GIBBSCAM | 2027 CAM for Production Machining

Version 2027 : September 2026

Plug-Ins

GIBBSCAM

Contents

Introduction 7

About Plug-In Manager 7

About the Plug-Ins 7

Geometry 7
Solids 8
Toolpath 8
Tool Libraries 8
Tool Managers 9
System 9
Main Tools 9
Posting 10
PLM 10
Tool Managers 10

Plug-Ins That Have Been Moved or Superseded 10

Plug-In Reference 14

3D Control Manager 15

5-Axis Toolpath Conversion 16

Add G-Code 16

Break Parent-Child Link 16

About Linked Operations (Parent/Child) 17
Links that are not broken 18

CoroPlus Tool Manager 18

What Is CoroPlus? 19

APT-CL Post 19

Auto CS Create 20

Autodesk Inventor Options 21

What Are Autodesk Inventor Features? 21

PLM > Vault 21

Servers for Files and for Database 22

Bottom Up Hit Flats 22

Change Backup File Location 22

Change Feeds and Speeds 22
Rapid to Feed 23
Operation Selection-Action 23
Operation Selection 23
Action 24
Log Any Changes 24
Milling Spindle Speed 24
Turning Spindle Speed 25
Milling FeedRate 26
Modifications 27
Turning Feedrate 28
Modifications 30
Change Hole Orientation 31
CleanUp 31
Cloud-Based Tool Libraries 32
Contour Trace 32
Convert to Trochoidal 33
Steps 34
Examples 34
Create D-Hole 35
Create Plunge Rough Process 36
Spiral 36
Plug-In options with no geometry selected 37
Plug-in options with geometry selected 37
Taper Thread 38
Custom Process Manager 40
Divide Contour 41
Face IJK to Polar 41
Find Ops 42
GARR Tool Library 43
GeoEdit 43
Get Draft Angle 45
GRANITE Information 45
Groove Cycle 45
Harvey Tool Library 46

Helical Solutions Tool Library 46
Helix Bore 46
Helix Builder 46
High Speed Machining (HSM) 47
Settings tab 47
Action 48
Arc Segmentation 49
Manage Dialog Settings 49
Extend Strokes tab 49
Entry Feed Lines tab 49
On/Off Moves tab 50
Lace Cut Stepovers tab 51
Sharp Corners tab 51
Uphill / Downhill 53
Results tab 53
Import Material 54
Import VNC 54
Interface 55
Files 56
Import Options 56
Action 57
File Handling 57
CS Matching 57
ISCAR Tool Advisor (ITA) 59
Machine Info 59
MachiningCloud Tool Library 59
Mirror Ops 60
Model Associativity 60
Offset Contour 61
Okuma Turn-Cut 61
Onshape 62
Settings 63
Update Check 63
Browse 64
Workflow 65
Pathfinder 66

Pinch Turning 67
Project Onto Solids 67
Reporter Basics 67
Part Report 68
Tool Report 69
Operation Report 70
Rotary Rough 71
Direction Settings 72
Stepover Settings 72
Machining Parameters 73
Set Up Post Editor 73
Show Surface Normals 73
Show Position 74
Solid Edge Options 77
Solid Inquiry 77
Solids Alignment 77
up2parts AutoCAM 79
Stock Wizard 80
Where Do I Find the Stock Wizard? 80
How Will the Stock Wizard Help Me? 81
Elements of the Stock Wizard 81
Dialog 81
Graphics 81
Stock Buttons 82
Action Buttons 82
Using the Stock Wizard 83
Surface Tools 85
Iso Curves 86
ISO Curves Examples 86
Center Curves 87
Center Curve Examples 88
Extend Surfaces 90
Extend Face Examples 90
Extract Edges 91
Extract Edges Examples 92
Swiss Data Setup 93
Thread Whirling 93

Tool Library Explorers 94
Using Tool Library Explorers 94
Tool Manager iQ 97
ToolsUnited Tool Library 97
Touch Interface Manager 97
Tp Transform / Transform Toolpath 98
About Linked Operations (Parent/Child) 98
Z Ramp Contour / Z Step 100

Distribution Only Plug-Ins 101
4/5 Axis Post Engine Selection 101
Deburring Process 101
Get Section 104
Line-Line Intersect 104
MDD Power Tools 105
Set Part Origin 106
Set Process to Face Approach 106

Conventions 107
Text 107
Graphics 107

Links To Online Resources 108

Introduction

This *Plug-Ins* guide provides information on the feature set referred to as plug-ins. Plug-ins are built into the system to provide enhanced functionality. They were created as a quick and flexible response to common feature requests. This document is intended to be a supplement to the full range of GibbsCAM guides, as the plug-ins are targeted at various sections of the system such as Geometry Creation, Mill, and SolidSurfacer. You should be familiar with these guides before using the plug-ins.
Note: The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer, GibbsCAM Operator and GibbsCAM Student Edition provide a subset of the full functionality.

About Plug-In Manager

Plug-In Manager, available under the Plug-Ins menu, lets you globally activate/deactivate individual plug-ins and plug-in groups.
Important: Deactivating a plug-in disables that plug-in's function *throughout GibbsCAM*. For example:

- If the OptiCAM EDM plug-in (`wire_gc_lh.dll`) is deactivated, then OPTICAM does not appear on the main menu.
- If the Transform Toolpath plug-in (`TpTrans.dll`) is deactivated, it does not appear on the list of available Operation Modifiers.
- If the Read SolidEdge Assembly plug-in (`ReadSEAsm.dll`) is deactivated, the SolidEdge Assembly (*ASM) choice does not appear among the file types offered in the Open part file or Import file dialogs.

About the Plug-Ins

The following is a list of plug-ins currently available in GibbsCAM.

Geometry

GeoEdit
Several plug-ins that help you to modify curves and shapes, often interacting with solids.
Taper Thread
Creates geometry for a tapered thread.
CleanUp
Deletes duplicate geometry within specified tolerances.
Contour Trace
Generates a contour based on a chain of lines and circles you select.
Create D-Hole
Creates D-shaped or double D-shaped geometry.
Spiral
Creates spiral geometry that may be machined or projected onto solids.

Divide Contour
Allows you to subdivide the elements of a selected contour into evenly divided segments, joined by points.
Helix Builder
Creates helical geometry.

Solids

Surface Tools: Iso Curves, Center Curves, Extend Surfaces, Extract Edges
Functions for creating geometry or surfaces from the selected faces of bodies.
Auto CS Create
Automatically creates coordinate systems based on the solid faces or features (such as holes, cylinders, or cones).
Model Associativity
Designed for use when a model has changed and you want to update the operations.
Show Surface Normals
Displays surface normals across selected faces to highlight the curvature across each face.
Solid Inquiry
Provides information about the faces of a selected body.
Solids Alignment
Allows for the manipulation of the position of solid models.
Get Draft Angle
Informs the user of the draft angle of a selected face.

Toolpath

Change Feeds and Speeds
Changes feedrates and spindle speeds for operations. Also available as an operation modifier.
High Speed Machining (HSM)
Provides functionality for high speed machining (HSM), allowing you to extend toolpath, put loops in lace cut toolpath, change rapid moves to feed moves, etc. Most capabilities are also available in operation modifiers: Rapid to Feed and High Speed Machining.
Mirror Ops
Mirrors selected geometry and toolpath.
Tp Transform / Transform Toolpath
Allows toolpath to be quickly rotated, duplicated, or repositioned. The Tp Trans plug-in creates copies and applies a Transform Toolpath operation modifier to each copy.
Break Parent-Child Link
Converts a child operation to full independence from its parent. Each independent operation is associated with its own independent process.

Tool Libraries

Tool Library Explorers
Provides an interface for importing tools from the catalogs for GARR Tools, Harvey Tool, and Helical Solutions.
CoroPlus Tool Manager
Internet connection required. Provides an interface for importing tools from Coromant CoroPlus Tool Manager catalog. Also available in the "Select Tool Type" fly-out summoned

when you right-click a process and choose "Change Tool".
For complete information, see the *CoroPlus Tool Library (CPTL) Plug-In* guide.

Cloud-Based Tool Libraries

Internet connection required. Provides an interface for logging in to your account and creating/importing tools from ToolsUnited and MachiningCloud.

Tool Managers

Tool Manager iQ

Internet connection required. Provides an AI-mediated interface for managing tools, toolholders, fixtures, and assemblies, and for importing them into GibbsCAM.

System

Pathfinder

A group of shortcuts to important folders and files.

Touch Interface Manager

Provides information on activating and using the touch gestural controls.

Machine Info

Displays various MDD and VMM information.

Plug-In Manager

See About Plug-In Manager.

Main Tools

Show Position

Provides for position inquiries on rendered parts.

Import VNC

Imports any VNC file with operations into any open operation in the system.

Stock Wizard

Defines simple stock in two or four steps: rectangular or cylindrical, with or without a single hole.

Find Ops

This plug-in is a search tool. It finds operations by the operation type, the tools used, geometry or solids that get machined.

Reporter Basics

Generates an array of reports on part files that are fully customizable. For complete information, see the *Reporter* guide.

3D Control Manager

Allows for the customization of a 3D controller device.

Project Onto Solids

Modifies the shape and or depth of geometry to lie on a body.

up2parts AutoCAM

Lets you design or choose models on the up2parts website and use AI to generate a solid model with feature tree, stock information, tooling features with operations, and meta information for orientation/clamping.

Posting

APT-CL Post
Provides one or more configurations for posting to APT-CL.

PLM

Onshape
Provides access to settings for Onshape, and lets you check for updates and browse CAD files in Onshape libraries and other sources.

PLM > Vault
Provides access to settings for Autodesk Vault, and lets you check for updates and browse files in the vault.

Tool Managers

Tool Manager iQ
AI-based tool management system for organizing cutting tools, holders, assemblies, fixtures, and suppliers.

Plug-Ins That Have Been Moved or Superseded

Add G-Code
No longer a plug-in.
This functionality is now covered in the *Multi-Task Machining (MTM)* guide.

Bottom Up Hit Flats
No longer a plug-in.
This functionality is now covered in the *Mill* guide, **Roughing** process dialog.

Build Machine
No longer a plug-in.
This functionality is now covered in the *Machine Simulation* guide.

Convert to Trochoidal
Generates trochoidal toolpath (circular toolpath with small fast cuts). Now available as an operation modifier only.

Create Plunge Rough Process
No longer a plug-in.
This item is now a process dialog, covered in the *SolidSurfacer* guide.

Custom Process Manager
No longer a plug-in.

Divide Contour
With one exception – Divide Contour – the tools in this plug-in have been broken out into other areas and incorporated into base functionality.

GRANITE Information
This plug-in was retired at version 2024. Its functionality is now provided natively and/or by **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel).

Groove Cycle
Allows you to cut geometry-independent rectangular grooves using Fanuc-style canned cycles for G74 and G75 output.
No longer a plug-in.
This functionality is now incorporated into the base Turning processes. For details, see the *Turning* guide.

Helix Bore
No longer a plug-in.
This functionality is now covered in the *Mill* guide, **Holes** process dialog.

Import Material
Provides for quick and/or bulk importing of custom material data into the Material database.
No longer a plug-in.
This functionality is now incorporated into the File > Materials dialog and the base processes that contain a Material button. For details, see the *Common Reference* guide.

Offset Contour
No longer a plug-in.
This functionality is now covered in the *Mill* guide, Contour process dialog.

Pinch Turning
Assists in turning a part on a twin turret lathe by using two tools simultaneously.
No longer a plug-in.
This functionality is now incorporated into the base Turning processes for machines with two or more spindles. For details, see the guides for *Turning* (Processes) and for *MTM* (Appendix).

Rotary Rough
This is designed to work with solids in Mill Turn or Advanced CS. Rotary Rough can create three types of toolpath – Linear, Rotary and Helical – along all of the axes.

Set Up Post Editor
No longer a plug-in.
This functionality is now incorporated into the File > Preferences dialog, Post Editor Settings tab. For details, see the *Common Reference* guide.

Solid Edge Options
No longer a plug-in.
This functionality was incorporated into the base Import and Open functionality

Surface Tools
A multi-function tool for creating geometry and surfaces from faces. Now broken out into four separate tools: Iso Curves, Center Curves, Extend Surfaces, Extract Edges.

Swiss Data Setup
No longer a plug-in.
This functionality is now incorporated in the Document Control dialog (DCD), covered in the *Turning* guide, topic "Cylindrical Stock (With Guide Bushing)".

Thread Whirling
A machining process where cutters are mounted on the inside of a cutting ring or cutting holder rather than the outside of a milling tool.
No longer a plug-in.
This functionality is now incorporated into the base Turning processes. For details, see the *Turning* guide.

Z Ramp Contour / Z Step
No longer a plug-in.
This functionality is now covered in the *Mill* guide, **Contour** process dialog.

Adveon Tool Manager
Replaced by CoroPlus Tool Manager in GibbsCAM 13.

ISCAR Tool Advisor (ITA)

Accesses a database of tools; populates the current part's Tool List with selected tools. This is now available in the "Select Tool Type" fly-out summoned when you right-click a process and choose "Change Tool".

Distribution Only plug-ins

Deburring Process

Creates a process to clean up selected edges of parts.

Get Section

Extracts geometry from the intersection between a solid and the HV or HD planes.

Line-Line Intersect

Creates workgroups of points based on where selected lines intersect and where they would leave the Workspace boundary.

MDD Power Tools:

Provides three basic functions: Show Toolgroup CSs, MDD Information List, and Preferences.

Set Part Origin

This plug-in was retired at version 11.0, when its functionality was moved into the **Document Control** dialog (DCD), Workspace tab, as values for Part Offset X Y Z.

Set Process to Face Approach

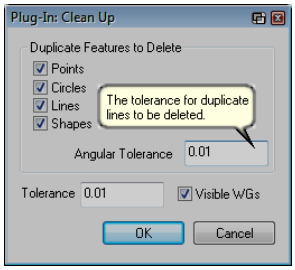
Changes the approach moves of a turning tool.

The plug-ins that are installed on your system will depend on the product options installed. For example, if you do not install a solids product, the plug-ins associated with solids will not be installed. However, there are several plug-ins that are installed (but not initially activated) that may apply to a function you do not have. These are the "Distribution Only" plug-ins. The Distribution Only plug-ins are items that were developed to fit a particular need but may be of use elsewhere. These items often do not adhere to the GibbsCAM look and feel, and are not considered a normal part of the system. For more information on how to activate these plug-ins, see About Plug-In Manager.

Product or Category	*Plug-In Name*
General (always installed)	3D Control Manager, Add G-Code, Autodesk Vault, CleanUp, Convert to Trochoidal (as an operation modifier), Create D-Hole, Create Spiral, Create Tapered Thread, Custom Backup Location, Divide Contour, GeoEdit plug-ins, Helix Builder, Machine Info, Reporter, Show Position, SurfaceTools, Stock Wizard, Touch Interface Manager, Transform Ops
2.5D Solids; **Solids Import**	Auto CS Create, Solid Inquiry, Solid Model Alignment, Solid Edge Options
SolidSurfacer	Solid Inquiry, Create Plunge Rough Process, Project Onto Solids, Solids Alignment, Auto CS Create, Get Draft Angle, Model Associativity, Rotary Rough, Show Face Normals, Solid Edge Options
Distribution Only (always installed)	Calculate Taper Angle, Deburring Process, Get Section, Line-Line Intersect, MDD Power Tools, Set Part Origin, Set Process to Face Approach

> In previous versions of GibbsCAM, users could modify the menu structure of the Plug-Ins by directly modifying the files and folders on the hard drive. You should now use Plug-in Manager to manage your plug-ins. because directly modifying the files on disk will cause duplicate versions to be created when GibbsCAM launches.

Most but not all plug-ins can provide balloon text as quick field-level help. To activate balloons, select the Balloons item under the Help menu . Then place the cursor in any text field within the plug-in. Balloon text will appear as you mouse over text fields and other controls.



Plug-In Reference

- (Main Tools >) “3D Control Manager” on page 15
- (operation modifier) “5-Axis Toolpath Conversion” on page 16
- (Main Tools >) “Add G-Code” on page 16: This plug-in was retired at version 14.0, when its functionality was moved into a utility process. For information on the Add G-Code utility process, see the *Multi-Task Machining (MTM)* guide.
- Adveon Tool Manager: See the *CoroPlus Tool Library (CPTL) Plug-In* guide.
- (Posting >) “APT-CL Post” on page 19
- (Solids >) “Auto CS Create” on page 20
- (PLM >) “PLM > Vault” on page 21
- “Bottom Up Hit Flats” on page 22: This plug-in was retired at version 11.0, when its functionality was moved into the process dialog. For information on the **Contour** and **Roughing** process dialogs, Bottom Up checkbox, see the *Mill* guide.
- (Toolpath >) “Break Parent-Child Link” on page 16
- (operation modifier or HSM >) “Change Feeds and Speeds” on page 22
- “Change Hole Orientation” on page 31: This plug-in was retired at version 11.0, when its functionality was moved into the Hole List context menu. For information, see the *Features* guide.
- (Geometry >) “CleanUp” on page 31
- (Geometry >) Contour Trace
- (operation modifier) “Convert to Trochoidal” on page 33
- CoroPlus Tool Manager: See the *CoroPlus Tool Library (CPTL) Plug-In* guide.
- (Geometry >) “Create D-Hole” on page 35
- (Geometry >) “Spiral” on page 36
- (Geometry >) “Taper Thread” on page 38
- (operation modifier) “Face IJK to Polar” on page 41
- (Main Tools >) “Find Ops” on page 42
- (Main Tools >) GARR Tool Library; see “Tool Library Explorers” on page 94
- (Geometry >) “GeoEdit” on page 43
- (Geometry >) “Divide Contour” on page 41
- (Solids >) “Get Draft Angle” on page 45
- (Tool Libraries >) Harvey Tool Library; see “Tool Library Explorers” on page 94
- (Tool Libraries >) Helical Solutions Tool Library; see “Tool Library Explorers” on page 94
- (HSM >) “Helix Bore” on page 46: This plug-in was retired at version 11.0, when its functionality was moved into the process dialog. For information on the **Holes** process dialog, tabs Drill and Bore, see the *Mill* guide.
- (Geometry >) “Helix Builder” on page 46
- (operation modifier or Toolpath >) “High Speed Machining (HSM)” on page 47
- (Main Tools >) “Import VNC” on page 54
- “ISCAR Tool Advisor (ITA)” on page 59
- (System >) “Machine Info” on page 59
- (Tool Libraries >) MachiningCloud Tool Library; see “ Cloud-Based Tool Libraries” on page 32
- (Toolpath >) “Mirror Ops” on page 60
- (Solids >) “ Model Associativity ” on page 60

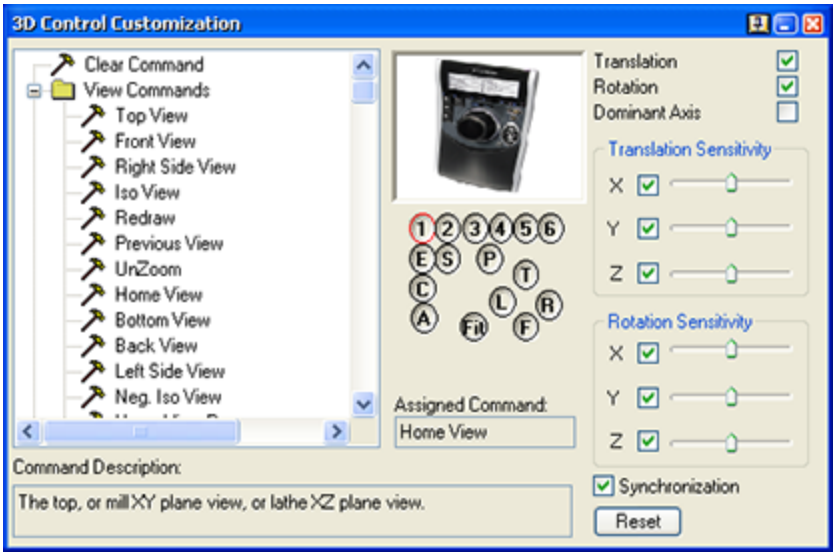
- "Offset Contour" on page 61: This plug-in was retired at version 10.5, when its functionality was moved into the process dialog. For information on the **Contour** process dialog, Contour tab, fields Number of Extra Offsets and Extra Stepover, see the *Mill* guide.
- (operation modifier) "Okuma Turn-Cut " on page 61
- (System >) "Pathfinder" on page 66
- (operation modifier) **Rapid To Feed**; see "Change Feeds and Speeds" on page 22
- "Reporter Basics" on page 67
- (Posting >) "Set Up Post Editor" on page 73: This plug-in was retired at version 12.0, when its functionality was moved into the **Preferences** dialog as a new tab: Post Editor Settings. For information, see the *Common Reference* guide.
- (Solids >) "Show Surface Normals " on page 73
- (Main Tools >) " Show Position " on page 74
- (Posting >) "Set Up Post Editor" on page 73: This plug-in was retired at version 12.0, with options incorporated into the File > Preferences dialog.
- (Solids >) "Solid Inquiry" on page 77
- (Solids >) "Solids Alignment" on page 77
- (Main Tools >) "Stock Wizard" on page 80
- (Solids >) "Surface Tools " on page 85
- (Tool Managers >) " Tool Manager iQ" on page 97
- (Tool Libraries >) ToolsUnited Tool Library; see " Cloud-Based Tool Libraries" on page 32
- (System >) "Touch Interface Manager" on page 97
- (Toolpath >) "Tp Transform / Transform Toolpath" on page 98
- (Main Tools >) "up2parts AutoCAM" on page 79
- "Z Ramp Contour / Z Step" on page 100: The Z Ramp Contour plug-in and Z Step operation modifier were retired at version 10.5, when their functionality was moved into the process dialog. For information on the **Contour** process dialog, Contour tab, Ramp Down checkbox, see the *Mill* guide.

3D Control Manager

When used with a properly installed 3D controller, the 3D Control Manager plug-in brings up a dialog that allows you to customize various aspects of the device, including the functionality of the buttons and the device's sensitivity. GibbsCAM supports the full line of 3D controllers produced by 3Dconnexion. Please consult your 3D controller's user manual and/or online help if you need more details on its operation and application.

1. Install the necessary driver for your 3D controller. A version of the driver is available on the CD (or other distribution media) that was supplied with the device, or you can check the manufacturer's website for the latest version of the driver.
2. Start GibbsCAM. After your 3D controller is properly installed, locate the 3D Control Manager... item in the Plug-ins menu. Select this item.

For example, in the **3D Control Customization** dialog pictured below (which might be different from the one you see), when the Synchronization checkbox is selected, you can assign the manufacturer-default functionality for View Commands, Modifier Keys, Device Control, and Centering Control by dragging the items on the left onto the buttons located at the center of the dialog. Click each of the commands to view a description of that command; click each of the buttons to view the command currently assigned to that button. You can choose to enable/disable the Translation, Rotation, and Dominant Axis functionality. You can also adjust the sensitivity settings for both translation and rotation. To restore the default settings, click the Reset button.



Important: If the Synchronization checkbox is selected, then GibbsCAM will interpret controller button actions according to the manufacturer-default meanings of the corresponding commands. If, instead, you want to customize the meanings of the commands (perhaps to map them to user-configurable macro-like sequences), follow these steps:

1. Deselect the Synchronization checkbox.
2. Open the controller's **Properties** dialog, **Button Configuration** tab.
3. Map one or more commands to the actions that you prefer.

Changes take effect only when the machine is restarted.

5-Axis Toolpath Conversion

Selecting the 5-Axis Toolpath Conversion operation modifier opens provides access to a subset of 5-Axis functionality, mainly parameters for Tool axis control and Gouge check.
For more information on this operation modifier, see the *5-Axis* guide.

> **Please Note:** Parts from v10.1 and v10.3 can contain operation modifiers that used an incompatible combination of choices for "Tool axis will..." in the **Tool axis control** tab. If such a part is opened in the current release, an error message will be displayed, and the settings will be made compatible.

Add G-Code

The Add G-Code plug-in was retired at version 14.0, when it was moved into mainstream GibbsCAM functionality as a utility process. For information on the **Add G-Code** utility process, see the *Multi-Task Machining (MTM)* guide.

Break Parent-Child Link

About Linked Operations (Parent/Child)

In some circumstances, such as using the TpTrans plug-in on a selected operation, one or more operations are created that are linked to the operation that was selected. Each new operation created in this way is called a *child op*, and the original from which a child op is created is called a *parent op*.

A child op can itself become a parent of other ops, each of which is a grandchild of the original op; likewise with great-[great-[...]]grandchildren.

A child op has no process of its own: its processes derive from the process (or process group) of its parent op. Therefore, deleting a parent op deletes any child/grandchild/... ops linked to it, and redoing the parent after changing its process propagates the same change to all its child/grandchild/... ops.

Op Tile Stacking. Since an op shares the same process or processes with all its children, such tile stack views as Process Mode and Process List group the parent together with all its children.

Breaking Links. To break the link from a child op to its parent, use the Break Parent-Child Link plug-in.

The purpose of the Break Parent-Child Link plug-in is to break the link and convert a child op into a fully independent operation, with its own separate process. After the link is broken, the parent can be deleted or modified without affecting its children.

> **Please Note:** If you are not sure which ops are parents to which children, change the op tile stack view to Individual (Unstacked) so that you do not inadvertently select more operations than you realize.

To break the link for a single child op:

Select the child op and, on the **Plug-Ins** menu, click Break Parent-Child Link.

Result: If possible (see below), the selected child op becomes independent. It now has a duplicate and independent copy of its parent's process parameters.

To break the link between a parent and all its child/grandchild/... ops:

Double-click the parent op, causing all of its child/grandchild/... ops to be highlighted. `Ctrl+click` the parent op so as to exclude it from the selection set. Then, on the **Plug-Ins** menu, click Break Parent-Child Link.

Result: If possible (see below), all selected child/grandchild/... ops become independent. Each has a duplicate independent copy of its parent's process parameters.

To break the link between a parent and some of its child/grandchild/... ops:

Double-click the parent op, causing all of its child/grandchild/... ops to be highlighted. `Ctrl+click` the parent op so as to exclude it from the selection set. `Ctrl+click` each

child/grandchild/... op whose linkage you want to keep, excluding it from the selection set. Then, on the **Plug-Ins** menu, click Break Parent-Child Link.
Result: If possible (see below), all selected child/grandchild/... ops become independent. Each has a duplicate independent copy of its parent's process parameters.

Links that are not broken

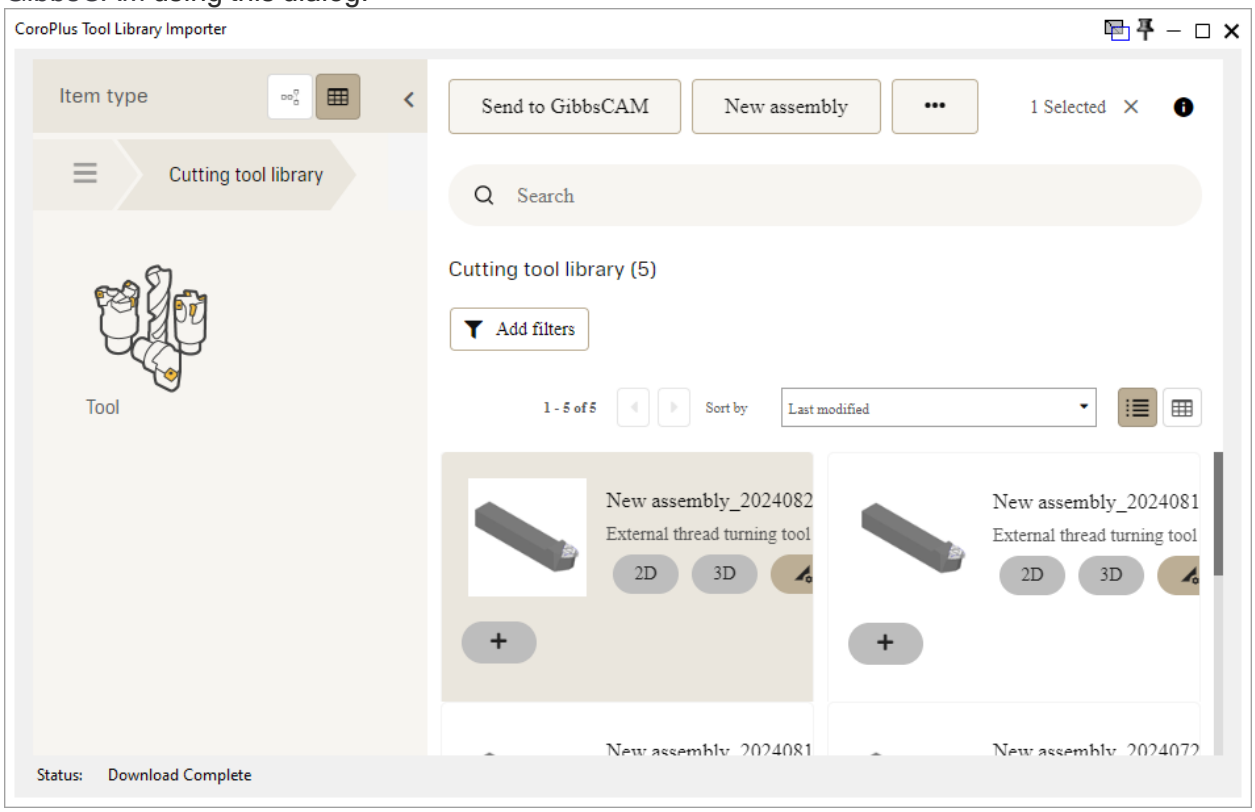
In two situations, the link from child to parent is not broken by the plug-in:

- If a single process created multiple operations (for example, two or more shapes were picked and each spawned an operation), and if such an operation is a parent, then the Break Parent-Child Link does not break links to child/grandchild/... ops this parent might have.
- If an operation was created from a process group that contains multiple processes, and if such an operation is a parent, then the Break Parent-Child Link does not break links from child/grandchild/... ops that this parent might have.

If an ineligible child op is part of a selection set, the plug-in keeps parent-child links that it cannot break and processes all links that it can break.

CoroPlus Tool Manager

Allows you to interface with the Sandvik Coromant CoroPlus Tool Library and to import tools and toolholders from assemblies that have been exported from CoroPlus to GibbsCAM. This requires an account with Coromant.
Typically, a tool or tool assembly is created in Coromant and exported to GTC format. This makes it available to GibbsCAM via the plug-in, where you select the item and send it to GibbsCAM using this dialog:



What Is CoroPlus?

GibbsCAM CoroPlus option provides access to CoroPlus ToolLibrary, Sandvik Coromant's system for managing libraries of cutting tools, toolholders, toolblocks, and the like, which complies with the ISO 13399 standard for tool exchange. For complete information, see the book *CoroPlus Tool Library (CPTL) Plug-In*.

APT-CL Post

A postprocessor for generating APT-CL from GibbsCAM toolpath is one of several available postprocessors. This topic provides instructions for installing, configuring, and running GibbsCAM APT-CL Post.

Names and Locations of Files needed by APT-CL Post

The two `*.dll` files required by GibbsCAM APT-CL must reside in specific places. (You can use Pathfinder to learn the location of the Plug-ins Folder.) For example, if you have not changed the defaults:

- `C:\Program Files*\Gibbs\GibbsCAM\<version>\Plugins\Posting\APTCL.dll`
- `C:\Program Files*\Gibbs\GibbsCAM\<version>\Modules\CLGen.dll`

If APT-CL configuration files (`*.cfg` files) are used, they are global data, and must therefore reside in the `APTCL` subfolder of your Plug-ins **Data** Folder. For example:

- `C:\ProgramData\Gibbs\GibbsCAM\<version>\Plugins\Data\APTCL\APTCL*.cfg`

User Interface

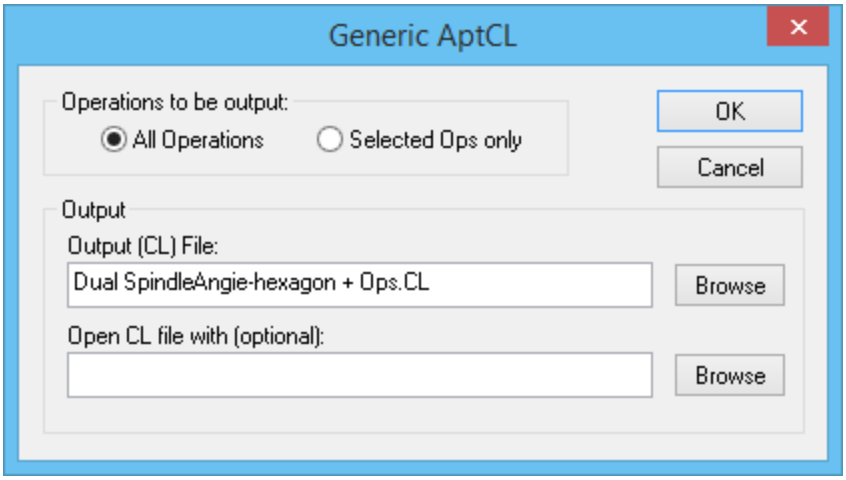
When GibbsCAM starts, it looks for `*.cfg` files under the Plug-ins Data Folder, typically `C:\ProgramData\...\Plugins\Data\APTCL`.

- If no configuration file is found, or only one file, then Plug-Ins menu item APT-CL Post will contain no sub-menu items.
- If more than one configuration file is found, then Plug-Ins menu item APT-CL Post will contain sub-items corresponding to the configuration files.

Creating APT-CL Output Files

When a part has been programmed with machining operations, you can output to an APT-CL file instead of the standard Post dialog using these steps.

1. On the GibbsCAM main menu, click Plug-Ins > Posting > APT-CL Post.
2. For all APT-CL except Mori: Supply values in the following dialog box. When you are done, click OK.



Operations to be output

To output all operations, choose All Operations. To output only the operations you were selected before you ran the plug-in, choose Selected Ops only.

Output (CL) File:

The output filename defaults to the filename and directory of the current part, adding a `.CL` extension. To view or change the output filename or folder, click Browse.

Open CL file with (optional)

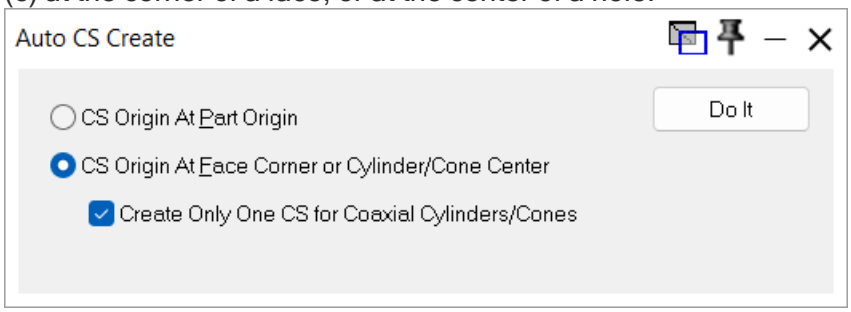
If this field is left blank, the `.CL` file is opened using the standard text editor (typically Notepad). If you want to specify a different editor, supply its path and filename, or click Browse to find and select the editor.

Notes

- Mori APT-CL provides no dialog for setting preferences: All operations will be output, and `.CL` filenames will be generated automatically.

Auto CS Create

The Auto CS Create plug-in automatically generates coordinate systems for a selected body. Simply select a solid face or faces and choose the Auto CS Create item from the Plug-ins menu. Depending on the method selected, coordinate systems will be defined with their origin ...
(a) at the part origin; or
(b) at the corner of a face, or at the center of cylinder or cone; or
(c) at the corner of a face, or at the center of a hole.



