specific turbo-driven applications and other useful notes about this type of gear (lubrication arrangement etc.). It is therefore always possible to perform the calculation according to AGMA 6011 using AGMA 2001 or 2101 without causing any problems. To input data correctly for AGMA 2001, as implemented in KISSsoft, that is also correct for AGMA 6011, you must be aware of the constraints, and take them into consideration when entering the parameters. Select the AGMA 6011 method to save the user having to do this. In this situation, the program checks whether all the constraints are set and, if not, displays a prompt asking the user if they want to make any modifications.

    Calculation according to API613 (Special Purpose Gear Units for Petroleum, Chemical and Gas Industry Services, 2003). API613 states that the calculation must be performed according to AGMA 6011. However, this also involves a number of special features. To perform the calculation correctly, you must use our information sheet which describes the necessary checks and inputs: kisssoft-anl-078-E-CylindricalGears API613.docx. The values required by API613, such as flank load K or the permitted value Ka, bending load S and the permitted value Sa, as specified in Annex J of API613, are documented.

13. **GOST-21354-87**
    Calculation according to the Russian guideline (latest edition, 1987). Take the following notes into account, see GOST-21354-87 (on page II-285).

14. **Plastic as defined in Niemann**
    Please refer to [65] and Table 13.3 to see the differences.

15. **Plastic as defined in VDI 2545 (YF, Method B)** (thermoplastic materials used in gears). This method has been withdrawn, and replaced by the new method, according to VDI 2736. This regulation defines how calculations are performed on gears made of plastic or combinations of plastic and

steel. See Plastic as defined in VDI 2545 (YF, Method B) (see section "Plastics according to Niemann, VDI 2545 or VDI 2736" on page II-287).

16. **Plastic as defined in VDI 2545 (YF, method C)**.
In this calculation method, the tooth form factor Y_F is calculated according to Method C.

17. **Plastic as defined in VDI 2545 modified (YF, Method B).**
This method was recommended for use by KISSsoft before VDI 2736 was published. VDI 2736 contains all the modifications recommended according to Tables 13.3 and 13.4. This method is recommended for plastics with normal toothing. Transverse contact ratio $\varepsilon_\alpha < 1.9$. See table in 14.4 for the differences between VDI and VDI modified.

18. **Plastic according to VDI 2545 modified (YF, Method C).**
This method is recommended for plastics with deep toothing. Transverse contact ratio $\varepsilon_\alpha > 1.9$. See table in 14.4 for the differences between VDI and VDI modified. See table in 14.4 for the differences between VDI and VDI modified. In this calculation method, the tooth form factor Y_F is calculated according to Method C.

19. **Plastic according to VDI 2736.**
We recommend you use this calculation method, VDI 2736, which was published for the first time in 2014/15. It includes all methods described in sheet 2 of VDI 2736 (empirical calculation, tooth root, tooth flank, deformation, wear).

20. **As in FVA program (DIN 3990)**. Supplies the same results as the FVA (Forschungsverein Antriebstechnik: German Research Society for Transmission Techniques) Reference Program. Based on DIN 3990 Method B with minor differences.

21. **BV/Rina FREMM 3.1 Naval Ships and Rina 2010 (ISO 6336)**
Calculation guidelines for ships' engines.

22. **DNV 41.2, Calculation guideline for ships' engines**
The Det Norske Veritas calculation guideline [93] for ships' engines corresponds in principle to ISO 6336 (root, flank) and ISO 13989 (scuffing). However, it does have some significant differences, especially where S-N curves (Woehler lines) are concerned. These differences are detailed in our kisssoft-anl-076-DE-Application_of_DNV 41.2.pdf information sheet, which is available on request.

23. **Lloyd's register, classification for ships**
Calculation guideline for ships' gears

24. **ISO 13691, High-Speed Special Purpose Gear Units**
Calculation guideline for high-speed gear units

14.2.1.1 Static calculation

Each coefficient (application factor, face load factor, transverse coefficient, dynamic factor) is set to 1.0. The load at the tooth root is calculated with the tooth form factor according to ISO 6336 Method B and the helix angle (without the stress correction factor).

$$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_\varepsilon \cdot Y_\beta \tag{12.1}$$

$$\sigma_F = K_A \cdot K_V \cdot K_{F\alpha} \cdot K_{F\beta} \cdot \sigma_{F0} \tag{12.2}$$

It also calculates the local tooth root stress multiplied by the stress correction factor Y_S. This stress is approximately the same as the normal stress calculated in an FEM model. This stress is then also output in the report:

$$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_S \cdot Y_\varepsilon \cdot Y_\beta \tag{12.3}$$

14.2.1.2 GOST-21354-87

Quality according to GOST 21354-87

GOST only takes into account one quality, which is why the poorer quality of the two gears is used during the calculation.

Q = max (Q1, Q2)

Endurance limit values for root and flank
The endurance limit values σFlim and σHlim are saved in the database, in KISSsoft, or you can enter them, if you select "Own input".

Endurance limit for tooth root
The endurance limit sFlim is calculated as follows, according to GOST:
σFlim = σFlim0 * Yz * Yg * Yd * YA * YT
σFlim0 – nominal endurance limit in the case of a limit load cycle (GOST 21354-87, Tables 14 to 17).
Yz - blank coefficient (GOST 21354-87 Table 13, Formula 10.3).
Yd – Takes into account the hardening of the root transition zone (GOST 21354-87, Tables 14-17).
Yg – takes into account the grinding of the root transition zone (GOST 21354-87, Tables 14 to 17).
YT – technology factor (GOST 21354-87, Table 13, Formula 10.2). The default technology factor setting is 1.0, but you can change it in KISSsoft. Select Factors – Z Y factors.
YA – alternating bending factor (GOST 21354-87 Table 13, formula 10.6). The alternating bending factor setting is 1.0, but you can change it to your own entry in KISSsoft. Select Factors – Alternating bending factor.
The Yd, Yg and Yz coefficients cannot be entered in KISSsoft and must be included directly when the endurance limit is entered.
In addition to the coefficients mentioned above, the endurance limit defined in GOST must be divided by 2.0 before being entered in KISSsoft. In the calculation, sFlim is then multiplied by the stress correction factor YST = 2, in a similar way to in ISO or DIN.

Consequently, the correct entry for sFlim in KISSsoft, for calculations according to GOST, is:
σFlim (KISSsoft entry) = sFlim0 (according to GOST) * Yd * Yg * Yz / 2.0

Required minimum safeties
GOST has the special property that the minimum safety set for tooth root fracture and the flank depends on the material type and the surface hardening. For this reason, you can enter minimum safeties for every gear individually, for GOST, under "Settings" > "Required safeties".

Information about root rounding
In various GOST formulae, a distinction is made between whether the root rounding is ground or not. To make this distinction, you must enter a suitable value under "Load" > "Details" > "Gear".

Face load factor flank calculation KHb
The face load factor (flank) is calculated according to GOST 21354-87, Table 6, Formula 7. The considerations described in GOST 21354-87, Annex 6, are ignored.

Face load factor root KFb
The face load factor (root) is calculated according to GOST 21354-87 Table 13, Formula 4.

Dynamic factor KV
The dynamic factor is calculated according to GOST 21354-87 Table 6, Formula 6. If conditions (34) and (35) specified in Formula 6 are not fulfilled, KISSsoft calculates the dynamic factor according to GOST 21354-87 Annex 5.

Load spectra
Calculations with load spectra are performed using the rules defined by Palmgren-Miner, according to ISO 6336-6.

Safety of the hardened layer
The safety of the hardened layer is calculated according to DNV 41-2.

14.2.1.3 Plastics according to Niemann, VDI 2545 or VDI 2736

The calculation methods used for plastics take special account of the fact that these materials are very sensitive to extremes of temperature. The types of lubrication used here include oil, grease or none at all (dry run). The acceptable load for each material is calculated from figures in data tables, in DAT format, while taking into consideration the local temperatures at the tooth flank and root, and the number of load cycles. The local temperature can be calculated when grease is used as the lubricant or during a dry run. However, when oil is used as the lubricant, the oil temperature is used as the local temperature. The calculation is performed for combinations of plastic/plastic and also steel/plastic. The acceptable deformation is also checked. KISSsoft supplies data for the following materials:

Polyamide (PA12, PA6, PA66, PA46)

Polyacetal (POM)

Polyetheretherketone (PEEK)

Polybutylene terephthalate (PBT)

Polybutylene terephthalate (PET)

Laminate

Data about other materials is available on request

All the specific properties of each material are stored in text tables (DAT files) to enable the integration of own materials (see page I-128). The strength of plastics can be calculated either as defined by Niemann 66], or according to VDI 2545 (1981)[3] [76] or VDI 2736 [102]. You can also use the modified calculation method as detailed in VDI 2545. This calculates the stress using the tooth root stress correction factor Y_s. The major differences between the two methods are:

Root	**Niemann**	**VDI 2545**	**VDI 2545-mod.**	**VDI 2736**
Y_F	C	B or C	B or C	C
Y_S	DIN 3990	1.0	DIN 3990	DIN 3990
Y_ε	1.0 [8)]	$1/\varepsilon_\alpha$ [7) 9)]	$1/\varepsilon_\alpha$ [7) 9)]	DIN 3990
Y_β	1.0	DIN 3990 [10)]	DIN 3990 [10)]	DIN 3990
σ_{FE}	$2 * \sigma_{Flim}$	σ_{Flim}	$2 * \sigma_{Flim}$	$2 * \sigma_{Flim}$

Table 14.4: Differences between the calculation methods used to calculate the root safety factor for plastics

Flank	**Niemann**	**VDI 2545**	**VDI 2545-mod.**	**VDI 2736**
Z_ε	1.0	DIN 3990	DIN 3990	DIN 3990
Z_V	DIN 3990 [5) 10)]	1.0	1.0	1.0
Z_R	DIN 3990 [6) 10)]	1.0	1.0	1.0

Table 14.5: Differences between the calculation methods used to calculate the tooth flank load capacity for plastics

Tooth deformation: Very different calculation methods!

[5)] For plywood only, otherwise 1.0

[6)] For steel/plastic combinations only, otherwise 1.0

[7)] For tooth form factor Y_F as defined in Method B: 1.0

[8)] The method sets the face contact ratio for the tooth root stress to the value 1.0. According to Niemann, this is because the material data is not always precise. The formulae used in VDI 2545 correspond to those used in ISO 6336:1996.

[3] *Calculation method VDI 2545 has been withdrawn and replaced by VDI 2736.*

[9] For crossed helical gears = 0.25 + 0.75/εγ
[10] For crossed helical gears = 1.0

14.2.2 Service life

Enter the required service life directly in the input field.
Click the button to size this value. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated according to ISO 6336-6:2006 using the Palmgren-Miner Rule. The system service life and the minimum service life of all the gears used in the configuration are displayed. You can size the service life using the button either with or without defining a load spectrum (see section "Define load spectrum" on page II-308).

14.2.3 Reliability

Reliability is calculated according to Bertsche's study [105 (see section "Bibliography" on page X-1214)], in which the possible methods have been described in great detail. The most commonly used approach, and one which is well suited to the results that can be achieved in "traditional" mechanical engineering calculations, is the "Weibull distribution". In this case, Bertsche recommends the use of the 3-parameter Weibull distribution. The reliability, R, of a machine element is calculated as a function of the number of load cycles, t, using the following equations.

$$R(t) = e^{-\frac{t-t_0}{(T-t_0)}^{\beta}} * 100\%$$

Parameters T and t_0 can be derived from the mathematically achievable service life of the component, H_{att}, as follows (with F_O according to the calculation method, Table 1, β and f_{tB} from Table 2 according to Bertsche).

Calculation method	**Damage probability Fo**			
	1%	10%	Other	Comment
Shaft, DIN 743			2.5%	Assumed, not documented
Shaft, FKM guideline			2.5%	
Shaft, AGMA 6001	*			If kC = 0.817

Rolling bearing, ISO 281		*		If coefficient a1 = 1.0
Tooth flank, ISO 6336; DIN 3990	*			
Tooth root, ISO 6336; DIN 3990	*			
Tooth flank, AGMA 2001	*			If randomness factor KR=1
Tooth root, ISO 6336; AGMA 2001	*			If randomness factor KR=1

Table 1 14.6a: Damage probability for different calculation methods when determining material properties

$$T = \left(\frac{H_{att} - f_{tB} * H_{att10}}{\sqrt[\beta]{-\ln\left(1 - \frac{F_o}{100}\right)}} + f_{tB} * H_{att10} \right) * fac$$

$$t_0 = f_{tB} * H_{att10} * fac$$

with

$$H_{att10} = \frac{H_{att}}{(1 - f_{tB}) * \sqrt[\beta]{\frac{\ln\left(1 - \frac{F_o}{100}\right)}{\ln(0.9)}} + f_{tB}}$$

	Coefficient ftB	**Weibull shape parameter** β
Shafts	0.7 to 0.9 (0.8)	1.1 to 1.9 (1.5)
Ball bearing	0.1 to 0.3 (0.2)	1.1
Roller bearing	0.1 to 0.3 (0.2)	1.35
Tooth flank	0.4 to 0.8 (0.6)	1.1 to 1.5 (1.3)
Tooth root	0.8 to 0.95 (0.875)	1.2 to 2.2 (1.7)

Table 2 14.6b: Factors for a Weibull distribution according to Bertsche, average values used in KISSsoft given in brackets

NOTE:

Calculating reliability using Weibull distribution uses the calculated service life, and so also takes into account the required safeties. To calculate reliability without taking required safeties into account, set the safeties to 1.0.

14.2.4 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 14.6 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336, DIN 3990 and DIN 3991.

When deciding which application factor should be selected, you must take into account the interrelationship between the required safety values, assumed loads and application factor.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 14.6: Assignment of operational behavior to application factor

DIN 3990, Part 41 (car gearboxes), distinguishes between application factors for flank strength K_{AH} and for tooth root strength K_{AF}. Except for flank strength calculations, all other calculations (e.g. resistance to scoring) use application factor K_{AF}.

However, according to DIN 3990 Part 41, the application factor can also be less than 1.0. This is intended to avoid the need to perform a calculation involving a load spectrum. For example, DIN 3990, Part 41, Annex A, suggests the following values for a 4-speed car gearbox:

Gear	**R**	**1**	**2**	**3**	**4**
N_L	10^5	$2 * 10^6$	$1.5 * 10^7$	$3 * 10^7$	$2 * 10^8$
K_{AH}	0.65	0.65	0.65	0.65	
K_{AF}	0.70	0.70	0.80	0.80	

Table 14.7: Application factor according to DIN 3990, Part 41

14.2.5 Power, torque and speed

Click the  button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-471) can be maintained.

14.2.6 Strength details

Click on the `Details...` button to open the `Define details of load` window which is divided into the `System data` and `Pair/Gear data` tabs. Note that a different window layout is used for calculations according to AGMA (see page II-307).

14.2.6.1 System data

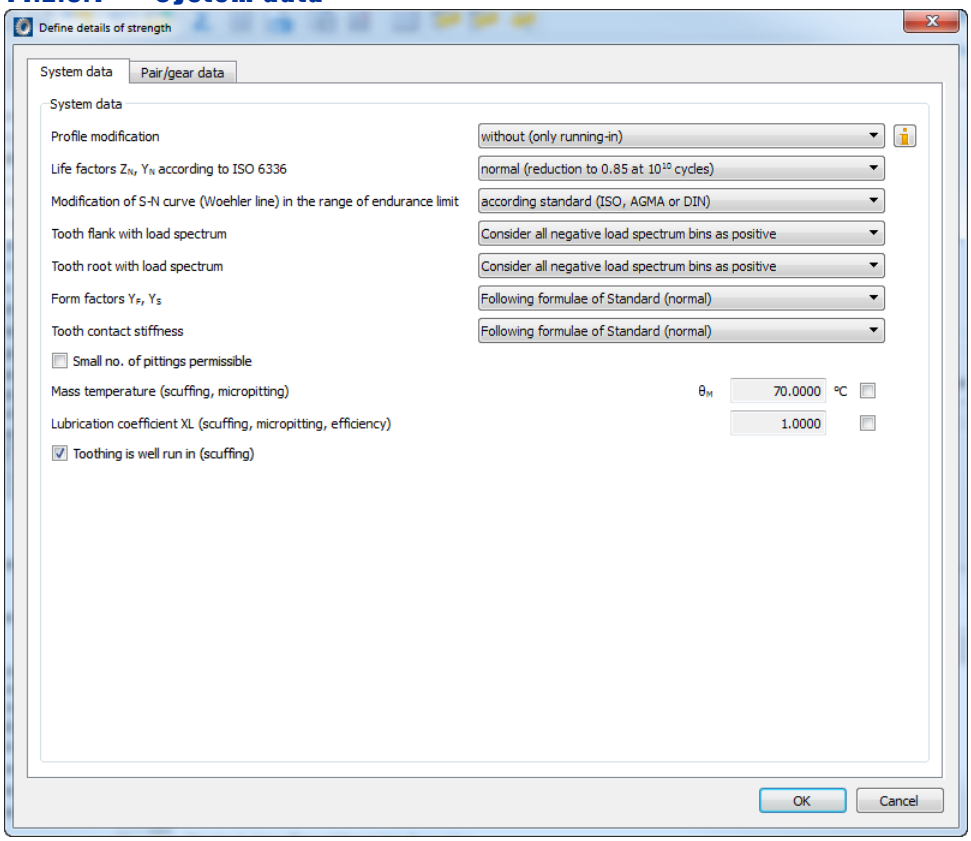


Figure 14.10: `System data` input window for a cylindrical gear pair

Profile modification

You can modify the theoretical involute in high load capacity gears by grinding/polishing the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft Module Z15 (see section "Modifications" on page II-361).

The method used to perform the profile modification has an effect on transverse load factors $K_{H\alpha}$ and $K_{H\beta}$ and also on the calculation of scuffing safety. The force distribution factor X_Γ is calculated differently according to the type of profile modification used. The main difference is whether the profile has been modified or not. However, the differences between `for high load capacity gears` and `for smooth meshing` are relatively small. The strength calculation standard presumes that the tip relief C_a is properly sized, but does not provide any concrete guidelines. The resulting force distribution factor X_Γ, according to DIN 3990, depends on the type of profile modification:

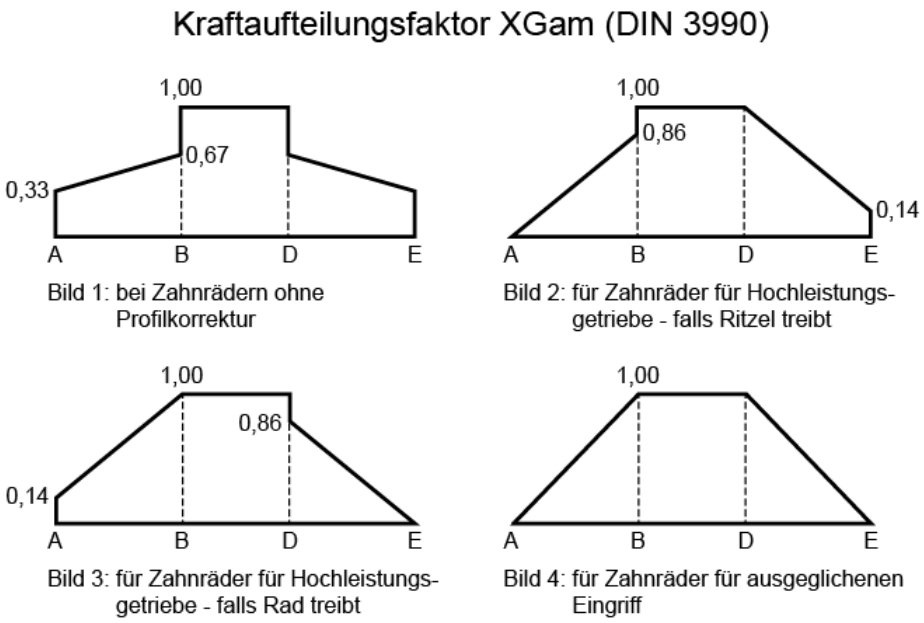


Figure 14.11: Load sharing factor X_Γ for different profile modifications

Lifetime factors as defined in ISO 6336

The fatigue limit factor Z_{NT} reduces the permitted material stress according to ISO 6336-2:2006:

$$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim} \quad (12.14)$$

$$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim} \quad (12.15)$$

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safety values for the range of endurance limit when compared with DIN 3990.

1. **normal (reduction to 0.85 at 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. Fatigue strength factors Y_{NT} and Z_{NT} are set to 0.85 for $\geq 10^{10}$ load cycles.
2. **increased with better quality (reduction to 0.92):** Y_{NT} and Z_{NT} at $\geq 10^{10}$ load cycles are set to 0.92 (in accordance with the data in ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

Modification of S-N curve (Woehler lines) in the range of endurance limit

In a standard Woehler diagram, the range of endurance limit is reached at a particular number of load cycles. From this point onwards, the dynamic strength no longer changes even when the number of load cycles increases. This behavior is called "according to Miner".

However, more recent investigations have revealed that there is actually no such thing as an endurance limit and that the S-N curve (Woehler lines) should be modified in the endurance limit range.

In the range of endurance limit, you can therefore select the following modified forms:

- Miner (corresponds to DIN 3990, Parts 2, 3 and 6). Lead ∞ (horizontal)
- According to Corten/Dolan. Lead p
- According to Haibach modified. Lead 2*p
- According to Haibach original. Lead 8*p-1 (according to [94])

The lead mentioned here matches the S-N curve (Woehler lines) according to

ISO, AGMA or DIN in the fixed period range, determined from YNT or ZNT. See also ISO 6336-6 [44]

Figure 14.12 shows the relevant characteristics. Experience has shown that performing a service life calculation with load spectra using the Miner method returns results that are far too optimistic. We recommend you use the Haibach method of approach.

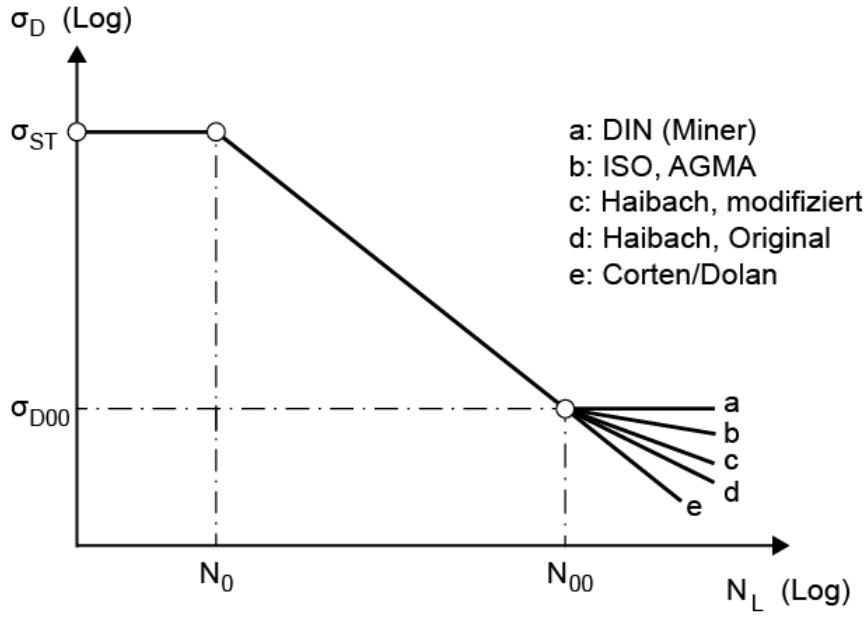


Figure 14.12: Endurance limit models

Note concerning calculations according to ISO or DIN:

The pitch (slope) of the S-N curve (Woehler lines) for bending in the time-dependent domain (between N0 and N00) is defined using the YNT, YdrelT, YRrelT and YX coefficients for the static and endurance cases, but in the endurance domain (NL > N00), only the YNT coefficient is used for the static and endurance cases. The same applies to pitting with the ZNT, ZL, ZV, ZR and ZW factors. This corresponds to the procedure used in ISO 6336 for the endurance domain. However, this does mean that buckling occurs on the S-N curve (Woehler lines) at N00, according to the Corten/Dolan rule.

As an example: for case-carburized steel, the pitch (slope) of the S-N curve (Woehler lines) in the endurance domain is 13.2, but in the time-dependent domain it is approximately 10, depending on the precise values for YdrelT, etc.

If all the factors, YdrelT, etc., are set to 1.0 using "Own input" then there will be no buckling of the S-N curve (Woehler lines) at N00.

Tooth form factors

The tooth form factor Y_F takes into account how the tooth form affects the nominal tooth root stress σ_{F0}. The stress correction factor Y_S takes into account the effect of the notch on the tooth root. These two factors can be calculated in three different ways:

1. **In accordance with the formulae in the standard (normal)**
   As defined in ISO 6336 or DIN 3990, the tooth form and the stress correction factors are calculated at the tooth root at the point at which the tangent and the tooth center line form an angle of 30°. However, it is generally acknowledged that this method is rather imprecise, for deep meshings in particular.

2. **Using graphical method**
   According to Obsieger [68], there is a more precise approach in which the product of the tooth form factor Y_F and the stress correction factor Y_S is calculated and the maximum value is determined. This method is based on the manufacturing process used for a specific tooth form and is applied to all points in the whole root area. This maximum value is then used in calculating the strength. Factors Y_F and Y_S are calculated in accordance with the formulae in ISO 6336 or DIN 3990.

   This is the recommended method, particularly for unusual tooth forms and internal teeth. If required, this calculation procedure can also be applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing.

   Note:
   If you use the graphical method here, KISSsoft will calculate the tooth form before it calculates the strength, each time. It takes its parameters either from the cutter data you entered previously in the `Tooth form` (see section "`Gear tooth forms`" on page II-636) input window or from the default settings in the `Reference profile` input window. The maximum value of the product of the tooth form and stress modification factor is calculated at the same time and included in the stress calculation.

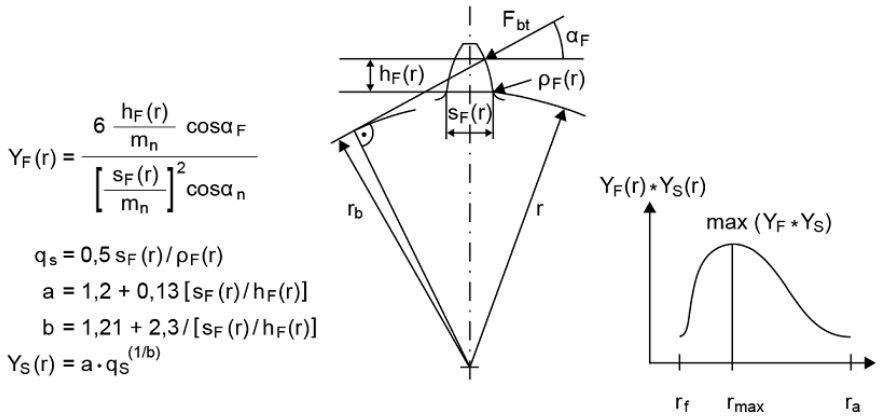


Figure 14.13: Tooth form factors using graphical method

3. **for internal toothing, in accordance with VDI Proposal 2737**
When calculating strength according to ISO 6336 or DIN 3990, selecting this option enables you to use the tooth form factor as defined in VDI 2737, which is more precise for internal teeth, because it evaluates the stress at the point of the 60° tangent and derives the tooth form from the manufacturing process with the pinion type cutter.

The tooth root stress calculation specified in ISO 6336 is more accurate than the one implemented in DIN 3990. However, the calculation applied to the root rounding in the critical point (for a 60° tangent) is still incorrect. The method defined in VDI 2737, Annex B is much more accurate, which is why we recommend you use this method. If you select this option, only the root rounding ρF and the root thickness sFn in the critical cross section is calculated according to the formulae in 2737. All other sizes are calculated according to ISO 6336.

The table (below) uses 4 examples to show the large variations that still arise in root rounding between the result defined in ISO 6336 and the effective values measured on the tooth form. However, the calculation method stated in VDI 2737 is very suitable.

Gear x=	**Pinion Cutter x0=**	**ρF in ISO 6336-3 2006 and 2007-02**	**ρF in ISO 6336-3 2007-04**	**ρF measured on the tooth flank**	**ρF with VDI 2737**
-0.75	0.1	0,201	**0,426**	**0,233**	**0,233**
-0.75	0.0	0,175	**0,403**	**0,220**	**0,220**
0.0	0.1	0,298	**0,364**	**0,284**	**0,286**

0.0	0.0	0,274	**0,343**	**0,265**	**0,264**

Table 14.8: Comparison of root roundings

Note about calculating YF:

- The theoretical profile shift is used for the calculation of the allowance is As < 0.05*mn (according to ISO 6336-3). Otherwise the larger manufacturing profile shift xE.e is used. This corresponds to the procedure used in the STplus program (from Munich, Germany). An exact definition is not provided in the ISO standard. However, if this is specified in Settings `Strength calculation using mean position in tolerance field (of tooth form)`, the calculation will always be performed with the average manufacturing allowance xE.m.
- According to the ISO standard, the reference profile for the entire toothing is to be used for the calculation. For this reason, if you input the reference profile for pre-machining with protuberance, and a manufactured profile with remaining protuberance is left after deduction of the grinding allowance, the reference profile for final machining is used for the calculation. In the case of the reference profile for pre-machining without a protuberance (or a protuberance that is too small), a grinding notch is produced. To ensure that this situation can be correctly taken into consideration, the pre-machining reference profile (with pre-machining manufacturing profile shift) is used to calculate YF. Furthermore, the final machining reference profile is used to calculate the grinding notch and therefore define YSg (section 7.3 in ISO 6336-3).

Tooth contact stiffness

Tooth contact stiffness is required to calculate the dynamic factor and the face load factor. You can use one of these calculation options:

1. **In accordance with the formulae in the standard (normal)**
   In the standard calculation, the tooth contact stiffness c_g is calculated using empirical formulae (in ISO 6336, DIN 3990, etc.).
2. **Using the tooth form**
   In this option, the tooth form stiffness c' is calculated in accordance with Weber/Banaschek's dissertation [69]. This takes into consideration tooth bending, basic solid deformation, and Hertzian pressure. The last condition determines the load dependency of c'. The contact stiffness is determined using the effective tooth form (see Meshing stiffness (Z24)). The mean value of the stiffness curve that is calculated using this method is then included in the calculation. If required, this calculation procedure can also be

applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing (Z04). The single spring stiffness c' is calculated from the c_g, by extrapolating c' from the formula for c_g (ISO or DIN).

3. **constant (20 N/mm/μm)**
   In this option, the tooth contact stiffness constant is replaced by

$$c_g = 20 N / mm / \mu m$$

Small no. of pittings permissible

In specific cases, the appearance of a small number of pittings on the flank may be permissible. In a range of materials, this results in higher flank safeties in the fatigue strength range due to the changed S-N curve (Woehler lines), as can be seen in either ISO 6336-2, Figure 6, curve 1 or DIN 3990-2, Figure 8.1.

Lubrication coefficient

The lubricant coefficient is needed to calculate the friction factor, loss, micropitting and scuffing.

These values are specified in ISO 15144:

- 1.0 for mineral oils
- 0.6 for water-soluble polyglycols
- 0.7 for non-water-soluble polyglycols
- 0.8 for polyalphaolefins
- 1.3 for phosphate esters
- 1.5 for traction fluids

14.2.6.2 Pair/gear data

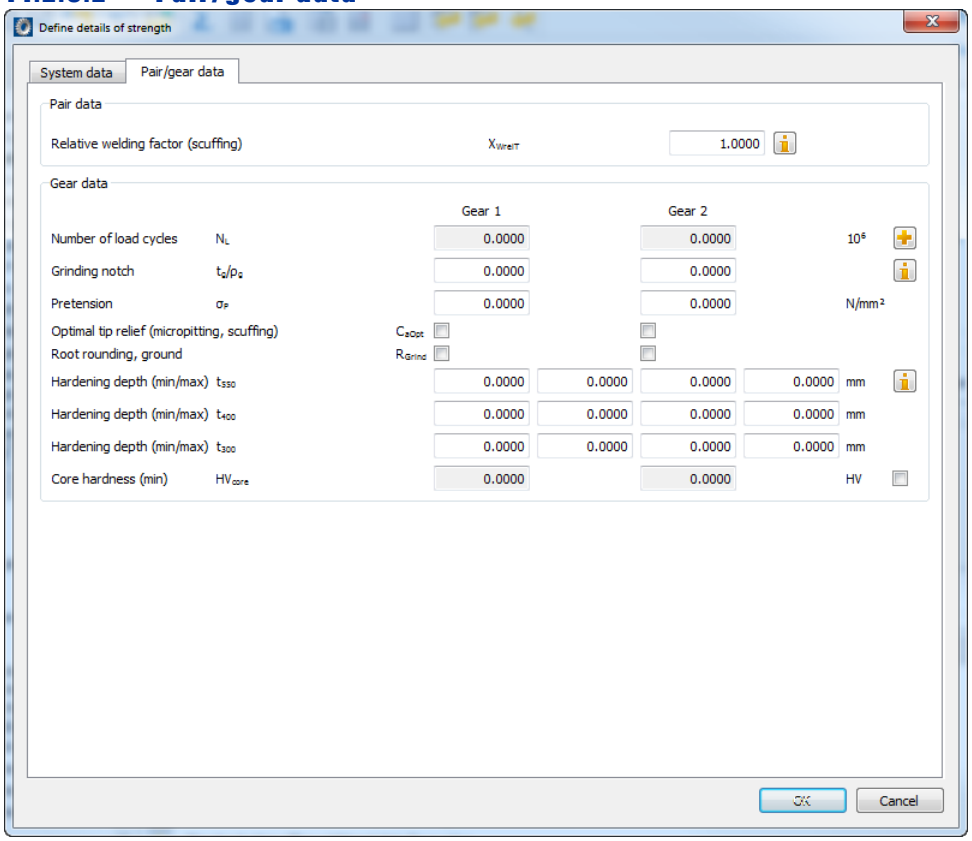


Figure 14.14: `Pair/gear data` input window for defining details of strength

Structural factor XwrelT or structural factor Xw (scuffing)

The structural factor takes into account differences in materials and heat treatment at scuffing temperature. Structural factor X_{wrelT} (in DIN 3990 and in ISO TR 13989-2) or structural factor X_w (in ISO TR 13989-1) is used, depending on which standard is used. However, in this case, $X_{wrelT} = X_w/X_{wT}$ and $X_{wT} = 1$ applies. This results in $X_{wrelT} = X_w$. The two factors are identical.

However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this factor, yourself because it is not set automatically by KISSsoft.

Through hardened steels	1.00
Phosphated steel	1.25
Coppered steel	1.50
Nitrided steel	1.50

Case-hardened steels	1.15 (with lower than average austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)
Stainless steels	0.45

Table 14.9: Structural factor as defined in DIN 3990, Part 4

The standard does not provide any details about how to proceed when the pinion and gear are made of different material types. In this case it is safer to take the lower value for the pair.

Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the `Number of load cycles for gear n window`. Click the  button to access this. In this window you can select one of five different options for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the service life, speed, and number of idler gears.
2. **Number of load cycles** Here you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear unit with three planets, enter `3` for the sun and `1` for the planets in the input field.

   Note:
   If the `Automatically` selection button in the calculation module is selected, KISSsoft will determine the number of load cycles in the `Planetary stage` calculation module, while taking into account the number of planets.
4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.
5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account that not every gear tooth will mesh with another. You must enter a value

here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/100 are set.

NOTE

This calculation method is used for transmissions with a slight oscillation angle.

This scenario assumes that a reduction is present

$$i = \frac{z_2}{z_1}$$

and an oscillation angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements by the angle value w. The effective endurance is given as the service life. The two coefficients N_1 and N_2, which reduce the absolute number of load cycles, N_L, are now calculated. To do this:

a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, a complete forwards/backwards movement is counted as a load cycle.

b) For the pinion, coefficient N_1 is determined as follows:

$$N_1 = 0.5 \qquad \text{für } w \cdot i > 360$$
$$N_1 = 0.5 \cdot \max(1.0, 360/w/i) \qquad \text{für } w \cdot i \le 360$$

c) The number of load cycles of teeth in contact in gear 2 is smaller by a coefficient of N_2 when compared with the number of load cycles during continuous turning.

$$N_2 = 0.5 \cdot 360/w$$

The coefficient 0.5 takes into account both the forwards and backwards movements.

d) Enter coefficients N_1 and N_2 in the `Load cycles per revolution` input field.

The correct number of load cycles can now be calculated on the basis of the data entered in steps a to d.

Grinding notch

As defined in DIN 3990 or ISO 6336, the effect of the grinding notch can be taken into account by the coefficient Y_{Sg}. Here, you input the ratio t_g to the radius of grinding notch ρ_g in accordance with the figure in DIN 3990-3, section 4.4 or ISO 6336-3, Figure 5. KISSsoft then calculates the coefficient $Y_g = Y_{Sg}/Y_S$ (a factor, which was multiplied with Y_S).

The distance between the 30_o tangents for the initial and final contour is used as the grinding notch depth t^g. If a pre-machining allowance has been entered in KISSsoft (see Figure 14.11), you can no longer enter the ratio t_g/ρ_g. It is calculated by the software instead. A grinding notch occurs when a grinding depth (see section "Modifications" on page II-361) has been entered, and no protuberances remain, either because no protuberance tool was used, or the selected allowance was too small. The fillet radius ρ_g is then calculated by generating the grinding wheel at the 30° tangent (or, for internal teeth, at the 60° tangent).

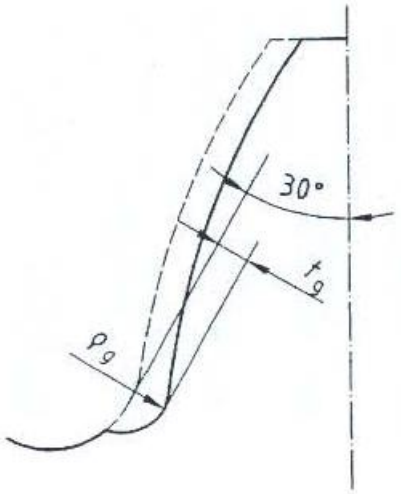


Figure 14.15: Grinding notch

Pretension

The influence of a press fit or other processing methods that influence tooth root stress can be taken into account with the pretension σ_P. This value influences the calculated tooth root stress as well as the safety according to the following formulae:

For static strength:

$$\sigma'_F = \sigma_F + \sigma_P$$

$$S'_S = \frac{R_P}{\sigma'_F}$$

$$S'_B = \frac{R_m}{\sigma'_F}$$

For fatigue strength:

$$\sigma'_{FG} = \sigma_{FG} \cdot \left(1 - \frac{\sigma_P}{R_m}\right)$$

$$S'_F = \frac{\sigma'_{FG}}{\sigma_F}$$

The pretension σ_P merely generates additional results in the reports. The results in the results window remain unchanged. You define this under "Strength" -> "Details".

NOTE 1

This rule is not documented in the ISO standard. **For this reason, we recommend extreme caution if the preload effect is to be taken into account.** The formulas are proposed by Alstom Ecotecnia. KISSsoft only shows this effect in the report.

NOTE 2

If the main calculation (single load or load spectra) requires the use of this rule, the value σ'Flim must be changed as follows, according to the equation for 'FG:

$$\sigma' F\lim = \sigma F\lim^* \cdot \left(I - \frac{\sigma_P}{R_m} \right)$$

σ'Flim must be used instead of σFlim in the material values. The main calculation will then be performed using this rule for pretension.

Moments of inertia

If required, you can enter the moment of inertia of the gears. This value is used when calculating the dynamic factor. If you enter an invalid value (deviation > 100% from the expected value), the system displays a warning. Normally in such cases, the wrong unit is used for the input.

Optimal tip relief

To calculate safety against micropitting as specified by Method B in ISO 15144, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large discrepancies, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and documented accordingly in the report.

Root rounding, ground

The setting specifying whether the root rounding is ground or not is only used in calculations according to GOST.

Information about material hardness

For more information about material hardness, refer to section 14.9a: Tooth flank fracture.

14.2.7 Strength details (AGMA)

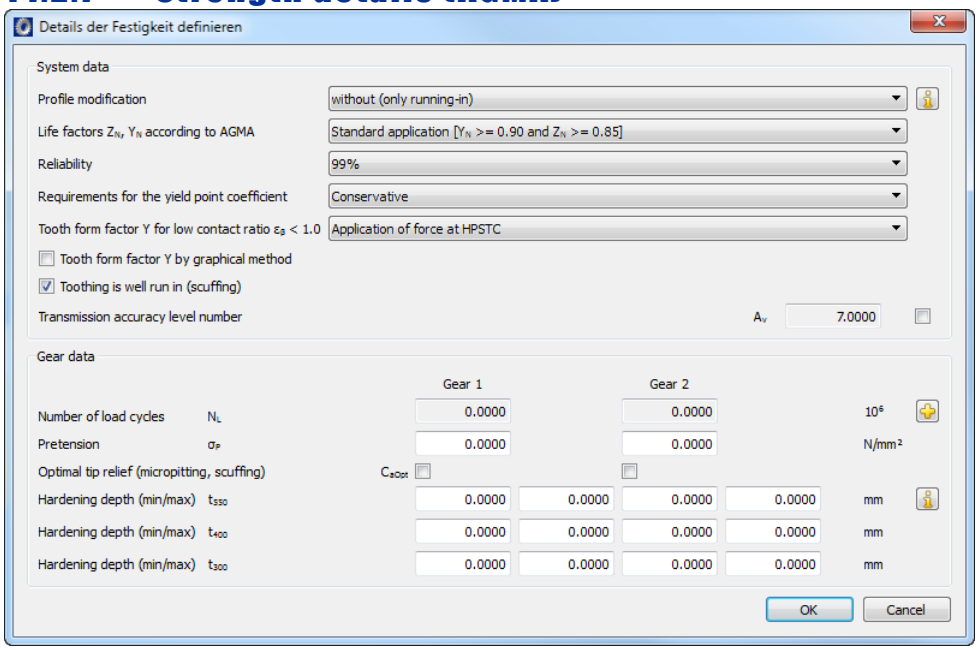


Figure 14.16: `Define details of strength` input window for calculating strength as defined in AGMA

NOTE

Only values in the input window that differ from those defined in ISO are described here.

14.2.7.1 Lifetime factors

The endurance limit factors determine which material values can be entered in the field for limited time and strength. In standard applications, endurance strength values up to 10^{10} load cycles are reduced from 100% to 90% for the root and to 85% for the flank. As stated in AGMA, the reduction in strength also extends beyond 10^{10} load cycles. In critical applications, where a gear breakdown must be prevented at all cost, the material values are further reduced in comparison to those used in standard application areas.

14.2.7.2 Tooth form factors

For cylindrical gears with low helix angles, or cylindrical spur gears, you can specify that the load is to be applied either at the tip or at the individual point of contact (the more precise option). For cylindrical gears with a large helix angle ($\varepsilon\beta$

≥ 1) according to AGMA the force is always applied to a single meshing point (HPSTC).

Calculating with the HPSTC results in a lower load at tooth root because the load is divided between the two teeth. However, if large single pitch deviations occur, this load distribution does not take place and therefore the force should be assumed to be placed at the tooth tip.

As stated in AGMA, the contact point between the tooth form and the Lewis parabola is selected as the critical root cross section. The stresses are determined here. AGMA does not provide a formula for calculating internal teeth. Instead, it recommends you use the graphical method to calculate the tooth form. The required data is to be taken from measurements. If you click the checkbox to select the graphical method of calculating the tooth form factor, the software automatically calculates the tooth form at the point where the K_f or I factor is greatest. In contrast to the method defined by Lewis, where the calculation is only performed at the contact point of the parabola, the calculation using the cross section with the greatest stresses gives more precise results, and is therefore the method we recommend for external gears too.

14.2.7.3 Transmission accuracy level number

A_V (or Q_V for AGMA 2001-C95 or earlier) is calculated in accordance with the formulae defined in AGMA 2001 or 2101 and is extremely dependent on the accuracy grade. However, the A_V value may be one level higher or less than the accuracy grade and is needed to calculate the dynamic factor. You can overwrite this value if required.

14.2.8 Define load spectrum

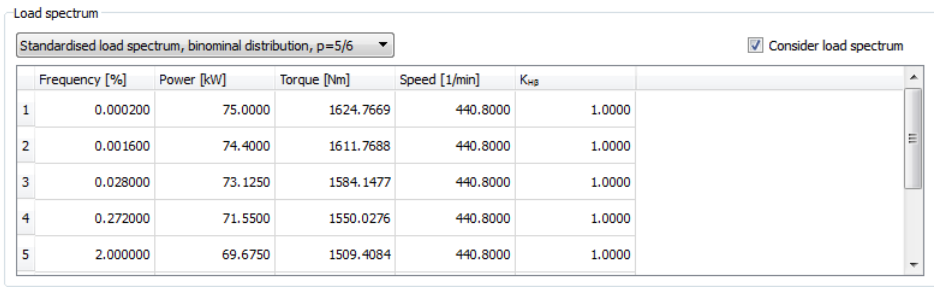


Figure 14.17: `Load spectrum` group

In this group, you can also access load spectra that have been stored in the database. You can also define the load spectra directly.

If you select `Read`, you can import a file (in either *.txt or *.dat format) with a load spectrum.

The "Example_DutyCycle.dat" file in the `dat` sub-folder in the KISSsoft installation directory is an example of a file that shows how a load spectrum can be defined.

If you want the calculation with load spectra to include separate factors ($K_{H\beta}$, K_{γ}, etc.) for each load spectrum element, you must make the appropriate settings in the `Factors` tab for the load distribution coefficient (on page II-314) K_{γ}, the alternating bending factor (on page II-315) Y_M and the face load factor (on page II-318) $K_{H\beta}$. You will find an example file that shows how a load spectrum with factors ($K_{H\beta}$, K_{γ}, etc.) can be defined in the "Example_DutyCycleWithFactors.dat" file in the `dat` sub-folder in the KISSsoft installation folder.

14.2.8.1 Type of load spectrum

The calculation of service life for load spectra is performed as specified in ISO 6336, Part 6, and is based on the Palmgren-Miner rule.

Three load spectra are predefined here, as shown in DIN 15020 (Lifting Appliances), along with many other standard spectra. You can enter your own load spectra.

A load spectrum consists of several elements (up to 50 in the database or an unlimited number if imported from a file). Each element consists of the frequency, speed, and power or torque. The data always refers to the reference gear you selected when you input the nominal power (Performance-Torque-Speed screen). The program stores these values as coefficients so that they are modified automatically when the nominal power changes.

If two speeds that are not equal to zero have been predefined for planetary stages, you can select two load spectra. In this case, only the speed factor is important for the second load spectrum.

NOTE

The load dependency of the K coefficients are included in the calculation (K coefficients: dynamic, face load and transverse coefficients). If you want to examine the result in greater detail, you will find the most interesting interim results in the Z18-H1.TMP text file (in the TMP directory).

14.2.8.2 Load spectra with negative elements

Load spectra with negative load bins (T < 0 and/or n < 0) can also be calculated as follows (this is only applied to bins whose alternating bending factor is YM=1.0).

IMPORTANT:

A load spectrum is considered to be negative if the non-working flank is placed under load.

Coefficient for torque	Coefficient for speed	Flank under load	Actual load bin
+	+	Working flank (*)	Evaluated as positive
+	-	Working flank (*)	Evaluated as positive
-	+	Non-working flank	Evaluated as negative
-	-	Non-working flank	Evaluated as negative

(*) Working flank as entered in the Strength tab

Table 14.10: Evaluation of a load bin, depending on the prefix operator

Under "`Details`" in the "Strength" section of the "`Load`" tab, you can select the following:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Consider only positive load bins
  - Consider only negative load bins
  - Check both cases and document the unfavorable case
- For calculating the tooth root safety
  - Evaluate all negative load bins as positive (as up to now)
  - For negative load bins, increase root stress by 1/0.7
  - Increase bending stress for positive load bins by 1/0.7
  - Check both cases and document the more realistic case

14.2.8.3 Classification of load spectra according to F.E.M. guideline

The "Supplementary Data" section in the report for a load spectrum contains the spectrum factor k_m, and the machine classes L, T and M for a load spectrum, according to F.E.M 1.001 [108 (see section "Bibliography" on page X-1214)].

The spectrum factor k_m lies between 0 and 1 and describes the load on a machine caused by active torque. If the load spectrum is 100% of the torque for 100% of the time, k_m is equal to 1. The spectrum class L is calculated from the calculated spectrum factor k_m, according to the table T.2.1.3.3 in F.E.M. 1.001, and increases from L1 to L4 with the active torque.

The application class T is determined from the entire duration of the load spectrum. The machine is assigned to one of the application classes T0 to T9, according to a duration of between zero and 50,000 operating hours (see table T.2.1.3.2 in F.E.M. 1.001). In each case, the predefined and achievable application classes T are output in the report.

A machine class M is determined according to the determined application class T and spectrum class L, according to table T.2.1.3.2 in F.E.M. 1.001. In each case, the predefined and achievable machine classes M are output in the report. Machine classes M1 (short operating time, low loads) to M8 (long operating time, high loads) are assigned, depending on the application class T and spectrum class L.

14.2.9 Calculate scuffing

The following selection options are available here:

☼ `Corresponding to the strength calculation method`

Here, if the DIN strength calculation method is used, scuffing is calculated as defined in DIN 3990-4. For all other calculation methods, scuffing is calculated as specified in ISO TR 13989.

☼ `Always according to ISO TR 13989`

Scuffing is always calculated as specified in ISO TR 13989.

☼ `Always according to DIN 3990-4`

Scuffing is always calculated as specified in DIN 3990-4.

Contrary to DIN 3990-4, the following formulae are used for the tooth mass temperature (analogous to ISO ST 13989):

$$\text{theMC} = \text{theoil} + XS * 0.70 * \text{theflaint}$$

$$\text{theMB} = \text{theoil} + XS * 0.47 * \text{theflamax}$$

For spray lubrication, XS=1.2 (otherwise 1.0). There is little point in multiplying the oil temperature (theoil) by the coefficient as specified in DIN 3990-4.

☼ `Always according to DIN 3990-4, similar to STplus`

STplus (Version 6.0) uses the original formulae according to DIN 3990-4 for the tooth mass temperature. In contrast, contrary to DIN 3990-4, the dynamic oil viscosity etaM is calculated with the oil temperature (instead of the tooth mass temperature).

Depending on which option is selected, the integral temperature and flash temperature are calculated according to the corresponding standard.

14.2.10 Calculating the internal temperature and the flash temperature

The calculation is performed for cylindrical gears and bevel gears. Here, you can specify whether the scuffing is calculated according to DIN or as specified in the selected strength calculation method as defined in ISO.

14.3 Factors

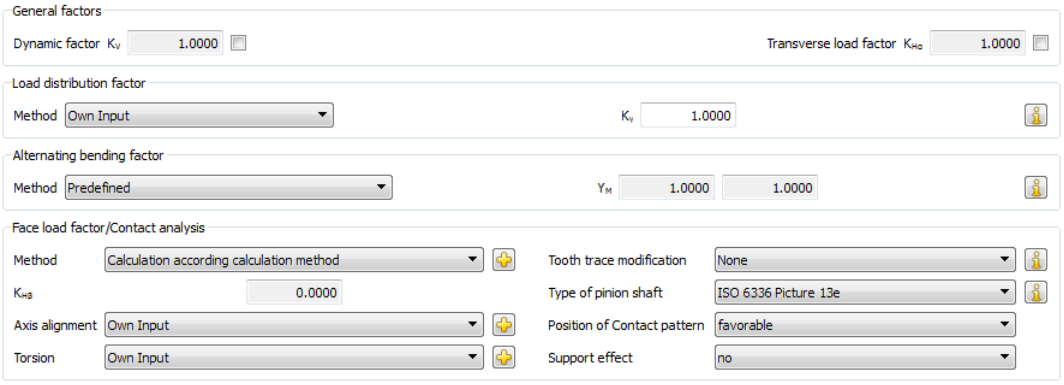


Figure 14.18: `Factors` input window for a cylindrical gear pair

The `Factors` input window is one of the standard (see page I-89) tabs.

14.3.1 Transverse coefficient

The transverse coefficient $K_{H\alpha}$ is calculated in accordance with the calculation method you selected. The transverse coefficient takes into account irregular contact characteristics across a number of teeth. When the contact ratio increases, the transverse coefficient also becomes larger depending on the predefined accuracy grade. A high contact ratio will result in a reduction of the root stresses. Large single pitch deviations, the transverse coefficient will compensate this effect.

In unusual cases, the transverse coefficient will be unrealistically high. If you want to reduce the transverse coefficient in this situation, simply click the checkbox to the right of the input field. You can then change this value.

14.3.2 Dynamic factor

The dynamic factor takes into account additional forces caused by natural frequencies (resonance) in the tooth meshing. It is usually calculated using the calculation method you selected, however you can also input the value if it has already been derived from more precise measurements. To change the value, click the checkbox next to the input field.

14.3.3 Load distribution coefficient

The load distribution coefficient takes into consideration the uneven load distribution across multiple planets or idler gears. In this case the load is multiplied by this coefficient. Dimensioning suggestion according to AGMA 6123-B06:

	Number of planets									
Application	2	3	4	5	6	7	8	9	Quality	Flexible
Level									ISO 1328 AGMA 2015	Mounting
1	1.16	1.23	1.32	1.35	1.38	1.47	1.60	-	≥ 7	without
2	1.00	1.00	1.25	1.35	1.44	1.47	1.60	1.61	5 ÷ 6	without
3	1.00	1.00	1.15	1.19	1.23	1.27	1.30	1.33	≤ 4	without
4	1.00	1.00	1.08	1.12	1.16	1.20	1.23	1.26	≤ 4	with

Table 14.11: Load distribution coefficient K_γ defined by the number of planets

Level of application	**Explanation**
1	Typical of large, slow-turning planetary gear units
2	Average quality, typical of industrial gears
3	High quality gears, e.g. for gas turbines

Table 14.12: Meaning of the level of application

NOTE

Level 2, or higher, requires at least one floating element.
Level 3, or higher, requires a flexible gear rim.
In a flexible assembly, the planets must be mounted on flexible pins/shafts or on bearings with couplings.

The `calculated according to AGMA 6123` method is used to calculate the load distribution coefficient K_γ for application level 1 ÷ 3 depending on the accuracy grade and the number of planets.

If a different load distribution coefficient is input for each element when load spectra are in use, you should select the `Own input, per load stage` method.

14.3.4 Alternating bending factor

The tooth root strength calculation is dedicated for the pulsating load on the tooth root. However, in some cases, the tooth root is subject to alternating bending loads (e.g. a planet gear in planetary gear units). In this scenario you can change the alternating bending coefficient of individual gears by selecting either the `Own input` or `Own input, per load spectrum element` methods. As an alternative to transferring these values directly, select the `Calculate in accordance with ISO 6336-3 Annex B` method to calculate the coefficient. To do this, you must then open the `Rating` tab, go to the Load spectrum section, and input the f_{low} and f_{high} parameters for each gear. f_{high} must always have the fixed default value of 100%.

ISO 6336-5:2003, Section 5.3.3 and DIN 3990-5, Section 4.3 have 0.7 as the value Y_M for pure cyclic load. In ISO 6336-3:2006, Annex B, the stress ratio *R* for idler and planetary gears is taken into account by using this formula:

$$Y_M = \frac{1}{1 - R \cdot \frac{1 - M}{1 + M}} \tag{12.16}$$

$$R = -1.2 \cdot \frac{f_{low}}{f_{high}} \tag{12.17}$$

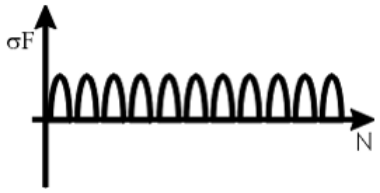
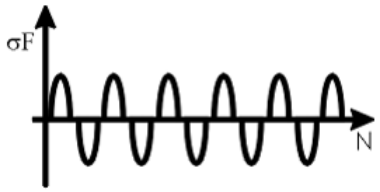
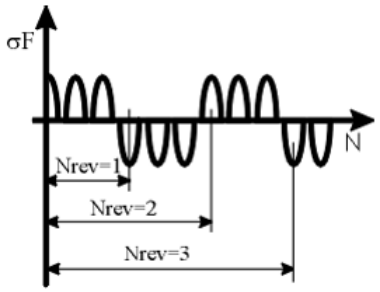
f_{high}	Load on the flank side that is subject to the higher load (must always have the fixed default value of 100%)
f_{low}	Load on the flank side that is subject to the lower load
M	Dimensionless number depending on the type of treatment and load type (see Table B.1 in ISO 6336:2006-3, Annex B)
R	Stress ratio
Y_M	Alternating bending factor

Treatment	**Endurance strength**	**Coefficient for static proof**
Steels		

case-hardened	0.8 ÷ 0.15 Y_S	0.7
Case-hardened and shot peened	0.4	0.6
Nitrided	0.3	0.3
Flame/induction-hardened	0.4	0.6
Not surface-hardened steel	0.3	0.5
Cast steel	0.4	0.6

Table 14.13: Mean stress ratio M as specified in Table B.1 - Mean Stress Ratio - in ISO 6336:2006-3

According to Linke [58], the alternating bending factor (described there as Y_A) is determined as shown in Figure 14.19. For plastics, Niemann recommends 0.8 for laminated fabric and 0.667 for PA (polyamide) and POM (polyoxymethylene).

Operating Mode	Alternating Bending Factor (Mean Stress Influence Factor) Y_M	Load Direction
Pulsating	1	
Alternating	0.7 (1) 0.65 (2)	
Oscillating	$0.85 - 0.15 \cdot \frac{\log N_{rev}}{6}$ (1) $0.85 - 0.20 \cdot \frac{\log N_{rev}}{6}$ (2) $(1 \le N_{rev} \le 10^6)$ 0.7 (1) 0.65 (2) $(N_{rev} \ge 10^6)$	

(1) Linke, H.: Stirnradverzahnung, Carl Hanser Verlag, 1996.
(2) Linke, H.: Stirnradverzahnung, Carl Hanser Verlag, 2010.

Figure 14.19: Alternating bending factor in accordance with Linke [58]

14.3.4.1 Load spectrum with changing torque

Load bins can also be entered with negative torques.

The problem:

until now, no calculation guidelines have been drawn up to describe how to calculate gears with changing load spectra.

The only unambiguous case is when a change in torque takes place, during every cycle (and in each element of the collective). At this point, the load change corresponds exactly to a double-load with +torque and then with –torque. This instance can be calculated correctly by entering the load spectrum of the +moments and the alternating bending factor YM for the tooth root. The flank is also calculated correctly, because the +moments always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal + moments and –moments for the tooth root are to be handled. For this type of case, only the +moments are considered for the flank (with the prerequisite that the +moments are equal to, or greater than, the –moments).

Note about handling load spectra with reversing torque:

A load progression as represented in the figure below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

13 loads with 100% of the nominal load (100 Nm) on the left flank, then

9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

11 load cycles with 100% load, positive torque, pulsating; then

1 load cycle with 100% load on the left and 80% load on the right; then

7 load cycles with 80% load, negative torque, pulsating; then

1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	Torque	Load left flank	Load right flank
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

Table 14.14: Load progression shown as a load spectrum

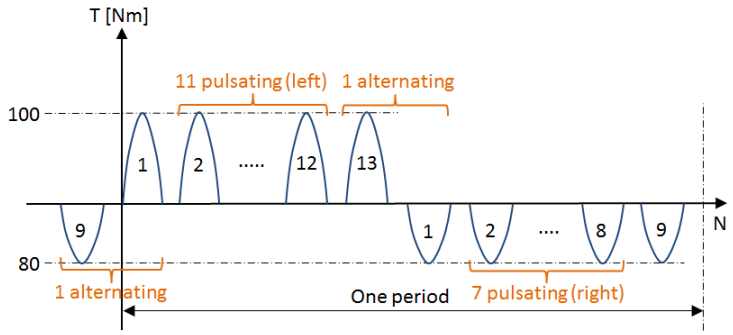


Figure 14.20: Load progression

14.3.5 Face load factor

The face load factors $K_{H\beta}$,$K_{F\beta}$,$K_{B\beta}$ take into consideration the influence of an uneven load distribution over the facewidth on the contact stress, tooth root stress and scuffing stress. You can specify that the face load factor is either to be set as a constant value or calculated from other values. If you already know the face load factor $K_{H\beta}$, select the `Own input` method and input this value. During a calculation according to DIN/ISO, click the  button to open the `Define face load factor` window, in which you can use a number of parameters to calculate the value you require.

The usual setting here is "Calculation according to calculation method". The face load factor is then calculated according to the formulae used in the strength calculation standard (ISO, AGMA or DIN). You will need to input some values for this. These values are displayed on the right of the window (tooth trace

modification, etc.) and are described in the sections that follow. You can input other values by clicking the  button in the "Define face load factor" window.

For the "**Coefficient K' with support effect**" entry: the pinion has the effect that it stiffens the shaft (support effect), if d_1/d_{sh} >1.15 and the pinion is manufactured with a fixed press fit or shaft/pinion on the piece.

The formulae proposed in the standards for defining face load factor KHb enable you to determine KHb very quickly (but only empirically, and therefore not very accurately). The KHb coefficient calculated using these formulae is usually higher than it actually is, so the calculated value is therefore on the conservative side. If you think the face load factor is too high (> 1.5), it is a good idea to perform a more accurate calculation. You can use the "Calculation according to ISO 6336 Annex E" method to do this.

Although the "Calculation according to ISO 6336 Annex E" method is very accurate, it requires quite a lot of time and effort. As described in [44], it calculates any gaping in the meshing, and therefore defines the load distribution over the entire facewidth. To perform this calculation, you will need to know the exact dimensions of the shafts and support. Click the "Define axis alignment" button to input the shaft values stored in the shaft calculation program for the relevant shafts.

The "Calculation with manufacturing allowance according to ISO 6336 Annex E" method is the most accurate. However, if you use this method, you must also enter the toothing tolerance fHb (tooth trace deviation over the carrying facewidth) and the axis alignment tolerance fma (angular deviation of the axis alignment in the plane of action). In this case, the load distribution over the facewidth is calculated 5 times (according to [44]): First without deviation, then sequentially using (+fHb,+ fma), (+fHb,- fma), (-fHb,+ fma) and (-fHb,- fma). The greatest face load factor KHb determined here is then taken as the end result.

NOTE

See `Module-specific settings -> Face load factor` for the settings involved in the calculation according to ISO 6336 Annex E.

If you want to calculate the face load factor by applying load spectra for each element, select either the `Own input, per load stage`, `Calculation according calculation method` or `Calculation with/without manufacturing allowance according to ISO 6336-1 Annex E, per load stage` method.

In the cylindrical gear pairs, three- and four-gear chains, and planetary systems, calculation module, shaft calculation files can be used to calculate the relative displacement between the gear flanks more accurately, based on the corresponding shaft bending lines (see page II-340). The torque, power, and force, for all the load

elements involved in the shaft calculation are then modified according to the partial load coefficient w_t.

This means you can include any torsion that occurs in the gear. Here the calculation assumes a solid cylinder or hollow cylinder (external diameter = root circle + 0.4*normal module or operating pitch circle, depending on what has been predefined under "Settings", bore = inside diameter) is involved. In other words, the inside diameter is taken into account and the torque on one side is zero. The torque is distributed in a linear fashion along the facewidth (parabolic course of deformation by torsion). You can select which side is to be subjected to torsion moment. In this case, I and II refer to the same side, as is also the case when you enter the toothing corrections. The increase in torque for a sun in planetary stages is taken into account by using multiple contacts (several planets). Multiple contact is not taken into consideration in any other configuration (e.g. for pairs of gears). In such situations, the correct torque curve can be used if the deformation is taken from the shaft calculation.

The facewidth is divided into slices, to help you calculate the face load factor as defined in ISO 6336, Annex E:

You can set the accuracy of the face load factor calculation according to Annex E in the "Define number of slices" dialog. Click the plus button next to the calculation method to open this dialog.

14.3.5.1 Tooth trace modification

You can achieve balanced contact characteristics if you perform targeted tooth trace modifications. Figure 14.21 shows the two most frequently used modifications.

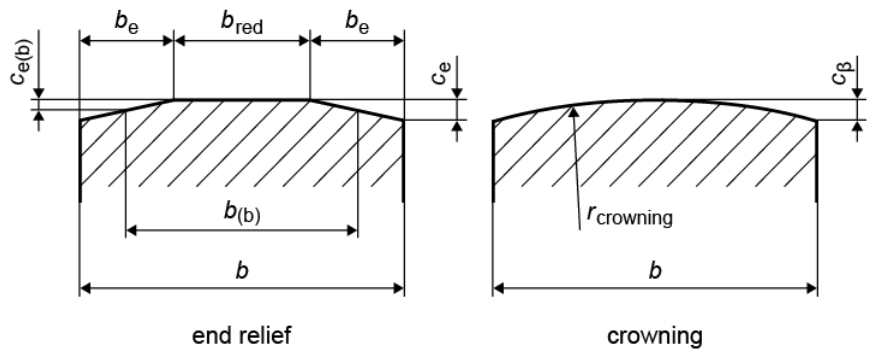


Figure 14.21: End relief and crowning

14.3.5.2 Cylindrical gear pairs

The calculation, as specified in ISO 6336, is based on an approximate estimate of the pinion deformation. In many cases, this is extremely inaccurate and usually results in face load factors that are much too high.

The face load factor is the ratio between the maximum and average line load. The basic equation used for the face load factor corresponds to equation (41) in the standard[4]:

$$K_{H\beta} = \frac{(F_m / b)_{\max}}{F_m / b} = 1 + \frac{F_{\beta y} \cdot c_{\gamma\beta}}{2 \cdot (F_m / b)} \qquad (14.4)$$

The effective tooth trace deviation $F_{\beta y}$, see equation (52) in the standard, is defined with the inclusion of a linearized, specific deformation component f_{sh}. The multiplier 1.33 in the equation stands for the conversion of the linearly determined deformation progression into the real parabolic progression - see equation. (14.5).

$$F_{\beta y} = F_{\beta x} \cdot \kappa_\beta = (1.33 \cdot f_{sh} + f_{ma}) \cdot \kappa_\beta \qquad (14.5)$$

The manufacturer component of the tooth trace deviation f_{ma} is derived from tolerances specified by the manufacturer. If a usual procedure for checking the accuracy grade is used, you can apply this formula (equation (64) in the standard):

$$f_{ma} = \sqrt{f_{H\beta 1}^2 + f_{H\beta 2}^2} \qquad (14.6)$$

If you have used KISSsoft's shaft calculation software to calculate the exact tooth trace deviation due to deformation (torsion and bending) in the plane of action, you can correct the approximate value f_{sh} extrapolated from the standard and therefore calculate the face load factors much more precisely! The formula as specified in

[4] *The equation numbers used in this section refer to ISO 6336:2006*

ISO 6336 only applies to solid shafts or hollow shafts that have an inside diameter that is less than half of the external diameter.

In Method C2, the face load factor is calculated using these equations:

Size	Drop-down list	Selection	Equation	No.
$K_{H\beta}$				(8.04)/ (8.06)
F_β				(8.08)
F_β	position of the contact pattern	not verified or inappropriate favorable optimal		(8.26) (8.27) (8.28)
f_{sh}				(8.39)
f_{sh0}	Tooth trace modification	None	0.023 • γ	(8.31)
		Crowning	0.012 • γ	(8.34)
		End relief	0.016 • γ	(8.35)
		Solid	0 • γ	a)
		Slight crowning	0.023 • γ	b)
		Helix angle modification	0.0023 • γ	b)
		Crowning + helix angle correction	0.0023 • γ	b)
γ	Gearing	straight/angled helical		(8.32) (8.33)
f_{ma}	Tooth trace modification	None	1.0 • $f_{H\beta}$	(8.51)
		Crowning	0.5 • $f_{H\beta}$	(8.53)
		End relief	0.7 • $f_{H\beta}$	(8.52)
		Total tooth trace modification	0.5 • $f_{H\beta}$	a)
		Slight crowning	0.5 • $f_{H\beta}$	b)
		Helix angle modification	1.0 • $f_{H\beta}$	b)
		Crowning + helix angle correction	0.5 • $f_{H\beta}$	b)

Table 14.15: Overview of equations used according to DIN 3990:1987

a) same as DIN 3990, Equation (6.20)
b) same as ISO 9085, Table 4

Size	Drop-down list	Selection	Value		No.
$K_{H\beta}$					(39)/ (41)
F_β					(43)
		not verified or inappropriate			(52)
F_β	position of the contact pattern	favorable			(53)
		optimal			(56)
f_{sh}					(57)/ (58)
f_{ma}					(64)
		None	1 /	1	
		Crowning	0.5 /	0.5	Table 8
		End relief	0.7 /	0.7	
B_1/B_2	Tooth trace	Full	0 /	0.5	(56)
	modification	Slight crowning	1 /	0.5	
		Helix angle modification	0.1 /	1.0	Table 8
		Crowning + helix angle correction	0.1 /	0.5	

Table 14.16: Overview of equations used according to ISO 6336:2006

Type of pinion shaft

Load as defined in ISO 6336:2006, Figure 13 (DIN 3990/1, Figure 6.8) or the bearing positioning is shown in Figure 14.22.

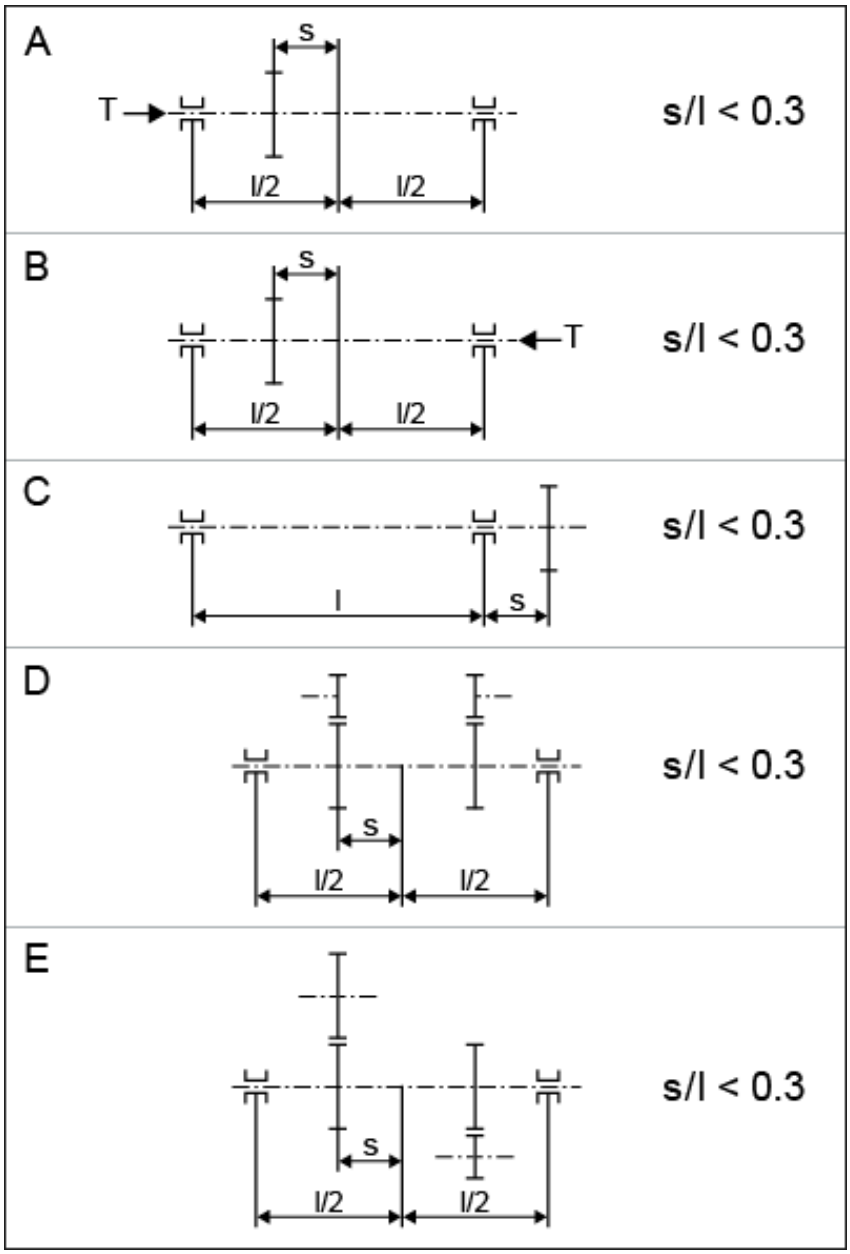


Figure 14.22: Load as defined in ISO 6336:2006, Figure 13.

Load in accordance with AGMA 2001

Definition of s and s_1 according to AGMA 2001, Figure 13-3. Figure 14.23 shows the bearing positioning as described in AGMA 2001.

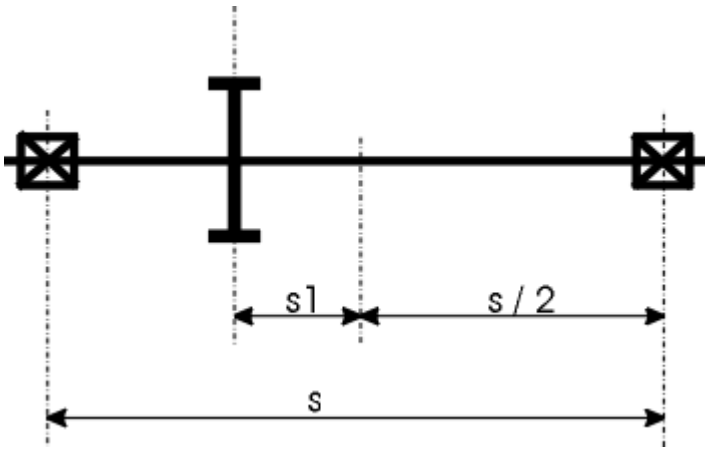


Figure 14.23: Load as defined in AGMA 2001, Figure 13-3

14.3.5.3 Planetary stages

The face load factor for planetary stages is calculated in a different way than for cylindrical gears. The deformation component f_{sh} is derived from the deformation of the mating gears on the shaft due to torsion and bending. In order to simplify the situation for a pinion-gear pair, only the pinion deformation (which is much greater) is taken into account.

Planetary stages are subject to the following sizable deformation: Since the sun has several tooth meshings, all radial forces are canceled out No bending takes place because deformation is caused solely by torsion. However, the multiple meshing which corresponds to the number of planets means this is greater than for normal pinion shafts. - A planet gear has two meshings with opposed torques, which prevents deformation due to torsion. Bending may be calculated in the same way as for pinion shafts; however, the circumferential force must be doubled because of the sun/planet and planet/internal gear. - In most cases, rim deformation can be ignored. As a result, the torsion at the pinion and the bending at the planet bolt must be taken into consideration for sun/planet meshing whereas, for planet/internal gear, only the bending at the planet bolt is important. For most support arrangements for planets, bending can be determined analytically using a procedure similar to that specified in ISO 6336. Figure <12>.24 shows the four most common cases.

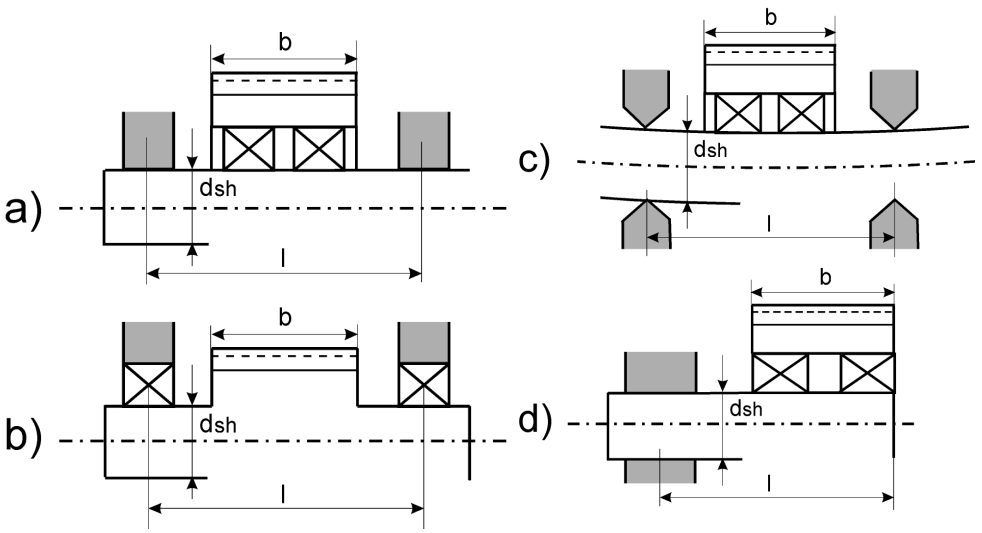


Figure 14.24: Support arrangement for planets

a) Planets mounted with fixed clamped bolts on both sides

b) Planets are on bolts, which have flexible bearings in planet carrier

c) Planets mounted with gently tightened bolts (flexible bearings) on both sides

d) Planets mounted with fixed clamped bolts on one side

Configuration	**ISO 6336**	**DIN 3990**	**AGMA 2001**
a	Part 1,	Formulae	Chapter 15, (37)
	Annex D	6.20/6.21/6.24/6.25/	
b	Part 1,	Formulae	Chapter 15, (37)
	Annex D	6.24A/6.24B/6.25A/6.25B	
c and d	Part 1,	Formulae as defined in Part 1,	Chapter 15, (37)
	Annex D	Annex C, see [49]	

Table 14.17: Configuration of planetary stages as defined in ISO, DIN and AGMA

For ISO 6336 see also the explanation in [49].

Equations (14.7a - 14.7d) show the bending components in relationship to the distance x from the planet's facewidth. As we are only interested in bending variation across the facewidth, the constant term was left out of the equations so that f_b(x =

0) is zero. Similar formulae can be found in other technical documentation [38]. The following equations apply for cases a to d, as illustrated in Figure 1.24.

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(3l - 6b + b^2\right)}{48} + \frac{b^2 x\left(3l - 4b + b^2 / l\right)}{48} \right] \quad (14.7a)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(l - b\right)}{8} + b^2 x\left(\frac{l}{8} - \frac{b}{12} \right) \right] \quad (14.7b)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(l - b\right)}{8} + b^2 x\left(\frac{l}{8} - \frac{b}{12} \right) \right] \quad (14.7c)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{6} - \frac{b^2 x^2}{4} + \frac{blx\left(l - b\right)}{2} \right] \quad (14.7d)$$

The sun's deformation due to torsion, as described in equation (14.8), can be calculated from Annex D (f_T in accordance with formula D.1).

$$f_{tso} = p \cdot \frac{8}{\pi} \cdot \frac{F_m / b}{0.39 \cdot E_{so}} \left(\frac{b}{d_{so}} \right)^2 \cdot \frac{x}{b} \left(1 - \frac{x}{2b} \right) \quad (14.8)$$

In order to stay as close as possible to the methods used in ISO 6336 (and be able to apply formula 2), the average deformation components f_{bmpla} (bending at the planet) and f_{tmso} (torsion at the sun) will be determined.

$$f_{tso} = \frac{1}{b} \int_0^b f_{tso}(x) dx \qquad f_{bmpla} = \frac{1}{b} \int_0^b f_{bmpla}(x) dx \quad (14.9)$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(-\frac{b}{5} + \frac{l}{6} + \frac{b^2}{18l} \right) \quad (14.10a)$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(\frac{l}{3} - \frac{b}{5} \right) \quad (14.10b)$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(\frac{l}{3} - \frac{b}{5} \right) \quad (14.10c)$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^2}{4} \left(\frac{b^2}{5} - bl + l^2 \right) \quad (14.10d)$$

$$f_{tmso} = p \cdot \frac{8}{3\pi} \cdot \frac{F_m / b}{0.39 E_{so}} \cdot \left(\frac{b}{d_{so}} \right)^2 \quad (14.11)$$

According to ISO 6336:2006, equation D.8, the linearized deformation components of the tooth trace deviation f_{sh} (in mm) will be:

$$f_{sh}(Paarung\ Sonne - Planet) = 2000 \cdot (f_{tmso} + f_{bmpla}) \quad (14.12)$$

$$f_{sh}(Paarung\ Planet - Ring) = 2000 \cdot f_{bmpla} \quad (14.13)$$

This can then be used with equations (14.4) and (14.5) to calculate face load factors for the sun/planet and planet/gear rim.

formula sym-bol	**Unit**	**Meaning**
b	mm	Meshing facewidth
$c\gamma\beta$	N/(mm μm)	Meshing stiffness
d_{pla}	mm	Planet reference circle
d_{sh}	mm	Planet shaft diameter
d_{so}	mm	Sun reference circle
E_p	N/mm^2	Young's modulus for planet bolt/shaft
E_{so}	N/mm^2	Young's modulus for sun
f_{bpla}	mm	Planet shaft deflection
$f_H\beta$	μm	Helix slope deviation according to ISO 1328

$f_{m\alpha}$	μm	Tooth trace deviation
		manufacture error
f_{sh}	μm	(Linearized) deformation components of the
		tooth trace deviation
f_{tso}	mm	Sun torsion deviation
F_m/b	N/mm	Average line load
$(F_m/b)_{max}$	N/mm	Maximum local line load
$F\beta_y$	μm	Actual tooth trace deviation
$K_{H\beta}$	[-]	Face load factor
l	mm	Planet bolt/shaft length
p	mm	Number of planets
x	mm	Distance to the left side of the facewidth
$\kappa\beta$	[-]	Run-in factor

Table 14.18: Overview of formula symbols

14.3.5.4 Calculation of KHβ with manufacturing errors

According to ISO 6336-1(E), the lead variation (fHb) and shaft misalignment (fma) errors are also taken into account in the plane of action. In such a case, their combined effect is taken into account for the flank gap in five cases:

- Case 1: fma = fHb = 0, i.e. no error
- Case 2: fma = |fma|, fHb = |fHb|, so positive values for both errors
- Case 3: fma = +|fma|, fHb = -|fHb|
- Case 4: fma = -|fma|, fHb = +|fHb|
- Case 5: fma = -|fma|, fHb = -|fHb|, so negative values for both errors

The face load factor $K_{H\beta}$ is calculated for all five cases, and the maximum value is selected as the face load factor of the gear pair.

The positive direction always lies in the direction of the pinion's material, seen from a common point of contact.

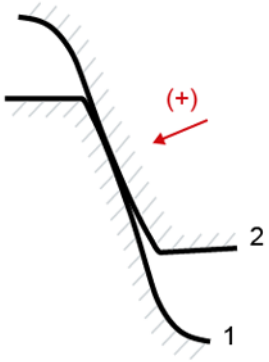


Figure 14.25: Definition of the positive direction

In all five cases, the manufacturing error is documented in the report and in the gaping and load distribution graphics.

Proposed value for fhb and fma

Click on the sizing button next to the input field for |fHb| to display suggestions of usable data for fHb and fma.

"Maximum" is the largest possible values for fHb and fma. The values are derived from the fHbT (helix slope deviation) tolerances of the two gears and from the axis alignment tolerance ($f_{\Sigma\beta}$ and $f_{\Sigma\delta}$).

The *"Statically evaluated"* proposal displays the probable maximum values (99.7% probability). This proposal is calculated as follows:

$$f_{tot} = 3 * \sqrt{(f_{HbT1})^2 + (f_{HbT2})^2 + (f_{\Sigma})^2}$$

$$f_{Hb} = 3 * \sqrt{(f_{HbT1})^2 + (f_{HbT2})^2}$$

$$f_{ma} = f_{tot} - f_{Hb}$$

14.3.5.5 Defining the misalignment for individual elements

The following elements are assumed in a planet system:

- Sun wheel
- Planet carrier
- N planetary gears with the corresponding n pins
- Internal gear

You can specify the position of these elements in the gear unit and the corresponding misalignment in the "`Define axis alignment`" dialog. To open this dialog, click on the "Axis alignment" button in the "`Factors`" or "`Contact analysis`" tab.

You can define more parameters in the "`Axis alignment, proportional`" tab for the load-specific alignment of system elements:

- Tilting of the sun to the gear axis (see Figure 14.26). If no shaft file is used, the sun can be handled as a "floating sun".
- Tilting of the planet carrier to the gear axis (see Figure 14.27).
- Tilting of the planet pin to the planet carrier in peripheral direction dt and in radial direction dr (see Figure 14.28). To model a carrier deformation due to torsion, you must first define a value for dt.
- The tilting of the planet gear is relative to the planet pin axis. The positive misalignment (in peripheral direction dt and radial dr) is defined according to the convention in Figure 14.28.
- The tilting of the internal gear relative to the gear axis (see Figure 1426). The conical extension of the internal gear can also be taken into consideration.

- The deformation of the planet pin is caused by the twisting of the planet carrier. If the direction of torque has been input in the "Torsion" tab, the software checks the values and issues a warning message if the prefix for dt has not been entered correctly.
  If the direction of torque has been input in the "Torsion" tab, the software assumes that dt represents the twisting of the carrier due to torque. For this reason, the prefix operator of dt is changed when KHβ is calculated for load spectra with a negative load factor

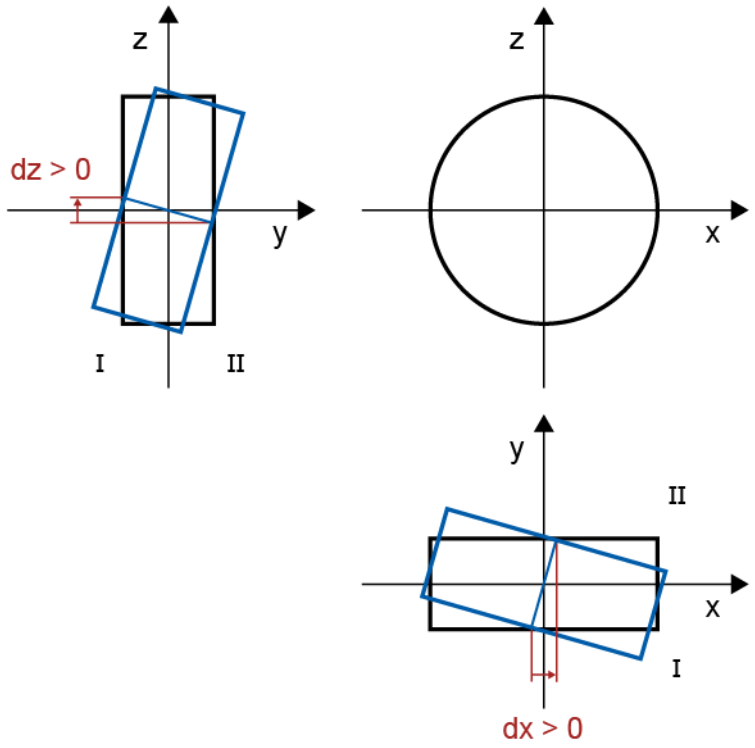


Figure 14.26: Tilting of the sun and internal gear to the gear axis

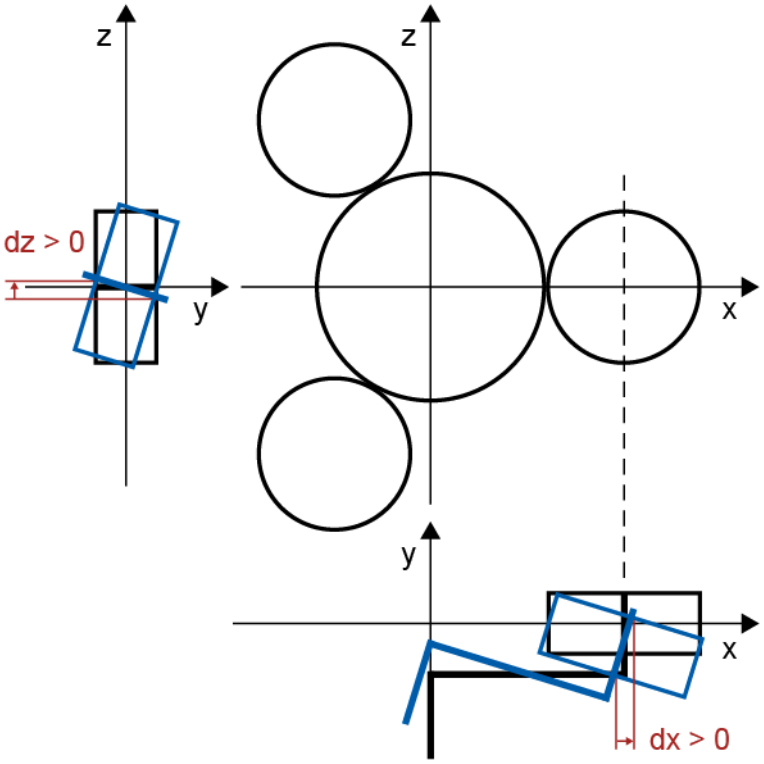


Figure 14.27: Tilting of the planet carrier to the gear axis

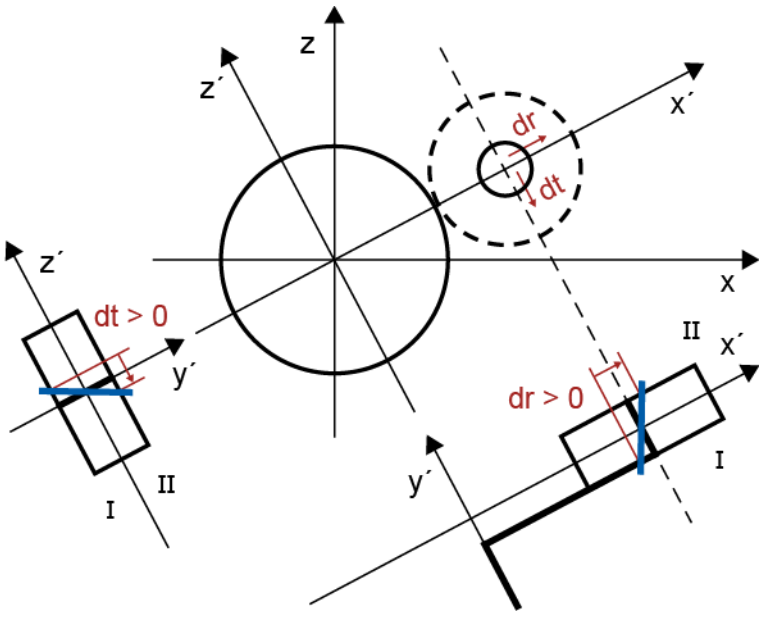


Figure 14.28: Tilting of the planet pin to the planet carrier

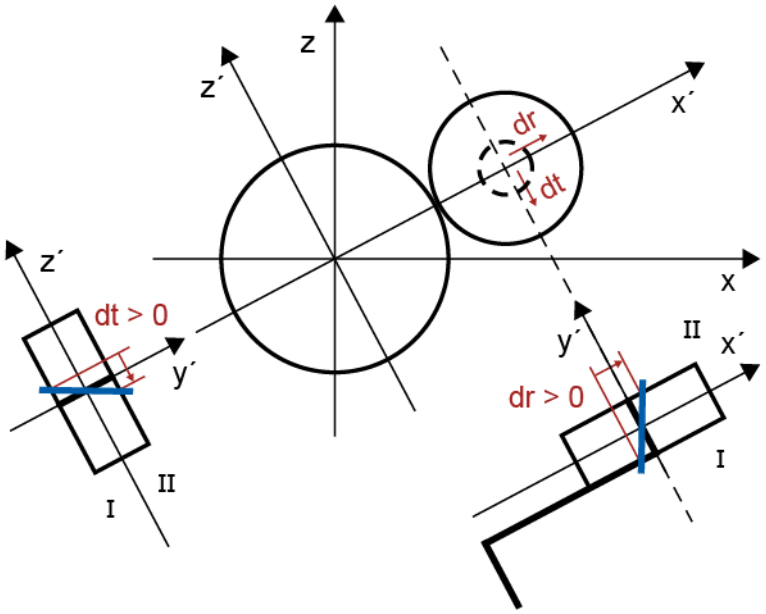


Figure 14.29: Tilting of the planet to the planet pin

You can also use shaft files to define the alignment of all the shafts, except the planet pin. The shaft files undergo the same checks as a gear pair. For example, the value input for gear torque in the shaft calculation files must match the value entered for the gears in the calculation module. The carrier shaft is characterized by its two couplings: one coupling transfers the torque to the sun wheel and the other transfers the torque to the internal gear. The "effective diameter" for both couplings must be the same as the sun-planet center distance. The "length of load application" must also be appropriate for the facewidth of the planet gear. If a shaft file is used for the sun, planet or internal gear, you must click on an additional plus button to select the meshing that must be taken into consideration.

The proportional axis alignment is scaled with the partial load wt (for contact analysis), or with the ISO factors K_V,K_A and K_γ.

The angle to the first planet Θ defines where the first planet gear must be located for each system definition. Every one of the subsequent planetary gears must have an angular offset of $2\pi / N$ to the previous gear. The load distribution on the planet for the specified planet carrier misalignment is dependent on the position of the planets. Modifying Θ will also change KHb, which is why this entry enables you to calculate the "worst case".

You can define the non load-specific inclination/deviation error of axis in the "Axis alignment, constant" tab.

In the "Torsion" tab, you define the side from which torque is introduced to the system or the side from which it is produced (depending on whether the element is a driving element or a driven element). You can select one of the following 3 options for inputting the direction of torque:

- Not taken into account
- Torque is applied/produced on side I
- Torque is applied/produced on side II

Each configuration is also displayed as a graphic so that the user can check their entries.

If a shaft file is used to define the shaft deformation, the torque is calculated automatically from the results of the shaft calculation.

The planet carrier is usually more complicated than is specified in the shaft calculation. For this reason, the carrier torsion is often greater than determined in the shaft calculation. Consequently, you can either take the torsion deformation value from the shaft calculation or enter it under dt for "planet pin" (or use an FEM calculation to determine it).

Calculating planet carrier deformation with FEM

The deformation of the planet carrier causes the planet pin to become misaligned (the pin tilts at dt and dr relative to the planet carrier axis). Use the Finite Elements Method (FEM) to calculate the exact planet carrier deformation. A range of different options are available here:

- The calculated FEM results can be input directly as point coordinates and point deformations (one node for each of side I and side II on a two-sided planet carrier; two nodes on one side for a single-sided planet carrier, (see Figure)

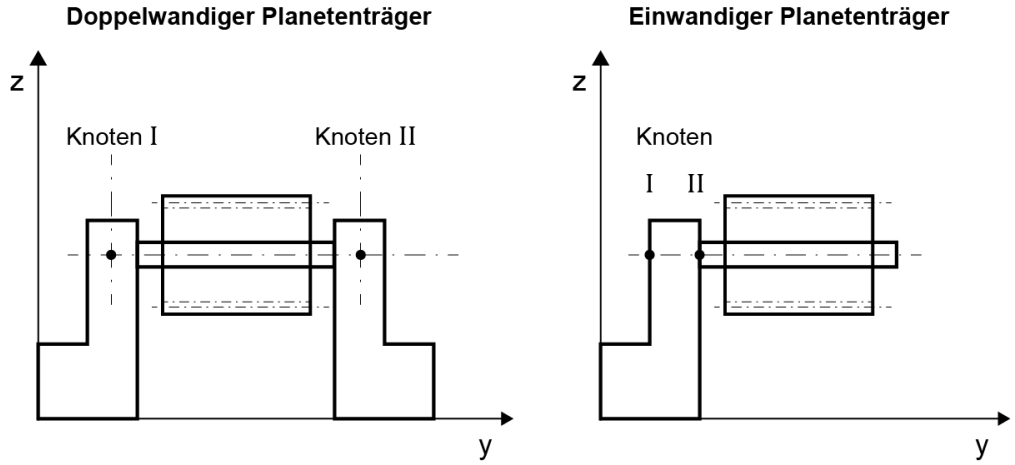


Figure 14.30: Planet carrier tab

- Import the file with the FEM results for the planet carrier deformation. The deformation in both nodes are then extracted from this file. The node coordinates do not need to be specified exactly. The deformation data of the adjacent node is transferred.

Input some of the planet carrier's fundamental dimensions. KISSsoft then generates the carrier in 3D and uses the relative torque to define the planet carrier's deformation.

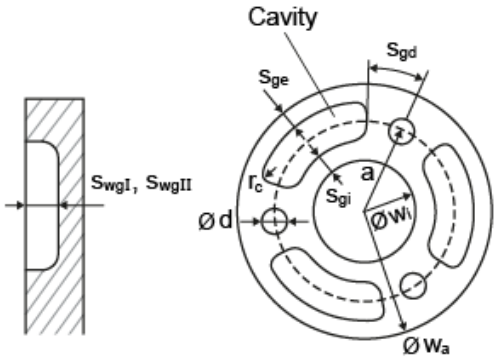
Input this data:

Single- or two-sided planet carrier
Pin diameter (d)
Coefficient for the external diameter of the planet carrier (f_{wa})
Coefficient for the inside diameter of the planet carrier (f_{wi})
Coefficient for the wall thickness of the planet carrier, which may be different for side I and side II (f_{swl} and f_{swll})
Width coefficient of the planet carrier (f_{bpc})
Coefficient for planet carrier's connector (f_{dcon})
Coefficient for the planet carrier's internal connector (f_{dicon})
External flange diameter on side I (d_{faI})
Flange length on side I (L_{fl})
Flange wall thickness on side I (S_{wfI})
External flange diameter on side II (d_{faII})
Flange length on side II (L_{fII})
Flange wall thickness on side II (S_{wfII})

Material of planet carrier (selected from the available database)

These coefficients can all be input under "Details", either as factors or directly as dimensions.
You can also click on the "Dimension planet carrier" and "Dimension flange" buttons to display standard entries for this data.
Please also note that mesh refinement can be also input by the user.

The coefficients and dimensions are shown in greater detail in the next figure.
Depending on how the direction of torsion is entered, side I or side II may not be required for a single-sided planet carrier.



N = Number of Planets

$$Ø w_a = 2 \cdot a \cdot f_{wa} \qquad r_a = f_r \cdot a \cdot 2 \cdot \frac{\pi}{N}$$

$$Ø w_i = 2 \cdot a \cdot f_{wi} \qquad s_{gd} = f_{gd} \cdot a \cdot 2 \cdot \frac{\pi}{N}$$

$$s_{wI} = b \cdot f_{swI} \qquad s_{ge} = f_{ge} \cdot \frac{(w_a - w_i)}{2}$$

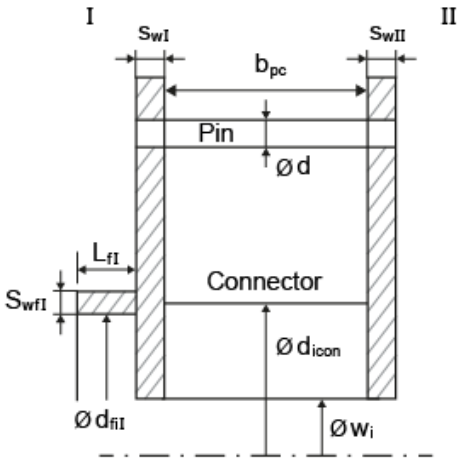
$$s_{wII} = b \cdot f_{swII} \qquad s_{gi} = f_{gi} \cdot \frac{(w_a - w_i)}{2}$$

$$b_{pc} = b \cdot f_{bpc} \qquad s_{wgI} = f_{wgI} \cdot s_{wI}$$

$$d_{con} = d_{apl} \cdot f_{dcon} \qquad s_{wgII} = f_{wgII} \cdot s_{wII}$$

$$d_{icon} = 2 \cdot a \cdot f_{dicon}$$

$$t = d_{apl} \cdot f_b$$



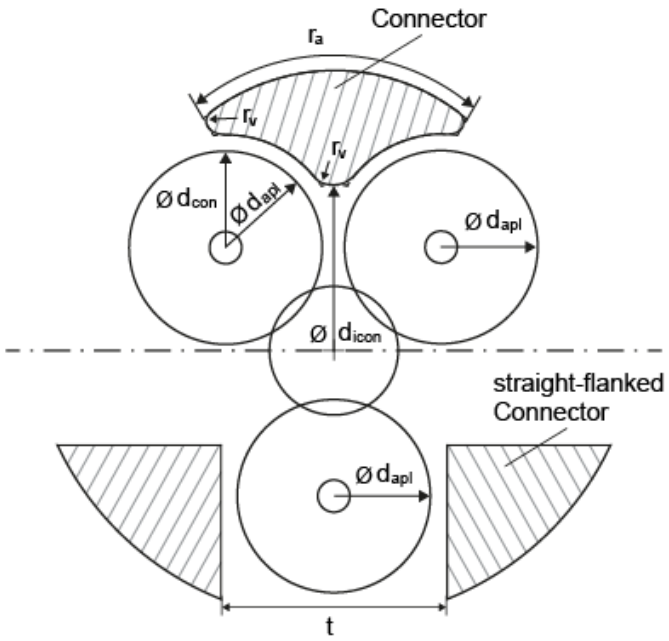


Figure 14.31: Coefficients and dimensions for planet carriers

In addition to the carrier variants shown above, you can also input a step model of the carrier directly.

The point to remember here is that the carrier is clamped on the inside diameter of the flange. If no flange is present, it is clamped on the inside diameter of the planet carrier. If both these diameters are identical, it is clamped along the entire length of the flange and the inside diameter of the carrier. If a step model is used, it is clamped at the specified flange diameter.

The FEM solver called by KISSsoft in the background is the Code_Aster open source solver, which you can obtain from the website at www.code-aster.org/. The preprocessor used to build the FE model is also an open source program, called Salome, located at www.salome-platform.org. To ensure you have the correct versions, install both programs from the KISSsoft DVD, or download them from the KISSsoft website. The only precondition for using this method is that Java is installed. (You can download it from www.java.com). Also ensure that the bin path, where java.exe is located, is set correctly in KISSsoft, in 'Extras->Settings->Directories' JAVADIR.
Remember that the folders for these FEM programs (FEPreProcessor and FESolver) can be copied to any location. In KISSsoft, this location should be defined in `Extras->Settings->Directories->FEMDIR` (this is usually the KISSsoft installation folder). In some computer configurations, MS-DOS naming conventions must be used.
Both the solver and the preprocessor are distributed under the GPL license, like the versions that can be found on the websites mentioned above. (More details about this license can be found in these programs' installation folders in KISSsoft, and on their websites).

II.14.3.5.5..1 Model and results view

To view the FEM model or the FEM results, start the Salome program. To do this, click on "Open Salome" after the calculation has finished. You can then either open the "PlanCarr.unv" file with the FEM mesh or the "PlanCarr.0.med" file with the FEM results. To view the mesh in Salome, select the Mesh module from the drop-down list in the Salome toolbar, and then select "File->>Import->>UNV file". To view the results, select the ParaViS module in Salome and then open the "med" file mentioned above. More information about how to work with meshes and the results files in Salome is provided in a special instruction file "kisssoft-anl-100-E-FEM-Planetencarrier.docx". You can request this documentation from the Hotline.

14.3.6 Taking into account shaft bending (face load factor and contact analysis)

Shaft bending can be taken into account using the "Define axis alignment" dialog. You can access this dialog either from the "Factors" tab (provided that either the "Calculation according to ISO 6336 Annex E" or "Calculation with manufacturing allowance according to ISO 6336 Annex E" option is selected, in the "Face load factor" field) or the "Contact analysis" tab.

14.3.6.1 Main settings

The `Define axis alignment` dialog is where you define the proportional and constant deviation error of axis ($f_{\Sigma\beta-p}$, $f_{\Sigma\beta-c}$) and the inclination error of axis ($f_{\Sigma\delta-p}$, $f_{\Sigma\delta-c}$). The proportional deviation/inclination error of axis is scaled with the partial load wt (for contact analysis) or with the ISO factors K_v,K_A and K_γ.

Instead of defining the allowance and inclination of the axes directly (linear deformation model), you can also use shaft calculation files for a more precise definition of the effect of bending and torsion on the shafts on which the gears are mounted.

The "`Define axis alignment`" dialog is described below. This is where you determine the axis alignment by using the shaft calculation files. In the "File Shaft Gear 1/Gear 2" fields, enter the file name for the shafts to which the pinion (1) or the gear (2) belong. You must input the file name with its entire path (for example C:\MyCalculations\ContactAnalysis\pinion_shaft.W10). However, if the shaft files are stored in the same folder as the gear calculation file Z12, you only need to input the name of the shaft calculation file (as shown in the figure).

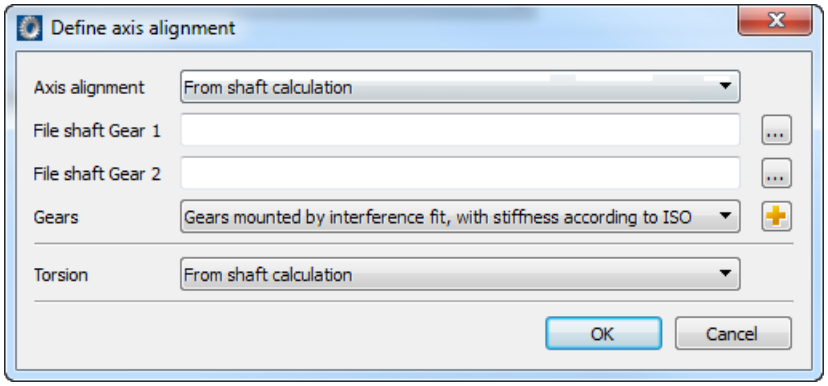


Figure 14.32: `Define axis alignment` (planets and gear pair)

If a shaft file is used, click the additional Plus button to select the meshing to be taken into consideration.

In the case of planetary stages, each meshing has a different load distribution, and consequently a different KHb value. If you select "KHb for strength calculation" in the "Define axis alignment" dialog, you can specify which KHb value is to be used in the strength calculation according to the required standard (ISO, AGMA or DIN). The standard provides no information on this point. It seems reasonable to use the mean value of the KHb values of all sun-planet pairings, and all the planet-internal gear pairings.

Conical expansion can be taken into account for hollow gears.

14.3.6.2 Conditions for using shaft calculation files

If you are working with shaft files, the sizing parameters in the gears module must match those in the selected W010 files. More specifically:

1. The pinion geometry must match the geometry defined for the pinion in shaft file 1. The selection is based on the operating pitch circle (with an error tolerance of 10%), the direction (driving/driven) and the contact flank. The same applies to the gear shaft.
2. The gear pair performance must match the gear performance defined in the shaft files (with an error tolerance of 5%).
3. The direction of rotation for both the pinion and the gear (according to shaft files W10) must be consistent. For example, if the pinion rotates in a clockwise direction, the gear must rotate counterclockwise. However, if the gear is an internal gear, both the pinion and gear must rotate clockwise in this example.

From these conditions you can also easily see whether the shaft files can be used for the contact analysis. If one of these conditions is not met, no calculation can be performed.

In addition to the conditions listed above, a number of other conditions (warnings) concerning the helix angle, the facewidth, and the gear's working transverse pressure angle, are also checked.

14.3.6.3 Effect of torsion on the body of the gear

You can take the effect of torsion on the body of the gear into account either by applying the results of the shaft calculation or by inputting your own data (the same applies to side I and II). Obviously, the results of the shaft calculation can only be referenced if shaft files have been used to define the axis alignment.

If you defined the gear's torsion in "Own input", then the torsion moment of resistance is calculated from the root circle df and the inner diameter.

14.3.6.4 Effect of partial load

You can use the partial load coefficient wt (Contact analysis tab) to modify the performance of all the force elements defined in the shaft calculation files, as shown in the following setting. The diagrams of bending are always modified by this setting. However, the effect of torsion can only be identified if the setting for torsion (previous section) has been made using the shaft calculation file.

For example, if all the force/power elements in a shaft file are 100N/100W, and the partial load coefficient is 85%, the force elements are calculated as 85N and the power elements as 85W.

14.3.6.5 Handling bending and torsion using the results for the shaft

If a gear pair has been found and the shaft calculations performed successfully, the bending and the effect of torsion are determined from the results for the shaft.

The results for bending in each shaft file are all transferred to a single coordinate framework, where pinion contact occurs at 0° and gear contact occurs at 180°. The torsional angle of each gear is assumed to be 0° on the side that is furthest to the left (side I, i.e. the side with the smallest Y-coordinate in the shaft file) and every torsional angle for this particular gear then refers to this side.

14.3.7 Z-Y coefficients and the technology factor

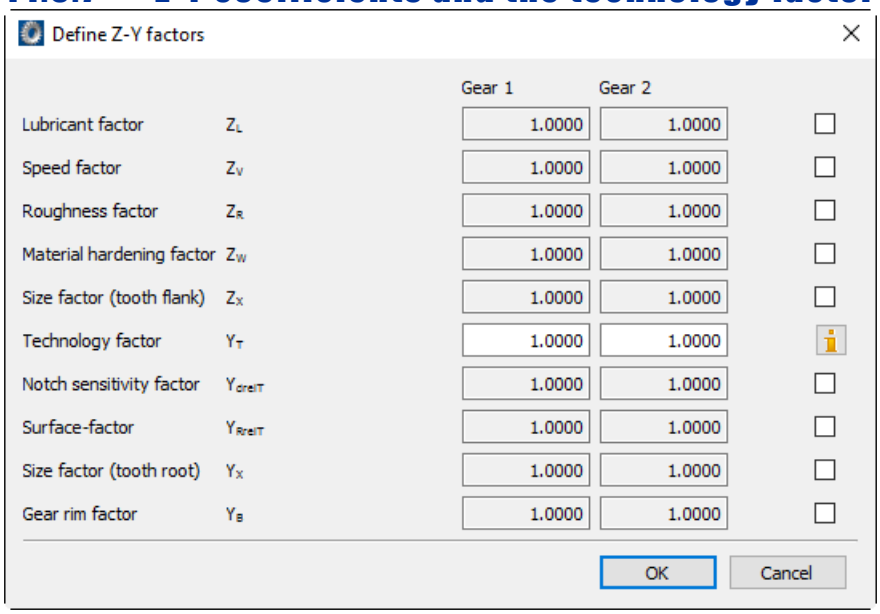


Figure 14.33: `Define Z-Y factors` tab

If necessary, you can modify any of the coefficients that affect the permitted material values (root and flank) as specified in ISO or DIN in the "Z-Y coefficients" window.

Coefficients ZL, ZV, ZR, ZW and ZX affect the pitting stress limit sigHG. Coefficients YT, YdrelT, YRrelT and YX influence the tooth root safety limit sigFG.

You can predefine any of these coefficients in the range 0.5 to 2.0. However, if you input a value that lies outside this range, it will be set automatically to 1.0.

The technology factor takes into account the change in tooth root strength caused by processing. In this situation, the material's permissible stress is multiplied by $Y_T \geq 1.0$. This coefficient is not specified in the DIN or AGMA standards and is therefore set to 1.0.

You can only input gear rim coefficient YB for calculation methods according to ISO 6336. If you select a different method, this flag is deleted and the coefficient is set to 1.0.

Tooth root area processing type	**Technology factor Y_T**
Shot peening	
Case-hardened/carbonitrided toothing	1.2
Not ground in the reinforced areas	
Rollers	
Flame- and induction-hardened toothing	1.3
Not ground in the reinforced areas	
Grinding	
For case-hardened	0.7 (general)
or carbonitrided toothing	1.0 (CBN grinding discs)
Cutting machining	
Not for profile ground teeth!	1.0

Table 14.19: Technology factor according to Linke

According to Bureau Veritas/RINA [70], the technology factors in Table 14.20.

Processing of tooth root area	**Technology factor Y_T**

Shot peening,	Case hardening steel	1.2
Shot peening,	Through hardening steel	1.1
Shot peening,	Nitriding steel	1.0

Table 14.20: Technology factors as defined by Bureau Veritas/RINA Directives

Table 14.21 shows the technology factors as defined in ISO 6336-5:2003, Section 6.7. These only apply to tooth root bending stresses and shot peened case hardening steel.

Material class	**Technology factor Y_T**
ML	1.0
MQ	1.1
ME	1.05

Table 14.21: Technology factor according to ISO 6336-5:2003, Section 6.7

14.3.8 General calculation procedure for KHbeta as specified in ISO 6336-1, Annex E.

1. Import the shaft files, select the correct gears, and then perform the initialization
2. Calculate the shafts and determine the diagrams of bending and torsion in the point of contact (if uniform load distribution is present, determine these values along the facewidth of the gear)
3. Take into account flank modifications from Z012 (not W010)
4. Calculate the gaps in the tooth contact, then the load distribution with tooth contact stiffness and finally calculate $K_{H\beta}$
5. Use the calculated load distribution to correct the load distribution on the original gears
6. Divide the gears into "sections" whose load values are defined in the previous step
7. Use the flank contact ratio (as a vector) from the previous iteration g_{k-1} and the current flank contact ratio g_k to calculate the root of the sum of the square error

$$\lambda = \sqrt{\sum \left(100 \cdot \frac{g_k^i - g_{k-1}^i}{g_{k-1}^i} \right)^2}$$

If λ>0.1%, go back to step 2 and perform further iterations. Otherwise finish.

This procedure exactly follows the method described in ISO 6336-1, Annex E, but uses a stricter iteration criterion.

14.4 Reference profile

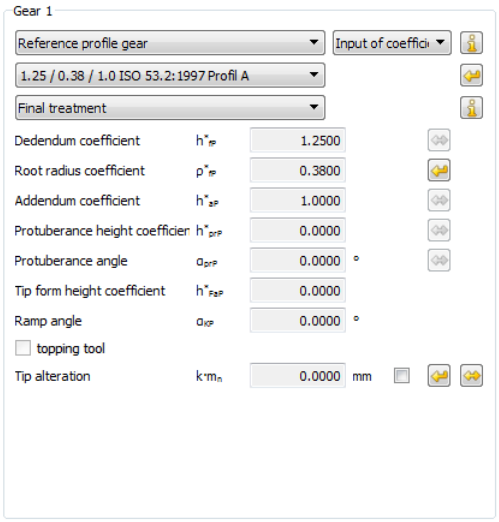


Figure 14.34: `Reference profile` input window

In contrast to traditional mechanical engineering, where a predefined standard reference profile is most commonly used, in precision mechanics the reference profile is often modified. Input the toothing reference profile or the appropriate tool in the `Reference profile` input window. You can input this data either as coefficients, as lengths or as the diameter.

14.4.1 Configuration

The reference profile of the gear toothing is usually predefined. However, you can also define your own hobbing cutter or pinion type cutter. The pinion type cutter parameters are also used in the strength calculation to calculate the tooth form factor. You can also select `Constructed involute` for precision engineering. In this case, the involute is defined directly together with a root radius.

14.4.1.1 Cutter: Hobbing cutter

Select the hobbing cutter you require from the selection list and then click the button, (see Figure 14.17.)

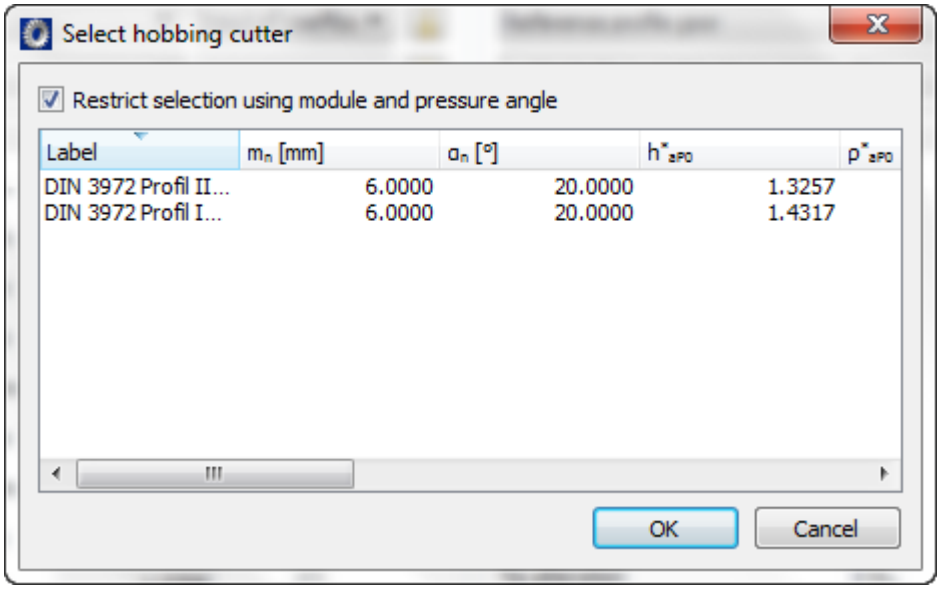


Figure 14.35: `Select hobbing cutter` window

If you select a standardized profile (e.g. DIN 3972 III), the list displays the tools that are present in the database. The name of the cutter file list is entered in the database. Click on the `Restrict selection using module and pressure angle` checkbox to only display tools whose modules and pressure angles match those defined in the gear geometry. By default, only tools that match the selected module and pressure angle are displayed. If tools are selected from the `Tooth form` tab, cutters that meet the condition $\alpha_n * m_n = \alpha_{n1} * m_{n1}$ are also displayed. The standard tolerance is set to + - 1°.

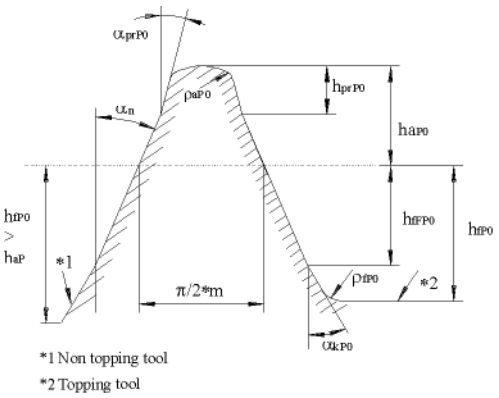


Figure 14.36: Reference profile for the `Cutter/tool` configuration: `Hobbing cutter`

Select `Own input` to directly define your own cutter:

- The cutter addendum coefficient h^*_{aP0} defines the cutter addendum, which defines the gear root circle. A usual value is 1.25.
- The cutter tip radius factor ρ^*_{aP0} defines the cutter tip radius, which then defines the gear root radius. The tip radius is limited by the maximum, geometrically possible radius, depending upon the profile addendum and the pressure angle. This value usually lies in the range 0.2 to 0.38.
- The cutter's dedendum coefficient h^*_{fP0} defines the cutter's dedendum that, with a topping tool, determines the tip circle. A usual value for this is 1. In a non topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The root radius coefficient ρ^*_{fP0} defines the cutter root fillet radius. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. Depending on the geometric conditions, a chamfer or corner may occur on the tip.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length, measured from the addendum. The protuberance is used as an artificial undercut to prevent a grinding notch from being created. The protuberance height can be calculated from the protuberance size and angle.
- The protuberance angle α^*_{prP0} is usually smaller than the pressure angle. However, in the case of some special cutters, it may also be larger.In this case, no undercut is present, but the tooth thickness at the root of the gear is larger. The protuberance angle can be calculated from the protuberance height and size. If you enter the value "0", no protuberance will be present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed in the profile modification. You can specify the threshold used to take into account the protuberance and buckling root flank for active diameters in the `Calculation > Settings (see page II-466)` menu option.
- The root form height coefficient $h_{FfP0}{}^*$ defines the end of the straight flank part of the tool with pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle $a_{KP0}{}^*$ defines a ramp flank or a profile modification that is present in the cutter. The length is determined by the root form height coefficient.

The angle must be greater than the pressure angle α_n. If you enter the value "0", this part will be ignored.

- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (→ more information (see page II-466)).
- The tooth thickness factor of the reference line s^*_{P0} for the usual tools $s^*_{P0} = \pi/2$. The value can be overwritten for special tools.
- The addendum coefficient of the gear reference profile h^*_{aP} for a non topping cutter/tool is defined with the usual value of $h^*_{aP} = 1$ of the gear reference profile or by the gear's tip circle. The value can be calculated from the tip circle.

14.4.1.2 Cutter/Tool: Pinion type cutter

Click the button next to the pinion type cutter designation to select a pinion type cutter for inside and outside gears from a list. Pinion type cutters as specified in DIN 1825, 1826 and 1827 are listed here. You use this window in the same way as the `Select hobbing cutter` window in Figure 14.37. The default setting is for the list to display only those tools that match the selected module, meshing and helix angle.

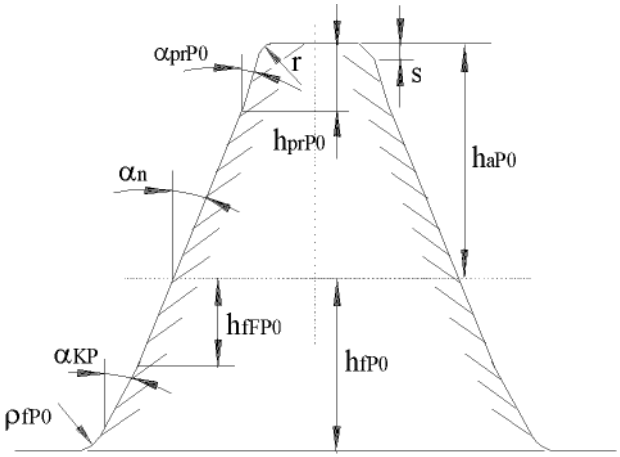


Figure 14.37: Reference profile for the `Cutter/tool` configuration: `Pinion type cutter`

Select `Own input` to directly define your own pinion type cutter:

- KISSsoft can prompt the number of teeth z_0 for the cutter. If the number of teeth is too small, it may not be possible to manufacture the tip form circle

and/or the root form diameter of the cylindrical gear. If the number of teeth is too great, it may cause collisions during manufacture.

- The pinion type cutter profile shift coefficient x_0 is often unknown. However, it does influence the root circle of the resulting gear. This value is set automatically, together with the number of teeth.
- A pinion type cutter tip often takes the form of a radius or a chamfer. Click the  button to define the corresponding numerical value.
- The pinion type cutter addendum coefficient h^*_{aP0} defines the pinion type cutter addendum that determines the pinion type cutter tip circle and the gear root circle. A usual value is 1.25.
- The pinion type cutter dedendum coefficient h^*_{fP0} defines the pinion type cutter dedendum height that determines the tip circle for a topping tool. A usual value for this is 1. In a non topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The pinion type cutter root radius coefficient ρ^*_{fP0} defines the pinion type cutter root radius. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. The input value is only displayed for a topping tool.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length, measured from the addendum. The protuberance is used as an artificial undercut to prevent a grinding notch from being created.
- The protuberance angle α^*_{prP0} is usually smaller than the pressure angle. If 0 is input, no protuberance is present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed in the profile modification. You can specify the threshold used to take into account the protuberance and buckling root flank for diameters in the `Calculation > Settings (see page II-466)` menu option.
- The root form height coefficient $h_{FfP0}{}^*$ defines the end of the tool involute with the pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle $\alpha_{KP0}{}^*$ defines a ramp flank or a profile modification that is present in the cutter. The length is determined by the root form height coefficient. The angle must be greater than the pressure angle α_n. If you enter the value "0", this part will be ignored.
- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (→ more information (see page II-466)).

- The addendum coefficient of the gear reference profile h_{aP}^* with the usual value of $h_{aP}^* = 1$ defines the tip circle of the gear for a non topping tool. The value can be calculated from the tip circle.

14.4.1.3 Reference profile

The reference profiles displayed here are taken from the database. If you can't find a suitable reference profile here, you must first enter it in the database (see page I-128) (`Z000.ZPROF`). Alternatively, select `Own input` from the drop-down list, to open a dialog in which you can edit all the input fields, and so change all the reference profile parameters. The `Label` input field is displayed under the `Reference profile` drop-down list. This is where you enter the name of your own profile, which will then appear in the calculation report.

NOTE

You do not create a new entry in the database when you define your own profile in the `Own input` field.

The reference profile details are according to ISO 53, DIN 867 or DIN 58400. This is the reference profile data for the gear. You can calculate the corresponding values in [mm] by multiplying it with the normal module. Please note the following points:

- If the reference profile is set to `Own input`, the tip alteration (see section "Modifications" on page II-361) is set to zero. For this reason, the addendum may change when you toggle from one window to another.
- If you are using the `BS4582-1:1970 Rack 2` reference profile to determine the correct tip and root diameter, you must input an appropriate tooth thickness tolerance of

$$A_s = -0.1572 \cdot m_n$$

. The tip and root diameter will then match the values defined in BS4582-1(8)).

- The ramp flank is usually used to generate a tip chamfer[5]. Alternatively, you can also use a small buckling root flank to generate a profile modification. However, profile modifications are usually defined in the `Modifications` (on page II-361) window.
- If the angle of the ramp flanks is only slightly different from the pressure angle, it is not taken into account in the contact ratio because the assumption for

[5] *also called semi-topping.*

profile modifications is that the contact ratio will not decrease under load. In contrast, the contact ratio should be reduced accordingly for a chamfer. In `Settings` (see page II-466), you can specify the difference in angle that is to be used as the threshold in profile modifications and chamfers.

- If a pre-machining tool is used, the additional measure for the pre-machining `must` be entered separately. You must input the gear's reference profile for the pre-machining. This data, and the grinding wheel data, is used to calculate the reference profile used during the final machining, and documented in the report (Processing (see page II-353)).
- For profile modifications, where the angle difference < threshold value (see

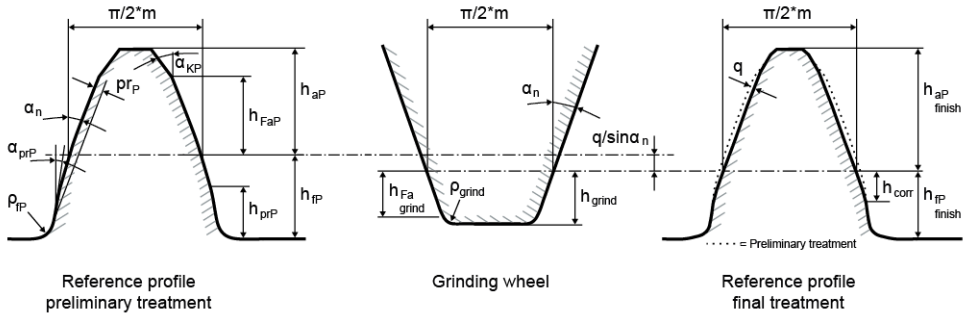


Figure 14.38: Reference profile for configuration: Reference profile gear

- Click the button next to the Reference profile drop-down list to open a dialog which contains proposals for reference profiles according to the following criteria:
  - High toothing, according to the theoretical profile contact ratio defined in the Sizing tab, in the "Module-specific settings" (`Calculation > Settings`).
  - Both gears at minimum top land (do not change x)
  - Both gears at minimum top land (x is optimized to suit sliding velocity)
  - Both gears with (dNf-dFf) minimum
- h_{aP}^* always applies as the normal gear reference profile. The tooth thickness on the reference line is

$$s_P = \frac{\pi}{2} \cdot m_n \tag{12.19}$$

14.4.1.4 Constructed involute

When you select `Constructed involute`, you do not need to enter as many parameters as you do when you select `Reference profile`. The essential difference is that no simulation of manufacture is performed, and the involute is generated directly.

In the gear root, the involute is closed by a radius that is defined by the root radius coefficient ρ_{fP}. In theoretical involutes, the root radius coefficient is usually greater than the coefficient for a reference profile, because the manufacturing process does not involve a meshing movement.

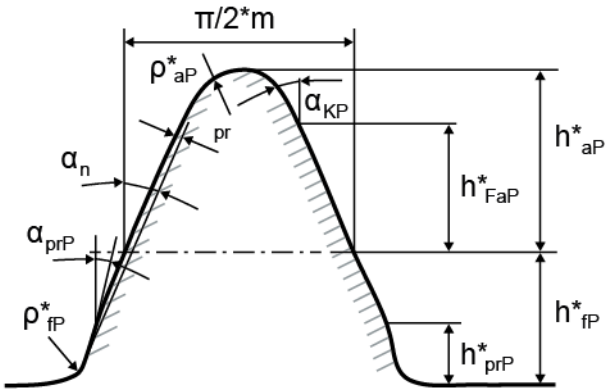


Figure 14.39: Reference profile for configuration: Constructed involute

14.4.2 Pre-machining and grinding allowance

Often gears are premachined with a grinding allowance, then hardened and then ground. The tooth flank, but not the tooth root, is usually involved in the grinding process.

Note: If a cutter, pinion type cutter or constructed involute is selected as the pre-machining tool, the gear reference profile for pre-machining is calculated internally from the tool data.

In this case, the root circle is created by the pre-machining cutter and the flank by the grinding process. To complete this process correctly, select either

`Preliminary machining` (with own input, or with reference profile for grinding allowance III or IV as specified in DIN 3972) or select `final machining`. If you decide to use pre-machining, the `Grinding allowance` field is displayed. You can also add your own tolerances to the database. Enter the profile of the pre-machining tool (except: h_{aP}^*) as the reference profile. As the tooth thickness deviations (tolerances) you have to enter the tooth thickness allowance of the finished gear teeth (A_s). In KISSsoft the grinding allowance is then calculated for the finished gear teeth. The pre-machining is then performed using the total tooth thickness allowance:

$$A_s' = A_s + \frac{2 \cdot q}{\cos \alpha_n} \qquad (12.20)$$

For special requirements, click the  button in the "`Define grinding allowance tolerance`" window to increase the tolerance. If a value is input for qmax-qmin, then qmax = q+(qmax-qmin)/2 and qmin = q-(qmax-qmin)/2 are used to define the pre-machining allowances.
The tolerance interval qmax-qmin is limited to the smaller value of either 200% of the tooth thickness tolerance interval (As.e-As.i), or 30%, or the grinding allowance (q).

KISSsoft then determines the reference profile that corresponds to the finished tooth form. This tooth form will also be used to calculate the factors Y_F and Y_S for the tooth root strength. The tooth form is then defined automatically by overlaying the pre-machining contour with the subsequent grinding process. The root diameters are derived from the reference profile for pre-machining. The control data (e.g. base tangent length) is calculated and printed out for both the premachined and the finished gear teeth.

IMPORTANT EXCEPTION

The addendum coefficient h_{aP}^* is the theoretical addendum coefficient that is used to calculate the theoretical tip diameter coefficient. The appropriate minimum root height of the hobbing cutter h_{fP0}^*, which is required to create the tooth form without topping, is printed in the report. h_{aP}^* always applies as the final machining reference profile for the gears. The tooth thickness on the reference line is $\pi/2 * m_n$.

14.4.3 Tip alteration

The tip alteration k*mn is usually calculated from the profile shift total to ensure that the tip clearance does not change. However, if the reference profile is set to `Own input`, the tip alteration will not be calculated. In an external gear pair, a reduction in the tip alteration in a negative value for the tip circle reduction. In

contrast, in internal toothings, the result is a positive value for both gears, and therefore also an increase in the tooth height. In KISSsoft, the tooth height of internal toothing is not increased, and therefore the tip alteration is limited to 0.

Alternatively, you can specify your own tip alteration. However, this only has an effect on non-topping tools. Otherwise, the value is set to 0 when it is calculated. Click a Sizing button  to calculate the proposed value for a constant tip clearance.

Click the Conversion button  to input a tip diameter (either d_a, d_{aE} or d_{ai}) which is used to convert the tip alteration, using the reference profile present.

14.5 Final machining

You can define the grinding process in this tab. These inputs are necessary if a grinding allowance is present in the `Reference profile` tab, and/or profile modifications are added in the `Modifications` tab. The start of modification at the tip or root specifies the height at which the grinding process processes the gear. In particular, the radius of the tip of the grinding wheel must also be predefined. If the grinding process reaches the diameter that matches the selected start of modification at the root, the software simulates the complete roll-off of the grinding tool. The grinding notch that may result is calculated and taken into account in the strength calculation according to ISO/DIN. You can input the data either as coefficients, as lengths or as the diameter. In the case of profile modifications, which are defined over a particular length (e.g. linear root relief), the length is measured from the selected start of the modification at the tip or root. The manufacturing process with a tool and gear can only be checked in the `Manufacture` 2D graphic.

Usually, the tooth root area is not included in grinding. When you enter a value for `Start of modification at root` you can, if required, also specify that the root area is included in grinding. The grinding wheel addendum [h^*_{grind}] is also usually entered in this case. The profile modifications in the root then start from the tip form height [$h_{Fa}{}^*_{grind}$] of the grinding wheel, but not before the gear's base circle.

NOTE:

Recommendation for the `Generate` or `Form grinding` setting:
if it is not known whether the grinding process is performed using the generation or form grinding process, we recommend you select the "Form grinding" process, if you input finished teeth without a pre-machining tool. We also recommend you select "Generate" if you input finished teeth with a pre-machining tool.

14.6 Tolerances

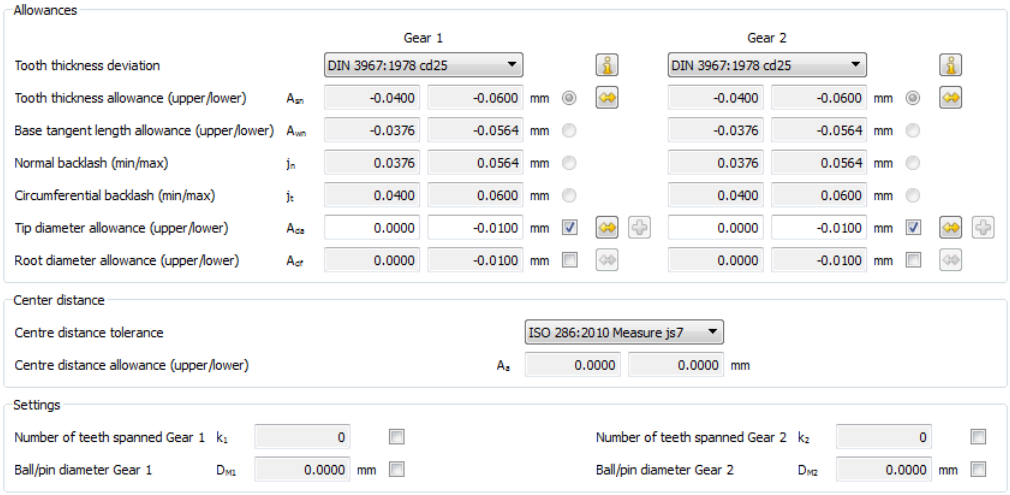


Figure 14.40: `Tolerances` input window

The toothing geometry is calculated for a backlash-free state. A slightly smaller tooth thickness is manufactured, to prevent the gears jamming in practice. This reduction in tooth thickness (in contrast to the backlash-free state) is known as the tooth thickness allowance. The upper tooth thickness allowance is the upper limit of the tooth thickness. The lower tooth thickness allowance is the lower limit of the tooth thickness.

EXAMPLE

Tooth thickness in a backlash-free state:	4.560 mm
Upper deviation of tooth thickness (top limit):	-0.050 mm
Lower tooth thickness allowance:	-0.060 mm
This results in the actual tooth thickness:	4.500 to 4.510 mm

14.6.1 Tooth thickness tolerance

This drop-down list contains the tolerances listed below. You can also include your own tolerance tables. You will find more detailed information about this in the section about the KISSsoft Database tool (see section "External tables" on page I-136).

14.6.1.1 DIN 3967

Selection of a tolerance as specified in DIN 3967 (for a gear unit with a module greater than 0.5 mm). Suggestions as defined by Niemann [65 (see section "Gear teeth when existing shaft data is present" on page I-229)](page 84):

Cast ring gears	a29, a30
Ring gears (normal clearance)	a28
Ring gears (narrow clearance)	bc26
Turbo gears (high temperatures)	ab25
Plastic machines	c25, cd25
Locomotive gears	cd25
General mechanical engineering,	
Heavy machines, non-reversing	b26
General mechanical engineering,	
Heavy machines, reversing	c25,c24,cd25,cd24,d25,d24,e25,e24
Vehicles	d26
Agricultural vehicles	e27, e28
Machine tools	f24, f25
Printing presses	f24, g24
Measuring gear units	g22

14.6.1.2 ISO 1328

The current edition of ISO 1328 no longer includes tolerance classes for tooth thickness allowances. For this reason, many companies have continued to use the tolerance classes specified in the old 1975 edition.

14.6.1.3 DIN 58405

Proposals as specified in DIN 58405, Part 2: Allowances for precision mechanics; usual gear modifications as defined in DIN 58405 Sheet 2

Material	Processing	Center distance tolerance	Base tangent length

			tolerance
Hardened steel	Ground	5J	5f
Heat treatable steel	finely milled	6J	6f
Light metal	finely milled	7J	7f
Light metal	finely milled	8J	8f
Steel/laminate	finely milled	6J	6e
Steel/laminate	finely milled	7J	7d/7c
Light metal	finely milled	8J	8d/8c
Plastic	milled	9J	9e/9d
Plastic	injection molded	10J	10e

14.6.1.4 Own input

Select this option to input your own data. However, you should note that the values for tooth thickness allowance, the normal or circumferential backlash (per gear) and the base tangent length allowance all depend on each other. The (negative) base tangent length allowance corresponds to the normal backlash.

14.6.2 Tip diameter allowances

You can specify the tip diameter allowances if a non-topping tool has been defined. In contrast, the tip diameter allowances for a topping tool are defined from the tooth thickness allowances. These allowances influence the effective contact ratio due to the effective tip circle.

Click the  button to specify a tolerance field according to ISO 286. The tolerances sign is changed in internal teeth because the tip circle is used as a negative value in the calculation.

Click the  button to specify the minimum and maximum tip diameter from which the allowances are to be calculated.

14.6.3 Root diameter allowances

Root diameter allowances are usually calculated from the tooth thickness allowances. In a gear cutting process, the backlash is produced by reducing the manufacturing distance of the tool. This is why the root diameter allowances depend on the tooth thickness allowances.

In special cases, a different manufacturing process is used e.g. for sintered gears or extruded plastic gears. The user can then input their own root diameter allowances.

Click the  button to specify the minimum and maximum root diameter from which the allowances are to be calculated.

14.6.4 Center distance tolerances

The center distance tolerances are defined either by a standard tolerance taken from the database or the value you enter in the `Own input` field. They influence the intermeshing allowance and the contact ratio.

14.6.5 Settings

The base tangent length and the mass across balls and rollers for the most suitable number of teeth over which the measurement is to be taken, or the roller diameters, are specified in the report. If you want to use a different number of teeth spanned, or a different diameter of ball/pin in an existing drawing, this is where you can overwrite the values selected by the software.

However, no results are output if you enter values for which a measurement cannot be performed. If the `do not cancel when geometry errors occur (see page II-458)` option is selected, test masses are also output for cases in which they could not be measured, for example, for points of contact above the tip circle.

NOTE

The proposed ball/pin diameters are taken from the `Z0ROLLEN.dat` file. In the case of splines as defined in ANSI 92.1, these values are taken from the `Z0ROLLENANSI.dat` file. This file corresponds to the recommended diameters specified in DIN 3977. You can then use an editor to modify the existing ball/pin. You will find more detailed information about how to handle external data records in the External tables (on page I-136) section.

14.7 Modifications

You can define profile and tooth trace modifications, a tip chamfer or a tip rounding in the `Modifications` input window.

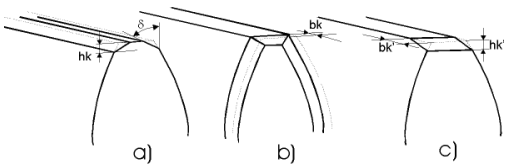


Figure 14: `Definition of modifications to the tooth end`

- a) tip chamfer
- b) chamfer at tooth end
- c) tip end chamfer

NOTE:

The tip end chamfer is not specified for gear calculations because it does not affect the strength. However, if an unusually large chamfer is involved, hk' and bk' can be simulated by inputting, for example, hk=0.3*hk'. The standards do not offer any guidance for this.

14.7.1 Type of modification

To create a new entry in the list of modifications, click the  button. Double-click on a cell in the `Type of modification` column to open a drop-down list if you want to change the value in that cell. Figure 14.25 shows an extract of the range of possible tooth corrections.

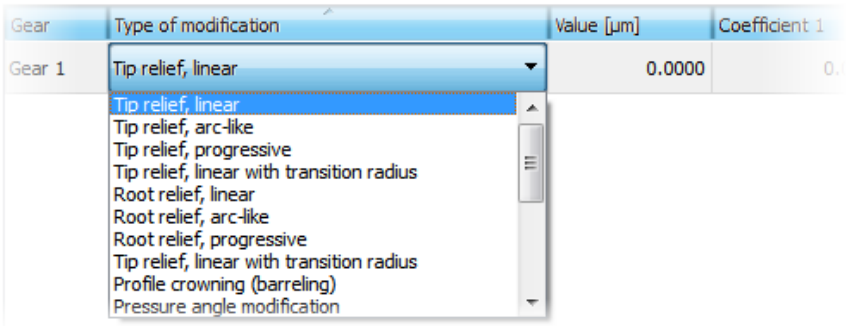


Figure 14.25: `Type of modification` drop-down list

The next two sections, 14.7.3 (see section "Profile modifications" on page II-364) and 14.7.4 (see section "Tooth trace modifications" on page II-372), provide descriptions of the modifications defined in ISO 21771.

Input different modifications for right or left flank: In the `Flank` drop-down list, you can specify whether a modification is to be applied to the right flank, the left flank or to both flanks.

Definition of the right-hand/left-hand tooth flank (according to ISO 21771):

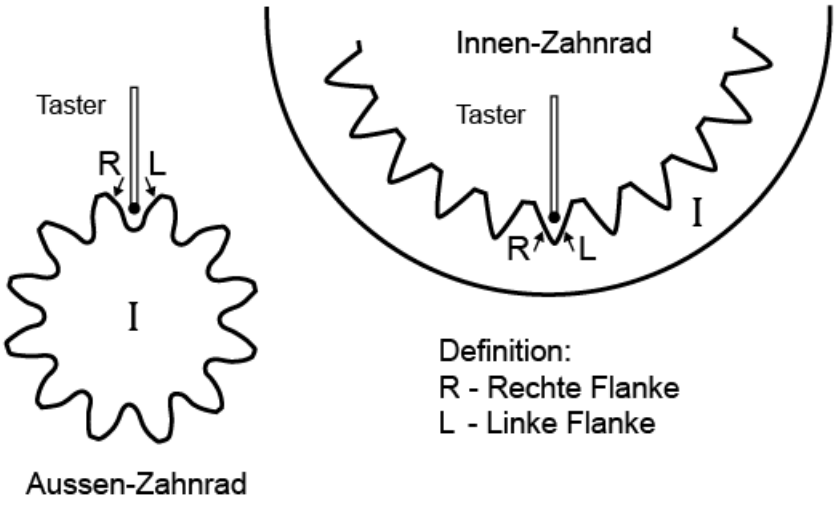


Figure 14.26: `Tooth flank definition`

14.7.2 Underlying principles of calculation

The geometry of straight or angled cylindrical gears is calculated according to ISO 21771 or DIN 3960. Many manuals and standards use very similar methods to calculate this geometry. In addition to calculating the geometry, it is very useful to have information about how to check for defects (undercut, insufficient active profile, etc.). Technical documentation provided by tooling manufacturers or machine tool manufacturers may also contain information about this.

Measurements for tooth thickness and backlash can be selected in accordance with different standards, such as ISO1328 (1970 edition) or DIN 3967. Manufacturing tolerances can also be defined using standards such as ISO 1328, AGMA 2000, AGMA 2015, DIN 3961, or DIN 58405, to suit the particular situation.

Strength is calculated in accordance with, for example, ISO 6336 or DIN 3990, by verifying common defects (tooth root fracture, pitting, scuffing, micropitting). These standards include the most comprehensive and detailed calculation methods currently available. There are two methods that can be used to calculate safety against scuffing. The integral temperature method of calculating scoring resistance is mainly used in the automobile industry, whereas the flash temperature method is used in manufacturing turbo gearboxes. It has not yet been established which of these two methods is the more reliable.

Micropitting is calculated according to ISO 15144, Method B. This method is very reliable for gears without profile modifications. However, in the case of gears with profile modifications, it has been specified that the tip relief Ca must correspond to the optimum tip relief Ceff (as proposed in the standard). If not, the verification must be performed without taking the modification into account. This is a significant disadvantage, because modifications have a considerable effect on micropitting. In this case, you should use Method A (Safety against micropitting using method A).

In the USA, the AGMA 2001 standard must be applied when calculating resistance. This calculation method differs so much from that required by DIN 3990 that the results cannot be compared. In addition, there are numerous different methods that are used to calculate the resistance of plastic gears.

One of the problems with applying DIN 3990 is the wide range of different calculation methods it contains. There are around 10 different calculation variants that can be applied between Method A (exact calculation involving measurements) and Method D (the simplest, rough calculation). It is therefore no surprise that very different results can be obtained from applying calculations according to DIN 3990 or ISO 6336 to the exact same gear wheel. KISSsoft uses the most detailed formulae for dimensioning and analyses during this calculation procedure. This procedure corresponds to Method B. However, calculations performed using different programs may also give very different results. It also takes a lot of time and effort to investigate the precise reasons for this. It is therefore much more effective and efficient to use a reference program to perform the comparison. One such program is the ST+ cylindrical gear program package developed by the FVA (Forschungsverein Antriebstechnik, (Research Society for Transmission Techniques, Germany)), at the Technical University in Munich. KISSsoft includes the `As in the FVA program (DIN 3990)` option, which supplies the same results as the calculation with the FVA code (see section "Calculation methods" on page II-281). The differences between results obtained by KISSsoft and the FVA are negligible. They are due to the minor differences between the FVA program and the regular version of DIN 3990. If requested, we can provide you with a number of different documents to help you compare these methods.

Other interesting results have been taken from Niemann's book [65]:

- Gear power loss with gear loss grade H_V according to equation (21.11/4)
- Average friction factor, value μ_m according to equation (21.11/6) with $1 \leq vt \leq 50$ m/s
- Gear power loss P_{VZ} according to equation (21.11/3)

14.7.3 Profile modifications

Profile modifications are actually variations of the involute and are known as height corrections. The following sections describe which profile modifications are included in the KISSsoft system.

NOTE 1:

Before you can define height corrections, you must first input the length factor L_{Ca*}. The length factor is the roll length Ly (from the modification to the tip or root form diameter) divided by the normal module: $L_{Ca*} = (L_{dFa} - L_{dC})/m_n$ or $L = (L_{dC} - L_{dFf})/m_n$. The roll length L_y is calculated according to ISO 21771, Equation 17, or DIN 3960, Equation 3.3.07.

NOTE 2:

Measuring tip relief C_a directly on the tip circle may be inaccurate. If tip reliefs have been defined, the report states the tip relief on a special measurement circle, dcheck, for measuring purposes.
Measurement circle dcheck = dFa.i - 0.02•mn

14.7.3.1 Linear tip and root relief

Figure 14.26 illustrates `tip relief`. The constantly increasing amount of material removed in the transverse section, starting at d_{Ca}, up to the tip circle, refers to the theoretical involute. The same applies to the `root relief`.

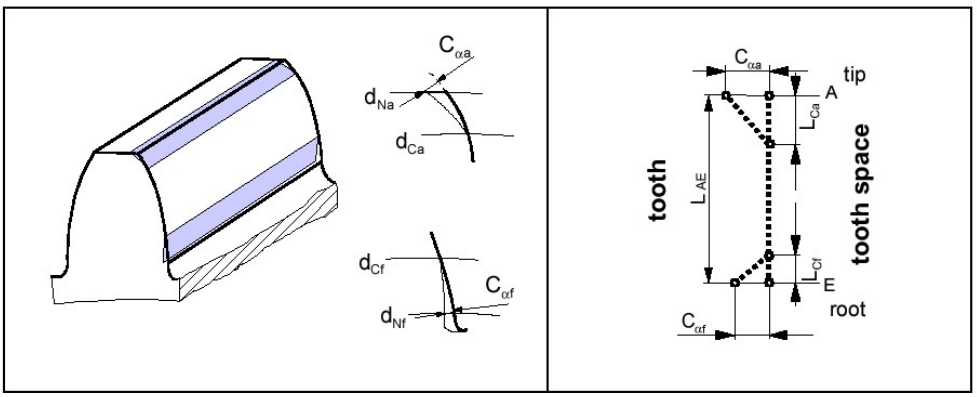


Figure 14.26: Linear tip and root relief

where

d_{Na}	Active tip diameter	d_{Nf}	Active root diameter
d_{Ca}	Modification end diameter (tip)	d_{Cf}	Modification end diameter (root)
L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief
A	Tip neighboring point	E	Root neighboring point
L_{AE}	Resulting tooth height length[1)]		

[1)] Corresponds to the meshing length g_α

In KISSsoft, input the size value $C_{\alpha a}$ in the `Value` input field, for tip relief. The `Coefficient 1` input field defines the quotient from the calculated tip relief length L_{Ca} and normal module m_n. Similarly, to represent root reliefs, input the values for $C_{\alpha f}$ and the quotient from L_{Cf} and m_n.

NOTE

In the "Modifications" tab, you can specify that the modification starts at the root. The figure shows the situation when the modification starts at the active root diameter d_{Nf}.

14.7.3.2 Arc-like profile modification

The procedure used here is similar to the one used for a linear profile modification. The difference is that this method involves approximating an arc of circle which starts at the point where diameter d_{Ca} intersects with the unchanged tooth profile. The tangents of the arc of circle are identical to the tangent of the unchanged tooth profile at this point. The benefit of this modification is that the tangents do not change abruptly in the unchanged tooth form - circular pitch approximation transition point.



Figure 14.27: Arc-like profile modification

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief

14.7.3.3 Progressive profile modification

The procedure used here is similar to the one used for a linear profile modification. The progressive profile modification is also detailed in the description of tooth form options (see Progressive profile modification (see page II-395))



Figure 14.28: Progressive profile modification

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief

14.7.3.4 Linear tip and root relief with transition radii

Figure 14.29 shows `tip and root relief with transition radii`. The constantly increasing amount of material removed in the transverse section, starting at d_{Ca}, up to the tip circle, refers to the theoretical involute. The same applies to the `root relief`.

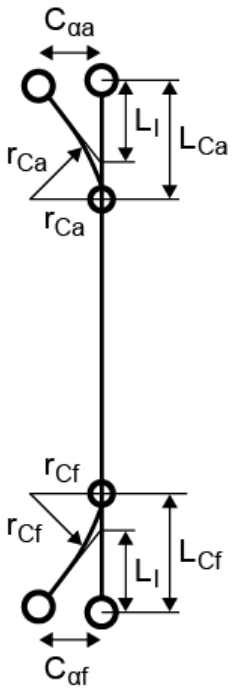


Figure 14.29: Linear tip and root relief with transition radii

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief
r_{Ca}	Transition radius in the tip area	r_{Cf}	Transition radius in the root area

Tip relief with transition radius: Enter a `Value` for $C_{\alpha a}$ in the input field. In the `Factor 1` input field, enter the quotient from the calculated tip relief length L_{Ca} and normal module m_n.
In the `Factor 2`, input the quotient from the transition radius in the tip area r_{Ca} and normal module m_n.

If coefficient 2 = 0, then r_{Ca} is calculated in such a way that $L_I = 0.8*L_{Ca}$ applies. The corresponding coefficient 2 is calculated and applied. If coefficient 2 is so large that $L_I < 0.75*L_{Ca}$ applies, then r_{Ca} is calculated in such a way that $L_I = 0.75*L_{Ca}$ applies. The corresponding coefficient 2 is calculated and applied.

Similarly, to represent root reliefs, input the values for $C_{\alpha f}$ and the quotient from L_{Cf} and m_n, and the quotient from r_{Cf} and m_n.

14.7.3.5 Profile crowning (depth crowning)

`Profile crowning (barreling)` is where a constantly increasing amount of material is removed in the transverse section in the direction of the tip and root circle, starting at the middle of the calculated tooth flank length. Points A, E and the value C_a define the arc-like progression. $C_a = C_{\alpha a} = C_{\alpha f}$ applies for profile crowning. `Eccentric profile crowning` can be used for different crowning at the tip and root.

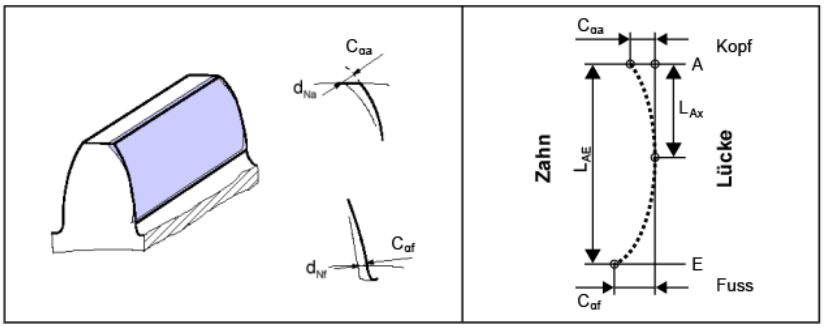


Figure 14.27: Profile crowning (depth crowning)

where

d_{Na}	Active tip diameter	d_{Nf}	Active root diameter
$C_{\alpha a}$	Crowning at tip	$C_{\alpha f}$	Crowning at root
L_{AE}	Resulting tooth height length[1)]	L_{AB}	Length from tip to center of crowning
A	Tip neighboring point	E	Root neighboring point

[1)] Corresponds to the meshing length g_α

In KISSsoft, enter the value `Ca` in the ςαλυε input field.

14.7.3.6 Eccentric profile crowning

In the "`Modifications`" tab, you can add eccentric profile crowning to the tooth profile.

The definition of eccentric profile crowning is the same as for eccentric crowning, but `Factor 1` corresponds to the diameter ratio $(d_A - d_X) / (d_A - d_E)$. Here, you should note that the modification is defined by the diameter, not by the length of path of contact. Therefore, if you input a value of 0.5 for `Factor 1`, this will not correspond to the profile crowning, which is symmetric to the center point of the path of contact (d_{Sm}).

`Factor 2` is used to define the root relief from the tip relief. You can use `Factor 2` to set a different value for $C_{\alpha a}$ and $C_{\alpha f}$. $C_{\alpha a}$ = 'Value'; $C_{\alpha f} = C_{\alpha a}$ • "Factor 2" then applies.

14.7.3.7 Linear tip relief with profile crowning

Linear tip relief with profile crowning is a combination of linear tip relief followed by crowning. The entry in the `Value` field is for the crowning value C_β. `Factor 1` defines the length of the linear tip relief (L_{Ca}/m_n). `Factor 2` defines the ratio of tip relief C_α(in µm) to m_n (in mm), therefore C_α/m_n (µm/mm).

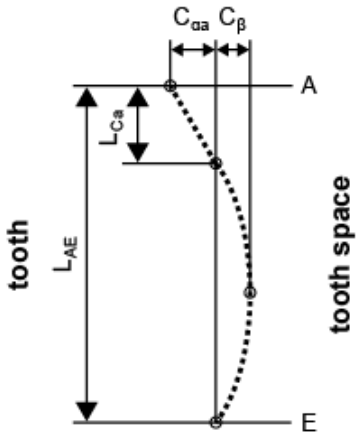


Figure 14.27. Linear tip relief with profile crowning

C_α	Tip relief	C_β	Crowning
L_{Ca}	Roll length of the tip relief	L_{AB}	Roll length of the active tooth height[1)]
A	Tip neighboring point	E	Root neighboring point

[1)] Corresponds to the meshing length g_α

This modification is usually applied to attempt to merge the linear tip relief without bending tangentially into the crowning. A value for Factor2_opt=... is output in the Info field for this purpose. If you input this value in the `Factor 2` field, you will achieve exactly this.

14.7.3.8 Shortened profile crowning

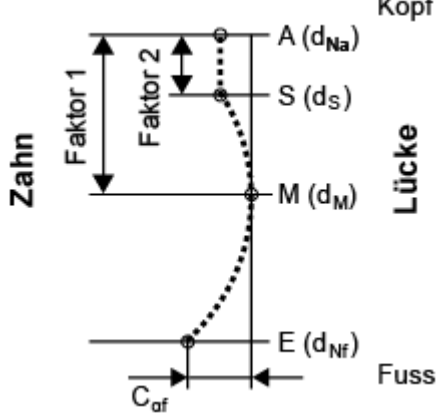


Figure 14.27a.: Shortened profile crowning

C_{af}: Root relief

Factor 1: Position of the arc midpoint

$$\frac{d_A - d_M}{d_A - d_E} = Faktor1$$

Factor 2: Shortening

$$\frac{d_A - d_S}{d_A - d_E} = Faktor2$$

NOTE:

Shortened profile crowning is used in combination with a tip relief. The tip relief should start at S. Otherwise, do not use this modification.

14.7.3.9 Pressure angle modification

You define the `pressure angle modification` in a similar way to tip/root relief (see section "Linear tip and root relief" on page II-364). However, the difference here is that the value C_H applies over the entire tooth height (see Figure 14.28).

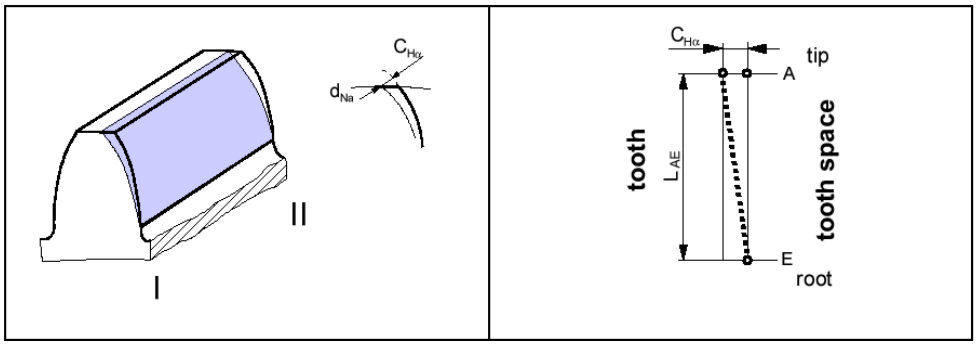


Figure 14.28: Pressure angle modification

where

d_{Na}	Active tip diameter	$C_{H\alpha}$	Pressure angle modification
A	Tip neighboring point	B	Root neighboring point
L_{AE}	Resulting tooth height length[1)]		

In KISSsoft, enter the value $C_{H\alpha}$ in the `Value` input field.

14.7.4 Tooth trace modifications

Tooth trace modifications are variations across the facewidth. The following sections describe which tooth trace modifications are implemented in the KISSsoft system.

14.7.4.1 Linear end relief I and II

A `linear end relief` is the constantly increasing removal of material from the tooth trace, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 14.29).

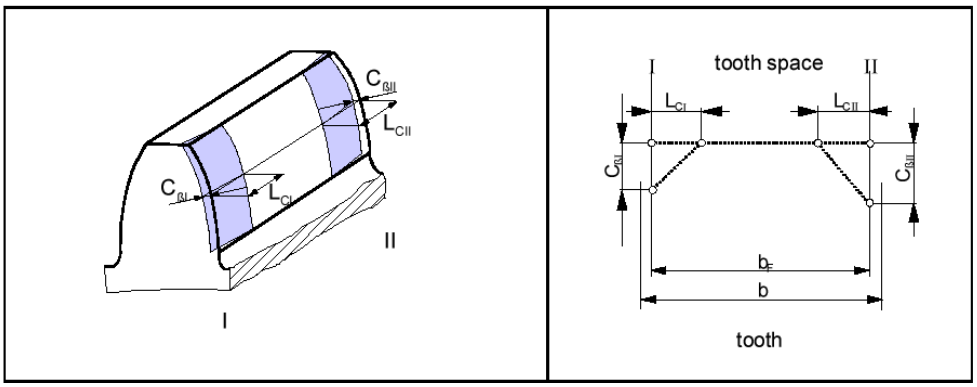


Figure 14.29: Linear end relief I and II

where

	Face I		Face II
L_{CI}	End relief length	L_{CII}	End relief length
$C_{\beta I}$	End relief	$C_{\beta II}$	End relief

In the KISSsoft system, go to the `Value` input field and enter the value $C_{\beta I(II)}$. Then go to the `Coefficient 1` input field, enter the quotient $L_{CI(II)}$ / b_F where b_F is the facewidth minus chamfer.

14.7.4.2 Arc-like end relief I and II

An `arc-like end relief` is the constantly increasing removal of material from the tooth trace, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 14.30).

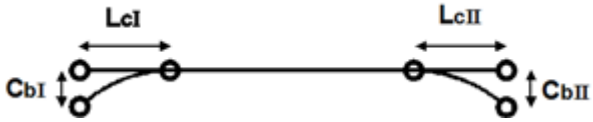


Figure 14.30: Arc-like end relief I and II

where

	Face I		**Face II**
L_{CI}	End relief length	L_{CII}	End relief length
$C_{\beta I}$	End relief	$C_{\beta II}$	End relief

In the KISSsoft system, go to the `Value` input field and enter the value $C_{\beta I(II)}$. Then go to the `Coefficient 1` input field, enter the quotient $L_{CI(II)}$ / b_F where b_F is the facewidth minus chamfer.

14.7.4.3 Helix angle modification

You define the `helix angle correction` in a similar way as end relief (see section "Linear end relief I and II" on page II-372). However, the difference here is that the mass L_{CI} applies over the entire facewidth (see Figure 14.30).

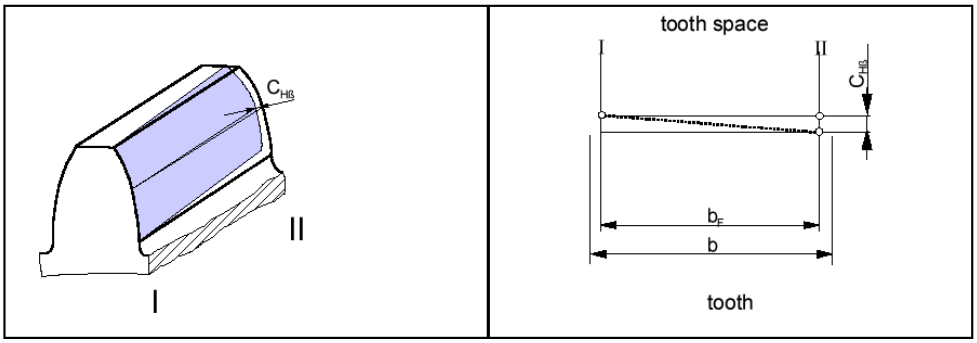


Figure 14.31: Helix angle modification

where

b	Facewidth	b_F	Usable facewidth
$C_{H\beta}$	Helix angle modification		

In the KISSsoft system, enter the value $C_{H\beta}$ in the `Value` input field.

14.7.4.4 Crowning

`Crowning` is where material is removed constantly and symmetrically in the direction of the face surfaces, starting from a common point and where the tooth trace remains constant. The material is removed in an arc-like progression with the maximum at the point $b_F/2$. $C_\beta = C_{\beta I} = C_{\beta II}$ applies.

In KISSsoft, transfer the CβI value to the "Value [µm] input field.

NOTE

Offset crowning, with its maximum to the right of the point $b_F/2$, is often used in practice. You can define this modification by inputting centrical crowning with an additional helix angle modification (on page II-374).

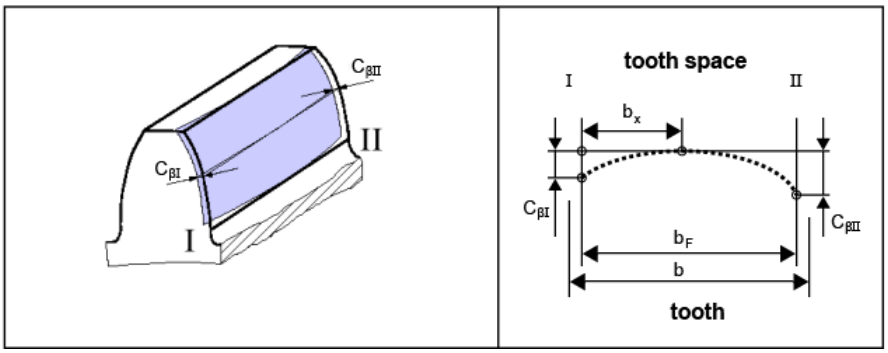


Figure 14.32:Crowning

where

b	Facewidth	b_F	Usable facewidth
$C_{\beta I}$	Crowning Side I	$C_{\beta II}$	Crowning Side II
B_X	Length I to crowning center point		

14.7.4.5 Eccentric crowning

In the "Modifications" tab, you can add eccentric crowning to the facewidth.

For eccentric crowning, the Value defines the value of the modification and Factor 1 defines the modification position from side I divided by the facewidth (b_X /b_F). The modification is defined as a part of an arc of circle, whose center runs along the vertical line defined by Factor 1. The radii are shown in the Information field according to your input. If you input a value of 0.5 for Factor 1, the modification corresponds to general crowning.

You can use Factor 2 to set a different value for the modification on side II ($C_{\beta II}$ =$C_{\beta I}$ • Factor 2).

14.7.4.6 Triangular end relief I and II

The corners are broken.

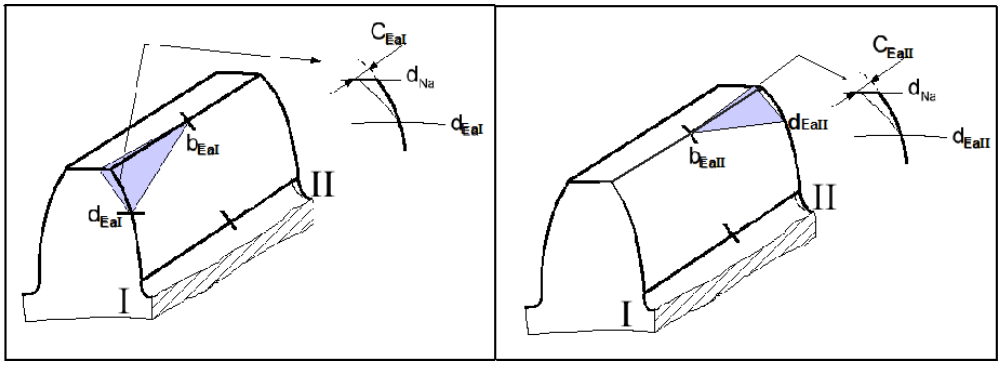


Figure 14.33: Triangular end relief I (left) and II (right)

where

C_{Ea}	Tip relief	d_{Ea}	Modification end diameter
L_{Ea}	Resulting triangular end relief length	b_{Ea}	Triangular end relief length
d_{Ef}	Modification end diameter	b_F	Usable facewidth

In the KISSsoft system, enter the value C_{Ea} in the input field. Then go to the Coefficient 1 input field and enter the quotient of L_{Ea} / m_n. Then go to the Coefficient 2 input field and enter the quotient of b_{Ea} and facewidth b.

14.7.4.7 Twist

`Twist` is the torsion of the transverse section profile along a helix. Usually, the angle increases in a linear progression from the start of the effective flank to its end. A positive directional torsion moves clockwise away from the observer. See also Figure 14.34. Modification C can be input as either a positive or negative value.

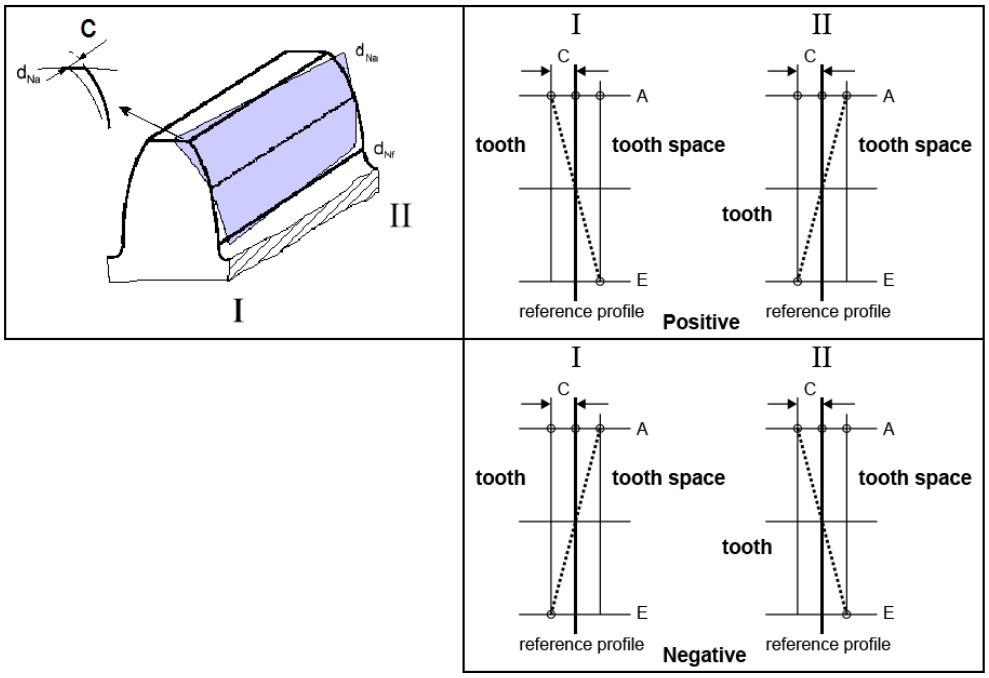


Figure 14.34: Twist

where

C	Relief on dNa at I		
dNa	Active tip diameter	dNf	Active root diameter

The notation used here is also shown in sections 14.8.4.2 (see section "Helix angle modification" on page II-374) and 14.8.3.4 (see section "Pressure angle modification" on page II-371).

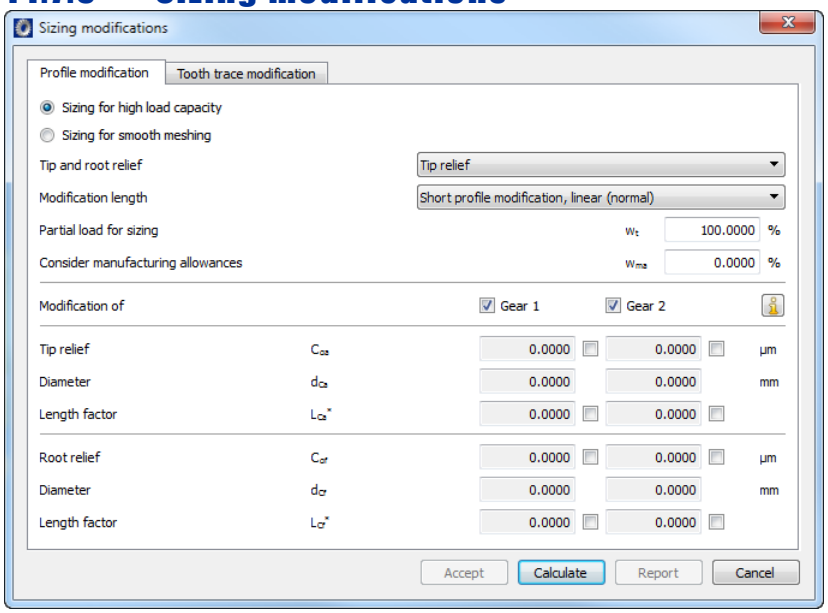
14.7.4.8 Topological modification

The Topological modification option enables you to define any type of modification. The actual modification is described in the file that is to be imported. You will find an example of this type of entry in the "topological_template.dat" file

in the dat directory. The file's name indicates its purpose. You can define coefficients in any slice and for any rolling depth. When the file is imported, these coefficients are multiplied by the value entered under Ca. You can display and check the modification by clicking `Graphics > 3D Geometry > Modifications`.

You will find an Excel application, "Topological Crowning.xlsx", in the \dat directory. You can use this to edit the table in which the topological modification is defined and then copy it to a .dat file. This Excel file also has an example of how to define a negative profile crowning.

14.7.5 Sizing modifications



Click the  button, as shown in Figure 14.23 on page II-361, to open the `Sizing modifications` dialog. The next two sections describe the basic method for performing profile and tooth trace modifications.

14.7.5.1 Profile modification

a) Tip relief on the driven gear reduces the entry impact, whereas tip relief on the driving gear reduces the exit impact. Tip relief is therefore usually applied to both gears. It is only applied to the driven gear alone in exceptional circumstances.

b) When calculating the profile modification, you must always specify the tip chamfer. If not, the active involute will not be included in the calculation.

c) Tooth contact stiffness is always calculated in accordance with the selected calculation method. Alternatively, you can derive the contact stiffness from the tooth form (see page II-298).

d) The points along the length of path of contact are labeled according to ISO 21771. In a situation involving a driving pinion, a tip correction must be applied on the pinion from H -DE to E (or D to E) and on a gear, from A to H -AB (or from A to AB). For a driven pinion, the descriptions are swapped according to ISO 21771 (A becomes E, E becomes A).

e) KISSsoft calculates the tip relief value for a nominal torque that was changed by a modification value. In the case of gears that do not always have the same operating torque, the modification value is assumed as approximately 50-75% of the maximum moment, evenly distributed across the pinion and the gear. The default value for tip relief C_α is defined using the mean value of the data as defined by Niemann. A (somewhat greater) value is set as the meshing start (C.I) at the tip of the driven gear. The value (C.II) is set as the value for the meshing end at the tip of the driving gear. When you select profile modification `For smooth meshing`, the value C.I is also set at the meshing end.
For deep toothing, where $\varepsilon_\alpha > 2$, the load-dependent portion of tip relief is reduced, depending on accuracy grade, to 12.5% (for accuracy grade 8 and poorer) and up to 50% (for accuracy grade 5 and better).

f) KISSsoft also calculates the modification length. The "long modification", as it is known, goes from point A to point B of the length of path of contact, but the "short modification" only goes to point H-AB (the midpoint between A and B). Usually, the short modification is selected. However, the modification length (from A to AB) should not be too short. A minimum length (related to the tooth height) of $0.2m_n$ should always be present. This value is checked during sizing. If the length from A to AB is too short, the program prompts you to use a minimum height of $0.2m_n$. However, the result of this is that the contact ratio in the unmodified part will be less than 1.0 (< 2.0 for deep toothing where $\varepsilon_\alpha > 2$). The program then displays an appropriate message.

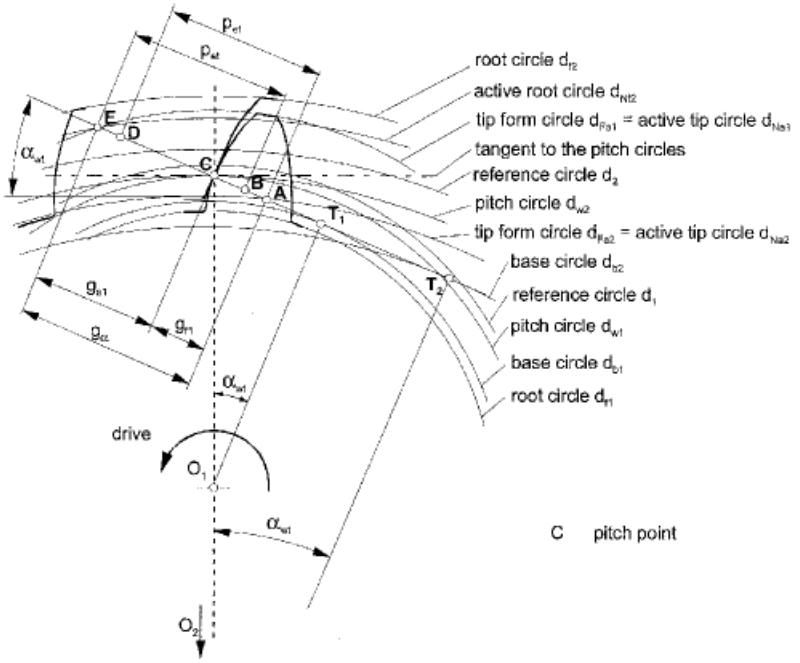


Figure 14.34: Length of path of contact for a cylindrical gear

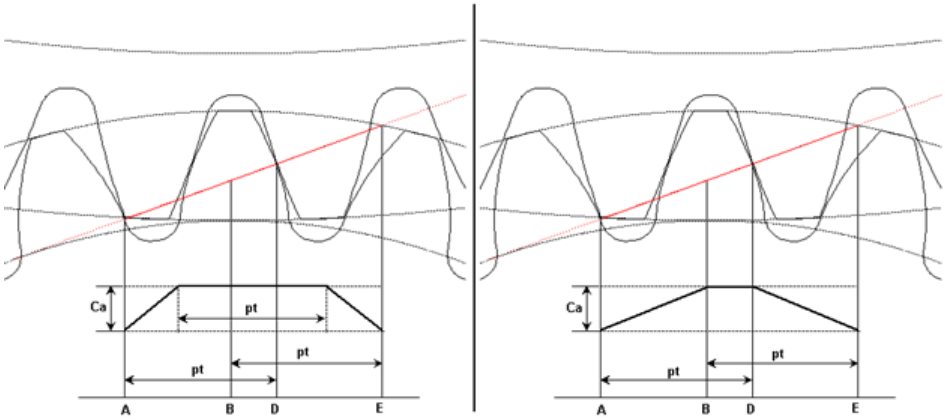


Figure 14.35: Short (left) and long profile modification

g) The type of `Profile modification` has an effect on how scuffing safety (see section "Structural factor XwrelT or structural factor Xw (scuffing)" on page II-301) is calculated.
If you select `For high load capacity gears` in accordance with the suggestion stated in Niemann, the profile modification at the end of the contact (point E on the path of contact) is somewhat less than that at the beginning of the contact.
If you select `For smooth meshing`, the profile modification at the end of contact is set to the same values as that for the beginning of contact.

14.7.5.2 Tooth trace modification

The procedure you use to size a width modification, for example, an end relief (see section "Linear end relief I and II" on page II-372) or crowning (on page II-375), is specified in ISO 6336, Part 1, Annex B.

If you are working with planet systems, the proposed tooth trace modification can be used to compensate for a misalignment of the planet and the sun. It can also take into account the effect of torsion on a particular gear. You will find more detailed information about the direction of torque and the axis alignment in the "Defining the misalignment of individual parts" section.

However, be aware that this sizing suggestion only applies to planets with a symmetrical misalignment because of the torsion that influences the beam.

The proposed modifications (KHβ = 1) are only then correct if the system has a single planet. If several planets are present, the program searches for the best compromise so that the proposed modification minimizes the maximum KHβ for all the planet contacts.

If ISO 6336-1, Annex E, is applied, an additional precise sizing of the tooth trace modification, as eccentric crowning or centrical crowning, can also be performed for calculating the face load factor.

14.7.6 Notes about profile modification

If you select a `short profile modification` , the length of the modification at the tooth tip (or at the tooth root) for both gears is defined such that the contact ratio of the part of the tooth flank that has not been changed by the modification is still exactly 1.0 (for deep toothing with $\varepsilon_\alpha > 2$ is still exactly 2.0). This type of profile modification is the one most frequently used because it always ensures a sufficiently large transverse contact ratio (no matter what load is involved).
This short profile modification runs from point A of the path of contact up to the point H-AB (the midpoint between point A and point B), or from E to H-DE. However, the result of this is that the contact ratio in the unmodified part is 1.0.

However, if you want to design a gear unit that runs as quietly as possible, it is usually better to select the long profile modification, because the transmission error is usually much lower in this case. To properly evaluate the effect of a profile modification, we recommend you calculate the meshing under load (see section "Contact analysis" on page II-411).

14.7.7 Using diamond dressing wheels and grinding worms

Click on the Conversion button  in the `Modifications` tab to call an option which enables you to find out whether suitable grinding worms (with their associated diamond dressing wheel) are present for processing the gear. A list of all suitable dressing wheels is generated from the «DressingWheel.dat» file. Dressing wheels which are not suitable for the currently entered module and pressure angle are ignored when the file is read.

The file to be loaded must be in the KISSsoft installation directory either in the …\ext\dat\ or …\dat\ sub-directory (although KISSsoft will searched for it in \ext\dat\ first). When the file is read, lines that start with a backslash are ignored. In a line, all entries after the first are separated by semicolons (starting from the

left):
1. Text, is ignored when the file is read
2. Text, is ignored when the file is read
3. Normal module [mm]
4. Pressure angle EW [°]
5. Profile crowning (depth crowning) radius HBR [mm] (during the reading of «straight», this radius is set to 10^{10}.
6. Length of the linear tip relief LKRN [mm]
7. Tip relief angle WKRN [°] or radius RKRN [mm] (if the angle value is in degrees and arc minutes: x°xx, or xØxx. If it is radius: Rxxx)
8. Text, is ignored when the file is read
9. Text, is ignored when the file is read
10. Text, is ignored when the file is read
11. Position HP of the high point of the dressing wheel profile crowning (depth crowning) [mm] (stated along the flank, from the tip)
12. Dressing wheel addendum hfP0 [mm]
13. Dressing wheel dedendum haP0 [mm]
14. Gap L* between the flank of the dressing wheel and the grinding worm [mm] (measured along the datum line)
15. Dressing wheel tooth root radius Rho [mm] of the dressing wheel
16. Dressing wheel article number/label
17. Root relief length LFRN [mm] (is read, but currently ignored)
18. Root relief angle WFRN [°] or radius RFRN [mm] (is read, but ignored)
19. Number of starts, grinding worm
20. Reference circle [mm] of the grinding worm

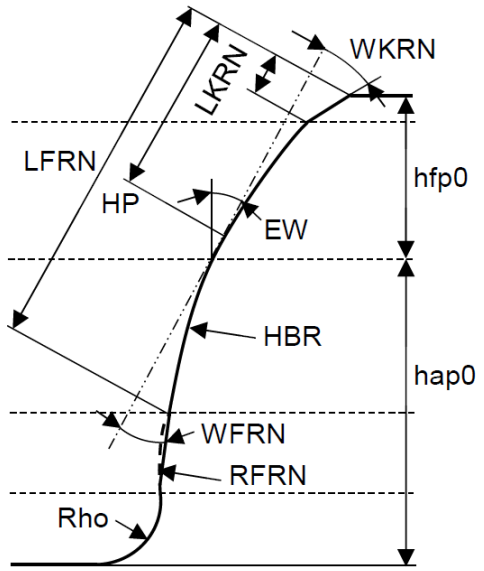


Figure 14.36a: Dressing wheel with linear tip relief

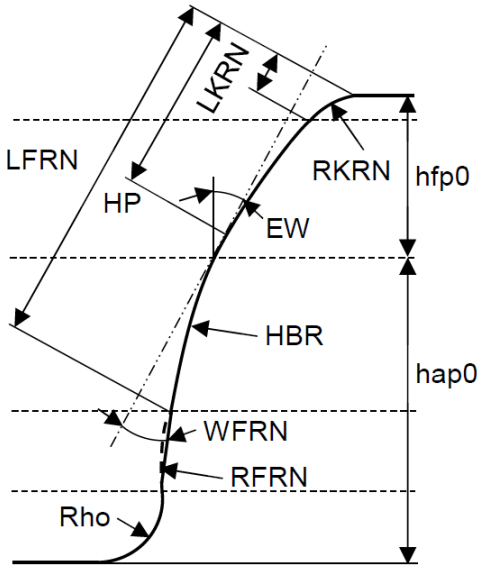


Figure 14.36b: Dressing wheel with radial tip relief

This enables you to load the same dressing wheel several times, in different lines, with different grinding worms, for example. Do not enter a semicolon at the start or end of the line. Do not leave the last line in the file empty.

The achievable tip and root modifications, according to the selected gear, with the loaded dressing wheels, are displayed in the first window. Only gears for which pre-machining (without a topping tool) has been defined, and a machining input has been entered, are displayed.
The system displays the suitability of the particular dressing wheel in the third column. Here, the following apply: "+" suitable without modification of the gap L* ("air between the flanks"), "+-" suitable if the gap is modified L* and "-" not suitable.
The modifications and toothing diameter are calculated using the tooth thickness tolerance in the `Tolerances` tab for the tooth form calculation. In addition, the target tip relief Ca and the target modification length LCa* are also displayed, according to the selected gear, (and according to the entries in the Modifications table in the `Modifications` tab).

When a dressing wheel is selected, the basic data for the grinding process is displayed in a second window called "Selected dressing wheel/grinding worm".

You can move the dressing wheel by Δh by modifying the gap L* and also modify the grinding worm's lead angle and pitch diameter. Modifications are recalculated and displayed.

If a grinding worm has been finally selected, the data in profile modifications that most closely match the dressing wheel, are adjusted. During this process, the data for the grinding worm and the dressing wheel is written to the "Dressingwheel.tmp" file. This file is stored in the Windows Temp folder.
The modifications are entered in the Modifications table, and all previous profile modifications are deleted. The gear contour is now defined in the same way as it will be produced in the grinding process with the selected grinding worm. The grinding depth in the root circle is set in the `Final machining` tab. The grinding process is set to "Generation grinding" with "Grinding of root and flank"

The report also contains the diameter of measuring ball circle dmess with the associated tip relief Ca. The measurement circle lies between the tip form circle dFa and the start of the tip relief dCa (dmess = (2*dFa + dCa) / 3).

NOTE:
You can perform all the standard manipulations with dressing wheels described here in the "Selected dressing wheel/grinding worm" window.

a) **Sharpening the grinding worm**
This is performed very frequently, and regularly after the production of a certain number of gears. Achieved by a radial reduction of around 0.5-1.0 mm on the dressing device. Here, the gap L* is kept constant (automatically by the dressing device). Resharpening with an unchanged L* only produces a minimal change in the worm lead angle.

b) **Modifying the gap L*** (without changing the radial reduction)
L* can be increased on a grinding worm that has already been profiled (i.e. dressed). L* can only be reduced on a grinding worm that is new, i.e. has not yet been profiled.

c) **Modify the radial reduction** ("center distance" dressing wheel-grinding worm a)
a center distance can only be increased on a grinding worm that is new, i.e. has not yet been profiled. a can normally be reduced on a grinding worm that has already been profiled (i.e. dressed). Theoretically, in this case (radial reduction), the gap L* could be reduced at the same time. This should prevent a minimal change to the grinding worm pitch (in a constant diameter dx), which would need to be corrected.

