

STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-FLOOR PRO 2010

USER MANUAL

Copyright 2010

support@adaptsoft.com www.adaptsoft.com
ADAPT Corporation, Redwood City, California, USA, Tel: +1 (650) 306-2400 Fax: +1 (650) 306-2401
ADAPT International Pvt. Ltd, Kolkata, India, Tel: +91 33 302 86580 Fax: +91 33 224 67281

LIST OF CONTENTS

1. GETTING STARTED 1

2. QUICK REFERENCE GUIDE 5

3. SIGN CONVENTION 13

3.1 SIGN CONVENTION RULES 15

3.1.1 User Defined Actions (Forces/Displacements) and Program Reported Displacements 15

3.1.2 Sign Convention for Actions Reported by the Program for User Generated Manual Design Sections 16

3.1.3 Sign Convention for Program Generated Tabular Reports for Automatically Generated Design Sections 17

4. STRUCTURAL MODELING 19

4.1 STRUCTURAL MODELING RULES 21

4.1.1 Default Position of Structural Components 21

4.1.2 Changing the Location of Structural Components 22

4.1.3 Verification of Geometry of the Structural Model 23

4.1.4 Position of Supports and Significance of Support Position 23

4.2 BOUNDARY CONDITIONS – RESTRAINTS/RELEASE 25

5. POST-TENSIONING 29

5.1 TENDON GEOMETRY 31

5.2 TENDON PROPERTIES 35

5.2.1 Tendon Shape 35

5.2.2 Cross-Sectional Geometry 35

5.2.3 Material Properties 38

5.3 POST-TENSIONING SYSTEM, STRESSING AND TENDON FORCE 39

5.3.1 Post-Tensioning System 39

5.3.2 Effective Force Method of Design 39

5.3.3 Variable Force Method of Design 40

5.3.4 Tendon Friction 41

5.3.5 Tendon Stressing 41

5.3.6 Long-Term Stress Losses 42

5.4 TENDON GENERATION 42

5.4.1 Single Tendon Generation 44

5.4.2 Tendon Mapping – Banded and Distributed 45

5.4.3 Dynamic Tendon Modeling (DTM) 55

5.4.4 Tendon Interference 56

5.5 POST-TENSIONING FABRICATION (SHOP) DRAWING 57

6. DESIGN CRITERIA – MATERIAL PROPERTIES 61

6.1 MATERIAL 63
6.1.1 Concrete 63
6.1.2 Mild Steel 64
6.1.3 Prestressing 64
6.2 CRITERIA 65

7. LOAD CASES AND LOAD COMBINATIONS 79

7.1 LOAD COMBINATIONS 81
7.1.1 Load Combinations Set Up 81
7.1.2 Load Combination Label 83
7.1.3 Analysis/Design Options 83
7.1.4 Long-Term Deflection Template 84
7.2 PATTERN LOADING 85
7.3 REDUCTION OF LIVE LOAD 89
7.4 LATERAL (WIND/SEISMIC) LOADS 90
7.4.1 Lateral Load Case Definition 91
7.4.2 Load Combinations 92
7.4.3 Input/Editing of Lateral Forces 93

8. ANALYSIS 97

8.1 OVERVIEW 99
8.2 MESHING 99
8.2.1 Imposed Location of Nodes 101
8.2.2 Basics of Automatic Meshing 103
8.2.3 Manual Meshing 103
8.2.3.1 Overview and Use of Quadrilateral Cells 103
8.2.3.2 Generation of Triangular Cells 110
8.2.4 Mesh Editing 113
8.2.5 Nodes and Finite Elements 113
8.2.6 Natural and Analysis Nodes 113
8.2.7 Node Consolidation – Node Shift 114
8.2.8 Excluder 117
8.3 ANALYSIS OPTIONS 118
8.4 ANALYZE THE STRUCTURE 119
8.5 DEFLECTIONS CALCULATION IN FLOOR PRO WITH ALLOWANCE FOR SLAB CRACKING 119
8.5.1 Overview 119
8.5.2 Procedure How to Calculate Cracked Deflection 120
8.6 CRACK WIDTH CALCULATION 123
8.7 DISPLAY LINE CONTOURS 124
8.7.1 Overview 124
8.7.2 Contour Toolbar 125

8.8 VIBRATION ANALYSIS 127
8.8.1 General 127
8.8.2 Run Vibration Analysis 128

9. VIEW/VALIDATE RESULTS 131

10. DESIGN 145

10.1 OVERVIEW 147
10.2 DESIGN A SPECIFIC LOCATION 147
10.3 DESIGN A SPECIFIC REGION 150
10.4 DESIGN THE ENTIRE FLOOR SYSTEM 151
10.5 PUNCHING SHEAR DESIGN 155
10.6 REINFORCEMENT SPECIFICATION AND DISPLAY 161
10.6.1 Basic Features 161
10.6.1.1 Mesh Reinforcement Specification 162
10.6.1.2 Generation of Rebar Drawing 165
10.6.1.3 Reinforcement Display Control 167
10.6.1.4 Detailing, Printing, Plot Preparation and Reporting 171
10.6.2 Advanced Features (DRD Module) 175
10.6.2.1 Bonded and Distributed Bars Specification 177
10.6.2.2 Beam Rebar Specification and Display 181
10.7 TRANSFER OF POST-TENSIONING DESIGN FROM ADAPT-PT TO ADAPT FLOOR-PRO 185
10.7.1 Overview 185
10.7.2 Export of Data from ADAPT-Floor Pro to ADAPT-PT 186
10.7.3 Execution of Data in ADAPT-PT 188
10.7.4 Recall of PT Data from ADAPT-Floor Pro 191
10.7.5 Validation of PT Data in ADAPT-Floor Pro 193

11. REPORTS 195

11.1 OVERVIEW 197
11.2 REPORT COMPONENTS 197
11.2.1 Default Reports of the Program 197
11.2.2 Program/User Generated Reports 197
11.2.3 Text-Based Information from Other Sources (External-Text) 198
11.2.4 Graphics-Based Information from Other Sources (External-Graphics) 198
11.3 REPORT GENERATION 198
11.3.1 Single Default Reports 198
11.3.2 Compiled Reports 200
11.4 REPORT GENERATION UTILITIES 203
11.4.1 Report Font 203
11.4.2 External-Text 204
11.4.3 External-Graphics 204
11.4.4 Program-User Report Components 204

11.5 LIST OF THE PROGRAM GENERATED REPORTS 204
11.6 SELECTED SAMPLE OF PROGRAM-S GENERATED REPORTS 206

12. INTEGRATION WITH MULTISTORY BUILDING PROGRAMS 223
12.1 OVERVIEW 225
12.2 IMPORT FROM THIRD PARTY PROGRAMS 227
12.3 GENERIC IMPORT/EXPORT 230

13. APPENDIX A 261
A1 NODE CONSOLIDATION 263
A1.1 Overview 263
A1.2 Consolidation of Analysis Nodes 264
A2 MESHING USING EXCLUDER 274
A3 VIEW FINITE ELEMENT NODE AND CELL INFORMATION 277
A4 FINITE ELEMENT CELL SIZE AND MESH DENSITY 279
A4.1 Maximum Mesh Size 279

14. APPENDIX B 289
B1 TIPS FOR MESHING 291
B1.1 Extend the Boundaries of Openings to Centerline of Adjacent Walls 291
B1.2 Extend Openings to Slab Edge 291
B1.3 Combine Adjacent Openings 292
B1.4 Break Up Walls and Beams That Intersect Outside a Slab Region 293
B1.5 Either “Disregard” Short Wall Projections, or Use Manual Shift Node 294
B2 SUGGESTIONS FOR IMPROVED MESHING 295
B2.1 Wall Intersections 297
B2.2 Break Walls Across Different Slab Regions 298
B2.3 Break Up Walls and Beams Adjacent to Jagged or Irregular Slab Boundaries 299
B3 MANUAL NODE SHIFT (NODE CONSOLIDATION) 301
B3.1 Exclusion of Locations of Your Choice from Node Shift 302
B3.2 Shift Node (Consolidate Node) Manually 304

15. APPENDIX C 305
C1 GLOSSARY OF TERMS 307
C1.1 Natural Nodes 307
C1.2 Analysis Nodes 307
C1.3 Offset 309

16. INDEX ..... ERROR! BOOKMARK NOT DEFINED.

Chapter 1

GETTING STARTED

This manual explains how to use the ADAPT-Floor Pro application to model, analyze, and design a concrete floor system. Floor systems designed with ADAPT-Floor Pro can be conventionally reinforced, or post-tensioned. ADAPT-Floor Pro also enables the user to generate comprehensive reports for purposes of performing code checks, and gauging design values and reinforcement of floor systems.

It is assumed that (i) you have already created a structural model of the floor system you intend to analyze, (ii) obtained a first finite element (FEM) solution for its selfweight, and (iii) have validated the solution to be reasonable – all using the information detailed in the Modeler User Manual. The Modeler User Manual covers the prerequisites to the process that leads to the final design of your floor system.

In addition, advanced topics of FEM processing that were not covered in the Modeler User Manual are discussed in this manual. Follow the sequence of steps listed in **Chapter 2** and detailed in the remainder of this manual to complete the analysis and design of your project.

Chapter 2

QUICK REFERENCE GUIDE

The natural sequence from creating a structural model, to generating structural documents and subsequently fabrication drawings, is listed in the following chapter. There are many short cuts and alternatives in the process of data generation and execution. These, however, are avoided in the following list in favor of a more common and straightforward approach. The items listed here are discussed in greater detail in the remainder of the manual.

Initial steps that were also covered in the Modeler User Manual are:

- Create a structural model.
  - Generate a structural model, either by importing from a third party program, or creating your own structural model using the tools of ADAPT-Modeler.
- Mesh the structure.
  - Go to the *FEM* pull-down menu and select *Automatic Mesh Generation*. Accept defaults of the program.
- Analyze the structure for selfweight, using programs defaults.
  - Once meshing is complete, click on *Analyze Structure* in the *FEM* pull-down menu.
- View the analysis results to validate the structural model and the solution.
  - Validate the solution obtained by examining the deflected shape of the structure in the 3D viewer and evaluating the value of the maximum deflection reported.
- Save the input and the solution.

Steps and considerations covered in this manual:

- Complete and finalize input data
  - Review the structural model for efficiency in analysis/design and mitigation of possible problems.
  - View/edit boundary conditions, if necessary.
  - Apply releases between the structural components, if necessary.
  - Add post-tensioning tendons, if the structure is post-tensioned.
    - From the *User Interface* pull-down menu, display the *Tendon Toolbar*.

- The tools available for tendon generation are: create tendon, map distributed tendon and map banded tendon. These tools and more can be found in the *Tendon Toolbar*.

- Review and finalize reinforcement properties.
  - Go to the *Materials* pull-down menu, and enter the material properties for nonprestressed steel and prestressed steel, if applicable.
- Specify base reinforcement to be included in your design, if any.
  - Enter mesh reinforcement, if any
  - Enter beam reinforcement (corner bars), if you require specific number and size
  - Enter single bars in size, length and number at locations of your choice.
  - Verify each of the reinforcement types entered through a visual check.
- Review and finalize design criteria.
  - Select/verify the building code to be used
  - View/edit the items listed under “analysis and design options”
  - Select the type of reinforcement you prefer for punching shear design, stirrups or shear studs.
  - Select the size of reinforcement bars to be selected by the program for bending and one-way shear.
  - View each of the other tabs of the design criteria and edit the default values if necessary.
- Define additional load cases; add loads if necessary.
  - If the load cases you plan to use are more than “dead”, “live”, “selfweight”, and “prestressing” do the following:
    - Go to *Load Case Library* and enter the label of the other load cases that you plan to use. Once the label is listed in the library, this enables you to enter the associated loads.
  - Enter the loads of each load case.
- View the program-generated load combinations; edit if necessary.
  - Open the *Load Combination* dialog window. Depending on the building code selected, the program will display a number of load combinations.

  - Review/edit the load combinations displayed in the *Load Combination* dialog window for relevance and accuracy.
  - When adding a new load combination, check that the correct *Analysis/Design Option* is selected. In most cases you would select *No Code Check*.
  - It is not necessary, but is advisable to create a load case for prestressing only "PT" with no code check for evaluation of the post-tensioning entered.
  - Likewise, if you have unusual load cases, make sure that for each of the load cases you define a load combination. The objective is to be able to view the results of the unusual load case on its own, in order to evaluate its validity.

- Perform analysis
  - Re-mesh the structure, if necessary.
  - Use the advanced features of meshing, if required to obtain optimum meshing.
  - Review analysis options–modify programs default, if necessary.
  - Obtain a solution.
  - View the deflected shape of the structure for selfweight, and other load combinations for validation.
    - Once the analysis is complete, select *View Analysis Results* from the *FEM* pull-down menu. This opens the 3D viewer of the program to display the solution.
    - Once in the 3D viewer, select the load combination interested in and Z-translation. This is the vertical displacement of the structure. Then click on the tool with two light-bulb graphics. This will display the deflected shape under selfweight only.
    - Zoom, rotate and view the results thoroughly to ensure that the deflected shape under selfweight looks reasonable. In particular, make sure there is no deflection where the structure was intended to have been supported. Correct the structural model if the deflected shape and values under selfweight do not appear reasonable.
- Prepare to Design
  - Create support lines, if not done already.
    - Use the support line wizard to create support lines along the principal axes of the structure.

- Create design sections automatically.
  - Use the *Generate Design Sections Automatically* option which is located in the *FEM* menu to generate the design sections.
  - View the automatically created design sections to ensure that the desired floor area is covered adequately.
  - In the case of more complex structural layouts, splitters are available to aid in customizing the design strips.
- Design the design sections.
  - From the *FEM* pull-down menu click on *Design the Design Section(s)*.
- View the outcome of the code check for the design sections in the X-direction.
  - If there are no purple lines (broken lines) the requirements of the building code you selected is satisfied.
  - If there are purple (broken) lines, the code requirements have not been met; investigate and fix the problem as described in this manual.
- View the outcome of the code check for the design sections in the Y-direction (follow the same instructions as with the X-direction).
- If any changes are made to the model during code check, go back to the "Analysis" step and repeat.
- Execute the punching shear option – if applicable.
  - Click on *Punching Shear Check* to perform the code check for punching shear. This can be found in the *FEM* pull-down menu.
- View the punching shear stress check on the screen.
  - View the values of the punching shear reported by the program on the screen. Display the *Support Line Results Scale* toolbar from the *User Interface* pull-down menu.
  - Click on the tool *Display Punching Shear Design* Outcome. This is the last tool on the right side of the toolbar.
  - If necessary and permissible, the program reports the necessary reinforcement. The program also reports if a section does not pass code by displaying it in red.
  - To view the stress ratios, click on the *Numerical Display Tool* on the same toolbar

- Generate Report
    - Generate tendon drawing. On engineering drawings, most engineers group tendons into tendons in one-direction (such as banded tendons) and tendons in the other direction (distributed tendons). [1]If you plan to show the tendons in two drawings, you must first group them, following the instructions below. If this is not the case, go to the next step.
    - Group Tendons.
        - From the *Settings* pull-down menu, select *Grouping*. This opens the group library. Add two group names, such as "banded tendons," and "distributed tendons."
        - Using *Select/Set View Items*, turn off everything except the tendons and other basic information you need to identify the tendons. In most cases, it is adequate to retain the tendons, slab outline and column supports.
        - Select as many tendons of one group as practical.
        - From the *Modify* pull-down menu, select *Modify Item Properties.*
        - Once the *Modify Item Properties* dialog window opens, select the *Tendon* tab.
        - In the *Tendon* tab, select the group to add the selected tendons. Press *OK* to close the *Modify Properties* dialog window.
        - Repeat the above steps, until all tendons are assigned to their respective groups.
        - Go to the *Grouping Dialog Window* and make only one group of tendons visible, such as distributed tendons. Once you have printed the drawing for this group, hid this group and make the next visible.
    - Generate single report
        - From the *Reports* pull-down menu, select *Single Default Reports/Graphical/Tendon Plan*.
        - In the dialog window that opens, select the following, then click OK.
            - Tendon ID
            - Control point heights

[1] For generating fabrication drawings, tendons are grouped more extensively, assigning unique group identification to tendons of same length and profile.

- Number of strands
- Elongation (if you selected the option in data generation)
- Stressing/dead end (if you selected the option in data generation)

- Generate rebar drawing.
  - From the *FEM* pull-down menu, click on *Generate Rebar Drawing* to compile a rebar drawing.
  - From the *User Interface* pull-down menu, display the *Reinforcement Toolbar*.
  - Use the capabilities of the tools on this toolbar to view and edit the display.
  - Select the reinforcement that you want to be shown on the structural drawing.
  - Edit/move the reinforcement annotation to make it arrive at a clear presentation.
  - Change the font size to values suitable for printing on the paper size you are going to select.
  - From the *File* pull-down menu, select print preview to examine the features of the drawing you are going to print.
  - Print the drawing or export it to AutoCAD, using the *Export DXF/DWG* tool of the program that is accessible from the File pull-down menu.
  - In the same manner, generate other rebar drawings.

- Generate compiled report for submittal to building officials.
  - From the *Reports* pull-down menu, click on *Complied Reports*. Select the items of your choice and send to the printer.

Chapter 3

SIGN CONVENTION

3.1 SIGN CONVENTION RULES....15

3.1.1 User Defined Actions (Forces/Displacements) and Program Reported Displacements....15

3.1.2 Sign Convention for Actions Reported by the Program for User Generated Manual Design Sections....16

3.1.3 Sign Convention for Program Generated Tabular Reports for Automatically Generated Design Sections....17

3.1 SIGN CONVENTION RULES

ADAPT - Floor Pro uses several sign conventions, each intended to reflect the common practice of many reinforced concrete designers. These are:

- Sign Conventions for user defined applied loads and software reported displacements
- Sign convention for the actions (forces) reported by user generated manual design sections
- Sign convention for program generated tabular reports from automatically generated design sections.

While in most instances the sign conventions are compatible among the various parts of the program, you are cautioned to interpret each of the actions reported by the program, using the associated sign convention.

3.1.1 User Defined Actions (Forces/Displacements) and Program Reported Displacements

The positive signs of the user defined **applied forces**, user defined **applied displacements**, and program reported **calculated displacements** are illustrated in **Fig. 3-1**. **Figure 3-1** does not apply to the program reported forces and moments.

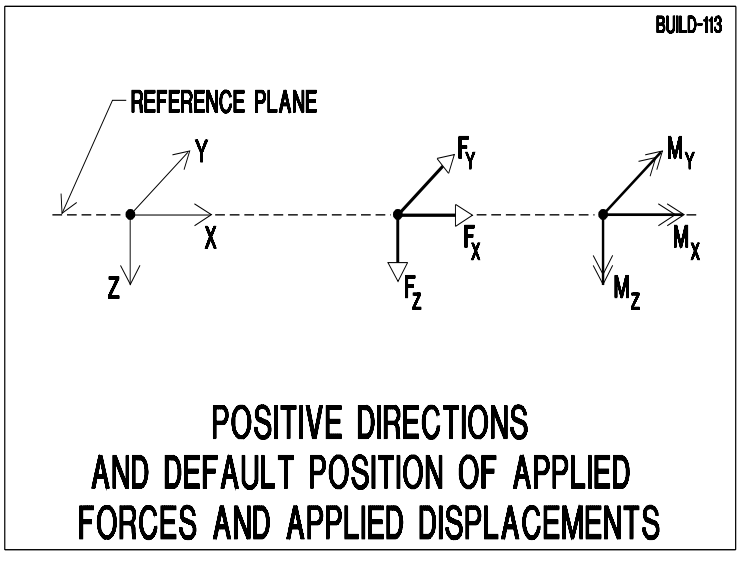


FIGURE 3-1

3.1.2 Sign Convention for Actions Reported by the Program for User Generated Manual Design Sections

The program resolves the stresses that act on the face of a design section into six actions at the centroid of the design section. The actions consist of three forces and three moments. The six actions along with their designation are illustrated in **Fig. 3-2**.

For manually generated design sections, the positive sign of each of the actions is shown in **Fig. 3-3**. The value of the actions can be read off from the property box of the design section displayed in **Fig. 3-4**.

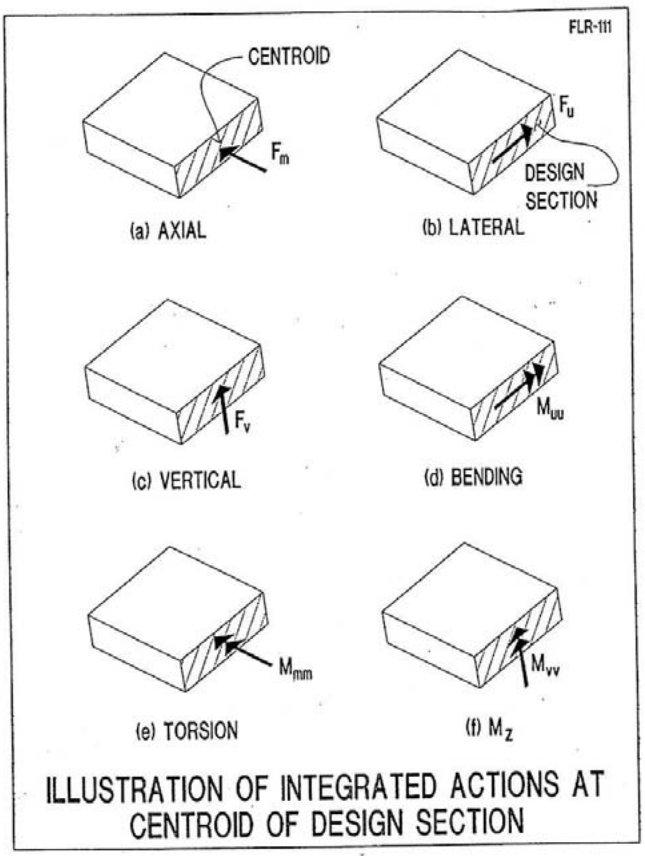


FIGURE 3-2

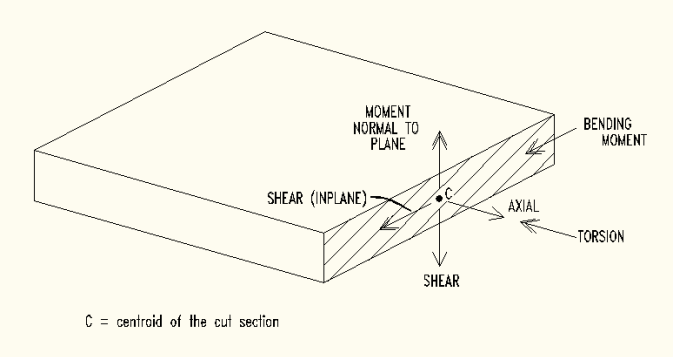


FIGURE 3-3 POSITIVE DIRECTION OF THE ACTIONS FOR MANUALLY GENERATED SECTIONS

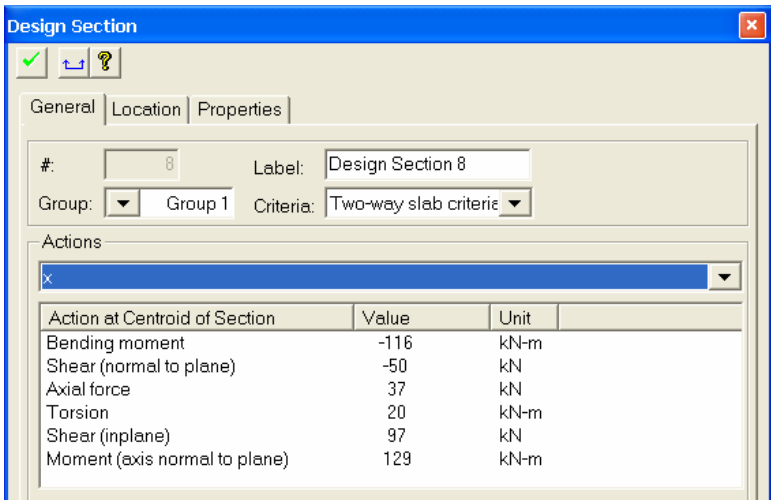


FIGURE 3-4 DIALOG WINDOW FOR REPORTING THE ACTIONS ON MANUALLY GENERATED DESIGN SECTIONS

The same sign convention is used for reporting the forces generated by a manual design section in the program's tabular compilation of the actions[1].

3.1.3 Sign Convention for Program Generated Tabular Reports for Automatically Generated Design Sections

In its automatically generated tabular reports, such as the sample shown in Table 3.1, the program lists the actions on the leading face of a design section, again expressed at the centroid of each design section (**Fig. 3-5**). Unlike the manual sections, where all six actions are reported, the

[1] This differs from the sign convention used in the tabular report for automatically generated sections.

automatically generated report is limited to four of the major actions as shown with positive directions in **Fig. 3-5**. These are bending moment, torsion, shear normal to the slab and axial force.

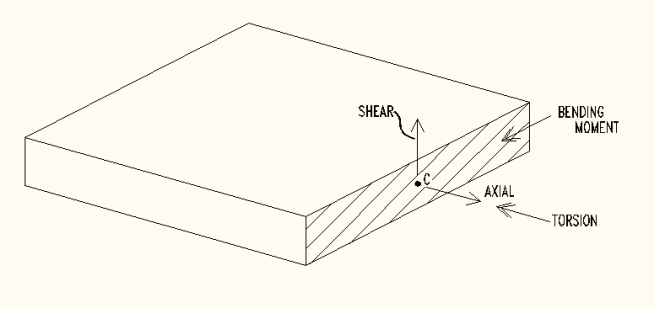


FIGURE 3-5 POSITIVE DIRECTION OF ACTIONS LISTED IN THE PROGRAM GENERATED REPORTS

TABLE 3.1 SAMPLE OF AN AUTOMATICALLY GENERATED TABULAR REPORT OF ACTIONS FOR AUTOMATICALLY GENERATED DESIGN SECTIONS

Design section	Moment	Shear	Axial	Torsion
	k-ft	k	k	k-ft
102000	-31.209	-20.096	-3.401	2.916
102001	4.63	0.087	-8.769	-3.69
102002	21.504	-11.101	-2.564	-5.403
102003	33.706	-3.878	-1.864	-9.488
102004	7.666	-0.075	-11.531	-2.554
102005	36.128	1.821	0.56	-8.532
102006	27.216	8.136	3.006	-7.477
102007	12.591	8.14	3.033	-7.817
102008	-12.374	14.717	6.368	-7.181
102009	-42.483	27.74	10.384	-20.19
102010	-92.597	27.756	10.716	-21.382
103000	-141.39	-39.53	9.568	20.247

Chapter 4

STRUCTURAL MODELING

4.1 STRUCTURAL MODELING RULES 21
4.1.1 Default Position of Structural Components 21
4.1.2 Changing the Location of Structural Components 22
4.1.3 Verification of Geometry of the Structural Model 23
4.1.4 Position of Supports and Significance of Support Position 23
4.2 BOUNDARY CONDITIONS – RESTRAINTS/RELEASE 25

The basic steps of structural modeling are outlined in the Modeler User Manual. In this Chapter we will review several of the salient features of modeling for complex structures and improvements in analytical efficiency.

4.1 STRUCTURAL MODELING RULES

4.1.1 Default Position of Structural Components

Each floor level is modeled together with the supports immediately below and above it. The supports can be columns, walls, springs, points or lines. Soil supports are treated separately in the ADAPT-MAT manual. The geometry of a floor is defined with respect to a horizontal plane. The distances from the top of different parts of the floor to this horizontal plane determine the changes in the elevation of the floor, if any. This horizontal plane is referred to as the *Current Plane*. It is also referred to as the *Current Reference Plane*. **Figure 4.1-1** shows an example of a current reference plane and the default positioning of several structural components with respect to this reference plane. For an example, note that the top of the beam and the drop cap are lined up with the current reference plane. When a structural component, such as a slab region, does not lie at the location shown, it has to be moved to its correct position. This is done using the "offset" feature of the program. The offset feature is described later in this document.

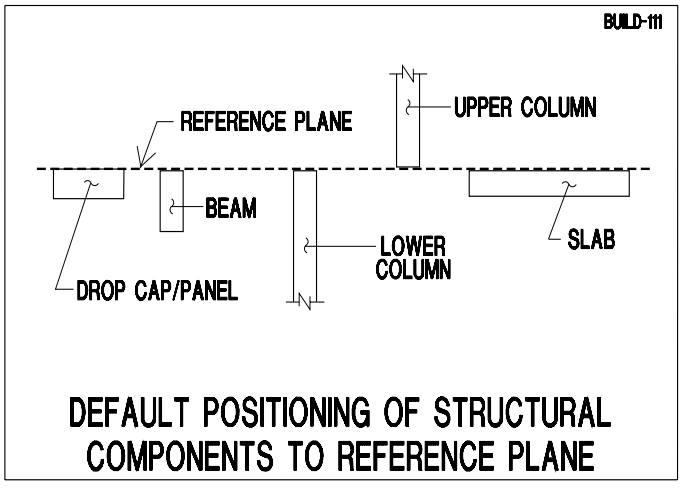


FIGURE 4.1-1

4.1.2 Changing the Location of Structural Components

The supports below the floor system are defined with reference to the current plane and bottom plane (**Fig. 4.1-2**). Similarly the supports above the floor system, such as columns, walls and ramps, are defined with respect to the current plane and the top plane. The top and bottom planes are used by the program to determine the height of each of the supports. For example, the length of an upper column is defined by the position of its top and bottom ends. The position of the bottom end is defined with respect to the bottom, and the position of the top end with respect to the top plane. The default of the program is that each wall or column support extends between the current plane and either the top or bottom plane. There are two conditions when you may wish to change the end position of a structural component along the vertical plane. The *Offset* and *Establish Component Connectivity* functions do this for you.

- **Offset**. For components that do not extend over the entire length between two adjacent reference planes, such as the short upper column shown in **Fig. 4.1-2**, the offset (Z distance) from the associated reference plane is used to adjust the end of the member. Create the structural component, then open its property box and change its offset to the position of your choice.

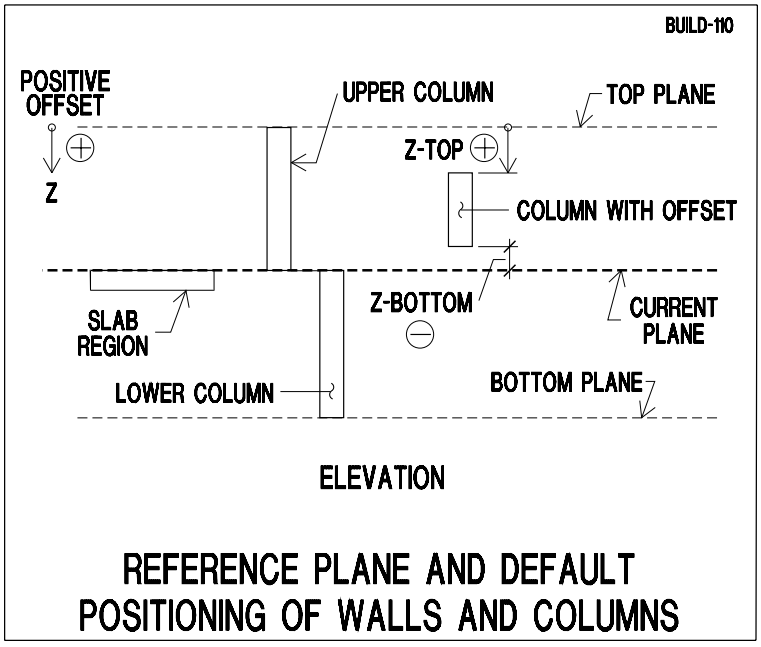


FIGURE 4.1-2

- **Establish Component Connectivity**. The default position of the structural components with respect to the current plane is shown in **Fig. 4.1-1**. As a result, a lower column will have an overlap with the slab, since both of them are adjusted to line up with the reference plane. However, once created, a structural component can be automatically adjusted by the program to its natural position. For example, the top of a column below a slab will be automatically moved to the soffit of the slab. Hence, there will be no overlap or interference between the top of a column and slab region. This is achieved if you click on the *Establish Component Connectivity* menu item from the *Build* pull-down menu and *Preprocessing* side menu.

4.1.3 Verification of Geometry of the Structural Model

Figure 4.1-3 illustrates the geometry of a complex slab in elevation. In this model, the position of each slab region is identified by its offset. You can access and adjust the offset value for each slab through its property box. The position of the supports (upper and lower columns) can be adjusted by using either the *Offset* tool, or the *Establish Component Connectivity* tool. The second tool is faster, since it automatically repositions all the walls and columns of your structural model to their correct locations.

In part (b), the slab region marked "F" is assumed to be at the level of the reference plane. The positions of the remaining components of the floor system are defined by their Z-offset. For example, region "D" has a negative offset (Z_F), while region "E" has a positive offset (Z_E).

It is strongly recommended that you verify the details of the geometry of the structure at complex locations. Do so by using the *Create a Cut at Specified Location* tool ( ).

4.1.4 Position of Supports and Significance of Support Position

Supports, as illustrated in **Fig. 4.1-4**, are positioned at the reference line when generated first by the program. If need be, they have to be shifted to their final location prior to the analysis.

The position of supports on the structure can significantly impact the solution. In simplified analysis, supports are implicitly assumed to be positioned at the centroid of a section. **Figure 4.1-5** illustrates the condition of a simple beam, where the solution is affected by the position

of supports. Your structural model is generally far more complex. This is discussed in greater detail in the section on Boundary Conditions – Constraint/Release.

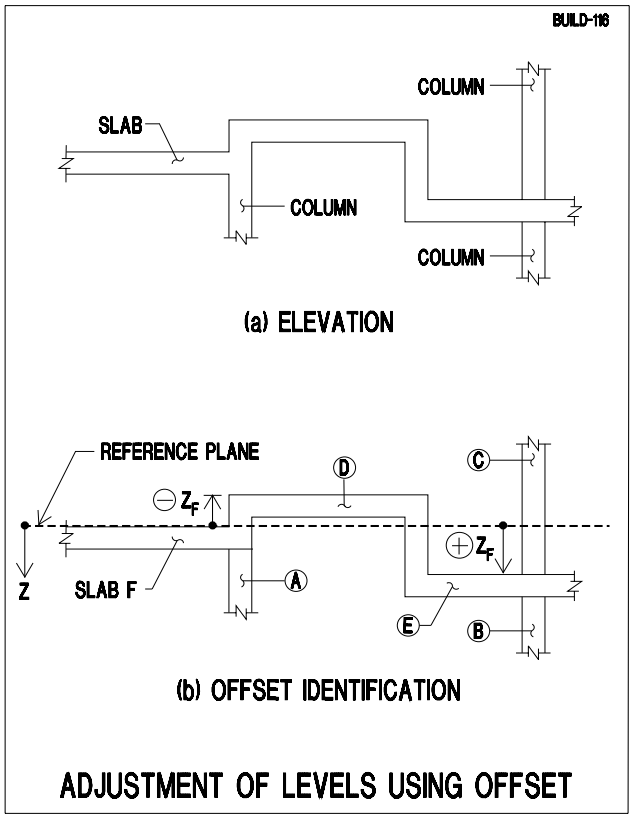


FIGURE 4.1-3

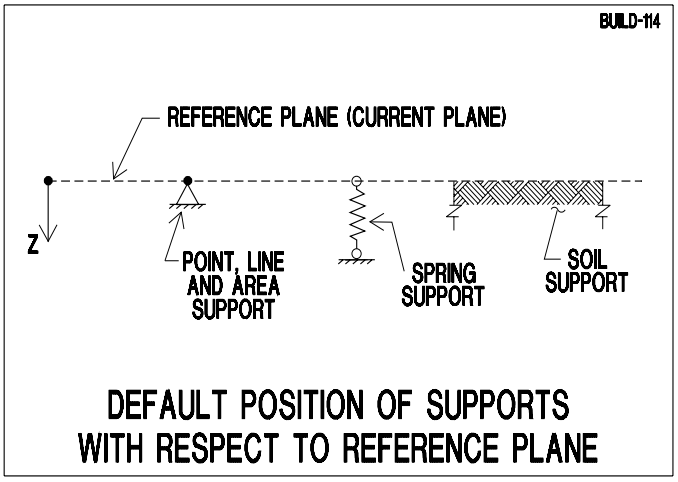


FIGURE 4.1-4

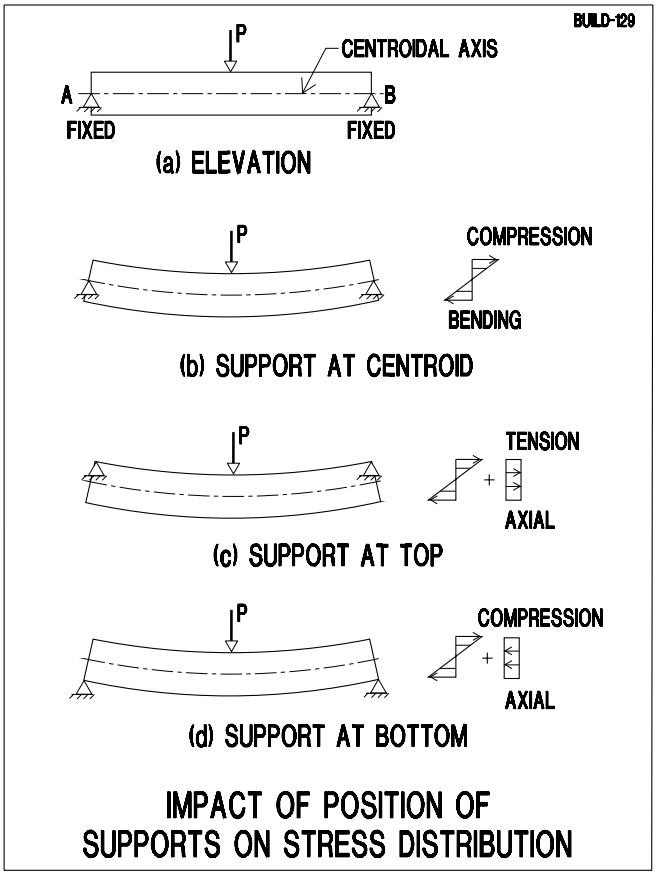


FIGURE 4.1-5

4.2 BOUNDARY CONDITIONS – RESTRAINTS/RELEASE

In the terminology of BUILDER, the connection between two structural components, such as a beam and a column, is defined by the "**release**" condition. This is shown symbolically in **Fig. 4.2-1** for point B between two structural components. If no release is specified, the program assumes rigid connection between the components, such as point D in the same figure. It is assumed that all structural components are rigidly connected together unless "released" by you. The release can be freedom of "translation" or "rotation." The translation allows one component to displace relative to the other in the direction specified by you. The freedom of rotation allows each component to rotate independent from the other, about the axis defined by you. The default directions are the global axes.

The connections between one or more structural components to the supports of the structure are defined by "**restraints,**" such as points A and C in **Fig. 4.2-1**. Restraints are the property of "supports." You first need to place a support at the location of your choice, and then define the manner in which you expect the support to behave. The default of the program for a support is full fixity. That is to

say, neither translation nor rotation is allowed at the location of the support. But, you can open the restraint properties of the support and change them as needed.

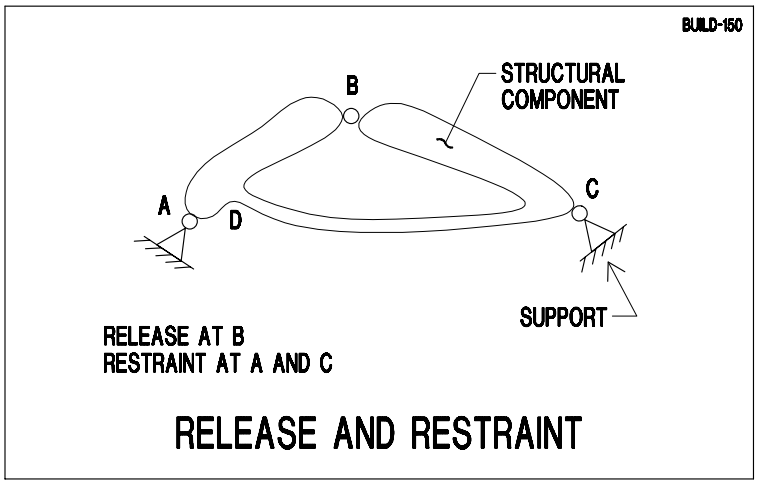


FIGURE 4.2-1

Figure 4.2-2 shows a beam resting on two columns. The columns are supported at their base by two user-defined "supports." The supports are shown by the small triangles. The type of fixity of the column at its base is defined in the property dialog window of the "supports."

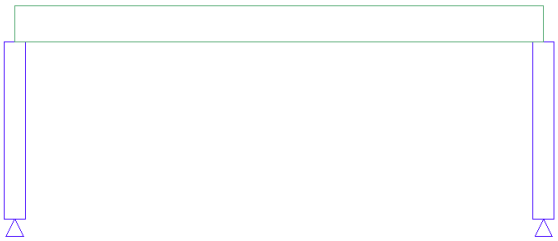


FIGURE 4.2-2 ELEVATION OF A BEAM AND TWO COLUMNS

The beam is shown resting on the column. The default of the program is full fixity between the top of the column and the beam. But, you can change the transfer of force between one and the other by specifying a "release" either at the end of the beam or the top of the column.

- **Release**. The dialog box for the release at the top or bottom of a column is given in **Fig.4.2-3**. As an example, referring to **Fig. 4.2-3**, the dialog box will be used to define the rotational fixity at the top of the column (imposition of a hinge between the beam and column).

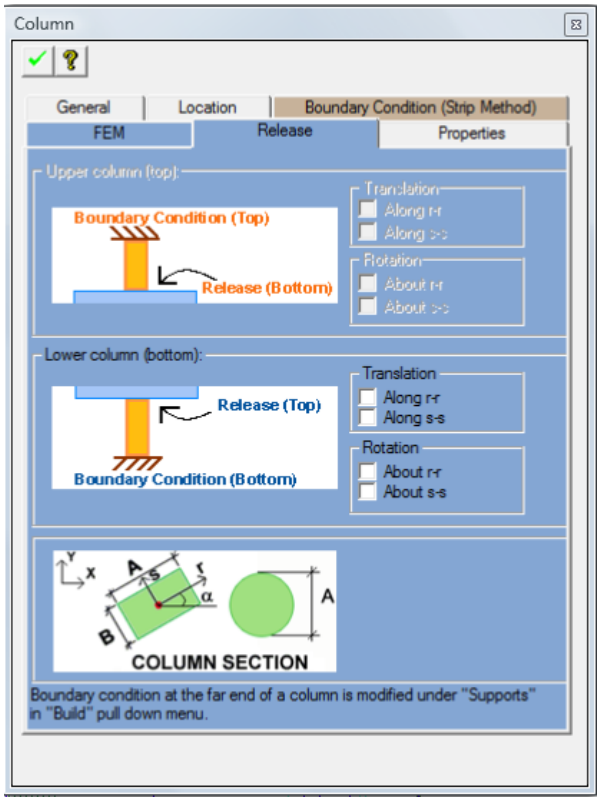


FIGURE 4.2-3 RELEASE DIALOG BOX FOR COLUMN

- **Restraint**. The condition of the support shown at the base of each of the columns in **Fig. 4.2-2** is defined by the restraint dialog box shown in **Fig. 4.2-4**.

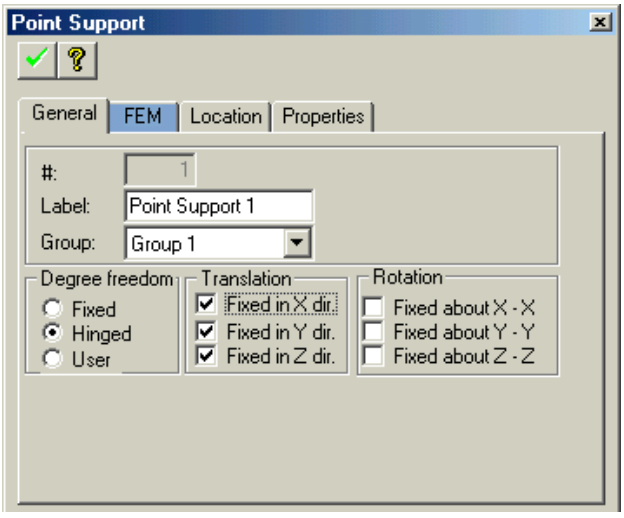


FIGURE 4.2-4 RESTRAINT DIALOG BOX FOR POINT SUPPORT

Chapter 5

POST-TENSIONING

5.1 TENDON GEOMETRY 31
5.2 TENDON PROPERTIES 35
5.2.1 Tendon Shape 35
5.2.2 Cross-Sectional Geometry 35
5.2.3 Material Properties 38
5.3 POST-TENSIONING SYSTEM, STRESSING AND TENDON FORCE 39
5.3.1 Post-Tensioning System 39
5.3.2 Effective Force Method of Design 39
5.3.3 Variable Force Method of Design 40
5.3.4 Tendon Friction 41
5.3.5 Tendon Stressing 41
5.3.6 Long-Term Stress Losses 42
5.4 TENDON GENERATION 42
5.4.1 Single Tendon Generation 44
5.4.2 Tendon Mapping – Banded and Distributed 45
5.4.3 Dynamic Tendon Modeling (DTM) 55
5.4.4 Tendon Interference 56
5.5 POST-TENSIONING FABRICATION (SHOP) DRAWING 57

5.1 TENDON GEOMETRY

Each tendon consists of one or more tendon spans. Each tendon span has a well-defined "Shape." The "Shape" can be a simple parabola, a straight line, a reversed parabola or otherwise. There are a number of pre-defined shapes to assist you in rapid modeling of a long tendon that may consist of several tendon shapes along its length. A tendon span is independent from the span of the structure between two supports (structure span), although in many instances you may select a tendon shape for each structure span.

Figure 5.1-1 shows the elevation of a tendon with six spans. Each span has a geometry that can be defined by the available "shapes" in the program's library.

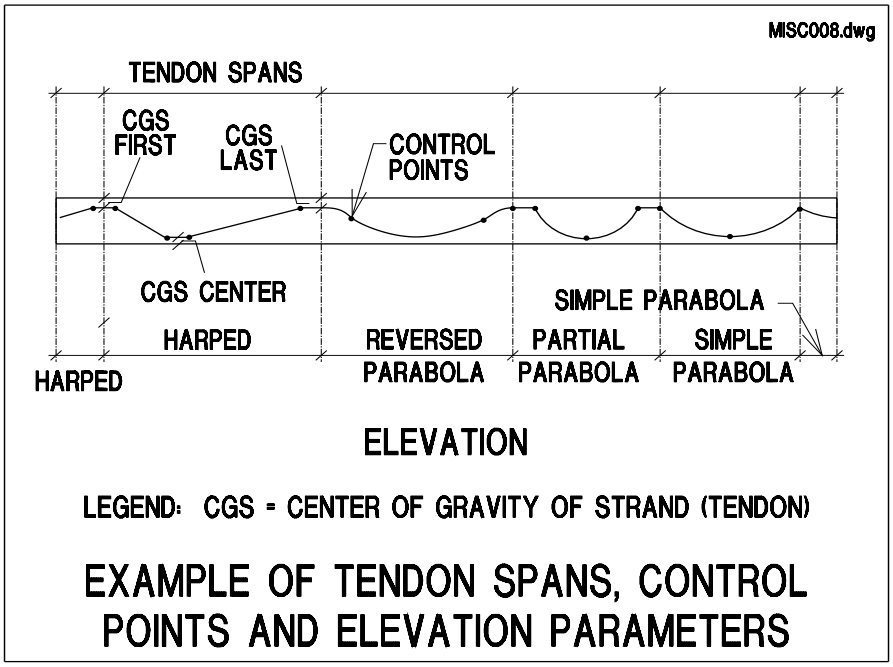


FIGURE 5.1-1

The tendon shapes currently available in the program's library are broken down into span tendons and cantilever tendons. These are (see **Figs. 5.1-2 and 5.1-3**):

- Shapes suitable for structural spans
  - Reversed Parabola (includes simple parabola)
  - Extended Reversed Parabola
  - Partial Parabola

- Harped
- Straight

- Shapes suitable for structural cantilevers
  - Parabola Up
  - Parabola Down
  - Harped
  - Partial Parabola

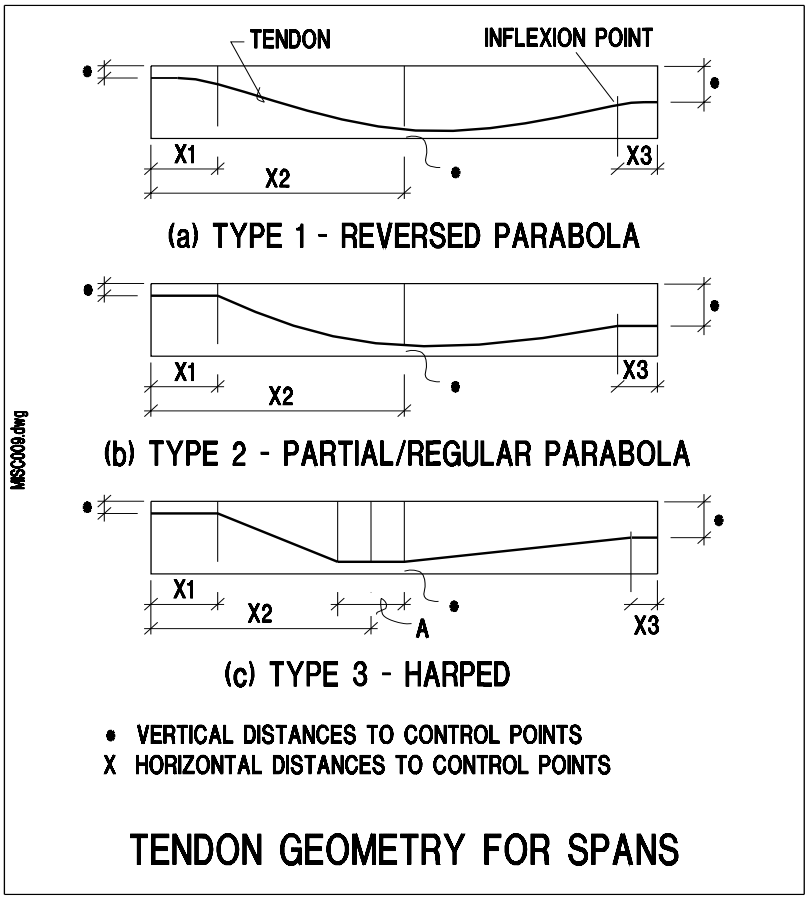


FIGURE 5.1-2

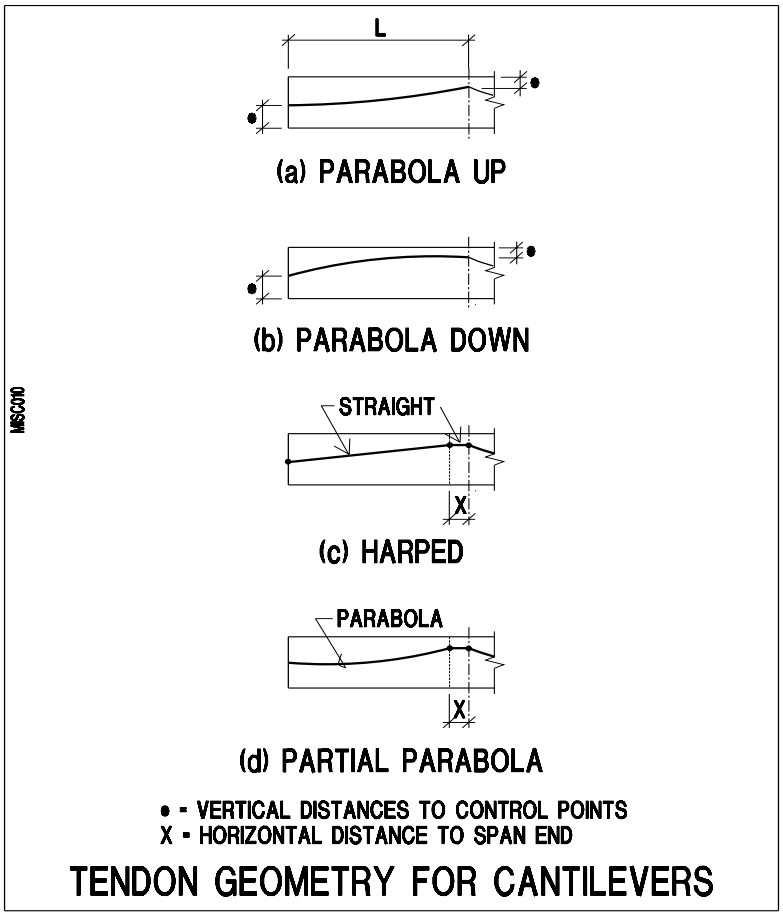


FIGURE 5.1-3

At creation, the program places each tendon within the outline of the concrete slab or beam at the location of the tendon. In doing so, the program observes the minimum cover you have specified at the start, end and low point of each tendon. Later, you will read how to view the tendon in elevation within its concrete outline and edit its geometry, if needed.

The shape of a tendon, and the values for its centroid to the top and bottom fibers (CGS) of the concrete outline, are given in the property box of the tendon (**Fig. 5.1-4**).

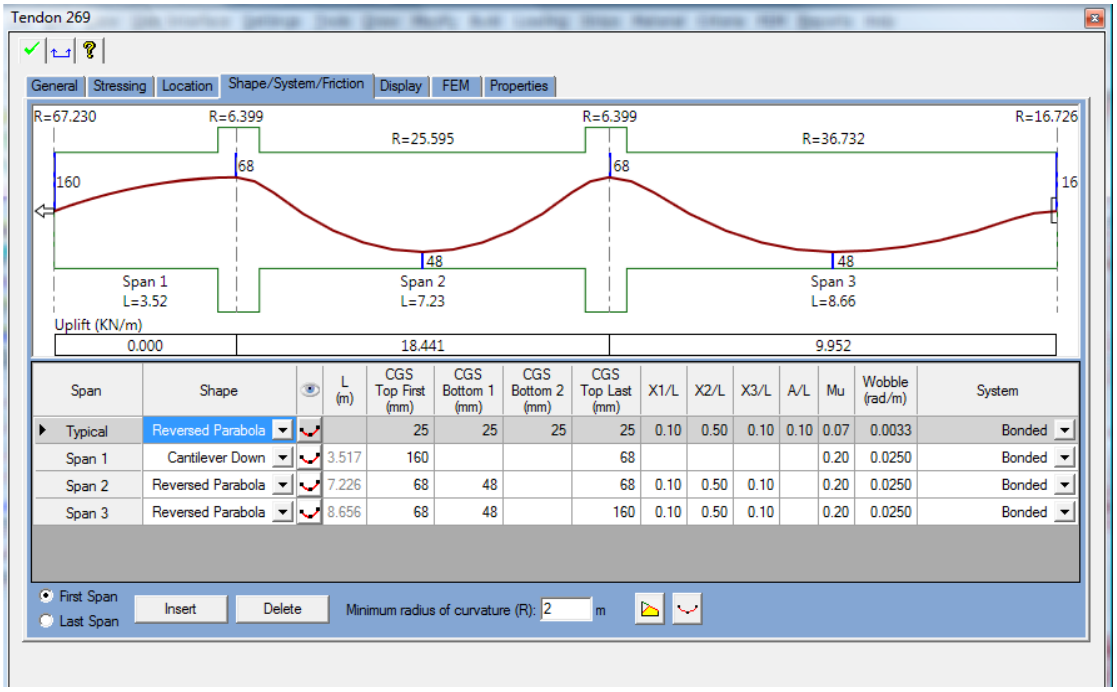


FIGURE 5.1-4 TENDON PROPERTY BOX

The Shape/System/Friction input tab contains both tabular and graphical interactive screens. The tabular data includes shape, span length, control points, inflection points, friction parameters and system type. All fields are editable except for the span length which can only be changed in the main modeling screen. To select a shape for a tendon span, click on the shape combo box for the desired span in order to activate the specific span and then using the combo box assign a shape. The Typical input row (top row) can be used if several rows in a column have the same value. Creation of tendons and other properties of it are discussed later in this section.

You can also add and delete spans in the current, displayed tendon by using the "Insert" and "Delete" tabs below the tabular input screen. The minimum radius of curvature can be assigned and the actual value is reported in the graphical screen at each high and low point along the length of the tendon. Where the radius of curvature does not meet the user-specified minimum value, the graphical output will report this as "NG" and will be highlighted in red.

In addition to the reporting of radius of curvature, the graphical screen reports CGS at high and low points along the length of the tendon, span length, stressing ends (dead and/or live) and the average uplift (force/length) for each span. The latter value can be used to expedite a hand validation of % balanced load.

When the post-tensioning design option of "Calculated force" is set under the Stressing input tab, the graphical screen in the Shape/System/Friction input can be toggled to display the following output:

- The calculated elongation at stressing ends
- Jacking ratios at stressing ends
- The average and maximum stress ratios (actual stress/ultimate stress) along the length of the tendon

You can initiate the calculation of the aforementioned data by pressing the "Friction and Elongation Calculation" button located at the bottom of the screen. This function is only available with the Post-Tensioning Fabrication (Shop) Drawing module. When the post-tensioning design option of "Effective force" is used, the button is grayed out and the graphical screen will default to the tendon and span geometry.

The graphical screen can be stretched so that you can view any span or a combination of spans at a time. To stretch the screen, place your cursor inside the graphical view and use the mouse roller to stretch in or out. The view can also be panned by clicking on the mouse roller and moving left or right.

5.2 TENDON PROPERTIES

5.2.1 Tendon Shape

The shape of a tendon determines its profile in the vertical plane. This is discussed in the preceding section. A tendon can also have a shape other than a straight line on the horizontal plane. This is discussed in a later section on Tendon Generation. A graphical representation of the shape selected in the pull-down menu can be viewed by selecting the tendon profile icon . The program duly accounts for the effects of tendon curvature on the horizontal plane.

5.2.2 Cross-Sectional Geometry

A tendon is made up of one or more strands encased in a sheathing. **Figure 5.2-1** shows a single strand with a plastic wrapping (mono-strand)[1]. The alternative, multi-strands in a single duct, is shown in **Fig.**

[1] Mono-strand refers to tendons in which strands are pulled one at a time. In multi-strand tendons the strands within a duct are all pulled at the same time.

5.2-2(b). Three values should be specified for the cross-section of each tendon. These are (**Fig. 5.2-3**):

- Number of strands
- Cross-sectional area of each strand
- Tendon diameter

The number of strands and the cross-sectional area of each strand determine the amount of steel available for prestressing. The diameter of the duct, along with the minimum specified concrete cover, is used to determine the location of the tendon's centroid (CGS) and also to avoid interference of intersecting ducts. The tendon's centroid (CGS) is the centroid of its strands. For multi-strand tendons, this is different from the duct's centroid. The distance between the tendon's centroid and the duct centroid (shown as 0.43" – 11mm in **Fig. 5.2-2**) is referred to as "z" distance and is used in the preparation of fabrication (shop) drawings.

The structural drawings show the distance of the tendon's centroid (CGS) from the member's soffit. The fabrication drawings show the distance of the duct's underside to the soffit of member[2].

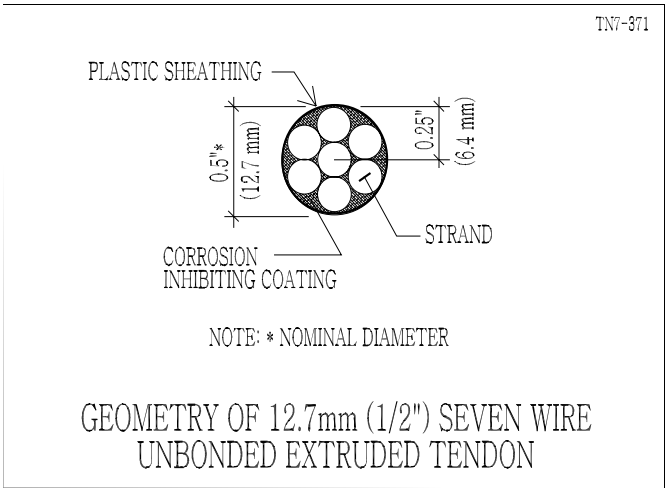


FIGURE 5.2-1 MONO-STRAND TENDON

[2] BUILDER Fabrication (Shop) Drawing Module uses the distance "z" to determine the chair heights.

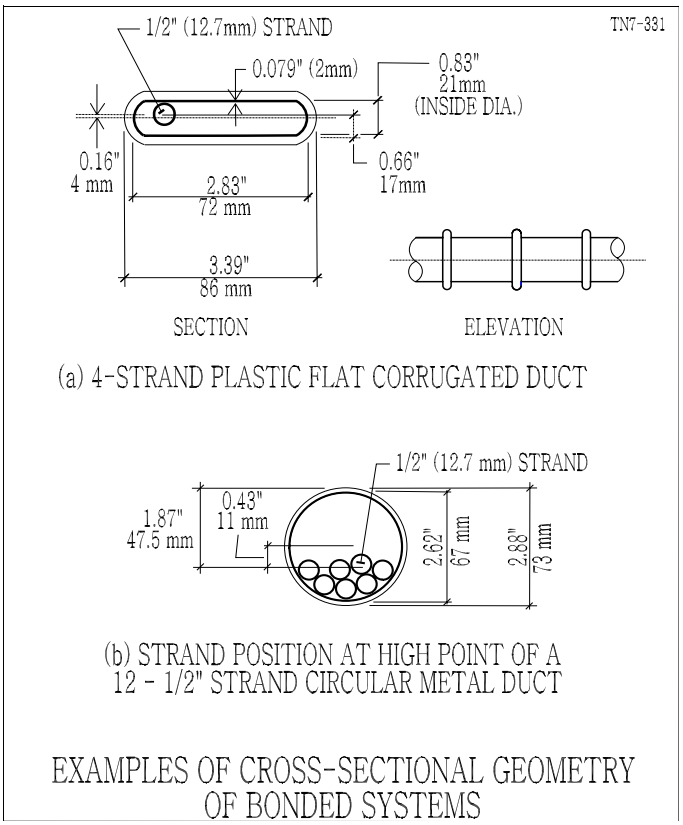


FIGURE 5.2-2 MULTI-STRAND TENDON

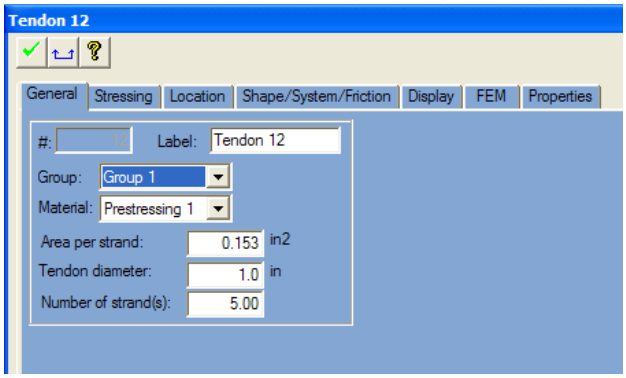


FIGURE 5.2-3

The total area of prestressing steel for each tendon is obtained from the data entered in **Fig. 5.2-3**. It is the product of the area per strand multiplied by the number of strands. The tendon diameter entered in the same figure is used only to check the interference of tendons in space, and in the fabrication (shop drawing) module of the program.

5.2.3 Material Properties

There are two types of material properties generally specified for a tendon. One is the material property of the prestressing steel (strands in most cases). The other defines the friction properties between the prestressing steel and its sheathing (duct). In this section we consider the properties of the prestressing material. These are:

- fpu (specified ultimate strength of prestressing material)
- fpy (yield stress of prestressing material)
- Eps (modulus of elasticity of prestressing material)

Material properties are specified in the dialog box as shown in **Fig. 5.2-4**, which can be opened from the *Material* pull-down menu *Prestressing/FEM*. Not all the tendons in your project need to have the same material properties. You can define more than one material property for prestressing steel in **Fig. 5.2-5**.

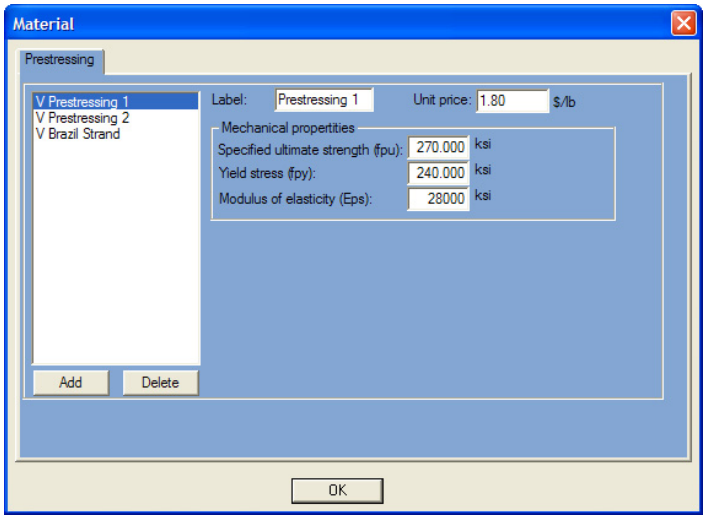


FIGURE 5.2-4 DIALOG BOX FOR MATERIAL DEFINITION OF PROPERTIES OF PRESTRESSING STEEL

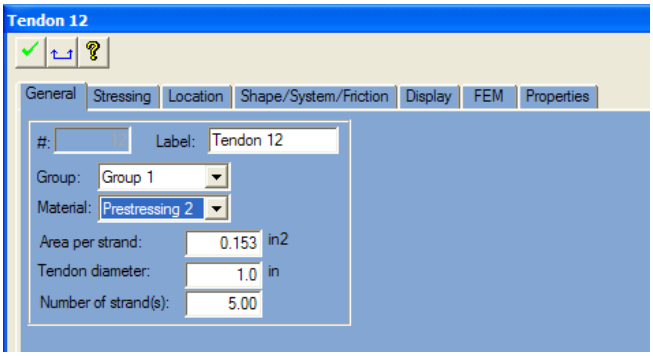


FIGURE 5.2-5 DIALOG BOX FOR ASSIGNMENT OF TENDON MATERIAL PROPERTIES

5.3 POST-TENSIONING SYSTEM, STRESSING AND TENDON FORCE

5.3.1 Post-Tensioning System

Both bonded (grouted) and unbonded systems of post-tensioning are covered in the program. You can define each tendon individually as bonded or unbonded. In addition, for a given tendon, you have the option of defining one or more of its spans as bonded, and the remainder unbonded.

The assignment of bonded, or unbonded system to each span takes place through the combo box shown at the far right of **Fig. 5.3-1**

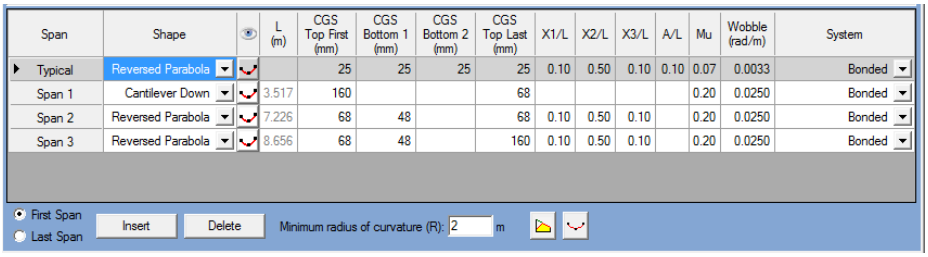


FIGURE 5.3-1 PRESTRESSING SYSTEM ASSIGNMENT DIALOG WINDOW

5.3.2 Effective Force Method of Design

Effective force design is used primarily in North America. With this method, you define the required post-tensioning force at each location in the floor system, along with the associated tendon profile. Hence, the structural drawings show the final post-tensioning force after all stress

losses (effective force) and the tendon profile. The post-tensioning supplier (contractor) can then convert your drawings (structural drawings) to fabrication (shop) drawings, on which the supplier will show the actual number and position of each tendon, together with stressing and elongation values. The value of the force you will use in your design is the force after all losses have taken place. These include friction, seating loss and long-term losses.

In common building construction, the effective force for unbonded tendons is calculated on the basis of 1,200 MPa (175 ksi) times the tendon area. The dialog window for the specification of effective force is shown in **Fig. 5.3-2**. Note that the force entered in this dialog window is the product of the effective stress times area of each strand.

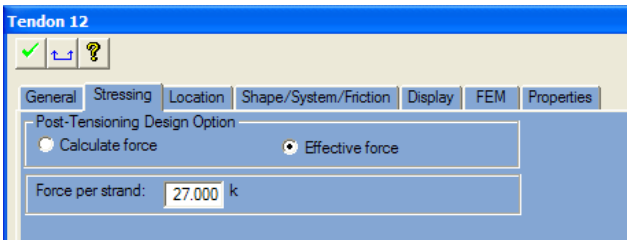


FIGURE 5.3-2 DIALOG BOX FOR SPECIFICATION OF EFFECTIVE FORCE IN TENDON

5.3.3 Variable Force Method of Design

With variable force design, you define the number, location, profile, and stressing of each tendon on your drawings. The program calculates the force immediately after seating of each tendon using the tendon's geometry, its friction, seating parameters and the jacking force. The forces calculated immediately after stressing are then adjusted for long-term stress losses before being used for code compliance in serviceability and strength requirements[3].

To use this option, you must define the friction and stressing characteristics of each tendon. These are defined in the following sections.

[3] In this method of analysis, the tendon force for each finite element cell of the floor system is based on the force calculated on that location from the friction and other stress losses. The program does not use the now obsolete and simplified "load balancing" or "equivalent load" procedures, where a tendon is removed from its housing and substituted by an upward uniform force over each span.

5.3.4 Tendon Friction

The friction properties of a tendon are defined by the following two friction coefficients:

- μ = coefficient of angular friction (meu); and
- K = coefficient of wobble friction (radians/ft or radians/m).

Each tendon span can have its own friction properties. This enables a tendon to be partly external, partly unbonded and partly bonded over its length. The friction properties of each tendon (or tendon span) are entered in the right data entry frame of the dialog window shown in **Fig. 5.3-3**. The input fields for the friction values appear only if you specify "Calculate Force" in the stressing tab of the tendon property box.

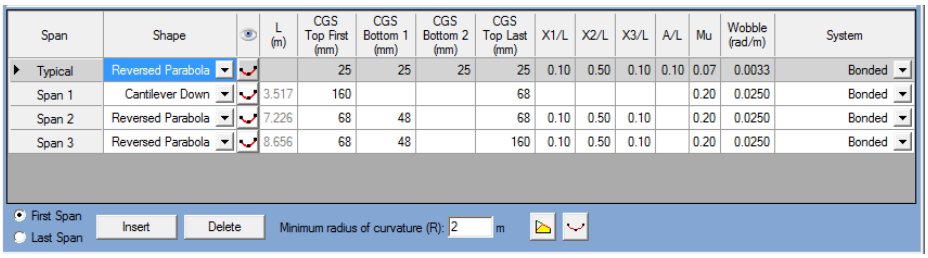


FIGURE 5.3-3 DIALOG WINDOW FOR FRICTION PROPERTIES

Obviously, for external tendon spans, you specify zero for the friction coefficients.

A full account of tendon stress losses, both due to friction and long-term effects, is given in an ADAPT-Technical Note[4].

5.3.5 Tendon Stressing

Each tendon can be stressed at either or both ends. At each end the stressing can be characterized by one of the following manners (**Fig. 5.3-4**);

- Specify jacking force, typically 80% of tendon's ultimate strength (0.80*fpu*tendon area);

[4] The Technical NoteTN-T9-04: "*Prestressing Losses and Elongation Calculations"* is on your program CD-ROM and can be downloaded from the ADAPT website, www.adaptsoft.com.

- Specify tendon stress at stressing end and prior to tendon seating, typically 80% of tendon's ultimate stress (0.8*fpu); and
- Ratio of jacking stress to ultimate strength of tendon (typically 0.80).

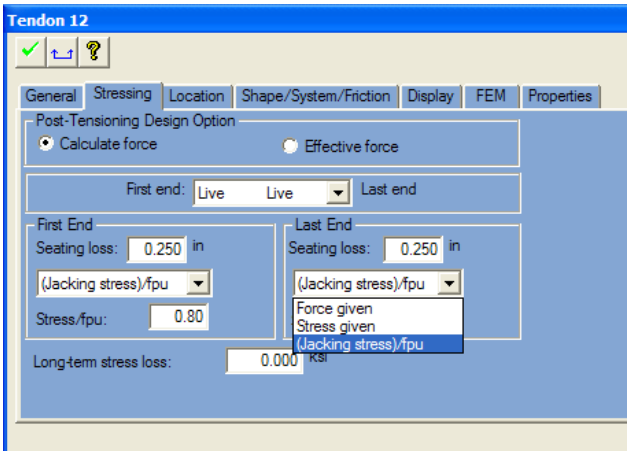


FIGURE 5.3-4 DIALOG BOX FOR SPECIFYING TENDON STRESSING

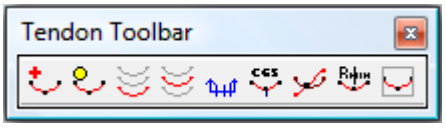
5.3.6 Long-Term Stress Losses

Stress losses due to creep, shrinkage, relaxation in prestressing and "elastic shortening" are grouped as a lump sum and must be entered by you in the data field "Long-term stress loss." The current version of the program does not calculate the long-term losses automatically. It is scheduled for release in future releases.

Typical long-term loss values for building structures are 75-100 MPa (11-15 ksi). Grouted tendons tend to have a larger friction loss, but a lesser long-term loss at the critical locations in a member.

The long-term stress losses are specified in the tendon's property dialog window, only if you select the "Calculate Force" option (**Fig. 5.3-4**).

5.4 TENDON GENERATION



Create Tendon Toolbar

The tools available for tendon generation, display and editing are listed in the *Create Tendon Toolbar*. These are:



Create Tendon. Creates a single tendon in the structure. Wherever the tendon has a high point, make a click with your left mouse button.



Display Tendon. Displays or hides all the tendons of the structural model.



Map Distributed Tendon. Replicates a selected tendon over a region that you define.



Map Banded Tendon. Generates a group of tendons parallel to a support line.



Display Tendon Elevation. Creates an elevation of the tendons selected within the outline of the concrete.



Show CGS Values from Bottom. Displays the distance from the centroid of a tendon's prestressing steel (CGS) to the outermost bottom fiber of concrete



Detect Tendon Interference . Detects the intersection of selected tendons in space.



Show/Hide Radius of Curvature. Click this button to display the results of the minimum curvature check on plan where the individual curvatures of each tendon get checked against the minimum radius of curvature defined in the tendon properties. The actual radius is displayed for each tendon as well as an OK in cases where the check passed, or a NG where the check failed.



Trim/Extend Tendon. This tool automatically trims or extends the tendons first and last span so that they terminate at a slab edge. The distance between the end of tendon and slab edge must be no greater than 0.5m by default. The default value can be changed in the initialization file in programs folder. Flag: "Extend/Trim_Tendon_to_SlabRegion_m".

5.4.1 Single Tendon Generation

Each tendon consists of one or more spans. A tendon span is a length of tendon that has a pre-defined geometry (shape). A tendon span can be a simple parabola, a reversed parabola, or other shapes that were described earlier in the section on Tendon Geometry. New tendons are generated by clicking on the *Create Tendon* tool in the create tendon toolbar. Click at the two ends of each tendon span as you move the cursor along the path of the tendon you intend to create. This will be treated as a continuous, multi-span tendon until you hit the "C" or "end" key on the keyboard[5].

Each tendon span will be displayed on a straight line between the two ends identified by the mouse clicks. However, in the vertical plane, the tendon will have the shape you specify in its property dialog box. Tendons that are curved in plan as well as in elevation need to be generated as multi-span tendons. In **Fig. 5.4-1** the tendon is swerved on plan to avoid an opening between two columns. The swerve of the tendon is modeled through a number of straight-line segments in plan. Each straight-line segment is viewed as a tendon span. The height of the tendon from the soffit of the slab is adjusted through the CGS parameters of each tendon span. The three-dimensional view of the adjusted tendon is shown in **Fig. 5.4-2.**

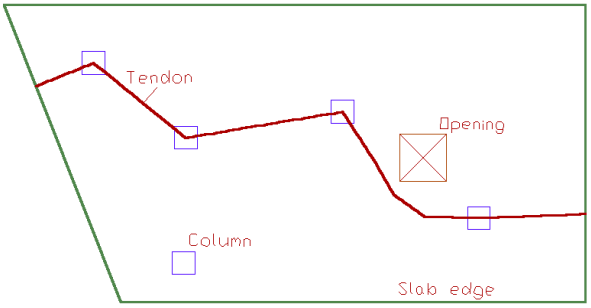


FIGURE 5.4-1 TENDON WITH CURVATURE ON PLAN

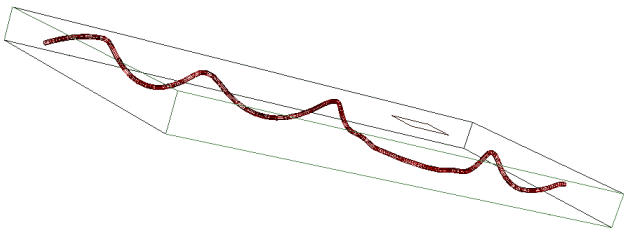


FIGURE 5.4-2 THREE-DIMENSIONAL VIEW OF TENDON WITH CURVATURE ON PLAN

[5] Alternatively, you can right click the mouse and select "Close."

Within each span, the tendon you have marked will automatically assume the default shape you have selected. For example, if the default shape selected is "reversed parabola," the tendon span created will have the shape of a reversed parabola. The program automatically detects the concrete outline within which the span is created and positions the high and low points of the tendon shape at the clearances specified. The clearances for each tendon are defined in its property box. Obviously, once generated, you can view a tendon in elevation and adjust its profile, if need be.

5.4.2 Tendon Mapping – Banded and Distributed

Tendon mapping is a process whereby you can replicate an existing tendon over a region of your choice—with some degree of intelligence from the tendon. The inbuilt intelligence in the current version of the program is as follows:

- The replicated tendon recognizes the change in the boundary of the region it is intended to cover and automatically adjusts its individual span length and consequently its total length to cover the entire area.
- The replicated tendon retains the minimum cover specified from the top and bottom, while cloning itself over regions of different length and depth. It also automatically adjusts its inflection point, or other parameters that define its shape.
- If there is a step, change in slab thickness, or a beam, the replicated tendon recognizes the change and automatically modifies its geometry to match the geometry of the concrete outline.
- The tendon can be automatically replicated using one or more of the following criteria defined by the user:
    - Minimum and maximum average precompression (P/A);
    - Minimum and maximum fraction of selfweight to be balanced; and
    - Maximum spacing between adjacent tendons,

The following describes the automatic generation of tendons in detail.



Map Distributed Tendon

This tool is used to cover an entire slab area with uniformly distributed tendons, each with the same cross-sectional area, material properties,

stressing conditions, and number of tendon spans, and each also adjusted to the geometry of its location.

To map the distributed tendons do the following:

- Select the tendon you want to replicate.
- Click on the map distributed tool. The dialog window shown in **Fig. 5.4-3** opens. The layout can either be optimized based on average precompression and percent of balanced selfweight or simply replicated by a given number input by the user. The option is also given to define the parameters by which the spacing should be determined. If do not optimize is selected, the total number of tendons will be one more than the number of spaces you enter in the data field of this dialog window.

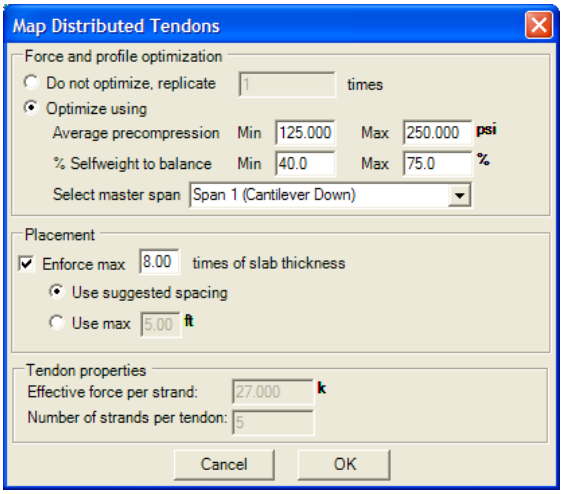


FIGURE 5.4-3 DIALOG WINDOW FOR MAPPING DISTRIBUTED TENDONS

We illustrate the mapping of a single span tendon over a region in the following steps (**Fig. 5.4-4**).

- Draw a single tendon anywhere on a slab region (part – (a) of the figure).
- Click on the tendon (select it), then click on the *Map Distributed Tendons*.
- With Do not optimize, replicate selected, enter 5 for the number of tendons to be duplicated.
- Next, follow the instructions that appear at the bottom of the screen. The instructions direct you to click at a point to represent

the uppermost corner of the field over which you want the tendon replicated, and then click at the uppermost right corner. Parts (a) and (b) of the figure show the two extreme corners selected.

