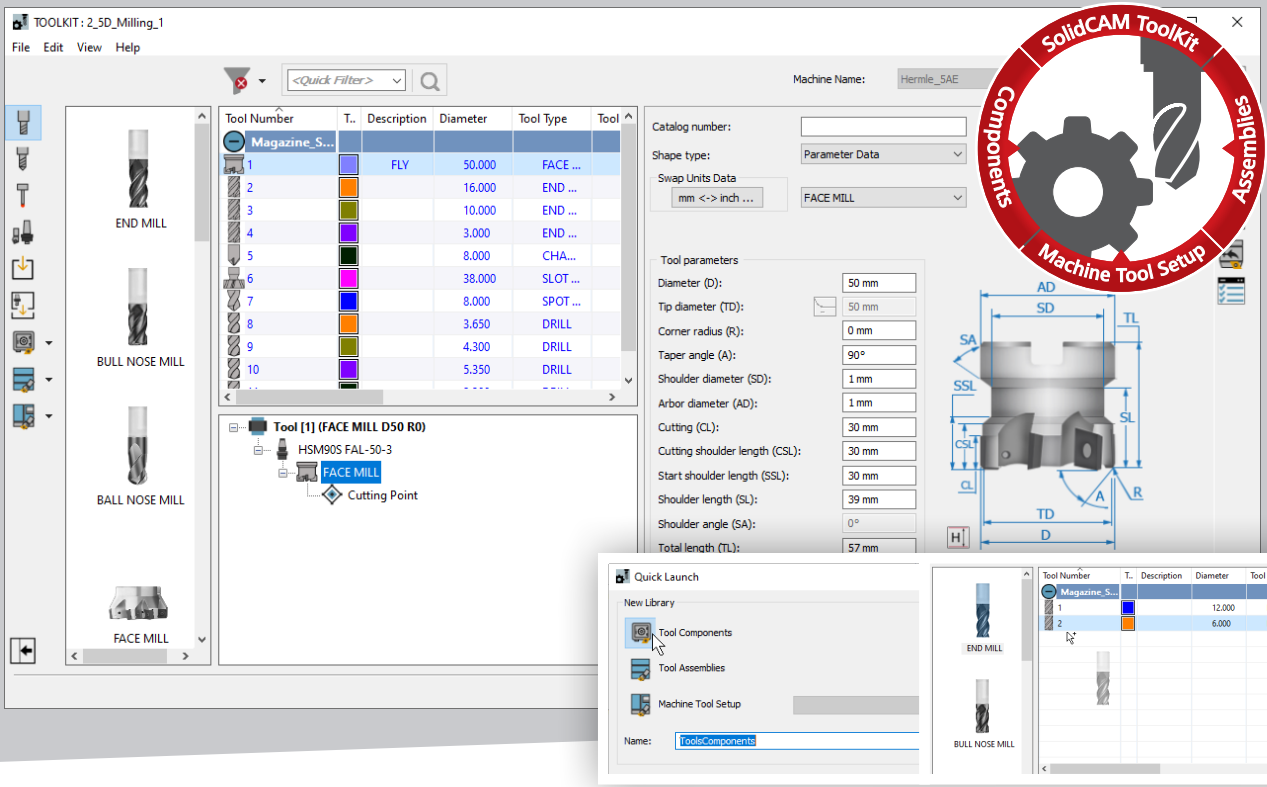
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SolidCAM Reference Guide

ToolKit

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1 Introduction

1.1 About this document

This Reference Guide is intended for all SolidCAM users.

The purpose of this document is to provide users a basic understanding of the ToolKit functionality, assist users in the getting started process and to give existing users, upgrading from SolidCAM 2020 and earlier versions, the information necessary to make the transition from Classic Tool Table to the New Tool Table as seamless as possible.

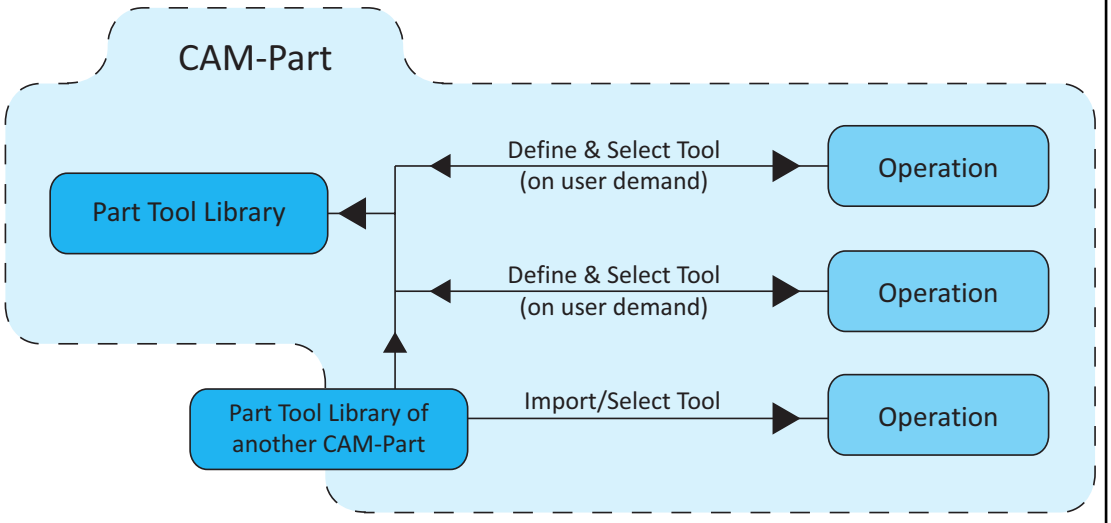
Not all ToolKit features and options will be covered in this document. For complete information, refer to the SolidCAM Help documentation. Inquiries should be directed to your Technical Support Team and/or Beta Software Supplier.

The ToolKit concept is indeed comprehensive, yet defining, editing and selecting tools for use in your operation(s) is a simple and straightforward process.

1.1.1 Document design

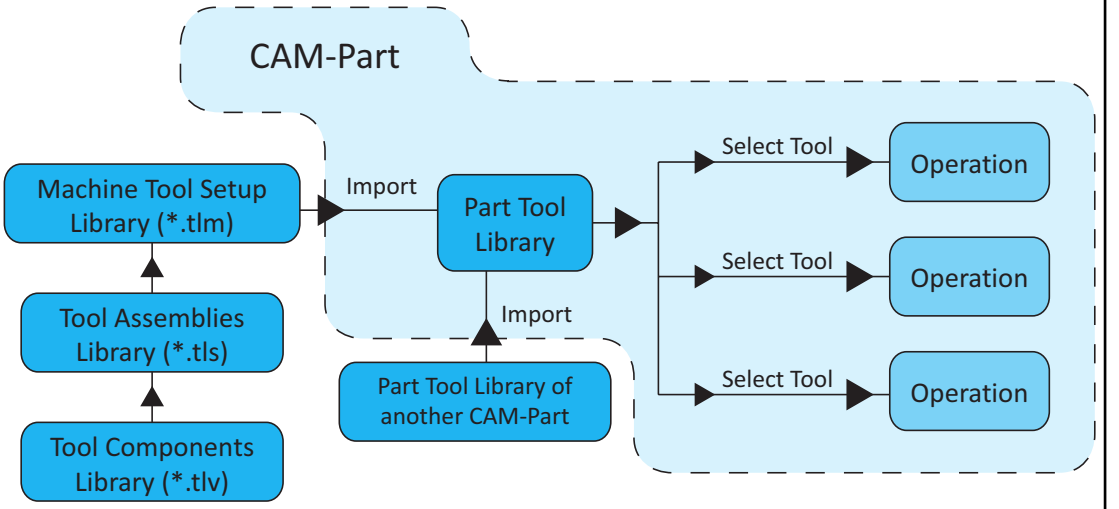
The information in this Reference Guide is presented in two main parts, for users with two different workflow styles, and it covers the important topics when working with SolidCAM's ToolKit.

Workflow Type 1 (Simple)	Users who prefer an on-the-fly way of working, where tools are often defined after setting up the CAM-Part, generally at the Operation stage.



Tools can also be modified on user demand and later acquired to suit one's needs.

Workflow Type 2 (Advanced)	Users who adhere to a more structured way of working, where it is preferred to use predefined Tool Libraries.



Serious considerations are made for inventory and availability of all tools and their components in the machine shop warehouse. Also important is the predefined matching/assembling of tools and their mounting positions on the CNC-Machine.

Illustrated in this document are examples of the two workflow types, as mentioned, and their corresponding procedures when working with ToolKit.

An elemental overview of ToolKit and its interface is provided in this Introduction chapter as it is relevant to both types of users.

1.1.2 Software versions used for this document

The screenshots in this Reference Guide were made using the latest development versions of SolidCAM 2022 integrated with SOLIDWORKS 2022, running on Windows 10.

If you are running on a different version of Windows, you may notice differences in the appearance of the menus and windows. Additionally, if a different background/color theme (i.e., light or dark) is chosen in the settings of your CAD application, you may notice differences in its appearance overall. It should be noted that these differences do not affect the performance of the software.

1.1.3 Recommended PDF Viewer for this document

It is recommended to use Adobe Reader when viewing this Reference Guide. If you do not already have Adobe Reader, it is a free PDF viewer that can be downloaded from Adobe's website using this link – https://get.adobe.com/reader/.

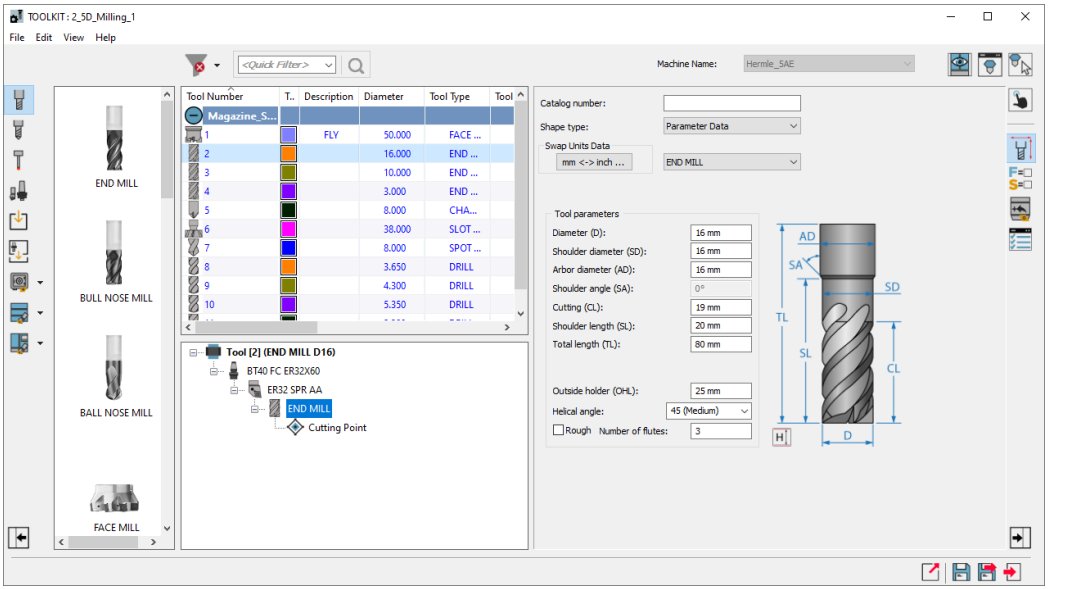
1.1.4 Conventions used in this document

This Reference Guide uses the following typographic conventions:

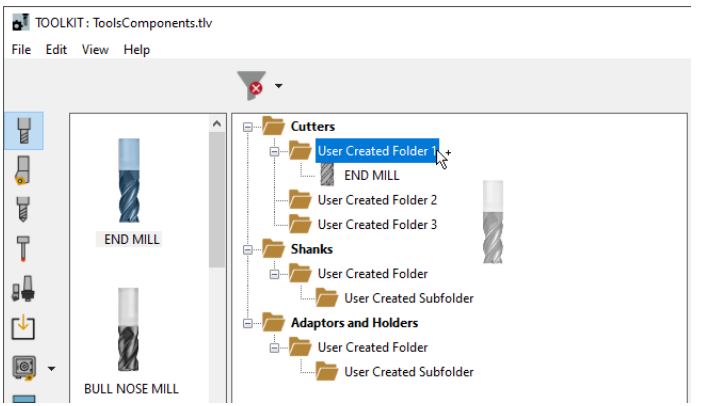
2.1.2 Adding new Tool Items...	This style indicates the topics relevant to the different Workflow Types.
 1. On the Toolbar...	The stairs icon indicates the steps used to complete applicable workflow procedures.
Bold	This style emphasizes SOLIDWORKS/SolidCAM options and commands or, in some cases, the name of buttons that require clicking. (e.g., ...clicking the **Select** button...)
 Note	The lamp icon is used for notes, tips, etc..

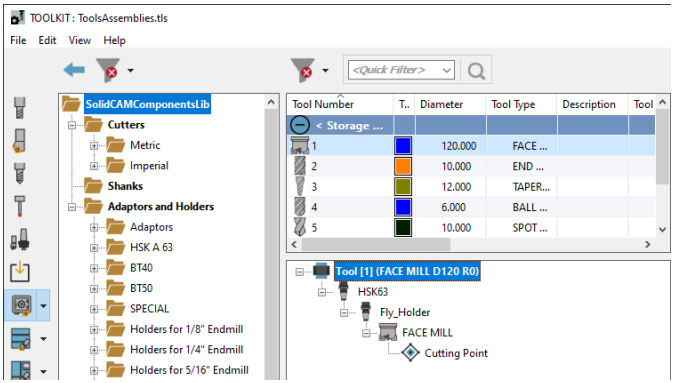
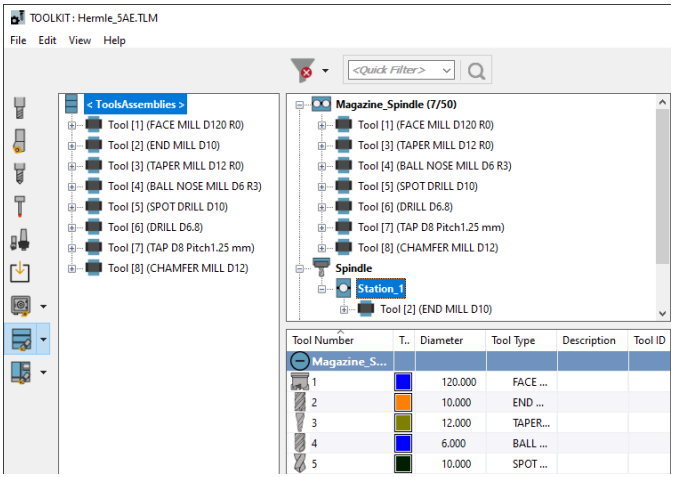
1.2 ToolKit Overview

SolidCAM's ToolKit is a feature-rich Tool Table system that facilitates better tool management and provides comprehensive Tool definition functionality.



ToolKit offers three Tool Library types, each having a graduated level of tool managing capabilities.

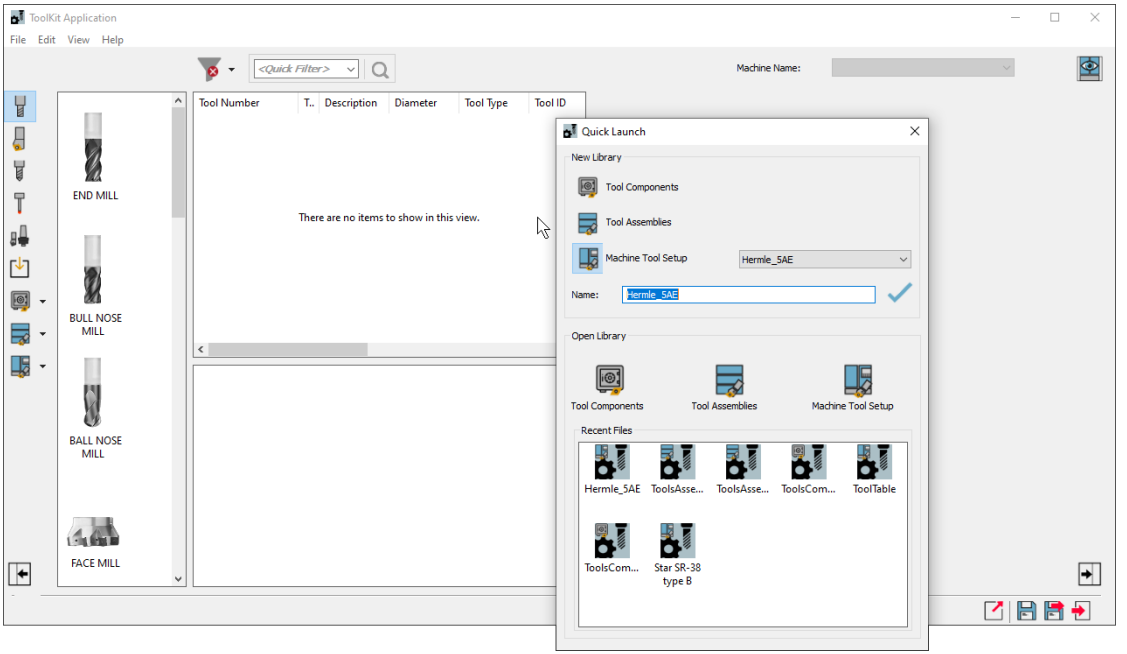
Tool Components Library	This library enables you to manage individual tool components.  A Tool Components Library contains individually defined tool components such as cutters, shanks, adaptors and holders that are typically stocked in your machine shop warehouse. Tool Components Libraries are given the *.tlv file extension.

Tool Assemblies Library	This library enables you to manage tool assemblies. Each tool assembly, referred to as a Tool Item, comprises a combination of tool components such as the cutter and the tool holding system (e.g., adaptor, extension and reduction).  A Tool Assemblies Library contains your defined tools existing in Tool Storage and prepared for use on any given CNC-Machine. Tool Assemblies Libraries are given the *.tls file extension.
Machine Tool Setup Library	This library enables you to manage Tool Items in the Machine environment, according to the VMID of a specific CNC-Machine.  (Advanced View shown for illustrative purposes) A Machine Tool Setup Library contains your defined tools and their mounting positions on the CNC-Machine tool holding devices (i.e., in the Magazine and Tool Stations of the Turret(s)). Machine Tool Setup Libraries are given the *.tlm file extension.

You can define any number of Tool Library variations. By default, the corresponding files are saved in the folder C:\Users\Public\Documents\SolidCAM\SolidCAM2022\Tables\Metric or ...\Inch depending on your SolidCAM Units selection.

The installation of SolidCAM includes a standalone version of ToolKit that enables you to access and manage your Tool Libraries totally independent of your CAD/CAM software. The full path to ToolKit standalone is C:\Program Files\SolidCAM2022\Solidcam\Toolkit-Y.exe

New and existing Tool Libraries can be opened from ToolKit's Quick Launch dialog box and managed using the standalone application.



Each Tool Library type enables you to perform different tasks. Whether creating a new or editing an existing, the Tool Library selection determines the Tool Table features and options that will appear in the ToolKit interface, thus making its GUI adaptive to the current task(s).

The final level of tool management is performed in the Part Tool Library.

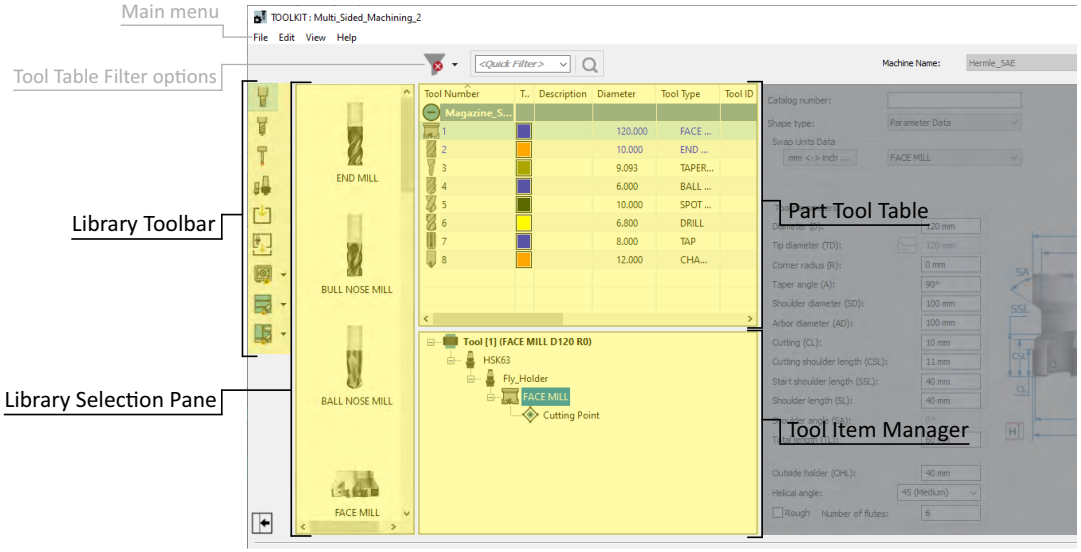
The Part Tool Library contains all the tools available for use in a specific CAM-Part and enables you to manage them in accordance with the CNC-Machine chosen for the CAM-Part Definition. The Part Tool Library and Machine Tool Setup Library interfaces are therefore identical in appearance, features and options.

The Part Tool Library, given the .tlp file extension, is stored within the CAM-Part.

SolidCAM enables you to import tools from any ToolKit Tool Library into the Part Tool Library. Predefining Tool Libraries and importing tools, however, is not required. If preferred, your tools can be defined at the Operation stage or you can import and select tools from the Part Tool Libraries of existing CAM-Parts.

1.3 ToolKit Interface

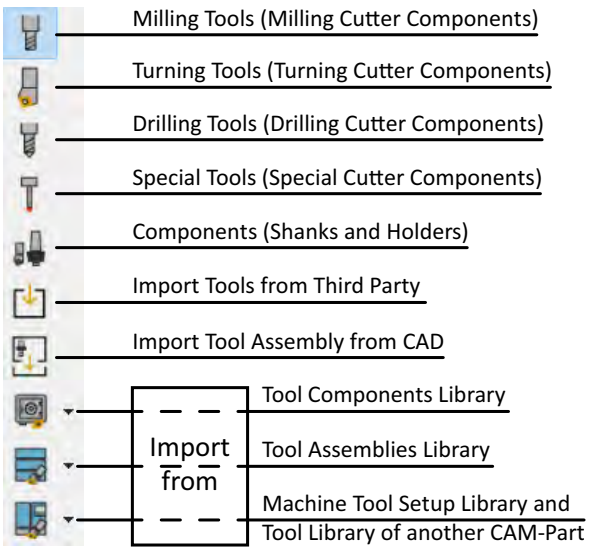
The left side of the TOOLKIT dialog box contains a number of important interface features.



The ToolKit main menu and Tool Table Filter options are covered later in this document.

Library Toolbar and Selection Pane

The Library Toolbar and corresponding Selection Pane enables you to add/import tools to the Part Tool Library using the available SolidCAM Components or the various import options.



(All available SolidCAM Components are shown)

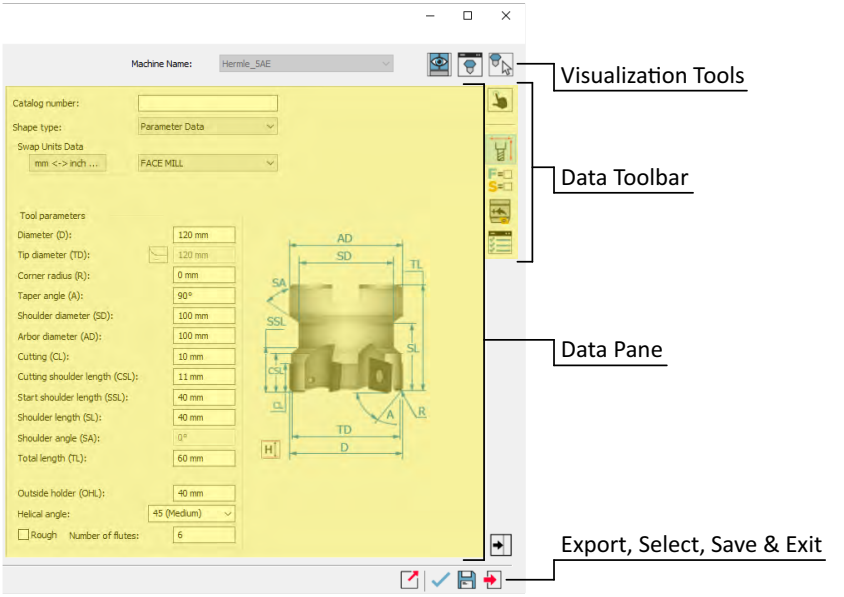
Part Tool Table

The Part Tool Table is the Tool Library that contains a structured list of all tools available for use in your CAM-Part. The Part Tool Table enables you to select a tool for the current operation, view the tool assembly in the Tool Item Manager, etc.

Tool Item Manager

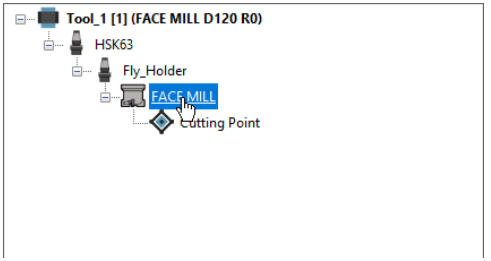
The Tool Item Manager displays the selected tool in a hierarchical tree structure as an assembly of individual tool components. The Tool Item Manager enables you to view/edit complete information about the tool.

The right side of the TOOLKIT dialog box displays the Tool Item/Component Data.



The Data Toolbar and corresponding Data Pane enables you to manage the combined and individual properties of the tool and its assembled components.

The Data Toolbar buttons and Data Pane parameters and options will vary depending on your selection in the Tool Item tree.

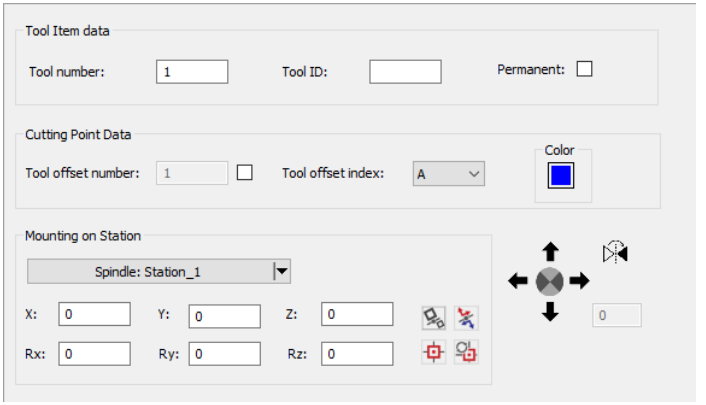


All available Toolbar buttons and their functions are summarized below.

Quick Access Data



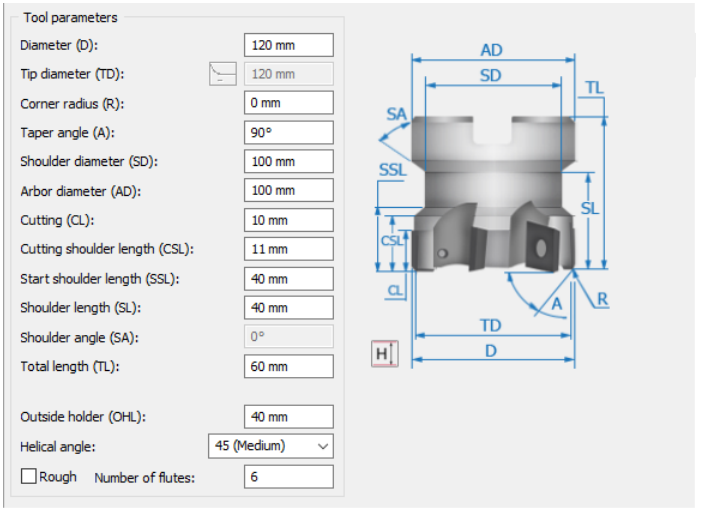
Quick Access enables you to define the tool identifiers (i.e., the Tool Number, Tool ID, Tool offset number/index, etc.) and allows you to quickly change the tool mounting position on the CNC-Machine.



Topology Data

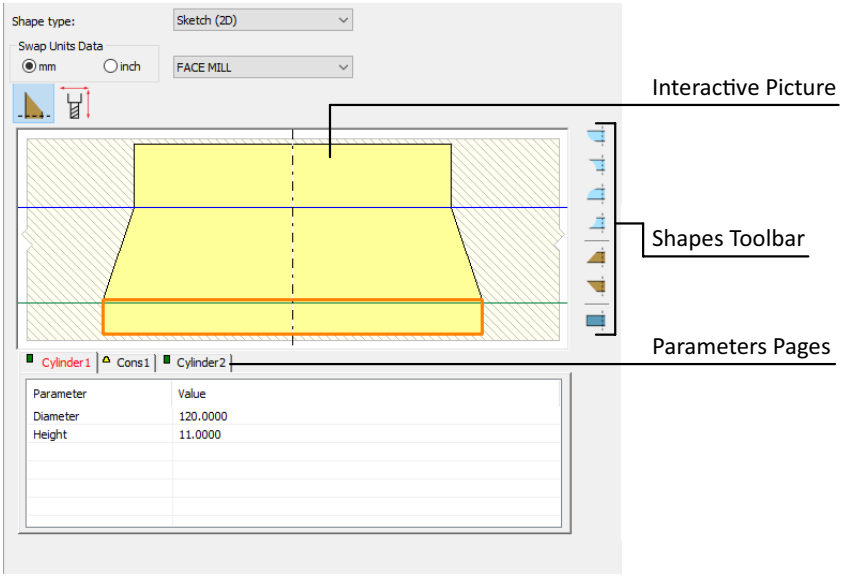


Topology enables you to define the geometry of the tool components according to the chosen Shape type.



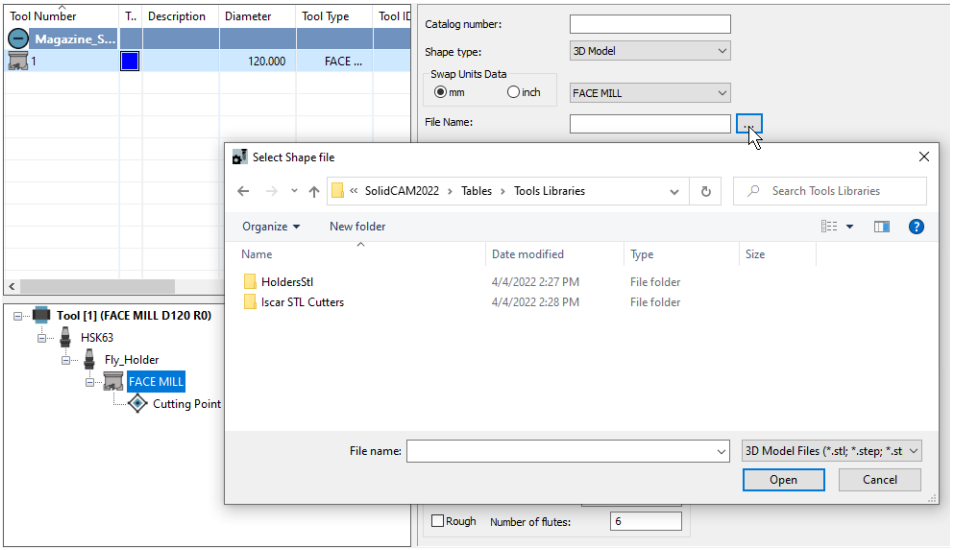
Different components (e.g., Cutter vs. Holder) offer different Shape type options, which determine how you want to specify the geometry. A Cutter component, for example, can be defined by its dimensions using the Parameter Data pictured in the image or it can be defined via a 2D Sketch or a 3D Model.

Using the Sketch (2D) option, components can be defined by segment shapes such as cylinders, cones and toruses. Shapes can be added using the Toolbar or the Interactive Picture right-click commands.



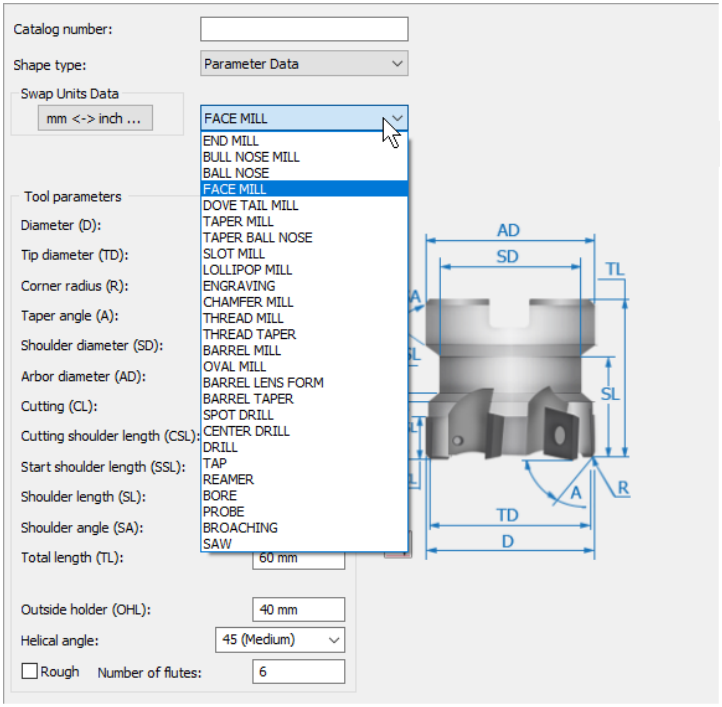
The actual dimensions can be edited by selecting the segments on the interactive picture and by using the corresponding Parameters pages.