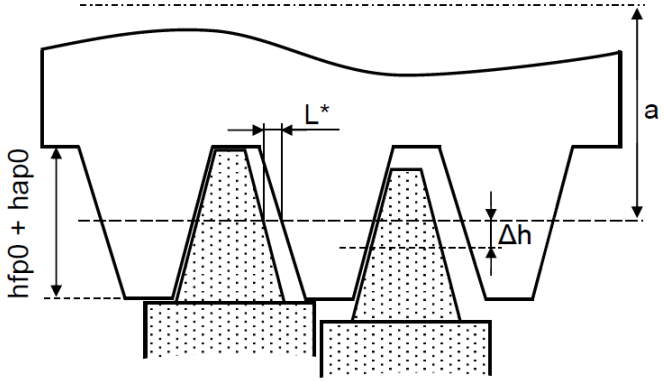


Figure 14.36c: Grinding worm with dressing wheels (examples shown with and without displacement Δh)

14.8 Tooth form

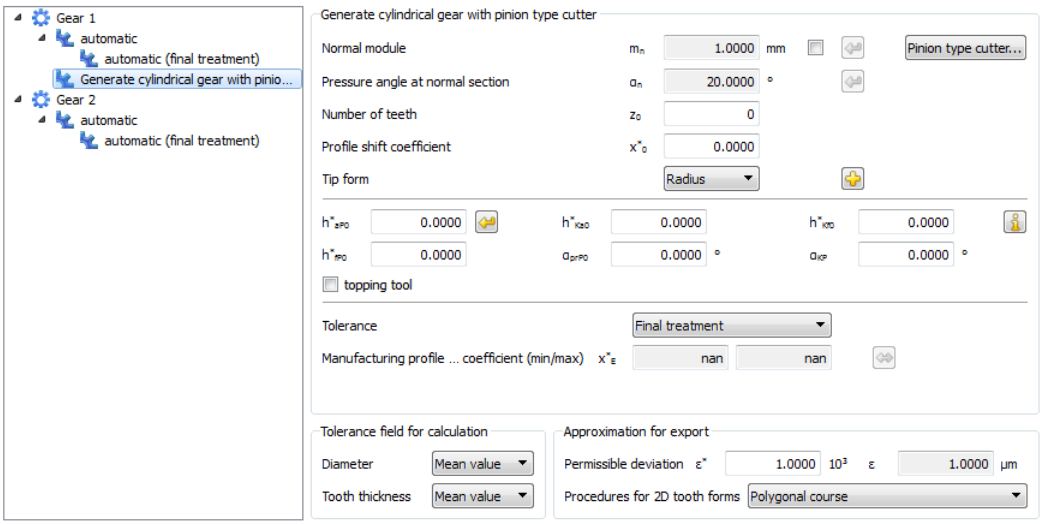


Figure 14.36: `Tooth form` input window

In addition to the actual calculation, the tooth form calculation offers a number of other options, because it simulates manufacture with a precisely defined cutter. These options include, for example,

- tooth form modifications with profile modifications and root contour optimization
- taking into account several steps in the manufacturing with different tools
- calculating the cutter (pinion type cutter or hobbing cutter) required to manufacture the toothing (for example, for tooth forms that have been imported from a CAD program or for modified tooth forms)
- tooth form modifications for injection molds or for use in manufacturing pinion type cutters

NOTE

Special tutorials that specifically deal with tooth form modifications have been designed and provided for use. You can download these tutorials from our website, `http://www.kisssoft.ch`.

The `Tooth form` calculation module input window consists of two columns. The left-hand column shows which operations are to be performed on the gears. The right-hand column consists of the `Tolerance field for calculation`

and `Approximation for export` groups and the corresponding operations group.

14.8.1 Context menu

Click the right-hand mouse button in the operation directory structure group to open a context menu. This menu refers to the active element in the directory (shown with a blue background).

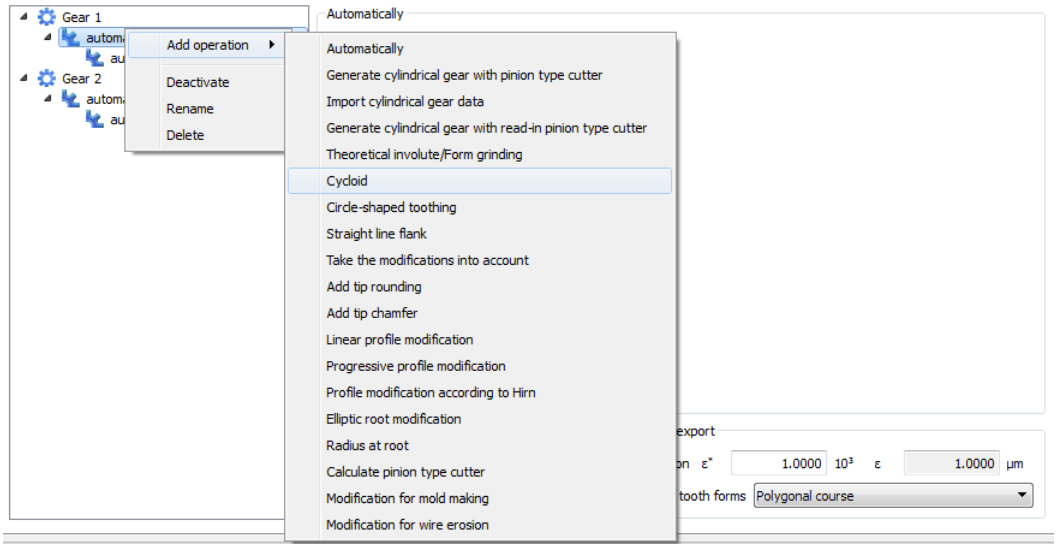


Figure 14.37: Context menu in the tooth form calculation

The context menu gives you these selection options:

- **Add operation** Select this menu option to open a sub-menu that lists the operations (see page II-389) that can be performed on a particular gear.
- **Choose as result** This result is usually displayed in the graphic and used in the strength calculations. The default setting is for the results of the last operation to be displayed here, unless the modification involves mold making, wire erosion, or a pinion type cutter.
- **Activate/Deactivate** Use this option to remove an operation that has been assigned to a gear from the list without deleting it. The icon is then marked with a red cross. The `Activate` menu option returns a deactivated operation to the list of active operations. The red cross then disappears.

- **Rename** Changes the name of an operation. Note that, if you change the name of an operation, this does not change the area name in the right-hand sub-window.
- **Delete** Permanently removes an operation entry,along with all its associated parameters.

14.8.2 Operations

You can calculate the tooth form by using a combination of various different operations. You can apply one processing step after another, for example, using a hobbing cutter or a pinion type cutter and applying modifications such as rounding or profile modifications. You can label each operation to make it easy to identify at a later point in time.

14.8.2.1 Automatic

The default operation for the tooth form calculation is `Automatically`. The tooth form (with all its pre-machining and final machining) is then generated using the data entered in the standard tabs (see page I-89). If you have defined modifications, these are taken into account when generating the tooth form. You can also disable this part of the operation in the context menu. The same applies to any tip chamfer or rounding you specify. If you select ZA as the flank shape, a ZA worm will be generated. Otherwise a ZI worm is created.

NOTE

If the `Automatically` operation has been disabled, none of the data input in the `Reference profile` or `Modifications` input windows will be taken into consideration.

14.8.2.2 Generate cylindrical gear with hobbing cutter

To generate a cylindrical gear with a cutter, input the gear reference profile. When you add this operation, the window is filled automatically, based on the values you defined in the `Reference profile` input window. If the tool is a non-topping tool, the addendum of the reference profile is determined automatically from the tip circle, and not transferred from the values you input. For special applications (manufacturing a gear with a cutter with a different module), you can modify the module mn and the pressure angle αn. You can then use the sizing buttons. The sizing buttons () calculate the correct value in each case for the specified base circle. Click the `Cutter...` button to open the `Define cutter` (see page II-347)

window which displays a list of tools. To define the tolerance field, you can either enter the generating profile shift coefficients directly (`Own inputs`), or use the pretreatment or final treatment tolerances.

The hobbing cutter data can also be input as factors or as absolute lengths (mm or inch). These selection options make your job much easier if the hobbing cutter data are the lengths (in mm or inches) given in a drawing.

When sizing haP0*, the system calculates the value, which is then used to manufacture the involute up to the active root diameter. The proposed value shown here is the exactly calculated value, to which 0.05 is added (to obtain a small distance between the root diameter and the active root diameter).

If you use the sizing button to define the grinding wheel, the radius ρaP0 should be small (e.g. 0.1*mn), otherwise the grinding process may reach the root radius.

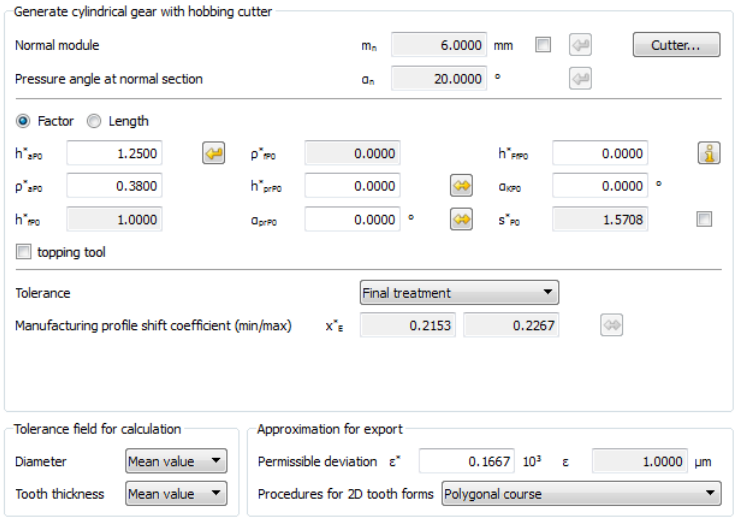


Figure 14.38: Operation: `generate cylindrical gear with hobbing cutter`

NOTE

The cutter information entered here is independent of the data specified in the `Reference profile` input window. In other words, the tooth form calculation is based exclusively on the values defined in the `Tooth form` input window.

14.8.2.3 Generate a cylindrical gear with an imported hobbing cutter

You can import the cutter contour from the CAD system in `dxf` format. To do this, define 1/2 tooth from the tip at A to the root at E:

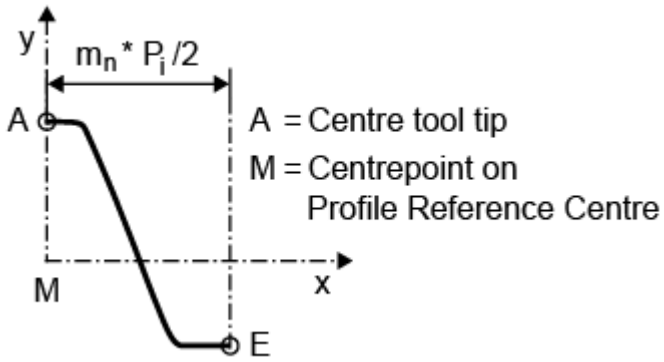


Figure 14.39: Tool profile

You can either specify the layer that includes the contour or select `ALL` for all the data. You can then decide whether to import the tool in transverse section or in normal section, and also change the module. The profile shift coefficient you enter here determines the tooth thickness.

Click on the "Cutter for displaced generation" option to select a normal module for the tool that differs from the cylindrical gear generated by the program.

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

14.8.2.4 Generating a cylindrical gear with a pinion type cutter

You must define the geometry of the pinion type cutter if you want to calculate the tooth form of gears manufactured using a shaping process.

Required input data:

- Reference profile of the pinion type cutter
  In the reference profile of the pinion type cutter, swap the values of the tip and root used in the reference profile of the work gear at $x_0 + x_E = 0$. In other cases, you need to input a displacement at x0.
- Z_0 Number of teeth on the pinion type cutter

- x_0 Profile shift for the pinion type cutter
  (if x_0 is not known, you can use the cylindrical gear calculation to define the profile shift from the tip diameter or the base tangent length → further info (see section "Profile shift coefficient" on page II-267))
- or determine the length of the chamfer on the pinion tooth tip s or the radius of the rounding r on the pinion tooth tip (see Figure 14.40)

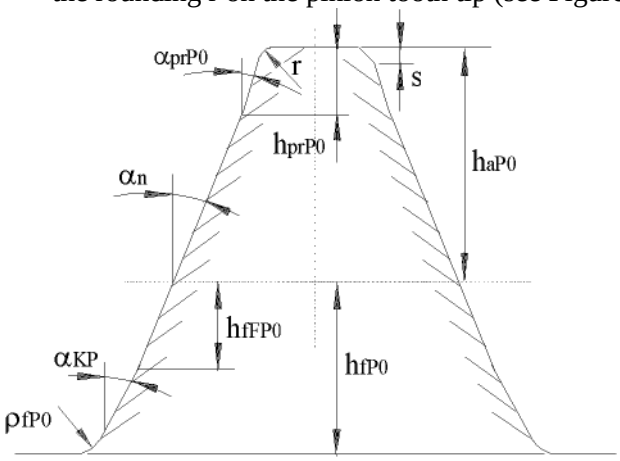


Figure 14.40: Tool profile

14.8.2.5 Generate a cylindrical gear with an imported pinion type cutter

You can import a pinion type cutter as a `*.dxf` file. In this situation, the system imports a half tooth from the specified layer (select ALL for all layers), as shown below:

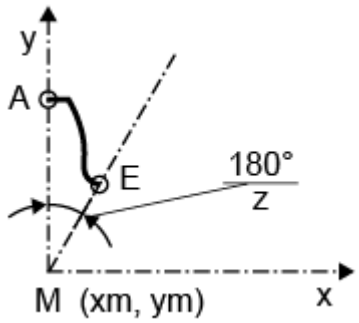


Figure 14.41: Pinion type cutter coordinates

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m is a required entry)
z	:	Number of teeth

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing). In this case, you must specify the number of teeth on the pinion type cutter and the manufacturing center distance.

14.8.2.6 Reading (importing) a cylindrical gear

You can import a cylindrical gear directly as a `*.dxf` file. To do this, define a half tooth in the selected layer:

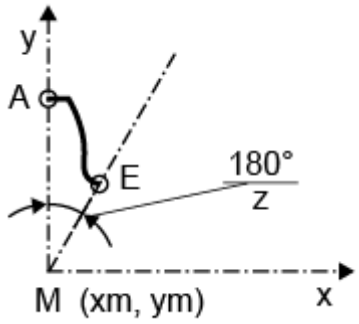


Figure 14.42: Coordinate system for the import

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m is a required entry)
z	:	Number of teeth

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing).

14.8.2.7 Adding tip rounding

You can add tip rounding as a tooth form modification. The rounding can be added either in the transverse or axial section.

14.8.2.8 Adding tip chamfer

You can add a tip chamfer as a tooth form modification. The chamfer can be added either in the transverse or axial section and is defined by the starting diameter and an angle.

14.8.2.9 Linear profile modification

In a linear profile modification, the tooth thickness is reduced in a linear progression from the starting diameter to the tip (relief C_a on each flank as a tooth thickness modification).

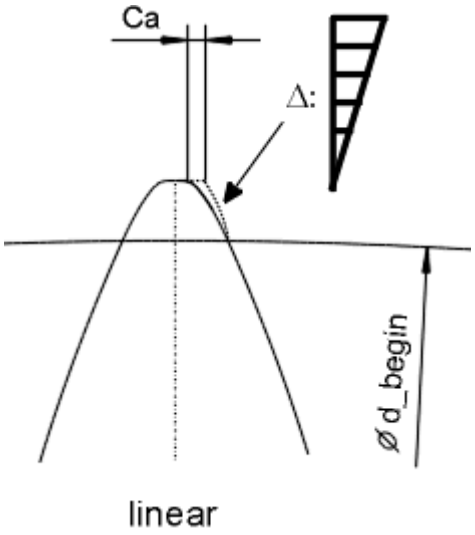


Figure 14.43: Linear profile modification

14.8.2.10 Progressive profile modification

In a progressive profile modification, the tooth thickness is reduced from a starting diameter to the tip (relief C_a on each flank as a tooth thickness modification) in accordance with

$$\Delta A_s = 2 \cdot C_a \left(\frac{d - d_{begin}}{d_a - d_{begin}} \right)^{factor\,/\,5} \qquad (14.21)$$

. The coefficient controls the course of the relief. A coefficient of 5 represents a linear relief. For more information, see also Figure 14.44. If a coefficient greater than 5 is used, the progressive profile modification moves tangentially into the unmodified tooth flank. This is the preferred option if larger reliefs are to be achieved. We do not recommend you use a coefficient of less than 5 (some of these

lower values are simply ignored by the program). Coefficients greater than 20 are also ignored. In this case, a coefficient of 20 is used.

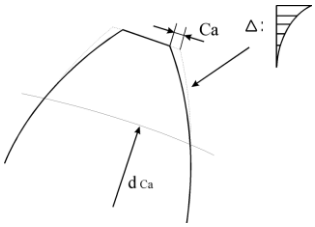


Figure 14.44: Progressive profile modification

14.8.2.11 Entry curve as specified by Hirn

An entry curve that passes into the involute tangentially is applied to the tooth tip starting from the specific diameter d_{begin}. This entry curve consists of three arcs of circle. The bend in the curve increases from arc to arc of circle, so that the final curve is tangential to the tip circle. This modified tooth form (also called a hybrid tooth) has significant benefits, because it results in extremely quiet running, despite relatively imprecise production methods. For this reason, the modification is applied for plastic products, for preference. See Figure 14.45.

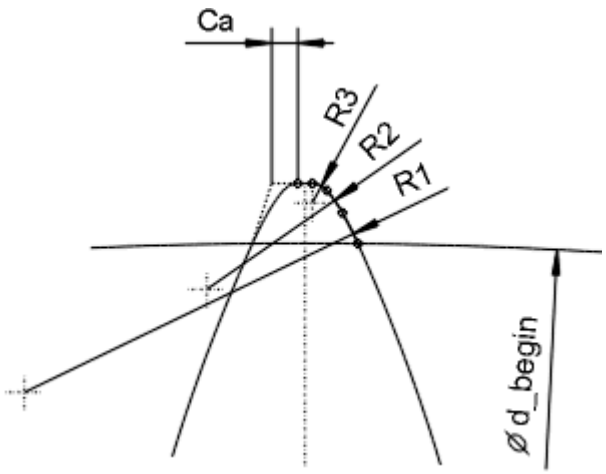


Figure 14.45: Profile modification as specified by Hirn

An entry curve is usually only applied to deep toothing with transverse contact ratios of greater than 2.1. In addition, KISSsoft can use its sizing function to suggest a suitable starting point (diameter) for the entry curve and the tip relief va-

lue. To do this, it uses the profile modification (see section "Modifications" on page II-361) calculation.

The start of the entry curve is defined as follows:

- For a transverse contact ratio of 2.0: The active involute is reduced until the transverse contact ratio is exactly 2.0.
- For a transverse contact ratio of less than 2.0: The diameter is calculated so that a medium tip relief is created, i.e. a transverse contact ratio of above 1.0 is reduced by approximately 50%.
  For example, from 1.8 to 1.8 - 0.5 . 0.8 = 1.4.

The exact definition is shown here:
For a transverse contact ratio > 2.0 : d_{start} = minimum (d_{PunktD}, $d_{PointE0.2}$)
For a transverse contact ratio < 2.0 : d_{start} = minimum (d_{PunktE}, $d_{PunktE0.2}$)

The relief C_a at the tip is defined as shown here:

- For top land less than 0.21 .m_n: 0.5 . Tooth thickness - 0.01 .m_n
- For top land greater than 0.21 .m_n: 0.10 .m_n to 0.12 .m_n

14.8.2.12 Elliptical root modification

The root fillet is replaced by an ellipse-shaped contour, which progresses tangentially in the flank and root circle. The aim is to achieve the greatest possible radius of curvature. The course of the contour can be influenced by the coefficient in the range 1 ÷ 20. If you click on the sizing button for the diameter, when you start the modification, the program suggests a value for the active root diameter (slightly increased, to prevent problems when applying the modification to an undercut). The definable length on the root circle is then set to > 0 if you want an area of the tooth form to run on to the root circle. For example, this is a good idea if the root circle is to be measured with measuring rollers.

The greater tooth thickness in the root area means that the generation process with the other gear in the pair must be checked.

14.8.2.13 Root radius

The root contour is replaced by an exact arc of circle with a specifically definable radius. After you make this modification, check the generation process using the other gear in the pair.

14.8.2.14 Theoretical involute/form grinding

The tooth form is construed mathematically. The involute is defined using the module and pressure angle along with the tip and root diameter. The tooth thickness is defined by the profile shift coefficients. You can also define a root radius (in the transverse section). This option is suitable for involute gears that cannot be manufactured by a gear generation process (e.g. internal gears with 4 teeth) or for a processing step involving form grinding.

14.8.2.15 Cycloid

You can select a cycloid as a special tooth form. The cycloid is defined with two rolling circles and the tip and root diameters. In the main calculation, the tooth thickness is defined by the allowances. Rolling circle 1 rolls on the inside on the reference circle and therefore cuts the dedendum flank. Rolling circle 2 rolls on the outside and generates the tip. Rolling circle 1 of the first gear should correspond to rolling circle 2 of the second gear. Sizing a cycloid toothing is made easier if you calculate the other gear in the pair using the data from the first gear during the optimization process.

Use the Stress curve and Kinematics analyses modules to analyze the strength and geometry properties of cycloid toothings.

14.8.2.16 Circular pitched teeth

The circular pitched teeth special toothing type can be defined using the tooth flank radius and the tooth thickness at the reference circle. An arc of circle is created in the root area.

A classic arrangement of circular pitched teeth, for example, as specified in NIHS 20-25 [67] consists of an arc of circle with radius r starting from the reference circle, a straight line that progresses in the direction of the center of the gear below the reference circle, and a full root rounding.

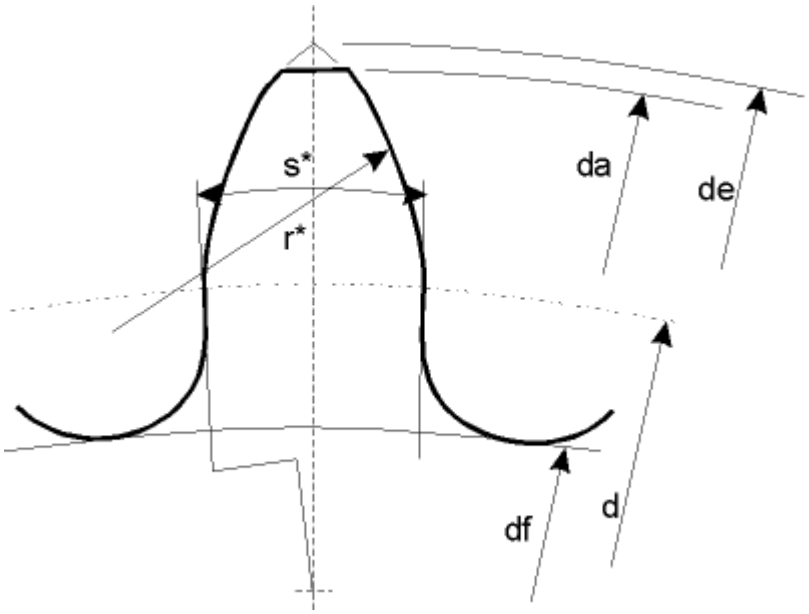


Figure 14.46: Arcs of circle on the tooth

14.8.2.17 Straight line flank

You can select a straight line flank as a special tooth form. The straight line flank is defined by the tooth thickness at the reference circle (theoretical toothing), the space width angle in transverse section, the tip and root diameter as well as the manufacturing profile shift coefficient (dependent on the tolerance). You can also predefine radii for tip and root rounding.

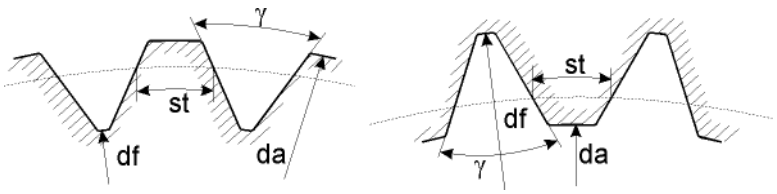


Figure 14.46b: Straight line flank

14.8.2.18 Generate with the other gear in the pair

You can use the other gear in the pair to calculate the tooth form on all the gears, except on gear 1 (gear number - 1). In this case, you can overwrite the manufacturing center distance and the tip circle. The clearance between the gears can be generated either by reducing the manufacturing center distance or by inputting the circumferential backlash. The tip clearance is achieved by increasing the tip circle of the tool.

14.8.2.19 Calculating the reference profile

You can calculate the reference profile of an existing tooth form. A hobbing cutter can then be used to manufacture it. The manufacturing center distance can be changed in this calculation. This has a significant effect on the practicability of creating a tooth form using the generation process. In contrast, the value you input for the profile shift has no effect on the profile. Instead this influences the null point.

The calculated reference profile is then used as a cutter to calculate the cylindrical gear again. By comparing the two tooth forms, you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click `Cutter/Tool` to display the reference profile in the graphic.

14.8.2.20 Calculating a pinion type cutter

You can calculate a pinion type cutter for an existing tooth form. To do this, enter the number of teeth on the pinion type cutter and the manufacturing center distance. The center distance has a significant effect on the practicability of creating a tooth form using the generation process. Try out a number of different values to find the best one.

The calculated pinion type cutter is then used as a cutter to calculate the cylindrical gear again. By comparing the two tooth forms you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click `Cutter/Tool` to display the pinion type cutter.

14.8.2.21 Generating a face gear with a pinion type cutter

This operation is not yet available. To generate a face gear, select the `automatic` option. Define the pinion type cutter in the `Reference profile` input window.

14.8.2.22 Generate a rack with a hobbing cutter

Once again, enter the rack's reference profile, as you do when generating a cylindrical gear using a milling cutter. In this case, the addendum is only relevant if you are using a topping tool. The profile shift is measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the pre-machining and final treatment tolerances.

14.8.2.23 Generating a rack with imported hobbing cutter data

You can define a cutter as a `*.dxf` file. In this case, the contour must be output as follows, so that the KISSsoft system can read the data correctly:

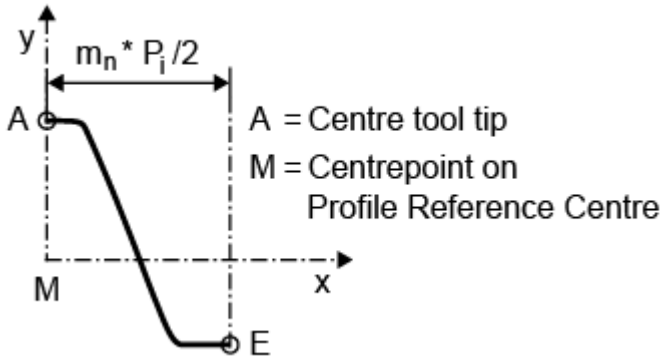


Figure 14.47: Tool profile

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing).

In addition to the contour, you must also define the manufacturing center distance. In this case, the reference line for the center distance is defined using the rack height.

14.8.2.24 Generate rack with a pinion type cutter

Once again, enter the reference profile of the pinion type cutter, as you do when generating a cylindrical gear using a pinion type cutter. The profile shift is measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the pre-machining and final treatment tolerances.

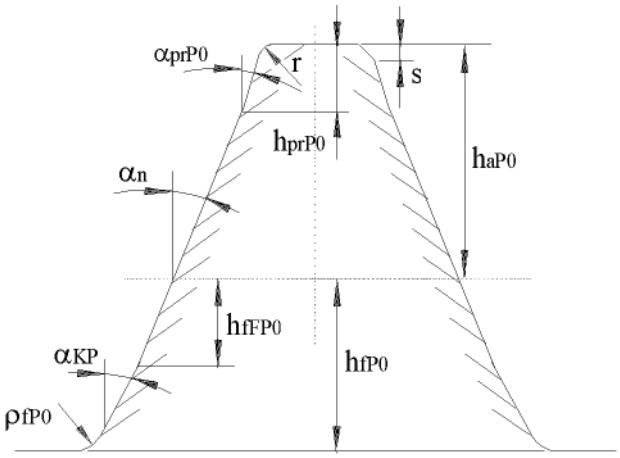


Figure 14.48: Cutter tooth geometry

14.8.2.25 Generating a rack with imported pinion type cutter

You can generate a rack with an imported pinion type cutter. In this case, you must specify the number of teeth on the pinion type cutter and the manufacturing center distance, in addition to the pinion type cutter contour in `*.dxf` format.

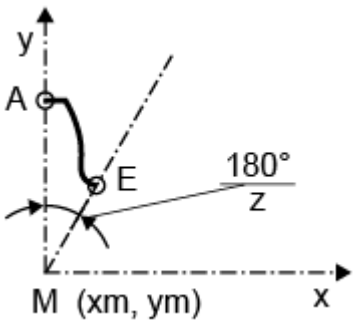


Figure 14.49: Coordinate system for the import

A : Mid tooth tip: Start of contour

E : Middle tooth space: End of contour

M : Center point (x_m, y_m is a required entry)

z : Number of teeth

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing).

14.8.2.26 Reading (importing) the rack

You can import a rack directly as a `*.dxf` file in the following format:

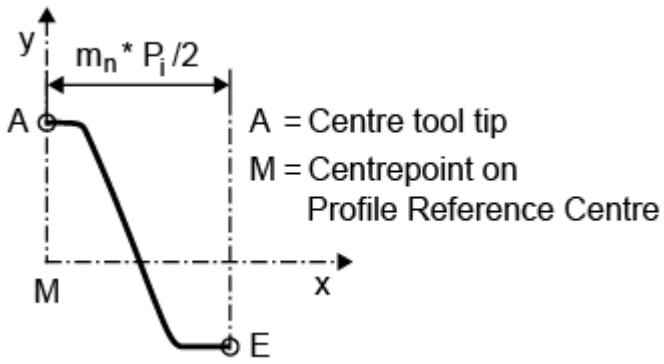


Figure 14.50: Tool profile

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing).

14.8.2.27 Generate a SA worm

This function is currently only available as the `automatic` option.

14.8.2.28 Reading (importing) a worm into the axial section

You can also import a worm in its axial section. In this case the contour is basically the same as the contour of the hobbing cutter, apart from the null point which forms the axis of the worm.

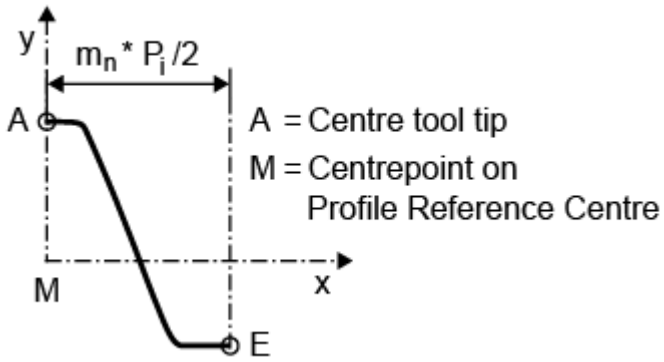


Figure 14.51: Tool profile

NOTE

The file (`dxf`) must only contain contours A to E in the layer you can specify for reading (importing).

14.8.2.29 Modification for mold making

When plastic gears are manufactured using the injection molding process, the material shrinks as it cools. To counter this effect, and manufacture precise tooth forms, the size of the cutter must be increased by the shrinkage amount. Depending on what type of material is involved, shrinkage may occur either radially or tangentially. If you enter the same values in the radial and tangential directions, the strain will be uniform in all directions

If the gear is injection molded around an inlay body, you must also input the external diameter of this body. The radial strains will then calculated using the "outside diameter of inlay body".

The modifications only affect the transverse section of the tooth form. No strain in the axial direction is present when a 3D volume model is generated. If you want to create an expanded 3D model of a helical toothed gear (if the strain is to be the same in all three axes), you can achieve this by scaling the module (m_n), the center distance and the facewidth.

EXAMPLE

In the main screen, increase the module, center distance and facewidths by the required strain coefficient.
Coefficient 1.02

$$m_n' = 1.02 \cdot m_n$$

$$a' = 1.02 \cdot a$$

$$b_1' = 1.02 \cdot b_1$$

$$b_2' = 1.02 \cdot b_2$$

Then, do not input a value for strain in the tooth form calculation.
This modification also increases the lead p_z by the same coefficient. However, the angle of rotation of the spirals across the facewidth remains the same.

Usual values are:

- Radial shrinkage approx. 2%
- Tangential shrinkage approx. 2%

14.8.2.30 Modification for wire erosion

In the erosion process, the electrodes must maintain a specific distance from the required shape, because additional material is removed due to the spark gap. This is usually taken into account by the machines involved in the wire erosion process.

When sink eroding an injection mold the eroding wire must therefore be thinner than the required shape by the amount of the spark gap. If a gear shaped electrode is used, the tooth will be correspondingly thinner. To achieve this, enter a negative value for the spark gap. Usual values for the spark gap are 0.03 to 0.07 mm.

After this modification you can also calculate the reference profile in the next step to determine the shape of a hobbing cutter for the electrodes.

NOTE

You can also use the wire erosion modification to check the practicability of using the wire erosion method. If the aim is to erode external teeth, enter one modification with a positive wire radius and then the second with a negative radius. If the aim is to erode an injection mold for external teeth, first input a negative radius and then run a modification with a positive radius. By comparing the tooth forms you can then see whether the form can be manufactured, or whether a practical form can be created using these two steps.

14.8.2.31 Modification for pinion type cutter

The effective cutting angle and the draft angle of the pinion type cutter cause a tooth form deformation in the projection of the pinion type cutter in the horizontal plane.
The conversion performed here deforms the tooth form in the horizontal plane so that the projection once again shows the exact tooth form once the pinion type cutter has been manufactured.

By grinding with angle ϕ(effective cutting angle) Q moves to P (see Figure 14.52). If the projection P' is to agree (exact contour in the horizontal plane), P must = Q must in the H plane.

$$PP' = \left(r_a - r_p\right) \cdot \tan(\phi) \tag{12.22}$$

$$P'Q = \left(PP'\right) \cdot \tan(\xi) \tag{12.23}$$

$$somit: \; P'Q = \left(r_a - r_p\right) \cdot \tan(\phi) \cdot \tan(\xi) \tag{12.24}$$

where

ϕ	Effective cutting angle
ξ	Tip draft angle in axial section
M	Pinion type cutter axis
r_a	Pinion type cutter tip circle radius
r_p	Coordinate of the point P

Conversion of the tooth form:

Given:	Exact tooth form in polar coordinates P = r (Angle)
Searched for:	Tooth form in H-plane P' = r' (Angle)
Solution:	$r' = r + \tan(\phi) \cdot \tan(\zeta)(r_{a-r})$

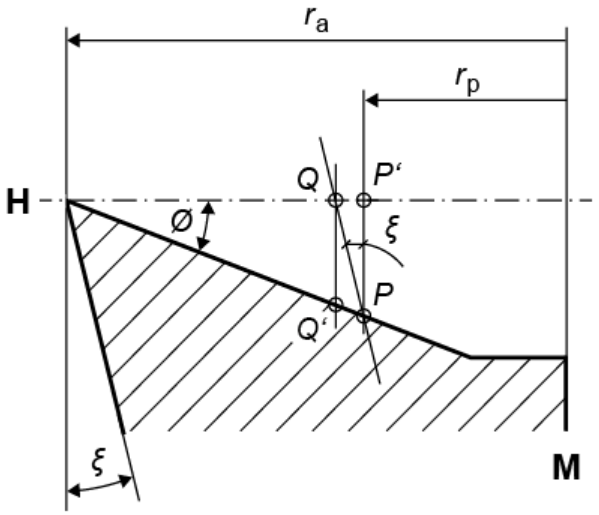


Figure 14.52: Pinion type cutter profile

14.8.2.32 Elliptical deformation

Applicable on the external gear (Gear1) of an internal-external cylindrical gear pair. This enables you to display the elliptical deformation of the race in a special gearbox in 2D. Typically z1+z2 = -2 applies here;

The contour of the race is stretched vertically by the lengthening factor and compressed horizontally so that the root circumference of the ellipse matches the root circle circumference of the undeformed gear. In a 2 D display, it is important you check:

- that the gear can be generated without collision over a pitch.

- that opposing sides mesh correctly.

If you need to make a modification, select a different lengthening factor or a different number of teeth (if the total number of teeth is an even number). Values between 0 and 5 % can be used as the lengthening factor.

Note: You cannot create a 3D output for this variant.

14.9 Tooth flank fracture

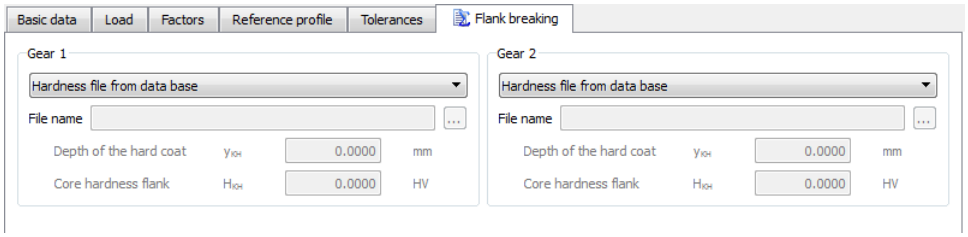


Figure 14.44: "Tooth flank fracture" input window

Tooth flank fracture appears in the area of the active tooth flank instead of in the area of the highest bending stress at the 30° tangent.

This calculation tab is designed to calculate the safety against tooth flank fracture in accordance with Dr. R.Annast [89]. The original calculation procedure from Dr. Annast requires detailed measurements of the gear hardness as a function of depth from the flank surface, to enable the depth of transition layer and the core hardness to be calculated.

Hardening depth ETH
You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final machining (after grinding).

When you input the data, the safety of the hardened surface layer is calculated automatically according to the DNV 41.2 [93]. A minimum value of t400 (nitrided steel) or t550 (all other steels) is used. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the section in [93] "Subsurface fatigue". The values required to define the EHT coefficient YC as specified in DNV 41.2 are also needed. The calculation is performed using different solutions than the calculation of the proposal for the recommended hardening depth, but still returns similar results (proposal for hardening depth). To obtain a proposal for a sensible hardening depth, we recommend you call the calculation in `Report->Proposals for hardening depth`. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

There are three calculation options:

- Using a hardness file for the gear material, if this file already exists in the database
- Selecting an independent file with the hardness information, or

☼ Direct input of core hardness and transition depth

If a file is used (case one and two) and only one pair of data is found, then it is assumed (in accordance with case three) that these values are the core hardness and the transition depth.

```
-- index number     depth (mm)  hardness (HV)
-- these lines are comments
DATA
1    0.2 700
2    0.3 675
3    0.4 650
4    0.5 625
5    0.6 600
*    *    *
*    *    *
*    *    *
END
```

Figure 14.45: Structure of the Hardness file

When a file containing hardness data is used (case one and two), the original data are used in the Annast algorithm. If the algorithm fails due to invalid data, the data is determined in accordance with the following regression formula (non-linear regression) $HV = a \cdot e^{-b \cdot y} + c$

. If the calculation with this data is also unsuccessful, a last attempt is performed (linearized regression) with the equation $\ln(HV) = \ln(a) + \ln(b) \cdot (-y)$

(as above, but without taking the constants into account).

The results of the tooth flank fracture calculation are in the special report (Report -> Tooth flank fracture).

14.10 Contact analysis

The load is taken into account when calculating the path of contact. This also calculates the face load factor $K_{H\beta}$ using the more precise method defined in ISO 6336-1, Annex E. In this case, the meshing stiffness can be calculated either according to Weber/Banaschek [69], ISO 6336-1 or `Own input`. (See "Contact analysis/Face load factor" section in the Settings chapter).
You can view the calculation results in the report or in the Graphic > Contact analysis menu option screen. The contact analysis can calculate either the transmission error as a length on the path of contact in µm or the angle of rotation error as an angle on the driven gear in °.

Contact data

You can predefine the single normal pitch deviation f_{pt} here. The proposed value for the single normal pitch deviation is then

$$f_{pt} = 0.5 \cdot \sqrt{f_{pt1}^2 + f_{pt2}^2}$$

calculated. You can enter the single normal pitch deviation with either a positive or negative algebraic sign. The results are then output for the case that the distance is too large or small. Contact analysis is performed over two pitches when single normal pitch deviation is taken into account.

Note:
Numerical problems may arise if the selected single normal pitch deviation is too large relative to the partial load.

The coefficient of friction between the flanks is assumed to be a constant in the meshing. Click the sizing button to accept the coefficient of friction as defined in ISO TR 15144. To take the effect of manufacturing errors (fma, fHβ) into account, select an appropriate value from the "Manufacturing allowances" drop-down list in the "Contact analysis" tab.
The manufacturing error increases the flank gap in the normal flank direction.

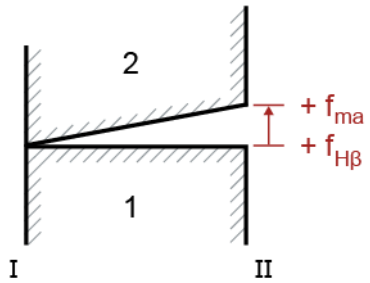


Figure 1: Definition of the positive direction of manufacturing errors fma and fHβ

A linear error distribution is assumed here, so the manufacturing error is 0 on side I, is at its maximum on side II, and increases in a linear progression along the facewidth. Manufacturing errors are taken into consideration in pairs, as either positive or negative values (up to a total of 5 combinations, but if there is only one combination, the manufacturing error is ignored).

Contact analysis

For the accuracy of calculation you can also choose between the levels "Own input", "low", "medium", "high" and "very high". The accuracy of calculation defines the termination criterion ε ($10e^{-3}$ to $10e^{-6}$) of the convergence condition

$$\left|\frac{T_c}{T_n} - 1\right| \leq \varepsilon$$

T_c= calculated torque
T_n= nominal torque

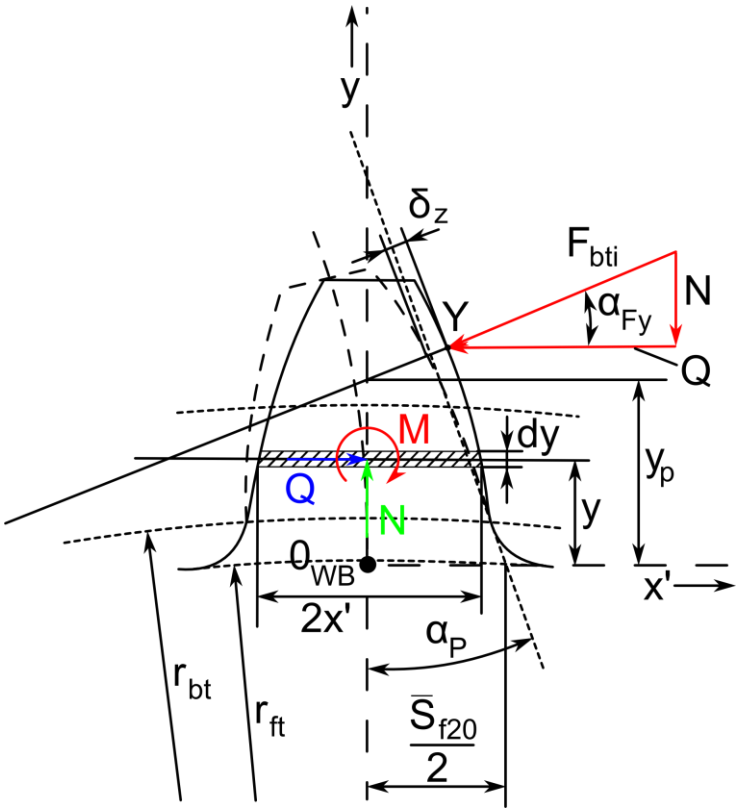
the contact analysis and the number of slices of the discretized model (see the Theory of contact analysis section). The number of slices is automatically set according to the gear geometry and the selected "Accuracy of calculation". The number of automatically determined slices increases correspondingly if a greater overlap ratio and "Accuracy of calculation" is selected. You can also enter the number of steps, slices and pitches manually by setting the "Accuracy of calculation" to "Own input" and clicking the Plus button next to it. If more than one pitch is calculated, the number of steps is distributed uniformly across all pitches. You can enter the "Partial load for calculation Wt" coefficient for the load. If you select the "Load distribution calculated with:…" option, the contact analysis's partial load wt is scaled according to the current setting in the module-specific settings. The partial load is taken into account both when calculating the shaft deformation and when calculating the nominal torque. To perform ISO-compliant calculations, make these settings in the "Module-Specific Settings" - for "Load factors", set "Load distribution calculated with to: $K_A K_\gamma K_\nu \cdot T_{nom}$", Axis alignment calculated with to: $K_A K_\gamma K_\nu \cdot T_{nom}$", and select the "Load distribution calculated with:…" option in the "Contact Analysis" tab.

14.10.1 Theory of Contact Analysis

As stated in Weber/Banaschek [69], the deformation of the meshing of gear pairs can be divided into three components:

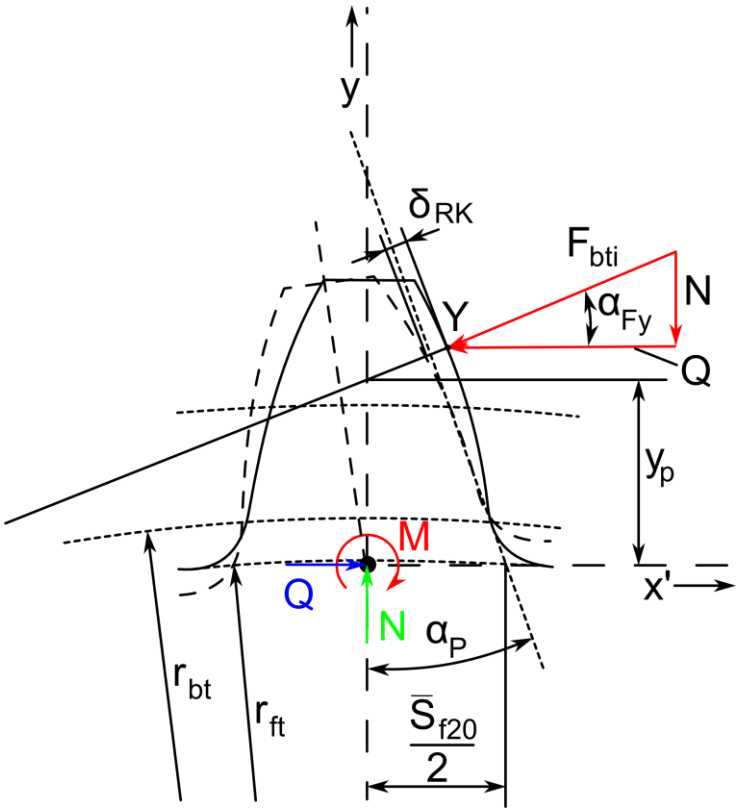
- Gear body deformation
- Bending
- Hertzian Flattening

Bending:



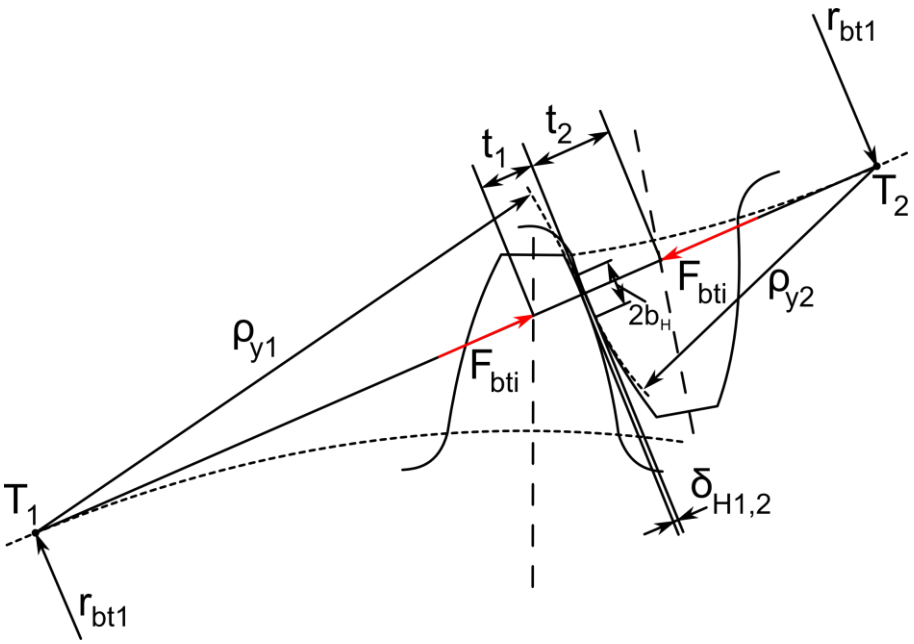
$$\delta_z = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\,\frac{1-\nu^2}{E}\left[12\int_0^{y_p}\frac{(y_p-y)^2}{(2x')^3}\,dy+\left(\frac{2.4}{1-\nu}+\tan^2\alpha_{Fy}\right)\int_0^{y_p}\frac{dy}{2x'}\right]$$

Gear body deformation:



$$\delta_{RK} = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\,\frac{1-\nu^2}{E}\left[\frac{18}{\pi}\,\frac{y_p^2}{\bar{s}_{f20}^2} + \frac{2(1-2\nu)}{1-\nu}\,\frac{y_p}{\bar{s}_{f20}} + \frac{4.8}{\pi}\left(1+\frac{1-\nu}{2.4}\tan^2\alpha_{Fy}\right)\right]$$

Hertzian flattening:



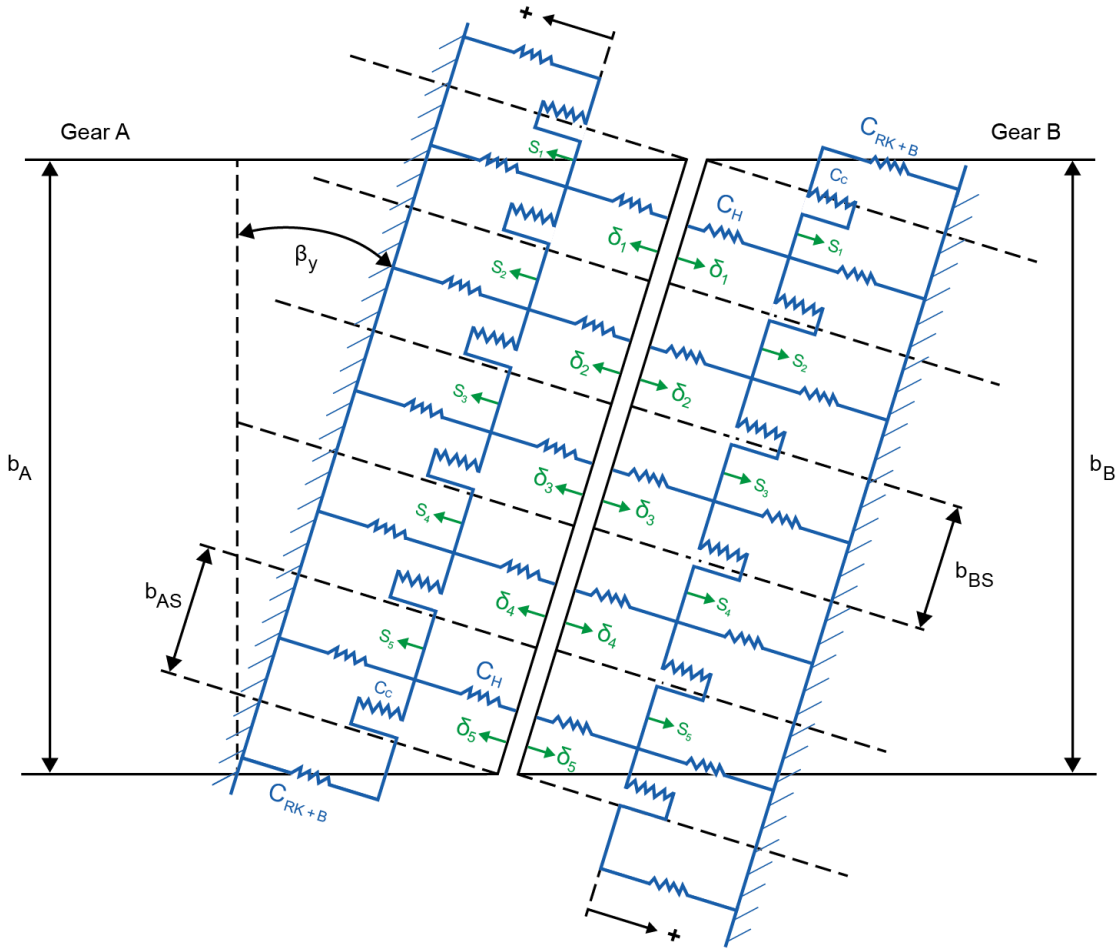
$$\delta_{H1,2} = \frac{F_{bti}}{\pi b_g}\left[\left|\frac{1-\nu_1^2}{E_1}\ln\left(\frac{b_H^2}{4t_1^2}\right)+\frac{\nu_1(1+\nu_1)}{E_1}\right|+\left|\frac{1-\nu_2^2}{E_2}\ln\left(\frac{b_H^2}{4t_2^2}\right)+\frac{\nu_2(1+\nu_2)}{E_2}\right|\right]$$

Total deformation has the effect that the contact point is displaced along the path of contact and the theoretical length of path of contact is elongated, in comparison to the actual length of path of contact. The transverse contact ratio under load is therefore greater than in the load-free state.

The spring equation F=d*C can be applied to calculate the components of the single contact stiffness from the individual deformation components and the normal force.

14.10.2 Discretized model

A discretized toothing model has been generated so that the deformation theory of meshing in gear pairs developed by Weber/Banaschek can be applied to three dimensional cylindrical gears with helical gear teeth.



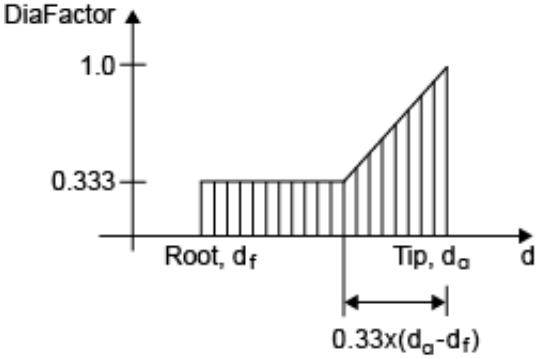
14.10.3 Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis

The large variations in curvature that occur during contact cause local, high peak values in Hertzian pressure and are a well-known problem. These values mean that any calculations, such as micropitting (frosting) according to Method A, which involve Hertzian pressure, will be incorrect. To avoid this, the peak values are filtered out after the calculation so that results that match the actual situation can be achieved.

This problem usually occurs on the tooth tip (where the curvature radius is 0). Therefore implement the following smoothing strategy to counter the curvature of the tooth form.

If the curvature radius ρ_y is less than 1.01 * m_n^*, then smoothing will be applied to ρ_y. (*: current setting in the code)

- SmoothFactor = 0.8 (=0: no smoothing, = 1: full smoothing)
- $\rho_{y+\Delta d}$ and $\rho_{y-\Delta d}$ are calculated.
- Δd = 0.3 * mn: the corresponding diameters are then applied to the diameter, i.e. d+Δd and d-Δd.
- If ($\rho_{y+\Delta d} > \rho_{y-\Delta d}$) then $\rho_{corr} = \rho_{y+\Delta d}$, otherwise $\rho_{corr} = \rho_{y-\Delta d}$,
- As this process has been designed for the critical tip area, the smoothing outside the tip area is reduced with the DiaFactor factor.



- This results in the "smoothed" ρ_y : ρ_{yNew} = SmoothFactor * DiaFactor * ρ_{corr} + (1 - SmoothFactor * DiaFactor) * ρ_{yOld}

14.10.4 Reduced stiffness on the side edges

The bending stiffness of the tooth in helical gears is reduced at the edges.

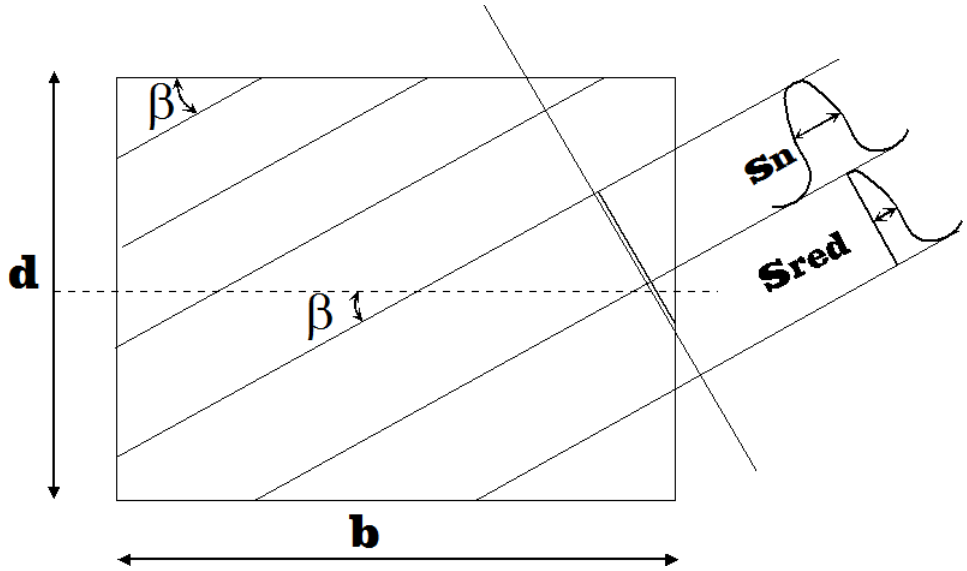


Figure 14.53.3: Illustration of two cuts for a gear with helical teeth

Cpet_border = Cpet*(sred/sn)^0.5

Exponent 0.5 was evaluated in comparative analyses with FEM and LVR. The reciprocal value of this exponent (border weakening factor (buttressing)) can be changed by the user. It has a significant effect on the buttressing effect that occurs in helical gear teeth.

14.10.5 Coupling the individual slices

The teeth are distributed in slices across the width and are coupled together by torsional stiffness.

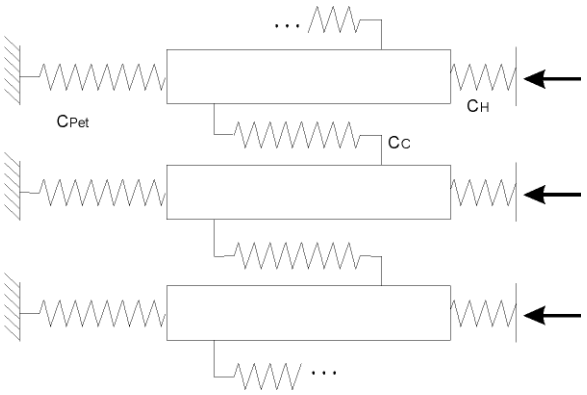


Figure 14.53.2: Linking the slices

Cpet = CZ + CRK = root stiffness as defined by Weber/Banaschek [69]

(CZ = bending stiffness and shearing strength as defined by Weber/Banaschek)

(CRK = deformation stiffness due to rotation in the tooth blank)

CH = stiffness from Hertzian pressure as defined by Weber/Banaschek

CC = coupling with stiffness

CC = 0.04*(Asec)^2*Cpet

Asec: Number of slices

All C are in N/µ/mm.

0.04: Empirical factor, confirmed by comparative calculations with FEM. The user can change this factor (slice coupling factor) in the `module-specific settings`.

(Asec)^2 is used because different numbers of slices must return the same result over the total width.

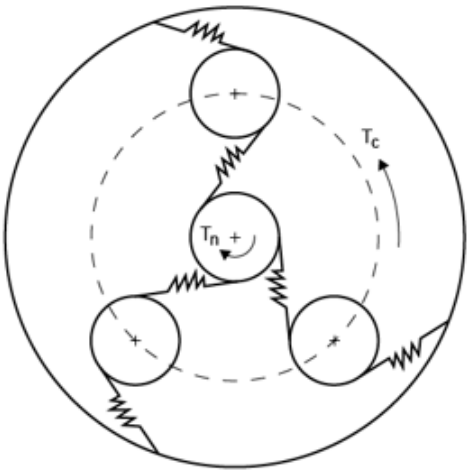
14.10.6 Contact analysis model for planetary systems

Planetary systems up to and including version 03-2014 adopt a cylindrical gear pair approach for the contact analysis. This treats all gear pairs (sun/planet, pla-

net/internal gear) for N planets as cylindrical gear pairs, and involves calculating the sun's resulting torque iteratively until the sun's nominal torque is reached.



The contact analysis for planetary systems in version 03-2015 is calculated using a systemic approach. This involves rotating the planet carrier around a fixed sun and internal gear. Each of the N planets uses the two pair stiffnesses of sun/planet and planet/internal gear to adapt its rotating position and thereby compensate for all torques. This approach also involves an iterative calculation of the system so that the sun's torque corresponds to the nominal torque.



In comparison to version 03-2014, the systemic approach to planetary contact analysis found in version 03-2015 may yield a range of different results, such as those relating to transmission errors and torque curves. This is because the systemic approach significantly improves both the zero-point search (the point of contact for each contact with no transferred forces) and the system iteration.

14.10.7 Base pressure angle of contact analysis

Normally, the contact analysis is performed for a pitch (or for a system period in the case of planetary systems).

For a `cylindrical gear pair`, for a pitch, the base position of meshing *i*, for *N* positions of meshing, is:

$$\varphi_{Ritzel,i} = -\frac{\pi}{z_{Ritzel}} + i\frac{2\pi}{z_{Ritzel} \cdot N}$$

$$\varphi_{Rad,i} = -\pi + \frac{\pi}{z_{Rad}} - \varphi_{Ritzel,i}\frac{z_{Ritzel}}{z_{Rad}} - TE$$

TE = transmission error

For a `planets system`, for a pitch, the base position of meshing (sun and planet carrier) *i*, for *N* positions of meshing, is:

$$\varphi_p = \frac{2\pi\left(z_S + |z_H|\right)}{z_S \cdot |z_H|}$$

$$\varphi_{C,i} = -\frac{\pi}{z_S} + i\frac{\varphi_p}{N} - \frac{\pi}{2} - TE_c\frac{z_S + |z_H|}{z_S}$$

$$\varphi_{S,i} = \varphi_{C,i} + \tan^{-1}\frac{c_y}{c_x} - \varphi_{C,i}\frac{z_S + |z_H|}{z_S} - \frac{s_n}{d_w}$$

Here, the following apply:

p = system period

C = planet carrier

S = sun

H = internal gear

TE_c = transmission error, planet carrier

s_n = tooth thickness, sun

d_w = operating pitch diameter

$c_{x,y}$ = position of the first planet in the Cartesian coordinate system

14.10.8 Wear iteration

You can use the wear iteration function to define wear along the tooth flank in more detail, because it performs several steps of the contact analysis with the worn tooth flank. However, this does significantly increase the time it takes this

calculation to run. Click the "Calculate wear iteratively" checkbox to select this option.

You can input a coefficient for the maximum permitted wear per step. This coefficient is then multiplied by the normal module. In the contact analysis shown below, the service life after one iteration was reduced by only applying the maximum permissible wear. The next contact analysis was then calculated with the worn tooth flank. This process was repeated until the total service life was reached.

By clicking on the sizing button, you can set the maximum permissible wear to half the wear calculated for plastic [delWn] for the entire service life. This should be used to perform roughly ten iterations.

14.10.9 Contact analysis with load spectra

The contact analysis can be performed either with the nominal power or with individual load bins and entire load spectra.

To perform a contact analysis with load spectra, you must select the `Consider load spectrum` option. To take into account individual load bins, you must select the element with the `Consider only one load bin in the load spectra` option in the `Rating` tab.

When load spectra are taken into account, the configuration of the driving wheel, the working flank, and the direction of rotation, change according to the load bin's algebraic sign.

14.11 Gear pump

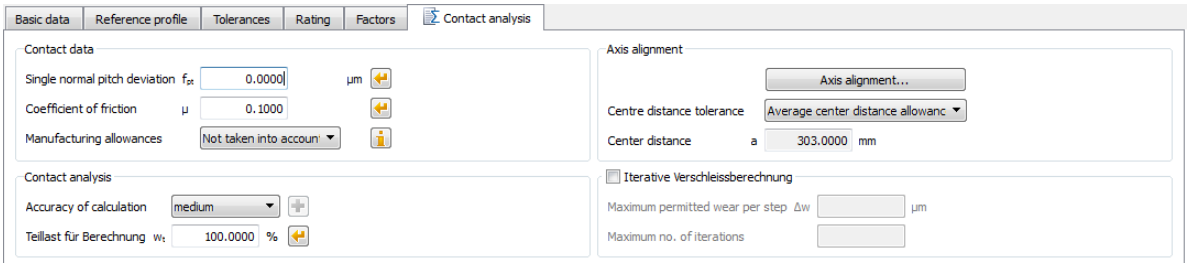


Figure 14.54: `Path of contact` input window for `Gear pump`

If you ignore the return volume, you can calculate the transport volume when you perform the normal calculation. You will find the parameter for this in the `Basic data input window.` (see section "`Basic data`" on page II-264) In this case, click the `Calculation of the displacement volume of gear wheel pumps` checkbox in the `Calculations` tab in the `Settings` window, which you open by clicking the `Calculation` menu.

In the lower part of the `Path of contact` input window you can then perform a detailed calculation for a gear pump.

The system calculates and displays the changes to the critical parameters of a pump that occur during meshing. These include geometric parameters such as the pinched volume (between two meshed tooth pairs, return volume), the volume with a critical inflow area (if possible, the flow of oil should be kept constant), the narrowest point (minimum distance between the first tooth pair without contact), inflow speed, oil inflow at the entry point (with Fourier analysis to evaluate the noise levels), volume under pressure at input. Other important information is the progression of torque on the two gears, the progression of the Hertzian pressure σ_H, the sliding velocity v_g and the wear coefficient $\sigma_H \cdot v_g$. Hertzian flattening can be included when calculating forces because this effect has a significant influence. The pinched volume depends on how the pump construction functions under pressure at input or output. This is defined by the appropriate input value and has a considerable effect on the torque curve. When the pinched volume is reduced, you see a significant momentary increase in compression in this volume. This produces strong pulsing forces on the support and therefore generates noise. A pressure release groove must be installed to avoid this increase in pressure. For this reason, it is very useful to calculate and display the pressure flow in the pinched volume.

Using this calculation, you can analyze any type of cylindrical gear with involute and non-involute teeth forms. At present, the only fundamental restriction is that this procedure is limited to spur gears.

Optimization strategies for gear pumps

The most important and critical problems regarding gear pumps are

- Noise
- Efficiency
- Size
- Wear

Here is some information that may help define the criteria according to which pumps can be evaluated.

- Noise:
  - Variations in flow through the pump generate noise in the pipes. For this reason, the flow (Q) should be as continuous as possible.
  - The enclosed volume (V1) should not be reduced during the generation process. A reduction in this volume would create a massive increase in compression in V1 and generate dynamic forces on both the bearing and the shafts. This effect can be reduced by the precise sizing of relief grooves.
  - The inlet speed of the oil through the narrowest point should be kept as low as possible
- Efficiency:
  - Return volume should be kept as low as possible
- Size:
  - The KISSsoft Fine Sizing functions provide a very efficient method of achieving the highest possible displacement volume for a specified size.
- Wear:
  - You must monitor the course of the wear values (sliding velocity and Hertzian pressure between the tooth flanks)

NOTE:

You will find more detailed information about gear pump analyses in KISSsoft-anl-035-E-GearPumpInstructions.doc [77] (available on request).

- The "Gear pump" report shows the input torque on gear 1 [T] and the torque transferred from gear 1 to gear 2 [T1Contact].

- You should use the torque at the point of contact in the strength calculation and the contact analysis (calculated from P_{out} and P_{in}). Enter this data in the "Basic data" tab.
- You should use the torque at the point of contact in the strength calculation and the contact analysis (calculated from P_{out} and P_{in}). Enter this data in the "Basic data" tab. The total power [P] and the torque [T1] at the pump inlet are only documented in the "Gear pump" report and are not otherwise used. All the graphics shown under "Graphics" "> Gear pump" are based on compression. The torque curve used in the graphic is the input torque [T1].

14.12 Operating backlash

In addition to calculating the theoretical backlash, the backlash after mounting can also be calculated as defined in DIN 3967 (this includes toothing deviations, deviation error of axis according to ISO 10064 or DIN 3964 (see also Table 14.15). The operating backlash (including the temperature differences between the gears and the housing) is also calculated. To calculate the operating backlash, the required input is a temperature range for the gears and the housing, and the maximum and minimum difference in temperature between them. Two cases are calculated simultaneously, one that produces the maximum operating backlash (with the given temperature inputs), and one that produces the minimum operating backlash.

If the module is < 1, the statistically evaluated circumferential backlash is also calculated according to DIN 58405.

The reduction of the backlash due to individual teeth deviations is then calculated with tolerances Fb, Ff and fp according to DIN 3961. These values as specified in DIN 3961 are not defined for module < 1. In this case, tolerances for module 1 are defined according to DIN 3961 and then reduced in proportion to the module. According to this formula: fp(mn) = fp(mn=1.0) * mn.

The reduction in clearance due to deviations in individual teeth is not taken into account for crossed helical gears.

The effect of the runout error can also be taken into consideration. In this case the roller runout tolerance (determined using the approximation formula Fr = Fi" - fi") is used instead of the runout error Fr for module < 1.

Bearing center distance L_G (nominal length) in mm	**Axis alignment accuracy class**											
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**
bis 50	5	6	8	10	12	16	20	25	32	40	50	63
over 50 up to 125	6	8	10	11 2	16	20	25	32	40	50	63	80
over 125 up to 280	8	10	12	16	20	25	32	40	50	63	80	100
over 280 up to 560	10	12	16	20	25	32	40	50	63	80	100	125
over 560 up to 1000	12	16	20	25	32	40	50	63	80	100	125	160
over 1000 up to 1600	16	20	25	32	40	50	63	80	100	125	160	200
over 1600 up to 2500	20	25	32	40	50	63	80	10 0	125	160	200	250

over 2500 up to 3150	25	32	40	50	63	80	100	125	160	200	250	320

Table 14.15: Deviation error of axis according to DIN 3964, values in [mm]

As shown in Table 14.15, the values in the Axis position accuracy and Distance between bearings input fields are used to calculate the axis deviation error according to DIN 3964.

Backlashes are calculated as specified in DIN 3967.

Circumferential backlash calculation:

The circumferential backlash is calculated according to DIN 3967 with the following formula on the reference circle:

$$j_t = (-A_s / \cos\ \beta) + 2 \cdot A_a \cdot \tan\ \alpha_t$$

In KISSsoft, the operating backslash is calculated using the more precise formula in the operating pitch circle:

$$j_t = (-A_s / \cos\ \beta \cdot \frac{\cos\ \alpha_t}{\cos\ \alpha_{wt}}) + 2 \cdot A_a \cdot \tan\ \alpha_{wt}$$

Planetary gear units are handled differently in the operating backslash calculation Here, there are 2 operating pitch diameters for the planets (sun/planet and planet/internal gear). The change in operating pitch diameter due to thermal expansion is defined here for the operating pitch circle determined in this process.

In addition, the change in tip clearance due to thermal expansion (and water absorption for plastics) is also calculated.

Any expansion that occur in the body of the gear also changes its pitch. A single pitch deviation occurs as soon as both gears show unequal expansion. The increase or decrease in pitch caused by thermal expansion is defined as follows:

$$\Delta pt_1 = pt_1 \cdot \alpha_1 \cdot \left(\Theta_1 - \Theta_{ref}\right)$$

$$\Delta pt_2 = pt_2 \cdot \alpha_2 \cdot \left(\Theta_2 - \Theta_{ref}\right)$$

$$fpt = \Delta pt_1 - \Delta pt_2$$

pt pitch

a coefficient of thermal expansion

Q temperatures

fpt single pitch deviation

Plastics also undergo expansion due to water absorption.

14.12.1 Temperatures

The `Reference temperature`T is the ambient temperature specified for manufacturing. The tooth thickness input here apply to this temperature.

The `Mass temperature` for the individual gears defines the thermal expansion of the individual gears, The Gear mass temperature of the scuffing calculation can be used as here as a starting point.

Taken together with the coefficient of thermal expansion, the `Temperature of housing` then defines the amount of thermal expansion that occurs for the housing.

14.12.2 Relative water absorption during swelling

You must input this value as a [%] of the volume. To calculate clearance, DIN 3967 specifies that: For plastics, the linear expansion due to water absorption detailed in DIN 3967 is approximately 1/3 of the amount of water absorbed. However, for fiber-reinforced plastics, it is only around 1/12 of the volume of water absorbed. If you click this checkbox, this phenomenon is taken into consideration when calculating the change in volume.

14.12.3 Coefficient of thermal expansion for housing

If you select a material from the database, this field merely provides information about the coefficient of expansion of the selected housing material. In this case,

you cannot change the value. However, if you have set the `Housing material` drop-down list to `Own input`, you can enter your own value.

14.12.4 Taking into account the bending of the shafts and width modifications

To enable you to use this option, the load distribution calculation (KHβ) according to ISO 6336-1, Appendix E, must be enabled (it is used to calculate the shaft bending). It then determines the place with the lowest backlash change Δjt.i across the facewidth. (This place is documented in the "Face load factor" report). For load spectra, the lowest value found in all bins is determined.
If Δjt.i is negative, the operating clearance is reduced. This therefore changes the minimum operating clearance. (The maximum operating clearance remains unchanged, as it represents the load-free state.) If Δjt.i is positive, the operating clearance increases. This therefore changes the maximum operating clearance. (The minimum operating clearance remains unchanged.)
To determine the backlash change caused by bending, only the components in the axial plane, including the component of the tooth trace modification in the peripheral direction, are taken into account. The bending component normal to the axial plane is not considered, as the flanks lie above the entire facewidth, under load (if KHβ < 2), and therefore do not cause any backlash change.

14.12.5 Tooth deformation

The tooth deformation is only taken into account if the line load w>=100 N/mm (otherwise the calculation of the bending according to ISO 6336 is too inaccurate). The tooth deformation is only taken into account in the case of the minimal operating clearance. (The maximum operating clearance remains unchanged, as it represents the load-free state.) It is questionable whether taking the tooth deformation into consideration is sensible. The calculation of the bending is only approximate and can result in the combined result being too conservative.

14.13 Master gear

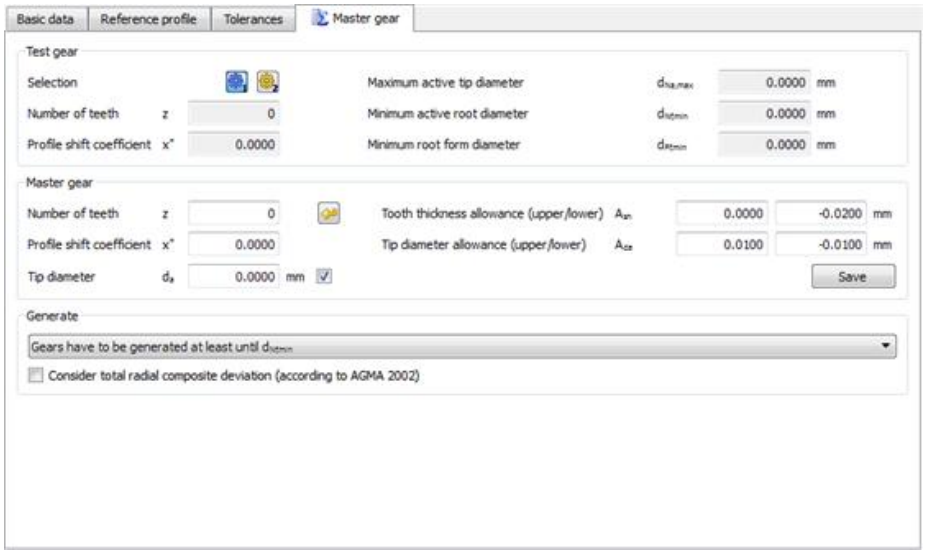


Figure 14.56: `Master gear` input window

This KISSsoft calculation module has been designed to enable you to size and check master gears.

To perform a test for double flank composite transmission, you require one master gear which is then rotated on a test device together with the gear you want to test. In the test run, the test gear and the master gear are pressed lightly together so that no backlash is generated. The deviation in center distances is then measured carefully. The difference between the minimum and maximum value calculated here is the tooth-to-tooth composite error. The active involute of the test gear should be processed as completely as possible during the test run, to ensure you can accurately assess how the test gear will behave when running, after it has been installed in the gearbox. However, it is essential that you prevent the master gear from meshing too deeply in the root area: If the value for the root form diameter of the test gear is not achieved, this will cause meshing interference which will in turn generate measurement results that are massively incorrect. You can call the master gear sizing function for each gear in a particular calculation. When you open the sizing window, the default values for a suitable standard master gear taken from DIN 3970 are displayed. The analysis functions check the maximum and minimum tolerance fields of the tooth thickness of the test gear whose involute is being processed. The report then show which area of the active involute has been tested, or not tested. If the value for the root form diameter is not achieved, the program issues a warning to prompt you to reduce the tip circle diameter of the master gear. This calculation is also available for cylindrical gears with a minimum number of

teeth greater than 4. Click the Save button to save the master gear data and the master gear/test gear pair as KISSsoft files.

Take into account total radial composite (according to AGMA 2002): When calculating the smallest test center distance [aMin], the theoretical center distance stated in AGMA 2002 (equation 8.5) is further reduced by the total radial composite (Vcq specified in AGMA 2000). If the manufacturing tolerances specified in ISO or DIN are being applied, Fi" is used for that purpose. If the tolerances specified in AGMA are applied, Vcq is used here:

14.14 AGMA 925

As specified in AGMA 925, you can use this input window to define the probability of scuffing and wear as well as susceptibility to micropitting (frosting).

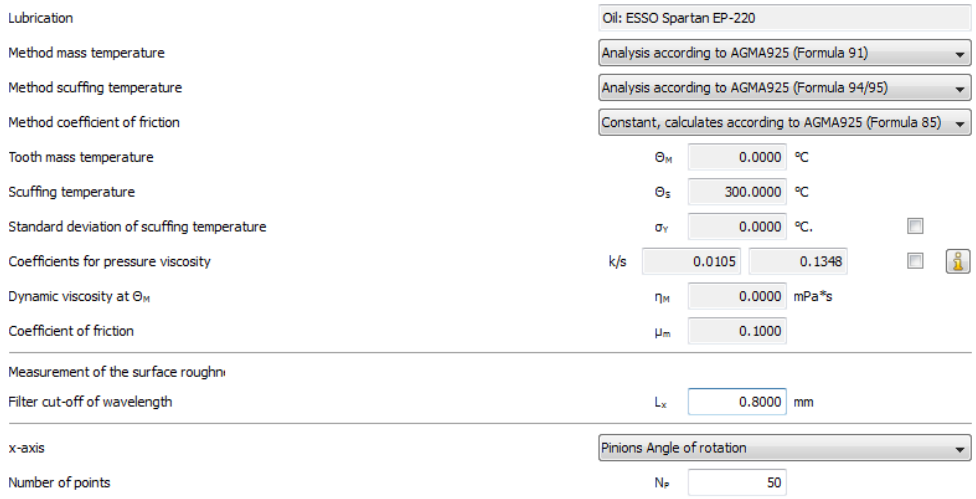


Figure 14.57: AGMA 925 input window

AGMA 925-A03 *Effect of Lubrication on Gear Surface Distress* calculates the conditions in the lubrication gap across the gear meshing. AGMA 925 defines how to calculate the lubrication gap height while taking into account the flank deformation, lubricant properties, sliding velocity, and local Hertzian stress. The standard then uses this base data to calculate the probability of wear. The wear is caused by the metal surfaces contacting each other if the lubrication gap is too narrow. The probability of wear calculated by the standard is greater than the values that occur in practice.

The standard does not give any indications about safety against micropitting (frosting). However, data provided by the relevant technical literature, and the results of research, reveal that there is a direct correlation between the minimum lubrication gap-to-surface roughness ratio and the occurrence of micropitting (frosting). You can therefore use this calculation method to optimize gear toothing for micropitting (frosting). AGMA 925 also includes a definition of the probability of scuffing. This analysis uses the same base data (Blok's equations) as the calculation of scuffing according to the flash temperature criteria given in DIN 3990, Part 4. However, defining the permitted scuffing temperature according to AGMA 925 presents more of a problem, because of the lack of comprehensive or generally applicable information. In particular, there is no reference to a scuffing

load capacity specification as given in the FZG test. There is therefore a tendency to underevaluate oils that have effective EP additives.

The values for the compression viscosity coefficient α of typical gear oils vary between 0.00725mm^2/N to 0.029mm^2/N, and are defined as follows in AGMA 925-A03:

$$\alpha = k \cdot \eta_M^5 \tag{14.25}$$

where

α	Compression viscosity coefficient	mm^2/N
k	see Table 2 in AGMA 925-A03	-
η_M	Dynamic viscosity for tooth temperature θ_M	mPa . s

In practice, calculating wear in accordance with Wellauer results in risk of wear values that are too high. For this reason, the analysis is performed according to Dowson (as in Annex E of AGMA 925). The report shows the results for both methods.

14.15 Root stress FEM

This special calculation is used to derive the stresses at the root of a gear tooth, using the FEM method. The user must select the gear pair and the specific gear that should be analyzed. The fixing area of the gear tooth (necessary boundary conditions for the FEM analysis) can be either the inner / outer diameter of the gear (for internal / external gears respectively), or the sides of the gear segment that is analyzed. Mesh refinement can be also defined, but we recommend you keep the highest possible value, since the accuracy of the stress calculation is strongly dependent on the mesh fineness.

During the calculation, the exact geometry of the gear is extracted and a segment of it is kept for the FEM mesh generation. The automatic mesh generation creates a finer mesh at the gear root area. You can find FEM mesh statistics in the final report. The load is calculated and applied at the point of single tooth contact, at the working angle.

In the case of helical gears, the equivalent spur gear is calculated and used, according to ISO 6336-3. You can save the data for the equivalent spur gear after the FEM calculation by selecting the relevant option. However, note that, if you save the equivalent spur gear data, it will overwrite the original data for the gear.

Finally, there is the option to automatically start the FEM pre- and post-processor used by KISSsoft, at the end of the calculation. This can be useful if you want to see the FEM mesh and/or the stresses on the gear. Special basic instructions on using this FEM tool are available from KISSsoft support.

14.16 Rough sizing

KISSsoft has exceptionally powerful sizing functions, which are described in this and the following sections.
The process for sizing a gear stage, from start to end, involves rough sizing, followed by fine sizing and finally, sizing the modifications.

Layout process of a gear set

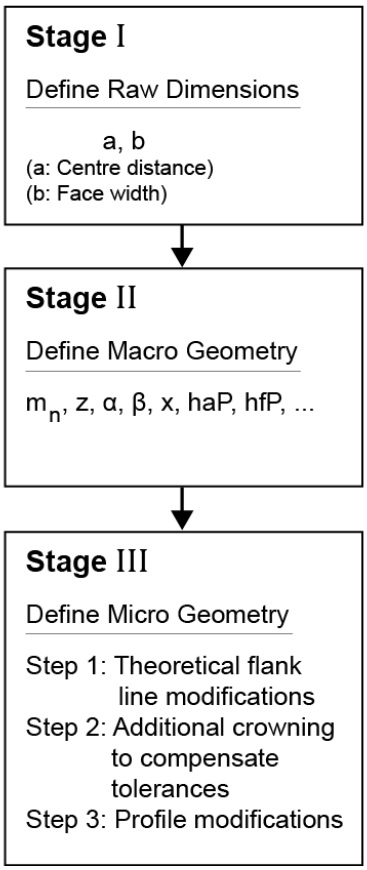


Figure 14.: Phases involved in sizing gears

Rough sizing suggests possible toothing configurations based on the data entered for the ratio and the load. The purpose of rough sizing is to ascertain the possible range of suitable solutions, all sized for the specified torque, according to all the specified required safeties. The total weight is possibly the most important output, because this can be regarded as roughly proportional to the manufacturing cost. The weight of the different solutions usually varies by a factor of up to 3!

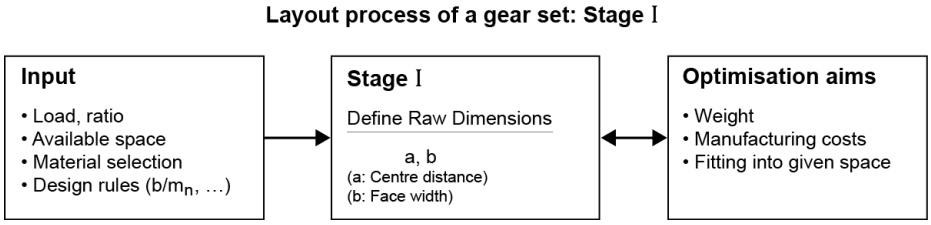


Figure 14: Gear sizing, Phase 1

To call the rough sizing function, either go to the `Calculation` menu and select the `Rough sizing` option or click on the  icon in the tool bar.

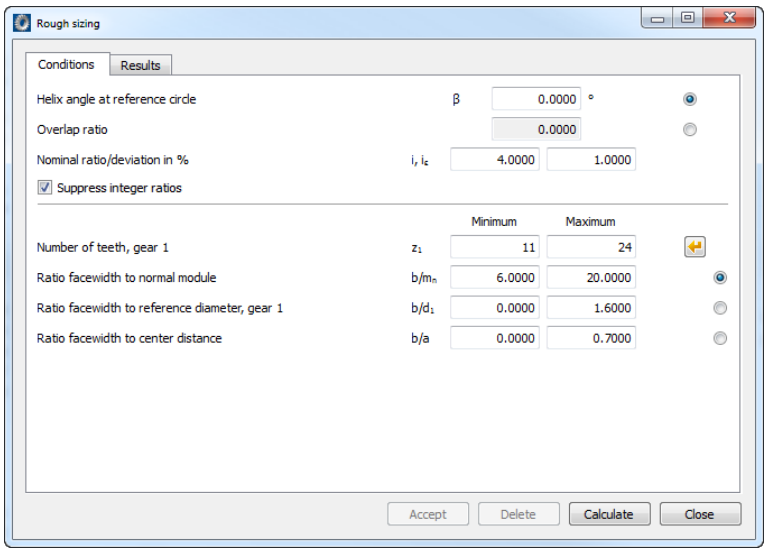


Figure 14: Dialog window: Rough sizing

At present, you can apply this to cylindrical gear pairs with internal or external teeth, and to planetary gears. The nominal ratio is the most important input parameter. For an internal gear pair, the ratio must be entered as a negative value in the `Geometry` area. In planetary stages, the nominal ratio must be > 2.0.

The operating data (power, speed, etc.) is taken from the KISSsoft main window (and can be changed there if required). You can also specify a helix angle or a required overlap ratio (e.g. ε_β= 1.0).

Some important design parameters for gear stages can be preset (the ratios b/mn, b/d1, b/a). Because these parameters may limit each other, you can click the appropriate button to specify which parameter is preferred.

Click on the `Calculate` button to open a list of suggestions that you can use to set the parameters for your gears.

The parameters in the results table are displayed with formula symbols which match the formula symbols used in the rest of the interface, and in the reports. Hover the mouse pointer over a formula symbol in the table to display a description of it in plain text. Right-click on the results table to open a dialog, in which you can either hide or display additional parameters.

Rough sizing automatically finds the most important tooth parameters (center distance, module, number of teeth, width) for the required power and ratio, using the strength calculation according to the selected calculation standard. Dimensioning is performed according to minimum safeties (Required safeties (see page II-471)).

You can specify the intervals for the relationships b/m_n-, b/a, b/d in the `Calculation` menu under `Settings > Sizings`. (Sizings (see page II-464))

The program displays a number of different solutions which you can select. You can then use them to perform an optimization in fine sizing. The window remains open, to enable you to select more solutions. You will find more detailed information about fine sizing in section 14.15.

The most important result of this sizing process is that it enables you to define the achievable center distance ranges and module ranges, as well as the facewidth. You can then decide how much space is required for the gearbox itself.

Solutions with a number between 1 and 5 show solutions with any modules. Solutions from 6 onwards show solutions with standardized modules according to DIN 780 (module series for gears).

Number 1: Solution with the most exact ratio

Number 2: Solution with the greatest center distance

Number 3: Solution with the smallest center distance

Number 4: Solution with the largest module

Number 5: Solution with the smallest module

You can fix the center distance for special cases. However, in these cases, you must remember that the program's sizing options are not exhaustive, and fine sizing represents a better alternative.

Sizing of strength for a planetary gear

When performing rough sizing for planetary stages, it is assumed that the rim is static. If the rim rotates, you must change the speed after sizing.

The system prompt suggests the number of teeth as defined by Niemann
Table of standard number of pinion teeth according to Niemann [65], Table 22.1/8.

Ratio *u*	**1**	**2**	**4**	**8**
Through hardened or hardened				
Counter-through hardened to 230 HB	32..60	29..55	25..50	22..45
Over 300 HB	30..50	27..45	23..40	20..35
Cast iron	26..45	23..40	21..35	18..30
Nitrided	24..40	21..35	19..31	16..26
case-hardened	21..32	19..29	16..25	14..22

Click the Sizing button to transfer these values to fine sizing automatically.

14.17 Fine sizing

The fine sizing function is one of KISSsoft's most powerful tools. It generates and displays all the possible geometry variants (module, number of teeth, etc.) for the specified facewidth and center distance (the gear rim diameter is usually specified for planetary stages and the center distance varied accordingly). The solutions are displayed as graphics, so you can easily see the best possible macrogeometric variant for your purpose.

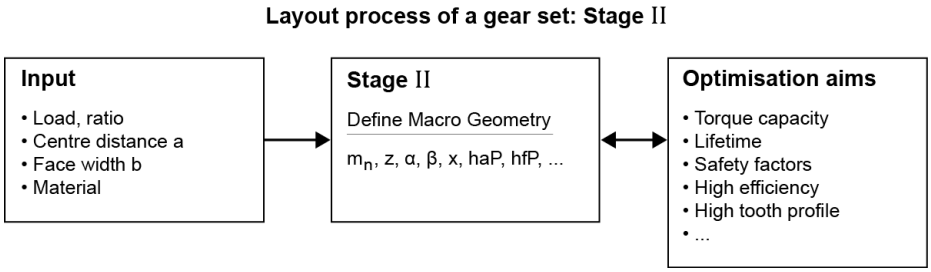


Figure 14: `Gear sizing, Phase II`

To call the fine sizing function, either go to the `Calculation` menu and select the `Fine sizing` option or click on the  icon in the tool bar.

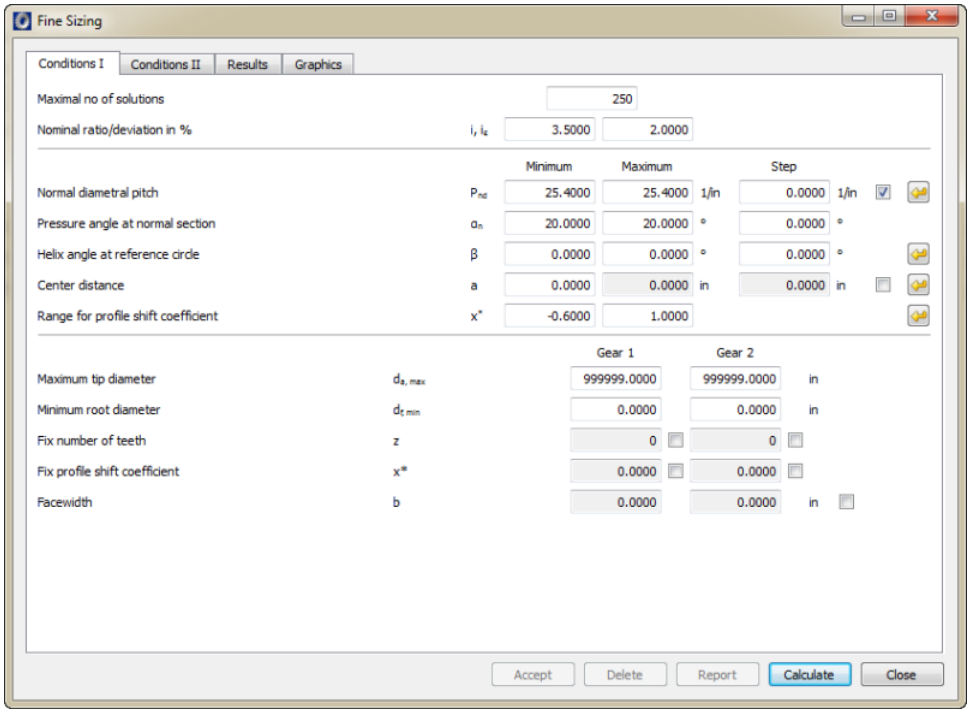


Figure 14: `Conditions I` tab in the `Fine Sizing` window

If you input a nominal ratio, a center distance, and intervals for the module and helix angle, as well as the pressure angle, KISSsoft calculates and displays suggestions for the number of teeth, module, helix angle and profile shift. It also shows the deviation from the nominal ratio, the specific sliding and the contact ratio. This module can also be used to size planetary stages and three gears trains.

All the variants found by this process can be evaluated by a wide range of different criteria (accuracy of ratio, weight, strength, tooth contact stiffness deviation etc.)

Depending on your requirements, limits can also be set on the most important parameters (tip circle, root circle, minimum number of teeth, tolerated undercut etc.). In addition to creating text reports detailing the solutions and the summary, the summary can also be displayed as a graphic.

The facewidth appears in the input screen, where you can modify it if required.

14.17.1 Necessary entries in the input window

Before you start the fine sizing process, you must enter the following data correctly in the `Basic data` or `Geometry` and `Strength` standard tabs to ensure the calculation returns the results you require.

Geometry:

- Reference profile
- Number of idler gears/planets (in a 3-gear configuration)

Strength:

- Materials
- Power/Speed
- Application factor
- Service life
- Lubrication

14.17.2 Conditions I

You can predefine the module range for cylindrical gears. If the module flag is set, you can predefine the increments. If the module flag is not set, you can only use modules from the standard module list.

For cylindrical gear pairs, you can either input a fixed center distance (the usual

approach) or specify an interval for the center distance. To do this, click the check-box to the right of the Center distance input fields.

If planetary gear units are involved, you can either perform the calculation with a predefined center distance or with a predefined V-circle (dp = d+2*x*mn) for the internal gear. In practice, it is usually the internal gear diameter that is fixed (gear size remains the same) and the center distance that is varied. In this case, we recommend you first input the required output reduction and the V-circle, then click the sizing button for the center distance.

Note:
You should check the center distance interval after you change the reference circle or select a variable center distance. You may then need to repeat the sizing process.

14.17.2.1 Limiting the tip diameter

Solutions whose tip circle exceeds the specified value are rejected. If you do not want to limit the tip value, you can input either 0 or 10^{10}.

However, the following problem prevents this option being used sensibly in practice: If a gear is to be installed in an existing housing, it is critical that it does not touch the walls of the housing.

14.17.2.2 Limiting the root diameter

Solutions whose root circle falls below the specified value are rejected. If you do not want to limit the root diameter, you can input 0.

However, the following problem prevents this option being used sensibly in practice: If a gear is mounted on rolling bearings in a speed change gear unit, you must guarantee a minimum thickness of material between the bore and the root circle.

14.17.2.3 Maximal no of solutions

Proposal: 50 ...250

If the program finds more than the specified number of solutions, you see a warning message and an appropriate note is entered in the report.

NOTE

You should only perform a final evaluation after all the possible solutions have been displayed. Otherwise, you run the risk that the optimum solution will not be displayed.

14.17.2.4 Limiting the number of teeth

You should not normally need to use this option, which is why the default setting is for it to be disabled. However, by clicking the individual checkboxes, you can still fix this parameter. A useful application for this option is when for sizing a planetary gear which has already been modified to fit inside a predefined internal gear. In this case, the module, the number of teeth and the profile shift are predefined for gear 3.

14.17.3 Conditions II

Here, the reference profile h^*_{aP} of the individual gears can be varied step-by-step. The dedendum h^*_{fP} is determined via the required tip clearance to the counter gear ($h^*_{fP}2$ - H^*_{aP} 1). If this value is not changed, the tip clearance value in every variant will be the same as the value entered in the Basic tab.
You can also specify that the maximum possible tip rounding radius, ρ^*_{fP}, is always set automatically.

14.17.4 Conditions II

You can specify other essential functions in the `Conditions III` tab.

1. **Show values of KISSsoft Basic Tab as additional variant with number 0**
   The toothing data in the KISSsoft Basic tab can also be displayed as a variant with the number 0 (table and graphic). However, **the data at the start of the fine sizing process must be consistent before this can happen**.
   This option can either be enabled or disabled. When you enable this option, you must restart the fine sizing process so the variant can also be displayed.
2. **Calculate geometry only**
   If you select this method, no strength calculation is performed.
3. **Strength calculation with load spectrum**

   Before you can perform calculations with a load spectrum, you must specify a load spectrum in the KISSsoft main window before you start the fine sizing process and run the calculation (to ensure the data is consistent). In this case, when you start the fine sizing process, you are prompted to confirm that you want to perform the calculation with a load spectrum. The flag in the window merely shows whether the calculation includes a load spectrum. You cannot reset this flag.
4. **Permit undercut**
   If this option is selected, solutions with undercut are not rejected.

5. **Reject results with specific sliding higher than 3**
   Usually specific sliding should not be greater than 3.

6. **Consider minimum tooth thickness**
   If this option is selected, solutions with a tooth tip thickness that is less than the predefined minimum tooth thickness (see `Calculation > Settings > General`) will be rejected.

7. **Allow small geometry errors**
   Minor meshing interference and similar geometry errors will now be tolerated when the system calculates variants! You can make separate settings to take into account the undercut and the minimum tooth thickness at the tip (see points 2 and 4).

   You must set this option if the program has to find solutions where the number of teeth is less than 7, or in other exceptional situations. We do not recommend you set this option in any other situation!

   Note:
   In these situations, you must also change the minimum number of teeth accordingly (see point 11).

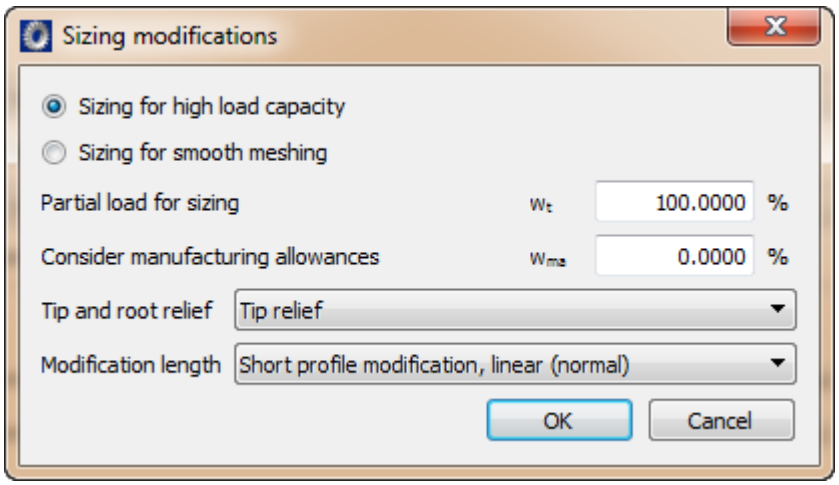
8. **Suppress integer ratios**
   If this option is selected, results with whole number gear ratios will be rejected.

9. **List of cutters for reference profile**
   Instead of using the predefined reference profile, you can use a list of hobbing cutters for fine sizing. In this case, the calculation is performed for every cutter in the given module and pressure angle range and the tool is displayed in the results list.
   The same hobbing cutter is used for each gear. Internal teeth are not affected by this setting.

10. **Sizing of deep tooth forms**
    Special reference profiles with larger addendums and dedendums are used for deep toothing. This sizing function εα=εαtarget calculates the necessary reference profile on the basis of the required transverse contact ratio εαtarget. If this function is active in fine sizing, the reference profile for every solution is calculated, so that the exact required transverse contact ratio is achieved. As a result, only solutions that have exactly the required transverse contact ratio are displayed.
    However, the εα>= εαtarget function only changes the transverse contact ratio when the transverse contact ratio original reference profile results in a transverse contact ratio that is smaller than εαtarget.

Note:
In both cases (εα= εαtarget and εα>= εαtarget), you must ensure that automatic tip alteration k*m_n is not executed (and is set to zero). Both the reference profile h*$_{aP}$ value and the tip alteration k*m_n have the same effect on the tip circle, which is why only one of these two values should be changed.

11. **Transmission error**
If the "Calculation of transmission error" option is selected, the contact analysis is performed for every variant. If the "Calculation of transmission error and profile correction" option is selected, the length and value of the profile modification (correction) is determined automatically, according to the settings made for the correction method. Click the  button to open the profile modification settings window.



The correction method takes into account the objective (for high load capacity gears or smooth meshing), tip and/or root relief, length (short or long), and the type (linear, arc, progressive, and linear with transition radius). It is important to note that the transmission error can only be minimized for one load, and the partial load for sizing should be set correctly according to the applied load.

When contact analysis is performed for transmission error, the default settings are used to prevent the calculation returning unusual results, apart from the coefficient of friction and accuracy of calculation. Input the settings in the main program, in the "Contact analysis" tab. You can also specify the accuracy of the calculation, however, we strongly recommend

you use "low" or "medium" to reduce the processing time. Therefore, the transmission error in fine sizing may not be exactly the same as the one from the contact analysis, according to the settings.
The default settings are:

Calculation for: right flank
- Torque for gear A: not considered
- Torque for gear B: not considered
- Partial load for calculation: 100 %
- Center distance: Average center distance allowance
- Single pitch deviation: 0 mm
- Deviation error of axis: 0 mm
- Inclination error of axis: 0 mm

The results list shows:
- Transmission error (PPTE)
- Average wear on the tooth flank (delwn1, delwn2
- Maximum flash temperature (theflamax)
- Variation in bearing force (VarL)

The calculation time increases significantly if the transmission error calculation option is used. We therefore recommend you limit the number of results before starting the calculation.

12. **Suspend results which do not meet required safety factors**
    Variants which do not meet the predefined minimum safety levels (see `Calculation > Settings > Required safeties`) will be rejected.
    Note:
    Variants with insufficient safety against scuffing will not be rejected.
13. **Sizing of profile shift coefficient x1**
    Fine sizing usually generates 3 or 4 variants, in which only the profile shift differs. In this case, the profile shift x1 is changed in increments of 0.1. Here you can specify the criterion used to determine the largest profile shift used, x1.
14. **Minimum number of teeth** z_{min}
    Practical values range for the minimum number of teeth:

    For helical gear teeth: 7 to 9
    For spur gear teeth: 10 to 12

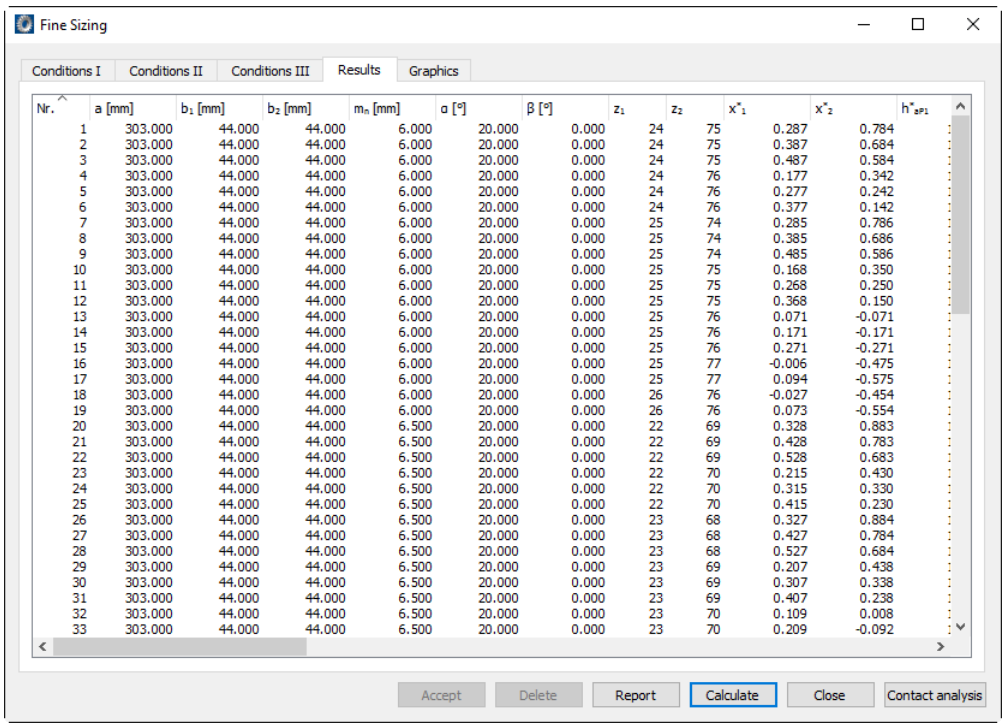
Click the  button to display a suggested value for the minimum number of teeth.

Note:
If you want to find solutions in which the number of teeth is less than 7, you must first select the `Allow small geometry errors` option.

15. **Minimum distance between root form diameter and active root diameter** d_{Nf} - d_{Ff}
Meshing interference occurs if the active root diameter is less than the root form diameter. Here you can specify a minimum value for the distance between the active root diameter and the root form diameter, i.e. between active and manufactured involutes. The input value is the minimum difference between the two diameters.

16. **Minimum distance between root form diameter and base circle** d_{Ff} - d_b
If the start of the manufactured involute is closer to the base circle, this will cause greater wear on a tool during the manufacturing process. Here you can specify a minimum value for the distance between the root form diameter and the base circle. The input value is the minimum difference between the two diameters.

14.17.5 Results



Nr.	a [mm]	b1 [mm]	b2 [mm]	mn [mm]	α [°]	β [°]	z1	z2	x*1	x*2	h*aP1
1	303.000	44.000	44.000	6.000	20.000	0.000	24	75	0.287	0.784	
2	303.000	44.000	44.000	6.000	20.000	0.000	24	75	0.387	0.684	
3	303.000	44.000	44.000	6.000	20.000	0.000	24	75	0.487	0.584	
4	303.000	44.000	44.000	6.000	20.000	0.000	24	76	0.177	0.342	
5	303.000	44.000	44.000	6.000	20.000	0.000	24	76	0.277	0.242	
6	303.000	44.000	44.000	6.000	20.000	0.000	24	76	0.377	0.142	
7	303.000	44.000	44.000	6.000	20.000	0.000	25	74	0.285	0.786	
8	303.000	44.000	44.000	6.000	20.000	0.000	25	74	0.385	0.686	
9	303.000	44.000	44.000	6.000	20.000	0.000	25	74	0.485	0.586	
10	303.000	44.000	44.000	6.000	20.000	0.000	25	75	0.168	0.350	
11	303.000	44.000	44.000	6.000	20.000	0.000	25	75	0.268	0.250	
12	303.000	44.000	44.000	6.000	20.000	0.000	25	75	0.368	0.150	
13	303.000	44.000	44.000	6.000	20.000	0.000	25	76	0.071	-0.071	
14	303.000	44.000	44.000	6.000	20.000	0.000	25	76	0.171	-0.171	
15	303.000	44.000	44.000	6.000	20.000	0.000	25	76	0.271	-0.271	
16	303.000	44.000	44.000	6.000	20.000	0.000	25	77	-0.006	-0.475	
17	303.000	44.000	44.000	6.000	20.000	0.000	25	77	0.094	-0.575	
18	303.000	44.000	44.000	6.000	20.000	0.000	26	76	-0.027	-0.454	
19	303.000	44.000	44.000	6.000	20.000	0.000	26	76	0.073	-0.554	
20	303.000	44.000	44.000	6.500	20.000	0.000	22	69	0.328	0.883	
21	303.000	44.000	44.000	6.500	20.000	0.000	22	69	0.428	0.783	
22	303.000	44.000	44.000	6.500	20.000	0.000	22	69	0.528	0.683	
23	303.000	44.000	44.000	6.500	20.000	0.000	22	70	0.215	0.430	
24	303.000	44.000	44.000	6.500	20.000	0.000	22	70	0.315	0.330	
25	303.000	44.000	44.000	6.500	20.000	0.000	22	70	0.415	0.230	
26	303.000	44.000	44.000	6.500	20.000	0.000	23	68	0.327	0.884	
27	303.000	44.000	44.000	6.500	20.000	0.000	23	68	0.427	0.784	
28	303.000	44.000	44.000	6.500	20.000	0.000	23	68	0.527	0.684	
29	303.000	44.000	44.000	6.500	20.000	0.000	23	69	0.207	0.438	
30	303.000	44.000	44.000	6.500	20.000	0.000	23	69	0.307	0.338	
31	303.000	44.000	44.000	6.500	20.000	0.000	23	69	0.407	0.238	
32	303.000	44.000	44.000	6.500	20.000	0.000	23	70	0.109	0.008	
33	303.000	44.000	44.000	6.500	20.000	0.000	23	70	0.209	-0.092	

Figure 14.62: `Results` tab in the `Fine Sizing` window

Click the `Report` button to open the editor and display a list of the best results. A brief description of the criteria used to evaluate the best variants is given here. Please note that these criteria are not relevant to every case, and only need to be queried in particular applications!

1. Summarize variants for accuracy of gear ratio:
   The difference between the actual gear ratio and the required gear ratio is evaluated here.
2. Weight: this is an indicator for the manufacturing price
3. Specific sliding: maximum value
4. Sliding velocity: maximum value
5. Relationship AC/AE
   AC: length of path of contact from meshing point to pitch point
   AE: total length of path of contact
   "Pushing" sliding occurs in the AC area of contact (sliding velocity of the driving gear is greater than that of the driven gear). As this area is critical for unlubricated plastic gears, the AC/AE relationship should be as small as possible in this case.
6. Summarize variants for vibrations:
   The variation in the total contact stiffness is evaluated here (the lower the variation the better).
   The calculation is based on empirical formulae, unless the "Calculate mesh stiffness" option is set in "Conditions II.
7. Summarize variants for strength:
   The root and flank safety are evaluated here, relative to the required safety. Although safeties of less than the required safety are given a very negative evaluation, large safety margins above the required safety have very little influence.
8. Transmission error (PPTE)
   Transmission error is displayed if the corresponding option is set in "Conditions II".
9. `Summary` evaluation:
   The `Summary` evaluation weights each component to form a total evaluation coefficient. Set the weighting of individual components in `Calculation > Settings > Summary`. This weighting depends to a great

extent on which solution you require, for example, whether you want a solution that is optimized for noise reduction or strength.

NOTE

The Rough sizing (on page II-435) section includes a complete list of all the available parameters. You will find information about noise optimization in [56].

14.17.6 Graphics

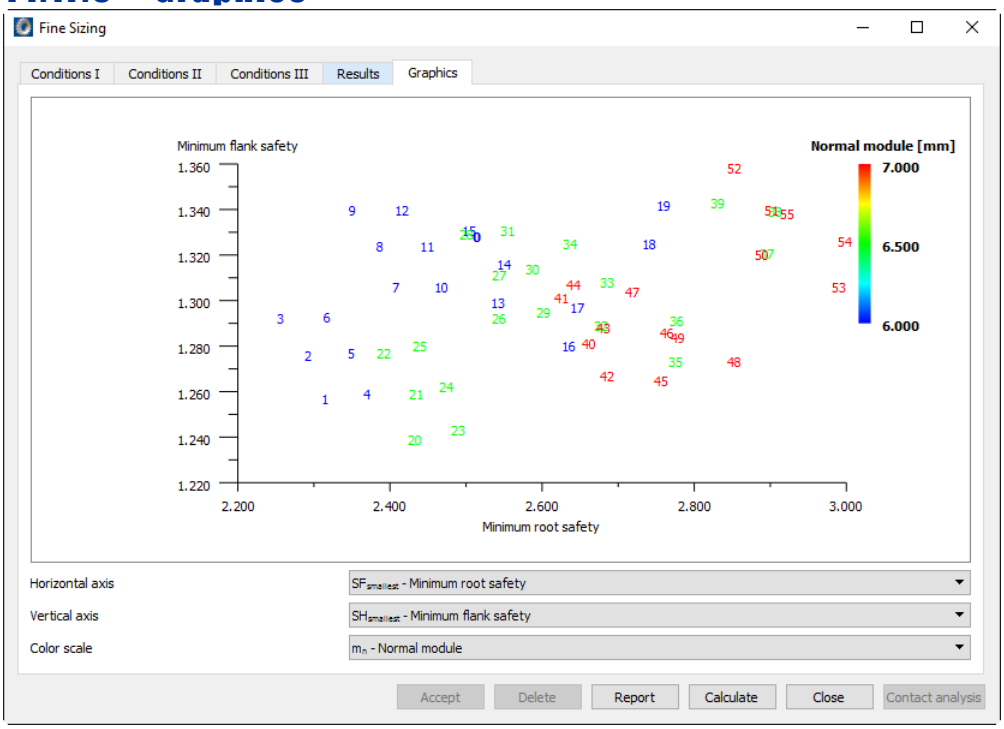
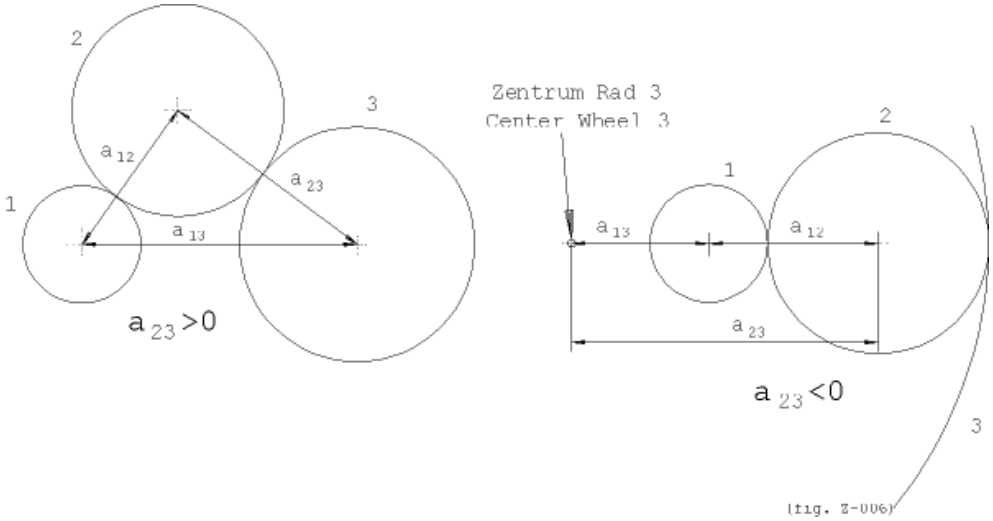


Figure 14.63: `Graphics` tab in the `Fine Sizing` window

The Graphics tab in the Fine Sizing window gives you a quick overview of the number of solutions. Three parameters can be displayed simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

14.17.7 Geometry fine sizing for 3 gears

Definition of center distances:



14.17.8 Additional strength calculation of all variants

The KISSsoft system also calculates the strength (tooth root, flank and scuffing) of each variant of geometry and displays the data as a list. This option can be used for cylindrical gear pairs, planetary stages and cylindrical gear stages that have an idler gear. If you click on the `Calculate geometry only` checkbox in the `Conditions II` tab, the calculation does not include tooth safeties.

14.18 Modifications sizing

Sizing the profile and tooth trace modifications is the last and most complex phase in sizing a gear. This modification variant generator can save you time and effort by calculating the optimal modifications directly.

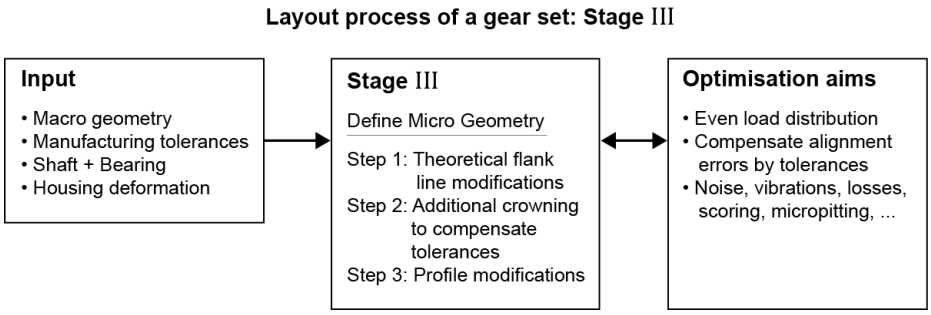


Figure 14.: `Gear sizing, Phase III`

To call the modification sizing function, click the  icon (tool bar) in the `Calculation` menu, and then click `Modification sizing`.

If you call the Optimization functions without opening the `Contact analysis` tab, the default settings in the tab will be used in the calculation.

14.18.1 Conditions I/II

Conditions I

The Conditions I tab is where you define basic modifications that will not be changed and which are valid for every solution. To automatically adopt modifications already present in this tab, click the "Import modifications" button.

Select the "Cross-vary value and factor 1/2" option to run an additional variation of the coefficients with the value of the modification value.

If the "Without contact analysis, only service life calculation with KHbeta according to ISO 6336-1, Annex E" option is enabled, the solution range is only performed using the service life and the calculation of KHbeta.

Every modification can be calculated for a larger partial load area. This can be specified under "Partial load area for the calculation".

Conditions II

The Conditions II tab is where you define the modifications you want to vary. You can enter 20 modifications for each gear, each one with a minimum value and a maximum value. By entering the number of steps per modification you can define

the number of steps between the minimum value and the maximum value. If the "Synchronize with no." column contains a different value than the line number of it's own, the modification is synchronized with the modification you selected, and all the variants are executed with the same number of steps.

14.18.2 Results

All the solutions are displayed as graphics in the "Results" tab. You can then select the solution that best suits your requirements. Click on Accept or double-click on the solution to transfer its data to the "Modifications" tab.

In the results overview, you see the following results for all the iteration steps:

- ID: solution ID. You can use this ID to search for more details about the results in the reports.
- Wt: Partial load of the calculated solution in % (depending on the number of iteration steps specified in the "Number of steps for partial load" field), e.g. 50% partial load relative to the nominal load defined in the "Basic data" tab.
- Hmin: The minimum service life achieved by the gear pair in hours
- PPTE: Amplitude of the transmission error of the driven gear along the path of contact in [µm] or angle of rotation error [°] of the driven gear.
- rel. PPTE: Relative amplitude of the transmission error/angle of rotation error in relation to the uncorrected toothing.
- ε_a: Transverse contact ratio under load
- K_{Hb}: Face load factor (if the calculation is performed with load spectra, only the face load factor of the last load bin is ever displayed)
- σ_{Hmax}: Maximum Hertzian pressure that occurs in the toothing
- Slam: Safety against micropitting as specified in ISO TR 15144
- η: Efficiency
- $\Delta Wn_{A/B}$: Wear on gear A/B
- ΔT: Torque amplitude of the driven gear
- Modifications: You can select options in the context menu to display all the modifications (click the right-hand mouse button in the Results window).

14.18.3 Graphics

All the solutions are displayed as graphics in the "Graphics" tab. You can display a maximum of up to 10 graphics at the same time. Each graphic can process its own data record.

Select the required partial load from the partial load selection list (red is the largest partial load, blue is the smallest partial load).

14.18.4 Report

The results are documented in three different, detailed reports. We suggest you begin by looking at the summary report which gives a broad overview. The other two types of report are considerably longer, and also document intermediate results.

The main calculation performs a series of contact analysis calculations. Each one has a different combination of modifications with all the intermediate steps, and for each wt% level. A contact analysis without modifications is also performed, to provide a basis for comparison.

A frequently asked question:

How can I use the "Optimize modifications" function to vary the length of the modification and the relief Ca independently of each other to find out which combination of length/value gives the best result?

Reply: For example, if you want to vary the tip relief Ca between100 and 220 mm, and vary the length factor between 0.78 and 1.56, to determine all the possible combinations of value and length.

14.19 Measurement grid

A measurement grid report is available for cylindrical and bevel gears (Calculations > Measurement grid). This report is not available for face gears and globoid worm gears.

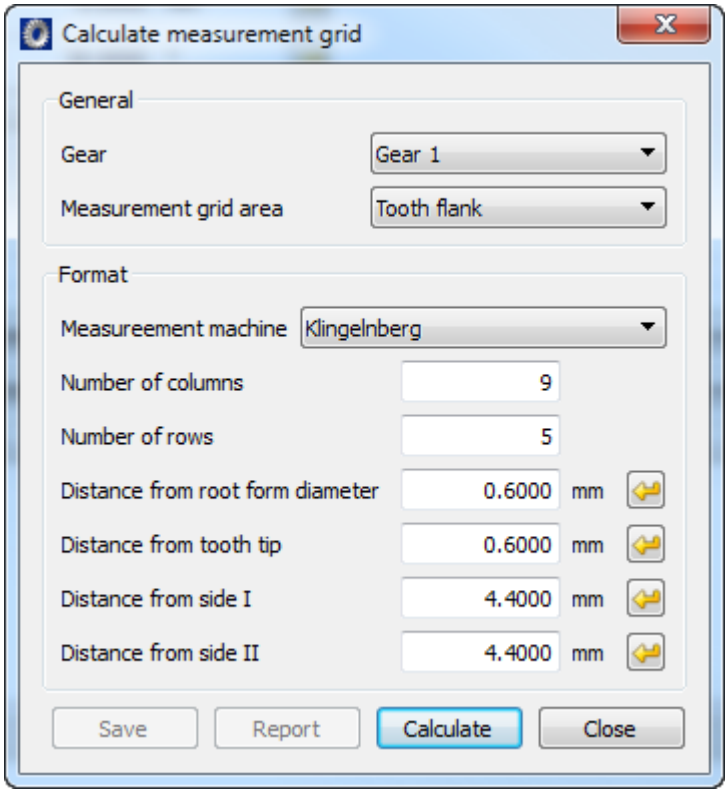


Figure 14.61: Calculating the measurement grid

Setting	Description
Gear	Setting the gear for calculating the measurement grid. If you select the "All" setting, the measurement grid will be calculated for every gear.
Measurement array	Setting the measurement array for the calculation. 0: Tooth flank 1: Root radius
Measurement machine	Setting the report format using a particular measurement machine 0: Klingelnberg

	1: Gleason
Number of columns	Setting the number of columns across the facewidth (>=3) Number of columns (number of sections – 2) for Parasolid settings, because the sections should not include both ends of a tooth.
Number of rows	Setting the number of rows across the tooth profile (>=3)
Distance from root form diameter	Distance from root form diameter. Default value 0.1* mean normal module.
Distance from tooth tip	Distance from tooth tip. Default value 0.1* mean normal module.
Distance from side I/toe	Distance from side I for cylindrical gears, distance from toe for bevel gears. Default value is (facewidth)/(number of columns + 1).
Distance from side II/heel	Distance from side II for cylindrical gears, distance from heel for bevel gears. Default value is (facewidth)/(number of columns + 1).

The report includes the coordinates and the normal vector of the grid points in the format [XP YP ZP XN YN ZN]. The reference point and the tooth thickness angle are displayed in the report header.

The reference coordinates of the data may differ according to which type of measuring machine is used. For example, the following convention applies to Klingelnberg machines.

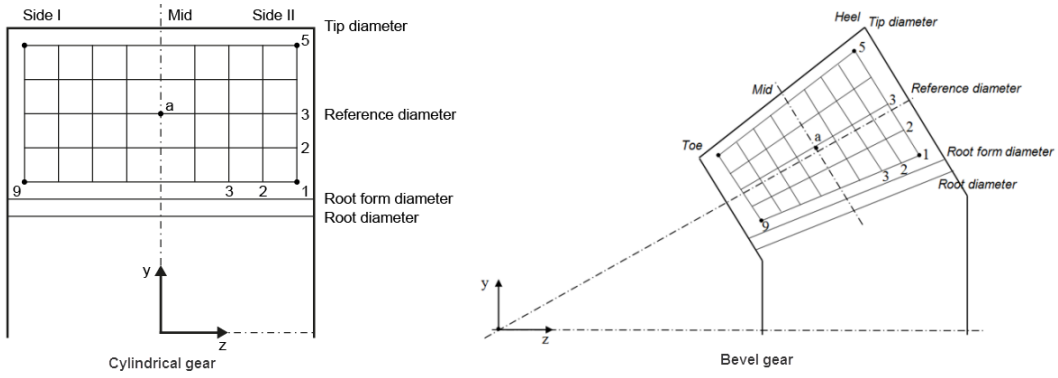


Figure 14.62: Measurement grid for cylindrical gears and bevel gears for Klingelnberg machines

The sequence of index numbers for points and sections is defined according to ISO/TR 10064-6. In other words, the index for lines runs from bottom to top, and the index for columns runs from side II (heel) to side I (toe).

14.20 Settings

You access the Module-specific settings window by opening the Calculation menu and then clicking on Settings. A huge number of these settings are available for cylindrical gear calculations. You can therefore enable the widest variety of possible special functions. Normally there is no need to change the settings.

14.20.1 General

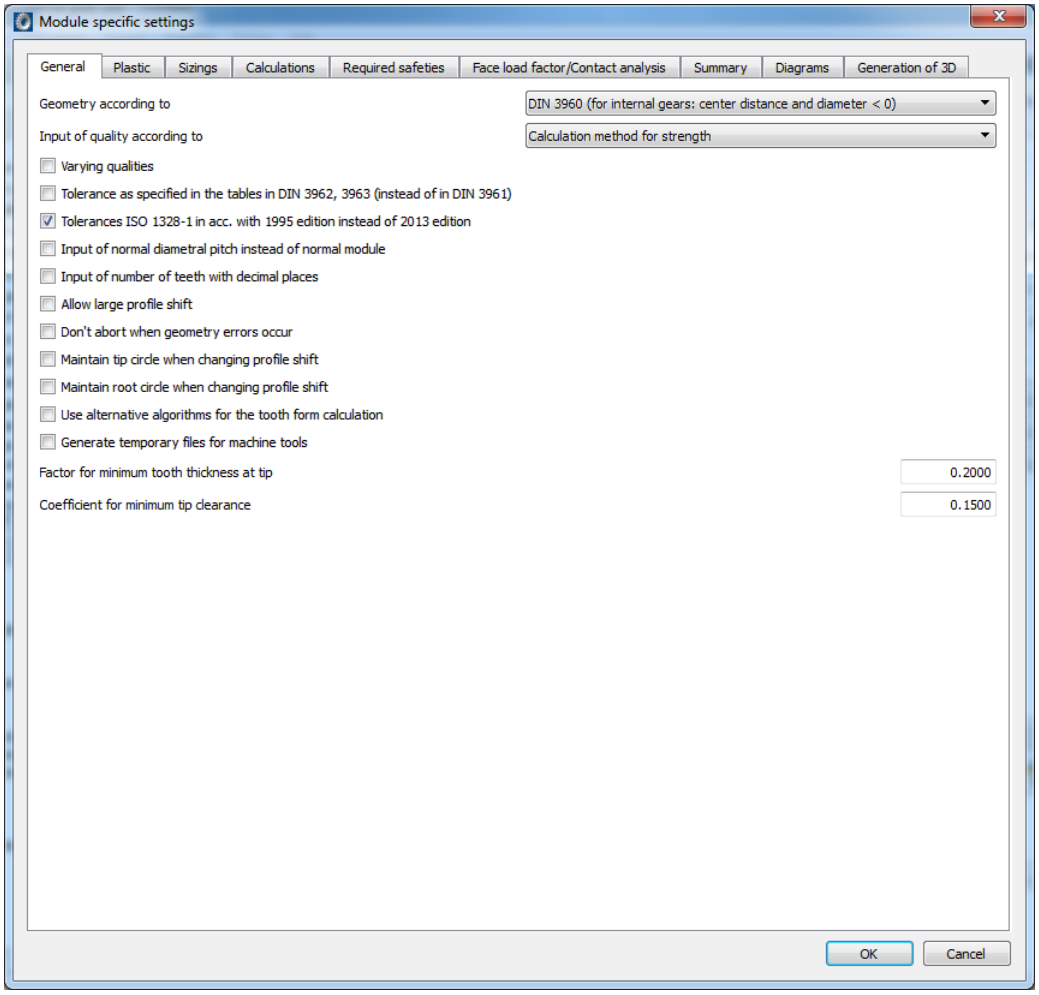


Figure 14.64: General tab in Module-specific settings

14.20.1.1 Input quality

The manufacturing allowances that are output in the report and used for particular factors in the strength calculation procedure are defined either in the DIN 3961, ISO 1328 or AGMA 2015 standards. You can specify which standard is to be used. If you click the `Calculation method for strength` option, the system applies the standard that is best suited to the strength calculation method (for example ISO 1328 is used if you are using the ISO 6336 calculation method).

14.20.1.2 Varying qualities

If you select this option, the plus button next to the Quality field in the main screen appears. You can then use this to input specific tolerances manually.

You will find a more detailed description of this in Qualities (see page II-270).

14.20.1.3 Fp tolerance as specified in tables in DIN 3962

The total cumulative pitch deviation Fp given in the tables in DIN 3962 is some ways very different from the Fp calculated in accordance with the formulae in DIN 3961.

14.20.1.4 Extrapolating tolerance values

The tolerances detailed in ISO 1328 (2013 edition), AGMA 2000 and AGMA 2015, are calculated using the formulae in each particular standard and with the effective geometric data (m_n, d, b…). The range of validity must be defined in each case. For example, the tolerances specified in ISO 1328 for a module range 0.5 mm <= m_n <= 70 mm. However, these formulae cannot be applied to toothing which lies outside of the range of validity. Despite this, these formulae are still used in such cases, due to the lack of any other information. In KISSsoft, the relevant limiting value is usually used to determine tolerances (for example, in the ISO standard, the tolerance is defined as 70 mm, and for a module it equals 80 mm). Alternatively, you can set the "`Extrapolate tolerance values`" flag to calculate tolerances using the effective value (i.e. with 80 mm).
In DIN 3961 and ISO 1328 (1996 edition), tolerances are calculated with the geometric mean values and therefore no extrapolations can be performed here.

14.20.1.5 Input normal diametral pitch instead of normal module

If you select this option, the `Normal module` input field in the `Basic data` or `Geometry` input window is replaced by the Diametral Pitch input field.

14.20.1.6 Input of number of teeth with decimal places

In KISSsoft, you can perform a calculation with a fractional number for the number of teeth. We recommend you use this option for arcs of circles or for nonsymmetrical teeth.

14.20.1.7 Allow large profile shift

Use this option to significantly increase the range of profile shifts that can be performed ($-1.2 \leq x^* \leq +1.5$). This is very useful for special cases. We recommend you use it for: cylindrical gears, bevel gears, worms, crossed helical gears.

14.20.1.8 Do not cancel if geometry errors occur

If you select this option, the program will continue the calculation even if severe geometry errors, such as pointed teeth, meshing interference etc. occur. This options enables you to continue the calculation in critical cases, however, you should then use its results with extreme caution!

14.20.1.9 Maintaining the tip circle when changing profile shift

In the KISSsoft system, the reference profile is usually maintained and the tip and root circle are modified to suit. If you select this option, the tip circle is retained, and the reference profile is modified to match it when the profile shift is changed. The tip circle value is retained as long as the number of teeth and the transverse module stay the same.

14.20.1.10 Maintain root circle when changing profile shift

In the KISSsoft system, the reference profile is usually maintained and the tip and root circle are modified to suit. If you select this option, the root circle is retained and the reference profile is modified to match it when the profile shift is changed. The root circle value is retained as long as the number of teeth and the transverse module stay the same.

14.20.1.11 Using alternative algorithms for the tooth form calculation

The tooth form calculation uses a very reliable algorithm for determining the points on a tooth form. However, in a few special cases this algorithm does not provide a good solution. In such situations, using an alternative algorithm may help.

14.20.1.12 Coefficient for minimum tooth thickness at tip

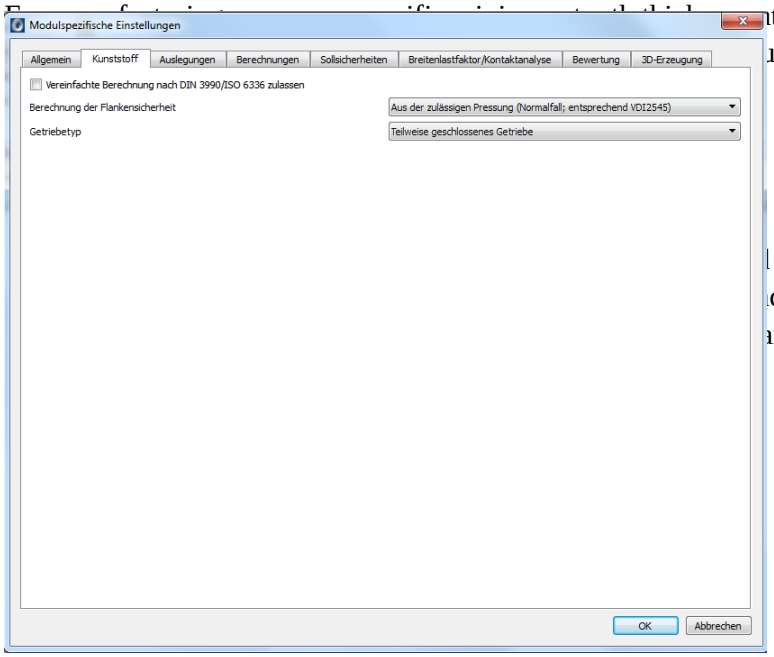


Figure 14.65: `Plastic` tab in `Module-specific settings`

14.20.2.1 Allow simplified calculation according to DIN 3990/ISO 6336

If you select this option, you can use the calculation methods for steel gears to calculate plastic gears. This calculation is performed in accordance with the endurance limit values listed in the materials database. The values for the plastics

given in this database apply for oil lubrication, a temperature of 70° and a number of load cycles of 10^{8}. In contrast to the calculation according to VDI 2545 or VDI 2736, the strength values do not depend on the temperature and lubrication type.

The calculation is performed in the same way as for through hardening steel, with the corresponding S-N curve (Woehler lines) as defined in ISO 6336.

14.20.2.2 Calculation of flank safety

Flank safety for plastic gears is calculated as specified in VDI 2545 or 2736 (in a similar way to the calculation for steel gears). In other words, flank safety is defined using Hertzian pressure with the permitted material value for pressure σ_{Hlim}. However, measurements taken from plastic gears have often shown that their tooth flanks wear in a similar way to the tooth flanks in worm gears. The KISSsoft system also provides the calculation of safety against wear as an alternative. Flank safety is calculated using σ_{Hlim} if the material database (or the material file with extended data in that database) includes data for σ_{Hlim}.
Safety against wear is calculated if wear data is present in the material database. If data for both calculations is present, the system performs both calculations. If only data for one calculation is present, the corresponding safety is displayed automatically.

14.20.2.3 Material pairing-dependent properties

The coefficient of friction, wear factors and heat transfer coefficients, which depend significantly on the selected material combination, can now be defined in a separate .DAT files. The material-dependent properties are defined in the CoefficientOfFriction.DAT, WearFactors.DAT and HeatTransferCoefficient.DAT files, stored in the kiss/dat directory.

To activate these options, go to `Module specific settings` -> `Plastic` and select the properties for which material-dependent properties should be read from a file. If the options are enabled, the values for the coefficient of friction and wear factor (defined in the individual material .DAT files) will be overwritten by the values in the material pairing .DAT files. For the heat transfer coefficient, the values defined in VDI 2736-2 will be overwritten by the values defined in the HeatTransferCoefficient.DAT file.

The characteristic values for the material pairings are shown in Figure 65a. Material pairings are defined as material ID number_material ID number_. A .DAT file contains information about different ways to describe a TABLE FUNCTION. You will find the material ID number shown as the database tool ->`Material`

of gears->Column ID. The function can also be used for user-defined materials.

```
-- Coefficient of friction for plastic/plastic combination
-- Combination PA66(VDI2736) and POM(VDI2736)
-- Lubricant type: 0-Oil 1-Grease 2-Dry running
:TABLE FUNCTION 10491_10492_COF
      INPUT X SchmierTyp TREAT LINEAR
DATA
      0      1      2
      0    0.2   0.40
END
-----------------------------------------
-- Wear factor for plastic/plastic combination for dry running
-- Combination PA66(VDI2736)_POM(VDI2736)
-- in 10^-6 mm^3/Nm
:TABLE FUNCTION 10491_10492_dry
      INPUT X MaterialID TREAT LINEAR
      INPUT Y ZahnTempFlanke  TREAT LINEAR
DATA
      10491    10492
 40    2.00     1.05
 60    5.60     6.50
 80    12.0     10.4
END
-----------------------------------------
-- Heat transfer factor Kroot for plastic/plastic combination
-- Combination PA66(VDI2736)_POM(VDI2736)
:TABLE FUNCTION 10491_10492_Kroot_dry
      INPUT X MaterialID TREAT LINEAR
DATA
      10491    10492
       1500     1500
END
```

Figure 14.65a: Defining the properties for material combinations in the corresponding DAT file

14.20.2.4 Permissible maximum wear of tooth thickness

When you calculate wear safety (see page II-460) you must specify a permitted wear threshold value. A usual value for plastic is 15% (tooth thickness wear in the reference circle). If no, or only very little, wear can be tolerated, we recommend you input a value between 5 and 10%.

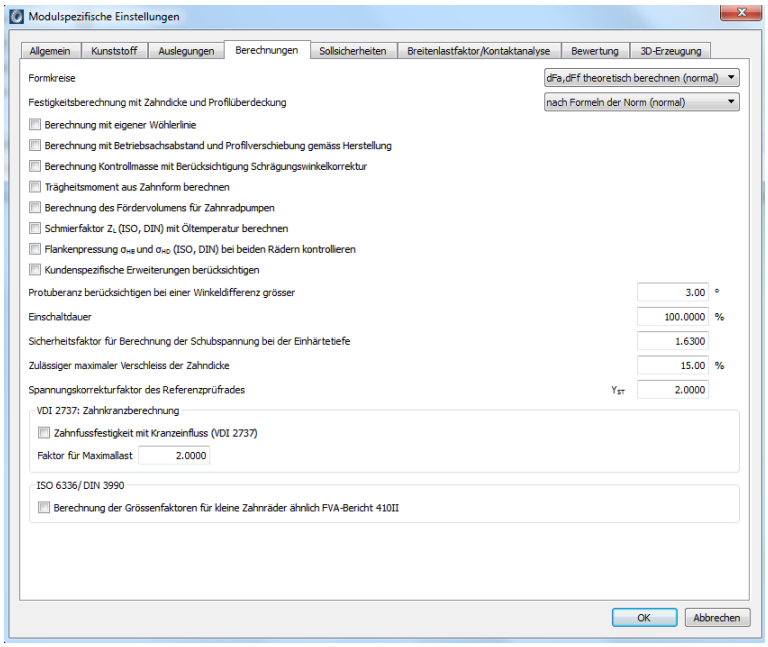


Figure 14: `Calculations` tab in `Module-specific settings`

14.20.3 Planets

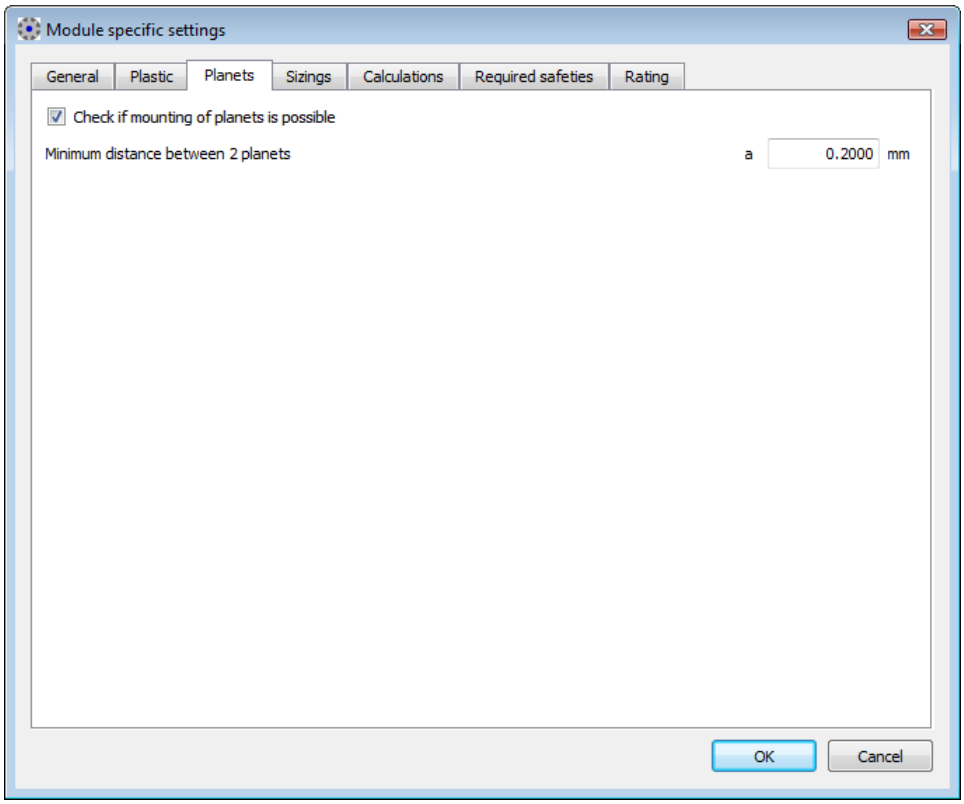


Figure 14.66: `Planets` tab in `Module-specific settings`

Check if mounting of planets is possible

Planets are usually arranged with a constant pitch on the planet carrier (if 3 planets are involved, they will each be at 120 degrees etc.). In these situations, the number of teeth must fulfill certain conditions to ensure the planets can be mounted correctly. KISSsoft performs the check if this checkbox is selected.

Minimum distance between 2 planets

You can specify a minimum distance between the tip circles of two planets in this input field. If this minimum distance is not reached, the system displays a warning message.

14.20.4 Sizings

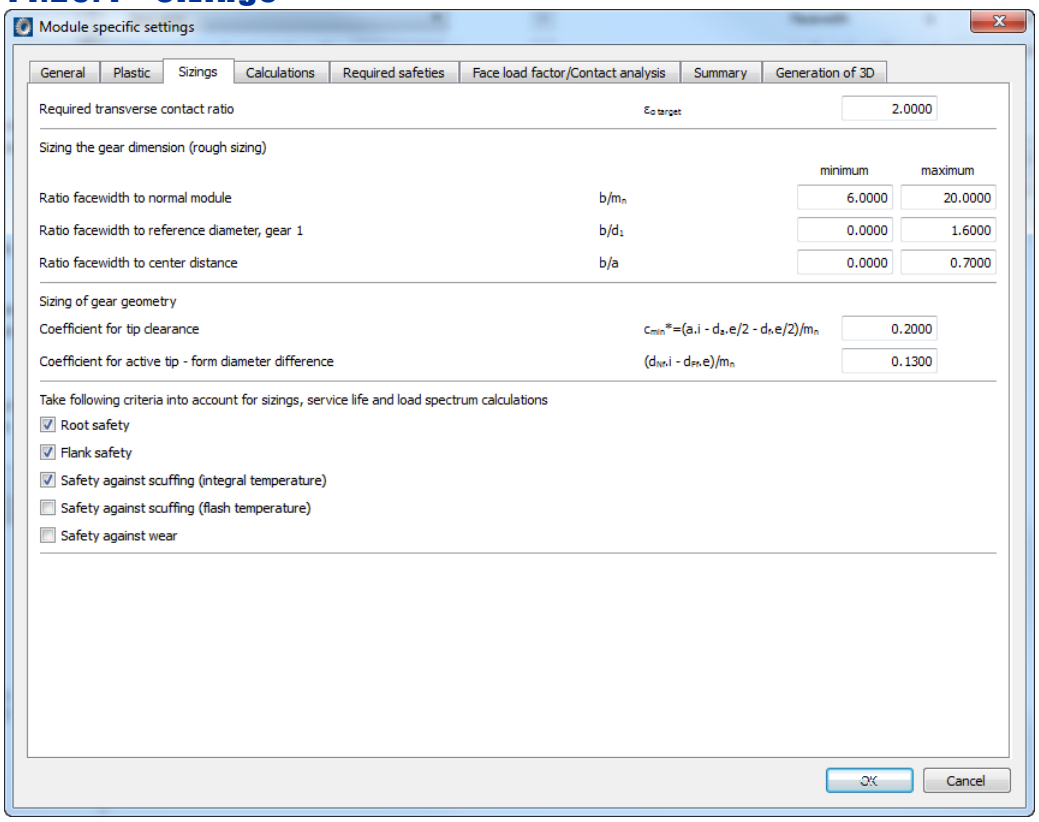


Figure 14.67: `Sizings` tab in `Module-specific settings`

14.20.4.1 Required transverse contact ratio

You can specify the required transverse contact ratio for sizing deep tooth forms (see page II-669).

14.20.4.2 Ratio facewidth to normal module

The facewidth to normal module ratio is a characteristic value used to achieve reasonable dimensions for gear stages. If very narrow gears are involved, the axial stiffness of the teeth cannot be guaranteed, for this reason, b/m_n should be greater than 6 (see Niemann, Table 22.1/7 [65]).

If the gears are too wide, it is essential that the load is spread evenly across the entire facewidth, depending on the gear type and accuracy grade, b/m_n should therefore be less than 15..40 (see Niemann, Table 22.1/10 [65]).

14.20.4.3 Ratio facewidth to reference circle, gear 1

The facewidth to normal module ratio is a characteristic value used to achieve sensible dimensions for gear stages. Depending on what type of heat treatment is involved, this ratio should be less than 0.8 to 1.6 (see Niemann, Table 22.1/5 [65]).

14.20.4.4 Ratio facewidth to center distance

The facewidth to center distance ratio is a characteristic value used to design standard gear units of modular construction. Depending on the stiffness of the housing, this value should be less than 0.3 to 0.5 (see Niemann, Table 22.1/6 [65]).

14.20.4.5 Sizing of gear geometry

The coefficients for the tip clearance and active diameter/form diameter difference (also known as "form clearance") are settings that are needed for the functions used to size the reference profile, in the "Reference profile" tab and in fine sizing. For example, you may need to adjust the reference profile there to achieve an exact form clearance and to ensure that the minimum tip clearance is also exceeded.

14.20.5 Calculations

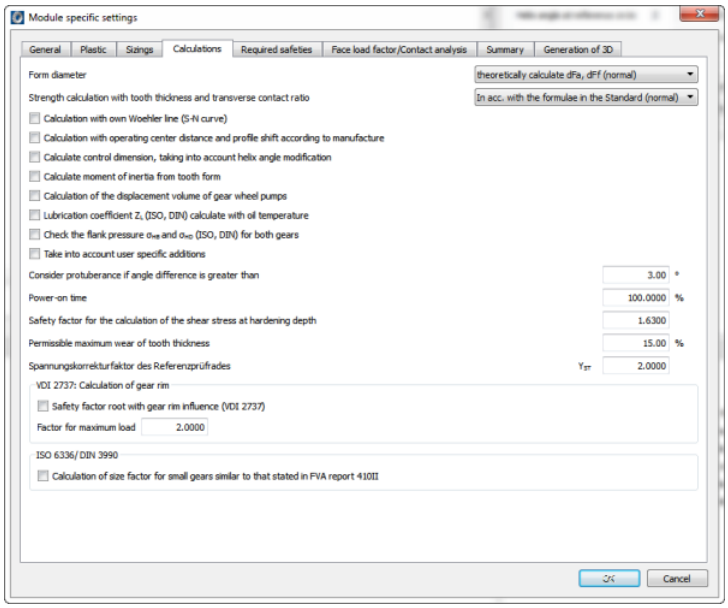


Figure 14.68: `Calculations` tab in `Module-specific settings`

14.20.5.1 Calculate form diameter from tooth form

The tooth form calculation simulates the manufacturing process. In doing so it calculates the effective undercut in the tooth root. Use the `Calculate form diameter from tooth form` option to calculate the tooth form in every calculation run, define any undercut that is present and include it in the calculation. This is then used to calculate the transverse contact ratio and the root and tip form circles (generated diameters). If this option is not set, the root and active tip diameter are defined with the usual method for involutes without taking undercut into account. See for example, DIN 3960. The note about the undercut is also only derived from DIN 3960 formulae.

You can select whether the root form diameter, the tip form circle, or both these values, are to be included in the tooth form. Up to now, the form diameter for racks has not been taken from the tooth form.

NOTE:

If this option is selected, and profile modifications have been predefined, the calculated form diameter will be at the beginning of the modification. This often results in very small transverse contact ratios εα.i and εα.e. This is correct because,

at the start of the modification, the tooth form no longer exactly matches the involute. However, the message that is displayed, to inform the user that the transverse contact ratio is too low, is rather confusing. If the profile modification has been sized correctly, so that meshing under load involves a whole tooth height, this message can be ignored. This is because the transverse contact ratio under load corresponds to the theoretical transverse contact ratio εα. Generally speaking, we recommend you do NOT use this option for profile modifications.

14.20.5.2 Calculation using your own S-N curve (Woehler lines)

The S-N curve (Woehler lines) of metallic materials is usually defined by the endurance limit values sigFlim, sigHlim, entered in the database, and the finite life calculation values Y_{NT} (root) and Z_{NT} (flank) according to ISO, AGMA or DIN. If this option is active, and you input your own S-N curves (Woehler lines) for material, the strength calculation is performed using your S-N curve (Woehler lines).

If you use your S-N curves (Woehler lines) to calculate plastics, the `Calculation with own Woehler line` flag has no effect.

Notes about calculation methods using your own S-N curves (Woehler lines):

- Here, you can use the calculation methods specified in ISO and DIN for metallic materials.
- The Woehler curves are stored in a file (see under: Database). The sustainable stress of the material (sigFadm for root and/or sigHadm for flank) is defined according to the number of cycles NL.
- The endurance limit values sigFlim and sigHlim, which are input directly in the database, are also required for documentation purposes, and should be stored conveniently together with the S-N curve (Woehler lines) data. We recommend you use the value of sigFadm/sigHadm if NL=10^7 for sigFlim/sigHlim.
- The service life factor, factor Y_{NT} and Z_{NT} is defined and reported as follows: Y_{NT} = sigFadm / sigFlim, Z_{NT} = sigHadm / sigHlim
- The other factors which influence the permitted material value, such as Ydrel, YRreIT, YX, ZL, ZV, ZR and ZW, are calculated and used in accordance with the selected calculation method (ISO or DIN). For this reason, the selected permitted material value sigFG or sigHG is not exactly equal to the value sigFadm/sigHadm from the S-N curve (Woehler lines).

14.20.5.3 Calculation with operating center distance and profile shift according to manufacture

Cylindrical gear geometry according to DIN 3960 is based on the calculation of the toothing (which is theoretically without clearance). This enables the total profile shift for the individual gears over the center distance to be specified.

Using this option, you can enter the profile shifts independently of the center distance. This is very useful as it provides a way to check the limits of a toothing (clearance, contact ratio etc.) if there are major variations in the center distance (e.g. in the case of center distance tolerance zones).

14.20.5.4 Calculate moment of inertia from tooth form

The toothing moment of inertia is calculated exactly from the tooth form in the tip to root diameter range. To achieve this, the KISSsoft tooth form calculation is run automatically for each calculation and defines the effective tooth form by the numerical integration of the moment of inertia. The result is output in the calculation report. The calculation is also performed in fine sizing and the results documented.

14.20.5.5 Calculating the displacement volume of gear pumps

This option calculates the transport volume without taking the return volume into consideration. If you activate this option, the tooth spaces are integrated numerically to calculate the transport volume and the result is output in the report. In Fine sizing, the transport volume of each variant is also calculated and output. This enables you to identify, for example, the variant with the largest displacement volume.

14.20.5.6 Calculating the displacement volume of gear pumps

According to ISO 6336 (or DIN 3990), the Hertzian pressure at the single contact point B is only monitored for the driving gear, and in D the pressure is only monitored for the driven gear. This option can be used to check both gears at points B or D, depending on which one is subject to the greater Hertzian pressure.

14.20.5.7 Take into account user specific additions

If KISSsoft AG has added customer specific upgrades to the software, you can select them and view them here.

14.20.5.8 Calculate lubrication factor with oil temperature

Unlike in ISO 6336 and DIN 3990, where the calculation is always performed with an oil viscosity of J= 40oC, when you click this checkbox the lubrication coefficient is calculated with oil viscosity at operating temperature. If this option is selected, the material pairing factor ZW is also calculated with the viscosity present at operating temperature.

14.20.5.9 Strength calculation using average position in tolerance field (of tooth form)

By default, values for the theoretical toothing (without allowances) are referenced for calculation. When you enable this checkbox, KISSsoft performs the calculation with the average allowances for the center distance, root circle and tooth thickness. This option is suitable for use where large tolerances are present.

This option has no influence on calculations performed according to AGMA.

14.20.5.10 Take protuberance into account

If the angle difference (protuberance, or buckling root flank) to the pressure angle is greater than the maximum difference defined here, its influence on the tip and root form diameters, and also the transverse contact ratio, are taken into account. The contact ratio then reduces accordingly.

14.20.5.11 Power-on time

The system also takes into account the power-on time when calculating the number of load cycles (multiplied by the service life).

The power-on time is also taken into account for plastic toothed gears when calculating the flank and root temperature. For worm gear units, this time is also included when calculating the thermal safety.

Note about calculating temperature:

It is assumed that heat is constantly dissipated, but that, in contrast, heat is only generated during the specified power-on time. The precondition for this is that the gear unit is only run for a short period of time (maximum 15 minutes), and is then stopped again. If this is not the case, the power-on time must be set to 100%.

14.20.5.12 Safety factor for calculating the shear stress at hardening depth

The safety factor is multiplied by the shear stress, which is then used to calculate the hardness. The hardening depth is then defined using this value.

14.20.5.13 VDI 2737: Calculation of gear rim

The strength calculation for inner gears is not very accurate. A significant improvement is needed. Gear rims are often subject to stresses that can affect their load capacity. At present, VDI 2737 is the only guideline that includes gear rim stress and the influences associated with this. The calculation is performed in two steps

1. Tooth root fracture safety (static and endurance) without taking the gear rim influence into account.
2. Tooth root fracture safety with gear rim influence. In this case, the maximum shear stress in the tooth root outside the meshing can in some conditions be greater than the actual bending stress in the tooth that is under load.

The notch factor Y_S, as in ISO 6336:2006, is defined as the place at which the tangent on the flank and the tooth center line form an angle of 60°.

The results of the calculation specified in VDI 2737 are detailed in their own section in the normal report.

Factor for maximum load (VDI 2737)

To calculate static safety in accordance with VDI 2737, input a maximum load factor. This is then multiplied with the nominal torque. To calculate the endurance limit, the nominal torque is, as usual, multiplied with the application factor K_A.

14.20.5.14 Calculating the size coefficients for small gears

Calculation of size coefficients for small gears similar to that stated in FVA report 410:

If mn < 1, the size coefficients (according to DIN or ISO) are ZX and YX > 1.

NOTE

If you use this method, you may need to adjust the required safeties that have been used up to now. This is because much higher computational safeties are produced.

14.20.6 Required safeties

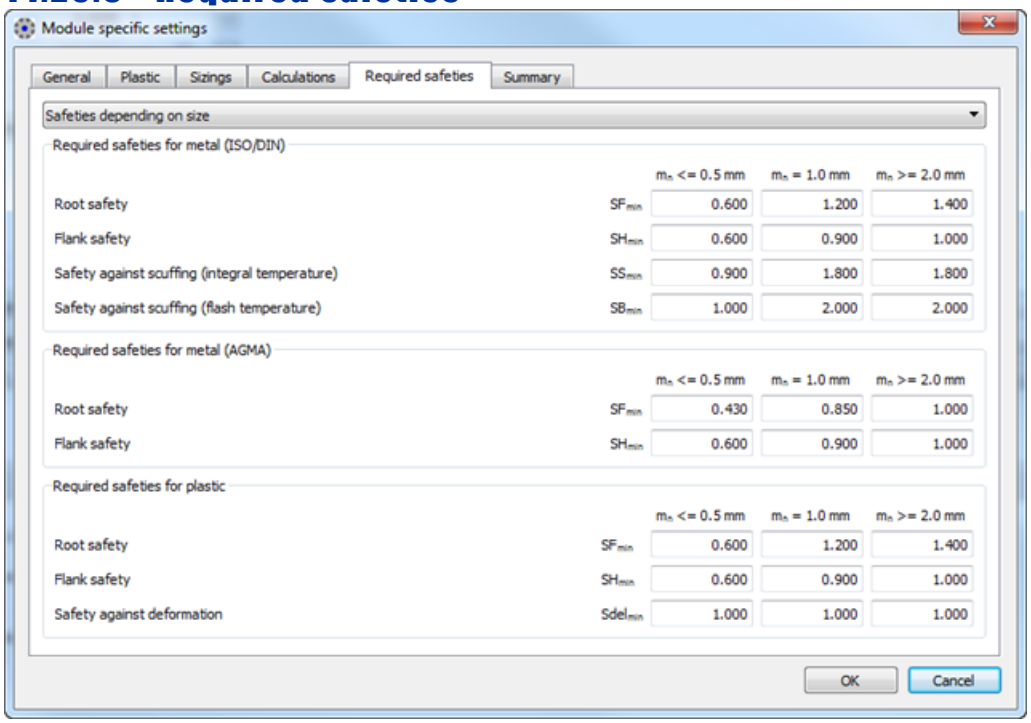


Figure 14.69: `Required Safeties` tab in the `Module-specific Settings` window

You must set required safeties, not only for every service life calculation, but also for rough sizing and fine sizing.

Calculating reliability using Weibull distribution uses the calculated service life, and so also takes the required safeties into account. To calculate reliability without taking required safeties into account, set all safeties to 1.0.

Safeties are not size-dependent

Experience has shown that much lower minimum safeties can be used for smaller modules. Although the standards do not provide any information about this, this knowledge is based on experience with many different applications. However, if you do not require size-dependent safeties, you can still select the "Safeties are not depending on size" variant.

Minimum safety for calculation according to AGMA

In the tooth strength calculation according to AGMA 2001, the permitted tooth bending stress sat is half the size of the value specified in ISO 6336. Although its meaning is similar, the corresponding s_{at} value in the ISO guideline must be

multiplied by a factor of 2, the reference gear's stress correction factor Y_{st}. Therefore, if the tooth strength is calculated according to AGMA 2001, the resulting safety is approximately 50% smaller than that in the calculation using ISO 6336. As a consequence, the safety required for the calculation according to AGMA 2001 is smaller.

Service coefficient

Some applications of the AGMA calculation method require a predefined service coefficient. In actual fact, this is merely a minimum safety. For this reason, if required, you can input service coefficient C_{SF} for flank strength and K_{SF} for tooth bending strength.

14.20.7 Contact analysis/face load factor

14.20.7.1 Contact analysis

Calculation method contact stiffness: Here you can select either the calculation method defined by Weber/Banaschek [69] (dynamic stiffness analysis: default setting), the method defined in ISO 6336-1 Method B and Own input.

Single contact stiffness: If "Own Entry" has been selected as the contact stiffness calculation method, you can enter your own value for the single intervention stiffness.

Slices coupling factor: Linking factor used to link the individual slices of the discretized toothing model.

Border weakening factor: Border weakening factor for a weakening of stiffness on the edge of helical gear teeth.

Stiffness correction factor (according to Winter): Correction factor for Hertzian flattening as described in the experiments performed by Winter/Podlesnik [95].

Take into account plastic deformation: Use this setting to specify whether plastic deformation is to be taken into account in the contact analysis. If plasticity is to be taken into account, the maximum contact stress, calculated using the elastic contact theory, is reduced on the basis of the specified "Maximum permitted flank pressure". If the maximum elastic flank pressure is exceeded, the radii of the contact body are changed locally so that the resulting elastic i.e. contact stress matches this maximum value. Only a percentage rate of the new radii is used, on the basis of the specified "Weighting of the plastic deformation".

14.20.7.2 Display

Unit of transmission error: Here you can select either the length on the length of path of contact (transmission error) or the angle on the gear (angle of rotation error).

Draw data for path of contact: If this option is enabled, the results of contact analysis are displayed quadratically in the 3D diagrams. This makes the data suitable for export as a matrix.

Take into account backlash in the graphic of transmission error: When this setting is selected, the backlash is taken into account in the transmission error. This causes a displacement of the value of the transmission error even though the amplitude remains the same.

Smooth results: This function uses a low-pass filter to smooth the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). By default, this function is deactivated, but can be used to smooth the results if they are affected by strong numerical noise.

Area analyzed on tooth depth: This defines the maximum area along the tooth height for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Area analyzed on facewidth: This defines the maximum area along the facewidth for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Interpolate stress increase caused by tip rounding: In the case of a tip rounding, the calculation of the tooth form results in a sudden change in the radii of curvature. This in turn results in stress increases at this transition point in the contact analysis calculation. For this reason, you can specify whether the mathematical solution is to be used, to perform the calculation, or whether this stress increase is to be interpolated.

14.20.7.3 Face load factor

Iterating the load distribution of the meshings: Only affects planetary stages: If shaft data is used to define the axis alignment, a constant load distribution over the facewidth is initially assumed in the shaft calculation, when the bending is

calculated. This is a satisfactory approximation if the load distribution is fairly well distributed, and the face load factor is therefore not greater than 1.3 (maximum 1.5). If the load distribution is less satisfactory, the load distribution value should be transferred into the shaft calculation, from the gear calculation, and the next calculation of bending should be performed with the modified (not linear) load distribution. This produces a more accurate, modified load distribution. This iterative determination of the load distribution across all the meshings is then performed until the load distribution stops changing in all the meshings. Be aware that this option only shows an effect if at least one of the deformation components is linked with the shaft calculation.

Tooth contact stiffness: This defines whether the tooth contact stiffness is calculated according to ISO 6226 ($C_{\gamma\beta}$) (default) or whether, as specified in AGMA 927-01, it is constant with C_m = 11 N/mm/µm.

Calculation ISO 6336, Annex E: This is where the ISO factors K_V,K_A and K_γ can be taken into account and defined for the proportional deviation and inclination error of axis ($f_{\Sigma\beta-p}$, $f_{\Sigma\delta-P}$) and the load distribution.

Calculating the torsion moment of resistance: If the calculation of torsion due to deformation in the "`Define axis alignment`" dialog is set to "Own input", the diameter specified here is used in the calculation.

14.20.8 Summary

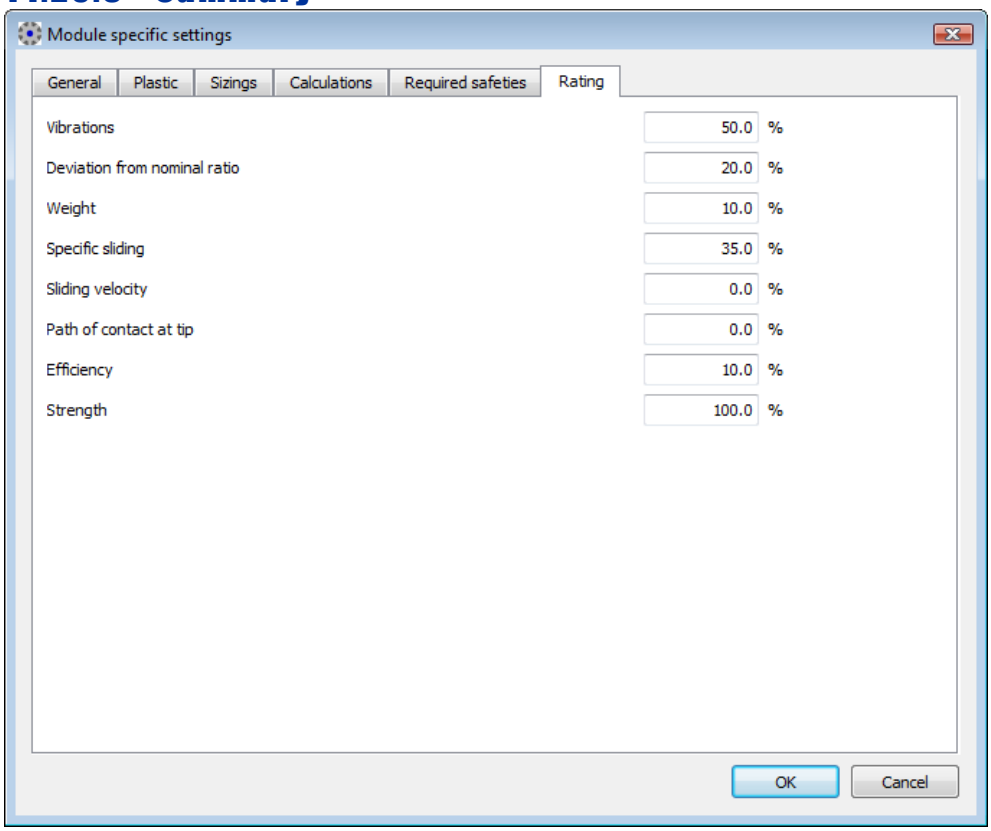


Figure 14.70: `Summary` tab in the `Module-specific settings` window

Weighting the individual component to evaluate the `Summary` coefficients in Fine Sizing. (see section "Results" on page II-446)

14.20.9 Diagrams

14.20.9.1 Values on the x-axis of diagrams

You can select different values for the x-axis from a drop-down list.

Here you can select the roll angle, the length (path of contact), the diameter of gear A and the angle of rotation.

You can also decide whether the X-axis (path of contact) and Y-axis (facewidth) are to appear as scales in the 3D diagrams or not at all.

NOTE

If you select the angle of rotation for the x-axis the gear axis is 0°.

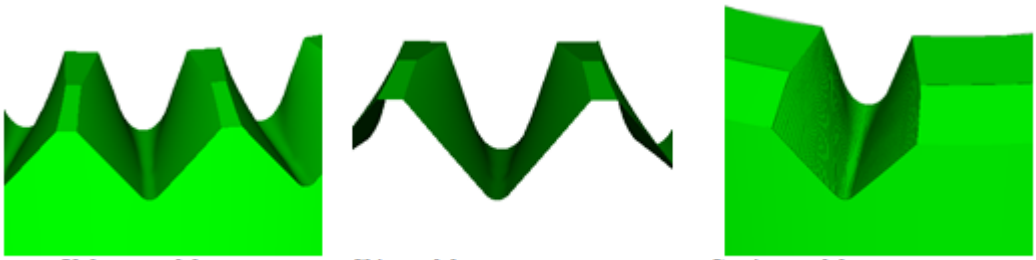
14.20.9.2 Helix angle modification, parallel

The entry concerns the display in the tooth trace diagram. If you select "Manufacture with modified helix angle β_{eff}", the helix angle modification is included in helix angle β, and the base tangent length and measurement over pins are defined with the modified angle β_{eff}.

14.20.10 Generation of 3D model

This is where you modify the parameters used to generate 3D models.

- Under **"Model"**, specify the type of model to be generated (volume model, skin model, cutting model). The volume model can be used for other applications such as machining by CNC or finite element analysis. The skin model is most suitable for contact analysis. The cutting model is only suitable for the gear models that use cutting simulation, such as face gear and enveloping worm gear, and is used to view the actual cutting simulation.



- The **Number of cutting steps** sets the number of cuts per half pitch for the cutting process. The minimum value is 1, and the default value is 20. The quality of the final model can be increased by increasing the number of cutting steps, but this also increases the probability of manufacturing errors. The "Scale factor" is used for solving the failure problem. If the operation fails, we recommend that you use a lower number of generation steps with a larger scale factor.
- The **Number of sections along facewidth** defines the number of sections along the facewidth for approximating the tooth flank form. The minimum value is 2, and the default value is 11. Normally, the quality of the final model can be improved by increasing this value, but we do not recommend that you set a number that is excessively high, compared with the facewidth. The factor is used for the gear models using cutting simulation and gear models using

multiple section curves such as spiral bevel gear and cylindrical gear with lead modification.

- The **Scaling factor for the cutting model** is used to scale the model during the cutting simulation process. The minimum value is 1, and the default value is 10. Sometimes the cutting simulation can fail due to an internal operation error in the Parasolid kernel, especially when the model has a very small module and/or a large number of generation steps. In order to prevent this type of operating error, use the model with its size set by by the scale factor in the cutting process. Consequently the cutting model can have different dimensions than the actual design. However, the volume and skin models are automatically returned to their original scale (size) after the operation, and therefore have the same dimensions as the entered gear.
- The **Modeling operation** sets the tolerance for the internal operations of the Parasolid kernel, such as the chordal approximation and clash detection in Boolean operations. The default value is 1 μm.
- The **Rendering quality** sets the resolution of the resulting graphics in the 3D geometry view. This is used only to improve the viewer display (usability) and does not affect the quality of the generated model. If the rotation operation in the viewer is slow, you can increase the quality value to speed up the operation. The default value is 5 μm.
- Click on "**Constant root radius along the facewidth**" to specify the method used to generate the root radius of a bevel gear. The root radius of the bevel gear changes by the factor of the normal module along the facewidth. If you activate this flag, we use the constant root radius defined by the normal module in the middle section. (Available for bevel gear)
- The "**Constant protuberance along the facewidth**" sets the protuberance of the reference profile of the bevel gear. The protuberance of the reference profile of the bevel gear changes by the factor of the normal module along the facewidth. If you activate this flag, we use constant protuberance defined by the normal module in the middle section. (Available for bevel gear)
- If "**Display 2D geometry for outside and inside**" is selected, the tooth forms on the internal and external sides are represented as a 2D graphic. (Available for bevel gear)
- If "**Generate tooth system model in the saved position**" is selected, the system model is generated at the position you saved. This position is saved in the calculation file, and you will be able to restore the checking position of the contact pattern in future. (Available for bevel gear)
- Click on "**Number of points on the edge of cut for spline approximation**" to specify the number of modeling (intermediate) points on each edge that are

used to approximate the spline curves for the root area or the tooth flank. The figure shows a diagram in which the points that are to be used are scanned. The end points (nodes) are removed since they are giving extreme waviness on the curve. We use only the intermediate points on the cutting edge supposing the parametric distance between the points is the same. We usually recommend that more points are used in the root area. However, this model will help the user determine the optimum value. (Available for enveloping worm gear)

- Click on "**Oversize factor for worm wheel cutter**" to define the coefficient used to increase the worm wheel cutter. There are different methods of implementing the interference hob. Those methods include the axial pitch method, the base pitch method, the extra thread method, and the normal pitch design method. KISSsoft is using the normal pitch method since the method is practically regarded as the industry standard. These methods are based on the principle, that the worm wheel cutter uses the same normal pitch and the same normal pressure angle in the normal section as the worm. The cutting distance between the hob and the gear will then be changed accordingly, to ensure a consistent result for the root and tip diameters on the gear. If you are using the oversize factor, the generated surface will not be conjugate with the worm surface and will not give the best contact pattern. Thus, especially if you are not using conventional cutting method but want to use theoretical surface geometry, you should not use the oversize factor. In practical case, the tooth thickness of the cutter will be increased to consider the tooth thickness tolerance of the worm wheel. In that case, it's recommended to use small oversize factor compensating the tolerance to get the best contact. (Available for enveloping worm gear)
- Click on "**Cutter shaft angle change**" to modify the shaft angle of the worm wheel cutter during the simulated milling run. The angle can be both positive and negative, and the positive angle is defined as shown below. (Available for enveloping worm gear)
- Click on "**Change in pressure angle of the worm wheel cutter in normal section**" to set the worm wheel cutter to a different pressure angle than the worm. (Available for enveloping worm gear)
- Click on "**Flank shape of worm wheel cutter**" to set a different tooth form for the worm wheel cutter than for the worm. There are many research works justifying the different combinations of the tooth form to get better contact pattern in worm gearing. This setting is used for this purpose. If this option is not selected, the same tooth form is used for both the worm wheel cutter and the worm. (Available for enveloping worm gear)

14.21 Tooth thickness

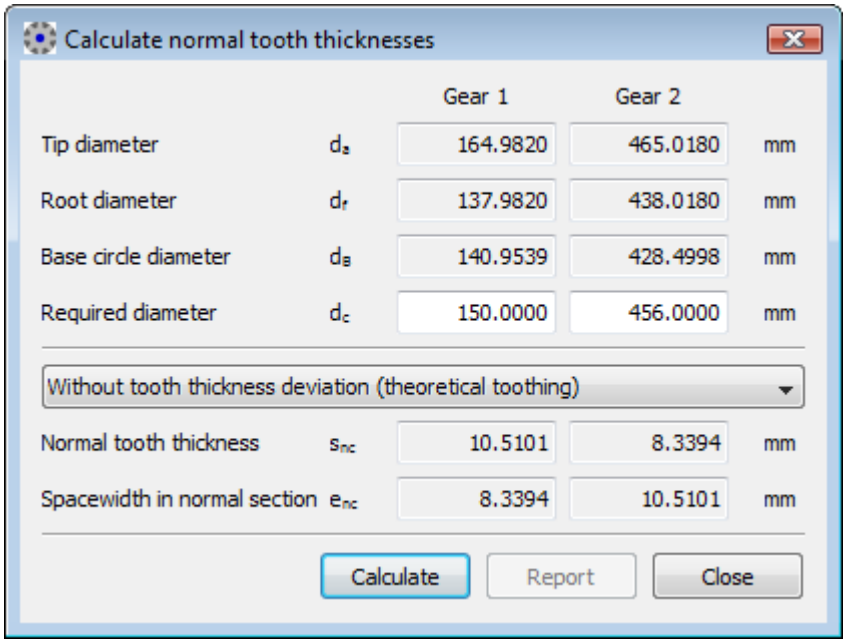


Figure 14.71: `Chordal tooth thicknesses` dialog window

Click the `Calculation > Tooth thickness` menu option to calculate the normal tooth thickness and the normal spacewidth for any diameter.

The tooth thickness is output as an arc length and as a chordal length. To help measure the tooth thickness, the chordal height is output along with the tooth thickness allowances.

Chapter 15
Bevel and Hypoid Gears

Use this module to calculate the geometry and strength of straight, helical and spiral bevel gears (gear axes intersect, offset is 0) and hypoid gears (crossed gear axes, offset not 0). Geometry as specified in ISO 10300, ISO 23509 and DIN 3971, tolerances according to ISO 17485 and DIN 3975, strength calculation as specified in ISO 10300 (equivalent cylindrical gear toothing method), AGMA 2003, DIN 3991, or Klingelnberg in-house standard KN 3030. The calculation only includes the geometry of bevel gears to the extent that is necessary for the strength calculation (see section "Methods used for strength calculation" on page II-501), no matter which manufacturing process is used.

15.1 Underlying principles of calculation

15.1.1 General

The geometry of bevel gears is calculated according to ISO 10300, ISO 23509 or DIN 3971. The strength calculation is performed in two steps. A virtual cylindrical gear toothing is defined first. This is then used for the strength calculation in a similar way to cylindrical gears. The process is described in [24], [45] and [66].

Bevel gear machine tool manufacturers (such as Klingelnberg in Germany) also have their own methods that differ slightly from the processes mentioned above.

Hypoid bevel gears and bevel gears with offset are primarily used in vehicle axle gear units. Strength is calculated by defining a virtual cylindrical gear toothing. The tooth root, flank and scuffing safeties, which are important for hypoid gears, are calculated as specified in the Klingelnberg in-house standard KN 3030.

15.1.2 Overview of the bevel gear manufacturing process and the terminology used in it

Various manufacturing processes are used to create bevel gears. Unlike cylindrical gears, the tooth length forms and tooth depth forms differ according to which manufacturing process is used. In particular, the process used to manufacture spiral teeth bevel gears uses a multitude of terms, the most important of which are described below.

The most important differences are shown in the tooth length form, which can be manufactured as an arc of circle (face milling procedure), epicycloid or involute toothing (face hobbing process). Circular pitched teeth were developed by the company Gleason, and are produced using the face milling approach, in which every gap is milled separately, and then the gear is rotated further by the width of that tooth space. Epicycloid toothing is used by Oerlikon and Klingelnberg. In this process, the gear rotates constantly during the milling process. Only the palloid manufacturing process is used to create the involute tooth length form. Although, nowadays, Klingelnberg and Gleason, the market leaders in machine manufacturing, can produce toothing using both the face milling and face hobbing processes, these companies are still associated with their traditional processes in the technical literature. You will find more details in section 15.1.3 and 15.2.1.

Although alternative processes for spur gears are available, they are not listed here.

15.1.3 Calculation according to Klingelnberg, Gleason and Oerlikon

The strength calculation defined in ISO 10300 DIN 3991 only includes the relationships (module, helix angle) in the middle of the facewidth in the equivalent cylindrical gear toothing method calculation. The shape of the bevel, and the process used to manufacture it, are ignored. As a result, the KISSsoft strength calculation method can be applied no matter which process is being used, especially for Klingelnberg and Gleason This also reflects the experience that the capacity of spiral bevel gears is only slightly affected by the manufacturing process.

The geometry calculation procedure in KISSsoft defines the dimensions, such as diameter and tooth thickness, in the middle of the facewidth. It also calculates the diameter at the outside and inside end of the facewidth. These dimensions depend on the type of the bevel. However, the results may differ from the actual conditions, because the processes are not described in sufficient detail. This is particularly true for the Gleason procedure.

- **Klingelnberg procedure:**
  The `Bevel gear (KN3028 and KN3030) and Hypoid gears (KN3029 and KN3030)` calculation methods enable to you calculate geometry and strength and check the manufacturing process in accordance with the Klingelnberg in-house standard. However, these methods do not calculate the machine settings for the selected Klingelnberg machine. When you input formula data from a Klingelnberg program, you must remember that the toothing data, such as module and helix angle, always applies to the middle of the facewidth (unless otherwise specified).

- **Gleason process:**
  Bevel gears are often designed by the Gleason company. Depending on which calculation program Gleason uses, toothing data such as the module and helix angle, is either predefined for the outside end of the facewidth or for the middle of the facewidth.
  The `Conversion from GLEASON data sheets` dialog window enables you to convert Gleason data from the outside end of the facewidth into data for the middle of the facewidth (see page II-495). Once this data has been converted, you can perform the strength calculation. Although the bevel dimensions (tip and root diameter) do not always exactly match the actual geometry they are close enough to enable you to check the assembly conditions (in a drive). This procedure does not check to see whether the part can be manufactured on Gleason machines.

- **Oerlikon process:**
  The Oerlikon process is broadly similar to the Klingelnberg process (select Klingelnberg bevel type).

15.2 Basic data

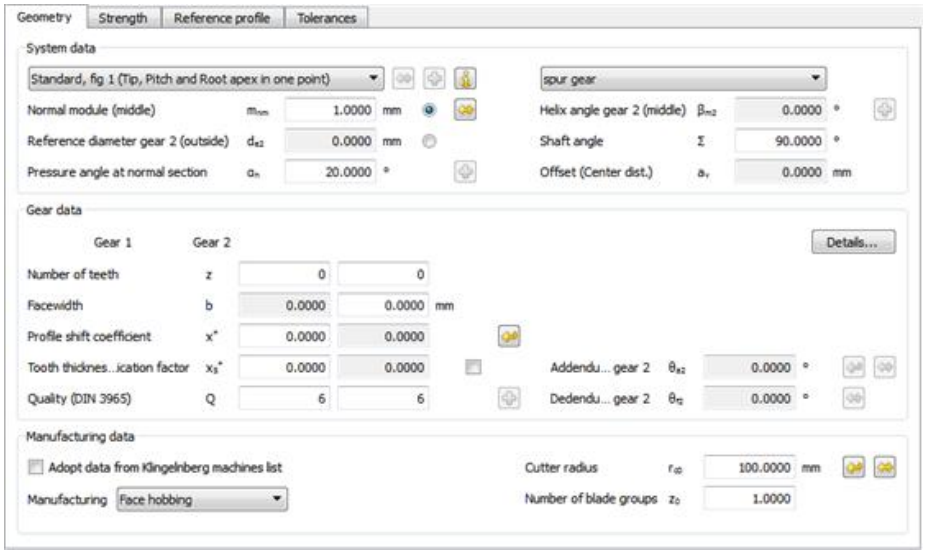


Figure 15.1: `Basic data` input window

15.2.1 Mean normal module

The reference circle of the external end of the bevel of Gear 2 (de2) is usually specified for bevel and hypoid gears. In the basic "Oerlikon" type, the reference circle of Gear 2 is predefined in the middle (dm2). Alternatively, you can specify the normal module in the middle of the facewidth. However, if you know the pitch, the transverse module, or the diametral pitch, instead of the normal module, click the  button to open a dialog window, in which the conversion will be performed. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` in `Calculation > Settings > General`.

15.2.2 Reference diameter gear 2

If there are changes to the sizes of the set of bevel gears, click the button to enter the new outer reference diameter of Gear 2. This is useful for designers since, very often, the amount of space available for installing the larger gear is predefined. The module is then recalculated (not optional).

15.2.3 Pressure angle at normal section

For standard meshings, the pressure angle is $\alpha_n = 20^o$. Smaller pressure angles can be used for larger numbers of teeth, to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases.

For hypoid gears, click the  button to input the pressure angle for the driving flank and the driven flank independently of each other. The driving flank is the concave flank of the pinion and the convex flank of the gear. The driven flank is the convex flank of the pinion and the concave flank of the gear.

15.2.4 Pressure angle driving/driven flank: Hypoid gears

Bevel gears are usually better able to withstand stress when driven by the concave pinion flank, i.e. when the spiral on the pinion and its direction of rotation run in the same direction.

The concave flank of the pinion is usually called the driving flank (index D for "Drive"), and the convex flank is known as the driven flank (index C for "Coast"). In the gear, the concave flank is the driven flank (index C) and the convex flank is the driving flank (index D). Since the effective nominal pressure angle on the driving flank is greater by the amount of the limit pressure angle, and on the driven flank it is smaller than the pressure angle in a normal section, by the amount of the limit pressure angle, the nominal pressure angle on the driving flank and driven flank can be entered independently.

For hypoid gears, as specified in ISO 23509, you should input the nominal design pressure angle as α_{dD}, α_{dC}. This is used to calculate the generated pressure angle ("effective pressure angle") α_{nD}, α_{nC} for the driving side (index D for "Drive") and the effective pressure angle α_{eD}, α_{eC} for the driven side (index C for "Coast").

The equations specified in ISO 23509 are:

$\alpha_{nD} = \alpha_{dD} + f\alpha_{lim} * \alpha_{lim}$

$\alpha_{eD} = \alpha_{nD} - \alpha_{lim}$

If, as a result α_{nD} has been specified, α_{dD} can be calculated as follows:

$\alpha_{dD} = \alpha_{nD} - f\alpha_{lim} * \alpha_{lim}$

$\alpha_{dC} = \alpha_{nC} + f\alpha_{lim} * \alpha_{lim}$

or if α_{eD} has been specified, α_{dD} can be calculated like this:

$\alpha_{dD} = \alpha_{eD} + \alpha_{lim} * (1- f\alpha_{lim})$

$\alpha_{dC} = \alpha_{eC} - \alpha_{lim} * (1- f\alpha_{lim})$

The limit pressure angle α_{lim} is calculated by KISSsoft and output in the report.

The influencing factor of the limit pressure angle $f\alpha_{lim}$ has been introduced so that you do not always need to take the total value of the limit pressure angle into consideration when calculating the flank angle on the tool. For forming tools (Klingelnberg process), $f\alpha_{lim} = 0$ is set. If you use the procedure with a constant slot width (Gleason), $f\alpha_{lim} = 0.5$ is set, otherwise $f\alpha_{lim} = 1.0$ is often used.

However, if precise data is not available, you can use the pressure angle in the normal section in the calculation (with $\alpha_{dD} = \alpha_{dC} = \alpha_n$ and $f\alpha_{lim} = 1.0$).

NOTE

These input fields are only available if you are calculating the strength of hypoid bevel gears (see section "Methods used for strength calculation" on page II-501).

15.2.5 Spiral and helix angle

The input of the angle is defined in the middle of the facewidth. In the case of helical bevel gears, the angle remains constant across the facewidth. However, in spiral bevel gears the spiral angle changes across the facewidth.

In hypoid gears, the spiral angle is specified in the middle of the facewidth for Gear 2. This value is then used to calculate the value for Gear 1 (pinion).

You can select any value as the spiral angle in the middle of the facewidth. However, we recommend you use a larger angle of between 30° and 45° to ensure optimum running performance. You should only select a value that is less than this guide value if the bearing load has to be reduced.

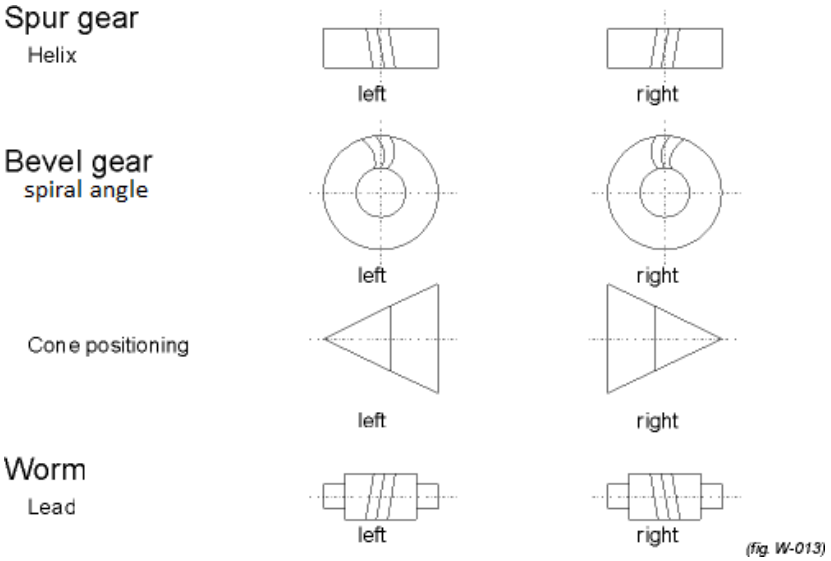


Figure 15.2: Spiral and helix angle

Click the button to the right of the spiral angle input field to open the `Additional data for spiral teeth` window, where you can input the internal and external spiral angle for spiral bevel gears.

15.2.6 Shaft angle

The shaft angle for bevel gears is usually 90°. However, you can perform the calculation for any shaft angle.

15.2.7 Hypoid offset

In the case of bevel gears without hypoid offset, the axes of the bevel gears intersect at one point. In the case of bevel gears with hypoid offset, the axes do not intersect. This application enables you to achieve a higher contact ratio and greater strength at the tooth root. It is primarily used in the vehicle construction industry. This pairing is known as a hypoid bevel gear and is illustrated in Figure 15.3.

NOTE

A positive hypoid offset is almost always applied to hypoid bevel gears, because this is the only way of achieving the improvements to the characteristics described above.

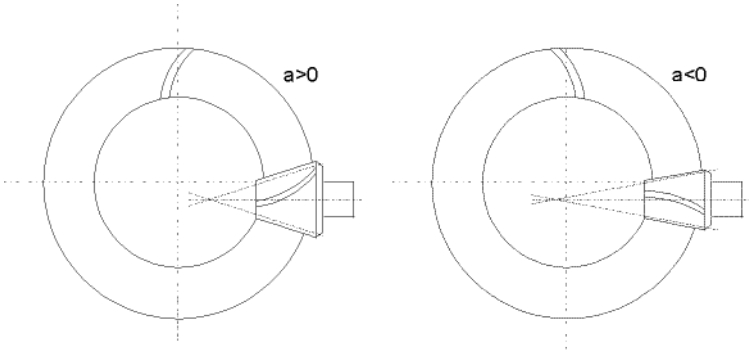


Figure 15.3: Hypoid bevel gear configurations. Positive hypoid offset (a > 0): Gear 1 left-hand spiral, Gear 2 right-hand spiral. Negative hypoid offset (a < 0): Gear 1 right-hand spiral, Gear 2 left-hand spiral

15.2.8 Number of teeth

You will find reference values for bevel gears with a shaft angle of 90 degrees in Table 15.1.

U	1	1.25	2	2.5	3	4	5	6
z_1	18..40	17..36	15..30	13..26	12..23	10..18	8..14	7..11

Table 15.1: Recommended pairing transmission ratio u - number of teeth, pinion z1 in accordance with Niemann [66]

15.2.9 Facewidth

The facewidth should not usually be larger than the one given in the recommendations (ratio to cone length, module ratio (see page II-516)). The contact pattern deteriorates if the facewidth is too great.

15.2.10 Profile shift coefficient

You will find reference values for the profile shift coefficient for bevel gears with a shaft angle of 90 degrees in Table 15.2.

U	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x^*	0.00	0.10	0.19	0.27	0.33	0.38	0.40	0.43	0.44	0.45

Table 15.2: In accordance with Niemann, 24/4 [66], recommended pairings for transmission ratio u- profile shift coefficient x^*

Click on the button to the right of the profile shift coefficient input field to display the minimum profile shift coefficient for the pinion required to prevent undercut as well as the recommended value according to Niemann [66].