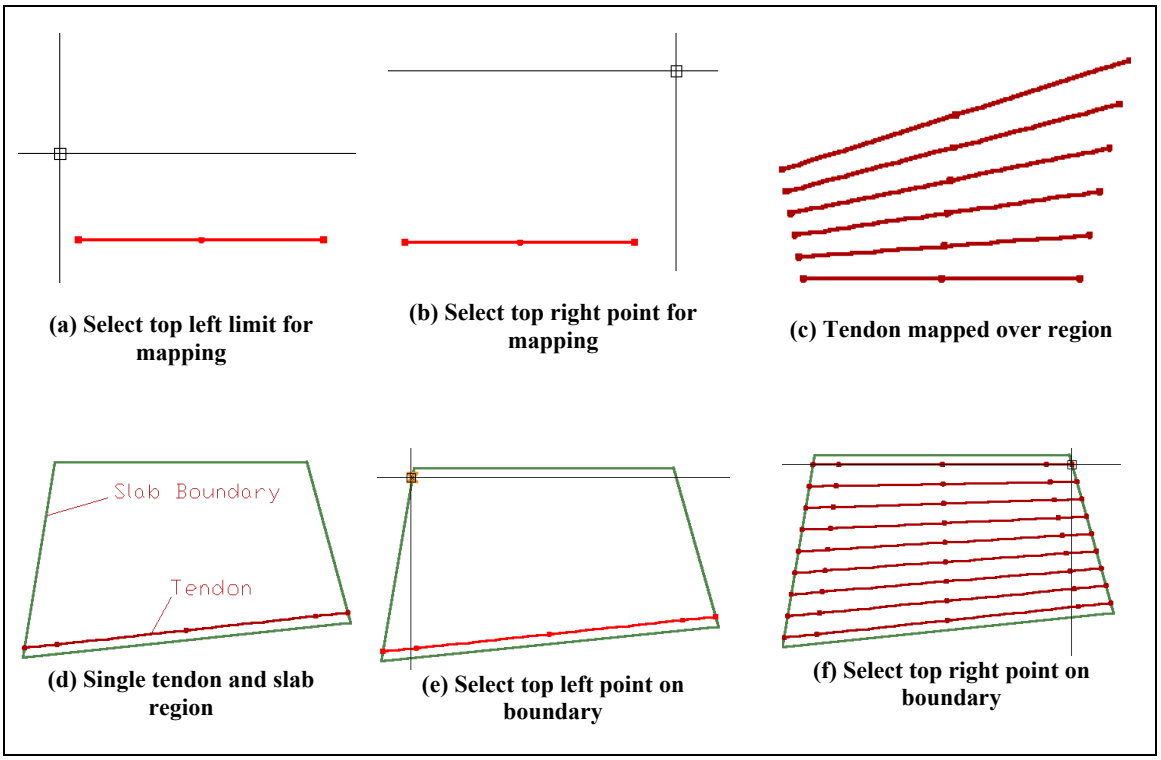
- When you click on the second point, the tendon replicates itself.

The area, shape and other parameters of the tendons will be identical to the master tendon used for mapping part (a) of the figure.

The same procedure is followed for a slab region shown in **Fig. 5.4-4** (d) through (f). When mapping over a slab region, make sure that:

- you use the snap tool (*Snap to Nearest* in this case) and snap on the slab boundary, and
- snap at a point somewhat away from the slab edge in order to avoid a tendon falling exactly at slab edge. Typically, 150mm (6 inch) is a reasonable distance.

The three-dimensional view of the mapped tendon over the slab region is shown in **Fig. 5.4-5**.



(a) Select top left limit for mapping

(b) Select top right point for mapping

(c) Tendon mapped over region

(d) Single tendon and slab region

(e) Select top left point on boundary

(f) Select top right point on boundary

FIGURE 5.4-4 STEPS IN MAPPING A TENDON OVER A SINGLE REGION

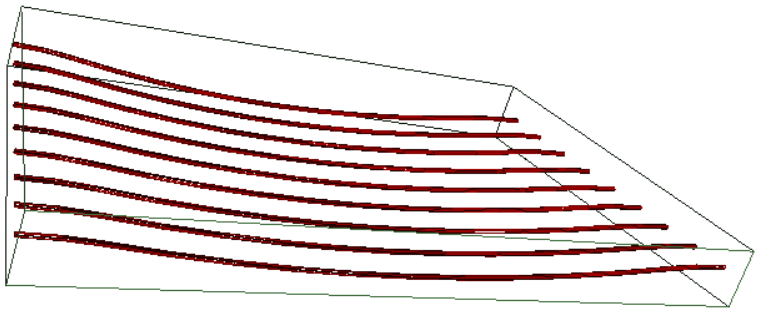


FIGURE 5.4-5 THREE-DIMENSIONAL VIEW OF TENDON MAPPED OVER THE REGION

Tendon mapping over two or more regions is done in a similar manner. The steps are illustrated in **Fig. 5.4-6**, with a three-dimensional view of the outcome in **Fig. 5.4-7**. Since there are two regions, the master tendon selected for mapping (**Fig. 5.4-6 (b)**) needs to have two spans.

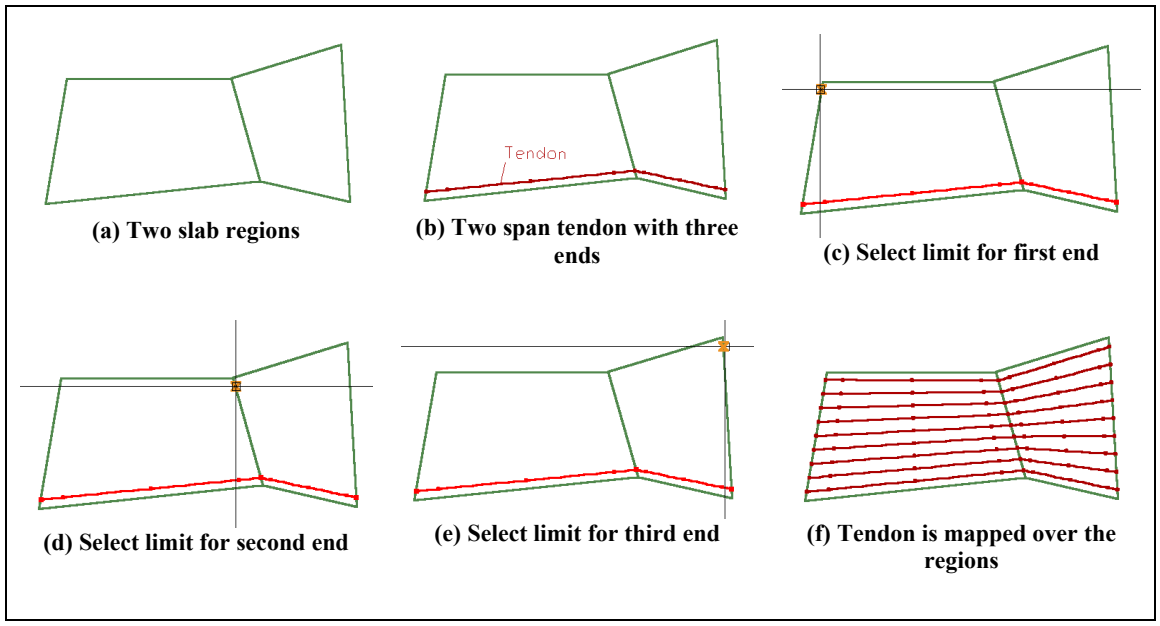


FIGURE 5.4-6 STEPS IN MAPPING A TENDON OVER MULTIPLE REGIONS

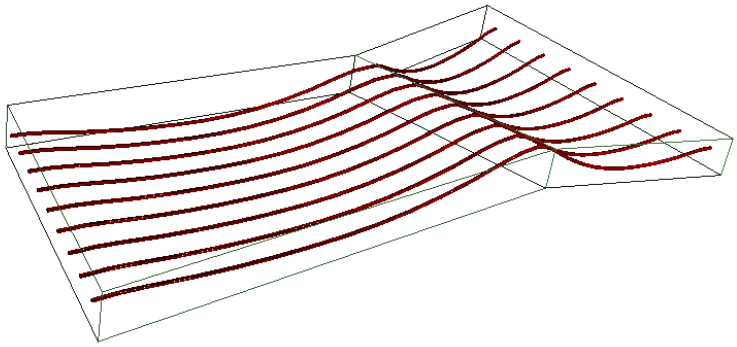


FIGURE 5.4-7 THREE-DIMENSIONAL VIEW OF TENDON MAPPED OVER TWO REGIONS

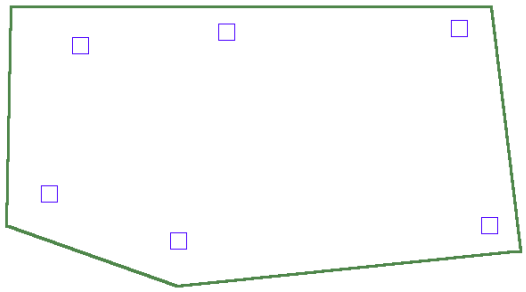
The procedure for tendon mapping using the optimized option works in a similar manner as explained previously except that the number of tendons is calculated using user predefined parameters.

The number of tendons distributed will be calculated based on the average precompression and percent balanced dead load which is input by the user. The program uses these parameters to optimize a number of tendons for the given region where the tendon is to be distributed. Limits for the spacing can also be defined by parameters entered by the user such as enabling to enforce 8 times slab thickness or setting a maximum spacing value. The tendon properties such as number of strands, effective force and shape will be inherited from the initial tendon selected.

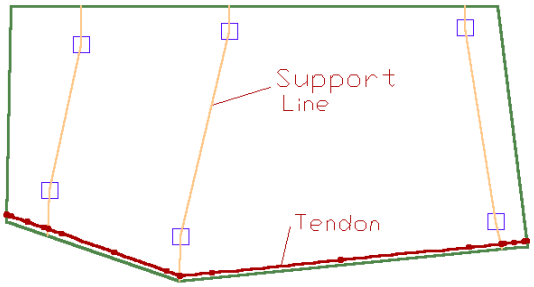
Example

Consider the column-supported slab shown in (**Fig. 5.4-8**). And, let us assume that we want the distributed tendons to be placed from left to right, and the banded tendons in the up-down direction. The three support lines drawn in part (b) of the figure break the slab into four parts in the up-down direction.

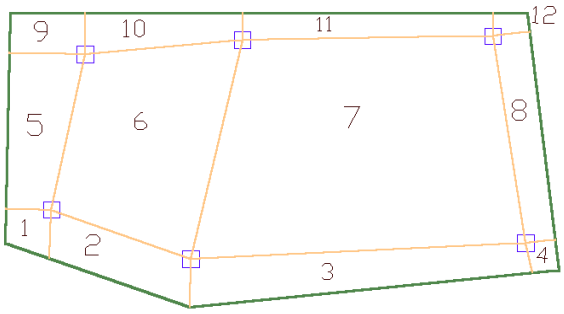
We start by drawing a master tendon next to the slab edge (b). The master tendon will be drawn with four spans, since there are four regions. We open the property box of this tendon and define the minimum cover, tendon area, profile, stressing and other features that we want common among all the distributed tendons. In the selection of profiles, we choose a cantilever profile for the first and last spans.



(a) Plan of a column supported slab. Tendons are to be distributed in left-right direction and banded in up-down direction



(b) Plan showing the support lines in up-down direction to be used for tendon mapping and a four span tendon to be used for mapping the distributed direction



(c) Using the support lines in right-left direction, the slab is subdivided into 12 regions for tendon mapping

FIGURE 5.4-8 TENDON MAPPING OF A COLUMN SUPPORTED SLAB

The steps to follow are illustrated in parts (a) through (d) of **Fig. 5.4-9**. Part (e) of the figure shows the completed mapping. Part (f) is its three-dimensional view.

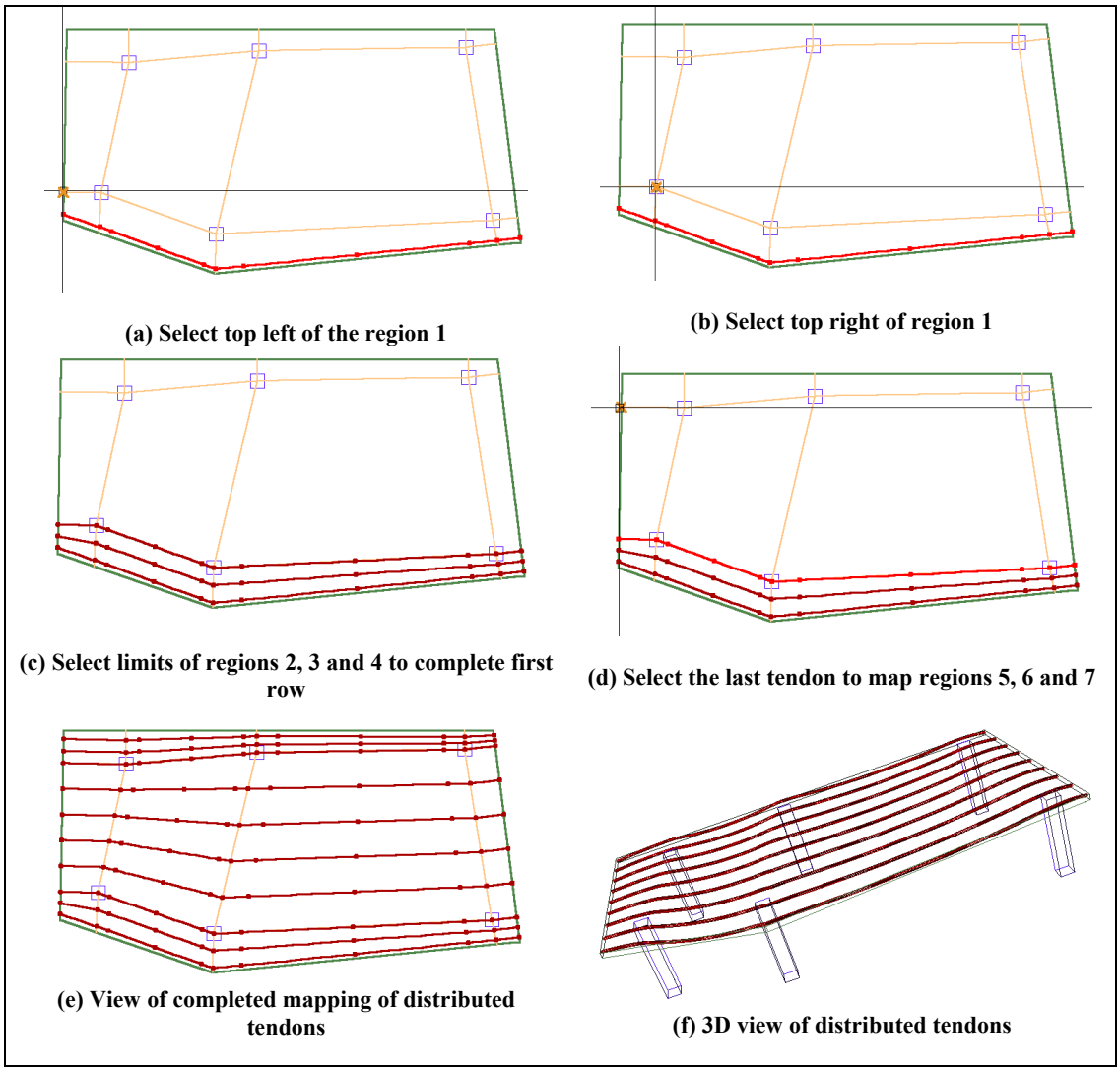


FIGURE 5.4-9 TENDON MAPPING OF COLUMN SUPPORTED SLAB

Map Banded Tendon

This tool is used to generate tendons that are banded along a user-specified guideline (Support Line). It generates a group of concentrated tendons on each side of the guideline. Each group will have the same total cross-sectional area, material properties, stressing conditions, and number of tendon spans, but each group will also adjust itself to the geometry of its location.

The procedure for mapping of banded tendons is illustrated by way of the same slab used in the preceding example. Click on the map banded tendons tool. You will be prompted to select a support line. In this example, the support line along the central columns is selected. This will

activate the dialog window for banded tendons which contains four sections as shown (**Fig. 5.4-10**).

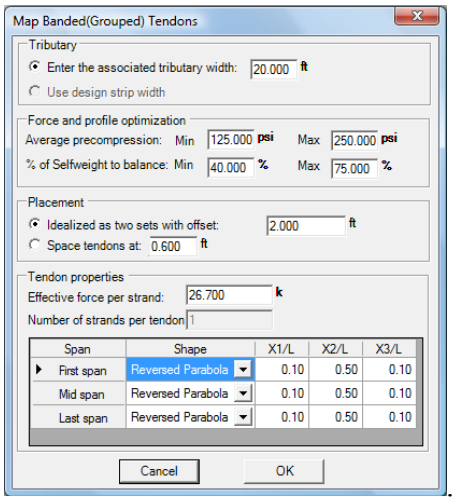


FIGURE 5.4-10 DIALOG WINDOW FOR BANDED TENDONS

If the design strips had been previously generated the option to use the design strip width for the tributary would be available. If this is not the case the tributary width can be manually entered.

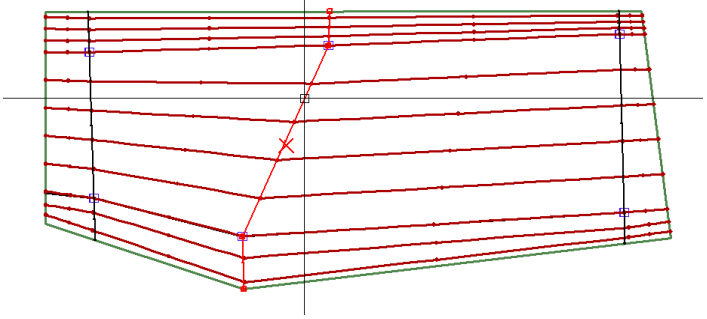
As in the case for the optimized distributed tendons, the banded tendons are automatically calculated using the average precompression and percentage of balanced dead load values entered by the user. The option is also given to idealize as two sets of tendons with a user defined offset (2ft) or to distribute the tendons at a specific spacing. If the second option is selected, the number of strands per tendon will become active and the program would calculate the number of tendons based on the total number of strands required. For common building structures and unbonded tendons this distance is 600 mm (2ft). The total width of the band would be 1200 mm (4ft). For this example the default of 600 mm (2ft) was selected.

Figure 5.4-11 (b) shows the band generated along the support line selected at specified spacing of 2ft.

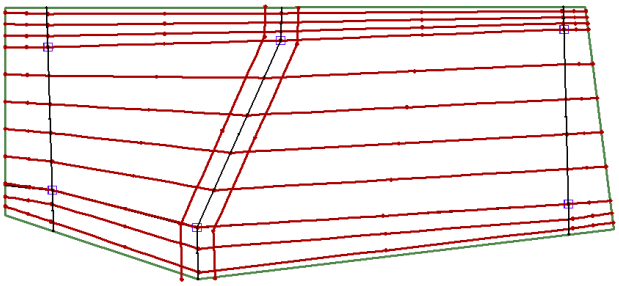
The three-dimensional view of the band created for the central support line is shown in **Fig. 5.4-11(c).**

The same procedure is repeated for the creation of banded tendons along the other two support lines in the up-down direction.

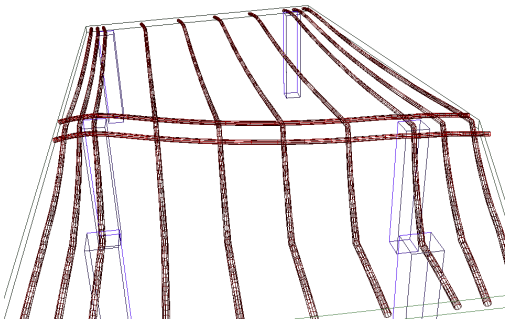
Since the tendon bands are created with a constant offset from their guide (support line), it is possible that their ends will not coincide with the slab edge, such as shown in **Fig. 5.4-12 (a)**. In such cases, you need to pick the end of the tendon and bring it to either the slab edge, if it is a stressing end, or within the slab, if it is a dead end. The steps are illustrated in **Fig. 5.4-12**[6].



(a) Select a support line for mapping the banded tendons. The band will be concentrated over a width along the selected support line.



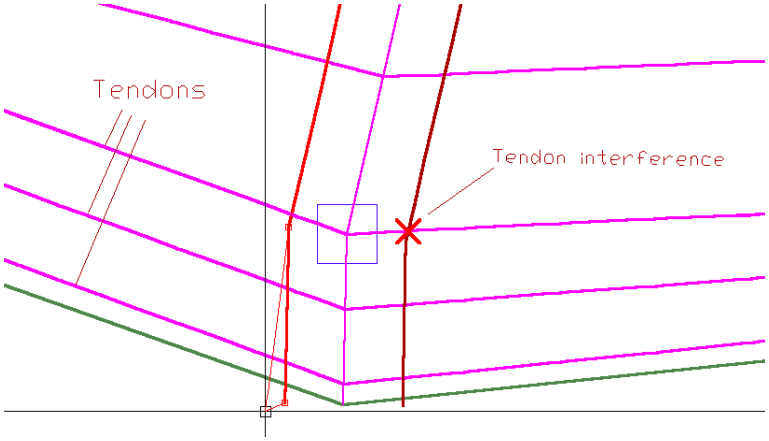
(b) The band created automatically using the values as input in the banded tendon dialog box.



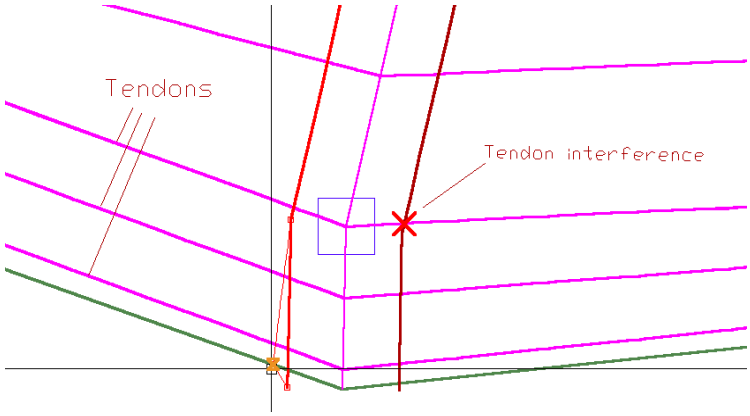
(c) Three-dimensional view of the slab showing the banded tendons over the central line.

FIGURE 5.4-11 STEPS IN THE CREATION OF BANDED TENDONS

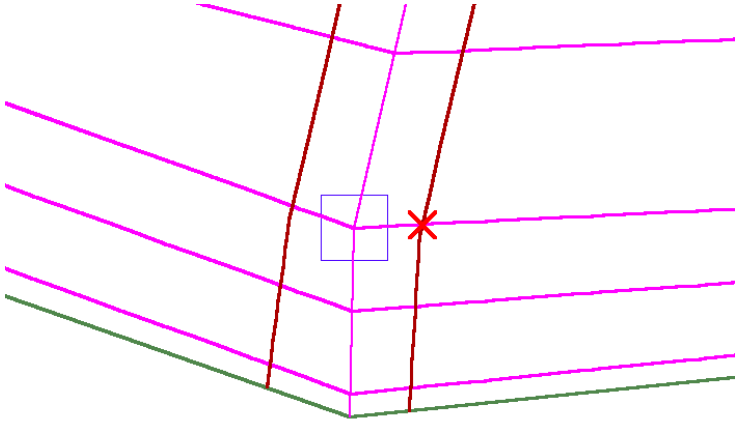
[6] The cross mark at one of the tendon intersections indicates interference of the two tendons in space. That is to say at the location marked, the two tendons partially or fully overlap. This is explained in a later section of the manual.



(a) Pick the tendon end that is outside the slab edge and drag it to the slab edge.



(b) Snap the tendon end to the slab edge using the "snap to nearest tool."



17.

(c) The ends of both banded tendons are adjusted to slab edge.

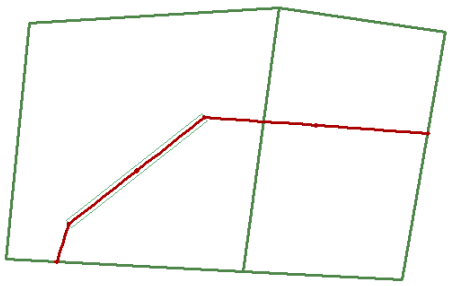
FIGURE 5.4-12 ADJUSTMENT OF TENDON END TO SLAB EDGE

5.4.3 Dynamic Tendon Modeling (DTM)

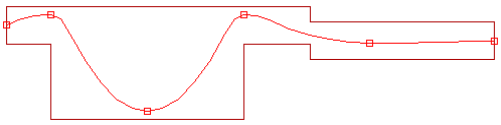
With the Dynamic Tendon Modeling of the program, you can view one or several tendons in elevation, each displayed within the outline of the concrete that contains it. More importantly, you can graphically edit each in a dynamic mode. That is to say, you can pick one of the hot spots of a tendon with the mouse, and drag it to a new location. The tendon profile adjusts itself to the new location graphically, and updates automatically the associated data in your structural model. The data updating is done in the background and will be fully transparent to you. Obviously, you can double click on each tendon, whether on plan or in elevation and open its property window to view/edit the updated values.

Select one or several tendons and click on the *Display Tendon Elevation* tool . The program will display the tendon elevations at an insertion point of your choice. For clarity of view, you may increase the display scale of the members depth[7].

Figure 5.4-13(a) shows a floor plan consisting of two slab regions, each with a different thickness and a beam. A tendon, as shown, was simply drawn on plan with the mouse. The program automatically detects the properties of the floor system and profiles the tendon as shown in part (b) of the figure. To display the elevation on the screen, pick the tendon and then click on the *Display Tendon Elevation* tool .



(a) Plan of Slab, Beam and Tendon



(b) View of Beam Within the Concrete Outline

FIGURE 5.4-13 VIEW OF TENDON WITHIN CONCRETE OUTLINE

[7] From *Settings* pull-down menu, select *Distortion Scale* and increase the scale factor in Z-direction.

The hot spot of the tendon will become visible, once you click on it. You can see the distance between the centroid of the tendon (CGS) and the soffit of its concrete container by clicking on the *Show CGS Values From Bottom* tool () on tendon toolbar (**Fig. 5.4-14**).

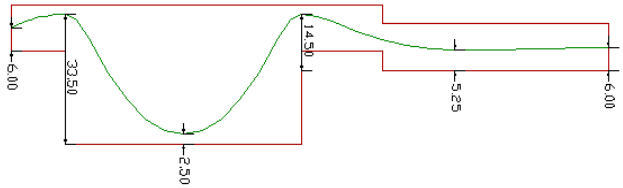


FIGURE 5.4-14 VIEW OF TENDON WITH DISPLAY OF ITS CGS FROM SOFFIT OF CONCRETE

In practice, tendon heights are rounded up or down to a user-selected value (0.25 inch for American practice, and 5mm for SI practice). You can set the rounding value and force the tendon to snap to the nearest rounding of your choice while you edit a tendon dynamically. From the *Criteria* pull down menu, open the *Tendon Height Defaults (FEM)* (**Fig. 5.4-15**). Adjust the round up value as indicated in the figure.

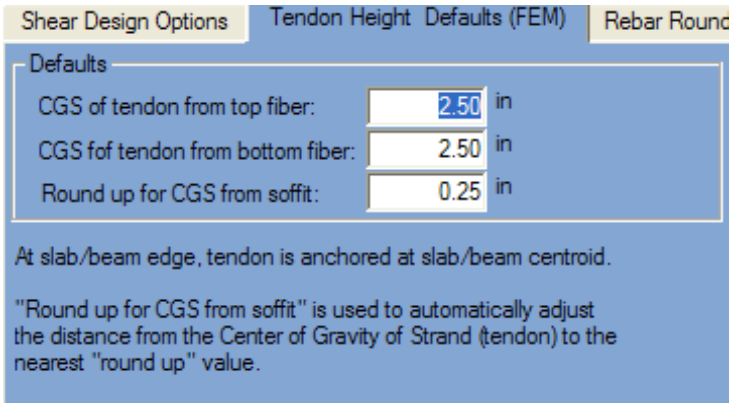


FIGURE 5.4-15 TENDON PROFILE DEFAULT SETTINGS

5.4.4 Tendon Interference

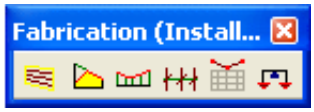
You can have the program search through the tendon layout and indicate the locations where tendons intersect. The program can detect full or partial intersection. In detecting the tendon intersection, the program uses the value of the tendon duct that you have entered in the property box of the tendons. Each tendon will be checked using its specific tendon duct for possible intersections with tendons that may have other duct diameters. In the case of flat ducts, use the width of the duct. This is the dimension that

occupies space in the vertical plane and can interfere with the path of other ducts.

To start the detection of tendon interference, simply click on its tool [8]. The locations where tendons interfere will be marked with a cross as is shown in **Fig. 5.4-12**.

You can also select two or more tendons and detect the intersection among them.

5.5 POST-TENSIONING FABRICATION (SHOP) DRAWING



Post-tensioning Fabrication (Shop) Drawing Toolbar

This toolbar contains the tools that you can use to generate a fabrication (shop drawing) for the installation of post-tensioning tendons from the design that you have completed using ADAPT-Floor Pro. It also aids you in creating reports associated with installation of tendons, such as tendon elongations and estimate of quantities. The function of each of the tools is explained below:



Tendon Display Manager. Opens the Tendon Display Manager (**Fig. 5.5-1**) where you can select which information to be displayed for tendons.

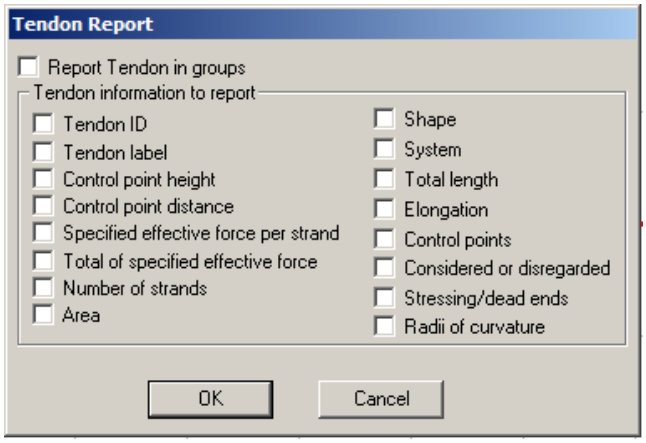


FIGURE 5.5-1 TENDON DISPLAY MANAGER

[8] This operation is very time consuming, since the proximity of each tendon in space is checked against all other tendons in the structure.

Elongation at Stressing. Compiles the elongation values of the tendons that you have selected as "calculate force" at stressing. The elongation reported accounts for the tendon's friction and seating loss. **Table 5.5-1** shows these reported values for one tendon.

TABLE 5.5-1 TENDON ELONGATION, INDIVIDUAL TENDON

Tendon(ID,Label)	Jack(1st,2nd)	Seating(1st,2nd)	Elongation (1st,2nd,Total)
	kN	mm	mm
1,Tendon 1	145824.000,145824.000	6,6	80,0,80

Display Tendon's Chair Height. Opens the Tendon Chair window shown in **Fig. 5.5-2**.

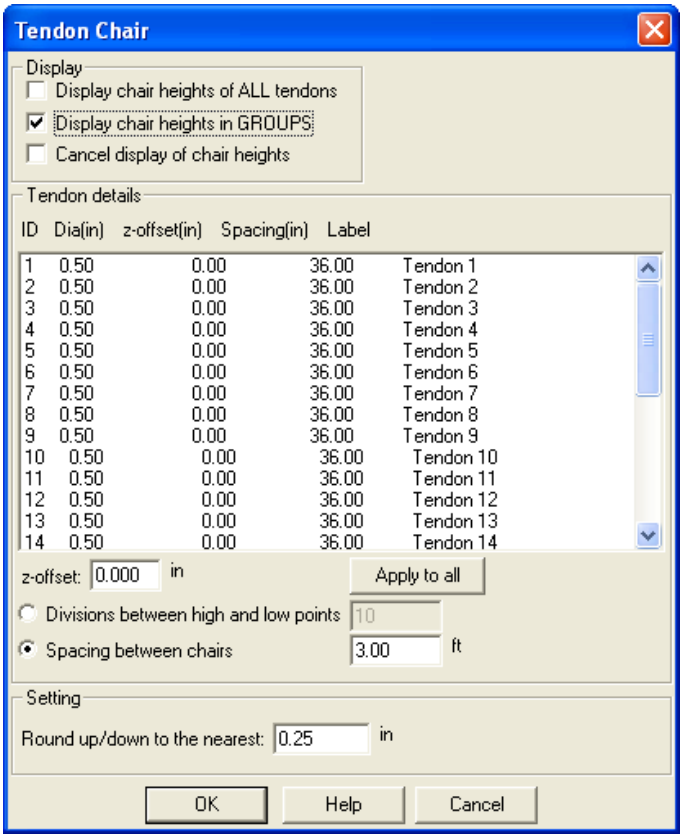


FIGURE 5.5-2 TENDON CHAIR WINDOW

In this window you can:

- Choose to display the chair heights of either all tendons or of tendon groups only.
- Specify the z-offset which is defined as the distance between the centroid of prestressing steel of a tendon to the tendon's geometry center.
- Define the spacing between the chairs as either the number of divisions between high and low points or as the distance between the chairs.
- For the setting of the chairs, you can make the program round up or down to the nearest value that you specify.

When the chair heights are displayed, the standard symbol indicating the number of strands per tendon is added to each tendon (**Fig. 5.5-3**).

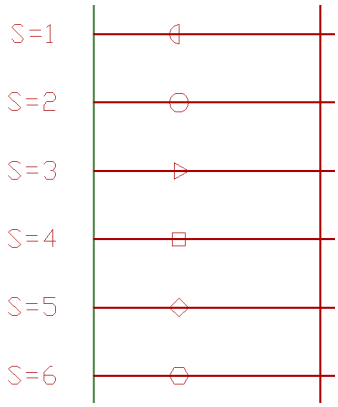


FIGURE 5.5-3 STANDARD SYMBOLS INDICATING THE NUMBER OF STRANDS PER TENDON



Tendon Spacing Tool. Displays the distances between selected tendons in your tendon layout as indicated in the following example (**Fig. 5.5-4**).

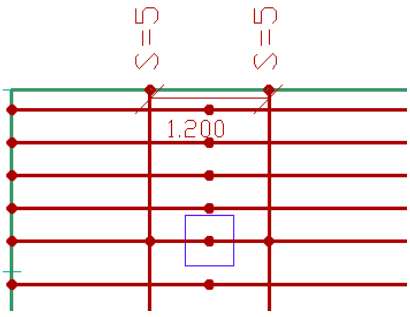


FIGURE 5.5-4 TENDON SPACING EXAMPLE



Tabular Reports. Lets you choose between several tendon-related reports as shown in **Fig. 5.5-5**.

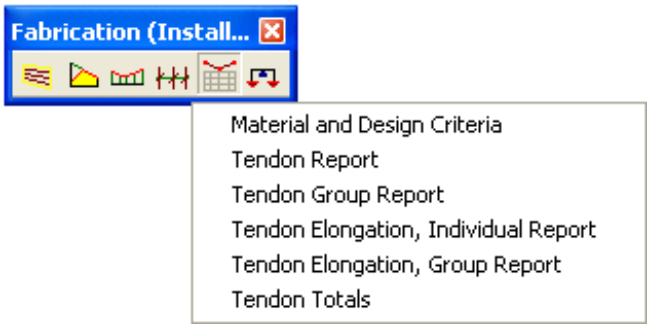


FIGURE 5.5-5 TABULAR TENDON REPORT OPTIONS



Quantities. Displays the report which contains the quantities for concrete, mild steel, and prestressing material required for your project. **Table 5.5-2** shows a typical report.

TABLE 5.5-2 TYPICAL QUANTITY REPORT

CONCRETE

Material	Unit Price	Quantity	Total Price
	Euro/m3	m3	Euro
Concrete 1	80.00	21.60	1728.00

MILD STEEL

Material	Unit Price	Quantity	Total Price
	Euro/kg	kg	Euro
MildSteel 1	3.00	224.49	673.48

PRESTRESSING MATERIAL

Material	Unit Price	Quantity	Total Price
	Euro/kg	kg	Euro
Prestressing 1	4.00	339.39	1357.57

Chapter 6

DESIGN CRITERIA – MATERIAL PROPERTIES

6.1 MATERIAL 63
6.1.1 Concrete 63
6.1.2 Mild Steel 64
6.1.3 Prestressing 64
6.2 CRITERIA 65

To a great extent, the design criteria is dictated by the building code. However, within the limits of each code, there are some parameters that you will have to decide. ADAPT-Floor Pro comes with default values for all the design criteria. The default values of the program are those most commonly used in the industry. For your first run, you may choose to execute your structural model with the program's default parameters. After a successful execution of data and its validation, browse through the default options and modify them to meet your particular requirements.

The specifics of implementation of each of the building codes in ADAPT-Floor Pro are given in its own Technical Note. The following covers an overview of the criteria. Where reference to a particular code becomes necessary, IBC (International Building Code) is used.

6.1 MATERIAL

You can access the input dialog box for material properties from the *Materials* pull down menu. It has three menu items as follows:

- Concrete
- Mild Steel
- Prestressing

6.1.1 Concrete

Select the *FEM* option of the *Concrete* menu item to open the materials input dialog box (**Fig. 6.1-1**). In this dialog window, you can define one or more concrete material properties. Each concrete material will have its own *Label*. The program will give a label to each material you add to the list, but you have the option of modifying the label.

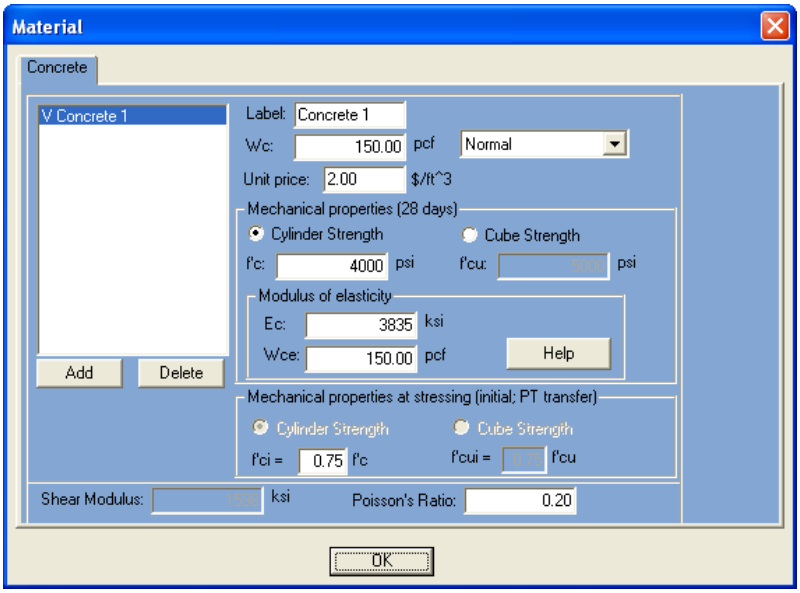


FIGURE 6.1-1 CONCRETE MATERIAL DIALOG WINDOW

6.1.2 Mild Steel

Non-prestressed steel is grouped in the *Mild Steel* category for input data. The properties that you need to specify are listed in **Fig. 6.1-2**. Material properties for mesh reinforcement should also be entered in the same dialog box.



FIGURE 6.1-2 NON-PRESTRESSED STEEL DIALOG BOX

6.1.3 Prestressing

You enter the material properties for prestressing steel in the input dialog window shown in **Fig. 6.1-3**.

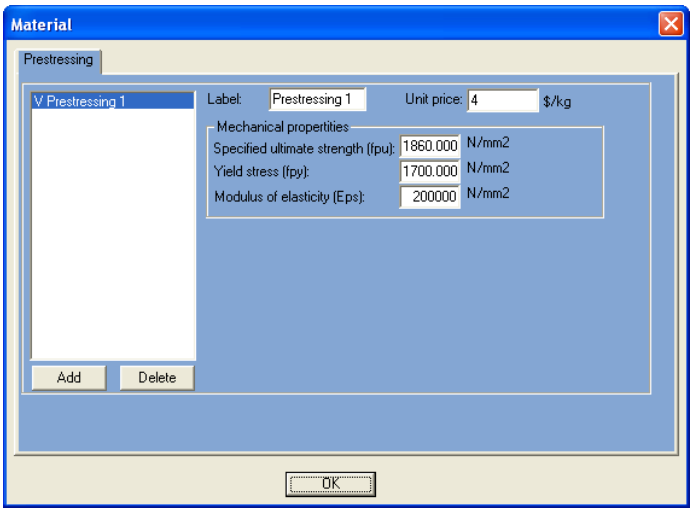


FIGURE 6.1-3 PRESTRESSING MATERIAL INPUT WINDOW

6.2 CRITERIA

- **Design Code**. **Figure 6.2-1** shows the list of building codes available at the time of preparation of this manual. Most probably, your program will display a larger selection, as more building codes are currently being added to the program. For each building code, the suggested strength-reduction factor, or material factors, will be displayed on the same screen. You have the option to edit these.

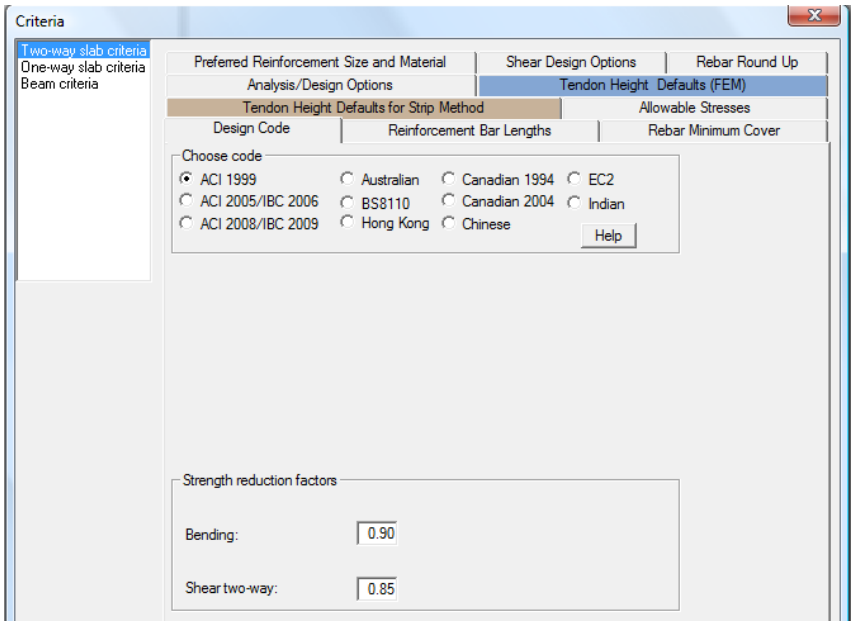


FIGURE 6.2-1 BUILDING CODE SELECTION WINDOW

- **Allowable Stresses**. The suggested values for the allowable stresses of each code are listed in the "Allowable Stresses" tab (**Fig. 6.2-2**). This tab will be displayed only if the program is opened with the prestressing option (PT) selected. Typically, the allowable stresses for both the final condition (service) and initial condition (transfer of prestressing) are listed.

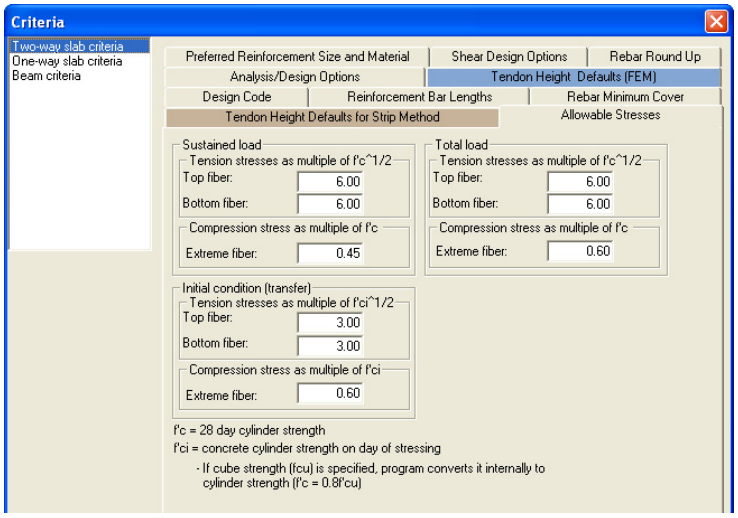


FIGURE 6.2-2 ALLOWABLE STRESSES FOR PRESTRESSED MEMBERS

- **Shear Design**. The shear design options are listed in a separate tab (**Fig. 6.2-3**). For one-way shear (beams and one-way slabs), the program uses stirrups (links, ties), where needed. For punching shear reinforcement, you have the option to select between shear studs or stirrups. Other options, such as shear band, are being added to the punching shear reinforcement alternatives.

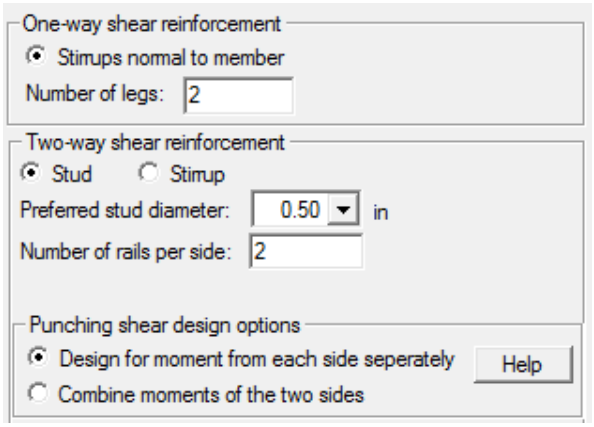


FIGURE 6.2-3 SHEAR REINFORCEMENT OPTIONS

- **Preferred Reinforcement Size and Material**. The bar sizes and material for the main reinforcement, one-way shear stirrups and two way shear reinforcement can be selected using the dialog window in **Fig**. **6.2-4**.

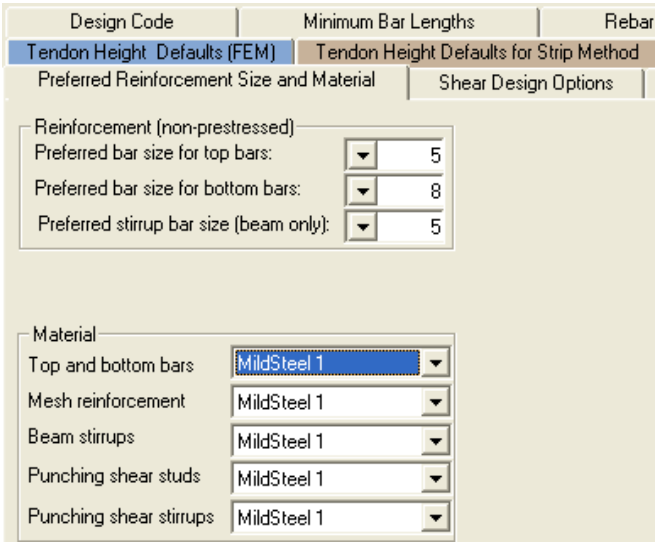


FIGURE 6.2-4 BAR SIZE AND MATERIAL SELECTION DIALOG WINDOW

- **Cover to Main Longitudinal Reinforcement**. The program assumes that your slab project is likely to have two layers of reinforcement at the top and two layers at the bottom. Each of the layers associates with design of the slab in one of the two orthogonal directions. You need to specify the cover to the outermost layer at the top and the outermost layer at the bottom. Knowing the size of the bar, the program will automatically calculate the cover to the inner layers. **Fig. 6.2-5** shows the dialog window you will use to specify the cover.

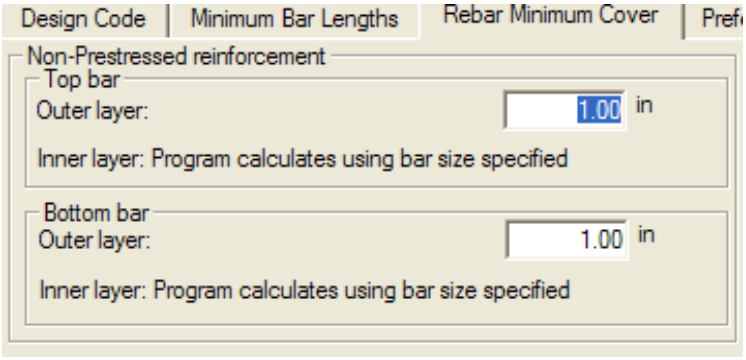


FIGURE 6.2-5 DIALOG WINDOW FOR REINFORCEMENT COVER

Choosing which rebar layers shall be considered for which direction is done in the property window of the design strips. Each design strip will be associated with one of the principal directions of the reinforcement layout.

- **Rounding Up Rebar Length**. The length of rebar calculated by the program based on the design criteria is generally not a round number. For practical purposes, however, it is common to specify the length in rounded up values. In the dialog window of **Fig. 6.2-6** you can specify the value to which the program should round up the length of the reinforcement calculated, before reporting it in its output.

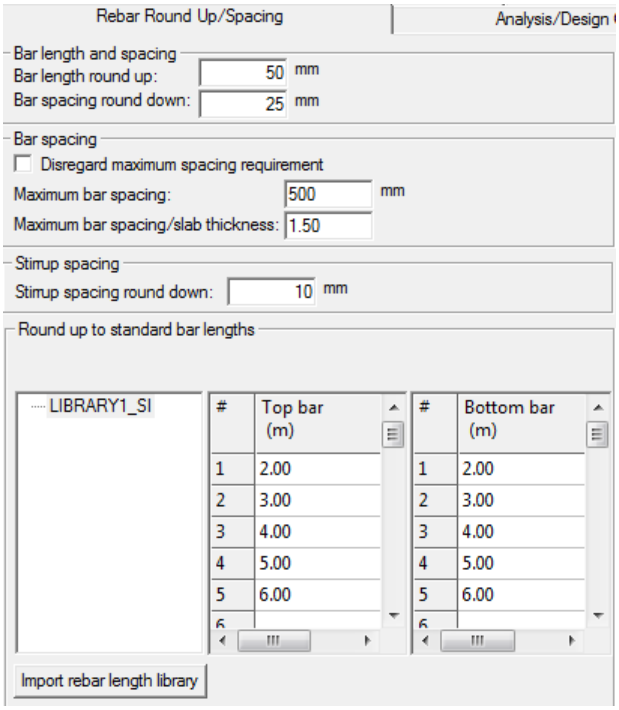


FIGURE 6.2-6 REBAR ROUND UP WINDOW

The second frame enables you to define the maximum spacing of bars on plan. This option is available only if the "RC only" option is selected.
The third frame enables you to specify the value to which the program should round down stirrup spacing.
The forth frame "Round up to standard bar lengths" allows you to define libraries with standard top and bottom bar lengths.
The left text field displays your libraries in a tree structure. You can add, clone, delete and rename libraries by right clicking on a library name in the text field.
Each bar length can be changed by double clicking on the cell in either the top bar or bottom bar table that displays the bar length. A cursor appears and the length can be edited.

The rebar library which is highlighted when clicking the OK button on the Criteria window becomes the active rebar library.

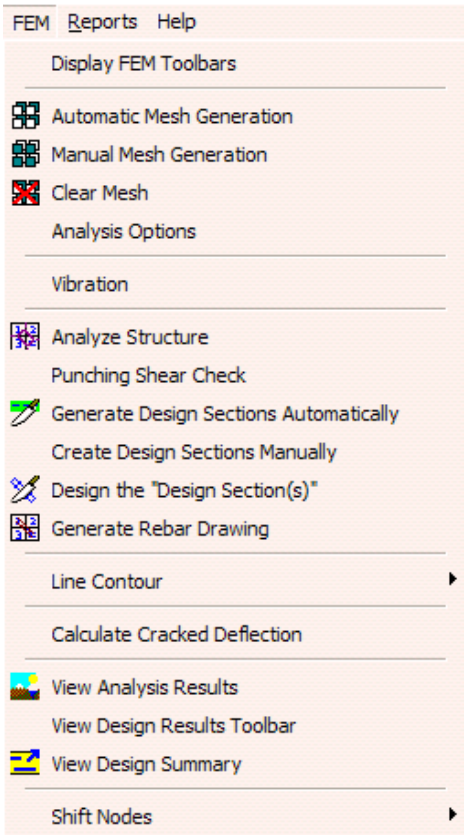


FIGURE 6.2-7 GENERATE REBAR DRAWING

When you generate the rebar drawing on the screen (FEM -> Generate Rebar Drawing, **Fig. 6.2-7**), you have two options. You can either choose "Calculated Lengths" which takes into consideration the values you entered for round up in the "Reinforcement Bar Length" tab or "Library Lengths," where the program takes the calculated rebar lengths and searches in the active rebar library for the next rebar length. . The next longer rebar length is then displayed on the screen. If there is no such rebar length specified in the rebar library, i.e. the calculated rebar is longer than the longest bar length in the active rebar library, the calculated rebar is displayed on the screen.

- **Rebar Length**. The bar length is determined by the program based on the stress values, or the strength requirements. However, IBC/ACI requires that regardless of the other considerations, the length of a bar deemed satisfactory for the minimum requirements of a post-tensioned slab shall not be less than a given value. The value is entered as default for the IBC/ACI in the dialog window of **Fig. 6.2-8**. Where in other codes no

prescription of minimum bar length is given, the IBC/ACI value is used. You have the option to edit these values. Specify zero to disable the provision.

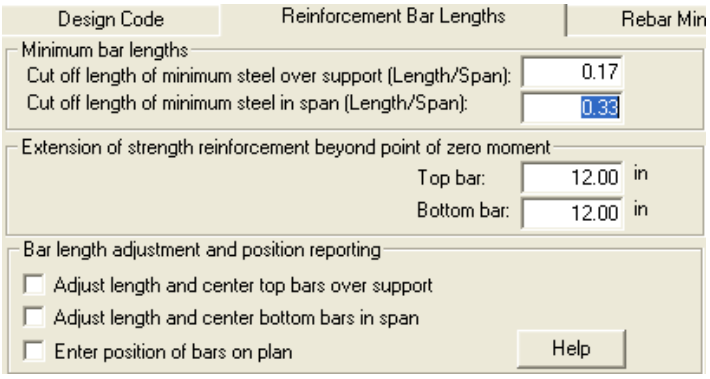


FIGURE 6.2-8 BAR LENGTH DIALOG WINDOW

The second frame in **Fig. 6.2-8** enables you to set minimum lengths for the strength reinforcement bars. Edit the default value as needed.

There is yet another consideration in the bar length reported by the program. The program checks the reinforcement requirements at the face of support, at the face of a drop cap/panel, and at the design sections created between two supports in any given span. If a span is subdivided into a large number of divisions, the program will have a rebar value at each division and can give a more accurate account for the length of the bar. The approximation in the bar length is equal to the length of a division (distance between two design sections) in a span. Hence, if you want an accurate length for the bar, you must change the default of the program (generally 12 divisions) to a finer division. You can make this change in the property box of a support line.

The third frame in **Fig. 6.2-8** enables you to display and/or adjust the position of reinforcement on plan. When the option “Enter position of bars on plan” is selected, the program will display the distance of a bar end to the centerline of the nearest support (**Fig 6.2-9**).

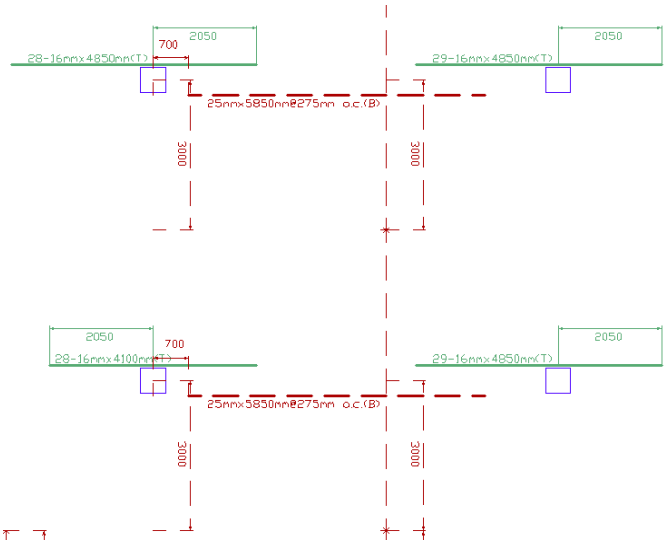


FIGURE 6.2-9

When "Adjust length and center bars …" is selected, the program finds the larger length of the bar over the support or span centerline, makes the total length twice that of the larger side and centers it over the support or midspan. Dimension line text size can be modified through the "Font height" column of the Rebar item in the Select/Set View Items, , tool.

- **Tendon Profile Values**. The program places a tendon with a minimum distance from the top and bottom fibers of the concrete outline as given in the first two data fields of **Fig. 6.2-10**. Once created, you can view and modify the tendon profile. If you choose to modify the tendon profile graphically, using the dynamic tendon modeling (DTM) as described earlier, the graphical work will move the tendon in steps of the distance given in the "round up" data field in the figure. This will result in a round number for the tendon height.

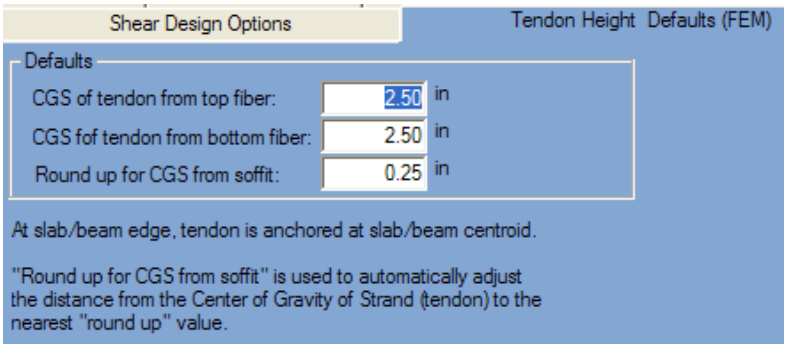


FIGURE 6.2-10 DIALOG WINDOW FOR TENDON PROFILE

- **Analysis/Design options**. Analysis and design options are listed in the "Analysis/Design Options" tab (**Figs. 6.2-11** and **6.2-12**).

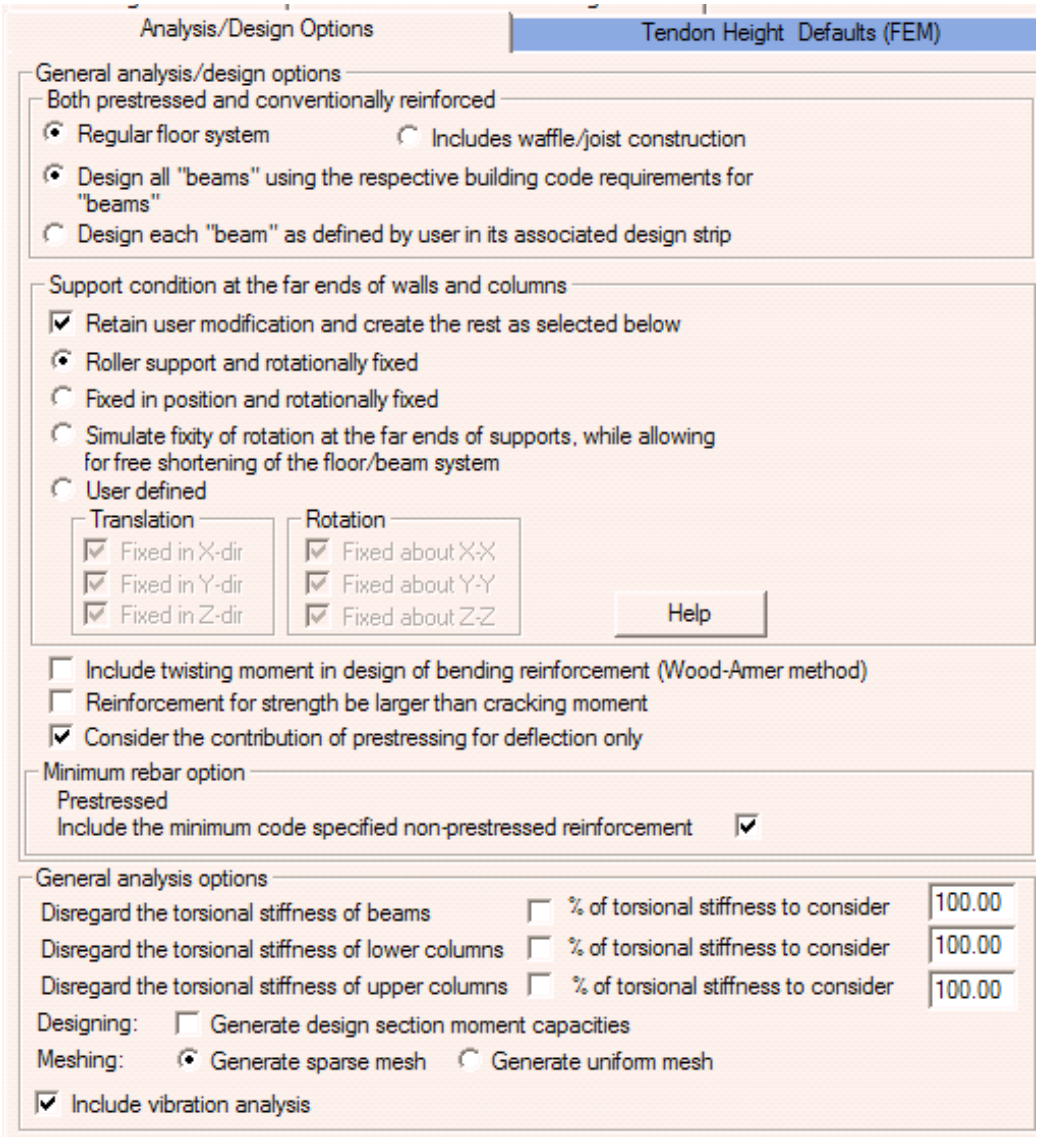


FIGURE 6.2-11 ANALYSIS/DESIGN OPTIONS FOR PRESTRESSED STRUCTURES

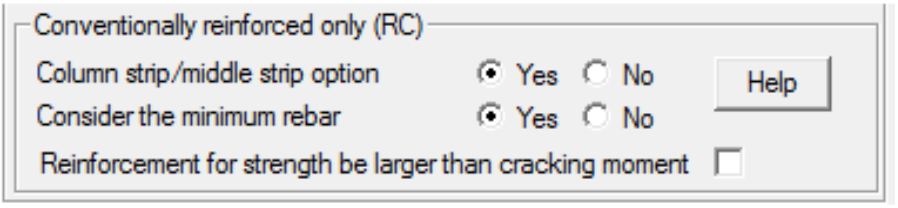


FIGURE 6.2-12 ADDITIONAL OPTIONS FOR CONVENTIONALLY REINFORCED STRUCTURES

The following explains the function of each of the entries of the Analysis/Design Options tab:

- *Non waffle/joist floor systems*. If "Regular floor system" is selected, at design stage the program differentiates between "structural" and "architectural" beams. In its code check the program disregards the stem of an architectural beam. That is to say, if a design section crosses an architectural beam, the program will not design that beam for reinforcement. The program will only place corner bars if entered by the user. In connection with a given support line, a beam is defined structural if it contains at least one "support" (vertex) of the given support line within, or on its boundary. **Figure 6.2-13** illustrates the three options for treatment of a beam with a support line vertex. The option depends on the angle between support line and the centerline of the beam.

    - For α <= "angle_for_beam" [30 degrees] consider as beam
    - For α >= "angle_for_support" [70 degrees] consider as support
    - For the remainder of the angles, consider as architectural beam

    The parameters "angle_for_beam" and "angle_for_support" are included in the program's initialization file and can be modified by the user.

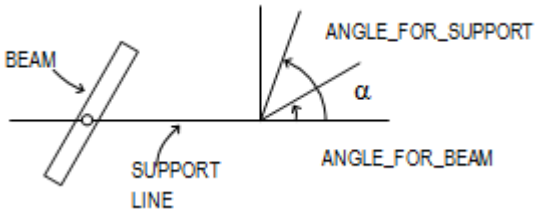


FIGURE 6.2-13 TREATMENT OF A BEAM WITH SUPPORT LINE VERTEX

The following example illustrates the treatment of beams based on two considerations, namely (i) the presence of at least one support line vertex on the beam and (ii) the angle between the beam's centerline and the support line. Refer to **Fig. 6.2-14**, where the support line has vertices within the boundary of beam A, B and D. In relation to the support line shown, only beams A and D are structural, where beam A is considered as support and beam D is considered as a proper beam. The other two beams B and C are architectural.

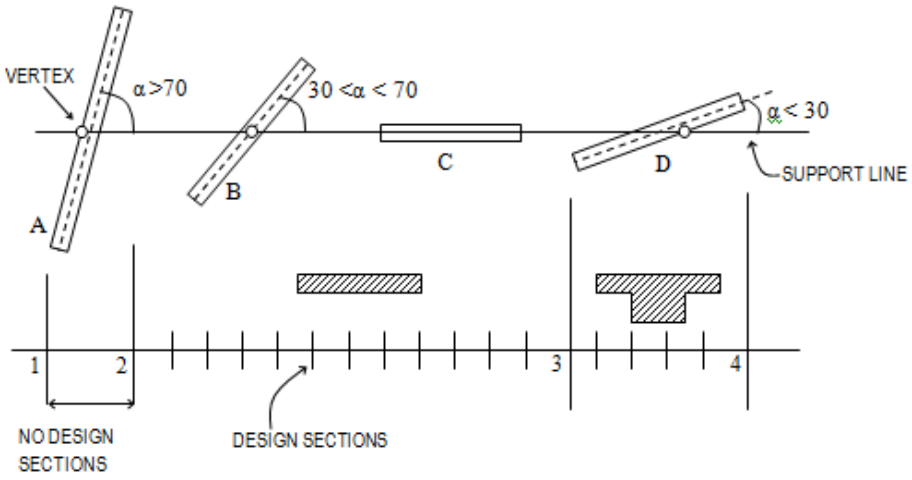


FIGURE 6.2-14

When a beam is considered as a support, the program will not cut a section through it for the slab design. The reinforcement for the beam will be determined from the design sections in the orthogonal direction. . For example, for region 1-2 shown in **Fig. 6.2-14** there will be no design sections. In a region where one or more structural beams are encountered, the program will consider the cross-section of the steam as shown in region 3-4 of **Fig. 6.2-14**.

Drop caps, drop panels, slab bands, and changes in slab thickness and changes in elevation of structural components will be faithfully represented in the geometry of a design section and accounted for regardless of their relation to support lines.

- *Includes waffle/joist construction*. If "Includes waffle/joist construction" is selected, the program considers each waffle or joist stem that is within a design section as a beam entity. For example, if a design section cuts 6 stems, the program does as follows:
    - Combines the six stems and their topping slab into an idealized single beam with the web thickness equal to the sum of the webs of the individual stems. Or, as the case may be, the entire will be converted to a single equivalent I-section.
    - The idealized section will be used for stress and strength checks.
    - The actions reported by the program will be that of the design section, not the individual stems.
    - The rebar displayed in builder sum and the report will be that of the entire design section.

- *Design all "beams" using the respective building code requirements for "beams."* The program will use the "beam" option of the respective code to design and detail the structural components that were created as "beams." In other words, if a design section cuts a beam the program will design it using the respective code provisions for beams.
- *Design each design strip as defined by user in its associated design strip*. In this case, the program will design each "support" line according to the user selection in the property box of the respective support line.
- *Create supports at far ends of walls and columns*. This feature automatically inserts supports at the far ends of walls and columns using the program's default restraints prior to the analysis. If conditions other than the program's default are selected in the "Criteria" pull-down menu, the program selects the latter. To modify any of the program specified boundary conditions, (i) Build->Support->Create supports at far ends of walls and columns (**Fig. 6.2-15**), (ii) double click on the support you want to view/edit to open its property window, and finally (iii) change the support restraints to the condition of your choice.

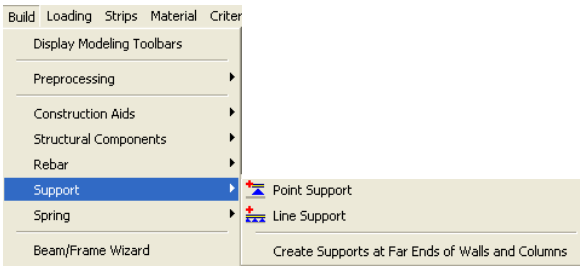


FIGURE 6.2-15

If in the Criteria pull-down menu the option "Retain user modification" is checked, the program will not overwrite these modifications when you analyze your model. If you do not check this option the program overwrites the user modifications with the boundary conditions specified in the Analysis/Design option tab.

- *Consider the contribution of prestressing in strength check*. When not checked, the program considers the force due to Prestressing for the tendons that are "considered," but does not allow for the contribution of tendons in the "strength" check. This scenario applies when a user adds tendons to a project for deflection or crack control.
- *Include the minimum code-specified non-prestressed reinforcement*. When this option is selected the program will

calculate minimum reinforcement based on the provisions of selected design code.

- *Column strip/middle strip option*. Column strip/middle strip option is available only if the "RC" option is selected. The Column strip/middle strip will be generated only when either of the following is triggered:
    - In "Strips" pull down menu user selects "Generate Design Strips for FEM."
    - When in FEM, pull-down menu user selects "Generate design sections automatically."

  When the support lines display is invoked, in addition to the original support lines, the program will also display the support lines that are associated with middle strips. For middle strips, the number of sections is determined as larger of two numbers defined with:

  "spacing_of_middle_strip_design_sections" 3 ft [1m]

  "Minimum_of_middle_strip_design_sections" 36

  The parameters "spacing_of_middle_strip_design_sections" and "Minimum_of_middle_strip_design_sections" are included in the program's initialization file and can be modified by user.
- *Consider code-required minimum rebar*. If selected by the user, the program calculates the minimum reinforcement that is based on respective code provisions.
- *Include twisting moment in design of bending moment (Wood-Armer method)*. If selected by the user, the program calculates the reinforcement for ultimate limit state according to the Wood-Armer method.
- *Reinforcement for strength be larger than cracking moment*. If the user selects "Yes" the program performs the code-specified calculation for the design moment to be larger than a multiple of the cracking moment.
- *Torsional Stiffness Options*. This control enables the user to disregard the torsional stiffness of beams, lower and upper columns as desired. The user also is given the flexibility to manually define the percentage of torsional stiffness to consider for the beams, upper and lower columns.
- *Generate design section moment capacities*. If the user selects this option, the design section moment capacities are calculated for all design sections. For automatically and manually generated design sections, the moment capacities can be observed in the appropriate tabular report (Reports->Single Default Reports->Tabular->Design Section Data->Design Section Moment Capacities). For manually

generated design sections the moment capacities can also be found on the "General" tab of their property box. Design section capacity graph is plotted together with the moment envelope (**Fig. 6.2-16**). To see the graph go to FEM->View Design Summary. Open moment diagram and select envelope from the drop down list. Click on the Design Moment Capacity frame and select positive or negative moment capacities.

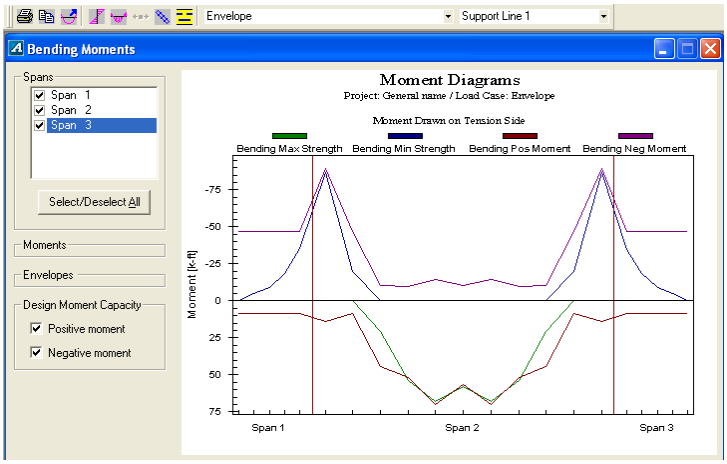


FIGURE 6.2-16

- *Meshing options*. This control enables the user to select the meshing option. The default is "sparse mesh."
- *Vibration Analysis*. This option enables the user to initiate an automatic vibration analysis of the structure. The analysis is performed according to the vibration load combinations and mass participation factors set by the user from the FEM->Vibration input window. After the analysis is complete, the user can view vibration results from the FEM-View Results menu.

Chapter 7

LOAD CASES AND LOAD COMBINATIONS

7.1 LOAD COMBINATIONS 81
7.1.1 Load Combinations Set Up 81
7.1.2 Load Combination Label 83
7.1.3 Analysis/Design Options 83
7.1.4 Long Term Deflection Template 84
7.2 PATTERN LOADING 85
7.3 REDUCTION OF LIVE LOAD 89
7.4 LATERAL (WIND/SEISMIC) LOADS 90
7.4.1 Lateral Load Case Definition 91
7.4.2 Load Combinations 92
7.4.3 Input/Editing of Lateral Forces 93

The definition of loads and load cases, including the creation of new load cases for gravity and other general conditions are described in the Modeler User Manual. Further, if you plan to import loads and or geometry from a general purpose multi-story building program, the ADAPT-Floor Pro program has special features which enable you to import wind, seismic and other loads from solutions obtained by other programs. In the following text, you will learn about load combinations.

7.1 LOAD COMBINATIONS

7.1.1 Load Combinations Set Up

To open the load case combinations dialog window (**Fig. 7.1-1**), click on the *Loading* pull-down menu, choose the *Load Combination* menu item, and finally click on the *FEM* side menu. Depending on the building code selection you have made, and whether or not you have implemented prestressing, the program automatically defines a number of load cases and load combinations. You can add to or edit the program's selection. The design code you have selected is displayed in the middle of the dialog box – ACI in the case illustrated in **Fig. 7.1-1**.

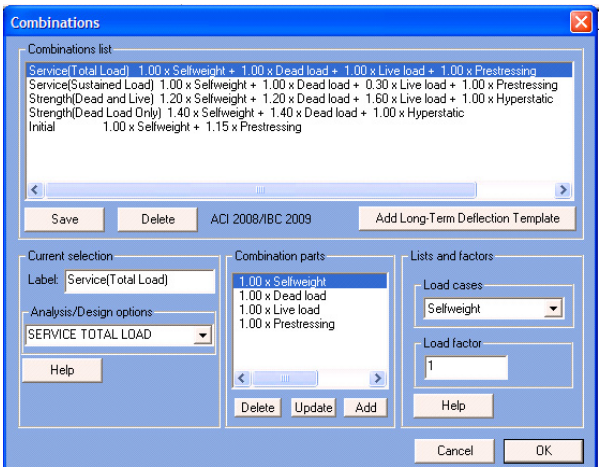


FIGURE 7.1-1 LOAD COMBINATION DIALOG WINDOW

The current load cases of the program are listed in the *Load cases* combo box on the bottom right of the figure. The program obtains a solution for each load case, then combines them, each with its respective *Load factor* as listed in the *Combination list*. Only the load combinations will be available for you to view, and are included in the program's reports. As a result, if you wish to see the program's solution for a particular load case,

such as “Selfweight,” you must define it as a load combination on its own (**Fig. 7.1-2**).

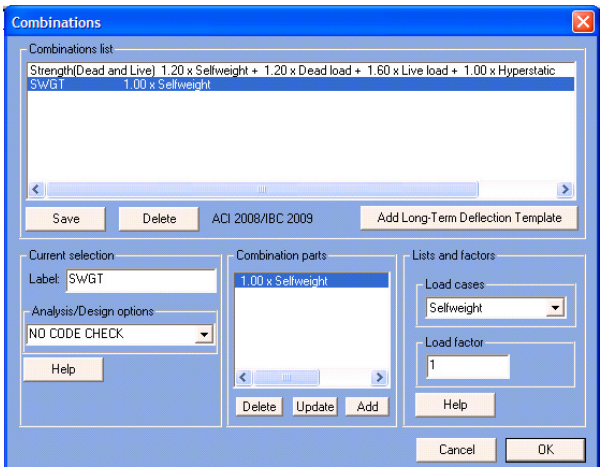


FIGURE 7.1-2 SELFWEIGHT LOAD COMBINATION

The default load cases of the program are:

- Selfweight
- Dead Load
- Live Load
- Prestressing
- Hyperstatic

The selfweight load case is determined automatically by the program using the volumetric geometry of your structural model and the unit weight you have assigned in the *Material* pull down menu. If you change the unit weight default of the program to zero, the program will automatically delete the “Selfweight” as a load case. Or, if you prefer, you have the option of excluding the selfweight load case in your load combination table.

Dead Load will be shown among the load cases only if you have assigned load to it and check-marked it in the *Load Case Library* dialog window. Likewise, Live Load will appear in the load case combo box, if you have assigned a load to it and check-marked it in the *Load Case Library* dialog window.

Prestressing and Hyperstatic load cases will appear in the load case combo box if you define at least one prestressing tendon in the structural model.

The prestressing load case determines the displacements and stresses in the structure resulting from the actions of the prestressing tendons. The hyperstatic (secondary) load case gives you the moments and shears in the structure due to the restraint of the supports to free the deformation of the structural model. The prestressing load case is the one you will be using for the service condition of the structure – that is to say for the stress check in concrete, crack control, and deflection checks. The hyperstatic load case is used only for strength (safety) load combinations.

7.1.2 Load Combination Label

Each load combination can be assigned a unique name. You can enter the load combination name in the field marked *Label* in **Fig. 7.1-2**. In its reporting, the program uses the name you entered in the label field to report the associated results.

7.1.3 Analysis/Design Options

Results of each load combination will be treated by the program using one of the following options:

- Serviceability
- Strength
- Initial
- Cracked Deflection
- No Code Check

For each of the load combinations, you need to select the option you want in the *Analysis/Design options* dialog box of **Fig. 7.1-2**.

The "serviceability" option applies the in-service requirements of the code you have selected to the results of the load combination. This includes stress checks, deflections, and crack control, as well as the minimum rebar. Likewise, the "strength" option uses the safety requirements of the code you selected to check the outcome of the load combination. The "Initial" option is used for the transfer of prestressing to the structure – jacking stage. The "Cracked Deflection" option is used for calculation of deflection with due allowance for loss of bending stiffness due to cracking. The "No Code Check" option simply reports the results of the FEM analysis without any additional processing. For example, if you are

interested to see the displacements and moments for the selfweight or prestressing, you would select this option.

7.1.4 Long-Term Deflection Template

You have the option to manually define load combinations for long-term deflection results or to use the "Add Long-Term Deflection Template" button as shown in **Fig. 7.1-3**. When this option is invoked, the program will prompt the user to enter a creep and shrinkage factor (See **Fig. 7.1-4**) and will automatically generate two additional load combinations and add these to the combination list. These combinations are "Sustained_Load" and "Long-Term." The "Sustained_Load" combination can be edited to include different load cases and factors using the input for "Combination parts" and "Load factor." This combination is automatically set to the Analysis/Design Option of "Cracked Deflection." The "Long-Term" load combination can also be edited in a similar way with the difference that "Combination parts" can include only existing "non-long-term" load combinations (see **Fig. 7.1-3**). The Analysis/Design Option of this combination is grayed out and cannot be modified. The user can add as many long-term combinations as required.

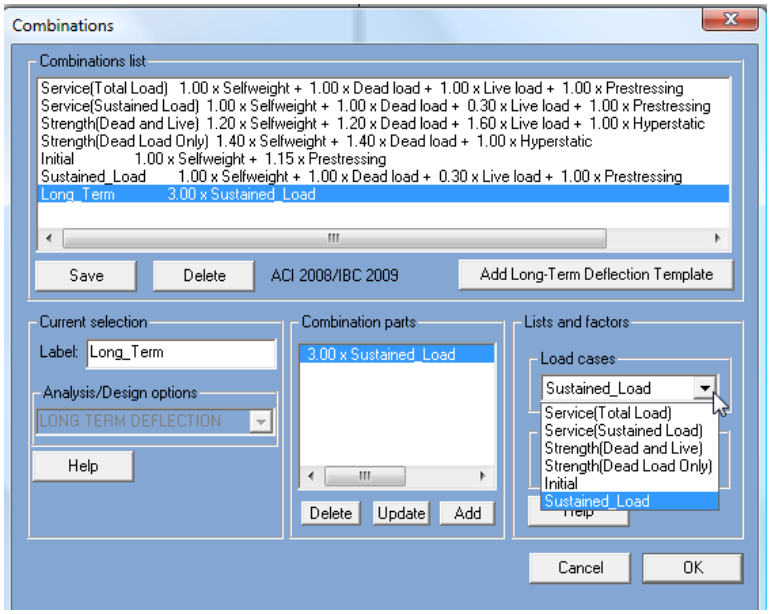


FIGURE 7.1-3 LONG-TERM DEFLECTION TEMPLATE



FIGURE 7.1-4 CREEP AND SHRINKAGE FACTOR

The purpose of this feature is to automate the calculation and reporting of long-term deflection results in lieu of manually applying a creep and shrinkage coefficient to elastic deflections in ADViewer (View Results from the FEM tab) to obtain long-term deflection results. It also serves as a template to setting up long-term deflection combinations and results. This feature can be used to combine results of different load combinations. It is important to stress that the program is not limited to its default combination and the user can edit or add combinations as required.

