User Guide

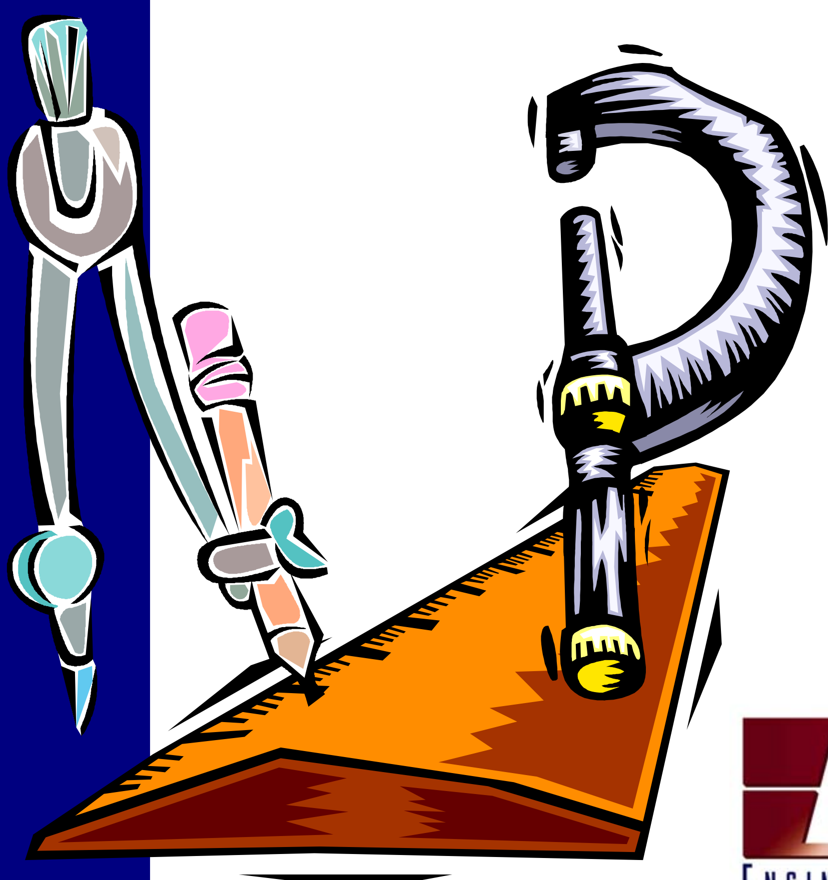
FeatureCAM

FeatureMILL • FeatureMILL3D • FeatureTURN

2004

Your guide to

Feature-based Manufacturing



E·G·S

ENGINEERING GEOMETRY SYSTEMS

June 2003
Tenth Edition

Engineering Geometry Systems
275 East South Temple, Suite 305
Salt Lake City, UT 84111

Table of Contents

CHAPTER 11 - GENERATING/SIMULATING TOOLPATHS

CHAPTER 12 - CONTROLLING MANUFACTURING

CHAPTER 13 - PART DOCUMENTATION

CHAPTER 14 - TOOLING

CHAPTER 15 - FEEDS AND SPEEDS

Chapter 1
Introduction

Getting Started with FeatureCAM manual

The FeatureCAM *User Guide* is a companion to the *Getting Started with FeatureCAM* manual. For information on the FeatureCAM interface or tutorials for the entire FeatureCAM product family, refer to the *Getting Started with FeatureCAM* manual.

FeatureCAM product family

FeatureCAM is a product family containing FeatureMILL™, FeatureMILL3D™ and FeatureTURN™, FeatureRECOGNITION™, Tombstone Option and 5-axis Positioning Option. This User Guide covers the entire family. All products share many features. If a chapter or section is specific to a particular product, this distinction will be mentioned at the heading of that section.

Evaluation options

Regardless of which modules you have purchased, the additional modules are available for your evaluation at any time. To evaluate other modules:

1. Close all FeatureCAM documents.
2. Click Evaluation Options from the File menu.
3. Click the modules you would like to try. Remember that FeatureMILL3D contains FeatureMILL so there is no reason to click both milling modules.
4. Click OK.

The program now allows you to access the new modules, but you cannot post code or save a file.

Embedded on-line help

Reference information is also available from within FeatureCAM. This style of help has the advantages that it is easier to search than written documentation and on-line help lends itself better to cross-referencing. To display this help, select *Contents* from the *Help* menu. A dialog box with three tabs is displayed:

Contents: This tab contains the table of contents for the on-line help. Click on a book to reveal the underlying pages and click on a book to display the help topic.

Find: Use this tab similarly to an index to search for topics.

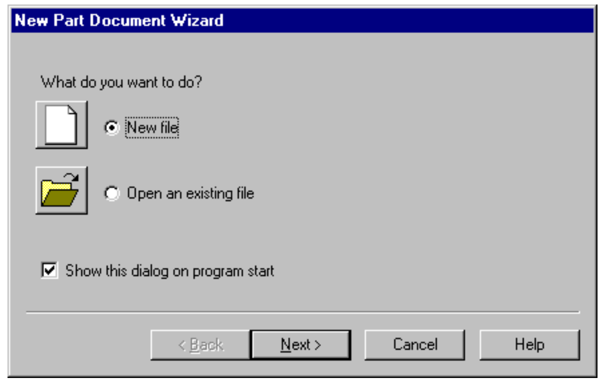
Search: Use this tab to search the entire help file for particular strings.

Context-sensitive help

Help is also available for any dialog box you are using. Just click the *Help* button to display the on-line help for your current task.

Chapter 2
Creating New Parts

New part document wizard



When you start FeatureCAM the *New Part Document Wizard* is the first thing you see. Click *Open an existing file* to work on an old FeatureCAM file. Click on *New file* to begin a new part. Click *Next* and follow the instructions in the next two sections.

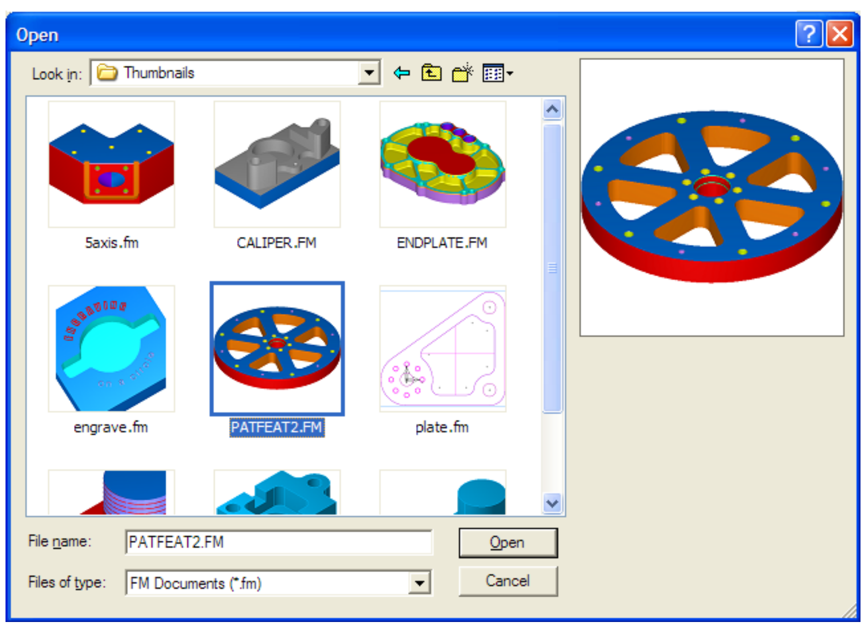
Open file button

Open loads the standard Windows browsing dialog for finding and opening files.

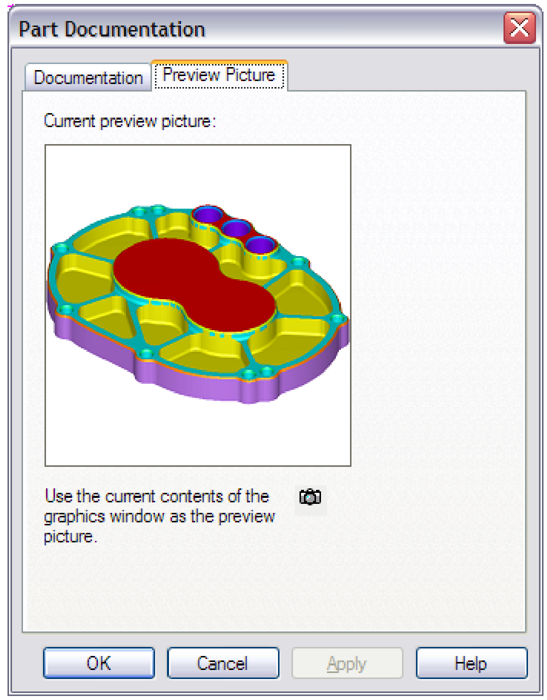
To modify an existing part file, click Open , on the Standard toolbar, select Open from the File menu, or click *Open an existing file* in the New Part Document Wizard. A standard Open dialog box for Windows appears. To use existing CAD files, have an open file ready to receive the data and use Import from the File menu. See page 153 for more information on importing CAD data.

Thumbnail pictures

When opening a part that was saved in version 10 or later, a preview image is displayed in the Open file dialog box. The saving of the image in the file is controlled by the Save options item of the file menu. This image will change each time you save the part unless you set a permanent image to store in the file.



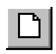
Preview Picture tab of part documentation dialog box



FeatureCAM's stores a small image of your part into the .fm file. This image is normally updated each time you save the part. If you want to store a permanent image with the part:

1. Create the view of the part you want to store.
2. Select *Part Documentation* from the *File* menu.
3. Click on the *Preview Picture* tab.
4. Click the camera button.

New FM document

To create a new file, click the New  button, select *New* from the File menu, or click New file in the New Part Document Wizard. The choices that you are offered will be limited by the FeatureCAM modules you have purchased.

Follow these steps:

1 Determine the type of the first setup of your part from among:

- Turn for 2-axis turned parts. Remember this is only the type of the first setup of your part. Your part can contain multiple setups and these setups can mix different manufacturing techniques.
- Mill for 2 ½ D or 3D milled parts. Once again, this is only the type of your first setup. Select this option for 5-axis positioning as well.
- Multiple fixture for laying out multiple parts on the table. This indicates that you will be mixing different milled parts for multiple part manufacturing. See *Multiple Fixture Parts* on page 305 for more information.
- Tombstone. See page 425 for more information.

2 Select the units you will use to model your part from among:

- Inch
- Millimeter

3 Click OK

System units

You choose dimension units when you create a new part file. You can choose from either Inch or Millimeter. If you wish to change dimension units later, select Default System of Units from the Options menu. If you always produce parts in the same units, you can turn off the automatic display associated with a new file by setting the *Don't ask at creation* check box.

Inch units set FeatureCAM to size the part, its features, and the tools in inches and fractions (decimal display) of inches. All measurements are in inches.

Metric units set FeatureCAM to size the part, its features and the tools in millimeters. There is no option for setting larger metric units as the default.

File options dialog box

The *File options dialog box* is displayed by selecting *File options* from the *Options* menu. This dialog box allows you to specify the defaults for opening new files.

The *File type* indicates the type of document you will create by clicking the *New*  button.

- Turn for 2-axis turned parts
- Mill for 2 ½ D or 3D milled parts. Select this type for 5-axis positioning as well.
- Multiple fixture for laying out multiple parts on the table.
- Tombstone. .

The *Unit of measure* is the units you will use for creating your part.

- Inch
- Millimeter

The choices for *Initial stock dialog* are:

- Wizard – The stock wizard will be displayed next. This option is best for novice users.
- Properties – The stock Properties dialog box. This dialog box presents the pages of the stock wizard in a more concise form.
- None – You will not be asked to setup the stock, but you will have to do it explicitly later.

If you want to or verify these choices each time you open a document check *Always ask when a new document is created*.

The *File Location* is the default directory for opening, saving, and importing files. If you specifically set the *File location* to a particular folder, then FeatureCAM will start in that folder whenever you start up FeatureCAM in the future and FeatureCAM will ignore the "Start In" folder that is listed in the shortcut to FeatureCAM. The starting folder is used when you first open, import, or save a part. But if you navigate to a new folder during your session, then the new folder is used for subsequent opens, imports and saves. FeatureCAM uses the idea of a current working folder, which means that if sometime during your session you navigate to some other folder to open a file, then any future opens or saves in that session will use the new folder. But the "File Location" folder will always be used as the starting folder for the next time you run FeatureCAM.

File options dialog box - Existing files tab

The following list of steps outlines what happens to tool cribs and tooling when a part is opened.

1. The name of tool crib that was originally used to create the file is saved in the .fm file. If there is a tool crib with the same name in the current installation of FeatureCAM, that tool crib is made current. If it doesn't exist, the last crib that was opened is used.

2. All tools that were explicitly overridden in the part file are copied to the current crib. These tools are only temporary and are only available when the current part is open.

3. A new crib is also created that contains only tools used for that part. This crib contains tools that were overridden and all tools that were automatically selected. The crib is called <filename>_tools_from_last_save. This crib is temporary and is only available when this part is opened.

4. If *Use the tool crib saved with the part document* is checked, then this new crib is selected as the active crib.

Note that if you check *Use the tool crib saved with the part document* the part will be cut with exactly the same tools. This is not recommended it you are going to make modifications to your part since this small tool crib will probably not have enough tools to cut additional features.

Feed and speed tables are handled similarly.

1. When a file is saved, the feed/speed tables for the part's material are saved with the file.

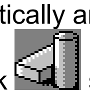
2. If *Use the F/S tables saved with the part document* is checked, then this new table is used as your feed/speed database.

3. If *Use the F/S tables from FeatureCAM's F/S database* is checked then the existing databases are used.

The *File Location* is the default directory for opening, saving, and importing files. If you specifically set the *File location* to a particular folder, then FeatureCAM will start in that folder whenever you start up FeatureCAM in the future and FeatureCAM will ignore the "Start In" folder that is listed in the shortcut to FeatureCAM. The starting folder is used when you first open, import, or save a part. But if you navigate to a new folder during your session, then the new folder is used for subsequent opens, imports and saves. FeatureCAM uses the idea of a current working folder, which means that if sometime during your session you navigate to some other folder to open a file, then any future opens or saves in that session will use the new folder. But the "File Location" folder will always be used as the starting folder for the next time you run FeatureCAM.

Stock

Setting up the stock with the stock wizard

The Stock dialog box and Setup wizard appears automatically any time you create a new part file. You can also access the wizard by clicking the Stock  step from the Steps toolbox. This wizard will help you specify the shape, location and material type of the stock and the part program zero.

Stock wizard – Dimensions page

The Stock Wizard assists you in defining the stock and coordinate systems. The Dimensions page helps you set the size of your stock.

To complete this page:

1. Click either the *Block*, *Round*, or *N-sided* radio button. If your stock is a different shape, see page 10.
2. If you selected *Block* enter the *Width* (X dimension) *Length* (Y dimension) and *Depth* (Z dimension).
3. If you selected *Round* enter the Axis, the *Length* and *OD*. If you are working with Tube stock, enter a positive number as the ID. If wrapping or indexing, the axis should correspond to your indexing axis.
4. If you selected *Round* enter the *Length* (Z dimension), and *OD*. If you are working with Tube stock, enter a positive number as the ID.
5. If you selected *N-sided* enter the *OD*, *Number of sides* and *Length* (Z dimension)
6. Click *Next*.

Note: If you are unsure of the size of your part, you can skip this step by simply clicking *Next*. Later, if you find that size of your stock needs to be changed, click the Stock step from Steps toolbox again to modify the size of the stock. If you are running the off-line version of FeatureCAM, you also have the option of automatically resizing the stock.

Stock wizard – Material page

This dialog box helps you specify the material type for the stock. Advanced users can also add new materials or view feed/speed tables.

To complete this page:

1. Select the material type from the *Material* drop-down list. The unit horsepower will be filled in for you.
2. A default hardness will be specified for the material, but if you know the specific hardness of the material, enter the numeric value in *Hardness* slot and then enter Hardness unit that the hardness is measured in.

Note: For some materials, the hardness is used in determining feeds and speeds. If you underestimate this value, your automatically generated feeds and speeds could be overly aggressive.

3. Click *Next*.

Stock wizard - Multi-axis positioning

In this dialog box you specify if you will be creating your part with 4^{th} axis indexing or 5^{th} axis positioning.

To complete this page:

1. If you are not using 4^{th} or 5^{th} axis milling, check *No*.
2. If you are using 4^{th} axis positioning, or 4^{th} axis wrapping check *4^{th} Axis Positioning*, then click the axis you index around.
3. If you are using 5^{th} axis positioning, check *5^{th} Axis Positioning.*

4. Click *Next*.

Stock wizard - Multi-axis options

This dialog box presents options for 4th wrapping and indexing and 5th axis positioning. It does not appear if you are not using the 4th or 5th axis options.

To complete this page for 4th axis milling:

1. If you want the ordering of operations to be tool dominant across all setups, click *Tool Dominant*. Note that you must also set the *Minimize tool changes* ordering attribute for Tool Dominant to work correctly.

2. If you want the ordering of operations to complete each setup before moving on to another setup, click *Setup Dominant*. The order that operations are performed within a setup is determined by the milling ordering attributes. Click *Generate Single Program* to create a single 5-axis indexed program.

3. Click *Next*.

To complete this page for 5th axis milling:

1. If you want to use fixture offsets, follow steps 2-6 in *5-axis positioning using fixture offsets* on page 435

2. If you want to use a single coordinate system follow steps 2-6 in *5-axis positioning using a single coordinate system* on page 433.

Stock wizard – Definition page

This dialog box allows you to name the setup and specify the fixture ID. Fixture IDs allow the machine to offset the fixture to correspond to the program zero, instead of having to move the fixture to the spot where the program is. For machines that use NC codes G54 – G59 or registers 54-59, the default should be 54. For machines that use NC codes H1- H3, the default should be 1.

To complete this page:

1. Optionally enter the setup name. This name is used only as a label for the setup.

2. Enter the Fixture ID for the setup. The default value should be correct since it is obtained from the current .cnc post processor template file.

3. Click *Next*.

Stock wizard - Part Program Zero

This dialog box lets you select a method of specifying part program zero.

To complete this page:

1. If you want to align program zero with a corner of the stock or explicitly type a location, click *Align to corner of stock*.

2. If you want to align with a previously created user coordinate system (UCS) click *Align with existing UCS*.

3. If you want to set the UCS relative to the index axis, click Align to index axis.

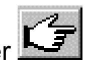
4. If you want to leave the part program zero at its current location, click *Use current location*.

5. Click *Next*.

Stock wizard – Align Part Program Zero with Stock

This dialog box helps you set the part program zero. This will be the origin of the coordinate system for the NC program.

To complete this page:

1. If you want to locate the origin at one of the points indicated by a pointing finger  button, click the button.

2. If you want to explicitly type the coordinates, enter the X, Y and Z coordinates.

3. If you want to pick the point with the mouse, click the *Pick point*  button. The dialog will warp into a button. Click the point with the mouse.

4. Click *Next*.

Stock wizard - Align Part Program Zero with UCS

This dialog box allows you to align the part program zero with an existing user coordinate system (UCS)

To complete this page:

1. Select the name of the UCS from the drop down list or click the Pick button and select it from the Graphics window.

2. Click *Finish*.

Stock wizard - Align with Index Axis

This dialog box allows you to specify the origin of the stock for a turn/mill or 4th axis indexed part.

To complete this page:

1. If you want to translate the origin off of the axis, enter the *Radius from rotation axis*.

2. If you want to rotate the coordinate system around the Stock axis, enter an *Angular location*.

3. Enter an *X Offset* if you want to translate the coordinate system along the X-axis.

4. Click *Finish*.

Stock wizard - Part Program Offset

This dialog box lets you translate your stock model. One reason to translate the stock is to model the extra stock on top of the part to that will be removed during a facing operation.

To complete this page:

1. If you want to translate the stock, enter the amounts to offset the stock as the *X, Y* and *Z Offsets*.

2. Click *Next*.

Irregularly shaped stock using stock curves

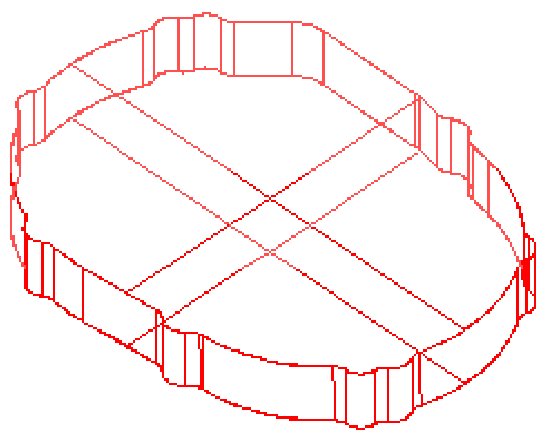
You may machine pieces from non-rectangular pieces of stock. The primary concern with these stock shapes is to create toolpaths that don't waste lots of time cutting air in regions without stock.

To create stock shapes other than block, round, hexagonal or octagonal, you must create a stock curve. This functionality is not available in the Stock wizard. Instead you must bring up the Stock Properties dialog box by either double-clicking on the stock, or selecting the stock and clicking the *Properties* button. To add a stock curve, click the *Stock curve*, button and check the curve in the list, or use the Pick  button and the mouse to pick the curve graphically.

Stock curves must be closed and lie in the stock XY plane. You may use a full circle, but only a single curve or circle can be selected. The curve also must not self-intersect, although FeatureCAM won't detect this condition.

A Stock curve is the default stock boundary for features on the top and bottom of the stock. For simplicity and flexibility, the stock curve should meet the positive X and Y axes. This location lets you easily calculate the width and length of the stock curve extent and position the origin at the corner of those rectangular extents.

Because of the nature of stock defined by a curve, aligning a UCS to custom stock ignores the stock curve and works with the rectangular extents.



Example of stock modeled with a stock curve.

Setting up user defined stock

Use this option if you want to use a solid model as the initial stock. Note that this only provides an initial simulation model, it does not force toolpaths to be within the solid stock boundary. To specify a solid as your stock:

1. Bring up the stock property dialog box by either double clicking the stock or selecting the stock and clicking the Properties button.

2. Click the *User defined* radio button.
3. Click the *Stock solid…* button.
4. Check the checkbox next to the name of the solid and click *OK*.
5. Click *OK* again.

Resizing the stock

If you created or imported geometry and want to resize the stock automatically:

1. Bring up the stock property dialog box by either double clicking the stock or selecting the stock and clicking the *Properties* button.
2. Click on the *Resize* button.
3. If you want to add additional material in any dimension, enter the amount in the Extra stock fields.
4. If you want to move the stock to the geometry, click the *Move Stock* button.
5. If you want to move the geometry to the stock, click the *Move Geometry* button.
6. Click *OK.*

Steps for changing material

To change the material type of your stock:

1. Double click the stock depiction in the Graphics window to open the Stock Properties dialog box.
2. Click *Material*.
3. Select your material type from the Material drop down list.
4. Set the Unit Horsepower or Specific cutting force if the default value is incorrect.
5. Specify the Hardness.
6. Select your Hardness scale if it is different than Brinell.
7. Click OK.

Material settings

Clicking Material in the Stock Properties dialog box opens another dialog box where you select defined materials for your part to be made from.

The Material drop-down list contains the following materials:

Aluminum	Bronze
Aluminum Cast <10% Si	Cast_Steel
Aluminum Cast >10% Si	Copper
Brass	Free cutting brass

Graphite

Grey cast iron

Magnesium

Plastic

Stainless steel (many sub-types)

Steel (many sub-types)

Titanium

Tool Steel (many sub-types)

Unit horsepower or cutting force

The power required to perform a cut is based on the rate the material is being removed and a power constant that is dependent on the material. This constant is known as the *unit horsepower*, *specific cutting force* or *power constant*. In FeatureCAM, this number is used to generate horsepower estimates for operations. These estimates are displayed in the operation list. If you enter an inaccurate value for the unit horsepower, no error will be generated. The only consequence is that improper horsepower estimates will be displayed.

For materials that are pre-defined in FeatureCAM, an approximate value is supplied that is independent of a material's hardness. For many materials, the unit horsepower is dependent on the hardness of the material, so you may have to adjust this constant to obtain more accurate horsepower estimates. A list of specific unit horsepower values can be found in a machinist's handbook. For defining new material tables, the Unit Horsepower constant must be supplied by the user.

Hardness

When you select a material name, the Hardness value is set to the low end of the defined hardness range for that material. Adjust the hardness to reflect the actual hardness of your material in this field. The feeds and speeds calculated for your part are influenced by the hardness of your material.

Hardness scale

Scale sets which scale your hardness setting is based on. The supported scales are Brinell, Rockwell C, Rockwell B, and Tensile Strength. Brinell is the default hardness scale.

New material button

New Material opens a small dialog for naming the new material.

Type in the name and click OK. If the material doesn't have an entry in the database, a warning appears. For a truly new material, click *Yes*. The Feeds and Speeds dialog box appears where you define the new material.

Chapter 3 Viewing

Interactive viewing

Use the View fly-out to select mouse controlled viewing modes and change your view of the part. Selecting any of the options from the view fly-out controls your view for one mouse click. Then you revert to your prior mode. You can stay in Viewing if you Ctrl+click the toolbar button. To turn it off again, click another toolbar button.

Your cursor shows the same icon as the viewing mode you selected. You can change the view in a number of ways as shown and described later with the fly-out toolbar. Viewing is performed interactively in FeatureCAM with the mouse.

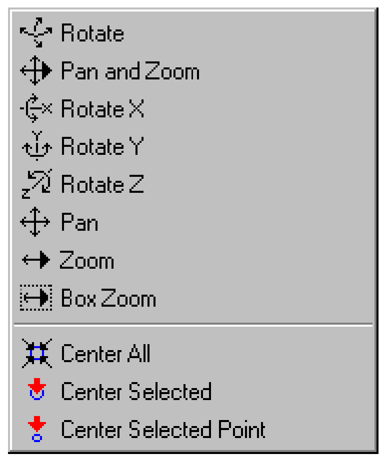
You can also use those modes with the Fast view function. Lastly, you can save views and return to them as you wish to compare different aspects of your part.

Select *Save view* in the User views submenu of the View menu.

You can save up to four views at any time. Select the numbered saved view to return to that view of your part.

You can toggle between your current view and last view with the Last View menu item, or its accelerator keys, ALT+L.

View fly-out



View changes your view of the part. Selecting any of the options from the view fly-out puts you in View mode. Your cursor shows the same icon as the viewing mode you selected. You can change the view in a number of ways as shown and described later with the fly-out toolbar. Viewing is performed interactively in FeatureCAM with the mouse.

Select the type of view manipulation you want to perform from the Viewing Modes sub-menu in the View menu or from the toolbar and its fly-outs. This sets the viewing mode.

Click and hold the mouse button, then move the mouse. Up or to the right changes the drawing one way, zooming or rotating clockwise for example. Down or to the left has the opposite effect.

Rotate

Rotate rotates the part around the X and Y axes with the movement of the mouse.

Rotate X, Rotate Y and Rotate Z

This command rotates the part around a single axis with the movement of the mouse.

Pan

This command translates the part.

Zoom

Moving the mouse to the right scales the part up. Moving to the left shrinks the part.

Pan and Zoom

This command combines the functionality of panning and zooming into a single mouse drag. The point you click on is centered in the screen and then the part is zoomed with the movement of the mouse.

Box Zoom

The portion of the model that is outlined with the mouse drag is zoomed to fill the Graphics window.

Center all

Changes to a complete view of the part, centering it in the Graphics display window. The view of the part automatically scales to fit in the window.

Center selected

Centers the entities you have selected in the Graphics window of FeatureCAM. The view of the part is automatically scaled to fill the window with the selected items.

Center selected point

Centers the entities you have selected in the Graphics window of FeatureCAM around the point you last clicked in the Graphics window.

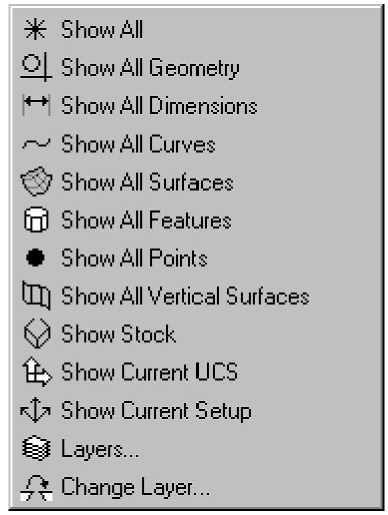
User views

The User Views (View1, View2, View3, View4) store various views of your model. When you select save view, the current view is saved under the next available view. Select one of these views to return to this view.

Save view

Saves the current view under one of the user views.

Show fly-out



Show functions help control what is displayed at any given time. This is useful as you place and model intricate features in a complex part.

Show UCS axis shows the icon and location for the current UCS.

Show UCS shows the UCS icon.

Show surfaces displays all surfaces. Remember that surfaces and features from surfaces are different. Only available in FeatureMILL3D.

Show stock shows the stock outline.

Show layers opens the Layer dialog box where you select which layer(s) to show hide, or make active. You can also define a new layer for the part.

Show geometry shows all geometry (points, lines, arcs and circles).

Show features shows all features.

Show points shows point objects.

Show all vertical surfaces shows all surfaces that are parallel to the Z-axis of the current setup.

Show dimensions shows all dimension information added with the Dimension tools.

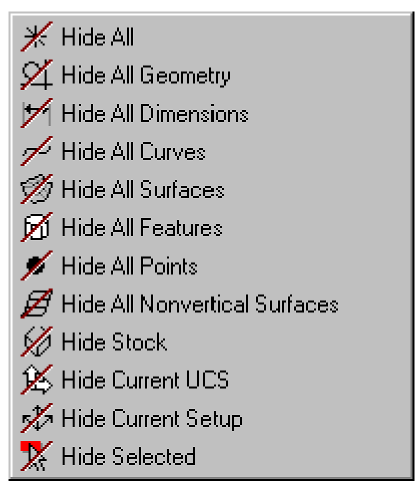
Show curves shows all curves.

Show current setup displays only those features and drawing elements that are in the current setup.

Layers displays the layers dialog box. In this box you can create or delete layers and decide what layers to display.

Change layers allows you to change the layer of the selected object(s).

Hide fly-out toolbar



Hide controls what is displayed at any given time. This is useful as you place and model intricate features in a complex part. Besides the display factors, you can't snap, select or build curves from hidden entities. The hide functions are not exclusive. You can click different buttons sequentially, hiding different entities until only the ones you want are still in view.

All hides all geometry, curves, features. The stock and axis icon remain visible. A common procedure is to Hide All, then Show only one type of entity, features, for example.

Stock hides the stock outline. All other entities remain visible.

Setup Axis hides the axis of the current setup. All other entities remain visible.

Current UCS hides the current user coordinate system.

Stock Axis hides the axis of the stock coordinate system.

Curves hides all curves. Other entities remain visible.

Surfaces hides all surfaces in the part model. Only available in the 3D version.

Geometry hides all geometry. Other entities remain visible.

Dimensions hides all dimension information added with the FeatureCAM Dimension tools.

Features hides all features.

Points hides point objects.

Hide all nonvertical surfaces turns off the display of surfaces that are not parallel to the Z-axis of the current setup.

Selected hides all selected entities. Non-selected entities are still visible.

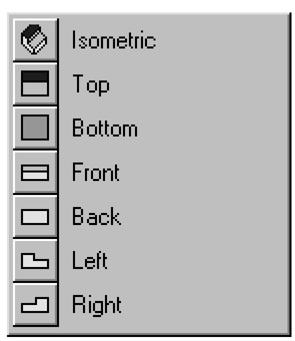
Unselected hides all entities other than the selected ones.

Select button

Select puts you in Select mode. You can tell you are in select mode because the mouse appears as a standard arrow pointer in the part window. In select mode, you can click features to select them. Then you can modify their attributes, turn their view on or off, change their layer, and so on. You can't pan or zoom or draw geometry in this mode.

Principal views fly-out

The *Principal views* fly-out changes the view to a predefined view.



Top view

Changes to a view of the part from the top only. Useful for drawing geometry, but harder to see the wire-frame model of the part.

Isometric view

Changes the view to a three-quarter view of the part showing the top and two sides with the current UCS near the bottom of the view area.

Front view

Changes to a view of the part from the front with no other surfaces visible.

Right view

Changes to a view of the part from the right side with no other surfaces visible.

Bottom view

Changes the view to the bottom of the part. No sides of the part are visible from this perspective.

Left view

Changes to a view of the part from the left side with no other surfaces visible.

Last view

Return to the last view of your part model with Last View.

Back view

Changes to a view of the part from the back with no other surfaces visible.

Perspective

The Perspective flag indicates the mode of viewing your part model. If selected you will view your model in a perspective view. Otherwise you will view the model in orthographic view.

Center all

Changes to a complete view of the part, centering it in the Graphics display window. The view of the part automatically scales to fit in the window.

Center selected

Centers the entities you have selected in the Graphics window of FeatureCAM. The view of the part is automatically scaled to fill the window with the selected items.

As on UCS

Changes your view to that of the current UCS.

As on setup

If *As on setup* is turned on, your viewing will be relative to the current setup. If it is turned off, your viewing will be relative to the stock coordinate system.

As on world

Changes your view to that of the Stock UCS.

View entities

Another useful viewing tool in the View menu is Entities. Select Entities in the View menu to open the Entities dialog box.

All geometry, curves, stock, and features, and surfaces are listed. You can sort the list by clicking one of the title buttons. The list reorders itself using that list as the alphabetical sorting field. This makes it easy to select by layers, construction type or category of entity.

Select an item or multiple items with Ctrl+click and then click a button to perform one of the following actions:

Close exits the dialog box.

Show displays the selected entities in the graphics window.

Hide removes the selected entities from the display in the graphics window.

Delete removes the selected entities from the part file.

Rename is only available for individual selections and opens the Rename dialog box where you change the name of the entity.

Properties is only available for some kinds of entities and only for individual selections. The properties dialog box that opens depends on the kind of entity. Features open the appropriate feature properties dialog box, for example.

Viewing options

Besides the Windows settings, you can control the quality of your part display in FeatureCAM after you've installed it. You have to balance the detail quality against the increased time it takes to generate more detail. Set these options in the Viewing Options dialog box, accessed through Viewing in the Options menu.

Curve fineness

Curve fineness adjusts the length of line segments for displaying curves. The smaller the line segments, the smoother the curve appears.

With small curve fineness values, more data is processed so the graphical performance slows down. If you increase the fineness value, graphical performance is improved but the display quality suffers, producing jagged, more notched curve representations.

Because it is easy to modify this value, you can use different settings at different stages of development.

Surface fineness

Surface Fineness adjusts the area of flat polygons (plane segments) for displaying a surface. FeatureCAM uses surfaces to display all features and stock models. The smaller the area of the polygons used to display a surface, the smoother the surface appears. There are separate surface fineness values for the shaded and wireframe representations of the surfaces.

With small Surface Fineness values, more data is processed, so the graphical performance slows down. If you increase the fineness value, graphical performance is improved, but the display quality suffers, producing more faceted, rougher surface representations.

Because it is easy to modify this value, you can use different settings at different stages of development.

Depth cueing

Depth Cueing is a toggle switch that provides a visual cue to mark distance in the graphical display. When Depth Cueing is on, shading as a function of distance is increased.

View animation

This option will provide smooth, animated transitions between principal views. With *View animation* turned on the part will smoothly rotate between two principal views. With this setting not enabled, the view will be abruptly changed to the new view.

Show surface boundaries only

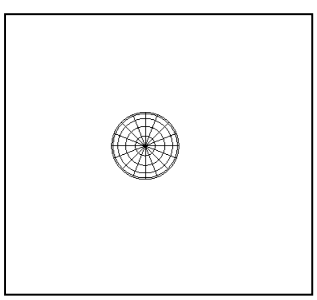
With this option enabled, surfaces are displayed as only their outer boundaries and trimmed loops. No additional lines will be drawn in the interior of the surface. This option makes the display of larger models much faster.

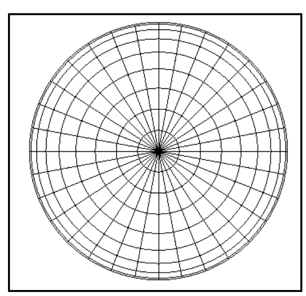
Selection radius

This option specifies the radius (specified in pixels) of the hit area for a selection pick. If this number is set small, then you must select very close to an object to select it. If it is set large, then picking may become more unpredictable

Refresh

The amount of surface detail that is displayed is dependent on the view. If you initially view a surface from far away, fewer lines (or polygons in the case of shading) are used to display the surface as shown on the left below.   If you zoom in on the surface and select *Refresh* from the View the part will be represented by in more detail shown on the right





Fast viewing

Fast viewing is a way to change your view on the fly without having to select a view mode in the middle of whatever you are doing. Fast view still relies on the last interactive viewing setting, but it can be accessed without changing modes. You can change the mode for fast viewing by selecting the viewing mode from the Fast View Mode submenu of the Viewing

Modes menu found in the View menu. Simply press SHIFT, click and hold the right mouse button, then drag the mouse. Your view changes just as if you were using the currently selected view mode.

Here is an example of fast view used during the chaining process.

1. Select Pan and Zoom in the Standard toolbar.
2. Now click Pick pieces in the Standard toolbar.
3. Pick some pieces with the cursor.
4. Use shift and a right mouse click to re-center your view.
5. A shift and a right mouse drag zooms in on your part where you clicked. In the zoomed view, you should have no trouble selecting specific pieces to build your curve.
6. Continue to pick more pieces now in view.
7. Press Alt+L to return to you last view.
8. Zoom in other parts as necessary to build your curve.

This same technique is useful for drawing parts too. You can set up this process by selecting Viewing modes from the View menu. Drag out the Fast view mode fly-out menu and select Pan and Zoom. A dot appears next to Pan and Zoom. Now use Shift+right mouse to pan and zoom as you work with your part.

Colors

Set your color preferences by choosing *Coloring* from the Options menu. A fly out menu appears where you select what item's color you want to set. You can set the colors for:

Change select, changes the color of the selected object regardless of the type of object.

Use default on selected, changes the selected object back to the color designated by its type.

Stock, the color of the stock wire frame model.

Feature, the color of unselected features.

Rapid, the color of the toolpath for rapid above-stock moves.

Toolpath, the line color for tools as they cut in a Show Centerline simulation.

Part line program, for toolpaths calculated with the part line program attribute.

Selection, the color of selected features.

Highlight, the color of highlighted items.

Background, the color against which all other colors are displayed.

Dimension, the color for dimension information and markings.

Curves, the color for curves after you have chained them.

Construction, the color for geometry as you are drawing it.

Surface, the color for surface wireframes and iso-lines.

After you select the item whose color you wish to change, a dialog box appears where you select the actual color. This dialog box is the same for each item you select.

Selecting objects by color or type

This dialog box allows you to select objects based on their color, their type or both color and type.

To select objects using the dialog box:

1. If you would like to select by color, select a color from the *Color* drop-down list, otherwise select *All* as the color.

2. If you want to select based on the entity type, select the entity type from the *Entity* drop-down list, otherwise select *All* as the entity type

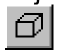
3. Click *OK*.

The settings of the drop-down lists are dependent on the objects that are shown in the graphics window. If you don't see the color or entity type listed, it is probably because there are no objects of that color or type currently displayed

Display mode bar and shading mode

By default, everything that you create in FeatureCAM is displayed as a line drawing in the graphics window. Objects can also be view as a 2D cross section, 3D shaded or 3D hidden line drawings. See the chart below for the display modes that apply to specific object types and how to activate each type of graphics.

Lines	Line drawings apply to points, geometry, dimensions, curves, milling features, turning features, surfaces, solids, stock. It is the default display mode.
2D view	Applies to stock for a turning setup and turning features. This mode is toggled by the 2D turned profiles button in the display mode toolbar.
Shaded	3D shading applies to milling features, turning features, surfaces, solids. There are a number of ways to shade and unshade objects. • Toggle between line drawings and shaded graphics by clicking the Shade button in the Standard toolbar. **Note that features are not shaded in this mode**. Use the shade selected button to shade features. • Shade specific objects on the screen by selecting the object and clicking the Shade selected button in the Display mode toolbar. • Click the Unshade selected button to return the selected object to lines or click the Unshade all button to return all objects to line drawings. These buttons arein the Display mode toolbar.
Hidden line	Hidden line graphics only apply to solids so it is best to hide all other

	objects and display only solids while in this mode. Toggle the Hidden line  button in the Display mode toolbar to clip and undisplay the hidden lines of a solid.

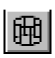
The following figures show a solid displayed as lines, hidden lines and shaded.

Display mode toolbar

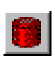
The buttons of Display mode toolbar, control the current display mode.

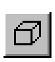
The Display mode toolbar has the following buttons to help you visualize and work with surfaces in the model.

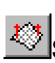
 **Shade selected** shades the selected entities. This gives you a solid like view of the surfaces.

 **Unshade selected** removes shading from the selected entities.

 **Unshade all** unshades everything that is displayed.

 **Hide shaded isoline** is a toggle switch to hide or show the lines of a surface or solid while it is shaded.

 **Hidden line mode** will display all solids as a hidden line drawing. Note that only lines within the solid are hidden. It is best to hide everything and show only the solids to use this mode. Unclick the button to turn off the mode.

**Show normals** displays the normals for selected surfaces in the model.

**Unshow normals** hides the normals for selected surfaces in the model.

 **Turning 2D view** toggles the view of a turning stage between 2D or 3D line drawing view.

Surface shading options

You can control how shading appears by setting different options by selecting Shading options in the Options menu. The basic options are set with checkboxes.

Lighting

Two lights uses light from two separate sources for more even lighting effects.

The lighting model has two different modes. With **Depth Cue** turned off, the lighting model uses light vectors (or directions). In this mode, the shading is not affected by the position of the light, but rather only the direction of the light. In this case you can specify the two light vectors as **Light Vector 1** and **Light Vector 2**.

With **Depth Cue** turned on, the lighting model uses lighting positions. Both the lighting direction and distance from the surface to the light affect the shading of a surface. In this case you can specify the two light vectors as **Light Position 1** and **Light Position 2**. The amount that the light is diminished is determined by the following formula:

1/(K + Linear * Distance of surface to light)

Both **K** and **Linear** variables are available for you to alter. Setting **K** or **Linear** to larger values this will darken the picture.

Material

Transparent enables the Opacity option.

Opacity sets how transparent or opaque the surface is. Settings range from 0 (transparent) to 1 (opaque).

Shininess affects how a highlight spreads. 0 provides a tight highlight. 1 creates a broad highlight.

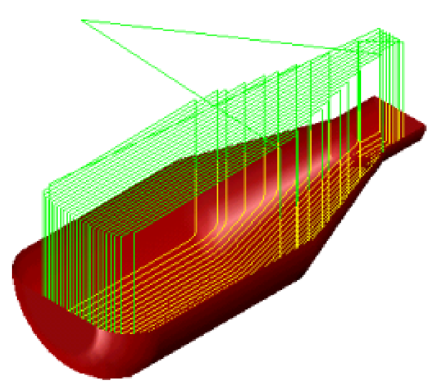
Ambient describes the amount of light present in the shading process besides the two light settings. This is a cumulative setting with Ambient set in the Lighting section.

Diffuse describes how the light spreads in the area occupied by the surface. This setting combines with the same setting in Lighting to determine the overall diffusing effect.

Display

The shading performed by FeatureCAM translates surfaces into triangles for shading. **Smooth shading** indicates that the shading of the triangles should be blended. With this turned off, the individual triangles will be visible.

Smooth Line will smooth out the jagged edges of lines. This applies to all line drawings including toolpaths. Note that this option can be time consuming.



Dithering is a setting applicable to shading with 256 colors. This mode is not recommended, but if you have only 256 colors, **Dithering** will use a dot pattern to simulate having more colors.

Z Buffer will sort the surfaces by their depth. It is not recommended that you turn this setting off.

Z Buffer Lines allows the lines to be sorted along with the surfaces. This allows surfaces to occlude lines that are positioned behind. This option is provided for viewing toolpaths in conjunction with shaded surfaces.

Backface Removed hides surfaces whose normals point away into the screen. For solid models, this option will speed up the shading. If portions of the model are visibly missing, turn this option off.

Use graphics hardware attempts to use your graphics card to speed up shading. Many graphics cards are not reliable for shading. If your shading looks bad or performs slowly, turn off this setting.

Backface Lighting applies when depth cue is off. It duplicates the light vectors in back of the model (for diffuse lighting).

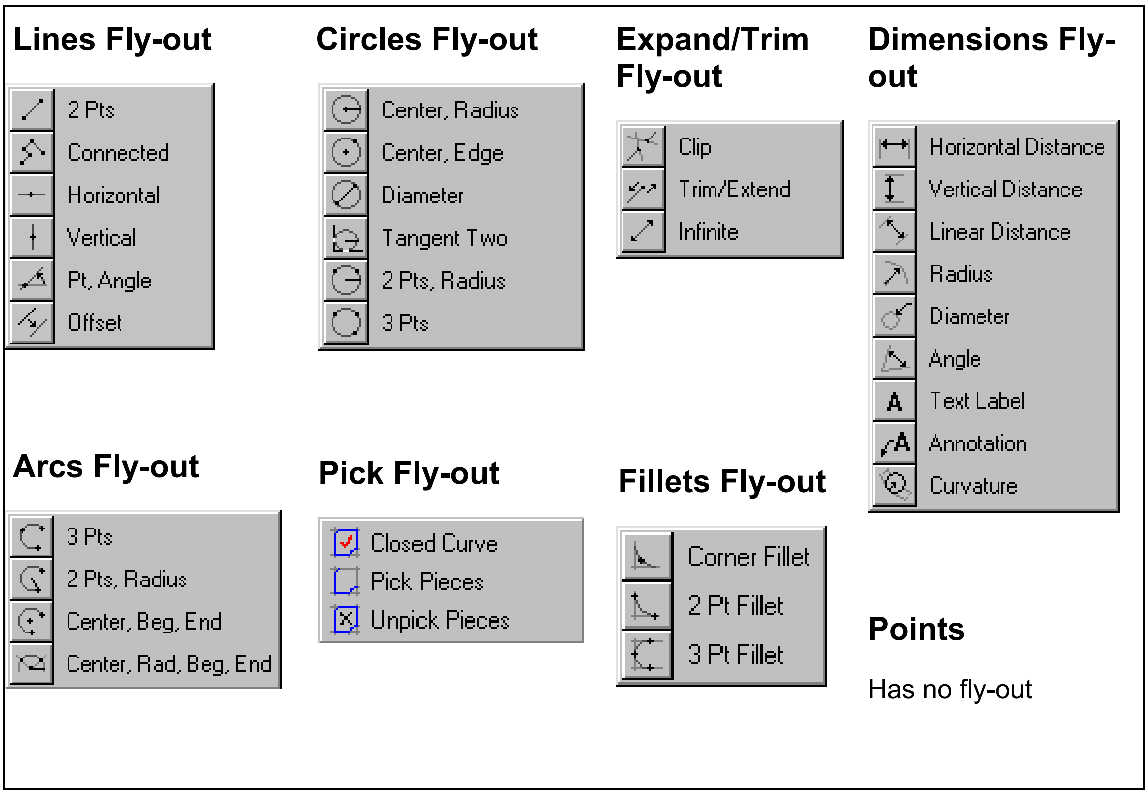
Backface Highlight applies when depth cue is off. It duplicates the highlighting affects for the duplicate lights.

Recommended shading procedures

1. By default turn Depth Cue off.
2. Adjust the Ambient material parameter to lighten or darken the shading.
3. If you are shading a model with many flat surfaces turn Depth Cue on.
4. If depth cueing is on, adjust the *Linear* parameter to lighten or darken the image.

Chapter 4
Drawing

The drawing tools are available from the Geometry toolbar. Each button in the toolbar is a fly-out toolbar with options shown below. See the *Getting Started with FeatureCAM Guide* for additional information on fly-out toolbars.



Points

Enter points either in absolute coordinates relative to your current UCS, or pick points directly with the mouse and snaps as needed.

Lines

Vertical line

Vertical line creates an unbounded vertical line through a point you select.

1. You can define points with the mouse or by entering coordinates.
2. You can edit the line by selecting it, then typing new values in the dialog bar clicking Update.
3. Trim the line later if you wish.

Offset line

Creates a line parallel to another existing line. The new line is also of the same length as the original line.

Select the existing line with the mouse.

1. Type your offset value in the dialog bar display at the Offset prompt. Note that you can also grab a line from the stock even though there is no line explicitly there.

2. Move the mouse pointer in the general area where you want the line to appear. The new line appears. If it is correct, click the mouse.

3. To change the offset line's position, select the offset line and change the sign and value at the D prompt in the dialog bar. Click *Update*.

Horizontal line

Horizontal line creates an unbounded horizontal line through a point you select.

1. You can define points with the mouse or by entering coordinates.

2. You can edit the line by selecting it, then typing a new value in the dialog bar and clicking Update.

3. Trim the line later if you wish.

Connected series of lines

Connected lines defines multiple lines in succession, where the endpoint of one line becomes the start point of the next.

1. You can define points with the mouse or by entering coordinates.

2. You can edit the line segments by selecting one, then typing new endpoint values, or angle and length values in the dialog bar.

3. Click Update.

Line from two points

Two Points defines a line by its endpoints.

1. Click the two endpoints of the line to define the line
   or

2. type in the coordinates of the endpoints in the dialog bar.

3. You can edit the line by selecting it, then typing new endpoint values in the dialog bar and clicking Update.

Arcs

Arc from two points and radius

Two points, radius constructs an arc through two points selected with the mouse or entered in the dialog bar, and with a radius as specified in the dialog bar.

1. Set the radius at the R prompt in the dialog bar.
2. You can select the points with the mouse and snaps, or by entering values in the dialog bar for any or all of the points.
3. If you use the dialog bar to enter the last point, click *Create* to make the arc.

Arc from center, radius, begin and end points

This selection constructs an arc with a center selected with the mouser or entered in the dialog bar of a specified radius and the starting and ending points of the arc.

1. Set the radius in the dialog bar.
2. If you select points with the mouse, the first point is the center.
3. Depending on the order sand where you place the next two points determines which direction the arc is built.
4. If you enter the last point in the dialog bar, click Create to make the arc.

Arc from center, begin and end points

Two Points, Begin, End Points constructs an arc through two points

1. If you select points with the mouse, the first point is the center.
2. Depending on the order and where you place the next two points determines which direction the arc is built.
3. If you enter the last point in the dialog bar, click Create to make the arc.

Arc from three points

Three Points constructs an arc through a start point, edge point, and a finish point.

1. You can select the points with the mouse and snaps, or by entering values in the dialog bar for any or all of the points.
2. If you use the dialog bar to enter the last point, click Create to make the arc.

Arc from center, radius, begin and end points

This selection constructs an arc with a center selected with the mouse or entered in the dialog bar. The arc has a radius, set in the dialog bar and begin and endpoints which you can set either with the mouse or with coordinates in the dialog bar. If you used the dialog bar to enter the last point, click Create to make the arc.

Circles

Circle from center and edge

Center, Edge defines a circle by its center point and a point on the circle's edge.

1. You define the center and then the radius of the circle with the mouse or in the text fields. The potential circle is drawn and follows the movements of the cursor between selecting the center point and selecting the point on the circle.

2. The default radius, (the last value entered, shown at the R prompt in the dialog bar) is automatically used unless you change it by typing or dragging.

3. For concentric circles, pick the center point, then set the Concentric switch in the dialog bar.

4. Pick the edge points for concentric circles.

Circle tangent to two entities

Tangent to two defines a circle by snapping tangent to both selected elements. When the elements are selected, the cursor position is very important in determining the position of the new circle.

1. Select the first point of the circle. The point selected will be a tangent point.

2. Select the second point. It too will be tangent. Select the point carefully, as it is your only control over the position of the circle.

Circle through three points

Three points defines a circle with three points on the circle's circumference. A circle tangent to three existing circles or arcs can be created by selecting the existing elements with the tangent snap active. If selecting arcs, the new circle may be tangent to the imaginary compliment of any of the arcs.

1. When selecting the lines, the cursor position determines on which side of the line the circle is drawn.

2. Use the mouse to pick the three points the circle must pass through. Or enter coordinates for any or all of the points. If you define the last point with numbers, click Create to make the circle.

Circle from two points and radius

Two points, radius defines a circle of a specific radius, set at the R prompt, with an unspecified center location and two individually defined point locations on the circle's circumference. The center of the circle is always placed to the left of the direction in which you define the two points through which the circle must pass.

1. Set the radius in the dialog bar.

2. Pick the first point with the mouse or enter coordinates for the point.

3. Pick the second point with the mouse or enter coordinates in the second point fields and click *Create*.

Circle from diameter and location

Diameter defines a circle by selecting two points that specify the diameter and location of the circle.

1. Select the first point. Select the point carefully, as the two points that you pick are your only control over the placement of the circle.
2. Select the last point. The distance between the two points is the diameter of the circle. The midpoint of the line between these points will be the circle center.

Circle from radius and center

Creates a circle from a center point and a radius value.

1. Set the radius at the R prompt in the dialog bar.
2. Select a center point with the mouse, aided by snaps as needed, or by entering coordinates in the dialog bar. If you entered the coordinates in the dialog bar, click Create to make your circle.

Fillets

Corner fillet

Corner fillet creates a fillet in a corner originally defined by an intersection of lines or arcs.

1. Select the corner fillet option.
2. Enter the radius in the dialog bar at the R prompt.
3. Now click in the corner where you want the fillet. As you move the mouse towards the corner FeatureCAM shows possible fillet positions. When you see the fillet you want, click the mouse button.
4. Creating the fillet automatically trims the existing lines.

Fillet through three points

Three points define a fillet by selecting three points, similar to the three point circle.

1. Select the Fillet option.
2. Use the mouse or dialog bar to enter the points.
3. The three points are the start, middle, and end points of the arc. The points are selected in CW or CCW order around the arc.
4. If the last point is entered in the dialog bar, click Create to make the fillet.

Two point fillet

Two point fillet creates a fillet in a corner originally defined by an intersection of lines or arcs.

1. The assistance bar prompts you to enter the radius of the fillet in the dialog bar and then

to select the corner lines.

2. When you select the first line, the fillet appears, snapping automatically to the closest valid snap point. You can move the cursor until the correct point is selected.

3. Click the second line of the intersection to create the fillet.

4. Creating the fillet automatically trims the existing lines.

Depending on your snap modes, you might not snap tangent to the existing lines. Non-tangent fillets don't chain automatically so watch the Status bar for the tangent message as you create two point fillets.

2D chamfer

Create a chamfer that trims surrounding geometry by:

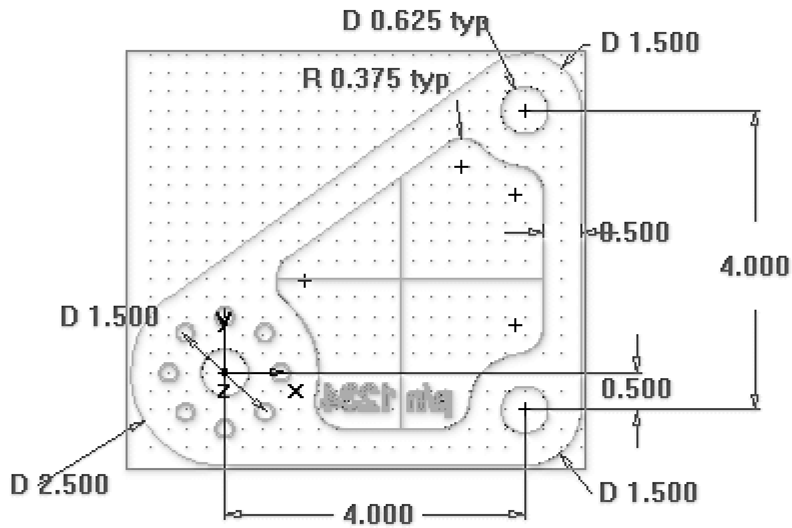
1. Selecting Chamfer from the Fillet fly-out menu.

2. Enter the Width and Height in the dialog bar.

3. Select the corner you want to fillet with the mouse.

Dimensioning

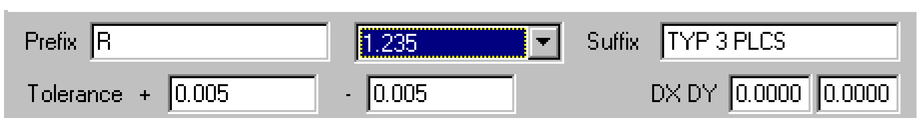
FeatureCAM has a complete set of dimensioning tools to create drawings like this one:



Inch

Dimension dialog bar

For dimensioning, the dialog bar has fields for including prefix text, suffix text, and other notations.



Prefix

Prefix is information entered in the dimension dialog bar that appears in front of the dimension in the drawing.

- Click the field.
- Type the information you want to precede the dimension.
- Remember to include a following space as needed.

Significant digits

Significant digits is a drop down list in the dimension dialog bar for setting how may significant digits are included in the dimension. Dimensions are shown to three decimal places by default.

Suffix

Suffix is information that appears after the dimension.

- Click the Suffix field in the dimension dialog bar.
- Type the information you want to follow the dimension.
- Remember to include a leading space as needed.

Tolerance

Tolerance + and - are fields in the dimension dialog bar for entering how much the dimension can deviate from the absolute listed dimension. The deviation allowances are printed along with the dimension.

DX DY

DX DY shows the relative offset of where the dimension will display from the last selected point that defines the dimension.

Horizontal dimension

Horizontal creates dimension information based on the horizontal axis of the part.

Vertical dimension

Vertical creates dimension information based on the vertical axis of the part.

Linear dimension

Linear creates dimension information based on the absolute distance between two points.

Radius dimension

Radius creates dimension information for the radius of the selected object.

Diameter dimension

Diameter creates dimension information for the diameter of the selected object.

Angle dimension

Angle creates dimension information for the angle between two selected lines.

Annotation

Annotation places explanatory text, entered in the dialog bar, with an arrow indicating what is being explained.

Label

Label places text, entered in the dialog bar, where you choose. The text is not available for engraving or other feature type modification.

Curvature

Curvature samples the surface and computes the curvature in two directions to describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real time rubber banding effect where you traverse the surface to find the point with the smallest curvature radius. Knowing the smallest radius tells you the smallest tool to use to manufacture the surface.

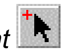
- Select curvature from the dimension fly-out.
- Move the cursor over the surface, particularly in the tight constrained areas of the surface or joints.
- Note the smallest value shown for curvature. That is the smallest tool end radius you need to accurately machine the surface.
- Set up rough and finish passes for the surface feature based on this knowledge, and make sure the tool is available for production.

Interrogation

This dialog box helps you extract numbers from the graphics window using snap modes and pick filters. You can then cut and paste these values into other dialog boxes.

To extract a value:

1. Select *Interrogation* from the *Dimensions* sub-menu of the *Construct* menu or select the *Interrogation* button from the Geometry toolbar.
2. The *Pick Dimension* dialog box is displayed.
3. Select a Pick type and Pick filter from among:
    - **Location**: Extracts the X, Y or Z coordinate of the point you snap to. The coordinate is dependent on the Filter setting. The coordinate system is based on the Alignment setting.
    - **Distance**: Extracts the distance between two points. The Filter setting controls whether the distance is measured in the X, Y, Z directions, in the XY plane (2D filter) or in 3D. The coordinate system is based on the Alignment setting.
    - **Same as**: Extracts the radius or diameter of a circle or an arc. With the depth filter enabled, the depth of a feature is extracted. Note that in the case of blind holes the overall depth of the hole is reported. This includes the extra depth to represent the drill tip.
4. Select an Alignment type from:
    - **Grid**: Calculates the values in the plane of the grid. See snap modes on turning on the display of the grid.
    - **UCS**: The points reported relative to the current UCS.
    - **Setup**: The points are reported relative to the current setup.
    - **None**: Uses the stock coordinate system

5. Click the *Pick point*  button and select one or two points depending on your pick type.

Using math to define fields and shapes

Turning input modes

For FeatureTURN, you can enter the coordinates of line geometry using one of three options available from the Options menu:

- 3D (XYZ): Coordinates are entered as X Y Z values.
- Radius (RZ): Coordinates are entered as a radius and Z coordinate.
- Diameter (DZ): Coordinates are entered as a diameter and Z coordinate.

Equations

You can use equations in numeric fields in the different properties dialog boxes. In parametric mode, the equation is always displayed. With parametric modeling off, the result of the equation is displayed.

Equations are input similar to the DOS command line format. The operators are listed in the Operators table. In a complex equation, multiplication and division operations are performed first, then addition and subtraction. Parentheses are also supported and can change the order of operation.

Real numbers can be specified in a number of ways, for example:

```
1.
.1
1.234
1.e2
.1e3
.1e-4
1.2e+6
```

Numeric arguments can be constants. The results of operators can be assigned to variables just like any other function:

```
x = 1
y = 2 * (x + 2)
z = y * 47.5
```

You can then use the variable in other numeric fields alone, or with other operations.

Operators table

+	addition, adds two numbers	acosd (num)	Computes the arccosine (in degrees) of a number.
-	subtraction, subtracts two numbers	atand (num)	Computes the arctangent (in degrees) of a number. Result range is -90 to 90.
*	multiplication, multiplies two numbers	atan2d (y,x)	Computes the arctangent (in degrees) of y/x. Result range is -180 to 180.
/	division, divides two numbers	ceil(num)	returns the nearest integer greater than or equal to a number.
sin (num)	Computes the sine of an angle (given in radians).	floor(num)	Returns the nearest integer less than or equal to a number.
cos (num)	Computes the cosine of an angle (given in radians).	fabs(num)	Returns the absolute value of a number.
tan (num)	Computes the tangent of an angle (given in radians).	sqrt(num)	Returns the square root of a number.
sind (num)	Computes the sine of an angle (given in degrees).	mm2in (millimeters)	Converts from milliliters to inches.
cosd (num)	Computes the cosine of an angle (given in degrees).	exp(num)	Returns e^x where e = 2.71828.
tand (num)	Computes the tangent of an angle (given in degrees).	log(num)	Returns ln(x) where ln is the natural logarithm.
asin (num)	Computes the arcsine (in radians) of a number.	log10 (num)	Returns the base-10 logarithm of a number.
acos (num)	Computes the arccosine (in radians) of a number.	pow(base, power)	Returns a base number raised to a power.
atan (num)	Computes the arctangent (in radians) of a number. Result range is -pi/2 to pi/2	degtorad (num)	Returns an angle in radians as converted from degrees
atan2 (y,x)	Computes the arctangent (in radians) of y/x. Result range is -pi to pi.	radtodeg (num)	Returns an angle in degrees as converted from radians
asind (num)	Computes the arcsine (in degrees) of a number.	pi	The mathematical value of pi to ten decimal places.
in2mm (num)	Converts inches to millimeters.	radiusxy(x,y)	Computes the distance from the origin to (XY)
anglexy(x,y)	Computes the angle between the X-axis to the line between (0,0) and (x,y)		

Layers

For viewing, editing and selecting convenience, you might want to create all of your geometry on a layer separate from the rest of your part. The same is true for curves and other elements. Layers are a good organizing tool.

You can't delete or rename the World Axis, Setup, UCS Axis or Stock layers. The program requires those three layers in the part file definition. When a layer is selected, every entity created afterward, including curves, is placed in the new active layer.   Elements can be changed from one layer to another using the Change Layer command in the right mouse button pop-up menu or from the Show fly-out. Change Layer is also available in the Edit Menu.

Layers have a long history in CAD tools. Traditionally, layers are used to organize similar parts of a design or drawing. In FeatureCAM, for example, you might choose to put all geometry on one layer, named geometry, and all features on another, named features. Besides organizing your design elements, layers are also useful for controlling the view. If your layers are well organized, you can turn the display of different layers on and off for even finer control than the Show and Hide menus offer. Setups and UCS can be used in similar ways.

Change layer

- Select or right-click a geometry element.
- Choose Change Layer from the pop-up menu, the Show fly-out or the Edit menu.
- Select the layer where you want to move the geometry in the list box.
- Click OK.

Creating a new layer

- Select Layers from the Show fly-out, or the Edit menu.
- In the Layers dialog box, click New.
- This same dialog box is accessible from the Layers button in the Show toolbar.
- In the Name Layer dialog box, type the name for your geometry layer, such as Geometry.
- Click OK

The new layer appears in the Layers dialog box. You can turn the view and editing capabilities of layers on or off with the check boxes in this dialog box. You could turn off the other layers now so your geometry is drawn only on the geometry layer. If you have other existing geometry or points you need to construct your geometry, leave the other layers active.

- Click OK to close the Layers window.

Editing drawings

Undo

Undo returns FeatureCAM to the state prior to your last change. Undo has multiple levels limited only by system resources.

Redo

Redo restores any undone changes.

Edit

The edit functions let you work with existing geometry to fine tune your drawing to the exact shapes needed, or to help you correct errors you made while drawing. How geometry is edited can be affected by two settings: Parametric Modeling and Multiple Regions.

Trim or extend

Trim/extend change the length of a line or an arc.

1. The element can be shortened or lengthened to any desired length by picking a new end point for the selected arc or line.
2. If the newly selected endpoint is not on the arc or line, the system automatically trims/extends the selected element to the perpendicular drop from the selected endpoint.
3. This does not change an element's orientation (location, angle or radius).

Clip

Clip edits an element to the boundaries closest to the cursor position.

1. Only click elements you want to delete.
2. The Assistance bar prompts you to select only the element being edited and then automatically searches for the boundaries of that element closest to the cursor position.
3. The element is then trimmed back to those boundaries.
4. Clipping provides interactive feedback as you position the mouse over geometry.

Restrictions:

1. You can trim curves against lines and arcs, but you cannot trim lines or arcs against curves.
2. You cannot trim curves against other curves.
3. Infinite lines and circles cannot be clipped unless they are crossed by a line or arc.

4. Interactive feedback works only for lines and arcs.

Infinite

Infinite, the last option in the fly-out, completes circles or arcs that have been trimmed or are otherwise incomplete. It can also extend lines to infinity.

1. Select Infinite from the fly-out.

2. Click the entity you want to complete or extend.

Multiple Regions

In the Options menu is an option, Multiple Regions, which affects how Trim, Extend and Clip functions work. For example, with Multiple Regions off, a clipped geometry might be considered as multiple separate lines. Turning on Multiple Regions, the clipped geometry is considered one line, even though it displays in multiple segments. Selecting either segment selects the entire line, or both segments. Trim and extend can extend separate segments, or extend visible portions of lines depending on the setting of Multiple Regions.

Transform

Right clicking a geometry entity opens a pop-up menu where you can select Transform. You can also select Transform from the Edit menu or the Standard toolbar.

1. Selecting Transform opens a wizard.

2. There are four radio buttons to set the type of transform:

3. Translate selected entity elements to a new location. You can move an absolute distance as specified in XYZ vectors, or you can move from point to point.

4. Rotate selected elements about a selected location to a specified angle, referenced from the positive X-axis.

5. Scale proportionally reduces or expands selected elements about a specified point.

6. Reflect mirrors the element about a line. The line can be an existing axis, or any other line including one created just for reflecting around. The object can be flipped top to bottom, left to right or even both depending on the line you choose to reflect around.

7. Set whether you want to **move** the original or **Copy** the original element. If you are copying, more fields appear for setting the number of copies you want in the new location. When copying with Parametric Modeling enabled, you can also set whether you want to link the copies to the original so any changes to the original are carried over to the copies.

Reflect (transform)

1. Right-click the entity.

2. Select *Transform*.

3. Select *Reflect*.

4. Set the *Move* or *Copy* radio button. If you chose Copy, set the number of copies and

Keep link to object box as needed.

5. Click *Next*.

6. Select the axis or line you want to reflect around.

7. Click *Finish*.

Rotate (transform)

1. Right-click the entity.

2. Select Transform.

3. Select Rotate.

4. Set the Move or Copy radio button. If you chose Copy, set the number of copies and Keep link to object box as needed.

5. Select the point to rotate about with the mouse and snaps, or enter an XYZ coordinate in the boxes.

6. Click OK.

Scale (transform)

1. Right -click the entity.

2. Select Transform from the pop-up menu.

3. Double click the Scale field and type in the scaling factor.
   Scale specifies the amount of scaling to be performed (e.g. a scale factor of 0.5 generates geometry at one-half the size of the original for the first copy).

4. Set the Move or Copy radio button.

5. If you chose Copy, set the number of copies and Keep link to object box as needed.

6. Select the point.
   The point selected sets a distance that the transformed element uses as a relative location. In the earlier example with a scale factor of 0.5, the distance between the point selected and the element would be decreased by half for the new element. This is even more obvious with multiple objects as you probably want to select the center point of the multiple objects to maintain proportional spacing in the final drawing.

7. Click Finish.

Translate (transform)

1. Right click the entity.

2. Select Transform from the pop-up menu

3. Select Translate in the dialog box.

4. Set the Move or Copy radio button.

If you chose Copy, set the number of copies and Keep link to object box as needed.

Distance

5. Enter the distance you want to move the entity in the X, Y, and Z directions.

Point to Point

5. Enter the coordinates of a point on the entity in the From fields or click in the From area and pick a point with the mouse.

6. Click OK.

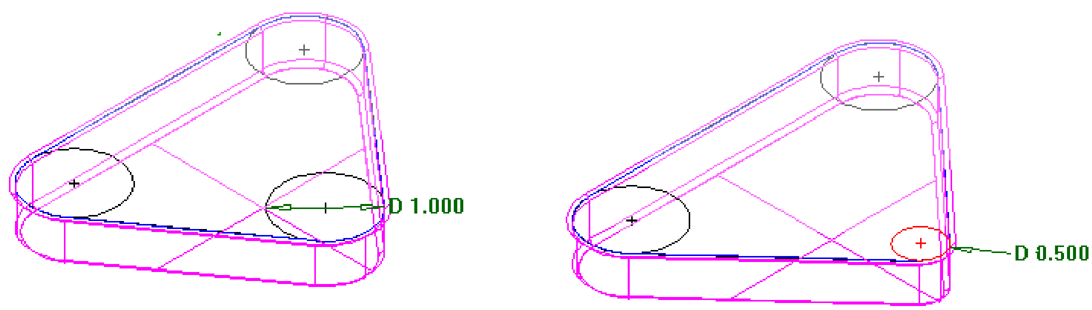
Parametric modeling

Parametric modeling means that objects that are created (lines circles, arcs, etc.) are linked to entities they snapped to during creation or that were used to build them.

By default, parametric modeling is off and is set with a toggle switch in the Options menu. If it is enabled, FeatureCAM remembers the connections between the objects you create on the screen. For example, if a line was created tangent to two circles, this relationship is stored. At a later date, if you change the location or radius of one of the circles, the line is updated to reflect that change. These relationships are maintained all the way through your FeatureCAM model.

In the example below, a pocket was created from a curve. This curve was created from a series of circles connected by tangent lines.

By editing the diameter of one of the circles, the end points of the lines change, the curve changes and the pocket changes as shown in the right-hand figure.



Decimal places dialog box

The number of decimal places that are displayed in FeatureCAM dialog boxes is determined by the settings in the *Decimal places* dialog box. There is a drop-down list for selecting the number of decimals for English (inch) and metric (millimeters) documents. Select the item in the list that shows the right number of decimal places. For example if you want four decimal places displayed, select *0.1234* from the drop-down list.

Even though you have selected a certain number of decimal places, the fields of a dialog box may not be wide enough to display a number that wide unless you click on the field and use the right and left arrow keys on the keyboard to scroll.

To set the number of decimal places:

1. Select *Decimal places* from the *Options* menu.

2. Select the number of decimal places for English and Metric documents
3. Click *OK*.

Chapter 5
Curves

Closed curve definition

Closed curves have start and end points in the same location, and at least one other point (not in that location) included in the curve. The closed curve clearly defines an area as the interior of the curve and completely separates this area from the exterior of the curve. Any ambiguities such as overlapping curve links or intersecting curve links cause failures and unpredictable results in the machining routines in FeatureCAM.

Open curve definition

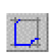
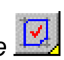
Open curves have end points that do not meet. Open curves may only be used in Side and Groove features.

Chaining

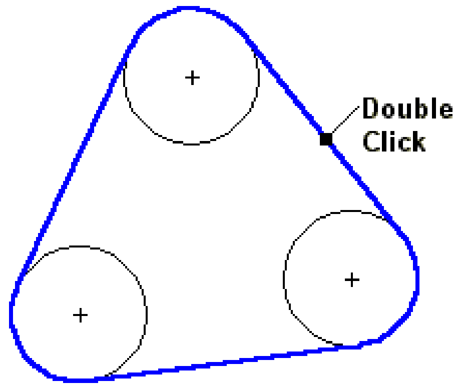
Overview of chaining

Lines, circles and arcs typically represent the shape of a part. To use a sequence of lines and arcs as the shape of a feature, you must chain them into a curve. Chaining is the primary way of creating curves by connecting pieces of geometry. In many cases you are not required to trim away pieces of geometry. Chaining will automatically prefer smooth, tangent-continuous paths since these paths are more conducive to manufacturing.

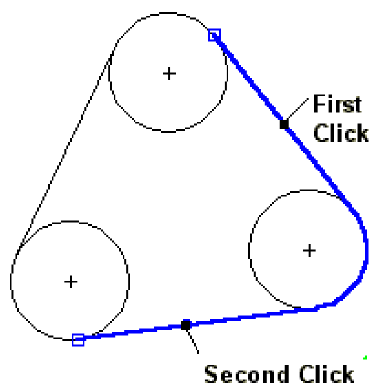
You can enter chaining by either:

Clicking the *Pick Curve Pieces*  button, or clicking the *Closed Curve*  button.

In both cases you enter chaining, but in different modes. In Closed Curve mode, you create closed loops by clicking once. If you do not receive the desired curve, change to Pick Curve Pieces mode. In Pick Curve Pieces mode, closed loops are created by double-clicking as shown below.

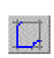
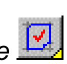


If you click twice in two places, chaining will select both the beginning and ending piece of geometry and attempt to find a smooth path that connect them as in the figure below. Notice the blue squares at the end of the blue curve. These indicate that the curve is not a closed loop. You can continue to click, adding pieces to the curve until you chain all the desired geometry. In Pick Curve Pieces mode, you can create open curves or closed curves that form loops.



In addition to showing the chaining mode, the dialog bar also contains radio buttons for controlling the plane of the chaining. The *Grid*, *UCS* and *Setup* buttons restrict the plane of chaining. The *Unrestricted* button, allows chaining in 3D.

How to chain lines and arcs into curves

1. Clicking the *Pick Curve Pieces*  button, or clicking the *Closed Curve*  button. The dialog bar changes to reflect the parameters useful for chaining.
2. Optionally name your curve. Good naming helps you keep track of what each curve in your part is for, so use the most descriptive name you can think of.
3. Select the plane you want to chain in. Your 2D choices are *Grid*, *UCS* or *Setup*. Note that the grid will rotate to the face of your stock that is most forward facing. Rotate your

part to change the plane of the grid. If you want to chain in 3D, click *Unrestricted.*

4. If you are trying to create a closed curve, click the *Closed Curve* button in the dialog bar and click on a segment of the desired geometry. If this does not chain the proper geometry click the *Pick Curve Pieces* button in the dialog bar and pick intermediate points along the curve to form the loop. If a click follows an undesired branch, click the undo button and click a point before the branch point.

5. If you are trying to create an open curve, click the *Pick Curve Pieces* button in the dialog bar and click on the beginning segment, and then click on the ending segment. If a click follows an undesired branch, click the undo button and click a point before the branch point.

6. Click *Create* in the dialog bar.

Restrictions of using pick pieces (chaining) for creating curves

- All of the geometry must be shown on the screen.
- Pick Curve Pieces doesn't work for curve pieces. You can only connect arcs, lines and circles. If you want to connect curve segments use Curve join.
- The geometry must be tangent continuous (smooth). Even if your geometry looks smooth, it may not be. Make sure that you have used the tangent snapping when creating the geometry.
- Chaining has a limit on its search path length in order to improve performance. If you are working with data with a lot of small pieces you may need to increase this limit.

Troubleshooting pick pieces (chaining)

1. **I can't form a closed loop.**

- Your data may be too complex to use the double click interface. If double clicking (or single clicking in closed curve mode) doesn't work, you can click the start segment, then click the next segment you want added to the curve and proceed until you have selected the complete curve. If there is only one simple path between points, you can click a couple of segments apart and FeatureCAM finds the path that connects them.
- The whole target curve must be visible on screen. Show the additional geometry if it is not displayed.
- If you are working with data with many small pieces you may need to increase the Double Click Depth or Single Click Depth found in Chaining Options. To see if this is the case, go into select mode and then pick pieces of your geometry.
- The intended curve must lie in one plane parallel to the current setup.
- Arcs and fillets need to be tangent or chaining may choose the other branch. Make sure that the geometry is really tangent by recreating the geometry that won't chain by using the Snap to tangent snap mode. You may want to turn off the other snap modes to ensure that you are truly snapping to the tangent.

- Gaps between endpoints must be less than that set in the Chaining Options. Look for blue squares in the chaining data. They indicate that the curve is not connected to another piece at that point. Zoom in on that data. Select the pieces in the gap. If there are no pieces, you may need to increase the End point tolerance.

2. **The loop is selected is wrong**.

- If double clicking is selecting the wrong loop, the data may have sharp corners or the algorithm may just be making some wrong decisions. If double clicking doesn't work, you can click the start segment, then click the next segment you want added to the curve and proceed until you have selected the complete curve. If there is only one simple path between points, you can click a couple of segments apart and FeatureCAM finds the path that connects them.

3. **I can't pick the pieces.**

- You are probably picking pieces that are already curves. You must use Curve join

4. **Chaining won't work in 3D.**

- When you enter chaining mode, click the *Unrestricted* button in the dialog bar. This will free chaining to pursue links that are not planar.

5. **Chaining is working in the wrong plane.**

- The *Grid*, *UCS* and *Setup* buttons restrict the plane of chaining. Click the appropriate button for the desired plane and re-chain.

Setting chaining options

You can improve your chaining results by setting up some options for FeatureCAM to follow. Select Chaining in the Options menu.

Avoid sharp corners is a checkbox that when set, causes the chaining process to choose other options first over a sharp (acute) corner option. You can turn this option on or off depending on what you are chaining or to match your preferences.

Endpoint tolerance is a field for setting distance sensitivity for the chaining process particularly designed to improve the chaining of imported geometry. The distance chosen sets how close geometry have to be for the chaining algorithm to assume they intersect. Your setting is saved and reused from session to session.

Double click depth sets the number of segments the program will analyze in each direction to connect your start and end points. Imported data may need a higher setting than data created in FeatureCAM.

Single click depth sets the number of segments the program will analyze in each direction to connect your start and end points for manual chaining. Imported data may need a higher setting than data created in FeatureCAM.

Unpick pieces

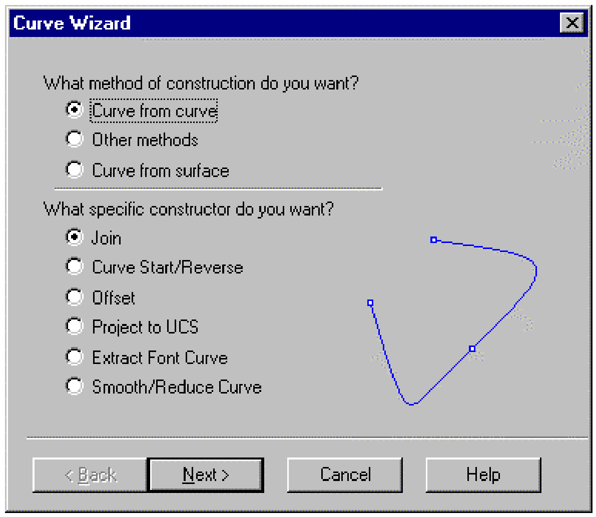
If you do get a curve, but it includes some incorrect pieces, use Unpick to clear the wrong sections from the curve, or click Clear Curve Pieces to discard the entire effort and start over.

When you have the correct curve chained, click Create to enter the final curve into FeatureCAM for later use.

Custom curve types

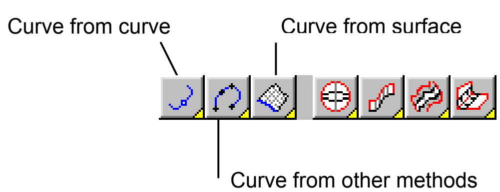
You can also create curves from methods other than chaining. These methods are available through the curve wizard or directly from the Curve and Surface toolbar.

To bring up the curve wizard click the curve wizard button. This brings up the following dialog box.



Select a category of curve construction methods in the top half of the dialog box. This reveals the constructors in the bottom half of the box. Select the constructor and click Next to display the constructor's dialog box.

The Curve and Surface toolbar also provides direct access to these constructors. There is also a fly-out menu for each category as shown below:

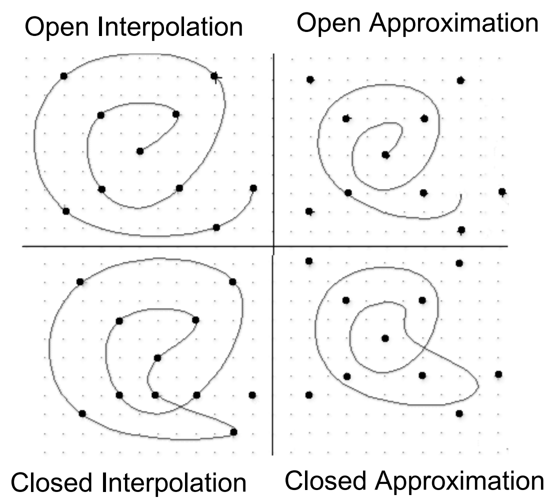


Curves from other methods

Splines

Splines are similar to Beziér curves. You specify points sequentially, either with the mouse, snapping, or directly through X Y Z coordinate prompts. You can enter as many points as you wish. As you create each point, a curve is drawn approximating each point or through each point.

The options button in the dialog bar toggles the spline through four settings. The settings control whether the curve is an open or a closed interpolation or an open or closed spline. The four splines in the diagram all have the same defined points. They each use a different spline option as shown.

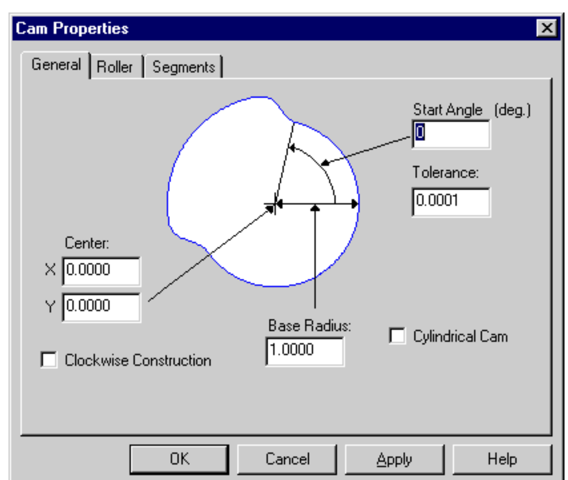


When you are done with the spline, press Create, or choose a new mode or tool and FeatureCAM automatically saves the spline curve. You don't have to chain splines to make them function as curves for building profiles.

Cams

General tab

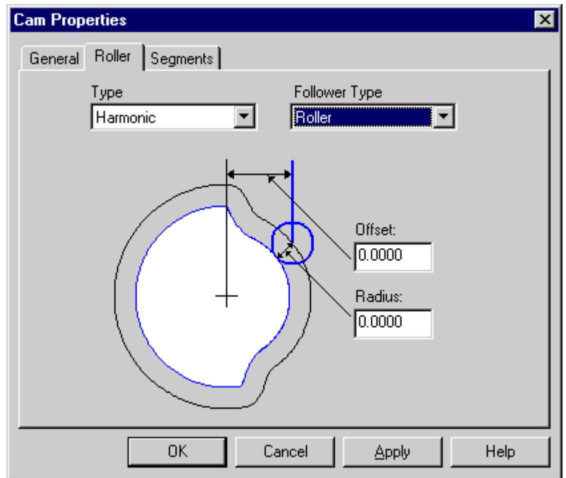
The general tab contains entry fields that define the basic dimensions on which the specific cam attributes are based.



- Set the coordinates for the center of the cam either explicitly or click the Pick button and pick the point with the mouse. **Center** sets the X and Y coordinates for the center of the cam body.
- Set the **Base Radius,** the radius of the circle that defines the body of the cam. This dimension is the minimum distance between the cam's center and the follower.
- Set the **Start Angle.** Start angle defaults to parallel to the X axis. Enter an angle in degrees to move the start angle. The direction of rotation for the start angle is controlled by the Clockwise Construction check box.
- Decide how to set **Clockwise Construction.** This sets the cam to be constructed from sequential segments arranged in a clockwise rotation. The default is for counterclockwise construction when the box is not checked. Note that the Start Angle is also changed by this setting.

- **Cylindrical Cam:** If you have 4th axis support, you can also create cylindrical CAMS or barrel CAMS. Design you CAM as usual, but check the Cylindrical CAM check box.

Roller tab



The Roller tab contains fields that describe the how the cam is followed (by a roller).

- Set the **Type** which describes the follower motion and its associated acceleration diagram commonly described as:
  Harmonic
  - Parabolic
  - Cycloidal
  - Modified Sine
  - Harmonic
  - Modified Trapezoid
  - 3-4 Polynomial
  - 3-4-5 Polynomial
  - 4-5-6-7 Polynomial

- Set the **Follower Types** for the kind of follower the cam is used with. It can be either Flat- or Roller. Setting the follower to Roller activates the last two fields for you to set.

- **Offset** sets the distance between the center line of translation for the follower and the cam's center which controls the pressure angle on the follower. This is not the cutter offset.

- **Radius** sets the radius of the roller that follows the cam's shape. If a roller-type follower is selected, then a radius must be entered. A zero roller radius simulates a knife-edge follower, or constructs the pitch curve of the cam.

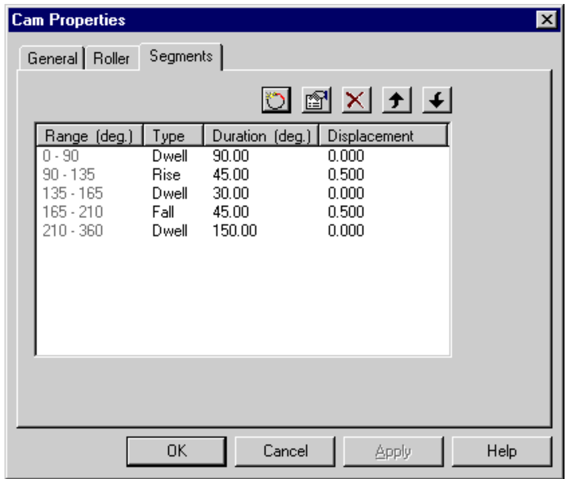
Segment tab

This tab defines the cam segments and their specific parameters. The segment arcs are listed in counterclockwise sequence in the list box area.

- Create new entries by clicking New , to edit an entry, double click the entry or select the entry with the mouse, then click Edit (the second button in from the left). This opens the Edit Cam Segment dialog box

- Use the jagged x button to delete a selected entry.

- Click the arrows to move entries up or down in the list box, changing how the cam is laid out.



Edit Cam Segment

Set the segment type:

- Dwell indicates an arc of rotation that neither rises nor falls but whose diameter from the center is determined from the ending displacement of the directly preceding segment. Duration sets how many degrees of rotation the dwell lasts. The Displacement field is not available for dwell segments

- Rise indicates that this segment rises to a greater diameter than the segment preceding this segment. If this segment is the first segment, rise is calculated from the base circle defined in the General tab. Duration sets how many degrees across which the rise occurs. Displacement sets how far the rise deviates from the previous segment or the base circle if the segment is the first segment defined for the cam.

- Fall indicates that this segment decreases from some displacement to a lower displacement, but never less than the base circle defined in the General tab. So a fall shouldn't be the first segment defined. Use Cams to create the geometric profile of various reciprocating cams. It is the actual profile of the cam, not the pitch curve (center line of the follower).

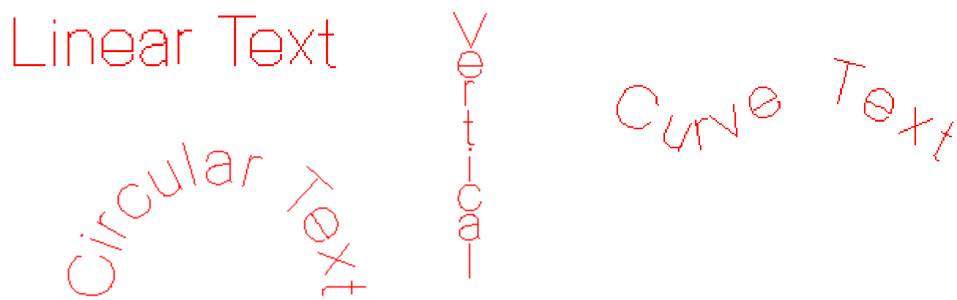
Engraving

Overview of text engraving

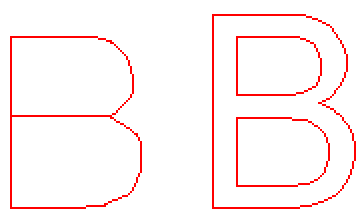
Engraving in FeatureCAM has two steps:

1. Create the text as a curve.
2. Use the curve for a simple groove feature.

Text can be created horizontally along a line, vertically along a line , along a circle or along a curve.



The fonts used are standard windows outline fonts and a custom single line font included with FeatureCAM called Machine Tool SanSarif. Single line fonts use single strokes for the letters. Outline fonts represents the boundaries of the letters. The figure below shows the difference between the two types of fonts.



Regardless of the font used, a groove feature will trace each line of the font, not the region between the outlines. If you are looking for simple engraving, the single line font is recommended.

Once you create the text, use the resulting curve to create a simple groove to engrave the text into your part. Bosses and pockets are possibilities too, but any feature based on text may require specialized small tools for their manufacturing processes unless you use the Machine Tool San Serif font.

The settings for scaling, spacing and fonts are saved for the next time you create a text string

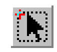
How to create linear text

1. Select Text from the Curve toolbar or from the Other Methods section of the Curve Wizard.

2. Type the text string as the Text.

3. Optionally type in a new name for the curve that will be created from the text as the Curve Name.

4. Select Linear as the path type.

5. Type in the X, Y and Z Locations or click the pick point  button and pick it with the mouse. This point will be used along with the Justification setting to locate the text.

6. To rotate the text counter clockwise around the Location point, enter a Direction angle between 0 and 360.

7. To create the text vertically, click the Vertical button.

8. To invert the text click the Reverse button.

9. Click the font button to select the font. Any Windows font on you system will be available. For simple engraving, the font, Machine Tool SanSarif is recommended. It is recommended that you set the font size to be 72 points. This will allow you to easily control the size using the scaling fields. See *Common text fields* for more information.

10. Optionally fill in the Common text fields

11. Click OK.

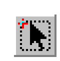
How to create text along a circle

1. Select Text from the Curve toolbar or from the Other Methods section of the Curve Wizard.

2. Type the text string as the Text.

3. Optionally type in a new name for the curve that will be created from the text as the Curve Name.

4. Select Circular as the path type.

5. Type in the X, Y and Z coordinates of the circle Center or click the pick point  button and pick it with the mouse.

6. Enter the Radius of the circle.

7. To rotate the text counter clockwise around the Location point, enter an Angle between 0 and 360.

8. To create text on the bottom quadrants of the circle, enter the appropriate angle and click Reverse.

9. Click the font button to select the font. Any Windows font on you system will be available. For simple engraving, the font, Machine Tool SanSarif is recommended. It is recommended that you set the font size to be 72 points. This will allow you to easily

control the size using the scaling fields. See *Common text fields* for more information.

10. Optionally fill in the Common text fields
11. Click OK.

How to create text along a curve

1. Select Text from the Curve toolbar or from the Other Methods section of the Curve Wizard.
2. Type the text string as the Text.
3. Optionally type in a new name for the curve that will be created from the text as the Curve Name.
4. Select Curve as the path type.
5. Pick the Curve from the drop down list or use the pick curve  button to graphically select the curve.
6. If you want to reflect the text to the other end of the curve, click reverse. (Note that changing the justification will move the text to the other end of the curve without flipping the text.)
7. Click the font button to select the font. Any Windows font on you system will be available. For simple engraving, the font, Machine Tool SanSarif is recommended. It is recommended that you set the font size to be 72 points. This will allow you to easily control the size using the scaling fields. See *Common text fields* for more information.
8. Optionally fill in the Common text fields.
9. Click OK.

Common text fields

Justification: Select from among left, center and right. These settings positions uses the point you enter as either the left end, center, or right end of the text.

Alignment: The alignment is a translation factor for X and Y for the entire text string. Use these fields to tweak the location of the string.

Scaling: Scale XY fields control scaling of the text. A value between zero and one shrinks the text. Values greater than one expands the text. You can have different values in the fields to stretch text for special effects. A negative value reflects the text, useful for making molds. If you set the font to be 72 points, this number reflects the size in inches of the font in the given direction.

Spacing: The spacing is the size of the gap between the letters. It is uniform across the entire text string.

Functions

The Functions dialog box is accessed by selecting Shapes in the Construct menu, then selecting functions, or from within the Curve wizard.

With functions, you create user-defined mathematical relationships to generate a graphical figure. Functions can be of four types:

y=F(x)
x=F(y)
r=F(a)
x=F(t), y=G(t)

The variables a, r, t, x, and y are local to the functions dialog box. Any previous values that you have set for these variables will be ignored. However, you are free to use any other previously defined variables. In addition to variables, you may use any predefined functions or constants discussed under Equations.

You build functions in the Function dialog box accessed by selecting Shapes in the Construct menu, then selecting Functions from the Shapes fly-out menu.

Trigonometric functions are often used in constructing these functions. Be careful to use sind(), cosd(), etc. when using degrees and sin(), cos(), etc. when using radians.

Chained

Chained is a checkbox to pre-chain your function shape for you. When set, you can't return and modify the function so be sure you have the function correct before setting Chained.

Start

Start sets the starting point for the range over which your function is evaluated

End

End sets the ending point for the range over which your function is evaluated

Increment

Increment sets the value added to or subtracted from the previous point evaluated for the function to determine the next value to be run through the function.

Preview (functions)

Preview generates the curve and displays it for your review but doesn't apply the generated function to the drawing as geometry. Depending on your start and end points and the increment, it may take a while to evaluate and build the preview image.

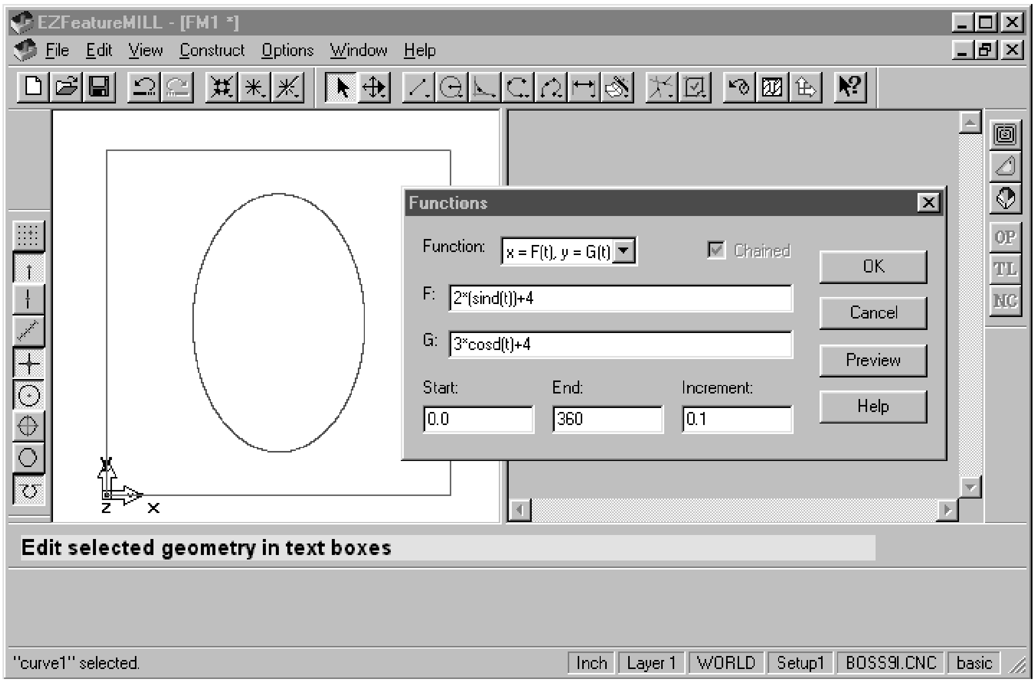
Function examples

Ellipse example

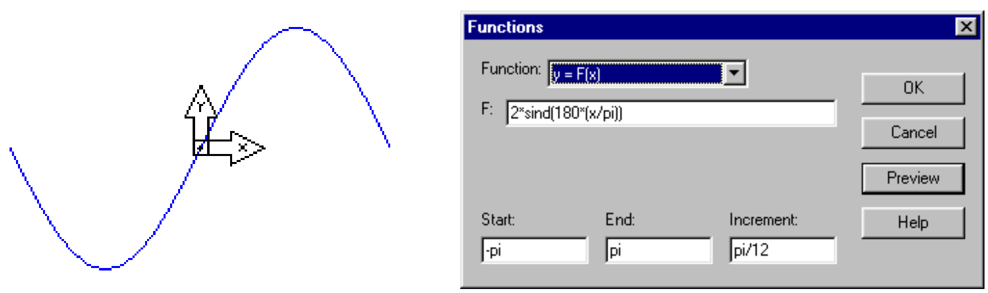
For a quick example, consider an ellipse. You can use either radian or degree based math, but be sure you use a range for x appropriate to your math system. Using a radian system, but specifying the range from 0-360 (degrees) does not work.