NOTE

The ISO 23509 standard defines two different data types that can be used to describe tooth height factors and profile shift. The formulae used to convert data between these two data types are listed in ISO 23509, Chapter 7. The Gleason calculation sheets also give partial descriptions of factors K and C1. Although these are very similar to data type II, there are slight differences. Click the button to convert data type II data.

15.2.11 Tooth thickness modification factor

You will find reference values for bevel gears with a shaft angle of 90 degrees in Table 15.3.

U	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x_s	0.00	0.010	0.018	0.024	0.030	0.039	0.048	0.065	0.082	0.100

Table 15.3: Recommended pairing transmission ratio u - tooth thickness modification factor xs in accordance with Niemann [66]

NOTE

If you are using standard cutters, such as those used for a Klingelnberg palloid or cyclo-palloid gear®, you must use the thickness modification coefficient specified in the standard.

15.2.12 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The accuracy grade defined in ISO 17485 is very similar to that specified in DIN 3965.

You will find notes about the toothing quality in the Manufacturing process (see page II-493).

15.2.13 Addendum angle and root angle

All the necessary data required to create the bevel gear drawing can be calculated from the addendum angle and dedendum angle. These are the tip and active root diameter on the outside and inside bevel, and the tooth thickness on the outside and inside cone diameter (see Figure 15.4). The values shown here are output in the main report. In the case of bevel gears with spiral teeth, the addendum angle and dedendum angle are calculated using the selected method [ISO 23509, DIN3971]. In the case of bevel gear type 2 (Gleason), you can input the addendum angle. The root cone of the counter gear is then calculated from this value.

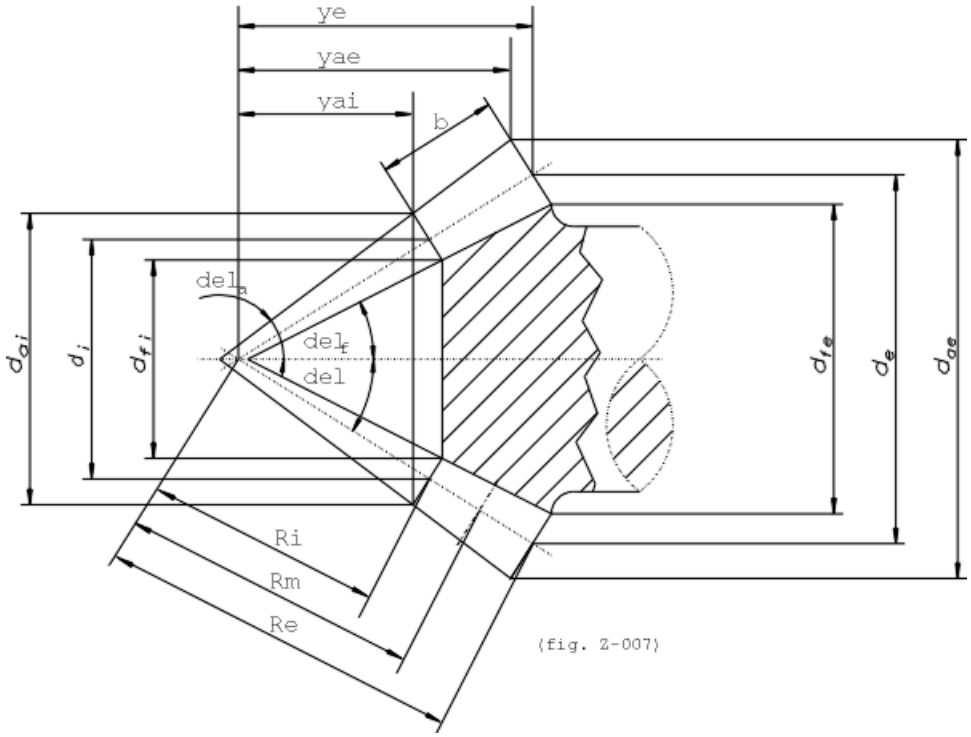


Figure 15.4: Dimensioning a bevel gear

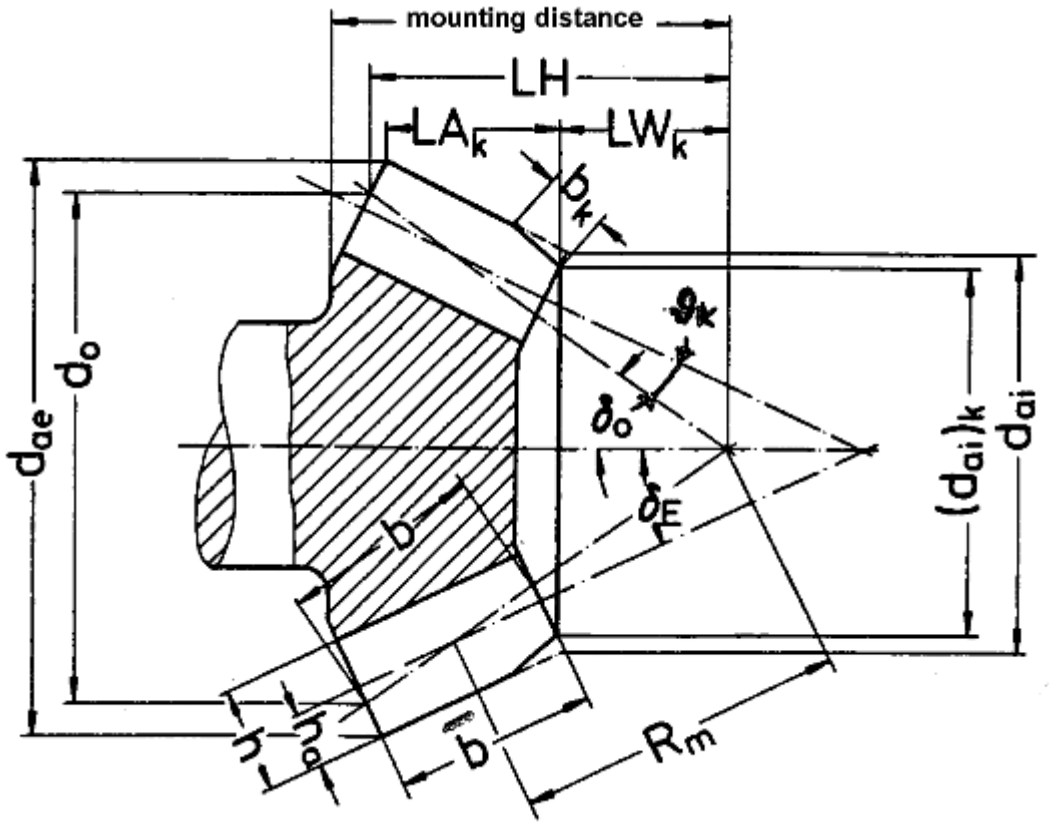


Figure 15.5: Dimensioning a bevel gear in accordance with Klingelnberg

15.2.14 Angle modifications

In some less than ideal situations it may happen that the cutter tip cuts into any shaft pins that are located immediately next to the toothing. If this cannot be prevented by modifying either the design or the toothing data, the cutter tip level at the calculation point at d_m of the gear and pinion can be tilted by a slight angle ϑ_c from its intended position $\delta_{o1,2}$ towards the reference cone angle $\delta_{E1,2}$. See Figure 15.7.

15.2.15 Geometry details

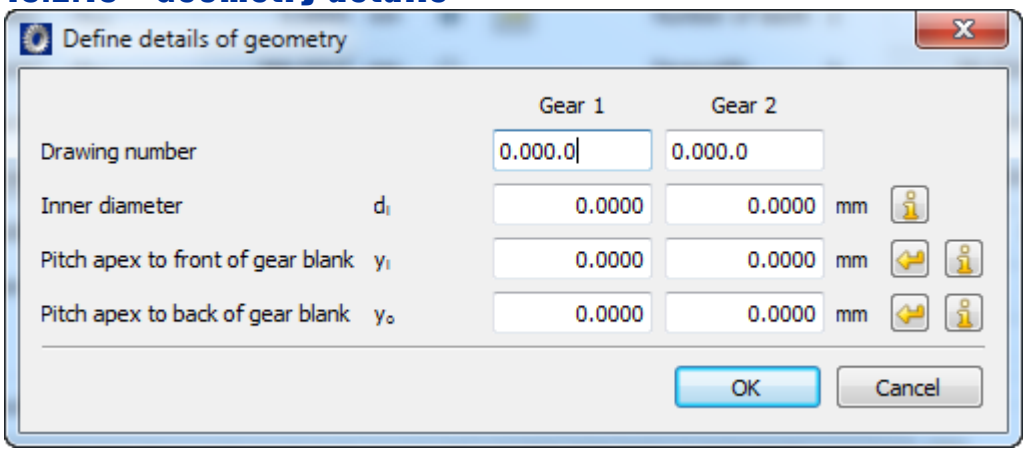


Figure 15.6: `Define details of geometry` dialog window

Click the `Details` button in the upper right-hand part of the Geometry area to open the `Define details of geometry` dialog window. You can enter these parameters here.

The following parameters are not described here:

- Inside diameter (see page II-273)

15.2.15.1 Reference cone apexes on the outside/inside of the blank part

The reference cone apex on the inside of the blank part is the distance from the reference cone apex to the front face of the unworked part, in the axial direction.

The reference cone apex on the outside of the blank part is the distance from the reference cone apex to the rear face of the unworked part, in the axial direction.

15.2.16 Manufacturing process

Tab. 15.10 shows the interrelationship between the manufacturing process and the achievable accuracy grade.

Process	**Achievable accuracy grade** **(ISO 17485, DIN 3965)**
Milling only	8
Lapping	7

Skiving	6
Grinding	6

Table 15.10: Interrelationship between manufacturing process and achievable accuracy grade

15.3 Type

You will find a drop-down list for the type on the top left of the screen, in the Geometry tab.

As you can see in Figure 15.7, the following bevel gear types are available for selection:

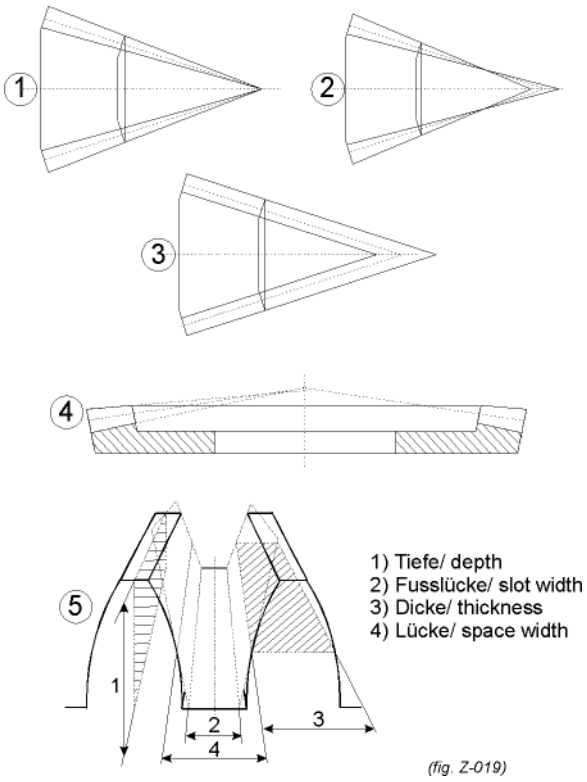


Figure 15.7: Basic types of bevel gears

- Default, Figure 1 (tip, reference and root cone apex in one point)
  The geometry is calculated according to ISO 23509. No hypoid offset is taken into account. If you click the sizing button, the cone angle is calculated so that the crossing points of the gear axes meet each other (similar to the standard specified in ISO 23509, Annex C.5.2). In this case, the tip clearance is not constant. Typical applications include form-forged, injection molded, or sintered, bevel gears, such as differential bevel gears.
- Standard, Figure 4 (reference and root apex in one point)
  The geometry is calculated according to ISO 23509. No hypoid offset is taken

into account. If you click the sizing button, the cone angle is calculated as specified in the standard (ISO 23509, Annex C.5.2). The tip clearance is constant.

- Default, Figure 2 (tip, reference and root cone apex NOT in one point)
  The geometry is calculated according to ISO 23509. No hypoid offset is taken into account. If you click the sizing button, the cone angle is calculated as specified in the standard (ISO 23509, Annex C.5.2). However, you can also input values for the addendum angle and the dedendum angle manually. The cone angle of the counter gear is calculated by taking a constant tip clearance into account.

- Constant slot width, Figure 2 (Gleason)
  The geometry is calculated according to ISO 23509. You can perform this calculation either with offset (method 1, hypoid gears) or without offset (method 0, spiral bevel gears). If you click the Sizing button, the cone angle is calculated with a "constant slot width" (ISO 23509, Annex C.5.2). The tip clearance is constant. Gap 2 in Figure 5 does not change. A typical application of this is a ground bevel gear toothing produced in the "completing" process (duplex), where the pinion and the bevel gear are each ground in one work step. This process requires machines that have an additional helical motion.

- Modified slot width, Figure 2 (Gleason)
  The geometry is calculated according to ISO 23509. You can perform this calculation either with offset (method 1, hypoid gears) or without offset (method 0, spiral bevel gears). If you click the Sizing button, the cone angle is calculated with a "modified slot width" (ISO 23509, Annex C.5.2). Gap 2 in Figure 5 changes. A typical application is the 5-section process, where the pinion is manufactured with 2 different machine settings, and a modified slot width is consequently created. The bevel shape is often also referred to as a TRL (Tilted Root Line). The toothing can be either ground or lapped.

- Constant tooth height, Figure 3 (Klingelnberg)
  The geometry is calculated according to ISO 23509. You can perform this calculation either with offset (method 3, hypoid gears) or KN 3028 and KN 3029, or without offset (method 0, spiral bevel gears). The tip and root cone are parallel. Applications are the cyclo-palloid® process and the palloid process. After quench hardening, the cyclo-palloid® toothing can either be hard-cut (HPG, HPG-S) or lapped. Palloid toothing is characterized by an evolvent tooth length form with a constant normal module over the facewidth. After quench hardening, the toothing is usually lapped.

- Constant tooth height, Figure 3 (Oerlikon)
  The geometry is calculated according to ISO 23509. You can perform this calculation either with offset (method 2, hypoid gears) or without offset (method 0, spiral bevel gears). The tip and root cone are parallel. Applications are Oer-

likon processes such as Spiroflex and Spirac. After quench hardening, the toothing is usually lapped.

15.3.1 Converting or inputting Gleason toothing data

The "System Data" group in the "Geometry" tab has a selection list (drop-down list) in its top left-hand corner. If the Gleason variant with "Constant slot width" or "Modified slot width" is selected here, the conversion and plus buttons are active. Using these two buttons, you can enter data in Gleason definition mode.

Select the conversion button if a Gleason data sheet is present. You can then input the data in the window as shown in Figure 14.3 and then click Calculate. When the calculation is finished, the Report and Accept buttons will be enabled. Click on the Report button to generate a partial report. If a complete report is needed, click on the  button in the main window. Click the Accept button to transfer the data to the main window.

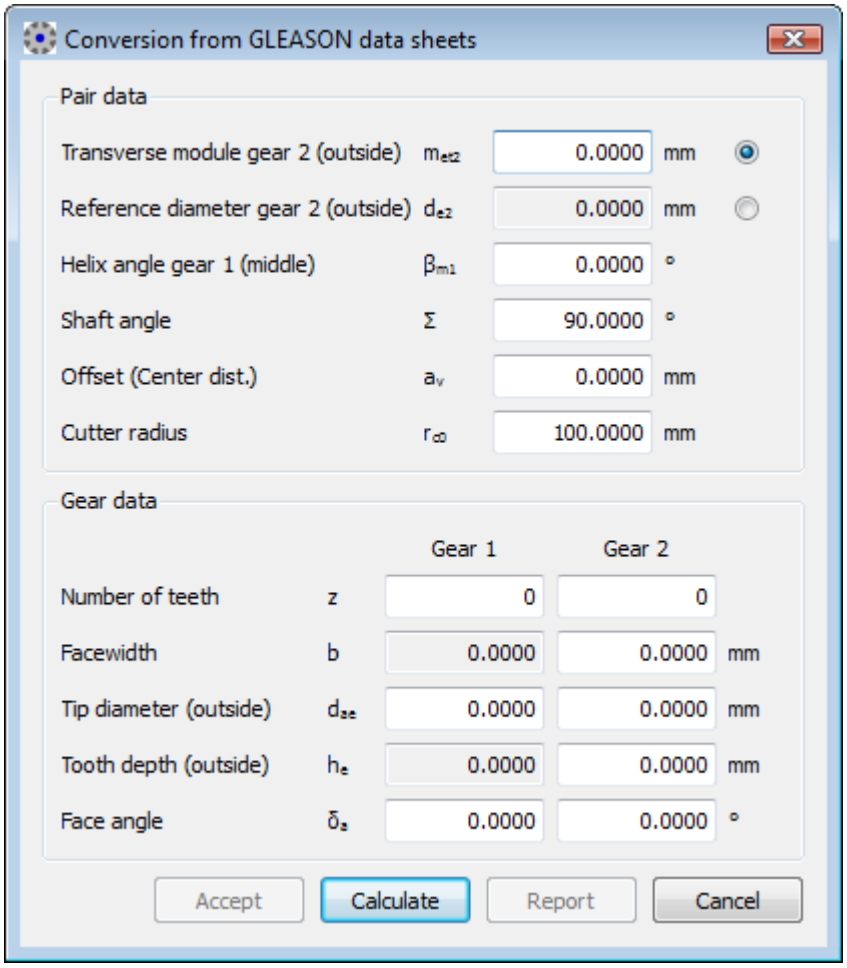


Figure 15.8: Conversion from Gleason data sheets

If you click the Plus button, the dialog window shown in Figure 15.9 appears. Here, you can enter raw bevel gear data according to the Gleason method. The results of the geometry calculation will not match the Gleason dimension sheet exactly, but are close enough for a calculation of strength according to ISO 10300 (or AGMA, DIN).

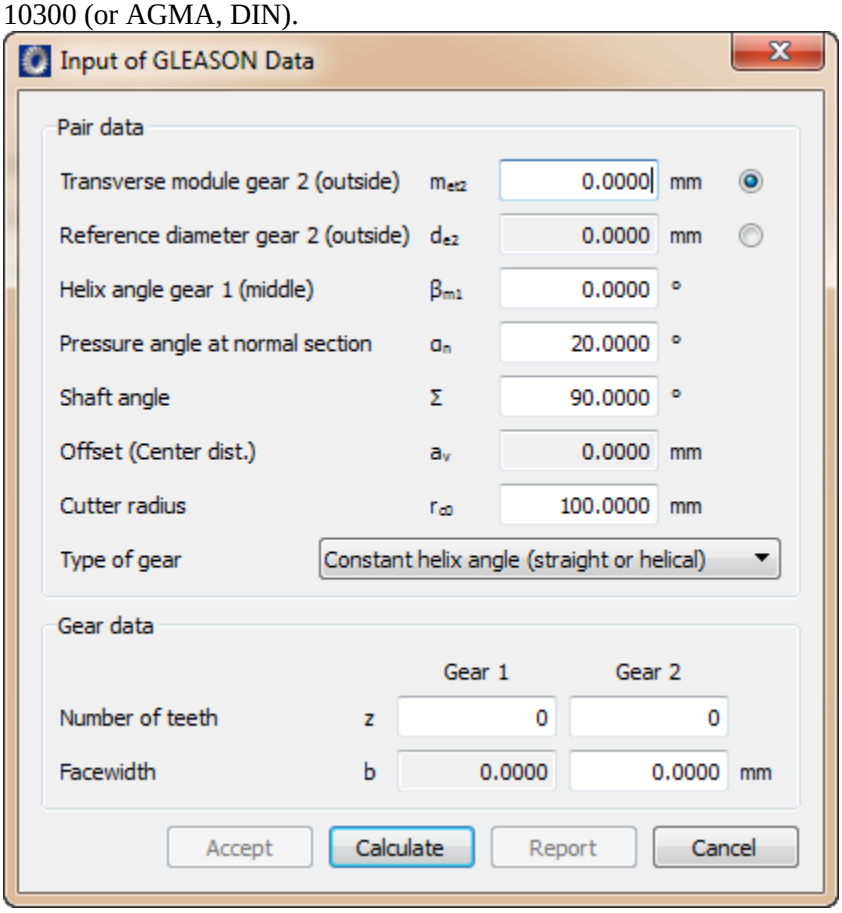


Figure 15.9: Inputting Gleason data

In the "Gear type" selection list, you can select one of a number of different Gleason methods (the default setting is to use a constant helix angle):

1. `Constant helix angle (straight or helical)`
   A constant helix angle represents a bevel gear with a constant helix angle. You can modify the mean Helix angle to compare the geometrical data with Zerol standard design if required. If you click the Accept button to close the dialog, the calculation is usually performed with the selection "Default, Figure 4 (part and root apex in one point)".

2. `Duplex (constant slot width)`
   The term "duplex" refers to bevel and hypoid gears that are manufactured with a constant slot width across the entire tooth length of both gears. These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle across its width. If you selected Duplex (constant root gap) and then clicked the Accept button to close the dialog, the calculation is usually performed with a "Constant slot width".

3. `Spiral toothing, default (modified slot width)`
   These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle in the axial direction. This gear type is described as having a "Modified slot width". If you select this gear type, and then click on Accept, the calculation is usually performed with a "Modified slot width". In this case, the root gap of the gear pair is constant over the entire tooth length, and any gap modifications are performed on the pinion.

4. `Zerol "Duplex taper"`
   This is a Zerol design (see Zerol), but a root angle variation is performed to achieve duplex dimensions. If you select Zerol duplex, and then close the dialog by clicking the Accept button, the calculation is usually performed with the "Constant root gap" selection.

5. `Zerol "standard"`
   The Zerol standard is a gear pair with a spiral angle of less than 10° in the middle of the facewidth, with a continuously changing spiral angle in the axial direction. In this case, the inner spiral angle is mostly negative. To ensure the program can take into account the change across the tooth length, a value of b=0.001 is assumed for the case b=0. If you close the dialog by clicking the Accept button, the calculation is usually performed with a "non-constant tooth gap".

15.4 Manufacture

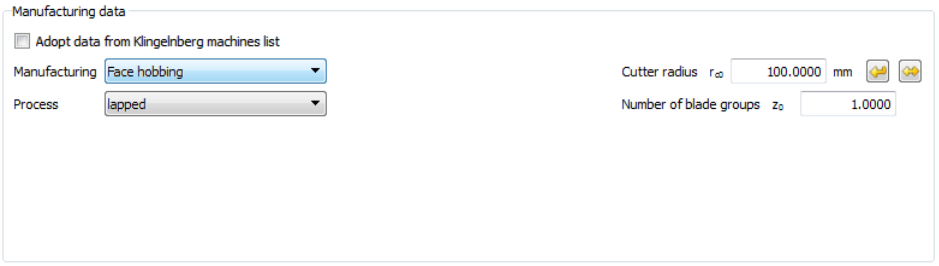


Figure 15.10: `Manufacture` input window

The process used to manufacture spiral teeth bevel gears is closely linked to this process. There are two basic processes used here. The arc of circle toothing process (traditionally known as the Gleason process) and the continuous face hobbing (traditionally referred to as the Klingelnberg and Oerlikon process). For more details see under Calculation process.

15.4.1 Cutter radius

In the case of spiral teeth bevel gears, the size of the cutter radius r_{c0} influences the beveling of the flanks and therefore also the properties of the pair of bevel gears. This effect applies both to the position of the contact pattern and the strength, and must be taken into account when calculating the transverse coefficient K_{Fa} according to ISO 10300.

NOTE

This parameter is not present if you use the Klingelnberg method to calculate strength. In that case, you select the cutter radius together with the machine type.

15.4.2 Number of blade groups

The number of blade groups describes the number of cutter blade groups on the cutter head used to manufacture bevel gears with spiral teeth and, when face hobbing is in use, this number, together with the cutter radius, influences the bevel of the tooth length. You must enter the number of blade groups as defined in ISO 23509, Annex E or as specified in the manufacturers' instructions.

15.5 Rating

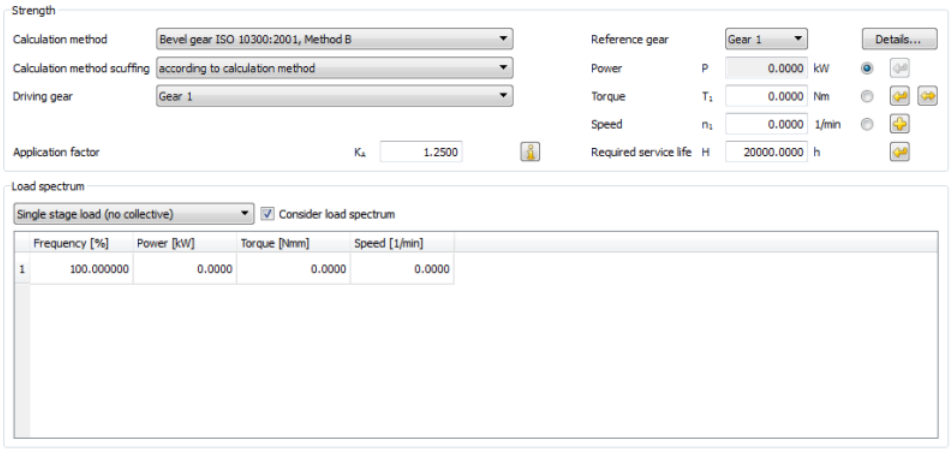


Figure 15.11: `Load` input window

15.5.1 Methods used for strength calculation

You can select the following methods:

1. **Bevel gears, only geometry calculation**
   Does not calculate strength. This method only calculates the geometric values, such as the path of contact.

2. **Bevel gears, static calculation**
   The strength calculation for cylindrical gears (see section "Calculation methods" on page II-281) is implemented here.

3. **Differential gear, static calculation**
   The static calculation method is used for differential gears. The calculation is performed with the greatest circumferential force F_1 or F_2, see Figure 15.12

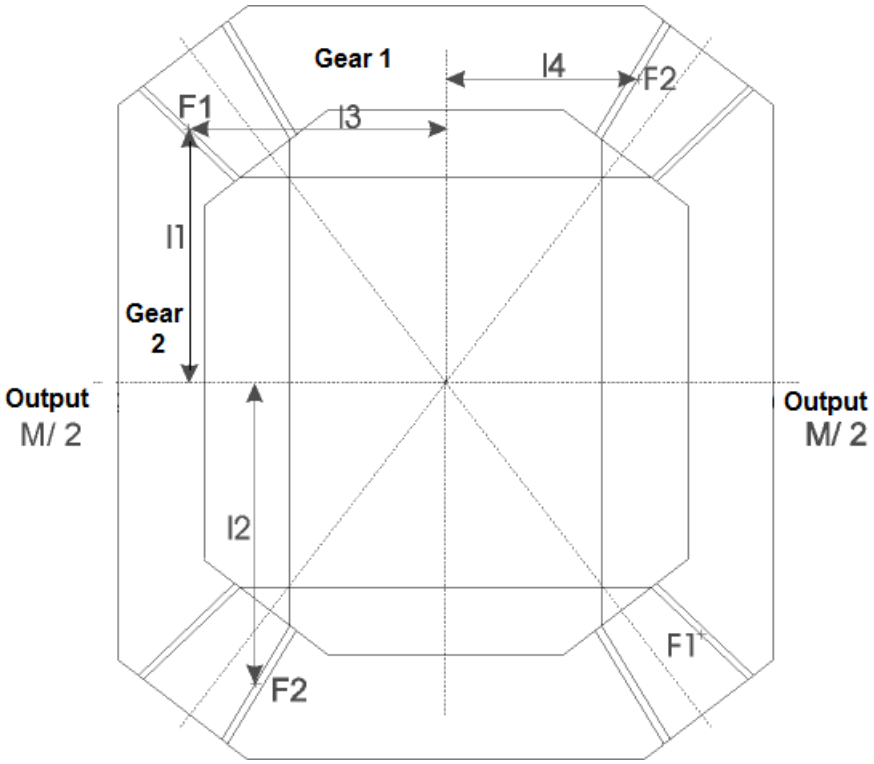


Figure 15.12: Bevel gears in differential gears

$$M = 2 \cdot F_1 \cdot l_1 + 2 \cdot F_2 \cdot l_2 \qquad (13.1)$$

$$F_1 = F_2 \cdot \frac{l_4}{l_3} \qquad (13.2)$$

$$l_1 = \sqrt{R_{m1}^2 - \left(\frac{d_{m1}}{2}\right)^2} - 0.5 \cdot \tan(\delta_1) \cdot (d_{am1} - d_{m1}) \qquad (13.3)$$

$$l_2 = \frac{d_{am2}}{2} \qquad (13.4)$$

$$l_3 = \frac{d_{am1}}{2} \qquad (13.5)$$

$$l_4 = \sqrt{R_{m2}^2 - \left(\frac{d_{m2}}{2}\right)^2} - 0.5 \cdot \tan(\delta_2) \cdot (d_{am2} - d_{m2}) \qquad (13.6)$$

4. **Bevel gears, ISO 10300, Method B (C)**
   ISO 10300, Parts 1, 2, 3: Load capacity calculation for bevel gears.

5. **Bevel gears as specified in ISO/CD 10300 (2011)**
   Draft version of the next edition of ISO 10300.

6. **Bevel gears as specified in AGMA 2003-B97 or AGMA 2003-C10**
   ANSI/AGMA 2003-B97 or AGMA 2003-C10: Rating the Pitting Resistance and Bending Strength of Generated Straight Bevel, Zerol Bevel and Spiral Bevel Gear Teeth

7. **Bevel gears**
   as specified in DIN 3991, Parts 1, 2, 3, 4: Load capacity calculation for bevel gears.
   This calculation is usually performed as defined in Method B, and the tooth form factor is calculated with Method C.

8. **Bevel gears Klingelnberg KN 3028/KN 3030**
   This calculation is the same as the Klingelnberg in-house standards KN 3028 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

9. **Bevel gears Klingelnberg KN 3025/KN 3030**
   This calculation is the same as the Klingelnberg in-house standards KN 3025 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

10. **Bevel gears Plastic**
    This calculates the equivalent cylindrical gear pair (see also DIN 3991). Here the calculation is performed according to Niemann/VDI/VDI-mod. in the same way as the cylindrical gear calculation (see page II-262) is finished.

11. **DNV41.2, Calculation standard for ships' engines**
    The Det Norske Veritas calculation guideline [93] for ships' engines corresponds in principle to ISO 10300 (root, flank) and ISO 13989 (scuffing). However, it does have some significant differences, especially where S-N curves (Woehler lines) are concerned. These differences are detailed in our kisssoft-anl-076-DE-Application_of_DNV42_1.pdf information sheet, which is available on request.

12. **Hypoid bevel gears in accordance with ISO 10300**
    Hypoid bevel gears as specified in ISO 10300 with the suggested extension in accordance with FVA411. ISO 10300 (2001 edition) applies to bevel gears. The feasibility of extending the calculation method to include

hypoid gears is under discussion. In Germany, it has been proposed to extend this calculation method as part of the FVA411 research project. This method is already documented in the "Bevel gear" manual produced by Klingelnberg [87]. The method specified in FVA411 is only slightly different from the proposed ISO 10300 extension, which has not yet been published.

13. **Hypoid bevel gears, geometry only**
14. **Hypoid bevel gears, in accordance with Klingelnberg KN3029/KN3030**
    This calculation is the same as the Klingelnberg in-house standards KN 3029 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.
15. **Hypoid bevel gears, in accordance with Klingelnberg KN3026/KN3030**
    This calculation is the same as the Klingelnberg in-house standards KN 3026 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

NOTE

You will find more information about the strength calculation in accordance with Klingelnberg in section 15.5.

15.5.2 Required service life

You enter the required service life directly in this input field.

Click the  button to size this value. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated according to ISO 6336-6 with the Palmgren-Miner rule. In the range of endurance limit, you can select a modified form of the S-N curve (Woehler lines) as an alternative to ISO 6336. The system service life and the minimum service life of all the gears used in the configuration is displayed. You can size the service life using the  button either with or without defining a load spectrum (see page II-308). You will find more detailed information about defining load spectra in section 14.19 (see page II-308).

15.5.3 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-471) can be maintained. Click the button next to the power input field to apply a frequency distribution for power, torque and speed in the `Define load spectrum (see page II-308)` window. Click the button on the right of the Speed input field to open the `Define sense of rotation` window, in which you can specify the direction in which the bevel gear rotates in accordance with Figure 16.6 on page II-552.

15.5.4 Strength details

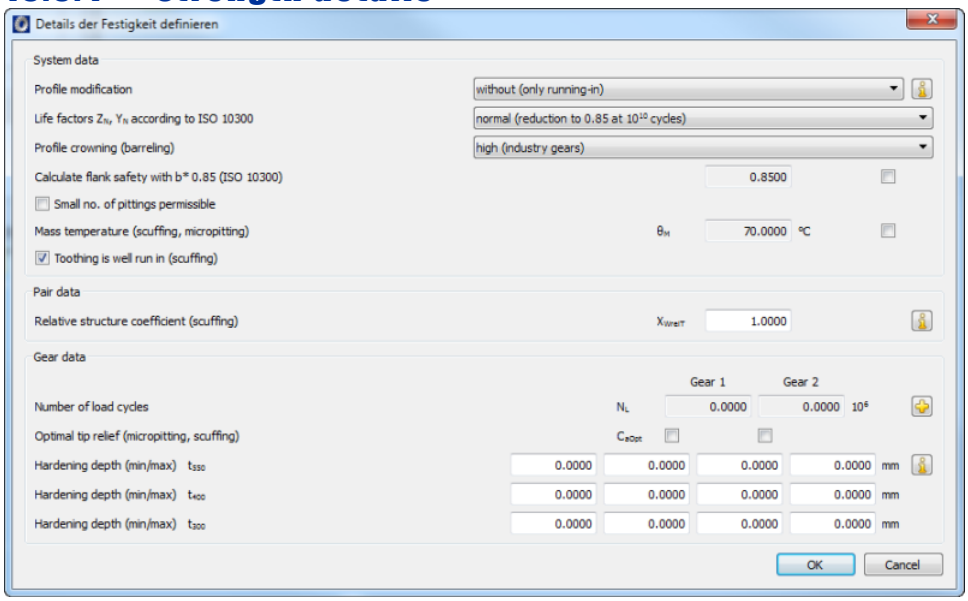


Figure 15.13: Dialog window: `Define details of strength`

Click the `Details...` button on the upper right-hand part of the `Strength` area to open the `Define details of strength` dialog window.

The parameters described in other places are:

- Fatigue strength (see page II-293)
- The input of the type of `profile crowning (barreling)`: influences the calculation of the contact area (only for ISO 10300) and the load distribution coefficient Z_{LS}. The 2001 edition of ISO 10300 does not yet use this variant.

- Small pitting (see page II-300)
- Relative structural factor (see page II-301)
- Number of load cycles (see page II-302)
- Alternating bending factor (see page II-315)

15.5.4.1 Profile modification

Modifying the profile of bevel gears is unusual. Please contact the manufacturer first to see whether it is feasible to do so. The run-in amount specified in ISO 10300 is the most commonly used.

15.5.4.2 Calculate flank safety with 0.85*b (ISO 10300)

Flank safety as defined in ISO 10300 is calculated with the length of the contact line on the middle of the tooth height l_{bm}. Select this checkbox to perform this calculation with a modified width instead of using the one defined in ISO 10300

$$b' = (faktor) \cdot b$$

.

The usual contact pattern width is 0.85*facewidth (for example, as specified by DIN 3991). If you have sufficient experience, you can modify this value.

NOTE

You can only input this value if you are using the ISO10300 calculation method.

15.5.4.3 Load spectra with negative elements

Load spectra with negative load bins (T < 0 and/or n < 0) can also be calculated as follows (this is only applied to bins whose alternating bending factor is YM=1.0).

IMPORTANT:

A load spectrum is considered to be negative if the non-working flank is placed under load.

Coefficient for	Coefficient	Flank under load	Actual load bin

torque	for speed		
+	+	Working flank (*)	Evaluated as positive
+	-	Working flank (*)	Evaluated as positive
-	+	Non-working flank	Evaluated as negative
-	-	Non-working flank	Evaluated as negative

(*) Working flank as entered in the Strength tab

Table 14.10: Evaluation of a load bin, depending on the prefix operator

Under "`Details`" in the "Strength" section of the "`Load`" tab, you can select the following:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Consider only positive load bins
  - Consider only negative load bins
  - Check both cases and document the unfavorable case
- For calculating the tooth root safety
  - Evaluate all negative load bins as positive (as up to now)
  - For negative load bins, increase root stress by 1/0.7
  - Increase bending stress for positive load bins by 1/0.7
  - Check both cases and document the more realistic case

15.5.5 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 15.4 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 10300, ISO 6336, DIN 3990 and DIN 3991.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**

uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 15.4: Assignment of operational behavior → application factor

15.6 Reference profile

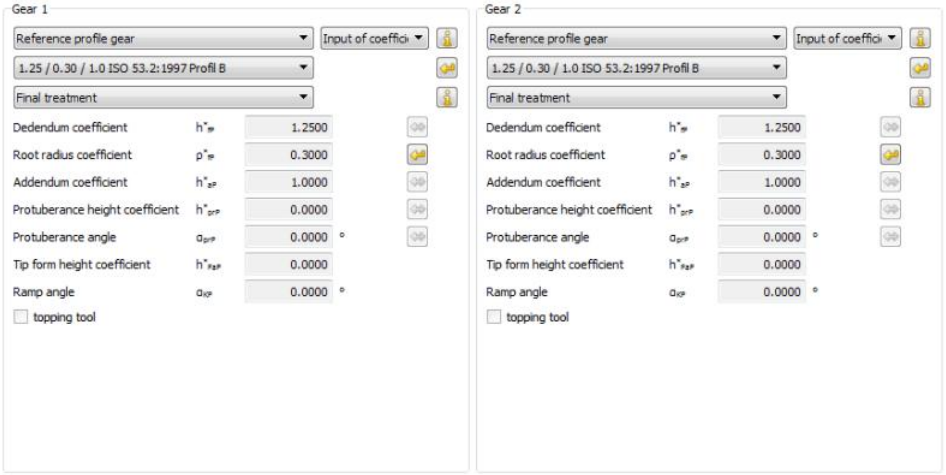


Figure 15.14: Reference profile tab

15.6.1 Default values for tip clearance

The tip clearance for spiral bevel gears is usually 0.2 to 0.3 times the mean normal module. However, a greater amount of clearance is used for toothing that is manufactured with tilt. This prevents the tooth tip intersecting with the root of the opposing gear.

Default values are (as stated in the "Kegelräder" book produced by Klingelnberg [87]):

"Gleason, modified slot width" process:	0.3
"Gleason, constant slot width" process:	0.35
"Klingelnberg, Palloid" process:	0.3
"Klingelnberg, Cyclo-Palloid" process:	0.25
"Oerlikon" process:	0.25

15.6.2 Default values for addendum coefficients

The addendum coefficient is usually 1.0.

15.7 Contact analysis

In the Bevel and Hypoid gears module, the contact analysis calculates the path of contact under load for bevel gears with straight, helical, and spiral teeth. Hypoid gears are not supported.

A pair of bevel gears with a substitute cylindrical gear toothing are approximated for the analysis. Each one of the gears in this cylindrical gear pair has a number of teeth that varies across the facewidth, an operating pitch diameter, and a helix angle (spiral toothing). For a more detailed description of the theory of contact analysis, refer to the Cylindrical gear contact analysis (see page II-411) section and [69].

Axis alignment

The contact analysis takes into account the specified H, G and V misalignment, and also the direction of the torsion. As for contact analysis for cylindrical gear pairs, the deformation of the shafts can also be taken into account when calculating bevel gears (see the Taking into account shaft bending (see page II-340) section). When the deformation of the shafts is taken into account, the equivalent H, G and V misalignment is also documented in the contact analysis report.

15.8 Modifications

The `Modifications` input window is where you define the profile and lead correction, and a tip chamfer or a tip rounding, and specify the depth of immersion of the grinding wheel.

You will find data about the tip chamfer, and also the profile and tooth trace modifications, in Kap.14.7 (see section "Modifications" on page II-361).
For technical, manufacturing-related reasons, not all modifications which could be used for cylindrical gears are used for bevel gears.

Tip alterations are used for bevel gears, in special cases, to ensure sufficient tip clearance is achieved. The way the data to be entered is defined is shown in Figure. 15.15.

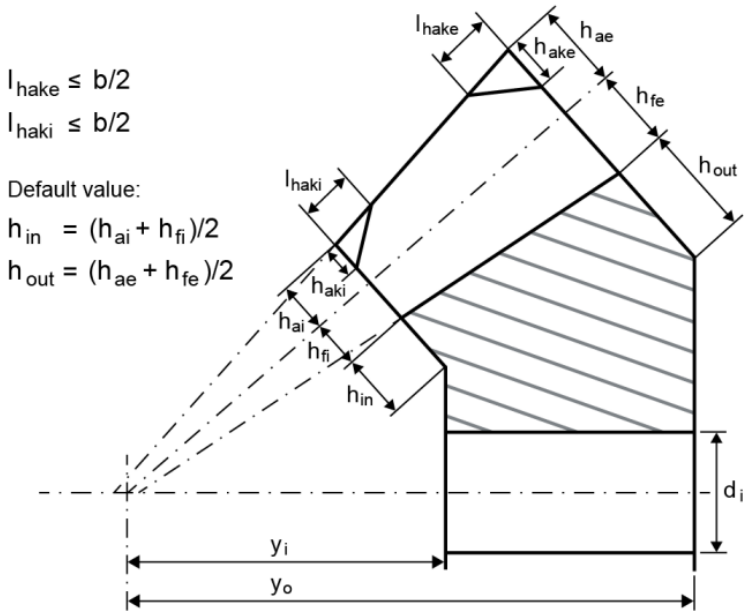


Figure 15.15: Tip alterations for bevel gears

Use the tip alteration sizing button for the inside face to generate a suggested value for a constant tip width with 0.2*normal module, corresponding to length bk (tip alteration width), as specified in the Klingelnberg in-house standard. To do so, the calculation according to Klingelnberg KN 3028 is required.

The length and width values of the gear body can be modified on the inside and

outside (for the 3D view). Then, click the conversion button to generate modifications with a parallel axis bearing. This then opens a window, in which you can use the sizing function for the external face to generate a proposed value for the maximum possible height of the modification, h_ake, up to the external cone length diameter. The maximum possible length of modification l_hake is limited to half the facewidth, so that the unchanged tooth height remains in the tooth middle, in the 3D model of the bevel gear.

Use the "`Modified blank`" option to generate special forms of the gear body (Figure 15.16)

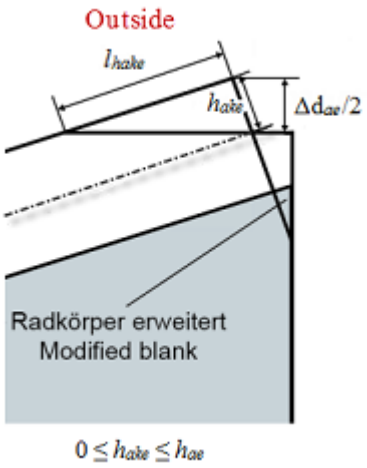


Figure 15.16: Gear body modification on the external face of the bevel gear

You can also click the conversion button to perform a conversion for the inside face. This then opens a window, in which you can use the sizing function for the inside face to generate a proposed value for the maximum possible height of the modification, h_aki, up to the internal cone length diameter.

Select the "`Modified blank`" option to generate special forms of the gear body (see Figure 15.17). The sizing button in this window for "`Distance in axial direction to the cone tip yaimod`" calculates yai in such a way that the bevel gear body is given a shape according to Δyai=0, (see Figure 15.17 below, left or right.

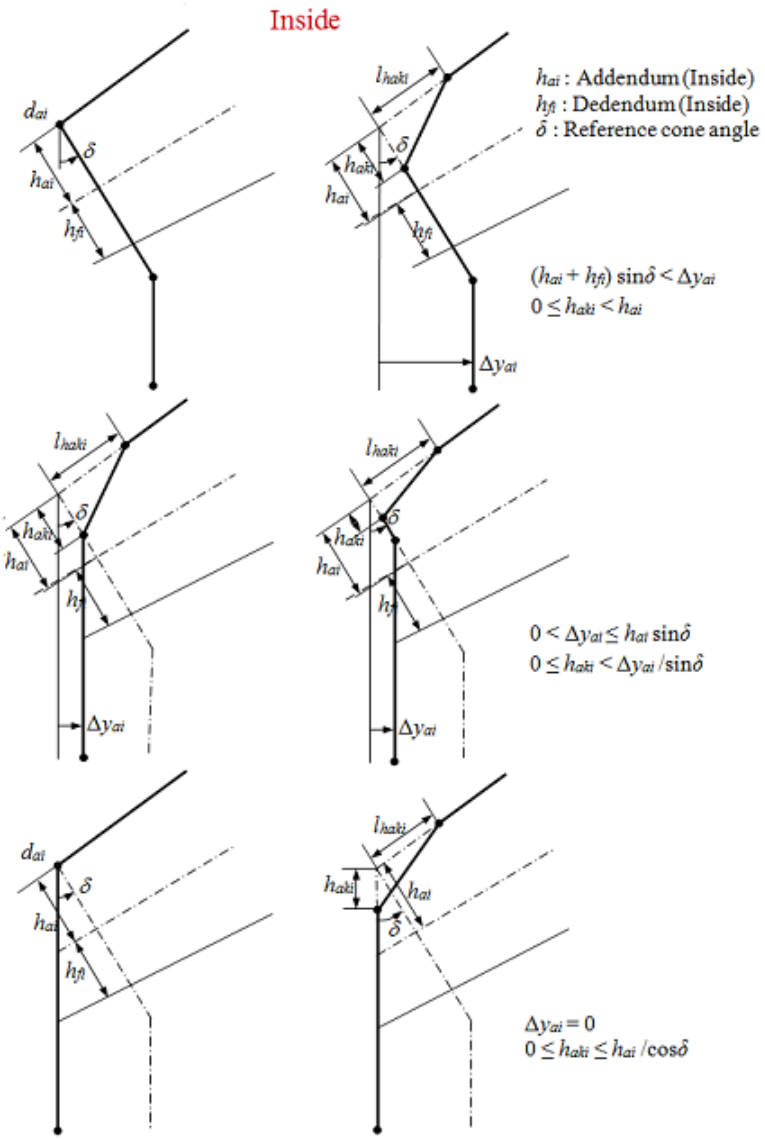


Figure 15.17: Gear body modification on the inside face of the bevel gear

15.9 Rough sizing

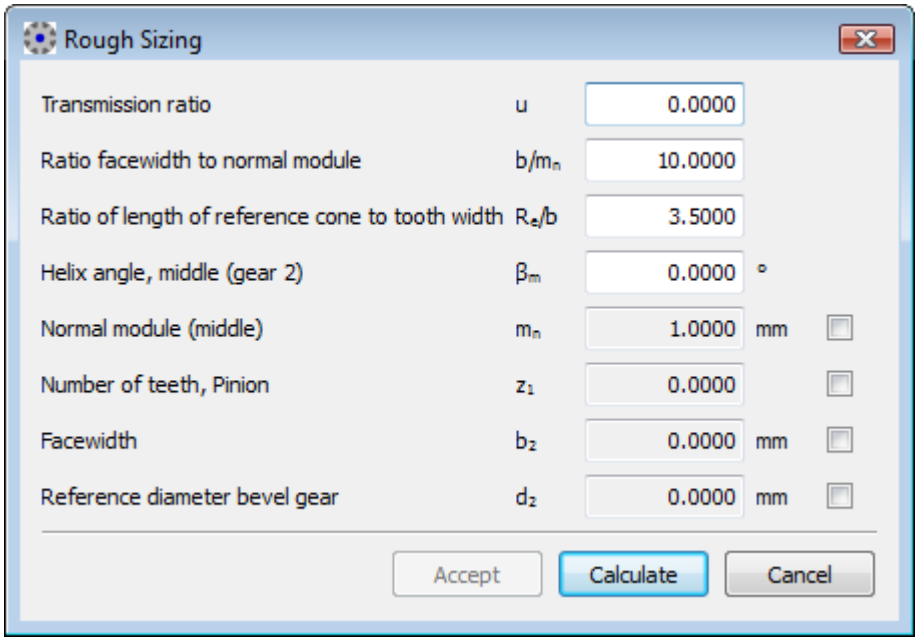


Figure 15.18: Dialog window: `Rough sizing`

The method used to size bevel and hypoid gears, in accordance with suggestions from technical literature [Kegelräder, pub. Klingelnberg] provides geometrically satisfactory sizing recommendations for gear pairs. This proposal does not provide sufficiently precise solutions to the problems of achieving the required safeties against tooth fracture and pitting, because it is based on values gathered through years of experience. If you verify gear teeth that have been dimensioned in accordance with this method, you may discover certain deviations from the required safety values.

However, you can easily achieve these safety levels by simply changing the module and the facewidth.

15.9.1 Facewidth ratio

Depending on how and where a gearbox is to be used, the facewidth should be in a specific ratio to the cone distance and correspond to the following values:

Light and medium-heavy load gear units for machines and vehicles	$3.5 \leq (Re/b) \leq 5.0$
Heavy load gearboxes for machines and vehicles	$3.0 \leq (Re/b) \leq 3.5$

15.9.2 Module ratio

The normal module mn should be in a ratio to the facewidth b within specific limits which can only be exceeded (or not reached) for exceptional reasons:

surface hardened bevel gears at risk of tooth fracture	$7 \leq (b/mn) \leq 12$
At risk of pitting or through hardened or not hardened	$10 \leq (b/mn) \leq 14$

15.10 Fine sizing

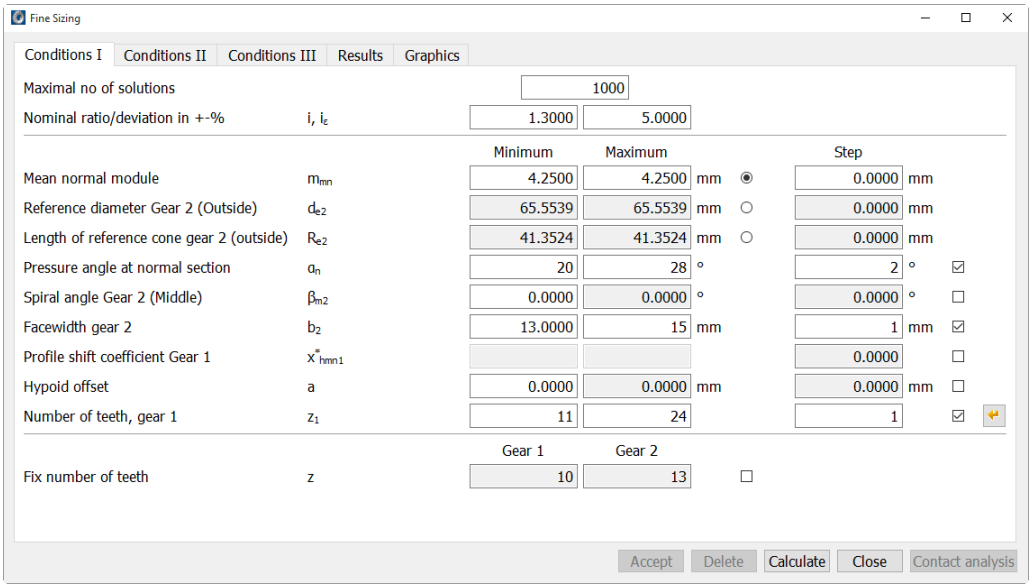


Figure 15: `Conditions I` tab in the `Fine Sizing` window

To start the `Fine Sizing` process, click the `Calculation` menu and select the `Fine Sizing` option or click the  icon in the tool bar.

If you input a nominal ratio, a center distance, intervals for the module and helix angle, and the pressure angle, the system calculates and displays all the possible suggestions for the number of teeth, module, helix angle and profile shift. It also shows the deviation from the nominal ratio, the specific sliding and the contact ratio. This module can also be used to size planetary stages and three gear chains.

All the variants found by this process can be evaluated by a wide range of different criteria (accuracy of ratio, weight, strength, etc.).

Depending on your requirements, limits can also be set on the most important parameters (minimum number of teeth, tolerated undercut, etc.). In addition to creating text reports detailing the solutions and the summary, the summary can also be displayed as a graphic.

15.10.1 Required entries in the standard tabs

Before you start the fine sizing process, you must enter the following data correctly in the `Basic data` or `Geometry` and `Strength` standard tabs to ensure the calculation returns the results you require.

Geometry:

- Reference profile
- Type: Standard, Gleason, Klingelnberg

Strength:

- Materials
- Power/Speed
- Application factor
- Service life
- Lubrication

15.10.2 Conditions I

15.10.2.1 Maximal no of solutions

If the program finds more than the specified number of solutions, you see a warning message and an appropriate note is entered in the report.

NOTE

You should only perform a final evaluation after all the possible solutions have been displayed. Otherwise you run the risk that the optimum solution will not be displayed.

15.10.2.2 Normal module (middle), reference diameter, length of reference cone

Use the three available options to vary and restrict the gear size.

15.10.3 Conditions II

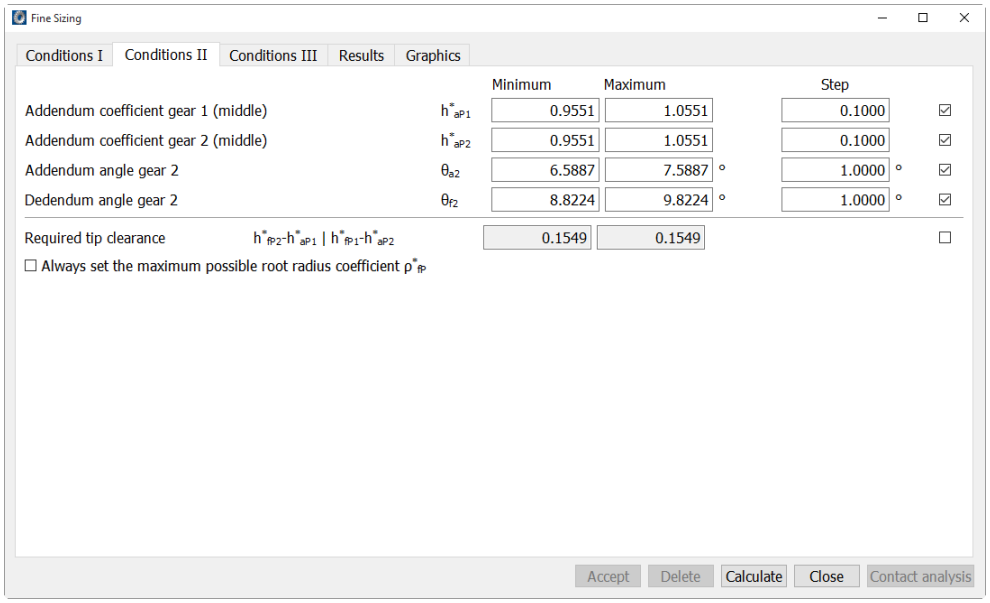


Figure 15: `Conditions II` tab in the `Fine Sizing` window

You can define more parameters in the `Conditions II` tab.

15.10.3.1 Addendum coefficient gear 1 (middle), addendum coefficient gear 2 (middle)

You can vary the reference profile of the bevel gears by changing the addendum coefficients of gear 1 and gear 2. You can then calculate the dedendum coefficients of the counter gear (gear 2 and gear 1) by specifying the addendum coefficient and the "Required tip clearance".

15.10.3.2 Addendum angle gear 2, dedendum angle gear 2

By varying the addendum and dedendum angle on gear 2 you can then vary the tooth height along the facewidth. To calculate the addendum and dedendum angle on the counter gear (gear 1), input a constant tip clearance (parallel tip cone and root cone for counter gear).

Restrictions due to gear type: You cannot vary the cone angle for gear types for which the angle cannot be changed. You cannot vary the addendum angle and root angle of the "default, Figure 1" type. Although the face angle can be varied for the "Standard, Figure 4" type, you cannot vary the root angle. The addendum angle and

root angle cannot be varied at all for the "Constant tooth height, Figure 3" types.

NOTE

These options for varying the parameters in Conditions II are useful for differential bevel gears, which are characterized by major geometric variations during manufacture. However, do remember that the usual conditions must be met when using conventional manufacturing methods such as Klingelnberg or Gleason.

15.10.4 Conditions III

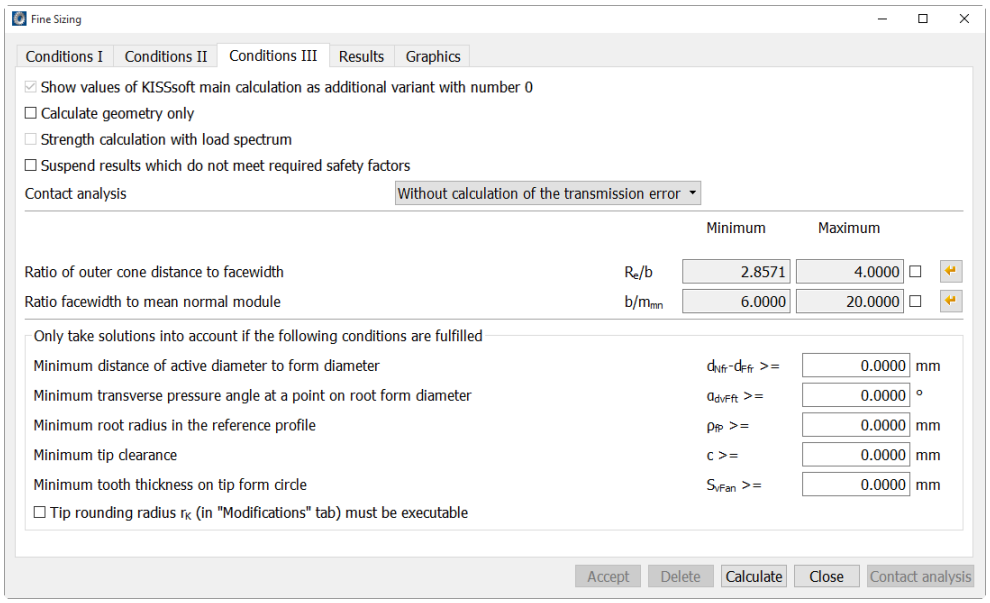


Figure 15: `Conditions III` tab in the `Fine Sizing` window

You can define more parameters in the `Conditions III` tab.

1. **Show values of KISSsoft main calculation as additional variant with number 0**

The toothing data displayed in the KISSsoft Basic tab can also be displayed as a variant with number 0 (table and

graphic). However, **the data at the start of the fine sizing process must be consistent before this can happen.**

2. Calculate geometry only

If you select this method, no strength calculation is performed

3. Strength calculation with load spectrum

Before you can perform calculations with a load spectrum, you must set a load spectrum in the main KISSsoft window BEFORE starting fine sizing,

and run the load spectrum calculation once (to ensure the data is consistent). In this case, when you start the fine sizing process,

you are prompted to confirm whether the calculation is to be performed with a load spectrum. The flag in the window only shows whether (or not) a load spectrum is being used.

You cannot change the flag.

4. Suspend results which do not meet required safety factors

Variants which do not meet the predefined minimum safety levels (see `Calculation > Settings > Required safeties`) will be

rejected.

5. Transmission error

If the "Calculation of the transmission error" option is selected, contact analysis is performed for every variant.

During the contact analysis for transmission error, the default settings, except for the coefficient of friction, and accuracy, are used to prevent the calculation

from behaving in any unusual way. Input the settings in the main program, in the "Contact analysis" tab. You can also

specify the accuracy of the calculation. However, we strongly recommend you select "medium" or "low" to reduce the processing time. Therefore,

the transmission error in fine sizing may not be exactly the same as you get in the contact analysis, depending on the

selected settings.

The default values are as follows;

- Calculation for: right flank
- Torque for gear A: not considered
- Torque for gear B: not considered
- Partial load range for calculation: 100 %
- Center distance: Mean center distance allowance
- Single pitch deviation: 0 mm

Then, the results list shows;

- Transmission error (PPTE)
- Medium wear on the tooth flank (delwn1, delwn2)
- Maximum flash temperature (theflamax)
- Variation in bearing force (VarL)

The calculation time increases significantly with the transmission error calculation option. For this reason, we recommend you limit the number of variants to be calculated

before you start the calculation

15.10.4.1 Ratio of length of reference cone to facewidth

A standard sizing characteristic value for bevel and hypoid gears is "Ratio of length of reference cone to facewidth". If this flag is set, solutions which lie outside this range are rejected.

NOTE

Make this range relatively small when calculating bevel gears with spiral teeth. Select a larger range for differential gear bevel gears.

15.10.4.2 Ratio facewidth to normal module

A standard sizing characteristic value for bevel and hypoid gears is "Ratio of facewidth to normal module". Small values result in modules that tend to be large and sizings that are optimized for root strength. If this flag is set, solutions which lie outside this range are rejected.

NOTE

Make this range relatively small when calculating bevel gears with spiral teeth. Select a larger range for differential gear bevel gears

15.10.4.3 Only take solutions into account if the following conditions are fulfilled:

The user can also define other criteria to ensure unsatisfactory solutions are rejected. These values are calculated and checked on the substitute cylindrical gear toothing.

1. Minimum distance of active diameter to form diameter d_{Nf} - d_{Ff}

Meshing errors occur if the active root diameter is less than the root form diameter. This is where you input a minimum value for the distance between

the active root diameter and the root form circle, in other words, between the used and manufactured involutes. The value you input here is the

minimum difference between the two diameters. Only solutions greater than, or equal to, the input value are taken into account in the results view.

2. Minimum transverse pressure angle at a point on root form diameter alphafF

For differential bevel gears, a minimum profile angle in the transverse section is required to ensure the axial demoldability

of the material from the forging tool. Only solutions greater than, or equal to, the input value are taken into account in the results view.

3. Minimum root rounding radius in the reference profile rhofp

A minimum root rounding radius may be required for reasons of manufacturability (absolute value in mm). Only solutions greater than, or equal to,

the input value are taken into account in the results view.

4. Minimum tip clearance c

A minimum tip clearance may be required for reasons of manufacturability (absolute value in mm). This is compared with tip clearance c. Only solutions

with a tip clearance greater than, or equal to, the input value are displayed in the results view.

5. Minimum tooth thickness on tip form circle sFvan

The minimum tooth thickness on the tip form circle, sFvan, is critical for achieving the required tip rounding radius. The

This calculation takes into account the tip alterations from the "Modifications" tab. Only solutions with a tooth thickness

on the tip form circle that is greater than, or equal to, the input value are displayed in the results view.

The tooth thickness sFvan is checked in the center of the facewidth. If you want that the tooth thickness is also checked outside and inside, in the sections I and II,

select the "Additions for differential gears" in the module-specific settings.

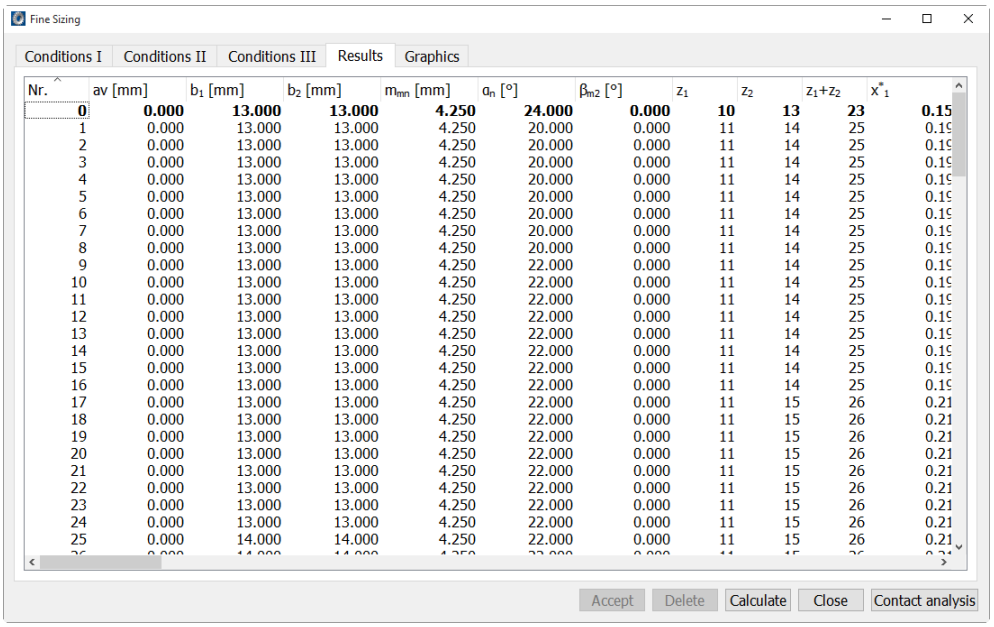
6. Manufacturing must be possible with tip rounding rK (in the "Modifications" tab).

Only solutions in which the tip rounding rK as defined by the entries in the "Modifications" tab can be executed are displayed in the results view.

NOTE

If "Module-specific settings – Differential gears" has been selected, these criteria are also checked in an "inside" and "outside" section. Only solutions which meet the predefined criteria are then taken into account.

15.10.5 Results



Nr.	av [mm]	b_1 [mm]	b_2 [mm]	m_{mn} [mm]	α_n [°]	β_{m2} [°]	z_1	z_2	z_1+z_2	x^*_1
0	**0.000**	**13.000**	**13.000**	**4.250**	**24.000**	**0.000**	**10**	**13**	**23**	**0.15**
1	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
2	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
3	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
4	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
5	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
6	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
7	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
8	0.000	13.000	13.000	4.250	20.000	0.000	11	14	25	0.19
9	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
10	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
11	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
12	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
13	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
14	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
15	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
16	0.000	13.000	13.000	4.250	22.000	0.000	11	14	25	0.19
17	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
18	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
19	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
20	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
21	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
22	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
23	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
24	0.000	13.000	13.000	4.250	22.000	0.000	11	15	26	0.21
25	0.000	14.000	14.000	4.250	22.000	0.000	11	15	26	0.21

Figure 15: `Results` tab in the `Fine Sizing` window

Click the `Report` button to open the editor and display a list of the best results. A brief description of the criteria used to evaluate the best variants is given here. Please note that these criteria are not relevant to every case, and only need to be queried in particular applications!

15.10.6 Graphics

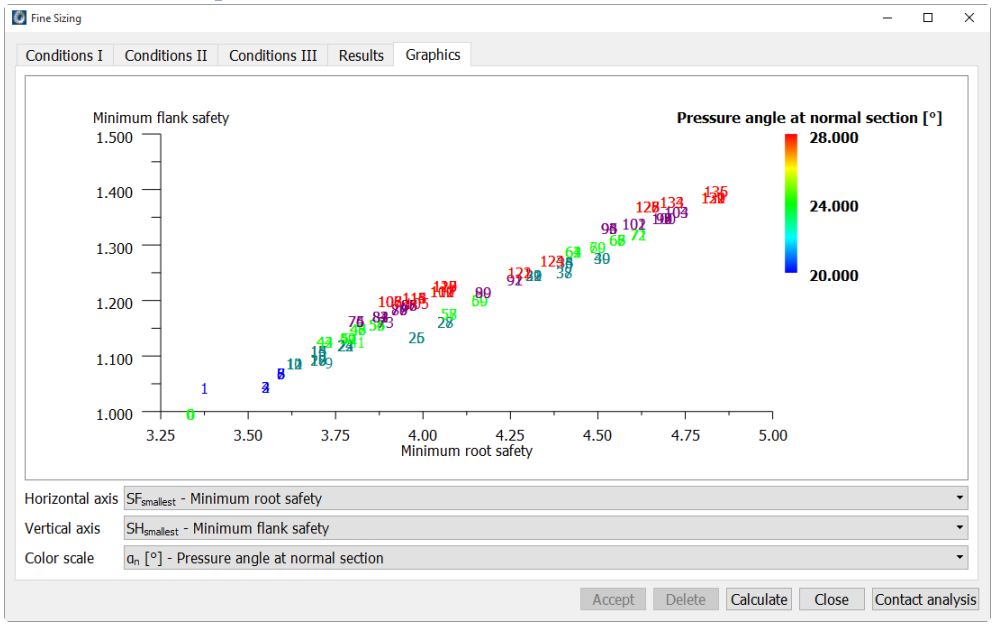


Figure 15: `Graphics` tab in the `Fine Sizing` window

The Graphics tab in the Fine Sizing window gives you a quick overview of the number of solutions. Three parameters can be displayed simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

15.11 Notes on calculations in accordance with the Klingelnberg standard

15.11.1 Bevel gears with cyclo-palloid® gear teeth

The geometry of bevel gears, their ease of manufacturing, and performing a strength calculation for them in accordance with the Klingelnberg cyclo-palloid® process.

As stated in the Klingelnberg KN 3028 in-house standard (geometry and manufacture) and KN 3030 standard (strength calculation), a complete calculation is performed for cyclo-palloid® toothing:

- Calculate machine distance for machine types FK41B, AMK400, AMK635, AMK855, AMK1602 with all corresponding cutters, cutter radii and numbers of starts. A warning is displayed if you select an incorrect machine type or cutter tip.
- You can specify any shaft angle, or angle modification here.
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, outside), checks on cut back, undercut space, calculation of profile shift for balanced sliding, checks on backwards cut, checking and calculating the necessary tip reduction on the inside diameter, profile and overlap ratio, tooth form factor and stress correction factor.
- Calculation of all toothing dimensions.
- Calculation of pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion) with all modifications in the in-house standard KN 3030.

15.11.2 Hypoid gears with cyclo-palloid gear teeth

The geometry of hypoid gears (bevel gears with offset), their ease of manufacturing, and performing a strength calculation for them in accordance with the Klingelnberg process.

As stated in the Klingelnberg KN 3029 in-house standard (geometry and manufacture) and KN 3030 standard (strength calculation), a complete calculation is performed for cyclo-palloid toothing:

- Calculate machine distance for machine types FK41B, KNC40, KNC60, AMK855, AMK1602 with all corresponding cutters, cutter radii and numbers of starts. A warning is displayed if you select an incorrect machine type or cutter tip.

- You can use any value as the shaft angle, angle modification, and ressure angle for the driving and driven flank.
- Overall geometry with calculation of the facewidths, modules (inside, middle, outside), spiral angle (inside, outside), undercut boundary, calculation of gap widths, checks on backwards cut, checking and calculating the necessary tip reduction on the inside diameter, profile and overlap ratio, tooth form factor and stress correction factor either for the tension (driving) or coast (driven) flank.
- Calculation of all toothing dimensions.
- Calculation of pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion for the replacement crossed axis helical gear wheel) with all modifications in the in-house standard KN 3030.

15.11.3 Normal module ranges for Klingelnberg machines (cyclo-palloid)

Machine	**Cutter radius** r_{∞}	**Normal module** m_{mn}	
FK41B	25	0.25 ...	1.6
	30	0.25 ...	1.6
	40	0.25 ...	1.6
AMK400	55	1.1 ...	4.0
	100	2.4 ...	5.2
	135	3.5 ...	8.0
	170	3.5 ...	13.0
AMK635	55	1.1 ...	4.0
	100	2.4 ...	5.5
	135	3.5 ...	8.0
	170	6.5 ...	13.0
	210	7.0 ...	13.0
AMK855	135	3.5 ...	8.0
	170	6.5 ...	13.0
	210	7.0 ...	15.5
	260	7.0 ...	15.5
AMK1602	270	8.0 ...	17

	350	14.0 ...	25.0
	450	17.0 ...	34.0
KNC25	30	0.5 ...	5.5
	55	0.5 ...	5.5
	75	0.5 ...	5.5
	100	0.5 ...	5.5
KNC40	30	1.0 ...	1.6
	55	1.1 ...	4.0
	75	2.0 ...	4.5
	100	2.4 ...	5.5
	135	3.5 ...	8.0
KNC60	75	2.0 ...	4.5
	100	2.4 ...	5.5
	135	3.5 ...	8.0
	170	6.5 ...	14.0

Table 15.11: Normal module ranges for Klingelnberg machines

15.11.4 Bevel gears with palloid toothing

Calculate the geometry and strength of bevel gears using the Klingelnberg process.

A complete calculation for palloid toothing is performed in accordance with the Klingelnberg KN 3025 in-house standard (Geometry, Edition No. 10) and KN3030 (strength calculation).

- Taking into account palloid cutter dimensions by including a smaller diameter dK and cutter cut length SF, you can also input special cutters here
- A warning is issued if the cutters do not cover the basic crown wheel at either the inner or outer end of the tooth
- You can select any shaft angle, or angle modifications
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, middle, outside), checks on profile shift for balanced sliding and undercut

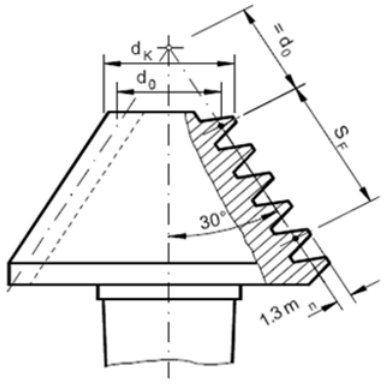
space, checking and calculating the necessary tip shortening on the inside diameter, profile and overlap ratio, tooth form factor and stress correction factor

- Calculation of all toothing dimensions
- Calculate forces for contact pattern core for reference cone length Rpr and Rm
- Calculate pitting, tooth root and scuffing load capacity (as defined by the integral temperature criterion for all modifications in Klingelnberg standard KN 3030 (taking into account the forces of cone distance Rpr)

NOTE

The forces at bevel length Rm are used for the transfer to KISSsys, to ensure that forces can be calculated independently of the toothing procedure. However, including the theoretical contact pattern core in the Klingelnberg in-house standard is very difficult to implement in the manufacturing process.

15.11.5 Definitions and dimensions of standard cutters for palloid toothing



$$d_0 = d_k - 2 \cdot 1.3 \cdot m_n \cdot \cos 30°$$

	Row A		Row B		Row C	
m_n	d_k	S_F	d_k	S_F	d_k	S_F
1	14	13	18	16	18	19
1.25	17	16	21	20	21	24
1.50	20	19	24	24	24	29
1.75	23	22	27	28	27	33
2	26	26	30	32	30	38
2.25	29	29	32	36	32	43
2.50	32	32	35	40	35	48
2.75	34	35	37	44	37	52
3	36	38	39	48	39	57
3.25	38	41	42	52	42	62
3.50	40	44	44	55	44	66
3.75	42	48	46	59	46	71
4	44	51	49	63	49	76
4.25	46	54	51	67	51	81
4.50	48	57	53	71	53	85
5	52	63	58	79	58	95
5.50	56	70	63	87	63	104
6	60	76	67	95	67	114
6.50	64	82	72	103	72	123
7	68	88	77	110	77	132
7.50			81	118		
8			86	126		

Figure 15.14: Dimensions of standard cutters

15.11.6 Minimum safeties

We recommend you use the following minimum safeties:

Application	Minimum safeties
Flank	1.1 ... 1.2
Root	1.5 ... 1.6
Scuffing	1.8 ... 2.0

Table 15.12: Recommended minimum safeties

15.11.7 Surface roughness at tooth root

Treatment	Roughness [mm]
through hardened	0.016
lapped	0.016
hard-cut	0.008

Table 15.13: Surface roughness values

15.11.8 Accuracy grade bevel gears

Treatment	Quality number
through hardened	7
lapped	7
hard-cut	6

Table 15.14: Accuracy grade for bevel gears

15.11.9 Characteristic number

The product of the lubrication, speed and roughness factor $Z_L Z_V Z_R$ for different surface treatments is shown in Table 15.15:

Treatment	Characteristic number $Z_L Z_V Z_R$

through hardened	0.85
lapped	0.92
hard-cut	1.0

Table 15.15: Characteristic number $Z_L Z_V Z_R$ for different surface treatments

NOTE

You will find a similar definition in ISO 10300-2:2001, Section 14.4. Here the characteristic number is also dependent on the defined level of roughness R_z.

15.11.9.1 Single normal pitch deviation

This is calculated according to DIN 3965.

15.11.9.2 Meshing stiffness

The contact stiffness is assumed to be constant.

$$c_g = 20 \frac{N}{mm \cdot \mu m}$$

15.12 Settings

In the `Calculation` menu you will find the `Settings` option. Click this submenu to open the `Module-specific settings` window. From here, you can access the tabs listed below to input other calculation parameters (the following parameters not described here (see page II-456)).

15.12.1 General Information

During the mounting process, you can modify the mounting distance to achieve additional backlash. You can also specify how much additional backlash you require with $\Delta j*$ (enter this as a coefficient in the module). The required axial displacement for the integral pinion shaft $\Delta\alpha 1$ and the gear shaft $\Delta\alpha 2$ is then calculated according to ISO/ST 22849. The additional backlash that would be achieved by entering a predefined modification to the mounting distance is also calculated.

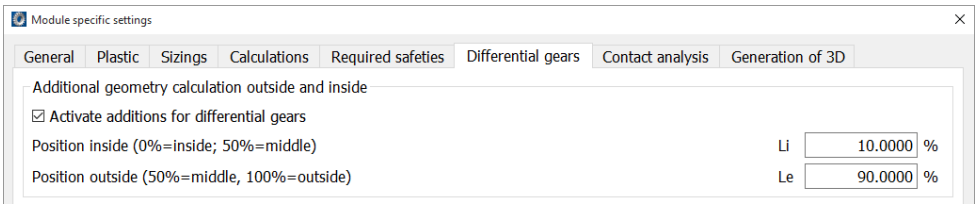
15.12.2 Calculations

15.12.2.1 Coefficient of friction for hypoid gears

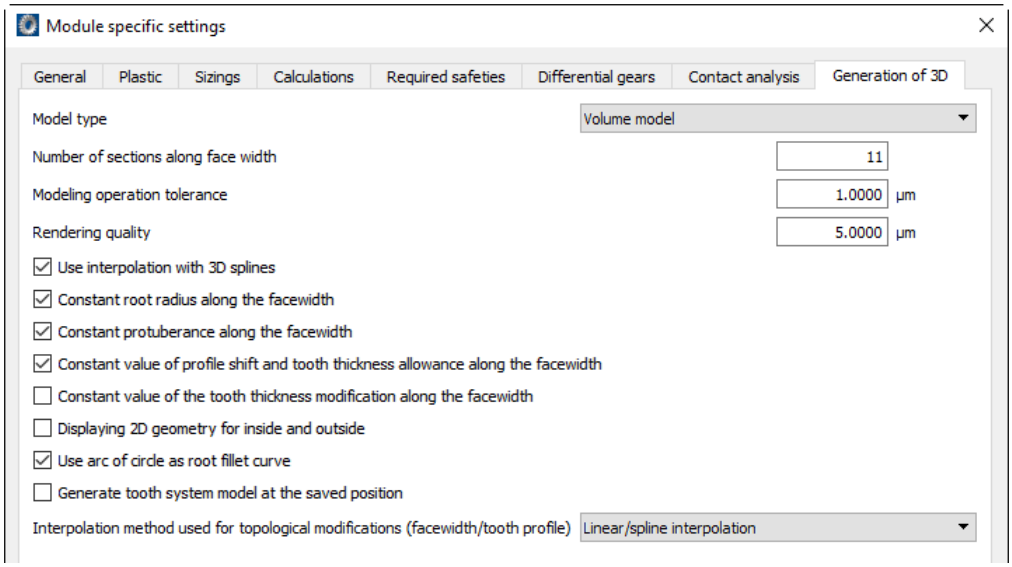
Due to longitudinal sliding, hypoid gears have more power loss than spiral bevel gears. For this reason, the calculation of toothing forces in KN 3030 takes the coefficient of friction account. If necessary, you can enter the size of the coefficient of friction in the Module-specific settings.

15.12.3 Differential gears

If the extensions for differential gears are enabled, the geometry parameters are calculated at positions Li and Le. The data for the equivalent cylindrical gear toothing at these two positions is then also documented in the report. The tip alteration can then also be applied up to underneath the cone length.



15.12.4 Helpful information about the Generation of 3D model tab



<Figure 15: `Generation of 3D model` in `Module-specific settings`

NOTE:

For more precise instructions about these entries, please contact KISSsoft Support and request the document KISSsoft-anl-068-E-3D Geometry of Spiral Bevel Gear.pdf.

15.12.5 Factors

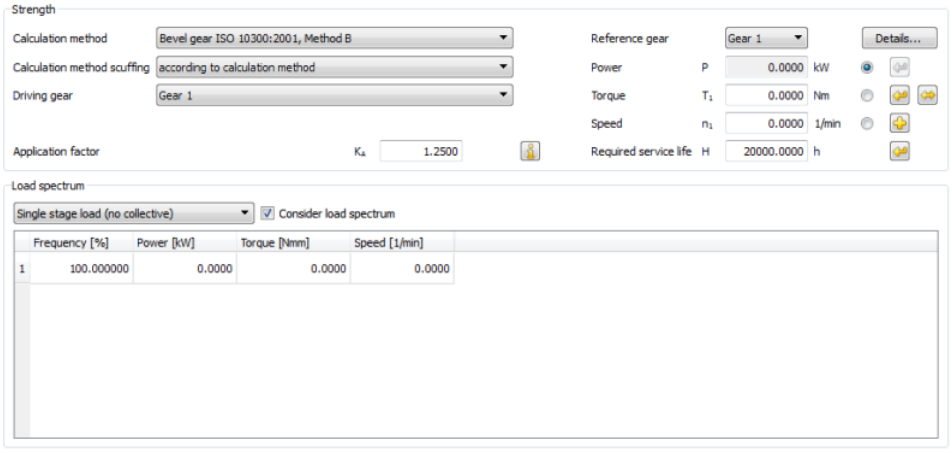


Figure 15.10: `Load` input window

15.12.5.1 Bearing application factor

Tables 15.5 ÷15.7 show the bearing type → bearing application factor for different standards.

Support for pinion and ring gear	**Bearing application factor**		
	a	**b**	**c**
both on both sides	1.00	1.05	1.20
one on both sides, one floating	1.00	1.10	1.32
both floating	1.00	1.25	1.50

a	:	Contact pattern in the gearbox tested under full load
b	:	Contact pattern in the gearbox tested under part load
c	:	Contact pattern only tested in specific tests

Table 15.5: Bearing application factor according to ISO 10300

Support for pinion and ring gear	**Bearing application factor**
both on both sides	1.10
one on both sides, one floating	1.25
both floating	1.50

Table 15.6: Bearing application factor according to DIN 3991

Support for pinion and ring gear	**Bearing application factor**
both on both sides	1.10
one on both sides, one floating	1.10
both floating	1.25

Table 15.7: Bearing application factor according to AGMA 2003

The face load factors $K_{H\beta}$,$K_{F\beta}$ and $K_{B\beta}$ are calculated as follows from the bearing application factor $K_{H\beta be}$ as defined in the standard:

$$K_{H\beta} = K_{F\beta} = K_{B\beta} = 1.5 \cdot K_{H\beta be} \quad (15.7)$$

15.12.5.2 Dynamic factor

To calculate the dynamic factor K_v, as defined by Klingelnberg, use the coefficient K_1 either for preliminary calculations based on the planned manufacturing method (lapped, HPG) or on the basis of the derived accuracy grade (see also Klingelnberg standard KN 3030, Table 5.2-1 or 5.2-2).

15.12.5.3 Bevel gear factor at flank and root

To calculate the strength of bevel gears, you use virtual cylindrical gear toothing with equations that apply to strength calculation for cylindrical gears. The bevel gear factors are then used to correct the systematic differences in the calculation between cylindrical gears and bevel gears. These factors are defined in the corresponding standards.

Standard	**Bevel gear factor of flank Z_K**
ISO 10300	0.80
Niemann	0.85

Table 15.8: Bevel gear factor of flank Z_K as defined in the standard

Standard	**Bevel gear factor of root Y_K**
ISO 10300	is calculated, see Part 3 of the standard
Niemann	1.00

Table 15.9: Bevel gear factor of root Y_K as defined in the standard

15.12.6 Contact analysis

Calculation method contact stiffness: Here you can select either the calculation method defined by Weber/Banaschek [69] (dynamic stiffness analysis: default setting), the method defined in ISO 6336-1 Method B and Own input.

Single contact stiffness: If "Own Entry" has been selected as the contact stiffness calculation method, you can enter your own value for the single intervention stiffness.

Slices coupling factor: Linking factor used to link the individual slices of the discretized toothing model.

Border weakening factor: Border weakening factor for a weakening of stiffness on the edge of helical gear teeth.

Stiffness correction factor (according to Winter): Correction factor for Hertzian flattening as described in the experiments performed by Winter/Podlesnik [95].

Take into account plastic deformation: Use this setting to specify whether plastic deformation is to be taken into account in the contact analysis. If plasticity is to be taken into account, the maximum contact stress, calculated using the elastic contact theory, is reduced on the basis of the specified "Maximum permitted flank pressure". If the maximum elastic flank pressure is exceeded, the radii of the contact body are changed locally so that the resulting elastic i.e. contact stress matches this maximum value. Only a percentage rate of the new radii is used, on the basis of the specified "Weighting of the plastic deformation".

15.12.7 Display

Smooth results: This function uses a low-pass filter to smooth the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). By default, this function is deactivated, but can be used to smooth the results if they are affected by strong numerical noise.

Area analyzed on tooth depth: This defines the maximum area along the tooth height for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Area analyzed on facewidth: This defines the maximum area along the facewidth for evaluating the results (Hertzian pressure, tooth root stress on gear 1/2, safety against scuffing and safety against micropitting). This setting generates additional results and does not change the results of the contact analysis.

Chapter 16
Face Gears

Face gears are a special type of bevel gear. Although a face gear pinion is a normal cylindrical gear, it has a complex 3D-tooth form. Unlike a bevel gear, a face gear is absolutely unaffected by axial displacement. For this reason, face gears are much easier to assemble.

The KISSsoft `Face gears` calculation module calculates the geometry of pairs of straight or helical cylindrical gear pinions with face gears with offset and with any shaft angle Σ. In this case, the strength and 2D geometry are calculated for an offset of 0 mm and a shaft angle Σ=90°. In every other case, you can perform the pre-sizing with these restrictions and then add the required hypoid offset and shaft angle to the 3D volume model. In the `Geometry` docking window you can display the tooth form of a face gear for its inside, middle and outside diameter or for any number of sections all at the same time. You use this tool to check for undercut and pointed teeth on the inside or external diameter of the face gear. In the `Modifications` input window (tab), you will find the value/length of tip alteration at outside (inside) hake(i), lake(i) input fields, which contain additional parameters that will help you prevent pointed teeth occurring in the gear. The tooth form on a face gear is calculated by simulating manufacture with a pinion type cutter. The strength calculation is based on the use of established standards for cylindrical or bevel gears.

16.1 Underlying principles of calculation

A face gear has features in common with a curved rack. However, unlike this simplest of all gears, when sizing a face gear, engineers are always confronted with the restrictions posed by that very bending. As the tooth flank in a spur geared face gear must run parallel to one radius of the face gear - the contacting pinion has flanks parallel to its own axis - the immediate result of the theorem of intersecting lines is that the pressure angle must reduce from outside to inside. The equation shown in [3] can be regarded as the main formula used to size the geometry of face gears. To keep this as simple as possible, only a gear with spur gear teeth is considered here

$$d_2 = \frac{m_n \cdot z_2 \cdot \cos\alpha_n}{\cos\alpha_2} \tag{16.1}$$

with

d_2	diameter of face gear
m_n	normal module pinion
z_2	number of teeth on face gear
α_n	pinion pressure angle on the reference circle
α_2	pressure angle on face gear for diameter d_2

From this, you can, for example, define the pressure angle from the external diameter to the inside diameter. If the inside tooth flanks are steep, the involute will be short and only bear a small part of the tooth height. The risk of an undercut grow in the direction of the crown gear center. Any undercut here would further reduce the usable area. The result is a minimum inside diameter and a maximum external diameter, which limit the total facewidth of the face gear. This is where a face gear differs fundamentally from a bevel gear: whereas you can increase the facewidth on a bevel gear to enable it to transmit higher torques, strict limits are set here for face gear to cylindrical gear-pinion pairs. However, if you select the right axial offset b_v, i.e. by moving the facewidth middle compared to the reference circle, you can optimize the maximum permitted facewidth.

When sizing a face gear, it is a good idea to define a minimum and a maximum pressure angle and then the achievable inside and external diameter. If external conditions limit this diameter (this usually affects the external diameter), you can use the conversion in equation (16.1) to change the range available for the module.

$$m_{\min/\max} = \frac{d_{2,\min/\max} \cdot \cos(\alpha_{2,\min/\max})}{z_2 \cdot \cos(\alpha_n)} \quad (16.2)$$

In addition to having the figures to hand, you may find it helpful to view the teeth as a graphic in this situation.

The vast majority of applications use face gears with spur gears. However, face gears with helical teeth, when arranged correctly, do offer a number of benefits such as noise reduction and strength. Unfortunately, these benefits are offset by the problem that the tooth flanks are not symmetrical, i.e. the left flank no longer matches the right flank. In practice, this means that any undercut that occurs will happen earlier on one flank than on the other. This differences in the flanks also have a significant influence on strength, which results in a difference between the directions of rotation when the gear transmits power. However, if only one direction of rotation is used, such as for electrical tools, you can optimize the flank involved without having to take the effect on the rear flank into account.

Experience has shown that theoretical observations of geometry to decide which involute functions, lines, and arcs of circles, to use, to describe a tooth form, will sooner or later reach their limit. A tried and tested, and much more reliable, means of calculating tooth forms is to simulate the generation process or, even better simulate the manufacturing process. To do this, the trajectory of a point on the active surface of the tool is followed until its speed relative to the tool surface normally reaches a zero crossing (see Figure 16.1).

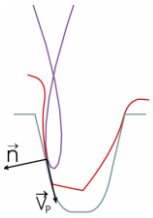


Figure 16.1: Spur curve (blue) of the pinion type cutter tool (red) on the face gear (green)

These points are potential points on the tooth form surface. You must then separate the actual points on the surface from the imaginary points at which the nominal speed also disappears but the underlying points are shown as being outside the material. One of the most difficult aspects of the procedure described here is how to separate the real points from the imaginary points. In addition to referring to the usual standard algorithms for classifying points in a level, you must also use

empirical approaches that use the known properties of the tooth form in order be sure of achieving a well-defined tooth form with sufficient safety. You can therefore match the data derived from calculating a 3D tooth form of a face gear with the data derived from generating using a pinion type cutter using a classic manufacturing method. By outputting the 3D body in IGES, STEP or SAT format you can then design the form in any CAD system. The face gears can then be manufactured in either an injection molding, sintered or precision forging process. However, 2D cross section view is much more suitable if you want to check a face gear for undercut or pointed tooth tips. This displays the inside, middle and outside of the face gear tooth form all at the same time. If you then rotate the gears step by step, you can check every aspect of the gear generation very precisely. If a tooth is pointed, or if the meshing ratios are not good enough, you must reduce the tooth height in the same way as you do for hypoid gears. To reduce the gear's sensitivity to errors in the axis alignment or the center distance, you can permit crowning on the tooth flank (tooth trace). You can generate this quite easily for face gears by using a pinion type cutter that has one or more teeth more than the pinion in the manufacturing process [79]. When you compare the tooth forms, you can see the effect the increased number of teeth on the pinion type cutter had on the generated tooth form. However, if the face gear has a large axial offset b_v, you can move the barreling to one side! In every axis section through the cylindrical gear, the face gear drive corresponds to a pinion-rack gear drive. Using the rack theory as a basis, you can therefore define the pressure angle, the lines of contact and the contact ratio in each section.

The examples in this section are based on a publication in [[50].

16.2 Basic data

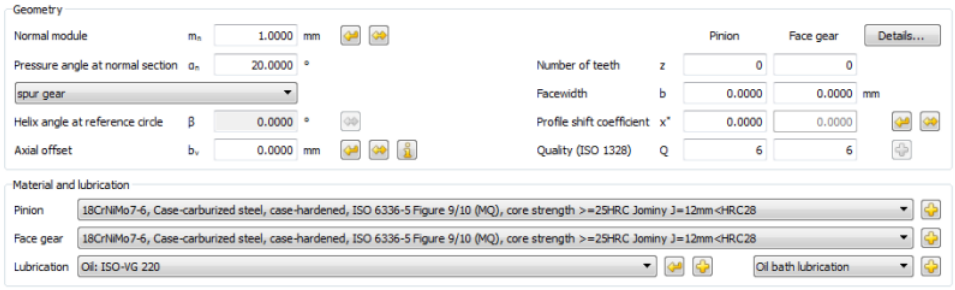


Figure 16.2: `Basic data` input window in the `Face Gears module`

16.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the  button to open a dialog window, in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

If you have already defined all aspects of the geometry of a face gear, the following message is displayed after you click the button:

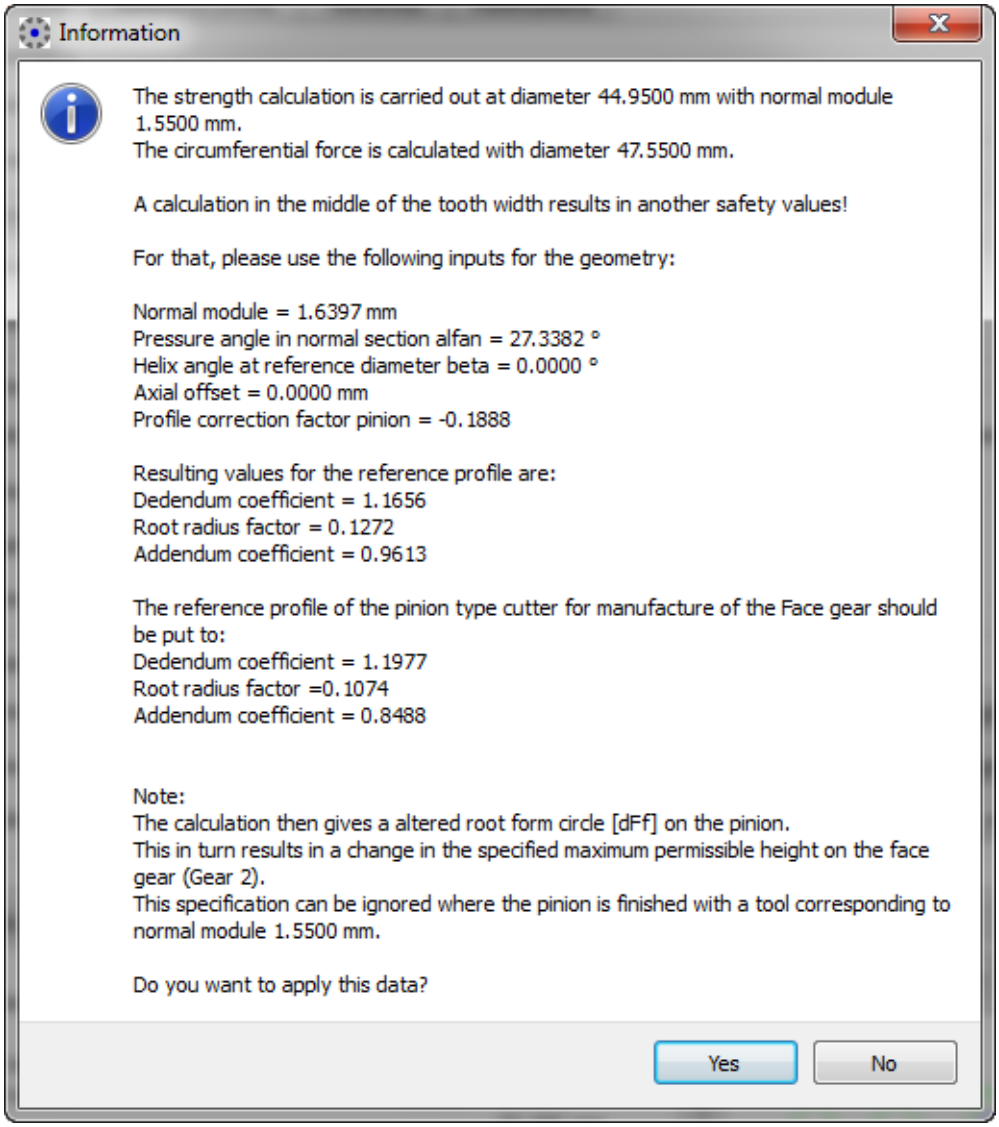


Figure 16.3: Information window for sizing the normal module

As part of the bevel gear calculation performed according to ISO 10300 or DIN 3991, the strength calculation is performed for the middle diameter of the face gear. If the axial offset $b_v <> 0$, the conditions for this type of calculation have not been met. For this reason, the button supports the conversion of normal module m_n and pressure angle α_n, to ensure that $b_v = 0$. Although this changes the root radius of the pinion, the shape of flank remains the same.

NOTE

We recommend you only use this conversion method when you perform the strength calculation. The conversion changes the module and you can no longer use the tool. This is why you must save your geometry data before you perform the conversion.

16.2.2 Pressure angle at normal section

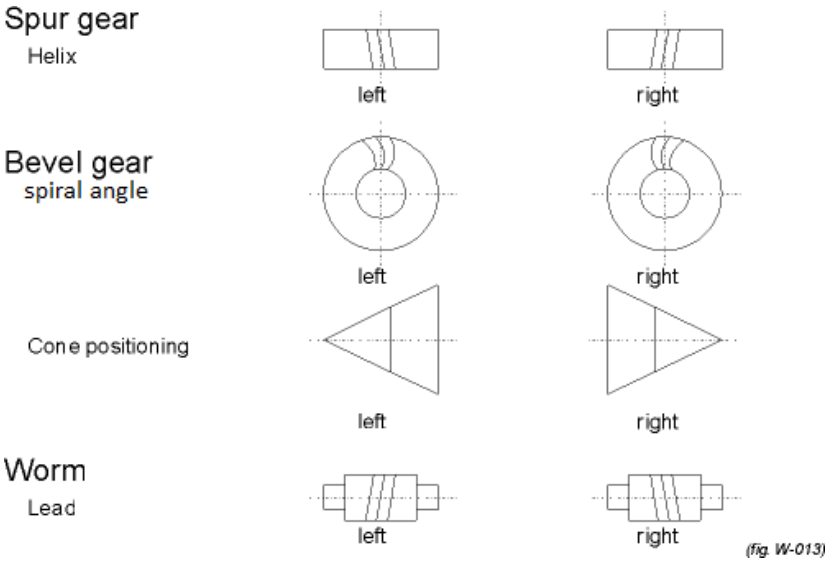
The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard meshings, the pressure angle is $\alpha_n = 20^o$. Smaller pressure angles can be used for larger numbers of teeth, to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

NOTE

The operating pressure angle α_{wt} changes across the width of the toothing.

16.2.3 Helix angle at reference circle

Enter the helix angle in [^{o}]. You can either convert this from the helix angle on the base circle β_b or from the helix angle at tip circle β_a by clicking the button in the `Convert helix angle` window. Helical gear teeth usually generate less noise than spur gear teeth. However, they also have the disadvantage that they involve additional axial force components.



(fig. W-013)

Figure 16.4: Helix angle

16.2.4 Axial offset

The axial offset is the distance of the pinion center from the middle diameter of the face gear.

Click the button to the right of the `Axial offset` input field to calculate the largest possible width of the face gear (see page II-557) b_2 and the corresponding axial offset b_v, so that the pressure angle lies within the predefined limits.

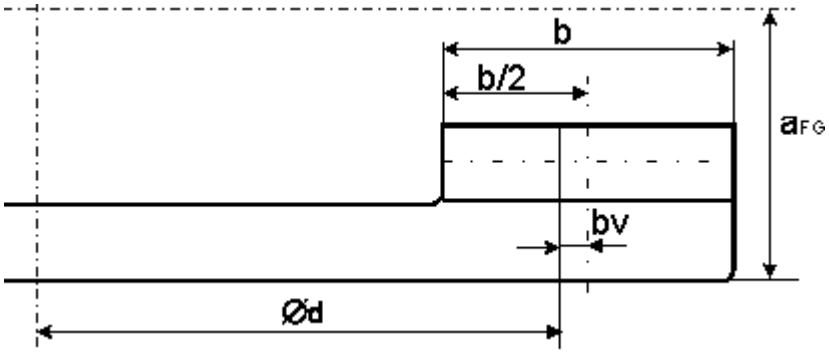


Figure 16.5: Axial offset of the face gear

16.2.5 Profile shift coefficient (on the pinion)

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth thickness is smaller and there is more danger of undercut. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

You can modify the profile shift according to different criteria. To do this, use the various sizing options in the `Sizing of profile shift` window. Here, click the relevant button for:

- **For undercut boundary**
- **For minimum top land per gear.**
  You can specify the minimum thickness of the top land in `Calculation >`

`Settings > General > Coefficient for minimum tooth thickness at the tip`.

NOTE

The pinion should have a reasonable high value for the tooth thickness at the tip because the pinion type cutter used to manufacture a face gear has a somewhat higher tip and, despite that, must not be permitted to become pointed.

Click the button, and KISSsoft will determine whether the profile shift coefficient (see page II-267) is taken from measured data or from values given in drawings.

16.2.6 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The accuracy grade according to ISO 1328 is very similar to that in DIN 3961 or AGMA 2015.

Achievable qualities are shown in Table 16.6.

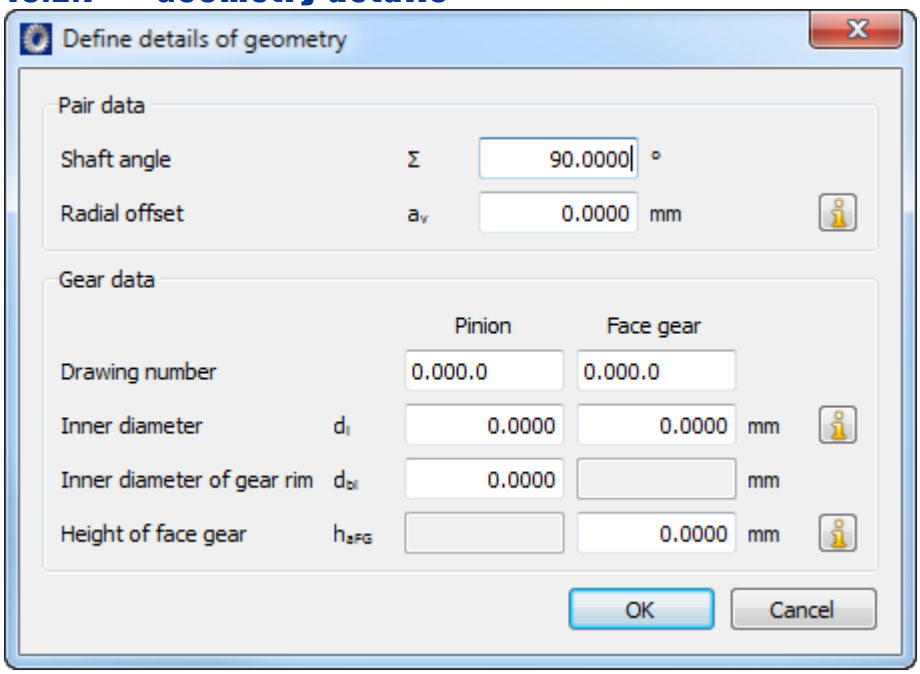
Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 16.6: Accuracy grades for different manufacturing processes

NOTE

The values in brackets can only achieved in exceptional situations.

16.2.7 Geometry details



Click the `Details` button in the upper right-hand part of the Geometry area to open the `Define details of geometry` dialog window. You can enter these parameters here.

16.2.7.1 shaft angle

You can select your own shaft angle here. However, to perform a strength calculation you should set it to Σ = 90.

16.2.7.2 Inner diameter

The inside diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For complete gears, enter 0, and for external gears with a web, enter the appropriate diameter d_i as shown in Figure 16.7.

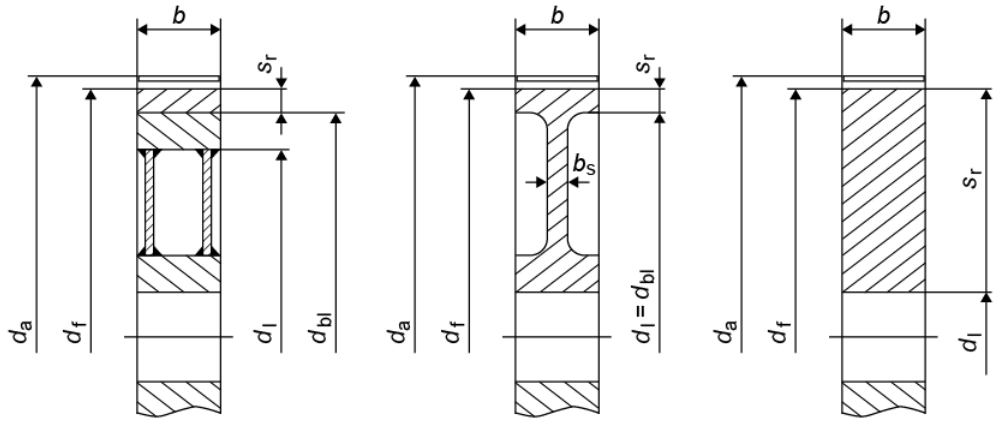


Figure 16.7: Measuring the diameter

The inside gear rim diameter is required for calculations according to ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation results. as illustrated by → Figure on page II-587.

16.2.7.3 Height of face gear

For information on defining the height of face gear h_{aFG} →see Figure (see page II-555)

16.2.8 Materials and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own input` from the list or enter the material in the database (see section "External tables" on page I-136) first. Click the  button to open the `Material pinion(face gear)` window, in which you can select a list of materials that are available in the database. Select the `Own input` option to enter specific material characteristics. This option corresponds to the `Create a new entry` window in the database tool.

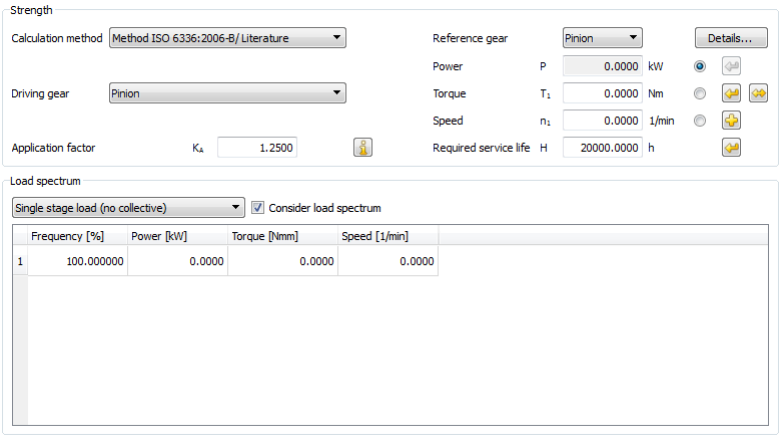


Figure 16.3: `Load` input window in the `Face gears` module

16.3.1 Methods used for strength calculation

To enable developers to use the calculation method they require, KISSsoft can perform strength calculation according to ISO 6336, DIN 3990, DIN 3991, ISO 10300 or DIN 3991.

16.3.1.1 Only geometry calculation

If you select this method, no strength calculation is performed. Therefore you no longer need to enter the data that is only required for the strength, such as power, application factor, etc.

16.3.1.2 Static strength

The strength calculation for cylindrical gears (see section "Calculation methods" on page II-281) is implemented here.

16.3.1.3 Method ISO 6336-B/Literature

We recommend you use the method described here.

The method used to calculate the strength of face gears as originally proposed by Crown Gear [3], is based on the cylindrical gear calculation according to DIN 3990. The inclined lines of contact in a face gear increase the total contact ratio due to pitch overlap. This can be compared with the overlap ratio in helical gear cylindrical gears (an overlap ratio is also present in helical face gears due to the helix angle β_n). You can therefore derive the virtual helix angle β_v from the inclination of the lines of contact.In the strength calculation this effect is taken into account by helix angle factors Y_β and Z_β. The value at the middle of the facewidth is then used as the transverse contact ratio ε_a. It is clear that the face load factor $K_{H\beta}$

and transverse coefficient K_{Ha} according to DIN 3990 cannot be used for face gears. In crown gear calculations, these values are usually set to $K_{H\beta} = 1.5$ and $K_{Ha} = 1.1$, and therefore enable the same procedure to be used as the one used to calculate bevel gears (DIN 3991, ISO 10300). However, the international acceptance of the strength calculation method specified in ISO 6336 makes it a logical alternative to DIN 3990. As ISO 6336 is very similar to DIN 3990, the same restrictions also apply.

In contrast to the Crown Gear program, the following data is used in the calculation:

- The arithmetical facewidth (pitting) corresponds to the minimum line of contact length (Lcont)

- The circumferential force Ft is derived from dPm (middle facewidth)

16.3.1.4 Crown Gear Method (DIN 3990)

This calculation method produces results that correspond to those produced by the Crown Gear program. The underlying principle of calculation is described earlier in the "ISO 6336/Literature" method (see page II-550).

The main differences between it and the "ISO 6336/Literature" method are:

- The calculation is based on the method defined in DIN 3990.
- The arithmetical facewidth (pitting) corresponds to the facewidth (even if the minimum line of contact length is shorter than the facewidth).
- The circumferential force Ft is derived from dPd (reference circle = module * number of teeth), even if dPd is not the middle facewidth.

16.3.1.5 Analogous to ISO 10300, Method B

As already mentioned, you can use ISO 10300 as a good alternative method for calculating the strength of bevel gears. Face gears are classified as bevel gears and can therefore be regarded as bevel gears where the cone angle is 0° (pinion) and 90° (face gear). The strength of bevel gears is calculated on the basis of the equivalent cylindrical gear (cylindrical gear with the same tooth form as the bevel gear). However, for a face gear, the virtual gear number of teeth for the pinion is $z_{1v} = z_1$ and for the gear z_{2v} it is infinite. If you verify the examples, using the Crown Gear program (method matches DIN 3990) and the ISO 10300 method in KISSsoft you will get a good match of values. The variation in root and flank safeties is less than 10% and usually less than 5%. This shows that both calculation methods in DIN 3990 and ISO 10300 (DIN 3991) are reliable and effective.

16.3.1.6 Analog to DIN 3991, Method B

The same notes as for the "Analog to ISO 10300" method (see page II-551) also apply here.

16.3.2 Service life

The value in the `Service life` input field is used together with the speed to calculate the number of load cycles.

16.3.3 Power, torque and speed

Click the  button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-471) can be maintained. Click the  button next to the Speed input field to enter the direction of rotation of the face gear as specified in the `Define sense of rotation` window in 16.9 .

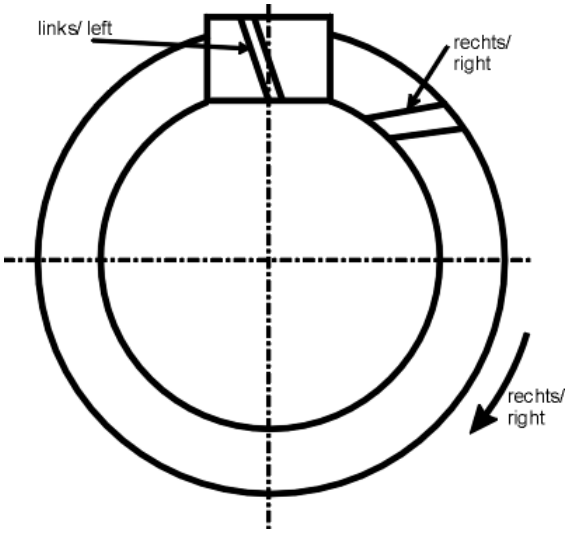


Figure 16.9: Helix angle on a face gear:right; helix angle on the pinion: left; direction of rotation: to the right

16.3.4 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$ applies. Table 16.8. illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

Operational behavior of the driving machine	Operational behavior of the driven machine			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 16.8: Assignment of operational behavior to → application factor

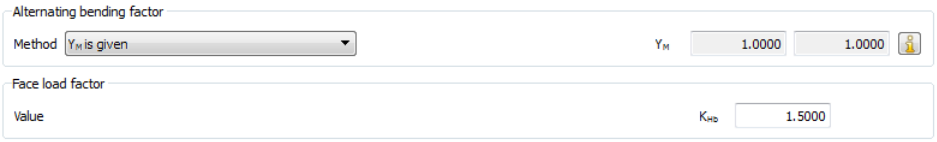


Figure 16.4: `Coefficients` input window in the `Face gears` module

16.4.1 Face load factor

The face load factors $K_{H\beta}$ take into account the effect of uneven load distribution across the facewidth on contact stress, tooth root load and resistance to scoring. For face gears, we recommend you use approximately the same coefficients (see page II-535) as for bevel gears.

16.5 Modifications

The Modifications (see page II-361) (tab) input window in the Face gears calculation module basically includes the same functionality as for cylindrical gears. Its special features are listed below:

16.5.1 Addendum reduction

You specify the tip alteration h_{ak} and the length of the tip alteration l_{hak} (see Figure 16.7) in the Modifications input window in the Modifications area. The tip is then altered to prevent the tooth becoming pointed. When you specify a tip alteration, we recommend you display the entire modification for the 3D export, so that you can increase the number of sections calculated under Calculation > Settings > General (→ Additional information (see page II-556)).

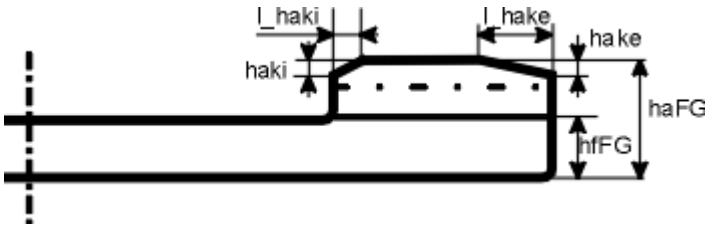


Figure 16.11: Characteristic values of a face gear

16.5.2 Type of modification

In the List of modifications (see section "Type of modification" on page II-361) you can only make changes to the pinion.

16.6 Settings

In the `Calculation` menu you will find the `Settings` option. Click this submenu to open the `Module-specific Settings` window. From here, you can access the tabs listed below to input other calculation parameters.

16.6.1 General

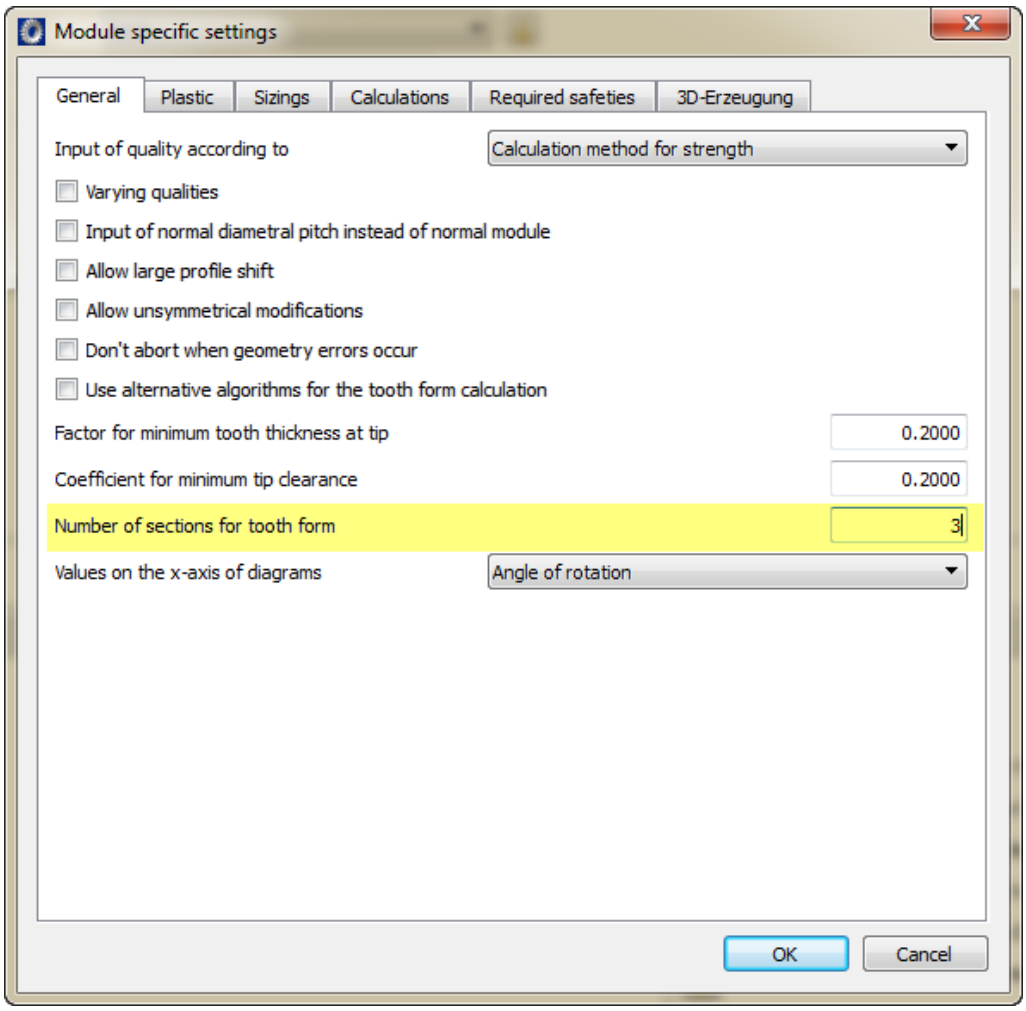


Figure 16.12: The `General` tab in the `Module-specific Settings` window

The `Number steps for tooth form calculation` input field defines how many equidistant section levels $N \geq 3$ are to be distributed between the outside and inside diameter of the face gear. The default value here is $N = 3$ which defines section levels $r_2 = d_{2i}/2$, $r_2 = d_{2e}/2$ and $r_2 = (d_{2i} + d_{2e})/4$.

NOTE

You should select *N* > *10* to ensure an adequate spatial resolution for your 3D export.

16.6.2 Sizings

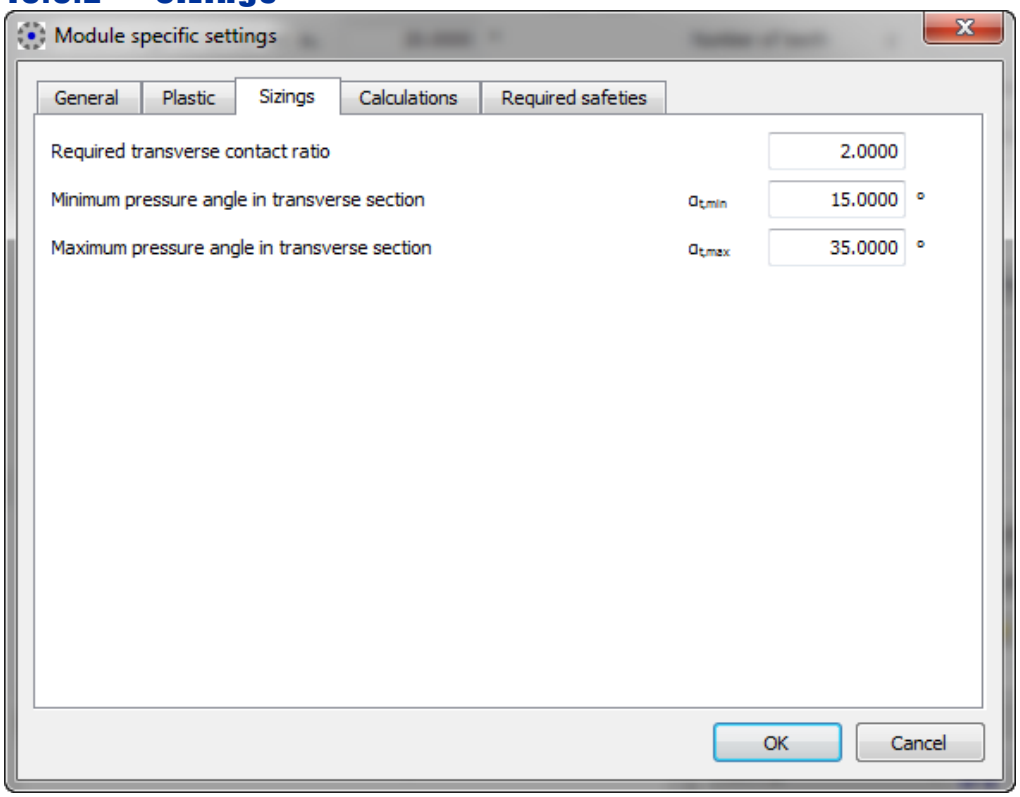


Figure 16.13: `Sizings` tab in the `Module-specific Settings` window

The Minimum/maximum pressure angle in transverse section $\alpha_{t,min/max}$ input fields define the range in which the values for the pressure angle for the tooth flank for the face gear across its width may lie. These values are used, for example, when sizing the facewidth of face gear b_2 and axial offset b_v.

16.7 Notes on face gear calculation

16.7.1 Dimensioning

The complexity of the tooth forms in face gears means that KISSsoft uses various procedures that differ extensively from other commonly used procedures, such as for cylindrical gears, when dimensioning face gears. For a face gear, you must select a geometry that prevents the creation of pointed teeth on the outside face of the gear and ensures that no (or very little) undercut occurs on the inside face. You must perform these checks when you calculate the tooth form. The actual geometry calculation procedure converts the equivalent bevel gear and the virtual cylindrical gear. In the tooth form calculation process, a face gear is calculated in a number of sections set along its facewidth. To specify the number of required sections, select the `Calculation` menu and then, under `Settings > General > Number of sections for the tooth form calculation` , define the number of sections. In the `Geometry` graphics window, you can display the tooth form simultaneously on the inside diameter, outside diameter and in the middle of the tooth. You can see here whether top land and undercut are tolerable.

You can take these measures to prevent pointed teeth and/or undercut:

- change facewidth offset b_v
- reduce the facewidth
- change the pressure angle
- alter tip in the outside part of the facewidth

NOTES

- To generate a crowned tooth form: You can generate crowning on the tooth trace of face gears by using a pinion type cutter that has one or two more teeth than the meshing pinion. Use the storage function in the `Graphics > Geometry > Meshing` 2D display to check the difference between the generated tooth forms. To do this, define a pinion type cutter with the same number of teeth as the pinion used to calculate the tooth form. Then save this cutter data by clicking the `Gear 2 Save` button  and then increase the number of teeth on the pinion type cutter. If the face gear has a large axial offset b_v, you can displace the crowning to one side.

16.7.2 Pinion - Face gear with Z1 > Z2

No provision has been made for calculating a pinion – face gear pairing when the number of teeth on the face gear (Z2) is less than the number of teeth on the pinion (Z1), because this situation does not happen very often. However, under certain conditions, you can still determine the geometry of this type of pairing.

To do this, go to `Settings` and set the `do not cancel when geometry errors occur` flag. Then, we recommend you follow these steps:

- Reduce the facewidth of the face gear (for example, by half)
- Starting with Z2 = Z1, zoom Z2 out step by step, performing a calculation after every step and correcting the inside, middle, and outside aspect of the cuts and, if necessary the tooth height, in the 2D display.
- Once you achieve the required number of teeth Z2, try to increase the face-width of the face gear again, and modify it if necessary.

Chapter 17
Worms with Enveloping Worm Wheels

Calculate worm geometry according to ISO 14521 or DIN 3975. Tooth thickness and control measurement values (base tangent length, rollers and measurement over balls of the worm gear) as specified in ISO 21771. Manufacturing tolerances according to DIN 3974.

You can size the facewidth, the center distance, the lead angle etc. Strength calculation as defined in ISO 14521 or DIN 3996 with: efficiency, temperature safety, pitting safety, wear safety, tooth fracture and deflection safety. Data for various different worm wheel materials are supplied.

You can also calculate the starting torque under load, which is a critical value when sizing gear drives.

Flank forms: ZA, ZC, ZH, ZI, ZK, ZN.

For more information about the dimensions of a worm wheel, refer to Figure 17.1.

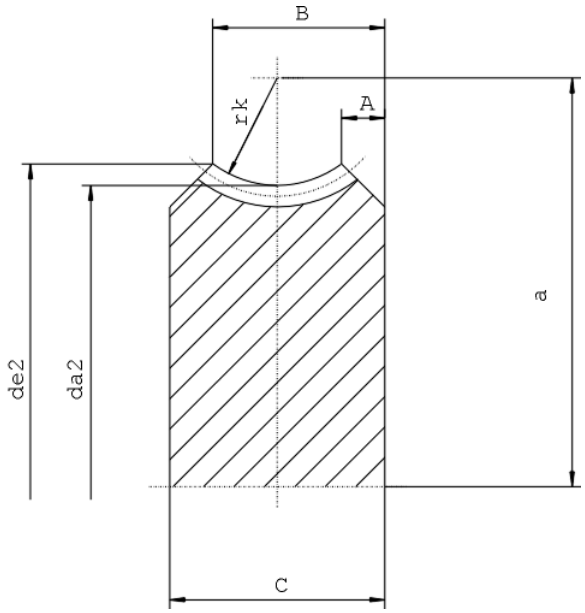


Figure 17.1: Dimensions of the worm wheel

17.1 Underlying principles of calculation

The underlying geometric relationships are defined in ISO 14521 or DIN 3975. You will find additional information, and other important definitions, such as the various worm flank forms (ZA, ZC or ZI, ZH, ZK, ZN), in [66]. You calculate strength (tooth fracture, pitting, wear and temperature safety) according to ISO 14521 or DIN 3996. These calculations take much less time and effort to perform than those required for cylindrical gears. Worms can be checked throughout the manufacturing process by using what are known as "three wire measurements". This corresponds to the principle of the measurement over two balls that is used for worm wheels (and also for cylindrical gears). However, the calculations involved in ascertaining the three wire measurement are very complex. A very useful method for standard flank forms has been developed by G. Bock [4] at the physikalisch-technisches Bundesanstalt (German national metrology institute) in Berlin. This method takes into account the shape of the worm's flank, which is why it is used in KISSsoft.

NOTE

When you use the term "module" you must differentiate clearly between the axial and the normal module.

Note about how to use the application factor

In cylindrical gear and bevel gear calculations, application factor KA is usually multiplied by the power, for example, so that KA=1 with P= 5 kW gives exactly the same safeties as KA=2 and P=2.5 kW. However, this is different for worm calculations performed according to ISO or DIN standards and may lead to confusion.

The forces and torques are multiplied with the application factor. In contrast, the power is not multiplied with the application factor when determining power loss PVLP and when calculating the total efficiency etaGes. Therefore, for CA=2 and P=2.5 kW instead of CA=1 with P= 5 kW - power loss [PV] will be smaller, and the total efficiency etaGes will be far too low.

Results for the example "WormGear 1 (DIN 3996, Example 1).Z80":

	KA=1; P= 5 kW	KA=2; P=2.5 kW	
PVLP	0.140	0.070	<< (* 1/KA)
PVD+PV0	0.199	0.199	=
PVZ	0.530	0.530	=
PV	0.869	0.799	<

etaz	90.00	90.00	=
etaGes	85.19	75.77	<<
theS	76.6	76.6	=
theM	80.9	80.9	=
SW	1.386	1.386	=
SH	1.143	1.143	=
Sdel	2.369	2.369	=
SF	2.251	2.251	=
ST	1.306	1.306	=

This difference in the results is not logical so therefore, to determine PVLP and etaGes, the power is also multiplied with CA to achieve the same results.

17.2 Basic data

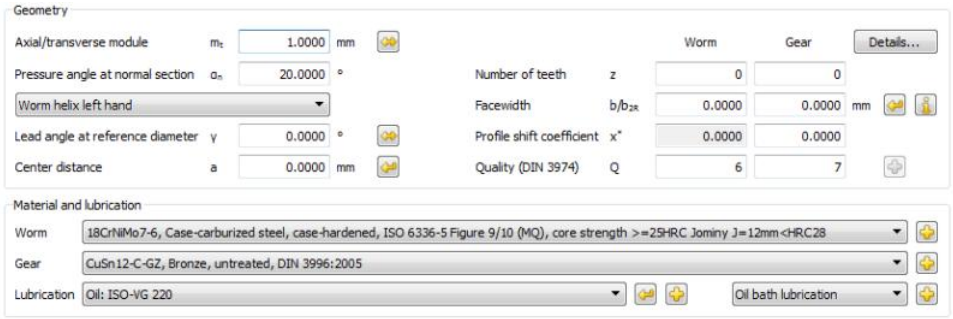


Figure 17.2: `Rating` input window in the `Worms with globoid worm wheels` module

17.2.1 Axial/transverse module

The axial module of the worm and of the transverse module on a worm wheel are identical. In the `Calculation` menu, select `Settings > Calculations > Calculation with normal module` instead of `axial module` to use the normal module m_n instead of the axial module in future calculations.

NOTE

This changes the way the tip and root circles (see page II-579) are calculated.

17.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard meshings, the pressure angle is $\alpha_n = 20^o$. Smaller pressure angles can be used for larger numbers of teeth, to achieve higher contact ratios. Larger pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

17.2.3 Lead angle at reference circle

The lead angle in a worm (gear 1) is the complement of the helix angle and is calculated in accordance with equation 17.1.

$$\gamma = 90° - \beta \tag{17.1}$$

Click the button to open the `Convert lead angle` dialog window, in which you can calculate the lead angle from other gear values. These options are available here: `from center distance`, `from reference diameter` and `from the reference circle and the center distance` (x_2* is modified). A larger lead angle produces greater efficiency, whereas you can design self-locking toothing if you use a smaller lead angle.

17.2.4 Center distance

Click the button to calculate the center distance from the values of profile shift coefficient x^*, number of teeth z and lead angle γ. In this case, you do not receive a message telling you that the calculation has been performed correctly.

17.2.5 Number of teeth

The number of teeth on a worm usually is in the range $1 \leq z1 \leq 4$.

17.2.6 Facewidth

For more information about the dimensions of gear teeth and wheel flange widths, please refer to Figure 17.3. Enter the width of the worm wheel in the facewidth b_{2R} input field. The facewidths b_{2H} and b_2 of the worm wheel are then calculated from this value.

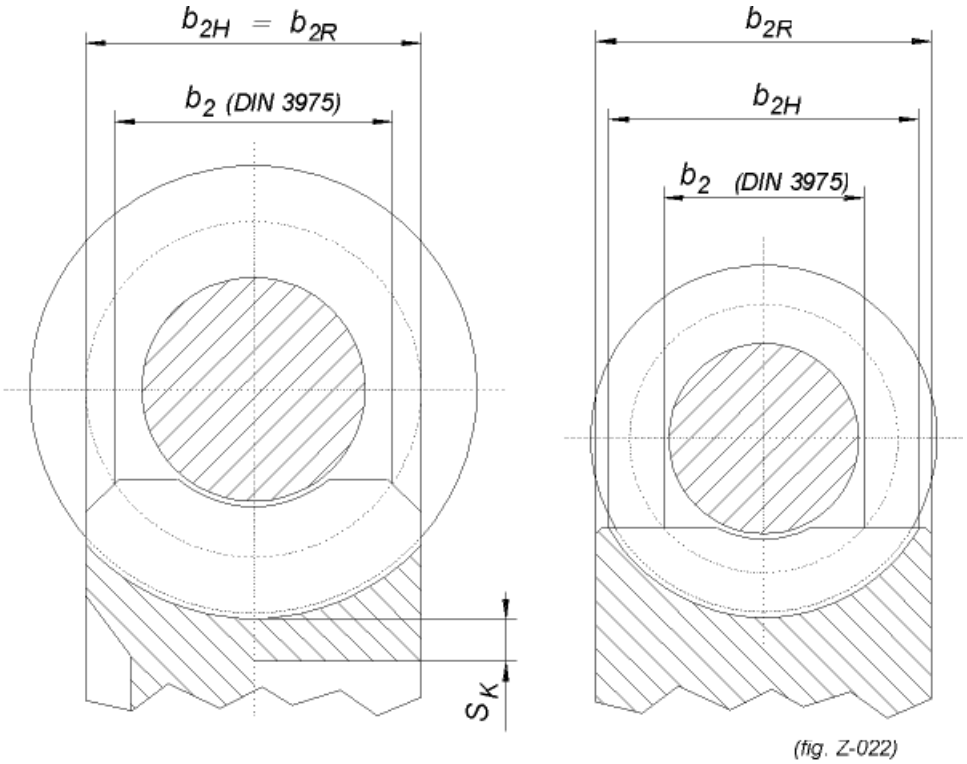


Figure 17.3: Dimensions of gear tooth and wheel flange width

17.2.7 Profile shift coefficient

In the `Worms with globoid worm wheels` calculation module, the profile shift for worm/gear 1 is set to zero (as defined in the ISO 14521 standard). You can only change the tooth thickness of the worm in the `Tolerances` input window.

NOTE

You should use the `Crossed helical gears and precision mechanics worms` calculation module if you require a worm where the profile shift coefficient is $x_1^* \neq 0$.

17.2.8 Tooth thickness modification factor

This factor should only be used in special cases. The factor xs changes the tooth thickness with ΔAs = 2 * xs * as it does with bevel gears. xs2 = -xs1 always applies so the clearance does not change if xs1 is changed Reasonable values for xs1 lie in the range -0.1 to +0.1.

It is useful to apply this factor in situations where: The worm is significantly harder than the gear. In operative use, the gear will wear and cause the drive to fail. This is because the gear teeth will have become thinner and thinner, and will break. If the worm's tooth thickness is changed, for example to xs1 = -0.1, the gear tooth will be thicker. This will result in a much longer service life.

17.2.9 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`.

Achievable qualities are shown in Table 17.1.

Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 17.1: Accuracy grades for different manufacturing processes

17.2.10 Geometry details

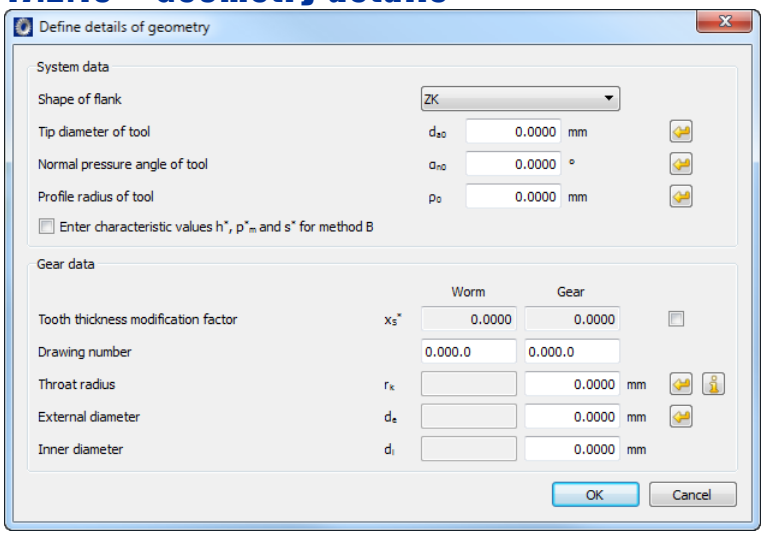


Figure 17.4: `Define details of geometry` window

Click the `Details...` button in the `Geometry` area to open the `Define geometry details` window, in which you can change the parameters listed below.

17.2.10.1 Shape of flank

The flank shape is a result of the manufacturing process. ZA, ZN, ZK and ZI worms have very similar levels of efficiency and flank load capacity. Although ZC and ZH worms (hollow flanks) have better load capacity in some situations, they do have other major disadvantages.

ZA form:	manufactured on turning machine with tool (straight flanks), mounted in axial section
ZN-form:	manufactured on turning machine with tool (straight flanks), mounted in normal section
ZI form:	manufactured with hobbing cutter (worm flank is involute)
ZK form:	manufactured with grinding wheel (straight flanks), mounted in normal section
ZC, or ZH form:	Manufacturing with special tools to generate a hollow flank

For more information, please refer to: Dubbel [38], with figures on pages G136 and S79.

17.2.10.2 External diameter and throat radius

You specify values for the external diameter d_{e2} and throat radius r_k as specified in DIN 3975-1:2002-7. In accordance with equations (59) and (67) the following values are suggested for these two dimensions:

$$d_{e2} \approx d_{a2} + m_x$$

$$r_k \geq a - \frac{d_{a2}}{2}$$

with:

d_{a2} - Tip diameter

m_x - Axial module

a - Center distance

17.2.11 Materials and lubrication

Materials

The strength calculation method used for worms according to ISO 14521 is based on empirical values determined using these materials:

Worm:

- Case hardening steels (especially 16MnCr5), HRC = 58 to 62
- Heat treatable steels (especially 42CrMo4), heat or induction-hardened, HRC = 50 to 56
- Nitriding steels (especially 31CrMoV9), gas-nitrided

Worm wheel:

- Bronze (GZ-CuSn12, GZ-CuSn12Ni, GZ-CuAl10Ni)
- Grey cast iron (GJS40, GC25)
- Polyamide (PA-12, cast)

To calculate strength you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited. As defining data for materials that are not already documented takes a great deal of time and effort, we strongly recommend you select a material from the list that is closest to the material you actually want to use.

Lubricants

Selecting the right lubricant for a worm gear is extremely important. Synthetic lubricants (polyglycols or polyalfaolephine) can reduce loss and wear enormously.

17.3 Rating

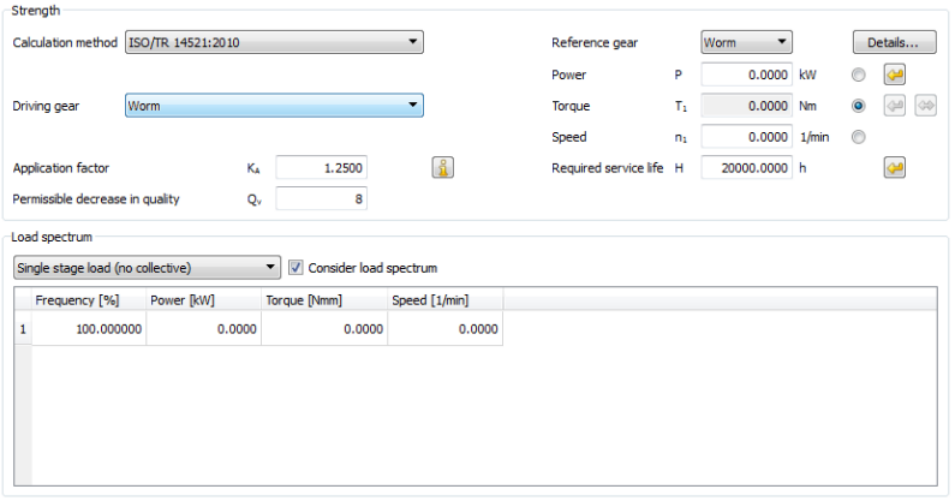


Figure 17.3: `Rating` input window in the `Worms with globoid worm wheels` module

17.3.1 Methods used for strength calculation

The calculations defined in ISO 14521 and DIN 3996:2006 are identical.

However, strength calculation as defined in ISO 14521 includes a number of different methods (A,B,C,D;). KISSsoft uses the most precise, documented method which usually corresponds to Method B. This calculation method is not suitable for every material (see section "Materials and lubrication" on page II-568), because some of the empirical values are missing.

The ISO 14521 standard provides a calculation method for determining:

- Efficiency
- Wear and Wear safety
- Pitting safety
- Root safety
- Deflection safety
- Temperature safety

NOTES:

- To calculate strength you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited.
- Grease lubrication: Grease lubrication is not mentioned in DIN 3996. In this situation, KISSsoft performs the calculation as for oil bath lubrication. This assumption is permissible, because the lubrication type has very little influence on the calculation.
- Endurance limit values for tooth root load capacity: The standard provides two different values. If you enter the smaller value in the database, no decrease in quality due to plastic deformation of the teeth will be accepted.

17.3.2 Service life

The value in the `Service life` input field is used together with the speed to calculate the number of load cycles.

17.3.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 17.2. illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 17.2: Assignment of operational behavior → application factor

17.3.4 Permissible decrease in quality

Depending on the construction type of the worm wheel, it may experience a decrease in quality over time due to wear. This value must not sink below the value

specified in this input field. A decrease in quality is linked to the plastic deformation of the material and therefore a higher material property. This, in turn, results in a higher safety against plastic deformation in the root.

17.3.5 Power, torque and speed

Click the  button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see section "Required safeties" on page II-471) can be maintained.

17.3.6 Strength details

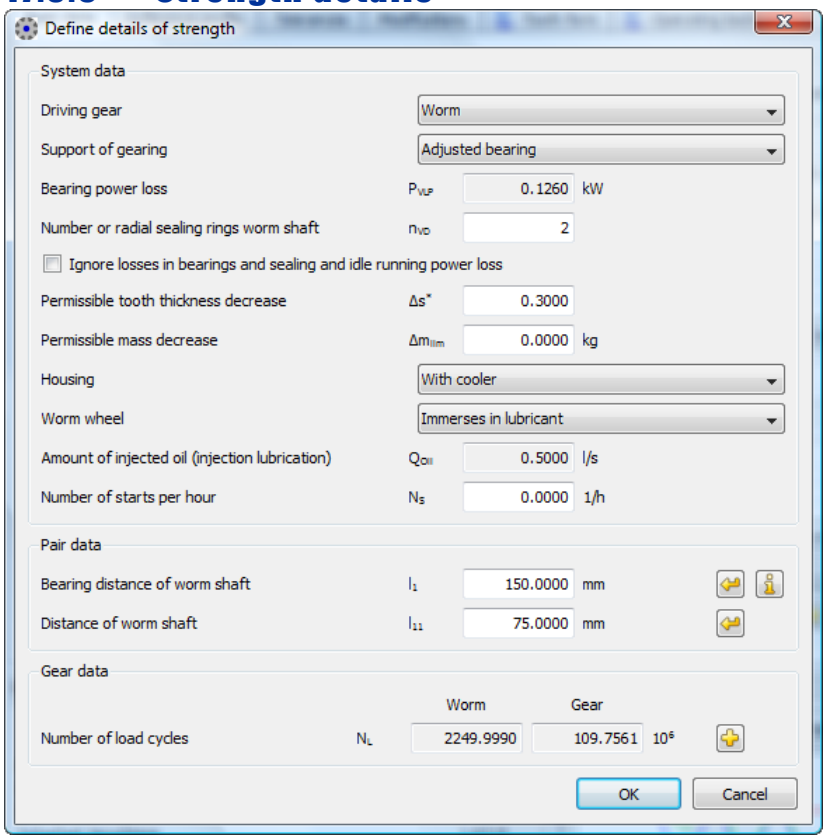


Figure 17.5: `Define details of strength` window

Click the `Details...` button in the `Strength` area to open the `Define details of strength` window, in which you can change the parameters listed below.

17.3.6.1 Support of gearing

The calculation method used to ascertain bearing power loss of the integral worm shaft identifies two different types of bearing.

17.3.6.2 Bearing power loss

If rolling bearings are used, the power loss is calculated using the empirical formulae defined in ISO 15451. If plain bearings are used, you must specify the power loss manually.

17.3.6.3 Number of radial sealing rings, worm shaft

To calculate the power loss in sealing, you must enter the number of radial sealing rings on the integral worm shaft. The sealing gaskets on the worm shaft are not taken into account because their slow rotation speed means they lose very little power (the calculation formulae are defined in ISO 15451).

17.3.6.4 Permissible tooth thickness decrease

The permissible tooth thickness decrease (on the gear) is necessary for calculating the wear safety and taken into account when calculating the root safety. If this input field contains the value 0, the permissible tooth thickness decrease is not checked.

17.3.6.5 Permissible mass decrease

You can limit the permissible mass decrease in kg on the worm wheel (for example, by specifying oil change intervals). This limiting value is also used to define wear safety. If this input field has the value 0, the mass decrease will not be checked.

NOTE

The decrease in mass experienced on the worm is not calculated, because the standard assumes that the worm is harder than the worm wheel and therefore will not be subject to wear.

17.3.6.6 Dimension of the integral worm shaft

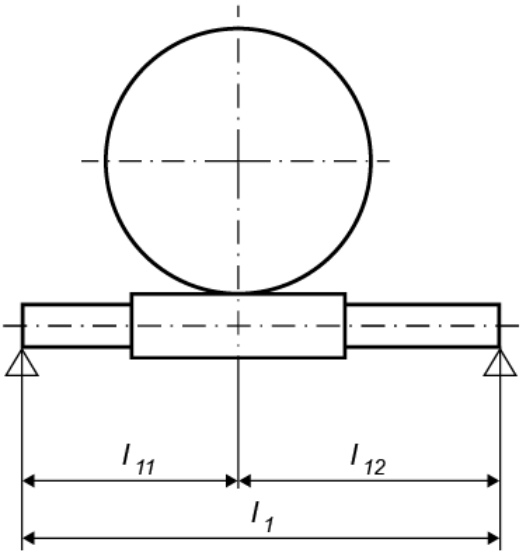


Figure 17.6: Dimensions of the worm-worm wheel

l_1 Distance between the bearings on the integral worm shaft

l_{11} Distance from bearing 1 to the middle of the worm

You need these values to calculate the bending safety. The position of the drive has no effect on the calculation.

17.3.6.7 Load spectra with negative elements

Load spectra with negative load bins (T < 0 and/or n < 0) can also be calculated as follows (this is only applied to bins whose alternating bending factor is YM=1.0).

IMPORTANT:

A load spectrum is considered to be negative if the non-working flank is placed under load.

Coefficient for torque	Coefficient for speed	Flank under load	Actual load bin

+	+	Working flank (*)	Evaluated as positive
+	-	Working flank (*)	Evaluated as positive
-	+	Non-working flank	Evaluated as negative
-	-	Non-working flank	Evaluated as negative

(*) Working flank as entered in the Strength tab

Table 14.10: Evaluation of a load bin, depending on the prefix operator

Under "Details" in the "Strength" section of the "Load" tab, you can select the following:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Consider only positive load bins
  - Consider only negative load bins
  - Check both cases and document the unfavorable case
- For calculating the tooth root safety
  - Evaluate all negative load bins as positive (as up to now)
  - For negative load bins, increase root stress by 1/0.7
  - Increase bending stress for positive load bins by 1/0.7
  - Check both cases and document the more realistic case

17.4 Tolerances

The structure and functionality of the Tolerances (see page II-357) input window in the Worms with globoid worm wheel calculation module is the same as the Tolerances input window for cylindrical gears. When you enter dimensions for worm calculations, we recommend you click on the Thickness tolerance drop-down list and select either the Worm as defined in Niemann or Worm wheel as defined in Niemann option. The corresponding data is based on recommendations in Niemann [66].

17.5 Settings

In the `Calculation` menu you will find the `Settings` option. Click this submenu to open the `Module-specific Settings` window. From here, you can access the tabs listed below to input other calculation parameters.

17.5.1 General

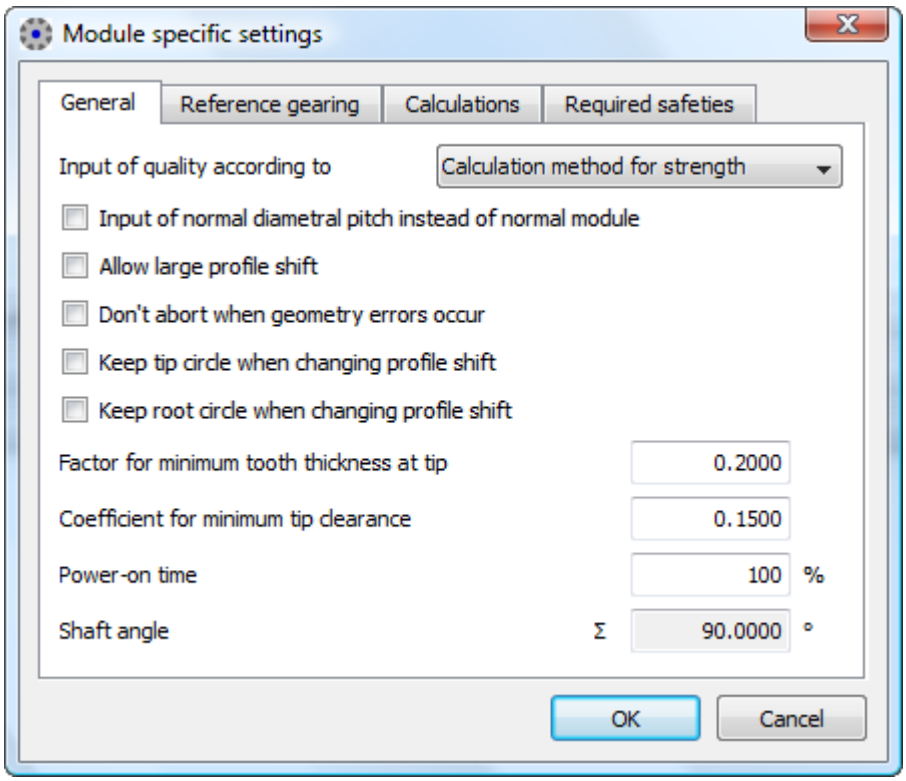


Figure 17.7: The `General` tab in the `Module-specific Settings` window

(entries that are not detailed here (see page II-456))

17.5.1.1 Einschaltdauer

Bei Berechnung der Lebensdauer wird die Einschaltdauer mit der Lastwechselzahl multipliziert. Die Temperaturberechnung berücksichtigt zusätzlich die Einschaltdauer bei der Berechnung der erzeugten Wärme.

Hinweis zur Temperaturberechnung:

Angenommen wird, dass Wärme dauernd abgeführt wird, hingegen nur während

der vorgegebenen Einschaltdauer Wärme zugeführt wird. Die Voraussetzung hierzu ist, dass das Getriebe während kurzer Zeit läuft (maximal 15 Minuten) und dann wieder abgestellt wird. Falls dies nicht zutrifft, muss die Einschaltdauer auf 100% gestellt werden.

17.5.1.2 shaft angle

The default value of the shaft angle is set to 90 degrees because this is the default value specified in the strength calculation method defined in DIN 3996. However, you can calculate the geometry with shaft angle that is not 90 degrees by using the `Crossed helical gears and precision mechanics worms` (see page II-582) calculation module.

17.5.2 Reference gearing

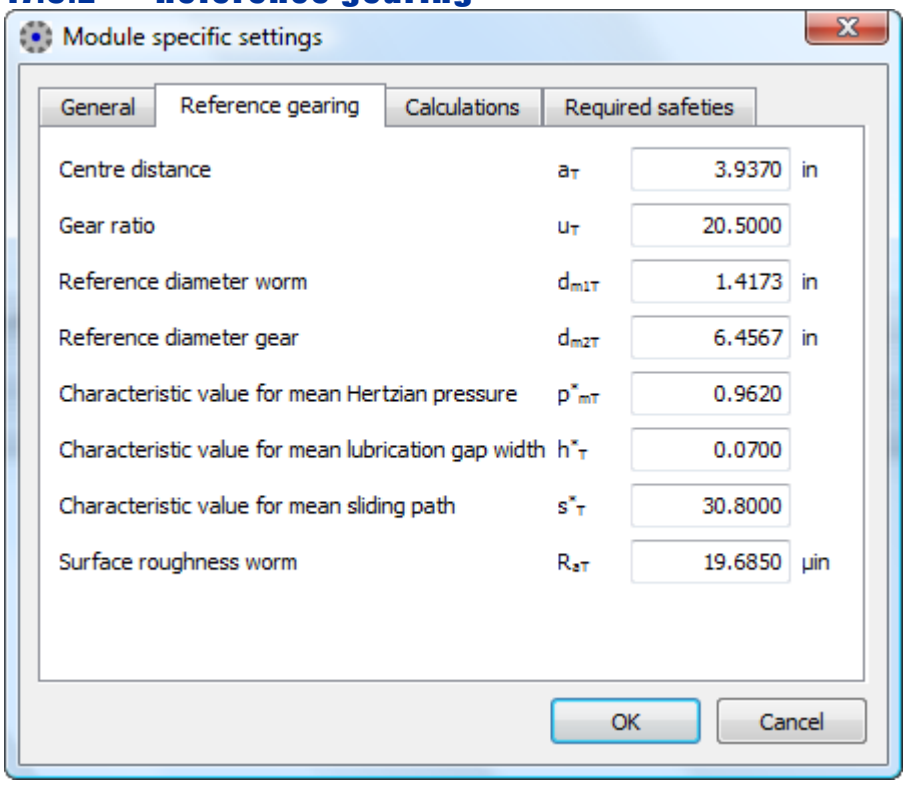


Figure 17.8: `Reference gearing` tab in the `Module-specific Settings` window

This calculation is based on a standard reference gearing, on which tests have been performed. The default data corresponds to the reference gearing in ISO 14521. However, if you have the results of your own tests or empirical values, you can modify this calculation to take advantage of this expertise. For a more detailed description, please refer to ISO 14521.

17.5.3 Calculations

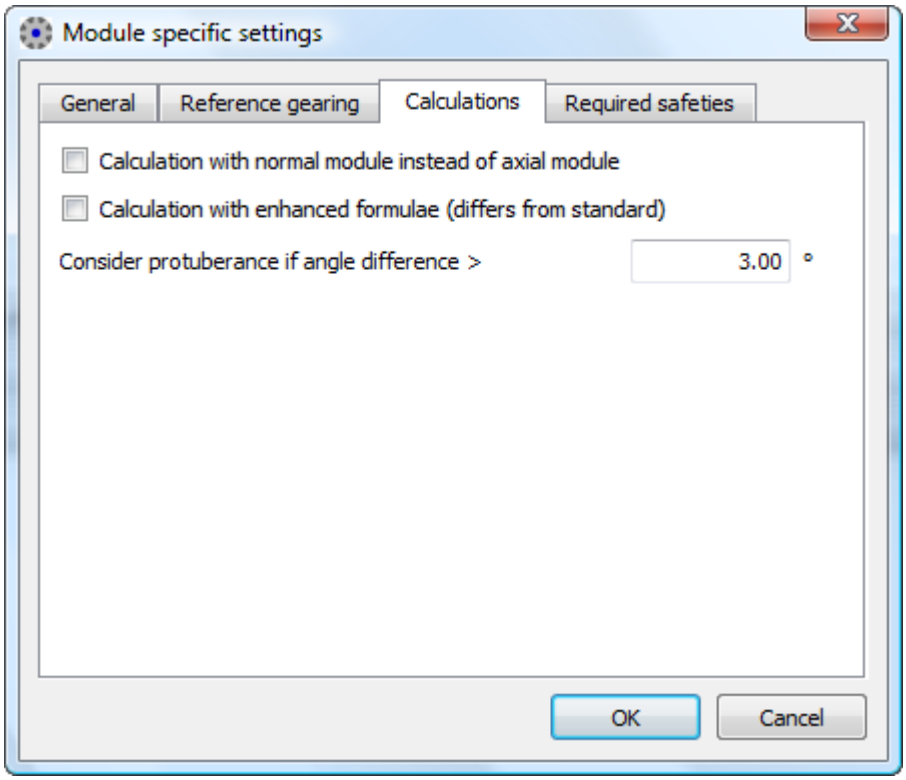


Figure 17.9: `Calculations` tab in the `Module-specific Settings` window

17.5.3.1 Calculation with normal module instead of axial module

The geometry of worm gear pairs is usually calculated with the axial module (or transverse module of the worm wheel). If you click on this checkbox, all the values used for the reference profile are calculated with the normal module (tool module). This particularly affects the tip and root circle. In contrast, the profile shift $x^*_x m_x$ (m_x for the axial module) remains unchanged.

The formula for the tip circle (m_n for the normal module) is then:

$d_{a1} = d_{m1} + 2\ m_n\ h_{aP}$
$d_{a2} = d_2 + 2\ m_x\ x_2 + 2\ m_n\ h_{aP}$

For the root circle, the following apply:

$d_{f1} = d_{m1} - 2\ m_n\ h_{fP}$
$d_{f2} = d_2 + 2\ m_x\ x_2 - 2\ m_n\ h_{fP}$

17.5.3.2 Calculation with enhanced formulae (differs from standard)

If you select this checkbox, alternative calculation methods are used at these points:

- Effective tooth thickness on the tip (instead of formula (84): calculated according to DIN or formula (110) according to ISO)
- Loss of power on toothing PVZ with coefficient 1/9.550 Instead of 0.1

17.5.4 Required safeties

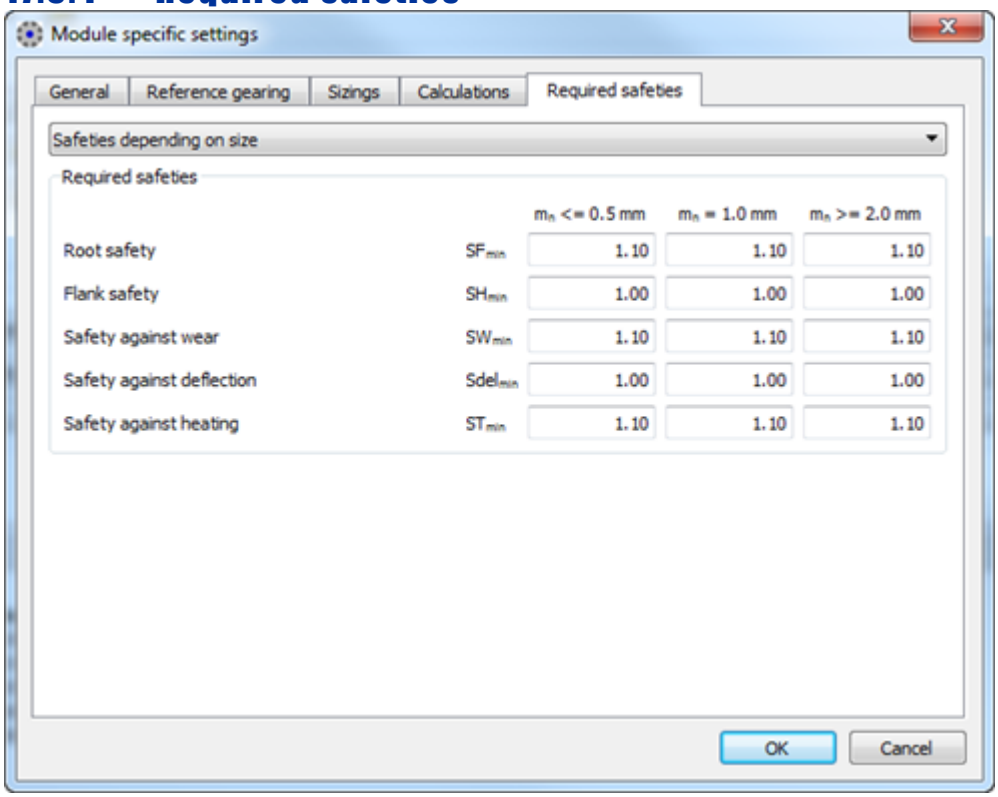


Figure 17.10: `Required Safeties` tab in the `Module-specific Settings` window

KISSsoft issues an error message if the specified required safeties have not been reached after you completed the calculation. Sizing is always calculated on the basis of the required safeties for tooth fracture, pitting and wear. If you do not wish to use one or more of these criteria, set the appropriate required safety to zero. According to ISO 14521, you must ensure the following safeties:

Root safety	:	1.1
Pitting safety	:	1.0
Wear safety	:	1.1
Deflection safety	:	1.0
Temperature safety	:	1.1

You can change these values as required to reflect your own findings.

Chapter 18
Crossed Helical Gears and Precision Mechanics Worms

Crossed helical gears are helical gears that are mounted on crossed axes. The shaft angle is usually $\Sigma = 90^{o}$. In contrast to the line contact shown in globoid worms, crossed helical gears only contact at one point during generation. As a result, they can only transmit very small forces, and are primarily used for control purposes.

In precision engineering, a worm wheel is often manufactured in the same way as a helical gear. This makes it easier to produce and assemble than a globoid gear manufactured using a worm-shaped cutter. In this situation, you should calculate the geometry of the worm wheel in the same way as a helical gear. This is because, if the profile shift total is not equal to zero, the helix angle of the gear will not match the lead angle of the worm. Both gears have the same hand of gear. If the worm has right hand teeth, then the worm wheel is also right hand. The total of both helix angles at the operating pitch circle/spiral is exactly the same as the shaft angle. However, due to the profile shifts, the total of helix angles at the reference circle is not identical to the shaft angle.

In special cases, the shaft angle can also be smaller than the helix angle of gear 1. In this situation, gear 2 has the opposite hand of gear to gear 1.

18.1 Underlying principles of calculation

The method used to calculate crossed helical gears (cylindrical gears with crossed axes) is defined according to Niemann [66]. The current version of this standard describes methods used to calculate and check the geometry of spiral toothed gears for any shaft angle. The measures used for checking and fabrication are determined arithmetically.

Although the method detailed in Niemann [66] is used to calculate the root and flank strength and the scuffing safety as concept, the individual equations used are following ISO 6336. (Niemann uses equations from an old edition of DIN 3990.)

18.2 Basic data

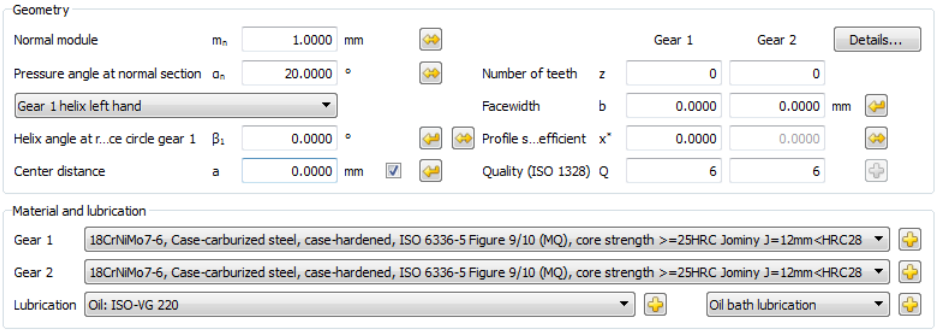


Figure 18.1: `Basic data` input window

18.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the button to open a dialog window, in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

18.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard meshings the pressure angle is $\alpha_n = 20^o$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and enable a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

18.2.3 Helix angle reference circle gear 1

The center distance, number of teeth, profile shift (x^*_1, x^*_2) and shaft angle are used to calculate the helix angle of gear 1. It often happens that several helix angles meet the requirements of the toothing geometry. In this situation, when you click the button you see an `Information` window that lists the possible values. Here the solution that is closest to **the current value** is selected automatically. However, if only one value is suitable for the sizing, it is transferred into the input

field without any messages being displayed. If the sizing function is unable to find any solutions, it displays a warning message and you must then change either the center distance or the module.

18.2.4 Center distance

The center distance is calculated on the basis of the helix angle of gear 1, the shaft angle, the profile shift (x^*_1, x^*_2) and the number of teeth.

18.2.5 Facewidth

Because the facewidth must have a minimum value, the input field has a  button which you can use to define the minimum width based on the parameters you have already defined.

18.2.6 Profile shift coefficient

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

Click the  button, and KISSsoft will determine whether the profile shift coefficient (see page II-267) is taken from measured data or from values given in drawings.

NOTE

If one of the two profile shift values appears in gray, this means it will be calculated by calculated. This is what happens when you activate the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for one of the other gears. calculated.

18.2.7 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The

accuracy grade according to ISO 1328 is very similar to that in DIN 3961 or AGMA 2015.

Achievable qualities are shown in Table 18.1.

Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 18.1: Accuracy grades for different manufacturing processes

18.2.8 Define details of geometry

Click the `Details...` button in the `Geometry` area to open the `Define geometry details` window, in which you can change the parameters listed below.

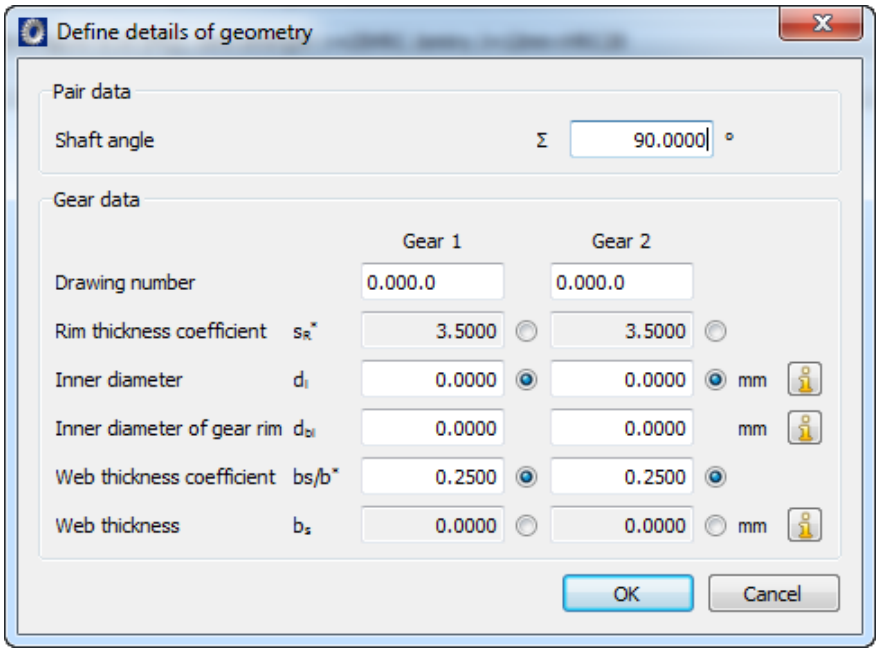


Figure 18.2: `Geometry details` input window

18.2.8.1 shaft angle

The shaft angle is usually $\Sigma = 90^{o}$, but you can specify your own value here.

18.2.8.2 Inner diameter

The inside diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For complete gears, enter 0, for external gears with a web, enter the appropriate diameter d_i as shown in Figure 18.3. For internal wheels, enter the external diameter of the gear rim.

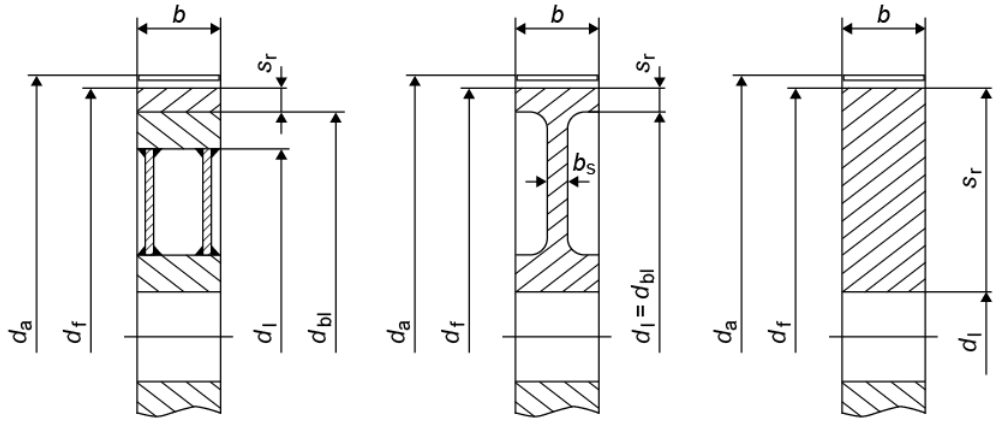


Figure 18.3: Measures of the diameter

The inside diameter of the gear's flange is required for calculations according to ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation results. See also the figure shown above <Kap16:>.3.

18.2.9 Material and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own input` from the list or enter the material in the database first (→ Additional information (see page I-128)). Click the  button to open the `Material gear 1(2)` window, in which you can select a material from the list of materials available in the database. Select the `Own input` option to enter specific material characteristics. This option corresponds to the `Create a new entry` window in the database tool.

18.3 Rating

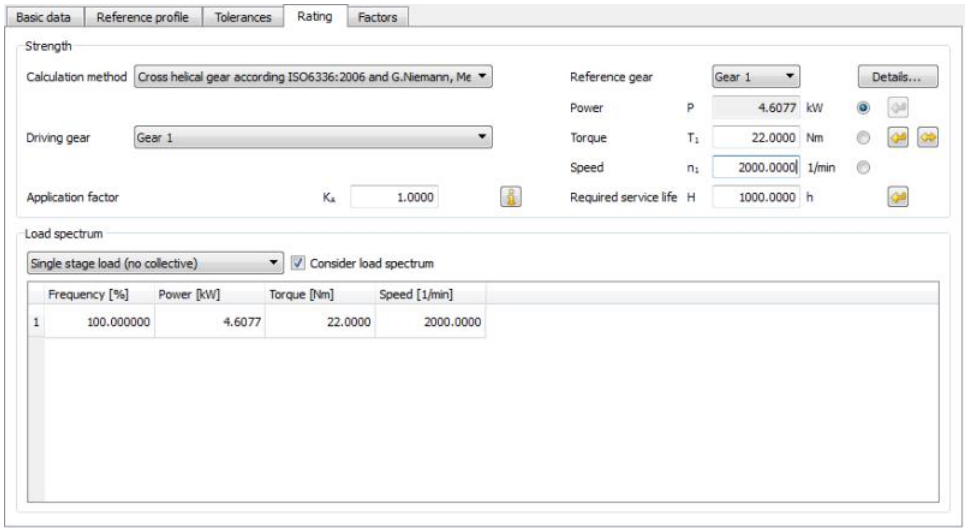


Figure 18.4: `Load` input window

18.3.1 Methods used for strength calculation

As yet, no binding standard has been drawn up for the calculation of crossed helical gears. KISSsoft therefore recommends you use the ISO 6336 (see page II-589) Calculation of load capacity of spur and helical gears.

You can use one of three different methods to calculate the strength of worms:

18.3.1.1 Strength calculation in acc. with Hirn

The method used to calculate worms as defined by H.Hirn is based on an obsolete edition of Niemann's machine elements. It calculates the temperature safety, the flank safety, the root safety, and the deflection safety. Although the material values cannot be compared with the values for worm calculation as defined in DIN 3996, the safeties are, however, similar.

We do not recommend you to use this obsolete method.

NOTE

The calculation method defined in Hirn also selects a material pairing. This must lie in the permitted range of `Materials and lubrication`. Shaft angle $\Sigma = 90^{o}$ and $z_1 < 5$.

18.3.1.2 Strength calculation in acc. with Hoechst

You can use the strength calculation according to with Hoechst for worm wheels made from Hostaform® (POM), paired with steel worm gears [80]. The permitted load coefficient is c [N/mm^2], see Gln. (18.1) ÷ (18.3) is a value that defines the temperature resistance. This method also checks the worm's permissible contact stress and blocking safety. The decisive value for blocking safety is maximum load, not continuous load.

$$c = \frac{F_2}{f_z \cdot b \cdot m \cdot \pi} \tag{18.1}$$

$$b = \sqrt{d_{a1}^2 - d_{m1}^2} \tag{18.2}$$

$$m = \frac{m_n}{\cos(\gamma_m)} \tag{18.3}$$

where

F_2	circumferential force on the worm wheel
f_z	Coefficient for number of teeth
b	useable width
m_n	Normal module
γ_m	Mean lead angle
d_{a1}	tip diameter of worm
d_{m1}	reference diameter of worm

NOTE:

Shaft angle $\Sigma = 90^o$ and $z_1 < 5$. The calculation method involves a worm made of steel and a crossed helical gear made of plastic.

18.3.1.3 Strength calculation in acc. with ISO 6336/Niemann

You can perform the strength calculation for crossed helical gears with $z_1 \geq 5$ as defined in Niemann [66]/ISO 6336. As stated in Niemann, the contact ellipse is calculated using a for the width and b for the height of the half axes. An effective facewidth of 2a is assumed for flank safety (pitting). The same value plus twice the

module value is used to calculation the strength of the tooth root. This corresponds to the specifications given in ISO 6336, if the facewidth is greater than the contact width. Scuffing safety is calculated as defined in Niemann [66] . This method differs from the DIN 3990-4 guideline because of the high sliding velocities of the crossed helical gears. It is more similar to the method applied to hypoid bevel gears. It supplies proof of tooth root resistance, flank load capacity and scuffing load capacity.

NOTE:

If the number of teeth is $z < 5$, this calculation supplies tooth root and contact stress safeties that are too high.

18.3.1.4 Strength calculation as defined in VDI 2736

This VDI guideline defines how precision mechanics worms are to be calculated.

18.3.1.5 Static calculation

This then makes it possible to define the safety against fracture and yield point Sl = lGFmin/lGFP. This calculation is performed in accordance with the formulae documented in 13.2.11 Static calculation.

The calculation for worm gears returns safeties that tend to be too great, because worms are usually checked for safety against shearing.

18.3.1.6 Static calculation on shearing

Verification of a worm wheel on shearing

$\tau_F = F_t^2 * K_A * Y_E / A_\tau$

$A_\tau = b_{max}/5*(4*s_{tda2}-s_{tdx2})$

$d_{x2} = 2* a-d_{a1}$

This calculation is performed automatically and is documented in the report in Section 6A.

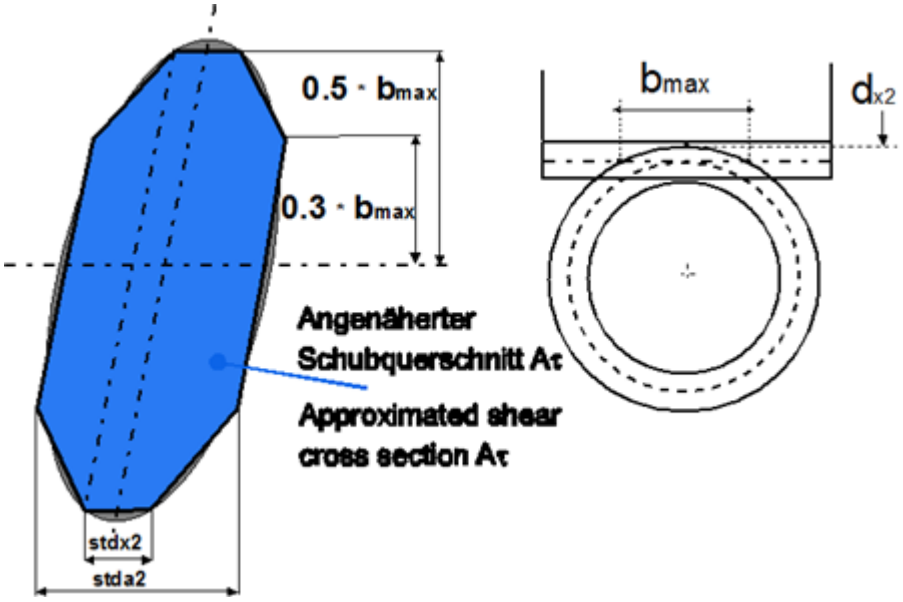


Figure 18.5: Dimensions of the shear cross section.

18.3.1.7 Calculating wear on worm wheels according to Pech

A calculation for determining the wear on crossed helical gears according to Pech [101] is now available. This process calculates the plastic deformation, the degree of wear and the overall wear (in the normal section on the operating pitch diameter) of plastic worm wheels. The following notes apply to this calculation:

- Cylindrical worm wheel pair with a shaft angle of 90°
- Grease lubrication
- Calculation without load spectrum
- Material of worm: Steel
- Material of worm wheel: POM, PEEK, PEEK+30% CF or PA46
- Driving gear: Worm

Using KISSsoft, you can also perform calculations for plastic/plastic combinations, but these are subject to special assumptions and limitations (see below).

The coefficient of friction (COF) taken from the material DAT file has no effect on

the calculation (the COF is calculated according to Pech). A (user-defined) COF is used to calculate plastic/plastic combinations (Load > Details).

The entries for the root temperature and flank temperature have no effect on the calculation of steel/plastic combinations (temperatures are calculated according to Pech). User-defined temperatures are used for plastic/plastic combinations. The grease temperature for plastic/plastic combinations is calculated as the mean value of the root temperatures of the two wheels.

The flank roughness of the worm wheel has an effect on the calculated coefficient of friction. A greater level of roughness causes a greater amount of wear.

Click on "`Module specific settings`" to input a coefficient for the permitted level of plastic deformation (Calculation > Settings > Plastic).

If you input your own material into the KISSsoft material database, you must enter additional data in the material DAT file (for example for PEEK).

```
-- Type of plastic material
-- Values: 0-not on the list, 1-POM, 2-PEEK, 3-PEEK+30%CF, 4-PA46, 5-PA66,
-- 6-PA6, 7-PA66+GF, 8-PPS, 9-PPS+GF, 10-PA12, 11-PBT, 12-PET
:TABLE FUNCTION MaterialType
    INPUT X None TREAT LINEAR
DATA
     0
     2
END
```

The table below shows the parameter limits for calculating wear according to Pech.

Number of teeth: Worm wheel	$16 \leq Z_2 \leq 80$
Center distance	$10 \text{ mm} \leq a \leq 80 \text{ mm}$
Axial module: Worm wheel	$0.5 \text{ mm} \leq m_x \leq 3 \text{ mm}$
Gear ratio	$10 \leq u \leq 80$
Pressure angle	$10° \leq \alpha_n \leq 22°$
Profile shift coefficient: Worm wheel	$-0.2 \leq x_2 \leq 1.5$

Table 18.2: Geometry limit values for calculating wear according to Pech

The development on the loaded and unloaded flank according to Pech can be seen in Figure 5.

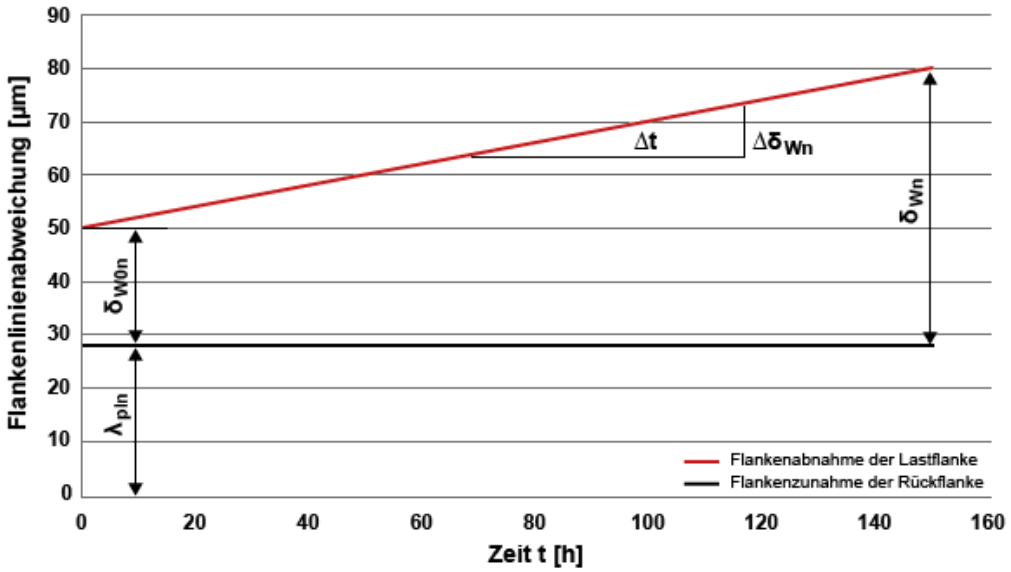


Figure 14.6): Development on the loaded and unloaded flank, according to Pech.

18.3.2 Service life

Click the button to enter the required service life directly in the input field. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated according to ISO 6336-6:2006 using the Palmgren-Miner Rule. In the range of endurance limit, you can also select a modified form of the S-N curve (Woehler lines) instead of ISO 6336 or DIN 3990. The system service life and the minimum service life of all the gears used in the configuration is displayed. You can size the service life using the button either with or without defining a load spectrum (see page II-308). You will find more detailed information about defining load spectra in section 14.19 (see page II-308).

NOTE

Only the ISO 6336 method includes a calculation for the service life.

18.3.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 18.3 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

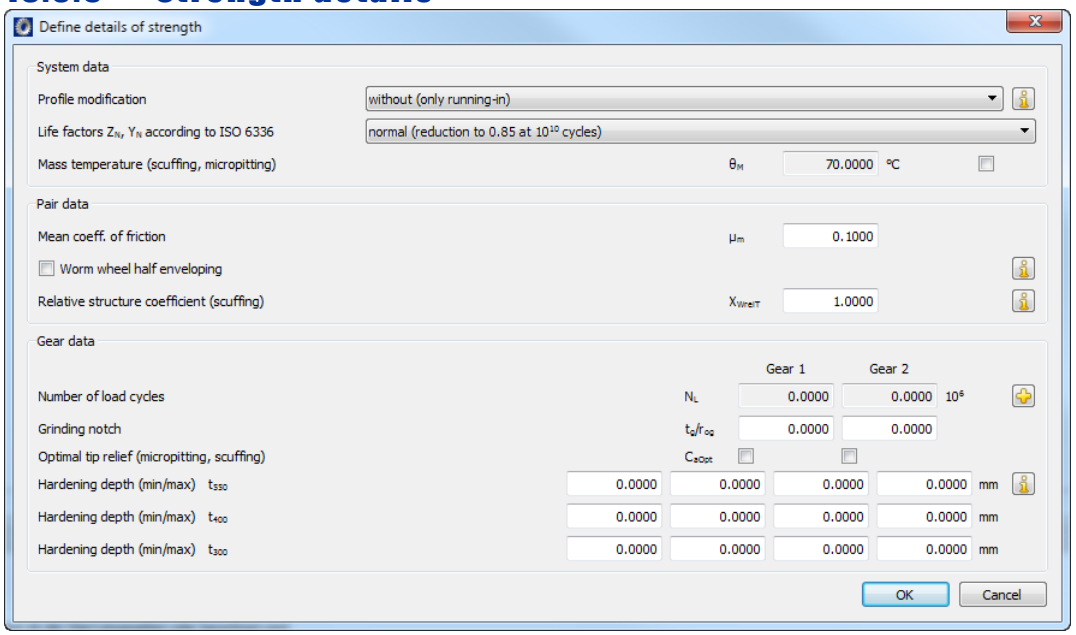
Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 18.3: Assignment of operational behavior to application factor

18.3.4 Power, torque and speed

Click the  button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-471) can be maintained. Click the  button next to the power input field to apply a frequency distribution for power, torque and speed in the `Define load spectrum (see page II-308)` window.

18.3.5 Strength details



Click the `Details...` button to open the `Define details of load` window, which is divided into `System data`, `Pair data` and `Gear data` areas.

18.3.5.1 Profile modification

You can modify the theoretical involute in high load capacity gears by grinding/polishing the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft Module Z15 (see section "Modifications" on page II-361). The type of profile modification has an effect on how scuffing safety is calculated. The force distribution factor X_Γ is calculated differently according to the type of profile modification used. The main difference is whether the profile has been modified or not. However, the differences between `for high load capacity gears` and `for smooth meshing` are relatively small. The strength calculation standard presumes that the tip relief C_a is properly sized, but does not provide any concrete guidelines. The resulting force distribution factor X_Γ, according to DIN 3990, depends on the type of profile modification:

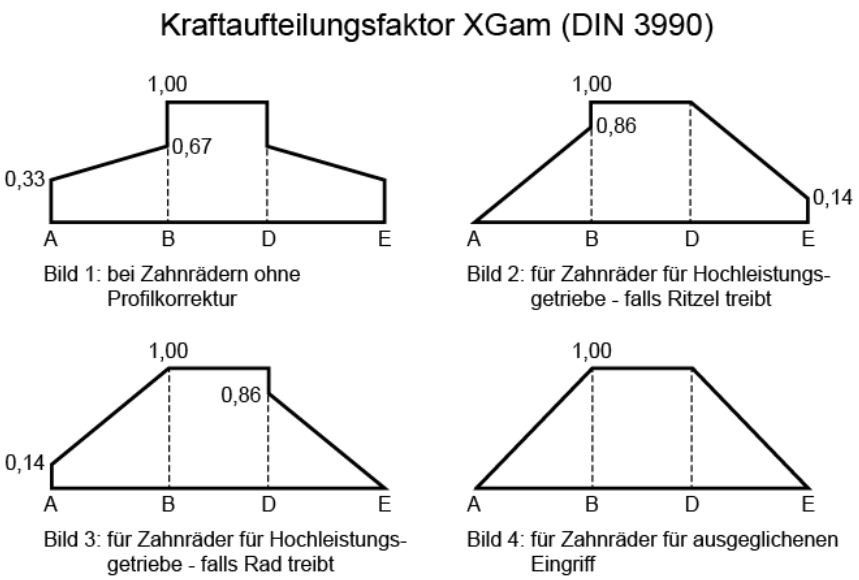


Figure 18.9: Load sharing factor X_Γ for different profile modifications

18.3.5.2 Lifetime factors as defined in ISO 6336

The fatigue limit factor Z_{NT} reduces the permitted material stress according to ISO 6336-2:2006:

$$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim} \tag{12.14}$$

$$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim} \tag{12.15}$$

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safety values for the range of endurance limit when compared with DIN 3990.

1. **normal (reduction to 0.85 at 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. Fatigue strength factors Y_{NT} and Z_{NT} are set to 0.85 for $\geq 10^{10}$ load cycles.
2. **increased with better quality (reduction to 0.92):** Y_{NT} and Z_{NT} at $\geq 10^{10}$ load cycles are set to 0.92 (in accordance with the data in ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

18.3.5.3 Structural factor XwrelT or structural factor Xw (scuffing)

The structural factor takes into account differences in materials and heat treatment at scuffing temperature. Structural factor X_{wrelT} (in DIN 3990 and in ISO TR 13989-2) or structural factor X_w (in ISO TR 13989-1) is used, depending on which standard is used. However, in this case, $X_{wrelT} = X_w/X_{wT}$ and $X_{wT} = 1$ applies. This results in $X_{wrelT} = X_w$. The two factors are identical.

However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this factor, yourself because it is not set automatically by KISSsoft.

Through hardened steels	1.00
Phosphated steel	1.25
Coppered steel	1.50
Nitrided steel	1.50
Case-hardened steels	1.15 (with lower than average austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)
Stainless steels	0.45

Table 14.9: Structural factor as defined in DIN 3990, Part 4

The standard does not provide any details about how to proceed when the pinion and gear are made of different material types. In this case it is safer to take the lower value for the pair.

18.3.5.4 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the `Number of load cycles for gear n window`. Click the  button to access this. In this window you can select one of five different options for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the service life, speed, and number of idler gears.
2. **Number of load cycles** Here you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation to ensure this value is taken into account.

3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear unit with three planets, enter `3` for the sun and `1` for the planets in the input field.

   Note:
   If the `Automatically` selection button in the calculation module is selected, KISSsoft will determine the number of load cycles in the `Planetary stage` calculation module, while taking into account the number of planets.

4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.

5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/100 are set.

NOTE

This calculation method is used for transmissions with a slight oscillation angle.

This scenario assumes that a reduction is present

$$i = \frac{z_2}{z_1}$$

and an oscillation angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements by the angle value w. The effective endurance is given as the service life. The two coefficients N_1 and N_2, which reduce the absolute number of load cycles, N_L, are now calculated. To do this:

a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, a complete forwards/backwards movement is counted as a load cycle.

b) For the pinion, coefficient N_1 is determined as follows:

$$N_1 = 0.5 \qquad \text{für } w \cdot i > 360$$
$$N_1 = 0.5 \cdot \max(1.0, 360 / w / i) \qquad \text{für } w \cdot i \le 360$$

c) The number of load cycles of teeth in contact in gear 2 is smaller by a coefficient of N_2 when compared with the number of load cycles during continuous turning.

$$N_2 = 0.5 \cdot 360 / w$$

The coefficient 0.5 takes into account both the forwards and backwards movements.

d) Enter coefficients N_1 and N_2 in the `Load cycles per revolution` input field.