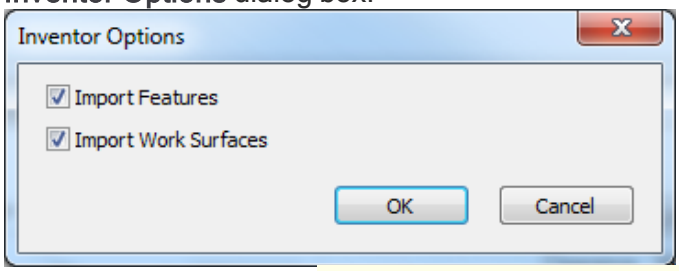
All coordinate systems are created relative to CS1. If a body lies in a plane other than CS1, the new planes will be created as if the body had undergone Change CS (HVD). This means the planes could lie well off the part. To avoid this, be sure the body is in CS1, using Change CS (XYZ) if needed.

Autodesk Inventor Options

Allows you to set preferences for importing from Autodesk Inventor: features only, work surfaces only, both, or neither.
In Autodesk Inventor, work surfaces are usually derived components. The term "features" covers many items (see below). Because work surfaces and features both consume a lot of disk space, it can sometimes be advantageous to omit one or both.

What Are Autodesk Inventor Features?

Autodesk Inventor defines several kinds of features: holes, extrusions, fillets, chamfers, and so forth. Features in Autodesk Inventor part files (`*.ipt`) and assembly files (`*.iam`) can be understood by GibbsCAM 2027, but only if they were saved while the GibbsCAM CAD Transfer Add-In for Autodesk Inventor was installed on the Autodesk workstation.
Such feature data can sometimes be quite large, and it is not always needed. In GibbsCAM, you can optionally set a switch that suppresses the importation of such feature data, using the **Inventor Options** dialog box:



To open this dialog box: Plug-Ins > (Misc) > Autodesk Inventor Options

PLM > Vault

This plug-in lets you access the server where your Autodesk Vault items are managed. For information on minimum requirements for obtaining and using the Vault client (included with Inventor and some other Autodesk products), refer to the Autodesk documentation.
If you have a Vault client installed, enabling the Autodesk Vault plug-in adds a new item to the **Plug-Ins** menu: PDM.
By default, in a new installation, Autodesk Vault is not enabled. To enable it: (1) Plug-Ins > Plug-In Manager; (2) In Plug-In Manager, under PDM, select the checkbox for Autodesk Vault; (3) Click Apply. *Result* (if your machine has a Vault client installed): When GibbsCAM is next re-started, a new Plug-Ins menu item, PDM, provides access to Autodesk Vault.
Settings: You can specify which GibbsCAM actions will trigger an automatic update check: On File > Import, when attempting a Do It or Redo, and/or when attempting to post.
Check for Updates: Persistent connection parameters allow you to have GibbsCAM find all Vault-monitored parts, report on their modification status, and re-import as needed.
Browse Vault: Using an explorer-like interface similar to Autodesk's Vault browser, you can navigate folders and subfolders, preview parts, query part properties, and import.
Disconnect from Server: When you do not need a persistent connection to Vault, you can disconnect from the server.

Servers for Files and for Database

By default, the same machine serves both the Vault database and files. If, instead, you prefer for these functions to be distributed to two separate machines, follow these steps after enabling the Autodesk Vault plug-in:

1. Navigate to the user plug-in data folder – typically `C:\Users\<username>\AppData\Roaming\Gibbs\GibbsCAM\<version>`**\Plugins\data**

> You can open this using GibbsCAM: Plug-Ins > Misc > Pathfinder > **User Plugin Data Folder**.

2. If necessary, close GibbsCAM.
3. Use a text editor to open the file *<path_to_user_plug-in_data>***\VaultData.xml** and, in the following line –
   `<file_server>localhost</file_server>`
   –, change the word `localhost` to the hostname of your file server.
4. Save your changes to **VaultData.xml** and close the file.

Bottom Up Hit Flats

This plug-in was retired at version 11.0, when its functionality was moved into the process dialogs: in the **Contour** process dialog, **Contour** page, or in the **Roughing** process dialog, **Pocket** page, within the Z Step area: Hit flats after each area.
For information on the Hit Flats options for contouring and pocketing, see the *Mill* guide.

Change Backup File Location

Allows the user to specify the backup file directory that backups are saved to if AutoSave is set to Enabled in the File Preferences settings.

Change Feeds and Speeds

The Change Feeds and Speeds plug-in can be used with any type or combination of operations. This plug-in is used to modify the feedrate and/or spindle speed associated with one or more of the selected operations. There are five tabs, Operation Selection - Action is the first. This tab is where operations and actions are selected. The system reads the dialog starting at the first tab at the top left and reads down each column moving through each tab in order. Therefore, it is recommended that you make your change selections in the same manner. For a detailed description of each tab, see "Operation Selection-Action" on page 23, "Milling Spindle Speed" on page 24, "Turning Spindle Speed" on page 25, "Milling FeedRate" on page 26, and "Turning Feedrate" on page 28.If you want to persist the settings and effects of this functionality (for example, preserving it through Redo or Redo Ops), you can apply it as an operation modifier instead of using the plug-in. For information on using operation modifiers, see the *Mill* and *Turning* guides.

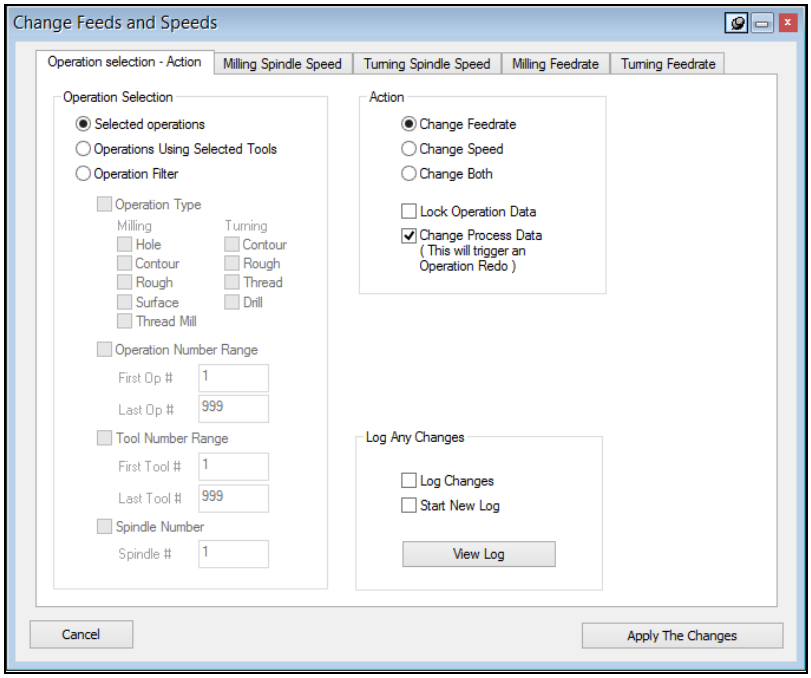
Rapid to Feed

Rapid to Feed is an operation modifier for both milling and turning operations. For the selected operation, it changes the feedrate for rapid moves to a value you specify for Fast Feedrate.

To Use Rapid to Feed

1. Right-click the operation whose rapid feedrate you want to change and, in the context menu, choose Operation Modifiers.
2. In the Manage Modifiers dialog box: Double-click Rapid to Feed.
3. In the Convert Rapid Moves to Fast Feed Moves dialog box: Enter a value for Fast Feedrate and then close the dialog.
4. Back in the Manage Modifiers dialog box: Click Do It.

Operation Selection-Action



For spindle speed and/or feedrate modifications there are a number of methods for selecting the operation(s) and the action(s) to be taken. There are three primary sections: Operation Selection, Action and Log Any Changes. An example is shown here. We began with our Operation Selection, we selected Operation Filter. Then we selected Hole, Contour and Rough for our Operation Type. Next we set our Operation Number range from 1 to 37 and our Tool Number Range from 1 to 16. We used Spindle #1. The Actions we chose will be to Change Speed and Change Feedrate, and to also change the Process Data. Our last selection was to Log Any Changes.

Operation Selection

Selected Operations

Modify operations that are currently selected.

Operations Using Selected Tools
Modify only the operation(s) associated with any of the currently selected tools.
Operation Filter
Use one or more filter to select operations. The filters are used separately, so if an operation matches any one of the filters, it is selected, it does not have to meet all the filter criteria.

Operation Type
Allows you to control what type of operations will be affected by the changes to feeds and speeds.

Operation Number Range
Allows you to filter for operations within a specified number range.

Tool Number Range
Allows you to filter for operations associated with any tool within a specified tool number range.

Spindle Number
Filters for any operation(s) machined on a specific spindle.

Action

The actions that can be taken to change feedrate and/or spindle speed for all selected operations. One or both of the items in this section must be selected to enable the feed and /or speed changes on the following tabs. In addition, the new feedrate/speed values can be locked at the operation level and the values for the associated processes can also be updated.
Change Speed
This option must be selected to activate Spindle Speed changes for the selected operations.
Change Feedrate
This option must be selected to activate Feedrate changes for the selected operations. Change the feedrate for all of the selected operations.
Lock Operation Data
New feedrate and/or speed values can be locked after changes are made.
Change Process Data
Changes the Process Data along with the Operation Data for feedrates and speed on selected operations.

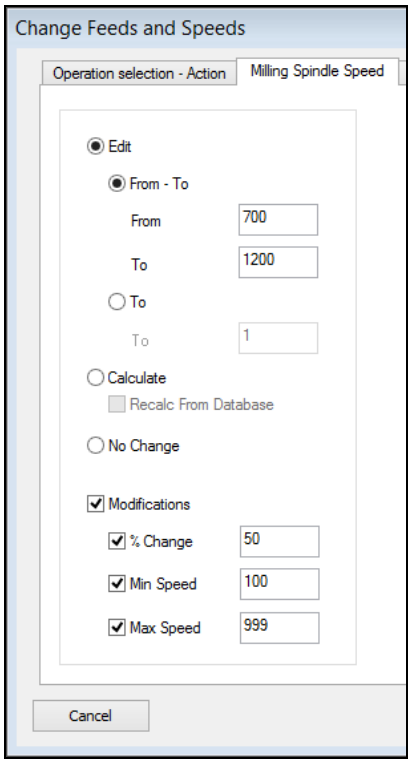
Log Any Changes

A log file is available to view all the changes made.
Log Changes
Creates a log file of speed and/or feed changes.
Start New Log
Create a new log file each time the plug-in is used, otherwise the log data is appended to the current log file.
View Log
The log file can be viewed from within the plug-in by using this button.

Milling Spindle Speed

The Spindle Speed for milling operations (and the associated processes) can be modified by first selecting Change Speed from the Operation Selection tab where there are a number of spindle speed options as described below. The system reads the dialog starting at the top left and reads

down each column adjusting the feedrate in that order. Therefore, it is recommended that you make your feedrate change selections in the same manner. Please note that while actual values are changed in the process, the Change Feeds and Speeds Plug-in does not differentiate between RPM and surface speed.
An example is shown below. We started by choosing Milling Operations, then we went to the Milling Spindle Speed Tab and made an Edit to our spindle speed from 700 to 1200. Lastly we made Modifications and set our Min Speed to 100 and our Max Speed to 999.



Edit
With this option you can change all speeds of a given value to another given value (From - To option) or change all speeds to a given value (To option).

Calculate
Recalculate the spindle speed based on the new spindle speed changes.

Recalc From Database
Recalculate the spindle speed based on the tool associated with the operation and the current material database.

No change
No change to the spindle speed.

Modifications
After changes have been applied, (including feedrate utility markers), changes can be further modified by checking Modifications and applying a Percentage (%) Change, setting a Min Speed and / or setting a Maximum Speed.

% Change
This option can change all speeds to the specified percentage of their current value. In the example above, all speeds that were 700 RPM are increased to 1200 RPM; then, under Modifications, we increased the current value by 50%. Our spindle speed of 1200 RPM has now been increased to 1800 RPM.

Min. speed
Sets any speed that is less than the specified value to this minimum value.

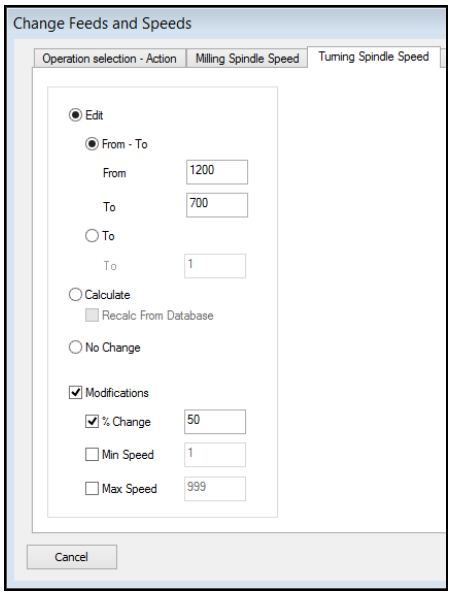
Max speed
Sets any speed that is greater than the specified value to this maximum value.

Turning Spindle Speed

The Spindle Speed for turning operations (and the associated processes) can be modified by first selecting Change Speed from the Operation Selection tab where there are a number of spindle speed options as described below. The system reads the dialog starting at the top left and reads down each column adjusting the feedrate in that order. Therefore, it is recommended that you make your feedrate change selections in the same manner. Please note that while actual values are changed in the process, the Change Feeds and Speeds Plug-in does not differentiate between RPM and surface speed.

Edit

With this option you can change all speeds of a given value to another given value (From - To option) or change all speeds to a given value (To option).



Calculate

Recalculate the spindle speed based on the new spindle speed changes.

Recalc From Database

Recalculate the spindle speed based on the tool associated with the operation and the current material database.

No change

No change to the spindle speed.

Modifications

After changes have been applied, (including feedrate utility markers), changes can be further modified by checking Modifications and applying a Percentage (%) Change, setting a Min Speed and / or setting a Maximum Speed.

% Change

This option can change all speeds to the specified percentage of their current value. In the example above, all speeds that were 700 RPM are increased to 1200 RPM; then, under Modifications, we increased the current value by 50%. Our spindle speed of 1200 RPM has now been increased to 1800 RPM.

Min. speed

Sets any speed that is less than the specified value to this minimum value.

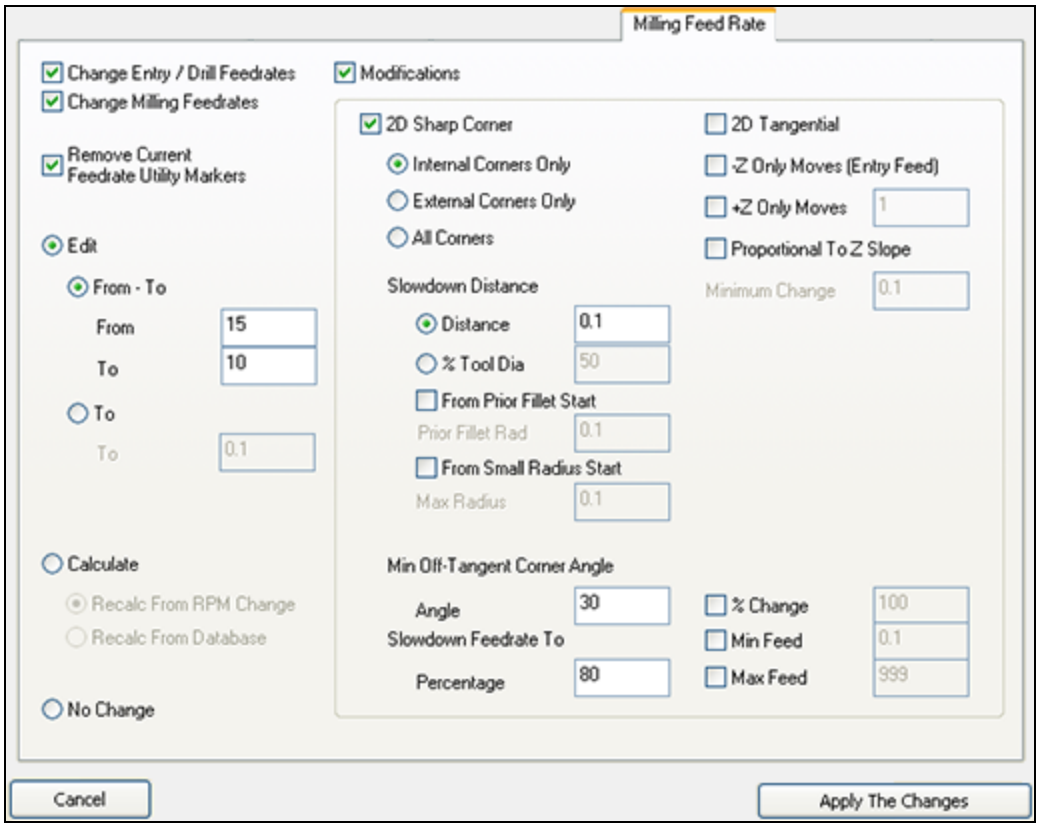
Max speed

Sets any speed that is greater than the specified value to this maximum value.

Milling FeedRate

This is where changes and / or modifications are made to the feedrate for Milling operations. The toolpath feedrates can be modified to slow down into an internal or an external corner and speed back up upon exiting the same corner. The distance before and/or after the corner can be a fixed value or a percentage of the current tool diameter. The system reads the dialog starting at the top left and reads down each column adjusting the feedrate in that order. Therefore, it is recommended that you make your feedrate change selections in the same manner. The Modifications checkbox provides for special situations.

An example is shown below. We started at Milling Operations, from there we selected to change both Entry / Drill Feedrates and Milling Feedrates. Then we removed our Current Feedrate Utility Markers. Next we made an Edit to our feedrate of 15 down to 10. Next we made Modifications to our Internal Sharp Corners Only, setting our Slowdown Distance to 0.1 and a Min Off-Tangent angle of 30°. Our last selection was to slow our feedrate at internal corners by 20%.



Change Entry / Drill Feedrates

Change Entry and / or Drill Feedrates.

Change Milling Feedrates:

Change Milling Feedrates.

Remove Current Feedrate Utility Markers

Removes all existing utility markers before performing any feedrate modifications.

Edit

Change all feedrates of one given value to another given value (from - to option) or changes all feedrates to a given value (to option).

Calculate

Recalculate the feedrate to adjust for any new spindle speeds.

Recalc From RPM Change

Recalculate the feedrate to adjust for any new spindle RPM.

Recalc From Database

Recalculate the feedrate based on the tool associated with the operation and the current material database.

No change

No change to the feedrate.

Modifications

2D Sharp Corner

The toolpath feedrates can be modified to slow down into the sharp corners and speed up upon exiting the same corner.

Slowdown Distance

To effect this slowdown a feedrate utility marker will be placed at the specified Slowdown Distance prior to a corner or at a given % of the Tool Dia. prior to a corner. The existing feedrate will be restored at the same distance upon exiting the corner.

From Prior Fillet Start

Allows you to specify the radius of a previously used tool to calculate where a blend radius would start and/or end on the two features that comprise each sharp corner. The slowdown distances will then be applied before and after the calculated blend rather than at the actual sharp corner.

From Small Radius Start

By using this option corners that already contain blends in the geometry can be treated as sharp corners. Any blend that does not exceed the given value will be treated as a sharp corner for slowdown purposes. The slowdown distances will be applied before and after the actual blend radius.

Min Off-Tangent Corner Angle

Takes the minimum difference between the angle at the end of one feature and the angle at the start of the next feature before a feedrate adjustment is made.

Slow Feedrate To

The current feedrate will be multiplied by the specified percentage for the slowdown adjustment.

2D Tangential

Modifies the toolpath feedrates by increasing the feedrate moves around the inside of an arc and slowing down the feedrate moves around the outside of an arc. The toolpath feedrate can then maintain a constant feedrate at the edge of the tool diameter around an arc.

-Z Only Moves (Entry Feed)

All moves in the Z minus direction (no XY moves) will be set to the entry feedrate.

+Z Only Moves

All moves in the Z plus direction (no XY moves) will be set to the specified value

Proportional To Z Slope

Sets the feedrate for moves that are in both Z and either X, Y, or both X and Y (XZ, YZ and XYZ). The feedrate will be calculated using the contour and the entry feedrates and is in proportion to the components movement in the Z and XY plane. The more movement in Z, the closer the feedrate will be to the entry feedrate, and the more movement in XY the closer the feedrate will be to the contour feedrate. For example if you have an entry feedrate of 10 and a contour feedrate of 20, a ramp down move that moved 4 inches in X while also moving 1 inch in Z could be cut faster than 10, but not as fast as 20. Since most of the movement is in the X direction the feedrate will be closer to 20 than 10.

% of Change

Modifies the feedrate by the specified percentage. 100% will leave the feedrate unchanged.

Min Feed

Sets any speed that is less than the specified value to this minimum speed.

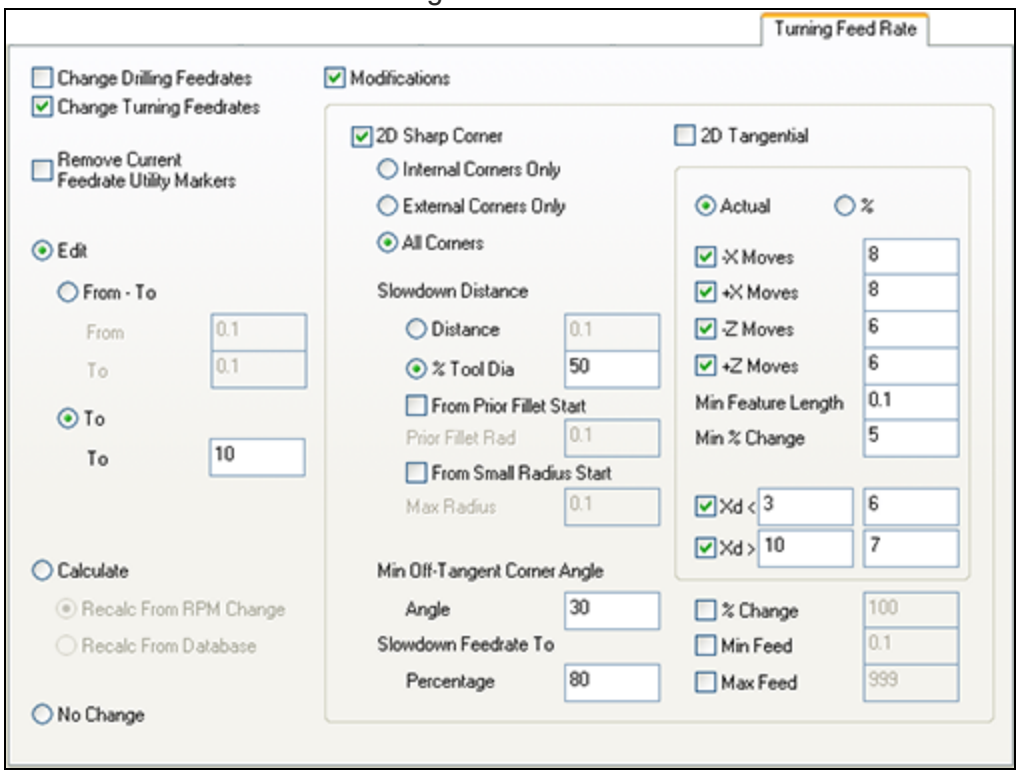
Max Feed

Sets any speed that is greater than the specified value to this maximum speed value.

Turning Feedrate

This is where changes and/or modifications are made to the feedrate for Turning operations.The toolpath feedrates can be modified to slow down into an internal or an external corner and speed back up upon exiting the same corner. The distance before and/or after the corner can be a fixed

value or a percentage of the current tool's diameter. The system reads the dialog starting at the top left and reads down each column adjusting the feedrate in that order. Therefore, it is recommended that you make your feedrate change selections in the same manner. The Modifications checkbox provdes for special situations.
An example is shown below. We started by selecting Turning Operations, next we selected to change the Turning Feedrates. We left our current feedrate utility markers in place, then we made an edit to change our feedrate to 10. Then we modified our 2D sharp corners, setting a slowdown distance at 50% of our tool's diameter and a minimum off-tangent corner angle of 30°. We also slowed our feedrate on internal corners by 20%. We specified Actual feedrates for all moves in -X, +X, -Z and +Z. Next we added a Min Feature Length and Min % change. For our last selection we set our X values and stated that if the value is greater than 10 (the value on the left) it will be changed to 7 (the value on the right) then we went through that process again but stating if a value is less than 3 it will be changed to 6.



Change Drilling Feedrates
Change the drilling feedrates.
Change Turning Feedrates:
Change the turning feedrates.
Remove Current Feedrate Utility Markers
Any existing utility markers will be removed before any feedrate modifications.
Edit
Change feedrate of a given value to another given value (From - To option) or change all feedrates to a specified value (To option).
Calculate
Recalculate to adjust for any new spindle RPM changes or recalculate the feedrate based on the tool associated with the operation using the CutDATA Material Database.
Recalc From RPM Change
Recalculate feedrate to compensate for any current RPM change.

Recalc From Database
Recalculate the feedrate based on the current material and tools associated with the operation using the CutDATA Material Database.

No change
No changes to the feedrate.

Modifications

2D Sharp Corner
Modify the toolpath's feedrate to slowdown into internal corners and speed up upon exiting the same corners. The distance before and after the corner can be a fixed value or a percentage of the current tool diameter.

Internal Corner Only
Feedrates will be modified at `Internal Corners Only`.

External Corners Only
Feedrates will be modified at `External Corners Only`.

All Corners
Feedrates will be modified at `All Corners`.

Slowdown Distance
To effect a slowdown the feedrate utility marker will be placed at the specified `Slowdown Distance` prior to the corner or a given `% of the Tool Diameter`. The prior feedrate will be restored at the same distance upon exiting the corner.

From Prior Fillet start
Allows you to specify the radius of the previously used tool and calculate where a blend radius would start and / or end on the two features that comprise each sharp corner. The slowdown distances will then be applied before and after this calculated blend rather than at the actual sharp corner.

From Small Radius Start
Corners that already contain blends in the geometry can be treated as sharp corners by using the `From Small Radius Start` option. Any blend that does not exceed the given value will be treated as a sharp corner for slowdown purposes and the slowdown distances will be applied before and after the actual blend radius.

Min Off-Tangent Corner Angle
The input amount will be the minimum difference between the angle at the end of one feature and the angle at the start of the next feature before a feedrate adjustment will be made.

Slow Feedrate To
The current feedrate will be multiplied by the input percentage for a slowdown adjustment. 100% will leave the feedrate unchanged. Therefore, if your current feedrate is 2000 and you input 50% your feedrate will be slowed to 1000.

2D Tangential
Modify the toolpath feedrates by increasing the feedrate moves around the inside of an arc and slowing down the feedrate moves around the outside of an arc. The toolpath feedrate can then maintain a constant feedrate at the edge of the tool diameter around an arc.

Actual or %
The specified value is the `Actual` Contour feedrate or a `Percentage` to be applied to the Contour feedrate. This option will adjust the Contour feedrates for all moves listed below in `-X`, `+X`, `-Z` or `+Z` to either an actual feedrate or a percentage of the current Contour feedrate.

Min Feature Length
This amount will be the minimum length of a feature that will be acceptable for a feedrate change. This option becomes available only when a X or Z axis move is selected.

Min % Change
Input amount will be the minimum percent for feedrate changes.This option becomes available only when a X or Z axis move is selected.

X<
All X values that are greater than the first input amount will be set to the second input amount.

X>
All X values that amount to less than the first input amount will be set to the second input amount.

% of Change
the Feedrate will be modified by the specified percentage. 100% will leave the feedrate unchanged.

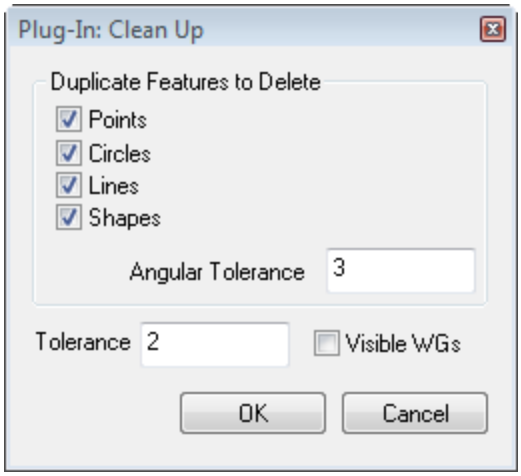
Min Feed
Sets any feedrate that is less than the specified value to this minimum value.

Max Feed
Sets any feedrate that is greater than the specified value to this maximum feedrate.

Change Hole Orientation

This plug-in was retired at version 11.0, when its functionality was moved into the Hole List context menu.
For information on Change Hole Orientation, see the *Features* guide.

CleanUp



This feature will delete duplicate points, circles, and lines within a given tolerance. CleanUp will also delete duplicate shapes (including splines) given that the shape is terminated at both ends (if it is not open). The user simply selects which feature type (or types) to delete, sets the tolerance and clicks the OK button. The system will keep the feature with the lowest number. Once CleanUp is finished, the user will be informed of the results.

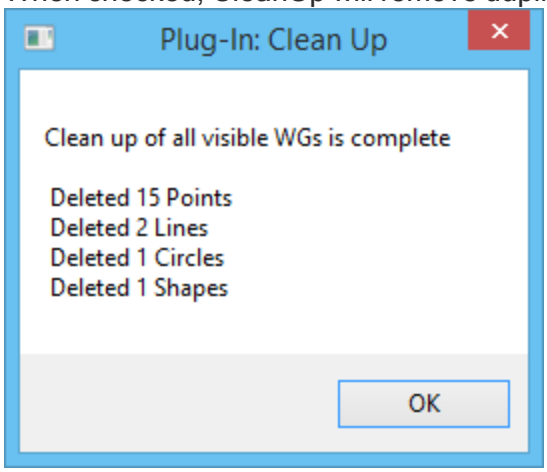
Angular Tolerance
Available when the Lines selection is checked. This is the allowable deviation (measured in degrees) between two or more lines. Lines that fall within the number of degrees entered in the Angular Tolerance field will be considered duplicates of one another.

Tolerance

Maximum allowable deviation between two or more features. Measured in either inches or millimeters, depending on which was specified in the Document Control dialog. Features that fall within the number of inches/millimeters entered in the Tolerance field will be considered duplicates of one another. This will apply to both the feature size and location.
For example, if a 10mm circle is created at X0Y0, another 10mm circle at X1Y0, then an 11mm circle at X0Y0, only the circle at X0Y0 will remain if the tolerance is anything greater than 1mm.

Visible Workgroups

When checked, CleanUp will remove duplicate items from all visible workgroups.



Cloud-Based Tool Libraries

The following three plug-ins require an internet connection. They allow you to import tools from cloud-based third-party tool libraries.

CoroPlus Tool Library:
See the book *CoroPlus Tool Library (CPTL) Plug-In*.

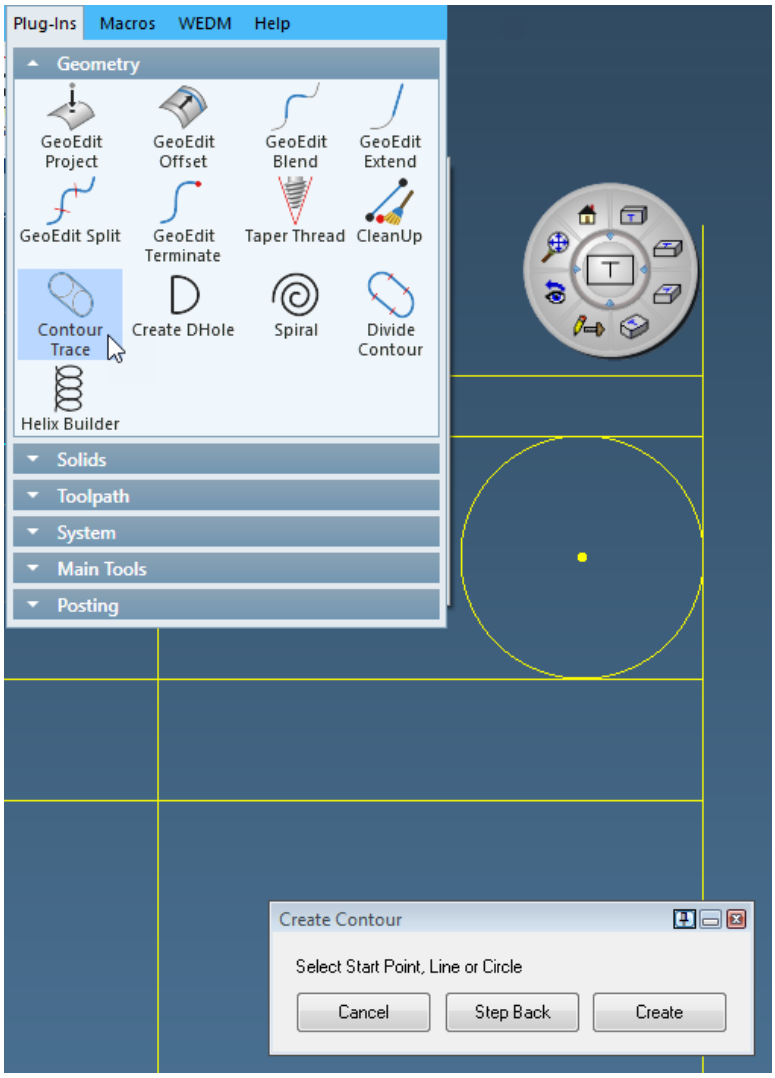
ToolsUnited:
https://www.toolsunited.com/

MachiningCloud:
https://www.machiningcloud.com/

All three use a similar approach, requiring you first to log in to online account with the third party.

Contour Trace

The plug-in on the Geometry menu, Contour Trace, lets you create a contour from a chain of lines and circles.



(`Click`) a series of lines/arcs. As geometry becomes connected (a line/arc element at both ends), it turns into a thick red line/arc. (`Click`) the Step back option in the Create Contour dialog to reverse as many selections as necessary.
When the contour is complete, click Create. You will be prompted: Delete construction geometry? Yes/No. Yes deletes the geometry associated with the current contour. This will also work with open shapes: simply select the points at the start and end of the shape.
The location of the first point selected becomes significant if a circle and closed shape are picked, as this will determine the direction of the contour.

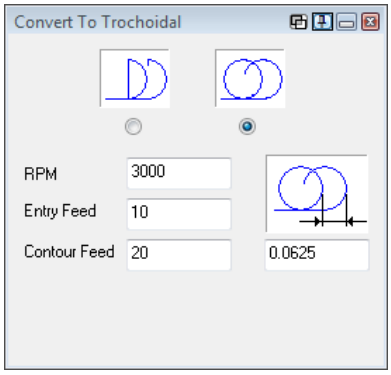
Convert to Trochoidal

This is useful for milling brittle materials, such as glass or granite, or extremely hard materials that generate a lot of heat upon cutting. It converts an existing Roughing (pocketing) or Contour operation into a new operation, generating circular toolpath at high feedrate with low load on the tool. The lower load diminishes heat and prolongs tool life.
Trochoidal toolpath loops back on itself at a specified offset, so as to maximize circular motion while keeping linear motion to a minimum; therefore, a smaller tool diameter is required. A tool

whose diameter is half the diameter of the existing tool will create trochoidal toolpath whose circular motion is centered along the old toolpath that it replaces.
If you want to persist the settings and effects of this functionality (for example, preserving it through Redo or Redo Ops), you can apply it as an operation modifier instead of using the plug-in. For information on using operation modifiers, see the *Mill* and *Turning* guides.