Using the 3D Model option, components can be defined by a STL or STEP file.



Clicking the ... button displays the Select Shape file dialog box, where you can browse for, select and open the STL/STEP file of your choosing.

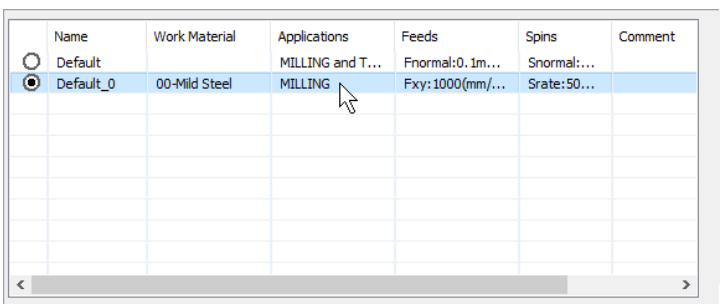
Additionally, each component can be given a Catalog number for referencing its catalog resource (if any). Options are also provided to quickly Swap Units Data and, for Cutters and Shanks only, to make on-the-fly Tool Type changes.



Cutting Condition Data

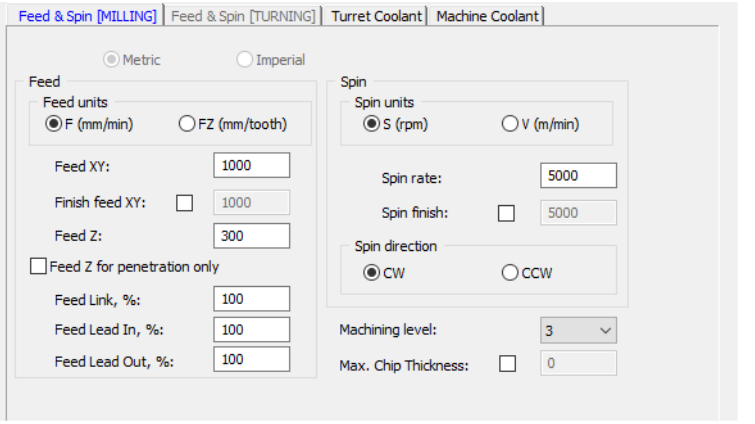
Cutting Condition enables you to define, for Cutter components only, default sets of cutting parameters for the machining of different Work Materials and for use in different Applications (MILLING and/or TURNING).

When a tool is added to the Part Tool Table, it is given a default set of Cutting Conditions which is indicated in the table.



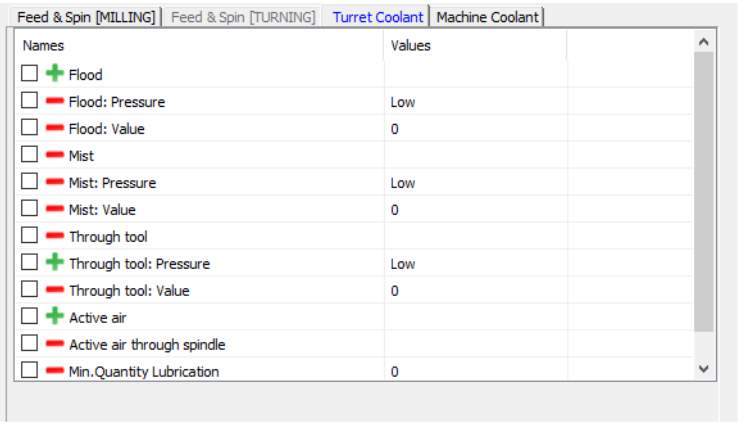
Using the table, you can manage any number of Cutting Condition sets.

For each Cutting Condition set, you can specify the default Feed & Spin data (initial values are defined in SolidCAM Settings).



When only MILLING is selected in the Applications column of the table, for example, only the Feed & Spin [MILLING] tab is active and the values can be edited.

For each Cutting Condition set, you can also specify the default Turret and Machine Coolant options.



If specified in the Tool definition, the default Coolant options are automatically applied to the chosen tool but can be changed for each operation.

The coolant options available for use for the chosen Turret and on the current CNC-Machine, according to the VMID settings, are marked with the ✚ icon. The coolant options not available for use for the chosen Turret and on the current CNC-Machine, according to the VMID settings, are marked with the ▬ icon.

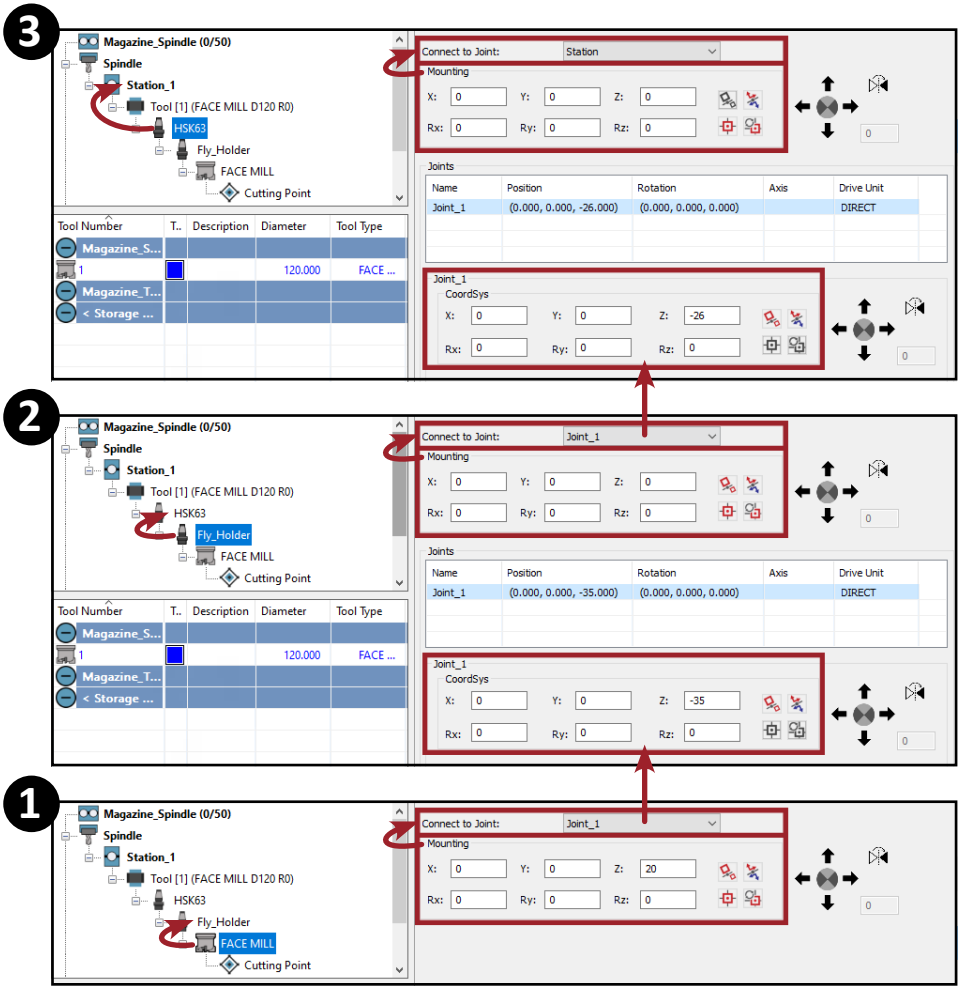
All the existing coolant types are listed with corresponding check boxes to enable or disable any of the available options.

Connection Data

Connection enables you to define, using Mounting and Joint CoordSys positions, the geometric relationships between components in the tool assembly. Default positions are provided in accordance with the components geometry.

In the following example, there are three connections:

1. Mounting position of cutter is connected to Joint_1 of the holder (Fly_Holder)
2. Mounting position of holder is connected to Joint_1 of the adaptor (HSK63)
3. Mounting position of adaptor (topmost component) is connected to the Station CoordSys position of the Spindle

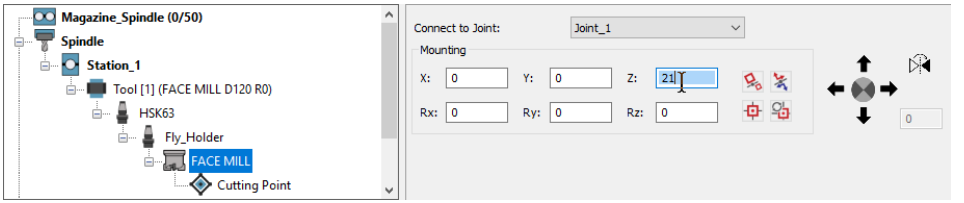


(Advanced View shown for illustrative purposes)

Except for the topmost component, the Mounting position of each component is connected to the Joint CoordSys position of the higher component. By clicking a component in the Tool Item tree, its Connection Data can be viewed and edited.

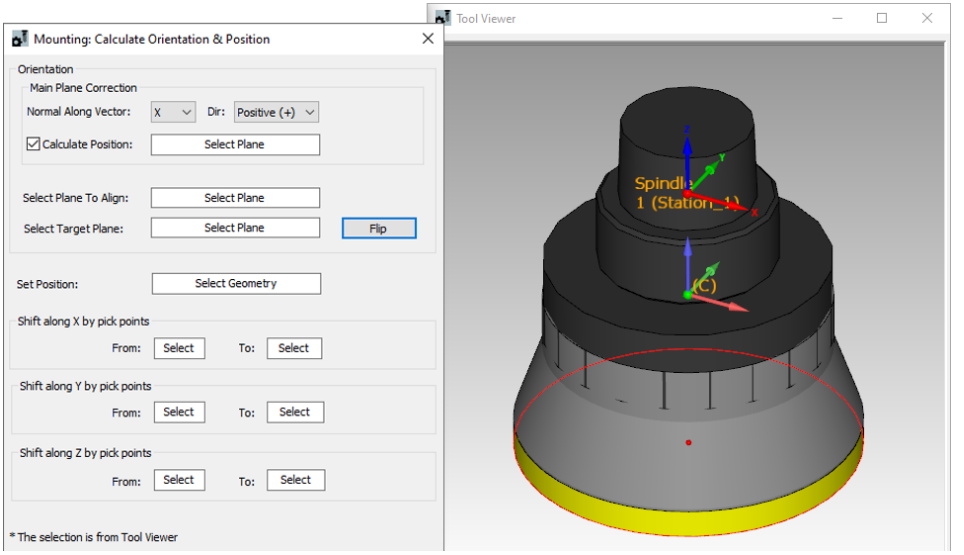
Modifying the Mounting/Joint CoordSys positions

The positions can be modified with linear and/or rotational movements using the corresponding fields. For example, you can shift a position along the X-, Y- and Z-Axes by entering a value in the X, Y and Z fields or by scrolling your mouse wheel to change the value gradually.



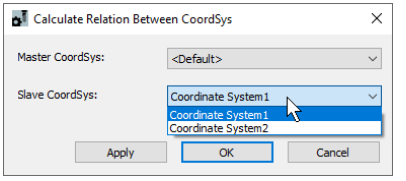
Additionally, the positions can be modified using the following options:

Calculate Orientation & Position – displays the Calculate Orientation & Position dialog box with Tool Viewer window that enables you to set the Coordsys by picking planes and by shifting along X, Y and Z pick points.



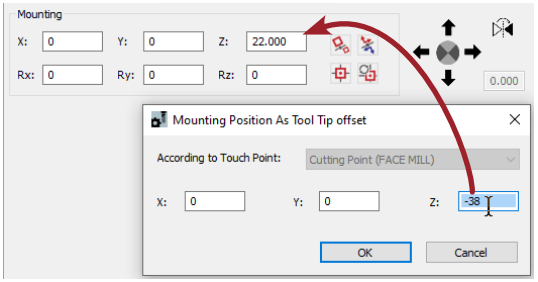
Calculate CS – displays the Calculate Relation Between CoordSys dialog box that enables you to set the CoordSys according to the orientational/positional relationship between two coordinate systems.

Master CoordSys selection can be <Default>, that which is provided in accordance with the components geometry, or a coordinate system defined in the CAD file. Slave CoordSys selection must be a CAD coordinate system.



For Cutter components, the default Mounting position is the tool tip. The positions of the tool components can be set/reset using the following options:

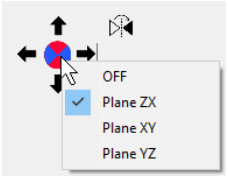
Set Position – displays the Mounting Position As Tool Tip offset dialog box that enables you to set the Mounting position according to the available Touch Point(s), the selected Touch Point being a Cutting or Non-Cutting Point. The X, Y and Z fields define the offsets.



As the X, Y and Z values are modified, the corresponding fields in the Mounting section of the Connection Data are updated to display the relative distance from the selected Touch Point to the Joint chosen in the Connect to Joint drop-down menu.

Reset Position – resets the Mounting position to the Cutter component tool tip. Accordingly, the updated values are populated to the Mounting fields.

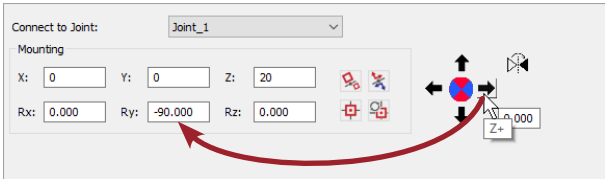
The Mounting/Joint CoordSys positions can also be modified with plane rotations about the X-, Y- or Z-Axis using the following icon.



Right-clicking the icon enables you to choose the plane to be rotated:

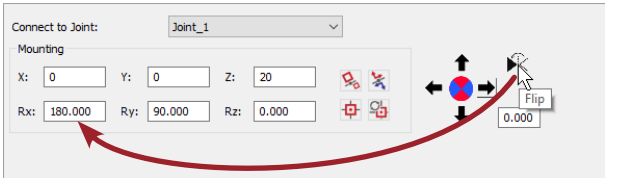
- **Plane ZX** – rotates the ZX-plane about the Y-Axis
- **Plane XY** – rotates the XY-plane about the Z-Axis
- **Plane YZ** – rotates the YZ-plane about the X-Axis

The ➡ ⬇ ⬅ ⬆ buttons enable you to rotate in 90° increments the selected plane around the rotation axis.



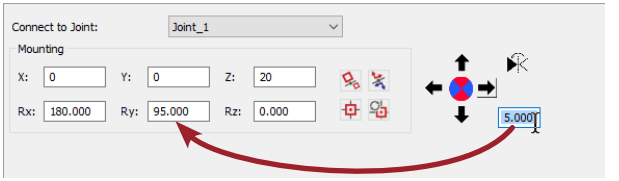
Accordingly, the corresponding component is turned and the angle value is reflected in the rotation axis field.

The  button enables you to flip the selected plane and the rotation axis 180°; the rotation axis faces the opposite direction.



Accordingly, the corresponding component is turned and the angle value is reflected in the rotation axis field.

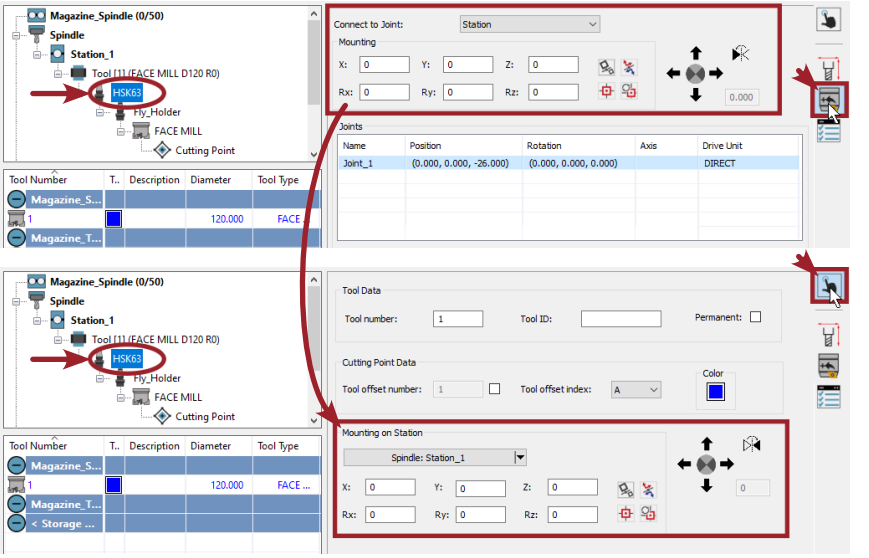
The plane rotation text field enables you to enter an angle value manually or change the value gradually by scrolling your mouse wheel.



Accordingly, the corresponding component is turned and the angle value is reflected in the rotation axis field.



The Mounting parameters of the topmost component in the Tool Item tree coincides with the Mounting on Station parameters in Quick Access Data.

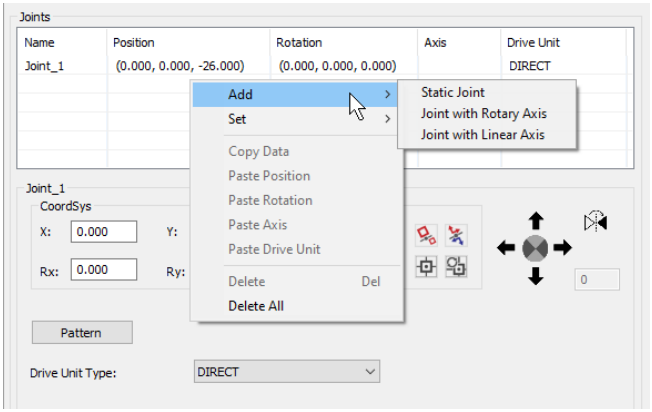


(Advanced View shown for illustrative purposes)

For all tool components, there can be only one Mounting position.

Working with the Joints Feature

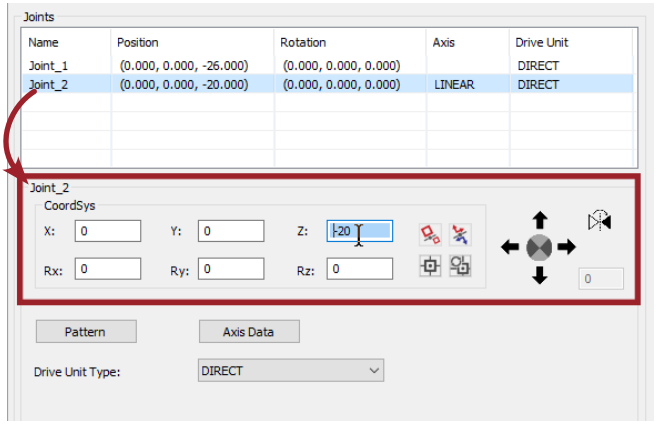
The Joints feature in Connection Data enables you to easily manage complex tool assemblies. When working with a complex tool assembly, multiple joints can be defined as necessary using the Joints table.



The Joints table right-click commands are as follows:

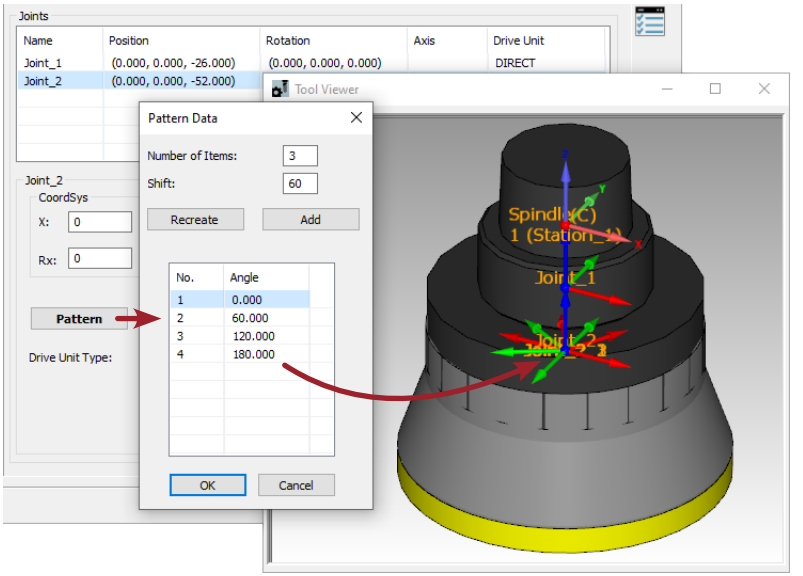
- **Add** – adds a new joint to the tool component and to the table for editing; it can be either a Static Joint, Joint with Rotary Axis or Joint with Linear Axis
- **Set** – sets the joint selected in the table as Static Joint, Joint with Rotary Axis or Joint with Linear Axis
- **Copy Data** – copies the data of the selected joint to enable pasting its data to another joint
- **Paste** – pastes the copied data (Position, Rotation, Axis or Drive Unit) individually from one joint to another joint
- **Delete/Delete All** – deletes the selected joint only or all joints at once

Joint CoordSys enables you to modify the orientation and position of the selected joint. As mentioned in the **Modifying the Mounting/Joint CoordSys positions** section, the position can be modified by shifting/rotating or by using the available options.

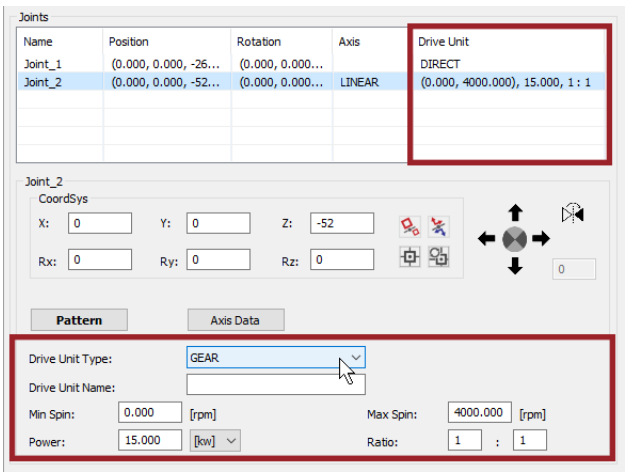


For the selected joint, you can also define the following:

- **Pattern** – this button displays the Pattern Data dialog box that enables you to create a rotation pattern for the joint.



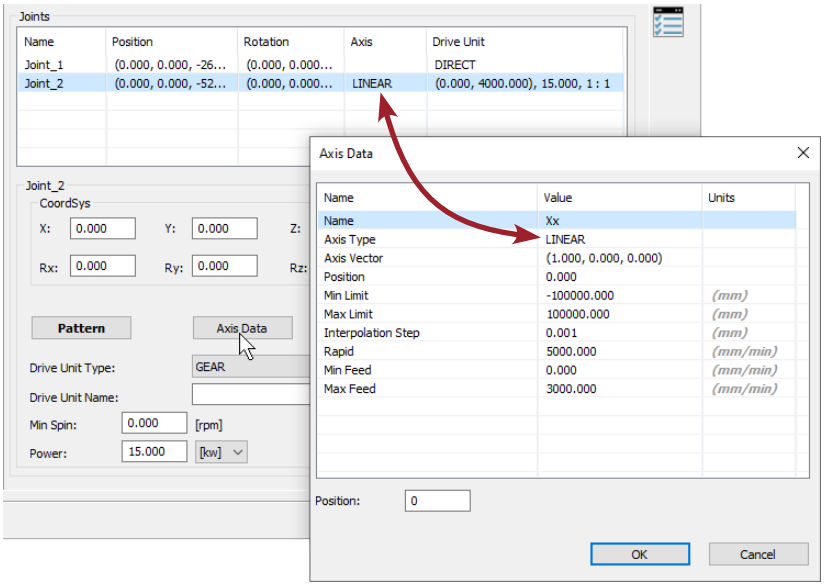
- **Drive Unit Type** – this option enables you to define for the joint the relative turret motor and transmission parameters, including the Min/Max Spin, the Power and, when GEAR is chosen in the drop-down menu, the Final Gear Ratio.



The Drive Unit definition is reflected in the Drive Unit column of the Joints table.

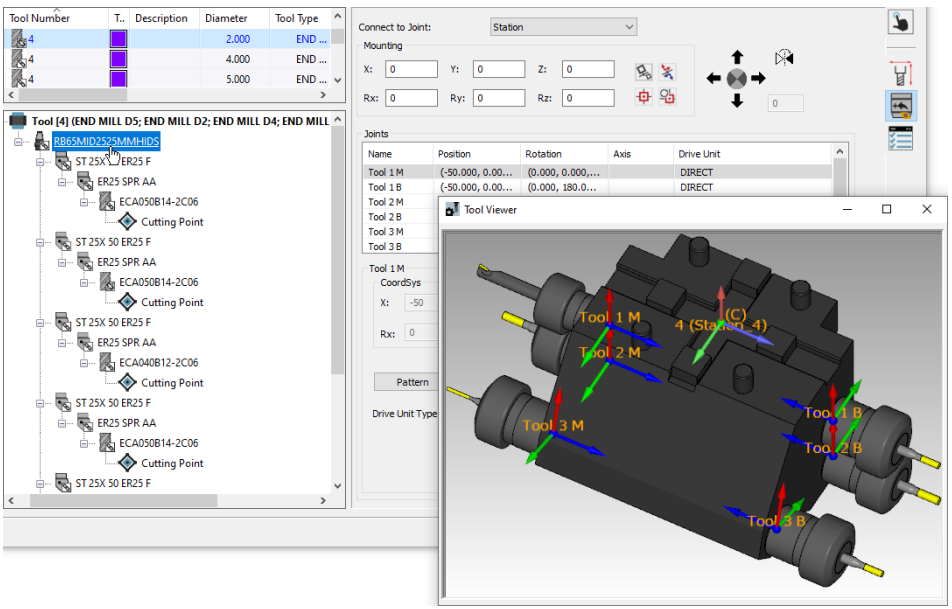
- **Axis Data** – this button is available for joints with a rotary/linear axis and it displays the Axis Data dialog box.

The Axis Data dialog box enables you to define the rotary/linear axis parameters of the joint.



Example – Tool Item with Multiple Joints

The Joints feature was used to create the below example, a complex tool assembly with mounting capacity for six tools. Each Cutter component is mounted to additional Shank components, which are mounted to the six defined joints of the topmost component (Adaptor/Holder RB65MID2525MMHIDS in the Tool Item tree).



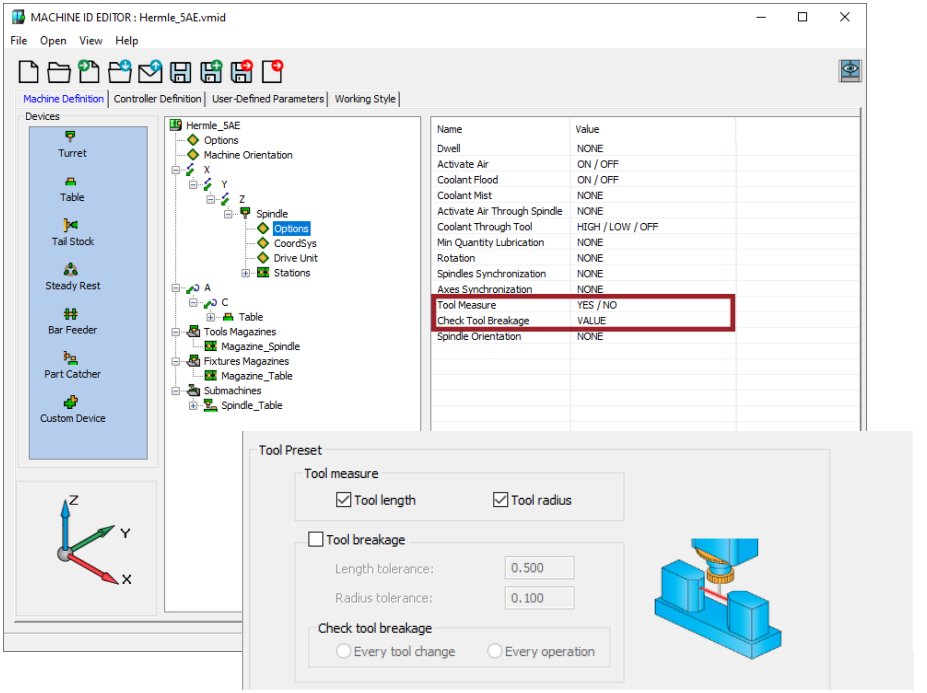
Offset Data

Offset enables you to define the Cutting Point and Tool Preset data of the tool as well as the relative Tool offset parameters for identifying its default position in the Tool Offset table of the CNC-Machine.

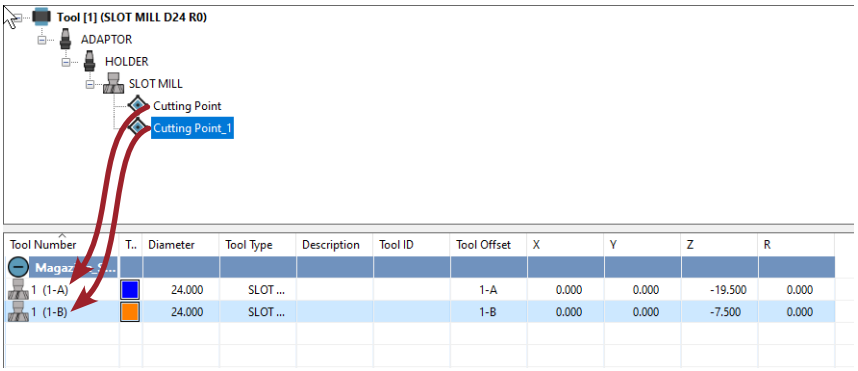


With the exception of Tool offset number/index, also available in Quick Access Data, the Offset button and its corresponding parameters are available only when the Cutting Point of the tool is selected in the Tool Item tree.

Tool Preset options are activated only if your CNC-Machine supports tool presetting and if, in the *.vmid file, Tool Measure and/or Check Tool Breakage for the Turret on which the tool will be mounted is set accordingly.



ToolKit supports multiple cutting points on a single Cutter component. If, for example, you want to use both top and bottom cutting faces of a SLOT MILL, a Cutting Point can be added using the Tool Item Manager right-click commands.



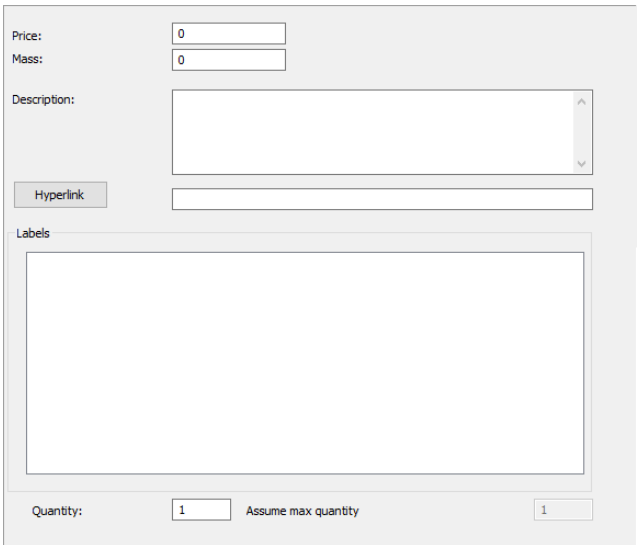
(Swap Views shown for illustrative purposes)

Upon adding a Cutting Point, it is assigned the next available Tool offset index, and the tool can be quickly and easily identified by its cutting points in the Tool Table list. When the tool is modified, all the shared properties are maintained.

Properties Data



Properties enables you to define additional information relevant to the tool and its assembled components, including Price and Mass (for each and sum total of all), Descriptions, Quantities, Hyperlinks for referencing local or online resources and, for the GCode, Tool Item Messages, etc.



The parameters and options in Properties Data will vary depending on your selection in the Tool Item tree.

ToolKit Visualization Tools

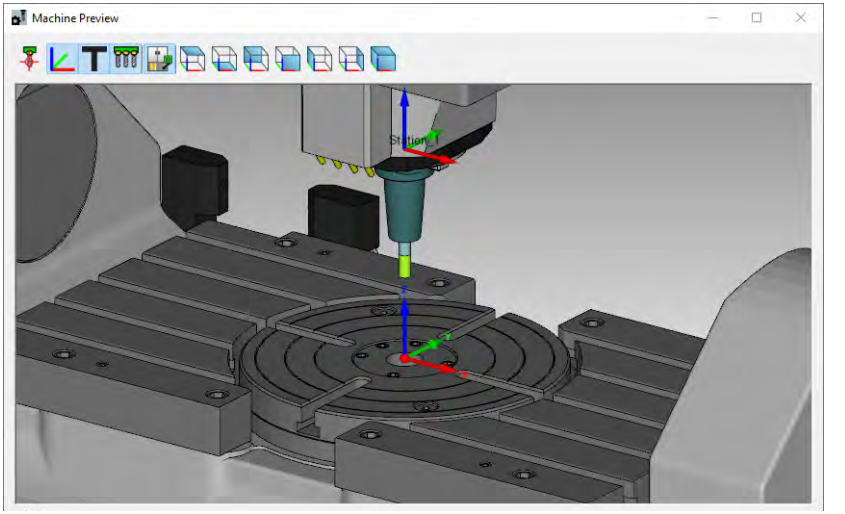
ToolKit offers several visualization tools that are accessible at the upper right corner of the TOOLKIT dialog box.