The correct number of load cycles can now be calculated on the basis of the data entered in steps a to d.

18.3.5.5 Optimal tip relief

To calculate safety against micropitting as specified by Method B in ISO 15144, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large discrepancies, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and documented accordingly in the report.

18.3.5.6 Hardening depth, known by its abbreviation "EHT"

You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final machining (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [93]. Here a minimum value of t400 (nitrided steel) or t550 (all other steels) is used. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the section in [93] "Subsurface fatigue". The values required to define the EHT coefficient YC as specified in DNV 41.2 are also needed. The calculation is performed using different solutions than the calculation of the proposal for the recommended hardening depth, but still returns similar results (proposal for hardening depth). To obtain a

proposal for a sensible hardening depth, we recommend you call the calculation in `Report->Proposals for hardening depth`. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

18.3.5.7 Load spectra with negative elements

Load spectra with negative load bins (T < 0 and/or n < 0) can also be calculated as follows (this is only applied to bins whose alternating bending factor is YM=1.0).

IMPORTANT:

A load spectrum is considered to be negative if the non-working flank is placed under load.

Coefficient for torque	Coefficient for speed	Flank under load	Actual load bin
+	+	Working flank (*)	Evaluated as positive
+	-	Working flank (*)	Evaluated as positive
-	+	Non-working flank	Evaluated as negative
-	-	Non-working flank	Evaluated as negative

(*) Working flank as entered in the Strength tab

Table 14.10: Evaluation of a load bin, depending on the prefix operator

Under "`Details`" in the "Strength" section of the "`Load`" tab, you can select the following:

- For calculating pitting safety
  - Evaluate all negative load bins as positive (as up to now)
  - Consider only positive load bins
  - Consider only negative load bins
  - Check both cases and document the unfavorable case
- For calculating the tooth root safety
  - Evaluate all negative load bins as positive (as up to now)

- For negative load bins, increase root stress by 1/0.7
- Increase bending stress for positive load bins by 1/0.7
- Check both cases and document the more realistic case

18.4 Settings

In the `Calculation` menu you will find the `Settings` option. Click this submenu to open the `Module-specific Settings` window. From here, you can access the tabs listed below to input other calculation parameters (the following parameters not described here (see page II-456)).

18.5 Notes

18.5.1 Checking the contact pattern

The collision check shown in the 2D graphic (Meshing (see page II-637)) can only be used to a limited extent for crossed helical gears because it only works for a shaft angle of 90°, does not take flank line modifications into account, and only represents generating in the axial section.

A better option here is to generate a 3D model which includes all the flank line modifications and works for any shaft angle. The "Skin model" 3D variant can be used to represent the contact pattern, so you can check the contact pattern accurately, when the gears are being generated. To do this, click the appropriate function button to rotate one gear slightly against the other until the contact pattern appears, and then generate the two gears. To ensure the gears do not engage too fully, we recommend you set the number of rotation steps to 30 or higher (in Properties).

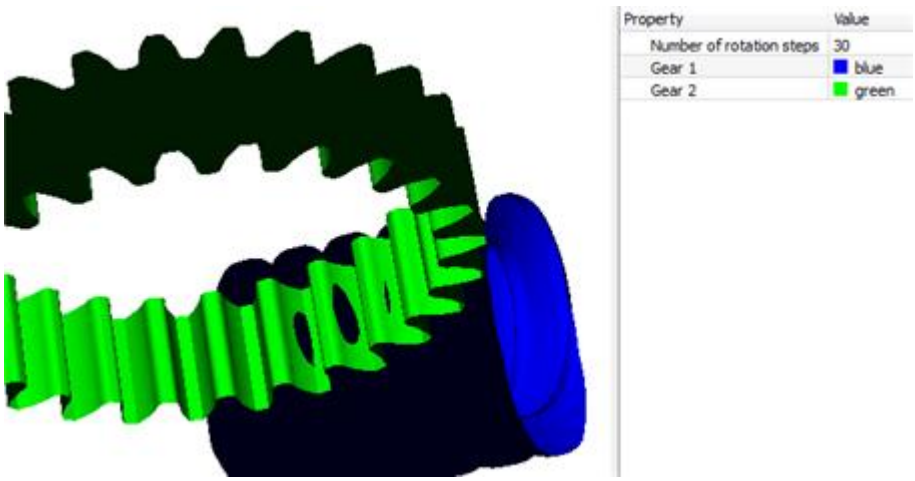


Figure 18: Contact pattern of a worm gear toothing

Chapter 19
Beveloid Gears

Beveloid gears, also known as conical gears, are generated by a rack-like cutter/tool which is tilted by a predefined angle (see K.Roth, Zahnradtechnik – Evolventen-Sonderverzahnung [79]).

Beveloid gears are primarily used in two particular areas: to generate a shaft angle between two meshing gears. Alternatively, two beveloid gears with opposing cone angles can be used to generate backlash-free toothing.
Beveloid gears with a shaft angle can be used to achieve a compact type of gear unit.

Unfortunately, no standards or guidelines have yet been drawn up for the calculation of the complex geometry, or for strength.
For this reason, the geometry calculation method used in KISSsoft is based on standard technical literature and publications. The main data used is taken from the publications mentioned in the next section.

For simplicity's sake the strength in the mid section is calculated as if for a cylindrical gear pair.

19.1 Underlying principles of calculation

The basic calculation of the geometry and tooth form for a single beveloid gear is based on K.Roth [79], and on well known standards for cylindrical gears (e.g. DIN 3960, DIN 867, etc.).

Therefore, the beveloid gear is generated using the same process as a cylindrical gear, except that the profile shift changes along the facewidth. And this therefore changes all the parameters which are affected by the profile shift.

When spiral toothed gears are involved, the cutter is not only tilted by a cone angle θ, but also by an additional helix angle β. In the transverse section, this creates a trapezoidal reference profile with different pressure angles α on the left and the right side. This has a significant effect on the tooth form, because it changes the base circles.

The changes to the profile shift across the facewidth mean that beveloid gears often run the risk of undercut at the root or having teeth with a pointed tip. The profile shift at the toe and heel is calculated by

$$x^*_{\min,\max} = x^*_m \mp \frac{b \bullet \tan\theta}{2 \bullet m_n}$$

The undercut limit and minimum top land are only output in the error message if the values are exceeded with the data that has just been entered. As the two sizes on the left and right may be different (in the case of helical gear teeth), the system displays the more unfavorable value in each case.

The beveloid pair's meshing conditions are calculated on the basis of the publications by S. J. Tsai [98] [99]. In this case it is important to note that the parameters are sub-divided into manufacturing and working parameters ("Manufacturing data and working data" section).

19.2 Basic data

19.2.1 Normal module

You can enter the normal module here.

However, if you know the "Pitch", "Transverse Module" or "Diametral Pitch" instead of this, click on the conversion button to open a dialog window, in which you can perform the conversion. If you want to transfer the "Diametral Pitch" instead of the normal module, you can select "Input normal diametral pitch instead of normal module" by selecting Calculation > Settings > General".

19.2.2 Pressure angle at normal section

This entry relates to the reference profile's flank angle. The normal pressure angle on the beveloid gear's reference circle is dependent on the cone angle and helix angle. [79]

19.2.3 Helix angle

Here you can enter the helix angle, or else select a spur gear toothing. The helix angle entry only applies to gear 1. Gear 2 may have a different helix angle value from gear 1, and is calculated. For toothing with total profile shift 0, the following equation applies for determining the second helix angle from the entered parameters:

$$\cos\sum = \cos\theta_1 \cos\theta_2 \cos(\beta_1 + \beta_2) - \sin\theta_1 \sin\theta_2$$

19.2.4 shaft angle

You can specify the shaft angle between the two axes of rotation here.

The shaft angle between any two straight pitches can be determined from the scalar product of the direction vectors of the two straight pitches. This corresponds to the angle between the two straight pitches in the plan view along the distance vector between the two straight pitches.

19.2.5 Number of teeth

The number of teeth defines the transmission ratio of the gearing. Only even numbered, positive values are permitted.

19.2.6 Width

Facewidth of the gears. Please note that, when the width and cone angle are very large, the profile shifts between the toe and heel may be very different. For this reason, you cannot input just any value for the width, because this might, for example, create a tooth that is too pointed.

At present, you cannot specify an axial offset. This means the gear pair contact is always in the middle of the gear.

19.2.7 Cone angle

The specified cone angle corresponds to the manufacturing parameter used to set the misalignment of the milling cutter to the gear. Both positive and negative cone angles are permitted, however, the total cone angle must be at least 0.

19.2.8 Profile shift coefficient (center)

The profile shift coefficient is defined in the same way as for a standard cylindrical gear, but the value relates to the value at the middle of the beveloid gear. When this calculation is performed, the Results window displays the size of the profile shifts at the toe and heel of the gear.

19.2.9 Quality

The quality achieved when generating the beveloid gear.

19.2.10 Material and lubrication

The entry is the same as the normal entry, as for cylindrical gears.

19.3 Reference profile

In the "Reference profile" tab, you can either input the reference profile for the manufacturing process in the same way as for a cylindrical gear calculation, or define the tools directly.

In this case, you must modify the height in the reference profile as follows to calculate the tooth form in transverse section (see K.Roth [79], section 5.2.6):

$$h_{aC} = \frac{h_{aP}}{\cos\theta} \quad , \quad h_{fC} = \frac{h_{fP}}{\cos\theta}$$

Here, the subscript C represents the heights in the transverse section of the beveloid gear (calculated values) and P represents the heights of the reference profile (input values). You can check these values in the main report by selecting "Summary / Reference profile / Gearing".

19.4 Modifications

The selection options for modifications in the beveloid gear module are limited.

In general, the contact pattern for beveloid gears with a shaft angle that is not 0 improves if negative crowning is used. To do this, you can input the "Crowning" modification and define a negative value.

19.5 Factors

The face load factor $K_{h\beta}$ cannot be calculated automatically for beveloid gears, and must therefore be set by the user. A value of 1.5 is used by default.

19.6 Dimensioning

As far as we know, no standards or research projects have yet been completed which involve the calculation of loads on beveloid gears. For this reason, the calculation of strength is performed using an equivalent cylindrical gear toothing in the mid section.

Note that the value for $K_{h\beta}$ in particular, can differ a great deal from the values in the common gear standards. For this reason, the factor must be entered manually.

Minor differences may occur in the calculated safeties produced during cylindrical gear calculation and beveloid gear calculation, which are caused by a slight difference in the way the contact ratio is calculated.

19.7 Manufacturing Data and Working Data

We perform the calculation of the beveloid pair according to J. Tsai [98], so it is important to know the difference between "manufacturing data" and "working data".

Manufacturing data is the data that is decisive for manufacturing. This category includes the values that the user enters in the 'Basic data' tab. In contrast to this is the working data, which relates to the generation geometry of the beveloid gears that are in use. An example is the cone angle θ of the angle at which the tool is tilted during manufacturing. In contrast, the working cone angle θw is the angle of the pitch cone of the beveloid gears in the meshing.

The working data is required to calculate a correct pairing, at which the contact point of the gears is in the middle of both beveloid gears. For example, if all the other parameters result in the helix angle value β_w from gear 2 at the operating point, this is then converted into a helix angle β for the manufacturing process.

The working data is also needed to position the two gears relative to each other. To position a gear pair in a 3D CAD environment, gear 2 is positioned relative to gear 1 as follows:

1. Displacement along the Y-axis at r_{w1}
2. Rotation around X-axis with θ_{w1}
3. Rotation around negative Y-axis with $\beta_{w1} + \beta_{w2}$
4. Rotation around X-axis with θ_{w2}
5. Displacement along the Y-axis at r_{w2}

Chapter 20
Non-Circular Gears

Using KISSsoft's non-circular gear analysis, you can calculate gears with non-circular gear bodies.

20.1 Input data

Input the geometry, generation and tolerance values in the **Basis data** tab.

Then, enter the details for generating non-circular gears in the **Reference profile** tab.

20.1.1 Geometry

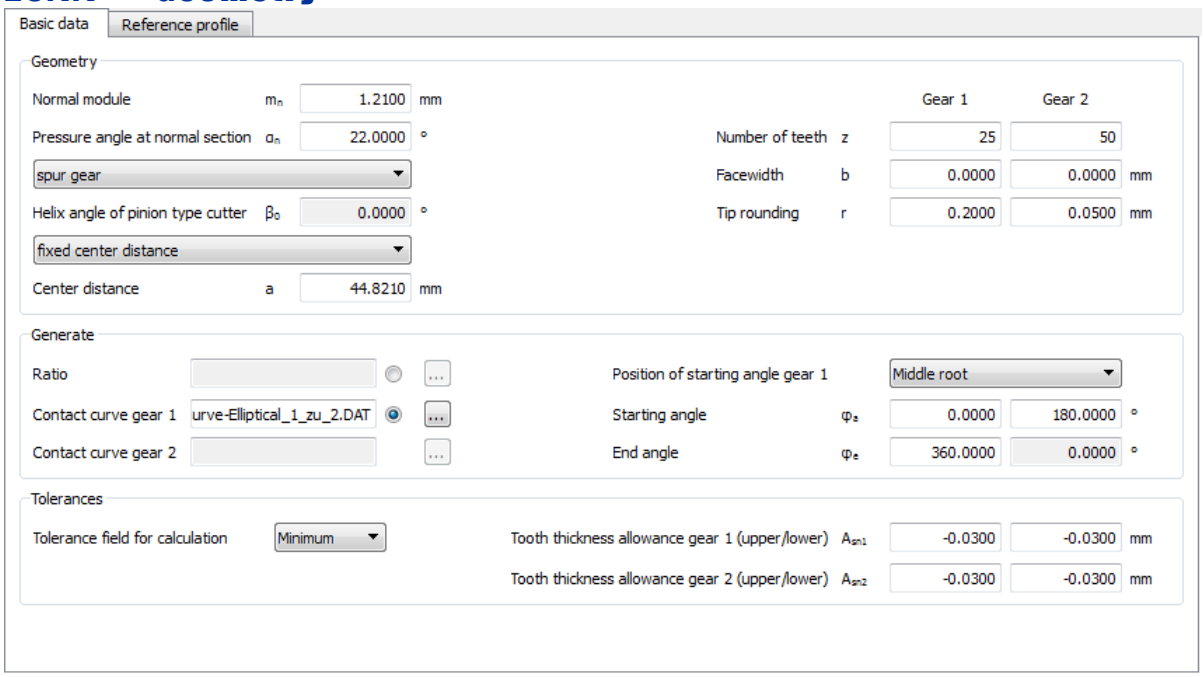


Figure 20.1: `Basic data` tab: entries for a non-circular gear pair

The module is defined from the "**Results window**" (total length of operating pitch line/[number of teeth* π]=module).

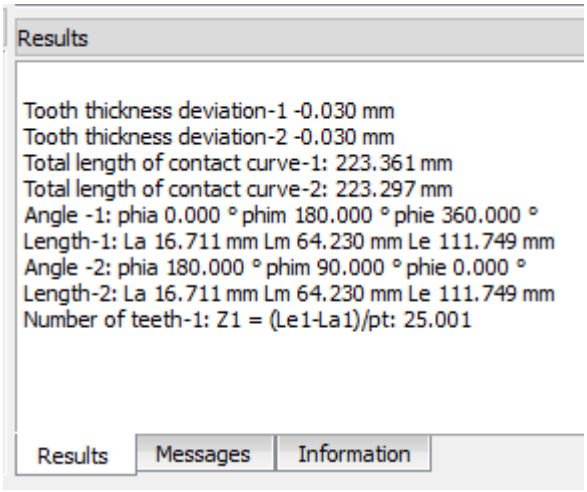


Figure 20.2: Results window

To save time in the first phase of the sizing process, we recommend you do not enter the total number of teeth z. We suggest you perform the calculation with a lower number of teeth (e.g. 2). In this case, although all the operating pitch lines are calculated completely, only the specified number of teeth (2) are calculated and displayed.

Initially, start the calculation with a pressure angle in the normal section α_n of 20°. Later on you can change this angle instead of the profile shift or to optimize the tooth form.

20.1.1.1 generation

The start and end angles φ_a and φ_e are important values because they determine the operating pitch line of gear 1, i.e. the area that will be generated. In closed curves the angle φ_a is 0° and φ_e is 360°.

The operating pitch lines or the ratio progression are then defined in files. The files must be in either "**dat**" or "**dxf**" format. These files can be stored in any directory.

It is important to register these files correctly, using the [...] button.

Operating pitch lines are also stored in the *.Z40 file. As a consequence, when you load a new calculation, you do not need to access the *.dat file. In this case you see a message to tell you the file cannot be found, and existing data will be used instead.

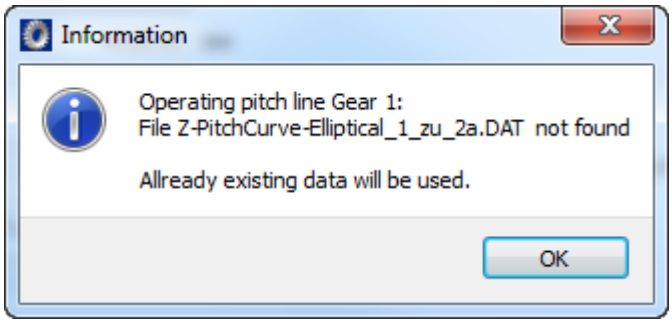


Figure 20.3: Message

NOTE

The progression (ratio or operating pitch line) must be defined from at least the starting angle to the end angle. To achieve clean toothing for the curve, the curve must have approximately 30° forward motion and follow-up movement. If the curve has no forward motion and/or follow-up movement, the software extends it automatically.

Input format for data in imported files

You can predefine one or two operating pitch lines or the ratio progression. The imported files must have ".dat" as their file extension.

A maximum of 7800 lines can be processed during non-circular gear calculation. Lines that start with # are comments and are ignored. To predefine the ratio progression, input the angle on gear 1 and the ratio.

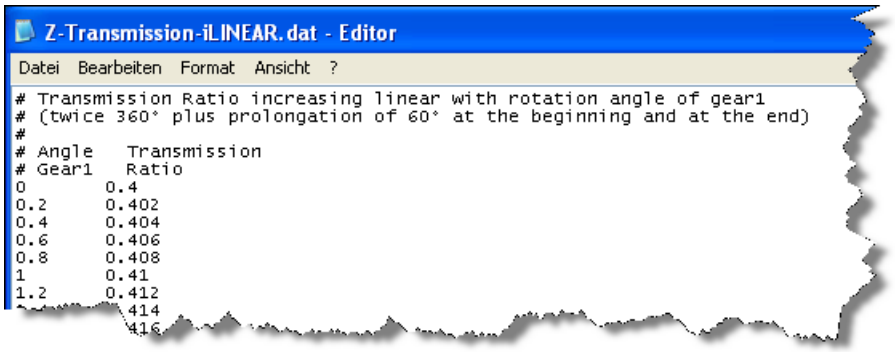


Figure 20.5: Example of ratio progression

To predefine the operating pitch line progression, input the radius and the angle.

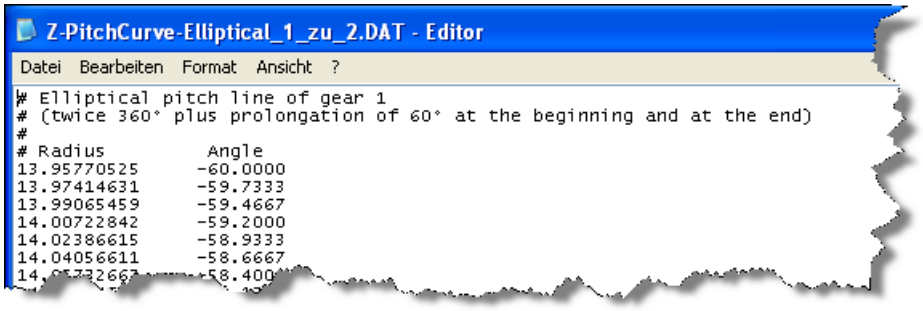


Figure 20.6: Example of an operating pitch line

20.1.2 Tolerances

We recommend you enter sufficiently large tooth thickness allowances A_{sn} (e.g. -0.10/-0.12 for module 2).

20.1.3 Reference profile

You must specify a topping pinion type cutter. The same pinion type cutter is usually defined for both gear 1 and gear 2.

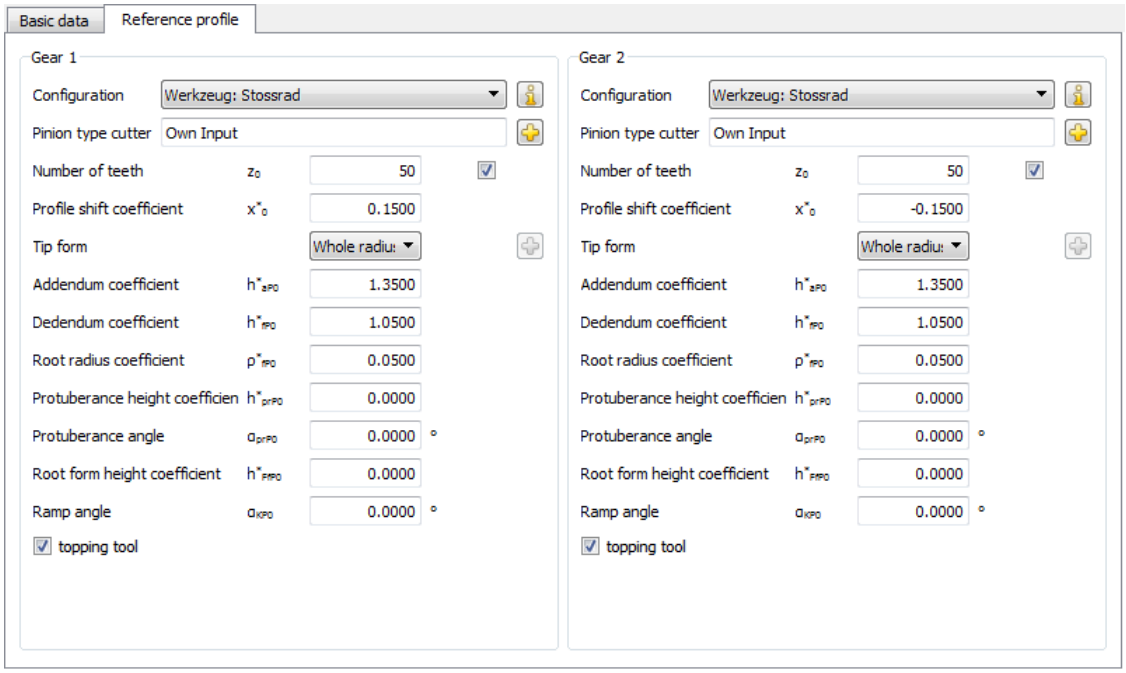


Figure 20.6: `Reference profile` tab: entries for a non-circular gear pair

Problems may arise unless the profile shift coefficient of the pinion type cutter is set to 0. You must then carefully check exactly how the gears are generated.

20.2 How to use KISSsoft

20.2.1 Angle error

When you input a closed curve (gear 1), using a operating pitch line or gear reduction progression, it must start at 0° and finish at 360°. For this reason, the rotation of gear 2 must also be 360° (or a multiple of this). If not, this will result in an error.

Starting angle	φ_a	0.0000	0.0000	°
End angle	φ_e	210.0000	-183.2583	°

Figure 20.7: Minor error for Gear 2: φ_e is 179.9489 instead of 180°

However, this error has no effect because the predefined intermeshing allowance is large enough.

20.2.2 Checking the meshing

A useful way of checking the meshing is to change the number of rotation steps (per 360°) to rotate the gear in larger or smaller steps. You change the step sizes, as usual, in the Graphics window.

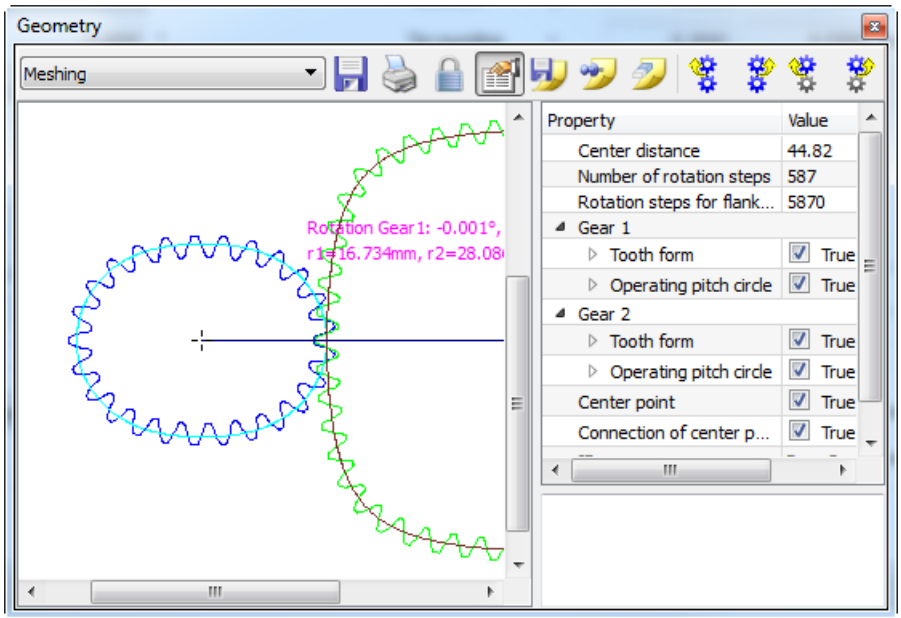


Figure 20.8: Changing the rotation steps

When you generate gears with allowances, we recommend you click the  button to bring the gears into flank contact with each other.

NOTE

If, when you click the "`Rotate independently to the right`" button, one gear rotates too far (or not far enough) against the other, you must adjust the number of "rotation steps" accordingly!

20.2.3 Improve tooth form

You can change the tooth form of circular gears quite significantly by changing the profile shift. In the current version of the program for non-circular gears, we recommend you set the profile shift coefficient of the pinion type cutter $x^*_0=0$. Despite this, you can still modify the tooth form by changing the pressure angle α_n.

20.2.4 Accuracy of the tooth form

Select `Calculations -> Settings` to predefine the accuracy (and therefore also the size of the file) for an IGES or DXF export.

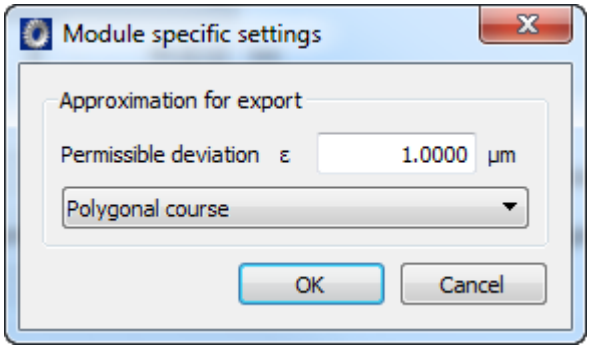


Figure 20.9: Module specific settings

This input only influences IGES or DXF files.

In the program, the tooth form (for each flank) is calculated with 100 points. You will find these results in the TMP files (and in the report). If you want to modify the number of internally calculated points, simply change the corresponding entry in the *.Z40 file:

Go to a saved *Z40. file and search for the lines:

ZSnc.AnzPunkteProFlanke=100;

and enter, for example, 40 instead of 100. As a result, only 40 points per flank will be calculated.

20.2.5 Export individual teeth

Go to a saved *Z40. file and search for the lines:

ZRnc[0].AusgabeKontur=0, for Gear 1 or

ZRnc[1].AusgabeKontur=0, for Gear 2.

There, change the variable to the required value, e.g. ZRnc[0].outputcontour=3.

The LEFT flank of the x-th tooth space (therefore the 3rd gap of Gear 1 in the example) is always output.

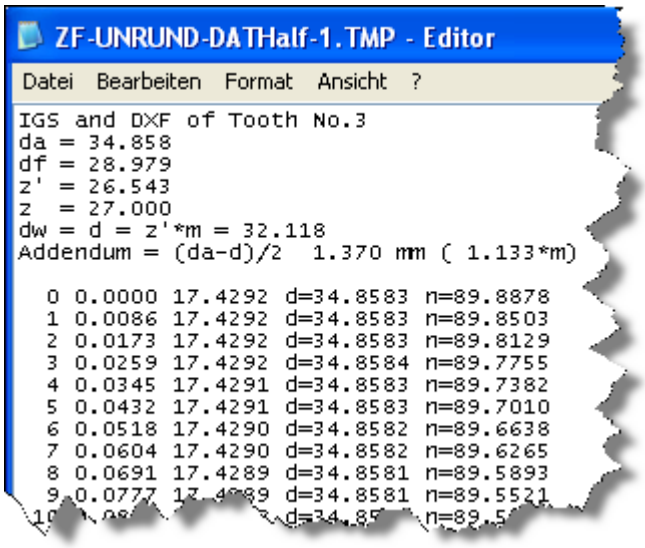


Figure 20.10: Temporary file for exporting teeth (ZRnc[0].outputcontour=3, for Gear 1)

20.2.6 Report

If you select Detailed in `Report settings`, this report will also be very extensive. If you want a shorter version, set "Extent of data" to 5 (standard).

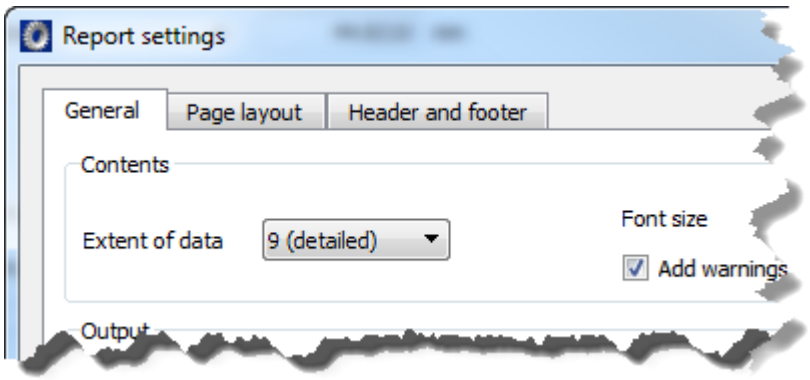


Figure 20.11: Report settings with a changed data scope for output to a report

20.2.7 Temporary files

When a calculation is performed, KISSsoft automatically generates temporary files. The directory in which these files are generated by KISSsoft must be specified in KISS.ini in the "Path" section. You will find KISS.ini in the KISSsoft main directory. Before changing the default setting you must ensure that you have read and write permissions for the changed directory. You will find more detailed information in Chapte 2 of the manual, "Setting Up KISSsoft".

ZF-H1_Gear 1 (step 1).tmp: ZF-H1_Gear 2 (step 1).tmp:	Insignificant, contains information about generating the pinion type cutter (cutter/tool)
ZF-H1_Gear 1 (step 2).tmp: ZF-H1_Gear 2 (step 2).tmp:	Not important information: contains details, flank for flank, about generating the non-circular gear
ZF-UNRUND-1.TMP:	Contains interesting information about operating pitch line 1: defining the contact points on operating pitch line 1, calculating operating pitch line 2 from operating pitch line 1, operating pitch line lengths, documentation about the intermeshing (individual points) of non-circular gear 1 with X, Y, normal, diameter and angle
ZF-UNRUND-2.TMP:	Contains interesting information: documentation about the intermeshing (individual points) of non-circular gear 2 with X, Y, normal, diameter and angle
ZF-UNRUND-DAT-1.TMP: ZF-UNRUND-DAT-2.TMP:	Possible further uses of the intermeshing (individual points) X,Y coordinates
ZF-UNRUND-OPLINE-1.TMP:	Possible further uses of the operating pitch line (indivi-

ZF-UNRUND-OPLINE-2.TMP:	dual points) X- and Y-coordinates
Z-WalzKurve-1.TMP: Z-WalzKurve-2.TMP:	Possible further uses of the operating pitch line (individual points) r, ϕ coordinates (*) Has exactly the same format as the DAT file (see "Import format" section)
Z-OpPitchPoints-1.TMP: Z-OpPitchPoints-2.TMP:	Can be used in further calculations of pitch points for each tooth in r, ϕ coordinates

Chapter 21
Report Menu

21.1 Drawing data

To display the toothing data you require to add to a drawing, select `Drawing data`. Use the `Z10GEAR1?.RPT` file (for Gear 1), and the `Z10GEAR2?.RPT` file (for Gear 2), etc. = d/e/f/i/s for the required language) to modify the template to your own requirements.

All the angle data for the user-specific Z10GEAR1?.rpt ... Z10GEAR4?.rpt reports is given in degrees-minutes-seconds, and displayed in brackets after the decimal point.

For example the number 20.3529° is displayed as:

20° 21' 10" (20.3529)

21.2 Manufacturing tolerances

Click on the `Manufacturing tolerances` menu option to generate a report that displays all the manufacturing tolerances as defined in the ISO 1328, DIN 3961, AGMA 2000, AGMA 2015 and BS 436 standards.

Notes on how to calculate tolerances as defined in ISO 1328 and DIN 3961

This calculation is performed in accordance with the formulae documented in the standard. As specified in the standard, these formulae use the geometric average of the edge values in the corresponding area instead of the effective geometric values (such as module, reference circle, etc.).

Example:

As specified in ISO 1328-1:1995, the following area limits are predefined for module m_n:

0.5 / 2 / 3.5 / 6 / 10 / 16 / 25 / 40 / 70 mm

For module m_n = 2.1, the equations use the geometric mean value of the range limits 2 and 3.5 instead of the ratio 2.1. Therefore sqrt(2*3.5)=2.645.

Exceptions (the standards do not provide any notes about exceptions):

- If the size is less than the lowest range limit, the lower range limit is used as the value if the value >= 0.8*range limit, otherwise, the effective value is used.

- If the size is greater than the upper range limit, the effective value is used.

Example:

Module m_n = 85 -> 85 is used.

Module m_n = 0.44 -> 0.5 is used (0.8*0.5 = 0.4, which is less than 0.44).

Module m_n = 0.31 -> 0.31 is used.

Note:

In some circumstances, calculations performed in accordance with the formulae in ISO 1328 may return data that is slightly different from the tables specified in the standard. We have also discovered that the formulae in DIN 3961 may return slightly different values from the data given in the tables in DIN 3962 and 3963. However, the majority of the variations are at most 1μm.

21.3 Summary

You use the summary function to compare the current toothing with the results of `fine sizing`.

21.4 Service life

This report shows the most important data that is used to calculate service life either with or without a load spectrum (see page II-308). You can also call the service life calculation by clicking the Sizing button next to the Service life input field. This then displays the service life that should be achieved if required safeties are used.

21.5 Sizing of torque

The sizing of torque displays the most important data required to calculate the transmissible torque (or the maximum transmissible power) with or without load spectrum. You can also call the sizing of torque function directly by clicking the checkbox next to the Torque or Power input fields. You then see a value for the torque that should be achieved if required safeties are used.

21.6 Proposal for the hardening depth EHT

A wide range of different proposals for the hardening depth EHT as specified in the standards have been documented. The data specified in the ISO, AGMA and Niemann standards are often very different, because of the very rough approximations involved. The most accurate calculation, which uses the shear stress criterion from the Hertzian law to define the required hardening depth, is documented in the upper part of the report. You can also specify the safety factor which is to be used for the calculation (safety factor for calculating shear stress for EHT (see page II-470)). You will see this displayed as a graphic in the "Hardening depth" section.

Chapter 22
Graphics Menu

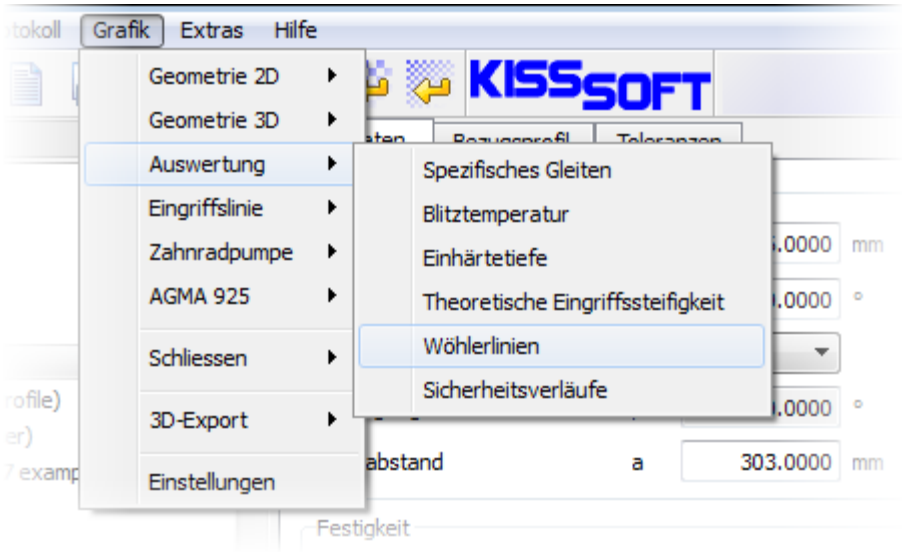


Figure 22: `Graphics` menu in the menu bar on the KISSsoft user interface

In the `Graphics` menu you can select various menu options to help you display toothing and functional processes.

NOTE

Click the right-hand mouse button to open a context menu that contains other operating functions.

Table 22.1 shows which of the options in the `Graphics` menu are supported by particular tooth calculation modules, and where you can find the relevant documentation in this section.

Menu option	Options	Section										
AGMA 925	Temperature in contact	22.1.1		✔	✔	✔	✔	✔				
	Lubricant film thickness			✔	✔	✔	✔	✔				

	Hertzian Pressure			✔	✔	✔	✔	✔					
	Specific thickness of film			✔	✔	✔	✔	✔					
Evaluation	Specific sliding	22.4.1		✔	✔	✔	✔	✔	✔				
	Flash temperature	22.4.2		✔	✔	✔	✔	✔					
	Contact temperature			✔	✔	✔	✔	✔					
	Hardening depth	22.4.3		✔	✔	✔	✔	✔	✔				
	Theoretical contact stiffness	22.4.7		✔		✔	✔	✔					
	S-N curve (Woehler lines)	22.4.4		✔	✔	✔	✔	✔	✔				
	Safety factor curves	22.4.5		✔	✔	✔	✔	✔	✔				
	Stress curve	22.5.9								✔			
	Path of contact (pinion/face gear)	22.4.8								✔			
	Safety scuffing	22.4.10								✔			
	Sliding velocity	22.4.10								✔			
	Oil viscosity	22.4.6	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
	Gaping			✔	✔	✔	✔	✔					
	face load distribution			✔	✔	✔	✔	✔					
	Tooth flank fracture			✔	✔	✔	✔	✔					
Contact analysis	Axis alignment	22.5.1 (see page II-653)	✔	✔	✔	✔	✔	✔					
	Specific sliding	22.5.8	✔	✔	✔	✔	✔	✔					
	Transmission error	22.5.2		✔	✔	✔	✔	✔					
	Transmission error acceleration	22.5.3		✔	✔	✔	✔	✔					
	FFT of the transmission error	22.5.4		✔	✔	✔	✔	✔					
	Normal force curve (line	22.5.5		✔	✔	✔	✔	✔					

	load)											
	Normal force distribution (line load)	22.5.5		✔	✔	✔	✔	✔				
	Torque progression	22.5.6		✔	✔	✔	✔	✔				
	Stiffness curve	22.5.7		✔	✔	✔	✔	✔				
	FFT of the contact stiffness	22.5.8		✔	✔	✔	✔	✔				
	Bearing force curve	22.5.9		✔	✔	✔	✔	✔				
	Bearing force curve in %	22.5.9		✔	✔	✔	✔	✔				
	Direction of the bearing forces			✔	✔	✔	✔	✔				
	Kinematics	22.5.10	✔	✔	✔	✔	✔	✔				
	Specific sliding per gear	22.5.11	✔	✔	✔	✔	✔	✔				
	Specific power loss	22.5.12		✔	✔	✔	✔	✔				
	Heat development	22.5.13		✔	✔	✔	✔	✔				
	Heat development along the tooth flank	22.5.13		✔	✔	✔	✔	✔				
	Flash temperature	22.5.15		✔	✔	✔	✔	✔				
	Lubricating film	22.5.16		✔	✔	✔	✔	✔				
	Specific thickness of film	22.5.16		✔	✔	✔	✔	✔				
	Safety against micropitting	22.5.16		✔	✔	✔	✔	✔				
	Stress curve	22.5.14		✔	✔	✔	✔	✔				
	Bending stress in root area	22.5.14		✔	✔	✔	✔	✔				
	Stress distribution on tooth	22.5.14		✔	✔	✔	✔	✔				
	Wear along the tooth flank	22.5.17		✔	✔	✔	✔	✔				
2D geometry	Meshing	22.2.4		✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Tooth form	22.2.1	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Tool	22.2.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Manufacture	22.2.3	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Profile diagram	22.2.5	✔	✔	✔	✔	✔	✔				✔	
	Tooth trace diagram	22.2.5	✔	✔	✔	✔	✔	✔				✔	

	Flank curvature radii	22.2.6	✔	✔	✔	✔	✔	✔					
	Angle of flank normal	22.2.7	✔	✔	✔	✔	✔	✔					
	Drawing	22.2.8	✔	✔	✔	✔	✔	✔	✔				
	Assembly	22.2.9		✔		✔	✔	✔	✔		✔		
3D geometry	Tooth system	22.3.1		✔	✔	✔	✔	✔	✔	✔	✔	✔	
	Tooth form	22.3.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
Graphics list		Graphics list	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
Manufacturing drawing		Manufacturing drawing	✔	✔	✔	✔	✔	✔					

Table22.1: Graphics menu in the KISSsoft interface menu bar.

- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear, - Three gears, - Four gears, - Bevel and Hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears and precision mechanics worms, - Splines (Geometry and Strength)

22.1 AGMA 925

22.1.1 Lubricant film thickness and specific oil film thickness

The lubricant film thickness h_e according to AGMA 925 is shown over the meshing cycle. Another figure shows the specific density of film λ, which is a critical value for evaluating the risk of micropitting. λ is the ratio of the lubricant film thickness to the surface roughness, expressed in simple terms.

22.2 2D geometry

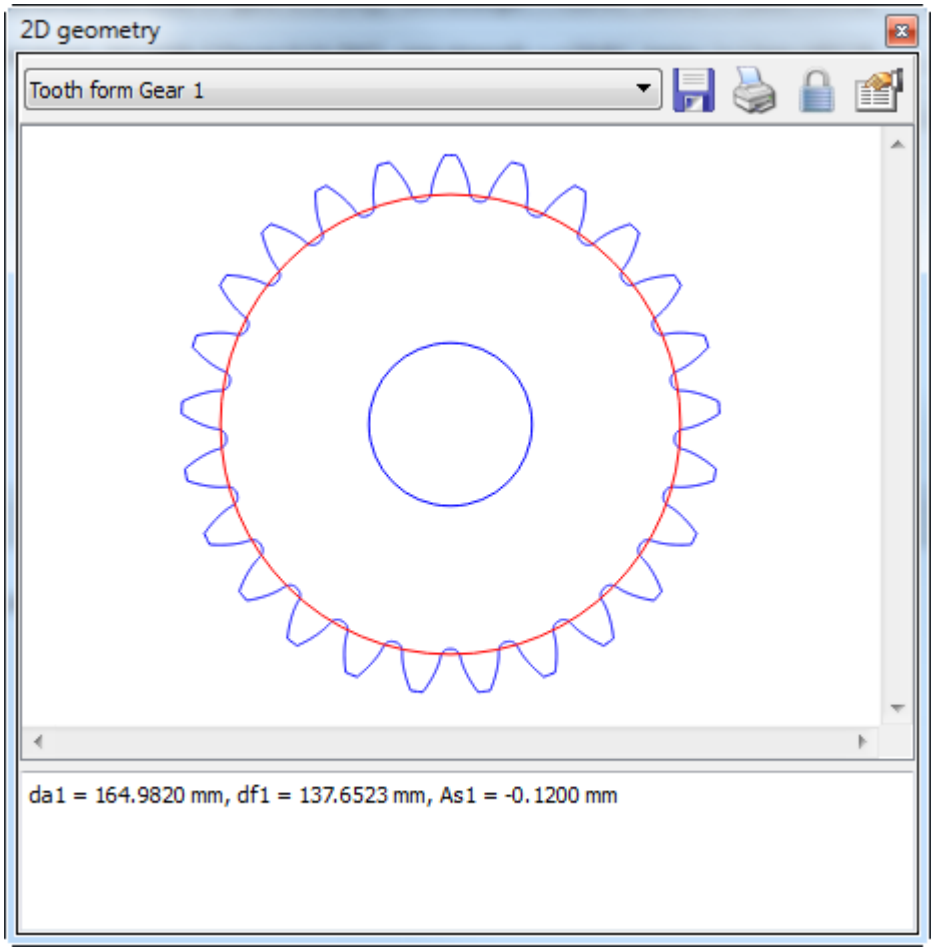


Figure 22.2: `Geometry` graphics window

You can select a number of different output options from the drop-down list in the tool bar in the `Geometry` graphics window (see Figure 22.2:

22.2.1 Gear tooth forms

Display a gear tooth form.

NOTE:

Click the Property button above the graphic to specify the number of teeth that are to be displayed. You can select whether to display it in transverse section, normal

section or axial section. Selecting the "Half tooth for export" option is also very useful if you want to export the tooth form and then reimport it into KISSsoft later on.

22.2.2 Gear tool

This displays the tool associated with the gear, if one is present.

22.2.3 Manufacturing a gear

Display the pairing: gear with cutter. Here the gear is shown in blue and the cutter in green.

22.2.4 Meshing

Displays the meshing of two gears.

NOTE ABOUT FACE GEARS:

In KISSsoft, the face gear is calculated by simulating the manufacturing process in different sections. You can display different sections at the same time. To do this, go to the `Property browser (PB)` in the graphics window and set the property in the `section` you require to `True` (see Figure 22.3).

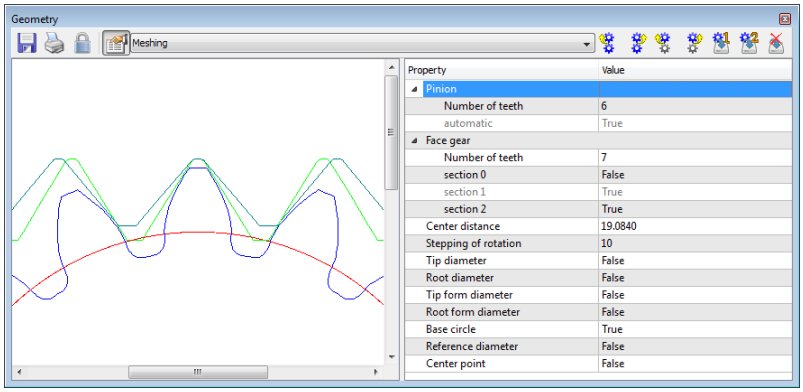


Figure 22.3: `Meshing` graphics window with `Property Browser`

The difference between the theory and the effective tooth form means that the tooth has an undercut! You can also see this clearly in the 2D view.

Collision check:

When generating two gears (in the graphical display) you can activate the collision display option. In the graphic, this shows (with squares) the points where the gears touch or where collisions may occur.

shown in brown: touch (between 0.005 * module distance and 0.001 * module penetration)

shown in red: collision (greater than 0.001 * module penetration)

The system identifies and marks collisions in all the meshing teeth. This option is particularly useful for analyzing the generation of non-involute tooth forms or measured tooth forms (using a 3D measuring machine) with a theoretical single flank generation check.

This function is also available for cylindrical gears and worm gears (but with restrictions for worm gears (see page II-603)).

NOTE:

If the "Make flank contact automatically" option is selected, you can only check the tooth meshing on contact. In this case, collisions are no longer displayed.

22.2.5 Profile and tooth trace diagram

These diagrams are generated by placing two lines diagonally over the tolerance band, as described in ANSI/AGMA: 2000-A88 (figures 1 and 2).

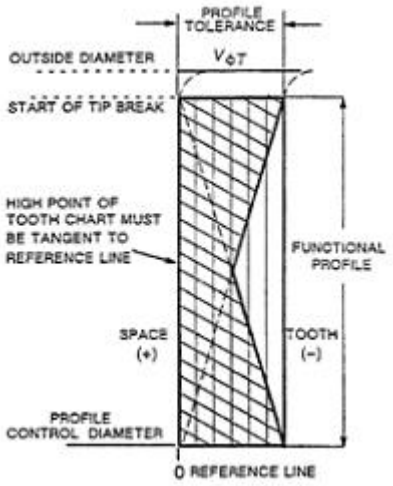


Figure 1 Profile diagram

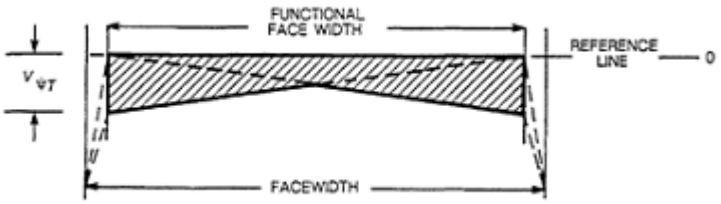


Figure 2: Tooth trace diagram

In the figures shown above, $V_{\phi T}$ is the profile tolerance and $V_{\psi T}$ is the tooth alignment tolerance which correspond to the profile total deviation (F_{α}) and the tooth helix deviation (F_{β}) as detailed in ISO 1328-1.

Although every company has its own method of creating profile and tooth trace diagrams, the AGMA is recognized as the standard in the industry. ISO TR 10064-1 (and ISO FDIS 21771) also include a general description of profile and tooth trace diagrams, however without any explanations about the construction method.

In KISSsoft the profile and tooth trace modifications are defined in the `Modifications` tab. The corresponding diagrams are then generated using this data.

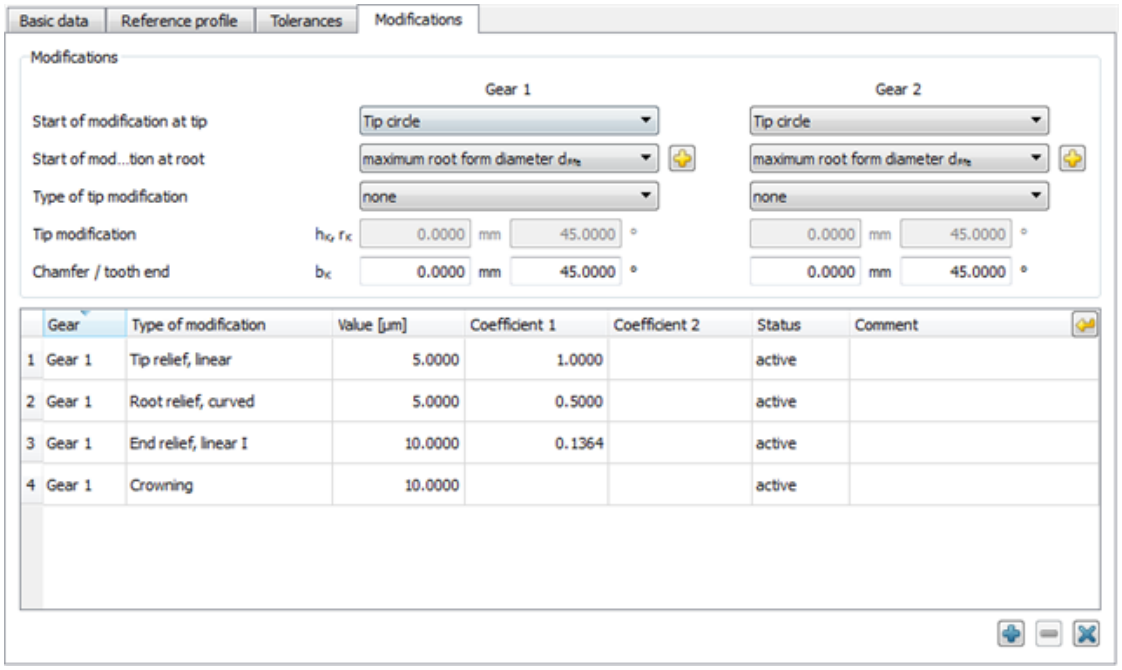


Figure 22.4: `Modifications` tab with modifications

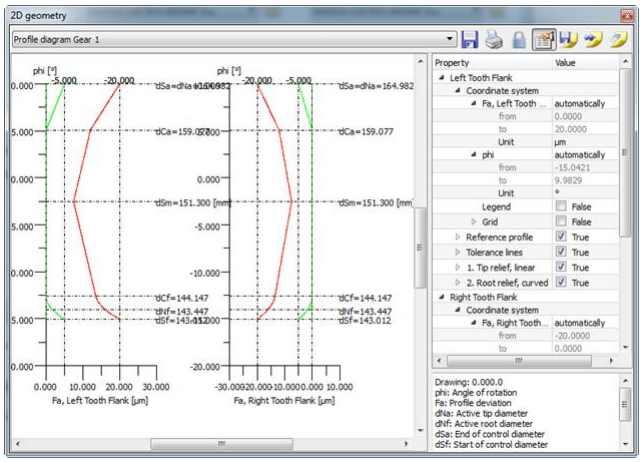


Figure 22.5: Profile diagram for gear 1 in accordance with the predefined modifications

The horizontal axis of the profile diagram shows the profile deviation values and the vertical axis shows the coordinates along the profile. You can select different values for the left-hand vertical axis (roll angle or path of contact length) (`Calculation→Settings→Contact analysis`). The values for the right-hand flank are always given as the diameter. You can also specify the tolerance type by clicking on `Calculation→Settings→General`. If you select the tolerance band type as specified in AGMA 20000-A88, the diagrams are generated according to the method mentioned above. If you set the tolerance band type to constant, the tolerance remains constant along the length or the width of the tooth flank. Click on the "Display profile in the middle of the tolerance band" checkbox to specify whether the central profile (see below) should usually be displayed.

Description of the specific diameter of the right-hand vertical flank:

- dSa: end diameter of the modifications (starting diameter of the modifications at the tip)
- dSf: starting diameter of the modifications (starting diameter of the modifications at the root)
- dCa: active tip diameter (starting diameter of the modification)
- dCf: tip form circle diameter (starting diameter of the modification)
- dCm: center point of the functional profile measured along the path of contact

NOTE:

The profile diagram is in the middle of the facewidth. The **Twist** profile modification is not possible.

Show curves in the diagram:

- green curve: Modifications of "1. Tip relief, linear" and "2. Tip relief, arc-like"
- blue curve: Reference profile (current function profile used for checking and generated from the total of the modified curves)
- red line: Tolerance curve generated by subtracting the profile total deviation from the reference profile. The profile deviation values are listed in the main report.
- green line (middle): Central profile, which can be entered as the target value for processing because it lies in the middle between the reference profile and the tolerance curve.

- gray lines: Tolerance range, which shows the range (as a crosshatched area) in which the actual manufacturing profile can lie.

The manufacturing profile (with tolerance) should lie between the tolerance curve and the reference profile.

You can use the properties to change the colors of the lines or to display or hide the individual curves.

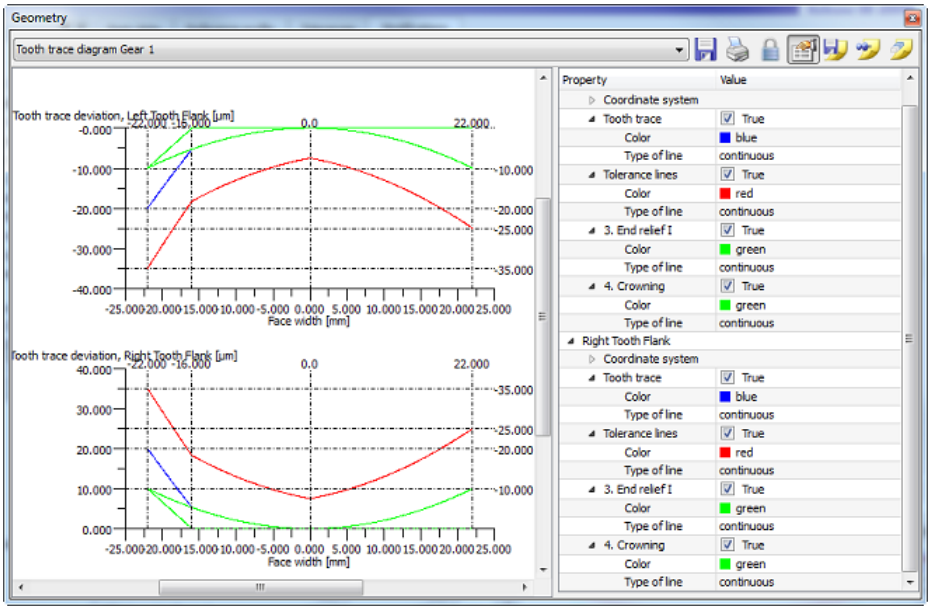


Figure 22.6: Tooth trace diagram for gear 1 with the predefined modifications

In the figure, the reference profile is shown in blue and the tolerance line is shown in red. The horizontal axis shows the coordinates along the tooth trace (facewidth) and the vertical axis shows the flank allowance as specified in the usual industrial conventions. The value of the total tooth trace deviation Fb is shown in the main report.

The manufacturing tooth trace (with tolerances) should lie between the tolerance curve and the reference tooth trace.

22.2.6 Flank curvature radii

In this graphic you see the flank curvature radii along the tooth flank. Along with the normal force, these are critical values for Hertzian pressure.

22.2.7 Angle of flank normal

The normal angle to the flank is shown in this graphic. Every point on the tooth form has a normal.

22.2.8 Drawing

Use this menu to display gears in diagram form. The gears are shown in transverse and axial section.

This option is primarily used for bevel gears and worms.

22.2.9 Assembly

Use this menu to create a diagram of how gears are assembled. The buildup (pair) of the gears is shown in transverse and axial section.

Two views, section and overview, are given for bevel gears with a shaft angle of 90°. For shaft angles <> 90°, only the section of the bevel gear pair is displayed.

22.2.10 Manufacturing drawing

22.2.10.1 General

Manufacturing drawings are designed to display a number of graphics on the same surface, and therefore create a print-ready image that can be used to manufacture a gear. You can also display the drawing data report at the same time. Use a control file to tailor the display to suit specific requirements. The control file is stored in the template directory (usually under KISSDIR\template). It has the module name and the file extension .grc (e.g. Z012gear1.grc).

You can also save the graphic generated here as a .dxf file in the usual way.

22.2.10.2 Editing the control file

By making changes to the control file you can modify the manufacturing drawing to suit your own requirements. The commands used to control the manufacturing drawing are described in the following table.

papersize: A4 papersize: A4 portrait	Specifies the required paper format. This refers to the standard terms used to describe commonly used paper sizes (A3,

papersize: 297, 210	A4, A5, B4, B5, Letter, Legal and Ledger), and also enables you to input your own dimensions for width and height. The default setting is for landscape format. However, you can switch to portrait format by entering the key word "portrait".
fontsize: 5	Specifies the required font size. The font size influences the size of the report and also the diagram titles.
units: inch	The default setting is that input values are assumed to be in mm. The system can handle these units: inch, mm and cm.

You can now add graphics that have specific characteristics. The table below gives an overview of the correct inputs.

draw 2DDiaProfileChart1	"draw" is the key word used to specify that a graphic is to be added. It is followed by the ID of the graphic you want to insert. The number at the end is part of the ID, and identifies the gear.
window: 160, 285, 0, 85	"window" identifies the window in which the graphic is displayed. NOT The values show the limits on the left, right, bottom and top.
scaletofit	This optional command forces the graphic to distort so that it fills the window in every direction. We recommend you do this for diagrams, but not for geometric figures. If this term is not used, the original size and shape of the graphic is retained.
fixedscale 2	This optional command generates a scaled output of the graphic. The number corresponds to the scale (in this case, 2 for M 2:1, 0.1 for M 1:10, etc.).

You can insert these graphics:

Tooth form	2DGeoToothDrawing
Drawing	2DGeoGearDrawing
Assembly	2DGeoAssemblyDrawing
Tool	2DGeoToolDrawing
Profile diagram	2DDiaProfileChart
Tooth trace diagram	2DDiaFlankLineChart
Angle of flank normal	2DDiaNormal

Finally, you can now display the report in the required location:

write report1	"write" is the key word used to create a gear data report. Enter report1 to select the gear data of gear 1, report2 to select the gear data from gear 2, etc.
topright: 297, 218	Unlike graphics, you must specify an alignment here. You define this with the first word. The correct commands are topright:, topleft:, bottomright: and bottomleft:. They represent the alignment (top right, top left, bottom right and bottom left). The next two values represent the particular reference point.

22.3 3D geometry

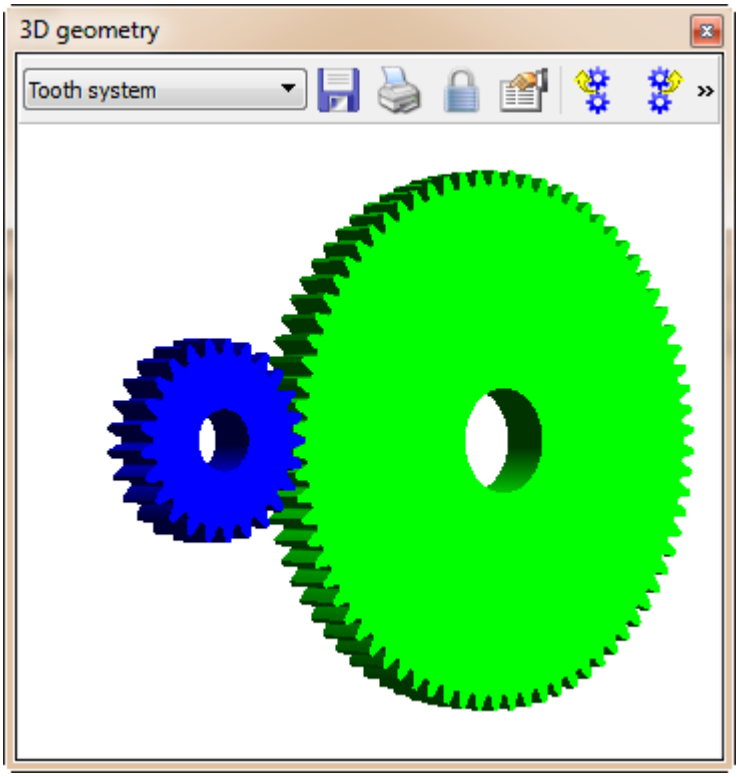


Figure 22.7: Graphics window `Tooth system`

The gears are displayed in the 3D Parasolid viewer.

You can select a number of different output options from the drop-down list in the tool bar in the `Geometry 3D` graphics window (see Figure 22.7). You can store the Parasolid viewer graphics in different file formats such as:

- Windows Bitmap (*.bmp)
- Joint Photographic Experts Group (*.jpg, *.jpeg)
- Portable Network Graphics (*.png)
- Standard for the Exchange of Product Model Data (*.stp, *.step)
- Parasolid Text File Format (*.x_t)
- Parasolid Binary File Format (*.x_b)

22.3.1 Tooth system

The tooth system displays the assembled system of gears in 3D.

You can display these gears in different views.

22.3.2 Tooth form

In the `Tooth form` menu, an individual gear is shown in 3D in the Parasolid viewer. There are the following restrictions on how these gears are generated. Only forms ZI and ZA can be generated for worm gears.

22.4 Evaluation

22.4.1 Specific sliding

The graphic in the Graphics window evaluation shows the specific sliding of the gears (ratio between sliding and tangential speed) over the angle of rotation. Different values can be displayed for each gear: gears without backlash, gears with upper center distance allowances (for lower tooth thickness tolerance) and gears with lower center distance allowances (for upper tooth thickness tolerance).

When you specify the profile shift (see section "Profile shift coefficient" on page II-267), click the  button to see a suggested value for balanced specific sliding.

22.4.2 Contact temperature

The contact temperature is the local temperature on the tooth flank at the moment of contact. It is displayed over the meshing. Based on the contact temperature values and its locations on the flank, appropriate action (e.g. profile modification) can be taken to reduce the temperature if necessary.

22.4.3 Flash temperature

The flash temperature graphic is displayed in the `Graphic` menu bar in the `Evaluation` graphics window.

The flash temperature is the increase in local temperature on the tooth flank at the moment of contact. It is displayed over the meshing. Depending on the values used for the flash temperature, and its position on the flank, a number of measures (e.g. profile modification) can be implemented to reduce the temperature.

22.4.4 Hardening depth

The hardening depth graphic is displayed in the `Evaluation` graphics window.

This calculates the optimum hardening depth (for case hardened or nitrided gears). It shows the stress progression in the depth vertical to the flank surface. This value is displayed directly in the HV values, because HV or HRC values are always used when specifying hardening depth and hardening measurements. If the materials database already contains values for a measured hardening progression, the hardening progression is displayed, accompanied by a warning message if the hardening properties are insufficient.

Proposed values for the recommended hardening depth are displayed in a special report, classified by calculation method, selected material and heat treatment process.

The various different methods are:

- The shear stress progression in the depth of the gear pair is calculated in accordance with Hertzian law. The shear stress is multiplied by a safety factor. (Enter this under "Settings". The default setting is 1.63). This defines the depth of the maximum shear stress (hmax). The program suggests the value 2*hmax as the hardening depth (EHT).
- For each individual gear in accordance with the proposals given in Niemann/Winter, Vol.II [65] (page 188)
- For each individual gear in accordance with the proposals given in AGMA 2101-D [1] (pages 32-34)
- For each individual gear according to the proposals given ISO 6336 Part 5 [44] (pages 21-23) (to avoid pitting and breaking up of the hard surface layer)

22.4.5 Theoretical contact stiffness

The graphic in the `Evaluation` graphics window shows meshing stiffness over the angle of rotation. The contact stiffness is calculated on the basis of the real tooth forms. The calculation takes into account tooth deformation, gear body deformation, and flattening due to Hertzian pressure. The calculation is performed according to Weber/Banaschek [69].

For helical toothed gears the overall stiffness is calculated with the section model (the facewidth is split into 100 sections and stiffness is added over all sections). See also [58], page 203. The transmission error is defined according to [65], and the transmission variation in the peripheral direction Δ:

$$\Delta s = \left(\frac{F_{bt}}{b}\right) \cdot \left(\frac{q}{c'}\right) \tag{22.5}$$

$$F_{bt} = \frac{F_t}{\cos(\alpha_t)} \tag{22.6}$$

where (q/c') is replaced by c_{gam}.

NOTE:

The theoretical contact stiffness and the contact stiffness of the effective toothing under load can be quite different.

22.4.6 S-N curve (Woehler lines) for material

The S-N curves (Woehler lines) graphic in the `Evaluation` graphics window displays the S-N curves (Woehler lines) for the tooth root and flank. The S-N curves are calculated using the selected calculation method for gears. The individual S-N curves are divided by a corresponding safety factor. The individual load spectra are also displayed in the same graphic.

If a load spectrum is taken into account when sizing the gears, the graphic also shows the curve for damage accumulation (not available for plastics).

22.4.7 Safety factor curves

The safety factor curves graphic in the `Evaluation` graphics window shows the progression of the safety factors over the service life. The safety factors are displayed for nominal operating conditions (i.e. without a load spectrum).

22.4.8 Oil viscosity, depending on temperature

The graphic shows kinematic viscosity at different oil temperatures.

22.4.9 Gaping

The graphic in the Evaluation graphics window shows a gap between the tooth flanks (in the direction of the path of contact) across the width of the contacting gears.

22.4.10 face load distribution

The graphical display in the Evaluation graphics window shows a line load across the width of the contacting gears.

22.4.11 Tooth flank fracture

The graphic shows the material utilization, the material's shearing strength, the equivalent shearing stress and the hardness curve for the selected gear.

22.4.12 Sliding velocity (face gear)

The face gear-sliding velocity graphic in the `Evaluation` graphics window shows the sliding velocity for the tip and root of the face gear.

22.4.13 Contact line (face gear)

The "Contact line (face gear)" graphic in the `Evaluation` graphics window shows the progression of the path of contact on the pinion and on the face gear.

22.4.14 Stress curve (face gear)

The graphic shows the stress curve (tooth root and flank) over the face gear's facewidth. The calculation splits the facewidth into individual segments, which can then be sized as rack pairs either according to ISO 6336, DIN 3990 or AGMA 2001. The calculation assumes a constant line load (which results in a slightly different torque for each segment due to the different pitch circle).

When you calculate data in order to represent the path of contact and the stress curve, the most important values are calculated in separate sections and saved to two tables. This data is stored in the Z60-H1.TMP and Z60-H2.TMP files.

22.4.15 Scuffing and sliding velocity (face gear)

The graphic displays scuffing safety for face gears. However, due to the very different sliding velocities and the changing contact stress across the tooth flank, calculating the safety against scuffing is actually very difficult. Akahori [2] reports massive problems with scuffing at higher sliding velocities. For this reason, it is appropriate to think about how to calculate the risk of scuffing. One sensible option, as described above for stress distribution, is to calculate scuffing safety in separate sections.

The figure in Kap18>.8 shows the progression of scuffing safety as defined by the flash and integral temperature criterion along the tooth flank. To achieve realistic results from this calculation, it must be ensured that every section is calculated with the same mass temperature. However, when you work through the calculation, you

will see there are significant changes in safety when the calculation is performed on the basis of the integral temperature. In particular, this happens as point E on the path of contact gets closer to the pitch point. If you then use the formulae in DIN 3990 to convert the flank temperature at point E to the average flank temperature the results you get will not be particularly precise. For this reason, we recommend you use the flash temperature as the criterion when you perform this calculation for face gears.

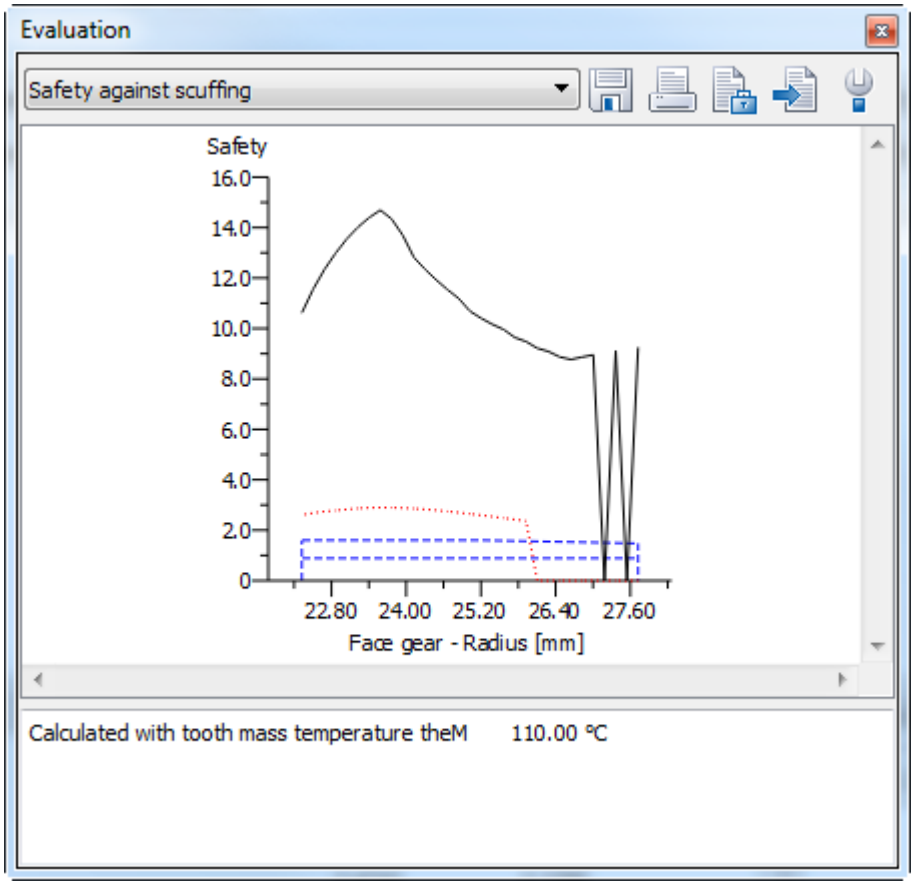


Figure 22.8: Graphics window `Safety against scuffing`

22.5 Contact analysis

NOTES:

The usual strength and speed calculations performed on gears assume that an involute tooth form is being used. However, if you use this program module, you can calculate and evaluate any type of toothing, such as cycloid toothing, just as accurately as involute tooth forms.

All the graphics can be exported:

- 2D diagrams as:
  - BMP
  - JPG
  - PNG
  - DXF
  - IGES
  - TXT
- 2D curves as:
  - TXT
- 3D diagrams as:
  - BMP
  - JPG
  - PNG
  - DAT (the y-axis is only output for the contact analysis if the "Draw data for path of contact" option is selected in the module-specific settings)

22.5.1 Axis alignment

Display the axis alignment of gear B relative to the axis of gear A. This display is a very useful way of checking the deviation error and inclination error of axis.

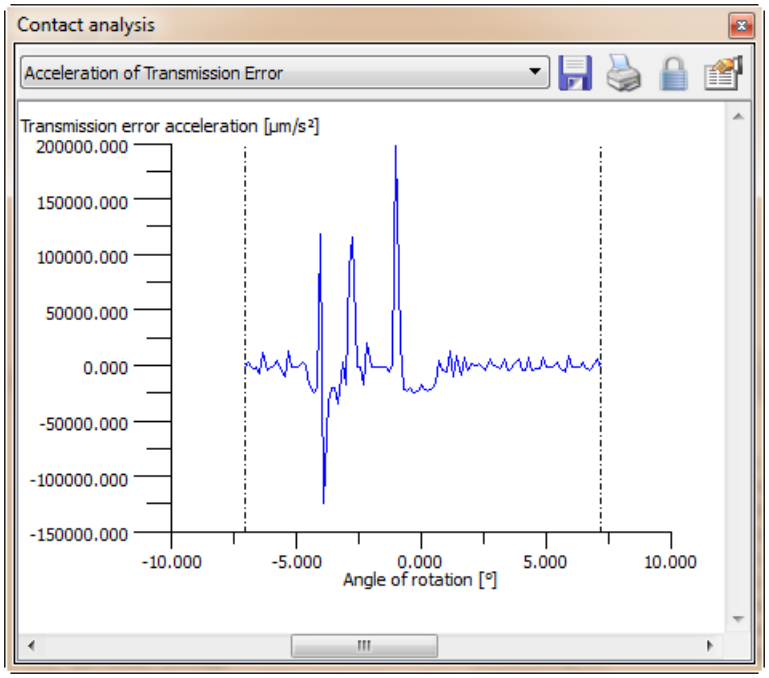
22.5.2 Transmission error

The path of contact under load is used to calculate transmission errors. The diagram shows the displacement of the contact point (μ) of the second gear on the length of the path of contact or the angle of rotation (°) of the driven gear.

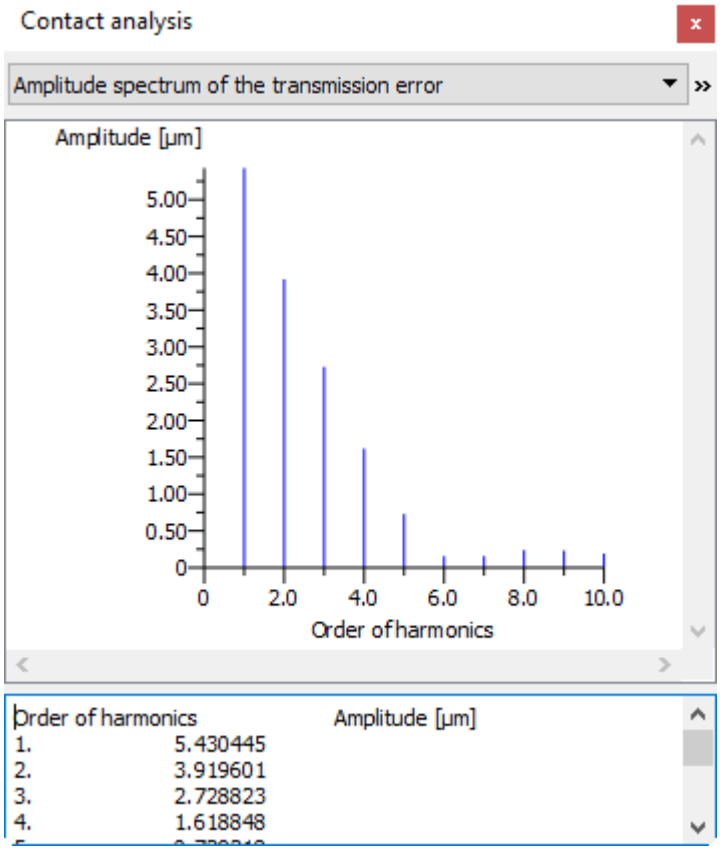
The amplitude of the transmission error plays a role in how much noise is generated but, despite this, you should not ignore the lead (how steep the slopes are), because high acceleration also generates high additional loads.

22.5.3 Transmission error acceleration

The Transmission error acceleration (second derivative with reference to time) is available as a graphic.



22.5.4 Amplitude of transmission error



The Amplitude spectrum of Transmission Error graphic displays the spectroscopic analysis results for the transmission error according to Fourier transformation.

You can compare the amplitudes of the spectra with the harmonic frequencies of transmission error in the comment window.

22.5.5 Contact lines on tooth flank

In this graphic you can examine the contact line along the facewidth. All the gear pairs in the meshing are shown at the same time in an engagement position.

22.5.6 Normal force curve

The normal force curve represents the line load per width for each tooth face in the middle of the cylindrical gear. In a well arranged profile modification, the normal force should increase steadily from zero. If you do not have a profile modification, an overlap length in the normal force curve shows the corner contact.

22.5.7 Normal force distribution

This graphic shows the normal force curve along the tooth flank and facewidth on a 3D gear.

22.5.8 Torque curve

The default value for torque defined in the main screen is kept constant during the calculation. The graphic then shows the torque for gear 1 and the torque for gear 2 divided by the ratio. If these two torque values are different, it means that torque has been lost. The loss is due to friction in the tooth contact.

Variations in the displayed moment course depend on the level of accuracy you have specified, and are caused by the accuracy of the iteration.

22.5.9 Single contact stiffness

This graphic shows the individual elements of single tooth contact stiffness. These are the stiffness of both gears and the single tooth contact stiffness of the gear pair.

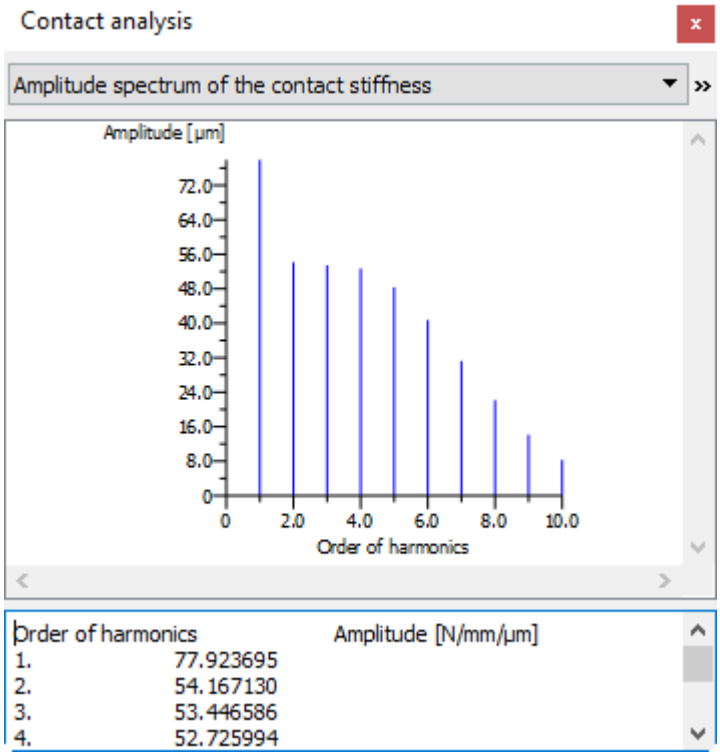
As this is a series-connected spring system, the following applies:

$$\frac{1}{C_{Pair}} = \frac{1}{C_{Gear\ 1}} + \frac{1}{C_{Gear\ 2}} + \frac{1}{C_{H\,1,2}}$$

22.5.10 Stiffness curve

The stiffness curve shows the local stiffness at the operating point. It is calculated from the torsion under load at every point of contact. The stiffness value for gears is usually specified per mm facewidth. To calculate the stiffness of the toothing of two gears, multiply the value you specify (c_γ) with the tooth facewidth.

22.5.11 Amplitude of contact stiffness



The "Amplitude spectrum" graphic displays the spectroscopic analysis results of the transmission error according to Fourier transformation.

You can compare the amplitude of the spectra with the harmonic frequencies of contact stiffness in the comment window.

22.5.12 Bearing force curve and direction of the bearing forces

The bearing force curve assumes that the support is mounted with a symmetrical bearing position. The value given for the face load factor calculation is used as the distance between the bearings. The purpose of this graphic is not to display the correct bearing forces, but to represent the variations in these forces.

Variations in the bearing forces cause vibrations in the shafts and changes in gear case deformations.

22.5.13 Kinematics

The effective tooth form and the effective path of contact are used to calculate a wide range of kinematic values which are then displayed along the length of path of contact:

- specific sliding
- sliding coefficients K_g
- sliding velocity
- variation in ratio

22.5.14 Specific sliding

You can display specific sliding either alongside the meshing cycle, in `Kinematics`, or alongside the tooth height. You can also clearly see the area of the tooth flank with contact.

22.5.15 Power loss

This calculates the power loss for a pair of teeth. Power loss is usually greatest at the start and at the end of the mesh because this is where the highest sliding velocities are generated. However, with a profile modification, you can reduce the load at these points so that the maximum value is shifted to the width between start of mesh and the operating pitch point and to the width between end of mesh and the operating pitch point.

22.5.16 Heat development

Heat development links power loss with specific sliding. If the contact point of a gear moves slowly, it creates a higher heat value per length than if the contact point moves more quickly.

High temperatures generated on the tooth flank should be in correlation with the tendency to scuffing. However, this is not directly attributable to temperature.

22.5.17 Stress curve

The effective tooth form is used to calculate and display the exact Hertzian pressure during generation. The same applies to calculating tooth root stress, as

defined in the Obsieger process (see page II-296), where the maximum stress in the tooth root area is shown by the angle of rotation.

Stresses are calculated with $K_{H\beta}$ = 1.0, $K_{H\alpha}$= 1.0, $K_{F\beta}$= 1.0, $K_{F\alpha}$= 1.0. Only K_A, $K_v$and K_γ are included.

22.5.18 Flash temperature

The effective local temperature shown in the graphic at each point in the path of contact is defined by the gear base temperature (the tooth mass temperature) plus additional local warming (the flash temperature).

At each point on the path of contact, the calculation uses the following data from the contact analysis calculation to calculate the flash temperature on the tooth flank:

- Sliding velocity
- Speed in tangential direction to the pinion and gear
- Curvature radii on the tooth flanks
- Hertzian Pressure

The coefficient of friction μ is taken from the value input for caculating the path of contact. The tooth mass temperature is calculated as specified in ISO TR 15144.

Flash temperature is calculated for:

- ISO as defined in ISO TR 15144

$$\theta_{fl,Y} = \frac{\sqrt{\pi}}{2} \cdot \frac{\mu_m \cdot p_{dyn,Y} \cdot 10^6 \cdot |v_{g,Y}|}{\sqrt{B_{M1} \cdot w_{Y1}} + \sqrt{B_{M2} \cdot w_{Y2}}} \cdot \sqrt{8 \cdot \rho_{rel,Y} \cdot \frac{p_{H,Y}}{1000 \cdot E_r}}$$

- AGMA according to AGMA 925 with equation 84

22.5.19 Micropitting (frosting)

Calculation method

The calculation is performed according to ISO 15144, Method A. All the required data is taken from the contact analysis.

Lubrication gap thickness h and specific thickness of lubrication film λ_{GFP}

The calculation of the progression of the effective lubrication gap thickness h and the effective specific lubrication gap thickness λ_{GF} across the meshing is precisely defined in the ISO TR 15144 proposal. The lubrication gap can vary significantly

depending on local sliding velocity, load and thermal conditions. The location with the smallest specific lubrication gap thickness is the decisive factor in evaluating the risk of micropitting.

Permitted specific thickness of lubrication film λ_{GFP}

To evaluate the risk of micropitting (frosting), it is vital that you know how large the required smallest specific lubrication film λ_{GFmin} is to be. The calculation rule states that:

$\lambda_{GFmin} >= \lambda_{GFP}$ to prevent micropitting (frosting), or to ensure safety against frosting $Sl = \lambda_{GFminP} / \lambda_{GFP}$.

If the lubricant's micropitting load stage is known, the permitted specific lubricant film thickness is calculated according to ISO TR 15144.

Otherwise, reference values for λ_{GFP} can be obtained from the appropriate technical literature.

In [81] you will see a diagram that shows the permitted specific lubrication gap thickness λ_{GFP} for mineral oils, depending on oil viscosity and the micropitting (frosting) damage level SKS.

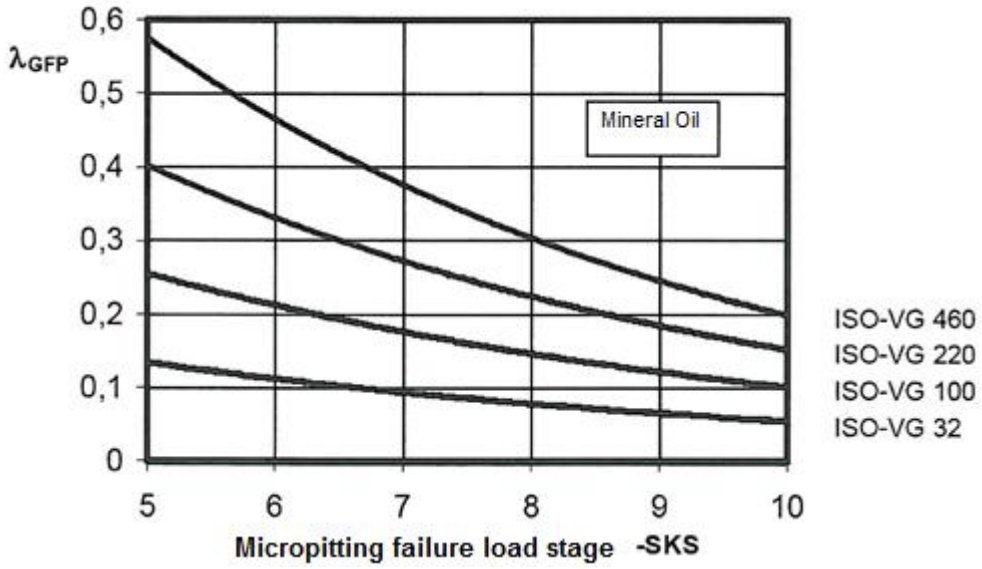


Figure 22.13: Minimum necessary specific thickness of lubrication film λ_{GFP}

The micropitting damage level SKS, determined according to FVA information sheet [82], is nowadays also stated in data sheets produced by various lubricant manufacturers. The data in the diagram applies to mineral oils. However, synthetic oils with the same viscosity and micropitting (frosting) damage level show a lower

permitted specific lubrication film λ_{GFP} [81]. Unfortunately, as no systematic research has been carried out on its effects, no properly qualified values are available.

Furthermore, you must be aware that the predefined values λ_{GFP} only apply to case-hardened materials. As specified in ISO TR 15144, for other materials, the permitted specific lubrication gap thickness λ_{GFP} can be multiplied by the coefficient Ww.

	Ww
Case hardening steel, with austenite content <= 25%	1.00
Case hardening steel, with austenite content > 25%	0.95
Gas-nitrided (HV > 850)	1.50
Induction or flame-hardened	0.65
Through hardening steel	0.50

Table 22.1: `Material coefficient`

It is interesting to note that, at least according to the table shown above, materials with a nitrite content are more prone to micropitting (frosting) than case hardened materials when the same lubrication gap is used. In contrast, through hardened materials that are not surface hardened are much more resistant.

You should be aware that the data shown here must be used with caution, because information about the micropitting (frosting) process is still incomplete, and even technical publications will sometimes contain contradictory data.

Safety against micropitting

If the load stage against micropitting as defined in FVA C-GF/8.3/90[82] is specified for the lubricant, the minimum required lubricant film thickness λ_{GFP} is calculated. The safety against micropitting can therefore be defined as $S\lambda = \lambda_{GFmin} / \lambda_{GFP}$.

22.5.20 Wear

Before you can calculate local wear on the tooth flank, you must first define the material's wear coefficient k_w. This coefficient can be measured using gear testing apparatus or by implementing a simple test procedure (for example pin and disc test rig) to determine the appropriate value. Investigations are currently being

carried out to see how the k_w coefficient, determined using a simpler measurement method, can be applied to gears. For exact forecasts, you will also need to determine the coefficient k_w for the material pairing. For example, POM paired with POM does not supply the same results as POM paired with steel.

Plastics

You can input the wear coefficient k_w for plastics in the polymer data file, depending on the temperature (for example, Z014-100.DAT for POM). The data is input in 10^{-6} mm^3/Nm.

As an example:

```
-- Data for wear (PROVISORY! From Dissertation R.Feulner, 2008) Dependency from Temperature unknown
-- in 10^-6 mm3/Nm
:TABLE FUNCTION KwearDry
       INPUT X ZahnTempFlanke TREAT LINEAR
DATA
    -20     23     40     90    120
   1.03   1.03   1.03   1.03   1.03
END
```

Steel

Plewe's investigations have revealed that a rough approximation of the wear coefficients for steel materials can be defined. See also the calculation of wear coefficients for steel (Calculation of wear coefficient kw for steel) (see page II-278)

Calculation

Wear is calculated according to the following base equation:

$$\delta_w = J_w \cdot P \cdot V \cdot T$$

(δw [mm], k_w [mm^3/Nm], P: Pressure [N/mm^2], V:Velocity [mm/s], T:Time[s])

As modified to suit gear conditions, local wear results from:

$$\delta_{w_i} = NL \cdot k_w \cdot w \cdot \zeta_{_i}$$

(i = 1.2)

(δw_i [mm], k_w [mm^3/Nm], NL: Number of load cycles, w:Line load [N/mm], ζ_i: specific sliding)

This equation also corresponds to the data in [83], Equation 6.1.

The calculation to determine wear on the tooth flank uses the following data at each point of contact taken from the calculation of the path of contact:

- Specific sliding

- Line load

For POM against steel (at 23°C), [83] gives a k_w of 1.03 * 10^{-6} mm^3/Nm, for PBT against steel, it gives a k_w of 3.7*10^{-6} mm^3/Nm.

When you interpret the results, you must note that the increasing wear on the tooth flank changes local conditions (line load, sliding velocity) to some extent, and therefore also changes the increase in wear itself. (For more information, see the Wear iteration section.)

22.6 Gear pump

Eleven different diagrams document in detail the progressions of the characteristic values in a gear pump when it is generated. You will find more detailed information about how to calculate gear pumps (see section "Gear pump" on page II-423) in KISSsoft-anl-035-E- GearPumpInstructions.doc [77] (available on request).

22.7 3D export

Click `Graphics > 3D Export` to export the geometry of the gears you have just designed to a specified CAD system. The next section (see page II-666) provides more detailed information about which CAD system you should use, and its interface.

NOTE:

Before you call this function for the first time, make sure you are using a suitable CAD system. If you have specified a CAD program that has not yet been installed, you may cause a problem when you call this function.

22.8 Settings

Click `Graphics > Settings` to define the background for 3D graphics and select your preferred CAD system. Here, you can select any of the interfaces for which you have the appropriate licenses.

22.9 Graphics list

The graphics list is where you save the graphics with , as in all the other toothing modules. This is attached to the end of the report, unless otherwise specified in the report template. In the graphics list you can open every graphic with , ,  and , depending on the graphic type, and then modify, enable or disable its properties, or delete it with .

Chapter 23
Answers to Frequently Asked Questions

23.1 Answers concerning geometry calculation

23.1.1 Precision mechanics

KISSsoft is an ideal tool for calculating the gears for precision mechanics.

The reference profile and the geometry are calculated as defined in DIN 54800 etc. The strength calculation is performed according to ISO 6336, DIN 3990, VDI 2545 or VDI 2736, since no special strength calculation exists for precision gears. For this reason, "defining required safeties for gear calculation (see section "Required safeties for cylindrical gear units" on page II-679)" is important when you are interpreting the results.

If gears are manufactured using topping tools, the tip circle can be used to measure the tooth thickness. In this situation, it is critical that you specify precise value of the addendum in the reference profile to match the corresponding cutter or tool. This is because this value is used to calculate the tip circle. The tip alteration $k*m_n$ is not taken into account when calculating the manufactured tip circle. The following formula is used:

$$d_{aeff} = d + 2 \cdot m_n \cdot x_{eff} + 2 \cdot m_n \cdot h_{aP} \qquad (23.1)$$

23.1.2 Deep toothing or cylindrical gears with a high transverse contact ratio

Using deep toothed gears is recommended for some specific applications (for example, for spur gears that should not generate a lot of noise).

In KISSsoft, you can easily calculate all aspects of deep toothed gears. To calculate the geometry, you must select a profile of a suitable height when you select the reference profile:

Normal profile height: e.g. m_n * (1.25 + 1.0)
For deep tooth form: e.g. m_n * (1.45 + 1.25)

You must be aware that this type of gear is more prone to errors such as undercut or pointed teeth. Experience has shown that you must select a value of 20 or higher as the number of pinion teeth to ensure that you can create a functionally reliable pair of gears. KISSsoft also has very effective and easy to use strength calculation

functionality; as specified in DIN 3990, Part 3, calculation of gears with a transverse contact ratio greater than 2.0 tends to be on the conservative side.

The Geometry-Variants calculation (Modules Z04 and Z04a) is very good at sizing optimum arrangements of deep toothed pairs of gears!

See also section 14.16.

23.1.3 Pairing an external gear to an inside gear that has a slightly different number of teeth

When you pair a pinion (for example, with 39 teeth) with an internal gear (for example, with 40 teeth) that has a slightly different number of teeth, the teeth may have a collision ("topping") outside the meshing area. This effect is checked and an error message is displayed if it occurs.

To size a functioning pairing of this type, select this strategy:

- Reference profile: Short cut toothing
- Pressure angle: the bigger the better
- Total profile shift: select a negative value
- Total pinion profile shift: approximately 0.4 to 0.7

23.1.4 Undercut or insufficient effective involute

(this triggers frequent error messages when you calculate the geometry of cylindrical gears.)

An insufficiently effective involute occurs if the tip of the other gear in the pair meshes so deeply with the root of the first gear that it reaches a point at which the involute has already passed into the root rounding. These areas are subject to greater wear and tear. Some gear calculation programs do not check this effect and suffer recurrent problems as a consequence.

To keep a close eye on the undercut and effective involute, you should always work with the `Calculate form circle from tooth form` (see page II-466) option. This function checks the tooth form every time a calculation is performed. It determines any undercut it discovers and takes it into account in the calculation.
(The tooth form calculation takes into account all aspects of the manufacturing process. In contrast, geometry calculation according to DIN 3960 uses simplified assumptions.)

23.1.5 Tooth thickness at tip

The tooth thickness at the tip circle is calculated for a zero clearance status. In addition, the maximum and minimum value is calculated using all tolerances.

When you check the tooth geometry, the tooth thickness at the tip must usually be at least 0.2 * module (according to DIN 3960). If this limit is not reached, KISSsoft displays the appropriate warning message. Click on `Calculation > Settings > General` to change this coefficient if required.

23.1.6 Special toothing

The term "special toothing" is used to describe toothing with non-involute flanks. The reference profile (or the normal section through the hobbing cutter or rack-shaped cutter) of special toothing is not straight (unlike involute toothing). However, the same generating process is used to manufacture both toothing types. As part of the tooth form calculation, special toothing (cycloid, circular pitch toothing) can either be imported from CAD or defined directly. In addition, a suitable counter gear can then be generated by clicking `Generate tooth form from counter gear`.

By simulating the generation process, the tooth form and, from this, the geometry can then be defined for special toothing. As no standards or documentation are available for strength calculations, analogies for these tooth form types must be drawn from the calculations used for the cylindrical gear process. For more information see the Path of contact (see section "Contact analysis" on page II-653) section.

23.1.7 Calculating cylindrical gears manufactured using tools specified in DIN 3972

Profiles I and II are profiles for the final machining. They can all be handled easily by KISSsoft. Simply select the tool you require from the selection list (Reference profiles).

Profiles III and IV are used for tools used in pre-machining. However, you should always use a finished contour to calculation the strength of a gear. These profiles should therefore only be used as a pre-machining cutter.

The reference profiles are dependent on the module, as defined in the following formulae

Profile III	$h_{fP} = 1.25 + 0.25\ m_n^{-2/3}$	$h_{aP} = 1.0$	$\rho_{fP} = 0.2$
Profile IV	$h_{fP} = 1.25 + 0.60\ m_n^{-2/3}$	$h_{aP} = 1.0$	$\rho_{fP} = 0.2$

In the `Reference profile` tab, if the configuration is set to `Tool: Hobbing cutter` is set, you can click the plus button to the right of the hobbing cutter to see a selection list that includes Profiles III and IV according to DIN 3972. Remember that the data you enter here depends on the module. If you want to change the module, you must select a the correct reference profile again.

Use the recommendations in the standard to select the correct allowance for pre-machining:

Profile III	Grinding allowance = $+0.5\ m_n^{1/3} \tan(\alpha_n)$
Profile IV	Grinding allowance = $+1.2\ m_n^{1/3} \tan(\alpha_n)$

If Pre-machining has been selected (in the Reference profile tab), you can set the appropriate Grinding allowance for Profile III or IV in the list in the Grinding allowance field.

Click on the + button next to Grinding allowance q to input a tolerance interval for the grinding allowance qTol (=qmax-qmin). The grinding allowance for pre-machining then lies in the range qmin ... qmax, where qmin = q - qTol/2. qmax = q + qTol/2 applies.

The control masses (base tangent length etc.) for pre-machining are then calculated with the following allowances:

Maximum grinding allowance with As.e + qmin*2 / cos(an)

Minimum grinding allowance with As.i + qmax*2 / cos(an)

Note:

If you want customer-specific tolerances to be processed automatically, you can define them in a file called "GrindingTolerance.DAT". The \dat directory has an example of this type of file, which is called "GrindingToleranceExemple.DAT". When this file is renamed to "GrindingTolerance.DAT" its tolerance values are used in the calculation.

23.1.8 Composites as defined in DIN 58405

DIN 58405 specifies the base tangent length allowances and permitted composite errors for toothing used in precision mechanics. In this case, the reference profile specified in DIN 58400 assumes a pressure angle of α_n=20°. If you use a operating

pressure angle that is not 20°, DIN 58405 Sheet 3, sections 1.2.10 and 1.2.11, state that the permitted composite error and the permitted rolling deviations must be multiplied with a coefficient L = tan(20°)/tan(abs). This must be performed because the base tangent length allowances are standardized and the center distance error increases as the pressure angle is reduced. KISSsoft takes coefficient L into account when calculating tolerances to comply with DIN 58405, because it is specified in the standard.

However, the tolerances specified in ISO 1328 and DIN 3961 do not include this coefficient because it is not listed in the standard.

23.1.9 Automatic change of reference profiles

Some calculations have revealed the problem that the reference profile changes automatically when the center distance changes. In the Reference profiles tab, the factors for the tool tip and dedendums change automatically. Why?

This is because the "Retain tip circle or dedendum when the profile shift changes" checkbox is active in the General tab in the module-specific settings.
If you change the center distance, the profile shift coefficient also changes. The setting you make automatically changes the factors for the reference profile.

23.1.10 Non-identical (mirrored symmetry) tooth flanks

If the tooth flanks (left, right) are not identical, will this cause an error when the tooth contour is exported?

The tooth flanks used in the calculation (sizing) are identical.

The export function used in the system not only exports the involutes but also the entire tooth form. This is an approximated curve.

With the export precision (permitted deviation ε) you can define how closely you want to approximate the calculated tooth form.

In each case, an approximate curve in the specified level of accuracy is given for either half of the tooth or the whole tooth. You can only use mirror symmetry with approximation accuracy.

This is the error you specified as the permitted deviation.

The smaller the selected deviation, the more detailed the curve.

23.1.11 Internal toothing - differences in the reference profile if you select different configurations

A gear pair with internal toothing has been calculated in the KISSsoft system. A pinion type cutter is then to be used to manufacture this internal gear. The tool is manufactured to suit particular customer requirements and is influenced by the particular tooth form which is used. This must reflect the reference profile geometry of the internal gear. How can you then determine the pinion type cutter geometry?

A gear's reference profile is the corresponding rack profile. A regular hobbing cutter for an outside gear has this rack geometry, which makes it easy to define the hobbing cutter profile. However, you must reverse the gear profile to obtain the hobbing cutter profile (the addendum of the gear reference profile becomes the dedendum of the hobbing cutter, and so on).

If the manufacturing tool is a pinion type cutter, the limited number of teeth on the pinion type cutter results in a different situation. Generally, the inverse gear reference profile corresponds to that of the pinion type cutter. However, after this, you must change the cutter addendum in a way that ensures you can achieve the necessary root diameter on the internal gear.

First of all, you must define the number of teeth on the pinion type cutter. Depending on the type of machine tool used to manufacture the gear, the reference diameter of the pinion type cutter is already predefined to some extent. This reference diameter must be greater than the diameter of the main shaft of the machine tool that is to be inserted in the pinion type cutter tool. However, if this diameter is too large in comparison with the size of the pinion type cutter, the shaft diameter will be too small. This will cause powerful vibrations during the production process and result in a poor accuracy grade. To prevent this, you must know the approximate pinion type cutter diameter. The reference diameter is then divided by the module to determine the number of teeth on the pinion cutter.

If you want to use the KISSsoft system to design the pinion type cutter geometry, you must first input the number of teeth on the pinion type cutter. You can start with 0.0 for the profile shift coefficient of the pinion type cutter. A pinion type cutter's profile shift changes as it is used. Every time the pinion type cutter is resharpened, the profile shift is reduced slightly. A new pinion type cutter usually has a positive profile shift (for example +0.2), a worn tool therefore has a negative profile shift.

After you have input the data for a pinion type cutter, you must first check all the entries, i.e. whether the required root form diameter has been achieved. If not, you must reduce the tip fillet radius of the pinion type cutter. If that does not help, you must increase the addendum of the tool reference profile. However, this also changes the active root diameter.

The same problem can also happen with the tip form diameter dFa. It often happens that you cannot generate the entire involute part up to the tooth tip. In this situation, you must either increase the number of teeth on the pinion cutter tool or reduce the tip diameter of the gear.

If you develop a gear that is manufactured by a pinion type cutter, it is always critically important that you investigate the production process early on in the development process. This is because not every gear geometry can be created with this production process.

23.1.12 Effect of profile modifications

Profile modifications are a popular topic of discussion. Where should these modifications start, and which amount should be used to make these modifications?

Linear tip relief is a type of profile modification. It has the following characteristics: Starting from a particular point, ever increasing amounts of material are removed from the involute toothing part up to the tip diameter.

The tooth contact in the modified area is disrupted. This is only a benefit when subject to the corresponding load. This entire area is taken into account when calculating the length of contact to determine the transverse contact ratio ε_a. Shouldn't this be different?

If you use profile modifications, you "delete" the real involute. Why is this a good idea?

This is a complex problem that must be taken into consideration when you design profile modifications. The amount of material removed (tip relief C_a is the reduction of tooth thickness at the tip due to the profile modification) and must be applied according to the tooth bending.

For example, if the tooth had infinite stiffness, and you ignore any of the possible effects of compensating for production errors, the profile modification would simply reduce the transverse contact ratio. If you did not take this profile modification into account, you would make an error in the geometry calculation. This is basically correct for a gear that is subject to a lower load. However, you will usually need to design gears for optimum performance at operating torque and the strain that this places on the teeth.

If the tip relief C_a is well arranged, the profile modification then compensates for the tooth deformation, so that the tooth contact across the entire tooth height is not compromised. In this case, the transverse contact ratio is not reduced. Here, you have a changed normal force curve over the meshing, when compared to a gear without profile modification.

However, the maximum force (in the operating pitch circle), where only one gear pair is in contact, is not changed. For this reason, the maximum root and flank strains, which determine the service life of the drive, remain unchanged. This profile modification reduces the normal force at the start and end of the tooth contact. This also leads to a significant reduction in the risk of scuffing. The risk of scuffing is due to contact stress and sliding velocity. Sliding is greatest at the start and end of the tooth contact, so, by reducing the contact stress in this area, you can also reduce the risk of scuffing. A profile modification can reduce the influence of tooth strain on stiffness fluctuations across the meshing, and therefore limit the number of transmission errors. This also lowers the levels of vibration and noise.

This clearly shows that a profile modification does not reduce the transverse contact ratio, as long as this has been properly sized, i.e. for the operating torque of the drive. However, when lower loads are involved, gears whose profile has been modified do not mesh as well as gears without profile modification. This is because the transverse contact ratio has been significantly reduced. In this case, although the load would increase, it would do so by a comparatively small amount, and can therefore be ignored.

23.1.13 Number of teeth with common multiples

A toothing with 15:55 teeth has been sized. Different documents state that you should avoid gear reductions (such as 11:22) that are whole numbers. Furthermore, you will also discover that you should also avoid using numbers of teeth that are common multiples (in this case the 5 in 3*5 to 11*5). Is that true, and is it displayed in KISSsoft?

Let's assume we have a gear which has a fault on one of its teeth. In a whole number gear reduction, this tooth will always come into contact with the same tooth in the counter gear. The error is then transmitted to the counter tooth. However, if the tooth with the fault comes into contact with a different counter tooth in every rotation, this error will be reduced as the gears wear in.

Nowadays, most gears are surface-hardened. Unlike weak gears, they hardly ever wear in. As a result, this problem is now less critical than it used to be, where it was important that whole number gear reductions (such as 11:22) were avoided even when hardened gears were used. In contrast, whole number toothing combinations with common multiples (such as 15:55) are quite unobjectionable for surface hardened gears.

In KISSsoft, you will find notes about whole number combinations with hunting multiples in both fine sizing and rough sizing under the keyword "hunting". If you see YES for "hunting" in the table, this means: no common multiple is present.

23.1.14 Allowances for racks

From Release 10/2003 onwards, allowances for racks are defined in conjunction with the paired gear.

This conforms to DIN 3961.

"The tolerances for the toothing of a rack should not be greater than the tolerances of its counter gear. If the counter gear's manufacturer is not known, the rack length should be the same as the counter gear circumference."

23.2 Answers to questions about strength calculation

23.2.1 Differences between different gear calculation programs

You will always discover differences in the results when you compare calculations performed with different gear calculation programs. Many of these differences are due to the different data entered. However, even if all the data entered is the same, you will still get different results.

One of the questions our users often ask is whether the results calculated by KISSsoft are correct.

The main calculation process used in the KISSsoft cylindrical gear calculation functions is based on DIN 3990, ISO 6336, and AGMA. It faithfully follows the procedure described in Method B. However, as DIN 3990 or ISO 6336 offer various different Methods (B, C, D) and sub-methods, it is no surprise that different calculation programs produce slightly different results. Most programs do not perform calculations that consistently use Method B. Instead, they partially use Method C or even D, which are easier to program.

To give our users additional reassurance, we have therefore integrated the `FVA program` calculation variant in KISSsoft. This variant supplies exactly the same results as the ST+ FVA program that was developed by the Technical University in Munich, and which can be used as a reference program.
The minor differences between KISSsoft's calculations according to DIN 3990 and the FVA programs are due to the slight (permissible) deviations of the FVA program from the standard process defined in DIN 3990.

23.2.2 Difference between cylindrical gear calculation following ISO 6336 or DIN 3990

The strength calculation method used in ISO 6336 is virtually the same as that defined in DIN 3990. The majority of the differences only affect minor details which have very little effect on the safeties calculated for tooth root, flank and scuffing.

The only significant difference happens to be the life factor (Z_{NT} and Y_{NT}). In the endurance area (according to DIN, depending on material type and calculation method 10^7 to 10^9 load cycles) this coefficient in ISO 6336 decreases from 1.0 to 0.85 at 10^{10} load cycles. Only with "optimum material treatment and experience" does the coefficient remain 1.0.

As a result, gears in the range of endurance limit range supply much smaller safeties (15% lower) when calculated according to ISO 6336 for root and flank! In the case of optimum material treatment, or for the number of load cycles in the fatigue strength range, the safeties are practically identical.

23.2.3 Calculation using Methods B or C (DIN 3990, 3991)

Cylindrical gears:

Calculation using Method B or C is described in DIN 3990. Method B is much more detailed and is therefore the method we recommend KISSsoft usually uses Method B. However, we do not consider Method B to be precise enough to calculate the form factors for internal teeth, which is why we recommend Method C.

Converting to using Method C means that most of the calculation is performed in accordance with Method B and only the tooth form factor is calculated as defined in Method C.

Note: The most precise way to calculate internal toothing is to take the exact tooth form into account (see "Tooth form factor using graphical method", section 14.3.16.3).

Bevel gears:

Tooth form factors are calculated in accordance with standard Method C.

23.2.4 Required safeties for cylindrical gear units

Defining the necessary safeties (for tooth root, flank, scuffing) for gears in a particular application, for example, in industry standard drives, vehicles, presses etc., is a very important step in the gear calculation process.

The (DIN 3990 or ISO 6336) standards give hardly any information about this. DIN 3990, Part 11 (industrial gears) has this data:

Minimum safety for root: 1.4

Minimum safety for flank: 1.0

AGMA 2001 does not specify minimum safeties. The AGMA 6006 guideline (for gearboxes in wind power installations) has a note that SFmin = 1.56 is specified for root safety for calculation according to ISO 6336. In contrast, SFmin = 1.0 is sufficient for calculations according to AGMA. This matches our findings that calculations performed according to AGMA give much lower root safeties. Therefore, we recommend a minimum safety of 1.4*1.0/1.56 = 0.90 for industrial gears calculated according to AGMA.

Scuffing is calculated according to DIN 3990, Part 4:

Minimum safety for scuffing (integral temperature): 1.8

Minimum safety for scuffing (flash temperature): 2.0

The standards do not specify this value for precision mechanics (module under 1.5). Despite this, in accordance with empirical values the required safeties are much smaller than for gears with a larger module (root 0.8; flank 0.6)! The reason for this: The formulae and methods used in strength calculation are all taken from tests with larger gears and only supply very conservative factors (values that err on the side of safety) for small modules.

Defining required safeties for gear calculation

You can use the simple method described here to obtain the required safeties:

1. Examine and define the basic settings of the calculation (e.g. application factor, lubricant, accuracy grade, processing etc.).
2. Then apply the gear calculation method (without changing the basic settings unless you absolutely have to!) on known set of gears. You should select gears that run reliably under operating conditions and also such that have failed.
3. You can then use the resulting safeties calculated with these gear sets to define the point up to which minimum service reliability can be guaranteed.
4. You can then use these parameters to calculate the sizing of new gears. You can, of course, change these minimum safeties to reflect the results of your own tests and experience.

23.2.5 Insufficient scuffing safety

You can increase scuffing safety by:

- Oil selection (higher viscosity at high temperatures)
- Tip relief (profile modification)
- Different distribution of the profile shift

The methods used to calculate scuffing safety (unlike those used to determine the tooth root and flank) is still a matter of controversy. For this reason, you should not pay too much attention to it, especially if the results of scuffing safety at flash temperature and the integral temperature process are very different.

23.2.6 Work hardening factor (strengthening an unhardened gear)

If you pair a hardened gear with an unhardened gear (e.g. pinion made of 17CrNiMo6 and gear made of 42CrMo4), you get the positive effect of increased load capability on the flank of the unhardened gear. This effect is taken into account by the work hardening factor (coefficient in the range 1.0 to 1.2). As stated in ISO 6336, the surface roughness of the hardened gear should be low (polished surface), otherwise the load capability will not increase. On the contrary, the tooth of the weaker gear may actually be ground off.

23.2.7 Defining the load stage scuffing (oil specification)

In accordance with Niemann [65], page 166, the torque on the test gear is gradually increased, on a test rig, until scuffing occurs. This torque level is then entered in the oil specification parameters (Example: no scuffing at load stage 10. Scuffing at load stage 11. The load stage scuffing of the oil is therefore 11).

To calculate the scuffing load capacity, you must then enter this load stage (for the oil specification). In the example described above, this is the value 11 (according to Niemann [65], page 341). The scuffing safety calculation defines the safety against scuffing with predefined safeties greater than 1.0. This creates a necessary reserve, because the gradual increase in torque used in the test only approximates the effective scuffing torque.

23.2.8 The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error.

When calculating a cylindrical gear according to ISO 6336, a higher value was determined for the tooth trace deviation f_{ma} when calculating the face load factor $K_{H\beta}$. This was due to a manufacturing error. The value for $K_{H\beta}$ does not change. Why then, does this value for $K_{H\beta}$ not change if a higher value for f_{ma} is used?

Before you can calculate $K_{H\beta}$ you must input the position of the contact pattern. If the contact pattern has been defined as "economical" or "optimum", $K_{H\beta}$ is calculated in accordance with the formulae in ISO 6336 or DIN 3990. f_{ma} has no influence on the calculation of $K_{H\beta}$ and is therefore ignored.

See formulae: (53) or (55) in ISO 6336:2006.

The reason for this is that a well designed contact pattern can compensate for manufacturing variations and variations due to deformation. If a higher value of f_{ma} is to be used in the calculation, this means, in reality, that a good contact pattern can never be present. That is why, in this situation, you should select the contact pattern position "not verified or inappropriate" when calculating the face load factor.

23.2.9 Load spectrum with changing torque

Load bins can also be entered with negative torques.

The problem:

until now, no calculation guidelines have been drawn up to describe how to calculate gears with changing load spectra.

The only unambiguous case is when a change in torque takes place, during every cycle (and in each element of the collective). At this point, the load change corresponds exactly to a double-load with +torque and then with –torque. This instance can be calculated correctly by entering the load spectrum of the +moments and the alternating bending factor YM for the tooth root. The flank is also calculated correctly, because the +moments always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal + moments and –moments for the tooth root are to be handled. For this type of case, only the +moments are considered for the flank (with the prerequisite that the +moments are equal to, or greater than, the –moments).

Note about handling load spectra with reversing torque:

A load progression as represented in the figure below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

13 loads with 100% of the nominal load (100 Nm) on the left flank, then

9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

11 load cycles with 100% load, positive torque, pulsating; then

1 load cycle with 100% load on the left and 80% load on the right; then

7 load cycles with 80% load, negative torque, pulsating; then

1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	Torque	Load left flank	Load right flank
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

Table 14.14: Load progression shown as a load spectrum

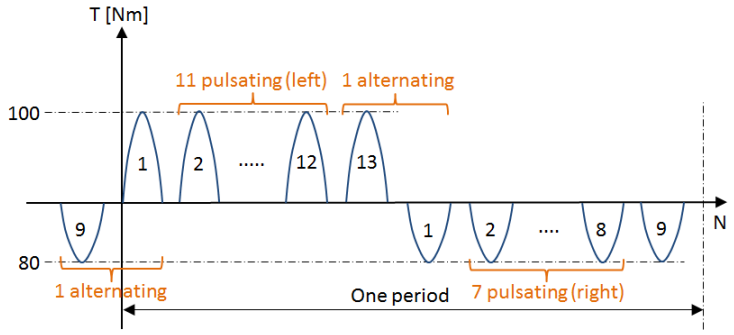


Figure 14.20: Load progression

23.2.10 Strength calculation with several geometries on one gear

How can you take several simultaneous meshing points on a motor pinion into account in the calculation?

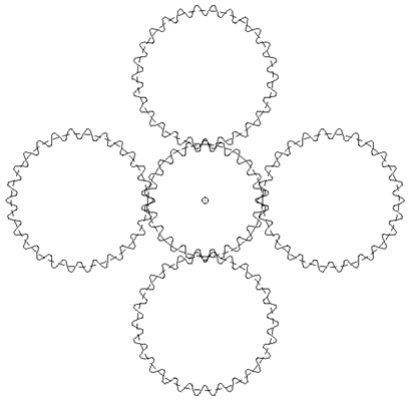


Figure 23.1: Fourfold meshing

You can solve this problem with a normal face gear pair calculation (Z12).

Simply divide the power by a factor of 4 (reduce by 25%)

Then click the Details... button in the Strength area after the reference gear.

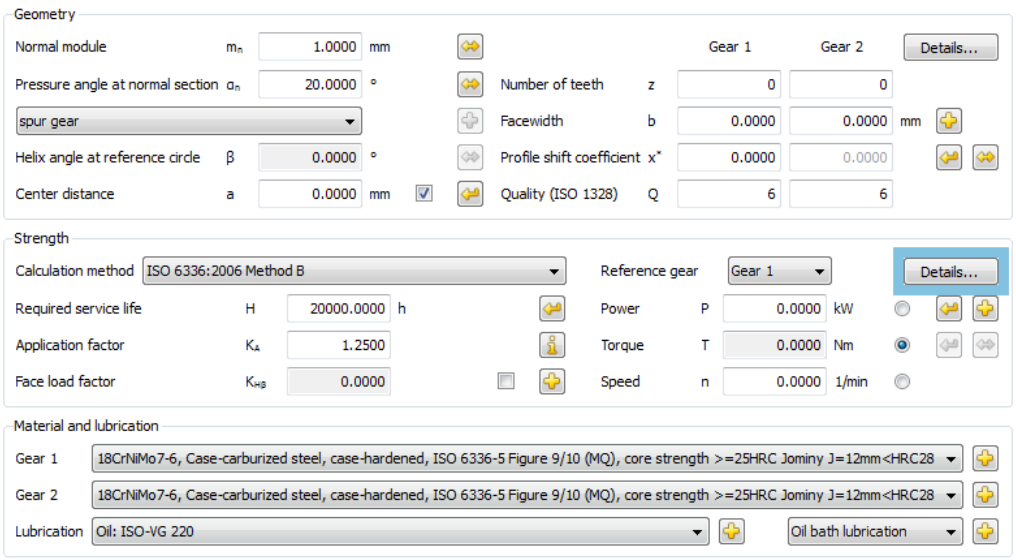


Figure 23.2: Details Strength

Then click the plus button after the number of load cycles to perform the change shown below. The number of load cycles for gear 1 is changed from "Automatically" to 4 load cycles per revolution.

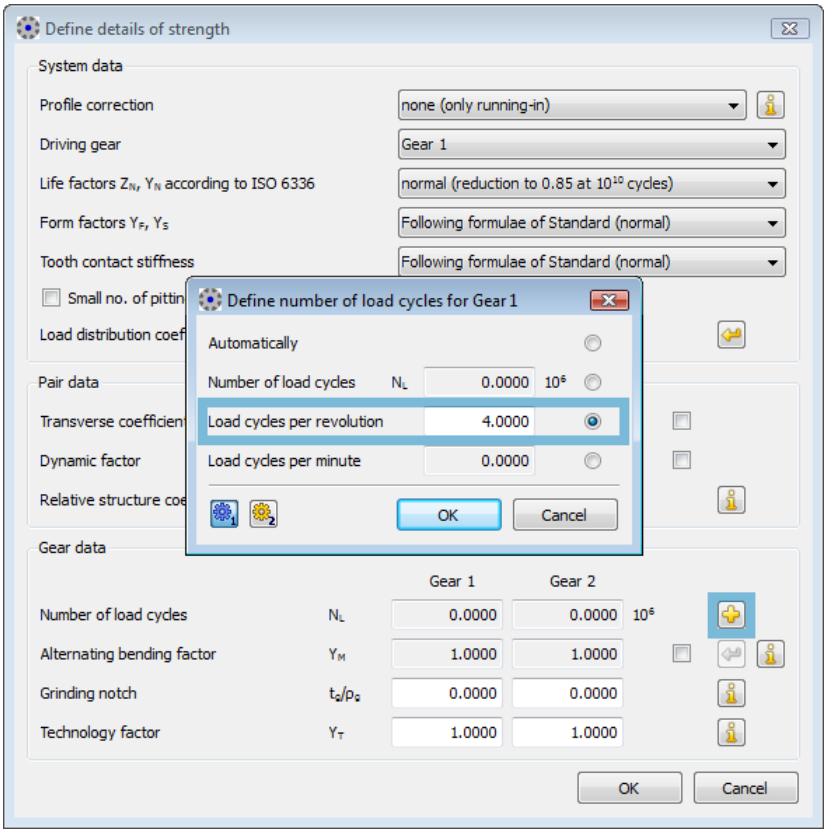


Figure 23.3: Define number of load cycles for gear 1

23.2.11 Bevel gears: – Determine permitted overloads

Can maximum overloads be taken into account when calculating bevel gears according to ISO standards?

AGMA norms have definitions that allow for a standard overload of 250%. This overload is defined as being present for less than 1 second, not more than 4 times in an 8 hour time period. Does the ISO standard have comparable regulations with regard to overloads (shock)? No references could be found about this subject in the ISO standard.

ISO 10300 does not give any information about permitted overloads. However, ISO has a different Woehler curve (for YNT and ZNT factors) than AGMA. Therefore, in principle if ISO 10300 is strictly adhered to, the total number of load

changes including the overload must be input. The application factor is 2.5 (which corresponds to 250% overload). After this, you must calculate and check the safety factors.

If the load only occurs very infrequently, (less than 1000 times during the entire service life), this can be handled in a static calculation. KISSsoft has a simplified version of the strength calculation process, specifically to cover this situation. This is based on the ISO method, but only takes into account the nominal stress in the tooth root (without stress correction factor YS). Here, please note that, in this case, you must maintain a minimum safety level of 1.5 relative to the material's yield point!

23.2.12 Take shot peening data into account in calculating the strength of toothed gears

On page 47 of AGMA 2004-B89 you will see a note about shot peening. This states that shot peening improves tooth root strength by 25%.

If you use KISSsoft to perform calculations according to DIN or ISO, you can achieve the increase in strength due to shot peening by inputting the corresponding technology factor. To do this, go to the Factors tab and click on "Z-Y factors..." in the General factors group.

You will find the details of useful entries as specified in Linke, Bureau Veritas/RINA or ISO 6336 in the manual. If you want to perform the calculation according to AGMA, you do not have the option of inputting the technology factor. In this case, you must increase the foot endurance limit by inputting the corresponding percentage rate directly when you enter the material data. To do this, go to the Basic data tab and then click the plus button behind the material selection. In the dialog window, then activate "Own input". Input the endurance limit as shown in the figure below.

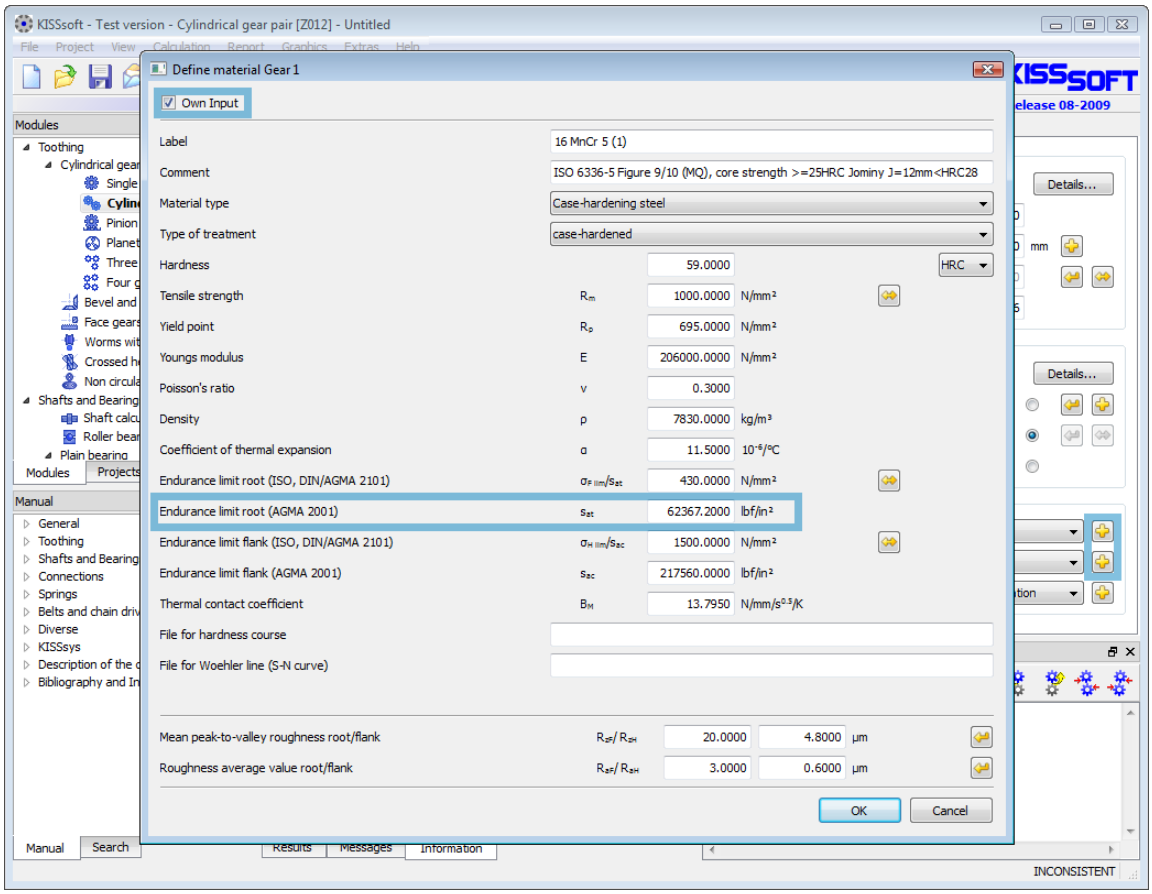


Figure 23.5: Material own input

23.2.13 Calculation according to AGMA 421.06 (High Speed Gears)

In the KISSsoft system, you perform calculations as specified by AGMA 421.06 for high speed gears in the following way.

AGMA 421 is an old, well-established standard (1968), and has since been replaced by AGMA 6011-I03 (2003).

Please note the information on this topic in section Calculation method (see section "Calculation methods" on page II-281)

23.2.14 Comparison of an FEM calculation with crossed axis helical gear wheel calculation

The accepted wisdom is that the differing results in the tooth root stress were primarily due to the lower value of the "Reference Facewidth" in the KISSSOFT calculation.

The effective contact of crossed axis helical gear wheels is included in our calculation of the "Reference Facewidth". This results from the pressure ellipse (flattening of the point of contact). In addition, if sufficient facewidth is present, 1x module per facewidth is added to each side, as specified in ISO 6336-3.

23.2.15 Estimating the strength of asymmetrical spur gear toothings

At present, KISSsoft does not have any algorithms that can be used to perform a direct strength calculation for asymmetrical gears. Safeties are determined using the calculation methods in ISO 10300 for hypoid gears (hypoid teeth are asymmetrical, and have an unequal pressure angle on the right-hand and left-hand flank).

This procedure is described below:

The calculation is run twice, each time with a symmetrical tooth, once with a high pressure angle (calculation I), once with small pressure angle (calculation II).

The safety factor for the required safety against pitting that corresponds to the calculation with the flank under load is applied here. Therefore, if the load flank is the one with the small pressure angle, the safety against pitting from the calculation with the smaller angle (SHII) is used.

Root safety is determined with the nominal stress (tooth form factor YF), which is derived from the loaded flank. The tooth thickness at root sFn is determined from both these calculations, so therefore:

$sFn = (sFnI + sFnII)/2$

The stress concentration (factor YS) is calculated with the formula given above, and using the root radius and the application of force lever arm of the flank under load, and also sFn. All the remaining factors for defining the root fracture safety SF are the same.

23.2.16 Determine the equivalent torque (for load spectra)

Some calculation guidelines require you to determine the equivalent torque of a load spectrum and use it to perform sizing. How can I determine the equivalent torque in KISSsoft?

The fundamental issue here is that the verification of a toothing with equivalent torque must give the same safeties as the verification with the actual load spectrum. For this reason, you can follow this procedure:

1. Input the load spectrum and calculate the toothing.

2. Make a note of the lowest root safety and the lowest flank safety for each gear.

3. In the `Module specific Settings` window, which you access from `Calculation` -> `Settings`, input the safeties you have noted as required safeties in the "Required safeties" tab. At this stage, we recommend you deactivate the "`Securities depend on size`" tab.

4. Delete the load spectrum by setting "Individual load".

5. Then click the sizing button next to the torque input field. This field will then be filled with the equivalent torque.

6. Now run the calculation to check the data. The safeties you have now defined for the root or flank of a particular gear must be exactly equal to the previous smallest value (as in step 2). None of the gears can have a safety that is less than the safeties you recorded in step 2.

23.2.17 Check changes in safeties if the center distance changes

Is it possible to check how the safeties change when gears are mounted with a different center distance?

Select `Calculation`-> `Settings` ->`Module-specific settings` in the `Calculations` tab and select `Calculation with operating center distance and profile shift according to manufacture`. You can then input the profile shift coefficients and center distance independently of

each other. The calculation then uses the circumferential forces in the operating pitch circle instead of the circumferential forces in the reference circle.

23.2.18 Warning: "Notch parameter qs .... outside RANGE (1.0 to 8.0) ..."

Stress modification factor Y_S is calculated with a formula that complies with ISO 6336, Part 3 or DIN 3990, Part 3. This formula uses a notch parameter q_s, which is also documented in these standards:

$$q_s = \frac{s_{Fn}}{\frac{r_{OF}}{2}} \tag{23.4}$$

The validity range for the formula for Y_S in accordance with the standard lies in the range 1.0 ...q_s... 8.0. This formula should not be used outside of this range.

If $q_s < 1$, Y_S (calculated with q_s=1), should be rather too large. In this case, the calculation results will fall in the validity area.

If $q_s > 8$, Y_S should be rather larger (than calculated with $q_s = 8$). In this case, the calculation results fall into the invalid range, and Y_s is therefore calculated with the effective q_s-value (>8).

In each case, if q_s exceeds, or falls below, the range 1...8, a warning is entered in the report. This report also shows which q_s value was used further on in the calculation.

If you want to change the procedure described here, you can do this either in the setup (STANDARD.Z12 file, etc) or in a saved file (*.Z12, etc.). To do this, open the file in Notepad and change this line:
ZS.qsLIMIT=0;
to: ZS.qsLIMIT=1; (q_s not changed)
or to ZS.qsLIMIT=2; (q_s<1 set to q_s=1, q_s>8 set to q_s=8)

23.3 Abbreviations used in gear calculation

Abb. in standards etc.	Abb. in KISSsoft	
a	a	Center distance (mm)
a_d	a.d	Reference center distance (mm)
A_a	A.a	Center distance allowance (mm)
As_e	As.e	Tooth thickness allowance at the normal section (mm)
α_{en}	alf.en	Angle at which force is applied (degree)
α_n	alf.n	Pressure angle at the normal section (degrees)
α_{Pro}	alf.Pro	Protuberance angle (degrees)
α_t	alf.t	Pressure angle on the reference circle (degrees)
α_{wt}	alf.wt	Operating pressure angle (degrees)
b	b	Facewidth (mm)
B_M	B.M	Therm. contact coefficient (N/mm/s.5/K)
β	beta	Helix angle at reference circle (degree)
β_b	beta.b	Base helix angle (degree)
c	c	Tip clearance (mm)
c'	c'	Single spring stiffness (N/(mm*µm))
c_γ	c.g	Contact spring stiffness (N/(mm*µm))
d	d	Reference diameter (mm)
d_a	d.a	Tip diameter (mm)
d_b	d.b	Base diameter (mm)
d_f	d.f	Root diameter (mm)
$d_f(x_E)$	d.f(x.E)	Root circle with profile shift for A_{se} (mm)
d_i	d.i	Inside diameter gear (mm)
d_{Na}	d.Na	Tip active circle diameter (mm)
d_{Nf}	d.Nf	Active root diameter(mm)
$d_{Ff(0)}$	d.Ff(0)	Root form diameter (mm)
d_{sh}	d.sh	External diameter of pinion shaft (mm)

d_w	d.w	Operating pitch diameter (mm)
D_M	D.M	Theoretical ball/pin diameter (mm)
	D.M eff	Effective ball/pin diameter (mm)
e_{fn}	e.fn	Normal gap width on the root cylinder (mm)
η_{tot}	eta.tot	Total efficiency
ε_α	eps.a	Transverse contact ratio
ε_β	eps.b	Overlap ratio
ε_γ	eps.g	Total contact ratio
f_f	f.f	Profile form deviation (mm)
$f_{H\beta}$	f.Hb	Helix slope deviation (mm)
f_{ma}	f.ma	Tooth trace deviation due to manufacturing tolerances (mm)
f_{pe}	f.pe	Single pitch deviation (mm)
f_{sh}	f.sh	Tooth trace deviation due to deformation of the shafts (mm)
F_a	F.a	Axial force (N)
$F_{\beta y}$	F.by	Actual tooth trace deviation (mm)
F_n	F.n	Normal force (N)
F_r	F.r	Radial force (N)
F_t	F.t	Nominal circumferential force in the reference circle (N)
	Fase.d	Tip chamfer (mm)
g_α	g.a	Length of path of contact (mm)
Γ	Gamma	Gamma coordinates (point of highest temperature)
h	h	Tooth height (mm)
h_{aP}	h.aP	Addendum reference profile (in module)
h_F	h.F	Bending moment arm (mm)
h_{fP}	h.fP	Dedendum reference profile (in module)
h_k	h.k	Protuberance height (in module)
ha	ha	Chordal height (mm)

H	H	Service life in hours
I	I	AGMA: Geometry factor for pitting resistance
Impulse	Impulse	Gear driving (+) / driven (-)
j_n	j.n	Normal backlash (mm)
j_t	j.t	Rotational backlash (transverse section) (mm)
j_{tSys}	j.tSys	Backlash of the entire system (mm); for planetary stages
k	k	Number of teeth spanned
$k * m_n$	k * m.n	Tip alteration (mm)
K_A	K.A	Application factor
$K_{B\alpha}$	K.Ba	Transverse coefficient - scuffing
$K_{B\beta}$	K.Bb	Width factor - scuffing
$K_{B\gamma}$	K.Bg	Pitch factor - scuffing
K_f	K.f	AGMA: Stress correction factor
$K_{F\alpha}$	K.Fa	Transverse coefficient - tooth root
$K_{F\beta}$	K.Fb	Width factor - tooth root
$K_{H\alpha}$	K.Ha	Transverse coefficient - flank
$K_{H\beta}$	K.Hb	Width factor - flank
$K_{H\beta be}$	K.Hbbe	Bearing application factor
K_V	K.V	Dynamic factor
K_{wb}	K.wb	Alternating bending coefficient
k_w	K.w	Wear coefficient (mm^3/Nm)
l	l	Bearing distance l on pinion shaft (mm)
m_n	m.n	Normal module (mm)
m_{Red}	m.Red	Reduced mass (kg/mm)
m_t	m.t	Transverse module (mm)
M_{dK}	M.dK	Diametral measurement over two balls without backlash (mm)
M_{dKeff}	M.dKeff	Effective diametral measurement over two balls (mm)

M_{dReff}	M.dReff	Effective diametral roller mass (mm)
M_{rK}	M.rK	Radial measurement over one ball without backlash (mm)
M_{rKeff}	M.rKeff	Effective radial measurement over one ball (mm)
μ_m	mu.m	Mean friction factor (as defined in Niemann)
μ_m	my.m	Mean coefficient of friction
μ_m	my.my	Friction factor
n	n	Speed (RpM)
ν_{E1}	n.E1	Resonance speed (min^{-1})
N	N	Resonance ratio
N_L	N.L	Number of load changes (in mio.)
ν_{100}	nu.100	Kinematic nominal viscosity of oil at 100 degrees (mm^2/s)
ν_{40}	nu.40	Kinematic nominal viscosity of oil at 40 degrees (mm^2/s)
p_{bt}	p.bt	Base circle pitch (mm)
p_{et}	p.et	Transverse pitch on path of contact (mm)
p_t	p.t	Pitch on reference circle (mm)
P	P	Nominal power (kW)
$P_{V Z}$	P.VZ	Gear power loss due to tooth load (kW)
$P_{V Ztot}$	P.VZtot	Total power loss (kW)
P_{WaelzL}	P.WaelzL	Meshing power (kW)
R_Z	R.Z	Average total height (mm)
ρ_F	ro.F	Tooth root radius (mm)
ρ_{fP}	ro.fP	Root radius reference profile (in module)
ρ_{Oil}	ro.Oil	Specific Oil density at 15 degrees (kg/dm^3)
s	s	Distance on pinion shaft (mm)
s_{an}	s.an	Normal tooth thickness on the tip cylinder (mm)
s_{Fn}	s.Fn	Tooth root thickness (mm)
s_{mn}	s.mn	Normal tooth thickness chord, without backlash

		(mm)
	s.mn e/i	Effective normal tooth thickness chord with clearance (mm) (e: upper, i: lower)
S_B	S.B	Safety factor for scuffing (flash temperature)
S_F	S.F	Safety for tooth root stress
S_H	S.H	Safety for pressure at single tooth contact
S_{Hw}	S.Hw	Safety factor for contact stress on operating pitch circle
S_{Sint}	S.Sint	Safety factor for scuffing (integral temperature)
S_{SL}	S.SL	Safety for transmitted torque (integral temperature)
σ_F	sig.F	(Effective) tooth root stress (N/mm^2)
σ_{F0}	sig.F0	Nominal tooth root stress (N/mm^2)
σ_{Flim}	sig.Flim	Endurance limit tooth root stress (N/mm^2)
σ_{FP}	sig.FP	Permitted tooth root stress (N/mm^2)
σ_H	sig.H	Contact stress on the operating pitch circle (N/mm^2)
σ_{H0}	sig.H0	Nominal contact stress on the pitch circle (N/mm^2)
$\sigma_{HB/D}$	sig.HB/D	Contact stress HPSTC (N/mm^2)
σ_{Hlim}	sig.Hlim	Endurance limit Hertzian pressure (N/mm^2)
σ_{HP}	sig.HP	Permissible contact stress (N/mm^2)
σ_s	sig.s	Yield point (N/mm^2)
Σx_i	Total x.i	Total profile shift
T	T	Torque (Nm)
θ_B	the.B	Highest contact temperature (^{o}C)
θ_{int}	the.int	Integral flank temperature (^{o}C)
θ_m	the.m	Tooth mass temperature (^{o}C)
θ_{M-C}	the.M-C	Tooth mass temperature (^{o}C)
θ_{Oil}	the.Oil	Oil temperature (^{o}C)
θ_s	the.s	Scuffing temperature (^{o}C)

θ_{Sint}	the.Sint	Scuffing integral temperature (^oC)
U	U	Gear ratio
v	v	Circumferential speed reference circle (m/s)
v_{ga}	v.ga	Maximum sliding velocity on tip (m/s)
	Vqual	Accuracy grade according to DIN 3962 or ISO 1328
w	w	Nominal circumferential force reference circle per mm (N/mm)
W_k	W.k	Base tangent length (no backlash) (mm)
	W.k e/i	Effective base tangent length (mm) (e: upper, i: lower)
x	x	Profile shift coefficient
x_E	x.E	Profile shift at manufacturing for A_{se}
$X_{\alpha\beta}$	X.alfbet	Angle factor
X_B	X.B	Geometry factor
X_{BE}	X.BE	Geometry factor
X_{Ca}	X.Ca	Tip relief factor
X_e	X.e	Contact ratio factor
X_Γ	X.Gam	Distribution factor
X_M	X.M	Flash factor
X_Q	X.Q	Meshing factor
X_S	X.S	Lubrication factor (scuffing)
X_{WrelT}	X.WrelT	Relative structural factor (scuffing)
y_a	y.a	Run-in amount (µm)
y_b	y.b	Run-in amount (µm)
Y	Y	AGMA: Tooth form factor
Y_b	Y.b	Helix angle factor
Y_{drel}	Y.drel	Notch sensitivity factor
Y_e	Y.e	Contact ratio factor
Y_F	Y.F	Tooth form factor

Y_{NT}	Y.NT	Lifetime factor
Y_R	Y.R	Surface factor
Y_S	Y.S	Stress correction factor
Y_{st}	Y.st	Stress correction factor test gear
Y_X	Y.X	Size factor (tooth root)
z	z	Number of teeth
z_n	z.n	Virtual gear no. of teeth
Z_β	Z.b	Helix angle factor
$Z_{B/D}$	Z.B/D	Single contact point factor
Z_E	Z.E	Elasticity factor ($N^{1/2}$/mm)
Z_ε	Z.e	Contact ratio factor
Z_H	Z.H	Zone factor
Z_L	Z.L	Lubricant factor
Z_{NT}	Z.NT	Lifetime factor
Z_R	Z.R	Roughness factor
Z_V	Z.V	Speed factor
Z_W	Z.W	Work hardening factor
Z_X	Z.X	Size factor (flank)
ζ_w	zet.W	Wear sliding coefficient according to Niemann
ζ_a	zet.a	Specific sliding at the tip
ζ_f	zet.f	Specific sliding at the root

Part III
Shafts and Bearings

Chapter 24
Defining Shafts

This program consists of a base package and different expert add-ins. The following calculations are available:

- Deformation, force, torque and stress curves
- Eigenfrequencies (bending, torsion and axial movements)
- Buckling loads
- Static and fatigue strength
- Rolling bearing calculation
- Plain bearing calculation (hydrodynamic)
- Necessary width modification of pinion

Base package

In this module, you can input and modify geometry and material data, shaft specifications, drawing numbers, bearings, peripheral conditions, external forces and torques (simplified input for couplings, spur and bevel gears, worms, worm gears, belt pulleys etc.).

A shaft with the machine elements mounted on it (for example, gears or bearings) is defined in the graphical shaft editor.
The properties required to define a shaft in this editor are:

- Any dimensions (cylindrical and conical), axially symmetric cross section, solid and hollow shafts, beams (H-, I-, L-profiles etc.)
- Integrated drawing tool that enables simple corrections to be made to the shaft contour (diameter, lengths). You can change any of these elements by simply clicking on them with the mouse.
- Definition of notch geometries for the automatic calculation of notch factors. The following notch geometries are available here:
  - Radius
  - Chamfer
  - Relief groove
  - Interference fit

  - Longitudinal groove
  - Circumferential groove
  - Square groove
  - V-notch
  - Spline
  - Cross hole
- Force and torque elements can be input in any spatial position. The following elements are predefined:
  - Cylindrical gear
  - Bevel gear
  - Worm
  - Worm wheel
  - Coupling
  - Rope sheave/V-belt
  - Centrical force
  - Eccentric force
  - External masses with moment of inertia (additional mass)
  - Power loss
- Calculation of:
  - Shaft weight
  - Moment of inertia
  - Axial force
  - Static torsion of the shaft

- Clear representation of geometry data and the calculated bearing and peripheral forces both on screen and on paper.

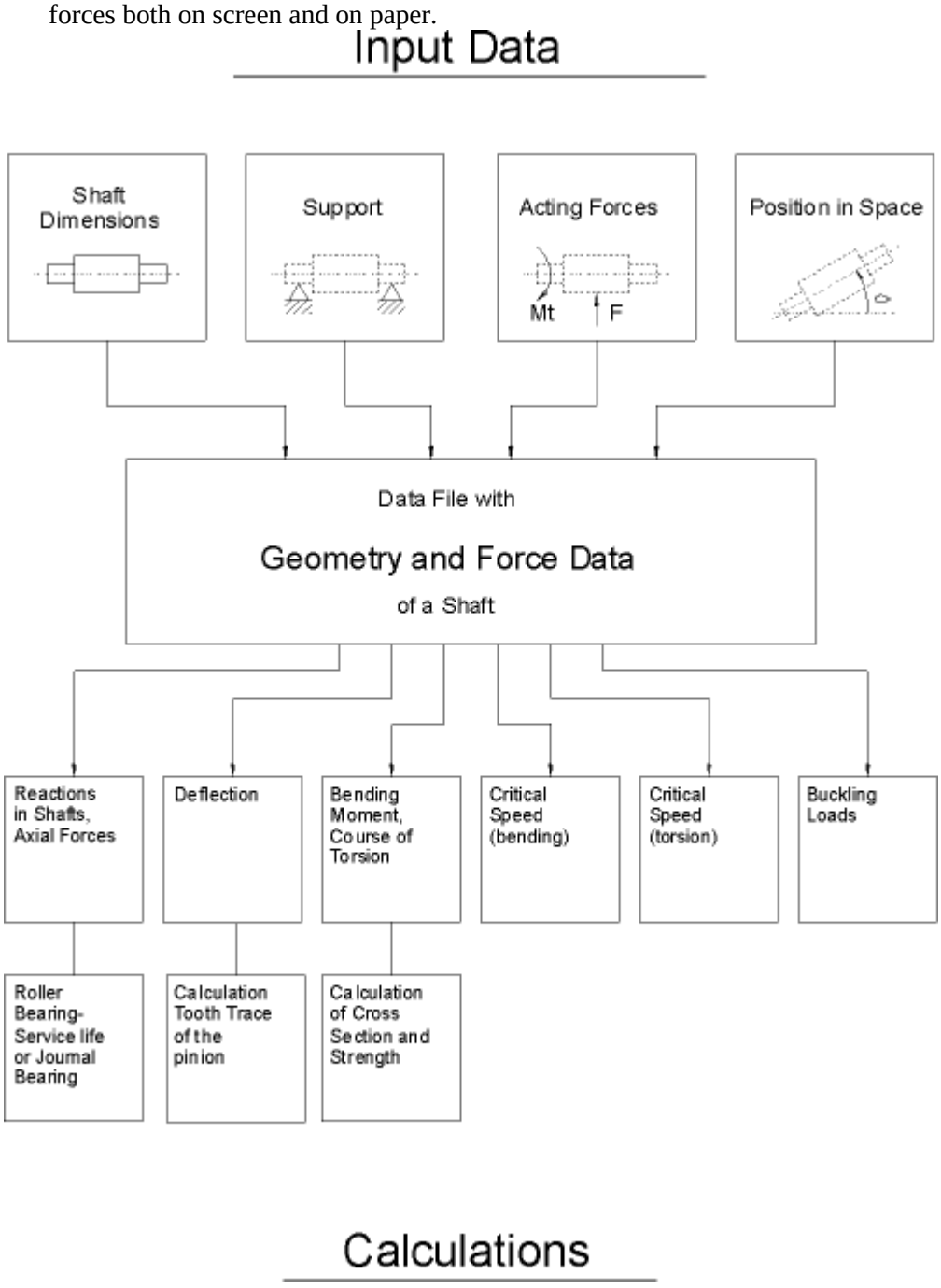


Figure 24.1: Flow-chart of the modules for shaft and bearing calculation in KISSsoft.

24.1 Input window

The KISSsoft system offers a range of different input windows, in which you can define shafts. The Shaft editor (see page III-702) shows a graphical representation of the shaft system. The Elements tree (see page III-703) illustrates the structure of the shaft system in a tree structure. You define the parameter of an element in the Elements editor (see page III-705).

24.1.1 Shaft editor

The shaft editor shows a graphical representation of the shaft system. Use the vertical tool bar on the right-hand edge of the Shaft editor to add the most frequently used elements. If your system has several shafts, the new element is always added to the active shaft. A shaft becomes active when one of its elements is selected. If no element has been selected, the last shaft is the active one.

You can select options in the context menu to print the graphic in the shaft editor and save it as a picture file. Each of the different elements also has interactive context menus.

24.1.2 Elements tree

The Elements tree illustrates the structure of the shaft system in a tree structure. Shafts are at the highest level. The connecting elements in systems with several shafts are also shown. Each shaft groups its main elements by Outer contour (see page III-711), Inner contour (see page III-719), by Strength (see page III-713), Bearings (see page III-725)and Cross sections (see page III-730). For the cylinder and conus main elements, the sub-elements are located on a further sub-level.

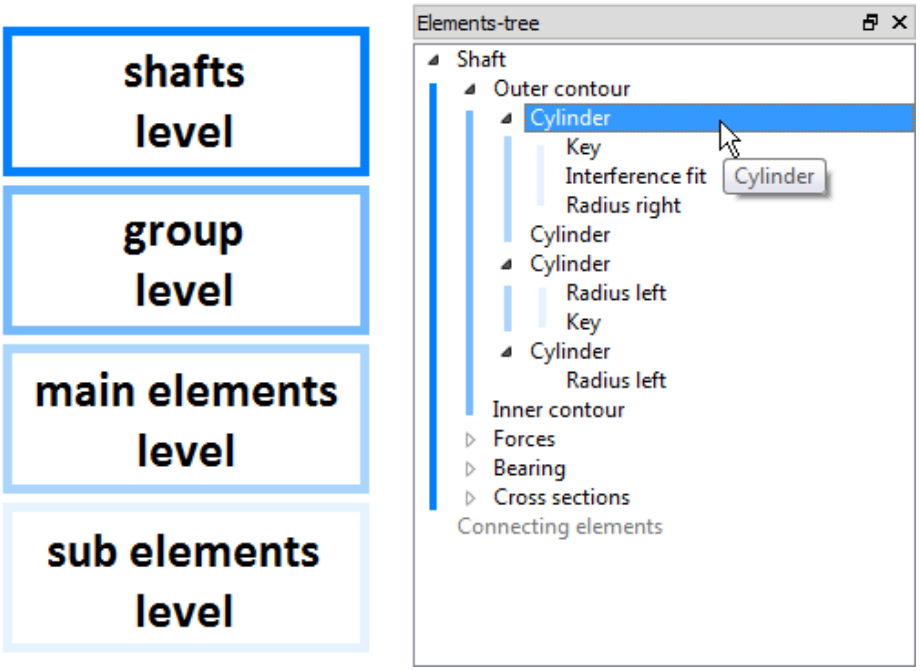


Figure: Levels in the Elements tree

You can select, copy, insert and delete elements via the Elements tree. You can select options in a context menu to display the actions that are available for each element. Special actions are available, depending on the element type. You can also arrange shafts, rolling bearings and cross sections. You can also import (see page III-717)/export (see page III-719) outer and inner contours to DXF.

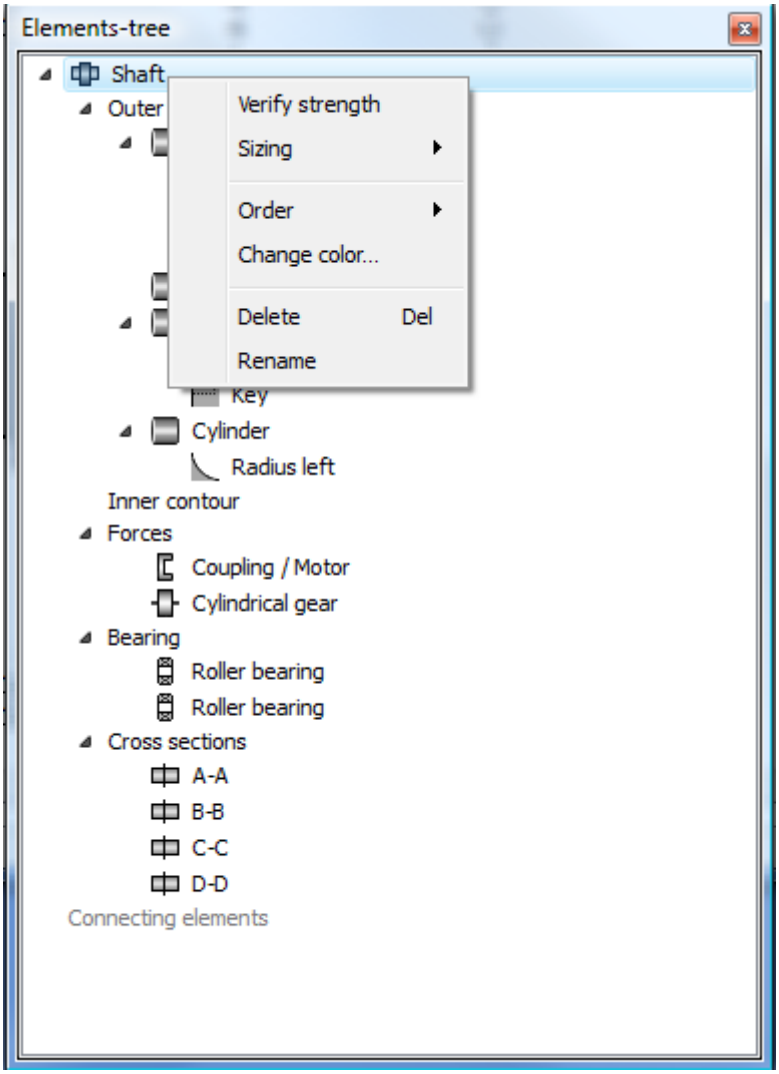


Figure: Context menu in the Elements tree

24.1.3 Elements list

The Elements list shows groups of elements in table format. You can edit the parameter listed in the table directly in the Elements list. Using the context menu, you can insert elements quickly and easily.

24.1.4 Elements editor

In the elements editor, you can edit any of the selected element's parameters.

24.2 Element overview

24.2.1 The Shaft element

To input a shaft, click on the first icon in the vertical tool bar in the Shaft editor (see page III-702). You will also find the `Add shaft` option in the Elements tree (see page III-703) context menu. A new entry appears at the end of the Elements tree. Single click on the shaft element in the elements tree to input parameters for the shaft in the elements editor (see page III-705), as shown in Figure 24.4).

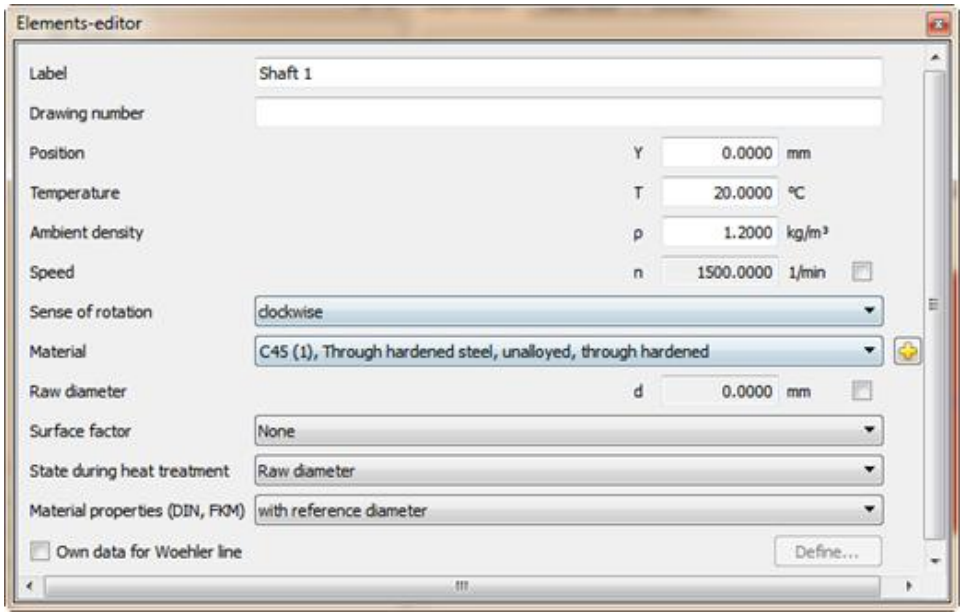


Figure 24.4: `Elements editor` for inputting shaft parameters

The next section describes the individual input fields, in which you enter parameters for a specific shaft.

24.2.1.1 Drawing number

In the `Drawing number` input field, you can enter a string of any characters apart from ";" (semicolons). The drawing number you enter here does not affect the calculation.

24.2.1.2 Position

The `Position` input field is where you enter the `Y`-coordinate of the starting point of the shaft relative to the global coordinates system.

NOTE

Global coordinates are indicated by upper case letters. Lower case letters indicate a shaft's local coordinate system.

24.2.1.3 Temperature

The shaft may undergo thermal expansion if the shaft's temperature is not the same as the Reference temperature (on page III-733). In addition to the thermal expansion of the shaft, the thermal expansion of the housing can also be taken into account by the housing temperature (see page III-734).

24.2.1.4 Ambient density

Bodies placed in hydrostatic fluids experience buoyancy. The value here is the same as the weight of the displaced medium, and is defined by the volume and the density of the displaced medium. KISSsoft takes the buoyancy effect into account, if you enter the appropriate ambient density value. The default setting is for air density. The next table lists technical values for other media.

Medium	**Air**	**Water**	**Oil**
Density ρ	1.2	998	772

Table 24.1: Densities [kg/m^3] of a few important fluids where ϑ = 20°C and p = 1016 mbar

NOTE

If a shaft is operated in different ambient media, for example, as is the case for input shafts in ships, you can combine two individual shafts, each of which has different ambient density data, by using the `Connections` element in the Elements tree, and then calculate them as a single shaft.

24.2.1.5 Speed

Shaft speed [1/min] around its longitudinal axis. If you click the checkbox to the right of the input field, you can change the speed independently of other shafts.

However, if this checkbox is not active, the value is taken from the Speed (see page III-732) input field in the Basic data input window.

24.2.1.6 Direction of rotation

The shaft's direction of rotation can influence the way loads are distributed along the shaft, for example, as the result of helical gear teeth, and therefore affect the service life of the bearing. Click the checkbox to the right of the Speed input field to view and select these entries from the drop-down list. However, if this checkbox is not active, the value is taken from the Sense of rotation (see page III-733) input field in the Basic data input window.

24.2.1.7 Material

You can select a shaft material from this drop-down list and therefore assign a specific material to each individual shaft. If you use this function together with the Connections element in the elements tree, you can generate shafts made of different materials.

24.2.1.8 Base size

The Raw measure input field is decisive for strength calculation. However, if you select the Pre-turned to actual diameter option in the State during heat treatment drop-down list, in the Strength input window, the setting for the raw measure value has no effect on the calculation. In contrast, if the selection is set to Raw diameter, the largest rounded shaft diameter will be selected, and the strength calculation will be performed using this value. Click the checkbox to the right of the input field to specify your own diameter for the blank before it is turned.

24.2.1.9 Hardening depth (FKM)

The hardening depth input field is required for estimating the fatigue strength of surface treated parts. The hardening depth is used to define the position of the transition surface layer relative to the core. It varies depending on which surface treatment process has been used. This input is not required for the main calculation. You will find another description of this calculation in the "Estimate the fatigue strength of surface treated parts" section.

24.2.1.10 Surface factor

In this selection list, you can define if an additional surface factor should be applied or not. Here, you can select either `Rollers` or `Shot peening`.

24.2.1.11 State during heat treatment

To define the technological size coefficient $K_{1,deff}$, select one of these three options:

- Pre-turned to actual diameter. The raw diameter has no influence on the technological size coefficient. The value $K_{1,deff}$ is recalculated for each cross section based on the actual diameter size.
- Raw diameter. $K_{1,deff}$ is determined once from the raw diameter and applied to all cross sections.
- Pre-turned to actual diameter (shoulders K1 from d)

NOTE

You can also define the `Base size` field in the `elements editor` for the corresponding shaft. To do this, input the dimension of the raw material which was used to generate the final material characteristics during the last heat treatment. If this involves a solid shaft, enter the external diameter of the blank part. For a pipe, enter the wall thickness and, for a cast part, enter the greatest wall thickness.

24.2.1.12 Heat treatment of hollow shafts in the full state

If you do not input the raw diameter for a "heat treatment state", and you are investigating a hollow shaft (di > 0.1*da), you can use this option to specify

whether the coefficients are determined using the solid shaft or hollow shaft. This option is only valid for the FKM and DIN calculation methods.

Table 1.2.3 in the 6th Edition of the FKM Guideline shows data for both a hollow shaft and a solid shaft, and also methods for Case 1 and Case 2. Case 1 is for parts made of treated steel, case hardening steel etc., Case 2 is for parts made of unalloyed steel, normally annealed through hardening steel, etc. KISSsoft automatically calculates values for these 2 cases when you input the material.

According to DIN 743-2, the shaft diameter is used as the deff for coefficients K2 and K3. If a solid shaft is involved, the diameter is used for coefficient K1. If a hollow shaft is involved, the wall thickness s or 2* the wall thickness is used for deff, according to FKM.

24.2.1.13 Material properties

In the `Material properties` drop-down list, you can specify how KISSsoft is to define the material characteristic values that are relevant to strength:

1. `with reference diameter` Values are taken from the database (at reference diameter) and multiplied with K_1
2. R_p, R_m `as stated in database, sW for reference diameter` The values R_p and R_m are determined according to size (excluding K_1), and the fatigue strength σ_W is determined for the reference diameter entered in the database and then it is multiplied with K_1.
3. R_p, R_m `as stated in database, σW constant` The values R_p and R_m are determined according to size, and the fatigue strength σ_W is taken from the database without being influenced by the geometric size factor. The size factor K_1 is not taken into account here.
4. R_p, R_M `according to the database, σW is calculated from` R_m The values R_p and R_m are taken from the database, and σ_W is determined from the tensile strength R_m in accordance with the standard.

The data of the material used to calculate the shaft strength is derived from the values in the database as follows:

- Fatigue limit factors (for tension/compression, bending, etc.) are taken directly from the material database. There, these values are defined for every calculation method. If data for these materials has been specified in the calculation method, it is these values that are used.
- Tensile strength values are stored in the database according to their diameter as defined in the specific EN standard. The actual tensile strength is used to fetch the tensile strength value from the database and use this in the calculation. This method of defining the effective tensile strength is very reliable and can be used for every calculation method. It has the effect that the same values are used for each calculation method.

  When you specify a calculation method, you can decide to use the material database on the basis of the requirements given in the corresponding standard. Then, the real tensile strength is defined using the thickness factor taken from the base tensile strength of the sample diameter (normally 10 mm), according to the standards (FKM or DIN. If you use Hänchen, this triggers an error message).

- The yield point or strain limits are taken either from the database or from the standard, in the same way as for the tensile strength.

24.2.1.14 Own data for S-N curve (Woehler lines)

Click the `Own data for Woehler line` checkbox to define your own S-N curve (Woehler lines). You can also enter values for the sustainable damage or Miner total here. If you do not activate this checkbox, the program will define the S-N curve (Woehler lines) in accordance with either DIN 743 or FKM. You should specify your own S-N curve (Woehler lines), or adjust the sustainable damage value if you are modifying your calculation to suit the results of specific tests.

24.2.1.15 Taking the results into account in the report

If this flag is set, the corresponding shaft is output in the main shaft report, along with all its elements (external/internal contour, force elements, bearing). However, this is only valid for inputs and does not affect the results of the calculation. By default, this flag is set.

24.2.2 Outer contour

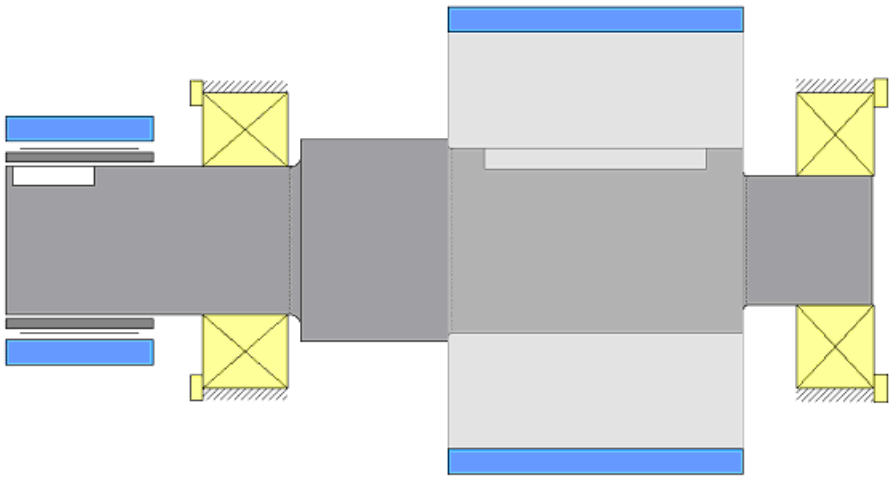
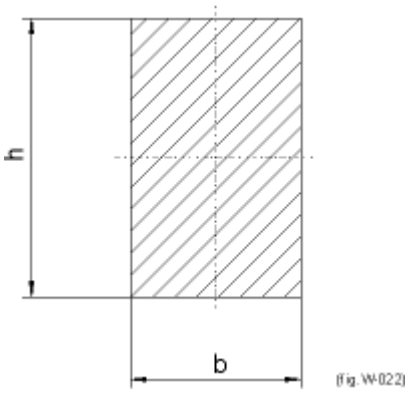


Figure 24.5: Display the outer contour in the `Shaft editor`

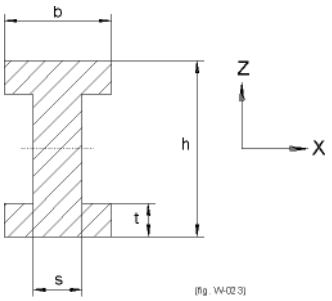
You can use (hollow) cylinders, (hollow) cones and beams to define the shaft geometry. To enter a new element, select the element you want at group level in

the elements tree, e.g.`Outer contour`. Click the right-hand mouse button on this element to add it to the group at the right-hand end of the shaft. Alternatively, you can select an existing element at element level (e.g. `cylinder`) and then right-click with the mouse to open a context menu. In it, select the `Add element before(after)` option to open another submenu, in which you select an element to be inserted at a position relative to the existing element.

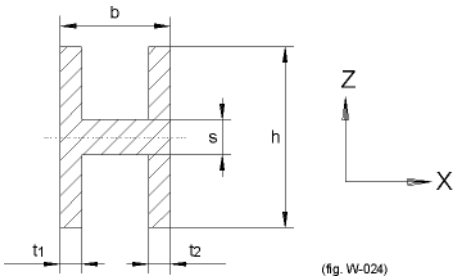
Possible profiles for beams are:



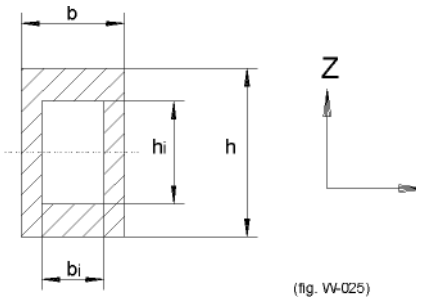
Rectangular profile



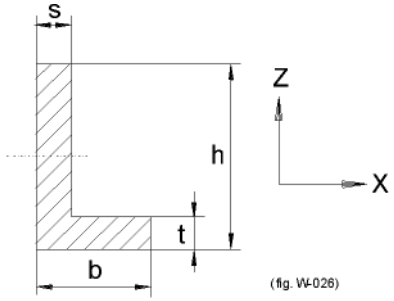
Double T profile



H profile



Rectangular profile (hollow)



L profile

24.2.2.1 Defining subelements

Before you can define a subelement, first select the main element to which you want to add this subelement in the Elements tree. Then right-hand mouse click to select the subelement you require. The inserted subelement now appears in the Shaft editor, and its corresponding notch factors are defined in the strength calculation. Once you have defined a subelement, you can activate it in the same way as a main element (see Activate).

Adding subelements:

- **Radius right/ left**
  Input values:
  - Radius: Size of the radius
  - Surface roughness: Radius surface
- **Chamfer right/left**
  Input values:
  - Length: Chamfer length
  - Angle: Chamfer angle
- **Relief groove right/left**
  Input values:
  - Relief groove form: Select the relief groove form as defined in DIN 509 or the FKM
  - series (DIN 509): (Selection: Series 1, radii as defined in DIN 250; Series 2, special radii)
  - Stress (DIN 509): (with conventional stress; with increased fatigue strength)
  - relief groove length: Length of the relief groove in the direction of the axle
  - Transition radius: Radius between the end of the relief groove and the next element
  - Depth of recess: Recess depth
  - Surface roughness: Recess surface
- **Interference fit**
  Input values:
  - Interference fit length: Interference fit length
  - Type of interference fit: (Selection: Slight interference fit, interference fit and interference fit with end relief)

- Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the interference fit

Longitudinal groove
Input values:

- Length: Length of the key groove (longitudinal groove)
- Standard: Standard used for the key groove
- Key groove width: Width of the key groove (can be entered if 'Own input' is selected)
- Key groove depth: Depth of the key groove (can be entered if 'Own input' is selected)
- Number of keys: (i > 2 not permitted according to standard)
- Manufacturing process: (Selection: end milling cutter, side milling cutter, combined with interference fit (FKM))
- Surface roughness: Key groove surface
- Reference measure: this specifies the measurement from the left end of the selected element up to the start of the key groove

Circumferential groove
Input values:

- Depth: Depth of the circumferential groove
- Rounding in the groove bottom: Radius of the circumferential groove
- Surface roughness: Surface of circumferential groove
- Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the circumferential groove

Square groove
Input values:

- Width: Width of the square groove
- Depth: Depth of the square groove
- Radius: Radius of the square groove
- Surface roughness: Surface of the square groove
- Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the square groove

V-notch
Input values:

- Depth: Depth of the V-notch
- Surface roughness: Surface of the V-notch
- Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the V-notch

Spline
Input values:

- Standard: Normal range of the spline (click the  button to select the required size from a list)
- Tip circle: you can either select this from a list of standards or input your own value
- Root circle: you can either select this from a list of standards or input your own value
- Number of teeth: you can either select this from a list of standards or input your own value
- Module: you can either select this from a list of standards or input your own value
- Surface quality: Spline surface quality
- Length: Spline length
- Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline

Spline shaft
input values:

- Tip circle: Tip circle of the spline shaft (straight-sided spline)
- Root circle: Root circle of the spline shaft (straight-sided spline)
- Number of keys: Number of keys
- Spline shaft (straight-sided spline) root rounding: (Selection: Shape A, Shape B and Shape C)
- Length: Length of the spline shaft (straight-sided spline)
- Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline shaft (straight-sided spline)
- Surface quality: Spline shaft (straight-sided spline) surface

Cross hole
Input values:

- Bore diameter: Diameter of bore
- Surface roughness: Axial boring surface
- Reference measure: this specifies the measurement from the left end of the selected element up to the position of the cross hole

Thread
Input values:

- Label: Thread label
- Thread depth: Thread depth
- Rounding: Rounding in the notch bottom of the thread
- Length: Thread length
- Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the thread
- Surface roughness: Thread surface

General notch effect
Input values:

- Width: Width of the overall subelement
- Notch factor bending/ torsion/tension-compression/shearing force: you can enter the notch factors directly here.
- Surface roughness: Surface of the overall subelement
- Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the overall subelement

You can enable the "Conical shoulder" notch type directly in the Strength calculation (see section "Cross section types" on page III-778).

24.2.2.2 Importing the shaft geometry

Right-hand mouse click next to the outside/ or inner contour to open a pop-up menu (see figure). Click `Import` to import a .ktx or a .dxf file.

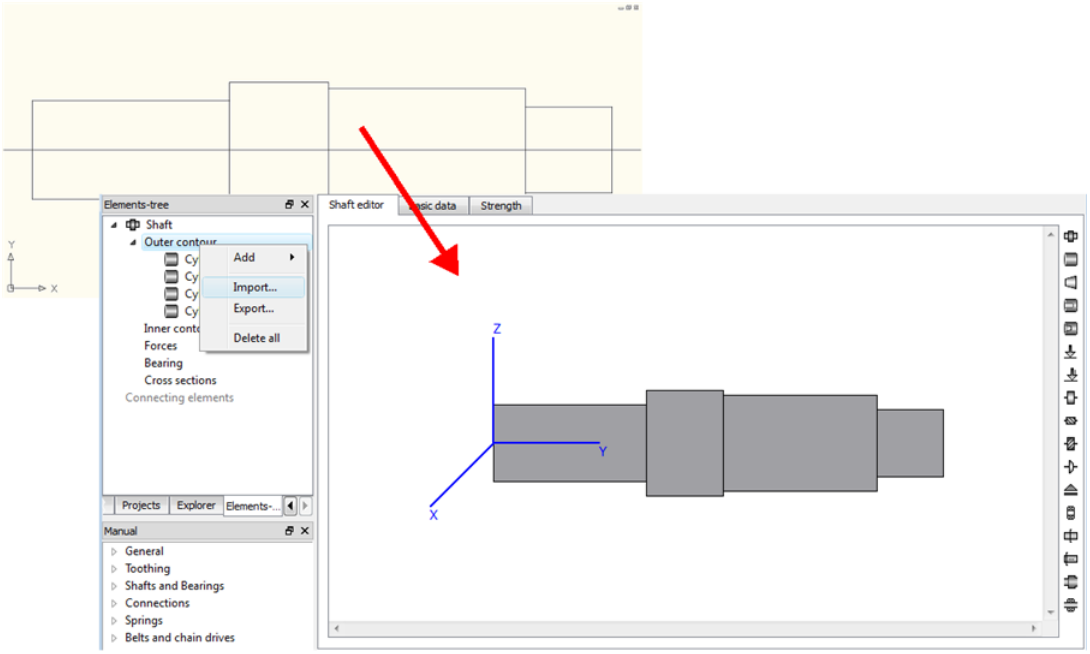


Figure 24.6: Importing the shaft geometry from a dxf file

Reading (importing) a ktx file:

In KISSsoft, go to the Shaft calculation elements tree and right-click on the `Outer contour element` to open a context menu, in which you select the `Import` option. Select the required *.ktx file and click on `Open`. The shaft contour is now uploaded into KISSsoft.

Reading (importing) a dxf file:

The outside and inner contour (if present) of the shaft should be output individually by the CAD system.

NOTE:

You can use the default value **ALL** for the layer name, so that all layers are imported. You can also import the contours as variants in different layers. To do this, enter the layer name in the appropriate input field. If you don't know the exact

layer name, you can input an invalid name as a test (for example, xxx). If you then try to read (import) this, the resulting error message will list the valid layer names.

- Draw the shaft contour with a mid line in a CAD system. Use the X-Y plane as the coordinates system (the X-axis is the rotational axis) to ensure the contour is interpreted correctly after it has been read imported and so that the shaft is drawn in KISSsoft in the Y-Z plane (the Y-axis is the rotational axis). Save the shaft geometry as a *.dxf file.
- In KISSsoft, go to the Shaft calculation elements tree and right-click on the `Outer contour element` to open a context menu, in which you select the `Import` option. Now select the *.dxf file you require, and click `Open`.
- This opens another dialog, in which you can define the layer, the point of origin (X/Y) and the angle of the symmetry axis. After you have input this data, click OK to close this dialog. The shaft contour is then loaded with these details.

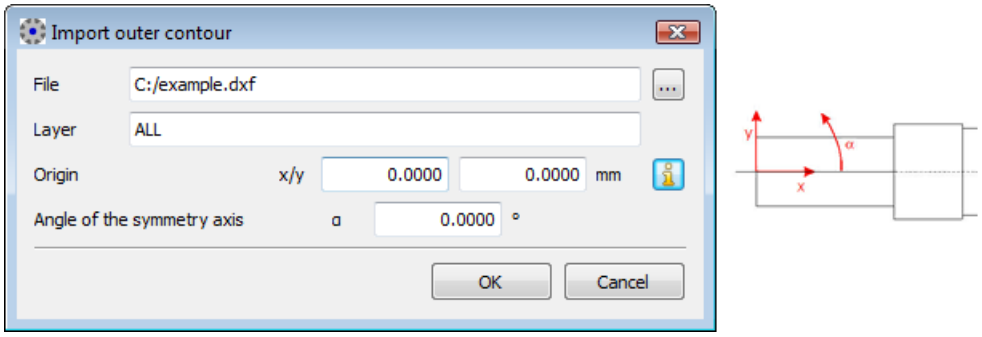


Figure 24.7: Import dialog for loading dxf files

24.2.2.3 Export shaft geometry

Right-hand mouse click next to the outside/ or inner contour to open a pop-up menu (see figure). If you select `Export`, you can create either a *.ktx or *.dxf file.

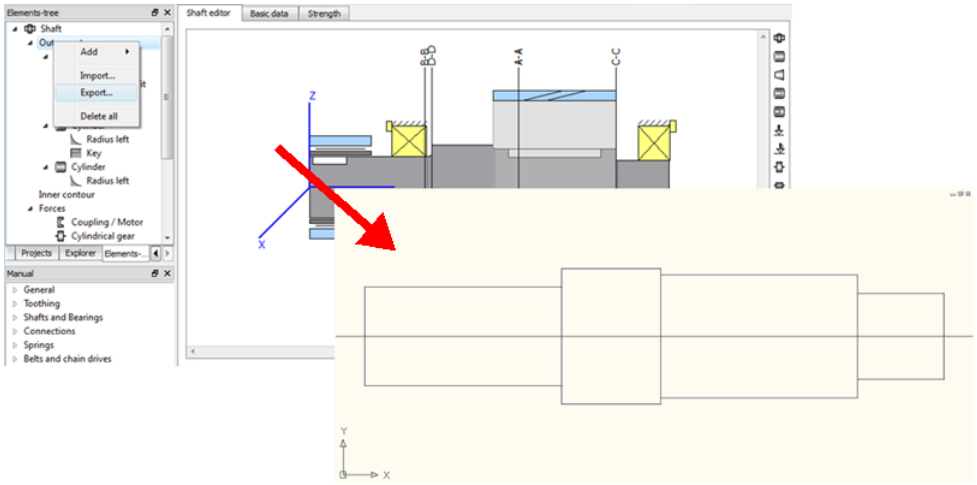


Figure 24.8: Exporting shaft geometry to a dxf file

Procedure for importing into a file:

- You can export previously defined shafts from the Shaft editor. In the KISSsoft Elements tree for shaft calculation, right-hand mouse click on the required element e.g. `Outer contour`, start the Popup menu start and select `Export`. You can export the inside or outer contours of the different shafts.
- After you select a contour, a dialog opens in which you can define the name of the *.ktx or *.dxf file.

24.2.3 Inner contour

The inner contour is generated from left to right (just like the outer contour). For example, if you want to generate a shaft with an axial hole from the right-hand side, you must first input data for an inside cylinder starting from the left-hand side with a diameter of 0 that extends up to the point where the bore begins.

24.2.4 Forces

24.2.4.1 Forces

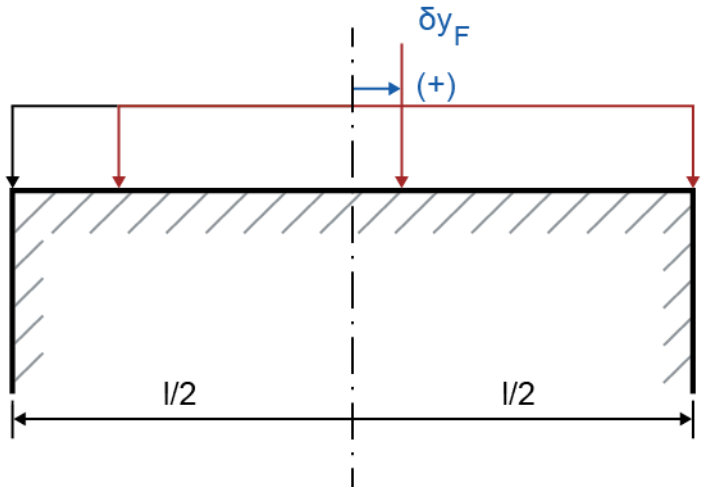
Forces can be applied to any point on the shaft and even outside(!) the shaft. Different methods are available for defining force transmitting elements (such as gears)

or even individual forces. In most force elements, the direction of the torque is defined by setting them as "driving" or "driven". "Driving" means that the shaft is the driving element or that the torque of the direction of rotation is counter to the direction of rotation. See also 24.3.5 (see page III-733).

Comments about special elements:

- Cylindrical gear
  Position of contact: specify the location of the point of contact with the paired gear according to Figure 24.3 on page III-731 (this point is where the forces apply).
  Instead of simply entering the reference diameter, you get a more accurate result if you enter the operating pitch diameter and the operating pressure angle instead of the nominal pressure angle.
  Click the Convert button to calculate these values.

  The center point of the load application is by default the center of the gear. This can be changed by defining the load application position offset δyF, according to the following formulae:



  Original starting position of gear load application: L0 = middle of the gear - (gear width/2)
  Original final position of the load application on the gear: R0 = middle of the gear + (gear width/2)

  * If δyF > 0
  New starting position of the load application on the gear: L1 = L0 + 2 * δyF
  New final position of the load application on the gear: R1 = R0

  * If δyF < 0
  New starting position of the load application on the gear: L1 = L0

New final position of the load application on the gear: R1 = R0 + 2 * δyF

The calculations shown above are only valid if the corresponding setting is enabled in the module-specific settings. Use the same process for all the gear elements.

- Bevel gear
  Position of contact: refer to the data for cylindrical gears.
  The position of the bevel gear can be converted using the bevel gear data. The reference point for positioning is the middle of the bevel gear width on the reference cone. The position of the bevel gear can be converted using the position of the axis crossing point on the shaft and other bevel gear data.

  An additional force component due to friction (μ = 0.05) is taken into account when calculating hypoid gears.

- Face gear
  The reference cone angle for face gears is always set to 90° (this input cannot be changed).

- The worm
  is usually a driving element. Its efficiency is included in the calculation of force components. Position of contact: refer to the data for cylindrical gears.

- The worm wheel
  is usually "driven". Its efficiency is included in the calculation of force components. Position of contact: refer to the data for cylindrical gears.

- Rope sheave
  Direction of rope sheave: Input the direction of the resulting belt forces as shown in Figure 24.3 on page III-731.

The direction of the helix angles and the positions of the elements are defined in Figure 24.9.

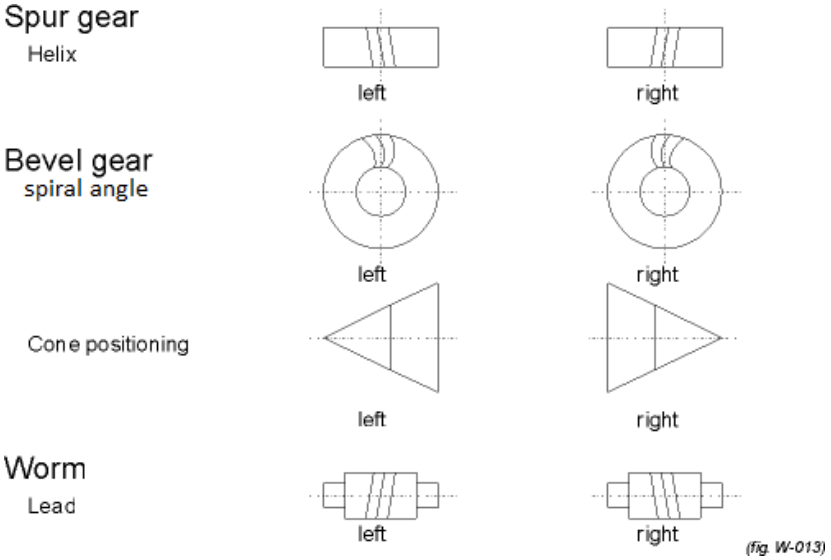


Figure 24.9: Defining the direction of force elements.

Eccentric force

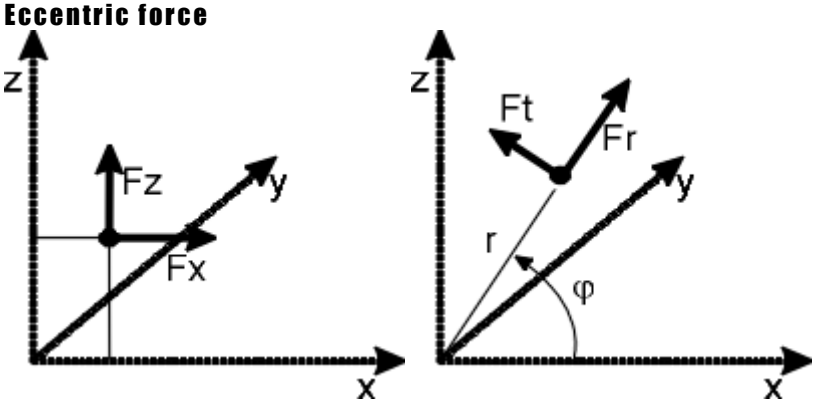


Figure 24.10: Cartesian/polar coordinates for eccentric force

You can enter values for eccentric force either in Cartesian or polar coordinates (see Figure24.10). You can change the coordinates system in the `Drawings/Settings` tab in the Shaft editor.

Transferring data from gear calculation

In the Elements editor, you can import the data used to define spur and bevel gears from a gear calculation file. Select the element you require in the Elements tree and then click on the Read data from file checkbox. Then select the gear number (1 to 4). The data relevant to these gear pairs is then imported directly. In this situation, the data at the pitch point is used instead of the data at the reference circle.

Important: If the Read data from file option in this input window remains active, data will be imported again from the gear calculation every time you call the shaft calculation function. If you then change the gear data later on, the new data will automatically be transferred with it! If the flag is not set, the data is only copied once from the gear calculation, and not updated later on. For this reason, you cannot change the contents of most input fields in the gear data input screen when linked files are involved. You can only change the Position of contact and the Y-coordinate.

24.2.4.2 Coupling

A coupling transmits torque and can also be subject to radial and axial forces. From the torque (or the specified power and speed) you can calculate the circumferential force to

$$F_t = \frac{M_t \cdot 2}{d} \tag{24.2}$$

F_t = Circumferential force

M_t = Torque

d = Effective diameter

Calculating radial force for a coupling:

$$F_r = K_2 \cdot F_t \tag{24.3}$$

F_t = Circumferential force

K_2 = Radial force factor

Define the direction of the force in the input window. You are also prompted to enter the mass of the coupling so it can be included in the calculation as a gravitational force.

Calculating axial force for a coupling:

$$F_a = K_3 \cdot F_t \tag{24.4}$$

F_t = Circumferential force

K_3 = Axial force factor

The axial force acts along the center line of the shaft.

24.2.4.3 Mass

Masses placed on the shaft are used as moments of inertia to determine the critical speeds. They are to be considered as a gravitational force.

24.2.4.4 Magnetic tension

The radial and axial forces produced by electromagnetic windings are included in the calculations.

Calculating radial force:

$$F_R = K_1 \cdot D \cdot L \cdot v \cdot \frac{f}{del0} \tag{24.5}$$

K_1	=	0.1 for three-phase motors in which the number of poles is 2
		0.2 otherwise
D	=	(mm) inside diameter of the stator of three-phase motors
		or external diameter of the rotor of direct current motors
L	=	(mm) Length of the active "packet of plates" (excluding the cooling slits)
v	=	Damping factor:
		Three-phase current asynchronous motor: Squirrel cage: v = 0.25
		Three-phase current asynchronous motor: Wound rotor: v = 0.7

		Three-phase current synchronous motor v = 0.5
		DC motor with wave formation v = 1.3
f/del0	=	Ratio of the mean eccentricity and the nominal air gap
		= 0.2 for three-phase motors
		= 0.1 for DC motors

Calculating axial force:

F^*_A	=	35 . α/D
F^*_A	=	Axial force factor
T	=	Torque (Nm)
α	=	Axial groove lead (deg)
D	=	(mm) inside diameter of the stator of three-phase motors
		or external diameter of the rotor of direct current motors

24.2.5 Bearing

In addition to calculating the shaft, you can export rolling bearings and general bearings as separate rolling bearing or plain bearing files (`File > Export`). Please note that the damping values entered for all types of bearings and connections, are used only for forced response recalculation.