7.2 PATTERN LOADING

Skipping of live load, also referred to as pattern loading, is the process in which a given load is selectively placed at different locations on the structure, in order to determine the maximum adverse effect of the load on the structure. This process applies to the loads that can be either present or absent at specified locations on the structure, such as live load. ADAPT-Floor Pro allows you to define as many number of skip load arrangements (patterns) as required. The program analyzes each pattern separately, and obtains an envelope of the maximum and minimum values for use in design.

The following are the steps for skipping of live loads (refer to **Figs. 7.2-1** and **7.2-2**):

- Define the structure. **Figure 7.2-1a** shows symbolically the floor plan of a structure. Supports and other features of the floor are not shown, since these are not required for the definition of skip loading, although you may use them to determine the subdivision of a floor into skip regions.
- From the *Loading* pull down menu, select *Skip Pattern/Create Subregions* (**Fig. 7.2-3**). The same menu item is used for the reduction of live load.
- Create “skip division lines” following the instruction at the bottom of the screen (hint line). This consists of using the mouse and drawing “division lines” to subdivide the floor plan into “skip regions” (**Fig. 7.2-1b**).
- Identify the “skip sub-regions” that form a load pattern, by selecting the *Define/Edit Load Pattern* option in **Fig. 7.2-3**.

- Define a skip factor, and/or request reduction of live loading through the dialog box of **Fig. 7.2-4**.
- Open the properties box (**Fig. 7.2-5**) of each skip sub-region by double clicking on its hot spot (circle displayed in its middle). Alternatively, select a skip region and then invoke its properties box by clicking on "*Item's Properties*" tool. ( )

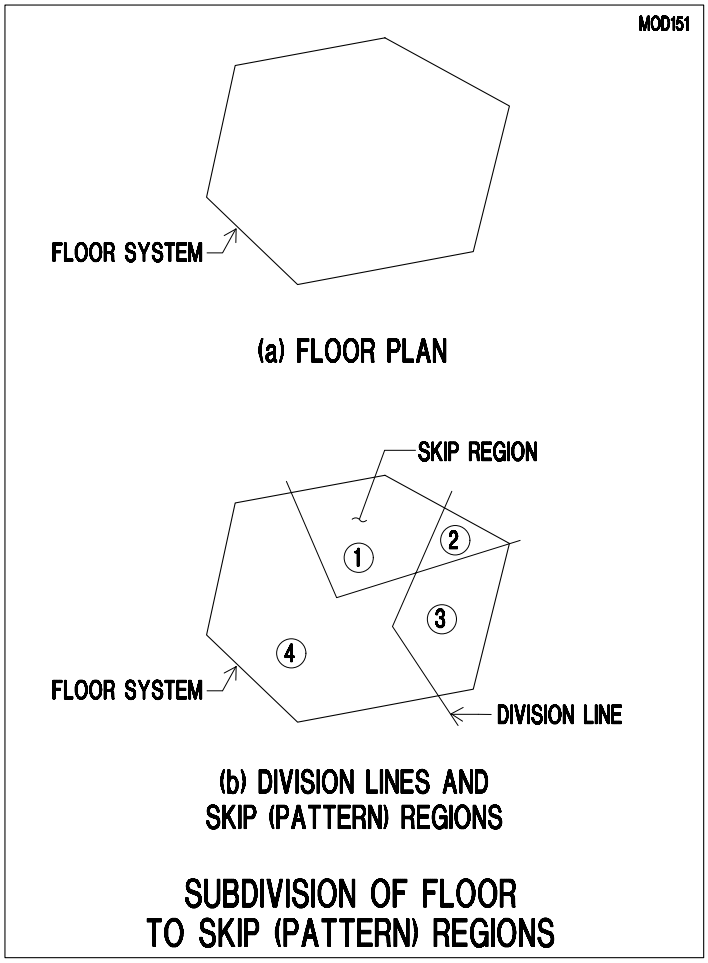


FIGURE 7.2-1

Once the skipping of live load is defined and saved, the program automatically generates a load case for each of the skip patterns along with two load case envelopes, one reporting the maximum and the other the minimum value obtained from the skipping of the live load.

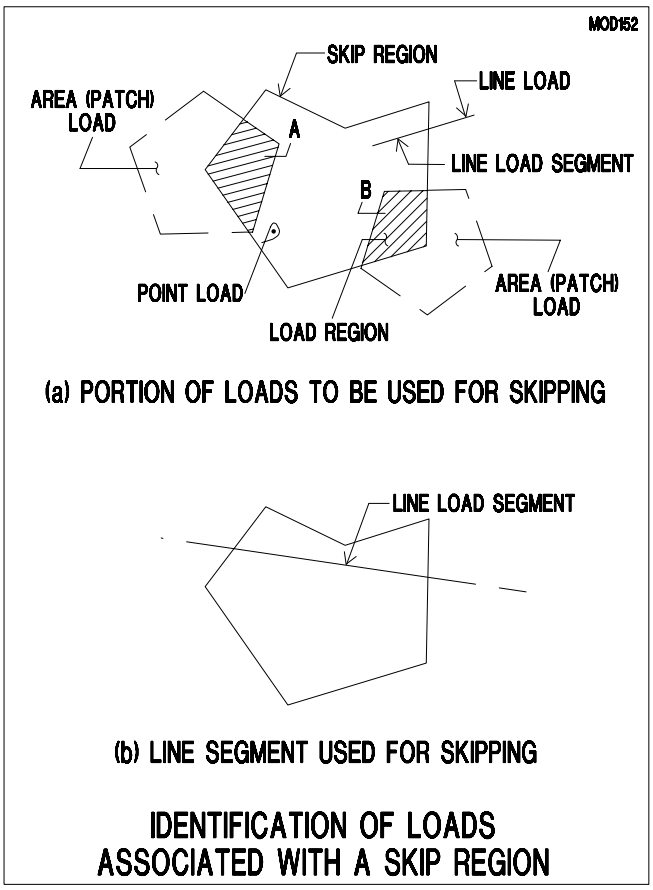


FIGURE 7.2-2

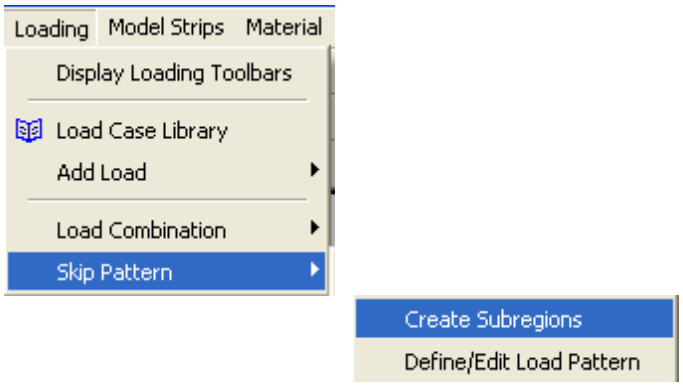


FIGURE 7.2-3 MENU FOR SKIP (PATTERN) LOADING

For example, in the floor plan shown in **Fig. 7.2-1**, four skip sub-regions are identified. If you select four skip patterns, the program will generate the following load cases and load envelopes.

Pattern_1
Pattern_2
Pattern_3
Pattern_4
Skip_####_Max
Skip_####_Min

The solution of each of the load cases can be viewed individually, reported individually and combined with other load cases.

The symbol #### represents the label you will select for the load you skip, such as "Live load." The Skip_####_Max and Skip_####_Min represent an envelope of the maximum and minimum values for all the patterns defined by you. In this example, these will be the envelope of the load cases Pattern_1, Pattern_2, Pattern_3 and Pattern_4.

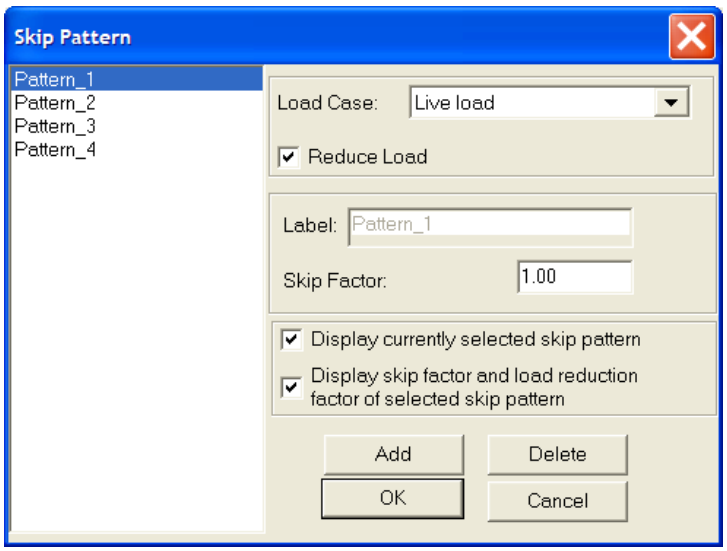


FIGURE 7.2-4 SKIP PATTERN SELECTION

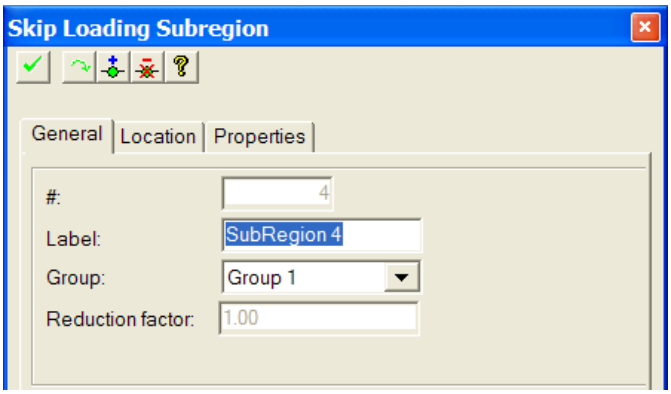


FIGURE 7.2-5 SKIP LOADING SUBREGION PROPERTIES BOX

7.3 REDUCTION OF LIVE LOAD

Live load values specified in the code are determined statistically, using a notional floor area. For floor areas larger than the notional areas used in the building codes, there is less of a likelihood of the load covering the entire area with the code-specified intensity. For this reason, the building codes generally allow a reduction in the intensity of the specified live load, when the load covers large areas. ADAPT-Floor Pro can automatically determine the reduction in the intensity of the live load specified, and accounts for it in its analysis. It also reports the reduction value used. An additional option allows the user to enter reduction values. Note that the reduction of live load for analysis is an option that you must select in your input data. Otherwise, the program applies the entire load you have specified over the area you selected.

The value used for the reduction of live load is calculated for each skip region (see the preceding section for the definition of skip regions). Hence, each skip region has its own coefficient of live load reduction. The area used for the calculation of the live load reduction in each skip region is strictly the area covered by the live load in that region. This is shown graphically by the hatched regions in **Fig. 7.2-2(a)**. The concentrated (point) loads and the line loads that are part of the live load being reduced, and that fall within the skip region under consideration, are not reduced. The reduction is applied only to the distributed loads.

If you do not define a skip region, the program applies the reduction to the entire area of each part of the live load you have defined.

For the reduction of live load, the program uses the recommendations of the IBC-2005 (International Building Code), as detailed below:

$$R = 0.08(A - 150)$$

Where,

R = reduction factor, not to exceed 40%; and
A = tributary in square feet (where the live load is applied.)

Nor more than

$$R = 23.1\ (1 + D/L_0) = < 40\%$$

Where,

D is the dead load, and L_0 the applicable live load.

7.4 LATERAL (WIND/SEISMIC) LOADS

The procedure how to manually apply lateral loads on a structure is explained step-by-step using the example shown in **Fig 7.4-1**. In general, it consists of the following steps:

- Define a lateral load case. This is done by going to the load case library and defining a new load case for the lateral forces you plan to enter.
- Using the loading toolbar, enter new loads:
  - Identify the position of a new load by selecting the wall/column of your choice.
  - Click Generate Lateral Point Loads tool, , in the Loading Toolbar. The program applies point loads at far ends of selected walls/columns.
  - To enter the load magnitude or edit the loads that you have already entered double click on the load symbol and insert the load values in the activated edit box.
- Each time a lateral load case is created, its name appears in the list of load cases in the load "combination" table. The next step is to go to the load combination table and specify how you want the solution of the load case you have defined to be combined with the other cases.

The following example will walk you through the process in a more detailed manner. It is a single level substructure with two lateral load cases, one applied in X-. and the other in the Y-direction (**Fig. 7.4-8**)

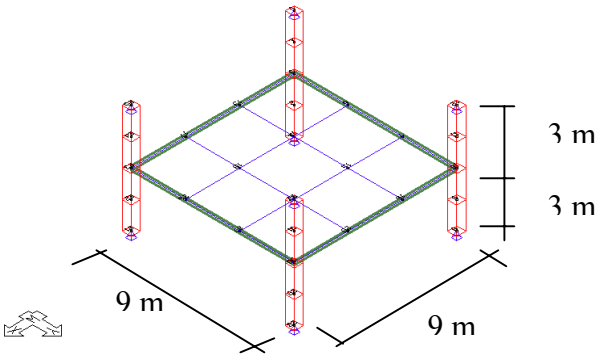


FIGURE 7.4-1 GEOMETRY OF THE SUBSTRUCTURE

7.4.1 Lateral Load Case Definition

Before entering any lateral loads, you must first define a lateral load case as described in the following.

- From the *Loading* menu, select *Load Case Library*. Check *Special lateral loads* option. The dialog window shown in **Fig. 7.4-2** will appear.
    - Click Add button. The program shows one load case with the name "Lateral_1." This is a default lateral load case which can be renamed by editing it in the data cell "Label." Note that the string "Lateral_" must be placed in front of the name selected. For example, if the load case is to be named "Wind-XX," it would have to read "Lateral_Wind-XX." It is the string "Lateral_" that signals to the program how to treat the load case.
    - Similar to gravity load cases, if you want to add or delete a lateral load case, you would use the "Add" or "Delete" buttons.
    - Once all loads are defined Click on the *OK* button to close the window.

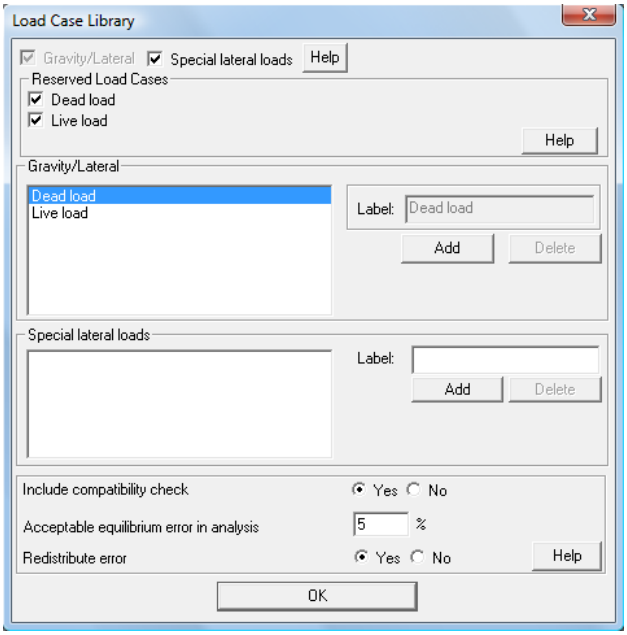


FIGURE 7.4-2 LOAD CASE LIBRARY DIALOG WINDOW

- Alternatively, the load case library can be accessed through the loading toolbar (**Fig. 7.4-3**). It is the tool third from the right.



FIGURE 7.4-3 LOADING TOOLBAR

As an example, **Fig. 7.4-4** shows the definition of two load cases Lateral_EQ_x and Lateral_EQ_y. "Dead load" and "Live load" are default load cases of the program.

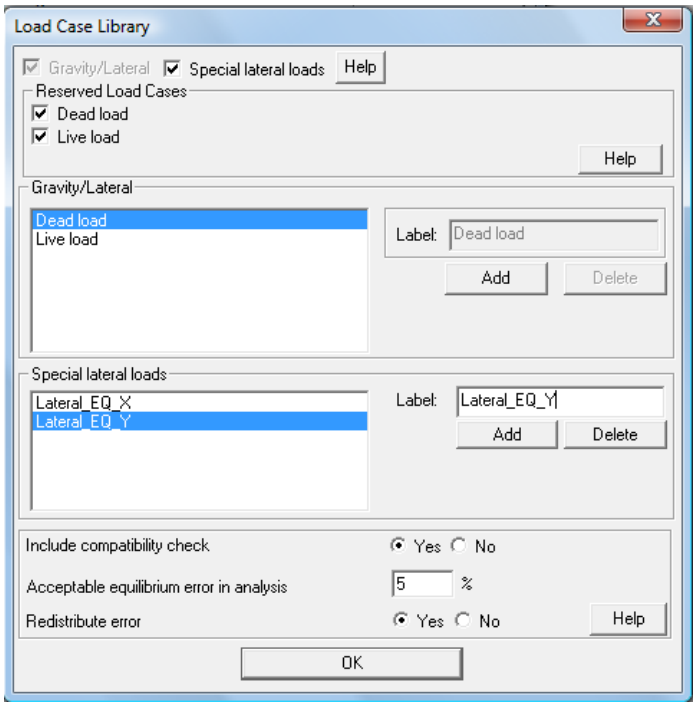


FIGURE 7.4-4 DEFINED LATERAL LOAD CASES

7.4.2 Load Combinations

To define load combinations do the following:

- From the *Loading* menu select *Load Combination*, which will display the *Combinations* dialog box.
- Click the *Load cases* drop-down box and choose the load case that will be included in the load combination. Combine this load case with the other load cases in the same manner that you would do for gravity loading.

Figure 7.4-5 shows the definition of two load combinations EQ_x and EQ_y.

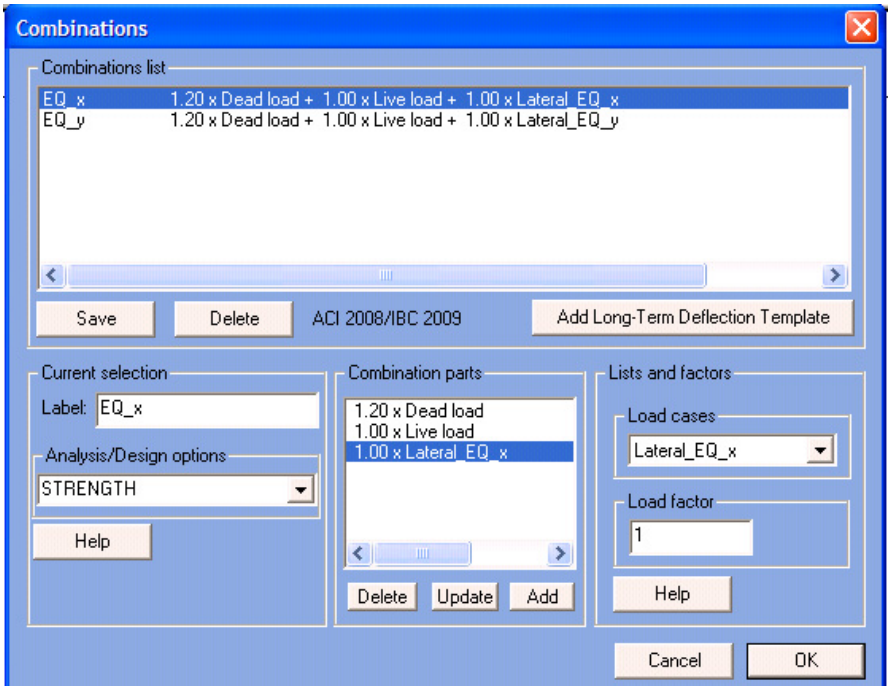


FIGURE 7.4-5

7.4.3 Input/Editing of Lateral Forces

You can input or edit lateral forces, after you have defined them in the load case library. This is explained in the preceding section.

- To input or edit forces of a previously defined load case, display the *Loading Toolbar* (**Fig. 7.4-3**). If you have already defined lateral load case, the tool will be lit.
- Select the walls and columns for which you wish to define lateral loads. Click on tool. Select lateral loads case from the drop down list that opens. Click Create button. The program gives the option of entering lateral loads for each of the columns or walls you click on. It shows the availability of the option by a hexagon symbol that appears on the component you click on. If there are columns or walls both below and above a floor slab, the best is to select a perspective view of the structure, instead of the plan view (use this tool). In the perspective view, click once on an upper wall/column and once on a lower wall/column to activate both. Obviously, you can use the convenient tools that are available for selection of all the columns or walls, if a lateral load acts on all of them.

- Next you have to define the magnitude and orientation of each of the loads. The program automatically positions the point of application of each of the loads at the top or bottom centroid of the wall/column selected by you. You have the option to change the point of application, as will be described later.

  Double click on the symbol generated for a point load to open its property box (**Fig. 7.4-6**). At this time the *Lateral Load Editing Mode* window will open (**Fig. 7.4-7**). Under the "General" tab of *Point Load* property box, enter the orientation and magnitude of the forces and moments you wish to specify.

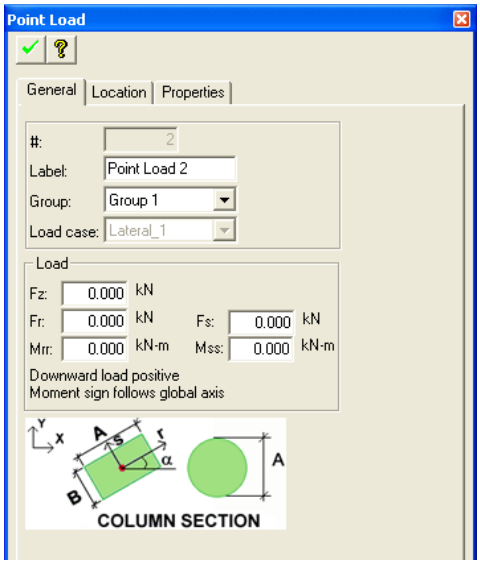


FIGURE 7.4-6 LOAD DEFINITION PROPERTY WINDOW

- When finished click the button to accept the values defined.
- Uncheck the tool.

When finished with editing of point loads for the given load case click *Exit Lateral Load Editing Mode*. (**Fig. 7.4-7**). Double click on the load that you want to modify. This will activate *Lateral Load Editing Mode*. When you finish with load editing close the *Lateral Load Editing* window.

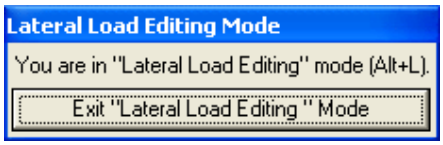


FIGURE 7.4-7

Figure 7.4-8 shows lateral forces defined for the X- . and Y-directions.

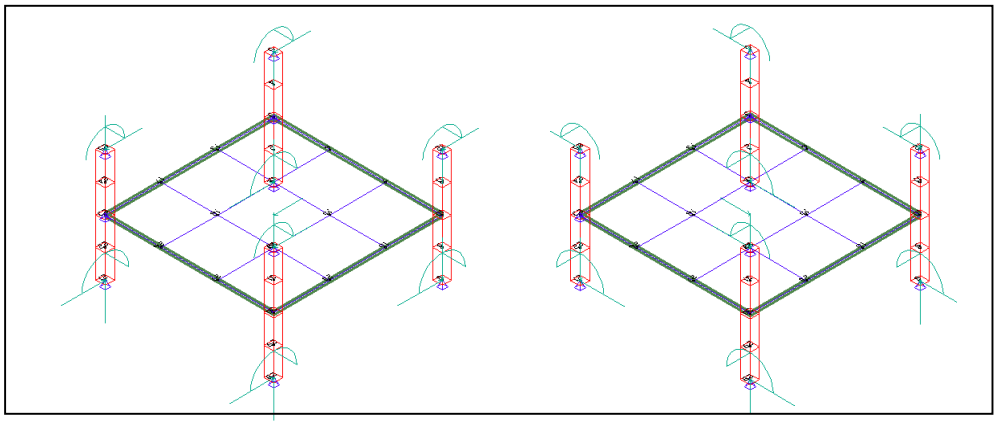


FIGURE 7.4-8 DISPLAY OF LATERAL LOADS FOR THE X AND Y DIRECTIONS

Chapter 8

ANALYSIS

8.1 OVERVIEW 99
8.2 MESHING 99
8.2.1 Imposed Location of Nodes 101
8.2.2 Basics of Automatic Meshing 103
8.2.3 Manual Meshing 103
8.2.3.1 Overview and Use of Quadrilateral Cells 103
8.2.3.2 Generation of Triangular Cells 110
8.2.4 Mesh Editing 113
8.2.5 Nodes and Finite Elements 113
8.2.6 Natural and Analysis Nodes 113
8.2.7 Node Consolidation – Node Shift 114
8.2.8 Excluder 117
8.3 ANALYSIS OPTIONS 118
8.4 ANALYZE THE STRUCTURE 119
8.5 DEFLECTIONS CALCULATION IN FLOOR PRO WITH ALLOWANCE FOR SLAB CRACKING 119
8.5.1 Overview 119
8.5.2 Procedure How to Calculate Cracked Deflection 120
8.6 CRACK WIDTH CALCULATION 123
8.7 DISPLAY LINE CONTOURS 124
8.7.1 Overview 124
8.7.2 Contour Toolbar 125
8.8 VIBRATION ANALYSIS 127
8.8.1 General 127
8.8.2 Run Vibration Analysis 128

8.1 OVERVIEW

In this chapter we will continue to explore meshing. It should be noted that many meshing stages take place in the background, without your intervention, others require that you interact with the program and make a selection. If in any of the steps the program encounters something unusual, it will stop and provide a detailed warning. You have the choice of disregarding the warning and continuing. If the program encounters an error, however, the program will in most cases not continue until you remove the error reported. Among the steps we will review:

- Create a finite element mesh for the slab regions
- Validate the mesh for cell angles, size and aspect ratio
- Apply default boundary conditions at locations that generally require support, but you have not specified one – such at bottom of a column
- Check the structure for stability in the horizontal direction, stabilize if needed
- Validate the structure, such as proper connection between a column and slab
- Generate finite element mesh for the columns and walls consistent with the cells in the slab regions
- Discretize the loads (and soil support, if applicable)
- Solve for structure's deformations for each of the load cases
- Check the validity of the solution obtained
- Combine the results obtained for each load case according to the load combination table.

8.2 MESHING

The basics of mesh generation are outlined in the Modeler User Manual. In this section, we will focus on the less common aspects of meshing. The break down of the structural model into finite elements is referred to as "discretization." In general, the discretization - also referred to as "meshing" - of the structure is done automatically by the program using ADAPT-Mesher. ADAPT-Mesher is based on adaptive mesh generation technology. It scans your structural model, detects its features and details, such as openings, steps, and creates a well proportioned set of quadrilateral cells to suit the detail of your structure. While the entire mesh generation operation is done in the background, some knowledge of the way

ADAPT does the discretization is of great help in achieving a good and expedient solution. This section explains the concept and the information necessary for a successful discretization of the structural model. Some of the less immediate, but important concepts and procedures are reported in the Appendices A and B of this manual.

ADAPT-Floor Pro uses several different types of elements in its discretization. These are:

- Flat quadrilateral or triangular shell elements
- ADAPT-Mesher uses an adaptive technology, and automatically creates almost exclusively well-proportioned quadrilateral elements, adjusted to the details of your structural model. You have the option to view and edit the mesh generated by the program. This includes creations of triangular cells, if you so decide. The shell elements are used for the slab regions, and walls.
- Frame elements
- Prismatic frames with one node at each end, having six degrees of freedom per node. The frame elements are automatically generated for beams and columns.
- Point spring elements
- These provide six degrees of restraint at a point of your choice. They are not generated automatically by the program.
- Soil support elements
- The soil support elements provide Winkler type foundation over a region defined by you.

Meshing is required for the slab regions only, including drop caps and drop panels. Meshing of other components, such as beams, columns, and walls, is carried out automatically, using the slab meshing as a template. Supports, such as the supports at the far ends of columns and walls are also handled by the program automatically, unless you wish to change the default boundary specifications, in which case, you can define the support conditions.[1]

The beam elements will be created by the program automatically to match those of the slab they support. The mesh generation of stand-alone beams, or grids with no slab support is also done automatically. You define the preferred element size.

[1] If the manual meshing option is used, point supports that are on a slab are to be identified with a "node.'

Columns are divided into a number of elements specified in the "initialization[2]" file of the program. Likewise, walls are subdivided into shell elements that line up with those of the slab they support. In the vertical direction, walls are subdivided between two or more elements as specified in the initialization of the program.

8.2.1 Imposed Location of Nodes

This section will only be of interest to users who are planning to either use manual meshing, or edit a mesh generated by the program automatically. Prior to meshing, the program scans your structural model and imposes a number of "leader" nodes and lines at selected locations. These are intended to delineate the bounds of the structural components for the finite element meshing. In meshing, nodes will be located at these points and over the imposed lines. In addition to the imposed nodes, others will be generated within the enclosed areas of slab regions.

The locations of imposed points are:
The end of walls, beams and columns

- At the intersection of structural components, such as a beam and a wall
- At the corners of slab regions, openings, drop caps and similar conditions

Similarly, there will be imposed lines that would form the sides of the finite element cells. No finite element cell cross these lines – they but against it.

- Along the edge of slab regions
- Along the edges of openings
- Along the boundary of drop caps and drop panels
- Along the length of beams, columns

Figure 8.2-1 is an example of imposed nodes and lines for meshing. It shows the partial plan of a slab with a wall and column support. The imposed nodes are at the ends of the wall (marked with A and B in part b of the figure). Other nodes will be generated between A and B along the imposed line AB. The number of elements generated along the line AB,

[2] Many of the parameters that govern the behavior of the program are listed in the program's initialization file. This is a text file "initialize_builder.txt" and is located in the subdirectory, where your ADAPT-Floor Pro program resides. The default subdivision of a column and wall in the vertical direction is two cells.

either automatically by the program or manually by you, depends on your choice and is determined by the cell size you choose.

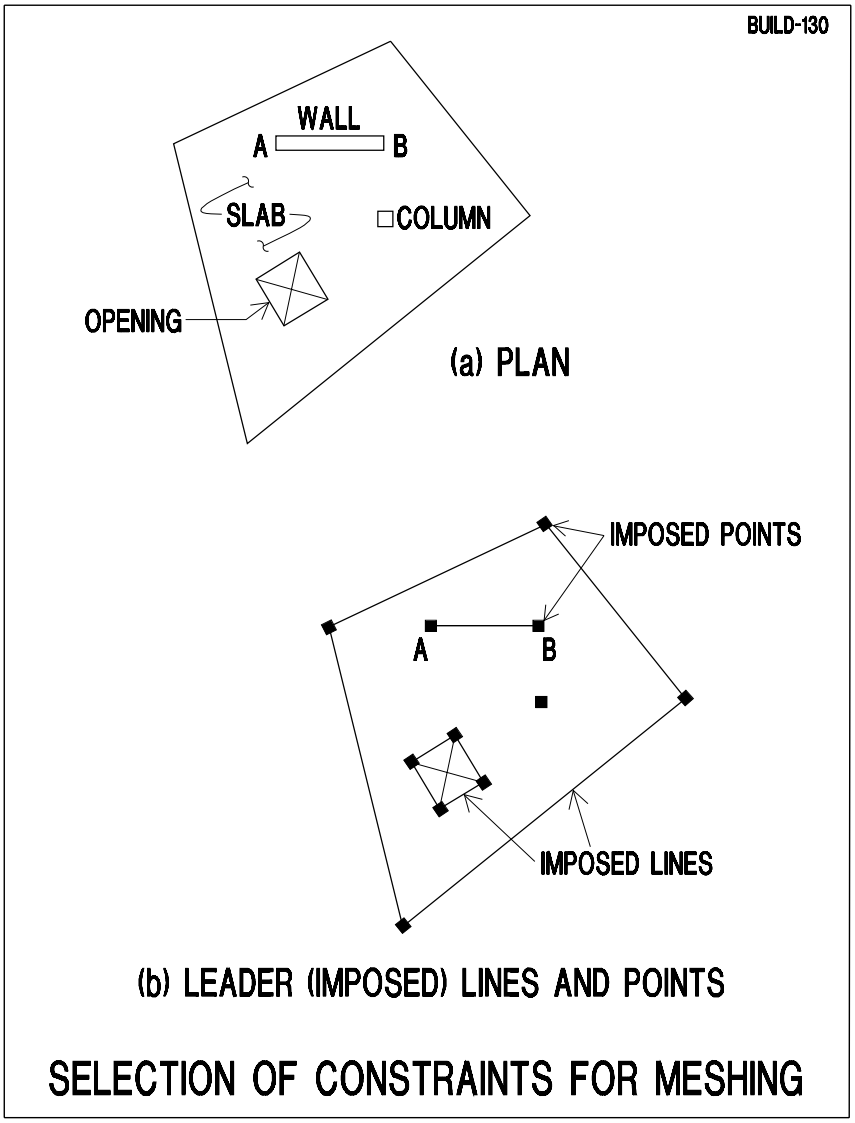
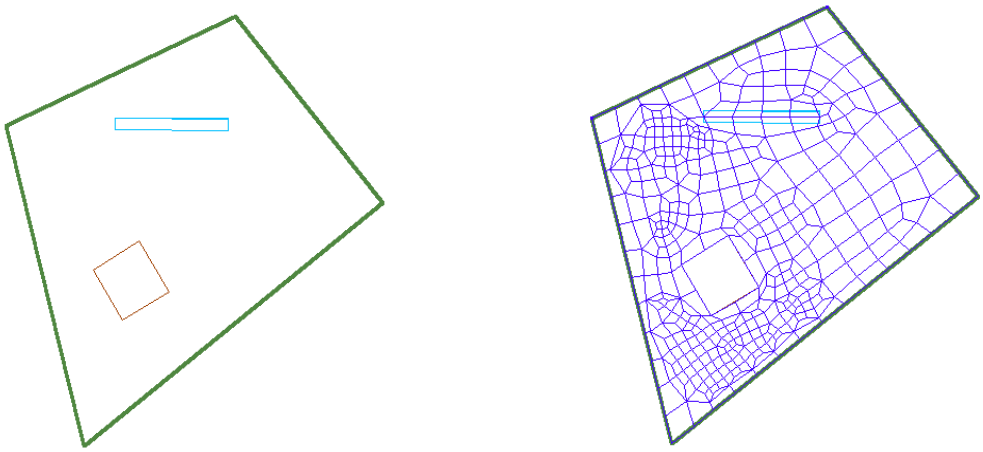


FIGURE 8.2-1

An example of imposed nodes, imposed lines and automatic meshing is illustrated in **Fig. 8.2-2**. Note that the meshing generated automatically by the program includes the imposed nodes placed along the imposed lines.



(a) (b)

FIGURE 8.2-2 IMPOSED NODES, LINES AND AUTOMATIC MESHING

The diagram of line and node constraints for the meshing (**Fig. 8.2-4**) is generated each time meshing is attempted. It is saved in a layer called "component representative."[3] You can view it on its own, by turning this layer on and all the others off.

8.2.2 Basics of Automatic Meshing

The basics of automatic meshing are outlined in the Modeler User Manual. This Chapter, and the Appendices A and B of this manual, cover additional details and the advanced features of meshing not covered in the Modeler User Manual. If it is your first attempt at meshing, it is highly recommended that you read the Modeler User Manual.

8.2.3 Manual Meshing

8.2.3.1 Overview and Use of Quadrilateral Cells

Only in rare cases will a structure need to be meshed manually. In the worst scenario, you are likely to use a combination of automatic and manual meshing. However, knowing how to perform manual meshing is of value, should you decide to edit

[3] The component representative will be displayed automatically, if you attempt manual meshing. It will also be saved in a dedicated layer called "component representative." The automatic meshing is also based on the component representative drawing. But, the drawing is not displayed on the screen. Neither is it saved in a layer, unless you do the following. From the FEM pull-down menu select Manual Meshing. Click on *Cancel* button, when the option opens. This will generate the component representative layer.

the mesh generated by the program, or should you in a special case have a reason to use manual meshing.

The node consolidation (node shift) feature of the software, described in the last chapter of this manual, is also available for manual meshing. If required, the node consolidation option has to be exercised before manual meshing is started. For your first reading, there is no need to use "node shift."

Once you click on the manual meshing tool, the program performs a "validation check" for the structural model to be meshed. The purpose of this validation check is to detect obvious errors, such as a column not having been attached to a slab. The validation also corrects, or improves a number of shortcomings in the structural model, such as merging points that are too close to one another, or multiple slab vertices.

Let's explore the process of manual meshing by way of an example. **Figure 8.2-3** shows the plan of a floor with an opening, three columns and a beam.

Click on the *Manual Meshing* tool on the *FEM* (Finite Element Method) pull down menu. The program performs a validation for the structural model in the background. If the program detects a problem, or shortcoming, it will display its finding on the computer screen either as an "error," or "warning." The errors must be corrected before manual meshing can proceed. The warnings, however, can be ignored, if these are acceptable to the user.

After the structural validation, the program converts the floor system to a simple drawing (component representation as described above) that shows only the outline to be covered by mesh cells. The component representation is shown in **Fig. 8.2-4.**

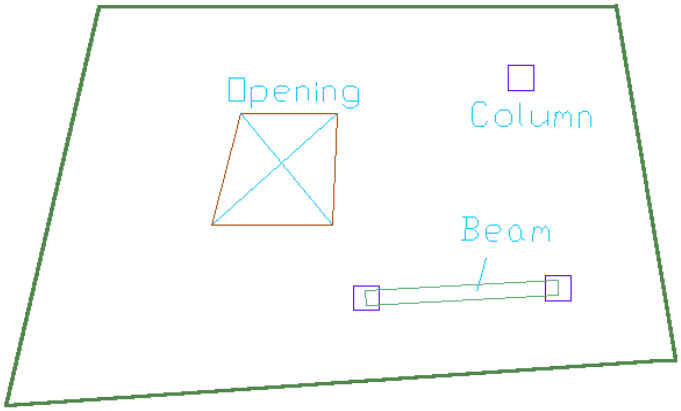


FIGURE 8.2-3 PLAN OF A SLAB REGION

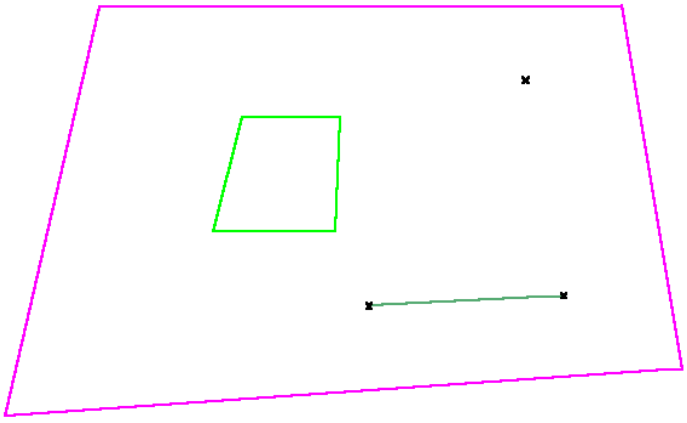
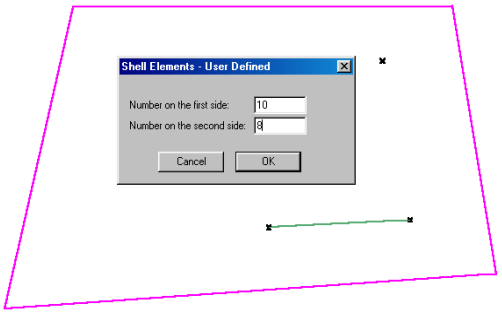


FIGURE 8.2-4 COMPONENT REPRESENTATION OF THE FLOOR

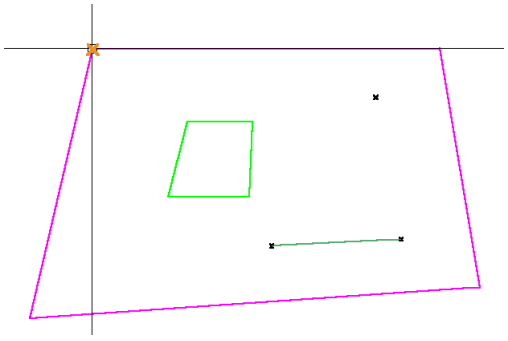
The columns are shown by points in the outline. Beams, walls, opening and slab boundaries are shown by lines, each with its own color. The manual meshing should conclude with a "node" on each of the points on the component representation plan, and cell boundaries on each of the lines. Further, the cells should cover the entire floor area, except the openings.

At the next step, the program prompts a dialog box for cell divisions in each direction of a mesh to be created. This dialog box is shown in **Fig. 8.2-5(a)** for the current example. A complex floor plan is generally covered by several mesh sets. For the current example, one mesh set will be adequate. As shown in **Fig. 8.2-5(a)**, a mesh with 10 by 8 is selected.

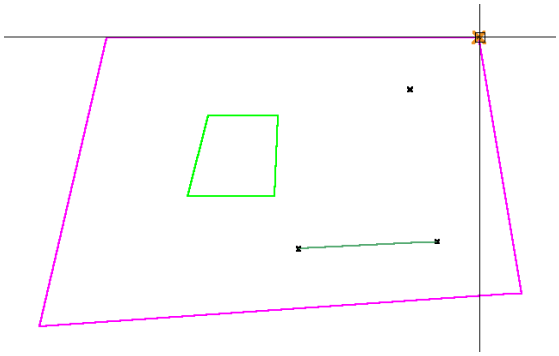
After pressing "OK" on the cell divisions dialog box, the program asks you to select the four corners of the region you want to apply the selection. Follow the prompts on the command line at the bottom of the computer screen. At this stage, use the snap tools of the program and match the corners/sides of the mesh to those of the points/lines on the component representation. In the current example the "snap to intersection" tool is used. As it is illustrated in **Figs. 8.2-5(b)** and **8.2-5(c),** the corners of the slab region are selected. Once the last corner at the bottom left is clicked, the mesh shown in **Fig. 8.2-5(d)** appears.



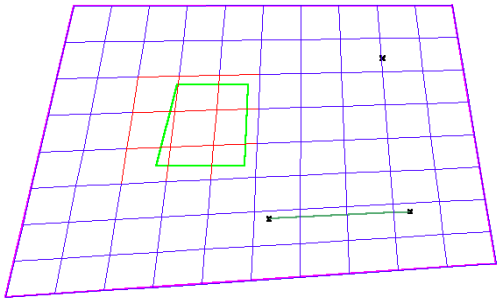
(a) Dialog Box for Number of Cells in the Mesh



(b) Selection of First Point at Top Left



(c) Selection of the Second Point at Top Right



(d) Generated Mesh Over the Slab Region

FIGURE 8.2-5 STEPS IN GENERATION OF A MESH REGION

Figures 8.2-6 through **8.2-12** illustrate the editing of the mesh to map it on the point and line restraints of the component representation in **Fig. 8.2-4.** The editing consists of the following steps.

- Select the common node of the cells next to column points, drag them and snap them to the column points (**Figs. 8.2-6** through **8.2-8**). In this operation, it is

important to select all the cells that share the common node to be shifted to the column.

- Select the common nodes of the cells that are close to opening corners and using "snap to intersection" snap them to the adjacent opening corner. Follow this by bringing the nodes next to opening sides and snapping them to the side. Use "snap to nearest" for this operation. Then select and delete the cells that are inside the opening. The mesh editing for the opening is illustrated in **Figs 8.2-9** through **8.2-11**.
- Using "snap to end," edit the common cell nodes next to the end of lines to snap to the line ends (**Fig. 8.2-12**). Then, using "snap to nearest," adjust the cells that straddle across the beam or wall line to have their boundary on the line.

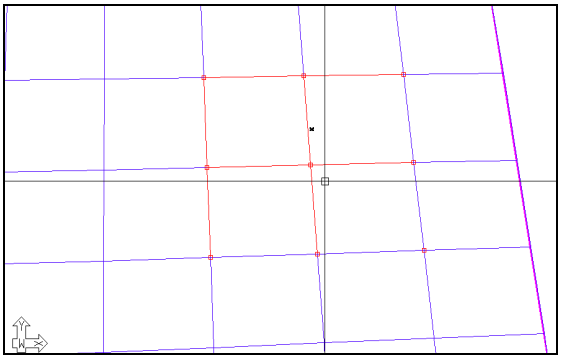


FIGURE 8.2-6 SELECTING FOUR CELLS ADJACENT TO A COLUMN POINT

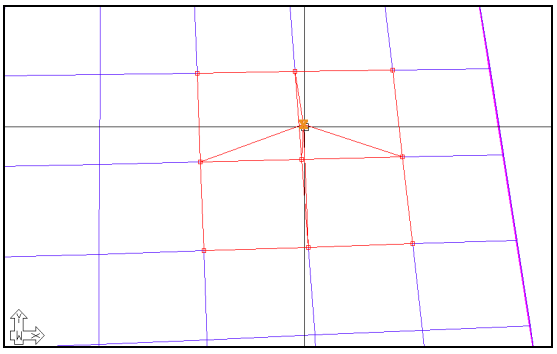


FIGURE 8.2-7 DRAGGING THE COMMON VERTEX OF FOUR CELLS AND SNAPPING IT ON THE COLUMN POINT

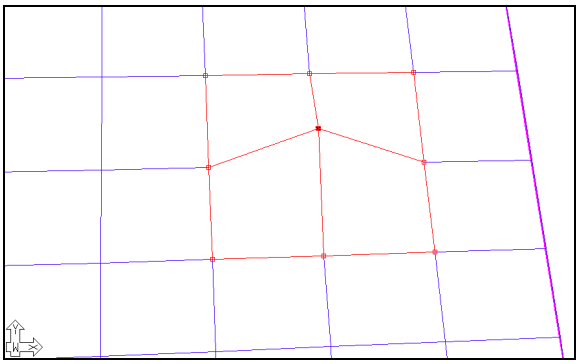


FIGURE 8.2-8 COLUMN POINT IS COVERED BY THE COMMON VERTEX OF ITS ADJACENT FOUR CELLS

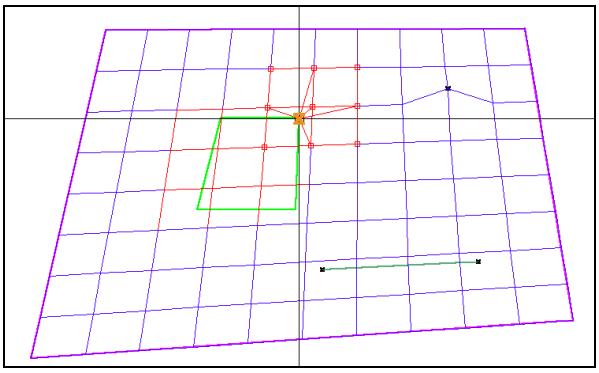


FIGURE 8.2-9 THE COMMON VERTEX NEXT TO A CORNER OF THE OPENING IS PICKED AND SNAPPED TO THE CORNER

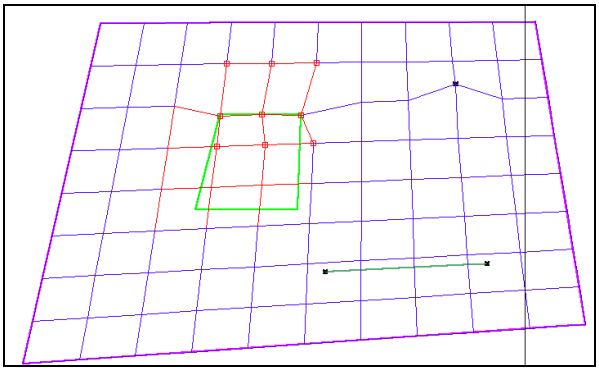
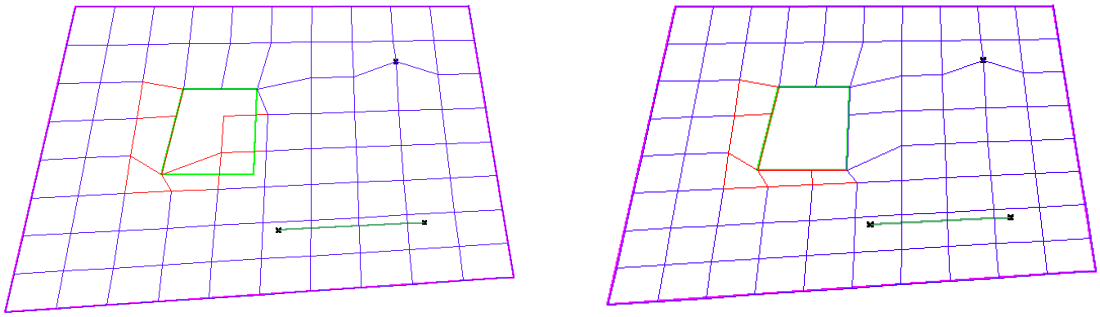


FIGURE 8.2-10 VERTICES ADJACENT TO THE OPENING ARE SELECTED, DRAGGED AND SNAPPED TO THE OPENING SIDE AND CORNER



(a) Opening Adjustment at Interim Stage (b) Completion of Mesh Editing at Opening

FIGURE 8.2-11 EDITING OF CELLS TO ACCOMMODATE THE OPENING

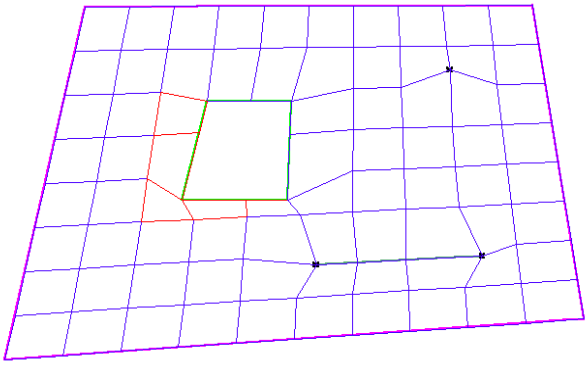


FIGURE 8.2-12 COMPLETED MESHING OF THE FLOOR

Once the meshing is complete, the program automatically generates the shell and frame elements for the beams, columns and the walls. The generation of these elements is achieved prior to the Finite Element Analysis of the floor system. Once the analysis is complete, you can view the meshing of the structure in three dimensions.

An alternative to the manual meshing is automatic meshing. The automatic meshing option for the current example is shown in **Fig. 8.2-13**. The cell size selected for the automatic meshing was the same used for the manual example.

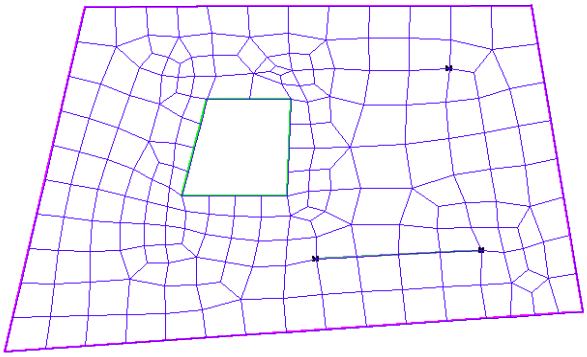


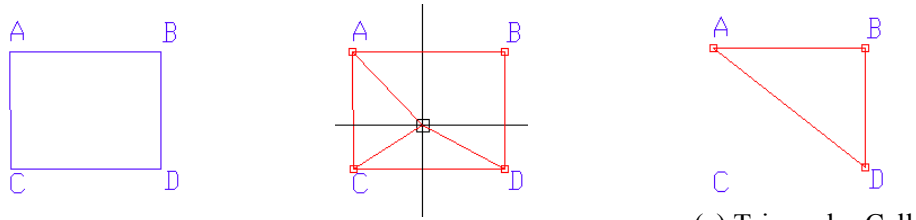
FIGURE 8.2-13 MESH FROM THE AUTOMATIC MESHING OPTION

8.2.3.2 Generation of Triangular Cells

Quadrilateral shell elements yield a more accurate solution. However, when using manual meshing, in some locations the geometry of a slab is more amenable to triangular elements. For such cases triangular cells are used.

A triangular cell is created by transforming a quadrilateral cell into a triangle. Click on one corner of the quadrilateral, drag it over and snap it onto another corner of the same quadrilateral. **Figure 8.2-14** illustrates the process. The quadrilateral cell ABCD shown in part (a) of the figure is transformed into the triangular cell ABD. The following are the steps.

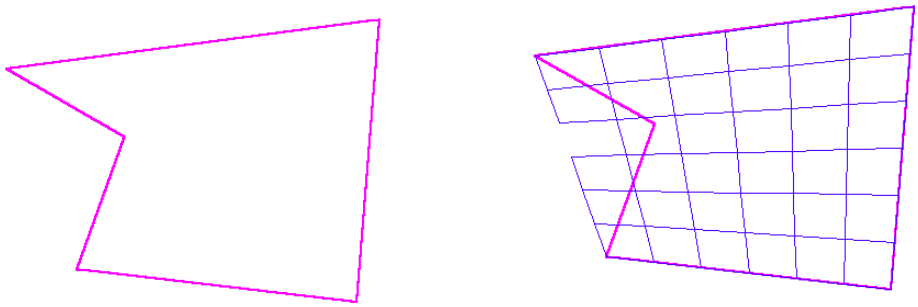
- Activate the snap to intersection tool and turn other snap tools off.
- Select the corner to be eliminated (point C in the figure).
- Snap the corner selected onto another vertex of the quadrilateral (point D).



(a) Quadrilateral (b) Editing (c) Triangular Cell

FIGURE 8.2-14 TRANSFORMATION OF A QUADRILATERAL TO A TRIANGULAR CELL

The following example illustrates the editing of quadrilateral cells and the application of triangular cells to a particular floor slab.



(a) Plan of Slab Region (b) 6x6 Cells Mapped Over the Slab Region

FIGURE 8.2-15 PLAN OF A SLAB REGION AND THE INITIAL MESH MAPPING

Figure 8.2-15(a) shows the plan of an irregular slab region. Selection of a 6x6 division when snapped at the four convex corners of this slab region give a mesh as shown in part (b) of the figure. Note that the cell falling entirely outside the boundary of the slab region is not generated by the program. This feature is designed to minimize the effort of manual meshing.

Figures 8.2-16 through **8.2-20** show graphically how nodes A and B of the quadrilateral cell next to corner C of the slab are edited to conform with the geometry of the slab boundary. The figures also illustrate the editing of the adjacent cells to complete the meshing along the slab boundary line CD.

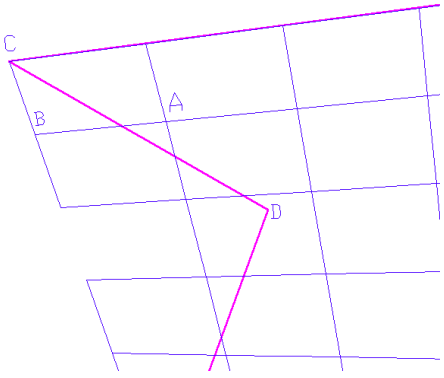


FIGURE 8.2-16 CLOSE UP OF THE TOP LEFT CORNER OF THE SLAB

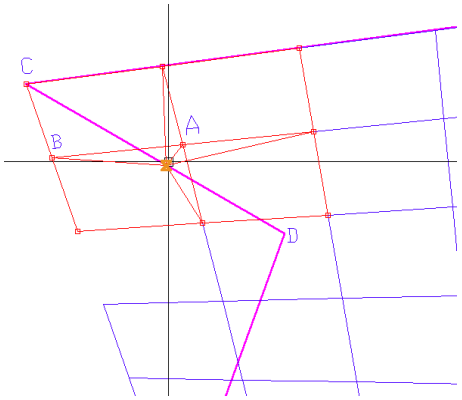


FIGURE 8.2-17 NODE A IS DRAGGED AND SNAPPED ON LINE CD

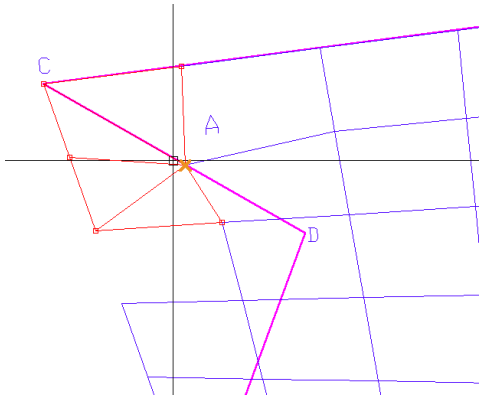


FIGURE 8.2-18 NODE B IS DRAGGED AND SNAPPED AT THE INTERSECTION OF A ON CD

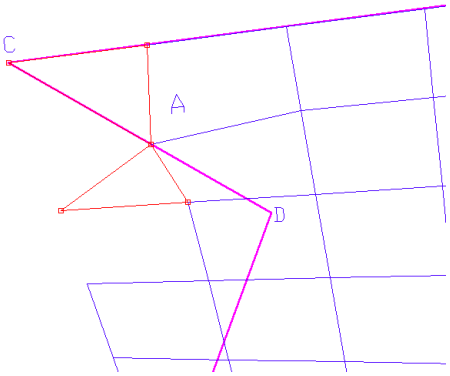


FIGURE 8.2-19 TOP LEFT QUADRILATERAL CELL IS TRANSFORMED TO A TRIANGULAR CELL MATCHING THE SLAB GEOMETRY

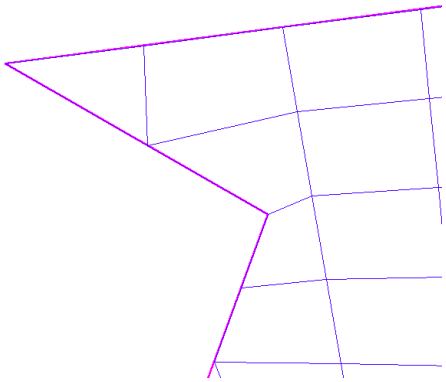


FIGURE 8.2-20 EXCESS CELLS ARE REMOVED

8.2.4 Mesh Editing

The program provides you with a set of effective tools to view and edit the layout of a mesh either generated automatically by the program, or otherwise. The process is detailed in the preceding section on manual meshing.

8.2.5 Nodes and Finite Elements

Prior to the analysis, the cells created in the mesh will be converted into finite elements and their vertices into nodes. You have full access to the node and element numbers, their connectivity and other properties. In the normal course of your work you would not need to access this information, since the program automatically performs all the operations necessary to obtain a complete finite element solution. The last Chapter of this manual describes how you can view both in graphical and tabular forms the internal information used by the program for the nodes and elements.

8.2.6 Natural and Analysis Nodes

In order to enable the program to model complex structures, it is necessary that each node of a finite element has a twin. Unlike simple finite element programs, the FEM processor of ADAPT-Builder is based on retaining the node definition and position that are commonly used. Again, this is all done in the background and will not be apparent either in data generation, nor the solution/results of the program.

Figure 8.2-21 illustrates an example.

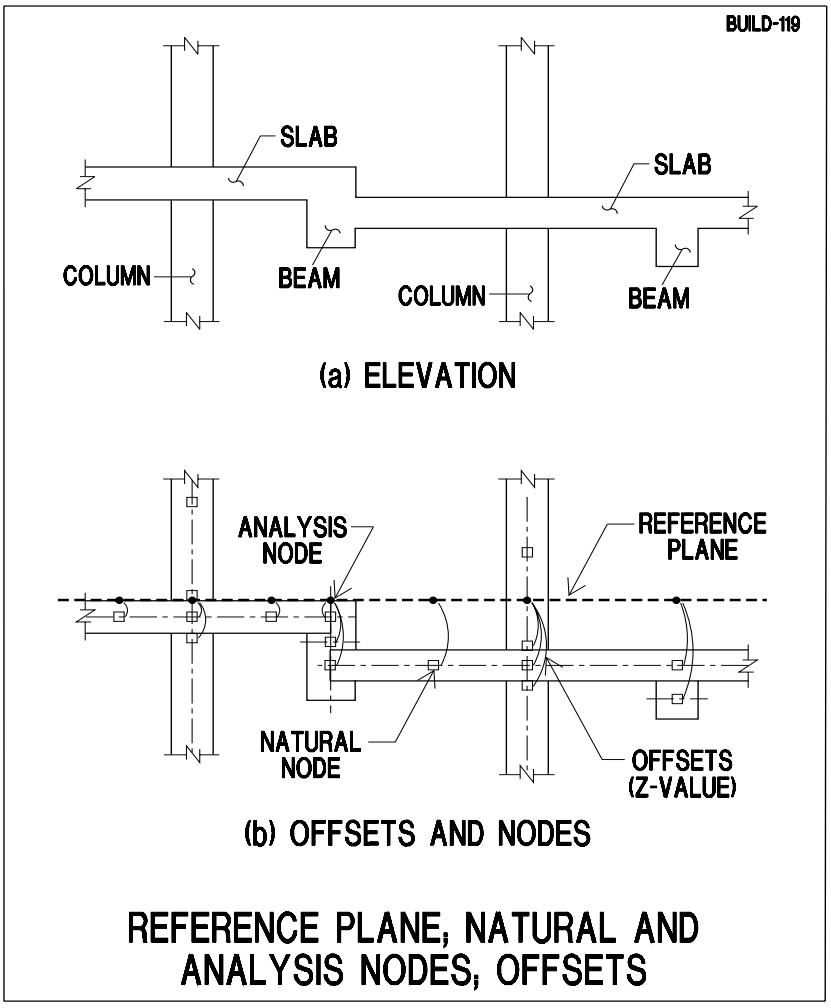


FIGURE 8.2-21

8.2.7 Node Consolidation – Node Shift

Another tool available to you is the node consolidation, referred to as Node Shift. Again, in the course of your normal work, you need not be conscious of it, nor use it. However, using it can help pave the way for a seamless integration of your design work with the architects work, and facilitate the structural drawings you generate. Node consolidation (Node Shift) also makes it possible to take advantage of the application of Component Technology in your modeling and designs.

The details of the implementation are given in the last chapter of this manual. The following describes the concept.

Consider the slab detail shown in **Fig. 8.2-22**. Part (a) of the figure shows the architect's drawing and the manner in which the structure will be built.

This is also the way you would show it on your structural drawings. However, in performing the analysis and design, traditional FEMs would model the structure as shown in part (b). That is to say, the column extends to the mid-depth of the slab, and the slab terminates short of the edge of opening. This may be fine for the analysis, but renders the structural model invalid for construction detailing. In summary, the model generated on the basis of traditional modeling does not lend itself for direct export as structural drawing. The alternative shown in part (c) of the figure overcomes this shortcoming. The slab is modeled to cover the entire area. Likewise, the column is terminated at the slab soffit. The geometry of the structural model (part (c)) of the figure matches that of the architect's. It can be directly used as a structural drawing for construction.

The handshake for force transfer between the column and the slab in part (c) of the figure takes place at the "analysis" node. The combination of the shift of the slab node to the slab edge and the selection of the analysis node is referred to as "node consolidation." The process takes place in the background, once you bring in the architect's drawing into the program. However, you have full control to edit and manipulate the process, if you so wish.

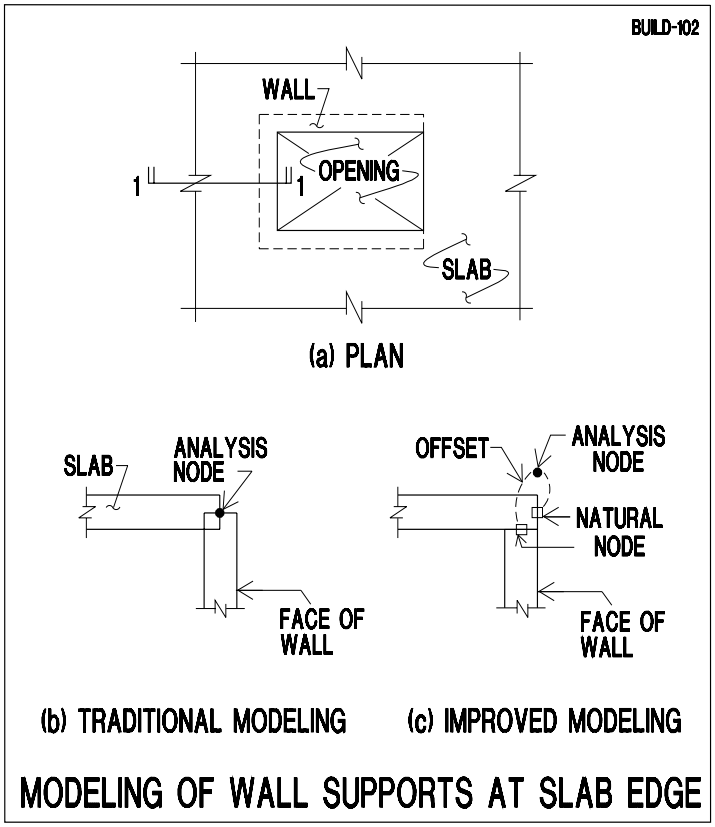


FIGURE 8.2-22

The seamless integration that can be achieved between the architect's presentations and the structural drawings you prepare is illustrated in **Figs. 8.2-23** and **24**. Again, **Fig. 8.2-23** shows the traditional method of creating a structural model from a simple portal frame. Note that neither the column, nor the beam lengths (h, l) used in the structural analysis match the construction length.

Again, the modeling technology used by ADAPT-Floor Pro (**Fig.8.2-24**) is Component Technology, which means that each component of your structure, such as a column or a beam, can be represented by their actual physical dimension. This process alleviates the discrepancy between the construction requirements and the structural engineering model. In addition, the analysis is based on the actual length. Your analysis model can be used to generate rebar layouts automatically (using the rebar module of the program). The rebar length, extent of ties and other particulars of the rebar layout would match the construction requirements.

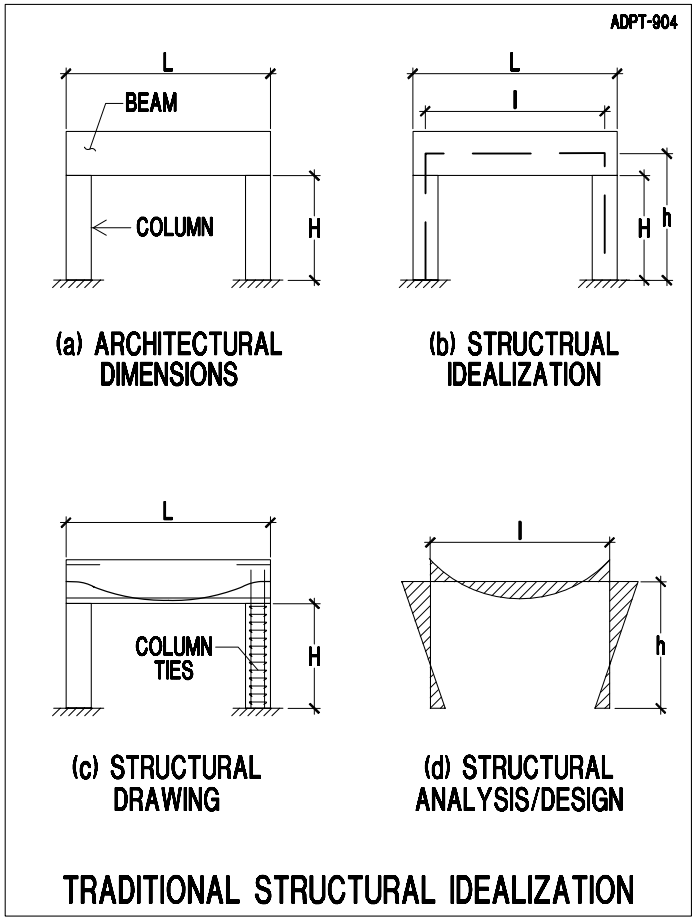


FIGURE 8.2-23

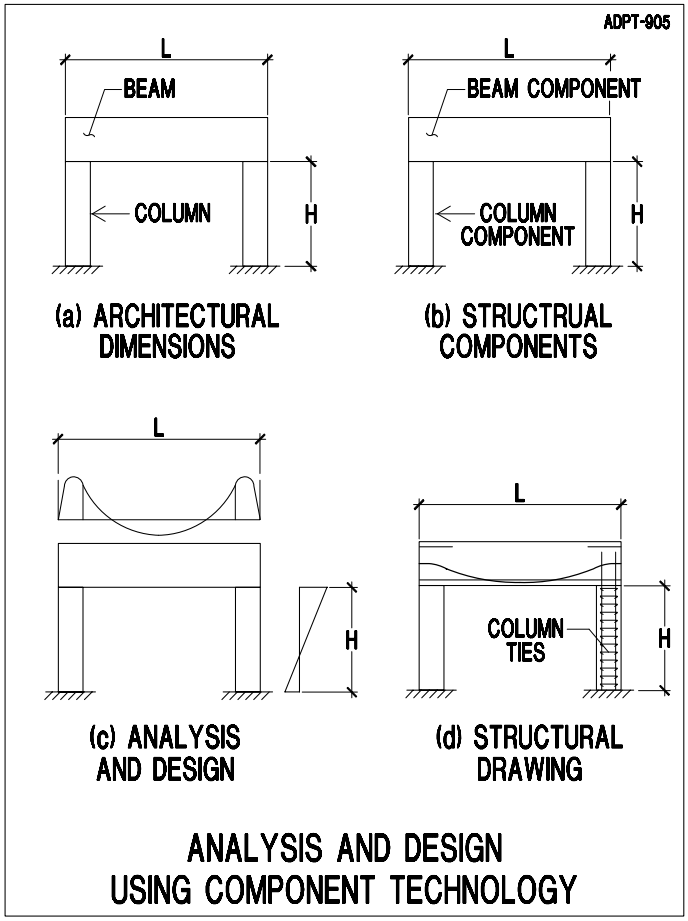


FIGURE 8.2-24

8.2.8 Excluder

Excluder is a feature, whereby you can bound one or more regions of a floor system and select to have the slab inside or outside the bounded region(s) meshed when the automatic meshing of the program is invoked. See **Fig. 8.2-25**. This feature applies to general analysis and vibration analysis. It is most useful when the features of a specific region require a manual mesh. Once the program concludes with the automatic meshing function, the areas of the slab that were excluded from the mesh can be completed using the manual meshing tools from the FEM menu. This process is described in the advanced features of the program in the last chapter of this manual. It is unlikely that you would use it for normal structures and in the course of routine work. This feature can be used for vibration analysis of a single region of a floor slab (see **Section 8.7.1**). Some users, however, may find this material beneficial for research and possibly with very complex structures.

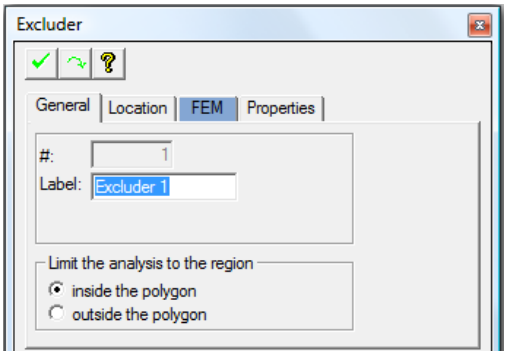


FIGURE 8.2-25 EXCLUDER OPTION

8.3 ANALYSIS OPTIONS

As it is detailed in the ADAPT-Modeler manual, the program automatically stabilizes your structural model against horizontal body movement, assuming that you are not doing so yourself. However, if you provide adequate boundary conditions, such as fixity at the supports in the X- and Y-directions, there is no need for the program to stabilize the model. If you have provided adequate supports in your structure to hold it tight in position, the "Stabilize the slab automatically" option in **Fig. 8.3-1** need not be checked.

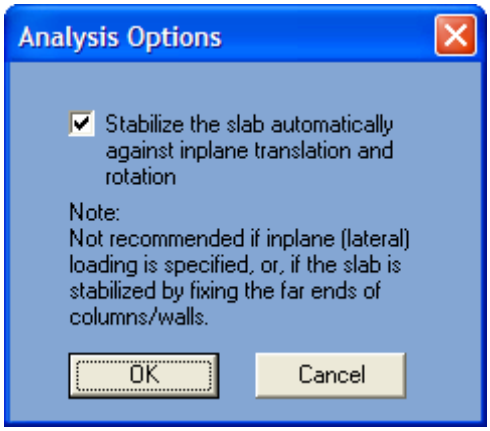


FIGURE 8.3-1 ANALYSIS OPTIONS DIALOG WINDOW

If you provide adequate supports, but do not uncheck the program's stabilization option in the above figure, the structure will end up with more supports than you intended.

8.4 ANALYZE THE STRUCTURE

Click on the *Analyze Structure* from the *FEM* pull down menu. The analysis does not require interaction from you. If no problems are encountered, the program prompts on the screen the successful completion of the analysis.

8.5 DEFLECTIONS CALCULATION IN FLOOR PRO WITH ALLOWANCE FOR SLAB CRACKING

8.5.1 Overview

ADAPT-Floor Pro has the option to calculate the deflection of a floor system with allowance for loss of bending stiffness due to cracking of either slab regions or beams. The formulation used in ADAPT-Floor Pro is rigorous, iterative, and accounts for loss of stiffness at the individual finite element cells over the entire floor system. The following is a brief outline of the algorithm used:

1. User identifies the load combination for which the deflection with cracking is to be calculated.
2. Using the geometry, material properties, information on current second moment of area of each finite element cell and support condition of the floor system, the program sets up the system stiffness matrix for the entire structure. From this stiffness matrix and the load identified above, the program calculates the distribution of moments (Ma) and axial forces over the entire structure.
3. The program scans the available reinforcement in the floor system. The reinforcement is either determined by the program prior to the initiation of deflection calculation, or is a combination of program-calculated and user-defined or user-edited reinforcement. Once the deflection calculation is initiated, the available reinforcement remains unchanged. There is no restriction on the orientation and length of reinforcement bars, nor the position of reinforcement within the slab The available reinforcement can be in one or more of the following forms:
    - User-defined one or more top and bottom mesh reinforcement;
    - User-defined grouped or distributed reinforcement bars at top and/or bottom of slab and beams;

- Reinforcement calculated and reported by the program for code compliance, such as minimum requirements of the code, strength check, initial conditions, or other criteria; and
- Post-tensioning tendons defined by the user, each with its own location and force.

4. The calculated moment (Ma) and axial force of each finite element cell is matched with the reinforcement in that cell. Using the parameters given below, the program calculates the effective moment of inertia of each cell:
    - Finite element cell thickness (to obtain uncracked second moment of area);
    - Available reinforcement associated with the cell (nonprestressed and prestressed);
    - Cracking moment of inertia associated with the cell in each direction (Icr);
    - Cracking moment associated with the cell (Mcr); and
    - Applied moment and axial force.
5. Using the modified moment of inertia due to cracking, the program re-calculates the stiffness matrix of each of the finite element cells.
6. The system matrix of the entire structure is then re-assembled and solved to obtain a new set of deflection values.

At this stage the solution given is a conservative estimate for the floor deflection with cracking. However, at user's option, the program can compare the calculated deflection with that of a previous cycle.

In its computations, the program accounts for loss of bending stiffness in beams as well as the slab. In the first release of the program, the formulation used for the calculation of stiffness loss from cracking is the one recommended in ACI-318-2005. However, the algorithm is general and can be readily expanded to relationships suggested in other codes.

8.5.2 Procedure How to Calculate Cracked Deflection

The following are the steps to calculate cracked deflection:

1. First, define the load combination for deflection with cracking.
    - Open Loading-> Load Combination/FEM. Define the load combination. In the "Analysis/Design Options"

combo box select option “CRACKED DEFLECTION” (**Fig 8.5-1**)

2. Analyze and design the structure.
3. View the reinforcement reported by the program. Modify as needed.
4. Go to FEM-> Calculate Cracked Deflection.
5. To view the deflection, select FEM->View Analysis Results. In the “Results” tab select “z-translation.” Go to the Load case/Combination tab. This tab lists the load combinations you defined together with the load combination for the cracked deflection. For this load combination the program creates two sets of results. The one with the prefix “Cracked” shows the cracked deflection. To view, select Display Results tool,  from the Results toolbar. The program will display the cracked_section deflection contour (**Fig. 8.5-2**)

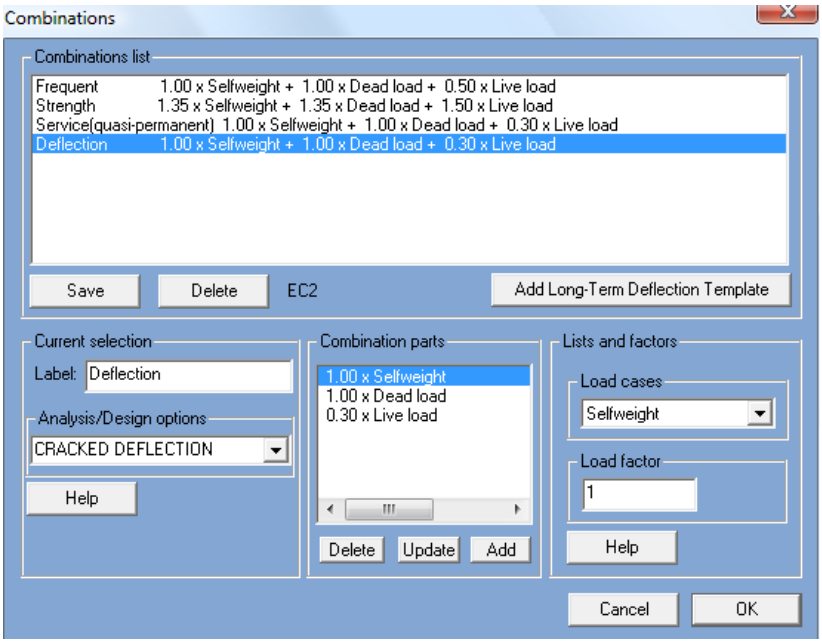


FIGURE 8.5-1 COMBINATIONS WINDOW

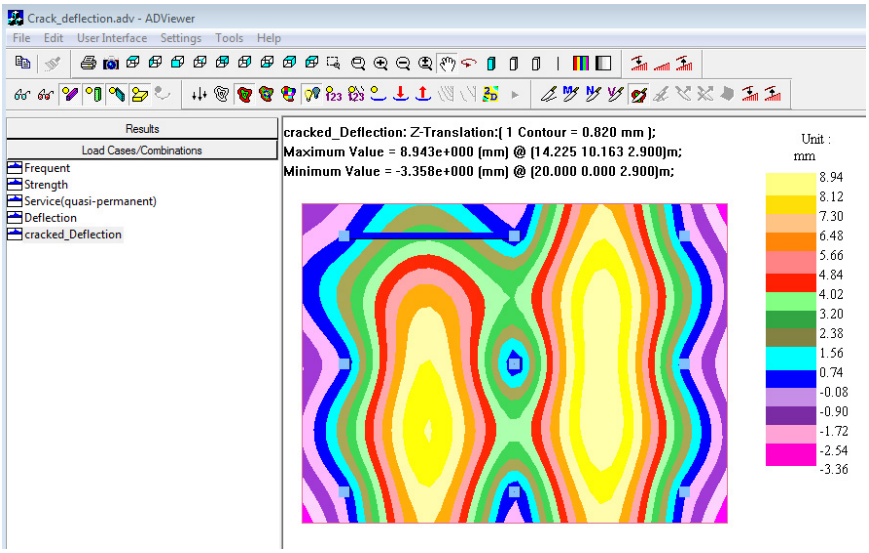


FIGURE 8.5-2 CRACKED_ SECTION DEFLECTION

To display the loss of rotational stiffness about X or Y direction of the slab, click on Results ->Slab Actions only -> Reduced Rotational Stiffness About XX or Reduced Rotational Stiffness About YY. The loss of stiffness is represented as a ratio of effective to gross second moment of area (**Fig. 8.5-3**).

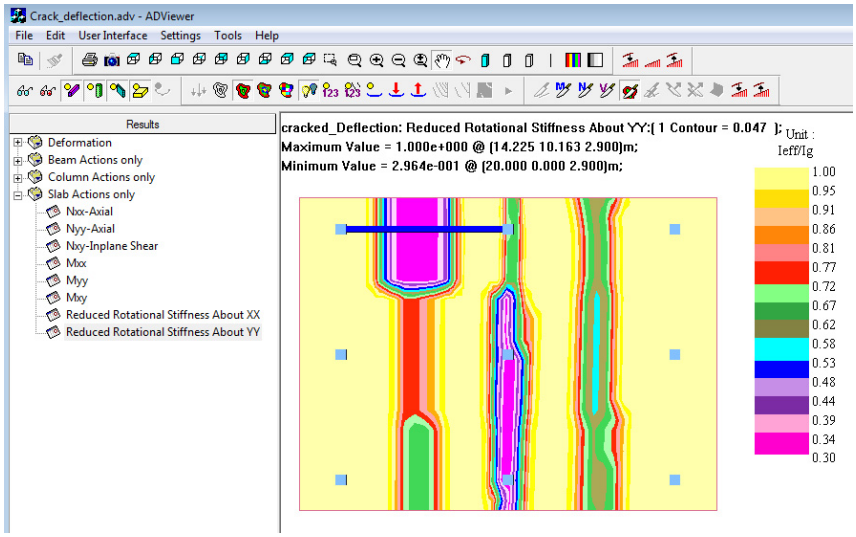


FIGURE 8.5-3 REDUCED ROTATIONAL STIFFNESS ABOUT YY

To display the stiffness ratio for beams, go to Beams action only ->Reduced Stiffness Ratio (**Fig.8.5-4**). To show values click on  tool.