Machine Preview

Clicking this button displays the Machine Preview window.

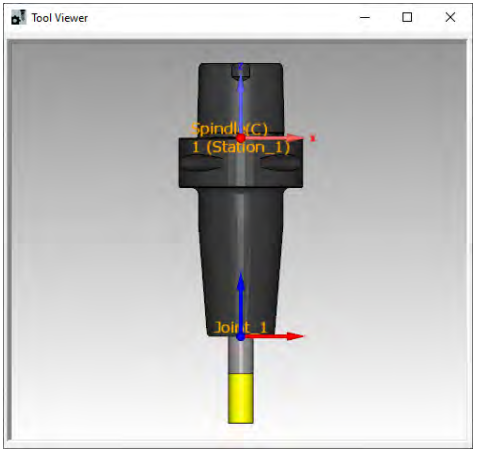


This window enables you to visualize the tool, its schematic position relative to the Machine Coordinate System and in connection with the machine devices, without entering the SolidCAM Machine Simulation mode.

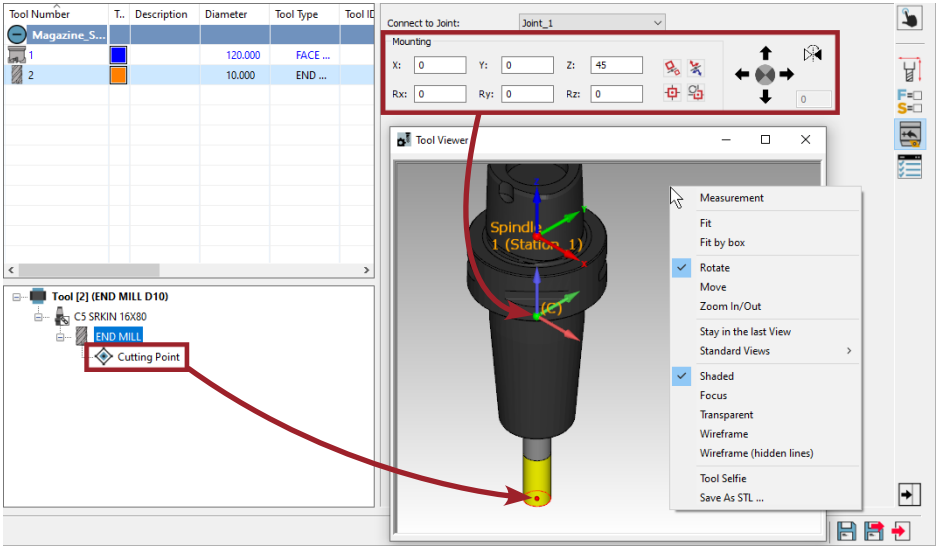


Show/Hide Tool Picture

Clicking this button shows/hides the Tool Viewer window that enables you to visualize a 3D model representation of the tool.



This window dynamically displays the Mounting/Joint CoordSys positions of the selected component in the Tool Item tree. When the Cutter component is selected, the Cutting Point(s) of the tool is also displayed.

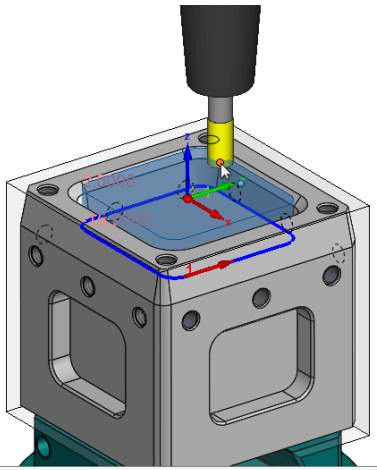


Additionally, this window enables you to view the tool in different orientations and display styles, take measurements of geometrical elements (i.e., lines, arcs, splines, edges and points), etc.



Show/Hide Tool Near Mouse

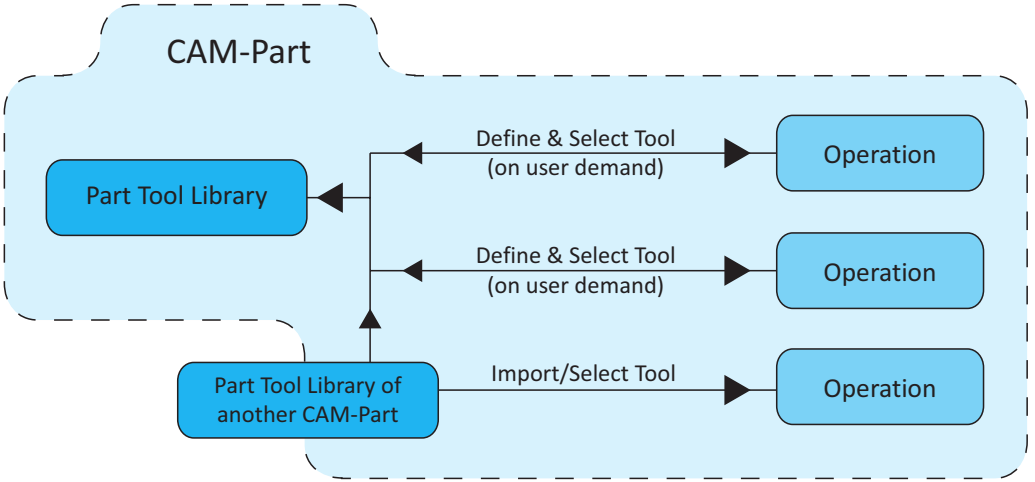
Clicking this button toggles on/off a 3D graphic representation of the tool in the SOLIDWORKS Graphics Area for performing a visual tool check.



The displayed tool follows the mouse pointer, and it can be dynamically zoomed and rotated together with the 3D CAD model.

Simple Workflow 2

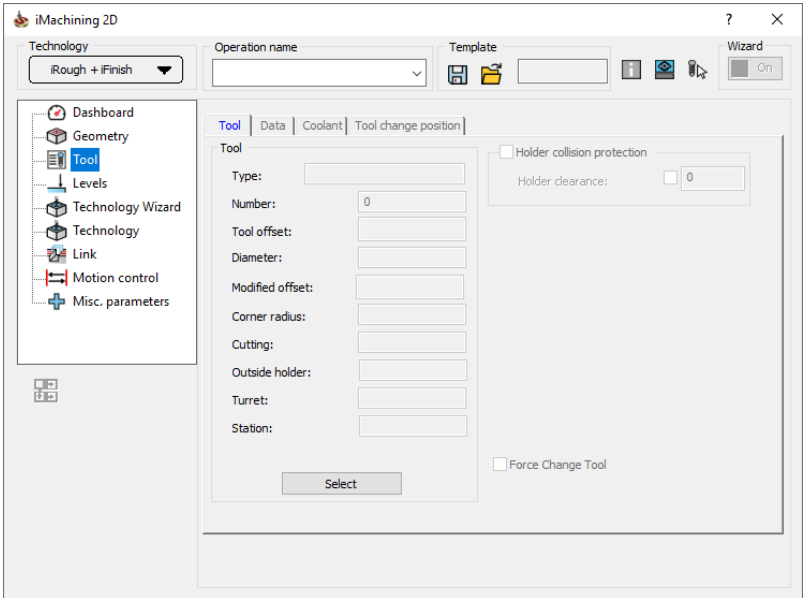
For Workflow Type 1 users, the Tool definition and selection in SolidCAM is generally performed at the Operation stage. Users who prefer this working style may also regularly import tools from the Part Tool Library of an existing CAM-Part in order to save time.



2.1 Working with ToolKit at the Operation stage

2.1.1 Opening ToolKit and starting the Tool definition

The Tool definition and selection for the operation is started by clicking the **Select** button on the Tool page of the Operation dialog box.

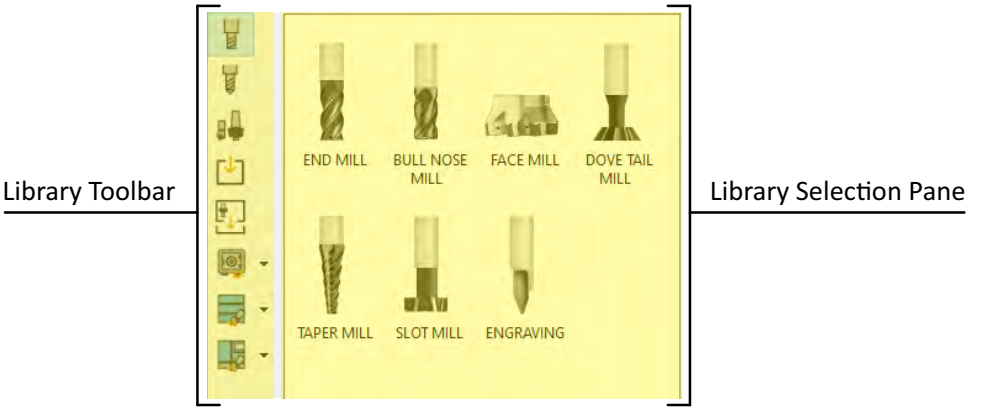


When the Tool Table list is empty or if no suitable tools are available for selection, one must be added and defined accordingly.

If Operation default templates are selected in the SolidCAM Settings or in the Part Settings, a tool suitable for the current operation may be loaded automatically. In such cases, the tool can be edited per your requirements.

Tools can be added to the Part Tool Library using the available SolidCAM Components or by importing them from a Third Party source, from CAD or from any predefined ToolKit Tool Library.

At the Operation stage, the availability of Cutter Components (on the Toolbar) and corresponding Cutter/Tool Types (in the Selection Pane) are determined by the CAM-Part type and the Operation type respectively.

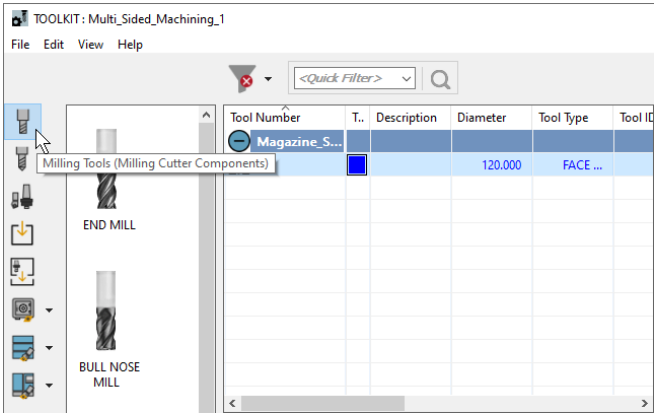


In the following example, a new tool is defined for a Milling CAM-Part and for use in an iMachining operation.

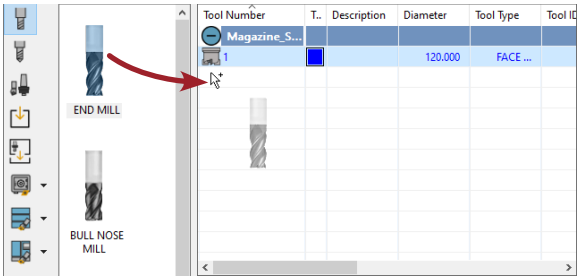
2.1.2 Adding a new tool and defining its properties

The steps to add and define a new tool using the available SolidCAM Components are as follows:

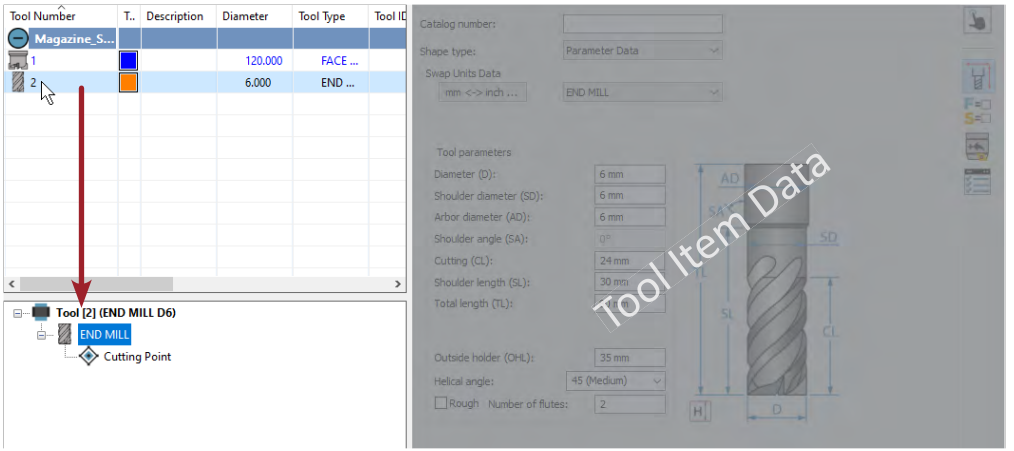
1. On the Toolbar, choose the **Cutter Components** you want displayed in the Selection Pane.



2. In the Selection Pane, double-click the desired Cutter/Tool Type or simply drag & drop it into the Tool Table window.



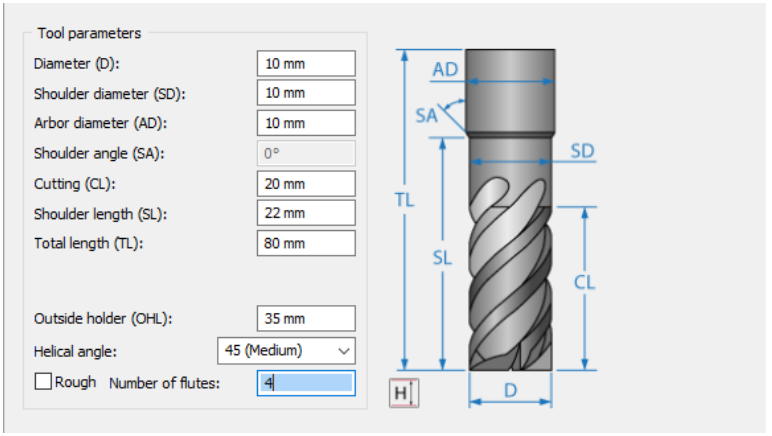
Upon adding the Cutter to the Part Tool Table, the corresponding Tool Item is created and displayed in the Tool Item Manager.



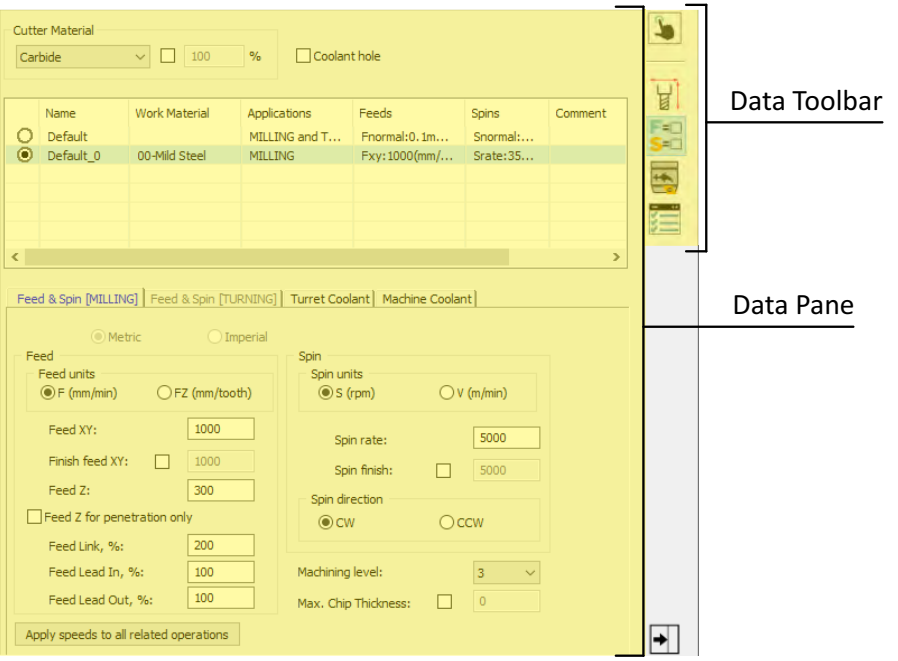
3. Define the Cutter geometry.



Topology is the default selection on the Data Toolbar, prompting you to specify the Cutter dimensions in the Data Pane.



4. Use the Data Toolbar and corresponding Data Pane to define the remaining properties such as the Feed & Spin data.



2.1.3 Tool properties important in iMachining operations

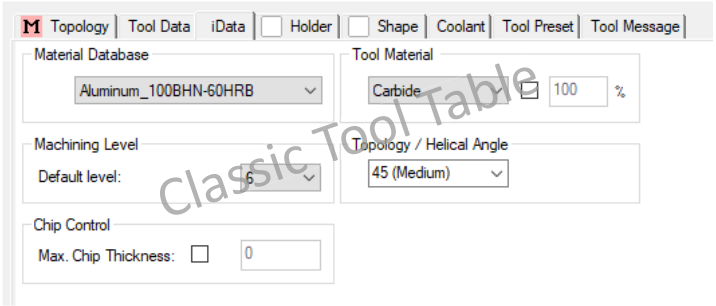
If you are defining a tool for use in iMachining operations, specifying the Feed & Spin data is not necessary. The iMachining Technology Wizard uses the Cutter geometry and other data to automatically calculate optimal sets of Cutting conditions specific to each operation.

The Topology parameters important in iMachining operations include:

- **Diameter (D):** defines the Cutter diameter, which affects the automatically calculated values of several parameters in iMachining operations.
- **Outside holder (OHL):** defines the Cutter length outside the Holder. The Technology Wizard modifies the chip thickness taking into account the Outside Holder Length (OHL). As the OHL increases, the chip thickness is reduced.
- **Cutting (CL):** defines the length of the Cutter that performs the cutting. The Technology Wizard in Automatic mode uses the Cutting length (CL) to calculate if multiple steps are needed to achieve the total depth.
- **Number of flutes:** defines the number of Cutter teeth. The Technology Wizard relies on the correct number of flutes to ensure the appropriate chip thickness is calculated per tooth. Changing the number of flutes will change the feed rate of the Cutting conditions.

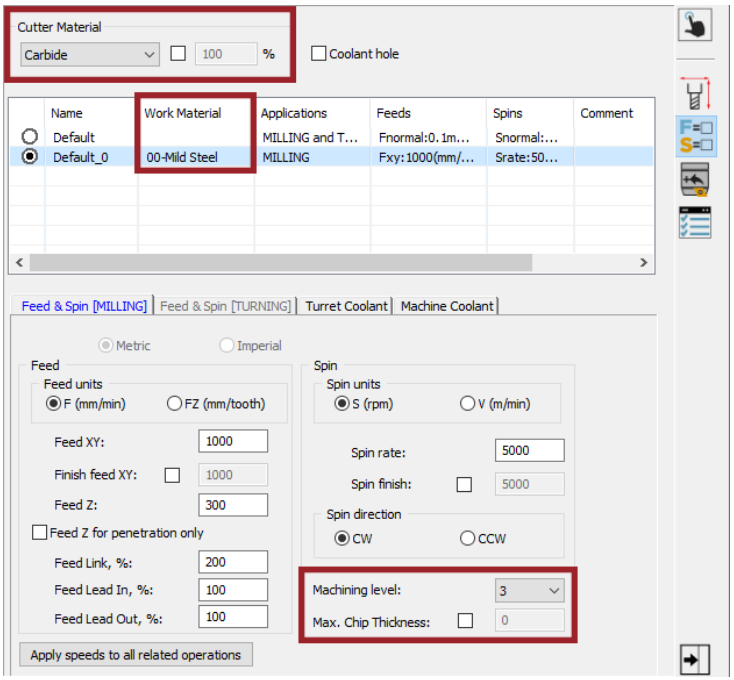
In addition to the parameters above, described below are several other Cutter properties specific to and used in iMachining operations.

In SolidCAM 2020 and earlier versions with Classic Tool Table, the iMachining specific properties could be found on the iData tab as shown.



In current SolidCAM versions with ToolKit, the iMachining specific properties can be found in the Cutting Condition Data and Topology Data.

Cutting Condition Data



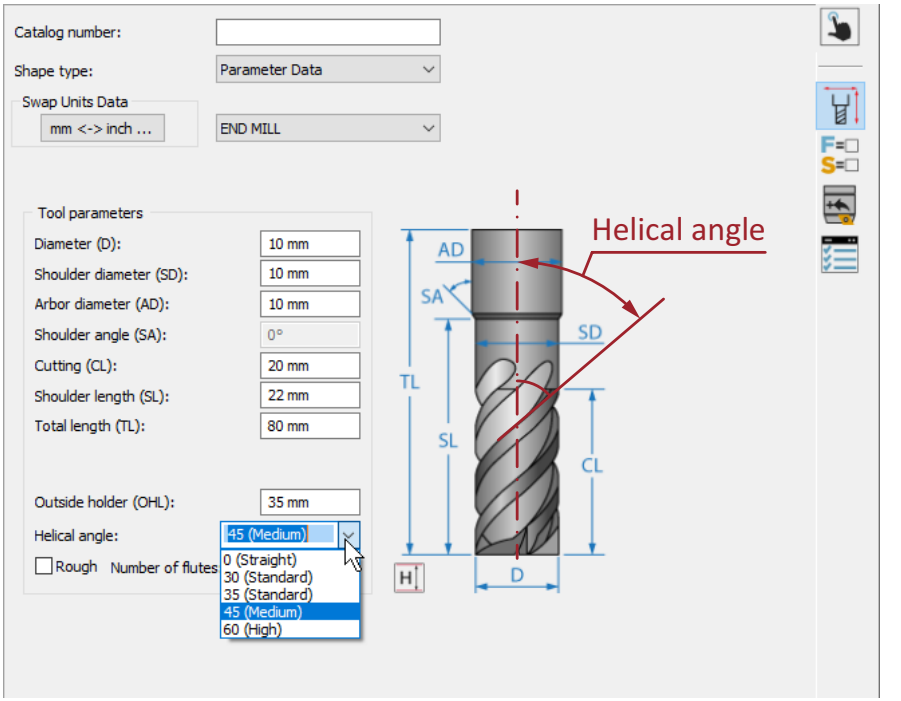
The **Cutter Material** option appears in the Cutting Condition Data. The predefined list enables you to select a given type of material from which the Cutter is made. The Cutter Material selection affects the Max. Cutting Speed adjustments generated by the Technology Wizard along with the associated Feed Rate Max and Max. Spin parameters.

The **Material Database** selection appears in the Work Material column of the Cutting Condition table. When clicking the cell, the appearing list enables you to define a different work material that is associated to the Cutter.

The **Machining level** and **Max. Chip Thickness** options appear on the Feed & Spin [Milling] tab of the Cutting Condition Data.

- **Machining level** enables you to assign a level different than the CAM-Part default that is specific to each tool.
- **Max. Chip Thickness** is automatically calculated by the Technology Wizard, but you can instead specify a preferred value that is associated to the Cutter (e.g., that which is recommended by the tool manufacturer for the given type of Cutter Material).

Topology Data



The **Helical angle** parameter appears in the Topology Data and is especially important for calculating depths based on Axial Contact Points (ACPs) of the Cutter. You can select from the predefined list a typical flute helical angle or you can manually enter a value.

The iMachining specific properties must be entered accurately in order for the Technology Wizard to calculate the correct and optimal Cutting conditions.

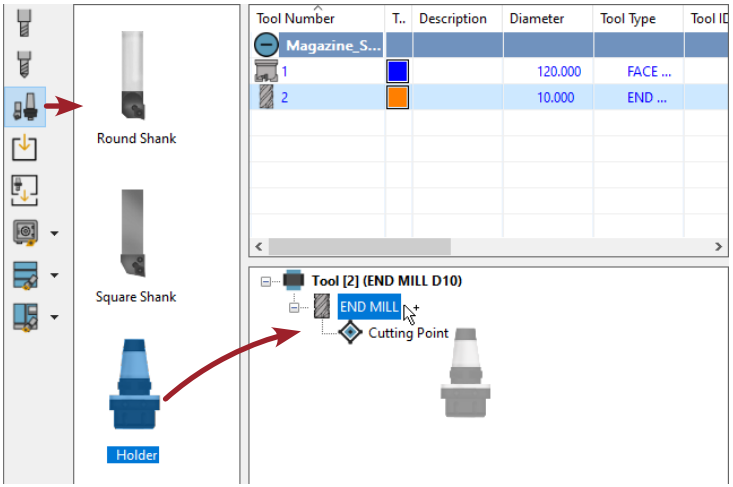
Once the Cutter component is added and its properties defined, you can create the tool assembly by adding Adaptor and Holder components.

2.1.4 Creating the tool assembly (Tool Item)

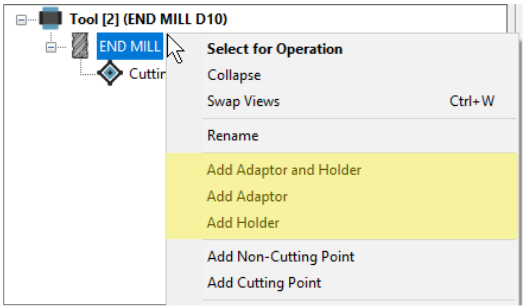
Additional components, such as Adaptors and Holders, can be added to the tool in several ways by using either automatic or manual methods.

You can manually add Adaptors and Holders using one of the following:

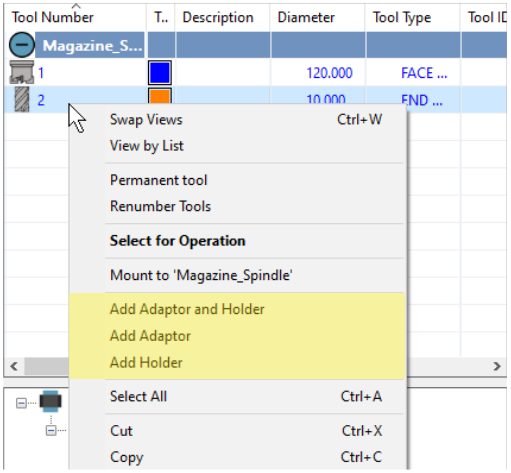
- Library Toolbar and Selection Pane to select from the available SolidCAM Components



- Tool Item Manager right-click commands

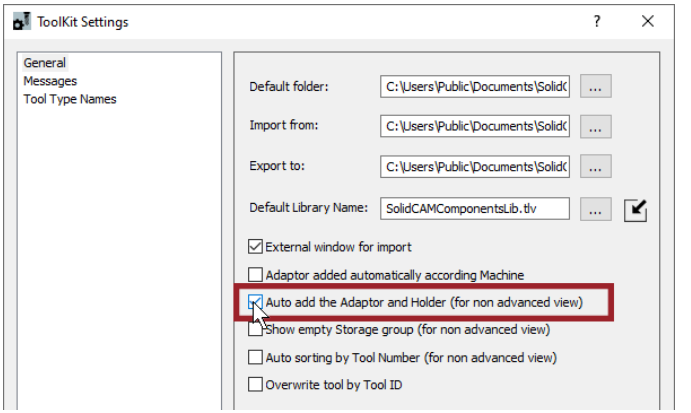


- Tool Table right-click commands



If preferred, you can have ToolKit automatically add the Adaptor and Holder upon adding your Cutter component to the Part Tool Table.

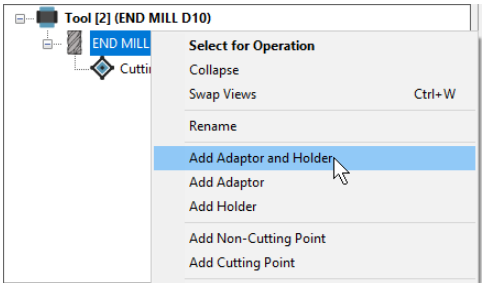
To do so, enable **Auto add the Adaptor and Holder** in the ToolKit Settings.



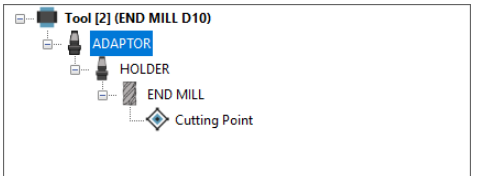
To access the ToolKit Settings, click **View > Settings** in the ToolKit main menu.

If not using the **Auto add the Adaptor and Holder** setting, the steps to add these components to the tool and define their properties are as follows:

1. In the Tool Item Manager, right-click the tool and choose the appropriate command to add the desired component(s).



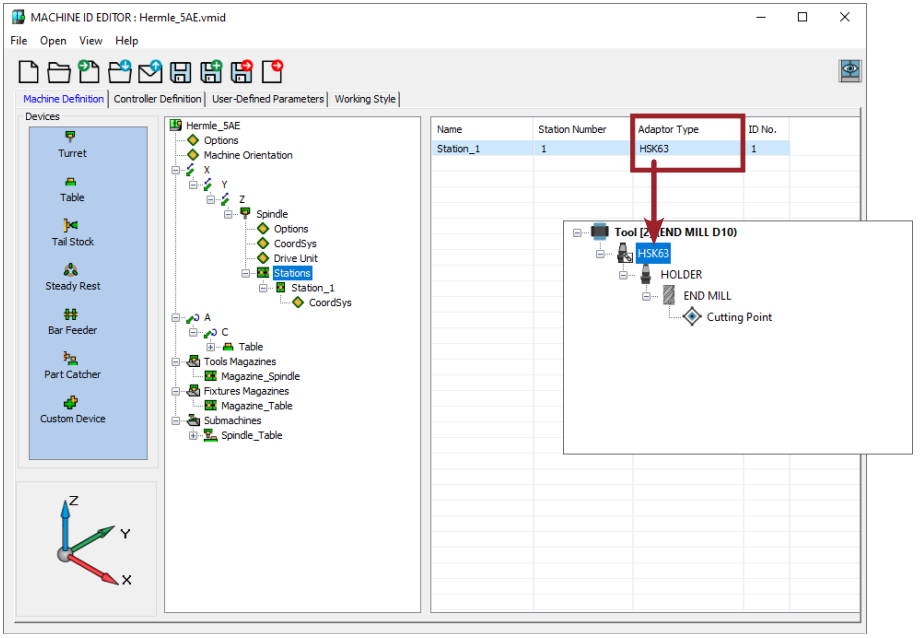
For example, if you select **Add Adaptor and Holder** from the appearing shortcut menu, the components are simultaneously added to the Tool Item tree as shown.



The Tool Item tree structure represents the actual arrangement of components when physically assembled and mounted on your CNC-Machine.

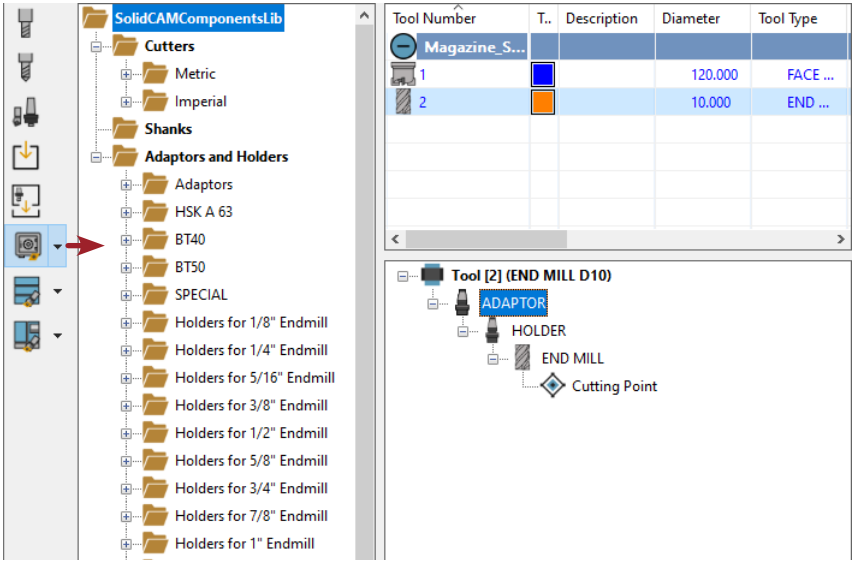
When the Tool Item/Tool Table commands are used to add a component, it is considered EMPTY and you are prompted to define its geometry.

If Adaptor Type is specified in the *.vmid file of your CNC-Machine, the Adaptor component is defined automatically.



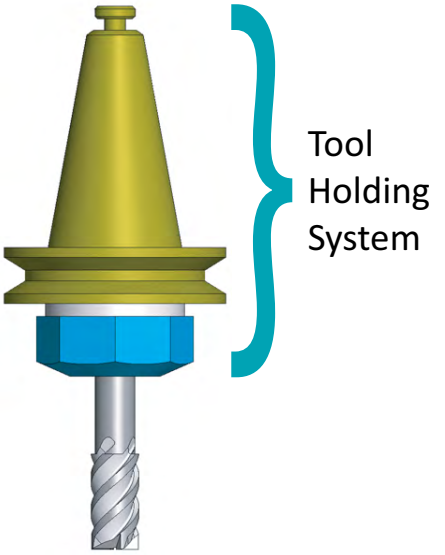
2. Define the Adaptor/Holder geometry.