24.2.5.1 Bearing (general)

All elements of a bearing (rigid or elastic) are considered to be a bearing. Input a fixed bearing, right mounted, left mounted, or thrust, bearing to determine the point on the shaft at which axial force is transmitted. This information is also used in the rolling bearing calculation. In taper roller bearings (or similar configurations) it is not always obvious which bearing is subject to the axial force. In this case, you must enter the mounting data for the bearings. You can also specify a radial or axial offset in the bearing alignment. This enables you to take into account other factors such as the simulation of assembly error.

24.2.5.2 Rolling bearings

In addition to general bearings, you can also select specific rolling bearings. The bearing data is then taken from the bearing database. This means the bearing's geometry data is already available, and you can draw the bearing using the width and external diameter values. In addition, for a bearing with a contact angle, the direction of the force can be taken into account in the calculation. You can also define a bearing clearance for each rolling bearing (according to DIN 620 C2, C0, C3, C4 or Own input).

If a suitable entry is present in the bearings database, the bearing stiffness value is taken from there. You can overwrite the stiffness value on the interface. You also have the option of defining stiffness in a file (for example W05-Stiffness.dat), and then use the data to calculate the local operating stiffness of the bearing. The file should include the bearing clearance in the displacement force curve, which is why the value input for clearance is set to 0 when the file is imported.

A pretension force, applied on the outer ring, can be used instead of the offset to define the pretension force on the bearing. This is only taken into account for bearings with inner geometry, and only if the corresponding bearing can accept an axial pretension force. The pretension force is then transformed internally to an equivalent axial offset of the outer ring. For bearings with inner geometry, you can also specify a rotation around axis X and Z of the outer ring. This could then be used, for example, to model the housing deformation, and make it possible for you to enter the FEM results directly.

The diametral bearing clearance defines the diameter-related clearance of a bearing. The diametral bearing clearance for a deep groove ball bearing is defined as:

$Pd = do – di – 2 * Dw$

Here, Pd is the diametral bearing clearance, do is the external raceway diameter, di is the internal raceway diameter and Dw is the rolling body diameter. Similar definitions are used for other bearing types, which vary depending on which particular type is involved.

24.2.5.3 Constraints on various bearings

Options for selecting a `Rolling bearing` with displacement and rotation options:

Rolling bearing selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating

Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating
Thrust bearing adjusted on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Thrust bearing, adjusted on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Thrust bearing adjusted on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating

Options for selecting a `Bearing (in general)` with displacement and rotation options:

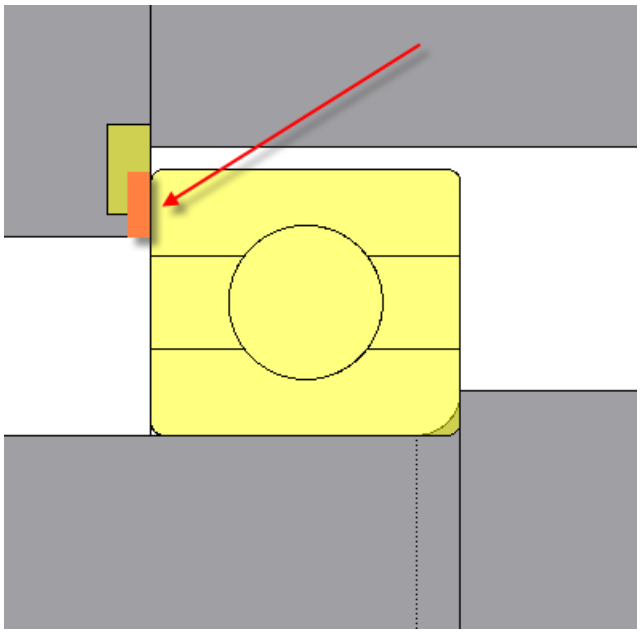
Bearing (in general) selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Own input	Own definition	Own definition	Own definition	Own definition	Own definition	Own definition
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating
Thrust bearing adjusted on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Thrust bearing, adjusted on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Thrust bearing adjusted on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating
Fixed	fixed	fixed	fixed	fixed	fixed	fixed

ux, uy, uz: Displacement in X-, Y-, Z-direction.

rx, ry, rz: Rotation about X-, Y-, Z-direction.

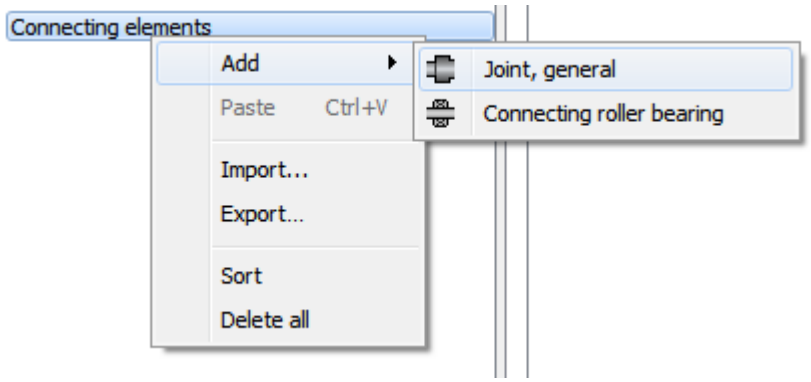
24.2.5.4 Pure thrust bearing

If a bearing is defined as a pure thrust bearing (left/right/both side layout) no radial forces can be transferred to it. Any radial forces created by the pressure of the outer ring on the housing and the corresponding coefficients of friction will be ignored.



24.2.6 Connection elements

A number of coaxial shafts can be connected by two different connection elements: a general connection or a connecting rolling bearing. The connection between these shafts, one defined as "shaft outside" and the other as "shaft inside", defines the constraints on the shafts over all possible degrees of freedom at that connection point, i.e. three relative displacements along axis x, y (the axial direction) and z, and three relative rotations around axis x, y and z.



NOTE

The number of coaxial shafts is limited to 30.

24.2.6.1 Connection, general

In general, you can define three different configurations of a connection element: a joint, a stiff connection or Own input.

Joint

All displacements are prohibited, but all rotations are permitted.

Stiff connection

All degrees of freedom (3 displacements and 3 rotations) are prohibited.

Own input

The user can define their own constraints on the translational and rotational degrees of freedom (DOF). The options are:

- free: no restrictions on the corresponding degrees of freedom.
- fixed: the associated DOF is constrained in both directions.
- fixed with stiffness: the associated DOF is constrained in both directions with additional stiffness.
- one-sided: the associated DOF is constrained in one axis direction (positive or negative, depending on the choice), but free in the opposite direction. The amount of allowable movement in the non-constrained direction can be input.
- one-sided with stiffness: similar to above, but an additional stiffness can be input.
- two-sided: the associated DOF (degree of freedom) is constrained in both directions. However, an allowable clearance can be input for both directions.
- two-sided with stiffness: as above, but with the option of specifying stiffness.

24.2.6.2 Connecting rolling bearing

A rolling bearing can be used to connect two shafts. The only additional information needed, compared to above, is to define the inside and outside shaft for that rolling bearing.

The bearing inner ring is assumed to be fixed on the inside shaft, and the bearing outer ring is assumed to be fixed on the outside shaft. The type of the bearing (fixed bearing, placed on left/right side, etc) defines how the axial forces are transmitted between the shafts through the bearing. Rolling bearings never

constrain y-axis rotation, so different rotation speeds between the connected shafts are permitted.

Due to the fact that both moments and forces need to be transmitted through a connecting element, we strongly suggest that you perform an inner geometry bearing calculation when using rolling bearings as connecting elements.

24.2.7 Cross sections

For information about the significance of cross sections in strength calculation, please refer to the corresponding section (see page III-776) in theCalculating Shafts section (see page III-750).

24.2.7.1 Free cross section

Using free cross sections, you can input the effects of notch, no matter what the actual definition of the shaft geometry.

24.2.7.2 Limited cross section

You should define the restricted cross section as the preferred cross section type in shaft calculations. The notch effect is determined automatically in accordance with the geometry data at this position in this cross section. If you make changes to the shaft geometry, you do not need to modify the cross section manually. The changes are transferred automatically. However, if you are working with restricted cross sections, you must input shaft geometry in detail.

24.2.7.3 Documentation point

Set a documentation point to document the equivalent stress, displacement, rotation, force and torque at a particular position on the shaft in the report.

NOTE:

If you want to document the reaction force/moment at a bearing area, you need to place the documentation point's position slightly to the right of the bearing area's position, and not exactly in the same spot. For more details, please refer to the diagrams of bending report.

24.3 Basic data

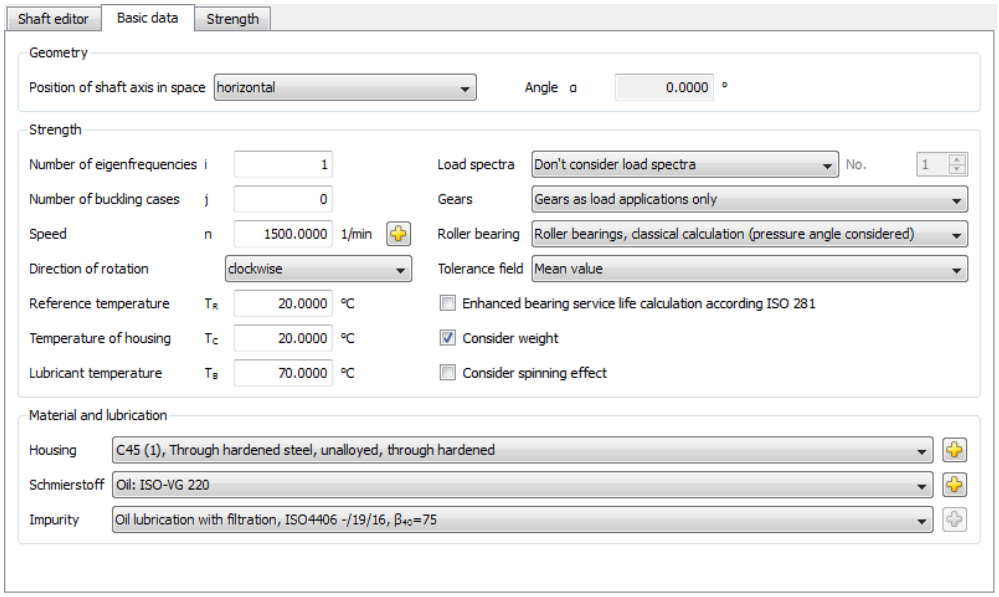


Figure 24.6: `Basic data` input window in `Shaft Calculation` module

In the `Basic data` input window, you can control the basic preliminary settings for shaft calculation. You can enter values for these parameters:

24.3.1 Position of shaft axis in space

You define the position of the shaft axis in space as shown in Figure 24.7.

The consequence of the shaft arrangement in space (horizontally, vertically or in a defined angle to the horizontal) is:

The weight of the shaft (in a horizontal position) is considered a gravitational force in the Z-Y plane when calculating the deflection. However, if the shaft is positioned vertically, the resulting axial force is, for example, included in rolling bearing calculations. If a shaft is positioned at an angle, the corresponding force components are distributed in the Z-Y plane and as an axial force.

Alternatively, you can enter the shaft mass direction vector using the 3-coordinate format.

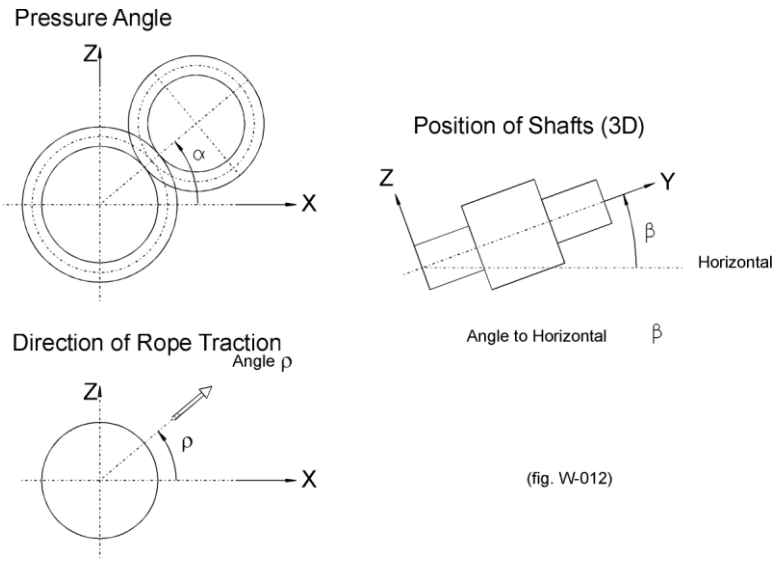


Figure 24.7: Defining the position of the shaft and the position of contact.

24.3.2 Number of eigenfrequencies

In this input field, specify the number of eigenfrequencies (see page III-756) that KISSsoft is to determine.

24.3.3 Number of buckling cases

In this input field, specify the number of buckling cases (see page III-757) that KISSsoft is to determine.

24.3.4 Speed

Enter the speed in revolutions per minute (rpm). Click the  button to open the `Define speed` window. You will see the default values for speed and shaft rotation direction for all the shafts. If you click the checkbox next to the Speed input field, you can overwrite the speed for a particular shaft.

The shaft speed used for the calculation can also be found in the corresponding "Report" -> "Shaft speeds" report.

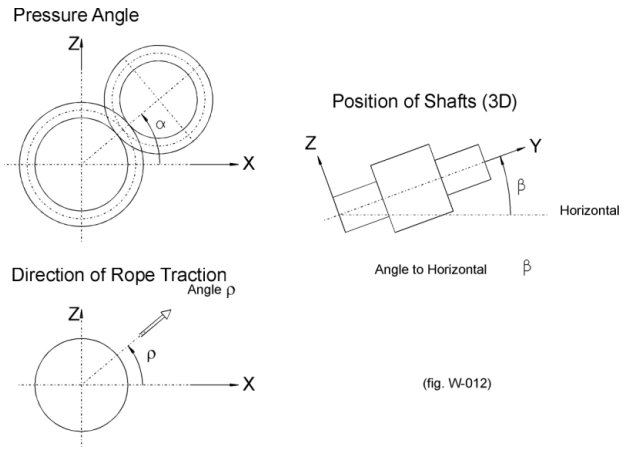
NOTE

If you change the speed, the effective torque and power change accordingly.

24.3.5 Direction of rotation

The shaft axis runs along the positive Y–direction (left to right in the graphical Shaft editor). In the Shaft editor, the Z–axis points vertically upwards and the X–axis points towards the user. A right-hand rotation of the shaft around the positive Y-axis direction is specified as "clockwise".

The next figure shows the direction of these coordinates and the positive direction of forces and moments. Please note that weight has an effect in the negative Z–direction if the shaft is positioned horizontally (see section "Position of shaft axis in space" on page III-731).



In most force elements, the directions of the moments is usually defined by the terms "driving" and"driven". The entry "driving" means either that the shaft drives (an external application) or that the torque acts opposite to the direction of shaft rotation (i.e. the shaft loses power). The entry "driven" means either that the shaft is driven from outside (e.g. by a motor) or that the torque runs in the same direction as the shaft rotation (i.e. the shaft is supplied with power).

24.3.6 Reference temperature

The `Reference temperature` is the temperature specified for the shaft dimensions. This is the temperature on which the drawing data or element testing is based.

24.3.7 Temperature of housing

When used together with the thermal expansion coefficient, the housing temperature defines a strain which changes the distance between the bearing points. In addition, the thermal expansion and Young's modulus of the gear case have an effect on the nominal operating clearance of rolling bearings.

NOTE

If you want to investigate the influence of thermal expansion in greater detail, you must also take the axial stiffness of bearings into account. If the bearings are assumed to be rigid, the load peaks will be too high.

The bearing's outer ring and the housing have the same temperature. The bearing's inner ring and the shaft also have the same temperature.

Thermal reference point housing

According to the previous paragraph (Temperature of housing), the position of the bearing outer ring follows the thermal expansion of the housing (if the bearing is a connecting bearing, this has no effect). The magnitude of the thermal expansion which is applied to the bearing's outer ring is given by ΔL, where

$$\Delta L = (T_c - T_r) \cdot \alpha \cdot (y_b - y_\theta)$$

T_c is the temperature of the housing

T_r is the reference temperature

α is the coefficient of thermal expansion of the housing material

y_b is the global axial coordinates of the bearing (relative to the global frame of reference, not the shaft)

y_θ is the housing temperature thermal reference point used to perform the calculation

For example, if $y_\theta = 0$, this means all thermal expansion is considered relative to the global frame of reference.

24.3.8 Lubricant temperature

The `Lubricant temperature` changes the lubricant's viscosity. This value is used to determine the extended bearing service life (aISO) and the moment of friction.

24.3.9 Load spectra

If the loads defined in the Shaft Editor have been assigned a load spectrum, the deformation can be calculated either for the nominal load or for any element of the load spectrum. To do this, select the `Consider load spectra` option from the `Load spectra` drop-down list. If you only want to take into account one element from the load spectrum, you should select `Consider only one element of the load spectrum`. Enter the appropriate element number in the input field to the right of the drop-down list.

If you select `Consider load spectra`, and define a load spectrum for a force element, the following adjustments are made, if the individual load bins have been defined incorrectly:

- if the frequency H = 0 is set, this is set to the value 10 ^-10
- if the speed factor nfact = 0, this is set to 10^-5 and the torque/load factor is set to 10^-10
- if the torque/load factor is set to Tfact = 0, this is set to 10^-10

24.3.9.1 Load spectrum with negative bins

Load spectra with negative load bins (T < 0 and/or n < 0) are treated as follows:

Coefficient for torque	Coefficient for speed	Shaft direction of rotation	Force element
+	+	-	-
+	-	C	D
-	+	-	D
-	-	C	-

- = unchanged

C = direction of rotation changes clockwise/counterclockwise

D = driving/driven changes

24.3.10 Gears

Select an option from this drop-down list to specify how gears are to be handled in the shaft calculation:

- Gears are only handled as load applications. The masses and stiffness of the gears are not taken into account.
- Consider gears as masses. The gear is handled as a mass in the bending calculation. The mass results from the difference between the reference circle and the outer shaft diameter as well as the gear width (same specific weight as the shaft).
- Consider gears as mass and as stiffness. The gear is handled as part of the shaft contour (for example, pinion shaft).
- Consider gears mounted by interference fit, with stiffness according to ISO 6336-1. The shaft is stiffened at the mid diameter d_m, with $d_m = (d_1+d_2)/2$, d_1 = shaft diameter, d_2 = reference circle of the gear.

NOTE

Gears set on shafts with a firm interference fit always pose the problem of how much they stiffen the shaft. Although KISSsoft cannot solve this problem, it can estimate how much influence the interference fit has: It is sufficient to perform the calculation for `Gears mass` and for `Gear as mass and stiffness` and note the difference in the diagrams of bending. If the difference is small, the interference fit has no influence. However, if the difference is significant, you must enter more precise information. To do this you must integrate a part of the gear in the shaft contour in the graphical shaft input.

If a gear has multiple contacts (for example, a sun wheel in a planetary system), then multiple identical gear elements must be defined at the same position. However, the weight is considered only once.

24.3.11 Rolling bearings

The `Rolling bearings` drop-down list contains four options:

- Classic rolling bearing service life (without taking the contact angle into account). **This method should no longer be used.**
  This option calculates how the bearing reacts at the specified bearing position. However, if you need a more accurate calculation of the way bearings with a contact angle (e.g. taper roller bearings) react, this must be determined at the

place where the direction force intersects with the shaft's symmetry axis. Calculation using the classic method (as described in manufacturers' catalogs).

- Classic rolling bearing service life (ISO 281 and manufacturer's notes). Calculation using the classic method (as described in manufacturers' catalogs). In the calculation for bearings with contact angle (e.g. taper roller bearings) react, the reaction is determined at the position where the direction force intersects with the shaft's symmetry axis.
  In this way, the interdependency between the axial and radial forces, such as exists in taper roller bearings, is included in the calculation. Rolling bearings primarily place constraints on the degree of freedom of movement found in displacement and/or rotation, which is why they are modeled in this way when you select this option. You can input displacement or torsional stiffness (if no values are input, the bearing is assumed to have infinite stiffness). This means the calculation is not affected by the type or size of the bearing.
- Rolling bearing stiffness from inner geometry; classical rolling bearing service life. With this option, the shaft's diagrams of bending are affected by the finite bearing stiffness which is calculated based on the bearing's geometry. However, the service life is calculated based on the forces, according to the manufacturer catalog (i.e. tilting moments are ignored in the service life calculation).
- Rolling bearing service life from internal geometry (ISO/TR 16281). Both the shaft diagrams of bending and the service life of the bearing are based on the inner geometry of the bearings according to ISO/TR 16281.

You will find more detailed information in the description of the bearing calculation (see page III-792).

24.3.12 Tolerance field

The definition of the bearing clearance class does not yet provide a definitive statement about bearing clearance because only a range of values has been defined for the bearing clearance class. The `Minimum` and `Maximum` options define the upper and lower limits of the range, whereas the `Mean value` is the arithmetical average of the `Maximum` and `Minimum` for (radial) bearing clearance.

The operating bearing clearance is defined using the selected bearing clearance class (e.g. "C0"), the selected tolerance (e.g. "mean value") and the working conditions, i.e. speed and temperature. For every rolling bearing, the calculation of operating clearance is described below. Starting from Fig. 1, the following variables are introduced in the calculation:

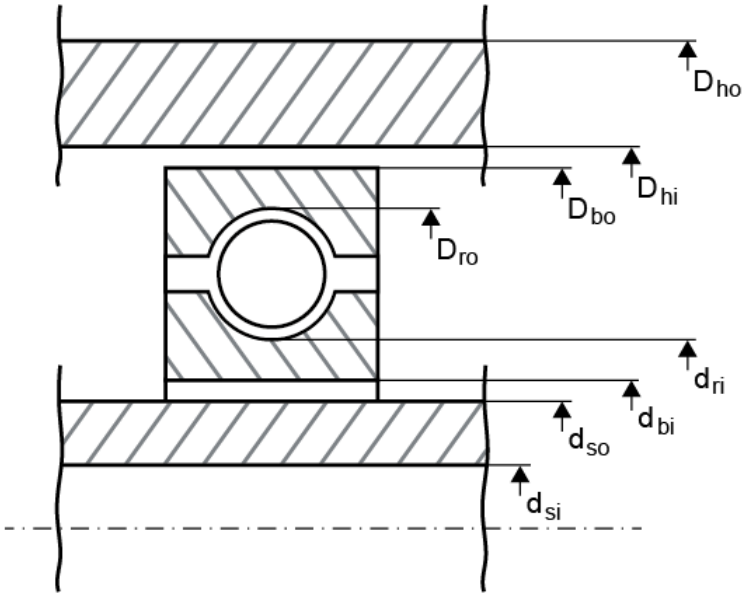


Figure 1: Diameters used for the calculation of bearing clearance

- d_{si}, d_{so} Inside and outside diameter of the shaft.
- D_{hi}, D_{ho} Inside and outside diameter of the hub. If the bearing is a connecting element, then these represent the inside and outside diameter of the external shaft. For simplicity's sake, the term "housing" is used here to mean either the housing or the external shaft (if present).
- d_{bi}, d_{bo} Inside and outside diameter of the bearing, and d_{ri}, d_{ro} for the diameter of the inner or outer race.

All diameter values represent actual diameters, that is taking the allowance of each part into account. The calculation steps are as follows:

- The ring race allowance is taken from the corresponding table (e.g. for tolerance "PN"), for the inner ring AB_i and the outer ring AB_o.
- The allowance for the shaft A_w and housing A_n are calculated from the user-defined data (e.g. "k6").
- The interference is calculated on the inner ring U_{wi} and on the outer ring U_{wo}.

$$U_{wi} = A_w - AB_i$$
$$U_{wo} = AB_o - A_n$$

- According to DIN 7190, the interference is reduced by the value 0.8*(R_{zA} + R_{zB}). In this case, R_{zA} and R_{zB} are the surface roughness of the contact bodies (A: Rolling bearing ring, B: shaft/hub). It is assumed that the roughness of the rolling bearing rings is much less than the roughness of the shaft/hub. For this reason, the roughness of the rolling bearing rings is not taken into account (R_{zA} = 0).

$$U_{wi} = A_w - AB_i - 0.8R_{zs}$$
$$U_{wo} = AB_o - A_n - 0.8R_{zh}$$

- However, the effect of temperature is taken into account,

$$U_{wiT} = U_{wi} + (a_s - a_b)(T_s - T_R)d_{nom}$$
$$U_{woT} = U_{wo} + (-a_h + a_b)(T_h - T_R)D_{nom}$$
$$d_{soT} = d_{so}[1 + a_s(T_s - T_R)]$$
$$d_{biT} = d_{bi}[1 + a_b(T_s - T_R)]$$
$$D_{hiT} = D_{hi}[1 + a_h(T_h - T_R)]$$
$$D_{boT} = D_{bo}[1 + a_b(T_h - T_R)]$$

  where $\alpha_s, \alpha_h, \alpha_b$ is the thermal expansion coefficient of the shaft, housing and bearing, T_s, T_h, T_R are the shaft, housing and reference temperatures and d_{nom}, D_{nom} is the reference diameter of the bearing as defined in the catalog.

- An interference fit calculation, which also takes into account the operating speed, is performed for the inner ring, if condition A applies, and for the outer ring, if condition B applies.

$$d_{soT} + U_{wiT} > d_{biT}(A)$$
$$D_{boT} + U_{woT} > D_{hiT}(B)$$

  The pressure generated in the interference fit changes the operating diameter of the bearing races, and therefore also changes the ΔP_d of the nominal bearing clearance.

NOTE

The selection you make in the Tolerance field has no effect on the general behavior of the bearing.

24.3.13 Modified service life according ISO 281

Click on this checkbox to include the lubricant state in the bearing life calculation. However, to achieve an accurate result, you must first have set the parameters for the `Lubricant` and `Impurity` drop-down lists, and entered a value in the `Lubricant temperature` input field. After the calculation is complete, you see a value for the modified service life L_{mnh} in the `Results` window and/or in the report.

24.3.14 Consider weight

Click this checkbox to include the shaft's dead weight in the section dimension calculation. Depending on the orientation of the shaft arrangement (see section "Position of shaft axis in space" on page III-731), you will see additional axial and shear forces which may have an influence on the diagrams of bending and/or axial displacement.

NOTE

In a global coordinates system, gravitational forces act on the shafts in the negative, Z-direction.

24.3.15 Consider gyroscopic effect

Click this checkbox to include the property of rotating shafts that have weights attached to one end to rotate either in the same (or opposite direction) around the longitudinal axis. Whereas, in situations that are not technically critical, the eigenfrequency sinks when the speed increases in a counter direction, the eigenfrequency increases when the speed is in the same direction. The number of eigenfrequencies that appear here is double the number that appear when the effect of spinning is not taken into account.

24.3.16 Housing material

The housing material value is only used to calculate the thermal expansion of the housing. The materials available for housings are identical to those used for shafts.

24.3.17 Lubrication

Your choice of lubricant only affects the bearing life calculation. Click the  button for your `Own input` for the lubricant.

24.3.18 Impurity

As defined in ISO 281, the impurity coefficient e_C depends on the type of oil filter, the bearing size, and the viscosity of the lubricant. This value varies within the range 0 (high level of impurity) ≤ eC ≤ 1 (ideal). Select the `Own input` option and then click the  button to specify your own e_C values.

Click on the `Enter impurity in each bearing separately` checkbox under `Calculation/Settings/Rolling bearings` to specify whether the value for impurity is to be applied globally in the basic data (for every bearing) or individually, for each bearing.

NOTE

Click the  button to enter your own values. You can define new values for the `Housing` and `Lubricant` that are based on existing data. However, these values are not stored permanently in the database.

24.4 Module specific settings

24.4.1 Non-linear shaft

Click this option to perform a calculation using geometric non-linear beam elements. Due to the planet shaft deflection, the results here also show a displacement in the axial direction because the arc length remains constant. In most situations where shafts are used, this non-linear model is irrelevant.

EXAMPLE

A shaft, which is fixed to its mounting on both sides, is subjected to centrical force. The linear beam model does not allow for an elongation of the beam because it ignores axial displacement during shear and moment loads. If you click on the `Non-linear shaft` field, you can select a calculation method that takes into account the bending effect on the shaft and therefore the elongation of the beam. This results in axial forces.

24.4.2 Take into account deformation due to shearing and shear correction coefficient

If this checkbox has not been selected, the shaft is modeled to be infinitely stiff. In this case, shearing forces have no effect on the diagrams of bending. However, if you do want to include deformation due to shearing, you can specify your own shear correction coefficient κ:

$$A' = A \cdot \frac{1}{\kappa} \tag{24.1}$$

where

A’ shear section

A cross-sectional area

The shear correction coefficient $\kappa \geq 1$ includes the irregular distribution of stress across the cross section and applies to the entire shaft system. $\kappa = 1.1$ applies for circular-shaped cross sections, and $\kappa = 1.2$ applies for rectangular-shaped cross sections.

NOTE

Note the definition of the shear correction coefficient used in KISSsoft, as shown in the previous equation. Some sources also use the reciprocal value for the formula symbol.

24.4.3 Activating offset of load application center point

This enables the gear load elements to define their load application center point offsets, as described in the corresponding chapter of the manual.

24.4.4 Using the 2013 solver

The new solver is used by default for shaft calculations. However, you can use the previous "2013 solver" instead. The new solver is more stable, which is why we recommend it.

24.4.5 Saving temporary results in CSV format with .tmp file extension

If the checkbox is selected, the results of the diagrams of bending are saved in temporary files (in the TMPDIR). The naming convention is W010-H3_bin_x.tmp, where "x" is the load spectrum's number. For example, for a load spectrum with 3 levels, the files W010-H3_bin_1.tmp, W010-H3_bin_2.tmp and W010-H3_bin_3.tmp are created.

1. For rolling bearings
    a. General results of the rolling bearing (displacement, tilting, reaction force)
    b. Results for each rolling body
    c. Results for each disc, if a roller bearing is involved
    d. The stiffness matrix

2. For shafts
    a. Diagrams of bending data

24.4.6 Standard radius at shoulders

To calculate the effect of notch on shoulders, you require a radius. This can be input as a sub-element. If no radius has been defined, the system uses the standard radius defined for calculating the effect of notch.

Generally, we recommend you define radii for each shoulder.

24.4.7 Node density

The user can influence how many nodes are used to calculate a beam. If you are performing a linear calculation, this has no effect on the result, apart from line moments which are distributed across the existing nodes. The beam elements supply the exact solution in the linear model independently of the length.

Reasons for influencing the density of nodes are, on one hand to speed up calculations (for example, in series calculations in KISSsys) and, on the other hand, to ensure the accuracy of the display of the diagrams of bending and the corresponding report.

The density of the nodes affects the accuracy of non-linear beam elements. For this reason, the maximum distance between two nodes for non-linear calculations, when compared with a linear calculation, is halved, no matter what value is predefined.

24.4.8 Iterative calculation of load distribution

If this selection box is enabled then the load distribution is calculated iteratively for the selected gear in the "Tooth trace modification" tab. The initial gear is replaced by a particular number of identical gears. The number of gears is set in the "Number of slices" field. The load on each replacement gear is set according to the current load distribution, and the load on each gear is adjusted iteratively until the root mean square (RMS) of the error in the line load difference is less than 1% between two sequential calculations.

You will find details of how KHβ is calculated in "Tooth trace modification" (see Chapter III-73).

Note: In the case of bevel gears, the selection box must be selected, so that the effect of the changeable operating pitch circle of the gear can be taken into account. Otherwise, the bevel gear is handled as a cylindrical gear whose pitch circle dw equals the pitch circle in the middle section.

24.4.9 Input different load cycles for bending and torsion (for finite life calculations)

Every time a shaft rotates, the bending load cycle changes. For this reason, the number of bending load cycles is calculated using the service life and the speed. The number of torsional load cycles is often very much lower, because not every rotation causes a torsional load cycle. For example, a gear unit may be switched on in the morning and run throughout the day with a constant torque, resulting in exactly one torsional load cycle per day. In contrast, a shaft running at 1000 rpm for 8 hours would be subject to 8000 bending load cycles in the same space of time. As a consequence, in this example, the ratio between the number of bending load cycles and torsion would be 8000 : 1. You can enter this ratio here.

24.4.10 Save user-defined rolling bearing in calculation file

If this flag is enabled, the rolling bearings used in the current calculation file are saved in the file, like the usual input parameters. This means that the calculation file can be opened on a different computer to run the calculation, without having to add data to the bearings database.

24.4.11 Entering an impurity separately in each bearing

If this checkbox is enabled, and `Modified rating life according to ISO 281` has been selected in the Basic data tab, you can enter a value for impurity for each rolling bearing, in the elements editor (in the `Lubrication` sub-tab). Otherwise, the impurity is defined globally in the Basic data tab and applies to all the rolling bearings.

24.4.12 Axial clearance

This is where you define the axial clearance for rigid fixed bearings. The clearance applies to both directions. As a result, a bearing that is fixed on both sides may deviate either to the right or to the left by this value. However, this clearance value is not used if the bearing stiffness is taken into account by the inner bearing geometry. Axial clearance only applies to rigid rolling bearings. You can either use this clearance value, or enter your own stiffness values for general bearings.

If an axially elastic shaft is mounted on several fixed bearings, for example, two bearings in a face-to-face arrangement, and the shaft is subject to a tension load, relatively high reaction forces are caused in the rolling bearings, which are not present in elastic bearings in real life. You can prevent this by entering a relatively small axial clearance for the bearings.

24.4.13 Failure probability

The failure probability value n is used to calculate the service life of rolling bearings. The default value is 10% but you can overwrite this here. The valid input range is $0.05\% < n < 10\%$.

24.4.14 Required service life

Required service life of rolling bearings. However, this value does not actually affect the rolling bearing calculation. However, if the calculated bearing service life is less than this value, the program displays a warning message. Click the sizing button next to the input field to size the service life of individual shafts. Sizing can only be performed in the fatigue strength range of the S-N curve (Woehler lines).

24.4.15 Maximum service life coefficient

In this input field you define the upper limit for the life modification factor: a_{ISO}:

$$a_{ISO} = a_{ISO} \quad falls \quad a_{ISO} \leq a_{ISO,\max}$$

$$a_{ISO} = a_{ISO,\max} \quad falls \quad a_{ISO} \geq a_{ISO,\max}$$

The default value, as defined in ISO 281-2007, is $a_{ISO,max} = 50$.

24.4.16 Displaying critical bearings

The shaft editor displays critical rolling bearings with colors to identify their service life. The color "orange" is used for critical bearings with a service life which is less than the required service life. The color "deep orange" is used for bearings with a minimum service life which is much less than the required service life. The color "blue" is used for bearings whose service life is longer than the required service life.

The lifetime value used to determine the bearing colors depends on which bearing calculation method has been selected, and whether the user has requested the extended bearing service life to be calculated.

Table 1: Lifetime value used for the bearing colors, based on the calculation settings

	Nominal service life requested	Modified service life requested
Rolling bearings, classic (without contact angle) Rolling bearing, classic (inc. contact angle) Rolling bearing stiffnesses, int. geom.	Lnh	Lnmh
Rolling bearing serv. life acc. to ISO/TS 16281	Lnrh	Lnmrh

24.4.17 Surface roughness of housing

The value of the surface roughness of the gear case is used to calculate the nominal operating clearance for rolling bearings. The pressure is calculated for a housing with an infinitely large external diameter. If different roughnesses are needed for

different bearings, or if you want to define the external diameter, you can specify an additional shaft that is then used for that purpose.

24.4.18 Calculation method for friction

From this list, you can select whether the calculation method described in the SKF catalog 1994, the SKF catalog 2013 or the Schaeffler catalog 2014 (INA, FAG) is to be used to calculate friction. These methods are described in more detail in the Moment of friction (see page III-809) section, in the Rolling Bearings chapter.

24.4.19 Oil level

If you select the calculation method described in the SKF catalog 2013 or the Schaeffler catalog 2014 (INA, FAG) to calculate friction, the oil level has an effect on the moment of friction which is determined by the amount of oil lost in the process. This is described in greater detail in the Moment of friction (see page III-809) section.

You input the oil level relative to the left-hand end of the first shaft (but only if "Oil bath lubrication" has been specified). The position of the shaft is then used to define a separate oil level for each bearing (h and H) which is then taken into account when calculating the loss. The oil level is displayed in the shaft editor, so you can check it.

24.4.20 Type of oil lubrication

The type of oil lubrication used is important if you are using the method described in SKF catalog 2013 to calculate friction. The method differentiates between oil bath and oil injection lubrication. This is described in greater detail in the Moment of friction (see page III-809) section.

24.4.21 Moment of friction, seals

This is where you select which method is to be used to determine the moment of friction for the seals. You can select:

- **SKF main catalog according to selected calculation method**
  - **According to SKF main catalog 4000/IV T DE:1994**
    You will find values from the SKF catalog for the seal types used in your bearings integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of

friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.

– **According to SKF main catalog 10000/1 EN:2013**
You will find values from the SKF catalog for the seal types used in your bearings integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
In KISSsoft, the diameter of the mating surface is calculated with ds = d + (D - d) * 0.2

- **according to ISO/TR 13593:1999 Viton** M_{seal} this diameter is calculated with the formula: M_{seal} = 3.736*10^-3*dsh; M_{seal} in Nm, d_{sh} shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{seal} this diameter is calculated with the formula: M_{seal} = 2.429*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm

24.4.22 Bearing manufacturer
Only bearings made by the selected bearing manufacturers are listed for selection.

24.4.23 Show coordinate system
This option toggles the coordination system in the Shaft editor on and off.

24.4.24 Show automatic dimensioning
This option toggles the mass line in the Shaft editor on and off.

24.4.25 Equivalent stress for sizings
This is the equivalent stress used to size a shaft for strength.

24.4.26 Maximum deflection for sizings
The maximum permitted deflection for sizing a shaft for deflection.

Chapter 25
Calculating Shafts

Once you have finished defining the shafts, either click the Σ button in the tool bar or press `F5` to calculate all the shaft-specific values. The results are shown either as a graphic or as a table of values. For example: click the `Graphics` menu in the menu bar, to display the shaft's diagrams of bending in `Shaft > Displacement` (see Figure 25.1).

Figure 25.1: Opening the Graphic window via the `Graphic menu`

Alternatively, go to the `Report` menu and select the `Diagram of bending` option, to display a list of the calculated values.

KISSsoft - Release 03/2017

KISSsoft-Entwicklungs-Version KISSsoft AG CH-8608 BUBIKON

Datei

Name : Shafts 1
Beschreibung: KISSsoft example
Geändert von: KISSsoft AG am: 15.02.2017 um: 09:38:29

y (mm)
d (mm)
u = [ux, uy, uz] (µm)
r = [rx, ry, rz] (mrad)
Q = [Qx, Qy, Qz] (N)
M = [Mx, My, Mz] (Nm)

Welle Shaft:

y	d	ux	uy	uz	rx	ry	rz	Qx	Qy	Qz	Mx	My	Mz
0.00	40.00	0.82	-89.01	4.88	-0.1754	0.0000	-0.0498	0.07	-0.99	-0.19	-0.03	0.00	-0.01
8.00	40.00	1.22	-89.01	3.48	-0.1754	-0.0294	-0.0498	0.07	-0.99	0.58	-0.03	-146.16	-0.01
16.00	40.00	1.61	-89.01	2.07	-0.1754	-0.1174	-0.0498	0.07	-0.99	1.35	-0.04	-292.33	-0.01
24.00	40.00	2.01	-89.01	0.67	-0.1755	-0.2642	-0.0498	0.07	-0.99	2.12	-0.05	-438.49	-0.01
32.00	40.00	2.41	-89.01	-0.73	-0.1755	-0.4698	-0.0498	0.07	-0.99	2.90	-0.07	-584.65	-0.01
40.00	40.00	2.81	-89.01	-2.14	-0.1755	-0.7340	-0.0498	0.07	-0.99	3.67	-0.10	-730.81	-0.01
47.50	40.00	3.18	-89.01	-3.45	-0.1755	-1.0093	-0.0498	0.07	-0.99	4.39	-0.13	-730.81	-0.01
55.00	40.00	3.56	-89.01	-4.77	-0.1756	-1.2845	-0.0498	0.07	-0.99	5.12	-0.17	-730.81	-0.01
60.00	40.00	3.80	-89.01	-5.65	-0.1756	-1.4680	-0.0498	0.07	-0.99	5.60	-0.19	-730.81	-0.00
65.00	40.00	4.05	-89.01	-6.52	-0.1756	-1.6515	-0.0498	0.07	-0.99	6.08	-0.22	-730.81	-0.00
65.00	40.00	4.05	-89.01	-6.52	-0.1756	-1.6515	-0.0498	3088.44	-3263.68	-4970.96	-0.22	-730.81	-0.00
70.00	40.00	4.47	-89.07	-7.67	-0.1733	-1.8350	-0.0483	3088.44	-3263.68	-4970.48	24.63	-730.81	15.44
75.00	40.00	4.87	-89.14	-8.79	-0.1661	-2.0186	-0.0438	3088.44	-3263.68	-4970.00	49.48	-730.81	30.88
75.40	40.00	4.90	-89.14	-8.88	-0.1654	-2.0332	-0.0433	3088.44	-3263.68	-4969.96	51.47	-730.81	32.12

Figure 25.2: Calculation report for `Diagram of bending`

The sections that follow provide more detailed information about the interim results of the values you are interested in.

25.1 Deflection and Bearing Forces, Distribution and Force of Torque

The stress, displacement and tilting calculation are based on the Finite Difference (new solver) or the Finite Element Method (2013 solver). The program determines the diagrams of bending by automatically splitting the shaft into 50 to 100 sections and by using as many points for the diagrams of bending. Boundary conditions and internal boundary conditions (bearing forces and torques) are found by solving a set of simultaneous equations with the same number of unknown variables.

Elastic bearings are considered by setting stiffness values (displacement and torsional stiffness).

The calculation enables you to:

- Calculate the diagrams of bending, course of transverse force, and torque diagram, in the XY and the ZY plane (the shaft rotational axis is always Y) with or without taking the dead weight into account.
- Calculate the axial force taking into account the mass (depending on the length of the shaft).
- Graphical display of all critical results on screen and as a printout: course of deflection, shearing force, bending moment in different levels, torsional moment and equivalent stress (GEH and SSH).
- Calculate the forces and torques in bearings (and ends of shafts) for an unlimited number and any type of bearing.
- The utilization and damage of a rolling bearing is calculated as follows:

$$usage = \frac{P}{P_{req}} = \left(\frac{L_{req}}{L} \right)^{1/k}$$

$$damage = \frac{L_{req}}{L}$$

In this case, L_{req} is the required rolling bearing service life, P_{ref} is the equivalent load which corresponds to L_{req}, L is the achieved service life and k is a coefficient that depends on the type of rolling bearing ($k = 3$ for ball bearings, $k = 10/3$ for roller bearings).

- Bearing clearance is always considered. If the bearing calculation method according to inner geometry is selected, then the bearing stiffness at the operating point and the static safety are also reported. 2 static safeties - S0w and S0r - are calculated.
  S0w is calculated as
  $$S_{0w} = (p_0 / p_{max})^n$$
  where p_{max} equals the maximum Hertzian pressure on the bearing ring. For ball bearings $p_0 = 4200$ N/mm^2 and n = 3. For roller bearings $p_0 = 4000$ N/mm^2 and n = 2.

  S0r is calculated with the following formula
  $$S_{0r} = C_0 / P_{0r}$$
  where C0 is the static load rating of the bearing, and P0r is the equivalent nominal load (i.e. tilting moments are ignored) which causes the same maximum contact pressure. The same calculations are available for standalone bearing calculations with internal geometry.

- The relative deformation and rotation of the inner ring to the outer ring is calculated and recorded.

  Note: the calculation assumes that the inner ring of the bearing is connected to the shaft. If a hollow shaft is connected to the inside of a rolling bearing, the bearing displacement and rotation are documented with the reversed sign.
- Calculate the inclination of the diagrams of bending in bearings, e.g. when calculating cylindrical roller bearings. The progression of the angle of inclination can also be displayed on screen and printed out.
- The diagrams of bending can be calculated with or without taking shear deformation into account.

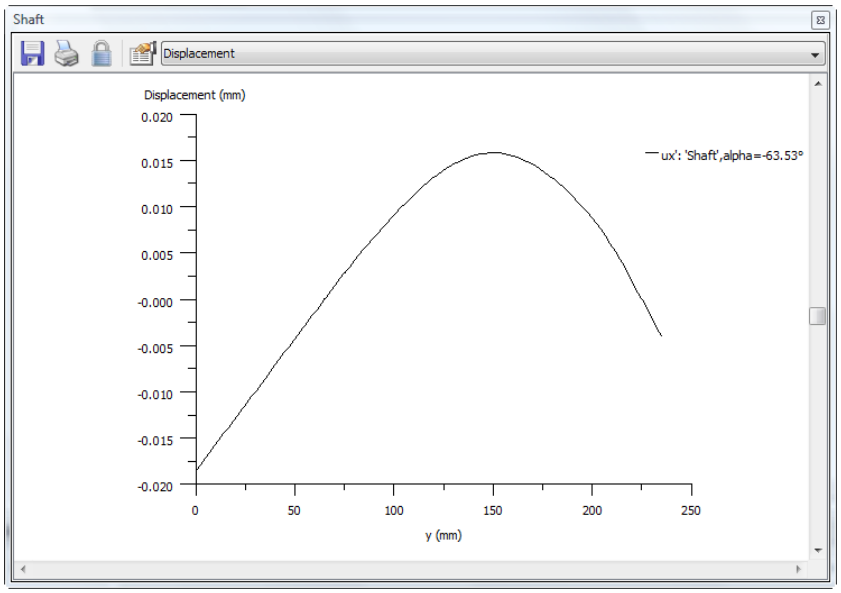


Figure 25.3: `Displacement` graphic displaying the diagrams of bending in the plane $\alpha = -63.53^{o}$

NOTE

Although the data about equivalent stress gives an initial indication of the static strength of a shaft, it cannot be used to calculate fatigue resistance. To do this, you must perform the actual strength calculation. However, this equivalent stress data is useful for beams, because the load they are subjected to is usually only a static load. If the moment of resistance in torsion has not been defined for beams, torsional stress is not included in the equivalent stress calculation. Despite this you can still perform the calculation.

25.1.1 Calculating force on bearings with a contact angle

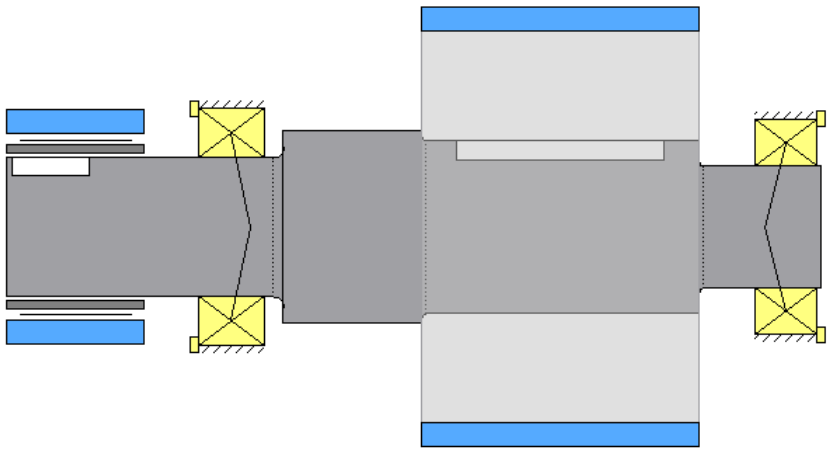


Figure 25.4: Representation of bearings with contact angles

Bearings with contact angle must be handled as a special case when you calculate shafts and bearings. The bearing center used to calculate the bearing reactions is determined at the point at which the compression force line of action intersects with the shaft centerline. In the rolling bearing manufacturers' catalogs, this is described as the axial force resulting from the oblique position of the bearing housing. You can use this to define the data (radial and axial loads) required to calculate the rolling bearing service life expectancy. It is more difficult - and also not clearly documented in the technical literature - to calculate the load progression in the shaft. Here, two modeling types are possible:

In bearings that have a contact angle, the effective bearing force line of action passes through the pressure center point. For this reason, you can calculate the bearing forces because, for calculation purposes, the bearing can be considered as being at the pressure center point. This corresponds to the procedures used to define the rolling bearing load (Variant I).

However, you cannot introduce the bearing force on the shaft outside the bearing width. This is why KISSsoft places the bearing force in the center of the bearing. At the same time, the eccentric application of force creates an additional bending moment which equals the product of the distance of the bearing- and pressure center point, times the radial force (Variant II).

Both variants supply the same progression of bending moment between the pressure centers. There is, however, a difference in the area of the pressure/bearing centers. The shoulder on the right of the picture would be considered as not subject to a force in Variant I (it could, therefore, be ignored), whereas Variant II displays both shearing force and a bending moment.

In real life, the load is not necessarily applied to the center of the bearing but to the entire area of the bearing. Therefore, the bending moment can be placed precisely on the shaft shoulder. However, this then causes a problem in the strength calculation if the load application acts directly on the proof point (i. e. when the proof point lies between the bearing center and the shaft shoulder).

The calculation of the diagrams of bending produces a difference in that, in Variant I, the deflection is zero in the pressure center and, in Variant II, it is at the bearing position. Here, Variant II is certainly more precise, especially when large pressure angles where the pressure center lies outside the bearing width are involved. Only Variant II enables the calculation to include cases in which the pressure center point lies outside the shaft.

As often happens in such cases, the reality lies somewhere between Variant I and II. More precise calculations can only be performed using time-consuming FEM calculations which take into account the properties of the bearing housing. Variant II is more precise and convenient for shaft calculations (because it allows for pressure center points being outside the shaft), which is why this variant has been included in KISSsoft shaft calculation functions. In special cases, when the modeling in Variant II is in question, you can modify the loads in the strength verification on the basis of more accurate observations, when the proof point lies between the bearing center and the pressure center points.

One more point about the shaft strength calculation: Any strength verification based on the nominal stress concept (DIN 743, FKM, ...), has limited validity, in the load application zone (e. g. internal rolling bearing ring on the shaft shoulder) when the local stress distribution does not correspond to the estimated nominal stress. In practice, the results calculated on these points must be interpreted with caution.

In KISSsoft, the additional internal axial force that is present in the case of bearings with a contact angle is calculated as Fr * 0.5/Y, as described in "Die Wälzlagerpraxis" and different bearing product catalogs. [FAG as here, NSK with a factor 0.6 instead of 0.5, SKF for taper roller bearings, as here, and for angular contact ball bearings with a factor 1.14 (Catalog 2004 as a function of Fa/C)]. If factor Y is not present in the bearing database, no additional axial force is taken into consideration. Consequently, the calculation process is the same as the KISSsoft bearing calculation.

25.2 Eigenfrequencies

Click on `Graphics > Shaft > Eigenfrequency` to access the results of eigenfrequency calculation on the modeled shafts system with or without additional masses. The calculation is based on a one-dimensional Finite Element Method (FEM) which takes into account the support and its stiffness.

The calculation enables you to:

- calculate any number of eigenfrequencies[6]
- display natural modes
- You can include the gyroscopic effect of large spinning masses if you click on the `Consider gyroscopic effect` checkbox in the `Basic data` input window. The critical speed (bending mode) is calculated for the forward and backward whirl. In a synchronous parallel run, an unbalance increases the bending oscillations, because the angular speed of the rotating shaft and angular speed of the shaft's peripheral center point are the same. However, the synchronous counter run is, in most cases, not technically important.
- For beam profiles, the critical bending mode (eigenfrequency) is calculated in both main planes.
- Gears can be included automatically and handled like masses. In this situation, KISSsoft takes into account the mass and moment of inertia of the gear (see section "Gears" on page III-736) located on the shaft.

25.2.1 Bending critical speed

The calculation of critical speed takes into account any masses located on the shaft. However, applied forces have no effect on the calculation. For this reason, additional masses must be handled as masses and not as loading forces.

The nodal points of the bending eigenmodes (vibration on plane x-z) are also documented in "Report" -> "Eigenmode nodes".

25.2.2 Torsion critical speed

- Calculation of the critical rotating eigenfrequencies of shafts.
- Calculation of any number of eigenfrequencies.
- Graphical display of natural oscillation.

[6] *Only limited by computing power.*

25.3 Buckling

You use this function to calculate the buckling load of shafts and beams. All peripheral conditions, bearings and effective axial forces (point or line loads) are taken into account in the calculations. Only the axial forces you specify are used to calculate the buckling load. This function calculates the factor by which all these forces have to be multiplied to create a situation under which buckling occurs. This factor therefore corresponds to the safety against buckling.

25.4 Rough sizing of shafts

The rough sizing of shafts is based on equivalent stress. A number of options affect the behavior of this functionality:

1. Equivalent stress: the maximum equivalent stress to which the shaft material is subjected.
2. Change only cylinder diameters: If this option is set to *true*, the length of the cylinders that form the outer contour is retained and only their diameter is changed. If this option is set to *false*, KISSsoft defines both the length and diameter of the cylinders. In this case, the inner contour is deleted.
3. Do not delete cross sections A-A etc.: If this option is set to *false*, the user-defined cross sections for the strength calculation (A-A, B-B etc.) are deleted, and KISSsoft attempts to find the most critical cross sections in the new design.
4. Consider bearings in sizing: If this option is set to *true*, the rolling bearings are sized according to their required service life.
5. Match shaft diameter to bearing bore: If this option is set to *true*, the software rounds up the final cylinder diameters to match the bore (inside diameter) of the bearing.
6. Take bearing type from model: If this option is set to *false*, you can replace the bearings in the model with a specific bearing type as necessary. Otherwise the current bearing is retained.
7. Move bearing if needed: When a bearing is being sized, it may happen that a larger, wider bearing is selected, and this interferes with the neighboring cylinder. If this option is set to *true*, the bearing is moved so that it does not overlap with the cylinder.

Once the calculation has finished, the system displays the old shaft contour so you can compare the old and new data more easily.

25.5 Strength

To access the strength calculation, click the `Strength` tab in the `Shaft calculation` module user interface.

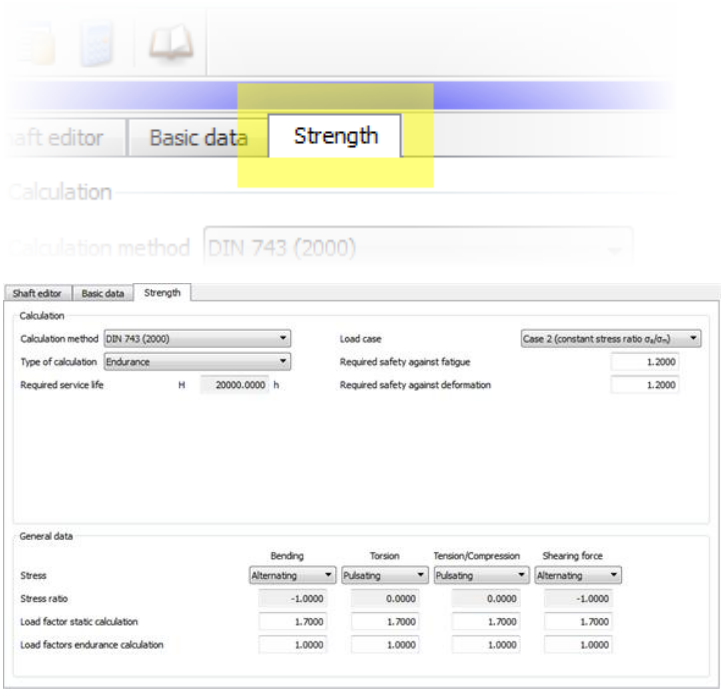


Figure 25.6: `Strength` input window in the `Shaft Analysis` module with the associated tab (above)

In KISSsoft, you can use the following methods to calculate the strength of shafts and axes:

- Hänchen & Decker
- DIN 743:2012-12
  Load capacity of shafts and axes [9] including FVA proposed update concerning fatigue strength and tensile strength []
- FKM Guideline (2012)
  Analytical strength verification of steel, cast iron and aluminum materials in mechanical engineering, 6th Edition 2012
- AGMA 6101-E08/ AGMA 6001-E08
  Design and Selection of Components for Enclosed Gear Drives
- No strength calculation
  In this case, the strength verification is not performed. However, all the other

results (diagrams of bending, equilibrium of forces, bearing reactions etc.) will still be calculated.

A static proof and proof of fatigue strength can be applied in each case. The proof according to FKM, DIN and AGMA can also be performed using a load spectrum.

Some of the shaft-specific data for the strength calculation can be defined in the Elements editor of a particular shaft.

25.5.1 Calculation method

In this drop-down list, you can select one of the calculation guidelines mentioned above. The sections that follow describe the guidelines in greater detail.

25.5.1.1 Hänchen & Decker

Strength calculation according to R. Hänchen and H. K. Decker [42] is an old, but well established method. If insufficient notch factor data is present, the equations produced by TÜV in Munich, Germany, are used: they are derived from known test results.

Material values

As shown in Figures 52, 56 and 60 according to [42] for construction, heat treated and case hardening steels. The empirical formula used is according to Hänchen [42], page 37

$$\sigma_{bw} = 0.4 \cdot \sigma_b$$

You can enter the material data in the database (see page I-128).

Calculation of equivalent stress

In the case of bending and torsion, KISSsoft calculates the equivalent stress value σ_V in accordance with the hypothesis of the largest distortion energy (see [42], section 3.2.5.).

Calculation of safety against fatigue failure

- Maximum load according to [42] equation (4a). Operating factor as defined in [42] Table 1 (page 24).
- Design bending fatigue limit under reversed bending according to [42] Equation (42a).

- Safety margin for fatigue failure according to [42] Equation (46).
- Required safety against fatigue failure in accordance with [42] Figure 156, depending on the frequency of the maximum load.
- The result of the calculation is the ratio of the required safety margin to the calculated safety margin, expressed as a percentage.

Important formulae

A)= Comparative stress (fatigue stress)

$$\sigma_v = \sqrt{\sigma_b{}^2 + 3 \cdot (\alpha_0 \cdot \tau_t)^2} \quad 25.1)$$

$$\sigma_b = \frac{M_b}{W_b} \quad 25.2)$$

$$\tau_t = \frac{M_t}{W_t} \quad 25.3)$$

A1) Comparative stress (strength against overload failure and deformation ($\tau_t = 0$)

$$\sigma_v = \sqrt{(\sigma_z + \sigma_b)^2 + 3 \cdot (\alpha_0 \cdot \tau_q)^2} \quad 25.4)$$

$$\sigma_z = \frac{F_z}{A} \quad 25.5)$$

$$\tau_q = \frac{F_q}{A} \quad 25.6)$$

B) Calculation of the safety against fatigue failure:

$$\sigma_{bWG} = \sigma_{bW} \cdot b_O \cdot \frac{b_d}{b_{kb}} \quad 25.7)$$

$$S_D = \frac{\sigma_{bWG}}{(\sigma_v \cdot f)}$$ 25.8)

α_0	a.0	Stress ratio factor	
A	A	Cross section area	*(cm3)*
b_d	b.d	Thickness number	
b_{kb}	b.kb	Notch factor (bending)	
b_o	b.o	Surface number	
f	f	Total load factor	
F_q	F.q	Shearing force	*(N)*
F_z	F.z	Tension/Compression force	*(N)*
M_b	M.b	Bending moment	*(Nm)*
M_t	M.t	Torque	*(Nm)*
σ_b	s.b	Bending stress	*(N/mm²)*
σ_{bW}	s.bW	Fatigue strength under reversed bending stresses	*(N/mm²)*
σ_{bWG}	s.bWG	Deformation strength under reversed bending stresses	*(N/mm²)*
σ_v	s.v	Equivalent stress	*(N/mm²)*
S_D	S.D	Safety against fatigue failure	
τ_q	t.q	Shear stress (shearing force)	*(N/mm²)*
τ_t	t.t	Torsional stress	*(N/mm²)*
W_b	W.b	Axial moment of resistance	*(cm³)*
W_t	W.t	Polar moment of resistance	*(cm³)*

Stress ratio factor

You will find values for the stress ratio factor in Table 25.2.

Bending	alternating	alternating	static	static	static	static
Torsion	pulsating	alternating	pulsating	alternating	static	static

Structural steel	0.7	0.88	1.45	1.6	1.0	1.0
Case hardening steel	0.77	0.96	1.14	1.6	1.0	1.0
Through hardening steel	0.63	0.79	1.00	1.6	1.0	1.0

Table 25.2: Stress ratio factor α_0 according to Hänchen page 28 [42] or Niemann, I, page 76 [64]

25.5.1.2 DIN 743 (2012)

The German DIN 743 standard [9] uses the most up to date information to calculate shafts and includes the following points:

- Consistent distinction between the different load classifications (tension/compression, bending, torsion) and between mean stress and stress amplitude.
- Surface factor: The influence on the strength is documented when using thermal methods (nitriding, case hardening) and mechanical processes (shot peening, rolling).
- Notch factors: Data for construction elements other than the usual notch factors is mentioned in all specialized books. This data, such as relief grooves, interference fit with relief groove or square notches (recesses for a Seeger ring) is widely used nowadays but has, until now only been poorly documented. All notch factors are documented for tension/compression, for bending and for torsion.
- Materials: it includes an extensive list of materials, as well as instructions on how to derive estimated values for undocumented steels.
- Fatigue strength: the calculation of load strength in accordance with the "Miner extended" method is described in Part 4 of the standard.

The critical limitations of the DIN 743 standard are:

- Shearing load (shear forces) is not included. This is not a disadvantage except for shafts with a very short distance between bearings.
- It only applies to steels and operating temperatures between -40°C and +150°C.

- As defined in the standard, the minimum safety margins for deformation and fatigue failure are defined as stated in 1.2. However, these safety factors only cover the lack of precision in the calculation method, and do not cover the problems encountered in load assumptions or the consequences if the material fails. The required safety factors must therefore be checked or agreed by both the customer and contractor.

25.5.1.3 FKM Guideline, Issue 2012

The FKM Guideline (FKM: Forschungskuratorium Maschinenbau e.V., Frankfurt [Board of Research in Mechanical Engineering]) is based on the standards of the former German Democratic Republic ("East Germany" as was), and includes the latest knowledge on materials theory. It will probably form the basis of a new VDI guideline. The FKM Guideline is extensive (running to approximately 175 pages plus 400 pages of commentaries), and includes not only fatigue strength calculations, but also endurance strength calculations and service life calculations, taking into account load spectra. It also provides calculation approaches for special problems such as operating temperatures above 100°C.

The calculation is performed in accordance with the 6th edition (2012) of the FKM Guideline, using the solutions proposed by Haibach.

Fatigue strength

The service strength coefficient $K_{BK,S}$ is determined according to section 2.4 of the guideline. The number of cycles at knee point N_D is 10^6.

$K_{BK,S}$ is greater than 1.0 if the number of load cycles is less than N_D. Above N_D, $K_{BK,S}$ usually equals 1.0.

Normal calculations with a given load (without load spectrum) are referred to as an "individual load". This is calculated in accordance with Section 2.4 of the guideline. For load spectra, three different processes (see section "Calculation type" on page III-765) are available.

25.5.1.4 AGMA 6101-E08/AGMA 6001-E08

AGMA 6101-E08/ 6001-E08 [9] describes how to calculate a closed gear. Calculations are described for shafts, interference fits, keys, bearings, housings and bolts in this AGMA standard.

- It distinguishes between the different load classifications (tension/compression, bending, torsion and shearing) and between mean stress and stress amplitude.

- Notch factors: Notch factors: the few notch factors given are only for bending. The same ones are used for the other loads.
- Materials: it includes an extensive list of materials, as well as instructions on how to derive estimated values for undocumented steels. The permitted values are converted from the core hardness value entered by the user.
- In KISSsoft, load spectra are not taken into consideration when the AGMA method is applied (as it is not described adequately).

The critical limitations of the AGMA standard are:

- Only for cylindrical steel shafts, but could maybe also be used for other materials.
- The only notch types defined in detail are shoulder, circumferential groove and cross hole.
- According to the standard, the set minimum safeties against peak load and fatigue are 1.0. However, these safety factors only cover the lack of precision in the calculation method, and do not cover the problems encountered in load assumptions or the consequences if the material fails. The required safety factors must therefore be checked or agreed by both the customer and contractor.

25.5.2 Calculation type

You can perform a safety analysis using one of these four different methods:

- **Static**. Calculates the safety against yield safety.
- **Endurance limit**. Calculates the safety against the endurance limit (in the horizontal section of the S-N curve (Woehler lines), no load spectrum used)
- **Fatigue strength**. Calculates the safety against fatigue for a given number of cycles. Here, a constant load is used (no load spectra). The calculation method according to AGMA only defines a one-step load spectrum. A one-step load spectrum is handled separately in the FKM method. According to the methods defined in DIN 743 and FKM, the S-N curve (Woehler lines) runs horizontally after it reaches the number of load cycles limit ND (10^6).

- **Miner consistent/elementary/extended**. These methods differ in the way they calculate the inclination of the S-N curve (Woehler lines) above the number of breakpoint cycles.

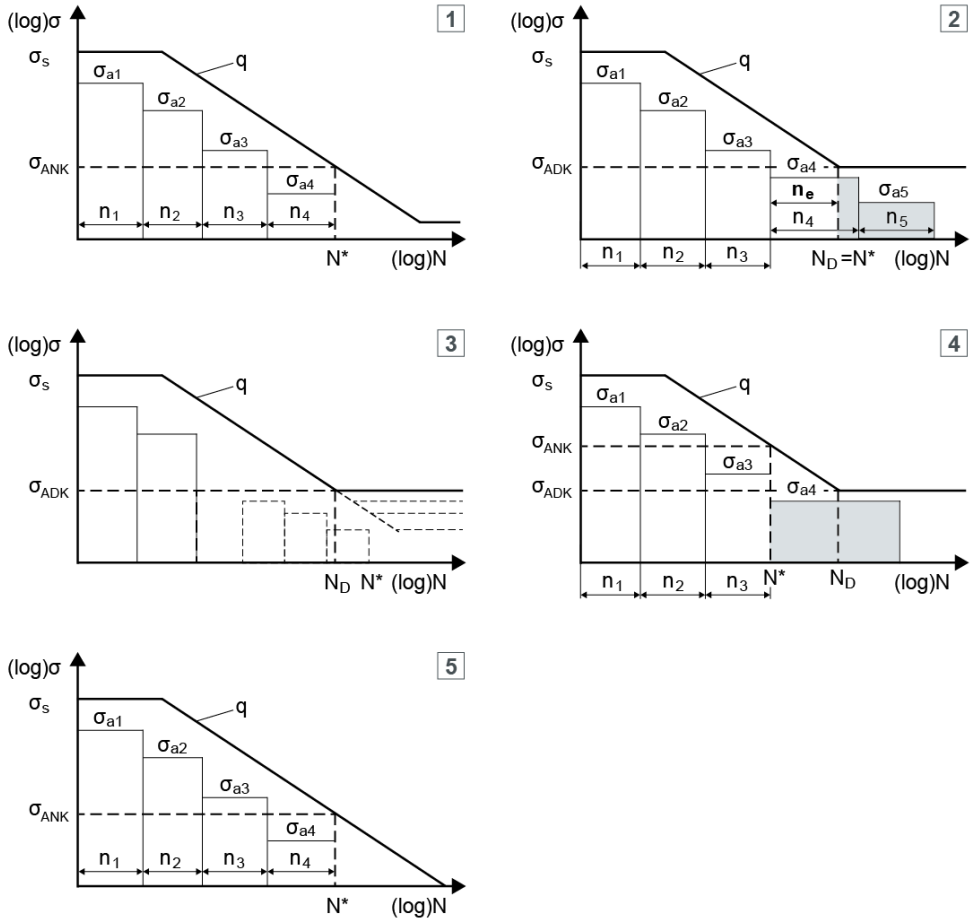


Figure 25.7: Miner hypotheses

Legend:

1) Miner elementary according to FKM guideline

2) Miner extended according to DIN 743-4:2012

3) Miner consequent according to FKM guideline

4) Miner original according to Haibach

5) Miner elementary according to Haibach

The gray fields are the fractions that are ignored.

NOTES

The calculation methods according to Miner are only available if you have selected the `Consider load spectra` option in the `Load spectra` drop-down list in the `Basic data` input window. As you can define load spectra (see section "Define load spectrum" on page II-308) in the KISSsoft database tool, you then only need to select them in the calculation.

25.5.3 Service life

The required service life in number of revolutions is calculated from the required service life in hours.

25.5.4 Strength parameters in accordance with Hänchen and Decker

25.5.4.1 Frequency of load

This value refers to the load value you entered previously (such as torque). If a load applies to the whole service life of the shaft, the frequency is 100%, otherwise it is correspondingly lower.

25.5.4.2 Notch factors

- Thickness number: in accordance with [42], Figure 120.
- Surface number: as stated in [42], Figure 119, Definition of the associated machining process in [42], Table 4.
- The following graphs have been pre-programmed:

Coarsely cut out	Graph with b_o = 0. 50 at 150 kp/mm^2
Milled/finely turned	Graph with b_o = 0. 70 at 150 kp/mm^2
Ground	Graph with b_o = 0. 94 at 150 kp/mm^2
Polished	Graph with b_o = 0. 97 at 150 kp/mm^2

- Shoulder notch effect coefficient during bending according to [42], Figure 131.
- Wheel seat with key: proposed values after consulting with TÜV, Munich. Only very few details are given in [42], section 6.4.
- Interference fit: Proposed values after consulting with TÜV Munich. Details given in [42], section 6.4.
- Bearings are handled as weak interference fits. Only very few details given in [42], section 6.4.
- Shaft-hub connections (multi-wedge toothing): Stress concentration factor and section modulus according to [42], section 8.5. Conversion of the theoretical stress concentration factor into the notch effect coefficient according to [42], section 5.6, Formula (36) and (37b) or (37c) with the radius for the substituting notch according to [42], Figure 112.
- Thread: Theoretical stress concentration factor in accordance with [42], Figure 123. Conversion into notch effect coefficient as shown above.

25.5.4.3 Safety against deformation/fracture

KISSsoft calculates the required safety against fatigue failure, depending on the frequency of the maximum load, using Hänchen's definitions. If the frequency is 100%, the specified safety is 2.0. At 0%, it is 1.0. However, in between these two extremes, the safety does not follow a linear progression.

The required safety against overload failure is 3.5 to 5.0, depending on the type of application or guideline involved. The required safety against deformation (yield point) is usually 2.0 to 3.5.

25.5.5 Strength parameters in accordance with FKM

25.5.5.1 Temperature duration

The FKM guideline takes into account thermal creep in various materials. Constant, high temperatures will reduce the shaft's strength and therefore also reduce its safety.

Part temperatures in the range from -40°C ÷ +500°C are taken into consideration in accordance with the FKM Guideline. For temperatures above 100°C (for fine grain steels above 60 degrees C), temperature factors (for the tensile strength, yield point, and resistance to change) are used to take the reduction in strength into account.

NOTE

You can define the shaft temperature in the `elements editor`: To do this, click on the shaft involved, in the `Elements tree`, and then enter the appropriate value in the `Temperature` field.

25.5.5.2 Protective layer thickness, Aluminum

If you selected aluminum as the shaft's material, enter the value for the thickness of the aluminum oxide layer in this field.

25.5.5.3 Enter safeties

Click on this checkbox to set required safety values on the right-hand side of the Calculation group. Alternatively, click the  button to open the `Define safeties` dialog window, in which you can specify safeties as defined in FKM.

The safety factors for the static strength calculation, j_m (for overload failure) and j_p (for deformation), are determined in accordance with section 1.5 of the Guideline, and the safety factor for fatigue resistance, j_D, is determined in accordance with Part 2.5 of the Guideline. You will find detailed comments in the guideline.

Steel		$j_m = 2.0$	$j_p = 1.5$	$j_F = 1.5$	$j_F = 1.5$
GS, GJS	-not checked	$j_m = 2.8$	$j_p = 2.1$	$j_{G*}j_F = 2.6$	$j_{G*}j_F = 2.6$
	-non-destruction tested	$j_m = 2.5$	$j_p = 1.9$	$j_{G*}j_F = 2.4$	$j_{G*}j_F = 2.4$
GJL, GJM	-not checked	$j_m = 3.3$	$j_p = 2.6$	$j_{G*}j_F = 3.1$	$j_{G*}j_F = 3.1$
	-non-destruction tested	$j_m = 3.0$	$j_p = 2.4$	$j_{G*}j_F = 2.9$	$j_{G*}j_F = 2.9$

j_m, j_p: The values apply for
- severe damage as the result of failure
- high probability of load occurrence

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided the probability of the same load occurring again is low, the safety factors can be reduced by about 10%.

$j_{G*}j_F$: The values apply for
- severe damage as the result of failure

- irregular inspection

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided inspections are carried out regularly, safety factors can be reduced by about 10%.

25.5.5.4 Load case

The load case identifies four hypothetical scenarios for the development of the stress ratio σ_a/σ_m if load increases, starting at the operating point.

25.5.5.5 Estimation of the fatigue strength for surface treated parts (section 5.5)

This calculation should only be used for surface treated rolled steel. The surface treatments include the following treatment methods applied to the materials:

- case-hardened
- nitrided, gas-nitrided, nitrocarburized
- induction hardened
- rolling
- shot peening

These types of treatment can be defined either when you input the material, or when you input the surface factor for the shaft in the elements editor.

This process is based on the concept of a local endurance limit. Two points on the component are observed. The first point is on the component's surface, and the second point is at the transition point between the hard surface layer and the core. The resulting stresses are converted into principal stresses σ1 and σ2. Only the largest principal stress σ1 is used in the subsequent calculation.

You can also input a hardening depth for this calculation in the elements editor. The hardening depth is then used to calculate the distance from the component's surface to the transition point between the hard surface layer and the core.

The Strength tab is where you define whether the constant Kf is to be calculated in accordance with formulae 4.3.2 and 4.3.3 or taken from Table 4.3.1. You also have the option of inputting the core hardness when you specify the material. Alternatively, this can also be estimated from the tensile strength.

This approach is used to calculate the internal stress, which is included in the calculation of mean stress sensitivity. In this case, the rate of utilization for the point on the component's surface is calculated first, followed by the rate of

utilization at the transition point between the hard surface layer and the core. The greater of the two rates of utilization is then used for the proof. Both rates of utilization should be < 1.

The results are only displayed in the report if this calculation method has been selected for rolled steel with the predefined treatment types.

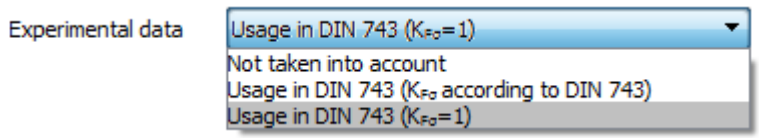
25.5.6 Strength parameters according to DIN

25.5.6.1 Load case

The load case identifies two hypothetical scenarios for the development of the stress ratio σ_a/σ_m if load increases, starting at the operating point.

25.5.6.2 Calculation with experimental data

Use this option to define a Haigh diagram, which has been determined from experimental data. If you input a file name (e.g. WMAT-001.dat) in the **Experimental data** field for module-specific material data as defined in DIN 743, a selection list appears in the **Strength** tab.



- **Not taken into account**: the data is ignored.
- **Use in DIN 743 (KFσ according to DIN 743)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is defined according to DIN 743.
- **Use in DIN 743 (KFσ=1)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is always set to 1.

The method used to define data from the file requires separate instructions, which we will supply upon request. The measured Haigh diagram is not interpreted exactly as described in DIN 743. The overall influence coefficient divides the Haigh diagram into X- and Y-coordinates so that the results are much smaller.

The influence of mean stress as defined in DIN 743 increases as the notches become sharper, and should not decrease. This modification ensures that this influence always increases.

If the comparative medium stress is σmv<0, the line of the Wöhler diagram is extended into the negative area of the Haigh diagram, and the curve is then broken in the negative area by compression yield point σDk. For more information, see also the description in DIN 743-1, p. 21.

25.5.6.3 Safety against fatigue/deformation

In these input fields, you specify the required safeties for endurance/yield. A warning message is displayed if these values drop below the limit you specified for any one cross section.

25.5.7 Strength parameters according to AGMA

25.5.7.1 Coefficients used to determine fatigue strength

The following coefficients are required to calculate the modified fatigue strength σf:

Surface finish coefficient ka

You can set the surface finish for each individual element in a list. The ka coefficient is then determined from the list entry, using the method shown in Figure 4, AGMA 6101-E08.

Size factor kb

The size factor kb is determined on the basis of the shaft diameter, using the method shown in Figure 5, AGMA 6101-E08.

Reliability coefficient kc

In the `Strength` tab, you can define the reliability (default 0.99%). This value is used to determine reliability coefficient kc, using the method shown in Figure 6, AGMA 6101.

Temperature coefficient kd

The coefficient kd=1 is set for temperatures between -30°C and 120°C. The entries for the shaft can be modified if the shaft temperature lies outside this range.

Life factor ke

This coefficient is defined by the stresses σe and σu and the number of load cycles, NL.

Surface processing coefficient kf

For key ways, this coefficient is taken directly from AGMA 2101, Table 2.

For straight-side or normal splines, a Kt coefficient of 2.0 is assumed.

If a combination of a key and a press fit is present, a kf coefficient of 0.33 to 0.4 is specified. In KISSsoft, the default kf coefficient is 0.33.

The notch effect type of the circumferential groove is used for V-notches and threads. It has a fixed ratio of h/r = 20.

The system takes an individual Kt coefficient from the diagrams for the shoulder, circumferential groove and cross hole (Figure 8-10, AGMA 2101) and uses them, together with coefficient q (Figure 7, AGMA 2101) to calculate coefficient kf.

Miscellaneous effects factor kg

This factor can be used to take into account, for example, heat treatments, residual stresses (shot peening, cold rolled, etc.), corrosion, and surface coatings.

The default value set for this factor is kg=1, however, it can be overwritten with the values entered for the shaft.

25.5.7.2 Calculating the Safeties

Load spectra are not taken into consideration when the AGMA method is applied.

To calculate the fatigue safety FSf, the average stress and the amplitude are determined using the von Mises stresses.

These values are later required, together with the modified fatigue strength and the tensile strength, to determine the fatigue safety.

The total stress is determined using the von Mises stresses to calculate peak load safety. The peak load safety coefficient is calculated using the total stress, the tensile strength, a coefficient for the yield point, and a peak load coefficient.

The peak load coefficient can be defined individually in the `Strength` tab for all loads (tension/compression, bending, torsion, thrust). In the KISSsoft system, the default setting for the peak load coefficient is 1.0. As specified in the standard, the setting for this coefficient is 2.0 for cylindrical gears with straight and helical flanks, bevel gears, and double helical gearing. For worm wheels, the setting for this coefficient is 3.0.

The yield point coefficient can be also defined in the `Strength` tab. In the KISSsoft system, the default setting for the yield point coefficient is 0.75. (As specified in the standard, it is 0.66 to 0.8).

The resultant safety factors for fatigue FSf and peak load FSp must be >= 1 according to the standard.

25.5.8 Stress

This is where, in particular, you define how the loads calculated by KISSsoft (e.g. the bending moment) are to be converted into amplitude or means stress. You can select usual loads (alternating, pulsating, static load) from the list. For exceptional cases, select `Own input` from the `Stress` drop-down list and enter the required value in the `Stress ratio` input field. (See next section.) Rotating shafts normally have an alternating bending and a pulsating or static torsion.

25.5.9 Stress ratio

You must also enter a stress ratio because KISSsoft requires this value to split the load on the corresponding cross section into mean load and load amplitude.

Maximum stress per load cycle:		σ_o
Minimum stress per load cycle:		σ_u
Stress ratio		$R = \sigma_u/\sigma_o$
Mean stress:	σ_m	$= (\sigma_o + \sigma_u)/2$
		$= (\sigma_o + R \cdot \sigma_o)/2$
		$= \sigma_o \cdot (1 + R)/2$
Stress amplitude:	σ_a	$= (\sigma_o - \sigma_u)/2$
		$= (\sigma_o - R \cdot \sigma_o)/2$
		$= \sigma_o \cdot (1 - R)/2$

For:

Pure alternating stress	$(\sigma_u = -\sigma_o)$	$R = -1$

Pulsating stress	($\sigma_u = 0$)	$R = 0$
Static stress	($\sigma_u = \sigma_o$)	$R = 1$

Normally valid for rotating shafts or axes:

Bending and shearing force: $R = -1$

Torsion and tension/compression: $R = 0$ (ev. R = 0...1)

NOTE

In contrast to the calculation according to DIN or FKM, where there is a clear differentiation between the mean stress and amplitude stress, when a strength calculation in accordance with Hänchen (see page III-760) is performed, the loads that are entered are converted into a comparative stress that is then compared with the fatigue limit for bending. For this reason, if you select this method, the stress ratio only affects the value of the stress ratio factor α_0.

25.5.10 Load factor for static analysis

The static calculation normally uses the greatest possible load. The maximum load factor covers the difference between the load value you specified and the peak value.

Maximum stress: $\sigma_{max} = \sigma_o \cdot f_{max}$

You can specify individual factors for every type of stress (bending, tension/compression, etc.).

The load factor is not used if the forces or power ratings are specified in free cross sections.

EXAMPLE

Electric motor with a permanent torque 100 Nm, starting torque 180 Nm. When you specify the shaft data, enter 100 Nm and set the maximum load factor to 1.8.

25.5.11 Load factor endurance calculation

If necessary, the mean stresses and the stress amplitudes can be multiplied by a load factor. As the DIN 743 standard does not include this factor, you should generally predefine it as 1.0. Using a factor > 1 is a good idea if you specify the nominal

torque in a shaft calculation without taking into account the increases in torque due to the vibrations caused when the shaft rotates.

The load factor is not used if the forces or power ratings are specified in free cross sections.

The calculation in accordance with Hänchen includes the following information:

Information on the total load factor f according to Hänchen [42], p. 24):

$$f = f_{un} \cdot f_{betr} \cdot f_{leb} \qquad 25.9)$$

f_{un} Uncertainty in maximum load (1.0 or 1.2 to 1.4)

f_{betr} Operational approach (shocks) (1.0 to 3.0)

f_{leb} Importance of part (1.0 or 1.2 to 1.5)

NOTE:

The Hänchen method uses only one load factor, which is the larger of the two values entered for bending and torsion.

25.5.12 Cross sections

Yield safeties and safeties for fatigue failure are evaluated at specific cross sections along a shaft that are defined by you. To define a cross section:

In the Elements tree you will see the Cross section entry at group level (→ see Figure on page III-702). Click the right-hand mouse button on this entry to open a context menu, in which you can select either Free cross section or Limited cross section.

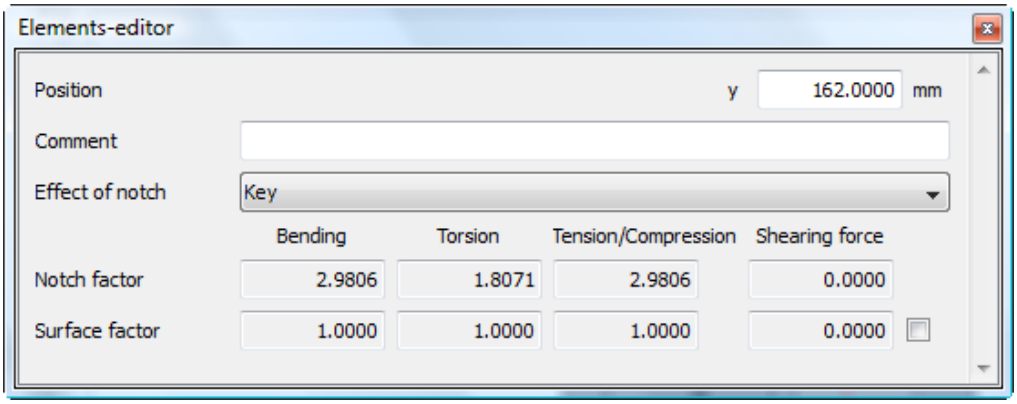


Figure 25.7: Elements editor for setting parameters for Restricted cross section

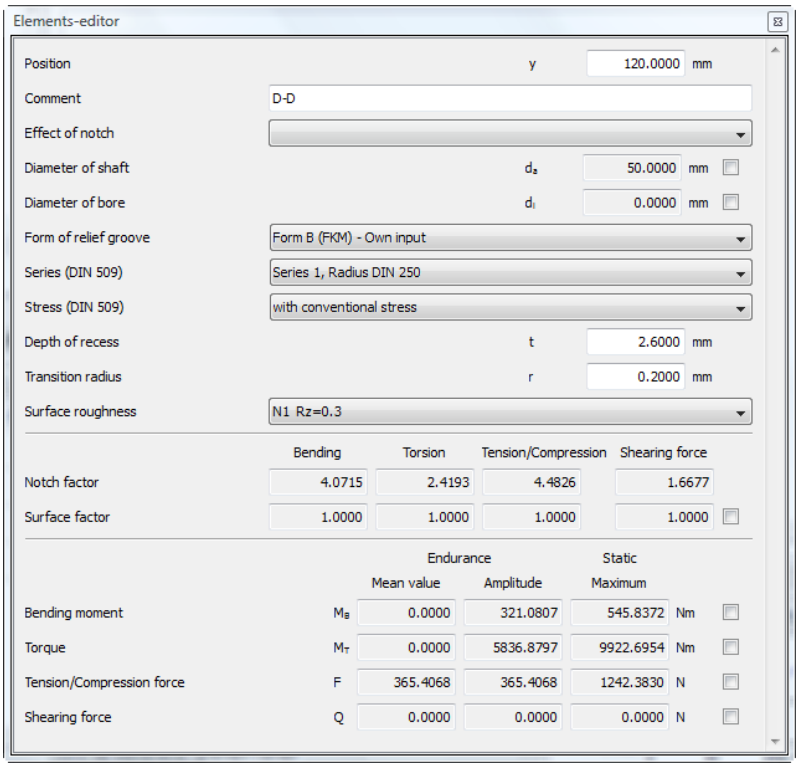


Figure 25.8: Elements editor for setting parameters for Free cross section

25.5.12.1 Surface roughness

If you enter a value for surface roughness as defined in ISO 1302, the corresponding surface roughness, R_Z, is displayed in the selection list. This value, R_Z, is then used in the calculation. In the calculation according to DIN or FKM, the surface roughness has already been included in the notch factor in some cases. In such situations, the surface factor is always 1.0, no matter what value you input as the roughness.

25.5.13 Sizing

You can select the `Size` option in the context menu for the `Cross section` entry in the elements tree, to make it easier for you to define the cross sections that need to be recalculated.

In this sizing, KISSsoft automatically finds cross sections (shaft shoulders, interference fits in bearings, key-grooves and other notch effects) which have been defined in the `graphical shaft input`, and in which a notch effect occurs. It displays the cross sections that have the lowest safety. You must check these cross sections carefully.

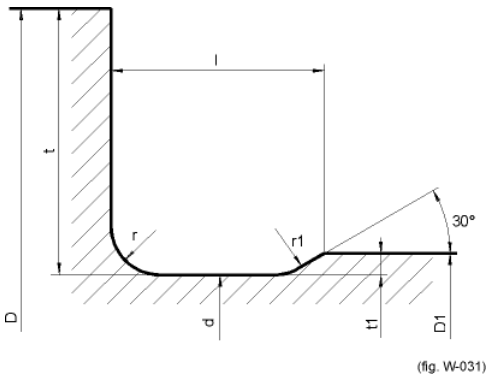
NOTE

Check for further notch effects, which KISSsoft cannot find, such as thread or cross hole.

25.5.14 Cross section types

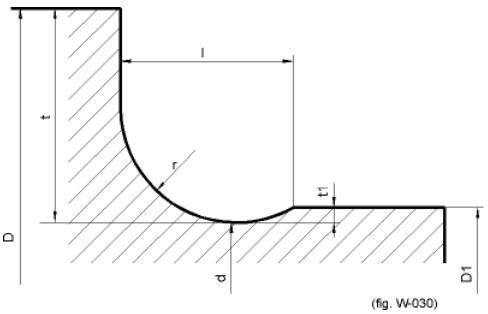
- **Shoulder**

- **Shoulder with relief groove**



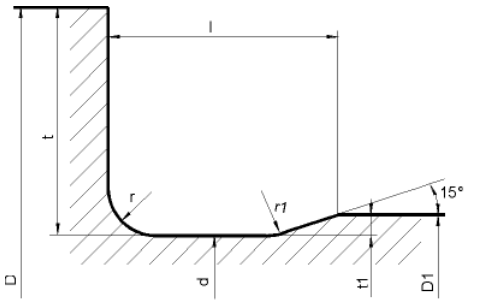
(fig. W-031)

FKM Form B



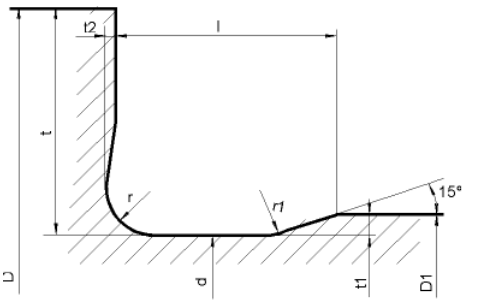
(fig. W-030)

FKM Form D



(fig. W-031-E)

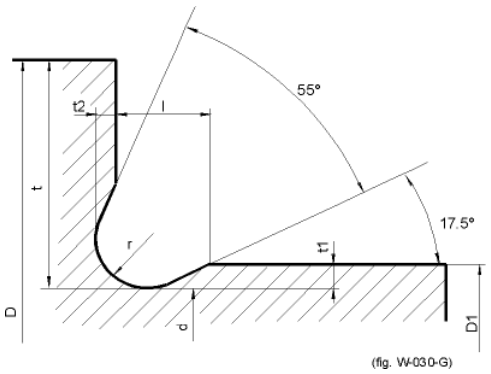
DIN 509 Form E



(fig. W-031-F)

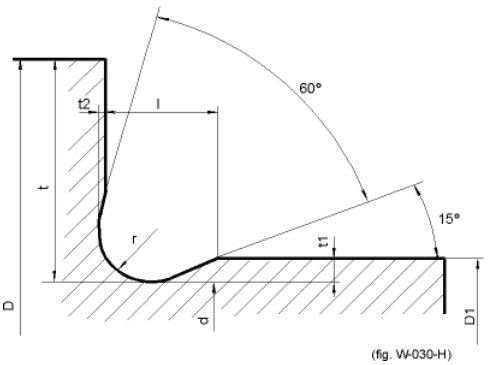
DIN 509 Form F

According to FKM, these shapes are handled like shape B.



(fig. W-030-G)

DIN 509 Form G

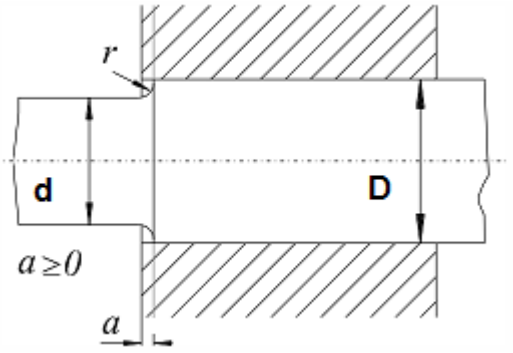


(fig. W-030-H)

DIN 509 Form H

According to FKM, these shapes are handled like shape D.

Shoulder with interference fit



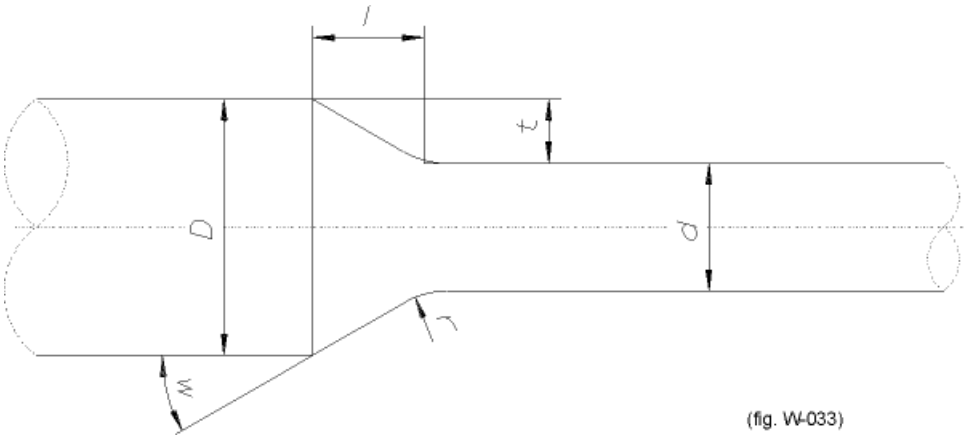
With Hänchen+Decker and AGMA 2101: not possible

In DIN 743: The notch factor will be calculated like a shoulder, but with the relationship d/(1.1*D). The maximum transmission for D/d ~ 1.1 and for r/(D/d) ~2. This condition is only applied if D/d >= 1.1, otherwise the notch effect of the shoulder is used.

In the FKM guideline: The notch effect coefficient is determined for the fit H7/n6. The notch effect coefficient is also calculated for a shoulder and then used, in the worst case, in subsequent calculations.

Notch factors are documented in the different methods. The notch factors calculated in FKM are usually significantly larger than in DIN.

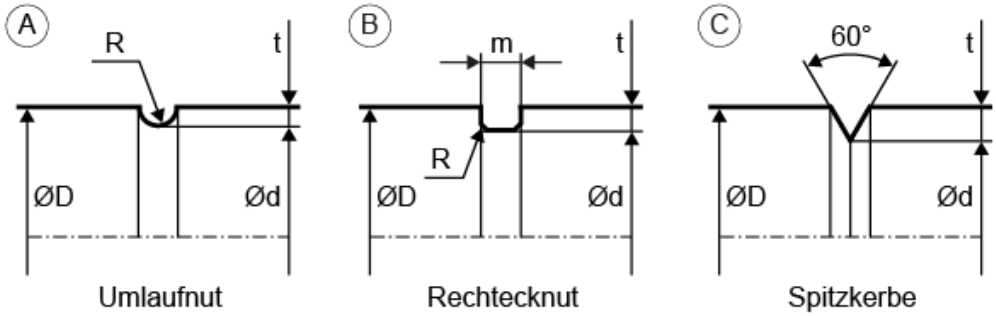
Shoulder with conical transition



(fig. W-033)

Shaft grooves

With the following variants:



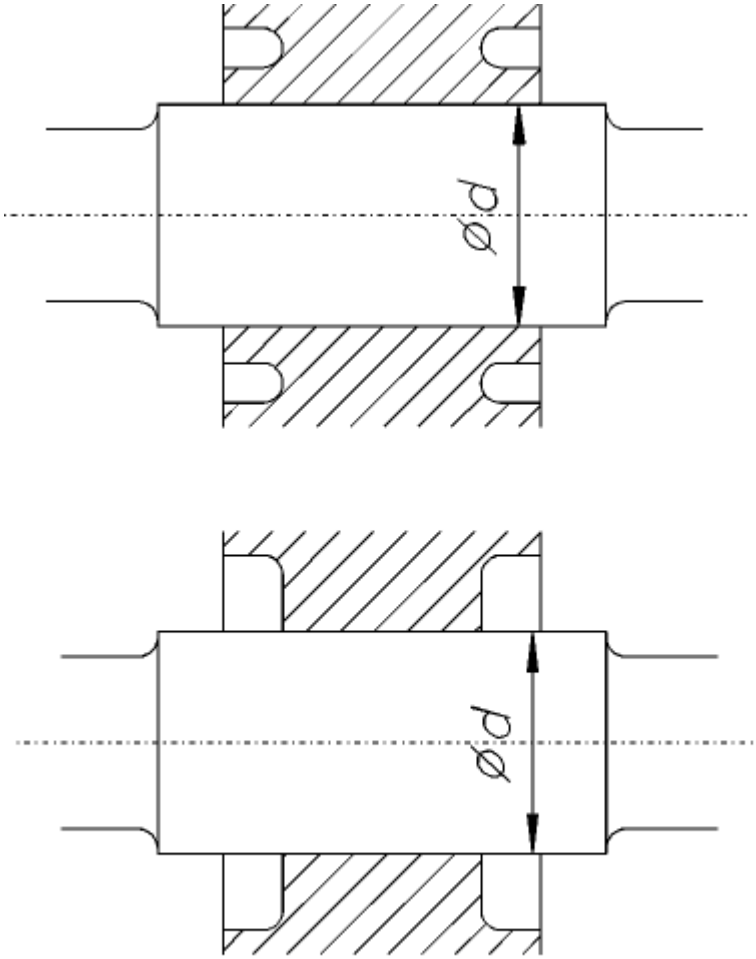
Thread

Notch factors for threads are not described as a separate topic in the specialist literature. For this reason, notch factors for threads are handled like those for V-notches.

Interference fit

Interference fit (Firm interference fit, Slight interference fit, Interference fit with relief grooves).

Only notch factors for the "fixed interference fit" are defined in DIN 743, which is why the FKM guideline is used to define the factors for the other types of interference fit.



(fig. W-021)

Above: Interference fit with relief grooves.
Below: Interference fit with end relief.

Key

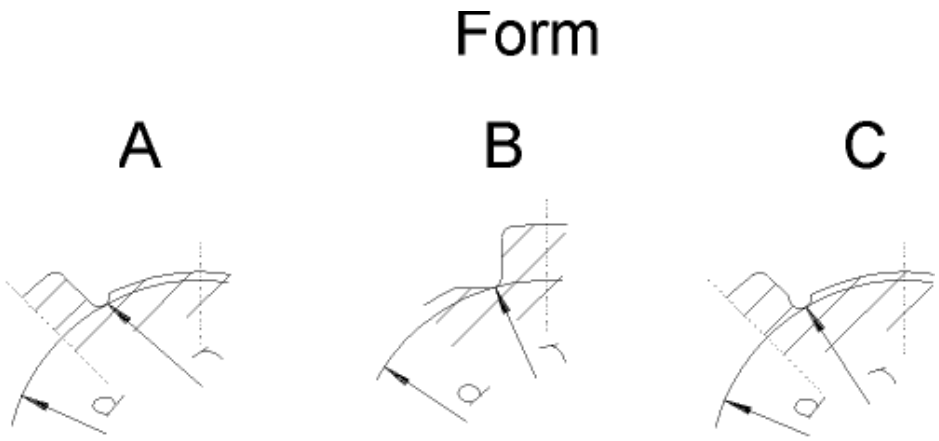
In every method, the moment of resistance for bending is determined from shaft diameter d. As described by Hänchen, the moment of resistance for torsion is calculated from the incorporated circle d - t. According to FKM, DIN and AGMA, it is calculated from the outer shaft diameter d.

Notch factors are documented in the different methods. However, Hänchen provides very little information about this that can be used to extrapolate values for steel of higher strength (with the appropriate comment about the calculation). In contrast, these values are well documented in the DIN standard and the FKM Guideline (in the tables for Interference fit with key). Two different production

methods are described for keys in AGMA 6101 (side milling cutter or keyway cutter). This standard also distinguishes between 2 different hardness ranges.

The program includes tables for cross sections with keys. The data is imported from a data file which includes the DIN 6885.1 (corresponds to ISO/R 773), DIN 6885.2 and DIN 6885.3 standards. You can also specify other standards.

- **Groove meshing and spline shaft**



(fig. W-032)

Shape of the spline shaft (straight-sided spine)

To calculate groove meshings or spline shafts you must first enter tip and root diameter data. All other values are used purely for documentation purposes.

To calculate the section modulus:

In Hänchen+Decker and FKM:	From the mean value ($d_a/2 + d_f/2$)
In DIN 743 and AGMA 6101:	From the root circle

Notch factors are documented in the different methods.

An exception occurs in the calculation according to FKM, where the root diameter of spline shafts (in this case: d) is used to calculate the notch radius.

- **Cross hole**

- **Smooth shaft**
  If you select `Smooth shaft` the notch factor is set to 1. You should select this for the cross section with the maximum stress.

- **Define your own input notch factors** (see page III-776)

- **Intersecting notch effects** (see page III-861)

25.5.15 General entries

25.5.15.1 Thickness factors from the shaft diameter

You can derive material values that depend on the diameter either from the effective shaft diameter (d or D) or from the thickness of the raw material. The choice based on the effective shaft diameter gives more reliable safety results, but can only be used if the shaft is through hardened before it is turned.

However, if you select `Pre-turned to actual diameter (for shoulders K1 from d)`, the material data for shoulders is derived from the smaller diameter (d). If you select `Pre-turned to actual diameter`, it is derived from the larger diameter (D). Although deriving these values from D gives slightly lower strength values, the results are therefore somewhat safer. The standard does not comment on this.

25.5.16 Thermally safe operating speed

The definition of the thermally safe operating speed is described in DIN 732 [8]. The calculation of the thermally safe operating speed is based on the thermal balance in the bearing. The thermally safe operating speed is derived from the thermal reference speed, using the speed ratio. The result of this calculation is the speed that will be reached by the bearing running at the permitted temperature in an actual situation. In order to define the thermally safe operating speed, you must first define the thermal reference speed for each case.

The thermal reference speed is defined in DIN ISO 15312 [7]. The thermal reference speed is the bearing-specific speed calculated under a given set of nominal operating conditions such that equilibrium is achieved between heat development (friction) and heat dissipation (through bearing contact and lubricant).

You can enter the values for the calculation in the special 'Thermally safe operating speed' tab and in the relevant rolling bearing in the elements editor.

The calculation is also available for use in the rolling bearing calculation module [W050], where the calculation process and the values you enter are described in more detail in Thermally permissible service speed (see page III-804).

25.6 Tooth trace modification

For various purposes, it is important that you know how much a specific point in the shaft cross section moves in a particular direction due to elastic deformation (bending and torsion). An example of this is calculating the gap between the two halves of a coupling that are mounted on each end of the same shaft. In this situation, the displacement of a point on the shaft cross section is calculated in the axial direction.

The most important application of this calculation is to determine shaft deformation in the meshing area. The deformation for the pitch point is calculated along the facewidth. In this situation, the displacement of a point on the shaft cross section due to bending and torsion is calculated only in the direction of the normal to the flank. A displacement parallel to the flank only results in a very minimal change in sliding velocity and can therefore be ignored.

In the "Tooth trace modification" tab, you can directly select the toothing currently present on the shaft. The data you have already input is used to define the necessary defaults for the calculation (Facewidth from and to, Coordinates meshing point, Direction of the normal to the tooth flank in the pitch point) which are displayed in the user interface. Therefore, assuming that the counter gear has infinite stiffness, the progress of the meshing point displacement due to deformation can be determined along the facewidth.

NOTE:

during the tooth trace modification calculation, any gear load application offset for the gear selected for the particular calculation (Calculation A or B) is temporarily deactivated. This means the gear load application offset of gear A is disabled when Calculation A is performed, but is re-enabled when Calculation B is performed.

To display this deformation, also called "gaping", click `Graphics -> Tooth trace modification -> Deformation`.

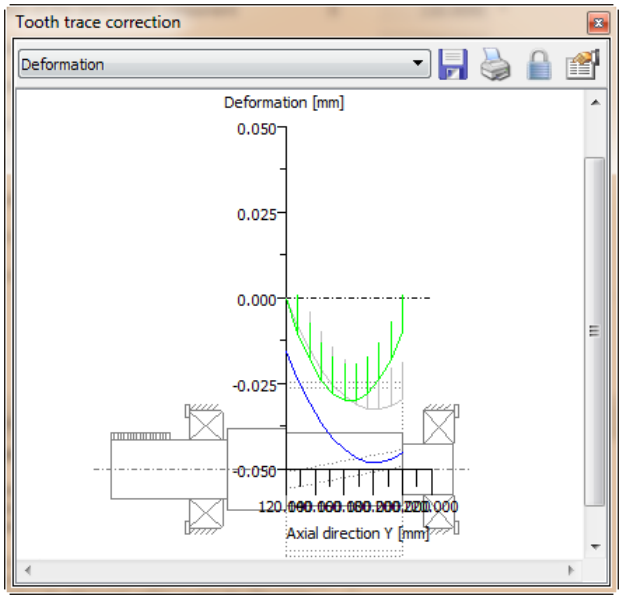


Figure 25.9:`Diagrams for tooth trace modification and> deformation`

This shows the deformation in the pitch point. It also shows a proposed value for an optimum tooth trace modification. This modification would achieve a homogeneous load distribution along the facewidth.

You can input the tooth contact stiffness cγ in another input field. For steel gears, the tooth contact stiffness per mm facewidth is approximately 20 N/mm/°. The values of cγ are calculated precisely and documented in the cylindrical gear calculation. This stiffness can then be used to calculate the load distribution along the facewidth. Click `Graphics -> Tooth trace modification -> Load distribution` to see the result.

Calculate the load distribution coefficient KHβ for gear calculations

The results window also shows the load distribution coefficient KHβ, calculated according to ISO 6336, with equation KHβ = wmax/wm from the average line load (wm) and the maximum line load (wmax). This calculation enables the face load factor to be estimated with significantly more accuracy, similar to Method B in ISO 6336. The procedure is basically similar to Annex E of ISO 6336. However, you must be aware that the shaft of the counter gear used here is assumed to have infinite stiffness. This is permitted if the shaft of the counter gear has much greater

stiffness. Manufacturing allowances are also only included if, for example, they have been defined by inputting an angular deviation of the shaft (bearing displacement) as part of the shaft data.

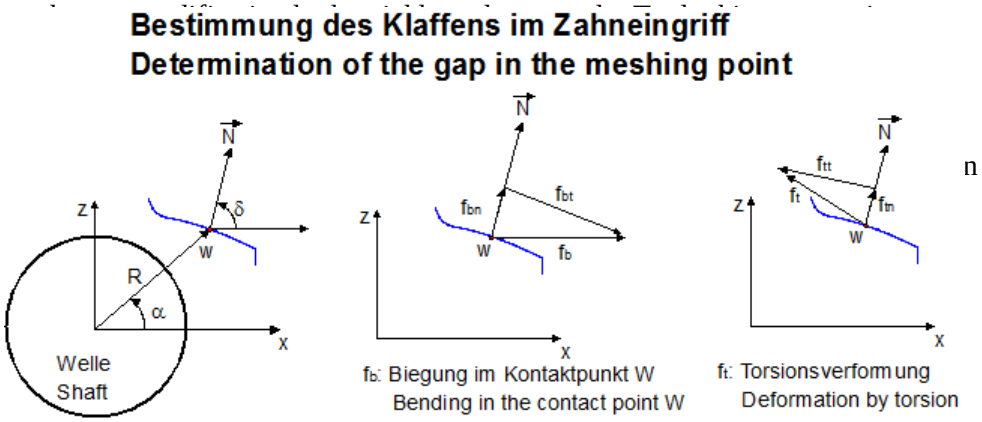
The normalized displacement of the gear body determined from an FE (Finite Element) calculation can also be taken into account as a displacement or stiffness matrix. To do this, select the "Take additional displacement matrix into account" option in the cylindrical gear force element. You will find the deviation.dat file, which gives an example of a displacement matrix, in the dat. directory. The stiffness matrix can be calculated in the gear body deformation module (K16), or imported from another FE program.

NOTE:

If KHβ is to be determined while taking into account the deformation of the two shafts: The deformation components of two shafts can be combined in the cylindrical gear calculation in the "Contact analysis" tab.

Sizing the tooth trace modification

This calculation module has been designed to enable you to define the best possible

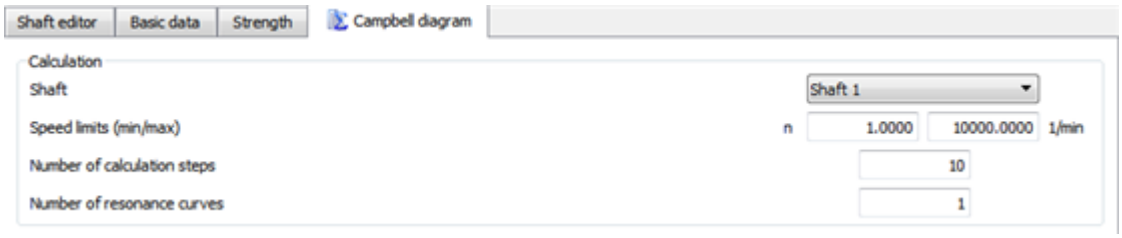


n

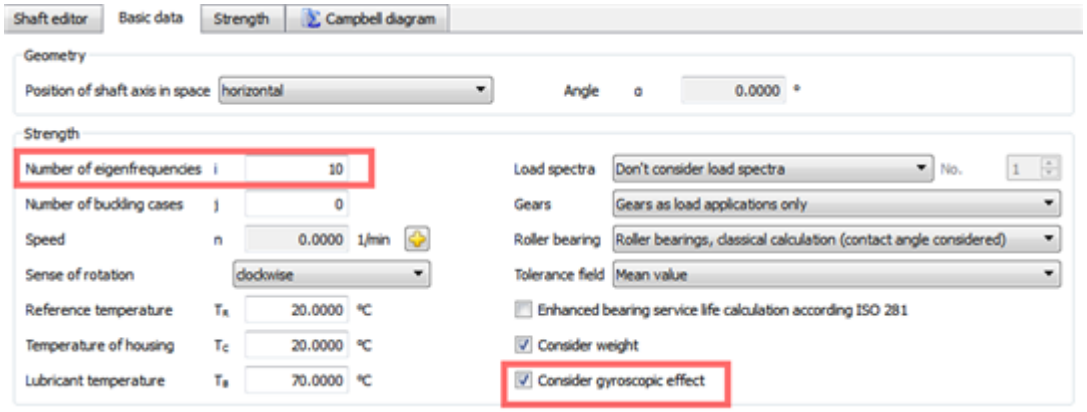
Figure 25.10: Determining the gap in the meshing point

25.7 Campbell diagram

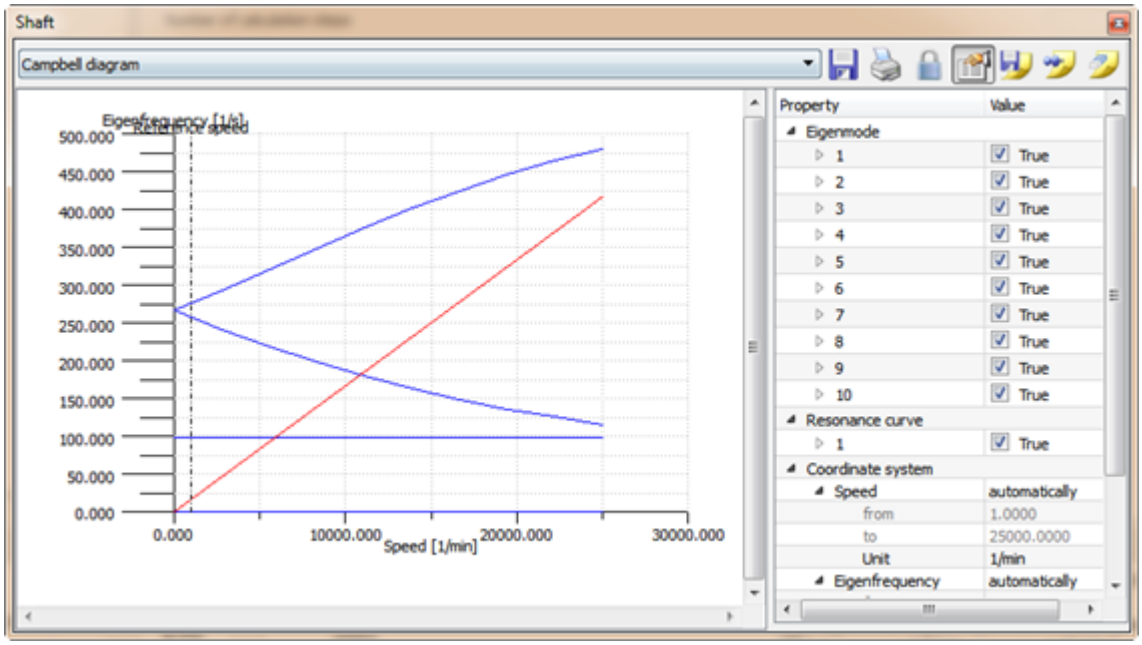
Select Calculation > Campbell diagram to enable the special calculation tab for the Campbell diagram. You can set the shaft to be analyzed, range of shaft speeds, the number of times the speed is calculated, and the number of resonance curves (synchronous speed curves) to be displayed.



The Campbell diagram shows the eigenfrequencies in a wider range of shaft speeds, and then we can follow the forward and backward whirls associated with the eigenmodes. In order to calculate the Campbell diagram, the number of eigenfrequencies should be set in the Basic data tab. The gyroscopic effect causes large changes in the eigenfrequencies and can be taken into consideration by setting the "Consider spinning effect" checkbox in the Basic data tab.



In normal cases, the backward mode drops in frequency, while the forward mode increases. For forward whirl, as shaft speed increases, the gyroscopic effects essentially increase the spring stiffness and increase the eigenfrequencies. The effect is reversed for backward whirl, and increasing shaft spin speed reduces the effective stiffness, and thus reduces the eigenfrequencies. The eigenfrequencies are also affected by the stiffness of the bearings.



25.8 Forced response

Use this module to calculate forced responses for the shafts defined in KISSsoft. The dynamic excitation of an unbalance mass is used to dynamically excite the shaft.

25.8.1 Calculation procedure

The first step in the calculation procedure is to enter this special calculation by selecting `Calculation` -> `Forced response`. There the user can define the shaft whose speed will be changed during the calculation, the relevant speed range and number of calculation steps in this range. Another input is the material (structural) damping for torsional, axial and bending vibrations. Note that the viscous damping of bearings must be defined separately for each bearing. In order for the calculation to run, at least one unbalance mass must be defined in a shaft ("Additional mass" element, mass and eccentricity fields).
Also note that, at the end of the calculation, results can be only requested at predefined documentation points. The documentation points in this case are used as measurement probes for the dynamic response of the shafts. Note that the resulting responses from different dynamic forces are added in the time range, and the final result is given based on the maximum value found during this operation.

Apart from the calculation for a range of speeds of the reference shaft, there is also an option for performing a calculation for a specific running speed of the reference shaft and determining the dynamic response results along its length.

25.8.2 Results

After the calculation has successfully finished, you can see the derived results by selecting `Graphics` -> `Shafts` > `Forced response`. Many different results can be plot there.

Chapter 26
General Bearing Calculation

26.1 Classification of bearings

Bearings can be classified according to:

- the type of motion, as for plain bearings, where the gliding motion takes place between the bearing and the supported part, and as for rolling bearings, where the rolling bodies describe a rolling motion.
- the direction of the bearing forces for radial and thrust bearings.
- the function in fixed bearings which can take up shearing forces and axial forces in both directions and in non-locating bearings which allows displacement in a longitudinal direction.

26.1.1 Properties

The most important properties for the operational performance and use of plain and rolling bearings can often be identified by examining their advantages and disadvantages.

There are hardly any rules to tell you how and when to use rolling bearings. The choice of bearing depends partly on the properties which are determined from the advantages and disadvantages and partly from the operational requirements such as size and type of load, maximum speed, required service life and practical experience.

26.1.1.1 Rolling bearings

Advantages: If used correctly, hardly any friction occurs when roller bearings are used, so their required starting torque is only slightly higher than their working moment (a major benefit when used for driving units!). They use little lubricant, are easy to maintain, do not require any running-in-time, and a large degree of standardization means that roller bearings are low-cost, and widely exchangeable with each other.

Disadvantages: They are especially sensitive to impacts and shocks, when they are not in use or running at low speeds. Their service life and maximum speed are limited. Their sensitivity to pollution can lead to added expense for sealing the bearing (wear, loss in efficiency!).

26.1.1.2 Plain bearing

Advantages: Due to their large, load-absorbing and lubrication area, plain bearings are insensitive to impacts and shocks they can run at unlimited speeds. If fluid friction is used, they have an almost unlimited service life. Their split construction

makes them easy to mount and dismount. Adjustable bearings give outstanding operational accuracy.

Disadvantages: Plain bearings require a higher starting torque (major disadvantage!). Because of their initial dry friction, they consume large quantities of lubricant, and require constant supervision. They are generally slightly less efficient than rolling bearings.

Chapter 27
Rolling Bearings (Classical Analysis)

Manufacturer catalogs (such as SKF) include fairly comprehensive methods for verifying the service life and static load capacity of rolling bearings. Specialized technical literature is also available to help you resolve more detailed problems [39].
KISSsoft includes data from well-known bearing manufacturers, to which you can add your own information.

In the KISSsoft start screen, select `Shaft and Bearings -> Rolling bearings` in the Module tree.

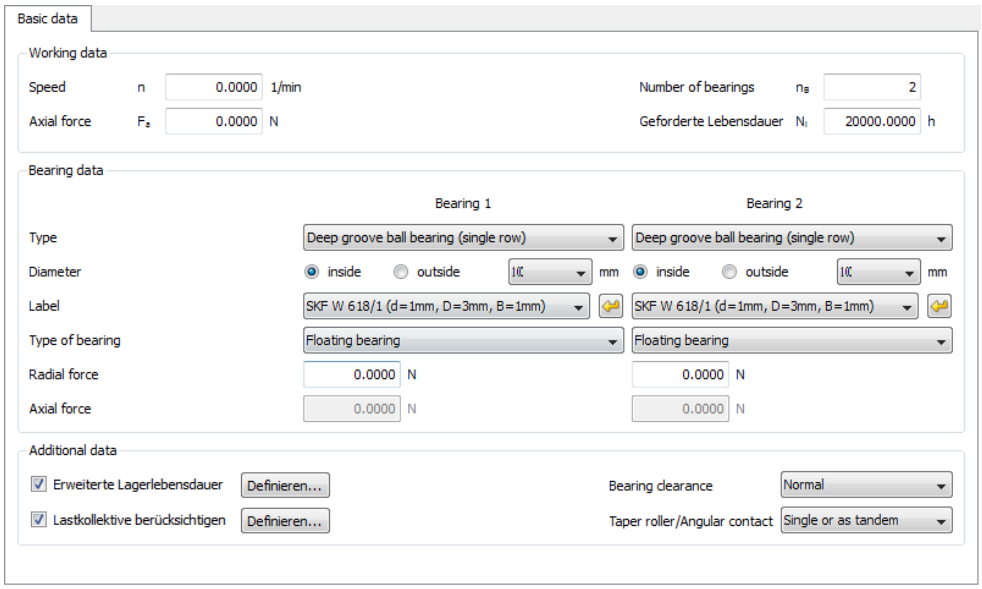


Figure 27.1: Basic data: Rolling bearings

There is not much to explain here, because the calculation provides numerous options, such as modified service life calculation or equivalent loads.

In the Basic data tab, you will see a  button for every bearing, next to its `Label` field. This function shows the service life of every bearing in the database (including the type and diameter). This makes it easy for you to select the best bearing for your purpose.

27.1 Selecting the type of rolling bearing

27.1.1 Properties of the most important bearing types

Selecting the most suitable type of rolling bearing is sometimes no easy matter. The table below presents an overview of the critical characteristics of the most important types of rolling bearing:

- **Deep groove ball bearing (DIN 625):**
  The single row radial deep groove ball bearing is the most commonly used, because it is both extremely versatile and also the most inexpensive form of rolling bearing, because of its simple design. This bearing can withstand relatively high radial and axial forces in both directions.

- **Single row angular contact ball bearing and four-point contact bearing (DIN 628):**
  Each ring of a self-holding single row angular contact ball bearing has one lower shoulder and one higher shoulder. The grooves on the higher shoulder are positioned so that the contact angle is normally $\alpha = 40^{o}$. The higher number of balls in this configuration means it can withstand not only radial forces but also larger axial forces in one direction (towards the higher shoulder) than deep groove ball bearings. Axial reaction forces due to the angle of the groove will be generated when the bearing is subjected to a radial load. You must take this into account when sizing the bearing. Because of their one-sided axial loading capacity, these types of bearings are usually installed in pairs in which the second bearing is mounted in the opposite direction. The axial force that acts on the bearing in the case of a back-to-back or face-to-face arrangement is calculated and displayed in the screen. See also 27.3.17.

- **Double row angular contact ball bearing (DIN 628):**
  The double row angular contact ball bearing corresponds to a pair of mirror image compounded single row angular contact ball bearings (back-to-back arrangement) with $\alpha = 25^{o}$ or 35°, and can therefore withstand radial and high axial forces in both directions.

  Areas of use: To support the shortest possible bending-resistant shaft that is subject to strong radial and axial forces: integral worm shafts, shafts with angled spur gears and bevel gears.

- **Double row self-aligning ball bearing (DIN 630)**
  This is a double row bearing with a cylindrical or conical bore (bevel 1:12). It can compensate for shaft displacement and misalignment (up to approximately 4° angular deviation) thanks to its hollow sphere race in the outer ring. It can be subjected to radial loads and axial loads in both directions.

Areas of use: bearings which are inevitably subject to mounting inaccuracies and bending of the shaft, e.g. transmissions, conveyors, agricultural machinery, etc.

- **Cylindrical roller bearing (DIN 5412)**:
  Cylindrical roller bearings can support larger radial loads than ball bearings of the same size (point contact area!) because the contact between the rollers and the races is made along a line. Demountable cylindrical roller bearings can only support small axial forces (if at all) and require accurately aligned bearings. There are different (construction) types (depending on the rim arrangement): N and NU have an unconfined outside and inside ring, and can be used as non-locating bearings, type NJ is a step bearing, and types NUP and NJ can be used as a fixed bearing or locating bearing for axial shaft support in both directions.

  Areas of use: in gearboxes, electric motors, for axles of rail vehicles, for rollers in a rolling mill. In general, suitable for bearing applications that are subject to large radial loads.

- **Needle roller bearings (DIN 617)**:
  A special type of cylindrical roller bearing, in which a cage separates the needle rollers to keep them at a specific distance from, and parallel to, each other. The bearing is supplied with or without an inner ring, and is only suitable for radial forces. It can be characterized by its small overall diameter, its high degree of rigidity in the radial direction, and by its relative insensitivity to an uneven load.

  Areas of use: Predominantly used at low to medium speeds and when oscillatory motion is present, e.g. as connecting rod bearings, rocker-shaft bearings, swivel arm bearings, jointed cross-shaft axle bearings (vehicles), spindle bearings, etc.

- **Taper roller bearing (DIN 720)**:
  The ring races in taper roller bearings are cone-shaped shells which must converge into one point due to the action of kinematic forces. The bearings with $\alpha = 15^{o}(30^{o})$ can support high loads both in the radial and axial directions. The detachable outer ring makes them easy to assemble and dismantle; Taper roller bearings are installed in mirror image pairs. Taper roller bearings are installed in mirror image pairs. The bearing clearance can be set and adjusted as required. Due to the angle of the race, a radial force produces an axial reaction force.

  Areas of use: Hub bearings of vehicles, cable pulley bearings, spindle bearings in machine tools, shaft bearings in worm gears and bevel gears.

  Calculation: The axial force that you must specify when calculating dynamic

equivalent loads is defined in several theories (see, for example, page 296 of FAG Wälzlager Catalog WL 41520DE (1995)). The axial force acting on the bearing is displayed in the screen. See also section 27.3.17. The bearing forces that include the contact angle can be calculated directly.

☼ **Barrel-shaped bearings (DIN 635), toroidal roller bearings (CARB), and double row self-aligning ball bearings (DIN 635)**:
Spherical races in the outer ring and barrel-shaped rollers (toroidal-shaped for CARB bearings), as in double row self-aligning ball bearings, enable barrel-shaped, toroidal roller (CARB) and double row self-aligning roller bearings with a cylindrical and conical bore (1:12) to compensate for misalignment and for the angular dislocation of the shaft (oscillating angle 0.5^o to 2°). Barrel roller bearings (single row self-aligning roller bearings) are suitable for high radial loads, but can only withstand low axial forces. In contrast, double row self-aligning roller bearings ($\alpha = 10^o$) can be used for the highest radial and axial forces. Toroidal roller bearings (CARB) have an extensive range of uses in many load applications. Toroidal roller bearings combine the angular flexibility of double row self-aligning roller bearings with the axial displacement options of cylindrical roller bearings.

Areas of use: for heavy wheels and cable pulleys, propelling shafts, rudder posts, crank shafts, and other heavily loaded bearings. Toroidal bearing: Paper making machinery, blowers and, generally, in planetary gear units.

27.1.2 Comparing types

Selecting the most suitable type of rolling bearing is sometimes no easy matter. The table below presents an overview of the most important characteristics. The bearing you select for specific operating conditions has often already been determined by its properties and characteristics. You can use this information to select the bearing you require for frequently occurring working cases and for specialized requirements. However, results may overlap, and therefore the cost factor may be decisive.

In addition to the specified rolling bearing types, some hybrid bearings (with ceramic rolling bodies) have been included, for some types. The special properties of these bearings are described in the "Hybrid bearing" section.

Radial bearing:

Features	a	b	c	d	e	f	g	h	i	j	k	l	m	n
Radial load capability	⊗	⊗	⊗	∅	⊗	+	+	+	+	+	+	+	+	+
Axial load capability	⊕	⊗	⊗	⊗	∅	-	⊕	⊕	-	⊕	⊕	+	⊕	⊗

Inside position adjustment	-	-	-	-	-	+	∅	-	+	∅	-	-	-	-
Mounting position adjustment	⊕	⊕	⊕	-	⊕	-	-	⊕	-	⊕	⊕	⊕	⊕	⊕
Dismountable bearings	-	-	⊕	⊕	-	+	+	+	+	⊕	-	+	-	-
Alignment error adjustment	∅	-	-	-	+	∅	∅	∅	-	∅	-	∅	+	+
Increased precision	⊕	⊕	⊕	∅	-	⊗	⊕	⊕	+	-	-	⊗	-	-
High speed running	+	+	⊕	∅	⊗	+	⊗	⊗	+	-	-	⊕	⊕	⊕
Quiet running	+	⊗	∅	∅	∅	⊕	∅	∅	⊕	-	-	∅	∅	∅
Conical bore	-	-	-	-	+	⊗	-	-	+	-	-	-	+	+
Seal on one/both sides	⊕	-	⊕	-	⊕	-	-	-	-	-	⊕	-	-	⊗
High stiffness	⊗	⊕	⊕	⊗	∅	⊕	⊕	⊕	+	+	+	+	⊕	⊕
Low friction	+	⊕	⊗	⊕	+	⊕	⊕	⊕	+	-	-	⊕	⊗	⊕
Fixed bearing	⊕	+	⊕	⊕	⊗	-	⊗	⊕	-	⊗	⊗	+	⊕	⊕
Non-locating bearing	⊗	⊗	⊗	-	⊗	+	⊗	∅	+	⊗	⊗	∅	⊗	⊗

+ very good ⊕ good ⊗ normal/possible ∅ with restrictions - not suitable/no longer relevant

a Deep groove ball bearing

b Angular contact ball bearing (single row)

c Angular contact ball bearing (double row)

d Four-point contact bearing

e Double row self-aligning ball bearing

f Cylindrical roller bearings NU, N

g Cylindrical roller bearings NJ

h Cylindrical roller bearings NUP, NJ+HJ

i Cylindrical roller bearings NN

j Cylindrical roller bearings NCF, NJ23VH

k Cylindrical roller bearings NNC, NNF

l Taper roller bearing

m Barrel roller bearing

n Double row self-aligning roller bearing

not listed here: Needle roller bearing, needle cage

Thrust bearing:

Features	o	p	q	r	s	t
Radial load capability	-	-	∅	-	-	∅
Axial load capability	⊗	⊗	⊗	⊗	⊗	+
Inside position adjustment	-	-	-	-	-	-
Mounting position adjustment	-	-	-	-	-	-
Dismountable bearings	+	+	-	+	+	+
Alignment error adjustment	⊕	⊕	∅	-	-	+
Increased precision	⊗	-	+	+	⊕	-
High speed running	⊕	∅	⊗	+	∅	∅
Quiet running	∅	-	∅	∅	-	-
Conical bore	-	-	-	-	-	-
Seal on one/both sides	-	-	-	-	-	-
High stiffness	⊕	⊕	⊗	+	⊕	⊗
Low friction	⊗	∅	⊕	⊕	-	-
Fixed bearing	⊕	⊕	+	+	⊗	⊗
Non-locating bearing	-	-	-	-	-	-

+ very good ⊕ good ⊗ normal/possible ∅ with restrictions - not suitable/no longer relevant

o Axial deep groove ball bearing (one-sided)

p Axial deep groove ball bearing (double direction)

q Axial angular contact ball bearing (single direction)

r Axial angular contact ball bearing (double direction)

s Axial cylindrical roller bearing

t Axial spherical roller bearings

not listed here: Thrust needle cages, axial tilted roller bearings, cross roller bearings

27.1.3 Hybrid bearings

In a hybrid bearing, the race is made of rolling bearing steel, and the rolling bodies are made of a ceramic material (silicon nitrite, Si_3N_4).

Hybrid bearings are included in the databases of standard rolling bearings. The rolling bearing database has a particular setting which identifies hybrid bearings.

The calculation basis is the same as for standard types of rolling bearing.

However, the thermal reference speed and thermally safe operating speed cannot be determined, because hybrid bearings are not covered by the standards.

The moment of friction for these bearings cannot be determined because the calculation methods used in rolling bearing catalogs do not cover hybrid bearings.

The most important benefits of hybrid bearings are:

- higher stiffness
- suitability for use at higher speeds
- reduced inertia and centrifugal forces in the bearing
- reduced frictional heat
- lower energy consumption
- longer bearing life and lubricant service life

27.2 Load capacity of rolling bearings

We distinguish between the dynamic load capacity of the rotating bearing and the static load capacity at standstill, at very slow speed or if very small oscillations are present, relative to the working state, but not to the effect of the load.

27.2.1 Dynamic load capacity

The dynamic load capacity is a property of the entire bearing. ISO 281 describes a number of various properties of a rolling bearing that occur, if the bearing experiences specific mechanical loading under specific conditions at specific speeds. This data is then used to calculate the number of operating hours (this is usually based on a failure probability of 10%).

27.2.2 Permissible static stress

The static load capacity includes properties that a rolling bearing must display in order to withstand certain mechanical loading situations at standstill, at very low speed (n < 20 rpm), or during oscillatory motion.

Plastic deformation (indentation) occurs between the roller bearings and the races when the bearing is subjected to a moderate static stress due to the weight of the shaft and the other elements. This value gradually increases as the stress increases. However, the plastic deformation must not be so great as to influence the operational properties of the bearing in its rotational movement. As defined in ISO 76, the static characteristic value $S_0 = C_0/P_0$ is a safety factor against detrimental plastic deformation which is a measure of the sufficient static load capacity.

The static load number, which is used to determine the bearing size, can be determined by taking into account the safety margin which depends on the operating conditions:

$S_0 > 2$	for shocks and impacts as well as exacting requirements for smooth operation and for axial spherical roller bearings
$S_0 = 1$	for normal operating conditions and low noise requirements
$S_0 = 0.5...0.8$	for smooth and non-impact operating conditions with few requirements (non-loaded bearing with adjusting or swivel motion)

27.2.3 Rolling bearing calculation with inner geometry

The rolling bearing reference service life calculation is based on ISO/TS 16281.

The results of this calculation are the reaction forces, torques, the displacements and rotations, the maximum Hertzian pressure on the inner and outer race (right and left ring for a thrust bearing), the statical safety, the reference and modified reference service life in hours, the stiffness matrix at the operating point, and the pressure curve on each rolling element.

For more detailed information, see 25 - Rolling bearing internal geometry (see section "Rolling Bearings (Internal Geometry)" on page III-823)

If the rolling bearing inner geometry is given by the manufacturer, then it is used in the calculation. If it is unknown, then KISSsoft runs an approximation method that tries to determine the internal geometry using the rolling bearing load ratings (both static C0 and dynamic C) given by the manufacturer. This procedure is based on ISO 76 and ISO 281-4, and normally produces quite useful results.

If you only know how many rollers or balls there are, and want to use this data when performing a calculation according to the standard, we suggest the following:

- Run a calculation based on bearing inner geometry. As the inner geometry is unknown, the system approximates it for you.
- Create a rolling bearing report, and note down the bearing inner geometry data.
- Open the KISSsoft database with write access authorization. Navigate to the needed bearing type, and add a new bearing. In the inner geometry tab, copy all internal geometry data you noted in step 2. In the number of rolling elements field (Z), input the appropriate number for your actual rolling bearing.
- Save and close the database.
- Go to the elements editor, and update your rolling bearings with the one you added to the database, above. Rerun the calculation and get the results.

If the inner geometry you enter in the database is either insufficient or incorrect, this data is then ignored, and the inner geometry is approximated. A note is then printed in the report that an approximation of the inner geometry has been used.

It is not possible to take the internal geometry for every type of rolling bearing into account, when performing a calculation for it. The calculations in which this is currently possible are listed in the database chapter, Rolling bearing internal geometry (see page I-163).

27.3 Thermally safe operating speed

The definition of the thermally safe operating speed is described in DIN 732 [8]. The calculation of the thermally safe operating speed is based on a heat balance at the bearing. The thermally safe operating speed is derived from the thermal reference speed, using the speed ratio. The result of this calculation is the speed that will be reached by the bearing running at the permitted temperature in an actual situation. This thermally admissible speed may differ greatly from other operating speed limits, depending on lubrication type, because the reference conditions only apply to quite specific cases. In order to define the thermally permissible operating limit, you must first define the thermal nominal speed for each case.

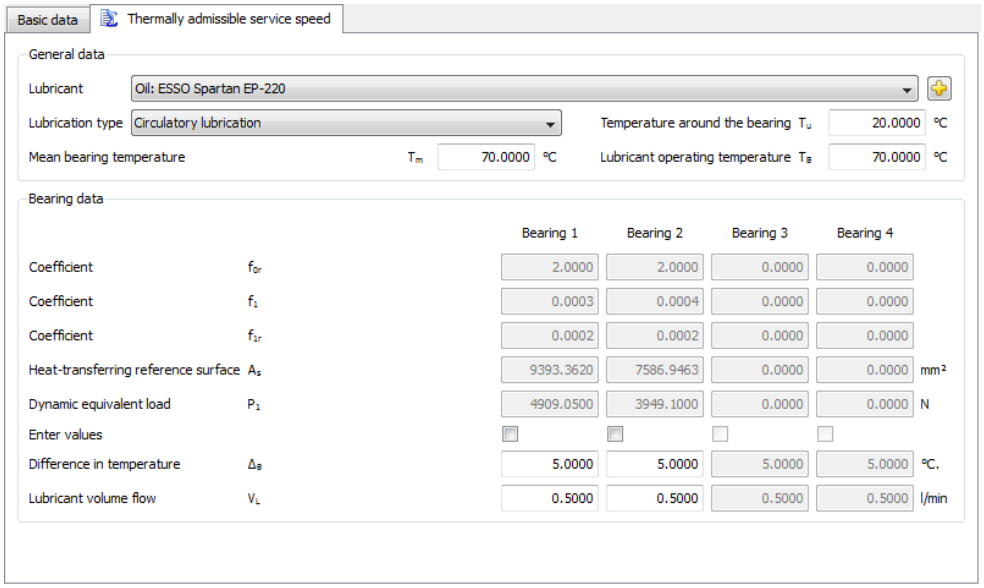


Figure 27.2: Thermally admissible operating speed

NOTE:

Calculations cannot be performed for barrel roller bearings (single row self-aligning roller bearings), axial tilted roller bearings, cross roller bearings, and all hybrid bearings, because none of the corresponding standards have values for them.

27.3.1 Thermal reference speed

The thermal reference speed is defined in DIN ISO 15312 [7]. The thermal reference speed is the bearing-specific speed reached under a given set of nominal operating conditions such that equilibrium is achieved between heat development (friction) and heat dissipation (through bearing contact and lubricant). Mechanical

or kinematic criteria are not taken into account for this speed. The reference values (temperatures, loads, viscosity of the lubrication, reference face of the bearing,. . .) are fixed so that the reference speed using oil or grease lubricated bearings will result in identical values.

27.3.1.1 Dissipated Heat Flows

The heat flow Qr is calculated from the reference heat flow density specific to a rolling bearing qr (for heat flow dissipated through bearing contact and lubricant), and from heat dissipation through the reference surface Asr.
$Qr = 10^{-6} * qr * Asr$
qr, Asr are defined under reference conditions according to DIN ISO 15312.

27.3.1.2 f0r and f1r coefficients

The coefficients f0r and f1r used to define the reference thermal operating speed are different, depending on which bearing type/series (also lubrication type for f0r) is used. They are shown in Table A.1 of the standard. Not all bearing variants are listed in the table.

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	f0r (tabular value)	f1r
Ball bearing	1.7	0.00015
Roller bearing	3	0.0003
Thrust ball bearing	1.7	0.00015
Thrust roller bearing	3.5	0.0015

27.3.1.3 Calculating the thermal nominal speed

The dissipating heat flows and the friction power are set as equal values so that the energy balance of the bearing is correct. The equation for the energy balance is:

$NFr = 10^{3} * Qr$

NFr: Friction power [W]
Qr : dissipated heat flow: [kW]

The subsequent equation becomes:

$(\pi * n\theta r)/30 * (10^{-7} * f0r * (\nu r * n\theta r)^{2/3} * dm^3 + f1r * P1r * dm) = qr * ASr$

nθr: thermal reference speed [1/min]
f0r: Coefficient from Table A.1, DIN ISO 15312 [-]
r: Reference viscosity [mm2/s]
dm: average rolling bearing diameter [mm]
f1r: Coefficient from Table A.1, DIN ISO 15312 [-]
P1r: Reference load [N]
qr: Rolling bearing-specific reference heat flow density (bearing contact, lubricant) [kW/m^2]
ASr: Reference surface area dissipating heat [mm2]

The value nθr can be determined from this equation.

27.3.2 Process for calculating thermally permitted operating speed (DIN 732-2)

As, when calculating the reference thermal operating speed, this calculation is based on equilibrium in the bearing. Dissipating heat flow:
$Q = Q_S + Q_L + Q_E$

Q_S: heat flow
$Q_L$dissipated across the bearing contacts by lubrication (only when there is circulatory lubrication) (the lubricant density ρ = 0.91 kg/dm3 and specific heat capacity cL = 1.88 KJ/(kg *K) are predefined.)
Q_E: additional heat flow (it is assumed that Q_E= 0 for the calculation)

27.3.2.1 Friction coefficients f0 and f1

The coefficient values f0 and f1 and the dynamic equivalent load P1 are only needed to define the load and lubrication parameters. These values differ depending on the specific bearing type/model, lubrication type, or load direction. They are listed in Table A.1 in the standard. Not all bearing variants are listed in the table. The following values for various types of lubrication have been defined (and incorporated in KISSsoft). They are based on the notes about f0 in Table A.1 in the standard.

- Oil, bath lubrication, bearing in oil mist: f0 = 0.5 * f0 (tabular value)
- Oil, bath lubrication, oil level up to middle bearing: f0 = 2.0 * f0 (tabular value)

- Oil, bath lubrication, oil level up to middle of the lowest rolling element: f0 = 1.0 * f0 (tabular value)
- Oil, circulatory lubrication: f0 = 2.0 * f0 (tabular value)
- Grease, run-in bearing: f0 = 1.0 * f0 (tabular value)
- Grease, newly greased: f0 = 2.0 * f0 (tabular value)

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	P1	f0 (tabular value)	f1
Ball bearing	3.3*Fa - 0.1*Fr (P1 <= Fr) where P1 = Fr	1.7	0.0007*(P0/C0)^0.5
Roller bearing	Fr	3	0.0003
Thrust ball bearing	Fa	1.7	0.0007*(P0/C0)^0.5
Thrust roller bearing	Fa	3.5	0.0015

27.3.2.2 Calculating the thermally permissible operating speed limit

The thermal permissible operating speed is calculated from the reference thermal operating speed with the help of the speed ratio, nθ = fn * nθr

The load and lubrication parameters must be calculated before the speed ratio can be iterated from this equation:

$KL * fn^{5/3} + KP * fn = 1$

Load parameter KL:

$KL = 10^{-6} * (\pi/30) * n\theta r * 10^{-7} * (f0r * n^{2/3} * n\theta r^{2/3} * dm^3)/Q$

Lubrication film parameter KP:

$KP = 10^{-6} * (\pi/30) * n\theta r * (f1 * P1 * dm)/Q$

nθr: thermal reference speed [1/min]
f0: Coefficient of friction from Table A.1, DIN 732 [-]
f1: Coefficient of friction from Table A.1, DIN 732 [-]
n: Lubricant viscosity [mm^2/s]
dm: average rolling bearing diameter [mm]

P1: Reference load [N]
Q: Total dissipated heat flows [kW]

27.4 Moment of friction

In a rolling bearing, the heat development, and therefore the operating temperature, is caused by the friction between individual components. The moment of friction in the rolling bearing is indicated by several distinct losses due to resistance during operation. Select this calculation method under `Calculation > Settings`.

You can only perform this calculation by clicking the "Modified rating life according to ISO 281" option (`Basic data/Strength` tab).

27.4.1 Calculation according to SKF Catalog 1994

The prerequisite for calculating the moment of friction is that the bearing rotating surfaces must be separated by a film of lubrication. The total bearing moment of friction results from the sum:

$$M = M_0 + M_1 \tag{27.1}$$

M_0: load-independent moment of friction

M_0 is determined by the hydrodynamic losses in the lubricant. It is especially high in quickly rotating, lightly loaded bearings. The value M_0 depends upon the quantity and viscosity of the lubricant, as well as the rolling speed.

M_1: load-dependent moment of friction

M_1 is determined by the elastic deformation and partial sliding in the surfaces in contact, especially due to slowly rotating, heavily loaded bearings. The value M_1 depends on the bearing type (bearing-dependent exponents for the calculation), the decisive load for the moment of friction and the mean bearing diameter.

For axially loaded cylindrical roller bearings, an additional axial load-dependent moment of friction, M_2, is added to the formula.

$$M = M_0 + M_1 + M_2 \tag{27.2}$$

M_2: axial load-dependent moment of friction

M_2 depends on a coefficient for cylindrical roller bearings, the axial loading and the bearing's average diameter.

For sealed rolling bearings, an additional axial load-dependent moment of friction, M_3, is added to the formula.

$$M = M_0 + M_1 + M_3 \quad (27.3)$$

M_3: **Moment of friction for grinding seals**
The moment of friction for grinding seals depends on the bearing type, the bearing size, the diameter of the seal-lip mating surface, and the layout of the seal. As the type of seal, the diameter of the seal-lip mating surface, and the seal layout, differ from one manufacturer to another, it is difficult to define a generally applicable moment of friction.

Under `Calculation/Settings` there are different options for determining this reference size:

- **according to SKF main catalog in selected calculation method**
  - *According to main catalog 4000/IV T DE:1994*
    You will find values for the seal types used in your bearings in the SKF catalog, which is integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
    Example of a seal label in the name of a rolling bearing:
    `SKF: 623-2RS`: this means that the bearing has a RS1 type seal on both sides. The KISSsoft system then searches for names containing "-2RS1". If this label is present, the coefficients from the SKF catalog are applied and the moment of friction for grinding seals is calculated.

- **according to ISO/TR 13593:1999 Viton** M_{seal} this diameter is calculated with the formula: M_{seal} = 3.736*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{seal} this diameter is calculated with the formula: M_{seal} = 2.429*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm

Coefficients f0, f1 (see page III-806) and P1 (values that depend on the bearing type and bearing load) used for the calculation have been taken from DIN 15312. The formulae, exponents and coefficients have been taken from the SKF Catalog, 1994 Edition.

27.4.2 Calculation according to SKF Catalog 2013

As this calculation has to take into consideration a myriad of factors and influences, it is only performed if selected as an option in the modified rating life calculation. However, this calculation can also be performed without these default values. The calculation of the total moment of friction according to the 2013 SKF catalog is determined by a combination of rolling and sliding friction in the roller contacts (between rolling bodies and cage, the bearing surface, the lubricant, and the sliding friction from grinding seals caused in sealed bearings). The calculation of the moment of friction depends on various coefficients:

- Rating
- Type of bearing
- Bearing size
- Operating speed
- Lubricant properties
- Lubricant quantities
- Seals

The following working conditions must be present for the calculation to be performed:

- Grease or oil lubrication (oil bath, oil mist, or oil injection process)
- Load equal or greater than minimum load
- Load constant in size and direction
- Nominal operating clearance

If the rating is less than the minimum load, the calculation continues using the minimum load. If a minimum load value has been entered in the database, this value is used. If not, the software will determine this value. In the case of radial bearings, the minimum load is converted into a minimum radial force. In thrust bearings, the minimum axial force is defined by the software. The value for the minimum load is not used here.

The formula for the total moment of friction is:

$$M = M_{rr} + M_{sl} + M_{seal} + M_{drag}$$

M_{rr}: **Rolling moment of friction**

The rolling moment of friction depends on the type of bearing, the average diameter, the radial and axial loading, the rotation speed, and the viscosity of the

lubrication. The design coefficients required to calculate the rolling moment of friction are defined using the rolling bearing's series. The design coefficients and coefficients used in the calculation are taken from the SKF Catalog 2013.

Coefficients used for rolling friction:

- ϕ_{ish}: **Lubricant film thickness factor**
  In a lubricant flow, the lubricant is exposed to shear forces caused by the movement of the rolling bodies. This produces heat and therefore reduces the rolling moment of friction.
- ϕ_{rs}: **Lubricant displacement factor**
  The constant rolling action squeezes excess lubricant away from the contact zone of the rolling body. This reduces the lubricant film thickness and therefore reduces the rolling moment of friction.

Assumptions have been made for bearing types and bearing series for which no design coefficients have been defined in the catalog, so that the rolling moment of friction can still be calculated despite their absence.

M_{sl}: **Sliding moment of friction**
The sliding moment of friction depends on the type of bearing, the average diameter, the radial and axial loading, and the viscosity of the lubrication. The design coefficients required to calculate the sliding moment of friction are defined using the rolling bearing's series. You will find the factors used for this calculation in the SKF 2013 catalog.

M_{seal}: **Moment of friction for grinding seals**
The moment of friction for grinding seals depends on the bearing type, the bearing size, the diameter of the seal-lip mating surface, and the layout of the seal. As the type of seal, the diameter of the seal-lip mating surface, and the seal layout, differ from one manufacturer to another, it is difficult to define a generally applicable moment of friction.

Under `Calculation/Settings` there are different options for determining this reference size:

- **according to SKF main catalog in selected calculation method**
  - *According to main 10000/1 EN:2013*
    You will find values for the seal types used in your bearings in the SKF catalog, which is integrated in the KISSsoft software. If the KISSsoft system finds a familiar seal label in the bearing label, it calculates the moment of friction for a grinding seal using the coefficients listed in the catalog. Otherwise, the moment of friction is set to zero.
    Example of a seal label in the name of a rolling bearing:
    `SKF: 623-2RS`: this means that the bearing has a RS1 type seal on both sides. The KISSsoft system then searches for names containing "-2RS1". If

this label is present, the coefficients from the SKF catalog are applied and the moment of friction for grinding seals is calculated.
In KISSsoft, the diameter of the mating surface is calculated with:
ds = d + (D - d) * 0.2

- **according to ISO/TR 13593:1999 Viton** M_{seal} this diameter is calculated with the formula: M_{seal} = 3.736*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm
- **according to ISO/TR 13593:1999 Buna N** M_{seal} this diameter is calculated with the formula: M_{seal} = 2.429*10^-3*dsh; M_{seal} in Nm, d_{sh} Shaft diameter in mm

M_{drag}: **Moment of friction caused by lubrication losses**
This moment of friction is caused by flow, splash, or injection losses during oil bath lubrication. To calculate this torque, you must also input the oil level depth (h_{Oil}), which you can specify in `Calculation > Settings`. You will find a more detailed description of this entry in the Oil level and lubrication type (see page III-822) section. The design coefficients KZ and KL for rolling bearings with a cage are also applied to toroidal roller bearings (CARB).

27.4.3 Calculation according to Schaeffler 2014 (INA, FAG)

To define the total moment of friction, the speed, load, lubrication type, lubrication method and viscosity of the lubricant at operating temperature must be known.

Formula used for the total moment of friction:

$$M_R = M_0 + M_1 \tag{27.1}$$

M_0: speed-independent (load-independent) moment of friction

M_0 is determined by the hydrodynamic losses in the lubricant. It is especially high in quickly rotating, lightly loaded bearings. The value M_0 depends upon the quantity and viscosity of the lubricant, as well as the rolling speed.

M_1: load-dependent moment of friction

M_1 is determined by the elastic deformation and partial sliding in the surfaces in contact, especially due to slowly rotating, heavily loaded bearings. The value M_1 depends on the bearing type (bearing-dependent exponents for the calculation), the decisive load for the moment of friction and the mean bearing diameter.

For axially loaded cylindrical roller bearings, an additional axial load-dependent moment of friction, M_2, is added to the formula.

$$M_R = M_0 + M_1 + M_2 \qquad (27.2)$$

M_2: axial load-dependent moment of friction

M_2 depends on a coefficient, k_B, for cylindrical roller bearings, the axial loading and the bearing's average diameter.

For bearings with a TB design (better axial load capacity achieved using new calculation and production methods), bearing factor f2 is displayed in a special diagram in the main catalog.

Coefficients f0, f1 (see page III-806) and P1 (values that depend on the bearing type and bearing load) used for the calculation have been taken from DIN 15312. The formulae, exponents and coefficients have been taken from the Schaeffler Catalog, 2014 Edition.

27.5 Maximum Speeds

Rolling bearings are reliable and can be expected to reach their calculated service life as long as the maximum speed (speed limit) is not exceeded. This depends on the type, size and lubrication.

A warning message is displayed if the maximum permissible speed is exceeded.

Depending on the lubrication type, the actually permitted maximum speed can be much lower. For more details, see the "Thermal admissible operating speed" section 27.3.

27.6 Service life

The nominal service life is calculated using the formulae given in ISO 281, and corresponds to the formulae that can also be found in the manufacturers' catalogs. Usually, the service life is calculated at an estimated survival rate of 90% (10% probability of failure, see also section 27.7), in hours. The label used here is L_{10h} (h: hours; 10: probability of failure).

27.6.1 Modified service life calculation in accordance with the Supplement to DIN ISO 281 (2007)

ISO 281 contains the regulations for "modified service life" which take into account the influence of loads, lubricant conditions, materials specifications, type, material internal stresses and environmental factors.

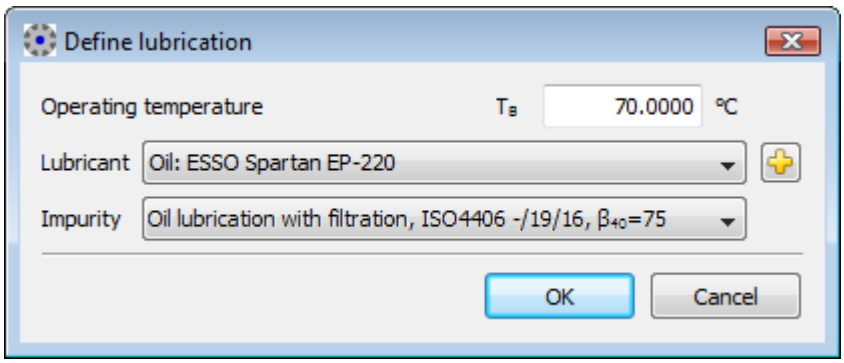


Figure 27.3: Dialog for modified rating life calculation

The service life coefficient a_{ISO} can be defined as follows:

$$a_{ISO} = f\left(\frac{ec \cdot Cu}{P}, \kappa\right) \tag{27.3}$$

a_{ISO}: service life coefficient from diagram [-]

ec : impurity characteristic value [-]

Cu : fatigue load limit [N]

P : dynamic equivalent load [N]

κ: viscosity ratio = n_u/n_{u1}

n_{u1}: reference viscosity diagram [$mm^2/2$]

n_u: VT diagram for the lubricant [mm^2/2]

The fatigue load limit Cu is specified by the bearing manufacturers. If none of these values are known, you can calculate them with the approximate formula as defined in ISO 281.
The impurity characteristic value ec (between 0 and 1) is taken directly from the degree of cleanliness.

27.6.2 Service life calculation with load spectra

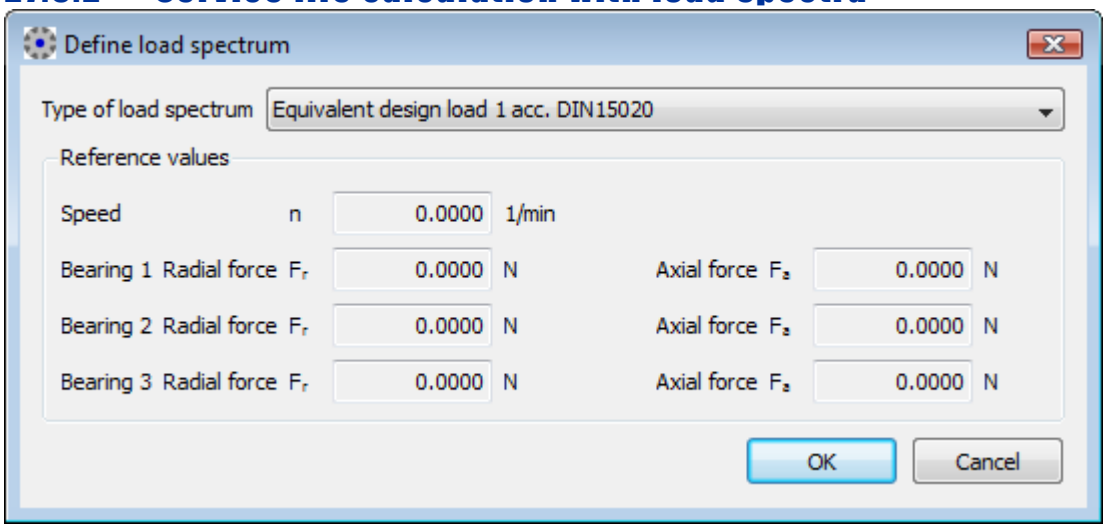


Figure 27.4: Dialog for selecting the load spectrum

The load spectrum on the bearing has these values:

k: number of elements in the load spectrum

q_i: frequency (load spectrum bin i) (%)

n_i: speed (load spectrum bin i) (rpm)

F_{ri}: radial force (load spectrum bin i) (N)

F_{ai}: axial force (load spectrum bin i) (N)

You can take this load spectrum data from the shaft calculation, in which case you may obtain different load spectra for radial and axial forces. Or alternatively you can select a load spectrum from the database. For bearing forces, the important factor here is the torque factor (not the load factor) and a negative sign operator will only affect the axial force.

Achievable service life with simple calculation approach:
You calculate the service life by defining an equivalent design load and the average speed. You can then use the usual formulae to calculate the service life.

$$n_m = \frac{n_1 \cdot q_1}{100} + \frac{n_2 \cdot q_2}{100} + \ldots + \frac{n_k \cdot q_k}{100} \tag{27.4}$$

$$P_m = \sqrt[p]{\frac{P_1^p \cdot q_1}{100} + \frac{P_2^p \cdot q_2}{100} + \ldots + \frac{P_k^p \cdot q_k}{100}} \tag{27.5}$$

n_m: average speed

p: exponent in the service life formula (3.0 or 10/3)

P_i: dynamic equivalent load (load spectrum bin i)

P_m: average dynamic equivalent load

Achievable service life with the modified rating life calculation:
When the modified service life calculation is used, the service life is calculated separately for every equivalent load spectrum element. The result is then used to determine the total service life:

$$L_{hna} = \frac{100}{\frac{q_1}{L_{hna1}} + \frac{q_2}{L_{hna2}} + \ldots + \frac{q_k}{L_{hnak}}} \tag{27.6}$$

L_{hnai}: service life (load spectrum bin i) in the case of speed n_i and load F_{ri}, F_{ai}

L_{hna}: Total service life

27.7 Failure probability

Normally, the failure probability is assumed to be 10%. This means there is a 90% probability that the nominal service life will be achieved. In this case the coefficient a_1 is equal to 1.0. If the failure probability value has to be lower, this coefficient must also be lower (at 1%, a_1 = 0.21).

You define the failure probability in `Calculation > Settings`.

27.8 Bearings with radial and/or axial force

For every bearing, you can specify whether it is subject to a radial or axial force. If the bearing is subject to axial force, you must also specify whether the force is applied in both directions (<>), in the direction of the Y-axis (- >) or in the opposite direction (< -).

27.9 Calculating axial forces on bearings in face-to-face or back-to-back arrangements

Because of the inclination of the races in the bearing, a radial load generates axial reaction forces in taper roller bearings, high precision angular contact ball bearings and angular contact ball bearings. This data must be taken into account when the equivalent design load is analyzed.

Axial reaction forces are calculated in accordance with SKF (rolling bearing catalog) which exactly match the values defined in FAG.

For bearings in an back-to-back arrangement, left bearing A, right bearing B, outer axial force in A-B direction, the following data applies:

Condition	Formula
$\frac{F_{rA}}{Y_A} \le \frac{F_{rB}}{Y_B}$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a > \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a \le \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + F_{aB}$ $F_{aB} = \frac{F_{rA}}{2 \cdot Y_A} - F_a$
F_{rA}, F_{rB} Y_A, Y_B F_a F_{aA}, F_{aB}	Radial force on bearing A, B Y coefficient of bearing A, B External axial force Axial force on bearing A, B

For all other cases, (face-to-face arrangement or axial force in the other direction) simply reverse the formula.

These calculated pretension force values are displayed in the main window. If the actual internal forces are higher, for example, due to the use of spring packages, you can change the value manually.

27.10 Oil level and lubrication type

Input the oil level and the lubrication type under `Calculation > Settings`. These entries are needed to define the moment of friction due to lubrication losses. The value h is given in the shaft calculation and results in the following formula for every bearing:

$$H = \frac{D}{2} - h \geq 0$$

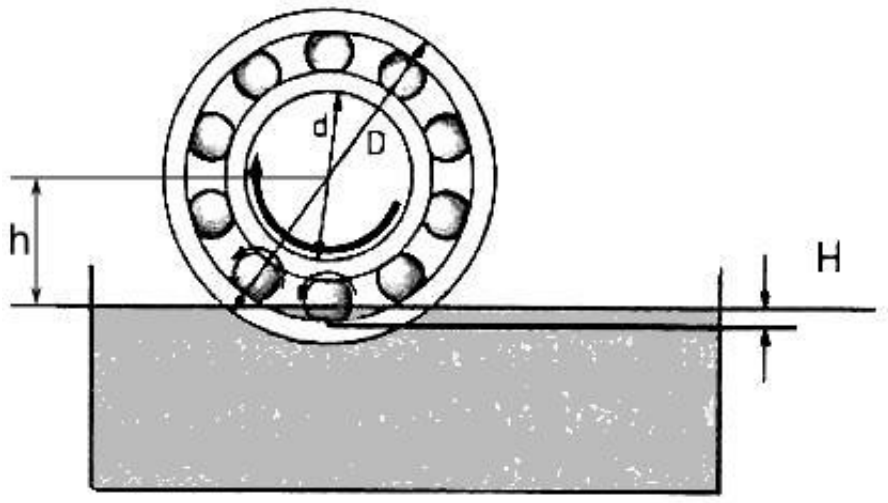


Figure 27.4: Oil level in the bearing

Two different types of lubrication can be defined:

- Oil bath lubrication
- Oil injection (spray) lubrication

If you select the `Oil injection lubrication` option, the value determined for the flow loss-dependent moment of friction for oil bath lubrication is multiplied by 2.

Chapter 29
Rolling Bearings (Internal Geometry)

In addition to the classic rolling bearing calculation method (see Rolling Bearings (on page III-795)), KISSsoft also provides a calculation method that complies with ISO 16281. This method calculates the bearing loads and the service life of the bearing based on its inner geometry.

This method is available either as an embedded feature in shaft calculation (see Rolling Bearings (on page III-736)) or as a stand-alone KISSsoft module. Unless otherwise specified, this chapter describes the stand-alone module, because the majority of the functionalities and features are used by both these two variants.

This module is designed to be used by bearing experts, or users who know the internal geometry of their bearings. Using it, you can calculate the life of a bearing, if you know the loading conditions. To start this module, go to the KISSsoft modules tree and double-click on "Shafts and Bearings" -> "ISO 16281".

EHL lubricant film thickness and spin to roll ratio

The minimum EHL lubricant film thickness is used for rolling bearings with known inner geometry according to the method described in [96]. The Barus equation is used to take into account the effect of pressure on viscosity, as documented in the same reference document.

The spin to roll ratio of ball bearings is calculated on the basis of the equations in [96]. The assumption of “outer raceway control” is used, meaning that no spin of the ball is present on the outer raceway. It is known that this assumption is primarily valid in the case of high speed lightly loaded bearings. Ball gyroscopic motions and cage effects are not considered any further than that.

28.1 Bearing data tab

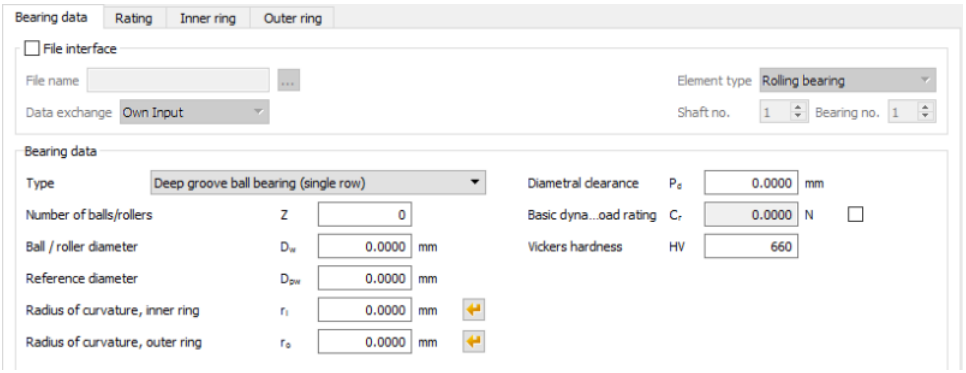


Figure 29.1 Bearing data tab

28.1.1 File interface

The user can use this module to link to a shaft calculation file. This enables bearing information to be transferred automatically from the shaft calculation file, without them having to reenter the data. The user must input the:

- File name: name of the shaft calculation file (extension .W10), from which the selected bearing data will be extracted
- Element type: Here, you select whether the bearing is a rolling bearing that belongs to a shaft, or a connecting rolling bearing
- Shaft number: If the bearing belongs to a shaft, you must input the shaft number here. The program then runs through the shafts Elements tree from top to bottom (see Figure 29.2)
- Bearing number: number of the selected bearing, either on the corresponding shaft or from the list of connecting elements. The program runs through the shafts Elements tree from top to bottom (see Figure 29.2)
- Data exchange: determines how data is exchanged between the shaft file and this module. In each case, the geometry of the selected bearing is transferred from the shaft file.
  - Bearing loads: the information transferred from the shaft file is the applied force and torque of the bearing as well as the lubricating conditions
  - Bearing displacements: the information transferred from the shaft file is the displacement and rotation of the inner ring of the bearing as well as the lubricating conditions

– Own input: only the geometry of the bearing is transferred. The user can determine their own load and lubrication conditions

Example:

a shaft file whose Elements tree has the structure shown in Figure 29.2. To extract the information for bearing "Rolling bearing 2" which belongs to "Shaft 1", the correct selection would be:

Element type = Rolling bearing, Shaft number = 1, Bearing number = 2

In contrast, to extract the information for bearing "Rolling bearing 1" from the list of connecting elements, the correct selection would be

Element type = Connecting Rolling bearing, Bearing number = 1

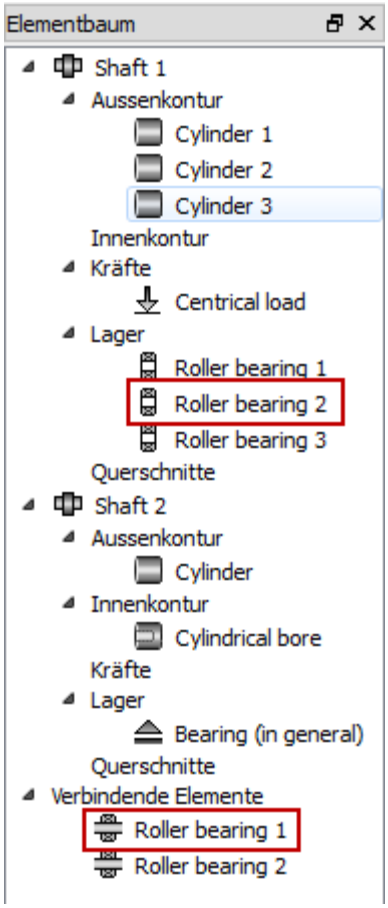


Figure 29.2 Example of how the shaft and bearing number are selected when a shaft file is linked.

28.1.2 Bearing data

This is where the geometry of the bearing is defined. You will find more detailed information about this in 24.2.3 Rolling bearing internal geometry (see section "Rolling bearing calculation with inner geometry" on page III-803). In addition to the geometry data, you can also specify the dynamic load rating, if you know it. If not, this rating is calculated using the current geometry data as specified in ISO 281. If you require a modified service life (see 24.6.1 Modified bearing service life (see section "Modified service life calculation in accordance with the Supplement to DIN ISO 281 (2007)" on page III-816)), input the fatigue load limit Cu. If Cu is not known, it will also be calculated on the basis of ISO 281.

Note for the shaft calculation: In this module, the effect of surface hardness on the static capacity can be taken into account by entering the Vickers hardness. You will find the formulae for this in [92]. The hardness value of every bearing calculated with their inner geometry is predefined as HV 660 for the shaft calculation.

28.1.2.1 User-defined roller profile

The default roller profile used for rolling bearings is the logarithmic profile as defined in ISO 16281. However, a user-defined roller profile can be used instead, if required. Click on the plus sign next to the roller length input field, and enter the name of the file with the required roller profile function (Figure 29.3a). The definition and coordinate frame of this file are shown in Figure 29.3b). The expected structure of this file is as follows:

-- this line is a comment

DATA

1	-0.45	0.000581256
2	-0.41	0.000390587
3	-0.37	0.000277616
4	-0.33	0.000200197
...		
...		
21	0.33	0.000200197
22	0.37	0.000277616
23	0.41	0.000390587
24	0.45	0.000581256

END

Notes:

- Lines that start with "--" are comments and are ignored.
- The profile function definition starts with the keyword DATA and ends with the keyword END
- Each line must contain three columns. The first column is the index. It is only included as a reference source for the user (its values have no effect, and will be ignored). The second column is the non-dimensional position x/Lwe, for which the profile is defined in mm/mm. The values in this column should range between -0.5 and +0.5. The third column is the non-dimensional profile f/Dw, in mm/mm. The values in this column cannot exceed 0.5.
- To save space, the data represented by "..." has been omitted

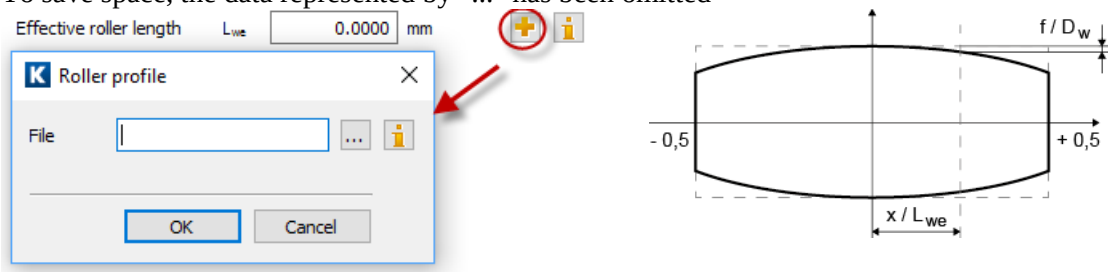


Figure 29.3 (a) Definition of a user-defined roller profile file, (b) Coordinate frame for defining the user-defined roller profile

28.1.2.2 Bearing ring deformations

The inside/outer rings are usually assumed to be rigid (non-deformable). To take ring deformations into account, click on the plus button next to the bearing type definition (figure 29.4a). The expected structure for both files is as follows:

-- this line is a comment

DATA

0	0	0.00E+00	0.00E+00	5.00E-03
1	8	0.00E+00	6.96E-04	4.95E-03
2	16	0.00E+00	1.38E-03	4.81E-03
3	24	0.00E+00	2.03E-03	4.57E-03

...

...

41 328 0.00E+00 -2.65E-03 4.24E-03

42.336 0.00E+00 -2.03E-03 4.57E-03

43.344 0.00E+00 -1.38E-03 4.81E-03

44.352 0.00E+00 -6.96E-04 4.95E-03

45.360 0.00E+00 -1.23E-18 5.00E-03

END

Notes:

- Lines that start with "--" are comments and are ignored.
- The ring deformation definition starts with the keyword DATA and ends with the keyword END
- Each row must contain 5 or 8 columns. The first column is the index. It is only included as a reference source for the user (its values have no effect, and will be ignored). The second column is the angle φ for which the deformation is specified. The next three columns are the X, Y, and Z components of the ring deformation, all defined in mm. If 8 columns are used, the last 3 columns represent the X, Y and Z components of the rim tilting, in rad.

Series	5 columns	8 columns
1	Index	Index
2	φ (°)	φ (°)
3	ux (mm)	ux (mm)
4	uy (mm)	uy (mm)
5	uz (mm)	uz (mm)
6	-	rx (rad)
7	-	ry (rad)
8	-	rz (rad)

To save space, the data represented by "..." has been omitted

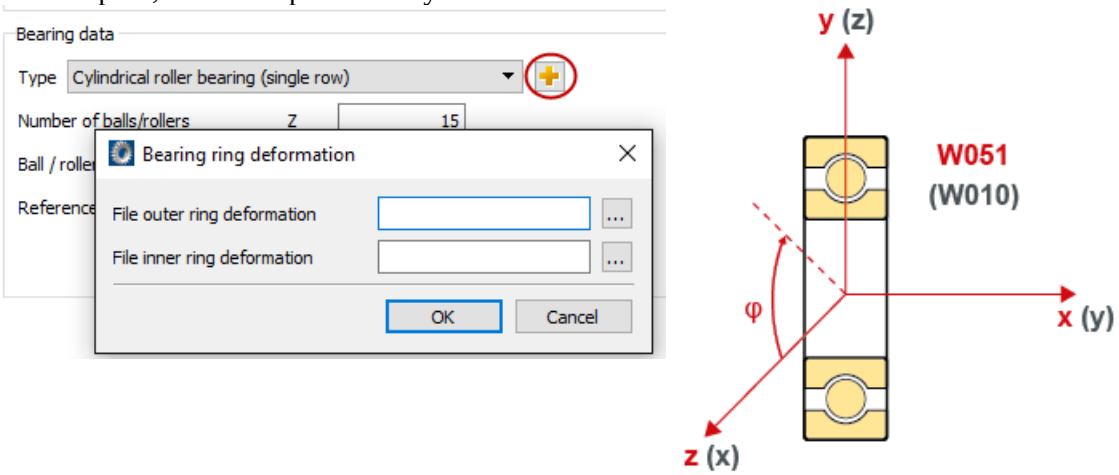


Figure 29.4 (a) Definition of ring deformations, (b) Coordinate frame of this module (W051), which defines the axial (x) and the radial directions (y, z). For the sake of clarity, the coordinate frame of the shaft module (W010) is also displayed.

Note for the shaft calculation: Ring deformations can only be processed in bearing calculation module W051, not in shaft calculation module W010.

28.2 Load tab

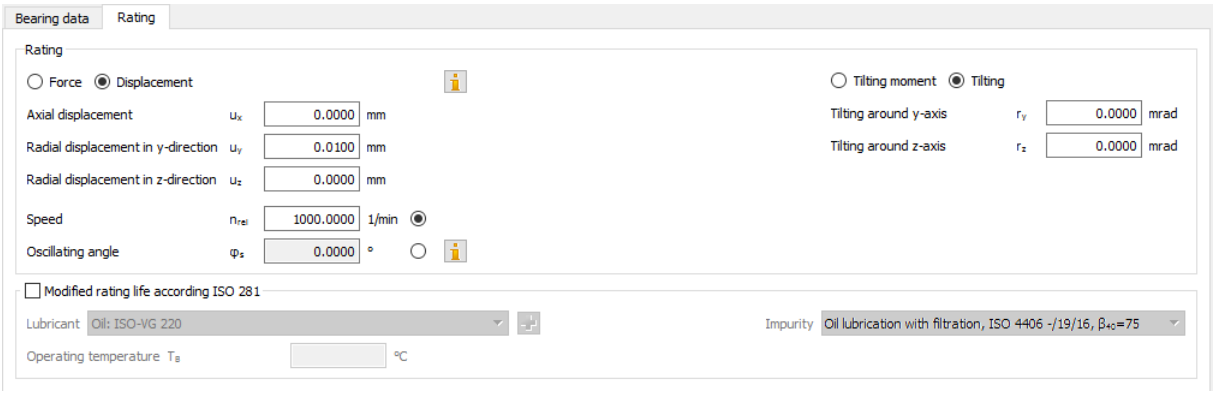


Figure 29.5 Load tab

This window is where you define the bearing's working conditions.

28.2.1 Rating

Four combinations of data can be entered here:

- (A) Force and Tilting moment
- (B) Force and Tilting
- (C) Displacement and Tilting moment
- (D) Displacement and Tilting

Speed: the speed of the inner ring relative to the outer ring. The outer ring is always assumed to be fixed (non-rotating).

Oscillation angle: the oscillation angle for partially rotating bearings. The service life in million oscillation cycles is determined according to [39].

Note for the shaft calculation: The default setting for the shaft calculation process is combination D.

Note: A complete oscillation is $2 \cdot \varphi_s$

28.2.2 Modified service life calculation according to ISO 281

The influence of lubrication, filtration and impurities on the service life can be taken into account here.

Lubricant: the lubricant used

Operating temperature: the temperature of the lubricant

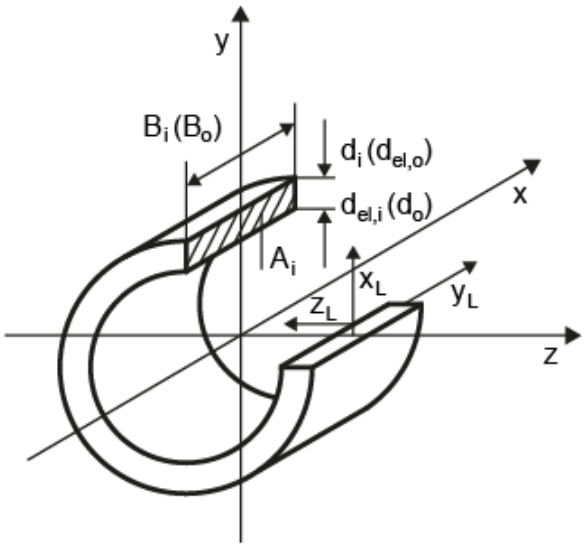
Impurity: the class of the impurity

28.3 Tab Elastic Rings

The calculation according to ISO/TS 16281 assumes that the bearing rings are rigid. However, in many cases the elasticity of the ring must be considered for realistic results. A typical application is the bearing which supports the planet gear in a planetary system; the ring is thin relative to its diameter, and therefore its elasticity cannot be neglected. The effect of the ring elasticity is that it typically increases the load zone and changes the load distribution within it, by “sharing” the maximum load to the neighboring rolling bodies. The calculation is based on the transfer matrix method [106 (see section "Bibliography" on page X-1214)].

28.3.1 Basic data

For this calculation, the ring geometry is needed. Unless you have selected `Own input`, the ring is assumed to be cylindrical, with a rectangular cross section.



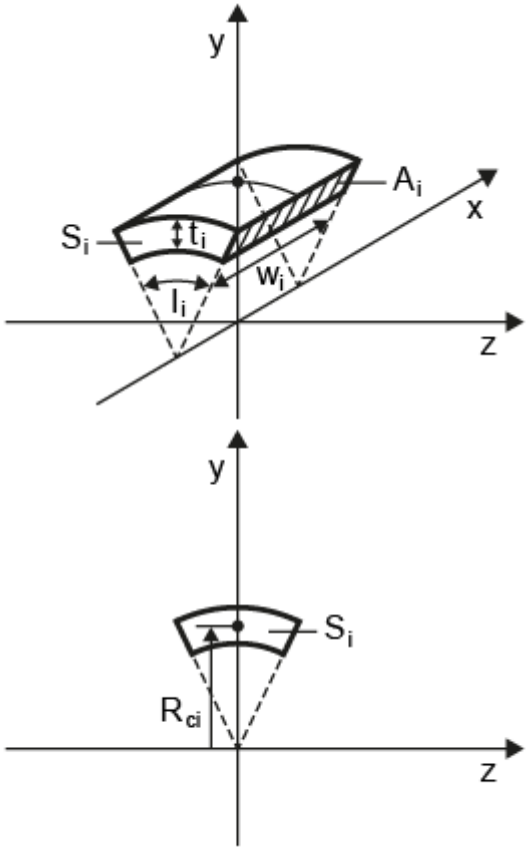
a. del, i is the inside diameter of the inner ring. The inner ring raceway diameter (di) is assumed to be the external diameter of the inner ring.

del,o is the external diameter of the outer ring. The outer ring raceway diameter (do) is assumed to be the inside diameter of the outer ring.

Bi is the inner ring width.

Bo is the outer ring width, which is usually not equal to Bi (for example, taper roller bearing).

If you have selected `Own Input`, the ring is no longer assumed to have a rectangular cross section, and you then need to input the geometric properties of its surface.



a. Ai is the cross-sectional area of the (inner) ring.

b. Jxi is the area moment of inertia (surface Si) around the X-axis, for an arc length li equal to the ring thickness ti. This is used to calculate the local bending of the ring in the transverse plane (normal to the radial plane).

Jyi is the area moment of inertia (surface Ai) around the y axis.

d. Jzi is the area moment of inertia (surface Ai) around the z axis.

e. Jpi is the polar moment of gyration of surface Ai around the local XL axis (normal to surface Ai.

f. Rci is the centroid radius of surface Si. It is essentially the radius of the center of gravity of surface Si on the plane yz.

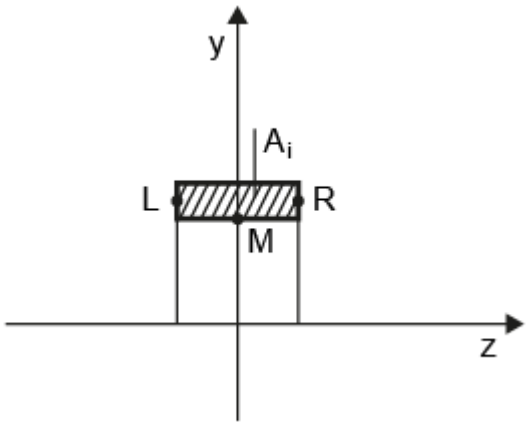
You can then determine the same variables for the outer ring by replacing each "i" subscript with "o".

28.3.2 Details

Here the boundary conditions which are applied on the elastic ring are defined. The constraints must be defined completely, to ensure that the solution for an elastic ring has the correct conditions applied to it (i. e. a specific degree of freedom (DOF) is free).

28.3.2.1 General constraints

The following boundary conditions are available, which act on predefined angular positions around the ring perimeter. Unless otherwise specified, all the coordinates refer to the global coordinate system.



a. Force: localized force and moment vector.

b. Distributed force: a force and moment loading which is applied on a given arc length starting at angle φ1 and finishes at angle φ2 (constant load distribution assumption). Since this is a distributed load, its components (X/Y/Z and RX/RY/RZ) are assumed to be constant relative to the **local** ring coordinates (XL/YL/ZL).

c. Elastic support: a spring support with constant spring stiffness.

d. Fixed support: a fixed support (on the ring mean line M) which blocks the corresponding DOF. A numerical value different than 0 in any component, means that this DOF is fixed. Example: (X, Y, Z, RX, RY, RZ) = (1, 1, 0, 1, 0, 0) means that ux, uy and rx are fixed, and the remaining DOFs are free.

e. Fixed support (left): a fixed support which blocks all DOFs, but instead of the ring mean line, the support is defined on the left side (L). A practical modeling case is a ring which is fixed on the shaft (or housing) with bolts.

f. Fixed support (right): similar to "Fixed support (left)", but the support is on the right side (R).

g. Deformation: Deformation: a given deformation and tilting value which is determined for the ring.

28.3.2.2 Gear contacts

A gear contact is internally identical to a general boundary condition for the "force" (a cylindrical gear is assumed). You must enter the following data:

a. αpos: the angular contact of the point of contact. This corresponds to "αpos" for a gear load element in the shaft calculation
b. mn: normal module
c. αn: nominal pressure angle
d. β: nominal helix angle (+: right-hand gear, -: left-hand gear)
e. a: center distance between gear and counter gear
f. z: number of teeth on the gear
g. zc: number of teeth on the counter gear
h. T/P (selection): select the torque or the power input
i. T/P (value): the numerical value of the load (+: driving gear, -: driven gear)
j. n: the speed of the gear around the ring's global X-axis (+: clockwise, -: counter-clockwise)

28.4 Graphics

The following graphics are provided:

1. Load distribution

This shows the load distribution over the rolling bearings (balls/rollers). For thrust bearings, the magnitude of the reaction force is used for the plot.

2. Deformation (elastic rings)

Shows the radial and axial deformation of the inner and outer ring.

3. Stress distribution on race

Shows the stress distribution on the inner and outer raceways.

4. Pressure curve

This shows how the pressure develops along the length of each roller, or at every contact point in a ball bearing.

5. Pressure curve for each rolling body

This graphic shows the pressure curve on each roller element along the roller profile.

6. Stiffness curve

This shows the force-displacement curve of the bearing. Both radial and axial stiffness are shown.

28.5 Fine sizing

After you have loaded the "Rolling bearing ISO/TS 16281" module, click on the Fine sizing option  to open a window in which you can perform fine sizing on the internal geometry of rolling bearings. In the same way as for fine sizing gears (see page II-439), you can vary the geometry parameters to generate a multitude of different bearing variants. The type you selected in the bearing data now determines which specific input parameters (for example, number of rolling bodies, radial clearance, osculation etc.) are available. Any existing bearing data for the bearing type is transferred directly to the fine sizing function.

You need a certain amount of experience before you can use the fine sizing function effectively. This is because the process may generate bearing variants whose internal geometry does not match either the currently applicable standards or rolling bearings that have actually been manufactured. The system does check some of the parameters you input, and displays warning messages when it encounters implausible data.

You input the external dimensions of the rolling bearing in the upper third of the first tab in the window. The middle third is where you input the parameters for the internal bearing geometry (according to the bearing type). In each case, the number of variants for the parameters are determined from the start values, the final values and the intervals (for example: If between 11 and 15 rolling bodies are involved, and the interval is set to two rolling bodies, the program calculates exactly three variants, if all the other intervals are set to zero).

The lower third of this tab is where you define the constraints that each of the rolling bearing variants the system generates has to fulfill. You can set the density of the rolling bodies to a number between 0% and 100% (if you set it to 100%, the bearing will have so many rolling bodies of this type that they will touch each other along the circumference). The minimum wall thicknesses you enter here apply to the inner and outer ring races.

Now, click on the "Calculate" button in the bottom area of the window to generate the bearing variants. The system now calculates the detailed inner geometry and the load rating and service life. In this case, the load data is taken from the "Load" tab in the "Rolling bearing ISO/TS 16281" module. A progress bar shows you how the calculation is progressing.

The bearing variants determined here are then displayed in a table in the "Results" tab in the Fine sizing window. These results can also be output in the report. The bearing variants are also displayed as a graphic in the "Graphic" tab (in a similar way to the results of the Gear fine sizing (see page II-448) function). Here you can

display the value of different parameters along the horizontal and vertical axes. You can also use a color scale to make the results easier to understand.

Chapter 29
Hydrodynamic Plain Radial Bearings

Niemann [64] provides a very accurate method for calculating plain radial bearings that can run at high speeds. This also produces excellent results for oval-clearance or tilting pad plain bearings.

ISO 7902 [33] or DIN 31652 include an excellent, detailed method for calculating stationary, hydrodynamic plain radial bearings that are to run at low to medium speeds. For those running at high speeds, use the equally excellent DIN 31657 [100].

29.1 Calculation methods

You can use one of these two methods to calculate oil-lubricated, hydrodynamic plain radial bearings:

a) According to G. Niemann, Maschinenelemente I, 1981, [64].
This method is very suitable for quickly rotating bearings. This also produces excellent results for special construction types such as tilting pad or oval-clearance plain bearings.
This method calculates the power loss, oil flow, oil temperature, and minimum lubricant gap thickness, according to [64] and [57]. This calculation can only be used for pressure lubricated bearings (circulatory lubrication) when the service reliability is also tested.

b) According to ISO 7902, Parts 1 to 3, 1998, 2013 [33].
This method is very suitable for slowly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance. Complete calculation according to ISO 7902, Parts 1 to 3 (1998 and 2013 Editions) for pressure-less and pressure-lubricated bearings. This takes into account the way in which lubricant is applied (lubrication holes, lubrication groove, lubrication glands). It calculates all the operating data as defined in ISO 7902, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.

c) According to DIN 31652, Parts 1 to 3, 2015, [103].
This method is very suitable for slowly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance. Calculation according to DIN 31652, parts 1 to 3 (2015 edition) for pressure-less and pressure-lubricated bearings. This takes into account the way in which lubricant is applied (lubrication holes, lubrication groove, lubrication glands). It calculates all the operating data according to DIN 31652, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.

d) According to DIN 31657, Part 1-4, 1996, [100].
This method is very suitable for quickly rotating bearings. It also determines the oil consumption, the oil flow and the entire heat balance. The calculation is suitable for multi-lobed plain bearings and tilting pad plain bearings.
Complete calculation according to DIN 31657, Parts 1 to 4 (1996 Edition) for pressure lubricated bearings. It calculates all the operating data according to DIN 31657, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.

29.2 Module-specific entries

Calculating the volume-specific heat c of the lubricant.

The volume-specific heat of lubricants can be calculated in two ways:

- `Take into account dependence on temperature`
- `Simplified assumption (as in ISO 7902/DIN 31657):` 1.8 . 10^6J/(m^3K)

If the "Take into account dependence on temperature" option has been selected, the specific heat capacity of the lubricant can also be specified, if it is known. For example, you must overwrite this value if you want to perform a calculation for a water-lubricated plain bearing, otherwise you will get incorrect results.

Run calculation with critical Reynolds number

If this flag is set, the calculation continues if the error message involving calculation using the critical Reynolds number is displayed (transition is laminar and turbulent). Otherwise, the calculation is canceled.

29.3 Thermal expansion coefficients

To calculate the clearance, you require the thermal expansion coefficients of the shaft and (wheel or pinion) center.