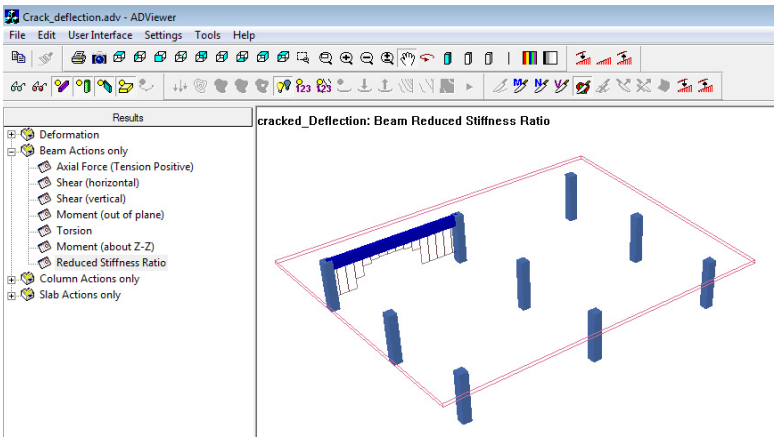


FIGURE 8.5-4 REDUCED STIFFNESS FOR BEAM

8.6 CRACK WIDTH CALCULATION

The program allows you to specify a limit for crack width (**Fig. 8.6-1**). If the calculated width exceeds the user-defined limit, the program will add reinforcement to limit the crack width as per design code provisions. The calculated rebar is included in the "service" envelope of reinforcement. The calculation of crack width will be available as an option when the selected design code doesn't require limitations for crack width (**Fig. 8.6-2**). You can select to only report crack width, or to report and limit crack width in which case the program adds reinforcement if required. For this scenario the calculations are based on European code provisions.

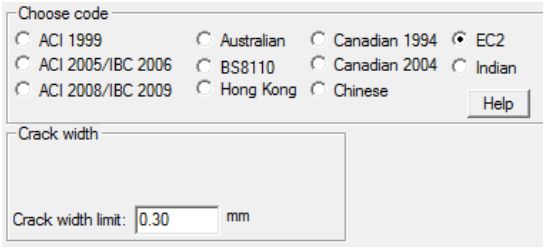


FIGURE 8.6-1 CRACK WIDTH LIMIT INPUT



FIGURE 8.6-2 CRACK WIDTH OPTIONS

To obtain results for crack width:

1. Go to Criteria->Design Code tab (if opened in RC only mode) or Criteria->Allowable Stress tab (if opened in PT and RC mode).
2. Depending on the selected design code you will have an option to enter the allowable crack width or to report crack width only.
3. Analyze and design.
4. To view crack width contours, go to FEM->View Analysis Results.
5. In the Load Cases/Combinations tab select "service" load combination.
6. In the Results tab->Slab Action only select Cracks along Strip Y – (width) or Cracks Along Strip X-(width). Note that these options will not be available if the crack width calculation does not apply to selected load combinations (for example "strength" load combination).
7. Select  and  tools. The color contour will display on the screen (**Fig. 8.6-3**)

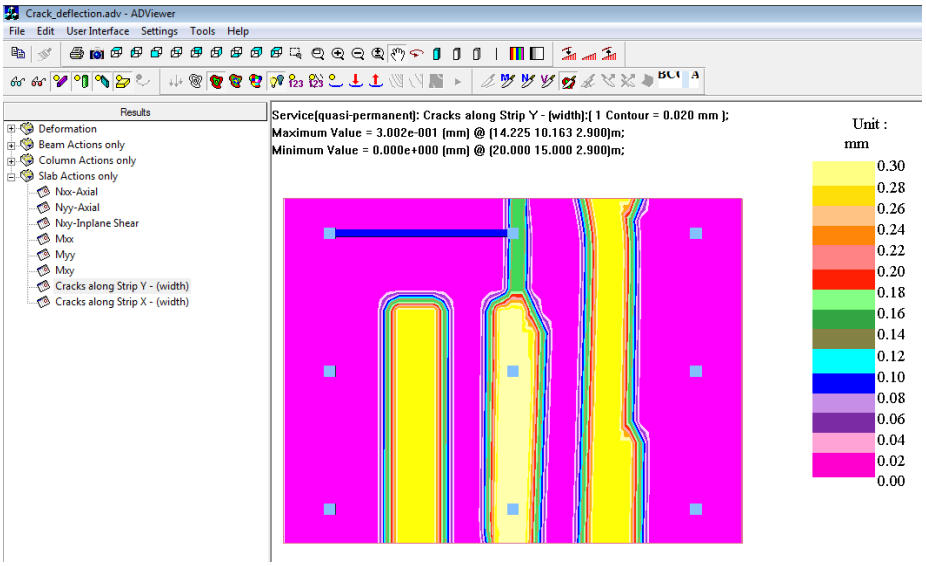


FIGURE 8.6-3 CRACKS ALONG STRIP Y

8.7 DISPLAY LINE CONTOURS

8.7.1 Overview

ADAPT-Floor Pro provides the option to display line contours for moments and shear forces in the modeling environment. After a successful analysis, go to FEM -> Line Contour -> Generate Line Contour to display the "Generate Line Contour" window (**Fig. 8.7-1**).

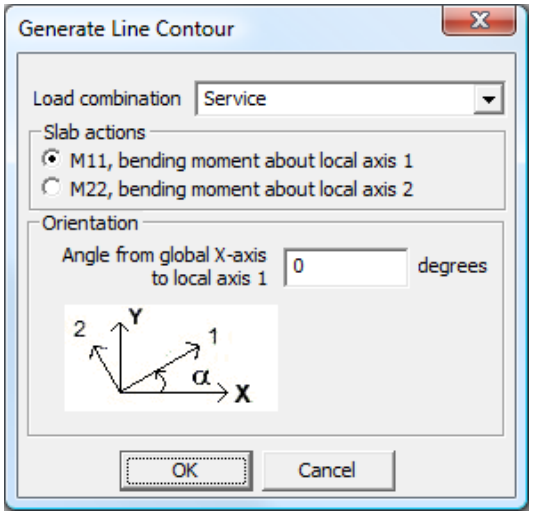


FIGURE 8.7-1 GENERATE LINE CONTOUR

From the Load combination drop down box select the load combination for which you want to display the line contours.

You have the option to select the following slab actions:

- Bending moment about the local axis 1 (M11)
- Bending moment about the local axis 2 (M22).

In the "Orientation" group box, you can specify the angle for the global X-axis to the local axis 1. This allows you to display line contours for slab actions in any direction.

Click "OK" to display the specified line contours for your model in ADAPT-Floor Pro's modeling environment.

8.7.2 Contour Toolbar

The "Contour Toolbar" allows you to display and manipulate the line contours (**Fig. 8.7-2**). These tools are also available through the FEM pull-down menu as a sub-menu of Line Contour.

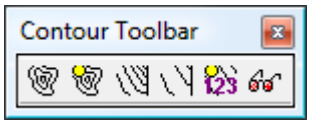


FIGURE 8.7-2 CONTOUR TOOLBAR

Generate Contour. Click this button to generate the line contours after analyzing the structure.

Display/Hide Contour. Use this button to either display or to hide the line contours after generating them.

Increase Number of Displayed Contour Lines. This tool allows you to increase the displayed number of contour lines.

Reduce the Number of Displayed Contour Lines. Click this button to decrease the number of contour lines for your model.

Display/Hide Text Shown on Contour Lines. Use this button to either display or to hide the labels for the contour lines that are labeled.

Contour Display Setting. Click this button to open the "Result Display Settings" (**Fig. 8.7-3**) to alter the contour line display.

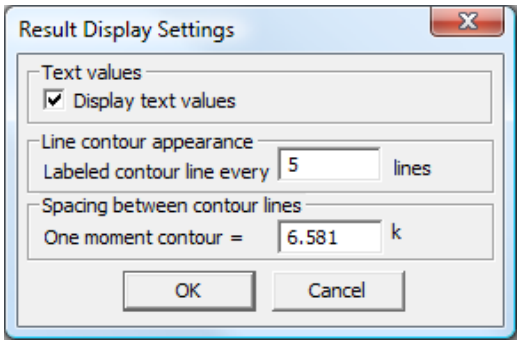


FIGURE 8.7-3 RESULT DISPLAY SETTINGS

To display the text values for labeled contour lines, check the check box in the "Text values" group box.

You can specify at what interval contour lines are labeled by defining the number in the "Line contour appearance" group box.

The spacing between contour lines can be specified by defining the distance between the contour lines.

8.8 VIBRATION ANALYSIS

8.8.1 General

ADAPT-Floor Pro includes the option to determine the vibration response of a floor system with respect to consideration of full bending stiffness and reduced bending stiffness due to cracking of either slab regions or beams. The formulation used in ADAPT-Floor Pro accounts for the loss of stiffness at the individual finite element cells over the entire floor system.

Inherent to the nature and type of slab systems designed using ADAPT-Floor Pro, the most common design carried out with the vibration module might be that associated with foot drop of walking occupants in residential and commercial buildings, however, the use of the tool is not limited to this. It is important for the user to determine the proper criteria for vibration excitation and susceptibility related to the source of vibration.

The results allow the user to obtain a simple and expeditious first estimate for a floor's vibration response where the objective is to determine whether a floor meets the serviceability requirements for vibration. The vibration response of a concrete floor is attributed to:

- Consequence of foot drop in residential and commercial floors;
- Rhythmic vibration, such as in dance and sporting events
- Vibration in laboratories or manufacturing facilities
- Vibrations due to vehicular traffic outside a facility
- Vibrations due to the operation of machinery
- Transient impulse due to earthquake, wind or other impact loads

ADAPT Floor-Pro has pioneered a method, whereby from the full 3D model of a building generated for overall design of a structure, a sub-region can be isolated for vibration analysis. Through use of the Excluder tool the program allows the user to draw a boundary around specific region of a floor slab and determine the vibration response of this area of slab. **Fig. 8.8-1** shows a portion of a slab identified for analysis as an example. Note that the option, "Limit the analysis to the region" is selected as "inside the polygon." This will give results for vibration and general FEM results only for the region selected. The support conditions selected for the boundary will have significant impact on the frequency of sub-region. To obtain the more reliable solution, the boundary of selected region should be extender farther away from the panel of interest, in order to minimize the impact of the user selected boundary restraints.

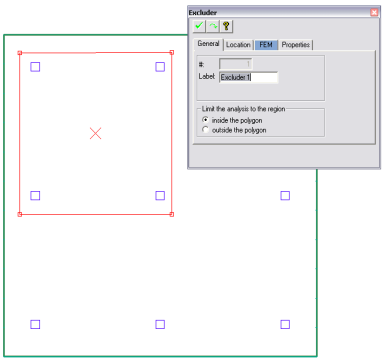


FIGURE 8.8-1 EXCLUDER OPTION

8.8.2 Run Vibration Analysis

Prior to executing the analysis of a model, to initiate the vibration analysis the user should select the option "Include vibration analysis" from the CRITERIA->Analysis/Design Options dialog window as shown in **Fig. 8.8-2.**

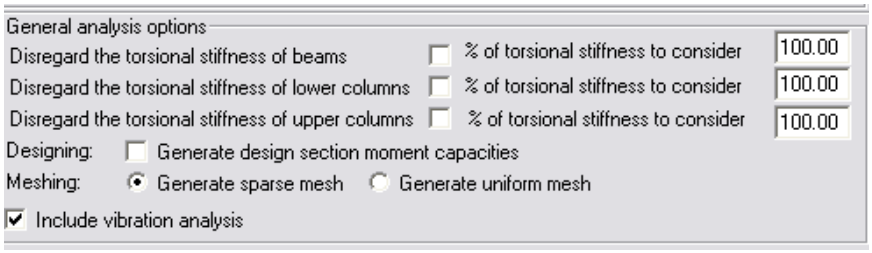


FIGURE 8.8-2 VIBRATION ANALYSIS OPTION

From the FEM->Vibration menu, the window shown in **Fig. 8.8-3** will appear. In this window, the user can input multiple load combinations for which vibration results will be generated. Load factors and cases can be combined in the window at the bottom, along with the Mass Participation Factors (Rx, Ry, Rz). The Mass Participation Factors are used to release the motion of the added mass from that of the structure in one or more of three global directions (1=fixed, 0=released). You can also set the number of modes for which results will be reported. After new combinations have been added and the number of reported modes is determined, select OK to proceed.

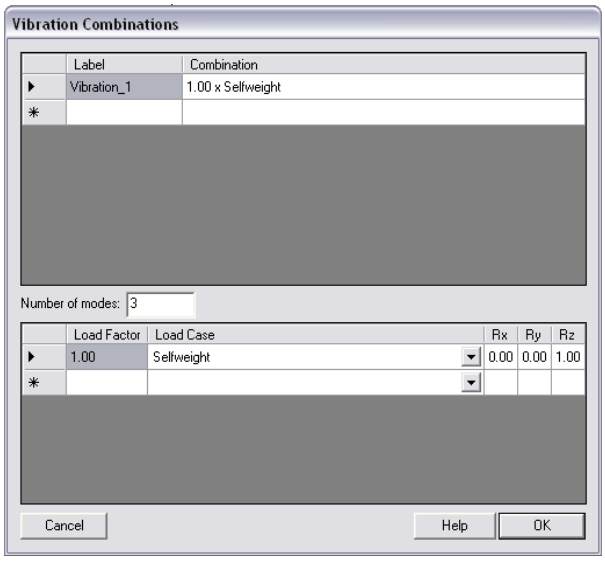


FIGURE 8.8-3 VIBRATION COMBINATION INPUT

After an analysis has been completed for a given slab, the user can view vibration results from FEM->View Results. If this is selected, the *ADViewer* screen will appear as shown in **Fig. 8.8-4**. Select the "Vibration Results" tab in the left-hand panel and the vibration analysis load combinations will appear for un-cracked and cracked results (if cracked analysis was initiated).

If either of the vibration results is expanded, by pressing the "plus" symbol, the mode numbers are listed and can be selected to view contour results in the screen.

The program reports the frequency (Hz), period (sec) and shows a normalized amplitude scale at the right edge of the screen. The maximum and minimum values in reference to the amplitude scale are also reported.

In order to obtain vibration results based on reduced stiffness due to cracking the user will have to run Cracked Deflection analysis (see **Section 8.5.2**). It is important to note that in the case when there is more than one load combination set to "cracked deflection" the vibration analysis will be based on stiffness that is calculated for the "cracked defection" load combination that is defined the last.

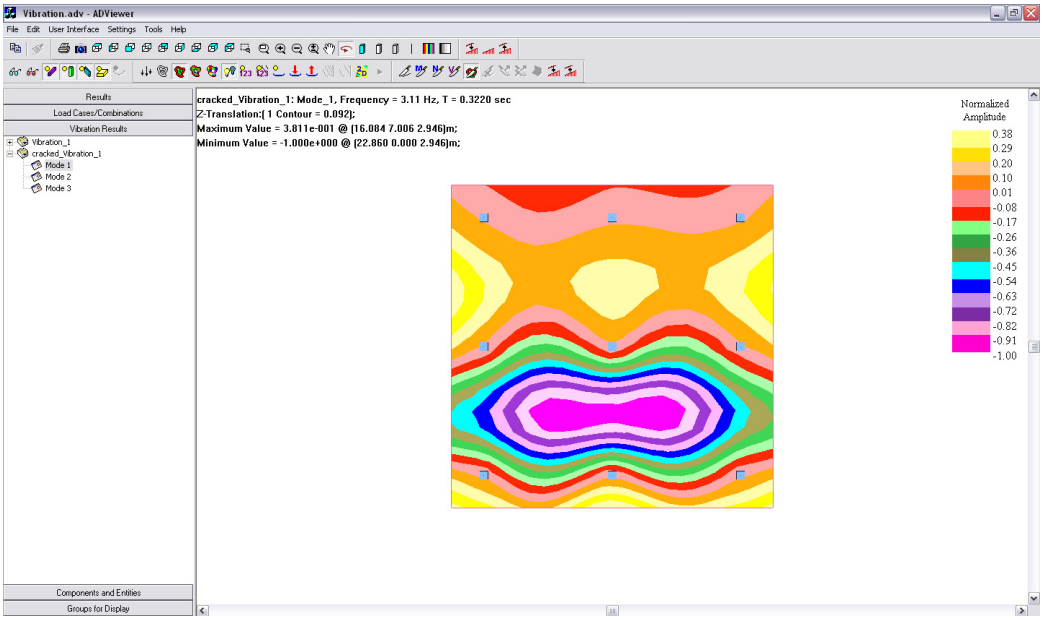


FIGURE 8.8-4 RESULTS OF VIBARTION ANALYSIS

Chapter 9

VIEW/VALIDATE RESULTS

Once the analysis is completed and saved, you can view and examine the results graphically. You can also obtain tabular reports of the analysis outcome. The focus of this chapter is the graphical view and queries from the graphical display to validate the solution. The tabular reports of the program are described in the chapter on "Reports."

Do the following to open the three dimensional viewer of the program and view the deflected shape of the structure.



View Analysis Results

Click on the above tool from the *FEM* pull-down menu to open the three dimensional viewer of the program. Once the 3D viewer opens, do the following:

- On the left margin of the screen, click on the *Load Cases/Combination*s button.
- From the list that opens, select *Service* or another load combination, if you did not select the service condition[1].
- Click on *Results* button on the top of the screen.
- From the list that opens, select Z-Translation
- Finally, click on the *Display Results* tool 

This will display the deflected shape contour of your structure. If the deflected contour does not display, click on this button: 

- Use the *Warp Display*  and *Rotate*  tools to examine the deformation of the structure. In particular, make sure that the slab does not displace excessively where it is supported on walls and columns.
- Use the other tools on the screen for a closer examination of the solution.

[1] Deflected shape will not be displayed for the load combinations assigned to "strength" code check, nor for "hyperstatic" load case.



Generate a Bitmap

Click on this tool to capture the 3D viewer image in bitmap format. You can ask the program to save this image for use in project reports or other purposes. Once clicked, the dialog window shown in **Fig. 9-1** opens. Note that apart from choosing your own title and location for saving the file, you can specify a title and subtitle for the image and its position of the report page. **Fig. 9-2** Illustrates the preview of such a screen capture. The preview shown is accessed from the File pull-down menu.

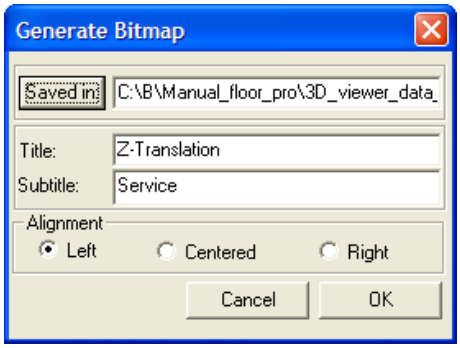


FIGURE 9-1 DIALOG BOX FOR SCREEN CAPTURE

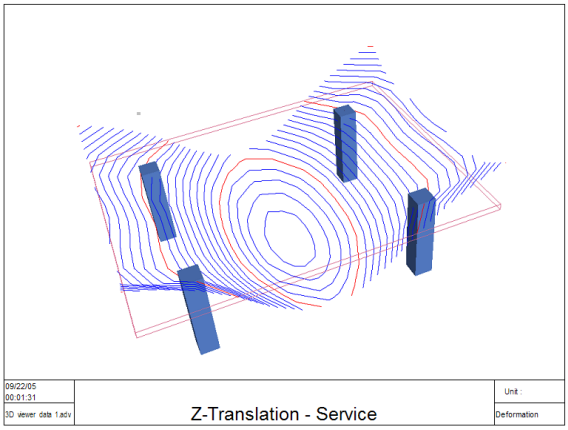


FIGURE 9-2 REPORT SHEET GENERATED BY 3D VIEWER SCREEN CAPTURE

Note that if you wish to copy the display into the Windows clipboard, simply use the standard copy tool that is on the main screen of the 3D viewer.

Model Display Tools



 **Display the model in solid form**

 **Display the model in transparent (see through) mode**

 **Display the model in wire frame**

 Display the frames (beams and columns) as lines (represented by their center lines)

 **Display the model in color**

 **Display the model in gray scale (black and white)**

Tools to manipulate the solutions displayed by curves are:



 **Scale down values** – reduce the ordinates of the results shown by curve(s)

 **Display values with default scale** – show the ordinates of the solution curve(s), according to the program's own choice

 **Scale up values** – enlarge the ordinates of the results shown by curve(s)

Display Components/Entities Toolbar





Entity Display Settings

This tool opens up the dialog window of **Fig. 9-3** that controls the attributes, such as color, line thickness, and font of each of the entities displayed on the screen.

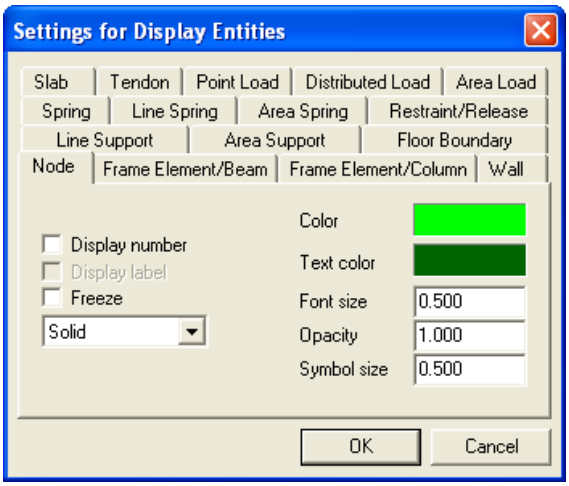


FIGURE 9-3 DIALOG WINDOW TO CONTROL THE ATTRIBUTES OF THE DISPLAYED ENTITIES



Result Display Settings

This tool opens up a dialog window as shown in **Fig. 9-4**. The dialog window controls the attributes of the displayed solution, such as line color and font.

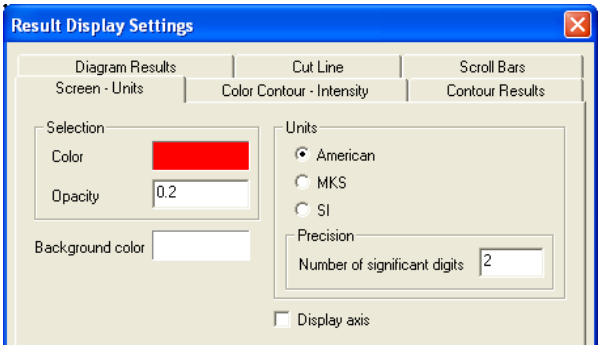


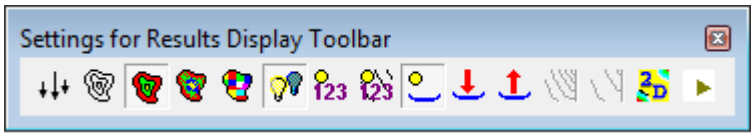
FIGURE 9-4 DIALOG WINDOW TO CONTROL THE ATTRIBUTES OF THE DISPLAYED ENTITIES

Tools to turn the display of the structural components on or off



The function of the above tools is the same as those in the Modeler screen. Each turns on or off the display of the structural component symbolized by its graphics.

Settings for Results Display Toolbar



This toolbar controls the display of the results shown in contour form. The following describes the function of each of its tools**.**



Vector plot (Fig. 9.5a)



Line contour (Fig. 9.5b)



Color contour (Fig. 9.5c)



Intensity contour



Max value contour (Fig. 9.5d)



Display results. This tool activates the display of the items you select. When you first enter the 3D viewer, the results of the calculations will be shown on the screen, but only after you select the values to be displayed followed by clicking on this tool**.**



Display numerical results for diagrams



Display numerical results for contours



Warp display (Fig. 9.5e). This tool displays the contour of your selection in three dimensions.



Increase warping



Decrease warping



Increase line contour density (Fig. 9.5f)



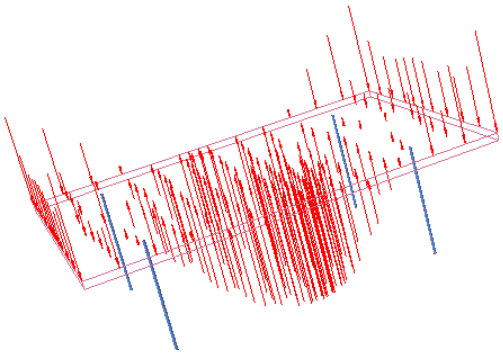
Decrease line contour density



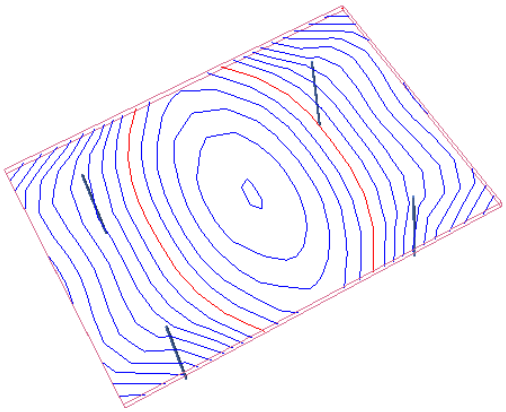
Two-dimensional viewer. To activate the two-dimensional contour viewer of the program, click on this tool first, then select the result you wish to be displayed.



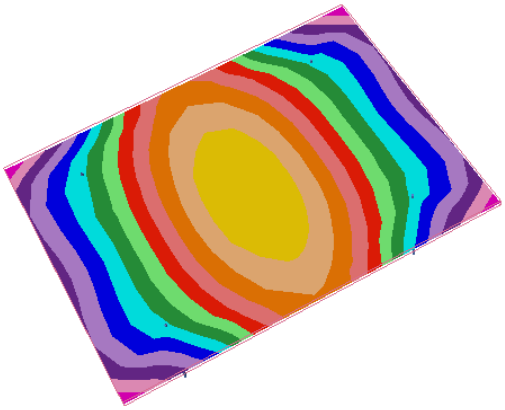
Start Animation



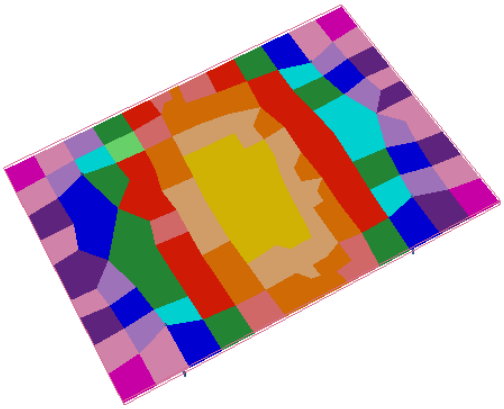
(a) Vector display of intensities



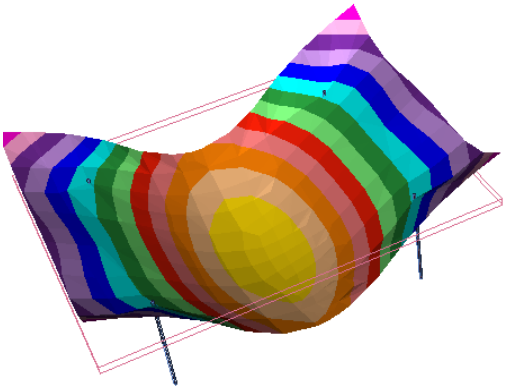
(b) Line contour display



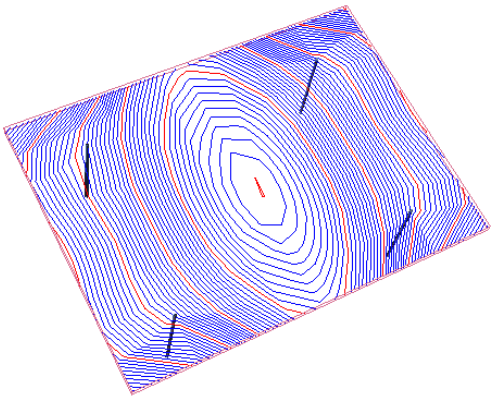
(c) Color contour display



(d) Max value display



(e) Warped display



(f) Increased contour density

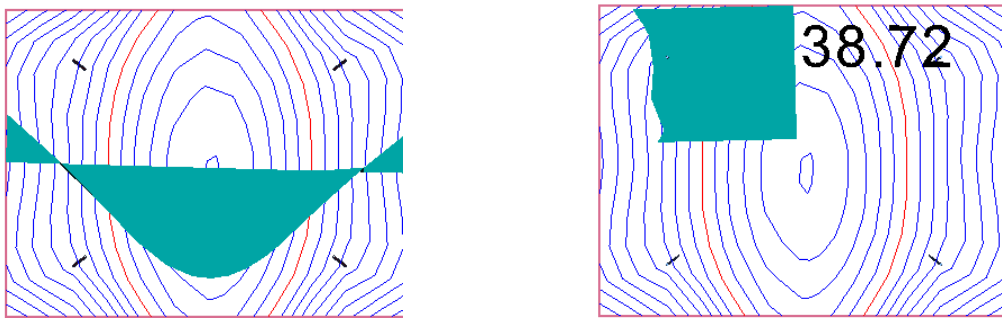
FIGURE 9-5 DISPLAY OPTIONS

Cuts Toolbar



This toolbar gives you the ability to view the distribution of the solution values along or across a line you draw (a cut) on the structural model. If applicable, it also gives you the total (integral) of the distribution you have selected over the length of the cut line

For example, a line drawn across the slab shown in **Fig. 9-6(a)** shows the distribution of slab deflection along this line. In a similar manner, the distribution of moment about a cut line is shown in **Fig. 9-6(b)**. In addition to the distribution, this figure shows the total value of moment across the cut line drawn. The unit of the distributions displayed will be shown on the hint line at the bottom of your computer screen. If the size of the text is not appropriate, you can change it to a size of your liking using the tool.



(a) Distribution of deflection

(b) Distribution of moment and its total value

FIGURE 9-6 DISTRIBUTION OF VALUES AT CUT LINES

Note that the distributions for the cut lines relate to the slabs, including drop cap/panels only. They do not include the values of beams, walls and columns. Later it is explained how to view the column values graphically.

To initiate a cut line, first click on the tool (Cut Line) located on the left of the toolbar. The program will be ready for a cut that would display any of the four options identified by the four tools on the left of the toolbar (moment, axial, shear and current display). The cut will show the option that is lit. If this is not your choice, click on the tool or the option of your choice, and follow the instructions at the bottom of the screen to proceed with the cut.

The following provides additional information**.**



Cut Line. This tool creates a vertical section through the floor system along a line that you draw. The program will then show the values of the solution entity you select along or across this line.



Show Moment About Cut Line. Shows moment about a cut line (**Fig. 9-7**)



Show Axial Force Across Cut Line (Fig. 9-7 (a))



Show Vertical Shear Along Cut Line (Fig. 9-7 (c))



Show Distribution of Contour Quantity Along Cut Line. This tool shows the distribution of the solution that you have currently selected. For example, if you select the distribution of stresses along the X-direction, and click on this tool, the program will show you how these stresses vary along the cut line you have selected. Unlike the previous tools, this tool applies to all the solution results.



Delete All Cut Line



Export Last Cut Line. You can retain one or more of the cuts you make, along with the associated solution for (i) further investigation, and/or (ii) inclusion in the report of your project. By clicking on this tool, the program stores the information on the cut. In addition, the program will enable you to review other information on the cut when you return to the program's main screen. For example, if the cut you have generated is to show the bending moment across the cut, when exported, the program will also calculate and display or report other actions, such as shear, stress and reinforcement associated with the cut.

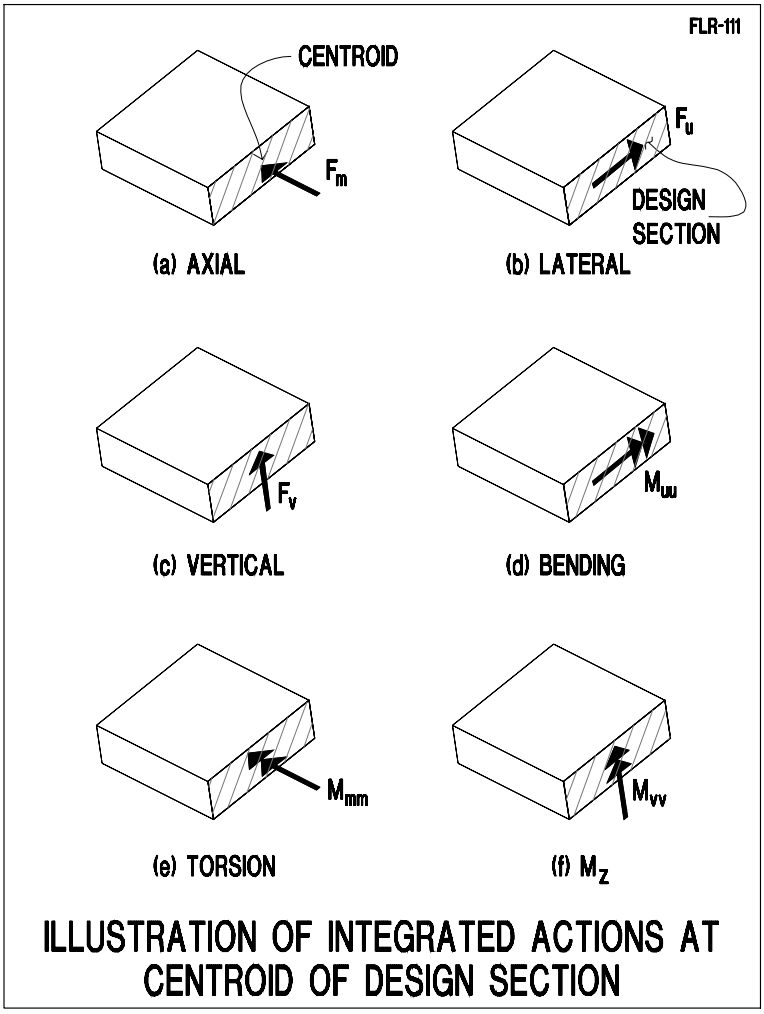


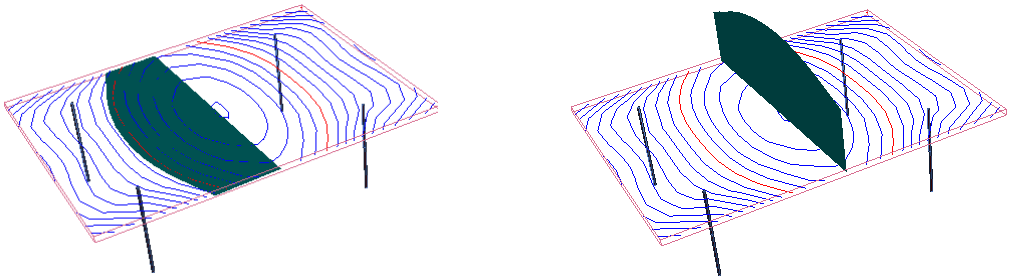
FIGURE 9-7 SIX ACTIONS AT THE CUT



Delete All Exported Cut Lines. This tool cancels the cuts that you had selected for further investigation using the tool described above.



Perpendicular Projection. The distribution of the variable you select can be displayed either on the plane of the floor system (**Fig. 9-8(a)**), or perpendicular to it (**Fig. 9-8(b)**). This tool toggles between the two display options.



(a) Display on plane of slab (b) Display normal to plane of floor

FIGURE 9-8 DISPLAY OPTIONS OF CUT LINE VALUE DISTRIBUTIONS

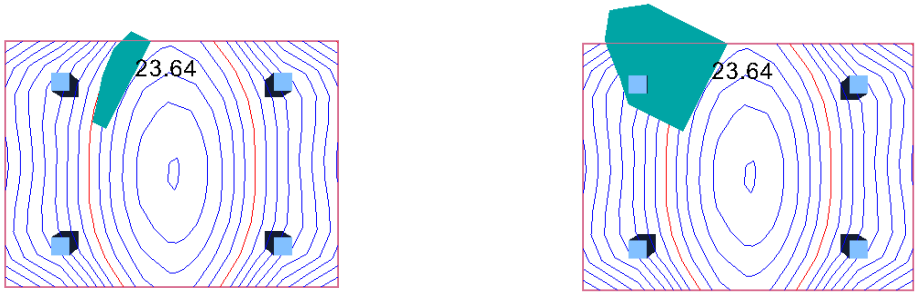


Scale Down Values

You can use this tool and the one shown below, to either reduce (**Fig. 9-9(a)**) or enlarge (**Fig.9-9(b)**) the display of the solution on a cut line.



Scale Up Values



(a) Display with reduced scale (b) Display with enlarged scale

FIGURE 9-9 DISPLAY OF DISTRIBUTION OF SOLUTION AT A CUT IN REDUCED OR ENLARGED SCALE



Scaling Toolbar

The functions of the tools in this toolbar are the same as *Support Line Result Scale Toolbar* that was explained earlier.

Chapter 10

DESIGN

10.1 OVERVIEW 147
10.2 DESIGN A SPECIFIC LOCATION 147
10.3 DESIGN A SPECIFIC REGION 150
10.4 DESIGN THE ENTIRE FLOOR SYSTEM 151
10.5 PUNCHING SHEAR DESIGN 155
10.6 REINFORCEMENT SPECIFICATION AND DISPLAY 161
10.6.1 Basic Features 161
10.6.1.1 Mesh Reinforcement Specification 162
10.6.1.2 Generation of Rebar Drawing 165
10.6.1.3 Reinforcement Display Control 167
10.6.1.4 Detailing, Printing, Plot Preparation and Reporting 171
10.6.2 Advanced Features (DRD Module) 175
10.6.2.1 Bonded and Distributed Bars Specification 177
10.6.2.2 Beam Rebar Specification and Display 181
10.7 TRANSFER OF POST-TENSIONING DESIGN FROM ADAPT-PT TO ADAPT FLOOR-PRO 185
10.7.1 Overview 185
10.7.2 Export of Data from ADAPT-Floor Pro to ADAPT-PT 186
10.7.3 Execution of Data in ADAPT-PT 188
10.7.4 Recall of PT Data from ADAPT-Floor Pro 191
10.7.5 Validation of PT Data in ADAPT-Floor Pro 193

10.1 OVERVIEW

Design consists of the calculation of (i) values (Design Values) that you use to determine the adequacy of the structure, and (ii) code compliance. Adequacy can be determined for the performance and safety of the structure under service conditions, and if the structure is post-tensioned, its adequacy at the time the tendons are being stressed (transfer of prestressing). In this section, you will learn how to use ADAPT-Floor Pro to achieve the deign objective.

Except for deflection, when you will be looking at specific points in the structure, all other design operations are based on *Design Sections* or in the case of punching shear an assumed *Critical Perimeter*.

ADAPT-Floor Pro offers you exceptional flexibility to perform this task because it allows you to.

- Choose to design a specific location of the structure
- Choose a specific region in the structure
- Have the program perform an automatic design for the entire floor system, and generate a professional quality report, ready for submittal

By way of an example, we will describe the three options of performing a design. **Figure 10.1-1** shows the floor system selected. The deflected shape of the floor is shown in **Fig. 10.1-2**.

10.2 DESIGN A SPECIFIC LOCATION

You can select specific locations, such as it is identified by design section 1 in **Fig. 10.2-1**. The program will determine the design values for this location; find the associated stresses; determine the reinforcement for each of the load cases you have defined; display the stresses in both a tabular and graphical form; find the reinforcement that may be required; find maximum and minimum deflection and report them all for your use.

The envelope of reinforcement required and the bar selection is also shown on the plan of the floor system (**Fig. 10.2-1**).

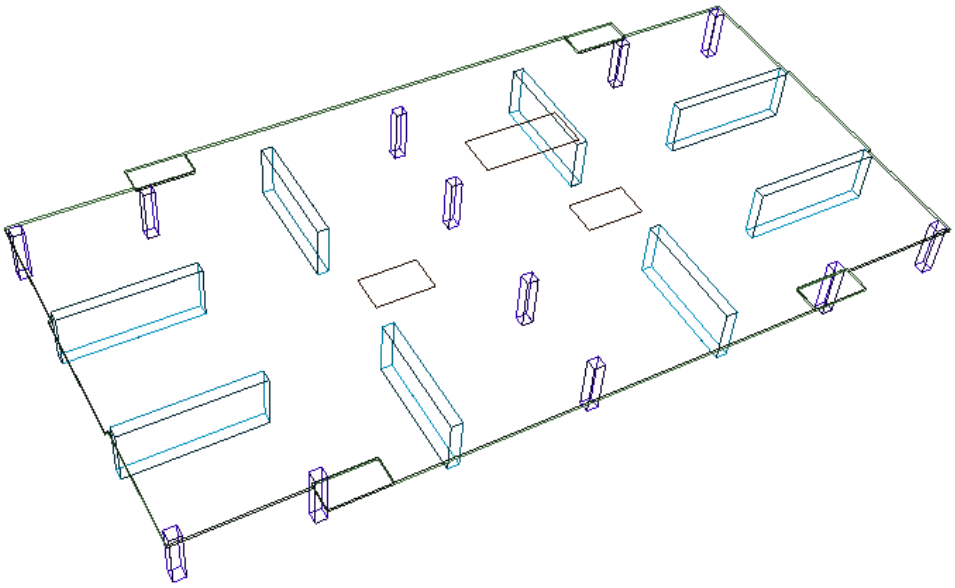


FIGURE 10.1-1 3D VIEW OF THE DESIGN EXAMPLE FLOOR SYSTEM

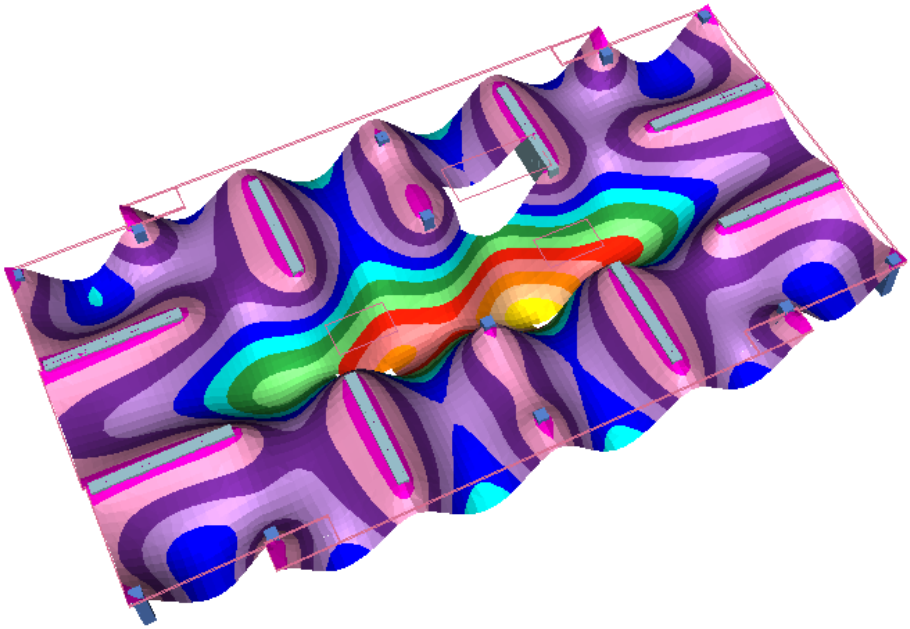


FIGURE 10.1-2 THE DEFLECTED SHAPE OF THE DESIGN EXAMPLE FLOOR

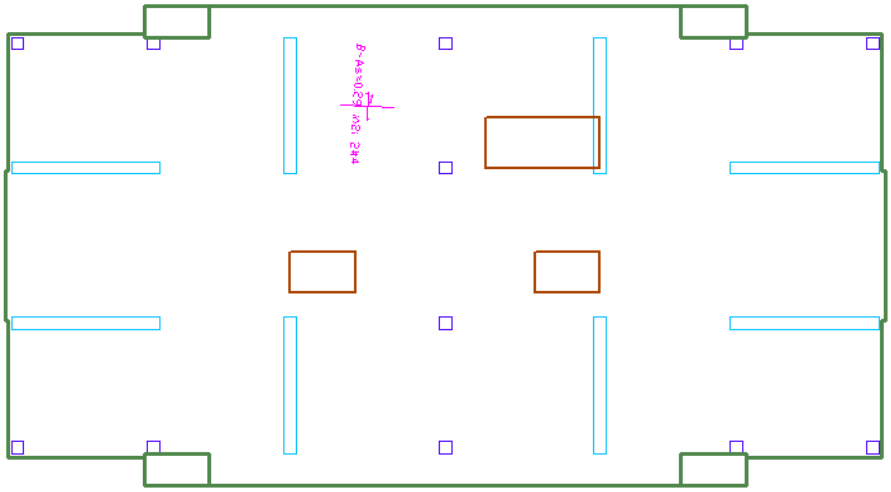


FIGURE 10.2-1 DISPLAY OF DESIGN SECTION 1

Figure 10.2-2 is the display of the actions at a manually drawn design section, along with the stresses for "service" condition and the envelope of reinforcement required.

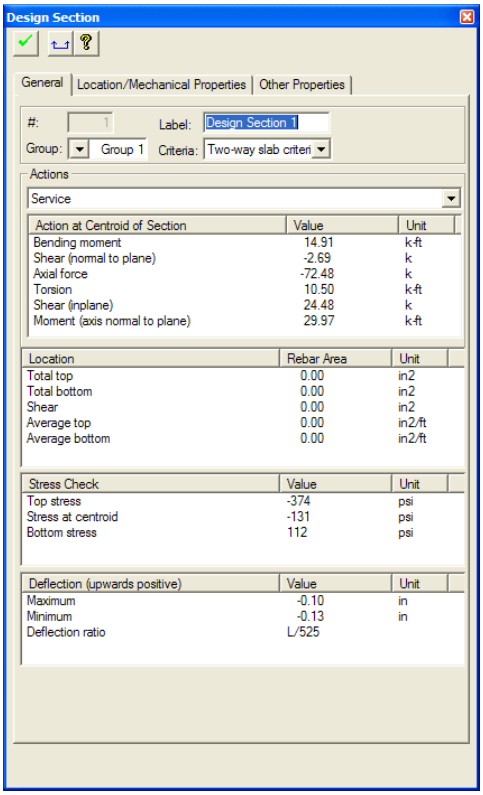


FIGURE 10.2-2 VIEW OF THE DESIGN OUTCOME FOR DESIGN SECTION 1

For a detailed account on the values reported for manual design sections, the computation of stresses and reinforcement, as well as the method you may use for their verification refer to the Technical Note.[1]

10.3 DESIGN A SPECIFIC REGION

Rather than a specific section, you can select a portion of the floor system and execute a design. **Fig. 10.3-1** illustrates a portion of the floor system selected for design. The selection is created by using a "splitter" around its boundary.

The direction of reinforcement to be reported by the program, as well as the direction of the stress for stress check, is identified by a support line that you draw across the entire length of the slab sub-region selected for design (**Fig. 10.3-2**).

Going through the standard design steps listed under the *FEM* pull-down menu, the program generates design sections (**Fig. 10.3-3**) within the slab sub-region selected and performs a complete code check and design. The outcome of the design, such as distribution of stress and the amount and length of reinforcement needed, will be reported by the program in the standard manner.

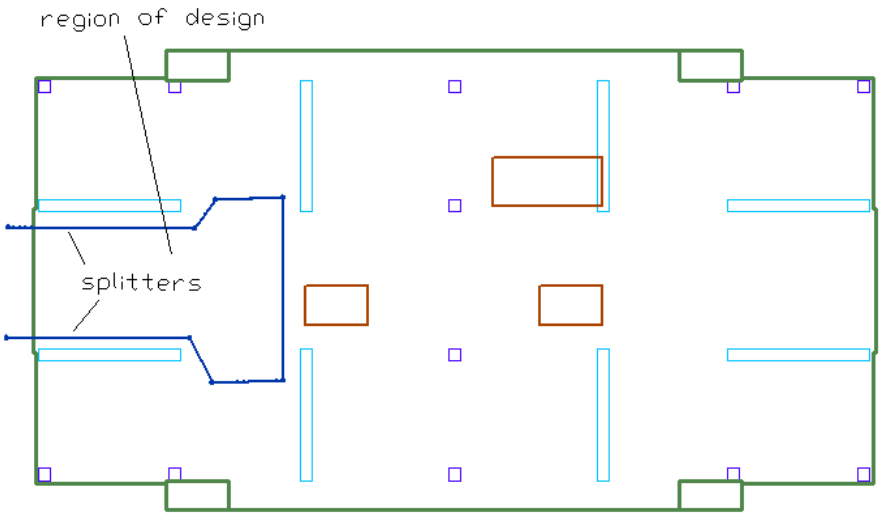


FIGURE 10.3-1 IDENTIFICATION OF A SLAB REGION FOR DESIGN

[1] This Technical Note TN-180: "*Stress Check and Rebar Verification*" is on your program CD-ROM and can be downloaded from ADAPT website, www.adaptsoft.com

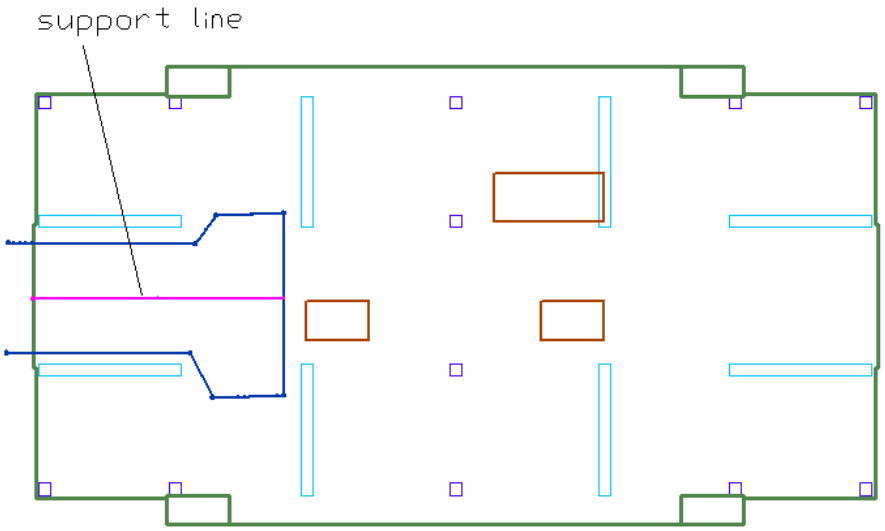


FIGURE 10.3-2 SUPPORT LINE FOR DIRECTION OF REINFORCEMENT

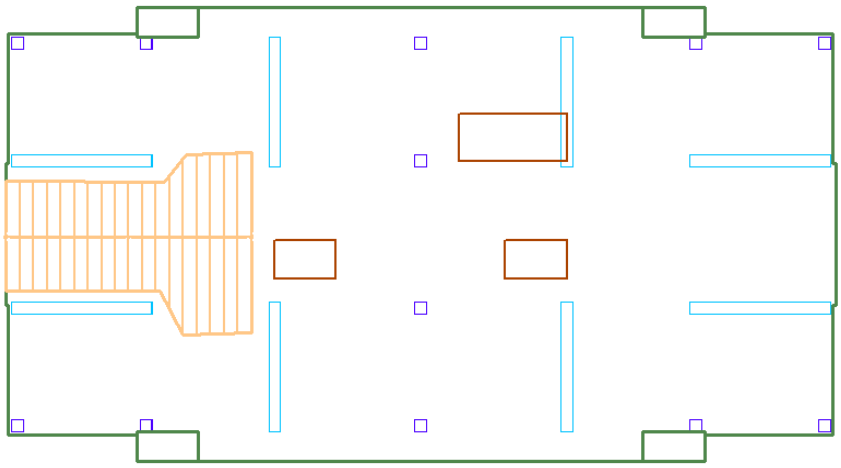


FIGURE 10.3-3 DESIGN SECTIONS FOR THE SLAB SUB-REGION

10.4 DESIGN THE ENTIRE FLOOR SYSTEM

Once the support lines are laid out for the entire floor system, you will need to generate the design sections by selecting “Generate Design Sections Automatically” from the *FEM* pull-down menu.

In a next step, select “Design the Design Sections” from the same pull-down menu. Once the design is complete, a message will appear asking if you would like to save the design results.

You can now review the design results for your model with the help of the Support Line Results toolbar (**Fig. 10.4-1**).

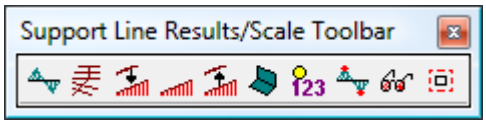


FIGURE 10.4-1: SUPPORT LINE RESULTS TOOLBAR

Display Graphically. Select this button to display those results for each support line that are selected in the "Results Display Settings" window (**Fig. 10.4-2**).

Display Design Sections. Click this button to display the design sections for all support lines. As soon as this button is selected, a floating toolbar is displayed that allows you to toggle between the display of support lines in the X and Y direction.

Scale Down Values. Use this button to scale down values for any curves that are displayed along the support lines.

Default Scale Values. In case where curves are displayed and the maxima are too large to fit, use this button to scale the curves back to a default scale.

Scale Up Values. Use this button to scale up values of any curves that are displayed along the support lines.

Perpendicular Projection. By default, all curves are displayed perpendicular to the slab surface. Select this button to flip the curves into the XY plane.

Numerical Display. Select this button to display the result values for each design section along the support lines.

Display Min/Max Values. Click this button to only display the minimum and maximum result values along the support lines.

Result Display Settings. Select this button to open the "Result Display Settings" window (**Fig. 10.4-2**) to select the results along the support lines such as actions, deflections, stresses etc.

Display Punching Shear Design Outcome. Once you have executed the punching shear design, the results can be reviewed in the model by clicking on this button

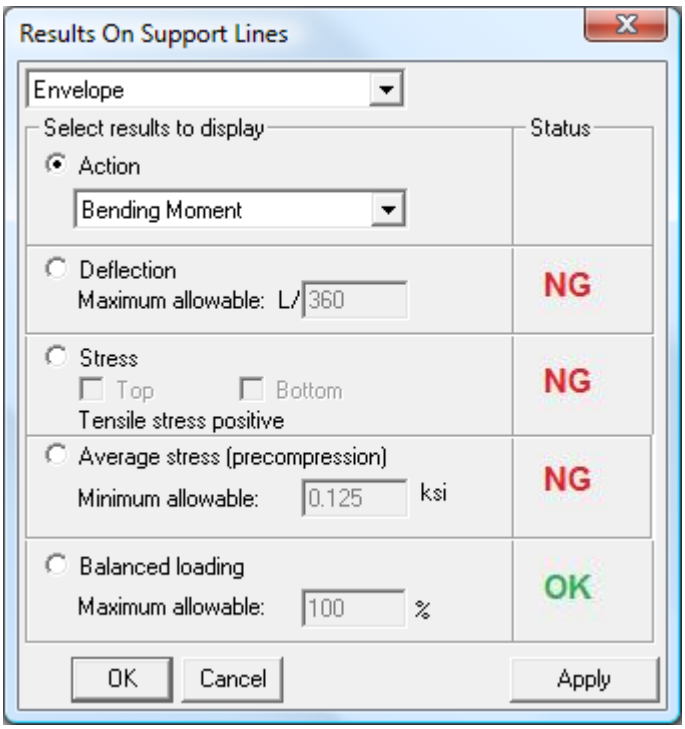


FIGURE 10.4-2: RESULT DISPLAY SETTINGS

The "Result Display Settings" window gives you the option to choose which results shall be reviewed for which load combination.

On top of this window the appropriate load combination has to be selected. Every load combination that was specified under Loading -> Load Combination -> FEM can be selected as well as the Envelope.

In the "Select results to display" group box you can select from five result groups: actions, deflection, stresses, precompression and balanced loading.

- Action: Select from the drop down box one of the six actions that you would like to review. To view the curve for the selected action, the "Display Graphically" button has to be clicked (**Fig. 10.4-1**).

- Deflection: Provide the maximum allowable deflection in the text box of the Deflection field. Clicking "Apply" will recalculate the status.

- Stress: Select either top and/or bottom stress to review. The stress limits are defined under Criteria -> General -> Allowable Stresses.

- Average stress (Precompression): You can provide the minimum allowable precompression and compare it to your actual average stress. Click “Apply” to recalculate the status.

- Balanced loading: Provide the maximum percentage of the balanced dead load and click “Apply” to recalculate the status.

The status for deflection, stress, average stress and balanced loading provides you an overall view on whether the criteria are met (OK) or not (NG). If the criteria for one result are not met, the color of the respective design section(s) will turn magenta, else the color will stay green.

Figures 10.4-3 and **10.4-4** show how the program covers the entire floor system by design sections. This is the only option that guarantees the safety and code compliance of the entire floor system.

A sample stress check for one of the design strips is shown in **Fig. 10.4-5**.

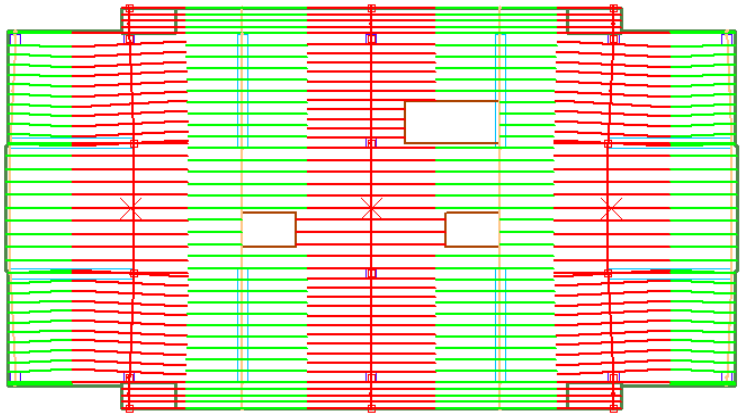


FIGURE 10.4-3 DESIGN SECTIONS ALONG Y-DIRECTION

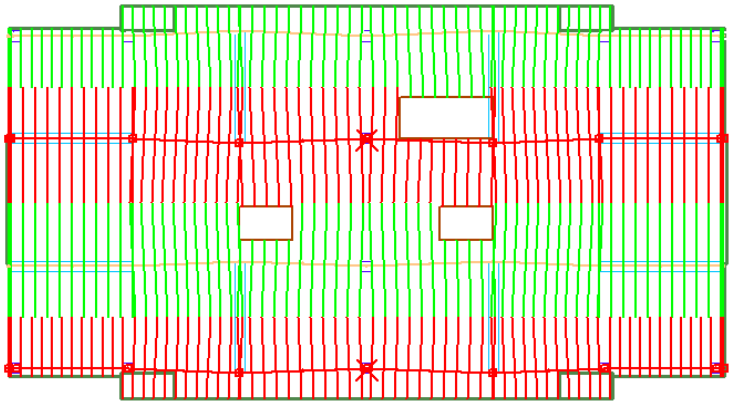


FIGURE 10.4-4 DESIGN SECTIONS ALONG X-DIRECTION

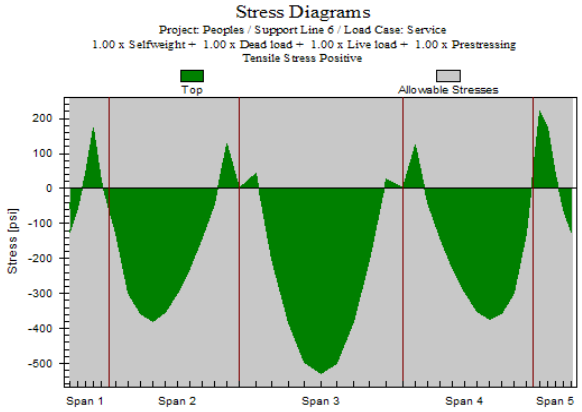


FIGURE 10.4-5 SAMPLE OF STRESS CHECK FOR A TYPICAL DESIGN STRIP

10.5 PUNCHING SHEAR DESIGN

ADAPT-Floor Pro performs an automatic punching shear design, reports the calculated punching shear stresses, and provides punching shear reinforcement, where required. You have the choice of specifying whether the punching shear reinforcement should be in the form of stirrups, studs or otherwise. In checking for punching shear, the program automatically recognizes whether a column's position is at interior, edge, or corner. Depending on the position of a column with respect to a slab edge, the program will automatically select the correct formulation and perform the relating code check.

Where there is a change in slab thickness at a column location, or its immediate vicinity, the program will conservatively select the smaller slab thickness for punching shear.

At locations of a drop cap or drop panel, the program performs the punching shear check both within the thickened drop cap/panel and in the slab outside the perimeter of the drop cap/panel.

Where punching shear reinforcement is required, the program continues checking the column-slab adequacy at code-specified successive layers (critical perimeters) around the column, until it reaches the perimeter, where no reinforcement is needed.

No punching shear check is performed for walls that have a length-to-thickness ratio larger than the program's default setting [4], or for columns butted by one or more beams. At these locations, the program reports (NA; Not Applicable). Punching shear design options are selected in the *Shear Design Options* tab (**Fig. 10.5-1**) of the *Criteria* pull-down menu.

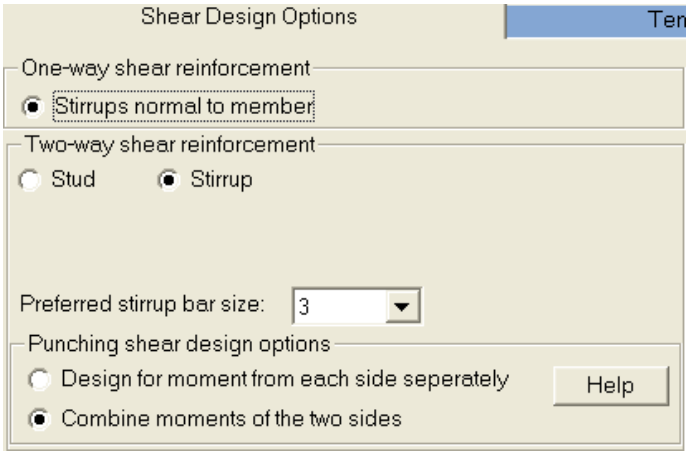


FIGURE 10.5-1 SHEAR DESIGN OPTIONS

The punching shear check can take place after you have successfully completed the analysis stage of your work. Click on *Punching Shear Check* item of the *FEM* pull-down menu.

If the model being checked for punching shear is set to the EC2, BS8110 or Hong Kong design codes an additional window, as shown in **Fig. 10.5-2**, will appear. If the design sections have not been designed and reinforcement has not been generated, the program will default to using the minimum reinforcement ratio per code in addition to contribution from precompression in calculating concrete shear strength.

If the design of sections and reinforcement has been carried out and the punching shear analysis is checked, the program gives the option to use the minimum reinforcement ratio or some percentage contribution of the program-calculated mild reinforcement envelope and/or bonded tendons in addition to contribution from precompression.

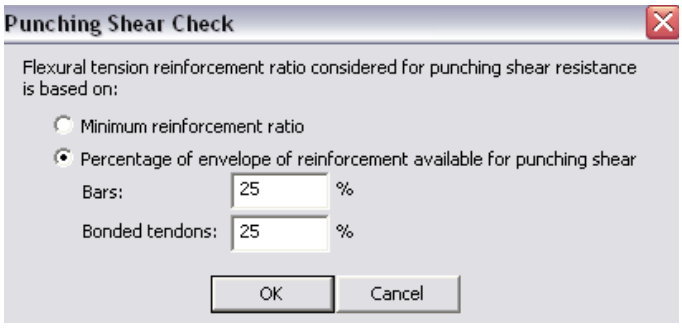


FIGURE 10.5-2 PUNCHING SHEAR CHECK REINFORCEMENT RATIO – AFTER REBAR DESIGN

The display of results on the screen and the tabular report of the program are illustrated by way of the following example. **Figure 10.5-3 (a)** shows the view of a slab supported on columns and a wall. The column identification is shown in part (b) of the figure. For the American code version of the program, you have the option of considering the moments in each of the orthogonal directions separately along with the column reaction (recommended), or consider the moments acting simultaneously. Parts (c) and (d) of the figure show the results from each of the two options. Two stress ratios (SR) are reported, if the moments are considered separately. Otherwise, one stress ratio (SR) for the combined moment will be reported.

To view the results of the punching shear design on the screen as shown in **Fig. 10.5-3**, there are two tools available in the *Support Line Result Scale* Toolbar:



Display Punching Shear Design Outcome. When this tool is selected the following design outcomes may be displayed:

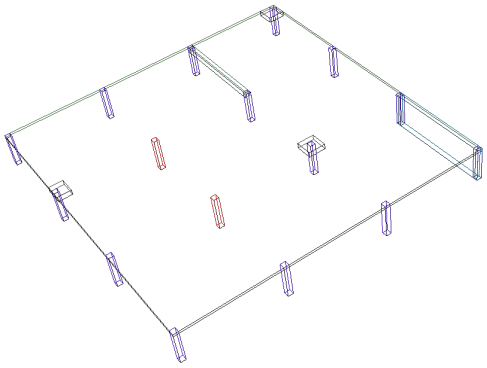
- "OK" – calculated stresses are below allowable stresses
- "Reinforce"- calculated stresses exceed allowable stresses and design code requires reinforcement
- "Exceeds Code" – design code requirements are not satisfied
- "NA"- punching shear design is not applicable



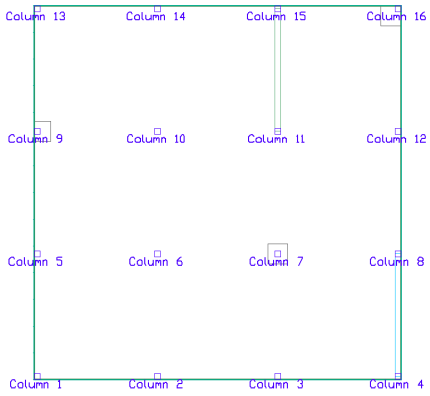
Numerical Display of Results. When this tool is selected the punching shear stress ratios will display on the screen.

ADAPT-Floor Pro provides a clear summary of the actions on each column, the design values and the outcome of the code check. The program's calculations and findings for the punching shear check are presented in the Reports section of the manual. A sample of punching shear reinforcement for stirrups is given in **Table 10.5-1**. A similar output for shear stud reinforcement is listed in **Table 10.5-2**.

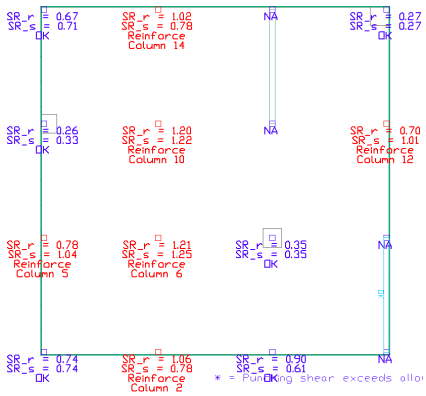
The reinforcement metrics reported in the two tables reads as follows: For stirrup reinforcement of Column 6 (**Table 10.5-1**): 12 legs at a distance of 3.25" from the face of column, followed by 10 legs at distance 6.50" from the face of column. The 12 and 10 legs are the total number around the perimeter of the column. Stirrup diameter is 0.38" (#3)



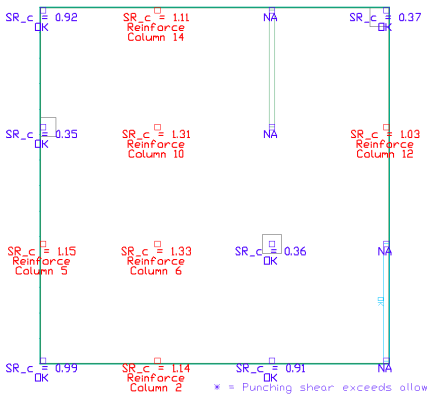
(a) 3D View of Floor System



(b) Column Identification



(c) Result Using Moments Separately



(d) Results Using Combined Moments

FIGURE 10.5-3 FLOOR SYSTEM, COLUMN IDENTIFICATION AND RESULTS

For shear stud reinforcement (**Table 10.5-2**):2 stud rails are specified per column side. For Column 6, each rail has 4 studs. The position of studs of each rail from the face of column is: 1.50, 3.25, 4.75, 6.50". The stud diameter is 0.38".

TABLE 10.5-1 PUNCHING SHEAR REINFORCEMENT USING STIRRUPS

180.80 SCHEDULE OF STIRRUPS FOR PUNCHING SHEAR REINFORCEMENT
Load Combination: Strength (Dead and Live)

Column ID	Perimeter	1	2	3	4	5	6
1	Distance	3.25					
	Legs	4					
2	Distance	3.25					
	Legs	7					
5	Distance	3.25					
	Legs	7					
6	Distance	3.25	6.50				
	Legs	12	10				
10	Distance	3.25	6.50				
	Legs	12	10				
12	Distance	3.25					
	Legs	6					
14	Distance	3.25					
	Legs	7					

Notes:
Distances are in (in)
Legs = total number of vertical reinforcing bars for the layer: bar size #4(0.50in)
Distance = distance of the vertical reinforcing bars from the respective column face
Distance (9)15.75 indicates layer 9 at distance 15.75 in
Legs (9) 12 indicates layer 9 with 12 vertical legs
Distance (9)15.75* indicates layer 9 at distance 15.75 in measured from the face of drop caps
"*" = exceeds maximum allowable by code

Refer to details for arrangement (**Fig 10.5-4**).

TABLE 10.5-2 PUNCHING SHEAR REINFORCEMENT USING SHEAR STUDS

180.90 SCHEDULE OF STUD RAILS FOR PUNCHING SHEAR REINFORCEMENT
Load Combination: Strength (Dead and Live)
Number of rails per side: 2

Column ID	Number of studs/rail	Stud diameter	Studs	1	2	3	4	5
2	1	0.38	Distance	3.25				
5	1	0.38	Distance	3.25				
6	4	0.38	Distance	1.50	3.25	4.75	6.50	
10	2	0.38	Distance	1.50	3.25			
12	1	0.38	Distance	3.25				
14	1	0.38	Distance	3.25				

Notes:
Dimensions are in (in)
Distance = distance of the vertical bars from the respective column face
(7) 16.5 indicates layer 7 at distance 16.50 in from face of support.

(7) 16.5* indicates layer 7 at distance 16.50 in from face of drop caps.
"*" = exceeds maximum allowable by code

Refer to details for arrangement (**Fig. 10.5-5**).

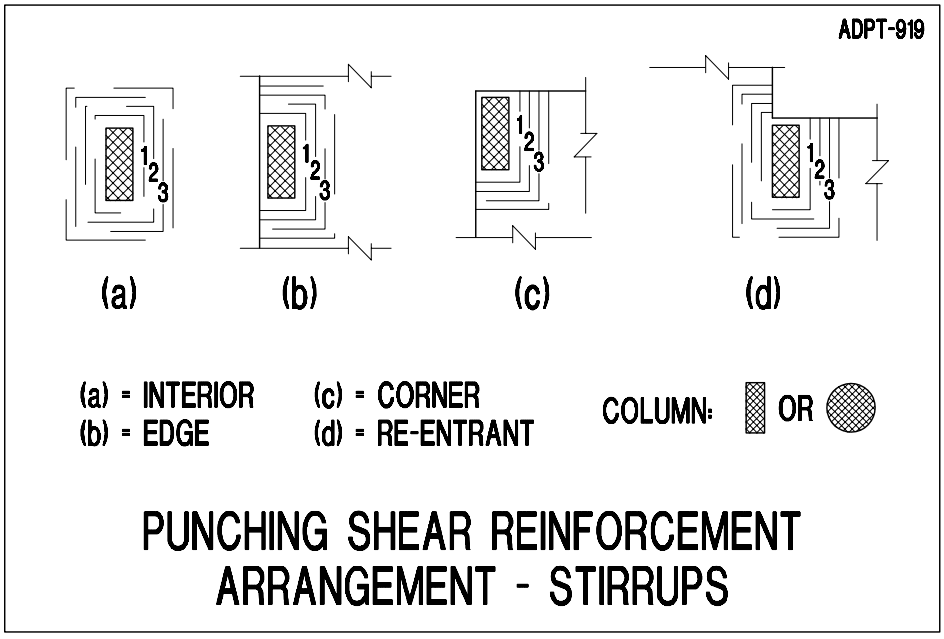


FIGURE 10.5-4 ARRANGEMNET OF PUNCHING SHEAR REINFORCEMENT USING STIRRUPS

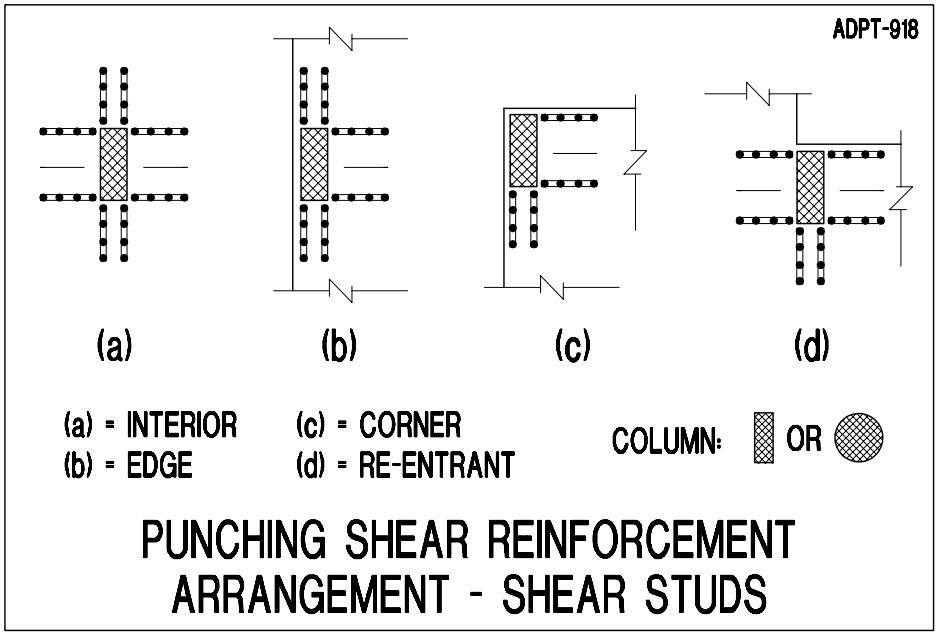


FIGURE 10.5-5 ARRANGEMNET OF PUNCHING SHEAR REINFORCEMENT USING SHEAR STUDS

10.6 REINFORCEMENT SPECIFICATION AND DISPLAY

This section describes the features available for reinforcement specification and display. Basic features allow you to specify mesh reinforcement prior to design, to display reinforcement plans, and to generate reinforcement reports. Advanced operations such as specifying rebar and editing rebar drawings, predefining beam rebar, inserting templates of beam details in your drawing, and more, are available in the "Dynamic Rebar Design" (DRD) module[2].

10.6.1 Basic Features

Basic features for reinforcement specification and display enable you to:

- Specify mesh reinforcement.
- Generate rebar drawing in plan.
- Control reinforcement display.
- Print/plot reinforcement plan.
- Generate tabular reports.

All basic tools related to reinforcement specification and display are grouped in the Reinforcement toolbar.



Reinforcement toolbar



Generate Rebar Drawing. This tool creates/refreshes the generation of rebar drawing.



Open Rebar Display Options. The dialog window gives you full control over the display of reinforcement.



Display/Hide Rebar. This is simply a toggle switch to turn the display of the entire reinforcement on the plan on or off.

[2] Contact ADAPT Co. to activate DRD module.



Create Mesh Reinforcement. This tool enables you to specify a wire mesh fabric or rebar layout over one or several areas of the floor system.



Mesh Rebar Wizard. The wizard lets you define a mesh reinforcement over a specific region of slab that you select.

10.6.1.1 Mesh Reinforcement Specification

You can specify a base mesh reinforcement in the slab prior to the design of your structural model. The program automatically accounts for the presence of the reinforcement specified by you, and will calculate and report the reinforcement required in addition to the amount you provided.

You can add mesh reinforcement using:

- Build pull-down menu/Rebar option, or
- Build toolbar/Show Rebar toolbar, , or
- User Interface menu/Reinforcement toolbar.

To add mesh reinforcement use Create Mesh Reinforcement tool, . Click on the tool to initiate the identification of the area over which you plan to specify the base reinforcement. Follow the instructions on the screen to draw the area covered by the mesh. **Figure 10.6-1** displays a region drawn to represent a mesh coverage. The program will apply the mesh only over the slab regions within the area specified by you.

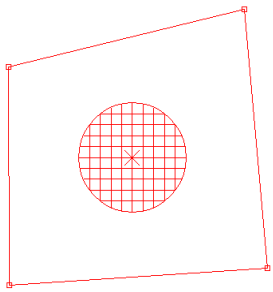
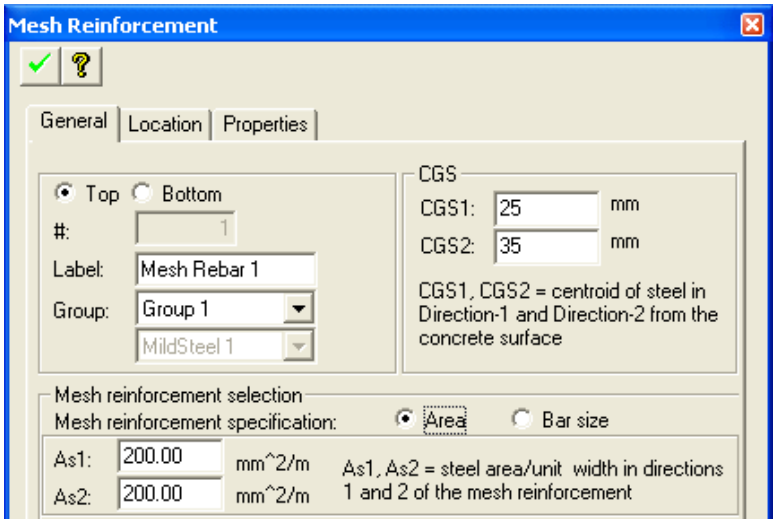


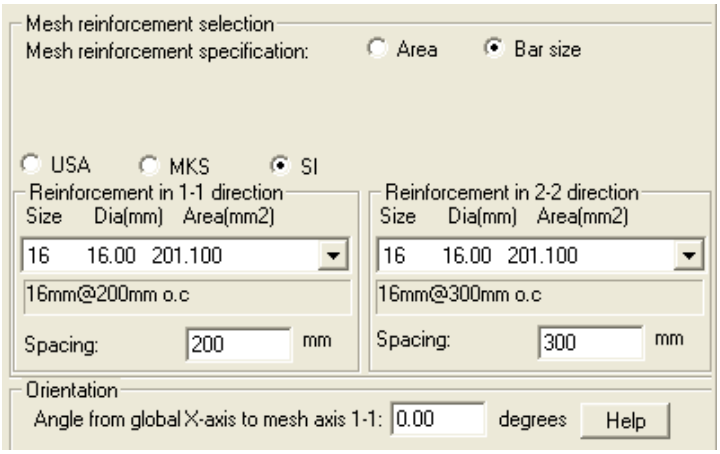
FIGURE 10.6-1 DEFAULT DISPLAY OF MESH

Double click on the mesh to open its property dialog box (**Figure 10.6-2**). In this dialog window, you can specify the following:

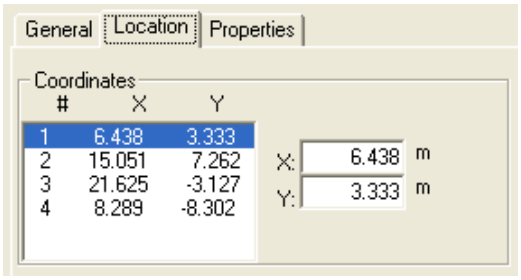
- Area of reinforcement in each direction
- Orientation of the mesh with respect to the global coordinates
- Position of the mesh at the top or bottom of the slab, and the distance to the centroid of the mesh bars in each direction from the slab surface



(a) Mesh Specification by Steel Area



(b) Mesh Specification by Bar Size and Spacing



(c) Editing of Mesh Region by Coordinates

FIGURE 10.6-2 DIALOG WINDOW OF MESH PROPERTIES

In **Figure 10.6-2**, and throughout the mesh reinforcement data input, it is assumed that the primary mesh reinforcement will be placed in direction 1-1. Optionally, a second set of parallel reinforcement can be specified in direction 2-2 normal to 1-1. The specified reinforcement need not be in the direction of the global axis. The orientation of the reinforcement with respect to the global axis X-X, however, must be specified. This is achieved by declaring the angle between the X-X axis and direction of the primary reinforcement in the mesh (1-1) measured counterclockwise. Further, by specifying the CGS (Center of Gravity of Steel) from the bottom fiber of the member for the bottom mesh, or from the top fiber of the member for the top mesh, you identify which layer of reinforcement is the inner and which is the outer.