A simple description of an ellipse in degrees is:
x = <width> * sind(t) + offset; y = <height> * cosd(t) + <offset>

If you don't specify an offset, the ellipse is centered on the current UCS. The diagram below shows an ellipse defined and previewed in FeatureCAM.

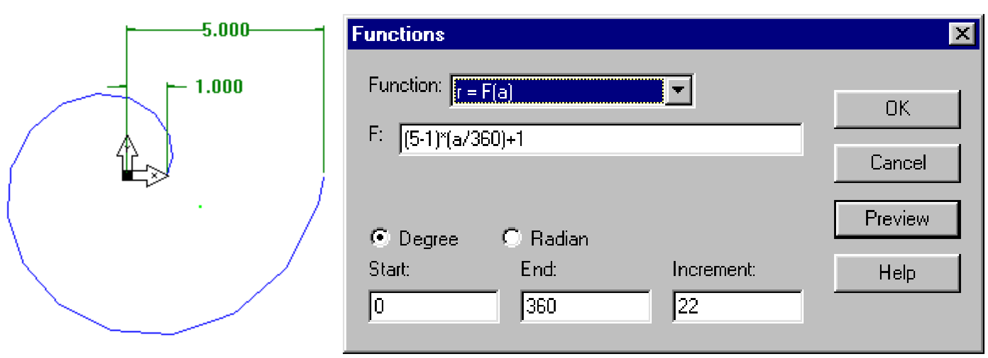


Y=F(x) example



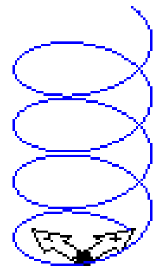
R=f(a) example

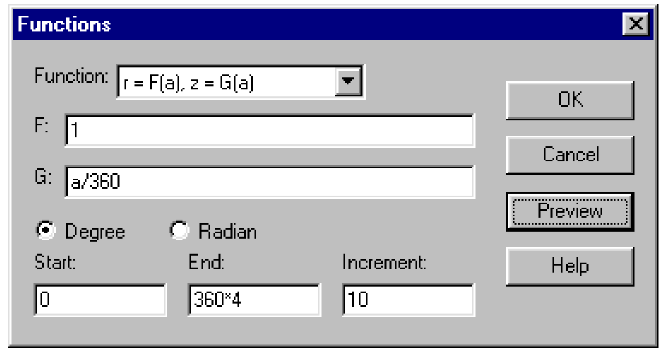
In the drop down list box, select the r=F(a), which is useful for describing polar functions where the radius is calculated as a function of the and angle or argument variable.



R=F(a), Z=G(a) example

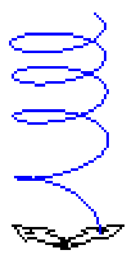
Use this function for polar functions with a Z coordinate that is specified as a function of the angle. A helix can be modeled with such a function.

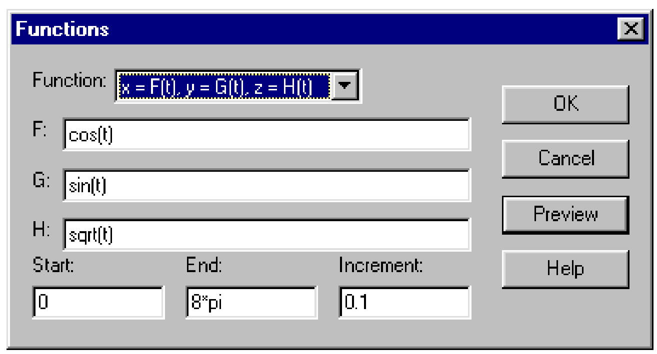




X=F(t), y=G(t), z=H(t) example

Use this function when x, y and z are parametric functions.





Polar coordinate examples

You can input polar coordinates in any field that accepts point locations.

- Specify the coordinate with the keyword *polar*, so an X coordinate is *polarx*, a Y coordinate is *polary* and so on.
- Sets the angle to be read as degrees, so polarxd means that you are entering a polar coordinate for X with the angle of rotation specified in degrees. Omitting the d specifies the angle in radians.
- In parentheses, set the length of the offset, the rotation angle (which can be negative) and optionally, a specific center point. The same point in degrees and radians is shown below:

polarXd(1,45,2.5)	polarYd(1,45,3.0)	degrees
polarX(1,pi/4,2.5)	polarY(1,pi/4,3.0)	radians

Notice that this point includes a specific center point. If you don't specify a center point, the current origin for the setup is used. Rotation is specified from or parallel to the x axis of the current setup.

The math for generating these points is:

polarXd(r,theta,Xc) = r(cos theta) + Xc
polarYd(r,theat,Yc) = r(sin theta) + Yc

Rectangle curve

The rectangle curve tool creates a rectangular shaped curve in the plane or parallel to the plane of the current UCS. It can be created two ways.

Use corner, width and height

Type or graphically pick the corner point and enter the width and height dimensions. Enter the elevation if you want the rectangle to be out of the UCS plane.

Use center, width, height

In this mode, create the shape by entering or picking the center point and the width and height dimensions. Enter the elevation if you want to translate the rectangle out of the plane of the UCS.

For either method, you can specify a corner radius and a counter clock-wise rotation angle.

Create as arcs and lines

If this checkbox is checked, the rectangle is converted to arcs and lines when the *OK* or *Finish* buttons are clicked.

Ellipse curve

The Ellipse curve tool creates an elliptically shaped curve in the plane of the current UCS or a plane parallel to the UCS plane.

Axis endpoint 1 is defines one end of the axis.

Axis endpoint 2 defines the other end of the axis.

Height defines the height above the axis. (Note that the total height of the ellipse is 2*Height.)

Elevation is the offset out of the plane of the UCS.

Create as arcs and lines converts the ellipse to arcs and lines when the *OK* or *Finish* buttons are clicked.

Curves from curves

Curve projected to UCS

This method assumes that you are projecting a curve from one UCS to the one you are currently in.

1. Name the curve in Curve name field.

2. Select the curve in the drop down list box or click  and select the source curve with the mouse.

3. Click Finish (or OK if you're not using the wizard) to create the curve.

A curve doesn't always exist where you need it or you may need to use it's "shadow" as it would lay on a surface or a in a different coordinate system. This function creates a new curve in a different UCS so you can use that curve for other purposes.

Curve start/reverse

While it's not clear in the graphical display, curves do have a direction and a start point. You might want to reverse a curve

Changing the direction of a curve has a number of possible uses:

1. to change the direction of a sweep in a swept surface

2. to change the direction of a curve in a surface

3. to change the direction of a curve as you are joining curves together

Changing the start point of a curve is helpful for operations such as Ruled surface. The start points of the two curves are used to form a correspondence between the curves when the surface is created.

How to reverse a curve

1. Select Curve start/reverse from the curve wizard or Construct menu.

2. Click Reverse.

3. If you want to replace the old curve with the new, click Modify existing curve.

4. If you want to create a new curve, click Create new curve and Name the curve in Curve name field.

5. Select the curve name in the drop down list or graphically pick the curve.

6. The direction of the curve is displayed.

7. Click Preview. The direction of the new curve is displayed.

8. Click Finish (or OK if you're not using the wizard) to create the curve.

How to change the start point of a curve

1. Select Curve start/reverse from the curve wizard or Construct menu.
2. Click Set start point.
3. If you want to replace the old curve with the new, click Modify existing curve.
4. If you want to create a new curve, click *Create new curve* and name the curve in Curve name field.
5. Select the curve name in the drop down list or graphically pick the curve.
6. The direction of the curve is displayed. The start point is the head of the direction arrow.
7. Click the pick point button next to the Start point and then click the new point on the curve.
8. Click Preview. The direction arrow is shifted so that the head is on the new start point.
9. Click Finish (or OK if you're not using the wizard) to create the curve.

Curve offset

Offsets a curve in the direction you pick by the amount you set. Note that offset is a mathematical function based on the curve. It is not necessarily a linear transformation. If you just want to move a curve, use the Transform function.

1. Name the curve in New Name field.
2. Set the offset distance value in the Offset field.
3. Set the Left or Right radio buttons as necessary to offset the curve in the right direction.
4. Click OK to create the curve.

Offsetting a curve can have somewhat surprising results depending on the nature of the curve. Arc sections of the curve might reverse themselves. Until you are familiar with the effects of offset, you should save your work before performing the offset to be sure you can return to a good set of data for your part model.

Curve join

Curve join connects a collection of curves, arcs or lines into a single curve. These objects must meet end to end and curve join will attempt to sort the curves for you. Join will not trim away any overlaps. If you want to connect arcs and lines into curves with trimming of sloppy overlaps use chaining.

To join curves:

1. Select the *Curves* step from the *Steps* toolbox.
2. Select *Create curves using the curve wizard* and click *Next*.
3. Select *Curve from curves* and select Join.
4. Name the curve in *Curve name* field.
5. Determine whether to set the Close curve checkbox. Set the Close curve checkbox if you

want a straight line drawn between the open end of the first curve and the open end of the last curve. You probably don't want the curve to cross itself so be sure the Close line doesn't cross the curve.

6. Select the curve to join in the drop down list box or click and select the source surface with the mouse.

As you select curves, watch the display window to verify that you are building the curve correctly.

7. Repeat step 3 until you have all the curves you want to join listed in the list box.

8. If you picked source curves out of order, use the up, down and delete buttons to rearrange the list. If a curve needs to be reversed, select the curve in the list and press the right arrow button to reverse it.

9. Click Finish (or OK if you're not using the wizard) to create the curve.

Extract font curve

You may need to move text characters from a font or some need to edit the curve. Normally, the text is all one object so this editing isn't possible without extracting the font curve.

1. To extract a font curve, you have to have some engraving text in the part model.

2. In the Curve wizard, set the radio buttons for *Curve from curve* and *Extract font curve*. Or select *Extract font curve* in the Curve from curve fly-out.

3. Name the curve in the *Curve name* field.

4. Select the source font text curve in the drop down list box or click  and select the text curve with the mouse.

The font you are using dictates some things about the behavior of the *Font segment* option. If the character encloses open space such as the letter "P" does, you can select either the inner or outer curve segment. If you are using the Machine Tool Sans Serif font, many of those characters are drawn in separate segments so you can only select individual segments of that font with this function.

1. If you only want one character or part of a character depending on the font, Click *Font segment* and pick the curve. You can only select one character or part of a character this way. Select all extracts every character in the text curve as a separate curve.

2. Click *Finish* (or *OK* if you're not using the wizard) to extract the font curve.

Reducing curve data (Smooth/Reduce Curve dialog box)

Curves that are created by approximations are often represented by a linear curve with hundreds or thousands of points. This data is accurate, but often is very inefficient to work with. The Smooth/Reduce Curve command (available from the curve wizard or the Construct menu) provides two methods for reducing these linear curves.

Smooth spline approximation: This method will approximate the curve with a smooth cubic spline curve. This method works best for three-dimensional curves that are made up of many small linear segments.

Arc/line approximation: This method approximates the curve with arcs and lines. It works best for planar curves that originally came from arcs and lines or piecewise linear curves from import, trim loop extraction or surface/surface intersection that are approximating arcs and lines. Arc/line does the best when the arc/line tolerance is BIGGER than the original sampling tolerance.

How to reduce curve data

1. Bring up the Smooth/Reduce Curve dialog box from either the curve wizard or construct menu.
2. Find the name of the curve in the drop down list or select the curve graphically.
3. If you want to replace the old curve with a new one, click Modify existing curve.
4. If you want to create a new curve, click Create new curve and enter a name of your new curve.
5. Select the curve reduction method by either

- Clicking Smooth spline approximation, OR
- Clicking Arc/line approximation and checking Chain arcs/lines

6. Enter a tolerance. This number indicates how close the new curve will approximate the original curve.
7. Click the Preview button.
8. The data reduction % will be displayed which shows you how much less data the new curves occupies versus the original curve.
9. If the tolerance you chose caused the new curve to actually be larger, you will get a warning dialog box. If you can increase your tolerance, you will probably get less data.
10. If you would like to see this curve extruded as a sample use of the new curve, click the Show preview surface button and hit preview again. A surface will be displayed. This surface can be shaded to check the quality of the curve.
11. Repeat steps 6 - 10 until satisfied with the resulting curve.
12. Click OK.

Curves from surfaces

You must license FeatureMILL or the FeatureRECOGNITION to use the functions described in this section.

Curve from surface boundary

Extracts the curve from a surface's boundary.

1. Name the curve in Curve name field.
2. Select the surface in the drop down list box or click  and select the source surface with the mouse.
3. Set the radio button for the desired boundary. The selected boundary is highlighted in the

display. You can also click and use the mouse to pick the boundary explicitly.

4. Click Finish (or OK if you're not using the wizard) to create the curve.

Surfaces are defined in a rectangular array of rows and columns, even though the surface itself may not look like that. Because of this definition, you can extract individual curves in the surface, especially those on the boundaries.

Boundary curves are particularly useful as a step in building and modifying surfaces. For example, you'll probably extract boundary curves to build a cap surface.

Curve projected onto a surface

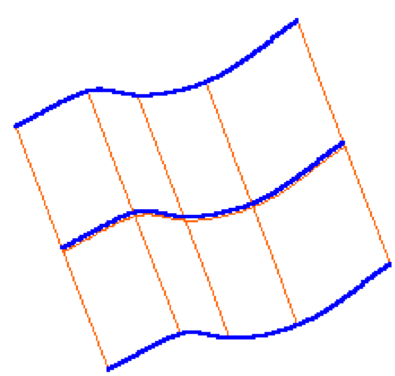
1. Name the curve in Curve name field.
2. Select the surface in the drop down list box or click and select the surface with the mouse.
3. Select the curve in the drop down list box or click and select the curve with the mouse.
4. Set the radio button for the correct direction relative to the current UCS.
5. Click Finish (or OK if you're not using the wizard) to create the curve.

Frequently, you'll want the outline of one surface as traced against another surface. This function generates that curve for you. You'll most often use this function as an intermediate step in constructing or modifying yet another surface.

Curve from surface isoline

1. Name the curve in the Curve name field.
2. Select the surface in the drop down list box or click and select the source surface with the mouse.
3. Set the Row or Column radio button. The selected part of the surface is highlighted in the display for confirmation.
4. Click Finish (or OK if you're not using the wizard) to create the curve.

Surfaces are defined in a rectangular array of points that are arranged in rows and columns. These points are not normally displayed. At any point on the surface curves exist that travel from one surface boundary to another boundary. These curves are called isolines. In the figure on the right the 3 dark curves are row isolines and the 5 lighter curves are the column isolines.



Curve from trimmed surface edge

1. Name the curve in Curve name field.

2. Select the surface in the drop down list box or click  and select the source surface with the mouse.

3. Click  and select the edge with the mouse.

4. Set the Join adjacent loops checkbox as necessary.

5. Join adjacent loops connects the curve to other trim loops that may be present in the source surface. You may need to use Undo so you can set/unset this checkbox to get the desired curve.

6. Click Finish (or OK if you're not using the wizard) to create the curve.

Surfaces are defined in a rectangular array of rows and columns, even though the surface itself may not look like that. Because of this definition, you can extract individual curves in the surface, In this case those on the first or last column or row.

Edge curves are particularly useful as a step in building and modifying surfaces. For example, you'll probably extract edge curves to build a cap surface.

Curve from surface intersection

Overview of intersection curves

The intersection of two surfaces the meet is generally a 3D curve. As long as the surfaces are not tangent (like a surface and its fillet) or do not have a complete region of overlap, you should be able to calculate their intersection curve using FeatureCAM. Intersection curves are often used for surface trimming.

How to create an intersection curve

1. Name the curve in Curve name field if you want a custom name.

2. Select the first surface in the drop down list box or click  and select the source surface with the mouse.

3. Select the second surface in the drop down list box or click  and select the source surface with the mouse.

4. Click Finish (or OK if you're not using the wizard) to create the curve.

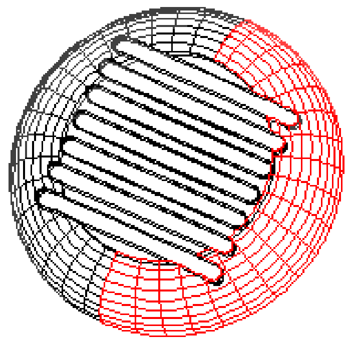
Curve from surface edges

Surface edges is useful for extracting trimming loops from 3D surface data, and projecting that curve onto the XY plane of the current UCS. These curves can be conveniently used in creating 2 ½ D features.

Overview of extracting curves from 3D data

For a customized interface for feature recognition, see page 411.

IGES files often contain complete solid models of a part. Features such as pockets and holes are subtracted from part surfaces resulting in a collection of trimmed surfaces. In the model shown below the seven pockets have been subtracted from the surrounding surfaces.



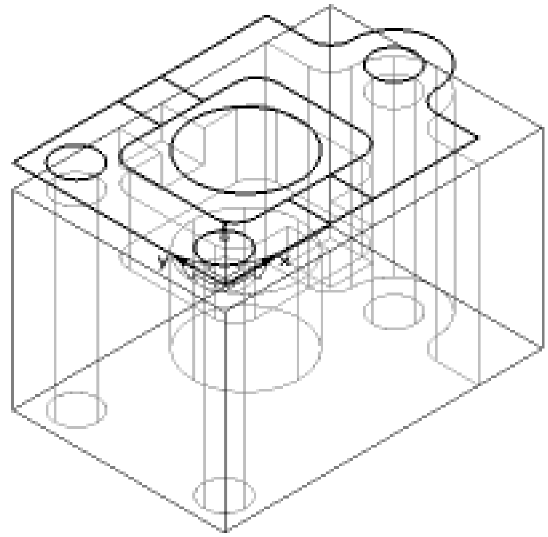
Rather than manufacturing all of these surfaces using 3D surface manufacturing techniques, you should use 2 ½ D pocket features for each pocket. Surface Edges allows you to easily extract trimming curves from a collection of surfaces, join them into a curve and then project the curve onto the XY plane of your current UCS. This curve can then be used to create features.

How to extract curves from 3D data

1. Name the curve in Curve name field if you want a custom name.
2. Click the pick surface button and select each edge that you would like to extract with the mouse. (Since the screen is often crowded with surfaces, the Select dialog box will often be displayed for you to help select the proper surface.)
3. The location of the pick is used to select a surface AND extract a trimming edge. Click the Use Next Edge of Surface if you picked the correct surface, but the wrong edge.
4. Continue selecting edges in order until you have selected all edges.
5. Use the arrow keys to reorder the edges if necessary.
6. If the edges are in the correct order, but the endpoints are matched up incorrectly, select the surface name in the table and click the reverse arrow.
7. Click *Connect Start and End Points* if you want the curve to be automatically closed.
8. Click *Project to UCS* to automatically project the curve to the 2D XY plane of the UCS.
9. If a hollow blue square is displayed at the intersection of two edges, then the edges have not quite matched up. You can correct this by either selecting the missing edge that will connect the endpoints, or if it is just an issue of floating point error, increase the tolerance parameter.
10. Click *Finish* (or OK if you're not using the wizard) to create the curve.

Curves from vertical surface projection

This method is a wizard that creates geometry by projecting straight-walled surfaces as in the figure on the right.



The method performs the following steps:

1. Identify straight vertical surfaces from the selected set or from all surfaces (depending on the option selected.)

2. Project these surfaces on to the current UCS.

3. Convert the curves into lines and arcs.

You are then ask to choose from:

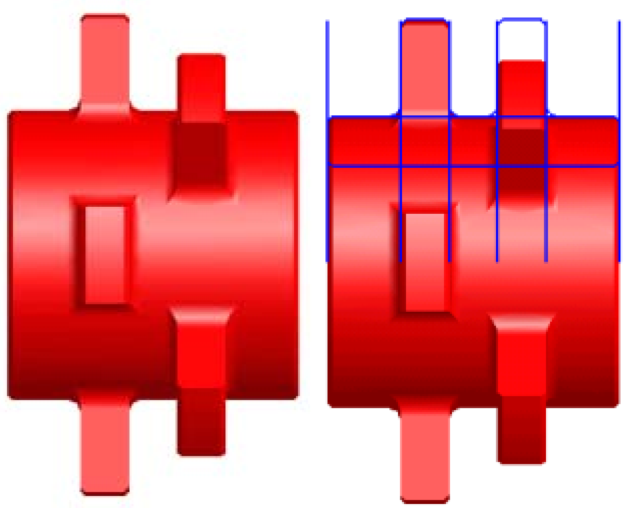
Remove after chaining - select this option if you only want chain this geometry once.

Keep all geometry – select this option if you want to make permanent arcs and lines from this process.

You are then put into chaining mode to create curves.

Curves from revolved surface boundary

Turned parts that are imported as solids are typically modeled using a series of surfaces of revolution. In order to accelerate the creation of turned features from these solid models, FeatureCAM provides a method of intelligently extracting geometry from these revolved solids that can be invoked from the process of importing a solid model into a turn or turn/mill document or by using the Revolved surface boundary curve creation tool. The figures below show the initial solid model and the geometry that is created from it.

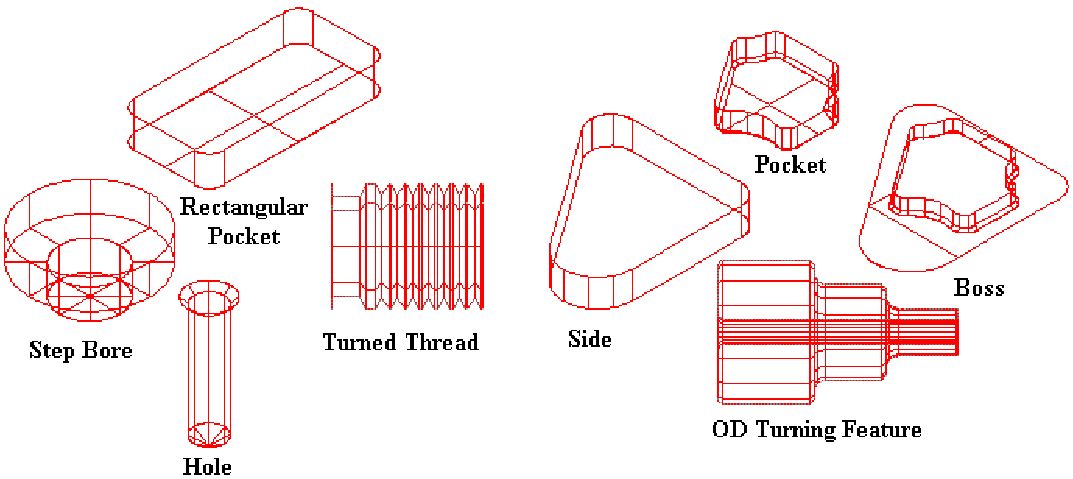


Geometry is created for every surface of revolution. If the *Include vertical plane projection*, option is selected, vertical lines are used to connect the geometry from each surface. In general, this option should be used. Once this geometry is created, it can be used to create turn, bore and groove features and quickly create a manufacturing strategy for your part.

Select *All surfaces* if you want to generate geometry from all of the surfaces and faces of your part. Select *Only selected* if you want to restrict the geometry creation to surfaces/faces that are currently selected.

Chapter 6
Working with Features

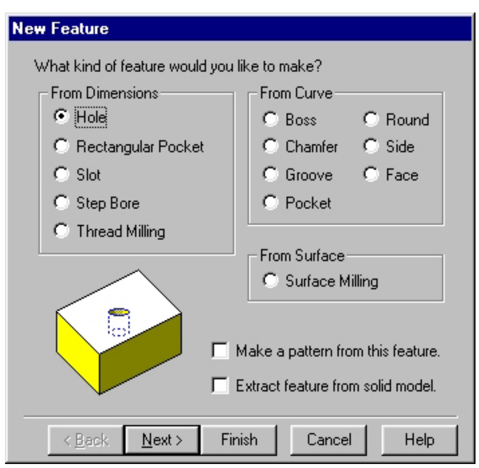
Features are the building blocks that you use to create parts in FeatureCAM. They are simple shop terms like hole, slot and thread. The figure below shows some of the FeatureCAM features.



New feature wizard

The *New Feature* wizard walks you through the creation of features. Features are objects like holes, pockets or threads. These are the objects from which you create and customize toolpaths.

To run the New Feature wizard, click the *New Feature*  step in the Steps toolbar.



The features listed under *From Dimensions* are created from numeric dimensions. The features listed under *From Curve* are created from curves and possibly some additional dimensions.

To complete this page:

1. Select the feature type.
2. If you want to create a pattern of these features, check *Make a pattern from this feature.*
3. If you are creating features from a solid model using the FeatureRECOGNITION option, click *Extract feature from solid model*. See page 411 for more information.
4. Click *Next* to run the rest of the wizard.

The wizard steps you through selecting curves, specifying dimensions, planning manufacturing strategies, verifying automatically selected tooling and automatically calculated feeds and speeds, and overriding the automated choices. Follow the instructions on each page and hit the *Help* button if you need further instructions.

Accelerated feature creation for experienced users

The New Feature Wizard presents a friendly step-by-step interface to creating features. Experienced users may want to directly access the Feature Properties dialog box that contains a concise presentation of the pages of the wizard as tabs of a single dialog box. To display this dialog box during feature creation:

1. Click on the triangle in the *Finish* button. A drop-down menu is displayed.
2. Select *Finish* 

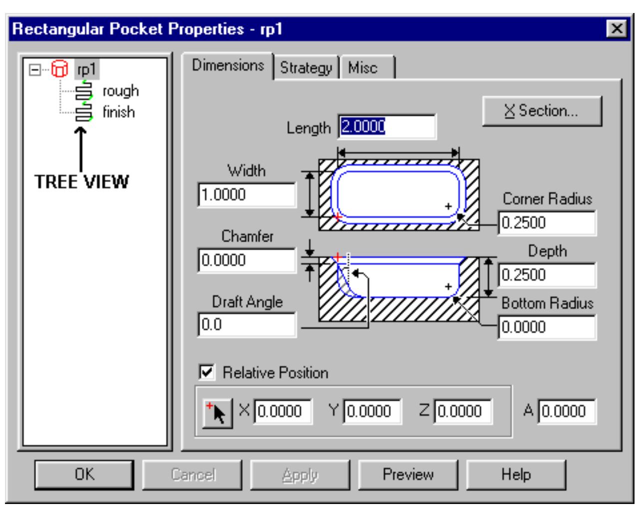
and *Edit Properties*.
3. The *Properties* dialog for the feature you are creating is displayed.

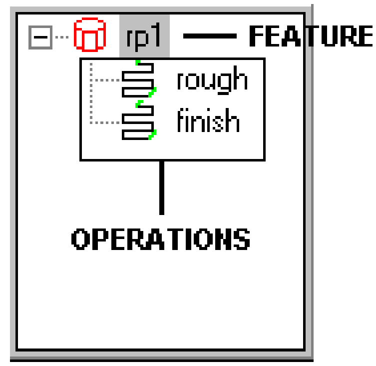
Note: The finish button will toggle to the *Finish and Edit Properties* 

button for subsequent uses of the New Feature wizard. This button can be toggled at any time to eliminate the display of the Feature Properties dialog box.

Feature properties dialog box

The Properties dialog box has a tree view on the left and tabs on the right. To change the feature, select the appropriate tab and set the feature attribute.





1. The dimensions tab contains critical dimensions for defining the feature's shape. This tab is different for each feature type.

2. The misc. tab contains various parameters for the feature.

3. The strategy tab controls the types of operations that are created from the feature.

By clicking on an operation in the tree view new tabs are revealed. The types of tabs depend on the type of operation.

- The tools tab controls tool selection
- The F/S tab controls feed and speed values.
- The drilling tab contains attributes for drilling
- The milling tab contains attributes for milling.
- The turning tab controls attributes for turning tabs.

Interrogating numeric values

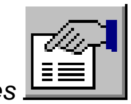
Any numeric value in a dialog box that has a blue label can be extracted from objects in the graphics window. Just click on the label and the dialog box will warp out of the way. Then just click in the graphics window to extract the numeric values. Snap modes control the points you snap to. Some values require one point and others require two.

Type of parameter	Number of points	How points are used
Length or Width	2	Calculates the difference between either the X or Y coordinates of the points, depending on the type of dimension. If the dimension is shown vertically in the dialog box, the Y coordinates are used. Horizontal dimensions use the X coordinates.
Angle	2	A line is created between the two points and the angle between that line and the Z axis is calculated.
Depth	2	The difference between the Z coordinates of the points is calculated.
X Y or Z	1	The X, Y or Z coordinate is extracted from the point.
Radius or Diameter	1	The radius or diameter of a circle can be extracted.

If you want to calculate the dimension differently, hold down the SHIFT key and click on a dimension, the Pick dimension dialog box is displayed which provides you with more options. When you click *OK* in the *Pick dimension* dialog box, the value is automatically inserted into your dialog box.

Modifying features

Edit features with the same dialog boxes used for creating the feature.

1. Double click the feature or select the feature and click the *Properties*  button to open the Feature Properties dialog box
2. Change pages as needed by clicking the different tabs.
3. Modify the parameters.
4. Click *OK* to save your changes and close the dialog box.

Renaming features

FeatureCAM automatically generates names for each entity. To change the name of an object:

1. Select the feature with the mouse.
2. Select *Rename* from the Edit menu, or right-click the object and select Rename.
3. Enter the new name.
4. Click *OK*.

Deleting features

To delete an entity, do one of the following:

- Select the entity and press the DELETE key.
- Select the entity and select *Delete* from the Edit menu
- Right click the entity and select *Delete* from the pop-up menu

Moving features to a different setup

To move a feature to a different setup:

1. Display the Part View by clicking on the *Part View* button in the Toolbox.
2. Click on the feature and drag it under the new setup.

See the *Getting Started with FeatureCAM* guide for more information on the user interface.

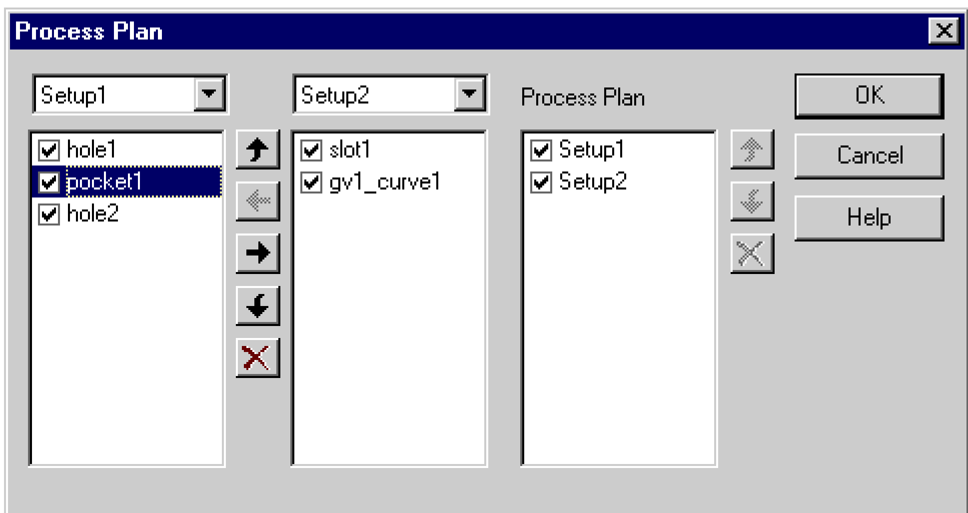
Including or excluding a feature for manufacturing

To toggle the inclusion of a feature in the toolpaths, toggle the checkbox next to the feature in the Part View.

Process plan dialog box

The Process plan dialog box provides an alternative to the Part View for moving features between setups, pulling features or setups out of the manufacturing plan, or adding features to the plan. The order of operations shown in the Process Plan dialog box, serves only as an initial ordering for manufacturing. Settings in the Default Operation attributes and the Ordering dialog boxes can change this order.

The left and middle boxes show the features contained in the setups whose name is listed above the box. If you only have one setup in your part, both of these boxes display the operations of your single stage. The Process Plan list contains the names of all setups in your process plan.



Down

Down moves a feature up in sequence by one place, thus altering the order in the current setup. The right-most Down button moves the selected setup down in the sequence.

Down setup

Down setup moves the selected setup down by one place in the Process Plan.

Delete

Delete removes the currently highlighted setup in the Process Plan list.

Left arrow

Left-arrow moves the selected manufacturing operation(s) from the currently assigned setup to another one. Refer to the Right-arrow description for an example.

Process plan feature

Feature shows the features for each setup.

Process plan list

Process Plan shows all of the setups that are defined for the current part model.

Right arrow

Right-arrow moves the selected manufacturing operation(s) from the currently assigned setup to another one. For example your part has a top and *roughbot* setup. With the *roughbot* setup selected, you see the operations currently assigned to the *first* stage of the manufacturing process. If you select the *top* setup in the pull-down menu above the right operations list box, you likewise see the operations listed for the *second* stage of manufacturing operations in that box. If you click on one of the operations in the *roughbot* setup (to select it) and then click the right-arrow button, that operation moves from the *roughbot* setup operations list to the *top* setup operations list.

Process plan setup

Setup lists of all the setups for your part. Select the name from the list and the features contained in the setup are displayed in the Feature List.

Up

Up moves a feature up in sequence by one place, thus altering the order in the current setup. The right-most Up button moves the selected setup up one place in the sequence.

Up setup

Up setup moves the selected setup up by one place in the Process Plan.

Including or excluding a feature for manufacturing

If there is a check mark next to the *Include In Plan*, the feature is part of the plan. If there is no check mark, it is not.

NOTE: If a feature is not included in the process plan, the toolpaths and the NC code that is generated by the post processor exclude that feature entirely.

Move features to a different setup

To move a feature to a different setup

1. Select Process Plan in the Edit menu.
2. Select the feature in the list.
3. Choose the new setup in the *Setup* drop down box in the *Process Plan* dialog box.

Paste special

The *paste special* command of the *Edit menu* provides a number of options for replicating objects or transferring objects or their settings between FeatureCAM part files.

The process begins by selecting objects and then using the *Copy* command of the *Edit menu* to place them onto the clipboard. The Paste special command then provides the following options:

Paste the clipboard contents into the current setup – This inserts the objects in the clipboard into the current FeatureCAM part document. This is the same functionality provided by the *Paste* command. If you paste into the same document that you copied from note that the second copy is placed on top of the first.

Paste the clipboard… Select a new location – This inserts the objects in the clipboard into the current FeatureCAM part document and subsequent dialog boxes assist in positioning the objects. Click the *Next* button to display additional dialog boxes.

Copy attributes from feature to another feature – This functionality allows you to apply the customization of the feature on the clipboard onto other features. This option is only available if the clipboard contains a single feature.   After selecting this option, click *Next* to specify what kind of attributes to transfer and to select the feature.

Paste special - attributes

Feed/speed overrides – This option copies the values for feed and speed that have been explicitly set on the feature onto another feature.   This option is available only if the feature on the clipboard has feed or speed values that have been changed from the defaults.

Tool overrides – This copies the tooling choices from the feature on the clipboard onto another feature.

Manufacturing attributes

Copy only the attributes that were set on the feature – All attributes that were set on the feature will be transferred to another feature. The settings that are transferred are all attributes of the feature properties dialog box except those on the dimension tab.

Copy all default attributes in effect for the feature – All feature attributes along with the default settings for every attribute on the clipboard feature are transferred to the second feature.

The feature to transfer the attributes to is selected in the drop down list labeled *And set them on the following feature.*   This feature can be selected by name or by clicking on the *Pick feature*  button and selecting the feature in the graphics window. Note that attributes, feeds and speeds and tooling can only be transferred to features of the same type.

Past special – reference

This dialog box allows you to identify the point that will serve as the reference point for pasting of the new feature. This is the point that will be mapped to the new location. To specify this point, type in the X, Y and Z coordinates or use the *Pick point*  button and click the *Next* button to bring up the Past special location dialog box.

Paste special - location

This location is the new location to map the reference point to. To specify this point, type in the X, Y and Z coordinates or use the *Pick point*  button and click the *Finish* button.

Part library

The library allows users to create commonly used configurations of objects and save them for easy use in FeatureCAM parts. Examples of library objects might include a collection of lines and arcs, single features whose attributes have been customized for a specific application or a collection of features that you might use multiple times.

Tree view shows the contents of a specific part library. These libraries contain objects, which you can paste into FeatureCAM parts, and folders which are used to organize your objects.

Add selected – The objects must be selected before opening this dialog box. If multiple objects are selected, they are added to the library as a stream. This is indicated in the tree view with the  button. A stream is a temporary group. When these objects are pasted into a document, the stream grouping is removed and just the objects are added directly to the document. If you want a permanent grouping, create a group of objects and then add the group to the part library.

Add folder – Adds a folder for organizing objects within the library.

Delete – Removes the selected object from the part library.

Rename – Renames the object that is selected.

Paste – Instigates the paste special command on the selected objects from the part library. The name of the pasted object is based on the name of the object in the library. If you paste multiple copies of the same type of object, the subsequent objects are named with a _1, _2 suffix. The only exception is a stream object. For stream objects, the objects are pasted directly into the document using the names of the objects contained in the stream.

Import – Imports objects of a part library into the current library.

Export – Exports all objects in the current library to an external library file. These files have a .nam extension.

Open library – Opens a part library.

New library – Creates a new part library.

Tips for library part objects

1. If you plan on modifying the dimensions of a feature, do not override tools on the library object. Stay with the automatically selected tools. This allows the automatic tool selection to modify the tooling when the dimensions are changed.

2. If you want to create a single library part that contains more than one object, use a group. If you want to ensure that the operations of that group are not rearranged, check the *Ordered* checkbox of the group.

3. Any object type can be added to the part library.

4. If you want to use a unique name for your object, rename the object either before you put it in the library, or rename the object in the part library.

5. Center the object around the origin. This makes it easier to place the feature when you paste it into a document.

Creating library part objects

1. Create the objects in a FeatureCAM document.

2. Select all of the objects you want to include in the part library.

3. Select *Part library* from the *Construct* menu.

4. Click the *Add selected* button to create the library part.

Part library example

1. FeatureCAM has a Counterdrill hole. It is manufactured by spotdrilling the top, drilling the smaller diameter hole and then drilling the top diameter hole. The order of these is fixed, you cannot rearrange them in the Op. List. An alternative manufacturing strategy would be to:

2. Spotdrill the top

3. Drill the larger hole on the top

4. Drill the smaller diameter

5. Let's use the part library to create a feature that uses the alternative manufacturing strategy.

1. Create a plain hole

   a. Set the *Diameter* to 0.375.

   b. Set the *Depth* to 0.25.

   c. On the Strategy tab, click the *Spot drill* and *Drill* checkboxes.

   d. On the Location tab, set the X, Y and Z coordinates to 0.0.

2. Create a second plain hole

   a. On the Dimensions tab, set the *Diameter* to 0.25 and the *Depth* to 0.75.

   b. On the Location tab, set the Z location to –0.25

   c. On the Strategy tab uncheck the *Spotdril*l checkbox and check the *Drill* checkbox.

3. Create a group of the two holes

   a. Arrange the holes so that the top hole is the first in the group.

   b. Check the *Ordered* checkbox.

4. Select the group and create a library part.

Creating a single part library object

You can insert a single object from a part library by either using the Paste special command from the *Edit* menu or by:

Click the Features {bmct btn-steps-feature.bmp} step.

Select the User radio button and click *Next*.

Click on the name of the part library object you want to add and click *Next*.

Enter the location and click *Finish*.

Creating a pattern of part library or UDF objects

These instructions assume that you already have an object in a part libraryor have defined a UDFthat you wish to add to a part.

1. Click the Features  step.
2. Select the User radio button and check the *Make a pattern from this feature* checkbox.
3. Click *Next*.
4. Click on the name of the part library object or UDF you want to add and click *Next*.
5. Follow through the rest of the wizard to create your pattern.

User defined features (UDF)

A UDF is a feature just like pocket or a slot, but it is created using FeatureCAM's application programming interface (API).   These features are often appeal only to small group of FeatureCAM customers so they are optionally loaded into FeatureCAM. This allows FeatureCAM's standard interface to remain as streamlined as possible.

Examples of user defined features that have been created include helical bores with cutter compensation, thin walled pockets and bosses and pulley grooves.

Loading user defined features and addins into FeatureCAM

Some UDFs are shipped with FeatureCAM, but new ones are constantly being added to the FeatureCAM website .

To save the file from the website:

1. Go to the website www.featurecam.com/udf for the latest set of available UDFs.
2. Save these files to your machine by right-clicking on the file name and specifying a directory on your machine. It is convenient to store these addins in Program Files\FeatureCAM\Addins directory, but you can store them anywhere you want.
3. Select the *Addin* option of the *Options* menu and check the checkbox next to the name of the addin you just downloaded.

Creating a user defined feature (UDF)

1. To create a user defined feature
2. Click the Features Step.
3. Select the User radio button and click *Next*.
4. Click on the name of the user defined feature and click *Next*. If the feature you want is not listed, you must load it first.
5. The actual parameters for each UDF will be different, but to set a parameter, click on the parameter name, enter the new value and click Set.

6. Continue through the wizard and click *Finish* when you are done.

Writing your own user defined feature (UDF)

Creating UDFs require the use of the FeatureCAM API. See www.featurecam.com/api for details.

Loading the Example Add-ins that Were Installed with FeatureCAM

The first step to understanding FeatureCAM's programming ability is to successfully use the programming examples that ship with FeatureCAM. Make sure that you install the latest version of FeatureCAM. You'll get a folder called "addins" in your C:\Program Files\FeatureCAM folder.

Choose the Add-Ins menu item in the Options menu. This brings up the Add-ins dialog that lists the add-ins available to you. The add-ins you see below are all installed with FeatureCAM. Click the MakeStockGeometry.bas file in order to load it into FeatureCAM.

Press OK on this dialog in order to dismiss it. Then look around your screen for a new floating toolbar that looks like this:

This button was created when the add-in was loaded, i.e. just after you pressed OK on the Add-in dialog. If you press this toolbar button, then a macro from the add-in will be executed that creates geometry that matches the size of the stock. You are done! You've loaded and used an add-in.

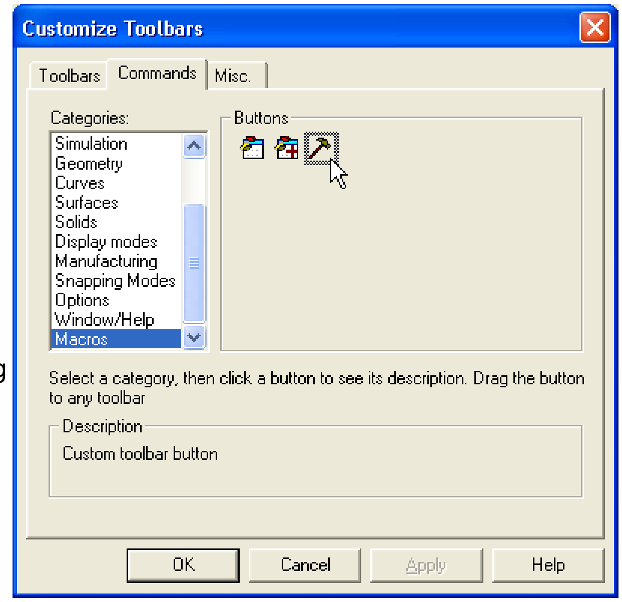
The following example add-ins create toolbar buttons for you. Try them all.

- CenterIndexedStock.bas
- MakeStockGeometry.bas
- MoveSetupToUL.bas

Neither the SelectHeight.bas nor HoleRecog.bas add-ins create toolbar buttons for you. So don't expect to find any if you load these add-ins.

Creating Toolbar Buttons for Macros

Neither the SelectHeight.bas nor HoleRecog.bas create buttons for you. It is the responsibility of the add-in author to create these buttons, but it is not mandatory. For the SelectHeight add-in, the add-in author declined to create a toolbar button for whatever reason. So in order to use SelectHeight.bas, you need to create a button yourself. Here's how:



1. First, make sure that SelectHeight.bas is loaded by using the Add-ins dialog.

2. Press OK on the Add-ins dialog to close it.

3. Make a new toolbar using the Toolbars tab of the Customize Toolbars dialog box.

4. Make a new toolbar button using the Commands tab of the Customize Toolbars dialog box.

Assigning a Macro to a Toolbar Button

The first time you click your new button, it will ask you for the name of the macro to run. Type "SelectHeight" and press OK.

You may return to this dialog box at any time to reassign any "custom toolbar button" (the hammer) to a different macro by right clicking on the button after it has been placed into a toolbar.

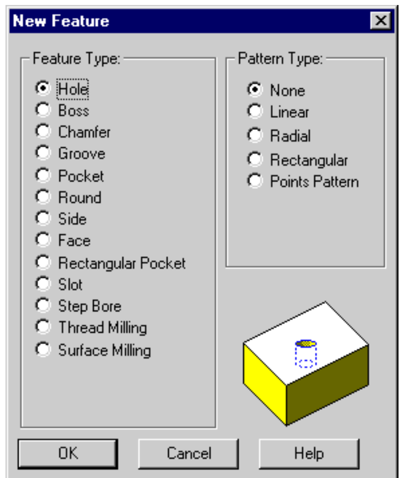
Now, whenever you press the toolbar button, the SelectHeight macro will be executed. If a message comes up saying that the macro doesn't exist, then either you spelled it incorrectly, or you failed to load the SelectHeight.bas add-in.

Even if you loaded the add-in and spelled the macro name correctly, pressing the button will probably cause some type of error that comes from the SelectHeight macro itself. This is because you don't know how to use the SelectHeight macro yet. What does the SelectHeight macro do? You can open the macro in the Sax Basic IDE to find out. Click here to learn how to do this

Chapter 7
2 1/2 D Milling Features

Milling features

You must license FeatureMILL for the functions described in this chapter.



Holes are created by drilling or boring and may have other characteristics such as a chamfer or tapped threads. They are manufactured using canned drilling cycles.

Boss mills a boss whose shape is determined by a curve.

Chamfer mills a chamfer that follows a curve. Most features include a chamfer option which you should use for chamfering entire features

Face is a milling operation to cut a smooth finish on a face of the stock and to cut the stock to exact dimension.

Groove is a groove that follows a curve. The curve is the centerline of the desired groove. Also used for engraving. Grooves support open curves.

Pocket mills an arbitrarily shaped cavity. It may contain a collection of island curves or bosses within the pocket. The islands can be at different heights.

Round mills a rounding operation that follows a curve.

Side is a general milling operation to cut all the material on one side of a curve. This feature works with open or closed curves.

Rectangular pocket mills a rectangular pocket with rounded corners. No curve is needed for this pocket.

Slot is a straight slot with rounded ends. No curve is needed for a slot.

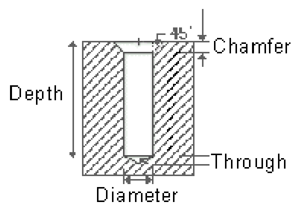
Step bore is a nested series of round pockets with a common center. No curve is needed for a step bore.

Thread mill is a milled thread on a circular pocket or boss.

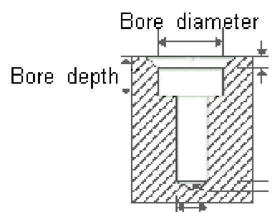
You can create patterns of features as well as groups of features which, in turn, can become the basis feature for even more sophisticated patterns.

Hole features

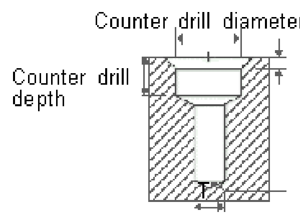
Hole features include 6 different types of holes.



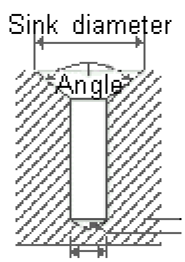
Plain hole



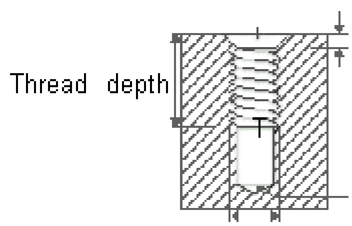
Counter bore hole



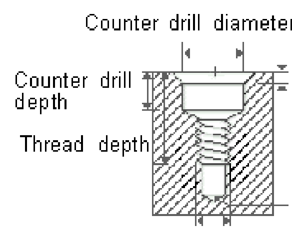
Counter drill hole



Counter sink hole



Tapped hole



Counter drill tapped hole

The different types of holes are defined by dimensional attributes. Not every hole has every attribute. This table shows all the hole types and dimension attributes associate with them.

Hole attributes and definitions	Plain	C-sink	C-bore	C-drill	Tapped	CD Tapped
Angle refers to the included angle of the counter sink. This value helps select an appropriate countersink tool.		•				
Back views the settings of the pattern object that contains the hole.	•	•	•	•	•	•
Bore depth refers to the depth of the counter bore.			•			
Bore diameter refers to the diameter of the counter bore.			•			
Chamfer sets the depth of a 45o chamfer at the top of the hole. Set to 0, no chamfer is cut.	•	•	•	•	•	•
Counter drill diameter sets the diameter of the counter drill.				•		•
Counter drill depth sets the depth of the counter drill.				•		•
Depth refers to the overall depth of the hole. Depth is measured from the top to the bottom of the hole and includes other parameter values such as the hole's *Chamfer* depth. For a blind hole, the screen picture shows a cone representing the drill tip. The cone distance is not part of the depth.	•	•	•	•	•	•
Diameter sets the finished diameter of the hole.	•	•	•	•	•	•
Location specifies location by coordinates. Click  to select the point with the mouse.	•	•	•	•	•	•
Metric toggles TPI (Threads Per Inch) to pitch for metric threads.					•	•
Relative position indicates that the *X, Y,* and *Z* coordinates are relative to the current UCS. Otherwise, coordinates are relative to the Stock Coordinate System.	•	•	•	•	•	•
Sink diameter refers to the diameter of the counter sink of the hole.		•				
Thread depth sets how much of the hole is threaded.					•	•
TPI/Pitch sets the threads-per-inch or pitch for metric of the hole's threads.					•	•
Thread diameter refers to the *nominal diameter* of the tap tool used to thread the hole. Underlying details such as selecting an under-sized drill for creating the initial hole are automatically computed.					•	•
Through is a checkbox that control the display of holes. For manufacturing it is only used for tapped holes.	•	•	•	•	•	•
XYZ Coordinates refer to the *XYZ* coordinate location of the top center of the hole. These values are relative to the Stock Coordinate System.	•	•	•	•	•	•

Creating a hole

1. Click *New Feature* .
2. Select *Hole* and Click *Next*.
3. Select the type of hole from the type drop-down list box.

- *Plain* hole is a simple hole with an optional chamfer
- *Counter sink* hole is a hole with a counter sink at the top
- *Counter bore* hole is a hole with a counter bore at the top
- *Counter drill* hole is a hole with a counter drill at the top
- *Tapped* hole is a hole that is under-sized drilled and then tapped
- *CD tapped* hole is a counter drilled hole with the bottom part of the hole tapped

4. Enter a *Diameter* value. (If you are building holes from circles you selected before entering the wizard, the diameter of the hole is set by the diameter of the selected circle.)
5. Set how deep the hole is in the *Depth* field.
6. Check *Through* if the hole is a through hole. See *Through hole details* for more information.
7. Depending on the type of hole you selected you may have still other dimensions to fill in such as tapped depth and thread pitch. Refer to the table above for details about all possible dimension attributes different types of holes can have. For tapped holes, you can click the *Standard Threads* button and select a thread type. Each thread type will set the *Thread depth*, *TPI* and *Diameter* dimensions.
8. Click *Next* and enter XYZ Location, or click to pick a location with the mouse.
9. Click *Next* to specify manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Through hole details

When set, FeatureCAM models the hole as a straight cylinder shape. Turned off, the hole is modeled as a blind hole, with a cone on the bottom for the drill tip..

Selecting Through does not ensure that the hole passes all the way through the material. You must assign a Depth in excess of the part thickness. For blind holes, the extra depth value required for the point of the tap is automatically included by ensuring that the initial drilled hole provides the required clearance. You can not make a blind tapped hole that does not have enough clearance to accommodate a bottoming tap. If you get an error message, increase Depth to at least the value shown as the bottom hole depth, or decrease the thread depth.

Manufacturing holes

FeatureCAM follows this process for drilling holes:

- Analyze the hole size, type, and attributes to determine what tools to use.
- Pick feeds and speeds based upon the material being drilled.
- Prepare the site with Spot-drill and pilot-drill operations.
- Drill to depth
- Size and counter cutting operations.
- Tap, or bore/ream if set.

There are infinitely many variations on this process, particularly with patterns of holes and hole macros. The basic hole process can by fine tuned primarily in two places. To fine tune for all holes in general, use the Drilling tab of the Machining Attributes dialog, and to tune only a feature, use the Tools, Drilling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a hole is drilled, and the feed/speed database to determines the feeds and speeds used.

Tool Selection

The first step of drilling a hole is to determine what kind of hole it is. Then collect a list of operations to produce that hole. After the analysis, FeatureCAM picks tools. The table below shows the tooling types that can be used for each operation type.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
spotdrill	spotdrill or centerdrill	spotdrill, centerdrill, countersink	Automatic behavior is dependent on the *prefer spotdrill or prefer centerdrill* attributes.
chamfer	countersink	spotdrill, centerdrill, countersink, chamfer	Circular interpolation is performed if the tool's diameter is smaller than the hole's. You can override the automatically selected tool with a tool that does not have a 90 ° included angle, but the chamfer will not be a 90 °chamfer.
twistdrill	twistdrill	twistdrills, endmills	No circular interpolation is performed with endmills, even if the diameter is smaller than the hole's. Use a step bore feature to mill a hole.
bore	boringbar	boringbar	
counterbore	counterbore	counterbore, endmill	Circular interpolation is performed if the tool's diameter is smaller than the counterbore's.
ream	ream	ream	

tap	tap	tap	For blind holes a fast spiral, plug tap is preferred. Through holes require a gun style plug tap

For drilling, the most important criteria are diameter and length. If a tool can't be found that satisfies the criteria, then you receive a tool selection error.

Feeds and Speeds

FeatureCAM chooses feeds and speeds for all of its drilling using the F/S database that you can customize. Feeds and speeds are determined based upon the stock material.

Site preparation

The critical aspects of roughing are as follows.

Spot-drilling aids later drill operations and is set with the Spot drill attribute. If you have set Attempt chamfer w/ spot, then the chamfer operation may be combined with this operation if the tool has the correct angle and size.

Pilot drilling is enabled by setting the Pilot drill attribute either as a Default Machining Attribute, or for the hole feature itself. How the Pilot drilling is performed is influenced by Chip break/Deep hole setting and Number of pecks and such.

Drill to depth

This is shown as a twist-drill operation in the tree view and is turned on or off with the Drill attribute. Other attributes affecting the behavior are Chip break/Deep hole, and Number of pecks. Actual diameter may be undersized depending on later actions such as tapping or reaming operations.

The actual depth of the twistdrill operation is determined as follows:

- Drilled Depth = depth + (diameter / 2.0) / tan(Angle_of_drill / 2.0)
- If the *Through* box is checked then add 0.1*Diameter

Auxiliary operations

Depending on the type of hole, one of either: Counterbore, Counterdrill, or Countersink, operations may be performed.

If a Chamfer was set, but not performed with the spot-drill operation, it happens now.

Threading, boring, and reaming

Last, the tap, or bore, or ream operations occur. If the hole is tapped, then no bore or ream operations are possible. If Bore and/or Ream is set, no tap operations are possible. if set, not available with Bore or Ream options.

Tap cuts threads in the hole and is measured in TPI (threads per inch) in inch units or Pitch (millimeters per thread) in metric units. Be sure to set the Max tap spindle RPM attribute.

For a tapping operation, the depth is determined as follows:

Tool	Diameter	Depth
Plug point tap	Less than 3/8 inch (9.5mm)	(diameter - 1.1047xPitch)/2 + 4.75xPitch
Plug point tap	3/8 – 7/16 inches (9.5 – 11 mm)	4.75xPitch
Plug point tap	>=7/16 inches (11.1mm)	4.75xPitch + .2xDiameter
Bottom point tap	< 7/16 inches (11.1mm)	1.75xPitch
Bottom point tap	>= 7/16 inches (11.1mm)	1.75xPitch + .2xDiameter

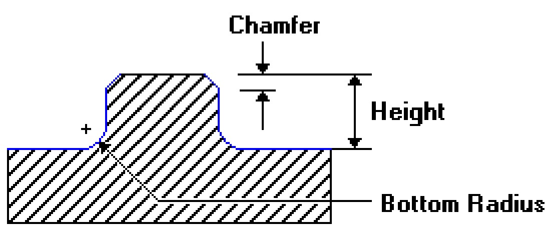
Bore uses a boring bar to position a hole exactly. Bore and Ream settings are not usually used together.

Ream drills a hole feature undersized and then reams it to size. The diameter of the drill will be between 93% and 97% of the final hole diameter. Bore and Ream settings are not usually used together.

Boss feature

The Boss feature removes all material between two curves: the Stock curve and the boss curve. Without a stock curve, the boundary of the stock serves as the stock curve.

Note: Producing multiple bosses on the surface of a part requires some special attention. By selecting multiple curves within a single boss feature, FeatureCAM becomes aware of the multiple bosses and correctly produces those features.



Creating a boss

1. Click *New Feature* .
2. Select *Boss* and click *Next*.
3. The boss curve controls the shape of your boss. Click *Curve* to specify a curve(s) for a boss feature. If you have multiple bosses at one height, select multiple curves. To select multiple curves within a feature, hold down the CTRL key while clicking on the curve name.
4. Click *Next*.

5. The Location page shows you the Z height of the curve. Enter an offset value if you want to change the height of the boss. Click *Next.*

6. Height sets the overall height of the boss feature as cut into the stock.

7. The Bottom Radius sets the bottom radius of the feature. The radius corresponds to the shape of the cutter. A value of 0 cuts a square corner and is the default.

8. Draft angle sets the slope of the feature wall. See page 117 for more information.

9. Chamfer sets the depth of a 45 degree chamfer cut at the top edge of the feature.

10. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

By default, a boss will use the stock boundary as the outer extent of the feature. To bound the extent of the boss cut:

1. Double click the feature. The Properties dialog box comes up.

2. Click the *Stock curve* button. Click the name of the curve you want to use as the outer extend of the boss.

Note: If ***stock boundary* is selected it indicates that the stock boundary will be used as the outer extent. The stock boundary will automatically be used as the outside extend of the boss only when the current UCS is parallel to one of the block faces. If your UCS isn't parallel to a stock face, boss features need to have a stock curve included with them.

3. If you want the wall of your feature to have a special cross section, click the X-Section button to open a dialog box to select the curve that matches your cross-section shape.

4. Click *OK*.

Boss curves

The curve selected defines the regions you want to keep after the boss is cut. To cut multiple bosses at the same height, the bosses all have to be part of the same feature. Simply select multiple curves for the boss feature. None of the boss curves can touch, nor can they contain one another.

Manufacturing hints for a boss

When building multiple bosses at the same height in the part, include all of the boss curves in the same boss feature.

Bosses remove material all the way to the stock boundary, including any other features you may have placed above the boss shoulder height. To limit the area milled away by the boss feature, use a Stock curve, or set the Total Stock attribute.

Manufacturing bosses

FeatureCAM follows this general process:

- Analyze the curve to determine what tool to use.

- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps based on the height of the boss.
- Generate a finishing pass.
- If the boss has a draft angle, cross-section curve or corner radius, see *Manufacturing draft angles or bottom radius regions*.

The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a boss is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib. The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the curve that defines the boss to determine what size tool to use. The largest tool that can cut the boss without gouging is selected (see *Tool % of arc radius* or Tool dia. override).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the boss height.

Endmills are automatically selected for the roughing and finishing passes, but you can override the automatically selected tool to specify a face mill.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and Speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by pre-drilling (see Pre-drill diameter and Pre-drill point).

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection After roughing, the roughing tool is used to finish the pocket. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on The finish pass ramps into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter).

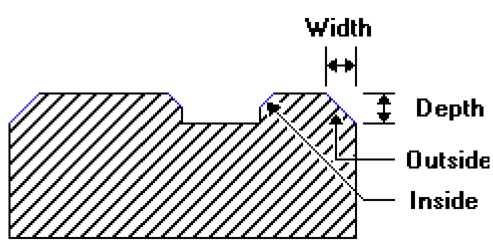
Finish passes and overlap The tool will go around the pocket a number of times set by Finish passes,), and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off Another arc of the same size as the ramp on moves the tool away from the finished wall.

Retract removes the tool from the stock area and sets up for the next operation.

Chamfer feature

The Chamfer creates an edge break along a curve with a chamfering tool. To chamfer the entire upper edge of a curved feature, use the optional *Chamfer* parameter on that feature. Use the Chamfer feature to chamfer only a portion of the edge of a feature.



Creating a chamfer feature

1. Click *New Feature* .
2. Click *Chamfer* and click *Next*.
3. Select a curve from the Curve list box.
4. An arrow is displayed showing the side of the curve that will be cut. If the arrow is pointing the wrong way, click the *Flip side* check box.

5. Click *Next*.
6. The Z height of the curve is displayed. If you want to offset the feature in Z, specify an Offset value.
7. Click *Next*.
8. Set the Width of the chamfer edge break.
9. Width and Depth settings are related to each other as they must match the shape of a tool you have in your tool crib in order to cut the described chamfer. For a 45° tool, Width and Depth must be equal.
10. Enter a value for Depth to set the depth of the chamfer edge break.
11. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Chamfer curves

The curve defines the shape of the chamfer. Select more than one curve if you want to create a chamfer along multiple curves with one feature. The curves can be open (ends do not touch) or closed (beginning and end points are the same.)

Manufacturing chamfers

In general FeatureCAM, uses the following process:

- Choose a tool based upon the width and depth of the chamfer, and on the tightest bend in the curve.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a chamfer is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool Selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The criteria used are corner radius and inner diameter of the rounding tool. The width of the tool must be large enough to cut the depth and width set for the chamfer. The inner radius of the tool is important because the tool must fit into tight corners of your curve. If a tool can't be found that satisfies the criteria, then you get an error and NC code isn't generated.

A chamfermill tool is automatically selected. You can override the selection to be a countersink tool. If you are using a countersink, you may need to adjust your touch-off point to mill an accurate chamfer.

Tool diameter: FeatureCAM picks a tool that has sufficient width to cut the chamfer feature.

Inner diameter: FeatureCAM analyzes the curve that defines the round to determine what size tool to use. The tool needs to fit into the smallest corner of the chamfer.

Feeds and Speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are as follows.

Getting to depth is accomplished by plunging.

There is no vertical step, but the horizontal step size is controlled by the *Distance between cuts* attribute on the Stepovers tab.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the finish pass is turned off and the entire feature is machined by the roughing pass. This can be changed on the Strategy page.

Tool selection After roughing, the roughing tool is used to finish the chamfer. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

There is no vertical step, but the horizontal step size is controlled by the *Distance between cuts* attribute on the Stepovers tab.

The contact point of the tool is controlled by the *Through depth* attribute.

Ramp on not available for chamfers.

Finish passes and overlap The tool goes around the chamfer a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

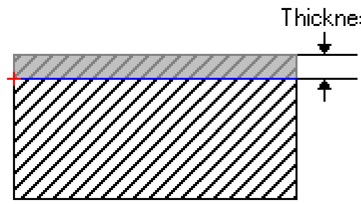
Ramp off not available for chamfers.

Retract removes the tool from the stock area and sets up for the next operation.

Face feature

FeatureCAM contains a fully integrated facing feature.

The Face feature is performed with facing tools and uses the facing feeds and speeds provided in the database. Facing removes all of the stock down to the Z=0 plane. If your stock does not extend above the Z=0 plane, or you don’t set a negative Z value for the feature, the machining simulation does not appear to cut.



Creating a face feature

1. Click *New Feature* .

2. Click *Face* click *Next.*

3. Enter the *Thickness* sets the thickness of the facing operation.

4. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Face curves

The curves for a Face feature define the regions to be faced. Select multiple curves to face multiple regions. All curves must be closed (beginning and end points are the same).

Manufacturing faces

FeatureCAM follows this general process:

- Select a facing tool.

- Pick feeds and speeds based upon the material being machined.

- Generate a facing pass, possibly in multiple z steps depending upon the amount of material to remove.

There are some variations on this process. The process can be controlled in the Stepover tab of the Default Machining Attributes, and on the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also affects the decisions, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and cutter height. If a tool can't be found that satisfy the criteria, then you get an error and NC code will not be generated.

Tool diameter is usually large for face operations as there are no tight spots or complex shapes to create.

Cutter height is usually small for facing tools. This prevents them from being used to cut inappropriately deep features and affects how many passes it might take to face the stock.

A face mill tool is automatically selected, but you can override this selection with an endmill.

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Facing operation

Face features are not roughed nor finished as other features. The critical aspects of a face operation are as follows.

Getting to depth is accomplished by plunging.

With only one pass, there is no **vertical step**.

Horizontal stepover is controlled in both the X and Y directions with Last pass overcut % and Lateral overcut %.

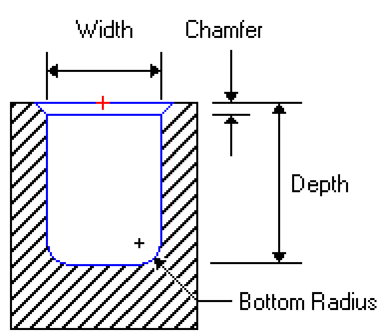
There is no **finish allowance** for face features.

Retract removes the tool from the stock area and sets up for the next operation.

Groove feature

The Groove feature creates a groove of any shape. There are, however, different kinds of grooves that have different limitations For one-pass engraving click *Simple* so only a single manufacturing pass is performed along the curve.

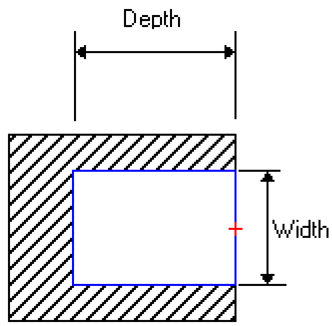
Face grooves



Face groove qualities

1. Can be of any shape and even intersect with itself.
2. Can cut multiple passes to cut grooves of any width when Simple is turned off.
3. If the groove is simple, the curve can be 3D.
4. Can have chamfers and bottom radii.

ID/OD groove



ID/OD groove qualities

1. Can be of any non-intersecting shape that is open enough to allow a cutter to enter, operate, and exit.

2. Can only cut to widths of tools you have available.

3. Can only cut to the tool's shape by default. If you load special tools and create multiple grooves, you can achieve special effects.

Creating a groove feature

1. Click *New Feature* .

2. Click *Groove* and click *Next*.

3. Select a curve from the Curve list box and click *Next*.

4. The Z height of the curve is displayed. If you want to offset the feature in Z, specify an Offset value. Click *Next*.

5. Decide whether your groove is a simple groove and set the check box accordingly. Simple designates that the groove should be cut with a single pass along the curve with a tool with a diameter equal to the *Width* of the groove. For engraving you most likely want to create a Simple groove.

6. Set whether your groove is a Face, Inside, or Outside groove. Face sets the groove to cut on the XY plane of the current setup. Inside and Outside set whether the groove runs inside or outside a closed curve.

7. If you are creating a Face groove:

   - Decide whether your groove is a simple groove and set the check box accordingly. *Simple* designates that the groove should be cut with a single pass along the curve with a tool with a diameter equal to the *Width* of the groove. For engraving you most likely want to create a Simple groove
   - Enter a value for the groove Width.
   - Set Through to change the modeling of the groove if it helps you display your part better.

- Set the Bottom Radiusif desired.
- Chamfer sets the depth of a 45 degree chamfer cut at the top edge of the feature.

8. If you are creating an Inside/Outside groove:
   - If the curve is at the bottom of the groove, check the *Curve at bottom* checkbox.
   - Enter a value for the groove Width.
   - Click *Next*. The Machining Side page is displayed and the groove is previewed in the graphics window.
   - An outside groove uses the curve as the outside of the groove. An inside groove uses the curve as the inside of the groove. Click the  button to toggle between the two types of grooves.
9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Manufacturing hints for a groove

- Simple face grooves are cut with one horizontal pass with a tool whose diameter matches the groove's width.
- Plunge points and ramping parameters are ignored for Simple grooves.

Manufacturing face grooves

Regular grooves are machined much like pockets, and include both a roughing and finishing pass. More often than not, a simple groove is preferred (see the Simple (Engrave) checkbox on the Dimensions page). Simple grooves are described in the next section. For a regular groove, FeatureCAM uses the following process:

- Determine what tool to use based only on the groove width and depth.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple z steps depending upon the depth of the groove.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a groove is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step of machining a groove is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and length. If a tool can't be found that satisfy the criteria, then you get an error and NC code will not be generated.

Tool diameter FeatureCAM uses the width of the groove to determine what diameter tool to use. The tool needs to fit into the groove, but still allow room for a finish allowance on both walls of the groove.

Tool length FeatureCAM picks a tool that has flutes long enough to reach the bottom of the groove.

Operation type	Automatically selected tool	Possible user overrides	Notes
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are as follows.

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by pre-drilling (see Pre-drill diameter and Pre-drill point). Note that zigzag is not an option if the groove has no linear portion. Simple grooves only support plunging.

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover Although not typically necessary, FeatureCAM can perform a horizontal stepover to manufacture a groove. FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection, after roughing, the roughing tool is used to finish the groove. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has the finish pass ramp into the material with an arc equal to a percentage of the tool diameter.

Finish passes and overlap The tool goes around the groove a number of times set by

Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall.

Retract removes the tool from the stock area and sets up for the next operation.

Manufacturing simple grooves

Simple grooves, also called engraving grooves, offer a single-pass approach to milling a groove. The following process is used:

- Determine what tool to use based only upon the groove width.
- Use slotting feeds and speeds based upon the material being machined.
- Generate a single pass, possibly in multiple z steps depending upon the depth of the groove.

Tool selection

Same as for a regular groove, except that the tool diameter must equal the groove width.

Feeds and Speeds

Same as for a regular groove, except that slotting feeds and speeds are used.

Roughing and finishing

Roughing and finishing are performed in a single pass. There is only a single operation shown in the tree view, and it is called “slot”. The critical aspects are as follows.

Getting to depth is accomplished by plunging.

Direction of cut can be controlled on the Strategy page.

Vertical step is no more than 100% the tool radius (see Rough depth, and Rough pass Z increment).

Horizontal stepover is not available for this feature type. The tool diameter must be equal to the groove width.

Finish allowance is not available for this feature type. The tool diameter must be equal to the groove width.

3D simple face grooves are approximated with lines and arcs, if the arcs lie in the XY, YZ or XZ planes.

Manufacturing OD/ID grooves

ID/OD grooves are machined using key-seat cutters, and include both a roughing and finishing pass. For an ID/OD groove, FeatureCAM uses the following process:

- Determine what tool to use based on the groove width and depth.
- Pick feeds and speeds based on the material being machined.

- Generate a roughing. The groove is first cut down the center of the groove, with subsequent passes alternating on either side of the center. Different allowances are possible on the walls and bottom of the groove.
- Generate a finishing pass. The finishing pass is based on the Finish walls and Wall pass attributes.

There are infinitely many variations on this process. The process can be fine-tuned primarily in two places: the Default Machining Attributes tabs, and the property tabs of a feature. The tooling database also has a large impact on how a groove is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step of machining a ID/OD groove is to pick a keyseat tool from the current tool crib (see the Manufacturing menu). The most important criteria are length and diameter. If a tool can't be found that satisfy the criteria, then you get an error and NC code isn't generated.

Tool length FeatureCAM uses the depth of the groove to determine what length tool to use. The tool needs to fit into all the way into the depth of the groove. If multiple steps need to be taken to manufacture the groove, then those steps can be made to reach the full depth.

Tool width FeatureCAM picks a tool that matches the width of the tool within a small tolerance. If the groove is too big, then you receive a tool selection error.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are as follows.

Getting to depth There are two aspects to getting the keyseat cutter to depth. The tool must first move down in Z, and then it must plunge into the metal. For the down move, the keyseat cutter simply plunges down. You must have air underneath the cutter for this move. You may set the predrill point to determine where the keyseat cutter is lowered in Z. Then the tool plunges into the material horizontally. There are no options to control the plunge into the material.

Vertical step No vertical steps are taken. You get a tool selection error if the tool width is not exactly as wide as the groove.

Horizontal stepover Rough pass stepover can override the horizontal stepover into the groove. Otherwise FeatureCAM automatically takes as many steps as it needs to cut to the full depth of the groove. 33% of the tool diameter is the default.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

Tool selection After roughing, the roughing tool is used to finish the groove. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on makes the finish pass ramps into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter).

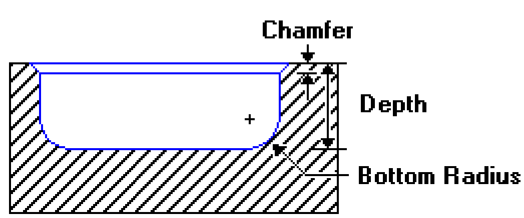
Finish passes and overlap The tool will go around the groove a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off uses another arc of the same size as the ramp.

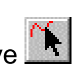
Retract removes the tool from the stock area and sets up for the next operation.

Pocket feature

The Pocket feature creates a pocket of any shape. If you want to create a simple pocket with a rounded-rectangular cross-section, use the Rectangular Pocket feature. A Pocket can have an arbitrary number of islands within the outer boundary. The island curves must be contained within the boundary curve and the island curves may not touch.



Creating a pocket feature

1. Click *New Feature* .
2. Click *Pocket* and click *Next*.
3. Select a curve from the Curve list box or select it graphically with the Pick curve  button. You can use one curve for the boundary, or if you have multiple pockets that have the same depth, you can select multiple curves in the feature properties dialog box all at the same time. Press and hold CTRL while you select the curves with the mouse to select multiple curves. Click *Next* after selecting the curve(s).
4. The Location page shows you the Z height of the curve. Enter an offset value if you want to change the height of the boss. Click *Next.*
5. Enter a value for Depth

6. Enter a Bottom Radius if desired.

7. Enter a Draft angle if the walls are at an angle.

8. Enter a Chamfer if you want a 45-degree edge break on the top of the feature.

9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Note that you can always edit this feature later.

If you need specify islands for the pocket or a different cross-section for the walls:

1. Double-click on the feature. The Properties dialog box comes up.

2. Click Islands to pick areas, defined by curves, that are not milled with the rest of the pocket.

3. Click X-section to enter a cross section curve for the feature.

Multi-height bosses and pocket islands

To make multi-height bosses or islands in a pocket with different heights:

1 Create the curves for the boss or island and translate it to its proper location in Z.

2 Use this curve as a boss curve or island curve for a pocket.

Manufacturing pockets

FeatureCAM follows this general process:

- Analyze the curve to determine what tool to use.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the pocket.
- Generate a finishing pass.
- If the pocket has a draft angle, cross section curve or corner radius, see Manufacturing draft angles or bottom radius regions

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a boss is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib. The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the curve that defines the pocket to determine what size tool to use. FeatureCAM uses the largest tool that can fit in the corner and the tightest passage of the feature. The largest tool that can cut the pocket without gouging is selected (see *Tool % of arc radius*).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the pocket.

Operation type	Automatically selected tool	Possible user overrides	Notes
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by pre-drilling (see Pre-drill diameter and Pre-drill point).

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection, after roughing, the roughing tool is used to finish the pocket. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has finish pass ramp into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter).

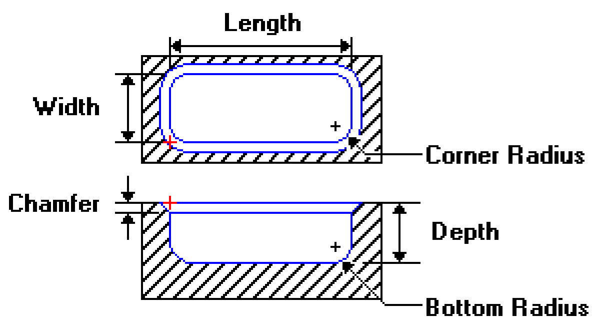
Finish passes and overlap The tool goes around the pocket a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall.

Retract removes the tool from the stock area and sets up for the next operation.

Rectangular pocket feature

For a pocket with a non-rectangular curve, use the more general *Pocket* feature. The Rectangular Pocket feature may be used to produce round pockets. To do so, input pocket diameter in both the length and width fields, input pocket radius (1/2 pocket diameter) in corner radius field.



Creating a rectangular pocket feature

1. Click *New Feature* .
2. Select *Rectangular Pocket* and click *Next*.
3. Enter a value for *Width*, the Y dimension of the pocket.
4. Enter a value for *Length*, the X dimension of the pocket.
5. Set the *Depth*, the distance cut into the material.
6. Rectangular pockets have a Corner radius that defines the four corners of the pocket. Enter a value for the *Corner radius*.
7. *Chamfer* sets the depth of a 45 degree chamfer cut at the top edge of the feature. Leave this value at 0, the default, for no chamfer.
8. Set the Bottom Radius if desired.
9. Set the Draft angle if desired.
10. Click *Next* and set the XYZ position of the lower left corner of the feature.
11. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

If you want the wall of your feature to have a special cross section:

1. Double click on the pocket. The Properties dialog box comes up.
2. Click *X Section* to open a dialog box.
3. Select the curve that matches your cross-section shape.
4. Click *OK*.

Manufacturing rectangular pockets

FeatureCAM follows this general process:

- Analyze the pocket dimensions to determine what tool to use.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the pocket.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a rectangular pocket is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the dimensions that defines the pocket to determine what size tool to use. FeatureCAM prefers large tools for pockets but is influenced by the corner radius. The largest tool that can cut the pocket without gouging is selected (see *Tool % of arc radius*).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the pocket.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by pre-drilling (see Pre-drill diameter and Pre-drill point).

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection, after roughing, the roughing tool is used to finish the pocket. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has the finish pass ramp into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter).

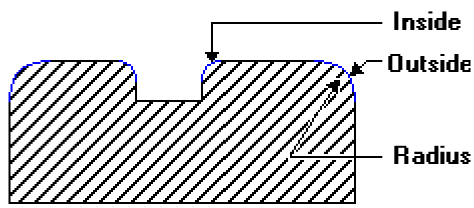
Finish passes and overlap has the tool goes around the pocket a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall.

Retract removes the tool from the stock area and sets up for the next operation.

Round feature

The Round feature creates an edge break along a curve with a rounding tool.



Creating a round feature

1. Click *New Feature* .
2. Click *Round* and click *Next*.
3. Select a curve from the Curve list box.
4. An arrow is displayed showing the side of the curve that will be cut. If the arrow is pointing the wrong way, click the *Flip side* check box.
5. Click *Next*.
6. The Z height of the curve is displayed. If you want to offset the feature in Z, specify an *Offset* value.
7. Click *Next*.
8. Enter a value for the *Radius* of the edge break. This corresponds to the radius of the rounding tool selected to cut the feature.
9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Manufacturing a round

In general FeatureCAM, uses the following process:

- Choose a tool based on the desired radius of the round, and on the tightest bend in the curve.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a round is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step of machining a round is to pick a tool from the current tool crib (see the Manufacturing menu). The criteria used are corner radius and inner diameter of the rounding tool. The corner radius of the tool must be the same as the desired radius of the round. The inner radius of the tool is important because the tool must fit into tight corners of your curve. If a tool can't be found that satisfies the criteria, then you get an error and NC code is not generated.

Corner radius FeatureCAM picks a tool that has exactly the same corner radius as the desired round feature.

Inner diameter FeatureCAM analyzes the curve that defines the round to determine what size tool to use. The tool needs to fit into the smallest corner of the round.

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are as follows.

Getting to depth is accomplished by plunging.

There is no vertical step, but the horizontal step size is controlled by the *Distance between cuts* attribute on the Stepovers tab.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the finish pass is turned off and the entire feature is machined by the roughing pass. This can be changed on the Strategy page.

Tool selection, after roughing, the roughing tool is used to finish the round. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on is not available.

There is no vertical step, but the horizontal step size is controlled by the *Distance between cuts* attribute on the Stepovers tab.

Finish passes and overlap has the tool go around the round a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off is not available.

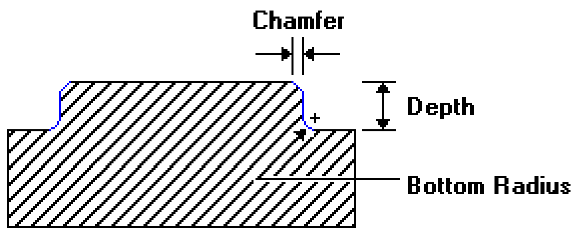
Retract removes the tool from the stock area and sets up for the next operation.

Side feature

The Side feature provides low-level manufacturing control when you need customized manufacturing that is not addressed by either the Boss feature or the Pocket feature.

The Side feature is useful for:

- Outside part boundaries where the Side feature has special attributes for controlling the starting point of the cut and for controlling the region that is cut.
- Features defined by open curves have endpoints which do not meet.



Creating a side feature

1. Click *New Feature* .
2. Click *Side* and click *Next*.
3. Select a curve from the Curve list box. The curve can be a closed loop or an open curve in which the end points do not meet.
4. An arrow is displayed showing the side of the curve that will be cut. If the arrow is pointing the wrong way, click the *Flip side* check box.
5. Click *Next*.
6. The Z height of the curve is displayed. If you want to offset the feature in Z, specify an *Offset* value.
7. Click *Next*.
8. Set the *Depth* of the pocket. The Depth sets to the overall height of the side feature as cut into the stock.
9. Set the Bottom Radius if desired.
10. Set the Draft angle if desired.
11. Chamfer sets the depth of a 45 degree chamfer cut at the top edge of the feature.
12. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

By default, an outer side feature will use the stock boundary as the outer extent of the feature. To bound the extent of the boss cut:

1. Double click the feature. The Properties dialog box comes up.

2. Click the *Stock curve* button. Click the name of the curve you want to use as the outer extend of the side.

Note: If **_stock boundary_ is selected it indicates that the stock boundary will be used as the outer extent. The stock boundary will automatically be used as the outside extend of the boss only when the current UCS is parallel to one of the block faces. If your UCS isn't parallel to a stock face, boss features need to have a stock curve included with them.

3. If you want the wall of your feature to have a special cross section, click the X Section button to open a dialog box to select the curve that matches your cross-section shape.

4. Click *OK*.

Side control tab

The Side control tab is where you set which side of the curve to mill. FeatureCAM makes its decision based on the direction of the curve. To see the side of the curve the will be cut, click on the curve in the list. An arrow shows you the side that will be cut. To change sides, select the curve and click the *Reverse* button.

Manufacturing hints for a side

The region that is cut for a side feature is controlled by one of the following:

- The stock boundary, which is extracted automatically for you if you do not explicitly specify a Stock Curve.

- The Stock Curve that you provide for the side feature.

- The offset distance specified by the Total Stock attribute.

If you set the *Total Stock* attribute to a positive number, then a roughing pass and finish pass are performed in the region between your side feature and a curve offset by the *Total Stock* value. To perform only a single pass around your feature, set the *Total Stock* attribute to a positive value and set *Finish Allowance* to 0 to turn off the finish cut.

Do not create a profile side from a curve that extends more than a tool radius from the stock boundary. The manufacturing of these features is unpredictable. To correct this problem, move your curve onto the stock boundary or to within a tool radius of the boundary.

Manufacturing a side

FeatureCAM follows this general process:

- Analyze the curve to determine what tool to use.

- Pick feeds and speeds based upon the material being machined.

- Generate a roughing pass, possibly in multiple Z steps based on the depth of the side.

- Generate a finishing pass.

- If the boss has a draft angle, cross section curve or corner radius, see Manufacturing draft angles or bottom radius regions

The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a side is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the curve that defines the boss to determine what size tool to use. The smallest corner and tightest passage determine the largest tool that can cut the side without gouging (see *Tool % of arc radius*).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the side.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by pre-drilling (see Pre-drill diameter and Pre-drill point). For open curves, Lead distance and Lead in /out angles control the horizontal approach to the material.

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection, after roughing, the roughing tool is used to finish the side. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has the finish pass ramp into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter). For open curves, Lead distance and Lead in /out angles control the horizontal approach and exit from the material.

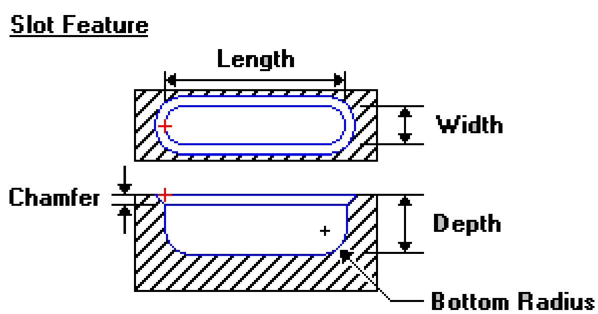
Finish passes and overlap make the tool go across or around the side a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall.

Retract removes the tool from the stock area and sets up for the next operation.

Slot feature

Slots are similar to rectangular pockets, but have round ends equal in diameter to the width of the slot.



Creating a slot feature

1. Click *New Feature* .
2. Select *Slot* and click *OK*.
3. Set the slot's *Length*, the long dimension of the slot.
4. Enter the slot's *Width*, the narrow dimension of the pocket. The width of a slot does not have to match the diameter of a standardly available endmill, unless you are making a Simple slot. If an exact match cannot be found, then a smaller tool is selected and multiple horizontal passes are performed.
5. Set the *Depth*, the distance into the material the slot is cut.
6. Set the Bottom Radius if desired.

7. Set the Draft angle if desired.

8. *Chamfer* sets the depth of a 45 degree chamfer cut at the top edge of the feature. Leave this value at 0, the default, for no chamfer.

9. Simple simplifies the manufacturing strategy for the Slot. If *Simple* is turned on, the slot is manufactured by making a single pass down the center of the Slot with a tool whose radius is equal to the *Width* of the slot.

10. Click *Next* and set the XYZ position of the center of the left corner arc of the feature.

11. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Manufacturing a slot

FeatureCAM follows this general process:

- Analyze the slot dimensions to determine what tool to use.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps based on the depth of the slot.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a slot is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool Selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the dimensions that defines the boss to determine what size tool to use. For a Simple slot, the tool must match the width of slot which also guarantees the correct radius at the ends of the slot. For normal slots, the largest tool that can cut the slot to width and still leave the Finish allowance is selected. (See *Tool % of arc radius*).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the slot.

Operation type	**Automatically selected tool**	**Possible user overrides**	**Notes**
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by predrilling (see Pre-drill diameter and Pre-drill point).

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02. For a simple slot, Finish allowance has no effect.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom.

Tool selection**, after roughing, the roughing tool is used to finish the slot. Use finish** tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has the finish pass ramp into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter). Not available for a Simple slot.

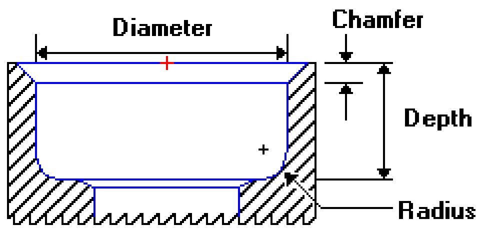
Finish passes and overlap makes the tool go around the slot a number of times set by Finish passes, and will overlap the starting point by an amount controlled by Finish overlap. Not available for a Simple slot.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall. Not available for a Simple slot.

Retract removes the tool from the stock area and sets up for the next operation.

Step bore feature

A Step Bore feature is a series of nested circular pockets. You can specify a Step bore step by step, or use a number of concentric circles as the part's curve where each circle defines a level's diameter.



Creating a step bore feature

1. Click *New Feature* .
2. Select *Step Bore* and click *Next*.
3. The default step bore feature has two steps. Each step corresponds to a row of the table.
4. To modify the top step:
5. Click the top row in the dialog box. Note that the initial values for this step are entered into the dimensioned drawing.
6. Edit the Diameter, Radius Chamfer and Depth for this step. The Depth measured from the top of the step bore, not the top of the current step.
7. *Through* sets display of the Step Bore without a bottom. Clicking Through does not make the feature pass all the way through the stock. You must set the depth value deep enough.
8. *Single Point Bore* sets the step to be finished with a boring bar. This mills the step to a tight tolerance. Set the checkbox if you want to finish the step with this technique.
9. Click Set to update the values in the table.
10. To modify the second step perform the same procedure on second row of the table.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

To add an additional step:

1. Click on the row in the table after which you want to insert.
2. Enter the parameters and click *Add* to insert the step parameters into the table.

If you want to delete a step:

1. Click on the row in the table
2. Click *Delete*.

When you have entered the step bore parameters

1. Click *Next* and enter the location of the center of the top bore.
2. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Manufacturing hints for a step bore

The Step Bore feature may also be used to produce a single depth round pocket. A Step Bore feature initially has two steps. Delete the second step and enter the appropriate dimensions for your round hole as the first step.

Manufacturing a step bore

FeatureCAM follows this general process:

- Analyze the dimensions to determine what tool to use.
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the step.
- Generate a finishing pass.

There are infinitely many variations on this process. The process can be fine tuned primarily in two places: the Default Machining Attributes dialog, to tune only a feature, use the Tools, Milling, Strategy and Misc. property tabs for the feature. The tooling database also has a large impact on how a step bore is machined, and the feed/speed database helps to determine the feeds and speeds used.

Tool selection

The first step is to pick a tool from the current tool crib (see the Manufacturing menu). The most important criteria are diameter and length. If a tool can't be found to satisfy the criteria, then you get an error and NC code is not generated.

Tool diameter FeatureCAM analyzes the diameter that defines the step bore to determine what size tool to use. The largest tool that can cut the step without gouging is selected (see *Tool % of arc radius*).

Tool length FeatureCAM picks a tool that has flutes long enough to cut to the bottom of the step bore depth for that level.

Operation type	Automatically selected tool	Possible user overrides	Notes
Roughing	endmill	face mill, endmill	If the feature has a bottom radius or draft angle see page 117.
Finishing	endmill	face mill endmill	
Chamfer	chamfer mill	spotdrill, centerdrill, countersink, chamfer mill	

Feeds and speeds

FeatureCAM chooses feeds and speeds for all of its milling using the Feed/Speed database that you can customize. Feeds and speeds are determined based upon the stock material.

Roughing

The critical aspects of roughing are:

Getting to depth The tool needs to get to depth, and this can be accomplished by a zigzag in Z (the default setting and influenced by Max ramp angle), by plunging, or by predrilling (see Pre-drill diameter and Pre-drill point).

Vertical step FeatureCAM cut depth is no more than 100% the tool radius (see Rough depth and Rough pass Z increment).

Horizontal stepover FeatureCAM moves over laterally a percentage of the tool diameter (controlled with Rough pass %,) as it steps across the feature.

Finish allowance controls how much material to leave for the finishing pass. By default this is 0.02.

Finishing

By default, the bottom is not finished. The roughing tool removes all of the material in Z. This is controlled by Finish bottom. Setting the Single point bore option for a step disables the other finishing options.

Tool selection, after roughing, the roughing tool is used to finish the step bore unless it will be finished as a Single point bore. Use finish tool commands FeatureCAM to choose a separate finishing tool (that has same characteristics unless you override them).

Ramp on has the finish pass ramp into the material with an arc equal to a percentage of the tool diameter (see Ramp diameter). Disabled for a Single point bore finish operation.

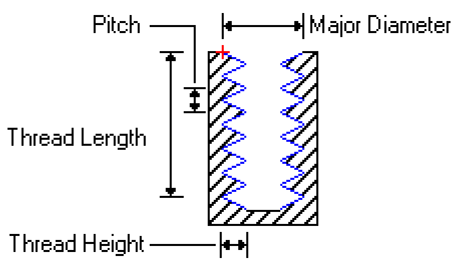
Finish passes and overlap has the tool go around the step a number of times set by Finish passes, and overlaps the starting point by an amount controlled by Finish overlap. Disabled for a Single point bore finish operation.

Ramp off uses another arc of the same size as the ramp on to move the tool away from the finished wall. Disabled for a Single point bore finish operation.

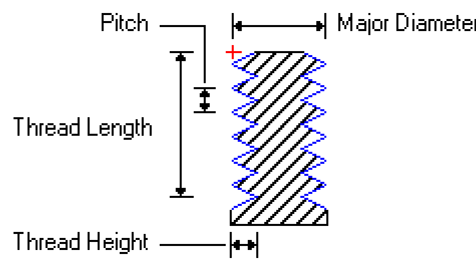
Retract removes the tool from the stock area and sets up for the next operation.

Thread mill feature

A thread mill feature allows you to mill a thread in a circular pocket or on a circular boss.



ID Thread Feature



OD Thread Feature

Creating a thread mill feature

1. Create a rounded step bore or pocket feature or a round boss to thread.
2. Click *New feature*  and select *Thread Milling*.
3. Click *Next*.
4. Enter ID for an inner diameter thread or OD for an outer diameter thread.
5. Select right or left hand.
6. Enter the Major Diameter, Pitch, Height and Thread Length. Note that for an OD thread, set the Major Diameter to the diameter of the boss. For an ID thread, set the Major Diameter to be the step bore diameter – Thread Height.
7. If tapered, enter the taper angle.
8. Click *Next and* Enter the X, Y and Z locations of the center of the top of the feature.
9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Dimensions that have blue labels can be extracted from objects in the graphics window. See 69 for more information.

Restrictions of thread milling

The toolpaths are accurate for UN or ISO metric threads. Adjustments must be made to the thread height or diameter to adjust for the different thread forms.

Manufacturing a thread mill

How a thread mill feature is manufactured

1. An appropriate tool is selected .
2. Feeds and speeds are calculated.
3. A single thread milling operation is created.

Thread milling tool selection

The default selected tool will have:

1. The same pitch as the thread.
2. The internal/external classification will match the feature.
3. The overall length will be greater than the thread length.
4. If the thread is tapered, the tool must have the same taper.

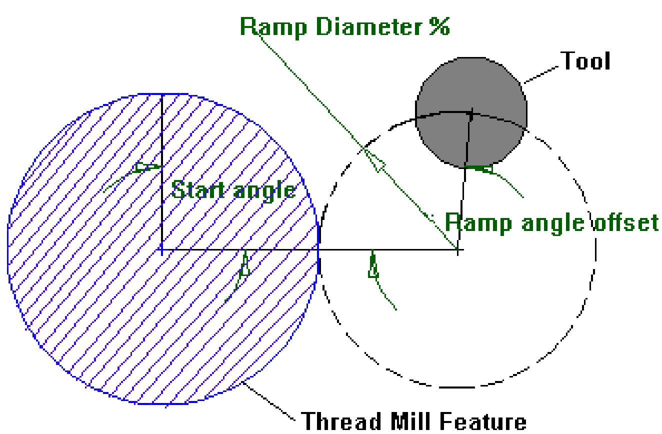
A tool with a longer cutter length is preferred.

Thread milling feeds and speeds

Thread milling uses the Profile finish speeds and feeds.

Thread milling operation

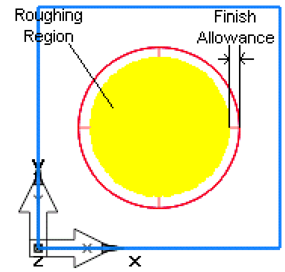
1. The tool ramps onto the feature according to the attributes shown below.