These are the coefficients for the most important materials:

Steel	$11.5 \cdot 10^{-6}$
Cast iron	$11 \cdot 10^{-6}$
White metal	$18 \cdot 10^{-6}$
Composite bronze	$18 \cdot 10^{-6}$

29.4 Average surface pressure

You will find the permitted values in:

- Niemann, Volume I, Table 15/1, [64]
- ISO 7902, Part 3, Table 2, [33]
- DIN 31657, Part 4, Table 1, [100]

Permitted maximum values for the surface pressure, depending on operating temperature (ISO 7902/DIN 31652):

- Pb and Sn alloys: 5 (15) N/mm^2
- Cu Pb alloys: 7 (20) N/mm^2
- Cu-Sn alloys: 7 (25) N/mm^2
- Al Sn alloys: 7 (18) N/mm^2
- Al Zn alloys: 7 (20) N/mm^2

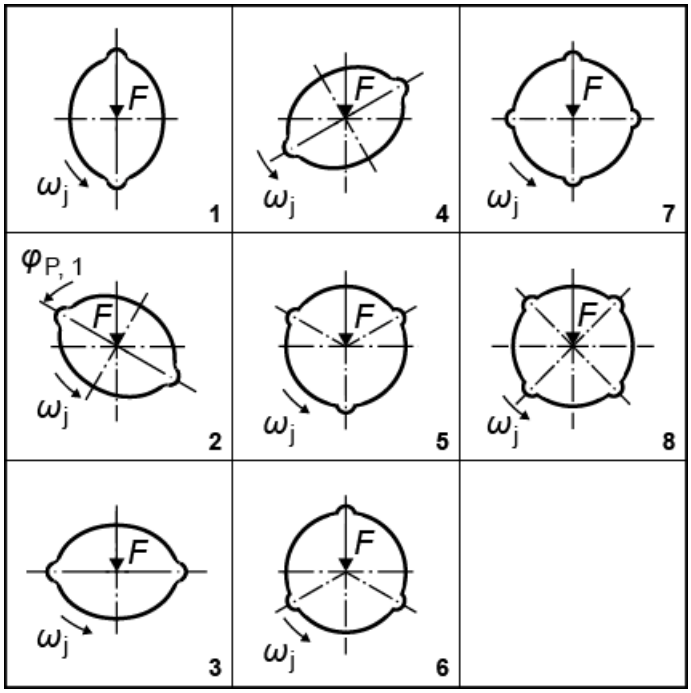
the values shown in brackets were recorded under special working conditions.

Permitted maximum values for the surface pressure, depending on operating temperature (DIN 31657):

- Lead alloys: 16 to 25 N/mm^2
- Tin alloys: 25 to 40 N/mm^2
- Copper alloys (bronzes): 25 to 50 N/mm^2

29.5 Geometries according to DIN 31657

Different load cases and arrangements of the multi-lobed plain bearings, as shown in DIN 31657-2 and present in the tables.



Arrangements of multi-lobed plain bearings

1) Z=2;Ω=150°;φP,1=180°;h*0,max=3,5;B/D=0.75

2) Z=2;Ω=150°;φP,1=240°;h*0,max=3,5;B/D=0.75

3) Z=2;Ω=150°;φP,1=270°;h*0,max=1,3,5;B/D=0.5,0.75,1

4) Z=2;Ω=150°;φP,1=300°;h*0,max=3,5;B/D=0.75

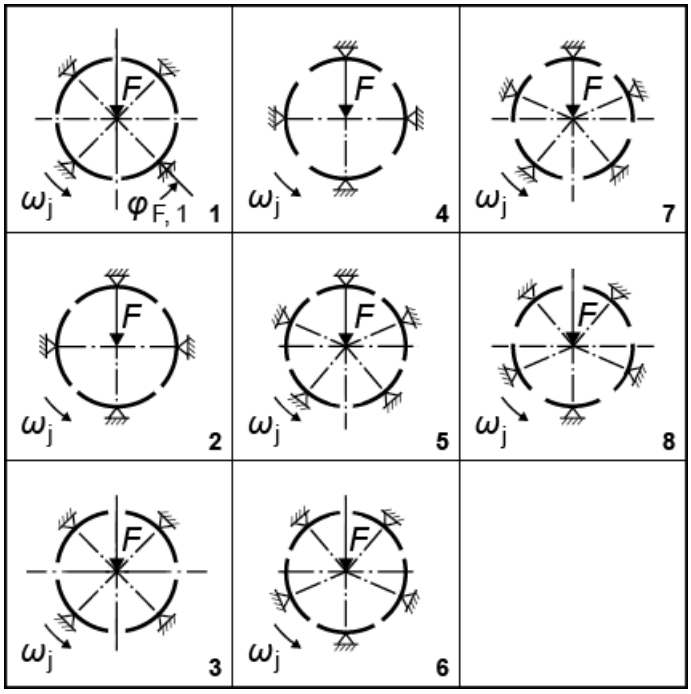
5) Z=3;Ω=100°;φP,1=240°;h*0,max=3,5;B/D=0.75

6) Z=3;Ω=100°;φP,1=300°;h*0,max=1,3,5;B/D=0.5,0.75,1

7) Z=4;Ω=70°;φP,1=270°;h*0,max=3,5;B/D=0.75

8) Z=4;Ω=70°;φP,1=270°;h*0,max=1,2,3,4,5;B/D=0.5,0.75,1

Different load cases and arrangements of the tilting pad plain bearings, as shown in DIN 31657-3 and present in the tables.



Arrangements of tilting pad plain bearings

1) Z=4;Ω=80°;φF,1=45°;h*0,max=2,3,5;B/D=0.5,0.75,1

2) Z=4;Ω=80°;φF,1=0°;h*0,max=3;B/D=0.75

3) Z=4;Ω=60°;φF,1=45°;h*0,max=2,3,5;B/D=0.5,0.75

4) Z=4;Ω=60°;φF,1=0°;h*0,max=3;B/D=0.5

5) Z=5;Ω=60°;φF,1=36°;h*0,max=2,3,5;B/D=0.5,0.75

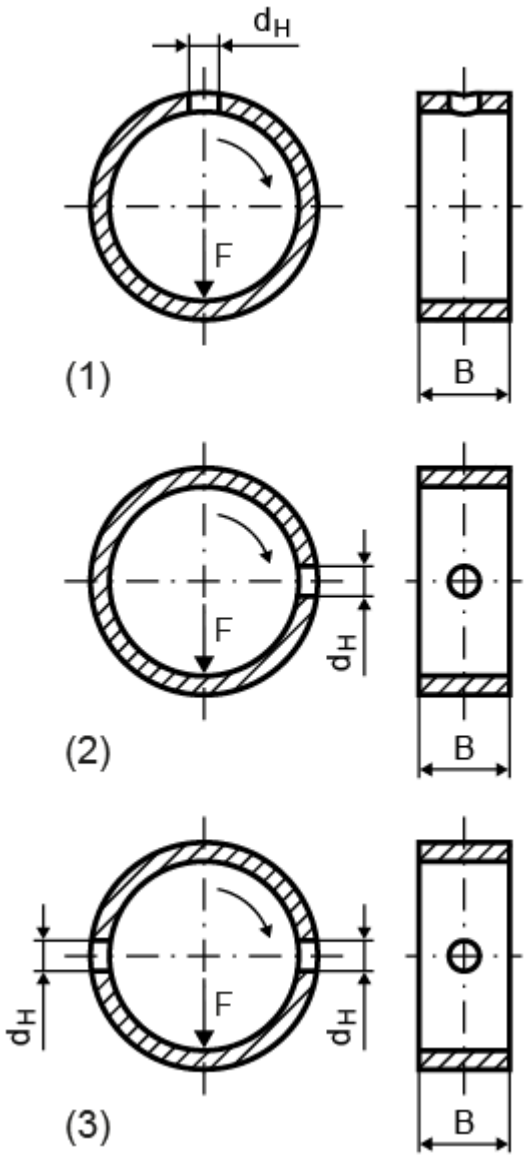
6) Z=5;Ω=60°;φF,1=0°;h*0,max=3;B/D=0.5

7) Z=5;Ω=45°;φF,1=36°;h*0,max=2,3,5;B/D=0.5

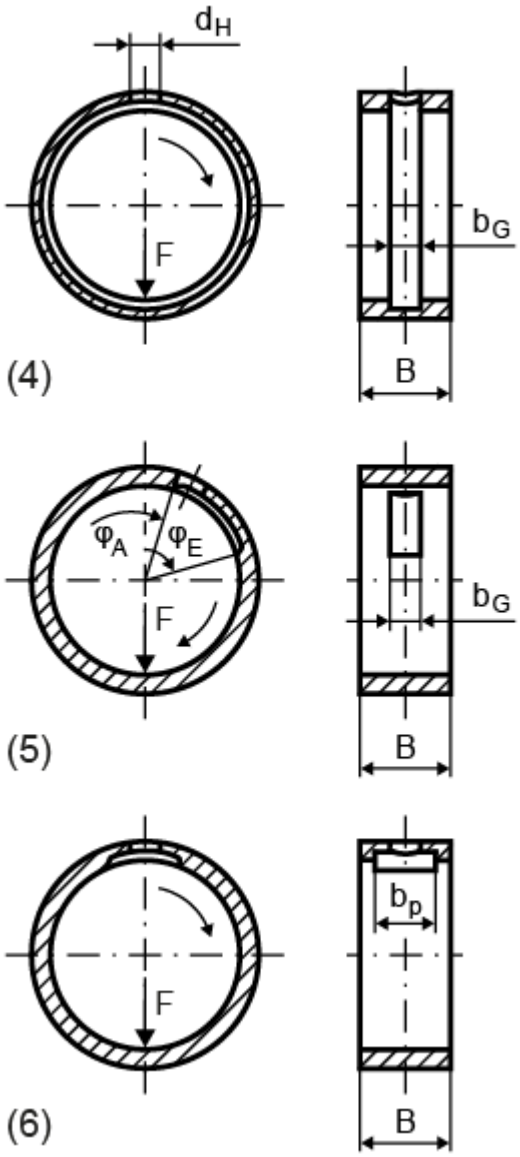
8) Z=5;Ω=45°;φF,1=0°;h*0,max=3;B/D=0.5

29.6 Lubrication arrangement

The different lubrication arrangements are shown in the next three figures.

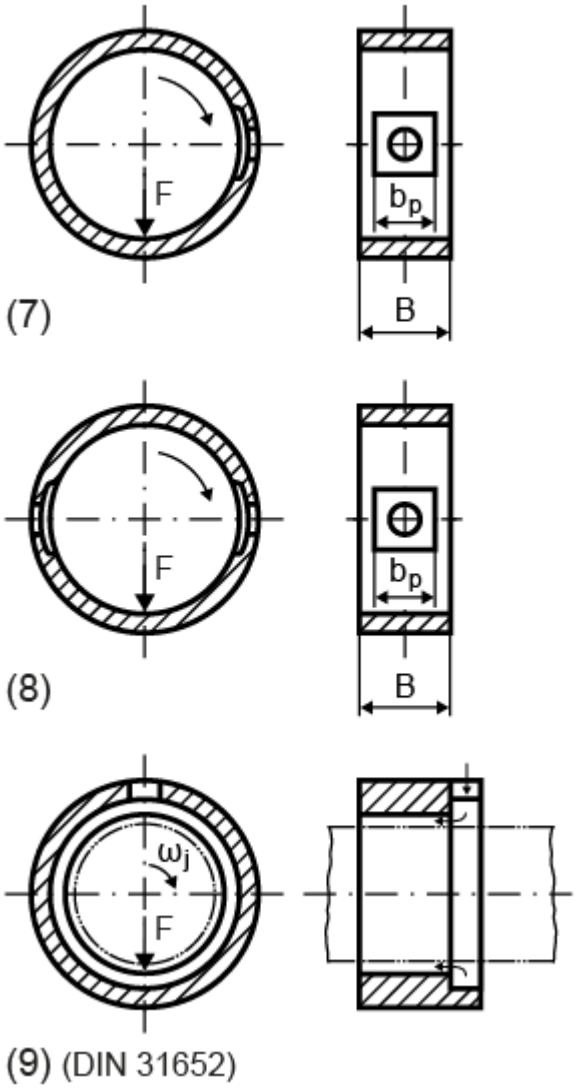


1: One lubrication hole opposite to load direction.
2: One lubrication hole positioned at 90° to the load direction.
3: Two lubrication holes positioned at 90° to the load direction.



4: Lubrication groove (ring groove).
5: Lubrication groove (circumferential groove).
Note: For lubrication with a circular groove, the calculation is performed for each bearing half with half the load! (see ISO 7902, Part 1, paragraph 3.4 [33]).

6: One lubrication pocket opposite to load direction.



7: One lubrication pocket positioned at 90° to the load direction.
8: Two lubrication pockets positioned at 90° to the load direction.
9: From one bearing edge across the entire perimeter of the bearing (only Draft DIN 31652)

29.7 Heat transfer coefficient

If the value of the heat transfer coefficient is not known, you can take 15 to 20 (W/m^2K) as a guide value.

29.8 Heat transfer surface

If the values of the heat transfer surface are not known, you can take 10 * d ***** b to 20 ***** d ***** b as a guide value. This value is only needed if heat is lost due to convection.

d : Bearing diameter
b : Bearing width

29.9 Oil temperatures

Oil exit temperature:

- Normally approximately 60°
- Upper limit for usual mineral oils: 70° to 90°

Oil inlet temperature:

- With the usual cooler: 10°C lower than the output temperature
- With a very efficient cooler: 20°C lower than the output temperature

29.10 Mixture factor

The mixture factor that is used for the calculation according to DIN 31657 should lie between 0.4 and 0.6.

If the mixture factor M=0, this would mean that there is no mixture in the lubrication pockets, or that the exiting lubrication flow rate Q2 flows entirely into the next lubrication gap.

If the mixture factor M=1, this would mean complete mixture in the lubrication pockets.

29.11 Sizing the bearing clearance

Bearing clearance = d_bore - d_shaft

In general, although a greater bearing clearance makes the bearing more stable, and enables it to cool more effectively, it also results in a reduction in load capacity.

- **Suggestion according to Niemann**
  Suggestion for metal bearings in mechanical engineering according to Niemann, Volume I, Table. 15/2, [64].

  The following applies for other materials:

Cast iron bearing	:	0.001 * d
Light metal bearing	:	0.0013 * d
Sintered bearing	:	0.0015 * d
Plastic bearing	:	0.003 * d

  d : Bearing diameter

- **Proposal according to ISO 7902**
  Proposal for metal bearings in mechanical engineering according to ISO 7902, Part 3, Table 4, [33].
  In this sizing method you can either use the proposal according to ISO 7902, or calculate the clearance from a predefined output temperature (only where the lubricant is used to dissipate the heat).

- **Proposal according to DIN 31652**
  Proposal for metal bearings in mechanical engineering according to ISO 7902, Part 3, Table 4, [33], as the DIN standard contains no proposed value.
  In this sizing method, you can either use the proposal according to ISO 7902,

or calculate the clearance from a predefined output temperature (only where the lubricant is used to dissipate the heat).

- **Suggestion according to DIN 31657**
  Suggestion for plain bearings in mechanical engineering according to DIN 31657, Part 4, [100].
  In this sizing method you can either use the proposal according to DIN 31657 or calculate the clearance from the entered output temperature.

- **Proposal according to K. Spiegel**
  Proposal for clearance according to K. Spiegel: Goettner equation
  clearance: (2.5+50.0/d)/1000.0*d

29.12 Sommerfeld number

You must calculate the Sommerfeld number because it is an important characteristic value for plain bearings.

Sommerfeld number > 1 occurs in heavily loaded bearings at the limit for b/d: $0 < b/d \leq 2$

Sommerfeld number < 1 occurs in quickly rotating bearings at the limit for b/d: $0.5 < d/b \leq 2$

d : Bearing diameter
b : Bearing width

29.13 Bearing width

Reference value for bearing width as defined in Niemann, Volume I, table. 15/1, [64]
Normal range: b/d = 1 to 2

Reference value for bearing width according to ISO 7902, [33]
Normal range: b/d = 0.125 to 1

Reference value for bearing width in accordance with DIN 31652, [103]
Normal range: b/d = 0.01 to 5

Reference value for bearing width according to DIN 31657, [33]
Normal range: b/d < = 0.8, b/d = 0.8 is assumed for the calculation of Qp

d : Bearing diameter
b : Bearing width

29.14 permissible lubricant film thickness

The suggested value is taken from ISO 7902, Part 3, Table 1, [[33] or DIN 31652, Part 3, Table 1, [[103].

The values in this table are all empirical values. They therefore assume an average roughness height of $< 4\mu$, low levels of form error, and that the lubricant is to be filtered appropriately.

For DIN 31657, the proposed value according to Niemann is taken, as there is no value present in this standard.

A value is determined according to the diameter, in the case of the proposed value according to Niemann.

Chapter 30
Hydrodynamic Plain Thrust Bearings

The DIN standard provides two methods for calculating hydrodynamic plain thrust bearings, according to type.

- Calculation of pad thrust bearings according to DIN 31653 [34]: This standard applies to bearings that have fixed sunken surfaces for lubrication (see Figure 30.2) which are separated from the rotating discs by a film of lubricant.
- Calculation of tilting-pad thrust bearings according to ISO 12130, DIN 31654 [35]: This standard applies to bearings that have movable tilting pads (see Figure 30.3) which are also separated from the rotating discs by a film of lubricant.

If you do not consider the influence of the center of pressure on the tilting-pad thrust bearings, the same calculation procedure is described in both standards, which is why it is described here only once. However, any significant variations to these two standards will get a special mention here.

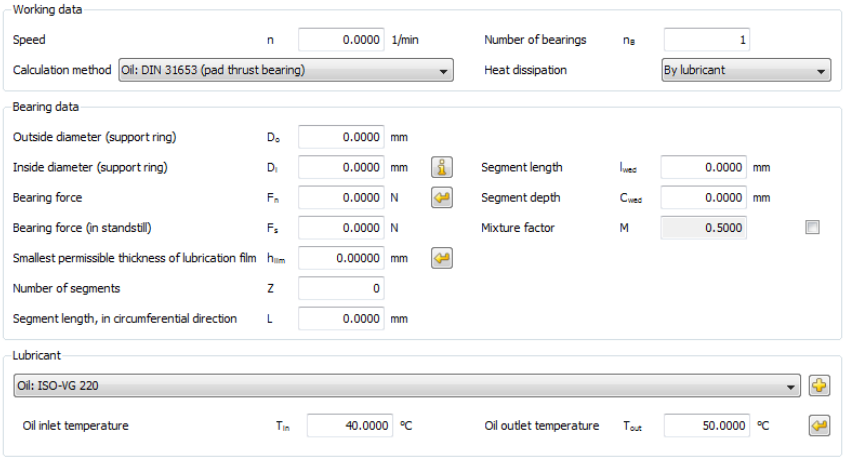


Figure 30.1: Basic data: Plain thrust bearing

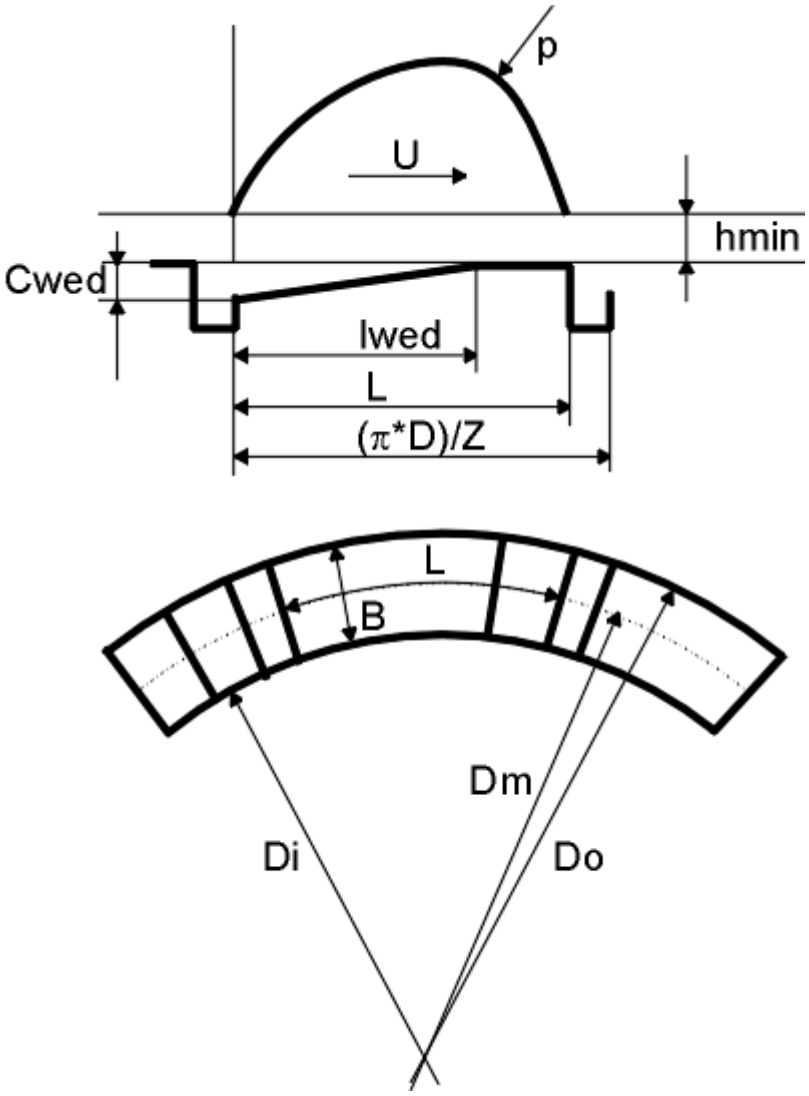


Figure 30.2: Pad thrust bearings as described in DIN 31653

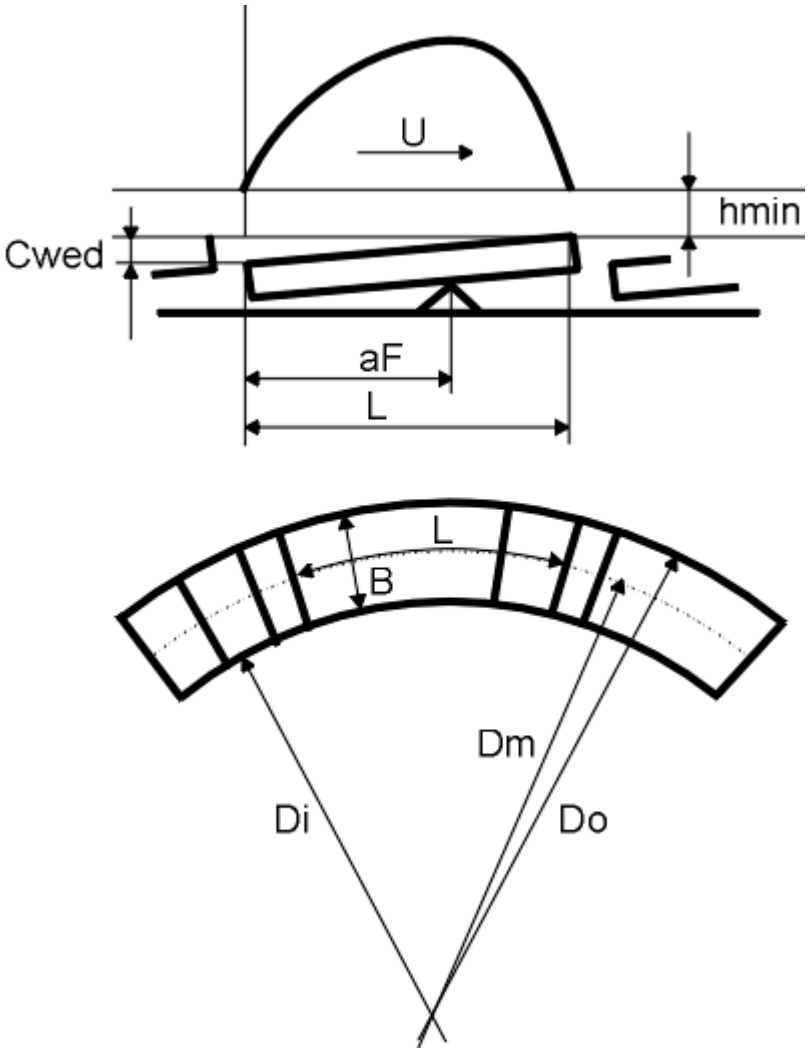


Figure 30.3: Tilting-pad thrust bearings according to ISO 12130, DIN 31654

30.1 Calculation

Essentially, both calculation procedures are based on the equation used to ascertain the thermal balance in the bearing. You can use either convection or circulatory lubrication in this calculation.

- Non pressure lubricated bearings (self-lubricating) dissipate heat out to the surrounding environment by convection. According to the standard, the coefficient of thermal expansion kA lies between 15 . . In the program the default value is 20 W/(m2*K), but you can change this as required.
- Pressure lubricated bearings mainly dissipate heat through the lubricant. Here you need to define a mixture factor M which must lie between 0 . . 1. Experience shows that this usually lies between 0.4 and 0.6. The default setting is 0.5, but this can be changed.

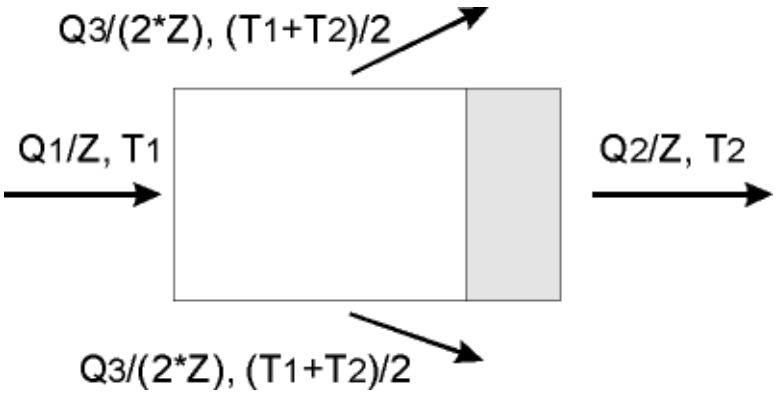


Figure 30.4: Segment lubricant and heat levels

These calculations provide values for the friction power, the lowest lubricant film thickness and the operating temperature. For circulatory lubrication, they also calculate the lubricant flow rate.

The bearing force (in standstill) is only used to determine the lowest admissible lubricant film thickness, and is otherwise irrelevant. The value of the load coefficient, the friction coefficient, and the lubrication flow rates are calculated according to the formulas (not according to the diagram or table) stated in DIN 31653/ 31654 Part 2. For tilting-pad thrust bearings, the ratio hmin/Cwed is calculated from the support position of the tilting-pad aF*. The formula for this is given in ISO 12130 Part 2 or in DIN 31654 Part 2.

30.2 Sizings

You can also calculate the bearing force (nominal load), but before you can do this, you must enter all the other input values. The bearing force is then calculated using the value you specified for the lowest lubricant film thickness, hlim.

The lowest possible lubricant film thickness hlim can be calculated according to DIN 31653 or 31654. This lowest possible lubricant film thickness is calculated depending on sliding velocity, average diameter and loading, according to the formula.

For convection:
can use a formula for approximation as defined in the standard:
A = (15 to 20) * B * L * Z
Click the  button next to the surface value input field to calculate this value using the formula A = 15 *B *L*Z.

For circulatory lubrication:
Experience shows that the exit temperature is between 10 and 30 K higher than the entry temperature. Click the  button next to the exit temperature to calculate a default value with a 10 K temperature difference.

30.3 Calculation of volume-specific heat

In `Calculations/Settings` there are two methods you can use to calculate volume-specific heat:

- `Take into account dependence on temperature`
- `Simplified assumption (as in ISO 7902):` $1.8 * 10^6 J/(m3 * K)$

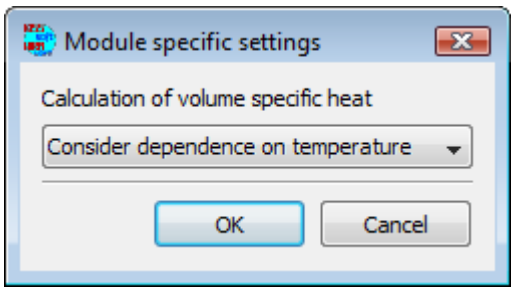


Figure 30.5: Module specific settings

30.4 Limiting values in the calculation

The standards only apply to laminar flow in the lubrication gap. For this to happen, the Reynold number must lie below the critical value of 600.
These results are also checked for highest permissible bearing temperature, Tlim, the smallest possible lubricant film thickness, hlim, and the specific bearing load. These limiting values are defined in DIN 31653/ISO 12130/DIN 31654 Part 3.

Chapter 31
Answers to Frequently Asked Questions

31.1 Intersecting notch effects

If at all possible, notch effects - for example in a shoulder with an interference fit - should not be overlapped when the shaft is designed. However, if this does happen, in the worst case scenario, the FKM Guideline should be applied to calculate the overall notch effect coefficient K_f:

$$K_f = 1 + (K_{f1} - 1) + (K_{f2} - 1)$$

from part notch effect coefficients K_{f1} and K_{f2}. In KISSsoft, this situation can be resolved by selecting `Own input` for the notch effect (see page III-778) of a free cross section (see page III-730).

The overall notch effect coefficient can then be calculated as follows:

1. Two cross sections (for example, A-A and B-B) are defined with the same Y-coordinate.
2. Cross section A-A is calculated by selecting notch type Kf1 (for example, shoulder). The notch factors are displayed directly in the elements editor (see page III-705).
3. The process described in 2. is then repeated for cross section B-B.
4. The resulting notch factors for both these notches are noted down, and the notch factors Kf are calculated in accordance with the formula given above.
5. Now, both cross sections (A-A and B-B) are deleted, and a new free cross section C-C with the same Y-coordinate is added. Open the Elements editor and select `Own Input` for the notch effect. Then, enter the overall notch effect coefficients calculated in point 4.

31.2 Notch effects on hollow shafts

All the notch factors described in the standards have been determined for solid shafts. No data is available for hollow shafts. KISSsoft calculates the nominal stresses for hollow shafts using the section modulus and taking into account the inside diameter.

31.2.1 Notches on the outer contour

For "small" inside diameters, the error due to calculating notch effect values for solid shafts is relatively small and you can use the results as approximations. However, when "large" inside diameters are involved, you must correct the notch effect values.

According to the FKM Guideline of 1998, you cannot accurately calculate the notch effect values of a round shaft that has a longitudinal bore for bending and tension/compression using the notch effect values of a round solid shaft. You should use the notch effect value of a round solid shaft for torsion and round shafts that have a circumferential notch, shoulder or conus, but use this value with nominal stress for a round shaft that has a longitudinal bore.

31.2.2 Notches on the inner contour

You cannot use these calculation methods to determine the notch factors of notches on the inner contour.

31.3 Fatigue Limits for New Materials

If you want to add a new material to the database, you must enter its endurance limits as well as the yield point and tensile strength.

Hänchen gives

$$\sigma_{bW} \approx 0.495 \cdot \sigma_B$$

as an approximation of the bending fatigue limit, and also other approximations from different sources. For the tension/compression fatigue limit, this states

$$\sigma_{zdW} \approx 0.45 \cdot \sigma_B \quad oder \quad \sigma_{zdW} = 0.5 \ldots 0.6 \cdot \sigma_B$$

, and for the torsion fatigue limit it states.

$$\tau_{tW} \approx 0.55 \cdot \sigma_{bW} \quad oder \quad \tau_{tW} = 0.4 \ldots 0.5 \cdot \sigma_{bW}$$

.

According to DIN 743, the following approximations can be made:

$$\sigma_{bW} \approx 0.5 \cdot \sigma_B \quad \sigma_{zdW} \approx 0.4 \cdot \sigma_B \quad \tau_{tW} \approx 0.3 \cdot \sigma_B$$

For through hardened steels (there can be different values for other material types), the FKM Guideline proposes:

$$\sigma_{W,zd} \approx 0.45 \cdot R_m \quad \tau_{W,s} \approx 0.26 \cdot R_m$$

31.4 Taking double helical gearing into account in the shaft calculation

In the shaft analysis process, when you input cylindrical gear data in "Hand of gear", you can select double helical gearing from the drop-down list. A gear with this characteristic always has an axial force 0 N. When double helical gears are transferred from the gear calculation (checkbox Read data from file active) the total width (= left side + gap + right side) is also transferred as is the total power. The shaft analysis then takes both the gap and the effective toothing into account. This generally results in a very useful model.

If you require a more precise model, input the two halves of the gear individually, one inclined to the right and the other inclined to the left. Unfortunately, you cannot do this by transferring the data directly from the gear calculation.

Part IV
Connections

Chapter 32
Cylindrical Interference Fits

- This calculation includes the entirety of the DIN 7190 standard (elastics) with longitudinal, radial and oil interference fits.
- Load in circumferential and axial directions.
- Load with bending moment and radial force.
- Calculating the maximum torque for a non-slipping fit. If slip occurs in the fit, micro gliding will cause corrosion due to friction.
- Influence of centrifugal force.
- Verification of an elastic-plastic loaded interference fit as specified in DIN 7190 with predefined interference (stresses and elongations are calculated only for the elastic case)
- Analysis of hubs with multiple interference fits
- Display stress curves (equivalent, tangential and radial stresses)
- Display tolerance fields:
  - only take into account allowances
  - take into account temperature and centrifugal force (without pressure)
  - take into account temperature, centrifugal force and pressure

You can calculate the safety of the interference fit against sliding and the safety of the shaft material and the hub against fracture and yield point. The calculation also takes into account the effect of centrifugal force on the expansion of the interference fit and on the stress in the shaft and hub. The tolerance system specified in DIN 7151 (e.g. with diameter input 60 H7/f6) has been implemented to make it easier to input data. You can either enter the tolerance manually, or use an automatic option to calculate the tolerance pairing based on the required safety against sliding and the permitted material stress. Input values for surface roughness with qualities as defined in ISO 1302.

Calculating the pressure: For elastic materials according to the theory of mechanics, for thick cylinders under internal pressure and thick cylinders under external pressure (e.g. [60], page 399, or [64]).

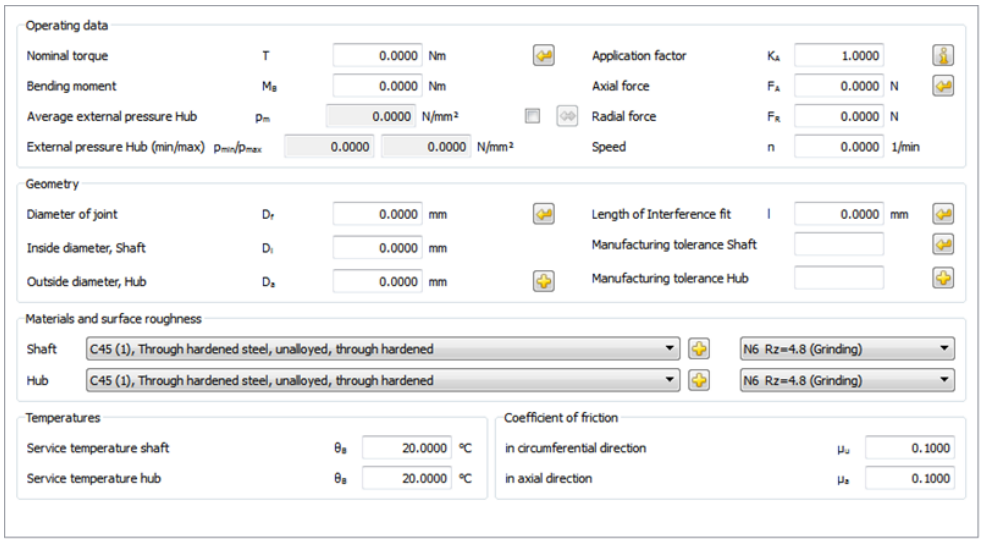


Figure 32.1: Basic data: cylindrical interference fit

Influence of speed: according to the theory of a cylinder in rotation ([38], page 219)

Amount of embedding: as defined in DIN 7190.

Equivalent stress:
you can change the hypothesis of equivalent stress in Calculations/Settings. The Settings chapter provides more information about this function.

Bending moment and radial force: this takes into account the effect of a bending moment and a radial force on the pressure. The additional amount of pressure is calculated as follows:

$$p_b = \frac{9}{2}\frac{M_b}{d_F l_f^{\,2}} \qquad p_r = \frac{F_r}{d_F l_F} \tag{32.1}$$

To ensure no gaping occurs between the hub and the shaft, the additional pressure must be less than the minimum interference pressure on the connection ($(p_b + p_r) < p_{min}$).

These formulae have been taken from the technical literature according to Prof. Hartmann, "Berechnung und Auslegung elastischer Pressverbindungen", pages 6 to 7.

Other values:

Dismounting force	Specified in [64], page 363
Transmission without slip	Specified in [55], equations 2.93 to 2.110

Micro sliding: If the torque of an interference fit is increased continuously until it exceeds the micro sliding limit, a local slip will occur at the position to which the torque is applied. As torque decreases continuously in the interference fit, the slip occurs only in one part of the fit, even if the torque then increases again. This effect is called micro gliding (shaft moving back and forth in the hub), and it can cause friction rust. Further explanations and details about this calculation are given in "Welle-Nabe-Verbindungen (Shaft-Hub Connections)", by Kollmann [55].
Comment about the calculation as defined by Kollmann:
To limit torque for micro sliding, use equation 2.110, with k use equation 2.107 and with ιr use equation 2.93.

Mounting: You will find details about mounting in the report. The temperature difference for mounting is calculated in such a way that, even if the maximum interference is reached (worst case scenario) there will still be enough play in the joint. Define the mounting clearance under `Settings`. Here the values for mounting the shaft at ambient temperature and for a cooled shaft (shaft at approximately -150°C) are calculated.

Verification of an elastic-plastic loaded interference fit according to DIN 7190:

Prerequisites: EI = EA, nyI = nyA, n = 0, diI = 0

If all the prerequisites are fulfilled, as defined in DIN 7190, the plasticity diameter DPA of the external component that is to be mounted can be calculated (diameter at which the plastic range ends). The corresponding compacting pressure, and the relationship between the ring surface qpA and the overall cross section qA, are also calculated. (Experiential limit according to DIN 7190 for heavily loaded interference fits in mechanical engineering qpA/qA <= 0.3)

32.1 Inputting Tolerances

Tolerances according to ISO/DIN:

To enter tolerances in the same way as, for example 60 H7/f6, you must:

- Enter 60 (mm) as the joint diameter.
- Enter H7 and f6 in the `Tolerances` fields. Here, the first field is where you enter the value for the wheel or hub and the second is where you enter the value for the shaft.
- The program checks automatically to see whether the tolerances you specified actually exist, and if you entered the data in the correct format!

Define own tolerances

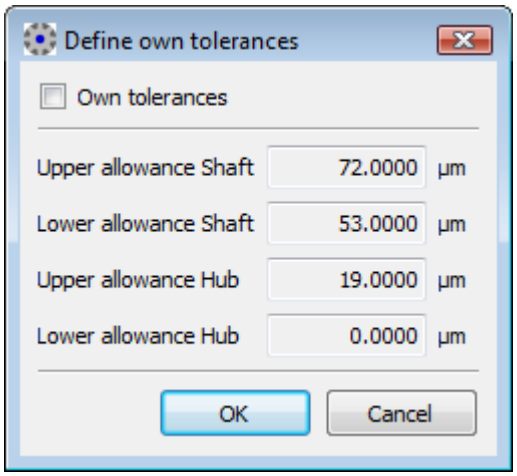


Figure 32.2: Tolerance values

Click the  button next to the `Tolerances` field to display the current allowances. You can then change them as required.

32.2 Coefficient of friction

Tables 32.2 and 32.4 contain coefficient of friction values as defined in DIN 7190.

Materials	**Coefficient of friction**			
	dry		lubricated	
	ν_{ll}	ν_{rl}	ν_{ll}	ν_{rl}
E 335	0.11	0.08	0.08	0.07
GE 300	0.11	0.08	0.08	0.07
S 235JRG2	0.10	0.09	0.07	0.06
EN-GJL-250	0.12	0.11	0.06	0.05
EN-GJS-600-3	0.10	0.09	0.06	0.05
EN-AB-44000 and following	0.07	0.06	0.05	0.04
CB495K	0.07	0.06	-	-
TiAl6V4	-	-	0.05	-
ν_{ll}: in axial direction - loosen				
ν_{rl}: lengthwise - sliding				

Table 32.2: Coefficient of friction for longitudinal fits under continuous stress as defined in DIN 7190

Material pairs, Lubrication, Joining	**Coefficients of friction ν_r,ν_{rl},ν_u**
Steel-steel pairing:	
Oil pressure connection normally joined with mineral oil	0.12
Oil pressure connection with degreased contact surfaces	0.18
joined with glycerin	
Shrink fit normally after warming the	0.14
External component up to 300°C in electrical oven	
Shrink fit with degreased contact surfaces after	0.20
heating in electrical ovens up to 300°C	
Steel-cast iron pairing:	
Oil pressure connection normally joined with mineral oil	0.10
Oil pressure connection with degreased contact surfaces	0.16
Steel-MgAl pairing, dry	0.10 to 0.15

Steel-CuZn pairing, dry	0.17 to 0.25
ν_r: Slipping	
ν_{rl}: lengthwise - sliding	
ν_u: in peripheral direction	

Table 32.4: Coefficients of friction for radial interference fits in longitudinal and peripheral direction subjected to sliding as defined in DIN 7190

32.3 Variable external diameter of the hub

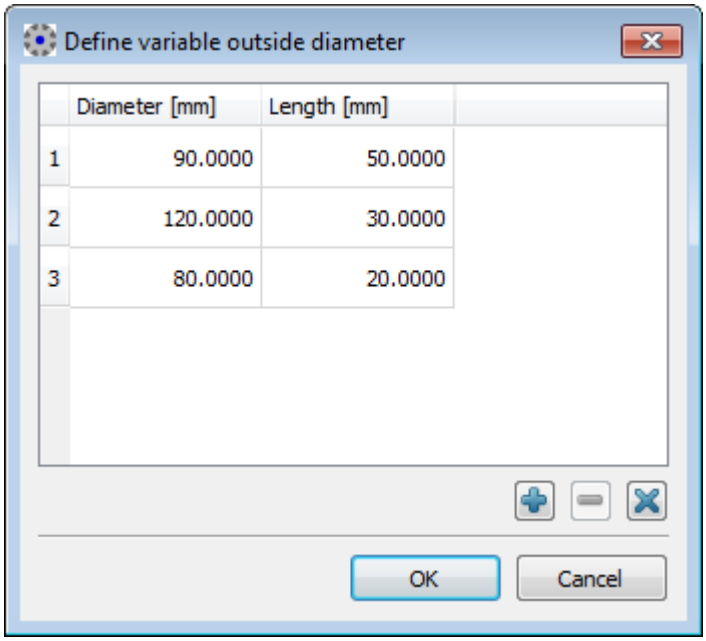


Figure 32.3: Variable external diameter

For a stepped hub out diameter, a single equivalent diameter and length are determined. These values are then used to calculate the stiffness of the external component.

32.4 Convert external pressure with multiple interference fit

The effect of outside pressure on the first hub can be taken into account for a multiple interference fit. This pressure is defined through a series of sequential interference fits, as shown in the following figure, and is calculated by the software (no direct user input).

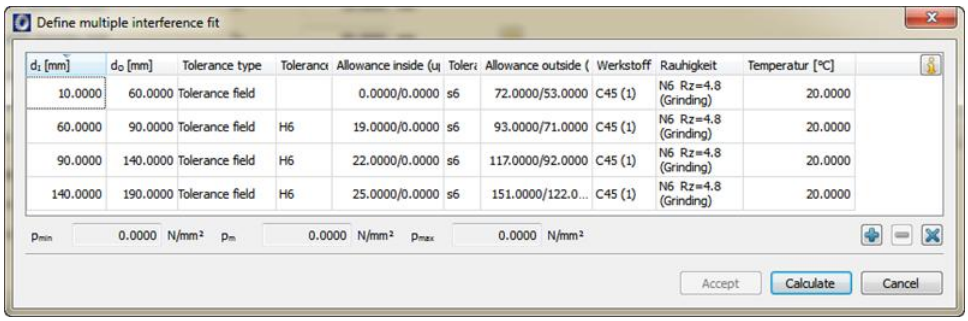


Figure 32.3 Multiple interference fit

The following notes apply to this calculation:

- The first two elements are the original shaft and hub (the first two lines in the table correspond to the first interference fit on the shaft), and are just shown in the table for the sake of completeness. The only user-editable values are the external diameter tolerances of the hub.
- The external diameter of each ring also serves as the inside diameter of the next ring.
- When a new ring is added the following default values are used:
  - External diameter = inside diameter + 50 mm
  - The Material and Roughness are those of the original hub
  - The operating temperature is the same as that for the hub
  - The tolerance class of the inner diameter is the same as the tolerance class of the shaft (for example, s6)
  - The tolerance class of the external diameter is the same as the tolerance class of the hub (for example, H7)

The pressure on the hub is calculated for the following three cases of tolerance values of all elements:

- The average tolerance at each contact surface
- The worst case, i.e. the maximum oversize at each contact surface
- The best case, i.e. the minimum oversize at each contact surface

32.5 Materials

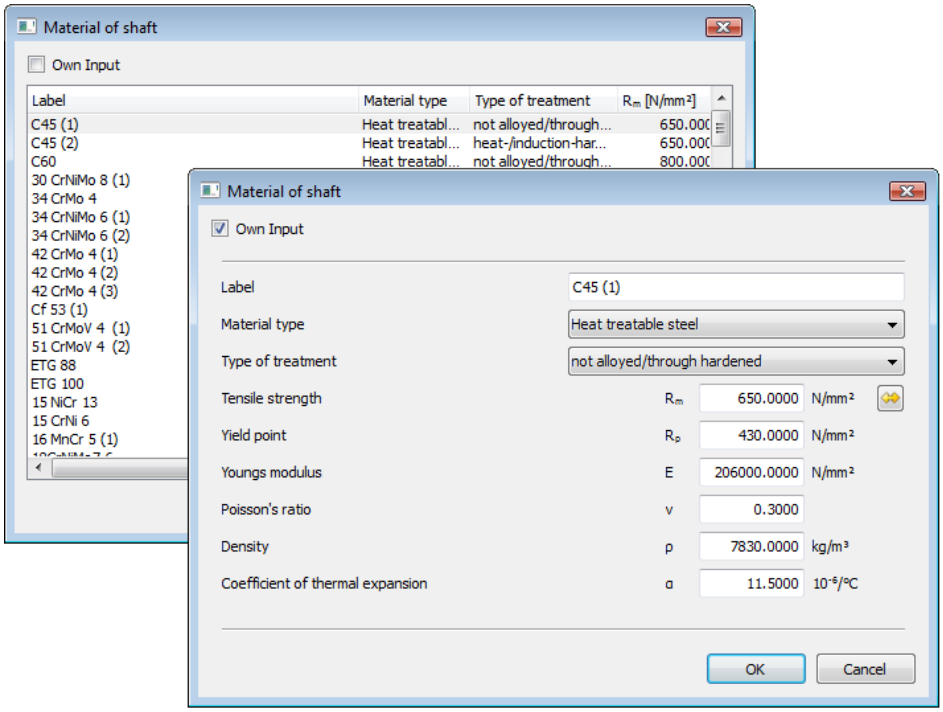


Figure 32.4: Materials screen: Cylindrical interference fit

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

32.6 Settings

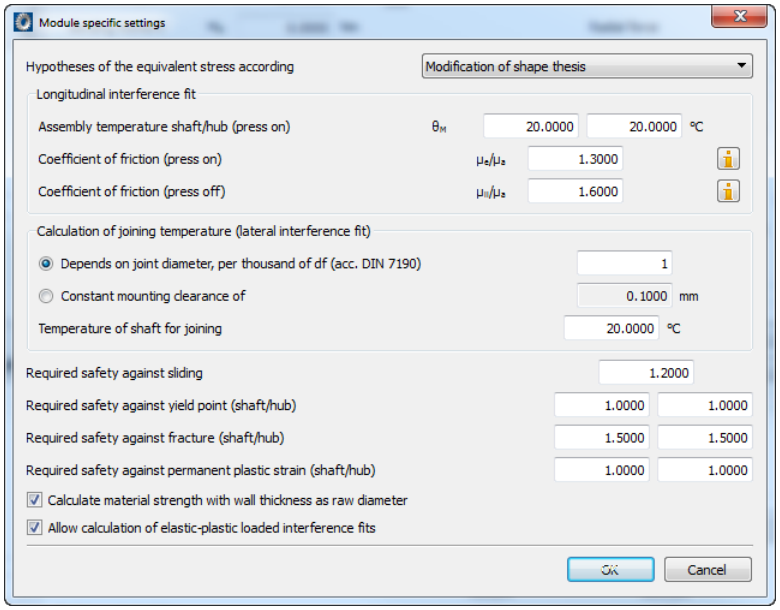


Figure 32.5: Settings: cylindrical interference fit

- Select equivalent stress
  According to [64], with the modification of shape hypothesis:

$$\sigma_v = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1 \cdot \sigma_2}$$

- According to [55], page 13, with the shearing stress hypothesis:
  $\sigma_v = \max(|\sigma_{\phi\phi} - \sigma_{rr}|, |\sigma_{\phi\phi}|, |\sigma_{rr}|)$
- `Assembly temperature shaft/hub`
  The press-on force can be calculated from the assembly temperatures.
- `Coefficient of friction coefficient (press on)`
  Here you can define the μe/μa ratio, i.e. the ratio of the coefficient of friction for press on to the axial coefficient of friction. The default value is 1.3.
  Only values for forcing off are given in DIN 7190. Niemann Volume I (1981 edition), Table 18/3 p. 363, also gives values for this press on.
- `Coefficient of friction coefficient (forcing off)`
  Here you can define the μe/μa ratio, i.e. the ratio of the coefficient of friction for forcing off to the axial coefficient of friction. The default value is 1.6.
  Only values for forcing off are given in DIN 7190. Niemann Volume I (1981 edition), Table 18/3 p. 363, also gives values for this press on.

- Calculation of joining temperature
  The mounting clearance can be input as either dependent on the diameter of joint D_F (modified for warming) or as a constant mounting clearance. The joining temperature of the external component is then calculated from this value. You can also define the **shaft temperature for joining**. This temperature and the mounting clearance are then used to calculate the joining temperature of the hub. The joining temperature of the hub is only output in the report if the shaft temperature during joining lies between -273 °C and 20 °C.
- Required safeties
  Under Settings you can input the required safeties against sliding, yield point and fracture. These safety factors are then used to define the values you require during sizing. The required safeties against plastic deformation are used to define the plasticity diameter that must be set when an interference fit is placed under plastic-elastic stress.
- Calculate material strength with wall thickness as raw diameter
  If you set this flag, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.
- Allow calculation of elastic-plastic loaded interference fits
  If you set this flag, the calculation is also performed for elastic-plastic stress (according to DIN 7190), otherwise, only elastic stress is included.
- For negative interference during operation, continue the calculation and document the allowances
  If there is a negative allowance during operation, an error message is displayed. This usually interrupts the calculation, because this value means the fit is no longer an interference fit. Set this flag to force the calculation to continue despite the error message so that you can display allowances for higher operating temperatures. Only the section with the allowances calculated for operation are then displayed in the report.

32.7 Sizings

- Tolerance according to ISO/DIN

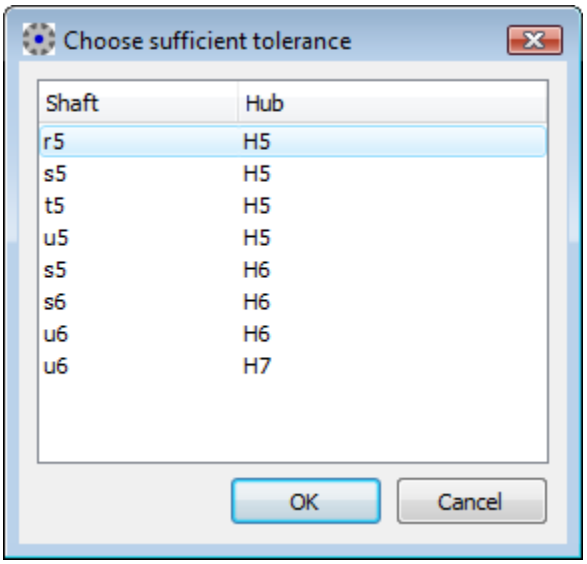


Figure 32.6: Display of possible tolerance pairings

KISSsoft has a very convenient sizing function that you can use for suitable tolerance pairs. Standardized tolerance pairs are stored in the M01-001.DAT file. Click the button next to the Tolerances field in the main screen to start the sizing process.

Based on the nominal required safety (which you can change in `Settings`), you can determine all the tolerance pairs which fulfill the requirements (sufficient safety against sliding, safety against fracture and yield point) and display these pairs in a list.

- Torque, axial force, diameter of joint, and length of interference fit
  KISSsoft can size the maximum transmissible torque, the transmissible axial force, the required length, and the diameter (according the required safety values you entered in Settings).

Chapter 33
Conical Interference Fits

Use this module to calculate the service reliability of a conical interference fit. Defining the mounting conditions.

The calculation methods are:

- Method as defined by Kollmann [55], Verification and sizing.

- Method as defined in DIN 7190-2 (2013)

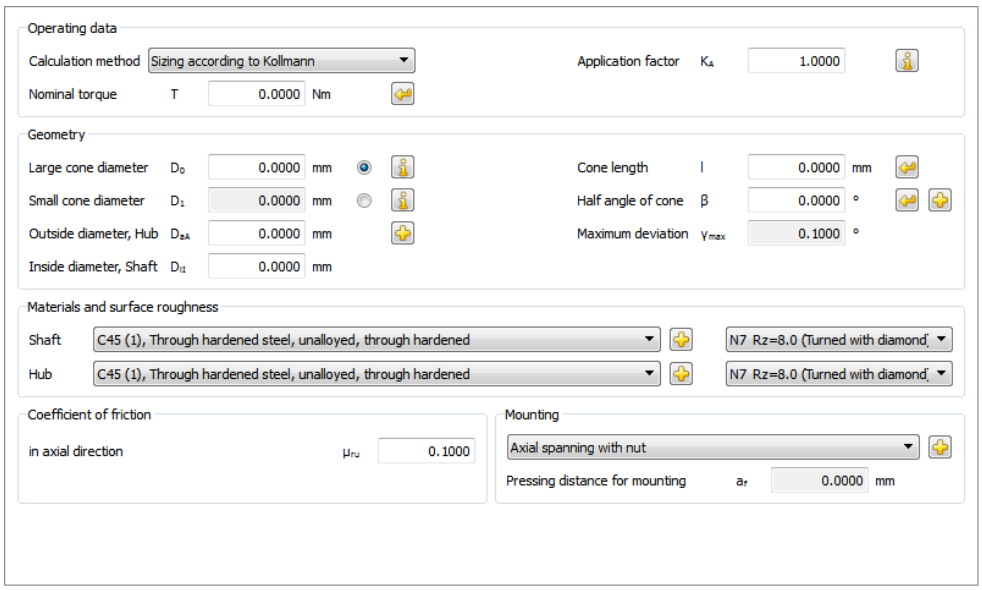


Figure 33.1: Basic data: Conical interference fit

Cone angle: The cone angle is the angle β between the flank of the cone and its axis. The opening angle α of the cone is twice the size of the angle of taper.

Calculating the conical interference fit:

- All known investigations focus on external and internal components made of materials that have the same Young's modulus and internal components that do not have any holes in the area of the cone. (Kollmann)
- If you select the method described in DIN 7190-2, and the shaft and hub are made of different materials, the material with the higher Young's modulus should be used for the shaft material.

- Conical interference fits must always have a stop at the upper end. For this reason, the program only deals with this situation.
- Conical interference fits are normally spanned (tightened) axially with a bolt. You must check the joint carefully by measuring the displacement of the cone. Just tightening with a torque wrench is not accurate enough. Conical interference fits are only joined by pressing them on in exceptional circumstances.

Adhesive coefficient in the case of slipping in the axial direction: Coefficient of friction (for slipping in axial direction) after investigations by Galle (see Kollmann [55], Table 2.20):

Material pairing	**Previous**	**Coefficient for sticktion**
	Rating	
Ck60/16MnCr5	-	0.299
42CrMo4/16MnCr5	-	0.269
31CrMoV9/31CrMoV9	-	0.247
Ck60/16MnCr5	U	0.407
42CrMo4/16MnCr5	U	0.297
31CrMoV9/16MnCr5	U	0.375
31CrMoV9/31CrMoV9	U	0.468
Ck60/16MnCr5	W	0.357
42CrMo4/16MnCr5	W	0.472
31CrMoV9/31CrMoV9	W	0.387

Load classifications:

- none

U Circumferential bending load

W Fatigue torsion load

No adhesive coefficients for other combinations of materials are available, so you will have to estimate them.

Adhesive coefficients for radial interference fits in longitudinal and peripheral direction subjected to sliding: Coefficient of friction as described in DIN 7190 [29], Table 4, see Cylindrical interference fit (see page IV-870) section.

33.1 Calculation

Sizing in accordance with Kollmann:

If you select this method, the maximum angular deviation γ_{max}, the pressing distance for mounting a_f, and the joining force, can be sized using the required safeties and the specified load and geometry.

Verification in accordance with Kollmann:

The transmittable torque is also calculated if no angular deviation, or maximum angular deviation is present.

You can also input the pressing distance for mounting af or the mounting pressing force Ffmin.

The safeties against the yield point and against sliding are also calculated in the verification process.

The safeties defined by Kollmann include the required safeties, which can be defined under `Calculation/Settings`.

As specified in DIN 7190-2:

In this calculation method, the sliding moment, joining force, and axial force are calculated for the state at which all the elements are already joined, from the input torque, mounting without force and all the geometry data. The safety against sliding is calculated in this way. If different external hub diameters or different inside shaft diameters are present, these can also be defined as slices (max. 7). The values are calculated for each individual slice and then added together. The maximum value is used for the pressure in the joint.

$$S_R = T_R / T_{\max}$$

In the case of a central load application, the safety against sliding S_R should have a value of $S_{Rmin} > 1.3$.

Furthermore, if you have defined your own joining force at mounting p_{FA} to calculate the effective equivalent stresses, the maximum equivalent stress is then compared with the yield point of that particular material in order to define the safety. The maximum equivalent stresses appear every time for the inner diameter of the hub or shaft when a hollow shaft is being used.

$$S_P = R_{p0,2} / \sigma_{v\max}$$

Safety against the yield point S_P (plastic flow) should have a value of $S_{Pmin} > 1.3$ to ensure that the elements are not affected by plastic deformation when they are assembled and disassembled.

33.2 Application factor

You define the application factor here in the same way as in the cylindrical gear calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 33.2 Application factor in calculations according to DIN 6892. You will find more detailed comments in DIN 3990, DIN 3991, ISO 6336.

33.3 Axial spanning with nut

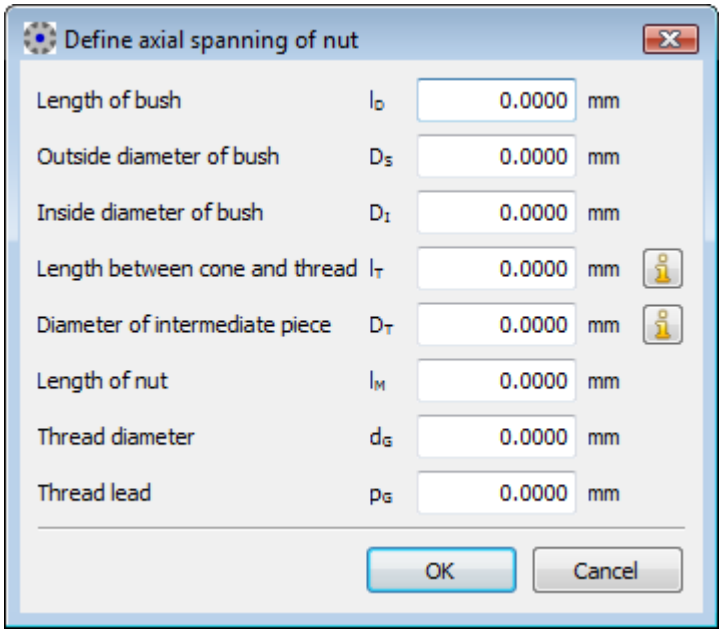
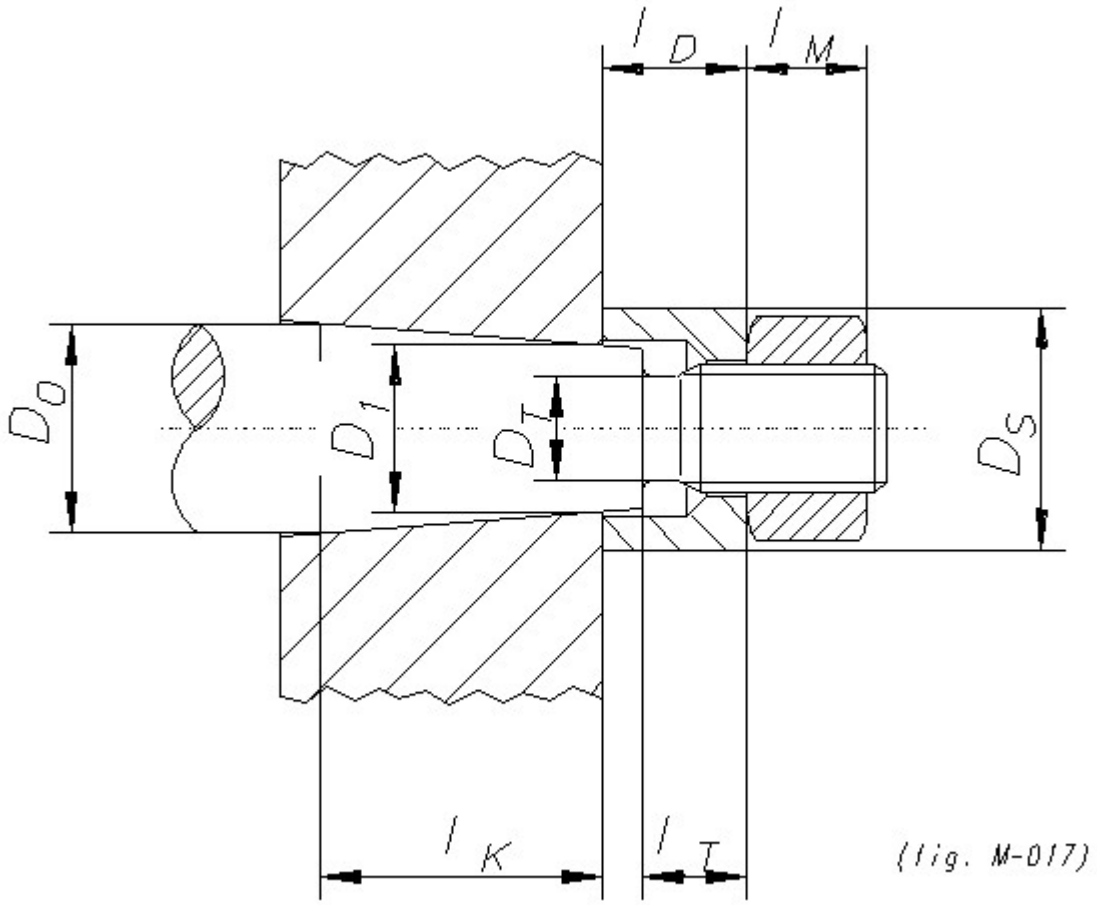


Figure 33.2: Axial spanning of nut

Axial spanning (tightening the nut) produces relative axial displacements which are applied to the individual parts. This causes lateral elongation and therefore increases the compacting pressure on the active surface. The values required for this calculation are shown in the diagram below.



33.4 Variable external diameter of the hub

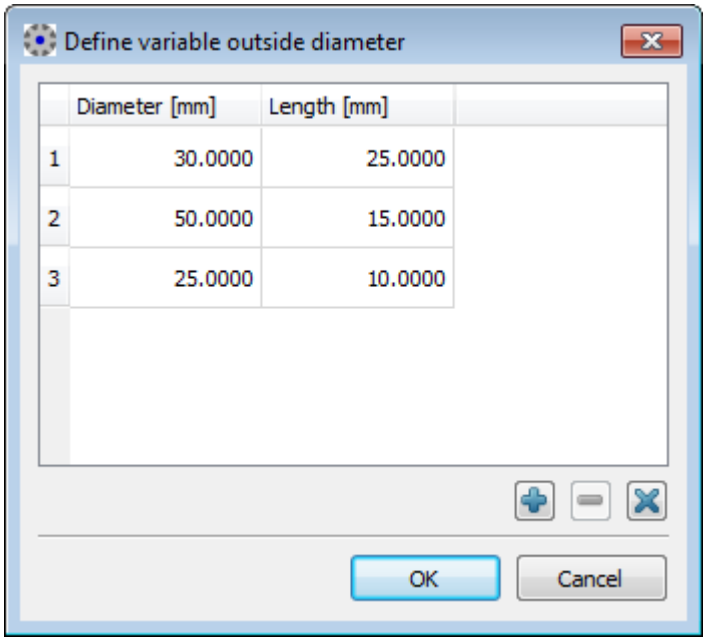


Figure 33.3: Variable external diameter

In case of a stepped external diameter, a single equivalent diameter is determined from the diameters and lengths. This value is then used to calculate the stiffness of the outer part.

33.5 Conicity

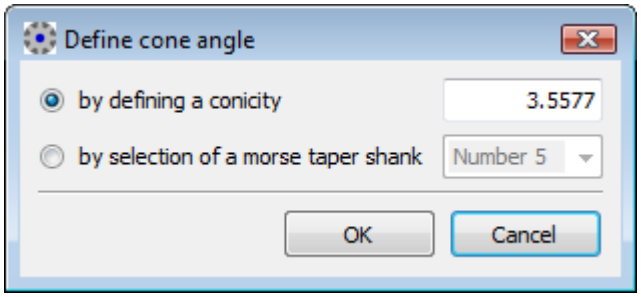


Figure 33.5: Conicity

This additional input dialog gives two methods for defining the conus:

- Conicity:
  The conicity is defined as follows:
  $x = l/(D_0-D_1)$. Here, x is the value that must be input.
- Morse tapers:
  Morse tapers are defined in DIN 228 and have a conicity of between 1:19.212 and 1:20.02.

33.6 Materials

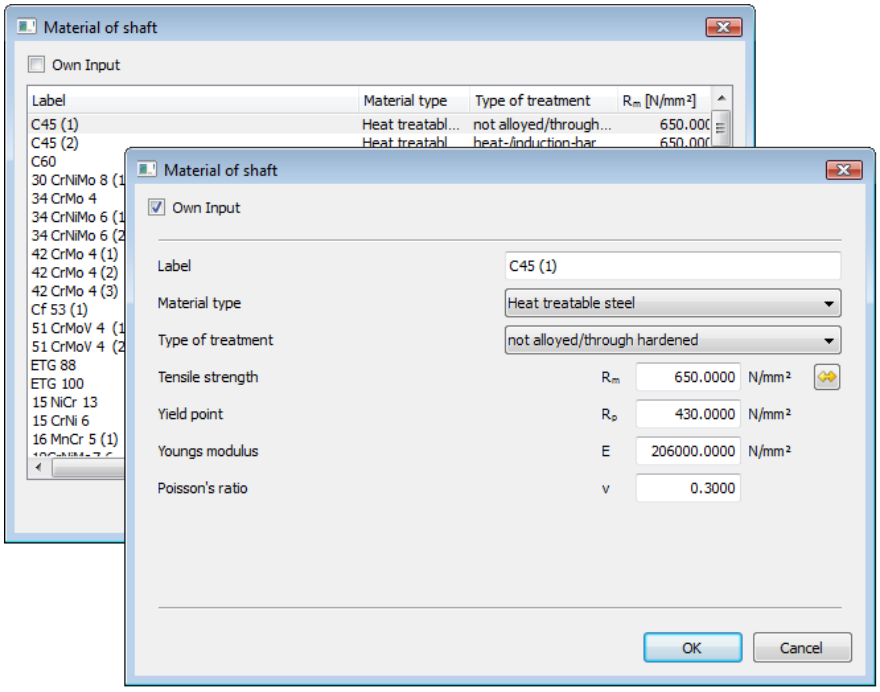


Figure 33.6: Materials screen for Conical interference fit

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

33.7 Settings

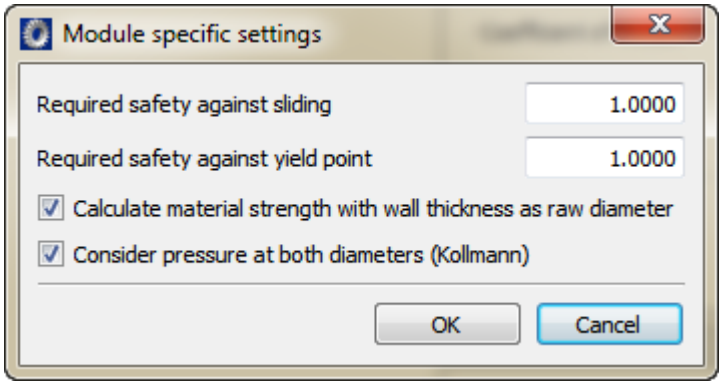


Figure 33.7: Settings: Conical interference fit

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

If you select the `Consider pressure at both diameters (Kollmann)` flag, the pressure at both the large and small cone diameters is taken into account, otherwise only the pressure at the largest diameter is used. However, this only applies to the method described by Kollmann. In DIN 7190-2, the average diameter is used for the calculation.

Enter the required safety factor for slipping and yield point under Settings. These safety factors are then used to define the values you require during sizing.

In the method defined by Kollmann, the default value set for required safety against sliding is 1.0 and the default value for required safety against the yield point is set to 2.0.

DIN 7190-2 recommends that safety against the yield point should be set to 1.2 to ensure that the elements are not affected by plastic flow when they are disassembled hydraulically.

In the case of a central load application, DIN 7190-2 recommends a safety against sliding of at least 1.2.

33.8 Sizings

KISSsoft can size the maximum transmissible torque, the permitted cone angle (for self-locking) and the length of interference fit for transmitting the maximum torque.

The torque and the length of interference fit are sized using the defined required safeties.

In a verification according to Kollmann, you can size the pressing distance for mounting or the mounting pressing force using the required safety against sliding.

As specified in DIN 7190-2, the sizings are calculated using the required safety against sliding, apart from the joining pressure which is sized using the required safety against yield point.

Chapter 34
Clamped connections

Clamped connections are only used to transfer low or medium torque (little fluctuation).

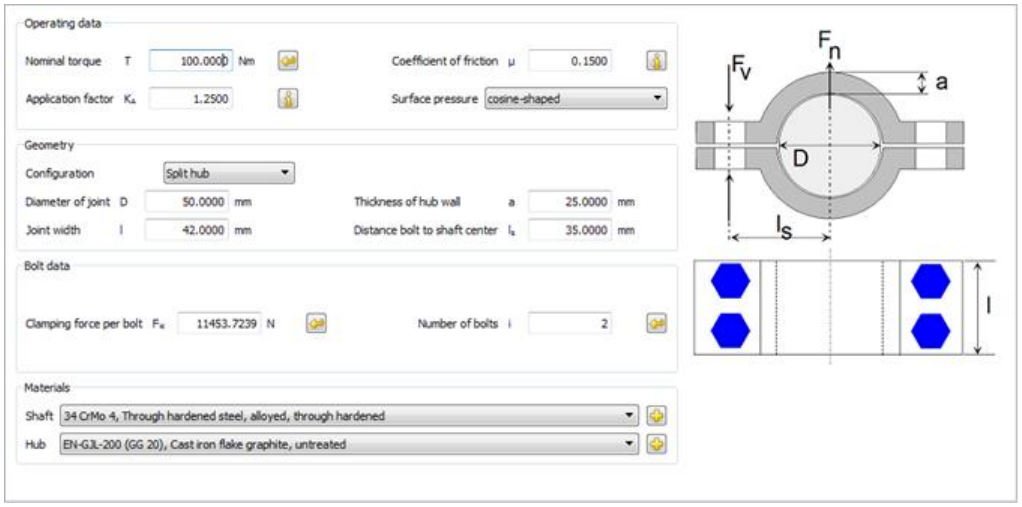


Figure 34.1: Basic data: Clamped connections

There are two different configurations of clamped connections that can be calculated:

- `Split hub`
  In the case of a split hub, it is assumed that pressure is distributed uniformly across the whole joint. The pressure can be equal or cosine-form surface pressure or linear contact.
- `Slotted hub`
  We recommend you use as narrow a fit as possible (hubs are also subject to bending) to ensure that the pressure is mostly of a linear nature. The calculation is performed for the least practical case of linear pressure.

Calculations of safety against sliding and surface pressure are described in literature by Roloff Matek [62]. The calculation of bending is performed as specified by Decker [86].

34.1 Calculations

Split hub:

Depending on the type of surface pressure, an additional factor for surface pressure and safety against sliding is used to calculate a shared hub:

- K = 1; uniform surface pressure
- K= π^2/8; cosine-form surface pressure
- K = π/2 linear contact

In KISSsoft you can select the type you require from a selection list.

Formula for surface pressure:

$$pF = \frac{2 \cdot KA \cdot T \cdot SH \cdot K}{\pi \cdot D^2 \cdot ls \cdot \mu}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot \pi \cdot D \cdot \mu}{2 \cdot KA \cdot T \cdot K}$$

Formula to calculate bending:

$$\sigma B = \frac{0{,}2 \cdot Fkl \cdot i \cdot ls}{2 \cdot Wb}$$

Slotted hub:

Formula for surface pressure:

$$pF = \frac{i \cdot Fkl \cdot l2}{D \cdot l \cdot l1}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot D \cdot \mu \cdot l2}{KA \cdot T \cdot l1}$$

Formula to calculate bending:

$$\sigma B = \frac{0{,}3 \cdot Fkl \cdot i \cdot (l2 - l1)}{Wb}$$

Description of codes:

pF: Surface pressure [N/mm2]

KA: Application factor

T: Nominal torque [N]

SH: Safety against sliding

K: Surface pressure correction factor

l: Joint width [mm]

D: Joint diameter [mm]

lS: Distance bolt to shaft center [mm]

l1: Distance normal force to center of rotation [mm]

l2: Distance - clamping force and center of rotation [mm]

μ: Coefficient of friction

σB: Bending stress [N/mm2]

Fkl: Clamp load per bolt [N]

i: Number of bolts

Wb: Moment of resistance [mm3]

34.2 Sizings

In these calculations, you can size the torque, the clamp load per bolt, and the number of bolts, to suit a predefined required safety value.

34.3 Settings

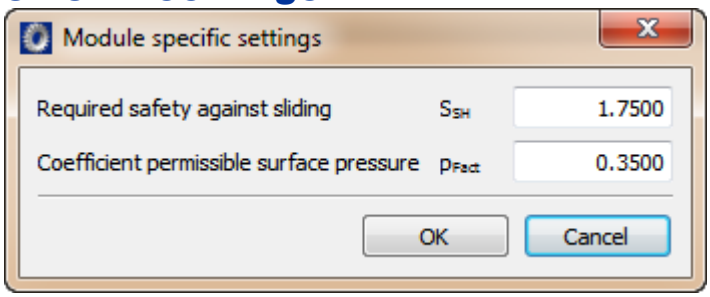


Figure 34.2: Settings Clamped connections

The required safety against sticking, SSH, is used to size the torque, the clamp load per bolt, and the number of bolts.

If the hub material is gray cast iron, this coefficient times the tensile strength is used to calculate the permitted pressure.

(pzul =pFact*Rm) (default value ~ 0.35 for an interference fit)

For all other materials, this coefficient times the yield point is used to calculate the permitted pressure.

(pzul =pFact*Rp) (default value ~ 0.35 for an interference fit)

34.4 Materials

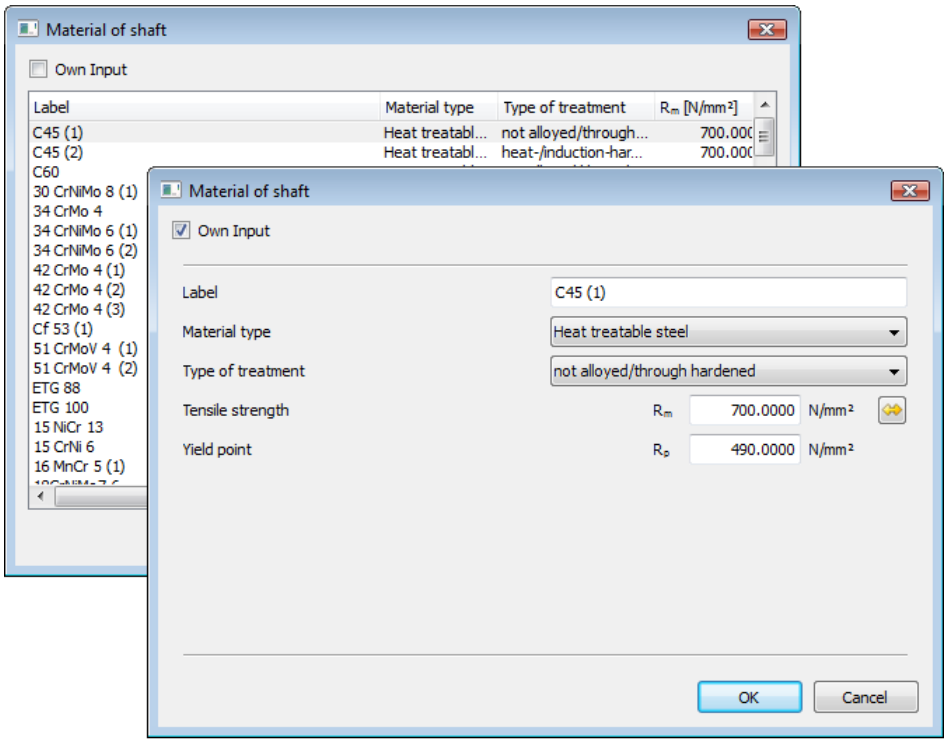


Figure 34.3: Materials Clamped connections

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

Chapter 35
Key

Keys are by far the most commonly used shaft-hub connections. In particular, they help to transmit the torque. Their geometry has long been standardized according to DIN 6885 [26]. However, to ensure adequate safety levels were achieved when transmitting torque, it was always necessary to refer to secondary sources of technical literature [64]. DIN standard 6892 [27] documents the different calculation methods that can be used for key connections.

You must perform two checks for keys:

1. Check the torque transmission by monitoring surface pressures on the shaft, hub and key.
2. Check the fatigue limit of the shaft due to the notch effect caused by the keyway. This effect is already described in DIN 743 [9]. We recommend you use this standard to verify the shaft strength, rather than DIN 6892.

Special characteristics of calculations according to DIN 6892:

- Key connections are mostly used with light interference fits. The calculation takes into account the decrease in torque on the key due to the interference fit.
- The calculation proves the nominal torque as well as the actual pitch torque over the entire operating period. The fatigue strength calculation based on the nominal torque also includes the number of load changes, which experience has shown to have a significant and damaging effect on the key.
- The type of load has a considerable effect on the service reliability of keys. This effect is taken into account by using a wide range of load distribution coefficients.
- The permissible pressure values are derived from the yield point. As a result, you can derive this for common and more unusual materials in accordance with the standard. The hardness influence coefficient is used to take the surface treatment into account.

Calculation Method B as defined in DIN 6892 recommends you use a differentiated calculation to prove the service reliability of key connections. Method C has been greatly simplified.

35.1 Main screen

For keys as defined in

- DIN 6885.1 (ISO/R 773, VSM 15161) Standard
- DIN 6885.1 Form G, H, J
- DIN 6885.2
- DIN 6885.3
- ANSI B17.1 Square
- ANSI B17.1 Rectangular
- Own input

you can calculate the load on shaft, hub and key (surface pressure) and the key (shearing) to determine the safeties.

The following calculation methods are available: DIN 6892 B/C [27].

The calculation takes into account the tolerances of the key radii and the direction of force. You can also enter your own value for the number of keys and the application factor.

Explanations for Figure 35.2:

→	Application or removal of torque
o	Start of key
F_u	Center of force application point on hub

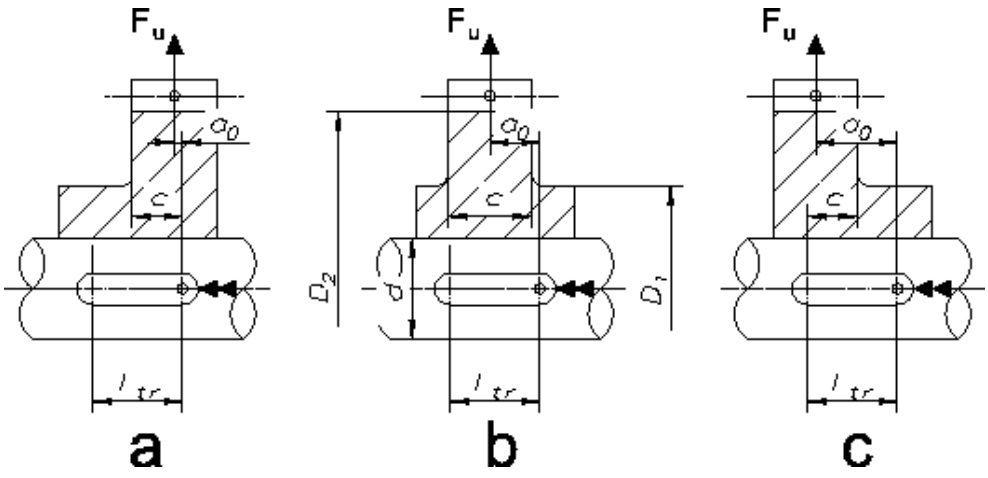


Figure 35.1: Key: Load application.

- Supporting key length: The supporting key length is defined according to DIN 6892:
  Helical key form (A, E, C according to DIN 6885) $l_{tr} = l_{eff} - b$
  Straight key form (B, D, F, G, H, J according to DIN 6885)$l_{tr} = l_{eff}$
  l_{eff} Actual key length
  l_{tr} Supporting key length
  b Key width

- Frictional torque
  Key connections are usually combined with a light interference fit. The calculation takes into account the decrease in torque on the key due to the interference fit. This effect is only relevant if you are performing the calculation as defined in DIN 6892 B.
- Frequency of load peak
  To determine the safety regarding the maximum torque, you must enter the approximate number of load peaks. This effect is only relevant if you are performing the calculation as defined in DIN 6892 B.

35.1.1 Additional inputs for DIN 6892 Method B

If you select the calculation method specified in with DIN 6892 B, you can enter the following data:

- Chamfer on shaft
- Chamfer on hub
- Small external diameter of hub D_1
- Large external diameter of hub D_2
- Width c for external diameter D_2
- Distance a_0 (→ see Figure on page IV-895)
- Torque curve: indication of whether this is alternating torque.
  If alternating torque is present, you can also define the backwards torque here. If this backwards torque is greater than the minimum effective frictional torque ($T_{maxR} > T_{Rmin}*q$; q=0.8), the load direction changing factor f_w is set to = 1.
  If ($T_{maxR} > T_{Rmin}*q$ and $T_{max} > T_{Rmin}*q$; q=0.8), the maximum torque is therefore also greater than the minimum effective frictional torque. In this case, the

frequency of the changes in load direction is taken into account when defining the load direction changing factor (from diagram; $f_w<1$)

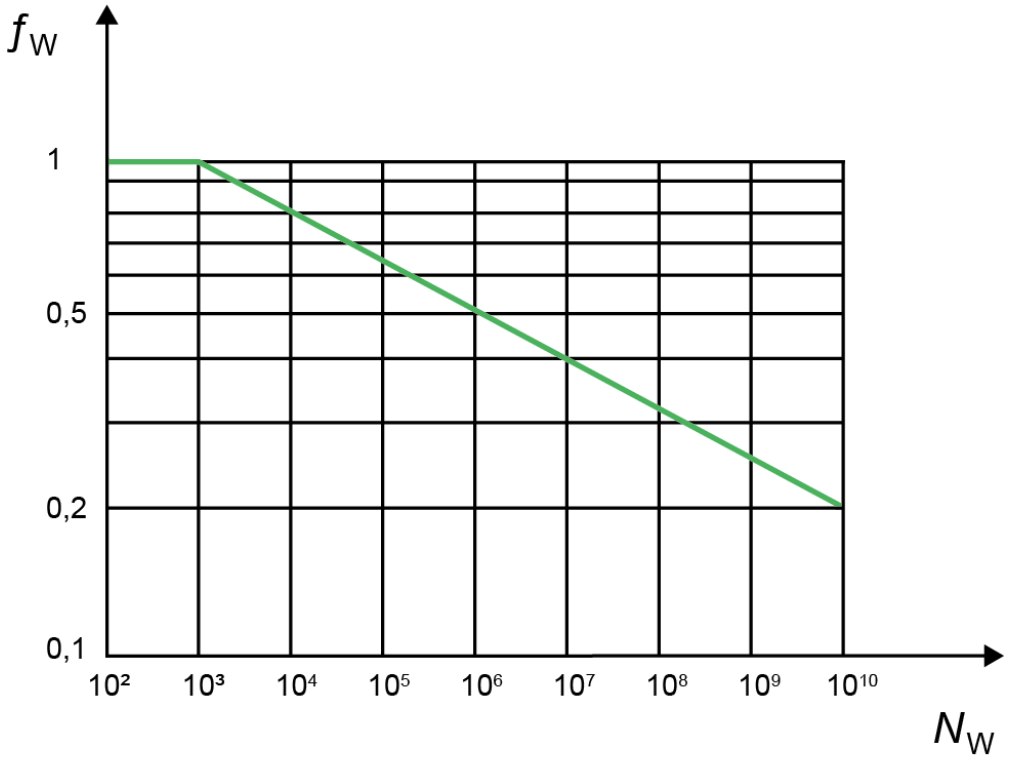


Figure 35.2: Load direction changing coefficient for reciprocal load, Figure 6, DIN 6892

To perform the calculation with the equivalent torsional moment, use coefficient q=0.5. The formula is the same as the one used for the maximum torsional moment, therefore ($T_{eqR} > T_{Rmin}$*q; q=0.5).

- Frequency of the changes in load direction
  In this situation you input the number of torque changes throughout the entire service life (but only if alternating torque is present).

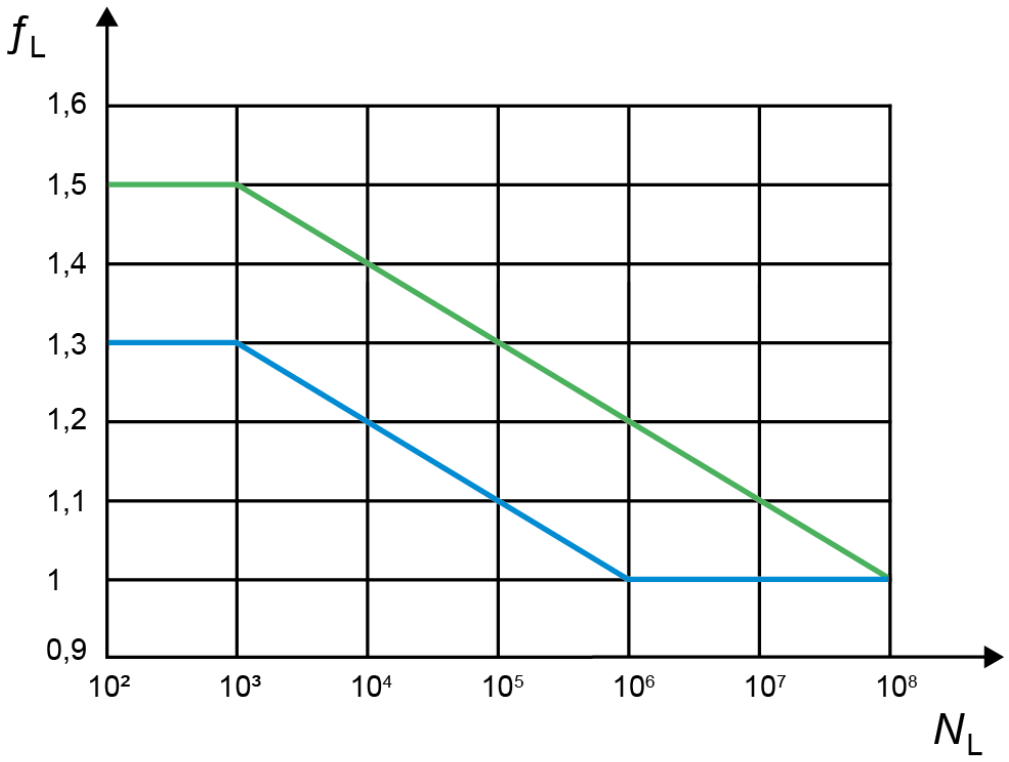


Figure 35.3: Load peak coefficient, Figure 7, DIN 6892

- Green line: ductile material
- Blue line: brittle material

35.2 Application factor

You define the application factor here in the same way as in the cylindrical gear calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 35.4: Suggestions for the application factor in calculations according to DIN 6892. You will find more detailed comments in DIN 3990, DIN 3991 and ISO 6336.

Suggestions for the application factor from other sources: See tables 35.3 and 35.4.

Type of	**characteristic**	**Type of**	**working**
machine	**Operational behavior**	**Shocks**	**Factor**
Turbine, blower	Uniformly rotating movements	Slight	1.0. . . 1.1
Internal combustion engine	Reciprocating movements	Medium	1.2. . . 1.5
Presses, saw frames	Reciprocating, impacting movements	Heavy	1.6. . . 2.0
Hammers, stone crushers	Impacting movements	Very heavy	2.1. . . 3.0

Table 35.5: Application factor according to Roloff/Matek [61].

	surfaces pressed together	**surfaces sliding against each other without load**	**surfaces sliding against each other under load**
constant load	1.0	2.0	6.0
Pulsating load, moderate im-	1.5	3.0	9.0

pacts			
Alternating load, moderate impacts	3.0	6.0	18.0
Pulsating load, heavy impacts	2.0	4.0	12.0
Alternating load, heavy impacts	6.0	8.0	36.0

Table 35.6: Application factor that takes into account the load behavior according to Professor Spinnler [[72].

35.3 Load factor

Contact coefficient as defined in DIN 6892, [27]:

$K_v=1/(i*\phi)$

$\phi = 1$ for one spring

$\phi = 0.75$ for two springs to calculate the equivalent surface pressure

$\phi = 0.9$ for two springs to the calculate the maximum surface pressure

more than two springs is unusual

KISSsoft calculates the contact coefficient on the basis of the number of springs.

35.4 Own inputs

If you set the `Own input` flag, you can enter your own data for the keys' geometry, that differs from the values specified in DIN 6885.

NOTE

If you already know the upper and lower allowance, you must enter the mean value for the chamfer and the two groove depths. The value for the peak incline a is only to be defined for key forms G, H, and J, in accordance with the DIN 6885.1 standard.

35.5 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials).

35.6 Materials

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

35.7 Settings

- `Calculation method`
  Here you can select either DIN 6892 Method B or Method C. The default setting is Method B, because Method C has been greatly simplified.
- `Take pressure on key into account`
  When this flag is set, the pressure on the key is taken into account when you size the permissible pressure by clicking the  button.
- `Calculate material strength with wall thickness as raw diameter`
  When the strength values for the hub are being set, either the external diameter (hub was turned from solid) or the wall thickness of the hub (hub was heat treated as a ring) is used.

35.8 Sizings

During the sizing process, the required value is defined such that the required safety (specified in `Calculations/Settings`) is only just achieved. To view the results in the lower part of the main window, you must perform the calculation immediately after the sizing.

Possible sizings:

- transmissible torque
- necessary length of key way in shaft and hub

The "Keys" tutorial has been created specially to describe how you verify these keys.

Chapter 36
Straight-sided spline

Spline shaft connections are often used for adjustable, form-closed shaft-hub connections. Main areas of use: Vehicle gear trains, machine tools.

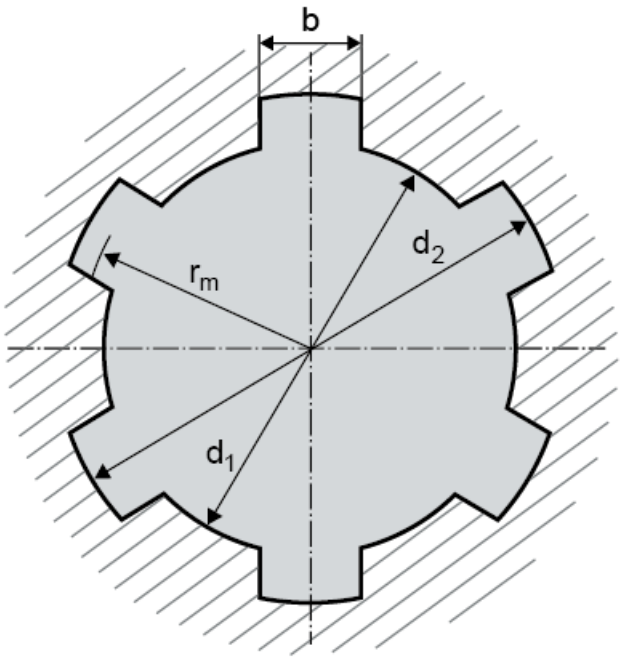
KISSsoft calculates the load on the shaft and hub (surface pressure) for splined shafts. The load on the shaft and hub (surface pressure) is calculated, and the safeties are determined, as described in classic technical literature ([64]). The calculation defined by Niemann forms the basis of DIN 6892 (key calculation).

36.1 Standard profiles

You can select one of these standards from the selection list:

- DIN ISO 14 (light series)
- DIN ISO 14 (medium series)
- DIN 5464 (heavy, for vehicles)
- DIN 5471 (for machine tools)
- DIN 5472 (for machine tools)
- Own input

In a splined shaft connection, after you select a standard, the program displays the corresponding external and inside diameters, and the number of keys, along with their width.
Own input: Select the `Own input` option to define your own splined shaft profile.



36.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 36.1: Application factor according to DIN 6892

36.3 Torque curve/Frequency of change of load direction

When you select the torque curve, you can choose one of two positions:

1. `No alternating torque`
2. `With alternating torque`

If you select item 2), the calculation not only defines the Frequency of change of load direction, as defined in DIN 6892, Figure 6, but also the frequency of change of load direction coefficient f_w. In the case of item 1), the coefficient will be set to 1.0.

36.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d_m * l_{tr} * h * z)$

$k_{\varphi\beta}$: Share factor

k_1: Length factor

T: Torque

d_m: average diameter

l_{tr}: supporting length

h: spline height

z: Number of keys

36.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

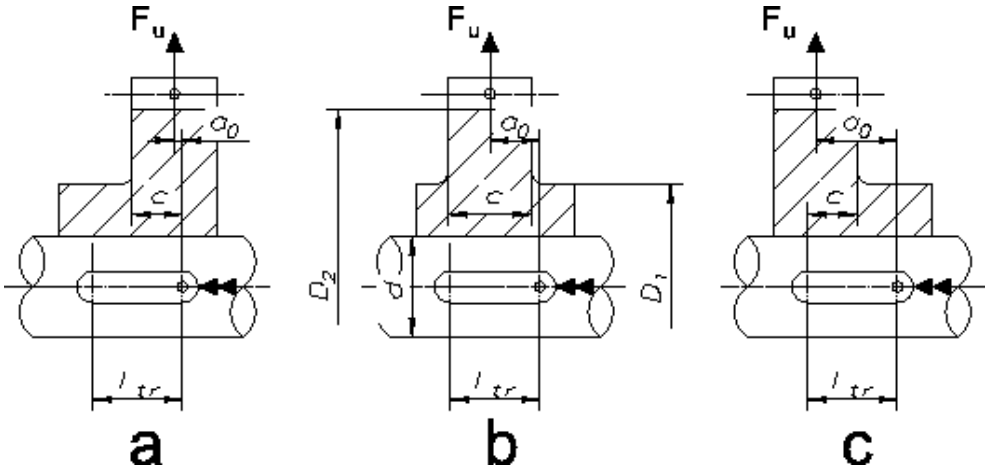


Figure 36.2: Spline shafts: Load application.

36.6 Share factor

To calculate the occurring contact stress, a share factor of $k_{\varphi\beta}$ is taken into account. This is then multiplied by the load. Interim sizes not shown in the table are interpolated linearly.

Form-closure	**spline connection with involute flank**					
connection	**Tolerance zones according to DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 36.3: Share factor in accordance with Niemann

36.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:
$p_{eq}=f_s * f_H * R_p$

- for brittle materials:
$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 24.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core hardness for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

36.8 Materials

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

36.9 Settings

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

36.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in `Calculations/Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing. Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max}
- Supporting length l_{tr}

Chapter 37
Splines (Strength)

Splines are spur gear meshings that have a shortened tooth height and a large pressure angle (usually 30°). In KISSsoft, you can use one of two different calculation modules to calculate splines. The geometry and tolerances required for manufacture, and the strength calculation, are described in the Splines chapter (Geometry and Strength) (Z09a (see page IV-929)) under Connections.

For splines, you must calculate the load on shaft and hub (surface pressure). You can also add additional standards. Toothing data is defined in the database and therefore you can make the use of in-house profiles mandatory. You can also use the KISSsoft Spline (geometry and strength) module, Z09a, to calculate the manufacturing mass and the tolerances. The load on the shaft and hub (surface pressure) is calculated, and the safeties are determined, as described in classic technical literature ([64]).

37.1 Standard profiles

You can choose one of these standards from the selection list:

- DIN 5480
- DIN 5481
- DIN 5482
- ISO 4156
- ANSI B92.1
- ANSI B92.2M
- Own input (tip diameter of shaft and hub, module, number of teeth, profile shift coefficient)

For splines, the corresponding values are displayed in the list after the norm selection.

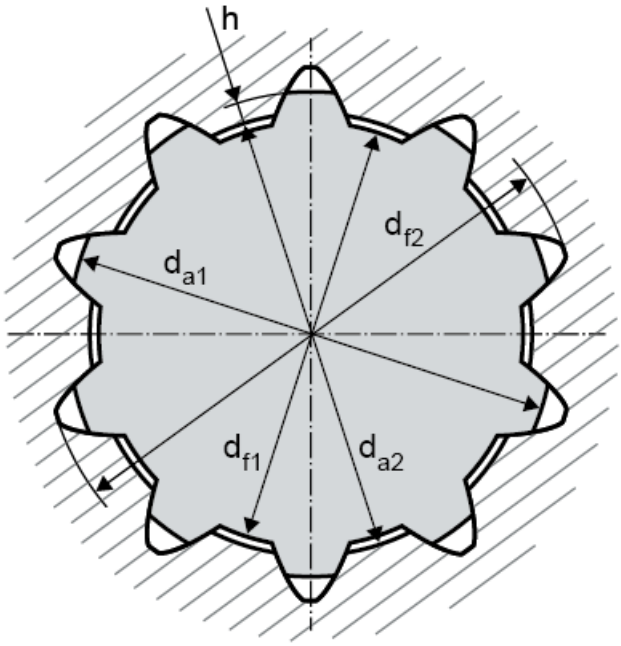
d_{a1}: Tip diameter of the shaft

d_{a2}: Tip diameter of the hub

m: Module

z: Number of teeth

x: Profile shift coefficient



Own input: Select the `Own input` option to enter your own data for the spline. The critical factor here is that the tip diameter of the shaft is greater than the tip diameter of the hub, if not, an error message is displayed.

37.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 37.1: Application factor according to DIN 6892

37.3 Torque curve/Frequency of change of load direction

When you select the torque curve, you can choose one of two positions:

1. `No alternating torque`
2. `With alternating torque`

If you select item 2), the calculation not only defines the Frequency of change of load direction, as defined in DIN 6892, Figure 6, but also the frequency of change of load direction coefficient f_w. In the case of item 1), the coefficient will be set to 1.0.

37.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d_m * l_{tr} * h * z)$

$k_{\varphi\beta}$: Share factor

k_1: Length factor

T: Torque

d_m: average diameter

l_{tr}: supporting length

h: spline height

z: Number of keys

37.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

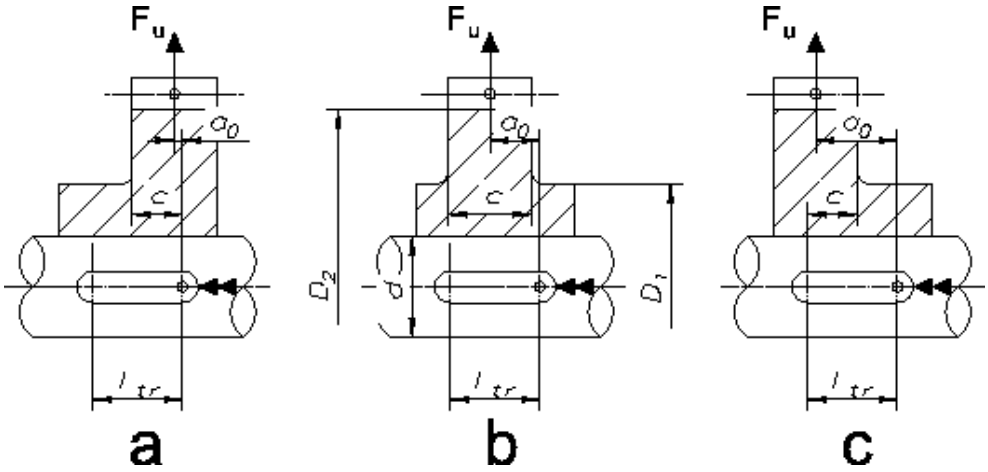


Figure 37.2: Splines: Load application.

NOTE:

The recommended values for k_1 according to Niemann apply to splined joints which are only subject to torsion. Any additional strain caused by bending or shearing forces can create very many, high local stresses. More precise methods must be used to investigate these cases.

37.6 Share factor

If you are working with DIN profiles, the "Manufacturing quality according DIN" can be used directly as a "Tolerance field according to DIN 5480" to define the value of $k_{\varphi\beta}$ according to the table shown below.

If you are working with ISO and ANSI profiles, the specified manufacturing quality +4 (see "Tolerances" section) is taken and used as the "Tolerance field according to DIN 5480". This value is then used to determine $k_{\varphi\beta}$.

Form-closure connection	**spline connection with involute flank** **Tolerance zones according to DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 37.3: Share factor in accordance with Niemann

NOTE:

The recommended values for $k_{\varphi\beta}$ according to Niemann apply to splined joints which are only subject to torsion. Any additional strain caused by bending or shearing forces can create very many, high local stresses. More precise methods must be used to investigate these cases.

Tolerances for splines

As a comparison, we have calculated an example for the tolerances specified in DIN 5480, ISO 4156 and ANSI B92.1 (spline) and for the tolerances given in ISO 1328 and DIN 3961 (cylindrical gear pair) with accuracy grade 6 (outside) and accuracy grade 7 (inside).

The results for the ISO 4156 accuracy grade are 3 to 4 accuracy grades higher than the same results for the DIN 5480 accuracy grade.

ANSI B92.1 returns the same results as ISO 4156.

This is all very confusing, because all other gears (cylindrical gears with straight or helical teeth, bevel gears, worms, etc.) return accuracy grades for DIN, ISO and AGMA tolerance results that are mostly very similar.

37.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:
$p_{eq}=f_s * f_H * R_p$

- for brittle materials:
$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 37.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core hardness for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

37.8 Materials

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes.
You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

37.9 Settings

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

37.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in `Calculations/Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing.

Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max}
- Supporting length l_{tr}

Chapter 38
Splines (Geometry and Strength)

You can calculate the geometry and the control measurement values of splines and hubs according to DIN 5480 (1986 Edition), ISO 4156, ANSI B92.1 or ANSI B92.2M. Strength calculations according to Niemann, DIN 5466 and AGMA 6123-C16 are also included.

The geometry profiles according to DIN 5481 (2005) and according to DIN 5482 (1973) are saved in files.

When you open the file for the profile you require, the KISSsoft screens are filled with all the necessary geometry settings.

38.1 Underlying principles of calculation

38.1.1 General

Involute short cut teeth are often used for couplings. Teeth with large pressure angles $\alpha_n = 30^o$ are very common and, to increase strength, they have a tooth height that is only half the size of normal cylindrical gears. Couplings with teeth as defined in DIN 5480 are very widespread, and are precisely described with regard to geometry and tolerance calculation. The strength calculation is performed according to the usual methods described in technical literature [5],[42].

NOTE

The moment of inertia is calculated as follows: the inside diameter of the shaft is $d_i = 0$, and the hub external diameter is the rounded result of $d_i = d_f + 4 \cdot m_n$. The moment of inertia is then determined for the cylinder between d_i and $(d_a + d_f)/2$.

38.1.2 Calculation of spline connections as described in DIN 5480 with diameter centering

Diameter centered connections are centered in the outside or inside diameters. The pinion center root diameter with the shaft tip diameter is used for outside centering. The pinion center root diameter with the pinion center tip diameter is used for inside centering. Here, the gear toothing is only used for rotational synchronization. The connection must therefore have sufficient flank clearance to prevent the center holes from intersecting. Due to the small tolerances of the centering diameter, diameter-centered connections require increased manufacturing effort to limit the central displacement. This is why they are only used in exceptional circumstances.

To calculate diameter-centered connections:

1. In the `Connections > Splines (Geometry and Strength)` calculation module, open the `Reference profile` input window, by clicking on its tab. Here, select the `DIN5480 Major diameter fit` option in the `Reference profile` drop-down list in the `Shaft` and `Hub` area.
2. Click the Tolerances tab to open the `Tolerances input` window. Check that no flag has been set in the checkbox to the right of `Tip diameter deviation (upper/lower)` and `Root diameter deviation (upper/lower)` input fields, for `Shaft` or for `Hub`. The program then prompts values from the DIN 5480 recommendations. For the tip circle, the following apply:

– for outside centering, H6 for the shaft tip diameter and H11 for the hub tip diameter

– for inside centering, h11 for the shaft tip diameter and H7 for the hub tip diameter

For the root circle, the following apply:

– for outside centering h14 for the shaft root diameter and H7 for the hub root diameter

– for inside centering, h6 for the shaft root diameter and H14 for the hub root diameter

9H/9e is recommended for the tooth thickness allowance.

38.1.3 Calculating spline connections according to DIN 5480 with diameter centering

In the case of flank-centered connections, the flanks are used for centering and rotational synchronization. The nominal lengths of the root diameter used for the shaft and the hub in flank-centered connections are the "theoretical root diameter" calculated with h_{fP} = 0.55*m. All reference profiles without the note `Major diameter fit` or `Minor diameter fit` are for flank-centered splined joints.

The allowances that apply where chip-producing manufacturing processes are involved cover the corresponding greatest possible dedendum (h_{fP}=0.65*m) and the root diameter allowances. The tolerance field for the root diameter of the tooth space is 9H. For the hub root diameter it is 11a.

The following values apply for flank-centered splines:

– Hub tip diameter d_{a2} with tolerance field H11

– Shaft tip diameter d_{a1} with tolerance field h11

– Allowance for the hub root diameter d_{f2}, calculated with A_{df2} = (0.2*m+1.73*(A_e+TG))

– Allowance for the hub root diameter d_{f1}, calculated for machining: with A_{f1} = -(0.2*m+(1.73*A_s+TG)). Calculated for cold roll forming: with A_{f1} = -0.76*m

This description is for calculating the root circle tolerances in accordance with the base standard, DIN 5480-1, Table 5.

Click on `Calculations/Settings/General` to select whether root circle tolerances are to be calculated according to the manufacturing process (DIN 5480-16, 4.3.1) or according to the base standard (DIN 5480-1, Table 5).

When the root circle tolerances are calculated according to the basic standard DIN 5480-1 (table 5), the root form diameter is calculated as specified by the formulae on page 13:

Shaft root form circle diameter: dFf1 = |da2| - 2*cFmin

Hub root form circle diameter: dFf2 = -(da1 + 2*cFmin);

cFmin comes from Table 4 in the DIN 5480-1 standard.

If either the reference diameter or the module lie outside of the defined range, the adjacent edge value in the table is used as cFmin.

The tolerance band calculated for the root diameter is also used here.

38.2 Basic data

38.2.1 Geometry standards

In the drop-down list in the upper left-hand part of the `Geometry` area, you see a list of the available geometry standards. To view a specific standard, click the  buttons to the right of the drop-down list to open the `Define profile view` dialog window. The complete standard and preference sequences are also available for most of the standards in this list. Use the database tool (see page I-128) to add your own standards to the list or extend existing guidelines. For example, the preference sequence for DIN 5480 is stored in the `M02C-001.dat` file in the `dat` folder of your KISSsoft installation folder. Each line corresponds to an entry in the `Define profile` list and uses the following syntax:

d_{a1} | d_{a2} | m_n | z | i^*

where

d_{a1}	Tip diameter, shaft
d_{a2}	Tip diameter, hub
m_n	Normal module
z	Number of teeth
x^*	Profile shift coefficient shaft

EXAMPLE:

KISSsoft Protokoll 27. February 2008, 09:35:12

KISSSOFT Calculation programs for machine design

```
#Units:mm|mm|mm||
3.50|2.72|.5|6|0.00
4.00|3.18|.5|7|0.00
4.50|3.66|.5|8|0.00
5.00|4.14|.5|9|0.00
5.50|4.62|.5|10|0.00
6.00|5.11|.5|11|0.00
6.50|5.60|.5|12|0.00
```

Figure 38.1: Example entry in `M02C- 001.dat`

The marked entry in `KISSedit` (see Figure 38.2) stands for d_{a1} = 5.5mm, d_{a2} = 4.62mm, m_n = 0.5mm, z = 10 and x^* = 0.

NOTE

You can only edit the Normal module, Number of teeth and Profile shift coefficient input fields if you have first selected `Own input` in the drop-down list for geometry standards.

38.2.2 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the button to open a dialog window, in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

38.2.3 Pressure angle at normal section an

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For splines the pressure angle is usually $\alpha_n = 30^o$.

38.2.4 Number of teeth

For internal toothed gears, you must enter the number of teeth as a negative value as stated in DIN 3960. The shaft and the hub must have the same number of teeth, but with different signs.

38.2.5 Profile shift coefficient

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. For pinion and gear factors:

$$x_1^* = -x_2^*$$

Click the button, and KISSsoft will determine whether the profile shift coefficient is taken from measured data or from values given in drawings.

The following options are available here:

- **Base tangent length**
  Here you must enter the base tangent length (span) and the number of teeth spanned. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.

- **Measurement over two balls**
  To do this, enter this dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins, see Measurement over pins.

- **Measurement over 2 pins**
  To do this, enter this dimension and the diameter of the pin. For helical gear teeth and gears with an odd number of teeth, you must also enter a minimum span, so the measurement can be performed. The measurement over pins cannot be measured in internal helix gears.

- **Tip circle**
  This is a rather imprecise calculation because the tip diameter does not always depend solely on the profile shift.

- **Tooth thickness at reference circle**
  Here, you specify the tooth thickness. You can also enter the arc length or chordal length, and whether the value is in transverse or normal section.

NOTE

The profile shift coefficient of the shaft and hub must be the same value.

38.2.6 Quality

Achievable qualities are shown in Table 38.2.

Manufacturing process	**Quality according to DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 38.2: Accuracy grades for different manufacturing processes

38.2.7 Niemann geometry data

This is geometry data that is only used to calculate strength as specified by Niemann:

- Shaft bore diameter d_I
- Large hub outside diameter D_2

If you perform the calculation as defined in Niemann, you must also enter additional values. Depending on the position of the load, you can enter the value a0. If a shouldered hub is present, you must also enter the small hub external diameter D_1 and the width of the center part c (with D_2). The following diagram shows how to define these values:

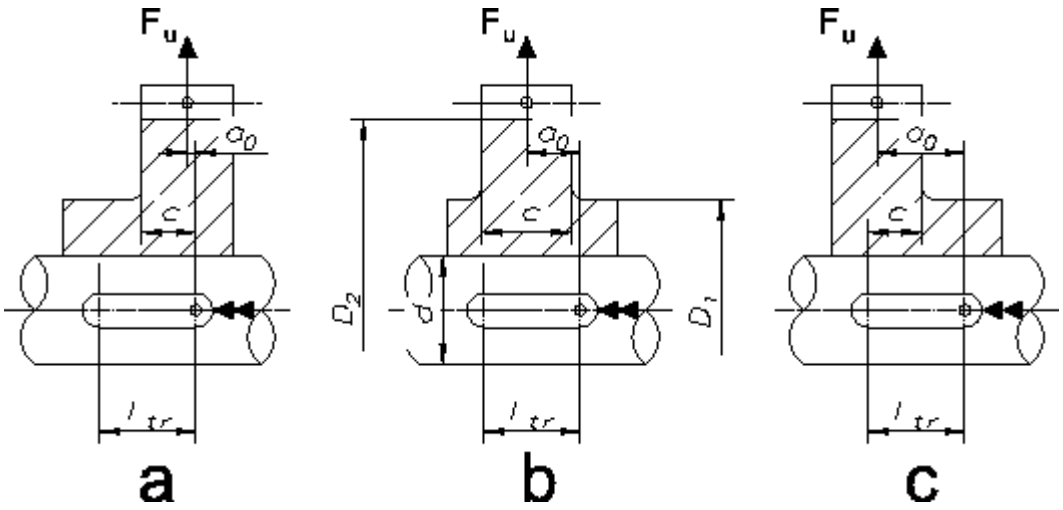


Figure 38.3: Niemann parameter definition

38.2.8 Geometry details

To open the `Define geometry details` window, click the `Details...` button in the upper right-hand part of the Geometry area. Here, you can change the values for the shaft and hub drawing numbers.

38.2.9 Define details of strength

The strength calculation is then performed either according to Niemann [64], DIN 5466 or AGMA 6123-C16. As DIN 5466 is still being developed, it is not described in any further detail here. To perform the calculation according to DIN 5466 and Niemann, you must enter additional data.

The strength method as described in Niemann is described in more detail in the Spline (strength) (see page IV-918) chapter.

38.2.9.1 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 38.2 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336, DIN 3990 and DIN 3991.

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 38.4: Assignment of operational behavior to application factor

38.2.9.2 Strength method as defined by Niemann

Type of loading/Frequency of change of load direction

If you open the `Type of loading` list, you can then select one of the two items shown in it:

1. `No alternating torque`
2. `With alternating torque`

If you select item 2) or 3), the calculation not only defines the `Frequency of change of load direction NW`, as defined in DIN 6892, Figure 6, but also the frequency of change of load direction coefficient f_w. In the case of item 1), the coefficient will be set to 1.0. This data is only used for calculations as described in Niemann.

$p_{eq}=f_w * p_{zul}$

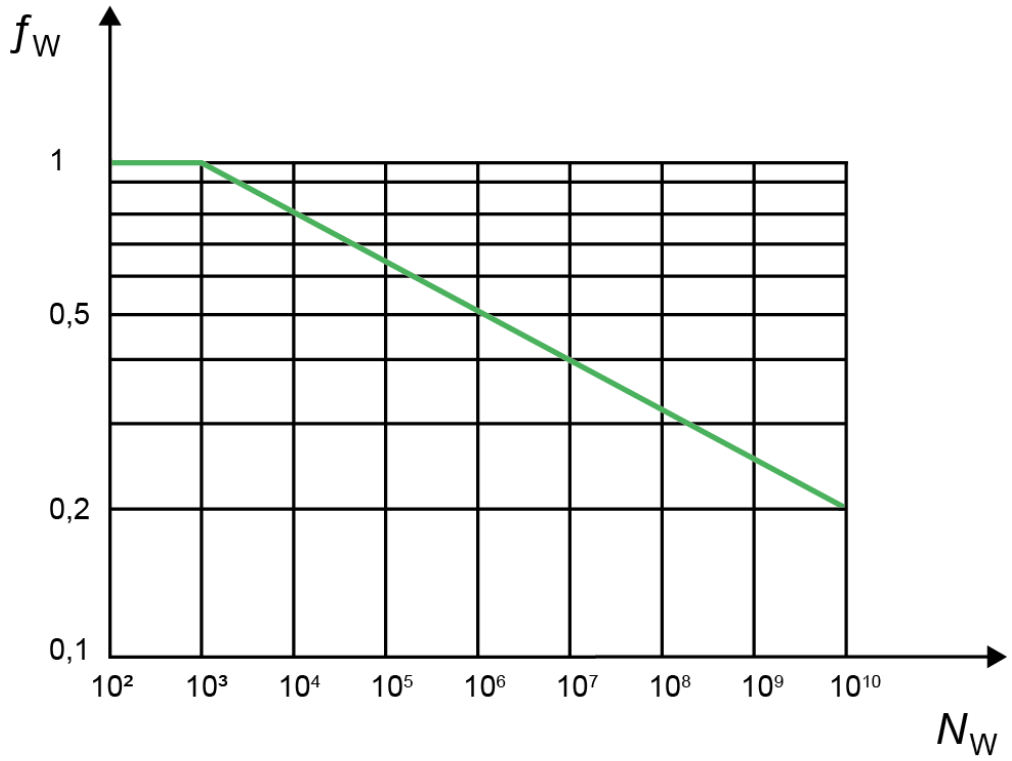


Figure 38.5: Graphic as described in DIN 6892 Figure 6: Frequency of change of load direction coefficient for reciprocal load

Number of load peaks

fL is the Load peak coefficient, which depends on the material type and the `Number of load peaks NL`. This coefficient is shown in a diagram in DIN 6892. This value is needed for calculations as described in Niemann.

For calculation with peak torque:

$p_{max}=f_L * p_{zul}$

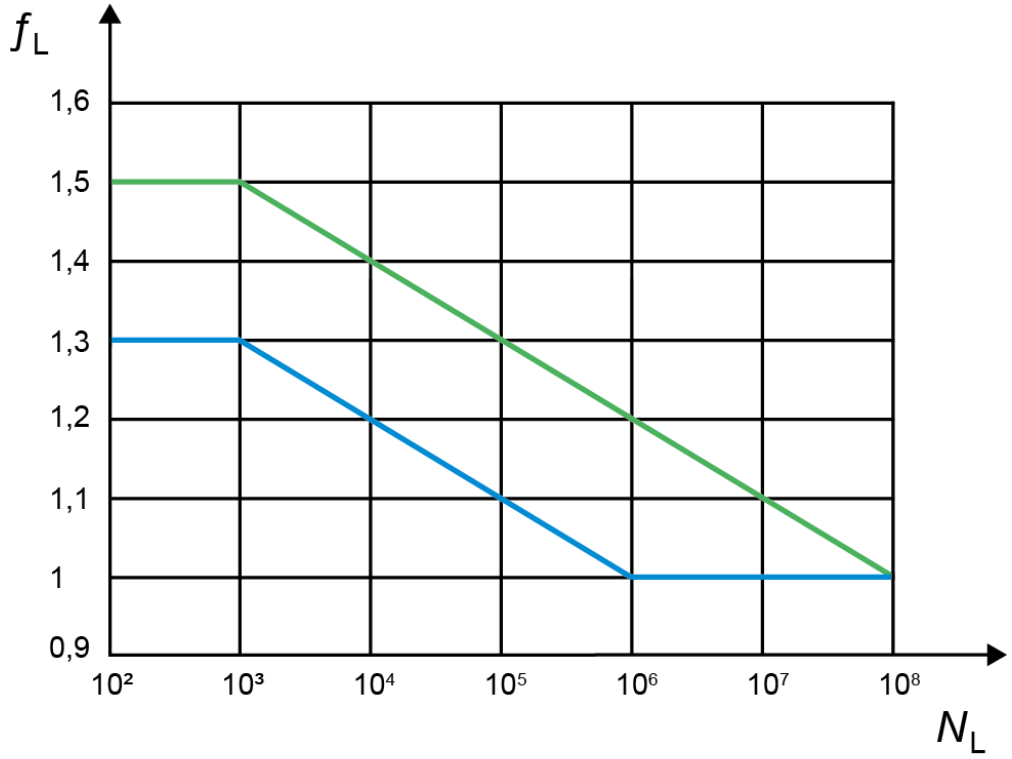


Figure 38.6: Graphic as described in Niemann (DIN 6892 Figure 7): Load peak coefficient

- Green line: ductile material
- Blue line: brittle material

Share factor

If you are working with DIN profiles, the accuracy grade Q (maximum value of shaft or hub) can be used directly as a "Tolerance field according to DIN 5480" to define the value of $k_{\varphi\beta}$ according to the table shown below.

If you are working with ISO and ANSI profiles, the specified quality Q (maximum value) +4 (see "Tolerances" section) is taken and used as the "Tolerance field according to DIN 5480". This value is then used to determine $k_{\varphi\beta}$.

Form-closure	**spline connection with involute flank**
connection	**Tolerance zones according to DIN 5480**

	H5/IT4	H7/IT7	H8/IT8	H9/IT9	H11/IT11	Maximum value
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 38.7: Share factor in accordance with Niemann

NOTE:

The recommended values for $k_{\varphi\beta}$ according to Niemann apply to splined joints which are only subject to torsion. Any additional strain caused by bending or shearing forces can create very many, high local stresses. More precise methods must be used to investigate these cases.

Tolerances for splines

As a comparison, we have calculated an example for the tolerances specified in DIN 5480, ISO 4156 and ANSI B92.1 (spline) and for the tolerances given in ISO 1328 and DIN 3961 (cylindrical gear pair) with accuracy grade 6 (outside) and accuracy grade 7 (inside).

The results for the ISO 4156 accuracy grade are 3 to 4 accuracy grades higher than the same results for the DIN 5480 accuracy grade.

ANSI B92.1 returns the same results as ISO 4156.

This is all very confusing, because all other gears (cylindrical gears with straight or helical teeth, bevel gears, worms, etc.) return accuracy grades for DIN, ISO and AGMA tolerance results that are mostly very similar.

Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

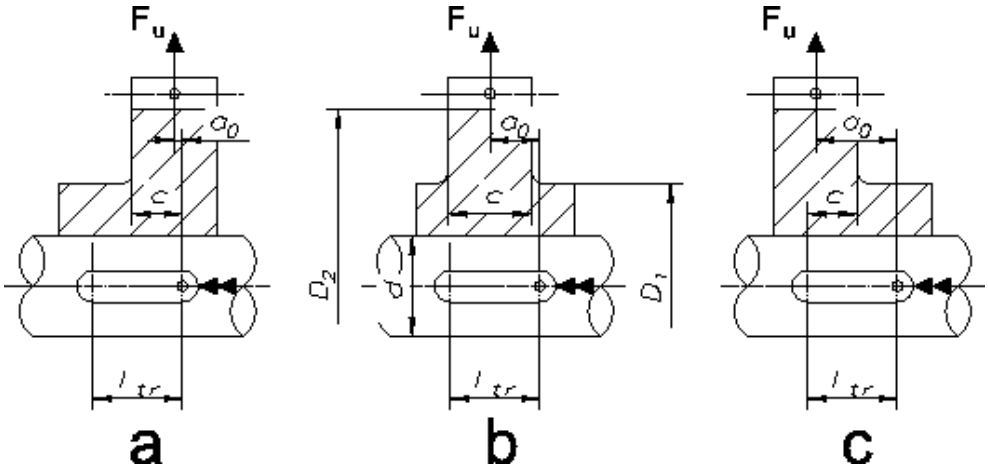


Table 38.8: Length factor according to Niemann

NOTE:

The recommended values for k_1 according to Niemann apply to splined joints which are only subject to torsion. Any additional strain caused by bending or shearing forces can create very many, high local stresses. More precise methods must be used to investigate these cases.

38.2.9.3 Geometries according to DIN 5466

Resulting shearing force

Shearing forces vertical to the shaft axis cause flank contact on both sides of the opposing side of the contact point.

Stress ratio R

Stress ratios are the ratios between under and over stress with regard to a particular type of load, such as torque. Here R = -1 and defines a pure alternating stress ratio, R = 0 defines a pure pulsating stress ratio.

Width and circumferential factor

Click on the checkbox to the right of the input field for one of these factors to enter a value for that factor. Otherwise, this value is calculated automatically and may vary within the range [3, 5]. As these are multiplied together to define the load increase, you can achieve safeties of up to 20 times smaller than is possible with the calculation method defined in Niemann.

38.2.9.4 Strength method AGMA 6123-B06

This standard is intended to be used to size a closed planetary gear unit. However, it also includes a part in which the spline is calculated (section 10.4).

This method calculates a permissible torque for contact stress and a permissible torque for scuffing and wear resistance.

It can be used to calculate both the permitted shearing stress, ssA, and the permitted contact stress, scA, from the core hardness of the material (HRC value).

The calculated permitted torques are then compared with the maximum torque, Tmax, (Tn*KA), to define a safety.

Click on `Calculation/Settings` to define the required safeties. These are then used to determine the safety factors and calculate the sizings.

Additional entries for this calculation method can include defining the alignment type (aligned; not aligned, not width-crowned; not aligned, width-crowned) and the alignment angle.

You can use these values to define the face load factor Fm.

NOTE:

The prerequisite for calculating shearing stress is that the load is carried by half the number of teeth. The prerequisite for performing a calculation against scuffing and wear resistance is that the load (wear) is carried by all the teeth.

If a thinner rim which runs at a faster speed is used, it must be checked for fracture. AGMA 6123 has a calculation for doing this. This calculation is split into three parts: the radial component, the tensile stress and the centrifugal ring stress.

These components can then be used to calculate the overall tensile stress on the ring, which is then compared with the permitted stress (converted from the core hardness) which corresponds to the safety against rim fracture.

NOTE:

A prerequisite for calculating the rim's safety against fracture is that the load is carried by half the teeth.

The formulae for the individual calculations are listed at the end of the calculation report.

38.2.10 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own input` from the list or enter the material in the database (see page I-128) first.

Click the  button to open the `Material pinion center/shaft` window, in which you can select your material from a list of materials that are available in the database. Select the `Own input` option to enter specific material characteristics. This option corresponds to the `Create a new entry` window in the database tool.

38.3 Tolerances

38.3.1 Tooth thickness tolerance

Select one of the options from the `Thickness tolerance` drop-down list.

The allowances for "Actual" (smax, smin, emax, emin) correspond to the individual measurements (base tangent length or measurement over pins measured on the toothing). The allowances for "Effective" (svmax, svmin, evmax, evmin) correspond to the measurement with gauges (all teeth checked together). The gear backlash of a spline connection is therefore derived from the "Effective" allowances. The effective allowances include not only the tooth thickness allowances of individual teeth but also a pitch and form error component. The "Effective" allowances are therefore theoretical values, and are smaller (the tooth is thicker) than for the "Actual" allowances.

NOTE

In accordance with the standard, the allowances for tooth thickness (smax, smin) are predefined for the shaft. In contrast, for the hub, the allowances apply to the tooth space (emax, emin).

If the tooth thickness tolerance has been set to your own specific value, you can input svmax for the shaft (maximum "Effective" allowance) to calculate svmin because the relationship applies in this case:

svmin – smin = svmax – smax

In addition, you can then use the flag to predefine the individual measurement allowances for "Actual". However, if this flag is not set, the difference svmax–smax (pitch and form error component), and the tolerance interval smax-smin are set according to the standard for the selected accuracy grade.

The same also applies to the hub.

38.3.1.1 DIN 5480

Unlike ISO 4156 or ANSI 92.1, DIN 5480 has the special feature that sveffmin = svmax always applies to the shaft and eveffmax = svmin to the hub. For this reason, sveffmin and eveffmax are not displayed.

NOTE

The tolerance widths in the specified values for gauges are larger because of Taylor's formula [25].

38.3.1.2 ANSI 92.1 and ISO 4156/ANSI 92.2M

If you have selected `Own input` and then entered your own tooth thickness tolerance value, you must take the following points into account:
You must enter the tooth thickness allowance s_v for the `Effective` tooth thickness for the overall measurement (caliber) to suit the tolerance system that you are using to calculate cylindrical gears. The `Actual` tooth thickness s for single measurements is defined using these equations.

$$s_{\max} = s_{v,\max} - \lambda \tag{38.1}$$

$$s_{\min} = s_{v,\min} - \lambda \tag{38.2}$$

These equations apply to the shaft tooth thickness or to the tooth space of the hub.

38.3.2 Effective/Actual

Click the button to open the `Convert total deviation of tooth thickness Effective(Actual) for shaft window` which uses the corresponding screen to convert the effective/actual tooth thickness allowance. Here, you can enter values either for the base tangent length, measurement over balls or pins, or the tooth thickness.

38.3.3 Shaft/hub: diameter of ball/pin

The implemented DIN 5480, Part 1, contains an extract of the measuring roller diameter as specified in DIN 3977 that must be used here. You can decide whether to extend the list of available roller diameters in the `Z0Rollen.dat` file in the `dat` dat directory in your KISSsoft installation folder.

38.4 Templates

Spline connections are often checked using templates.

Go gauges are always fully toothed (teeth all around the perimeter) and are used to test the **effective** tolerance limit. For hubs this is the **min. effective** tooth space and for shafts this is the **max. effective** tooth thickness.

No-go gauges are always toothed by sector (depending on the number of teeth of the test piece, 2 to 7 teeth located opposite each other) and are used to test the **actual** tolerance limit. For hubs, this is the **max. actual** tooth space and for shafts, this is the **min. actual** tooth thickness. The externally located flanks of each sector are given sufficient clearance (flank relief, see 1 in the Figure), as they cannot be measured exactly.

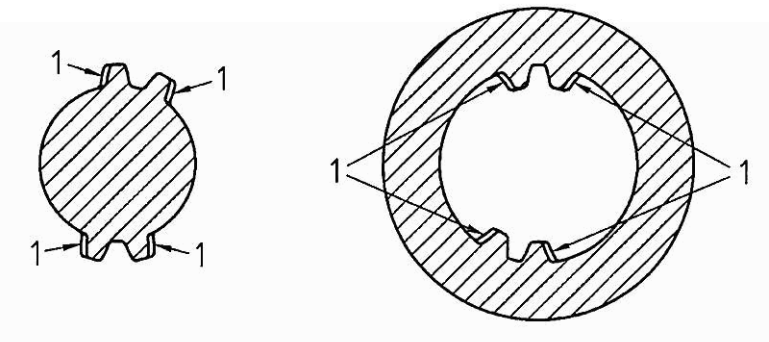


Figure 38.9: Displaying templates

The KISSsoft system can calculate all the gauge allowances specified in ISO 4156 and DIN 5480-15. To do this, select "Reports" and then "Construction of gauges". The system does not automatically calculate the gauge dimensions for profiles that comply with ANSI.

DIN 5480-15 is limited to the pressure angle of 30°. The pressure angles 37.5° and 45° are defined in ISO 4156. DIN 5480-15 contains data for a module range of 0.5 to 10 mm and a maximum number of 100 teeth on the sample. No information is provided in the report for sizes that exceed the data contained in DIN 5480-15.

According to DIN 5480-15, a calculated allowance coefficient **AF1** should be entered for the measuring rollers, if the measuring rollers do not exactly match the specified dimension. Then the distance **AF1** is calculated as a multiplication coefficient on the mean value of the allowances of the two measuring rollers, enabling the tolerance for the distance over rollers to be determined. As this value is not known, this calculation is not performed in KISSsoft.

38.5 Tooth form

If you want to generate the tooth forms of a splined joint, you can select the data you require from the six different tolerance fields (actual, effective) and the three different diameter tolerances displayed in the `Tooth form` tab.

The default settings here are the average allowances for tooth thickness and diameter. After the calculation has been performed, the resulting diameter and tooth thickness are output in the tooth form report.

NOTE

To ensure the selected diameter tolerance is applied to the root diameter, set the flag next to the root diameter allowance in the `Reference profile` tab. If you do not set this flag, only the tooth thickness tolerance will be included in the calculation of the root thickness.

If this flag is not set, the calculation uses a default tolerance for the root diameter as defined in the standard. In other words, the diameter tolerance you selected will not be used to calculate the root diameter.

Chapter 39
Polygons

You use polygon connections to create shaft-hub connections that can withstand very heavy loads. In particular, the low notch effect present in this connection does not reduce shaft strength.

For polygon shafts, you must calculate the load on the shaft and hub (surface pressure). You can also add additional standards.

You can use one of these two methods to calculate the load on the shaft and hub (surface pressure) and to define the safeties:

- Niemann, Volume I (4th Edition) [64].
- DIN 32711-2 (P3G profiles) [84]/ DIN 32712-2 (P4C profiles) [85]

In the calculation according to DIN, only the static load is observed. In the method according to Niemann, the influence of alternating torque can be observed or

load peaks can also be calculated.

39.1 Standard profiles

You can select one of these standards from the selection list:

- DIN 32711-1 (P3G profile)
- DIN 32712-1 (P4C profile)

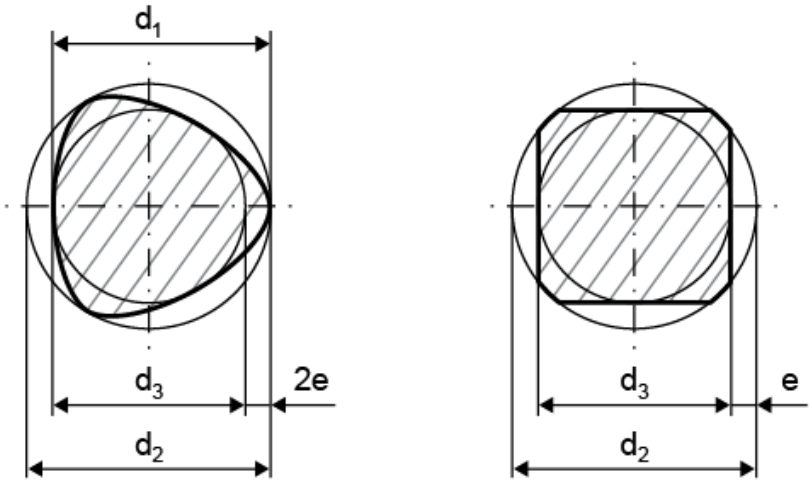


Table 39.1: Profiles

In a P3G profile, which varies according to which standard you select, the list displays the diameter of mean circle d1, the diameter of outer circle d2, the diameter of inner circle d3, the eccentricity e and the coefficient y.
In a P4C profile, the list displays the diameter of outer circle d2, the diameter of inner circle d3, the eccentricity e and the coefficient y.

39.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 39.2: Application factor according to DIN 6892

39.3 Torque curve/Frequency of change of load direction

This influence can only be made to apply using the Niemann calculation method.

When you select the torque curve, you can choose one of two positions:

1. `No alternating torque`
2. `With alternating torque`

If you select item 2), the calculation not only defines the Frequency of change of load direction, as defined in DIN 6892, Figure 6, but also the frequency of change of load direction coefficient f_w. In the case of item 1), the coefficient will be set to 1.0.

39.4 Occurring contact stress

Method in accordance with Niemann:

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

Profile P3G:

p(eq,max)=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

Projection area = l_{tr} * n * 2 * e; (n = 3)

d_1: Diameter of mean circle T: Torque

l_{tr}: supporting length e: Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p(eq,max)=T * 1000/(l_{tr} * (π *dr* er + 0.05 * d2^2))

Projection area = l_{tr} * n * 2 * er; (n = 4)

d_2: Diameter of outer circle T: Torque

l_{tr}: supporting length e: Eccentricity

d_r: Mathematically theoretical diameter e_r: Mathematical eccentricity

d_3: Diameter of inner circle

Method in accordance with DIN:

The following formula is used to calculate the occurrence of contact stress:

Profile P3G:

p=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

d_1: Diameter of mean circle T: Torque

l_{tr}: supporting length e: Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p=T * 1000/(l_{tr} *dr (π * er + 0.05 * dr))

d_2: Diameter of outer circle

l_{tr}: supporting length

d_r: Mathematically theoretical diameter

d_3: Diameter of inner circle

T: Torque

e: Eccentricity

e_r: Mathematical eccentricity

39.5 Permissible pressure

Method in accordance with Niemann:

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:
$p_{eq}=f_s * f_H * R_p$

- for brittle materials:
$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 39.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core hardness for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

Method in accordance with DIN:

The permissible surface pressure on the shaft or hub for polygon profiles P3G and P4C is:

$p_{zul} = 0.9 * R_{p0.2}$

39.6 Materials

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

39.7 Settings

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

39.8 Sizings

During the sizing process, the required value is defined such that the required safety (specified in `Calculations/Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing. Possible sizings:

- Transmissible nominal torque T_n
- Transmissible maximum torque T_{max} (only for Niemann)
- Supporting length l_{tr}

39.9 Graphics

The polygon form is defined using the formulae in the relevant DIN standard (32711-1/ 32712-1) and is displayed as a graphic which can be exported either as a picture file or as a DXF file.

Polygon curve equation (profile P3G, DIN 32711-1):

$$x(\alpha) = \left[Dm/2 - e \cdot \cos(3 \cdot \alpha)\right] \cdot \cos(\alpha) - 3 \cdot e \cdot \sin(3 \cdot \alpha) \cdot \sin(\alpha)$$

$$y(\alpha) = \left[Dm/2 - e \cdot \cos(3 \cdot \alpha)\right] \cdot \sin(\alpha) - 3 \cdot e \cdot \sin(3 \cdot \alpha) \cdot \cos(\alpha)$$

Polygon curve equation (profile P4C, DIN 32712-1):

$$x(\alpha) = \left[Dm/2 - e \cdot \cos(4 \cdot \alpha)\right] \cdot \cos(\alpha) - 4 \cdot e \cdot \sin(4 \cdot \alpha) \cdot \sin(\alpha)$$

$$y(\alpha) = \left[Dm/2 - e \cdot \cos(4 \cdot \alpha)\right] \cdot \sin(\alpha) - 4 \cdot e \cdot \sin(4 \cdot \alpha) \cdot \cos(\alpha)$$

Chapter 40
Woodruff Keys

Connections that use Woodruff keys are no longer commonly used, because the deep keyway in these keys causes too great a notch effect. However, this connection still widely used in precision mechanics.

For Woodruff keys, you calculate the load on the shaft and hub (surface pressure). You can also add additional standards. The load on the shaft and hub (surface pressure) is calculated, and the safeties are determined, as described in classic technical literature ([64]). The calculation defined by Niemann forms the basis of DIN 6892 (key calculation).

40.1 Standard profiles

You can select one of these standards from the selection list:

- `DIN 6888, series A` (high pinion groove)
- `DIN 6888, series B` (low pinion groove)
- `Own input`

After you select the standard for calculating the Woodruff key, a list of corresponding values is displayed.

b: Width d: Diameter

h: Height t_1: Shaft groove depth

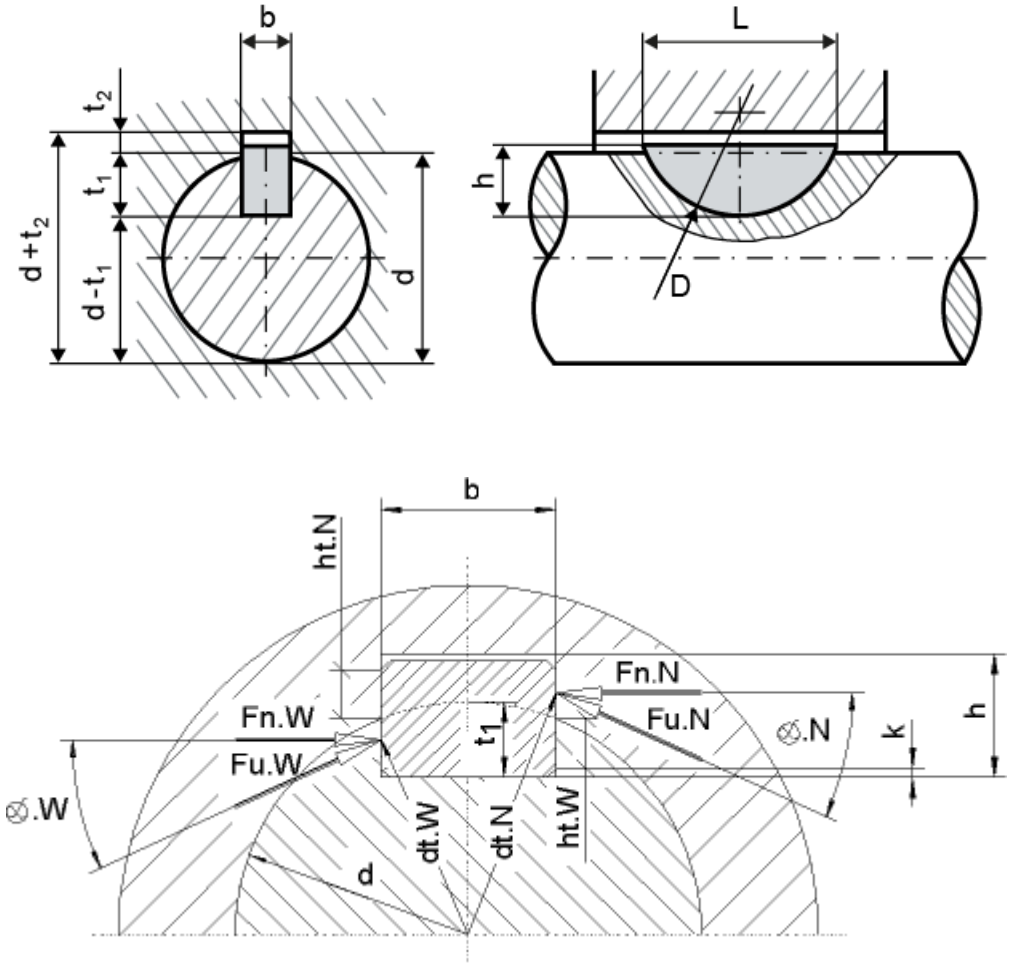


Figure 40.1: Woodruff key with circumferential and normal forces for the calculation as defined in Niemann

Own input: Select the `Own input` option to define your own Woodruff keys.

40.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior	**Operational behavior of the driven machine**			
of the driving	equal-	moderate	medium	strong
Machine	moderate	Shocks	Shocks	Shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 40.2: Application factor according to DIN 6892

40.3 Torque curve/Frequency of change of load direction

When you select the torque curve, you can choose one of two positions:

1. `No alternating torque`
2. `With alternating torque`

If you select item 2), the calculation not only defines the Frequency of change of load direction, as defined in DIN 6892, Figure 6, but also the frequency of change of load direction coefficient f_w. In the case of item 1), the coefficient will be set to 1.0.

40.4 Occurring contact stress

This formula is used to calculate occurrences of contact stress. The formula is used both for the equivalent load and for the maximum load:

$$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d * l_{tr} * h_{tw} * z)$$

$k_{\varphi\beta}$: Share factor

k_1: Length factor

T: Torque

d: Shaft diameter

l_{tr}: supporting length

h_{tw}: supporting height (shaft)

z: Number of Woodruff Keys

40.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large external hub diameter and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

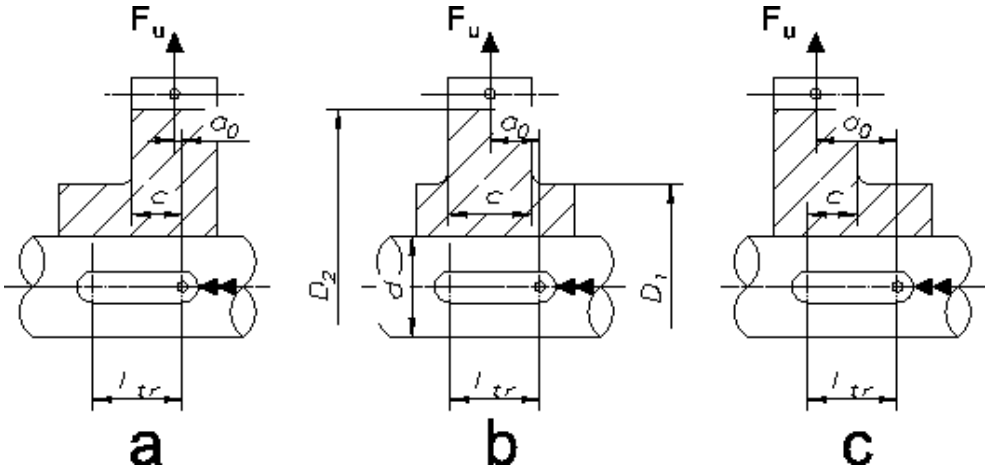


Figure 40.3: Woodruff key: Load application.

40.6 Share factor

To calculate the occurring contact stress, a share factor of $k_{\varphi\beta}$ is taken into account. This is then multiplied by the load. Interim sizes not shown in the table are interpolated linearly.

Form-closure	**spline connection with involute flank**					
connection	**Tolerance zones according to DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 40.4: Share factor in accordance with Niemann

40.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:
$p_{eq}=f_s * f_H * R_p$

- for brittle materials:
$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, through hardening steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 1.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core hardness for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15. Otherwise, it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

40.8 Materials

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes.
You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

40.9 Settings

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

If the `Take pressure on key into account` flag is set, the values of the Woodruff key are included in sizings. Otherwise, the sizing procedure will be carried out on the basis of the shaft and hub.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

40.10 Sizings

During the sizing process, the required value is defined such that the required safety (specified in `Calculations/Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing.

Possible sizings:

- Transmissible nominal torque T_n

Chapter 41
Bolts and Pins

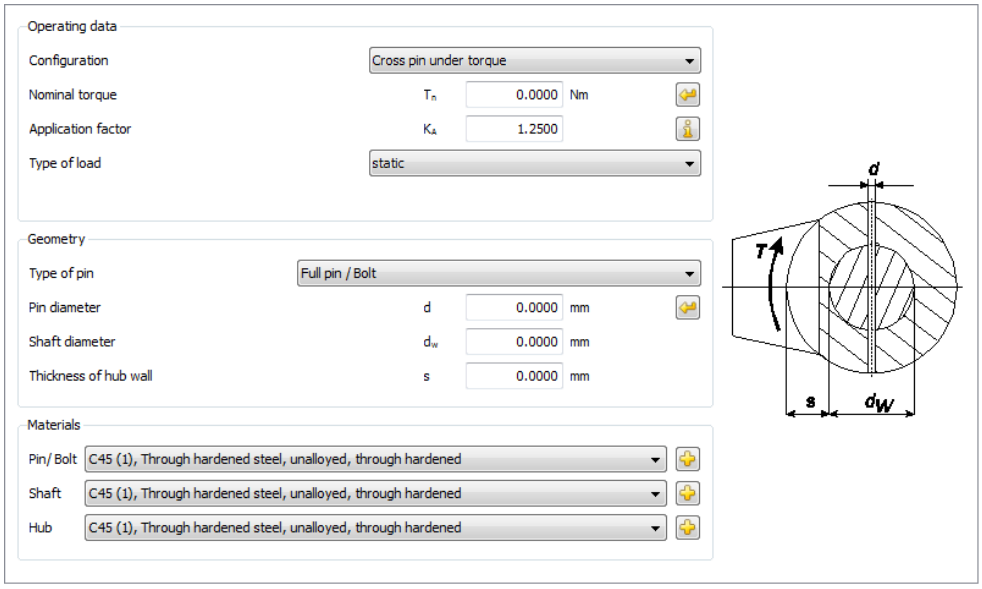


Figure 41.1: Basic data for bolts and pins

The bolt/pin connections are divided into four types of calculation depending on where they are used:

- `Cross pin under torque`
  With cross pin connections where large forces are in play, the surface pressure of the shaft and hub, and the shearing of the pin, will be checked.
- `Longitudinal pin under torque`
  Longitudinal pin connections are subject to surface pressure in the shaft and hub and shearing on the pin.
- `Guide pin under bending force`
  Guide pin connections are subject to bending stress due to torque and to shearing force caused by transverse forces. The shearing force, surface pressure and bending of the pin, and the surface pressure on the element, are calculated here.
- `Bolt connection subjected to transverse load (in double shear)`
  The pin is subject to bending and shear stress and to surface pressure in this ar-

rangement. You can use different calculation methods, depending on the fit of the rod/bolt and fork/bolt connections. Experience shows that the decisive value in sizing non-sliding surfaces is the bending stress. In sliding surfaces, it is the surface pressure.

☼ `Bolts in a circular array (in single shear)`
In this arrangement, the effective torque is distributed uniformly across the individual bolts/pins and therefore the shaft and hub are subject to surface pressure from the individual bolts/pins, creating a shearing force. In addition, the maximum shear stress (calculated as specified in Roloff Matek [62], p. 254) and the minimum safety for the bolts are also output.

The load placed on the bolts, shaft and hub (or part) is calculated, and the safeties are determined, as described in classic technical literature (Niemann, Maschinenelemente I, 4th Edition 2005 [64]),

apart from bolts in a circular layout.

The cross-sectional area and moment of resistance to bending in the spring dowel and coiled spring pins (bushes) is calculated according to Decker [86]. In those configurations where the bolts, spring dowels and coiled spring pins (bushes) are only subjected to shearing, the permitted shearing force specified in the relevant DIN standard can be applied to the pins.

41.1 Influence factors

When calculating individual connections you must include a number of influencing factors which are defined depending on the load, stress, type, etc.:

- Application factor
- Dynamic factor:
  fixed load: Cd = 1; pulsating load: CD = 0.7; alternating load: CD = 0.5;
  for coiled spring pins and spiral pins (bushes)
  fixed load: Cd = 1; pulsating load: CD = 0.75; alternating load: CD = 0.375;
- Reduction factors for full/grooved dowel pin
  Full pin: CK = 1; grooved dowel pin (bending, shear): CK = 0.7; grooved dowel pin pressure: Ckp = 0.8;

Since the permissible stress values in the literature are very low, other material factors have been added to obtain the values in the table.

41.2 Materials

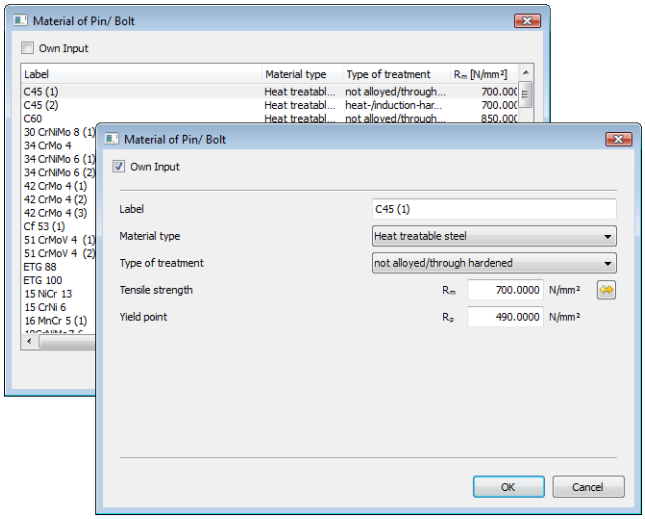


Figure 41.2: Material screen: Bolts and Pins

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

41.3 Settings

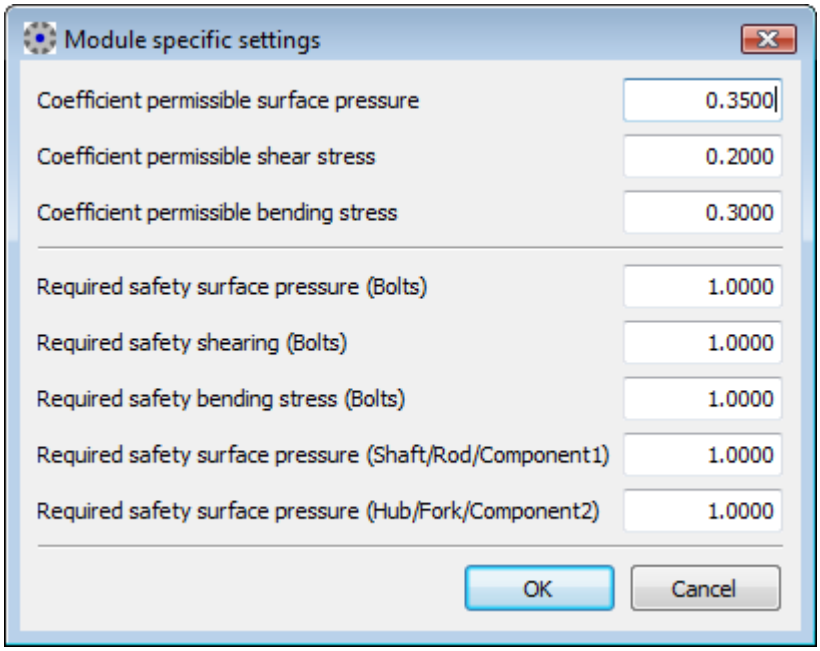


Figure 41.3: Settings for bolts and pins

In this subwindow, you can view and change the materials factors and required safeties for each calculation.

This factor is multiplied by the tensile strength Rm for all parts/bolts and pins apart from coiled spring pins and spring dowel pins (bushes) to calculate the permitted value.

In the case of coiled spring pins and spring dowel pins (bushes), the permitted values are taken directly from the file and do not depend on tensile strength Rm.

41.4 Permitted values

Parts/Full pins/Bolts/Grooved dowel pins

For each part/bolt and pin, depending on the load, the coefficient you find in `Calculation/Settings` is multiplied with the tensile strength Rm to define the permitted value.

Coiled spring pins and spiral pins (bushes)

The permitted values for coiled spring pins and spiral pins (bushes), are imported from a file.

The permitted values for transverse force, for configurations that are only subject to shearing, can be taken from the relevant DIN standard for the pins.

These values for the minimum safety against shearing are displayed in the standards for a case involving double shear. If the calculation involves single shear, the minimum shearing force (double shear) used in the standard can be halved. This was described in the old standard (e.g. DIN 1481:1978).

The permitted values for shear and bending stress under different loads can be taken from the technical documentation provided by Decker:
Bending Stress: $\sigma b = 380 N/mm^2$
Shear stress: $\tau = 160 N/mm^2$
Surface pressure: $p = 208 N/mm^2$

Half the permitted values from other arrangements are used for the arrangement "Longitudinal pin under torque".

(Recommendation in accordance with Decker)

41.5 Sizings

Click the  buttons next to Diameter and Load to size the values that are beside them to suit the required safeties.

Chapter 42
Bolts

KISSsoft calculates bolted joint in accordance with VDI 2230 (2015). In addition to providing tables with standard values, the program also has a range of options that enable you to enter your own definitions for most of the constraint values (such as geometry and material data). Although the VDI 2230 standard does not have iteration functionality, i.e. it can be calculated manually, the flexible input and modification options give you a user-friendly software solution at your fingertips. However, you must be familiar with VDI 2230 before you can interpret the results and enter the required values correctly in the program.

VDI 2230 compares the permissible assembly pretension force (F_M and also, to some extent, F_{Mzul}) with the minimum and maximum assembly pretension force (F_{Mmax} and F_{Mmin}). Here the first is a value calculated with 90% of the bolt yield point and the last two are determined by the loads required to guarantee that the connection functions correctly. Assembly pretension force F_{Mzul} is therefore determined from the strength of the bolt, while assembly pretension forces F_{Mmin} and F_{Mmax} are determined from the function of the connection. The necessary assembly pretension force F_{Mmin} is calculated from the axial force F_A and the resilience of the parts and the bolt ϕ, the embedding loss F_Z, the thermal forces $F_{V\,th}$ and the required clamp load F_{Kerf}. F_{Mmax} can be calculated from Φ_{Mmin} while taking into consideration the coefficient of friction scatter and the tightening technique (tightening factor a_A).

$$F_{M\max} = \alpha_A \cdot F_{M\min} \tag{42.1}$$

$$F_{M\max} = \alpha_A \cdot \left[F_{Kerf} + (1-\phi)\cdot F_A + F_Z + \Delta F_{vth}\right] \tag{42.2}$$

The necessary assembly pretension force F_{Mmax} must now be smaller than the sustainable pretension of the bolt F_{Mzul}. Similar to this comparison is the comparison between the minimum required mounting pretension force F_{Mmin} and the minimum pretension force achieved by tightening at, for instance, 90% of the yield point F_{Mzul}/α_A:

$$F_{M\max} \overset{!}{<} F_M \qquad F_{M\min} \overset{!}{<} \frac{F_M}{\alpha_A} \tag{42.3}$$

42.1 Special features in KISSsoft

In VDI 2230, the values for pretension force F_M when utilizing 90% of the yield point, and for the tightening torque, are to be found in Tables A1 to A4. These values are rounded (rounding off error <= 1%). However, KISSsoft calculates the values using the equations on which these tables are based. The results are therefore more general than the ones that use the values given in the tables and therefore may also differ slightly from them.

42.2 Inputs for Basic data

The entries you make in the Basic Data tab form part of the operating and bolt data, and include the type of bolting, washers, and tightening technique.

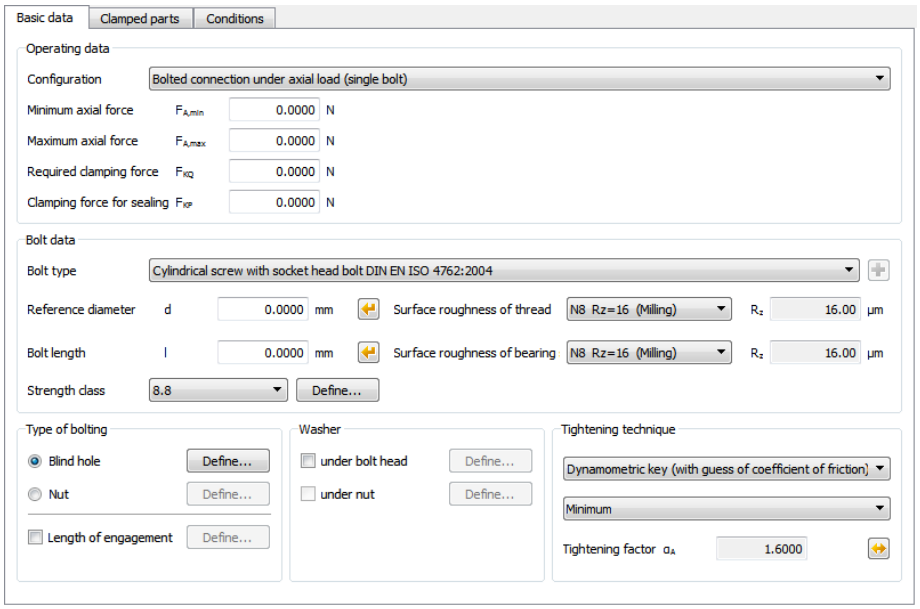


Figure 42.1: Basic data tab.

42.2.1 Operating data

You input operating data in the `Basic data` tab. You can then use it for the following clamping configurations:

1. Bolted joint under axial load
2. Bolted joint under axial load and shearing force
3. Flange connection with torque and forces
4. Multi-bolted joint with any bolt position
5. Proof for bolts with FEM results

An axial loading F_{Amax}, F_{Amin} and a required clamp load F_{Kerf} are determined each time, from the operating data.

External forces and torques, which must be transmitted via the bolted joint, are to be converted into axial force FA and into the required clamp load F_K. VDI 2230

assumes these values are known. In the KISSsoft system, you can input the appropriate configuration (bolt under shearing force and flange connection) as external forces and torques. These values are then used to calculate the axial load and pretension force on an individual bolt.

For a bolted connection under shear load, this shear load is represented by the friction between the bolted parts. The friction is determined by the coefficient of friction and the pretension force. The torsional moment is then applies at the location of the friction radius. This radius is determined from the dimensions of the clamped parts (connecting solids).

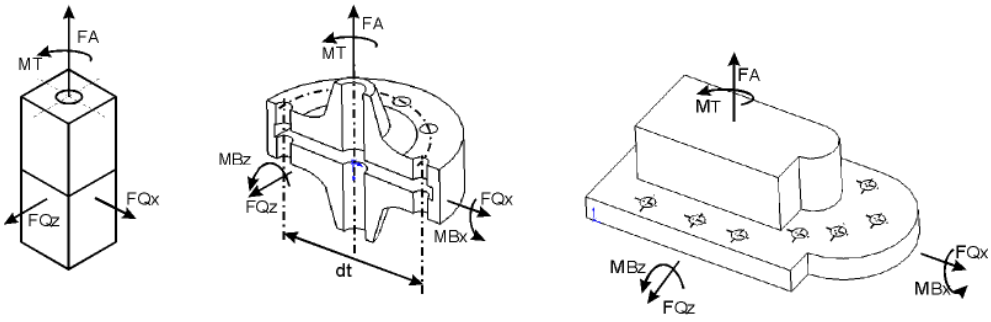


Figure 42.2: Bolting configurations: 1/2, 3 and 4

42.2.1.1 Bolted joint under axial and shearing force

In the second configuration, the required clamp load for axial load transmission is calculated from the shearing force F_Q, the torque M_T, the coefficient of friction m_T, the diameter d_a and the number of force transmitting parting lines q_T:

$$F_{KQ} = \frac{1}{\mu \cdot q_T} \cdot \left(F_Q + \frac{2000 \cdot M_T}{ra} \right) \quad (42.5)$$

$$F_{Kerf} \geq \max\left(F_{KA} + F_{KP}, F_{KQ}\right) \quad (42.6)$$

F_{KQ} Required clamp load for transferring a shearing force and/or a torque

through friction grip (e.g. for friction grip)

F_{KP} Clamp load required to guarantee a sealing function (required when internal pressure is present)

μ_T Interface coefficient of static friction (when shearing force or torques are present), → see Figure on page IV-985.

ra Friction radius resulting from the dimensions of the clamped parts where the torsional moment is acting

42.2.1.2 Bolted joint under axial load

The occurring axial forces F_{Amax} and F_{Amin} are entered directly. The necessary clamp load F_{Kerf} is defined in accordance with

$$F_{Kerf} = \max\left(F_{KA} + F_{KP}, F_{KQ}\right) \tag{42.4}$$

and the required clamp load for axial load transmission F_{KQ} and the sealing function F_{KP} are calculated. F_{KA} is present to prevent gaping in the required clamp load and is calculated by the program.

42.2.1.3 Flange connection with torque and forces

The forces on the single bolt in the case of flanged joints (with stress from torque and/or shearing force and/or bending moment and/or axial force) are calculated according to [63], and also partially according to [61], Example 8.4:

$$F_{KQe} = \frac{1}{n \cdot \mu_T}\left(\frac{2000 \cdot M_T}{dt} + F_Q\right) \tag{42.7}$$

$$F_{KPe} = \frac{F_{KP}}{n} \tag{42.8}$$

$$F_{Kerf} = \max\left(F_{KA} + F_{KP}, F_{KQ}\right) \tag{42.9}$$

$$F_{Bo} = \frac{1}{n}\left(F_{Amax} + \frac{4000 \cdot M_B}{dt}\right) \tag{42.10}$$

$$F_{Bu} = \frac{1}{n}\left(F_{A\min} - \frac{4000 \cdot M_B}{dt} \right) \quad (42.11)$$

dt	Bolt pitch diameter
n	Number of bolts
μ_T	Coefficient of friction between the parts, see Figure 42.3
F_Q	Shearing force on configuration
F_{Amax}	Axial force on configuration (maximum)
F_{Amin}	Axial force on configuration (minimum) Resultant operating force on the bolt that is subject to the greatest stress:
F_{Bo}	Upper limiting value
F_{Bu}	Lower limiting value
F_{KP}	Configuration sealing load
M_B	Bending moment on configuration
M_T	Torque on configuration
F_{Kerf}	Required clamp load
F_{KQ}	Required clamp load (e.g. for friction grip)
F_{KP}	Required clamp load to ensure sealing (atinternal pressure)
F_{KA}	Required clamp load to prevent gaping under eccentric load

If you select a flanged joint configuration, we strongly recommend that you define the geometry of the clamped parts as `individual annulus segments`. The program then automatically generates proposed values for the pitch (t) and the bolt radius of the annulus (r_s).

Experience shows that the results of VDI 2230 are usually very conservative for flanged connections. In order to achieve realistic results, you should increase the coefficient of friction between the parts.

Approximate values for coefficient of static friction μ_T in the interface (VDI 2230)

Material pairing	Coefficient of static friction μ_T	
	dry	lubricated
Steel – steel/cast steel (general)	0.10 ... 0.30	0.07 ... 0.12
Steel – steel; cleaned	0.15 ... 040	/
Steel – steel; case-hardened	0.04 ... 0.15	/
Steel – GJL	0.11 ... 0.24	0.06 ... 0.10
Steel – GJL; cleaned	0.26 ... 0.31	/
Steel – GJS	0.10 ... 0.23	/
Steel – GJS; cleaned	0.20 ... 0.26	/
GJL – GJL	0.15 ... 0.30	0.06 ... 0.20
GJL – GJL; cleaned/degreased	0.09 ... 0.36	/
GJS – GJS	0.25 ... 0.52	0.08 ... 0.12
GJS – GJS; cleaned/degreased	0.08 ... 0.25	/
GJL – GJS	0.13 ... 0.26	/
Steel – bronze	0.12 ... 0.28	0.18
GJL – bronze	0.28	0.15 ... 0.20
Steel – copper alloy	0.07 ... 0.25	/
Steel – aluminium alloy	0.07 ... 0.28	0.05 ... 0.18
Aluminium – aluminium	0.19 ... 0.41	0.07 ... 0.12
Aluminium – aluminium; cleaned/degreased	0.10 ... 0.32	/

Figure 42.3: Interface friction grip coefficients according to [75]

42.2.1.4 Multi-bolted joint with any bolt position

In a multi-bolted joint you can define bolts in any position. These are then affected by shearing force, a bending moment in two directions, and a torsional moment. The load distribution on the bolts is calculated on the assumption that rigid plates are connected by springs at the bolt positions. Forces which do not affect the center of gravity must be moved to the center of gravity so they can be entered. By using a stiffness coefficient, you can model different bolt diameters (doubled diameter equals fourfold rigidity). Set the coefficient = 1 for the bolt with the smallest diameter. For larger bolts, use their diameter ratio to the smallest bolt as the coefficient. The forces are then distributed across the bolts. Only one bolt where the coefficient = 1 can be proved because otherwise the bolt data is incorrect.

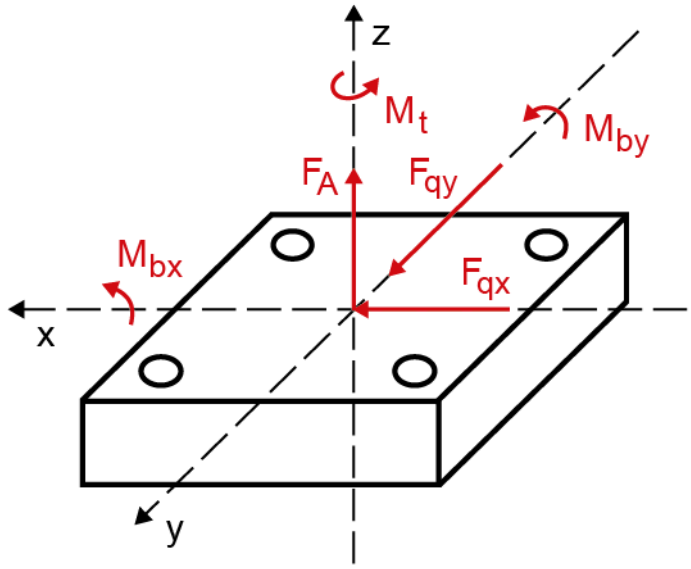


Figure: Definition of forces and torques in KISSsoft

Once you have entered the operating data in the `Basic data` tab, you can define the bolt positions in the `Position of bolt` tab. You can either enter the bolt positions in a table or import them from a file. The resulting axial forces, and also the clamp loads required to transmit shearing force, are also displayed in the table.

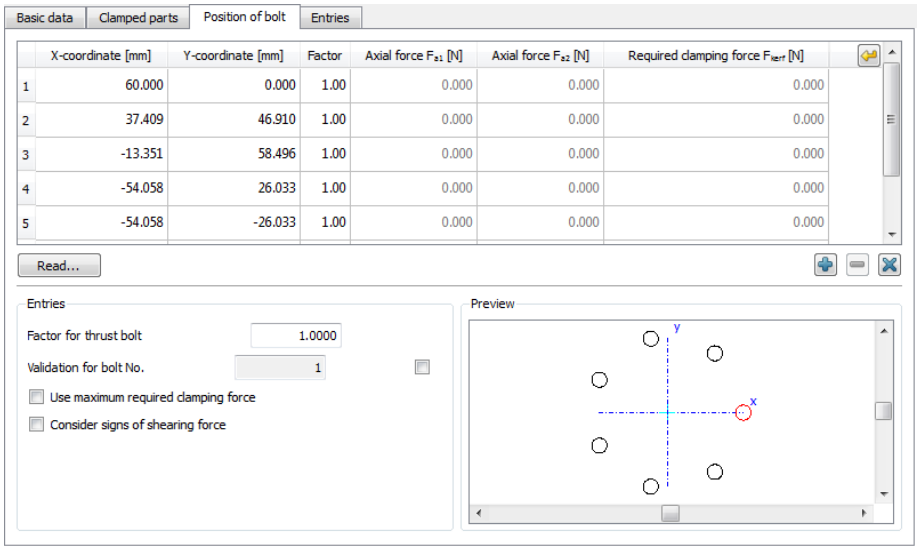


Figure 42.4: Position of bolt tab

Optionally, you can define an additional coefficient for set screws, in which it is assumed that compression is transmitted directly via the plates. However, use of this coefficient assumes you have expertise in this area. In [62], under the keyword "Multi-bolted Plate Joint", for example, an average pressure point of 1/4 plate height is assumed. You use the coefficient for set screws to set this status.

The program then automatically selects the bolt with the highest axial force for the calculation. As a rule, and to ensure that the calculation results are on the safe side, the maximum required clamp load is used for all bolts. However, you can select a further option to deactivate this function.

When you calculate the necessary clamp load, you can also take the sign of the shearing force into account. Shearing forces caused by torsion and shearing force are then added at specific points and subtracted at other points. You should only include the sign if you know the direction of the shearing force and if this force is constant.

To prevent you having to specify the arrangements in which the incremental distances between the bolts repeat themselves, for every single bolt, you can define the position of particular bolts in the Sizing function. Click the  button in the table (above, on the right) in the "Position of bolt" tab to open this window, in which you can enter different configurations.

You can enter these values here:

- line (values for: starting point, end point, number of bolts)
- circle (values for: center point, radius, number of bolts)
- circle segment (values for: radius, starting angle, end angle, number of bolts)

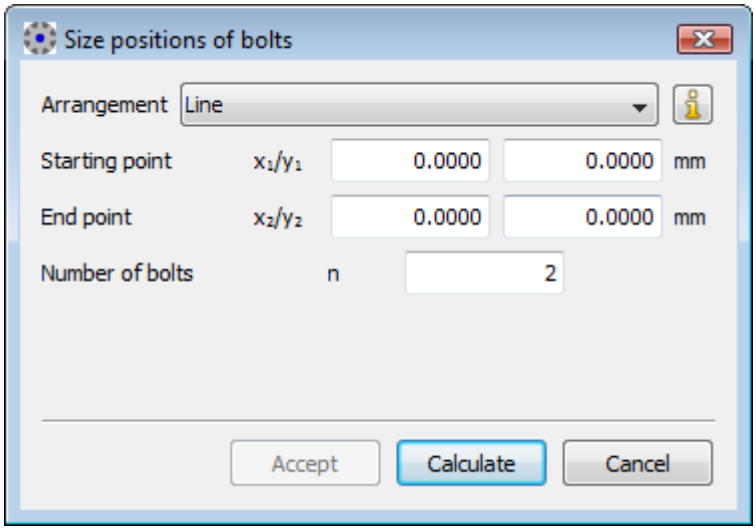


Figure 42.5: Sizing the position of a bolt

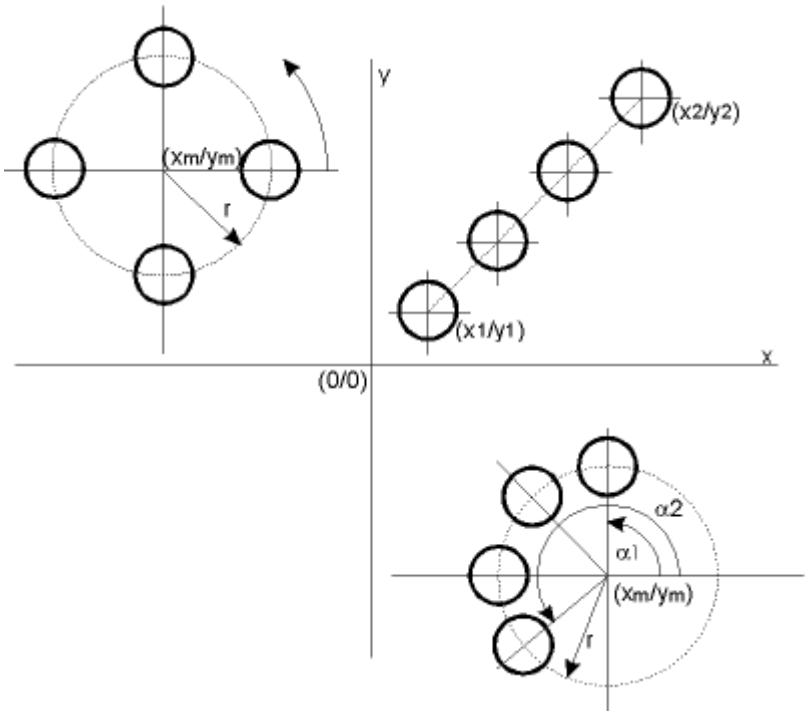


Figure 42.6: Position sizing options

You can add positions by transferring existing positions. However, if you only want to use the positions shown in the configuration, you must delete all the others.

42.2.1.5 Results from FEM calculation

The following behavior can be taken into account in an FEM calculation (instead of a calculation according to VDI 2230 Sheet 1):

- Non linear material behavior (for example, plasticity or yield)
- Non linear boundary conditions (for example, load dependency of the contact surfaces),
- Geometric non linear behavior (for example, significant deformation occurs when the tightening process is simulated)

As specified in VDI 2230 Sheet 2, Multi bolted joints, the FEM modeling is subdivided into 4 model classes:

Model class I:

The bore is not represented. The interface is either completely integrated or connected by a rigid coupling and a reference node for every bolt.

Model class II:

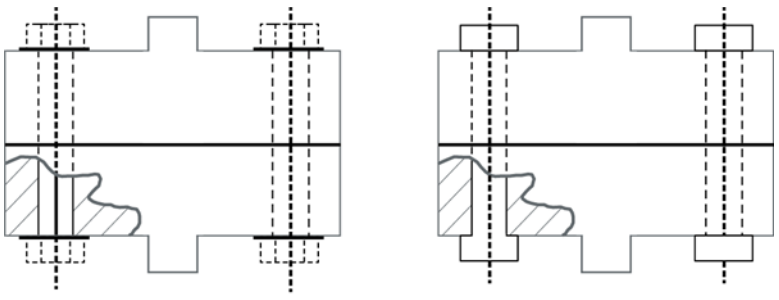
The bolt is represented as a beam or a spring element (with the ideal translational and rotational degrees of freedom for each section). Interface with contact definition.

Model class III:

The bolt is shown in 3D as a cylinder with large cylinders at its ends. The interface and support areas are assigned contact definitions in the FEM calculation. The cylinder has the same core diameter as the thread.

Model class IV:

The bolt is modeled with the exact thread geometry. The assembly and working states are treated separately.

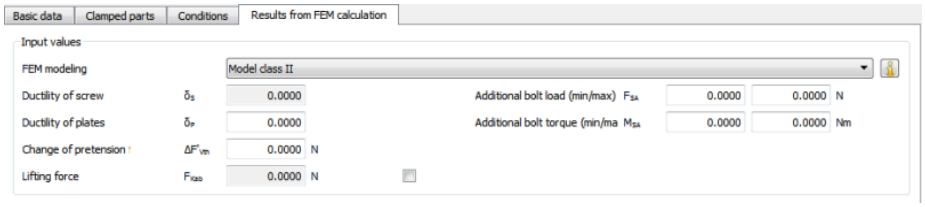


Modellklasse / model class II Modellklasse / model class III

Using VDI 2230 Sheet 1 to prove FEM results is really only a sensible idea for model classes II and III. Model class I does not supply enough informative results

to create a proof according to VDI 2230 Sheet 1. If you are using model class IV, you already have all the necessary results and should use a different method from the one in VDI 2230 Sheet 1 for the proof.

Implementation in KISSsoft – VDI 2230 Sheet 2 FEM results



You can select these model classes directly from the list.

List items model class II and model class III:

the main difference between the models is that bolt resilience can only be defined in model class III.

In the case of these list items, the "Bolted joint under axial and shearing force" configuration is required for calculating the amount of embedding.

The integration of FEM results in the calculation according to VDI 2230 Sheet 1. So that a proof can be performed, this is implemented as follows in KISSsoft (subdivided as specified in VDI 2230 Sheet 1).

R0, R1	according to VDI 2230 Sheet 1
R2	You can input the clamp load for the sealing function FKP directly in the basic data, if it has already been determined in the FEM calculation. The lifting force can be defined directly in the FEM calculation, and specified here. If the results have not been defined, the program determines them on the basis of the values in VDI 2230 Sheet 1. The axial load transmission FKQ is determined as specified in VDI 2230 Sheet 1.
R3	In model classes II and III, the resilience of the plates is taken from the results of the FEM calculation. In model class III, the bolt resilience is also taken from the results of the FEM calculation. In model class II, the bolt resilience is derived as specified in VDI 2230 Sheet 1. The amounts of embedding are estimated according to VDI 2230 Sheet 1.
R4	Amounts of embedding defined as specified in VDI 2230 Sheet 1. Fv'th can be input directly as a result from the FEM calculation.
R5	Fv'th can be input directly as result from the FEM calculation. Fkerf, φen, Famax and Fz can be taken from VDI 2230 Sheet 1 -> determination of FMmin.
R6	Fmmax = αA*FMmin.

R7	In KISSsoft, assembly stress is calculated according to VDI 2230 Sheet 1. This would take an exceptional amount of time and effort if an FEM calculation were used instead.
R8	FV'th is included in the calculation of Fsmax (total bolt load). If FV'th > 0, it is set to 0, as specified in R8/1, VDI 2230 Sheet 1. All others σz, τmax, σredB are calculated according to VDI 2230 Sheet 1.
R9	FSA, MSA as input from the FEM results. You can specify the upper and lower limit. Proof of dynamic strength according to VDI 2230 Sheet 1 (R9/2). The permitted values are defined according to VDI 2230 Sheet 1. The effects of temperature are included directly in the FEM results.
R10	pBmax can be derived from the FEM results if model class III is being used, otherwise it is calculated according to VDI 2230 Sheet 1. You should only calculate the values for permissible surface pressure pGmax directly in FEM if these values are not already available.
R11	The minimum length of engagement cannot be represented realistically in model classes I to III. It would take a great deal of time and effort to model this in model class IV. The calculation is performed according to VDI 2230 Sheet 1.
R12	The calculation In KISSsoft is performed according to VDI 2230 Sheet 1. The values you need to input here are determined from the FEM calculation.
R13	not applied.

List item model class III (only forces and torques, without resiliences):

The main difference between this and the other methods is that the calculation is performed without defining the resiliences.

When this list item is used, the amount of embedding is permanently set to n=1.

The FEM results needed to create a proof according to VDI 2230 Sheet 1 are grouped in the summary of the calculation steps below:

R0	The geometry is to be defined according to VDI 2230 in FEM. In KISSsoft, set this to "Plates" for clamped parts.
R1	Tightening factor according to VDI 2230 Sheet 1
R2 to R6	Are determined in the FEM model. The results are: FM, FM/α (may be different, because this is a real value), FKP, FSmax, FKR. FKQ which is determined with the values FQ, Mt and ra, is used to calculate safety against sliding.

R7	Mounting conditions: FMzul = FMtab at 90% load, otherwise σredMzul =ν*Rp0.2. With FM value for calculating σredM from FE.
R8	Working conditions: FSmax is a result in the FEM calculation. σz, τmax, σredB are calculated according to VDI 2230 Sheet 1. SF = Rp0.2/σzmax
R9	FSA and MSA values from the FEM results. You can specify the upper and lower limit. Proof of dynamic strength according to VDI 2230 Sheet 1 (R9/2). The permitted values are defined according to VDI 2230 Sheet 1. The effects of temperature are included directly in the FEM results. σAbo = FSA,o/As + MSA,o/Ws; σAbu = FSA,u/As + MSA,u/Ws SD = σAS/σab σAS calculated according to the formulae in VDI 2230 Sheet 1.
R10	Input values: permissible surface pressure pGzul (under head and nut). Mounting: pMmax = FMzul/Apmin Operation: pBmax/Apmin SP = pGzul/PM,Bmax
R11	Minimum length of engagement is not calculated.
R12	FKR is an input value. FKerf is calculated according to VDI 2230 Sheet 1. FKQ interim results are calculated from input values FQ, Mt and Ra. SG = FKR/FKerf
R13	not applied.

42.2.2 Bolt data

The type, geometry, surface roughness and strength class of a bolt can all be defined as bolt data.

`Bolt type:` You can use the following standard bolt descriptions from the database to define the bolt type:

DIN EN ISO 4762/	Hexagon socket head cap screw
DIN 912	Standard thread M1.6 to M64
DIN 7984	Hexagon socket head cap screw with low head
	Standard thread M3.0 to M24.0
DIN EN ISO 4014/	Hexagon headed bolts with shank (formerly DIN 931 T1)
DIN EN 24014	Standard thread M1.6 to M64
DIN EN ISO 4017/	Hexagon headed bolt with thread to head (formerly DIN 933)
DIN EN 24017	Standard thread M1.6 to M64
DIN EN ISO 1207/	Slotted cheese head screw
DIN 84	Standard thread M1.0 to M10
DIN EN ISO 8765	Hexagon headed bolt with shank
	Fine thread M8.0 to M64
DIN EN ISO 8676	Hexagon headed bolt without shank
	Fine thread M8.0 to M64
DIN EN 1662	Hexagon headed bolt with flange, light series form F
	Standard thread M5.0 to M16
DIN EN 1662	Hexagon headed bolt with flange, light series form U
	Standard thread M5.0 to M16
DIN EN 1665	Hexagon headed bolt with flange, heavy series form F
	Standard thread M5.0 to M20
DIN EN 1665	Hexagon headed bolt with flange, heavy series form U
	Standard thread M5.0 to M20
ASME B18.2.1	Square bolts, UNC thread, 0.25-1.5in
ASME B18.2.1	Hex bolts, UNC thread, 0.25 to 4 in
ASME B18.2.1	Heavy hex bolts, UNC thread, 0.5-3 in
ASME B18.2.1	Hex cap screws, UNC thread, 0.25-3 in
ASME B18.2.1	Heavy hex screws, UNC thread, 0.5-3 in

`Reference diameter:` You can either input your own value for the reference diameter or click the  button to calculate an approximate value after you input the operating data. This sizing function usually leads to bolt diameters that are too large. We therefore recommend you input a value that is 1 or 2 standard sizes less than the system's proposed value.

`Bolt length:` You can input any bolt length if you are inputting your own bolt geometry. Otherwise, after you input the bolt length, the system sets it to the next standard length.

`Surface roughness of thread/head bearing area:` The surface roughnesses influence the amount of embedding and therefore also the preload loss of the bolted joint.

`Strength class:` After the entry for standard strength classes, you can click the `Define...` button to define your own strength values. The shearing strength ratios are set according to Table 5.5/2 in VDI 2230 Sheet 1 (2015), according to strength class. The values can be overwritten.

`Own definition of bolt geometry:` To define your own bolt geometry, you must set the `Bolt type` selection list to `Own input`. This activates the `Define...` button, which you can click to input your own bolt geometry values.

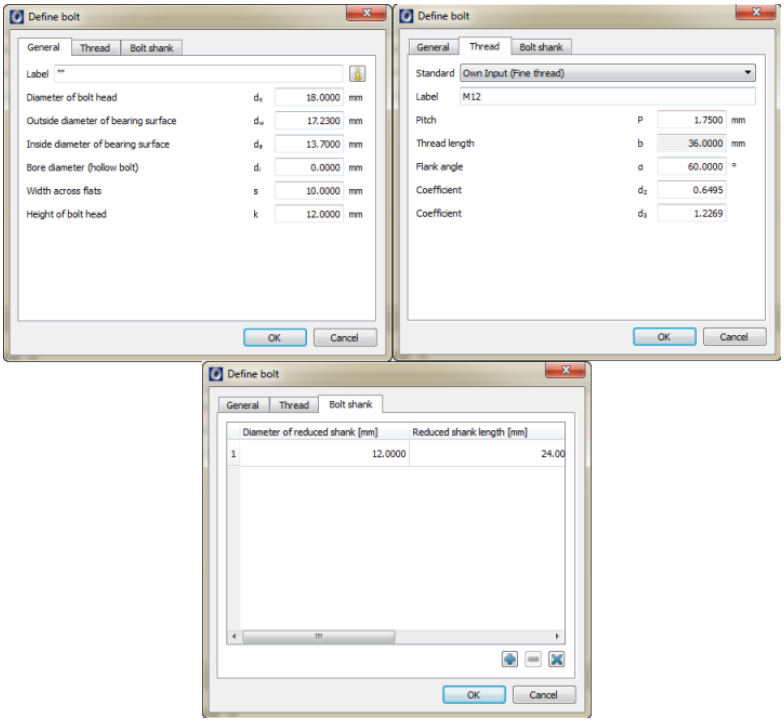


Figure 42.7: Dialog with three tabs for defining your own bolt geometry.

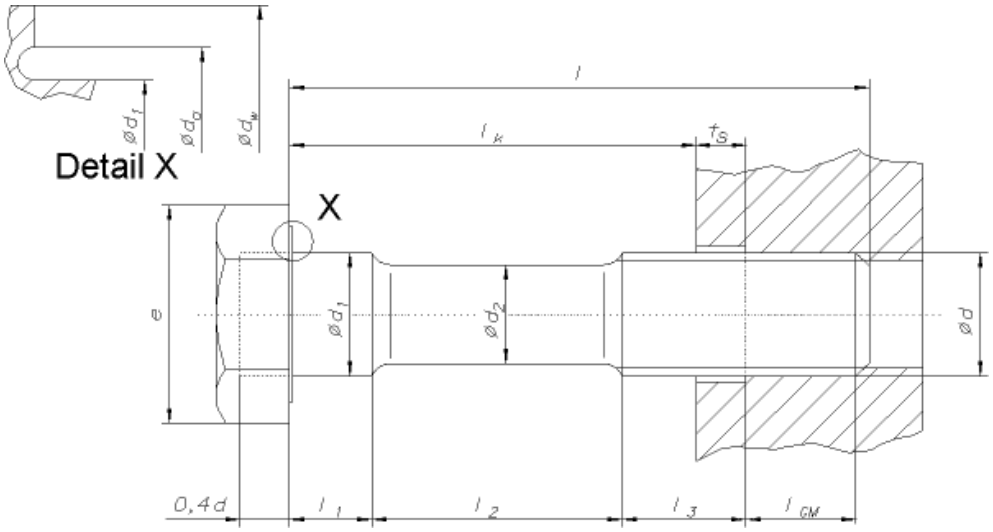


Figure 42.8: Bolt geometry

- **General** tab: If you are using a bolt with a bore, input the dimensions of the bolt head as well as the bore diameter.
- **Thread** tab: Data from the standard, the size of the thread, the lead, and the thread length. This is where you define the factors used to calculate the flank diameter d_2 and the core diameter d_3 (d_2 = d - $d_2$factor*P; d_3 = d - $d_3$factor*P).
- **Bolt shank** tab: Data about the individual bolt cross sections. Click the  button to add a new cross section. Click the  button to remove the selected cross section. Click the  button to delete all the cross sections.

42.2.3 Type of bolted joint

To define the type of bolting, enable either `Nut` or `Blind hole`. This corresponds to the difference between through-bolt and single-bolted (tapped thread) connections as defined in VDI. Click on the appropriate `Define...` button to open the appropriate dialog, in which you input additional data about the nut or threaded part.