Steps

1. Select the operation you want to convert: a mill Roughing (pocketing) or Contour operation.
2. Select the new tool you want to use. Its diameter must be less than the diameter of the operation's current tool.
3. Right-click the operation, choose Operation Modifiers, and then add Convert To Trochoidal.
4. In the **Convert To Trochoidal** dialog box:



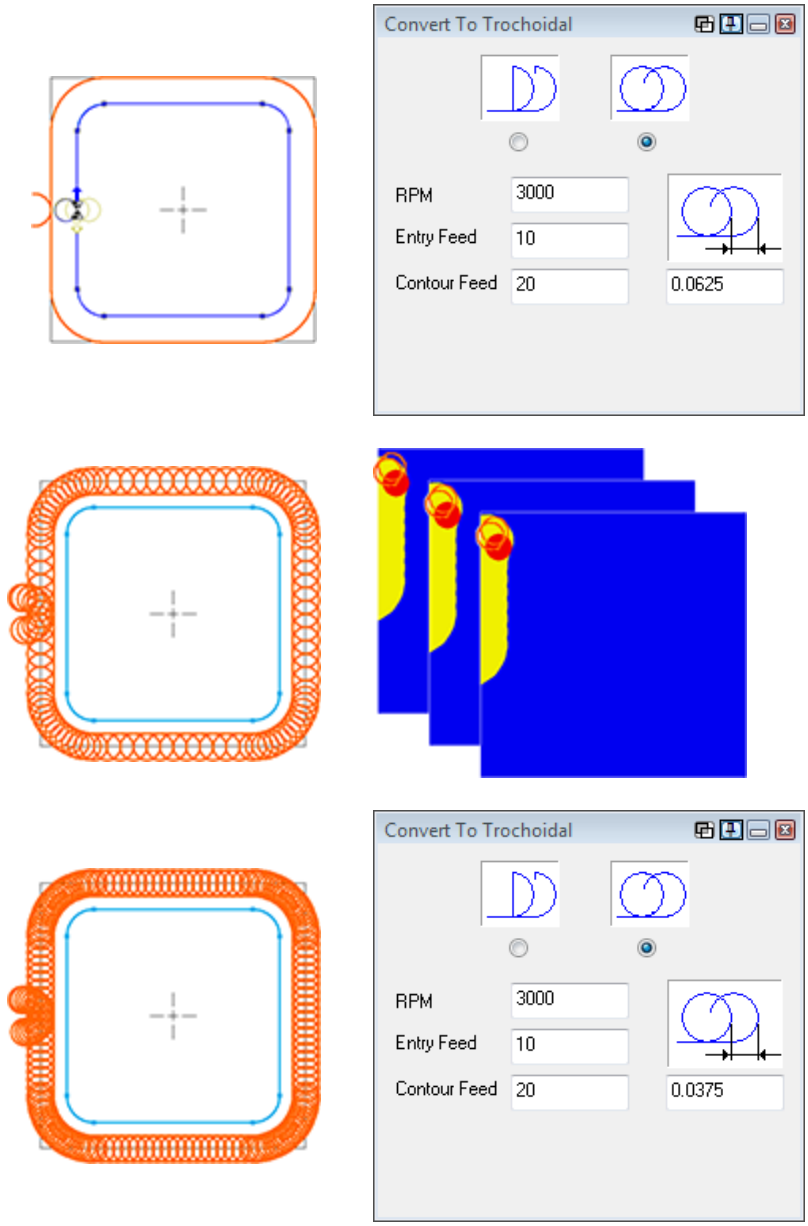
   - Choose the type of rotation. Full-circle motion is ideal for high speed machining, but half-circle can be better for traditional cutting at lower speeds, because of less motion.
   - If necessary, modify the existing values for RPM, Entry Feed, and Contour Feed (inherited from the previous operation) and supply a value for offset between rotations.
     Note: Although an offset greater than the tool diameter may be accepted, we recommend using a value less than or equal to the tool radius, for a cleaner finish and less linear motion.
5. Click Do It.

Result:

- If Convert to Trochoidal was used as a *plug-in*, then the old operation is replaced and the process is removed; no further changes can be made to the new operation. (However, a Redo or Redo Ops owill restore the previous operation and the old process.)
- If Convert to Trochoidal was used as an *operation modifier*, then the old operation is marked with a plus symbol and the process is modified. The changed operation and process will persist through further modifications, if they occur.

Examples

This series of images illustrates the effect of Convert to Trochoidal, starting with an existing Contouring toolpath and using a tool half the diameter of the tool used in the original operation. In the Convert to Trochoidal dialog, the offset is designated. The toolpath is created and rendered. Note how wide a path the tool is able to cut while not being stressed. Also note that the trochoidal toolpath's centerline follows the original contour's toolpath.



Two examples of Trochoidal toolpath

Create D-Hole

This feature creates single- or double-sided DHole geometry. The geometry will be created at the coordinates specified in the dialog at a depth of zero.

Shape

Tells the system to create either a DHole or a double DHole.

X Coordinate

Horizontal coordinate of the origin.

Y Coordinate

Vertical coordinate of the origin.

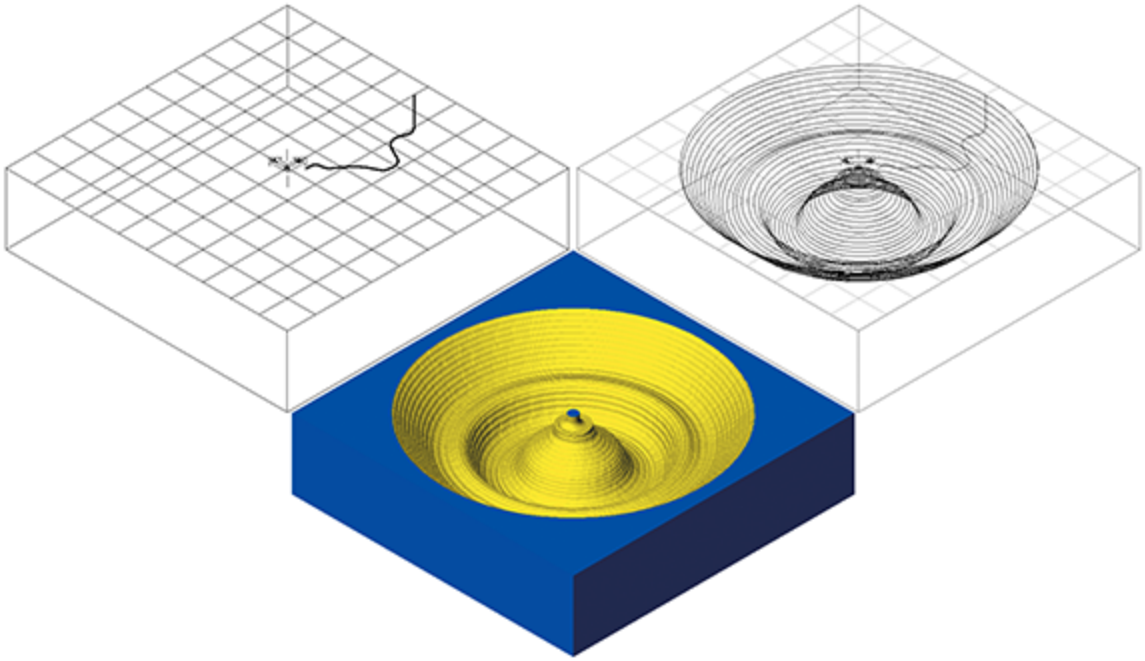
Diameter
Diameter of the DHole.
Angle to Flat
Degree of rotation from the horizontal axis to the flat side of the DHole.
Height
Distance from the flat to the arc on a DHole or the distance between flats on a double DHole.

Create Plunge Rough Process

This plug-in was retired at version 9.5, when its functionality was moved into the CAM palette. For information on the **Plunge Rough** process, see the *SolidSurfacer* guide.

Spiral

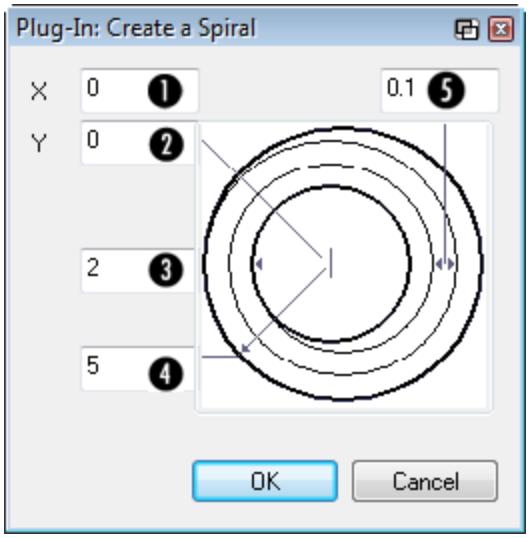
This feature creates spiral geometry which may then be used to generate spiral cuts. The geometry will be created at the coordinate specified in the dialog at a depth of zero.



An example of a spiral created from geometry

There are two distinct modes for creating a spiral, depending on whether or not geometry is selected when the plug-in is launched.

Plug-In options with no geometry selected

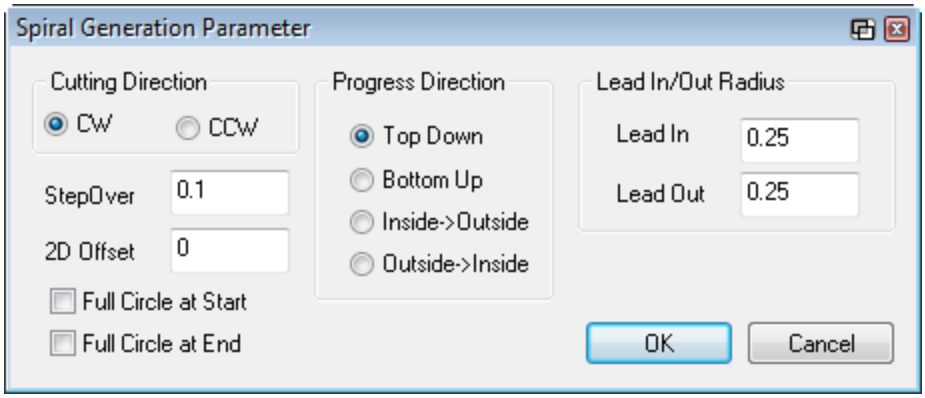


1. X Coordinate
2. Y Coordinate
3. Inner Radius
4. Outer Radius
5. Offset

X Coordinate
Horizontal coordinate of the origin.
Y Coordinate
Vertical coordinate of the origin.
Outer Radius
Radius of the outer edge of the spiral or the starting point.
Inner Radius
Radius of the inner edge of the spiral or the ending point.
Offset
Gap between revolutions of the spiral.

Plug-in options with geometry selected

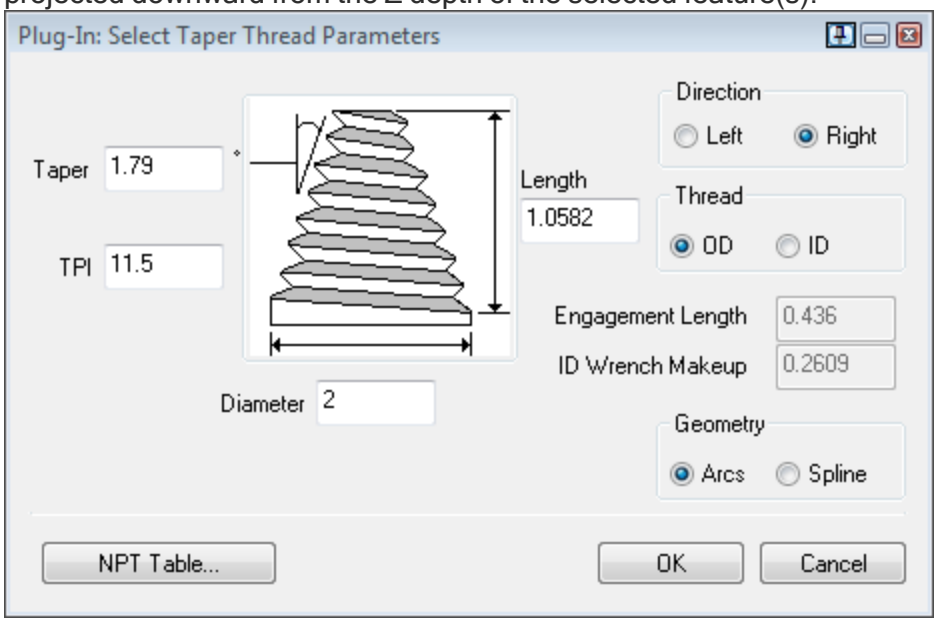
The `Spiral` Plug-in can also generate a Z-axis revolved spiral around `X0Y0` when the plug-in is activated with selected geometry. The geometry can be an arc, line, or spline and must be created in the XY plane. The system will generate a spiral with the profile of the selected geometry; it can then be machined with a Contour process. Note that the geometry should not cross `X0` (or the Y axis), as the shape will be revolved.



Cutting Direction
Direction of the spiral, either clockwise or counterclockwise.
StepOver
Length of the gap between revolutions of the spiral.
2D Offset
Additional distance at which the geometry will be generated. Both positive and negative values are allowed.
Full Circle at Start
A generated spiral will include a complete circle at its start point. The circle is flat and perpendicular to the depth axis. This creates a start loop on the spiral geometry.
Full Circle at End
A generated spiral will include a completed circle at its end point. The circle is flat and perpendicular to the depth axis. This creates a finish loop on the spiral geometry.
Progress Direction
Direction in which the spiral is created. Use Top Down or Bottom Up for geometry whose height (or Y values) changes. Top Down will generate the spiral beginning at the highest Y value while Bottom Up starts at the lowest Y value. Use Inside->Outside or Outside->Inside for flat geometry (such as a horizontal line). Inside->Outside will generate the spiral beginning at the lowest X value while Outside->Inside will start at the highest X value.
Lead In/Out Radius
Creates an arc of the specified radius value to function as a Lead In or Lead Out when machining the geometry.

Taper Thread

This feature creates the geometry for a tapered thread. The geometry will be created at the coordinates specified in the dialog. Prior to selecting Taper Thread, one or more points and/or circles must be selected as the centerpoint of each tapered thread. The tapered thread will be projected downward from the Z depth of the selected feature(s).



Outer Diameter
The largest diameter of the tapered thread. The OD will be at the bottom of the thread.

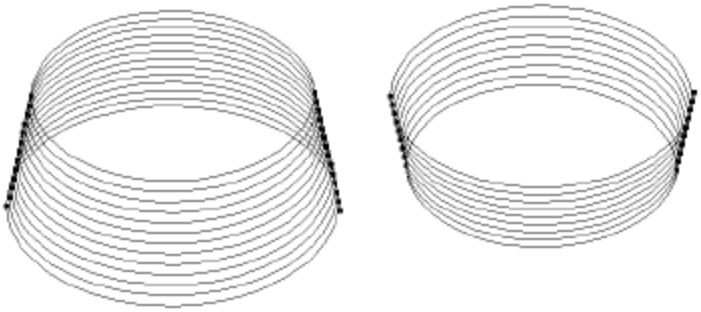
Thread Length
Height or length of the tapered thread. The tip of the tapered thread will begin at the same depth as the selected centerpoint geometry.

Degree of Taper
Angle of the tapered thread. A positive number means that the taper gets larger as it moves down in Z, and a negative number signifies that the taper gets smaller as it moves down in Z.

TPI/Pitch
Lets you set the threads per inch (TPI) or Pitch of the tapered thread. The label depends on the units of measurement specified in the Document Control dialog (DCD).

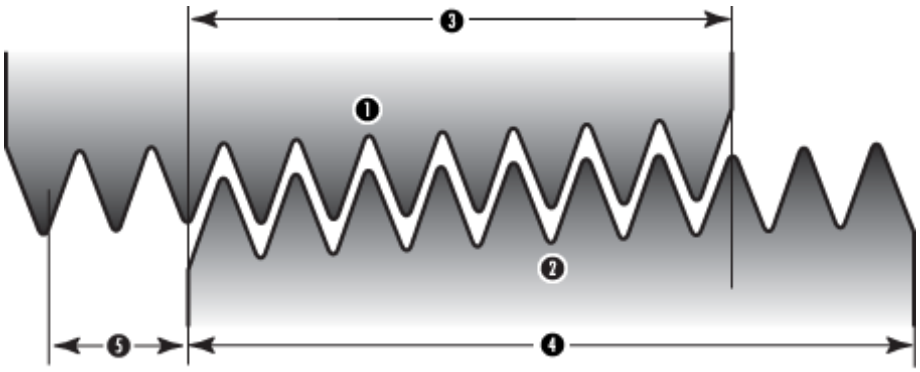
Taper Direction
Lets you select left-hand or right-hand thread direction.

Thread Orientation
Lets you specify the orientation of the thread, either OD or ID. An OD thread's radius increases as it gets deeper, whereas an ID thread's radius decreases as it gets deeper:



OD Thread	ID Thread

Engagement Length
Length of the area in which the interior and exterior threads meet when screwed hand tight. Available when ID is selected.



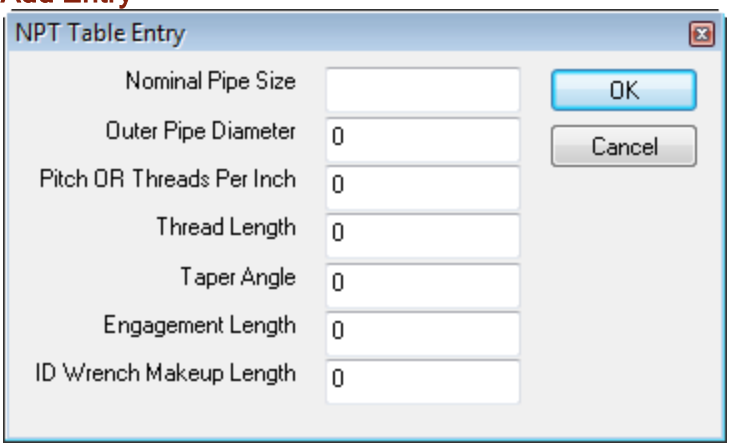
1. Interior Thread
2. Exterior Thread
3. Engagement Length
4. Length
5. Wrench Makeup

ID Wrench Makeup
When ID *is selected:* Length added to the inner thread (to facilitate tightening the pipe).

Thread Geometry
Lets you specify whether Arcs or Splines will be used to create the tapered thread.

NPT Table Button
Accesses a table of standard tapered thread sizes. Selecting an entry in the table and clicking OK enters thread values into the corresponding fields in the tapered thread dialog. These values are taken from *Machinery's Handbook*.

Add Entry



Enter a custom thread. The new entry will be appended to the end of the table.

Edit Entry

Edit an existing thread entry.

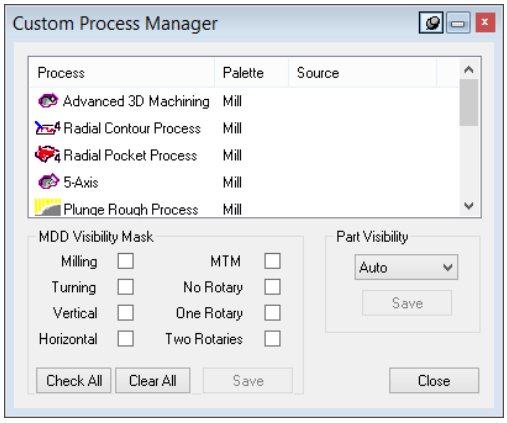
Delete Entry

Deletes the selected thread entry.

Default List

Restores the NPT Table to its original state.

Custom Process Manager



The Custom Process Manager Plug-in can control the visibility of Custom Processes on the Machining Palette. This is accomplished by changing the process visibility based on the MDD types, or changing the visibility for a specific part. The visibility of the custom process can also be changed at the time of installation by providing .CPV files that control the default visibility of the processes based on the MDD types.

MDD Visibility Mask

Once a process is selected, use this field to change the MDD mask settings.

Mill

All non-turning machines.

Turning

All turning machines, including MTM.

Vertical

All vertical machines.

4 Axis

All milling machines with exactly 4 axes and all turning machines with only a C rotary axis.

5 Axis

All milling machines with exactly 5 axes and all turning machines with at least a C and A rotary axis.

MTM
All machines that use VMMs and all machines with multiple toolgroups, flows or workpieces.

Horizontal
All horizontal machines.

3 Axis
All milling machines with exactly 3 axes and all lathe machines with no rotary axis.

Part Visibility
Once a process is selected, use these fields to modify the process visibility for the open part, and click on the Save button to update the MDD mask settings for the selected process or processes.

Auto
There is no part specific setting. The current MDD mask settings will be used to determine if the custom process should be visible based on the MDD type.

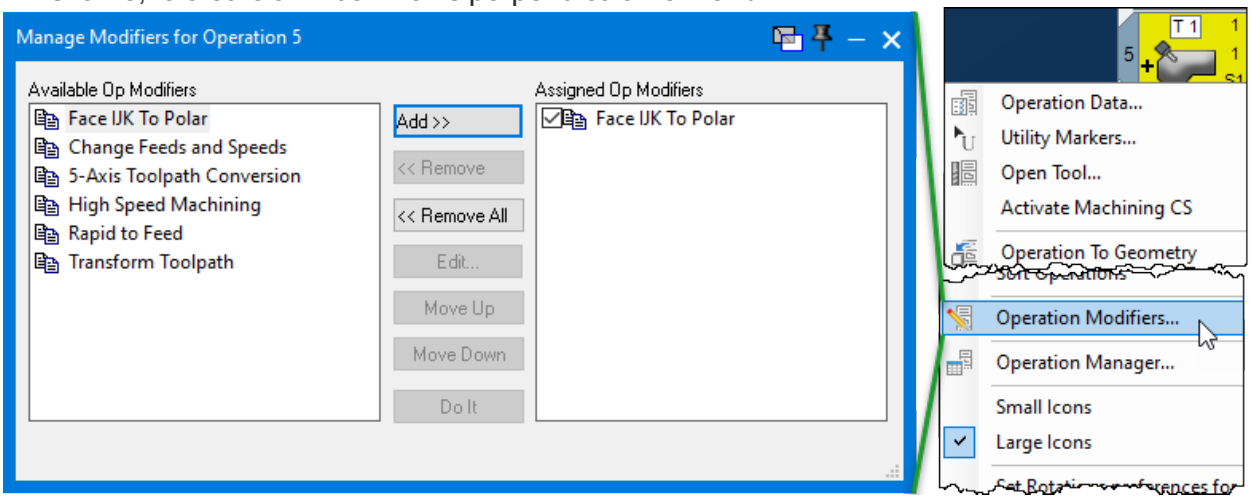
Visible
The Custom process will be visible for the open part. The part visibility information is saved with the part, so don't forget to save the part after changing those settings on a specific part.

Hidden
The Custom process will be hidden for the open part. The part visibility information is saved with the part, so don't forget to save the part after changing those settings on a specific part.

Divide Contour

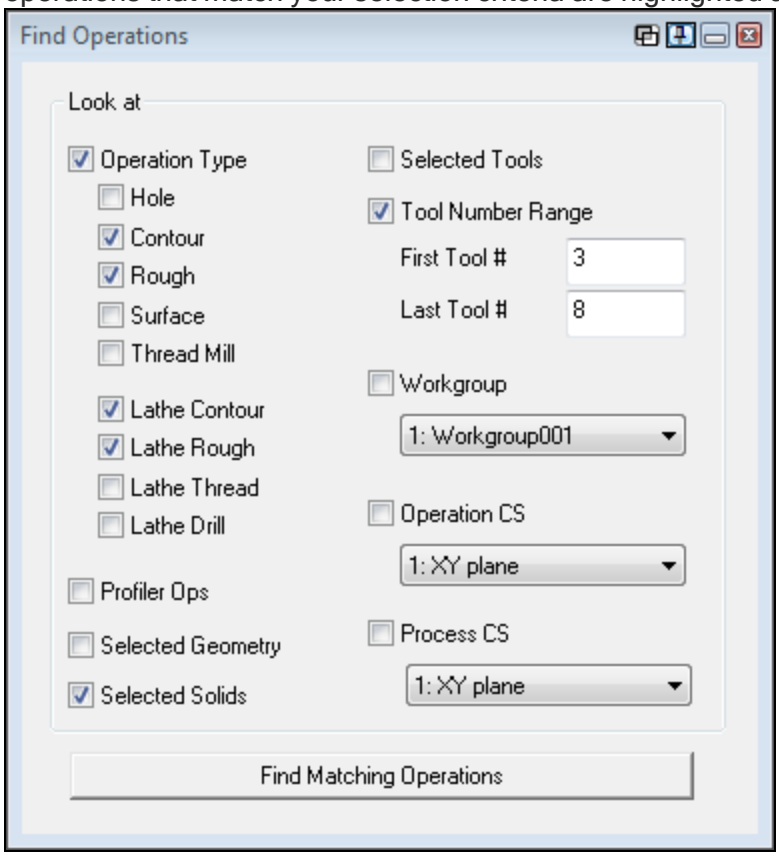
Divide Contour
This plug-in creates points at a given distance around a selected contour. The total length of the geometry is measured, and if the length is not evenly divisible by the specified Division distance, the system will set a value according to which it will segment the geometry.
The division distance is calculated by dividing the geometry's length by the (rounded up) results of the length divided by the Division distance specified in the dialog. This results in a number that is smaller than your specified Division distance that will still produce the same number of points.

Face IJK to Polar
This Operation Modifier replaces a distribution-only plug-in. It lets you use polar kinematics in 5-Axis operations where the tool aligns with the part's rotary axis. The most common use is in an XZC lathe, to create a virtual Y-axis perpendicular to X and Z.



Find Ops

This feature is a search tool. It finds operations by operation type, by tools used, by geometry or solids to be machined, and by workgroup and CS. When you click Find Matching Operations, all operations that match your selection criteria are highlighted and selected in the Operation List.



Operation Type:
Find operations whose process type matches *any* of the selections – Hole, Contour, ...
Profiler Ops:
Find only the operations that use Profiler.
Selected Tools:
Find only the operations that use the tools selected in the Tool List.
Tool Number Range:
Find only the operations using tools in the range you specify.
Selected Geometry:
Find only the operations that use the geometry selected in the workspace.
Selected Solids:
Find operations that use the solid body or bodies selected in the workspace.
Workgroup:
Find only the operations that use geometry in the workgroup chosen in the pull-down list.
Operation CS:
Find only the operations that machine in the CS chosen in the pull-down list.
Process CS:
Find only the operations whose original CS – that is, the CS of the process before any Op Modifiers were applied – matches the CS chosen in the pull-down list. The toolpath for such

operations, might be in a different CS. For other searches, clear the Process CS checkbox.

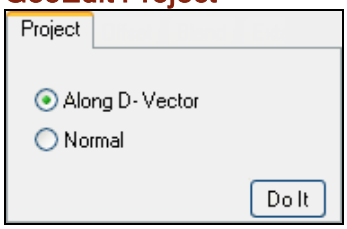
GARR Tool Library

See "Tool Library Explorers" on page 94.

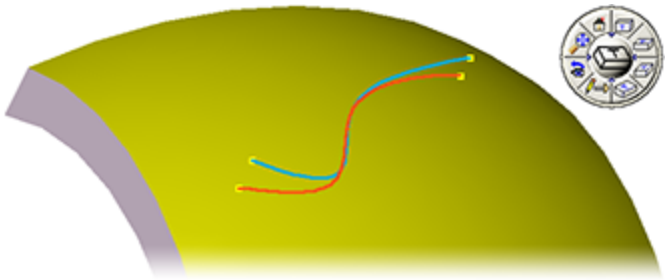
GeoEdit

These plug-ins provide you with several tools for modifying geometry, often based on curves and solids.

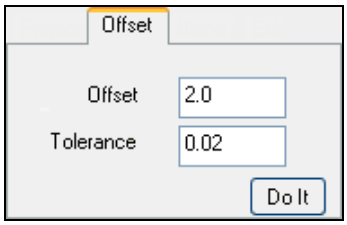
GeoEdit Project



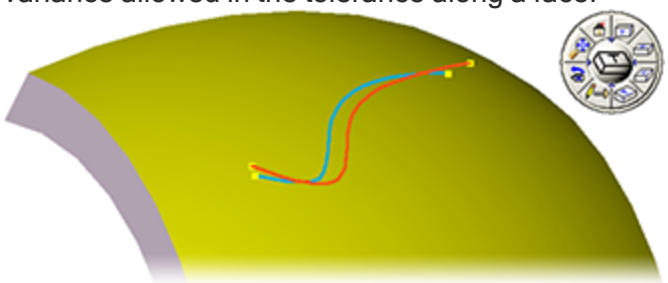
This option will project a shape onto a face. Select the shape you want to project and the face you want to project the shape on to. Make sure the shape is located completely above the selected face, and projects completely inside the boundary of the face. The Along D-Vector option will project the shape along the depth vector of the current CS. The Normal option projects the shape normal to the face selected.



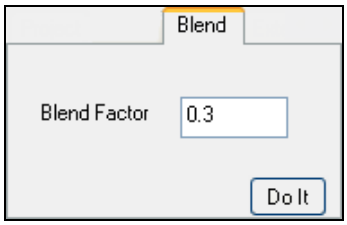
GeoEdit Offset



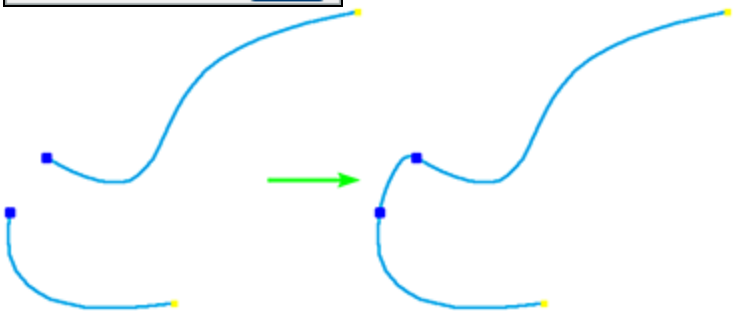
This option will offset a shape along a face. Select the shape you want to offset, and the face on which the shape is lying. The selected shape must not have any sharp corners and the curvature radius at every point must be larger than the offset value to be used. The Offset value is the distance the shape will be offset by. The sign affects to which side the offset will be calculated. The Tolerance is the variance allowed in the tolerance along a face.



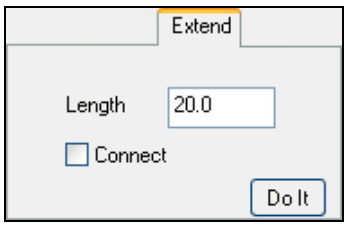
GeoEdit Blend



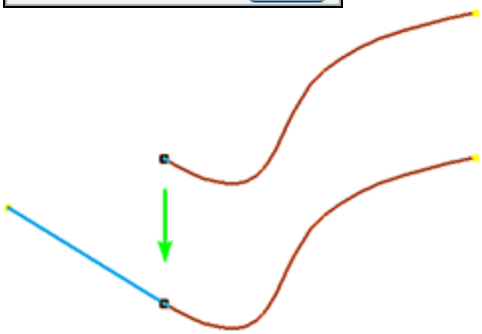
This option will create a curve between two curves. Select the two curves you want to blend, by selecting the terminator corresponding to the side you want to blend. The Blend Factor changes the shape of the curve that will be blended between the 2 selected curves. The value must be between `0.0` and `1.0` and the recommended value is `0.3`.



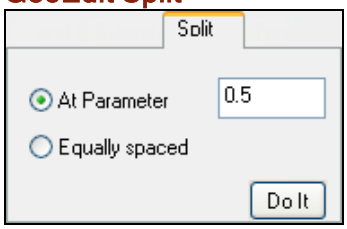
GeoEdit Extend



This command will create a line extending out from a curve. The line is created from a selected terminating point. The Length value is the length of the line that will be added to the selected curve. Depending on the selection for the Connect option, the new line will or will not be automatically connected to the shape.



GeoEdit Split



This command will break a curve into multiple components. Select the curve you want to split.

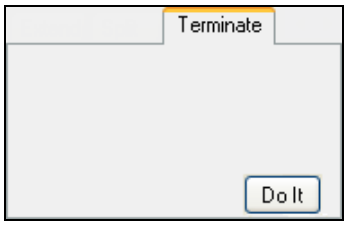
At Parameter

This option lets you specify a parametric value corresponding to the location where the curve will be split, e.g. `0.5` is halfway along the curve, `0.76` is 76% of the way along the curve. The value must be between `0.0` and `1.0`. When using this option you must select the end (connector/terminator) corresponding to the 0.0 value for the parameter.

Equally Spaced

This option will split the curve into the number of segments specified.

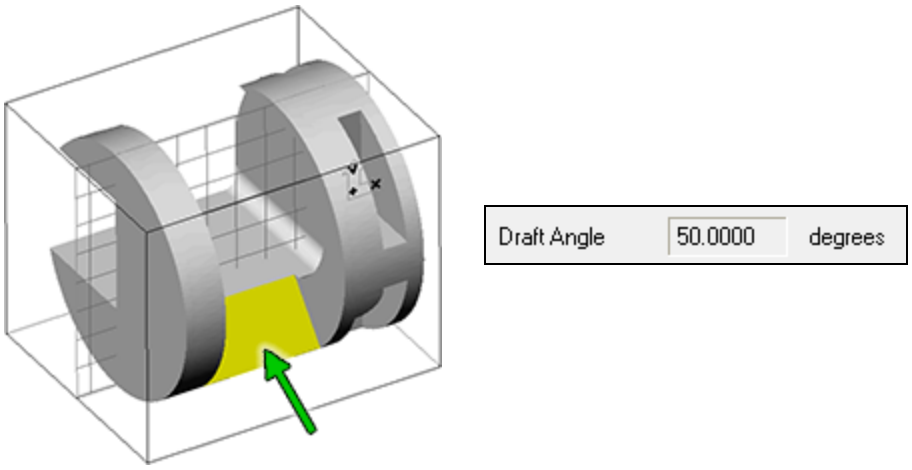
GeoEdit Terminate



This option terminates a selected B-spline curve.

Get Draft Angle

The Get Draft Angle Plug-in informs the user of the draft angle of a selected face. Simply select a face on the model and select Get Draft Angle to bring up a dialog with the draft angle of the face in degrees.



The angle returned is calculated based on the current CS. Since the draft angle is based on the current CS, the value may not be able to be calculated, as it is not really applicable. Note that this plug-in does not support multiple selections.

GRANITE Information

This plug-in was retired at version 2024. Its functionality is now provided natively and/or by **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel).

Groove Cycle

Designed around the Fanuc-style canned cycles for G74 and G75 output, Groove Cycle allows you to cut geometry-independent rectangular grooves.
This plug-in was retired at version 13.0, when its functionality was moved into the Groove Cycle G74/75 process dialog.
For information on the Groove Cycle process, see the *Turning* guide.

Harvey Tool Library

See “Tool Library Explorers” on page 94.

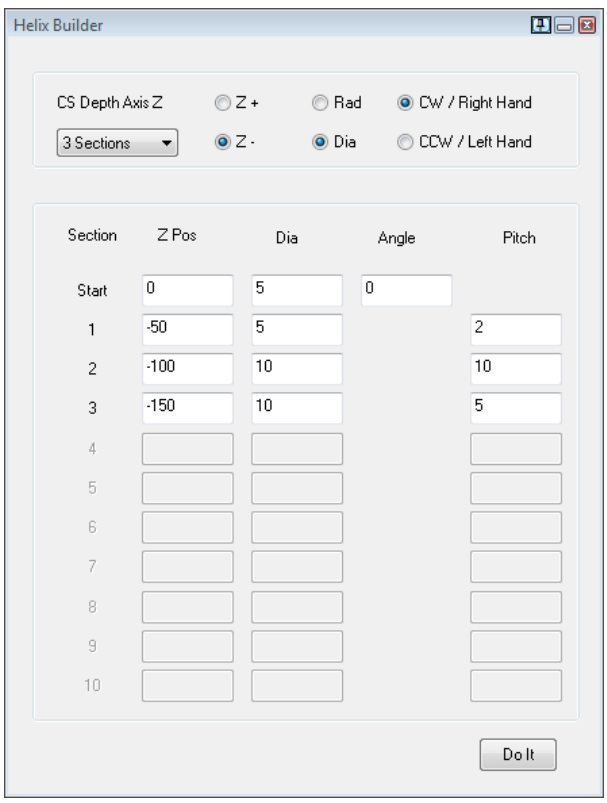
Helical Solutions Tool Library

See “Tool Library Explorers” on page 94.

Helix Bore

This plug-in was retired at version 11.0, when its functionality was moved into the **Holes** process dialog. Choosing Helix Bore as an Entry/Exit Cycle in the **Drill** tab causes the **Bore** tab to become available.
For information on controls presented by the **Bore** tab, see the *Mill* guide.