To specify mesh reinforcement you can also use Mesh Rebar Wizard tool, . The advantage of the wizard is that you no longer need to draw the boundaries of the mesh, if the slab region over which the mesh applies has already been defined. When you activate the Mesh Rebar Wizard, the program first asks you to identify the slab region over which you want to specify a mesh. Once you identify the region, the program places a mesh using its default values. Next, you will edit the properties of the mesh specified by the program, if the default values are not as you choose. To edit the mesh reinforcement, double-click on it.

The mesh reinforcement input dialog window (**Figure 10.6-3**) will open. Enter the mesh details.

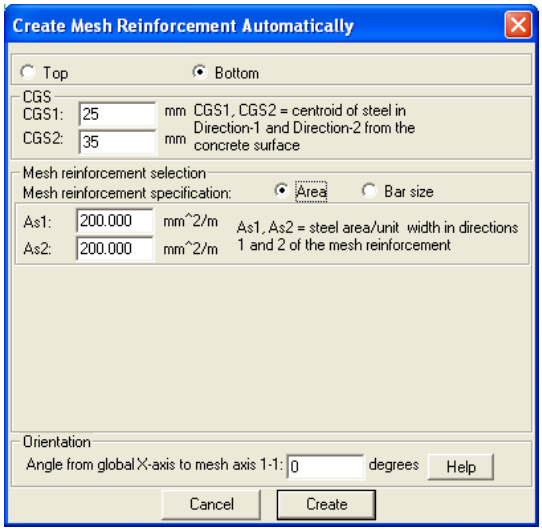


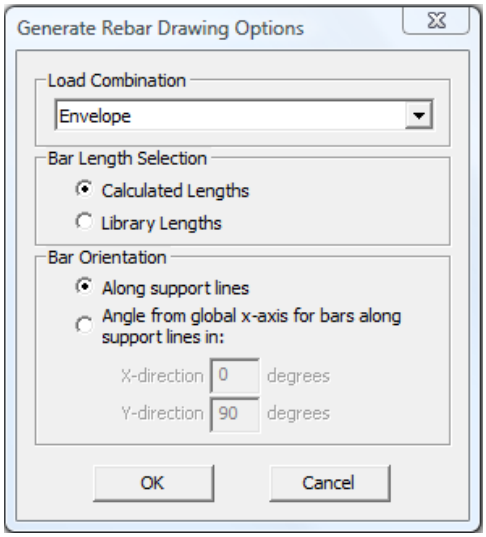
FIGURE 10.6-3 CREATE MESH REINFORCEMENT AUTOMATICALLY DIALOG WINDOW

10.6.1.2 Generation of Rebar Drawing

The program can generate a plan view of the reinforcement for each load combination after you complete the design of your structural model. To create the drawing follow the steps below:

1. Go to FEM -> Generate Rebar Drawing, or select  in the Reinforcement toolbar.

2. The dialog window shown below opens



3. From the drop down list select the load case for which you would like to display the calculated reinforcement

4. Select if the displayed bar length will be equal to the calculated length or taken from the library you previously defined.

5. Set bar orientation.

6. Click OK.

7. The program displays a plan such as shown in **Figure 10.6-4 (1)**, with the reinforcement displayed according to the program's default settings. You can control the display as described below.

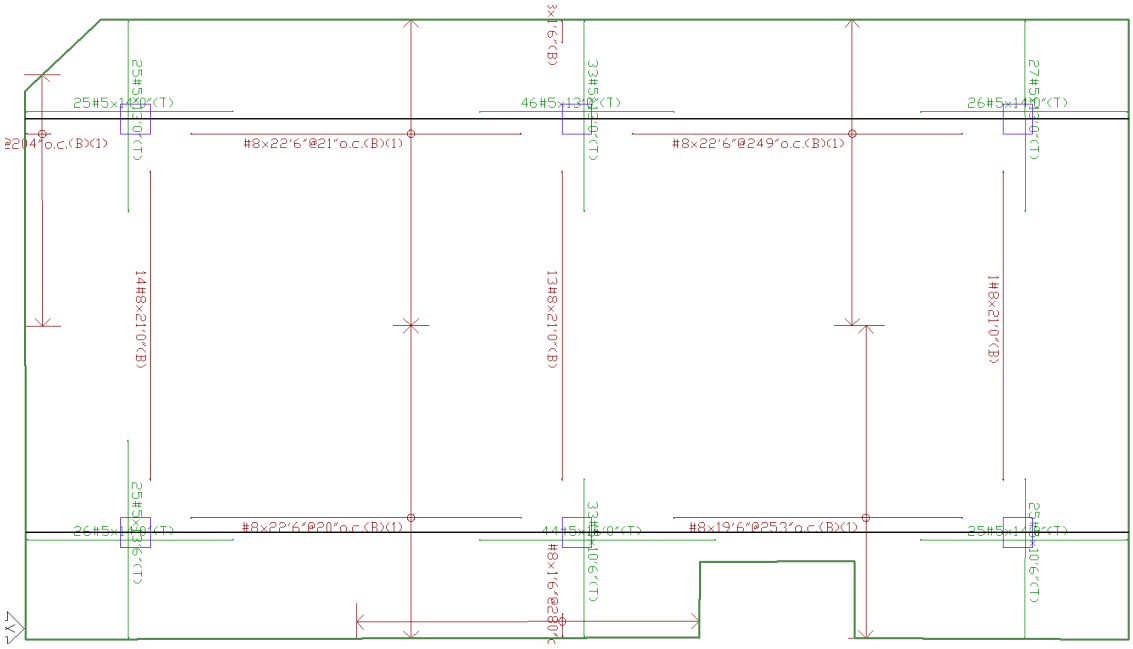


FIGURE 10.6-4 (1) SAMPLE REBAR DRAWING IN PLAN

In addition to the plan view you can also see calculated rebar in elevation along each support line.

To see the reinforcement in elevation do the following:

- Turn on the support line.
- Select the support line (double-click on it).
- From the support line property box, select the section display tool and click at the location where you want to insert elevation view. The program will display the reinforcement along the support line as shown in Figure 10.6-4(2)

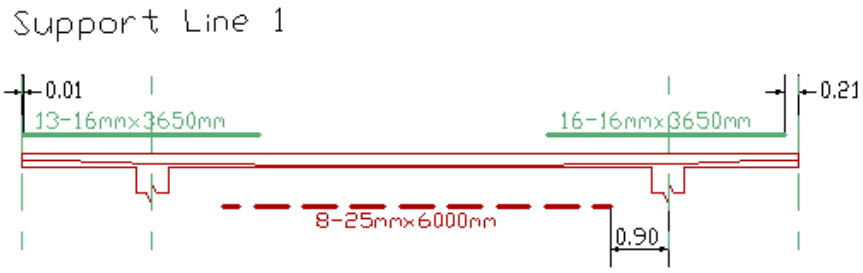


FIGURE 10.6-4(2) SAMPLE REBAR DRAWING IN ELEVATION

10.6.1.3 Reinforcement Display Control

The display of reinforcement can be controlled using the "Open Rebar Display Options" tool, , in the Reinforcement toolbar.

This tool opens the dialog window shown in **Fig. 10.6-5**. The dialog window gives you full control over the display of reinforcement. Several of the options available in the dialog window are explained it its Help file, reproduced in **Fig. 10.6-6**. The remainder is either self-explanatory, or described next, by way of using a partial plan of a typical rebar display. The partial plan (**Fig. 10.6-7**) displays two columns and the span between them. In this figure, the top bars (marked "T") over the column are grouped together, but the bottom bars in the span (marked "B") are distributed over a range identified by the extents of the arrows.

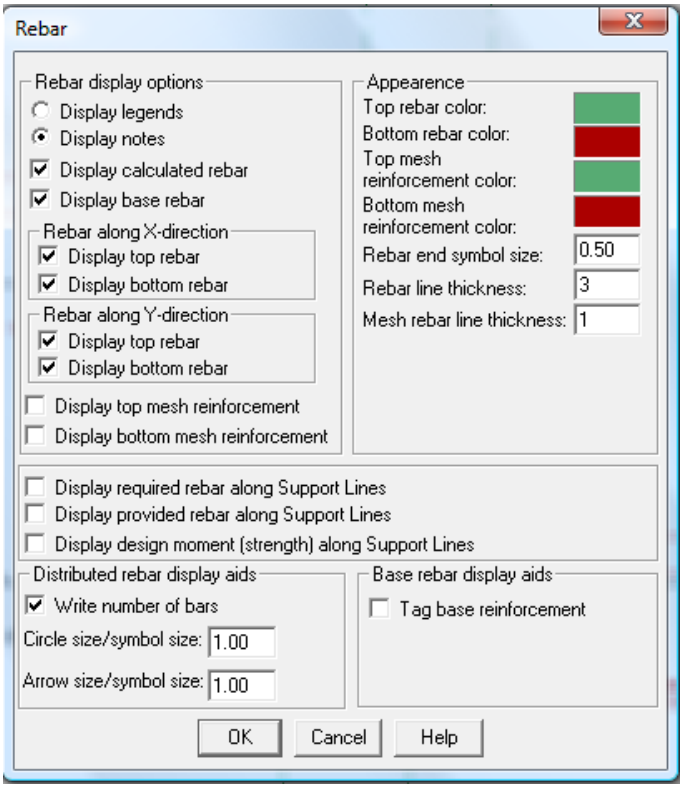


FIGURE 10.6-5 REBAR DISPLAY OPTIONS WINDOW

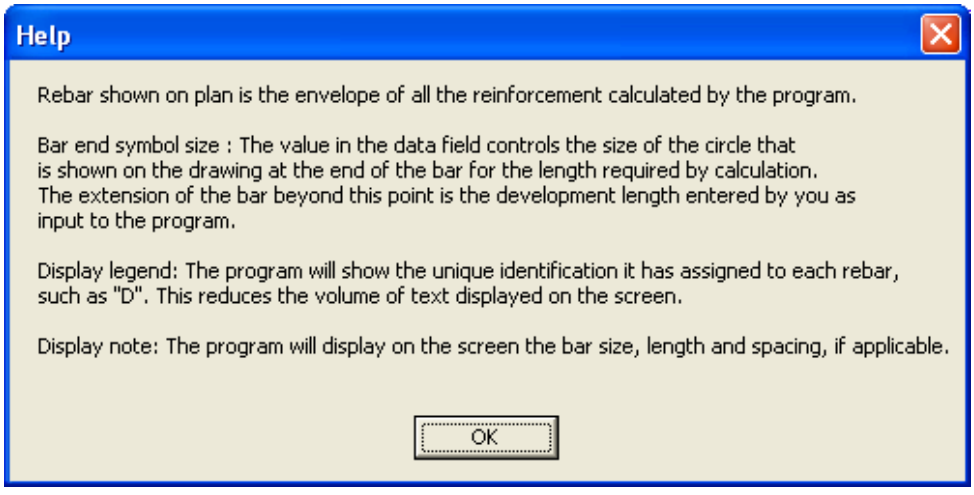


FIGURE 10.6-6 HELP FILE OF REBAR DISPLAY DIALOG WINDOW

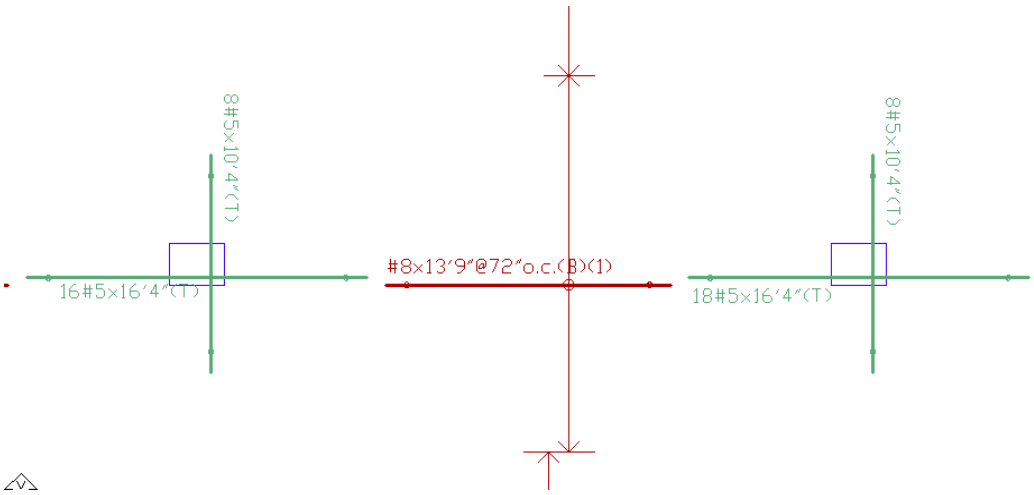


FIGURE 10.6-7 PARTIAL VIEW OF PLAN SHOWING DISPLAYED REBAR

Refer to **Fig. 10.6-8** for descriptions of some of the other options listed in the rebar display dialog box. For clarity in illustration, only the bottom bars are displayed. The solid line between the two columns represents the support line for which the reinforcement was calculated.

- **Display required rebar along Support Lines**. The required rebar calculated by the program is displayed in the form of a bar chart drawn normal to the support line. The position of each line of the bar chart (displayed normal to the support line) represents the location where reinforcement was required. The length of the line in the bar chart drawn normal to the support line is an indication of the amount of reinforcement needed. In the figure shown, bottom reinforcement was needed only over the central region of the span.
- **Display provided rebar along Support Lines**. This tool displays a rectangle normal to the Support Line. The height of the rectangle represents the area of reinforcement provided and reported in the text legend below the rebar line. The width of the rectangle along the support line represents the length of the bar. The short piece of rebar extending beyond the width of the rectangle representing the provided rebar is the development length of the bar. By design, the rectangle of provided rebar envelops the bar lines of the chart that represent the required reinforcement, unless edited otherwise by the user.

- **Display design moment (strength) along Support Lines**. In a similar manner, the design moment used to calculate the reinforcement for strength condition of the design can be displayed on the support line along the rebar reported by the program. This is critical information, in particular for non-prestressed slabs, where the inflection points along the moment diagram determine the cutoff points of the bars.
- **Circle Size/Symbol Size**. The number in this data field controls the size of the circle that is drawn at the intersection of a bar and the line that represents its extent.
- **Arrow Size/Symbol Size**. The number in the data field controls the size of the arrow arms at the end of the extent line of distributed reinforcement.
- **Rebar End Symbol Size**. This determines the size of the small square that marks the end of a bar required by calculation. The extension of a bar beyond this symbol is the development length. You define the development length as part of the input data in the "Criteria" input tab of the program.
- **Rebar Line Thickness**. This data field specifies the thickness of the rebar line shown on the computer screen.
- **Mesh Rebar Line Thickness**. This data field specifies the line thickness of the display symbol selected for the mesh reinforcement on the plan.
- **Tag base reinforcement**. This option allows you to easily distinguish between base and calculated rebar on the rebar plan. Once checked the program marks all base rebar with label "BR" (see **Fig. 10.6-9**)

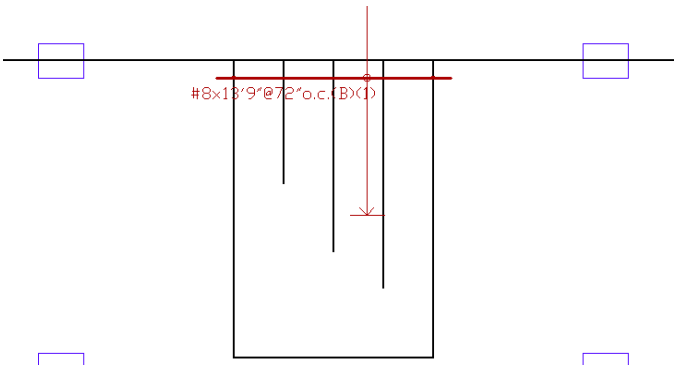


FIGURE 10.6-8 PARTIAL VIEW OF PLAN SHOWING DISPLAYED REBAR

You can control the size of the display font (size of text) by changing the font size of the "Rebar" item in "Select/Set View Items",  This adjusts the size of the font on the screen for better viewing.

The text of the bar normally appears below its line. However you can pick the cross (shown in **Fig. 10.6-9** over the numeral "6") and move it to the location of your choice. The ability to move the text helps to alleviate congested displays.



FIGURE 10.6-9

10.6.1.4 Detailing, Printing, Plot Preparation and Reporting

You can add previously prepared details, such as standard details for your project, to your drawing, prior to printing it. These details can be added at the location of your choice and scaled to suite the drawing you are working on.

Text, line thickness and symbol sizes that are suitable for viewing a drawing on the computer screen are not necessarily optimal for large-size drawings common in construction practice. As a result, prior to either plotting or exporting your drawing, you would select a set of suitable text and line sizes. The program allows you to set default values for your print and plot outputs, without affecting the view of the structure on the screen.

Open the Rebar Draft/Print/Detail Toolbar from the user interface pull-down menu.



Rebar Draft/Print/Detail/Report

This toolbar enables you to add details to your drawings and set the parameters of printing or plotting your model. It also includes tools for reporting the quantity of reinforcement used. The latter feature is described in the next section.



Prepare for DWG. To set the parameters that are suitable for export of your rebar drawing to an AutoCAD DWG file, click on this tool to open its dialog window, shown in **Fig. 10.6-10**.

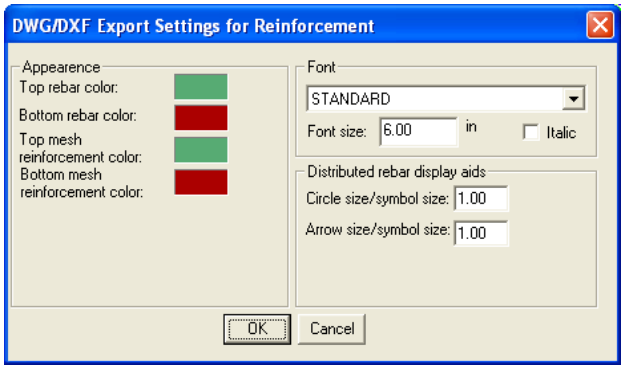


FIGURE 10.6-10 DIALOG WINDOW FOR EXPORT TO AUTOCAD



Configure Plot/Print Settings. Click on this tool to open the dialog window (**Figure 10.6-11**). This dialog window allows you to set the display parameters suitable for printing directly from your computer using the Builder platform print option. Obviously, parameters such as the size of the text and the thickness of the lines will depend on the size of the drawing you plan to print/plot. These are generally different from the values you select to view the structural drawing on plan.

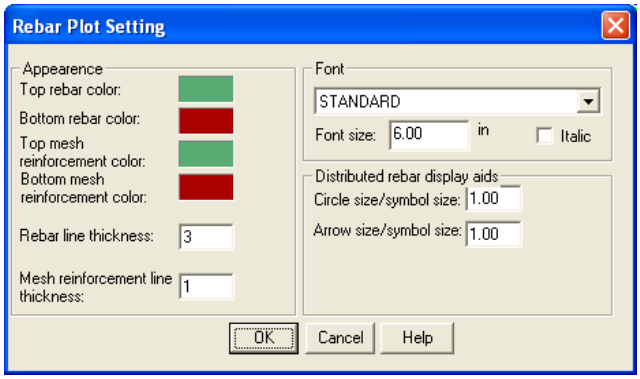


FIGURE 10.6-11 DIALOG WINDOW FOR SETTING PLOT PARAMETERS OF FLOOR-PRO



Insert Pre-prepared Details. Activate this tool and follow the instructions on the screen to place images of the

basic templates at a location of your choice on your plan. The templates are intended to form part of the rebar drawing you prepare from your structural design.

Once inserted, you can adjust the size of the image to match the layout of your drawing by using the "Scale Selected Item(s)" tool from the "Modify" pull-down menu

The reinforcement shown on your drawing (all base rebar and calculated rebar for the selected load combination) is organized in summary reports. The reports give the details of each bar along with the total weight of reinforcement and the average reinforcement per unit area of floor slab or per unit volume of concrete used.

 **Report Rebar Legends**. This tool generates a list of bar legends. A sample report of this tool is given below.

199.00 REBAR LEGEND

Bar Reinforcement (calculated and user defined base reinforcement)
Load combination: Envelope

#	Legend	Note
1	A	#8x3'6"@167"o.c.(B)
2	B	1#5x4'6"(T)
3	C	#8x5'0"@148"o.c.(B)
4	D	#8x17'0"@170"o.c.(B)
5	E	3#5x7'0"(T)
6	F	4#5x16'0"(T)
7	G	4#5x16'6"(T)

 **Rebar List.** This tool generates a detailed list of all the reinforcement used in the structure you have analyzed. A sample report of this tool is given below.

200.00 REBAR LIST

Bar Reinforcement (calculated and user defined base reinforcement)
Load combination: Envelope

#	Legend	Size	System	No.	Length	Unit Weight
					in	lb/ft
1	F	#5	US	4	192.00	1.04
2	F	#5	US	4	198.00	1.04
3	D	#8	US	1	204.00	2.67
4	G	#5	US	9	144.00	1.04
5	B	#5	US	1	54.00	1.04
6	E	#5	US	3	84.00	1.04
7	C	#8	US	1	60.00	2.67
Total Length(ft):	289.00	Total Weight(lb):	342.93			

 **Rebar Totals.** This tool generates a summary of total reinforcement type and weight of the project, as well as the rate of reinforcement per unit area of floor system and per unit volume of concrete used. A sample report generated by this tool is given below.

210.00 REBAR TOTALS

Bar Reinforcement (calculated and user defined base reinforcement)
Load combination: Envelope

Group	Quantity	Size	Length/bar	Length	Weight	Unit cost	Total cost
	bars		ft	ft	lb	$/lb	$
A	1	#8	3.50	3.50	9.35	1.35	12.62
B	1	#5	4.50	4.50	4.69	1.35	6.34
C	1	#8	5.00	5.00	13.35	1.35	18.02
D	1	#8	17.00	17.00	45.39	1.35	61.28
E	3	#5	7.00	21.00	21.90	1.35	29.57
F	4	#5	16.00	64.00	66.75	1.35	90.12
G	4	#5	16.50	66.00	68.84	1.35	92.93
H	9	#5	12.00	108.00	112.65	1.35	152.07
Total				289.00	342.93		462.95

Note:
Group = Identification assigned to a group of bars with the same diameter and length.

Mesh Reinforcement

ID	Size 1	Spacing 1	As1	Size 2	Spacing 2	As2	Area	Weight	Unit cost	Total cost
		in	in2/ft		in	in2/ft	ft 2	lb	$/lb	$
1	#5	24.00		#5	24.00		53.874	56.52	1.35	76.30
2			0.10			0.10	32.578	22.05	1.35	29.77
Total								78.56		106.06

Architectural Beam Reinforcement

Beam label	Quantity	Size	Length/bar	Length	Weight	Unit cost	Total cost
	bars		ft	ft	lb	$/lb	$
Beam 1	4	#5	24.00	96.00	100.13	1.35	135.18
Beam 2	4	#5	24.00	96.00	100.13	1.35	135.18
Total				192.00	200.26		270.35

Total volume of concrete = 572.80 ft^3 (21.21/cu yd)
Total surface of floor slabs = 859.20 ft^2
Average rebar usage = 0.72 lb/ft^2, 1.05 lb/ft^3 (0.05 lb/cu yd)
Total cost of reinforcement = 839.36 $

 **Rebar Summary.** Click on this tool to open the dialog window (**Figure 10.6-12**). This dialog window displays which rebar sizes are used in your structure including their length and quantity.

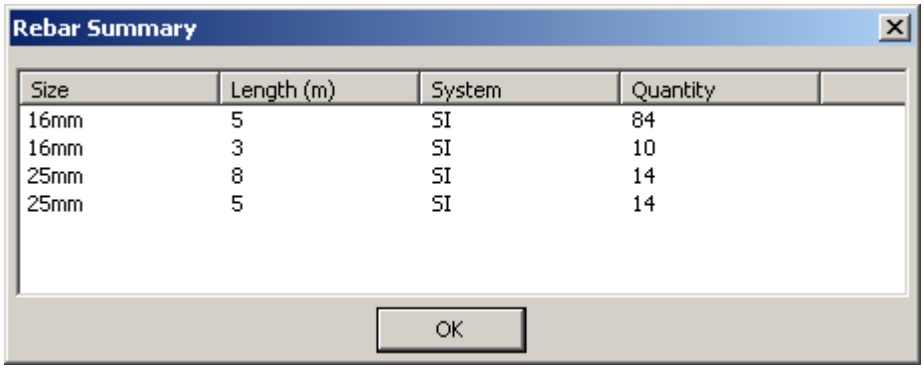


FIGURE 10.6-12 REBAR SUMMARY

10.6.2 Advanced Features (DRD Module)

In addition to the basic features already explained in previous sections, you can use advanced features. Advanced features are specific for the "Dynamic Rebar Design Module" (DRD). These features enable you to:

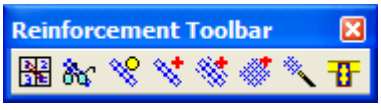
- Interactively edit the reinforcement calculated by the program and shown on its rebar drawings. Dynamic editing means that any change you make in any of the rebar properties, such as its size, results in an automatic change in the associated properties of the same bar. For example, if you change the diameter of a bar, the program automatically changes the number of bars to keep the total area the same. In addition to links among the properties of a single bar, such as its area and nominal size, each rebar, being a live object, is linked upstream with the information in your structural analysis model, and downstream with the list of quantities and reporting. A change you make in the layout of a bar will automatically be reflected in your structural model and your report.
- Define a single bar or a group of reinforcing bars at the location of your choice prior to the structural calculation. At design stage, the program will recognize the pre-defined reinforcement (base reinforcement), and report added reinforcement, should the rebar you define not be adequate.
- For beam design, you can define the two top and the two bottom corner bars of the reinforcing cage. The program will determine the reinforcement in addition to the four bars selected to construct the cage.
- Once the program completes the design, it reports the entire reinforcement necessary, including the base reinforcement that you may have pre-defined as input.

- In line with the practice of creating engineering drawings, the program displays the slab rebar on plan, and the beam rebar on the elevation of the beam.
    - Each reinforcement entity on plan is clearly annotated, giving the number of bars, size, and length. Once the entities are displayed on plan, you have full control to change the bar number, its size and length. However, since the bars displayed on plan are not merely lines, but are defined as "objects" linked with the remainder of your design parameters, the program can guide you in your bar editing and let you know whether the change of length, location and other characteristics that define a reinforcing bar are in violation of the code requirements used by the program.
    - Along the length of a bar on plan, you can request that the program display the bending moment diagram used in the determination of the bar's value and length. Also, the program can show you graphically the minimum amount of the reinforcement necessary for each location, alongside the reinforcement provided by you or the program.
    - For the beams, the program displays on the beam elevation the reinforcement that is necessary along with the four corner bars you select. The location of each added bar is shown clearly on the elevation. Again, you have full control to edit the reinforcement selected by the program.
    - The stirrup requirements of a beam are also shown graphically on the elevation of the beam, ready to be exported to the structural drawings.

There are other features that facilitate the creation of structural drawings. These are reported elsewhere[3].

- A complementary feature to the rebar module is the summary report of the entire non-prestressed reinforcement shown on your drawing. This consists of the reinforcement that was used in the calculation and reported by the program, along with the reinforcement that you may have added manually and subsequent to the calculations for structural detailing.

All DRD module features are grouped in the Reinforcement toolbar.



Reinforcement Toolbar

[3] Refer to the "workshop/tutorial" package for details.

The tools not listed below were explained in Section 10.6.1.



Create Grouped Rebar. With this tool, you define an isolated bar, or a group of reinforcement bars at the location of your choice. This tool is available in the Build/Rebar menu.



Create Distributed Rebar. The distributed rebar tool is used when a number of identical bars with identical spacing are to be specified.



Beam Rebar. The Beam Rebar tool is used to specify the size of the two corner bars at the top, at the bottom, or both at the top and the bottom of the reinforcing cage.

10.6.2.1 Bonded and Distributed Bars Specification

You can specify reinforcement for the program to account for in its calculation as pre-defined (base reinforcement), add reinforcement subsequent to the program's design and display of reinforcement, and modify the reinforcement reported by the program. There is full flexibility both in the specification of reinforcement, and its editing. The following distinction is important, however:

If you want the given base reinforcement you define to be accounted for by the program when it goes through its code check and design process, you have to define the base reinforcement before the program goes through its design process. The reinforcement that you define, or edit subsequent to the calculation by the program, will appear on your drawings and cause a change in the base reinforcement that you may have defined, but will not go through the code check of the program, unless you reactivate the design process of the program.

To pre-define a base reinforcement prior to the calculation, use:

- The Rebar menu item from the Build pull-down menu; or
- The Build toolbar/Show Rebar toolbar, ; or

- The User Interface menu/Reinforcement toolbar.

To create banded rebar click on Create Banded Rebar tool, . Once you click on it, the program prompts you to draw on the plan the location and length of the reinforcement, using the mouse. Depending on the system of units you selected, the program will display a bar with its default values, such as shown in **Figure 10.6-13**. Once the default rebar is created, you can modify it to the bar of your choice as described below:



FIGURE 10.6-13 CREATION OF DEFAULT REBAR

If you double-click on the rebar shown, its property window (**Fig. 10.6-14**) opens. Using the options provided in the property window of the rebar, you can fully edit the bar selected. Among the many properties, you can edit the number, size, length, cover, and whether the bar is at top or bottom. The program allows you to specify any rebar as base reinforcement. If you select “Base”, the program keeps this rebar in case you re-design the structure. When re-designing, the program would only provide reinforcement that is needed in addition to the base reinforcement. Once you change the bar to “Base” you cannot switch it back to “Calculated”. The bar can be deleted or modified.

If distributed is selected for rebar distribution you have the option to fix the number of bars and/or the spacing as desired. If the number of bars is fixed the spacing and extents will be newly calculated as necessary. The same applies if the spacing is fixed the number of bars and extents will be calculated accordingly. If a specific number of bars and spacing is required and fixed the extents will be adjusted accordingly.

The “Location” tab (**Figure 10.6-15**) allows you to edit the rebar by specifying its coordinates, as opposed to using graphical tools. The “Help” button on the dialog box provides additional guidance.

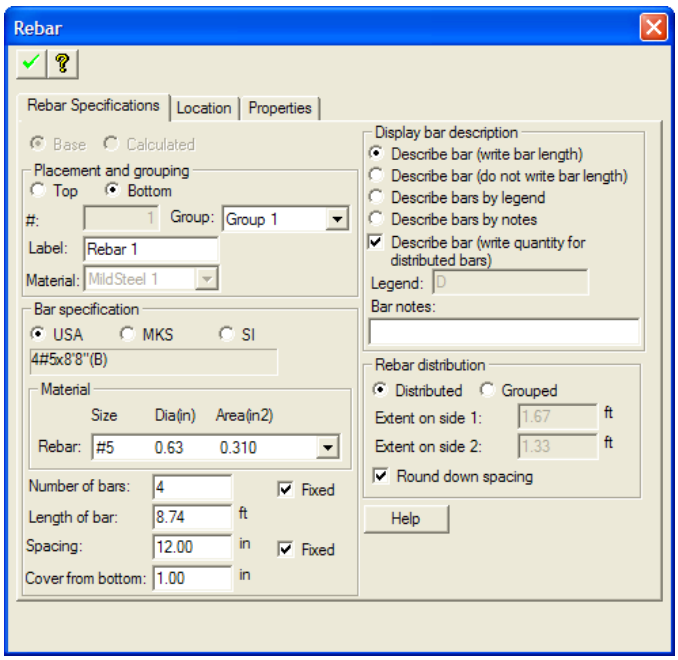


FIGURE 10.6-14 DISTRIBUTED REBAR PROPERTY WINDOW

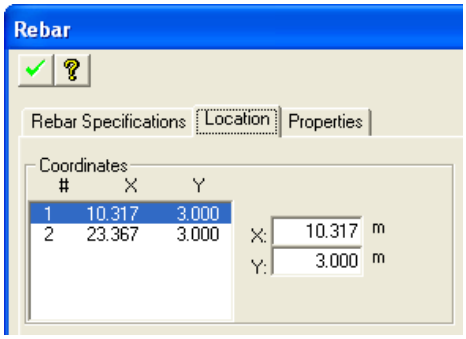


FIGURE 10.6-15 DEFINITION OF REBAR BY COORDINATES

There are several convenient graphical tools that allow you to edit the length, location and display of bar reinforcement.

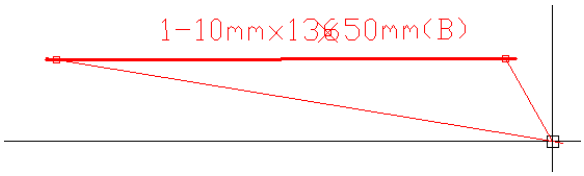
- If you click on a bar (select it) as shown in **Figure 10.6-16(a)**, two small squares, one at each end, appear on it. The distance between each small square and the tip of the bar represents the development length of that bar. You define the development length in the "Criteria" pull-down menu.
- To change the length of a bar graphically, or to rotate it, select one of the squares at its ends and drag it (part **(b)** of the figure).

- To move a bar parallel to itself, select one of the squares, hold down the "Control" key and drag (part **(c)** of the figure).
- To change the extension of distributed bars, select the arrow and drag it to the new location.

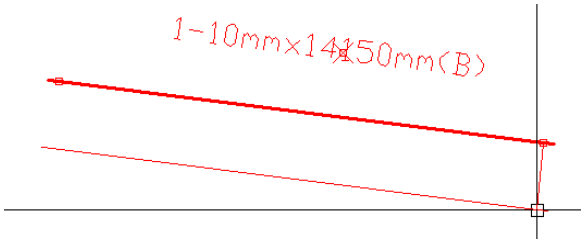
Other editing tools, such as move and copy, that are available for other components can be used for rebar also.



(a) Rebar Activated for Editing



(b) Pick a Square and Drag to Edit



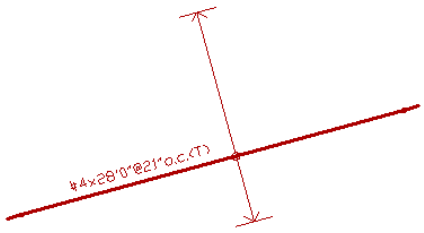
(c) Pick a Square, Drag While "Control" Key is Depressed

FIGURE 10.6-16 MODIFICATION OF BAR REINFORCEMENT LENGTH

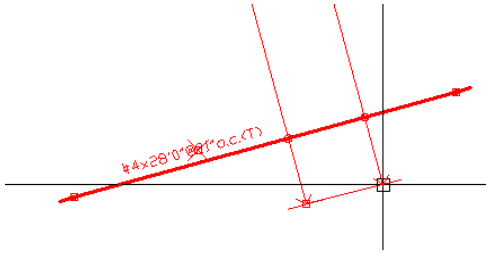
To create distributed rebar, click on the Create Distributed Rebar tool, . The distributed rebar tool is used when a number of identical bars with identical spacing are to be specified over a given area of a slab, such as #4 bars, each 6'0" in length, spaced at 16" on center over a 14-foot width of slab (T12x2300@300 mm o.c. over a distance of 4m).

- Click on the tool and follow the instructions on the command line (bottom of the screen) to complete the display of a group of distributed bars as shown in **Figure 10.6-17(a).**

- You can edit the features of the distributed bars displayed in a similar manner to the single bar (part **(b)** of the figure).
- In addition, you can move the extension line of the bar parallel to itself, by selecting the tip of one of the arrows and dragging it to a new location.
- Using the same procedure as above, you can change the extent of a bar. Once the extent of a bar is changed graphically, the program automatically adjusts the bar spacing, so as to maintain the total area of steel over the selected region.



(a) Display of Distributed Bars



(b) Shift of Extension Line

FIGURE 10.6-17 DISPLAY OF DISTRIBUTED BARS

10.6.2.2 Beam Rebar Specification and Display

To specify beam rebar click on the Beam Rebar tool, , in the Reinforcement toolbar. The Beam Rebar tool opens the dialog window shown in **Fig. 10.6-18**. In this window you can specify the size of the two corner bars at the top, at the bottom, or both at the top and the bottom of the reinforcing cage. The reinforcement calculated by the program will be in addition to the bars you specify. No reinforcement will be reported, if the

bars you select are adequate to meet the design criteria specified.

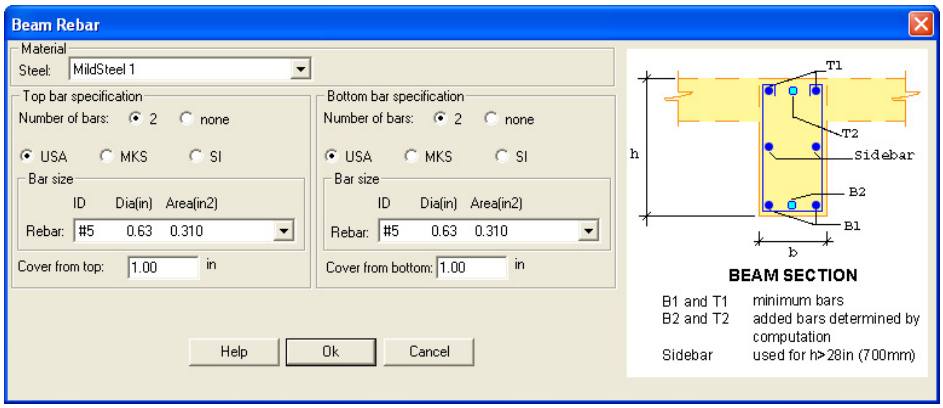


FIGURE 10.6-18 BEAM CORNER BAR SPECIFCATION WINDOW

The Help button provides you with additional information on the treatment of beam corner bars (**Figure 10.6-19**).

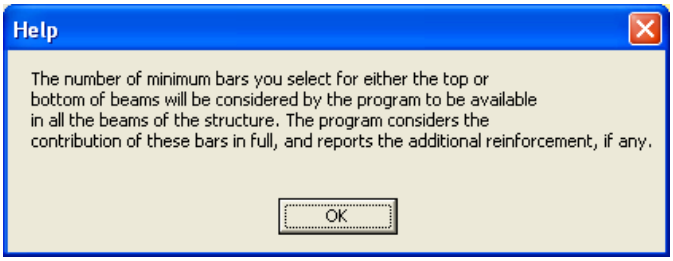


FIGURE 10.6-19 HELP MENU FOR BEAM REBAR

The program shows the required reinforcement of each beam on its elevation. This includes both the post-tensioning tendons and the non-prestressed reinforcement. If there are no tendons, the display will be limited to non-prestressed reinforcement. The elevation also specifies the location and amount of stirrups needed to resist one-way shear.

To see the reinforcement of a beam on its elevation do the following:

- Turn on the support line associated with the beam.
- Select the beam's support line (double-click on it).
- From the support line property box, select the section display tool . This will guide you to display the

beam rebar at the location of your choice. If no rebar is displayed, turn on the "Display/Hide Rebar" tool .

- Once the beam rebar is displayed, you can use the other tools of rebar editing and display available to finalize the beam details.
- If you wish to remove the display, you need to go through the same steps and reactivate the toggle switch "Display/Hide Beam Rebar" tool . Do not use the "Delete" tool, since this will remove the rebar from the design of the structure, as opposed to hiding its display.

Note: Click the Generate Rebar Drawing tool to create a beam reinforcement drawing. Beam reinforcement will not show in plan.

All beams are detailed following a standard layout. The standard layout consists of a template that shows the location, length and shape of the minimum reinforcement in each beam (**Figures 10.6-20** to **10.6-22**). Each beam is reinforced with a minimum of four bars, one at each corner. These bars can be selected by you, as part of the input to the program, prior to the design. The remainder of the bars, if they are required, are calculated and shown on the elevation of each beam.

The template covers the basic conditions of cantilever, end span, and interior spans.

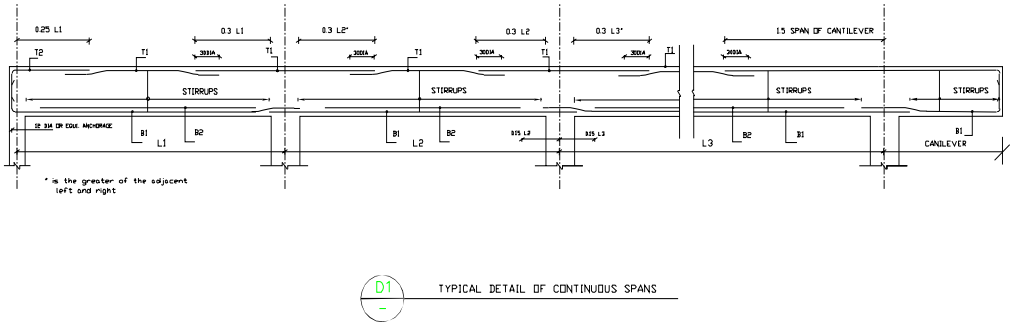


FIGURE 10.6-20 TEMPLATE OF BASIC BEAM REINFORCEMENT

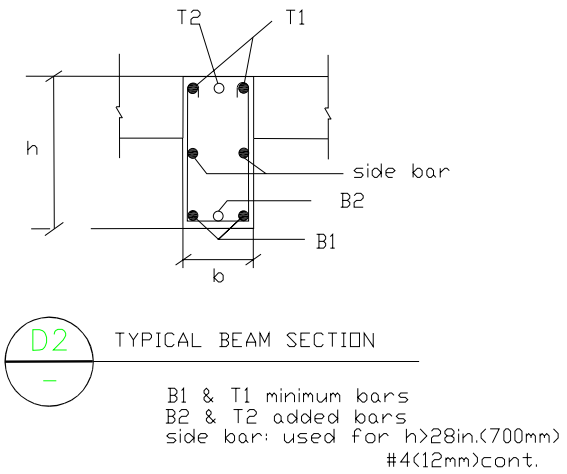


FIGURE 10.6-21 TEMPLATE OF BASIC BEAM CROSS-SECTION

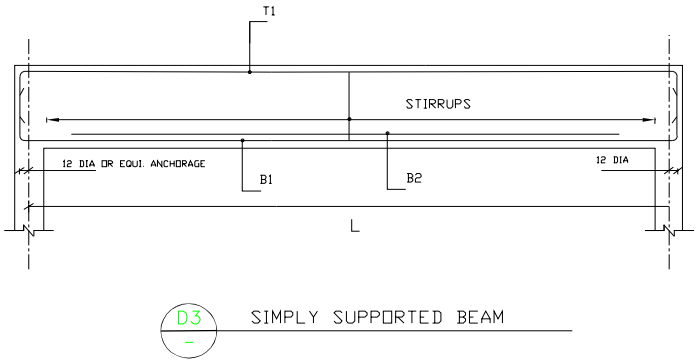


FIGURE 10.6-22 TEMPLATE OF SIMPLY SUPPORTED BEAM REINFORCEMENT

Figure 10.6-23 shows the elevation of a typical multi-span beam. The elevation shows the number, size and length of each added bar. The position of each bar is identified with the distance of its starting or end point from the nearest support centerline.

For shear reinforcement, the program reports number of bars and spacing.

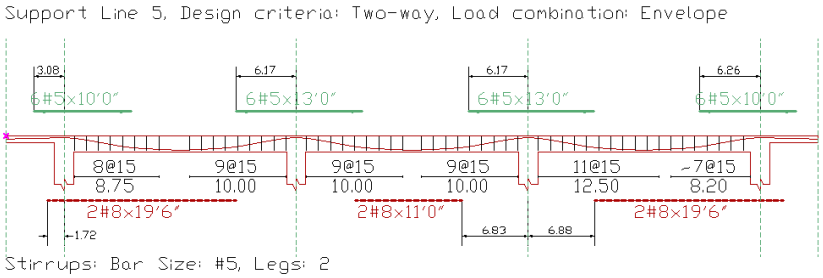


FIGURE 10.6-23 ELEVATION OF A BEAM REINFORCEMENT

To change the text display, use the Settings/Environment/Text option.

10.7 TRANSFER OF POST-TENSIONING DESIGN FROM ADAPT-PT TO ADAPT FLOOR-PRO

10.7.1 Overview

This section explains how you can perform an optimal design for your post-tensioning project using ADAPT-PT, and have the details of your design automatically transferred to ADAPT-Floor Pro. The backward transfer of design information from PT to Floor Pro eliminates the guesswork involved in the tendon and force layout that are prerequisite to the designs performed using finite element programs. The steps involved are:

- Generate a three-dimensional model of your structure in ADAPT-Floor Pro, including the design criteria and loads.
- Define support lines and create design strips in ADAPT-Floor Pro.
- Export the design strips to ADAPT-PT.
- Execute the imported design strips in ADAPT-PT and obtain optimum designs for each.
- Transfer the tendon profile and force obtained in ADAPT-PT to the associated design strip in ADAPT-Floor Pro.
- Validate the imported tendons in ADAPT-Floor Pro.
- Execute ADAPT-Floor Pro using the information obtained and validated from ADAPT-PT.

- Fine-tune the design, if needed.

The procedure is detailed, using the simple example shown in **Fig. 10.7-1**.

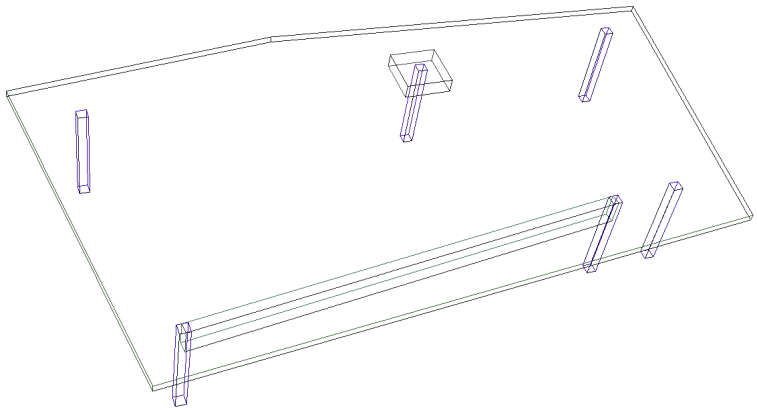


FIGURE 10.7-1 THREE-DIMENSIONAL VIEW OF FLOOR SYSTEM

10.7.2 Export of Data from ADAPT-Floor Pro to ADAPT-PT

Create support lines and design strips using the "Generate Design Strips for Strip Method" from the Strips pull down menu (**Figs. 10.7-2** and **10.7-3**).

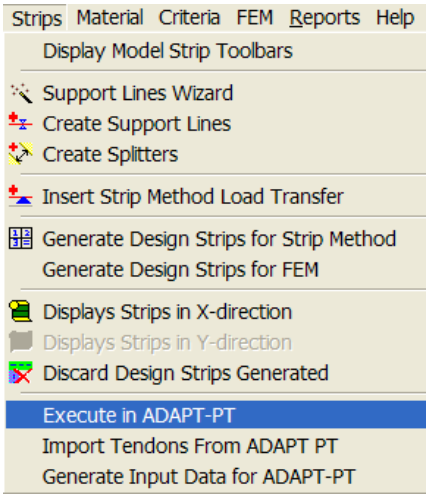
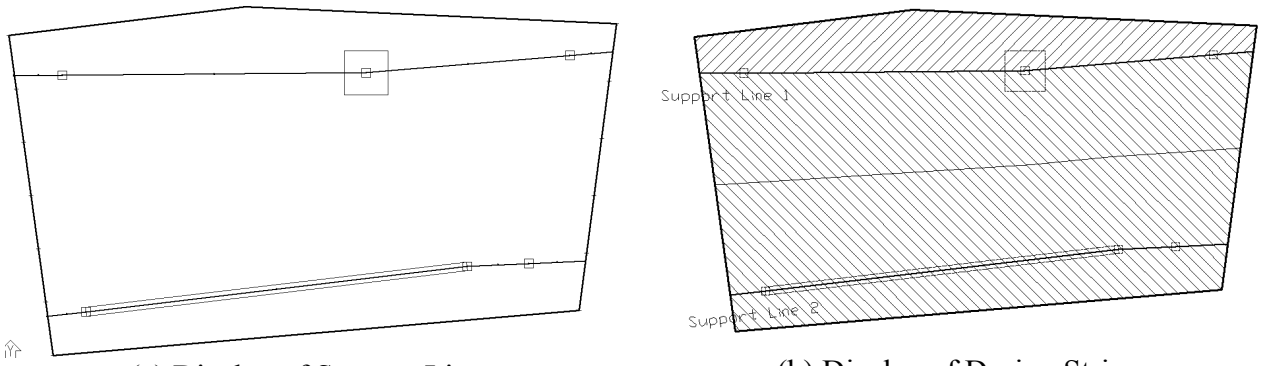


FIGURE 10.7-2 STRIPS PULL-DOWN MENU ITEMS



(a) Display of Support Lines (b) Display of Design Strips

FIGURE 10.7-3 CREATION OF SUPPORT LINES AND DESIGN STRIPS

Once the design strips are successfully created, you can export them one by one to ADAPT-PT for execution. There are two ways you can export a design strip to ADAPT-PT. These are:

Method 1

- Select a support line.
- Click on Execute ADAPT-PT in Strips pull-down menu (**Figure 10.7-2**).

This opens ADAPT-PT with the information associated with the selected support line.

Method 2

- Select a support line.
- Open the property box of the support line (**Figure 10.7-4**).
- Using the 3D viewer tool ( ), obtain an idealized view of the design strip ready to be exported to ADAPT-PT (**Figure 10.7-5**).
- Using the view and editing tools on the right side of the screen, modify the model if need be.
- Once you are satisfied with the idealization of the design strip, click on the PT ( ) tool to open ADAPT-PT with the information from the displayed design strip.

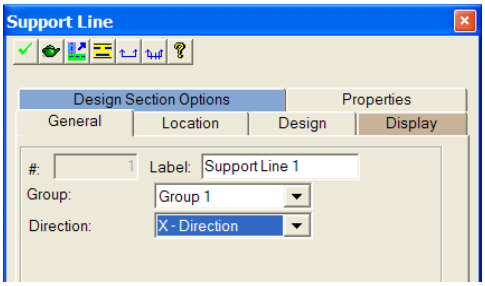


FIGURE 10.7-4 SUPPORT LINE PROPERTY BOX WINDOW

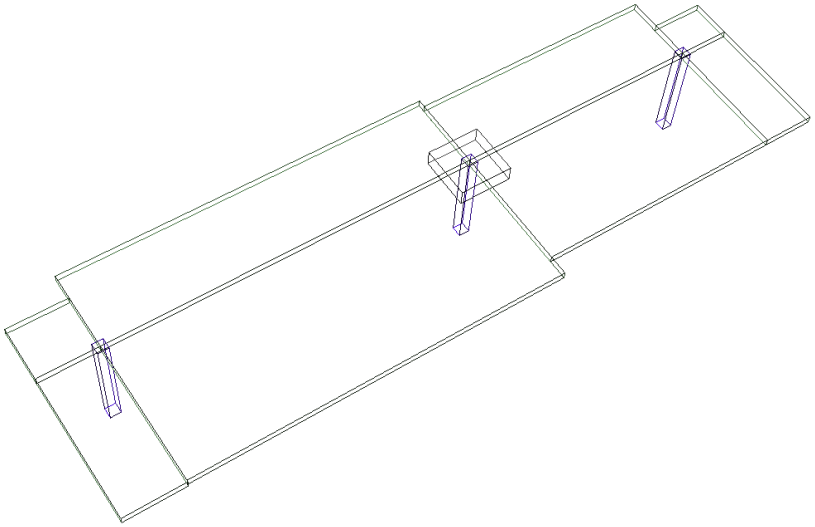


FIGURE 10.7-5 IDEALIZATION OF DESIGN STRIP 1 FOR STRIP ANALYSIS AND DESIGN

The data exported in ADAPT-PT is saved in its own folder under the name "Support Line *name*" in the "STRIP_MODEL/Current_plane" folder. ADAPT-PT recognizes that the loaded data is from ADAPT-Floor Pro and that the outcome of the design is likely to be sent back to ADAPT-Floor Pro.

10.7.3 Execution of Data in ADAPT-PT

Once in ADAPT-PT, the imported data fields will be grayed out, and will not be accessible for editing[4]. Other information necessary for the execution of ADAPT-PT will be filled in with the program's default and is available for editing.

[4] The intent of locking certain data fields is to retain the compatibility of the imported data with its source in ADAPT-Floor Pro.

If necessary, you can unlock the data fields imported from ADAPT-Floor Pro, using the "Unlock" tool shown in **Figure 10.7-6**. Once you unlock the data, it is deemed to have lost its compatibility with ADAPT-Floor Pro and cannot be transferred back to ADAPT-Floor Pro (**Figure 10.7-7**).

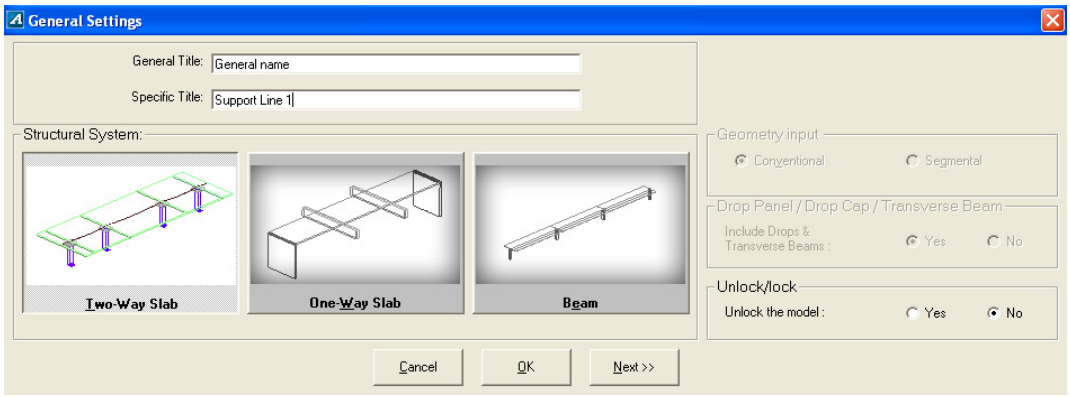


FIGURE 10.7-6 DIALOG WINDOW FROM ADAPT-PT SHOWING THE UNLOCK TOOL

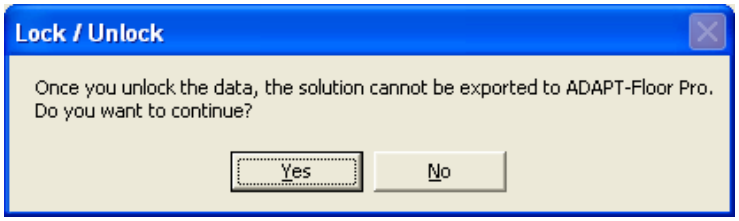


FIGURE 10.7-7 MESSAGE BOX FOR UNLOCKING THE IMPORTED DATA FIELDS

The reference plane used in ADAPT-Floor Pro is generally different from that used in ADAPT-PT. When data is transferred to ADAPT-PT, the program uses a default value of zero (**Figure 10.7-8**). Since it is common to define tendon heights from the soffit of a member, the reference height data field is left unlocked for you to either retain the values imported to ADAPT-PT, or change them to the measurements from the soffit of the member, without losing the ability to export the data to ADAPT-Floor Pro.

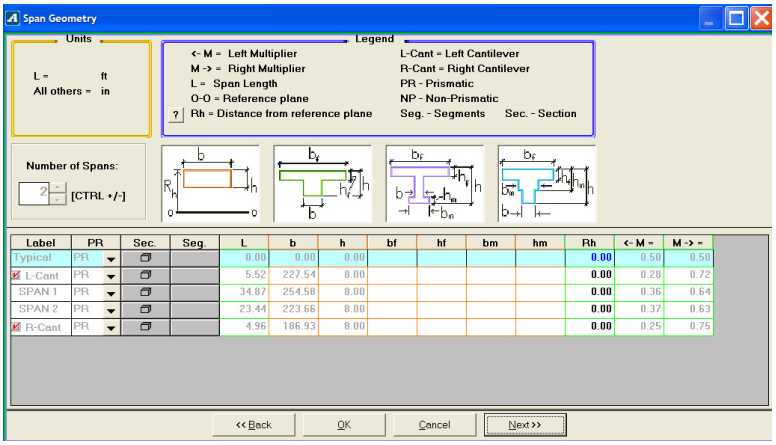


FIGURE 10.7-8 GEOMETRY INPUT SCREEN SHOWING THE INPUT FIELDS FOR REFERENCE HEIGHT

Execution of the program in ADAPT-PT is described in detail in the ADAPT-PT's user manual. Once the design is completed and you exit the program, the message box shown in **Figure 10.7-9** will advise you that your design is available to be imported in ADAPT-Floor Pro. The data for import of design values to ADAPT-Floor Pro is saved under the file name "Support Line *name*.tdn" in the folder "STRIP_MODEL/Current_plane/Support Line *name*". You need not open ADAPT-Floor Pro for processing the data from ADAPT-PT immediately. You can continue with the optimization of the post-tensioning of other design strips, and recall the saved data in ADAPT-Floor Pro when it is convenient.

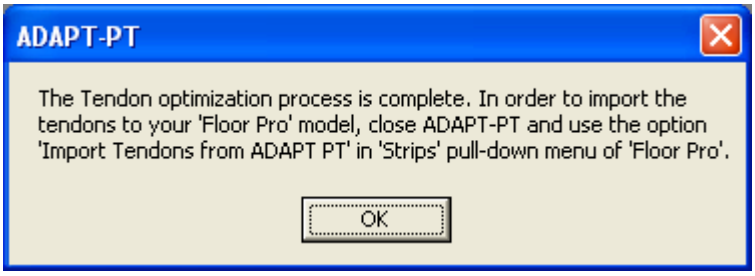


FIGURE 10.7-9 MESSAGE BOX FOR COMPLETION OF DESIGN OPTIMIZATION IN ADAPT-PT

Figure 10.7-10 shows the optimization screen of ADAPT-PT for the design strip 1. The tendon profile and force selected are the minimum to meet the stress requirements of the support line.

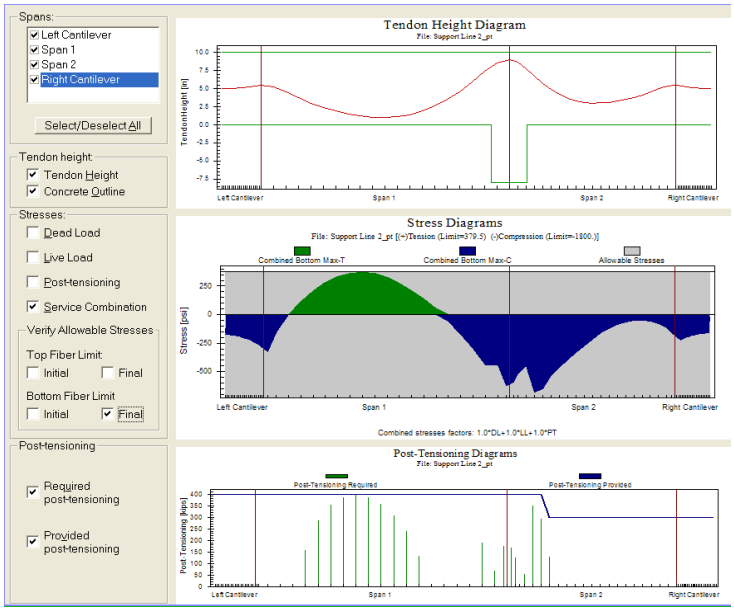
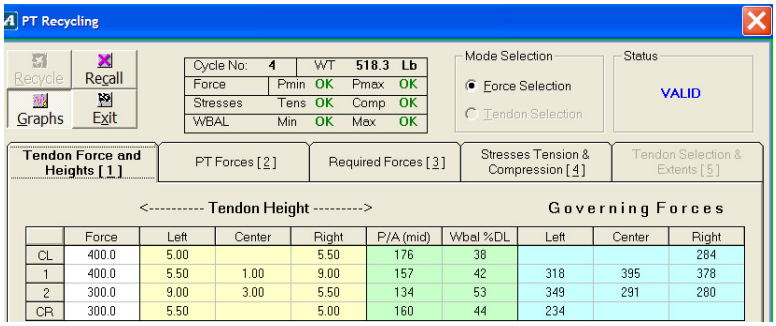


FIGURE 10.7-10 OPTIMIZATION SCREEN OF ADAPT-PT FOR DESIGN STRIP 1

Details of the force selected, the profile of the tendons at the control points, and the status of the design obtained are shown in the tables of **Figure 10.7-11**. The values in the table show the validity of the design obtained for having met the stress, force and balanced loading, and the weight of the post-tensioning used.



	Force	Left	Center	Right	P/A (mid)	Wbal %DL	Left	Center	Right
CL	400.0	5.00		5.50	176	38			284
1	400.0	5.50	1.00	9.00	157	42	318	395	378
2	300.0	9.00	3.00	5.50	134	53	349	291	280
CR	300.0	5.50		5.00	160	44	234		

FIGURE 10.7-11 TABULAR DISPLAY OF OPTIMIZATION OPTAINED IN ADAPT-PT FOR DESIGN STRIP 1

10.7.4 Recall of PT Data from ADAPT-Floor Pro

To recall the optimized tendon design in ADAPT-Floor Pro, do the following:

- Open the file of the floor system and display the support lines.
- Select the support line for which you intend to import the data. You may select more than one support line, if you wish to import data for multiple design strips.
- From the Strips pull-down menu (**Figure 10.7-2**) select "Import Tendons From ADAPT-PT." The program reads the data prepared by ADAPT-PT and displays the tendons overlapped on the support line. As an example, the data imported for design strip 1 is shown in **Figure 10.7-12**. The design obtained in ADAPT-PT consisted of a continuous tendon over the entire length of the support line and a second tendon that covered the left cantilever, span 1 and is anchored beyond the second support in span 2. The short tendons terminating in span 2 are identified with a handle (short sloping line to the right of the second column). The handle is a visual display intended to (a) indicate the location of terminated tendons, and (b) provide a graphic that you can select for editing the associated tendons.

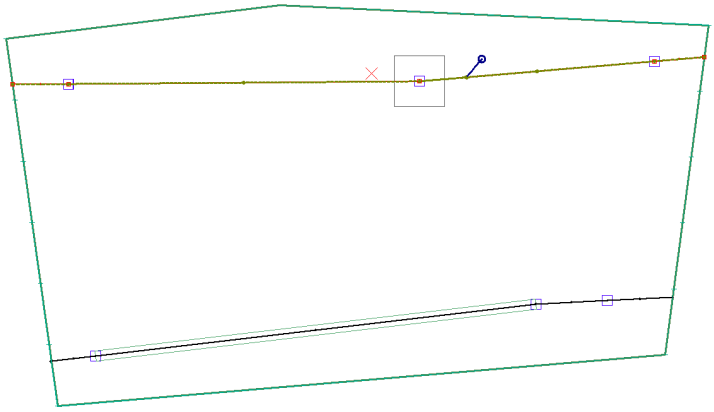


FIGURE 10.7-12 DISPLAY OF IMPORTED TENDONS FOR SUPPORT LINE 1

- While being imported from ADAPT-PT, the program displays the message box of **Figure 10.7-13**, and it overwrites any previously imported tendons from ADAPT-PT for the same design strip that you did not validate to be retained.
- The imported tendons are initialized with the "Disregard" status. They remain disregarded until you validate them.

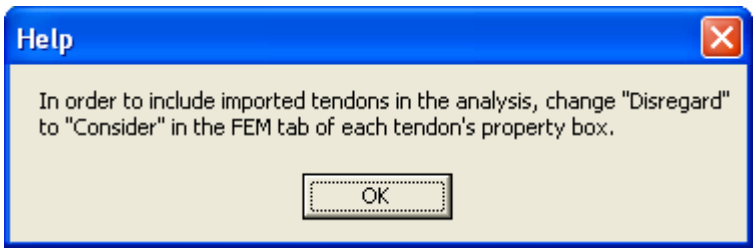
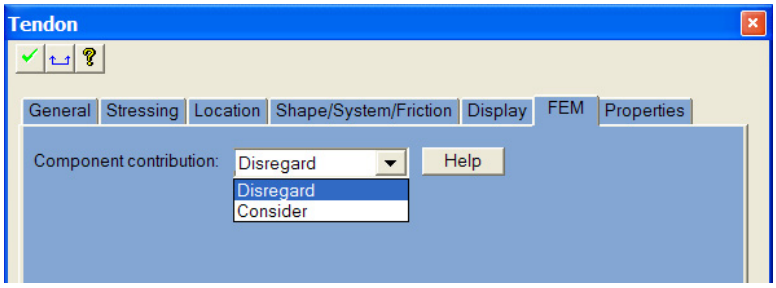


FIGURE 10.7-13 TENDON STATUS MESSAGE BOX

10.7.5 Validation of PT Data in ADAPT-Floor Pro

- The imported tendons will be initially given a "Disregard" status by the program. That is, they will be displayed in the model; you can edit them, but they will not be considered in the analysis and design, unless you change their status to "Consider." You can change the status of a tendon in the FEM tab of the tendon property box (**Figure 10.7-14**)
- If (a) there are any imported tendons from a previous ADAPT-PT execution for the same design strip, and (b) their status is not yet changed to "Consider" by you, the latest import of design data will overwrite the existing information. In other words, at any given stage, the tendons displayed in ADAPT-Floor Pro are either from the latest ADAPT-PT import, or are those you have viewed and determined to be retained and considered in your design.



FIUGRE 10.7-14 TENDON PROPERTY WINDOW AND REGARD/DISREGARD OPTION

- To validate a tendon, you can open its property window and view its particulars. A powerful tool to view the tendon in elevation is the elevation viewer tool ( ) from the tendons property window (**Figure 10.7-14**). **Figure 10.7-15** shows the elevation of the continuous tendon of support line 1.

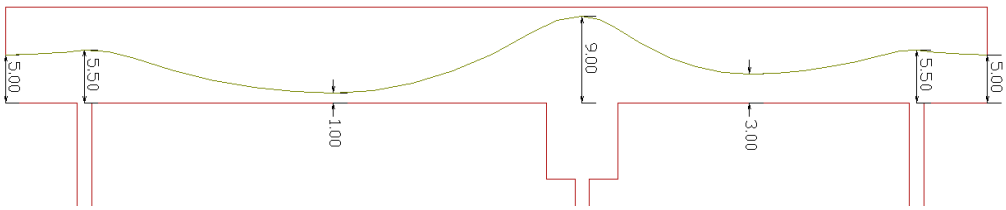


FIGURE 10.7-15 ELEVATION OF CONTINUOUS TENDON OF DESIGN STRIP 1

- **Figures 10.7-16** and **10.7-17** show the view of the idealized design strip 2 and the optimum tendon profile obtained from ADAPT-PT.

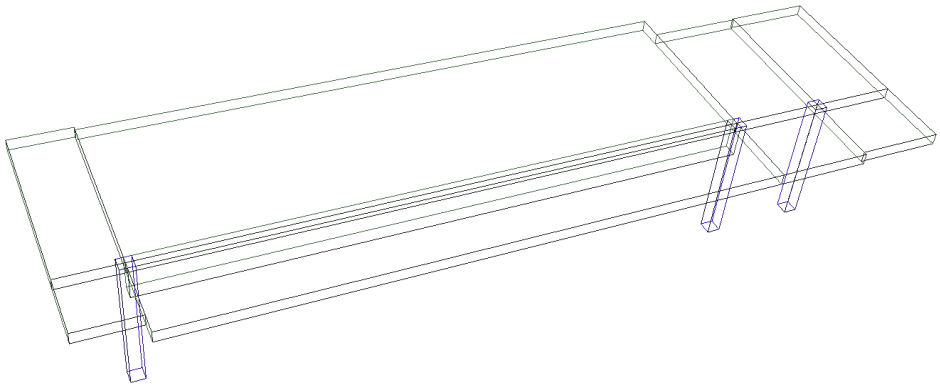


FIGURE 10.7-16 IDEALIZATION OF SUPPORT LINE 2

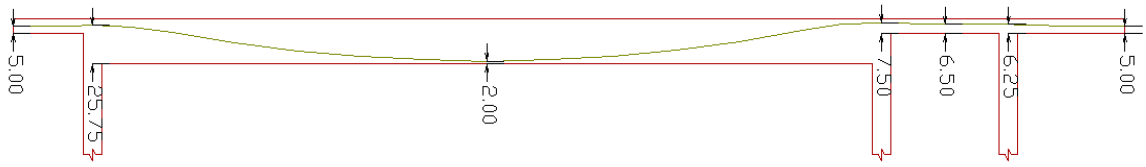


FIGURE 10.7-17 TENDON PROFILE FROM SUPPORT LINE 2

Chapter 11

REPORTS

11.1 OVERVIEW 197
11.2 REPORT COMPONENTS 197
11.2.1 Default Reports of the Program 197
11.2.2 Program/User Generated Reports 197
11.2.3 Text-Based Information from Other Sources (External-Text) 198
11.2.4 Graphics-Based Information from Other Sources (External-Graphics) 198
11.3 REPORT GENERATION 198
11.3.1 Single Default Reports 198
11.3.2 Compiled Reports 200
11.4 REPORT GENERATION UTILITIES 203
11.4.1 Report Font 203
11.4.2 External-Text 204
11.4.3 External-Graphics 204
11.4.4 Program-User Report Components 204
11.5 LIST OF THE PROGRAM GENERATED REPORTS 204
11.6 SELECTED SAMPLE OF PROGRAM-S GENERATED REPORTS 206

11.1 OVERVIEW

ADAPT-Floor Pro's report-generation capability enables users to create highly professional and targeted reports. Intuitive and highly versatile, the report-generation features make it ease for you to create detailed documentation on all aspects of your work, from basic structural layout all the way to quantities and fabrication drawings. These reports are ideal for a variety of uses:

- For submittal to building officials and plan check
- For validation of the program, verification, post-processing, documentation
- For creations of structural documents
    - Structural drawings
        - Rebar drawings
        - Post-Tensioning drawings
    - Fabrication (shop) drawings
    - Cost estimating, and quantities

11.2 REPORT COMPONENTS

11.2.1 Default Reports of the Program

ADAPT-Floor Pro automatically generates a comprehensive list of pre-programmed reports, including:

- Stress check results
- Column reactions
- Live load plan

The default reports are broken down into tabular and graphical groups

11.2.2 Program/User Generated Reports

During the creation of data, execution of the program and the design process, you have the option to capture the image on the computer screen, crop it, annotate it, edit it, save it, and include it in your final report. These

are referred to as "Program/User" generated report components. Once created, they remain available to you to edit, delete and save at your current session or subsequent sessions.

11.2.3 Text-Based Information from Other Sources (External-Text)

The program can import a text-based file and include it in your report at the location of your choice. The text file can be of any length. Once imported, it can be viewed and edited prior to compilation and printing of the report. The program refers to imported text as "External-Text."

11.2.4 Graphics-Based Information from Other Sources (External-Graphics)

Graphics stored in bitmap format can also be imported by the program, viewed, edited, and included in the report.

11.3 REPORT GENERATION

There are two options in report generation: (i) you can generate a "Single Report" for the purpose of viewing or printing a given set of information, or (ii) you can create "Compiled Reports" that are made up of more than one report component. Again, you can view, edit, save or discard either type. The compiled reports are made up of a grouping of selected single reports.

Report generation is initiated from the pull-down menu *Reports*, which has three sub-menus (**Fig.11.3-1**)



FIGURE 11.3-1 REPORTS SUBMENUS

11.3.1 Single Default Reports

Through a number of side menus, you can access the entire program generated single reports. These are grouped in four categories (**Fig. 11.3-2**).

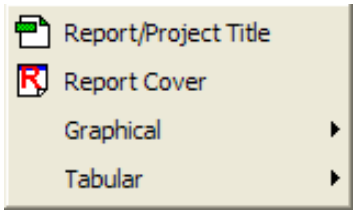


FIGURE 11.3-2 GROUPING OF SINGLE REPORTS

The following is the list of single reports:

- Report/Project Title
- Report Cover
- Graphical
  - Plan Geometry
  - Tendon Plan
  - Rebar Plan
  - Loading
    - Dead Load
    - Live Load
    - User Defined Load Cases
  - Design Plans
    - Support Lines in X-direction
    - Support Lines in Y-direction
    - Design Strips in X-direction
    - Design Strips in Y-direction
    - Stress Check in X-direction
    - Stress Check in Y-direction
  - Mesh Plan
- Tabular
  - Structural Geometry
    - Slab Region
    - Openings
    - Beams
    - Columns
    - Drop Caps/Panels
    - Walls
    - Tendons
    - Springs
    - Releases and Restraints
  - Material and Design Criteria
  - Analysis Data
    - Equilibrium Check
    - Solution Statistics
    - Nodes
    - Elements

- Load Cases and Combinations
- Design Section Data
  - Design Section Forces Tabular Report
  - Design Section Rebar Tabular Report
  - Design Section Moment Capacities
  - Design Section Dimension
  - Design Section Geometry
- Column Forces
  - Upper Columns
  - Lower Columns
- Wall Reactions
  - Upper Walls
  - Lower Walls
- Other Support Reactions
  - Point Support Reactions
  - Line Support Reactions
  - Point Spring Reactions
  - Line Spring Reactions
- Punching Shear Design
  - Punching Shear Stress Check Results
  - Punching Shear Stress Check Parameters
  - Punching Shear Reinforcement
- Skip Pattern
- Quantity and Cost
- Tendon Totals
- Applied Loads
- Rebar Totals
- Rebar Summary
- Rebar Legend

11.3.2 Compiled Reports

Compiled reports are created through *Report Generation Manager* (RGM) (**Fig. 11.3-3**). The following four functions/features of the *Report Generation Manager* are critical to its effective use.

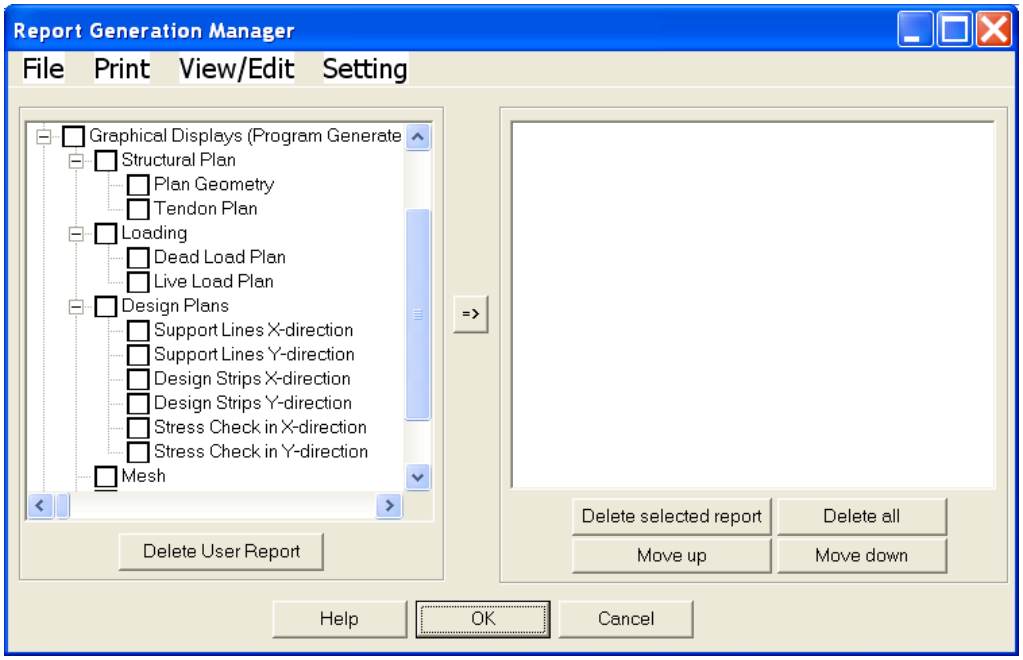


FIGURE 11.3-3 REPORT GENERATION MANAGER (RGM) DIALOG BOX

- RESOURCES. The Report Generation Manager uses the resources available to it, and your instructions to compose a report. The resources available to the RGM are:

    - **Default Reports of the Program.** You can access the program-generated default reports through the check boxes on the left side of the RGM dialog window. These include both the tabular and graphical reports. The default reports include all that is listed under single reports in 11.3.1 plus a "Content List."
    - **Program/User Generated Reports**. Any screens captured by you during the course of data generation and execution will all be listed in the RGM for your view and possible inclusion in the finished report. **Figure 11.3-4** shows an example for a list containing three user-generated screens from the program.

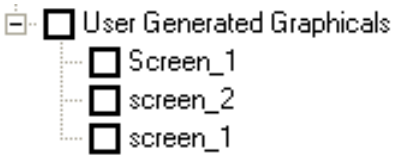


FIGURE 11.3-4 EXAMPLE FOR A LIST OF USER-GENERATED SCREENS

- **External Text.** Text files you have imported for possible inclusion in your report will be listed in RGM as shown in **Fig. 11.3-5.**



FIGURE 11.3-5 EXAMPLE FOR A LIST OF IMPORTED EXTERNAL-TEXT FILES

- **External Graphics.** The graphics that you have imported for possible inclusion in your report will be listed under in the RGM as illustrated in **Fig. 11.3-6.**

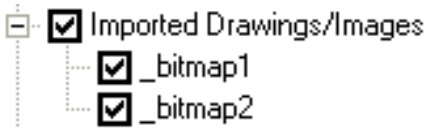


FIGURE 11.3-6 EXAMPLE FOR A LIST OF IMPORTED EXTERNAL-DRAWINGS/IMAGES

- COMPOSITION. Reports are composed as follows:
  - **Select Report Components**. You select the report parts to be included in the compilation by (i) checking the box in front of them, and (ii) shifting them to the window on the right side of the RGM, using the arrow "=>" between the two windows.
  - **Arrange the Report Components**. Arrange the sequence of the report parts with the following tools:
    - Move up: shifts the selected report component toward the beginning of the report
    - Move down: shifts the selected report component toward the end of the report
    - *Delete selected reports*: removes the report components marked from the compiled report. If a report component is removed accidentally, you can always go back to the list of report parts in the left window of the RGM and add them back to your compiled report.

- VIEW/EDIT. You can view and edit any of the report components you have included in the compiled report as follows:
    - Select the report component by placing the cursor on it and left clicking the mouse to highlight the item;
    - Click on the View/Edit menu item at the top of the RGM.

  This will open the report component for your view. At this time, you have the option of editing the displayed report component. If you save the edited report component, it will be included in your final report in its edited format. However, if you decide to restore the report to its original un-edited format, you can do so. Simply select "Regenerate Reports" from the "File" pull down menu of the RGM screen.

- PRINT PREVIEW. The print preview displays the report in three parts as follows:
    - Content List
    - Selected Graphical Reports. This includes all the graphics-based report components

  The sequence of the printed report components is given in the "Content List" In print preview, however, these are grouped into tabular and graphical parts.
- PRINT. You can create either a hard copy or electronic copy of your compiled report.
    - For a hard copy, simply click on the "Print" tool. The program sends a copy of the report to the default printer of your computer.
    - Electronic copies of the compiled report can be generated by selecting the "Acrobat Distiller" from the list of printers. This will create a PDF file. The individual report components can be printed using other printers.

11.4 REPORT GENERATION UTILITIES

The principal tools for creating reports tailored to your needs are described below:

11.4.1 Report Font

For the graphical components of your report, you can select the font shape and size. To do so, click on the *Settings* pull down menu of the RGM and select *Graphical Report Font Size*. Follow the instructions on the screen.

11.4.2 External-Text

To add an existing text file to your compiled report, select *Import Text File* from the *File* menu of the RGM and follow the instructions prompted on the computer screen.

11.4.3 External-Graphics

To add an existing graphics file saved in bitmap format to your compiled report, select *Import Bitmap* from the *File* menu of the RGM and follow the instructions prompted on the computer screen.

11.4.4 Program-User Report Components

To save a display of the program during data generation, analysis or design, do the following:

- Compose the display on the computer screen as you want it to appear in your report;
- From the *Report* pull down menu, select *Save Current Screen*.

The program captures the screen and saves it in a layer with the label "Report_*name*," where "*name*" is the label you will give to the captured screen. The layer generated will be turned off, in order not to interfere with the original flow of your work. At any given time, however, you can go to the list of layers from the main screen's pull down menu, turn all layers off, except the layer you generated from the screen capture. You can view and edit the information on this layer for possible inclusion in your report.