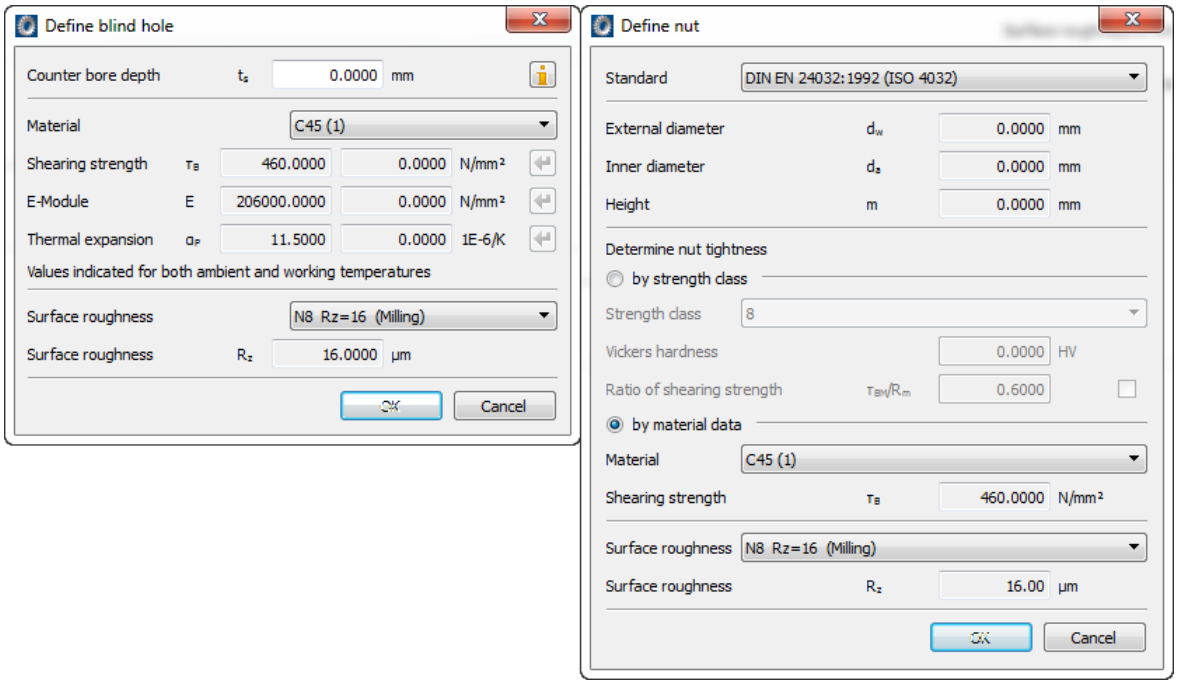


Figure 42.9: Input dialog for data about cut threads and nuts

Blind hole

For cut threads, the counter bore depth t_s describes milling without thread that is primarily used to elongate the clamp length (see also Figure on page IV-995).

Nut

In the nut definition screen you can either select a standard for the geometry or define one yourself.

For example, when calculating the length of engagement, you can either define the hardness from the strength class (as specified in DIN EN ISO 898-2) or define the shearing strength directly from the material.

The "Own input" option is also available in both variants. However, when you input the strength class, you must also define the ratio of the shearing strength to tensile strength (τBM/Rm).

The system then converts the hardness value you enter here into tensile strength as part of the hardness conversion process. The tensile strength Rm is then multiplied by the ratio (τBM/Rm) to calculate the shearing strength (τBM).

The minimum hardness value for nuts with a standard thread (including UNC) is taken from the strength class in Table 6 in DIN EN ISO 898-2. The minimum

hardness value for fine threaded nuts (including UNF) is taken from the strength class in Table 7 of the same standard.

NOTE

If the dimension of the interface area DA is only slightly larger than the bearing diameter of the bolt head dw, it must be calculated as a through-bolt connection (note the deformation cone).

(DA to ~1.4*dw)

42.2.4 Washer

If this flag is set, a washer is inserted between the nut and the part and/or the head and part.

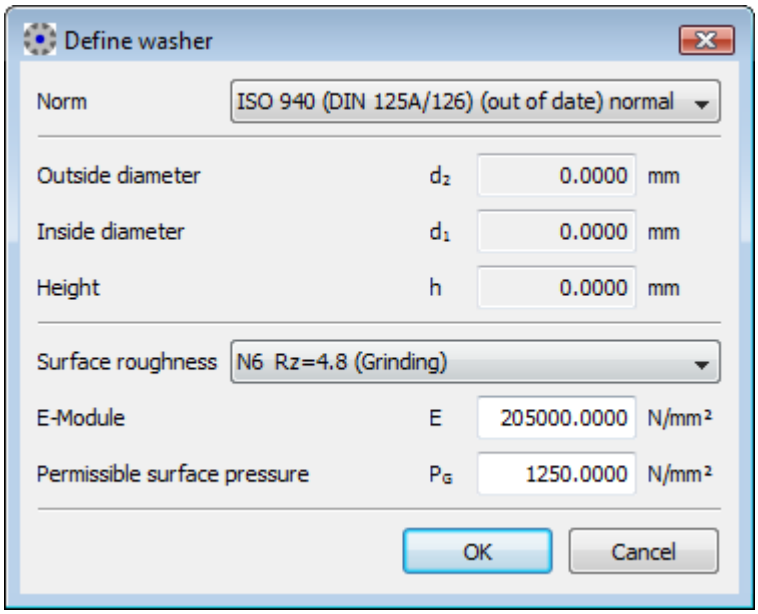


Figure 42.10: Defining washers.

In the `Calculations/Settings` tab, if you select the "Determine specific thermal expansion of washers" flag, you can also define the thermal expansion coefficients that are used to calculate the difference in pretension force. You will find a more detailed description in the "Settings" section.

42.2.5 Extension sleeves without external forces

You can specify the length of an individual extension sleeve in the extension sleeves dialog. In a single-bolted (tapped thread) joint, a extension sleeve can be

used under the bolt head. In a through-bolt connection, an extension sleeve can be used under the bolt head and under the nut.
No external forces are to act on the extension sleeves defined here. These sleeves are to be used to create the distance between the bolt head or nut and the part.

The extension sleeves are taken into account when sizing the length, calculating the resilience, and in the length expansion at operating temperature.

42.2.6 Tightening technique

Uncertainties such as, for example, the scatter of coefficients of friction, differently precise tightening techniques, instrument, operating and reading errors result in the scatter of the achievable assembly pretension force. For this reason, oversizing the bolt is necessary, and is expressed by the tightening factor $\alpha_A = F_{Mmax}/F_{Mmin}$. If the required minimum pretension force F_{Mmin} remains constant, then an increasing tightening factor α_A means that the bolt must be sized for a greater maximum assembly pretension force F_{Mmax} (due to the greater scatter). Tightening technique and associated tightening factors:

Tightening factor α_A	**Tightening technique**	**Adjusting technique**
1.0	Yield point-determined tightening mechanically or manually	
1.0	Angle of rotation-controlled tightening mechanically or manually	Experimental determination of the preload moment and angle of rotation
1.2 to 1.6	Hydraulic tightening	Adjustment by means of measuring length or pressure
1.4 to 1.6	Torque-controlled tightening with a torque wrench, torque indicating wrench or a precision torque wrench with dynamic torque measurement	Experimental determination of the required tightening torques on the original bolting part, e.g. by measuring the length of the bolt
1.6 to 1.8	ditto	Defining the nominal tightening torque by estimating the friction factor, value (surface and lubrication ratios)
1.7 to 2.5	Torque-controlled tightening with a bolt installation spindle	Torque wrench adjustment with a tightening-up moment, set to the required tightening moment (for an estimated friction factor, value) plus a supplement.
2.5 to 4	Pulse-controlled tightening with	Torque wrench adjustment with

	an impact wrench	tightening-up moment as described above

42.3 Data input for clamped parts

The `Clamped parts` screen displays data about the materials and geometry of the clamped parts (connecting solids), the distances involved for eccentric loading/clamping and data about the load application factor.

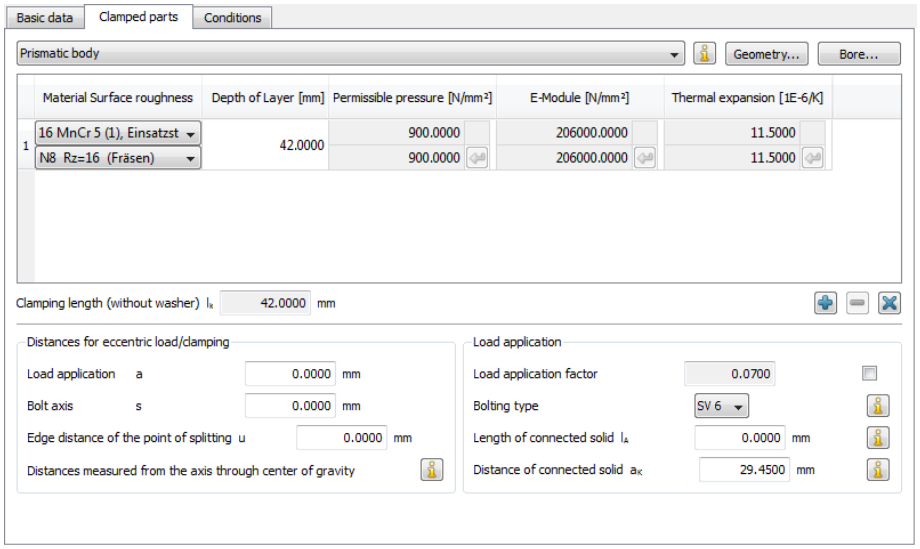


Figure 42.11: Clamped parts (connecting solids) input tab

42.3.1 Geometry of clamped parts (connecting solids)

There are several basic types of clamped parts (connecting solids):

- Plates
- Cylinder
- Prismatic body
- Annulus segment

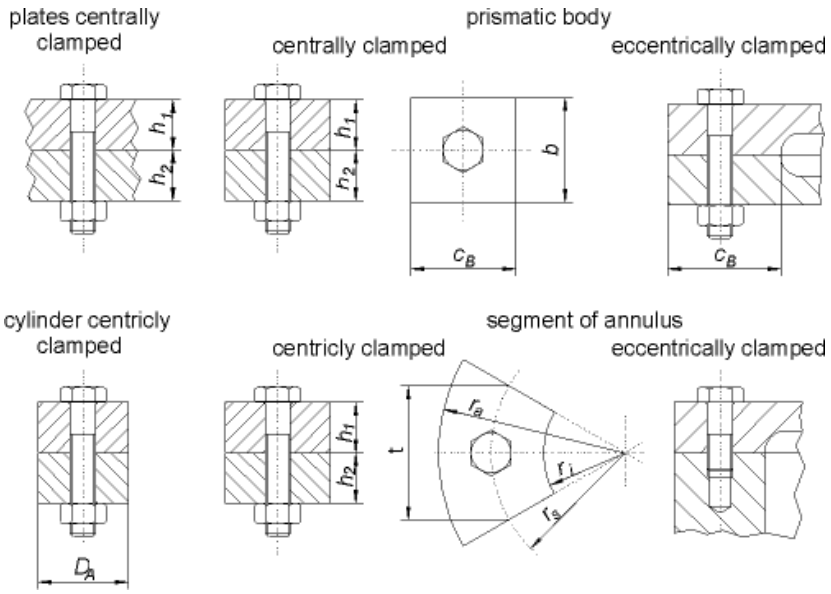


Figure 42.12: clamped parts (connecting solids).

If you select Plates, it is assumed that the clamping deformation cone will be able to expand freely sideways. For all the other selection options, click the `Geometry` button to enter the type of clamped part you want to use in the calculation.

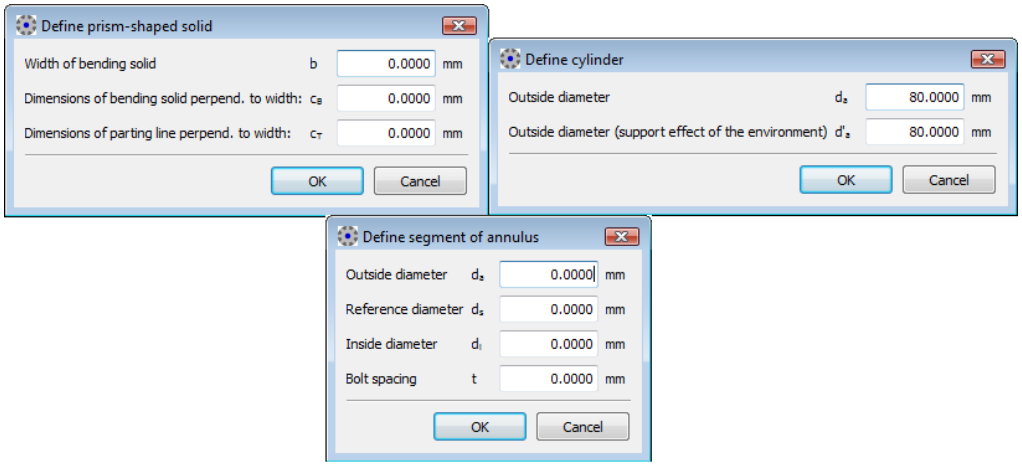


Figure 42.13: Geometry inputs for the cylinder, prismatic body and annulus segment.

Click the `Bore` button to define a threadless through-bore in the part. You can also define chamfers under the head or nut here. These chamfers are then included when the bearing areas are calculated. The chamfer reduces the outside radius of the bearing area, therefore increasing the surface pressure.

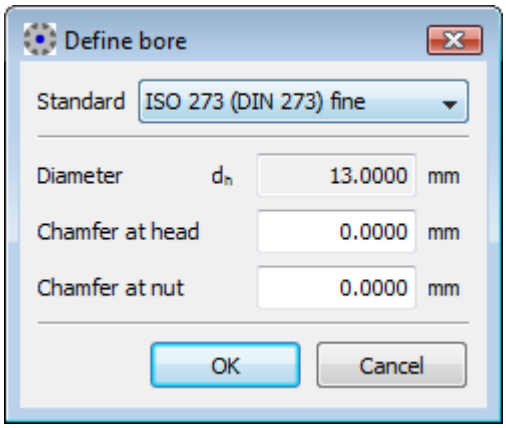


Figure 42.14: Defining through-bores and chamfers under head and nut.

You simply enter the different material situations in the list. The upper values for the permissible pressure, Young's modulus and thermal expansion are material values that apply at room temperature and, unless they are values you have entered, are always shown with a gray background. If the "Calculate temperature dependent material data automatically with estimation formulae" flag is set in `Calculations/Settings`, the values for the operating temperature are calculated empirically and displayed in the lower half of the particular material. You cannot edit these values. If the flag is not set, you must input your own values. Click the buttons to call the particular empirical formulae so they can be applied in the calculation. Click the button to add a material, and click the button to delete the selected element. Click the button to delete all the elements. The calculated clamp length is displayed in the l_k field.

42.3.2 Distances for eccentric clamping/load

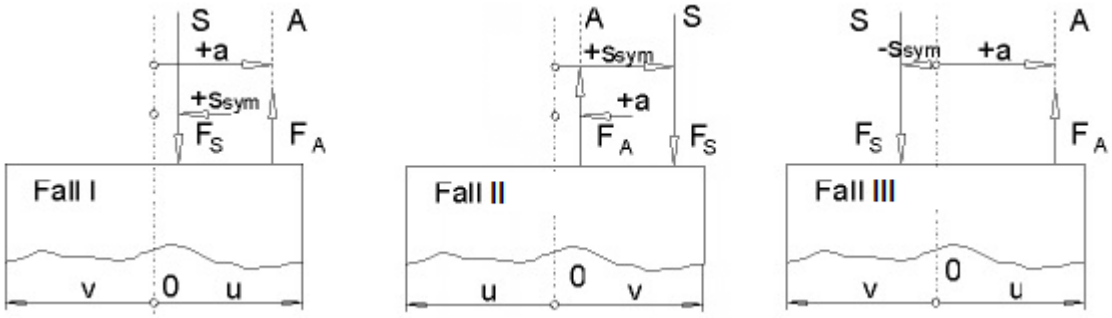


Figure 42.15: Possible load cases in the case of eccentric clamping.

As you can see in Figure 42.15, the axis of the center of gravity of the clamping solid 0 - 0 determines the null point (origin) of the X-axis. The distance between load line of action A - A and the center of gravity axis 0 - 0 is always positive. The distance s between bolt axis S - S and center of gravity axis 0 - 0 is set as positive, if the bolt axis S - S and the load line of action A - A lie on the same side as the center of gravity axis 0 - 0, if not, this value is negative.

The dimension u defines the distance from the center of gravity axis 0 - 0 to the point at which gaping first occurs. In Figure 42.15 this is the distance to the right-hand side in cases I and III, but the distance to the left-hand side in case II. In cases I and II, u must be positive, and in case II it must be negative. The guidelines governing the use of signs specified in VDI 2230 Sheet 1 are applied here.

42.3.3 Load application

The VDI guideline issued in 2014 defines equations for calculating the load application factor. Here, you must select a configuration in accordance with Figure 42.13. The interface must lie within the range shown in gray. The length of the clamped parts h, the distance to the connection piece $a_k$and the length of the connected solid l_A as shown in Figure 42.14 define the position of the application of load point and therefore also the load application factor.

In single-bolted (tapped thread) connections, only configurations SV1, SV2 and SV4 are available. You must use the height h_{ESV} up to the interface as the height h.

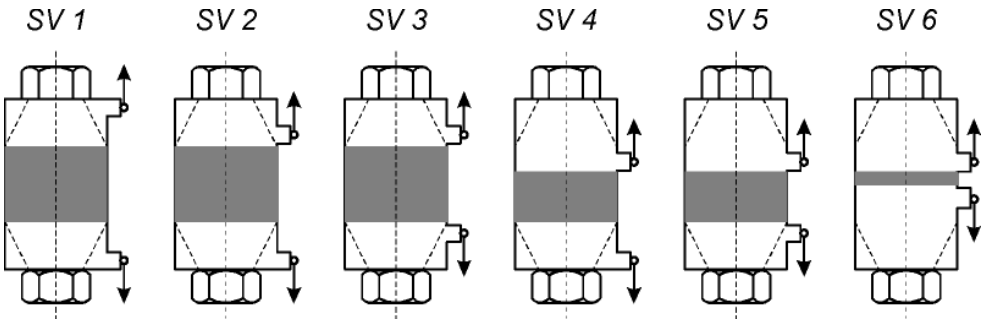


Figure 42.16: Configurations for defining the load application factor as shown in VDI 2230 (2015 edition).

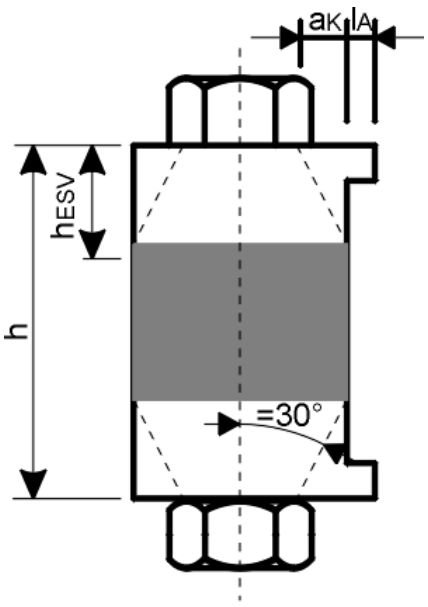


Figure 42.17: Inputs for defining the load application factor as shown in VDI 2230 (2015 edition).

42.4 Constraints data

In this calculation, you can define the yield point utilization, the maximum assembly pretension force or both tightening torques as constraints. If you define the maximum and minimum tightening torque as constraints, the tightening factor is then calculated from this torque variation and the friction coefficient variation.

You can also enter values for the number of load cycles, amount of embedding, preload loss and temperatures for the bolted joint in this window.

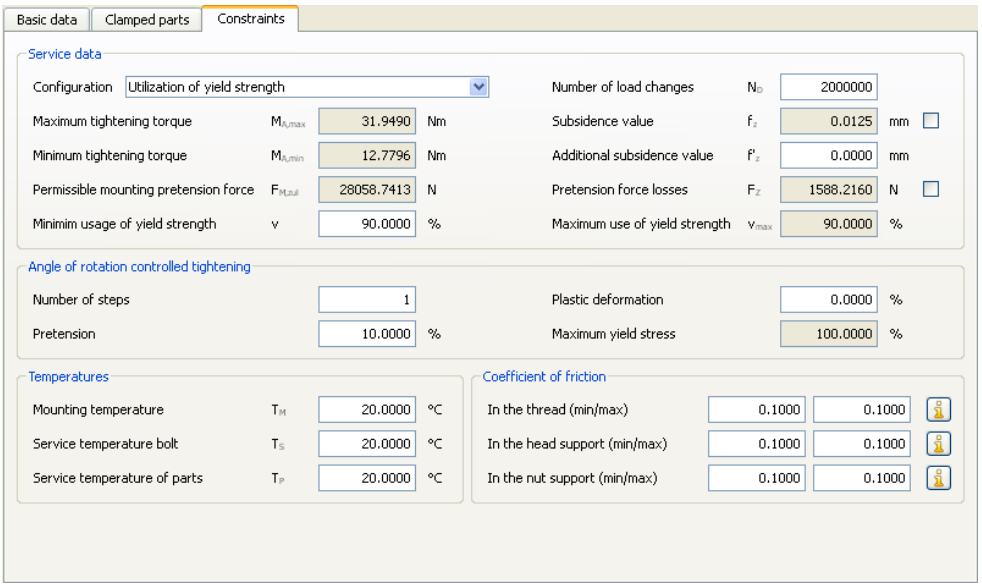


Figure 42.18: Preset values, ready for input.

Use of the yield point
In usual bolt layouts, the bolt is tightened to 90% of its yield point to calculate the pretension force. However, if you use yield point or angle of rotation-controlled tightening, you can increase this utilization value up to 100%.

Number of load cycles
If this number of load cycles is ND >= $2*10^6$ the fatigue lives for final heat treated and final rolled bolts are calculated in accordance with the formulae specified in VDI (VDI 2230, 5.5/20 and 5.5/21). If smaller values are involved (ND < $2*10^{6)}$, a limited life fatigue strength sizing is performed for the connection (5.5/22 and 5.5/23).

Amount of embedding

The amount of embedding is calculated according to which calculation method you use. You can also input an extra amount of embedding value for flat seals. In addition, you can overwrite the calculated amount of embedding with your own value or input the preload loss directly. If you input your own preload loss, the amount of embedding is no longer taken into account.

Assembly and operating temperature
Now KISSsoft's bolt calculation function has been extended, it can be used for the calculation guideline specified in VDI 2230, which also calculates bolted joints for

operating temperatures between -200 and +1000 degrees Celsius. You can specify different temperatures for the bolt and the clamped parts (connecting solids). You can also take into account the temperature-dependent changes of the Young's modulus, the thermal expansion coefficients, the yield point and the pressures permitted for the materials. You can either use empirical formulae to calculate these temperature-dependent values, or specify your own values. Since the empirical formulae for steel have already been determined, you should check the values for high-temperature changes or, even better, enter your own values here.

All the criteria for the bolted joint are checked in the assembled state at ambient temperature, and also in a stationary or non-stationary state at operating temperature (according to VDI 2230: pretension force, bolt load, fatigue life, and surface pressure).

KISSsoft automatically performs the calculation for assembly and operating temperatures at the same time. This calculation should also be performed for a higher temperature difference between the bolt and the parts. The minimum temperature difference between the parts or the bolt and the assembly temperature must at least equal 30° C, so that results appear in the report.

42.4.1 Technical explanations

The critical influences of temperature on the operating properties of bolts are:

- Change in pretension force due to thermal expansion
- Change in pretension force due to relaxation (at high temperatures)
- Brittleness (at high and low temperatures)

The lack of sufficient general data for materials (bolt materials and clamped parts (connecting solids)) means the number of calculation options is also limited. The change in pretension force due to thermal expansion can be calculated very accurately because, as a first approximation, the thermal expansion value can be viewed as linear (with the temperature) (in a temperature range: from -100 to +500°C). The other effects (relaxation and brittleness) can be minimized by selecting appropriate materials and taking precautionary measures (see the relevant literature).

The calculation of the change in pretension force due to thermal expansion is performed as specified by H. Wiegand in Schraubenverbindungen, 4th Edition 1988, section 7.1.3.1 (with temperature-dependent thermal expansion value and Young's modulus). All other calculations are based on the equations in VDI 2230 with the appropriate values at operating temperature.

KISSsoft suggests sensible values for much of the data you can input (Young's modulus, thermal expansion value, yield point at operating temperature) which are

based on current technical literature (DIN standards, technical documentation from the company Bosshard, in Zug, Switzerland). These suggestions are based on the Young's modulus for ambient temperature and, of course, also on the operating temperature. When calculating the suggestion for the permissible pressure at operating temperature, the proportional change to the yield point was assumed. The suggestions are average values for "commonly used steels". They do not refer to one specific material and must therefore be checked carefully in critical situations because the influence of temperature also varies according to the type of material involved. If you want to calculate material data automatically using empirical formulae, simply click on the `Calculation>Settings` tab.

42.4.2 Coefficient of friction

In KISSsoft you can specify an interval for coefficients of friction. The minimum value is used for calculation with F_M, F_{Mmax} and the maximum value is used for calculation with F_{Mmin} and F_M/α_A. The maximum value therefore affects the scatter of the tightening torques.

Friction coefficient μ_G according to Wiegand, Schraubenverbindungen

thread					outside thread										
thread	material				steel									A2	
		surface			black finish or phosphate treated				electro galvanised (Zn6)		cadmium galvanised (Cd6)		adhe-sive		
			production		rolled			cut	cut or rolled						
	material	surface	produc-tion	lubri-cation	dry	oiled	MoS_2	oiled	dry	oiled	dry	oiled	dry	dry	oiled
inside thread	steel	bare	cut	dry	**0.12** - 0.18	**0.10** - 0.16	**0.08** - 0.12	**0.10** - 0.16	/	**0.10** - 0.18	/	**0.08** - 0.14	**0.16** - 0.25	/	/
		electro galvanised			**0.10** - 0.16	/	/	/	**0.12** - 0.20	**0.10** - 0.18	/	/	**0.14** - 0.25	/	/
		cadmium galvanised			**0.08** - 0.14	/	/	/	/	/	**0.12** - 0.16	**0.12** - 0.14	/	/	/
	GG/GT	bare			/	**0.10** - 0.18	/	**0.10** - 0.18	/	**0.10** - 0.16	/	**0.08** - 0.16	/	/	/
	AlMg	bare			/	**0.08** - 0.20	/	/	/	/	/	/	/	**0.32** - 0.43	**0.28** - 0.35
	A2				/	/	/	/	/	/	/	/	/	**0.26** - 0.50	**0.12** - 0.23

Figure 42.19: Friction coefficients in the thread.

Friction coefficient μ_K and μ_M according to Wiegand, Schraubenverbindungen

support area				screw head/nut											
	material			Steel										A2	
		surface		black finish or phosphate treated						electro galvanised (Zn6)		camium galvanised (Cd6)			
			production	pressed			twisted		grinded	pressed					
support area	material	surface	production	lubrication	dry	oiled	MoS_2	oiled	MoS_2	oiled	dry	oiled	dry	oiled	dry	oiled
counter surface	steel	bare	grinded	dry	/	0.16 - 0.22	/	0.10 - 0.18	/	0.16 – 0.22	0.10 - 0.18	/	0.08 - 0.16	/	/	/
			machined		0.12 - 0.18	0.10 - 0.18	0.08 - 0.12	0.10 - 0.18	0.08 - 0.12	/	0.10 - 0.18	0.10 - 0.18	0.08 - 0.16	0.08 - 0.14	/	/
		electro galvanised			0.10 - 0.16		/	0.10 - 0.16	/	0.10 - 0.18	0.16 - 0.20	0.10 - 0.18	/	/	/	/
		cadmium galvanised			0.08 - 0.16						/	/	0.12 - 0.20	0.12 - 0.14	/	/
	GG/GTS	bare	grinded		/	0.10 - 0.18	/	/	/	0.10 - 0.18			0.08 - 0.16	/	/	/
			machined		/	0.14 - 0.20	/	0.10 - 0.18	/	0.14 - 0.22	0.10 - 0.18	0.10 - 0.16	0.08 - 0.16	/	/	/
	AlMg				/	0.08 - 0.20				/	/	/	/	/	0.35 - 0.50	0.08 - 0.11
	A2				/	/	/	/	/	/	/	/	/	/	0.08 - 0.11	0.08 - 0.12

Figure 42.20: Friction coefficients in head bearing area and nut bearing area.

You can also use the sizing according to friction classes A to E as specified in VDI 2230 Sheet 1, Annex to Table A5 to define the values for the coefficients of friction. The minimum and maximum coefficients of friction for the thread, the head bearing area and the nut support are then imported into KISSsoft.

42.4.3 Angle of rotation-controlled tightening

For angle of rotation-controlled tightening, the report displays a preload torque and an angle of rotation split into a number of steps. Here you can enter the value for this preload torque and the number of steps. The angle of rotation is then calculated with the medium assembly pretension force ($F_M + F_M/\alpha_A)/2$. If the yield point

utilization is 100%, this force is applied up to the yield point. To calculate the tightening angle of rotation you can also enter the required plastic strain of the weakest cross section.

42.5 Stripping strength

Click on the `Length of engagement` flag to calculate the length of engagement and check the stripping strength of the thread according to VDI 2230 section 5.5.5.

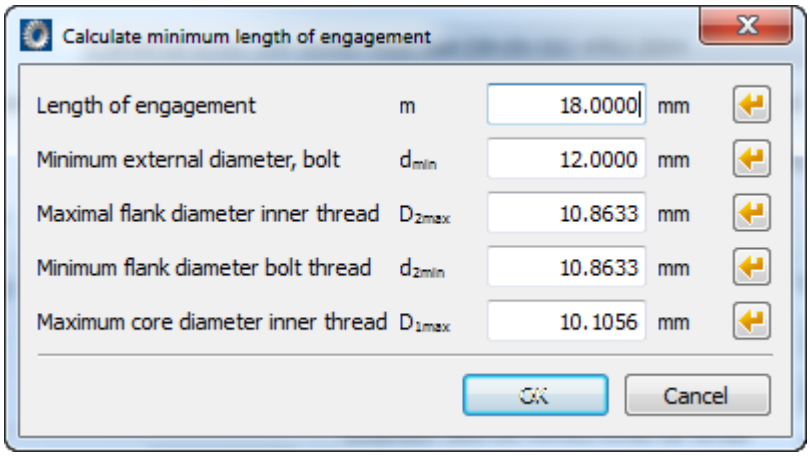


Figure 42.21: Entries for calculating the length of engagement and stripping strength

In this screen you can enter the data for the length of engagement. Use the sizing buttons to set the individual defaults which were calculated from the entries in the main screen.

The length of engagement meffmin is calculated from the (theoretical) tensile strength Rm of the bolt material, the length of engagement meffmax is calculated for the bolt and internal thread (with Rmmax, dmin or d2min and D2max or D1max according to VDI 2230 Sheet 1, Equation 210/213). The more critical case is then displayed in the results.

The default value for the Rmmax/Rm coefficient is 1.2. This is also stated as a practice-related value in VDI 2230.
You can change the Rmmax/Rm coefficient in `Calculation > Settings`.

To calculate the worst case (VDI 2230, formula 210), you must also take the thread tolerance into account. To define this influence, the minimum external diameter of the bolt dmin, the maximum flank diameter of the internal thread D2max, the minimum flank diameter of the bolt thread d2min and the maximum core diameter of the internal thread D1max can be entered in this window.

The main report lists the stresses, the minimum length of engagement, and the safety against shearing under load, with the maximum pretension force for the connection.

42.6 Settings

In `Calculations > Settings` you can enable additional settings:

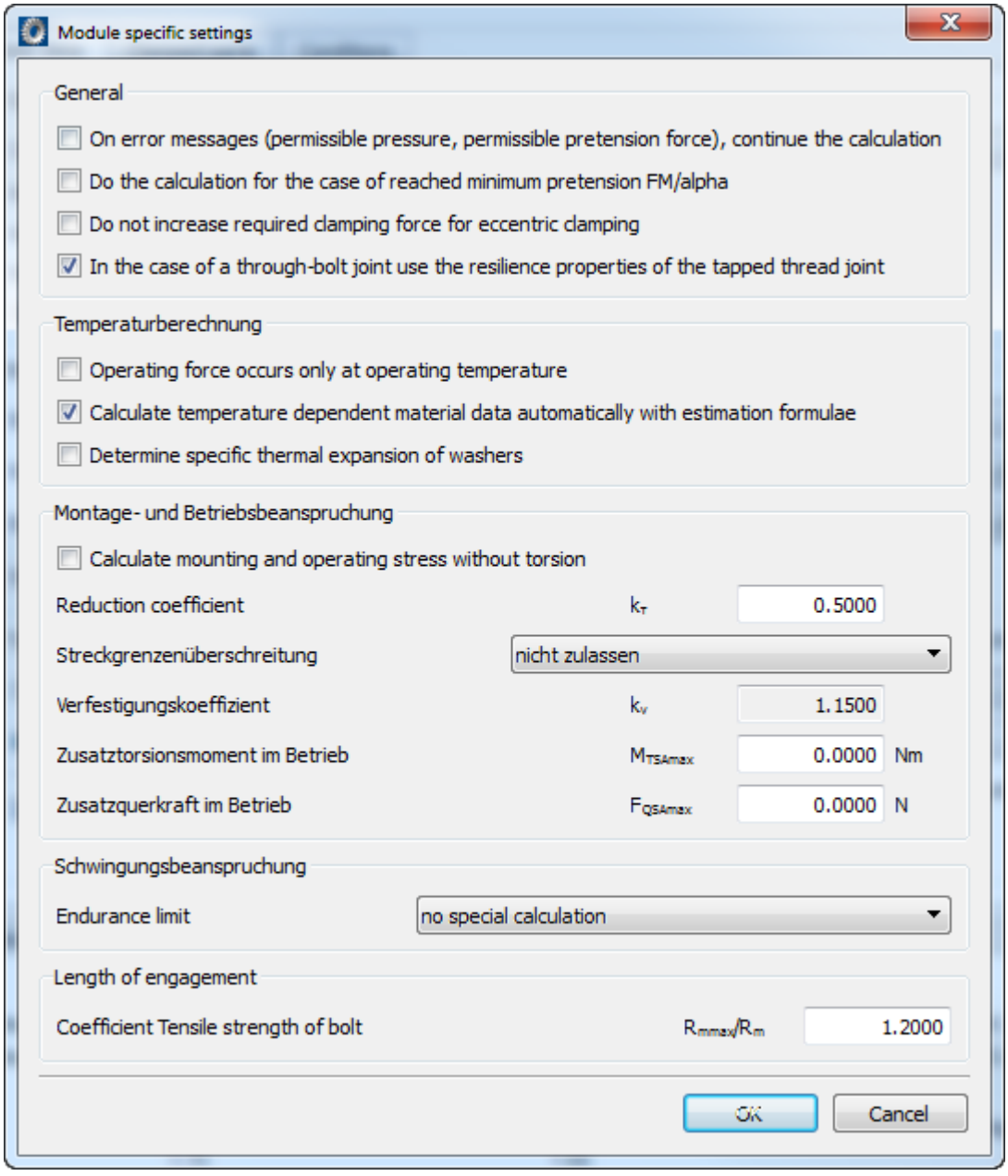


Figure 42.22: Settings for bolts.

- **Continue calculation if error messages appear**
  If you set this flag, the calculation will continue even if error messages are displayed, for example to warn you that the yield point or the permitted pressure has been exceeded.

- **Calculate minimum preload force F_M/α_A achieved**
  If this flag is set, the load F_M/α_A is also calculated. The pretension force F_M/α_A is the minimum pretension force that must be present, if the entered pretension force F_M is included. α_A is the tightening factor. It describes the scatter of the pretension force. If this option is set, the results overview in the main screen shows the results of the calculation with F_M, otherwise the results with F_{Mmax} are displayed.

- **Do not increase required clamp load for eccentric clamping**
  KISSsoft increases the required clamp load to prevent gaping for eccentric clamping. You can switch off this function here. You can then specify your own required clamp load. Take care when using this option. The calculation assumes that gaping does not occur!

- **Use the resilience of the single-bolted connection for a through-bolt connection**
  If the diameter of the clamped part and the bolt head support is relatively small, the connection can be calculated in the same way as a through-bolt connection. In this case, the resiliences can still be defined as a "single-bolted connection" (i.e. a single-bolted (tapped thread) joint).

- **Operating force only at operating temperature**
  Normally, KISSsoft calculates the minimum pretension force based on the required clamp load and loading at ambient and operating temperatures. This flag can be set when the operating force only occurs at operating temperature. In this case, the minimum pretension force is then only calculated at operating temperature. The mounting pretension force F_M in the working state is reduced by the proportion $(1-\phi)*F_A$ of the axial bolt load.

- **Calculate temperature dependent material data automatically with estimation formulae**
  KISSsoft can automatically calculate material data at operating temperature by using empirical formulae. These empirical formulae do not take into account the material data you entered: they use an average dependency for "commonly used steels"! Delete this flag if you want to enter your own materials data at operating temperature.

- **Determine specific thermal expansion of washers**
  This opens the input field for thermal expansion values in the sub-window for (flat) washers. If this flag is not set, the difference in pretension force is calculated using the average thermal expansion of the plates. In other words, the flat

washer has the same thermal expansion as the plates. This is why you have the option of inputting this value. If you do so, the difference in pretension force is calculated using the value you specified, but the resilience of the plates is still used in this calculation. VDI 2230 does not specify that a special thermal expansion calculation is to be used for washers.

- **Calculate mounting and operating stress without torsion**
  Select this option if the connection is fully preloaded to the extent that torsional stress no longer occurs. If you do this, the torsion is set to 0 when the system calculates the equivalent stresses that occur in the assembled and working states.

- **Reduction coefficient**
  The reduction coefficient is used to calculate equivalent stress when the machinery is in its working state. In many cases, the torsional stress in elastically preloaded connections reduces by 50%. This is why VDI 2230 recommends the value 0.5 is used here.

- **Exceeding the yield point**
  Three selection options are available here: yield point cannot be exceeded, yield point can only be exceeded during operations, or yield point can be exceeded during operation and mounting. This enables you to select your preferred calculation variant.

- **Hardening coefficient**
  An additional hardening factor, kv, is used when calculating whether the yield point has been exceeded during mounting and during operation. The default value for the hardening factor is 1.15. The VDI standard specifies that it should lie between 1.1 and 1.2.

- **Additional torsional moment during operation**
  An additional torsional moment can be defined when calculating working stress. This torsional moment is then used in the shearing load calculation. This applies both if the yield point cannot be exceeded and in cases in which the yield point can be exceeded.

- **Additional torsional moment during operation**
  An additional torsional moment can be defined when calculating working stress. This torsional moment is then used in the shearing force calculation. This applies in cases where the yield point is exceeded.

- **Endurance limit**
  Selection list for specifying the kind of bolt for which the endurance limit calculation is to be performed. In the case of high-strength friction-grip fasteners, the sustainable fatigue life is reduced by 10% because of special geometrical features. In the case of hot-galvanized high-strength friction-grip fasteners, the

sustainable fatigue life is reduced by 30%. (Comment in VDI 2230, chapter on alternating stress)

- **Tensile strength of bolt coefficient**
  This coefficient is used to calculate the minimum length of engagement required to achieve a practical value for Rm (as in VDI 2230). You will find a more detailed description in the section on Stripping strength (see page IV-1012).

Chapter 43
Welded Joints

Underlying principles of calculation: DIN 18800, Part 1, Edition November 1990, in particular section 8.4 "Conections with arc welding"

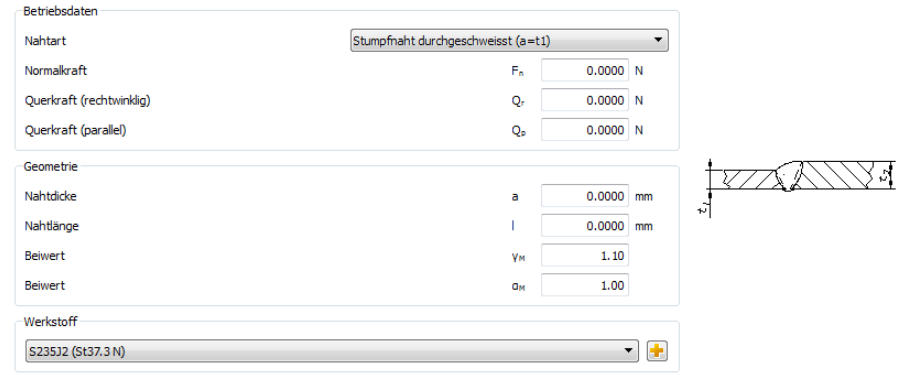
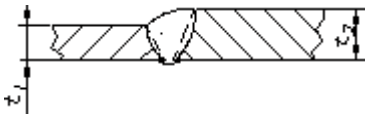


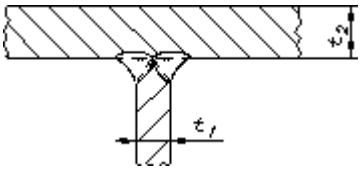
Figure 43.1: Basic data: Welded joints

43.1 Welded joints

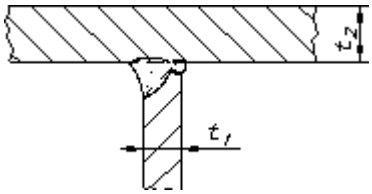
You can apply the calculation method defined in DIN 18800 to these welded seam types:



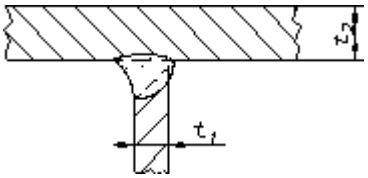
Butt seam through welded



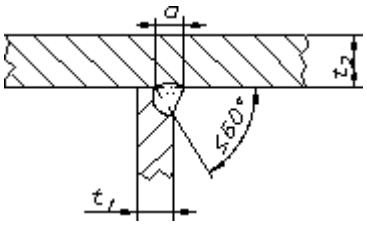
Double-HV-seam, counter welded



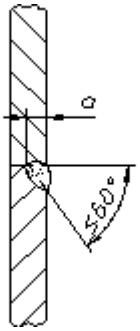
HV welded seam, cap position counter welded



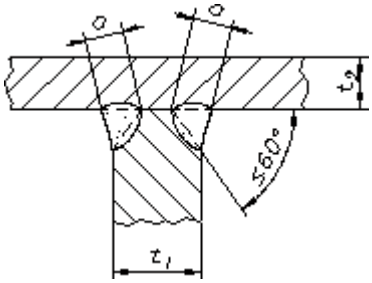
HV welded seam, root through welded



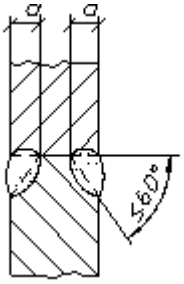
HY seam with fillet weld, not through welded



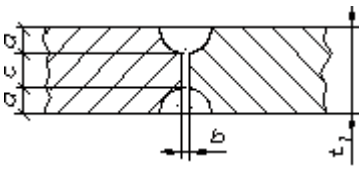
HY seam, not through welded



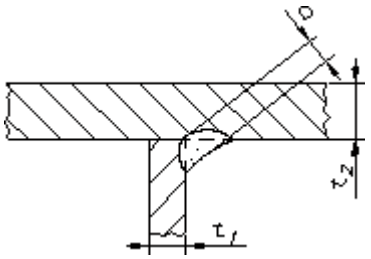
Double HY seam with fillet weld, not through welded



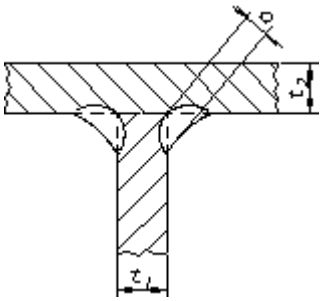
Double HY seam, not through welded



Double-I-seam, not through welded



Fillet weld, not through welded



Double-fillet weld, not through welded

43.2 Seam length

Table 20 in DIN 18800 shows various configurations that use welded seam length l.

43.3 Welded seam equivalent stress

Use the following formula to calculate the equivalent stress for butt and filled welded seams:

$$\sigma_{W,V} = \sqrt{\sigma_r^2 + \tau_r^2 + \tau_p^2} \qquad (43.1)$$

$\sigma_{W,V}$: Equivalent stress [N/mm^2]
σ_r: Normal stress (vertical to the welded seam) [N/mm^2]
τ_r: Shear stress (vertical to the welded seam) [N/mm^2]
τ_p: Shear stress (parallel to the welded seam) [N/mm^2]

43.4 Weld seam boundary stress

The weld seam boundary stress $\sigma_{W,R,d}$ is calculated with:

$$\sigma_{W,R,d} = \alpha_W \cdot \frac{R_p}{\gamma_M} \tag{43.2}$$

$\sigma_{W,R,d}$: Weld seam boundary stress [N/mm^2]
α_W : Weld seam boundary coefficient [-]
R_p: Yield point [N/mm^2]
γ_M: Part safety coefficient [-]

43.5 Part safety coefficient

The part safety coefficient γ_m is usually 1.1 as specified in section 7.3 in DIN 18800. However, you can also use the value 1.0 to prove the suitability for use or reduced stiffness.

43.6 Boundary safety coefficient

The weld seam boundary coefficient α_W is defined as specified in Table 21 of the standard:

Weld seam type	**Seam quality**	**Stress type**	**St37-2 and similar**	**St52-3 and similar**
1 - 4	All seam qualities	Compression	1.0	1.0
	Proven seam quality	Tension	1.0	1.0
	Unproven seam quality	Tension	0.95	0.85
5 - 15	All seam qualities	Compression, tension	0.95	0.85
1 - 15	All seam qualities	Shear	0.95	0.85

43.7 Materials

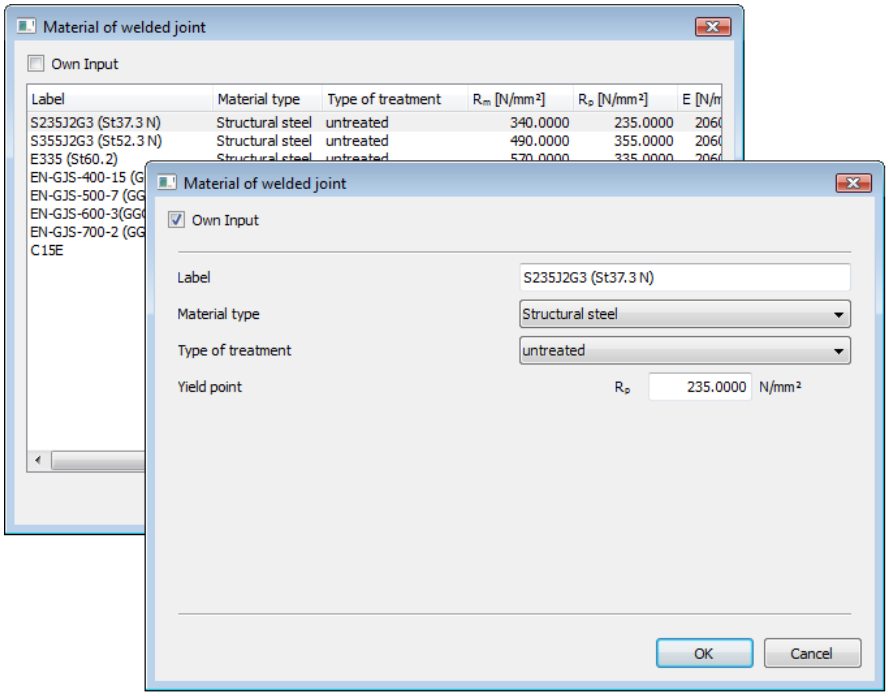


Figure 43.2: Materials screen: Welded joints

The selection list contains materials from the DIN 18800 standard.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

Chapter 44
Glued and Soldered Joints

Calculation basis: [64]. The calculation is performed for glued and soldered connections that are subject to shear.

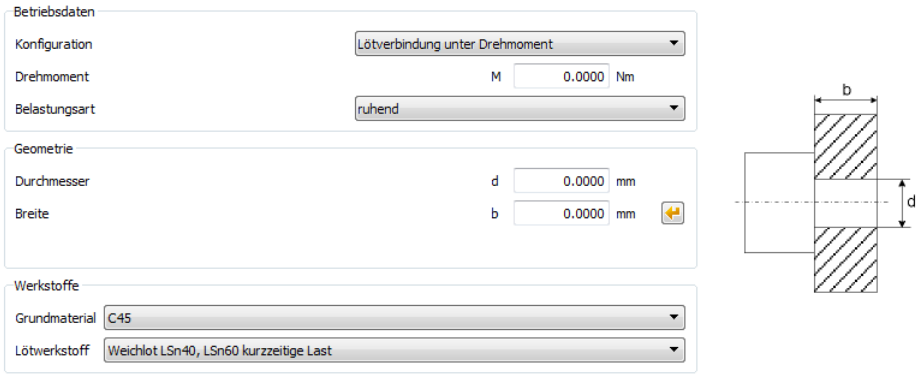


Figure 44.1: Basic data: Glued and soldered connections

Two different load cases are treated:

- Shearing force: Transmission of shearing force between two surfaces.
- Torque: shaft-hub connection with a torque load.

The joint can be subject either to static or dynamic (usually pulsating) load.

The guideline values for the static strength of soldered joints are taken from [64], Table 8/8 (average values of resistance to fracture due to shearing). The guideline values for glued joints are taken from Table 8/9. For the pulsating load on soldered joints, 50% of the static strength is assumed as the permitted limit (data not available: you must check these connections to ascertain the endurance limit of the base material. This may reduce the element safety of the soldered joint by approximately 80%). For glued joints, 30% of the static strength is permitted (as defined in Table 8/9).

At present, the following materials can be used for glued joints:

- cured at ambient temperature.

- cured at increased temperature.

To calculate the shearing strength value, the program uses the mean value of the minimum and maximum value from the database. The value achieved by optimum implementation as defined in Niemann is not used.

At present, the following materials can be used for soldered joints:

- soft solder LSn40, LSn60 for short-term loads
- soft solder LSn40 at continuous load
- Brass solder: Steel NE heavy metals
- New silver solder-copper: Steel
- Silver solder: Steel NE heavy metals

There is no point calculating and sizing soldered joints with light Al-based metals, because the strength of the underlying material is usually less than that of the joint. To calculate the shearing strength value, the program uses the mean value of the minimum and maximum value from the database.

44.1 Basic materials

These materials are only used to size the width, on the basis of the strength of the base material.

At present, you can select these materials:
Ck 45 N, Ck 60, CrNiMo, CrNi 4, CrNiMo, CrMo, St 37.3, St 52.3, St 60.2, Ganevasit, PA 12, PA 66, POM, laminated wood.

You must then still decide which material will be the best for your joint. For example, you should not select PA 12 if you are using a soldered joint.

44.2 Settings

In this window, you can view the required safety value and the shearing strength to be used in the sizing. You can adjust this value as required.

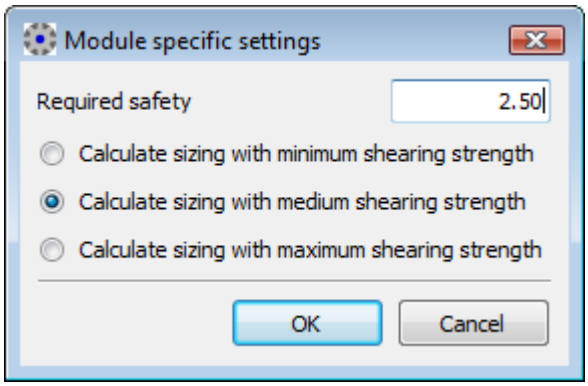


Figure 44.2: Settings: Glued and soldered joints

44.3 Sizings

- Sizing the width on the basis of the base material
  Sizing the adhesion width (for shaft hub), or the adhesion length (for brackets), on the basis on the strength of the base material. The tear resistance of the joint is sized so that it equals the tear resistance of the underlying material or the fatigue strength under pulsating stress of the shaft.
- Sizing the width on the basis of load
  Sizing the adhesion width on the basis of load. the tear resistance of the joint is sized so that it can withstand the forces it is subjected to without compromising the required safety.

44.4 Bracket connection

Calculating a glue or soldered joint with sheets or plates. You must specify the tension or compression force, the adhesion length and the thickness of the metal sheet or plate.

44.5 Shaft connections

Calculating a glued or soldered connection for shaft-hub connections. You must specify the transferring moment in Nm, the joint diameter and the length of the adhesion point.

Chapter 45
Retaining Rings (Self-Locking Rings, Seeger Rings)

This module is used to perform calculations for shaft or hub snap rings. The calculations are carried out in accordance with the manual published by the company Seeger (http://www.seeger-orbis.com/). To open the module, navigate to the module tree and double-click the appropriate module (see Figure 45.1).

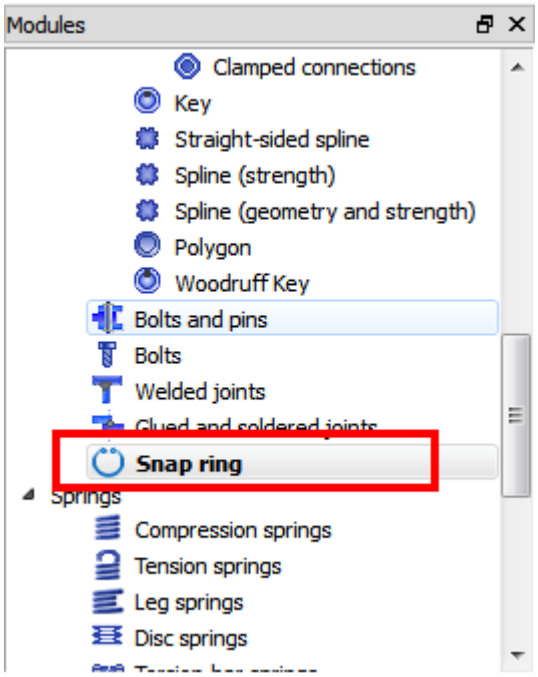


Figure 45.1 The Seeger ring calculation module

45.1 Basic data

Input the following data in the Basic data tab:

- "Geometry" group
  - Shaft/bore ring: specifies whether the calculation is to be performed for a shaft or for a bore ring
  - Retaining ring/circlip: specifies whether the calculation is to be performed for a circlip or a retaining ring
  - d1: nominal length, the shaft diameter for a shaft ring, or the bore diameter for a bore ring
  - d2: Groove diameter
  - d3: inner diameter of the Seeger ring for shafts or external diameter of Seeger ring for bores in the unstressed state
  - b: the maximum radial width of the Seeger ring
  - Dimension l: see Figure 45.2
  - s: the thickness of the ring
  - ψ: permissible dishing angle of the Seeger ring (see Figure 45.3)
  - sharp-cornered bearing area: Controls whether conditions with a chamfer, the corner distance or the radius of g should be considered (see Figure 45.3)
  - g: the chamber or corner distance/radius

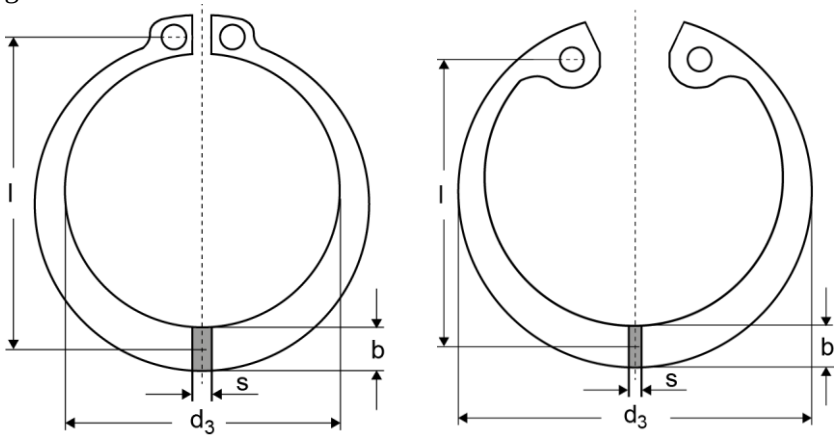


Figure 45.2 Geometry of shaft ring (a) and bore ring (b)

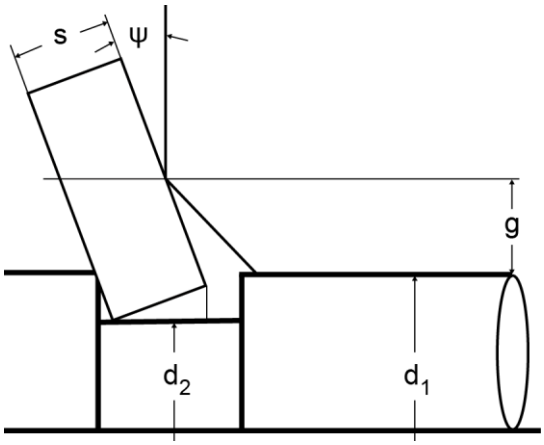


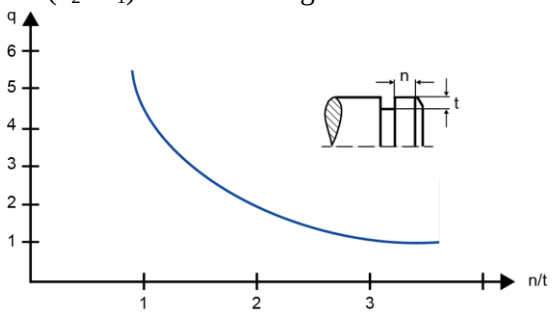
Figure 45.3 Definition of geometric values s, ψ, g

- "Operating data" group
  - q: the load factor, taking into consideration the effect of the shoulder length ratio (see Figure 45.4)
  - μ: the coefficient of friction between the ring surface and the shaft/bore surface
- "Materials" group
  - In this group you can define the material of the ring and shaft/bore. The functionality is similar to the rest of the KISSsoft modules which are located in the "Connections" module group.

45.2 Automatic calculation of load factor q

If you click on the button next to q, the system displays a window, in which you can calculate q, based on the ratio of the shoulder length n to the groove depth t (see Figure 45.4). The groove depth is defined as:

- $t = (d_1 - d_2)/2$ for shaft rings
- $t = (d_2 - d_1)/2$ for bore rings



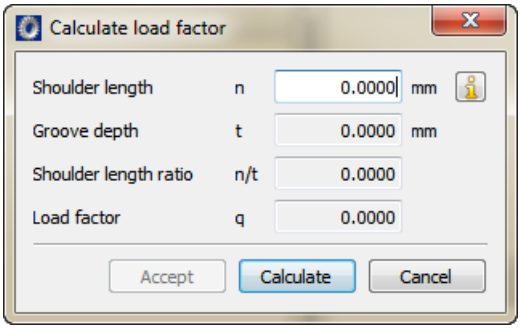


Figure 45.4 (a) Definition of load factor q, shoulder length n and groove depth t. (b) Sizing window for load factor q.

45.3 Automatic calculation of the dishing angle ψ

Use Figure 45.5 to calculate ψautomatically.

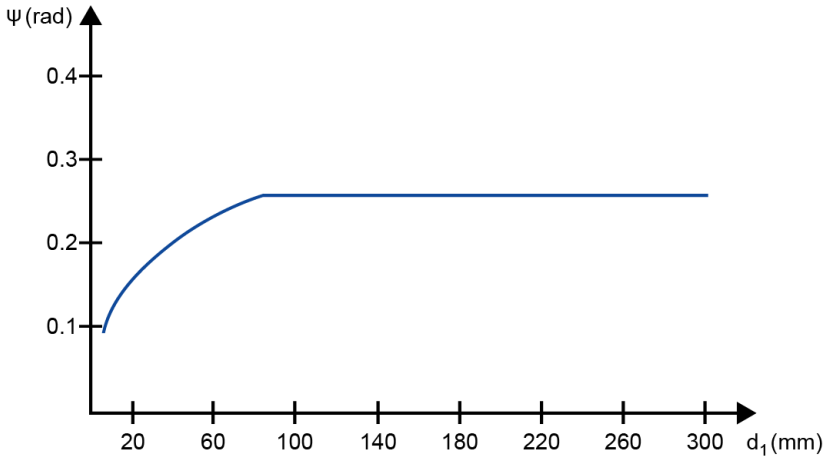
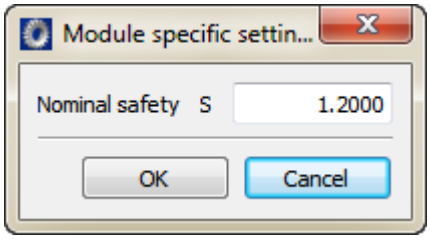


Figure 45.5 Calculation of ψ, based on d1.

45.4 Module specific settings



Here you can define the minimum required safety S.

Chapter 46
Answers to Frequently Asked Questions

46.1 Adding new bolt types to the database

The KISSsoft database includes the following bolt types:

- hexagon socket head cap bolts EN ISO 4762
- hexagon headed bolts with shank (AB) EN ISO 4014
- hexagon headed bolts without shank (AB) EN ISO 4017
- slotted cheese head bolts EN ISO 1207
- hexagon headed bolts with shank, metric fine thread (AB) EN ISO 8765
- hexagon headed bolts without shank, metric fine thread (AB) EN ISO 8676
- hexagon headed bolts with flange, light series, shape F EN 1662
- hexagon headed bolts with flange, light series, shape U EN 1662
- hexagon headed bolts with flange, heavy series, shape F EN 1665
- hexagon headed bolts with flange, heavy series, shape U EN 1665
- Own definition of bolt geometry

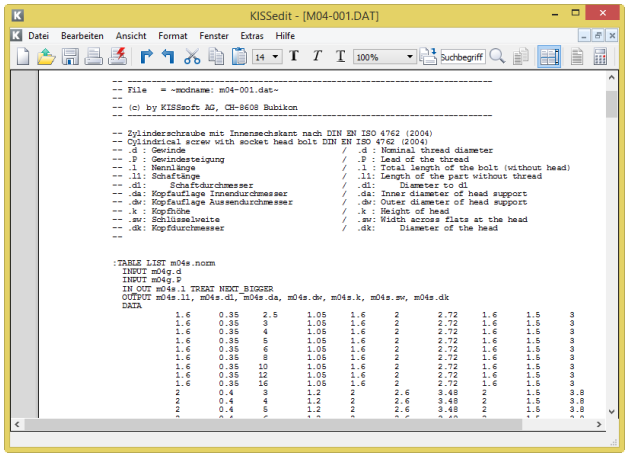
For each of these bolts types, a number of tables list the various bolts sizes (= bolts series). You will find the name of the file that contains the bolts series data in the database (see page I-128).

You enter a new size within an existing bolt type i.e. extend the bolt series (see page IV-1041) or input a new bolt type (see page IV-1043).

46.1.1 Extending an existing bolt series

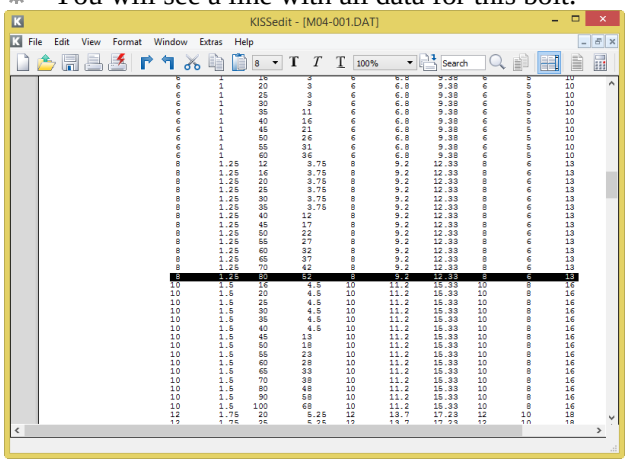
Example: Enter the data for an M8 bolt with a length of 100 mm in the "hexagon socket head cap bolts EN ISO 4762" series.

Then start the database tool. Open the `Screw Type M000.KDB, M040Typ` table. Select the `Hexagon cap screw EN ISO 4762` data record in that table. In the `File name` field, you will see the name of the file which contains the table with the bolt series data. Click the `Edit` button at the end of the input line to open the file in the Editor:



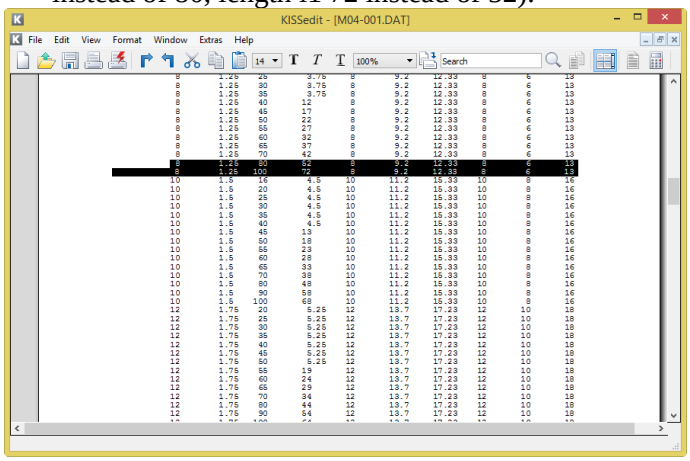
To enter a new bolt:

- Look for a similar bolt (M8, length 80mm).
- You will see a line with all data for this bolt.



- Copy this line. When you do so, note the exact sequence of the lines.

- Change the data in accordance with Table 1 in EN ISO 4762 (length 100 instead of 80, length l1 72 instead of 52).



- Save the file.
- Document any changes for other users.

46.1.2 Create a new bolt type

You must already be familiar with the table structure before you can add a new bolt type. You must know which value goes in which column (use the variable names from the descriptions in the table header).

Then, proceed as follows:

- In the database, open the data record most similar to the new bolt type.
- Copy this data record and rename it to the new bolt type.
- Click the `Edit` button at the end of the input line for the file names. This opens a file which still contains the "old" values.
- Overwrite these values with the new values. Note the variables structure (i.e. a specific variable is assigned to a number, depending on where the number appears) and the sorting of the lines.
- Save the updated file with a new name and close the Editor.
- Transfer the new file name to the database (to create the cross reference).
- Then save the new data record.

Part V
Springs

Chapter 47
Compression Springs

The calculation of compression springs is based on DIN EN 13906-1 (2013)[30].

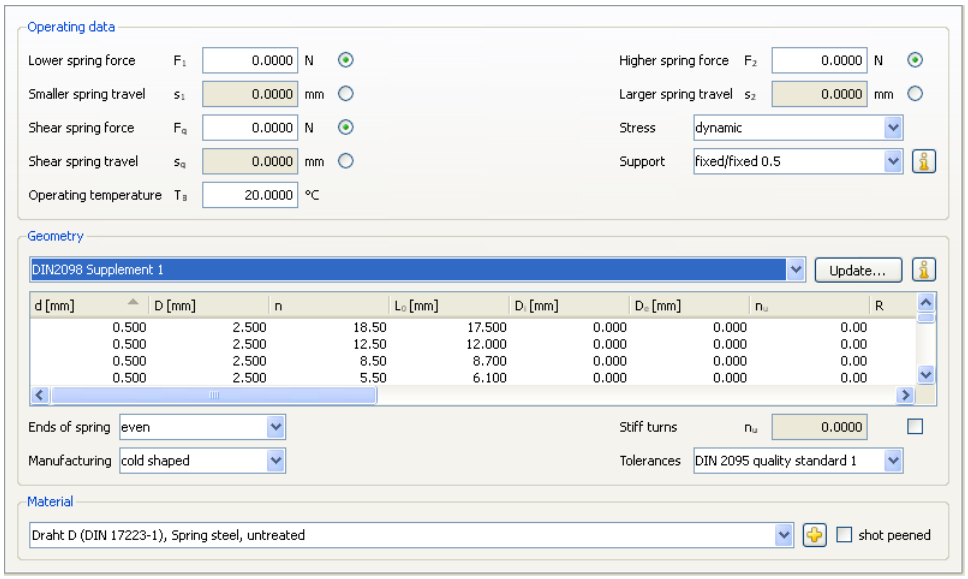


Figure 47.1 Basic data for compression springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, force.

Geometry

You can select the geometry data according to DIN 2098 Part 1 directly from this table.

If you select `Own input`, you can either take selected values from the list or enter your own values. Select `Own input` to specify your own spring length and the diameter.

Instead of using the spring length in its non-stressed state L_0 you can also use a spring length in its stressed state L_1 or select L_2.

The choice of the `End of spring` and `Manufacture` affects the calculation of the block length L_c.

Click the `Update` button to calculate the block lengths and the resulting values of the current situation for individual springs and display them in a table.

47.1 Strength values

The material strengths are stored in different files, depending on diameter. The transverse strength is either saved in the tables, as in DIN EN 13906-1 for thermo-formed springs, or calculated from the predefined tensile strength as $\tau_{czul} = 0.56 \cdot R_m$.

To calculate the endurance limit, use either the Goodman diagram as defined in EN 13906-1 or an approximation. The approximation assumes a dynamic strength of $0.25 \cdot R_m$ and a lead of 0.75 for the graph of the upper stress in the Goodman diagram. For shot peened materials, the dynamic strength is increased by 20%. These values roughly correspond to the diagrams in the DIN EN 13906-1 standard, but you should regard the safeties more conservatively.

47.2 Shear stress values

The calculation of the highest shear stress also calculates the axial and shear spring travel.

$$\tau_{max} = \frac{8}{\pi \cdot d^3} \cdot \left[F\left(D + s_Q\right) + F_Q \cdot \left(L - d\right) \right] \tag{47.1}$$

τ_{max}: highest shear stress [N/mm^2]
d: wire diameter [mm]
F: spring force [N]
D: average coil diameter [mm]
s_Q: shear spring travel [mm]
F_Q: shear spring force [N]
L: spring length [mm]

The highest corrected shear stress is calculated by:

$$\tau k_{max} = k \cdot \tau_{max} \tag{47.2}$$

τk_{max}: highest corrected shear stress [N/mm^2]
τ_{max}: highest shear stress [N/mm^2]

K: stress correction factor
(dependent on the ratio D/d)

47.3 Bearings coefficient

The `Support` you select defines the value of the support coefficient ν, as shown in Figure 47.2.

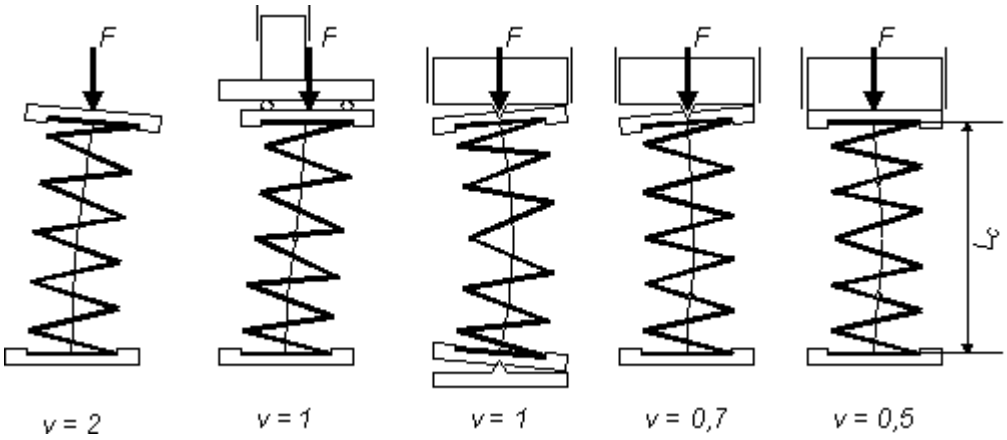


Figure 47.2: Support with associated support coefficients for axially stressed compression springs

The support coefficient ν is used for calculating the buckling spring travel sk. If the buckling safety factor is not reached then the spring must be guided, otherwise it will buckle.

47.4 Materials

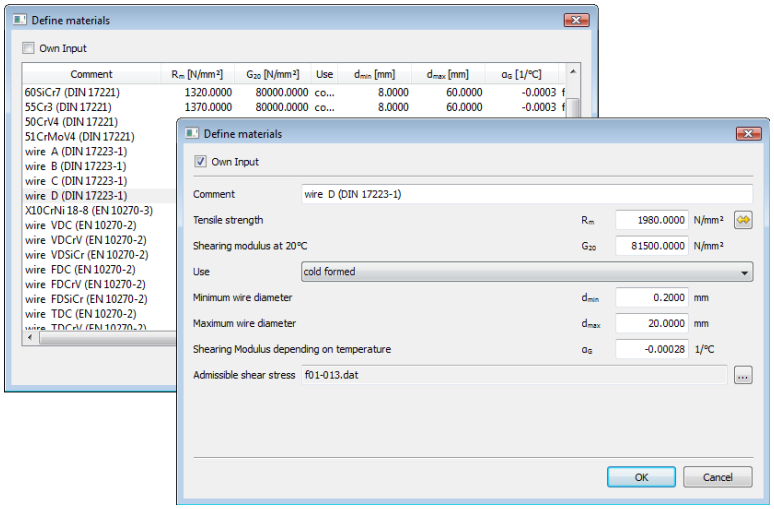


Figure 47.3: Compression springs

The selection list includes materials from the DIN 10089, DIN 17223-1 (withdrawn), DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

47.5 Tolerances

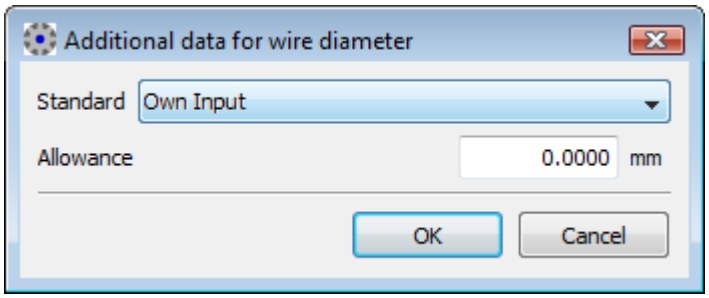


Figure 47.4: Additional data for compression springs wire diameter

When you select a spring from the table (according to DIN 2098 Part 1), the tolerance of the diameter used here is specified in DIN 2076 C. To change the diameter tolerance, toggle to the Own input list to open the input fields. To change the diameter tolerance, change to the `Own input` list to display the input fields which you can then use. Click the  button next to the wire diameter field to open another screen. (see Figure)

In the Tolerances screen, you can select wire diameters according to DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), DIN EN 10270-1 (2012), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012) or DIN EN 10218 (2012), or select `Own input` to enter your own value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the `Geometry Tolerances` and `Force Tolerances` list in the basic data you can choose one of the quality standards according to DIN 15800 (2009) [14] or DIN 2096 Part 1 (1981) [15].

47.6 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard, and shown in diagrams. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By noting the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics are also available that show the progress of relaxation over time and the spring force. The results for the specified conditions are then displayed in the relaxation report for 48 h. The value of the spring force is also calculated after 48 h.

To extend the data for the materials relaxation curves, or add new data, add this new information to the *.dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 predefined measurement points. The curves are then calculated from these points.

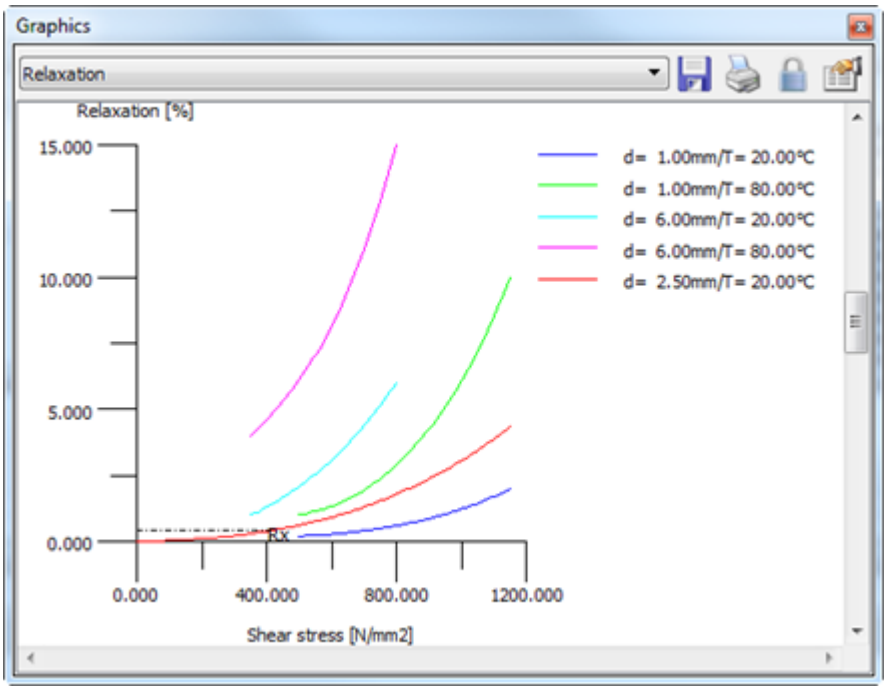


Figure 47.5: Relaxation for compression springs

47.7 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

47.8 Sizings

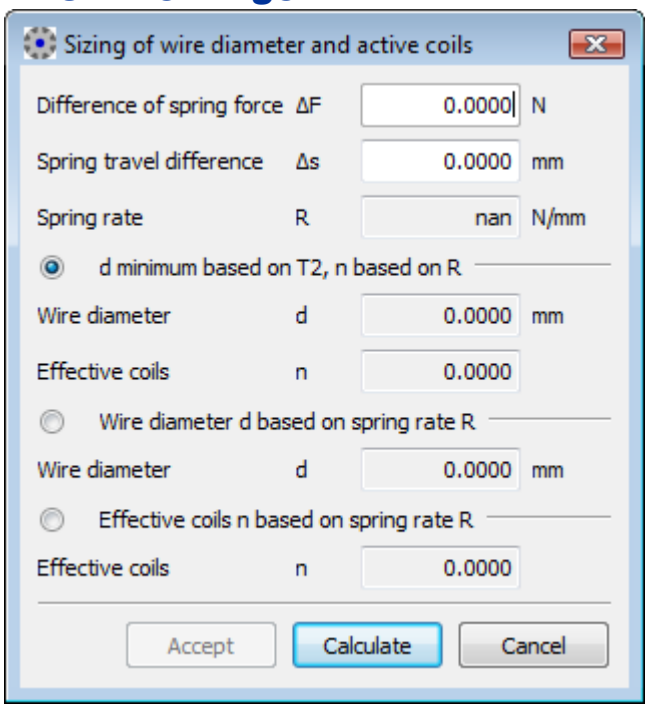


Figure 47.5: Sizing screen: Compression springs

If you selected `Own input` in the list under Geometry, you now see input fields here instead of a table showing the values defined in the standard. Next to the `Wire diameter` and the `Effective coils`, you can click the  button to size the following values.

Using the predefined spring rate R = ΔF/Δs, the number of turns n can also be calculated if the wire diameter has been predefined. The number of turns is defined by this value, but the strength and the geometric constraints are not checked. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 48
Tension Springs

The tension spring calculation is described in the DIN EN 13906-2 (2013)[31] standard.

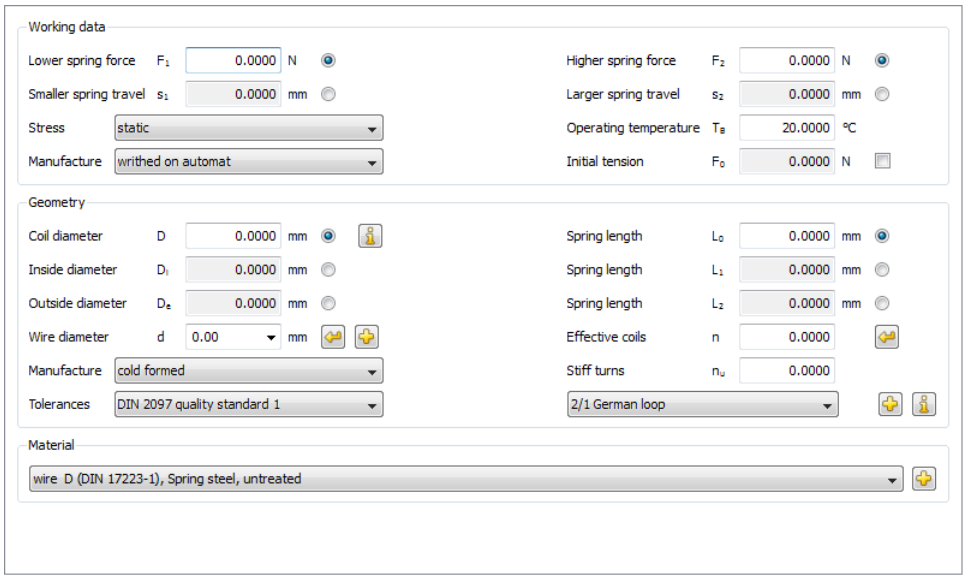


Figure 48.1: Basic data: Tension springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. This force is defined as the `initial preload force` F0, which is required to open the coils which lie one on top of the other. This force is only present if the spring is pretensioned.
If the flag for `Inner preload` is not set, you can influence the number of effective coils.
You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

Geometry

You can specify the spring length and the spring diameter directly in the main screen.
Instead of using the spring length in its non-stressed state L_0 you can also use a spring length in its stressed state L_1 or select L_2.

For the wire diameter, you can either select the diameter values as defined in DIN 2098 supplement 1 from the list or enter your own value directly in the list.

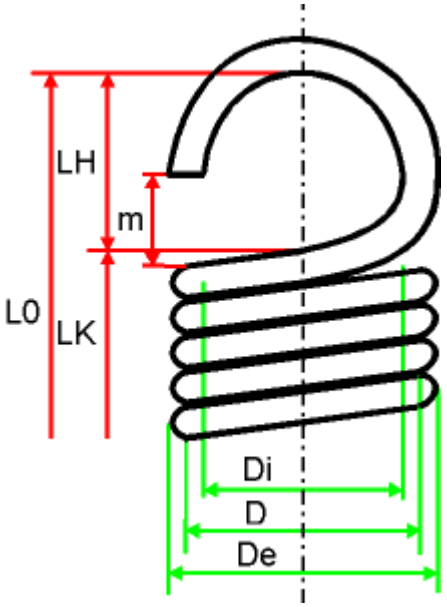


Figure 48.2: Definitions used for tension springs

48.1 Strength values

Permissible shear stress is calculated from the tensile strength of cold formed tension springs. The tensile strength values are determined by diameter values stored in various files. The shear stress is calculated using the formula $\tau_{zul} = 0.45 \cdot R_m$. Thermo-formed tension springs should not exceed the permissible shear stress of $\tau_{zul} = 600 N/mm^2$. These values apply to static or quasistatic cases. Tension springs as defined in DIN 2097 should not be subjected to dynamic stress if at all possible. Shear stress is distributed very unevenly over the cross section of the wire or pin of a tension spring. You can use an intensity factor k to approximate the highest arithmetical stress. Additional stresses are present at the transitions to the eyes. As they may be well above the permissible shear stress, no generally applicable fatigue strength values can be given.

48.2 Shear stress values

The shear stress τ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$$\tau = \frac{8 \cdot D \cdot F}{\pi \cdot d^3} \tag{48.1}$$

τ: Shear stress [N/mm^2]
D: medium coil diameter [mm]
F: spring force [N]
d: wire diameter [mm]

Calculating shear stress for springs subjected to dynamic stress:

$$\tau_k = k \cdot \tau \tag{48.2}$$

τ_k: corrected shear stress [N/mm^2]
τ: Shear stress [N/mm^2]
k: stress correction factor
(dependent on the ratio D/d)

48.3 Manufacturing type

Thermo-formed tension springs cannot be produced with inner pretension force because the heat treatment creates an air gap between the coils. Cold-shaped tension springs can be manufactured in two ways, either by winding on a coiling bench or by winding on a spring winding machine. As defined in DIN EN 13906-2, a formula is specified for each manufacturing method which gives the permissible inner shear stress τ_0.

48.4 Eyes screen

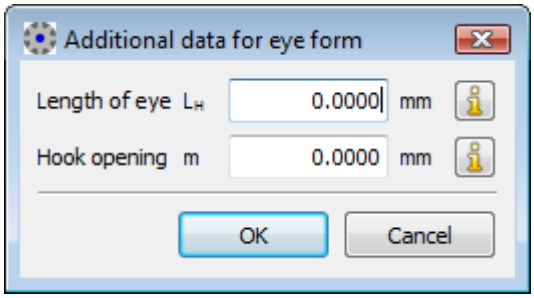
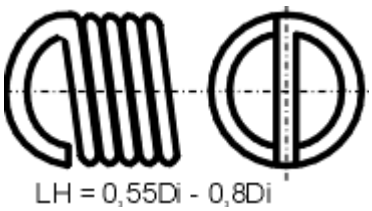


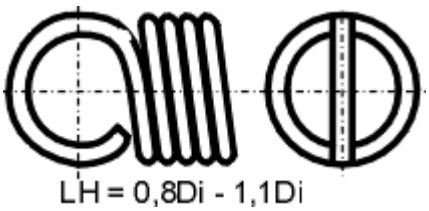
Figure 48.3: Definitions used for eyes

Using the definitions of the `Length of eye` L_H in each case, in this screen, you can then determine the total length of the spring. In contrast, the `Hook opening` m is a reported value that is not used in this calculation.

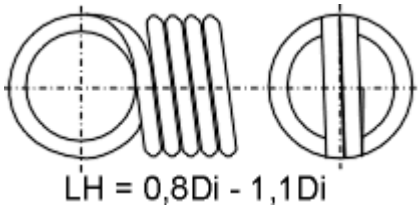
DIN 2097 defines 13 different eye shapes for tension springs. The program suggests different eye lengths depending on the shape of the eye. The position of both eyes is also handled separately in this DIN standard.



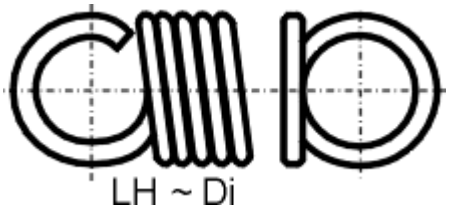
1/2 German loop



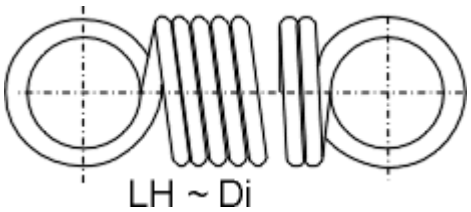
1/1 German loop



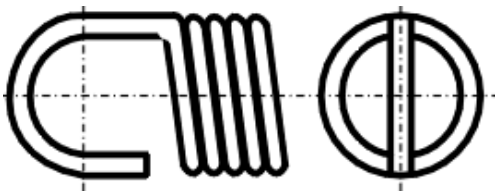
2/1 German loop

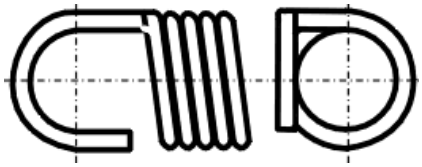


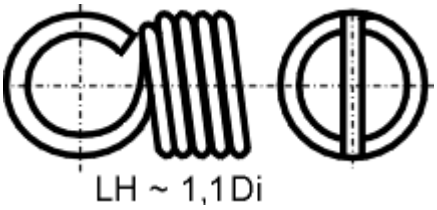
1/1 German loop at side



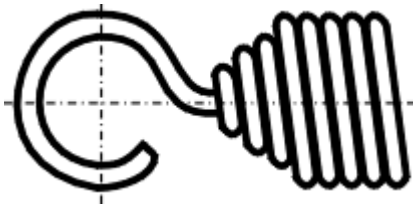
2/1 German loop at side

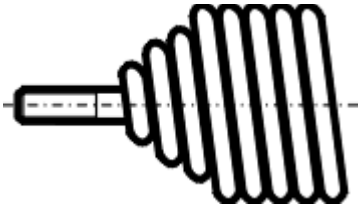

Hook

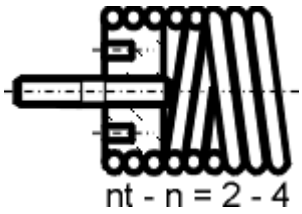

Extended side hook



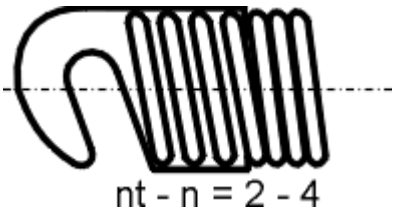
English loop


Coiled-in hook

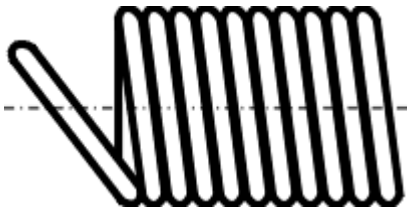

Screwed plug



Screwed-in screw cap



Screwed-in shackle



1/1 German loop inclined

48.5 Materials

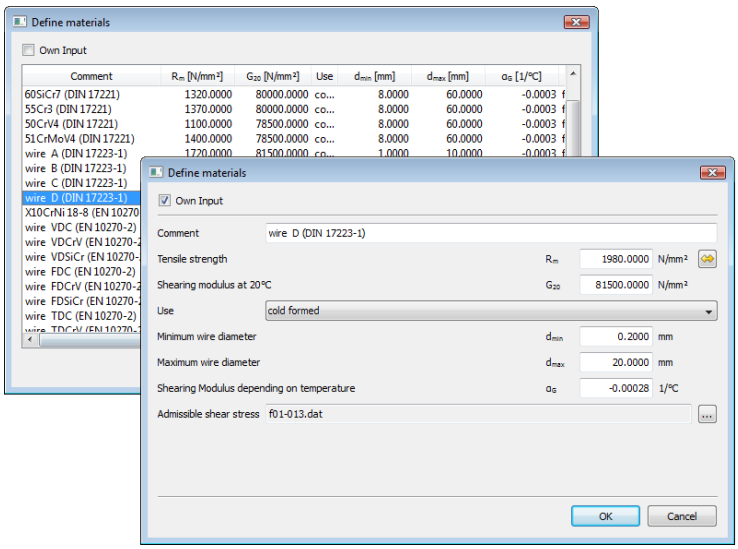


Figure 48.4: Materials screen: Tension springs

The selection list includes materials from the DIN 10089, DIN 17223-1 (withdrawn), DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

48.6 Settings

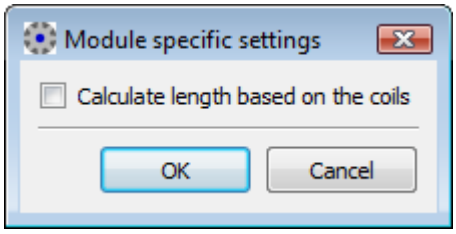


Figure 48.5: Settings: Tension springs

If the `Calculate length using coils` flag is set, and the spring is prestressed (`Initial tension force` flag is set), the length of the spring is calculated from the number of coils. You can no longer input the length in the dialog.

48.7 Tolerances

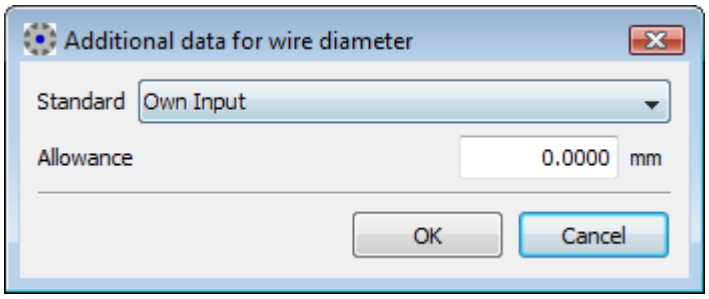


Figure 48.6: Additional data for tension springs wire diameter

Click the  button next to the Wire diameter field to open the Tolerances screen. In this screen, you can select a wire diameter as defined in DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), EN DIN EN 10270-1 (2012), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012), DIN EN 10218 (2012) or select `Own input` to enter your own value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the `Tolerances` list in the basic data you can choose one of the quality standards according to DIN 2097[16] or DIN 2096 Part 1 (1981)[15].

48.8 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard (the standard for compression springs), and shown in diagrams. It was assumed that the same relaxation values can be used for both for tension springs and compression springs. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By noting the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics are also available that show the progress of relaxation over time and the spring force. The results for the specified conditions are then displayed in the relaxation report for 48 h. The value of the spring force is also calculated after 48 h.

To extend the data for the materials relaxation curves, or add new data, add this new information to the *.dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 predefined measurement points. The curves are then calculated from these points.

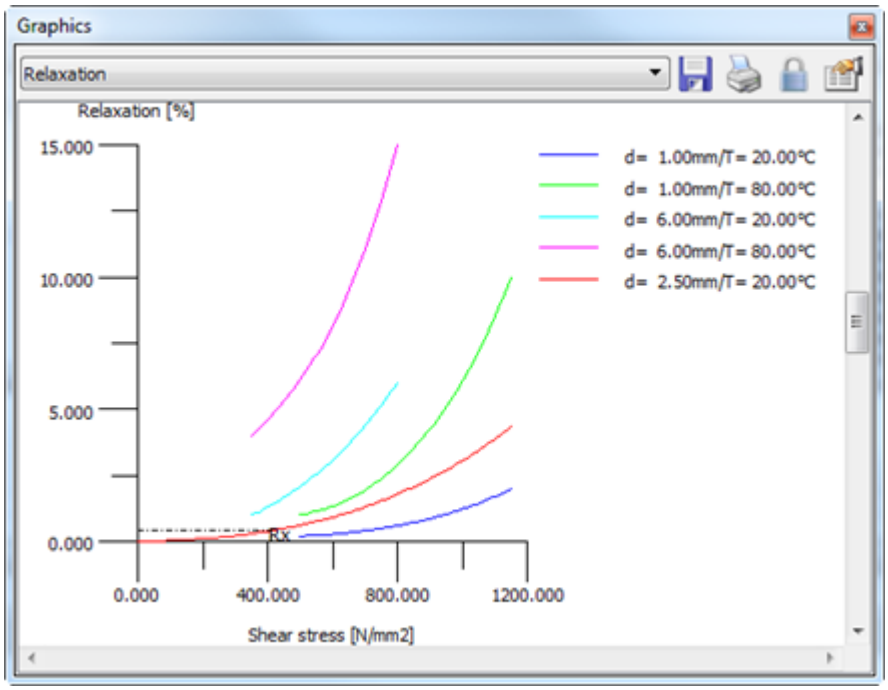


Figure 48.5: Relaxation for tension springs

48.9 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

48.10 Sizings

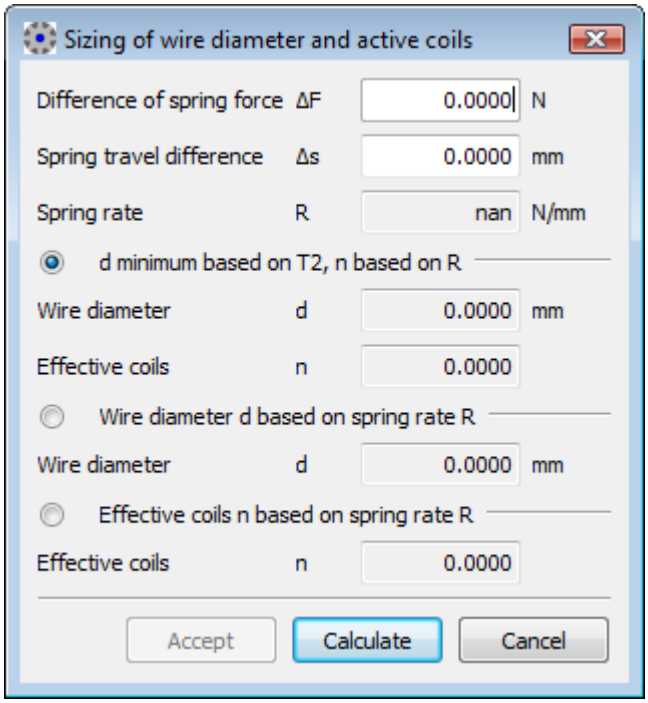


Figure 48.7: Sizing screen: Tension springs

Click the  buttons next to the `Wire diameter` and `Effective coils` fields to use the spring rate $R = \Delta F/\Delta s$ to calculate the number of turns n for the predefined wire diameter. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 49
Leg Springs

The calculation used for leg springs is defined in DIN EN 13906-3 (2014) [32].

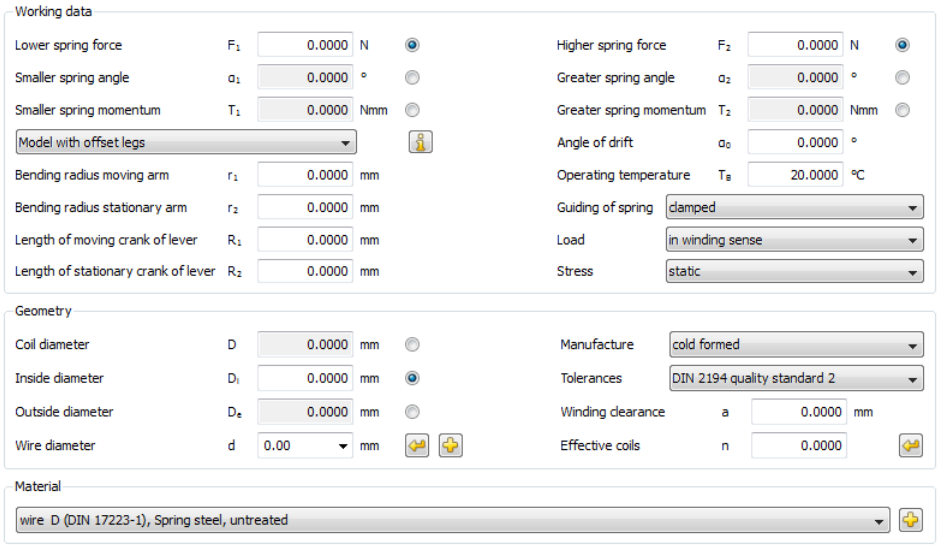


Figure 49.1: Basic data: Leg springs

Operating data

When you define a load you can either enter a value for the spring force, spring angle, or spring torque.
To do this, you must first specify the torsion arm (R_1,R_2) on which the force is applied to the spring.
The value α0 is used to identify the starting angle. This is used together with the direction of load (sense of winding) to calculate the maximum angle of the spring.
Depending on which value you select in the `Guiding of spring` list, the report will also include a reference value for the diameter of the working mandrel or the working bush.
You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

Geometry

You can select the geometry data according to DIN 2098 Part 1 directly from this table.
If you select `Own input`, you can either take selected values from the list or enter your own values. If you select `Own input` you can select a value for the spring diameter and enter it directly.
The winding clearance is the distance between the coils.

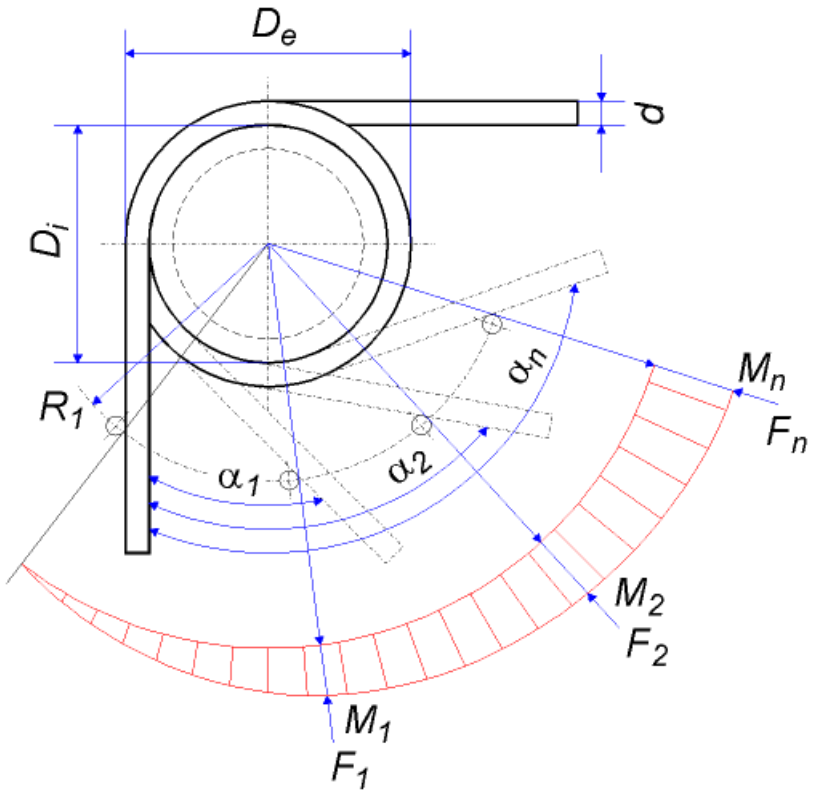


Figure 49.2: Definitions used for leg springs

49.1 Strength values

The permissible bending stress for cold formed leg springs is calculated from the tensile strength. The tensile strength values are determined by diameter values stored in various files. The bending stress is calculated using the formula $\sigma_{zul} = 0.7 \cdot R_m$. These values apply to static or quasistatic cases. The bending of the wire or pin axle due to the load causes an asymmetrical distribution of the spring stresses. In order to approximate the arithmetical stress (dynamic case), you can use the stress coefficient *q* in the calculation.

49.2 Bending stress values

The bending stress σ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$$\sigma = \frac{32 \cdot T}{\pi \cdot d^3} \tag{49.1}$$

σ: shear stress [N/mm^2]
T: spring torque [Nm]
d: wire diameter [mm]

Calculating the bending shear stress for springs subject to dynamic stress:

$$\sigma_q = q \cdot \sigma \tag{49.2}$$

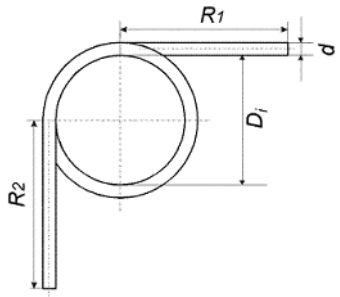
σ_q: corrected bending shear stress [N/mm^2]
σ: bending shear stress [N/mm^2]
q: stress correction factor
(dependent on the ratio D/d)

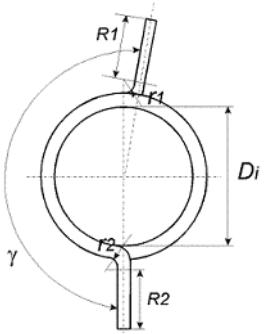
49.3 Spring design

In order to prevent friction, the coils either do not touch each other or under only slight stress. The following applies, for the greatest achievable winding clearance:

$$a_{max} = (0.24 \cdot w - 0.64) \cdot d^{0.83}$$

Generally, leg springs are wound. There are two options for the leg design: they can be either bent with offset (the radius must be specified) or tangential.





with tangential legs

with offset legs

49.4 Assumptions made for the calculation

The calculations apply only to leg springs with fixed or circular guided spring ends. If the leg is not fixed, the spring must be guided by means of a pin or sleeve.

49.5 Materials

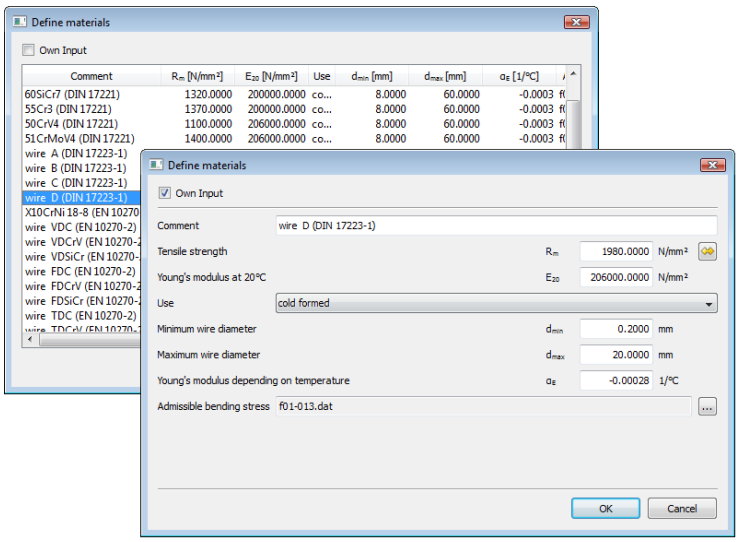


Figure 49.3: Materials screen: Leg springs

The selection list includes materials from the DIN 10089, DIN 17223-1 (withdrawn), DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

49.6 Tolerances

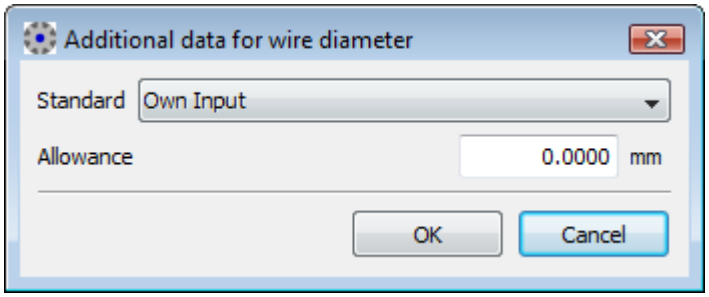


Figure 49.4: Additional data for leg springs wire diameter

Click the button next to the `Wire diameter` field to open the Tolerances screen. In this screen, you can select a wire diameter as defined in DIN 2076 (1984, withdrawn), DIN 2077 (1979, withdrawn), EN DIN EN 10270-1 (2012), DIN EN 10270-2 (2012), DIN EN 10270-3 (2012), DIN EN 10218 (2012) or select `Own input` to enter your own value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

In the `Tolerances` list in the basic data you can choose one of the quality standards according to DIN 2194 (2002)[17].

49.7 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

49.8 Sizings

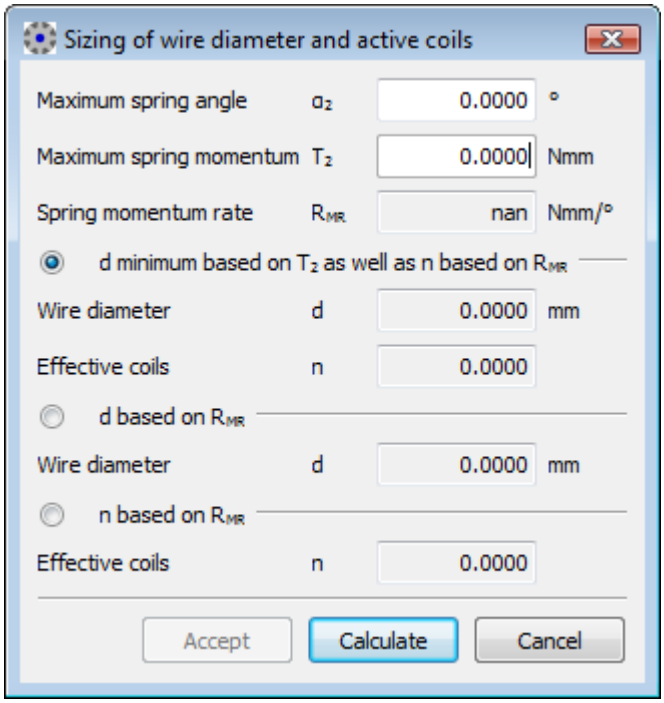


Figure 49.5: Sizing: Leg springs

Click the  buttons next to the `Wire diameter` and `Effective coils` fields to use the spring momentum rate *RMR* = $\Delta M/\Delta\alpha$ to calculate the number of turns n for the predefined wire diameter. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 50
Disc Springs

The calculation for disc springs is described in DIN 2092 (2006) [12]. The mass and quality requirements are handled according to DIN 2093 (2006)[13].

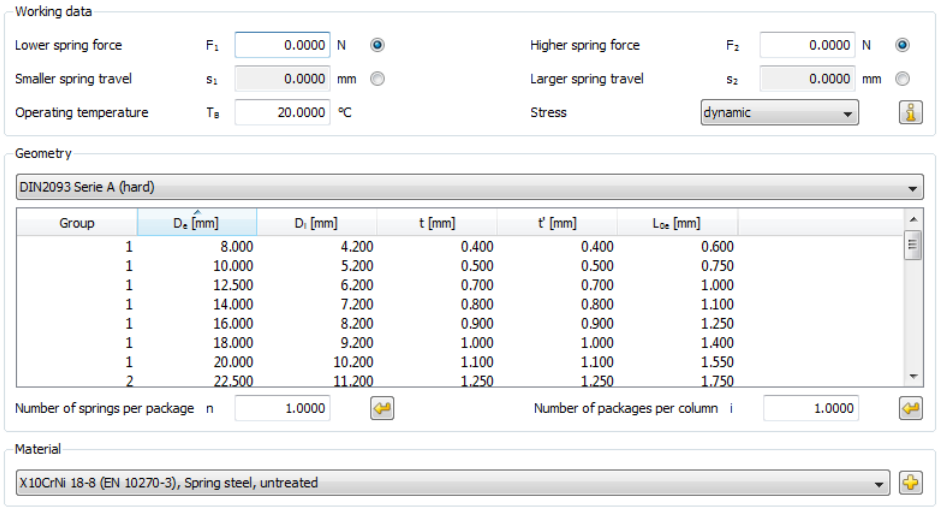


Figure 50.1: Basic data: Disc springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

The calculations specified in DIN 2092 are for disc springs with or without bearing areas for the ratios 16 < De/t < 40 and 1.8 < De/di <2.5 and materials specified in DIN 2093.

Geometry

As specified in DIN 2093, disc springs are divided into 3 groups and 3 sequences. Groups 1 and 2 contain the springs without a bearing area, whereas group 3 has the springs with a bearing area. The disc thickness for group 1 is less than 1.25 mm, in group 2 it is between 1.25 and 6 mm, and in group 3 it lies between 6 and 14 mm. These groups differ in the hardness of the springs in them. Series A includes hard springs, i.e. they can withstand larger forces in a smaller travel of spring. This is followed by series B and series C which can withstand the least force in a larger

travel of spring. If you select `Own input`, the input fields for geometry data become active, and you can enter your own values in them. This type of calculation only applies to springs without a support area, because the ratio of the thicknesses t'/t is not known, but it is still required for the calculation.

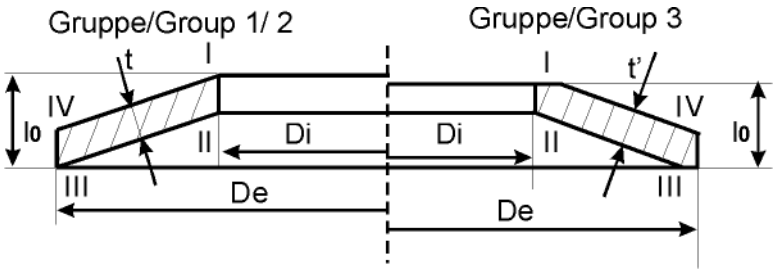


Figure 50.2: Dimensions of the disc springs

50.1 Strength values

In the case of springs that are subject to static or quasistatic load ($N <= 10^4$), the maximum force on the spring is calculated. The formula is predefined in DIN 2092. This force then is compared to the effective force Fn of the spring (at s = $0.75 \cdot h_0$) and the utilization of the spring force is calculated. If the required force is greater than the spring's effective force, Fn, the deviation in the calculation is too large. The DIN formula for calculating force only applies where the travel of the spring is $s = 0.8 \cdot h_0$. The springs can be used in packages or columns to handle larger forces. The calculated total force of the system is then displayed in a force travel diagram. The calculation for disc springs is performed without taking friction into account.

Furthermore, the stresses are calculated at edge points I to IV. Points I and IV are subjected to compression, and points II and III are put under tension. Under dynamic load, the stress range is calculated using the maximum stress (either at point II or III) with the associated lower stress level. The permissible permanent stress range is defined using a Goodman diagram. These values are then compared to give the number of load cycles the springs should be able to withstand under load. DIN 2093 includes Goodman diagrams which are only valid for materials specified in DIN 17221 and 17222. If any other materials are involved, you must contact the spring manufacturer.

50.2 Stress values

Stresses are calculated for edge points I to IV.

Point I:

$$\sigma_I = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^{\,2}} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right] \quad (50.1)$$

Point II:

$$\sigma_{II} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^{\,2}} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right] \quad (50.2)$$

Point III:

$$\sigma_{III} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^{\,2}} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right] \quad (50.3)$$

Point IV:

$$\sigma_{IV} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^{\,2}} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right] \quad (50.4)$$

σ_I - σ_{IV} : Stress at points I to IV [N/mm^2]
E: Young's modulus [N/mm^2]
μ: Poisson's ratio [-]
D_e: External diameter [mm]
s: Spring travel of an individual disc [mm]
t: Thickness of an individual disc [mm]
h_0: Travel of spring until flat [mm]
δ: Diameter ratio (D_e/D_i)
K_1 - K_4: Variables calculated from formulae (DIN 2092)

50.3 Materials

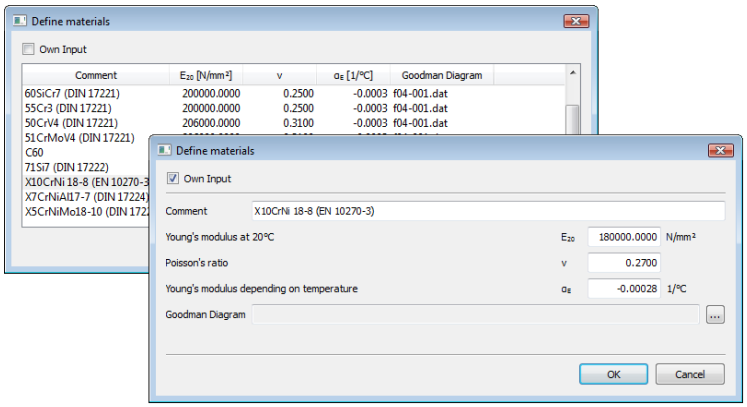


Figure 50.3: Materials screen: Disc springs

The selection list includes materials from the DIN 17221, DIN 17222, DIN 17224 and DIN 10270-3 standards.

If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

50.4 Calculate number

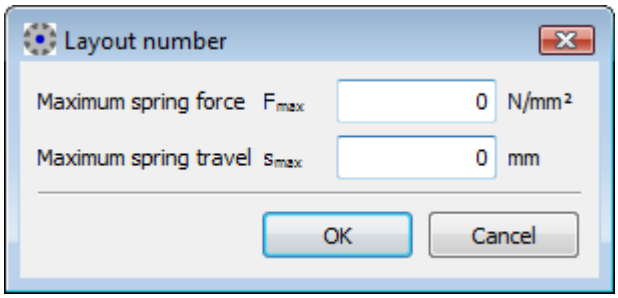


Figure 50.4: Sizing the number of packages/ columns

To estimate the number of discs or columns required, click the button next to the fields for number of springs per package or number of packages per column. In this screen, you can define the maximum force and the maximum travel can be defined. These values are then used to calculate and display the number of springs per package or the number of packages per column.

50.5 Limit dimensions

For all disc springs, the external diameter D_e must lie in the tolerance field h12 and the inner diameter D_i must lie in a tolerance field H12.

Chapter 51
Torsion-Bar Springs

The calculation for torsion-bar springs is defined in DIN 2091 (1981) [[11].



Figure 51.1: Main screen: Torsion-bar calculation

Operating data

When you specify the load, you can enter a value for either an angle of rotation or a torsional moment.

If a torsion-bar is set as the default (`Torsional bar preplaced` flag) the permitted shear stress of the torsion-bar, τ_{zul}, is increased.

Geometry

Enter specific parameters to define the geometry of the spring.

For the `toothed` head form, you must also specify the number of teeth, although this is purely for documentation and is not used in the calculation. The standard assumes shearing modulus G as a constant. However, the calculation is still permitted even if this value is slightly different.

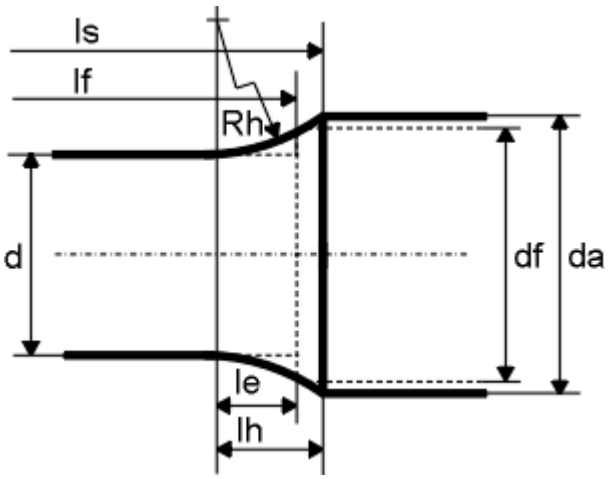


Figure 51.2: Defining a torsion-bar

51.1 Head forms

Torsion-bar springs as defined in DIN 2091 can have one of three different head forms: rectangular, hexagonal and toothed. Toothed torsion-bar heads are usually produced according to DIN 5481 Part 1 or SAE J 498 b. However, they can also be manufactured with special toothings. The body forms of the heads apply only to bars that are loaded in one shaft rotation direction. Alternating loads require special design measures.

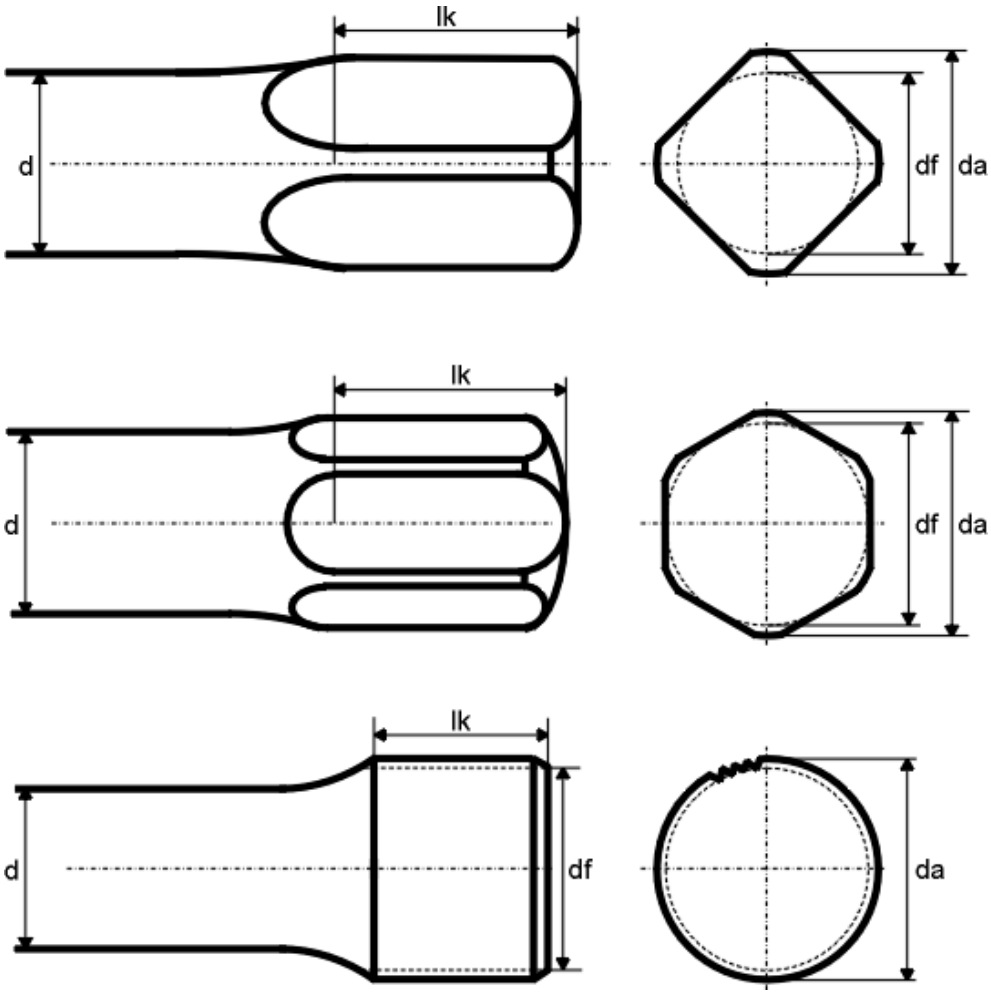


Figure 51.3: Shapes of torsion-bar heads (rectangular, hexagonal, toothed)

51.2 Strength values

Definitions in DIN 2091:

DIN 2091 only applies to the materials defined in DIN 17221.
The permitted shear stresses:
For non-preloaded bars: $\tau zul = 700$ N/mm^2
For preloaded bars: $\tau zul = 1020$ N/mm^2
The heat treatment strength for these values is: $Rm = 1600 - 1800$ N/mm^2
For the shearing modulus, $G = 78500$ N/mm^2 is used as the default. Due to a preload (above the yield point, deformed in the direction of operation) after the torsion-bar springs have been heat treated, there will be a better distribution of the working stress, and a relief in the boundary zone will be achieved.

51.3 Shear stress

Calculating shear stress τ:

$$\tau = \frac{16 \cdot T}{\pi \cdot d^3} \qquad (51.1)$$

τ: shear stress [N/mm^2]
T: Torsional moment [Nm]
d: wire diameter [mm]

51.4 Limiting values

The following limit ratios for torsion-bar heads apply to torsion-bar springs:
rectangular, toothed: $df/d > 1.3$; hexagonal: $df/d > 1.25$
The strength values from the DIN standard apply to bar diameters of 10 to 60 mm.
The guide value is a head length between $0.5 \cdot$and $1.5 \cdot d$.
The Rh/d ratio should be between 1 and 50.

51.5 Sizings

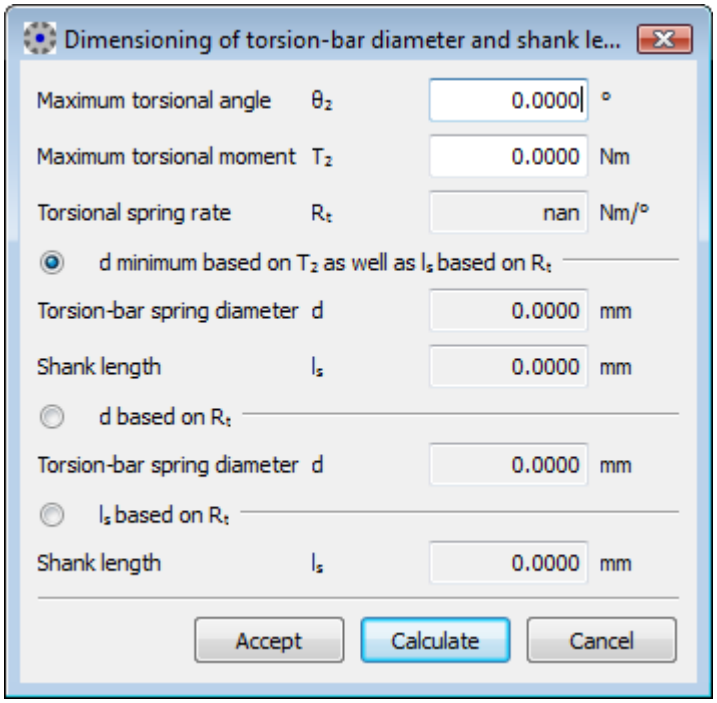


Figure 51.4: Sizings screen: Torsion-bar springs

Click the  buttons next to the `Wire diameter` and the `Shaft length` fields to open the screen described above. You can enter the torsional moment and the angle of rotation in the sizing screen. These values are used to calculate the torsional spring rate, which is then used to size bar diameter d or shank length ls. When sizing d and ls, you must first calculate d with the permitted shear stress value. You can then calculate the shank length ls from the bar diameter d. Various values are assumed so that you can size the dimensions (rectangular, toothed: df = 1.35·d; hexagonal: df = 1.3·d; da = df + df/7; Rh = (da-d)·1.2).
These values are not transferred to the main screen.

Part VI

Belts and chain drives

Chapter 52
V-Belts

Preamble:
Follow the manufacturer’s instructions when sizing and verifying V-belt drives. Most catalogs detail the entire calculation method. As the amount of power V-belts can transfer improves because of better materials and flank shapes, manufacturers' data provides the only really reliable values.

Fully automated calculation including standard V-belt lengths and standard effective diameters. Determining transmittable power per belt while taking into account the speed, effective diameter, transmission ratio and belt length. All the data is taken from manufacturers tables (for example, ContiTech). This also includes a belt stress calculation module that uses data from belt-bending tests. This calculates the end of rope force and axle load at standstill and in operation for optimum setting as well as for setting in accordance with data in the catalogs.

As a variant, the calculation can also be performed with a third roller (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **V-belt** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular X and Y value is displayed in the status row. This roller can be positioned outside or inside as required.

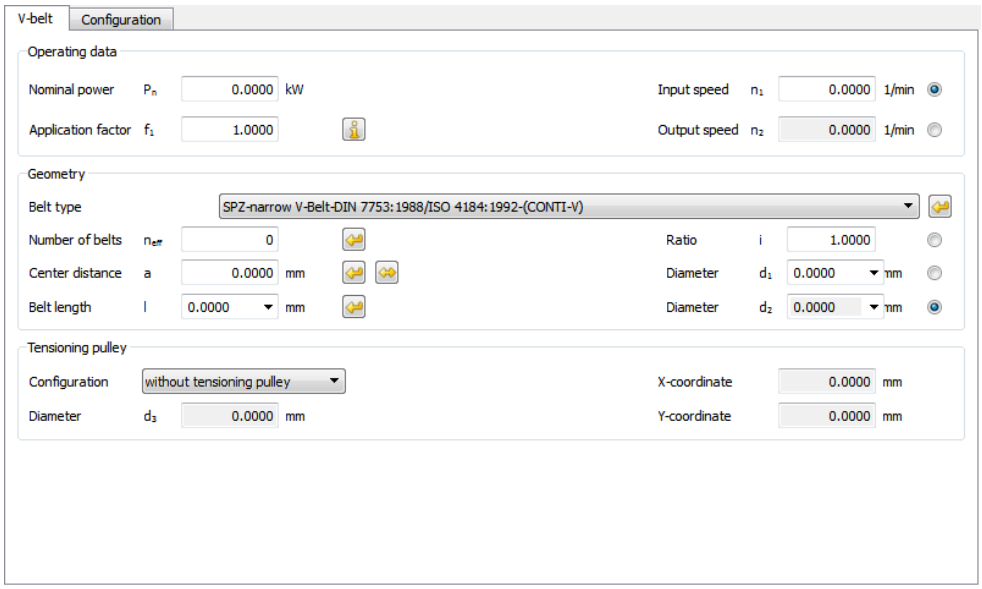


Figure 52.1: Basic data: V-belt calculation

52.1 V-belts data

KISSsoft stores the tabular values (catalog data) in files which you can then edit. You will find these file names in the KISSsoft database tool for the corresponding belt types (for example, Z090-015.dat for XPA narrow V-belts).

52.2 V-belt standards

The following belt types are available:

- XPA-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- XPB High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- XPC-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- XPZ-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- 5/-High-performance V-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- 6/Y-High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- 8/- -High-performance v-belts-DIN7753/ISO4184-(CONTI-FO-Z)
- SPZ-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPA-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPB-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- SPC-Narrow V-belts-DIN7753/ISO4184-(CONTI-V)
- 8/-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 10/Z-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 13/A-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 17/B-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 20/-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 22/C-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 25/- -Multiflex-v-belt-DIN7753/ISO4184-(CONTI-V STANDARD)
- 32/D-Multiflex V-belts-DIN7753/ISO4184-(CONTI-V STANDARD)
- 40/E Multiflex V-belts DIN7753/ISO4184 (CONTI-V STANDARD)
- 3V-9J Force-belts
- 5V-15J-Force-belts
- 8V-25J-Force-belts

- 3V-9N Narrow V-belts-USA-standard
- 5V-15N Narrow V-belts-USA-standard
- 8V-25N-narrow-v-belt-USA-standard

52.3 Configuring tensioning pulleys

Here, you can select either:

- non-tensioning pulley
- inside tensioning pulley
- outside tensioning pulley

If you selected an inside/outside tensioning pulley, you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the `Configuration` tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

52.4 Application factor f1

You can enter this factor in the basic data screen. If you selected a configuration with a tensioning pulley, you should increase factor f1 by 0.1. The table shown below is used to define the f1 factor (refer to the catalogs for more information):

	Drive motor with normal starting torque, combustion engines and turbines with *n* over 600 min^{-1} For daily operation hours:			**Drive motor with high starting torque, combustion engines and turbines with *n* over 600 min^{-1} For daily operation hours:**		
	up to 10 h	from 10 h to 16 h	over 16 h	up to 10 h	from 10 h to 16 h	over 16 h
light drives	1.0	1.1	1.2	1.1	1.2	1.3
medium-heavy drives	1.1	1.2	1.3	1.2	1.3	1.4
heavy drives	1.2	1.3	1.4	1.4	1.5	1.6
very heavy drives	1.3	1.4	1.5	1.5	1.6	1.8

Figure 52.2: Application factor V-belts

52.5 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

The conversion process uses a suggestion for new designs according to the ContiTech catalog. Here:

```
0.7*(d1+d2) <= a <= 2.0*(d1+d2)
```

In KISSsoft, the average coefficient for the suggested value is used in the conversion.

```
a = 1.35*(d1+d2)
```

52.6 Belt length

You need to know the belt length before you can calculate a V-belt. If you have not specified a length, or if you change to a configuration that involves a tensioning pulley, you must ensure that the program recalculates the belt length.

52.7 Effective number of V-belts

The effective number of V-belts is calculated from the theoretical number by rounding this value up to the next highest whole number.

52.8 Tensioning pulley diameter

The tensioning pulley diameter should be at least as big as the smallest belt sheave. If at all possible, you should not use tensioning pulleys, in particular outside tensioning pulleys. However, if you have to use a tensioning pulley, its diameter should be at least 1.33·d if it is an outside pulley or 1.0·d if it is an inside pulley (d: diameter of the smaller sheave).

Every manufacturer provides slightly different information about tensioning pulleys.

52.9 Position of tensioning pulley (x/y)

When you configure the tensioning pulley, you can enter the position of the pulley (in x/y-coordinates). Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the `Configuration` tab, you can only move the tensioning pulley within the valid area.

52.10 Inspecting V-belts

(belt bending test)

The actual axial stress of V-belt drives is calculated from data provided by the belt bending test. Enthusiastic mechanics have a tendency to over-tension belt drives, and therefore subject them to loads that are too high for their capabilities.

Chapter 53
Toothed Belts

Use this method to calculate and size all aspects of toothed belt drives, including the tooth number and belt length, while taking into account standard numbers of teeth. When you enter the required nominal ratio and/or the nominal axis distance, the program calculates the best possible positions. You can also calculate the required belt width, taking into account the correction factors, the minimum tooth numbers, and the number of meshing teeth. You can also print out assembly details (belt bending test). The data for each type of belt is saved to text files, whose names indicate their purpose, which can be edited as required.
You can also perform calculations for special stress-resistant toothed belts with integrated steel ropes (e.g. AT5mm).

As a variant, the calculation can also be performed with a third roller (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **Toothed belts** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular x and y value is displayed in the status row. This roller can be positioned outside or inside as required.

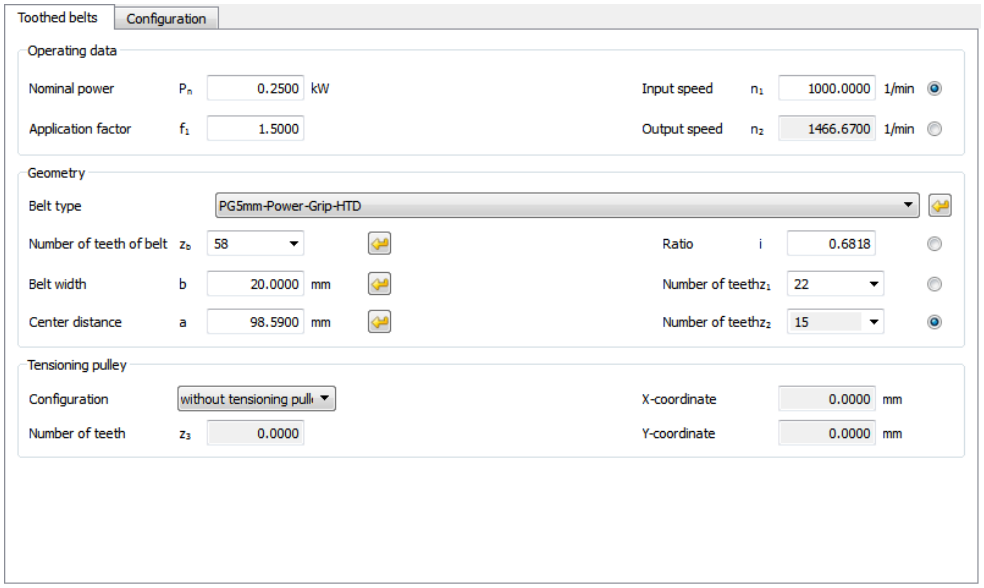


Figure 53.1: Basic data: Toothed belt calculation

53.1 Technical notes (toothed belts)

Preamble:
Follow the manufacturer’s instructions to achieve the best results when sizing and verifying toothed belt drives. Most catalogs detail the entire calculation method. As the amount of power V-belts can transfer improves because of better materials and flank shapes, manufacturers' data provides the only really reliable values.

Elasticity:
As the manufacturers catalogs provide very little data on this subject, you must treat the belt elasticity constraint values with caution. The elasticity (in N) is the force required to lengthen the belt by 100%.

Weight:
As the manufacturers catalogs provide very little data on this subject, you must treat these values with caution.

Pretensioning the belt:
As the manufacturers catalogs provide very little data on this subject, you must treat the constraint values with caution. The calculation method and the factors it uses are stored in the Z091-0??.DAT files where they can be changed if required. You can use one of the processes shown below to calculate the required pretensioning values for various types of belts. The data here is taken from the catalogs:

Belt type:	**Pretension:**	
Breco AT5, AT10, AT20	0.5	* Circumferential force
Synchroflex AT3, AT3 GIII, AT5 GIII, AT10 GIII	0.5	* Circumferential force
Isoran XL, L, H, 8, 14	0.625	* Circumferential force
HTD 3, 5, 8, 14	0.25	* max. permitted circumferential force
8MGT, 14MGT Poly Chain GT2	0.5	* Circumferential force
RPP-HPR 8, 14	0.5	* Circumferential force

Table 53.1: Pretension

Forces in no load/load are calculated according to [66], equation 27/23.

$$F_V = \frac{F_1 + F_2}{2} \quad (53.1)$$

$$F_1 = F_V + \frac{F_t}{2} \quad (53.2)$$

$$F_2 = F_V - \frac{F_t}{2} \quad (53.3)$$

53.2 Toothed belt standard

You can select one of these standards:

- XL-ISORAN RPP (FENNER)
- L-ISORAN RPP (FENNER)
- H-ISORAN RPP (FENNER)
- 8mm ISORAN RPP (FENNER)
- 14mm ISORAN RPP (FENNER)
- 8mm-ISORAN-RPP-GOLD (Megadyne)
- 14mm-ISORAN-RPP-GOLD (Megadyne)
- 8mm-ISORAN-RPP-SILVER (Megadyne)
- 14mm-ISORAN-RPP-SILVER (Megadyne)
- RP8mm-Pirelli RPP-HPR
- RP14mm-Pirelli RPP-HPR
- PG3mm-Power Grip-HTD
- PG5mm-Power Grip-HTD
- PG8mm-Power Grip-HTD
- PG14mm-Power Grip-HTD
- 8mm-DAYCO-RPP-(Panther)
- 8MGT-Poly-Chain-GT-Carbon (Gates)
- 14MGT-Poly-Chain-GT-Carbon (Gates)
- 8mm MGT-Poly Chain-GT2

- 14mm MGT-Poly Chain-GT2
- AT3mm SYNCHROFLEX
- AT3mm GEN III-SYNCHROFLEX
- AT5mm GEN III-SYNCHROFLEX
- AT10mm GEN III-SYNCHROFLEX
- AT5mm-BRECOflex
- AT10mm-BRECOflex

AT20mm-BRECOflex*Additional standards on request.*

53.3 Possible sizings/suggestions

The following sizings are possible if you select the different  buttons:

Variable	**Influencing/necessary variables**
Belt profile	Power
	Speed (small disc)
	Operating factor
Number of teeth on belt	Center distance
	Number of teeth on sheave
Center distance	Number of teeth on belt
	Number of teeth on sheave (all)
Ratio	Center distance
	Nominal ratio
	Speed (small disc)
Number of teeth on tensioning pulley	Number of teeth (small disc)

Table 53.2: possible sizings

53.4 Configuring tensioning pulleys

Here, you can select either:

- non-tensioning pulley
- inside tensioning pulley
- outside tensioning pulley

If you selected an inside/outside tensioning pulley, you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the `Configuration` tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

53.5 Application factor and summand for operational behavior

You can either enter the application factor manually in the load factor interface, or have the program define it from the operating parameters. If you selected a configuration with a tensioning pulley, you must increase the operating factor by 0.1. Use the data in this table to define the factor (refer to the catalogs for more information):

Driven machine	**Operating hours per day**		
	0-10	**10-16**	**16-24**
Light drive	1.2	1.3	1.4
Medium-light drive	1.4	1.5	1.6
Medium-heavy drive	1.5	1.6	1.7
Heavy drive	1.7	1.8	1.9
Heavyweight drive	1.8	1.9	2.0

Table 53.3: Application factors

Summand for operational behavior
(This summand is added to the operating factor in the calculation)

Operational behavior:	**Summand**
continuous, 0-10 hr/day	0
continuous, 10-16 hr/day	+0.1
continuous, 16-24 hr/day	+0.2
intermittent or with alternating load	-0.1

Table 53.4: Summand

53.6 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

53.7 Belt length and number of teeth on belt

In toothed belt drives the number of teeth on the belt is used to define the belt length. You need this value when you perform the calculation for the belt. If you did not specify the number of teeth on the belt or, if you switched to configuration with a tensioning pulley, you must ensure that the program recalculates the value for the number of teeth on the belt.

53.8 Effective belt width

The theoretical belt width (minimum width required to transmit the torque) can be calculated from the data in the manufacturer catalogs. The effective belt width is then taken as the next largest standard belt width.

As a general rule, the belt width should not be larger than 5*pitch. A warning message is displayed if you select a belt that is either too wide or too narrow. Although the calculation continues, you use the data it provides at your own risk.

Defining the effective belt width/factor for the belt width:

To define the belt width, you will need the belt width factor (f_b). Use this formula to calculate this factor:

$$f_b = \frac{Betriebsleistung\ /\ operating\ \ power}{Nennleistung\ nach\ Katalog\ /\ No\,minal\ \ power\ according\ to\ catalog\,e} \qquad (53.4)$$

The nominal power as specified in the catalog is a tabular value taken from the manufacturers' catalogs and is dependent on the speed and number of teeth on the smaller belt sheave.

With the calculated coefficient *f_b* you can then define the effective belt width from a catalog table. However, if the value of *f_b* does not match a standard belt width, the next biggest width will be used.

Remarks:
The theoretical belt width in the KISSsoft calculation reports corresponds to an interpolated value, according to calculated factor *f_b*.

KISSsoft stores the tabular values (catalog data) in files which you can then edit. Use the KISSsoft database tool to find the exact file name for a specific belt type (e.g. Z091-001.DAT for XL-Isoran).

53.9 Number of teeth in tensioning pulley

The value you use for the number of teeth on a tensioning pulley should be at least as large as the value given for the diameter of the smallest belt sheave.

Where possible, tensioning pulleys should be used as inside tooth sheaves. However, if necessary, they can also be used as smooth sheaves from outside. The diameter of the tensioning pulley should be at least 1.2 *d if positioned outside, or 1.0 *d if positioned inside (d: diameter of the smaller sheave). Every belt manufacturer provides very different data about tensioning pulleys.

For Poly Chain GT:

> An outside tensioning pulley reduces service life and should be avoided if possible.

For AT-belts:

AT5mm	with tensioning pulley inside:	25 mm ($z > 5$)
	with tensioning pulley outside:	50 mm ($z > 10$)
AT10mm	with tensioning pulley inside:	50 mm ($z > 5$)
	with tensioning pulley outside:	120 mm ($z > 12$)
AT20mm	with tensioning pulley inside:	120 mm ($z > 6$)
	with tensioning pulley outside:	180 mm ($z > 9$)

53.10 Position of the tensioning pulley x/y

You must enter this value when you configure a tensioning pulley. Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the `Configuration` tab, you can only move the tensioning pulley within the valid area.

Chapter 54
Chain Drives

Use this module to calculate chain drives with roller chains as defined in ISO 606 (with standardized roller chain values taken from a database). The chain geometry (center distance, number of chain elements) for simple and multiple chains and the transmissible power, axial forces, and variation in speed, are calculated by the polygon effect, etc. Basis: DIN ISO 10823, [38] and [64].
During this calculation, the program checks the highest permitted speed and shows a suggested value for the required lubrication.

As a variant, the calculation can also be performed with a third roller (tensioning pulley). The X and Y coordinates of the tensioning pulley can be defined in the **Chain drives** tab. If you open the **Configuration** tab, you can use the mouse to move the tensioning pulley. In this case, the particular x and y value is displayed in the status row. This roller can be positioned outside or inside as required.

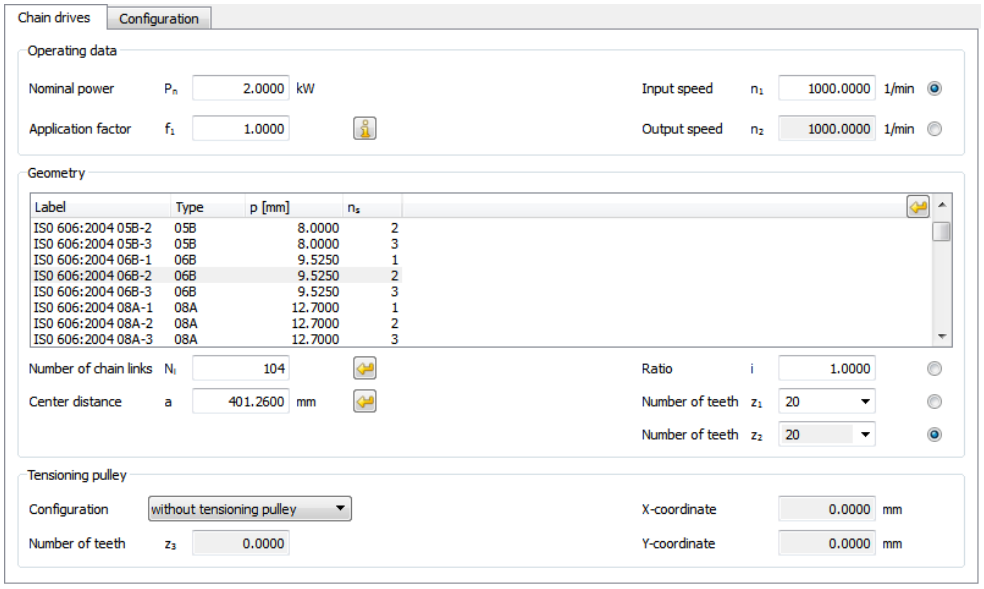


Figure 54.1: Basic data: Chain calculation

54.1 Sizings

Using the drive data as a starting point, the program displays a list of suggested values for suitable chain drives.

- Calculating the center distance from the chain length.
- Calculating the chain length from the center distance.

54.2 Tensioning pulleys

You require tensioning pulleys if you need to limit the chain deflection or keep to a minimum wrap angle. You must arrange the tensioning pulleys under no load. They must have at least three teeth.

54.3 Standard

Chain profile standard:

- Roller chain ISO 606

The roller chain standard, ISO 606, includes chains as defined in the DIN 8154, 8187 and DIN 8188 standards. Roller chains are the most frequently used type of chain because lubricated rollers reduce noise and wear. The chains defined in DIN 8187 correspond to the European type, those defined in DIN 8188 correspond to the American type. You should only install bush chains as defined in DIN 8154 in closed gear cases with sufficient lubrication.

54.4 Chain type

The data shown below depends on the type of chain:

- Chain pitch.
- Maximum permitted speed of the small gear.
- Nominal power at maximum permitted speed.

Tables in ISO 606 pages 8 to 0.

54.5 Number of strands

You can achieve high levels of power by using multiple chains. Chains are often arranged in two or three strands (Duplex, Triplex). The values for duplex and triplex chains are also given in the same standard.

54.6 Application factor

Guide values according to DIN ISO 10823, Table 2:

Working characteristic of the driven machine	Working characteristic of the driving machine		
	uniform	light shocks	moderate shocks
uniform	1.00	1.10	1.30
moderate shocks	1.40	1.50	1.70
heavy shocks	1.80	1.90	2.10

Figure 54.2: Application factor for chain calculation

54.7 Speed/number of teeth/ratio

Range of ratio:

favorable i = 1. . . 5,

good i = 1. . . 7,

unfavorable i = > 10.

Number of teeth:
Due to the polygon effect, we recommend a minimum number of teeth of between 17 and . . 25. Tooth numbers of less than 17 should only be used to produce low levels of power. The preferred numbers of teeth for use in chain gears, as stated in ISO 606, are: 17, 19, 21, 23, 25, 38, 57, 76, 95, 114.

You should use at least three teeth for tensioning pulleys.

54.8 Configuration

You can select one of these configurations:

- without tensioning pulley
- with tensioning pulley inside
- with tensioning pulley outside

In a configuration involving tensioning pulleys, you must specify the number of teeth and the position of the tensioning pulley (x/y). In the `Configuration` tab, you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

54.9 Center distance

Recommended center distance: $a = 30 \cdot p \ldots 50 \cdot p$ (p: pitch)

You should avoid: $a < 20 \cdot p$ and $a > 80 \cdot p$

Click the button to calculate the center distance from the number of chain links.

54.10 Polygon effect

When calculating chains, you must take the polygon effect into account both for the reference circle and the center distance.

- Formula for the reference circle:

$$d = \frac{Teilung}{\sin\left(\frac{\pi}{z}\right)} \tag{54.1}$$

- (see also [66], equations 26/46)
- Formula for the center distance:
  The length of the loop on the sprocket differs as follows from the formula used for V-belts/toothed belts:

$$l_{UK} = l_{UR} \cdot \frac{\sin\left(\frac{\pi}{z}\right)}{\frac{\pi}{z}} \tag{54.2}$$

- l_{UK}: Length of loop for chains
  l_{UR}: Length of loop for V-belts

54.11 Number of links

The number of links should usually be an even number.

Click the button to calculate the number of links from the center distance.

54.12 Sprocket geometry

In KISSsoft, you can display and print out the sprocket geometry as a graphic as defined in ISO 606. The graphics are created with a mean deviation.

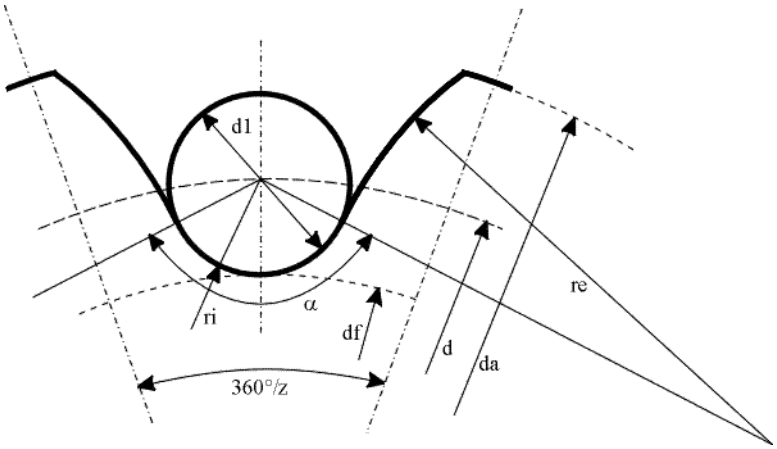


Figure 54.3: Geometry of chain sprocket

You can also output other values for a sprocket in a report. The figures in this section show how specific information is represented in this report.

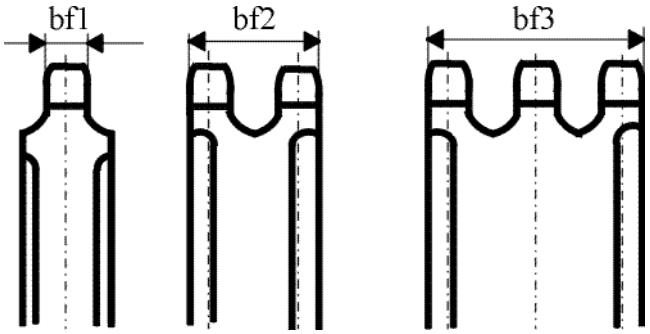


Figure 54.4: Chain sprocket width

Part VII

Automotive

Chapter 55

Synchronization

Use this module to calculate the gear synchronization time and total time, based on the specified geometry, forces and application data. Some additional calculations for heat development, frictional power, and wear resistance, are also performed. Calculations can be performed for common types of synchronizations for a given number of cones (single, double or triple cone).

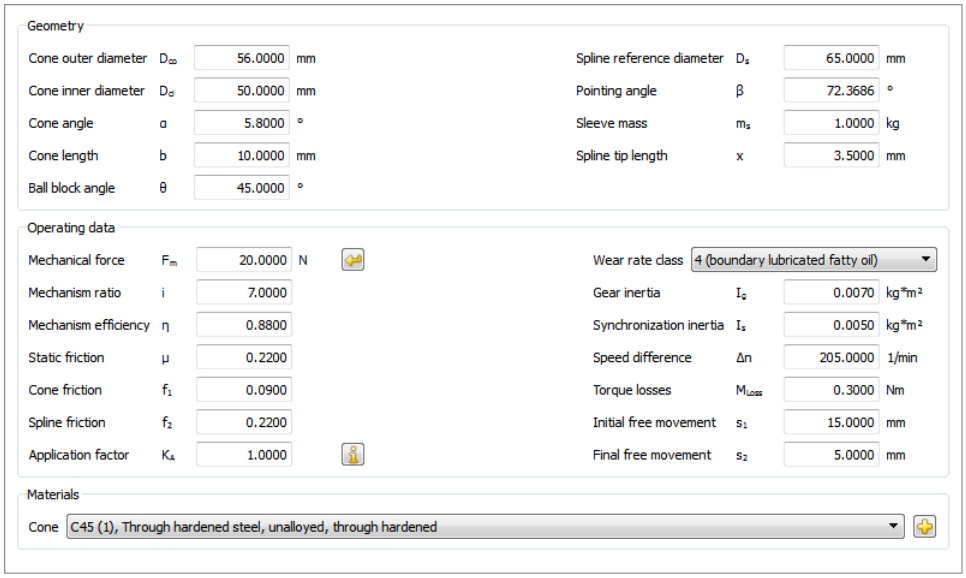


Figure 55.1 Synchronization module tab

55.1 Geometry

Geometry data is needed for the synchronization ring, also called the cone. Additional data is needed for the spline tip definition (the indexing) and ball block angle. This is the external ball angle which holds the synchronizer at its position (engaged or disengaged). Specific limit values have been defined for the angle input to ensure synchronization can be guaranteed.

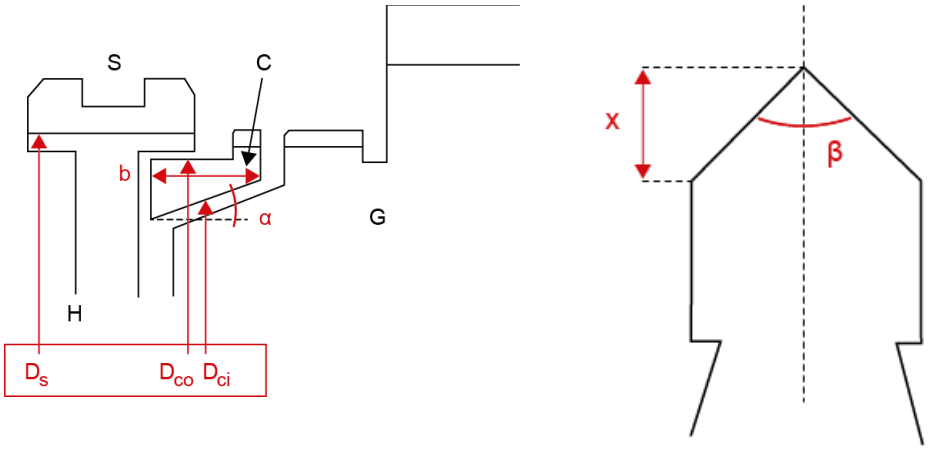


Figure 55.2: (a) Description of main geometry: S = Sleeve, C = Ring/Conus, H = Hub, G = Gear, (b) Spline tip geometry

55.2 Operating data

The mechanical force is the force applied to the shifting handle. This force is multiplied by the mechanical ratio and applied to the sleeve. The coefficient of friction at the beginning of the synchronization can be defined, for the conus and the sleeve.

The gear inertia and the speed difference are required entries. Torque losses due to mechanical friction, oil splashing, and other sources, can be defined. The defined losses during the shifting will either help or hinder the process depending on the shifting direction. If there is a small amount of clearance on the synchronizer sleeve before and after the actual synchronization, the distances can be entered here to enable the total time (from another gear to the final end position of the sleeve) to be calculated correctly.

Chapter 56
Friction clutches

This module is used to calculate friction clutches and brakes in accordance with VDI 2241 [90]. The results of this calculation can then be used to select a suitable clutch or brake. The clutches are operated either mechanically, electromagnetically, or by pressure (e.g. hydraulically), thereby either generating or removing pressing force. The clutches can be designed to run either dry or with lubrication. This has a significant effect on the coefficient of sliding friction and the coefficient of static friction.

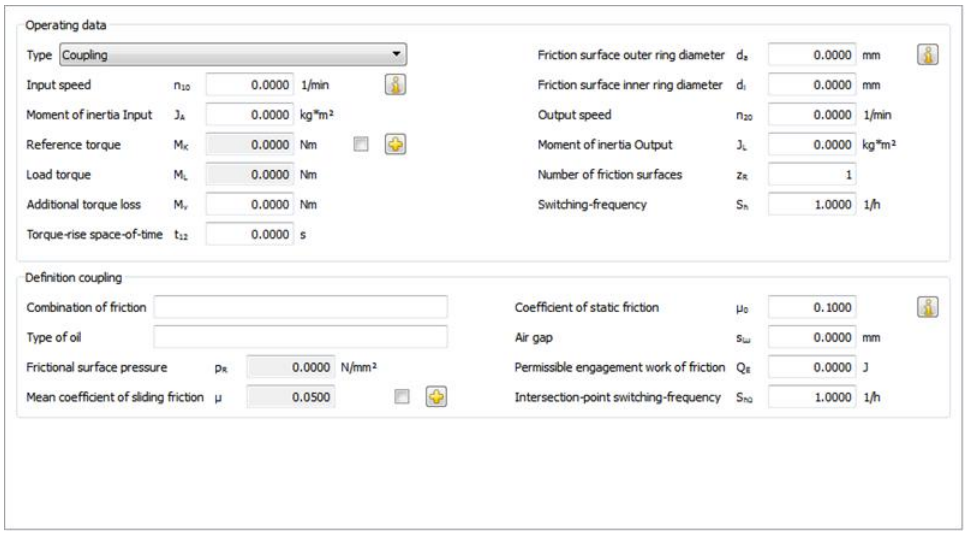


Figure 56.1: Basic data: Friction clutches

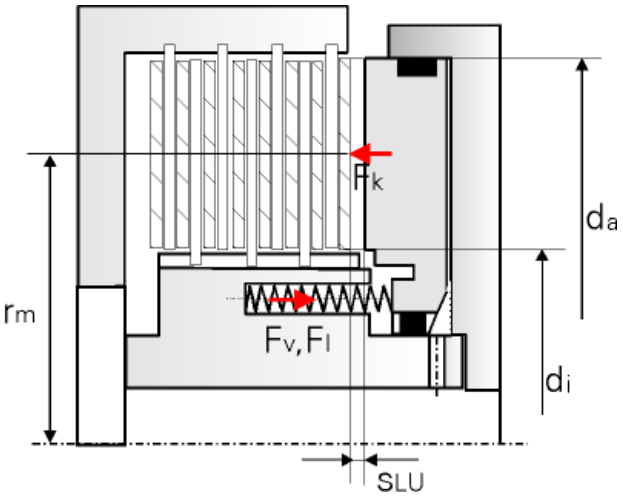


Figure 56.2: Display of a clutch

Force is stored in a spring. When the spring is released, the force returns the clutch to its open state. Compression or disc springs are usually used here. Both types of spring are pretensioned in their open state. In this example the compression is created hydraulically, and therefore affects the piston. This additional definition of storing force is not included in the VDI guideline. The guideline assumes that frictional surface pressure is applied directly to the plate. As the dynamic characteristics of the springs can also be non-linear, the force generated by the contact with the first plate is used in the calculation.

In KISSsoft, you can either define the spring forces or input the reference torque MK and the load torque ML directly. As specified in VDI 2241, the work of friction and the switching capacity are defined using an average sliding velocity and an average coefficient of sliding friction. You can also specify the coefficient of sliding friction as a dependency of 5 sliding velocities, because this coefficient can vary greatly depending on which sliding velocity is present. However, this does not take into account the aging of the oil, which would reduce the coefficient of sliding friction.

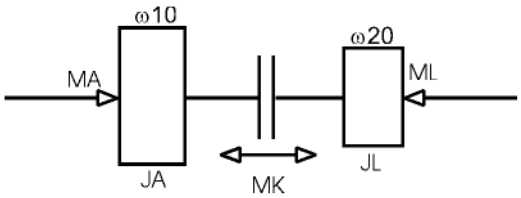


Figure 56.3: Schematic display of a clutch

The dynamic moment of inertia JL can also be made up of a number of different parameters. If there is a mass m at the distance r from the rotational axis, its moment of inertia can be calculated with $JL2 = r^2*m$.

This can then be added to the existing moment of inertia JL1.

JL =JL1 + JL2.

Ratios can then reduce the moment of inertia on the clutch shaft $J2red = J2*(n2/n1)^2$. This reduced moment of inertia can then be added to the clutch shaft's moment of inertia.

JL = JL1 + J2red

56.1 Calculation

Inputting the spring forces/defining the reference torque

If you decide to input the spring forces (Reference torque flag is set), the reference torque is calculated as follows:

$$F_{stat} = Fl$$

$$F_{dyn} = Fk - Fv$$

Fk (N): Piston force (FK = pK/AK)

pK (N/mm^2): Compression on the piston

AK (mm^2): Piston surface area

Fl (N): Spring force to plates contact

Fv (N): Pretension for spring force

F (N): Resulting force on the first plate (N)

The accelerating torque or the holding torque is then determined from this. Using the different coefficients of sliding friction, if these have been defined, otherwise using the average coefficient of sliding friction:

$$M_A = F_{dyn} \cdot \mu \cdot r_m \cdot z_R$$

$$M_L = F_{stat} \cdot \mu 0 \cdot r_m \cdot z_R$$

MA (Nm): Accelerating torque

Fstat, Fdyn (N): Resulting force on the first plate

μ (-): Coefficient of sliding friction

rm (mm): average friction radius

zR (-): Number of friction surfaces (plates)

ML (Nm): Holding or load torque

μ0 (-): Coefficient of static friction

The reference torque is then defined from MA+ML. You can also define a torque loss, which has a negative sign for a clutch and a positive sign for a brake.

However, if you define the reference torque directly, you cannot also define a torque loss. This must then be taken into account with the reference torque.

The formulae specified in VDA 2241 [90] are used to define the sliding time t_3.

For a clutch:

MK=MA, with a specified MK, with the influence of t12 (in this sequence):

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = (J_A \cdot J_L)/(J_A \cdot (M_K - M_L) + J_L \cdot (M_K - M_A)) \cdot (\omega_{10} - \omega_{20})$$

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12}/2 \cdot (M_L/M_K + 1)$$

For a brake:

with a specified MK, with the influence of t12 (in this sequence):

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12}/2 \cdot (M_L/M_K + 1)$$

The engagement work of friction Q is then calculated with, or without, taking the torque-rise time t12 into account, depending on whether or not this value has been defined. The switching capacity on the total friction surfaces and the maximum switching capacity are also calculated.

If you input curve points for the coefficient of sliding friction, the area below the calculated curve in the torque diagram is calculated as the engagement work of friction. The switching capacity is then derived from the time-based conclusion of this calculation.

Each of these values must be input as specific values for the friction surfaces because these are provided by the manufacturers in the relevant catalogs.

Furthermore, when you input the switching frequencies and the permitted engagement work of friction (one-time switching) the program calculates a utilization to show whether the selected clutch will be adequate.

$$Q_{zul} = Q_E \cdot (1 - e^{-Shü/Sh})$$

Qzul (kJ): permissible engagement work of friction

QE (kJ): permissible engagement work of friction (one-time switching)

Shü (1/h): Intersection-point switching frequency

Sh (1/h): Switching frequency per time unit

The utilization AQ is then determined from this permitted value and the calculated engagement work of friction:

$$A_Q = Q_{zul} / Q$$

When you select a clutch, you must take into account the reference torque, and most importantly, the permissible engagement work of friction QE (one-time

switching) and the calculated permissible engagement work of friction (for higher switching frequencies).

56.2 Definition of spring forces

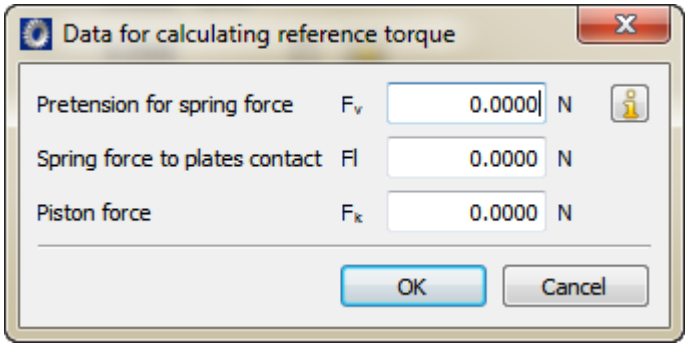


Figure 56.4: Definition of spring forces

These additional inputs, `Pretension for spring force Fv` and `Piston force Fk`, are used to determine the characteristic values required to calculate the resulting spring force. The coefficient of sliding friction and the average radius rm and the number of plates are then applied to determine the accelerating torque.

The coefficient of static friction from the `Spring force to plates contact Fl` is then used to define the holding torque.

56.3 Definition coefficients of sliding friction and velocities

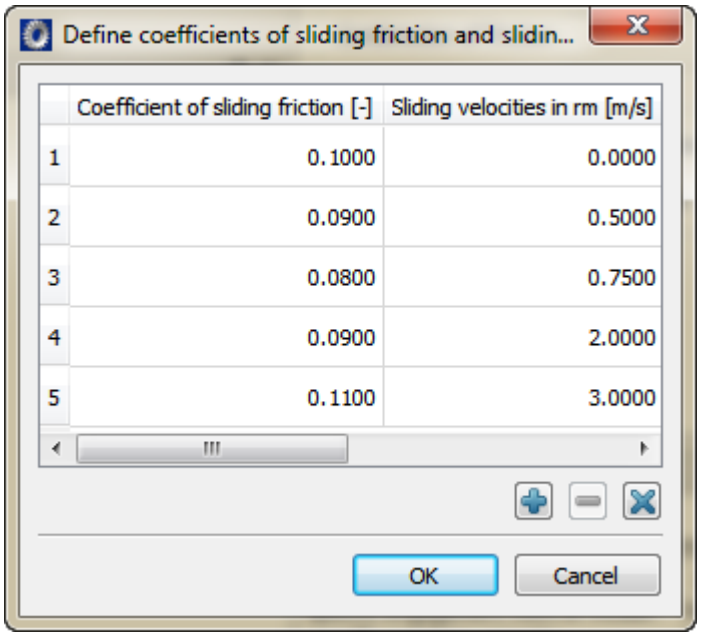


Figure 56.5: Definition coefficients of sliding friction and velocities

The coefficients of sliding friction are specified by the manufacturers in accordance with the sliding velocities. The VDI 2241 standard assumes that a constant value is used. However, this may result in a large deviation in results. By inputting a maximum of 5 points you can create a poly line that connects these points. From this line the program can then derive 10 values for the coefficients of friction in the sliding velocity areas at the start and at the end. The 10 different accelerating torques derived from this can then be used later on in the calculation.

56.4 Graphics

The graphics show the speed curve over sliding time t3, the torque diagram over sliding time t3, and the coefficient of sliding friction curve for the sliding velocity, of which a maximum of 5 points have been entered (if defined by the user).

56.5 Settings

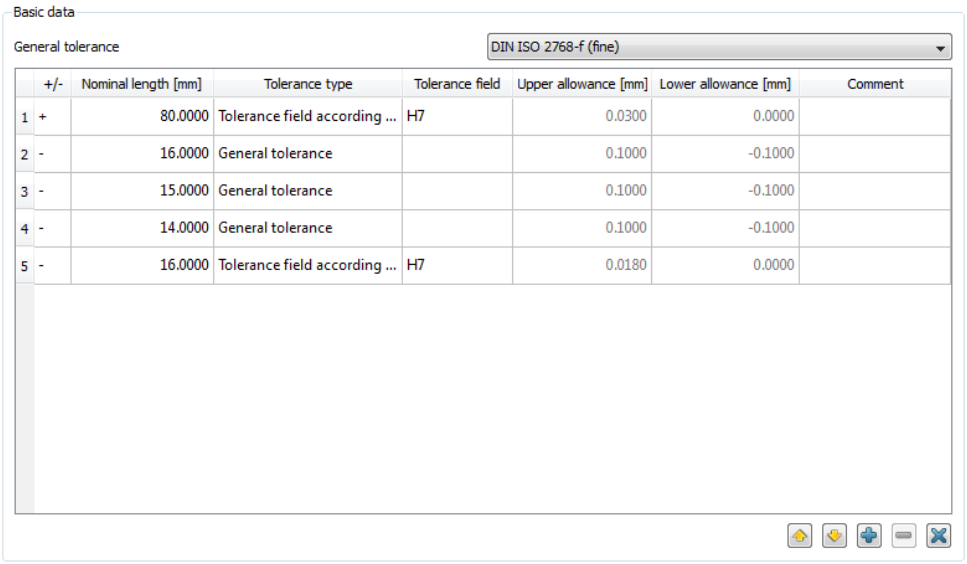
If the `Use radius to plates gravity center for the calculation` flag is set, the radius at the center of gravity for the plates is used in the calculations instead of the mean radius of the plates rm. This radius is calculated with:

$$rs = \frac{2}{3} \cdot \frac{ra^3 - ri^3}{ra^2 - ri^3}$$

Part VIII
Various

Chapter 57
Tolerance Calculation

In this module, you enter the nominal lengths and their corresponding allowances for various elements. These values are then used to calculate an overall tolerance. This calculation uses a constant distribution (arithmetical sum) and the square root of the tolerance squares (standard distribution) to define the maximum and minimum size of the measurement chains. You can also use the appropriate dimensions to calculate the nominal length/expected value of the measurement chain. The `Tolerance field specified in ISO` is defined according to ISO 286, in which the tolerances are defined up to a size of <= 3150 mm. In KISSsoft, for tolerance classes H, h. JS and js, the values used in the standard are extrapolated up to a value of 10,000 mm.



	+/-	Nominal length [mm]	Tolerance type	Tolerance field	Upper allowance [mm]	Lower allowance [mm]	Comment
1	+	80.0000	Tolerance field according ...	H7	0.0300	0.0000	
2	-	16.0000	General tolerance		0.1000	-0.1000	
3	-	15.0000	General tolerance		0.1000	-0.1000	
4	-	14.0000	General tolerance		0.1000	-0.1000	
5	-	16.0000	Tolerance field according ...	H7	0.0180	0.0000	

Figure 57.1 Basic data

Chapter 58
Strength Verification with Local Stresses

58.1 General

You can start this calculation in the `Various` section of the modules tree.

58.1.1 Functionality of the software

The calculation program supplies a complete, written proof of integrity for static and fatigue strength at the proof point W.

The strength verification is supplied in accordance with the local stress concept as described in the FKM "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" guideline. The idea behind the local stress concept is to estimate the service life on the basis of the elastic-plastic, local stress at the critical point compared to the S-N curve (Woehler lines) elongation derived from an unnotched test bar. The local concept is implemented as a stress-based variant within the framework of the FKM Guideline. Therefore, before it can be used, the material must be in an elastic state. In this context, the concept used is not really a local concept like the elastic-plastic notch root strain concept, but a concept close to the nominal stress concept, except that the theoretical stress concentration factor stands on the other side of the equation. It is a useful tool for calculating a static proof and proof of fatigue strength in the high cycle range ($N > 10^4$).

Input: You can enter stress amplitudes and stress ratio at a proof point W and at a neighboring point B. Alternatively, the stress ratio at the proof point and the support effect are estimated mathematically. You must also specify parameters such as surface roughness, part size, etc., before you can calculate the design coefficients. Additional load data, such as number of cycles, spectrum, temperature etc. are also predefined.

Output: The calculation calculates the utilization factors for static cases and fatigue. It creates a complete set of documents for this.

58.1.2 Areas of application for the FKM guideline

The software is based on the FKM "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" Guideline, Chapters 3 and 4. The guideline applies to mechanical engineering and its associated industrial sectors. In real life scenarios, contractual partners must agree how this guideline is to be implemented. For parts that are subject to mechanical stress, this guideline can be used to calculate the static and fatigue strength either for a finite or infinite working life. However, this guideline does not cover other mathematical proofs such as brittle fracture stability, stability or deformation under load, or experimental strength verifications. Before the guideline can be applied, it is assumed that the parts have been manufactured so that all aspects of their design, material and operation are technically free of error

and fit for purpose. The guideline is applicable for parts made of iron and aluminum alloys, even at elevated temperatures, either by machining or welding, and in particular for

- parts with geometric notches
- parts with welded joints
- static stress
- fatigue loads ranging from approximately ($N > 10^4$) cycles as an individual or collective load
- rolled and forged steel, including stainless, mix cast iron alloys as well as forged and cast aluminum alloys
- different temperatures
- a non-corrosive ambient medium.

Supplementary agreements must be drawn up if this guideline is to be used outside the specified area of application. The guideline does not apply if a strength verification is required using other standards, codes or guidelines, or if specific calculation data, such as VDI 2230 for bolted joints, is applicable.

For simple rod-shaped and planiform elements, we recommend you use a calculation method that involves nominal stresses. The calculation using local stresses is to be used for volumetric parts or, in general, where stress is to be calculated using the finite element method or the boundary element method, if no specifically defined cross sections or simple cross section forms are present or if the shape values or notch effect values are unknown.

58.2 Background

58.2.1 The FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile

The idea for this guideline was proposed at a meeting of the DVM in Berlin, Germany, in May 1990, when experts from the then Federal Republic of Germany met together with experts from the then German Democratic Republic. The objective was to combine the standards from what was then two separate standards (VDI in the West and TGL in the East) to create one new strength assessment guideline. The new guideline was to be based, in particular, on the former TGL standards for strength calculation, VDI guideline 2226, DIN 18800, Eurocode 3 and the recommendations of the IIW. The FKM Guideline was also updated to take into account the latest discoveries from research into the fatigue strength of metallic parts. The FKM Guideline is designed for use in mechanical engineering and associated industrial sectors. The first edition of the FKM Guideline, "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" appeared in 1994, followed in 1998 by a third, completely reworked and extended edition (characterized by its much more practical updates and a more user-friendly structure). A fourth edition, which was even more comprehensive, was published in 2002. The main innovation of this edition was the inclusion of aluminum materials. An English translation of this guideline appeared as the fifth edition. The sixth edition of the guideline (2012) has, once again, been completely reworked, and now includes the results of new research, such as the data supplied by the tests in "Margin of safety static resistance" and "Improved Calculation Concept in the FKM Guideline". In the meantime, the FKM Guideline has been widely accepted and is regarded as the best reflection of the current state of technology.

58.2.2 Usefulness of the service life calculation

It is a well known and proven fact that the service life calculation is not sufficiently accurate. In other words, factors in the range from 0.1 to 10, and in some cases even greater, may occur between the calculation and the test. However, a basic, if somewhat simplified, statement about the difficulties in achieving a reliable service life calculation has been made: In this case the strength verification is based on a comparison of the stress values and the stress itself. In a static strength verification, the occurring force can be compared with the sustainable force. For a proof of service strength, the characteristic functions, i.e. the stress spectrum and the S-N curve (Woehler lines) are compared. If the total damage, which is of central significance to the service life calculation, is then understood as a quotient of the characteristic functions for stress and sustainable stress, it is clear that this quotient is very sensitive to changes in these critical values. This means errors in determining the

characteristic functions will have a significant effect on the result. In addition, by influencing the critical values, for example, by implementing specific measures when selecting materials and at the production stage, the long-term sustainable service life can be increased.

Three different concepts can be used to calculate the service life of components that are subjected to cyclical stress. These are: the nominal stress concept, the local concept and the fracture mechanics concept. These concepts have specific application areas. For many decades, the technical set of rules was based solely on the nominal stress concept. However, nowadays the local concept and the fracture mechanics concept are being used more and more frequently in this set of rules. In the nominal stress concept, the complex transfer function between stress and service life contained in the total stress-elongation event in critical material volumes (notch bottom area) is given directly with the component S-N curve (Woehler lines) for nominal stresses. By contrast, in the local concept, this must be represented mathematically by a number of relatively complex modules. This may be the reason for results according to previous experience not being any more accurate than those achieved with the nominal stress concept.

Possible sources of errors in calculating the local concept:

Load assumptions

It must be emphasized that the load assumption must be as precise as possible to ensure an accurate calculation of component service life. Any errors in load assumption can have significant effects on the service life calculation results. The effect may even be greater than those due to insufficient accuracy of the different methods used for service life estimations. We recommend you check the load assumptions carefully and test them if necessary. In this way, any uncertainties in the load assumptions can be resolved by actual measurements performed at a later date. That is particularly because this type of measurement can be performed non-destructively, and can usually provide important information for subsequent designs.

Local stress

Local stresses can be determined either mathematically or by measurement. It is essential that the part's geometry is entered accurately, especially the splines and wall thicknesses. A convergence check must also be performed to ensure the effective stresses are not underestimated. However, a problem still to be resolved in productive operation is how to calculate the effective level of internal stresses in a part cross section, or in a surface layer, so that this can be evaluated when subjected to load stresses in a service life calculation.

Combined stress

In the case of combined stress, a strength calculation should fulfill the instance of the invariant (results independent of the selected coordinate system). However, as S-N curve (Woehler lines) with different inclinations are used for normal and shear stresses, the resulting calculated service life/damage is no longer separate from, and independent of, the selected coordinate system.

Material characteristics

Since it is usually not possible to ascertain material characteristics by simply measuring the finished part, we recommend you use standardized or, at least, well-documented values. It is acknowledged that these values may be dispersed and not always suitable. It is also not possible to determine reliable endurance limit values from tensile strength Rm alone. The fatigue limit can be estimated using the proof stress Rp02. The FKM Guideline defines the values from Rm and also for the material type.

Cyclical deformation characteristic

A check to see whether cyclical compaction or loss of cohesion is present must be performed to see whether or not the sequence of load cycles plays a significant role. This effect is not considered in the calculation program.

Support effect

A number of different models can be used to ascertain the support effect. As many comparisons between calculated results and test results have shown, a mathematical estimate of the support effect is fraught with uncertainties.

Production processes

When a local concept is applied, it is assumed that the volume element displays cyclical material behavior. Influences encountered during the production process, in particular surface layer characteristics, surface roughness, material state and internal stresses must be taken into consideration. Currently used calculation methods also have their limitations here.

Damage parameters

A number of damage parameters have been proposed to help determine the influence of mean stress and the influence of multiple shafts. PSWT, the most well-known damage parameter, corresponds to a mean stress sensitivity of M=0.41, which is present in this order of magnitude for heat treated steel, but assumes entirely different values for low strength steels or wrought aluminum alloys. The use of PSWT should be seen as a major source of errors. Also in question is the extent to which the influence of internal stresses can be determined. In the latter case, this is only known for exceptional cases in practice. Damage parameters are still widely used by researchers to determine multi-shaft behavior, excluding pro-

portional stress. The influence of multi-shaft stress states on service life depends greatly on which material is being used. This is because the material's resilience determines which different damage mechanisms are present.

Damage accumulation

In practice, damage accumulation occurs almost exclusively in accordance with the Palmgren-Miner linear hypothesis. Although the shortcomings of this hypothesis were recognized early on, no significant advances that would lead to tolerable errors in the service life calculation have been made in this area, despite decades of intensive international research. The only progress is that, by summarizing the amplitudes below the endurance limit, various modifications have been proposed, which achieve much better results than the original Palmgren-Miner rule, and in which no damage is caused when amplitudes are below the endurance limit.

Even if the service life calculation methods for evaluating variants and analyzing weak points are implemented correctly, it is not certain that the current level of knowledge can achieve a reliable service life calculation for new parts. This requires the use of strategies where calculations are validated and calibrated by specific experimental analyses. At the current level of knowledge, it is only possible to make relative forecasts about service life on a purely mathematical basis.

58.3 Implementation in KISSsoft

58.3.1 Main screen

58.3.1.1 Selection of the part form

Selection of the part form: you can choose between parts that are rod-shaped (1D), shell-shaped (2D) or block-shaped (3D). They each have different stress components or stress types, and different indexing. If the local concept is applied, block-shaped (3D) parts are usually present. The selected part form influences the data input for the stress components.

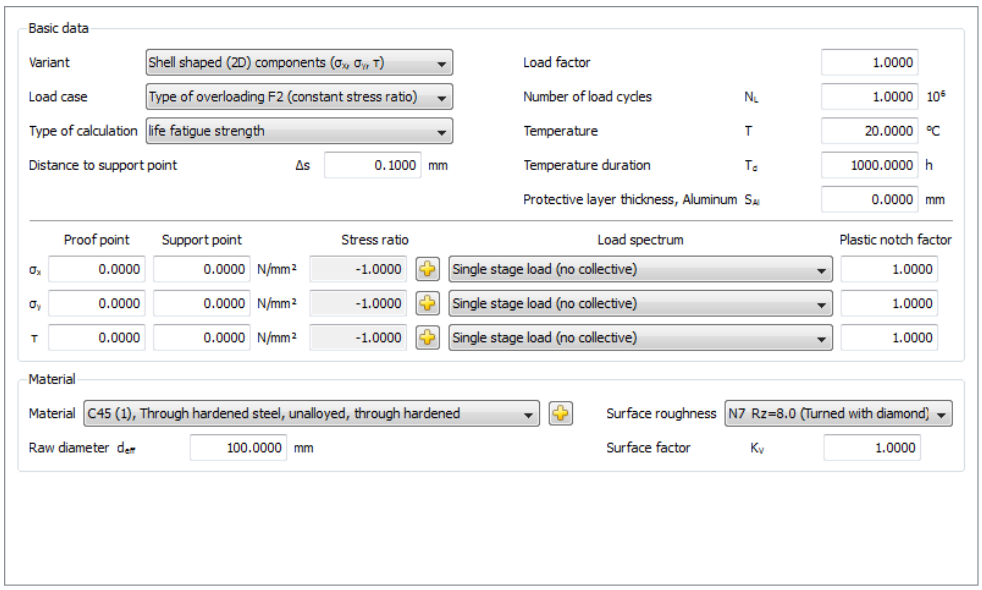


Figure 58.1: Main screen for the proof with local stresses

Rod-shaped parts: the following part-related coordinates system applies, for rod-shaped parts - rods, beams, shafts: The X-axis lies in the rod axis, and the Y- and Z-axes are the main axes of the cross section, and need to be specified in such a way that Iy > Iz applies for the moment of inertia.

For planiform (flat) parts - disc, plate, shell, - the following part-related coordinates system should apply in the proof point: the X- and Y-axes lie in the plane, and the Z-axis is vertical to it in the direction of thickness. The normal stress and the shear stresses in the direction of Z should be negligible.

Block-shaped parts: volume-related coordinates systems apply. The primary stresses σ1, σ2 and σ3 need to be calculated. In the proof point W on the free surface of a 3D part, the primary stresses σ1 and σ2 should act in the direction of the surface and the primary stress σ3 points into the interior of the part, vertically to them. Generally, there is one stress gradient that runs vertically to the surface, and two stress gradients in the direction of the surface, for all stresses. However, only the stress gradients for σ1 and σ2, running vertically to the surface, can be taken into account in the calculation, and not the stress gradients for σ1 and σ2 in both directions on the surface, and none of the stress gradients for σ3.

58.3.1.2 Inputting the stress values on the proof point and on the neighboring point

If the support factor is determined according to the stress state on the neighboring point, then the stresses on the proof point W and on the neighboring point B, and also the distance from point B to point W, will be entered. (Enter compressive stresses as negative values):

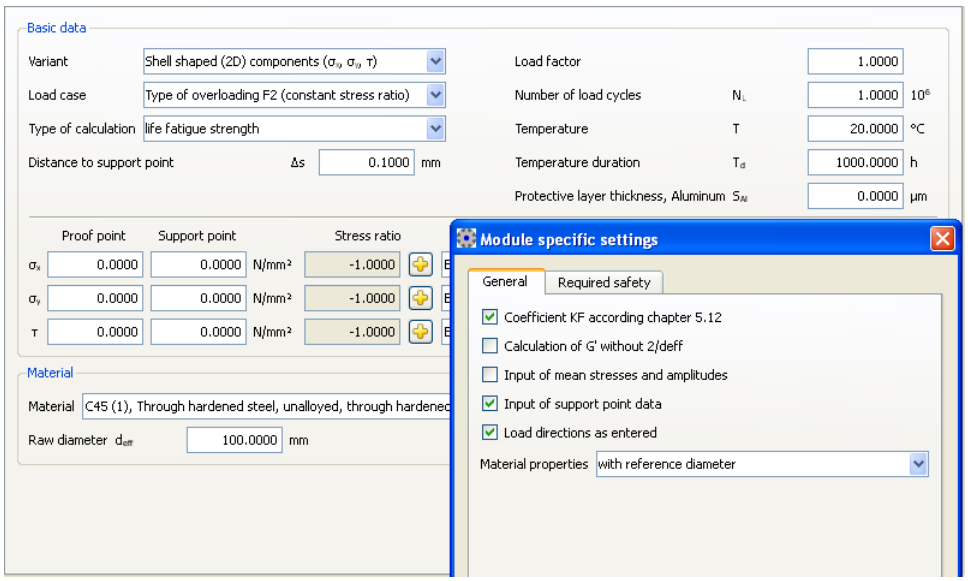


Figure 58.2: Inputting the stress values on the proof point and on the neighboring point. Inputting the neighboring point distance.

58.3.2 Load cases

In the endurance limit diagram, different assumptions are used to determine different levels for the maximum stress amplitude S_{AK}. Assumptions where sm=const. result in a larger S_{AK} than for R=const. This is because the limit lines in the Smith diagram rise by an angle < 45° (mean stress sensitivity). The most suitable assumption depends on the expected change in stresses in the part when it is subjected to permitted operational fatigue load. The overload case can therefore be a decisive factor in whether or not a part is overloaded [62].

Load case

- Type of overloading F1 (constant mean stress): at a constant mean stress, the stress amplitude increases as the decisive operating force increases.
- Type of overloading F2 (constant stress ratio): when the operating force increases, the ratio between the maximum stress and minimum stress remains the same. This overload case usually returns conservative results (compared to other overload cases) and should therefore be used in case of doubt.
- Type of overloading F3 (constant minimum stress): when the operating force increases, the minimum load remains the same.
- Type of overloading F4 (constant maximum stress): when the operating force increases, the maximum load remains the same.

58.3.3 S-N curve (Woehler lines)

Miner elementary, section 4.4.3.5.2 of the FKM Guideline

If a stress collective is present instead of individual stress, the calculation should usually be performed using the Miner elementary process.

Miner consistent, Section 4.4.3.5.2 of the FKM Guideline

The Miner consistent process (derived from Haibach, see [94]) takes into consideration the fact that the part endurance limit will reduce as damage increases. The reduction applies from $K_{D,}\sigma$=1*10e6.

58.3.4 Number of load cycles

Number of load cycles. If calculation according to Miner elementary is selected, then inputs greater than ND result in constant use.

58.3.5 Temperature

Inputting the temperature in degrees Celsius. The area of application of the FKM Guideline is limited according to material, see section 1.2.1.7. The temperature factor $K_{T,D}$ is defined on the basis of the temperature and the material type.

58.3.6 Temperature duration

time period during which the part is subjected to the temperature.

58.3.7 Protective layer thickness, aluminum, section 4.3.4, Figure 4.3.4

Protective layer factor K_S (which is defined via the protective layer thickness) takes into account the influence of a protective layer on the fatigue strength of a part made of aluminum.

58.3.8 Stress ratios

The mean stress is recorded in the R-value. In comparison to the mean stress-free case (cyclic loading, R=-1), the S-N curve (Woehler lines) is moved to higher sustainable stress amplitudes in the case of trials with mean compression stresses. In the case of trials with mean tensile stresses, the S-N curve (Woehler lines) is moved to lower sustainable stress amplitudes. The sustainable stress amplitude's dependency on the mean stress is material-specific, and is called the influence of the mean stress. This usually increases along with the tensile strength of the material.

Here R is defined from -1 up to +1

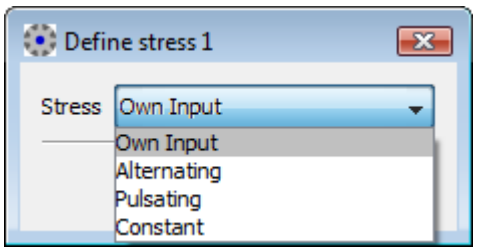


Figure 58.3: Inputting the specific R-value

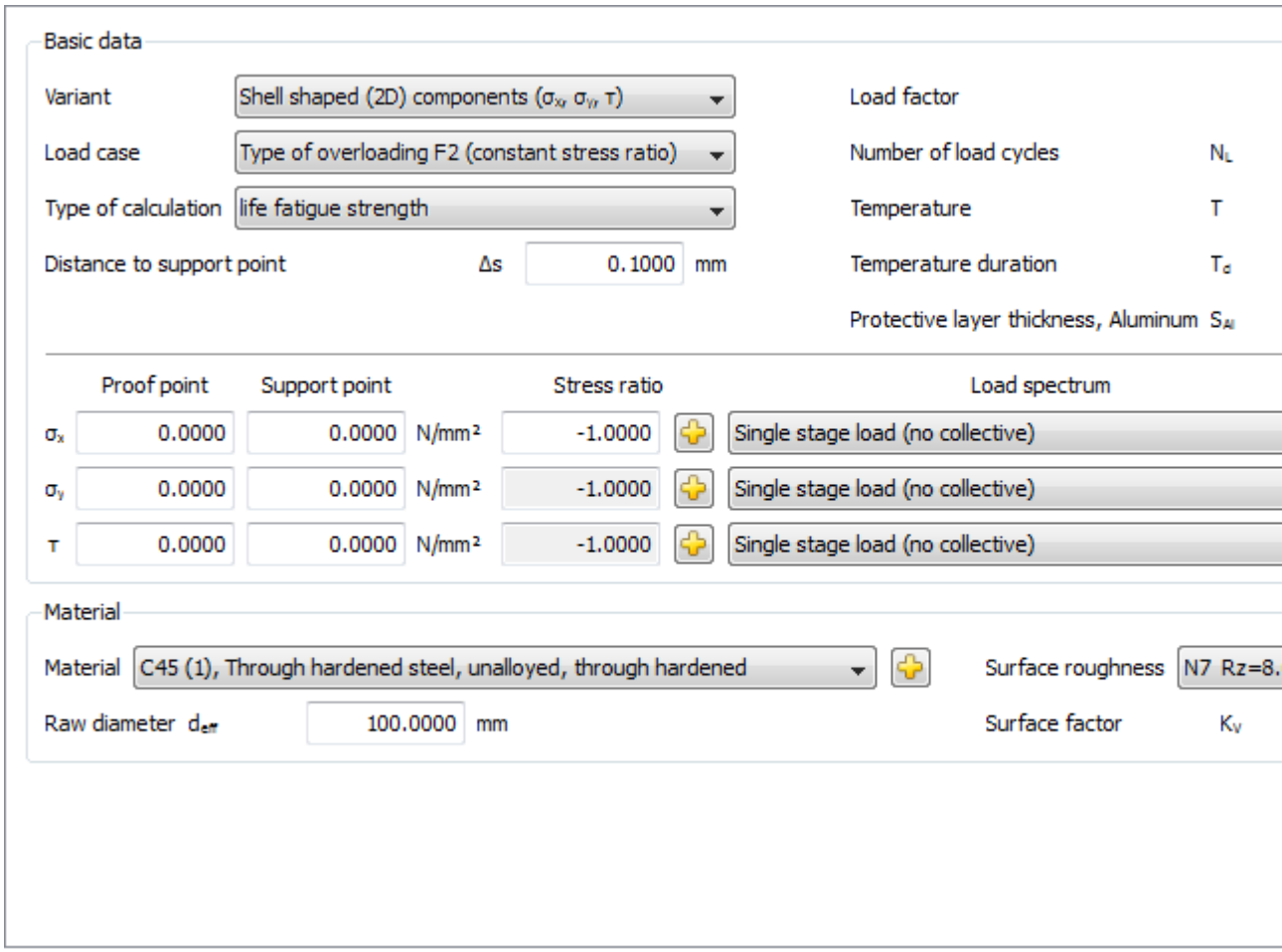


Figure 58.4: Inputting your own R-value.