Helix Builder



The Helix Builder Plug-in provides a quick and easy way to create helical geometry.

CS Depth Axis ‘X’

Z+ / Z-

This defines the geometry along either the positive or the negative direction of the current CS depth axis.

Rad

This defines the geometry using radius values (measured in part units).

Dia

This defines the geometry using Diameter values (measured in part units).

CW / Right Hand

The geometry created will be defined in a clockwise direction (Right Hand) as viewed along the CS depth axis.

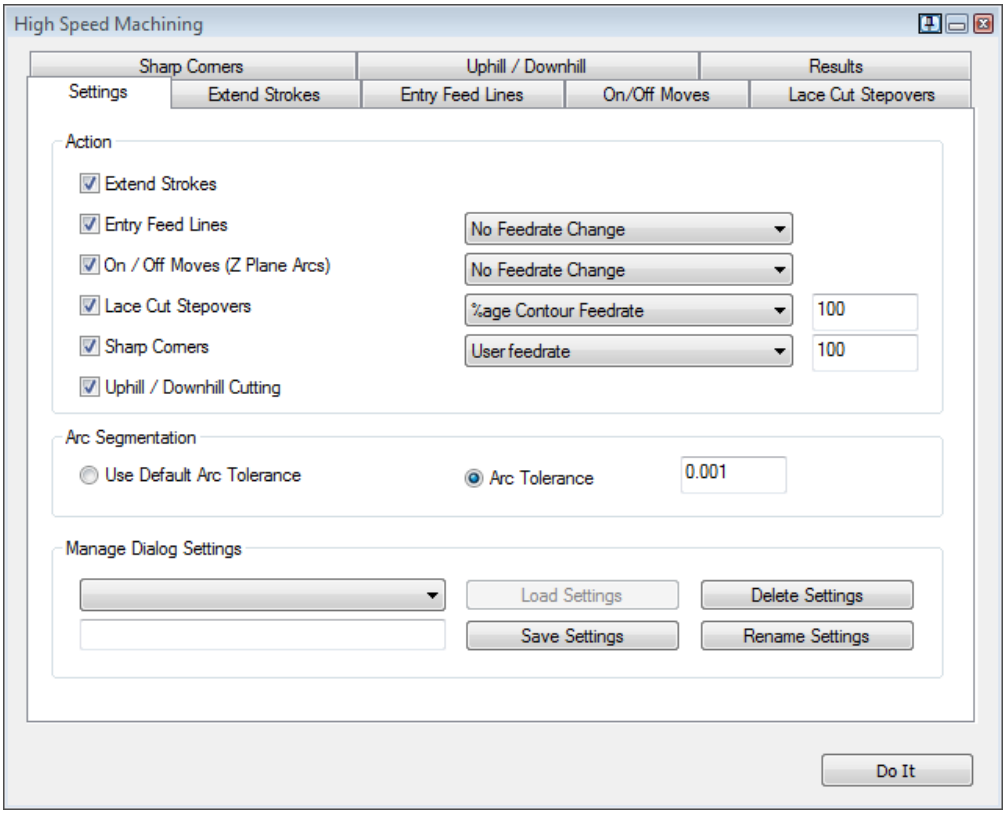
CCW / Left Hand
The geometry will be defined in a counter-clockwise direction (Left Hand) as viewed along the CS depth axis.
Sections
This drop-down list determines the number of sections.
Section
Z Position
This is the start position of each individual section beginning with the Start position.
Rad / Dia
The Diameter or Radius for the specified section.
Angle
This is the Start Angle of the Start section.
Pitch
This value is the part units per revolution.

High Speed Machining (HSM)

The High Speed Machining (HSM) plug-in can extend the toolpath, change rapid moves into feed moves and /or add loops to a lace cut toolpath or add a radius to the entry and exit moves of a toolpath. The HSM plug-in has these capabilities and others that are necessary for high speed machining. For a detailed description of each tab, see "Settings tab" on page 47, "Extend Strokes tab" on page 49, "Entry Feed Lines tab" on page 49, "On/Off Moves tab" on page 50, "Lace Cut Stepovers tab" on page 51, "Sharp Corners tab" on page 51, "Results tab" on page 53, and "Uphill / Downhill" on page 53.If you want to persist the settings and effects of this functionality (for example, preserving it through Redo or Redo Ops), you can apply it as an operation modifier instead of using the plug-in. For information on using operation modifiers, see the *Mill* and *Turning* guides.

Settings tab

The Action items listed below give options for changing various operation feedrates. Checking the Action item(s) on this tab is necessary to activate the correlating tab or tabs. Additionally, set the preferences for Arc Segmentation. The Manage Dialog Settings area lets you save setups.



Action

Extended Strokes
Extends the toolpath.

Entry Feed Lines
Change a rapid move to a feed move when entering a part. From the Settings tab you can also select to change the feedrate. The selections are, No Feedrate Change, Entry Feedrate or use a Percentage of Contour Feedrate or determine your own User Feedrate.

On / Off Moves (Z Plane Arcs)
Add Z Plane Arcs to entry and exit moves on the toolpath. From the Settings tab you can also select to change the feedrate. The selections are, No Feedrate Change, Entry Feedrate or use a Percentage of Contour Feedrate or determine your own User Feedrate.

Lace Cut Stepovers
Put loops on the Lace Cut toolpath ends. From the Settings tab you can also select to change the feedrate. The selections are, No Feedrate Change, Entry Feedrate or use a Percentage of Contour Feedrate or determine your own User Feedrate.

Sharp Corners
Add Fillets or Loops to Sharp Corners. From the Settings tab you can also select to change the feedrate. The selections are, No Feedrate Change, Entry Feedrate or use a Percentage of Contour Feedrate or determine your own User Feedrate.

Arc Segmentation

Use Default Arc Tolerance

This Plug-in creates 3D arcs. GibbsCAM has a default segmentation tolerance defined under Preferences that can be used to output code for 3D arcs and splines. If you want to override the default setting and set your own segmentation Arc Tolerance value here, otherwise Use Default Arc Tolerance.

The segmentation tolerance is the maximum distance you will permit the small lines to deviate from the original arc, so the bigger the tolerance the fewer lines you have, but a rougher output. When using this plug-in you may want to output rough 3D arcs, since they are not actually used for cutting material, this will get the tool off the part and keep the tool moving without any abrupt changes in direction. It may be more efficient to have an arc converted into 5 segments by this plug-in rather than a smoother fit of 20 segments using the Preference settings.

Arc Tolerance

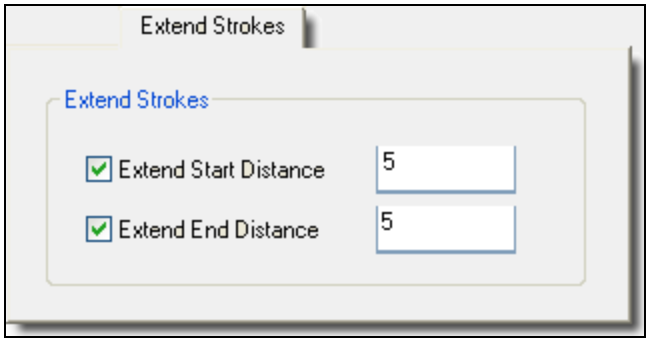
Segment all arcs using the specified Arc Tolerance.

Manage Dialog Settings

This feature enables the user to save a variety of different settings, that will then be available on the pull down menu. When using this feature the user will setup all the HSM Tabs as desired, when finished, name the settings and select the Save Settings button. That selection will become available to you on the pull down menu in Manage Dialog Settings for future use.

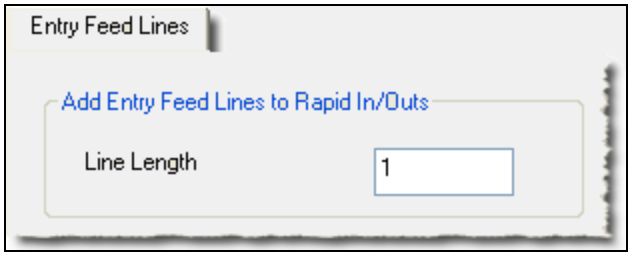
Extend Strokes tab

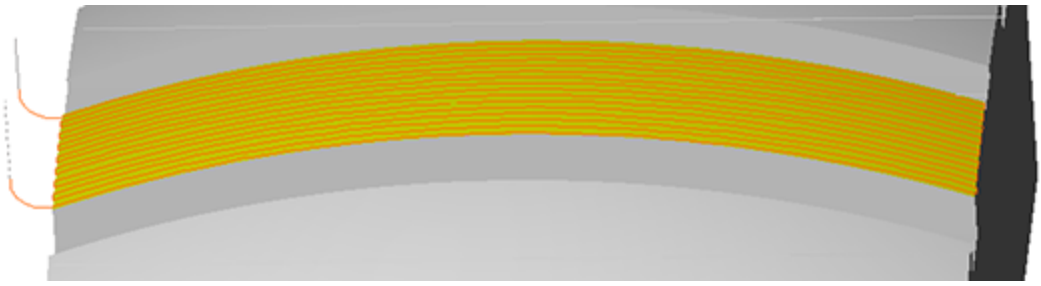
Extend Strokes will extend the stroke Start Distance and / or the End Distance by a user specified amount.



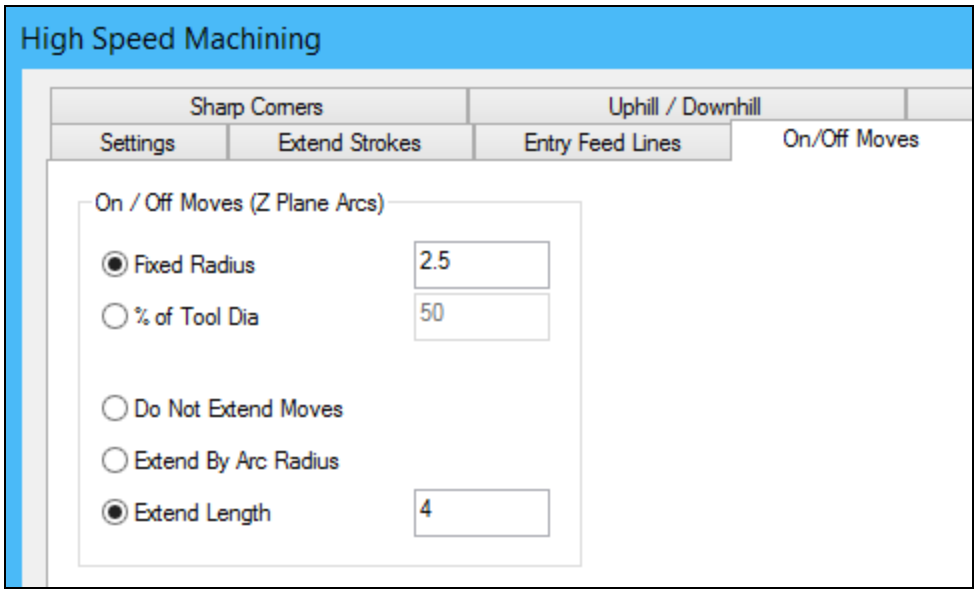
Entry Feed Lines tab

The Entry / Exit Feed Lines tab will change all Z rapid moves to feed moves to the user specified Line Length. Shown below is a Lace Cut toolpath where one inch Entry Feed Lines were added to all of the Rapid In/Out moves.





On/Off Moves tab



On /Off Moves (Z Plane Arcs) puts a radius (arc) on the entry and exit moves of a toolpath. You can choose between a Fixed Radius or use a Percentage of the Tool Diameter associated with the operation. The On/Off Moves can not be extended or they can be extended either by the Arc Radius or by a user specified length.

Fixed Radius
Radius (arc) will be the specified amount.

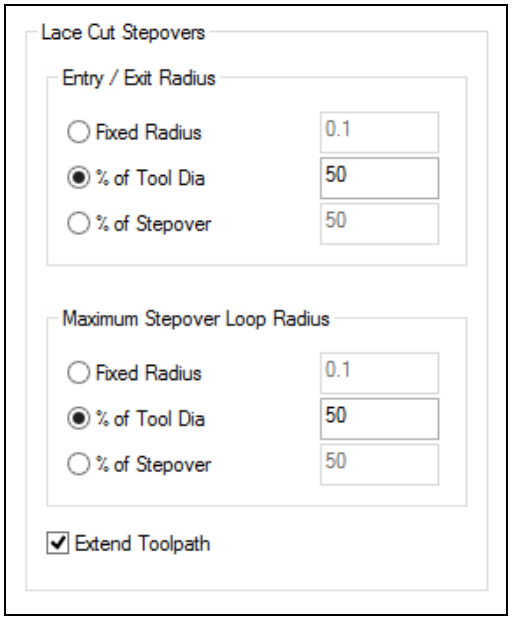
% of Tool Dia
Radius (arc) on the entry and exit moves will be the input percentage of the diameter of the tool associated with the operation.

Do Not Extend Moves
Adds the radius (arc) but will not extend the toolpath.

Extend By Arc Radius
The amount the toolpath extends will be the arc radius.

Extend Length
The toolpaths entry and exit moves will extend by the specified length.

Lace Cut Stepovers tab



Entry / Exit Radius

Lace Cut Stepovers will put loops on the lace cut toolpath ends. The Entry / Exit Radius will determined by one of the following radio buttons.

Fixed Radius

The Entry / Exit Radius of the arc on each lace cut will be the specified amount.

% of Tool Dia

The Entry / Exit Radius of the arc on each lace cut will be the input percentage of the diameter of the tool associated with the operation. If the tool associated with our operation is a 1 inch endmill and we have the Entry / Exit Radius at 50% of our tool diameter as shown on the right, our Entry / Exit Radius of the arc on each lace cut would be 0.5 inch.

% of Stepover

The Entry / Exit Radius of the arc on each lace cut will be the specified percentage of the stepover. If we have a 1.5 inch XY Stepover and we want our Entry / Exit Radius at 50% of the stepover as shown above, our Entry / Exit Radius of the arc on each lace cut will be 0.75.

Maximum Radius for Stepover Loop

Lace Cut Stepovers puts loops on the toolpath ends. The maximum radius of any stepover loop will determined by Fixed Radius, % of Tool Dia or % of Stepover radio buttons.

Fixed Radius

The maximum radius of each stepover loop will be the specified amount.

% of Tool Dia

The maximum radius of each stepover loop will be the input percentage of the diameter of the tool associated with the operation.

% of Stepover

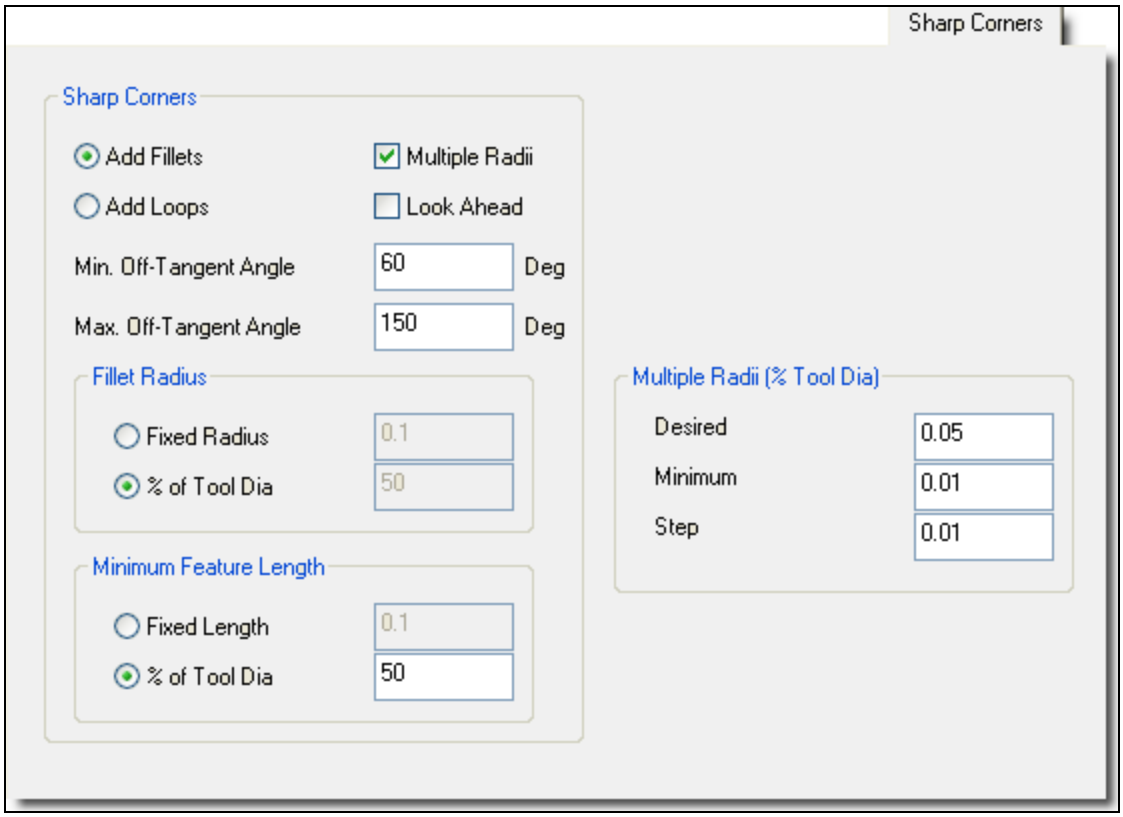
The maximum radius of each stepover loop will be the input percentage of the Stepover.

Extend Toolpath

Extends the radius that is tangent to the end of the stroke move off the part. To extend only the toolpath, see the Extend Stokes tab. Extend Toolpath is the default.

Sharp Corners tab

Sharp Corners will add fillets or loops to inside sharp corners.



Sharp Corners

Add Fillets or Loops to all inside sharp corners.

Add Fillets

Adds Fillets to inside sharp corners.

Multiple Radii

Used with the Add Fillets option. This begins adding fillets at the Desired Radius then decreasing in Step amounts until all sharp corners have been filleted down to the specified minimum radius.

Look Ahead

When a fillet will not fit, the plug-in will look to either side of the corner for features where a Fillet will fit.

Add Loops

The two features that make up a sharp corner will be extended and a loop of the given radius will be added between these extensions.

Min. Off-Tangent Angle (deg)

A fillet or loop will not be added to features that are less than the Min. Off-Tangent Angle (measured in degrees).

Max Off-Tangent Angle (deg)

A fillet or loop will not be added to features that are more than the Max. Off-Tangent Angle (measured in degrees).

Fillet Radius

The maximum radius of each fillet will be the specified Fixed Radius or the specified percentage of the diameter of the tool associated with the operation, the % of Tool Dia.

Minimum Feature Length

Fillets or loops will not be added where either of the two features that make up a corner are less than this minimum feature length. This is based on either a specified Fixed Length or a

specified percentage of the diameter of the tool associated with the operation, the % of Tool Dia.

Multiple Radii (Fixed Radius or % Tool Dia)

The multiple radii of the fillets can be an Actual Value or a percentage of the tool diameter. The fillets will start with the Desired radii then decrease in Step amounts until reaching the Minimum.

Desired

The specified Desired radius of fillets added to sharp corners.

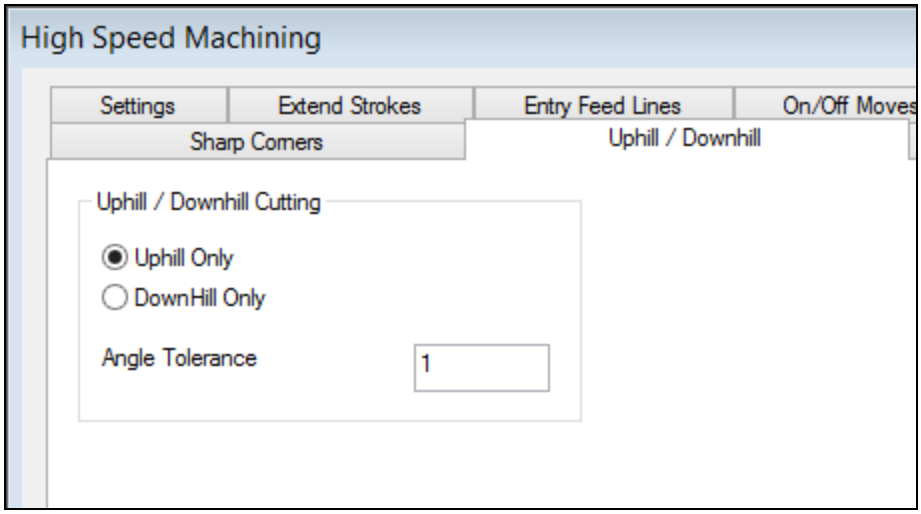
Minimum

The specified Minimum radius used when adding Fillets to Sharp Corners.

Step

The specified amount either Actual or % of Tool Diameter there will be between the decreasing steps.

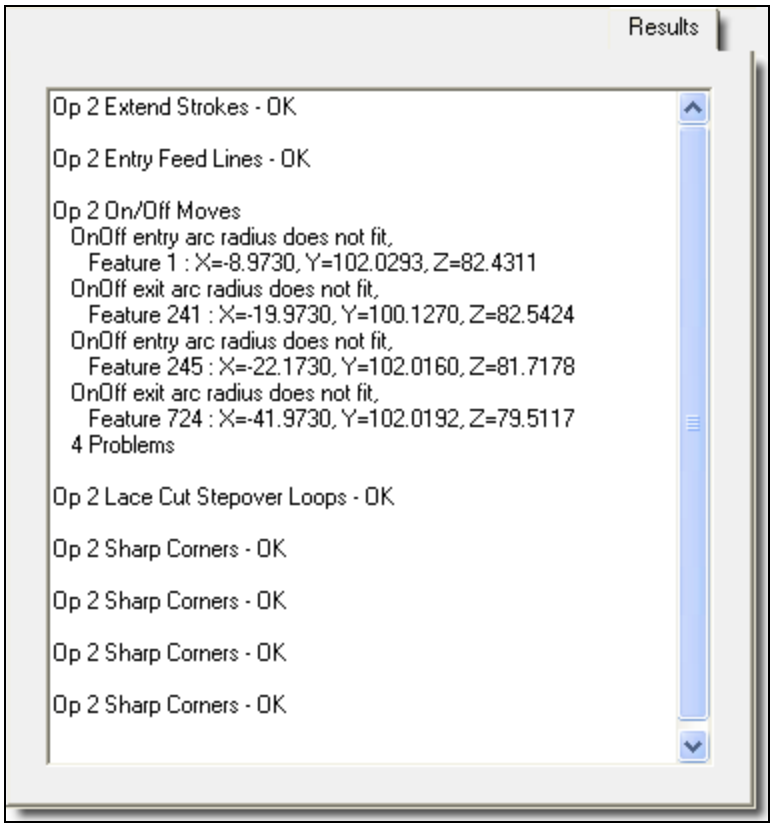
Uphill / Downhill



The Uphill / Downhill tab will cut only in the selected direction, an uphill direction (bottom to top) or a downhill direction (top to bottom). Any moves that are above the Angle Tolerance will be deemed to be in the opposite direction and the cuts will be output in the reverse direction.

Results tab

The Results tab will give you information regarding the action(s) that were performed by this plug-in and /or the action(s) it was unable to complete.



Import Material

This plug-in was retired at version 13.0, when its functionality was moved into the **Materials** dialog box, which can be summoned from the main menu (**File > Materials**) or by clicking the Material button in a process dialog.
For information on the Materials dialog box, see the *Common Reference* guide.

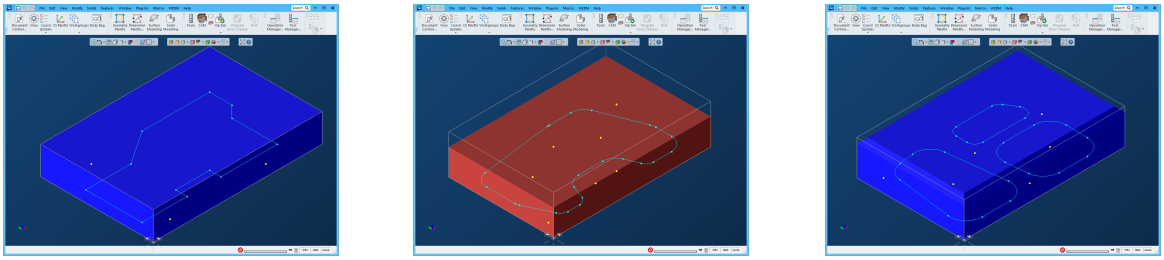
Import VNC

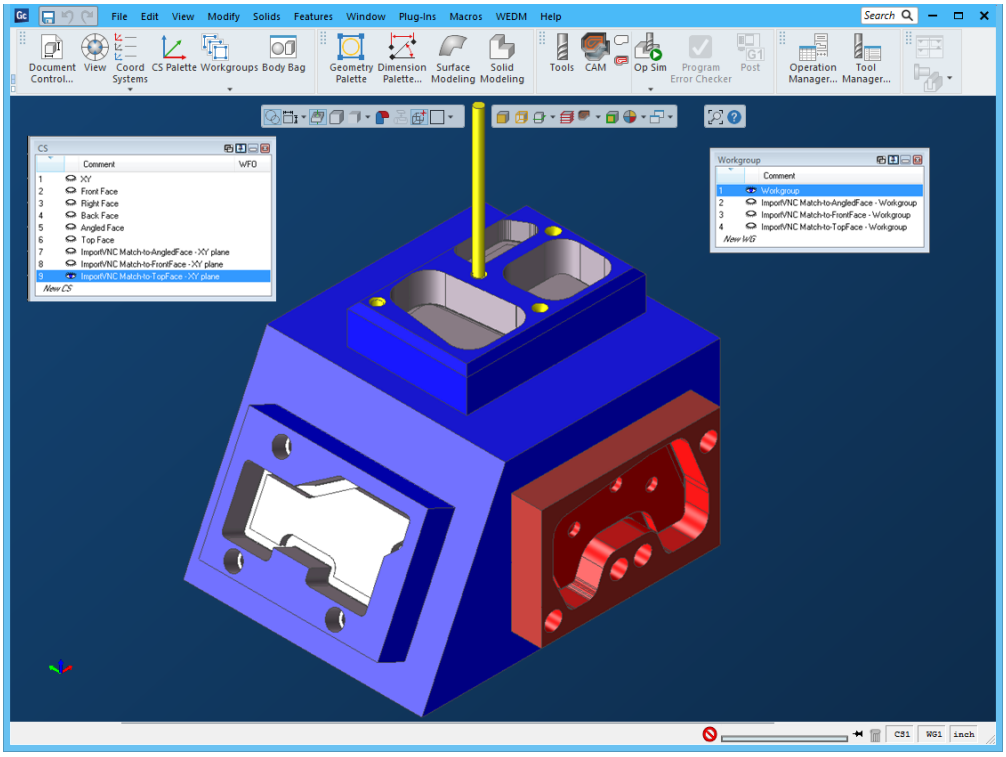
The Import VNC plug-in imports any VNC file with operations into any open file in the system. Although located by default under Plug-Ins >Mach.Sim.-TMS, the plug-in has more uses than just for Tombstone Machining. Because it imports processes and workgroups as well as bodies and elements, you can make adjustments to processes (if needed) and use Redo to regenerate toolpath.
Important: Unregenerated toolpath is not aware of other bodies added to the part, such as newly imported parts, stock bodies, or fixtures.

> ⓘ Pre-existing toolpath remains unchanged until a Redo occurs. Changes to geometry or processes are not dynamically accompanied by toolpath updating. You could move or sort tiles, edit values, or even add new bodies and processes, but all previous toolpath will remain the same (and therefore possibly wrong) until the next Redo.

> When toolpath is recalculated (either explicitly when you click Redo, or implicitly when you edit and save an operation modifier), the new operations reflect the new conditions. For example, if tools, geometry/bodies/fixtures, or processes have been removed, modified, or added (such as through Import VNC or copy-and-paste), then regenerated operations will match the changes, and will vanish whenever there is no toolpath to generate.
> *Exception:* An operation data value that is locked ( ) is protected from modification by process edits. If process settings change, Redo affects only the settings that are unlocked ( ).

Import VNC has the same effect as Copy/Paste from one GibbsCAM session to another.



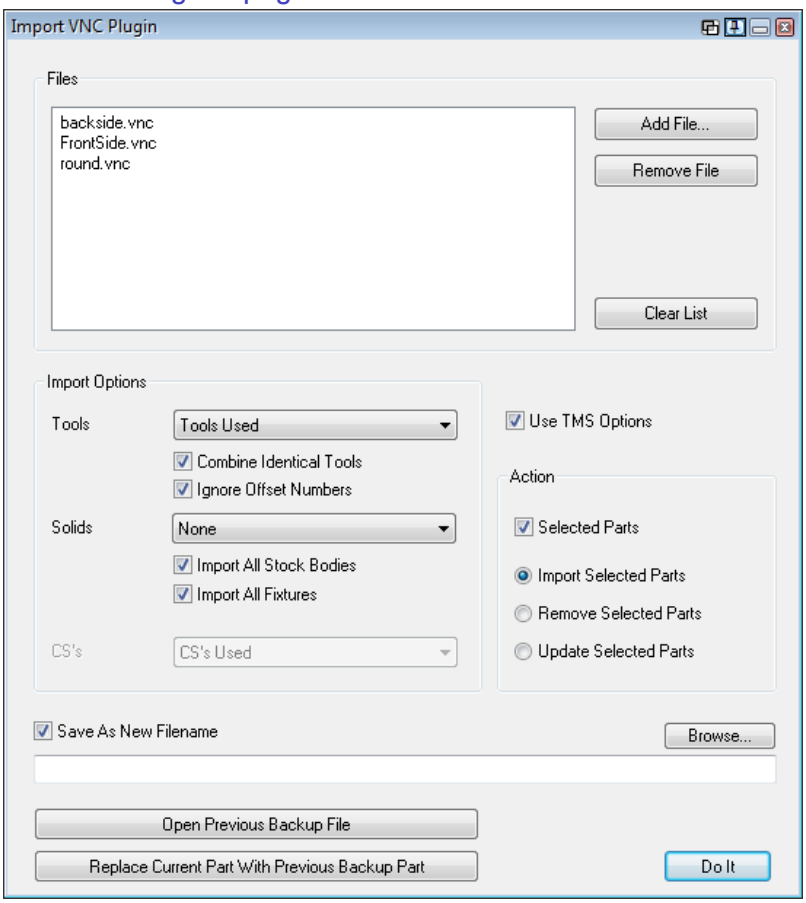


Interface

The interface for Import VNC has five components, described in detail below:

- “Files” on page 56
- “Import Options” on page 56
- “Action” on page 57
- “File Handling” on page 57

- “CS Matching” on page 57



Files

Add File
Allows browsing for files that are to be added to the part.

Remove File
Removes selected file(s) from the list.

Clear List
Removes all files from the list.

Import Options

Tools
This pull down menu allows the user to import only the Tools Used or to import All Tools.

Combine Identical Tools
All criteria for identical tools will be combined.

Ignore Offset Numbers
This option will ignore all imported tool Offset numbers, (Tool length Offset # and Cutter Comp. Offset #) and give new offset numbers.

Solids
This pull down menu allows the user to import None of the solids, the Part Master Solid, the Solids Used, or both Part Master and Used Solids. A Part Master Solid is a solid you select to import. It can be a used solid or a unused solid, but it must be named “Part Master”.

Import All Stock Bodies
All Stock Bodies will be imported from the listed files.

Import All Fixtures
All Fixtures will be imported from the listed files.

CS’s
This pull down menu allows the user to import only CS’s Used in the part or All CS’s.

Use TMS Options
If you are importing into a part ready for use with Tombstone (TMS) this box must be checked. This will automatically check the Match CS’s option and select the Only Import Ops that Use Selected CS radio button, while leaving Create New CS’s unchecked. Using these options, only one CS may be used in each imported part.

Action

Selected Parts
Checking this box will Import Selected Parts, Remove Selected Parts or Update Selected Parts only to or from the current part.

Import All Parts
Import all parts from the Files list in the dialog box into the current part.

Remove All Parts
Remove all parts, tools and operations from the Files list in the dialog box into the current part.

Update All Parts
Update the files in the current part. Any files not already in the current part will be added to the part.

File Handling

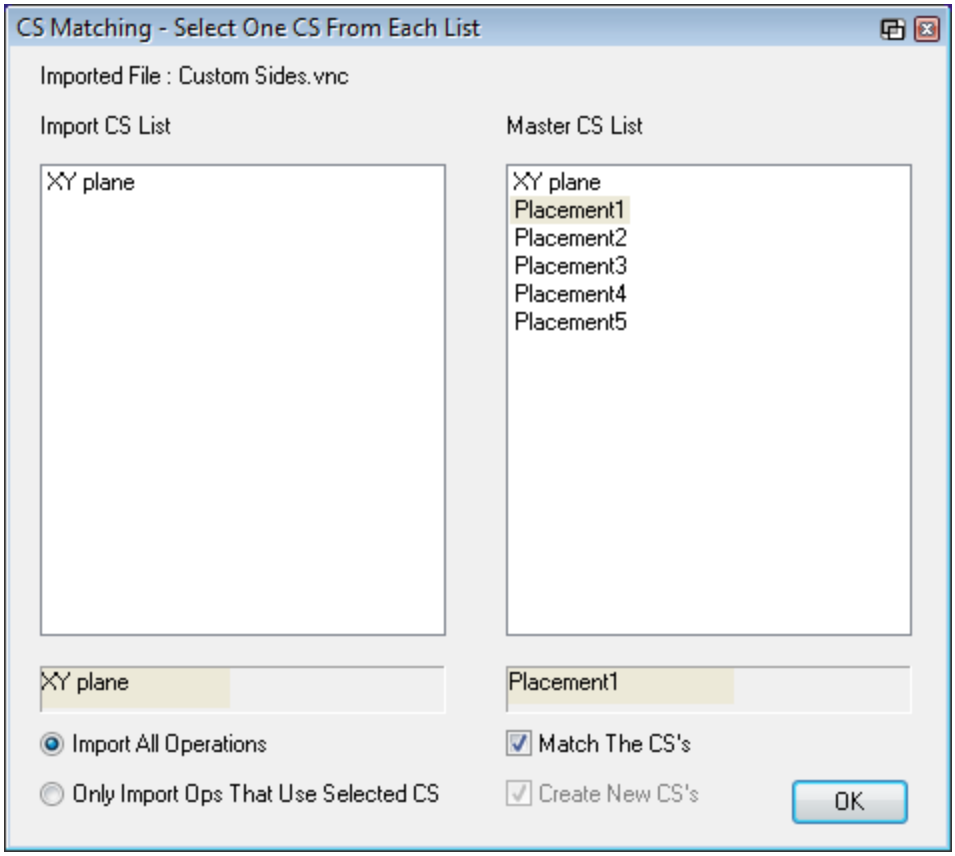
Save As New Filename
This selection will save the current file with a new filename.

Open Previous Backup File
This selection closes the part file that is currently open and will open the Previous Backup file (this is the original open part file before any imports).

Replace Current Part With Previous Backup Part
This selection opens the Previous Backup Part file and saves it under the current filename replacing the part file that is currently open.

CS Matching

Clicking Do It in the **Import VNC** dialog will bring up the CS Matching dialog as shown below. This lets you control how coordinate systems should be mapped.



Import CS List
This is the list of all of the CS's in the part to be imported.

Master CS List
This is list of all the CS's in the original part (the Master list) into which the new part is to be imported.

Import All Operations
This selection is for when you have a part that you want to import completely; it imports all operations.

Only Import Ops That Use Selected CS
This selection allows you to import a selected toolpath and then pick the relevan CS from the import list.

Match The CS's
When this is selected the part is positioned by associating it with a selected CS in the Master list. If the imported part uses multiple CS's, then the plug-in calculates where each CS needs to be and looks for a CS that is at the same location in the master list. If a matching CS is found it will be used, otherwise a new CS will be created. If this option is not selected the imported part will be positioned by rotating/translating it.