11.5 LIST OF THE PROGRAM GENERATED REPORTS

110 STRUCTURAL COMPONENTS
110.10 SLAB REGIONS (Summary Report)
110.15 SLAB REGIONS (Detailed Report)
110.35 OPENINGS (Detailed Report)
110.40 BEAMS
110.50 COLUMNS
110.55 DROP CAPS/PANELS
110.60 WALLS

111 PRESTRESSING
111.10 TENDON MATERIAL
111.20 TENDON GROUPS GEOMETRY, SYSTEM AND FRICTION

111.30 INDIVIDUAL TENDON'S GEOMETRY, SYSTEM AND FRICTION
111.50 TENDON ELONGATION, BY GROUPS
111.60 TENDON ELONGATION, INDIVIDUAL TENDONS

112 RELEASES AND RESTRAINTS
112.20 POINT SUPPORT LOCATION AND RESTRAINT
112.40 LINE SUPPORT LOCATIONS AND RESTRAINT

113 OTHER SUPPORTS
113.10 POINT SPRINGS (Summary Report)
113.15 POINT SPRINGS (Detailed Report)
113.35 AREA SPRINGS/SOIL SUPPORTS (Detailed Report)

117 MATERIALS
117.20 CONCRETE MATERIAL PROPERTIES
117.40 REINFORCEMENT (NONPRESTRESSED) MATERIAL PROPERTIES
117.60 PRESTRESSING MATERIAL PROPERTIES

120 APPLIED LOADS
120.10 DEAD LOAD
120.20 LIVE LOAD
120.30 LATERAL

132 ANALYSIS DATA
132.10 EQUILIBRIUM CHECK
132.20 SOLUTION STATISTICS
132.30 NODES
132.40 ELEMENTS

142 CODES AND ASSUMPTIONS
142.10 DESIGN CODE SPECIFIED
142.20 MATERIAL AND STRENGTH REDUCTION FACTORS
142.30 COVER TO REINFORCEMENT
142.40 MINIMUM BAR LENGTH

143 DESIGN CRITERIA
143.10 ONE-WAY SLABS
143.20 TWO-WAY SLABS
143.30 BEAM

146 LOAD CASES AND COMBINATIONS
146.20 LOAD CASES
146.30 SKIP PATTERN
146.40 LOAD COMBINATIONS

152 DESIGN SECTION GEOMETRY
152.20 AUTOMATICALLY GENERATED DESIGN SECTIONS
152.40 MANUALLY GENERATED DESIGN SECTIONS

153 DESIGN SECTION DIMENSION
153.20 AUTOMATICALLY GENERATED DESIGN SECTIONS
152.40 MANUALLY GENERATED DESIGN SECTIONS

154 DESIGN SECTION ACTIONS
154.20 DESIGN ACTIONS FOR AUTOMATICALLY GENERATED SECTIONS
154.40 DESIGN ACTIONS FOR MANUALLY GENERATED DESIGN SECTIONS

156 DESIGN SECTION REBAR
156.20 DESIGN SECTION REBAR FOR AUTOMATICALLY GENERATED SECTIONS
156.40 DESIGN SECTION REBAR FOR MANUALLY GENERATED SECTIONS

158 DESIGN SECTION CAPACITY
158.20 DESIGN SECTION CAPACITIES BASED ON REINFORCEMENT REPORTED IN THE PROGRAM
158.40 DESIGN SECTION CAPACITIES FOR MANUALLY GENERATED

SECTIONS

160 COLUMNS

160.20 LOWER COLUMNS

160.40 UPPER COLUMNS

170 WALLS

170.20 LOWER WALLS

170.40 UPPER WALLS

180 PUNCHING SHEAR DESIGN

180.60 PUNCHING SHEAR STRESS CHECK PARAMETERS

180.40 PUNCHING SHEAR STRESS CHECK RESULTS

199.00 REBAR LEGEND

200.00 REBAR LIST

210.00 REBAR TOTALS

202 Point Support Reactions

204 Line Support Reactions

11.6 SELECTED SAMPLE OF PROGRAM-S GENERATED REPORTS

110 STRUCTURAL COMPONENTS

110.10 SLAB REGIONS (Summary Report)

ID:1 Label: Slab Region 1 Material: Concrete 1 Thickness: 200 mm Offset: 0 mm

Vertex	X	Y	Z	Comments
	m	m	m	
1	-7.688	3.209	3.000	
2	6.210	0.608	3.000	
3	16.207	1.649	3.000	

110.15 SLAB REGIONS (Detailed Report)

ID	Label	Thickness	Offset	Material	Comments
		mm	mm		
1	Slab Region 1	200	0	Concrete 1	

110.35 OPENINGS (Detailed Report)

Opening:1 Label: Opening 1 Slab region IDs: 1

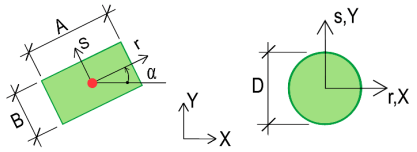
Vertex	X	Y	Z	Comments
	ft	ft	ft	
1	24.00	23.00	10.00	
2	28.00	23.00	10.00	
3	28.00	21.00	10.00	
4	24.00	21.00	10.00	

110.40 BEAMS

ID	Label	First (X,Y); Second (X,Y)	Width	Depth	Offset	Material ID
		ft	in	in	in	
1	Beam 1	0.52,31.54; 56.43,31.54	12.00	22.00	0.00	Concrete 1

110.50 COLUMNS

COLUMN SECTION

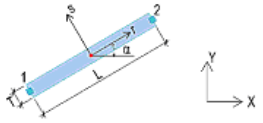


Column Dimensions and Material Property

Label	#	Centroid X,Y,Alpha	A or D	B	Length	f'c
		ft ft °	in	in	ft	psi
Column 1	1	9.50,10.00,0.00	12.00	24.00	10.00	4000

110.60 WALLS

WALL SECTION



Wall Dimensions and Material Property

Label	#	Centroid X,Y,Alpha	T	L	Height	f'c
		ft ft °	in	ft	ft	psi
Wall 1	1	28.74,10.41,0.00	8.00	7.85	10.00	4000

111 PRESTRESSING

111.10 TENDON MATERIAL
Material Label: Prestressing 1
fpu = 270.000ksi fpy = 240.000ksi Eps = 28000ksi

111.20 TENDON GROUPS GEOMETRY, SYSTEM AND FRICTION
Material Label: Prestressing 1

Group	Span	Span Length	Shape	Start (from bottom)	Center (from bottom)	Right (from bottom)	Code	System	K *0.001	Meu *0.01
		ft		in	in	in			rad/ft	
Red	1	9.50	7	4.00	---	7.00	-1	U	1.00	7.00
	2	38.00	1	7.00	1.00	7.00	-1	U	1.00	7.00
	3	9.50	7	7.00	---	4.00	-1	U	1.00	7.00
	Total	57.00								
Blue	1	10.00	7	7.25	---	7.25	-1	U	4.00	20.00
	2	40.00	1	7.25	0.75	7.25	-1	U	4.00	20.00
	3	10.00	7	7.25	---	7.25	-1	U	4.00	20.00
	Total	60.00								

TENDON GROUPING, IDENTIFICATION AND STRESSING
GROUP: Red Number of tendons: 34 Material ID: Prestressing 1
Tendon length(ft) = 57.00 Strands area (in2): 0.15
Stressing: first end Seating (in): 0.25
Elongation (in): first end: 5.15 second end: 0.00 Total: 5.15

Tendon ID	Number of Strands
1	1
2	1
3	1

GROUP: Blue Number of tendons: 4 Material ID: Prestressing 1

Tendon length(ft) = 60.00 Strands area (in2): 0.15
Stressing: both ends Seating (in): 0.25
Elongation (in): first end: 4.90 second end: 0.11 Total: 5.01

Tendon ID	Number of Strands
35	1
36	1
37	1
38	1

Legend

Shape
1-----Reversed Parabola
2-----Partial Parabola
3-----Harped
4-----Cantilever Up
5-----Cantilever Partial
6-----Cantilever Harped
7-----Cantilever Down
System
U-----Unbonded
B-----Bonded
Code
First-----Stressed at first end
Last------Stressed at last end
Both------Stressed at both ends

111.30 INDIVIDUAL TENDON'S GEOMETRY, SYSTEM AND FRICTION
Material Label: Prestressing 1

Tendon ID	Label	Span	Span Length	Shape	Start (from bottom)	Center (from bottom)	Right (from bottom)	Code	System	K *0.001	Meu *0.01
			ft		in	in	in			rad/ft	
1	Tendon 1	1	9.50	7	4.00	---	7.00	-1	U	1.00	7.00
		2	38.00	1	7.00	1.00	7.00	-1	U	1.00	7.00
		3	9.50	7	7.00	---	4.00	-1	U	1.00	7.00
		Total	57.00								
2	Tendon 2	1	9.50	7	4.00	---	7.00	-1	U	1.00	7.00
		2	38.00	1	7.00	1.00	7.00	-1	U	1.00	7.00
		3	9.50	7	7.00	---	4.00	-1	U	1.00	7.00
		Total	57.00								
3	Tendon 3	1	9.50	7	4.00	---	7.00	-1	U	1.00	7.00
		2	38.00	1	7.00	1.00	7.00	-1	U	1.00	7.00
		3	9.50	7	7.00	---	4.00	-1	U	1.00	7.00
		Total	57.00								

TENDON GROUPING, IDENTIFICATION AND STRESSING

Tendon: Tendon 1 Material ID: Prestressing 1
Tendon length(ft) = 57.00 Strands/tendon: 1 Strands area (in2): 0.15
Long-term stress loss specified: 11.000ksi
Stressing: first end

First end	Second end
Ratio of jacking force:0.85	---
Seating loss: 0.25	---

Tendon: Tendon 2 Material ID: Prestressing 1
Tendon length(ft) = 57.00 Strands/tendon: 1 Strands area (in2): 0.15
Long-term stress loss specified: 11.000ksi
Stressing: first end

First end	Second end
Ratio of jacking force:0.85	---
Seating loss: 0.25	---

Tendon: Tendon 3 Material ID: Prestressing 1
Tendon length(ft) = 57.00 Strands/tendon: 1 Strands area (in2): 0.15
Long-term stress loss specified: 11.000ksi
Stressing: first end

First end	Second end
Ratio of jacking force:0.85	---
Seating loss: 0.25	---

Legend

Shape
1-----Reversed Parabola
2-----Partial Parabola
3-----Harped
4-----Cantilever Up
5-----Cantilever Partial
6-----Cantilever Harped
7-----Cantilever Down

System
U-----Unbonded
B-----Bonded

Code
First-----Stressed at first end
Last------Stressed at last end
Both------Stressed at both ends

111.50 TENDON ELONGATION, BY GROUPS

Tendon (Group Label)	Jack(1st,2nd)	Seating(1st,2nd)	Elongation (1st,2nd,Total)
	k	in	In
Red	35.114,N/A	0.25,N/A	5.15,N/A, 5.15
Blue	35.114,35.114	0.25,0.25	4.90,0.11,5.01

111.60 TENDON ELONGATION, INDIVIDUAL TENDONS

Tendon(#,Label)	Jack(1st,2nd)	Seating(1st,2nd)	Elongation (1st,2nd,Total)
	k	in	In
1,Tendon 1	35.114,N/A	0.25,N/A	5.15,N/A, 5.15
2,Tendon 2	35.114,N/A	0.25,N/A	5.10,N/A, 5.10
3,Tendon 3	35.114,N/A	0.25,N/A	5.15,N/A, 5.15

112 RELEASES AND RESTRAINTS

112.20 POINT SUPPORT LOCATION AND RESTRAINT

|--------Location-------|--Restraint: Translation---|---Restraint: Rotation-----|

ID	Label	X, Y, Z (m)	X	Y	Z	XX	YY	ZZ
1	Point Support 1	-5.000;12.000;0.000	No	No	Yes	Yes	Yes	Yes
2	Point Support 2	5.018;10.073;0.000	No	No	Yes	Yes	Yes	Yes
3	Point Support 3	14.000;11.000;0.000	No	No	Yes	Yes	Yes	Yes
4	Point Support 4	14.993;2.978;0.000	No	No	Yes	Yes	Yes	Yes

112.40 LINE SUPPORT LOCATIONS AND RESTRAINT

Line Supports Locations

Label	#	Length	Centroid X;Y;Z	Start X;Y;Z	End X;Y;Z
		ft	ft	ft	ft
Line Support 5	5	7.85	28.74;10.41;0.00	24.81;10.41;0.00	32.66;10.41;0.00

Line Supports Releases |----Release: Translation---|------Release: Rotation-----|

Label	#	X	Y	Z	XX	YY	ZZ
Line Support 5	5	No	No	Yes	Yes	Yes	Yes

113 OTHER SUPPORTS

113.10 POINT SPRINGS (Summary Report)

ID	Label	Location	Offset	Kz	Type
		m	mm	kN/mm	
1	Point Spring 1	6.308,11.332,3.000	0	1.00E-006	T/C
2	Point Spring 2	6.308,11.332,3.000	0	1.00E-006	T/C

Spring Type:
T = Tension only
C = Compression only
T/C = Tension and compression

113.15 POINT SPRINGS (Detailed Report)

ID	Label	Location	Offset	Kx	Ky	Kz	Kxx	Kyy	Kzz
		m	mm	kN/mm	kN/mm	kN/mm 1E-6			
1	Point Spring 1	6.308,11.332,3.000	0	0.00	0.00	1.00	0.00	0.00	0.00
2	Point Spring 2	6.308,11.332,3.000	0	0.00	0.00	1.00	0.00	0.00	0.00

Units of Kxx, Kyy, Kzz are kN-m/rad

113.35 AREA SPRINGS/SOIL SUPPORTS (Detailed Report)

Area spring/soil support:1 Label:Area Spring 1 Stiffness:2.72E-2 N/mm3

Vertex	X	Y	Z	Comments
	m	m	m	
1	9.003	8.771	3.000	
2	12.232	10.807	3.000	
3	12.382	5.980	3.000	
4	10.032	5.892	3.000	

117 MATERIALS

117.20 CONCRETE MATERIAL PROPERTIES

ID	Label	f'c	Weight	Type	Ec	Creep coefficient
		psi	pcf		ksi	
1	Concrete 1	4000	150.00	Normal	3600	2.00

f'c = strength at 28 days
Ec = modulus of elasticity at 28 days

117.40 REINFORCEMENT (NONPRESTRESSED) MATERIAL PROPERTIES

ID	Label	f'y	fvy	Es
		ksi	ksi	Ksi
1	MildSteel 1	60.000	60.000	30000

f'y = yield stress of longitudinal reinforcement
fvy = yield stress of one-way shear reinforcement
Es = modulus of elasticity

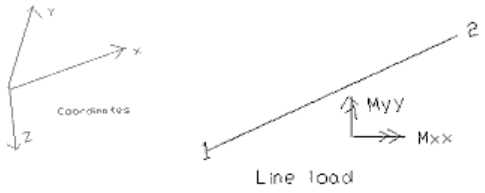
117.60 PRESTRESSING MATERIAL PROPERTIES

ID	Label	fpu	fpy	Fps
		ksi	ksi	Ksi
1	Prestressing 1	270.000	240.000	28000

fpu = ultimate stress
fpy = yield stress
Fps = modulus of elasticity

120 APPLIED LOADS

120.10 APPLIED LOADS



120.20 DEAD LOAD

Concentrated Loads

No	X	Y	Fz	Mxx	Myy	Fx	Fy
	ft	ft	k	k-ft	k-ft	k	K
1	7.09E+001	2.15E+001	1.00E+001	0.00E+000	0.00E+000	0.00E+000	0.00E+000

Line Loads
Geometry

No	X1	Y1	Z1	X2	Y2	Z2
	ft	ft	ft	ft	ft	ft
1	3.54E+001	3.22E+000	1.00E+001	6.11E+001	4.96E+000	1.00E+001

Values

No	Fz1	Fz2	Mxx1	Mxx2	Myy1	Myy2
	k/ft	k/ft	k-ft/ft	k-ft/ft	k-ft/ft	k-ft/ft
1	1.00E-001	1.00E-001	0.00E+000	0.00E+000	0.00E+000	0.00E+000

Area Loads
No = 1 Z=10.00ft

Vertex	X	Y	Fz	Mxx	Myy
	ft	ft	ksf	k-ft/sf	k-ft/sf
1	2.86E+001	-7.78E+000	2.00E-002	0.00E+000	0.00E+000
2	1.59E+001	2.48E+001	2.00E-002	0.00E+000	0.00E+000
3	9.35E+001	2.96E+001	2.00E-002	0.00E+000	0.00E+000
4	9.06E+001	-1.35E+001	2.00E-002	0.00E+000	0.00E+000

120.30 LIVE LOAD

Concentrated Loads

No	X	Y	Fz	Mxx	Myy	Fx	Fy
	ft	ft	k	k-ft	k-ft	k	K
2	3.66E+001	8.24E+000	1.50E+001	0.00E+000	0.00E+000	0.00E+000	0.00E+000
3	4.91E+001	7.71E+000	1.00E+001	0.00E+000	0.00E+000	0.00E+000	0.00E+000

Line Loads

Geometry

No	X1	Y1	Z1	X2	Y2	Z2
	ft	ft	ft	ft	ft	ft
2	4.32E+001	2.27E+001	1.00E+001	6.06E+001	2.47E+001	1.00E+001

Values

No	Fz1	Fz2	Mxx1	Mxx2	Myy1	Myy2
	k/ft	k/ft	k-ft/ft	k-ft/ft	k-ft/ft	k-ft/ft
2	2.00E-001	2.00E-001	0.00E+000	0.00E+000	0.00E+000	0.00E+000

Area Loads

No = 2 Z=10.00ft

Vertex	X	Y	Fz	Mxx	Myy
	ft	ft	ksf	k-ft/sf	k-ft/sf
1	6.70E+001	1.81E+000	3.00E-002	0.00E+000	0.00E+000
2	6.82E+001	1.11E+001	3.00E-002	0.00E+000	0.00E+000
3	7.98E+001	6.66E+000	3.00E-002	0.00E+000	0.00E+000
4	7.51E+001	-3.39E-002	3.00E-002	0.00E+000	0.00E+000

132 ANALYSIS DATA

132.10 EQUILIBRIUM CHECK

Unit: kN

LOAD CASE NAME =SELFWEIGHT
TOTAL APPLIED LOAD (ALONG X,Y,Z)============ 0.000E+000 0.000E+000 1.262E+003
TOTAL REACTION (ALONG X,Y,Z)================ 4.285E-014 1.381E-012 1.262E+003
%DIFFERENCE (ALONG X,Y,Z)=================== 0.000E+000 0.000E+000 0.000E+000

LOAD CASE NAME =PRESTRESSING
TOTAL APPLIED LOAD (ALONG X,Y,Z)============ 0.000E+000 0.000E+000 0.000E+000
TOTAL REACTION (ALONG X,Y,Z)================ 1.373E-013 4.850E-012 5.253E-010
%DIFFERENCE (ALONG X,Y,Z)=================== 0.000E+000 0.000E+000 0.000E+000

LOAD CASE NAME =HYPERSTATIC
TOTAL APPLIED LOAD (ALONG X,Y,Z)============ 7.247E-012 1.096E+001 8.138E+001
TOTAL REACTION (ALONG X,Y,Z)================ 7.753E-012 1.096E+001 8.138E+001
%DIFFERENCE (ALONG X,Y,Z)=================== 0.000E+000 0.000E+000 0.000E+000

132.20 SOLUTION STATISTICS

Number of shell elements
- -Slab regions 713
- -Walls 4

Number of frame elements
- Beams 24
- Columns 14

Total number of equations

Number of load cases 5
Number of load combinations 2

Total time taken for Analysis 0 Hours 1 Minutes 11 Seconds
Total time taken for Design 0 Hours 1 Minutes 11 Seconds

132.30 NODES
Units: in

NODE	X	Y	Z
1	0.00000E+00	0.00000E+00	1.20000E+02
2	2.48128E+01	0.00000E+00	1.20000E+02
3	4.96257E+01	0.00000E+00	1.20000E+02
4	7.44385E+01	0.00000E+00	1.20000E+02
5	9.92514E+01	0.00000E+00	1.20000E+02
6	1.24064E+02	0.00000E+00	1.20000E+02

132.40 ELEMENTS
Slab Shell Elements

Element	I	J	K	L
1	73	102	106	77
2	102	134	136	106
3	134	184	180	136
4	184	214	213	180
5	214	243	246	213

Wall Shell Elements

Element	I	J	K	L
730	162	163	165	164
731	164	165	167	166
732	166	167	169	168
733	168	169	171	170

Beam Frame Elements

Element	I	J
153	387	388
154	388	389
155	389	390

Column Frame Elements

Element	I	J
177	148	149
178	149	150
179	615	616
180	616	617
181	151	152

152 DESIGN SECTION GEOMETRY

152.20 AUTOMATICALLY GENERATED DESIGN SECTIONS
Design Strip: Support Line 1

Design Section	Start(x,y)	End(x,y)	Centroid (local x,z)	Area	I	Ytop	Ybot
	ft	ft	ft	in2	in4	in	In
1100	(0.05,60.00)	(0.05,40.77)	(9.62,9.67)	1846.16	9845.97	4.00	4.00
1101	(1.58,60.00)	(1.58,40.77)	(9.62,9.67)	1846.16	9845.97	4.00	4.00
1102	(3.17,60.00)	(3.17,40.77)	(9.62,9.67)	1846.16	9845.97	4.00	4.00

152.40 MANUALLY GENERATED DESIGN SECTIONS

Design Section	Start(x,y)	End(x,y)	Centroid (local x,z)	Area	I	Ytop	Ybot
	ft	ft	ft	in2	in4	in	in
1	(44.86,60.71)	(45.01,41.19)	(10.11,9.67)	1805.55	9629.75	4.00	4.00
2	(35.83,60.86)	(35.75,41.04)	(10.34,9.67)	1820.12	9707.11	4.00	4.00

154 DESIGN SECTION ACTIONS

154.20 and 154.40 Option 1: report without twisting moment and eccentricities
The following report section is available if the value for the item "Report_Mxy" in the initialization file is set to 0.

154.20 DESIGN ACTIONS FOR AUTOMATICALLY GENERATED SECTIONS

Load Case: Strength
Design Strip: Support Line 1

Design section	Moment	Shear	Axial	Torsion	Top Stress	Bottom Stress	Centroid Stress
	k-ft	k	k	k-ft	psi	psi	psi
1100	-5.092	1.319	-339.801	-165.658	-0	-0	-0
1101	-3.070	1.319	-339.765	-165.641	-0	-0	-0
1102	-16.421	-10.760	-342.519	-155.396	-0	-0	-0

154.40 DESIGN ACTIONS FOR MANUALLY GENERATED DESIGN SECTIONS

Load Combination: Strength

Design section	Moment	Shear	Axial	Torsion
	k-ft	k	k	k-ft
Design Section 1	4.760	7.974	0.000	21.168

154.20 and 154.40 Option 2: report with twisting moment and eccentricities
If twisting moment as well as eccentricities shall be reported, set the value for the item "Report_Mxy" in the initialization file to 1.

154.20 DESIGN ACTIONS FOR AUTOMATICALLY GENERATED SECTIONS
Load Combination: Strength

ecc = Eccentricity for force
Normal shear eccentricity (ecc) is the distance from the first end of design section toward the second end
Inplane shear eccentricity (ecc) is the distance from the centroid of design section (upward positive)

Design Strip: Support Line 1

Design section	Bending Moment	Twisting Moment	Axial Force	Normal Shear	Inplane Shear	Normal Shear Ecc	Inplane Shear Ecc	Top Stress	Bottom Stress	Centroid Stress
	k-ft	k-ft	k	k	k	in	In	psi	psi	psi
101000	2.180	3.012	-296.864	-15.635	-35.520	78.05	4.00	-244	-214	-229

154.40 DESIGN ACTIONS FOR MANUALLY GENERATED DESIGN SECTIONS

Design section	Moment	Shear	Axial	Torsion	Top Stress	Bottom Stress	Centroid Stress
	k-ft	k	k	k-ft	psi	psi	psi
Design Section 1	-47.369	-48.314	-378.670	21.726	0	-0	-0
Design Section 2	196.532	-14.634	-381.401	-43.596	-1	1	-0

154.20 and 154.40 Option 3: In addition to Option 1 or Option 2 table the program reports table that includes moments with contribution of twisting moments using Wood-Armer approximation. This table is available if design is based on Wood-Armer method.

Design moments with inclusion of twisting moments (Wood-Armer method)

Design section	Positive Bending Moment	Negative Bending Moment
	k-ft	k-ft
101000	3.451	-4.321

NA = Not applicable for "One-way" or "Beam" design sections.

156 DESIGN SECTION REBAR

156.20 DESIGN SECTION REBAR FOR AUTOMATICALLY GENERATED SECTIONS
Load Combination: Service
Design Strip: Support Line 1 Design Criteria: SERVICEABILITY
<Bending rebar>

Design section	As top	As bot	Top bar	Bottom bar
	in2	in2		
1100	0.00	0.00	0#5	0#8
1101	0.00	0.00	0#5	0#8
1102	0.00	0.00	0#5	0#8

156.40 DESIGN SECTION REBAR FOR MANUALLY GENERATED SECTIONS
Design Criteria: SERVICEABILITY
<Bending rebar>

Design section	As top	As bot	Top bar	Bottom bar
	in2	in2		
Design Section 1	1.51	0.00	5#5	0#8
Design Section 2	1.50	0.00	5#5	0#8

158 DESIGN SECTION CAPACITY

158.20 DESIGN SECTION CAPACITIES BASED ON REINFORCEMENT REPORTED IN THE PROGRAM

Design Strip: Support Line 1

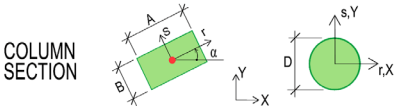
Design section	Moment positive	Moment negative
	KN-M	KN-M
101000	105.300	-129.600
101001	68.560	-174.800
101002	42.690	-201.000
101003	29.510	-221.500
101004	19.160	-288.900
102000	17.640	-258.800

158.40 DESIGN SECTION CAPACITIES FOR MANUALLY GENERATED SECTIONS

Design section	Moment positive	Moment negative
	K-Ft	K-Ft
Design Section 1	39.158	-9.154

160 COLUMNS

160.20 LOWER COLUMNS



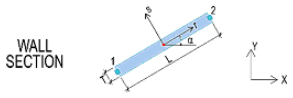
Service

Label	#	Fz	Fr	Fs	Mrr	Mss	Mzz
		k	k	k	k-ft	k-ft	k-ft
Column 1	1	-76.923	0.000	0.000	-93.397	7.963	-0.140
Column 2	2	-145.585	0.000	0.000	28.465	58.813	-0.197
Column 3	3	-74.877	-0.000	-0.000	-50.214	-9.768	0.060

Strength

Label	#	Fz	Fr	Fs	Mrr	Mss	Mzz
		k	k	k	k-ft	k-ft	k-ft
Column 1	1	-110.179	0.000	0.000	-136.065	8.654	-0.225

170 WALLS



Service

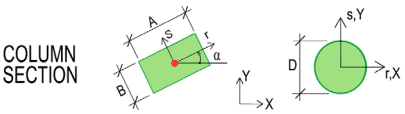
Label	#	Fz	Fr	Fs	Mrr	Mss	Mzz
		k	k	k	k-ft	k-ft	k-ft
Wall 1	1	-111.789	0.000	0.000	-69.125	-7.553	-1.015

Strength

Label	#	Fz	Fr	Fs	Mrr	Mss	Mzz
		k	k	k	k-ft	k-ft	k-ft
Wall 1	1	-172.591	-0.000	0.000	-102.639	-12.251	-1.492

180 PUNCHING SHEAR DESIGN

180.60 PUNCHING SHEAR STRESS CHECK PARAMETERS

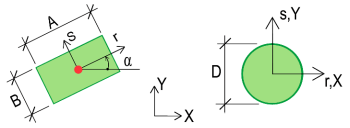


Label	Condition	Axis	Effective depth	Design length rr	Design length ss	Design area	Section constant	Gamma
			mm	mm	mm	mm2	mm4	
Column 1	Interior	rr	175	675	675	4.73E+005	3.65E+010	0.40
Column 1	Interior	ss	175	675	675	4.73E+005	3.65E+010	0.40
Column 2	Interior	rr	175	675	675	4.73E+005	3.65E+010	0.40

Column 2	Interior	ss	175	675	675	4.73E+005	3.65E+010	0.40
Column 3	Interior	rr	175	675	675	4.73E+005	3.65E+010	0.40
Column 3	Interior	ss	175	675	675	4.73E+005	3.65E+010	0.40

180.40 PUNCHING SHEAR STRESS CHECK RESULTS

COLUMN SECTION



Load Combination:Strength

Label	Condition	Axis	Factored shear	Factored moment	Stress due to shear	Stress due to moment	Total stress	Allowable stress	Stress ratio	Case
			kN	kN-m	N/mm2	N/mm2	N/mm2	N/mm2		
Column 1	Interior	rr	-207.050	117.170	0.438	0.434	0.872	1.313	0.66	1
Column 1	Interior	ss	-207.050	127.110	0.438	0.470	0.909	1.313	0.69	1
Column 2	Interior	rr	-377.390	118.680	0.799	0.439	1.238	1.313	0.94	1
Column 2	Interior	ss	-377.390	-67.336	0.799	0.249	1.048	1.313	0.80	1
Column 3	Interior	rr	-136.130	32.940	0.288	0.122	0.410	1.313	0.31	1
Column 3	Interior	ss	-136.130	-17.137	0.288	0.063	0.352	1.313	0.27	1

199.00 REBAR LEGEND

#	Legend	Note
1	A	#8x3'6"@167"o.c.(B)
2	B	1#5x4'6"(T)
3	C	#8x5'0"@148"o.c.(B)
4	D	#8x17'0"@170"o.c.(B)
5	E	3#5x7'0"(T)
6	F	4#5x16'0"(T)
7	G	4#5x16'6"(T)
8	H	9#5x12'0"(T)

200.00 REBAR LIST

#	Legend	Size	System	No.	Length	Unit Weight
					in	lb/ft
1	F	#5	US	4	192.00	1.04
2	F	#5	US	4	198.00	1.04
3	D	#8	US	1	204.00	2.67
4	G	#5	US	9	144.00	1.04
5	B	#5	US	1	54.00	1.04
6	E	#5	US	3	84.00	1.04
7	C	#8	US	1	60.00	2.67
8	A	#8	US	1	42.00	2.67
Total Length(ft):	289.00	Total Weight(lb):	342.93			

210.00 REBAR TOTALS

Bar Reinforcement

Group	Quantity	Size	Length/bar	Length	Weight	Unit cost	Total cost
	bars		ft	ft	lb	$/lb	$
A	1	#8	3.50	3.50	9.35	1.35	12.62
B	1	#5	4.50	4.50	4.69	1.35	6.34
C	1	#8	5.00	5.00	13.35	1.35	18.02

D	1	#8	17.00	17.00	45.39	1.35	61.28
E	3	#5	7.00	21.00	21.90	1.35	29.57
F	4	#5	16.00	64.00	66.75	1.35	90.12
G	4	#5	16.50	66.00	68.84	1.35	92.93
H	9	#5	12.00	108.00	112.65	1.35	152.07
Total				289.00	342.93		462.95

Note:
Group = Identification assigned to a group of bars with the same diameter and length.

Mesh Reinforcement

ID	Size 1	Spacing 1	As1	Size 2	Spacing 2	As2	Area	Weight	Unit cost	Total cost
		in	in2/ft		in	in2/ft	ft 2	lb	$/lb	$
1	#5	24.00		#5	24.00		53.874	56.52	1.35	76.30
2			0.10			0.10	32.578	22.05	1.35	29.77
Total								78.56		106.06

Architectural Beam Reinforcement

Beam label	Quantity	Size	Length/bar	Length	Weight	Unit cost	Total cost
	bars		ft	ft	lb	$/lb	$
Beam 1	4	#5	24.00	96.00	100.13	1.35	135.18
Beam 2	4	#5	24.00	96.00	100.13	1.35	135.18
Total				192.00	200.26		270.35

Total volume of concrete = 572.80 ft^3 (21.21/cu yd)
Total surface of floor slabs = 859.20 ft^2
Average rebar usage = 0.72 lb/ft^2, 1.05 lb/ft^3 (0.05 lb/cu yd)
Total cost of reinforcement = 839.36 $

202 POINT SUPPORT REACTIONS

Service

ID	Label	Fx	Fy	Fz	Mxx	Myy	Mzz
		kN	kN	kN	kN-m	kN-m	kN-m
1	Point Support 1	-0.000	-0.000	161.830	-82.480	-87.750	0.000
2	Point Support 2	-0.000	-0.000	289.950	-86.708	37.450	0.000
3	Point Support 3	0.000	0.000	120.790	-27.172	19.589	-0.000

Strength

ID	Label	Fx	Fy	Fz	Mxx	Myy	Mzz
		kN	kN	kN	kN-m	kN-m	kN-m
1	Point Support 1	0.000	0.000	0.000	0.000	0.000	0.000
2	Point Support 2	-0.000	-0.000	372.190	-111.900	104.600	0.000
3	Point Support 3	0.000	0.000	119.820	-20.188	-8.580	0.000

204 LINE SUPPORT REACTIONS

Service

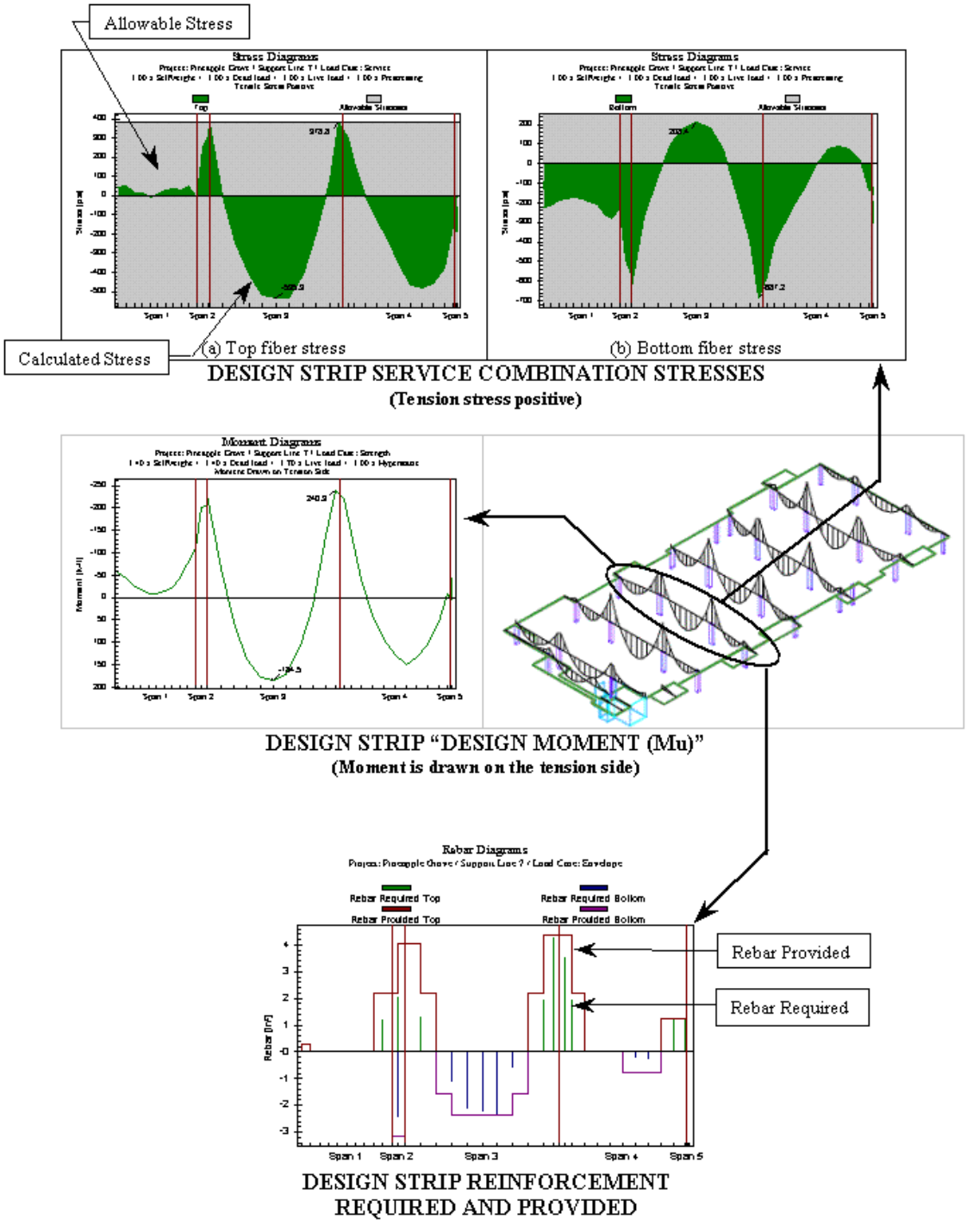
ID	Label	Fz	Fr	Fs	Mrr	Mss	Mzz
		kN	kN	kN	kN-m	kN-m	kN-m

Strength

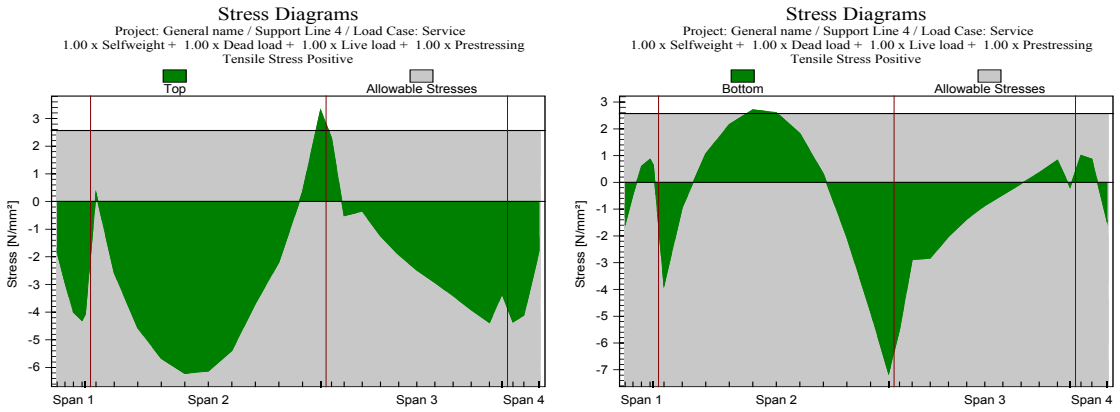
ID	Label	Fz	Fr	Fs	Mrr	Mss	Mzz
		kN	kN	kN	kN-m	kN-m	kN-m

DESIGN LEGEND

This legend is provided as an aid for the interpretation of the results that follow.

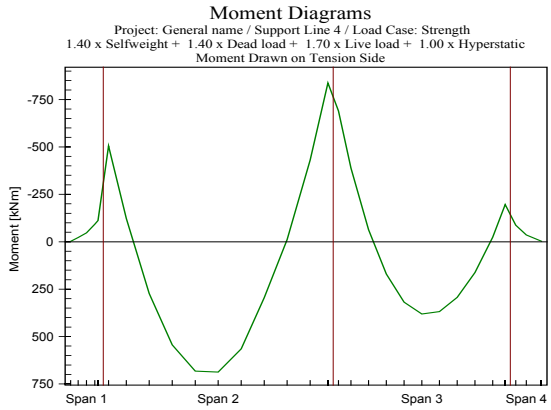


SUPPORT LINE 4

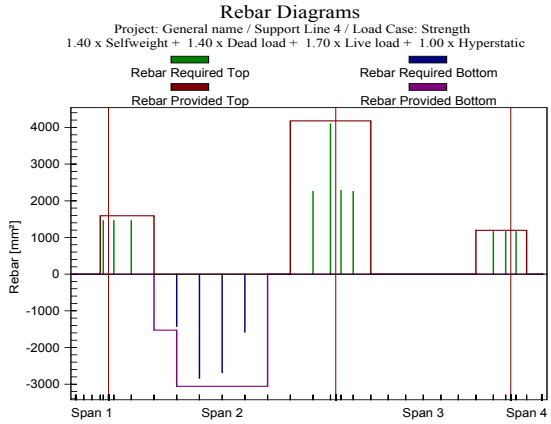


(a) Max tension 3.3 N/mm2, Allowable 2.6 N/mm2 - (b) Max tension 2.7 N/mm2, Allowable 2.6 N/mm2
Max compression -6.2 N/mm2, Allowable 11.9 N/mm2 - Max compression -7.1 N/mm2, Allowable 11.9 N/mm2

DESIGN STRIP SERVICE COMBINATION STRESSES
(Tension stress positive)



DESIGN STRIP "DESIGN MOMENT (MU)"
(Moment is drawn on the tension side)



DESIGN STRIP REINFORCEMENT REQUIRED AND PROVIDED

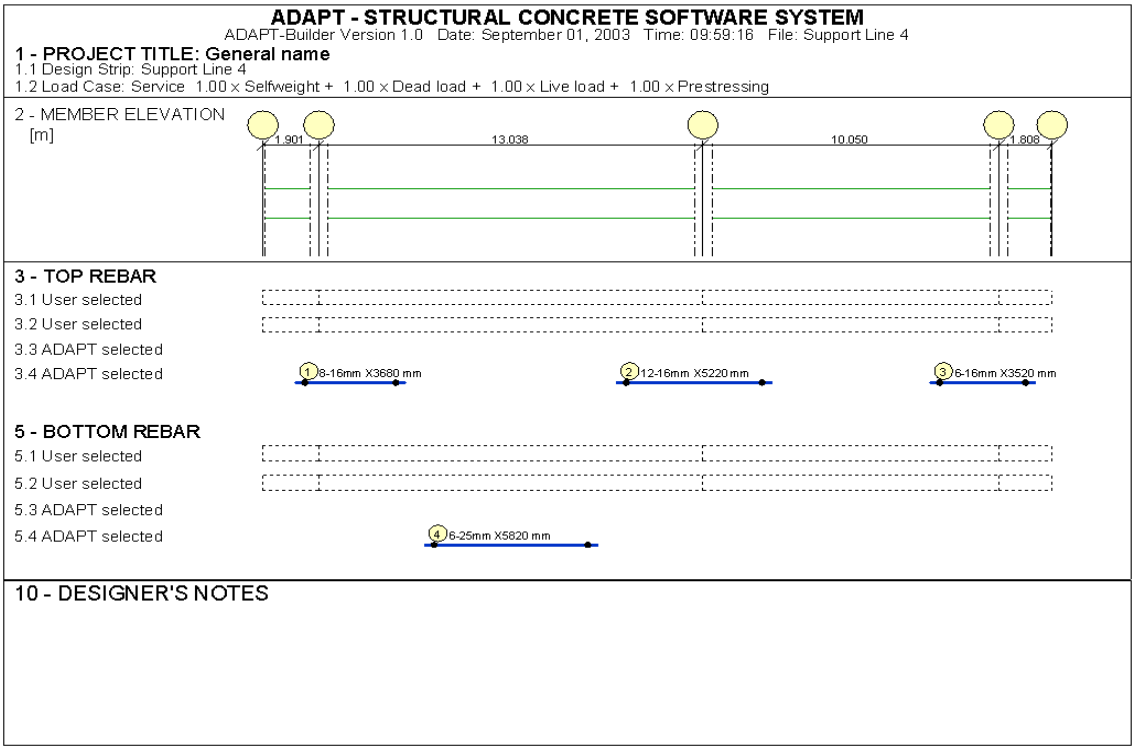


FIGURE 11.6-1 REINFORCEMENT LAYOUT FOR SUPPORT LINE 4

The reinforcement summary created for each design strip shows the location, number of bars, length of the bars and the bar size required to meet all the design criteria specified by you (**Fig. 11.6-1**). This includes, but is not limited to, the initial condition (transfer of prestressing), service, and strength checks.

In order to obtain other reports, you must navigate through the reporting options and select the report of your choice. For example, for the report on punching shear check, the following pull-down and side menus apply (**Fig. 11.6-2**):

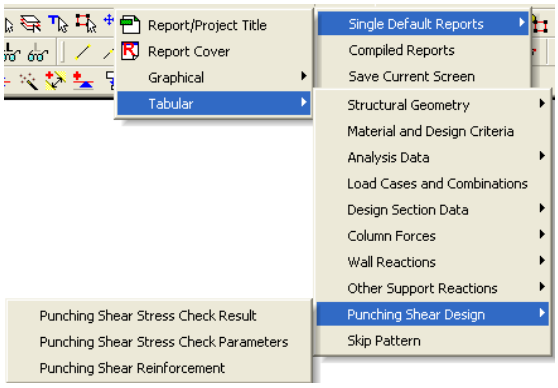


FIGURE 11.6-2

Chapter 12

INTEGRATION WITH MULTISTORY BUILDING PROGRAMS

12.1 OVERVIEW 225
12.2 IMPORT FROM THIRD PARTY PROGRAMS 227
12.3 GENERIC IMPORT/EXPORT 230

12.1 OVERVIEW

ADAPT-BUILDER Platform supports an open data exchange scheme. Through this you can bring into the Builder program the critical information you have generated in, or obtained from other programs. The data exchange scheme allows you to import the geometry of a floor system and its supports, gravity loading and solutions from wind and seismic load cases.

You can use the "ADAPT Input File" (INP) file of the Builder platform generated with the ADAPT Integration Console. The Integration Console reads data from third party programs such as REVIT, ETABS or ROBOT Millennium and generates the INP file for ADAPT-Builder. You can then call this file directly from the ADAPT-Builder. Alternatively, you can manually generate the INP file that has a well-defined text format.

The following describes the basics for data import from the output of lateral analysis programs to ADAPT-BUILDER. An important point to consider is that not all the solutions obtained for the design of the lateral force resisting system of a building can be directly used for the combination of gravity and lateral loads. The following is the reason.

The design values reported for the members designated to resist lateral forces are at times the envelope of several load combinations. While these are valid for the design of each member, such as a column or a shear wall, the collection of these values over the entire floor system is neither in equilibrium, nor does it relate to any specific load case. Hence, the reported values cannot meaningfully be used to determine the resistance to be provided by the floor system. For this reason, for the design of a floor system, the actions from each load case of a lateral analysis should be entered (exported) in ADAPT-BUILDER separately. ADAPT-BUILDER will use the values of each load case and combine them with the gravity loads using the applicable load combinations. ADAPT-BUILDER will conclude with an envelope solution from the combinations of individual lateral load cases and the gravity actions. The envelope will be used by the program for the design of the floor.

Figure 12.1-1 shows the elevation of a frame modeled for lateral analysis by a non-ADAPT program. ADAPT recognizes five levels for this structure, and assigns them levels 01 through 05. Level 01 is the foundation level and the last assigned level is the roof.

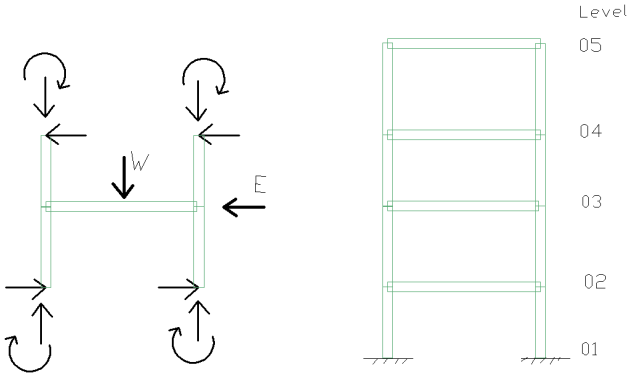
In ADAPT-BUILDER each level is analyzed and designed separately for gravity and combinations of gravity and lateral loading. The structural model for the analysis of a typical level is shown in **Fig. 12.1-1**.

The structural model of a typical level consists of a substructure consisting of the entire floor system of that level, together with the upper and lower walls and columns of the stories immediately above and below the level being modeled. The lateral loads for the substructure are extracted from the analysis of the entire frame and consist of the following:

- Actions from the lateral load analysis at the far ends of the upper walls and columns that are attached to the floor system;
- The horizontal force due to lateral loading associated with the floor system (shown as force E in **Fig. 12.1-1**);

In addition to the lateral loads, for combination with gravity forces, these will be gravity loads as outlined below:

- Gravity load on the floor system (shown as W in **Fig. 12.1-1**);
- Gravity load from levels above and the weight of the walls and columns associated with the substructure being modeled. These are shown with load vectors in the figure. The load vectors represent both gravity and vertical in the figure, but must be reported separately, if these are to be used with different load factors for design.



(a) Substructure of Level 03 (b) Elevation of Frame

FIGURE 12.1-1 VIEW OF A FRAME AND ITS SUBSTRUCTURE

The loads shown in **Fig. 12.1-1** must satisfy static equilibrium in their entirety. Also, the loads due to gravity and the loads due to the lateral forces must each satisfy equilibrium separately.

In addition to the elevated decks, Builder program can import lateral loads necessary for the design of the foundation system. **Fig. 12.1-2** shows the load components that are applicable to level 01 (foundation level).

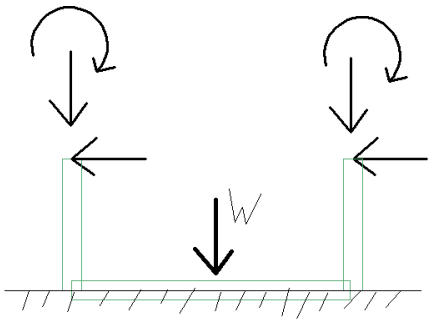


FIGURE 12.1-2 LEVEL 01 – FOUNDATION (MAT)

12.2 IMPORT FROM THIRD PARTY PROGRAMS

When in ADAPT- Builder select option *File/Import.*

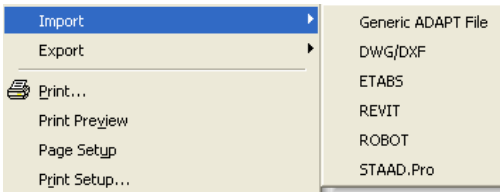


FIGURE 12.2-1

Select the third party program that you are importing from (**Fig. 12.2-1**). The dialog box shown in **Fig. 12.2-2** will display. Navigate to the location where you saved the "input" file and load the file.

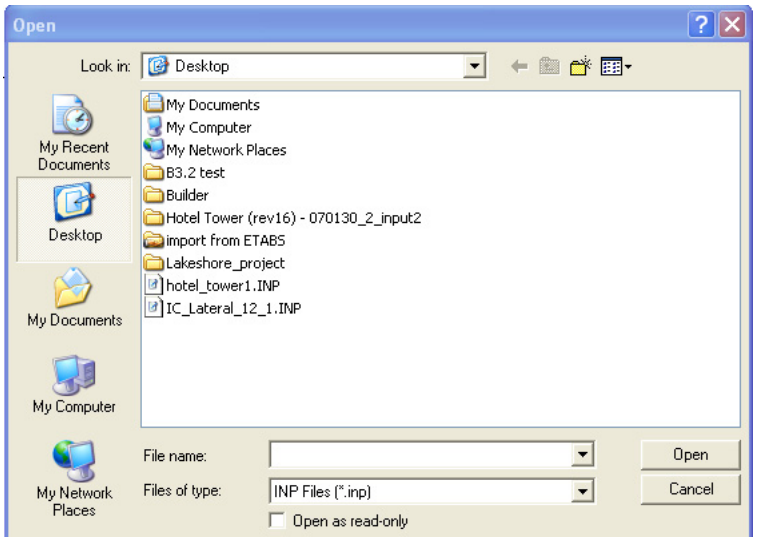


FIGURE 12.2-2

In the *Import options* dialog box enter the level and select components that you want to import (see **Fig. 12.2-3**).

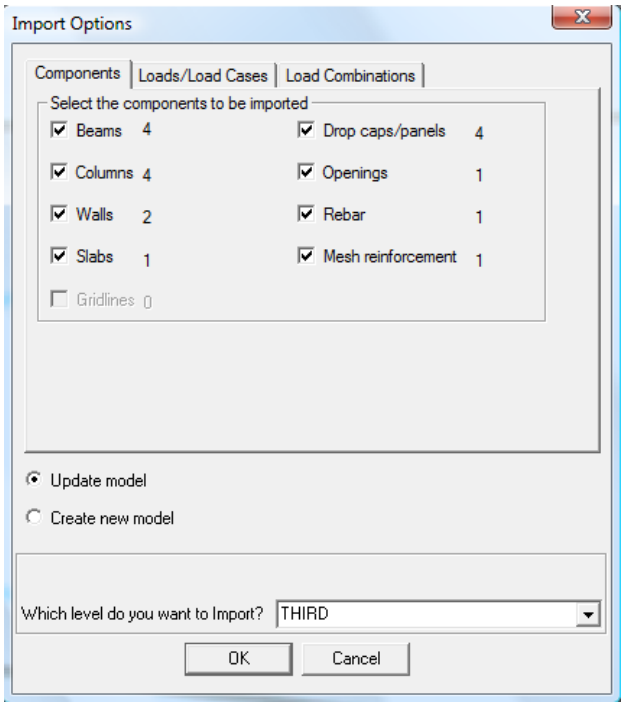


FIGURE 12.2-3

If you check option *Keep original model* the program will import the selected components into the already opened model.

Click OK and the components of level THIRD of the structure will display on the Builder screen.

If you selected to import lateral loads the *Lateral Load Status Check* dialog box shown in **Fig. 12.2-4** will display.

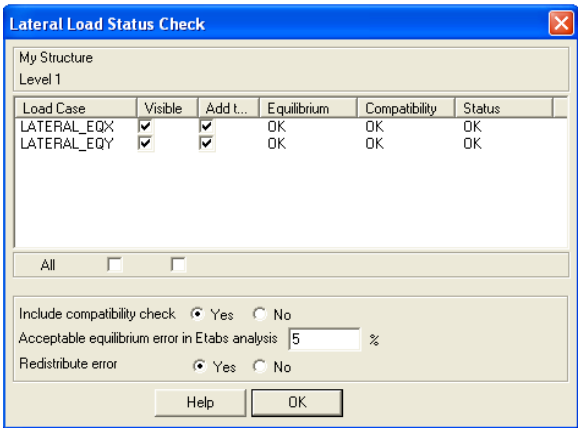


FIGURE 12.2-4

Prior to importing the loads into Builder the program by default does two checks, namely equilibrium and compatibility. The equilibrium check uses equations of statics to verify that the data reported by ETABS and imported to the Builder program is valid. A set of loads that do not meet static equilibrium will be reported by the program as not valid, along with the details of the lack in equilibrium. An acceptable equilibrium error is by default 5% and can be modified by the user. If the user selects "Yes" for "Include compatibility check" option, the program attempts to match each of the loads imported with an existing valid structural component (such as wall, slab) in the structural model. If any of the imported loads is not found to be compatible with a structural component in the model, the program prompts the load with instructions for its editing. If the user selects "No" the program will not perform the compatibility check. If both checks equilibrium and compatibility if applicable are satisfied, the program will import the selected lateral loads and display the loads that are marked "visible" in Builder lateral load dialog box.

If the status is NG for the Compatibility check you have the option to modify the loads.

As the first step in editing a load, you should mark it as "visible" in the *Lateral Load Status Check* table (**Fig. 12.2-4**). This will display the details of the compatibility check. On exiting the report the loads will show on the screen. Double click on the load that you want to modify. This will activate the Lateral Load Editing Mode. When you finish with the load editing close the *Lateral Load Editing* window. If you do not correct the problem fully, the program will re-display the compatibility status report and you have to go through one more cycle of load editing. Once the

prompted problem is edited correctly, the load case will be imported into Builder.

This procedure will be repeated for all the loads that need correction.

Once the lateral forces have been imported, the program automatically creates lateral load cases (see **Fig. 12.2-5**).

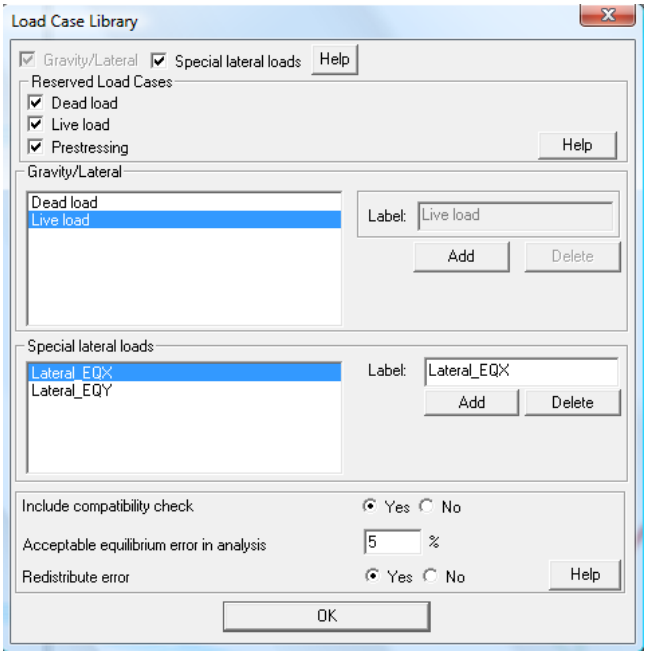


FIGURE 12.2-5

To include the imported lateral loads into the analysis, create load combinations that will include this lateral load.

The boundary conditions for a lateral load case and a gravity load case are different. The program calculates the solution for the lateral loads and the solution for the gravity loads separately and combines them as defined in the load combinations.

12.3 GENERIC IMPORT/EXPORT

In order to automatically import the geometry, gravity loads and solution of a lateral analysis from a program other than the Builder platform, the output from the other programs has to be recast in the format of the ADAPT Input File (INP) file. Once generated, the INP file can be imported into ADAPT-Floor Pro through "File/Import/Generic ADAPT file" menu option. This section introduces the format, and explains its contents.

To export a model containing geometry, loads, load combinations and base reinforcement to an ADAPT generic file, select the "File/Export/Generic ADAPT file" menu option.

The INP file contains the necessary information on the geometry, material and loading of the structure. The structure of the ADAPT file is shown in **Figure 12.3-1**. The description of the data blocks is as follows:

ID_OF_THIS_FILE: This block gives the name of the file, and its version.

```
~DATA ID_OF_THIS_FILE
"Building_B.amx"    // name of the AMX file
"2.00"              // version of the AMX file
~END_DATA ID_OF_THIS_FILE
```

NAME_AND_VERSION: This block in the data file gives the particulars of the program that generated, or last edited the data. An example is given below:

```
. ~DATA NAME_AND_VERSION
"Buiolding_B.rvt"   // name of the data file used to generate AMX file
"March 20, 2006"    // generation date
"11:40"             // generation time
"Revit"             // name of the program that generated data
"5.00"              // release number of the original software
~END_DATA NAME_AND_VERSIO
```

FOLDER_DIRECTORY: This block consists of 5 sub blocks:

UNITS: The block that defines the units. All values must be expressed in N and mm. ADAPT understands and displays its report in user selected units.

BASIC_STRUCTUREAL_INFORMATION: The block with the basic structural information – defines number of levels and elevation. LEVEL_1 is always the base of the structure.

MATERIALS: The block that will define material properties - different types of concrete

LOAD_COMBINATIONS: This data block lists all the load combinations that will be imported to the ADAPT program. Each load combination is defined with load cases and their load factors, analysis/design option and label.

DATA_OF_EACH_LEVEL: The block that has information about the geometry and loads for each level of a structure.

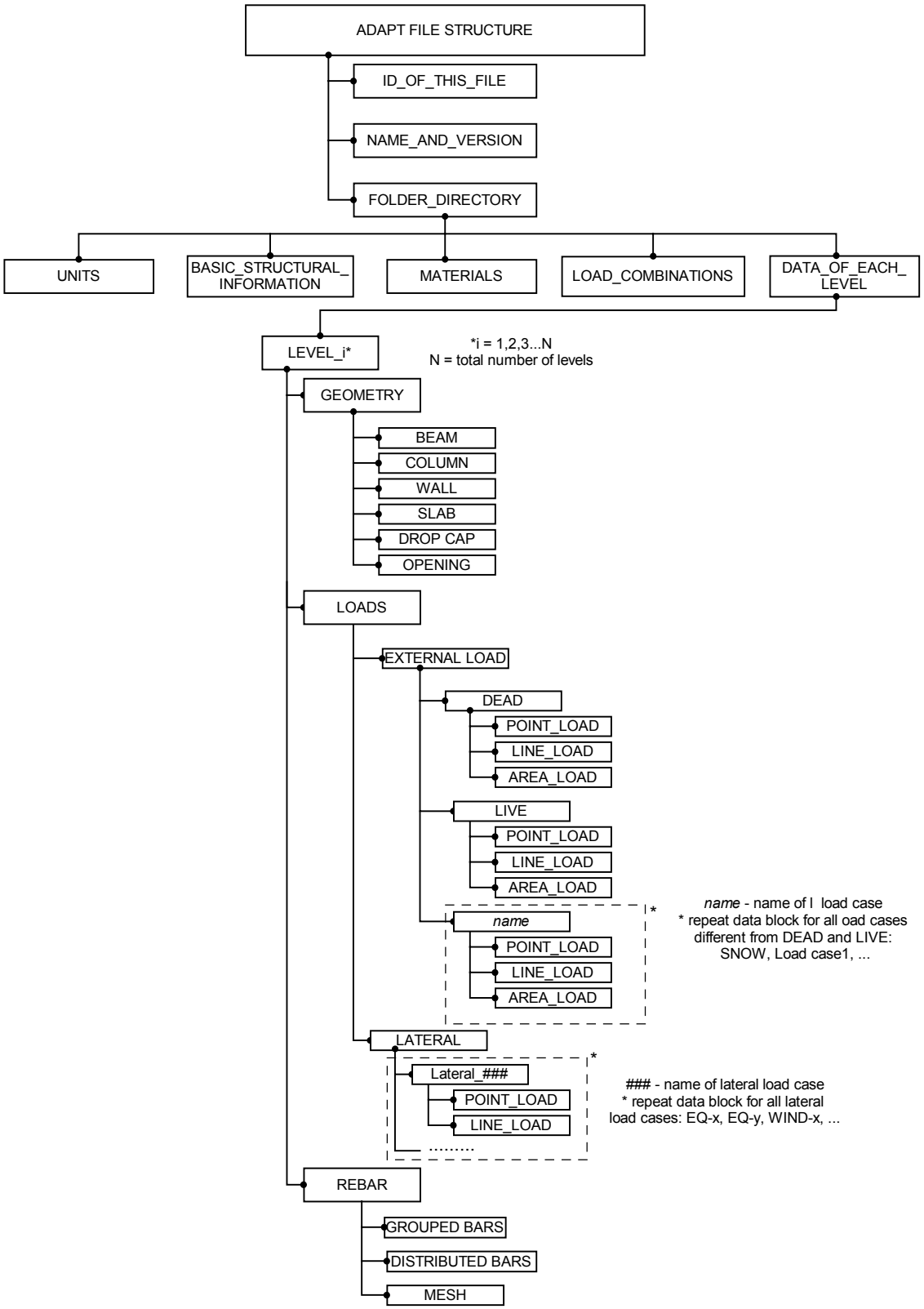


FIGURE 12.3-1 STRUCTURE OF ADAPT FILE

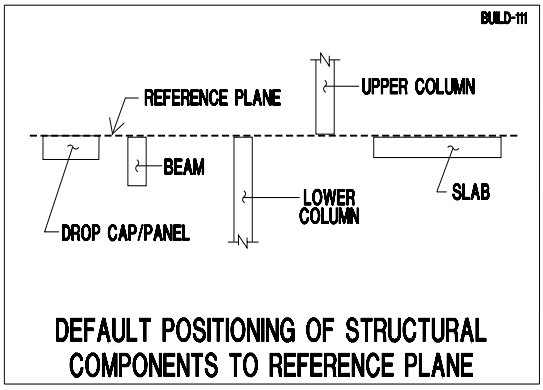
Builder defines the geometry of the structural elements as follows:

GEOMETRY: The geometry block contains all the information regarding coordinates of the structural elements, cross-sections, and materials used. Each "LEVEL" consists of several sub-blocks named **"BEAMS", "COLUMNS", "WALL", "SLAB", "DROP CAP/PANEL",** and **"OPENING"**. The following is a detailed description of the geometry block.

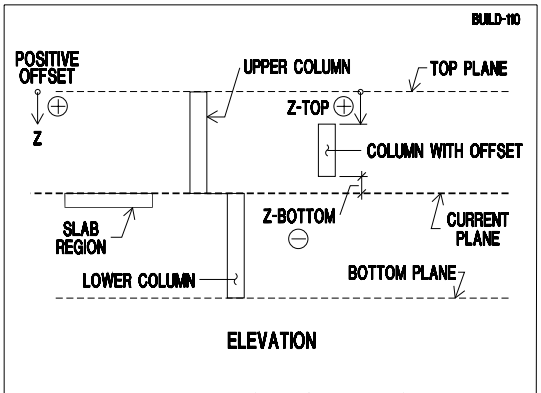
Global axes
The position of the structural components is expressed in terms of the global axes X, Y and Z.

Reference Plane (Fig. 12.3-2)
The reference place is a user defined (envisaged) horizontal plane, that is used as a datum, with respect to which the position of the structural components are given. In the current data file, the position of the reference planes is defined by the parameter "Level-###" The coordinate Z defined for each "Level_###" defines the position of the reference plane of each level. The reference plane is generally selected to be the top surface of the primary floor slab.

Offset (Fig. 12.3-2)
Offset is the vertical distance (along Z-direction) from the reference plane. A distance below the reference plane is positive – above is negative. This is contradictory to the positive direction of the global Z-axis.



DEFAULT POSITIONING OF STRUCTURAL COMPONENTS TO REFERENCE PLANE



REFERENCE PLANE AND DEFAULT POSITIONING OF WALLS AND COLUMNS

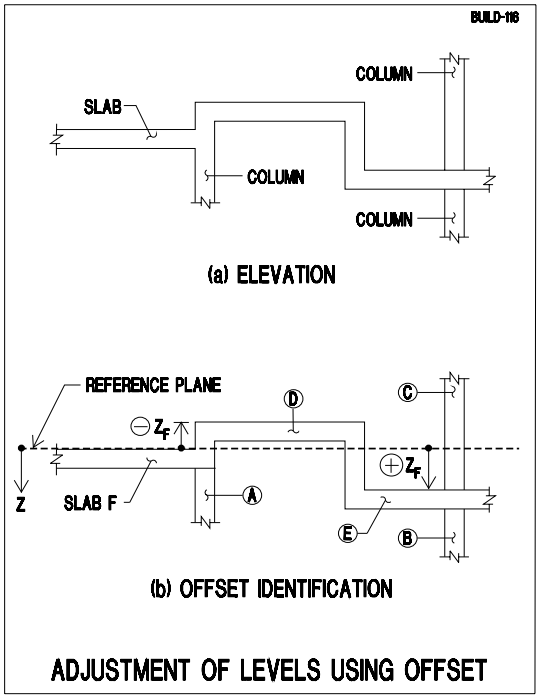


FIGURE 12.3-2 REFERENCE PLANE AND OFFSET

BEAM: The beams are defined by the position of the reference plane, two points in the reference plane, cross-sectional properties (width and depth), offset, material type and label (**Fig. 12.3-3**). Note that the "offset" parameter used, specifies the distance of the top fiber of a beam from a user defined reference plane. The offset below the reference plane is assumed "positive."



FIGURE 12.3-3 BEAM GEOMETRY

Data block "BEAM" should contain the following:

```
            ~DATA BEAM
            5011100 "BEAM"
//X1,Y1,X2,Y2 = cords of end of beam;  width, depth = stem of beam to top of slab
//offset is distance from top of beam stem to reference line.
//----X1-------Y1------X2---------Y2-------WIDTH------DEPTH------OFFSET------MATRIAL_ID---LABEL---
//----1--------2-------3----------4----------5----------6-----------7-------------8---------9-----
        ~END_DATA BEAM
```

COLUMN: Columns are assumed to be vertical (along Z-axis). The position of a column is defined by the the X, Y coordinates of its centroid (**Fig. 12.3-4**). Other parameters (flag) define whether a column is above, or below the reference line. The length of a column is defined by the position of its top and bottom ends with respect to the reference planes that associate with its top and bottom ends. For example, the position of the top end of an upper column for Level_3 is defined with its offset from the Z-coordinate of Level_4. The cross-sectional geometry of a column must be either rectangular (square) or circular. The orientation of a column is given by the angle of its first dimension from X-axis, measured counter-clockwise.

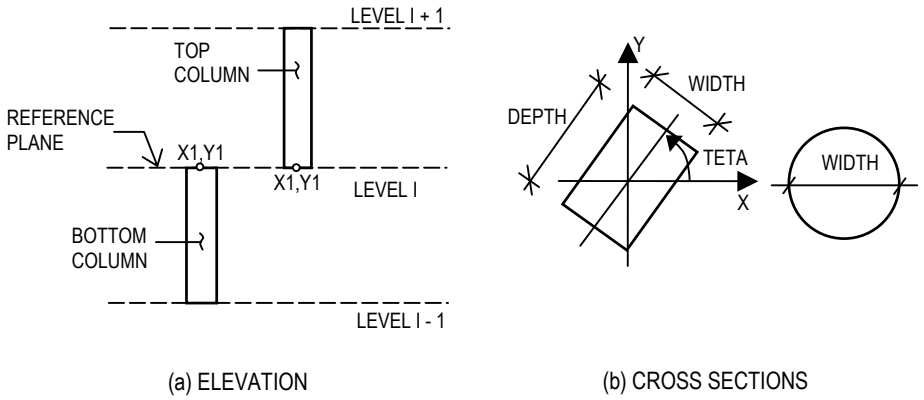


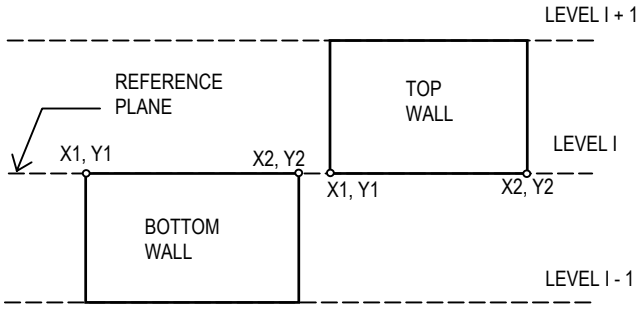
FIGURE 12.3-4 COLUMN GEOMETRY

Data block "COLUMN" should contain the following:

```
            ~DATA COLUMN
            5011200 "COLUMN"
//bottom columns
//Flag (1 = top column, 2 = bottom column), Type (1 = rectangular column, 2 = circular column)
//-Flag--X1---Y1---THETA---WIDTH---DEPTH----TYPE---OFFSET_TOP---OFFSET_BOTT---MATRIAL_ID---LABEL--
//--1----2----3-----4--------5-------6-------7---------8-------------9-----------10---------11----

//top columns
//Flag (1 = top column, 2 = bottom column), Type (1 = rectangular column, 2 = circular column)
//-Flag--X1---Y1---THETA---WIDTH---DEPTH----TYPE---OFFSET_TOP---OFFSET_BOTT---MATRIAL_ID---LABEL--
//--1----2----3-----4--------5-------6-------7---------8-------------9-----------10---------11----
        ~END_DATA COLUMN
```

WALL: Walls are assumed to be straight, vertical and without an opening. The position of each wall is defined by the coordinates X, Y of its two ends (X1, Y1 and X2, Y2) on the reference plane (**Fig. 12.3-5**). Like columns, the top and bottom of each wall is specified by its offset from the respective reference planes (Level_###).



(a) ELEVATION

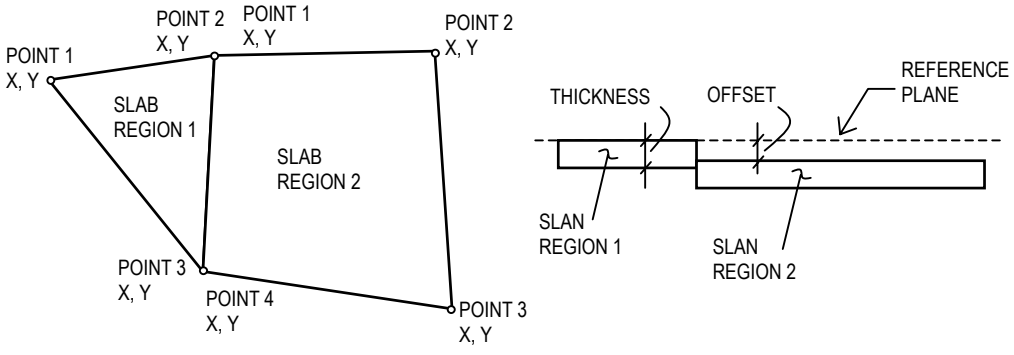
FIGURE 12.3-5 WALL GEOMETRY

Data block "WALL" should contain the following:

```
            ~DATA WALL
                    401300 "WALL"
            ~DATA WALL
            5011300 "WALL"
//bottom walls
//Flag (1=top wall, 2=bottom wall)
//-Flag---X1-----Y1----X2----Y2---THICKNESS----OFFSET_TOP----OFFSET_BOTT----MATRIAL_ID-----LABEL--
//--1-----2------3-----4-----5-------6-------------7--------------8-------------9------------10---

//top walls
//Flag (1=top wall, 2=bottom wall)
//-Flag---X1-----Y1----X2----Y2---THICKNESS----OFFSET_TOP----OFFSET_BOTT----MATRIAL_ID-----LABEL--
//--1-----2------3-----4-----5-------6-------------7--------------8-------------9------------10---
        ~END_DATA WALL
```

SLAB: Slabs are assumes to be horizontal and in form of irregular closed polygons. Their position is defined at a user defined distance (offset) from the reference plane. The plan geometry of a slab region is defined by the X, Y coordinates of its vertices (**Fig. 12.3-6**). A floor system can consist of more than one slab region.



(a) PLAN VIEW (b) ELEVATION

FIGURE 12.3-6 SLAB GEOMETRY

Data block “SLAB” should contain the following:

```
~DATA SLAB
5011400 "SLAB"
~DATA CAPSULE_DIRECTORY
    5011410 “SLAB PARAMETERS”
    5011420 “SLAB GEOMETRY”
~END_DATA CAPSULE_DIRECTORY

    ~DATA SLAB_PARAMETERS
    5011410 “SLAB PARAMETERS”
            “Number of slabs”    2
    ~END_DATA SLAB_PARAMETERS

    ~DATA SLAB_GEOMETRY
    5011420 “SLAB GEOMETRY”
    ~DATA CAPSULE_DIRECTORY
        5011421 “SLAB_1”
        5011422 “SLAB_2”
        // add or delete as necessary, but remain sequential
    ~END_DATA CAPSULE_DIRECTORY

        ~DATA SLAB_1
        5011421   “SLAB_1”
//---NUMBER_OF_POINTS-----THICKNESS-------OFFSET-------MATERIAL_ID-------LABEL------
//-----------1---------------2---------------3--------------4--------------5--------

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
        ~END_DATA SLAB_1

        ~DATA SLAB_2
        5011422   “SLAB_2”
//---NUMBER_OF_POINTS-----THICKNESS-------OFFSET-------MATERIAL_ID-------LABEL------
//-----------1---------------2---------------3--------------4--------------5--------

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
        ~END_DATA SLAB_2
    ~END_DATA SLAB_GEOMETRY
~END_DATA SLAB
```

DROP CAP/PANEL: Drop caps/panels are defined by (i) the position of their centroid (X,Y), the offset from the reference plane, and the angle of their first side from the X-axis measured counter-clockwise. The thickness of a drop cap/panel refers to the total depth, including any overlap with a slab region (**Fig. 12.3-7**).

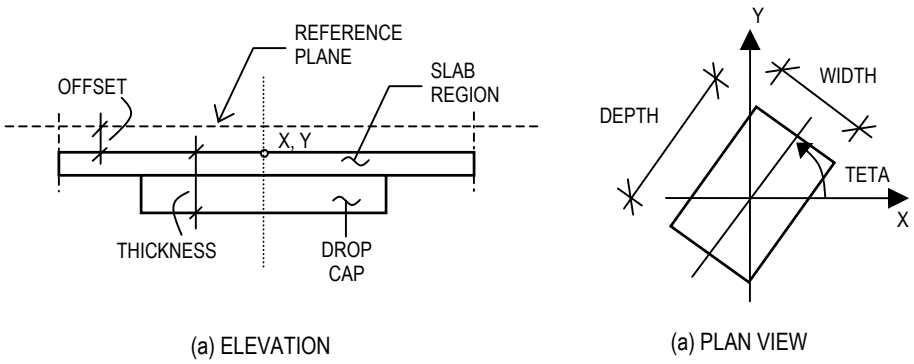
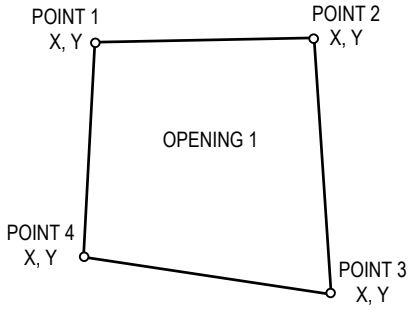


FIGURE 12.3-7 DROP CAP/PANEL GEOMETRY

Data block “DROP CAP/PANEL” should contain the following:

```
        ~DATA DROP_CAP/PANEL
        5011500 "DROP_CAP/PANEL"
//Thickness = stem of drop cap/panel to top of slab
//---X1------Y1------WIDTH------LENGTH------THICKNESS----TETA-----OFFSET-----MATERIAL_ID---LABEL--
//---1-------2---------3----------4------------5----------6---------7------------8-----------9----
        ~END_DATA DROP_CAP/PANEL
```

OPENING: Openings are defined in the same manner as slab regions (**Fig. 12.3-8**).



(a) PLAN VIEW

FIGURE 12.3-8 OPENING GEOMETRY

Data block “OPENING” should contain the following:

```
            ~DATA OPENING
            5011600 "OPENING"
            ~DATA CAPSULE_DIRECTORY
                5011610 “OPENING PARAMETERS”
                5011620 “OPENING GEOMETRY”
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA OPENING_PARAMETERS
                5011610 “OPENING PARAMETERS”
                       “Number of openings”    1
                ~END_DATA OPENING_PARAMETERS

                ~DATA OPENING_GEOMETRY
                5011620 “OPENING GEOMETRY”
                ~DATA CAPSULE_DIRECTORY
                    5011621 “OPENING_1”
                ~END_DATA CAPSULE_DIRECTORY

                    ~DATA OPENING_1
                    5011621 “OPENING_1”
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
                    ~END_DATA OPENING_1
                ~END_DATA OPENING _GEOMETRY
            ~END_DATA OPENING
```

LOADS: The folder with loading information consists of several data blocks: **DEAD, LIVE, OTHER, LATERAL_*name***. Blocks DEAD, LIVE and OTHER contain gravity forces

applied on the slab, while lateral contain forces that are results of a lateral analysis. The blocks contain the following:

DEAD: dead load case forces acting on the slab

LIVE: live load case forces acting on the slab

OTHER: gravity forces different from dead and live load cases acting on the slab

LATERAL: Lateral load data blocks report lateral forces at the top and the bottom of the columns and walls for the specified level. A force acting at the column end is a point load, while forces acting at the wall ends can be either a point or line load.

Each data block contains sub blocks: **POINT_LOAD, LINE_LOAD** and **AREA_LOAD**. Each of these blocks is described in the following.

POINT_LOAD: **A** point load is defined with the global coordinates X, Y, Z of its point of action and intensity in X, Y and Z directions (Fx, Fy, Fz, Mxx, Myy, Mzz) (**Fig. 12.3-9**).
Note: The Z-component of a point load is downward positive.

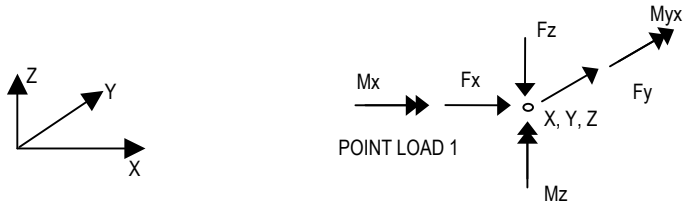


FIGURE 12.3-9 POINT LOAD

Data block "POINT_LOAD" should contain the following:

```
                ~DATA POINT_LOAD
                "POINT_LOAD"
//--X1------Y1------Z1------Fx------Fy------Fz------Mxx-------Myy-------LABEL-------
//--1---------------3-------4-------5-------6-------7---------8-----------9---------
                ~END_DATA POINT_LOAD
```

LINE_LOAD: A line load is defined with its position and intensity. The coordinates of the beginning (point 1) and end point (point 2) of the line load determine the position of the line load. The intensity is specified in point 1 and point 2, between these two points the intensity changes linearly (**Fig. 12.3-10**).

Note: The Z-component of a line load is downward positive.

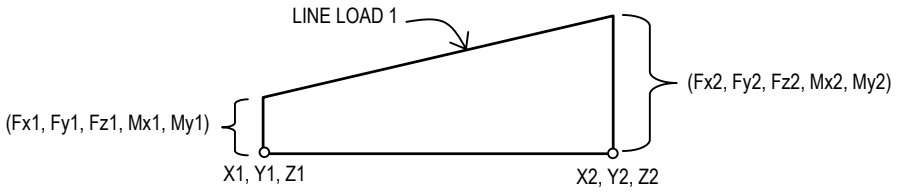


FIGURE 12.3-10 LINE LOAD

Data block "LINE_LOAD" should contain the following:

```
    ~DATA LINE_LOAD
            "LINE_LOAD"
//--X1--Y1--Z1--X2--Y2--Z2---Fx1----Fx2---Fy1---Fy2---Fz1---Fz2---Mx1---Mx2---My1---My2--LABEL---
//--1---2---3---4---5---6----7------8------9----10----11----12-----13----14----15----16-----17----
-
            ~END_DATA LINE_LOAD
```

AREA_LOAD: **An** area load is described with the global coordinates of its points and forces defined in three points. The program calculates the intensity of the other points of the area load automatically by determining the intersection of the point vertex and plane defined with the coordinates and intensity of three points (**Fig. 12.3-11**).