2. The tool then spirals either up or down the feature depending on the setting of Feed Dir. Cutter compensation is used on the toolpaths if cutter comp is turned on. Both of these settings are found on the Strategy page.
3. The overlap between revolutions is controlled by the Tooth overlap attribute.
4. The tool then ramps out. This move is controlled by the same attributes as the ramp in.

FeatureMILL 2.5D rough milling algorithms

For boss and pocket features, FeatureCAM provides two different milling methods for roughing.

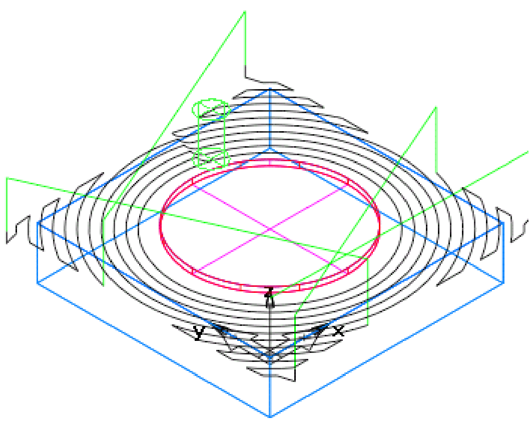


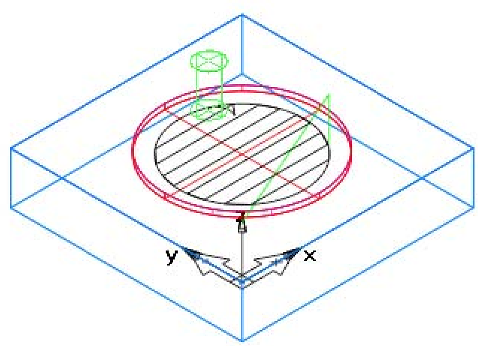
The offset method uses a series of offset curves as the shape of the toolpaths. The zig-zag method uses straight toolpaths that are parallel to each other. Regardless of the roughing method selected, the feature is roughed to within the finish allowance of the boundary. The roughing region is determined by offsetting the boundaries of the feature by the finish allowance

Offset roughing milling method

For bosses, the curves of the boss are offset and then clipped against the shape of the stock. When using a square piece of stock the toolpaths tend to cut the four corners first, and then work their way inward. The extent of the toolpaths can be altered by using a stock curve of total stock.

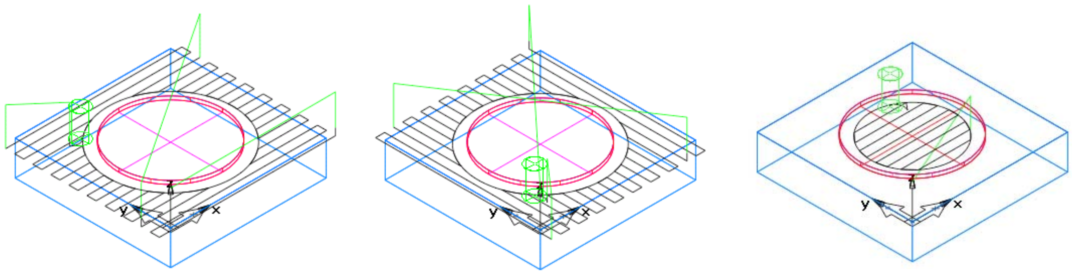
For pockets, the boundary of the pocket is offset and the toolpaths are cut starting from the center of the pocket. The shape of the stock does not affect the toolpaths.





Zig-zag milling method

For bosses the toolpaths are laid in parallel lines across the stock and clipped against the boundaries of the bosses.   The starting point is one of the four corners of the stock. The angle of the toolpaths can be changed, but the neighboring toolpaths are always parallel. For pockets, the parallel toolpaths are laid inside of the pocket boundary.

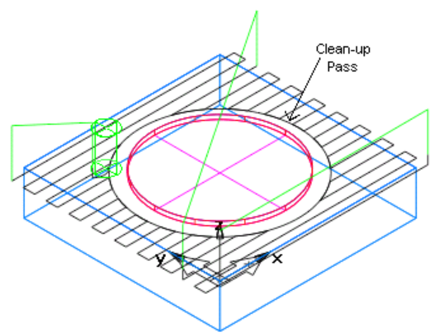


After the parallel paths, a cleanup pass is then performed around the boundaries of bosses, boundaries of pockets and boundaries of pocket islands.

Zig-zag roughing applies to rectangular pockets, step bores, slots, bosses, pockets, and non-simple grooves only.

For features that are cut multiple depths or multiple tools, you can even mix roughing methods within a feature. It is recommended to compare cutting times when deciding which method is correct for a particular feature.

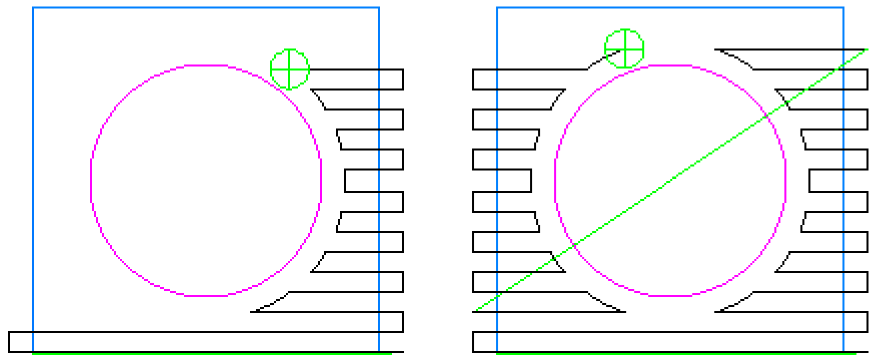
Zig-zag cleanup pass



The zig-zag roughing pass has two phases, the parallel roughing phase and the boundary cleanup phase. The cleanup phase, cleans up the boundaries of the feature to ensure a uniform finish allowance. The tree view for the feature only shows a single feature, so the cleanup phase uses the same feed and speed values as the roughing pass The number of cleanup passes is determined by the *Cleanup passes* attribute. If *Zig-zag cleanup passes* is set to 0, the cleanup pass is not performed. If *Zig-zag cleanup pass* is set to 1, a single pass is performed along the boundaries of the roughing region. If set to a number larger than 1, multiple cleanup passes are performed. The default spacing of these passes is controlled by the *Cleanup stepover* attribute. To more finely control the spacing of multiple cleanup passes, set the zig-zag cleanup stepover milling attribute. The ramping onto the cleanup pass is controlled by the ramp diameter of the Stepover tab.

Reordering zig-zag paths

If bi-directional cut is turned on on the strategy page or the reorder attribute is checked on the milling page, the toolpaths are sequenced so that it finishes one region before moving on to the next. The figures below demonstrate this reordering. The toolpaths finish the region on the right of the boss before moving on to the region on the boss’s left side. If bi-directional cut is off and reorder is unchecked, the toolpaths move across the part without any reordering.

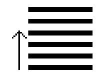
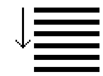
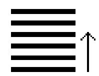
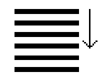
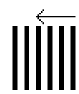
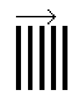
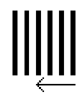
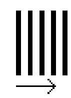


Comparison of offset and zig-zag milling methods

1. The zig-zag toolpaths are parallel lines. The offset toolpaths are offsets of the part boundaries.
2. The zig-zag method has even tool load across each line of the toolpath. The offset method has higher loads at corners.
3. Zig-zag usually requires a clean-up pass.
4. Zig-zag only applies to pockets and bosses.
5. The offset method is limited to stepovers of 50% of the tool diameter or less. Zig-zag can use stepover values of up to 99% of the tool diameter.
6. Zig-zag is often a better technique for cutting pockets on horizontal mills. Zig-zag can start at the bottom of the part and work its way up. In this case, the chips will fall away and not be re-cut. Offset toolpaths start in the middle of the pocket and for concentric rings. Chips can be re-cut at the bottom of the pocket.
7. For bosses with square stock, the offset method cuts the corners first, while the zig-zag technique cuts along an edge of the stock.
8. For climb milling a pocket, the offset method will retract less than zig-zagging. The zig-zag method must retract after each row of the toolpath, while the offset method can feed to the next concentric path.
9. If finishing the bottom of a boss or pocket, the texture of the bottom of the features will reflect the shape of the toolpaths.
10. The offset method supports helical ramping and plunge points, while the zig-zag method does not.

Controlling zig-zag milling

The following chart shows the relationship between the zig-zag angle and the climb mill setting. The picture in path column indicates the direction, start point and sequencing of the toolpaths. For example,  indicates toolpaths that are parallel to the X-axis. The start point is the lower left and the paths are sequenced from the bottom to the top. In the figures the X-axis of the setup is the horizontal axis, and the Y-axis is the vertical axis.

Zig-zag Angle	Climb Mill	Path	Zig-zag Angle	Climb Mill	Path
0	No		180	Yes	
0	Yes		180	No	
90	Yes		-90	No	
90	No		-90	Yes	

Zig-zag finish passes

The zig-zag technique only applies to a finishing pass if the bottom of the feature is being finished. In this case it behaves just like a single roughing pass

Manufacturing draft angles or bottom radius regions

Draft angles

Draft angle sets an angle for the feature wall. Use only positive numbers. Using tapered tools or ball end taper tools improves the quality of tapers and bottom and corner radii. These manufacturing attributes affect draft angles:

- Draft flat scallop height
- Draft radius scallop height
- Radius tool scallop height

Draft angles are cut as much as possible with flat tools and floors are finished with flat tools. Tapered operations are used only if requested and an appropriate tapered tool exists. An error appears if the operation is requested and an appropriate tool is not found. Bull-nosed tools are supported and used if no ball-end tools are found. No differentiation is made between bull-nosed and ball-end tools.

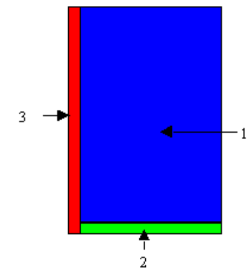
Here is a short list of criteria that affect the manufacture of draft angles:

- A tool of the exact size radius is not required to finish the feature, but without the exact size, the pass generally leaves a scallop.
- The finish operation with a tapered tool only goes down to the intersection of the straight side and the bottom radius.
- The rough operations are generated unless the scallop height is set to 0.

- Scallops are never at the top edge of the feature except when cut with flat-end tools.

Manufacturing steps for basic milled features

Features without a bottom radius are milled with the following steps.



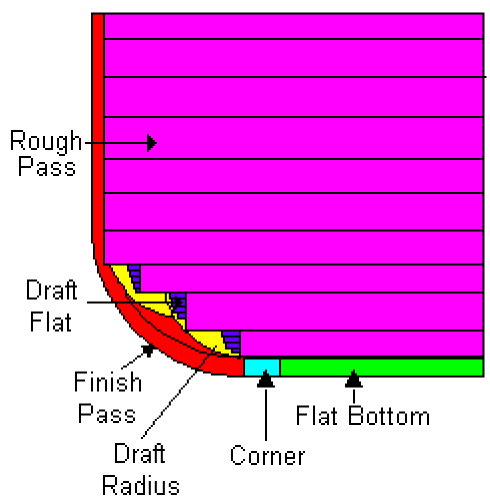
1. Rough middle of tool leaving finish allowance on bottom and sides with flat end tool

2. Finish bottom of feature

3. Finish walls

Manufacturing steps for milled features with bottom radius regions or cross sections

Features with a bottom radius are manufactured with some combination of the operations shown below.



1. The *Rough pass* cuts starting at the top and either roughs down to the bottom of the feature (if you are not finishing the bottom) or leaves a finish allowance at the bottom.

2. The stair steps of the roughing operation are knocked down by the *Draft flat* operation. This operation will only take a single pass at each Z level.

3. The stair steps of the draft flat operation can be further smoothed by the *Draft radius* operation. This operation will also only take a single pass at each Z level.

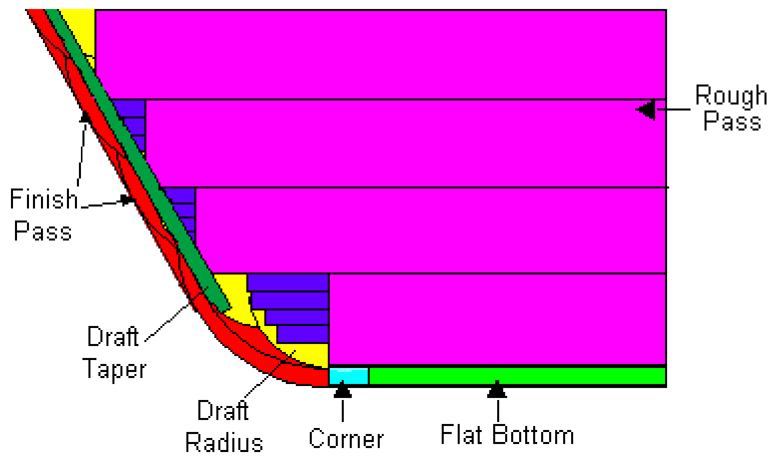
4. If the bottom of the feature is finished the *Flat bottom* operation is performed next with a flat-end mill.

5. The bottom radius and the walls of the feature are finished by the *Finish pass*.

6. If the feature has a tight corner on the floor that could not be finished by the flat bottom operation, a *Corner* operation is performed.

Manufacturing milled features with tapered walls

Features with tapered walls are manufactured similarly to those with just a bottom radius. In addition to the bottom radius operations there is an optional *Draft taper* operation shown below.



The *Draft taper* operation is a roughing operation for the tapered wall region. Typically, you would either use a draft taper operation or a draft radius operation.

For tapered features, the *Finish Pass* can be performed with a tapered or ball-end tool.

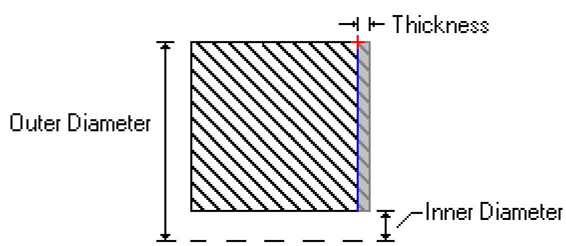
Chapter 8
Turning Features

This chapter provides details of turning features. You must license FeatureTURN to use these features. These features include:

- OD turning
- ID boring
- On-axis holes
- Grooves
- Threads
- Facing
- Cutoff
- Barfeed/barpull

Face feature

The Face feature faces off the front of a turned part.



Creating a turned face feature

1. Click *New Feature* 
2. Select Face as the Feature type and click *Next*.
3. Enter the top X value as the Outer Diameter.
4. Enter the bottom X value as the Inner Diameter.
5. Enter the amount of material to remove in Z as the thickness.
6. Click Positive if you want to cut in the +X direction or click Negative if you want to cut in the –X direction.
7. Click *Next*
8. Enter the Z coordinate of the left edge of the feature. Click *Next*.
9. By default, face features only generate a finish pass. If you want a roughing pass as well click the *Rough* button.
10. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's

automatic selections.

Manufacturing turned faces

FeatureCAM follows this general process:

- Determine what tool to use
- Pick feeds and speeds based upon the material being machined.
- If roughing has been requested generate a roughing pass possibly in multiple X steps depending upon the depth of the feature.
- Generate a finishing pass.

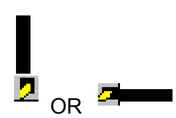
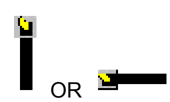
Turned face feature tool selection

The default selected tool for turning will have:

1. Lathe – turn tool tool type.
2. The proper orientation for the type of cut. See *Turn face feature tool orientations* for more information.
3. An 80 degree diamond is preferred but the default selected tool must have an included tip angle of at least a 55 degrees. You can override the tooling selection with a tool with a narrower diamond insert, but such a tool will not automatically selected.

Currently tip radius, tool length and insert grade are not taken into account.

Turn face feature tool orientations

Turret	Cut direction	Orientation
Back	-Z	
Front	-Z	

Turn face feature feeds and speeds

To view the recommended feed or speed value for a turned face operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended feeds and speeds are derived from the Face column of the turning feed/speed tables.

Turn face feature roughing

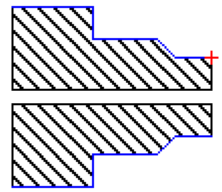
The roughing pass of the turn face feature is turned of by default. Click the Strategy page and click Rough to turn on the roughing pass. If ID has been checked on the Dimensions page, the roughing will be performed in the +X direction. If the OD has been checked, the roughing will be performed in the –X direction. The algorithm is the same as the turn feature roughing, except that the Withdraw Angle is fixed at 90 degrees and the Engage Angle defaults to 45 degrees and must be set to less than 90 degrees.

Turn face feature finishing

If ID has been checked on the Dimensions page, the finishing will be performed in the +X direction. If the OD has been checked, the finishing will be performed in the –X direction. The algorithm is the same as the turn feature finishing .

Turn feature

The Turn feature cuts an OD profile.



Creating a turn feature

Create the curve that defines the shape of the feature.

1. Click *New Feature* 
2. Click *Turn* and click *Next*.
3. Select the name of the curve from the Curve List box and click *Next*.
4. If you want to offset the feature in the Z direction, enter an *Offset* value. Click *Next.*
5. A turned feature will automatically create a roughing, and finishing operations. If you want to create fewer operations, select the appropriate operations. Click *Next*.
6. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

If you are working from a casting, you can include a stock curve to limit the extent of the roughing pass. To set a stock curve:

1. Double-click on the feature. The Properties dialog box comes up.
2. Click the Stock Curve button and select the curve from the Curve list box.
3. Click *OK*.

Restrictions on turn features

1. The curve must not cross the X-axis.
2. For face or backface roughing, the curve must cross the top of the stock or the max diameter must be set to the maximum X of the curve.

Manufacturing turn features

FeatureCAM follows this general process:

- Determine what tool to use
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the feature.

- Generate a semi-finishing pass.
- Generate a finishing pass.

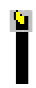
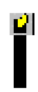
Turn feature tool selection

The default selected tool for turning will have:

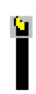
1. Lathe – turn tool tool type.
2. The proper orientation for the type of cut. See sections below for more information. An 80 degree diamond is preferred but the default selected tool must have an included tip angle of at least a 55 degrees. You can override the tooling selection with a tool with a narrower diamond insert, but such a tool will not be automatically selected.

Currently tip radius, tool length and insert grade are not taken into account.

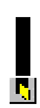
Turn feature turn rough, semi-finish or finish operation orientations

Turret	Cut direction	Orientation
Back	-Z	 OR 
Front	-Z	 OR 
Back	+Z	
Front	+Z	

Turn feature face rough operation orientations

Turret	Cut direction	Orientation
Back	-Z	 OR 
Front	-Z	 OR 

Turn feature backface rough operation orientations

Turret	Cut direction	Orientation
Back	-Z	
Front	-Z	

Turn feature feeds and speeds

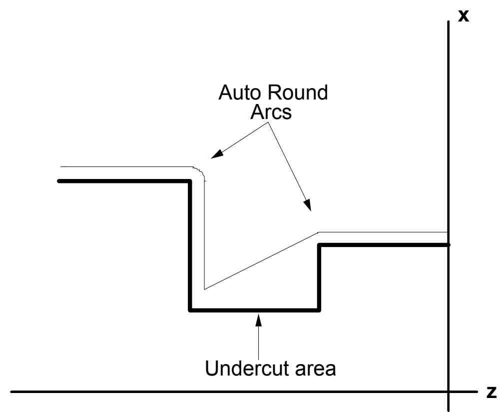
To view the recommended feed or speed value for a turning operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended feeds and speeds are derived from the OD column of the turning feed/speed tables.

Turn operation

The turn feature is manufactured the same as a Bore feature. See page 129 for more information.

Manufacturing hints for turn features

If the entire feature was not cut, it could be that the feature could not be cut entirely with the selected tool.   FeatureTURN checks the tool path to make sure that the tool can cut the specified path without "crashing" into the part itself. If a conflict is found, the system automatically alters the tool path so that a "safe" path is maintained. A message is displayed on the screen warning the user that the path has been changed.

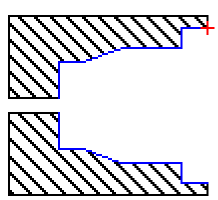


Turned hole feature

The Turned Hole feature is an on-axis hole. It is manufactured the same as a milled hole. See page 80 for details.

Bore feature

A Bore feature cuts an ID profile.



Creating a bore feature

1. Create the curve that defines the shape of the feature.
2. Click *New Feature* 
3. Click *Bore* and click *Next*.

4. Select the name of the curve from the Curve List box and click *Next*.

5. If you want to offset the feature in the Z direction, enter an *Offset* value. Click *Next.*

6. A bore feature will automatically create a roughing, and finishing operations. If you want to create fewer operations, select the appropriate operations.. Click *Next*.

7. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

If you are working from a casting, you can include a stock curve to limit the extent of the roughing pass. To set a stock curve:

1. Double-click the feature. The Properties dialog box comes up.

2. Click the Stock Curve button and select the curve from the Curve list box.

3. Click *OK*.

Manufacturing bores

FeatureCAM follows this general process:

- Determine what tool to use
- Pick feeds and speeds based upon the material being machined.
- Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the feature.
- Generate a semi-finishing pass.
- Generate a finishing pass.

Bore feature tool selection

The default selected tool for turning will have:

1. Lathe – bore tool type.

2. The proper orientation for the type of cut. If you have a Back turret the  orientation is preferred. For Front turrets, the  orientation is preferred.

3. An 80 degree diamond is preferred but the default selected tool must have an included tip angle of at least a 55 degrees. You can override the tooling selection with a tool with a narrower diamond insert, but such a tool will not be automatically selected.

Currently tip radius, tool length and insert grade are not taken into account.

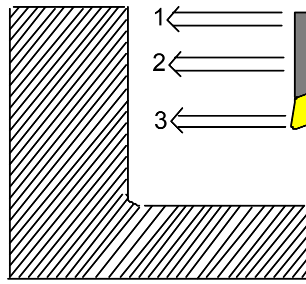
Bore feature feeds and speeds

To view the recommended feed or speed value for a bore operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended feeds and speeds are derived from the ID column of the turning feed/speed tables.

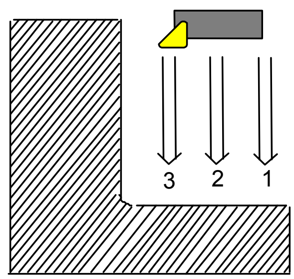
Turn and Bore feature roughing

Select one of the following options from the Strategy page:

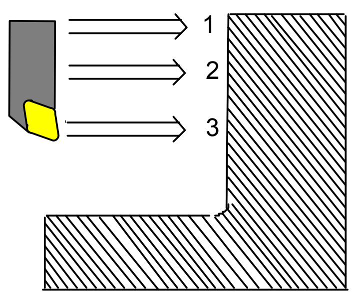
- Turn: roughs within the defined material boundaries by feeding parallel to the part's center line along the Z-axis while stepping down in the X axis. If the Negative checkbox is checked, the tool moves from right to left. If the Positive checkbox is checked, the tool moves from left to right.



- Face: roughs within the defined material boundaries by feeding from the outside of the part to the center while stepping into the face of the part along the Z-axis in the negative direction.



- Back face: roughs within the defined material boundaries by feeding from the outside of the part to the center while stepping into the face of the part along the Z-axis in the positive direction.

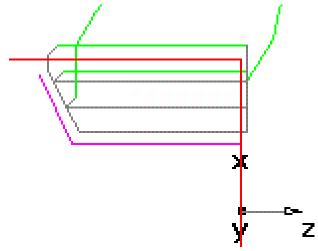


If you are actually creating your part from a casting instead of from bar stock, use the Stock Curve to limit the extent of the roughing pass. The details of this operation are controlled by the manufacturing attributes contained on the Turning tab. To display this tab:

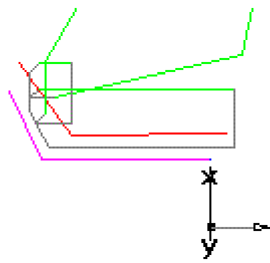
1. Click on the **Rough** operation in the tree view
2. Click on the Turning tab.

Stock curve for turn feature roughing

A stock curve controls the boundaries of the roughing pass.   This figure shows roughing without a stock curve.



This figure shows the use of a stock curve to limit the area that is roughed. For a turn roughing pass the curve must have a single value of X for every value of Z. For a facing pass or back facing pass the curve must have a single value of Z for every X.



The Boundaries parameter further restrict the region that is roughed.

Turn and Bore feature semi-finishing

The semi-finishing pass cuts a path that is offset from the part's surface to the tool's tip center for the entire (defined) curve. The offset value is ½* the X Finish Allowance and ½ * the Z Finish Allowance of the roughing pass. Profiling proceeds in the –X direction unless the Positive direction button is checked on the Strategy page.

Turn and Bore feature finishing

The finishing pass cuts a path that is offset from the part's surface to the tool's tip center for the entire (defined) curve. The offset value is just the tool tip radius. Profiling proceeds in the –Z direction unless the Positive Feed direction button is checked on the Strategy page. Use Finish passes to cut more than one finish pass.

Iscar® CUT-GRIP® tooling

FeatureTURN has special support for these tools. To activate this support:

1. Create a turning feature. Note that using this tool on a grooving feature does not create specialized toolpaths.

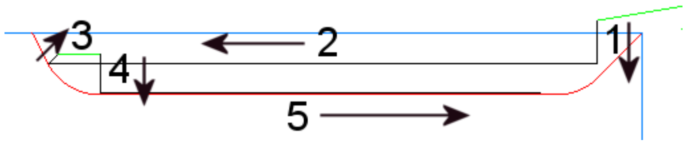
2. On the Strategy tab, set toolpath typeto *Cut-grip* to generate specialized rough and finishing strategies.

3. Override the tool to a cut grip tool. This is a tool with the *Cut grip tooling* attribute checked.

4. After cutting a part measure the deflection, set this parameter on the turning tab and regenerate toolpaths.

Cut-grip roughing

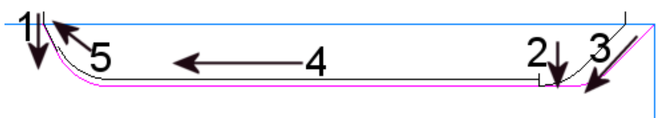
Roughing with Iscar cut grip tools is done in a bi-directional fashion. The steps of the cuts are as follows.



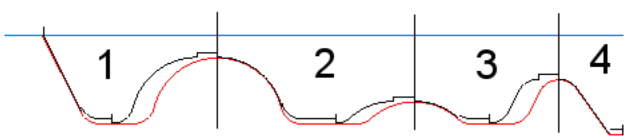
1. Feed straight down into the part. The distance is based on the depth of cut.
2. Feed straight over in Z.
3. Withdraw away from the wall and rapid back slightly in Z.
4. Feed straight down again based on the depth of cut.
5. Feed straight in the –Z direction.

Cut-grip finishing

The cut gripstyle of finishing is performed using a unique strategy that is enabled by having a grooving tool that cuts in both directions.



1. Cut down the left hand wall up to the bottom radius.
2. Rapid up and over and plunge a relief groove.
3. Cut down the right-hand wall, through the bottom radius into the relief groove.
4. Cut along the bottom of the groove. This move is offset by a deflectionamount.
5. Cut up the left-hand bottom radius. This move is offset by a deflectionamount.
6. If the feature has multiple groove cavities, each cavity is cut in the manner and the cavities are ordered in a left to right fashion.

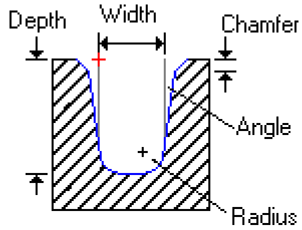


Deflection

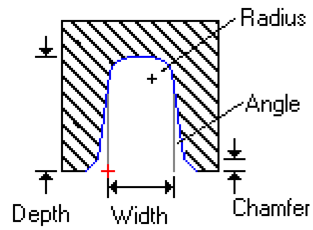
This attribute applies to finishing a turning feature with a cut grip turning tool. When using these tools, it is assumed that the tool will deflect during moves 4 and 5 in the figure below. This deflection will causes the tool to gouge. The toolpath is offset for these moves by the Deflection amount specified on the turning tab. This amount is obtained by cutting an initial part and measuring the deflection.

Groove feature

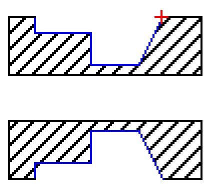
The Groove feature cuts an OD or ID groove from either dimensions or a custom curve.



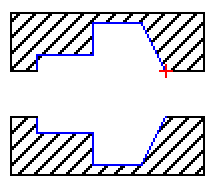
OD groove from dimensions



ID groove from dimensions

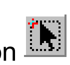


OD groove from a curve



ID groove from a curve

Creating a turned groove feature from dimensions

1. Grooves such as o-ring, radiussed, or necking grooves can be created from dimensions. There is no need to explicitly create a curve first. To create a groove from dimensions:
2. Click New Feature 
3. Select *Groove* from the *From Dimensions* category and click *Next*.
4. Specify the Orientation.
5. Choose X-axis if the groove’s depth is along the X-axis and then specify the Location as either OD for an outer diameter groove or ID for an inner diameter groove.
6. Choose Face if the groove’s depth is in the –Z direction.
7. Choose Backface if the groove’s depth is in the Z direction.
8. Specify the Diameter by either entering a number or clicking the pick point button  and picking a location.
9. Specify the Depth and Width.
10. For each side you can specify a chamfer, wall angle and bottom radius.
11. Set the Z coordinate of the left edge of the groove.
12. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM’s

automatic selections

Creating a turned groove feature from a curve

1. Create the curve that defines the shape of the groove.
2. Click *New Feature* 
3. Select *Groove* under the *From curve heading* as the Feature type and click *Next*.
4. Click the Curve button, and select the name of the curve from the Curve List box. Click *Next*.
5. If you want to offset the feature in the Z direction, enter an *Offset* value. Click *Next.*
6. Click OD if you want to cut from the top of the part down. Click ID if you want to cut from the X-axis up.
7. Click your *Orientation* as, *X-axis*, *Face*, or *Backface*. Click *Next*.
8. A turned groove feature will automatically create a roughing, and finishing operations. If you want to create different operations, click the operations you want. Click *Next*.
9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Manufacturing turned grooves

FeatureCAM follows this general process:

1. Determine what tool to use
2. Pick feeds and speeds based upon the material being machined.
3. Generate a roughing pass, possibly in multiple Z steps depending upon the depth of the feature.
4. Generate a finishing pass.

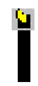
Turned groove tool selection

The default selected tool for turning will have:

1. Lathe – groove/cutoff type.
2. An insert Width of at least Groove Width – Finish Z Allowance
3. The shortest depth that will cut to the bottom of the groove.
4. The proper orientation for the type of cut. See *Groove tool orientations* for more information.
5. For ID grooves the Length of the groove is checked against the length of the holder to ensure that it will extend far enough into the part.

Currently tip radius, and insert grade are not taken into account.

Groove tool orientations

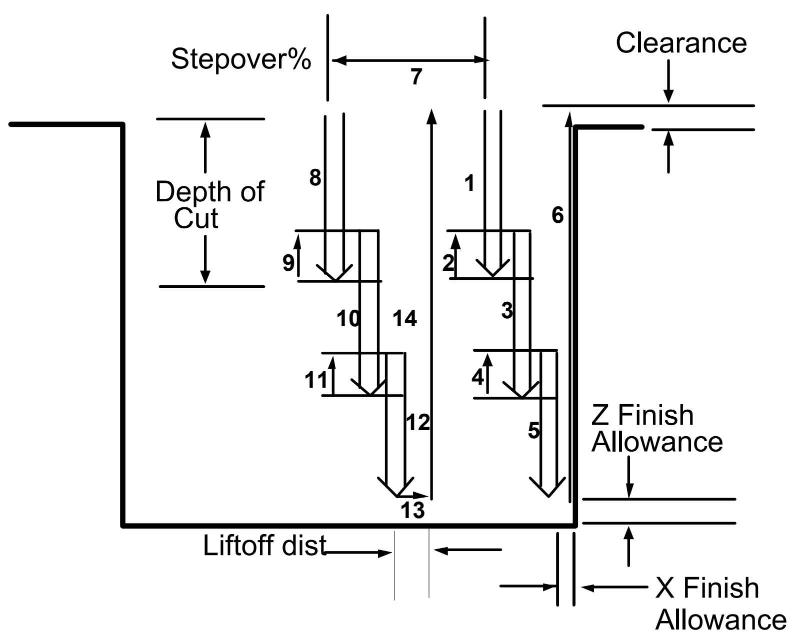
Turret	Diameter	Orientation
Back	ID	
Front	ID	
Back	OD	 OR 
Front	OD	 OR 

Turn groove feeds and speeds

To view the recommended feed or speed value for a turned groove operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended feeds and speeds are derived from the groove column of the turning feed/speed tables.

Turn groove roughing

The Groove feature is used to machine smaller slots or undercut regions. Grooves are roughed by plunging parallel to the X-axis retracting , stepping over in the –Z direction and then plunging again. The following figure shows the groove roughing algorithm and the manufacturing attributes that control the process.



If your groove has angled walls, the rectangular middle portion of the groove is roughed first and then the slanted walls are roughed as shown in this figure.

The details of this operation are controlled by the manufacturing attributes contained on the Turning tab.

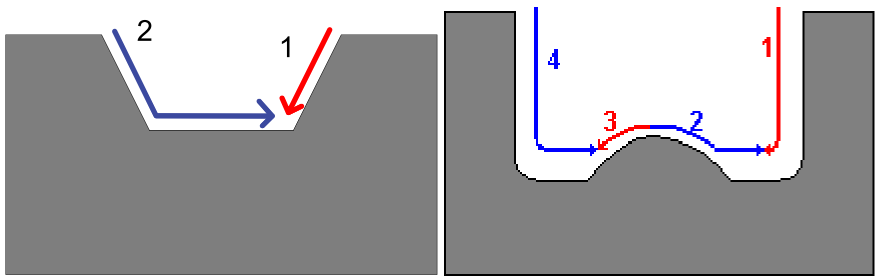
To display this tab:

1. Click on the Rough operation in the tree view

2. Click on the Turning tab.

Turn groove finishing

The finishing pass cuts a path that is offset from the groove's profile to the tool's tip center for the entire (defined) curve. The offset value is just the tool tip radius. Grooves are finished using a technique called shoulder stroking. This technique ensures that the grooving tool never cuts in the upward direction. Profiling proceeds in the –Z direction until the curve moves up in X. The tool then rapids to the highest point and cuts back in the +Z direction. This process repeats until the entire groove is finished. The left-hand figure shows an example for a symmetric groove with angled walls. The right-hand figure shows an example of a groove with multiple valleys.



The details of this operation are controlled by the manufacturing attributes contained on the Turning tab.

To display this tab:

1. Click on the Finish operation in the tree view

2. Click on the Turning tab.

Cutting wide face grooves

FeatureCAM grooves rough in only one direction. The groove can be cut in the positive X or negative direction. Due to the curved shape of the face groove tool holders, face grooving tools have a limited range (between the Min plunge diameter and Max plunge diameter in which they can plunge.

To cut wide face grooves you sometimes have to create three grooves as follows:

1. Call the Diameter of your wide groove *D* and the width of *W.*

2. Pick a diameter between the Min and Max plunge diameter of the tool as your plunge diameter. (Call this diameter *P*),

3. Create one groove for roughing only and set the *Diameter* to *P* and the *Width* to *W – (D-P)*. On the *Strategy* tab set the Rough *Feed dir* to *Negative*.

4. Create a second roughing-only groove and set the *Diameter* to *D* and the *Width* to *(D-P)*. On the *Strategy* tab set the Rough *Feed dir* to *Positive*.

5. Create a third groove with only a finishing pass with the *Diameter* set to *D* and *Width* set to *W.*

Groove canned cycle options

Use simple canned cycle or computer cycle

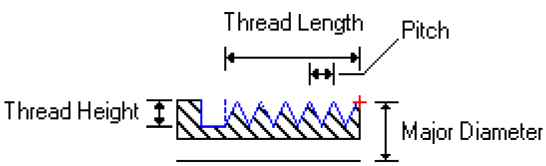
If this option is checked on the strategy page, a groove is output as series of computed moves or in the case of a straight walled dimensioned groove, the roughing operation is output as a canned cycle.

Use path canned cycle

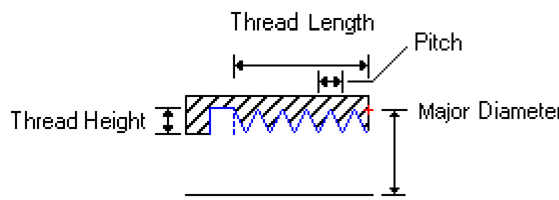
If this option is checked on the strategy page, the curve is output in the NC code twice and the roughing and finishing canned cycles reference the curves. If *reuse path* is checked then the curve is output only once and the roughing and finishing cycles reference the same curve.

Thread feature

The Thread feature cuts an OD or ID thread.

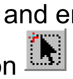
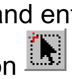


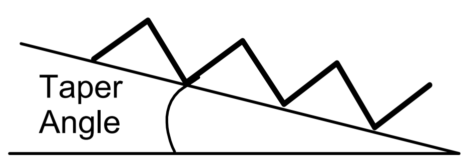
OD Thread



ID Thread

Creating a thread feature

1. Click *New Feature* 
2. Select *Thread* as the Feature type and click *Next*.
3. If you are threading the outer diameter, click OD and enter the Major diameter by either entering a number or clicking the pick point button  and picking a location.
4. If you are threading the inner diameter, click ID and enter the Major diameter by either entering a number or clicking the pick point button  and picking a location.
5. Click either Left hand or Right hand as the Thread.
6. Enter the Length, Height and Pitch. (Remember Pitch = 1/TPI)
7. If you are creating a tapered thread, check the Taper checkbox and enter the <u>angle</u>



8. Click *Next*.
9. Enter the Z location of the starting thread location.

10. A turned groove feature will automatically create a groove roughing pass for a relief groove, and a threading operation. If you want to create fewer operations, click the Strategy tab and select the appropriate operations. On the Strategy page you can also create roughing and turning operations for turning the part down to the thread diameter.

11. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Manufacturing threads

FeatureCAM follows this general process:

1. Determine what tool to use

2. Pick feeds and speeds based upon the material being machined.

3. Optionally create a rough and/or finish pass to turn the part down to the diameter of the thread. The creation of these operations is controlled by the Rough and Finish checkboxes on the Strategy page. See How a turn feature is manufactured for more details.

4. Optionally generate a roughing pass for the relief groove. The existence of this operation is controlled by the Relief Groove check box on the Strategy page.

5. Generate a threading pass.

Thread tool selection

The default selected tool for turning will have:

1. Lathe – thread tool type.

2. The insert must have a TPI range that contains the pitch of the desired thread.

3. The proper orientation for the type of cut.

Currently tip radius, tool length and insert grade are not taken into account.

Thread feeds and speeds

To view the recommended speed value for a threading operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended speeds are derived from the thread column of the turning feed/speed tables. Feed values are determined by the canned cycle on the machine tool.

Threading operation

The number of passes is controlled by the Number of Steps parameter on the Strategy tab. You can specify either Fixed or Calculate.

- If you select Fixed, then you must enter the total steps required for the threading operation in the Passes field. In this case, the passes are of a fixed depth.

- If you select Calculate, then the number of steps for the threading operation is calculated by the system. Additionally, if you select Calculate, then you must supply data for the Step 1, Step 2 and Minimum Infeed fields. In this case, the first step is cut at a depth of

Step1. The second and successive cuts are at a depth of Step 2. When the remaining depth is less than Minimum Infeed, it is cut with a single pass.

The other details of this operation are controlled by the manufacturing attributes contained on the Turning tab.

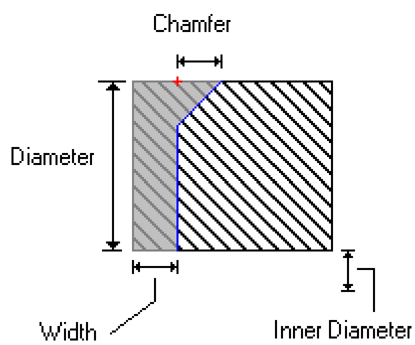
To display this tab:

1. Click on the Thread operation in the tree view

2. Click on the Turning tab.

Cutoff feature

A Cutoff feature is used to cut off the part with an optional chamfer.



Creating a cutoff feature

1. Click New Feature 

2. Select *Cutoff* as the Feature type. Click *Next*.

3. Enter the Outer Diameter.

4. Enter the Inner Diameter.

5. Enter the Width as the width of the cutoff tool you will use.

6. If you have a back chamfer, enter the value as the Chamfer.

7. Click *Next*.

8. Enter the Z coordinate as the left edge of the cutoff feature.

9. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Manufacturing cutoffs

1. Determine what tool to use

2. Pick feeds and speeds based upon the material being machined.

3. Generate a cutoff pass.

Cutoff feature tool selection

The default selected tool for turning will have:

1. Lathe – groove/cutoff type.
2. An insert Width equal to the Cutoff feature Width
3. The shortest depth that will cut to the bottom of the feature.
4. The proper orientation for the type of cut. For Back turrets, the  orientation is required. For Front turrets, the  orientation is required.

Currently tip radius, and insert grade are not taken into account.

Cutoff feature feeds and speeds

To view the recommended feed or speed value for a cutoff operation, click on the operation in the tree view and then click on the Feed/speed tab. The recommended feeds and speeds are derived from the Cutoff column of the turning feed/speed tables.

Cutoff feature operation

If there is no chamfer, the cutoff is performed as a simple plunge.

If there is a chamfer:

1 The cutoff groove is plunged down to the depth of the chamfer

2 The tool traces along the chamfer and then down the cutoff groove.

If there is a chamfer and Plunge Rough Chamfer is checked on the Strategy page:

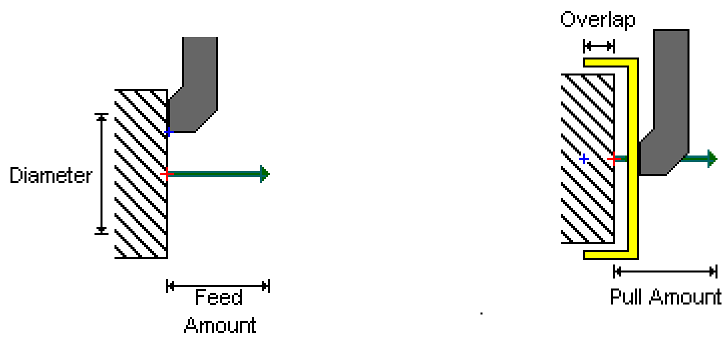
3 The cutoff groove is plunged down to the depth of the chamfer.

4 The chamfer is plunged roughed.

5 The tool traces along the chamfer and then down the cutoff groove.

Barfeed feature

A Barfeed feature provides support for both bar feeders and bar pullers.



Creating a barfeed feature

1. Click *New Feature* 
2. Click *Barfeed* and click *Next.*
3. Select *Barfeed* from the *Type* pull-down menu.
4. Enter the *Diameter* as the Y coordinate for the barfeed.
5. Enter the *Feed amount* as the amount of material you want to feed in the Z direction.
6. Click *Next*.
7. Enter the *Z* coordinate. This will be the initial z position of the feed motion.
8. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's automatic selections.

Creating a barpull feature

1. Click New Feature 
2. Click Barfeed and click *Next.*
3. Select *Barpull* from the *Type* pull-down menu.
4. Enter the *Overlap* distance. This will be the amount of material you will hold in the puller (in the Z direction).
5. Enter the *Feed amount* as the amount of material you want to feed in the Z direction.
6. Click *Next*.
7. Enter the *Z* coordinate. This will be the initial Z position of the feed motion.
8. Click *Next* to specify more manufacturing details or click *Finish* to accept FeatureCAM's

automatic selections.

How a barfeed/barpull is performed

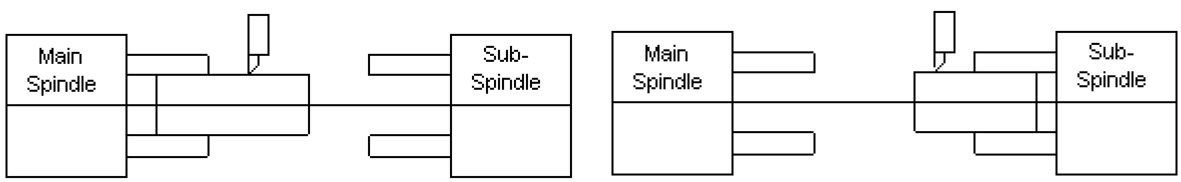
1. For barfeed operations, the tool rapids to the point (Diameter, 0, Z – Clearance).
2. For barpull operations, the tool rapids to a point in front of the stock along the Z-axis and then feeds to the point (0,0, Z-Clearance-Overlap)
3. The tool then feeds out the *Pull amount* or the *Feed amount.*
4. The feedrate is controlled by the *FPM* attribute.

Restrictions on barfeed/barpull operations

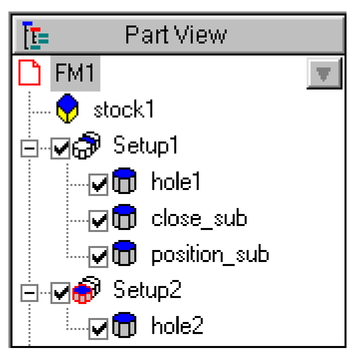
Barfeed features are only simulated with centerline simulations. 2D and 3D simulations ignore these operations.

Subspindle feature

Subspindles can be used to support the part from both ends of the stock or to change which spindle is used to hold the stock. To cut features on two different ends of your stock, you must first create setups at each end of your part with the Z directions pointing out from the stock, as shown below. (If your machine requires that the Z's of each setup point in the same direction, this can be changed in the post.)



The features from must be included in the proper setups. The subspindle commands can be located at the end of the main setup or the beginning of the subspindle setup. The part view, shown on the right, has the subspindle commands at the end of the first setup. If you are using the subspindle to support the end of the stock, order the subspindle features so that they occur when you need the extra support.



Subspindle feature types

A subspindle feature allows you manipulate the main and sub-spindles.

There is only one feature-type for sub-spindles, but it performs many different tasks.

First you specify which spindle you want to control, either the main or sub-spindle. You then select the action you want to perform from among:

Open the spindle – Opens the current spindle.

Close the spindle – Closes the current spindle.

Orient the spindle – Rotates the current spindle. Note the FeatureCAM will orient the spindle during cutting. This feature type is only needed to orient the spindle before grabbing the part or initializing the spindle position.

Turn spindle on/off – Direct control over rotating the spindle or turning it off. FeatureCAM automatically controls the spindle, but this feature type may be necessary to provide precise control of the spindle when moving from one spindle to the other.

Position the spindle – Direct positioning of the current spindle

Subspindle feature examples

FeatureTURN provides individual control over the different subspindle functions.

Switch from the main spindle to the subspindle is

1. Subspindle position
2. Subspindle close
3. Main spindle open
4. Subspindle position

Switch from the main spindle to the subspindle using a cutoff feature to cut the stock from the bar:

1. Subspindle position
2. Subspindle close
3. Cutoff feature
4. Subspindle position

How to create a subspindle feature

1. Click New Feature .
2. If you have the turn/mill option activated, click *Turning* as the feature type and click *Next.*
3. Select *Subspindle* as the Feature type. Click *Next*.
4. On the dimensions page, first select which spindle you want to control from *Main spindle* and *Sub spindle*.
5. Then select the action you want to perform and click *Next*.
6. If you selected *Open the spindle*, the strategy tab is displayed. You can then specify to extend the part catcher before opening and an optional dwell after spindle is opened.
7. If you selected *Close the spindle*, the strategy tab is displayed where you can specify an optional dwell after spindle is opened.
8. If you selected *Orient the spindle*, specify the orientation angle on the strategy tab.
9. If you selected *Turn the spindle on/off*, you are presented with the options of Off, CW (Clock-wise) and CCW (Counter Clock-wise). If you are turning on the spindle, you must specify the speed.

10. If you selected *Position the spindle*, the Location tab is displayed.

    - First specify the Z-coordinate of the *final sub-spindle location*.
    - Next decide how you want the sub-spindle to arrive at that location:
    - Rapid directly – The spindle will make a single rapid move from its current location to the final sub-spindle location.
    - Feed to intermediate location, then rapid to final location – The spindle will feed to the intermediate location and then rapid to the final location. This is useful for gradually removing the support of one of the spindles.
    - Rapid to intermediate location, then feed to final location - The spindle will rapid to the intermediate location and the feed the rest of the way. This can be applied to approach the part.
    - If you are using an intermediate location, specify its Z-coordinate.

11. Click *Finish*.

Generate single program for all setups

This option, located on the Indexing tab of the stock, for turn or turn/mill documents combines the toolpaths of all setups into a single setup. This means that when simulating the toolpaths, the toolpaths from all setups will be displayed. Only a single NC program will created. **This option must be set if you are using subspindle**. With this option turned off each setup will create a separate NC file.

Chapter 9
Groups and Patterns

Patterns allow you to quickly create the same feature at multiple locations. Groups allow you to collect together any set of objects so that they can be treated as one object.

Groups

Creating groups

To create a group of unlike features:

1. Build the various features that you want to group through the New Feature dialog box.

2. Control+click to select the features and patterns you want to group. Make sure that only the items you want grouped are highlighted.

3. Select *Patterns and Groups* from the Construct menu.

4. From the Pattern and Group sub-menu, select *Group*.

You can also create a group using the *New Feature* wizard:

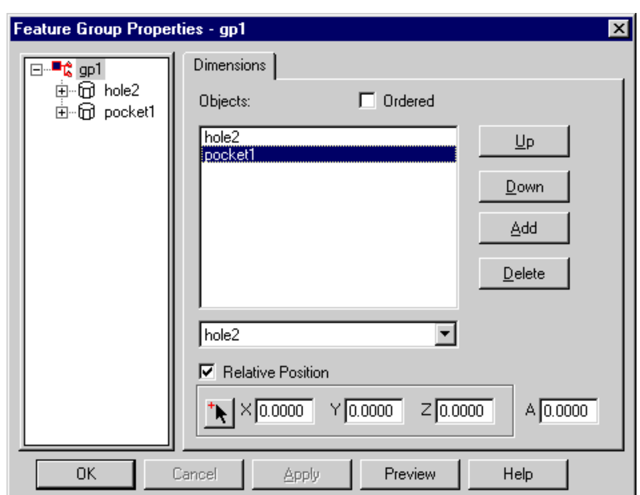
1. Create the features.

2. Click on the New Feature  step.

3. Click *Group* in the *From Feature* category.

4. Click *Next* and follow the instructions in the wizard.

Group dialog box



To enable you to efficiently model parts with repeated geometry elements in their design, FeatureCAM lets you specify groups of different features, in addition to feature patterns. When you group unlike features, you are essentially creating your own (user-defined) feature, that can then, in turn, be used as the basis feature for a pattern.

Groups are collections of features. Members of groups do not have to be the same object. Groups are often used to collect a set of objects for a pattern.

Specify patterns of a single object through the *Feature Pattern Properties* dialog box, accessed through the *Construct* menu's *Pattern and Group* option. The Milling, Drilling, and Misc. tabs are the same as for the individual features in the group. Use the tree view to move between the pattern properties, the object properties, and the manufacturing operation properties. Simply select the item in the tree view whose properties you want to see.

The features contained in the group are shown in the *Objects* list. Click on name in the list to highlight the object in the graphics window.

Object list box

Object list box shows a comprehensive list of all the objects that currently exist on the active part model. You select the object you want to add to the group from this menu.

Ordered

Ordered check box controls whether the objects listed in the Grouped Objects list are manufactured in the order in which they are listed in that list.

NOTE: If you are having trouble with the order in which features are manufactured, put the features into a group, arrange the order you would like the features cut and click *Ordered*.

Down

Down moves the object that is currently selected in the Grouped Objects list down by one place.

Up

Up moves the object that is currently selected in the Grouped Objects list up by one place.

Add

Add lets you add the object currently selected in the Object drop-down list box to the current group definition.

Delete

Delete removes the object currently selected in the Grouped Objects list from the current group definition.

Angle

Angle rotates the current group counterclockwise from its X-axis location by the angle specified.

Group XYZ

Group X Y Z location allow you to move the current object to a new location. If the X Y Z coordinate position is (0,0,0), then the group's objects all remain in the same location at which they were independently defined.

Relative position

Relative Position box indicates that you want the current object (feature/pattern/group) to be located relative to the user coordinate system rather than the stock coordinate system.

Ungrouping objects

Once you have created a group, the objects cannot be selected individually.

To remove the grouping of objects without removing the objects themselves:

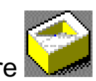
1. Select the group with the mouse.
2. Select *Ungroup* from the *Pattern and Group* sub-menu of the Construct menu.

NOTE: If you select the group and press Delete, you remove the group AND delete the objects in the group.

Patterns

Creating patterns

Patterns can be created in two ways.

1. While creating a feature, also select a pattern type and you will automatically create a pattern of the desired feature.
2. Create a feature first and then click on the New Feature  step. Click *Pattern* in the *From Feature* category. Click *Next* and follow the instructions in the wizard

Pattern dialog box

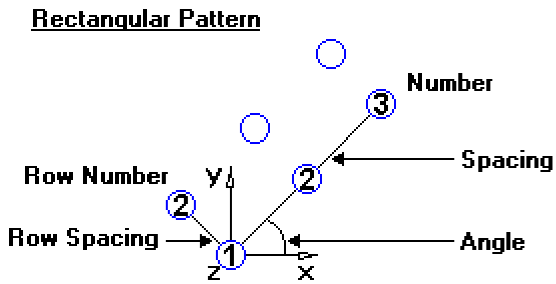
Use the tree view to move between the pattern properties, the object properties, and the manufacturing operation properties. Any box with a + in it can be expanded by clicking it. Similarly, a box with a - in it can be collapsed by clicking it. Simply select the item in the tree view whose properties you want to see. To modify pattern characteristics:

1. Select the pattern. The selection's Pattern Properties dialog box appears.
2. Modify the Type, Spacing, Number, or its basis feature's X, Y, Z coordinate location.
3. Click on a feature in the tree view to modify the dimensional and type characteristics of features within an existing pattern.
4. The Properties dialog box for the pattern's basis feature appears.
5. Modify the pattern's feature dimensions and type.

Rectangular pattern

A Rectangular Pattern arranges the specified number of like objects in two linear rows, with each row separating the objects at the specified spacing, and with the rows spaced as specified in row spacing, starting at the specified XYZ location.

Use the tree view pane to control whether you are looking at the pattern, the object, or the manufacturing operations for the pattern. Simply highlight your choice with the mouse. The rest of the dialog box changes to reflect your choice.



Row number

Row Number sets the number of rows of objects.

Spacing

Spacing sets the spacing between the columns. If you are creating a pattern in the standard Top view with the Y axis pointing up and the X axis pointing to the right, the Row Spacing is a distance in the X direction.

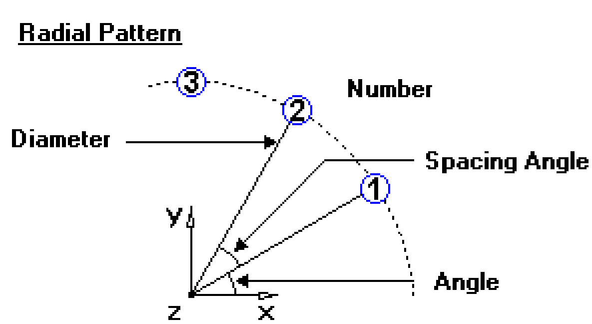
Row spacing

Row Spacing sets the spacing between the rows. If you are creating a pattern in the standard Top view with the Y axis pointing up and the X axis pointing to the right, the Row Spacing is a distance in the Y direction.

Radial pattern

A Radial Pattern arranges the specified features spaced along the circumference of a circle. The spacing can be set so you only arrange features along an arc instead of the whole circle. Spacing and angles are set with dimension settings.

Use the tree view pane to control whether you are looking at the pattern, the object, or the manufacturing operations for the pattern. Simply highlight your choice with the mouse. The rest of the dialog box changes to reflect your choice.



Diameter

Diameter sets the overall diameter of the pattern.

Spacing angle

Spacing Angle sets the space between the objects specified in degrees.

4th axis wrapping

4th axis wrapping offers the following additional parameters:

- *Setup XY plane* orients the features in the XY plane of the setup.
- *Around index axis* aligns the features on the OD of the part, pointing toward the index axis.

Turn/mill

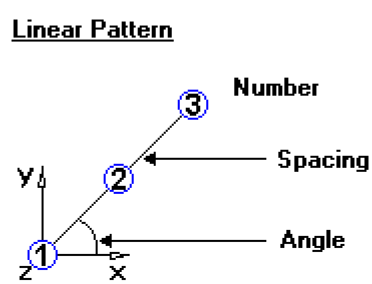
For turn/mill features there are the following additional orienting parameters:

- *Axial* orients the features to be parallel the Z-axis.
- Radial orients the features so that they are perpendicular to the Z-axis.

Linear pattern

A Linear Pattern arranges the specified number of objects in a line, at the specified distance apart, starting at the specified XYZ location.

Use the tree view pane to control whether you are looking at the pattern, the object, or the manufacturing operations for the pattern. Simply highlight your choice with the mouse. The rest of the dialog box changes to reflect your choice.



Spacing

Spacing controls the distance between objects in the pattern.

Number

Number sets the number of objects in the pattern.

XYZ

XYZ sets the location of the lower left-hand object of the pattern. Or click to pick a point with the mouse.

Points List Pattern

A Points Pattern lets you specify the location of each object explicitly. Use the tree view to move between the pattern properties, the object properties, and the manufacturing operation properties. Simply select the item in the tree view whose properties you want to see. There is a shortcut for some points patterns of holes. The different fields and buttons are described below.

Angle

Angle sets the angle of rotation around the Z-axis, measured counter-clockwise from the X-axis, for the first object in the pattern.

Local offset

Local offset controls whether the initial position of the object of the pattern is ignored. The position of the object is completely determined by its position in the pattern. If *Local offset* is checked, then the feature's position influences the created pattern.

Using an object's position in a pattern can be tricky. For example, use local offsets to create a pattern of profiled features, like a radial pattern of profile pockets. The recommended procedure is:

1. Create the profile relative to the UCS.
2. Enter the center of the pattern as XYZ coordinates.
3. Enter 0.0 as the radius.

The pockets share the relative position to their center as the initial curve did to its UCS.

Object list box

Object list box highlights the name of the object repeated in the pattern. To change the object click the down-arrow, then select the object to use in the pattern.

Point list

Point List contains a table of locations for each object of the pattern. If you pre-selected holes or points, the order in which you picked these objects is reflected in the Point List. Clicking a row of the table displays the values in the *X*, *Y*, *Z* and *A* options in the dialog box. You can then modify the values using the *Set*, *Add*, *Delete*, *Up* and *Down* options.

To highlight the location of a specific feature:

1. Click the *Pick feature*  button.
2. Select the feature in the graphics window.
3. The row of the point list table is highlighted to show the location of that feature.

Set

Set takes the values in the *X, Y, Z* and *A* options and overwrites the selected row in the table. You select a row of the table by clicking on any item in the *Point List* table.

To change a location in the *Point List* table:

1. Click the row in the table you want to change. The values are inserted into the X, Y, Z and A options.
2. Change the values in the X, Y, Z and A options.
3. Click Set.

Add

Add takes the values in the X, Y, Z and A options and inserts them at the end of the Point List table.

Delete

Delete removes the selected row of the Point List table.

Up

Up moves the selected row of the Point List table up one spot in the table.

Down

Down moves the selected row of the Point List table down one spot in the table.

X, Y, Z, and A

X, Y, Z and A contain a single location for an object. The X, Y, Z options specify the location of the feature and the A parameter specifies a local Z rotation about the object's center.

Curves

Curves loads point locations from a linear profile. Click Curves to bring up the Select Curves dialog box. Select the profiles that contain points you want to use as feature locations. Hold down the Control key (denoted by Ctrl on most keyboards) to select multiple profiles.

Sorting

The *Sorting* button brings up the *Point list sorting* dialog box. This dialog allows you to sort the objects in the following manners:

Shortest path: Starting with the first object in the list, a path is created by moving to the next closest object.

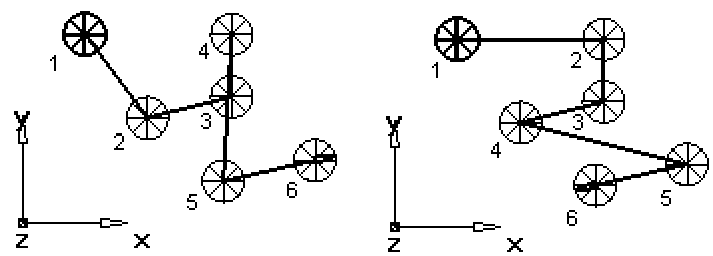
X ascending: Objects are sorted in increasing order according to their X coordinates.

X descending: Objects are sorted in decreasing according to their X coordinates.

Y ascending: Objects are sorted in increasing order according to their Y coordinates

Y descending: Objects are sorted in decreasing according to their Y coordinates

The left-hand figure shows a shortest path sorting. The figure on the right shows a y descending ordering.



Select circles

In the Edit menu is an option named Select Circles. It opens a dialog box where you enter a value for the radius of the circles you want to select. Click OK and FeatureCAM automatically selects all circles that are visible and in the current setup that have the radius you specified. You can then easily turn them into a Points List pattern for holes, or other operations as you wish.

To select circles of similar radius:

1. Pick one circle in the graphics window. If you do not pick a circle prior to bringing up the dialog box, you can pick one later.
2. Select *Select circles* from the edit window.
3. If you pre-selected a circle, the radius of selected circle will be displayed in the dialog box. If you want to select the radius from this dialog box, click on the *radius* label and then pick a circle.
4. Click *OK*. All circles of that radius will be selected.

The *tolerance* setting is used to determine the range of radii to select. All circles with radii greater than *radius-tolerance* and less than *radius+tolerance* are selected. You can now use these circles to create patterns of holes.

Chapter 10 Import/Export

Importing files

To import other CAD files:

1. Open a new or existing part file. You must have a part open to import geometry
2. Select *Import* from the File menu.
3. The Import function is not available unless you already have a part file open. A standard Browse dialog box for Windows appears. You can also set a few parameters in the Import Options dialog box such as replacing existing profiles with newly imported profiles with the same name. You can also set how fine to smooth the curves and to keep an IGES log file.
4. You can set the file type and directories you want to import from or simply browse until you locate the file you want to import.

FeatureCAM can import the following formats. If the format has a “(3D)” following it, FeatureMILL3D or the FeatureRECOGNITION Module is required to import this format.

DWG	AutoCAD files (3D for embedded solids)
DXF	AutoCAD files (3D for embedded solids)
IGES	Industry standard neutral CAD files (3D for surfaces)
GEO	Bridgeport EZ-CAM
XMT and XMB	Parasolid-based solid models (3D)
SAT and SAB	ACIS-based solid models (3D)
SLDPRT	Solidworks files
MOD and MODEL	Catia files (note this is priced separately).
IPT	Autodesk Inventor solid models

Importing IGES files

It is helpful to set the export settings of your CAD system for proper import.

- See SolidWorks and AutoCAD sections below.
- For other systems try and ensure that 3D surfaces are exported as trimmed NURBS.

Importing files from Pro/Engineer

If you are working with Pro/Engineer from Parametric Technology Corporation set the following IGES import options:

- Physically dependent - OFF
- Logically dependent - ON
- Both physically and logically dependent - ON
- See *Import/Export Options* for more details on these parameters

IGES entities

FeatureCAM can read the following IGES entities of version 5.3 or earlier. If the 3D column is checked, then FeatureMILL3D or the Solid Import Module is required to read these entities.

Entity	IGES Description	FM Object Type	3D
100	Circular Arc	Circle and arc	
102	Composite Curve	Curve	
104	Conic Arc	Curve	
106	Copious Data	Line	
108	Plane	Not supported	
110	Line	Line, multiple lines	
112	Parametric Spline Curve	Curve	
114	Parametric Spline Surface	Surface	x
116	Point	Point	
118	Ruled Surface	Surface	x
120	Surface of Revolution	Surface	x
122	Tabulated Cylinder	Surface	x
126	Rational B Spline Curve	Curve	
128	Rational B Spline Surface	Surface	x
141	Boundary	Curve	x
142	Curve on Parametric Surface	Curve	x
143	Bounded Surface	Surface	x
144	Trimmed Surface	Surface	x
186	Brep	Solid with Solid Modeling, Surfaces without Solid Modeling	x
304	Line Font Definition	Not supported	
408	Subfigure Instance	Geometry, curves, or surfaces	
410	View	Not supported	
502	Vertex	Point	X
504	Edge	Curve	X
508	Loop	Curve	X
510	Face	Surface without Solid Modeling	X
514	Shell	Surfaces without Solid Modeling, Solid with Solid Modeling	X

All other IGES entities are ignored. Look in the status bar and the log file for feedback about the IGES file import. After importing an IGES file you will told either “The IGES file has been imported successfully” or “The file “example.igs” is corrupted and cannot be imported.” A log file is created which details the IGES import process. The name of this file is specified in the Import Settings dialog box which is displayed with the Import Settings option of the File menu.

Importing DWG, DXF files

FeatureCAM supports the import of the below DWG and DXF entities. Up to the AutoCAD 2000 version 15 is supported for importing DWG and DXF files. DXF files have both ASCII and binary forms, DWG files have only a binary form.

FeatureCAM supports the import of the following DWG and DXF entities. If the entity has a “(3D)” listed after it, then FeatureMILL3D or the Solid Import Module is required.

DXF Entity	**FeatureCAM Object**
LINE	line from 2 points
POLYLINE	lines from 2points
POLYLINE with smooth vertices	curve (see Digitized data import/export options)
POLYGON MESH	surface (see Digitized data import/export options)
LINE3D	line from 2 points
POINT	point
SPLINE (DXF only)	curve
UCS	UCS
SEQEND	line from 2 points
LAYER	layer
VERTEX	line from 2 points
COLOR	object colors are maintained upon import
CIRCLE	circle center radius
	arc center begin end
	arc center begin end
	curve
ARC	arc center begin end
	curve, if asymmetrically scaled
INSERT	DXF BLOCK could contain any of the above
BODY	trimmed surfaces (3D)
SHEET	trimmed surfaces (3D)
3DSOLID	trimmed surfaces (3D)
DIMENSION	dimension

Importing dimensions from DXF and DWG files

AutoCAD files can contain two different spaces, model space and paper space. Model space is where you draw up geometry, 3D solids, etc. Paper space is the formatted space where multiple views are formatted. Most AutoCAD files have only model space. FeatureCAM can import dimensions in model space only.

AutoCAD dimensions are imported as FeatureCAM dimensions. That means that you can modify the color of the dimension, easily delete the entire dimension or use the Show or Hide fly-out menus to toggle the display of dimensions.

Simplifying 3D AutoCAD data for 2D import

Many AutoCAD models are comprised of higher level objects such as 3DSOLIDs or BODYs. If you have licensed FeatureMILL3D or the Solid Import Module, you can import these entities directly into FeatureCAM. If not, you must simplify this data before importing.

To import the geometry that is used to create the 3DSOLIDs, select your model and then use the EXPLODE command, or the EXPLODE fly-out from the modify toolbar, to repeatedly reduce your model into these more primitive elements.

When exploding solids you often get duplicate lines and circles. Any edge that was shared between two different faces will be duplicated. This repeated geometry causes trouble if you try to chain the geometry by double clicking. To work around this, you can

- Chain manually, by picking each piece individually
- Use AutoCAD to explode specific faces of the model (not the entire model), and then do chaining normally with double click or piece-to-piece
- Remove duplicate pieces in FeatureCAM and then do chaining normally
- To see if you have imported geometry:
- Select a line or circle.
- Look at the object's name listed in the status bar. Imported lines have names that begin with "_ln" and circles begin with "_circ"
- Select the object again.
- See if the same name is listed in the status bar.
- If a duplicate name appears and selected geometry is the same, delete the second occurrence of the geometry.

If your AutoCAD model contains layers, those layers will be retained upon import.

Importing ACIS files (3D)

This function requires FeatureMILL3D or the Solid Import Module.

CAD systems that use the ACIS® kernel can create files of type .SAT. These files contain the solid models created by systems including Mechanical Desktop®, CadKey® and others. These solids can also be imported in DXF files. FeatureCAM can import ACIS files of version 6 or earlier.

Note that FeatureCAM cannot import wire frame models, only solids.

Mechanical Desktop is a registered trade name of Autodesk Inc.

CadKey is a registered trade name of Baystate Technologies.

Importing Parasolid files (3D)

This function requires FeatureMILL3D or the Solid Import Module

CAD systems that use the Parasolid ® kernel can create files of type .XMT. These files contain the solid models created by systems including SolidWorks®, SolidEdge® and Unigraphics®. FeatureCAM can import ASSEMBLIES, and PARTS that are made up of BODIES. FeatureCAM can import files of version 12 or earlier of the Parasolid library.

SolidWorks is a registered trade name of Solidworks Inc.

Parasolids, SolidEdge and Unigraphics are registered trade names of Unigraphics Solutions Inc.

Importing Solidworks files

SolidWorks97, SolidWorks99 and SolidWorks2000 solid model files can be read directly into FeatureCAM. The files have a .sldprt extension. Only the final solid model is imported. No geometry or curves are imported and the construction history is also not imported. This means that you only get a single solid and not the individual design features from SolidWorks. Assemblies are also not supported.

Importing Autodesk Inventor files

Autodesk inventor .ipt files can be directly imported into FeatureCAM. Only the final 3D solid model is imported, 2D geometry, assemblies, sketches are not imported.

Importing Catia Files

Catia version 4 files can be imported directly into FeatureCAM. Note that importing Catia files requires the purchase of a separate optional module. These files must have a .mod or .model extension. In general, only 3D surface and solid models can be imported, but composite curves can also be imported. A log of the import is recorded in windows\temp\catia.txt.

Catia Object Type	**FeatureCAM Object Type**
Point	Not translated
Line	Curve
Circle	Curve
Ellipse, Hyperbola, Parabola	Curve
Curve	Curve
Composite curve	Curve
NURBS curve	Curve
Polynomial surface, B-spline polynomial surface	Surface
NURBS surface	Surface
Skin	Surfaces or solid depending on option settings.
Exact solid	Surfaces or solid depending on option settings.
Polyhedral solid	Surfaces or solid depending on option settings.

Working with imported geometry

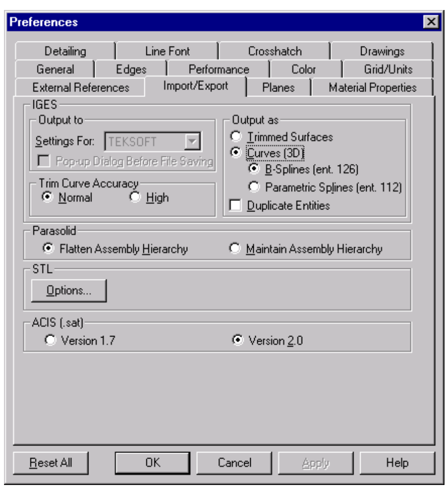
Imported geometry is often less exact than geometry you create directly in FeatureCAM . End points may not match and you may find that you have a lot of extraneous data on your screen. Deleting or hiding the extraneous data greatly speeds up the display of your part. You may also want to chain your data one piece at a time instead of using the automatic chaining.

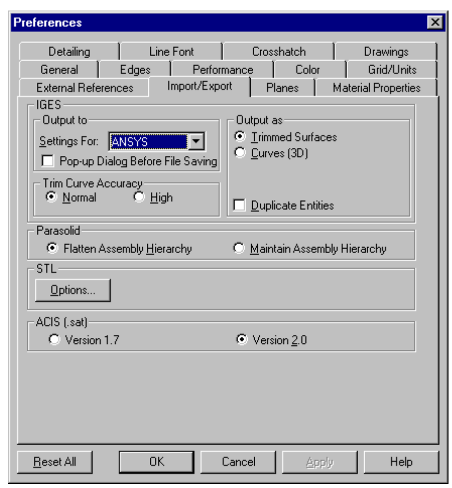
Gaps in your data make chaining your geometry into curves more difficult. With small gaps you may have to adjust the chaining tolerance set in Chaining in the Options menu. This tolerance represents the distance between endpoints that will automatically be bridged by the chaining algorithm. By increasing this tolerance you may be able to automatically close the gaps between endpoints during chaining. You can change this tolerance in the Chaining dialog box. On some data you may find that you must manually insert line segments or arc to close the gaps in the data. After closing these gaps, you should find that the data will chain more easily.

SolidWorks export settings

Use the following steps to set the IGES export settings in SolidWorks before exporting for FeatureCAM:

1. In the Tools menu choose User Preferences
2. Choose the Import/Export tab
3. Under Output As, choose Curves (3D)
4. Choose B-splines as shown below.
5. Under Output As, choose Trimmed Surfaces
6. Under Output To, select ANSYS as the Settings For field as shown below.
7. Click OK
8. Use Save As to export the part to an IGES file





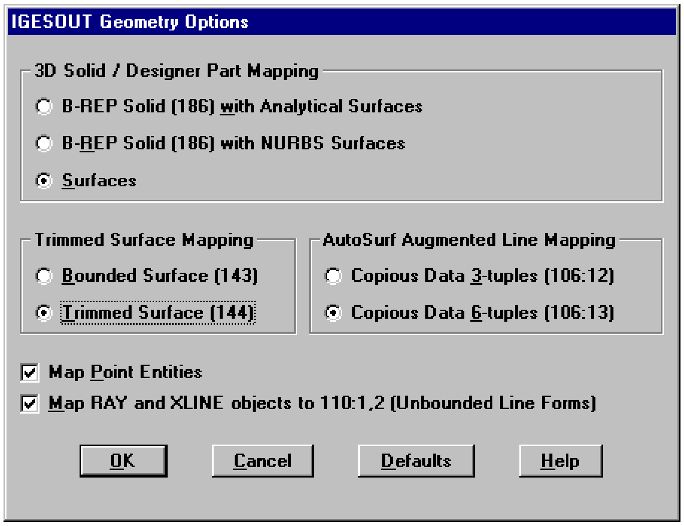
AutoCAD export settings

Use the following steps to set the IGES export settings in AutoCAD before exporting for FeatureCAM:

1. Select IGES Out… from the File menu
2. Select Edit Options…

3. Select Geometry …

4. Set the parameters as in this figure.



Importing Catia Files

Catia version 4 files can be imported directly into FeatureCAM. These files must have a .mod or .model extension. In general, only 3D surface and solid models can be imported, but composite curves can also be imported.   A log of the import is recorded in windows\temp\catia.txt.

Catia Object Type	**FeatureCAM Object Type**
Point	Not translated
Line	Curve
Circle	Curve
Ellipse, Hyperbola, Parabola	Curve
Curve	Curve
Composite curve	Curve
NURBS curve	Curve
Polynomial surface, B-spline polynomial surface	Surface
NURBS surface	Surface
Skin	Surfaces or solid depending on option settings.
Exact solid	Surfaces or solid depending on option settings.
Polyhedral solid	Surfaces or solid depending on option settings.

Exporting files

Send (File menu)

Send in the File menu is a simple way to send the active document to someone through e-mail. Trying to send a file with unsaved changes forces a file save operation. The specifics of the Send feature vary from installation to installation depending on your choice of e-mail client, network setup, and Internet connection. The behavior should be something similar to the following:

- Select Send in the File menu.
- The email client is started and a new message window appears.
- Address the message and give it a subject.
- Verify that the .fm file is attached (It should be attached automatically).
- Send the message.

If you don't have an email service to configure, this function won't work for you.

Exporting IGES Files

The different FeatureCAM categories of geometry, curves, features and so on are exported to IGES as shown in the following table:

FeatureCAM Entity	IGES Entity
2.5D Feature	144 = Trimmed surface(s) or
	142 = Curve(s) on parametric surface
Surface	128 = NURBS surface
Curve	126 = NURBS curve
3D feature	144's and 142's
Point	116 = Point
Line	110 = Line
Arc/Circle	100 = Circular arc
Dimensions, layers, & other attributes	not exported

Exporting DXF and DWG files

The DXFand DWG files exported are version 11 files. DXF files are exported in the ASCII form.

FeatureCAM Object	DXF Entity
Point	POINT
Circle	CIRCLE
Arc	ARC (Note that all arcs normal points in the +Z

Line	LINE
Layer	LAYER
UCS	UCS

Exporting XMT files

All solids in the part can be exported in an XMT file.

General import/export options

Set these options to control how FeatureCAM imports or exports files.

Always replace object on import when checked, overwrites objects of the same name when you import. Unchecked prompts for every replacement. You can't have two objects with the same name

Smooth EZ-MILL curves sets how to improve EZ-MILL curves on import as FeatureCAM has a higher resolution curve format.

Keep IGES import log file sets whether to keep a log of the import process for later review or troubleshooting.

Review IGES log displays the log file (in Notepad) after importing a file.

IGES log file name sets the path and filename for saving a record of the import process.

Number of decimal places in IGES export determines how finely data is exported to the IGES format.

Center stock automatically will alter the stock to automatically size and position itself so that it covers the imported data.

Heal .MOD solid will try and repair the faces or surfaces contained in the Catia .MOD file. This option can be time consuming since it attempts to:

- retrim the surfaces/faces against each other
- force edges of a surfaces actually lie on the surface
- simplify surfaces like converting a general surface into a cylinder.

Import .SAT .SLDPRT, .MOD, .DXF and .X_T as solids will import these files as solid models. If this option is turned off, the models are imported as surface models. It is recommended that you import these models as solids. If this option is enabled and the solid fails to import properly, you are asked if you would like to attempt to heal the solid to try and fix the import problem

IGES import filters

The following settings filter out entities during IGES file import.

Physically dependent controls which entities in an IGES file are imported. Checking this setting imports additional low level geometry. Physically dependent entities can not exist by themselves but is referenced by another entity. For example, a circular arc could be dependent on a composite curve. Without the Physically dependent switch being checked, only the composite curve would be imported. Can not exist unless the parent entity exists.

Logically dependent entities are entities that are part of a group. It is generally recommended to turn this switch on for import.

Both physically and logically dependent meets both logically and physically dependent criteria. It is recommended to keep this setting checked.

Digitized data import/export options

Digitized data is imported into FeatureCAM through DXF POLYLINE entities. The Digitized data import/export options affect how this data is imported. These settings only affect the import of the data. If you want to change one of these options, you must remove the old data and import the DXF file again.

Non-smooth polyline import

If a non-smooth polyline is imported it is imported as a curve if *As curve* is checked. If *As connected lines* is checked, the polyline is imported as a series of line segments.

Smooth polyline import

If the POLYLINE is has smooth vertices, it is imported as a curve. The DXF Smooth Polyline Import Method has two settings:

Interpolation: The curve will pass through the data. If the data is very dense, interpolation will sometimes cause the curve to wiggle in between the points. The spline interpolation technique is used to create the curve.

Approximation: The curve will come very close to the data points and is unlikely to wiggle for densely spaced data.

Polygonal mesh import

If the POLYLINE is a polygonal mesh it is imported as a surface. The DXF Polygonal Mesh Import Method has two settings:

Interpolation: The surface will pass through all of the data points. If the data is very dense, some wiggles may appear in the surface. A cubic hermite technique is used to create the surface.

Approximation: The surface will come close to the data points. The points are used as control points to the surface and the surface is unlikely to wiggle for densely spaced data.

Digitized data formats

Example for a two point curve

Note that text preceded by <--- is included only as a comment.

```
0
SECTION
2
ENTITIES
0
POLYLINE
0
VERTEX
10
0.0000                    <--- x0
20
0.0000                    <--- y0
30
0.0000                    <--- z0
0
VERTEX
10
1.0000                    <--- x1
20
1.0000                    <--- y1
30
0.0000                    <--- z1
0
SEQEND
0
ENDSEC
0
EOF
```

Example of digitized data format for a 3D point

Note that text preceded by <--- is included only as a comment.

```
0
SECTION
2
ENTITIES
0
POINT
10
0.0000                         <--- x
20
0.0000                         <--- y
30
0.0000                         <--- z
0
ENDSEC
0

EOF
```

Example for a surface defined by four corner points

Note that text preceded by <--- is included only as a comment.

```
0
SECTION
2
ENTITIES
0
POLYLINE
70
16
71
2              <--- Number of rows
72

2              <--- Number of columns
0
VERTEX
70
64
10
0.0000         <--- x of row 0, column 0
20
0.0000
30
0.0000
0
VERTEX
70
64
10
1.0000         <--- x of row 0, column 1
20
0.0000
30
0.0000
0
VERTEX
70
64
10
0.0000         <--- x of row 1, column 0
20
1.0000
30
0.0000
0
VERTEX
70
64
10
1.0000         <--- x of row 1, column 1
20
1.0000
30
0.0000
0
SEQEND
0
ENDSEC
0
EOF
```

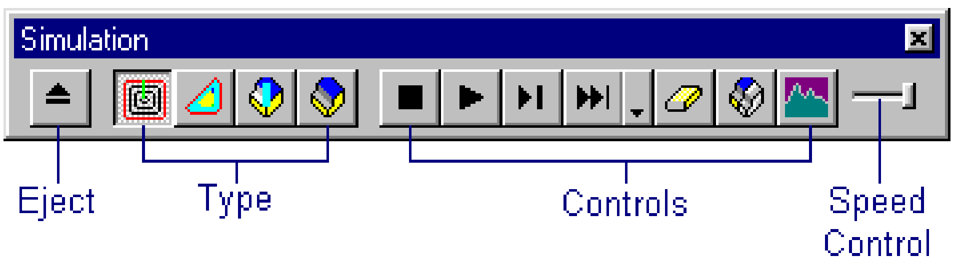
Example of digitized data format for a spline curve

Note that at least four points are needed for a spline. Also note that text preceded by <--- is included only as a comment.

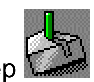
```
0
SECTION
2
ENTITIES
0
POLYLINE
0
VERTEX
70
8
10
0.0000                         <--- x0
20
0.0000                         <--- y0
30
0.0000                         <--- z0
0
VERTEX
70
8
10
0.0000                         <--- x1
20
1.0000                         <--- y1
30
0.0000                         <--- z1
0
VERTEX
70
8
10
1.0000                         <--- x2
20
1.0000                         <--- y2
30
0.0000                         <--- z2
0
VERTEX
70
8
10
1.0000                         <--- x3
20
2.0000                         <--- y3
30
0.0000                         <--- z3
0
SEQEND
0
ENDSEC
0
EOF
```

Chapter 11 Generating/Simulating Toolpaths

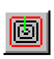
Generating toolpaths

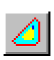


Simulating toolpaths requires three steps:

1. Display the Simulation toolbar by selecting the Toolpaths step  from the Steps toolbar.

2. Select a simulation type from among:

 **Show centerline**: A line drawing of the center of the tip of the tool is displayed.

 **2D simulation**: A two-dimensional simulation showing the regions cut by each operation is displayed. The view is changed to the top view automatically.

 **3D simulation**: A 3D solid simulation is displayed.

 **Rapidcut simulation**: In this mode a 3D simulation is performed but the tool is not animated. Only the final result is displayed. For most parts, the simulation takes only a few seconds to complete. Note this type of simulation is only available in FeatureMILL3D.

3. Click the Play button.

Using simulation VCR controls

When you generate toolpaths, they are displayed in the graphics window using the technique (centerline, 2D, etc.) that you selected. You can control the simulation more finely using the VCR buttons in the Simulation toolbar.

 The **Pause** button pauses the simulation. If this button is pressed before generating

toolpaths, the tool moves to its initial position and then pauses. If the Pause button is pressed during a simulation, it pauses the graphics. After the simulation is paused, the three other VCR buttons become active.

The **Stop** button cancels a simulation.

The **Play** button restarts a paused simulation.

The **Single** step button moves the simulation ahead one tool move. The keyboard accelerator for this button is *ALT+F3*.

The **Next** operation button continues to simulate until the next operation. This button is actually a fly-out menu. By clicking on the triangle to the right of the button the following additional options are revealed:

The **Next rapid** button simulates until the next rapid tool move.

The **Next tool** change button simulates until the next tool change.

The **Erase** button erases any centerline toolpaths on the screen.

The **Region of interest** button limits the portion of part that is simulated.

The **Show tool load** button indicates whether or not to display a graph of the tool load when the next 3D Simulation is performed.

The **Eject** button removes the Simulation toolbar from the screen and erases the simulation from the graphics window.

To adjust the speed of a centerline, 2D or 3D simulation, use the slider on the right-hand side of the controls. Slide to the right to speed up, and move to the left to slow the simulation down. The slider of the simulation toolbar also affects the display for rapidcut simulation. If the slider is all the way to the right, only the final simulation result is displayed. Position the slider bar further to the left to see intermediate results.

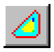
Pausing a toolpath simulation

A simulation can be paused using any of the following techniques:

1. Using the Pause button in the simulation toolbar.
2. Setting a break point.
3. Using the Single step Next operation Next rapid or Next tool change buttons in the simulation toolbar

Interaction between viewing and simulation

Show centerline: Uses the current view to draw the toolpaths over whatever is currently displayed. If Keep toolpaths with view change is checked, you can dynamicall view the part and the toolpaths. If Keep toolpaths with view change is not checked, the toolpaths are erased when the view is changed.   .

 **2D simulation**: The view is changed to the top view automatically and anything on the screen is temporarily erased until the simulation is complete. The view cannot be changed during the simulation. Once the simulation is complete you can change the view, but the toolpaths are erased.

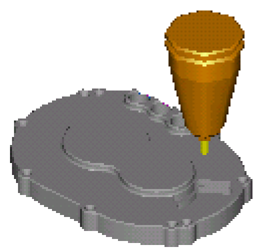
 **3D simulation**: The simulation is performed in the current view. You can interactively view the simulation at the conclusion of the simulation or after pausing it. The simulation does not have to be recalculated, so the view change is instantaneous. After changing the view, you can continue the simulation with the play button.

 **Rapidcut simulation**: This simulation can be dynamically viewed. For zooming, the image must be recalculated, but this calculation time is quick. For panning or rotating, the image does not have to be recalculated, but a lower resolution version is displayed while you are transforming the part. This allows you to more interactively change the view.

Region of interest

The region of interest  button in the simulation toolbar allows you to limit the portion of the part that is rendered during simulation. There are two primary reasons to do this:

1. The rest of the part is not simulated and it is therefore faster to simulate your region.
2. If you are using rapidcut, the region is simulated at a finer resolution.

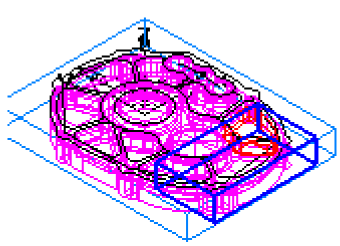


There are three different types of regions

Stock: This is the default type of simulation where the entire stock is rendered during simulation

Feature: Select the name of the feature in the drop down list and a region around the feature will be selected as the region.

XYZ Location: The region is defined by either a box you drag or by two points whose coordinates you can enter. Note that the sides of the box are aligned with the X and Y-axes.



Rapidcut simulation

This mode is available only in FeatureMILL3D. Rapidcut simulation does not apply to turned parts. In this mode the tool is not animated, but rather the results of the simulation are directly displayed. For most parts, the simulation takes only a few seconds to complete.

To enable this simulation mode, click the  button.

If a break point is set, the result of the simulation is performed up to the breakpoint. If no breakpoint is set, then the entire NC program is simulated.   The slider of the simulation toolbar also affects the display. If the slider is all the way to the right, only the final simulation result is displayed.   Position the slider bar further to the left to see intermediate results.

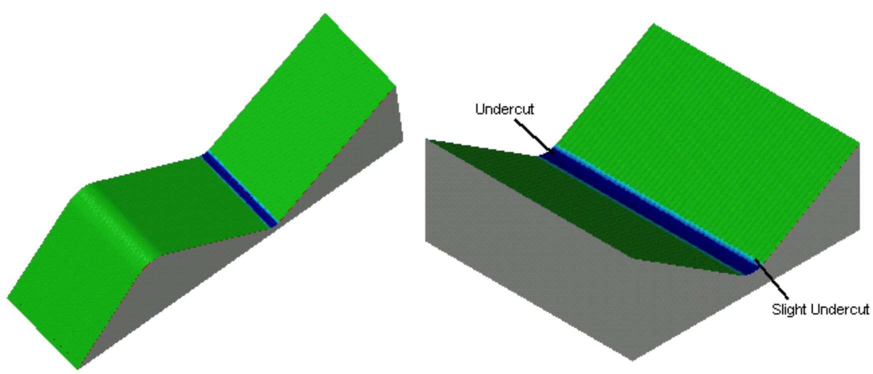
Note that rapidcut simulation does not detect insufficient cutter length or tool holder gouges.

Once you have run a rapidcut simulation you can then run a part compare to see how the toolpaths you generated compare to the final desired shape.

Part Compare

Part compare is a feature of rapidcut simulation that allows you to compare the results of a toolpath simulation with the actual part model. Regions that are properly cut are displayed in green. Regions where extra material remains are shown in light or dark blue. Gouged regions are shown in yellow or red.

In the example you can see that the trough is undercut. Examining the trough more closely, you can see that the bottom of the trough is greatly undercut. This is indicated by the dark blue color. The light blue regions indicate a slightly undercut region.



To run part compare:

1. Set the desired Part compare rest material > tolerance.

2. Run a rapidcut simulation.

3. Select *Simulation* from the *View* menu and then select *Show part compare*.

To view the actual part model that the simulation is compared with:

1. Select *Simulation* from the *View* menu and then select *Show target part*.

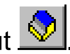
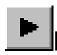
If you adjust the view after a part compare has been performed, a new rapidcut simulation is automatically performed, but you must run another part comparison.

Part compare also works with the region of interest.

Mixing 3D simulation and rapidcut

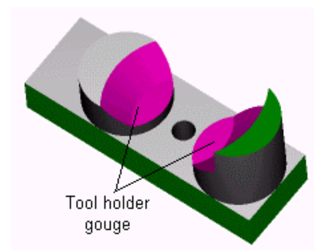
You can switch between 3D simulation and rapidcut anytime the simulation is paused.

To switch between these two simulation modes:

1. Begin the simulation.
2. Pause the simulation.
3. Change simulation types to either 3D solid or Rapidcut .
4. Click the Play button to restart the simulation.

By using this technique, you can use rapidcut to fast forward to a point and then use 3D simulation to more carefully study a particular set of toolpaths.

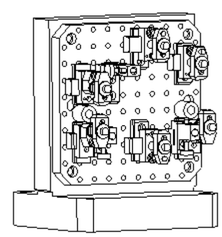
Detecting gouges



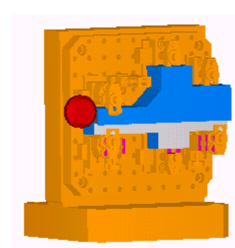
Using 3D simulation, you can visually detect gouges with the tool (during rapid moves), the tool holder or the spindle. Any gouge is displayed in pink. If you want the simulation to pause when it detects a gouge, set the *Pause on gouge* simulation option. If this option is set, the simulation will automatically pause if a gouge is detected. Click the Play button in the simulation toolbar to continue the simulation

Fixture and clamp collision detection

Normally, only the part model is rendered during a 3D simulation. If *Pause on gouge* is set, the simulation will stop if the tool rapids into the part. In FeatureMILL3D, 3D simulation can help you detect collisions with models of clamps or fixtures that you create as solid models. To detect these collisions:



1. Create a solid model of a fixture. It can be as elaborate as you need it to be.
2. Right-click on the solid in the graphics widow. Select *Use solid as clamp* from the pop-up menu.
3. If you want the simulation to alert you to gouges, select Simulation from the Options menu. Click on the 2D/3D simulation tab and check *Pause on gouge*.



4. Run a 3D simulation. During the simulation both the part and the clamp model are rendered. It may take longer for the simulation to start up because the clamp model is being rendered.

Remember, FeatureCAM does not come with a library of clamps, but the 3D solid modeling capabilities of FeatureMILL3D allows you to create these models from scratch. You can also import clamp models for other solid modeling systems.

If a clamp is defined in a .fm part file, this clamp will be displayed during 3D simulation in a tombstone document.

Simulating 3D toolpaths

Due to the large number of moves in the toolpaths for a typical 3D part, animating the entire toolpath using 3D solid simulation is often time consuming. Instead, consider one of the following techniques:

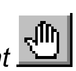
1. Mix 3D solid and rapidcut simulation.
2. Use centerline simulation and pause the simulation periodically and erase the displayed toolpaths to clear the screen.
3. Use rapidcut and use the Next operation  button to quickly see the results of each operation.
4. Use rapidcut and slow down the speed control so that you see some intermediate results.
5. Use a region of interest in combination with either 3D solid simulation or rapidcut.

Setting simulation breakpoints

Clicking on the Op list tab in the Manufacturing Feedback window brings up a table of operations.

Each row displays the operation, feature that the operation came from and the tool that will be used to cut the feature. As the toolpaths are simulated, the current operation is indicated with a yellow arrow in the left-hand margin. You can also set break points in the simulation. A break point will pause the simulation at a particular operation. You can then use the VCR controls to control the simulation.

To set a breakpoint:

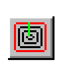
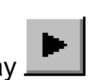
1. Click on the operation that you would like to pause on.
2. Click the *Breakpoint*  button. A maroon dot will appear in the left-hand margin. The simulation will now pause when it reaches this operation.

To remove a breakpoint:

1. Click on the operation that has a break point set on it. (It will have a maroon dot in the left-hand margin.)
2. Click the *Remove breakpoint*  button.

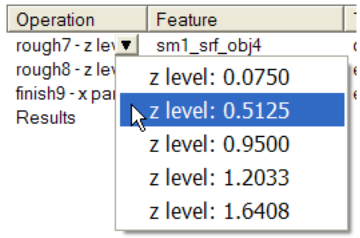
Display a single Z level

To display a single Z level of a Z rough or Z finish operation:

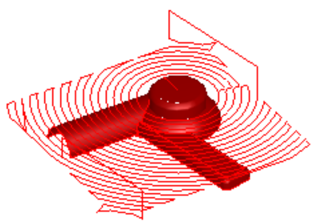
1. Click the centerline simulation  button and click the play  button to simulate all of the toolpaths for your part.

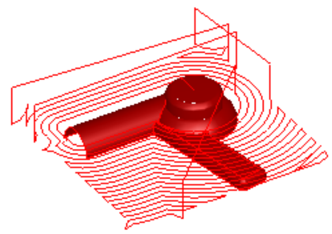
2. Click the *Erase*  button.

3. Click on a Z level operation in the op list tab. All of the toolpaths of that operation will be displayed and a drop down menu is available under the operation name.



4. Click a Z level from the drop down menu and the toolpaths from that level will be displayed.





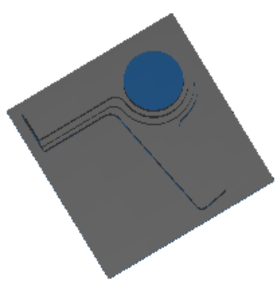
Viewing intermediate shaded simulations

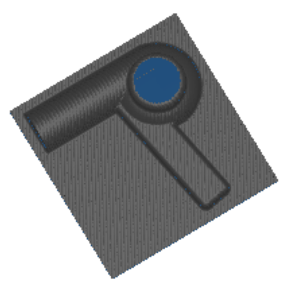
To view intermediate shaded simulations:

1. Click the 3D shaded  button or the rapid cut simulation  button and click the play  button.

2. Click on an operation to display the machined model up to that point in the process.

3. At the end of the list of operations is the word, Result. Click on this word to view the final simulation image for your part.





Note the you must turn on the 2D/3D shaded simulation option *Save result files during rapid cut* for this procedure to work for rapid cut simulations.

View toolpaths for a single setup

When toolpaths are generated, they are generated for all setups, but only the toolpaths for the current setup is displayed on the screen.

To view the toolpaths for a particular setup:

1. Select the name of the setup in the *Part view*.
2. Run any simulation mode you wish.

Simulating multiple setups

1. Arrange the view of your part so you can observe the manufacturing involved in all stages as you can't change the view after you start the simulation.
2. Click the Toolpaths step  in the Steps Toolbox to display the Simulation toolbar.
3. Click on the *Part View*. It is located at the bottom of the Steps toolbox.
4. Click on the first setup in the Part view.
5. Click *3D simulation*  from the Simulation toolbar and click Play.
6. Click on the second setup in the Part view.
7. Click Play again.
8. Repeat steps 6 and 7 until you have simulated your entire part.