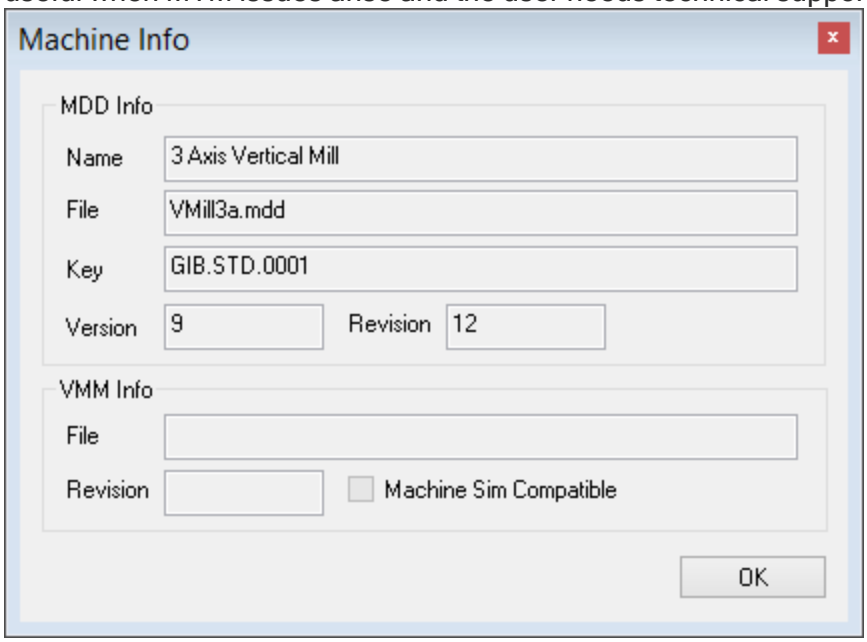
Create New CS's
When this option is selected, new CS's will always be created for each CS in the imported part. When it is unselected, Master CS's will be reused where it is possible.

ISCAR Tool Advisor (ITA)

You can use the ITA (Iscar Tool Advisor) pull-down choice on the title bar of the **Select Tool Type** flyout dialog.

Machine Info

The Machine Info Plug-in, for use with MTM, displays various MDD and VMM info. This Plug-in is useful when MTM issues arise and the user needs technical support.



Name
Name of the chosen MDD.
File
Name of the physical MDD or VMM file.
Key
Unique identifying number for that particular MDD.
Version
Version number of the MDD.
Revision
Revision number of the MDD or VMM.
Clicking OK will save the information in the Machine Info Plug-in as a text (.txt) file.

MachiningCloud Tool Library

See " Cloud-Based Tool Libraries" on page 32.

Mirror Ops

This feature mirrors selected geometry and toolpath. Select one or more ops and the Plug-in will make copies of these with the toolpath mirrored in X or Y. The toolpath can be reversed or modified to maintain the same G41/42 direction so the tool will continue to cut on the same (left/right) direction.

Mirror Type:

This selection Mirrors the selected item(s) in part units from the coordinate system origin either Vertically or Horizontally by a specified distance.

Workgroups:

There are three available Workgroup options. When the Use Same Workgroup option is selected all new mirrored geometry will go into the same workgroup as the original geometry. If One New Workgroup option is selected a new workgroup called either “Mirror X” or “Mirror Y” will be created and all new geometry will be put into the new “Mirror X” or “Mirror Y” workgroup. If a Mirror workgroup already exists, new geometry will be placed in the existing Mirror workgroup. The Separate New Workgroups option creates a new workgroup for each workgroup used by the original geometry, adding the “Mirror X” or Mirror Y” to the start of the original workgroup for the new workgroup name.

Contour and Pockets

Keep Same Tool Side (G41/G42) will keep the tool on the same side after the mirror operation.

Delete prior mirrored workgroups and solids

Deletes all prior mirrored workgroups and solids.

Replace selected operations

Replaces selected operations with the mirrored operations.

Model Associativity

The Model Associativity plug-in is designed for use when a solid has changed and you want to update the operations. This plug-in allows you to associate a toolpath with a changed solid. A toolpath can be unpredictable. It is recommended that you keep a copy of the original file.
The plug-in updates operations relating to faces that have been selected in the model. It does not update operations relating to Profiler selections. Therefore, to update operations created using the Profiler, you would need to update each such operation manually.

Action

Choose one action from the following three, specifying how to associate a toolpath with a changed solid.

Set To Selected Body:

Select one solid and update each selected op to now be associated with that new solid.

Replace First Solid With Second:

Select two solids and update each op that was associated with the first selected solid to now be associated with the second solid.

Replace With Child Solid:

Any operations that were associated with a parent (grandparent etc.) or the selected solid will now be associated with the selected solid.

Selection

You can select to update Selected Operations or All Operations.

Redo Modified Ops

This item reprocesses the Modified Ops. If changes have been made to the part geometry a new toolpath will reflect the geometry changes.

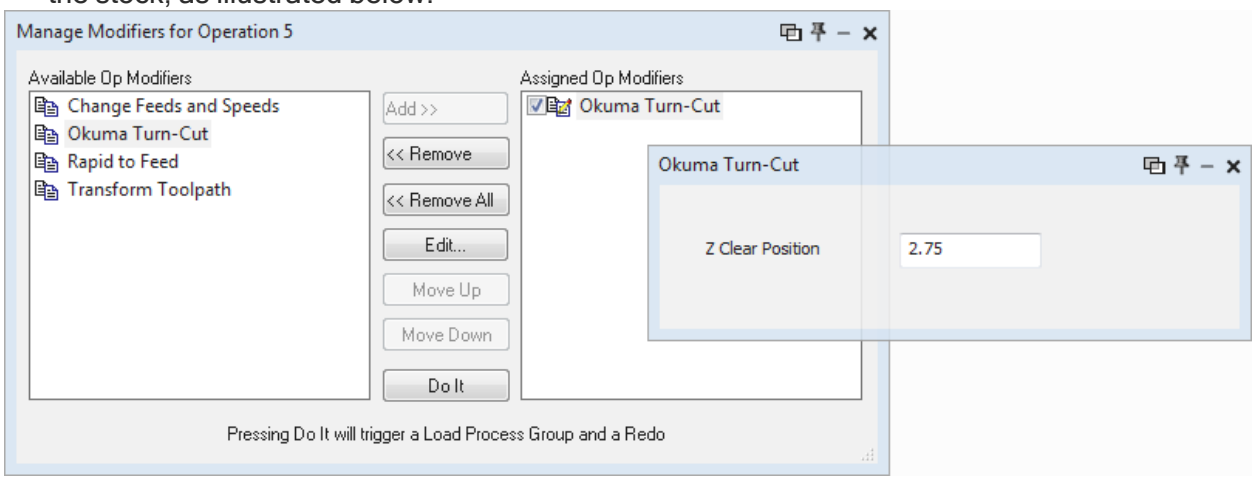
Offset Contour

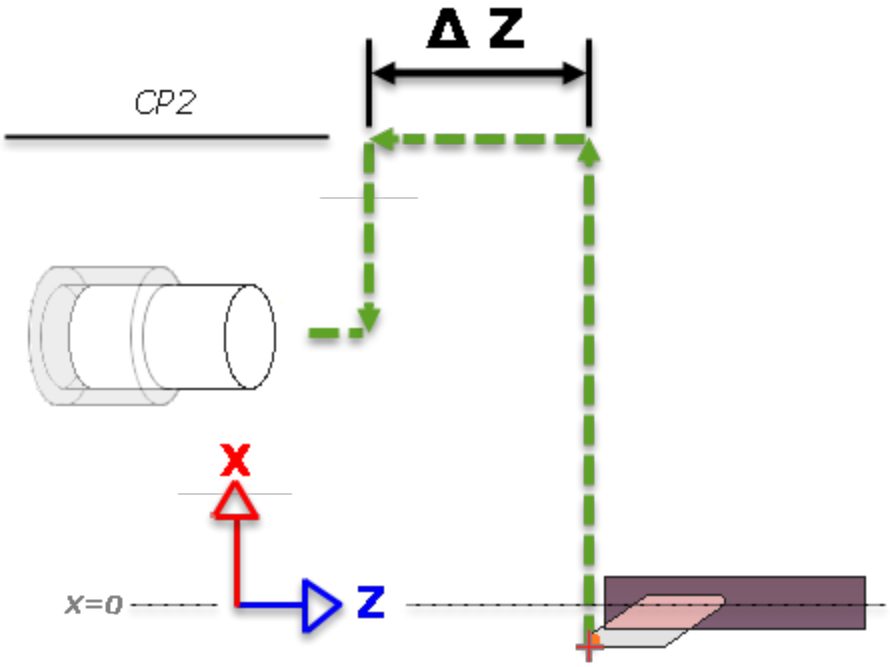
This plug-in was retired at version 10.5, when its functionality was moved into the process dialog. For information on the **Contour** process dialog, Contour tab, fields Number of Extra Offsets and Extra Stepover, see the *Mill* guide.

Okuma Turn-Cut

The Okuma Turn-Cut operation modifier compensates for a machine that constrains the toolholder to be centered on the X-axis.
This operation manager must be enabled using Plug-In Manager. This operation modifier cannot be applied to operations that use CSS (Constant Surface Speed).
Steps:

1. Right-click the operation you want to modify.
2. On the context menu, choose Operation Modifiers.
3. Click Add to add it to the operation.
4. In the **Okuma Turn-Cut** dialog, provide one value, Z Clear Position.
   This value specifies an extra clearance in Z to allow the system to use fast feedrate to bring the tooltip up to clearance plane 2 (CP2), over to the starting Z of the cut, and then down to the stock, as illustrated below.



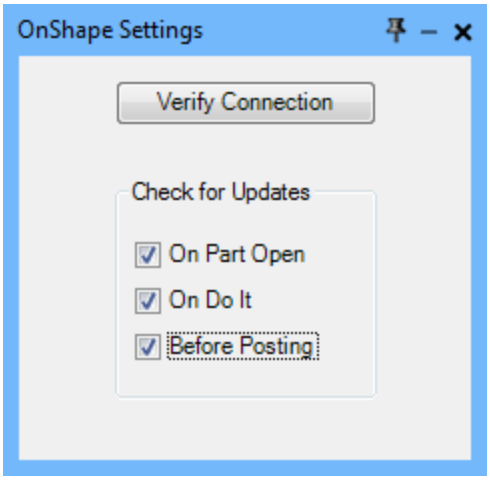


Onshape

Onshape is a SAAS (software-as-a-service) application that uses a cloud-based approach to sharing and collaborating on CAD models. In GibbsCAM, Plug-Ins > PLM offers three commands to provide access to your Onshape libraries:

	Settings – Opens a dialog that lets you verify the connection with the Onshape server, and to specify which actions in GibbsCAM trigger an update of any imported Onshape models.
	Update Check – Causes the Onshape server to check whether Onshape models in the current GibbsCAM part are up-to-date and, if any are not, opens a dialog that lets you update as needed.
	Browse – Opens a dialog that lets you view and choose Onshape models from various sources.

Settings



OnshapeSettings Dialog

Verify Connection
To check your connection to an Onshape server.
If you have never previously logged in, you will be prompted to sign in to your Onshape account.
After you sign in, you will not be prompted again in future GibbsCAM sessions.

Check for Updates
Use these checkboxes to instruct the system when to for fresher versions of all Onshape models in the current GibbsCAM part.

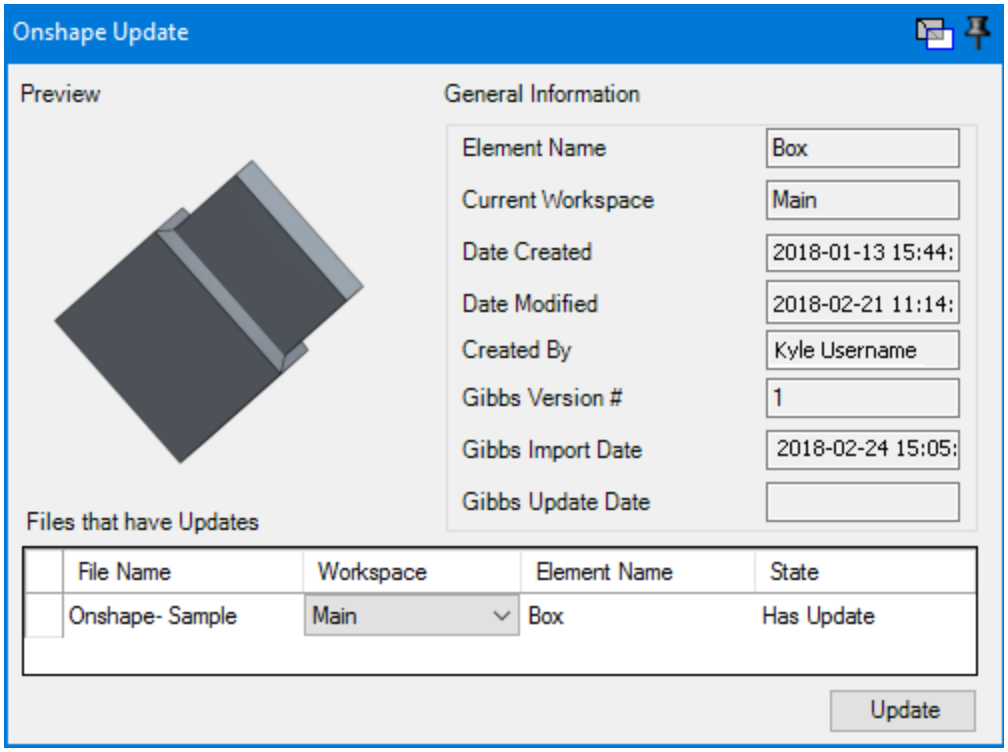
On Part Open
If selected, the system will check for revised versions of all Onshape models in the GibbsCAM part being opened.

On Do It
If selected, the system will check for revised versions of all Onshape models in the current GibbsCAM part whenever you generate (Do It) or regenerate (Redo) operations.

Before Posting
If selected, the system will check for revised versions of all Onshape models in the current GibbsCAM part when you attempt to post.

Update Check

Queries the Onshape server for any fresh versions of any Onshape models in the current GibbsCAM part.
If all Onshape models are up-to-date, the system replies, "There are no updates for this file."
If one or models are not up-to-date, the following dialog appears:

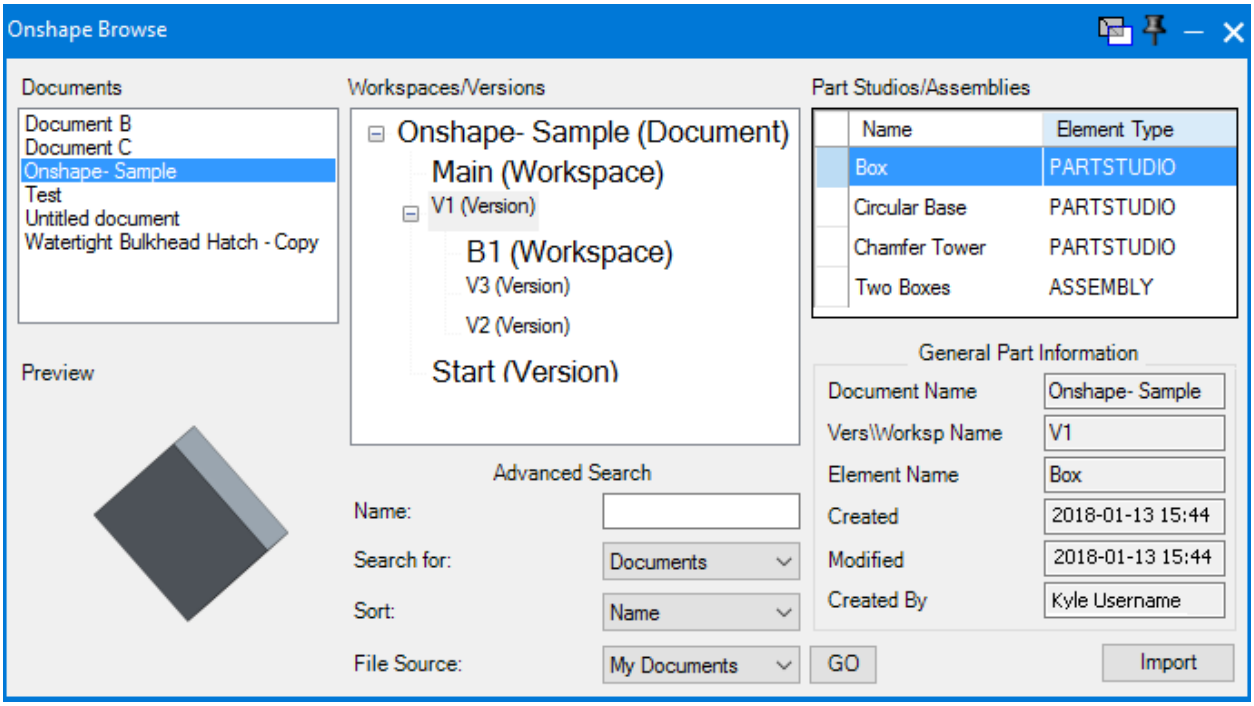


Onshape **Update Check** Dialog
(when a newer version of an
Onshape model is found)

At the bottom of the dialog is a list of all files that are not up-to-date. When you select an item, the upper half of the dialog displays preview and general information about the model.
The Workspace column provides a pull-down control that lets you select a different version of the model residing in a different workspace.
To update a single model: Choose the correct workspace and version and then click **Update** button.
To update multiple models: Use the far left column to select each model, you want to update, choose the workspace and version for each selected model, and then click **Update** button.

Browse

Use this dialog to view and import Onshape models.



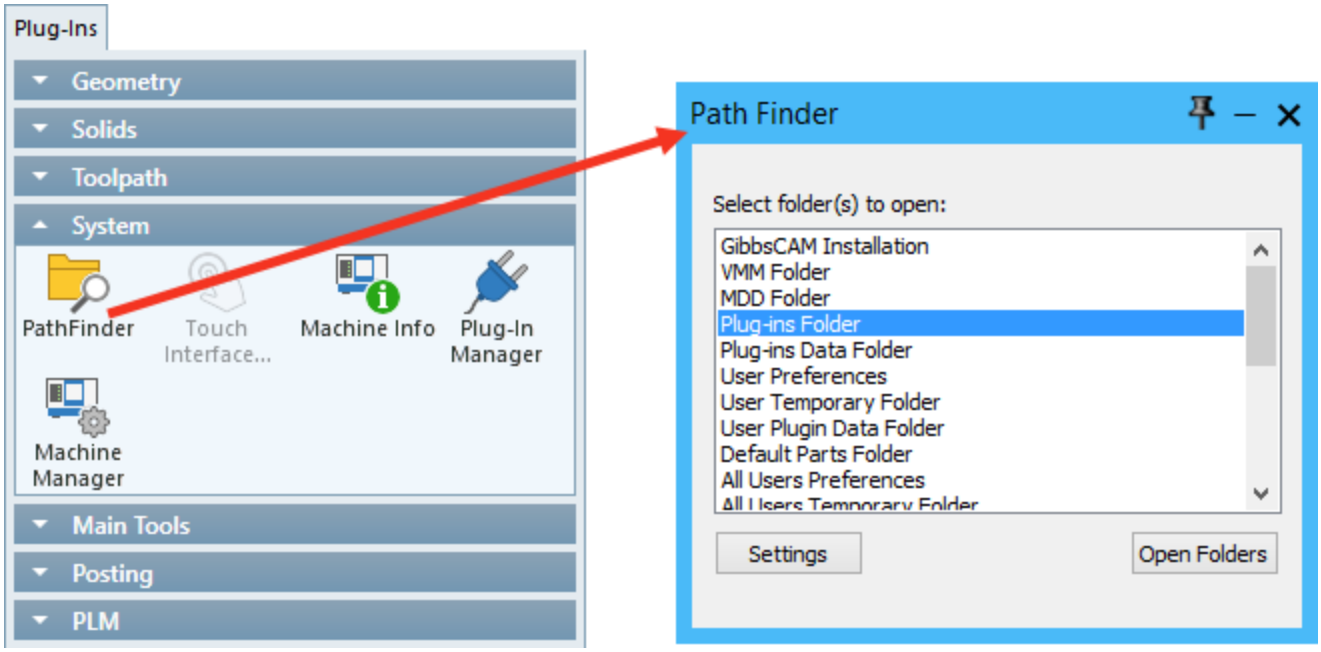
Onshape **Browse** Dialog

In the screenshot above, when File Source is set to My Documents, five documents are found. When document "Onshape - Sample" is selected, the tree showed that the Main workspace contains version V1 and the B1 workspace contains versions V3 and V2. When the V1 version is selected, the three parts and one assembly. When the "Box" part is selected, the dialog displays a preview and general information on the model.

Workflow

1. Use the File Source pull-down to choose a source: My Documents, Created, Shared, Trashed, or Recent.
   Result: In the **Documents** area, a list of files appears.
2. Optionally, use the **Advanced Search** controls to filter and sort results.
3. Select a document from the list in the upper left.
   Result: The **Workspaces/Versions** area in the upper center displays a tree of Onshape workspaces and the versions of this model that they contain.
4. Navigate through the **Workspaces/Versions** tree to find the version you want to inspect or import.
   Result: The **Part Studios/Assemblies** area in the upper right displays a list of models in the selected workspace.
5. **To inspect a model:** Click on a model name in the list to see a preview and general information about the model.
6. **To import a model:** Click the cell to the left of the model name, and then click the **Import** button.
   Result: The selected Onshapemodels are imported into the current GibbsCAM part. The GibbsCAM actions that will trigger their updating are governed by the checkboxes selected in the **Settings** dialog.

Pathfinder

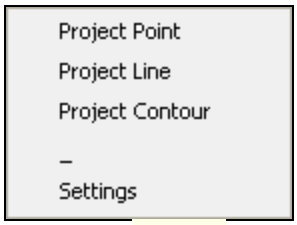


The Pathfinder is actually a collection of shortcuts to folders where important system and user data is stored. There are many places where various types of data is stored that you may need to access, particularly if you are getting help from Technical Support. Most of these items are in the application's installation directory, but some are not. Simply select an item and the corresponding folder will open.

Pinch Turning

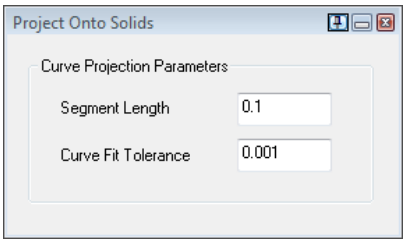
This plug-in was retired at version 12.0, when its functionality was incorporated into base turning processes.
Pinch Turning lets you rough a part on a twin turret lathe using two tools simultaneously. This can reduce cycle times and can provide support for a long part away from the chuck. Both tools begin with each stroke together, with an optional lag distance between turrets. The second cut can finish sooner than the first, depending on the length of cuts.
To use Pinch Turning in MTM at GibbsCAM MTM 2027 and later, create a new turning process on a subspindle, choose either Pinch Contour or Pinch Rough, and designate the master process and a value for Lead/Lag. For complete information on conditions required for pinch turning, see the *Turning* guide.

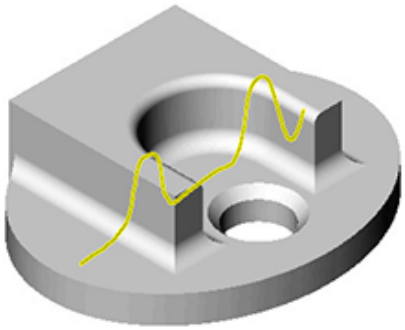
Project Onto Solids



The Project Onto Solids Plug-in will project points, lines, and contour shapes onto bodies, modifying their depth and/or shape as needed. Note that the resulting geometry will lie on the first surface that is encountered on a body. Lines will be converted into b-splines while contours will become segmented.

Selecting Settings will bring up the dialog shown below. The value of the Segment Length field determines the size of the line segments to be produced, and the Curve Fit Tolerance sets the tolerance to be used in segmenting the lines.

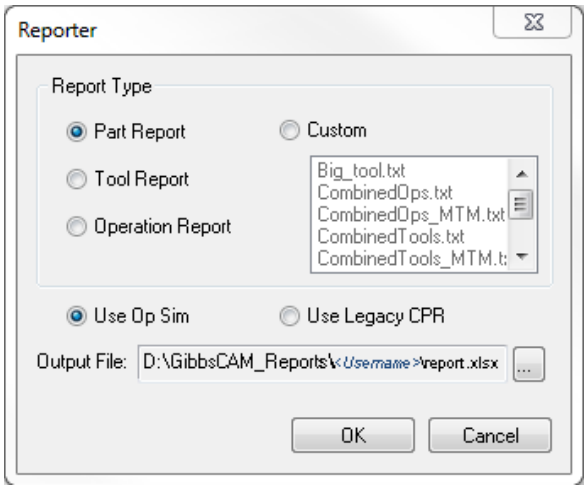




Reporter Basics

Reporter is used to generate predefined or custom reports from the data in the current part in Excel. You do not need to have Excel installed to use Reporter.
Three predefined reports are provided:

- “Part Report” on page 68
- “Tool Report” on page 69
- “Operation Report” on page 70



In addition to the three basic reports, you can create your own custom templates. Several pre-made custom reports are available. Please note that reports tend to be specific to machine type. For example, the “Big_tool” custom report is for Mill machines only and will create invalid output if used for a turning or broaching part.

The default rendering mode is Op Simulation. If it better suits the needs of your report or if speed is an issue, you can select the checkbox Use Legacy CPR to override the default rendering mode and render the part in CPR instead. This can be significantly faster when generating a large report.

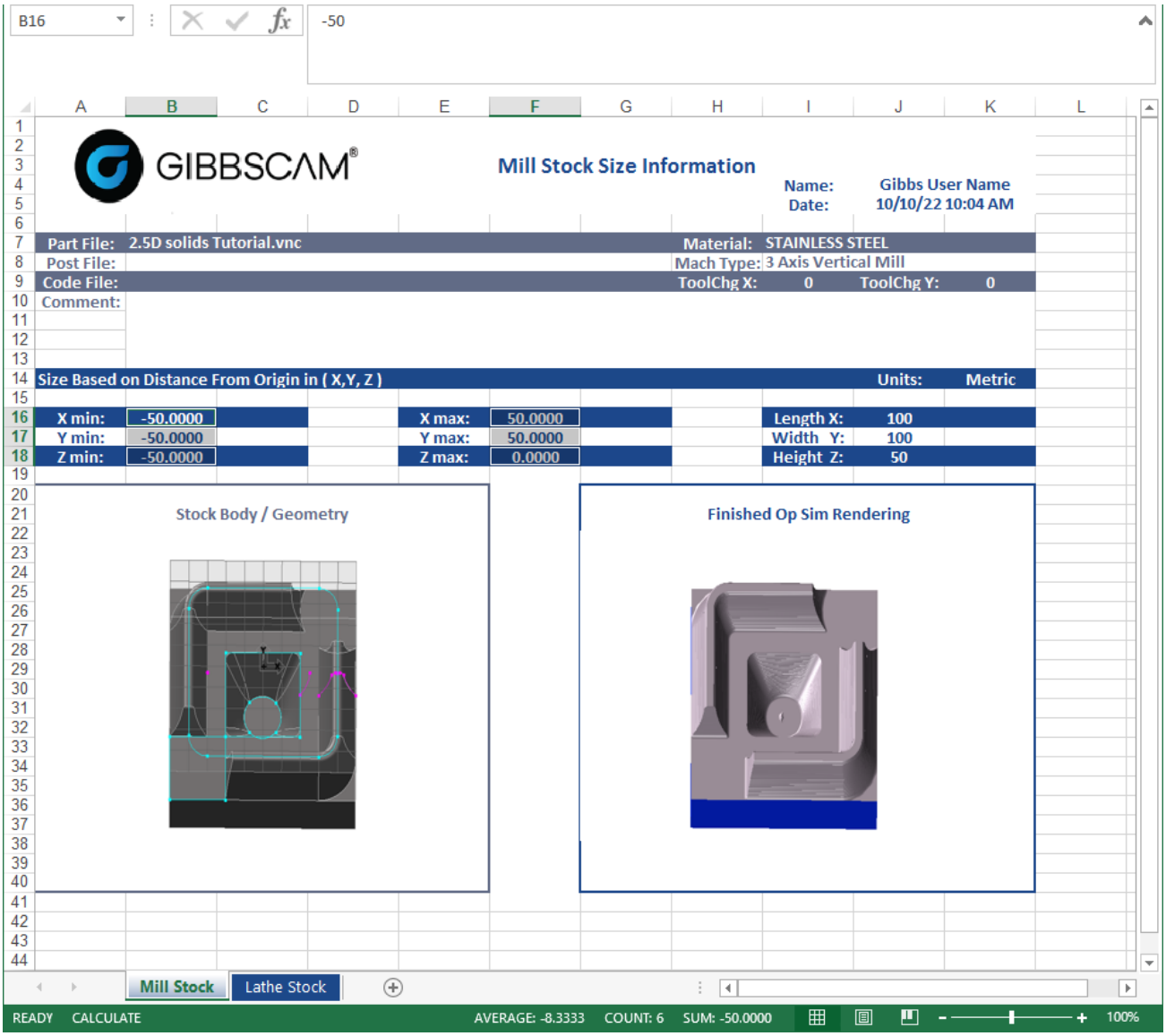
To generate a report, open a part and, on the **Plug-ins** menu, click **Reporter**. Select the type of report to generate, and then click OK. This will launch Excel and create the report. Creating Part and Tool Reports is a very quick process. Generating an Operation Report will cause the part to be fully rendered before the report is generated (the report captures an image of the finished part for each operation). When the report is complete, the Excel file can be named, saved, and printed for record-keeping.

This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

For full details on the standard reports, as well as using, editing, and customizing reports, see the Reporter guide.

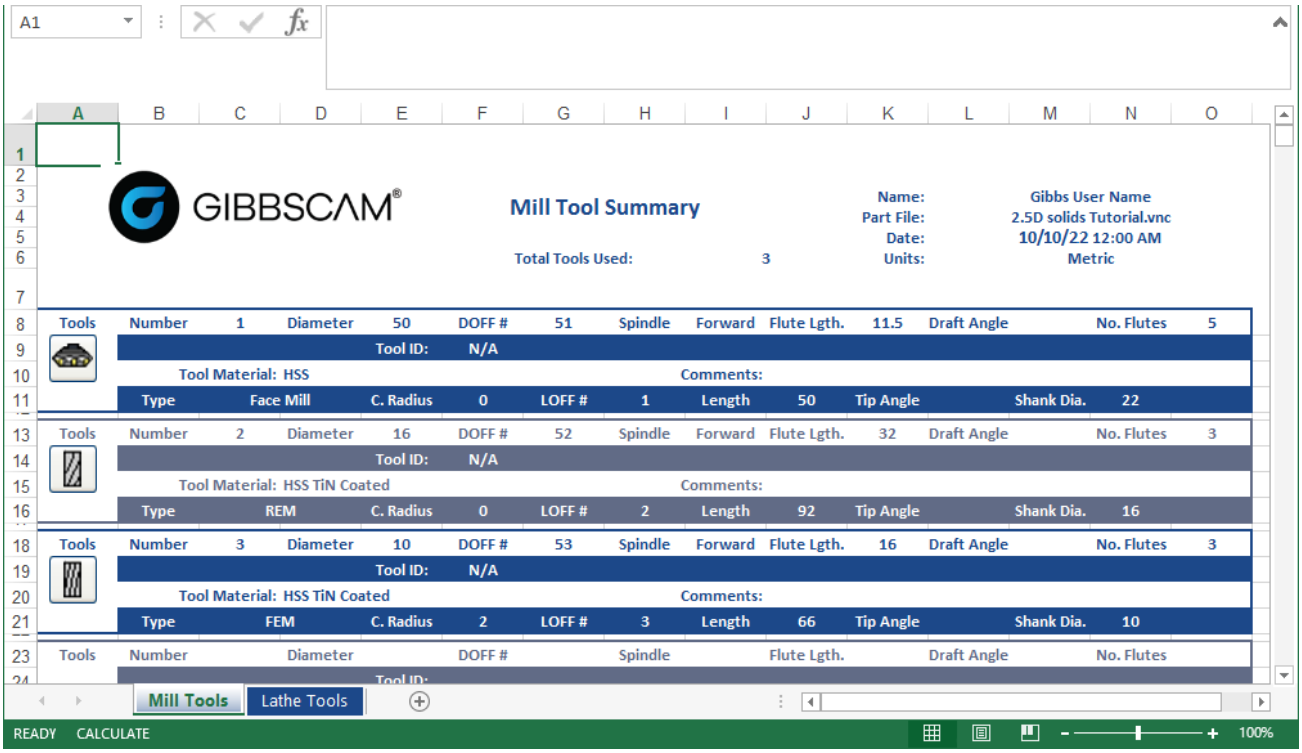
Part Report

The Part Report is an overview of the current part file and provides basic information about the part. The standard Part Report includes the user’s name, the current date, the saved name of the part file, the type of machine on which the part is programmed, the part material, the name of the Post Processor used on the part, and the name of the saved NCF file for the part. Additionally, the report contains the dimensions of the stock, an image of the part geometry or solid the part is created from, and an image of the final rendered part. Note that the report uses an image of the last item rendered. It is recommended that cut part rendering be run before generating the Part Report. This will ensure that the proper rendered image is displayed.



Tool Report

The Tool Report is an overview of the tools in the Tool List of the current part file. The standard Tool Report includes the user's name, the current date, the saved name of the part file, and the part's units of measurement. Additionally, the report contains details about each tool, including a graphic of the tool, the tool type/number/size, tool material, CRC number, spindle direction, the number of flutes, and any tool comments.



GIBBSCAM®

Mill Tool Summary

Total Tools Used: 3

Name: Gibbs User Name
Part File: 2.5D solids Tutorial.vnc
Date: 10/10/22 12:00 AM
Units: Metric

Tools	Number	1	Diameter	50	DOFF #	51	Spindle	Forward	Flute Lgth.	11.5	Draft Angle		No. Flutes	5
				Tool ID:	N/A									
	Tool Material:	HSS					Comments:							
	Type	Face Mill	C. Radius	0	LOFF #	1	Length	50	Tip Angle			Shank Dia.	22	
Tools	Number	2	Diameter	16	DOFF #	52	Spindle	Forward	Flute Lgth.	32	Draft Angle		No. Flutes	3
				Tool ID:	N/A									
	Tool Material:	HSS TiN Coated					Comments:							
	Type	REM	C. Radius	0	LOFF #	2	Length	92	Tip Angle			Shank Dia.	16	
Tools	Number	3	Diameter	10	DOFF #	53	Spindle	Forward	Flute Lgth.	16	Draft Angle		No. Flutes	3
				Tool ID:	N/A									
	Tool Material:	HSS TiN Coated					Comments:							
	Type	FEM	C. Radius	2	LOFF #	3	Length	66	Tip Angle			Shank Dia.	10	
Tools	Number		Diameter		DOFF #		Spindle		Flute Lgth.		Draft Angle		No. Flutes	
				Tool ID:										

Operation Report

The Operation Report is a detailed summary of the operations used to create the part. Each operation in the part is fully described, including the starting and ending condition of the stock for each operation. The standard Operation Report includes the user's name, the current date, the saved name of the part file, the calculated cut time, and the part's units of measurement. Additionally, the report contains tool information for each operation, the operation type (Roughing, Lace Cut, Contouring, etc.), the amount of stock left by the operation, feedrates, depth of cut, the number of cuts taken, cut times, and more. Please note that an Operation Report can take up to several minutes to generate if there are many operations in the part. This applies to active operations only; inactive operations are unaffected. For more information about active and inactive operations, see *Common Reference*, chapter "Miscellaneous", section "Lists", subsection "Active and Inactive Operations".