As the surface roughness increases, the S-N curve (Woehler lines) moves to lower stress amplitudes, but the surface roughness alone is not the cause for this. The strength is much more affected by the detailed properties of the surface. In addition, despite similar surface characteristics and the same surface roughness, different processing procedures can cause different material internal stress states, resulting in S-N curves (Woehler lines) that differ from each other greatly.

58.3.9 Spectra

You can select existing load spectra directly.

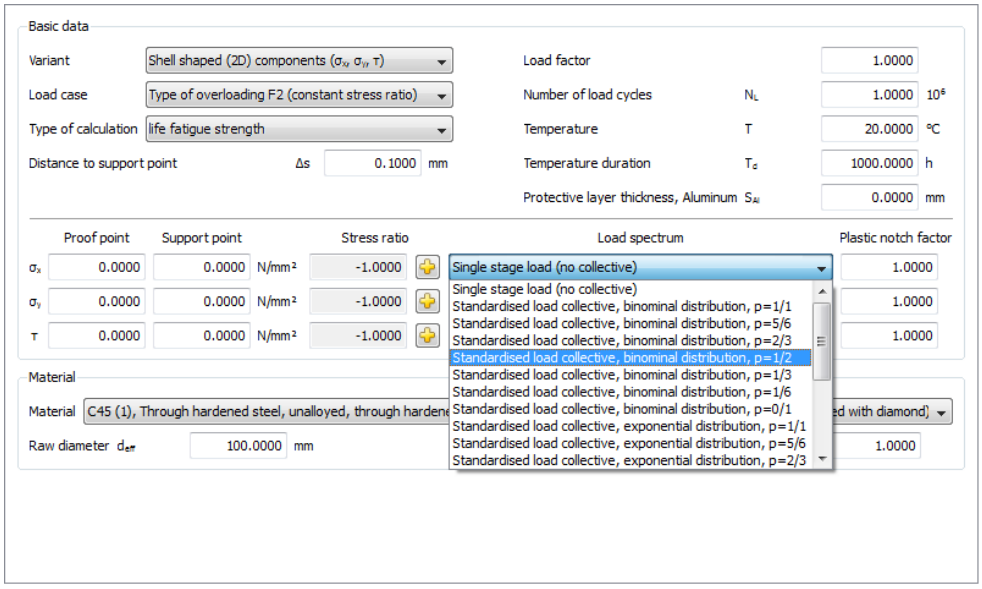


Figure 58.5: Selecting spectra

You can create a new load spectrum in the database tool (see section "Define load spectrum" on page II-308).

58.3.10 Surface factor KV, section 4.3.3, Table 4.3.7

Case factor K_V takes into account the influence of surface treatment on the fatigue strength.

58.4 Materials

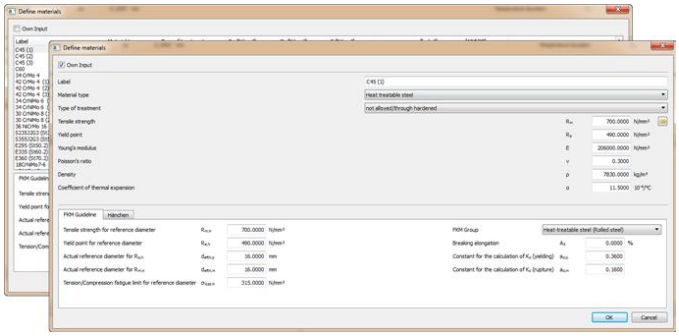


Figure 58.6: Materials screen: Strength verification using local stresses

The selection list contains materials from the FKM Guideline.
If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

58.4.1 Surface roughness

The roughness factor takes into account the influence of the surface roughness on the part's fatigue strength. Experiments are performed to derive it from the endurance limits of unnotched test rods with and without surface roughness, and shown in dependency of the material's total height Rz and tensile strength Rm. For polished surfaces it has the value 1.0. For rolled, forged and gray cast scale, the average roughness Rz=200µm applies. The average roughness can also be defined as your Own input.

58.5 Settings

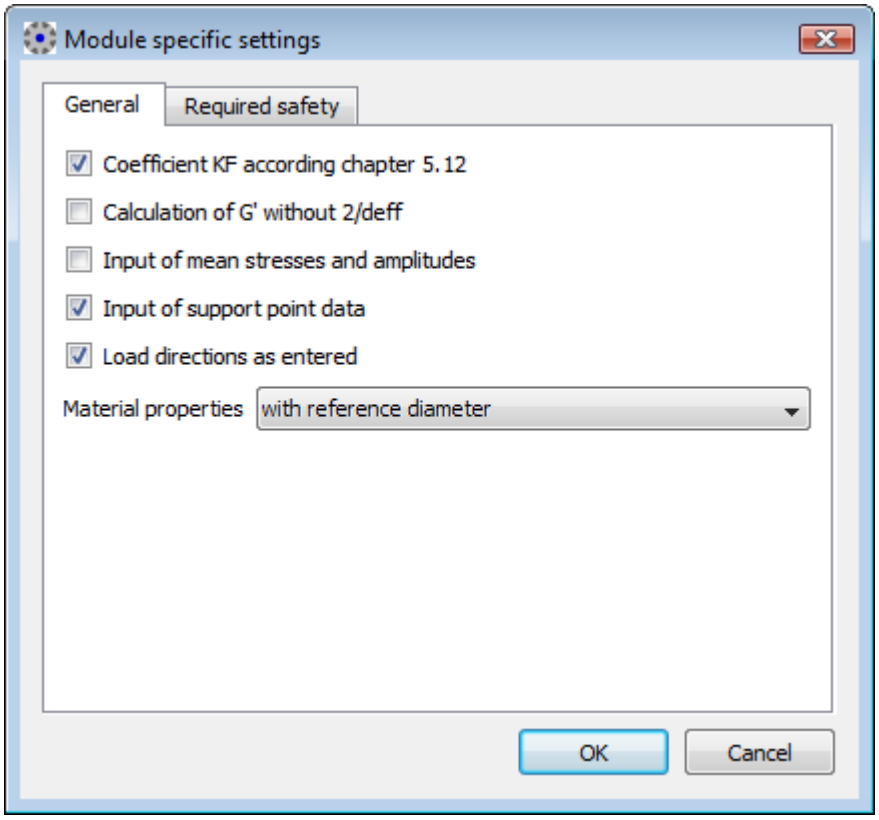


Figure 58.7: Settings

58.5.1 General settings

The references are to sections in the FKM guideline.

Coefficient KF according to equation 4.3.2 and 4.3.3, section 4.3.1.2

Notch effect coefficient as an estimated value to enable the effect of the roughness coefficient to be determined, according to the nominal stress concept, when the local stress concept is in use.

Flag set: Kf is defined according to formulae 4.3.2 and 4.3.3, described in section 4.3.1.2.

Flag not set: The KF coefficient is set as shown in Table 4.3.1.

Calculating G without 2/deff, section 4.3.1.3.3

If the flag is not set in General data, Neighboring point data entry, then an approximation of the related stress gradient is calculated, using the calculation based on the equations in 4.3.18. This contains terms for tension/compression, torsion, and bending. If no bending is present, it is questionable whether the second term (2/d) in the formulae makes any sense. The option programmed here is not provided in the FKM Guideline!

Flag set: the stress gradient is defined without applying the second term in formula 4.3.18.

Flag not set: the stress gradient is defined while also applying the second term in formula 4.3.18.

Input of mean stresses and amplitudes

If the flag is set, then the stresses are input in the main screen via the medium and amplitude stress.

Input of mean stresses and amplitudes

- **Material values at reference diameter:**
  values are taken from database (at reference diameter) and multiplied by K1
- **Rm, Rp depending on value from database, sigW at reference diameter:**
  Rm, Rp are read from the database according to size (excluding K1), and the fatigue strength is determined for the reference diameter entered in the database, and then it is multiplied by K1
- **Rm, Rp depending on value from database, sigW constant:**
  Fatigue strength not multiplied by K1, correct value must be in database
- **Rm, Rp depending on value from database, sigW calculated from Rm:**
  Fatigue strength is calculated from Rm. Rm is in database according to size, conversion according to FKM

Neighboring point data entry, section 4.3.1.3.3, Formula 4.3.17

Flag set: Notch sensitivity factor-related stress slope is defined in the neighboring point via the stress state. To do this, the stress values and the distance between the proof point and neighboring point must be entered in the main screen.

Flag not set: Notch sensitivity factor-related stress gradient is not determined from the values at a neighboring point. The related stress gradient at the point of maximum stress is estimated using formula 4.3.18. To do this, two radii (Radius 1 and Radius 2) must be defined (for the two directions on the surface), and also a typical part dimension d. See also: module-specific settings, Calculation of G without 2/deff, above.

Direction of load as specified, sections 4.1.0, 4.6.2

Flag set: the calculation is carried out for synchronous stresses.

Flag not set: the calculation is performed for asynchronous stresses (4.6.2.2). It can safely be assumed that this method of approach is a cautious one.

Use the mechanical material support coefficient, sections 4.3.1, 4.3.1.3.2

If this flag is selected, the mechanical material support coefficient is used for the calculation, otherwise the Stieler support coefficient is used. If sharp notches are present, the mechanical material support coefficient takes into account the strength reserves and contains the static size coefficient. The mechanical material support number (nwm) is made up of three parts: the static support number (nst), the mechanical deformation support number (nvm) and fracture mechanical support number (nbm). Assumption: nst = 1 is applied to the "Smooth shaft" and "Own input" notch types.

Selecting materials data, section 3.2.1

The part standard values Rm and Rp must be calculated from the semi-finished product or test piece standard values $R_{m,N}$ and $R_{p,N}$ or from the part drawing value $R_{m,Z}$. In exceptional situations, the part actual values $R_{m,I}$ and $R_{p,I}$ can be applied.
Refer to section 3.2.1.2.

Calculate the endurance limit for surface treated parts (section 5.5)

If this checkbox is enabled, and you have selected a rolled steel and a surface treatment method, the calculation method specified in the FKM Guideline, 6th Edition, section 5.5, is performed.

Surface factor

This selection is only used for calculating the endurance limit of surface treated parts.

Hardening depth

You can use this entry to define the location of the transition point between the hard surface layer and the core.

Determine core hardness from tensile strength Rm

If this checkbox is selected, the core hardness is estimated from the tensile strength instead of using the core hardness entered when the material was specified.

58.5.2 Required safeties

The FKM Guideline is one of the few calculation guidelines that lists the required safeties according to the consequences of failure etc. In combination with safe load assumptions and an average probability of survival of the strength variables Pü=97.5%, they apply for both welded and non welded parts. Safety factor are defined on the basis of the selected material and the defined consequences of failure, probability of occurrence of the load, and also inspection and test. It differentiates between steel, cast iron (ductile or non ductile), and also aluminum (ductile or non ductile), i.e. five different classes. Alternatively, you can also set the safety factors manually.

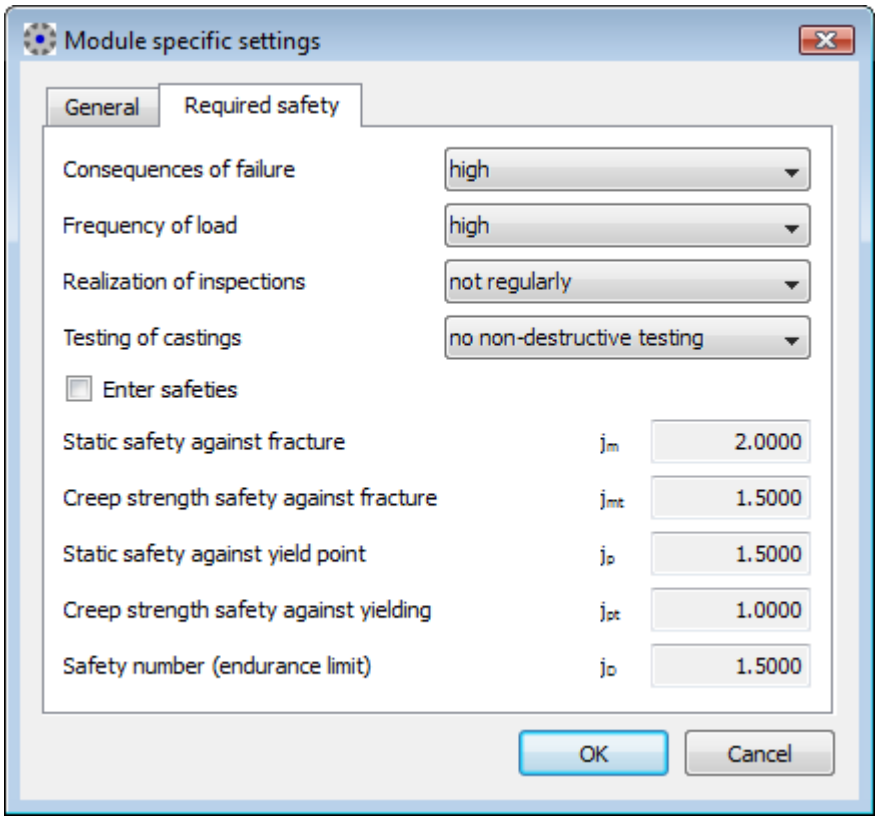


Figure 58.8: Selecting the safeties according to material and load properties

jmt	Safety margin against creep strength depending on time
jp	Safety margin against yield point
jpt	Safety margin against creep limit
jF*jG	Safety against the endurance limit

58.6 Estimation of the fatigue strength for surface treated parts (section 5.5)

This calculation should only be used for surface treated rolled steel. The surface treatments include the following treatment methods applied to the materials:

- case-hardened
- nitrided, gas-nitrided, nitrocarburized
- induction hardened
- rolling
- shot peening

You can define these treatment types either when you specify the material or when you input the surface factor (Settings).

This process is based on the concept of a local endurance limit. Two points on the component are observed. The first point is on the component's surface, and the second point is at the transition point between the hard surface layer and the core. This is calculated with the greatest principal stress $\sigma 1$. If no principal stresses have been entered, they are first calculated.

You can also input a hardening depth for this calculation in the Settings. The hardening depth is then used to calculate the distance from the component's surface to the transition point between the hard surface layer and the core.

The constant Kf can either be calculated in accordance with formulae 4.3.2 and 4.3.3 or taken from Table 4.3.1. You also have the option of inputting the core hardness in the Settings when you specify the material. Alternatively, this can also be estimated from the tensile strength.

This approach is used to calculate the internal stress, which is included in the calculation of mean stress sensitivity. In this case, the rate of utilization for the point on the component's surface is calculated first, followed by the rate of utilization at the transition point between the hard surface layer and the core. The greater of the two rates of utilization is then used for the proof. Both rates of utilization should be < 1.

The results are only displayed in the report if this calculation method has been selected for rolled steel with the predefined treatment types.

Chapter 59
Hertzian Pressure

Use this module to calculate the contact pressure between two bodies. In the case of a load on a rolling pair that is applied vertically to the contact surface, elliptical flattening occurs for point contact, and rectangular flattening occurs in the case of linear contact. Apart from the case of general contact (see next page).

The theory presented in the reference document [107 (see section "Bibliography" on page X-1214)] is used for the case of general contact. Based on this reference, we use a grid of triangular pressure distribution elements, to calculate the line contact between bodies with an arbitrary profile, under the application of a given vertical line load. The bodies are assumed to extend infinitely vertical to their profile. The profiles of the two bodies are given with the use of two data files, containing three columns with the index, the position and the value of the profile.

```
:TABLE LIST Profile
  INPUT index TREAT NEXT_BIGGER
  OUTPUT Position, ProfileValue

DATA
1       -5      0.635083269
2       -4.95   0.622242132
3       -4.9    0.609538427
4       -4.85   0.596971886
5       -4.8    0.584542241
6       -4.75   0.57224923
7       -4.7    0.560092593
8       -4.65   0.548072075
9       -4.6    0.536187424
10      -4.55   0.524438391
11      -4.5    0.51282473
12      -4.45   0.5013462
13      -4.4    0.490002563
14      -4.35   0.478793582
15      -4.3    0.467719027
16      -4.25   0.456778669
17      -4.2    0.445972282
END
```

Figure 59.1: Format of the data file of the body profile

All values in this file should be given in millimetres.

Each body’s profile file can have different discretization and position values. It is advised though to use the same discretization and position for both bodies. The accuracy of the results will depend on the density of the profile points

(discretization length). Edge contact effects are also considered. The contacting surfaces are assumed frictionless. With the use of this general contact option, the user can also create a 2D plot of the subsurface stresses and a plot of the contact pressure along the profiles. You will find more information about the calculations and equations used in the reference document mentioned above.

The Hertzian equations are used to help calculate the maximum pressure (Hertzian pressure) and also the proximity of the two bodies (ball, cylinder, ellipsoid, plane; convex or concave). The calculation formulae have been taken from "Advanced Mechanics of Materials, 6th Edition [78]. The underlying principle for calculation for point contact is that the diameter of the bodies is defined on two principal planes, from which an equivalent diameter is then defined. In the case of linear contact, the calculation is performed in one main plane, so there is only one equivalent diameter. In addition, the location and value of the maximum principal shear stress in the interior of the body are determined.

An approximation of the cylinder/cylinder configuration has been calculated using the dissertation from Weber/Banaschek [69]. Formula (55) from Norden's book [89] is used to calculate the approximation of the cylinder area.

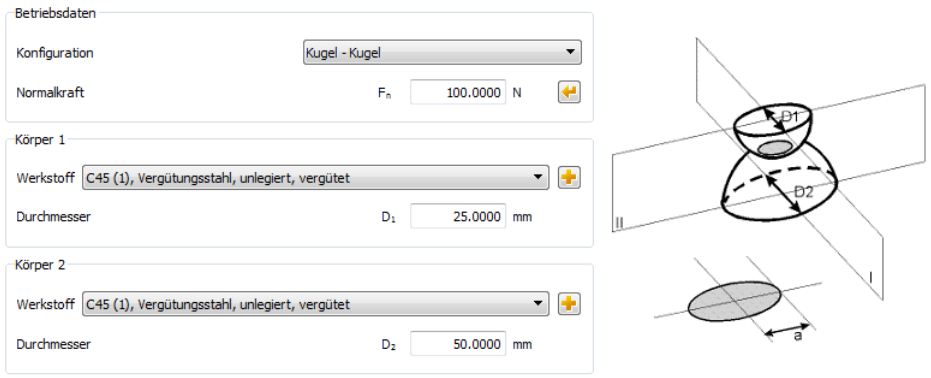


Figure 59.2: Main window for Hertzian pressure

The main window for Hertzian pressure (see Figure 59.1) is where you define the normal force, the configuration, and the diameter (in addition to the supporting length for linear contact), and the materials used in the body.
You can select one of these configurations:

- Ball - ball
- Ball - cylinder
- Ball - ellipsoid

- Ball - plane
- Ellipsoid - ellipsoid
- Ellipsoid - cylinder
- Ellipsoid - plane
- Cylinder - cylinder
- Cylinder - plane
- General contact

On the right, in the main screen, an image of the current configuration is displayed to help you input the values more easily.
For normal force there is also a sizing option. If you click the sizing buttons next to the normal force, you can enter the required Hertzian pressure, and the system will then calculate the normal force from it.
If the bearing area has a concave bend, you must enter the diameter as a negative value. Negative diameters are only possible in the case of Body 2.

59.1 Settings

Use the `Depth display factor` to define the depth display in the graphic. The depth of the point τmax is multiplied by this coefficient. The resulting depth is then displayed in the graphic. The default setting of this coefficient is 6.

Chapter 60
Hardness Conversion

You access the Hardness conversion module in the `Extras > Hardness conversion` menu. The hardness conversion is also present in the materials screens as a sizing function, where, for example, the tensile strength can be defined by means of a hardness value.

This module contains the hardness conversion calculation as specified in DIN EN ISO 18265, Edition 2-2014. During the conversion, select `Extras > Hardness conversion` to display a selection list in which you can select the required material. The other conversions (for the materials) use the table for unalloyed and low-alloy steels and steel casting. As required in each particular case, the stored tables can be used to convert the value of the tensile strength into Vickers, Brinell or Rockwell hardness, and vice versa. Due to possible variations, the received values should only be used if the default testing process cannot be applied. The interim values of the value conversion table will be interpolated from the neighboring values.

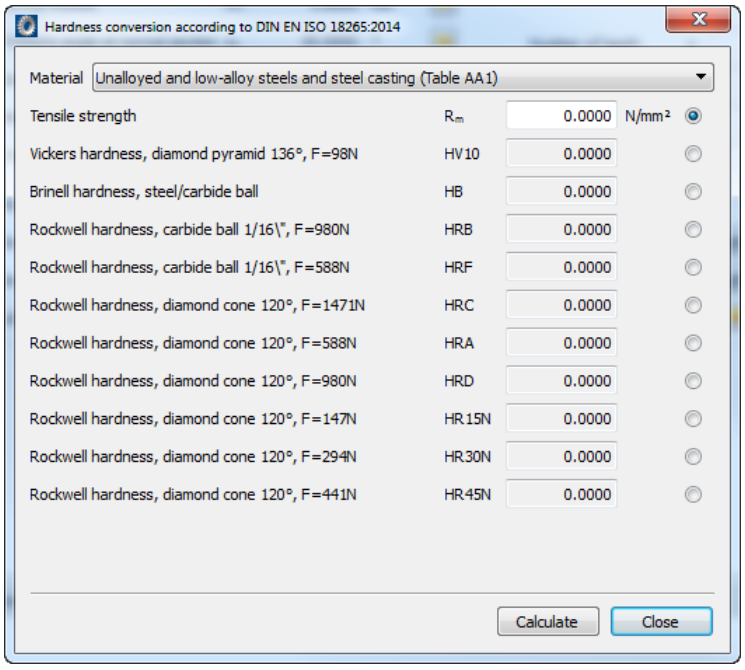


Figure 60.1: Hardness conversion input screen

Integrated conversions of the steels and steel groups according to DIN EN ISO 18625:

- Unalloyed and low-alloy steels and steel casting (Table AA1)
- Heat treatable steel in heat treated state (Table B.2)
- Heat treatable steel in untreated, soft-annealed or normally annealed state (Table B.3)
- Heat treatable steel in hardened state (Table B.4)
- Cold-working steels (Table C.2)
- High-speed steels (X80WMo6.5, X82WMo6.5, X90WMo6.5, X97WMo3.3, X100WMo6.5, X85WMoCo6.5.5, X105WMoCo6.5.5 and X79WCo18.5) (Table D.2)

The range of validity for unalloyed and low-alloy steels and steel casting (with the conversion in the material screens applied) is limited as follows:

- Tensile strength Rm: 255 to 2180 N/mm2
- Vickers hardness HV: 80 to 940 HV
- Brinell hardness HB: 76 to 618 HB
- Rockwell hardness HRB: 41 to 105 HRB
- Rockwell hardness HRF: 82.6 to 115.1 HRF
- Rockwell hardness HRC: 20.3 to 68 HRC
- Rockwell hardness HRA: 60.7 to 85.6 HRA
- Rockwell hardness HRD: 40.3 to 76.9 HRD
- Rockwell hardness HR 15N: 69.6 to 93.2 HR 15N
- Rockwell hardness HR 30N: 41.7 to 84.4 HR 30N

Rockwell hardness HR 45N: 19.9 to 75.4 HR 45N

Chapter 61
Linear Drive Train

Use this calculation module to calculate drive screws. Drive screws are used to convert rotational movement into longitudinal movement or to generate great forces.

Although trapezoidal screws are almost exclusively used as drive screws, some rough operations also use buttress threads.

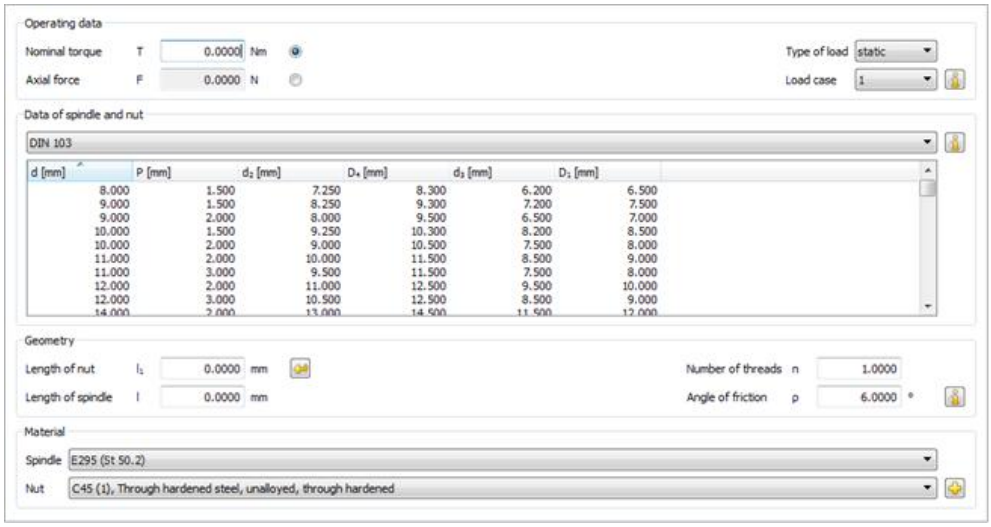


Figure 61.1: Linear drive train basic data

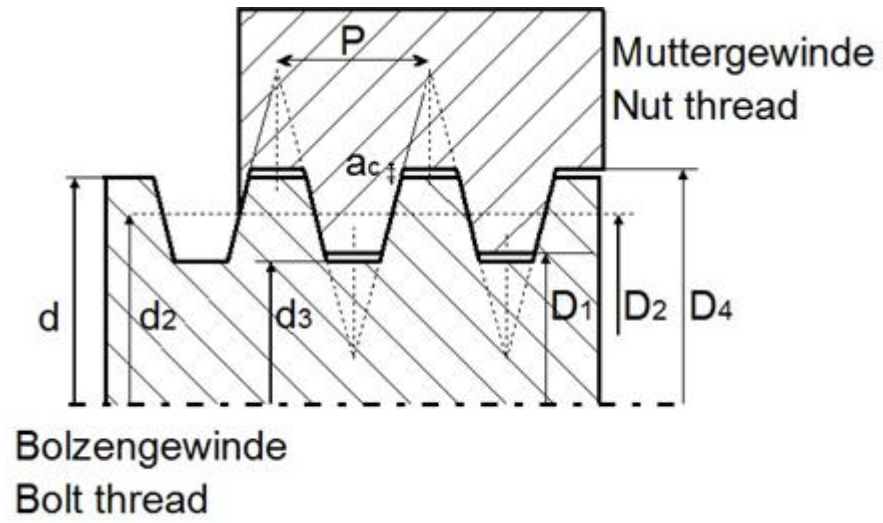


Figure 61.2: Dimensions of trapezoidal screws

Two different linear drive train configurations can be calculated:

- `Load case 1`
  Stress on the spindle in a spindle press
- `Load case 2`
  Stress on the spindle in a gate valve

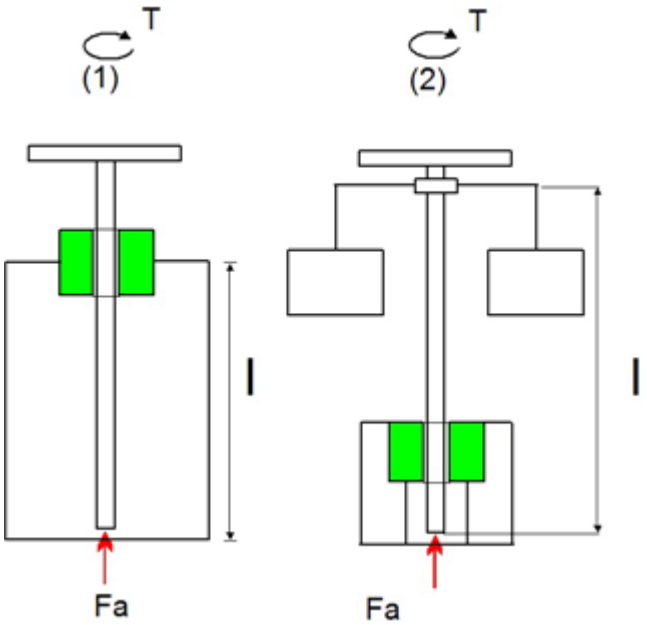


Figure 61.2: Linear drive train stress cases

The information provided in Roloff Matek [62] is used to calculate linear drives (drive screws).

61.1 Calculation

Short and long linear drives subjected to pressure are handled separately in the calculation process.

Short pressure stressed drive screws

Short pressure stressed drive screws are not at risk of buckling and therefore are not tested for this.

The required cross section of the thread can therefore be defined using the formula:

$$A_3 \geq \frac{F}{\sigma_{d(z)zul}}$$

σd(z)zul: under static load: Rp/1.5; under pulsating load σ zdSch/2.0; under alternating load: σ zdW/2.0;

Long pressure stressed drive screws

The formula for calculating the necessary core diameter of the thread is taken from the Euler equation:

$$d_3 = \sqrt[4]{\frac{64 \cdot F \cdot S \cdot lk^2}{\pi^3 \cdot E}}$$

S: Safety (S≈6 to 8)

lk: mathematical buckling length, lk ≈ 0.7*l (Euler buckling case 3 used for general, guided spindles)

Calculation of the strength:

Load case 1:

The upper part of this configuration is subject to torsion and the lower part is subject to compression and therefore buckling.

Torsional stress:

$$\tau_t = \frac{T}{Wp} \leq \tau_{tzul}$$

Wp: polar moment of resistance Wp≈0.2*d3^3

τtzul: permissible torsional stress; static load τtF/1.5; pulsating load τtsch/2.0; alternating load τtW/2.0;

Compressive (tensile) stress:

$$\sigma_{d(z)} = \frac{F}{A_3} \leq \sigma_{d(z)zul}$$

A3: Thread minor diameter cross section

σd(z)zul: permissible compressive (tensile) stress

Load case 2:

The upper part of this configuration is subject to torsion and the lower part is subject to compression, infrequent tension and torque.

Formula for the part to be checked:

$$\sigma_v = \sqrt{\sigma_{d(z)}^2 + 3 \cdot (1 \cdot \tau_t)^2} \leq \sigma_{d(z)zul}$$

The required torque corresponds to the thread torque, if not subject to any moments of friction.

$$T = F \cdot d_2 / 2 \cdot \tan(\varphi \pm \rho')$$

d2: Flank diameter of the thread

φ: Lead angle of the thread (for single thread trapezoidal screws φ≈3°...5.5°)

ρ': Thread friction angle

Material		Lubricant	Friction angle ρ' [°]
Screw	Nut		
Steel	Cast iron	Dry	12
Steel	CuZn – CuSn alloys	Dry	10
Steel	Special plastics	Dry	6
Steel	Cast iron	Lubricated	6
Steel	CuZn – CuSn alloys	Lubricated	6
Steel	Special plastics	Lubricated	2.5

Figure 61.3: Values for the friction angle

The + in the formula stands for "tightening the spindle", and - stands for "loosening the spindle". The KISSsoft procedure calculates both situations and outputs the results in a report.

Calculation for buckling (only for long spindles):

First of all, calculate the slenderness ratio.

$$\lambda = \frac{lk}{i} = \frac{lk}{\sqrt{I / A_3}} = \frac{lk}{\sqrt{\frac{\pi \cdot d_3{}^4 \cdot 4}{64 \cdot d_3{}^2 \cdot \pi}}} = \frac{lk \cdot 4}{d_3}$$

λ: Slenderness ratio of the spindle

lk: mathematical buckling length

i: Gyration radius

Only 3 different materials can be used for the spindle so that the slenderness ratio can be defined correctly.

Elastic buckling is present if λ>=λ0 = 105 for S235; λ>=89 for E295 and E335.

$$\sigma_K = \frac{E \cdot \pi^2}{\lambda^2}$$

The non-elastic area as defined by Tetmajer and l λ<105 for S235.

$$\sigma_K = 310 - 1.14 \cdot \lambda$$

For λ<89 and for E295 and E335:

$$\sigma_K = 335 - 0.62 \cdot \lambda$$

For a non-elastic case, the Johnson parabola equation can also be used for the calculation. (also for other materials)

$$\sigma_K = \sigma_{dS} - (\sigma_{dS} - \sigma_{dP}) \cdot \left(\frac{\lambda}{\lambda 0} \right)^2$$

The safety can then be calculated as follows:

$$S = \frac{\sigma_K}{\sigma_{vorh}} \geq S_{erf}$$

The required safety for elastic buckling is Serf≈3 to 6. For non-elastic buckling, it is Serf≈4 to 2.

Buckling no longer needs to be calculated for a slenderness ratio < 20.

Analysis of the nut:

The surface pressure of the nut is calculated from the nut length:

$$p = \frac{F \cdot P}{l_1 \cdot d_2 \cdot \pi \cdot H_1} \leq p_{zul}$$

P: Pitch of thread

l1: Length of the nut thread

d2: Flank diameter of the thread

H1: Flank engagement of the thread

pzul: permissible surface pressure

Due to the uneven distribution of surface pressure, the nut length should be no greater than 2.5*d. During sizing, the length is limited to 2.5* d even if a longer one is input.

Efficiency and self-locking:

The efficiency of the conversion from rotational movement into longitudinal movement:

$$\eta \approx \frac{\tan \varphi}{\tan(\varphi + \rho')}$$

The conversion of movement is only possible for non-self-locking threads, because the limiting value in this case is that, if φ=ρ', the efficiency is 0.5.

If φ>ρ', the thread is no longer self-locking.

Each permissible value is taken from the Roloff Matek tables.

61.2 Sizings

This calculation module can calculate the core diameter d3 of a long spindle that is subject to pressure, when you select "Own Input".

In addition, it can also define the nut length on the basis of permissible surface pressure and the required safety.

61.3 Settings

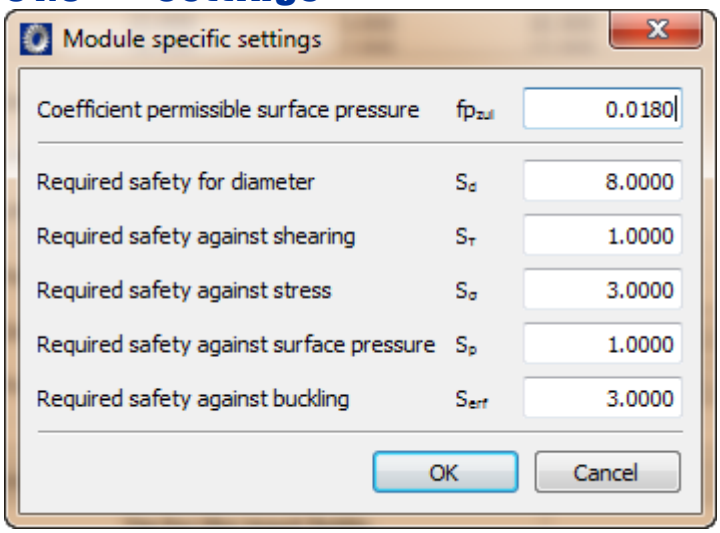


Figure 61.4: Settings input window

- Coefficient of permissible surface pressure; this factor is used to define the ratio to Rm, in other words pzul = fpzul*Rm
- Required safeties for diameter, shearing, stress, surface pressure and buckling: for the calculation and the sizings

61.4 Materials

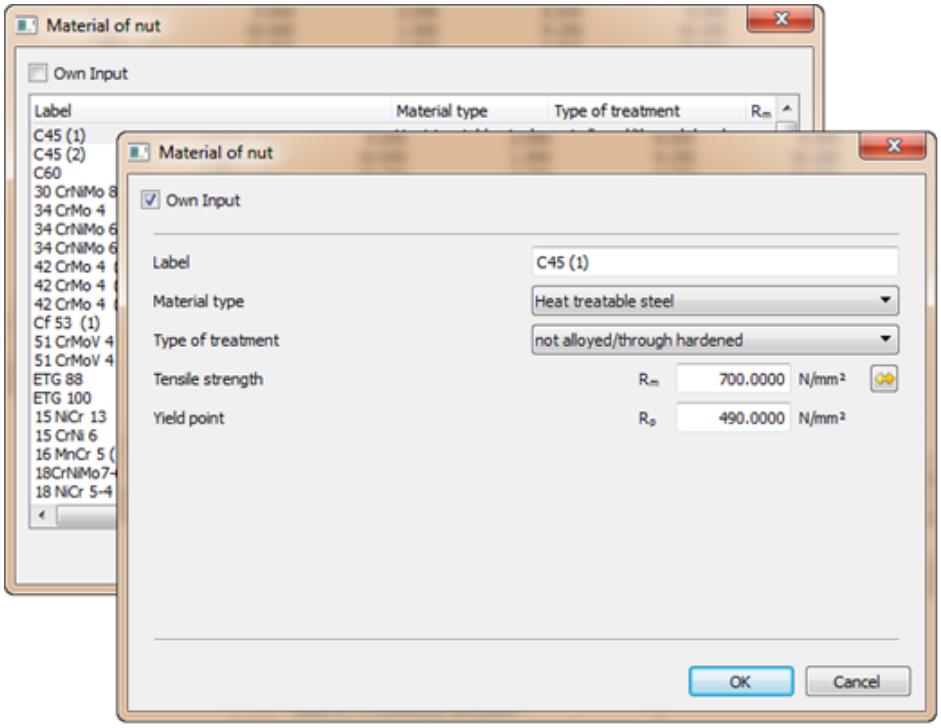


Figure 61.5: Nut materials input window.

In the selection list, you can select materials in accordance with the standard. If you have set the `Own input` flag, the system displays a new dialog here. In it, you can define the material data used in the calculation to suit your own purposes. You can also define your own materials directly in the database (see page I-128), so that these can also be used in subsequent calculations.

You can only select these different materials for nuts. For the spindle material you can choose E295 (St 50.2), E335 (St 60.2) and S235 (37.3) materials, because the calculation of buckling is only designed for use with these materials.

The strength values for the 3 materials have been fixed:

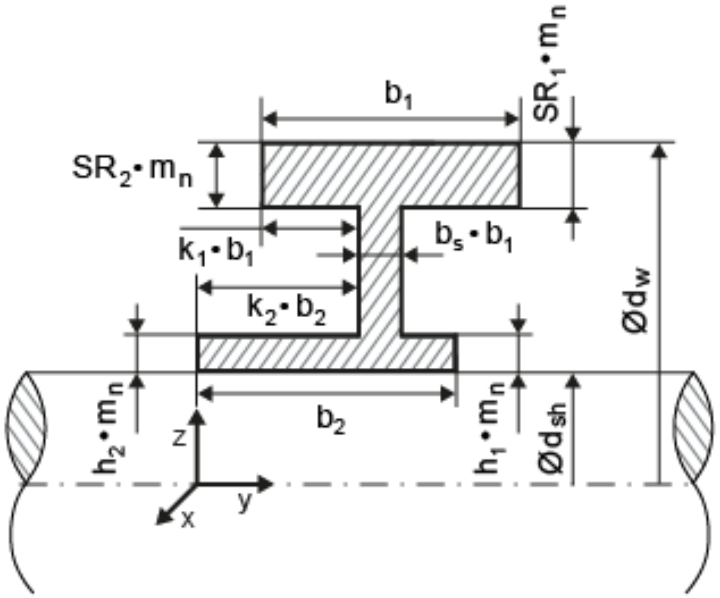
- E295 (St 50.2): Rp02 = 295 N/mm^2; σzdSch = 295 N/mm^2; σzdW = 195 N/mm^2; λ0 = 89; τtSch = 205 N/mm^2; τtW = 145 N/mm^2
- E335 (St 60.2): Rp02 = 335 N/mm^2; σzdSch = 335 N/mm^2; σzdW = 235 N/mm^2; λ0 = 89; τtSch = 230 N/mm^2; τtW = 180 N/mm^2
- S235 (St 37.3): Rp02 = 235 N/mm^2; σzdSch = 225 N/mm^2; σzdW = 140 N/mm^2; λ0 = 105; τtSch = 160 N/mm^2; τtW = 105 N/mm^2

Chapter 62
Deformation of the Gear Body

This model is designed to calculate the deformation of a gear body using the finite elements method (FEM).

62.1 Calculation procedure

The first step of the calculation involves importing the required data, which is listed in the following figure:



The most important values are:

Facewidth (b_1)
Hub width (b_2)
Operating pitch diameter (d_w)
Shaft diameter (d_{sh})
Normal module (m_n)
Rim thickness coefficient (SR_1)
Rim thickness coefficient (SR_2)
Hub width coefficient (h_1)
Hub width coefficient (h_2)
Web thickness coefficient (b_s)
Cavity factor (k_1)
Cavity factor (k_2)

Please note that this input can be given either as factors or as dimension. Internal gears can be also handled. FEM mesh density can be also input by the user.
Some of this data can also be taken directly from a specified gear pair calculation file.
Once you have set the geometry data, you can select the hub material from the database.

Once you have set the geometry data, you can select the hub material from the da-

tabase. Then, you must define the applied load. To do this, apply line load to the gear body (in a specified load direction). Another option is to calculate the stiffness matrix of the gear body, and then save this data in a file. This file can then be imported directly into the shaft calculation file, to define the stiffness matrix of the gear body and also to take this into account in the shaft calculations.

The FEM Solver KISSsoft calls in the background is the Code_Aster open source solver, which you can find by going to this website: http://www.code-aster.org/. The pre-processor used to generate the FE model is also an open source program, called Salome, which can be found at http://www.salome-platform.org/. To ensure you have the correct versions, install both programs from the KISSsoft DVD, or download them from the KISSsoft website. The only precondition for using this method is that Java is installed (it can be downloaded from https://www.java.com) and you have its bin path set correctly in KISSsoft in "`Extras->Settings->Directories-` JAVADIR", where the java.exe can be found. Remember that the folders for these FEM programs (FEPreProcessor and FESolver) can be copied to any location. In KISSsoft, they should be defined in `Extras->Settings->Directories`->FEMDIR (this is usually the KISSsoft installation folder). Do not use path names that contain accented characters such as German umlauts or other special characters.

Both the solver and the pre-processor are available with a GPL license, as are the versions on the websites mentioned above. (You will find more details about this license in the installation directories for these programs in KISSsoft, and on the corresponding web sites.)

Note that you will be prompted to give the pre-processor network access through the Windows firewall the first time you start it. However, you can simply click on "Cancel" and ignore this message.

62.2 Results

Once you have defined a single line load, the maximum deformation of the gear body and the coordinates of the point of maximum deformation are displayed. Select the `Gear body deformation->Graphics` menu option to also display the deformation of the line to which the line load was applied. If necessary, you can then start the FE program yourself. To do this, select the `Calculation->Open Salome` menu option once the calculation has finished running. You can then either open the "GearBody.unv" file to view the FEM meshing model, or open the "Aster.0.med" file to display the results. To view the mesh in Salome, select the Mesh module from the drop-down list in the Salome toolbar and then select "File->>>Import->>UNV file". To view the results, select the ParaViS module in Salome and then open the "med" file mentioned above. You will find more detailed information about working with meshes and the results files in Salome in a special set of KISSsoft instructions. You can request this documentation from the Hotline.

Chapter 63
Plastics Manager

You use this module to add new plastic materials to the KISSsoft material database, if the material characteristics are available. When you add a material to the database, KISSsoft generates its associated .DAT material file.

Currently, KISSsoft has over 45 plastic materials in its database. However, with a large number of different plastic materials available for gear calculation, it is often necessary to enter own materials into the KISSsoft database. To add a new material into KISSsoft database, at least the following material properties must be known:

- Poisson's ratio
- Coefficient of friction for oil, grease and dry run
- Temperature-dependent Young's modulus

If other material properties (tensile strength and yield point, temperature-dependent wear factors, thermal expansion coefficient, permissible root/flank stresses, etc.) are also known, they can also be added to the KISSsoft database. The more gear test results are available, the more accurately plastic gear calculations can be performed.

If gear test results are available, the permissible root or flank stresses can also be calculated, and the results entered in a .DAT file for the material. This data can then be used to calculate safety factors over the entire service life of the gears.

In the `Basic data` tab, you can input general material characteristics (density, material type, absorption of water etc.) and also tribological characteristics (coefficient of friction and wear coefficient). Specific heat capacity and conductivity are used purely for information purposes in the material database.

Tribological properties can be individually entered for oil, grease and dry runs. If the data is available, you can enter temperature-dependent wear factors for all lubrication conditions in the corresponding temperature-dependent wear factor table.

If the temperature dependence of wear factor is not known, KISSsoft assumes the same temperature range for wear factor as was used for the Young’s modulus data. Additional comments are also displayed in a corresponding .DAT file.

You can also input temperature-dependent material data (Young's modulus, tensile strength and yield point) in the `Test data` tab.

If results from gear testing are available, it is possible to calculate permissible fatigue strength for root and/or flank. You input this test data in the "Test gear allowances" table in the `Test data` tab.

63.1 Gear test results

2 different types of test results can be used to calculate the permissible root and/or flank strength. The first type of results (Case I) is measured on an identical cylindrical gear pair (.Z12). The second type of results (Case II) can be measured on different cylindrical gear geometries (2 gears (.Z12), 3 gears (.Z15), 4 gears (.Z16) and planetary gears (.Z14)) under different lubrication conditions. It is not possible to enter and calculate both sets of data at the same time. To switch between the 2 cases, click on `Module specific settings` -> `Identical test gears`.

63.1.1 Case I. Test results measured on identical test gears

Such test results are usually measured on test rigs. The geometry of the test gears must be defined in a separate KISSsoft .Z12 file, which can be selected on the user interface (`Test data` tab). In this case, only 1 lubrication regime is possible, and this must be defined in the test file.

Before entering data in the test table, a file for the test gear must be created and then selected. Based on the settings of the test file, the user interface is adjusted. The material properties of the failed gear in the test file are not important, because the material properties are overwritten with user-defined values set in the `Basic data` and `Test data` tabs. The selected material of the failed gear must be plastic (for instance select PA66 (VDI 2736)). However, the correct material properties must be set for the counter material.

If the test file is selected, the failed gear must also be selected. If both materials from the selected test file are plastics, then the user can select between gears 1 and 2. In other cases, failed gear is selected automatically. The failure can only happen on gear 1 or on gear 2, not on both gears simultaneously.

The following limitations apply to the test files:

- Calculation method for plastic gears must be selected
- The `Calculate temperatures root/flank` option must be disabled (for grease and dry running)
- Calculation without load spectrum
- At least 1 gear must be made of plastic material
- The `Rating` tab must be visible

To calculate the permissible root and/or flank strength, the following test results must be available:

- Torque, speed and cycles to failure of the **failed** gear. According to the VDI 2736, all the tests should run until failure
- Ambient, root, flank temperature (depending on the failure mode) or oil temperature (if oil lubricated conditions) of the failed gear
- If the counter gear is plastic and the failure is set to `flank`, also the flank temperature of the counter gear must also be set
- Type of failure mode (can be selected as no failure, root, flank or wear failure)

The test gear measurements table can also be read in from a file, however an appropriate file structure must be used (please see example file K17_testResults.DAT).

Additional settings, that control the calculation of permissible root and/or flank strength are available in the tab `Test data` and in `Module specific settings`. Additional explanation of the individual parameters is given in chapters 1.2 and 1.3.

63.1.2 Case II. Test results available from gears with different gear geometry

To activate this option, uncheck the "Simple test file" option in `Module specific settings`. Depending on the selection, the necessary input fields in the test data table are automatically adjusted.

In the "Test gear measurements table",failed gear, failure mode and counter gear must be selected for every test file. Gear numbers for .Z12 file are [1,2], for Z15 [1,2,3], for Z16 [1,2,3,4] and for Z14 [1=sun, 2=planets, 3=internal gear]. Torque, speed cycles to failure and temperatures must be set in the individual test gear files.

Additional settings, that control the calculation of permissible root and/or flank strength are available in tab `Test data` and in `Module specific settings`. Additional explanation of the individual parameters is given in chapters 1.2 and 1.3.

63.2 Additional settings in the "Test data" tab

Damage probability: The calculated average number of cycles to failure (50 % probability of damage) are recalculated to a selected damage probability according to the selected statistical method.

Merge temperature deviation $\Delta\theta_{merge}$: Acc. to the VDI 2736-4 [109], at least 3 tests should be performed for every test condition (torque and speed). In theory, gears running at the same test conditions should have identical root and flank temperatures. However, in reality, this is often not the case. Use $\Delta\theta_{merge}$ to take these temperature deviations into account.

	Moment	Speed	Root temperature	NL
1	1.00 Nm	750 rpm	100	$1.01 \cdot 10^6$
2	1.05 Nm	750 min	105	$1.13 \cdot 10^6$
3	1.00 Nm	750 rpm	120	$0.82 \cdot 10^6$
4	1.00 Nm	750 rpm	98	$1.05 \cdot 10^6$

Table 1 63.1: Merging test results

If $\Delta\theta_{merge}$= 8°C, tests 1, 2 and 4 from Table 1 will be merged together and used for further evaluation. Test 3 will be omitted.

Merge deviation: If deviations of torque (or stress) are smaller than the set merge deviation (in %), then these measurements and taken for further evaluation. Example from Table 1: if merge deviation is 5%, then tests 1, 2 and 4 are merged together. If merge deviation is smaller, then just tests 1 and 4 are merged together (test 3 is not taken into consideration because of $\Delta\theta_{merge}$ condition).

Group temperature deviation $\Delta\theta_{group}$: After merging, the results must also be grouped according to the root or flank temperature to be able to write them to a material .DAT file. Use $\Delta\theta_{group}$ to group together results even though the measured temperatures are not identical. If $\Delta\theta_{group}$=5°C, tests 1, 2 and 3 in Table 2 are grouped and recorded with the same temperature of 20 °C (the lowest temperature is always used as the reference temperature).

	Stress	Root temperature	NL	DATA 20 0.000e+000 100.0 1.010e+006 100.0
1	100 MPa	20°C	$1.01 \cdot 10^6$	

2	80 MPa	22°C	$2.13 \cdot 10^6$	
3	60 MPa	25°C	$3.82 \cdot 10^6$	
4	40 MPa	98°C	$5.14 \cdot 10^5$	
5	30 MPa	102°C	$9.02 \cdot 10^5$	

Table 2 63.2: Grouping results in a .DAT file

The following **statistical methods** are available for calculation of cycles to failure (calculation is based on the user defined damage probability):

- Statistical method according to VDI 2736-4
- Normal distribution
- Weibull distribution (2 parameter)

Statistical method according to VDI 2736-4
The process detailed in VDI 2736-4 [109 (see section "Bibliography" on page X-1214)] for the statistical calculation of the cycles (load cycles) to failure is applied. The only difference is that a default damage probability of 10% is used in the VDI standard. In KISSsoft, it is possible to calculate cycles to failure with a user defined damage probability.

Normal distribution
A normal (Gaussian) distribution can be used to calculate the number of cycles to failure. You will find more detailed information about this method in the technical literature [110 (see section "Bibliography" on page X-1214)].

Weibull distribution (2 parameters)
A 2-parameter Weibull distribution can be used to calculate the number of cycles to failure. You will find more detailed information about the method applied here in [111]. To calculate Weibull parameters, relative least square method is used.

63.3 Settings in the Module specific settings

Min. number of data points for merging:
You can limit the minimum number of tests needed for merging. The minimum possible number is 2, however, VDI 2736 specifies that at least 3 measurements are necessary.

Min. number of data points for grouping
Use this setting to specify the minimum number of merged points needed to group results in a .DAT file. The minimum possible number is 2.

Allowable cycles to failure deviation:
However, at least 3 tests should be performed for each test condition (torque and temperature) to get an accurate statistical distribution of cycles to failure. If standard deviation of cycles to failure divided by the mean value of cycles to failure exceed the set value, then a warning message is displayed.

Allowable speed deviation:
A warning message displayed if the standard deviation for speed divided by the mean value for speed exceeds the set value.

Stress correction factor of reference test gear (Y_{ST}):
Calculated tooth root stresses are normalized by the stress correction coefficient and then recorded in a .DAT material file.

$$\sigma_{F,DAT} = \frac{\sigma_{F,CALC}}{Y_{ST} \cdot S_F}$$

Root safety (S_{FMIN}):
Calculated tooth root stresses are divided by the root safety factor and then written to a .DAT file.

Flank safety (S_{HMIN}):
Calculated tooth flank stresses are divided by the flank safety factor and then written to a .DAT file.

Power on time, Housing heat-transfer resistance, Housing heat-dissipating surface:
The values set here are used to calculate gear heat transfer coefficients based on the measured root and flank temperatures. For more information, see VDI 2736-2 [109].

63.4 Extrapolation of the calculated permissible root and/or flank stresses

If root and/or flank fatigue results are available, it is possible to extrapolate the calculated results. This can be done in the tab `Data extrapolation`. With method 1, it is possible to extrapolate data up to the user defined number of cycles (Extrapolate to cycles). Extrapolation can be done with an average slope or 0.5k average slope. With method 2, you can determine what to do after the user defined cycles – either set permissible strength to 0 or set cycles to infinity (at a given permissible stress).

Individual extrapolation options are available for root, flank oil, flank grease and flank dry results. The effect of Data extrapolation can be seen in Graphics under S-N curve.

It is possible to extend the calculated results to lower temperatures by selecting the `Extend temperature range` option. Set the parameters for the minimum temperature (must be lower than the lowest calculated temperature in the results) and the coefficient for increasing permissible stress (the coefficient must be ≥ 1). The results can be seen in the S-N graphics or in the generated .DAT file.

63.5 Other calculation options

If results are available, you can calculate the wear factor k_w from the worn tooth geometry as defined in the VDI 2736-2. This option must be enabled in `Module specific settings` (select `Calculate wear factor from worn gear`). An additional field (the "Average wear" field) is then displayed in the measurement table. To calculate the factor k_w, select "wear" as the cause of failure. The calculated wear factors are displayed in the report. The wear factors are not written to the .DAT file.

If lubrication is either grease or dry running, heat transfer coefficients k_θ for the root and flank of the failed gear are automatically calculated according to the VDI 2736-2. The calculated factors are also shown in the report.

63.6 Writing material data to the KISSsoft database

In the tab `DAT file`, it is possible to generate and open .DAT data file and automatically save material to a KISSsoft database.

63.7 Graphics

Based on the material inputs and calculated results, the following graphics are available.

- Young’s modulus vs. temperature
- Ultimate and/or yield material strength vs. temperature
- Wear factor vs. temperature
- S-N curves (Woehler lines) for root and flank (all lubrication regimes)

Part IX

KISSsys

Chapter 64
KISSsys: Calculation Systems

64.1 General

KISSsys is an extension to the KISSsoft calculation program. With KISSsoft, you can size, optimize and recalculate individual shafts, gears or shaft-hub connections. In contrast, KISSsys is suitable for administering machine element systems.

Some special links between different calculations are already present in KISSsoft. For example, bearing forces can be transferred from the shaft calculation and gears can be placed onto a shaft. However, in the case of larger systems, such a multi-level gearbox with several shafts and gears, separate performance data and speeds must be entered for each individual stage. If several loads need to be calculated, the rating must be updated in each calculation.

In contrast with KISSsoft, where the individual calculation takes center stage, KISSsys provides a way to observation a system as a whole. However, KISSsys has not been designed to replace KISSsoft. Instead it is an extension that uses the existing, tried and tested calculation modules. You could say that KISSsys administers the relationships between individual elements but leaves the calculation of the individual elements to KISSsoft.

64.1.1 Structure of KISSsys

KISSsys is based on an object management system called Classcad. Classcad manages the administration of KISSsys elements, evaluates the expressions for variables and provides an interpreter with which the user can also generate functions for special purposes.

This forms the basis for a user interface and a link with KISSsoft. The functionality of the user interface is different for administrators, who generate new systems or change the systems' structure, and for normal users who, while using the same structure, merely want to change data, recalculate, and observe results. It takes more effort, and a better understanding of the program structure, to generate new systems than to use an existing system, which is easy to do.

64.1.2 Ways in which KISSsys can be used

At the most basic level, KISSsys provides a way for grouping calculations. All calculations belonging to a system can be called up from one interface. In addition, you can get an overview of the most important results of all calculations. This makes it immediately obvious which particular gear pair or shaft is critical.

Even just this view of all the calculations that are of interest makes work considerably easier.

KISSsys then offers a way for you to specify relationships between variables. For example, you can calculate the speeds in a gearbox from the initial speeds and the transmission ratios. Moreover, KISSsys can also describe the power flow. Consequently, in KISSsys you only need to enter the load for the calculations in a few places. This enables you to quickly recalculate a complex system for varying load cases.

KISSsys enables you to store tables for loading cases or even variants. In this way, you do not have to constantly reenter the load data. KISSsys can also store the data for variants of a construction. With one click of a button you can then perform all the calculations for a selected load or variant.

For example, imagine a shaft with a radial force of unknown direction (e.g. via a belt drive/ belt force, whose direction is only determined when the equipment is installed). If it is necessary to define the worst case scenario, you could use KISSsys to rotate this force in steps of up to 360°.

KISSsys is not only of great benefit during design, but it is also useful in the sales environment. With KISSsys, you can, for example, store a standard gearbox in your computer. If the client later requests different loads on a gearbox of this kind, instead of the ones originally used for its construction, KISSsys lets you quickly check whether the gearbox will meet the new load requirements.

Different example applications are illustrated on the KISSsoft CD or website.

64.1.3 The user interface

The user interface provides several views of the administered data. There are table views, which are primarily designed to provide you with a good overview of the calculations. Another view, which has a tree structure, represents the hierarchy of an assembly structure, while the two-dimensional power flow diagram is primarily designed to display the kinematic coupling of the system. In addition, you can produce a three-dimensional display of the entire system or of subsystems.

This section details the options for using the KISSsys system without administrator rights.

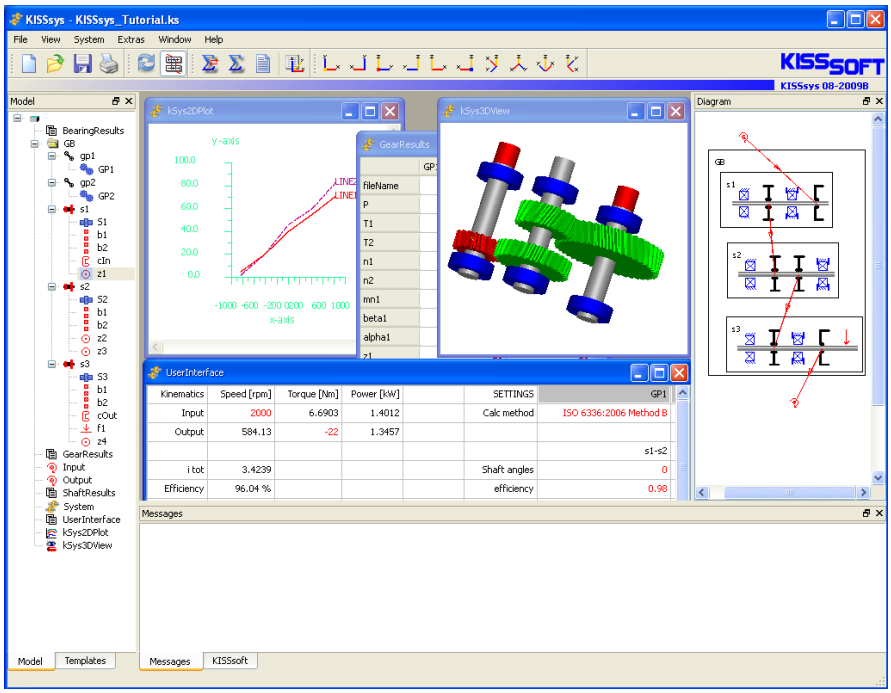


Figure 64.1: The KISSsys user interface with tree view, diagram view, 3D view, tables and 2D diagram

64.1.3.1 Tree view

The tree view (on the left in → Figure on page IX-1165) lists all elements present in the system, hierarchically. This provides a way to display an assembly structure. A bitmap image is displayed next to the element's name. This image identifies the type of the element. Bitmaps with a blue background are KISSsoft calculations, and bitmaps with a gray background are variables such as figures, functions or character strings. With a click with the right-hand mouse button you can open a separate context menu which provides functions for an element.

Each element has a `Properties` dialog which you can display here. The `Properties` dialog contains an overview of the available data elements or variables. However, these can only be changed by the administrator.

In the case of the KISSsoft calculations you can select `kSoftInterface` in the context menu to start the appropriate KISSsoft module. The calculation data can then be changed or evaluated in KISSsoft. Select `kSoftReport` to display the

calculation report and select `Calculate` to perform the calculation in the background without a user interface. Data is only exchanged with KISSsoft via the KISSsys calculation elements.

64.1.3.2 Diagram view

Diagram view (on the right in → Figure on page IX-1165) shows the kinematic coupling of the elements. To start with, the element structure has nothing to do with the calculations. The calculations only use the data that relates to the shafts, gears and connections, and they can be added or deleted as you wish.

The structure consists of shafts and their sub-elements: gears, forces, couplings and bearings. The kinematic coupling and the power flow between the shafts is achieved via `connections`. The connection has the calculation standard for transferring the speed to the next element (usually simply the ratio). It also transfers a torque and a loss of efficiency.

The externally supplied torque and a speed are defined with speed/torque elements. In each case, you can specify whether the speed or the torque are known, or whether they should be calculated by KISSsys. The number of predefined values must correspond with the number of degrees of freedom.

The elements in the diagram view can be moved with the left-hand mouse button. Click with the right-hand mouse button to display a context menu like the one in tree view. You can change the zoom factor by pressing the '+' or '-' keys, or in the context menu which you access by right-clicking.

64.1.3.3 Table view

To display the tables, select `Show` in the context menu in tree or diagram view, or double-click on the tree view table element with the left-hand mouse button. The contents of the tables are defined during system set-up. The values displayed in black cannot be changed, but the red numbers or strings can be edited. A special table for user interfaces contains fields with a gray background. Start these functions by double-clicking on them.

You can print the contents of the table, or press `Ctrl-C` to copy it and, for example, paste it into a spreadsheet.

64.1.3.4 3D view

To display the windows for the 3D view, select `Show` in the context menu in tree view. You can rotate the view with the left-hand mouse button, enlarge or reduce it

with the right-hand one, and move it with the center mouse button. One of the main views can be selected via the menu or the tool bars.

In 3D view you can export the 3D geometry into the CAD system (via the context menu). If you want to display graphical elements (see `System.kSys3DElements`), you can export the elements from that file if you have the appropriate license. If a 3D kernel is present and you want to generate solid elements, a CAD file will be generated directly.

64.1.3.5 Message output

In the lower part of the program window (see → Figure on page IX-1165) there is an output window for messages. Error messages and warnings from KISSsoft calculations are displayed under `Messages`. Calls by `KISSsoft` are reported under KISSsoft, so this view is usually not required.

In the lower program window, as in KISSsoft, you will see an information tab. If information about a particular function is present, it is displayed in this tab.

64.1.3.6 Menus, context menus and the tool bar

In the File you can open, store and save models, open and close projects, and close KISSsys. You can also open or close KISSsys templates.

Each individual docking window in the user interface can be hidden or displayed in the View main menu. You can also refresh all views.

In the System main menu you can generate the KISSsoft report and perform kinematics and KISSsoft calculations. You can also select a load spectrum for the model from the KISSsoft database or define your own in KISSsys. Use these functions to call the characteristics of the element and variable overviews.

In the Insert menu, the Assistant will guide you through the steps required to create a simple gear unit or a single stage planetary gear unit. The Elements Box operates like the Assistant in that it lets you add elements, but without any structural limitations. Default templates are used to add prefined models or templates to an existing model.

In the Extras menu you will find the administrator settings, the license tool, the configuration tool and the language setting. Select Extras > Settings to change general program settings such as the names of individual elements or table settings.

As in Windows, at the end of the menu bar you will see the Help icon which you can use to navigate in the KISSsoft manual. Click on `Help > Info` you will find information on the program version and on the support provided by KISSsoft.

In the Window main menu you will find actions for organizing the opened subwindows such as tables and 3D views. The print option is only enabled if a table is open.

In addition to the main menu, KISSsoft uses context menus in many places. Use context menus to access actions for a particular area or model element. Normally, you click the right-hand mouse button to display context menus.

Select the Tool bar for rapid access to the menu actions that you need to use most often. Also note the tool tips: they display information about the actions in the tool bar and also the more detailed explanations in the status bar.

64.2 Creating Models in KISSsys

This chapter is intended for KISSsys users.

There are four ways for you to create new models in KISSsys. They are described in the next four sections.

64.2.1 Classic method

If you enable the Administrator in `KISSsys Extras > Administrator`, the system displays, in the Template tab, all elements that are required for creating a model. You create a model by copying the particular elements from the template and inserting them in the navigation tree. It is also possible to create the elements by moving them from the template to the navigation tree. All possible models can be created with the classic method.

64.2.2 Element Assistant

The `Insert > Elements` Assistent is based on the classic method. If you use this assistant, you no longer have to drag and drop or cut and paste data. To insert an element, click on it, and the system automatically inserts it in the current structure in the navigation tree. Using the Element Assistant you can create all possible kinds of variants.

64.2.3 System Assistant

Under `Insert > System of Shafts` KISSsys provides an Assistant with which you can create cylindrical gear stages. This Assistant leads you through the model step by step when you are creating it. You use it in the same way as the Assistant. Use this function to create a cylindrical gear unit.

Use the KISSsys Assistants to create parallel shafts with the following combinations of gears:

Cylindrical gear

Bevel gear

Worm wheel

Face gear

Use the Planetary gear Assistant to create a single stage planetary gear unit.

This Assistant leads you through the model step by step when you are creating it. You use it in the same way as the Assistant.

64.2.4 Setup with icon

The default setting is that all the elements are listed on the left-hand side of the screen as icons, so you can construct a model from the very beginning.

64.2.5 Creating and modifying tables

To create your own tables in KISSsys Model you can use a predefined table called "UserInterface", which is stored in the template in "Tables". Using this "UserInterface" table, you can add all parameters from the elements and your own texts. You can change the table's name to suit your needs.

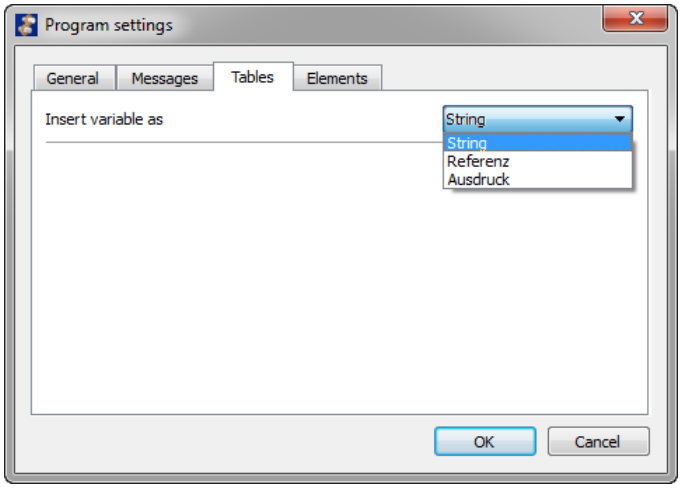
For each cell, no matter what it contains, you can right-click with the mouse to select Format in the context menu. There you can set the font, color, background color, and position, of the text for that cell.

64.2.6 Adding variables to tables

One way of inserting variables in the "UserInterface" table is to select a variable from the "Properties" dialog and then click on the corresponding icon in the "Toolbar" menu. You can then insert the variables either as text (Name), as a reference, or as an expression.

 Text

You can also input texts directly in the cell. Alternatively, you can use the Text icon to help you. Select the parameter you want to insert from an element's property and preselect a cell in the table. Then click on the Text icon. This transfers the required value to the cell. You can also hold down the left-hand mouse button and drag and drop the required parameter to insert it directly. The default setting is for text to be inserted. Click on `Extras > Settings > Tables tab` to define the default setting as a text, reference or expression



 Reference

Referenced data is displayed in red. These values are referenced with a parameter in the element property. You can modify the value both in the table and in the property.

You can add referenced values to cells in the same way as you add texts. Select the parameter you want to insert from an element's property and preselect a cell in the table. Then click on the Reference icon. This transfers the required value to the cell. Alternatively, you can hold down the right-hand mouse button and insert the parameter in the appropriate cell. A selection window is displayed, in which you can select a text, reference or expression, as required.

 Expression

The expression is merely shown as a value and cannot be modified in the table.
You can insert an expression into a cell in the same way as a reference.

64.2.7 Individual names for elements

Individual names can be used for all KISSsys elements. The individual name is assigned to the element automatically when it is assigned to the model. This behavior can be set individually for each element (`Extras->Settings->Elements`).

Use the <autoInc> and <localInc> tags to add an index to the individual name at the insertion position. The first of these tags increments the index globally. This means that no other element in the model can have the same name. The second of these tags increments the index locally (in the same folder). Use the <parentName> tag to add the name of the hierarchically superior item.

The "Automatically/Ask" option is set to suppress or display the dialog that prompts the user to define the name for the new element for the model. Click on "Reset" to select the KISSsys default name. You can use the "All questions", "All automatic" and "Reset all" functions to modify all the elements at the click of a button.

64.3 Extended functionality for developers

In addition to the functionality already described, more functions are available for developers.

- To open a template file, click on File>Open Templates. Alternatively, click on Insert > Default templates to load the template file. It is displayed as a tree under Templates.
- To add new elements in tree view, you can "Copy" and "Paste" them. The new elements are added as copies from a template file.
- You can rename and delete elements via context menu functions.
- The data in the `Properties` dialog can be edited. New variables can be added and deleted
- Hidden variables will be displayed and all functions can be performed.
- Hide messages by selecting `Extras>Suppress messages`.

64.3.1 Properties dialog

In tree view, or in the diagram for an element, you can open the KISSsys `Properties` dialog via the context menu. In it, you can add new variables or change existing ones. Only one `Properties` dialog is available. A second one will not be displayed.

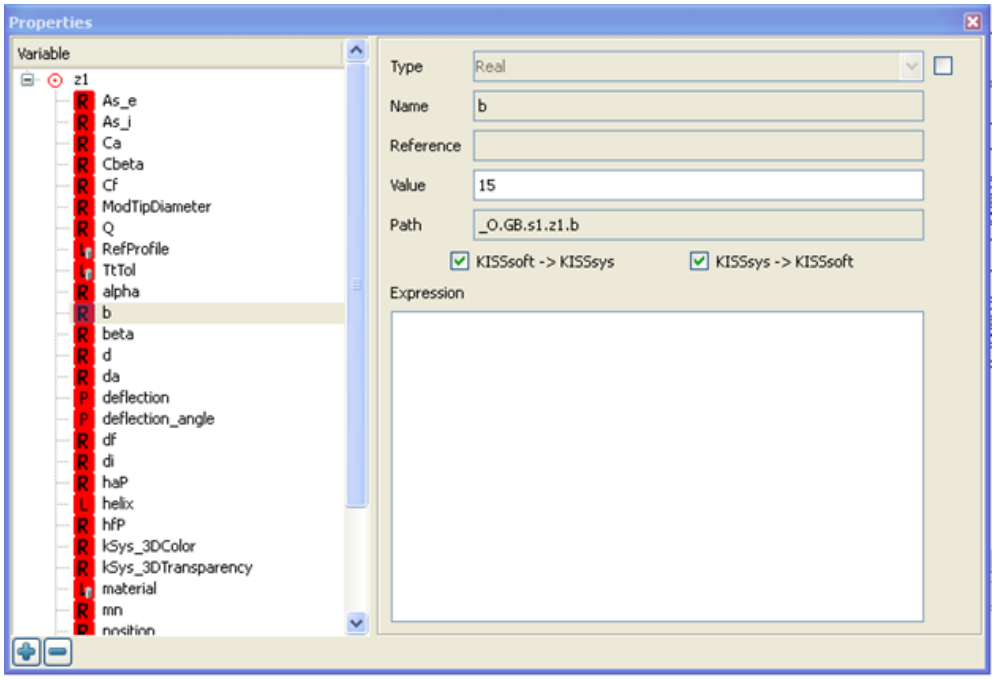


Figure 64.2: The KISSsys Properties dialog

Figure 64.2 shows the `Properties` dialog. On the left you see a tree view in which you can select data elements or variables, and on the right you see a dialog for the selected variable. The following fields are available for the variables:

- **Type:** Display the Variable type (see section "Variables" on page IX-1181).
- **Name:** The name of the variable. You can change the name here. However, if a variable has to be used in formulas or references, you must also change the name there, as otherwise the variable cannot be found.
- **Reference:** Enter the reference target here for reference elements. A name must be entered in quotation marks. An alternative would be the name of a string variable (see page IX-1182). In the case of Variants (see page IX-1182) the index must be entered here in an array. Here, an invalid reference will be marked in red.
- **Value:** The current value of the variable.
- **Expression:** An Expression used for calculating the variable (see page IX-1188). The value will be calculated on the basis of the expression, if an expression is present.
- "**KISSsoft** → KISSsys" flag. The variable can be transferred from KISSsoft to KISSsys.
- "**KISSsys** → KISSsoft" flag. The variable can be transferred from KISSsys to KISSsoft.

You can convert the variable into a `reference` or `variant` variable and vice versa by clicking on the Reference or Variant button.

64.3.2 Table view

The format of the tables is defined in the hidden `definition` variables. There are different types:

- **Table for calculations:** This table is best suited for displaying the data for several elements of the same type. The definition format is:

```
[[type,rows,columns],['variable1','variable2',..],
[element1,element2,..]]
```

In the case of type 1, you can edit each displayed value, in the case of type 2 you can edit all values that have no expression, and in the case of type 3 you can edit all values for which the `KISSsys` → KISSsoft flag has been set. The Number of Rows or Columns is not used.

- **Table for arrays or variants**: In this table, the arrays or variant variables are each displayed in a separate column. The definition format is:

  ```
  [[type,rows,columns],['variable1','variable2',..]]
  ```

  In the case of type 21, you can edit each displayed value, in the case of type 22 you can edit all values that have no expression, and in the case of type 23 you can edit all values for which the `KISSsys` → KISSsoft flag has been set. The Number of Columns is not used.

- **Table for user interface:** You can configure this table to suit your needs. The definition is `[[type,rows,columns],[[A1,B1],[A2,B2]]]`. The contents can be inserted via a context menu in the table, and should not be changed in the definition. Since the definition is changed interactively, you must not set an expression here. The number of rows or columns should also only be changed via a dialog, as otherwise information on reference elements will be lost.

64.4 The existing elements

64.4.1 Variables

The following variables can be used:

- **Real**: A numerical value.
- **String**: A character string. Input in quotation marks e.g. "Text".
- **Point**: A coordinate or vector with 3 components. Input in {1,2,3} format.
- **Array**: A one-dimensional or multidimensional field. Input e.g. as ["Text",1.23,{1,2,3},[1,2]].
- **Function**: An executable function. Input best entered via the special input screen.
- **ElementID**: The ID of a Classcad object. Output as $31, input as name of the object with no quotation marks.
- **List**: Displayed as selection list and acts as a number in the Interpreter (index of the list beginning with 0). The selection list is defined as an array via the Edit list menu option, e.g. ["one","two","three"].
- **Database List**: The name from the KISSsoft database is displayed in a selection list. In the Interpreter, this type also acts as a number according to the database ID. The database assignment is defined as an array via the Edit list menu option: ["database","table"]

Each of the variables has a name, a value, an expression and different flags. If an expression is present, the value of the variables is defined via this expression. The expression is therefore particularly suited for the input of formulas. If, in contrast, a formula is entered in place of the value, this formula will be evaluated and the result will be assigned. The actual formula will be lost. The KISSsoft->KISSsys and KISSsys- >KISSsoft flags determine how data is exchanged between the two programs. Only variables with the appropriate flag activated will be exchanged.

In the case of functions, the function is placed in the expression, and the value has no meaning.

For the Real, String, Point, List and Database List data types there are additional reference elements and variant elements.

64.4.1.1 References

A reference element behaves like any other variable, with the difference that another variable fetches the data. A valid variable name must be entered as the target for the reference element. The reference target must be entered as a character string. This will be either an actual name in quotation marks or an expression resulting in a character string, e.g. a concatenation of character strings (e.g. `gear1+'.z'` with the string variables `gear1` or `'gearwheel1.z'`). The system marks an invalid reference in red.

64.4.1.2 Variants

Internally, the variant elements administer a field of variables, whereas externally they behave like a normal variable. As additional data, the variant is assigned an index variable, which indexes the field. The index variable must be entered as an `array` of variables (e.g. `[system.index]`). With these data types you can store load spectra or system variants and the results can be displayed in tables.

64.4.2 Calculation elements

All elements for KISSsoft calculations are derived from classes which begin with the name `kSoft`. In tree view they have a blue background.

The calculation elements have a series of functions:

- **Calculate**: performs a KISSsoft calculation in the background.
- **kSoftInterface**: starts KISSsoft interactively
- **kSoftReport**: performs the calculation and displays the report.
- **SetFlags**: Sets the flags for data exchange between KISSsoft and KISSsys to suit the required storage location.
  - **Save in KISSsys**: The data will be passed on in both directions.
  - **Save in KISSsoft**: Data with a stored expression will be transferred from KISSsys to KISSsoft, and all other data will only be transferred in the other direction.

  This function sets the flags only once when selected. It therefore has no effect on later changes.
- **kSoftModul**: This hidden function displays the KISSsoft module descriptor.
- **getTranslationTable**: This hidden function shows the translation table for variable names from KISSsys to KISSsoft. In the calculation element, the translation table can be extended via the `TranslationTable` array: For example,

an entry of `[['eps_a_min','ZP[0].Eps.aEffI'],['eps_a_max','ZP[0].Eps.aEffE']` adds a link between the variables `eps_a_min` and `eps_a_max` and the corresponding KISSsoft variables. Until now the names of the KISSsoft variables could only be taken from the report templates, `*.rpt`.

- **getUtilization**: This function returns the utilization, and the required safety/safety ratio.

In the `fileName` variable you can specify a KISSsoft calculation file which will automatically be loaded at the start of the calculation, before any other variables are transferred. You can use the `savingMode` variable to specify whether this KISSsoft calculation file should be saved automatically:

- **Don't ask and don't save** When KISSsoft is shut down, you will not be asked if the file should be saved after changes have been made to it.
- **Ask for saving** When KISSsoft is shut down you will be asked if the file should be saved. (KISSsoft default response)
- **Save automatically** When KISSsoft is shut down, the calculation file will be saved automatically without a user confirmation prompt.
- **Save file in KISSsys** No file name will be entered in `fileName`. Instead, the entire calculation file will be saved in the KISSsys element.

The shaft calculation contains the special method UpdateShaftElements. This must be called up if an element of force is to be added/deleted on a shaft. It evaluates the type and number of elements of forces on the shaft and transfers them into the 'forces' array in the shaft calculation. This array is a defining factor for the forces in the shaft calculation.

64.4.2.1 Relationship of calculations with elements

Templates are provided which automatically link the calculation with the shafts and gears. To do this, use the `Dialog` function. In the case of fundamental changes, i.e. when more elements of forces are added to the shaft, this dialog must be called up again to update the relationships.

64.4.2.2 Storage strategies for calculations

There are different options for saving the calculation data:

1. All data is to be saved in the KISSsys file and the calculations can only be accessed via KISSsys: Select `Save file in KISSsys`, in `savingMode`. It is best to set the flags bidirectionally.
2. All data is to be saved in a KISSsoft file and the file can also be changed outside of KISSsys: Select `Ask for saving`, or `Save automatically` in `savingMode`. The flags must be set to `Save in KISSsoft` with `SetFlags`. Note here that the calculation data will only be loaded from the KISSsoft file when the calculation is called up for the first time. After the KISSsys file is opened, you should therefore call up `kSoftCalculate` once again.

64.4.2.3 Importing existing KISSsoft calculations

If there are already KISSsoft calculations present for elements of a new KISSsys system, you can simply load the files into the KISSsoft window. However, you should note a few points:

- The file name under `fileName` in the KISSsys calculation element will be changed. The name must either be deleted or modified.
- During the shaft calculation the elements of forces and the bearings are overwritten. For this reason, you need to call up the `dialog` or the `UpdateShaftElements` function after importing the calculation. The elements of forces and bearings cannot be imported, and neither can the positions. This data must be entered in KISSsys.
- In the case of gears you must ensure that the sequence of the gears matches up.

64.4.3 Elements for shafts

Different elements can be placed onto shafts. They will also be transferred into the KISSsoft shaft calculation. The position on the shaft is defined with the `position` variable.

- **kSysHelicalGear**: A cylindrical gear.
- **kSysBevelGear**: A bevel gear. The position of the peak is defined by the `direction` variable.
- **kSysWorm**: A worm.
- **kSysWormGear**: A worm wheel.
- **kSysCoupling**: A coupling. Diameter `d` and Width `b` can be entered for the 3D display.

- **kSysBearing**: A normal type of bearing. Losses can be recorded in `Tloss`. The direction of the loss torque should be defined with a `-sign(speed)` in the expression.
- **kSysRollerBearing**: A rolling bearing. The bearing geometry will be loaded from the KISSsoft bearings database during each refresh. Losses can be recorded in `Tloss`. The direction of the loss torque should be defined with a `-sign(speed)` in the expression.
- **kSysCentricalLoad**: A centrical load. KISSsys will always prompt with a torque (Ty) but no power. This torque will also be included in the kinematics calculation.
- **kSysMass**: An additional mass on the shaft.
- **kSysRopeSheave**: A rope sheave. Unlike the torque, the belt force will not be calculated via the connection. It is up to you to ensure that the belt force in two belt sheaves matches up.
- **kSysFaceGear:** A face gear.

64.4.4 Connection elements

- **kSysGearPairConstraint**: a connection between two cylindrical or bevel gears.
- **kSysPlanetaryGearPairConstraint**: a connection between a gear and a planet. You can select the type of pairing: sun-planet, planet-internal gear or planet-planet. Both gears must also be entered in this sequence. A planet carrier must also be selected. The number of planets must be defined in the `NofPlanets` variable in the planet carrier coupling.
- **kSysPlanetaryBevelGearConstraint**: a connection between a bevel gear and a rotating bevel gear for bevel gear differentials. As in the case of the planetary connection, the sequence of the bevel gears and the number of planets must be defined. An efficiency cannot be specified here.
- **kSysWormGearConstraint**: a connection between a worm and wormwheel. You can also define two efficiencies (`eta1` and `eta2`) for the driving worm or driving gear.
- **kSysCouplingConstraint**: A connection with ratio 1 between two couplings. The kinematic force of the coupling can be activated or deactivated. Additionally, it is possible to specify a slip, e.g. for flake graphite couplings or synchronizations. The torque in the connection will usually be calculated, but it can also be specified.

- **kSysBeltConstraint**: A connection between belt sheaves. The transmission ratio is calculated from the diameter ratio. A slip and an efficiency can be specified. When you are inputting the slip, take into account the ratio and the sign. The calculation is performed in accordance with: $n_1 - d_2/d_1 \cdot n_2 = slip$
- **kSysConnectionBearing:** A connection between two shafts
- **kSysConnectionRollerBearing:** A connection between two shafts
- **kSysSynchronizer:** A shaft synchronizer

  Use the `setConfig(slipConstraint_r/[slipConstraint_r, slip_r], torqueConstraint_r/[torqueConstraint_r, torque_r])` function to activate or deactivate the connection:
  1. Closed, without slip: `setConfig([TRUE, 0], FALSE)`,
  2. Open, without torque: `setConfig(FALSE, FALSE)`,
  3. Open, with torque: `setConfig(FALSE, [TRUE, 20])`

- **kSysSpeedOrForce**: An element for specifying speed or torque. Both values can either be specified or else calculated by the system. You can also preset the power as an alternative to the torque.

  Use the `setConfig(speedConstraint_r, torqueConstraint_r/[torqueConstraint_r, type_r, torque_r])` function to change the presets. If you specify a load type, the values below have these meanings: 0..torque with sign, 1..torque driving, 2..torque driven, 3..power driving, 4..power driven. Examples:
  1. Speed and torque specified: `setConfig(TRUE, TRUE)`,
  2. Speed and torque with value specified: `setConfig(TRUE, [TRUE, 0, 20])`,
  3. Only driving power specified: `setConfig(FALSE, [TRUE, 3, 20])`

64.4.5 Displaying elements in 3D graphics

Each element has an `OnRefresh3DView` function which generates the 3D display. If necessary, this function can be overwritten. You can set the color of an element in the range from 0 to 255, with the `kSys_3DColor` variable, and set the transparency with the `kSys_3DTransparency` variable. These two variables must be created if necessary.

64.4.6 System settings

You can make use of a series of setting options in the `System` element:

- **kSoftAcceptChanges**: default setting `yes`, the changes will be transferred from KISSsoft. If the setting is `no`, nothing will be transferred. If it is `asked`, you will be prompted to confirm whether the changes should be accepted when KISSsoft is shut down.
- **kSysKinematicFunc**: while the kinematics are being calculated, you can call up the `OnCalcTorque` function. The standard implementation of this function calls up the calculation of the bearing actions for all shafts.
- **kSysKinematicMode**: the calculation of the kinematics can either be iterative or not. Iterations for the torque must be activated if the efficiency needs to be included. Iterations for speeds are only necessary if formulas for speeds have been entered.
- **kSys3DElements**: you can optionally display graphical elements or solid elements (3D kernel required). Graphical elements will be generated more quickly, although solid elements are more detailed, and it is, for example, possible to also display a loaded housing.
- **project_name**: the project name will be displayed in the KISSsoft calculation reports.
- **project_contract**: The commission number will be displayed in the KISSsoft calculation reports.

64.5 Programming in the Interpreter

There are programming options in the expressions used in variables and in functions.

64.5.1 Expressions in variables

The programming options in expressions are restricted. No local variables may be used.

Between the data types, the operators are defined in accordance with Table 64.1. A series of mathematical functions is also available. They are listed in table 64.2.

Data type	Operations	Description
`Real`	+,-	Addition and subtraction
	*,/	Multiplication and division
	<,>=,=,!=,>=,>	Relational operators
	!,AND,OR	Logical operators
`String`	+,LEN	Concatenation and length operators
	<,>=,=,!=,>=,>,!	Relational operators
`Point`	+,-	Addition and subtraction
	*,**	Scalar and vector multiplication
	:x,:y,:z	Access to components
	LEN	Vector length
`Array`	[],+,LEN	Indexing, concatenation and length operator

Table 64.1: Permitted operators for data types

`abs(x)`	Supplies the value of x
`sign(x)`	Supplies the sign of x (+1, -1 or 0 if x=0)
`min(a,b,...)`	Supplies the smallest value of the arguments
`max(a,b,...)`	System supplies the largest value of the arguments
`a_r(x)`	System converts from degrees to radian measure
`r_a(x)`	System converts from curve to degrees
`sin(x)`	System calculates sin of x in the radian measure
`sinh(x)`	System calculates sinh of x

`asin(x)`	System calculates arcsin of x
`cos(x)`	System calculates cos of x in the radian measure
`cosh(x)`	System calculates cosh of x
`acos(x)`	System calculates arccos of x
`tan(x)`	System calculates tan of x in the radian measure
`tanh(x)`	System calculates tanh of x
`atan(x)`	System calculates arctan of x
`atan(y,x)`	System calculates arctan of y/x
`exp(x)`	System calculates e to the power of x
`ln(x)`	System calculates the natural logarithm of x
`log(x)`	System calculates the decadic logarithm of x calculates
`sqrt(x)`	System calculates square root of x
`pow(x,y)`	System calculates x to the power of y
`fmod(x,y)`	System calculates x modulo y

Table 64.2: Predefined mathematical functions

A variable's expression can contain the specified operations and any function calls. If limited expressions are to be used, the expression must begin with `#` and the result must be returned with `RETURN`:

```
#
IF a>b THEN
   RETURN a;
ELSE
   RETURN b;
ENDIF
```

64.5.2 Functions

The different options for programming in functions are best described with the help of examples. A function's header looks like this:

```
// Transferred Variables of calling Program
PAR Parameter1, Parameter2;

// Declaration from Constants
CONST PI=3.1415926, E=2.71828;
```

```
// Declaration from local Variables
VAR a,b,c,d;
```

Here, the lines that begin with `//` are comments. Each of these three lines may only occur once, and the declared variables must be separated with a comma. A non-initialized parameter or variable is `VOID`. This can be checked with `ISVOID(variable)`.

Limited statements have two variants: `IF` or `SWITCH` statements:

```
// IF statement with optional ELSIF and ELSE Block
 IF Parameter1 > 5 THEN
     a = sin(PI*Parameter1);
 ELSIF Parameter1 < 0 THEN
     a = Parameter1;
 ELSE
     a = 0;
 ENDIF

 // SWITCH Statement with Selection of figures or texts
 SWITCH Parameter2
 CASE 'Zero': b = 0;
 CASE 'one': b = 1;
 DEFAULT: b = 5;
 ENDSWITCH
```

For loops, there are four program variants:

```
// FOR Loop with optional intervals
FOR a = 1 TO 8 STEP 2 DO
    b = b + a;
    IF b>100 THEN
        BREAK;      // Ends the Loop
    ENDIF
NEXT

// WHILE Loop
WHILE b<100 DO
    b = b*10;
WHEND

// DO Loop
DO
    b = b*10;
UNTIL b>100;

// FORALL Loop calls for all Elements of an an array
c = [1,2,3,4,5,6,7,8,9];
a = 0;
FORALL c d DO   // d receives each of the Value of an an element from c

    a = a + d;
NEXT
```

There is a special syntax for calling up functions that belong to objects. The standard method is to specify the object name followed by a point and the name of the function. However, the name of an object can also be contained in a local variable. This enables you to change the object for the function call at runtime.

```
    // The Function function calls for for object.
    Object1.OBJ_GetMember('variablenname');

    // a is a local Variable of Type String with the the of an object
    a = 'Object1';

    // This calls a Service function for called hierarchically superi-
or with the a a
    b = a.OBJ_GetMember('variablenname');

    // calls a of User by Function for for Object1.
    a.Userfunction();

    // the of User by Function calls for called
    // for the hierarchically superior object.
    Userfunction();

    // the of User by Function calls for called
    // for the hierarchically superior object.
    ^.UserFunction();
```

The system searches for variable names relatively to the current object. If `object.z` is used in an expression, the system will first attempt to find this variable below the current object. If it is not present, the search will continue in the hierarchically superior object (in accordance with `^.object.z`) and so on.

64.5.3 Important service functions

`OBJ_GetChildren()`	Supplies an array with all child objects.
`OBJ_GetName()`	Supplies the name of the object.
`OBJ_GetId()`	Supplies the ID of the object.
`OBJ_GetId()`	Supplies the ID of the object.
`OBJ_HasMember()`	Tests whether a variable is present
`OBJ_GetMember()`	Supplies the variable of the current object.
`OBJ_FindMember()`	Supplies the variable of the current or hierarchically superior object.

Table 64.3: Important service functions

64.5.4 Variable dialogs

In interpreter functions, variable dialogs can be generated for the input of variables. The call is:

```
res = CADH_VarDialog(["Title", Width, Height, Pitch],
[Dialogelement1], [Dialogelement2], ...);
```

The title is displayed in the dialog title line, and the width and height values show the dialog's dimensions in pixels. The pitch (between 0 and 1) describes the relationship between the width of the field description and the dialog width (default value 0.4). This definition of the dialog size can be followed by any number of arrays that each define the individual dialog elements.

The return value is an array. Its first value is `res[0] =1` if the dialog has been closed by the user clicking `OK`. Otherwise, it is zero. The other elements of the returned array supply the results of the input fields.

Below, the following convention is used to define the type of a variable: _str=String, _n=Int, _r=Real, _b=Bool. For example, in the case of Caption_str, this means that the Caption variable is of the type String.

64.5.4.1 Dialog elements for the variable dialog

The following dialog elements are available for the variable dialogs:

Horizontal grouping:

The horizontal grouping provides a framework in which the individual dialog elements are lined up beside each other. Their position must always be defined by a vertical group, which means that all dialog elements contained within a horizontal grouping must be defined in a vertical group. A horizontal group is defined as follows:

[C:VDGL_HORZ,Caption_str,DistAbove_n,DistAfter_n,[Dialogelem]]

- C:VDGL_HORZ: Type definition for horizontal grouping.
- Caption: The horizontal grouping's caption. If "Caption" is not an empty string, a frame will be drawn around the horizontal group.
- DistAbove: distance above the horizontal group to the next dialog element.
- DistAfter: distance behind the horizontal group to the next dialog element. "DistAfter" and "DistAbove" are specified in pixels.
- [Dialogelem]: element array used to define the dialog elements located in the horizontal grouping. This array must only contain elements of the type VDGL_Vert.

Vertical grouping:

The vertical grouping provides a framework in which the individual dialog elements will be lined up below each other. The width of the dialog elements is defined by the vertical group. A vertical group is defined as follows:

[C:VDGL_Vert,Caption_str,[XStart_r,XEnd_r],XPart_r,[Diag],Marg_n]

- C:VDGL_Vert: Type definition for vertical grouping
- Caption: The vertical grouping's caption. The vertical grouping always has a frame drawn around it.
- [XStart,XEnd]: XStart and XEnd define a factor (between 0 and 1) for the width of the vertical group relative to the width of the hierarchically superior dialog. Additionally, they define the X-position of the vertical group.
- XPart: A factor between 0 and 1 that defines the ratio between the prompted value and the input value for the dialog fields (the text assigned to an input field is called the "prompt"). If XPart=-1, the prompt is positioned above the dialog element.
- [Diag]: Element array used to define the dialog elements located in the vertical grouping.
- Marg (margin): An optional parameter that defines the displacement of the dialog elements relative to the edge of the vertical group. This means that the dialog elements contain the distance "Marg" (margin) from both the left-hand and right-hand edge of the vertical group.

RealEditFeld:

Provides an edit box in which you can input a floating comma number.

[C:VDGL_Real,Prompt_str,Preset_r,res,res,Places_n]

- C:VDGL_Real: Type definition of RealEditFeld.
- Prompt: text assigned to the input field.
- Preset: Preset value.
- res: Here, a space has been reserved for two optional parameters which are not currently in use. However, these spaces must not be left empty in the definition (e.g. [C:VDGL_Real,Prompt,Preset,0,0,Places] would be a correct solution but not [C:VDGL_Real,Prompt,Preset,,,Places]).
- Places: An optional parameter defining the number of decimal places in the input field.
- ReturnVal: (return value). The return value is the input string.

IntEditFeld:

Provides an edit box in which you can input a whole number.

[C:VDGL_Int,Promt_str,Vorgabe_n]

- C:VDGL_Int: Type definition of IntEdit field.
- Prompt: text assigned to the input field.
- Preset: Preset value.
- ReturnVal: (return value). The return value is the input string.

StringEditFeld:

Provides an edit box in which you can input text.

[C:VDGL_Str,Promt_str,Vorgabe_str]

- C:VDGL_Str: Type definition of the StringEdit field.
- Prompt: text assigned to the input field.
- Preset: preset value.
- ReturnVal (return value): The return value is the input string.

Text display:

The system generates a text display. If an empty string is entered instead of text, the text field can also be used to define a distance.

[C:VDGL_Prompt,Prompt_str,Fieldheight_n]

- C:VDGL_Prompt: Type definition of text display.
- Prompt: Field text.
- Fieldheight: Height at which the text is displayed.

IntComboBox:

Provides a combo box in which you can input a whole number.

[C:VDGL_IntCom,Prompt_str,[Entr_n],Sign_n/[Ind_n],0,0,AsVal_b]

- C:VDGL_IntCom: Type definition of IntComboBox.
- Prompt: Text assigned to the combo box.
- [Entr]: Element array containing the available list items (in the case of an IntComboBox, the components must be whole numbers).
- Sign/[Ind]: Here, you have the option of using "Sign" to either set a constraint value, which is contained in the list, directly, or using "Ind" to select a value in a particular list position as a constraint value (the first element in the list is located at position 0)."Sign" or "[Ind]" are optional parameters.

- AsVal: If the optional "AsVal" parameter has been set, and does not equal 0, the return value becomes the input. Otherwise, the return value is the index of the selected entry.

IntEditComboBox:
Provides a editable combo box in which you can input a whole number. Please ensure that the values entered here are whole numbers.

[C:VDGL_IntComE,Prompt_str,[Entr_n],Sign_n/[Ind_n]]

- see IntComboBox
- ReturnVal: (return value). The return value is the input string.

RealComboBox:
Provides a combo box in which you can input a floating comma number.

[C:VDGL_RealCom,Prompt_str,[Entr_r],Sign_r/[Ind_n],0,0,AsVal_b]

- see IntComboBox

RealEditComboBox:
Provides a editable combo box in which you can input a floating comma number.

[C:VDGL_RealComE,Prompt_str,[Entr_r],Sign_r/[Ind_n]]

- see IntComboBox
- ReturnVal: (return value). The return value is the input string.

StringComboBox:
Provides a combo box in which you can input a string.

[C:VDGL_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n],AsPos_n]

- see IntComboBox
- AsPos: Contrary to the IntComboBox, the return value here represents the index of the selected field, if the optional parameter "AsPos" has been set and does not equal 0. Otherwise, the return value is the input.

StringEditComboBox:
Provides a editable combo box in which you can input a string.

[C:VDGL_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n]]

- see IntComboBox
- ReturnVal (return value): The return value is the input string.

Code button:

Service button:

64.5.4.2 Example application of a variable dialog

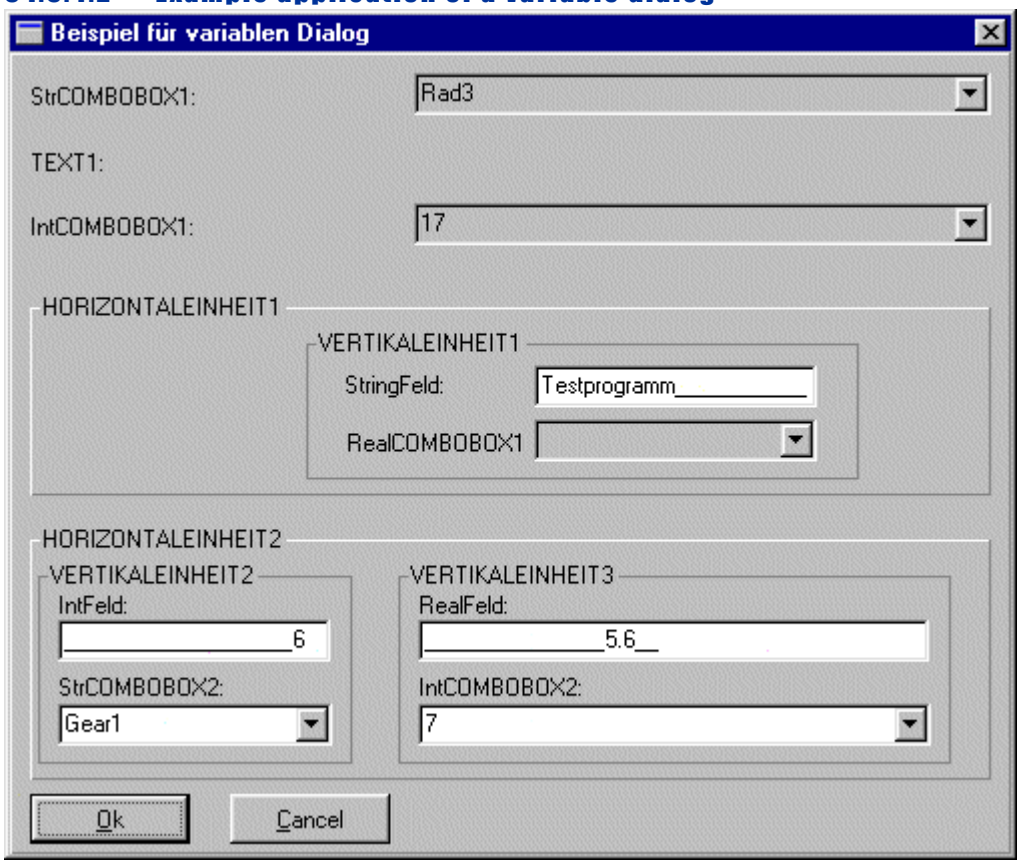


Figure 64.3: Example of a variable dialog

The section below uses the example of the program code for the variables dialog shown in Figure 64.3, in which the greatest possible number of elements are used:

```
//  VARIABLES DECLARATION

VAR res,result1,result2,result3,result4,result5,fullResult;

// DIALOG AND INPUT DATA

res = CADH_VarDialog(["Example of Variable Dialog",500,400,0.4],

      [C:VDLG_StrCom,"StrCOMBOBOX1:",["Gear1","Gear2","Gear3"],[2],0],
```

```
[C:VDLG_Prompt,"TEXT1:",30],
[C:VDLG_IntCom,"IntCOMBOBOX1:",[12,17,19],17,0,0,1],
// HORIZONTAL GROUP WITH ONE VERTICALGROUP
[C:VDLG_HORZ,"HORIZONTAL UNIT1",20,10,
  [     // Warning: remember brackets!
    [C:VDLG_VERT,"VERTICAL UNIT1",[0.3,0.9],-1,
      [
        [C:VDLG_Str,"StringFld:","Test Program"],
        [C:VDLG_RealComE,"RealCOMBOBOX1",[5.3,7.1,9.1],[2]]
      ],
      20
    ]
  ]    // Warning: remember brackets!
],
// HORIZONTAL GROUP WITH  TWO VERTICAL UNITS GROUPS
[C:VDLG_HORZ,"HORIZONTAL UNIT2",10,10,
  [
    [C:VDLG_VERT,"VERTICAL UNIT2",[0.01,0.35],-1,
      [
        [C:VDLG_Int,"IntFld:",6],
        [C:VDLG_StrComE,"StrCOMBOBOX2:",["Gear1","Gear2"],[0]]
      ],
      10
    ],
    [C:VDLG_VERT,"VERTICAL UNIT3",[0.4,1],-1,
      [
        [C:VDLG_Real,"RealFld:",5.6,0,0,3,3],
        [C:VDLG_IntComE,"IntCOMBOBOX2:",[5,7,9],7]
      ]
```

```
          ]

        ]

      ]

    );

// res [0] contains 1 if OK was clicked, or else

IF res[0] THEN

    // READ RESULTS FROM DIALOG:

    result1 = res[1];    //res [1]= Gear3

    result2 = res[2];    //res[2]= TEXT1:

    result3 = res[3];   //res[3]= 17

    result4 = res[4];   //res [4]= [["Test Program",9.1]]

    result5 = res[5];   //res[5]= [[6,"Gear1"],[5.6,7]]

    fullResult=res;     //res=["Gear3","TEXT1:",17,[['Test Pro-
gram",9.1]],[[6,"Gear1"],[5.6,7]]]

    CADH_Message(fullResult);

ENDIF
```

64.5.4.3 Interactions with variable dialogs

It is possible to interact with variable dialogs. Selections in lists, changes in input fields, and selections in lists, can trigger callbacks to a user-defined function. Then, it is also possible to change dialog elements from this callback routine.

You set a local function as a callback via the title input in the variable dialog:

```
res = CADH_VarDialog([[“Title“,PROC(Callback)], Width,
Height, Pitch], [Dialogelement1], [Dialogelement2],
...);
```

The local callback function will now be called up if there are changes in the dialog. The function is declared as follows:

```
PAR res;

PROC Callback
PAR handle, elemNo, event, eventPar;

IF TYP(elemNo)=STRING THEN
    IF elemNo='@combo' AND event=C:CBN_SELCHANGE THEN
        IF eventPar=0 THEN  // own input, enable input
            CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,TRUE]]);
```

```
        ELSE  // disable input, set value to zero
            CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,FALSE],
                                         ['@input1',c:VDLG_ASSIGN,0]]);
        ENDIF
    ENDIF
ENDIF
ENDPROC

res = CADH_VarDialog([['Title',PROC(Callback)], 400, 400 0.4],
                     [[C:VDLG_Real,'@input1'],'Input1:',2],
                     [[C:VDLG_StrCom,'@combo'],'Selection:',
                         ['own input','calculate'],[0],TRUE]);
```

A handle is transferred to the dialog as a code parameter, along with an element identifier, the event, and additional parameters. The possible events are:

Element type	Event	Parameter
Dialog	Initialization WM_INITDIALOG	none
Combobox	Selection CBN_SELCHANGE	Current value
Input field	Leave field WM_KILLFOCUS	Current value
Button	activated BN_CLICKED	none

Either the number of the element according to the index in the results array is transferred as the element number, or the name of the element is transferred. As in the example, a name can be defined by transferring an array, with a type and name in its first element, for the dialog element.

Use this function to access the dialog from the callback routine:

```
CADH_VarDialogAccess(handle, [[elemNo, action,
param],[elemNo, action, param],...]
```

Here, the following actions are permitted:

Action	Description	Parameter
DLG_ASSIGN	Assignment to input field	New value
VDLG_SELECT	Selection in combo box	[position]/value
VDLG_ENABLE	Activate or deactivate	TRUE/FALSE
VDLG_SETFOCUS	Focus on new element	Element's ID

If no action is specified, the value in the input field will be returned. The data is returned as an array, with as many elements as code parameters.

64.5.5 Defining 2D graphics

In KISSsys, you can generate two-dimensional graphics for displaying results which are present in arrays. You can store the definition of the graphic in the data variable expression of the kSys2DPlot graphical element. Bar and line graphics can be displayed in parallel. The definition of the graphic consists of three parts:

- Axis system (1- or 2-axis systems can be defined)
- XY-line graphics
- Bar graphic

Below, each of these parts is described in more detail.

64.5.5.1 The definition of the axis system (af)

At least one axis system must be defined. The second one is optional. The definition for the axis system is as follows:

```
[ | Xaxisname_str , | min_x_r , | max_x_r ] , [ | Yaxisname_str , | min_y_r
, | max_y_r ] , [ axiscolor_str/array , | axiscross_x_r , axiscross_y_r ] ,
[ | scaleinterval_x_r , | scaleinterval_y_r , [ | exponential_x_n , | expo-
nential_y_n ]
```

where :

- **XAxisname**: Name of the X-axis.
- **YAxisname**: Name of the Y-axis.
- **min** : Minimum value of the axis (optional).
- **max** : Maximum value of the axis (optional).
- **axiscolor** : The axis color is defined in a string (red, green, blue, yellow, white, gray, cyan, brown, magenta, purple, black) or as an array [r_n , g_n , b_n] (where r, g, b represent the red, green and blue color values from 0 to 255 (optional).
- **axiscross** : The intersection point of the axes (optional).
- **scaleinterval** : Increment of the axis scaling.
- **exponential** : If 1 is input, the axis will be logarithmically subdivided.

64.5.5.2 The definition of an XY-line graphic (dg_l)

The following information is required for an XY-line graphic:

```
grouptype_n , [ dataarray_x_r ] , [ dataarray_y_r ] , [ | linename_str , |
|linecolour_str/array , | linestyle_n ] , | assignaxis_n
```

where :

- **grouptype** : =1 (for lines graphic).
- **dataarray** : Contains the X- or Y-coordinates of the data.
- **linename** : Name of the element.
- **linecolour** : Line color.
- **linestyle** : Line type (0- solid, 1- interrupted, 2- dashed, 3- semicolon, 4- dash dot dot)
- **assignaxis** : Number 1 or 2 of the coordinates system

64.5.5.3 The definition of a bar chart (dg_b)

For a bar chart, a group of data is defined as follows:

```
grouptype_n , [ dataarray_1_r , ... , |dataarray_n_r ] , [ barco-
lor_str/array ] , | bargroupname_1_str , [ | barelementlabel_1_str , ... ,
barelementlabel_1_str ] , | barclass_n
```

where :

- **grouptype** := 2 (for bar chart).
- **dataarray** : Contains the data for the group.
- **barcolour** : Color of the group's bars.
- **bargroupname** : Name of the group.
- **barelementlabel** : Names for individual elements.
- **barclass** : Display as group (=0) or sorted by elements (=1).

64.5.5.4 The entire definition

The entire definition must begin with the definition of the axis system. After this, you can list any number of definitions for line and bar charts. Each part definition must be enclosed in square brackets, just like the entire definition:

```
[ [af_1] , | [ af_2] , | [dg_l_1] , ..., | [ dg_l_ n1 ] , | [ dg_b_ 1 ] ,
..., [ dg_b_ n2 ]]
```

If lines and bars are to be used simultaneously, a second coordinates system will automatically be applied. This can, however, be changed by the definition of a second coordinates system. An example of the available options is listed as follows:

```
[
   [['x-AXIS'],['y-axis',0],[[40,250,150],[-1000,-10]],[30,20,0,0]],
   [['x-AXIS 2'],['y-axis 2',0],['blue',[0,0]],[30,20,0,0]],
   [1,[-1000,-500,0,500,1000],[5,20,40,55,71],['LINE1','red',0]],
   [1,[-1000,-500,0,500,1000],[2,20,46,60,83],['LINE2',[200,5,150],3]],
   [2,[5,25,16,10,4],['red',3],'group 1'],
   [2,[40,35,25,20,12],['red',3],'group 2']
]
```

The example shows two lines and two groups of bars in two separate coordinate systems.

64.5.5.5 Displaying the graphic

After you have defined the graphic in the `data` variable, you can use the graphical element's `Show` function to display the graphic. Later, you can update it, using the `Refresh` function in the menu or the graphics window.

64.6 Specific functionalities

Some specific calculations are integrated in KISSsys.

64.6.1 Load spectrum calculation

With KISSsys, you can generate an entire gearbox or drive train in a single file. Calculations can be then performed easily for the whole system using KISSsoft modules. If several load cases are present (load spectrum or load cycle), KISSsys can be used to analyze a complete system using this load spectrum.

The user is then able to define load cases for the whole system and perform the calculation for all the components. Lifetime calculations with a load spectrum can be used with the same components as in KISSsoft.

The safety factors and service lives of different gears and bearings can be calculated. Shaft fatigue and static safeties can also be calculated using this load spectrum. The remaining components (which cannot be used to perform a calculation with a load spectrum) are regarded merely as a test for the "weakest" part.

The load spectrum functionality can be used for the following types of calculation:

- Calculation of a gearbox with a user-defined load
- Calculation of a gearbox with a single load stage taken from the load spectrum.
- Calculation of a gearbox with a predefined load spectrum (similar to the ones defined in KISSsoft)

It is also possible to extend the spectrum by adding extra additional values or settings, for example to consider different shift speeds for each bin, which is itself defined by a frequency, torque or power, and a speed.

To perform the calculations, the following KISSsoft modules are also required to perform at least one strength calculation for the elements that need to be calculated:

- Gear service life and safety factors with load spectrum
  ZZ1 (load spectra) including modules Z16, Z16a, Z18 and Z18a
- Shaft calculation with load spectrum
  WA8 (load spectra) including modules W01s and W06s
- Full version of KISSsys for template implementation and setup
  K11c SYS module

64.6.2 Efficiency Calculation

You can use the efficiency calculation in KISSsys to calculate the heat level in a particular gearbox. Several different methods are implemented to enable you to select how the calculation is to be performed, according to the ISO/TR 14179 standard, Part 1 and Part 2.

Thermal analysis can be defined in two sections: power loss and heat dissipation. An external cooler can also be taken into account. The power loss and heat dissipation can be divided up into several sections to enable the effect of all the individual gear unit components to be taken into consideration.

There are two main types of power loss: load-dependent and non-load-dependent. Both types of loss are usually present when a gear unit is operating. Power loss can also be subdivided into gear unit elements, such as gears, bearings and seals. Meshing and churning losses are taken into account for gears, whereas rolling, sliding, seal and drag friction are taken into account for bearings, and seal friction for seals. In some cases, the results must be treated with caution because the calculation methods used may not fully support the type of geometry.

Heat dissipation can be divided into heat dissipation through the housing, base, and rotating parts (input/output shafts and couplings) and cooling oil flow.

You can then simply calculate the total efficiency and the total heat dissipation capacity of a gear unit for a given lubricant temperature, cooler power and input power. You can also specify two of these three entries and calculate the optimum value for the third parameter, which is the value with which you achieve the best heat level for the gear unit. In other words, the value where the heat dissipated equals the heat generated through the power loss.

The difference between Part 1 and Part 2 of the standard is the way in which the different values are entered for the calculations. The main benefit of Part 1 is that it enables you to enter your own heat transfer coefficient for the heat dissipated through the housing (if it has a very specific shape), whereas, in Part 2, this coefficient is calculated using an approximation of the shape of the housing. The main benefit of this part is that it also takes fins, bases and rotating parts into consideration when calculating the amount of heat that is dissipated.

64.6.3 Taking into account housing deformation in static KISSsys calculations

64.6.3.1 Introduction

The inclusion of housing deformation in KISSsys calculations is based on the use of a reduced stiffness matrix for the housing, as calculated by the Finite Element Method (FEM). This reduced stiffness matrix should include the nodes that refer to the center position of the bearings that connect the shafts of the gearbox to the housing.

64.6.3.2 Main calculation steps

The calculation steps that must be followed to perform this kind of analysis are summarized next. Please note that the process to generate the reduced stiffness matrix is not described, since it is different for each FEM computer program used. For more information on this, please refer to your FEM program manuals.

- Step 1: Import the stiffness matrix and the FEM nodes coordinates

The first step is to read the stiffness matrix and the FEM node coordinates. This is achieved by calling the relevant function in the housing element, in the KISSsys model (right-click, select ImportStiffnessMatrix). Both the stiffness matrix and the nodes coordinates should be positioned in the same file, together with some information on the system of units used. An example of such a file (that can now be handled by KISSsys) is given next:

```
Unit System      (1 = SI, 2 = CGS, 3 = BFT, 4 = BIN, 5 = MKS, 6 = MPA, 7 = uMKS)

Active Unit System = 1

--------------------------------------------------------------------------------

MASTER NODE POSITION

        Number    x-coord        y-coord        z-coord

          1       *******        *******         *******

          2       *******        *******         *******

**********Stiffness matrix data as exported by the FEM program**********

                                                          *******
```

More information on the supported FE programs and the format requirements can be found in the relevant instructions document (available on request).

- Step 2: Position the housing correctly in the KISSsys model

  Since the FEM model and the KISSsys model may not have a matching coordinate system (CS), you should then position the housing correctly in the KISSsys model. To do so, right-click on the housing element again, and select the ResetPosition function. In the next dialog, you can either input the origin and orientation of the housing CS directly, or use the ThreePointPositioning function. To do so, select three points (e.g. bearings) in the KISSsys model from a drop-down list, and then enter the coordinates of the same three points in the housing CS. Make sure that these three points are not collinear. This procedure returns the housing CS with respect to the KISSsys CS. A visual check of the correct positioning can be also done by reading a simplified step file of the housing (e.g. only wireframe) and showing it in the KISSsys 3D viewer. To import a step file into the housing, you can right mouse-click on it and select Dialog. We recommend you use a simplified version to avoid overloading the KISSsys model. Finally, you can also decide how the FEM nodes are displayed on the KISSsys model, by right-clicking on the housing element and selecting ShowNodes in the context menu. Every time the orientation of the housing is changed, you must select ShowNodes again, to update the nodes view in KISSsys. If you click on ShowNodes, you also see the IDs of the displayed FEM node, which makes it easier to validate the positioning. At this point, we must mention that you do not need to add a step model to the housing element, although you can use this model as an additional aid for validating the correct positioning.

- Step 3: Perform

  Click the housing calculation button to start the analysis. The first step in the calculation is the mapping of the FEM nodes on KISSsys bearings. The program gives a message of any nodes that could be mapped and their distance to the closest bearing. At this point, you need to know if the rejected nodes actually correspond to bearings, or not. You can then decide whether to continue with the calculation or cancel it. One possible reason for not mapping bearings of interest to nodes, is the wrong positioning of the housing in the KISSsys CS. If this is the case, then the previous step must be repeated. If this is not the case, and the difference between FEM nodes and bearings (as reported in the mapping message) is not that great, you can change the tolerance that is used in the mapping process. This can happen for example in the case that the FEM node is not positioned in the middle of a bearing, but at its edges. The tolerance used in the mapping can be changed from the housing properties in KISSsys

(right mouse click on the housing and select Properties window). There, the tolerance is given in millimeters.

If you continue the calculation, the program reduces the stiffness matrix to the part that corresponds to the mapped nodes. Nodes that have not been not mapped to bearings are ignored. The calculation also ignores any predefined offsets and tilting values given previously in the bearings and sets them to zero. The algorithm performs all KISSsoft calculations and derives the forces on the bearings. From these forces the program calculated the offsets and tilting on the bearings (using the FEM stiffness matrix). Then, using the resulting offsets, the KISSsoft calculations are run again, resulting possibly on new bearing forces and offset values. This procedure is continued iteratively until there is convergence between successive forces and offset calculations. There may be cases where due to a housing with small stiffness, the maximum number of iterations is reached. If you cancel the calculation, the system will display the percentage difference between the two last iterations and apply the results from the last iteration. You can set the maximum number of iterations in the housing element's properties (right-click on the housing element, and select Properties). The relevant property is called "`maxNumberOfIterations`". You should input a number greater than 4 to ensure the algorithm finds a useful solution. After the calculation has completed, you can perform a range of other investigations, for example, analyzing the gearing contact to determine the effect of housing stiffness on the parameters used to size the gear unit.

You can also use more than one housing, each with a different stiffness matrix, in the KISSsys model. In this case, the program prompts you to select the housing it should use, before starting the calculation. This can be very useful if you want to compare the effect of different housing designs on the gear unit design. The results for each housing calculated using this method are then stored in the housing element. These results can then be viewed again by clicking on the housing element's `RestoreOffsetResults` function (right-click on the housing). The following functions for handling displacement are also available (the tolerances remain the same).

- ResetBearingOffsets: reset all bearing offset values to zero
- SaveBearingOffsets: save the current displacement values.
- RestoreBearingOffsets: recover the saved displacement values.

64.6.4 Modal analysis of shaft systems

64.6.4.1 Introduction

Users can use the modal analysis of shaft systems function in KISSsys to calculate the eigenfrequencies and eigenmodes of a complete shaft system, including the effect of the gear connection between shafts. Performing a modal analysis for individual shafts is not realistic. It must be performed for the entire topology of the shaft system. The necessary calculation steps, together with important restrictions are given next.

64.6.4.2 Calculation procedure

To calculate the system dynamics, you must import a ShaftSystem KISSsys calculation in the model. Right-click, and select Modal analysis from the context menu, to open a dialog in which you can set various parameters for the calculation. You must define the number of eigenfrequencies that must be calculated, and specify whether only torsional or all vibration types are to be included, whether gyroscopic effects are to be taken into account (only applies to torsional vibrations), and which method is to be used to calculate gear mesh stiffness. The following selections are available for this last option:

- According to ISO 6336, where the tooth contact stiffness as described in this standard is used.
- Using the KISSsoft Contact Analysis (CA) algorithm, where a full contact analysis is performed in the gear connections. If KISSsoft does not have a CA calculation for a particular gear pair type, or the gear pair transfers no power, the ISO 6336 process is used for that specific pair.
- Infinite, and the tooth contact stiffness is assumed to be infinite. This option can be selected if you want to check limiting conditions.
- Ignore, and the tooth contact stiffness is assumed to be zero, and hence there is no connection between the vibrating shafts (each one is vibrating independently).

All the dynamic calculation properties mentioned above are also available in the calculation's Properties window (right-click on Calculation and select Properties window).

64.6.4.3 Results

After the calculation is finished, a new tab opens, in which a 3D animation of the vibrating system can be displayed. There, you can select the eigenfrequency to view and also define the animation speed and the scaling of the deformation. The eigenfrequency values and tooth contact stiffness used for each gear pair are also displayed in the system dynamics report, together with other useful analysis results and a 2D plot diagram. To display this report, right-click on `ShaftSystem calculation` and then select `ShowReport`. If necessary, click on the `SavePlot` button to save the 3D plot, if it is to be used (unchanged) in subsequent calculations, (so that a new plot is generated each time). After the calculation is finished, the program also generates a table that shows the mode shape data of all the shafts in the system.
Please note that the only gears displayed in the animation window are those that belong to a shaft calculation file. Nevertheless, all the gears are taken into account in the modal analysis.

Finally, also note that, if a modal analysis is performed for a planetary system, this does not take into account the effect of the rotating planets' position on the system bending stiffness. This is in agreement with the quasi-static calculation procedure followed normally in eigenfrequencies analysis.

64.6.5 Campbell diagram for shaft systems

64.6.5.1 Introduction

A Campbell diagram can be used to investigate the effects of shaft speed on the eigenfrequencies. In this way, the critical eigenfrequencies can be determined for each speed or multiple of that speed.

64.6.5.2 Calculation procedure

To run a calculation for shaft systems with a Campbell diagram, click on the `ShaftSystem` calculation element in KISSsys. Right-click on the element, and then select the `CampbellDiagram` option. The Campbell diagram dialog contains all the necessary entries. You can select the reference shaft for the calculation from a list of shafts in the system that include a coupling with a defined boundary condition. In this dialog, you can also select the calculation method for calculating the gears and the speed range of the reference shaft. You can define the various different speeds, together with the number of eigenfrequencies, that are to be taken into account in the Campbell diagram. Finally, you can select the number of resonance curves that are to be drawn in the Campbell diagram. The calculation

starts with a kinematic analysis of the system for each speed of the reference shaft. The speeds of all of the shafts are updated and then a modal analysis is performed for each of these speeds.

64.6.5.3 Results

Once the calculation is finished, you can view the 2D plot of the Campbell diagram directly in KISSsys. A more detailed 2D display, and a number of other useful analysis results, appear in the report which is generated when you right-click on `ShaftSystem Calculation` and then select `ShowReport`. All the calculation data is also available in the results table that is generated in KISSsys. You can also click on the "SavePlot" button to save the 2D plot, if it is to be used (unchanged) in subsequent calculations, (so that a new curve is generated each time).

64.6.6 Analysis of unbalance response of shaft systems

64.6.6.1 Introduction

The unbalance response analysis functions can be used to calculate the real dynamic behavior of a shaft system that is subjected to dynamic loads (unsymmetrical masses and/or deviations in path of contact). The calculated behavior includes deformation, rotation, forces and moments. The necessary inputs and the results achieved by the calculation are described below.

64.6.6.2 Calculation procedure

To call the unbalance response analysis, click on a `ShaftSystem calculation element` in KISSsys. Right-click on the element, and then select the `Unbalance response` option. The next dialog you see contains all the inputs required to perform the calculation. You can select the reference shaft for the calculation from a list of shafts in the system that include a coupling with a defined boundary condition. You can then select the X-axis of the unbalance response diagram for the calculation (and therefore also define the type of calculation to run). Two options are available here:

- **Reference shaft speed.** The reference shaft speed is modified within the range you have specified (min/max speed) with the predefined number of steps. A kinematic calculation is performed for each speed in the entire shaft system and the speed of all the shafts is calculated. These speeds are then used to calculate the dynamic loads, which are then applied to the model. The result is the unbalance response in the specified reference position on the shaft.
- **Y-coordinate of the reference shaft.** In this case, the length of the reference shaft is subdivided into the predefined number of sections, and the unbalance response calculation is performed for the specified speed. This results in the exact shape of the reference shaft at this speed.

You can then also select the calculation method you want to use to calculate meshing stiffness, which is similar to the modal analysis calculation.

The effect of speed on the stiffness of rolling bearings (only for bearings with internal geometry) can also be taken into account. If this option is selected, a static calculation is performed for each speed, and the bearing stiffness used in the dynamic analysis is modified accordingly.

Finally, you can also define the damping for torsion, axial and bending vibration in this dialog. Note that the viscous damping of bearings must be defined separately for each bearing in the shaft calculation (freely definable units) or in its properties in KISSsys (SI units).

64.6.6.3 Results

Once the calculation is complete, a 2D plot is generated from the data you have entered. More detailed analysis results and other plot data are displayed in the report which is generated when you right-click on the `ShaftSystem` KISSsys element and then select `ShowReport`. You can also click on the `SavePlot` button to save the 2D Plot, if it is to be used (unchanged) in subsequent calculations, (so that a new curve is generated each time). A table containing all the data used in the plot is also generated. Note that this method not only calculates results at the refe-

rence position of the reference shaft (as defined in the input data for the unbalance response analysis), but also calculates analysis results for all the documentation points that are defined in the system's shaft calculations. These documentation points can therefore be used as measuring points for dynamic behavior. The results of the documentation points are displayed both in the report and in the results table.

Part X

Bibliography and Index

[1] A.G.M.A.: Fundamental Rating Factors and Calculation Methods for Involute Spur and Helical Gear Teeth. Standard 2001-B88, 1988.

[2] Akahori, H., Sato, Y., Nishida, T., Kubo, A.: Prove di durata di Face Gear (Testing the durability of face gears). Organi di trasmissione, 2002, No. 12 or MTP2001-Fukuoka, The JSME Int. Conference, Japan, 2001.

[3] Basstein, G., Sijtstra, A.: Neue Entwicklung bei Auslegungen und Fertigung von Kronenrädern. Antriebstechnik, 32 (1993), No. 11.

[4] Bock G., Nocj R., Steiner O.: Zahndickenmessung an Getriebeschnecken nach der Dreidrahtmethode. Physikalisch-Technische Bundesanstalt, Braunschweig, 1974.

[5] Decker, K.H.: Maschinenelemente. Carl Hanser Verlag München, 10th Edition, 1990.

[6] Dietrich G., Stahl H.: Matrizen und Determinanten in der Technik. VEB Verlag Leipzig, 5th Edition, around 1960.

[7] DIN ISO 15312: Wälzlager - Thermisch zulässige Betriebsdrehzahl - Berechnung und Beiwerte, October 2004.

[8] DIN 732: Wälzlager - Thermische Bezugsdrehzahl - Berechnung und Beiwerte, May 2010.

[9] DIN 743: Tragfähigkeitsberechnung von Wellen und Achsen, December 2012.

[10] DIN 867: Bezugsprofile für Evolventenverzahnungen an Stirnrädern (Zylinderrädern) für den allgemeinen Maschinenbau und den Schwermaschinenbau, February 1986 Issue.

[11] DIN 2091: Drehstabfedern mit rundem Querschnitt: Calculation and Design. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[12] DIN 2092: Tellerfedern: Berechnung. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[13] DIN 2093: Tellerfedern: Masse, Qualitätsanforderungen. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[14] DIN EN 15800: Zylindrische Schraubenfedern aus runden Drähten: Quality specifications for cold coiled compression springs. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[15] DIN 2096: Zylindrische Schraubenfedern aus runden Drähten und Stäben: Gütevorschrift für warmgeformte Druckfedern. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[16] DIN 2097: Zylindrische Schraubenfedern aus runden Drähten: Gütevorschriften für kaltgeformte Zugfedern. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[17] DIN 2194: Zylindrische Schraubenfedern aus runden Drähten und Stäben: Kaltgeformte Drehfedern (Schenkelfedern), Gütenorm. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011.

[18] DIN 3960: Begriffe und Bestimmungsgrössen für Stirnräder und Stirnradpaare mit Evolventenverzahnung. December 1987 Issue.

[19] DIN 3961: Toleranzen für Stirnradverzahnungen, Grundlagen, 1978.

[20] DIN 3967: Flankenspiel, Zahndickenabmasse, Zahndickentoleranzen, 1978.

[21] DIN 3971: Begriffe und Bestimmungsgrössen für Kegelräder und Kegelradpaare, July 1980 Issue.

[22] DIN 3975: Begriffe und Bestimmungsgrössen für Zylinderschneckengetriebe mit Achsenwinkel 90 Grad, July 1976 Issue.

[23] DIN 3990: Tragfähigkeitsberechnung von Stirnrädern. Parts 1, 2, 3, 4, 5, 11 and 21. December 1987 Issue.

[24] DIN 3991: Tragfähigkeitsberechnungen von Kegelrädern, 1990.

[25] DIN 5480: Zahnwellen-Verbindungen mit Evolventenflanken. Parts 1 to 15. March 1986.

[26] DIN 6885: Passfedern. Sheet 1-3, 1968.

[27] DIN 6892: Passfedern - Berechnung und Gestaltung, 2012.

[28] DIN 7151: ISO Grundtoleranzen für Längenmasse bis 500 mm, 1964.

[29] DIN 7190: Berechnung und Anwendung von Pressverbänden. February 2001.

[30] DIN EN 13906-1: Compression springs: Calculation and Design. DIN Taschenbuch 29, 12th Edition, Beuth Verlag Berlin, 2015.

[31] DIN EN 13906-2: Tension springs: Calculation and Design. Beuth Verlag Berlin, 2013.

[32] DIN EN 13906-3: Turnsprings: Calculation and Design. DIN Taschenbuch 29, 12th Edition, Beuth Verlag Berlin, 2015.

[33] ISO 7902: Hydrodynamic plain journal bearings under steady-state conditions. Part 1, 2013, Part 2 and 3, 1998.

[34] DIN 31653: Hydrodynamische Axial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 1991.

[35] DIN 31654: Hydrodynamische Axial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 1991.

[36] DIN 58400: Bezugsprofil für Evolventenverzahnungen an Stirnrädern in der Feinwerktechnik. June 1984 Issue.

[37] DIN 58405: Abmasse für die Feinwerktechnik, Part 2.

[38] Dubbel, H.: Taschenbuch für den Maschinenbau. Springer Verlag Berlin, 15th Edition, 1986.

[39] Schaeffler Technologies AG: Wälzlagerpraxis, Handbuch zur Gestaltung und Berechnung von Wälzlagerungen. Vereinigte Fachverlage GmbH Mainz, 2015.

[40] FAG: Standard program. Catalog WL 41510, 3rd Edition 1995.

[41] FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile. VDMA Verlag Frankfurt am Main, 6th Edition, 2012.

[42] Hänchen, R., Decker, K.H.: Neue Festigkeitslehre für den Maschinenbau. Carl Hanser Verlag, Munich, 3rd Edition, 1967.

[43] Hirn, H.: Computergestützte Zahnradoptimierung. Fink GmbH, Druck und Verlag Pfullingen, 1999.

[44] ISO 6336: Calculation of load capacity of spur and helical gears. Parts 1, 2, 3, 4, 5, 6. 2006 Edition.

[45] ISO/DIS 10300: Calculation of load capacity of bevel gears. Parts 1, 2, 3. ISO 10300 2001 Edition, ISO/DIS 10300 Draft 2011.

[46] Kissling, U.: KISSsoft - eine praxisgerechte Maschinenelemente-Software. Antriebstechnik 27, 1988, No. 12, pp. 34-40.

[47] Kissling, U.: Auslegung von Maschinenelementen. CIM Management 11 4, 1995.

[48] Kissling, U.: Technische Berechnungen auf Personal Computern. VDI-Z 130, 1988, No. 5, pp. 45-52.

[49] Kissling, U.: Sicher dimensioniert. Antriebstechnik 6, 2007. Pp. 64-68.

[50] Kissling, U., Beermann, S., Hirn, T.: Kronenräder: Geometrie und Festigkeit, Antriebstechnik 10, 2003.

[51] Klingelnberg in-house standard 3028: Auslegung eines Kegelradgetriebes ohne Achsversatz. Issue No. 2.

[52] Klingelnberg in-house standard 3029: Auslegung eines Kegelradgetriebes mit Achsversatz. Issue No. 2.

[53] Klingelnberg in-house standard 3030: Tragfähigkeits-Berechnung für Spiral-kegelräder. Issue No. 1.

[54] Klotter, K.: Technische Schwingungslehre, Volume 2. Springer Verlag Berlin, 2nd Edition, 1960.

[55] Kollmann, F.: Welle-Nabe-Verbindungen. Springer Verlag Berlin, 1984.

[56] Lachenmaier, S.: Auslegung von evolventischen Sonderverzahnungen für schwingungs- und geräuscharmen Lauf von Getrieben. VDI Verlag Düsseldorf, WZL Range 11 No. 54, 1983.

[57] Lang, O., Steinhilper R.: Gleitlager. Konstruktionsbücher Volume 31, Springer Verlag Berlin, 1978.

[58] Linke, H.: Stirnradverzahnung. Carl Hanser Verlag, Munich, 1996.

[59] MAAG: Pocket Book. 2nd Extended Edition, Zürich, 1985.

[60] Massa, E.: Costruzione di macchine. Editori Masson Italia, Milan, 1981.

[61] Matek, W., Muks D., Wittel H.: Roloff/Matek Maschinenelemente. Vieweg Verlag, Brunswick, 11th Edition, 1987.

[62] Matek W., Muks D., Wittel H., Becker M., Jannasch D.: Roloff/Matek Maschinenelemente. Vieweg Verlag, Brunswick, 15th Edition, 2001.

[63] Matthias, K.: Schraubenkräfte in einer Flanschverbindung. Maschinenbau, Berlin 34 (1985) 11, p. 517.

[64] Niemann G.: Maschinenelemente, Volume 1. Springer Verlag Berlin, 2005.

[65] Niemann G.: Maschinenelemente, Volume 2. Springer Verlag Berlin, 1983.

[66] Niemann G.: Maschinenelemente, Volume 3. Springer Verlag Berlin, 1985.

[67] NIHS 20-25: Uhrenindustrie, Schweizer Norm SN 282 025, October 1993.

[68] Obsieger: Tooth form factors used for external and internal teeth Zeitschrift Konstruktion 32 (1980), p. 443-447.

[69] Weber C., Banaschek K.: FVA-Bericht 129 und 134, Elastische Formänderung der Zähne und der anschliessenden Teile der Radkörper von Zahnradgetrieben, FVA 1955.

[70] Rules for The Classification of Naval Ships (FREMM 3.1), Bureau Veritas, March 2004.

[71] SKF: Main Catalog 4000 T. 1989 Edition.

[72] Spinnler, Prof.: Manual de calcul d'organes des machines. EPFL Lausanne, 1990.

[73] VDI 2226: Festigkeitsberechnung metallischer Bauteile.

[74] VDI 2227: Festigkeitsberechnung.

[75] VDI 2230: Systematische Berechnung hochbeanspruchter Schraubenverbindungen, Sheet 1. December 2014.

[76] VDI 2545: Zahnräder aus thermoplastischen Kunststoffen. 1981 Edition.

[77] KISSsoft: Klassische Anleitungen zu den Berechnungsmodulen: KISSsoft Gear Pump Analysis, Hombrechtikon, 2005.

[78] Boresi, A.P., Schmidt R.J.: Advanced Mechanics of Materials, 6th. Edition, John Wiley and Sons, Inc., 2002.

[79] Roth, K.: Zahnradtechnik - Evolventen-Sonderverzahnungen zur Getriebeverbesserung, Springer DE, 1998.

[80] Hoechst, High Chem: Technische Kunststoffe - Berechnen, Gestalten, Anwenden, B.2.2, Hoechst AG, 1992.

[81] Theissen, J.: Berechnung der Sicherheit gegen Graufleckigkeit von Industriegetrieben auf der Grundlage des neuen Rechenverfahrens nach FVA 259. Dresdner Maschinenkolloquium, TU Dresden, Sept. 2003. Tagungsband pp. 195-212.

[82] FVA-Informationsblatt Nr. 54/7: Testverfahren zur Untersuchung des Schmierstoffeinflusses auf die Entstehung von Graufleckigkeit bei Zahnrädern, FVA Vereinigung, Frankfurt, 1999.

[83] Feulner, R.: Verschleiss trocken laufender Kunststoffgetriebe, Lehrstuhl Kunststofftechnik, Erlangen, 2008.

[84] DIN 32711: Welle-Nabe-Verbindung - Polygonprofil P3G. March 2009 Issue.

[85] DIN 32712: Welle-Nabe-Verbindung - Polygonprofil P4C. March 2009 Issue.

[86] Decker, K.-H.: Maschinenelemente, Funktion, Gestaltung und Berechnung, Hanser Verlag Munich, 2001.

[87] Klingelnberg, J.: Kegelräder Grundlagen, Anwendungen, Springer Verlag Berlin Heidelberg, 2008.

[88] Norden, N.: On the compression of a Cylinder in Contact with a Plane Surface, National Bureau of Standards, 1973.

[89] Annast, R.: Kegelrad-Flankenbruch, Technical University Munich, 2002.

[90] VDI 2241: Schaltbare fremdbetätigte Reibkupplungen und -bremsen, Sheet 1: 1982; Sheet 2: 1984.

[91] Burdick, R.: Manufacturing Single-Enveloping Worm Gear Sets, Gear Solutions, April 2003.

[92] Harris, T., Rumbarger, J.H., Butterfield, C.P.: Wind Turbine Design Guideline DG03: Yaw and Pitch Rolling Bearing Life. 63 pp.; NREL Report No. TP-500-42362, 2009.

[93] DET NORSKE VERITAS: Calculation of gear rating for marine transmissions, Norway, 2003.

[94] Haibach, E.: Betriebsfestigkeit, Verfahren und Daten zur Bauteilberechnung, 2nd Edition, Springer Verlag 2002.

[95] Winter, H., Podlesnik B.: Zahnfedersteifigkeit von Stirnradgetrieben, Parts 1 to 3, Antriebstechnik 22 1983.

[96] Harris, T.: Rolling Bearing Analysis, 4th Edition, John Wiley & Sons Inc., 2001.

[97] Schlecht, B: Maschinenelemente 2, Getriebe - Verzahnungen - Lagerungen, Pearson Studium 2010.

[98] Tsai, S.-J., Wu, S.-H.: Geometrical Design of Conical Gear Drives with Profile-shifted Transmission; 12th IFToMM World Congress, 2007.

[99] Tsai, S.-J., Wu, S.-H.: Designing Skew Conical Gear Drives in Approximate Line Contact for Power Transmission; Proceedings of MPT2009-Sendai JSME International Conference on Motion and Power Transmission, 2009.

[100] DIN 31657: Hydrodynamische Radial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 2015.

[101] Pech, M.: Tragfähigkeit und Zahnverformung von Schraubenradgetrieben der Werkstoffpaarung Stahl/Kunststoff, Lehrstuhl für Maschinenelemente, Getriebe und Kraftfahrzeuge, Bochum, 2011.

[102] VDI 2736: Thermoplastische Zahnräder, Sheets 1 to 4, 2014/2015

[103] DIN 31652 1-3: Hydrodynamische Radial-Gleitlager im stationären Bereich. Beuth Verlag Berlin, 2017.

[104] Masudo, T.; Abe, T.;Hattori, K.: Prediction Method of Gear Noise Considering the Influence of the Tooth Flank Finishing Method. Journal of Vibration, Acoustics, Stress, and Reliability in Design, JANUARY 1986, Vol. 108 / 95.

[105] Bertsche, B.: Reliability in Automotive and Mechanical Engineering; Berlin, Heidelberg: Springer Verlag, 2008.

[106] Pestel, E.C.; Leckie, F.A.: Matrix Methods in Elastomechanics; McGraw-Hill, 1963.

[107] Johnson, K.L.: Contact Mechanics, Cambridge University Press, 1987

[108] F.E.M. 1.001; Rules for the design of hoisting appliances: Booklet 2. Fédération Européenne de la Manutention, 1998.

[109] VDI 2736: Thermoplastische Zahnräder, Sheets 1-4, 2016.

[110] Ribeiro, M.: Gaussian Probability Density Functions: Properties and Error Characterization, 2014.

[111] Zaka, A.; Akhter, A.: Methods for Estimating the Parameters of the Power Function Distribution, 2013.

1

12. Calculating the required amount of lubricating oil - II-280

2

2D geometry - II-636

3

3D export - II-665
3D geometry - I-196, II-646
3D interface to Autodesk Inventor - I-217
3D interface to CATIA - I-250
3D interface to Creo Parametric (ProEngineer) - I-237
3D interface to Solid Edge - I-209
3D interface to SolidWorks - I-201
3D interface to Unigraphics NX - I-223
3D interfaces - I-190
3D view - IX-1167

A

Abbreviations used in gear calculation - II-691
Accuracy grade bevel gears - II-531
Accuracy of the tooth form - II-620
Activating offset of load application center point - III-743
Add your own texts in the results window - I-111, I-258
Addendum angle and root angle - II-489, II-491
Addendum angle gear 2, dedendum angle gear 2 - II-519
Addendum coefficient gear 1 (middle), addendum coefficient gear 2 (middle) - II-519
Addendum reduction - II-549, II-555
Add-in (menu options in CAD) - I-218, I-225
Add-in functions (calls) - I-206, I-215, I-221
Adding additional tabs and dialogs - I-102
Adding and deleting files - I-96
Adding manufacturing data - I-201, I-206, I-209, I-215, I-217, I-221
Adding manufacturing data to the drawing - I-231
Adding new bolt types to the database - IV-1041
Adding tip chamfer - II-394
Adding tip rounding - II-394
Adding variables to tables - IX-1175
Additional inputs for DIN 6892 Method B - IV-896
Additional settings in the - VIII-1155

Additional strength calculation of all variants - II-449
Adhesives - I-146
AGMA 6101-E08/AGMA 6001-E08 - III-764
AGMA 925 - II-432, II-635
Allow large profile shift - II-458
Allow simplified calculation according to DIN 3990/ISO 6336 - II-459
Allowances for racks - II-677
Alternating bending factor - II-309, II-315, II-506
Ambient density - III-707
Amplitude of contact stiffness - II-657
Amplitude of transmission error - II-655
Analog to DIN 3991, Method B - II-552
Analogous to ISO 10300, Method B - II-551, II-552
Analysis of unbalance response of shaft systems - IX-1210
Angle error - II-619
Angle modifications - II-492
Angle of flank normal - II-643
Angle of rotation-controlled tightening - IV-1011
ANSI 92.1 and ISO 4156/ANSI 92.2M - IV-945
Answers concerning geometry calculation - II-669
Answers to Frequently Asked Questions - I-253, II-668, III-860, IV-1040
Answers to questions about strength calculation - II-678
Application factor - II-282, II-291, II-507, II-552, II-571, II-594, IV-882, IV-899, IV-909, IV-920, IV-937, IV-951, IV-964, VI-1092
Application factor and summand for operational behavior - VI-1086
Application factor f1 - VI-1078
Arc-like end relief I and II - II-373
Arc-like profile modification - II-366
Areas of application for the FKM guideline - VIII-1111
Assembly - II-643
Assumptions made for the calculation - V-1063
Automatic - II-389
Automatic calculation of load factor q - IV-1037
Automatic calculation of the dishing angle ψ - IV-1038
Automatic change of reference profiles - II-673
Automotive - VII-1096
Average surface pressure - III-843
Axial clearance - III-746
Axial offset - II-546
Axial spanning with nut - IV-883
Axial/transverse module - II-563
Axis alignment - II-632, II-653

B

Background - VIII-1113
Base pressure angle of contact analysis - II-420
Base size - III-708

Basic data - II-264, II-423, II-484, II-543, II-563, II-584, II-606, III-731, III-832, IV-933, IV-1035
Materials - I-154
Basic installation - I-43, I-45, I-256
Basic material, glued and soldered joints - I-145
Basic materials - IV-1029
Beam profiles - I-153
Bearing - III-703, III-725
Bearing (general) - III-725
Bearing application factor - II-535, II-554
Bearing data - III-826
Bearing data tab - III-824
Bearing force curve and direction of the bearing forces - II-657
Bearing manufacturer - III-749
Bearing power loss - II-573
Bearing ring deformations - III-827
Bearing width - III-851
Bearings coefficient - V-1047
Bearings with radial and/or axial force - III-819
Belt length - VI-1079
Belt length and number of teeth on belt - VI-1087
Belts and chain drives - VI-1075
Bending critical speed - III-756
Bending stress values - V-1062
Bevel and Hypoid gears - II-480
Bevel gear - generating a 3D model - I-198
Bevel gear factor at flank and root - II-536
Bevel gears
– Determine permitted overloads - II-685
Bevel gears with cyclo-palloid® gear teeth - II-527
Bevel gears with palloid toothing - II-529
Beveloid Gears - II-604
Bibliography - II-289, II-311, III-832, VIII-1131, VIII-1156, X-1214
Bibliography and Index - X-1213
Bolt data - IV-995, IV-999
Bolted joint under axial and shearing force - IV-984
Bolted joint under axial load - IV-985
Bolts - IV-980
Bore - I-150
Coefficients of friction classes - I-151
flat washer - I-152
Nuts - I-152
Nuts strength class - I-151
Strength class - I-151
Thread type - I-152
Tightening factor - I-150
Type - I-152
Bolts and Pins - IV-973
Bolts/pins - I-148
Boundary safety coefficient - IV-1025

Bracket connection - IV-1032
Buckling - III-732, III-757

C

Calculate flank safety with 0.85*b (ISO 10300) - II-506
Calculate form diameter from tooth form - II-466, II-670
Calculate lubrication factor with oil temperature - II-469
Calculate moment of inertia from tooth form - II-468
Calculate number - V-1069
Calculate scuffing - II-311
Calculating a pinion type cutter - II-400
Calculating and generating a report - I-92, I-112
Calculating axial forces on bearings in face-to-face or back-to-back arrangements - III-820
Calculating cylindrical gears manufactured using tools specified in DIN 3972 - II-671
Calculating force on bearings with a contact angle - III-754
Calculating planet carrier deformation with FEM - II-336
Calculating Shafts - III-730, III-750
Calculating spline connections according to DIN 5480 with diameter centering - IV-931
Calculating the displacement volume of gear pumps - II-468
Calculating the internal temperature and the flash temperature - II-312
Calculating the reference profile - II-400
Calculating the Safeties - III-773
Calculating the size coefficients for small gears - II-470
Calculating the thermal nominal speed - III-805
Calculating the thermally permissible operating speed limit - III-807
Calculating wear on worm wheels according to Pech - II-591
Calculation - III-856, IV-880, VII-1101, VIII-1139
Calculation according to AGMA 421.06 (High Speed Gears) - II-687
Calculation according to Klingelnberg, Gleason and Oerlikon - II-482, II-500
Calculation according to Schaeffler 2014 (INA, FAG) - III-813
Calculation according to SKF Catalog 1994 - III-809
Calculation according to SKF Catalog 2013 - III-811
Calculation elements - IX-1182
Calculation method - III-760
Calculation method for friction - III-748
Calculation methods - II-281, II-363, II-501, II-550, II-688, III-840
Calculation of flank safety - II-460, II-462
Calculation of KHβ with manufacturing errors - II-330
Calculation of spline connections as described in DIN 5480 with diameter centering - IV-930
Calculation of the wear coefficient kw for steel - II-278, II-662
Calculation of volume-specific heat - III-858
Calculation procedure - III-791, VIII-1148, IX-1208, IX-1209, IX-1210
Calculation reports - I-112
Calculation type - III-764, III-765
Calculation using Methods B or C (DIN 3990, 3991) - II-679

Calculation using your own S-N curve (Woehler lines) - II-467
Calculation variables - I-120
Calculation with enhanced formulae (differs from standard) - II-580
Calculation with experimental data - III-771
Calculation with normal module instead of axial module - II-563, II-579
Calculation with operating center distance and profile shift according to manufacture - II-468
Calculations - II-348, II-349, II-350, II-352, II-466, II-533, II-579
Calculations - IV-891
Campbell diagram - III-789
Campbell diagram for shaft systems - IX-1209
Case I. Test results measured on identical test gears - VIII-1153
Case II. Test results available from gears with different gear geometry - VIII-1154
Center distance - II-265, II-267, II-564, II-585, VI-1078, VI-1086
Center distance - VI-1093
Center distance tolerances - II-360
Center distance tolerances - I-144
Chain Drives - VI-1090
Chain profiles ISO606 - I-146
Chain type - VI-1091
Change to the parameter for generation - I-209
Changing base settings in the interface - I-240, I-248
Changing the output of angles in reports - I-254
Characteristic number - II-531
Check changes in safeties if the center distance changes - II-689
Checking the contact pattern - II-603, II-638
Checking the meshing - II-619
Circular pitched teeth - II-398
Clamped connections - IV-890
Classic method - IX-1171
Classification of bearings - III-793
Classification of load spectra according to F.E.M. guideline - II-311
Coefficient for minimum tip clearance - II-459
Coefficient for minimum tooth thickness at tip - II-459
Coefficient of friction - IV-870, IV-879
Coefficient of friction - IV-1010
Coefficient of friction for hypoid gears - II-533
Coefficient of thermal expansion for housing - II-428
Coefficients used to determine fatigue strength - III-772
Columns - I-105
COM interface - I-181
Comment field - I-81
Comments - I-106, I-119
Comparing types - III-798
Comparison of an FEM calculation with crossed axis helical gear wheel calculation - II-688
Composites as defined in DIN 58405 - II-672
Compression Springs - V-1045
Compression springs standard - I-144
Condition query IF ELSE END - I-90, I-123

Conditions for using shaft calculation files - II-341
Conditions I - II-440, II-518
Conditions I/II - II-450
Conditions II - II-442, II-519
Conditions III - II-520
Cone angle - II-607
Configuration - II-346
Configuration - VI-1092
Configuration tool - I-62
Configuring tensioning pulleys - VI-1078, VI-1085
Conical Interference Fits - IV-878
Conicity - IV-886
Connecting rolling bearing - III-729
Connection elements - III-728, IX-1185
Connection, general - III-729
Connections - IV-865
Consider gyroscopic effect - III-740
Consider weight - III-740
Constraints data - IV-1007
Constraints on various bearings - III-726
Constructed involute - II-353
Contact analysis - II-382, II-411, II-472, II-510, II-537, II-653, II-671
Contact analysis model for planetary systems - II-419
Contact analysis with load spectra - II-422
Contact analysis/face load factor - II-472
Contact line (face gear) - II-651
Contact lines on tooth flank - II-655
Contact temperature - II-648
Context menu - I-78, I-81, II-388
Convert external pressure with multiple interference fit - IV-873
Converting or inputting Gleason toothing data - II-497
Coupling - III-723
Coupling the individual slices - II-419
Create a new bolt type - IV-1041, IV-1043
Creating and modifying tables - IX-1174
Creating Models in KISSsys - IX-1170
Creating, opening and closing projects - I-95
Cross section types - III-716, III-778, III-861
Cross sections - III-703, III-730, III-776, III-784
Crossed Helical Gears and Precision Mechanics Worms - II-578, II-582
Crown Gear Method (DIN 3990) - II-551
Crowning - II-375, II-381
Cutter
  Hobbing cutter - II-347, II-389
Cutter radius - II-500
Cutter/Tool
  Pinion type cutter - II-349
Cutting teeth on an existing shaft - I-240, I-245
Cycloid - II-398
Cylindrical gear pairs - II-321

Cylindrical Gears - II-262, II-503
Cylindrical Interference Fits - IV-866

D

Data input for clamped parts - IV-1003
Database Tool and External Tables - I-128, II-288, II-351, II-587, III-760, IV-874, IV-887, IV-893, IV-904, IV-915, IV-926, IV-933, IV-943, IV-957, IV-970, IV-976, IV-1026, IV-1041, V-1048, V-1055, V-1063, V-1069, VIII-1123, VIII-1145
Deep toothing or cylindrical gears with a high transverse contact ratio - II-464, II-669
Default values for addendum coefficients - II-509
Default values for tip clearance - II-509
Define details of geometry - II-586
Define details of strength - IV-936
Define load spectrum - II-289, II-308, II-504, II-505, II-593, II-594, II-628, III-767, VIII-1122
Defining 2D graphics - IX-1200
Defining input and output - I-175
Defining Shafts - III-699
Defining subelements - III-703, III-713
Defining the load stage scuffing (oil specification) - II-681
Defining the misalignment for individual elements - II-331
Defining your own default files - I-52, I-73, I-95, I-176
Definition coefficients of sliding friction and velocities - VII-1105
Definition of spring forces - VII-1104
Definitions and dimensions of standard cutters for palloid toothing - II-530
Definitions in [CADEXPORT] - I-58
Definitions in [CATIA] - I-60
Definitions in [GRAPHICS] - I-57
Definitions in [HICAD] - I-61
Definitions in [INTERFACES] - I-58
Definitions in [INVENTOR] - I-59
Definitions in [LICENSE] - I-57, I-71
Definitions in [PATH] - I-43, I-47, I-49, I-52, I-53, I-54, I-70, I-98, I-112, I-117
Definitions in [PROENGINEER] - I-60, I-239
Definitions in [REPORT] - I-56, I-114
Definitions in [SETUP] - I-45, I-46, I-47, I-50, I-51, I-55
Definitions in [SOLIDEDGE] - I-59, I-209
Definitions in [SOLIDWORKS] - I-59, I-193
Deflection and Bearing Forces, Distribution and Force of Torque - III-751
Deformation of the Gear Body - VIII-1147
Deleting a database entry - I-134
Description of database tables - I-144
Description of the Public Interface - I-171
Details - III-834
Determine the equivalent torque (for load spectra) - II-689
Diagram view - IX-1167
Diagrams - II-475
Dialog elements for the variable dialog - IX-1192

Difference between cylindrical gear calculation following ISO 6336 or DIN 3990 - II-678
Differences between different gear calculation programs - II-678
Differential gears - II-533
Dimension of the integral worm shaft - II-574
Dimensioning - II-558, II-611
DIN 3967 - II-358
DIN 5480 - IV-944
DIN 58405 - II-358
DIN 743 (2012) - III-763
Direction of rotation - III-708, III-720, III-733
Directory structure - I-49
Disc spring standard - I-153
Disc Springs - V-1066
Disconnect license from the network - I-71
Discretized model - II-416
Display - II-473, II-537
Displaying critical bearings - III-747
Displaying elements in 3D graphics - IX-1186
Displaying the graphic - IX-1202
Dissipated Heat Flows - III-805
Distances for eccentric clamping/load - IV-1006
Do not cancel if geometry errors occur - II-360, II-458
Docking window - I-73, I-75, I-78
Documentation point - III-730
Downloading a license file - I-44, I-45, I-46, I-47
Drawing - II-643
Drawing data - I-73, I-113, II-625
Drawing data - V-1050
Drawing data - V-1058
Drawing data - V-1064
Drawing number - III-706
Dynamic factor - II-313, II-536
Dynamic load capacity - III-802
Dynamic user interface - I-100

E

Eccentric crowning - II-376
Eccentric profile crowning - II-369
Editing the control file - II-643
Effect of partial load - II-342
Effect of profile modifications - II-675
Effect of torsion on the body of the gear - II-341
Effective belt width - VI-1087
Effective number of V-belts - VI-1079
Effective/Actual - IV-945
Efficiency Calculation - IX-1204
Efficient interfaces - I-172

Eigenfrequencies - III-732, III-756
Einschaltdauer - II-577
Element Assistant - IX-1172
Element overview - III-706
Elements - I-104
Elements editor - III-702, III-705, III-706, III-861
Elements for shafts - IX-1184
Elements list - III-704
Elements of the KISSsoft User Interface - I-72
Elements tree - III-702, III-703, III-706
Elliptical deformation - II-407
Elliptical root modification - II-397
Enter safeties - III-769
Entering an impurity separately in each bearing - III-746
Entry curve as specified by Hirn - II-396
Equivalent stress for sizings - III-749
Estimating the strength of asymmetrical spur gear toothings - II-688
Estimation of the fatigue strength for surface treated parts (section 5.5) - III-770, VIII-1130
Evaluation - II-648
Example
  Interference fit calculation - I-138, I-174, I-178
Example application of a variable dialog - IX-1196
Example of a call from Excel - I-184
Explicitly reading (importing) and generating data - I-177
Export individual teeth - II-621
Export shaft geometry - III-704, III-719
Expressions in variables - IX-1179, IX-1188
Extended functionality for developers - IX-1178
Extending an existing bolt series - IV-1041
Extension sleeves without external forces - IV-1000
External diameter and throat radius - II-568
External tables - I-128, I-136, I-144, I-145, I-146, I-148, I-150, I-152, I-153, I-154, I-156, I-157, I-158, I-159, I-168, I-169, II-274, II-357, II-360, II-549
Extrapolating tolerance values - II-457
Extrapolation of the calculated permissible root and/or flank stresses - VIII-1158
Eyes screen - V-1053

F

f0r and f1r coefficients - III-805
Face gear - 3D geometry - I-197
Face gears - II-539
face load distribution - II-650
Face load factor - II-309, II-318, II-473, II-554
Facewidth - II-266, II-489, II-565, II-585
Facewidth ratio - II-515
Factors - II-313, II-534, II-554, II-610
Failure probability - III-746, III-819

Fatigue Limits for New Materials - III-863
File interface - III-824
Final machining - II-356
Fine sizing - II-439, II-517, III-837
FKM Guideline, Issue 2012 - III-764
Flange connection with torque and forces - IV-985
Flank curvature radii - II-642
Flash temperature - II-648, II-659
FOR loop - I-120, I-125
Forced response - III-791
Forces - III-719
Formatting - I-104, I-112, I-117, I-118
Formula entry and angle input - I-90
Fp tolerance as specified in tables in DIN 3962 - II-457
Free cross section - III-730, III-861
Frequency of load - III-767
Friction clutches - VII-1098
Friction coefficients f0 and f1 - III-806, III-810, III-814
Functionality of the software - VIII-1111
Functions - IX-1189
Functions tables - I-137

G

Gaping - II-650
Gear contacts - III-835
Gear pump - II-423, II-664
Gear teeth in the case of an existing basic solid - I-201, I-209, I-210
Gear teeth when existing shaft data is present - I-217, I-224, I-229, II-358
Gear test results - VIII-1153
Gear tool - II-637
Gear tooth forms - II-296, II-636
Gears - III-736, III-756
General - I-41, I-114, II-456, II-481, II-555, II-556, II-577, II-643, IV-930, VIII-1111, IX-1164
General Bearing Calculation - III-737, III-792
General calculation procedure for KHbeta as specified in ISO 6336-1, Annex E. - II-345
General constraints - III-834
General entries - III-784
General Information - II-533
General information about 3D modeling in Parasolid - I-200
General settings - VIII-1124
Generate a cylindrical gear with an imported hobbing cutter - II-391
Generate a cylindrical gear with an imported pinion type cutter - II-393
Generate a rack with a hobbing cutter - II-401
Generate a SA worm - II-403
Generate cylindrical gear with hobbing cutter - II-389
Generate rack with a pinion type cutter - II-401

Generate with the other gear in the pair - II-400
Generating 3D shafts - I-194, I-201, I-209, I-217, I-224
Generating a cylindrical gear with a pinion type cutter - II-391
Generating a database entry - I-133
Generating a face gear with a pinion type cutter - II-400
Generating a rack with imported hobbing cutter data - II-401
Generating a rack with imported pinion type cutter - II-402
generation - II-615
Generation of 3D gears - I-192, I-201, I-206, I-209, I-215, I-217, I-221, I-224, I-229, I-238, I-250
Generation of 3D model - II-476
Geometries according to DIN 31657 - III-844
Geometries according to DIN 5466 - IV-941
Geometry - II-614, VII-1097
Geometry data - I-180
Geometry details - II-273, II-493, II-548, II-567, IV-936
Geometry fine sizing for 3 gears - II-449
Geometry of clamped parts (connecting solids) - IV-1003
Geometry standards - IV-933
Global settings - KISS.ini - I-53, I-54, I-70, I-71, I-112, I-117
Glued and Soldered Joints - IV-1027
GOST-21354-87 - II-283, II-285
Graphics - II-448, II-452, II-526, III-836, III-837, IV-960, VII-1106, VIII-1161
Graphics list - II-667
Graphics Menu - II-631
Graphics window - I-73, I-78
Grinding notch - II-303
Groups - I-105

H

Hänchen & Decker - III-760, III-775
Hand of gear for gear teeth - II-264
Handling bending and torsion using the results for the shaft - II-342
Hardening depth - II-648
Hardening depth (FKM) - III-708
Hardening depth, known by its abbreviation - II-599
Hardness Conversion - VIII-1135
Head forms - V-1072
Header and footer - I-114
Heat development - II-658
Heat transfer coefficient - III-848
Heat transfer surface - III-849
Heat treatment of hollow shafts in the full state - III-709
Height of face gear - II-549
Helix angle - II-606
Helix angle at reference circle - II-264, II-265, II-545
Helix angle modification - II-374, II-375, II-377
Helix angle modification, parallel - II-476

Helix angle reference circle gear 1 - II-584
Helpful information about the Generation of 3D model tab - II-534
Helptext Viewer - I-77, I-86
Hertzian Pressure - VIII-1131
Housing material - I-156, III-740
How can I test the software? - I-256
How to use KISSsoft - II-619
Hybrid bearings - III-801
Hydrodynamic Plain Radial Bearings - III-839
Hydrodynamic Plain Thrust Bearings - III-853
Hypoid gears with cyclo-palloid gear teeth - II-527
Hypoid offset - II-488

I

Implementation in KISSsoft - VIII-1117
Import and export data from the database tool - I-135
Important service functions - IX-1191
Importing existing KISSsoft calculations - IX-1184
Importing the shaft geometry - III-704, III-717
Improve tooth form - II-620
Impurity - III-741
Individual names for elements - IX-1177
Influence factors - IV-975
Information about material hardness - II-306
Initial parameters - I-70
Inner contour - III-703, III-719
Inner diameter - II-549, II-587
Input data - II-614
Input different load cycles for bending and torsion (for finite life calculations) - III-745
Input elements - I-90
Input file - I-176
Input format for data in imported files - II-616
Input materials for gear calculations in the database - I-255
Input normal diametral pitch instead of normal module - II-458
Input of number of teeth with decimal places - II-266, II-458
Input quality - II-457
Input window - III-702, III-777
Inputs for Basic data - IV-983
Inputting the stress values on the proof point and on the neighboring point - VIII-1118
Inputting Tolerances - IV-869
Inspecting V-belts - VI-1081
Installation on the server - I-46
Installing KISSsoft - I-42
Insufficient scuffing safety - II-680
Integrating the KISSsoft Add-in - I-218, I-225, I-240
Integrating the KISSsoft Add-in (menu options in CAD) - I-203, I-211
Interactions with variable dialogs - IX-1198
Interfaces between calculation programs and CAD - Overview - I-172

Internal toothing - differences in the reference profile if you select different configurations - II-674
Intersecting notch effects - III-784, III-861
Introduction - II-261, IX-1205, IX-1208, IX-1209, IX-1210
ISO 1328 - II-358
Iterative calculation of load distribution - III-745

J

Joint - III-729

K

Key - IV-894
Key standard - I-148
Kinematics - II-658
KISSsoft Calculation Modules - I-73, I-88
KISSsys - IX-1162
Calculation Systems - IX-1163

L

Label - I-107
Language settings - I-50, I-73
Lead angle at reference circle - II-563
Leg Springs - V-1060
Length factor - IV-912, IV-923, IV-940, IV-967
Licensing - I-43, I-45
Licensing the KISSsoft system. - I-47
Lifetime factors - II-307
Lifetime factors as defined in ISO 6336 - II-293, II-505, II-596
Limit dimensions - V-1070
Limited cross section - III-730
Limiting the number of teeth - II-442
Limiting the root diameter - II-441
Limiting the tip diameter - II-441
Limiting values - V-1073
Limiting values in the calculation - III-859
Linear drive train - VIII-1136
Linear end relief I and II - II-372, II-374, II-381
Linear profile modification - II-395
Linear tip and root relief - II-364, II-371
Linear tip and root relief with transition radii - II-367
Linear tip relief with profile crowning - II-370
List of key words used - I-140, I-142
List tables - I-140
Load application - IV-1006
Load capacity of rolling bearings - III-802

Load case - III-770, III-771
Load cases - VIII-1119
Load distribution coefficient - II-309, II-314
Load factor - IV-901
Load factor endurance calculation - III-775
Load factor for static analysis - III-775
Load spectra - I-146, III-735
Load spectra with negative elements - II-310, II-506, II-574, II-600
Load spectrum calculation - IX-1203
Load spectrum with changing torque - II-317, II-682
Load spectrum with negative bins - III-735
Load tab - III-830
Lubricant film thickness and specific oil film thickness - II-635
Lubricant temperature - III-735
Lubricants - I-148
Lubrication - II-279, III-740
Lubrication arrangement - III-846
Lubrication coefficient - II-300

M

Machining allowance for cylindrical gear - I-144
Magnetic tension - III-724
Main calculation steps - IX-1205
Main input area - I-85
Main screen - IV-895, IV-896, VIII-1117
Main settings - II-340
Maintain root circle when changing profile shift - II-458
Maintaining the tip circle when changing profile shift - II-458
Managing database entries - I-130, I-133
Manual and Search - I-77
Manufacture - II-500
Manufacturing a gear - II-637
Manufacturing Data and Working Data - II-612
Manufacturing drawing - II-643
Manufacturing process - II-490, II-493
Manufacturing process for bevel and hypoid gears - I-145
Manufacturing tolerances - II-626
Manufacturing type - V-1053
Mass - III-724
Master gear - II-430
Material - III-708
Material and lubrication - II-587, II-607
Material Disc spring calculation - I-156
Material Interference fit - I-155
Material of bolts - I-156
Material of enveloping worm wheels - I-155
Material of gears - I-157
Material of plain bearings - I-155

Material of shaft-hub connection - I-156
Material pairing-dependent properties - II-460
Material properties - III-710
Material Shaft calculation - I-156
Material Spring calculation - I-154
Material Welded joints - I-156
Materials - I-255, II-274, IV-874, IV-887, IV-904, IV-915, IV-926, IV-943, IV-957, IV-970, IV-976, IV-1026, VIII-1123, VIII-1145
Materials - I-153
Materials - IV-893
Materials - V-1047
Materials - V-1055
Materials - V-1063
Materials - V-1068
Materials and lubrication - II-274, II-549, II-568, II-570
Maximal no of solutions - II-441, II-518
Maximum deflection for sizings - III-749
Maximum service life coefficient - III-747
Maximum Speeds - III-815
Mean normal module - II-484
Measurement grid - II-453
Menus, context menus and the tool bar - IX-1169
Menus, context menus and the Tool Bar - I-73, I-75, I-76, I-95, I-96, I-99
Meshing - II-603, II-637
Meshing stiffness - II-532
Message output - IX-1168
Messages - I-93
Method ISO 6336-B/Literature - II-550, II-551
Methods used for strength calculation - II-480, II-486, II-501, II-550, II-570, II-588
Micropitting (frosting) - II-659
Minimum safeties - II-531
Mixture factor - III-850
Modal analysis of shaft systems - IX-1208
Model and results view - II-339
Modification for mold making - II-405
Modification for pinion type cutter - II-406
Modification for wire erosion - II-406
Modification of S-N curve (Woehler lines) in the range of endurance limit - II-294
Modifications - I-146, II-293, II-304, II-351, II-361, II-378, II-397, II-511, II-555, II-595, II-609
Modifications sizing - II-450
Modified service life according ISO 281 - III-740
Modified service life calculation according to ISO 281 - III-830
Modified service life calculation in accordance with the Supplement to DIN ISO 281 (2007) - III-816, III-826
Modified tabs and dialogs supplied with the system - I-101
Modifying the selected 3D model - I-239, I-244
Modifying the teeth on an existing shaft - I-247
Module ratio - II-489, II-516
Module specific settings - III-742, IV-1039

Module-specific entries - III-841
Moment of friction - III-748, III-809
Moment of friction, seals - III-748
Moments of inertia - II-306
Multi-bolted joint with any bolt position - IV-988
Multi-Spline standard - I-153

N

Necessary entries in the input window - II-440
Network version with dongle (protection key) - I-46
Network version with the license code - I-47
Niemann geometry data - IV-936
Node density - III-744
Non-Circular Gears - II-613
Non-identical (mirrored symmetry) tooth flanks - II-673
Non-linear shaft - III-742
Normal force curve - II-655
Normal force distribution - II-656
Normal module - II-264, II-543, II-584, II-606, IV-934
Normal module (middle), reference diameter, length of reference cone - II-518
Normal module ranges for Klingelnberg machines (cyclo-palloid) - II-528
Notch effects on hollow shafts - III-862
Notch factors - III-767
Notches on the inner contour - III-862
Notches on the outer contour - III-862
Notes - II-603
Notes about profile modification - II-382
Notes on calculations in accordance with the Klingelnberg standard - II-527
Notes on face gear calculation - II-558
Number of blade groups - II-500
Number of buckling cases - III-732
Number of eigenfrequencies - III-732
Number of links - VI-1093
Number of load cycles - II-302, II-506, II-597, VIII-1119
Number of load peaks - IV-938
Number of radial sealing rings, worm shaft - II-573
Number of strands - VI-1091
Number of teeth - II-266, II-488, II-564, II-607, IV-934
Number of teeth in tensioning pulley - VI-1087
Number of teeth with common multiples - II-676

O

Occurring contact stress - IV-911, IV-922, IV-953, IV-966
Oil level - III-748
Oil level and lubrication type - III-813, III-822
Oil temperatures - III-849

Oil viscosity, depending on temperature - II-650
Only geometry calculation - II-550
Only take solutions into account if the following conditions are fulfilled: - II-523
Open interfaces concept in KISSsoft - I-173
Opening the calculation file - I-235
Opening the calculation file for the created gear - I-207, I-216, I-222
Operating backlash - II-426
Operating data - IV-983, VII-1098
Operations - II-388, II-389
Optimal tip relief - II-306, II-599
Other calculation options - VIII-1159
Outer contour - III-703, III-711
Output file - I-176
Overview of the available CAD interfaces and their functionality - I-191, I-196
Overview of the bevel gear manufacturing process and the terminology used in it - II-481
Own data for S-N curve (Woehler lines) - III-711
Own input - II-359, III-729
Own inputs - IV-902

P

Page layout - I-114
Pair/gear data - II-301
Pairing an external gear to an inside gear that has a slightly different number of teeth - II-670
Parasolid Export of 3D Shafts - I-196
Part safety coefficient - IV-1024
Permissible decrease in quality - II-571
permissible lubricant film thickness - III-852
Permissible mass decrease - II-573
Permissible maximum wear of tooth thickness - II-462
Permissible pressure - IV-903, IV-914, IV-925, IV-955, IV-969
Permissible static stress - III-802
Permissible tooth thickness decrease - II-573
Permitted values - IV-978
Pinion - Face gear with Z1 > Z2 - II-559
Plain bearing - III-793
Planetary stages - II-326
Planets - II-463
Plastic - II-459
Plastics according to Niemann, VDI 2545 or VDI 2736 - II-284, II-287
Plastics Manager - VIII-1151
Polygon effect - VI-1093
Polygon standard - I-148
Polygons - IV-949
Position - III-707
Position of shaft axis in space - III-720, III-721, III-731, III-733, III-740
Position of tensioning pulley (x/y) - VI-1080

Position of the tensioning pulley x/y - VI-1089
Possible sizings/suggestions - VI-1085
Power loss - II-658
Power, torque and speed - II-292, II-505, II-552, II-572, II-594
Power-on time - II-469
Preamble - I-175
Precision mechanics - II-669
Pre-machining and grinding allowance - II-352, II-353
Pressure angle at normal section - II-265, II-485, II-545, II-563, II-584, II-606
Pressure angle at normal section an - IV-934
Pressure angle driving/driven flank
  Hypoid gears - II-485
Pressure angle modification - II-371, II-377
Pretension - II-305
Procedure for toothing creation - I-201, I-210, I-217
Process for calculating thermally permitted operating speed (DIN 732-2) - III-806
Profile and tooth trace diagram - II-638
Profile crowning (depth crowning) - II-369
Profile modification - II-293, II-378, II-506, II-595
Profile modifications - II-362, II-364
Profile shift coefficient - II-267, II-392, II-489, II-547, II-565, II-585, II-648, IV-934
Profile shift coefficient (center) - II-607
Profile shift coefficient (on the pinion) - II-546
Programming in the Interpreter - IX-1188
Progressive profile modification - II-367, II-395
Project Management - I-52, I-73, I-76, I-94
Project properties - I-96, I-99
Properties - I-78, I-80, I-81, I-84, III-793
Properties dialog - IX-1178
Properties of the most important bearing types - III-796
Proposal for the hardening depth EHT - II-630
Protective layer thickness, Aluminum - III-769
Protective layer thickness, aluminum, section 4.3.4, Figure 4.3.4 - VIII-1120
Pure thrust bearing - III-728

Q

Quality - II-270, II-457, II-490, II-547, II-566, II-585, II-607, IV-935

R

Range tables - I-139
Rating - II-281, II-501, II-550, II-570, II-588, III-830
Ratio facewidth to center distance - II-465
Ratio facewidth to normal module - II-464, II-523
Ratio facewidth to reference circle, gear 1 - II-465
Ratio of length of reference cone to facewidth - II-522
Reading (importing) a cylindrical gear - II-394

Reading (importing) a worm into the axial section - II-404
Reading (importing) the rack - II-403
Reduced stiffness on the side edges - II-418
Reference cone apexes on the outside/inside of the blank part - II-493
Reference diameter gear 2 - II-484
Reference gearing - II-578
Reference profile - II-346, II-351, II-509, II-608, II-618
Reference profiles - I-144
Reference temperature - III-707, III-733
References - IX-1179, IX-1182
Registering the interface - I-250
Registering the server - I-181
Relationship of calculations with elements - IX-1183
Relative water absorption during swelling - II-428
Relaxation - V-1049, V-1057
Reliability - II-289
Report - II-452, II-622
Report Menu - II-624
Report settings - I-114
Report templates - I-79, I-112, I-117, I-177, I-254
Report Viewer - I-73, I-85, I-112
Required entries in the standard tabs - II-517
Required safeties - II-292, II-437, II-471, II-505, II-552, II-572, II-580, II-594, VIII-1128
Required safeties for cylindrical gear units - II-669, II-679
Required service life - II-504, III-746
Required transverse contact ratio - II-464
Requirements placed on the third party program - I-176
Restoring a database entry - I-134
Restoring the previous stages of the calculation - I-259
Resulting shearing force - IV-941
Results - II-446, II-451, II-475, II-525, III-791, VIII-1150, IX-1209, IX-1210, IX-1211
Results and Reports - I-102, I-110
Results from FEM calculation - IV-992
Results of a calculation - I-111
Retaining Rings (Self-Locking Rings, Seeger Rings) - IV-1034
Rights - I-53
Rolling bearing - I-160
Rolling bearing basic data - I-160
Rolling bearing calculation with inner geometry - III-803, III-826
Rolling bearing Internal geometry - I-163, III-803
Rolling bearing tolerance - I-168
Rolling bearing Tolerance classes - I-168
Rolling bearings - III-726, III-736, III-793, III-795, III-823
Rolling Bearings (Internal Geometry) - III-803, III-823
Root diameter allowances - II-359
Root radius - II-397
Root rounding, ground - II-306
Root stress FEM - II-434
Rough sizing - II-435, II-448, II-515

Rough sizing of shafts - III-758
Rules - I-66
Running KISSsoft via the Add-in - I-206, I-215, I-221, I-228

S

Safety against deformation/fracture - III-768
Safety against fatigue/deformation - III-772
Safety factor curves - II-650
Safety factor for calculating the shear stress at hardening depth - II-470, II-630
Save the files to the PTC Windchill working directory - I-249
Save user-defined rolling bearing in calculation file - III-745
Saving temporary results in CSV format with .tmp file extension - III-743
Scope of a report - I-112, I-114, I-118
Scuffing and sliding velocity (face gear) - II-651
Seam length - IV-1021
Selecting the type of rolling bearing - III-796
Selection of hobbing cutters - I-145
Selection of pinion type cutters - I-152
Selection of the part form - VIII-1117
Separator - I-106
Server functionality - I-181
Service life - II-289, II-552, II-571, II-593, II-628, III-767, III-816
Service life calculation with load spectra - III-817
Service life of files - I-177
Setting Up KISSsoft - I-48
Settings - II-360, II-456, II-533, II-556, II-577, II-602, II-665, II-666, IV-875, IV-888, IV-905, IV-916, IV-927, IV-958, IV-971, IV-977, IV-1014, IV-1030, VII-1107, VIII-1124, VIII-1134
Settings - IV-892
Settings - V-1056
Settings - VIII-1144
Settings in the Module specific settings - VIII-1157
Setup with icon - IX-1173
shaft angle - II-548, II-578, II-587, II-606
Shaft angle - II-487
Shaft connections - IV-1033
Shaft editor - III-702, III-706
Shaft/hub
  diameter of ball/pin - IV-945
Shafts and Bearings - III-698
Shape of flank - II-567
Share factor - IV-913, IV-924, IV-939, IV-968
Shear stress - V-1073
Shear stress values - V-1046, V-1052
Shortened profile crowning - II-370
Show automatic dimensioning - III-749
Show coordinate system - III-749
Simplified view of the gears - I-207, I-216

Single contact stiffness - II-656
Single normal pitch deviation - II-532
Single user version with dongle (protection key) - I-45
Single user version with license code - I-46
Sizing - III-778
Sizing modifications - II-378
Sizing of gear geometry - II-465
Sizing of torque - II-629
Sizing the bearing clearance - III-850
Sizings - II-437, II-464, II-546, II-557, III-857, IV-877, IV-889, IV-906, IV-917, IV-928, IV-959, IV-972, IV-979, IV-1031, V-1059, V-1065, V-1074, VI-1090, VIII-1144
Sizings - IV-892
Sizings - V-1050
Sliding velocity (face gear) - II-651
Small no. of pittings permissible - II-300, II-506
Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis - II-416
S-N curve (Woehler lines) - VIII-1119
S-N curve (Woehler lines) for material - II-650
Solders - I-148
Sommerfeld number - III-851
Special elements - I-106
Special features in KISSsoft - IV-982
Special toothing - II-671
Specific functionalities - IX-1203
Specific sliding - II-648, II-658
Spectra - VIII-1122
Speed - III-707, III-708, III-732
Speed/number of teeth/ratio - VI-1092
Spiral and helix angle - II-486
Spline Standard - I-146
Splines (Geometry and Strength) - IV-918, IV-929
Splines (Strength) - IV-918, IV-937
Spring design - V-1062
Springs - V-1044
Sprocket geometry - VI-1095
Standard - VI-1091
Standard and special tabs - I-89, I-92, I-112, II-264, II-281, II-313, II-389
Standard profiles - IV-908, IV-919, IV-950, IV-962
Standard radius at shoulders - III-744
Start and end block - I-115
Starting KISSsoft - I-69
State during heat treatment - III-709
Static calculation - II-281, II-285, II-590
Static calculation on shearing - II-590
Static strength - II-550
Stiff connection - III-729
Stiffness curve - II-656
Storage locations - I-98

Storage locations and descriptions - I-117
Storage strategies for calculations - IX-1183
Straight line flank - II-399
Straight-sided spline - IV-907
Strength - III-759
Strength calculation as defined in VDI 2736 - II-590
Strength calculation in acc. with Hirn - II-588
Strength calculation in acc. with Hoechst - II-589
Strength calculation in acc. with ISO 6336/Niemann - II-588, II-589
Strength calculation using average position in tolerance field (of tooth form) - II-469
Strength calculation with several geometries on one gear - II-684
Strength details - II-292, II-505, II-572, II-595
Strength details (AGMA) - II-282, II-292, II-307
Strength method AGMA 6123-B06 - IV-942
Strength method as defined by Niemann - IV-937
Strength parameters according to AGMA - III-772
Strength parameters according to DIN - III-771
Strength parameters in accordance with FKM - III-768
Strength parameters in accordance with Hänchen and Decker - III-767
Strength values - V-1046, V-1052, V-1061, V-1067, V-1073
Strength verification with local stresses - VIII-1110
Stress - III-774
Stress curve - II-658
Stress curve (face gear) - II-651
Stress ratio - III-774
Stress ratio R - IV-941
Stress ratios - VIII-1120
Stress values - V-1067
Stripping strength - IV-1012, IV-1017
Structural factor XwrelT or structural factor Xw (scuffing) - II-301, II-381, II-506, II-597
Structure of KISSsys - IX-1164
Student version - I-45
Summary - II-475, II-627
Support of gearing - II-573
Surface factor - III-709
Surface factor KV, section 4.3.3, Table 4.3.7 - VIII-1122
Surface roughness - III-778, VIII-1123
Surface roughness at tooth root - II-531
Surface roughness of housing - III-747
Surface roughness of shafts and shaft-hub connections - I-148
Synchronization - VII-1096
System Assistant - IX-1173
System data - II-292
System of units - I-51, I-73
System settings - IX-1187

T

Tab Elastic Rings - III-832
Table view - IX-1167, IX-1179
Tables - I-91
Tabs - I-106
Take into account deformation due to shearing and shear correction coefficient - III-742
Take into account user specific additions - II-468
Take protuberance into account - II-469
Take shot peening data into account in calculating the strength of toothed gears - II-686
Taking double helical gearing into account in the shaft calculation - III-864
Taking into account housing deformation in static KISSsys calculations - IX-1205
Taking into account shaft bending (face load factor and contact analysis) - II-319, II-340, II-510
Taking into account the bending of the shafts and width modifications - II-429
Taking the results into account in the report - III-711
Technical explanations - IV-1009
Technical notes (toothed belts) - VI-1083
Temperature - III-707, VIII-1120
Temperature duration - III-768, VIII-1120
Temperature of housing - III-707, III-734
Temperatures - II-428
Templates - IV-946
Temporary files - II-622
Tension Springs - V-1051
Tensioning pulley diameter - VI-1079
Tensioning pulleys - VI-1091
Test version - I-45, I-256
Text formatting features - I-119
The active working project - I-49, I-52, I-76, I-95, I-97, I-98
The definition of a bar chart (dg_b) - IX-1201
The definition of an XY-line graphic (dg_l) - IX-1201
The definition of the axis system (af) - IX-1200
The entire definition - IX-1201
The existing elements - IX-1181
The FKM Guideline
Rechnerischer Festigkeitsnachweis für Maschinenbauteile - VIII-1113
The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error. - II-681
The info window - I-76, I-90
The Messages window - I-76, I-93
The module tree - I-75
The project tree - I-52, I-73, I-76, I-94
The Results window - I-76, I-89, I-111
The Shaft element - III-706
The user interface - IX-1165, IX-1166, IX-1167, IX-1168
Theoretical contact stiffness - II-649
Theoretical involute/form grinding - II-398
Theory of Contact Analysis - II-413

Thermal expansion coefficients - III-842
Thermal reference speed - III-804
Thermally safe operating speed - III-784, III-785, III-804
Thickness factors from the shaft diameter - III-784
Tightening technique - IV-1001
Tip alteration - II-354
Tip diameter allowances - II-359
Tolerance Calculation - VIII-1109
Tolerance field - III-737
Tolerances - II-357, II-576, II-617, IV-944
Tolerances - V-1048
Tolerances - V-1056
Tolerances - V-1064
Tolerances standard - I-153
Tool bar and context menu - I-78, I-79
Tooltips and status bar - I-74, I-87, I-91, I-92, I-111
Tooth contact stiffness - II-298, II-379
Tooth deformation - II-429
Tooth flank fracture - II-306, II-409, II-651
Tooth form - II-387, II-647, IV-948
Tooth form factors - II-296, II-307, II-659
Tooth system - II-647
Tooth thickness - II-479
Tooth thickness at tip - II-671
Tooth thickness modification factor - II-489, II-566
Tooth thickness tolerance - II-357, IV-944
Tooth thickness tolerances - I-168
Tooth trace modification - II-320, II-381, III-786
Tooth trace modifications - II-362, II-372
Toothed belt standard - I-169, VI-1084
Toothed Belts - VI-1082
Toothing - I-83, II-260
Topological modification - II-377
Torque curve - II-656
Torque curve/Frequency of change of load direction - IV-910, IV-921, IV-952, IV-965
Torsion critical speed - III-756
Torsion-Bar Springs - V-1071
Transmission accuracy level number - II-308
Transmission error - II-653
Transmission error acceleration - II-654
Transverse coefficient - II-313
Tree view - IX-1166
Triangular end relief I and II - II-376
Twist - II-377
Type - II-482, II-495
Type of bolted joint - IV-998
Type of load spectrum - II-309
Type of loading/Frequency of change of load direction - IV-937
Type of modification - II-361, II-555
Type of oil lubrication - III-748

U

Undercut or insufficient effective involute - II-670
Underlying principles of calculation - II-362, II-481, II-540, II-561, II-583, II-605, IV-930
Unit switch - I-91
Used files - I-176
Usefulness of the service life calculation - VIII-1113
User-defined roller profile - III-826
User-defined settings - I-62
User-defined variables - I-108
Using alternative algorithms for the tooth form calculation - II-459
Using diamond dressing wheels and grinding worms - II-382
Using the 2013 solver - III-743

V

Value input fields - I-51, I-76, I-90, I-91
Values on the x-axis of diagrams - II-475
Variable dialogs - IX-1191
Variable external diameter of the hub - IV-872, IV-885
Variables - IX-1179, IX-1181
Variants - IX-1179, IX-1182
Various - VIII-1108
Varying qualities - II-457
V-belt Standard - I-145
V-belt standards - VI-1077
V-Belts - VI-1076
V-belts data - VI-1077
VDI 2737
  Calculation of gear rim - II-274, II-470
Viewer with neutral format interface - I-196
Viewing database entries - I-130, I-133

W

Warning
Washer - IV-1000
Ways in which KISSsys can be used - IX-1164
Wear - II-661
Wear iteration - II-421, II-663
Weld seam boundary stress - IV-1023
Welded joints - IV-1019
Welded Joints - IV-1018
Welded seam equivalent stress - IV-1022
What licenses are available? - I-257
Width - II-607
Width and circumferential factor - IV-941

Woodruff Key standard - I-148
Woodruff Keys - IV-961
Work hardening factor (strengthening an unhardened gear) - II-681
Worm wheel - generating a 3D model - I-199
Worms with enveloping worm wheels - II-560
Writing material data to the KISSsoft database - VIII-1160

Z

Z-Y coefficients and the technology factor - II-342