Previewing toolpaths for one feature or operation

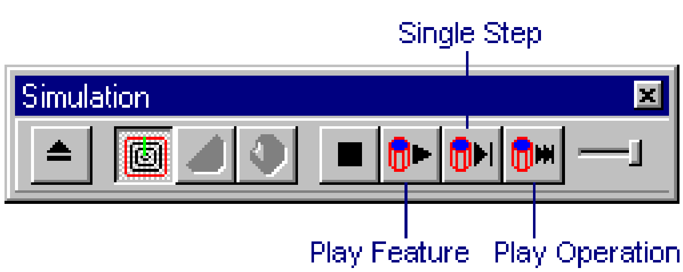
Simulations are normally performed on all features in a setup. To simulate a single feature or operation:

1 Double-click on a feature.

2 The Properties dialog box comes up.

3 Select the feature or an operation in the tree view.

4 Press the *Preview* button. The Properties dialog box is hidden. A special version of the Simulation toolbar is displayed.

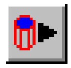


5 Click the *Play Feature* button.

You can also use the Op List to simulate a single operation.

To perform a centerline simulation of a single operation:

1. Click on the operation that you would like to simulate in the Op List window.

2. Click the *Preview toolpaths*  button. The simulation toolbar is displayed.

Skip toolpath on error

If Skip Toolpath On Error is turned on in the *Options* menu, no toolpaths will be generated if an error is encountered when generating toolpaths. If this flag is turned off, toolpaths will be generated for all valid features and features that caused the toolpath error will be skipped.

Simulation options

The behavior of 2D and 3D simulations is controlled by the settings in the *Simulation Options* dialog box, available in the Options menu. You can open this dialog box while a simulation is running and refine or coarsen the simulation on the fly. If you do open Simulation Options during a simulation, the simulation pauses until you close the dialog box. You'll get an error message if you run the 3D simulation in a display showing fewer than 256 colors. The simulation will still run, but won't look as good.

General Options

White background

White Background sets the background color for shaded simulation. If set, then the background of the shaded graphics is white. If not set, the background is black. To change the background color for line drawing, use Coloring in the View menu to set the Background.

Show holder

Show holder toggles the display of the tool holder during 2D and 3D solid simulation.

Simulation speed

Control the speed of the simulation with the simulation speed slider. This slider controls the speed of 2D and 3D solid simulation.

Show turn chuck

Show turn chuck displays a disk representing the chuck during 3D simulation.

Tool colors

This simulation option simulates the cuts of each tool with a different color. This allows you to graphically view which portions of the part are cut with which tool. If after you run a 3D simulation, you may toggle this setting without rerunning the simulation.

For centerline simulation this option will display each tool in a different color during the simulation.

Status options

The Status section of the *Simulation Options* dialog box shows the information is displayed in the Status bar during simulation. Check boxes show which information in the status bar.

Time turns on the display of the machining time estimate.

Feed displays the feed rate for the operation currently being simulated.

Speed displays the spindle speed rate for the operation currently being simulated.

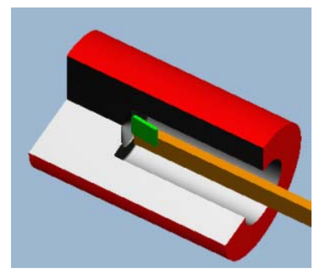
Operation displays the name of the operation currently being simulated.

Tool displays the name of the tool performing the current operation.

Position shows the X, Y and Z positions of the tool on the screen. If you are using indexing, the angle you are rotated around the axis is also shown. Note this option can slow the simulation.

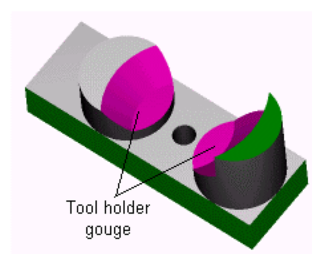
3D Shaded

¾ view with lathe ID work

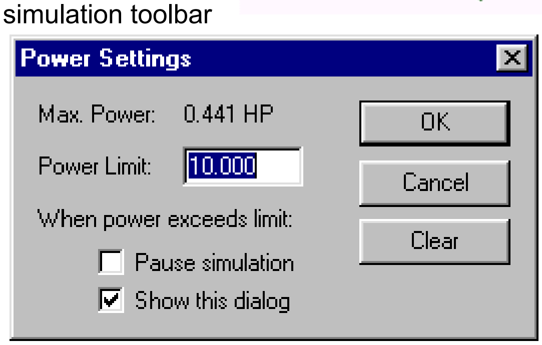


This option enables a ¾ cut-away view for ID work for turned setups. This figure shows an example of the ¾ view:

Pause on gouge



Using 3D simulation, you can visually detect gouges with the tool (during rapid moves), the tool holder or the spindle. Any gouge is displayed in pink. If you want the simulation to pause when it detects a gouge, set the *Pause on gouge* simulation option. If this option is set, the simulation will automatically pause if a gouge is detected. Click the Play button in the simulation toolbar to continue the simulation.



Show pause on gouge dialog

This option turns on the display of a dialog box to warn you that a gouge has taken place. If this option is unchecked, then the simulation just pauses without displaying a dialog box when a gouge is detected.

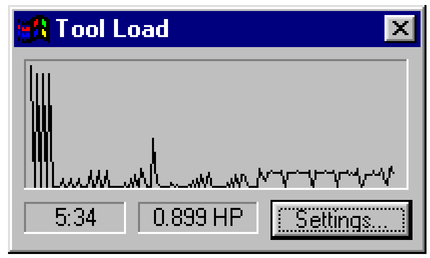
Power graph samples/min

The tool load graph is determined by measuring the tool load a certain number of times per minute. The *power graph samples/min* is the number if samples to take per simulation minute

Rotate view when indexing

This setting applies to 4^{th} or 5^{th} axis indexed parts and turn/mill parts. With *Rotate view when indexing* checked, the simulation will rotate the part for an A-axis or B-axis indexing move in milling or for a C-axis rotation in turn/mill. While these rotations provide for a more accurate simulation, they can slow down the simulation especially with simultaneous X and C-axis moves in turn/mill. To speed up the simulation, uncheck this setting and the part will stay fixed and the tool will move around the part.

Tool load



3D toolpath simulation can be used to estimate and graph horsepower requirements. These estimates can be used to fine-tune your program for maximum performance. By clicking the *Show tool load*  button on the simulation toolbar, the *Tool Load* dialog box is displayed when a 3D simulation is performed. This dialog box graphs the horsepower requirements of the part program and displays the current simulation time and instantaneous horsepower.

Clicking the *Settings* button displays the *Power Settings* dialog box.

Power limit sets the top power value that is displayed in the tool load dialog box.

Max Power displays the maximum horsepower value required by the program so far. If you display this dialog box at the conclusion of a simulation, you can view the maximum horsepower required for the entire program.

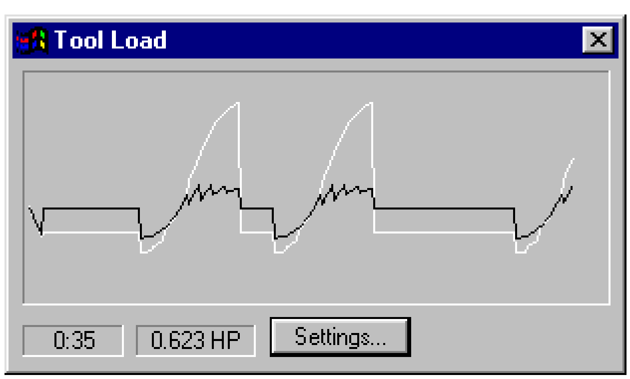
Pause simulation will pause the 3D simulation every time that the power limit is exceeded.

Show this dialog will display the *Power Settings* dialog when the power limit is exceeded.

Clear will clear the graph when the simulation it resumes.

Note that the horsepower values are estimates. If you are approaching the power limits of your machine, you should lower your feed rates, or decrease the width or depth of your cuts.

If you have run a feed optimization, the tool load graph shows the current loads in black and the previous (unoptimized) feedrates in white.



Preview only

3D solid simulation has two modes. With *Preview only* turned on, memory usage is minimized, but it takes longer to rotate, translate or scale the simulation. With *Preview only* turned off, FeatureCAM creates an internal model of the simulation that allows you to change the view of the simulation faster, but memory usage is higher.

Resolution

Resolution controls the roughness or quality of the image and affects the speed of the simulation. Set it to a positive value. The default value is 1. If you double the value (by setting it to 2.0), the tool is subtracted from the block half as often. If you decrease the value, the tool is subtracted from the block more often. If the simulation is too chunky, decrease Resolution by half. If the simulation quality is acceptable, but it is running too slowly, double the Resolution or increase the Simulation Speed.

Translucent part

Check this checkbox to change the stock of a 3D solid simulation to be slightly transparent.

Translucent tool

Check this checkbox to change the tool of a 3D solid simulation to be slightly transparent.

Tool cutting tolerance

This tolerance affects the fineness of the 3D simulation. If this is set high, the simulation will appear more faceted. The smaller this tolerance is set, the smoother the simulation will appear.

Tool visual tolerance

This tolerance affects the appearance of the tool. By increasing this tolerance, the tool appears chunkier. By decreasing the tolerance the tool will look rounder and smoother, but the simulation will take up more memory and may be slower.

Warning for display that is using less than 256 colors

Check this checkbox to enable a warning if you are using a display with less than 256 colors.

2D Solid

Steps % of normal

Steps % of Normal is a value between 0 and 200 that changes the animation step size for 2D Simulation. Set it to less than 100 to decrease the step size and to more than 100 to increase it.

White Background

White Background sets the background color for shaded simulation. If set, the color of the background of the shaded graphics is white. If not set, the background is black. To change the background color for line drawing, use Coloring in the View menu to set the Background.

Show turn chuck

Show turn chuck displays a disk representing the chuck during 3D simulation.

Centerline

Show centerline prior to 2D/3D simulation

If the 2D or 3D simulation button is pressed without performing a centerline simulation first, the toolpaths must be generated. If *Show centerline prior to 2D/3D simulation* is checked, a centerline simulation is shown while the toolpaths are being generated and then the 2D or 3D simulation is then performed. If it is not checked, then the initial toolpaths are generated without display and then the 2D or 3D simulation is performed.

Keep toolpaths with view change

Keep toolpaths with view change, leaves the toolpaths on the screen while you change the view of the part. Without this setting, the toolpaths are erased when the view changes. Note that saving the toolpaths for interactive viewing requires extra computer memory (RAM).

Show tool animating

Check *Show tool animating* to view the tool as a line drawing as the centerline toolpaths are displayed. This setting must be checked to see the toolpaths in an incremental manner. If *Show tool animating* is not checked, the toolpaths for the entire part are shown all at once at the conclusion of the calculation.

Smooth animation

Smooth animation *controls the display of the tool during centerline simulation. With Smooth animation* turned on, the tool will be displayed without flickering during the simulation. With this setting unchecked, the simulation will require less memory, but will flicker some during the simulation.

At maximum animation speed, only update display every

Only update display every controls the display of the centerline toolpath as it is calculated. This number controls how often the toolpaths are displayed. This number is specified in minutes of tool travel, i.e. if the feed rate is 100 inches per minute and *Only update display every* is set to 0.5 minutes, then the screen will be updated after the tool has moved 50 inches. A zero setting causes the screen to be updated for every block of NC code. If the speed control slider is not set to the maximum value the *Only update display every* setting to be ignored.

In previous versions of FeatureCAM, this was called *Toolpath update*.

Part compare options

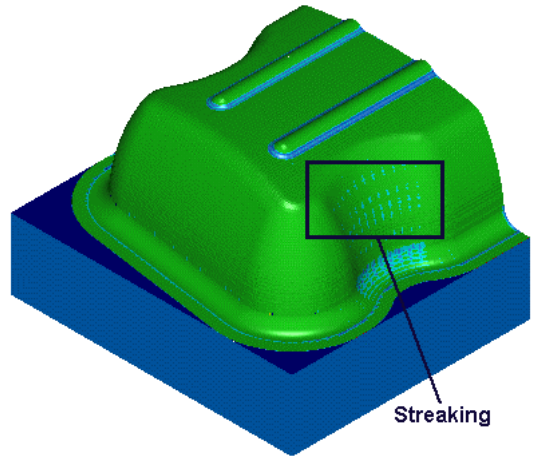
Regions that have extra material greater or equal to the *Show rest material* parameter this amount are shown in blue. The more rest material there is, the darker the shade of blue.

Regions with gouges greater than or equal to the *Show gouge* parameter are shown in yellow. Larger gouges are shown in red.

Regions with rest material less than the *Show rest material* parameter or gouges less than the *Show gouge* number are shown in green. These areas are considered correctly cut.

Note, if either the *Show rest material* number or the *Show gouge* number is set too small, then the part comparison will be very noisy and difficult to interpret.

Target part tessellation tolerance



Rapidcut works by comparing the part cut by the toolpaths with the actual surfaces of your part, known at the target part. The target part is approximated by triangles for the comparison. If this tolerance is too coarse, then the part compare will report gouges or rest material where there actually is none and the model will have streaks. Decrease this tolerance to remove such streaks.

Rapidcut conversion tolerance

When mixing 3D solid and rapidcut simulations, the images must be converted from one type to the other by FeatureCAM. If you want to improve the quality of the part when it switches representations, decrease this tolerance.

Operation ordering

You create features and FeatureCAM breaks them down into operations for manufacturing. For example, a hole is translated into spot-drill, drill and countersink operations. The individual operations are shown in the Op List tab in the order in which they will be performed. The ordering is either *Automatic*, based on rules, or *Fixed*, based on manual sequencing.

For automatic ordering:

1. Click the *Automatic ordering* radio button.
2. Click the *Ordering* {bmc btn-ordering-options.bmp} button.
3. The *Automatic ordering options* dialog box is displayed. Select the ordering criteria from among:
    - Minimize rapid distance
    - Minimize tool changes
    - Cut higher operations first
    - Do finish cuts last

See page 204 for information on the automatic ordering options.

4. As you select ordering options, the operations are resorted.
5. Click OK.

With automatic ordering, any new operations that are created are automatically sequenced using the ordering options.

For fixed ordering:

1. Click the *Fixed ordering* radio button.
2. Reposition the operations by clicking a row and using the arrow buttons, or click on the operation name and drag the operation to its new position.

If you add new operations with *Fixed ordering* selected, the new operations are inserted at the end of the list.

Using groups to determine manufacturing order

Groups collect features together, but they also provide a powerful level of control over the order of manufacturing operations.

Check *Ordered* in the Group Properties dialog box when you create a group to set manufacturing in the exact ordered specified. If you created an ordered group with a hole and a rectangular pocket, then the operations are performed as follows:

- spot-drill hole
- drill hole
- rough pocket
- finish pocket

You can mix ordered and unordered groups in your setup for more flexibility in ordering operations. For example, perform all drilling operations first and all milling operations last:

1. Make all of your features.
2. Create an *Unordered* group of holes.
3. This indicates no preference about the order for manufacturing the holes.
4. Use Rename from the Edit menu to name the group *holegroup*.
5. Create a second *Unordered* group of your milling features name *millgroup*.
6. Create a third group that contains *holegroup* and *millgroup* in that order.
7. Mark this group as *Ordered* and call this group *allgroup.*

When *allgroup* is manufactured, all the holes are made first, then the milled features are made. Within each group, FeatureCAM optimizes the order so that tool changes are minimized. That means for *holegroup*, all spotdrilling operations are done first, followed by all drilling operations. Then all milling operations are completed. Using groups, you control the order of manufacturing operations without sacrificing the help that FeatureCAM can provide.

Changing your display driver

FeatureCAM has two different modeling procedures, manufacturing simulation and surface shading. Simulation works best with at least 256 simultaneous colors. Shading works best with at least 16000 simultaneous colors.

In fewer colors, the image may appear flat and not accurately convey the information it was designed to provide. In low color modes, FeatureCAM displays a warning message.

Most display adapters can display 256 or more colors but Windows may not be configured for it. Use *Display* in the Control Panel to configure your screen resolution and color display. The Settings tab contains the display size and color depth controls. After changing these settings, you might have to reboot. You can also update your video driver on this tab.

You may need to confirm that your display adapter is actually capable of displaying 256 colors. In the hardware documentation for your computer, look for information regarding the display adapter or graphics card. Details such as the manufacturer, model number, resolution options, number of colors, and display memory available on the display adapter are useful in configuring your display. You may also need to provide the make and model and refresh rates of the monitor you use. If you don't have any of this information, ask your computer vendor for help.

If the shading looks acceptable, then you do not need to change your display driver.

Chapter 12
Controlling Manufacturing

The default behavior for manufacturing is controlled by *default machining attributes*. These attributes can be modified so that the system's default behavior can better represent the practice of your shop. To override these settings for a particular feature, use *feature attributes*. These attributes are set directly on the feature.

Default machining attributes

Select *Machining Attributes* in the *Manufacturing* menu to open the Machining Attributes dialog box. The machining attributes are broken into two different categories, milling and turning.

Milling default attributes

Drilling tab

Attempt chamfer with spotdrill

Default value for the *Attempt chamfer with spotdrill* milling strategy attribute documented on page 225.

Spot drill

Default value for *Spotdrill* checkbox on the Strategy page. See page 228 for details.

Spotdrill diameter

This percentage is used to select a spot drilling tool. A value of 100 specifies that the spotdrill should be the same diameter as the hole. A smaller value will create only a starter hole.

Use L/D compensation

Use L/D Compensation reduces speed and feed for holes that have a ratio of hole depth (L) to hole diameter (D) of greater than 2.5. the greater this ratio, the greater the speed/feed reduction.

Retract to plunge clearance

Retract to plunge clearance is a default machining attribute only for milling. It is set off by default. See page 224 for information on the feature level attribute.

Pilot drill diameters

Pilot drill diameters sets the default list of diameters for pilot drilling. See page 228 for more details.

Bore cycle

Bore Cycle sets the default for the bore cycle drilling attribute documented on page 229.

No drag X shift and No drag Y shift

These attributes set the default values for *No drag X shift* and *No drag Y shift*. See page 230 for further details.

Ream cycle

Ream cycle sets the default for the *Ream cycle* drilling attribute documented on page 229.

Tap cycle

Tap cycle sets the default for the *Tap cycle* drilling attribute. See page 229.

Max tap spindle RPM

Max tap spindle RPM sets the default value for the *Max tap spindle RPM* drilling attribute documented on page 230.

Drill cycle

Drill cycle sets the default value for the *Drill cycle* strategy attribute documented on page 229.

Spot drill edge break

Spot Drill edge break drives the spot drill an extra amount (in terms of depth) such that when the operation is complete an edge break or chamfer is produced by the spot-drill by the amount indicated. Amount is radial distance, 0.005 edge break results in chamfer 0.010 greater than hole size. The angle of the chamfer depends on the spot-drilling tool used.

Dwell

Dwell sets the default for the *Dwell* drilling attribute documented on page 229.

Combine with similar holes into canned cycle

Combine with similar holes into canned cycle sets the default value for the feature level attribute with the same name documented on page 226.

Milling tab

Bi-directional rough

Bi-directional rough sets the default value for the *Bi-directional rough* strategy attribute. See page 208 for more information.

Climb mill

Climb mill sets the default value for the *Climb mill* strategy attribute described on page 206.

Cutter comp

Cutter comp sets the default value for the *Cutter comp* strategy attribute. See page 185 for more information.

Minimum ramp dist%

Minimum ramp dist % is specified as a percentage of the tool diameter and is the default value for the *Minimum ramp dist* milling attribute. See page 219 for further information.

Depth first and Individual rough levels

These checkboxes are the default values for the *Depth first* and *Individual level* feature attributes described on page 207.

Part line program

Part line program sets the default for the *Part line program* strategy attribute documented on page 207.

Reorder

This is the default machining attribute for the *Reorder* feature attribute. See page 401 for an explanation.

Last pass overcut %

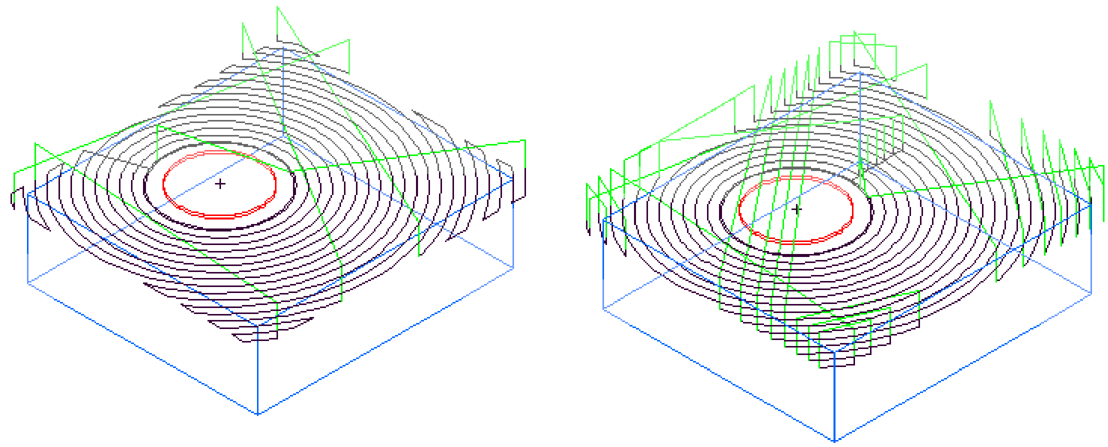
Last pass overcut % is specified as a percentage of the tool diameter and sets the default value for the *Last pass overcut%* milling attribute. See page 220.

Lateral overcut %

Lateral overcut % is specified as a percentage of the tool diameter, and sets the default value for *Last overcut %* milling attribute. See page 220 for further information.

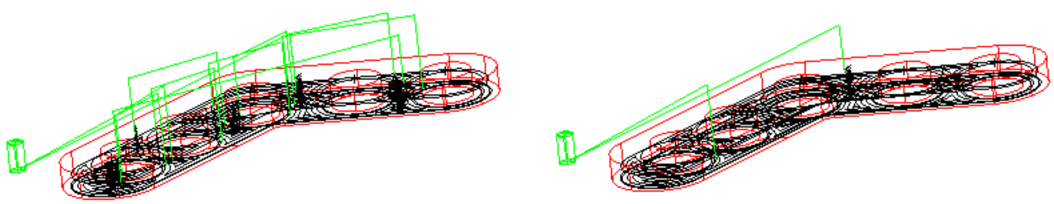
Min rapid dist %

Min rapid dist % is the lower threshold that defines when a tool move is performed as a rapid move, or just a move at the current feed rate. The number is specified as a percentage of tool. The left-hand figure shows a boss cut with a value of 400%. Note that as the tool moves inward, there are few rapid moves. Instead, the tool is fed between passes. In the right-hand figure, *Min rapid dist %* is set to 10% and the tool retracts and rapids between passes.



Minimize tool retract

Minimize tool retract is a milling attribute that reduces that amount of retracting that the tool will perform while milling. Instead of retracting, the tool will continue feeding to its next location. The figure on the left shows normal retracting. The right-hand figure shows the same feature with *Minimize tool retract set.*



This attribute only affects how the tool retracts within a single operation. It does not control how operations are ordered. For this functionality, see *Minimize retract distance* on page 180.

Side roughing bottom up

This attribute is the default value for the bottom up attribute described on page 209.

Side finish bottom up

This attribute is the default value for the bottom up attribute described on page 210

Stepover tab

Do rough pass

Do rough pass sets the default value for the *Rough* checkbox on the strategy tab. See page 208 for more information.

Spiral %

Spiral % is the percentage of tool diameter to use for radial depth of cut for rough milling or finishing the bottom of a milled feature when using the offset method

Zig-zag %

Zig-zag % is the percentage of tool diameter to use for radial depth of cut for rough milling or finishing the bottom of a milled feature when using the zig-zag method.

Zig-zag angle

The default value specified in degrees for the *Zig-zag angle* attribute. See page 119 for more details.

Depth %

Rough depth % is specified as a percentage of tool diameter and sets the default value for *Rough pass Z increment* on the milling tab for rough milling operations. See page 215 for information on this feature attribute.

Finish bottom

Finish bottom sets the default value of the *Finish bottom* strategy checkbox which is documented on page 209.

Facing pass %

Facing pass % is specified as a percentage of the tool diameter and sets the default value of the *Stepover %* milling attribute of a facing operation. See page 220.

Do finish pass

Do finish pass sets the default value for the *Finish* checkbox on the strategy tab. See page 209 for more information.

Finish allowance

Finish allowance sets the default value of the *Finish allowance* milling attribute which is documented on page 218.

Bottom finish allowance

This attribute set the default for the bottom finish allowance attribute described on page 218.

Do semi-finish pass

Do semi-finish pass sets the default value of the *Semi-finish* checkbox on the strategy tab. See page 208 for more information.

Semi-finish allowance

Semi-finish allowance sets the default value for the *Semi-finish allowance* milling attribute. See page 218 for more information.

Finish passes

Finish passes sets the default value of the *Finish passes* milling attribute that is documented on page 218.

Finish overlap

Finish overlap sets the default value for the *Finish overlap* milling attribute. See page 218.

Wall pass

The default value for the *Wall pass* attribute documented on page 210.

Tool selection tab

Tool selection attributes

The **Counter bore** radio buttons control the default behavior for tool selection for counter bore operations. Select *Use counter bore* to default to a counter bore tool or select *Use endmill* to make circular interpolation with an endmill the default. Click *Automatic* to have FeatureCAM first attempt to select a counter bore to and to use an endmill as a secondary choice.

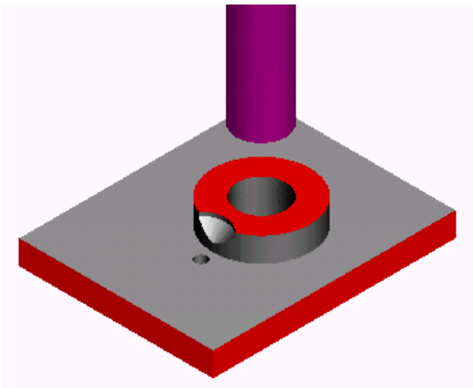
Prefer spot drill use a spot drill tool for spot drill operations if possible.

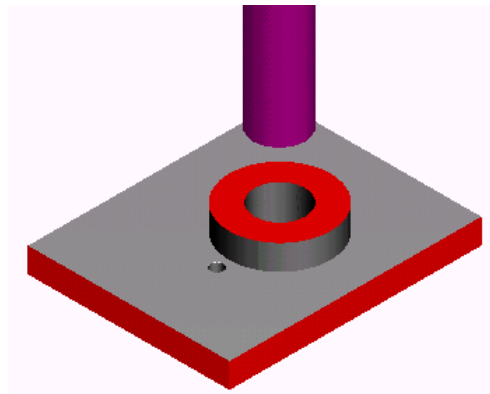
Prefer center drill use a center drill tool for spot drill operations if possible.

Tool diameter tolerance is the tolerance used when selecting a tool for an operation. If the tool diameter is within the tolerance of the desired tool, the tool will be selected.

Drill % of ream/bore is the percentage of the hole diameter to use for selecting tooling for undersize drilling operation. For example, if set to 95, a drilling operation will be created with a diameter that is 95% of the nominal hole diameter.

Optimize spot drill tool selection will automatically spot-drill all holes with the largest spot-drill that would be used for a collection of holes. For example if you have 0.25, 0.375 and 0.5 inch holes, a 0.5 inch spot-drill will be used for all holes. It is possible that you could get a gouge with this setting turned on, if a large spot-drill might gouge neighboring features. The left-hand figure shows a situation that would gouge. The right-hand figure shows the proper result with *Optimize spotdrill tool selection turned* off.





Optimize chamfer tool selection will automatically chamfer all holes with the largest chamfer that would be used for a collection of holes.

Preferred spot drill dia is the diameter of the spot-drill (or center drill) that will be preferred for all holes.

Tool % of arc radius is located on the Misc. tab and controls the size of the tool that FeatureCAM automatically selects. In earlier program versions this attribute was called Default tool %.

If *Tool % of arc radius* is set to 100 then a tool equal to the smallest corner radius is selected for a feature such as a pocket, and the finish tool path for the pocket looks like the toolpath shown above. With *Tool % of arc radius* set to 100 the tool dwells in the corners as it changes direction. This can sometimes nick the part. To avoid this problem, set *Tool % of arc radius* to a slightly smaller number, such as 98.

Multiple roughing tools for milling. FeatureCAM has the option of rouging a 2.5d milling feature with a single tool or using a sequence of tools.

If you use multiple roughing tools, FeatureCAM cuts all the parts of a feature that it is capable of cutting with the larger tool, and cuts only the remaining portions of the feature with the smaller tool. You do not have to manually create these separate regions. FeatureCAM automatically calculates them for you.

Use the Mult. rough diameters feature attribute to request multiple roughing tools for a single feature. Use the default attribute to set this behavior on all 2.5D milling features.

The default behaviors available are:

- **Use a single tool that is automatically selected**. FeatureCAM will select a tool based on the smallest radius of the feature and use that tool to cut the entire feature. If your part has broad corners or you need to minimize the number of tools you are using, this is a good option to pick. If you have features with tight corners, you should not choose this option. If you are using automatic feature recognition, this option can result in a number of tool selection errors for parts with sharp corners.

- **Use Multiple roughing tools**. This option will cut the part with a list of tools you supply. Each tool will cut only in the regions that have not been cut previously. As soon as the part has been cut completely, no more operations are created with the smaller tools. This is a good option to select if your part has sharp corners, but be aware that your part will require more tools and tool changes. You should enter the too diameters you want to use. Separate each diameter with a comma. As part of this option, you can ask FeatureCAM to **automatically select an additional tool that fits the smallest radius of the contour**. If you check this option, you should list some larger diameters for multiple roughing and then let FeatureCAM select the tool for the final roughing pass.

As a separate option, you can limit the smallest tool FeatureCAM will select by checking **the minimum tool diameter of an automatically chosen roughing tool** and specifying a diameter. Since many CAD models have sharp corners, it is recommended that you use this option when working with imported models with feature recognition.

Surface leadin tab

Use this tab to set the default values for the attributes described on page 394

Surface mill tab

Tolerance (3D)

Tolerance sets how close the milling will be to the mathematically ideal surface. This does not guarantee that your feature is machined to this tolerance in all locations if the tool you select is incapable of cutting within that tolerance in constrained areas. If your part shows a faceted appearance, set you tolerance to a lower value.

Z slice tolerance

Sets how finely to compute the intersection of the different Z levels with the surface that is being milled. Use this setting to refine the intersection curves shown in the Z level rough preview before using the simulations. This is a shortcut to get good toolpaths without spending lots of time in simulations.

Parallel angle

Sets the angle used by default for X parallel, or Y parallel milling. Defaults to 0. It can be overridden on individual features as needed.

Tool diameter

Tool diameter sets the default tool diameter for 3D surface milling features.

Tool end radius

Tool end radius is set with radio buttons. Ball end sets the default tool to a ball end tool with a radius equal to half that of the tool diameter setting. Flat sets the default tool to a flat endmill. It can be overridden on individual features as needed.

Scallop height

Sets the default scallop height allowed for surface milling features. Can be overridden on individual features.

Check allowance

Sets the check allowance used by default in Surface milling features. By default, it has no value associated with it. It can be overridden on individual features as needed.

Finish allowance

Finish allowance is the amount of material left after a 3D roughing pass.

Slope limitation angles

These attributes set the default values for the Slope limits tab described on page 391.

Scallop height stopovers

Click this attribute to set the default stepover type for projection milling finishing and Z-level finishing to be specified by scallop height instead of a linear stepover distance.

Operations tab

Base priority

Base priority changes a feature's priority. The features are then sorted by their priority to determine the order in which they are manufactured. The *Priority* attribute on each feature can be used to order the features for manufacturing. Among features with identical priority values, FeatureCAM will use minimization of tool changes and other criteria to determine a manufacturing order. The Base Priority attribute is the initial priority that each feature will be assigned by default. Setting the *Base Priority* default machining attribute establishes the priority value for each feature. If you explicitly set a feature's priority, it overrides any priority value that the feature had previously.

Cut higher operations first

This attribute only affects milling setups. *Cut Higher Operations First* Set this checkbox to mill the features from the top of the stock first and work toward the bottom. If you disable this attribute, you should carefully graphically verify the toolpath before cutting your part.

Do finish cuts last

Do Finish Cuts Last affects the order in which operations are manufactured. If you want to perform all rough milling operations before finish milling operations, check the Do Finish Cuts Last attribute.

Do not ask at tool path generation

Do not ask at tool path generation is a toggle for whether you are prompted with the dialog box when you run a simulation. The Ordering dialog box settings override the operation defaults set on this tab.

Minimize rapid distance

This attribute only affects milling setups. *Minimize Rapid Distance* moves to the next closest feature that uses the same tool as the last operation. This checkbox must be off if you want to generate hole macros in the NC code

Minimize tool changes

Minimize Tool Changes groups operations together that use the same tool. This saves time for you by eliminating or reducing needless tool changes. This checkbox must be set if you want to generate hole macros in the NC code.

Cost estimation attributes

These attributes can be customized to fit the behavior of your particular machine. These attributes affect the machining time estimates printed in the operation sheets.

Rapid Traverse is the feed per minute of rapid moves.

Tool Change is the time in seconds it takes to change a tool (not including the rapid to get to tool change location).

Go to Start is the time it takes for the tool head to move to the start location and the spindle or tool head to come to a stop.

X-Y Acceleration is used in the formula below to calculate the time for a particular tool move.

Z Acceleration is used in the formula below to calculate the time for a particular tool move.

Formula for a particular tool move

The formula for the time estimate for a particular tool move is:

time = dist/fpm + fpm * (1 + zdist/dist * (1/z-accln - 1)) / xy-accln

fpm = feed per minute (Rapid Traverse for rapids, feed rate for cutting moves)

xy-accln = X-Y Acceleration

z-accln = Z Acceleration

dist = total distance of tool move

zdist = total distance the tool moves in the Z-direction

Acceleration conversions

If the acceleration rates for your machine are reported in different units, use the following conversions

Current Units	Desired Units	Multiply by
Meter per second squared	Millimeters per minute squared	3,600,000
Feet per second squared	Inches per minute squared	43,200

Inches per second squared	Inches per minute squared	3,600

Assumptions used:

Most often, because of change of direction (and/or several other factors), the tool must effectively accelerate to final fpm from a stop. So the acceleration is calculated as if the tool head must always accelerate to final fpm from 0.

Also, most mills take longer to accelerate in the Z-direction than in either X or Y (and that X and Y acceleration are equal).

Misc. tab

Z rapid plane

Z rapid plane is the minimum safe distance in Z above your part. Before performing a rapid move in X or Y the tool tip is retracted to this height. This value is relative to the top of your stock in the current user coordinate system.

Min rapid distance

Min rapid distance is the lower threshold that defines when a tool move is performed by a rapid move, or just a move at the current feed rate. The number is specified as a percentage of tool's diameter.

Maximum spindle RPM

Maximum spindle RPM sets the default for the *Max. spindle RPM* strategy attribute documented on page 225.

Coolant

Coolant sets the default value of the *Coolant* strategy attribute documented on page 224.

Speed

Speed is the default value for the *Speed override%* attribute on the misc. tab. See page 225 for more information.

Feed

Feed is the default value for the *Feed override%* attribute on the misc. tab. See page 225 for more information.

Plunge feed override %

Plunge feed override % sets the default value for the *Plunge feed override %* attribute on the misc. tab. See page 225 for further details.

Proportional plunge feed

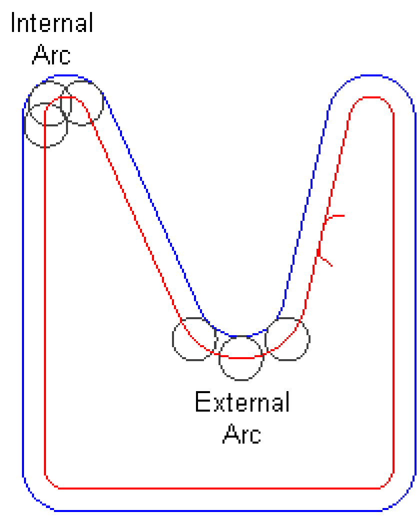
If *proportional plunge feed* is set, the feed rate of the ramping move is scaled based on the *Max ramp angle* attribute. A ramp angle of 1 degree will set the feed rate of the plunging moves to approximately the milling feedrate. An angle of 90 sets the feed rate of the plunging moves to the value determined by Plunge feed override %. If *Proportional plunge feed* is not set, then the feed rate of plunging moves are determined by *Plunge feed override %* regardless of the ramp angle.

Spline tolerance

Spline Tolerance parameter sets the default for the *Spline tolerance* attribute on the misc. tab. For more details on this attribute see page 225.

Peripheral feed dialog box

These parameters allow you to adjust the feedrates of arc moves for 2 ½D milling features. The concept is that by slowing the feedrate on internal arcs and increasing the feedrate on external arcs, you get a more consistent finish.



Display this dialog box by clicking on the *Peripheral feed …* button in the Misc. tab of the Machining Attributes.

Click the *Decrease feed on internal arcs* checkbox to slow down on concave moves. Set the lower limit by entering a percentage of the linear feed.

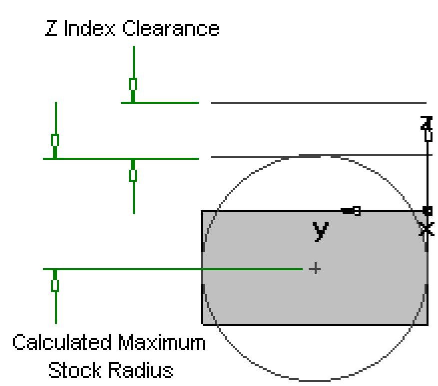
Click the *Increase feed on external arcs checkbox* to speed up on convex moves. Set the upper limit by entering a percentage of the linear feed.

There is a section for roughing and finishing and the options are identical. The finishing section applies to semi-finishing and finishing 2 ½ D milling operations. The roughing section applies to 2 ½ D roughing operations.

It is recommended to use these adjustments for finishing and leave the settings for roughing unchecked. Use feed optimization for roughing instead. In earlier versions of FeatureCAM, this functionality was called *corner feedrate reduction* and only allowed you to slow down concave corners.

Post variables

On the Turning or Milling tab of each FeatureCAM feature is a Post Vars. Button. This button brings up the Post Variables dialog box that contains 9 separate variables that are passed directly to the post processor. You can use these variables to pass strings directly to the post processor.



Retract to plunge clearance

Retract to plunge clearance can lessen wasted time by using a lower Z clearance. By default, FeatureCAM retracts the tool to the Z Clearance Plane between operations. While this is a safe assumption, it can result in inefficient NC part programs. Instead, set the *Retract to Plunge Clearance* manufacturing attribute. See page 224 for further details.

Z index clearance

In 4th axis positioning and 5th axis positioning the tool must retract to a safe distance so that it will not collide with the part while it is indexing. To achieve this, FeatureCAM calculcates the maximum stock radius and adds to that the *Z index clearance* to determine the appropriate retract distance.

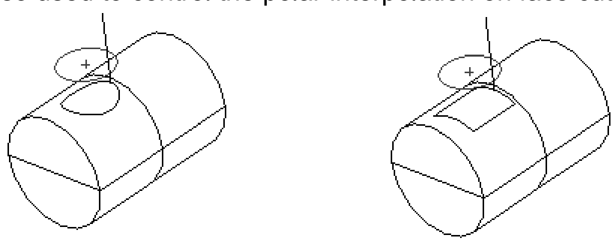
Chamfer depth

The default value for the feature attribute described on page 222.

Wrap tolerance

Arcs that are wrapped must be converted to small 3D line segments. The *Wrap tolerance* is used to determine the acceptable distance between the line segments and the initial arc. The figure on the right shows a lower tolerance for a wrapped circular groove. The left-hand figure shows a large tolerance. In this case the circle is approximate by a square.

This attribute is also used to control the polar interpolation on face cuts for turn/milling. If FeatureCAM is performing the polar interpolation, any linear move or off center arc move on the face of the part must be interpolated by linear moves and rotations about the C axis. Wrap tolerance is used to control the fineness of this linear interpolation. The figure on the left shows a straight face cut with loose wrap tolerance. Notice the dent in the middle of the cut. The right-hand figure shows the same cut with a tighter tolerance.



Lead/ramp tab

The lead/ramp parameters are the same as the equivalent feature attributes, except that Lead distance% and Ramp diameter% are specified as a percentage of the tool diameter. See page 222 for descriptions of these attributes.

Helical ramping

Helical ramping sets the default value for the *Helical ramping* milling attribute documented on page 211.

Max. ramp angle

Max. ramp angle sets the default value for the *Max ramping angle* milling attribute documented on page 211.

Helix linear approx tol

Helix linear approx. tol sets the default value for the *Helix linear approx tol* milling attribute. See page 211 for more information.

Thread mill tab

Linear ramp dist

Linear ramp dist. sets the default value of the *Linear ramp dist.* milling attribute documented on page 221.

Ramp diameter %

Ramp diameter%. sets the default value of the *Ramp diameter%* milling attribute. See page 221 for more information.

Ramp angle offset

Ramp angle offset. sets the default value of the *Ramp angle offset* milling attribute documented on page 221.

Taper approx angle

Taper approx. angle. sets the default value of the *Taper approx. angle* milling attribute documented on page 221.

Helical ramping

This milling attribute sets the ramping into thread mill features to a helical ramp. See page 195 for more information.

Starts

Starts sets the default value for the *Start Threads* milling attribute documented on page 221.

Start angle

Start angle sets the default value for the *Start angle* milling attribute. See page 221 for details.

Tooth outside

Tooth outside sets the default value for the *Tooth outside* milling attribute documented on page 196.

Tooth overlap

Tooth overlap specifies the default setting of the *Tooth overlap* milling attribute described on page 221

Feed Dir

Feed dir sets the default value for the *Feed dir* strategy attribute documented on page 209.

Cutter comp

Cutter comp sets the default value of the cutter comp strategy attribute for thread mill features. Cutter comp is described on page 207.

Pecking tab

Pecking overview

Pecking applies to Deep Hole, Chip Break and Tap operations. FeatureCAM supports four styles of pecking. These styles are listed in the post processor. Three different attributes control the pecking and they are used differently depending on the style of pecking. FeatureCAM checks the pecking type in the currently loaded post processor to duplicate canned cycles when simulating toolpaths.

Fixed Steps
The NC code specifies one depth (*First peck*) and all the steps peck at that depth.   *Second peck* and *Minimum peck* have no effect in this case.

Two Steps
The NC code specifies two depths. The first step pecks at the first depth (*First peck*) and all the subsequent steps peck at the second depth (*Second peck*). *Minimum peck* has no effect in this case.

Value Reduction
The NC code specifies the first depth depth (*First peck*), a reducing value (*First peck - Second peck*). and a minimum depth (*Minimum peck*). The first step pecks at the first depth. Each subsequent step is reduced by the reducing value until the minimum depth is reached.

Factor Reduction
The NC code specifies the first depth (*First peck*), a reducing factor (*Second peck/First peck* and a minimum depth (*Minimum peck*). The first step pecks at the first depth. Each subsequent step is reduced by the reducing factor until the minimum depth is reached.

Drilling first peck

This is the depth of the first peck of a drilling operation specified as a percentage of tool diameter. If the depth of the hole is less than *First peck* the hole will be drilled in a single peck. This attribute sets the default value of the *First peck* attribute of a drilling operation.

Drilling second peck

This is the depth of the second peck of a drilling or operation specified as a percentage of tool diameter. If your control uses a value reduction method where subsequent pecks are reduced by a fixed amount or a factor reduction method where subsequent pecks are reduced by a percentage, still specify the full depth of the second peck. The post will handle the conversion. This attribute sets the default value of the *Second peck* attribute of a drilling operation.

Drilling minimum peck

This is the minimum step size for a peck used for value reduction pecking methods or factor reduction pecking methods. This attribute sets the default value of the *Minimum peck* attribute of a drilling operation.

Tapping first peck

This is the depth of the first peck of a tapping operation specified as a percentage of tool diameter. If the depth of the hole is less than *First peck* the hole will be drilled in a single peck. This attribute sets the default value of the *First peck* attribute of a tap operation.

Tapping second peck

This is the depth of the second peck of a tapping operation specified as a percentage of tool diameter. If your control uses a value reduction method where subsequent pecks are reduced by a fixed amount or a factor reduction method where subsequent pecks are reduced by a percentage, still specify the full depth of the second peck. The post will handle the conversion. This attribute sets the default value of the *Second peck* attribute of a tap operation. This attribute sets the default value of the *Minimum peck* attribute of a tap operation.

Tapping minimum peck

This is the minimum step size for a peck used for value reduction pecking methods or factor reduction pecking methods.

Turning default machining attributes

Drilling tab (turning)

See page 183 for a complete listing of drilling default machining attributes.

Pecking tab (turning)

See page 197 for a complete list of pecking default machining attributes.

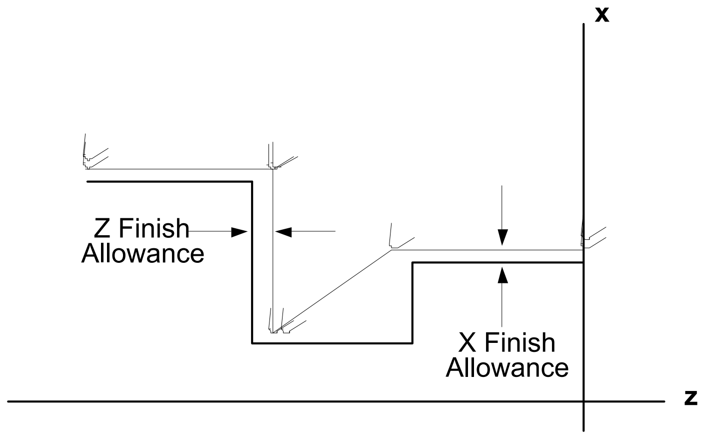
Turning tab

Auto round

This parameter is set either On or Off. When turned On, the system automatically inserts arc moves to connect two non-tangent elements. This results in a minimum of wasted motion by the machine; however, the posted part program may be slightly longer in the number of blocks used.

Rough depth %

Rough depth % is the percentage of tool diameter to use for axial depth of cut for rough milling.



X finish allow

This parameter allows you to specify a separate finish allowance in the X axis direction.

Z finish allow

This parameter allows you to specify a separate finish allowance in the Z axis direction.

Engage angle

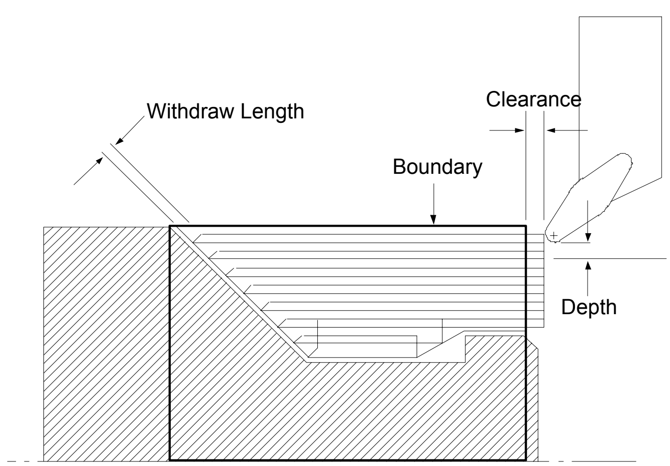
The Engage angle controls the approach of Turn and Bore features. The engage angle is measured away from the part. An angle of 0 will approach along the path. An angle of 90 will approach perpendicular to the path.

Rough and finish withdraw angle

These parameters set the default values for withdraw angle parameters documented on page 234.

Withdraw length

This parameter is the distance along the withdraw angle line in which the tool withdraws before returning for the next step.



Tool nose radius compensation

Tool nose radius compensation ignores the tool radius when generating semi-finishing and finishing passes for turn, bore and facing features. The actual part geometry is output as the toolpath. It is assumed that the operator at the machine tool will perform the tool radius compensation when Tool Nose Radius Compensation is activated.

This option is turned on in the Strategy page of Turn, Bore and Face features.

Use canned cycle

This attribute is located on the Strategy page of Turn or Bore features. If it is checked, then these operations are performed using canned cycles. You must use a post that has support for roughing and finishing canned cycles. Note that the post processor you are using must support canned cycles.

Reuse profile in canned cycle

This attribute is used in conjunction with Use canned cycle . If *Reuse profile in canned cycle* is checked, then the curve is output to the NC file once and then referenced in both the rough and finish canned cycles.

Threading tab

Rough turn and finish turn

Check these boxes to automatically turn the piece down to the thread diameter.

Relief groove

A thread feature has an option of cutting a relief groove at the end of the thread. Click the Relief Groove checkbox on the Strategy page and enter the groove parameters.

Side wall angle

This is the default angle of the relief groove of the thread.

Groove addl. depth

The depth of the relief groove is the depth of the thread plus the *Groove addl. depth*.   This prevents the threading tool from dragging on the bottom of the groove.

Start clearance

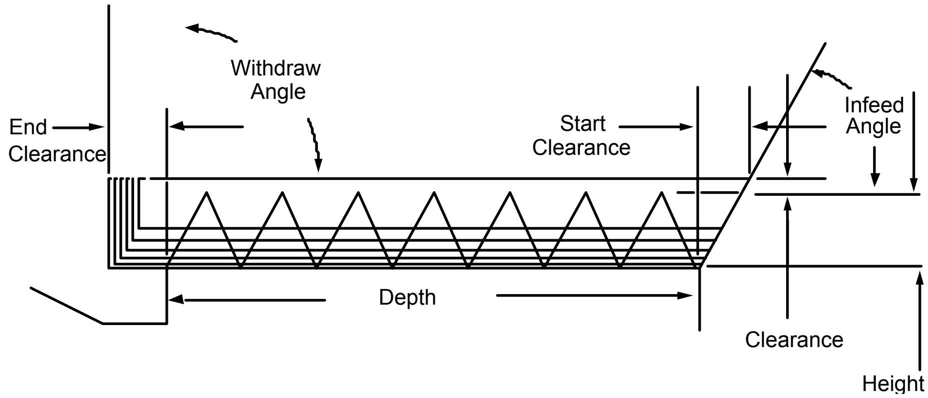
The start clearance value is the position to which the tool traverses before engaging into the work piece.

End clearance

This parameter controls the distance that the tool feeds past the end of the thread (into the relief groove) before retracting from the part's surface.

Infeed angle

The infeed angle is specified as an unsigned, incremental value from the positive Z-axis.



Withdraw angle

This parameter specifies the angle between cross feed movement and the withdraw move. This angle is measured away from the part against the direction the tool was traveling. An angle of 180 will continue in the direction the tool was traveling. An angle of 0 will withdraw back along the path. An angle of 90 will with draw perpendicular to the direction of travel.

Depth %

The depth of a thread is based on the thread pitch. The thread depth is calculated as Depth% * pitch.

Step 1

This parameter is used to specify the incremental step for the first pass across the thread.

Step 2

This parameter specifies the second pass and is used by the system to determine subsequent passes on the thread, reducing in depth until the minimum infeed value is reached.

Minimum infeed

This parameter is only applicable to Thread cycle and is only accessible when the Number of Steps parameter has been set to Calculate. This parameter specifies the minimum infeed distance. The system automatically reduces the infeed distance for each pass after the second step, until the minimum infeed distance is reached or the tool reaches full depth.

Spring passes

A spring pass is a duplicate of the final threading pass. *Spring Passes* indicates the number of spring passes that are to occur at the completion of the thread.

Grooving tab

Dwell

Dwell defines the time that the tool will dwell at the bottom of the groove or cutoff feature.

Stepover %

This parameter is expressed as a percentage of the tool's width. It is the distance by which the tool shifts to position itself for the next plunge cut.

Side lift off dist

This parameter sets the distance by which the tool moves at the end of a plunge cut. The shift is in the direction opposite to the cutting direction. *Side Lift Off Dist* is used to avoid tool contact with the uncut material when the tool is retracting at a rapid feed rate. Side Lift Off Dist is ignored for the retract move at the end of the first plunge. The actual lift off move is performed at the plunge feed rate. If the groove is a round-bottomed groove, then lift off is not utilized, even when specified.

Depth of cut

The Depth of cut parameter specifies a step increment for each pass that the roughing routine performs on the part.

Chamfer extend dist

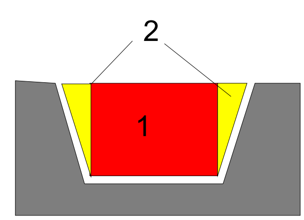
Chamfer Extend Dist provides extra space for the tool so that the tool does not start on the metal for the groove finish pass.

Peck retract dist

For cutoff and groove features, Peck Retract Dist is the distance the tool retracts between plunges.

Plunge center first

If *Plunge Center First* is checked, the straight portion of the groove is roughed first and then the angled portions are roughed separately. If Plunge center first is set, the region 1 of this figure is roughed first and then the region 2 is roughed.



Feed dir

The direction the tool will feed. The choices are either *Neg Z* (-Z direction) or *Pos Z* (+Z direction).

Barfeed tab

Dwell

Dwell defines the time that the tool will dwell at the end of a barfeed operation.

Feed

Feed is the feedrate of the barfeed/barpull tool.

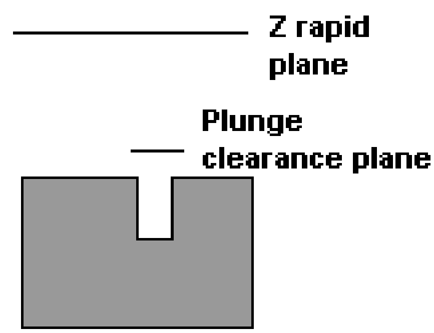
Speed

Speed is the turning velocity in RPMs. This is normally 0.

Misc. tab

Plunge clearance

Plunge clearance is the distance above a feature at which the tool starts to feed. In the case of deep hole drilling, the drill will retract to this distance between pecks. The relationship with the rapid Z plane is shown in the figure.



Coolant

Coolant indicates the type of coolant to use for a feature. The choices are:

- Flood - a continuous stream of coolant
- Mist - coolant mixed with air
- None - no coolant

Turret

Sets the default to either the primary or secondary turret for tooling.

Turret location

The turret location is either Front or Back indicating that the lathe's turret is in front or in back of the spindle. For Front turrets, tools operate below the spindle axis. For Back turrets, tools operate above the spindle axis. This setting will limit the tools that are displayed in the Tools page to only tools appropriate for the machine's turret location.

Turret direction

Turret direction is best left as *Automatic* so that FeatureTURN can calculate the direction for a particular operation. It can also be set explicitly to CW (clockwise) or CCW (counter-clockwise).

Max CSS RPM

Max CSS RPM is the top RPM value for constant surface speed turning.

Tool programming point

Tool tip center: Select this default option if you want to adjust the tool touch-off points by the insert radius compensation at the machine. In this case set the tool's programming point to (0,0).

Tool tip edge: Select this default option if you want to adjust the tool programming points in FeatureCAM by the insert radius compensation. In this case adjust the tool's programming point by the radius compensation.

Turret location

The turret location is either Front or Back indicating that the lathe's turret is in front or in back of the spindle. For Front turrets, tools operate below the spindle axis. For Back turrets, tools operate above the spindle axis. This setting will limit the tools that are displayed in the Tools page to only tools appropriate for the machine's turret location.

Spindle RPM override %

Spindle RPM override % scales the system-generated speed rates for a particular operation, enter the percentage change as the Spindle RPM Override % attribute. A value of 100 leaves the speed rate unchanged. A value of less than 100 will reduce the speed rate, and a value of greater than 100 increases the speed.

Remachining

Automatically sets the boundaries for subsequent operations that use the same curve. This minimizes air cutting and Works between turn features, bore features and between holes and bore features.

Starting offset number for shared tool slots

The *Starting offset number for shared tool slots* is the first length offset register to use for tools that share the same tool slot.

Ordering tab

Do finish cuts last

Do Finish Cuts Last affects the order in which operations are manufactured. If you want to perform all rough milling operations before finish milling operations, check the Do Finish Cuts Last attribute.

Minimize rapid distance

This attribute only affects milling setups. *Minimize Rapid Distance* moves to the next closest feature that uses the same tool as the last operation. This checkbox must be off if you want to generate hole macros in the NC code.

Use operation template

This attribute is applicable only to turning setups. If *Use operation template* is checked then the order of operations is determined by the outline of operations listed Feature order dialog box.

Turn operation order

The Turn operation order option of the Manufacturing menu displays the general outline that is used for ordering turning operations if the Use operation template default ordering attribute is set. The dialog box that comes up lists the types of operations that are created in a general turned part. The order of the list determines the order that the operations in your part will be cut. If your part does not have a particular operation in it, that operation is skipped.

By selecting an operation type and using the arrow keys, you can change the order in which the features are manufactured. After rearranging the list, click OK. This order will be remembered for all of your parts.

Cost estimation attributes

See page 192 for complete details.

Machining Configurations

A configuration is a collection of machining attributes. The defaults for values such as for stepovers, ramping, canned cycle use, or operation ordering are all stored as default machining attributes. Default machining attributes are stored in collections called *configurations*.

This dialog box lists all of the available configurations. All configurations with the document symbol, , in the left-hand margin represent open files.   The other names are configurations that are independent of a particular file.

Note that merely selecting a configuration from the Available configurations list does not have any affect unless you then select one of the actions below.

New

Creates a new machining configuration that is independent of any part file. You are prompted for the name of the new configuration and the configuration to copy the initial values from.

Rename

Rename allows you to rename a configuration that is independent of a part file.

Copy

Allows you to copy attributes from one configuration to another. Select the configuration you want to copy to from the Available configurations list before clicking the Copy button. You are then prompted for the configuration to copy from.

Delete

Delete allows you to remove a machining configuration. You can only remove configurations that are not associated with an opened file. You are not allowed to delete the last configuration that is independent of opened files.

Import

Allows a configuration file with a *cdb* extension to be imported into FeatureCAM. Note that these attributes are not applied to a file unless they are copied or unless these attributes are used as an initial configuration for a document.

Export

Allows configurations to be exported to a .cdb file. A dialog box is displayed in which you can select the configurations to export and to specify the name of the file to be exported.

Edit

Allows configurations to be exported to a .cdb file. A dialog box is displayed in which you can select the configurations to export and to specify the name of the file to be exported.

The drop-down list at the bottom of the dialog box is used to specify the configuration that will be used as the initial configuration for new documents. New documents will copy the attributes from this configuration when the document is created.

Changes in default attribute handling in version 9

1. In version 9, the default attributes are now called *Machining attributes.*
2. In version 9, the machining attributes are saved with the FeatureCAM file. This means that when this file is opened in another copy of FeatureCAM, it will behave the same. A FeatureCAM file is now completely self-contained.
3. Each FeatureCAM file has its own attributes, but the initial values for these attributes when the file is opened are set by an *initial configuration*.
4. Configurations can still be imported or exported through .cdb files, but this is only necessary if you wish to share a configuration with another person without sending the entire FeatureCAM file.
5. If machining attributes are modified in the *Machining Attributes* dialog box, it only affects the current document's use of that attribute.
6. Machining attributes can still be saved independent of a file.
7. It is still possible to change machining attributes that are independent of a file, but they must be edited through the machining configuration dialog box.
8. The attributes that are in effect for a file is named the same as the file.
9. You can simulate the behavior of default attributes prior to version 9.

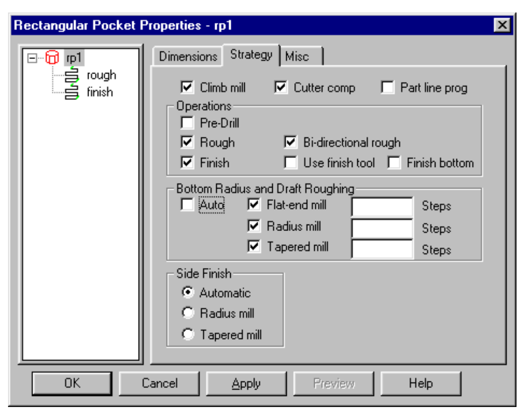
How to use default attributes in version 9 the same way you did in previous versions.

Default attributes are now called Machining attributes.

If you want to use a set of Machining attributes to initialize the default attributes for a FeatureCAM file:

1. Open your FeatureCAM file.
2. Select *Machining Configurations* from the *Manufacturing* menu.
3. Select the name of the file you are working on in the *Available configurations*.
4. Click the *Copy* button.
5. Select the configuration you want to copy from in the drop down list.
6. Click *OK* twice.

Milling feature attributes



To override default system settings you change attributes on specific features. If you open the properties dialog box for a feature you will see a dimensions tab and strategy tab. The strategy tab contains attributes for determining what operations will be generated for a feature.

Strategy tab attributes

Climb mill

Climb mill determines whether the tool is on the left side of the machined edge (in the direction of tool travel). If it is off, conventional milling occurs and the tool is on the right side of the machined edge.

Cutter comp

Cutter comp turns on cutter compensation for the finish pass of a milled feature. Cutter compensation is a feature of a machine control that will offset the lines and arcs of a toolpath to account for the tool's actual diameter. The direction of the compensation depends on the value of *Climb mill*. If *Climb mill* is on, the cutter compensation direction is left, and right if off.

If you use cutter compensation, you must enable this capability in the *Post Options* dialog box. Cutter compensation commands are output for finish passes with the Cutter Comp. attribute set. If the *Cutter Comp* check box is not checked in the *Post Options* dialog box, then cutter compensation is disabled for the entire part regardless of the value of the *Cutter Comp*. attributes on each feature.

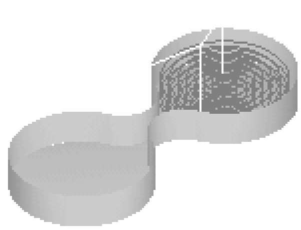
Part line program

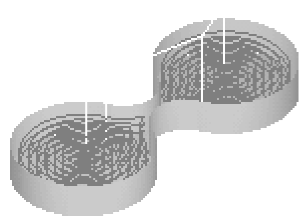
Part line program is a sub-mode of cutter compensation that applies to the finish pass of milled features. If enabled, the actual drawing dimensions of the feature are output as the toolpath rather than the center-line of the tool. The tool selected to cut the feature is still important even when using part line programming. If the same tool is used for roughing, be sure not to deviate far from the diameter of the selected tool to ensure proper area coverage for the roughing passes. Also ensure that the diameter of the selected finishing tool is small enough to cut your entire feature. If you have selected a tool too large to fit into a tight corner, you cannot correct the toolpath with just cutter compensation.

FeatureCAM automatically calculates the entrance point of your finish pass and adds a linear move and a ramping move (based on the *Ramp diameter* attribute) to your finish pass to accommodate cutter compensation. If you receive a warning in the operations list such as "Can't find ramp in/out arc" or "Can't extend end of open profile" then correct the problem by decreasing the *Ramp diameter* attribute or changing the *Pre-drill point*.

Depth first

Checking the *Depth first* option will cut each region of the feature completely before moving on to another region. The toolpaths descend in Z. If this option is unchecked then all regions of a feature are cut at one Z level before descending to a deeper Z-level. Note that this attribute has no effect if the toolpaths for your feature do not rapid between regions of the feature. The figures below show how regions of a feature are completely cut before moving on to another region.

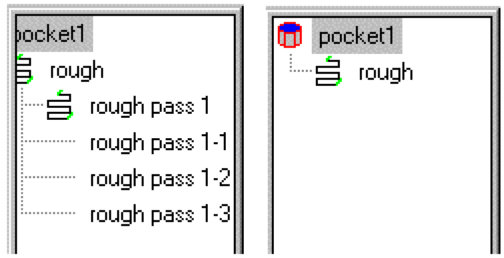




If you are using multiple roughing tools or multiple finishing tools, to efficiently rough out tight corners, *Depth first* is also useful. The corners are milled from top to bottom completely before the tool moves to another corner.

Individual levels

Many roughing cuts are performed at multiple Z levels due to the depth of the feature. If you check *Individual levels* on the Strategy page, you will be able to customize the manufacturing attributes of each level. If you have a feature that is roughed in four levels then the tree view for that part is shown on the left. Note that each pass is listed underneath the *rough* operation. By clicking on any of the passes, you can set attributes in the milling tab. If *Individual levels* is not checked, then only the rough pass is listed in the tree view and you can only make changes to milling tab attributes that will apply to all levels. The figure on the right shows the tree view for the same feature with *Individual levels* unchecked.



Checking the *Individual levels* checkbox also triggers selecting the shortest tool available in the current tool crib for each level. For example, if cutting a 1 inch deep pocket with 0.25 inch corner radii with a z increment of 0.25 inches the following tools would be selected:

Individual levels off			**Individual levels on**		
Operation	Tool Name	Cutter Length	Operation	Tool Name	Cutter Length
Rough	Endmill0375:high+	1.5	Rough pass 1	Endmill0375:reg	0.56
			Rough pass 1-1	Endmill0375:reg	0.56
			Rough pass 1-2	Endmill0375:high	0.75
			Rough pass 1-3	Endmill0375:high +	1.5

Pre-drill

Pre-drill adds or deletes a predrill operation to a feature's manufacturing plan.

Rough

Rough adds or deletes the roughing operation to a feature's manufacturing plan.

Bi-directional rough

Bi-directional rough mills in both directions. If it is Off, conventional roughing happens and the cutting path always moves in one direction with rapid, above stock return movements to set up for the next pass. *Climb mill* controls the cutting direction.

Semi-finish

Semi-finish toggles a semi-finishing operation in a feature's manufacturing plan. Cutter compensation can be applied to this operation. This operation helps to ensure a consistent width of cut for the finish pass. If the finish pass is cut at multiple z depths, a semi-finish pass is also cut as each Z depth.

Finish

Finish toggles the finishing operation to a feature's process plan.

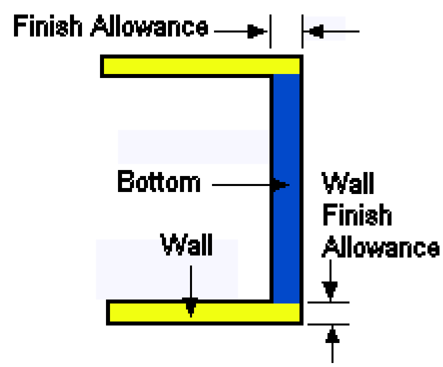
Use finish tool

If *Use finish tool* is not checked, the same tool is automatically selected for both the rough and finish passes.   If *Use finish tool* is checked, FeatureCAM creates a new tool for finishing. This finishing tool is identical to the tool that was selected for roughing. The string, "-finish" is appended to the name of the roughing tool. For example if the roughing tool is named "endmill1.0", the finishing tool is called "endmill1.0-finish". This finishing tool is not permanently assigned to a tool crib. Instead it is a temporary tool for use in this part only. Using *Use finish tool* assumes that you want to use a separate tool for finishing, but that tool is identical to the roughing tool.

If you want to use different types of tools for roughing and finishing, like different length tools or tools with a different number of flutes, it is best to uncheck *Use finish tool* and explicitly change the tool to use for finishing.

Finish bottom

Finish bottom finishes the bottom of a feature with a flat endmill up to the beginning of any bottom radius if present.



Finish walls

If this attribute is checked, then the finish allowance is left on the walls of the groove. Checking this attribute also enables the Wall pass option.

Feed dir

This thread milling attribute sets the direction of the cuts. A negative setting will move the tool down the thread mill feature. A positive setting will move the tool up the feature. Note that Climb/Convention, CW/CCW distinctions are automatically calculated and reported next to these settings.

Bottom radius and draft roughing

Auto automates the roughing of features with a bottom radius or a draft angle. FeatureCAM automatically chooses tools, and number of steps to manufacture the part. If you want specific control, turn off the *Auto* check box and the other fields become active.

- Enter a number of Steps to use (in the Z direction) or leave the Steps field blank to use the *Draft flat scallop height* attribute on the operation to control the number of steps.
- *Radius-mill*, when checked, creates a *draft radius* operation which will rough the bottom radius and possibly the walls with a ball-end or bull nose mill. Enter a number of Steps to use (in the Z direction) or leave the Steps field blank to use the *Draft radius scallop height* attribute on the operation to control the number of steps.
- *Tapered mill* is only available for tapered features. When *Tapered mill* is checked, a *draft taper* operation is created which will rough the walls with a tapered endmill. Enter a number of Steps to use (in the Z direction).

If the *Bottom up* checkbox is checked, the bottom radius or tapered region is roughed from the bottom up, otherwise the roughing is performed from the top down.

All of these operations are discussed on page 122.

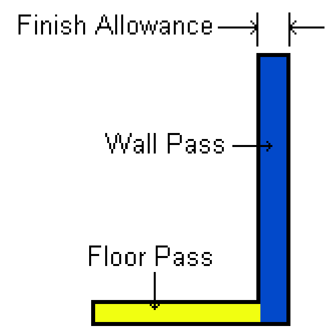
Side finish

These settings affect how FeatureCAM performs finishing of a draft angle feature including bottom radius features.

- *Automatic*, when set, FeatureCAM decides how to finish the side wall and/or bottom radius of the feature.
- *Radius mill* creates a *Finish pass* operation that finishes the wall and bottom radius with a radius tool. This could be either a bull nose or ball end tool. The *Radius tool scallop height* attribute controls the scallop height of the *Finish pass* operation.
- *Tapered mill* creates a *Finish taper* operation that finishes the side wall up to the bottom radius, if any, with a tapered mill that matches the features defined taper. If no tool matches the draft angle, an error message is generated to warn you of that fact.

If the *Bottom up* checkbox is checked, the bottom radius or tapered region is finished from the bottom up, otherwise the roughing is performed from the top down.

Wall pass

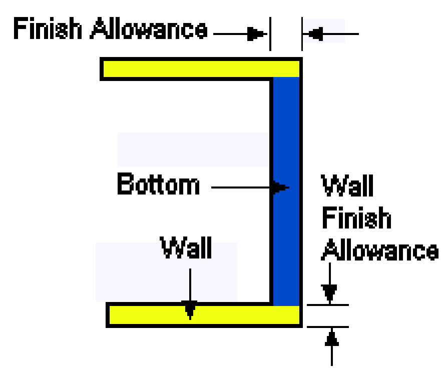


Wall pass only applies to milling features where the bottom is finished. If wall pass is on, then the bottom will be finished up to the finish allowance on the wall. The walls are then finished in a separate pass.

If wall pass is off, then the floor will be finished all the way out to the wall in a single pass. The wall is not finished separately.

For OD/ID grooves if this attribute is checked, then the bottom is finished separately from the walls of the groove.

Wall finish allowance

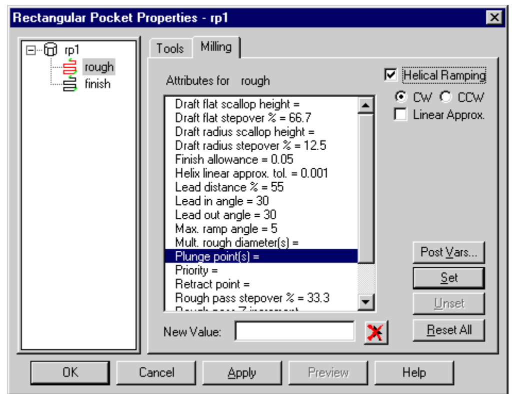


The amount of material to leave on the walls of an ID/OD groove.

Milling tab attributes

If you display the Properties dialog box for a feature (by double-clicking in a feature), and then click on an operation in the tree view, tabs that are specific to the operation will be displayed.

Milling tab controls



The milling tab for milled features looks like this figure.

New Value is the place for changing an attribute's value.

To change an attribute:

1. Double click the feature to open the feature's Properties dialog box.
2. Click the appropriate tab in the dialog box.
3. Set the check box or select the attribute.

- For parameters with a check box, click the check box to toggle the attribute's value.
- For parameters with a pull down list (indicated by an arrow to the right of the value), click the arrow, then select the new value.
- For numeric attributes, click the name. The current value appears in the New Value line at the bottom of the dialog box.

4. Type in the new value.
5. Click Set or press Enter.

Once a parameter has been changed from the system supplied default, an * appears next to the name to indicate that the value has been altered.

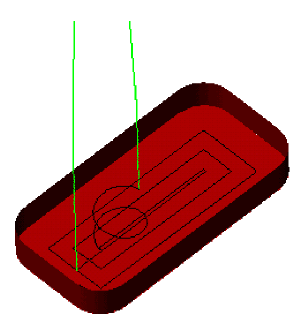
Set applies the *New Value* on the attribute that you have selected.

Unset sets the selected attribute back to the system-supplied default.

Reset All sets the default value for all attributes listed on that page.

Ramping attributes

Helical ramping



This milling attribute sets the ramping into milled features to a helical ramp. The angle of the ramp is controlled by the *Max. ramp angle* attribute. Click CW for a clock-wise ramp or CCW for a counter-clock-wise ramp. If Helical ramping is not set then multiple zigzag passes are used ease into the material. If Linear Approx is set, then the arc moves are approximated by linear moves. . The diameter of the helix (or the length of each linear move if helical ramping is off) is controlled by Max ramp distance.

Helical and zigzag ramping restrictions

FeatureCAM tries to automatically determine locations for ramping into the part using the following criteria:

1. The ramping move should not gouge.
2. For zigzag ramping, the XY distance of each linear move must be at least one tool diameter for non-center cutting tools. Center cutting tools only require an XY move of 20% of the tool diameter.
3. For helical ramping, the same restrictions apply mentioned above, except that the distance applies to each 360° helical move.

If you ask for ramping and do not receive the ramping move set a plunge point or pre-drill the entry point.

Note that helical ramping only applies to the offset style of milling, not zig-zag milling.

Helix linear approx tol

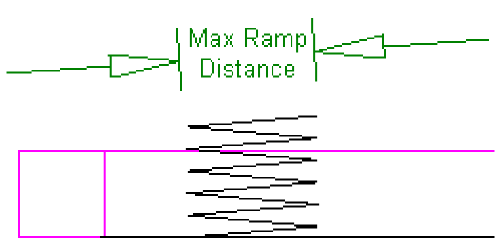
This attribute is associated with helical ramping and Thread mill features. If *Linear Approx* is set, then the arc moves are approximated by linear moves. *Helix linear approx tol* controls how accurate the approximation is relative to the theoretical spiral. Set this tolerance to a smaller number to more accurately approximate the spiral.

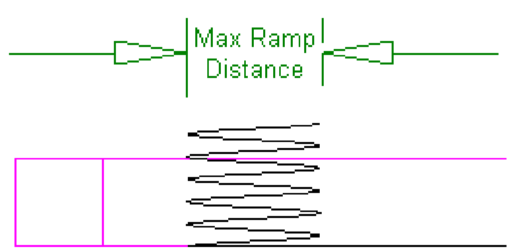
Max ramp angle

Max ramp angle is the maximum usable slope (in degrees) for ramping down to depth. It applies to helical ramping or linear ramping. FeatureCAM won't exceed whatever you set this to and may use lesser slopes for ramping cuts. Default maximum ramp angle is 30. Setting this value to 0 causes a plunge cut.

Max ramp distance

Max ramp distance applies to linear or helical ramping. For linear ramping it is the distance for each linear move. For helical ramping it is the diameter of the helix. If this attribute is not set, then the default distance is the tool's diameter. If ramping at this distance would cause a gouge, the distance is reduced.

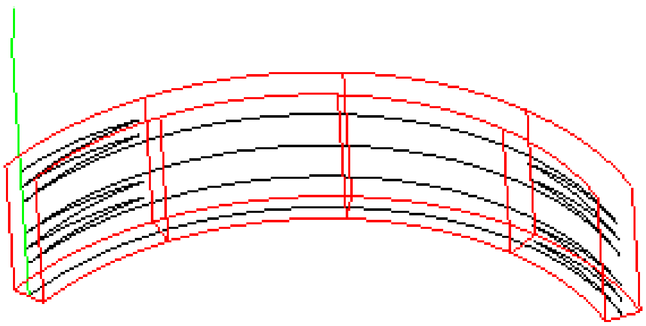




Zig-zag ramping

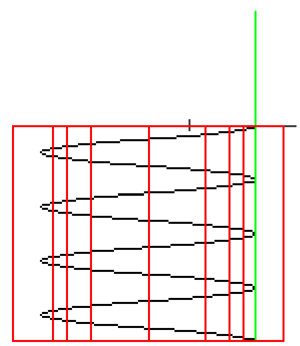
Zig-zag ramping occurs when the *Helical ramping* milling attribute is unchecked. Zig-zag ramping typically moves in linear segments. The length of these segments is controlled by the *Max ramp distance* attribute. The slope of the linear moves are controlled by the *Max ramp angle* attribute. If a plunge point is specified, zig-zag ramping is still available, but the distance of the ramping moves is calculated automatically. FeatureCAM determines the starting point for milling the feature and the tool zig-zags between the plunge point and the automatically calculated start point.

For simple grooves or for plunge points that are located in narrow regions of a feature, straight, linear zig-zag ramping may not be possible since these moves would gouge the feature. Instead, the tool will zig-zag along a 3D arc or a combination 3D arcs and lines that would follow the shape of the feature. In this case, 3D arc moves are output in the NC code. There is currently no way to approximate these moves with 3D line segments. The *Linear approx*. parameter only applies to helical ramping



Using zig-zag ramping to mill a helical path for a simple groove

Zig-zag ramping can be used with a simple groove feature to create a generalized helical toolpath.



To produce this toolpath:

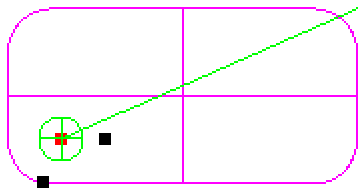
1. Set *Max ramp angle* to 90.
2. Set *Max ramp distance* to a very large number like 1000.
3. Set *Rough pass Z increment* to a value less than the depth of the feature. In one pass around the groove, the tool will spiral down in Z an amount equal to *Rough pass Z increment*.

An alternative way to create the generalized helical toolpath is to:

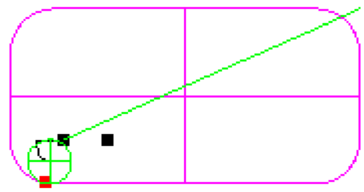
1. Set *Max ramp distance* to 1000.
2. Set *Rough pass Z increment* to the depth of the feature.
3. Use the *Max ramp angle* to control the slope of the toolpath.

Start point(s)

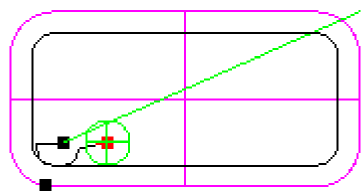
Use *Start point* to explicitly set where you would like the tool to contact a 2.5D milling feature on the finishing pass. If this attribute is not set the start point is automatically calculated.   If the feature has only one boundary curve, enter a single point.   If your feature has two boundary curves use a line or linear curve to specify the start points of each curve. If your feature has more than two curves enter a linear curve with a point indicating the start point for each curve. See the figure below for how start points interact with plunge points and retract points.



Plunge Point
If the tool retracts after the roughing pass, it will plunge at the plunge point. If the tool feeds to the start of the finish pass, it goes straight to the start point.



Start Point
This point is the contact point of the tool with finish pass. The tool ramps onto the feature at this point.



Retract Point
After ramping off the part, the tool moves to the retract point before withdrawing from the feature.

Pre-drill points and plunge points

Pre-drill points sets explicit pre-drill and plunge points. Plunge points just set the plunge location(s) for an operation. Use the *Curve start/reverse* to discover and set where the curve starts. If you pick a plunge point that does not gouge, then the start point will be the nearest point on the curve to the plunge point. To explicitly enter these points follow these steps:

- For a single point, when you highlight the attribute, you can enter a coordinate point, or select a point with the mouse
- For two points
    1. Create a line whose endpoints describe two plunge points
    2. Chain it into an open curve
    3. Enter the name of the curve as the setting for Pre-drill Points attribute on the feature
- For three or more points
    1. Create a polyline whose endpoints describe plunge points
    2. Chain it into an open curve
    3. Enter the name of the curve as the setting for Pre-drill Points attribute on the feature
    4. Milling proceeds from whichever plunge point is closest

This attribute can also control the starting point of a Profile Side feature. To change the starting point, create a curve containing a single point and enter the name of the curve as the Pre-drill Points attribute on the Profile Side feature. In this case you do not have to enter a value for the Pre-drill Diameter attribute.

Note that plunge points only apply to the offset style of milling, not zig-zag milling.

Using plunge points to control the start point of a finish milling pass

A plunge point is one in which the tool makes a downward movement in Z. The user has control over these by using the "plunge point" attributes in his roughing and finishing passes. A common point of confusion is when the user tries to set a plunge point for his finish pass. If you look at the toolpaths carefully, you'll notice that in many cases no "plunge" occurs between the roughing and finishing passes -- the tool simply moves via a feed move from the roughing pass and into the finish pass. So the technicality comes into play, and the plunge point is not used. Unfortunately, we have no way to tell FeatureCAM where to start the finishing pass in this case. But you'll notice that if you turn off the roughing pass and the finish bottom, then the finish plunge point will be honored. So a workaround would be to break up your feature into to separate features: one for roughing and one for finishing.

Plunge points are ignored if they would cause a gouge

A plunge point will be ignored if it results in a gouge to the part. You will see a warning in your Operations sheet when the program ignores a plunge point. For example if you specify a plunge point that is closer to a wall than the tool radius, then the plunge point will be ignored. There is also a more subtle variation of this rule that may come into play: if the user's chosen plunge point gouges on its way to the starting point of the toolpath, then here too the plunge point will be ignored.

Pre-drill diameter

Pre-drill diameter determines the diameter of the hole used to pre-drill the plunge points of pockets and bosses. Make sure the diameter is large enough to allow the milling tool to enter the stock. FeatureCAM automatically selects the plunge points for you unless you explicitly set the *Pre-drill points* attribute.

Z ramp clearance

The Z ramp clearance is the distance above the feature where a zig-zag or helical ramping move will begin. It is bound by the plunge clearanceattribute.

Roughing attributes

Rough pass stepover %

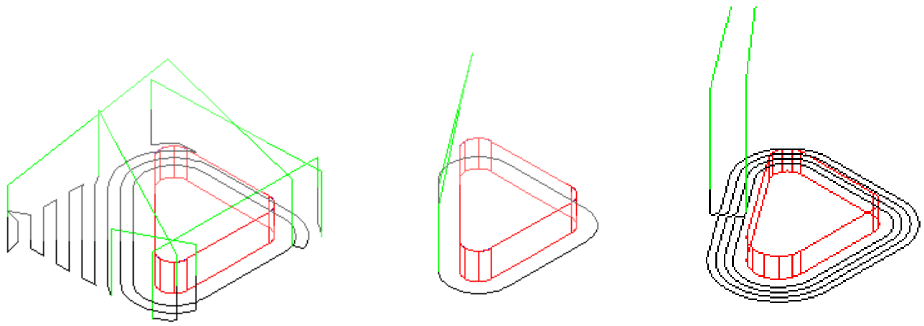
Rough pass stepover % is the percentage of tool diameter to use for radial depth of cut for rough milling.

Rough pass Z increment

Rough pass Z increment is the attribute for setting the roughing depth of cut. This value is the maximum Z increment that will be used to rough the feature. If the *Rough pass Z increment* evenly divides the depth of your feature, your increment will be used. If it results in a final pass that is quite shallow, the depth of cut will be adjusted to result in even roughing passes. For example if you have a feature that is 0.5 inches deep and specify a *Rough pass Z increment* of 0.4, the pocket will be roughed in two even passes 0.25 inches deep rather than one pass of depth 0.4 inches and another pass with depth of 0.1 inches.

Total stock

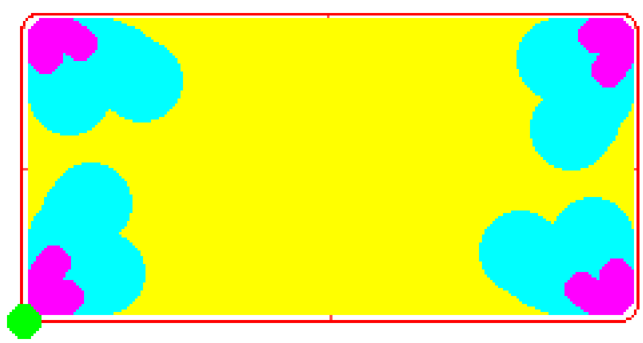
Total stock changes the roughing strategy to use a constant offset distance from the profile of a milling feature. So instead of cutting to the stock boundary, which may already have been cut away, and without having to specify a stock curve, you can still minimize redundant cutting by using *Total stock*. The figure on the left-hand side has the *Total stock* attribute unset. The figure in the middle has the *Total stock* set to 0.25 inches. The right-hand figure has *Total stock* set to 1.0 inches.



Mult. rough diameters

Mult. rough diameters controls the use of multiple roughing tools and is specified as a list of diameters separated by commas. The last diameter is also used for the finish pass. If you want FeatureCAM to select the tool to use for the final roughing pass, set the last diameter to 0.

Your diameter list should work its way gradually down to the desired roughing tool. One way is to let the system pick the tooling initially, then set up your diameter list to work gradually toward the last value. If the system recommends a 0.125 inch endmill, set *Mult. Rough diameters* attribute to “1.0, 0.5, 0”. The figure below shows the cuts performed by each of the three tools.

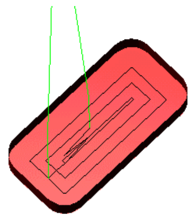


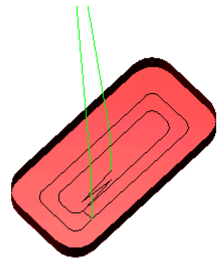
By default, FeatureCAM creates one roughing and one finish pass for all of the milled features. If you use multiple roughing tools, FeatureCAM cuts all the parts of a feature that it is capable of cutting with the larger tool, and cuts only the remaining portions of the feature with the smaller tool. You do not have to manually create these separate regions. FeatureCAM automatically calculates them for you.

For cutting pockets or bosses, FeatureCAM automatically selects a single tool diameter for roughing and finishing. For a large feature with small corner diameters, this method results in a small tool cutting the middle of the pocket and wastes time. A better strategy uses a larger tool for the wide areas and a smaller tool for the tight corners.

Toolpath corner %

This attribute rounds the corners of milled roughing passes. It is specified as a percentage of the tool diameter. Rounding the sharp corners of the toolpaths provides a more constant tool velocity and reduces the tool load. It applies to all 2.5D milling features and Z level rough passes. The left-hand figure shows a rectangular without toolpath corner % set. The right-





hand figure shows the same pocket with Toolpath corner % set.

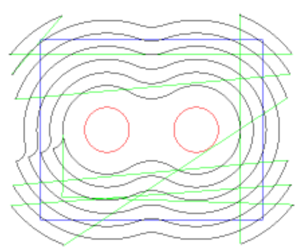
High speed machining application of toolpath corner %

Toolpath corner % can be set to any positive percentage. Setting it to a large percentage like 200% or 300% provides a high degree of toolpath smoothing. In this case, where the radius of the cutter is significantly less than the radius of the corner, the percentage of the tool that contacts the part is minimized. This allows the tool to cool and also avoids sharp increases in tool load as it enters the corners.

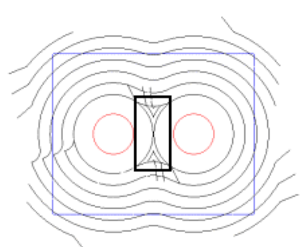
For high speed machining applications:

1. Create at least two roughing passes using Mult. rough diameters parameter. You can set the tool diameters to be the same if you want to use the same diameter for each pass.

2. Set the *toolpath corner %* to a high value (for example 200%) for the initial roughing pass. This pass will cover the majority of the part with smooth toolpaths. This figure shows an example of smoothing toolpaths using a high corner %.



3. Set *toolpath corner %* to lower values for subsequent roughing passes. **These toolpaths will only cover the remaining regions of the part**. This figure shows the second roughing toolpaths. It is best to set *tool corner %* to less than 25% for the last roughing pass to ensure that the entire part is roughed.



These toolpaths will have more inconsistent tool loads, but you can adjust the stepovers, depth of cut, or feed rate separately for these passes to create acceptable tool loads. You can use the tool loads dialog box during 3D simulation to verify the tool loads of your paths before cutting.

Finishing attributes

Semi-finish allowance

Semi-finish allowance is the amount of material to leave after the semi-finishing pass.

Finish overlap

Finish overlap applies to features defined by closed profiles and is the distance that the tool overlaps its starting point on the finish pass.

Finish allowance

Finish allowance is the amount of material left on the walls of a milled feature after the rough passes. *Bottom finish allowance* controls the amount left on the floor of milled features.

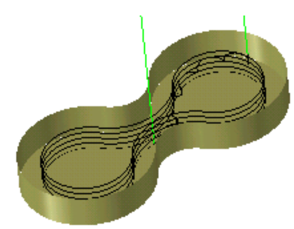
If *Finish allowance* is equal to 0, then FeatureCAM tries to leave a nice finish on the part by ramping in and out on the final cut of the roughing pass. Usually, when *Finish allowance* is non-zero, then FeatureCAM doesn't ramp out -- it just retracts. This happens only if you've selected ramped stepovers, but not when you've selected direct stepovers.

Bottom finish allowance

Bottom finish allowance is the amount of material to leave on the floor of a milled feature. It only applies if finish bottom is checked. The attribute, finish allowance, controls the allowance on the walls of a feature.

Finish pass z increment

By default, a milling feature will be finished with a single pass along the wall of the feature. If *Finish pass z increment* is set to a positive number, the feature will be finished in a series of vertical passes. The depth of these passes will equal the *Finish pass z increment.* The finishing tool need only have enough cutter length greater than or equal to *Finish pass z increment.*



Finish passes

Finish passes is the number of finish passes to perform. If you want to compensate for tool deflection, set Finish Passes to more than 1.

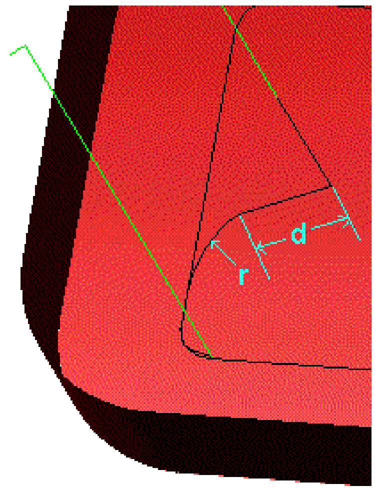
Mult. finish diameters

Mult. finish diameters controls the use of multiple finishing tools and is specified as a list of diameters separated by commas. If you want FeatureCAM to select the tool to use for the last pass, set the last diameter to 0.

Your *Mult. finish diameter* list should work its way gradually down to the desired finish tool. One way is to let the system pick the tooling initially, then set up your diameter list to work gradually toward the last value. If the system recommends a 0.125 inch endmill, set your *Mult. Finish diameters* attribute to “1.0, 0.5, 0”.

By default, FeatureCAM creates one finish pass for all of the milled features. If you use multiple finishing tools, FeatureCAM cuts all the parts of a feature that it is capable of cutting with the larger tool, and cuts only the remaining portions of the feature with the smaller tool. You do not have to manually create these separate regions. FeatureCAM automatically calculates them for you.

Default ramping for milled finish passes



The lead in moves for finish passes for closed milled features like pockets and bosses consist of a short linear move and an arc ramp on move. These moves are included to accommodate the cutter compensation requirements of many controllers.

The radius of the arc, r, is controlled by the *Ramp Diameter%* attribute found on the Stepover/Lead tab. It is specified as a percentage of the tool diameter. The default length of the linear move, d, is the *Finish Allowance* of the roughing pass of the feature. The length of d can also be altered by setting the *Minimum Ramp Dist* milling attribute on the finish pass to an explicit distance or by setting the *Minimum Ramp Dist %* default machining attribute as a percentage of the tool diameter. A setting of 0 for *Min Ramp Dist %* indicates that the default machining attribute is not active.

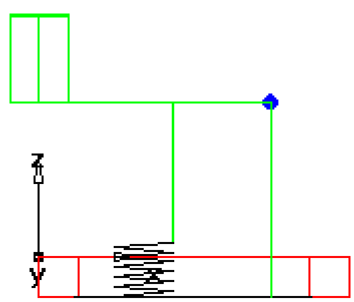
Ramp diameter

Ramp diameter sets a percentage of the tool diameter to generate a tool motion that approaches the stock along a curve on the finishing pass. The tool only arcs within the distance set in the finish pass allowance so the ramping effect is small.

Side leave allowance

This is the amount to leave on the walls of a milled feature after the finish pass.

General attributes



Retract point

This is the point that the tool will retract to after the operation.

Priority attributes

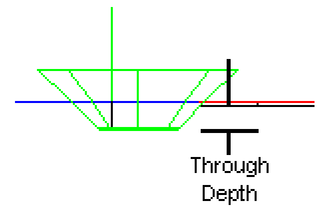
Operation ordering should be performed in the Op. List tab of the Results window.   The *Priority* attribute is included for compatibility with older versions of FeatureCAM.

If you want to ensure that an individual feature is cut before anything else, you can set its *Priority* attribute in the Misc. tab. All features have a *Priority* manufacturing attribute. By default, the value is 10. To make sure that a feature is manufactured first, set its priority to a lower value. To make a feature last, set its priority to a higher value. For example, if you set the *Priority* of a pocket to 8, its roughing pass is the first operation performed, its finish pass is second, and the rest of the operations are ordered according to the scheme described above.

NOTE: While you can specify the exact order of every feature by priority, you shouldn't do so casually because you lose the automatic optimization sequences built into FeatureCAM. It's harder to maintain or change the part too.

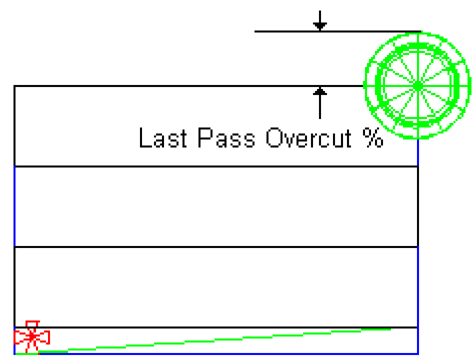
Through depth

Through depth is extra depth that will be added to the operation. This is essentially equivalent to changing the feature depth for a single operation. It applies to slots, step bores, grooves, sides and chamfer features.



For chamfers, *Through depth* controls the depth of the tool and therefore the contact point. The default *Through depth* for a chamfer is 0.1 inches. This means that the tool will extend 0.1 inches below the bottom of the chamfer. A setting of 0.0 will place the bottom of the tool at the bottom of the chamfer. A larger value will move the contact point down the tool.

Facing attributes

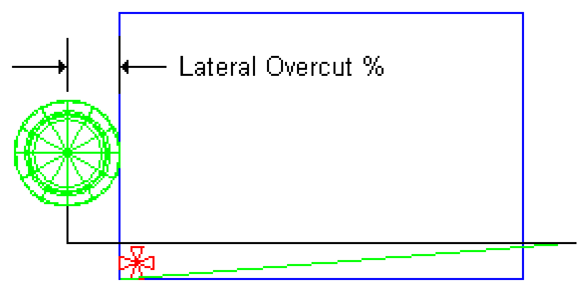


Last pass overcut %

Last pass overcut % is the distance, specified as a percentage of the tool radius, that the tool will cut past your stock boundary in the direction perpendicular to the cut. This is typically the Y direction. The figure shows a *Last pass overcut%* of 100.

Lateral overcut %

Lateral overcut % is the distance, specified as a percentage of the tool radius, that the tool moves past the stock boundary in the direction of the cut, normally the X direction. The figure shows a *Lateral overcut%* of 100.



Stepover%

Distances between passes of a facing operation specified as a percentage of tool diameter.

Z increment

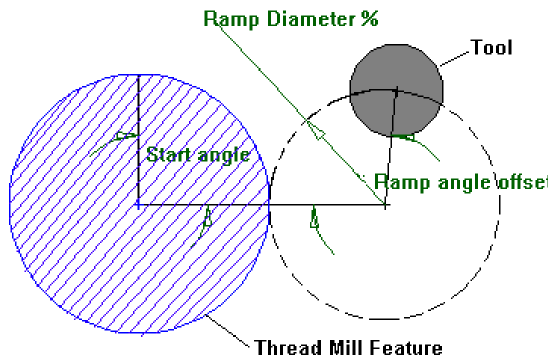
Z Increment is the Z depth of each cut of the facing operation. It is specified in absolute terms.