GIBBSCAM®

Mill Operation Summary

Total Run Time: 0:21:59

Name:	Gibbs User Name
Part File:	2.5D solids Tutorial.vnc
Date:	10/10/22 9:59 AM
Units:	Metric

Start Condition								End Condition
CRC	Off	Workgroup	1	Coordinate	1	Coolant	On	
Operation	1	Group #	2	Entry Feed	1478	Tool Dia.	50	
Tool #	1	Cut Depth	-5	Con. Feed	1478	Op. Type	Rough	
DOFF #	51	Step Depth	5	RPM	1164	Tool Type	FaceMill	
LOFF #	1	Surf. Stock		TP Length	619.5000	Time	0:00:25	
# Cuts	1	Poc. Stock	2	Isl. Stock	2	Utility Data	Unused	

Start Condition								End Condition
CRC	Off	Workgroup	1	Coordinate	1	Coolant	On	
Operation	2	Group #	4	Entry Feed	763	Tool Dia.	16	
Tool #	2	Cut Depth	-45	Con. Feed	763	Op. Type	Rough	
DOFF #	52	Step Depth	6.666667	RPM	1668	Tool Type	REM	
LOFF #	2	Surf. Stock		TP Length	#########	Time	0:03:32	
# Cuts	1	Poc. Stock	2	Isl. Stock	2	Utility Data	Unused	

Start Condition								End Condition
CRC	Off	Workgroup	1	Coordinate	1	Coolant	On	
Operation	3	Group #	4	Entry Feed	763	Tool Dia.	16	
Tool #	2	Cut Depth	-45	Con. Feed	763	Op. Type	Rough	
DOFF #	52	Step Depth	6.666667	RPM	1668	Tool Type	REM	
LOFF #	2	Surf. Stock		TP Length	466.3290	Time	0:00:37	
# Cuts	1	Poc. Stock	2	Isl. Stock	2	Utility Data	Unused	

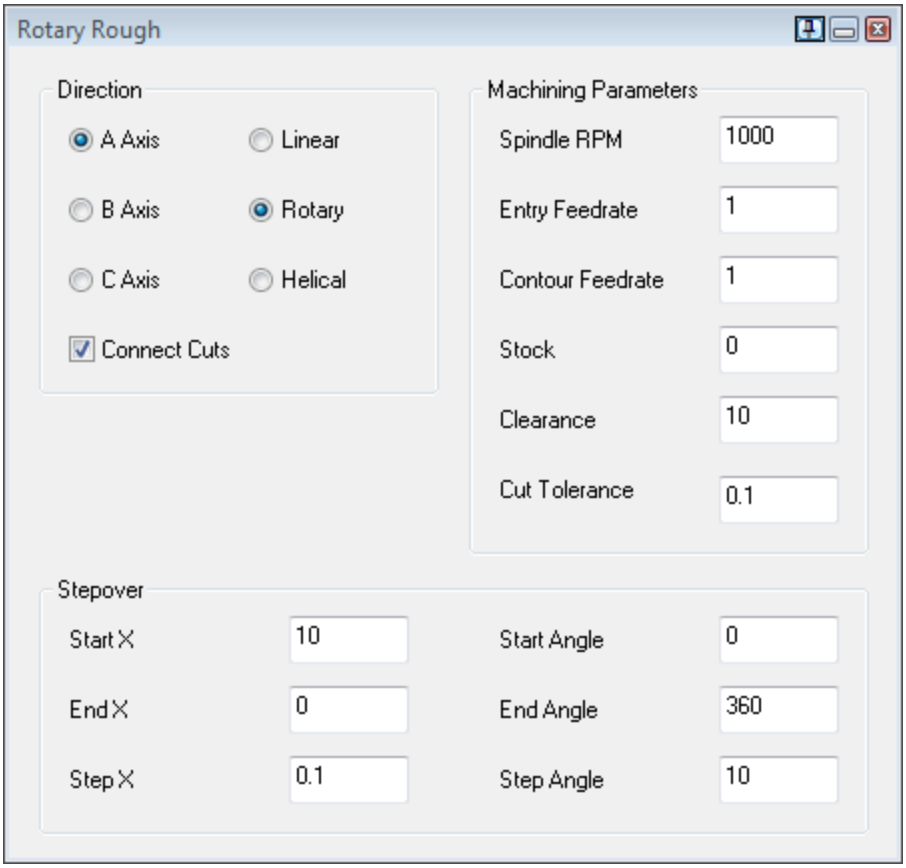
Start Condition								End Condition
CRC	On	Workgroup	1	Coordinate	1	Coolant	On	
Operation	4	Group #	6	Entry Feed	1183	Tool Dia.	10	
Tool #	3	Cut Depth	-25	Con. Feed	1183	Op. Type	Contour	
DOFF #	53	Step Depth	0.689655	RPM	7762	Tool Type	FEM	
LOFF #	3	Surf. Stock		TP Length	#########	Time	0:07:29	
# Cuts	1	Poc. Stock	0	Isl. Stock		Utility Data	Unused	

Start Condition								End Condition
CRC	On	Workgroup	1	Coordinate	1	Coolant	On	
Operation	5	Group #	6	Entry Feed	1183	Tool Dia.	10	
Tool #	3	Cut Depth	-25	Con. Feed	1183	Op. Type	Contour	
DOFF #	53	Step Depth	0.689655	RPM	7762	Tool Type	FEM	
LOFF #	3	Surf. Stock		TP Length	#########	Time	0:07:29	
# Cuts	1	Poc. Stock	0	Isl. Stock		Utility Data	Unused	

Mill Ops

Rotary Rough

This plug-in is designed to work with solids in Mill/Turn or Advanced CS. The plug-in can cut asymmetrical solids and create three types of toolpaths – Linear, Rotary and Helical – along all axes. Activating this plug-in creates a process tile. Dropping a tool on the process tile opens a process dialog where you can set the operation parameters.



Direction Settings

Direction
Axis of rotation (A, B or C) is determined by these radio buttons.
Linear
Cuts along the linear axis of rotation with indexing (measured in degrees) between each cut around the axis of rotation.
Rotary
Cuts around the axis of rotation, keeping the tool normal to the surface, with a step over in the direction of the linear axis rotation between each cut.
Helical
Cuts in a continuous helical movement around the axis of rotation.

Stepover Settings

Start X, Y or Z
This is the cut start location on the X, Y or Z axis in part units.
End X,Y or Z
This is the cut end location on the X, Y or Z axis in part units.
Step X, Y or Z
Specifies the distance the tool will move over while roughing. This is the distance between rotary cuts, the pitch of helical cuts or the segmentation distance of linear cuts. This distance should be

less than or equal to the tool radius.

Start Angle
The angle of the starting location measured in degrees around the rotary axis.

End Angle
The angle of the ending location measured in degrees around the rotary axis.

Step Angle
Angular stepover around the X, Y or Z axis. This is the segmentation angle for rotary and helical cuts and will define the angular rotation between each cut for linear cuts.

Machining Parameters

Spindle RPM
Specifies the rotation speed of the spindle in revolutions per minute.

Entry Feedrate
Specifies the entry feedrate in millimeters per minute or inches per minute.

Contour Feedrate:
Specifies the contour feedrate in millimeters per minute or inches per minute.

Stock
Specifies the thickness of material left on the part. This value is in part units.

Clearance
The incremental distance measured up from the finish cut depth.

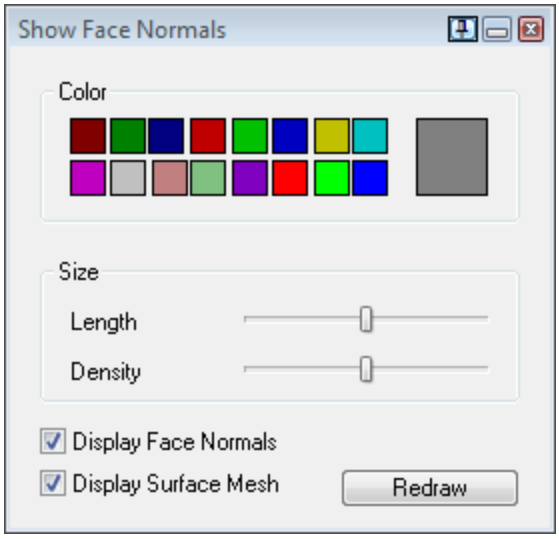
Cut Tolerance
Set the accuracy of the toolpath along selected geometry. This value is in part units.

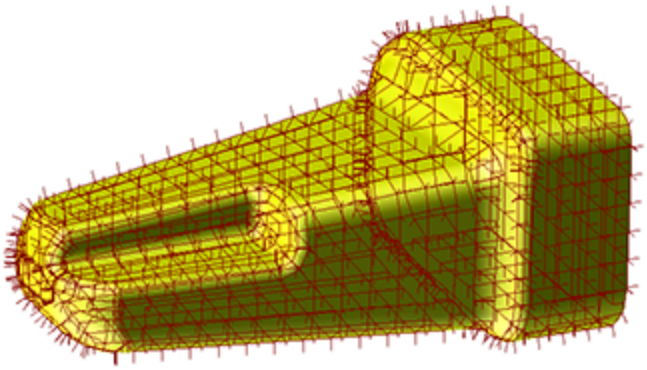
Set Up Post Editor

The Set Up Post Editor plug-in was retired at version 12.0, when its functionality was moved into the **Preferences** dialog as a new tab: Post Editor Settings. For information on the **Preferences** dialog, see the *Common Reference* guide.

Show Surface Normals

This Plug-in displays Surface normals across selected faces to highlight the curvature across each face as shown in the image below. The length and density (number of vectors) of the normals are defined by the user.



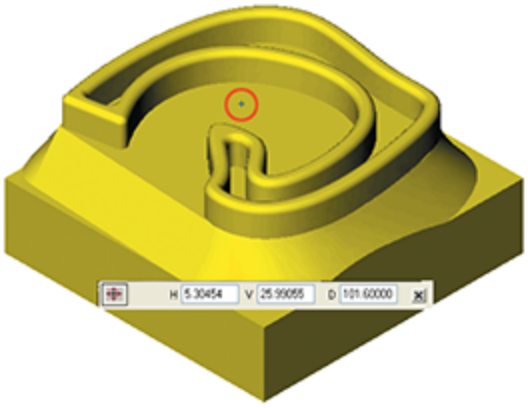


Color
Select the color off Face Normals and/or Surface Mesh.
Length
Select the length of Face Normals by slider.
Density
Select the density of Face Normals by slider.
Display Face Normals
Face Normals are displayed when checked.
Display Surface Mesh
Surface Mesh is displayed when checked.

Show Position

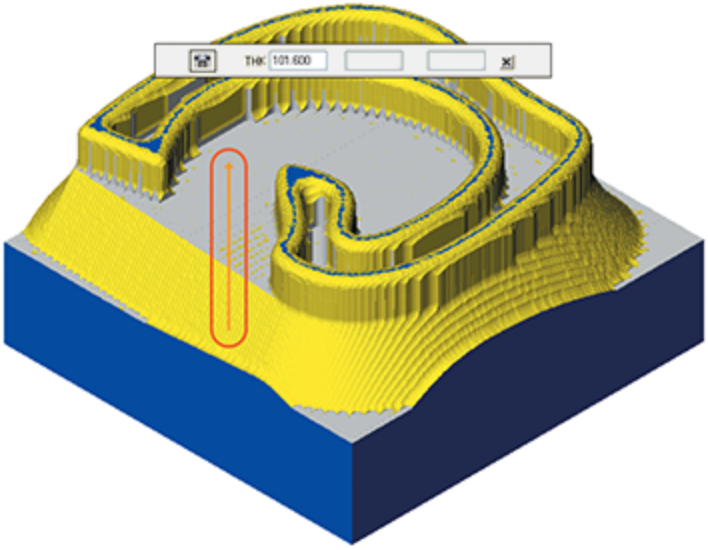
Show Position provides positional data on solids as well as rendered parts. The data may be the exact XYZ position of the cross hair on the solid, the position of the tool tip during rendering, or the position of the marker once Cut Part Rendering (CPR) is complete. Additionally, this plug-in can display the depth of the solid/stock from a selected point to the opposite side of the solid/stock or the curvature of a specific point on the solid/stock. The information gathered is displayed in a floating palette that can be moved anywhere on the screen. Except while Cut Part Rendering is playing, you can switch between the modes by clicking the icon on the left side of the dialog.
You can copy data from the Show Position plug-in. To copy the data, click in the desired field, select the data, and then right-click to bring up the context menu. Choose Copy to save the data to the clipboard. The data may then be pasted into any other text entry box.
Surface Coordinate

This tool can be used to display the XYZ coordinates of the any point on a solid. The Show Position plug-in can be switched to the Surface Coordinate mode when Cut Part Rendering is not playing but is currently active; it is generally used when Cut Part Rendering is complete. The Show Position dialog will also display the XYZ coordinates of a mouse click a rendered part. Additionally, the location of the click will be marked by a cross-hair until another point is selected or when Cut Part Rendering is replayed.

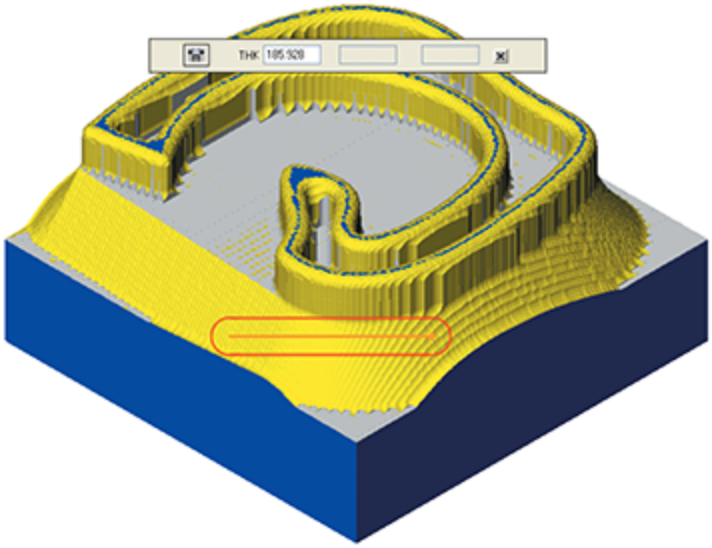


Depth of Solid/Stock

The Show Position Plug-in can be switched to the Depth of Solid/Stock mode when Cut Part Rendering is not playing but currently active. Clicking (or right-clicking in Cut Part Rendering) will determine the depth of the solid or stock directly perpendicular to the clicked spot. This perpendicular position is referred to as the surface normal.



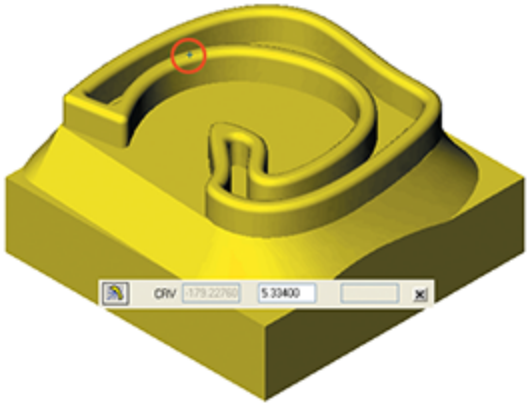
In Cut Part Rendering, the depth of the part can be determined from both cut areas and stock that has not been cut. “ERR” will be displayed in the text box if you click off the part. Determining the depth of the stock from the surface normal of an area cut by a flat tool is fairly straightforward. When selecting an area cut by a ball endmill, this process can become more difficult, because the system will likely be selecting an area on a ridge. The depth of the part is determined from the surface normal of the point on the ridge. Therefore, the system will be determining the depth of the part at an angle, as shown in the following image.



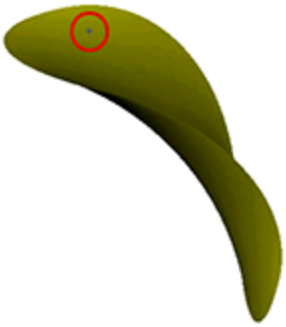
An example of a surface normal from a ridge

Curvature

This tool is used to measure the curvature of a specific point on a solid. Clicking on a solid will determine the two principle curvatures of the selected point.



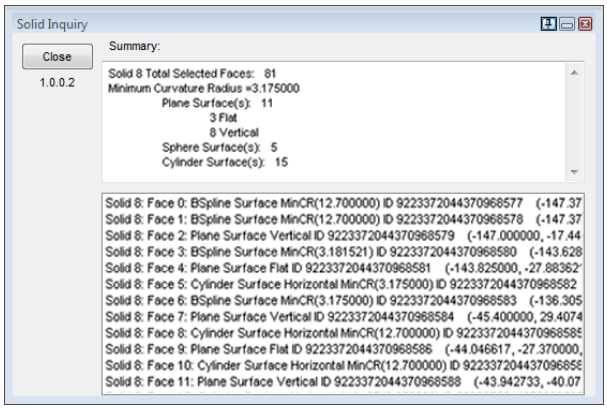
In the example below to the left, a mouse-click is performed on a cylinder with a 1” radius while in Curvature mode. One of the principal curvatures of this solid runs along the radius of the cylinder. The other shows up as “+INF” because the other principal curvature, which runs along the height of the cylinder, is flat. In the example to the right, the positive number is the curvature along the convex side of the surface, and the negative number is the curvature along the concave side of the surface.



Solid Edge Options

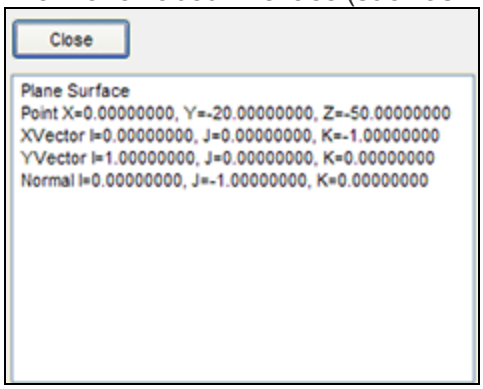
The Solid Edge Options plug-in was retired at version 12.0, as part of the streamlining of the Import process.

Solid Inquiry



The Solid Inquiry Plug-in provides information about the faces of a selected body. Select a body and run the plug-in to activate the Solid Inquiry dialog shown at the right. The left side of the dialog features a Close button to close the dialog.

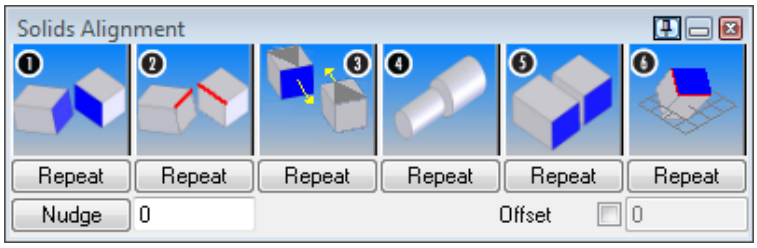
The Summary text box displays a listing of the faces of the body, grouped by type. The text box below the Summary displays a listing of all the faces of the body, including the type of each face and the dimensions of its bounding box. Double-clicking on any of the items will highlight that particular face as well as open the Face Surface Details dialog which includes more detailed information about the face (such as the face's vectors).



Solids Alignment

The Solids Alignment Plug-in allows you to manipulate the positions of solid models. The positioning is done by rotating/translating the first of the two selected models. The solid that was selected first becomes aligned to a second model. Cylindrical bodies can be aligned by making them concentric. Further functions are provided to align bodies using two edges or two points.

Once two bodies have been aligned, additional models may be selected and aligned using the same transformation, enabling multiple models to be aligned.



1. Align Two Faces
2. Align Two Edges
3. Flip About A Face
4. Move/Rotate
5. Face-To-Face Mate
6. Align With CS

Align Two Faces
Select two models by picking one face on each. The first model is rotated to align the two faces so that the faces are parallel.

Align Two Edges
Select two models by picking one edge on each. The first model is rotated and translated so that the two edges become parallel.

Flip About A Face
This item is used to flip a body about the selected face. Select one face (it must be a flat face) and the body is flipped around so that the face normal for the selected face points in the opposite direction.

Move/Rotate
If you only select one solid (instead of 2), then the solid is moved and/or rotated so that the center of the selected face (you must pick a single cylindrical face) lies along the Horizontal axis of the current CS. The solid is then translated so that it's maximum H(X) value is zero. This is designed to enable a solid imported into turning to be positioned to lie along the Z axis, with the front at Z zero, by picking just one cylindrical face.
If you select two solids, the behavior is slightly different. Select two models by picking a cylindrical face on each model. The second model is rotated and translated so that the two cylindrical faces share the same axis of rotation. Please note that the second cylinder may be placed inside the first. By checking the Offset box and giving an offset value, the second model will be translated along the axis of the cylinders by the offset value.

Face-to-Face Mate
Select two models by picking one face on each. The face normals for the two faces must be in the same or exact opposite direction (as they would be after using the Align Two Faces command). The first model is translated along the face normal until the two faces are parallel and coincident.

Align With CS
Select one face on one solid and that solid will be moved so that the selected face is aligned with the HV axes of the current CS. The center of the face will lie at the HV origin. If you also pick one edge on the selected face, then once positioned, the solid is rotated so that the selected edge lies along the H axis. If you pick two edges on the selected face (which must be at 90° to one another), then, once positioned, the solid is rotated so that the first selected edge lies along the H axis and the second edge lies along the V axis.

Repeat
The six Repeat buttons are used to repeat the command associated with the icon above the button. The repeat buttons can be used with multiple models. For example, there may be a group of ten models that all need to be moved together. Move any one of the models first, using one of the alignment commands. After the first alignment, select the other nine models and use the

repeat function to apply the same translation/rotation so that all ten models maintain their position relative to each other.

Nudge

The Nudge button is designed for use after a Face-to-Face Mate command or a Cylindrical alignment. The first model will be moved by the nudge distance along the direction of the face normal so that the faces are no longer coincident but still remain parallel.

Offset

Once aligned, one of the models can be translated in the direction normal to the selected face so that the two faces become parallel. The two models can also be moved apart by a given distance.

up2parts AutoCAM

The up2parts AutoCAM plug-in lets you import up2parts *.zip files and use the results to generate GibbsCAM operations, toolpath, simulation, and G-Code.

Workflow: On the up2parts Website

The following steps can be done by anyone familiar with up2parts, not necessarily a GibbsCAM user.

1. The user supplies credentials to the up2parts website and then uses up2parts to:
   a. Specify a solid model in STEP format. Note that AP242 includes PMI and tolerance information, but AP203 and AP214 do not.
   b. Start the up2parts geometry analysis and AI-based feature recognition.
   c. Get a technical description that includes recognized features like pockets, holes, countersinks, and surfaces.
4. Then, up2parts calculation is used to generate a work plan by defining the machine, material specification, and general tolerance (ISO-2768).
5. Next, up2parts proposes a CAM project for six-sided machining that includes “clamping” (approach from top/bottom/left/right/front/back), tools, machining steps, and operations. The up2parts interface allows adjustments and refinements to this proposed project.
6. After the project is confirmed, up2parts bundles its proposals and recommendations into a *.zip file readable by GibbsCAM. This .zip file includes: the solid model in STEP format with complete feature tree; stock information; tooling; features with operations; and meta information for clamping. Associated PMI (product manufacturing information), if any, is preserved.

Workflow: Within GibbsCAM

Before you begin: Set the units in your DCD to match the units of the up2parts bundle (usually mm).

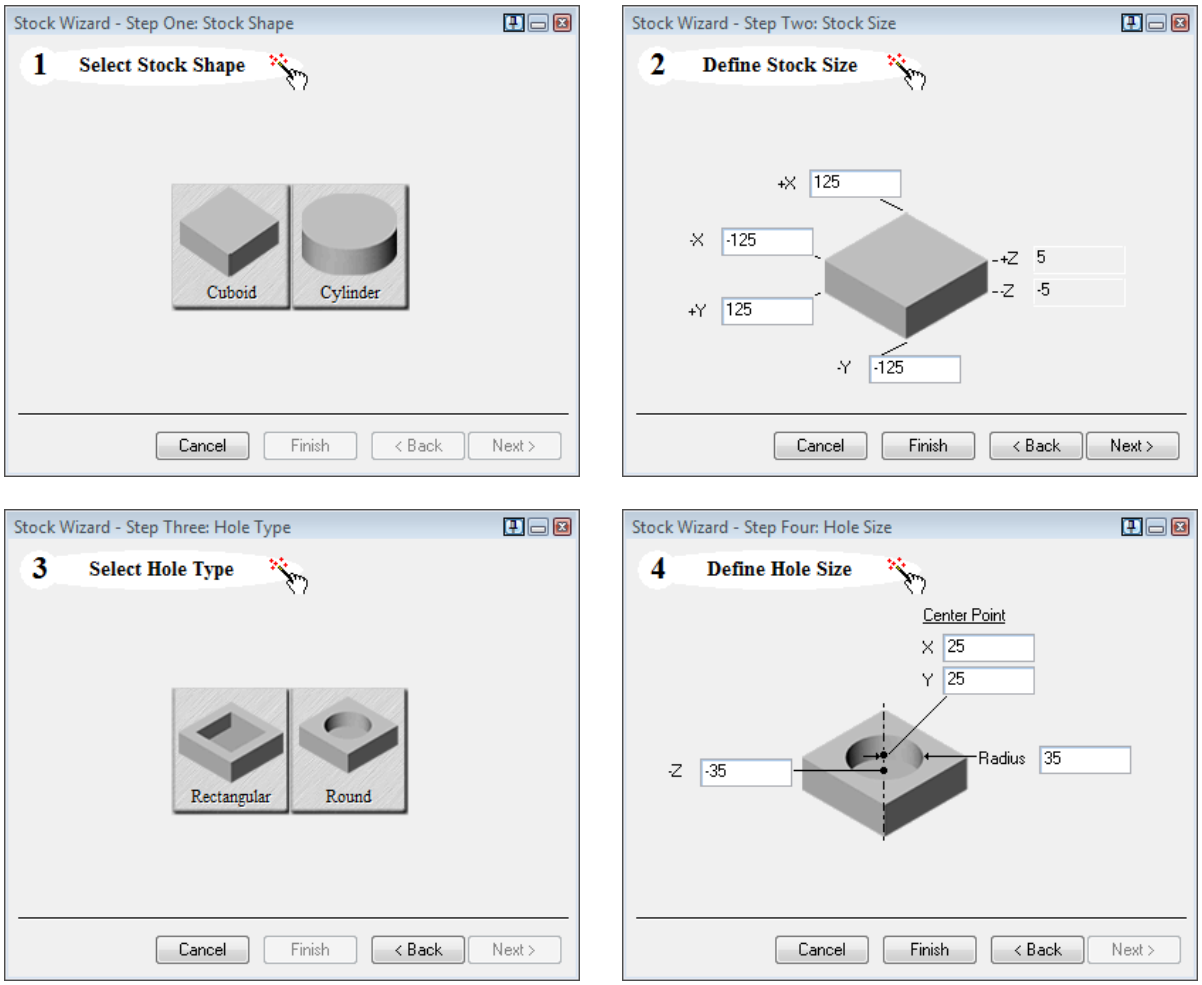
1. On the **Plug-Ins** menu, under **Main Tools**, click up2parts autoCAM.
2. In the **up2parts program arguments** dialog:
   (for Selected file): Supply the *.zip filename of the bundle prepared by up2toparts.
   (for Select clamping): Select a folder to specify approach: top/bottom/left/right/front/back.
   Click Import.
3. Notice that progress is shown by status bar at the bottom of the GibbsCAM window. When the import process is complete, GibbsCAM displays the model in the workspace. Then, watch as features are processed and operations are generated. Finally, in response to the message “The part is loaded and processes/operations have been created”, click OK.
4. You can inspect and optionally modify any aspect of the GibbsCAM model, including tooling, processes, parameter values, and operations. You can then use any of the native GibbsCAM

Simulation options to visualize the machining operations and to find/correct any toolpath issues.

5. Use the appropriate GibbsCAM postprocessor to generate G-Code for your specific machine.

Stock Wizard

The Stock Wizard helps to create a rectangular or cylindrical stock definition with or without a single hole, rather than manually creating it. The hole may be a rectangular or circular shape. The hole in the stock may be a blind hole (a pocket) or a through hole. This is all accomplished in four easy steps.



An example of defining stock with the Stock Wizard

Where Do I Find the Stock Wizard?

The Stock Wizard is found under `Misc` in the `Plug-Ins` menu. Selecting `Stock Wizard` from the pull-down menu will open the Stock Wizard.

How Will the Stock Wizard Help Me?

Typically the stock for a part is defined as a rectangular block in the DCD (Document Control dialog). Custom stock shapes can also be defined using geometry in a workgroup defined as Stock. Using this method you may define nearly any shape as stock. The Stock Wizard simplifies stock creation by automating this process. This eliminates the need for the user to create additional geometry or workgroups in order to set up many custom stock conditions.

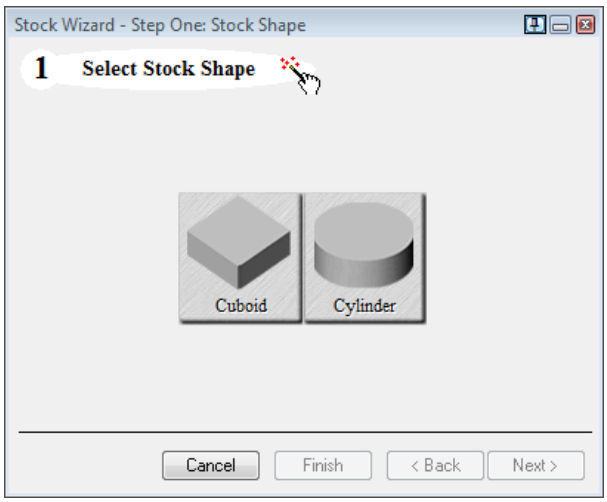
Please Note: The Stock Wizard was not designed to work with spinning Part Stations.

Elements of the Stock Wizard

The elements in the Stock Wizard dialog are quite simple. There are graphics, text boxes, selection buttons and action buttons. Each are described on the following pages.

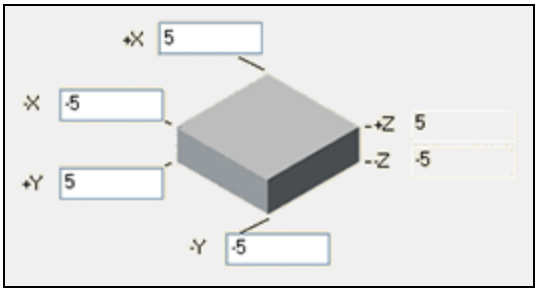
Dialog

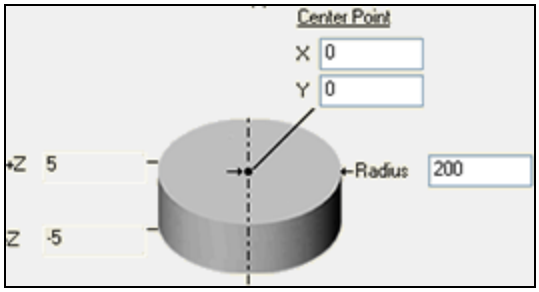
The interface for the Stock Wizard leads you through creating your custom stock very easily. The Stock Wizard is a single dialog box that walks you through four simple steps to define a custom stock with or without an initial hole.

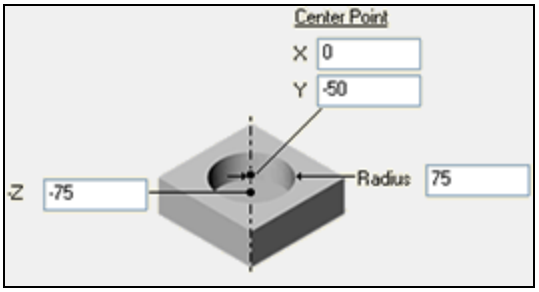


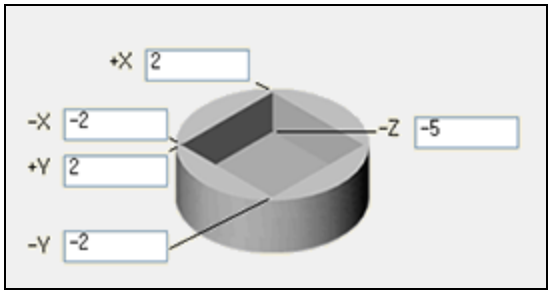
Graphics

The images provide a visual clue as to what your choices are in a dialog. The image will show the shape of the stock and the shape of the hole in the stock if there is one. Some examples are shown below.



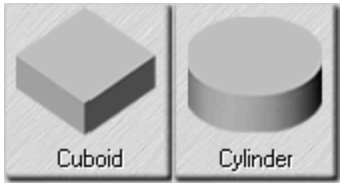




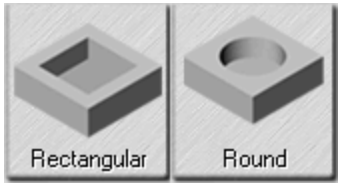


Stock Buttons

These are buttons you will see when choosing a stock shape and the shape of the hole in the stock. The first set of stock buttons you will see will decide the basic shape the stock. The choices are Cuboid and Cylinder. The second set of stock buttons will define the shape of any existing hole in the stock. The hole may be Rectangular or Round. The stock shape does not matter. The choices are Rectangular or Round.



Stock Shape



Hole Shape

Action Buttons

Action buttons help you to navigate through the Stock Wizard.

Cancel

This button will close the current dialog. Any changes you have made to the dialog will not be saved and your stock definition remains unchanged.

Finish

This button will become active at any point at which the dialog has sufficient data to define stock.

Back

This button will bring you to the previous step. This will allow you to make any needed modifications.

Next

The Next button becomes available once the needed data for the current step has been supplied. Clicking the Next button will advance you to the next step.

Using the Stock Wizard

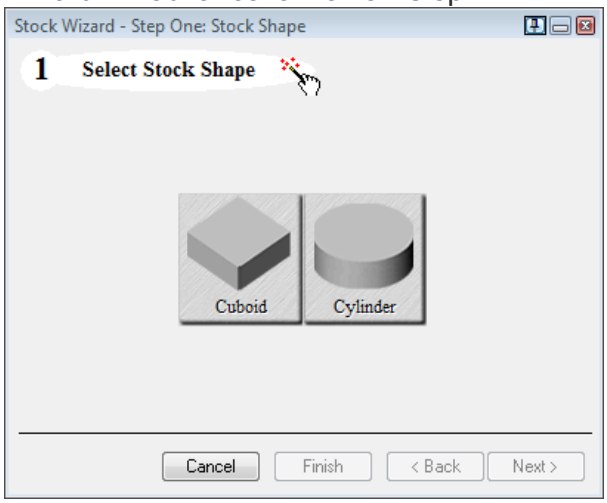
It is typical to define your stock when a part is first made, however, this is not required. You may define custom stock using the Stock Wizard at any time during the part creation process. Any changes made to the stock size in the Stock Wizard will override the Document workspace stock size definition.

The Stock Wizard allows you to visualize the part you are working on and the Stock Wizard will take the stock depth from the Document Control dialog. The measurement unit is also taken from the Document Control dialog. The Stock Wizard determines the stock depth based on the value specified in the Document Control dialog.

Please Note: The Stock Wizard was not designed to work with spinning Part Stations.

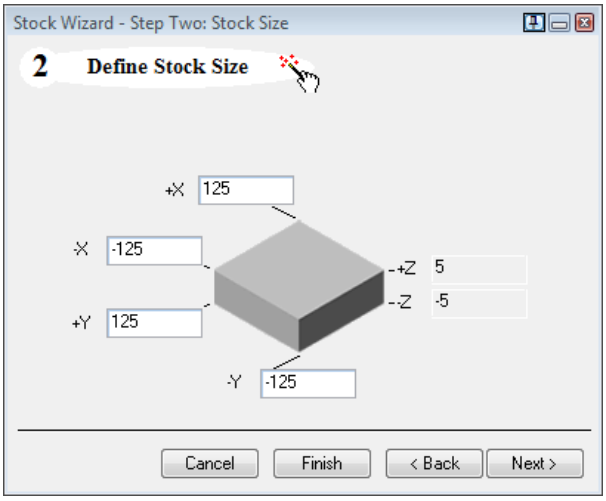
1. Select the shape of the stock.