Automatically loaded in the Selection Pane is **SolidCAMComponentsLib**, which facilitates defining the component geometry.



SolidCAMComponentsLib and the Tool Holding System

SolidCAM enables you to define, edit and use a variety of Adaptors and Holders to help you check and prevent all possible collisions between your Tool Holding System and the workpiece.



The SolidCAM Tool Holding System also provides a more realistic visualization of the machining simulation in the SolidCAM Simulator and SolidVerify Simulation modes.

SolidCAMComponentsLib is a predefined Tool Components Library that is included with the installation of SolidCAM. SolidCAMComponentsLib.tlv is installed in C:\Users\Public\Documents\SolidCAM\SolidCAM2022\Tables\Tool Libraries.

The included components assist in the Tool Holding System definition for your tool. As described below, two types of Holders can be used.

Standard/Shape Holder

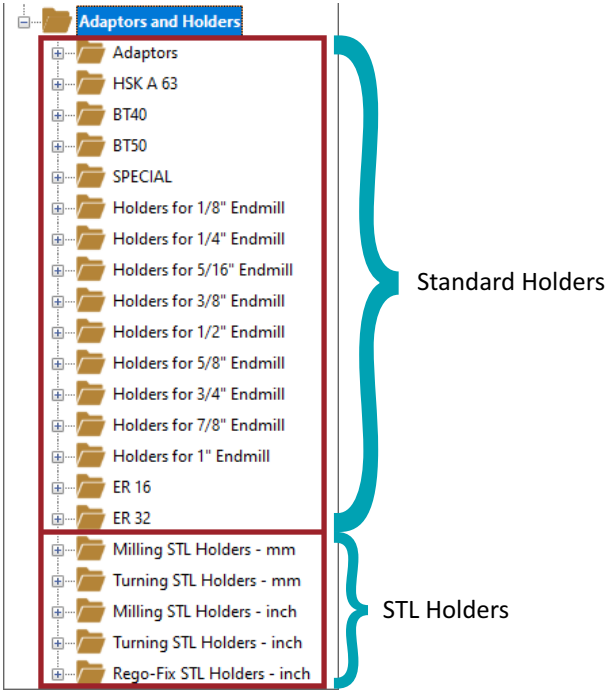
A Holder of this type is comprised of two components. The first component is the adaptor and the second component can consist of different types of extensions and reductions such as collet chucks, arbors, shanks and other components you may have.

There are a series of Standard Holders included in SolidCAMComponentsLib.

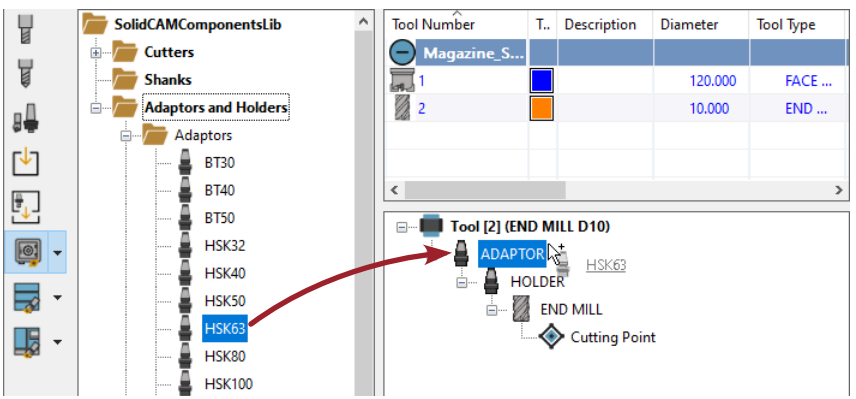
STL Holder

A Holder of this type is defined in the STL format. The STL file contains data on the shape and the visual properties of the actual Holder component.

Like the provided Standard Holders, SolidCAMComponentsLib also includes a series of STL Holders.



In the Selection Pane, expand the **Adaptors and Holders** folders as needed and double-click the desired component or simply drag & drop it into the Tool Item window.

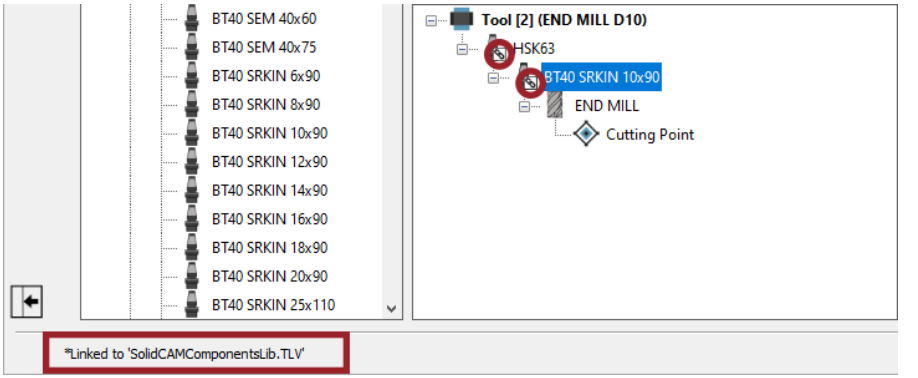


The EMPTY component will be replaced by the one of your choosing.

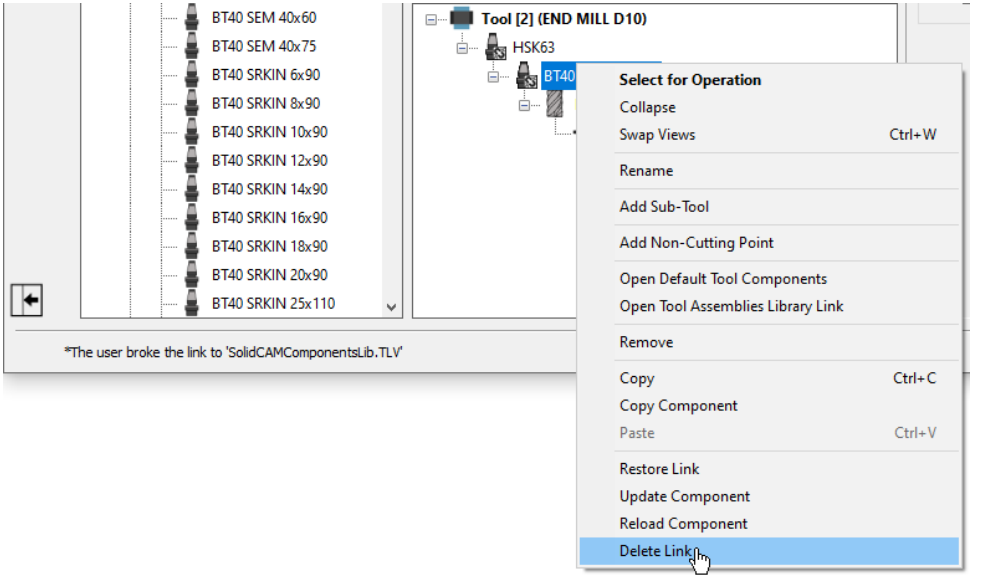
If the included components do not suit your needs exactly, new ones can be defined or the provided ones can be edited.

Editing linked components

Components added from SolidCAMComponentsLib are linked to the .tlv library and shown with a lock icon in the Tool Item Manager.



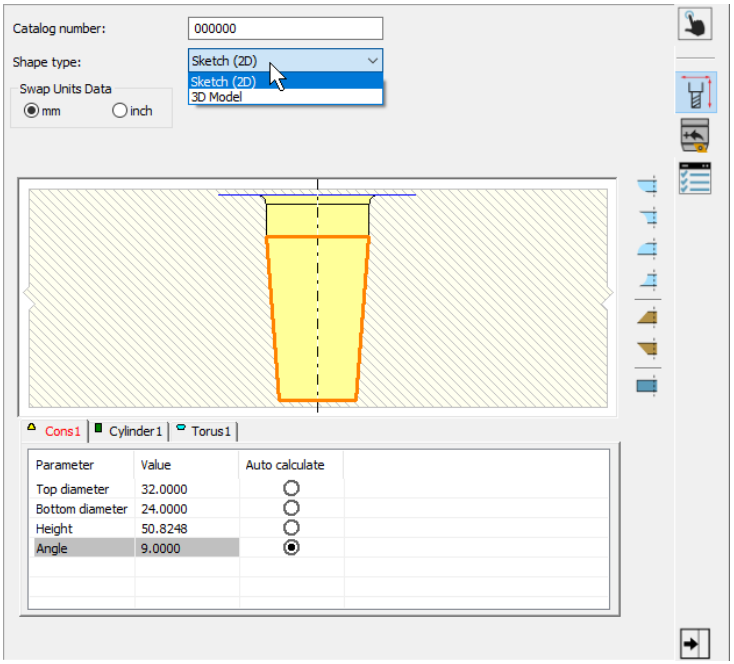
If you want to edit a linked component for use in your Tool Library, editing can be enabled with the **Break/Delete Link** commands in the right-click menu of the Tool Item Manager.



If you use the **Break Link** option, it is possible to restore the link and revert all changes back to the original component stored in SolidCAMComponentsLib.tlv.

On the other hand, it is not possible to restore the link if you use the **Delete Link** option.

Whether new or existing, the Adaptor/Holder geometry can be defined and edited in the Topology Data of the components.



The Adaptor and Holder components from SolidCAMComponentsLib.tlv are provided in Standard/Shape or STL formats, and you can edit them according to the Shape type as mentioned in the Introduction page 11.

Sketch (2D)

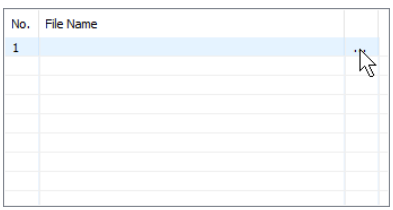
The **Sketch (2D)** option displays an interactive picture of the component, which is divided into segment shapes such as cylinders, cones and toruses.

Shapes can be added using the Toolbar or the Interactive Picture right-click commands. The actual dimensions can be edited by selecting the segments on the interactive picture and by using the corresponding Parameters pages.

3D Model

The **3D Model** option enables you to define the component by one or more STL/STEP files.

Using the ... button in the STL Files table, you can replace the current file with one of your choosing or you can define an additional one in the next available row.

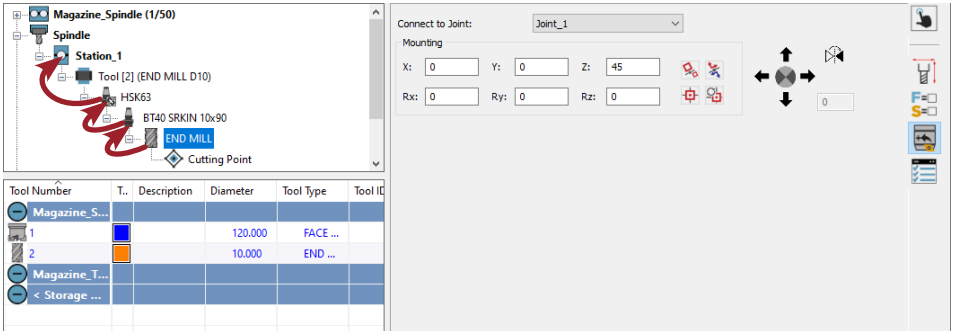


3. Use the Data Toolbar and corresponding Data Pane to define the remaining properties such as the Connection Data.

   As mentioned in the Introduction page 15, default Mounting and Joint CoordSys positions are provided in accordance with the components geometry.

   In this example, there are the following three connections:

   1. Cutter (END MILL) is mounted to the holder (BT40 SRKIN 10x90)
   2. Holder is mounted to the adaptor (HSK63)
   3. Adaptor (topmost component) is mounted to the Spindle

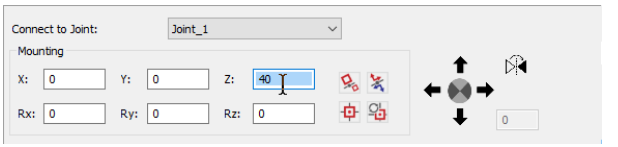


(Advanced View shown for illustrative purposes)

Except for the topmost component, which is mounted to the Station CoordSys position of the Spindle in this case, the Mounting position of each component is connected to the Joint CoordSys position of the higher component.

By clicking a component in the Tool Item tree, its Connection Data can be viewed/edited.

The Mounting and Joint CoordSys positions can be modified in several ways such as using linear and/or rotational movements. For example, the cutter can be moved along the Z-Axis by entering a value in the corresponding Z field or by scrolling your mouse wheel to change the value gradually.



Modifying the Z value of the cutter's mounting to the holder directly affects the Outside holder length (OHL) parameter defined in the Topology Data. As one is modified the other is updated automatically.

Such movements can be seen using the available ToolKit visualization tools.

2.1.5 Visualizing the Tool and selecting it for the operation

As mentioned in the Introduction page 24, ToolKit offers several visualization tools that are accessible at the upper right corner of the TOOLKIT dialog box.



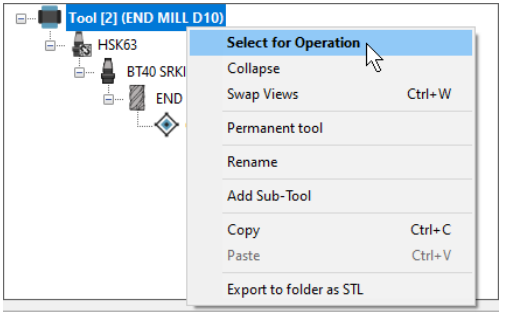
The ToolKit visualization tools can be used to facilitate, preview and check your Tool definition. Changes you make are updated instantaneously and can be viewed on the fly.

Once checked and ready for use, you can select the tool for the operation by using:

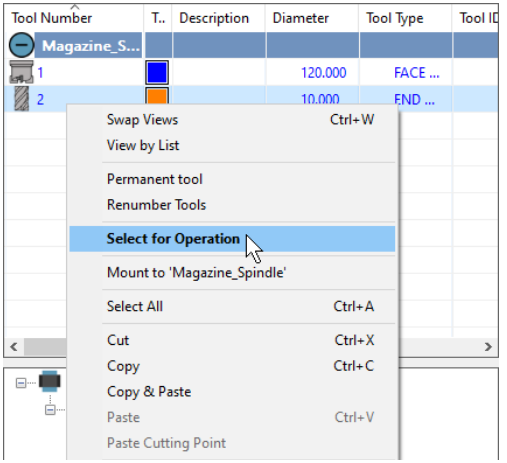
- the **Select** button



- the Tool Item Manager right-click command



- the Tool Table right-click command

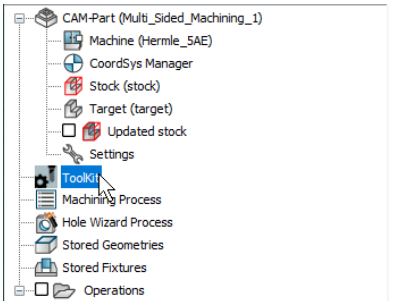


2.1.6 Importing Tool(s) from an existing CAM Tool Table

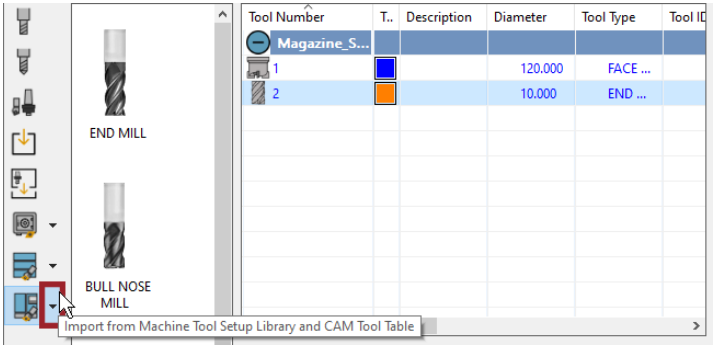
To simplify the Tool definition process, already defined tools from existing CAM projects can be imported to the Part Tool Library of your current CAM-Part.

The steps to import tools from an existing CAM Tool Table are as follows:

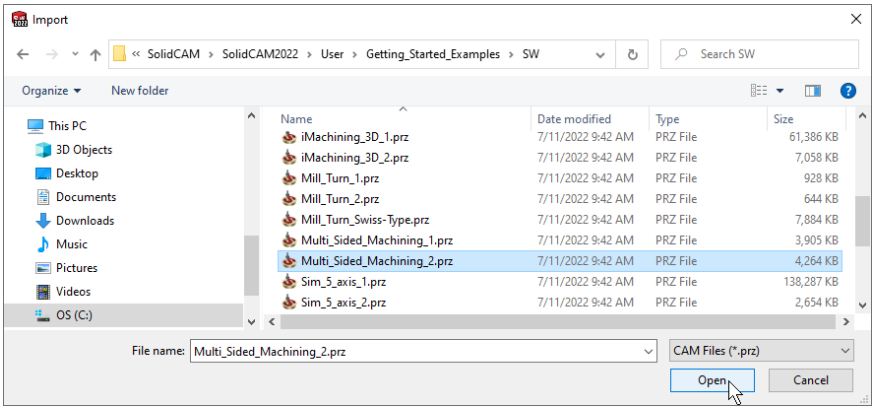
1. In the SolidCAM Manager, double-click the **ToolKit** header to **Edit Part Tool Library**.



2. On the Library Toolbar, click **Import from Machine Tool Setup Library and CAM Tool Table** arrow icon as shown.

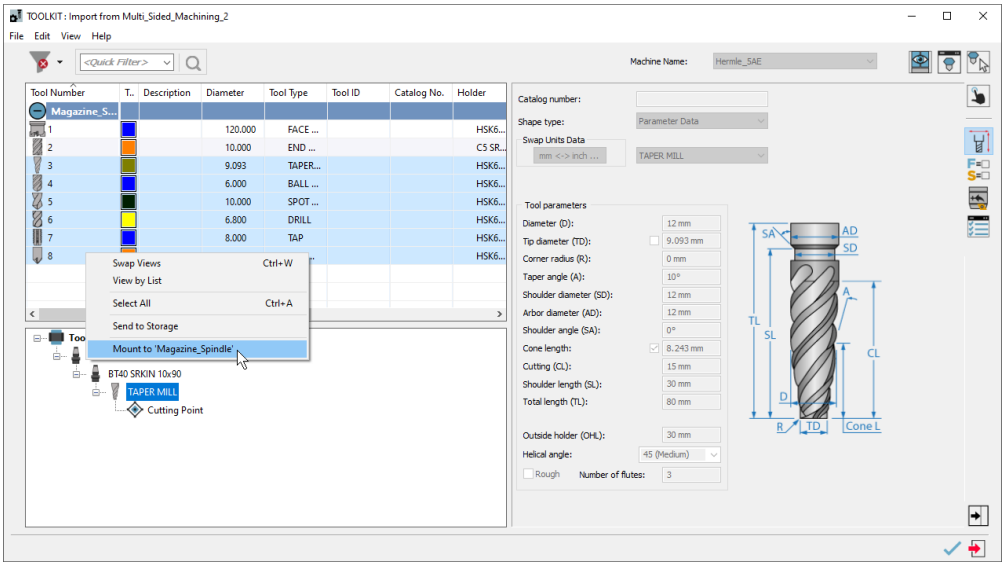


3. Using the Import dialog box, browse to and **Open** the CAM-Part file from which you want to import the tools.



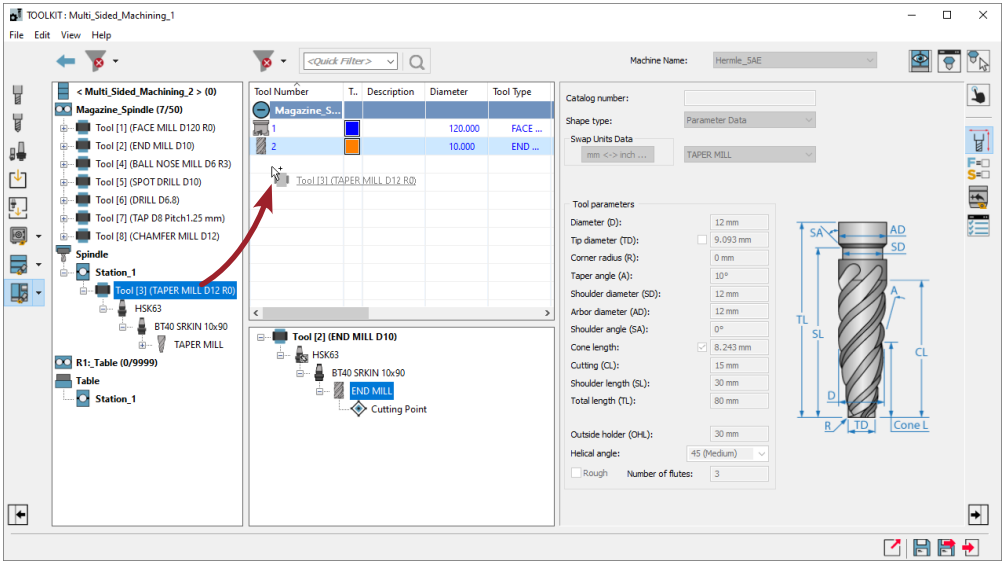
Tools can be imported from either type of CAM-Part, non-compressed (*prt) or compressed (*prz).

The CAM Tool Table (ToolTable.tlp) from the chosen CAM-Part is opened in an external window, where you can view and select the tools for use in your current CAM-Part.



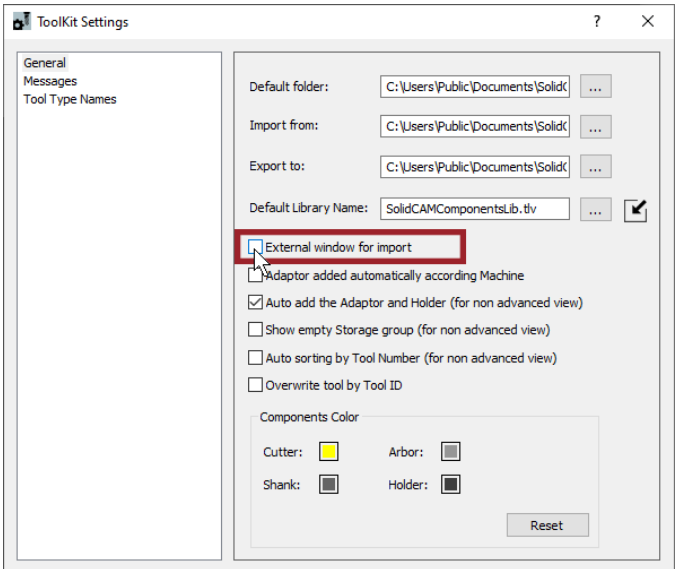
The tools you select are imported to the Part Tool Table in sequence but are assigned the next available Tool Numbers/Positions in Magazine.

If you exit the external window, the tools of the chosen CAM Tool Table also appear in the Selection Pane from which you can import the tools to the Part Tool Table in the order of your choosing.



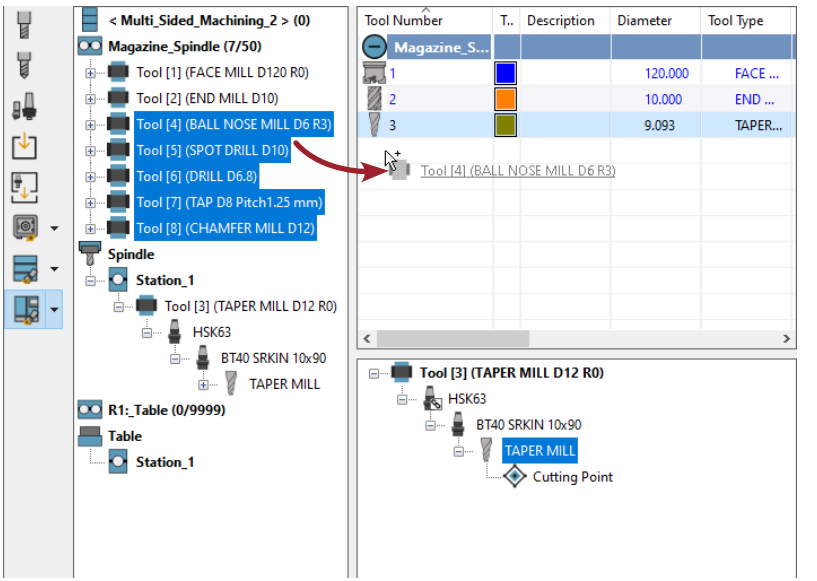
If the Selection Pane is preferred for importing tools, you can disable the external window that is displayed on import.

To do so, disable **External window for import** in the ToolKit Settings.



To access the ToolKit Settings, click **View > Settings** in the ToolKit main menu.

4. Per your preference (Selection Pane or external window), import the desired tool(s) to the Part Tool Library of your current CAM-Part.



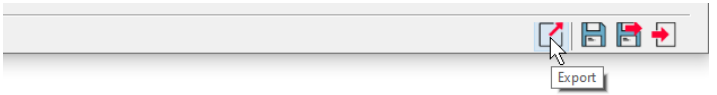
Once imported, the tools can be edited if necessary and used in your operations.

2.1.7 Exporting Tool(s) for later use

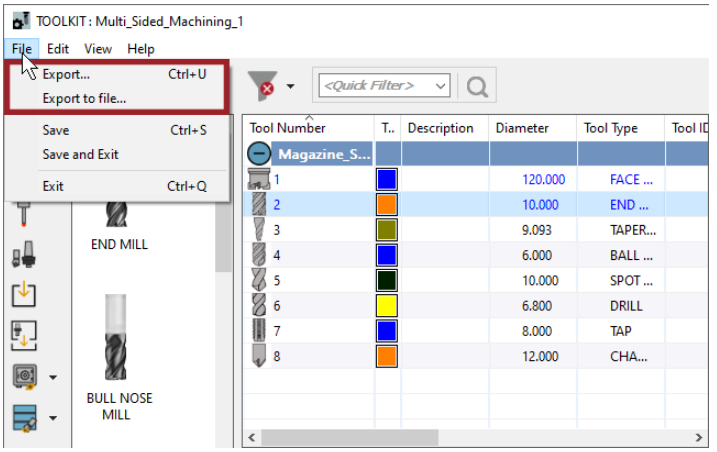
ToolKit offers several export options for when you want to later use the defined tools from your current CAM-Part.

From the Part Tool Library, exporting is permitted to any ToolKit Tool Library (i.e., Tool Components *.tlv, Tool Assemblies *.tls or Machine Tool Setup *.tlm).

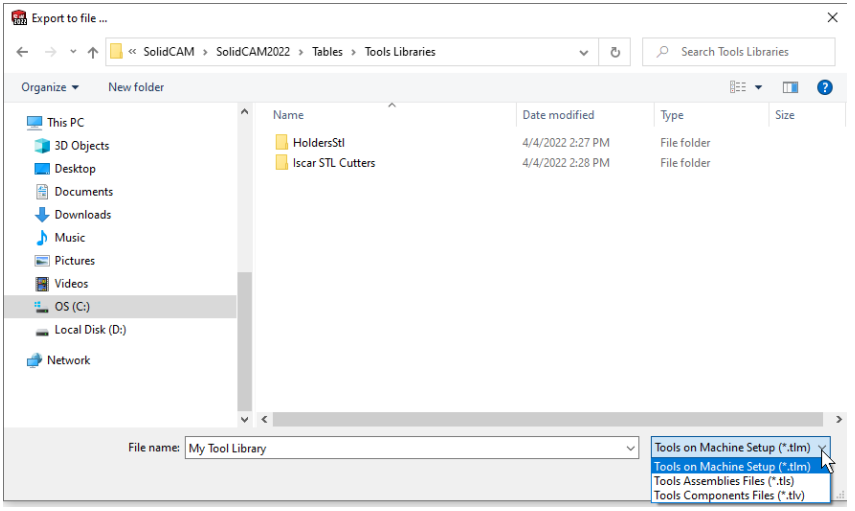
To do so, you can use the **Export** button at the bottom right corner of the TOOLKIT dialog box



OR you can use the **Export/Export to file** commands in the ToolKit main menu.

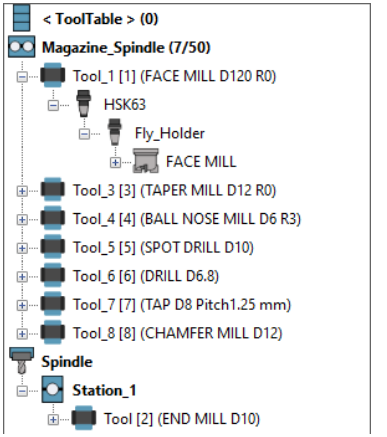


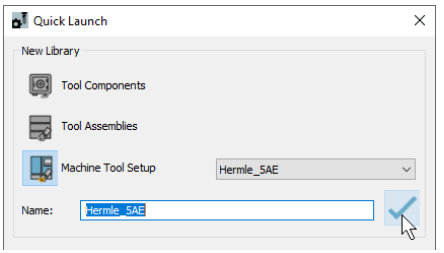
Whichever method you choose displays the Export to file dialog box that enables you to name the Tool Library, select the file type and the directory where you want to save the file.



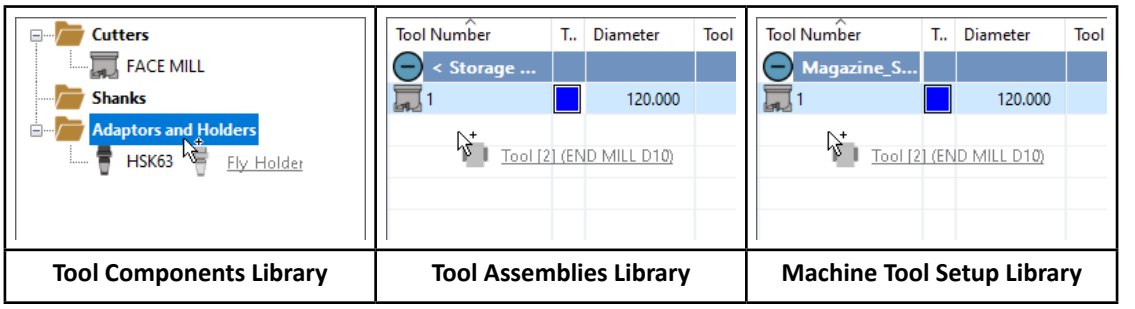
According to the chosen Tool Library file type, the appropriate TOOLKIT dialog box will be opened and all tools existing in the Part Tool Library are displayed in the Selection Pane.

For *.tlm files, you are prompted to choose the CNC-Machine for the basis of the Machine Tool Setup Library.



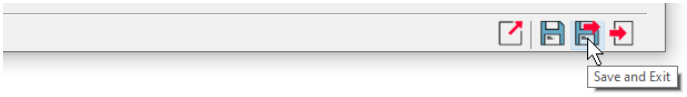


Drag & drop the components you want to export into the folders (for Tool Components Library) or drag & drop the tools you want to export into the Tool Table window (for Tool Assemblies and Machine Tool Setup Libraries).