Thread mill attributes

Linear ramp dist

Linear Ramp Dist controls the length of the linear approach move to a thread milling feature. For this attribute to be active, you must set Ramp diameter % to 0.

Ramp diameter



Ramp diameter sets a percentage of the tool diameter to generate a tool motion that approaches the stock along a curve on the finishing pass. The tool only arcs within the distance set in the finish pass allowance so the ramping effect is small.

Ramp angle offset

This angle controls the starting and ending points of the ramp moves of a thread milling feature. The tool will start ramping along the arc of radius Ramp diameter % using the Ramp angle offset to determine the start point of the ramping move. If positive, the arc will be counter-clockwise.

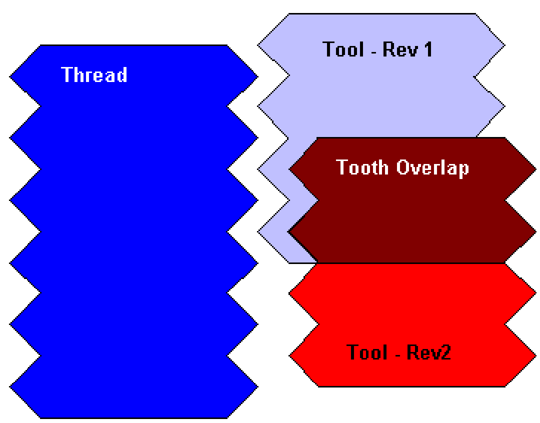
Start angle

Measured counter-clockwise, the angle determines the starting point of the thread.

Start threads

Set Start Threads to a value greater than one for multiple start threads.

Taper approx angle



For tapered threads the toolpath is increasing in diameter as well as moving in Z. These moves are approximated with 3D arcs. The Taper Approx Angle is the angle around the thread that will be approximated by a single arc. 360 must be evenly divisible by the Taper Approx Angle. For example, if set to 90, a single revolution of the tool will be broken into 4 arcs.

Tooth outside

This is the number of teeth that will be above (if feeding in negative Z) or below (if feeding in positive Z) the thread mill feature for the first pass.

Tooth overlap

This thread milling attribute controls the amount one revolution of a multi-thread tool will overlap the previous revolution. It is an integer that represents the number of threads. It is recommended to overlap at least 1 thread.

Ramp angle offset

This angle controls the starting and ending points of the ramp moves of a thread milling feature. The tool will start ramping along the arc of radius Ramp diameter % using the Ramp angle offset to determine the start point of the ramping move. If positive, the arc will be counter-clockwise

Draft angle or bottom radius attributes

The operations used to manufacture drafted features or features with a bottom radius are discussed on page 122.

Draft flat scallop height

Draft flat scallop height affects the roughing of the walls of tapered features or features with a bottom radius. It sets the scallop height of the *Draft flat* operation. Note that this attribute has no affect if the *Flat endmill* Strategy attribute is checked and you enter a specific number of Steps.

Draft radius scallop height

Draft radius scallop height affects the roughing of the walls of tapered features or features with a bottom radius. It sets the scallop height of the *Draft radius* operation. Note that this attribute has no affect if the *Radius endmill* Strategy attribute is checked and you enter a specific number of Steps.

Bottom radius corner stepover %

Bottom radius corner stepover % is a percentage distance based on the size of the finish tool diameter. This value sets how far out the ball end finish tool will start for a corner operation.

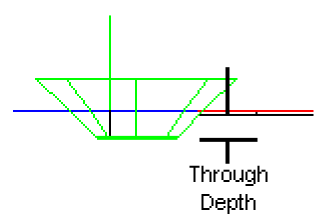
Radius tool scallop height

The *Radius tool scallop height* attribute affects the scallop height of the *Finish pass* operation of a drafted feature or feature with a bottom radius.

Chamfer attributes

Chamfer depth

For milled chamfers, *Chamfer depth* controls the depth of the tool and therefore the contact point. The default *Chamfer depth* for a chamfer is 0.1 inches. This means that the tool will extend 0.1 inches below the bottom of the chamfer. A setting of 0.0 will place the bottom of the tool at the bottom of the chamfer. A larger value will move the contact point down the tool.



Stepover/lead attributes

The stepover/lead tab contains controls for toolpath transitions for 2 ½ D milling. The type of transitions that occur at the beginning and ending of a toolpath depends on whether that portion of the toolpath is a closed or an open toolpath

An *open* toolpath plunges at one point and retracts at another point. This type of toolpath is shown in the lower right-hand corner of the figure. A *closed* toolpath forms a loop and begins and ends at the same point. Closed toolpaths are shown in the upper left of the figure. Note that rough or finishing passes can contain closed, open or both types of toolpaths.

The stepover parameters control transitions between closed loops. The Leads parameters control open toolpath transitions.

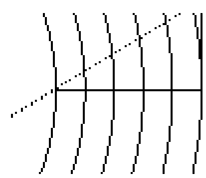
Distance between cuts

For roughing, the *stepover distance* is the horizontal distance between roughing toolpaths. The automatically calculated distance is based on the setting of Rough pass stepover %. For finishing, it is fixed to be the *Finish allowance*.

For chamfers and rounds this attribute enables multiple rough or finish passes. The default values for roughing is the radius of the round feature or the largest dimension of the chamfer feature. The default value for finishing is the *Finish allowance*. By decreasing this value multiple roughing or finishing passes are created by stepping in horizontally.

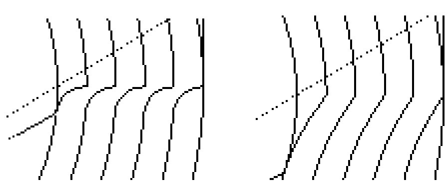
Direct stepover style

The direct stepover style creates a straight linear transition that is perpendicular to the toolpath.

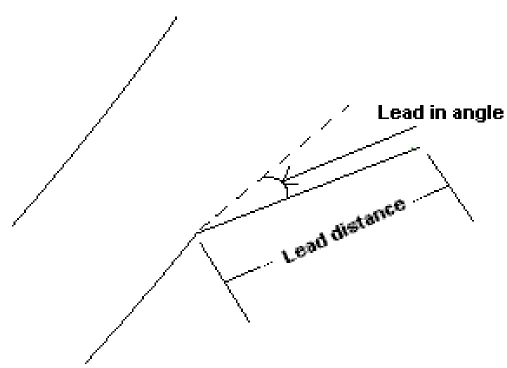


Ramp stepover style

The ramp stepover style creates an arc transition. Set the *Diameter* parameter to specify the radius of the arc as a percentage of the tool diameter. The left-hand figure shows ramping with *Diameter* set to 55%. The right-hand figure shows ramping with a 600% *Diameter* setting.



Lead distance



Lead distance is the linear distance that a tool path extends beyond the ends of an open toolpath or toolpaths that are clipped against the stock profile. This parameter is specified as a percentage of the tool's diameter. If Lead Distance is set to 0.0, the tool path stops exactly at the ends of the profile.

Lead in angle

Lead in angle is the angle applied to the start of an open toolpath. The lead in angle occurs only over the Lead distance, so if Lead distance is 0.0, Lead in angle has no effect.

Lead out angle

Lead out angle is the angle applied to the end of the finish pass for open toolpath. It also applies to the last toolpath of a roughing pass if the *Finish allowance* is set to 0.0. The lead out angle occurs only over the Lead distance, so if Lead distance is 0.0, Lead out angle has no effect.

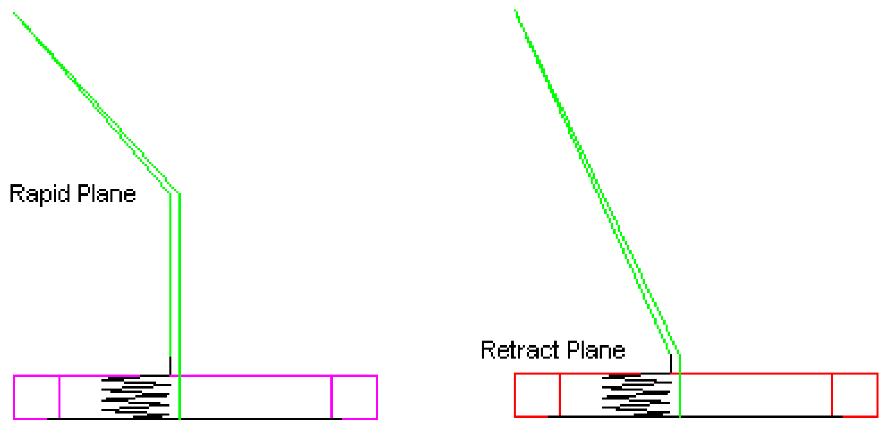
Insert arc

Insert arc changes the lead in or lead out move to be an arc. The endpoint of the arc is determined by the Lead distance and either the Lead in angle or Lead out angle

Misc. attributes

Retract to plunge clearance

Retract to plunge clearance can lessen wasted time on milling features by retracting a lower Z clearance after cutting the pocket.



Plunge clearance

Plunge clearance is the distance above an operation at which the tool starts to feed. In the case of deep hole drilling, the drill will retract to this distance between pecks. For milling features, the default is to use the same value for roughing and finishing. As a result, the tool will feed from the top of a pocket to the floor before cutting. To make the tool feed down into the feature, set the *Plunge clearance* for an operation to a negative value, but make sure the value is above the floor of the feature.

Coolant

Coolant indicates the type of coolant to use for a feature. The choices are:

- Flood - a continuous stream of coolant
- Mist - coolant mixed with air
- None - no coolant

Base priority

Base priority changes a feature's initial priority. The features are then sorted by their priority to determine the order in which they are manufactured. The *Priority* attribute on each feature can be used to order the features for manufacturing. Among features with identical priority values, FeatureCAM will use minimization of tool changes and other criteria to determine a manufacturing order. The Base Priority attribute is the initial priority that each feature will be assigned by default. Setting the *Base Priority* default machining attribute establishes the priority value for each feature. If you explicitly set a feature's priority, it overrides any priority value that the feature had previously.

Feed override %

Feed override % is a scaling factor for the feed rates generated by the system. A value less than 100 reduces the calculated feed rates. A value more than 100 increases the rates.

Max. spindle RPM

Max. spindle RPM is the maximum spindle speed (in RPM) that FeatureCAM will output.

Plunge feed override %

Plunge feed override % gives the scaling value for the feed rate used during the initial plunge into the material and the initial horizontal step over for milling operations.

Speed override %

Seed override % is a scaling factor for the speeds generated by the system. A value less than 100 reduces the calculated speeds. A value more than 100 increases the rates.

Spline tolerance

Spline tolerance approximates the profile with arcs and lines if a profile is defined as a spline. The smaller the value of the parameter, the smoother the profile.

Tool % of arc radius

Tool % of arc radius controls the size of the tool that FeatureCAM automatically selects. In earlier program versions this attribute was called Default tool %.

If *Tool % of arc radius* is set to 100 then a tool equal to the smallest corner radius is selected for a feature such as a pocket, and the finish tool path for the pocket looks like the toolpath shown above. With *Tool % of arc radius* set to 100 the tool dwells in the corners as it changes direction. This can sometimes nick the part. To avoid this problem, set *Tool % of arc radius* to a slightly smaller number, such as 98.

Drilling feature attributes

Strategy tab attributes

Attempt chamfer with spotdrill

Attempt chamfer with spot for spot drilling with a 90 degree spotdrill, set the Attempt Chamfer w/Spot to cut the chamfer during spot drilling. If your tool does not have a 90 degree profile and this checkbox is set, FeatureCAM automatically ignores this setting. If a tool cannot be found that will spot and chamfer without gouging the hole, a separate chamfer operation is created.

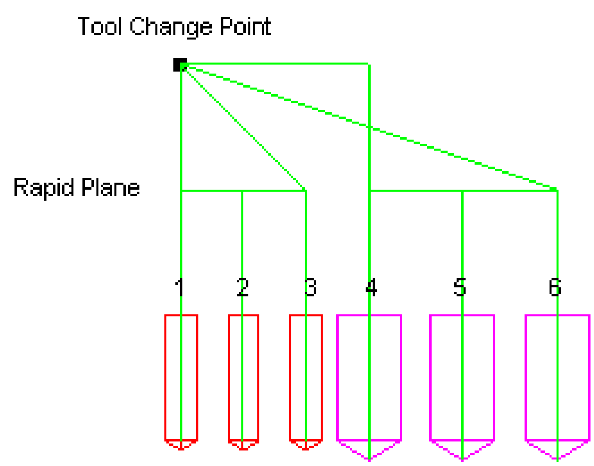
Bore

Bore adds a boring operation to the to manufacture of a hole. Boring places a hole very accurately.

Combine with similar holes into canned cycle

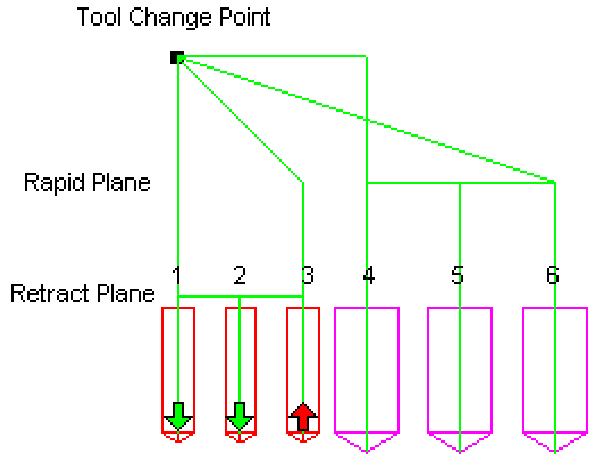
This attribute applies to drilling operations. In previous versions of FeatureCAM this attribute was called Retract to plunge clearance. Note the retract to plunge clearance still applies to milling operations. By default, FeatureCAM retracts the tool to the *Z Rapid Plane* between operations. While this is a safe assumption, it can result in inefficient NC part programs. Between each operation the program cancels (G80) and then re-establishes (G83) the canned cycle mode as shown in the NC code below.

```
:10
(9-13-2001)
N25G00G17G40G49G80
N30G30G91Z0
N35T1M6
N40G00G54G90X0.Y0.S3819M03
N45G43H1Z1.0M08
N50Z0.1
N55G83R0.1Z-1.0Q0.25F14.3
N60G80
N65Z1.0
N70X0.5
N75Z0.1
N80G83R0.1Z-1.0Q0.25F14.3
N85G80
N90Z1.0
N95X1.0
N100Z0.1
N105G83R0.1Z-1.0Q0.25F14.3
N110G80
N115Z1.0
N120G0G91G28Z0M09
N125G49G90
N130M30
```



The *Combine with similar holes into canned cycle* attribute serves two functions. First it creates more efficient NC code by entering canned cycle mode only once. It also lowers the retracts that occur after drilling each hole to the Retract Plane. If the Post option variable, *Disable Macros*, is unchecked, the hole locations are included in a macro.

```
:10
(9-13-2001)
N25G00G17G40G49G80
N30G30G91Z0
N35T1M6
N40G00G54G90X0.Y0.S3819M03
N45G43H1Z1.0M08
N50Z0.1
N55G83R0.1Z-1.0Q0.25F14.3
N60P1001M98
N65G80
N70G00Z1.0
N75G0G91G28Z0M09
N80G49G90
85M30
:1001
N90G91
N95X0.5
N100X1.0
N105G90
N110M99
```



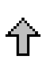
If *Disable Macros* is checked, the NC code is still efficient since canned cycle mode is only entered once.

```
:10
(9-13-2001)
N25G00G17G40G49G80
N30G30G91Z0
N35T1M6
N40G00G54G90X0.Y0.S3819M03
N45G43H1Z1.0M08
N50Z0.1N55G83R0.1Z-1.0Q0.25F14.3
N60X0.5
N65X1.0
N70G80|
N75G00Z1.0
N80G0G91G28Z0M09
N85G49G90
N90M30
```

Once *Combine with Similar holes into canned cycles* is checked on a feature you can specify the retract plane for each of the feature's operations individually in the Retract column of the Oper List. The column will contain one of the following symbols:

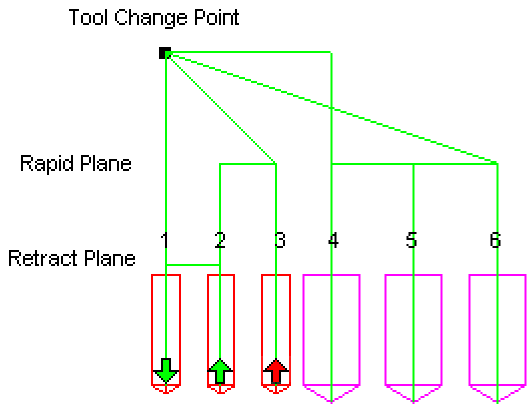
This green down arrow indicates that the tool will retract to the lower retract plane after performing the operation. This arrow can be toggled to an up arrow (Retract to Rapid Plane) by clicking the arrow with the left mouse button and selecting *Rapid plane* from the pop up menu.

This green up arrow means that the tool will retract to the higher Clearance plane after the operation. It can be toggle to a down arrow by clicking the arrow with the left mouse button and selecting *Plunge clearance* from the pop up menu.



This gray arrow means that the tool will retract to the higher Clearance plane after the operation and it cannot be changed.

The figure below shows two hole patterns. The first pattern that contains holes 1, 2 and 3 has *Combine with Similar holes into canned cycles* checked. Hole 2 has been modified to retract to the Retract plane.



If you are using a post that supports different rapid planes inside a canned cycle, you can create G-code that is more efficient. For example, Fanuc supports G98 for retracting to the rapid plane and G99 for retracting to the Z rapid plane. These G-codes are entered in the post processor as the *R plane retract* and *Z rapid retract* respectively. The resulting program is as follows:

```
N65 G83 G98 Z-1.0751 R0.1 Q0.25 F14.3
N70 X0.0
N75 X0.5
N80 X1.0  G99
N85 X1.5  G98
N90 X2.0
N95 X2.5 G99
N100 G80
```

Drill

Drill attribute adds a drilling operation to the manufacture of the hole. This operation is usually undersized in preparation for later reaming or boring.

Pilot drill

The *Pilot drill* attribute adds a pilot drilling operation to the hole. You must then set the Pilot drill diameters drilling attribute of the pilot drill operation to specify the diameters that will be used to create the hole.

Pilot drill diameters

Pilot drill diameters turns on and sets a list of drill sizes used to drill pilot holes. Specify a list of diameters with a comma between each drill size. Specifying .5, 1, 1.5 in inches for example causes holes to be pilot drilled with the half-inch drill for final hole sizes up to an inch. A hole in excess of 1.5 inches would be pilot drilled with all three of the specified drills before being drilled to size. No list of drill sizes turns off pilot drilling for the feature, although this attribute can also be set up as the default for all parts.

Ream

Ream drills a hole feature undersized and then reams it to size. The diameter of the drill will be between 93% and 97% of the final hole diameter.

Spot drill

Spot drill tells the system whether or not to spot drill (or center drill) a hole before drilling. This operation has some wide-ranging effects, however, especially when used with the Attempt Chamfer w/ Spot and tool optimization. Of those three settings, tool optimization has the highest priority and it's decisions override settings with a lower priority.

For example, a spot drill operation could be performed with either a spotdrill or a centerdrill. Spotdrills with a tip angle of 90 degrees can also perform a chamfering operation. You specify a specific tool to cut the hole's chamfer and also turn on Attempt Chamfer /w Spot and tool optimization. If there is an appropriate spotdrill in the toolcrib FeatureCAM will optimize things and use this tool in spite of your lower priority override. Even though you selected a specific tool, your other settings conflicted with and superseded your choice.

Note: If you are using an insert drill for the twistdrill operation of a hole, a spotdrill operation will not be created for the hole regardless of the setting of this attribute.

Cycle tab attributes

This tab allows you to specify a variety of canned cycles for a hole operation. Select the type of drilling cycle that will perform the operation. After each operation the G-code contained in the current post processor for each cycle is listed in parenthesis.

Drill cycle

Drill cycle chooses a simple drilling operation for the operation. The tool drills down and up in one motion. No pecking is performed.

Spot face cycle

A *Spot face* cycle is a drilling cycle with an optional dwell.

Bore cycle

Bore cycle affects how a bore is performed. The choices are FF, FDF, FSR (feed-stop spindle-retract) and No drag. If FF is chosen the cycle is posted using the Bore(F-F) format in MBUILD. FDF will use the Bore(F-D-F) format.

Tap cycle

Tap cycle affects how a tap operation is performed. The choices are floating, rigid, deep hole and chip break. All cycles use the same Tap program format, but logical reserved words exist in Mbuild to distinguish the desired tap type.

Ream cycle

Ream cycle affects how a ream is performed. The choices are FF (feed-feed), FDF (feed-dwell-feed) FSR (feed-stop spindle-retract) and No drag . If FF is chosen the cycle is posted using the Bore(F-F) format in MBUILD. FDF will use the Bore(F-D-F) format, FSR uses the Bore (F-S-R) format and No drag uses the Bore (No drag) format.

Deep hole cycle

Drill cycle affects how the chips are cleared from the hole in a drilling pass. In a Deep Hole cycle, the drill retracts all the way above the stock a number of times during the process to clear debris from the hole. Each plunge and retract is called a peck. FeatureCAM starts pecking when the hole depth is greater than 2.5 times the drill diameter.

Chip break cycle

In a Chip Break cycle, the drill retracts a short distance to clear chips before plunging again. Each plunge and retract is called a peck. FeatureCAM starts pecking when the hole depth is greater than 2.5 times the drill diameter.

Cycle parameters

Dwell

The amount of time to dwell for a feed-dwell-feed cycle.

No drag X shift and No drag Y shift

These attributes affect the amount that the boring tool shifts prior to retracting in No-drag boring.

First peck

This is the depth of the first peck of a tapping operation. If the depth of the hole is less than *First peck* the hole will be drilled in a single peck. See page 197 for an overview of pecking.

Second peck

This is the depth of the second peck of a tapping operation. See page 197 for an overview of pecking.

Minimum peck

This is the minimum step size for a peck used for value reduction pecking methods or factor reduction pecking methods. See page 197 for an overview of pecking.

Drilling tab attributes

Drill depth

Drill depth sets the depth that a twistdrill, bore, countersink or reaming operation is driven to in the stock, not including a point allowance. The depth setting in the dimension attributes automatically includes a point allowance so use this attribute to override the point allowance.

Max tap spindle RPM

Max tap spindle RPM refers to the maximum speed (in RPM) for tapping.

Spot drill depth

Spot drill depth sets how deep the spotdrill operation proceeds into the stock.

Tap depth

Tap depth is an override for setting the depth of a tapping operation. FeatureCAM will automatically set a depth based on the thread depth and the geometry of the tap that is chosen. If you set this attribute, no additional adjustment is made for the tap geometry. The *Tap depth* is simply passed directly into the NC code.

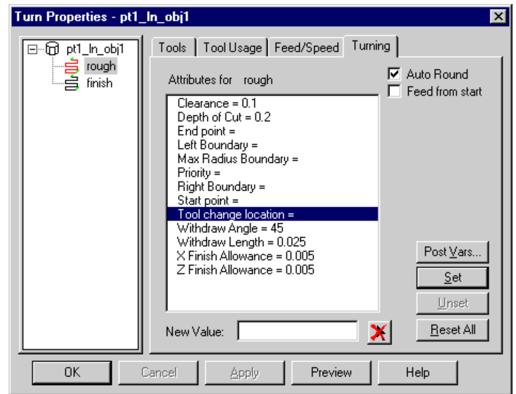
Chamfer depth

The absolute depth of a drilled chamfer operation.

Turning feature attributes

Manufacturing tab controls

The turning tab for turned features looks like this:



New Value is the place for changing an attribute’s value.

To change an attribute:

1. Double click the feature to open the feature's Properties dialog box.
2. Click the appropriate tab in the dialog box.
3. Set the check box or select the attribute.

- For parameters with a check box, click the check box to toggle the attribute's value.
- For parameters with a pull down list (indicated by an arrow to the right of the value), click the arrow, then select the new value.
- For numeric attributes, click the name. The current value appears in the New Value line at the bottom of the dialog box.

4. Type in the new value.
5. Click Set or press Enter.

Once a parameter has been changed from the system supplied default, an * appears next to the name to indicate that the value has been altered.

Set applies the *New Value* on the attribute that you have selected.

Unset sets the selected attribute back to the system-supplied default.

Reset All sets the default value for all attributes listed on that page.

Away from chuck

If clicked, the threading will be performed in the direction away from the chuck.

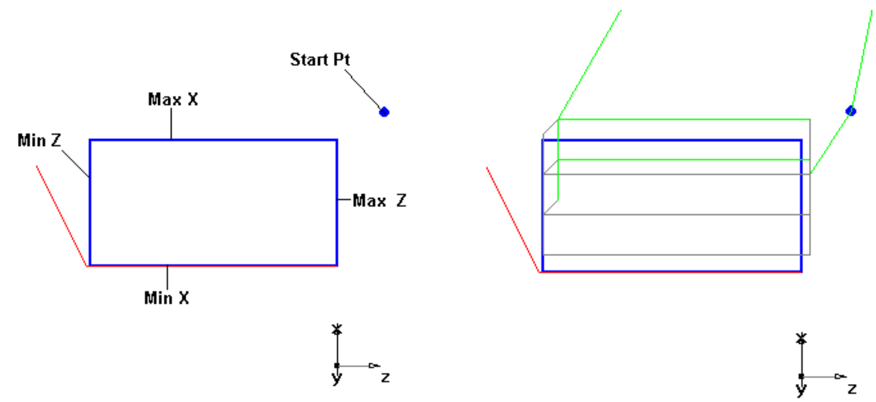
Auto round

This parameter is set either On or Off. When turned *on*, the system automatically inserts arc moves to connect two non-tangent elements. This results in a minimum of wasted motion by the machine; however, the posted part program may be slightly longer in the number of blocks used.

Boundaries

The Left, Right, Max Diameter and Min Diameter boundary parameters limit the portion of the feature that will be roughed. These boundaries are displayed in blue whenever you select the rough operation in the tree view. When the material boundaries are defined to machine a part, the boundary must be specified so that it completely encloses the path (e.g., the path cannot start or end in the middle of the rectangular box; it must start on, or outside of the boundary).

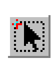
When the path is defined, it may extend beyond the material boundary. This is a powerful technique in roughing, since a long path can be defined, then an area in which a specific portion is roughed only between material boundaries can be established. In a second segment, the path could be copied, then the boundaries to rough another section of the path can be redefined, and so on. Any change that is made to a boundary parameter is automatically displayed on the screen.

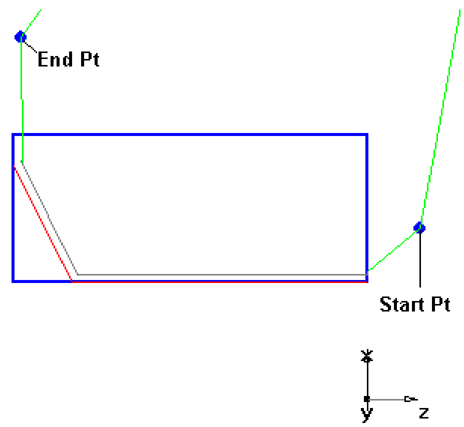


Start point and End point

Start point and *End point* attributes are available on all turn operations. If the *Start point* is set, the tool will rapid to this point at the start of the operation. If the *End point* is set, the tool will rapid to this point at the end of the operation.

To set the start or end point:

1. Double click on the feature to bring up its Properties dialog box.
2. Click on the operation in the tree view.
3. Click on the Turning tab.
4. Click on the Start point or End point parameter.
5. Click the Pick XYZ Location button  next to the New Value slot at the bottom of the dialog box.
6. Click the location in the graphics window.
7. Click Apply



Chamfer extend dist

Chamfer Extend Dist provides extra space for the tool so that the tool does not start on the metal for the groove finish pass.

Depth of cut

The Depth of cut parameter specifies a step increment for each pass that the roughing routine performs on the part. This value is the maximum depth of cut for the feature. If the *Depth of cut* evenly divides the depth of your feature, your increment will be used. If it results in a final pass that is quite shallow, the *Depth of cut* will be adjusted to result in even roughing passes. For example if you have a feature that is 0.5 inches deep and specify a *Depth of cut* of 0.4, the feature will be roughed in two even passes 0.25 inches deep rather than one pass of depth 0.4 inches and another pass with depth of 0.1 inches.

Dwell

The number of seconds the tool will dwell after plunging during a groove roughing pass. It also applies to the roughing of the cutoff chamfer.

End clearance

This parameter controls the distance that the tool feeds past the end of the thread (into the relief groove) before retracting from the part's surface.

Engage angle

The *Engage angle* controls the approach of Turn and Bore features. The *Engage angle* is measured away from the part. An angle of 0 will approach along the path. An angle of 90 will approach perpendicular to the path.

Clearance

At the beginning of an operation the tool rapid traverses to a point that is a distance away from the beginning of the toolpath. This distance is the *Clearance*. The *Clearance* is also used to calculate the move at the end of the operation.

The location of these points is also controlled by the *Engage Angle*, *Withdraw Angle*.

If you are using *Tool Nose Radius Compensation*, *Lead In Dist* is used instead of *Clearance*.

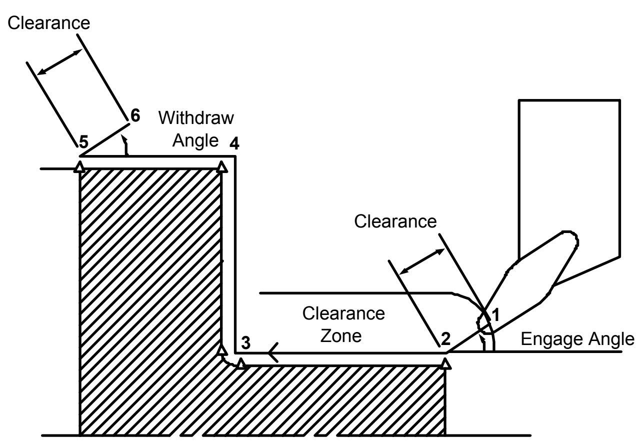
Withdraw angle

This parameter specifies the angle between cross feed movement and the withdraw move. For both roughing and finishing passes this angle is measured against the roughing scanlines.

An angle of 90 will retract perpendicular to the roughing pass. An angle of 45 will pull back away from the part and the chuck. An angle of 135 will pull toward the chuck. **As of version 9, the withdraw angle for finishing is not dependent on the shape of the feature curve**. Even if no roughing pass is created, the withdraw angle for finishing pass is measured against the roughing scanlines.

Illustrating engage and withdraw angles

Engage and *Withdraw angles* are specified from the path (or extension of the path), relative to the side of the path that the tool is on, and the direction in which the tool is traveling. In the graphic below, Point 1 is the calculated engage point and the Point 6 is the calculated withdraw point.

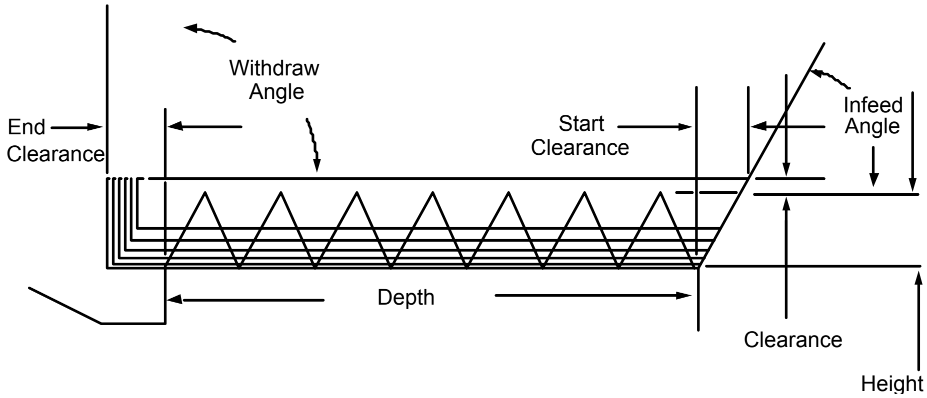


Feed Dir

This attribute controls the direction the tool will feed.

Infeed angle

The infeed angle is specified as an unsigned, incremental value from the positive Z-axis. This figure illustrates the infeed angle.



Lead in angle

This parameter applies only for tool nose radius compensation. It is the angle for the lead in move for semi-finishing and finishing of turn and bore features The Lead in angle is measured away from the part. An angle of 0 will approach along the path. An angle of 90 will approach perpendicular to the path.

Finish passes

Normally, Finishes Passes is set to 1 and a single pass is generated offset by the tool tip. If Finish Passes is set greater than 1, then the region to be finished is divided into equal parts and finished in sequential passes. The region to be finished is the X Semi Finish Allowance and the Z Semi Finish Allowance if the feature has a semi-finish pass, and it is the full the X Finish Allowance and Z Finish Allowance if the feature has no semi-finish pass.

Lead in dist

This parameter applies only for tool nose radius compensation. *Lead in dist* is the distance for the lead in and lead out moves.

Lead out angle

This parameter applies only for tool nose radius compensation. It is the angle for the lead out move for semi-finishing and finishing of turn and bore features. It is measured clock-wise. The *Lead out angle* is measured away from the part. An angle of 0 will exit along the direction of path. An angle of 90 will exit perpendicular to the path.

Side lift off dist

This parameter sets the distance by which the tool moves at the end of a plunge cut. The shift is in the direction opposite to the cutting direction. *Side Lift Off Dist* is used to avoid tool contact with the uncut material when the tool is retracting at a rapid feed rate. Side Lift Off Dist is ignored for the retract move at the end of the first plunge. The actual lift off move is performed at the plunge feed rate. If the groove is a round-bottomed groove, then lift off is not utilized, even when specified.

Minimum infeed

This parameter is only applicable to Thread cycle and is only accessible when the Number of Steps parameter has been set to Calculate. This parameter specifies the minimum infeed distance. The system automatically reduces the infeed distance for each pass after the second step, until the minimum infeed is reached (or full depth).

Number of passes

This parameter specifies the number of steps to the bottom of the thread. You can specify either Fixed or Calculate. If you select Fixed, then you must enter the total steps required for the threading operation in the Passes field. If you select Calculate, then the number of steps for the threading operation is calculated by the system. Additionally, if you select Calculate, then you must supply data for the Step 1, Step 2 and Minimum Infeed fields.

Parts catcher

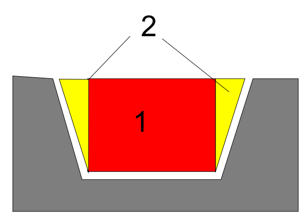
Parameter located on the Strategy page for the cutoff parameter. If checked, the parts catcher code is output after the cutoff operation. The code for activating the parts catcher must be listed in your cnc file.

Peck retract dist

For cutoff and groove features, Peck Retract Dist is the distance the tool retracts between plunges.

Plunge center first

If Plunge Center First is checked, the straight portion of the groove is roughed first and then the angled portions are roughed separately. If Plunge center first is set, the region 1 of this figure is roughed first and then region 2 is roughed.



Plunge rough chamfer

If there is a chamfer on a cutoff feature and Plunge Rough Chamfer is checked on the Strategy page:

1. The cutoff groove is plunged down to the depth of the chamfer.

2. The chamfer is plunged roughed.

Spring passes

A spring pass is a duplicate of the final threading pass. *Spring Passes* indicates the number of spring passes that are to occur at the completion of the thread.

Start Clearance

The start clearance value is the position to which the tool traverses before engaging into the work piece.

Start threads

If set to 1, a single thread is created. If set to 2 or 3, multiple start threads are created. The number of threads per inch (or per mm) for each thread is divided by the number of threads. For example, if you create a thread with 10 threads per inch with 2 start threads, then each thread is 5 threads per inch 180 degrees apart.

Step 1

This parameter is used to specify the incremental step for the first pass across the thread.

Step 2

This parameter specifies the second pass and is used by the system to determine subsequent passes on the thread, reducing in depth until the minimum infeed value is reached.

Stepover %

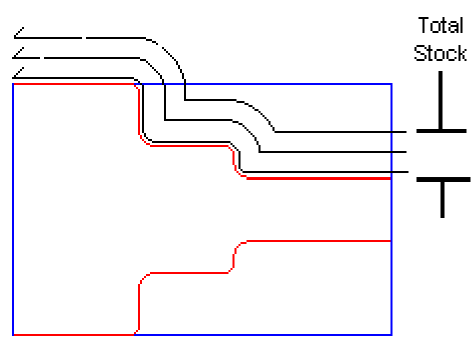
This parameter is expressed as a percentage of the tool's width. It is the distance by which the tool shifts to position itself for the next plunge cut. This value specifies the maximum stepover distance. If this value evenly divides the width of your feature, your increment will be used. If it results in a final pass that is quite shallow, the width of the cuts will be adjusted to result in even roughing passes. For example if you have a feature that is 0.5 inches wide and specify a width of cut of 0.4 (specified as a *Stepover %* of 80 for a tool with a width of 0.5 inches), the feature will be roughed in two even passes 0.25 inches wide rather than one pass of 0.4 inches and another pass with a width of 0.1 inches.

Tool change location

Tool change location is the point where the tip of the tool moves prior to a tool change. It can be set in the Post Options dialog box , or from the Turning tabs of individual operations.

Total stock

The total stock attribute changes the way that the feature is roughed. Instead of roughing within the boundaries of the stock, the region that is roughed is determined by offsetting the feature's curve by the total stock amount. The toolpaths are then performed parallel to the feature's curve.



Towards chuck

If clicked, the threading will be performed in the direction toward the chuck.

Post variables

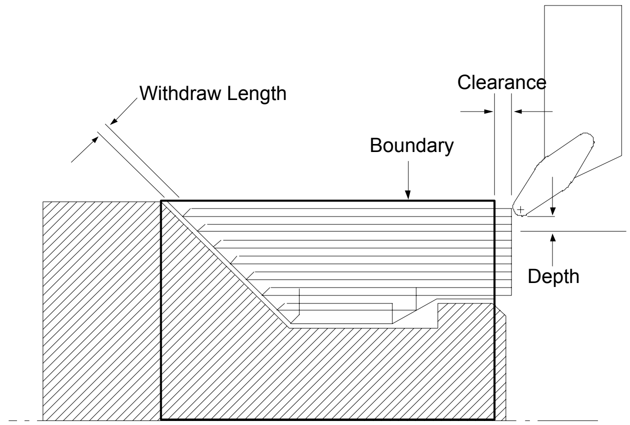
On the Turning or Milling tab of each FeatureCAM feature is a Post Vars. Button. This button brings up the Post Variables dialog box which contains 9 separate variables that are passed directly to the post processor. You can use these variables to pass strings directly to the post processor.

Use finish tool

Use finish tool selects different tools for the rough and finish passes of a feature. Otherwise, the same tool is used for both passes.

Withdraw length

This parameter is the distance along the withdraw angle line in which the tool withdraws before returning for the next step.

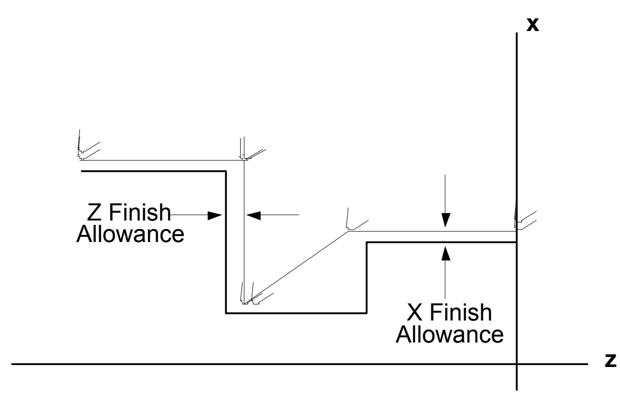


X finish allow

This parameter allows you to specify a separate finish allowance in the X axis direction.

Z finish allow

This parameter allows you to specify a separate finish allowance in the Z axis direction.



Chapter 13
Part Documentation

Manufacturing reports

After you generate a toolpath for a particular part, you can get feedback through the Manufacturing Feedback window or printouts by clicking the details tab located at the bottom of the manufacturing results window.

At the top of the manufacturing feedback window, there are two buttons.

- *Operations List* displays a detailed report on each manufacturing operation
- *Tooling List* displays a report on the automatically selected tools.   See Tooling groups for specific parameters for each tool type.

To view the NC code for your part click on the NC Code tab. This posts the part program and shows the M & G-codes. It is not a report in the normal sense, but is often reviewed as a report by those who understand NC code.   The NC code can also be displayed by clicking the NC code  button of the Steps toolbar.

Note that if you hold the SHIFT key down and then click the NC code tab, the underlying ACL code is displayed in the Manufacturing Feedback Window.

You cannot click the NC tab unless you have a dongle installed. Clicking one of the report buttons shows you the documentation, but it doesn't save it to disk. Use Save NC from the File menu to save the documentation.

Changing the fonts for the reports in the Manufacturing Feedback window

You have a choice of three fonts for the text in the Manufacturing Feedback window

To change the font:

1. Click the right-mouse button in the window
2. Select either *Small Font, Medium Font,* or *Large Font*.

Manufacturing operations sheet

The *Manufacturing Operations* sheet lists information for each manufacturing step on cut type, cut depth and center point, tool details, speed/override, feed/override, and estimated manufacturing time. It is displayed when you click the *OP Sheet* tab.

Part name

Part name shows the file name of the part. If you haven't saved your part, the default part name is "FM1".

Setup name

Setup name shows the name of the coordinate system that defines this setup.

Computed at

Computed at lists the date and time that the NC code was generated.

Stock

Stock reports the dimensions of your initial stock.

Stock material

Shows the material name and hardness of the stock material.

Step no

Step no refers to the number of the current operation. The step numbers start at 1 and are incremented by one for each additional step.

Feature name

Feature name is generated by the program. Change it with Rename in the Edit menu.

Pass

Pass identifies the pass as either a roughing or finishing pass. If multiple roughing tools are used, then the roughing passes are labeled "rough0" and "rough1". If a tool roughs at multiple depths, then the passes are labeled with a dash. For example, if a pocket is roughed at two different depths with the same tool, then the passes are labeled "rough" and "rough-1".

Tool slot no

Tool slot no. is where the tool should be loaded in the tool changer.

Tool diameter

Tool diameter is the diameter of the tool used.

Cut depth

Cut depth is the axial depth of cut for the current operation.

Cut step

Cut step is the radial depth of cut for the current operation.

Cut type

Cut type denotes the type of operation performed.

Speed

Speed is the speed in RPM of the current operation.

Feed

Feed indicates the feed rate for the current operation.

Est. HP

When toolpaths are generated, the estimated horsepower required for each operation is calculated by using the depth of cut, width of cut, feedrate and stock material. The actual shape of the feature is not taken into account. Instead just a simple straight cut is assumed when calculating the estimated horsepower. This number is reported in the Operations list of the Details tab as the Est. HP. The estimated horsepower is only calculated for operations that are cut with flat bottom tools with a fixed depth. This includes 2 ½ D milling features, and 3D Z level roughing and other 3D features with Z increment set.

Manufacturing tool detail sheet

Whenever you simulate a part, the *Manufacturing Tool Detail* sheet is automatically generated. The Manufacturing Tool Detail sheet provides in-depth information on the following three items:

- **Tool name** shows name of the tool.
- **Tool slot no**. shows the tool slot (or tool pocket) that contains the tool.
- **Tool offset no.** shows the tool cutter comp. offset register. If the offset register has the same number as the *Tool Slot No*. it is not reported.

Op list tab

Clicking on the Op list tab in the Manufacturing Feedback window brings up a table of operations.

Each row displays the operation, feature that the operation came from and the tool that will be used to cut the feature. This tab has three main functions:

- Simulation control. See page 171 for details.
- Operation ordering. See page 180 for further information.

- Operation editing.

Feature and operation editing using the op list tab

The Op. list tab also provides for convenient editing of features and operations. Double clicking on a column brings up the following dialog boxes.

Column	**Dialog box**
Operation	Mill or drill tab for the operation. This provides for easy modification of manufacturing attributes.
Feature	Dimensions tab for the feature. Change dimensions for the feature or click on the Strategy tab to alter the types of operations that will be used to manufacture the feature.
Tool	Tools tab for the operation. Review tooling details or modify the tooling for the operation.
Feed	F/S tab for the operation.
Speed	F/S tab for the operation.
Retract	Single click on this column to change the retract plane for drilling features. See *Combine with similar holes into canned cycle* on page 226 for more information.

If a manufacturing error is detected for an operation that would prohibit generating toolpaths, a stop sign  icon is displayed in the left-hand side of the row. Click on this icon to attempt to fix the error.

Note that you can also right-click on a row of the table and a menu of feature tabs is displayed. Select the tab name to go directly to that tab.

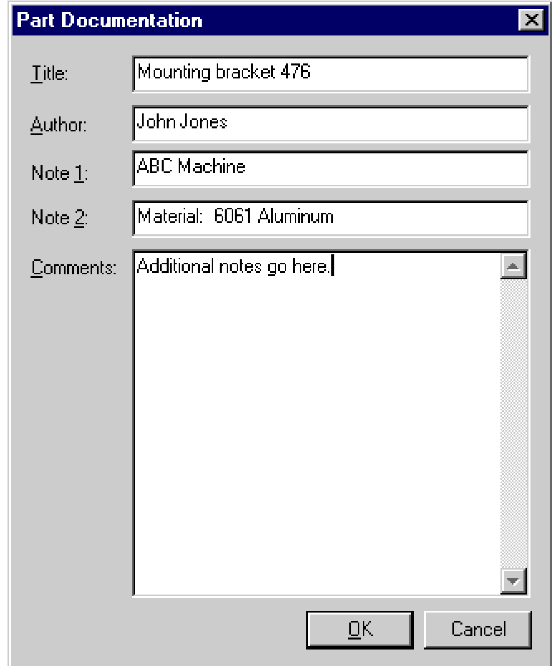
Overriding the tool, feed or speed for multiple operations at once

1. Select the first operation in the Op List tab in the Results window.
2. Hold down the SHIFT key and select the last operation so that a block of operations are selected.
3. Click the right mouse button on the selected group. A popup menu will be displayed with three items that begin with the word, *override*. Select the proper item to override the tool, feed or speed of all operations so that they are identical to the first selected operation.

Printing

Part documentation

The part documentation dialog box allows you to add comments to the printed documentation. Fill in the Title, Author, Note fields and comments and these text strings are printed along with the documentation. Toggle the printing of these values with the Comments check box in the print dialog box.



Printing

You can print the tooling lists, operations sheets, NC programs, and drawing for a part using Print from the File menu. You can also preview these documents with Print Preview. Some options may be grayed out in the dialog box if you haven't generated toolpaths and thereby created the lists and some of the graphics options. The files are all ASCII text files that print like a text document.

1. Select what you want to print in the Print Range section of the dialog box.
2. If there is a toolpath displayed and you want to print it, click *Print tool path*.
3. If you want to print so that the units of your part are honored, click *Print to scale*. This means that a 1 inch line segment will measure 1 inch on the paper. If your part is larger than your physical sheet of paper only a portion of your part will be printed. If *Print to scale* is unchecked, your drawing will be scaled to fit the paper.
4. Set your print quality. The specific options depend on your printer.
5. Set the number of copies to print.
6. Click OK.

Setup opens the standard Windows printer configuration window.

Print preview command (File menu)

Use this command to display the active document as it would appear when printed. When you choose this command, the main window will be replaced with a print preview window in which one or two pages will be displayed in their printed format. The print preview toolbar offers you options to view either one or two pages at a time; move back and forth through the document; zoom in and out of pages; and initiate a print job.

Print preview toolbar

The print preview toolbar offers you the following options:

Print Bring up the print dialog box, to start a print job.

Next Page Preview the next printed page.

Prev Page Preview the previous printed page.

One Page / Two Page Preview one or two printed pages at a time.

Zoom In Take a closer look at the printed page.

Zoom Out Take a larger look at the printed page.

Close Return from print preview to the editing window.

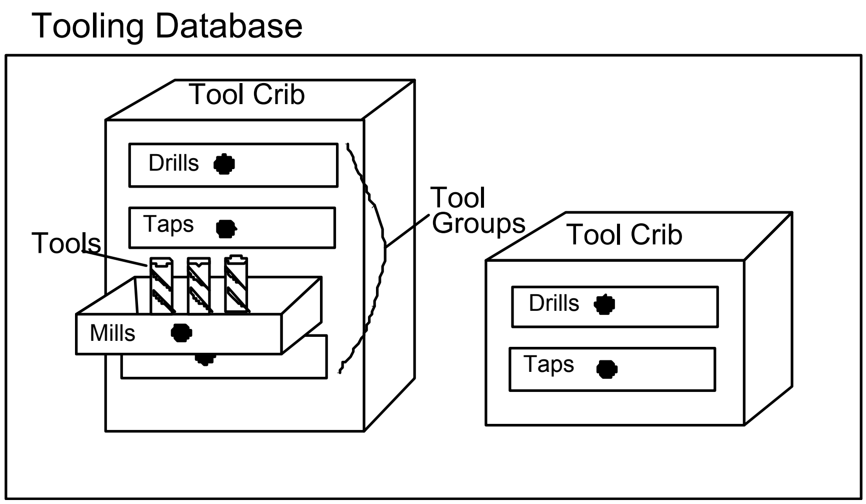
Print setup command (File menu)

Use this command to select a printer and a printer connection. This command presents a Print Setup dialog box, where you specify the printer and its connection.

Chapter 14 Tooling

Overview of tooling

For each manufacturing operation in created for the features of a part, a tool must be selected from the internal tooling database. This database is broken into separate tool cribs to represent individual collections of tools that your organization might have. This figure shows the structure of the tooling database.



FeatureCAM comes with two different built-in tool cribs. The *basic* crib is the default crib that contains tools that most shops will own. The *tools* crib is a large, crib containing thousands of tools. This crib is most often used as a source to copy from into custom tool cribs or into the basic tool crib. Only one tool crib is available at a time and all tools are selected for a part from only the current tool crib. Tool cribs contain individual tools and they are classified into tool groups such as drills, end mills and boring bars. You cannot create new tool groups, but you can create new tools to reflect the specific tools that your shop owns.

Each feature-type has rules for tool selection. By double clicking on a feature and clicking on the Tools tab, you can see the specific tool that has been selected to perform the operation. A feature's Tools tab is also a convenient place to override the tooling choice if you prefer another tool.

How to import tooling

1. Select *Tool Manager…* from the Manufacturing menu.
2. Click the *Import* button. The Tool Import dialog box comes up.
3. Enter the name of the file to import under *Import File*.
4. Select a toolcrib to add the new tools.
5. Select an option for handling new tools with the same name as existing tools.

6. Select *Overwrite* if you want the new tool to be copied over the old one. Select *Add 2nd Copy* to add the new tool under a different name. Select *Skip Tool*, to ignore any new tools with the same name as old tooling.
7. Click Import.
8. Click Close.

How to export tooling

1. Select *Tool Manager…* from the Manufacturing menu.
2. Click the *Export* button. The Tool Export dialog box comes up.
3. Enter the name of the file to export under *Export File*.
4. Select the crib to export.
5. In the Tool Groups list select the groups you want to export.
6. Use the button to select all groups. Use the to unselect all groups.
7. Click *Export*. Click *Close*.

The file created is tab-delimited text. You can edit the file in a spreadsheet if desired. This file can now be imported into FeatureCAM.

Previewing the automatically selected tool

To see the parameters of the tool that was selected for an operation:

1. Double-click on the feature to bring up the *Properties* dialog box.
2. Click on the operation in the tree view.
3. Click on the *Tools* page.
4. The selected tool is highlighted in the list and it has a "**D**" in the checkbox.

How to explicitly set a tool

1. Double-click the feature to bring up its Properties dialog box.
2. Click the operation in the tree view.
3. Check the checkmark next to the tool name in the table to select a specific tool

Tools attributes

Tools page

The table lists the recommended tools (marked with a "D") and other tools in the current tool crib that fit the tool selection criteria. Other tools can be selected from the table by checking the checkbox next to the tool name. The tools that are listed in the table are controlled by the filter settings.

Any tool that is highlighted in the table is displayed in the upper right-hand corner of the dialog box. This display can be panned and zoomed by clicking and dragging the left mouse button in the tool graphic window. Clicking the right mouse button in the tool graphic window and selecting Center all will center the entire tool and holder.

The tools that are listed in the table can be sorted by any column by clicking the title of the column.

 The Undo Override button will remove the tool override and return the selected tool to the recommended tool

 The New Tool button will allow you to create a brand new tool and add it to the current crib. The tool that is selected in the table will be used to fill in the initial values for the tool.

 The Tool Manager button brings up the tool manager.

 The Properties button shows you the details of the tool that is selected in the table.

Filtering criteria

The tools displayed in the table are filtered using the criteria listed above the table. The filter criteria are

- Orientation – select an orientation from among the icons. Select ✸ to see tools of all orientations.
- Presentation angle – Enter a number for the presentation. Select **Anything** to see tools with all presentation angles.
- Insert shape – Select an insert shape from among the icons. Select ✸ to see tools with all insert shapes.

Regardless of the filtering criteria, the automatically selected tool will remain in the list.

Tools usage tab

The tools usage tab contains information about spindle directions and the turret.

Tool: The name of the tool is listed at the top of the dialog box.

Spindle direction: The direction that the spindle will turn for the current operation using the tool.

Turret: Is the tool in the Primary or Secondary turret.

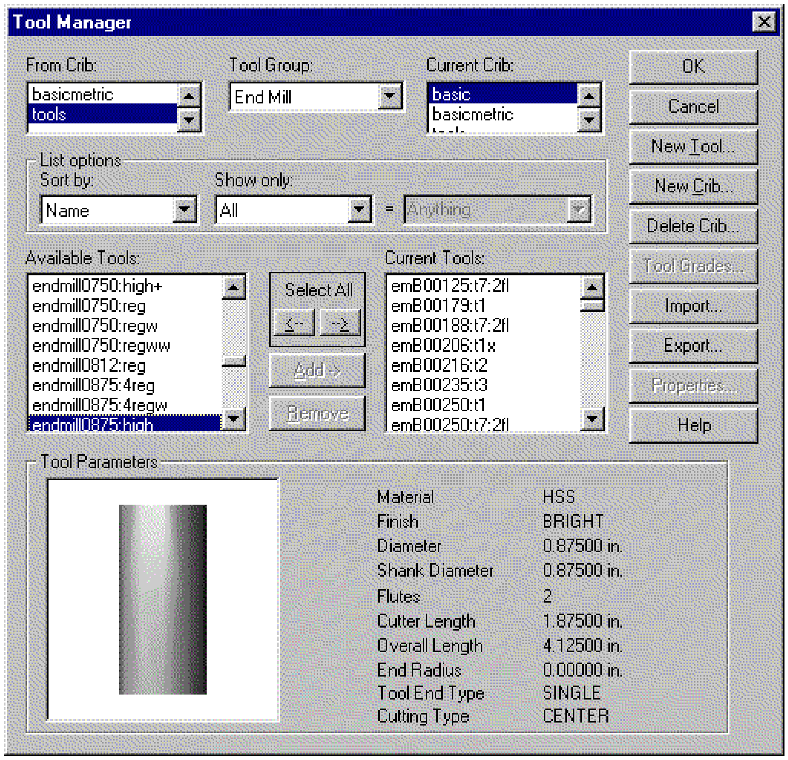
Turret direction: Which direction does the turret turn for tool changes. If set to Auto, the machine will determine the most efficient direction.

Tool manager

Use the Tool Manager to view, modify or add tools to a toolcrib. Tools are separated into groups. The Tool Group drop-down list contains the groups supported by FeatureCAM.

The Tool Manager only shows one type of tool at a time. The dialog box is arranged so it's convenient to move tools from one crib to another. In general you want to have the *tools* toolcrib listed as the *From crib* and the *basic* toolcrib listed as the *To crib*.

The fields of the *Tool Manager* dialog box are explained below.



Tool groups

Tool Groups list for toolcribs are separated into related groups, according to the type of manufacturing operation that class of tools typically performs. The following tool-group divisions are defined for FeatureCAM.

- Countersink
- Counterbore
- Ream
- Spotdrill
- Tap
- Twistdrill
- Boring Bar
- Chamfer Mill
- End Mill
- Face Mill
- Rounding Mill

From crib list

From crib list specifies the toolcrib from which you want to take tool definitions to copy to the toolcrib highlighted in the To Crib list.

Using Add, you copy the selected tool definition(s) in the Available Tools list (under the From Crib list) to the Current Tools list, thus building a set of tool definitions for the toolcrib highlighted in the To Crib list.

Available tools list

Available tools list shows the tools defined for each tool group within the toolcrib listed as the

From Crib in the Available Tools list.

Once you define a new toolcrib, you'll see the tools you defined for each tool group within that crib in the Available Tools list when you highlight that toolcrib's name in the From Crib list.

Current tools

Current tools shows tool definitions by tool group for the toolcrib highlighted in To Crib. (When you first access the Tool Manager, both From Crib and To Crib are the same because initially only one toolcrib exists: the pre-defined *tools* toolcrib.)

To crib

To crib is the toolcrib into which tool definitions are copied from Available Tools. The tool definitions listed in Current Tools are the tool definitions associated with the toolcrib highlighted in To Crib (for the tool group listed in the Tool Group drop-down list).

Immediately after you define a new toolcrib name and click OK, you see your newly defined toolcrib name selected in the To Crib list.

New tool

New tool creates a new tool definition for a particular tool group. The new tool definition will show up in the *Current Tools* list and consequently it will belong to the toolcrib that is listed in the *To crib* field of the *Tool Manager* dialog box. When you click *New Tool*, the New Tool dialog box comes up, showing the currently selected tool group's parameters. To define a new tool, you must provide values for all of the parameters defined for tool definitions within that tool group.

If you currently have a tool highlighted in either the *Available Tools* or *Current Tools* list, the *New Tool* dialog box will use parameter values for that particular tool definition for its initial values. This enables you to use values contained in existing tools as initial values for new tool definitions.

New crib

New crib defines a new tool crib. See Creating a New Toolcrib for more details on creating a new toolcrib.

Select all

Select all arrows are used to select an entire set of tool definitions for a particular tool group from either the Available Tools list (use the right arrow) or the Current Tools list (use the left arrow).

Add

Add includes the tool definition(s) highlighted in the Available Tools list to the Current Tools list. Available Tools lists tool definitions for the toolcrib highlighted in From Crib. Current Tools lists tool definitions for the toolcrib highlighted in To Crib.

Importing tooling

The import button brings up the Tool Import dialog box. This allows you to import tools from other FeatureCAM users. See *How to import tooling* for more information.

Exporting tooling

The export button brings up the Tool Export dialog box. This allows you to export tools so that you can share them with other FeatureCAM users. See *How to export tooling* for more information.

Remove

Remove deletes tool definitions selected in Current Tools from the toolcrib set in To Crib.

Tool grades

This button brings up the Turning Tools Grades dialog box.

OK

OK confirms the toolcrib modifications you have made since last entering this dialog box.

Cancel

Cancel negates the modifications you have made since last entering this dialog box, leaving the toolcrib definitions as they were before the dialog box was entered.

Tooling speed and feed overrides

In the tool manager you can set speed and feed overrides for a particular tool. Whenever this tool is used, the feed and speed values will be scaled by this percentage value. Select a tool and click Properties to reveal these settings.

Default tool registers

In the tool manager you can set the tool number, cutter comp register and offset register for the tool. These settings will be used for this tool whenever it is used to cut a part. Select a tool and click Properties to reveal these settings.

See also *Tool Mapping* on page 252.

Creating a new toolcrib

To define a new toolcrib, designate a unique name for the new crib. Click *New Crib* from the *Tool Manager* dialog box to open the *New Tool* dialog box where you name the new toolcrib. Click *OK* to create an empty crib. It automatically becomes the crib listed in the *To Crib* list so that you can easily add tools to the new crib.

If you highlight the toolcrib name in either the *From Crib* or *To Crib* list immediately after you define a new toolcrib name, notice that the *Available Tools* or *Current Tools* list is empty. It is empty because you haven't specified any tools for the new toolcrib yet.

How to delete a toolcrib

1 Select *Tool Manager …* from the Manufacturing menu.

2 Select the crib to delete as the Current Crib.

3 Click the *Delete Crib* button.

4 Confirm the deletion.

5 Click *OK*.

Creating new tools

1. Select the name of the destination toolcrib in the *To Crib* list.
2. Select the type of tool you wish to create in the *Tool Group* menu.
3. If there is a tool that is close to the new tool you wish to create, click the tool name in the *Available Tools* or the *Current Tools* list. The dimensions of this tool can be used as the initial values of the new tool.
4. Click New Tool.
5. Each tool has different parameters. The general parameters are shown in Tool groups.
6. Fill in the values for the new tool and click OK.

This tool is now available in FeatureCAM.

Adding tools

1. Select the name of the toolcrib that contains the tool definition in the *From Crib* list.
2. Select the name of the destination crib in the *To Crib* list.
3. Click the name of the tool you wish to add in the *Available Tools* list. You can select more than one tool definition at a time by pressing the ctrl key while making each selection, or you can select a tool group's entire tool list using the left *Select All* arrow.
4. Click *Add*.

Deleting a tool from a toolcrib

1. Designate the toolcrib containing the tool as the *To Crib*.
2. Select the name of the tool in the *Current Tools* list.
3. Click *Remove*.

Tool mapping

Tool Mapping is where you change the tool slot assigned to the selected tool. You can change the cutter comp. offset register for any tool here too.

The *Tool Map* dialog box has these options:

- **Same** sets the cutter comp. offset registers for all tools to the value of their tool slot number.
- **Sort by tool** sorts by the tool numbers for the Tool Mapping dialog box.
- **Reset** returns all tool slot numbers and cutter comp offset registers to their initial values.
- **Cancel** rejects the changes you have made to tool numbering
- **OK** accepts the changes you have made to tool numbering.

The **Save in Crib** button will permanently assign the tool number with the tool in the database. The tool will then be locked in this position for all parts that use the tool.

Note: if you do not use the Save in Crib feature, then FeatureCAM may renumber your tools if you make a change to your part.

The table in the middle of the dialog box contains a row for each tool currently involved in the manufacturing process plan. The following information is displayed for each tool:

Tool is the current tool slot number for that tool. Note that tools can occupy the same slot.

Diameter offset register number is the diameter cutter comp. offset register for that tool.

Length offset register number is tool length offset register. Most lathe controllers have a single register that contains the length and diameter offset values. In this case, the Length offset register number is the important field to set in FeatureCAM.

ID is the tool ID register for the tool. This is a seldom used field that is used by Bridgeport lathes and occasionally for Cincinnati Machines lathes

Name of the tool.

Instructions for changing tool number or cutter comp

1. Press Tool Map in the Post Options dialog box.
2. Click the row of the table with the tool you want to change. The tool slot number and cutter comp offset register appear in the Tool and Comp text boxes.

NOTE: If you want to change the cutter comp offset register and the Comp text box is dimmed, click the *Same* check box. This box indicates that the tool slot number and the cutter comp. offset register have the same value.

3. Enter the new values in the *Tool* text box and the *Comp* text box. Click *Set*.

How to put two tools in the same tool slot

There are situations in turning when two different tools are loaded in the same tool slot. For example, two drills may be loaded with one facing the main spindle and the other facing the subspindle. Use the following steps to implement this tooling setup.

1. Program the part as if the tools are in separate tool slots.
2. Open the Tool mapping dialog box.
3. Set the *Tool number* of the two tools to the same number.
4. Set the *Length offset register number* to different values for the two tools in the same tool slot.

How to use an insert drill to drill and bore in the same program

1. Create the hole feature and override the tool for the drilling operation to be an insert drill.
2. Create the bore feature and override the tool to be the same insert drill.
3. If you view the tools in the Tool mapping dialog box, you will see that there are two drills listed in the same tool slot, but they have different *Length offset* registers.

Tool life management

Tool life management allows you to limit the use of a tool and automatically switch to another tool when that limit is reached. It is useful when cutting hard material that may wear out a tool during a single program run. The tool mapping dialog box displays the number of holes that will be cut by each drilling tool and the number of minutes that each milling tool will be used during a single run of the NC program.

To specify tool life information for a tool:

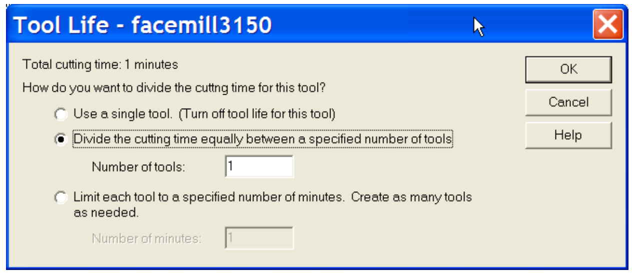
1. Select the tool in the tool mapping dialog box.
2. Click the *Tool life* button.
3. The Tool life dialog box is displayed.

Restrictions for tool life management

1. Tool life management is active for multiple fixture parts and tombstone parts.
2. For milled parts, tool life management is only active for the creation of a single program. Therefore it is active for single setup parts, or 4 or 5 axis indexed parts. For 4 and 5 axis parts, if you have *setup dominant* checked, you must also check *generate single program* to enable tool life management. See the index tab of the stock dialog for more information on indexing. If you have a multiple setup milling part without indexing, you will have to uncheck all but one setup in the Part View to enable tool life management for this setup.
3. It is not active for turning, turn/mill or wire EDM parts.
4. Tool life only applies to the use of a tool during the running of a single program. Tool life information is not stored permanently in the tooling databases.

Tool life dialog box

Use this dialog box to limit the use of a tool during the running of a single program.



The choices for each tool are:

Use a single tool – Select this option to disable tool life management for this tool. A single tool will perform all operations that are scheduled for this tool slot.

Divide time equally – This option will divide the work of the tool, equally among additional tools. For drills, the number of holes drilled will be divided among multiple tool slots. For mills, the milling minutes will be divided the same way.

Limit each tool – For this option you specify a limit (in number of holes for drills, milling minutes for mills) and additional tools are assigned when the limit is reached.

Additional tool slots are then assigned to the same tool and the work is divided among those slots. When the NC code is created, a tool change is inserted when the limit is reached.

Removing tool life restrictions on a tool

1. Select Tool mapping from the Manufacturing menu.
2. Select the tool and click on the Tool life button
3. In the Tool life dialog box select *Use single tool* and click *OK*. The additional tool slots that were assigned to this tool are deleted.

Milling tools

Tool Groups list for toolcribs are separated into related groups, according to the type of manufacturing operation that class of tools typically performs. The following tool-group divisions are defined for FeatureCAM. The following figures illustrate the parameters that define all of the tooling groups.

Common milling tool parameters

All milling tools have the following parameters.

Default Tool Registers

If these values are set permanently on a tool, it will fix these values for every time a tool is used. If tools have the same permanent setting, FeatureCAM will change the tool registers for the second tool.

Name is a string that identifies the tool.

Measure indicates the units that are used for reporting the tool’s dimensions. Check the *inch* checkbox for inches or uncheck the box for millimeters.

Material indicates what the tool is made out of. This information is important when calculating the feeds and speeds.

Finish is the coating or finish on the tool. This information is also used in feed/speed calculations.

Exposed length is the amount of the tool that will stick out of the holder if the holder is simulated.

Milling tool overrides tab

Use the default tool registers if you want to permanently assign tool registers for this tool every time it is used. If the tool number is 0, then this indicates that no default value is assigned to this tool and that FeatureCAM will automatically assign a tool number when the tool is used. If you want to change these values only for a particular part, use the tool mapping dialog box.

The picture in the upper right-hand corner shows the tool. You can use the pan and zoom function by right-clicking on the image and dragging the mouse.

Tool Number is the current tool slot number for that tool. Tools can occupy the same tool slot.

Diameter offset register number is the diameter cutter comp. offset register for that tool.

Length offset register number is tool length offset register. Most lathe controllers have a single register that contains the length and diameter offset values. In this case, the Length offset register number is the important field to set in FeatureCAM.

ID is the tool ID register for the tool. This is a seldom used field that is used by Bridgeport lathes and occasionally for Cincinnati Machines lathes.

% Speed override is the percentage to adjust the speed value whenever this tool is used. 100% means that the values chosen by FeatureCAM or overriden by the user will not be scaled.

% Feed override is the percentage to adjust the feed rate whenever this tool is used. 100% means that the values chosen by FeatureCAM or overriden by the user will not be scaled.

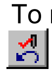
Comments are random comments that can be associated with a tool. A post processor can be configured to output these comments.

Milling tool holders tab

This tab allows you to see the tool holder that has been automatically selected for the tool and to permanently assign a holder to the tool.

The *Tool name slot* shows the name of the tool. *Current spindle* indicates which spindle type matches the current holder.

Tool holders are shown in the table at the bottom of the dialog box. Two radio buttons control which holders are displayed in the table. Select *Only show holders that match the current spindle* if you want to eliminate holders that done fit in the current spindle. Select *Show all holders* to show them all.

The holder that was automatically selected for the tool is shown with a "D" in the check box. To explicitly select a holder to use with this tool anytime the tool is used, check the checkbox next to the tool name. To remove an association of the tool with a holder, click the *Undo holder override* button .

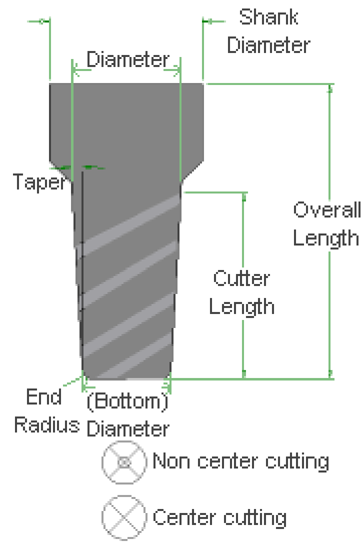
You can also create new holders or modify a holder by using the following buttons.

 Create a new tool holder for the current spindle. The dimensions of the current tool holder are used as initial dimensions.

 Displays the properties of the currently selected tool holder in the table and allows you to alter them.

Endmill tools

Endmill tools are used to represent flat end mills, ball-end mills, bull-nose mills and tapered end mills. The basic parameters are shown on the right.



For ball end tools, enter the diameter and click the *ball end* checkbox and the radius will be calculated.

For flat end tapered tools there are three different ways to specify the taper angle, cutter length and diameters

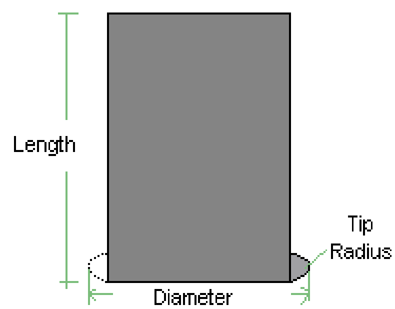
Enter the Taper Angle, Diameter (which is the diameter of the tool at the top of the taper) and the Cutter Length.

Enter the Taper Angle, click the Diameter at Bottom checkbox, enter the Bottom Diameter and enter the Cutter Length.

Enter the Taper Angle, click the Diameter at Bottom checkbox, enter the Diameter (which is the diameter at the bottom of the taper) and Click the Compute from Shank button to have the Cutter Length computed for you.

Boring bars for milling

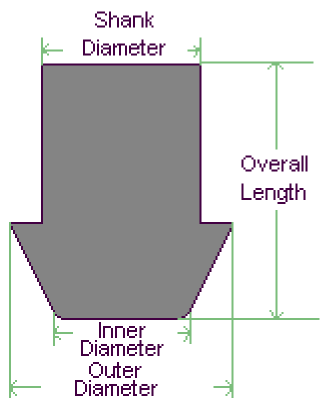
Boring bars are used in milling for boring operation for a hole or for a step of a step bore feature. The tip radius is only taken into account for step bores. It is assumed that the user has an adjustable boring bar. For boring operations, if there is not an appropriate tool in the tool crib FeatureCAM will create a tool with the name “user_adjust…”.

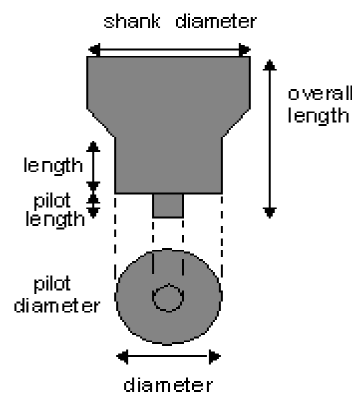


Chamfer mills, and Counter bores

Chamfer mills are used for chamfer features or for chamfering large diameter holes.

Counter bore tools are used for the counterbore operations of counter bored holes.



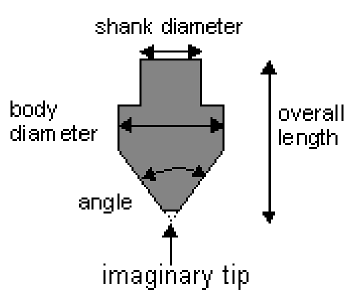


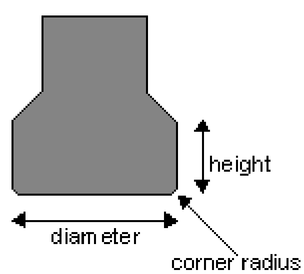
Counter sinks and Face mills

Counter sink tools are used for counter sink operations on holes. Note that the Overall Length is measured down to the imaginary sharp tip of the tool.

If you are using a counter sink to mill a chamfer feature and your tool has a radius or flat rather than a pointed tip, then you must account for this difference. If you touch off your tool on the rounded or flat tip, you must add an extra amount to your touch off point to move it to the imaginary sharp tip.

Face mills are used for facing features.



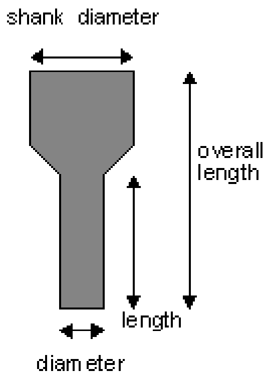


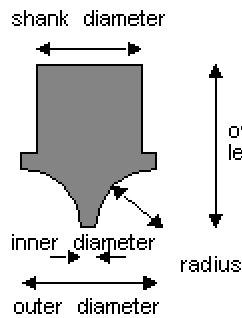
Reams Rounding mills and Spotdrills

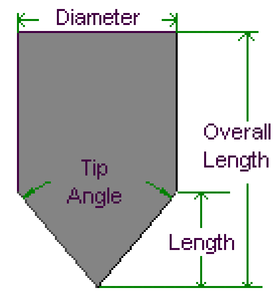
Reams are used for reaming operations on holes.

Rounding mills are used for round features.

Spotdrills are used for spotdrilling starter holes. Note that center drills are preferred for spotdrill operations.

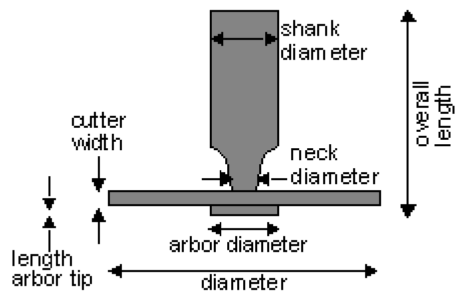






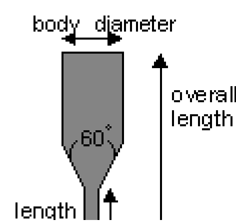
Side mills

Side mills are used for OD and ID grooves. The *Slitting saw* parameter is only used as documentation on the tool.

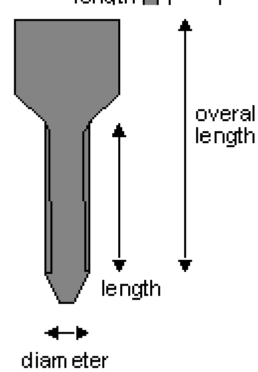


Center drills

Center drills are used to drill starter holes. They are preferred over spot drills for spotdrilling operations. Spotdrill tools can also be used for spotdrill operations.



Taps



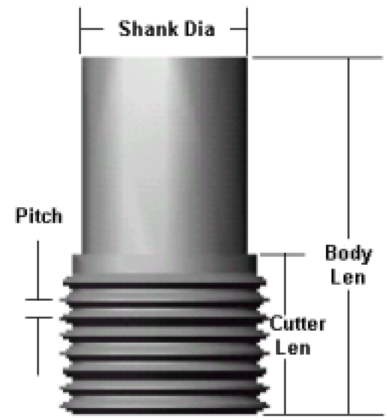
Taps are used to tap holes. For inch tools, you specify the threads per inch (TPI), and for metric tools, you specify the pitch of the threads.

Spiral-style taps have twisted flutes while *Gun*-style taps have straight flutes. *Bottom* taps have a flat bottom, while *plug* taps come to a point. Bottom taps have a flat bottom, while point taps come to a point. If a bottom tap is required for a blind hole, a warning is issued in the operations sheet. Regardless of the type of tool used for tapping, the hole must allow enough clearance for the tap. If the hole violates this rule, an error message is generated when you enter the hole dimensions.

Thread mills

Thread mills are used for OD and ID thread milling operations.

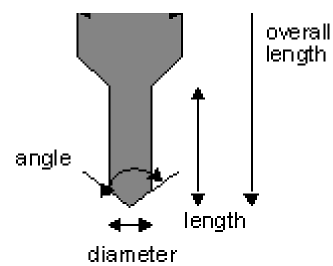
For single point tools, set Max Pitch equal to Cutter Length.



Twist drills

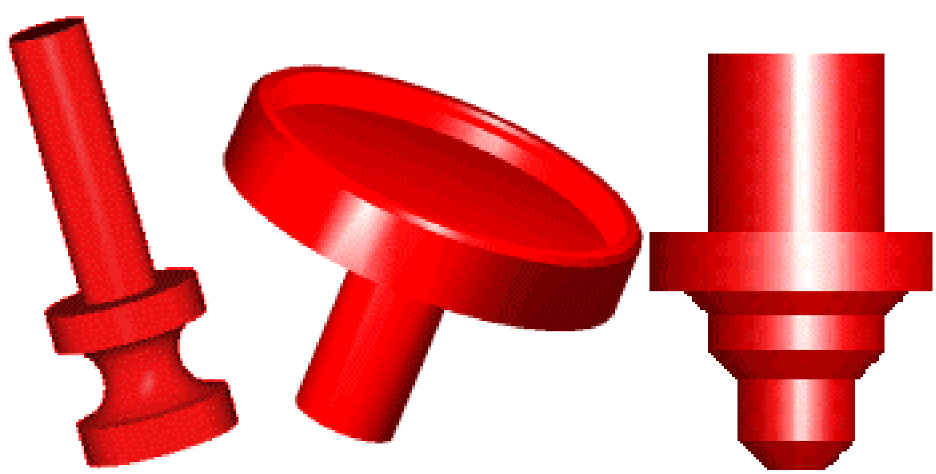
Twist drills are used for drilling operations.

For insert drills, click the Insert Drill checkbox and enter the Insert Depth instead of the Angle. If an insert drill is initially chosen for a drilling operation, the spotdrill operation will be omitted



Form Tools

The FeatureCAM toolcribs contain thousands of industry-standard tools in a wide variety of types. Custom-shaped form tools are not included in the default tool cribs, but you can create these tools and use them to cut features. The pictures below show examples of form tools.

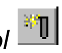


Concave Cutter | Tripan Cutter | Port-Entrance Tool

Using these tools does not change the toolpaths generated for features, but the 3D simulation will simulate proper shape of form tools. Form tools are never automatically selected, but they can be manually be selected by the user.

Form tools have unique shapes, but their type must be endmill, twistdrill or sidemill. If you want to use the form tool to perform a milling operation it must be an endmill form tool. Drilling operations can be performed with endmill form tools or twistdrill form tools. OD/ID grooves can only be performed with sidemill form tools.

Creating a form tool

1. Create a tool using either the *New tool* button of the tool manager or the *New tool*  button of the tools page.

2. Make sure you are creating an endmill, sidemill or twistdrill by having a tool of that type selected before pressing the button.

3. Click *Use curve to define tool shape.*

4. A drop down box is displayed. Select the name of the curve you would like to use. Note that only curves that meet the requirements for form tools will be listed.

5. Click *OK*.

When you specify the shape of a tool with a curve, many of the dimensions normally entered for a tool are no longer used, but the *diameter* dimension is still critical for milling operations. The diameter is still used calculating stepovers and generating the paths.

Requirements for form tool curves

1. The curve must lie in the XZ plane of the stock axis.

2. The endpoint of the curve must be on the origin (0,0,0).

Displaying a profile of an existing tool

When creating form tools, it is often helpful to get the profile of an existing tool to use as a reference. Use the following procedure to get a tools profile curve:

1. Bring up the tool manager.

2. Double click on the tool to bring up its property dialog box.

3. Click the *Paste copy of curve* button.

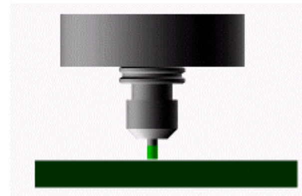
4. Click *Cancel* in the Property dialog box.

Using a form tool or insert drill for drilling operations

When drilling with a standard twistdrill, extra depth is added to the operation to account for the drill tip geometry. When a form tool or insert drill is used for drilling, no extra depth is added automatically. You can still use the Drill depth attribute to manually add extra depth if desired.

Milling tool holders and spindles

During toolpath centerline or 3D toolpath simulation, the tool is always displayed. This allows you to see the tool moving relative to the part and with the 3D simulation to detect gouging of the part with the tool. For some parts, that have closely spaced features or that approach the part from an angle, it is possible that the tool holder or spindle might gouge the part. In these cases, you can simulate the toolpaths while displaying the tool, tool holder and spindle to ensure that you can clear the part properly. The figure on the right shows a 3D simulation in which the spindle and tool holder are displayed during toolpath simulation



Displaying spindles or tool holders during simulation

1. Select the *Simulation…* option from the *Options* menu.
2. To display the tool holders, check the *Show Holder* checkbox.
3. To display the spindle, check the *Show Spindle* checkbox. Note that you must also check the *Show Holder* checkbox to display the spindle.
4. Click *OK*.

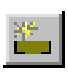
Setting the length of tool extending past the holder

Each tool has a parameter called *Exposed Length*. This is the length of the tool that will stick out past the holder. The properties of a tool can be accessed in the tool manager or selecting tools tab of a feature's property dialog box.

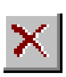
Spindles and tool holders dialog box

This dialog box displays the current spindle and the holders that are defined for it. To change the current spindle, select the spindle from the *Current Spindle* list. Click on a tool holder to display the spindle and tool holder combination.

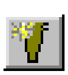
The spindle buttons are:

 Create a new spindle. The dimensions of the current spindle are used as initial dimensions.

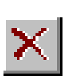
 Displays the dimensions of the current spindle and allows you to alter them.

 Delete the current spindle and its tool holders.

The tool holder buttons operate on the selected tool holder.

 Create a new tool holder for the current spindle. The dimensions of the current tool holder are used as initial dimensions.

 Displays the properties of the current tool holder and allows you to alter them.

 Delete the current tool holder.

Creating or modifying tool holders

1. Select Spindles and Toolholders… from the Manufacturing menu.
2. Select the desired spindle as the Current Spindle. The tool holders for that spindle are listed.
3. To modify a tool holder, double-click on the tool holder name or click on the tool holder name and click the Properties  button. The toolholder properties dialog box comes up.
4. To create a new tool holder, click on an existing tool holder name (the dimensions of this holder will be used as the initial dimensions for your new holder) and click the Create new toolholder {bmc btn-new-holder.bmp} button. The toolholder properties dialog box comes up.

Creating or modifying spindles

1. Select Spindles and Toolholders… from the Manufacturing menu.
2. To modify a spindle, select the desired spindle as the Current Spindle and click the Properties  button. The spindle properties dialog box comes up.
3. To create a new tool holder, select an existing spindle as the Current Spindle (the dimensions of this spindle will be used as the initial dimensions for your new holder) and click the Create new spindle  button. The toolholder properties dialog box comes up.

Toolholder properties dialog box

Name: Each toolholder must have a unique name

Measure: Click the *Inch* check box if the toolholder dimensions are in inches. For millimeter dimensions leave this checkbox unchecked.

Holder type: Holders are either *Endmill* or *Collet*. The major difference between these two types of holders are that Collets are adjustable to fit a range of tool diameters.

Tool groups: This button displays a list of tooling types that will use this holder.

Use curve to describe holder shape: If you want to make custom holder shape, click this option and select a curve in the drop down menu that describes the shape. This curve must be in the XZ plane with one endpoint at the origin.

Paste copy into graphics window: Click the *Paste* button to copy into the graphics window a set of lines and arcs for the current holder shape. This can be useful as a reference if you are going to draw a custom holder shape.

Tip Diameter: This is the outer diameter of the holder at the tip for collets. For convenience, you can set the Tip diameter for endmill holders to *Based on tool dia* so that the holder is scaled to fit a tool.

Tool Diameter: For collets this can be specified as a minimum and maximum value, to accurately reflect actual collets, or it can be set to *Fit any tool* so that the diameter is adjusted to fit any tool with a diameter less than the *Tip Diameter.*

Dimensions The other dimensions of the holders are shown in the figure on the right for endmills and in the left-hand figure for collets.

Tool holder selection

Toolholders are selected from the set of *Defined toolholders* in the Spindles and toolholders dialog box. In general, if a tool holder that exactly matches your tool cannot be found, FeatureCAM will scale an existing toolholder to fit the tool. If you are just looking for an approximate toolholder shape, this scaling should create sufficient simulation of the holder.

Selection details

Certain operations are associated with collets and others with the endmill holders. See Creating or modifying spindles for how to change this association.

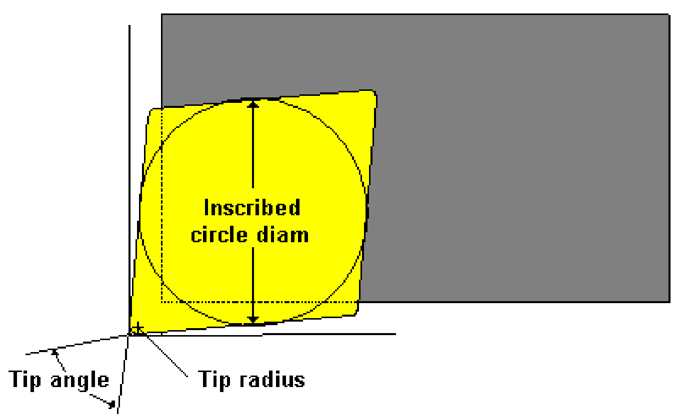
Once the holder type is determined, tool selection is based solely on matching the diameter of the tool with the tool diameter parameter of the holder. For endmill holders, the holder with the smallest tool diameter that is greater than or equal to the tool's actual diameter will be preferred. For collects, the tool's diameter must be within min. and max. tool diameter of the collet. For both the endmill and collet holders, if a match cannot be made from the existing toolholders, an existing holder is scaled by adjusting the tool diameter and tip diameter.

Currently, there is no way to explicitly associate a tool with a holder. If two holders exist with the same tool diameter, there is no way to determine which holder will be selected for that tool diameter.

Turning tools

Insert tab

The insert tab describes the characteristics of the tool's insert. The specific values contained in this tab are dependent on the type of tool.



I

Inscribed circle diameter

The *Inscribed circle diameter* is the diameter of a circle that fits inside the insert shape.

Tip angle

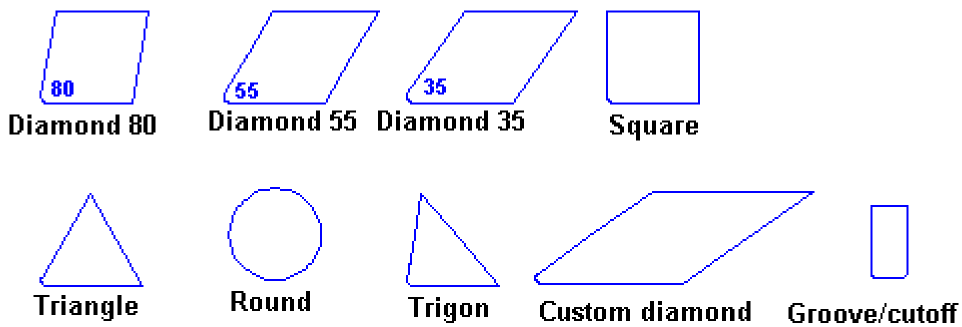
The included angle of the insert.

Tip radius

The radius of the cutting tip of the insert.

Insert shape

The valid insert shapes are listed. For the custom diamond shape, the user enters the tip angle.



Grade

The grade of material that the insert is made out of.

Measure

The unit of measure of the insert. Click the Inch checkbox to specify the insert in inches. Uncheck the Inch checkbox to specify the insert in millimeters.

Name

The name of the insert. The name must be unique among all the tools in the crib.

New material button

Click this button to create a new material name. To create a new material from the Tools Properties dialog box:

1 Click the *New material* button. The Turning Tools Material dialog box comes up.

2 Click the *New* button.

3 Enter the new material name and click *OK*.

4 Click *OK* in the Turning Tools Material dialog box.

You will still have to define feed/speed tables for your new material.

Width

The width along the Z-axis of a grooving/cutoff tool.

Max TPI

The maximum threads per inch that the tool can cut.

Min TPI

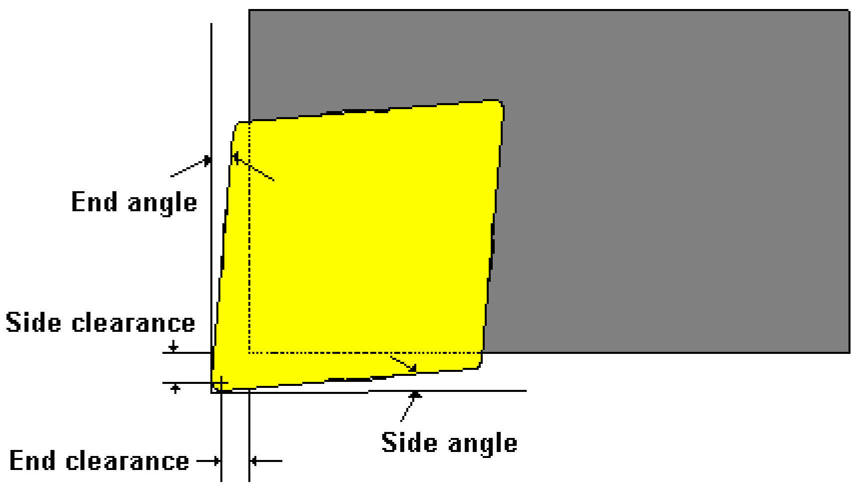
The minimum threads per inch that the tool can cut.

Max Diameter

This is the maximum diameter that a bar puller can accommodate.

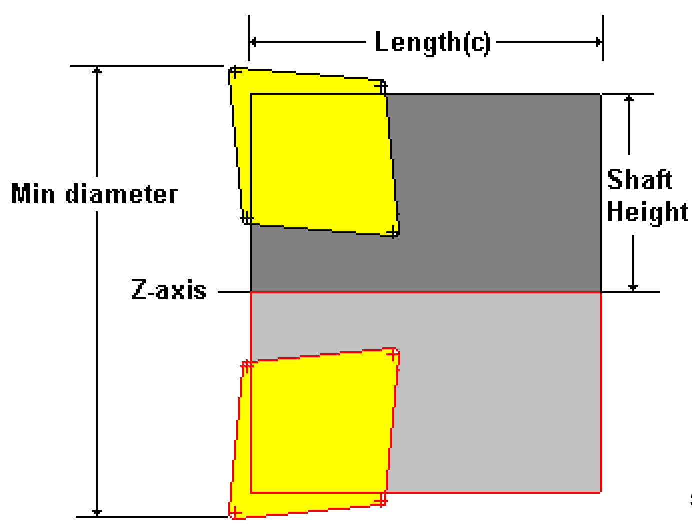
Holder tab

The holder tab describes the characteristics of the tool holder and how the insert is oriented relative to the holder. The specific values contained in this tab are dependent on the type of tool.



End angle

The angle, off of vertical, of the end of the insert.



End clearance

The distance from the end edge of the insert to the holder.

Side angle

The angle of the side of the insert.

Side clearance

The distance from the side

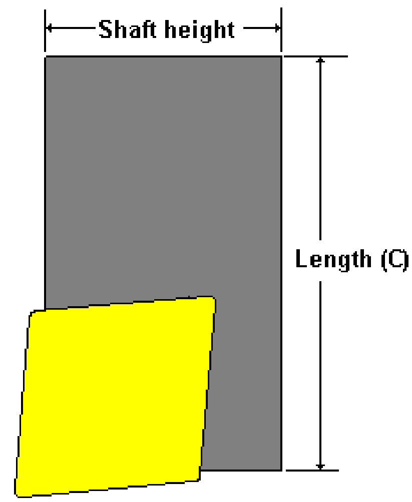
edge of the insert to the holder.

End cut

A classification of a tool that indicates that the tool will cut in a direction parallel with the length of the holder

Min diameter

The min diameter is the smallest diameter that will accommodate an ID tool.



Length (C)

The length of the holder.

Shaft Height (A)

The height of the holder.

Side cut

A classification of a tool that indicates that the tool will cut in a direction perpendicular with the length of the holder.

Cut depth

The maximum depth of cut for grooving tools.

Holder Type

Threading and groove/cutoff tools have additional subtypes listed under the Holder Type parameter.

For threads possible holder types are:

- OD Threads – Enter this value for outer diameter threading tools
- ID Threads – Enter this value for inner diameter threading tools

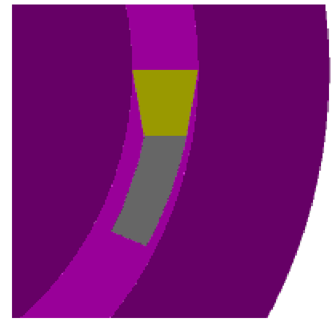
For grooves possible holder types are:

- OD Groove – Enter this value for outer diameter grooving tools
- ID Groove – Enter this value for inner diameter grooving tools
- Face Groove – Enter this value for face grooving tools
- Cutting – Enter this value for cutoff tools

Head width

This is the width of the head of a bar pull or bar feed tool.

Min plunge diameter and Max plunge diameter



Many face groove tools have curved holders. Due to the curvature of the holders the tools have a limited set of diameters at which they can plunge. The figure on the right illustrates the curved shape of the supporting holder. These two diameters are the minimum and maximum diameters between which the tool can plunge.

The tool inside edge of the groove must be between the *Min plunge diameter* and *Max plunge diameter* if the groove is being cut in the positive direction. The outside edge must be between these two diameters if the groove is being cut in the negative direction.

Holder drawing tab

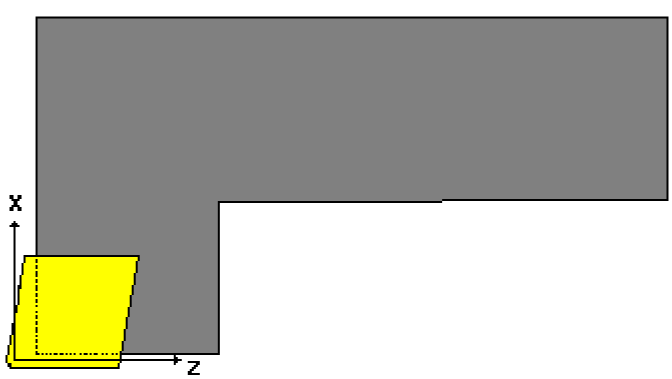
The holders that are created automatically are simple rectangular shapes that are created based on the holder dimensions.

You can also create custom holders by modeling a curve that more accurately reflects the holder's shape. The main motivation for User Defined Holders is that the simulations will be more accurate enabling you to visually verify holder collisions.

Creating holder curves

1. If you want to display the insert and current holder shape for a particular tool, see Displaying holder and insert geometry in the graphics window. This enables you to have a reference to draw on top of.
2. Regardless of the eventual orientation of the holder in the machine, draw the holder curve in the XZ plane with the length of the tool holder in the +Z direction and the leading edge of the insert at the origin. The curve you draw must be a closed curve.
3. Name the curve appropriately so that you can remember is later when you associate the curve with a user defined holder.

Custom holder orientation



Associating curves with user defined holders

Remember that you must create the curve first.

1. Bring up the Tools Properties dialog box for the desired tool.
2. Click the *Holder Drawing* tab.
3. Click the *Use custom drawn holder* radio button.
4. Select the name of the curve from the drop-down list.
5. Click the *Set selected curve* button.
6. Click *OK*.