The two basic stock shapes are Cuboid and Cylinder. When a shape has been selected the Stock Wizard will advance to the next step.



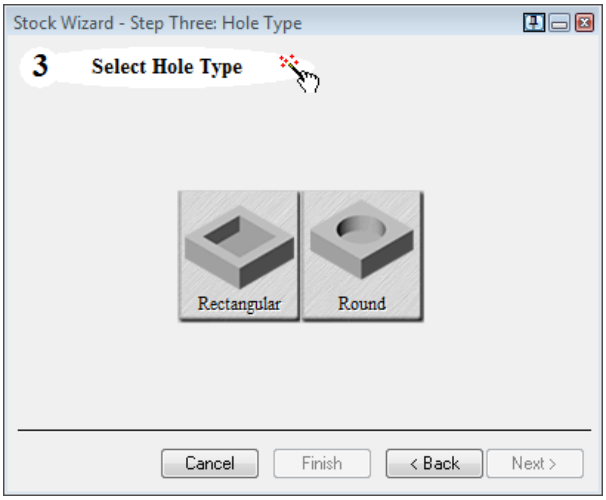
2. Define the size of the stock shape.

The size of the stock will always be equal to or less than the workspace size as defined in the Document Control dialog. If you enter values greater than the workspace the Document Control dialog will automatically be updated to fit the stock. The stock's depth values are defined by the Document Control dialog. If the part does not have a hole in it, you may click Finish and generate the stock, otherwise click the Next button to define a hole.



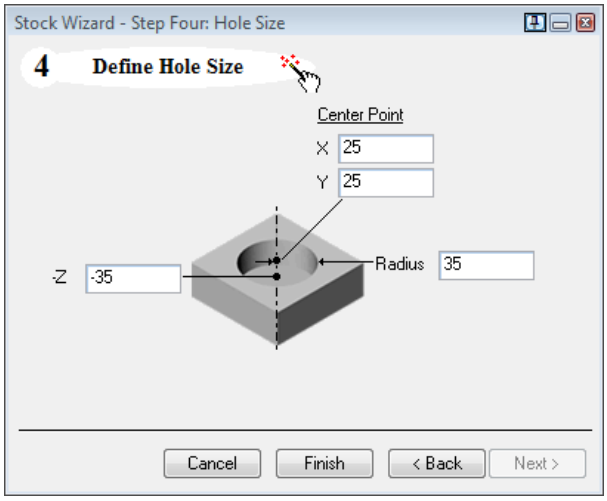
3. Select the type of hole in the stock.

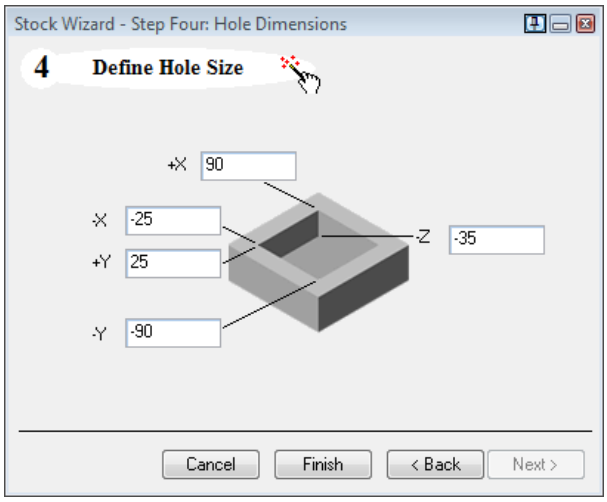
A hole may be Rectangular or Round. When a hole shape is selected the Stock Wizard will advance to the next step.



4. Define the size, location and depth of the hole.

A Circular hole is defined by a Radius, a Center Point and the bottom of the hole, the Z value.





A rectangular hole is defined by its X and Y boundaries and the bottom of the hole, the Z value. A hole does not have to go through the part, blind holes from the top of the part are valid. Simply enter the Z value of the bottom of the hole. The Z value of a through hole does not have to be coincident with the Z value of the stock. Any value below the bottom of the stock is valid. However, holes must always be contained within the X and Y bounds of the stock. If the hole overlaps the stock no hole will be produced. When the size of the hole has been defined, `click` the Finish button and your stock will be updated.

Surface Tools

Surface Tools is a collection of functions that create geometry or surfaces from selected faces of bodies.

Iso Curves

Creates isoparametric curves of the selected face. Those curves can extend to the UV bounding box of the face or be trimmed at the edges of the face.

“ Center Curves” on page 87

Creates a curve located at the center of a face with an additional offset along the face normal.

“ Extend Surfaces” on page 90
Creates a new surface tangent to the selected face and extending by the specified length from the selected edge.
“ Extract Edges” on page 91
Extracts the selected edges by closing automatically the gaps between extracted curves when the gap is smaller than the specified value. Those gaps are typically due to tolerant edges (faces stitched with a non-zero tolerance).

Iso Curves

Select the face (or faces) you want to create isoparametric curves on. If you want to create isoparametric curves passing through a specific location instead of a grid, select also the point or points located at the specific position.

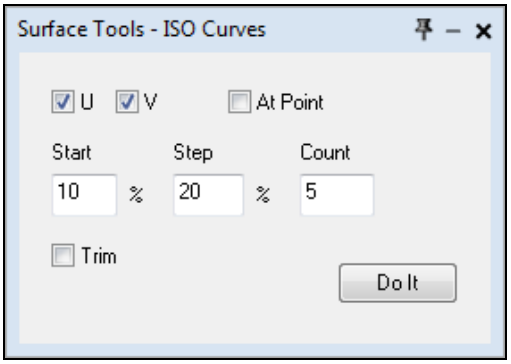
U / V direction
Select the direction or directions in which you want to create the isoparametric curves.

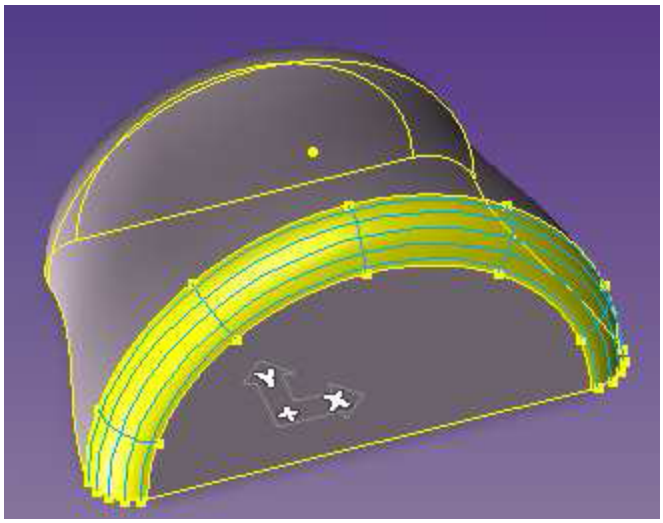
At Point
Select this option to create curves passing through the selected points (the normal projection of the point on the surface).

Curve Definitions
Specify the number of isoparametric curves, the parametric increment between the curves and the position of the first curve, expressed as a percentage value of the whose surface. Make sure you use adequate values so that the isoparametric values of all the curves are strictly between 0 and 1.

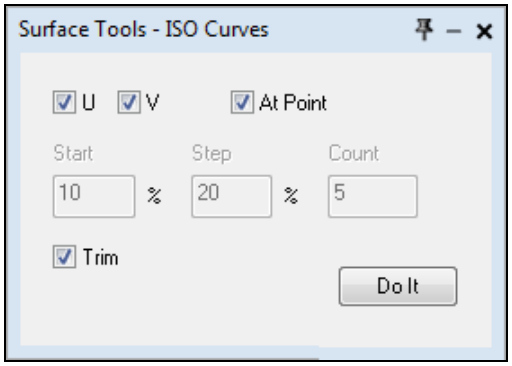
Trim
Use this option to trim the curves to the edges of the face instead of extending to the UV bounding box of the face.

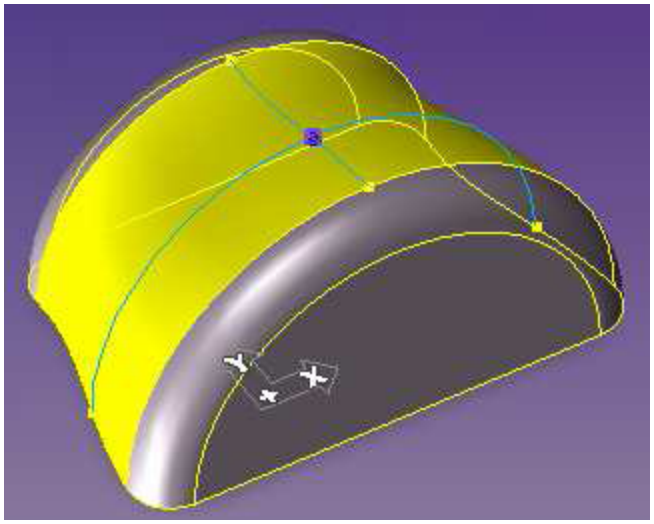
ISO Curves Examples



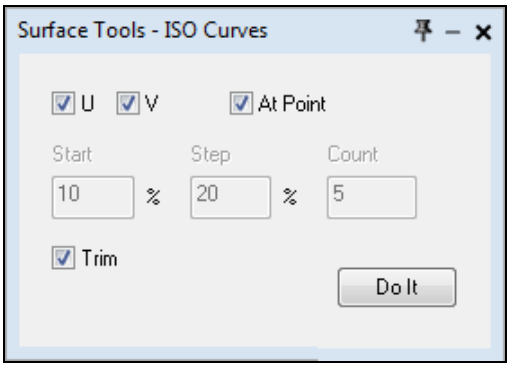


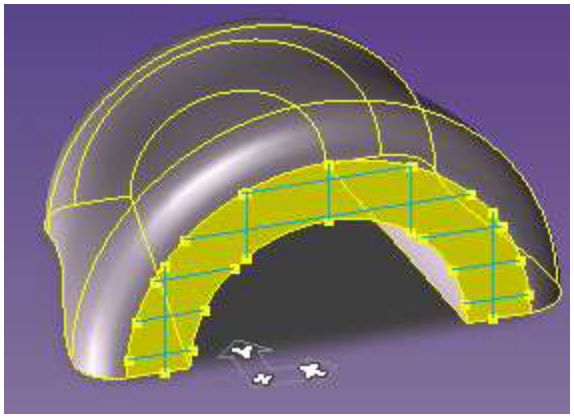
ISO Curves Example #1





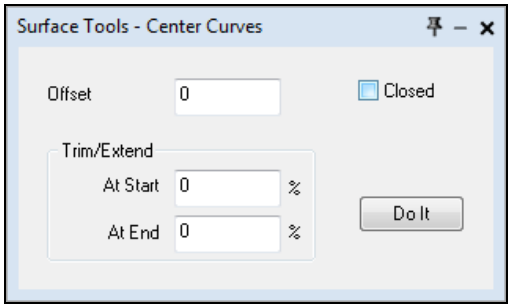
ISO Curves Example #2





ISO Curves Example #3

Center Curves



This function will create a curve that runs along the center of a face or faces.

Selection mode 1

Select one face. The curve will be created along the isoparametric curve at the center of the UV bounding box of the face, along the longest direction.

Selection mode 2
Select starting and ending edge. The curve will be created along the face common to the selected edges from the middle of the first edge to the middle of the second edge. If there are 2 common faces between the edges, select the face you want to use.

Selection mode 3
Select a set of contiguous faces. The curve will be created along the selected faces, going through the middle of the common edges between the faces. You can also select the start edge of the first face and the end edge of the last face to control the start and end location of the curve

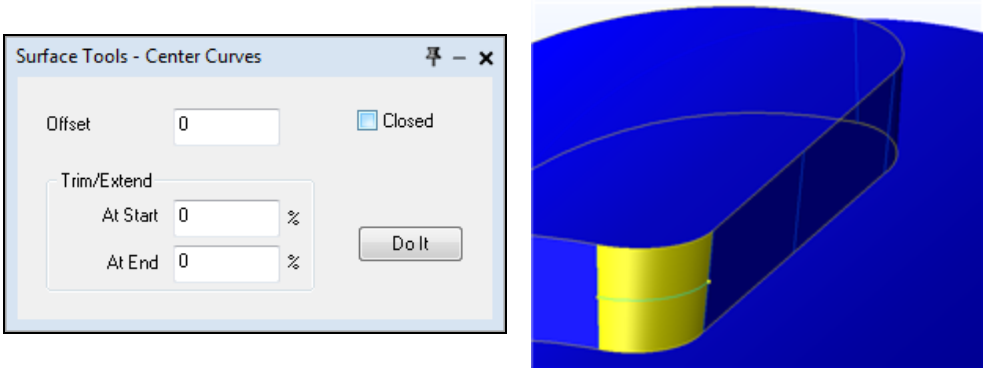
Offset
The offset distance by which the curve will be shifted along the normal vector of the face. The sign will affect the side on which the offset will be calculated. A value of 0.0 will create a curve lying on the face.

Trim/Extend At Start
Use this value to trim or extend the start of the curve. A positive value will trim the curve, and a negative value will extend it. The value is specified as a percentage of the total length of the curve.

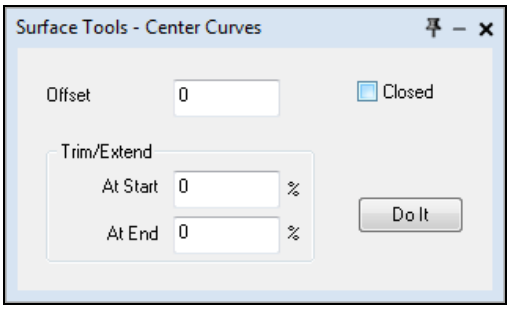
Trim/Extend At End
Use this value to trim or extend the end of the curve. A positive value will trim the curve, and a negative value will extend it. The value is specified as a percentage of the total length of the curve.

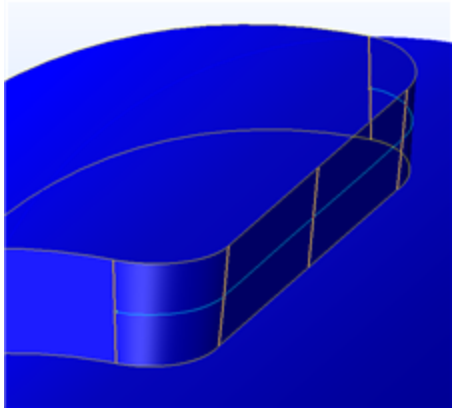
Closed
Check this option to create a closed curve.

Center Curve Examples

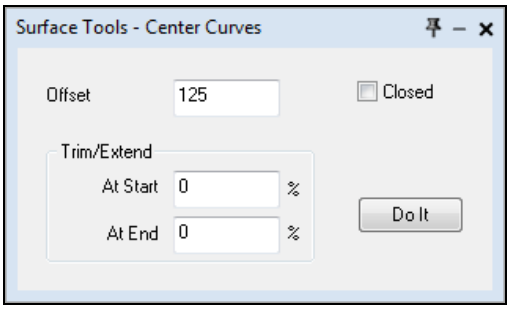


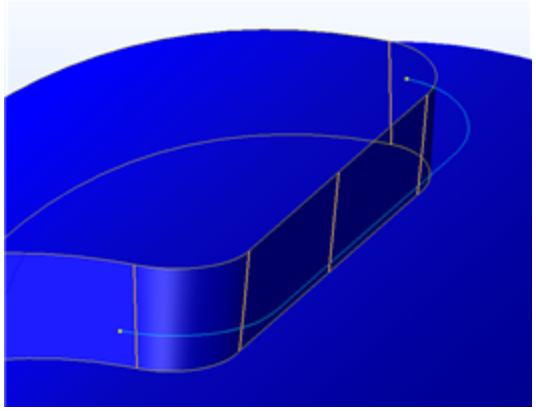
Center Curves Example #1 - Face Selection



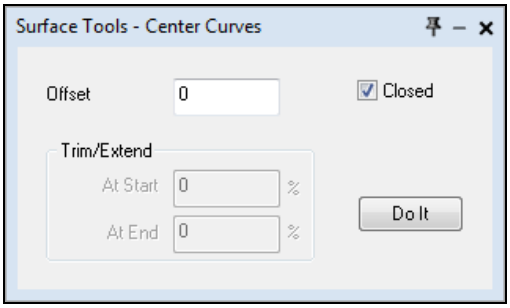


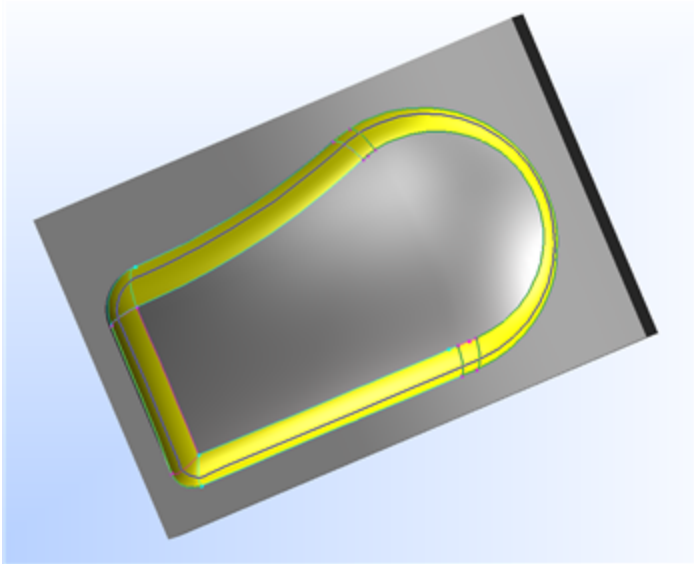
Center Curves Example #2 - Edge Selection





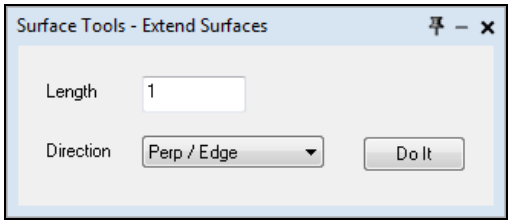
Center Curves Example #3 - Edge Selection + Offset + Extension





Center Curves Example #4 - Multiple Faces Selection

Extend Surfaces



Select the face and the edge or edges corresponding to the side you want to extend. Select the extension length and the direction of the extension.

Length

The length of the extension.

Direction - Perp/Edge

The extension surface will be created normal to the selected edges, and tangent to the face

Direction - Parallel/Side

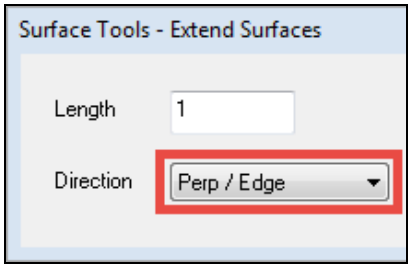
The extension surface will be created by matching the directions of the adjacent edges (the sides) at each end of the selected edge

Direction - ISO Curves

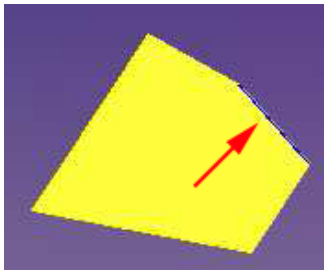
The extension surface will be created along the isoparametric direction of the selected face. The direction closest to the edge normal will be used.

Extend Face Examples

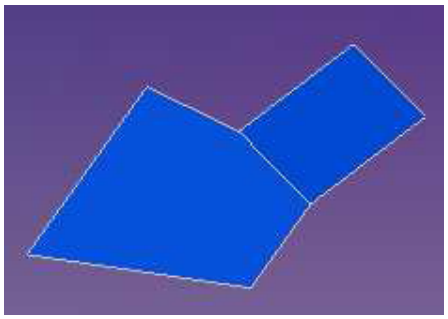
Parameter



Selection

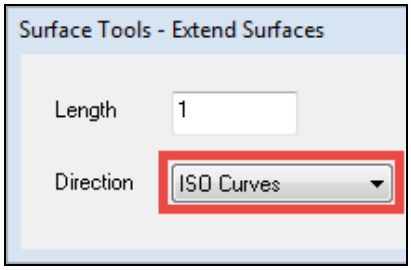


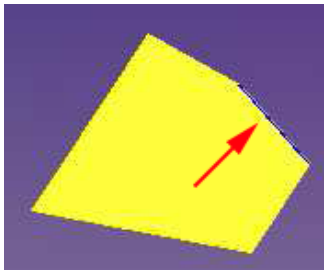
Result

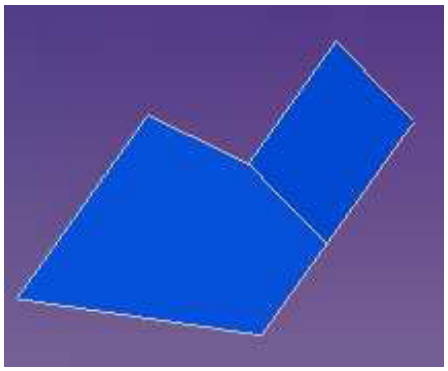


Surface Tools Center Curves Example #1

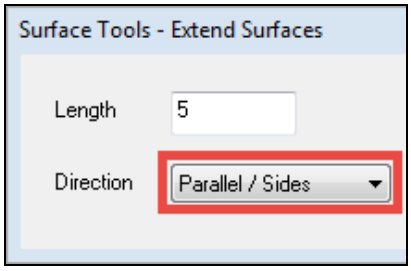
Parameter | Selection | Result

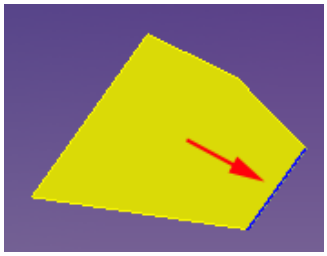


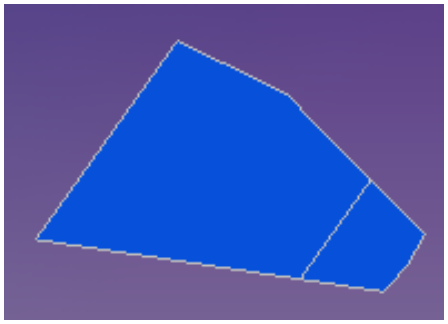




Surface Tools Center Curves Example #2

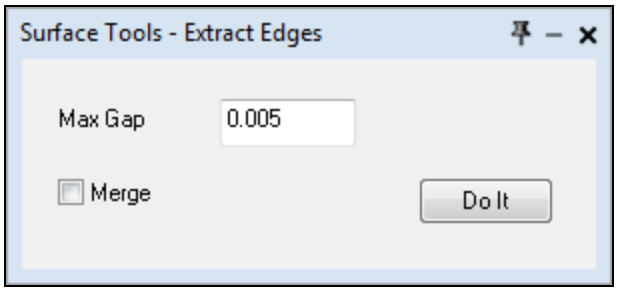






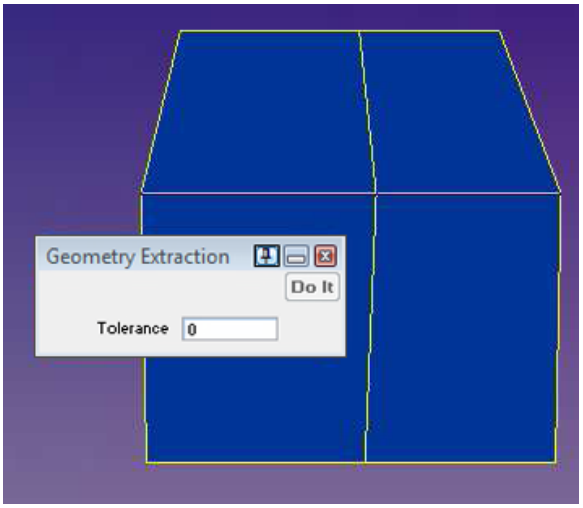
Surface Tools Center Curves Example #3

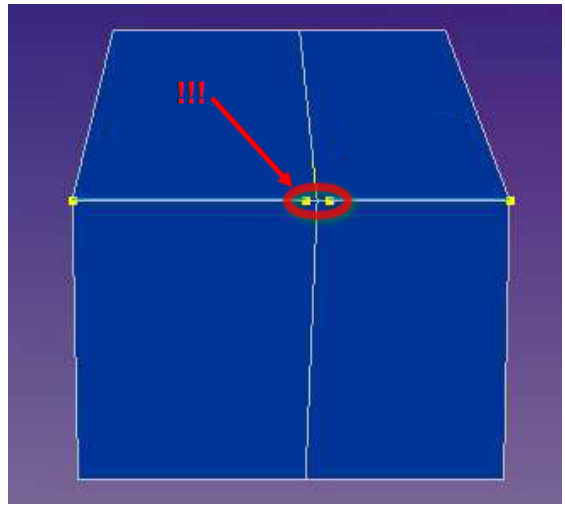
Extract Edges



This feature is useful to extract edges from solids that were stitched with a large tolerance, such as solids imported from other solid modelers through IGES or STEP, or imported from PTC Creo Parametric (Pro/E) or from Catia.

The gaps between faces still exist in the solid, but are not visible in the solid representation because of the tolerant capabilities of the solid modeler. Those gaps will become visible when extracting the edges with the standard edge extraction tool, as soon as the selected edges cross the gap between the faces. Below is an example showing the problem.





Selection using Geometry Extraction Result (undesirable)

To use this function, select the edges you want to extract and set the Max Gap value. In a case where selected edges could belong to multiple faces, you can optionally also select the one face that contains all edges you want to extract.

Max Gap

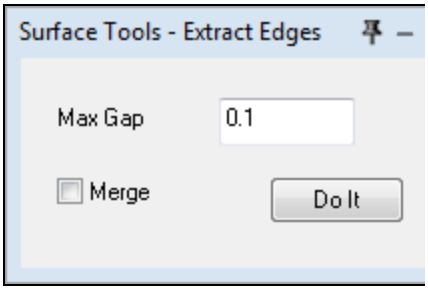
The maximum gap that will be closed when extracting the edges. If the gap between two selected edges is larger than this value, the gap (or overlap) will remain in the extracted edges. You have to make sure that this value is smaller than the length of the shortest selected edge.

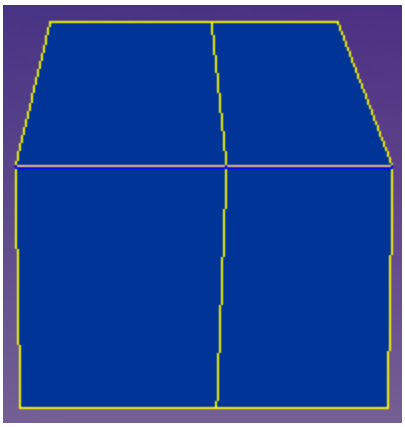
Merge

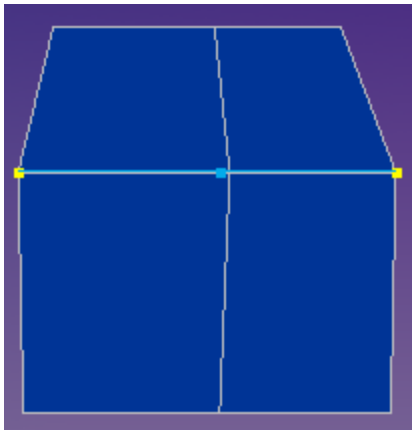
Select this option to merge the selected edges in one curve.
You should only use this option when all the selected edges are connected (with a gap smaller than Max Gap) ant tangent to each other.

Extract Edges Examples

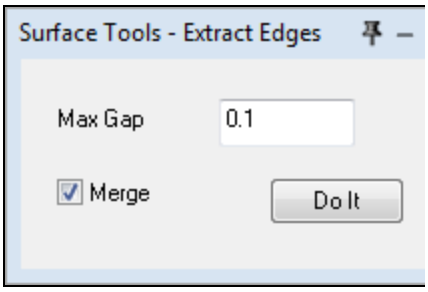
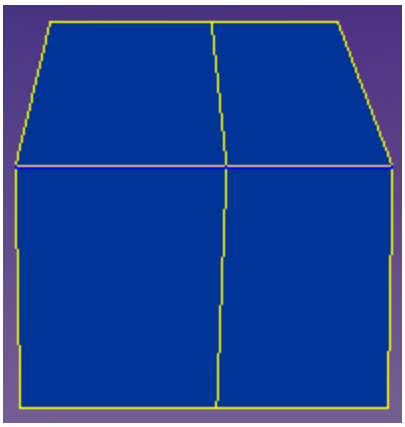
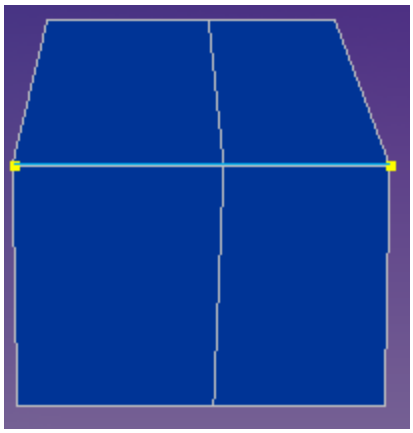
Parameter Selection Result



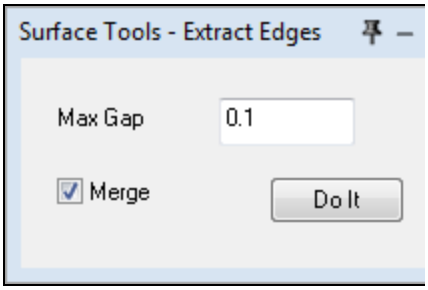
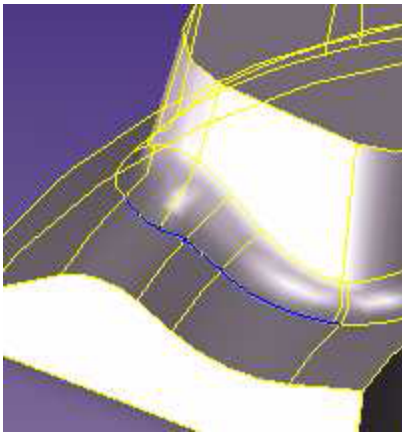
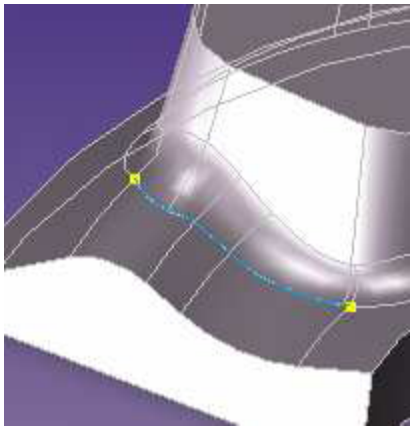




Extract Edges Example #1

Parameter	Selection	Result
		

Extract Edges Example #2

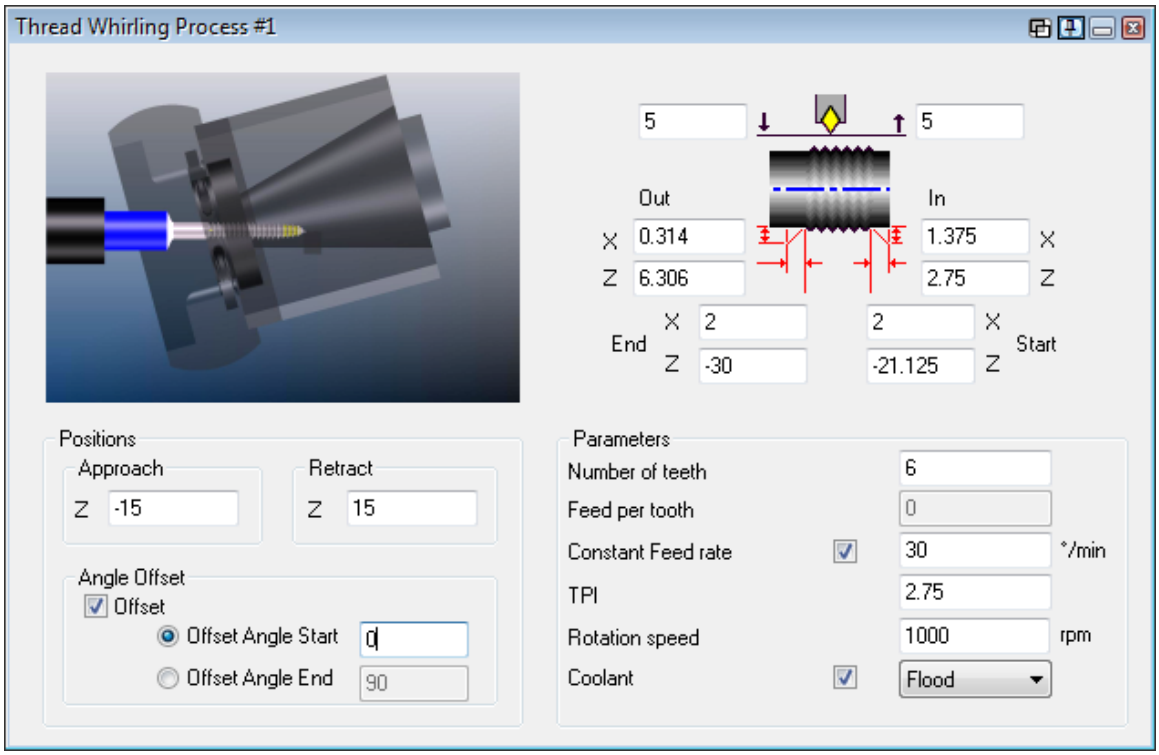
Extract Edges Example #3

Swiss Data Setup

This plug-in was retired at version 11.0, when its functionality was moved into the **Document Control** dialog (DCD), Workspace tab, for machines whose MDD includes a part station whose Has Guide Bushing checkbox is selected.
For information on guide bushing controls presented by the DCD, see the *Lathe* guide, topic "Cylindrical Stock (With Guide Bushing)".

Thread Whirling

Thread Whirling is a machining process where the cutters are mounted on the inside of a cutting ring or cutting holder rather than the outside of a milling tool.



This plug-in was retired at version 13.0, when its functionality was moved into the Thread Whirling process dialog.
For information on the Thread Whirling process, see the *Turning* guide.

Tool Library Explorers

These three plug-ins allow you to import tools from third-party tool libraries:

Harvey Tool:
https://www.harveyperformance.com/partners/cam-tool-libraries-gibbscam/

Helical Solutions:
https://www.helicaltool.com/

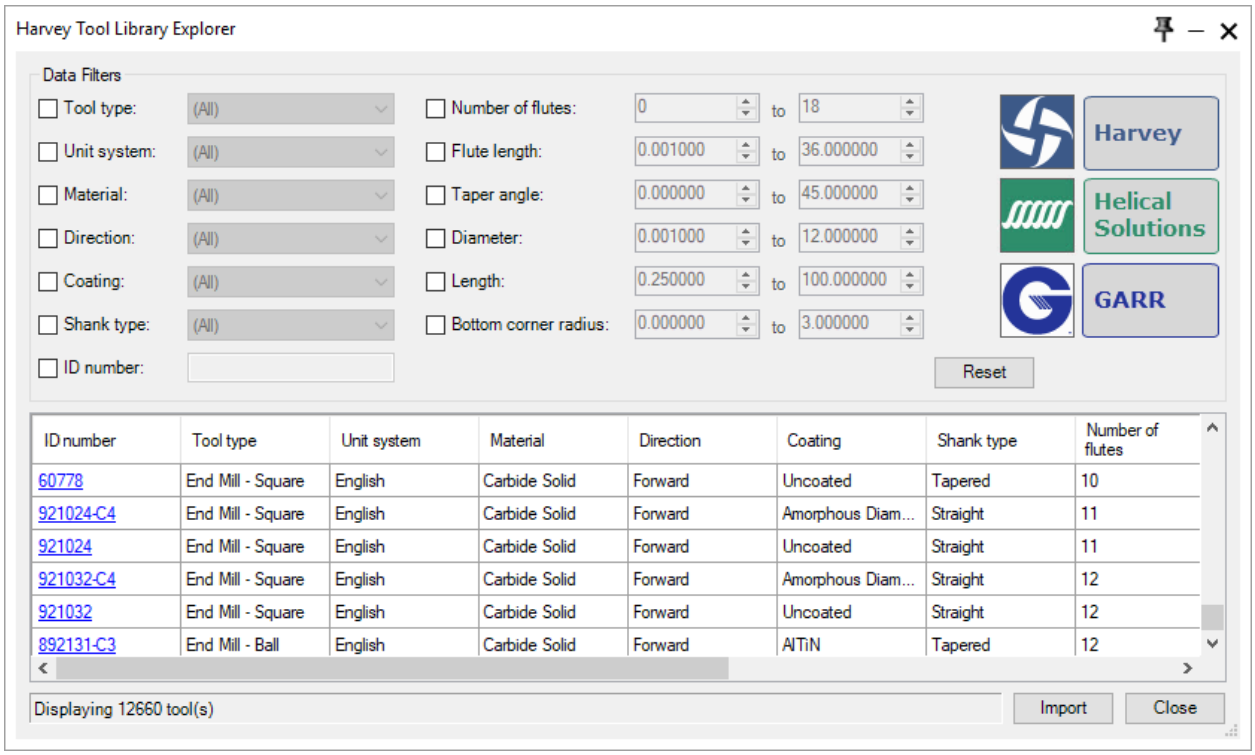
GARR TOOL:
https://www.garrtool.com/downloads/

All three use a very similar interface to filter, list, and import tools, as described below.