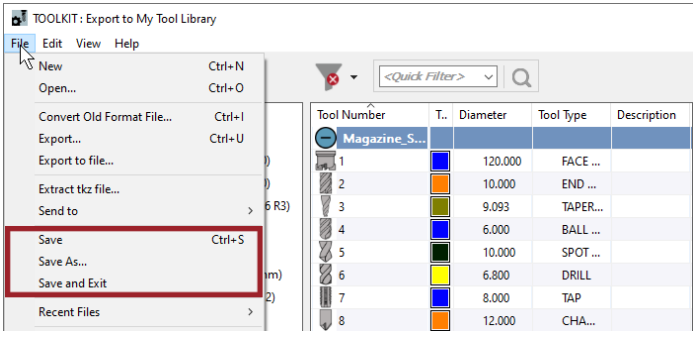


Tool Components Library	Tool Assemblies Library	Machine Tool Setup Library

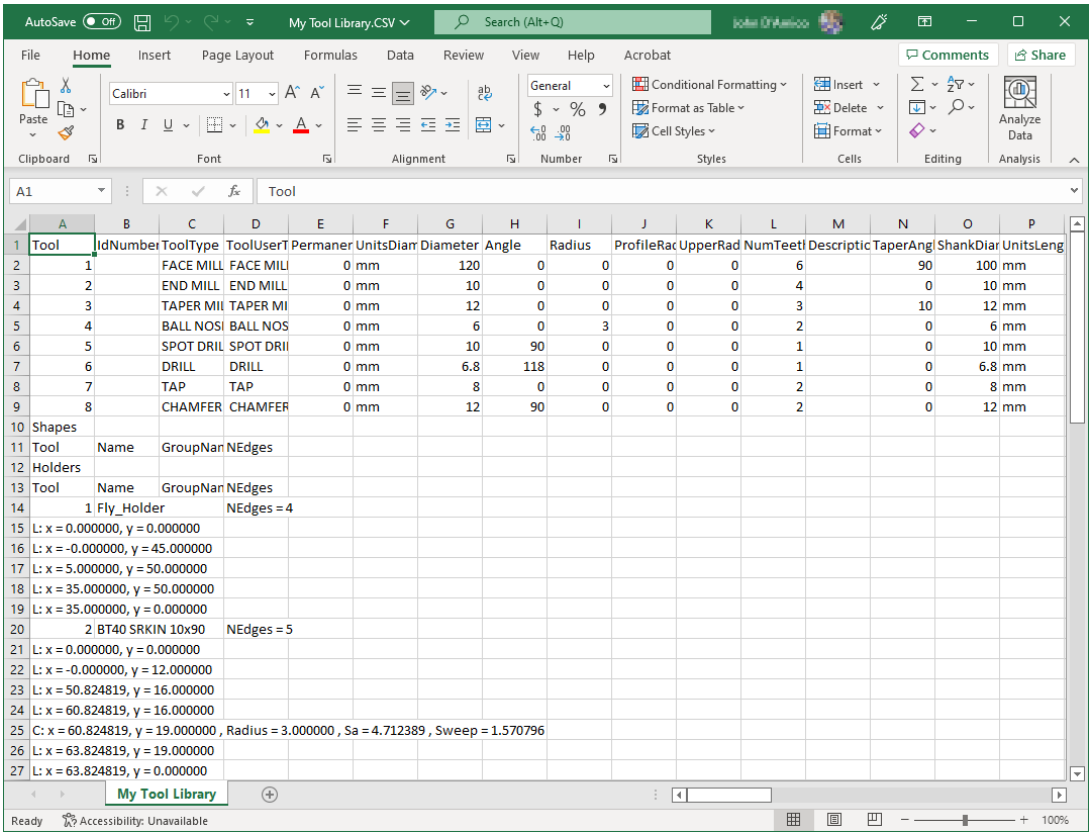
Once your new Tool Library is ready for exporting, save it using the **Save/Save and Exit** buttons at the bottom right corner of the TOOLKIT dialog box



OR using the **Save/Save As/Save and Exit** commands in the ToolKit main menu.



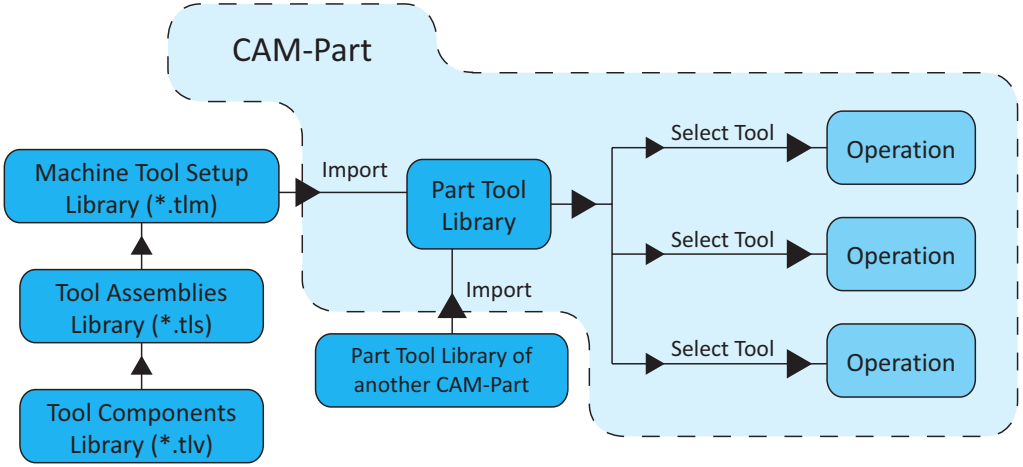
There is also the option to export the data of all your tools to an Excel-compatible file using the **Export to file** command.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1	Tool	IdNumber	ToolType	ToolUserT	Permaner	UnitsDiam	Diameter	Angle	Radius	ProfileRac	UpperRad	NumTeetl	Descriptic	TaperAng	ShankDiar	UnitsLeng
2	1		FACE MILL	FACE MILI	0	mm	120	0	0	0	0	6		90	100	mm
3	2		END MILL	END MILL	0	mm	10	0	0	0	0	4		0	10	mm
4	3		TAPER MII	TAPER MI	0	mm	12	0	0	0	0	3		10	12	mm
5	4		BALL NOSI	BALL NOS	0	mm	6	0	3	0	0	2		0	6	mm
6	5		SPOT DRIL	SPOT DRI	0	mm	10	90	0	0	0	1		0	10	mm
7	6		DRILL	DRILL	0	mm	6.8	118	0	0	0	1		0	6.8	mm
8	7		TAP	TAP	0	mm	8	0	0	0	0	2		0	8	mm
9	8		CHAMFER	CHAMFER	0	mm	12	90	0	0	0	2		0	12	mm
10	Shapes															
11	Tool	Name	GroupNan	NEdges												
12	Holders															
13	Tool	Name	GroupNan	NEdges												
14	1	Fly_Holder		NEdges = 4												
15	L: x = 0.000000, y = 0.000000															
16	L: x = -0.000000, y = 45.000000															
17	L: x = 5.000000, y = 50.000000															
18	L: x = 35.000000, y = 50.000000															
19	L: x = 35.000000, y = 0.000000															
20	2	BT40 SRKIN 10x90		NEdges = 5												
21	L: x = 0.000000, y = 0.000000															
22	L: x = -0.000000, y = 12.000000															
23	L: x = 50.824819, y = 16.000000															
24	L: x = 60.824819, y = 16.000000															
25	C: x = 60.824819, y = 19.000000 , Radius = 3.000000 , Sa = 4.712389 , Sweep = 1.570796															
26	L: x = 63.824819, y = 19.000000															
27	L: x = 63.824819, y = 0.000000															

Advanced Workflow 3

For Workflow Type 2 users, the Part Tool Library is typically prepared before the Operation stage with the use of predefined Tool Libraries. Although tools can be imported from any ToolKit Tool Library, users who prefer this working style may use all levels of tool management in their intended order – Tool Components Library > Tool Assemblies Library > Machine Tool Setup Library > Part Tool Library.

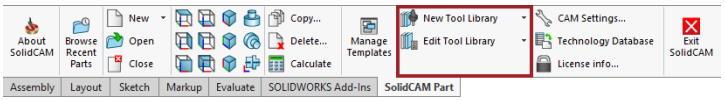


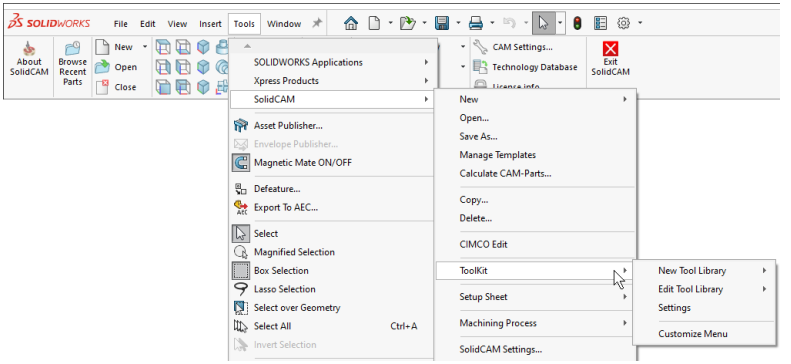
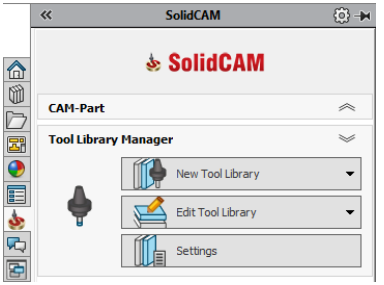
Described below are topics relevant to Workflow Type 2 users and insight to what can be expected when working with SolidCAM ToolKit at each tool management level.

3.1 Working with ToolKit at Library stages

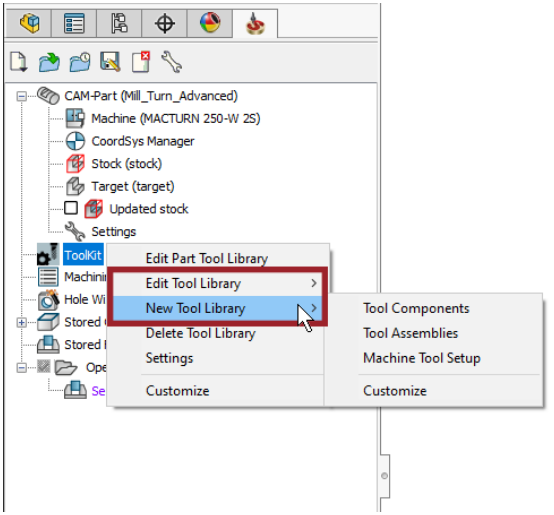
3.1.1 Defining and editing ToolKit Tool Libraries

SolidCAM enables you to create new or edit existing ToolKit Tool Libraries in various ways.

You can use the available SolidCAM controls in the SOLIDWORKS environment, including:	
SolidCAM tabs in SOLIDWORKS CommandManager	**SolidCAM Part** tab > **New/Edit Tool Library** options 
SolidCAM toolbars in SOLIDWORKS window	**SolidCAM New Tool Library** toolbar  **SolidCAM Edit Tool Library** toolbar 

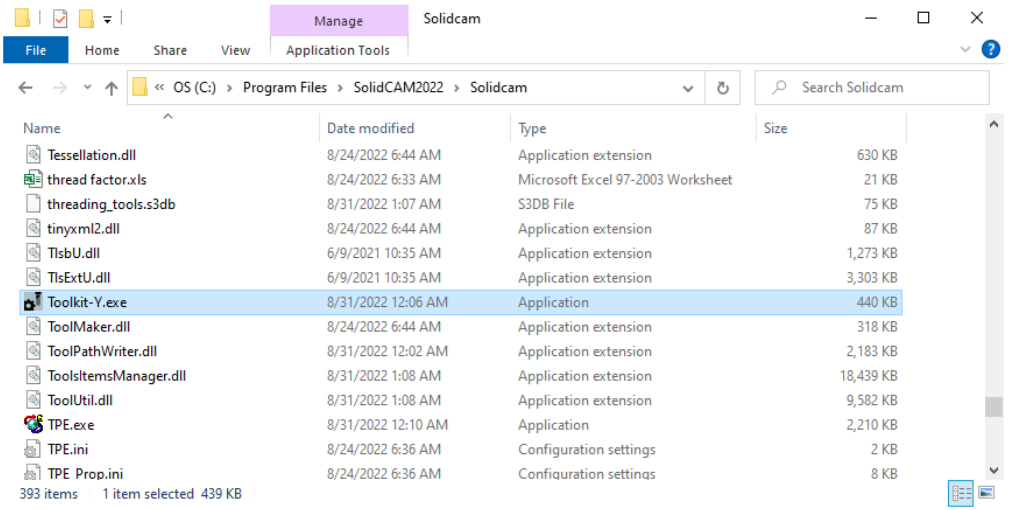
SOLIDWORKS Menu Bar	SOLIDWORKS Menu Bar > **Tools** > **SolidCAM** > **ToolKit** 
SOLIDWORKS Task Pane	**SolidCAM** tab > **Tool Library Manager** 

If a CAM project is already started, you can also create new or edit existing ToolKit Tool Libraries from the SolidCAM Manager > **ToolKit** header > **New/Edit Tool Library** right-click commands.

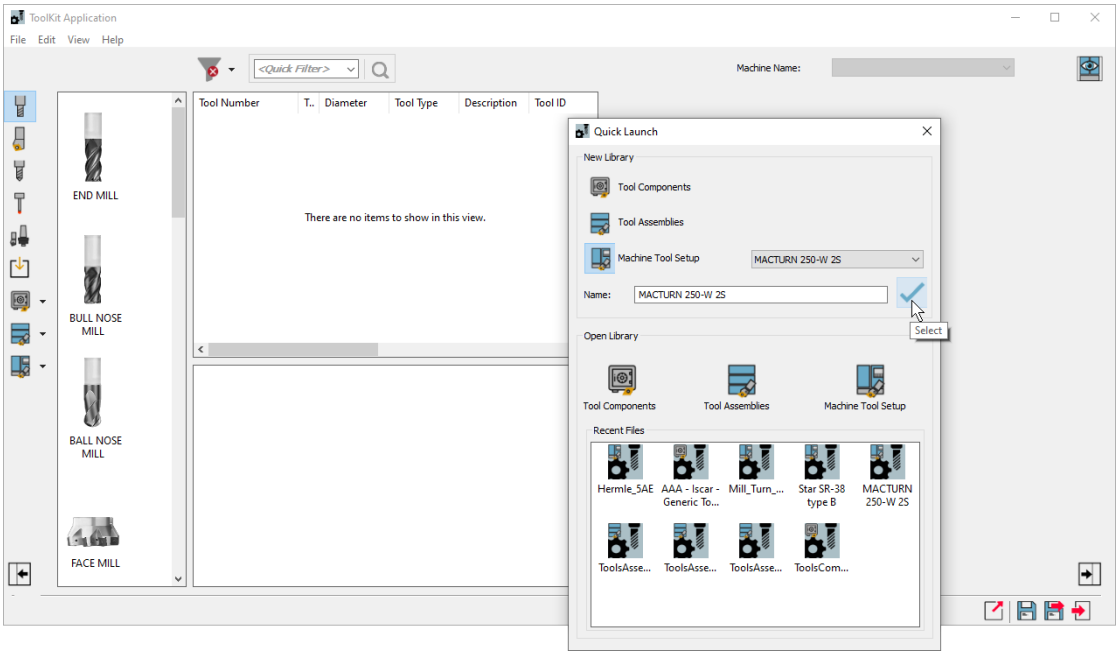


In addition, ToolKit is provided as a standalone application, enabling you to create new or edit existing Tool Libraries when SOLIDWORKS and SolidCAM are not actively running.

As mentioned in the Introduction page 8, the full path to SolidCAM's standalone ToolKit application is C:\Program Files\SolidCAM2022\Solidcam\Toolkit-Y.exe

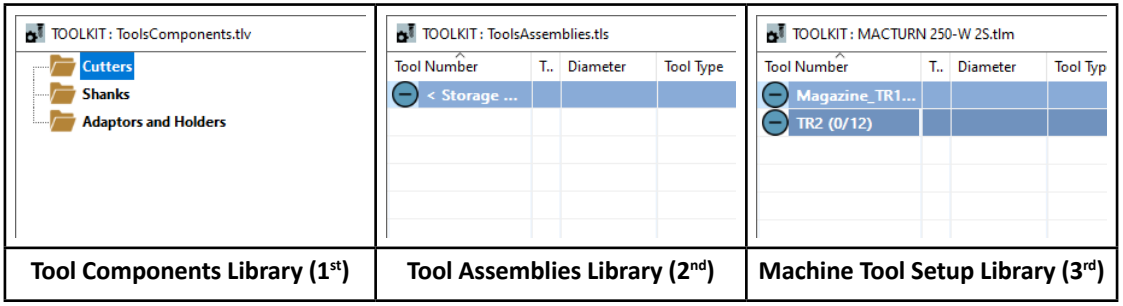


When running ToolKit from Toolkit-Y.exe, the type of Tool Library you want to create is chosen in the Quick Launch dialog box.



Whether you are running ToolKit independently or from within SOLIDWORKS/SolidCAM, a Machine Tool Setup Library requires you to choose the CNC-Machine for the basis of your Tool Library creation.

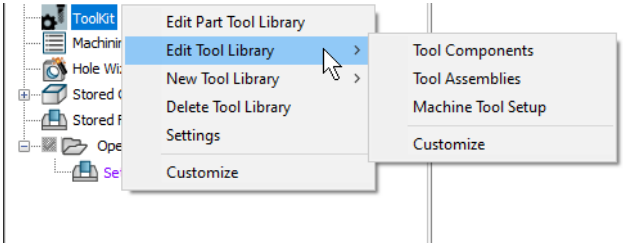
Your **New Tool Library** selection will display the appropriate TOOLKIT dialog box that enables you to create the Tool Library of your choosing.



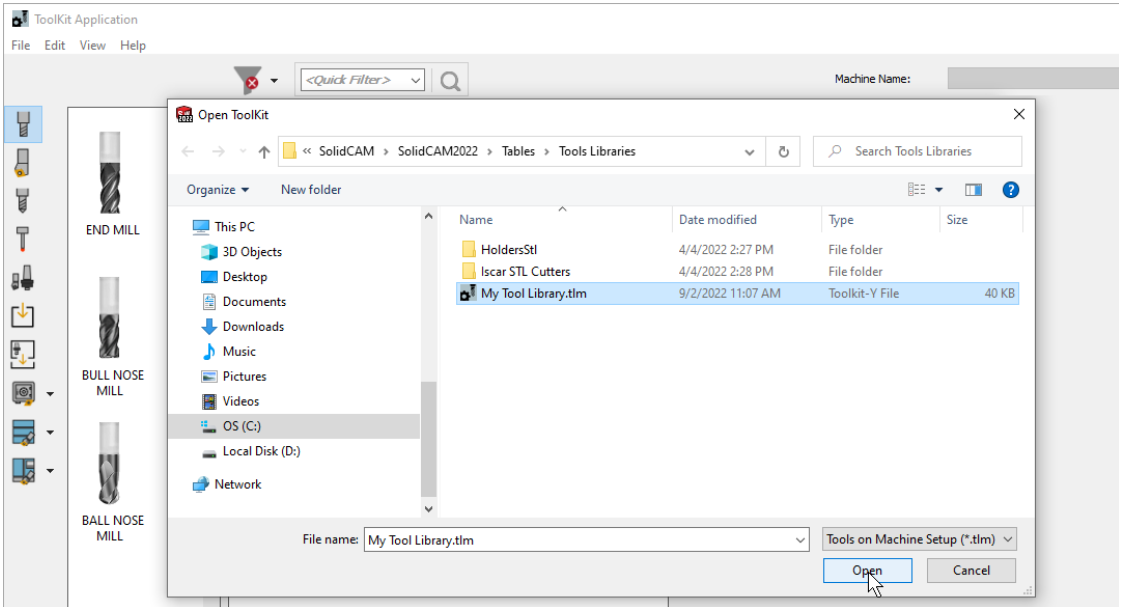
Tool Components Library (1st)	Tool Assemblies Library (2nd)	Machine Tool Setup Library (3rd)

For the purpose of this document, the ToolKit Tool Libraries will be defined in the order as shown.

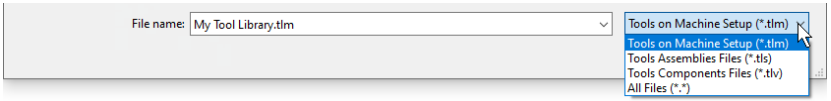
When you want to edit an existing Tool Library from within SOLIDWORKS/SolidCAM, choose the type using one of the described methods.



Your **Edit Tool Library** selection will display the Open ToolKit dialog box that enables you to browse and select for editing the Tool Library of your choosing.

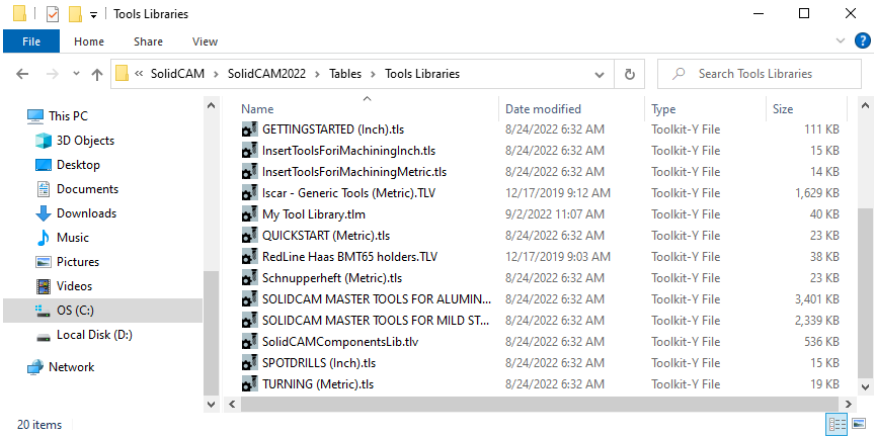


The Tool Library file type is selected automatically, but it can be easily changed on the fly if you want to choose a different Tool Library for editing.

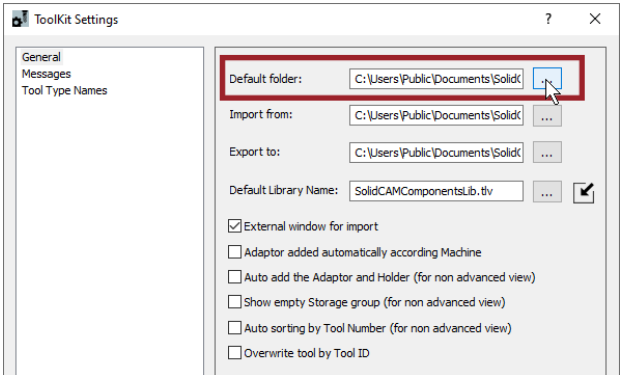


When you want to use SolidCAM's standalone ToolKit application to edit an existing Tool Library, it can be opened from Windows/File Explorer.

By default, all ToolKit Tool Libraries are saved in the folder C:\Users\Public\Documents\SolidCAM\SolidCAM2022\Tables\Tools Libraries



If you want to specify a different default directory, you can do so in the ToolKit Settings.



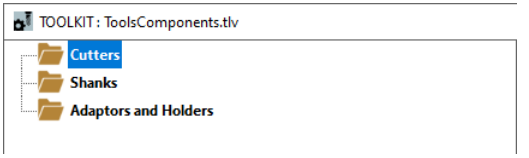
The **Default folder** specifies the directory where newly created ToolKit Tool Libraries are saved and, when editing a Tool Library from within SOLIDWORKS/SolidCAM, where browsing starts in the Open ToolKit dialog box.

By default, ToolKit specifies the same directory for importing and exporting Tool Libraries.

In the following examples, new ToolKit Tool Libraries are created for a Mill-Turn CAM-Part.

3.1.2 Defining a new Tool Components Library (*.tlv)

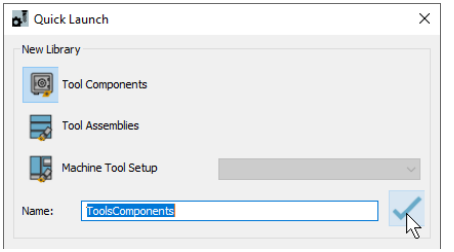
The first level of tool management offered by ToolKit is the Tool Components Library that enables you to manage various types of individual tool components such as your cutters, shanks, adaptors and holders.



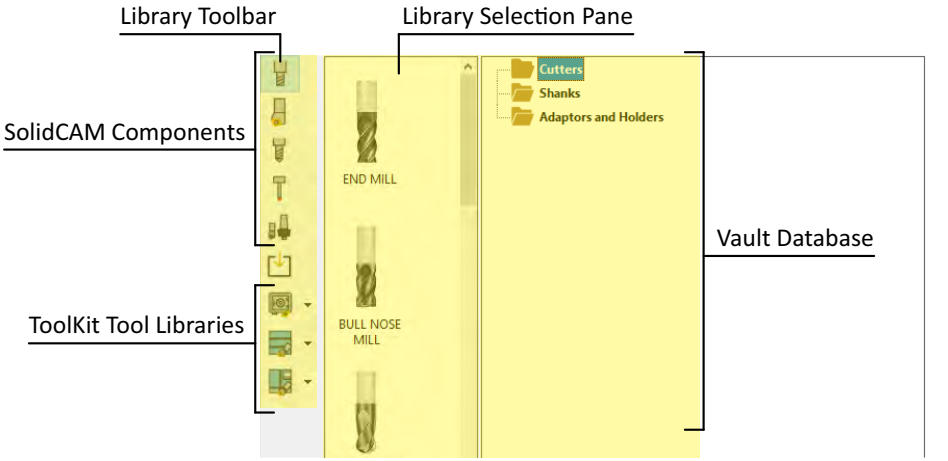
For the purpose of this document, a new Tool Components Library is defined by adding components from the available SolidCAM Components.

The steps to define a new Tool Components Library are as follows:

1. Start a new Tool Components Library using one of the previously described methods (from within SOLIDWORKS/SolidCAM or with ToolKit standalone).

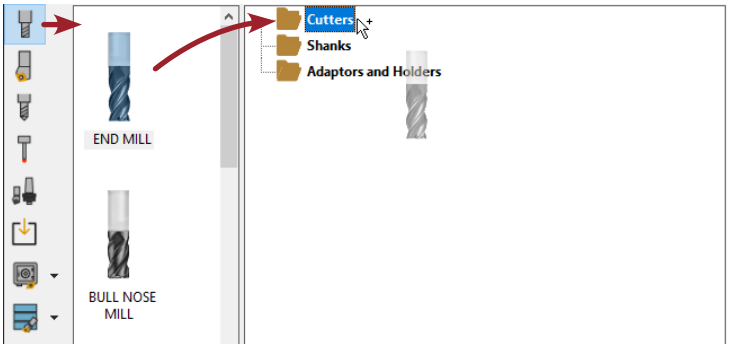


Displayed is the TOOLKIT dialog box at the Tool Components Library level.



While most interface features are shared between all ToolKit Tool Libraries, the main difference in the Tool Components Library is the Vault Database that enables you to manage your components using an organized folder structure.

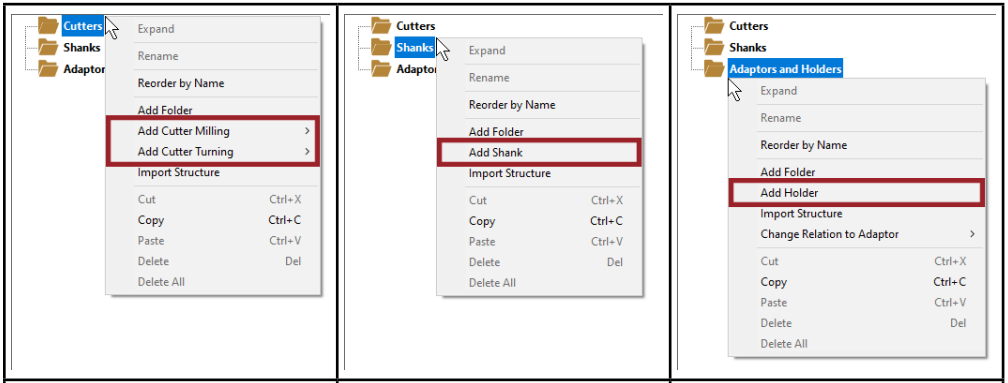
2. Using the Library Toolbar and corresponding Selection Pane, add the desired component to the appropriate folder in the Vault Database.



For Cutters, you can either double-click the component or simply drag & drop it into the Cutters folder in the Database window. Shanks, Adaptors and Holders must be dragged & dropped into the appropriate folders.

When double-clicking a Cutter component, it is automatically added to the predefined folder, not to any nested folders (if defined).

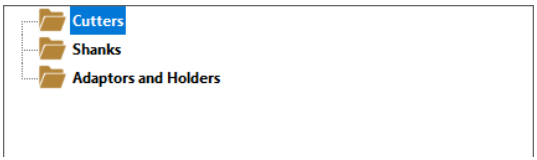
For Cutters, Shanks, Adaptors and Holders, you can also use the **Add...** commands in the right-click menus of the predefined folders.



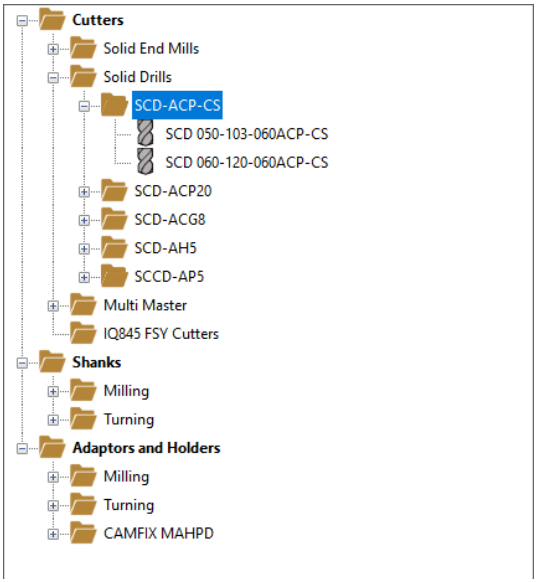
Cutters folder	Shanks folder	Adaptors and Holders folder

Tool Components Library folder structure

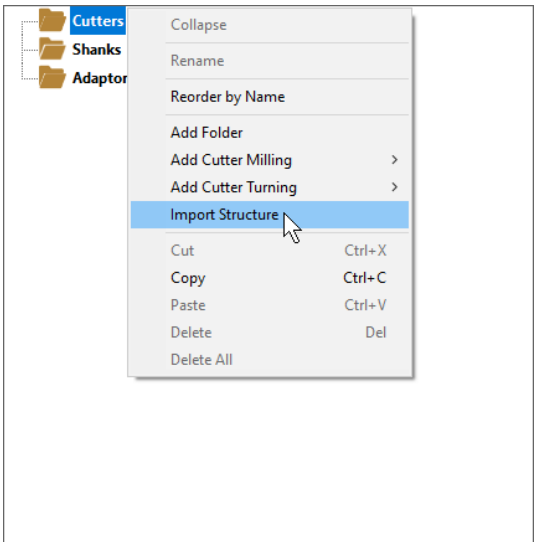
Currently, ToolKit provides you with three predefined folders as shown below.



The provided folders cannot be renamed, deleted, etc. However, you can manage any number of nested folders in the instance you want to organize your components into specific subcategories such as by type, manufacturer, application, and so on.

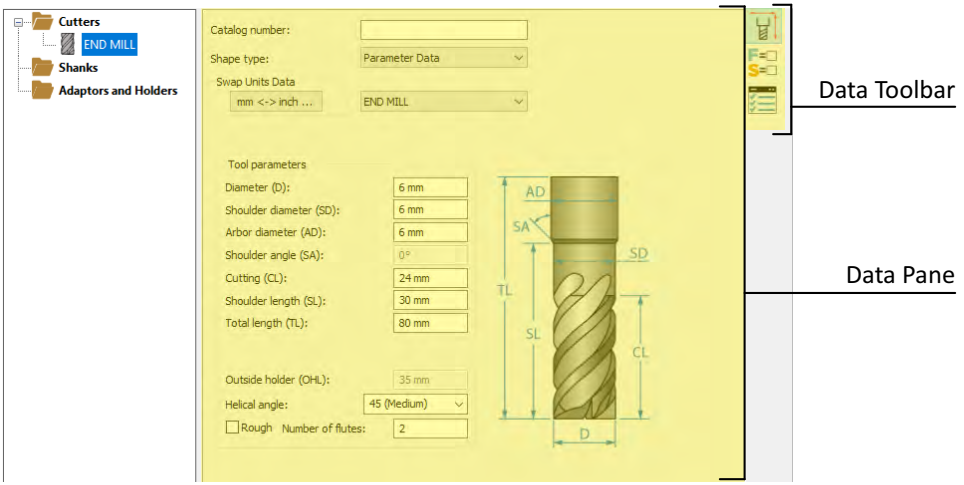


If you prefer creating a folder structure with Windows/File Explorer or if one is already defined on your computer, it can be easily imported using the **Import Structure** command in the right-click menu of the predefined folders.



Imported folder structures can contain any number of nested folders.

Upon adding your component to the Vault Database, its properties can be defined using the Data Toolbar and corresponding Data Pane.



3. Define the component geometry.



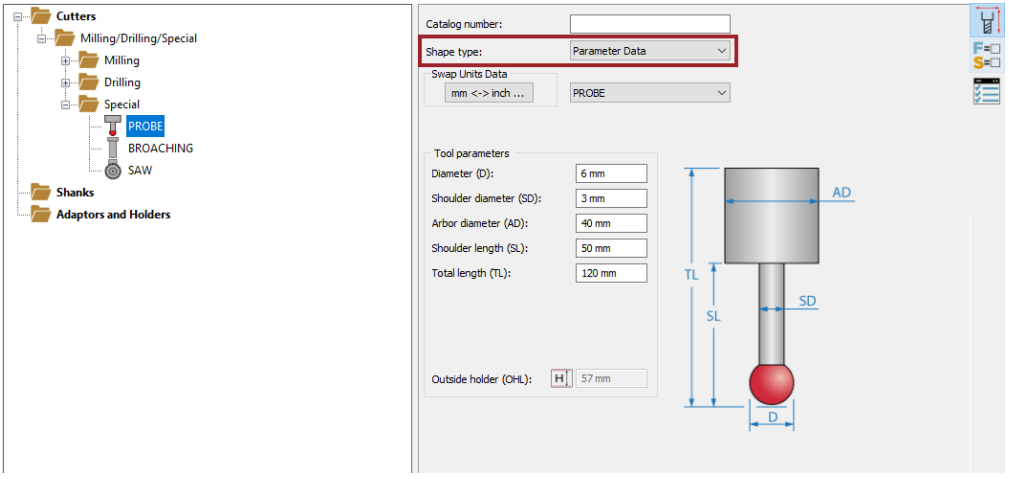
With each newly added component, the **Topology** selection on the Data Toolbar prompts you to specify the geometry in the Data Pane.