The curve is saved in tooling database independently from the part file. Any subsequent use of this tool in any part file will now use the custom holder.

Bringing up a tool in the Tools Properties dialog box

Via the Tool Manager:

1. Bring up the Tool Manager by selecting *Tool manager …* from the *Manufacturing* menu.
2. Select the desired tool group.
3. Double-click on the name of the desired tool.

Via a feature that utilizes the desired tool:

1. Double-click the feature.
2. Click on the operation that uses the tool.
3. Click the Tools tab.
4. Double click on a tool name.

Displaying holder and insert geometry in the graphics window

1. Bring up the Tools Properties dialog box for the desired tool.
2. Click on the *Holder Drawing* tab.
3. Click the *Paste to part file* button. The holder and insert geometry are pasted into the graphics window as two separate curves.

Orientation tab

On the Orientation tab, you select the orientation of the holder in the machine and the handedness of the tool. To set the options on this tab:

1. Select an orientation by clicking the button at the end of the tool that indicates the desired orientation. Remember that nothing is implied by the shape of the inserts in the icons. The actual insert of the tool can be any shape.
2. Select the handedness of the tool from either Right Hand or Left Hand.

Prog pt. tab

On this tab you set the point of the insert that is actually programmed. You can set both the X Programming Pt. and the Z Programming Pt. If they are both set to 0.0 then the center point of the tip arc is programmed. In this case it is expected that you will perform insert radius compensation at the machine tool. Note that you should also set the default Tool Programming Point attribute to Tool Tip Center.

If you wish to perform insert radius compensation in FeatureCAM, set X Programming Pt. and Z Programming Pt to the radius compensation values provided with your tool. Note that you should also set the default Tool Programming Point attribute to Tool Tip Edge.

Turning tool overrides tab

Use the default tool registers if you want to permanently assign tool registers for this tool every time it is used. If the tool number is 0, then this indicates that no default value is assigned to this tool and that FeatureCAM will automatically assign a tool number when the tool is used. If you want to change these values only for a particular part, use the tool mapping dialog box.

The picture in the upper right-hand corner shows the tool. You can use the pan and zoom function by right-clicking on the image and dragging the mouse.

Tool Number is the current tool slot number for that tool. Tools can occupy the same tool slot

Diameter offset register number is the diameter cutter comp. offset register for that tool.

Length offset register number is tool length offset register. Most lathe controllers have a single register that contains the length and diameter offset values. In this case, the Length offset register number is the important field to set in FeatureCAM.

ID is the tool ID register for the tool. This is a seldom used field that is used by Bridgeport lathes and occasionally for Cincinnati Machines lathes.

% Speed override is the percentage to adjust the speed value whenever this tool is used. 100% means that the values chosen by FeatureCAM or overriden by the user will not be scaled.

% Feed override is the percentage to adjust the feed rate whenever this tool is used. 100% means that the values chosen by FeatureCAM or overriden by the user will not be scaled.

Comments are random comments that can be associated with a tool. A post processor can be configured to output these comments.

Tooling databases

The tooling database defines the set of tools from which FeatureCAM selects tools to perform manufacturing operations. These tool sets are called cribs. FeatureCAM comes with two standard tool cribs, the *tools* toolcrib and the *basic* toolcrib. The *tools* toolcrib is a comprehensive toolcrib that contains more tools than your shop probably owns. The *basic* toolcrib contains a smaller set of tools such as HSS end mills and common drills. By default FeatureCAM is set to use the *basic* toolcrib. You should modify the cribs to reflect the tools your shop has. You might also want to construct some cribs of commonly used tools that you use over and over again. This can simplify setting up for making the part.

The currently active toolcrib is displayed in the status bar. To change which toolcrib you are currently using, click on the crib name in the status bar. A menu of cribs is displayed you can select from. .

To create or modify a toolcrib, you have to have a part file open. Then access the Tool Manager dialog box by selecting *Tool Manager* from the *Options* menu.

Initializing FeatureCAM databases

The tooling and feed/speed databases are created using the INITDB Initialization program in the FeatureCAM. This is the program that is run the first time that you run FeatureCAM to create your initial database. You may also want to use INITDB for the following reasons:

- Adding default tools and feed/speed tables to the database
- Recreating tooling and feed/speed databases if they become corrupt

Adding default tools and feed/speed tables to the database

Each tool is identified by a tool name and each feed/speed table is defined by a combination of the stock material name, tool material and tool grade. Any tool or feed/speed table that you add to the database will remain even if you rerun the INITDB program. INITDB will not overwrite any existing tools or feed/speed table in the database. The only way to remove them is to explicitly delete them in the Feed/speed table dialog box or the Tool manager.

The two most common reasons for rerunning INITDB are:

1. Restoring default tools or default feed/speed tables that you have deleted.
2. Adding tools specified in other units. You may have elected to only load the inch tools the first time and you may now want to add the metric tooling.

To perform these tasks:

1. Exit FeatureCAM if you are running it.
2. Run INITDB from the FeatureCAM group in the Start menu.
3. Select the units of tools you want to add. Remember FeatureCAM will not alter any items that you changed or delete any items that you added.
4. Click OK.
5. If you added tooling of both units, you will be asked which unit you will most often use to model parts.

Recreating tooling and feed/speed databases if they become corrupt

If your tooling or feed/speed database becomes corrupt, you can recreate the databases using the INITDB program. **Remember that this procedure will erase any changes you have made to existing tools or feed/speed tables and will delete any custom tooling or new feed/speed tables that you have created. Contact your FeatureCAM support person before performing this task.**

To remove your database and recreate it:

1. Exit FeatureCAM if you are running it.

2. Run Windows Explorer.
3. Open the directory in which you installed FeatureCAM.
4. Open the database directory.
5. Click in the right-hand contents window.
6. Click Select All from the Edit menu.
7. Press the Delete key on the keyboard.
8. Run INITDB from the FeatureCAM group in the Start menu.
9. Select the units of tools you want to add. Remember FeatureCAM will not alter any items that you changed or delete any items that you added.
10. Click OK.
11. If you added tooling of both units, you will be asked which unit you will most often use to model parts.

Chapter 15
Feeds and Speeds

Feed and speed values are calculated automatically for each operation based on FeatureCAM's build-in feed/speed databases. You can also explicitly set the feed or speed value for any operation.

Overview of feeds and speeds

Each feature is automatically broken down into a collection of operations. Feeds and speed settings for each operation are automatically calculated. To view the recommended feed or speed value for an operation, click on the operation in the tree view and then click on the Feed/speed tab for turned features or the Tools tab for milling operations.

The recommended feed and speed values are extracted from a built-in database in FeatureCAM. The combination of the stock material, the tool material and the manufacturing operation are used to find the suitable values in the FeatureCAM feed/speed tables.

You can customize this database by changing the values for an existing table or by creating a new table for an additional material. By customizing the database in this way, you are ensuring that FeatureCAM will make future decisions about feed and speed values just as you would.

You can also explicitly set the feed rate or spindle speed for an individual operation, but these changes will only affect a single feature in a single part file.

Setting a feed or speed value for a milled operation

1. Double-click on the feature to bring up the Properties dialog box.
2. Click on the operation in the tree view.
3. Click on the Tools tab.
4. Delete the current speed or feed value
5. Type in the desired speed or feed value
6. Click *OK* to set the value and remove the dialog box or click Apply to set the value and leave the dialog box for further input.

Feed

Feed displays the feed of the tool that performs the currently selected operation. Override the setting by typing the new value directly in the display field. The Override check box automatically selects itself.

The feed is calculated from the speed and feed table for the stock material. You can change the material's table to affect speeds and feeds for that material in all cases by selecting Feeds and Speeds Tables in the Options menu. Or you can set global percentages to change feeds and speeds for all parts by selecting Machining Attributes in the Options menu.

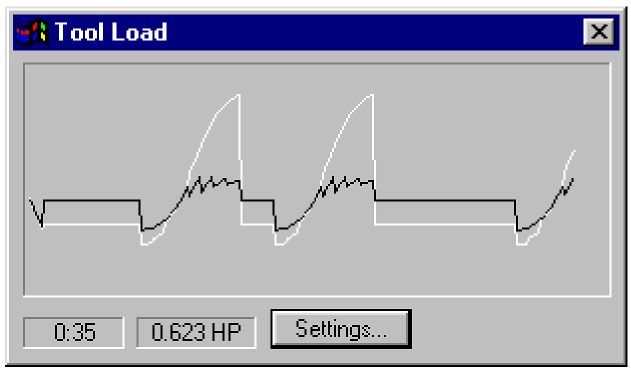
Speed

Speed displays the speed of the tool that performs the currently selected operation. Override the setting by typing the new value directly in the display field. The Override checkbox automatically selects itself.

The speed is calculated from the speed and feed table for the stock material. You can change the material's table to affect speeds and feeds for that material in all cases by selecting Feeds and Speeds Tables in the Options menu. Or you can set global percentages to change feeds and speeds for all parts by selecting Machining Attributes in the Options menu.

Feed optimization

Feed optimization is an optional process that evens out the tool load by adjusting a program's feedrates. This optimization is performed so that the horsepower required for the cuts approximates a *target horsepower*. In the graph shown below, the initial feedrates are shown in white. The optimized feedrates are shown in black. Notice how the black curve removes the spikes in the tool load.



To optimize the feedrates of your program:

1. Generate toolpaths
2. Set the target horsepower milling attribute for the operations you would like to optimize.
3. Select *Feed optimization* from the *Manufacturing* menu.
4. Set the parameters appropriately. These parameters are documented below.
5. Click OK.
6. For complicated parts this task can take several minutes.

Note that if the NC code is regenerated, the optimized feedrates will be lost.

To remove the optimized feed rates without regenerating the NC code, click on the *Clear optimized feeds* option in the *Manufacturing* menu.

It is helpful to graph the tool load during 3D simulation before and after the optimization to see the changes. See page 176 for more information.

Target horsepower

Set this milling attribute to the desired horsepower for the operation. This number is then used in feed optimization to even out the tool loads.   This attribute is automatically set to the estimated horsepower for 2D roughing passes performed with flat end tools. (See page 241 for additional information on estimated horsepower.) For 3D roughing passes performed with flat end tools, this attribute is also set to the estimated horsepower **as long as the z increment attribute is set**. For all other operations, this attribute has no default value.

If the *Target horsepower* is not set for an operation, then its feedrates are not modified by feed optimization. This means that if you do not explicitly set the target horsepower for individual operations, feed optimization will only affect 2D roughing operations and 3D roughing operations with flat end mills and the z increment parameter set.

Feed optimization parameters

The following parameters are contained on the Feed optimization dialog box.

Increase feed when load is below % of programmed power – The programmed power is the estimated horsepower displayed in the Operations List. This percentage is applied to the estimated horsepower and the feedrate is increased for any move that requires less power.

But do not increase feed beyond % of programmed feed – This is the maximum amount to increase the feedrate over the operation's initial feedrate.

Decrease feed when load is above % of programmed power - The programmed power is the estimated horsepower displayed in the Operations List. This percentage is applied to the estimated horsepower and the feedrate is decreased for any move that requires more power.

But do not decrease feed below % of programmed feed – This is the minimum amount to decrease the feedrate.

Perform super sampling to calculate instantaneous tool load – Setting this option creates a more accurate sampling of the tool loads by sampling the loads at a number of points along the toolpath and then averaging these loads for each NC block. If this option is turned off a single tool load is calculated for each block of NC code.

Measure tool load – If *Perform super sampling* is checked, then you can enter how many times per minute to calculate the tool load. To calculate the load for each NC block, the loads are averaged. The larger this number is, the longer the Feed optimization will take. Note that this number is relative to the running time of your NC program. If your program will run more than an hour, it is recommended that you set this number to 100 or less.

Use different feed rates on a single block – A long cut may have varying tool loads. If you want to break up NC blocks so that you can more finely control the feedrates, check this option. Be aware that this option will increase the size of your NC program.

Split toolpath every - If you check *Use different feed rates on a single block*, then this parameter controls how often the toolpath will be broken up. It is specified as a percentage of the tool diameter.

Setting a feed or speed value for a turned operation

1. Double-click on the feature to bring up the Properties dialog box.
2. Click on the operation in the tree view.
3. Click on the Feed/speed tab.

4. If you want to specify the speed as constant surface speed:

- Click Constant Surface Speed checkbox.
- Enter the Surface Speed.
- Optionally specify the Approach RPM, the initial speed for the operation while still off the part.
- Optionally specify the Maximum RPM, the initial speed for the operation while still off the part.

5. If you want to enter the speed as a single revolution per minute value, unclick Constant Surface Speed and enter the Spindle speed.
6. If you machine has explicit spindle speed ranges, you may want to set the RPM Range.
7. If you want to specify the speed as feed per revolution, click the IPR checkbox and enter the Feed Rate.
8. If you want to specify the speed as feed per minute, unclick the IPR checkbox and enter the feed rate.
9. Click OK

RPM range

Some turning centers have gearboxes that set the maximum spindle speed of the machine. The RPM Range drop down list sets the gearbox to a specific maximum range. If RPM Range is set to a value of 1-4, then the range is set explicitly. If RPM Range is set to Auto then FeatureTURN sets the range for you based on the following rules:

1. If the feature is a turned hole or another turned feature without Constant Surface Speed set, then the range is determined based on the Spindle Speed.
2. If the feature is a turned feature with Constant Surface Speed set, then the range is determined based on the MAX RPM.

Note: You must physically switch the gearbox to match the RPM output by FeatureCAM.

Feed speed tables

Viewing feed/speed tables

1. Select Feed/Speed Tables from the Manufacturing menu.
2. The Feed/Speed Tables dialog box comes up with the current material type selected and the appropriate tab displayed based on the type of setup you are in.
3. Select the desired stock material from the drop-down list.
4. Select either the Milling/drilling tab or the turning tab.
5. For turning, select the desired tool grade. If the tool table is displayed dimmed, click the Tool Grades button to see which other table is being scaled to obtain values for this tool grade. Click OK.

6. For milling, select the desired tool grade and tool finish.

7. For certain materials there are actually two tables, each associated with a hardness value. If there are two table, the Hardness2 button will not be dimmed. Click Hardness 2 to see the other table. Feeds and speeds are interpolated for hardness values in between the two hardness values associated with the two tables.

8. Click OK when done viewing the table

Range

A rangeless table is for hardness values up to the value listed and higher values are invalid.

For a ranged table, two values of hardness must be given. For a hardness below the first value, the table for the first hardness is used. For a hardness between the two values, a table of interpolated values is used. Hardness above the last hardness value is invalid.

FeatureCAM supplies values for the fields for the selected combination of material, tool material, and tool finish.

Adding a new material

1. Select Feed/Speed Tables from the Manufacturing menu.

2. The Feed/Speed Tables dialog box comes up with the current material type selected and the appropriate tab displayed based on the type of setup you are in.

3. Click the *New Material* button.

4. Enter the name of the new material and click OK. The first character of the material is restricted to an uppercase letter (A-Z) or an underscore (_).

5. Click the *New* button to add values.

6. For milling, select the tool grade and tool material. For turning, select the tool grade.

7. Enter speeds and feed values (for a 1 inch tool) for each operation type.

8. For initial values you can click the *Copy Values* button to select a table to copy from.

9. If the feed or speed are dependent on the hardness,

- Click the *Use Range* button and enter the upper hardness value as *Hardness 2*.

- Enter the lower hardness value as *Hardness 1*.

- Go to step 7.

10. Click OK

Adding a new tool grade for turning operations

1. Select Feed/Speed Tables from the Manufacturing menu.

2. The Feed/Speed Tables dialog box comes up with the current material type selected and the appropriate tab displayed based on the type of setup you are in.

3. Click the *Turning* tab if not already displayed.

4. Click the *Tool grades* button.
5. Click the *New* button and enter the new tool grade name
6. Under When F/S table is undefined select either:

- Generate Toolpath Error if you want to define all table entries.
- Scale Existing Material to get values from another material. Select the existing material from the drop-down list and enter the scaling values for speeds and feeds.

7. Click *OK*.
8. If you selected *Generate Toolpath Error*, fill in all table values.
9. Click *OK*.

Modifying existing feed/speed tables

1. View the appropriate table.
2. Adjust the values as desired.
3. Click *OK*

Deleting a feed/speed table

1. View the appropriate table.
2. Click *Delete*
3. Click *OK* when asked for confirmation.
4. Click *OK*

How to import feed/speed tables

1. Select *Feed & Speed Tables…* from the Manufacturing menu.
2. Click the *Import* button. The Feed/Speed Import dialog box comes up.
3. Enter the name of the file to import under *Import File*.
4. Select an option for handling new materials with the same name as existing tools.
5. Select *Overwrite* if you want the new tool to be copied over the old one. Select *Skip Materia*l to ignore any new materials with the same name as an existing tool. .
6. Click *Import*.
7. Click Close.

How to export feed/speed tables

1. Select *Feed & Speed Tables…* from the Manufacturing menu.
2. Click the *Export* button. The Tool Export dialog box comes up.

3. Enter the name of the file to export under Export File.

4. In the *Material Types* list select the materials you want to export.

5. Use the  button to select all materials. Use the  to unselect all materials.

6. Select the units to express the tables in. Either *Inch* or *Metric*.

7. Check *Milling Feeds and Speeds* to export milling tables. Check *Turning Feeds and Speeds* to export turning tables.

8. Click *Export*.

9. Click *Close*.

The file created is tab-delimited text. You can edit the file in a spreadsheet if desired. This file can now be imported into FeatureCAM.

Feed speed databases

Initializing FeatureCAM databases

The tooling and feed/speed databases are created using the INITDB Initialization program in the FeatureCAM. This is the program that is run the first time that you run FeatureCAM to create your initial database. You may also want to use INITDB for the following reasons:

- Adding default tools and feed/speed tables to the database
- Recreating tooling and feed/speed databases if they become corrupt

Adding default tools and feed/speed tables to the database

Each tool is identified by a tool name and each feed/speed table is defined by a combination of the stock material name, tool material and tool grade. Any tool or feed/speed table that you add to the database will remain even if you rerun the INITDB program. INITDB will not overwrite any existing tools or feed/speed table in the database. The only way to remove them is to explicitly delete them in the Feed/speed table dialog box or the Tool manager.

The two most common reasons for rerunning INITDB are:

1. Restoring default tools or default feed/speed tables that you have deleted.

2. Adding tools specified in other units. You may have elected to only load the inch tools the first time and you may now want to add the metric tooling.

To perform these tasks:

1. Exit FeatureCAM if you are running it.

2. Run INITDB from the FeatureCAM group in the Start menu.

3. Select the units of tools you want to add. Remember FeatureCAM will not alter any items that you changed or delete any items that you added.

4. Click OK.

5. If you added tooling of both units, you will be asked which unit you will most often use to

model parts.

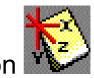
Recreating tooling and feed/speed databases if they become corrupt

If your tooling or feed/speed database becomes corrupt, you can recreate the databases using the INITDB program. **Remember that this procedure will erase any changes you have made to existing tools or feed/speed tables and will delete any custom tooling or new feed/speed tables that you have created. Contact your FeatureCAM support person before performing this task.**

1. To remove your database and recreate it:
2. Exit FeatureCAM if you are running it.
3. Run Windows Explorer.
4. Open the directory in which you installed FeatureCAM.
5. Open the database directory.
6. Click in the right-hand contents window.
7. Click *Select All* from the Edit menu.
8. Press the DELETE key on the keyboard.
9. Run INITDB from the FeatureCAM group in the Start menu.
10. Select the units of tools you want to add. Remember FeatureCAM will not alter any items that you changed or delete any items that you added.
11. Click OK. If you added tooling of both units, you will be asked which unit you will most often use to model parts.

Chapter 16
Creating NC Code

Posting your program

After running a simulation, click the NC code button  of the Steps toolbar to display the NC code dialog box. Click the *Display the NC code*  button to list the G-code in the Manufacturing Results window. You can also click the NC Codes tab of the Manufacturing Results Window.

NOTE: You can not post without the dongle connected to the computer.

Milling post options

Post Options in the Options menu controls the type of CNC machine that FeatureCAM targets for NC output.

The dialog box contains some auxiliary parameters used in post processing.

CNC file is the file name for the kind of machine. The file is either one that comes standard with FeatureCAM or one that you created with the Mbuild program. Click *Browse* to select your CNC file from the list of available files. Browse presents a list of on disk. You may put your files anywhere on disk, but look in the *M-LBRY* subdirectory of the *EZFM* directory to find the CNC files that come standard with FeatureCAM. For example, if you installed FeatureCAM in "*c:\ezfm*," the pre-defined CNC files are in one of the following directories:

- c:\ezfm\m-lbry\inch
- c:\ezfm\m-lbry\metric

Select the file name and click *OK* to select a CNC file.

Edit will start the MBUILD milling post processor on the current post processor.

Defaults returns all the Post Options to their default values.

Max. speed is the maximum spindle speed of your machine.

Min. arc defines the limit for any arc whose radius is transferred to the CNC machine as a line. Arcs greater than this limit are sent as arcs.

Max arc defines the maximum arc radius. Arcs greater than this limit are translated into lines.

Block start sets the starting line number for your CNC programs.

Block increment sets the increment between line numbers in your CNC programs.

Enable Cutter comp. turns off cutter compensation for the entire part.

Disable macros turns off macro generation for the NC code. This option is not available for all posts. Refer to *Hole macros* for more information about setting up macros.

Tool change location is the point where the tip of the tool moves prior to a tool change.

OK accepts your changes.

Cancel discards your changes.

Non-modal Decel override is an optional setting for the G-code for overriding the automatic deceleration of a control.

Milling post processors

850SXM	Cincinnati Avenger 850 Machining Center
AB845I/M	Allen Bradley 7-845 (Inch/Metric). Bridgeport Series II BMC 7-845 (Automatic tool change) with an Allen Bradley 8200 control.
ABS7DI/M	Allen Bradley S7D3 (Inch/Metric). Bridgeport Series II CNC S7D3 (Manual tool change) with an Allen Bradley 8200 control.
AC2100	Acramatic 2100 CNC Control (Inch). General post.
AC950	Acramatic 950 mill control. 3 axis mill post. Inch units.
AC950-10	Acramatic 950 mill control. 3 axis mill post with 10 block look ahead. Inch units.
ACLOC	Acraloc CNC (Inch). General post for Acraloc mills.
AGE3	Southwestern AGE3 controller, the newer form of ProtoTrakMx3. Inch units. Similar to PTRAKMX3 except: circular moves output absolute IJ values.
AN1100C	Anilam 1100M Conversational post (Inch).
AN1100MG	Anilam 1100M Control (Inch). G&M output.
AN1400G	Anilam 1400M mill control. Inch format.
ANCRUSAD	Anilam Crusader. Inch format.
ANILAM	Anilam (Inch). General post designed for mills using the Anilam control.
ARRW2100	Acramatic 2100 control for Arrow or Sabre 1000. Inch units.
BANDIT	Bandit CNC (Inch). General post for mills using the Bandit control.
BOSS15I V2XT KNEE MILL (INCH)	Bridgeport BOSS 15 (SX) based control. This post generates M26 for tool changes & M22 for program rewind.
BOSS3I/M	Bridgeport Operating Software Systems 3 (Inch/Metric). Bridgeport machines with BOSS3 control software.
BOSS4I/M	Bridgeport Operating Software Systems 4 (Inch/Metric). Bridgeport machines with BOSS4 control software.
BOSS6I/M	Bridgeport Operating Software Systems 6 (Inch/Metric). Bridgeport machines with BOSS6 control software.
BOSS7I/M	Bridgeport Operating Software Systems 7 (Inch/Metric).Bridgeport machines with BOSS7 control software.
BOSS8I/M	Bridgeport Operating Software Systems 8 (Inch/Metric).Bridgeport machines with BOSS8 control software.
BOSS9I/M	Bridgeport Operating Software Systems 9 (Inch/Metric). Bridgeport machines with BOSS9 control software.

control software.

BOSSSX	This post for the BOSS SX15 controls supports a pneumatic indexer using an M51 code. Wrapping is not supported.
BP320I/M	Bridgeport 320 Machining Center (Inch/Metric). Bridgeport 320 series horizontal machining centers with Fanuc 6M/11M controls.
BP380I/M	Bridgeport 380 Machining Center (Inch/Metric). Bridgeport 380 series horizontal machining centers with Fanuc 6M/11M controls.
BP520I/M	Bridgeport 520 Machining Center (Inch/Metric). Bridgeport 520 series vertical machining centers with Fanuc 11M controls.
BPSER2	Bridgeport Series II NC (Inch). Series II incremental controls.
BTC2I/M	Bridgeport BTC II (Inch/Metric). BTC II controls.
CEN400	Centroid M-400 CNC Control, Automatic tool change, CNC spindle control, G&M (Inch).
CINACR	Cincinnati Acramatic CNC-MC 2200B (Inch). General post for Cincinnati Milacron mills using the Acramatic CNC-MC 2200B control. This post is setup for vector type cutter compensation.
DSC300I	Discovery 300 (Inch) Bridgeport BOSS 15(SX) based control. This post generates M6 for tool changes & M22 for program rewind.
DSC308I	Discovery 308 (Inch) Bridgeport BOSS 15(SX) based control. This post generates M6 for tool changes & M22 for program rewind.
DX32I	DX 32 Control (Inch).
DYNA10C	Dynapath Delta 10M/20M controls, Conversational (Inch).
DYNA10G	Dynapath Delta 10M/20M controls, G&M (Inch).
DYNA40C	Dynapath Delta 40M/50M/60M controls, Conversational (Inch).
DYNA40G	Dynapath Delta 40M/50M/60M controls, G&M (Inch).
DYNAMTC	Dynapath 40M Control (Inch). Manual tool change (knee mill). No tapping.
EMCO	Emco Maier. Inch format.
EZTRAK	Bridgeport EZ-Trak (Inch)
EZTRAKM	Bridgeport EZ-Trak (Metric).
EZTRAKSX	Bridgeport EZ-Trak 2 or 3 AXIS conversational post; no cut comp.
FADAL	Fadal Controls (Inch) Compatible with all Fadal Machines.
FAN0M	Fanuc -0- M General post designed for mills using the Fanuc 0 M control.
FAN0MR	Fanuc 0m control Inch format, with rigid tapping.
FANU6M	Fanuc 6M (Inch). General post for mills using the Fanuc 6M control.
FANUC11M	Fanuc 11M Control (Inch). General post.
FANUC18B	Fanuc 18M Control (Inch). Similar to FANUC18M except: G28 home, G98 canned cycle retract.
FANUC18M	Fanuc 18M Control (Inch). General post.
FANUC21M	Fanuc 21M Control (Inch). General post.
GE2000	General Electric 2000 MC control. 3 axis mill post, Inch units. Originally developed for Monarch VMC-75.
GE845I/M	General Electric 7-845 (Inch/Metric). Bridgeport Series II BMC 7-845 (Automatic tool change) with a General Electric Mark Century 2000 control.

GES7DI/M	General Electric S7D3 (Inch/Metric). Bridgeport Series II CNC S7D3 (Manual tool change) with a General Electric Mark Century 2000 control.
HAAS	General post for HAAS mills (Inch). 3 axis.
HAAS2	HAAS mills (Inch). 3 axis. Similar to HAAS except: "O" used for program name, canned cycle executed from canned cycle line (machine setting #28 ON).
HD2500AI	Heidenhain 2500 Automatic Tool Change (Inch) General post for machining centers with Heidenhain 2500 control. This post does not use tool change positions and uses M6 to change tools.
HD2500MI	Heidenhain 2500 Manual Tool Change (Inch) Interact I Mark II knee mills with Heidenhain 2500 control. This post uses tool change positions and M25 code to change tools.
HURCO	Hurco control. Inch format.
INATCI/M	Interact Automatic Tool Change (Inch/Metric). Bridgeport Machines with Heidenhain 150,151,155 controls.
INMTCI/M	Interact Manual Tool Change (Inch/Metric). Bridgeport Machines with Heidenhain 150,151,155 controls.
INT300I	Heidenhain 2500 (Inch) Interact 300 Machining Center with Heidenhain 2500 control.
INT308I	Heidenhain 2500 (Inch) Interact 308 Machining Center with Heidenhain 2500 control.
INTGI	Heidenhain G-Code (Inch)
MAHO60	Maho Model 600 (Inch). This post is specifically designed for the Maho Model 600 horizontal machine center with the Phillips CNC 432 Control. It will support the model 500 with a minimal amount of changes to the output.
MAZAK	Mazak CNC (Inch). General post for Mazak mills.
MILLTRON	Milltronics Centurion 1 Control. Inch units.
MITSU50M	Mitsubishi 50M. Inch format.
NEWINTI/M	Revised post processor for Interact machines with Heidenhain 2500 controls and later (Inch/Metric).
OK5MGI	Okuma OSP5000 (Inch). General post for Okuma vertical mills using the OSP5000 control.
OKUMA5	Okuma OSP5000 (Inch). General post for Okuma vertical mills using the OSP5000 control.
OM550I	Bridgeport 550 Machining Center (Inch). Bridgeport 550 series horizontal machining centers with Fanuc -0-M controls.
PTRAKMX3	Proto Trak MX3 This post will generate 3 axis G-Code format programs.
R2G4I/M	Bridgeport Series II Mill (Inch/Metric). Bridgeport Series II vertical knee mills with Fanuc G4 controls.
SERVO	Servo. Inch format.
SLOMO	Slo Motion (retrofit control). Inch format.
SYS10	Bendix system 10 (Inch). General post for mills using the Bendix system 10 or 20 controls.
TN145I/M	Heidenhain TNC 145 (Inch/Metric). Bridgeport machines using the TNC 131,135,145 controls.
TNC426HS	Heidenhain TNC426 high-speed control (Inch). Uses datum table for tool definitions and has corner rounding tolerance for high-speed machining.
TORQCUTI	Bridgeport TORQ CUT 22 CNC uses 4th axis and MCSID
TRAKDX_3	Inch format of the trakdx3m.

TRAKDX3M	EZTRAK 3 axis metric conversational post. Metric format.
TREE	General post for TREE mills with Dynapath controls. G&M output (Inch). 3 axis.
VMCH400I/M	Updated post processor for Bridgeport VMC's, with Heidenhain 400 series controls, and tool pre-select includes 200 series cycles (Inch/Metric).
VMC-HRGI/M	Updated post processor for Bridgeport VMC's, with Heidenhain 2500 controls or later, includes 200 series cycles (Inch/Metric).
VMC-HTTI/M	Updated post processor for Bridgeport VMC's, with Heidenhain 2500 controls or later using machine tool table, includes 200 series cycles (Inch/Metric).
VMCQCYCI/M	Updated post processor for Bridgeport VMC's, with Heidenhain 370 controls (with level 8 PLC software) or later, includes 200 series cycles (Inch/Metric).
YASNAC50	Yasnac J50 milling post processor. Inch units.
YASNAC	Yasnac. Inch format.

Milling macros

Hole macros

Macros can be generated in the NC code for holes. To generate these macros, your post processor must support them, and you must turn this function on for the post.

1. Turn on *Retract to plunge clearance* for the hole pattern or set this attribute individually for individual holes.
2. Open the Post Options dialog box by selecting *Post Process* in the Manufacturing menu.
3. Select your post processor.
4. Set the *Enable Macros* checkbox.
5. Change any other appropriate settings.
6. Click *OK*.
7. Select *Machining Attributes* from the Options menu.
8. Set the *Minimize tool changes* checkbox.
9. You could set Minimize tool changes in the Ordering dialog box instead. Using the Default machining attributes setting includes macros for any parts you create.
10. Turn off *Minimize rapid distance*.
11. Click *OK*.

Macros cause FeatureCAM to analyze the NC code and generate macros for hole operations if it finds sets of repetitive tasks. This method may ignore sets of operations that don't have one-to-one correspondence with the other sets in the macro. For example, if you are drilling and reaming a pattern of six holes, and another hole in the setup is drilled with the same tool, the drilling operations, are not included in the macro because there are more drilling operations than reaming operations.

Z-level macros

Macros can be generated in the NC code for multiple Z levels of a milled feature. To generate these macros, your post processor must support them, and you must turn this function on for the post.

1. Open the Post Options dialog box by selecting *Post Process* in the Manufacturing menu.
2. Select your post processor.
3. Set the *Enable Macros* checkbox.
4. Click *OK*.
5. Select *Machining Attributes* from the Options menu.
6. Set the *Minimize tool changes* checkbox.
7. You could set Minimize tool changes in the Ordering dialog box instead. Using the Default machining attributes setting includes macros for any parts you create.
8. Turn off *Minimize rapid distance*.
9. Click *OK*.

Now when you generate NC code, you will get macros for the milled features that are milled at multiple Z depths.

Macros for repeated parts

If you are cutting multiple parts at once and want to use macros to decrease your code size see *Multiple Fixture Parts* on page 305.

Post options for turning

Post Options in the Options menu controls the type of CNC machine that FeatureTURN targets for NC output. The dialog box contains some auxiliary parameters used in post processing.

CNC file is the file name for the kind of machine. The file is either one that comes standard with FeatureTURN or one that you created with the Tbuild program. Click *Browse* to select your CNC file from the list of available files. Browse presents a list of on disk. You may put your files anywhere on disk, but look in the *T-LBRY* subdirectory of the *EZFM* directory to find the CNC files that come standard with FeatureTURN . For example, if you installed FeatureCAM in "*c:\ezfm*," the pre-defined CNC files are in one of the following directories c:\ezfm\t-lbry\inch or c:\ezfm\t-lbry\metric.

Select the file name and click *OK* to select a CNC file.

Defaults returns all the Post Options to their default values.

Max. speed is the maximum spindle speed of your machine.

Min. arc defines the limit for any arc whose radius is transferred to the CNC machine as a line. Arcs greater than this limit are sent as arcs.

Block start sets the starting line number for your CNC programs.

Block increment sets the increment between line numbers in your CNC programs.

A number of system features are enabled at the bottom of the dialog boxes. These settings allow these options to be individually set on features. Note that these settings must be set on the feature to be activated. By turning off these settings, the option is turned off system-wide. These options include:

Drill canned cycle activates canned cycles for drilling. This is a global setting and cannot be set on individual hole features. Note the post processor must support canned cycles.

Groove canned cycle activates canned cycles for drilling. This is a global setting and cannot be set on individual groove features. Note the post processor must support canned cycles.

Turned canned cycle activates canned cycles for turning and boring. It must also be turned on for individual features.

Tool change location is the point where the tip of the tool moves prior to a tool change.

Turning post processors

850SXT	Cincinnati 850 Talon Lathe
850TC	Acramatic 850TC control. Inch format, OP STOP after tool changes, BLOCK DELETE (SKIP) at sync blocks. Turn off BLOCK DELETE to access sync block when restarting at other than start of program. Uses TOOL CHANGE location as positioning move in sync blocks.
ANILAMT	Anilam Lathe
FAN0TC	Fanuc 0TC controller. Inch format. Supports multiple repetitive cycles (G70/G71) and threading canned cycles (G76).
FAN6TI/M	Fanuc6T Control (Inch/Metric). General post for Lathes using the Fanuc6T control. The post is designed for computed threading and drilling cycle output, with canned grooving cycle output.
FANUC11T	Fanuc 11T Lathe
FANUC3T	Fanuc 3T Lathe. This post is for a Fanuc 3TD control on a Mori Seiki SL - 1. G50's are used on every tool and after completion of the segment it is returned to the Home position by a G28 on the X axis first followed by the Z axis. X axis output is negative for all values since all cutting is done on backside of part.
FANUCEZ	EZ-Path Lathes with Fanuc controllers. Supports Fanuc Canned Cycles: G83 Drilling, G84 Taping, G70 Finishing Cycle, G71/72 Roughing Cycles, G76 Threading Cycle.
G10_CYC	G10 Romi lathe with Fanuc21i control. Supports Fanuc Canned Cycles: G83 Drilling, G84 Taping G70 Finishing Cycle, G71/72 Roughing Cycles, G76 Threading Cycle.
G30_CYC	G20 & G30 Romi lathes with Fanuc21i control. Supports Fanuc Canned Cycles: G83 Drilling, G84 Taping G70 Finishing Cycle, G71/72 Roughing Cycles, G76 Threading Cycle.
GE1050	General Electric 1050T Control. This is a general post designed for lathes using the General Electric 1050T control.
HARD6T	Hardinge Fanuc6T Control (Inch). General post for Hardinge Lathes using the Fanuc6T control. Instead of using G0 for rapid movements it uses feed rates of 200 inches per minute. The post is designed for computed threading, with canned drilling and grooving cycle output.
JLAB73	Jones & Lamson Allen Bradley 7365 Control. Jones & Lamson TNC lathes with the Allen Bradley 7365 control.
MAZAK2TI	Mazak lathe with radius output. Computed thread and groove. G41 left compensation. G42 right compensation.
MAZAKTI	Mazak lathe with diameter output. Computed drilling and canned thread and groove. G42 left compensation. G41 right compensation.
MORSEK	Moriseki (Inch). General post for Moriseki lathes. The post is designed for computed drilling, threading and grooving output.

	computed drilling, threading and grooving output.
MSL200	Moriseki SL200 Lathe. Inch format
NC-RUN	This is a special post created just for NC-RUN verification, it is based on the Okuma OSP5000 control.
NEW6T	New Hardinge Fanuc6T Control (Inch). This is a modified version of the HARD6T post for Hardinge, It uses G0 for all rapid movements. The post is designed for computed threading, drilling and grooving cycle output.
OK500I/M	Okuma OSP5000 Control (Inch/Metric). General post for Okuma lathes using the OSP5000 control. The post is designed for canned cycle drilling, threading and grooving output.
OKLC3I/M	Okuma LC30 Lathe OSP3000 Control(Inch/Metric). General post for Okuma lathes using the OSP3000 control. The post is designed for canned cycle drilling, threading and grooving output.
PATH	*Bridgeport EZ-Path Lathe (without CSS). This post generates native conversational format programs.
PATH_CYC	Bridgeport EZ-Path Lathe (without CSS) with canned cycles. Conversational format programs. Supported Canned Cycles: ROUGH, PROFIL, GROOVE, THREAD, and DRILL.
PATHG	Bridgeport EZ-Path Lathe (without CSS). This post generates G-Code format programs.
PATHS	*Bridgeport EZ-Path S Lathe (with CSS). This post generates native conversational format programs.
PATHSG	Bridgeport EZ-Path S Lathe (with CSS). This post generates G-Code format programs. (with CSS)
PP15_CYC	Bridgeport PowerPath 15 Lathe with canned cycles. Supported Canned Cycles: ROUGH, PROFIL, GROOVE, THREAD, and DRILL.
PPATH	*Bridgeport Power-Path-15 Lathe. This post generates native conversational format programs.
PPATHG	Bridgeport Power-Path-15 Lathe. This post generates G-Code format programs.
PTHS_CYC	Bridgeport EZPath S Lathe (with CSS) with canned cycles. Supported Canned Cycles: ROUGH, PROFIL, GROOVE, THREAD, and DRILL. (with CSS)
ROMI35I	Romi Lathe. This post is designed for the Romi
SBLI	South Bend Lathe (Inch) This post is designed for the Magna Turn Slant Bed CNC2000 lathe.

* Note: For Plain Language Dialog Post (P1 = Tool I.D.)

Turning canned cycles

Canned cycles can be generated in the NC code for nearly every turned feature.   To generate these macros, your post processor must support them, and you must turn this function on for the post and for some features you must also activate the canned cycles on the feature level.

Hole features

To turn on canned cycles for holes check *Enable drilled canned cycles* in the Post options dialog box. This turns on canned cycles for all holes. There is no way to control this on an individual feature basis.

Turn/Bore features

Canned cycles for turn and bore features must be enabled by checking *Enable turn canned cycles* in the Post options dialog box. You must then go to the Properties dialog box for each turn/bore feature click on the Strategy tab and check *Use canned cycle*. Also check *Reuse path in canned cycle* if you want to output the path geometry only once for both roughing and finishing. You can also set these values in the default machining attributes, but remember these values will only apply to features you create after making this change.

Groove features

Enable grooving canned cycles in the Post options dialog box by checking *Enable groove path canned cycle*. Then turn on canned cycles for each groove by bringing up the feature's Property dialog box, clicking on the Strategy tab, and then clicking *Use path canned cycle*. You can also set this attribute on the Groove tab of the default machining attributes, but remember this will only apply to features you create after changing this setting.

Thread features

Thread features will always use canned cycles.

Chapter 17
4th Axis

Fourth axis overview

You must have a CNC control that supports a fourth axis, and a rotary table or native fourth axis that can be controlled by the CNC machine. The fourth axis can be used in two different ways. It can be used as an indexer, to rotate the part between machining operations so that the machining takes place on different planes of the part. This is called indexing.

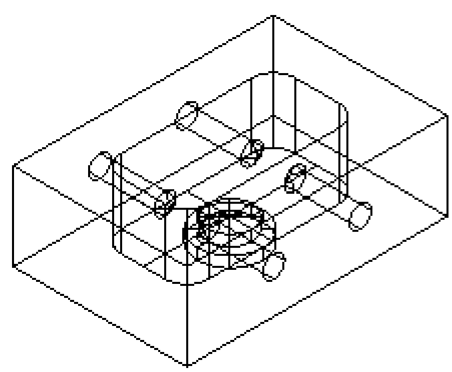
The second use of the rotary table is as a continuously moving axis. Rotation occurs during the machining operation and the tool movement is limited to either the X or Y-axis and the Z-axis. The part itself is rotated to take the place of the axis not used. This feature is called wrapping and is capable of wrapping any feature while the part is rotated.

Indexing

Indexing uses the fourth axis to rotate the part between machining operations so that the machining takes place on different planes of the part

Overview of indexing

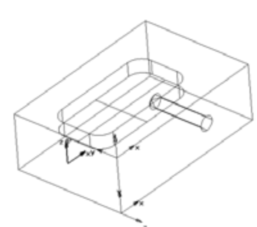
Parts whose features can be accessed by rotating around a single axis are candidates for fourth axis wrapping. Indexing uses the fourth axis to rotate the part to access each feature. In FeatureCAM each face of the part can be assigned to a separate setup or features can be placed radially around the center of rotation by using only one setup.



Note that you must have a CNC control that supports a fourth axis, and a rotary table or native fourth axis that can be controlled by the CNC machine in order to utilize indexing.

4th axis indexing with multiple setups

Normally each setup generates a separate program. If the setups of your part are a simple rotation around your machine's fourth axis as shown below you can use indexing to combine all the setups into a single program that rotates each setup into position using the fourth axis.



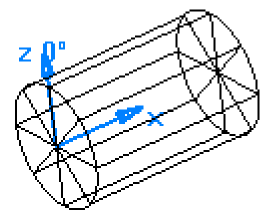
Indexing can be performed around the X or Y-axis of the Stock Axis. The post processor you use must match have the same indexing axis as your part. For each setup the corresponding axis must be parallel to the indexing axis. For example, if you are indexing around the world coordinate X-axis, the X-axes of each setup must be parallel to the stock axis.

When using indexing, the part documentation is combined for all setups. This means that you will have just one operations list, one tool list and one NC part program for all setups.

When positioning features in the Location dialog box, use the XYZ or polar types of positioning.

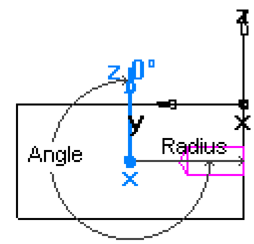
The Stock axis

The *Stock axis* corresponds to machine zero and the axis of rotation for an indexer. It is not normally displayed, but it can be show by clicking on the *View* menu and selecting *Show stock axis* from the show menu. It is displayed as two vectors. One shows the axis of rotation and the other indicates the orientation of a 0 degree rotation.

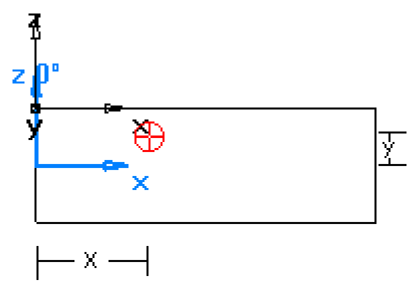


4th axis indexing with a single setup - Positioning features

When creating 4^{th} axis indexing or turn/mill parts, the Location_dialog_box of the feature wizard provides an option for positioning called *Radial about the X axis* (or *Radial about the Y axis* if indexing around the Y axis.) Use this method of positioning 4^{th} axis features, unless you want to create setups on each face you want to machine. Use the angle and radius dimensions to orient the feature on the proper face.



In the case of indexing around the X-axis, the X coordinate will move the feature along the X-axis and the Y coordinate will translate the feature in the perpendicular direction.

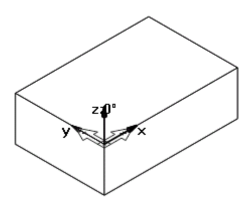


In the case of indexing around the Y-axis, the Y coordinate will move the feature along the Y axis and the X coordinate will translate the feature in the perpendicular direction

4th axis indexing - Positioning the stock

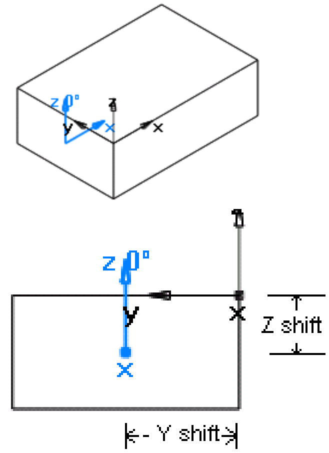
Often, the initial part setup is positioned at the same point as the stock axis.

You can position the setup axis at whatever point is convenient for locating part zero, but the stock must be moved so that the stock axis is positioned at the center of rotation of your indexer. This cannot be performed with the *Stock* step, instead it must be done using the stock properties dialog box as follows:



1. Select the stock in the graphics window.

2. Click the Properties  button at the bottom of the screen. The box comes up.

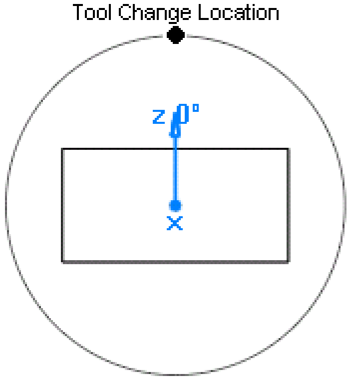


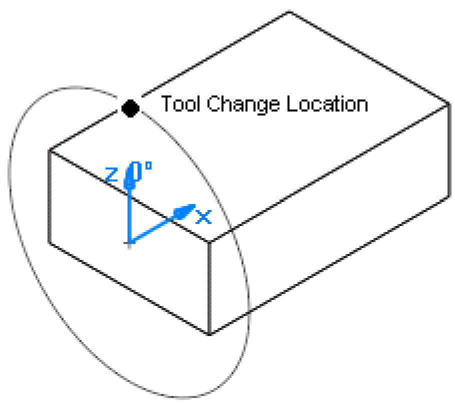
3. On the Dimensions page, adjust the X, Y and Z coordinates to move the stock. In the example on the right, it is assumed that the center of the stock will be aligned with the axis of the indexer. The stock is shifted by a negative amount in Y equal to half the stock width and a positive amount in Z equal to half the stock thickness.
4. Click the *Apply* button to see the results.
5. Click *OK* when the stock has been moved correctly.

4th axis indexing - Specifying the tool change position

The tool change position must be set correctly for the simulation of 4th axis indexing parts to be accurate. Note that this does not affect the NC code, only the graphics.

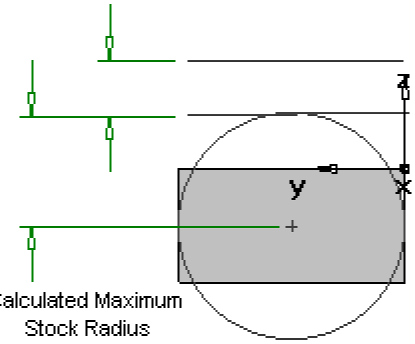
The distance from the stock axis origin to the tool change point must be long enough so that the tool clears the part as it rotates. The figures below show the circle that would be formed by sweeping a tool change location around the stock axis. Notice that this circle clears the entire part.





4th axis indexing - How the clearance plane is calculated

In 4th axis positioning and 5th axis positioning the tool must retract to a safe distance so that it will not collide with the part while it is indexing. To achieve this, FeatureCAM calculcates the maximum stock radius and adds to that the *Z index clearance* to determine the appropriate retract distance.



How to create an indexed program

1. Display the stock axis. This will be used for reference.

2. Click on the Stock  step.

3. Click *Next* twice.

4. Click the *4th Axis Positioning* radio button.

5. Decide what world coordinate axis you will rotate about. The X-axis is recommended since the majority of the post processors support only X-axis indexing. (For the rest of this description it is assumed that you have selected the X-axis.) For the X-axis, click *Index around the X-axis*. For the Y-axis, click *Index around the Y-axis*.

6. Position the stock appropriately.

7. Specify the tool change position.

8. If you want to create setups on each face,

9. Create the setups for indexing.

10. Switch to each setup and create your features.

11. If you want to use a single setup, create each feature and follow this procedure on the location page to orient each feature.

12. Generate the tool paths.

13. Click on *Post Process* in the *Manufacturing* menu and select a post from the 4thxs directory.

14. Click the NC tab in the Manufacturing Results window.

Restrictions of indexing

1. You must use a post from the 4thxs directory. Normal posts do not support indexing.

2. The post CNC files have the indexing axis hard coded. Most are setup to rotate about the X-axis. The axis of rotation in your program must match the rotation axis of the post.

3. If you are using multiple setups and are rotating about the world X axis the X axes of the setups must be parallel to the world X axis.

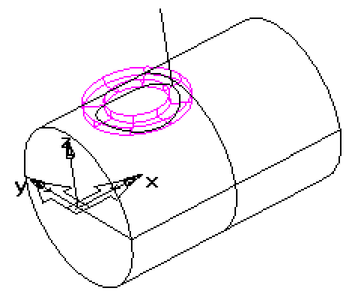
4. If you are using multiple setups and are rotating about the world Y axis the Y axes of the setups must be parallel to the world Y axis.

Fourth axis wrapping

4th axis wrapping allows you to wrap a 2 ½ D feature around either the X or Y-axis. You must have a machine tool with 4th axis capabilities to take advantage of this feature.

Overview of 4th axis wrapping

4th axis wrapping uses the rotary table as a continuously moving axis. Rotation occurs during the machining operation and the tool movement is limited to either the X or Y-axis and the Z-axis. Wrapping is limited to cylindrical stock shapes. A side effect of wrapping is that all arc moves must be converted to linear moves for posting. See *Wrap tolerance* for further information.



Any milling feature can be wrapped using 4th axis wrapping or turn/milling. The figure shows a simple groove that is wrapped around the X-axis.

Note that the features are not displayed as wrapped, but when you generate toolpaths for a wrapped setup, the toolpaths are wrapped.

How to create an NC program using 4th axis wrapping

1. Select Show Stock Axis from the Show submenu of the View menu. This is the wrapping axis.
2. Click on the Stock  step in the Steps toolbox. This will walk you through the stock wizard. During this process make sure that:

a. On the Dimensions page of the Stock wizard, you create round stock. For the *Axis* select the desired wrapping axis.

b. On the Multi-axis positioning page of the Stock wizard, you click *4th axis positioning* and *Index around X* if you chose *X* as your Axis, or click *Index around Y* if you chose the *Y* axis. Note that most 4th axis post processors provided with FeatureCAM wrap around the X axis.

3. When creating a feature that you would like to wrap:

a. Position it using the *Radial about X axis* option in the Location dialog box.

b. After creating the feature, edit it and check the *Wrap feature around X axis.*

4. Select *Post Process…* from the Manufacturing menu.
5. Click the *Browse* button to specify your post processor.
6. Select a post from the *4thxs* directory. Make sure to select a post processor that matches the index axis you selected in the Stock wizard.
7. Click OK.
8. Generate the tool paths with Show centerlines. Notice that the toolpaths are wrapped around a cylinder. Centerline and 3D solid simulation will allow you to preview the wrapped toolpaths.
9. If the wrapped feature is too chunky, you may need to lower the Wrap tolerance default

machining attribute. It is located on the Misc. tab of the machining attributes.

10. Click the NC tab in the Manufacturing Results window.

Restrictions of 4th axis wrapping

1. You must use a post from the 4thxs directory. Normal posts do not support 4th axis wrapping.
2. The post CNC files have the indexing axis hard coded. Most are setup to rotate about the X-axis. The axis of rotation in your program must match the rotation axis of the post.
3. If you are rotating about the stock X-axis the X axes of the setups must be parallel to the stock X axis.
4. If you are rotating about the stock Y-axis the Y axes of the setups must be parallel to the stock Y axis.
5. 4th axis wrapping is not simultaneous motion of 4 axes. Only three axes are active. In the case of X-axis wrapping, you get X and Z translational motion and a rotation around Y.
6. 2D simulation does not work for 4th axis wrapping.

Cylindrical cams

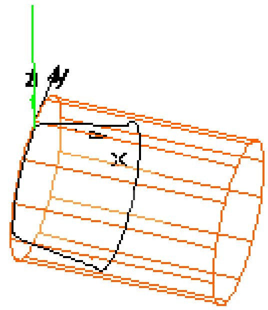
Note: You must have 4th axis wrapping to cut cylindrical CAMS.

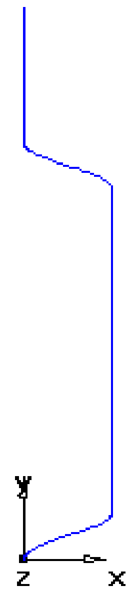
Cylindrical, or barrel, cams are specified as you would a normal cam. See page 48 for more information on reciprocating cams. For simple cams, the rise and fall are radial distance. On Cylindrical cams the rise and fall displacements are along the wrapping axis.

For example, if you create a barrel cam that is wrapped around the X-axis with the parameters shown below, the right-hand curve is created. Notice that the displacement is in the X direction.

If you create a groove feature with using the barrel cam curve, you get the toolpath shown below.

Range (deg.)	Type	Duration (deg.)	Displacement
0 - 30	Rise	30.00	2.000
30 - 240	Dwell	210.00	0.000
240 - 270	Fall	30.00	2.000
270 - 360	Dwell	90.00	0.000





How to create a cylindrical cam

Follow the steps for *How to create a NC program using 4th axis wrapping*. For Step 3, *Create your part features*, follow these steps:

3a. Bring up the cam dialog box from the Create menu or the curve wizard.

3b. Click *Cylindrical Cam* check box.

3c. Specify the *Radius* as the radius of your cylindrical stock

3d. Specify either *Wrap X Axis* or *Wrap Y Axis*. Make sure that this setting matches the wrapping axis you specified in your setup.

3e. Enter the *Segment* parameters.

3f. Click *OK*.

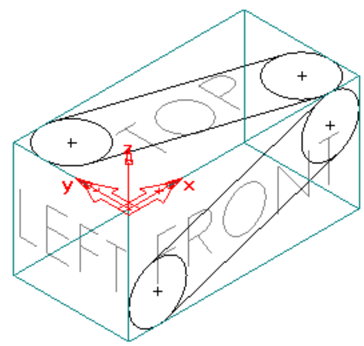
3g. Use this curve to create a groove.

You now have a feature that you can use for 4th axis wrapping.

Chapter 18
Coordinate Systems

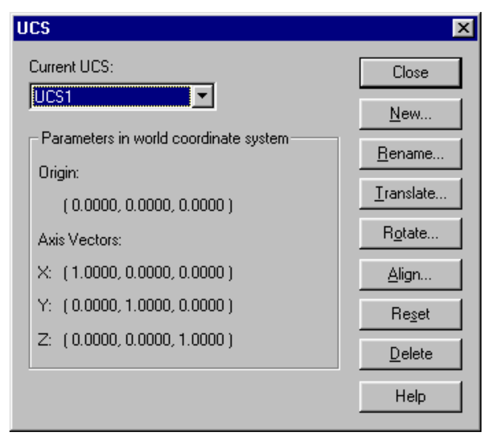
Overview of user coordinate systems

A User Coordinate System (UCS) is an origin, X direction, Y direction and Z direction used for modeling. Two-dimensional geometry is created in the principal plane (defined by the current UCS) that is closest to the plane of the screen. For example, if you are in the Top view the geometry is created in the XY plane. Viewing in the Front view will result in geometry being created in the XZ plane. The figure shows geometry that was created in both the top and front views. Note that the part is being viewed in the isometric view in the figure.



If you want to create geometry out of one of the principal planes of the existing UCS, you will need to create additional UCSs. You can use many different UCSs while modeling your part. When you create a manufacturing setup, you select a single UCS to determine the part program zero and set the direction for the X, Y and Z axes.

It is easy to create UCSs in many different locations. You can create a UCS by explicitly transforming an existing UCS or by using the Alignment wizard to create UCSs at convenient locations on your part. The name of the current UCS is displayed in the status bar. To change to a different UCS just click on the current UCS name in the status bar and select the new UCS from the popup menu.



UCS dialog box

The UCS dialog box allows you to create and manipulate User Coordinate Systems. It is displayed by selecting *UCS* from the *Construct* menu, or by clicking the UCS  button from the Advanced toolbar. The Current UCS drop down list shows the current dialog box. Its coordinates in the Stock Coordinate System are displayed in the Parameters in Stock Coordinate Space box. By selecting a new UCS in the Current UCS drop down list, you change the active UCS

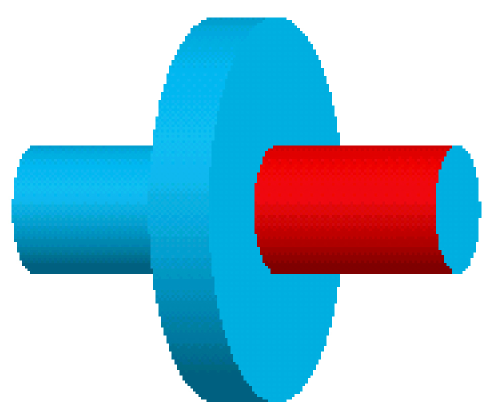
The *New* button allows you to create a new UCS. The translate, rotate, align, rename and Reset buttons allow you to modify the current UCS.

To delete the current UCS, select the UCS name in the Current UCS drop down list and click the Delete button.

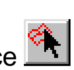
Align Wizard

The Align Wizard allows you to place a UCS using a number of methods. Select the method and hit the Next button. The next page then prompts you for the information listed below.

Method	Next Page Prompts To
Block stock	Select the face, and any corner of that face. You can graphically pick either the face or corner.
Feature	Select the feature from a list of features in the part. The UCS is placed at predetermined locations on the feature.
Three points	Pick an origin, a point anywhere on the new X axis, and any point in the new XY plane
Two lines	Select from any two intersecting lines in your part drawing to specify the new X and Y axes.
Circle	Select a circle. UCS is positioned at the origin.
Curve	Select a curve. UCS is positioned the start or end point. The Z axis is aligned with the curve normal.
Surface	Select the surface and a point on the surface. UCS is positioned at the point with the Z axis aligned with the surface normal.
UCS	Select another UCS. The new UCS is now a copy of the UCS you selected. You can then edit it with other functions in the UCS dialog box.
Align to revolved surface	This method will typically be used to align the UCS to the axis of a turned part. Remember that this alignment method only works for surfaces of revolution. For many imported models, flat disks are not represented as surfaces of revolution. For the model below you should select the cylindrical surfaces instead of the disk in order to position the UCS at the end of the part.



To complete this dialog box:

1. Click the Pick surface  button.

2. Select the surface of revolution in the graphics window. If the surface you select is not a surface of revolution, you will

receive the error message, “Can’t construct UCS from the selected surface.” In that case, you must select a different surface.

3. If you want to work from the other end of the surface, select *Flip direction*.

Close

Close applies your changes, if any, and closes the dialog box.

Rename

Rename opens a dialog box where you can rename the current setup and Fixture ID.

Reset

Reset makes the UCS equivalent to the STOCK coordinate system.

Rotate

Rotate specifies rotation about the X, Y and/or Z axes. The rotation is given in degrees, either positive or negative.

Translate

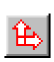
Translate specifies coordinates for the new UCS or use Pick to select the point with the mouse.

How to change user coordinate systems

There are two ways to change your current UCS

1. Click on the current UCS name displayed in the status bar
2. Select the desired UCS from the popup menu.

OR

1. Click up the UCS  button from the Advanced toolbar.
2. Select the desired UCS from the drop down list.
3. Click Close

How to create a user coordinate system

1. Open the UCS dialog box by selecting the UCS  button from the Advanced toolbar.
2. Click the New button.
3. Name the new UCS.

4. If you want to create a copy of an existing UCS, Click From UCS and then select the UCS in the drop down list. Click OK. You can then modify your new UCS using the UCS dialog box.

5. If you want to use the alignment wizard, Click Use alignment wizard. Click OK. Use the alignment wizard to position your new UCS.

How to modify a user coordinate system

1. Open the UCS dialog box by selecting UCS from the Advanced toolbar.

2. Select the UCS from the Current UCS drop down list.

3. Click one of these buttons:

- Translate
- Rotate
- Align
- Rename
- Reset

4. Click Close

How do setups relate to UCSs?

A user coordinate system (UCS) is an origin and three vectors (X, Y and Z) that determine a position and orientation in 3-dimensional space. You can use an unlimited number of these to conveniently model your part.

One particular UCS is associated with a setup. A setup is an orientation and part program zero for a physical setup on the machine tool. The orientation and program zero are determined by the associated UCS and the setup contains additional information like the fixture ID and the name of the NC program that will be generated.

If setups are created directly by aligning with the stock, special UCSs are created with the string, *UCS*, appended to the setup name. For example a UCS called *UCS_setup2* is automatically created for *Setup2.* These UCSs are used to store the location/orientation information for the setup. They cannot be deleted as long as their setup exists.

Setups

Adding setups to a part lets you create multiple NC programs for cutting the different sides of the part. You could create different FeatureCAM part files for each setup, but creating a single part with multiple setups has the following advantages:

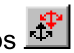
1. It is easier to organize the different setups if they are contained in a single FeatureCAM file.

2. All setups can be simulated together to provide a graphical model of the finalized part.

Setups can be anywhere in three dimensional space. They can even exist in the same location as other setups. When you create a setup, you need to keep in mind what it means for the part. The setup is the part origin, (0, 0, 0) on the machine and in the NC code. You have to place the setup where that origin will also work with the manufacturing of features in the setup.

The origin and coordinate system for each NC part program is determined by the setup. When you Save NC, the part program and manufacturing documentation is generated and saved for all setups of your part.

How to create a setup

1. Open the Setups dialog box by clicking Setups  in the Advanced tool bar.
2. Click the *New* button.
3. Follow the steps of the wizard.

Fixture ID

Fixture ID has two related contexts.

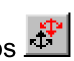
- One specifies the fixture offset used to model the part within FeatureCAM, especially in multiple fixture situations.
- The other context specifies the fixture offset used to produce the part in NC code.

For NC code, FeatureCAM passes the Fixture ID to the Post Processor that then uses the reserved word Fixture to pass the fixture offset information to the machine. While your part may have been displayed and modeled at one location, the fixture offset may override that location in actual production depending on your machine tool system.

You must set the Fixture ID to correspond to your machine tool. If your machine uses G54 or G55, set the fixture ID to 54 or 55. If your machine uses H1, set the fixture ID to 1.

The other fixture offset type reserved words, Datum Shift and Datum Set, are not supported. Datum Shift and Set are commonly seen as G92, or G97 codes.

How to edit an existing setup

1. Open the *Setups* dialog box by clicking Setups  in the Advanced toolbar.
2. Select the setup name as the Current Setup
3. Click *Edit*.
4. Change the name, fixture ID, type or UCS.
5. Click *OK*
6. Click *Close*

How to delete a setup

1. Click on the Part View.
2. Either delete or move each feature in this setup to another setup.
3. Click on the setup name and click the DELETE key on the keyboard.

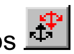
How to change the setup

To change the active setup,

1. Select *Setups* in the Edit menu or from the Advanced toolbar
2. Choose the setup from the list box
3. Click Close.

Create your feature.

How to set the current setup

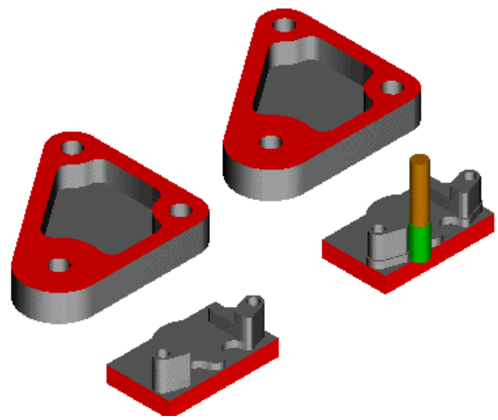
1. Open the Setups dialog box by clicking Setups  in the Report bar.
2. Select the setup name as the Current Setup
3. Click Close

OR

1. Hold down the left mouse button on the setup name in the status bar
2. Select the setup name from the popup menu.

Chapter 19 Multiple Fixture Documents

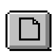
If you want to cut different parts on your machine at once, use the Multiple Fixture part. With Multiple fixture parts, you can mix different setups from FeatureCAM parts. FeatureCAM can then create a single program for cutting all the parts and it can minimize tool changes across all parts.

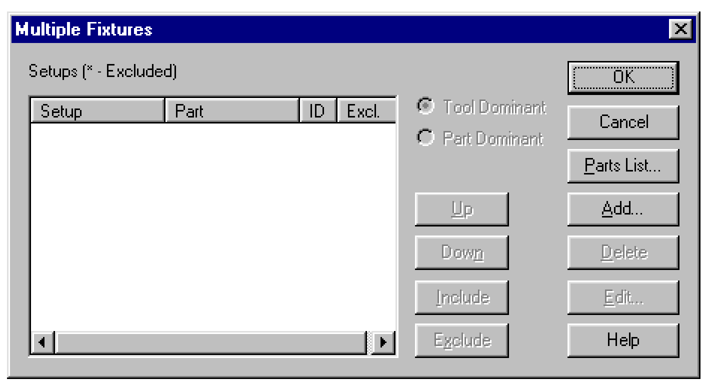


Saving and opening multiple fixture parts

Multiple fixture parts are saved in files with the ".mf" extension. When opening multi-fixture documents change *the Files of type* to be *MF Documents (*.mf)* to view the multi-fixture documents saved on your disk.

How to create multiple fixture parts

1. First create the parts you would like to cut and save them on disk.
2. Click the New  button.
3. Select Multiple Fixture Document. The following dialog box comes up.
4. Click Parts List and follow the instructions in the next section.



Parts list

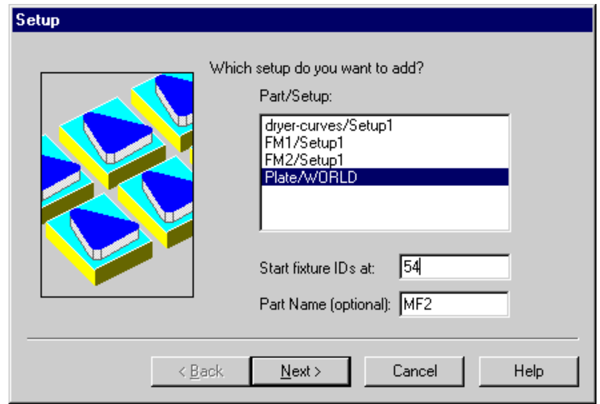
1. To open a part file for use in a Multiple Fixture part, use Parts List in the Multiple Fixtures dialog box.
2. Then Browse for the parts you want to include, one by one.
3. Perform the Add process.

Besides the Browse and select functions, Parts List also is where you **update** a Multiple Fixture document if the source files have changed.

4. Select the source file and click *Reload*.

Add

1. To put parts in the dialog box, use *Add* which opens the Setup dialog box. The Setup dialog box has the parts and setups from all the .fm files that FeatureCAM currently has open or that were selected in Parts List. Using Parts List now is a little more work than already having the files open.



2. Pick the part and setup in the list that you want to place in the multiple part file.

3. The fixture ID numbers itself, but you can override it if you wish.

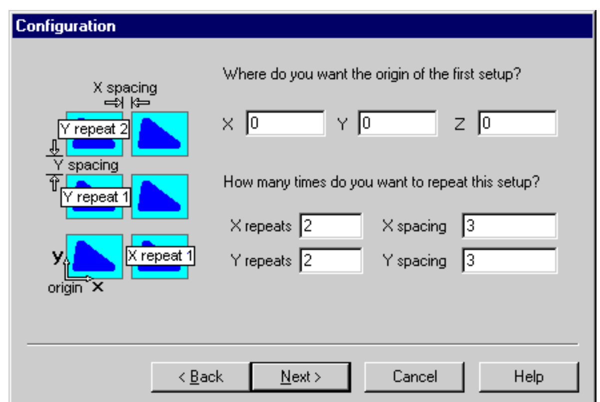
4. Then click *Next*. Subsequent Fixture IDs number sequentially from the last Fixture ID input, so you can automate and customize fixture numbering at the same time.

5. Click *Next* to open the Configuration dialog box.

Configuration

1. The XYZ part origin is read from the file and setup you selected. You can also override it if you wish.

2. X and Y repeats define a rectangular array of the selected part setup. Use these numbers to determine how many copies of the part to mill at the same time.



3. X and Y spacing is the distance between part origins. Depending on your post, these spacing numbers might not have any effect on the part as produced at the machine. That is controlled by the Fixture ID and how the Fixture ID is used to locate other parts relative to each other.

4. Click *Next* to open the Layout dialog box.

Layout

This is where the multiple part file setup diverges. You can build each part in its own piece of stock material, or build them from one larger block of stock material.

- Individual blocks are simpler to design, but perhaps less efficient.

- Single block may minimize waste, but is more difficult to layout and fixture.

Individual blocks

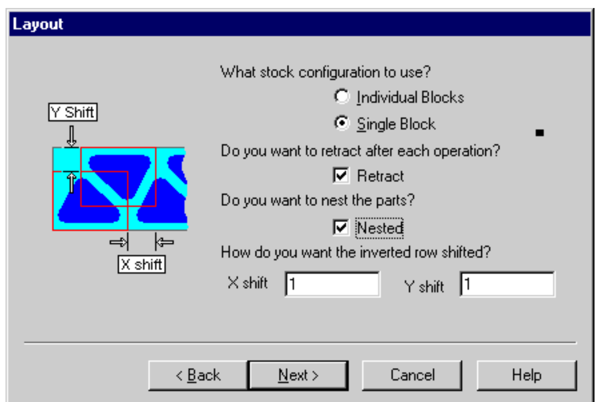
1. Set *Individual Block* and click *Next*. FeatureCAM constructs the layout in the Graphics window and displays the Preview dialog box.
2. Depending on your screen layout, drag the Preview box out of the way so you can preview your parts.
3. If the layout is acceptable, click *Finish*.
4. Otherwise, click *Back* until you reach the dialog box with the settings you want to modify.
5. Change those parameters and click *Next* until you return to the Preview dialog box.
6. Repeat this process until the part layout is correct then click *Finish*.
7. You can repeat the whole Add process to place more setups or parts in the whole layout as needed to complete the task at hand.

Single block

1. When you set Single block, the dialog box changes. The Nested and Retract options are described below.
2. Click *Next* to specify the stock.

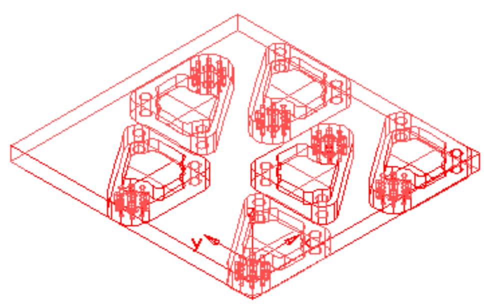
Nested

Depending on your part geometry, you might be able to make better use of a given stock size by nesting your parts. Nesting involves flipping the part around so that it can nest in the blank spots between other parts and can minimize waste in cutting some parts.



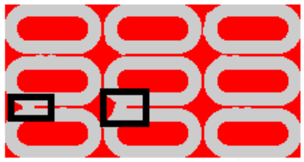
Setting Nested in the Layout dialog box changes the picture and adds some more parameter fields.

X /Y shift is set for the inverted row. The important concept is that you don't set space directly between neighboring shapes. That parameter was handled earlier when you specified how many repeated columns and rows to use. For shift spacing, you are setting the distance between the two rows' top edges (Y shift) and the columns' right edge offset (X shift). You probably will need to adjust your X and Y spacing in addition to the X and Y shifts to find the optimum spacing. Don't overlook negative values either.



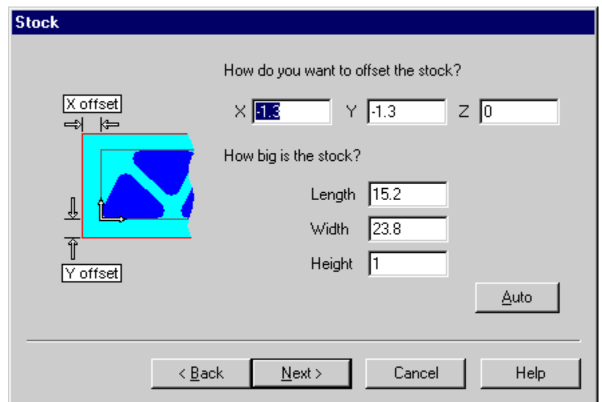
Retract between each operation

This setting is specific only to parts with outer side features or bosses that have the Total Stock attribute set. It is not recommended to set this attribute for parts that are not in this category. For parts of this type, FeatureCAM can occasionally violate the regions between the parts by feeding between features. Set the *Retract* checkbox to ensure that the tool will retract between each feature. The figures below show a multiple fixture part which contains individual parts that are side features with Total Stock set to 0.0. The left-hand figure has the *Retract* box unchecked. The right-hand figure is the same part with the *Retract* checkbox checked.



Stock

1. The part origin is read from the file. You can override it if you wish.



2. Notice that the block is automatically sized based on the part size and the spacing you specified. If your stock has a different size, enter those dimensions here. Clicking *Auto* calculates the minimum block the parts will fit in as described in the Configuration dialog box.

3. Click *Next* to open the Preview dialog box. FeatureCAM constructs the layout in the Graphics window and displays the Preview dialog box.

4. Depending on your screen layout, drag the Preview box out of the way so you can preview your parts. If the layout is acceptable, click Finish.

5. Otherwise, click *Back* until you reach the dialog box with the settings you want to modify.

6. Change those parameters and click *Next* until you return to the Preview dialog box.

7. Repeat this process until the part layout is correct, then click *Finish*.

8. You can repeat the whole Add process to place more setups or parts in the whole layout as needed to complete the task at hand.

Completing the multiple fixture part

When you click Finish in the Preview dialog box, you return to the Multiple Fixture dialog box. You can continue to Add other parts and setups as needed. You can even mix and match single and individual block layouts if you need to.

Adding parts is only part of the process. Optimizing the manufacture of these parts is another process you can perform in the Multiple Fixtures dialog box. Assuming that you already optimized the parts and setups, you can select Setups in the list and move them up and down in the process plan, or exclude/include them as needed.

After clicking OK, you can work with the view, run simulations, and fine tune the manufacturing as you would for any other part. And you can generate NC code for the multiple parts all in the same NC part program.

Macros for multiple parts

To incorporate macros in your program:

1. Open the Post Options dialog box by selecting *Post Process* in the *Manufacturing* menu.
2. Select a post processor that supports macros
3. Check the Enable Macros checkbox.

Now when you generate your NC code, it will contain macros.

Editing a multiple fixture design

Double click the multiple fixture display to open a tabbed dialog box containing the information from the Name, Configuration, Layout, and Stock dialog boxes. You can modify the spacing, shift spacing, nesting, and so on in the Properties dialog box. You can also change the setup name and fixture ID directly in the Name tab.

Selecting a part and using the Alt+Enter shortcut opens the Multiple Fixtures dialog box.

Chapter 20
3D Surface Modeling

You must license FeatureMILL3D to use the functions described in this chapter.

Surface definition

To build and use surfaces in FeatureCAM, you need to understand how FeatureCAM defines a surface. A surface is defined by a rectangular set of points. The set of points determines the shape, or geometry, of the surface.

The rectangular nature of the control point mesh means that a surface has four boundary curves. In some cases, surfaces are constructed where one or two (opposite) boundary curves collapse into a single point such as the poles of a sphere. In other cases where surfaces wrap around, such as a cylinder, two opposite boundaries can be the same curve and are called seams. With seams collapsed edges, a surface may appear to have only three, or two, or even no boundaries (a sphere), but the four boundaries are always defined. With these four boundaries, you can break a surface into rows and columns so surfaces have a table-like structure.

Surface wizard

You can build surfaces from the Surface toolbar, the wizard or selecting Surfaces in the Construct menu. With the surface wizard:

1. Select the category of surfaces from among:

- From curves
- Predefined surfaces
- From one surface
- From multiple surfaces

2. Pick a construction method from that category.
3. Click *Next*. The surface construction dialog box appears.
4. Look through the construction methods and verify that they will construct the kind of surface you want.
5. If the methods don’t match what you are trying to do, click *Back* and use a different method. The same surface can often be made in different ways.
6. When you have the construction method you want, fill in the fields with the correct information. Surfaces use curves, other surfaces, points, surface normals, and other kinds of user specified information to create your part model.
7. Click *Finish* to add the surface design to the part model.

Surfaces from curves

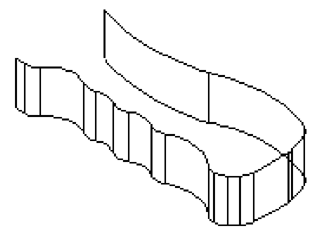
Extrude surface

Extrude creates a surface from a curve by extending that line sideways a specified distance. Sideways can be a linear distance in any direction. This is important as the extrude doesn't have to be only in the direction of an X,Y or Z axis. Extruding along an axis is an easy way to build such a surface that you could then move or transform into the final position if you prefer. Extrudes are a shortcut to create a ruled surface between a curve and a transformation of the curve.

Extruded surfaces are exact.

Extrude sample

The handle on the blow dryer is an extruded surface. The curve for the handle was drawn with the arcs and lines tools. The geometry was chained into a single curve which was then transformed down in Z to the depth where the handle exists. This transform operation puts a bounding curve in location for the top surface of the handle as well.



Coons surface

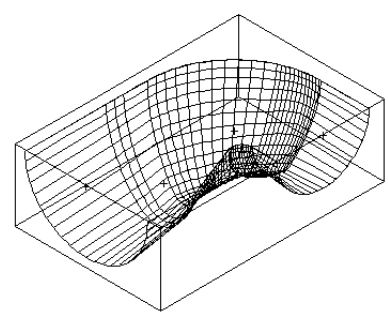
A Coons surface defines a surface between four bounding curves. For planar curves, Cap surface is probably a better option, although the curves must be joined into a loop first. For a grid of curves, Coons surfaces do not produce smooth results. Use Curve Mesh or join the curves into cross-sections and use Lofted surfaces instead.

When using the three curve Coons option, the ordering of the curves makes a difference in the appearance of the surface. Try different sequences until the result is correct.

Chained curves behave differently in a Coons surface than spline curves. If your surface doesn't look quite right in the preview set the Reparameterize curves checkbox and try the Preview again. Reparameterize curves analyzes the curves and may adjust the control points of the curves to yield a better surface result. Reparameterize affects both chained and spline curves, but the effect is stronger on chained curves. Coons surfaces are approximate.

Coons example

You might use a Coons surface to create a trough shape that necks down as shown on the right. This is easy to draw as two bounding curves on the stock surface, and two arcing end curves. You don't have as much control over the behavior of the surface between curves as if you had used a lofted surface to create a similar shape.



Ruled surface

A Ruled surface creates a linear surface between two curves. The curves can be open or closed, planar or non-planar. For closed curves, the starting points of the curves should line up or the surface may twist in odd ways.

Ruled surface defines a surface by plotting rules between two different curves. The curves can be open or closed, planar or non-planar. For closed curves, the starting points of the curves must line up or the surface may twist in odd ways. See *Twists in surfaces with closed cross sections* for more details.

Ruled surfaces are useful for filling a region and blending between two curves and can be used with more than two curves to create a single surface where each section is the same as if each pair of curves were used to create a ruled surface.

Chained curves behave differently in a ruled surface than spline curves. If your surface doesn't look quite right in the preview set the Reparameterize curves checkbox and try the Preview again. Reparameterize curves analyzes the curves and may adjust the control points of the curves to yield a better surface result. Reparameterize affects both chained and spline curves, but the effect is stronger on chained curves.

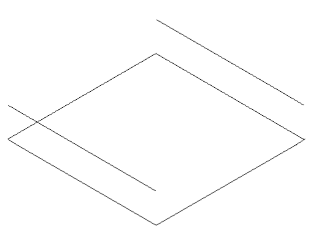
Ruled surfaces are exact.

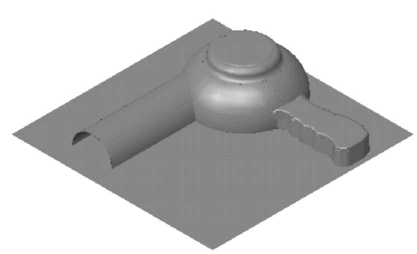
Follow these steps to build a ruled surface:

1. You must have at least two curves before you can create a ruled surface fly-out.
2. Use the Surface wizard or select Ruled from the surface toolbar.
3. Name the surface for future reference.
4. Click the Pick curve  button and pick the curve or select the name of the curve in the Curve list and click the plus  button.
5. Continue picking curves either with the Pick button and mouse or from the drop down list. If you use the list, use the Plus button to add the curve to the list.
6. Use the Up, Down, Reverse, and Delete buttons to set the sequence and direction of the curves in the ruled surface.
7. Generally you will want to set *Reparameterize curves*. If the correspondence between neighboring curves seems wrong, try turning this option off.
8. Click *Preview* to see how the current settings affect the surface. Make adjustments as necessary until the surface is how you want it.
9. Click *Finish* or *OK*.

Ruled example

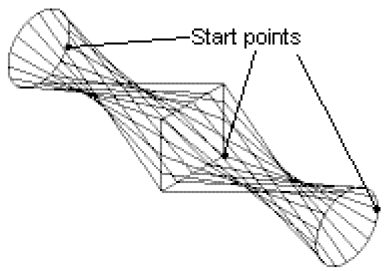
The blow dryer uses a ruled surface as a parting surface in the model. The two curves (lines in this case) are drawn along opposite sides of the stock and are then translated down in Z to the appropriate depth. Then they are selected as the two curves for the ruled surface.

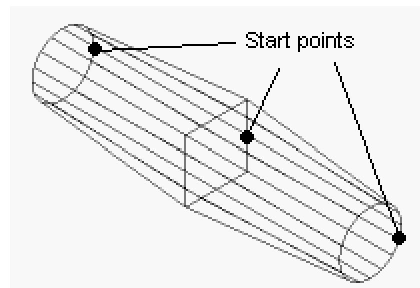




Twists in surfaces with closed cross sections

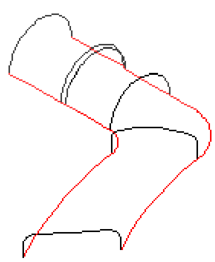
When creating ruled surfaces of lofted surfaces or solids with closed cross sections, you want to make sure that the start points of the curves line up with the way you would like to create the lines of your shape. The figure on the left shows a ruled surface created from cross sections with misaligned start points. After using Curve start/reverse to change the start point of the square center curve, the twist is removed, as shown on the right.



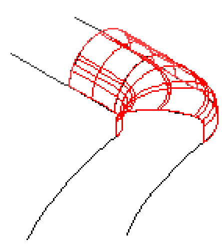


Curve mesh

The curve mesh constructor makes a smooth surface from a mesh of "row" curves and "column" curves. For example, if you have the curves shown below, the curve mesh constructor will create the surface shown in the right-hand figure below.



The mesh of curves may only partially intersect. This allows the surface to be built up incrementally. The surface shown below is an example of such a surface.



The surface is made up of Coons patches. If the input curves are smooth, the surface will be smooth (equal tangent plane planes along the curves). The input curves must intersect at crossing points, but explicit curve points are not necessary at the crossings. The curves are reversed automatically. Triangular patches are supported when a pair of row or column curves meet at their ends.

How to create a surface from a curve mesh

1. The curves of your mesh must already exist in position in the drawing before you can create a surface.
2. Use the Surface wizard or select *Curve mesh* from the surface toolbar fly-out.
3. Optionally name the surface for future reference.
4. Click the Pick curve  button and pick the curve or select the name of the curve in the Curve list and click the plus  button.
5. If you pick them out of sequence, change the sequence with the Up and Down arrow buttons.
6. Pick the Column curves with the mouse and pick button, or choose them in the list and click Plus.
7. If you pick them out of sequence, change the sequence with the Up and Down arrow buttons.
8. Specify an acceptable Tolerance. This tolerance is used in approximating some of the curves.
9. Click *Preview* to view the surface.
10. Click Zero twists if you want the surface to be flatter.
11. Click *Preview* to see how the current settings affect the surface.
12. Click *Finish* or *OK* to create the surface.

Restrictions of surface from curve mesh

1. The input curves must intersect at crossing points, but explicit curve points are not necessary at the crossings.

Cap surface

Cap surfaces can be used to make a cylinder into a closed solid, for example. Extract the boundary curve at each end of cylinder and make a cap surface out of each one. You can also create closed solids from extrudes, sweeps, and revolutions.

Supporting multiple curves in the cap surface allows you to nest your curves inside of other curves. Depending on the direction of the curve, you can then include cutouts and surfaces within cutouts all in one operation.

A cap surface may be either exact or approximate depending on your settings.

Cap surface creates a surface to cover an “open end” of another surface or surfaces. The surface(s) to be capped should form a closed surface by itself, or when considered as a group. While you can cap open surfaces, a straight line is drawn between the open points of the surface, and can generate inappropriate surfaces in cases where the endpoints cause the closing line to cross the surface boundary.

Cap example

The blow-dryer has a cap surface on the handle, but the handle is an open curve. So you have to close the curve and create a new curve from the handle curve and the geometry you used to close the curve. Snapping a line between the endpoints creates the needed geometry. Chain it all into a curve and build the Cap surface.

Lofted surface

Lofting has its heritage from shipbuilders as they laid out ship hulls. They passed imaginary lines between multiple sequential cross-sections. A lofted surface creates a smooth surface from cross-sectional curve data. The curves can be non-planar as well.

Lofting defines surfaces that project between multiple curves. A lofted surface is used to create a smooth surface from cross-sectional curve data. The curves can be open or closed, but for closed curves the start points need to line up for good results. The curves can be non-planar as well. See *Twists in surfaces with closed cross sections* for more details.

Lofted surfaces are commonly used when there are many uniformly spaced cross-section curves available. A lofted surface has to fill in a lot of data between the cross-sections and so is an approximate surface. The data calculated to pass through the surface can be tweaked with the Uneven spacing switch for a better fit when the data points are not uniformly spaced.

The curves can be open or closed, but for closed curves the start points for each curve need to line up with the other curve start points for good results. The start points for closed curves are not obvious. If you build the curve with Curve join or with a closed spline, you can control the start point. Re-chaining the curve where you want a start point may work, but is not guaranteed because some of it depends on the characteristics of the curve. You can see the start point of a closed curve by looking at it in the Curve Reverse dialog box.

Lofted surfaces are commonly used when there are many cross-section curves available. A lofted surface has to fill in a lot of data between the cross-sections so it is an approximate surface. The data calculated to pass through the surface can be tweaked with the Uneven spacing switch for a better fit when the input curves are not uniformly spaced.

Spline approximates the surface with input curves as control points/curves. They are smooth between points. Interpolate uses the input curves as explicit curves for the surface to pass through and the surface may wave, or bend between points. Setting the Uneven spacing switch might improve the fit if the input curves aren't uniformly spaced. A lofted surface may be either exact or approximate depending on your settings.

The Select degree spinner allows you to vary the tightness by changing the degree of the polynomial used to calculate the resulting surface. A degree of one passes straight lines between curves (like a ruled surface). Higher degree curves allow a looser result between input curves. The highest degree possible is three or one less than the total number of curves, whichever is greater. If you are going to export the part file to another CAD package, some other software doesn't support degree values higher than three.

Chained or joined curves behave differently in a lofted surface than spline curves. If your surface doesn't look quite right in the Preview, set the Reparameterize curves checkbox and try the Preview again. Reparameterize curves analyzes the curves and may adjust the control points of the curves to yield a better surface result. Reparameterize affects both chained and spline curves, but the effect is stronger on chained curves. Lofting shouldn't be used when a sweep, or other more exact constructor could be used.

Follow these steps to create a lofted surface:

1. The curves for the lofted surface must already exist in position in the drawing before you can create a lofted surface.
2. Use the Surface wizard or select Lofting from the surface toolbar fly-out.
3. Name the surface for future reference.
4. Click the Pick curve  button and pick the curve or select the name of the curve in the Curve list and click the plus  button.
5. If you pick them out of sequence, change the sequence with the Up and Down arrow buttons.
6. Specify an acceptable Tolerance. Tolerance means that the resulting surface may deviate from the exact offset by no more than value you enter.
7. Click Preview to view the surface.
8. Set either the Interpolated or Spline radio button. Use Preview to see the difference.
9. Set the Uneven spacing switchbox as needed for the best surface match. Use Preview to see the difference.
10. It is quite possible that your surface will flip between adjacent curves. Use Curve reverse to reverse the curve. If you have multiple flips, do each curve separately as it is easy to get confused about how the surface might flip after you have reversed many curves.
11. Decide whether to set Reparameterize curves.
12. Click Preview to see how the current settings affect the surface. Make adjustments as necessary until the surface is how you want it.
13. Click Finish or OK to create the surface.

Lofted example

The soap dish model uses a lofted surface. Its source curves are worth understanding. Using a UCS on the side of the stock as shown in the diagram, simple geometry was drawn at depth (relative to the setup) in Z. The geometry was then transformed in X to set the begin, middle and end curves of the top surface of the soap dish electrode. These curves were then used to build the lofted surface. The geometry may have to be reversed, to get the right surface.

Surface of revolution

A surface of revolution is created by spinning a curve about an axis. The revolution is any amount from -360 to 360 degrees. These are similar to a swept surface, and can be used to create other primitive shapes not provided such as a torus or a cone. These surfaces are exact.

While you can use a 3D curve as the curve to spin around an axis, there is a higher chance of creating a self-intersecting surface. Where possible, it’s best to use a 2D curve for input to this surface.

The revolution is any amount greater than 0 up to 360 degrees. These are similar to a swept surface, but can be used to create other primitive shapes not provided such as a torus or a cone.

These surfaces are exact.

Revolved surfaces may have no cap on the ends depending on the source curve. Depending on the milling technique you select, such as Z level roughing, you may need a cap surface.

Follow these steps to create a surface of revolution:

1. To create a surface of revolution, you need a curve and either the X or Y axis for rotation, or a custom line for the axis.
2. Use the Surface wizard or select Surface of revolution from the surface toolbar fly-out to open the dialog box.
3. Name the surface for future reference.
4. Click the Pick curve  button and pick the curve you are revolving.
5. Set the *Start angle* in degrees.
6. Set the *End angle* in degrees.
7. Pick the Construction method. This is where you set what is revolved around, or the pole. You can pick a custom axis, or the X or Y axis.

8. For a custom axis, pick the axis either from the list box or click the Pick line  button and select it with the mouse.

9. Click *Apply*.

10. Click *Preview* to see if you got the shape you wanted.

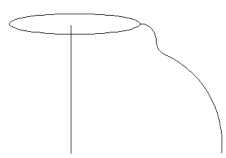
11. If the shape is acceptable Click Finish, or OK. Otherwise change some settings and preview again until the surface is right.

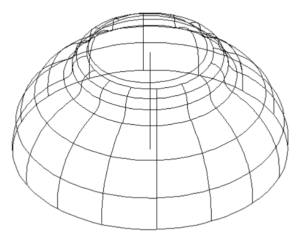
Revolved surfaces may have no cap on the ends depending on the source curve. Depending on the milling technique you select, such as Z level roughing, you may need a cap surface.

Revolved example

The motor housing on the blow-dryer model is a surface of revolution. The source curve for the housing was created from a top view and then rotated 90 degrees around the X-axis to orient the curve for the revolve construction. A line was drawn from the center of the housing down in Z to act as the axis for the revolution. Later, this model uses a cap surface to complete the housing as an example of the cap surface.

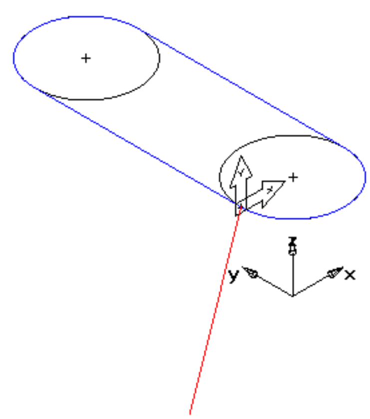
Set the surface for a full 360 degrees of revolution. Select the axis and the curve to pass around the axis and the result is shown on the right. You could also have created this same surface as a sweep around the circle.





If the source curve had extended all the way to the rotation axis, the housing could be completed in one step. A drawback to this method is that the central boundary would be a point and may not provide good machining results.

Swept surface



A sweep creates a surface by replicating the shape of the curve at multiple positions along a path. The path is not necessarily a straight line. A sweep moves one curve along another and is useful for making many shapes. By creating complex curves for both the axis and the cross-section, complex fillets and blends can be directly achieved in a single surface.

Swept surfaces are exact unless the path or axis curve is a spline curve, not a curve built from lines, and arcs and chained together. Along a spline curve axis, the shape may deform.

There are two kinds of sweeps: a regular sweep and a translational sweep. Translational sweeps maintain the same relationship between the curve and axis normals at all points throughout the sweep. Otherwise, the sweep curve stays in its drawn position at all points on the axis.

The swept curve needs to be hooked to the axis curve at the start point of the curve. The simplest way to create a curve at this location is to create a UCS at the start point of the axis curve and create your cross section curve in that UCS. The figure on the right shows an example of a cross section curve that is properly defined. A UCS was created at the start point of the axis curve. The cross section curve was then defined relative to this UCS. Note that the setup axes were not changed. The UCS is used as a design coordinate system only.

Swept surfaces are exact.

Follow these steps to make a swept surface:

1. You must have two curves before you can create a swept surface.

2. Use the Surface wizard or select Swept surface from the surface toolbar fly-out.

3. Name the surface for future reference.

4. Pick the curve that defines the axis the sweep will follow.

5. Decide and set whether to use a Translational sweep.

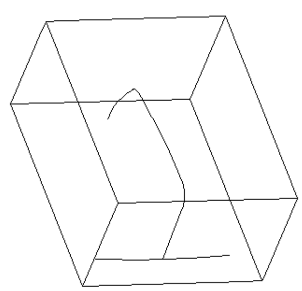
6. Pick the curve that defines the cross section that will be swept along the axis.

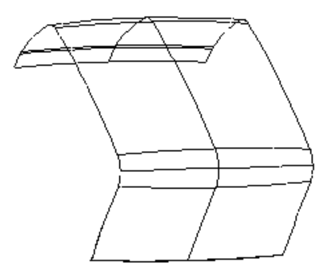
7. Click Preview to see whether you got the desired results. You might need to toggle the Translational sweep setting to get the surface you are looking for.

8. Click *Finish* or *OK*.

Sweep example

The speaker housing uses a swept surface to define the top, back, and bottom sides in one surface. The source curves are a large diameter circle segment (1000 inches) and a center line through the three sides (at maximum diameter of the segment). The Translational sweep switch was turned off for this surface to keep the bow in the same orientation relative to the axis throughout the sweep. The slightly bowed bottom is flattened later as a Modify surface operation.



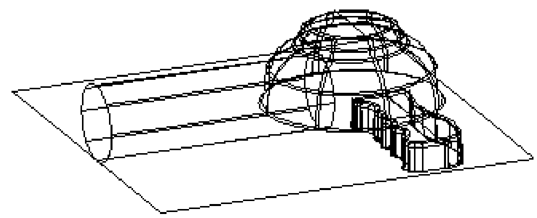


Surface primitives

Cylinders

Cylinders are exact surfaces but contain a seam where two edge boundaries meet. Be careful not to trim across the seam. Editing along the seam is fine.

Cylinder example



The vent tube of the blow dryer can be modeled as a cylinder. Pick end and center points with the mouse or pick buttons, or enter explicit coordinates. Remember to set the points 2 inches in Z in the stock to create the cylinder at the right depth. Now set the radius to 1 inch. The cylinder can be trimmed to the rest of the model with the ruled surface.