Note: The Z-component of an area load is downward positive.

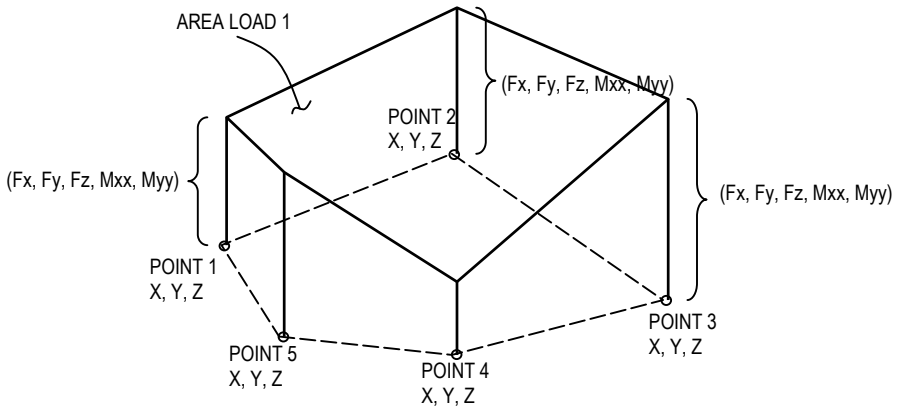


FIGURE 12.3-11 AREA LOAD

Data block AREA_LOAD should contain the following information:

```
            ~DATA AREA_LOAD
            "AREA_LOAD"
            ~DATA CAPSULE_DIRECTORY
                5012110 "AREA_LOAD1"
                //add or delete as needed
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA AREA_LOAD1
                5012110 "AREA LOAD1"
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------

//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2---------------3--------------4---------------5---------------6-------
                ~END_DATA AREA_LOAD1
            ~END_DATA AREA_LOAD
```

LATERAL_*name*: This data block contains the reactions (point or line loads) acting at the far ends of supports obtained for the lateral load case **LATERAL_*name*** and story shear for the level under consideration.

Data block "LATERAL_name" should contain the following:

```
        ~DATA Lateral_EQ
        5012400 "Lateral_EQ"
            ~DATA POINT_LOAD
            "POINT_LOAD"
//At far ends of UPPER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1-------Y1-------Z1------Fx-----Fy------Fz------Mxx------Myy------Mzz------LABEL-----
//-1--------2--------3----------------5--------------7-------8--------9--------10--------11-------

//At far ends of LOWER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag----X1-------Y1--------Z1------Fx-----Fy------Fz------Mxx------Myy-------Mzz-----LABEL-----
//-1-------2--------3---------4-------5------6-------7-------8--------9---------10-------11-------

//Story Shear Forces
//Flag (2 = upper columns, 1 = story shear, 0 = lower column
//-Flag----X1-------Y1--------Z1------Fx------Fy------Fz------Mxx------Myy-------Mzz----LABEL-----
//-1--------2-------3---------4-------5-------6-------7--------8-------9---------10------11-------
            ~END_DATA POINT_LOAD

            ~DATA LINE_LOAD
            "LINE_LOAD"
//At far ends of UPPER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1------Y1-----Z1-----X2----Y2----Z2-----Fx------FY-----FZ----Mxx----Myy---Mzz--LABEL-
//--1-------2-------3------4------5-----6-----7------8-------9------10----11------12----13---14---
---


//At far ends of LOWER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1------Y1-----Z1-----X2----Y2----Z2-----Fx------FY-----FZ----Mxx---Myy---Mzz--LABEL--
//--1-------2-------3------4------5-----6-----7------8-------9------10----11----12-----13---14----
--
            ~END_DATA LINE_LOAD
        ~END_DATA Lateral_EQ
```

REBAR DISPLAY PROPERTIES. This data block contains all the properties and attributes of all the reinforcement that is part of the structure. The rebar included is calculated and base reinforcement, that can be grouped, distributed or mesh rebar. When importing rebar to ADAPT-Builder the reinforcement is tagged as "base reinforcement" in the program.

```
~DATA REBAR_DISPLAY_PROPERTIES
    ~DATA REBAR_COLOR
        1         R=87 G=171 B=115
        2         R=171 G=0 B=0
    ~END_DATA REBAR_COLOR

    ~DATA REBAR_LINE_TYPE
        1        "CONTINUOUS"
        2        "Dashed"
    ~END_DATA REBAR_LINE_TYPE

    ~DATA REBAR_LINE_THICKNESS
        1        3
        2        1
    ~END_DATA REBAR LINE_THICKNESS

    ~DATA REBAR_DISPLAY_TEXTURE
    ~END_DATA REBAR_DISPLAY_TEXTURE
~END_DATA REBAR_DISPLAY_PROPERTIES

~DATA REBAR_DRAWING_GRAPHICS
    ~DATA REBAR_CALL_OUT_GRAPHICS
        "line_thickness"            3
        "line_color"      R=87 G=171 B=115
        "arrow_size"                        10
        "circle_size"               17
    ~END_DATA REBAR_CALL_OUT_GRAPHICS

    ~DATA REBAR_TEXT
        "Font_size"              12
        "Font_type"                 "STANDARD"
        "Text_color"                R=87 G=171 B=115
    ~END_DATA REBAR_TEXT

~END_DATA REBAR_DRAWING_GRAPHICS

~DATA REBAR_GRAPHICAL_DISPLAY
//-Mark--Color--LineType-LineThickness-TextThickness----Texture---
//-1------2-------3---------4----------5----------------6------

~END_DATA REBAR_GRAPHICAL_DISPLAY

~DATA REBAR_DISPOSITION
        "Single_bar"            1
        "Bundled_bars"          2
        "Grouped"               3
        "Distributed"           4
~END_DATA REBAR_DISPOSITION

~DATA REBAR_MARK_AND_PROPERTIES
// 1 - Rebar "Mark" determined by ADAPT, starting by 1 and increasing numerically
// 2 - Rebar Material number from the material library
// 3 - Rebar source, such as from computation, or detailing
// 4 - Group, which group the bar belongs to
// 5 - Rebar associate member type, such as beam, slab, drop cap
// 6 - Rebar associate member ID, such as beam number
// 7 - Cross sectional area
// 8 - diameter
// 9 - cover
// 10 - weight density in kg/mm
// 11 - Rebar size from previous data blocks
// 12 - User defined mark on the drawing
//                         <-Associate->
//-Mark-Mat--source-group---Type-ID------Area---Diameter----Cover---density--visibility---size--
--user mark-
//-1----2------3-----4------5----6---------7----------8-------9---------10---------11------12---
----13

~END_DATA REBAR_MARK_AND_PROPERTIES

~DATA RBAR_SHAPE_AND_NUMBER
```

```
// 1 - Rebar "Mark" determined by ADAPT, starting by 1 and increasing numerically
// 2 - Rebar number of times it is repeated in the drawings
// 3 - Rebar length
// 4 - Rebar shape number as defined above
// 5 - Rebar length A
// 6 - Rebar length B
// 7 - Rebar length C
// 8 - Rebar length D
// 9 - Rebar length E
//10 - or beyond other lengths, if applicable
// last string user defined annotation for the specific bar
//Mark-Number--length---shape-----A----------B--------C----------D-----------E-----annotation
//-1-----2-------3--------4-------5----------6--------7----------8-----------9----------10------

~END_DATA REBAR_SHAPE_AND_NUMBER

~DATA REBAR_LOCATION
    ~DATA REBAR_GLOBAL_LOCATION
           //top bar = 2, bottom bar = 1
    // Xi,Yi,Zi = global coordinates of the first hot point of the bar
    // Xj,Yj,Zj = global coordinates of the last hot point of the bar
    // Xm,Ym,Zm = global coordinates of the positioning hot point of the bar
    // Mark top/bottom < -------GLOBAL START----------->  < -------GLOBAL END--------->   <------
GLOBAR THIRD HOT POINT---->
    //--1--2---Xi--------Yi----------Zi----------Xj---------Yj---------Zj--------Xm---------Ym--
-------Zm------

    ~END_DATA REBAR_GLOBAL_LOCATION

    ~DATA REBAR_LOCAL_LOCATION
    // TYPE is like "beam, crop cap, slab" and is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52"
refers to a specific beam
    // xi,yi,zi = local coordinates of the first hot point of the bar
    // xj,yj,zj = local coordinates of the last hot point of the bar
    // xm,ym,zm = local coordinates of the positioning hot point of the bar
    // Mark           < -------LOCAL START----------->  < -----------LOCAL END--------->   <----
--LOCAL THIRD HOT POINT---->
    //--1---TYPE---ID-----xi--------yi----------zi----------xj---------yj---------zj--------xm--
-------ym---------zm------
    ~END_DATA REBAR_LOCAL_LOCATION

    ~DATA REBAR_DISPOSITION_LEGEND
    // TYPE is like "beam, crop cap, slab" is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52"
refers to a specific beam
    // Number = number of bars with given Mark
    // Disposition
    //    1 = single bars
    //    2 = bundled bars
    //    3 = grouped bars
    //    4 = distributed bars
    //-Mark-TYPE---ID---Number-----Dispositon--

    ~END_DATA REBAR_DISPOSITION_LEGEND
~END_DATA REBAR_LOCATION
~DATA REBAR_LAYOUT
// This data block lists only the bars that are distributed.
    ~DATA GLOBAL_EXTENT_OF_DISTRIBUTED_BARS
    // Xi,Yi,Zi = global coordinates of the first end of the rebar range, (closer to global
origin)
    // Xj,Yj,Zj = global coordinates of the second point of the rebar range
    // distribute=1 show bars individually. Do not show the extent call out
    // distribute=0 show a single bar with the range line
    // adjust = 1 truncate the bar, if when distributed can falls outside a slab region or
enters an opening
    // adjust = 0 do not adjust the bar length. Show and annotate them as given in input data
    // Mark < -------GLOBAL START----------->  < -------GLOBAL END--------->
    //--1-----Xi--------Yi----------Zi----------Xj---------Yj---------Zj------distribute-adjust-
-

    ~END_DATA GLOBAL_EXTENT_OF_DISTRIBUTED_BARS

    ~DATA LOCAL_EXTENT_OF_DISTRIBUTED_BARS
    // TYPE is like "beam, crop cap, slab" and is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52"
refers to a specific beam
    // xi,yi,zi = local coordinates of the first hot point of the bar (closest to the origin)
    // xj,yj,zj = local coordinates of the last hot point of the bar
    // distribute=1 show bars individually. Do not show the extent call out
    // distribute=0 show a single bar with the range line
```

```
    // adjust = 1 truncate the bar, if when distributed can falls outside a slab region or
enters an opening
    // adjust = 0 do not adjust the bar length. Show and annotate them as given in input data
    // Mark           < -------LOCAL START---------->  < -----------LOCAL END--------->
    //--1---TYPE---ID-----xi--------yi----------zi----------xj---------yj---------zj------
distribute-adjust-
    ~END_DATA REBAR_LOCAL_LOCATION
    ~END_DATA REBAR_LAYOUT

~DATA MESH_REINFORCEMENT
~DATA MESH_GRAPHICAL_DISPLAY
//-Mark--Color--LineType-LineThickness-TextThickness----Texture---
//-1------2-------3---------4----------5----------------6------

~END_DATA MESH_GRAPHICAL_DISPLAY
~DATA MESH_ID_AND_SIZE_BOTTOM
// Mark = ADAPT defined sequential number
// Material ID
// Area = cross-sectional area in mm2/mm
// Diameter
// size = bar size as given in rebar library
// CGS = centroid distance of bar from bottom surface of member in mm
//angle = measured  counterclockwise from positive X to rebar direction
// Mark AssociateType AssociateID< ------------first bar in mesh---------------------->< --------
-second bar in mesh----------------------------------->
//--Mark-MatID--Type---ID---Area----Diameter---size-------CGS--------angle----spacing------area-
--Diameter----size-------CGS--------angle----spacing---user mark-
~END_DATA MESH_ID_AND_SIZE_BOTTOM

~DATA MESH_ID_AND_SIZE_TOP
// Mark AssociateType AssociateID< ------------first bar in mesh---------------------->< --------
-second bar in mesh----------------------------------->
//--Mark--MatID--Type---ID---Area----Diameter---size-------CGS--------angle----spacing------
area---Diameter----size-------CGS--------angle----spacing---user mark-

~END_DATA MESH_ID_AND_SIZE_TOP

~DATA MESH_LOCATION
//xi,yi = coordinates of the first hot point of the bar
//-MARK---Vertex------Xi---------Yi------


~END_DATA MESH_LOCATION
```

Notes on integration:

- Slab regions and beams with a slope: if the slope is less than 10%, consider the member horizontal. Retain the actual thickness, but express its elevation as the average between its highest and lowest points.

- Curved members on Plan: Curved members and boundaries must be simplified, before they can be exported to ADAPT. Break the curved boundary (member) into straight segments of (i) 1 m (3ft) in length, or 10 degrees subtended angle, whichever results in a longer chord.

- Create structural components only from (i) members that are made of concrete, and (ii) are expected to participate in the load carrying system of the building.

- Do not export members that are sloped beyond the 10% limit. In other words, ramps and sloping roofs shall not be exported.

- The current release of adapt can handle structural members that are of rectangular geometry. Idealize the non-rectangular cross-sections as follows:

  - For column cross-sections, substitute the original column by a square column of equal area.
  - For beams, replace the actual geometry by a rectangular beam with a width and depth that circumscribes the actual beam.
  - For slab regions, assume the same plan (X-Y plane), with an average thickness. Use the z-offset of the location with average thickness.
  - Substitute the drop caps and drop panels that are not rectangular, by a rectangular shape inscribed in each.

- Model foundation mat and grade beams, but not spread footings, piles and other foundation parts.

- Treat non-prismatic members, such as columns and beams with a taper as follows:
  - Substitute tapered columns or non-prismatic columns by a column having the cross-sectional geometry of the column at its mid-height between successive levels.
  - Substitute tapered beams, by a beam with the geometry of the beam's mid-length. For long beams, model the beam as several pieces, each with the cross-sectional geometry at half length.

- Disregard openings in walls.

The following is the example of the inp file format.

```
// ----------------------------------------

~DATA ID_OF_THIS_FILE
    "Adapt_model_exchange.amx"
    "2.00"
~END_DATA ID_OF_THIS_FILE


~DATA NAME_AND_VERSION
    "vista_grande.rvt"
    "March 20, 2006"
    "11:40"
    "Revit"
    "5.00"
~END_DATA NAME_AND_VERSIO


~DATA FOLDER_DIRECTORY
    1000000 "UNITS"
    2000000 "BASIC_STRUCTURAL_INFORMATION"
    3000000 "MATERIALS"
    4000000 "LOAD_COMBINATIONS"
    5000000 "DATA_OF_EACH_LEVEL"
~END_DATA FOLDER_DIRECTORY


    ~DATA UNITS
        10000000 "UNITS"
        "UNITS"                  N-mm
    ~END_DATA UNITS


    ~DATA BASIC_STRUCTURAL_INFORMATION
        20000000 "BASIC_STRUCTURAL_INFORMATION"
        ~DATA CAPSULE_DIRECTORY
            201000 "STRUCTURE_PARAMETERS"
        ~END_DATA CAPSULE_DIRECTORY

            ~DATA STRUCTURE_PARAMETERS
                2010000 "STRUCTURE_PARAMETERS"
                "Number of levels"        3
                //= Level # === Label ======== z-coord ========STORY=====
                //---1-----------2-----------------3--------------4-------
                     3        "LEVEL_3"       1.23456e+00      "STORY2"
                     2        "LEVEL_2"       1.23456e+00      "STORY1"
                     1        "LEVEL_1"       1.23456e+00       "BASE"
                // add or delete as needed
            ~END_DATA STRUCTURE_PARAMETERS
    ~END_DATA BASIC_STRUCTURAL_INFORMATION
```

```
~DATA MATERIALS
    3000000 "MATERIALS"
    ~DATA CAPSULE_DIRECTORY
        3010000 "MATERIAL_PROPERTIES"
    ~END_DATA CAPSULE_DIRECTORY
        ~DATA MATERIAL_PROPERTIES
        3010000 "MATERIAL_PROPERTIES"
                "Number of material types"   2

//Type (1=normal weight; 2=semi-lightweight; 3=lightweight)
//See the bottom of the file for the rest of legend
//==ID===Type=====FC============Wc===========Ec============meu==========alfa============G===========Label========
//--1-----2-------3-------------4------------5-------------6------------7-------------8-------------9----------
    1    2   1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    "Concrete 1"
    2    1   1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    1.23456e-05    "Column_concrete"
        ~END_DATA MATERIAL_PROPERTIES
~END_DATA MATERIALS


~DATA LOAD_COMBINATIONS
    4000000 "LOAD_COMBINATIONS"
    ~DATA CAPSULE_DIRECTORY
        4100000 "COMB_1"
        4200000 "COMB_2"
        // add or delete as needed
    ~END_DATA CAPSULE_DIRECTORY
    ~DATA COMB_1
        4100000 "COMB_1"
//1 = serviceability; 2 = service sustained load 3 = service sustained load 4 = service frequent 5 = service quasi-permanent 6 = strength; 7 =
initial 8 = no //code check
//====Analysis/design_option============Label==========
              6               "Strength requirement"

//===ID=====Load_case========Load_factor====
//---1----------2----------------3----------
    1        "DEAD"            +1.2
    2        "LIVE"            +1.6
    3       "OTHER"            +1.2
    ~END_DATA COMB_1

    ~DATA COMB_2
        4200000 "COMB_2"
//1= serviceability; 2= service sustained load 3= service sustained load 4= service frequent 5= service quasi-//permanent 6 = strength; 7 = initial
8= no code check
//====Analysis/design_option============Label==========
              6                   "Earthquake X"

//===ID====Load_case=======Load_factor====
//---1--------2---------===-----3-----
    1        "DEAD"            +1.4
    2        "LIVE"            +1.6
    3     "Latheral_X"         -1.4
```

```
    ~END_DATA COMB_2
~END_DATA LOAD_COMBINATIONS


~DATA DATA_OF_EACH_LEVEL
    5000000  " DATA_OF_EACH_LEVEL "
    ~DATA CAPSULE_DIRECTORY
        5010000 "LEVEL_1"
        5020000 "LEVEL_2"
        5030000 "LEVEL_3"
        //add or delete as needed, but entries must be sequential
    ~END_DATA CAPSULE_DIRECTORY

        ~DATA LEVEL_1
        5010000 "LEVEL_1"
        ~DATA CAPSULE_DIRECTORY
        5011000 "GEOMETRY"
        5012000 "LOADS"
        ~END_DATA CAPSULE_DIRECTORY

        ~DATA GEOMETRY
        5011000 "GEOMETRY"
        ~DATA CAPSULE_DIRECTORY
            5011100 "BEAM"
            5011200 "COLUMN"
            5011300 "WALL"
            5011400 "SLAB"
            5011500 "DROP_CAP/PANEL"
            5011600 "OPENING"
        ~END_DATA CAPSULE_DIRECTORY

            ~DATA BEAM
            5011100 "BEAM"
//X1,Y1,X2,Y2 = cords of end of beam;  width, depth = stem of beam to top of slab
//offset is distance from top of beam stem to reference line.
//-------X1--------------Y1------------X2-------------Y2-------------WIDTH----------DEPTH----------OFFSET-------MATRIAL_ID------LABEL------
//-------1---------------2-------------3--------------4----------------5--------------6--------------7--------------8-------------9--------
   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00       1         "Beam1"
   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00       1         "Deep beam"
        ~END_DATA BEAM

            ~DATA COLUMN
            5011200 "COLUMN"
//bottom columns
//Flag (1 = top column, 2 = bottom column), Type (1 = rectangular column, 2 = circular column)
//-Flag--- TYPE -----X1--------------Y1----------THETA----------WIDTH--------DEPTH------ ----OFFSET_TOP----OFFSET_BOTT----MATRIAL_ID-----LABEL-----
//--1-------2---------3--------------4-------------5--------------6------------7-----------------8--------------9-------------10----------11-------

//top columns
//Flag (1=top column, 2=bottom column), Type (1=rectangular column, 2=circular column)
//-Flag- TYPE -----X1--------------Y1----------THETA----------WIDTH--------DEPTH------ ----OFFSET_TOP----OFFSET_BOTT----MATRIAL_ID-----LABEL-----
//--1------2-------3--------------4-------------5--------------6------------7---------------8--------------9-------------10----------11-------
    1    1    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00        1        "Column1"
    1    1    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00        1        "C2"
```

```
    1    1    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00        1        "Column3"
        ~END_DATA COLUMN

            ~DATA WALL
            5011300 "WALL"
//bottom walls
//Flag (1=top wall, 2=bottom wall)
//-Flag-------X1--------------Y1------------X2------------Y2----------THICKNESS-----OFFSET_TOP-----OFFSET_BOTT----MATRIAL_ID-----LABEL------
//--1---------2---------------3-------------4-------------5--------------6--------------7---------------8-------------9-----------10--------

//top walls
//Flag (1=top wall, 2=bottom wall)
//-Flag-------X1--------------Y1------------X2------------Y2----------THICKNESS-----OFFSET_TOP-----OFFSET_BOTT----MATRIAL_ID-----LABEL------
//--1---------2---------------3-------------4-------------5--------------6--------------7---------------8-------------9-----------10--------
    1    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00        1        "Column1"
        ~END_DATA WALL

            ~DATA SLAB
            5011400 "SLAB"
            ~DATA CAPSULE_DIRECTORY
                5011410 "SLAB_PARAMETERS"
                5011420 "SLAB_GEOMETRY"
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA SLAB_PARAMETERS
                5011410 "SLAB_PARAMETERS"
                        "Number of slabs"    2
                ~END_DATA SLAB_PARAMETERS

                ~DATA SLAB_GEOMETRY
                5011420 "SLAB_GEOMETRY"
                ~DATA CAPSULE_DIRECTORY
                    5011421 "SLAB_1"
                    5011422 "SLAB_2"
                    // add or delete as necessary, but remain sequential
                ~END_DATA CAPSULE_DIRECTORY

                    ~DATA SLAB_1
                    5011421   "SLAB_1"
//---NUMBER_OF_POINTS-----THICKNESS-------OFFSET-------MATERIAL_ID-------LABEL------
//-----------1---------------2---------------3-------------4--------------5--------
            3          +1.23456e+00    +1.23456e+00        1        "Slab_Region1"

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
         1      +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00
                    ~END_DATA SLAB_1

                    ~DATA SLAB_2
                    5011422   "SLAB_2"
//---NUMBER_OF_POINTS-----THICKNESS-------OFFSET-------MATERIAL_ID-------LABEL------
//-----------1---------------2---------------3-------------4--------------5--------
```

```
            4          +1.23456e+00    +1.23456e+00        1       "Slab_Region2"

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
         1      +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00
                     ~END_DATA SLAB_2
                ~END_DATA SLAB_GEOMETRY
        ~END_DATA SLAB

        ~DATA DROP_CAP/PANEL
        5011500 "DROP_CAP/PANEL"
//Thickness = stem of drop cap/panel to top of slab
//-----X1-------------Y1-----------WIDTH---------LENGTH-------THICKNESS--------TETA---------OFFSET--------MATERIAL_ID-----LABEL-----
//-----1--------------2--------------3--------------4-------------5--------------6-------------7---------------8------------9-------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00         1       "Drop_Cap 1"
        ~END_DATA DROP_CAP/PANEL

            ~DATA OPENING
            5011600 "OPENING"
            ~DATA CAPSULE_DIRECTORY
                5011610 "OPENING_PARAMETERS"
                5011620 "OPENING_GEOMETRY"
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA OPENING_PARAMETERS
                5011610 "OPENING_PARAMETERS"
                          "Number of openings"    1
                ~END_DATA OPENING_PARAMETERS

                ~DATA OPENING_GEOMETRY
                5011620 "OPENING_GEOMETRY"
                ~DATA CAPSULE_DIRECTORY
                    5011621 "OPENING_1"
                ~END_DATA CAPSULE_DIRECTORY

                    ~DATA OPENING_1
                    5011621 "OPENING_1"
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------
            4          "Opening_1"

//-----POINT----------X--------------Y--------
//-------1------------2--------------3--------
         1      +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00
                     ~END_DATA OPENING_1
                ~END_DATA OPENING_GEOMETRY
            ~END_DATA OPENING
    ~END_DATA GEOMETRY
```

```
    ~DATA LOADS
    5012000  "LOADS"
    ~DATA CAPSULE_DIRECTORY
        5012100 "DEAD"
        5012200 "LIVE"
        5012300 "OTHER"
        5012400 "Lateral_EQx"
        //add or delete as needed
    ~END_DATA CAPSULE_DIRECTORY

        ~DATA DEAD
        5012100 "DEAD"
            ~DATA POINT_LOAD
            "POINT_LOAD"
//-----X1------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy----------LABEL-------
//-----1-------------2--------------3--------------4-------------5--------------6-------------7---------------8-------------9---------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_1
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_2
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_3
            ~END_DATA POINT_LOAD

            ~DATA LINE_LOAD
            "LINE_LOAD"
//-----X1------------Y1-------------Z1-------------X2------------Y2------------Z2--------------Fx1------------Fx2-----------Fy1------------Fy2-------
//-----1-------------2--------------3--------------4-------------5-------------6---------------7--------------8-------------9-------------10--------
//-----Fz1-----------Fz2------------Mx1------------Mx2-----------My1-----------My2------------LABEL------
//-----11------------12-------------13-------------14------------15------------16-------------17--------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_1"
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_2"
            ~END_DATA LINE_LOAD

            ~DATA AREA_LOAD
            "AREA_LOAD"
            ~DATA CAPSULE_DIRECTORY
                5012110 "AREA_LOAD1"
                5012120 "AREA_LOAD2"
                //add or delete as needed
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA AREA_LOAD1
                5012110 "AREA LOAD1"
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------
            4          "Area Load 1"

//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00
```

```
//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2---------------3--------------4--------------5---------------6-------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
                 ~END_DATA AREA_LOAD1

                 ~DATA AREA LOAD2
                 5012120 "AREA_LOAD2"
//-----POINT----------X-------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2---------------3--------------4--------------5---------------6-------
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
                 ~END_DATA AREA_LOAD2
             ~END_DATA AREA_LOAD
        ~END_DATA DEAD

        ~DATA LIVE
        5012200 "LIVE"
             ~DATA POINT_LOAD
             "POINT_LOAD"
//-----X1-------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy----------LABEL-------
//-----1--------------2--------------3--------------4-------------5--------------6-------------7---------------8-------------9---------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_1
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_2
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_3
             ~END_DATA POINT_LOAD

             ~DATA LINE_LOAD
             "LINE_LOAD"
//-----X1-------------Y1-------------Z1-------------X2------------Y2------------Z2--------------Fx1------------Fx2-----------Fy1------------Fy2-------
//-----1--------------2--------------3--------------4-------------5-------------6----------------7--------------8-------------9-------------10--------
//-----Fz1------------Fz2------------Mx1------------Mx2-----------My1-----------My2------------LABEL------
//-----11-------------12-------------13-------------14------------15------------16-------------17--------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_1"
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_2"
             ~END_DATA LINE_LOAD

             ~DATA AREA_LOAD
             "AREA_LOAD"
             ~DATA CAPSULE_DIRECTORY
                 5012110 "AREA_LOAD1"
```

```
                5012120 "AREA_LOAD2"
                //add or delete as needed
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA AREA_LOAD1
                5012110 "AREA LOAD1"
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------
            4          "Area Load 1"

//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2---------------3--------------4--------------5---------------6-------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
                ~END_DATA AREA_LOAD1

                ~DATA AREA LOAD2
                5012120 "AREA_LOAD2"
//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2---------------3--------------4--------------5---------------6-------
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
                ~END_DATA AREA_LOAD2
            ~END_DATA AREA_LOAD
        ~END_DATA LIVE

        ~DATA OTHER
        5012300 "OTHER"
            ~DATA POINT_LOAD
            "POINT_LOAD"
//-----X1-------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy----------LABEL-------
//-----1--------------2--------------3--------------4-------------5--------------6-------------7---------------8-------------9---------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_1
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_2
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   Point_load_3
            ~END_DATA POINT_LOAD
```

```
            ~DATA LINE_LOAD
            "LINE_LOAD"
//-----X1-----------Y1-------------Z1-------------X2------------Y2------------Z2--------------Fx1------------Fx2-----------Fy1------------Fy2-------
//-----1------------2--------------3--------------4-------------5-------------6----------------7--------------8-------------9-------------10--------
//-----Fz1----------Fz2------------Mx1------------Mx2-----------My1-----------My2------------LABEL------
//-----11-----------12-------------13-------------14------------15------------16-------------17--------
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_1"
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00
  +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   "Line_load_2"
            ~END_DATA LINE_LOAD

            ~DATA AREA_LOAD
            "AREA_LOAD"
            ~DATA CAPSULE_DIRECTORY
                5012110 "AREA_LOAD1"
                5012120 "AREA_LOAD2"
                //add or delete as needed
            ~END_DATA CAPSULE_DIRECTORY

                ~DATA AREA_LOAD1
                5012110 "AREA LOAD1"
//---NUMBER_OF_POINTS------LABEL------
//-----------1---------------2---------
            4          "Area Load 1"

//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2--------------3--------------4--------------5---------------6-------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
                 ~END_DATA AREA_LOAD1

                 ~DATA AREA LOAD2
                 5012120 "AREA_LOAD2"
//-----POINT----------X--------------Y---------------Z--------
//-------1------------2--------------3---------------4--------
         1      +1.23456e+00    +1.23456e+00    +1.23456e+00
         2      +1.23456e+00    +1.23456e+00    +1.23456e+00
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00
         4      +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00

//-----POINT----------Fx--------------Fy-------------Fz-------------Mxx-------------Myy------
//-------1------------2--------------3--------------4--------------5---------------6-------
         3      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
         5      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
```

```
 1      +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00    +1.23456e+00
          ~END_DATA AREA_LOAD2
      ~END_DATA AREA_LOAD
    ~END_DATA OTHER

    ~DATA Lateral_EQ
    5012400 "Lateral_EQ"
      ~DATA POINT_LOAD
      "POINT_LOAD"
//At far ends of UPPER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1-------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy-------------Mzz---------LABEL-
------
//-1--------2--------------3--------------4--------------5-------------6--------------7--------------8--------------9--------------10-----------11---
------
   2   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    Column_1
   2   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    Column_2
   2   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00    Column_3

//At far ends of LOWER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1-------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy-------------Mzz---------LABEL-
------
//-1--------2--------------3--------------4--------------5-------------6--------------7--------------8--------------9--------------10-----------11---
------

//Story Shear Forces
//Flag (2 = upper columns, 1 = story shear, 0 = lower columns)
//-Flag-----X1-------------Y1-------------Z1-------------Fx------------Fy-------------Fz------------Mxx------------Myy-------------Mzz---------LABEL-
------
//-1--------2--------------3--------------4--------------5-------------6--------------7--------------8--------------9--------------10-----------11---
------
      ~END_DATA POINT_LOAD

      ~DATA LINE_LOAD
      "LINE_LOAD"
//At far ends of UPPER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1-------------Y1-------------Z1------------X2------------Y2-------------Z2-------------Fx-------------FY------------FZ------------Mxx---
-----
//--1-------2--------------3--------------4-------------5-------------6----------------7-------------8-------------9-------------10------------11----
-----
//-----Myy------------Mzz------------LABEL-----
//-----12-------------13-------------14-------
   2   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00
   2   +1.23456e+00   +1.23456e+00   "Pier_1"
   2   +1.23456e+00   +1.23456e+00   +1.23456e+00    +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00   +1.23456e+00
       +1.23456e+00   +1.23456e+00   "Pier_2"

//At far ends of LOWER supports
//Flag (2 = upper supports, 1 = story shear, 0 = lower supports)
//-Flag-----X1-------------Y1-------------Z1------------X2------------Y2-------------Z2-------------Fx-------------FY------------FZ------------Mxx---
-----
```

```
//--1-------2--------------3--------------4-------------5-------------6----------------7-------------8-------------9-------------10------------11----
-----
//-----Myy-----------Mzz-----------LABEL-----
//-----12------------13-------------14-------
            ~END_DATA LINE_LOAD
         ~END_DATA Lateral_EQ
    ~END_DATA LOADS

  ~DATA REBAR_DISPLAY_PROPERTIES
      ~DATA REBAR_COLOR
          1         R=87 G=171 B=115
          2         R=171 G=0 B=0
      ~END_DATA REBAR_COLOR

      ~DATA REBAR_LINE_TYPE
          1        "CONTINUOUS"
          2        "Dashed"
      ~END_DATA REBAR_LINE_TYPE

      ~DATA REBAR_LINE_THICKNESS
          1        3
          2        1
      ~END_DATA REBAR LINE_THICKNESS

      ~DATA REBAR_DISPLAY_TEXTURE
      ~END_DATA REBAR_DISPLAY_TEXTURE
  ~END_DATA REBAR_DISPLAY_PROPERTIES

  ~DATA REBAR_DRAWING_GRAPHICS
      ~DATA REBAR_CALL_OUT_GRAPHICS
          "line_thickness"            3
          "line_color"      R=87 G=171 B=115
          "arrow_size"                        10
          "circle_size"               17
      ~END_DATA REBAR_CALL_OUT_GRAPHICS

      ~DATA REBAR_TEXT
          "Font_size"              12
          "Font_type"                 "STANDARD"
          "Text_color"                R=87 G=171 B=115
      ~END_DATA REBAR_TEXT

  ~END_DATA REBAR_DRAWING_GRAPHICS

  ~DATA REBAR_GRAPHICAL_DISPLAY
  //-Mark--Color--LineType-LineThickness-TextThickness----Texture---
  //-1------2-------3---------4----------5---------------6------

  ~END_DATA REBAR_GRAPHICAL_DISPLAY

  ~DATA REBAR_DISPOSITION
          "Single_bar"            1
          "Bundled_bars"          2
          "Grouped"               3
```

```
        "Distributed"           4
~END_DATA REBAR_DISPOSITION

~DATA REBAR_MARK_AND_PROPERTIES
// 1 - Rebar "Mark" determined by ADAPT, starting by 1 and increasing numerically
// 2 - Rebar Material number from the material library
// 3 - Rebar source, such as from computation, or detailing
// 4 - Group, which group the bar belongs to
// 5 - Rebar associate member type, such as beam, slab, drop cap
// 6 - Rebar associate member ID, such as beam number
// 7 - Cross sectional area
// 8 - diameter
// 9 - cover
// 10 - weight density in kg/mm
// 11 - Rebar size from previous data blocks
// 12 - User defined mark on the drawing
//                         <-Associate->
//-Mark-Mat--source-group---Type-ID------Area---Diameter----Cover---density--visibility---size----user mark-
//-1----2------3-----4------5----6---------7----------8-------9---------10---------11------12-------13

~END_DATA REBAR_MARK_AND_PROPERTIES

~DATA RBAR_SHAPE_AND_NUMBER
// 1 - Rebar "Mark" determined by ADAPT, starting by 1 and increasing numerically
// 2 - Rebar number of times it is repeated in the drawings
// 3 - Rebar length
// 4 - Rebar shape number as defined above
// 5 - Rebar length A
// 6 - Rebar length B
// 7 - Rebar length C
// 8 - Rebar length D
// 9 - Rebar length E
//10 - or beyond other lengths, if applicable
// last string user defined annotation for the specific bar
//Mark-Number--length---shape-----A----------B--------C----------D-----------E-----annotation
//-1-----2-------3--------4-------5----------6--------7----------8-----------9----------10------

~END_DATA REBAR_SHAPE_AND_NUMBER

~DATA REBAR_LOCATION
    ~DATA REBAR_GLOBAL_LOCATION
          //top bar = 2, bottom bar = 1
    // Xi,Yi,Zi = global coordinates of the first hot point of the bar
    // Xj,Yj,Zj = global coordinates of the last hot point of the bar
    // Xm,Ym,Zm = global coordinates of the positioning hot point of the bar
    // Mark top/bottom < -------GLOBAL START----------->  < -------GLOBAL END--------->   <------GLOBAR THIRD HOT POINT---->
    //--1--2---Xi--------Yi----------Zi----------Xj---------Yj---------Zj--------Xm---------Ym---------Zm------

    ~END_DATA REBAR_GLOBAL_LOCATION

    ~DATA REBAR_LOCAL_LOCATION
    // TYPE is like "beam, crop cap, slab" and is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52" refers to a specific beam
    // xi,yi,zi = local coordinates of the first hot point of the bar
```

```
    // xj,yj,zj = local coordinates of the last hot point of the bar
    // xm,ym,zm = local coordinates of the positioning hot point of the bar
    // Mark            < -------LOCAL START----------->  < -----------LOCAL END--------->   <------LOCAL THIRD HOT POINT---->
    //--1---TYPE---ID-----xi--------yi----------zi----------xj---------yj---------zj--------xm---------ym---------zm------
    ~END_DATA REBAR_LOCAL_LOCATION

    ~DATA REBAR_DISPOSITION_LEGEND
    // TYPE is like "beam, crop cap, slab" is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52" refers to a specific beam
    // Number = number of bars with given Mark
    // Disposition
    //    1 = single bars
    //    2 = bundled bars
    //    3 = grouped bars
    //    4 = distributed bars
    //-Mark-TYPE---ID---Number-----Dispositon--

    ~END_DATA REBAR_DISPOSITION_LEGEND
~END_DATA REBAR_LOCATION
~DATA REBAR_LAYOUT
// This data block lists only the bars that are distributed.
    ~DATA GLOBAL_EXTENT_OF_DISTRIBUTED_BARS
    // Xi,Yi,Zi = global coordinates of the first end of the rebar range, (closer to global origin)
    // Xj,Yj,Zj = global coordinates of the second point of the rebar range
    // distribute=1 show bars individually. Do not show the extent call out
    // distribute=0 show a single bar with the range line
    // adjust = 1 truncate the bar, if when distributed can falls outside a slab region or enters an opening
    // adjust = 0 do not adjust the bar length. Show and annotate them as given in input data
    // Mark < -------GLOBAL START----------->  < -------GLOBAL END--------->
    //--1-----Xi--------Yi----------Zi----------Xj---------Yj---------Zj------distribute-adjust--

    ~END_DATA GLOBAL_EXTENT_OF_DISTRIBUTED_BARS

    ~DATA LOCAL_EXTENT_OF_DISTRIBUTED_BARS
    // TYPE is like "beam, crop cap, slab" and is defined in earlier blocks
    // ID refer to identification of the type, for example beam 52. "beam" is the type and "52" refers to a specific beam
    // xi,yi,zi = local coordinates of the first hot point of the bar (closest to the origin)
    // xj,yj,zj = local coordinates of the last hot point of the bar
    // distribute=1 show bars individually. Do not show the extent call out
    // distribute=0 show a single bar with the range line
    // adjust = 1 truncate the bar, if when distributed can falls outside a slab region or enters an opening
    // adjust = 0 do not adjust the bar length. Show and annotate them as given in input data
    // Mark            < -------LOCAL START----------->  < -----------LOCAL END--------->
    //--1---TYPE---ID-----xi--------yi----------zi----------xj---------yj---------zj------distribute-adjust-
    ~END_DATA REBAR_LOCAL_LOCATION
    ~END_DATA REBAR_LAYOUT

~DATA MESH_REINFORCEMENT
~DATA MESH_GRAPHICAL_DISPLAY
//-Mark--Color--LineType-LineThickness-TextThickness----Texture---
//-1------2-------3---------4----------5---------------6------

~END_DATA MESH_GRAPHICAL_DISPLAY
~DATA MESH_ID_AND_SIZE_BOTTOM
```

```
      // Mark = ADAPT defined sequential number
      // Material ID
      // Area = cross-sectional area in mm2/mm
      // Diameter
      // size = bar size as given in rebar library
      // CGS = centroid distance of bar from bottom surface of member in mm
      //angle = measured  counterclockwise from positive X to rebar direction
      // Mark AssociateType AssociateID< ------------first bar in mesh---------------------->< ---------second bar in mesh--------------------------------
      ---->
      //--Mark-MatID--Type---ID---Area----Diameter---size-------CGS--------angle----spacing------area---Diameter----size-------CGS--------angle----
      spacing---user mark-
      ~END_DATA MESH_ID_AND_SIZE_BOTTOM

      ~DATA MESH_ID_AND_SIZE_TOP
      // Mark AssociateType AssociateID< ------------first bar in mesh---------------------->< ---------second bar in mesh--------------------------------
      ---->
      //--Mark--MatID--Type---ID---Area----Diameter---size-------CGS--------angle----spacing------area---Diameter----size-------CGS--------angle----
      spacing---user mark-

      ~END_DATA MESH_ID_AND_SIZE_TOP

      ~DATA MESH_LOCATION
      //xi,yi = coordinates of the first hot point of the bar
      //-MARK---Vertex------Xi---------Yi------

         ~END_DATA MESH_LOCATION
   ~END_DATA LEVEL_1


~END_DATA DATA_OF_EACH_LEVEL
~END_DATA FOLDER_DIRECTORY
~END_OF_FILE

//  LEGEND:
//  - Flag at the beginning of the column/wall block is: 0 for bottom columns/walls
//                                                       1 for top columns/walls
//
//  - Flag in front of the offset in the column/wall block is: 0 for offset from the bottom
//                                                             1 for offset from the top
//
//  - TETA is the angle between global and local axis. Angle is positive in clockwise direction.
//===ID====lightweight?====FC========Wc============Ec==========meu===========alfa ============G==========Label=====
// Fc = cylinder strength at 28 days. If user specified cube strength (fcu), enter here (0.8fcu)
// Wc = unit weight of concrete acting as force on the structure in N/mm3
// Ec = modulus of elasticity
// meu = Poisson's ratio
// alfa = coefficient of thermal expansion - not used in the first release of the program, but enter a
// value
// G = shear modulus
// Label = name given by the user or the program to the material
// Offset = distance from a reference plane (typically top of slab) to the top of the structural
// component, such as a slab, or beam
// The position of a reference plane is arbitrary and determined by the user.
```

```
// Label of each structural component must be unique. That is to say, two beams cannot be given the same // name. Name is a string that the program uses
to identify // the members
// Flag = for columns,  0=bottom column   1=top column
// Flag at the beginning of the Lateral Point/line load block is:
// 0 = bottom of lower columns/walls
// 1 = story shear (current level)
// 2 = top of upper columns/walls
```

APPENDIX A

A1 NODE CONSOLIDATION 263
A1.1 Overview 263
A1.2 Consolidation of Analysis Nodes 264
A2 MESHING USING EXCLUDER 274
A3 VIEW FINITE ELEMENT NODE AND CELL INFORMATION 277
A4 FINITE ELEMENT CELL SIZE AND MESH DENSITY 279
A4.1 Maximum Mesh Size 279

A1 NODE CONSOLIDATION

A1.1 Overview

The structural model is made up of structural components, such as beams, columns, slab regions, drop caps and more. Each of these components is meshed (subdivided into elements) on its own with the provision that there will be common analysis nodes at the intersection, or a connection of each structural component with those adjoining it.

As they are created in the program, each of the structural components is positioned with respect to the current reference plane as shown in **Fig. A1.1-1**. Note that the "analysis node" of the structural component is positioned on the reference plane immediately above the associated "natural node." The natural node is typically at the centroid of each structural component, whereas the analysis node is the location at which the response of the element is formulated. The program solves its equations at analysis nodes. Once a solution is obtained, it is mapped back to the natural nodes, before reporting it to you. As a result, the presence of analysis nodes will be transparent to you.

Strictly speaking, the program is organized such that the existence of both analysis and natural nodes are transparent to the user. All matters related to these items are handled automatically in the background by the program. While the creation of analysis nodes, and the mapping of the natural nodes onto analysis nodes, is carried out by the program in practically all cases, the consolidation of the analysis nodes is an optional function. The default setting of the program is to activate this option with a value 1 to 3 times the slab thickness. However, you may choose to cancel or change it.

This section describes in greater detail the concept of the dual nodes (analysis and natural), as well as the consolidation of the analysis nodes.

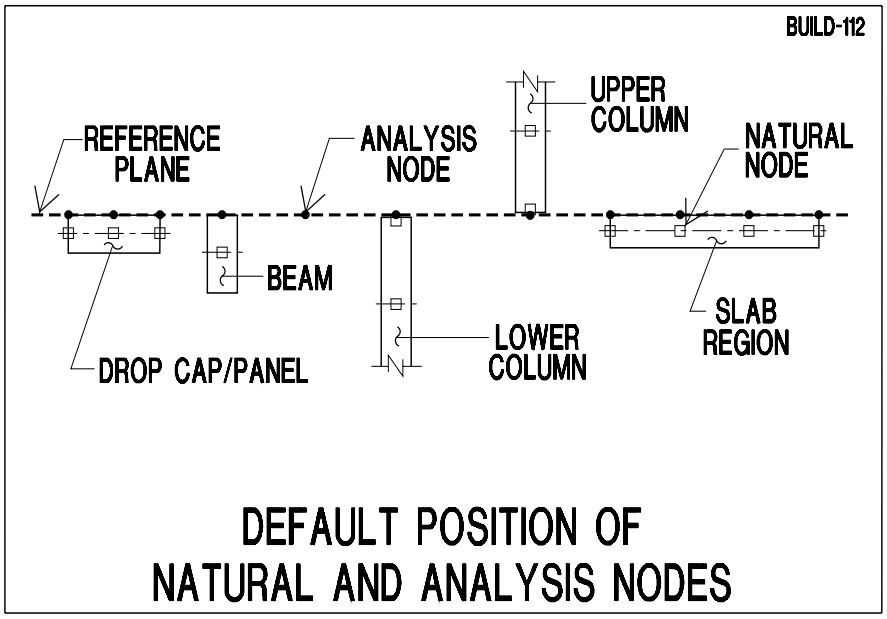


FIGURE A1.1-1

A1.2 Consolidation of Analysis Nodes

Consider the wall-supported slab shown in **Fig. A1.2-1.** The gap between the two walls, as shown in part (b) of the figure, is about the same as the slab thickness. For a gap of up to approximately four times the slab thickness, the behavior of the slab region spanning the gap is significantly different from the assumption of "plane sections remain plain," which is a key aspect of ADAPT's software algorithm and essentially all other linear elastic finite element analysis software. Such regions are referred to as discontinuities (**Fig. A1.2-2b**). The smaller the gap, the closer is the response of the region to a solid volume. Normally, natural nodes are placed at the corners of the walls together with natural lines as shown in parts (c) and (d) of the figure.

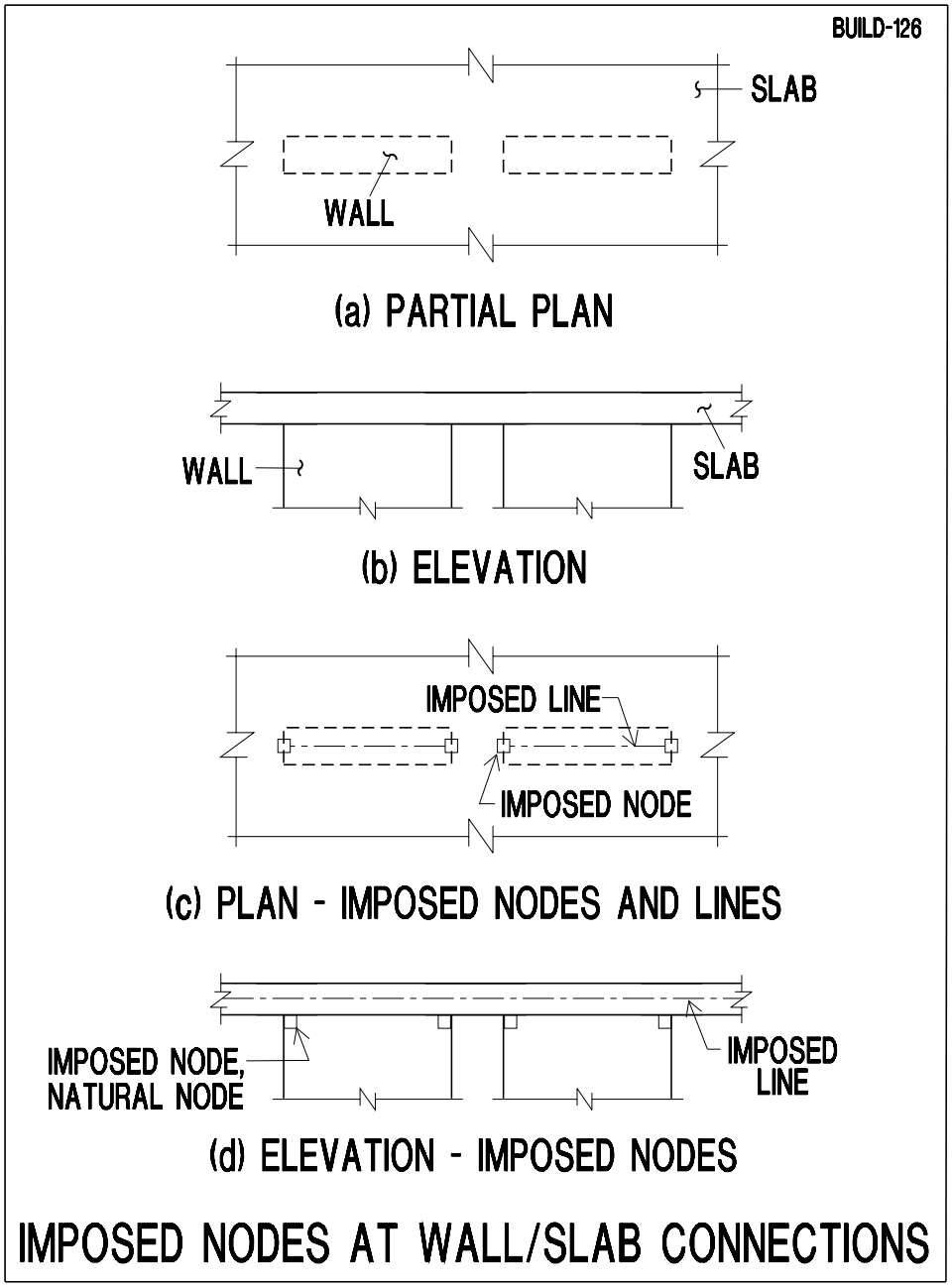


FIGURE A1.2-1

It is expedient and safe, from a structural design standpoint, if the natural nodes of the members terminating at such regions are consolidated into a single analysis node, such as node "A" shown in part (c) of **Fig. A1.2-2.** Node "A" will be linked to the natural nodes 1, 2 and 3 by way of three offsets. The consequence of this modeling is that the three points 1, 2 and 3 will be acting as if rigidly connected together. The conditions of equilibrium with the applied loading necessary for the safety of the structure are maintained. The approximation affects the distribution of

stresses within the "D" region. The analysis, however, will conclude with each of the three natural nodes having its own displacement and forces.

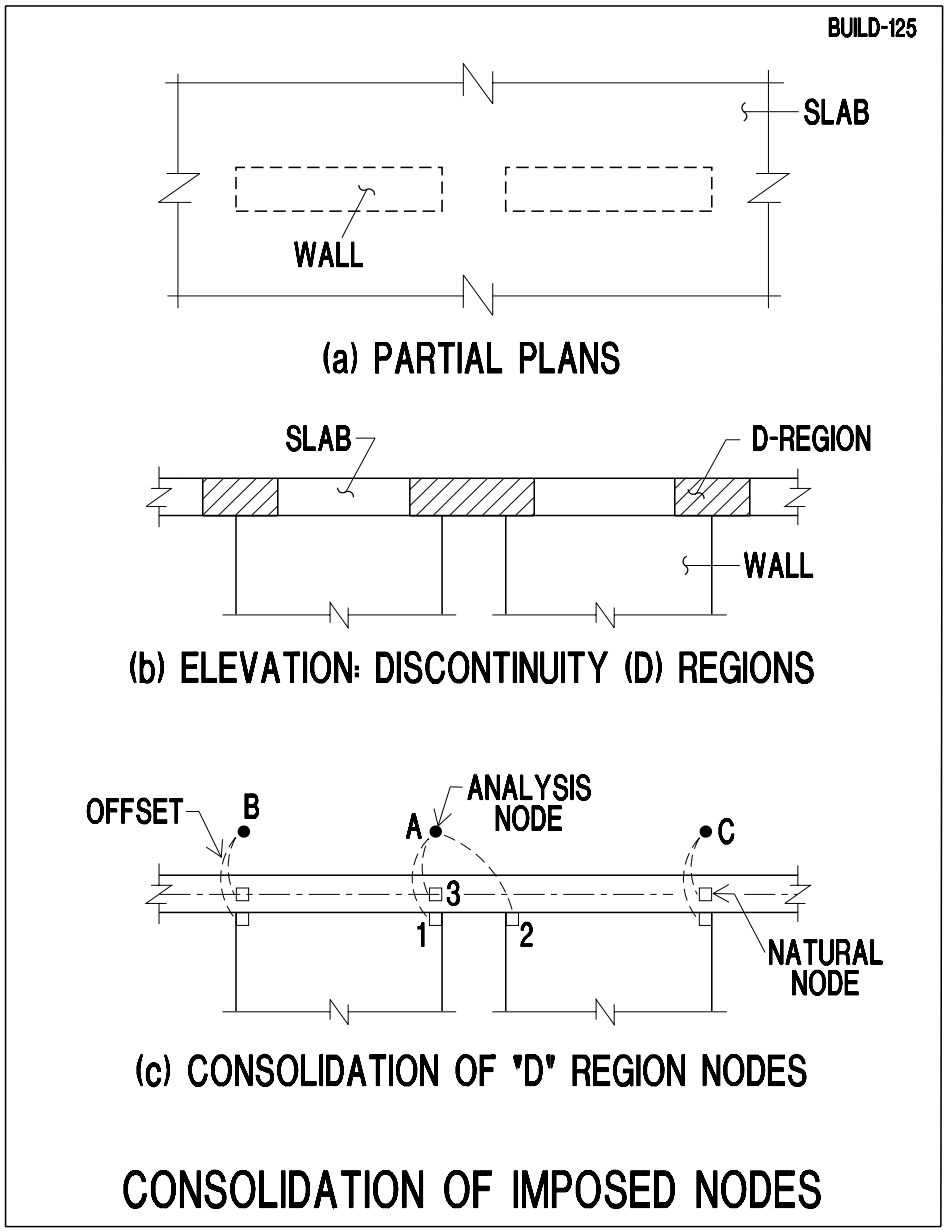


FIGURE A1.2-2

Structural design is based on the magnitude of the actions (moments, shears, axial loading) at selected "design sections." In ADAPT's formulation, contrary to traditional FEM [Aalami, Dec. 2001], the design actions at a section are not derived from the distribution of internal stresses. The design actions are based on the forces at the nodes adjacent to the section. The consolidation of nodes at a discontinuity region leads to

an expedient and safe design. The regions are designed to satisfy the equilibrium of forces and detailed for ductility.

In **Fig. A1.2-2(c)**, the position of node 2 with respect to its analysis node is shifted both horizontally and vertically. The horizontal shift is referred to as "Node Shift." The node shift is generally done automatically by the program. You have the option to shift nodes manually and consolidate several natural nodes of a discontinuity region into a single analysis node, or to suppress node shift altogether. The automatic shift node of the program takes place when two or more "imposed" natural nodes fall closer to one another than the minimum cell size you define in the automatic meshing option. This distance is generally two to four times the slab thickness.

One of the great advantages of node consolidation is the substantial reduction in the number of nodes necessary to complete a design. Clusters of closely spaced imposed nodes result in a dense meshing around them and subsequently a greater demand on the computational resources with no significant increase in the accuracy of the design values.

As an example, consider the partial plan of slab shown in **Fig. A1.2-3(a).**The plan illustrates a closely spaced wall, beam and column. The natural imposed nodes of each are shown in part (b) of the figure. The region bound by the closely spaced components acts essentially as a chunk of solid and rigid concrete. The shift node option of the program would represent the region by a single analysis node as shown in part (c) of the figure. It is important to note that while the three natural nodes shown are consolidated into a single analysis node, the program will automatically determine and report the displacement and actions at each of the natural nodes separately. This determination is based on satisfying the equilibrium of the forces within the region common to the natural nodes consolidated and considering the deformation of the region as a rigid chunk of concrete.

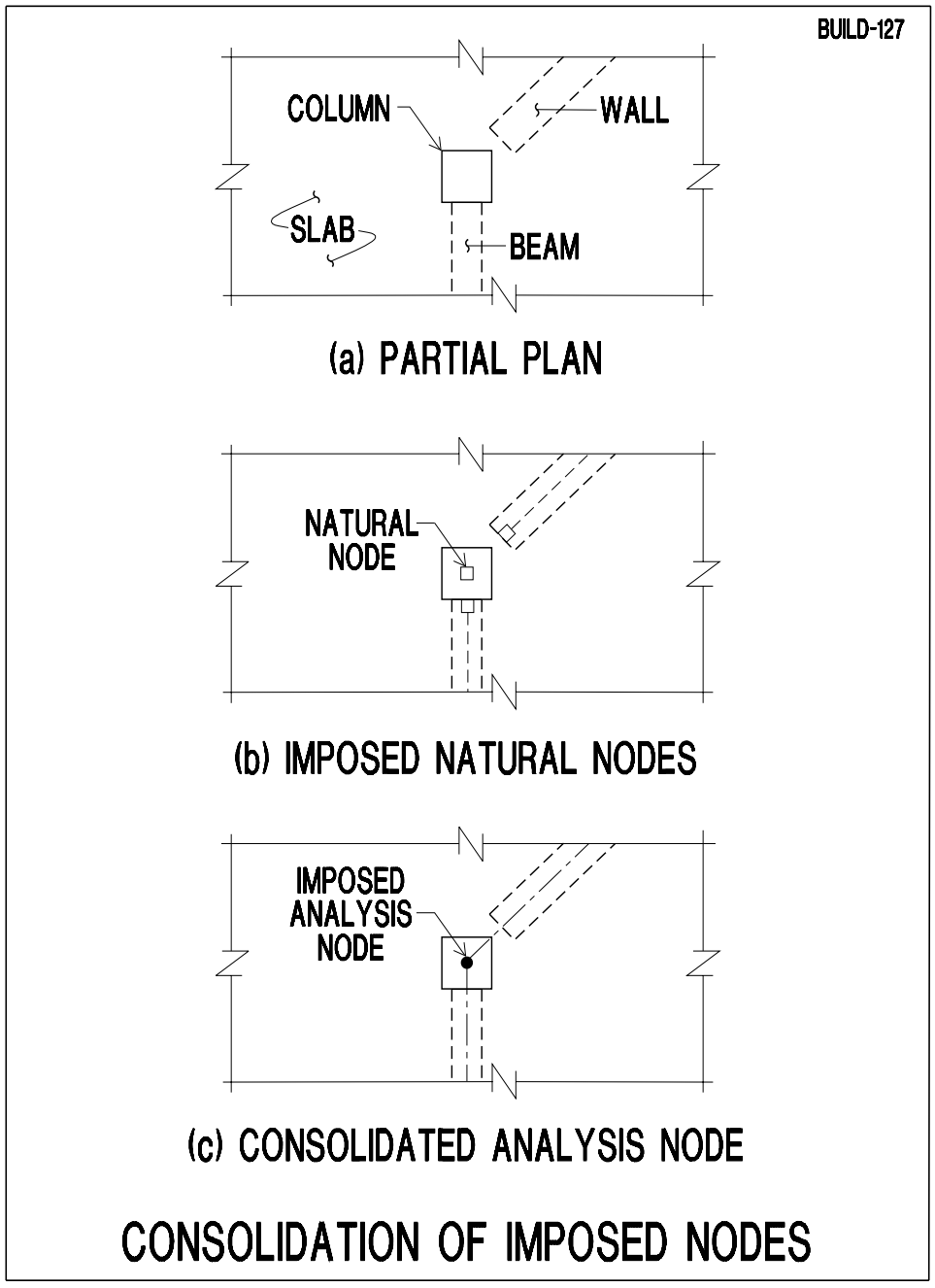


FIGURE A1.2-3

A common practical scenario of node consolidation is the condition of walls terminating at the slab edge. **Figure A1.2-4** illustrates the traditional FEM modeling (part b). The alternative available in ADAPT-Floor Pro is shown in part (c) of the figure.

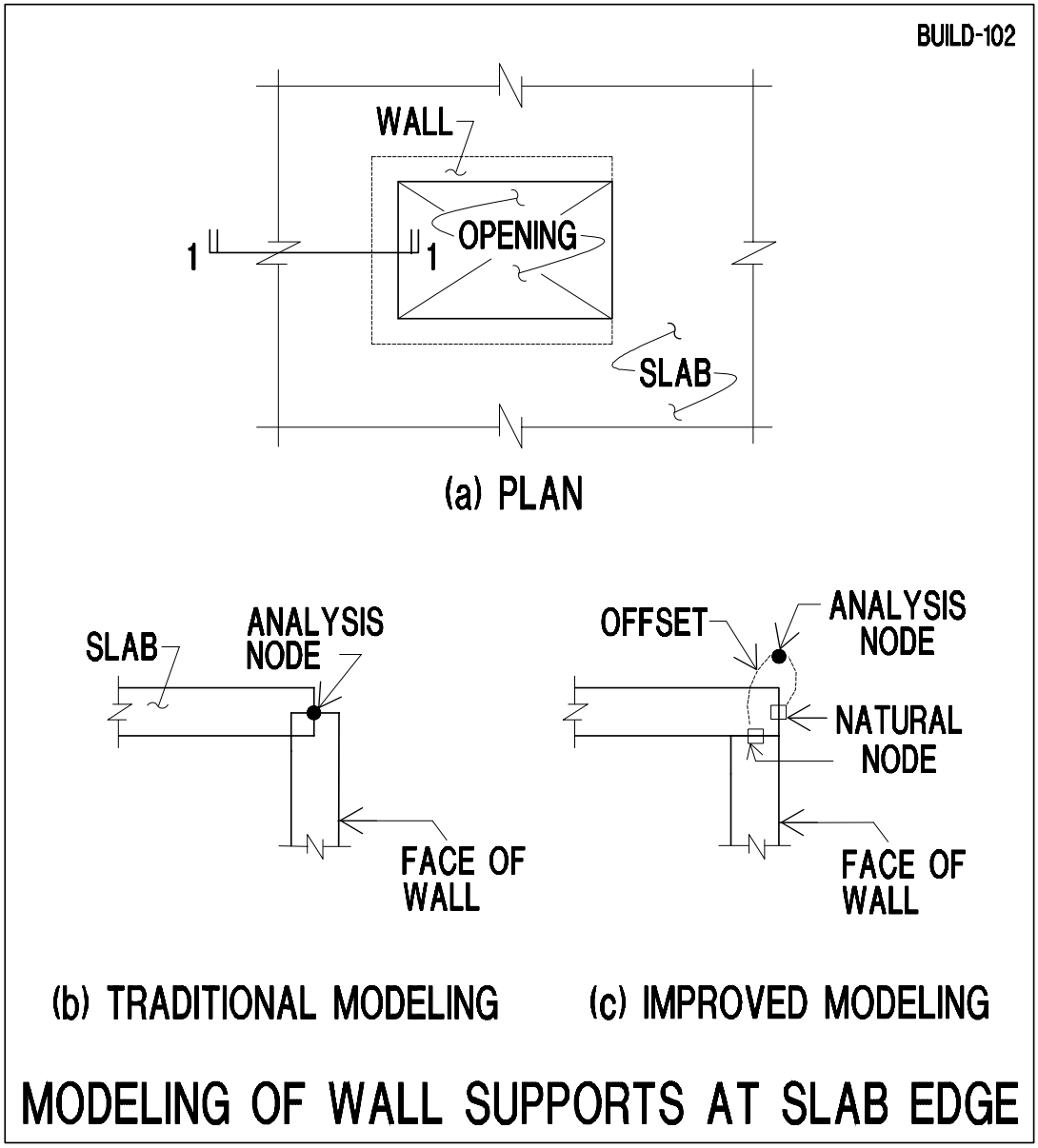


FIGURE A1.2-4

The common cases of node consolidation that are available to be performed automatically by the program are listed in **Figs. A1.2-5A, A1.2-5B,** and **A1.2-5C.**

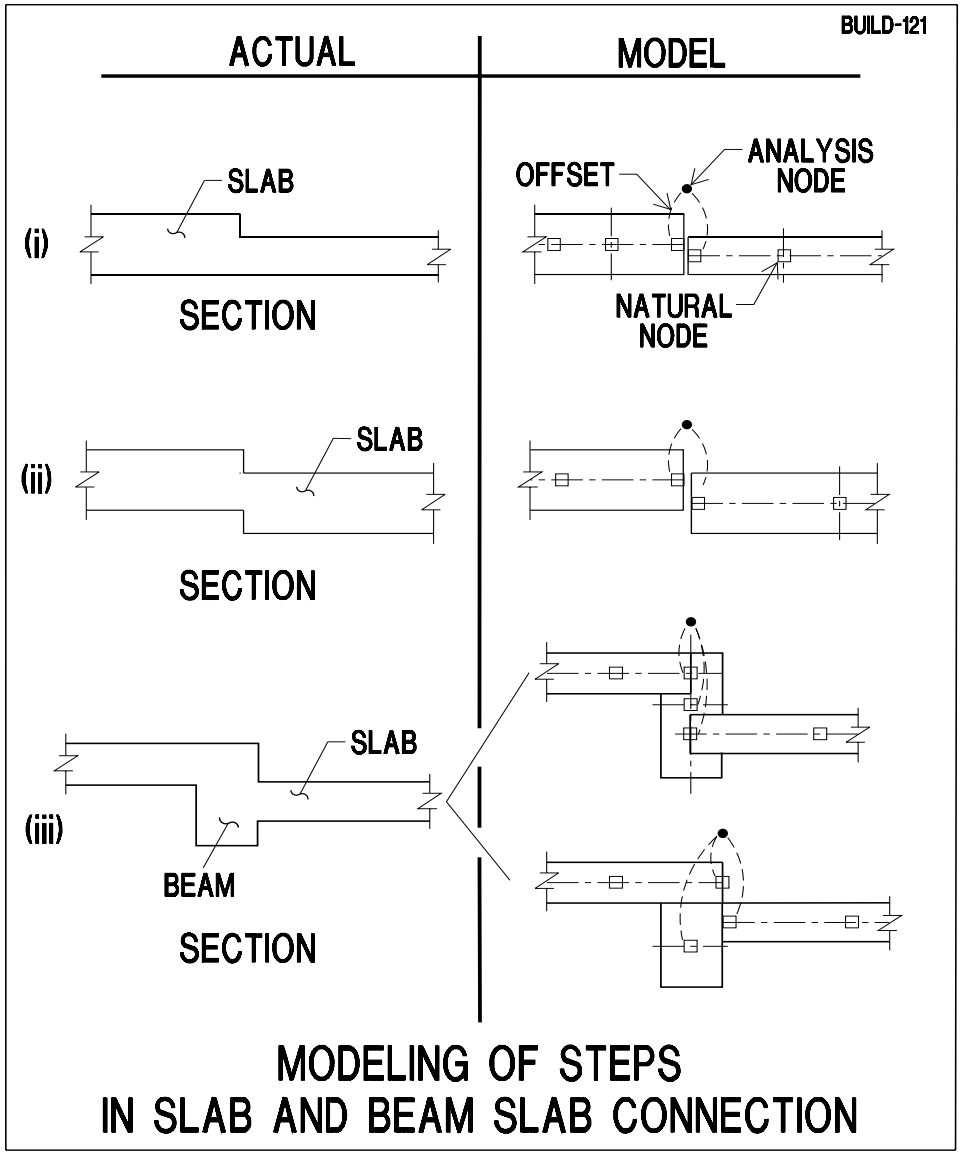


FIGURE A1.2-5A

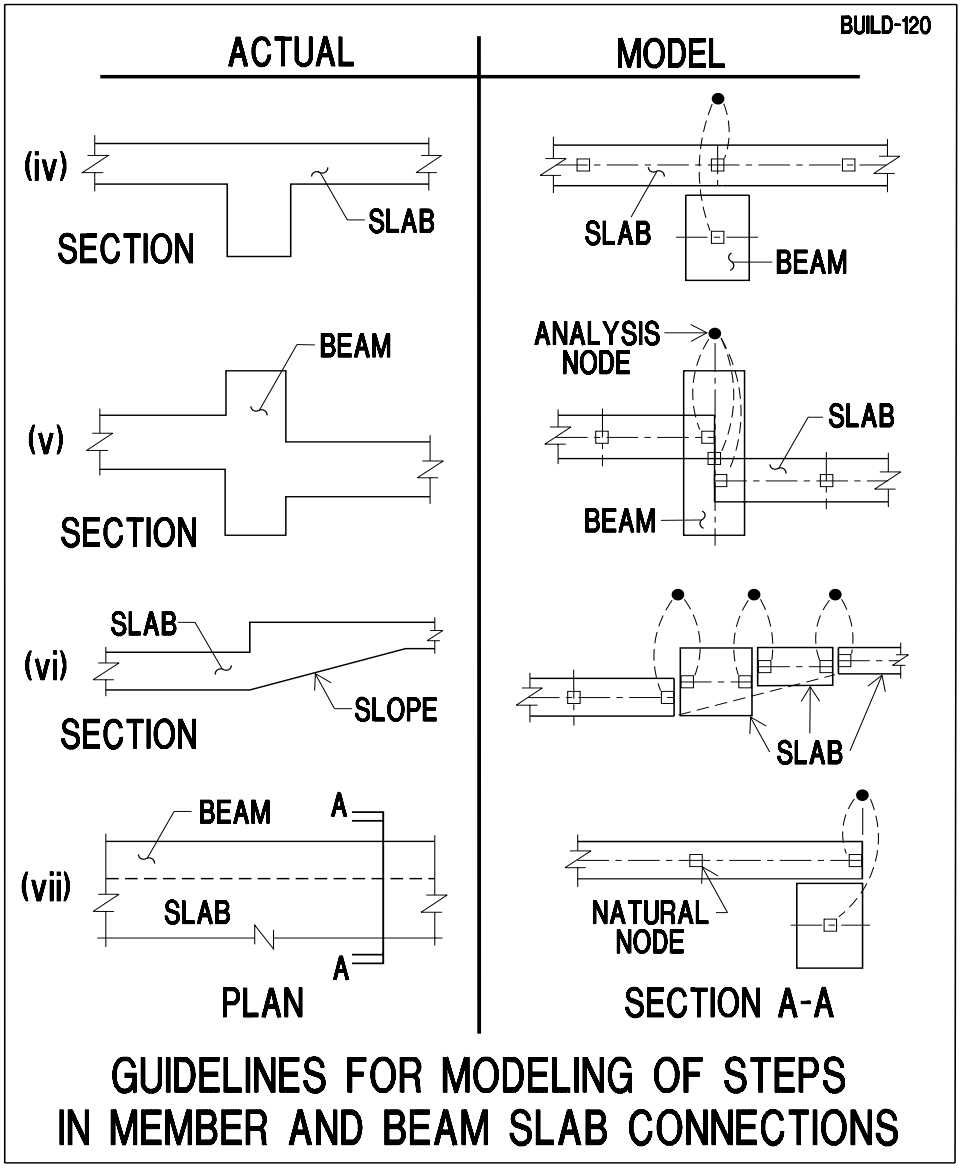


FIGURE A1.2-5B

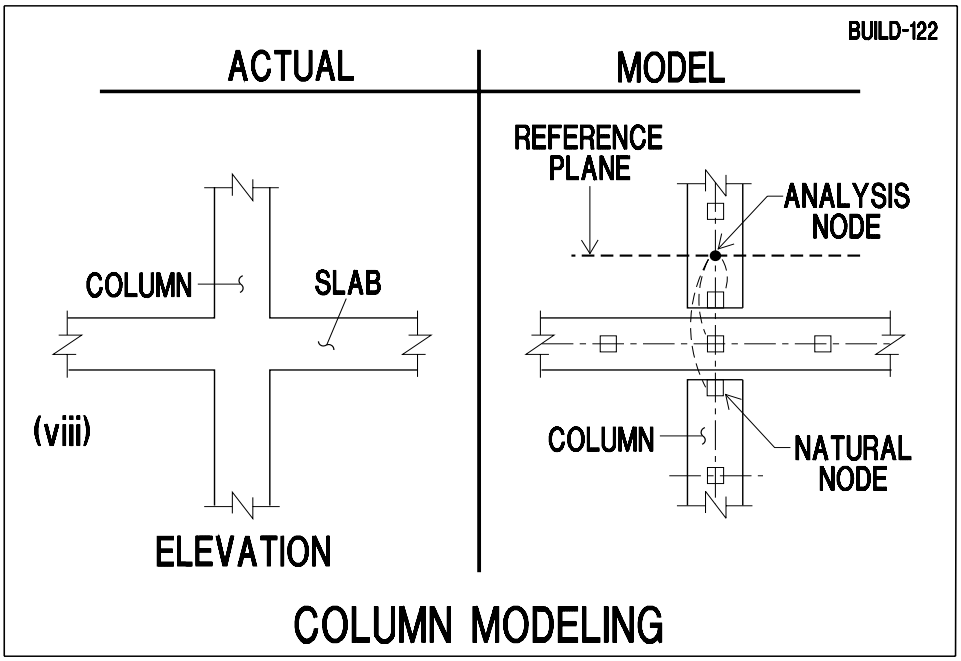


FIGURE A1.2-5C

Figures A1.2-6 and **A1.2-7**show other examples of node consolidation and offset.

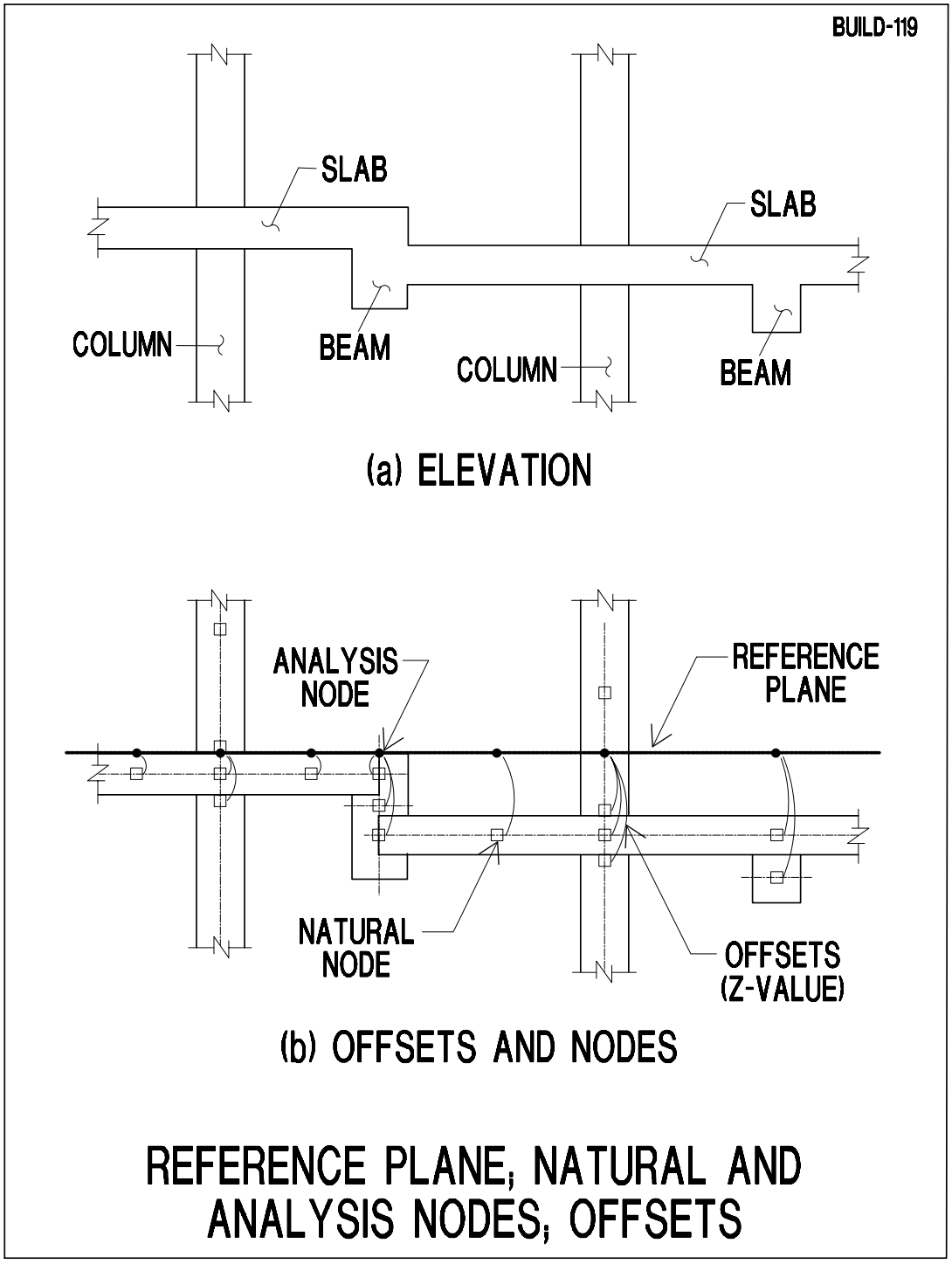

BUILD-119
SLAB
SLAB
COLUMN
BEAM
COLUMN
BEAM
(a) ELEVATION
ANALYSIS NODE
REFERENCE PLANE
NATURAL NODE
OFFSETS (Z-VALUE)
(b) OFFSETS AND NODES
REFERENCE PLANE; NATURAL AND ANALYSIS NODES; OFFSETS

FIGURE A1.2-6

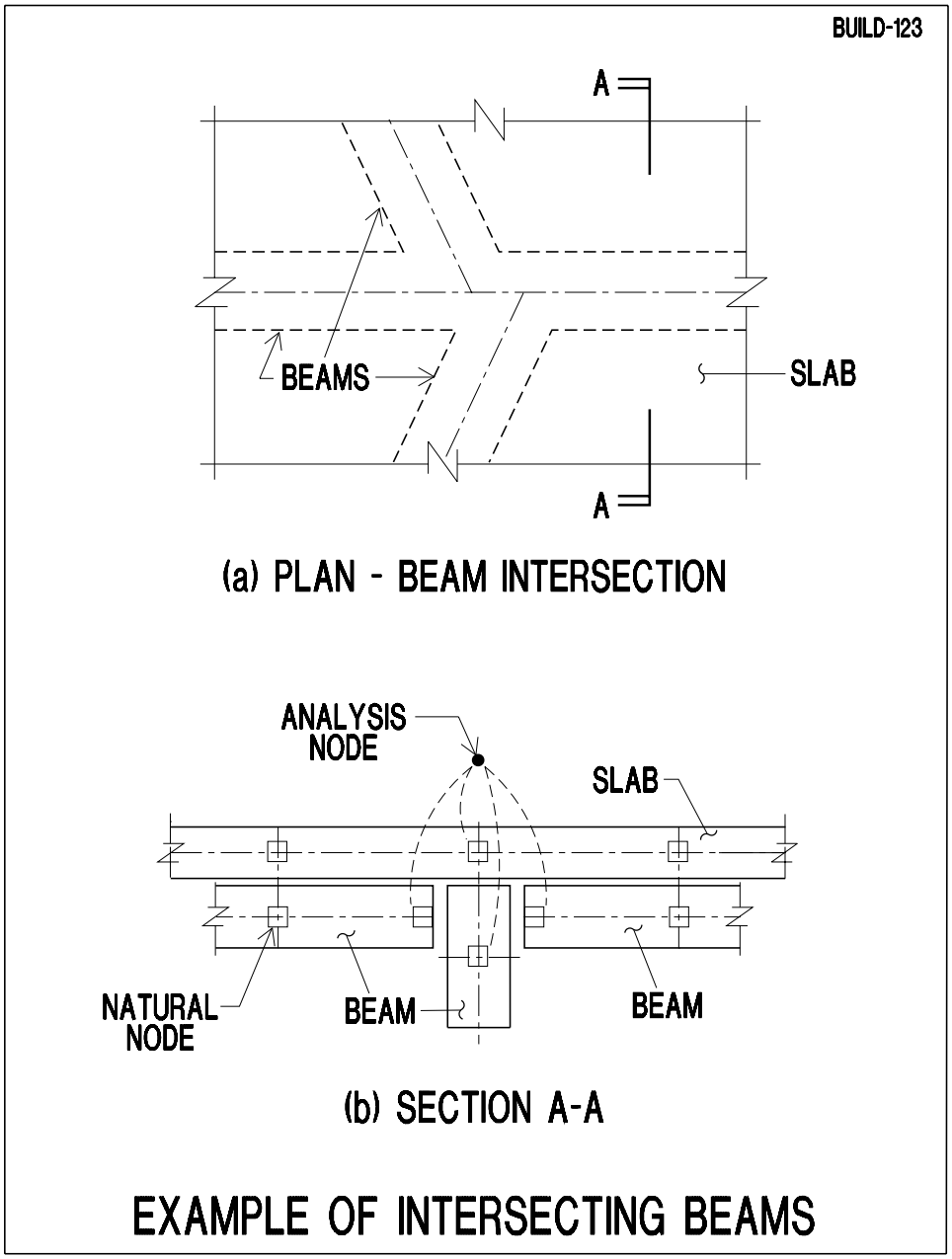


FIGURE A1.2-7

A2 MESHING USING EXCLUDER

Figure A2-1 shows a slab with a wall, columns and an opening. The proximity of the wall to the cut-out corner and also the proximity of one of the columns to an opening are likely to cause a dense mesh within the region. If meshing is attempted, a layout as shown in **Fig. A2-2** with 541 elements will result. In this example, since the total number of cells is small, the dense mesh generated is of no practical computational consequence. However, for more complex floor

systems, while the program can handle a mesh with a large number of cells, the computational time may not be justifiable, especially if the required accuracy can be obtained by a much smaller number of cells. Typically, standard floor systems do not require more than 4,000 cells. Complex slabs may require up to 6,000 cells.