Tool Components geometry

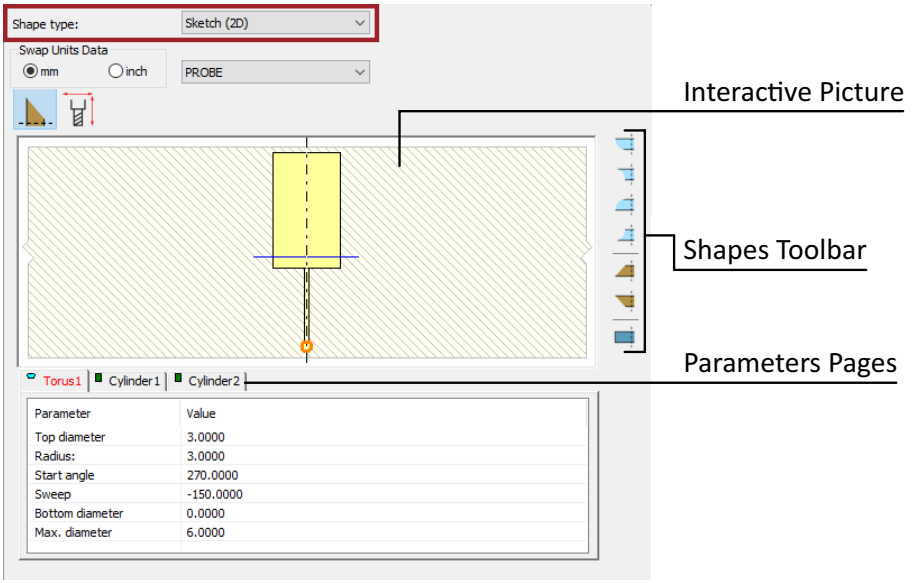
As mentioned in the Introduction page 11, different components (e.g., Cutter vs. Holder) offer different Shape type options, which determine how you want to specify the geometry.

Cutter components

For Milling, Drilling and Special Tools, the component can be defined by its dimensions using the **Parameter Data** pictured in the image.

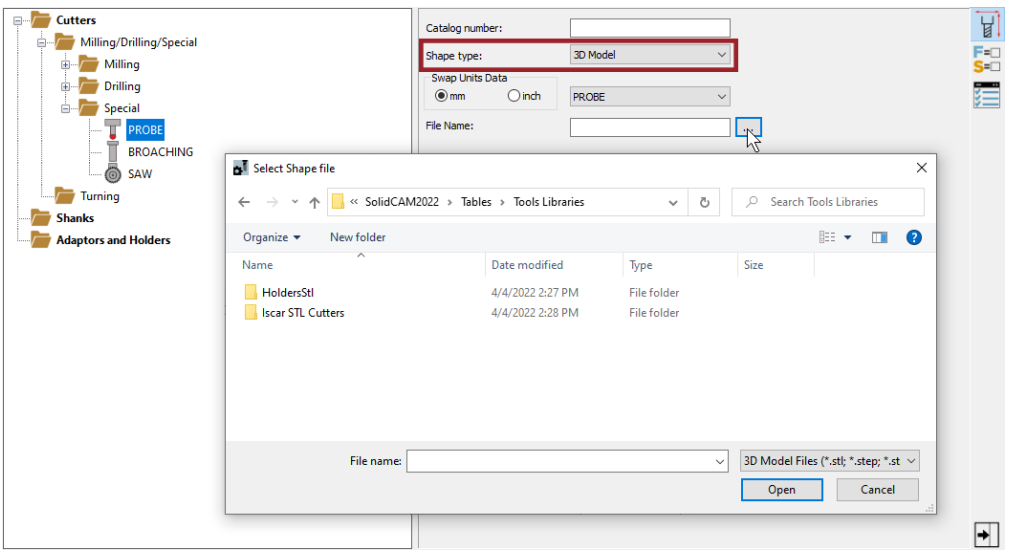


Using the **Sketch (2D)** option, the Cutter component can be defined by segment shapes such as cylinders, cones and toruses. Shapes can be added using the Toolbar or the Interactive Picture right-click commands.



The actual dimensions can be edited by selecting the segments on the interactive picture and by using the corresponding Parameters pages.

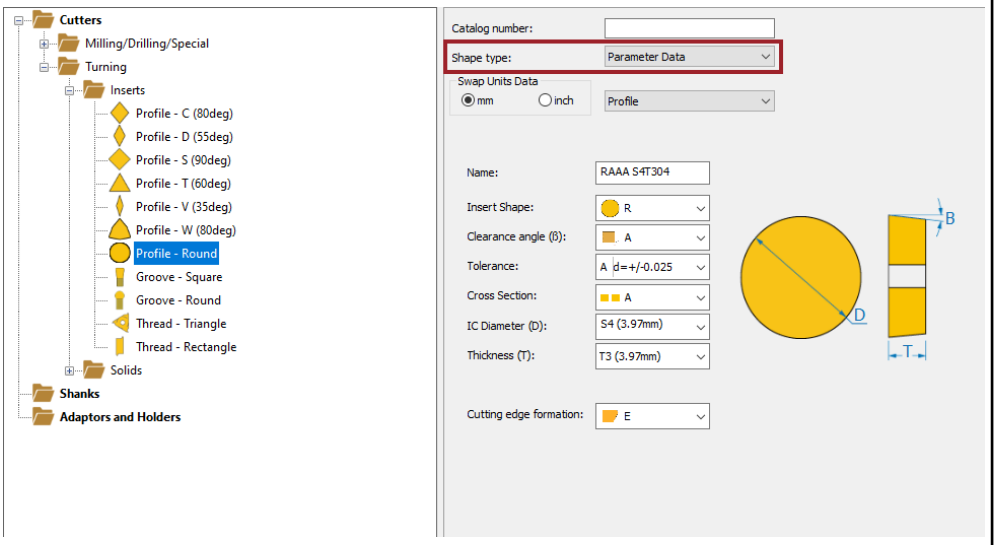
Using the **3D Model** option, the Cutter component can be defined by a STL or STEP file.



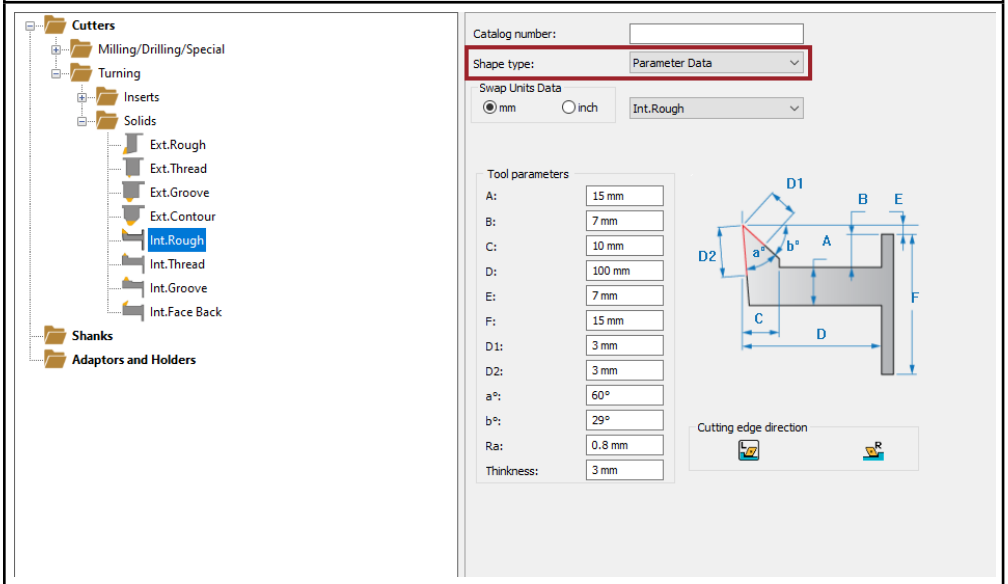
Clicking the ... button displays the Select Shape file dialog box, where you can browse for, select and open the STL/STEP file of your choosing.

For Turning Tools, whether an insert or a solid, the component can be defined using the **Parameter Data** or the **3D Model** option.

Parameter Data definitions for insert tools will vary depending on the Tool Type (Profile, Groove or Thread) and Insert Shape (C, D, S, etc.).



Parameter Data definitions are similar for solid tools of all types.

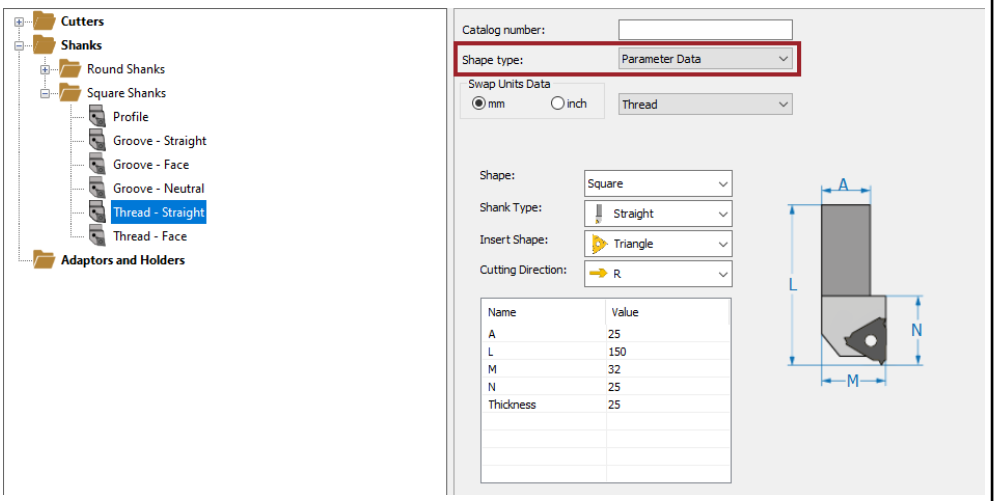


3D Model definitions are similar to Milling/Drilling/Special Tools in that Turning inserts and solids can be defined by a STL or STEP file using the Select Shape file dialog box.

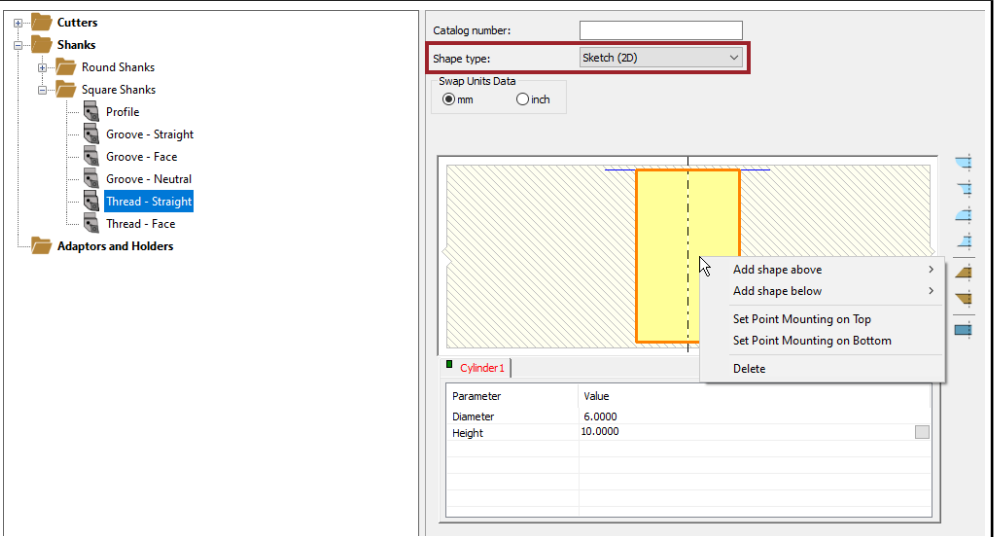
Shank components

Shanks can be defined by one of three Shape type options: **Parameter Data**, **Sketch (2D)** or **3D Model**.

Parameter Data specifies the Shank dimensions and, similar to Turning inserts, the parameters by which it is defined will vary depending on the Tool Type, Shank Type and Shape, and Insert Shape (for Profile and Thread types).

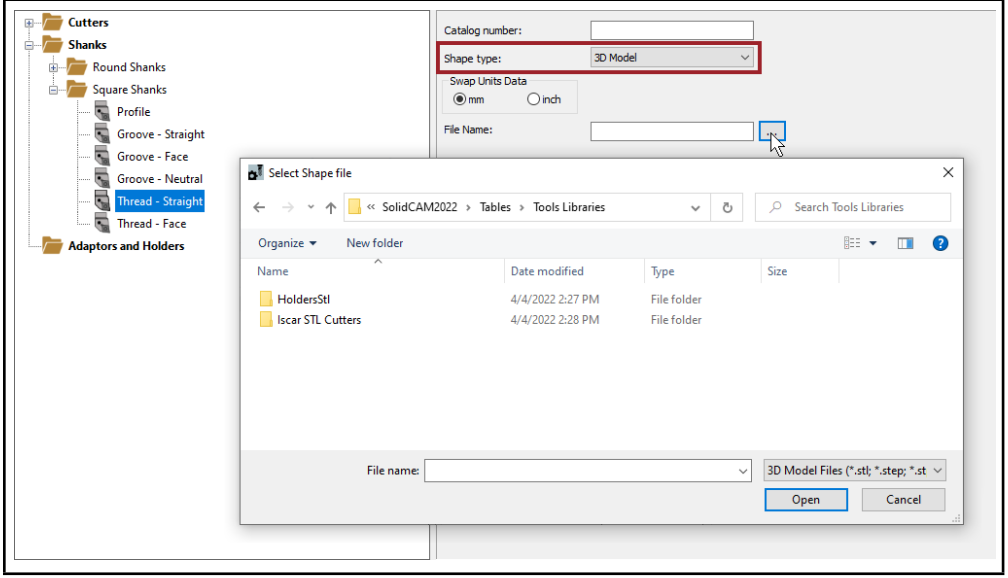


Sketch (2D) specifies the Shank dimensions using segment shapes (similar to Milling/ Drilling/Special Tools).



You can add shapes using the Interactive Picture right-click commands or the Shapes Toolbar. The actual dimensions can be edited by selecting the segments on the interactive picture and by using the corresponding Parameters pages.

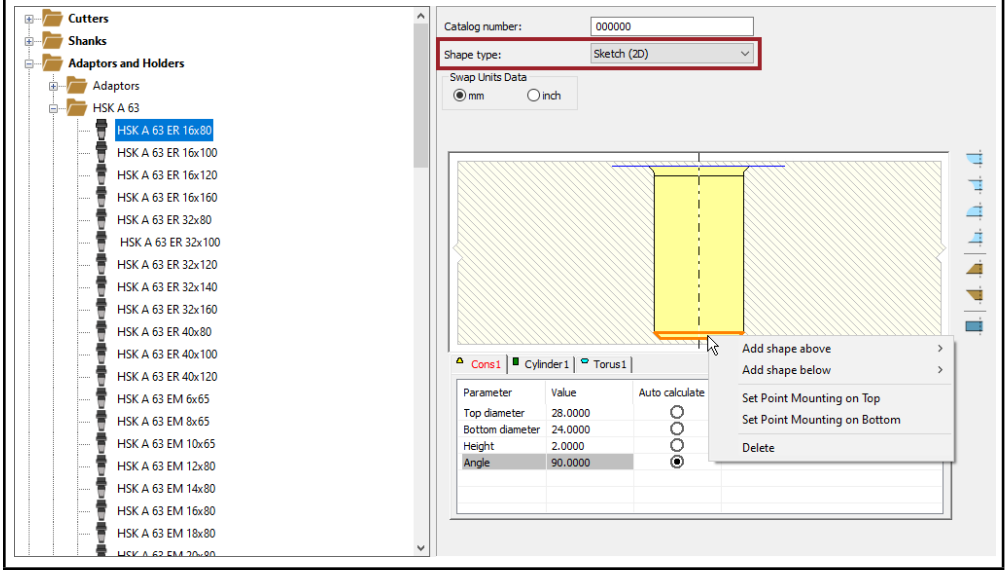
3D Model defines the Shank component according to an STL or STEP file using the Select Shape file dialog box.



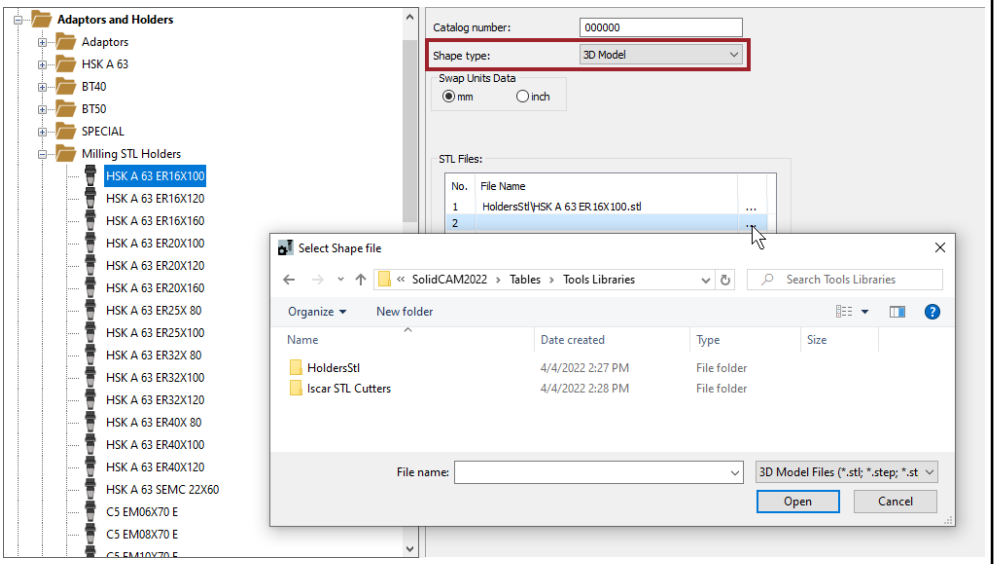
Adaptor and Holder components

Adaptors and Holders can be defined by one of two Shape type options: **Sketch (2D)** or **3D Model**.

Sketch (2D) specifies the Adaptor/Holder dimensions using segment shapes (similar to Milling/Drilling/Special Tools and Shank components).



3D Model defines the Adaptor/Holder component by one or more STL/STEP files.



Each row in the STL Files table enables you to define an additional file.

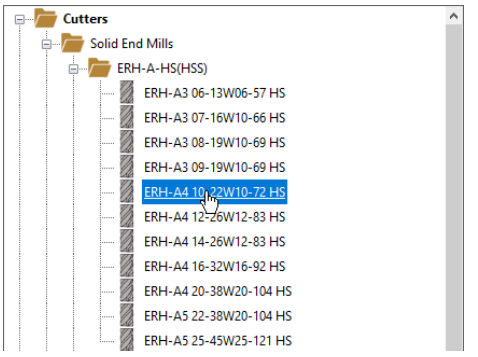
To simplify your selections, default Shape types are set for the components as follows:

- Cutter and Shank components default to **Parameter Data**
- Adaptor/Holder components default to **3D Model**

4. Use the Data Toolbar and corresponding Data Pane to define the remaining properties of your component.

Tool Components data

The Data Toolbar buttons and Data Pane parameters and options will vary depending on your selection in the Database window.



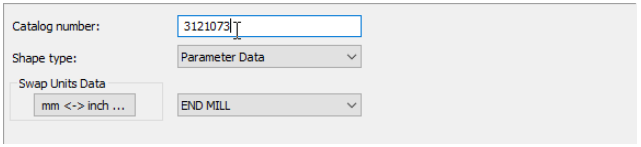
All available Toolbar buttons and their functions are summarized below.

Topology Data



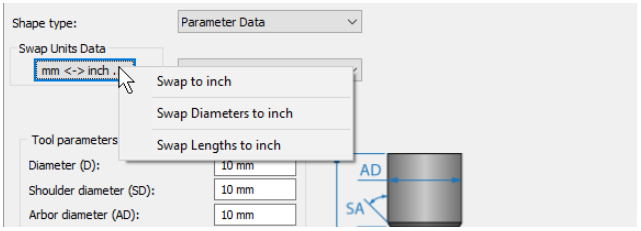
Topology, available for all components, defines the geometry of the component according to the chosen Shape type as mentioned.

In addition, each component can be given a Catalog number for referencing its catalog resource (if any).

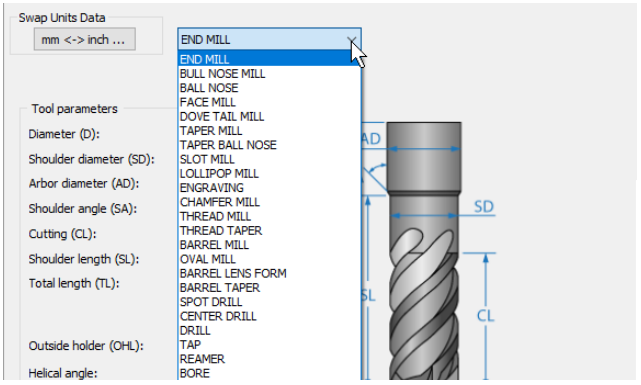


Options are also provided to quickly Swap Units Data (mm to inch and vice versa). Swapping the units of all parameters at once is permitted with all Shape types (Parameter Data, Sketch (2D) and 3D Model).

Milling, Drilling and Special Tools defined by the Parameters Data also allow the swapping of Diameters only and Lengths only.



Also available is the option to make on-the-fly Tool Type changes for Cutter and Shank components.

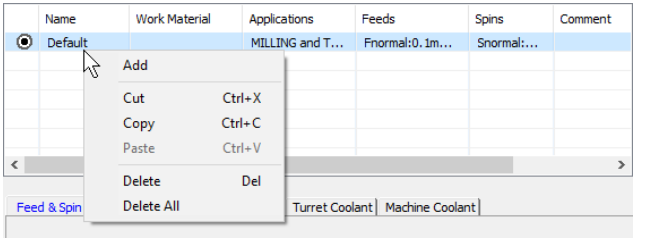


Cutting Condition Data

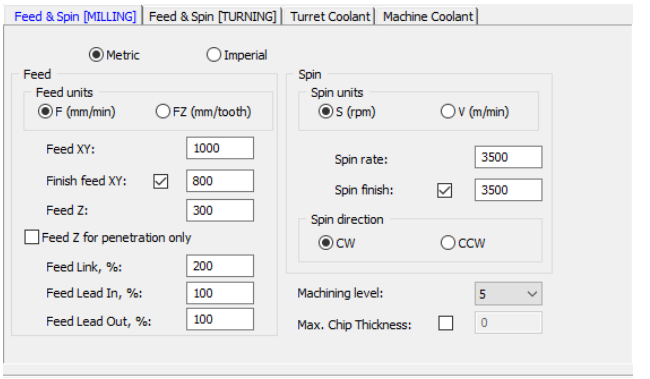


Cutting Condition, available for Cutter components only, enables you to define default sets of cutting parameters for the machining of different Work Materials and for use in different Applications (MILLING and/or TURNING).

Using the table, you can manage any number of Cutting Condition sets.



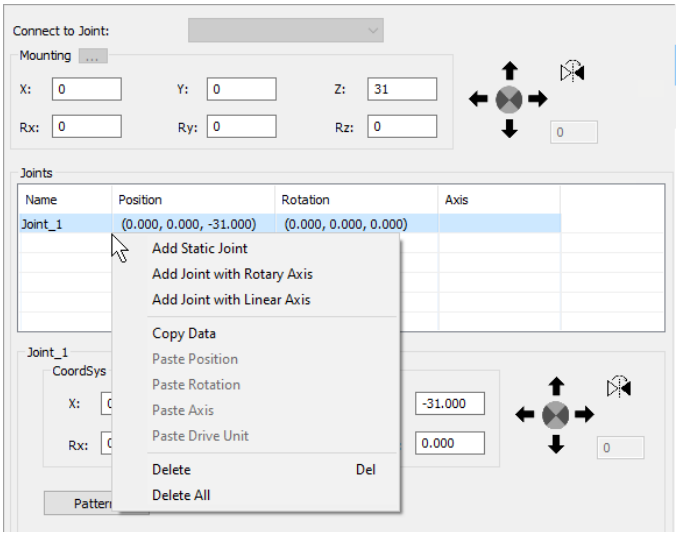
For each Cutting Condition set, you can use the corresponding tabs to specify the default Feed & Spin data (separately for MILLING and TURNING operations) and the default Turret and Machine Coolant options.



Connection Data

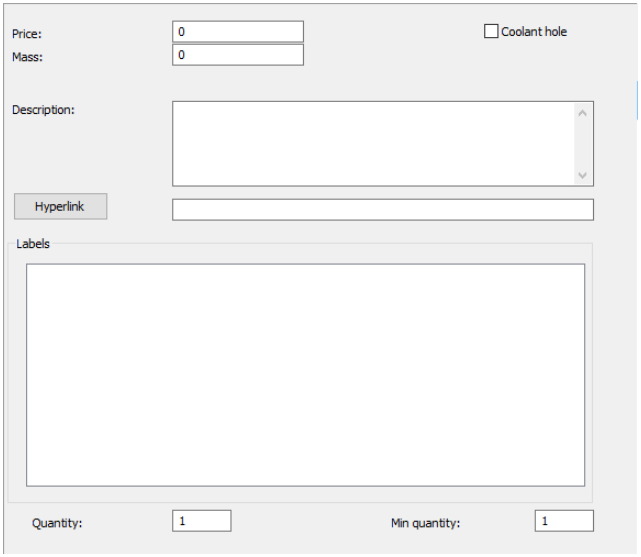


Connection, available for all components except Cutters, enables you to define the default Mounting and Joint CoordSys positions used in tool assemblies.



Properties Data

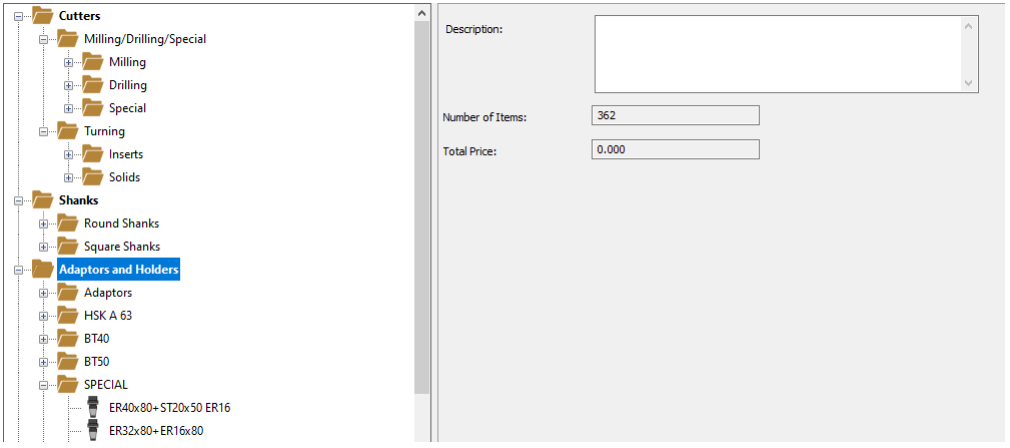
Properties, available for all components, enables you to define additional information relevant to each component, including Price, Mass, Description, Quantity and a Hyperlink for referencing a local or online resource.



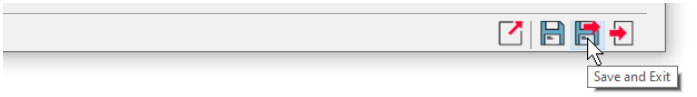
Also available for all Database folders is the Properties Data, which sums the Number of Items and Total Price of all components contained in your folders.



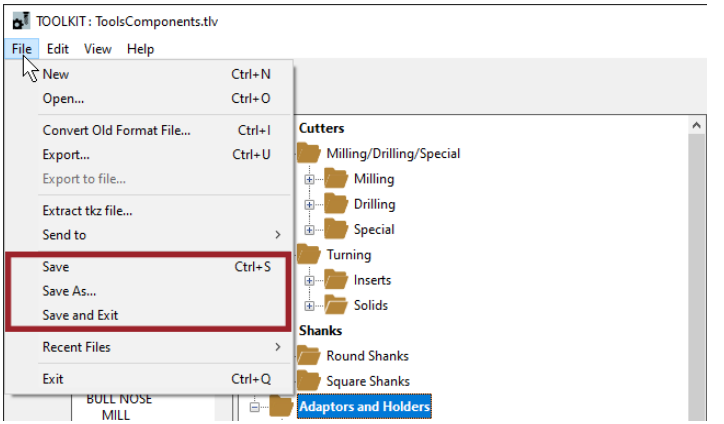
5. Continue adding and defining your components, organizing the Database folders to suit your needs, etc.



6. Once your Tool Components Library is ready, save it using the **Save/Save and Exit** buttons at the bottom right corner of the TOOLKIT dialog box.

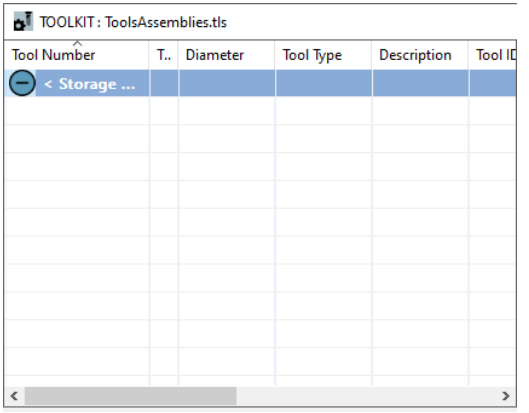


OR using the **Save/Save As/Save and Exit** options in the ToolKit main menu.



3.1.3 Defining a new Tool Assemblies Library (*.tls)

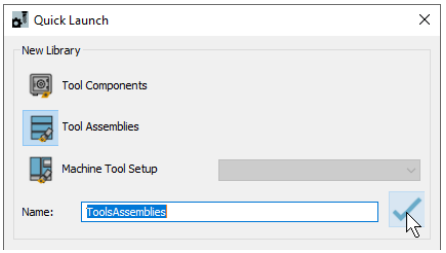
The second level of tool management offered by ToolKit is the Tool Assemblies Library that enables you to manage simple and complex tool assemblies. As mentioned in the Introduction page 7, each tool assembly is referred to as a Tool Item and comprises a combination of tool components such as the cutter and the tool holding system. The defined tools represent those existing in Tool Storage and prepared for use on any given CNC-Machine.



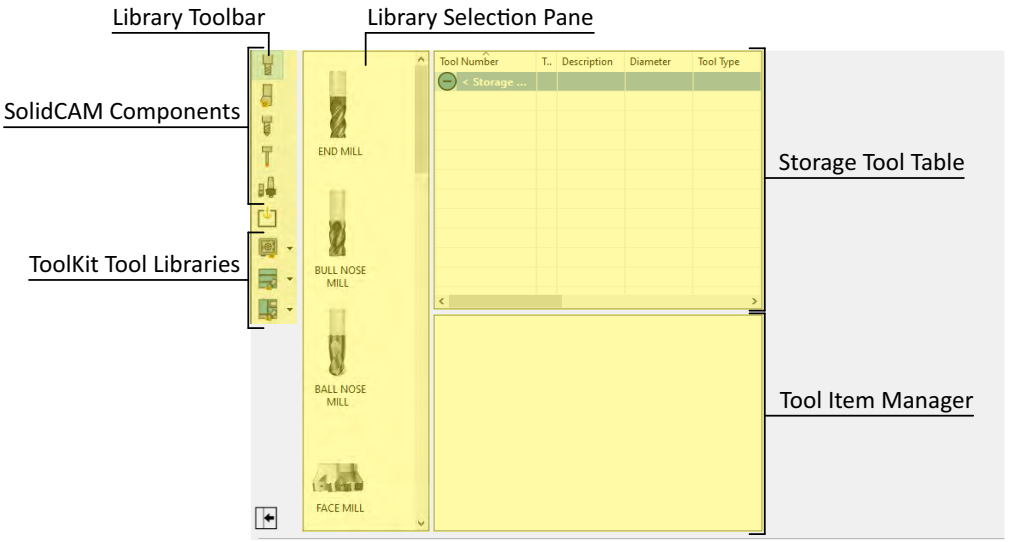
For the purpose of this document, the tools in the new Tool Assemblies Library are defined by importing components from the previously created Tool Components Library.

The steps to define a new Tool Assemblies Library are as follows:

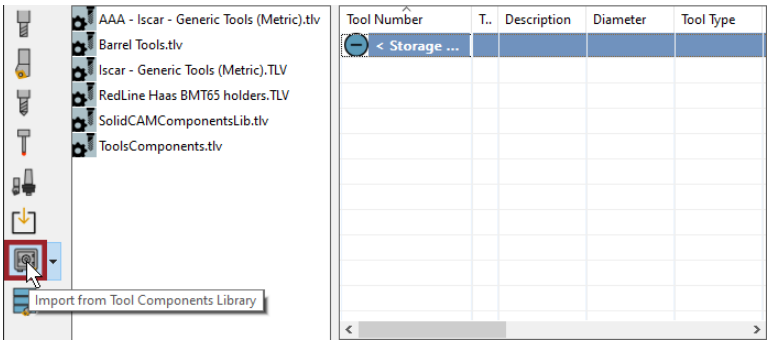
1. Start a new Tool Assemblies Library using ToolKit standalone or one of the various options within SOLIDWORKS/SolidCAM.