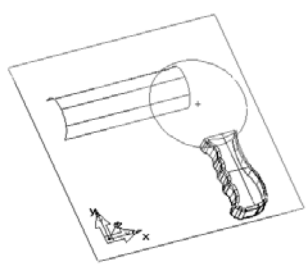
Spheres

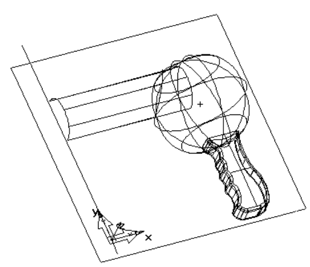
Sphere constructs a spherical surface around a center point and of the specified radius.

Spheres are exact surfaces, but contain seams and degenerate points at their poles.

Sphere example

The blow dryer could be redesigned to use a spherical motor housing. So if the design called for 2.25 inch diameter sphere instead, you could create such a surface by drawing the 2.25 inch diameter circle at depth in the block. Now use that circle to construct a sphere, which you could trim with the ruled parting surface.





Flat surface

Flat creates a rectangular surface between two diagonally opposite corners. This surface is a shortcut to create a ruled surface without having to build bounding curves if the desired surface is rectangular. Flat surfaces are exact.

Follow these steps to create a flat surface:

1. Flat surfaces require no pre-existing curves or geometry.

2. Use the Surface wizard or select Flat surface from the surface toolbar flyout.

3. Name the surface for future reference.

4. Enter the coordinates for a corner point, or click the Pick point {bmct btn-pick-xyz.bmp} button and pick the point with the mouse.

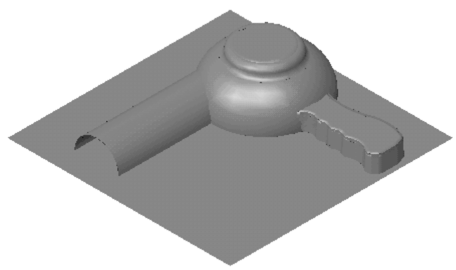
5. Enter the coordinates for the diagonally opposite corner point, or click Pick point button and pick the point with the mouse.

6. Enter an (optional) elevation offset for the surface. You can also pick the z coordinate of any point using the Pick point button.

7. Click Finish or OK to create the surface.

Flat example

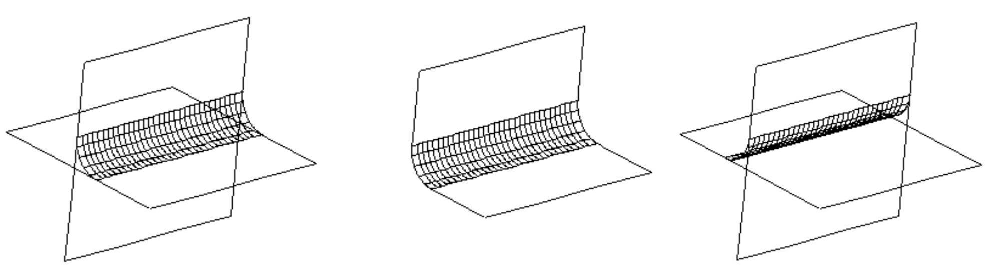
The blow dryer's parting surface can also be built with a flat surface. Simply create a flat surface at depth with corner points that align with two opposing block stock corners.



Surface from multiple surfaces

Fillet

A fillet is a surface that creates a smooth tangent-continuous blend between two surfaces. The left-hand figure shows a fillet surface that blends two flat surfaces. The middle figure shows the same fillet with the surfaces automatically trimmed against the two fillet boundaries. Just like the case of an arc between two lines, there is more than one possible fillet between two surfaces. The right-hand figure shows the same two flat surfaces with a fillet on another corner.



Setting the Begin radius and End radius to the same value creates a constant radius or rolling ball fillet. Entering different values for the two radii creates a variable radius fillet.

Filleting multi-step process:

1. Intersect the two surfaces. If you explicitly specify the intersection curve this step is skipped. The most common reason for explicitly entering the intersection curve is to calculate the curve separately and then extend it so that the fillet extends beyond the surface boundaries.

2. Construct fillet surface

3. Optionally trim one surface against the top of the fillet. The top boundary curve of the fillet must be a valid trimming curve for the top surface.

4. Optionally trim the other surface against the bottom of the fillet. The bottom boundary curve must be a valid trimming curve for the bottom surface.

It is possible to create a valid fillet surface that does not allow you to trim the surfaces against the fillet. In this example, the fillet is in the middle of both surfaces and as a result the boundaries of the fillet are not valid trimming curves.

How to create a fillet surface

1. You must have two surfaces that intersect to create the fillet between.

2. Use the Surface wizard or select Fillet surface from the surface toolbar fly-out.

3. Name the surface for future reference

4. Click Pick and select surface 1, or select the surface from the dropdown list box.

5. If you want to trim surface 1 with the fillet, set the Trim to fillet check box. Set the Trim Tolerance to affect the accuracy of your trimming. It is recommended for most parts (less than 1 foot cubed) that you set this between 0.001 and 0.0001 inches (0.0254 to 0.00254 mm).

6. Click Pick and select surface 2, or select the surface from the dropdown list box.

7. If you want to trim surface 2 with the fillet, set the Trim to fillet check box. (See step 5 regarding trim tolerance.)

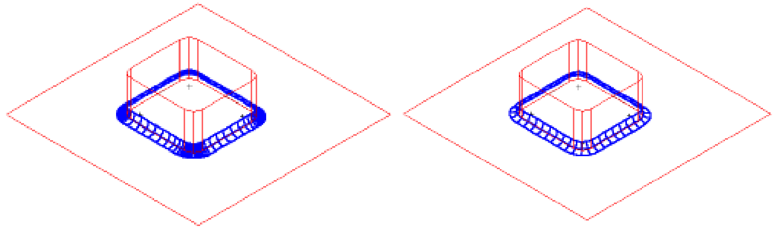
8. You may also set an Intersection to limit the range of the fillet, or to clarify which intersection of the surfaces to consider. Intersection places the fillet along the curve described by the intersection of the surfaces. This curve must actually exist in the drawing.

9. Set the Arcs only construction switch box if you want to create just the arcs that would comprise the fillet. You could then combine these arc curves to create other surfaces.

10. Choose the Corner for the fillet. Corner refers to the drawing in the dialog box to help you specify which of the four possible sides of the intersection you want the fillet to be in.

11. Set Tolerance for how much the resulting surface may deviate from the exact fillet. It will vary by no more than value you enter. If you want high quality fillets for parts that are approximately 1 foot cubed, the recommended settings are between 0.001 and 0.0001 (or 0.0254 and 0.00254 mm).

12. Define the Arc step of the fillet. If you select Variable, the spacing of the fillet cross sections will be calculated automatically based on the Tolerance. More cross sections will be placed in curved regions of the fillet as in the left-hand figure below. If you uncheck the Variable button, you must set an Arc Step. The spacing will then be constant between cross sections as in the right-hand figure.



13. Set the Begin radius. Having two settings for radius lets you create a fillet that grows or shrinks along the path of the fillet as needed. For a closed fillet, the Begin and End radius need to be the same.

14. Set the End radius of the fillet as applicable.

15. Click Finish or OK to create the filleted surface.

Fillet restrictions

1. The fillet constructor works with two surfaces only.

2. The surfaces must intersect.

3. If trimming, the top and bottom boundary curves of the fillet must form a valid trimmed surface in the respective surfaces. See *Trimming restrictions* for the rules of valid trimming curves.

4. Fillet creation does not consider trimmed portions of a surface so if you fillet a trimmed surface, the fillet will extend for the complete length of the original surface. You can either trim the fillet away after it has been created or use an intersection curve to limit where the fillet runs.

5. The input surfaces must be smooth with no sharp corners.

6. The fillet radius should be less than the smallest radius of the intersection curve. If it is larger, the fillet may overlap itself.

Surface from feature

You can create a 3D surface from any 2.5D feature. You can use these surfaces in your part modeling or even to manufacture them using the 3D techniques, or to modify with the Surface editing tools so you can join your 21/2D features to your 3D features.

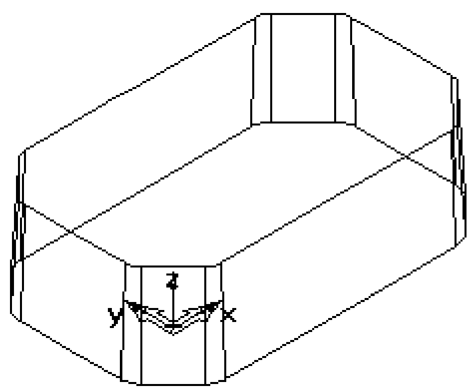
Features can be a shortcut to create swept and cap surfaces with the familiar dimension driven feature interface. The draft-angle, or radiused features produce a swept surface for the side walls (including chamfer and bottom/top radius) and a cap surface for the bottom in one operation.

Follow these steps to create a surface from a 2½D feature:

1. You must have a 2½D feature present in the part model.
2. Use the Surface wizard or select Surface from feature in the surface toolbar fly-out.
3. Name the surface for future reference
4. Click Pick and select the source feature, or select the feature from the dropdown list box.
5. Click Finish or OK to create the new surface.

Feature to surface example

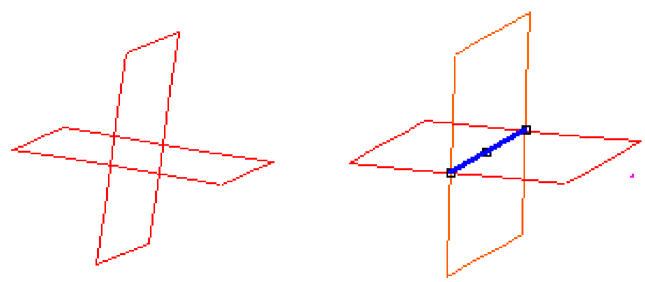
The soap-dish model can be built with a Boss feature for the side. In that method, you need to trim the fillet against the boss, but you have to create a surface to trim against. Select Surfaces from Feature, name the new surface. Select the boss feature as the source feature. FeatureCAM calculates the corresponding surface.



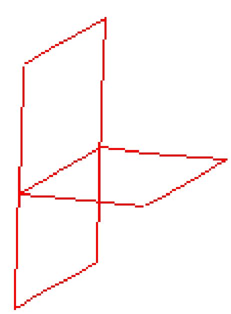
Surface-surface trimming

Surface/surface trimming has two types of surfaces, trimm**ing** surfaces and trimm**ed** surfaces. The trimming surface is the surface that does the cutting. Think of it as a pair of scissors or a knife. The trimmed surfaces are the surfaces that are cut. Think of them as the paper. You can have only one trimming surface, but you may have more than one trimmed surfaces.

The first step of surface/surface trimming is calculating the intersection of the trimming surfaces and the trimmed surface(s). This step happens behind the scenes and is only visible if you are previewing your calculation before actually performing it. This curve is then used to trim away a portion of the trimmed surfaces. If the surfaces shown in the left-hand figure are the original surfaces, the blue curve shown in the left-hand figure is the intersection curve between the two surfaces.



The surfaces shown in the left-hand figure are the result of using the vertical surface as the trimming surface to cut the horizontal surface. If you check the Trim this surface also checkbox, the trimming surface will also be trimmed using the intersection curve as shown in the right-hand figure.



How to trim surfaces against other surfaces

You must have two surfaces in the model to use Surface/Surface trimming.

1. Set whether to *Create a new surface* or *Modify the existing surface*.
2. If you're creating a new surface, name the new surface for future reference.
3. Click *Pick* and select a trimming surface, or select the surface from the dropdown list box. When you pick the surface with the mouse, pick the surface in an area you want to keep.
4. Decide whether to set the *Trim this surface also* check box. If so, then set the *Side kept* field to choose the correct side.

5. Click *Pick* to select the other surface, or select the surface from the dropdown list box and click Add to add it to the list. You can add more surfaces to the list the same way and trim multiple surfaces all at once.

6. If the side you selected the surface from isn't the side you want to keep, then set the *Side kept* field for the surfaces in the trimmed list. The options for *Side kept* are:

   - *Picked side* will keep the portion of the trimmed surfaces where you picked with the mouse.

   - *Normal* will keep the side of the trimmed surfaces that are on the opposite side as the normal of the trimming surface.

   - *Reverse* will keep the side of the trimmed surfaces that are on the same side as the normal of the trimming surface.

7. Set the *Tolerance* for the operation. This value specifies the distance between calculations along the intersection area. Set the tolerance lower if the intersection curve appears too coarse.

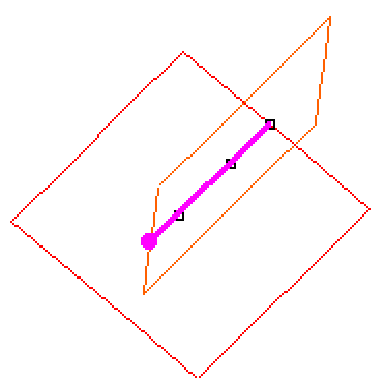
8. Click *Preview* to check your results against your expectations. You may need to work with the Side kept fields and multiple Previews to get the final surface you expect.

9. Click Finish or OK to complete the operation.

Surface surface trimming restrictions

1. The intersection curve is calculated using the surface/surface intersection technique. If you receive the message, *No surface-surface intersection*, then FeatureCAM cannot determine the intersection curve for your surfaces. For additional details see these restrictions.

2. If you receive the following message, *Can't trim, trim curve does not end on a boundary*, then an intersection curve has been calculated, but it is not a valid trimming curve for your surfaces. This figure below shows an example. See *Trimming restrictions* for all the restrictions for trimmed surfaces.



3. If you have multiple trimm**ed** surfaces, the intersection curve must form a valid trimmed surface for each of the trimmed surfaces.

4. If you have checked the Trim this surface also checkbox, then the intersection curve must form a valid trimmed surface with respect to the trimm**ing** surface.

Merged surface

You can merge with a blend so that the two surfaces transition into each other more gradually. Or you can make an exact merge so that the new surface keeps all of the original surface data of the two source surfaces.

Surfaces for a merge ideally have the same number of rows or columns so the two surfaces The exact option is best when the two surfaces share a common boundary curve. When they don't meet, exact inserts a ruled surface between the boundaries being merged.

If you had two neighboring flat surfaces, you could combine them with merge into one flat surface.

The blend option is best when the surfaces don't meet. A fillet-like blend is created to fill the gap. Merging automatically untrims trimmed surfaces. Merged surfaces may be trimmed again as needed.

The exact option is best when the two surfaces share a common boundary curve. When they don't meet, exact inserts a ruled surface between the boundaries being merged.

The blend option is best when the surfaces don't meet. A fillet-like blend is created to fill the gap. Merging doesn't work for trimmed surfaces. Surface can be untrimmed, merged and then retrimmed.

Modify surface

Modify works with the surface by manipulating links and surface curves. Modify works with the surface by manipulating links and surface curves. The links of a surface characterize the shape of the surface. The collection of links in one direction across the surface is known as a surface curve. When you select the surface to modify a number of blue lines appear with intersection points. These are the links and surface curves. A link is any segment between two intersection points.

Isolines are displayed on the screen, but they are not the surface curves that are edited. To see the isolines while editing, check the Show surface option. With Modify you can change specific points in the surface to fine tune your surfaces to match your needs. Changing the surface with Modify breaks the parametric link of the surface to the original construction data.

Follow these steps to use Modify surface:

1. You must have a surface in the part model to use Modify surface.
2. Click the Pick surface  button and select the surface, or select the surface from the dropdown list box.
3. Set whether to show the isolines or not with the Show surface option.
4. Select the type of operation you want to perform on the surface. Choices are:

- Change point
- Change link to line
- Change link to arc
- Add surface curve
- Delete surface curve

5. Click Finish or OK to complete the surface edit.

Change point

Change point lets you change any intersection point on the surface to any other point. Use the pick button and the mouse to select the point. Then type in the point's new coordinates. The surface is redefined to maintain the overall smoothness of the surface so the shape is still tangent-continuous.

This edit breaks the link between the surface and the method originally used to construct it. You can't open the surface's properties dialog box now and modify it based on the parameters that originally defined it as those parameters no longer apply directly to this surface.

Change link to line

Change link to line takes any selected segment and changes it to a straight line between the two points that bound the segment. The surface is recalculated to properly incorporate this change in its definition.

This edit breaks the link between the surface and the method originally used to construct it. You can't open the surface's properties dialog box now and modify it based on the parameters that originally defined it as those parameters no longer apply directly to this surface.

Change link to arc

Change link to arc changes a selected segment to an arc that follows the original segment path. This does not obviously change the shape of the surface but you can use it to control surface behavior between defining curves, for example.

This edit breaks the link between the surface and the method originally used to construct it. You can't open the surface's properties dialog box now and modify it based on the parameters that originally defined it as those parameters no longer apply directly to this surface.

Add surface curve

Add surface curve adds another surface curve to the surface you selected with the mouse or defined by coordinates. This is often a preliminary step for a subsequent modify operation based on the new surface curve if there wasn't a surface curve or link where you needed one.

This edit breaks the link between the surface and the method originally used to construct it. You can't open the surface's properties dialog box now and modify it based on the parameters that originally defined it as those parameters no longer apply directly to this surface.

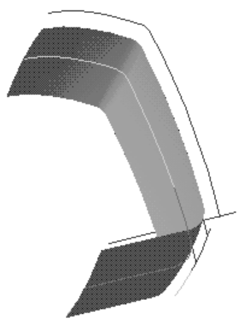
Delete surface curve

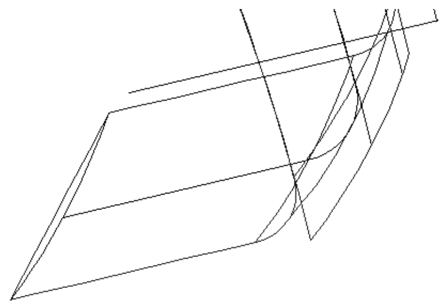
Delete surface curve removes a surface curve from the surface. This can drastically affect the shape of the surface. You can not delete any boundary.

This edit breaks the link between the surface and the method originally used to construct it. You can't open the surface's properties dialog box now and modify it based on the parameters that originally defined it as those parameters no longer apply directly to this surface.

Modify example

The speaker case, for example, has a slightly bowed bottom. With Modify, you can flatten that part of the model. Orient the model so you can see the bottom. The view of the model is shown on the left. On the right is an unshaded zoom of the model in the same orientation so you can see the result of the Modify operation. Using Modify, set Change link to line to make a straight line between the points and select the front edge of the bottom. Click Apply. Notice how the bottom surface flattened out in comparison to the source curve which is also visible. This surface can no longer be edited from the Sweep operation used to create it because the Modify operation breaks the relationship to the original data. Another Modify operation is needed to flatten the back edge of the surface.



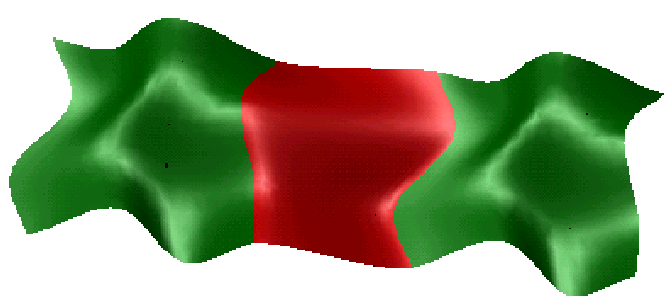


Blend surface

A fillet is a type of blend surface, but it requires that the two surfaces actually intersect. They also have the restriction that they can only blend two surfaces. A blend surface is a more general blend of 2, 3 or 4 surfaces. This surface constructor requires that you be familiar with creating curves on a surface and tangency, but it creates complex blends if these concepts are mastered by the user.

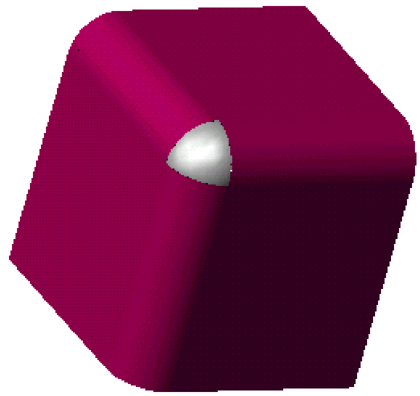
2 Surface Blends

For this blend you must supply the curve on each surface to create this surface.



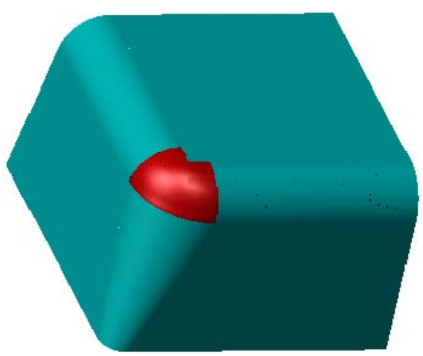
3 Surface Blends

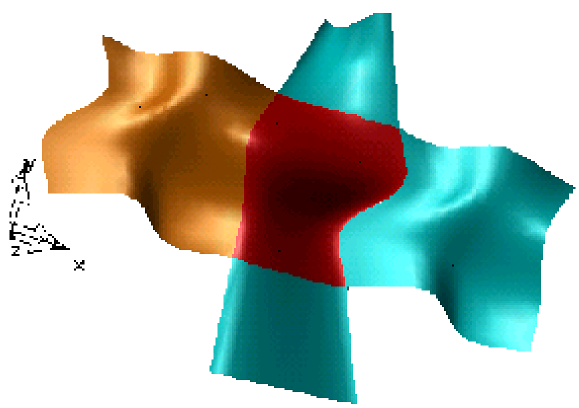
This blend is used to create what is known as a “suit case fillet.” This means that you want to create the intersection of three fillets like the corner of a suit case. If you create the fillets with FeatureMILL3D, the curves can usually be automatically detected.



4 Surface Blends

The left-hand figure shows another suit case fillet. In this case one surface is not a fillet. The curve for this surface must be specified. Four surface blends can also be used to join four distinct surfaces as shown in the right-hand figure.





How to create blend surfaces

1 Select *Corner Blend* from the Surface Wizard or *Curve and Surface* fly-out.

2 Optionally name the surface.

3 Select the number of surfaces you wish to blend as the Surfaces value.

4 Select each surface by either the drop down list or by picking the graphically using the pick surface  button.

5 For each surface, select the Curve on the Fillet by either using the drop down list or using the pick curve  button. Select ***Automatic* to have the FeatureCAM calculate the curves. See *Restrictions on blend surfaces* for further information.

6 Click *Apply* to preview the blended surface.

7 If the edges of the fillet are too coarse set the Tolerance to a smaller value.

8 If you want the fillets to be trimmed against the blended surface, click Trim fillets.

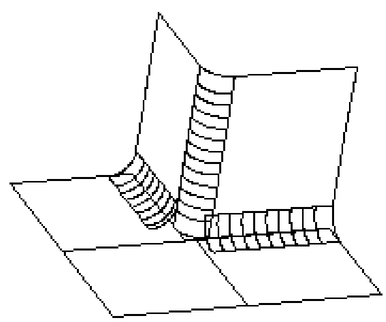
9 Click *Apply* again to preview the final blended surface.

10 Click *OK*.

Restrictions on blend surfaces

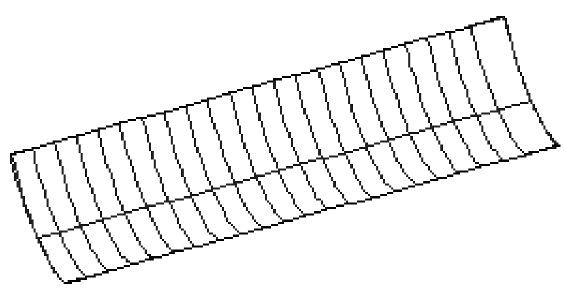
For three or four surfaces, if you choose ***Automatic* as the curve to indicate that the system should automatically calculate the curve then the following restrictions apply:

1 The fillet surfaces must extend to where they intersect with the other fillet surfaces. This figure shows an example of surfaces that do not intersect.



2 If the surfaces are not a FeatureCAM-created fillet (either not a fillet at all, or an imported surface), the automatic corner boundary calculation may fail. In these cases it is better to specify the curve-on-surface for that particular surface.

3 FeatureCAM attempts to find a closed loop across the three or four surfaces by intersecting the rail curve boundaries of each fillet with the other fillet's rail curves. The figure below shows rail curves. Therefore, if any of the curves on surfaces are specified, they must intersect with these rail curves of the fillet surfaces where a curve on a surface is not supplied.



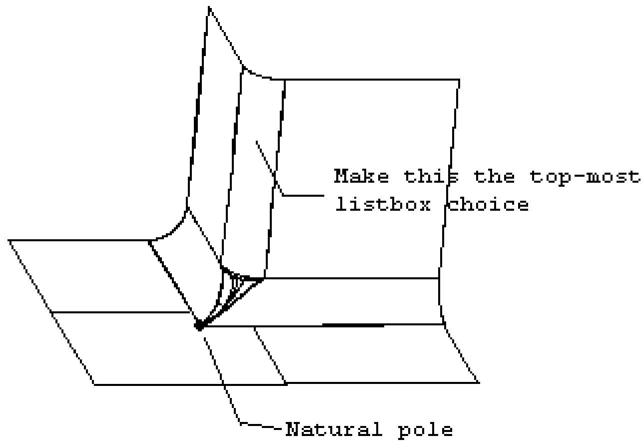
4 If a fourth surface is supplied, it is usually not a fillet surface. For these surfaces you must supply a curve on the fourth surface. Remember that the curve supplied must be on the fourth surface and should be tangent to the rail curves or the neighboring fillets.

5 If the automatic calculation fails, try a bigger tolerance, supplying more curves on the surfaces or extending the fillet surfaces so that they intersect. Sometimes it is helpful to recreate the fillet surfaces with a tighter tolerance or a variable arc step.

General restrictions

1 With only two surfaces, you must indicate curves on each surface to connect.

2 For three sided cases, if there is a natural pole as shown below, it is best to but the fillet surface that is furthest from the pole at the first surface.



Surfaces from surfaces

Extended surface

Extend adds a linear extension to the selected surface similar to extruding the boundary curve of the surface however far you set. The direction of the extend is the tangent direction at the boundary. Remember that the surface is defined with rows and columns as you select which boundary to extend. Surface extends create extra surface to fill gaps or to be trimmed away cleanly by a nearby surface. For example, open fillets often have to be extended slightly before they will cleanly trim the surfaces they fillet (and so they can be cleanly trimmed by another surface). Extend is an exact operation.

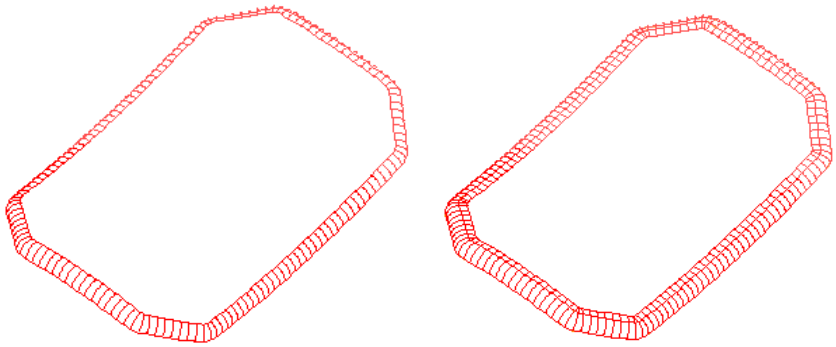
Follow these steps to extend a surface:

1. Extend requires an existing surface to work with.
2. Use the Surface wizard or select *Extend surface* from the *Curve and surface* toolbar fly-out.
3. Set whether you want to modify the existing surface or create a new surface from the process.
4. If you are creating a new surface, name the surface for future reference.
5. Click the Pick surface button  and select the source surface, or select the surface from the dropdown list box.
6. Select the edge to extend either with the radio buttons, or click Pick and select the edge with the mouse.
7. Set the distance for how far to linearly extend the surface.
8. Click Finish or OK to create the new surface.

Extend example

Consider the fillet in the soap dish model. If the top surface were undersized for some reason, or the fillet radius didn’t reach to the top surface, an extend operation could resolve the problem. In this case, select the Extend surface option from the Surface toolbar.

Unless you're sure of your process, it's safer to create a new surface instead of modify an existing surface. Set the Create new surface radio button and name it. Working through column and row options and previewing the results, the intended surface is achieved using First Column and an extension of 0.1 inch.



Split surface

A sub-region of a surface is sometimes necessary for an auxiliary surface to use for surface/surface intersection, filleting, milling, and so on. Split is an exact operation within the limits of the original surface. Splitting the Top surface of the soap dish model creates two surfaces.

Surface offset

An offset surface is a constant distance away from the original. The offset produces an approximation of this, which is why there is a tolerance. The original surface must be smooth (no sharp corners) or the offset result is not correct.

Use offsets to "thicken" an object or to account for shrinkage in mold making.

Use offsets to intersect two (offset) surfaces to get the center curve of an implied fillet. This curve could be used for a groove milling operation to cleanly mill a fillet without actually making the fillet surface.

If an offset can be produced by an exact operation, for example, offsetting a curve and using revolution or a sweep, then use the exact operation.

Surface reverse

Because of the rectangular definition of surfaces, you have three options in the surface reverse process. You can keep the same surface, but reverse the direction of the calculated normals, thereby turning the surface inside out. This is perhaps the most common function as it affects isoline milling

You can reverse the layout of the surface by swapping all the row and column layout with each other. You can also reverse the trim loops of the surface. This process takes a trimmed surface and changes the trim operation so that what was trimmed away before is now the retained surface and the discarded surface is the surface you selected for the surface reverse operation.

Surface reverse is an exact operation. Follow these steps to reverse a surface:

1. You must have an existing surface to use Surface reverse.

2. Use the Surface wizard or select Surface reverse from the surface toolbar flyout.

3. Set whether to create new surface or modify an existing surface.

4. Name the surface for future reference if it's a new surface.

5. Click Pick and select the source surface, or select the surface from the dropdown list box.

6. Select whether you want to:

   - reverse normals

   - transpose row/column

   - reverse trim loops

7. Click Finish or OK to create the new surface or edit the existing surface, whichever matches your settings.

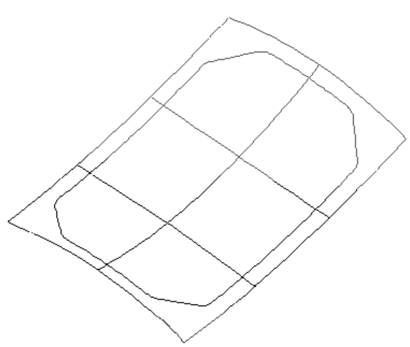
Sheets can also be reversed using the Surface Reverse operator. They cannot be selected with the mouse, but instead should be picked from the drop down list box in step 5 above.

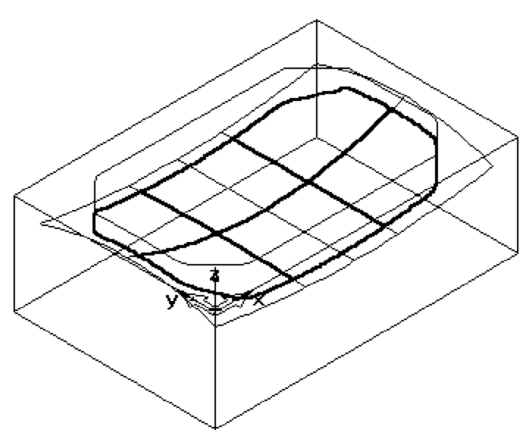
Trimmed surface

Overview of trimmed surface

A trimmed surface is a surface that has a portion removed by an embedded curve. The curve will divide the surface into two pieces. For example a square surface with a circular trimming loop will form either a square with a hole in it or a disk depending on which portion of the surface you want to keep.

The surface shown on the left is the top portion of a soap bottle mold. It must be trimmed by the curve to reflect the correct shape. This surface on the right shows the result of trimming the outer surface with the curve.





Trimming a surface with a curve

1. To trim a surface, you must have a surface and a curve that bounds the area to be

trimmed in the part model. The curve must cut across the surface boundary at two points for an open trim, or be contained completely within the surface.

2. Use the Surface wizard and set the first radio button to From one surface and the second button to Trim. Or use the From one surface fly-out and select Trim.

3. Set whether to Create a new surface or Modify the existing surface.

4. Name the new surface for future reference.

5. Click the Pick surface button  and select the source surface, or select the surface from the dropdown list box. and select the source surface, or select the surface from the dropdown list box. **The location that you select will determine the portion of the surface that is kept.**

6. Click the Pick curve  button and select the trimming curve, or select the curve from the dropdown list box.

7. Use Preview to verify the surface. If it's not right, reset some of the options and verify it again with Preview. You may need to toggle the Side kept switch to trim the desired part of the surface.

8. Click Finish or OK to create the new surface.

How to trim surfaces against other surfaces

1. You must have two surfaces in the model to use Surface/Surface trimming.

2. Set whether to Create a new surface or Modify the existing surface.

3. If you're creating a new surface, name the new surface for future reference.

4. Click the Pick surface button  and select the source surface, or select the surface from the dropdown list box. and select a trimming surface, or select the surface from the dropdown list box. When you pick the surface with the mouse, pick the surface in an area you want to keep.

5. Decide whether to set the Trim this surface also check box. If so, then set the Side kept field to choose the correct side.

6. Click the Pick surface button  to select the other surface, or select the surface from the dropdown list box and click Add to add it to the list. You can add more surfaces to the list the same way and trim multiple surfaces all at once.

7. If the side you selected the surface from isn't the side you want to keep, then set the Side kept field for the surfaces in the trimmed list.

8. Set the *Tolerance* for the operation. This value specifies the distance between calculations along the intersection area. Set the tolerance lower if the intersection curve appears too coarse.

9. Click *Preview* to check your results against your expectations. You may need to work with the Side kept fields and multiple Previews to get the final surface you expect.

10. Click *Finish* or *OK* to complete the operation.

Trimming restrictions

Trimming curves have two main restrictions:

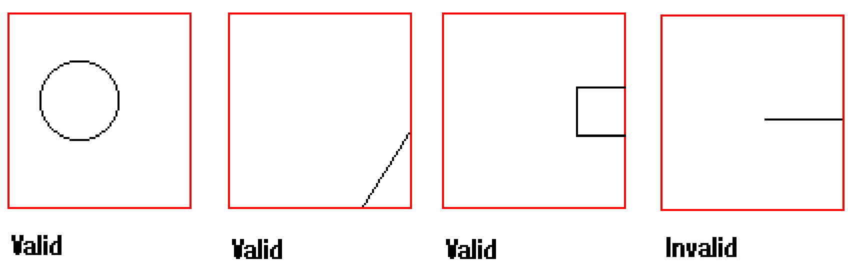
1. The curve must lie on the surface. Convenient ways of getting a curve on a surface are:

- If the surface is a flat surface, model the curve in the plane of the curve.
- Project a curve onto the surface.
- Intersect two surfaces to get a curve.

2. The curve must divide the surface in two distinct regions. If not, you will get the error *Can't trim, curve does not end on a boundary*. Think of the surface as a piece of paper and the trimming curve as a path for a pair of scissors. After the cutting operation, you should be left with two pieces of paper. The trimming curve must do one of the following:

- Form a loop in the interior of the surface
- Cut across two surface boundaries
- Cut across the same boundary twice

Trimming examples



1. Do not include portions of the boundary when constructing curves for trimming. Trimming curves may cross or end on boundaries.
2. Avoid cutting through the degenerate boundary a surface such as the pole of a sphere or surface of revolution.
3. Avoid cutting along (across is OK) a seam of a surface. A seam exists, for example, at the left/right boundary of a cylinder. You can detect seams with the curve extraction dialog box, and then canceling out after you know where the seam is. Avoid cutting a seam rotating the object because many objects with seams are symmetrical.

Comparison of Surface surface trimming and Trim a surface with a curve

Surface/surface trimming is more accurate and is the preferred method of trimming surfaces against each other.

Surface/surface trimming should be used:

f the intersection curve of the two surfaces is a valid trimming curve for the surface to be trimmed.

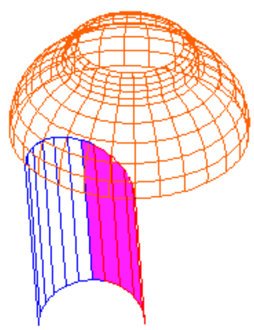
Surface/surface trimming performs surface/surface intersection as its initial step. The resulting curve is then used to trim the surfaces. This curve must be a valid trimming curve for the surfaces to be trimmed. See trimming restrictions for more information. If the intersection curve does not cut all the way across the surfaces to be trimmed, you may have to calculate the intersection curve separately and then join it with other curves to form a valid trimming curve for the surface.

Trim a surface with a curve should be used:

1. If you only have a curve with which to trim a surface.

2. If the surfaces are tangent at their intersection such as a fillet and its blending surfaces. In this case, extract the boundary curve of the fillet surface and use it to trim the other surface.

3. Where one surface needs to be trimmed by multiple other surfaces before a complete loop is formed. You may need to extract trimmed edges or calculate surface/surface intersection curves and join them to create the trim loop. In the example shown below, the revolved surface cannot be trimmed by either the left-hand or right-hand nozzle surfaces alone.

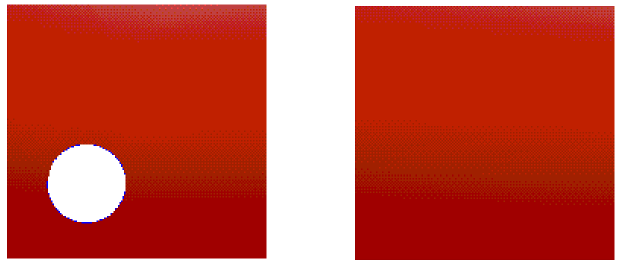


Instead:

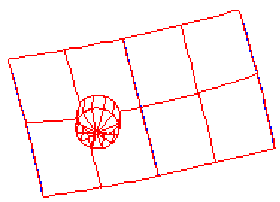
- Calculate the intersection curve of the left-hand nozzle surface vs. the revolved surface.
- Calculate the intersection curve of the right-hand nozzle surface vs. the revolved surface.
- Join the two curves.
- Trim the revolved surface with the joined curve.

Untrim surface

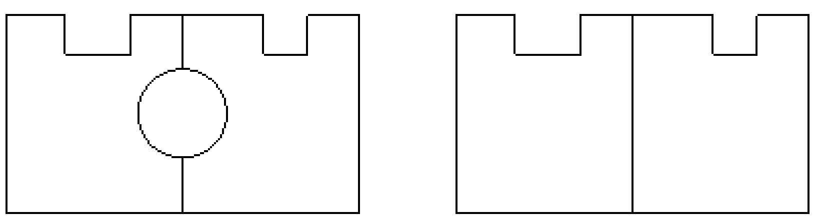
A trimmed surface is a surface with an embedded curve that removes a portion of the surface. When you untrim a surface you are removing the trimming curve from the surface and therefore adding the material back to the surface. The right-hand surface becomes the surface on the right after you remove its circular trimming loop.



If you import a part model, it is often useful to untrim portions of the part that you would like to machine with 2 ½ D features. For example, in the part shown below surfaces are used to represent the geometry of the hole. Since it is preferable to drill the hole using FeatureCAM's Hole feature, we can untrim the surface by removing the entire trimming loop and mill it as if the hole did not exist and then later add in a hole feature for drilling the hole.



For some parts, a feature such as a hole may result in trimming edges in multiple surfaces. In the figure on the right, a hole trims away two surfaces. Removing the entire outer trimming loop is undesirable since the notches at the top of the surfaces would also be removed. Instead, only the semi-circular edges from both surfaces need to be removed as shown on the left.



How to untrim a surface or fill a hole

1. You must have a trimmed surface or face of a solid available to untrim.
2. Use the Surface wizard or select *Untrim surface* from the surface toolbar flyout. All of the trimming loops are displayed.
3. Set whether to *Create a new surface* or *Modify the existing surface*.
4. Name the new surface for future reference.
5. To remove all trimming loops, click *Untrim all*.
6. To remove a single loop,

a. Click *Untrim selected loop,*

b. Click the Pick loop button. The dialog box warps into a button labeled *Untrim a trimmed….*

c. Select the trim curve(s) in the graphics window.

d. Double-click on the button labeled *Untrim a trimmed ….* The dialog box appears again.

7. Click *Finish* or *OK* to complete the operation.

Surface region

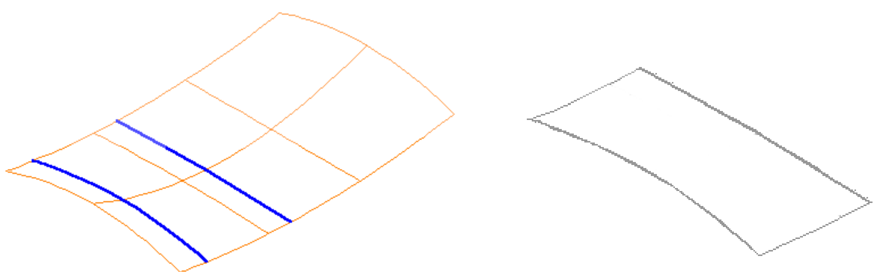
Surface region derives a second surface from part of an original surface. A subregion of a surface is sometimes necessary for an auxiliary surface to use for surface/surface intersection, filleting, milling, etc.

When you extract a region, you are getting a band of either columns or rows and this is an exact operation within the limits of the source surface.

Surface region is an exact process, however, if the source material is approximate, the resulting region can be no more precise than the source material.

Region example

For the top surface of the soap model, you could extract a row or column slice between any two points you choose. The picture below shows a potential region of rows from the Top surface of the model.



Surface design hints

Here are some concepts and techniques that have proved helpful in designing 3D surfaces.

- Design first, edit last. Put all the surfaces in place first, then trim and fillet the surfaces together to create the final shape and boundaries.
- Use quality source curves. Quality curves have as much detail as possible designed into the curves before the curves are used to make surfaces. Curves are much simpler to design and edit. With parametric modeling active, changes to your source curves propagate automatically into the surface.
- Build as much detail into a single surface as possible. It's easier to split and reduce surfaces than to merge and combine them. Similarly, machining with isoline operations uses one surface at a time and filleting and trimming work with two surfaces at a time so the more detail the better.
- Learn which surface methods are approximate and which are exact. Approximate methods are not better or worse than exact methods, but have more degrees of freedom in filling in the space between input curves or surfaces. Other surface methods are conditional. They are exact to what they do, but when used with approximate surfaces, the resulting surface is still approximate.
- **Approximate**: surface intersection, trim, untrim, fillet, lofting, merge, modify, offset (Another key is any operation that includes a tolerance setting. That is a flag that the source material is being approximated.)
- **Exact**: extrude, surface of revolution, sweep, ruled, sphere, cylinder, flat, from a feature
- **Conditional**: split, region, reverse, coons, cap. Coons is neither approximate nor exact, but a mathematical definition of how four boundary curves describe a surface. Learn which dialog boxes give you the option to modify-in-place or create a new object.
- Learn when and how to use the option to modify-in-place or create a new object. Both methods have their uses. Modifying an object in place breaks the parametric link for which constructor was used for the original object and prohibit parametric modeling from updating an object in the future. This limits you to further modification operations only. Creating new objects can result in excessive clutter on the screen.
- Don't use self-intersecting curves or surfaces. Curves and surfaces with that characteristic are not viable for predictable editing, construction or machining.

Surface editing hints

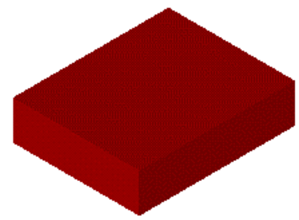
Here are some hints for using the Surface surface trimming, Trim, Surface/Surface intersection and Untrim operations.

- Do NOT include portions of the boundary when constructing curves for trimming. Trimming curves may cross or end on boundaries.
- Do not use surface/surface intersection to get a boundary of a surface. Use surface boundary extraction.
- Intersecting tangent surfaces produces inaccurate or broken intersection curves, such as a fillet surface and the surface it's tangent to. Instead, extract curves and trim with a curve instead.
- Do not intersect coincident surfaces (surfaces that overlap). Extract curves and trim with a curve instead.
- Avoid cutting through the pole of a surface with a degeneracy such as the tip of a sphere or surface of revolution.
- Avoid cutting along (across is OK) a seam of a surface. A seam exists, for example, at the left/right boundary of a cylinder. You can detect seams with the curve extraction dialog box, and then canceling out after you know where the seam is. Cutting a seam usually can be avoided by rotating the object.
- Surfaces can be trimmed multiple times.
- Curves for surface/curve trimming can extend off of the surface and/or cut the surface multiple times.

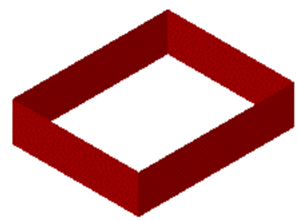
Chapter 21
Solid Modeling

Overview of solids in FeatureCAM

The FeatureCAM support of solids allows you to work with imported CAD designs from solid modeling systems and to create 3D solid models using FeatureCAM's solid modeling tools. Solids are a convenient representation for 3D parts since they group collections of surfaces into 3D volumes. By working with a solid instead of all of the individual surfaces, you are provided with a more convenient representation and more powerful modeling tools.

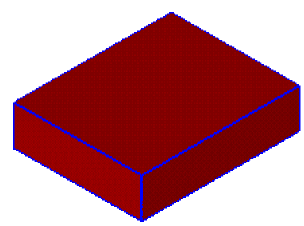


A *solid* is a collection of surfaces (called *faces)* that define a 3D volume. The edges that join faces are shared between the faces. A solid cannot have any holes. If you filled it with water and tumbled it around, it would not leak. For example, a box is a solid, but a box with a missing face is not. FeatureCAM automatically names these objects with the *solid* prefix like "solid123".

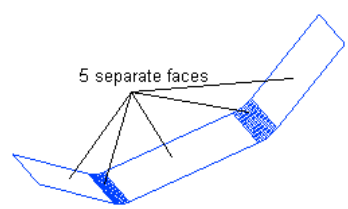


A *face* is an individual surface contained in a solid. FeatureCAM automatically names these objects with the *face* prefix as in "face472".

An *edge* is a curve that connects two faces. FeatureCAM automatically names these curves with the *edge* prefix like "edge857". These curves are not normally displayed, but they are displayed by dialog boxes that require an edge.



A *Sheet* is a group of surfaces that are created using the solid modeling tools, but that do not create a solid. Sheets have restrictions on how they can be used since they do not enclose a 3D volume. FeatureCAM automatically names them with the *sheet* prefix like "sheet902".



A *surface* is a single 3D surface that is created with the tools of the FeatureCAM surface wizard or imported from a CAD system (usually through IGES file transfer). These surfaces do not share edges with neighboring surfaces even though the boundaries may overlap. The edges individually defined in each surface. Certain collections of surfaces may be converted into sheets or solids using the design features available in the *From surfaces* category of the solid wizard.

Comparison of surface and solid modeling

The advantages of surface modeling are:

1. For representing curved shapes, surface modeling tools are more powerful. Operators

like Curve mesh, Ruled surface, cap surface and modify surface are only available for surface modeling. Surface lofting is more powerful than the solid fillet design feature.

2. With surfaces you are not forced to always work with 3D volumes. You can build your model face by face with surfacing tools without having to worry about 3D volumes.

The advantages of solid modeling are:

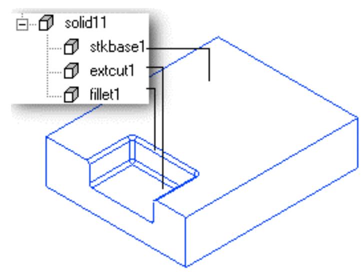
1. Many surfaces are grouped together in a single solid so solids can be a more efficient way to model.

2. Modeling operations that need to know the connections between surfaces are more powerful in solids. Filleting is stronger in solids since you only need to pick shared edges.

3. Useful modeling tools like shell, combine shells and silhouette curves are only available for solids.

Part View for solids

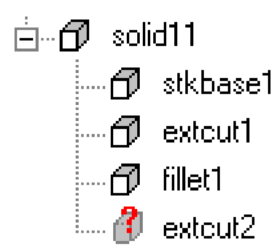
All solids are listed under the Solid category of the part view. Underneath each solid is a sequence of design features that were used to create the solid. Click the + next to the solid to reveal the design features. The first design feature listed for a solid is the *base* feature. Base design features define the initial shape of the solid. Base design features have the suffix, *base*, in their name. Subsequent design features listed under the solid further modify the shape of the solid. The prefix of a design feature name indicates the type of design operation that was used to create it. In the example below, *solid11* is made up of three separate design features. The first feature is named *stkbase1*. As the first feature of the solid, it is the base feature. The prefix, *stk,* indicates that the stock was used to create the feature. The second design feature is named *extcut1*. The name indicates that an extrude design operation was used to cut out the pocket shape. The final design feature is named *fillet1* and was created using a fillet design operation.



Note that the list of design features is only available for solids that are created in FeatureCAM. If you import a solid, only the resulting solid model is imported. You can access the faces of the solid, but you have no history of how the model was constructed.

Unattached design features

If a solid design feature becomes disconnected from the solid, a question mark is displayed over its icon. These disconnected design features are called *unattached* features. They can occur if you place a feature outside of the boundaries of a solid, or an edit to the solid makes a feature become unattached. For example, if you fillet an extrude feature and then delete the extrude feature, the fillet feature becomes unattached.



Unattached features do not cause any harm, but they are no longer contributing to the shape of the solid. If you select the solid, the faces of the unattached features are not selected

Selecting and deleting solids

Solids can be selected by:

- Clicking on the name of the solid or one of its design features in the part view; or
- Right-clicking on a face of the solid and selecting *Select solid* from the pop-up menu.

Clicking in the graphics window only selects a face of the solid.

Solids can only be deleted by selecting them and either pressing the DELETE key or selecting *Delete* from the *Edit* menu.

Verifying that a solid is valid

Occasionally, a solid model can be imported that looks good, but is not a valid solid. That can cause problems performing further solid modeling operations or recognizing features from the solid. To check that the solid is valid:

1. Select *Verify solid* from the *Solid* submenu of the *Construct* menu or click the Automatic Feature Recognition {bmct btn-steps-afr.bmp} button.
2. Click the *Verify* button.
3. A dialog box is displayed indicating if the solid is valid.
4. Click *Cancel* to remove the dialog box.

Solid wizard and solid toolbar

Solid modeling commands are available from either the solid wizard or the solid toolbar.

To work with the solid wizard:

1. Click Solid wizard  button to bring up the solid wizard.
2. Select a method of solid construction
3. Select a specific solid constructor
4. Click *Next* to bring up the dialog for the specific solid constructor.

If you run the solid wizard without a base solid defined, a page is displayed that helps you create the base solid. Your choices are:

- Use the stock as a base solid. A dialog box is presented for you to create the base solid from the stock or make another cube shape.
- Design a custom shape. In this case the normal solid wizard is presented.

To work with the solid toolbar:

1. Display the solid toolbar by selecting *Toolbars* from the *View* menu and checking *Solid* from Toolbars tab.
2. Select the constructor from the toolbar.

The specific solid constructors are:

1. From curves
    - Extrude solid design feature

- Revolved solid design feature
- Swept solid design feature
- Lofted solid design feature

2. From surfaces/primitives
   - Stitching
   - Solid from 2 ½ D feature
   - Solid from stock or cube
   - Extrude a surface
   - Revolve a surface
   - Sweep a surface
3. Shape modifiers
   - Solid Fillets
   - Cut solid with parting surface
   - Combine solids
   - Shell solid design feature
4. Manufacturing
   - Silhouette Curves
   - Select core/cavity
   - Split face
   - Delete face
   - Explode
   - Parting surface
   - Draft a face

Type of design feature

All of the solid design features created from curves have the following options:

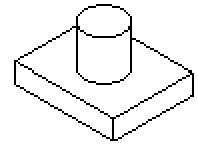
As new base solid – This will create an independent base solid that is not subordinate to any other solid.

As add – This will create the design feature and append it to the solid.

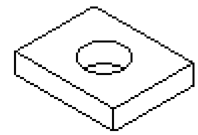
As cut – This will create the design feature and subtract it from the solid.

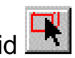
In this example a circle is extruded down into a block

If the extrude feature type is an **add** then the cylinder is trimmed against the surfaces of the box and the portions of the cylinder outside of the box is added to the solid.



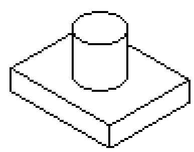
If the extrude feature type is a **cut** then the cylinder is trimmed against the surfaces of the box and the portions of the cylinder inside of the box are subtracted from the solid.

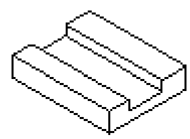


For add or cut, if there is more than one base solid, a pick solid  button and solid drop down list is presented for selecting the solid.

Extrude solid design feature

Extrude creates a solid design feature by pushing a curve in a straight line.   Below are examples of an extrude add with a circle as the curve and an extrude cut with a square as the curve.

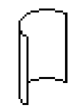




To create a solid extrude design feature:

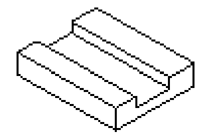
1. Select *Extrude* from the solid wizard or the solid toolbar.
2. Select the curve name in the *Curve* drop down list or click the pick curve  button and select the curve in the graphics window. Note that the curve must be planar.
3. Select the type as *New base solid*, *Add*, or *Cut.*
4. Specify the parameters of the extrude. You can do this in one of two ways.
   - Enter the specific vector of the extrude,
   - Enter the point to extrude from and then enter the point where the extrude ends.
5. If you want to draft the walls of the surface, enter a *Draft angle*.
6. Click the *Preview* button to see a line drawing of the feature.
7. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.
8. Click *OK* or *Finish*.

Using open curves for solid extrudes

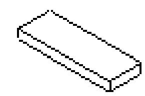


Open curves can be used for solid base extrudes or solid cut extrudes. They cannot be used for solid add extrudes. For base extrude solids, a sheet is created without any end caps.

For cut extrude solids, you must ensure that the cut will divide the solid into two distinct pieces.



When working with open curves, an additional checkbox called, *Flip side to cut*, is displayed. After previewing your result, you can check this checkbox if you want to keep the solid on the other side of the extrude.



Revolved solid design feature

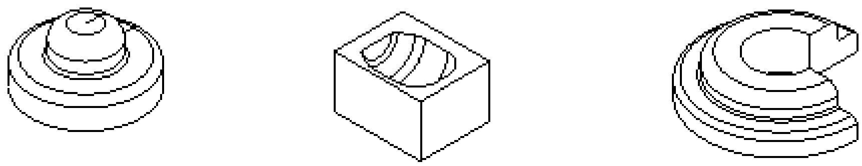
Solid of revolution creates a solid design feature by revolving a curve around a line.

To create a solid of revolution design feature:

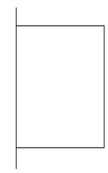
1. Select *Solid of revolution* from the solid wizard or the solid toolbar.
2. Select the curve name in the *Curve* drop down list or click the pick curve  button and select the curve in the graphics window. Note that the curve must be planar.
3. Select the type as *New base solid*, *Add*, or *Cut.*
4. Set the Start angle in degrees.
5. Set the End angle in degrees.
6. Pick the Construction method. This is where you set what is revolved around, or the pole. You can pick a custom axis, or the X or Y axis.
7. If you select *Custom line*, select the name of the line in the *Line* drop down menu or click the pick line  button and select the line in the graphics window.
8. Click the *Preview* button to see a line drawing of the feature.
9. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.
10. Click *OK* or *Finish*.

Examples of solids of revolution

The left-hand figure shows a base solid created with a curve on the axis revolved 360 degrees. You can tell that the curve was on the axis since there is no hole in the middle of the solid. The middle figure shows an extrude cut design feature rotated 180 degrees. The figure on the right shows a base solid created with a curve off of the axis (note the hole in the middle) and revolved 270 degrees.

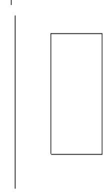


Using open curves for solids of revolution

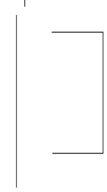


If the curve is open, a dialog is displayed with the following options:

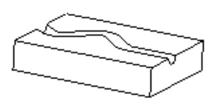
- Close curve using lines to axis– creates a straight line from the curve endpoints to the axis.



- Close curve by joining endpoints – creates a straight line between the endpoints of the curve.
- Continue with curve as-is – no change is made to the curve and a sheet is created.



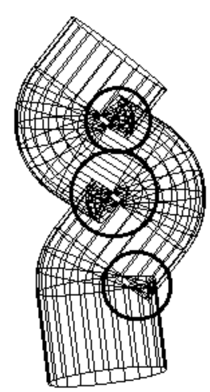
Sweep solid design feature



A swept solid takes a curve and runs it along another curve. This example shows a circle swept along a curved path to create a groove shape.

To create a swept solid design feature:

1. Select *Swept* from the solid wizard or the solid toolbar.
2. Select the curve name in the *Axis* drop down list or click the pick curve  button and select the curve in the graphics window. Note that the curve must be planar.
3. Select the curve name in the *Cross section* drop down list or click the pick curve  button and select the curve in the graphics window.
4. Select the type as *New base solid*, *Add*, or *Cut.*
5. Click the *Preview* button to check out your shape.



6. If the sweep does not follow the Axis curve, click *Sweep from the other end* and click *Preview* again. If you still receive an error see troubleshooting swept solids.

7. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.

8. Click *OK* or *Finish*.

Troubleshooting swept solids

If you use a cross section that is too large, it can result in surfaces that self-intersect. If sweeping from both ends of the curve does not work, cancel the swept solid and try creating a swept surface. If the result shows overlapping surface regions like shown on the right, then the cross section is too large.

Lofted solid design feature

This design feature takes a series of curves and fits a surface to them.

To create a solid lofted design feature:

1. Select *Lofted* from the solid wizard or the solid toolbar.

2. Select the curve name in the *Curve* drop down list and click the add  button, or click the pick curve  button and select the curves in the graphics window.

3. Select the type as *New base solid*, *Add*, or *Cut.*

4. If you want to connect the first and last curves, click the *Close* button.

5. Click the *Preview* button to confirm that the surface is correct.

6. If you are working with open curves and the surface twists, click on the name of the curve in the curve list and click the *Reverse selected curve*  button and click *Preview* again.

7. If you are working with closed curves and the surface twists, you may need to change the start point of some of the curves.

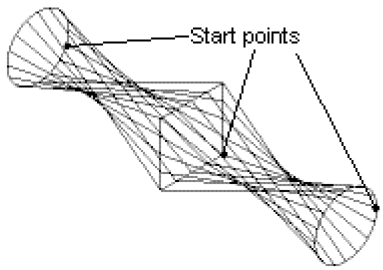
8. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.

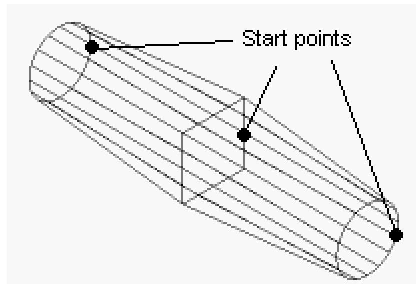
9. Click *OK* or *Finish*.

If you cannot get the desired shape, you may want to create the shape with a lofted surface.

Twists in surfaces or solids with closed cross sections

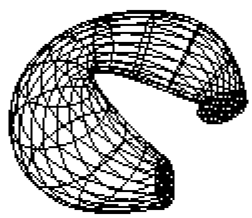
When creating ruled surfaces of lofted surfaces or solids with closed cross sections, you want to make sure that the start points of the curves line up with the way you would like to create the lines of your shape. The figure on the left shows a ruled surface created from cross sections with misaligned start points. After using Curve start/reverse to change the start point of the square center curve, the twist is removed, as shown on the right.

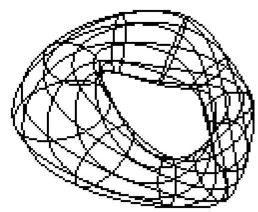




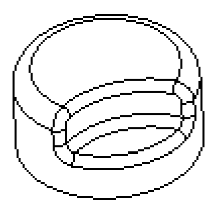
Closed lofted solids

The Closed checkbox for lofted solid design features, connects the first and last curves. Note that you cannot repeat curves in a loft. In this example, the curves on the left are lofted. In the middle picture, the loft is not closed. The loft on the right shows a closed loft.

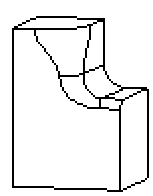




Solid Fillets



Since solids share edges among faces, solid filleting is simpler and more powerful than surface filleting. You simply select edges of the solid and enter fillet radii. The surfaces are then blended across the edge and the faces are automatically trimmed. You can create constant radius fillets as shown on the left or variable radius fillets as shown on the right.



Creating a solid fillet

1. Select the solid in the part view.
2. Click the Solid wizard button and select *Fillet Edge* from the *Solid Modifier* category or Select the *Fillet Edge* button from the Solid toolbar.
3. The edges of the solid are highlighted in blue.
4. Enter the *Radius*.

5. If you want to set the radius for a single edge, click the *Pick edge*  button and pick the edge in the graphics window.

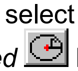
6. If you want to fillet all the edges of a face , click the *Pick face*  button

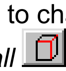
7. The name of the edge and its fillet radius is entered into the list of edges and the edge is shown as a red arrow.

8. If you want to use a different radius on the next edge, enter a different *Radius*.

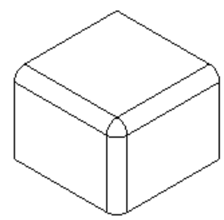
9. Continue picking edges until you have selected all the desired edges.

10. Click *Preview* if you want to preview the fillet before committing to the changes.

11. If you need go change the radius on an edge, select the edge in the edge list, enter a new radius and click the *Set radius on selected*  button.

12. If you want to change the radius of all the edges, enter the new radius and click the *Set radius on all*  button.

13. Click *Finish* or *OK*.

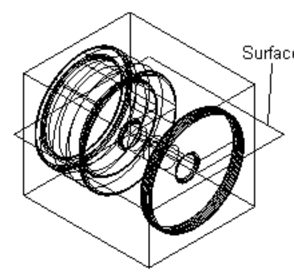


Controlling the shape of solid fillet joints

The joints where fillets intersect can either be mitered or blended (as shown in the figure).

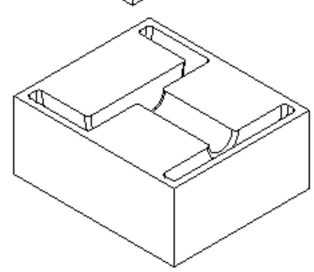
To create a blended fillet, fillet the adjacent edges in a single fillet. The blend will automatically be calculated based on the radii or the fillets.

To create a mitered fillet, fillet the adjacent edges in different fillets. The fillets are intersected and trimmed against each other.

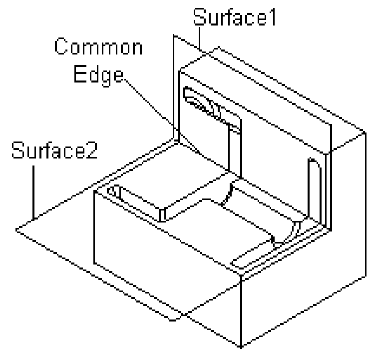


Cut solid with parting surface

Once you have a solid, you can cut it with a surface or a collection of surfaces with common edges. This is a useful operation for separating the solid into two mold halves. In the example below, the solid is a revolved glass that was subtracted from a box.



This solid is then cut with a flat surface resulting in half of the solid being cut away.

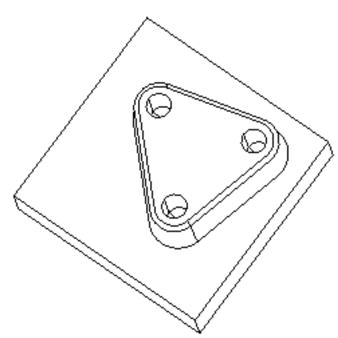


You can also cut a solid with a collection of surfaces that have common edges. The surfaces are stitched together and the solid is then cut with the resulting surfaces.

How to cut a solid with parting surfaces

1. Click the Solid wizard button and select *Cut with parting surface* from the *Solid Modifier* category or Select the *Cut with parting surface* button from the Solid toolbar.
2. Select the name of the solid in the *Cut this solid* drop-down list or click the *Select solid* button and pick the solid in the graphics window.
3. Select the name of the surface in the *Surface* drop-down list and click the button or click the *Select surface* button and pick the surface in the graphics window.
4. If you have additional surfaces to cut with, repeat step 3. Remember that the surfaces must have common edge curves that allow them to be stitched together.
5. If you wish to create a solid for both sides, check the *Keep both sides* checkbox.
6. Click the *Preview* to preview the cut. The result will be shown in thick blue lines. If you want to cut on the other side of the surface(s), click on the surface name and click the *Reverse* arrow.
7. Click *Finish* or *OK*.

Solid from 2 ½ D feature



When a 2.5D feature is created, a 3D surface representation is created for display. While you can generate toolpaths and create a 3D shaded simulation of the part, you must convert the features into solid modeling operations to subtract them from a solid. In the example below a boss feature and three holes were converted to a solid.   It is important to remember that all solids created from 2 ½ D features are **subtractive**, even boss features.

Creating a solid modeling operation from a 2 ½ D feature

1. Click the Solid wizard button and select *Solid from 2 ½ D feature* from the *From surface/primitive* category or Select the *Solid from 2 1/2D feature* button from the Solid toolbar.

2. Select the feature name in the feature drop-down list or pick the *Pick feature* button and pick the feature in the graphics window. You can convert all 2 ½ D features except side features. Patterns are also supported.

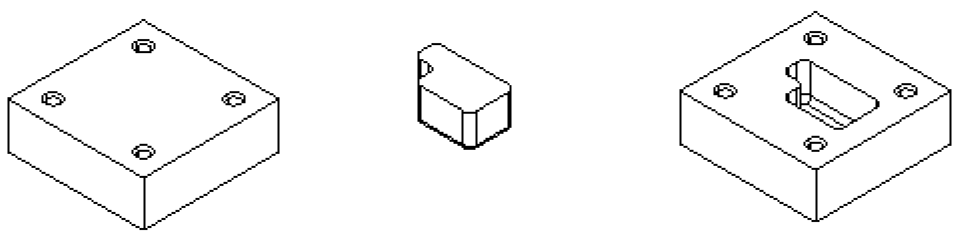
3. If you have more than one base solid created, select the name of the solid.

4. Click *OK* or *Finish*.

Remember that all features are subtracted from the solid, even bosses.

Combine solids

Normally you work only on a single solid model. You start with a base and create additional design features that alter the shape of the solid. Sometimes it is convenient to work on two different solids separately and then combine them into a solid. An example would be combining an imported solid and a mold base. The figure on the right is a solid mold base. The middle figure is the cavity solid. The left-hand figure shows the result of combining the mold base and the cavity.



How to combine solids

1. Click the Solid wizard button and select *Combine solids* from the *Solid Modifier* category or Select the *Combine solids* button from the Solid toolbar.

2. Select the name of a solid from the *Solid1* drop-down list or click the *Pick solid* button and select the solid in the graphics window.

3. Select the name of a solid from the *Solid2* drop-down list or click the *Pick solid* button and select the solid in the graphics window.

4. Select the *Operation*.

5. Click *Preview* to preview the results. Remember that for the Difference operation, the order of the solids matters. Solid2 is subtracted from Solid1.

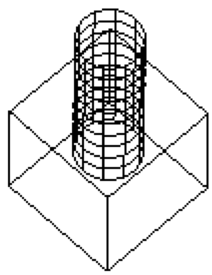
6. Click *Finish* or *Ok*.

Solid operations

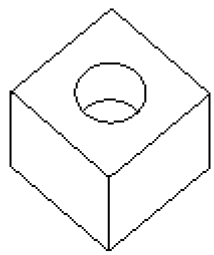
A Difference B is A minus the portion of B that is inside of A.

A Union B is A plus the portion of B that is outside of A.

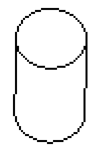
A Intersection B is the volume that is common to both A and B.



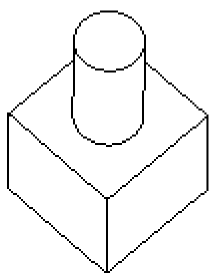
Initial solids



Square **difference** Cube



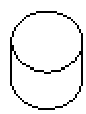
Cube **difference** Square



Cube **union** Cylinder

Or

Cylinder **union** Cube



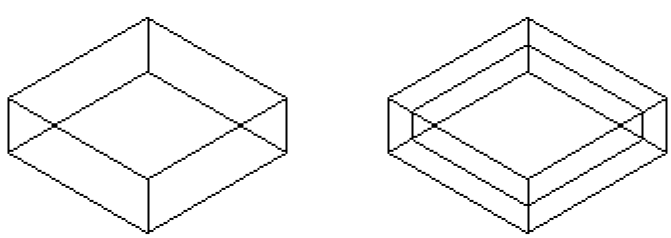
Cube **intersection** cylinder

Or

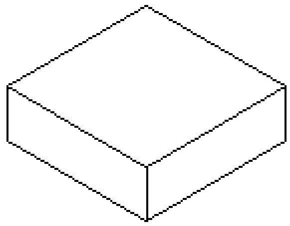
Cylinder **intersection** Cube

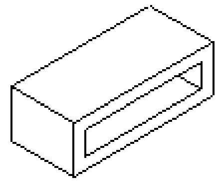
Shell solid design feature

Shell creates a thin-walled solid from another solid. In the example below, all the walls of the box are offset to create a rectangular void in the center of the solid.

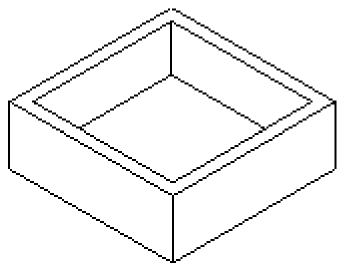


If you are using a hidden line view or shaded view, results of the shell design feature will not be visible, but if you cut the solid, you can see the void.





You can optionally select faces of the solid not to offset. The faces that are offset create openings into the void.



Creating a shell solid design feature

1. Click the Solid wizard  button and select *Shell* from the *Solid Modifier* category or Select the *Shell*  button from the Solid toolbar.

2. Select the name of a solid from the *Offset this solid* drop-down list or click the *Pick solid*  button and select the solid in the graphics window.

3. Enter the *Offset distance*. A negative distance will offset the faces into the solid. A positive distance will offset the surfaces out of the solid. Note that for solids with tight regions or fillets, you cannot offset the faces more that smallest fillet radius or the smallest gap between faces.

4. If you want to exclude any faces, select the name of the face in the *Surface* drop-down list and click the  button or click the *Select surface*  button and pick the faces in the graphics window.

5. Click *Preview* to preview the results.

6. Click *OK* or *Finish*.

Troubleshooting shell solid design features

If the *Offset distance* is too large, the offset surfaces will intersect with itself and cause an error. Your only choice in this situation is to use a smaller *Offset distance*.

Solid from stock or cube

This feature creates a base feature from the stock dimensions or allows you to create a cube design feature.

Creating a solid from the stock dimensions

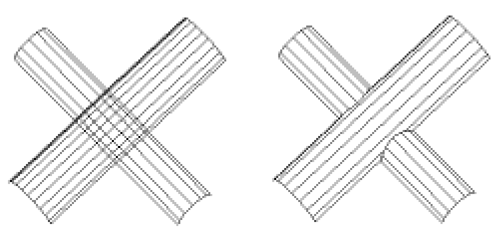
1. Click the Solid wizard  button and select *From stock (cube)* from the *From surfaces/primitives* category or Select the *Stock*  button from the Solid toolbar.
2. Click the *Stock* radio button.
3. Click *OK* or *Finish*.

Creating a cube solid design feature

1. Click the Solid wizard  button and select *From stock (cube)* from the *From surfaces/primitives* category or select the *Stock*  button from the Solid toolbar.
2. Click the *Cube* radio button.
3. Enter the coordinates of each corner or click the *Pick point* button and pick the point in the graphics window.
4. Select the type as *New base solid*, *Add*, or *Cut.*
5. Click *OK* or *Finish*.

Stitching

Stitching converts well-trimmed surface models with shared edges and shared boundaries into solid models. Once the surfaces have been stitched together into a solid, you can perform any solid modeling operation on the solid. **The surfaces must not overlap or have gaps between them.** The surfaces on the left will not stitch because they overlap. The right-hand figure will stitch because the overlap has been trimmed away.



Types of models that are good candidates for stitching are:

- IGES files containing trimmed surfaces from solid modeling packages
- Surfaces that you model and trim using FeatureCAM surfacing tools

Stitching surfaces into solids

1. Click the Solid wizard button and select *Stitching* from the *From surfaces/primitives* category or select the *Stitch* button from the Solid toolbar.

2. Click the Pick surface button and pick the surfaces in the graphics window. Each surface you pick will be entered in the surface list. Note that if you have the surfaces already selected before bringing up the dialog box, these surfaces will be listed in the surface list.

3. If the surfaces form a sheet instead of a solid, click the *Non-solid results OK* checkbox. If you do not check this box, you are given an error if the surfaces do not form a complete solid. Click *OK* or *Preview*.

4. If the surfaces form a solid, a new solid will be listed in the part view.

5. If the surfaces to not form a solid or a sheet, an error message is displayed.

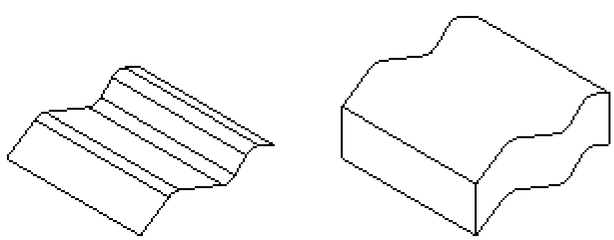
Troubleshooting stitching

1. Make sure that no duplicate surfaces are displayed.

2. Make sure that the surfaces do not overlap. In this case, you must trim the surfaces before stitching.

Extrude surface solid design feature

Extrude creates a solid design feature by pushing a *surface* in a straight line. This operation is identical to Extrude_solid_design_feature except it works with surfaces not curves as the input. Below are examples of a surface and its extrusion.



To create a solid extrude design feature:

1. Select *Extrude a surface* from the solid wizard or the solid toolbar.

2. Select the surface name in the *Surface* drop down list or click the pick surface button and select the surface in the graphics window. Note that you cannot use a face of a solid.

3. Select the type as *New base solid*, *Add*, or *Cut.* See Type of design feature for further information.

4. Specify the parameters of the extrude. You can do this in one of two ways.

5. Enter the specific vector of the extrude,

6. Enter the point to extrude from and then enter the point where the extrude ends.

7. If you want to draft the walls of the surface, enter a *Draft angle*.

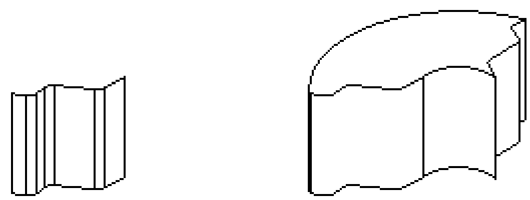
8. Click the *Preview* button to see a line drawing of the feature.

9. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.

10. Click *OK* or *Finish*.

Revolved solid from surface design feature

Solid of revolution creates a solid design feature by revolving a *surface* around a line.



1. Select *Revolve a surface* from the solid wizard or the solid toolbar.

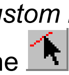
2. Select the surface name in the *Surface* drop down list or click the pick surface  button and select the curve in the graphics window. . Note that you cannot use a face of a solid.

3. Select the type as *New base solid*, *Add*, or *Cut.* See Type of design feature for further information.

4. Set the Start angle in degrees.

5. Set the End angle in degrees.

6. Pick the Construction method. This is where you set what is revolved around, or the pole. You can pick a custom axis, or the X or Y-axis.

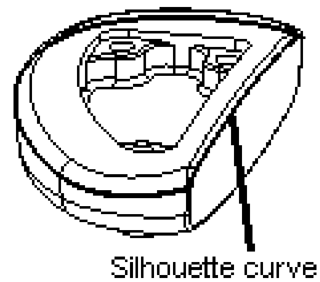
7. If you select *Custom line*, select the name of the line in the *Line* drop down menu or click the pick line  button and select the line in the graphics window.

8. Click the *Preview* button to see a line drawing of the feature.

9. If you displayed this dialog box from the solid toolbar, you can click *Apply* to preview the feature as a solid. If the cut is on the wrong side of the curve, click *Flip side to cut* and click *Apply* again to verify.

10. Click *OK* or *Finish*.

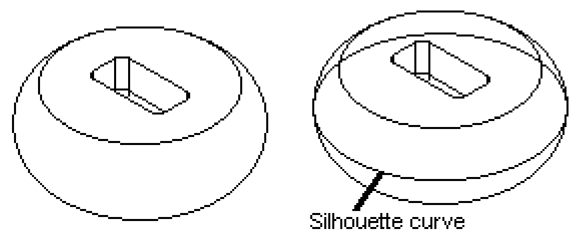
Silhouette curves



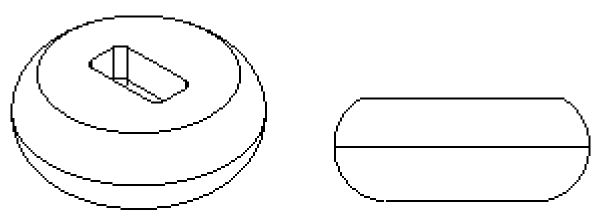
Silhouette curves represent the widest part of a solid and serve as useful parting curves for molds. These curves can also be used to split the faces of a solid.

One of the tasks in creating a mold for a solid model is determining the parting lines and splitting the part into at least two parts. Silhouette curves represent the widest extent of a part when viewed from the +Z direction. These curves often are helpful for determining parting lines.

The left-hand figure shows a solid model. Initially, the side of the part is a single surface. As a first step in creating two mold halves, we would like to split the side face at its widest part. The right-hand figure shows the silhouette curve for the solid.

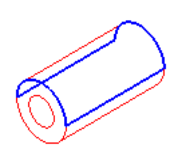


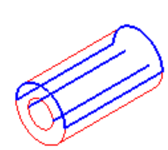
In the silhouette curves dialog box, you can also use the calculated silhouette curves to split the faces of the solid. The figures below show that the side face is now split into two separate pieces along the silhouette curve. Use Select core/cavity to split solids into two halves.



. This curve can then be used to create parting surfaces.

Creating silhouette curves or splitting faces of a solid at silhouette curves



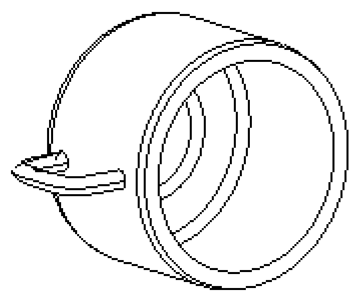


1. Click the Solid wizard button and select *Silhouette curves* from the *Manufacturing* category or select the *Silhouette curves* button from the Solid toolbar.
2. Select the name of a solid from the *Solid* drop-down list or click the *Pick solid* button and select the solid in the graphics window.
3. The silhouette curves are determined from the +Z direction of the UCS. Select the appropriate UCS from the *UCS* drop-down list.
4. If you want to extract only the silhouettes visible from the top silhouettes, uncheck the *Include hidden silhouettes* checkbox.

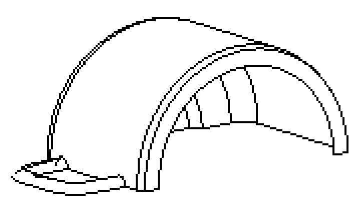
5. If you want to extract all silhouettes, check the *Include hidden silhouettes* checkbox.
6. If you want to join the resulting curves that touch into a single piece, click*Join resulting curves*.
7. Silhouette curves can often be made up of many small pieces. To reduce the number of these pieces, check *Smooth/reduce*. The number after the *Smooth/reduce* label is the tolerance for the data smoothing. The smaller the number, the tighter the curve will approximate the original silhouette.
   Note: This option can smooth out sharp corners in your curve, so use it with caution.
8. If you want to split the faces of the solid at silhouette curves, check the *Split faces at silhouettes* checkbox.
9. The *Tolerance* affects how tightly the silhouette curve approximates the actual silhouette. Reduce this number if the silhouette misses regions of your part.
10. Click *OK* or *Finish.*

Select core/cavity

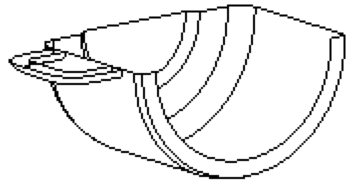
Select core/cavity segregates the surfaces of a solid into one of three different groups.



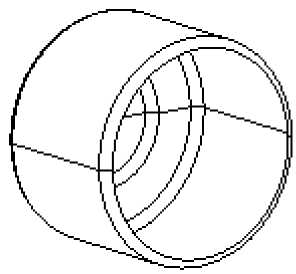
Top
All surfaces (or portions of surfaces if the Automatic split option is enabled) that are visible from the top.



Bottom
All surfaces (or portions of surfaces if the Automatic split option is enabled) that are visible from the top.

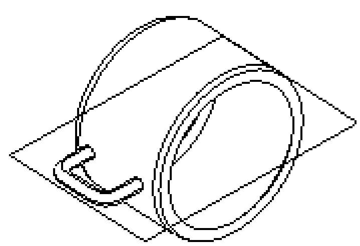


Other
All surfaces that do not fall into either the top of bottom category. These surfaces are usually referred to as the core.



It is recommended that the core/cavity separation be performed using the *Automatic split* option. This option will split the faces of the solid at either the silhouette of the solid, along a parting surface that you provide as the *Parting surface*. With the *Parting surface* option set to ***Automatic*, the silhouette curve is automatically calculated and used to split the faces prior to the classification into either the *top, bottom* or *other* category.

You can also specify a parting surface for the Automatic splitting. This surface must pass all the way through the solid as shown in this figure.



Selecting core or cavity faces from a solid

1. Click the Solid wizard button and select *Select core/cavity* from the *Manufacturing* category or select the *Select core/cavity* button from the Solid toolbar.

2. Select the name of a solid from the *Solid* drop-down list or click the *Pick solid*

button and select the solid in the graphics window.

3. The silhouette curves are determined from the +Z direction of the UCS. Select the appropriate UCS from the *UCS* drop-down list.

4. If you want the faces that are on the top, click the *Top* radio button.

5. If you want the faces that are on the bottom, click the *Bottom* radio button.

6. If you want the core surfaces, the surfaces that are neither on the top nor on the bottom, click the *Other* button

7. If you want to include the vertical surfaces that are next to the core/cavity surfaces, click the *Include vertical surfaces* radio button.

8. If there are surfaces you would like to explicitly exclude, click the *Pick Surface*  button and select the surfaces or select the name of the surface in the drop-down list and click the  button.

9. If you would like to split the surfaces at the silhouette curve, check *Auto split* and leave the *Parting surface* field set to ***Automatic*.

10. If you would like to split the surfaces at a parting line check Auto split and set the *Parting surface* field to the name of the parting surface.

11. Click *OK* or *Finish*.

12. The core/cavity surfaces are now selected so that you can easily create 3D surface milling features.

Creating a solid from core or cavity faces of a solid

1. Click the Solid wizard  button and select *Select core/cavity* from the *Manufacturing* category or select the *Select core/cavity*  button from the Solid toolbar.

2. Select the name of a solid from the *Solid* drop-down list or click the *Pick solid*  button and select the solid in the graphics window.

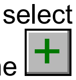
3. The silhouette curves are determined from the +Z direction of the UCS. Select the appropriate UCS from the *UCS* drop-down list.

4. If you want the faces that are on the top, click the *Top* radio button

5. If you want the faces that are on the bottom, click the *Bottom* radio button.

6. If you want the core surfaces, the surfaces that are neither on the top nor on the bottom, click the *Other* button.

7. If you want to include the vertical surfaces that are next to the core/cavity surfaces, click the *Include vertical surfaces* radio button.

8. If there are surfaces you would like to explicitly exclude, click the *Pick Surface*  button and select the surfaces or select the name of the surface in the drop-down list and click the  button.

9. If you would like to split the surfaces at the silhouette curve, check *Auto split* and leave the *Parting surface* field set to ***Automatic*.

10. If you would like to split the surfaces at a parting line check Auto split and set the *Parting surface* field to the name of the parting surface.

11. Click the *Make solid from result* checkbox.

12. Click *OK* or *Finish*.

13. The surfaces are selected and then stitched together into a solid.

Transforming a solid

1. Select the solid

2. Select the transform {bmct btn-xform.bmp} button from the Standard toolbar.

3. The *Transform* dialog box comes up. See page 38 for information on this dialog box.

4. If you selected to move the solid, a solid operation is created in the tree view.

5. If you select to copy the solid, a new solid is displayed in the tree view.

Split face

The split face operator takes a curve or list of curves, projects them onto the selected face and splits the face into multiple faces using the projected curves as the boundaries.

To create a solid lofted design feature:

1. Select *Split face* from the solid wizard or the solid toolbar.

2. Select a face name in the *Face* drop down list or click the Pick surface button and select the face in the graphics window.

3. Select the curve name in the *Curve* drop down list and click the add button, or click the pick curve button and select the curves in the graphics window.

4. Repeat step 3 if you want to use more than one curve to split the face.

5. Click the *Preview* button to confirm that the surface is correct.

6. Click *OK* or *Finish*.

Explode solid

This constructor copies the faces of a solid into surfaces.

To explode a solid:

1. Select *Explode* from the solid wizard or the solid toolbar.

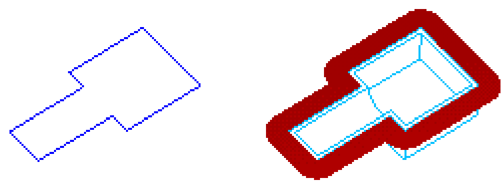
2. Select the name of a solid from the *Solid* drop-down list or click the *Pick solid* {bmc btn-pick-solid.bmp} button and select the solid in the graphics window.

3. If you only want to copy the selected faces, click the *Selected faces only* checkbox.

4. Click *OK* or *Finish*.

Parting surface

This function, takes a curve and creates a parting surface.



The *Parting* surface constructor does not part the model. Use Cut solid with parting surface or Select core/cavity for that functionality. The curve can be a 2D or 3D curve. The curve can be obtained any many ways including Silhouette Curves, Surface edges, Curve projected onto a surface The Z-axis of the UCS indicates the parting direction of the mold.   For 3D curves the curve is divided at any corners.

To create a parting surface:

1. Select *Parting surface* from the solid wizard or the solid toolbar.

2. Specify the name of the surface, if desired.

3. Specify the *Land width*. This is the width of your parting surface.

4. Specify the UCS you would like to use. The Z axis of this UCS will be used as the parting direction.

5. Select the *Parting curve* by name and click the “+” button or use the Pick curve  button to select the curve in the graphics window.

6. If your curve is 3D, it will be broken subdivided at sharp corners into a number of curves. Each of these names will be listed under your curve name.

7. Click the *Preview* button to see your surface.

8. If a section of your curve seems to be oriented incorrectly follow this procedure:

    a. Click on the names in the curve list until the desired segment is highlighted.

    b. The direction that this segment is extruded will be displayed in the *Extrude direction* field.

    c. To change this direction, enter a new vector in the form “(X,Y,Z)”. You must enter the parenthesis and the commas. For example (1,0,0) is the X direction.

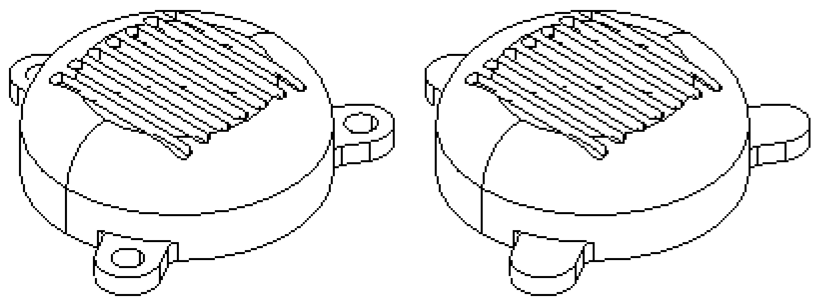
    d. Click the *Set* button.

    e. Click the *Preview* button to see the results.

Deleting faces

If you have a solid that you designed in FeatureCAM, you can easily delete a design feature. If you delete an extrude feature that cut a slot in a part, the slot is removed and the material is filled back in that region. If you have a solid model that you imported or stitched it is difficult to remove a design feature since you do not have the design features that were used to create the part, so you must use delete faces to remove the feature and heal the model back together.

The example shown below was a imported surface model that was stitched into a solid. The left-hand figure is the original solid and the right-hand figure shows the same model with the faces that represent the three holes deleted.



Most of the regions that you will want to remove are the equivalent of 2.5D features in your solid. This includes extruded holes, pockets or bosses. It is important that you select all surfaces that represent the feature in the solid. For example for a blind hole, the walls and bottom must be removed. If you only select the walls to be removed, the bottom would be left floating in space and cannot be healed back into the model.

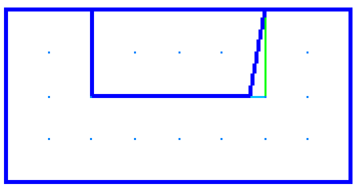
To delete faces from you solid

1. Select *Delete face* from the solid wizard or the solid toolbar.
2. If you want to create a new solid with the faces removed, click *Create new solid* and enter the name of the new solid.
3. If you want to modify an existing solid, click *Modify existing solid.*
4. Select the faces by either:
    a. Selecting the surface name and clicking the “+” button
    b. Picking the surfaces in the graphics window and clicking the “+” button.
    c. Clicking the pick surface button and picking the surfaces in the graphics window
5. If you want the gaps left by the deleted surfaces to be filled in, check *Heal remaining faces*.
6. Click the *Preview* button.
7. If you get an error message, it is probably because you left out some of the surfaces that need to be removed. Add the other surfaces and try *Preview* again.
8. Click *Finish*.

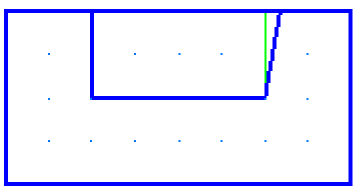
Draft a face

Many of the solid design features allow you to include a draft angle, but they require that all faces be drafted the same amount. *Draft a face* allows you to set a draft angle on one or more faces individually.

To draft a face properly, the *fixed reference surface* determines how the face is rotated. When a face is drafted, the surface must be rotated around a particular axis. This axis is determined by the intersection of the face and the fixed reference surface. By default a face is rotated around its intersection with the XY plane of the current UCS. This means that for a face that intersects the XY plane, a positive angle keeps the top of the face fixed and rotates the bottom of the face in, as shown below. The same result is calculated if a surface connected to the top edge of the face is selected as the fixed reference face.



If the surface connected to the bottom edge of the face is selected as the fixed reference, the bottom of the face stays fixed and the top edge of the face is moved out.



To draft a face:

1. Select *Draft a face* from the solid wizard or the solid toolbar.
2. Enter the draft angle in degrees.
3. Select the faces to draft by:
    a. Selecting the surface name and clicking the "+" button
    b. Picking the surfaces in the graphics window and clicking the "+" button.
    c. Clicking the pick surface button and picking the surfaces in the graphics window.
4. To keep the top of the face fixed, select the Fixed reference surface by clicking the pick surface button and picking a surface connected to the top of the face.
5. To keep the bottom of the face fixed, select the Fixed reference surface by clicking the pick surface button and picking a surface connected to the bottom of the face.
6. Click the *Preview* button to review the results.
7. Click *Finish* or *OK*.

Chapter 22
3D Surface Manufacturing

Overview of surface manufacturing

You must license FeatureMILL3D to use the functions described in this chapter.

To mill 3D surfaces you must create a Surface milling feature from the surfaces. Surfaces can be used as part surfaces, the surfaces that you want to machine, and check surfaces, surfaces that you would like to avoid with the tool. You specify these surfaces on the Surface milling feature dimensions tab.

After specifying the surfaces to machine, select the manufacturing operations to use to generate toolpaths on the Strategy tab. Use the *Tools* tab to specify the desired tooling.

Finally, the attributes on the milling tab are used to fine tune the manufacturing process. The moves between toolpaths are controlled by the leads/step tab. To limit the toolpath to regions of the part based on slope see the Slope limits tab. For controlling the side of surfaces that are machined with isoline milling or Z level finishing see the Surface control tab. For information on 3D toolpath techniques and these tabs see page **Error! Bookmark not defined.**.

How to create a surface milling feature

1. Use the New feature  button and select *Surface milling* and click *Next*. (If you graphically select the your surfaces first, they will automatically be classified as part surfaces.)
2. The New feature new strategy page comes up. This page allows you to select the cutting strategy you want to use for your feature. You can only specify one strategy in the New feature wizard. Don't worry if it is a roughing of finishing operation. This information is specified further into the wizard. The strategy choices are:
    - Predrill (available only from the feature properties dialog box) – drill the part to create a hole for plunging.
    - Parallel toolpaths – toolpaths that are parallel to the X or Y axes
    - Z level – toolpaths that are parallel to the Z axis. Applies to Z level rough and Z level finish
    - Isoline milling– toolpaths that follow the rows or columns of individual surfaces
    - Spiral toolpaths – toolpaths that move out radially from the center of the feature, or in radially toward the center of the feature.
    - Pencil milling– a single clean-up pass for corners
    - Horizontal + Vertical– machine steep and shallow regions using different techniques.

- Parallel + 90(remachining– a combination operation which includes a parallel toolpath follosed by another parallel operation perpendicular to the first

3. Click on the desired strategy.

4. Click the *Next* button and follow the steps of the wizard.

3D milling methods

You have a number of options for milling a 3D feature. The object is to select a method that is efficient for your feature's shape and that also gives an acceptable finish. All methods are available for roughing. The available methods are:

- Projection milling methods
- Z level rough and Z level finish
- Isoline milling
- Pencil milling and remachining

Z level rough

Z level roughing slices the model at various depths to obtain planar contour curves. It then uses these curves to rough the model using a pocketing or boss toolpath generation technique. For a generalized boss, FeatureCAM uses the stock boundary or stock curve as the outer boundary of the boss.

The part surfaces are analyzed to determine if they represent a boss or pocket, but you can explicitly classify the surfaces using the Boss attribute on the 3D Milling tab

Advantages:

1. Constant depth of cut for minimized tool wear.
2. Constant width of cut with a reduced chance of getting a slotting cut.
3. It handles nearly-vertical walls well

Disadvantages:

1. Requires a well-trimmed model to work appropriately.
2. Does not handle gaps in the model as well as the projection techniques.
3. Does not detect undercut regions.

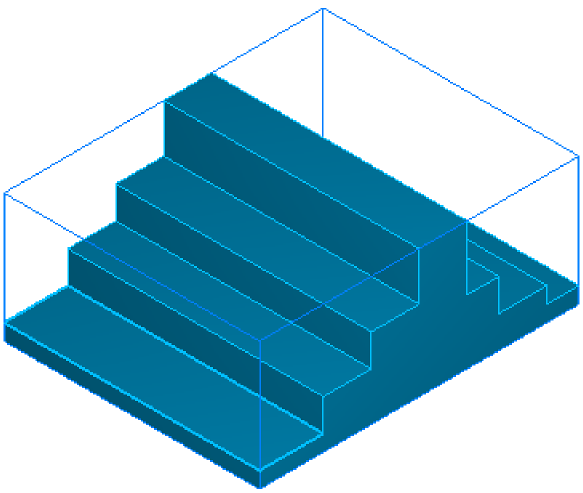
Methods of Z-level roughing

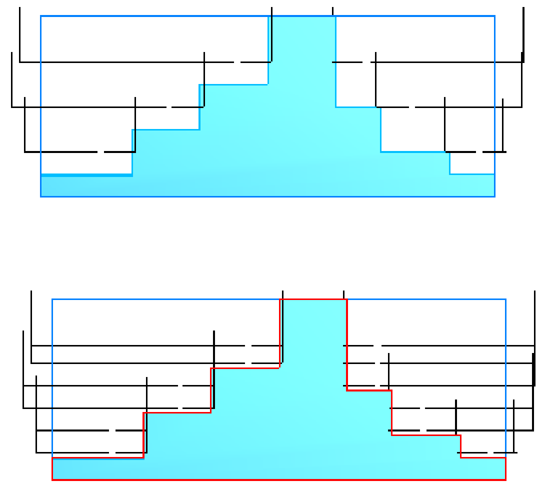
There are three different methods of performing Z-level roughing in FeatureCAM. These methods are controlled by the *Visible surfaces* attribute and the *Handles undercuts* attribute.