Referring back to the example of **Fig. A2-1**, the mesher displays with red circles at the locations, where details of your structural model require a cell with a size smaller than the one suggested (**Fig. A2-3**). You may override the mesher's message and continue with the meshing. The outcome will be cells of smaller side dimensions at the locations indicated by the program. This, obviously, is acceptable. The alternative is to exclude the regions of complex details from the automatic meshing and fill in the cells within these regions at later time. This is achieved using manual meshing within the excluded regions.

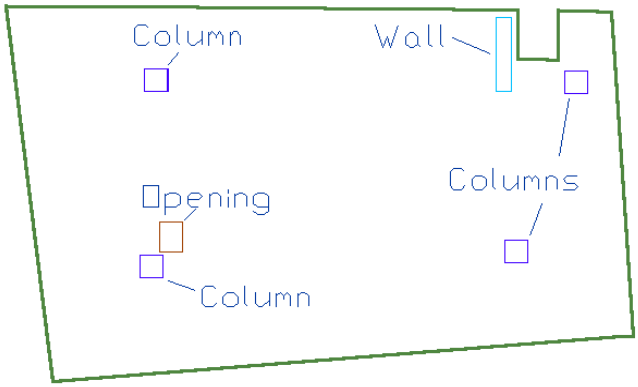


FIGURE A2-1 LAYOUT IN PLAN VIEW

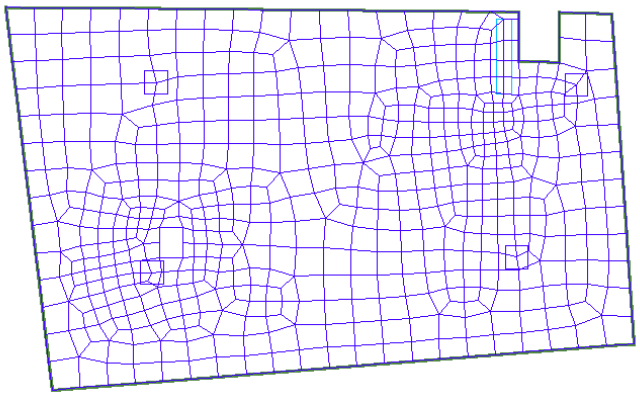


FIGURE A2-2 MESHING WITHOUT EXCLUDER (541 ELEMENTS)

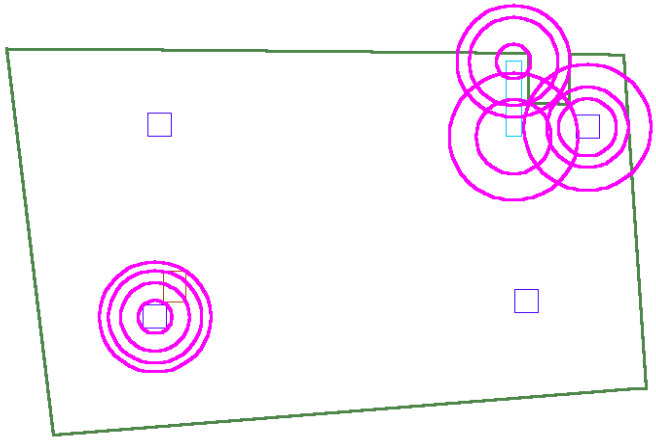


FIGURE A2-3 PROBLEM REGIONS INDICATED BY CIRCLES

To exclude a slab region from automatic meshing, click on the *Excluder* tool located in the *FEM* menu under *Automatic Mesh Generation* dialog box. Draw a polygon around the region to be excluded. For the program to exclude the region successfully, there are four rules that need to be followed.

Rule #1: The first point of the excluder shall fall within a slab region.

Rule #2: The excluder cannot cross the boundary of a slab region in more than two locations.

Rule #3: The area bounded by the excluder shall be convex. That is to say, none of the internal angles of an excluder shall exceed 180 degrees.

Rule #4: Excluder cannot intersect an opening, but can fully contain one or more openings.

Using the above rules, two excluders are drawn as illustrated in **Fig. A2-4**. The remainder of the slab region is now ready for *Automatic Mesh Generation* ( ) at a recommended mesh size of 3ft. The resulting mesh (**Fig. A2-5**) has a coarser and more even grid. The components located inside the excluder will be treated according to the following rules.

Rule #1: Automatic shift node does not apply within the excluded area.
Rule #2: Manual shift node is permissible within the excluded area but shall not be used to shift nodes to the excluder boundary, or across it.

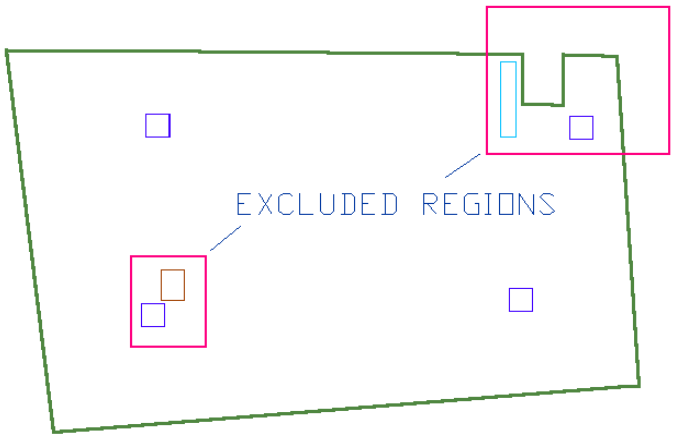


FIGURE A2-4 TWO EXCLUDERS AROUND PROBLEM AREAS

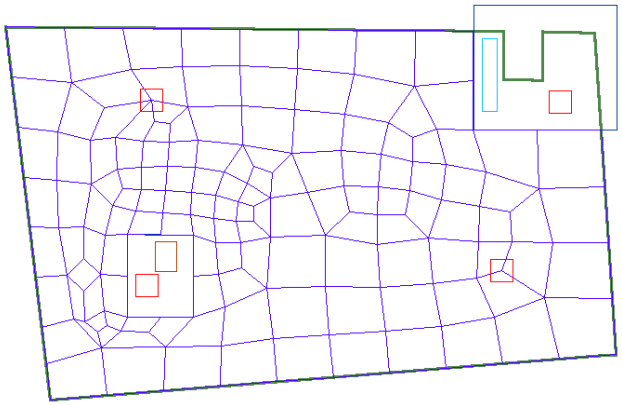


FIGURE A2-5 AUTOMATIC MESH GENERATION OUTSIDE EXCLUDED REGIONS

The regions excluded are now ready to be meshed manually. To fill in the regions within the excluded areas, refer to the Modeler Manual for manual meshing.

A3 VIEW FINITE ELEMENT NODE AND CELL INFORMATION

You can view the finite element cell and node numbers used by the program to obtain your solution. Also, you can obtain a text-based list of both the nodes and cells. The list of cells also includes the node connectivity.

One of the several ways to obtain the graphical display of cell node numbers is to select their display check mark from the *Select/View Items* ( ) tool . Figures A3-1 and A3-2 show views of cell and node numbers.

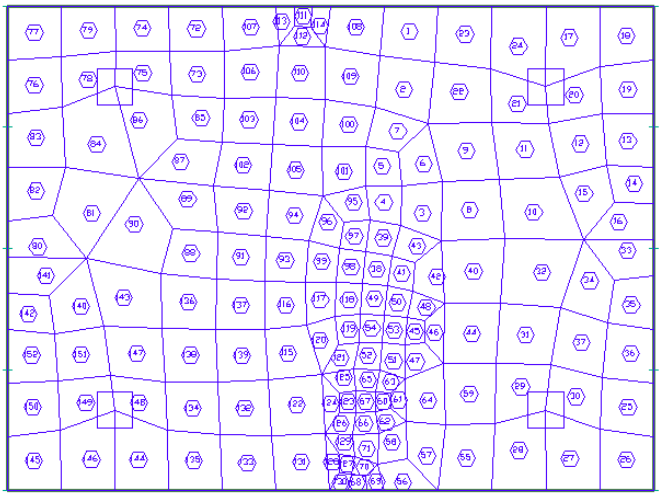


FIGURE A3-1 VIEW OF CELL NUMBERS

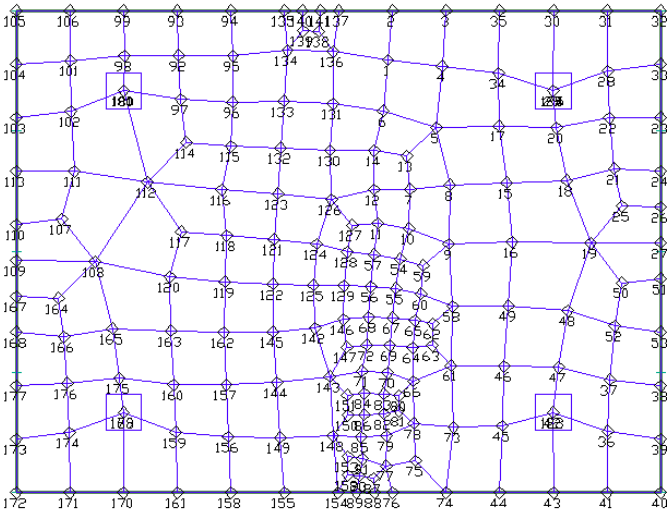


FIGURE A3-2 VIEW OF NODE NUMBERS

Samples of node numbers and their coordinate are shown in TABLES A3-1 and A3-2.

TABLE A3-1 EXAMPLE OF SHELL ELEMENTS AND THEIR CONNECTVITY

Element	I	J	K	L
1	113	116	136	135
2	135	134	107	113
3	124	139	138	123
4	123	106	102	124
5	122	124	102	104
6	122	134	139	124
7	122	104	107	134
8	149	151	138	139
9	139	134	144	149

TABLE A3-2 EXAMPLE OF NODE NUMBERS AND THEIR COORDINATES (INCH)

NODE	X	Y	Z
1	2.49132E+02	2.90839E+02	1.20000E+02
2	2.52144E+02	3.24000E+02	1.20000E+02
3	2.88108E+02	3.24000E+02	1.20000E+02
4	2.85440E+02	2.86592E+02	1.20000E+02
5	2.81406E+02	2.45109E+02	1.20000E+02
6	2.46168E+02	2.56720E+02	1.20000E+02
7	2.63865E+02	2.03854E+02	1.20000E+02
8	2.90595E+02	2.06793E+02	1.20000E+02
9	2.89845E+02	1.67141E+02	1.20000E+02
10	2.63171E+02	1.77661E+02	1.20000E+02
11	2.42210E+02	1.81068E+02	1.20000E+02

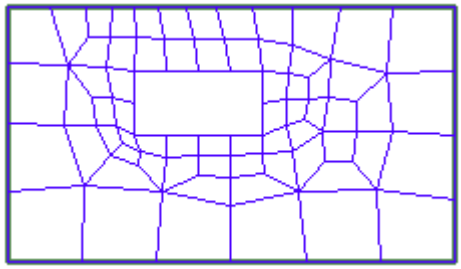
A4 FINITE ELEMENT CELL SIZE AND MESH DENSITY

A4.1 Maximum Mesh Size

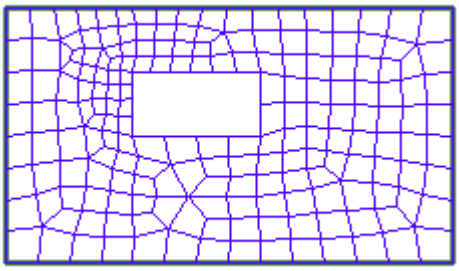
Automatic meshing is performed by the program using a maximum mesh size defined by the user. Generally, 6 to 8 divisions for the typical span of a floor system provide a good solution for design. If geometric complexity in the structural model makes it impractical to mesh with the user-suggested size, the program automatically reduces the mesh size and makes a new attempt. The reduction of maximum mesh size followed by a new attempt at meshing is continued through many iterations until either a successful meshing is achieved, or the number of iterations allowed is exhausted. If the automatic meshing is successful, the user has the option to view/edit the meshing selected. Otherwise the solution is ready to be converted into elements. The default number of iterations for selection of an optimum cell size is 2. The user can change this default to a larger number of iterations, each successive iteration with a smaller cell size.

If an attempt at automatic meshing is not successful, several options are available:

- Re-attempt automatic meshing, but specify a smaller "maximum mesh size." The value to be specified should be less than 80% of the previous attempt, since the program automatically reduces the suggested size in each of its iterations by 20%. For example, if the first user suggestion was 2m, the second suggestion should less than 1.60m.
- Use the *Node Proximity Detection* tool from the *Shift Nodes* option in the *FEM* pull down menu. The program will encircle and display the location(s). A visual evaluation of the location reveals the preparatory steps necessary for the meshing. Invariably, these are locations where clusters of closely spaced imposed nodes, such as an intersection of several beams at an uncommon location, make it difficult for automatic meshing. In this case the user should rectify the location indicated by the program by manually consolidating two or more of the imposed nodes. This is referred to as the *Shift Nodes Manually* operation.



(a) Selection of a large "Maximum Mesh Size"



(b) Selection of a small "Maximum Mesh Size"

FIGURE A4.1-1 EFFECT OF "MAXIMUM MESH SIZE" SELECTION

The effect of the maximum mesh size selection on the outcome of the program's automatic meshing is illustrated in **Figure A4.1-1,** which shows the plan of a slab with an opening.

The maximum mesh size selection is made in the dialog box shown in **Figure A4.1-2** when invoking the automatic mesh generation button .

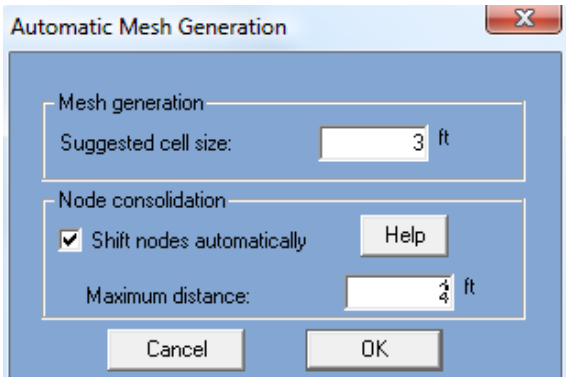


FIGURE A4.1-2 AUTOMATIC MESH GENERATION DIALOG BOX

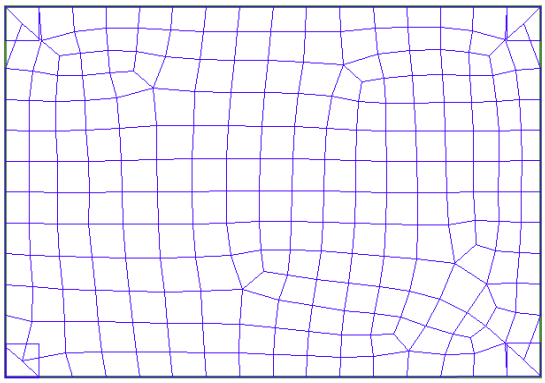
The final mesh size selected by the program is governed by the location on the floor system where closely spaced imposed nodes dictate a dense meshing. The following example illustrates the point.

Figure A4.1-3(a) shows a slab region with corner columns. For analysis, it is advantageous to consolidate the analysis node of this column with that of the slab corner, in other words shift the analysis node of the column to the corner of slab. If the column's natural node is not shifted to the slab corner, the program will impose an analysis node at the column centerline and mesh the slab region accordingly (part (b) of the figure). The outcome is generally a denser mesh, since the distance between the column centerline and the slab edge must be represented by at least one slab element. However, if the column's analysis node is shifted to the slab corner (part (c) of the figure), the ensuing mesh generation will be independent of the column's natural node. The outcome is a larger mesh size (fewer elements) as shown in part (c) of the figure. It should be emphasized that the location of the corner column remains unchanged, but for purposes of analysis, its analysis node is advantageously shifted to the slab corner. Moments, axial forces, punching shear and other analysis and design values reported by the program refer to the actual (natural) column and member locations.

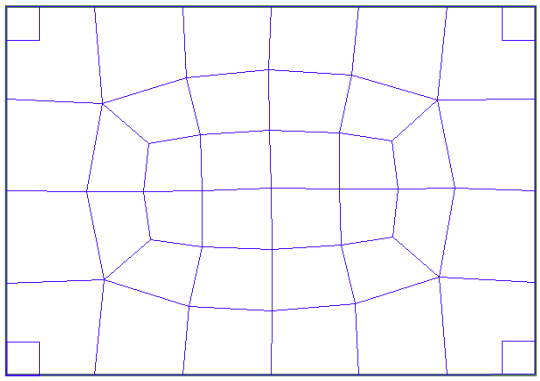
Edge columns are treated in a similar manner. Their analysis node is best shifted to the slab edge.



(a) Plan of slab with four columns



(b) Nodes not shifted



(c) Nodes shifted

FIGURE A4.1-3 IMPACT OF NODE SHIFT ON MESHING

A5 MANUAL EDITING OF AUTOMATICALLY GENERATED MESH

This section describes how you can edit a mesh that is automatically generated by the program. Manual mesh generation is described in detail in Modeler User Manual. Editing of automatically generated mesh becomes of interest in at least two conditions.

First, in some instances the automatic mesher creates an overly dense mesh at complex details of your structural model. While not essential, it is often desirable from a PC-performance standpoint to reduce the number of cells.

Second, at some locations the aspect ratio for sides of a given cell might be too large. Again, while from computational standpoint this is acceptable, some users may wish to bring the aspect ratio to a more acceptable limit. The formulation of the program gives solutions acceptable for design, with aspects ratios of individual cells well beyond 50. For quadrilateral shell elements this is the ratio of the element length to the element width. For triangular elements, if any, this is the

ratio of the triangle base to its height. When aspect ratio reaches a limit that might not be acceptable, the program generates an onscreen message prompt and identifies the cell in question.

The following walks you through the steps of manual mesh editing by way of an example. Consider the floor slab shown in **Fig. A5-1**. As an exercise, let us edit the cells around the upper right hand column to a lower density.

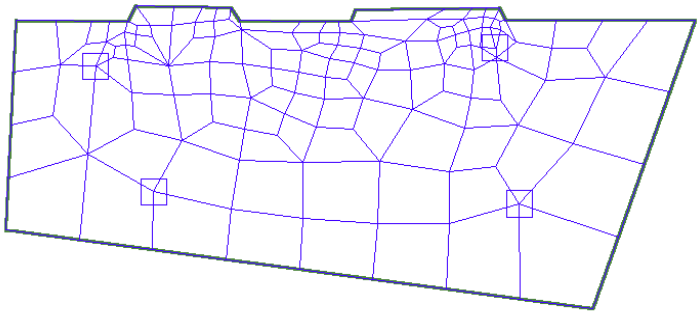


FIGURE A5-1 FLOOR PLAN SHOWING FINITE ELEMENT MESHING

The enlarged view of this region on the computer screen is given in **Fig. A5-2**. The quadrilaterals and/or triangles intended for editing are first selected **(Fig. A5-3)**. The selected items are identified on the screen by a change in color. The selected items are then deleted **(Fig. A5-4)**.

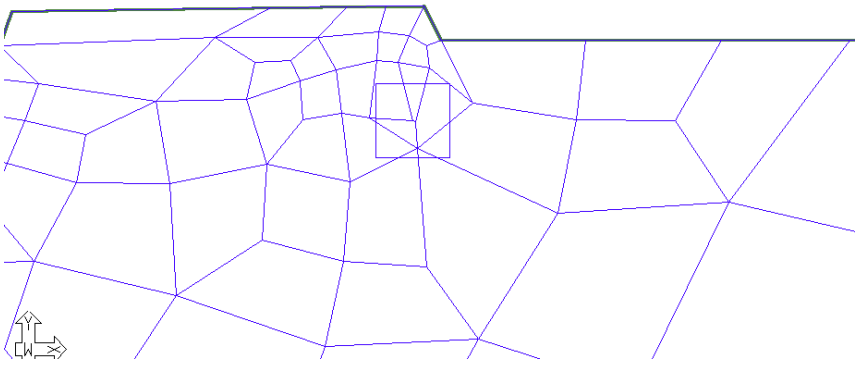


FIGURE A5-2 ENLARGED VIEW OF LOCATION TO BE EDITED

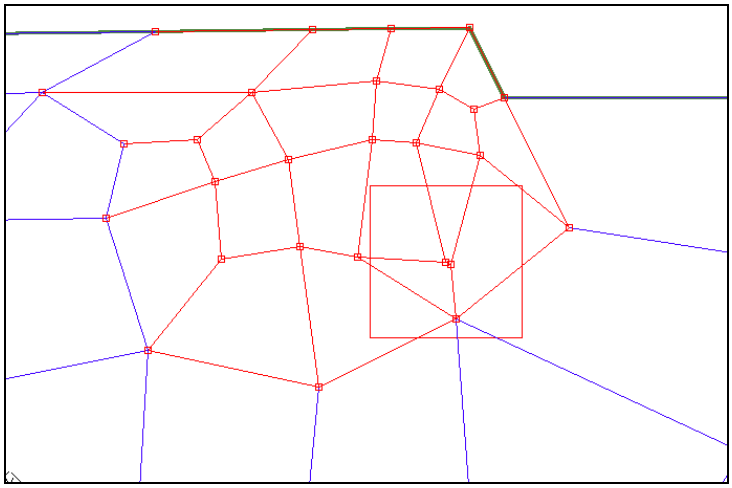


FIGURE A5-3 QUADRILATERALS INTENDED FOR EDITING ARE SELECTED

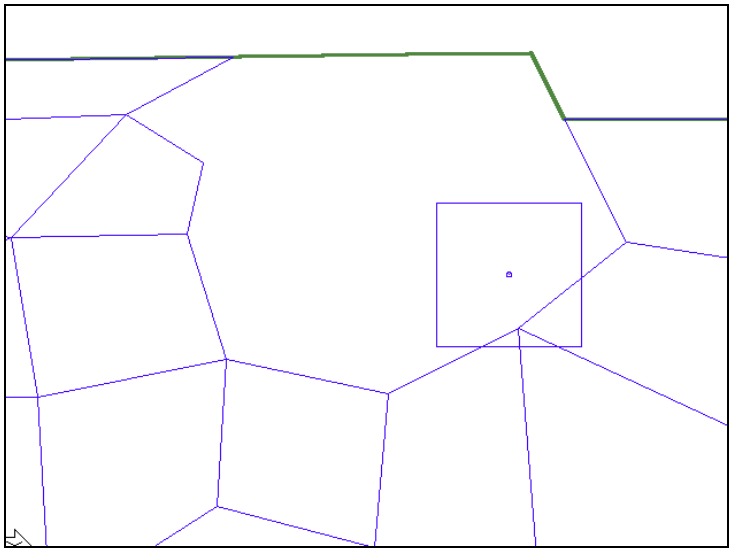


FIGURE A5-4 ITEMS SELECTED FOR EDITING ARE DELETED

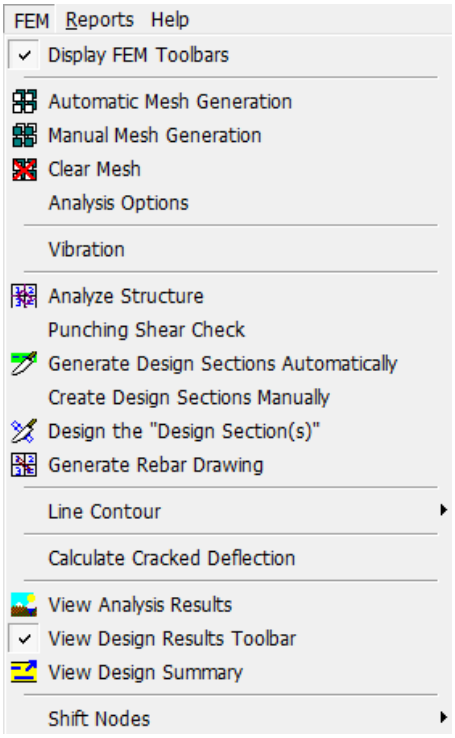


FIGURE A5-5 SELECTION OF MANUAL MESHING COMMAND

From the *FEM* pull down menu, select *Manual Mesh Generation*, the dialog box of *Shell Elements - User Defined* will appear (**Figs. A5-5 and A5-6**).

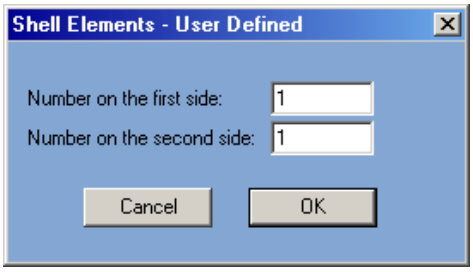


FIGURE A5-6 MANUAL MESH DIALOG BOX

Set the number of cells to be created to 1 as shown in **Fig. A5-6**. Press *OK* to close this dialog window and start drawing quadrilateral or triangular cells to fill in the region to be meshed. Complete each cell first before moving on to the next. In drawing the cells you have to use the appropriate snapping tool from the *Snap Toolbar* as shown in **Fig. A5-7**. Use *Snap to Intersection* (fourth button from the left) wherever possible. This will ensure the connectivity of adjacent cells at the nodes. If the vertices of the cells you create do not snap to one another, there will be no force transfer among them.



FIGURE A5-7 SNAP TOOLBAR

Fig. A5-8 shows the first two cells created. Note that column center is selected as one of the nodes. The completion of the work is shown in **Fig. A5-9**.

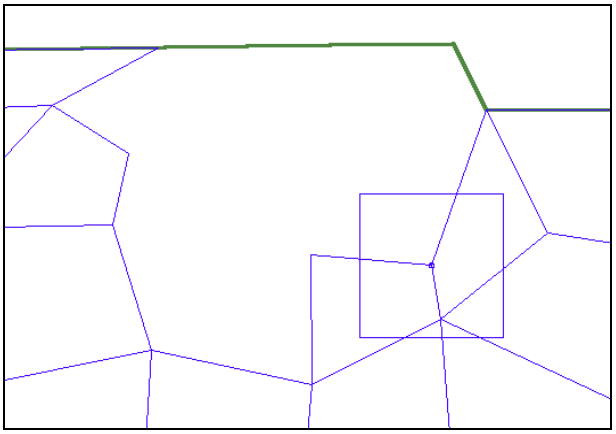


FIGURE A5-8 CREATION OF THE FIRST TWO QUADRILATERALS

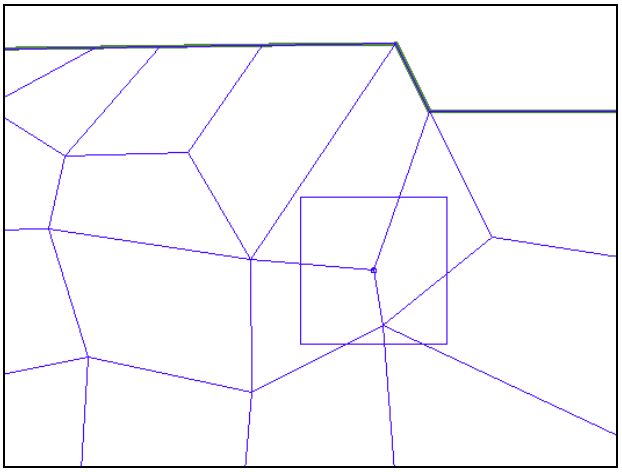


FIGURE A5-9 VIEW OF COMPLETED EDITING

A faster method to fill in a specific region in a mesh is to generate a group of cells that cover the region, and then snap the perimeter nodes of the new mesh to the boundary of the existing mesh cells. Before getting started, estimate how many cells are necessary to fill the region. From the *FEM* pull-down menu select *Manual Mesh Generation* (**Fig. A5-5**). In the *Shell Elements – User Defined*

dialog box, define several cells at a time instead of one-by-one. When using this process, a mesh of 3x2 will be used to fill the region (**Fig. A5-10**).

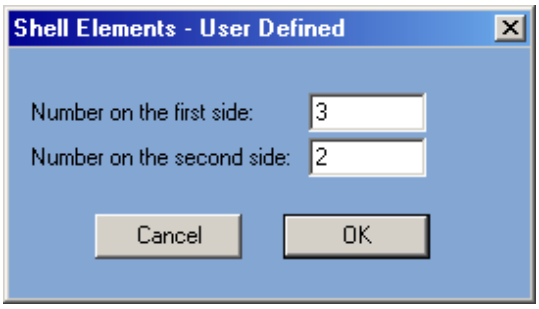


FIGURE A5-10 USER-DEFINED MESH DIALOG BOX

Once the number of cells has been defined, the program will prompt you to enter the four vertices of the 3x2 mesh. It is good practice to position the four points so that the new mesh's boundary is close to the boundary of the region it is expected to cover (**Fig. A5-11**). If the new mesh is positioned well, then there will be less work in adjusting the cells to obtain a well-balanced mesh.

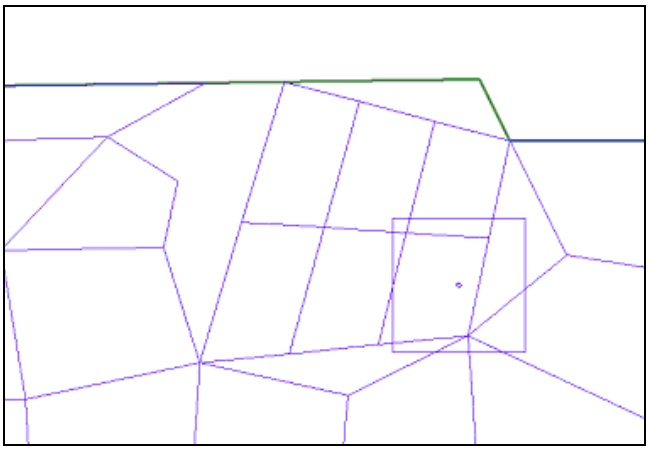


FIGURE A5-11 VIEW OF 3 X 2 MESH

Use the diagrams in **Fig. A5-12** as a guideline in adjusting the cells. Note that to reposition a common vertex shared by two or more cells, you must select all cells that meet at that common vertex. To do so, click on the first cell and then hold down the "Ctrl" key to add more cells to the selection. If you have difficulty selecting a cell, you may find the "tab" key on your keyboard helpful in selecting overlapping cells. In order for the column to be considered in the analysis and design, you must snap the vertex of a cell on the center of the column as shown in **Fig. A5-12 (d)**.

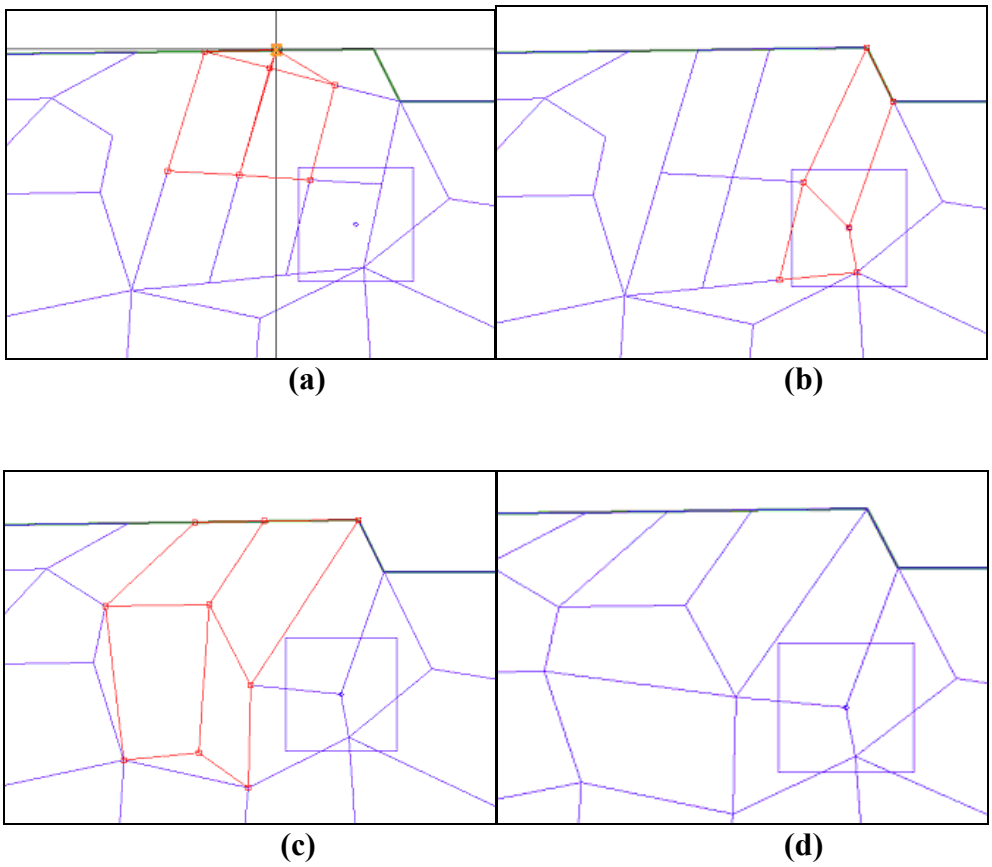


FIGURE A5-12 NODE-ADJUSTMENT PROCESS

APPENDIX B

B1 TIPS FOR MESHING 291
B1.1 Extend the Boundaries of Openings to Centerline of Adjacent Walls 291
B1.2 Extend Openings to Slab Edge 291
B1.3 Combine Adjacent Openings 292
B1.4 Break Up Walls and Beams That Intersect Outside a Slab Region 293
B1.5 Either "Disregard" Short Wall Projections, or Use Manual Shift Node 294
B2 SUGGESTIONS FOR IMPROVED MESHING 295
B2.1 Wall Intersections 297
B2.2 Break Walls Across Different Slab Regions 298
B2.3 Break Up Walls and Beams Adjacent to Jagged or Irregular Slab Boundaries 299
B3 MANUAL NODE SHIFT (NODE CONSOLIDATION) 301
B3.1 Exclusion of Locations of Your Choice from Node Shift 302
B3.2 Shift Node (Consolidate Node) Manually 304

B1 TIPS FOR MESHING

B1.1 Extend the Boundaries of Openings to Centerline of Adjacent Walls

For purposes of analysis, it is recommended that the user extend the side of the openings with perimeter walls or beams to the centerline of the walls or beams that form the opening's boundary (**Fig. B1.1-1**). If you follow the modeling technique that is outlined in the workshop tutorial, you will find that this style of modeling does not impact the authenticity of the structural drawings you create. Your drawings for construction will show the openings extending to the sides of the perimeter walls/beams.

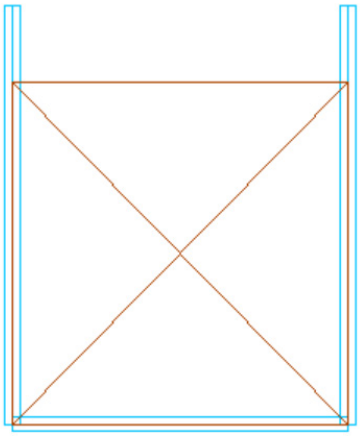


FIGURE B1.1-1 OPENING EXTENDED TO CENTERLINE OF WALLS

B1.2 Extend Openings to Slab Edge

Where openings are next to the slab edge, such as shown in three different scenarios in **Fig. B1.2-1**, extend the opening to the slab edge. The extension is justified where the narrow slab band between the opening and slab edge does not make a notable difference in the load carrying capacity of the floor.

Part (b) of the figure shows the treatment in each of the three scenarios.

- If the strip of slab band between the opening and slab edge is not supported, move the slab boundary around the opening.

- Where a wall or a beam separates the opening from the slab edge, extend the opening to the slab edge, with no modifications either in the wall or the beam. The analysis will automatically mesh the beam and the wall forming the outer boundary of the opening. The contribution of the beam and the wall on the outer edge of the opening will be fully accounted for by the program.

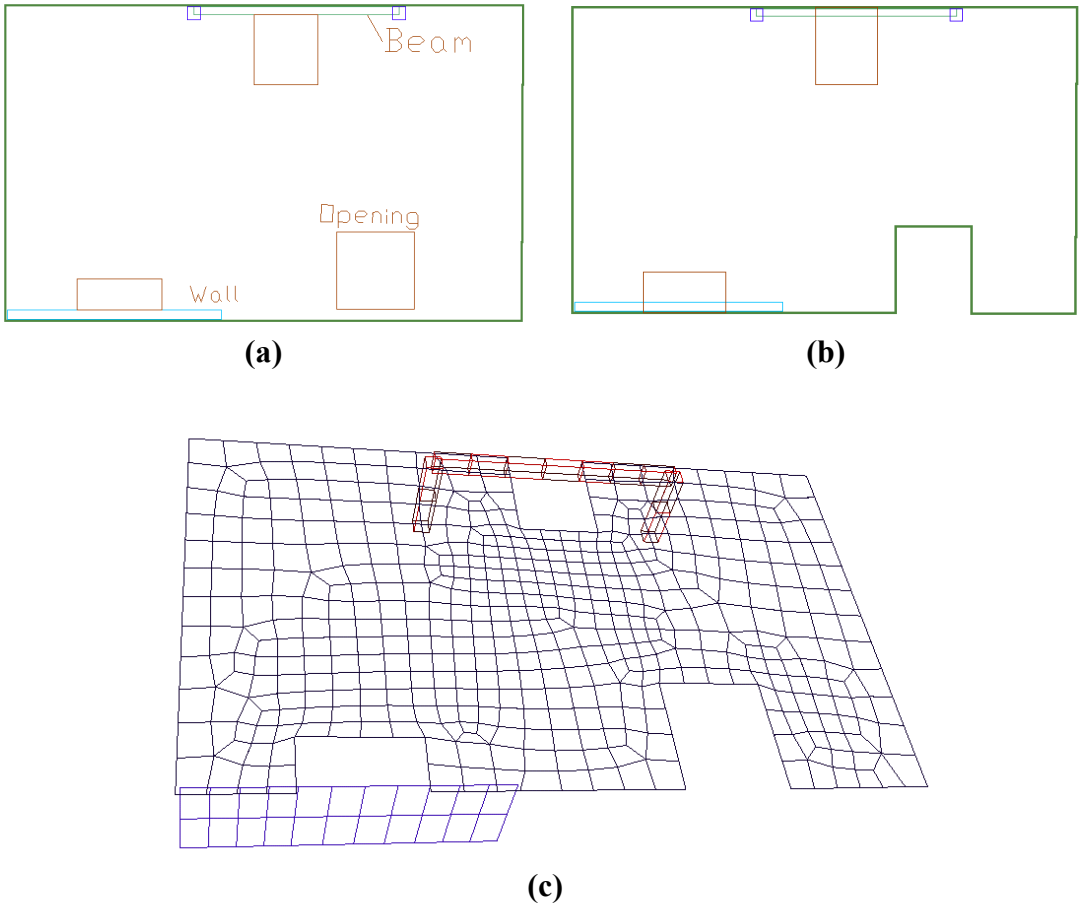


FIGURE B1.2-1 TREATMENT OF OPENINGS NEXT TO SLAB EDGE

B1.3 Combine Adjacent Openings

When two or more openings are next to one another, such as the two openings at the left of **Fig. B1.3-1**, it is best to combine them into one opening as shown in part (c) of the figure. If left as is, an unnecessarily denser mesh can result (part (b) of the figure).

Likewise, when two openings are drawn one on each side of a wall or a beam, it is best to combine the openings as shown in part (c) of the figure.

When combined, the program correctly accounts for the continuity and load carrying features of the wall or beam that passes through the opening.

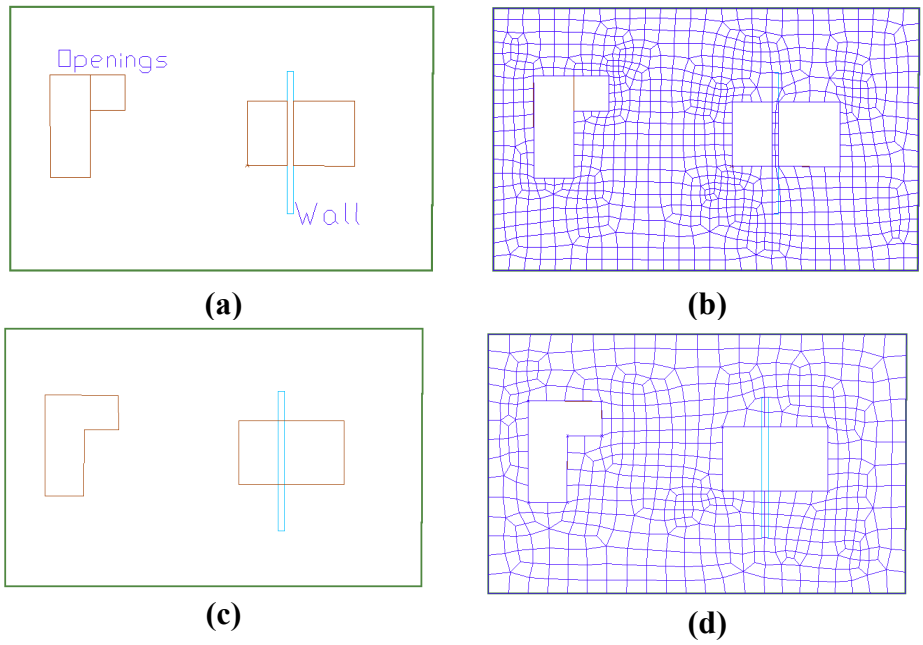


FIGURE B1.3-1 COMBINE ADJACENT MULTIPLE OPENINGS INTO ONE

B1.4 Break Up Walls and Beams That Intersect Outside a Slab Region

Figure B1.4-1 shows a slab region with three walls A, B and C. Wall A does not intersect with another wall outside the slab area. This wall does not need special treatment. Walls B and C, however, intersect at a point outside the slab region. You must break each into two segments, one being the length within the slab region and one outside it. In other words, walls, or portions of walls that fall outside a slab region should be modeled as a separate wall. This is a modeling requirement. From an analysis standpoint, the program will automatically consider the walls contiguous.

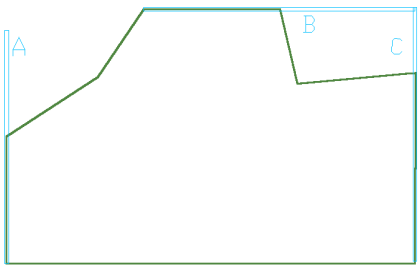


FIGURE B1.4-1 WALLS OR BEAMS EXTENDING BEYOND SLAB BOUNDARY

B1.5 Either "Disregard" Short Wall Projections, or Use Manual Shift Node

In cases where short walls project from longer walls for reasons of architectural detailing, such as the example shown in **Fig. B1.5-1(a)**, there are several options for their efficient modeling.

- Use the scheme suggested in workshop tutorial, whereby the walls appear on your construction (structural) drawings, but not in your structural model for analysis;
- Bring the walls into your structural model, but disregard them in your analysis; and
- Use manual node shift, in order to simplify the modeling and avoid a dense finite element mesh.

The following gives the details.

Disregard Option

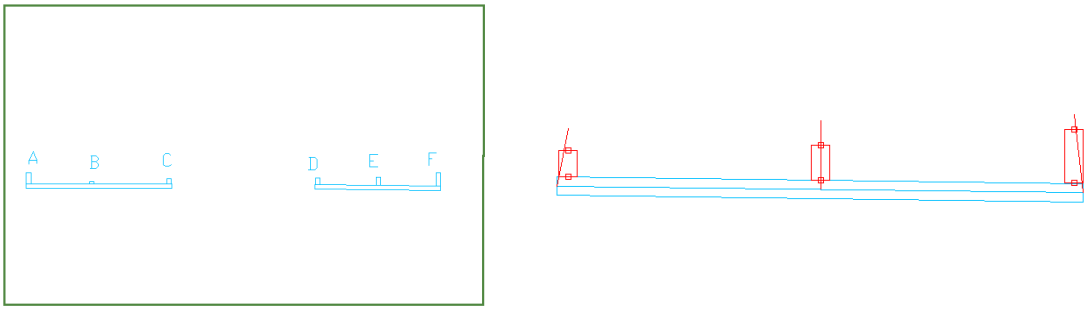
If a wall is created to complete a structural detail, as opposed to carry load, such as projection B in **Fig. 1.5-1**, open its property window and select "disregard." This will retain the wall in your structural model for completeness; will not mesh it; will not consider its stiffness in resisting the applied load – but will consider it in the calculation of reinforcement and quantity take off. In other words, by selecting "disregard" you eliminate the inclusion of that wall in the formation of a stiffness matrix of the structural model, but account for its presence in other aspects of your work, including to resist the design values, such as moments and punching shear.

Manual Node Shift

If a wall projection created to complete an architectural detail is large enough to have a notable effect on the local stiffness of the floor system, you may choose to retain it in your model. Select the "consider" option in its property box. In addition, the mesh density resulting from the inclusion of the wall projection can be improved if you manually consolidate its end node with that of the primary wall. An example of this option is shown in the projections D, E and F of **Fig. B1.5-1**. This kind of node-consolidation is referred to as "manual node shift," and is explained in other sections of this manual. **Figure B1.5-1(b)** shows the shifted axis of the wall projections D, E and F.

Do Not Pre-process

In general, for wall projections with an aspect ratio greater than four, preprocessing treatment is not necessary.



(a) **(b)**

FIGURE B1.5-1 TREATMENT OF SHORT WALL PROJECTIONS

B2 SUGGESTIONS FOR IMPROVED MESHING

The following recommendation can help to improve the meshing of your structural model, especially in cases where closely spaced multiple constraints result in a dense mesh.

- In the *Automatic Mesh Generation* dialog window (**Fig. B2-1**), select a suggested maximum cell size 1/6 to 1/8 times the average span.
- Select *Shift node automatically* and consider a distance for node shift equal to three times the average slab thickness. A distance between two to four times gives reasonable results.

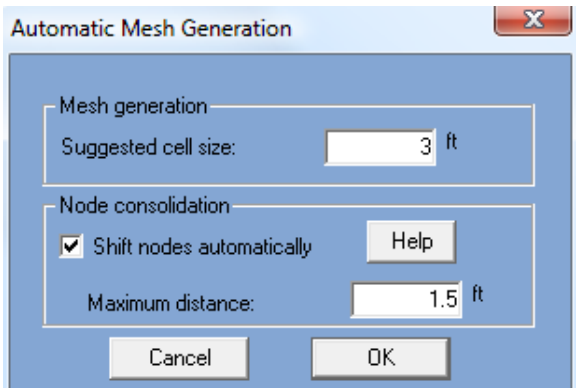


FIGURE B2-1 AUTOMATIC MESH GENERATION DIALOG WINDOW

Figure B2-2 shows plan of a floor system with closely spaced columns. When using the *Shift nodes automatically* option, the imposed top node of the closely spaced columns will be consolidated into a single analysis node. The resulting mesh is shown in **Fig. B2-3**. A close up of the three-column cluster (**Fig. B2-4a**) indicates a node at center of one of the columns only.

Figure B2-4b displays the consolidation of the natural nodes of the two left columns to the column at right. This is illustrated by the lines connecting the natural nodes of the shifted column to the one on the right. These "shift nodes" lines can be displayed onscreen by turning on the "symbol" of the column. It should be emphasized again that the solution proceeds with each column being duly meshed and concludes with separate forces for each column reported at each column's centroid (**Fig. B2-5**)

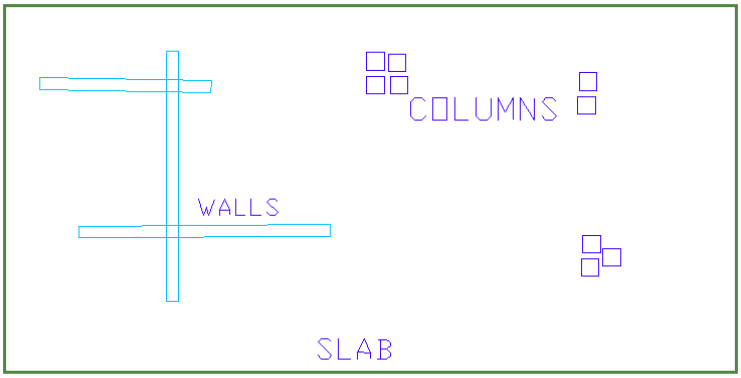


FIGURE B2-2 PLAN OF SLAB WITH CLOSELY SPACED COLUMNS

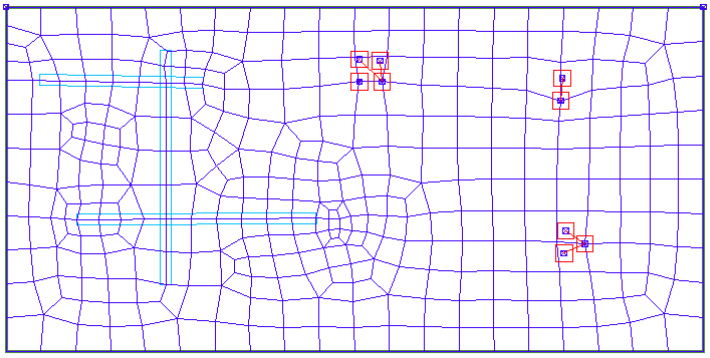
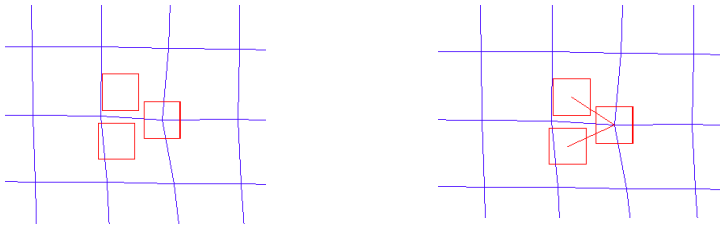


FIGURE B2-3 MESHING OF THE STRUCTURE USING NODE SHIFT



(a) Frame Element Symbol Not Shown (b) Frame Element Symbol Shown

FIGURE B2- 4 CONSOLIDATIONS OF THREE CLOSELY SPACED COLUMN TOP NODES INTO ONE (NODE SHIFT)

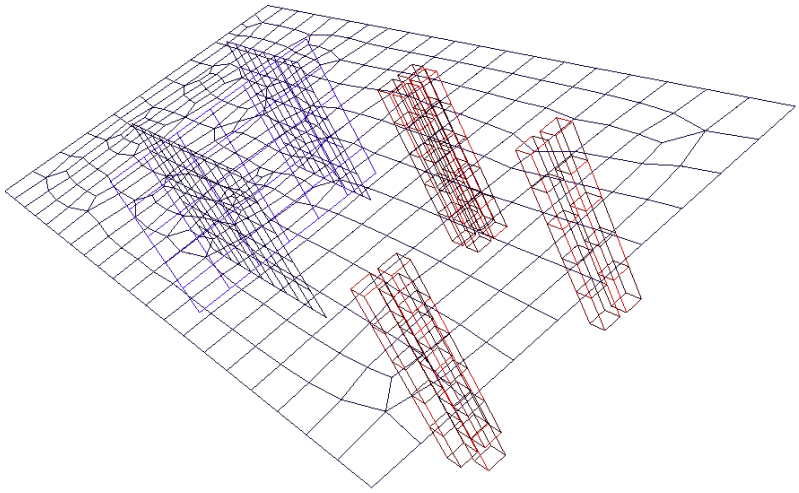
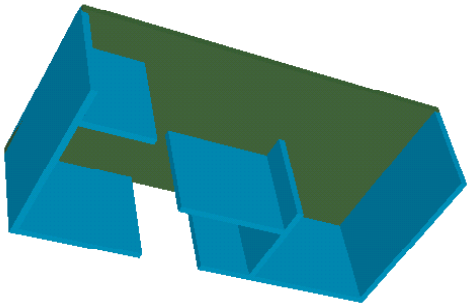


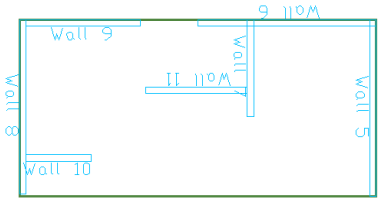
FIGURE B2-5 VIEW OF MESHING SHOWING INDIVIDUAL CELLS OF ALL STRUCTURAL COMPONENT

B2.1 Wall Intersections

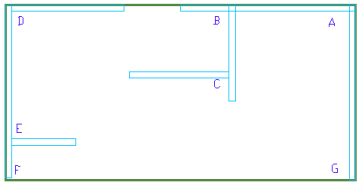
There are several options to model the intersection of walls. These are illustrated in **Fig. B2.1-1**. At "A" (part (c) of the figure), the walls intersect (overlap). Likewise, at "B," Wall 7 is modeled to overlap with Wall 6. Wall 9 butts to Wall 8 at intersection "D." Using the node shift option, the resulting meshing is shown in part (c) of the figure. Note that in all instances the intersection is consolidated into a common node. The difference between the various modeling schemes lies in the meshing and design of the wall itself. For example, for calculation of building weight and wall design, the length of Wall 10 is calculated to the face of Wall 8. But the length of Wall 5 is calculated to the outside face of Wall 6, that is to say, Wall 5 is assumed to extend over the entire width of the slab.



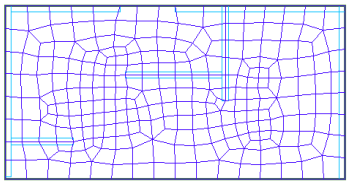
(a) Wall Supported Slab



(b) Wall Identification



(c) Wall Intersection Identification

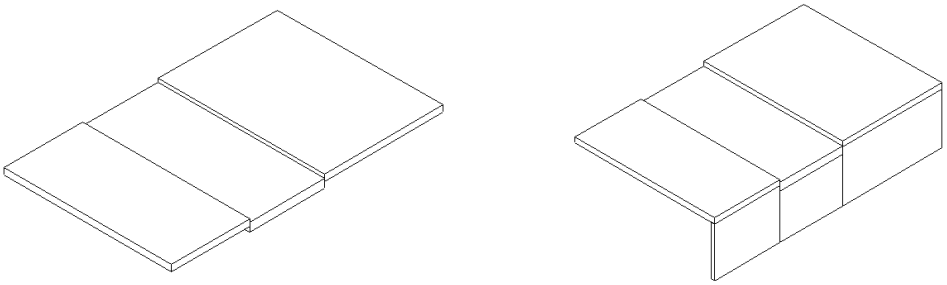


(c) Meshing Using Node Consolidation (Node Shift)

FIGURE B2.1-1 WALL INTERSECTION MODELING ALTERNATIVES

B2.2 Break Walls Across Different Slab Regions

While not essential, where walls extend over more than one slab region, such as shown in **Fig. B2.2-1** (b), you may break them into several walls for greater accuracy in your solutions. Break the wall, such that each sub-wall covers one of the slab regions. The wall in part (b) of the figure is continuous, but it is broken into three parts. When the *Establish Component Connectivity* tool of the program is invoked, the program automatically adjusts the top of each sub-wall to be in line with the soffit of respective slab region. The separation of the walls shown in part (b) of the figure does not affect the analysis of the structure. In the analysis, the walls that touch one another, such as the scenario in this case will be considered contiguous and will be treated as one. There will be full compatibility of deformation among the sub-walls.



(a) (b)

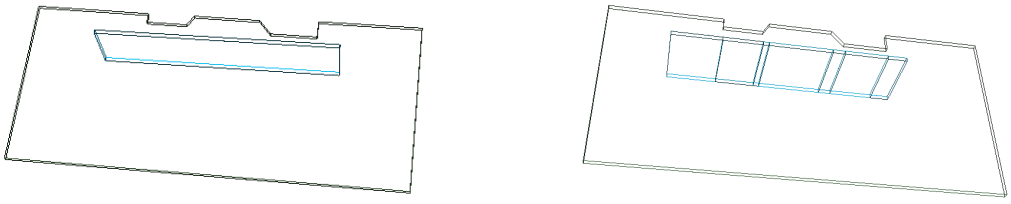
FIGURE B2.2-1 WALLS EXTENDING ACROSS SLAB REGIONS

If you have already created a wall as one whole and want to break it into parts, the simplest procedure is to use the program's "split" tool, which is accessed from the *Modify* pull-down menu. Once you split a wall, drag the sub-walls with the mouse and snap at their ends to the boundary of the slab regions.

B2.3 Break Up Walls and Beams Adjacent to Jagged or Irregular Slab Boundaries

Where a wall or a beam is close to a multiple discontinuity (jaggedness) at slab edge or opening, as illustrated in **Fig. B2.3-1**, it may become necessary to break the wall up into sections. This is mostly the case, where the structural modeling is not done accurately, and a condition as shown in **Fig. B2.3-1** arises. The break up is only necessary (i) if there is more than one protrusion or cut out along the slab edge, and (ii) if you invoke the node shift option of the program. When the node shift option is invoked, the program attempts to shift the analysis axis of the wall/beam onto the slab edge or opening. If the slab edge has many discontinuities, it would not be possible to line up the wall axis with more than one segment of the slab edge (or opening). For this reason, the wall has to be broken down into segments as shown in part (b) of the figure. The number of segments should correspond to that of the slab edge/opening adjacent to the wall/beam.

Again, it should be noted that the breakdown of the wall is not necessary if the shift node option is not used. Also, for the analysis, the program will automatically consider the subdivided pieces of a wall/beam as a whole. In other words, the breakdown of a wall/beam into segments does not impact the solution, nor the treatment of the wall/segment as a contiguous member in resisting the load.



(a) (b)

FIGURE B2.3-1 BREAKUP OF WALLS NEXT TO JAGGED SLAB EDGE

Consider wall A in **Fig. B2.3-2.** This wall needs not be subdivided regardless of whether or not shift node option is used, since its analysis axis can be shifted (lined up) to the slab edge next to it. Wall B adjacent to the two openings, however, must be broken down if shift node is used, since its axis cannot be simultaneously be lined up with the edges of both of the openings. If node shift option is used, it is best to break up this wall into two segments.

A variation of the above conditions is shown in **Fig. B2.3-3**, where a wall is adjacent to a slab edge and at the same time its extension is close to an opening. Again, it generally is not possible to line up the wall axis with both the slab edge and the edge of the opening. If automatic shift node is to be used, it is "best to break the wall into segments.

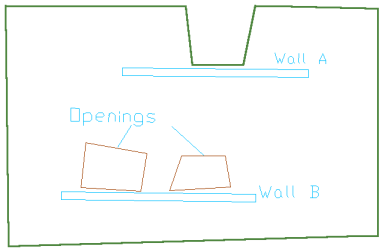


FIGURE B2.3-2 WALL "A" NEED NOT BE BROKEN UP, BUT WALL "B" MAY HAVE TO

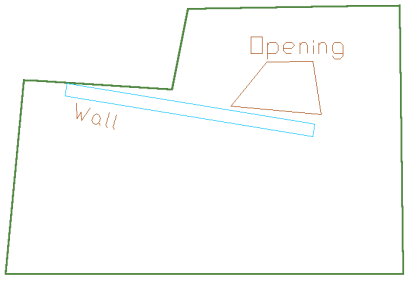


FIGURE B2.3-3 WALL ADJACENT TO SLAB EDGE AND OPENING

B3 MANUAL NODE SHIFT (NODE CONSOLIDATION)

You can manually consolidate analysis nodes at locations of your choice. Also, you can prevent the program from doing so automatically when you do not wish nodes to be consolidated. This section describes these options in great detail.

Access the manual shift node through the *FEM* pull-down submenu *Shift Node*s (**Fig. B3-1**). The following options are available:

- Node Proximity Detection
- Shift Nodes Automatically
- Shift Nodes Manually
- Cancel Node Shift

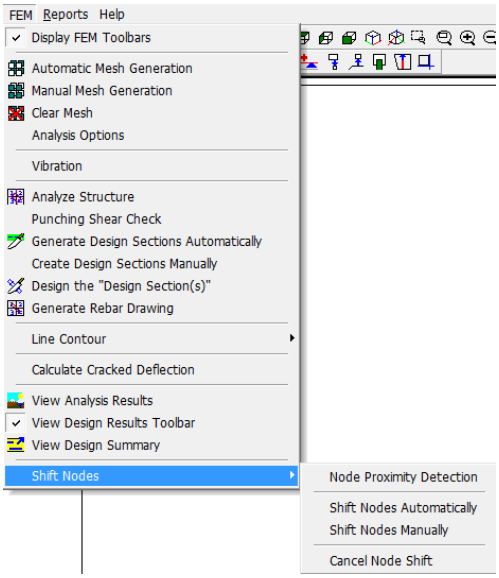
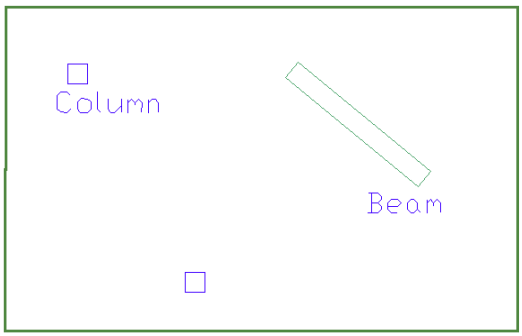


FIGURE B3-1 NODE SHIFT MENUS

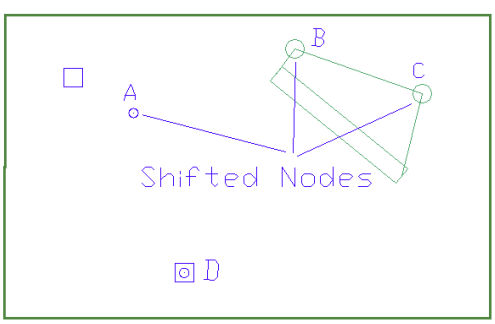
Shift node option applies to columns, beams, walls, point supports, and line supports. Once a node is shifted manually, it will remain at its imposed position until you activate the *Cancel Node Shift*. The following illustrates the procedure.

Figure B3-2(a) shows a floor slab with two columns and a beam. The node of column A and the nodes of the beam are shifted manually to the locations A for the column and B, C for the beam (**Fig. B3-2(b)**). The new locations are displayed with the symbol of the column (small circle) and the beam (large circles). The analysis nodes generated in meshing for the beam will be along line BC. Likewise, the node of column at the top left of the slab will be at point A. The

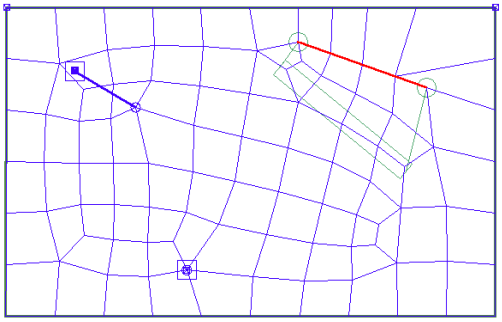
analysis node of column D was not shifted. Hence, a node is generated at its center (part (c) of the figure).



(a)



(b)



(c)

FIGURE B3-2 MANUAL NODE SHIFTS AND THE RESULTING MESHING

B3.1 Exclusion of Locations of Your Choice from Node Shift

You can selectively "freeze" one or more components from node shift, while allowing the analysis nodes of the remainder of the eligible components to be automatically or manually shifted. Consider the floor plan in **Fig. B3.1-1(a)**. Column A is well within the interior of the slab and is not a natural candidate for automatic node shift. The remainder of the columns B, C and D, along with the two beams E and F, are close enough to the slab edge to be subject to node shift. Let us assume that you do not want the node of column C and the nodes of beam E to be shifted. The following is the procedure.

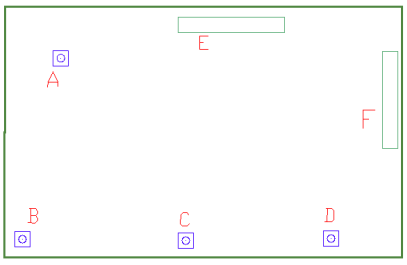
- Double click on column C to open its property dialog window (**Fig. B3.1-2**).
- Select the *FEM* tab.

- Select the check box for *Do not shift nodes automatically*. This will lock the position of the analysis node of this column at its natural position (centroid of the column). The two data fields to the top right of this check box refer to the offset values of the node shift in the X- and Y-directions.
- Likewise, double click on beam E to open its property box and select the *Do not shift nodes automatically* check box in its *FEM* tab.

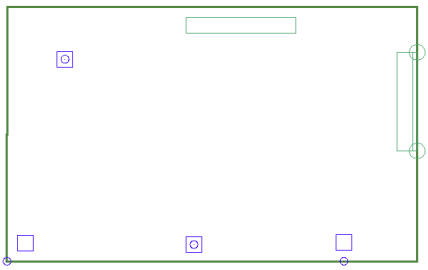
Figure B3.1-1(b) shows the position of the shifted analysis nodes of the columns and the beam, which were either not frozen, or were not a candidate (column A). The outcome of meshing is shown in part (c) of the figure. Observe that in part (c) of the figure there are nodes on column C and along beam E, the two structural components with frozen node shifts.

The mesh layout resulting from this operation is shown in **Fig. B3.1-1(c)**.

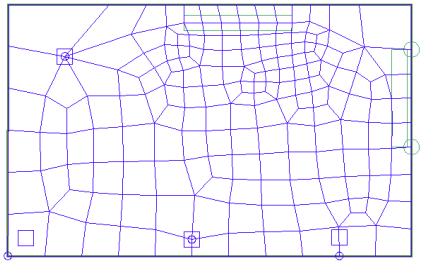
Note that once you manually consolidate nodes, the program locks your choice and will not override it in its automatic node consolidation. In other words the work done by you manually remains frozen, until you cancel your work.



(a)



(b)



(c)

FIGURE B3.1-1 COMPONENTS WITH NODE PERMITTED AND LOCKED NODE SHIFT

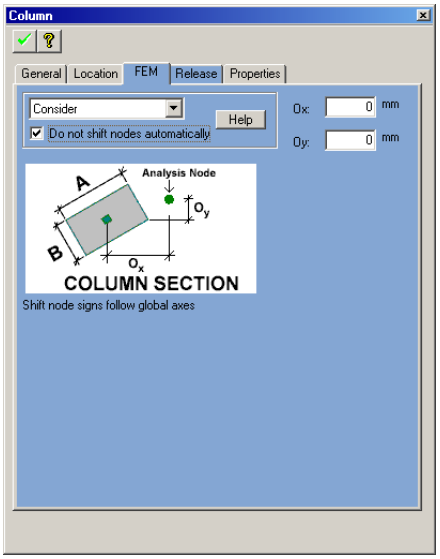


FIGURE B3.1-2 NODE SHIFT CHECK BOX IN COLUMN PROPERTY WINDOW

B3.2 Shift Node (Consolidate Node) Manually

To shift (consolidate) an analysis node manually, click on the *Shift Node Manually* side menu item of **Fig. B3-1,** and follow the instructions on command line at the bottom of the screen. To verify your work do the following:

- Turn on the symbol of the structural component that you selected for node shift;
- Click on the structural component you selected for node shift; and
- Operation is successful when the symbol of the structural component you node shifted is displayed in red color.

APPENDIX C

C1 GLOSSORY OF TERMS 307
C1.1 Natural Nodes 307
C1.2 Analysis Nodes 307
C1.3 Offset 309

C1 GLOSSORY OF TERMS

C1.1 Natural Nodes

Each structural component has a number of "natural nodes" when converted into finite elements for analysis. For example, the natural nodes of a beam frame are located at the centroid of its end sections. Hence, each frame element has two natural nodes. For shell elements, the natural nodes are at each corner of the shell element and mid-depth of its thickness. The natural nodes are located on the midplane of the shell element. **Figure C1.1-1** illustrates several structural components and the position of the natural nodes.

C1.2 Analysis Nodes

Unlike the traditional finite element formulations, in order to handle post-tensioning and also to take advantage of the features of component technology, ADAPT's formulation is based on "analysis nodes." Each finite element of the structure is mapped onto a reference plane as illustrated in **Fig. C1.2-1**. The image of a natural node on the reference plane is its "Analysis Nodes." For example, I' is the analysis node of the natural node I. ADAPT's calculations are performed on the analysis nodes. However, the solution is reported and displayed with respect to the natural nodes. In summary, the analysis node concept is an analytical feature necessitated to move beyond the traditional application of the Finite Element Method and use the more intuitive design approach of "component technology." The introduction and description of "analysis nodes" in this document is intended to provide a smooth informational transition from the traditional FEM to Component Technology. Otherwise, the knowledge of analysis nodes for design is not necessary.

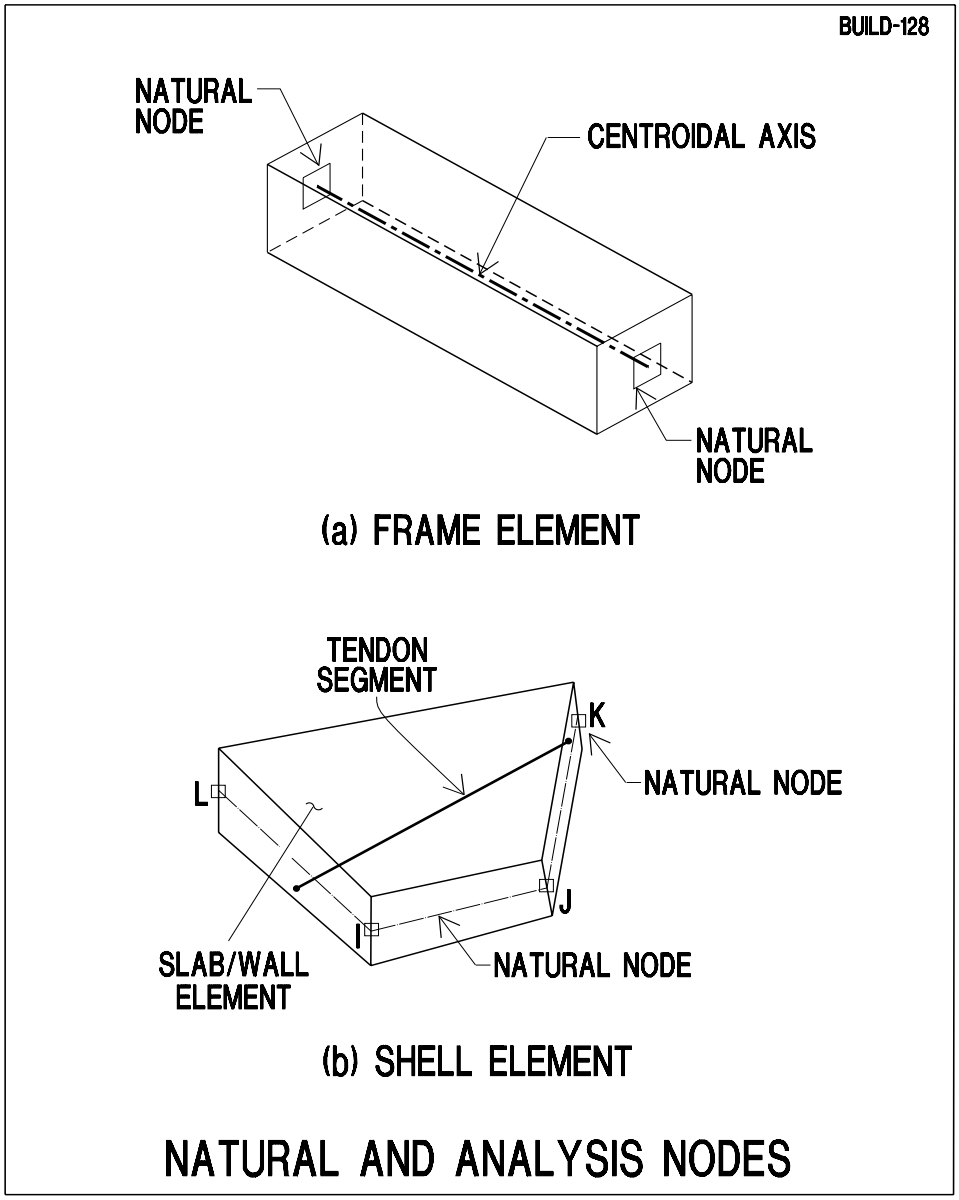


FIGURE C1.1-1 NATURAL NODE

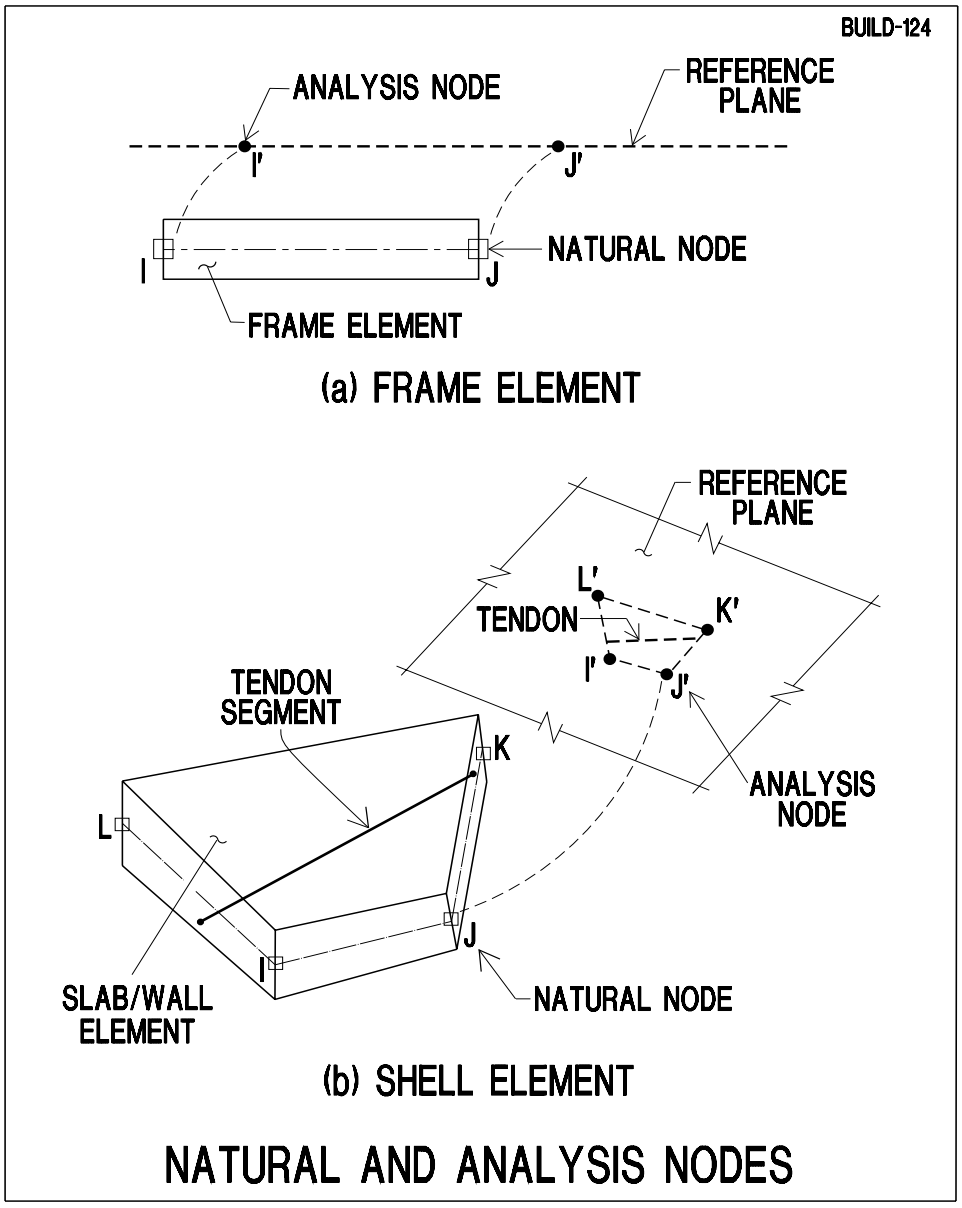


FIGURE C1.2-1 NATURAL AND ANALYSIS NODES

C1.3 Offset

Offset is the distance a natural node must be shifted to reach its analysis node. Refer to **FigC1.3-1** and note that offsets are expressed in terms of translations along the global coordinate axes. For example, OX is offset along X-direction.

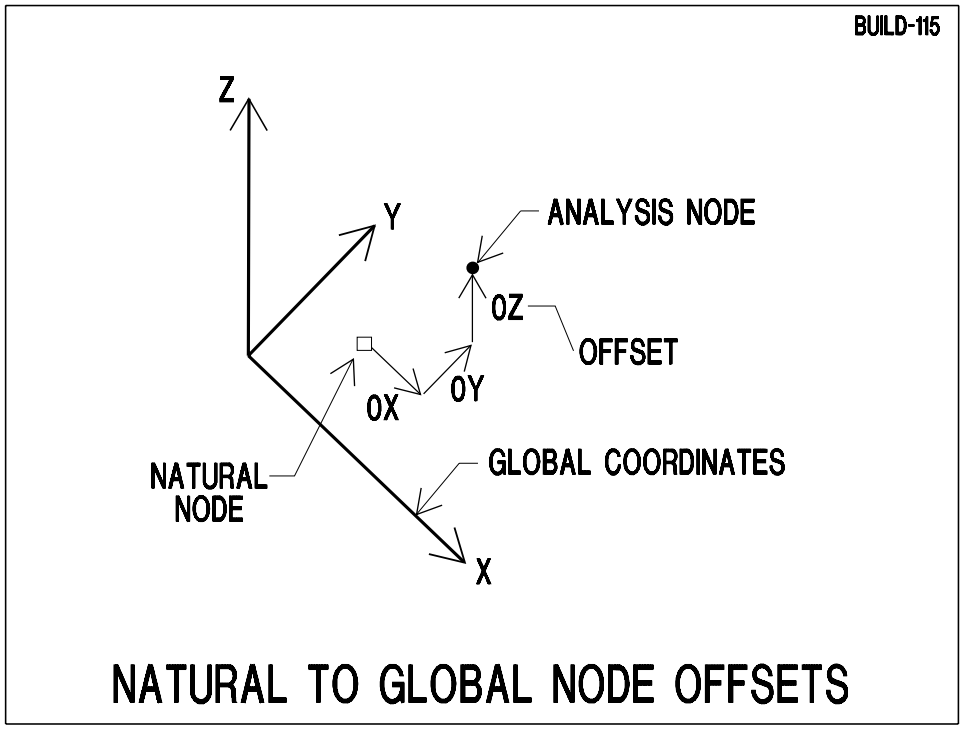


FIGURE C1.3-1 NATURAL AND ANALYSIS NODES

INDEX

Actions at the cut, 144
Adapt Model Exchange (AMX) file, 232
Allowable Stresses, 68
Analysis node, 117, 265, 309
Beam rebar, 183
Beam templates, 185
bitmap, 136
Bonded system, 39
Boundary conditions, 25
Component representative, 105
Component Technology, 116, 118
Concrete Cover, 69
cracked_section deflection, 121
Create Tendon tool, 43
Current Plane, 21
Cut line results, 142
Design Code, 67
Design Sections, 149
Detect Tendon Interference tool, 43
Display settings, 139
Display Tendon Elevation tool, 43
Display Tendon tool, 43
Display tools, 137
Dynamic Tendon Modeling (DTM), 55
Effective Force Method, 39
Entity display settings, 138
Excluder, 278
Generic Import/Export, 232
Hyperstatic load, 85
Import from ADAPT-PT, 187
Imposed lines, 103
Imposed nodes, 103
Jacking force, 41
Lateral Load Editing Mode, 96
Lateral loads, 92
Load cases, 83
Load Combination, 83
Long-term stress loss, 42
Map Banded Tendon tool, 43
Map Distributed Tendon tool, 43
Material properties
  concrete, 65
  mild steel, 66
  prestressing steel, 66
Mesh reinforcement, 164

Meshing, 101
Natural node, 265, 309
Node consolidation, 303
Node Proximity Detection, 282
Node shift, 106, 116, 269
Offset, 22, 311
Pattern loading, 87
Preferred Reinforcement Size, 69
Prestressing load, 85
Punching shear design, 157
Rebar display, 169
Rebar Length, 71
Rebar round up, 70

Report Generation Manager, RGM, 202
Reports
  default reports, 203
  External Graphics, 204
  external text, 204
  User Generated Reports, 203
Result display settings, 138
Shear Design, 68
Show Tendon CGS Values, 43
Skipping of live load, 87
Stress losses, 42
Tendon
  cross-sectional geometry, 35
  friction, 41
  material properties, 38
  profile, 73
  shape, 31
  Stressing, 41
Tendon mapping, 45
Tendons intersection, 57
Unbonded system, 39
Variable Force Method, 40
View analysis results, 135