Displayed is the TOOLKIT dialog box at the Tool Assemblies Library level.

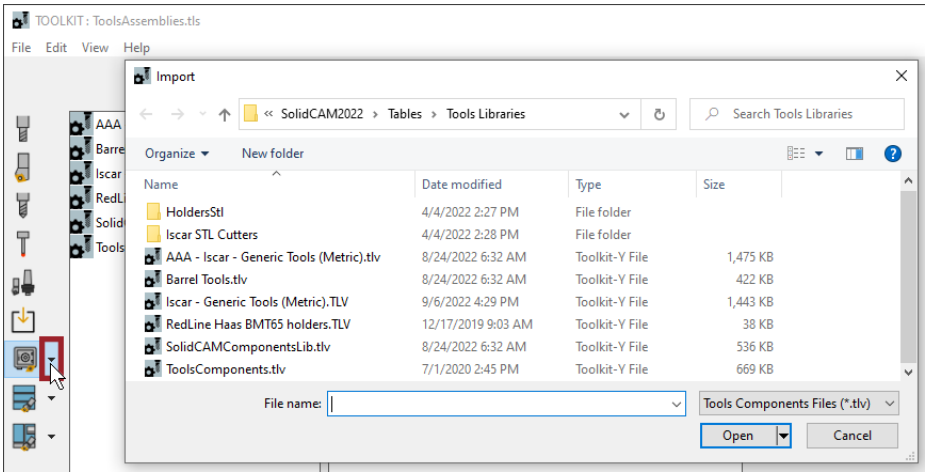


2. On the Library Toolbar, click **Import from Tool Components Library** as shown.

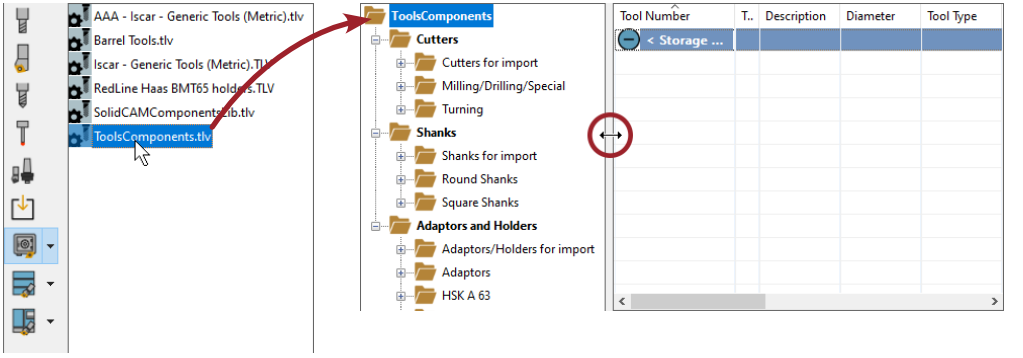


Displayed in the Selection Pane are the Tool Components Libraries (*.tlv files) existing in the **Default folder** directory (defined in ToolKit Settings).

If you instead click the arrow icon, you can use the Import dialog box to browse to and open a Tool Components Library that is saved in a different directory.

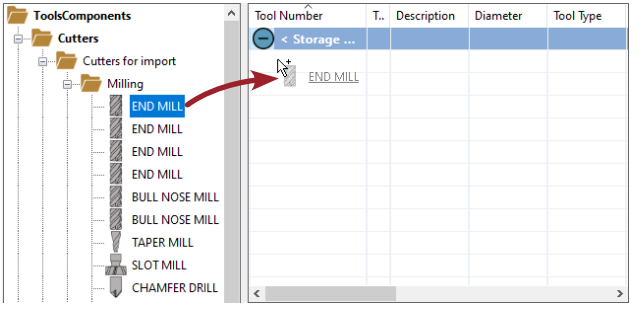


3. In the Selection Pane, double-click the Tool Components Library from which you want to import the components.

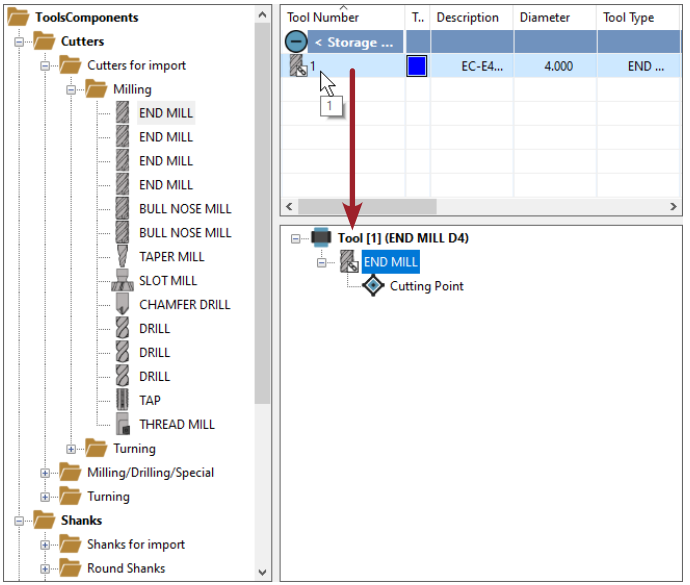


The Selection Pane displays the contents of the Tool Components Library. If necessary, the window can be expanded for better viewing as shown.

4. In the Selection Pane, navigate the folders as needed and double-click the desired Cutter or simply drag & drop it into the Tool Table window.

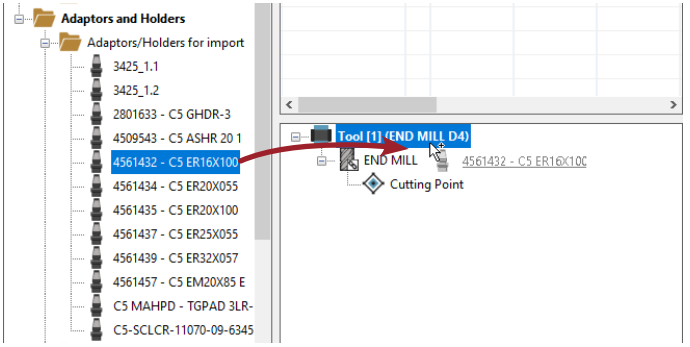


Upon adding the Cutter to the Storage Tool Table, the corresponding Tool Item is created and displayed in the Tool Item Manager.

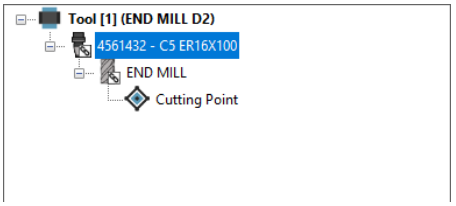


Using the Tool Item Manager, you can create simple and complex tool assemblies and you can view/edit complete information about the tool.

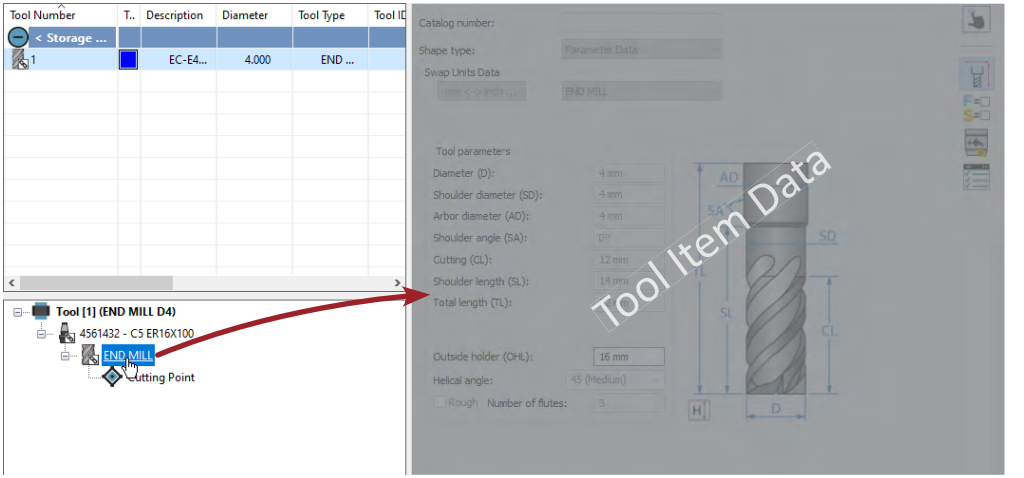
5. Complete the tool assembly by importing the necessary additional components such as a Holder. Double-click in the Selection Pane or drag & drop the desired component(s) into the Tool Item window.



In the example shown below, the Holder component is added to the Tool Item tree.



Displayed on the right side of the TOOLKIT dialog box is the Tool Item/Component Data. Similar to Tool Components Libraries, the Data Toolbar buttons and Data Pane parameters and options will vary depending on your selection in the Tool Item tree.



6. Use the Data Toolbar and corresponding Data Pane to view and edit the properties of your tool assembly.

   Imported components are linked to the ToolKit Tool Library chosen for import (Tools Components.tlv in this case) and have editing restrictions. Only the properties relevant to the tool assembly can be edited.

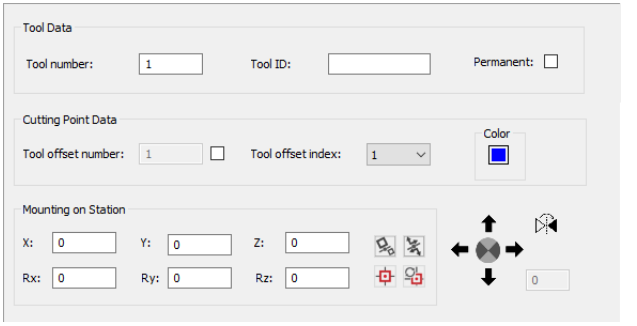
Tool Assemblies data

The properties relevant to the tool assembly include those in Quick Access Data, Connection Data, Offset Data and Properties Data.

As summarized in the Introduction page 11:

Quick Access Data

Quick Access defines the tool identifiers (i.e., Tool Number, Tool ID, Tool offset number/index, etc.) and the tool mounting position on any given CNC-Machine.

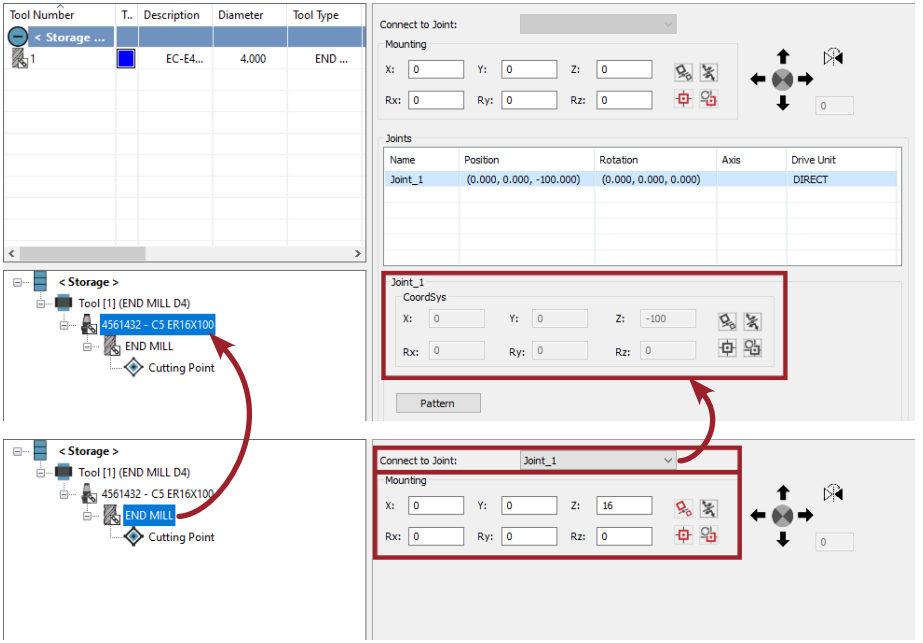


Connection Data



Connection enables you to define, using Mounting and Joint CoordSys positions, the geometric relationships between components in the tool assembly. The positions are inherited from the ToolKit Tool Library chosen for import.

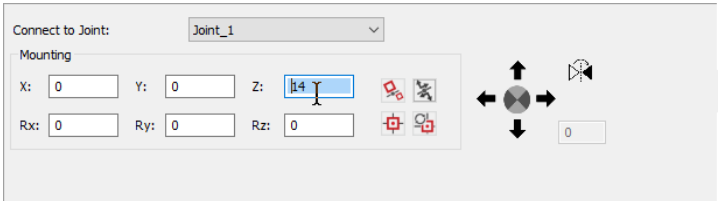
In this example, there is only one connection since the tool is being defined for Tool Storage. The Mounting position of the cutter (END MILL) is connected to Joint_1 of the holder (4561432 – C5 ER16X100).



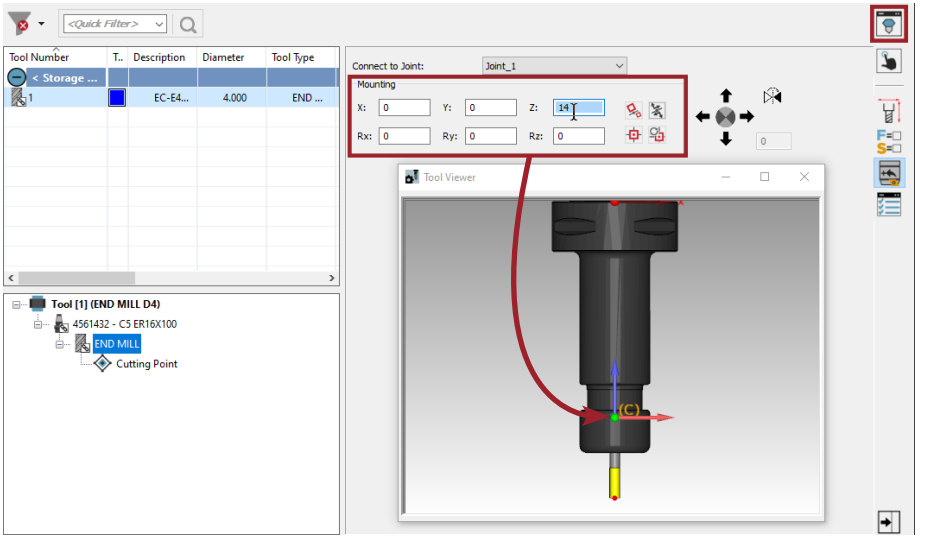
(Advanced View shown for illustrative purposes)

For imported components linked to another ToolKit Tool Library, the Joints feature is disabled and the Joint CoordSys position is locked from editing. Only Mounting positions can be modified.

The Mounting positions can be modified in several ways such as using linear and/or rotational movements. For example, the cutter can be moved along the Z-Axis by entering a value in the corresponding Z field or by scrolling your mouse wheel to change the value gradually.



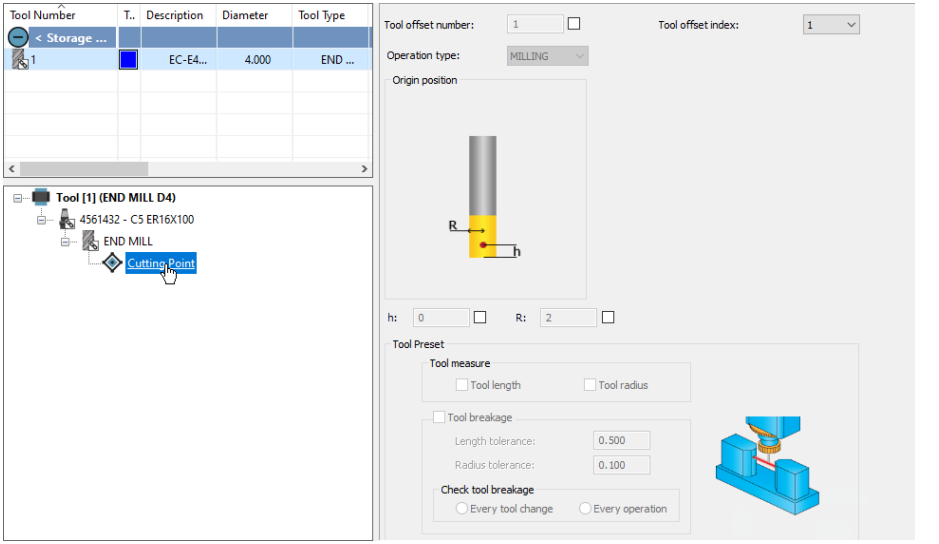
Such movements can be seen in the Tool Viewer window by clicking the **Show/ Hide Tool Picture** button at the upper right corner of the TOOLKIT dialog box.



Offset Data



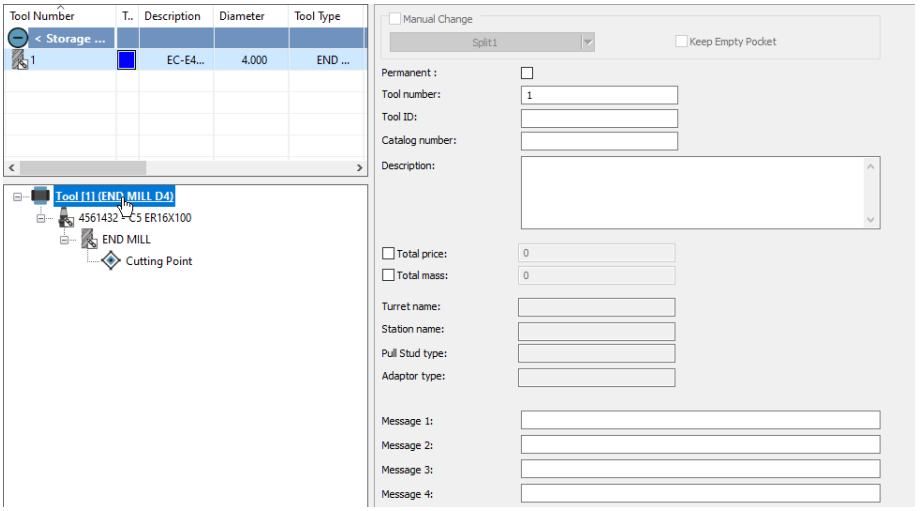
Offset, when **Cutting Point** is selected in the Tool Item tree, enables you to define the Cutting Point data of the tool and the relative Tool offset parameters for identifying its default position in the Tool Offset table of any given CNC-Machine.



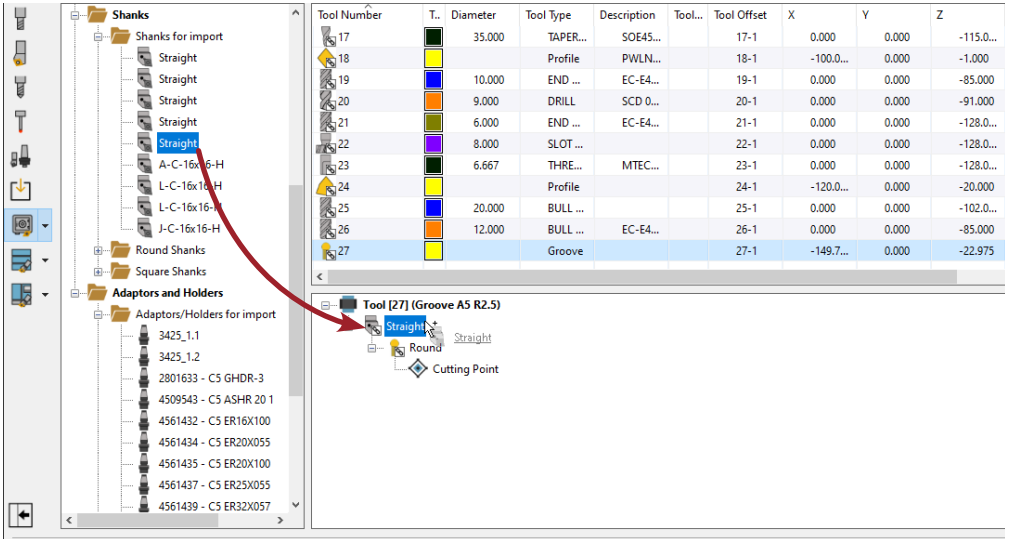
Since a Tool Assemblies Library is not associated with a specific CNC-Machine, the Tool Preset options are disabled.

Properties Data

Properties, when **Tool Item** is selected in the Tool Item tree, enables you to define additional information relevant to the tool assembly, including the Description, Total price and Total mass (if defined for each component) and, for the GCode, Tool Item Messages.



7. Continue importing components and creating your tool assemblies.

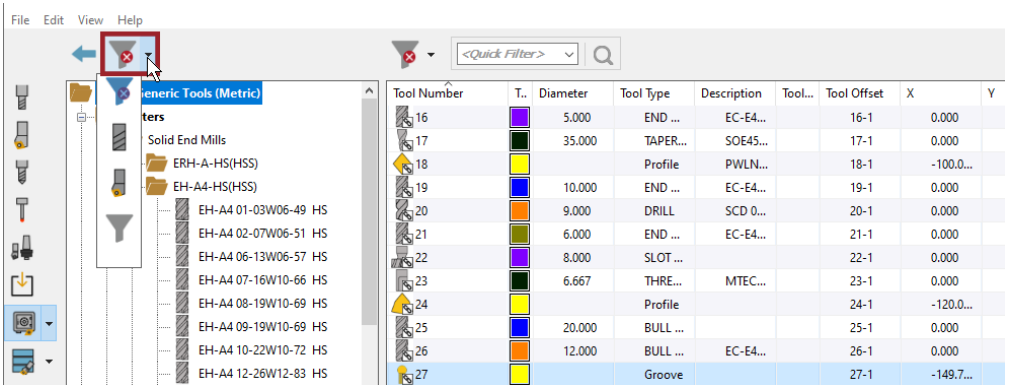


If you are importing from a Tool Components Library that has numerous components, the available ToolKit filter options can help save time when searching for components of specific criteria.

ToolKit filter options

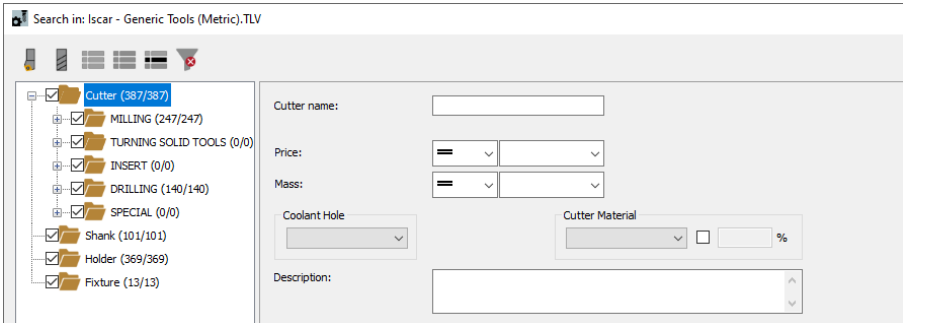
ToolKit offers an extensive range of simple and advanced filters that can be used to facilitate component searches.

Clicking the **Filter** arrow icon as shown displays the options for filtering components in the Selection Pane window.



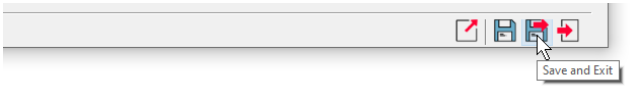
Using the drop-down list, you can select:

- **Cancel Filter** to cancel any active filters.
- **Milling Tools** to display only Milling Cutter Components.
- **Turning Tools** to display only Turning Cutter Components.
- **Advanced Filter** to display the Search in... dialog box that enables you to search for components of specific criteria using various parameters and options.



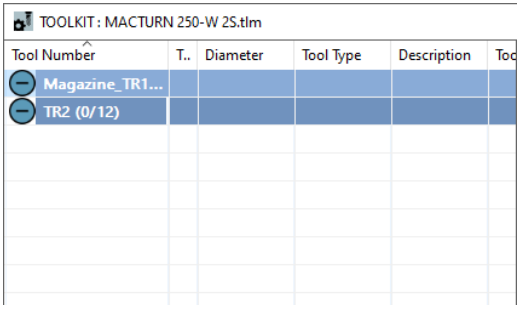
Similar filter options are available for searching tools in the Tool Table window.

8. Once your Tool Assemblies Library is ready, save it using the available **Save...** options.



3.1.4 Defining a new Machine Tool Setup Library (*.tlm)

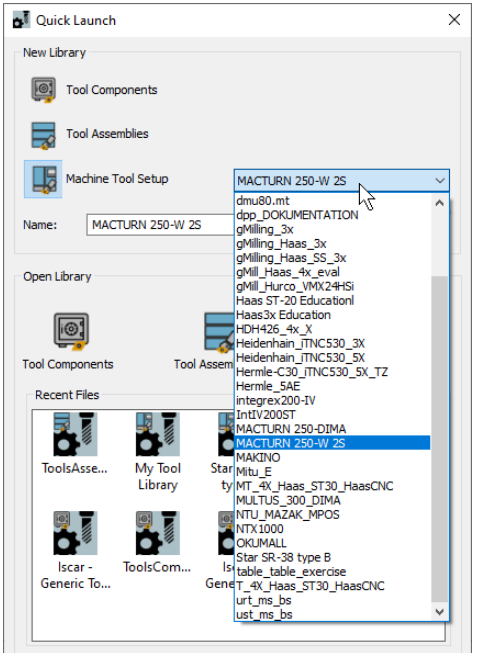
The third level of tool management offered by ToolKit is the Machine Tool Setup Library that enables you to manage tools in the Machine environment. As mentioned in the Introduction page 7, you can define tools and their mounting positions, such as in the Magazine and Tool Stations of the Turret(s), specific to the VMID of your CNC-Machine.



For the purpose of this document, the tools in the new Machine Tool Setup Library are defined by importing tools from the previously created Tool Assemblies Library.

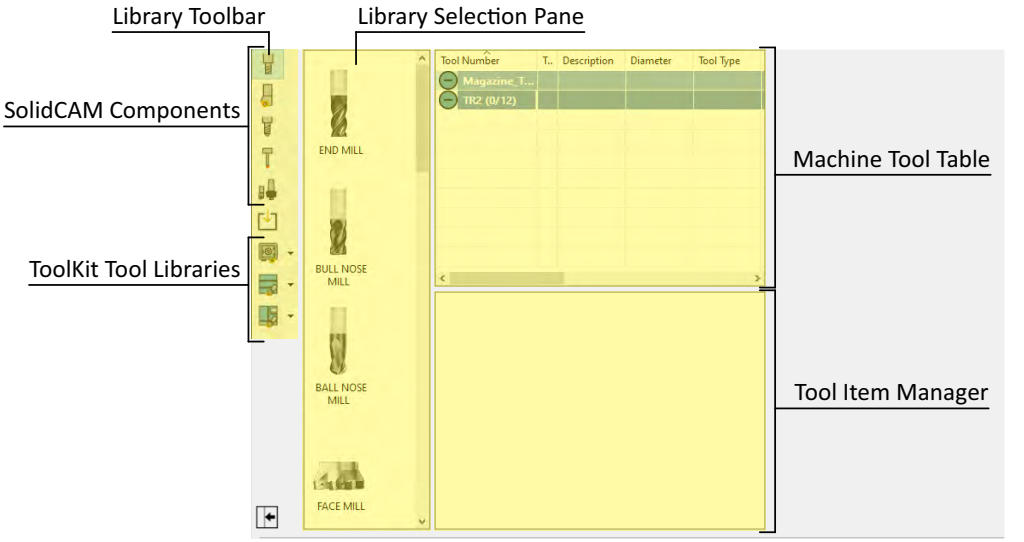
The steps to define a new Machine Tool Setup Library are as follows:

1. Start a new Machine Tool Setup Library.

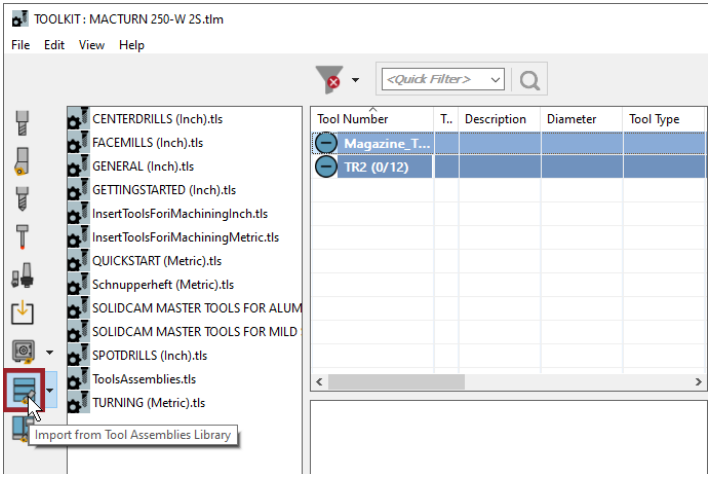


When creating a ToolKit Tool Library of this type, you have to click the corresponding field to display the list of installed post-processors and choose the CNC-Machine you want to use for the basis of your Machine Tool Setup Library.

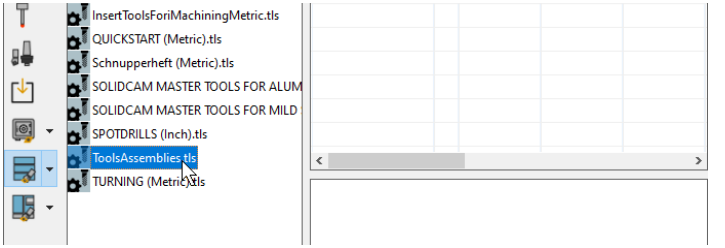
Displayed is the TOOLKIT dialog box at the Machine Tool Setup Library level.



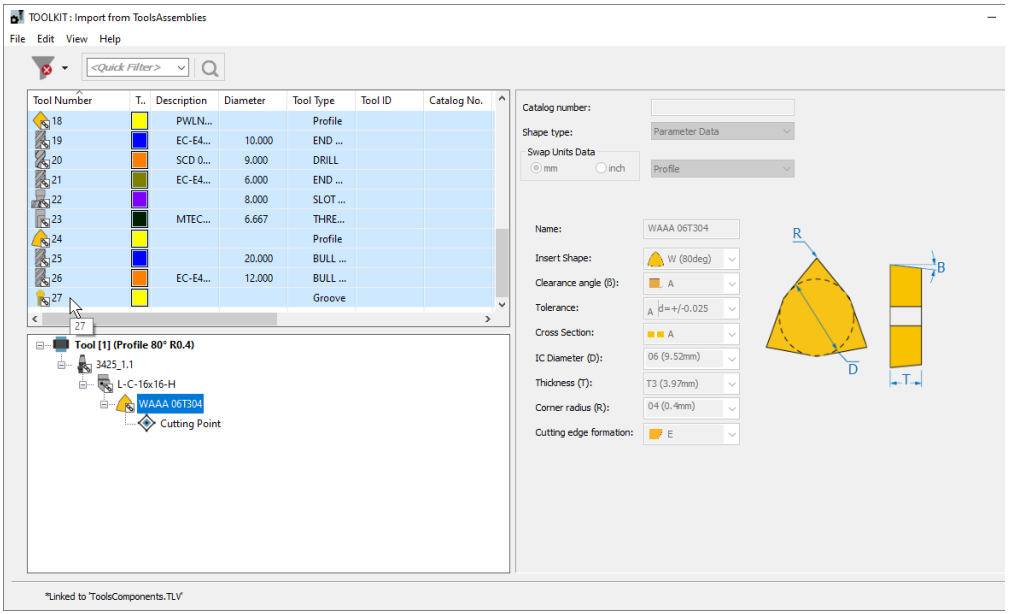
2. On the Library Toolbar, click **Import from Tool Assemblies Library** as shown to display the available Tool Assemblies Libraries (*.tls files) in the Selection Pane.