Using Tool Library Explorers

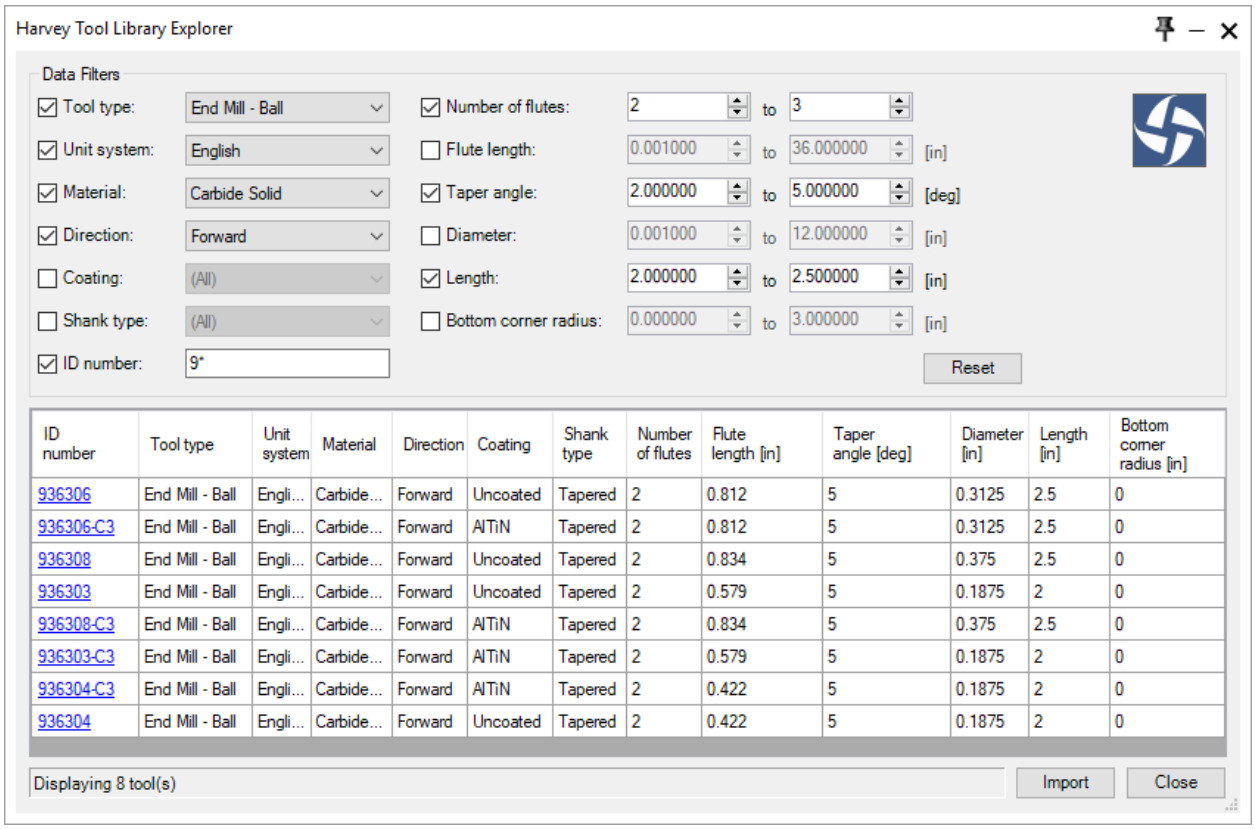
Before you begin: A GibbsCAM model must be open.

1. On the GibbsCAM Plug-Ins menu, under Main Tools, click the tool library you want to use, of:
   GARR TOOL, Harvey Tool, or Helical Solutions.
   Result: The corresponding tool library explorer dialog appears:

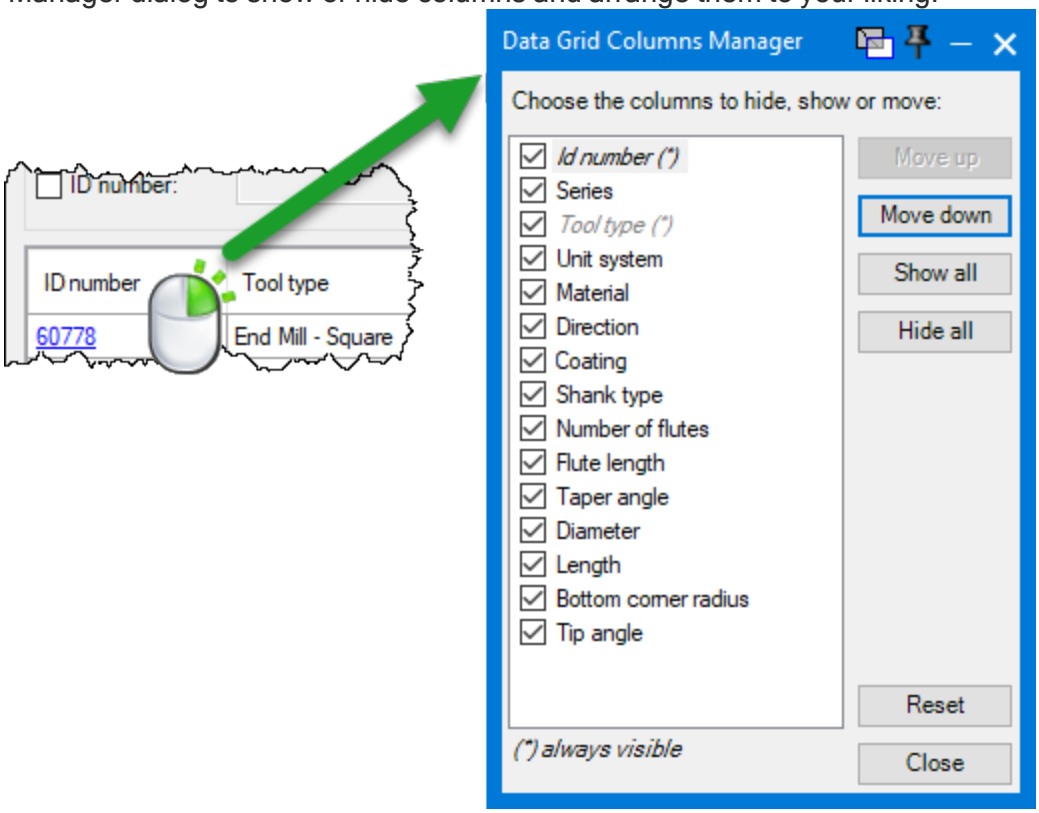


2. Use the checkboxes under Data Filters to narrow the choices:
   - Checkboxes in the left column, except for ID number, provide pull-down choices.
   - Checkboxes in the right column let you specify ranges.
   - The ID number checkbox lets you specify a matching pattern. You can use _ (underscore) as a wildcard to match any single character, and/or * (asterisk) as a wildcard to match 0 or more characters.

   Result: The list displays only and all tools that match the filters you have set, as shown in this example:



3. Optionally, you can right-click the top row of the table and use the Data Grid Columns Manager dialog to show or hide columns and arrange them to your liking.



4. Optionally, you can make table's columns wider or narrow, you can sort by values in any column, and you can rearrange columns by dragging them to the left or right.
5. Optionally, you can visit the tool manufacturer's website to view the tool's specifications by clicking the hyperlink under the ID number column.
6. Select one or more rows, optionally using context menu item Select All and/or the SHIFT key for range-select and/or the CTRL key to add or remove a row from the selection set.
7. Click the Import button (or right-click and, on the context menu, choose menu item Import Selection). You can import a maximum of 25 tools at a time.
   Result: The selected tools are added to the Tool List in GibbsCAM. In the Tool dialog, the Comment field is populated with text from the manufacturer.

Tool Manager iQ

ToolManager*iQ* (TMiQ), developed by CAM Solutions, is an AI-based tool management system for organizing cutting tools, holders, assemblies, fixtures, and suppliers. It is supplied in the GibbsCAM installer and fully integrated with GibbsCAM, accessed via the **Plug-Ins** menu. Tier 1 support for TMiQ is provided by the GibbsCAM team; advanced or in-depth support is handled directly by CAM Solutions.

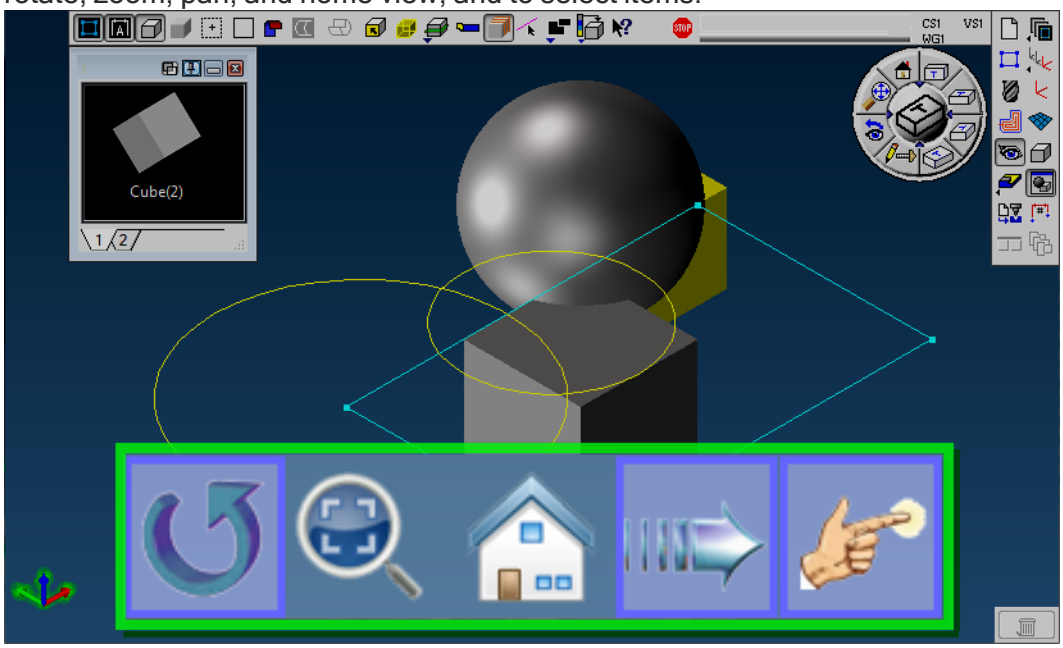
– An internet connection is required.

ToolsUnited Tool Library

See " Cloud-Based Tool Libraries" on page 32

Touch Interface Manager

With suitable hardware, you can use touch gestures to control common viewing actions, such as rotate, zoom, pan, and home view, and to select items.



Icon	Name	Icon	Name
	Rotate View		Pan View
	Zoom View		Select Element
	Home View		

Tp Transform / Transform Toolpath

Modify the location, orientation, and size of selected toolpath. When using the Tp Transform plug-in, one or more operations must be selected prior to choosing Do It; the plug-in makes one or more copies of the selected operations and applies a Transform Toolpath operation modifier to each copy.
The Translate option moves toolpath horizontally, vertically, or along the depth axis. Toolpath can be rotated around a point by a specified number of degrees using Rotate.
The Mirror options duplicate and flip toolpath along the horizontal or vertical axis. Toolpath can be resized using the Scale option.
Toolpath can be shrunk using Shrinkage option.
If you want to persist the settings and effects of this functionality (for example, preserving it through Redo or Redo Ops), you can apply it as an operation modifier instead of using the plug-in. For information on using operation modifiers, see the *Mill* and *Turning* guides.

About Linked Operations (Parent/Child)

In some circumstances, such as using the TpTrans plug-in on a selected operation, one or more operations are created that are linked to the operation that was selected. Each new operation created in this way is called a *child op*, and the original from which a child op is created is called a *parent op*.
A child op can itself become a parent of other ops, each of which is a grandchild of the original op; likewise with great-[great-[...]]grandchildren.
A child op has no process of its own: its processes derive from the process (or process group) of its parent op. Therefore, deleting a parent op deletes any child/grandchild/... ops linked to it, and redoing the parent after changing its process propagates the same change to all its child/grandchild/... ops.
Op Tile Stacking. Since an op shares the same process or processes with all its children, such tile stack views as Process Mode and Process List group the parent together with all its children.
Breaking Links. To break the link from a child op to its parent, use the Break Parent-Child Link plug-in.

Translate

Select this option button to move the operation along the H, V, and D of the Operation CS.

Trans H
Horizontal value by which to offset the duplicate toolpath.
Trans V
Vertical value by which to offset the duplicate toolpath.
Trans D
Depth value by which to offset the duplicate toolpath.

Rotate
Select this option button to 2D rotate the operation in the Operation CS.
Center H
Horizontal value about which to rotate the duplicate toolpath.
Center Y
Vertical value about which to rotate the duplicate toolpath.
Angle
Angle by which to rotate the duplicate toolpath around the XY centerpoint.

CS
If your part contains more than one CS, you can select the CS to be used as the reference for the transformation.
Transform in This CS
Performs the transformation within the current CS.
Create New CS
Performs the transformation in a new CS that will be created.
Repeat
Available only in the Transform Toolpath plug-in. You can specify multiple copies of the selected operation and transform the copies. Available only for `Translate` or `Rotate`. The default value, `1`, transforms only the original operation without making further copies.

Mirror
Mirroring will flip operations about an axis. This can be performed on the operations or a copy of the operations. `Mirror Horizontally` and `Mirror Vertically` are mutually exclusive options. And in the Transform Toolpath operation modifier, both of the Mirror options are mutually exclusive with `Scale`, `Reverse`, `Reverse HV`, or `Shrinkage`.
H coord
Horizontal value along which to mirror the duplicate toolpath.
V coord
Vertical value along which to mirror the duplicate toolpath.

Scale
Available only in the Transform Toolpath operation modifier. Shrink or grow the selected toolpath.
Factor
Amount (times) by which to resize the duplicate toolpath.

Reverse
Available only in the Transform Toolpath operation modifier. Select this option button to use the reverse direction instead of the original toolpath direction.

Reverse HV
Available only in the Transform Toolpath operation modifier. Select this option button to use toolpath HV loops in reverse direction instead of the original direction.
These options are rarely used, because they affect only the toolpath, with no corresponding changes to connecting moves, climb/conventional, entry or exit moves, and so forth. They are mutually exclusive with other options.

Shrinkage
Available only in the Transform Toolpath operation modifier. Select this option button to shrink the toolpath either uniformly or non-uniformly with user-defined X,Y,Z percentages.

Uniform %
Enter the percentage of shrinkage to be applied to all axes.

X%, Y% Z%
Enter a percentage of shrinkage for each of the three axes.

Z Ramp Contour / Z Step

The Z Ramp Contour plug-in and Z Step operation modifier were retired at version 10.5, when their functionality was moved into the process dialog. For information on the **Contour** process dialog, Contour tab, Ramp Down checkbox, see the *Mill* guide.

Distribution Only Plug-Ins

Distribution Only plug-ins are items that were developed to fit a particular need and may be of use elsewhere. These items often do not adhere to the GibbsCAM look and feel and are not considered a normal part of the system. Nevertheless, for your convenience we have provided documentation and tutorials covering the use of these items.

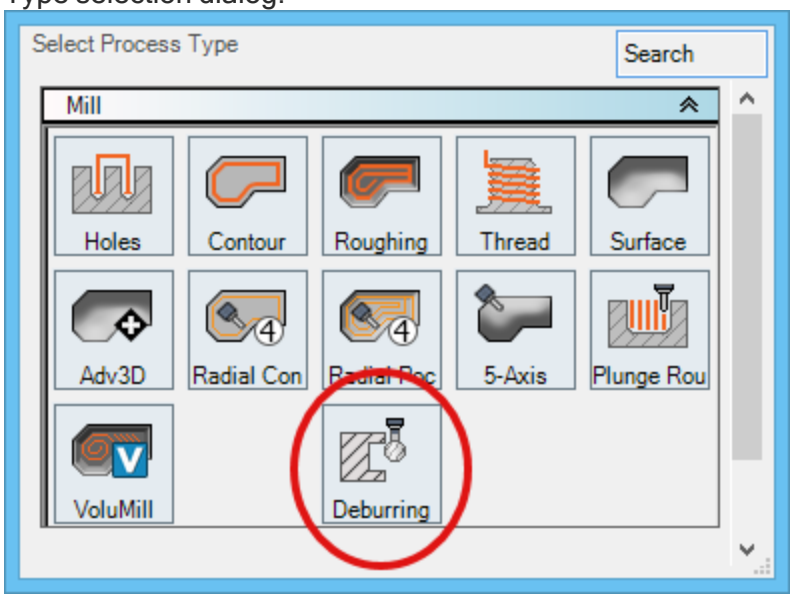
- "4/5 Axis Post Engine Selection" on page 101
- " Deburring Process" on page 101
- "Get Section" on page 104
- "Line-Line Intersect" on page 104
- "MDD Power Tools" on page 105
- "Set Process to Face Approach" on page 106

4/5 Axis Post Engine Selection

This changes the way 5-axis post processors are run. This plug-in is not installed by default and should only be installed if you are directed to do so by the post department. This plug-in is available in the extras\plugins folder and is called Post5Select.dll.

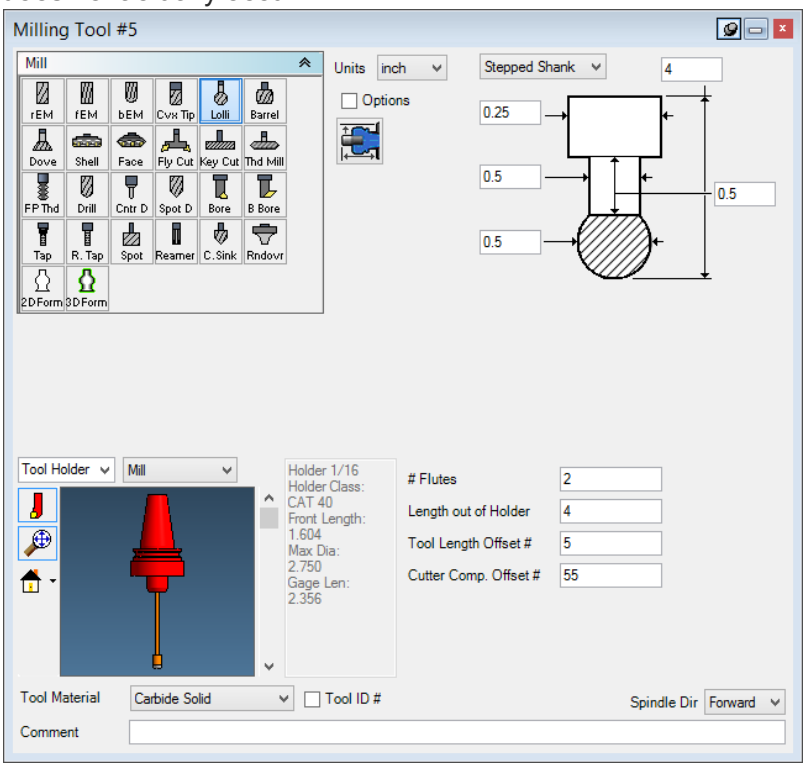
Deburring Process

The Deburring Process will clean up selected edges of parts. The user must first activate the Plugin in the Plugin Manager dialog. Deburring will then appear as a process in the Processes Type selection dialog.

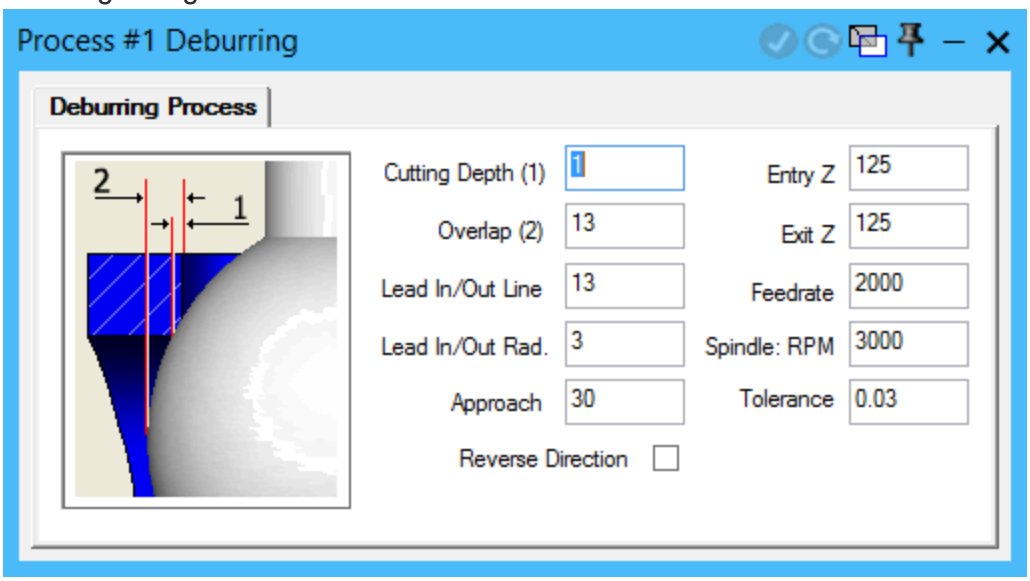


While any tool can be used (the tool will be considered a sphere with a diameter equal to the tool's diameter), it is recommended that a lollipop tool (like the one shown below) be created to

represent the most accurate rendering of the part. Using an endmill may show a collision that does not actually occur.



Drop the tool onto an empty process tile and select the Deburring Process to bring up the following dialog.



Cutting Depth

This value represents the size of the chamfer (as the graphic in the dialog illustrates) and specifies how deep the tool will penetrate into the material. The distance is measured along the normal to the selected face.

Overlap

This item specifies how deep the tool will move inside the opening. The value must always be smaller than the tool's (or the tool shank's) radius, or else the tool shank will collide with the part.

Lead In/Out Line
This value represents the length of the lead in and lead out moves, or the length of the tangent line to be used with a 90 degree arc to approach and exit the deburring operation. Enter a value of zero for an arc move.

Lead In/Out Radius
This value represents the radius of the circular lead in and lead out moves; in other words, the radius value for the 90 degree arc following the tangent line. Enter a value of zero for a straight line move.

Approach
This item determines the length of a linear approach or retract move, parallel to the normal to the selected face at the start or end point.

Entry Z
This value represents the Z value at the start of the operation (Z CP2).

Exit Z
This value represents the Z value of the end of the operation (Z CP3).

Feedrate
This item determines the feedrate value in part units.

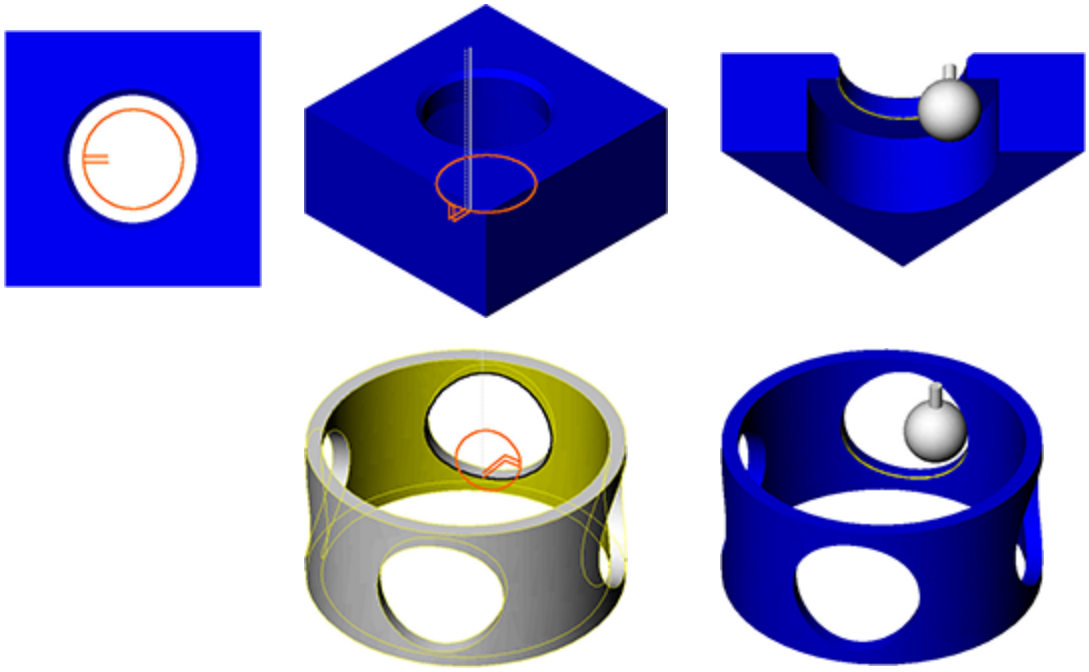
Spindle RPM
This item determines the spindle rotation in revolutions per minute.

Tolerance
This value specifies the tolerance used to approximate the edge.

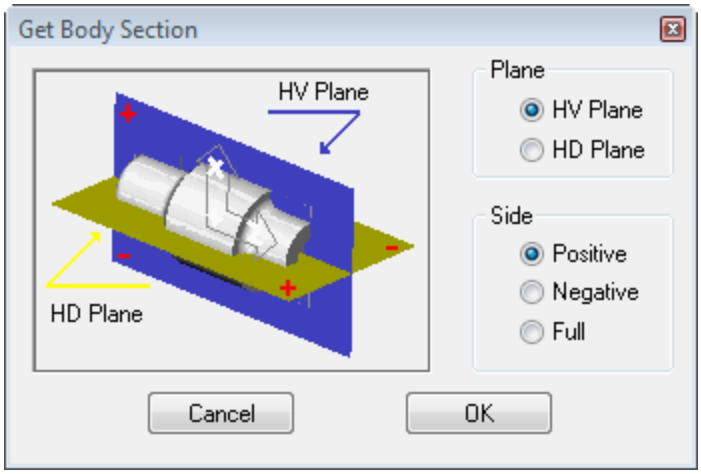
Reverse Direction
Select this checkbox to reverse the calculated toolpath.

Click Close in the dialog and then Do It in the Machining palette to create the deburring operation.

The images below illustrate practical use of the Deburring Process. The spherical tool approaches a hole in the part, feeds onto the edge, cuts around the select edge, pulls away, and then retracts.



Get Section



The Get Section Plug-In extracts geometry from the intersection between a solid and the HV or HD planes. This Plug-In is useful for extracting flat profile geometry in solid Lathe parts to be used as a basis for machining, and requires that Solid Import be enabled for it to work. A solid must be selected before executing this Plug-In. If no points of intersection exist between the solid and HV or HD planes, a sheet will be created along the selected plane.

HV Plane
This option extracts geometry from the intersection of the selected solid and the current CS.

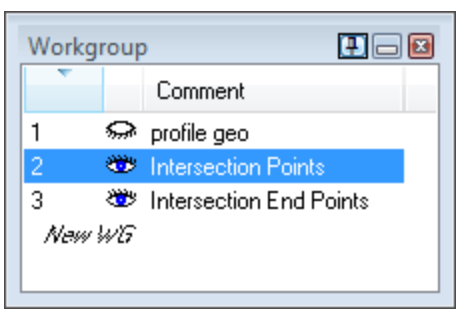
HD Plane
This option extracts geometry from the intersection of the selected solid and the plane perpendicular to the current CS.

Positive
Geometry is extracted from the intersection of the selected solid and the positive side of the V or D axis.

Negative
Geometry is extracted from the intersection of the selected solid and the negative side of the V or D axis.

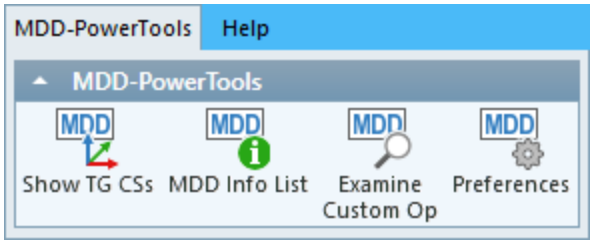
Full
Geometry is extracted from the intersection of the selected solid and both sides of the V or D axis.

Line-Line Intersect



The Line-Line Intersect plug-in creates points based on where selected lines intersect and where they would leave the stock boundary. Selecting at least two lines and then running this plug-in creates two new workgroups. The workgroup named Intersection Points contains the point(s) where the lines intersect each other; the workgroup named Intersection End Points contains the points where the lines would leave the stock boundary when fully extended.

MDD Power Tools

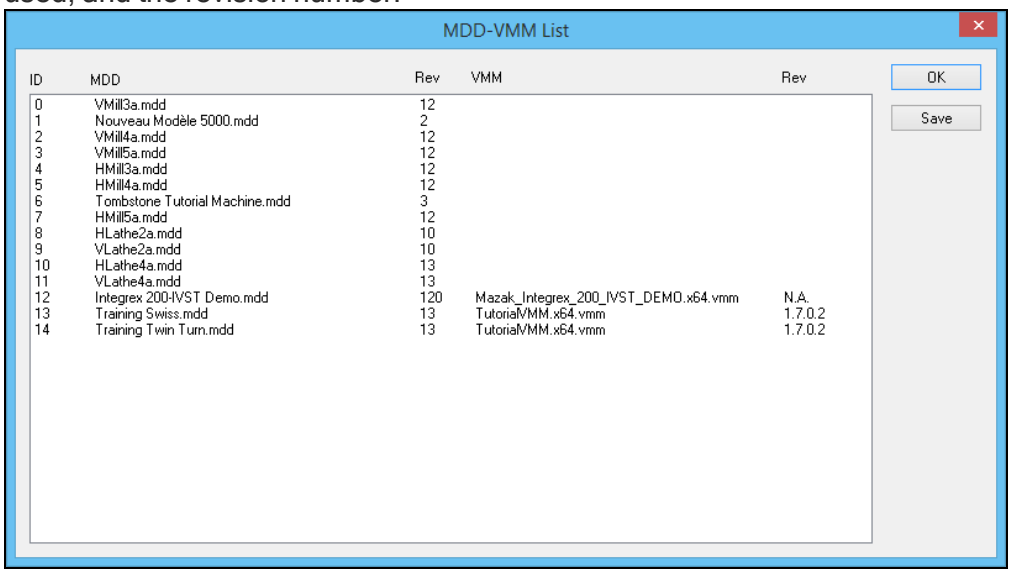


Show TG CSs

This option creates a CS for every toolgroup in the MDD and shows where each toolgroup is and in what direction it points.

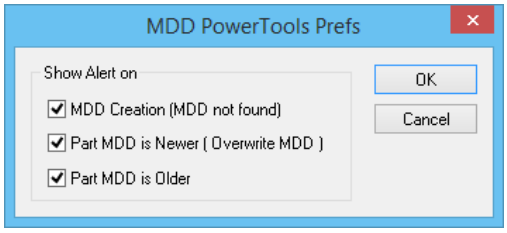
MDD Info List

This option shows every MDD that GibbsCAM has currently loaded, along with what VMM is used, and the revision number.



ID	MDD	Rev	VMM	Rev
0	VMill3a.mdd	12		
1	Nouveau Modèle 5000.mdd	2		
2	VMill4a.mdd	12		
3	VMill5a.mdd	12		
4	HMill3a.mdd	12		
5	HMill4a.mdd	12		
6	Tombstone Tutorial Machine.mdd	3		
7	HMill5a.mdd	12		
8	HLathe2a.mdd	10		
9	VLathe2a.mdd	10		
10	HLathe4a.mdd	13		
11	VLathe4a.mdd	13		
12	Integrex 200-IVST Demo.mdd	120	Mazak_Integrex_200_IVST_DEMO.x64.vmm	N.A.
13	Training Swiss.mdd	13	TutorialVMM.x64.vmm	1.7.0.2
14	Training Twin Turn.mdd	13	TutorialVMM.x64.vmm	1.7.0.2

Preferences



This gives you the option to turn on the three alerts shown here.

MDD Creation (MDD not found)

A new MDD is created when no MDD is found.

Part MDD is Newer (Overwrite MDD)

The part MDD is newer than the generic default MDD.

Part MDD is Older

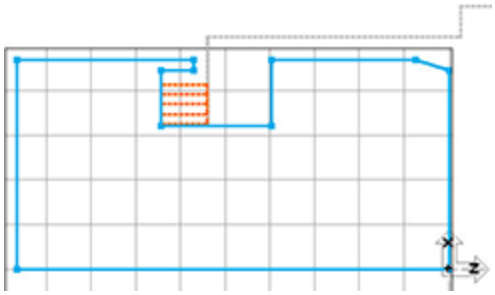
The Part MDD is older than the generic default MDD.

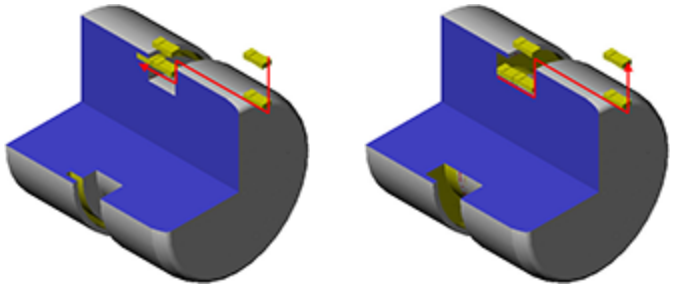
Set Part Origin

This plug-in was retired at version 11.0, when its functionality was moved into the **Document Control** dialog (DCD), Workspace tab.
It does no harm to continue to use this plug-in, which now simply sets values for X Y Z in the Part Offset section of the DCD Workspace tab. As always, these are absolute values in part units *from* the machine origin *to* the part origin.

Set Process to Face Approach

The Set Face Approach plug-in is only for use in lathe to change the approach moves of a turning tool. The tool normally moves in X and then Z for an ID operation; for an OD operation, the tool approaches the part by moving from the tool change position (first in Z, then down in X) to the start position. There are occasions when it is more desirable for a tool to approach an OD operation as if it were an ID operation (i.e. when machining a groove in the front of a turned part)–first in X, then in Z. The Set Face Approach plug-in can be used for this purpose.
Select either a roughing or contouring process and then run the plug-in. If the operation in question is an OD operation, the system changes the operation's information so that now the tool will approach the part as if it were an ID operation, and the graphic of the tool tile will be updated to reflect this change (as shown at the right). The results of the plug-in may be verified by running Cut Part Rendering.





Conventions

GibbsCAM documentation uses two special fonts to represent `screen text` and `keystrokes or mouse actions`. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

Text

`Screen text`. Text with `this appearance` indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.
`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A “torn” edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:

Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

Links to Online Resources

Please contact your reseller for support.

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens Gibbs Online page to download GibbsCAM and all supported material.