3. Double-click the Tool Assemblies Library from which you want to import the tools.

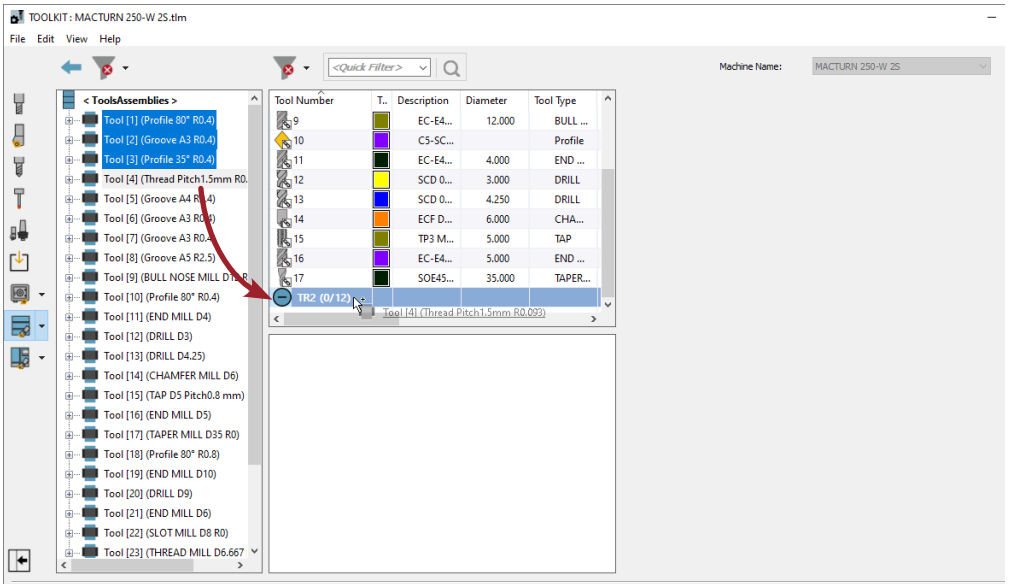


4. Import your tools from the appearing external window.



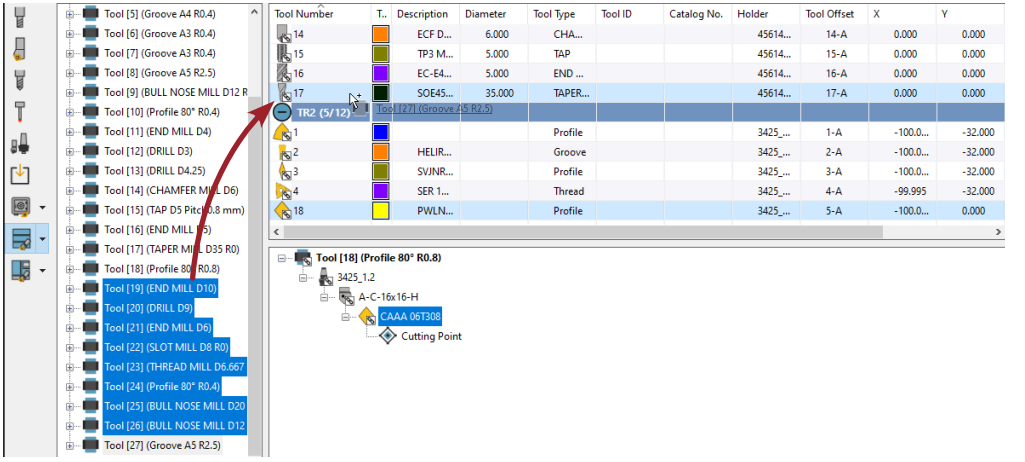
(Using this method, the tools you select are imported to the Machine Tool Table in sequence with matching Tool Numbers/Positions in Magazine.)

OR exit the dialog box and import your tools from the Selection Pane.



(Using this method, you can import tools into the Machine Tool Table in the order and mounting positions of your choosing.)

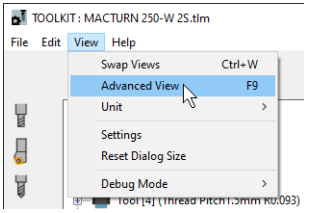
5. Continue importing your tools into the Machine Tool Setup Library.



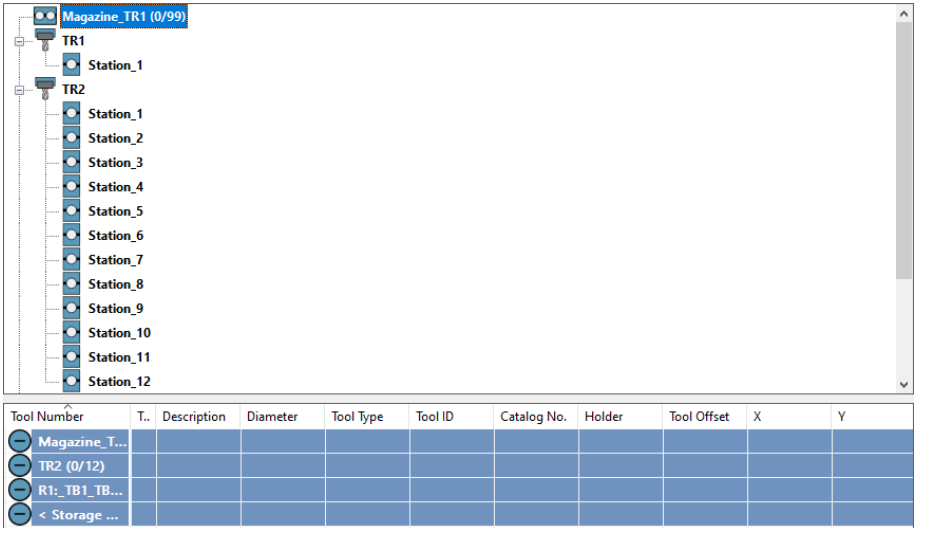
There is also the option to import your tools using the **Advanced View**.

Advanced View

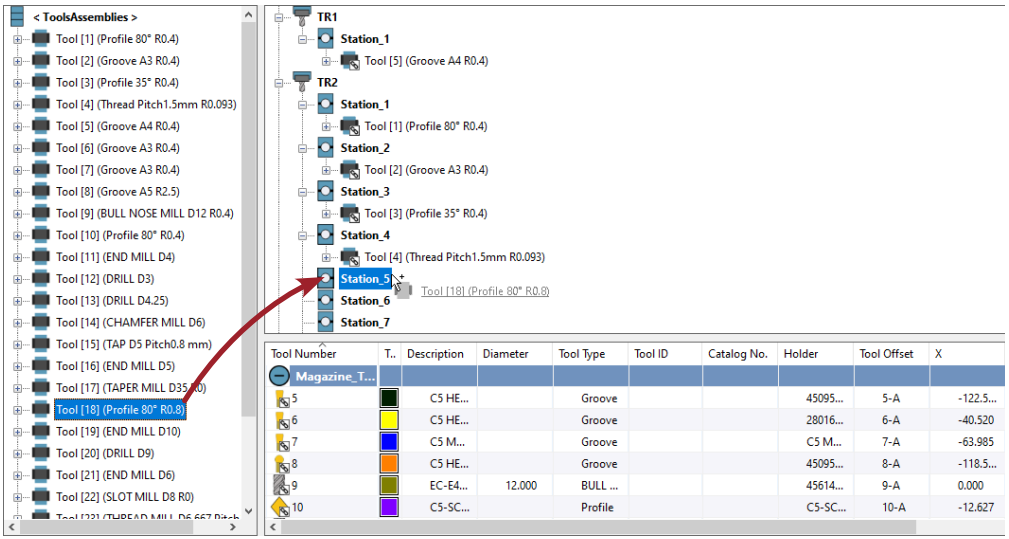
Advanced View can be enabled in the ToolKit main menu or by clicking the **F9** shortcut key on your keyboard.



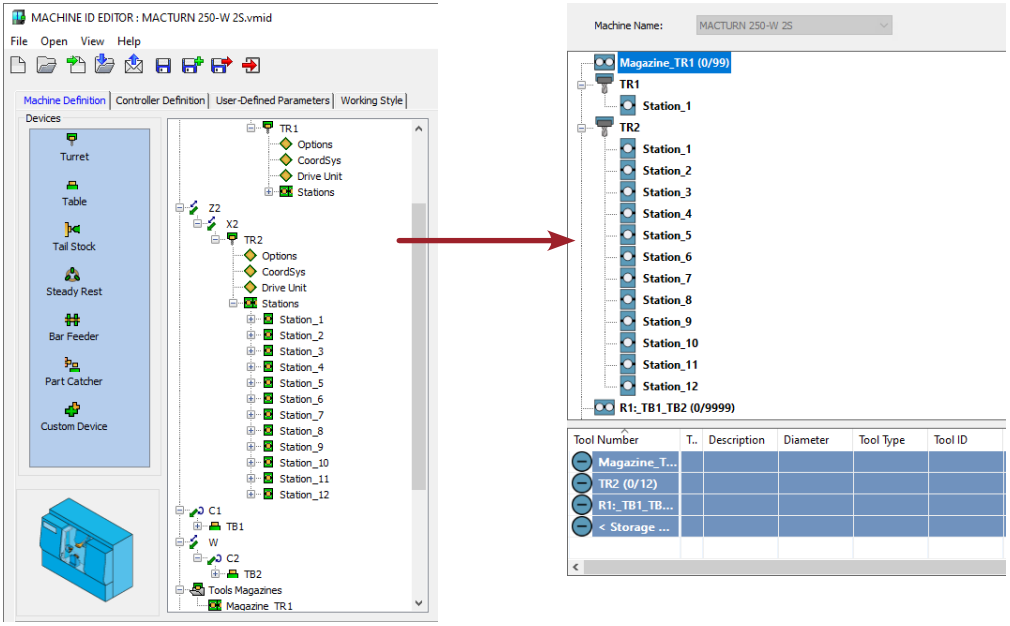
When enabled, the Tool Table and Tool Item windows are swapped in the ToolKit interface.



Displayed in the Tool Item Manager are the tool holding devices of the CNC-Machine. Using the Advanced View, you can import by dragging & dropping your tools directly to the Magazine or to the Tool Stations of the Turret(s). Once mounted, you can use drag & drop to move tools from one Position/Station to another.



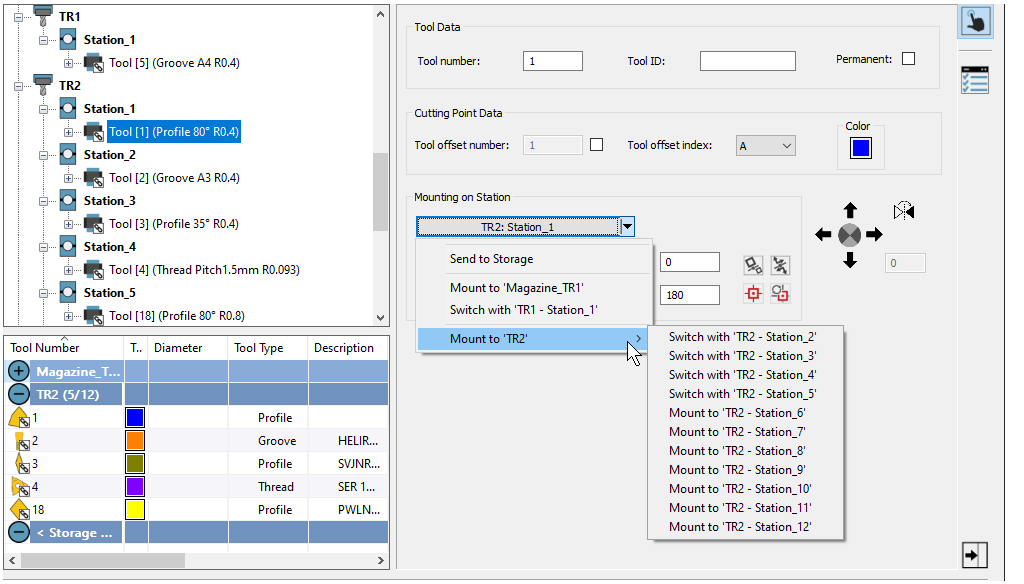
Advanced View provides additional functionality, enabling you to view/edit your tools and their mounting positions in the Machine environment. The hierarchal tree structure reflects that which is defined in the *.vmid file of your CNC-Machine.



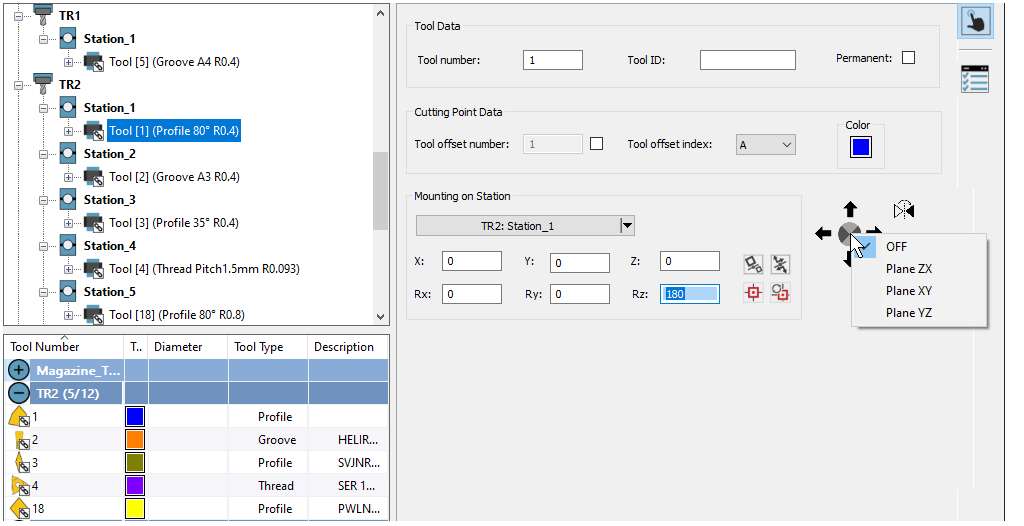
6. After you have imported your tools, define/modify their mounting positions on the CNC-Machine using the Mounting on Station options in Quick Access Data.

Mounting on Station

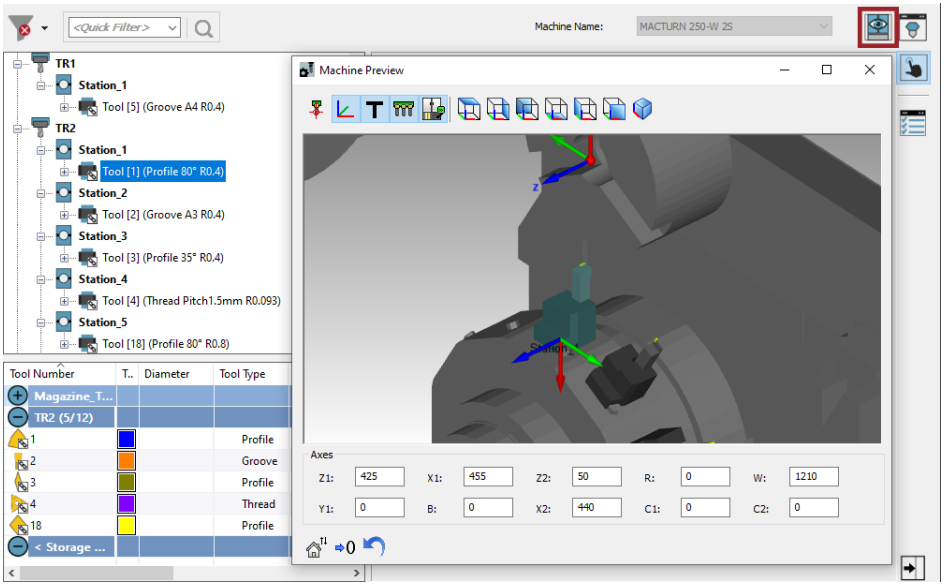
In Quick Access Data, Mounting on Station drop-down list enables you to quickly change the mounting position of Tool Items on the CNC-Machine.



You can also modify, using linear and/or rotational movements, the tool position relative to the CoordSys position of the Station to which it is mounted.



Such positions and movements can be visualized in the Machine Preview window by clicking the **Machine Preview** button at the upper right corner of the TOOLKIT dialog box.



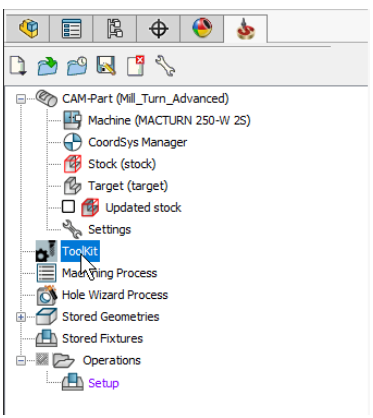
7. Once ready, save your Machine Tool Setup Library.

3.1.5 Importing ToolKit Tool Libraries to the CAM-Part

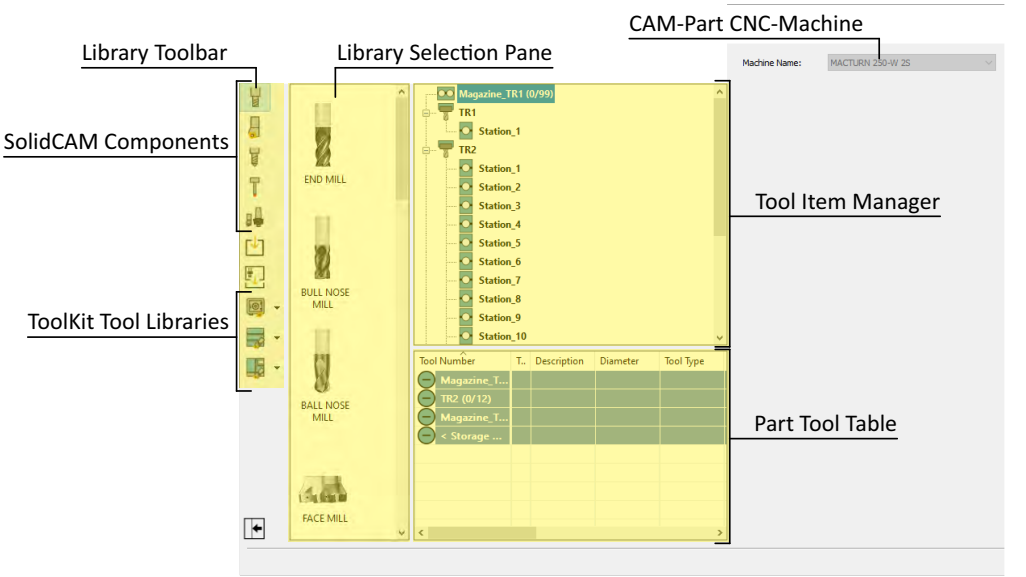
SolidCAM enables you to import tools from any ToolKit Tool Library into the Part Tool Library of your CAM-Part. For the purpose of this document, a new Part Tool Library is defined by importing tools from the previously created Machine Tool Setup Library.

The steps to define a new Part Tool Library are as follows:

1. With your CAM-Part opened, double-click the **ToolKit** header in the SolidCAM Manager.

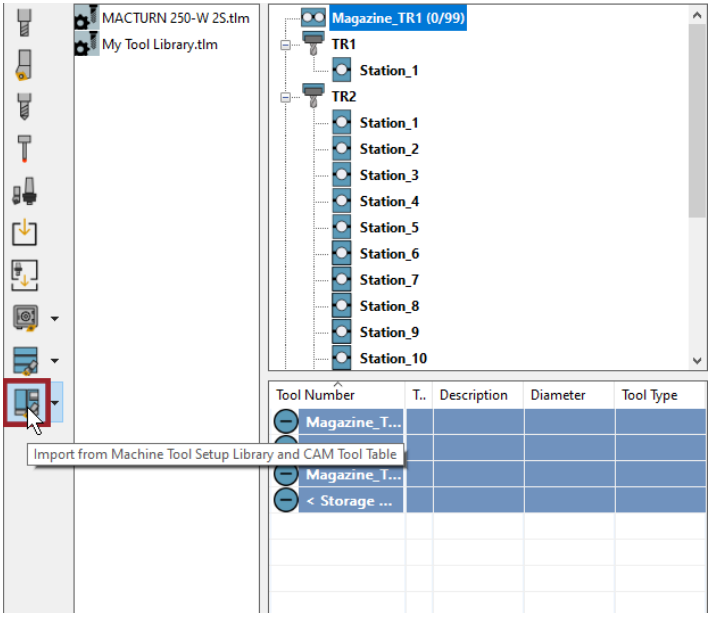


Displayed is the TOOLKIT dialog box at the Part Tool Library level with Advanced View enabled. As mentioned in the Introduction page 8, the Part Tool Library and Machine Tool Setup Library interfaces are identical in appearance, features and options.

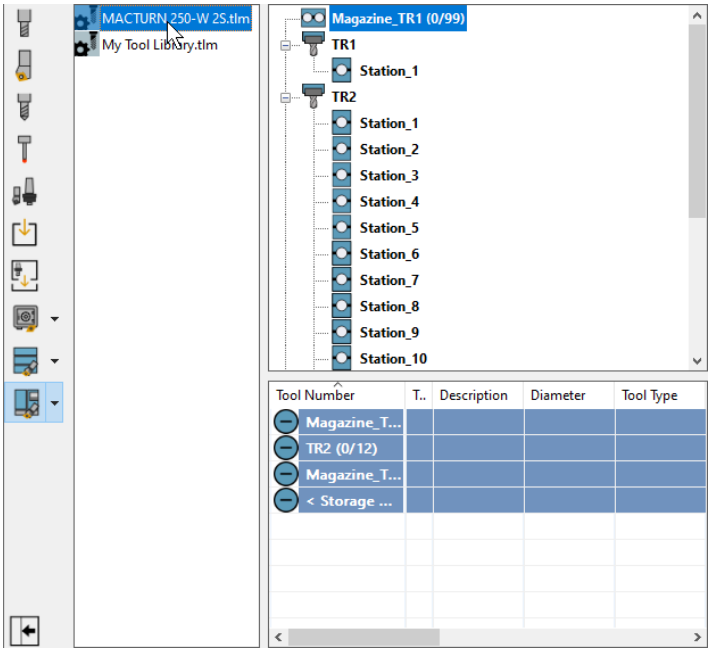


In the following example, the CAM-Part CNC-Machine exactly matches the CNC-Machine used for the basis of the previously created Machine Tool Setup Library.

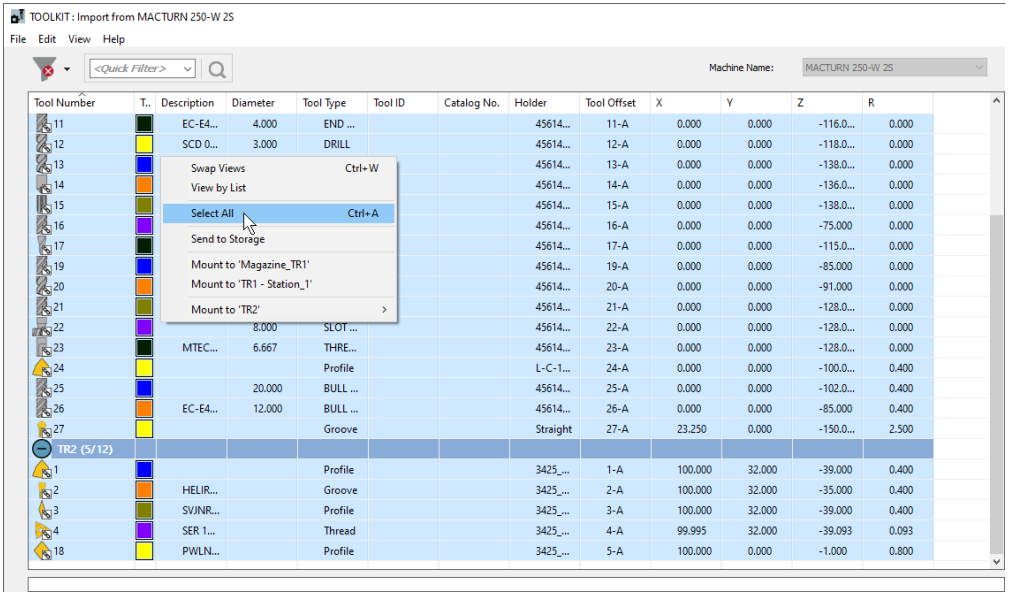
2. On the Library Toolbar, click **Import from Machine Tool Setup Library** as shown to display the available Machine Tool Setup Libraries (*.tlm files) in the Selection Pane.



3. Double-click the Machine Tool Setup Library you want to use for importing tools.

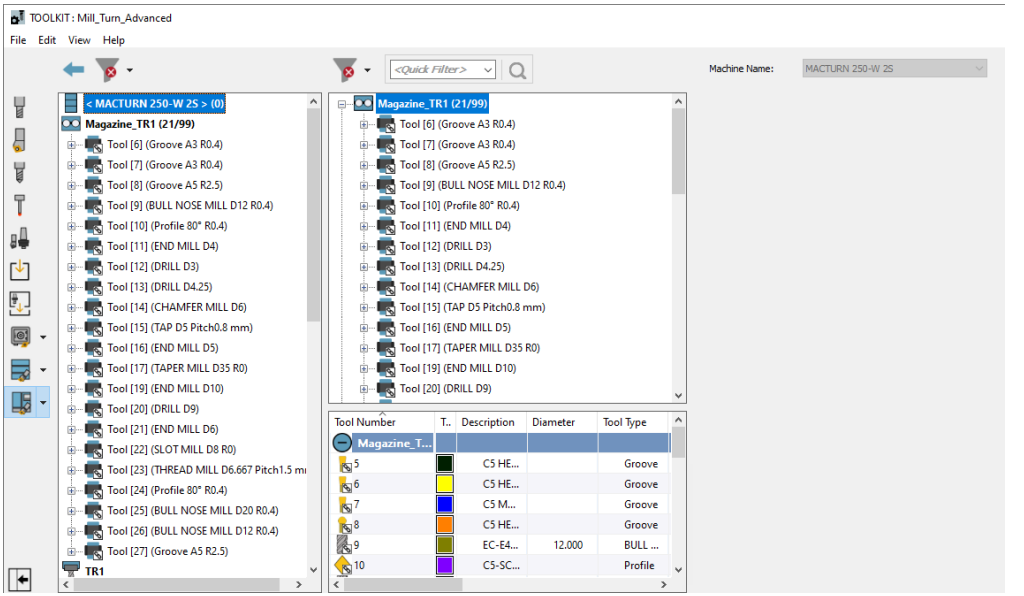


4. Import your tools from the appearing external window.



All tools are selected in this case since the Machine Tool Setup Library was predefined for use with the current CAM-Part.

Because the Part Tool Library was empty, the tools are imported with their order and mounting positions preserved.



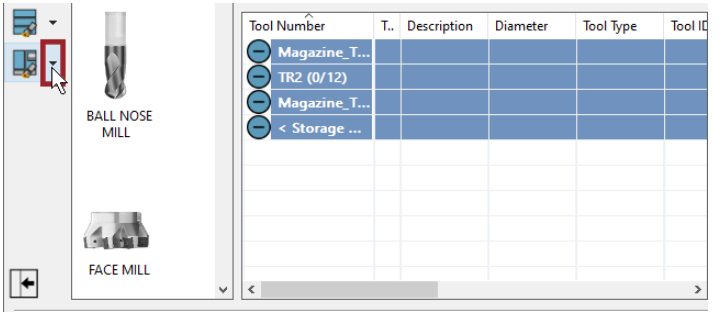
5. Once ready, save your Part Tool Library. The imported tools can be selected for use in your operations.

3.1.6 Importing from TAB Tool Libraries of previous versions

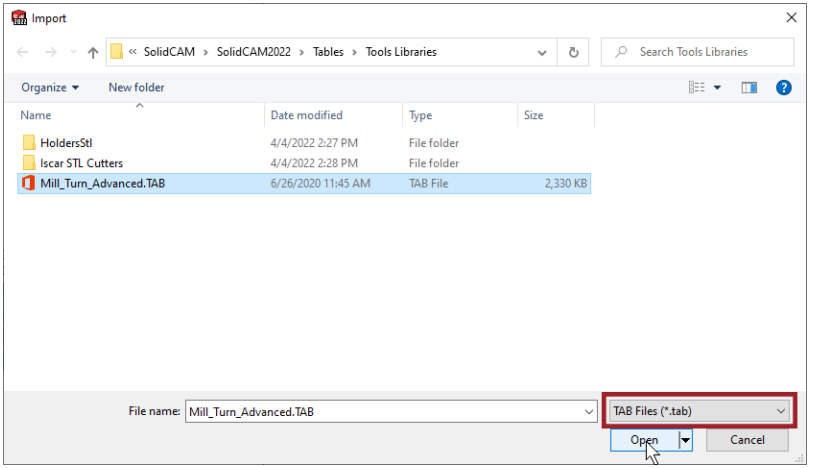
If you have a TAB Tool Library (*.TAB file) from a previous version, it can be used for importing. During the import process, your TAB tools are automatically converted to Tool Items that can be imported to Tool Assemblies, Machine Tool Setup and Part Tool Libraries.

The steps to import TAB tools to either of the three mentioned ToolKit Tool Libraries are as follows:

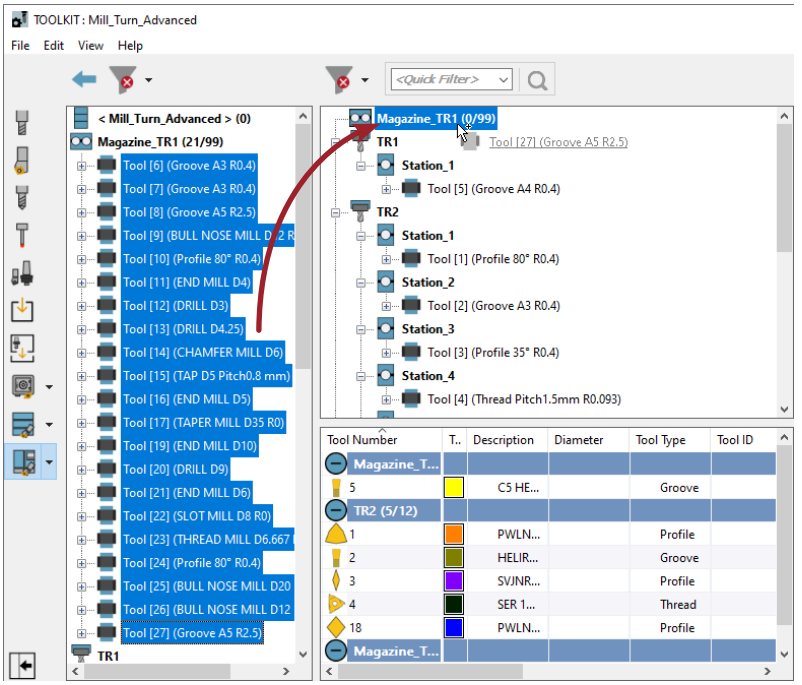
1. On the Library Toolbar, click the arrow icon on either **Import from Tool Assemblies Library** or **Import from Machine Tool Setup Library** as shown.



2. Using the Import dialog box, select **TAB Files (*.tab)** for the file type and browse to and **Open** the TAB Tool Library you want to use for importing.



3. Import your tools from the appearing external window or you can exit the external window and import your tools from the Selection Pane.

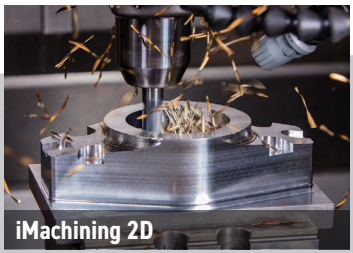


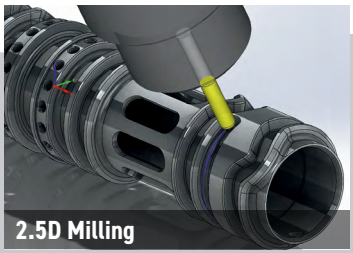
Once imported, the tools can be edited and, when imported to a Part Tool Library, selected for use in the operations of your CAM-Part.

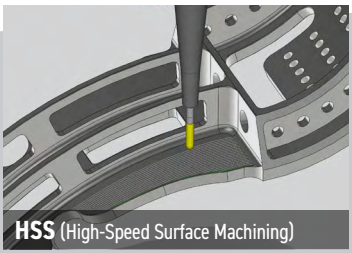
SolidCAM 2022

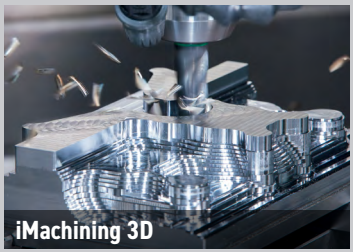
SolidCAM Reference Guide

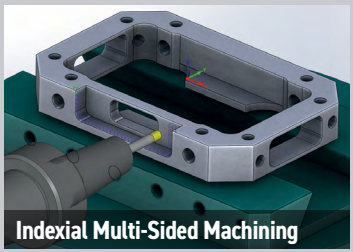
ToolKit

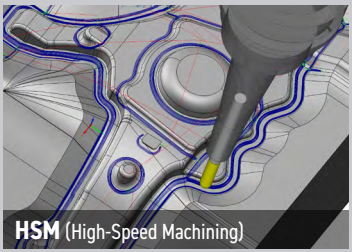

iMachining 2D

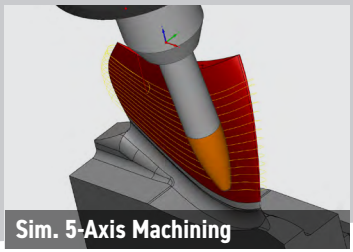

2.5D Milling


HSS (High-Speed Surface Machining)

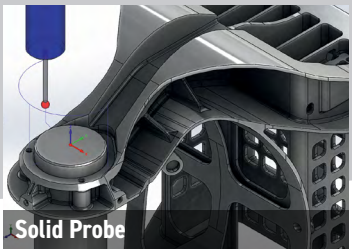

iMachining 3D


Indexial Multi-Sided Machining


HSM (High-Speed Machining)


Sim. 5-Axis Machining


Turning & Advanced Mill-Turn


Solid Probe

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