Flat srf support

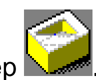
Without *Flat srf support* checked, the Z levels are calculated at a constant Z increment. With *Flat srf support* checked, extra levels are inserted above each flat surface in the model. This ensures that the finish allowance will be accurately left on all flat surfaces.

The images below illustrate the differences in the slicing. The top picture shows the slices without *Flat srf support*. The bottom picture shows the additional slices that are added with that option invoked.





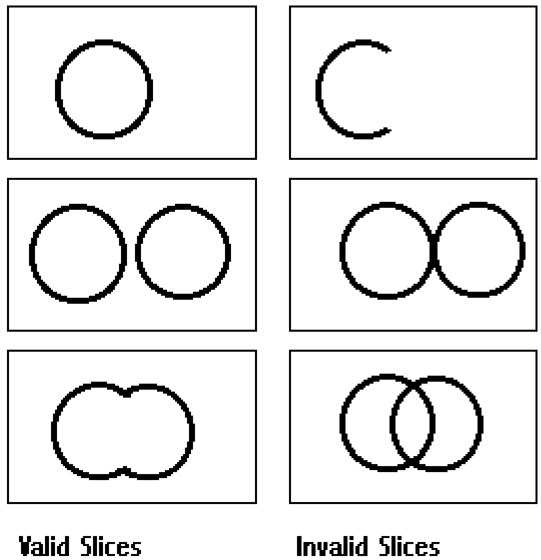
How to create a Z level roughing operation

1. Create a surface milling feature using the Feature Step .
2. Specify *Z level* as the *Strategy* on the New Strategy page and *Rough* as the operation on the Strategy page.
3. If your shape is a 3D boss, check the *Boss* checkbox until a dark check appears. If your shape is a cavity, click on the *Boss* checkbox until no checkmark appears. If the *Boss* checkbox contains a gray checkmark, then this parameter is set automatically by FeatureCAM.
4. At the conclusion of the wizard, click the *Finish and edit properties* button. If this button is not displayed, hold down the triangle on the right side of the *Finish* button and select *Finish and edit properties* from the menu.
5. Click on the Milling tab.
6. Select the Z-level roughing method.
7. Click the *Preview slices* button to preview the slices of your model. This figure shows how the slices should look. See Restrictions of Z level roughing for more information.
8. Click on the Leads tab and set the options for the moves between toolpaths.
9. Click OK.
10. Generate toolpaths. If you receive an error, see Troubleshooting Z level roughing for more information.

Note: The toolpaths for Z level rough always use both linear and circular interpolation where appropriate. While Arc/line approximation.is an option for other types of 3D toolpaths, it is always used for Z level rough.

Restrictions of Z level roughing

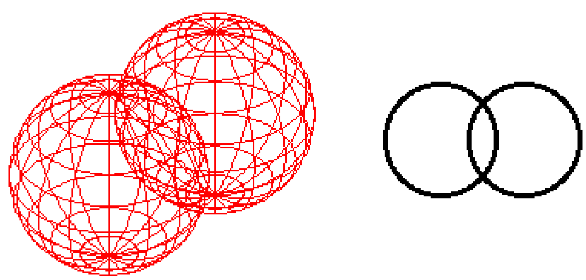
1. The slices of the model must formed closed contours that form single loops. Z level roughing requires a well-trimmed model. This figure illustrates valid and invalid slices.



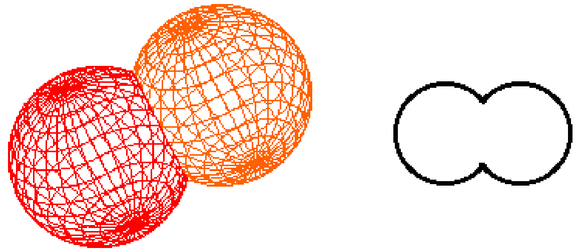
To view the slices, click the Preview button in the Milling tab. Remember that you can change your view to more closely study the slices.

2. As a result of restriction 1, Z level roughing requires:

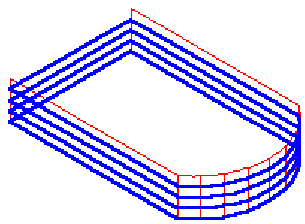
- A well trimmed model without overlapping surfaces. When this untrimmed model is sliced, it will result in these bad contours.



Once this model is trimmed as shown below the loops in the contours are eliminated and Z slice roughing should work on the model.



- Any gaps in slices to be closed. FeatureCAM attempts to close gaps in contours with straight lines. In this U-shaped surface the contours are closed with straight lines. If you want the toolpaths to behave differently in the gap region, you must explicitly model surfaces in the gap.



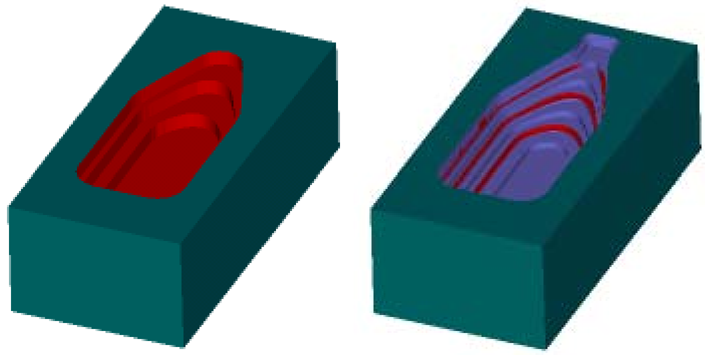
3. Gaps in between surfaces in the model are not handled as robustly as with projection techniques.
4. Only contours inside of the stock boundary are cut.
5. Undercut regions are not detected.

Restrictions of Handles Undercut method

1. This option will only *complete* visible surface slice loops with silhouettes.
2. There must be some VISIBLE surface slice to begin with. If the entire part is an undercut region, no slices will be generated.

Z-level roughing with multiple tools

Z-level roughing can be performed with multiple tools. Once a larger tool is used to rough out the majority of the part, FeatureCAM can automatically apply a semi-roughing pass to the uncut regions of the part without recutting the previously roughed region. The smaller tool will rough around the walls of the part with a smaller Z-increment than the previous tool and will rough regions in which the larger too will not fit. The figures below show a bottle that is first roughed with a 0.75 inch endmill and then roughed with a 0.25 inch endmill.



Roughing with multiple roughing tools is much more efficient than simply roughing with a small tool. In the case of the bottle, roughing with two tools takes 9 minutes and 34 seconds, while roughing the entire part with only the smaller tool takes 19 minutes and 8 seconds.

To perform Z-level roughing with multiple tools:

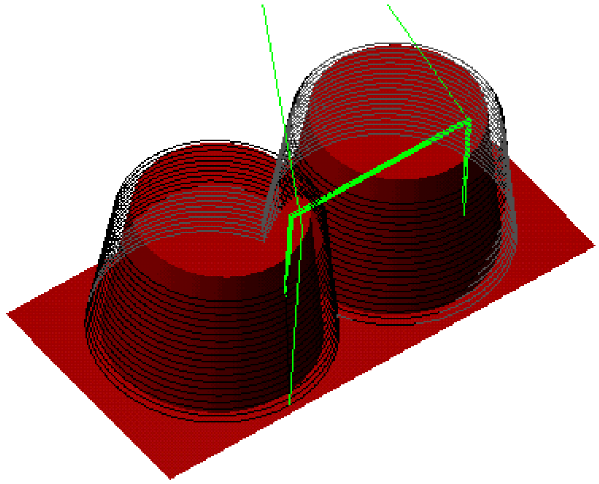
1. On the Strategy tab of the New Feature wizard or the Feature Properties dialog box, check the *Multiple rough* check box.
2. Enter a list of *Tool diameters*. Separate the diameters with a comma. Each diameter must be smaller than the preceding diameter.

Each of the roughing passes is listed in feature's tree view so you are able to further edit these operations to customize attributes such as step over values or z increments.

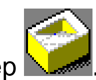
Each of the roughing passes is affected by the visible surfaces attribute.

Z level finishing

Z-level finishing is a good technique for finishing steep walls or when you require a consistent depth of cut. This technique is sometimes called "waterline milling." Just like Z level rough, it starts with slicing the model and then creating toolpaths from the slices. Z-level finishing is more forgiving than its roughing counter-part, Z-level roughing, in that it does not require the slices to form closed loops and it handles undercuts.

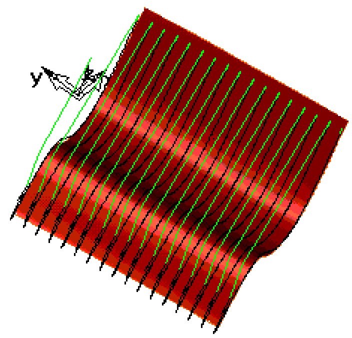


How to create a Z level finishing operation

1. Create a surface milling feature using the Feature Step .
2. Specify *Z level* as the *Strategy* on the New Strategy page and *Finish* as the operation on the Strategy page.
3. At the conclusion of the wizard, click the *Finish and edit properties* button. If this button is not displayed, hold down the triangle on the right side of the *Finish* button and select *Finish and edit properties* from the menu.
4. Click on the Milling tab. Click the Preview button to preview the slices of your model. This figure shows how the slices should look.
5. Click on the Leads/steps tab and set the options for the moves between toolpaths.
6. Optionally set the Start point(s) attribute to adjust the start point of the toolpaths for each loop.
7. To enable zone machining click the Reorder attribute on the Milling tab.
8. Click OK.
9. Generate toolpaths.

Isoline milling

Isoline milling uses the isoline curves of a surface to mill the surface. These curves can be in the row direction or column direction. This figure shows an example of isoline toolpaths on a single surface.



Advantages

- Uniform finish with scallop height control. The toolpaths are spaced based on the distance along the surface.
- Nearly-vertical walls handled well. Since this is not a projection technique, nearly vertical walls are cut correctly.

Disadvantages

- Toolpaths generated on a surface by surface basis.
- Non-uniform depth of cut if used as a roughing technique.

Restrictions of isoline milling

- Isoline milling works on a surface by surface basis. This can result in a lot of retractions
- The orientation of the surfaces matters. Toolpaths are generated for surfaces whose normals point up. Surfaces are “auto flipped” where possible, but for vertical and some other cases, the machining side must be specified. Select the surface in the list box and click Switch machining side. Isoline milling may mill on the wrong side of the surface or if certain flags are set, it may skip the surface.

Troubleshooting isoline milling

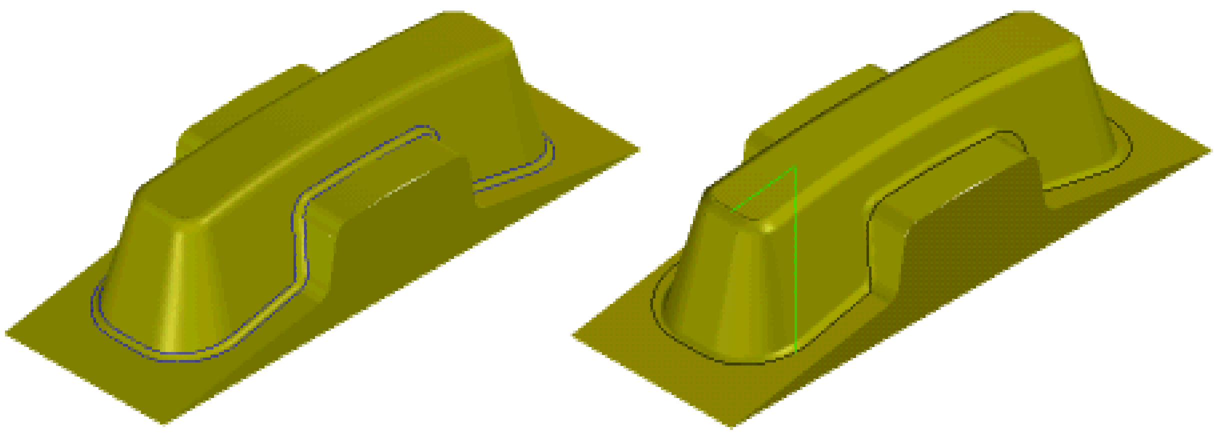
1. **The toolpaths are on the wrong side of the surface**. Select the operation in the feature’s tree view. Go to the Surface control tab. Change the machining side to Reverse using the  button.
2. **Toolpaths should go the other direction or start at the other end of the surface**. In the Isoline control tab the Start curve has the options of first row, last row, first col, last col. Select the surface in the table and toggle the Start curve using the  button.   An icon appears on the screen that indicates the start point and the direction of the first toolpath.
3. **Surfaces are cut in the wrong order**. Select the operation in the feature’s tree view. Go to the isoline control tab. Use the arrow keys to rearrange the order of the toolpaths.
4. **Toolpaths for a surface are begin reorder strangely**. Select the operation in the feature’s tree view. Go to the Milling tab. Turnoff the Reorder option.

Pencil milling

Pencil milling is used to clean up corners or fillets of a part. Pencil milling operations automatically detect corners less than or equal to the specified radius (including sharp corners) and then creates a single toolpath to clean out these corners. Applications of pencil milling include:

- Finishing fillets in a part with a single toolpath
- Cleaning up sharp concave corners
- Pre-relieving corners before high-speed finishing the part with a small tool
- Roughing fillets by using the finish allowance.

The first step is to determine the regions that contain the corners as shown on the left. Depending on the number of surfaces in the model, this step may take a little time. After the regions are detected, a single toolpath is created that will clean out the corners regardless of the number of surfaces contained in the region.



How to create a pencil mill operation

1. . Create a surface milling feature using the Feature Step .
2. Specify *Pencil* as the *Strategy* on the New Strategy page and *Finish* as the operation on the Strategy page.
3. At the conclusion of the wizard, click the *Finish and edit properties* button. If this button is not displayed, hold down the triangle on the right side of the *Finish* button and select *Finish and edit properties* from the menu. Create a surface milling feature.
4. Select a ballend tool. The radius of the tool will be used to locate the corners in the part.
5. Click on the Remachine tab to customize the operation.
6. Click on the Leads/steps tab and set the options for the moves between toolpaths.
7. Click OK.

Troubleshooting pencil milling

1. **Calculating the regions takes too long.** Use a stock boundary on the Remachine tab to limit the search for remachining regions or create a separate feature with only the surfaces of interest.

2. **Some fillets/corners are not being detected.** Decrease the tolerance on the Remachine tab.

Projection milling methods

Projection milling techniques are a robust and easily understood method of generating 3D toolpaths. They work by taking a pattern of curves and projecting points from these curves onto the surfaces of the part.

Advantages

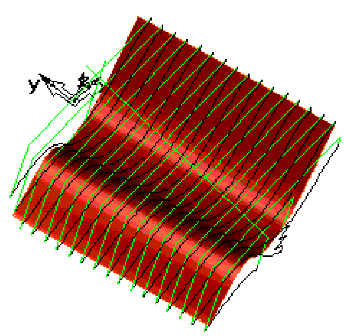
- Robust
- Handles overlapping surfaces well.
- Good for multiple surface manufacturing .
- Surface normals are not considered for the manufacturing computation

Disadvantages

- Non-uniform finish.
- Poor handling of nearly vertical surfaces
- Roughing passes can lead to non-uniform depth of cut.
- See *Restrictions of projection milling techniques* for more details.

X parallel

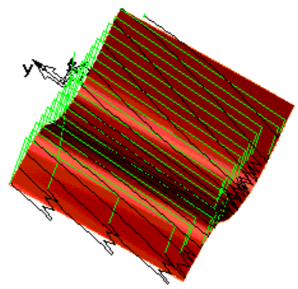
X parallel roughs the feature in cuts that are parallel to the X-axis of the current setup. To slant the toolpaths relative to the axis, set the Parallel angle attribute.



This method mills only the area above and down to the surface(s) in the feature. If part of the stock does not have a feature surface in it, that area won't be milled away, except perhaps incidentally to the manufacturing of another surface feature. For roughing use the Z increment (3D) attribute to cut at multiple depths.

Y parallel

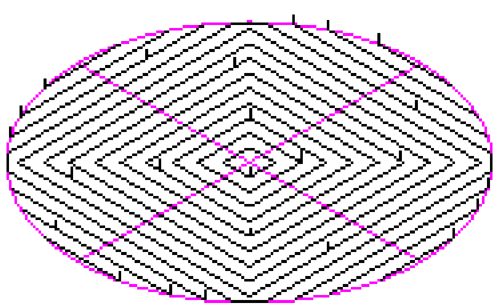
Y parallel roughs the feature in cuts that are parallel to the Y-axis of the current setup. To slant the toolpaths relative to the axis, set the Parallel angle attribute.

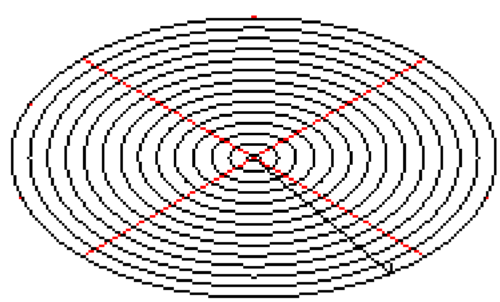


This method mills only the area above and down to the surface(s) in the feature. If part of the stock does not have a feature surface in it, that area won't be milled away, except perhaps incidentally to the manufacturing of another surface feature. For roughing use the Z increment (3D) attribute to cut at multiple depths.

Spiral

Spiral toolpaths mill a feature in a continuous spiral either towards the feature center or away from the feature center. Spiral is only a general description of the actual path as not all feature shapes lend themselves to a truly spiral pattern. The pattern is obtained by taking the stock boundary, the feature boundary or the curve, as specified in the Boundaries tab, and offsetting this curve toward the center of the part. To use the stock boundary, click *Use stock dimensions as boundary* on the Boundaries tab. This will result in a square shape to the toolpaths. To automatically calculate the silhouette boundary of the surfaces of your feature, click *Use part surfaces as boundary*. This toolpath will mimic the boundary of your feature. The figure on the right shows a spiral toolpath using the stock boundary. The left-hand figure shows the same operation using the part surfaces as it boundary.



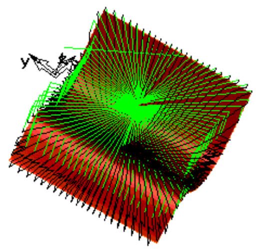


If you want to use a different curve as your toolpath shape, click *Use a curve as boundary* on the Boundaries tab. See page 390 for additional details on additional options.

For roughing use the Z increment (3D) attribute to cut at multiple depths. The Spiral Milling tab controls the shape of the spiral.

Radial inward

Radial inward milling works its way from outside boundary of the feature towards the middle of the feature with each pass. This figure shows a radial toolpath example.



The center of the pattern is automatically calculated unless the *Center point* milling attribute is set. Not all feature shapes lend themselves to this manufacturing style. Features long in one direction and skinny in the other probably aren't best milled in this way. For roughing use the Z increment (3D) attribute to cut at multiple depths. The radial term is only a rough description of the actual path as not all feature shapes lend themselves to a truly radial pattern.

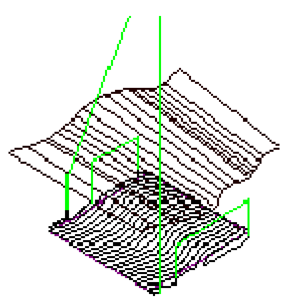
Radial outward

Radial milling works its way from the middle out. Not all feature shapes lend themselves to this manufacturing style. The center of the pattern is automatically calculated unless the *Center point* milling attribute is set. Features long in one direction and skinny in the other probably aren't best milled in this way. The radial term is only a rough description of the actual path as not all feature shapes lend themselves to a truly radial pattern. Again, long skinny shapes will not be efficiently machined with this pattern.

For roughing use the Z increment (3D) attribute to cut at multiple depths.

Flowline guide surface

Setting a Flowline guide surface is a way to manufacture the surface milling feature using the isolines of a different surface to generate the projected toolpaths. This figure shows a flat surface cut with the upper surface used as the flow line guide surface.



How to use a flow line guide surface

1. Select the surfaces you want to cut as the part surfaces for your surface milling feature.

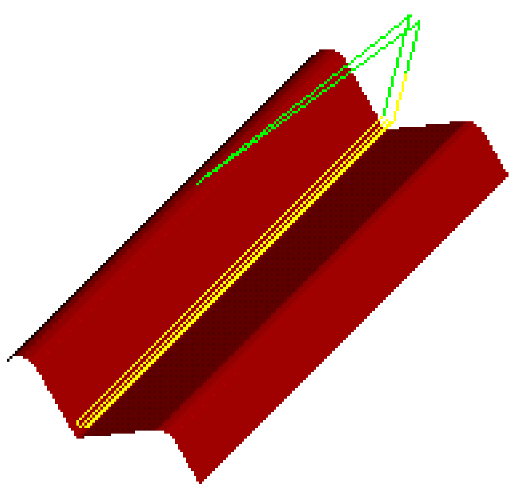
2. Create an isoline milling operation.
3. Edit the feature.
4. Click on the word, *isoline*, in the tree view and click on the surface control tab.
5. Set the flowline guide surface checkbox, click on the Pick surface  button and click on the flowline guide surface.

How to create a projection milling operation

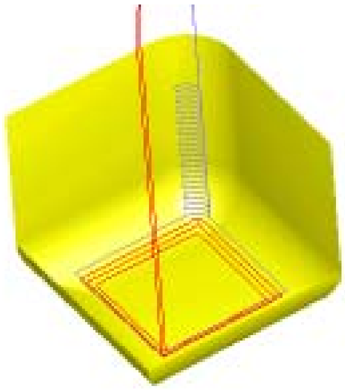
1. Create a surface milling feature using the Feature Step .
2. Specify *Parallel, Radial or Spiral* as the *Strategy* on the New Strategy page and *Rough* or *Finish* as the operation on the Strategy page.
3. At the conclusion of the wizard Click *Finish*.

Remachining

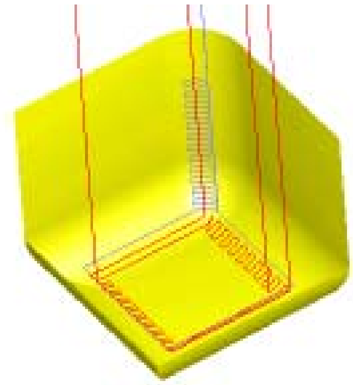
Remachining is used to automatically mill regions that were not cut by previous operations. You provide the diameter of the previous tool that was used to cut the part and FeatureCAM automatically determines the uncut regions and applies a toolpath to them. In the example below, the trough region is automatically calculated and the parallel toolpath is limited to this region.



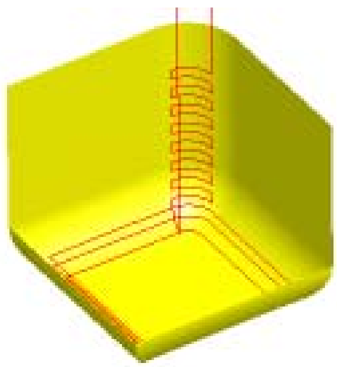
Remachining applies to spiral, parallel, z-level and horizontal+vertical combination operations. Since most remachining areas have both steep and shallow regions, horizontal+vertical is recommended.



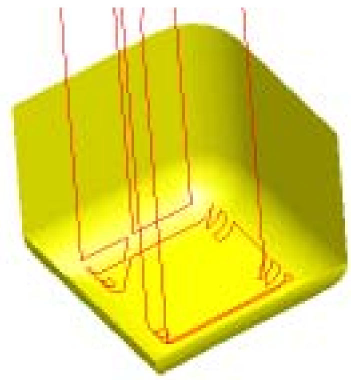
Horiz Spiral + Vert



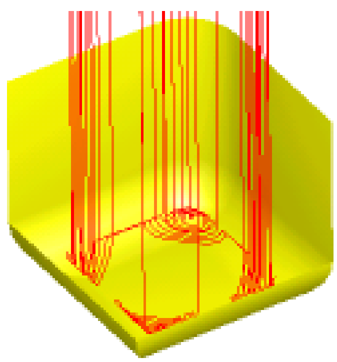
Horiz Parallel+ Vert



Z-level



Parallel



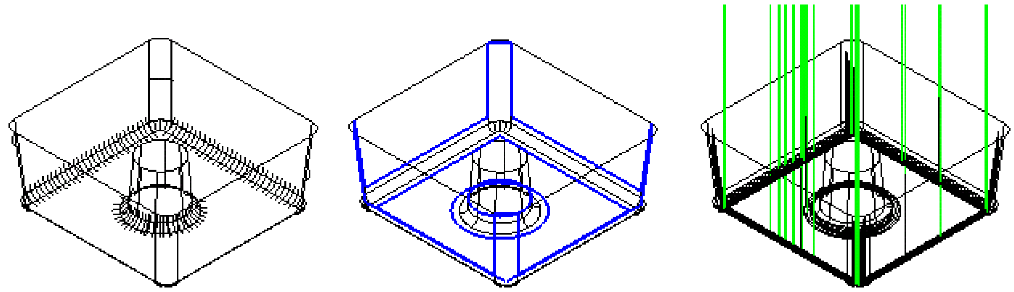
Spiral

Remachining can be used to:

Remove material that could not be reached with a larger tool for both finishing and roughing.

Pre-relieve corners roughed with a larger tool for high speed finishing with a smaller tool.

The process of remachining has three steps. First, locations are identified where the previous tool would have contacted the part at two or more points. These locations are shown during toolpath creation by the arcs shown in the left-hand figure. Second, these locations are joined into regions as shown in the middle figure. Third, toolpaths are created within this region.



How to create a remachining operation

1. Create a parallel, spiral , -level horizontal+vertical finishing surface milling feature

2. On the Strategy page, click *Remachining* and enter the Previous tool diameter. This diameter will be used to calculate the region for remachining.

3. At the conclusion of the wizard, click the *Finish and edit properties* button. If this button is not displayed, hold down the triangle on the right side of the *Finish* button and select *Finish and edit properties* from the menu.

4. Click on the operation you just created and click on the Remachine tab. See Tree view for surface milling features for more information on the surface milling dialog box.

5. If using a spiral operation, it is recommended to click the *Improve region* parameter to prevent the calculated regions from becoming ragged.

6. Click OK.

7. If the toolpaths do not look correct see Troubleshooting remachining for hints.

Limitations of remachining

1. Parallel and spiral remachining operations have the same limitations as projection toolpath techniques.

2. Z-level Remachining operations have the same restrictions as normal Z-level finishing operations.

3. Remachining provides approximate boundary curves to limit the machining to the appropriate areas. Spiral operations near vertical walls are not very smooth. In general the Horizontal+Vertical technique is recommended.

Troubleshooting remachining

1. **Remachining is not generating toolpaths.** Use the *Preview region* button on the Remachining tab to see if the regions look smooth and connected.

   - Examine your part and make sure there are really regions where the previous tool diameter would not fit. Remember that fillets are specified by their radii and tools are specified by their diameters.

   - If small jagged regions that you do not want to cut are displayed, increase the *Minimum arc angle* parameter or decrease the *Boundary tolerance.*

- If the boundaries of the regions cross themselves, increase the *Minimum arc angle* or use *Improve region*.

- If corners where three fillets (or tight regions) are not being properly included in the boundaries, Decrease the *Grid size* parameter, and possibly decrease the tolerance as well.

2. **Spiral toolpaths are jagged near vertical walls**. Use the Horizontal+Vertical technique instead.

3. **Calculating the regions takes too long.** Use a stock boundary on the Remachine tab to limit the search for remachining regions or create a separate feature with only the surfaces of interest.

4. **Regions that are exactly the same diameter as the previous tool are being ignored.** If there is a region that you would like to remachine that the previous tool will exactly fit into it is best to slightly overstate the *Previous tool diameter* to ensure that the region is properly remachined. For example if you have a pocket with a 0.25 radius corner fillet, you should set the *Previous tool diameter* to 0.51 inches. The region will be larger than necessary, but the remachining boundaries will be more predictable.

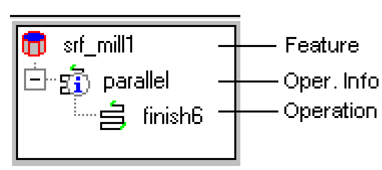
Restrictions of projection milling techniques

- A drawback of projection manufacturing is a non-uniform finish. Because the pattern is uniformly spaced across the XY plane (rather than at a uniform distance across the surface), the cusp height of the passes varies according to the surface. This can results in uneven surface finishes for walls that are tangent to the toolpath direction.

- Vertical surfaces are almost invisible to projected manufacturing methods because the vertically projected lines may not detect the vertical surface. The top edge of such a surface is protected, but the actual surface may not be milled. Surfaces that are exactly and completely vertical must be machined with another technique. Vertical surfaces might be better candidates for a Side feature instead of a surface.

3D operation attributes

Tree view for surface milling features



The tree view for surface milling features has changed with Version 8 and the tabs of the feature properties dialog box have been reorganized. The tabs displayed in the dialog box change depending on what level in the tree view you have clicked on.

The top level of the tree view is the *Feature Level*. The tabs available at this level are:

Dimensions– specify the part and check surfaces
Process– create, delete and reorder the operations of the feature
Machining Side – control which side of surfaces to machine
Misc. – a variety of feature-level attributes.

The next level is the *Operation Info*. level. The tabs available at this level are:

Strategy– rough/finish classification, edge roollover and Remachining.
Boundaries– boundary curves (formerly known as stock curves) for all operations.
Slopes– angle limits for applying toolpaths

Surface Control – exclude feature surfaces for specific operations

The third level is the *Operation Level*. The tabs available are:

Tools– view selected tool or change to a different one
F/S – view automatically calculated feed or speed or change feed or speeds
Milling– operation level attributes
Leads – control leads and ramps

Feature level tabs

Dimensions tab

The dimensions tab has three buttons that open dialog boxes where you select surfaces to manufacture, surfaces to use as protected areas where the tool must not go, and set a Stock curve to use for localizing the milling.

Part surfaces

Use this dialog box to pick surfaces you want to include in your 3D part feature. To specify part surfaces:

1. Click the *Part surfaces* button.

2. Select surfaces by either:

- Checking the surface(s) in the list box;

- Clicking the Pick surface button and select a surface with the mouse. To pick additional surfaces, click Pick again before selecting each additional surface; or

- Select the surface(s) in the Graphics window and hitting the  key to add the surfaces to the selected list

3. Click OK to return to the 3D feature properties dialog box.

You need to consider the following when specifying part surfaces:

1. You can't manufacture undercut surfaces using 3-axis machining, so it's a good idea to use only surfaces in the feature that can be cut from the setup.

2. Some surfaces may be cut from multiple setups to manufacture all parts of the surface. In such situations, a Stock curve is helpful in limiting the machining area to just those spots that need it.

Check surfaces

Use this dialog box to select surfaces you want to use to limit machining in a 3D feature. Check surfaces are surfaces that denote area that won't be milled away. You should select surfaces that are more horizontal than vertical. The check surface acts as a boundary up to which milling occurs. With vertical check surfaces the milling will stop there, but may resume on the other side of the check surface if the surface to be milled extends beyond the check surface. To add check surfaces, use the same procedure described above for adding part surfaces.

Process tab

The Process tab shows the operations that are included for milling the feature have checks set in their checkboxes in the operations list. You can turn off operations by unchecking them. This allows you to customize operations and then turn them off to reduce screen clutter when working on subsequent operations.

The buttons on the right have these functions:

New operation: Allows you to create a new operation using a wizard.

Delete operation: discards an operation from the feature. You can disable an operation instead by unchecking the operation. The checkbox, when set, includes that operation in the feature.

Move operation up: changes the order of operations in the milling feature by moving the selected operation up one place in the feature.

Move operation down: changes the order of operations in the feature by moving the selected operation down one place in the feature.

Misc tab

These attributes are the same as the 2.5D attributes described on page 224.

Machining side tab

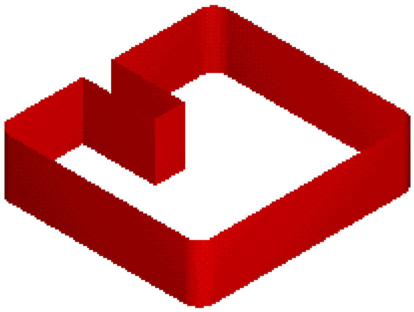
FeatureCAM attempts to cut on the appropriate side of a surface based on the surface normals, but occasionally you will need to explicitly orient a surface. This will mainly be for the isoline, Z level rough or z level finish techniques.

If you click on a surface name, an arrow is shown in the graphics window that indicates which side of the surface will be cut. If you want to reverse the side:

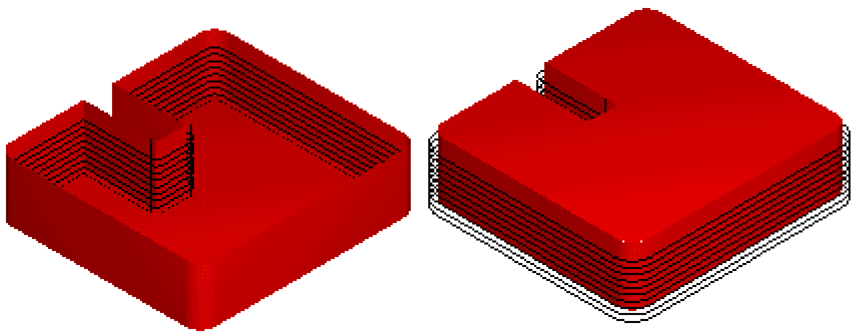
1. Select the name of the surface in the list. If you do not know the name of the surface, click the pick surface button and select the surface in the graphics window. The name of the surface is then highlighted in the list.
2. Click the Switch machining side button.
3. If you want to return the machining side of all surfaces to the FeatureCAM defaults, click the *Reset normals* button

Automatic normal flipping

Vertical surface are usually the only surfaces that will need to be manually oriented in the Machining side tab. FeatureCAM tries to automatically flip surface normals so that they are consistent. For this automatic flipping to work on vertical surfaces, there must be some non-vertical surfaces nearby. For parts that have only vertical surfaces, FeatureCAM does not automatically flip any normals. That means that you will have to ensure that all surface normals point either in or out by manually flipping the normals to ensure a consistent cutting side.



For open surfaces with a floor, the normals will be flipped to create a pocket shape and the insides will be milled. For open surfaces with a top, they will be treated as a boss shape and the outsides of the surfaces will be milled.



Operation info level tabs

Strategy

This tab allows you to fine tune your cutting strategy.

To complete this page:

1. Click *Rough* for a roughing pass, or *Finish* for a finishing pass.
2. The following options are available for certain types of operations
   a. If you are performing radial milling, select *Radial in* to cut from the edge of the part toward the center. Select *Radial out* to cut from the center toward the edge.
   b. For spiral milling, select *Spiral in* to spiral from the edge in toward the center. Select *Spiral out* to spiral away from the center.
   c. For parallel milling, select *X parallel* to cut parallel to the X axis, *Y parallel* to cut parallel to the Y axis. Also specify an *Angle* (measured in degrees in the counter clock-wise direction) to rotate the toolpaths off of the principal axis.
   d. For Z level roughing or Z level finishing, classify the slices as either:
      - 3D Boss - mills outside the Z level slice curve.
      - 3D Pocket - mills inside the Z level slice curve.
      - Automatic - analyzes the surface and its normals to decide where to mill on the surface

e. For Z level roughing set the Min. retract attribute. See page 186 for details on this attribute.

f. For Isoline and pencil milling, there are no options.

3. Edge rollover – Set the edge rollover method as described below.

4. If you wish to use this operation to clean up areas that were missed by previous toolpaths, check the *Remachining* checkbox. See also page 397 for information on the operation level Remachining tab.

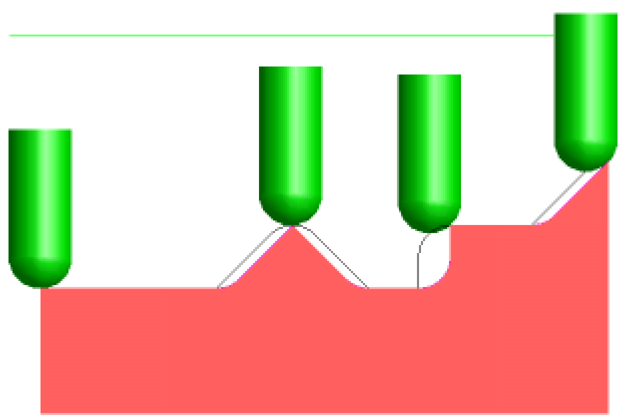
Edge rollover

Edge rollover gives you three choices on how the toolpaths are generated by near surfaces edges. It also affects the speed of toolpath generation. Edges are the boundary curves of the surfaces or sharp "creases" internal to a surface is considered an edge as well. Edge protection affects only Projection milling methods.

Edge protection internal only

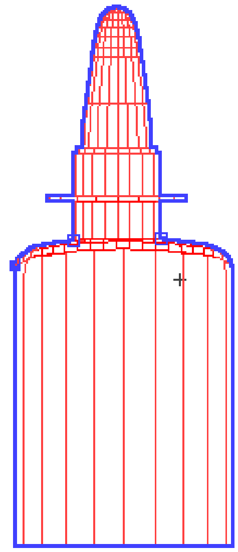
This type of edge rollover is the default method. For the majority of parts, this type of edge rollover should be used.

Internal only protects only those edges internal to the surface or group of adjacent surfaces up to the silhouette edge. The milling doesn't move beyond the boundary edge, but instead rolls up to the edge until the tool just contacts the edge. In the figure below, note that the tool rolls over the edges that are internal to the surface, but the tool stops on the right edge as soon as the side of the tool contacts the edge. The tool does not roll over the boundary edge.

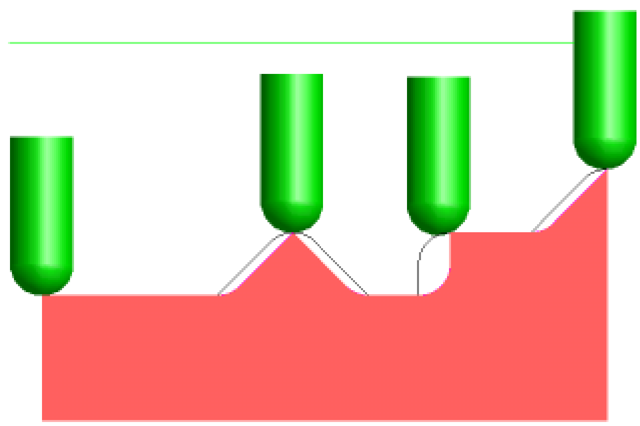


Edge rollover all

All is a way to protect all silhouette edges of a part by rolling over these edges. The silhouette edges are the edges that would cast a shadow if a light were held directly above the part. The blue edges in the figure below are silhouette edges.



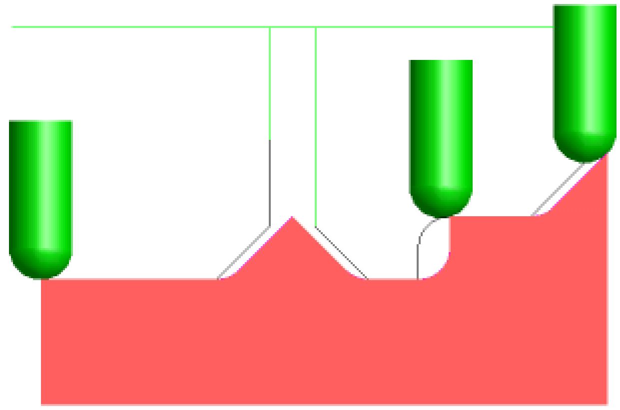
This option is more efficient and uses less memory than Internal edges only, but may cause undesirable results for some models by rolling too far over the edge. For edges that are not parallel to the direction of the cut, the tool will travel up and over the silhouette.   In the figure below, note that the tool rolls up over the sharp corners that are internal to the surface. In addition the tool rolls up and over the part boundary on the right side until the center of the tool contacts the edge.



For some parts this behavior may be undesirable. For most cases, the default setting of *internal only* should be used.

Edge rollover none

None provides no rolling over of edges. The toolpath stops when the center of the ball of the tool is in line with the surface normal. For surfaces with sharp corners the tool will retract. This technique works best for generating toolpaths for a single surface. For most cases, the default setting of *internal only* should be used.

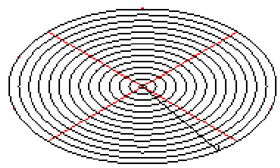


Warning: This technique is not recommended for multi-surface machining as it may result in gouging.

Boundaries tab

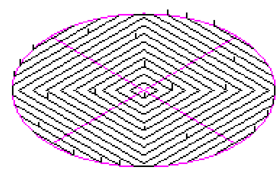
This tab controls the boundaries of the toolpaths.

Note: The Boundaries page does not apply to Z level rough operations.

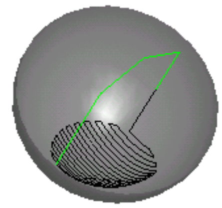


To complete this tab:

1. Click *Use part surfaces as boundary* to create toolpaths on the surfaces regardless of the location of the stock. If you are using a spiral toolpath, this option automatically calculates the silhouette curve of the feature and uses this curve as the shape of the spiral. This generally has the best shape for spiral toolpaths.

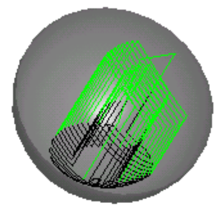


2. Click *Use stock dimensions as boundary* to restrict the toolpaths to the portions of the surface feature that are within the stock. If you are using a spiral toolpath, this option uses the outline of the stock as the shape of the spiral.

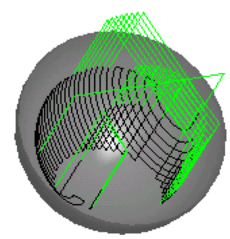


3. If you want to use curve(s) to restrict the toolpath boundaries or to affect the shape of a spiral toolpath, click *Use a curve as the boundary*. The following options are available.

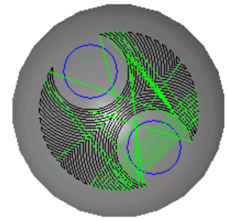
   - *3D pocket*– The toolpaths are restricted to the regions inside of the curves specified as the *boundaries*. Island curves can also be specified for 3D pockets. The toolpaths will be generated outside of the island curves, but inside the boundary curves. The island curves must be inside of the boundaries and must not touch the boundaries.



   - *3D boss* - The toolpaths are restricted to the regions outside of the curves specified as the boundaries.

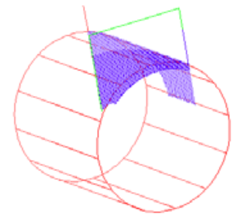


   - Boundary curves may extend beyond the stock or beyond the surfaces of the feature. Regardless of the curves that are specified, toolpaths will not extend beyond the surfaces of a feature.

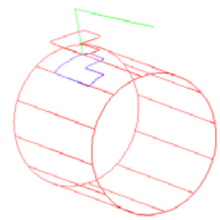


- *Curve allowance*– The distance to stay away from the boundary or island curves. This must be a positive number and is an absolute distance.

4. If you are using spiral toolpaths, the shape of the spiral toolpaths are determined by the boundary and island curves and the classification of *boss*, *pocket*, *side* or *wall only. 3D side* and *Wall only* apply to spiral toolpaths are are defined as follows:



- The 3D side setting will project the boundary curve onto the surface and then cut on one side or the other as shown in the figure on the right. In this case the side curve is a straight line.



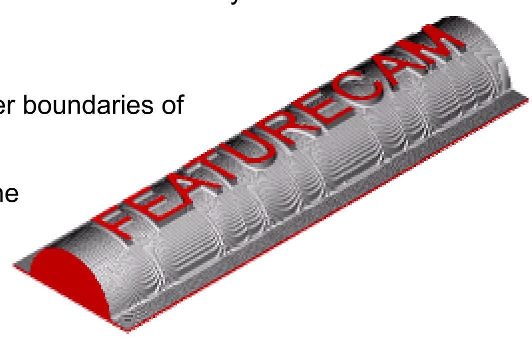
- The Wall only setting will trace along the curve as shown with the letter “P”. Note that for *Wall only* profile types, click the *Other side* button to offset to the other side of the curve.

  Multiple boundary curves can be specified, but these curves must not touch.   If no boundary curve is specified, then the stock boundary will be used.

5. *Total offset* - This is the total distance away from the boundary curve to cut for boss or side profile types.

How to create a 3D boss from font curves

1. Create the surface or surfaces you want to use as the floor of your feature.
2. Create your text.



3. Use Extract font curve and extract the outer boundaries of the text.
4. Use Extract font curve again and extract the islands.
5. Create a spiral out 3D boss operation for the floor surfaces using the outer boundary curves as the boundaries.
6. Set the curve allowance to the tool radius.
7. Create another spiral out 3D pocket operation for the floor surfaces using the island surfaces as the boundaries.

Again, set the curve allowance for this operation to the tool radius.

Slope limits tab

Many of the 3D toolpath techniques can be limited to regions of the part by angles. These controls are located on the Slope limits tab. The options are:

None: All regions of the surfaces of the model will be included regardless of slope.

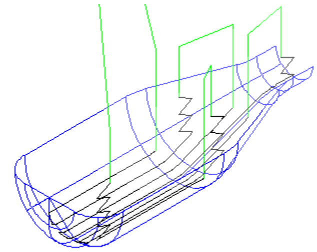
Horizontal only: Limit cutting to regions with a slope less than Maximum surface slope

Vertical only: Limit cutting to steep regions with slope greater than Minimum surface slope .

Steep slope remachining: Limit cutting to areas where the toolpath has a slope at least

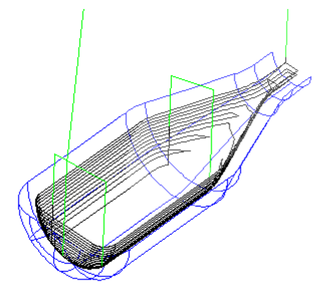
Minimum surface slope in direction of toolpath.

Maximum surface slope



Maximum surface slope limits the toolpaths to portions of surfaces with a slope less than that angle. This is a good way to limit milling to the "flatter" portions of your model. This attribute applies to X parallel, Y parallel, Radial inward and Radial outward milling techniques. This figure shows an example of limiting X parallel toolpaths to the bottom of the mold by setting Maximum slope angle to 45 degrees. Note that the toolpaths do not climb up the steep walls.

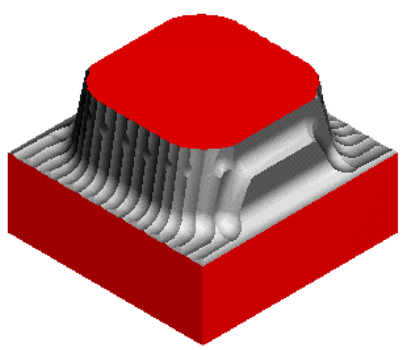
Minimum surface slope



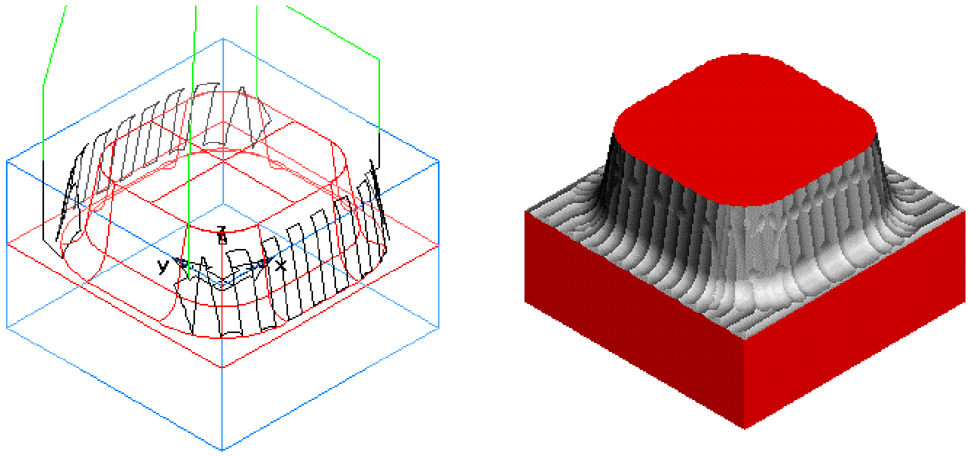
Minimum surface slope limits toolpaths to surface portions that have a slope greater than that angle. It works well to limit toolpaths to the "steeper" portions of the model. It applies only to the Z level finish milling operation. This figure shows an example of using Minimum surface slope to limit the finishing passes to the walls. Note that the bottom of the mold is skipped.

Minimum surface slope in direction of toolpath

Minimum surface slope in direction of toolpath will limit toolpaths to the regions where the toolpath itself has a minimum slope. It applies to X parallel and Y parallel techniques only. It works best when you create two parallel cuts in sequence where the second is 90 degrees off of the first one. For example, this figure shows a four-sided surface after roughing in the X direction. It is badly cut on the front and back walls.



A second operation with a Y parallel limited by a Minimum surface slope in direction of toolpath of 45 degrees results in the toolpaths shown on the left. Note that the toolpaths are only where the paths are steep. The result of both operations are shown in the figure on the right.



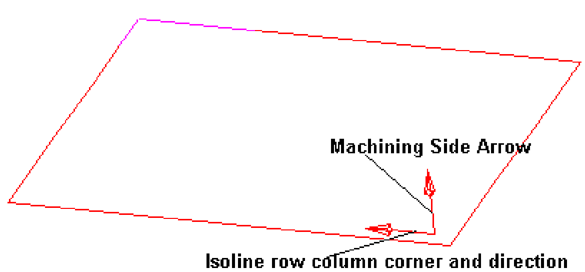
Surface control tab

The Surface control tab allows you to include or exclude surfaces for a particular operation. All surfaces are listed as a row in the table. If the row has a check next to it, the surface is included in the current operation. Uncheck it to remove the surface from the operation. If you select a row in the table, the surface is highlighted in the graphics window. Use the pick surface button  on the right to select a surface graphically and select the appropriate row in the table.

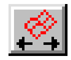
Isoline milling only

The order that the surfaces appear is the order that the surfaces will be milled. The Up and Down arrows on the right side will move the operations accordingly. When a surface is selected in the table, an arrow indicating the starting corner and isoline direction is displayed along the surface normal direction as show above.

When a surface is selected two arrows are displayed as shown in this figure.



The side with the arrow facing up will be machined. If the wrong side is selected, you must go to the Machining side tab to change this. The arrow that is tangent to the surface indicates the starting corner and direction of the first isoline toolpath.

The Start Curve column indicates whether you are going to start milling at the First row, Last row, First Column or Last column. If you want to change start curve of the isoline paths, click the Set Isoline Row Column  button until the arrow in the graphics window is aligned with the correct row or column of the surface. The Cut direction column of the table indicates the direction of the toolpaths. Click the Cut direction  button to toggle the direction along the row or column.

Flowline Guide Surface

If you select a flowline guide surface the isolines of another surface are projected onto all the surfaces of the feature. See page 380 for more information.

Operation level tabs

Tools tab

The Tools tab displays the tool, speed, and feed settings for the operation selected in the tree view.

Tool Diameter sets a specific value for the tool diameter if you want something different than chosen by FeatureCAM. The override checkbox sets itself. If you uncheck the override box, the tool diameter is no longer overridden and FeatureCAM selects the tool to use. For a 3D feature, default diameter is set strictly by the default machining attribute, not dimensions of the feature as for 2 1/2D features.
End radius sets a specific value for the tool end radius. The override checkbox sets itself. If you uncheck the override box, the end radius is no longer overridden and FeatureCAM selects the tool to use.

Speed determines how fast the tool spins in RPM. You can set a specific value if you wish. The override checkbox sets itself. If you uncheck the override box, the speed is no longer overridden and FeatureCAM selects the speed to use.

Feed determines how fast the tool moves through the stock. You can set a specific value if you wish. The override checkbox sets itself. If you uncheck the override box, the feed is no longer overridden and FeatureCAM selects the feed to use.

Reset All clears any overrides you made to the tool selection and feed and speed and returns the settings to the default value for tool attributes listed on that page.

3D leads tab

The Leads/step tab controls how the tool moves on and off the surface milling feature (lead in/out) and how the tool moves between toolpaths (stepovers).

Ramping

If Ramp to depth is not checked, the tool will plunge to depth. If Ramp to depth is checked and Helical is not checked, the tool will zigzag into the material. The angle of the zigzag passes is controlled by Ramp angle. If Helical is checked the tool will spiral into the material. See *Helical ramping* for more information.

Lead moves

The Use lead in/out controls when lead in/out moves are applied. If Normal to surface is checked, the lead in/out moves are performed with respect to the surface normal. If it is not checked the moves are performed in the plane of the toolpath.

Use lead in/out

The Use lead in/out drop-down list controls when lead moves will be applied. The options are:

- **Never**: Do not use lead moves on this feature.
- **On all plunges/retract**: Apply the leads on all plunge and retract moves.
- **On first plunge/retract**: Apply on the first plunge and first retract move only.
- **On all toolpaths**: Apply the leads on every toolpath.

Lead in/out plane

The arcs and ramps are measured relative to one of the following options:

Normal: Lead in/out moves are relative to the surface normal.

Horizontal: Moves are relative to a horizontal plane.

Vertical: Moves are relative to a vertical plane.

Default: The default for z level finish is *horizontal*, and *vertical* for all others.

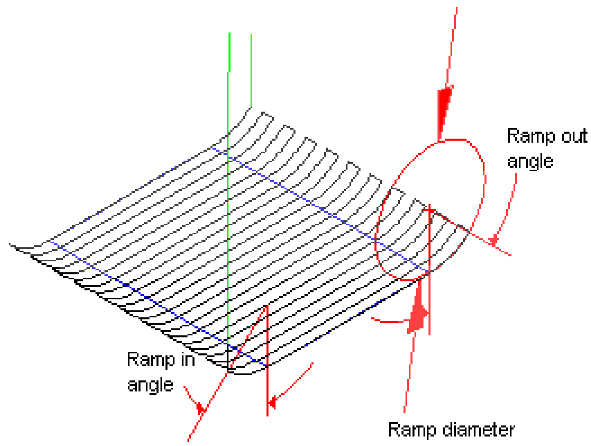
Lead moves are either performed as arcs or linear moves by selecting one of the following categories:

- Use arc ramp in/out
- Use linear lead in/out

Use arc ramp in/out

If Use arc ramp in/out is selected the following parameters are used to control the ramping on and off the part feature:

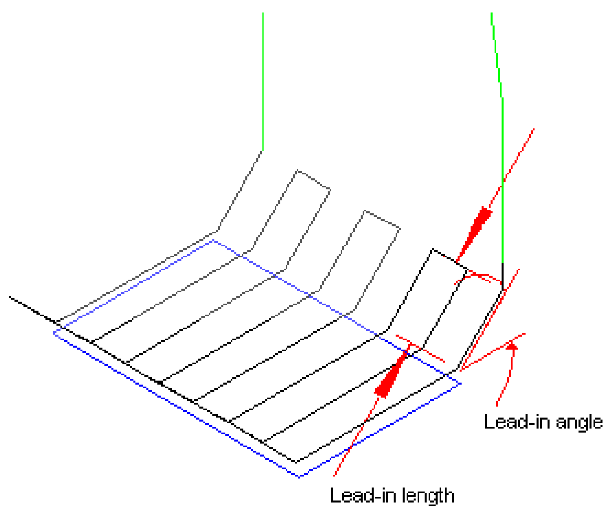
- **Ramp diameter**: The diameter of the ramp move.
- **Ramp in angle**: The angle of the ramp in move.
- **Ramp out angle**: The angle of the ramp out move.



Use linear lead in/out

If Use linear lead in/out is clicked then the following parameters controls the move off of the feature:

- **Lead-in angle**: Angle measured away from the toolpath for the lead-in move.
- **Lead-out angle**: Angle measured away from the toolpath for the lead-out move.
- **Lead-in length**: Length of the linear lead-in length.

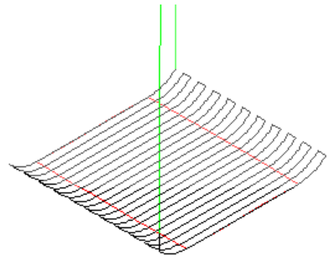


- **Lead-out angle**: Length of the linear lead-out length.

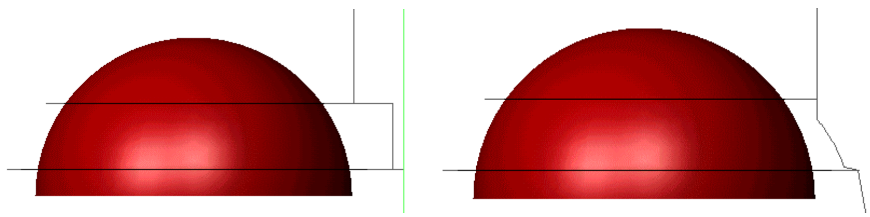
Stepover type

The Stepover type controls the type of transition move that is inserted between toolpaths. The choices are:

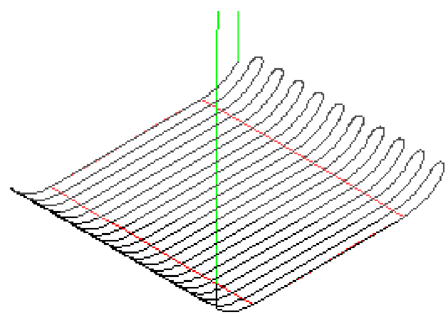
- **Direct**: The tool moves straight over to the next position. The tool can move in all 3 axes. This figure shows a direct stepover move on a flat surface feature.



- **Stair step:** The tool moves up in Z and then over in X and Y. The left-hand figure shows a stair step transition move on a spherical surface. The right-hand figure shows the same surface feature using a direct stepover.



- **Loop**: The tool makes an arc move out of one toolpath and an arc move into the next toolpath. These transitions are actually programmed from linear moves and may move all three axes. This figure shows a loop move on a flat surface feature.



Remachine tab

This tab is available on pencil milling operations and on spiral and parallel Remachining operations only.

Tool diameter

Either the *Previous tool diameter* for remachining, which is settable on the tab, or the pencil milling tool diameter is shown. The pencil milling diameter is not settable. Use the *Tools* tab to choose the tool.   For remachining, *Previous tool diameter* is used to determine the region for remachining.

Stock boundary

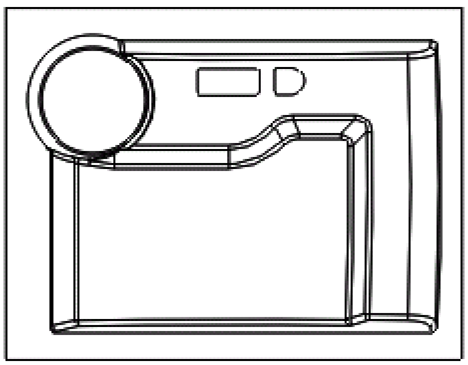
Choose a stock boundary to roughly limit the portion of the part that will be searched for remachining regions.

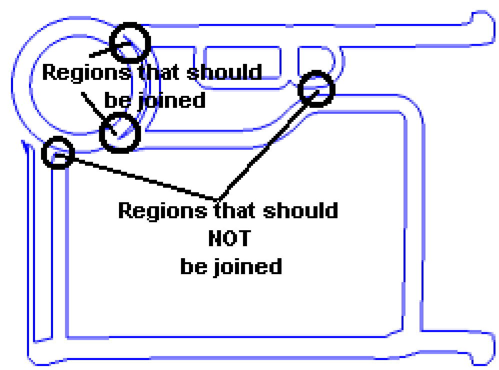
Tolerance

For pencil milling, this is the tolerance for the final toolpaths. For remachining it's called *Boundary tolerance* and is used to find the remachining regions and generate the boundary curves. The *Tolerance* on the Milling tab is for the actual toolpaths.

Grid size

The *Grid* size is used in the second phase of remachining or pencil milling to sort and connect the arcs into regions. See page 381 for information about the phases of remachining. The default value for *Grid size* depends on the size of the part (or stock curve) and the *Previous tool diameter* for remachining or *Tool* diameter for pencil milling. In order for 3 or more regions to be joined correctly at a corner where they come together the grid size has to be sufficiently small to distinguish which region an arc belongs to. Generally the default value is used and it is not recommended that this value be increased. Smaller values need to be used to correctly join multiple regions in certain instances. Using smaller values will decrease performance, so try the default value before adjusting this parameter. The right-hand figure shows the proper regions for the camera mold displayed on the left.





Preview region

Shows the boundary curves of the remachining region. Forces the region to be computed and saved on the feature if necessary. Clicking a second time unshows the curves.

Improve region

If checked, this option will perform additional processing to improve the calculated boundary of the region. This setting is recommended if remachining using spiral milling techniques. It should prevent the boundary from becoming ragged.

3D milling attributes

Switches

Not all switches are available for all operation types.

Bi-directional sets the operation to mill in each direction. You can combine this setting with Climb Mill to force the first pass to be a Climb mill cut.

Climb Mill sets the operation to use climb milling. When set in conjunction with Bi-directional milling, it ensures that the first pass will climb mill.

Improved uses a finer algorithm to approximate the feature boundary but takes longer to compute. You can combine this switch with any of the other settings.

Arc/line approx. creates an arc line approximation for toolpaths that are contained in the XY, YZ and XZ planes. This allows 3D programs to be smaller and to result in smoother surface finishes for certain types of parts.

Arc/line approximation applies to the following 3D techniques

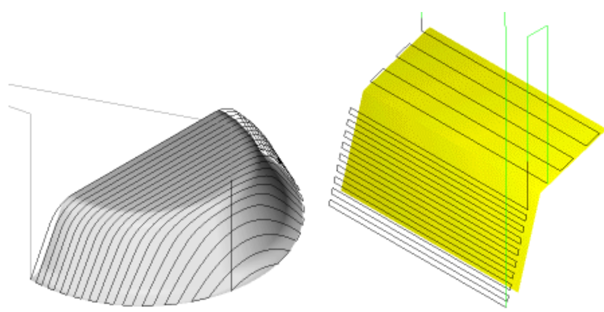
- X-parallel and Y-parallel roughing or finishing with Parallel Angle set to 0
- Z-level finishing
- Isoline finishing where the toolpaths line in a plane

Note that there is no arc/line approx attribute for Z level rough. Arc/line approximation is always performed on Z-level roughing passes.

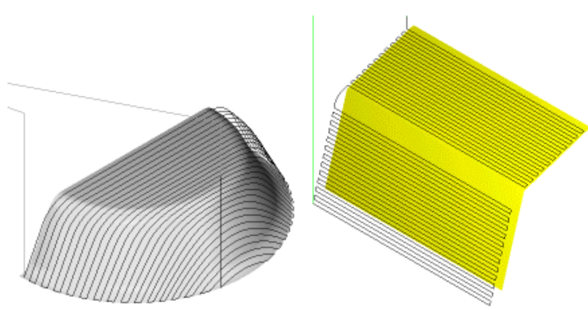
Scallop height is available on the Milling page of projection milling finish operations and Z-level milling finish operations. For projection milling methods, it toggles the way that you specify how far the tool moves over between passes. With Scallop height unchecked, you specify the stepover. With it checked, you specify the scallop height.

For Z-level finish operations it toggles how you specify how the tools moves down in Z. With Scallop height unchecked, you specify the Z-increment. With it checked, you specify the scallop height.

With scallop height checked, spacing of the toolpaths are calculated along the surfaces to provide a uniform surface finish. These figures shows surfaces cut without using scallop height.



These figures show the same surfaces cut with scallop height.



Check allowance

Check allowance sets the distance that the tools will approach, but not pass closer than for a check surface. If Check allowance is set to a specific value, that value is used for both rough and finish passes. If it is left blank then if behaves differently for roughing and finishing. If left blank for a roughing pass then check allowance is set to finish allowance + leave allowance. If left blank for a finishing pass then check allowance is set to leave allowance.

Cut direction

Click the *Cut* direction button to reveal the cut direction dialog box. This dialog box provides controls for the direction the tool will cut. Different options are available for certain toolpath techniques.

Style Options

- *Unidirectional* – Toolpaths will only go in one direction. For Z-level toolpaths, the conventional/climb mill parameters control the direction. For other toolpaths, the decreasing/increasing controls the direction.
- *Bidirectional* – If active, the decreasing or increasing parameters control the initial cut's direction.
- *Uphill only* – Breaks toolpaths up into segments that increase in Z. If this option is checked all direction parameters are dimmed since Up hill only fully determines the cut direction.
- *Downhill only* -- Breaks toolpaths up into segments that decrease in Z. If this option is checked all direction parameters are dimmed since Down-hill only fully determines the cut direction.

Direction

- *Conventional* – Only applies to Z level with unidirectional set. The tool rotates against the direction of the cut.
- *Climb mill* – Only applies to Z level with unidirectional.   If climb mill is set the tool will rotate in the direction of the cut.
- *Decreasing* – Forces the cut to decrease in its principal direction. For X-parallel operations, the tool will start at the maximum X value and cut in the negative direction.
- *Increasing* – Forces the cut to increase in its principal direction. For X-parallel operations, the tool will start at the minimum X value (or Y value for Y-parallel) and cut in the positive direction.

If unidirectional is set, the Increase and Decrease options only affects the direction of the initial cut. For spiral milling use Decrease/Increase to toggle the clockwise/counter-clockwise nature of the paths. For pencil milling Decreasing/Increasing options toggle the direction of cut. For radial milling, use Decrease/Increase to toggle the clock-wise and counter-clockwise ordering of each radial pass around the center.   For parallel milling set Parallel angle to 180 to cut from the opposite end of the part.

First feed override %

The initial pass of many 3D toolpaths are slotting cuts or cuts with increased tool load. This attribute slows the feed rate for these initial pass as a percentage. This percentage is applied to the operation's feed rate to determine the feed rate for the initial pass. This attribute applies to all 3D toolpaths except Z level roughing.

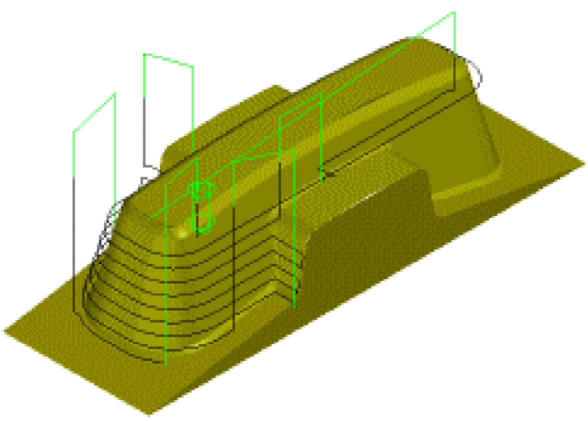
Leave allowance

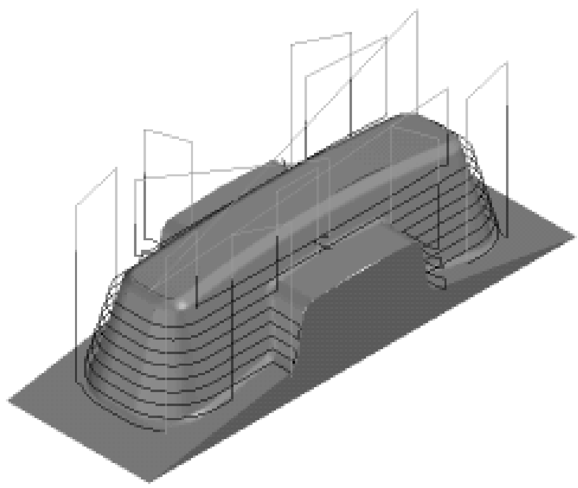
The *Leave allowance* is the amount of material to leave after a 3D finish pass. If unset leave allowance defaults to 0. This number can be set to a negative number (up to - tool radius) to allow for shrinkage or spark gaps. If set to a negative number, the part will be machined into the part surfaces by the negative amount specified.

Reorder

The *Reorder* attribute tells FeatureCAM to sequence the toolpaths to minimize retractions while avoiding full width cuts. Use Reorder when you have a part where several separate regions are cut. If you want the toolpaths to move directly across a surface without worrying about retractions, turn off Reorder.

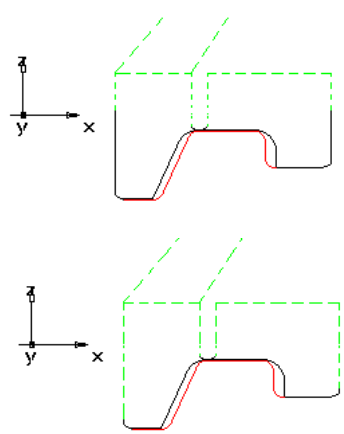
For Z-level finish operations, the Reorder attribute will enable zone machining, where the toolpaths will descend in the Z (or –Z) direction if that is more efficient than cutting the entire part in complete Z levels. The phone handset example below shows that the toolpaths will





cut the top of the part in complete Z levels and then cut one side and the other.

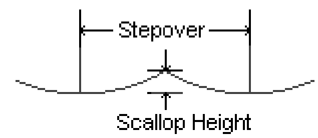
Relative plunge



The *Relative plunge* attribute affects how the Plunge Clearance attribute is used when plunging in 3D machining. Without *Relative plunge* checked, the tool will plunge to the *Plunge clearance* as an absolute value. This can cause the tool to feed an unnecessary amount for parts that are not flat. In the figures on the right an up-hill only toolpath is applied to a simple surface. The dotted lines represent rapid or plunge moves. Relative plunge is turned off in the top example. The tool plunges to a set Z value and then feeds a long way down to the part on each end. In the bottom figure, *Relative plunge* is checked. With this feature enabled, Plunge clearance is used as a relative distance from the surface and the tool plunges down close to the part.

Stepover

Planar stepover distance for finishing with projection techniques. This is the distance between toolpath center line. This distance is measured in the XY plane and then the toolpaths are projected onto the surfaces of your feature. This attribute only applies if the scallop height checkbox is unchecked.



Scallop height

Absolute scallop height between passes for isoline milling, projection milling finishing passes and Z-level finishing. This distance is measured along the surface and represents the maximum cusp height between neighboring passes. This attribute only applies if the scallop height checkbox is checked.

Parallel angle

Parallel angle is available only in X and Y parallel roughing and sets what angle the roughing passes occur in reference to the X or Y axis. This value can be anywhere from -360 to 360 degrees. A positive value rotates counterclockwise from the axis, and a negative value rotates clockwise from the axis. Examples of using *Parallel angle* are:

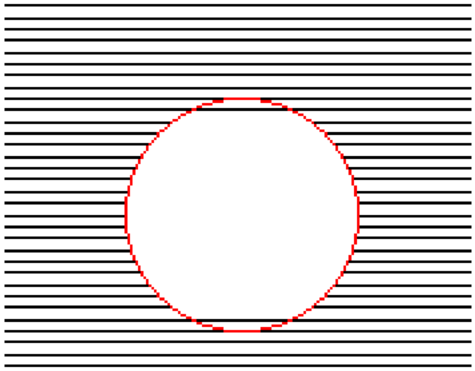
1. Setting the angle to 90 on an X-parallel operation causes it to affectively become a Y-parallel operation.

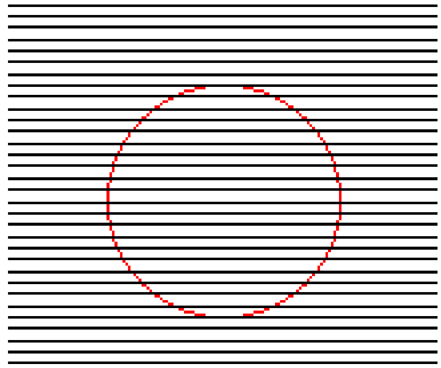
2. Setting the angle to 180 will cause the toolpaths to cut from the opposite side of the part. For example, an X-parallel operation with the angle set to 0 will start at the minimum Y coordinate. With the angle set to 180, the toolpaths will start at the maximum Y coordinate.

Retract gap distance

Retract gap distance controls whether the tool will retract over a gap in a toolpath. It defaults to 55% of the tool diameter. If the distance between toolpath points is greater than Retract gap distance, the tool will retract. If less than, the tool will feed between the points. To override this parameter, enter an absolute distance. This distance is measured in 2D before the toolpaths are projected onto the 3D surfaces of the part.

In the left-hand figure, *Retract gap distance* is set to the default. Note that the tool retracts over the gap on all toolpaths except the first and last toolpaths that travel over the gap. The tool does not retract on these toolpaths because the gap is less than 55% of the tool diameter. In the figure on the right, *Retract gap distance* is set to a large value so that the tool never retracts.

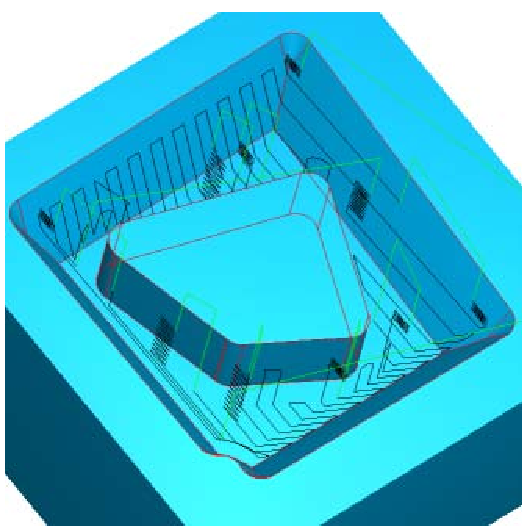


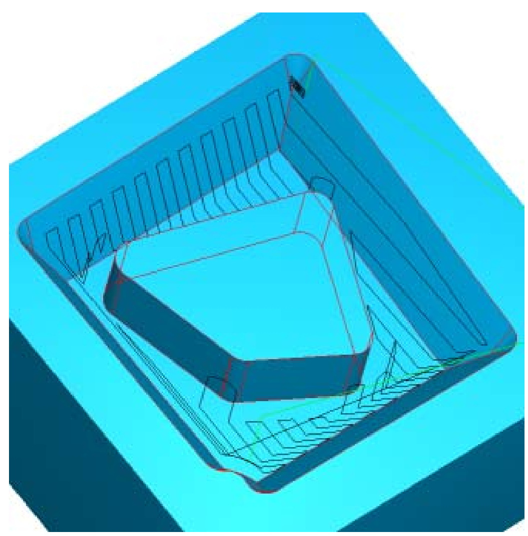


Stepover rapid distance

Stepover retract distance is used just to determine when to feed vs. rapid to the next toolpath. Note that this only applies to stepovers between toolpaths. Use Retract gap distance to prevent retracting over a gap in a toolpath. *Stepover retract distance* is a 3D distance. If not explicitly set, the stepover threshold is the tool diameter plus twice the allowance. For a rough operation, allowance in this context is the *Finish allowance*. For finish operation it is *Leave allowance*.

The practical use of this attribute is to prevent stepovers from climbing up or down a big wall. The left- imahandge shows toolpaths before increasing *Stepover retract distance*. The right-hand figure shows how the tool stays on the metal with an increase in *Stepover retract distance*.

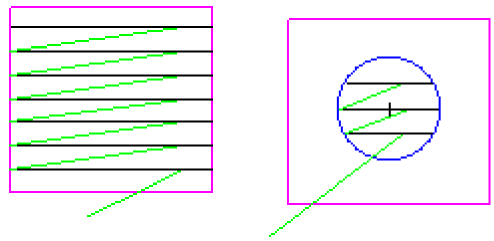




Stock curve

Stock curve sets the area that will be milled away for a specific projection milling operation. Regardless of the stock curve that is specified, projection milling toolpaths will not extend beyond the surfaces of a feature. Stock curves do not affect toolpaths generated with Isoline milling or Z level rough. A stock curve can either be set on a surface milling feature or on the stock itself. A stock curve that is set on a surface milling feature or on a specific manufacturing operation will over ride the curve that is set on the stock object. The stock curve must be a closed curve in the XY plane of the setup.

Without a stock curve a flat surface is cut with an X parallel operation as in the left-hand figure With a stock curve the toolpaths are restricted as in the figure on the right.



Stock curve overcut % affects how closely the toolpaths come to the stock curve.

Stock curve overcut % (3D)

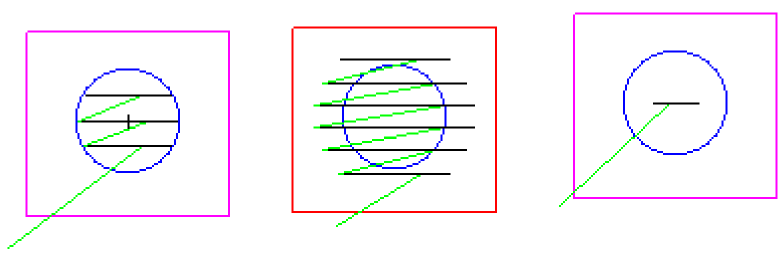
Stock curve overcut % is used in conjunction with stock curve and specifies what percentage of the tool approaches or passes beyond the stock curve boundary.

It can have a value between -100 and 100 with the following meanings:

0 Puts the centerline of the tool on the stock curve.

100 Over cuts the region by a tool radius.

-100 Stops one tool radius short of the stock curve.



Overcut% of 0 Overcut% of 100 Overcut% of -100

Pre-drill (3D)

Pre-drill adds a drilling operation so the end mill doesn't have to ramp in to the stock. The point location and size are determined from your tool settings. If you want to specify those explicitly, set the Pre-drill and Plunge points manufacturing attributes in the Milling tab for the specific operation selected in the tree view window pane.

Skip final pass

For roughing using Y-parallel or Y-parallel milling techniques, the part is roughed at constant Z levels and then an optional pass is performed that is an offset of the feature's surfaces. *Skip final pass* controls whether this optional pass is performed.

Tolerance (3D)

Tolerance sets how close the milling will be to the mathematically ideal surface. This does not guarantee that your feature is machined to this tolerance in all locations if the tool you select is incapable of cutting within that tolerance in constrained areas. If your part shows a faceted appearance, set you tolerance to a lower value.

X Start and X End

For Y parallel projection milling, X Start and X end give the X values for the first and last toolpaths. These attributes are only available for finish operations.

Y Start and Y End

For X parallel projection milling, Y Start and Y end give the Y values for the first and last toolpaths. These attributes are only available for finish operations.

Z end

Z end sets the distance along the Z-axis below which the operation will not mill. You may want to use Z end on an earlier operation then follow it with an operation using the Z start attribute so you control the toolpaths efficiently.

Z increment

Z Increment is the depth of each cut of the facing operation.

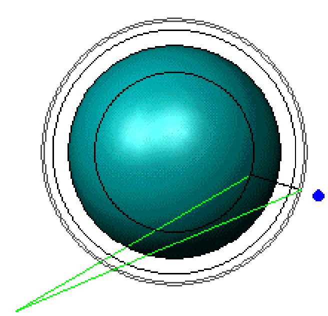
Z start

Z start sets the distance along the Z axis where the milling operation will start. You can use this to save time in manufacturing if the stock material has already been machined away in an earlier operation.

Start point(s)

The *Start point(s)* attribute controls the initial points for Z-level finish toolpaths. Note that it only works for toolpaths that are closed loops. Start point(s) can either be a single point or a curve made from a connected line. The closest point on the toolpath loop relative to the start point is found and that point is used as the starting point for the loop.

In this figure a spherical surface feature is viewed from the top. The blue start point controls the location of the start of each toolpath.



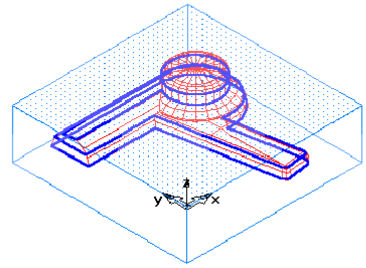
Center point

The *Center point* attribute applies to the Radial inward and Radial outward milling techniques. This point will be projected down onto the surface to become the center of the radial pattern.

Recommended machining strategies

Roughing well-trimmed models

These models should have well-defined cross-sections when sliced so Z level rough is the recommended technique.

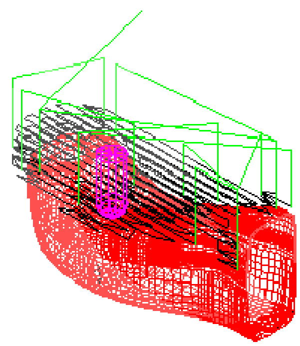


1 Rough with Z level rough with Flat End tool.

- Z Increment = DEFAULT,
- Stepover = 50%
- Tolerance (3D) = 0.005 inches is adequate for most models that fit within a 1 foot cube.

Roughing sloppy data or untrimmed models

If your data has overlapping surfaces or data that does not slice into good contours, use one of the projective techniques to rough the model.



1. Rough with X or Y parallel in longest part direction. Use Flat end tool

- Z Increment = DEFAULT.
- Stepover = 50%.

2. Semi-rough with X or Y parallel finishing perpendicular to roughing direction. Use Flat end tool.

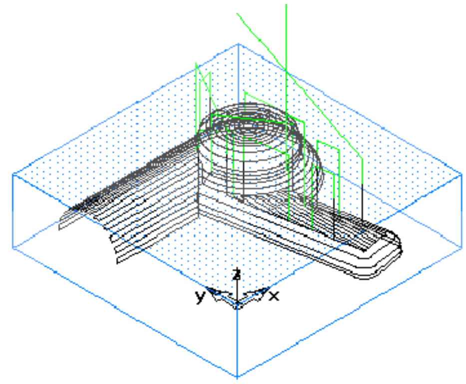
- Z Increment = leave Blank.
- Stepover = 50%.

Semi-finishing and finishing strategies

For semi-finishing and finishing, an effective technique is to use a parallel technique on the flat portions of the model and Z-level finishing on the steeper regions as follows:

1. Create an X parallel or Y parallel finishing operation. Set horizontal-only on the Slope limits tab.
2. Create a Z level finish operation and set vertical-only on the Slope limits tab.
3. Set the angles for each operation so that the two passes overlap by at least 10 degrees.
4. Set the Tolerance (3D) to between 0.001 and 0.0005 for a semi-finishing pass and to between 0.0001 and 0.00005 (if the part has a lot of small details) for a quality finishing pass.
5. Set the Leave allowance to 0 for a finishing pass or to the desired finish allowance for a semi-finishing pass.

Finishing models with few surfaces



Models created in FeatureCAM typically have fewer surfaces than imported data. You also have a better chance of created a well-trimmed model. Isoline milling is recommended since it will give you the best surface quality.

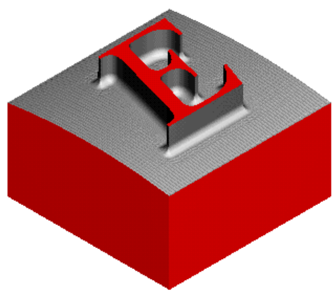
1. Finish with isoline milling using the longest direction of each surface.

- Tolerance = 0.0001 (for parts modeled in inches)

- Scallop height = 0.001 (for parts modeled in inches)

Finishing walls of pocket or boss shapes with a 3D floor

To manufacture this part we will use a spiral milling operation.



1. Create the curve for the boss shape and create the floor surfaces.

2. Create a surface milling feature for the floor surfaces.

3. Rough with a Spiral inward operation. Select 3D boss in the Spiral mill tab and select your boss curve as the boundary curve.

4. Finish with a spiral out operation. On the Spiral Mill tab indicate a 3D boss using the 2D curve as the boundary curve. Leave total stock unspecified if you want the walls and floor finished.

Troubleshooting 3D toolpaths

Troubleshooting Z level roughing

1. **Mills the wrong side of the surfaces**. Toggle the Boss attribute on the milling tab.

2. **Slices are very coarse**. Decrease the tolerance attribute on the milling tab.

3. **Slices are missing**. There is probably a gap in your model and the slice is hitting the gap. Adjust the Z start, Z end or Z increment milling attributes on the milling tab to avoid the gap.

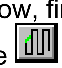
4. **Can't generate toolpaths**

- Preview the slices by selecting the operation in the feature's tree view and selecting the milling tab. Click the Preview button.
- If your slices are not closed non-overlapping contours as show in this figure, you probably need to trim your model. See Restrictions of Z level roughing for more information.
- If the slices look valid, decrease the tolerance and regenerate the toolpaths. Make sure your part is completely inside the stock. You may have to increase the size of your stock.

5. **Cuts an undercut region**. Z level roughing does not detect undercut slices. You must either:

- Remove the undercut surfaces from your feature
- Limit the extend of the Z roughing with the Z start, Z end or Z increment milling attributes

Troubleshooting isoline milling

1. **The toolpaths are on the wrong side of the surface**. Select the operation in the feature's tree view. Go to the Surface control tab. Change the machining side to Reverse using the  button.
2. **Toolpaths should go the other direction or start at the other end of the surface**. In the Isoline control tab the Start curve has the options of first row, last row, first col, last col. Select the surface in the table and toggle the Start curve using the  button. An icon appears on the screen that indicates the start point and the direction of the first toolpath.
3. **Surfaces are cut in the wrong order**. Select the operation in the feature's tree view. Go to the isoline control tab. Use the arrow keys to rearrange the order of the toolpaths.
4. **Toolpaths for a surface are begin reorder strangely**. Select the operation in the feature's tree view. Go to the Milling tab. Turnoff the Reorder option.

Troubleshooting projection milling methods

1. **Bad surface finish on near vertical walls**. Portions of a surface that are tangent to the direction of the toolpath will have larger scallops than other parts of the surface. Possible solutions include:

- Use scallop height instead of fixed planar distance stepovers. This will provide a constant scallop height over surfaces of the model. See item 3 below.
- Finishing the surface with isoline milling
- Finishing the surface in another direction with a projection technique. Use a stock curve if you only want to mill a small portion of the surface milling feature.
- Finishing the surface again with the same technique using a much smaller stepover value.

2. **The toolpaths extend beyond the edge of the surface.** See Stock curve (3D) or Edge Protection.

Troubleshooting Z-level finishing

1. **The toolpaths are on the wrong side of the surface**. Select the operation in the feature's tree view. Go to the Surface control tab. Change the machining side to Reverse using the  button.

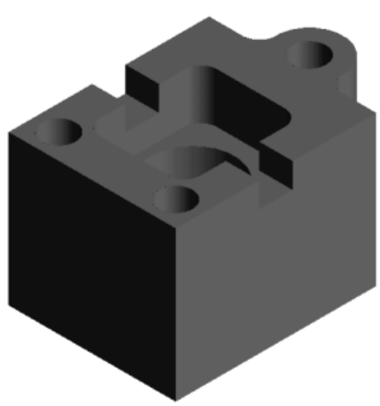
Chapter 23 Feature Recognition

You must license the FeatureRECOGNITION option for the functions described in this chapter.

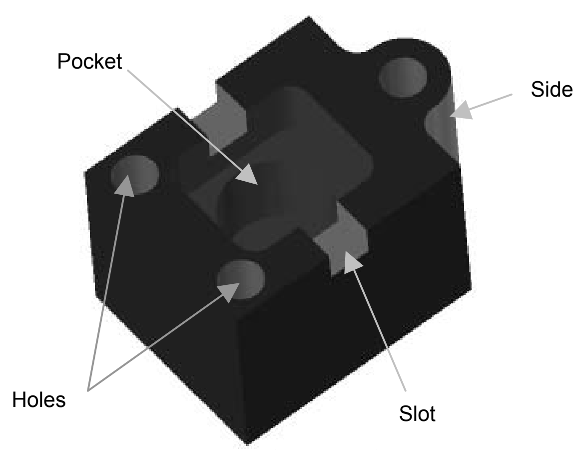
When you create models from scratch in FeatureCAM, you use the set of manufacturing features that the program provides. While this process results in a 3D representation of the feature on the screen, the type of feature and dimensional information you enter provides information that is crucial to the efficient manufacture of the part. For example, knowing that a hole is a counter bored hole, aids in deciding on the appropriate operations for manufacturing the feature. The dimensions are also helpful in determining the dimensions of the resulting operations.

With the popularity of solid modeling, more and more surface/solid models are being created that contain 2 ½ D features (like holes, pockets, or slots). While they look the same as a FeatureCAM feature-based model, they are actually only surfaces. A sample solid model is shown below on the left. The cylindrical voids are only surfaces, not holes. What looks like a FeatureCAM pocket feature is actually only a collection of surfaces. While it has been possible to extract edges from the solid model and join them into curves for features, this process can be time consuming.

FeatureCAM now provides the ability to create features directly from solid or surface models. With this process you classify surfaces in the model as FeatureCAM manufacturing features. The figure on the right shows a classified solid model. The features that you recognize are the same as any that you create from scratch. The same feature-based automation applies so that you can quickly and easily create part programs.



Unclassified solid model



Model classified into features

This set of features should cover the features contained in most parts. If you need to extract other features from a solid or surface model, you should use the curve from surface tools and then create your features from this geometry.

Methods of feature recognition

FeatureCAM provides four different methods of feature recognition:

Automatic feature recognition. Two different styles of feature recognition are available that attempt to recognize as many features as possible. These techniques require a solid model.

Recognition from side surfaces. With this method, a feature's shape and depth are determined solely from selected surfaces. For holes and slots this is the only applicable method.

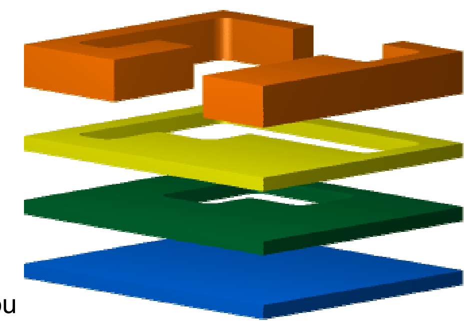
Recognition from horizontal surfaces. Recognize pockets from bottom surfaces or bosses from their top surfaces.

Recognition from feature curves. For milled features that require curves (bosses, pockets and sides) the shape of the features can be determined by chaining the curves in the plane of the current UCS. Use this technique for features that are made up of too many surfaces to conveniently pick.

Styles of automatic feature recognition

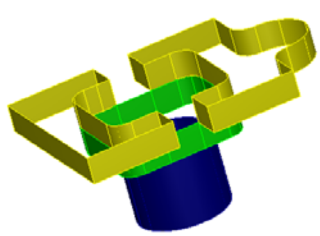
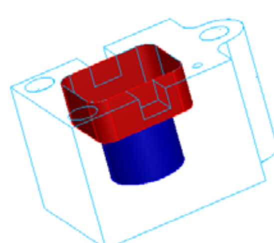
There are two styles of automatic feature recognition available in FeatureCAM. Both methods require a solid model.

1. Automatic feature recognition wizard - The automatic feature recognition wizard attempts to create all features for milling a part. It creates features by dividing the model into horizontal slices. All features of the model are recognized at once. It creates plain, counter drilled, counter bored and counter sunk holes and all milling features are created as side features. It will often create a set of features that will completely cut your solid, but it may create more features that you might create if you modeled the features yourself.



2. Automatic option of the New feature wizard - The automatic option of the new feature wizard is applicable to holes, pockets, bosses and side features. It is more limited to how much of the part can be automatically recognized and you must recognize each type of feature with separate runs of the wizard.

The following example shows the milling features that are extracted using each method. The figure on the left shows the three side features created by the automatic feature recognition wizard. The model has the advantage that the two slots are cut by the top feature. It has the disadvantage that top circular pocket is not represented by a single pocket. The image on the right is the result of automatically recognizing the pockets in the model using the automatic option of the new feature wizard. Each pocket is represented by a separate pocket, but the two slots are ignored and must be recognized separately using another technique.

Note that if automatic feature recognition does not create the features you desire, you should use an interactive feature recognition technique such as recognition from feature curves or from individual surfaces.

Invoking the automatic feature recognition wizard

1. Select the AFR  step from the Steps toolbox.
2. Follow the instructions of the wizard.

Invoking the automatic option of new feature wizard

1. Select the Feature  step from the Steps toolbox.
2. Select either the *Hole*, *Boss*, *Pocket*, or *Side* radio button.
3. Check the *Extract with FeatureRECOGNITION* check box and click *Next*.
4. Select the *Automatic recognition* radio button and follow the rest of the wizard.

Note: You must have a solid model to use this feature recognition method.

Feature rerecognition wizard

The rerecognition wizard compares a new solid model with an existing set of features. It is assumed that you have imported a new solid model that is a variation on the initial model. If are trying to recognize the initial features from a part model use automatic feature recognition.

The rerecognition wizard automatically recognizes features from the new solid and then compares the new features to the existing features and classifies the new features into the categories of unchanged, new, modified and deleted. Unchanged features are identical to existing features. They will be ignored in the rerecognition process. New features do not part of the existing features and therefore are assumed to be new. The recognition wizard will offer you the opportunity to add these features to your part. Modified features have the same shape or size as existing features, but parameters like depth, bottom radius or chamfer distance have been modified. The rerecognition wizard will ask you if want to replace the existing features with their modified versions. Deleted features exist in the current set of part features, but were not found in the new model. You are offered by the rerecognition wizard the option of deleting these features. There are many possible reasons that this feature is in your existing set of features but was not automatically recognized, so you should be careful about deleting these features. If the feature has clearly been deleted, you can comfortably remove it. If the feature was created through interactive methods, you probably want to keep this feature. If you are uncertain, you should preview each feature and decide on an individual basis.

Invoking the rerecognition wizard

1. Import another version of the solid model from which you have already created features.
2. Select *rerecognition wizard* from the Construct menu.
3. Follow the instructions of the wizard.

How to recognize features from surfaces

1. Import your solid model or IGES file.
2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z-axis.
3. Click the Feature step from the Steps tool bar or the Advanced toolbar.
4. Select your feature type. Remember only holes, slots, pockets, bosses and sides can be recognized.
5. Click *Extract feature from solid model* and click *Next*.
6. If you are creating a pocket, boss or side, you are presented with a choice of methods. Click *Select surfaces* and click *Next*.
7. Select surfaces using the following procedure:

    a. Select the surface(s) in the Graphics window. Any surfaces that were selected prior to entering the wizard are listed.

    b. Hit the button to add the surfaces to the selected list. The surfaces change to the highlight color (green by default).

    c. To remove a surface from the list, select the surface in the Graphics window or click on the name in the list and hit the button. To select a range of names from the list, click on the top name and then hold the SHIFT key down and click on the bottom name.

    d. If you want to hide the selected surfaces after creating the feature, click *Hide checked surfaces when finished*. This option can be useful to clear the screen for subsequent surface selections. These surfaces are not deleted, only hidden. They can be reshown by using the *Show all surfaces* option from the *Hide* fly-out.

    e. When you have selected the surfaces required to define the feature, click *Next*.

    f. If you receive an error, the surfaces you selected cannot be used to create the specified feature. See *Types of features that can be recognized* for requirements for recognizing specific features.

8. If you are making a feature other than a hole, the top and bottom page is displayed. The top and bottom surfaces are extracted from the surfaces so you will probably not have to adjust the top or bottom values. Just click *Next*.
9. Confirm the extracted dimensions of the feature. Modify the feature's dimensions if desired and click *Next*.
10. The Strategies page is displayed. Click the desired attributes depending on your manufacturing preferences. Click *Next*.
11. The Operations page is displayed. This page displays the operations that will be used to manufacture the feature along with the names of the selected tools and calculated feed and speed values. If these values are acceptable, click *Finish.* If you want to change the tooling or feeds and speeds click *Next* and follow the instructions on the screen.

How to recognize features from surfaces using curve chaining

1. Import your solid model or IGES file.

2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z-axis.

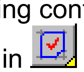
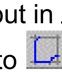
3. Click the Feature step  from the Steps tool bar or the Advanced toolbar.

4. Select your feature type. Remember only pockets, bosses and sides can be recognized using curve chaining.

5. Click *Extract feature from solid model* and click *Next*.

6. Click *Chain curves* and click *Next*.

7. Geometry is projected from the surfaces onto the plane of the UCS and chaining is enabled.

    a. The dialog bar at the bottom of the screen is changed to contain chaining controls. If you are creating a pocket or boss feature you will automatically be put in  Closed curve chaining mode. For side features you are automatically put into  Picking pieces mode.

    b. It is often easier to chain from the top view. If you want to change to this view, click the *Switch to top view* button.

    c. Chain your curve using the desired method.

    d. Click *Next*.

    e. If you receive the message “Profile curve is not closed. Curve must be closed for this feature type”. Click *OK* and re-chain the curve so that it forms a loop and click *Next*.

8. The top and bottom page is displayed. Enter the top and bottom values or click the pick buttons and graphically pick the locations. Click *Next*.

9. Confirm the extracted dimensions of the feature. Modify the feature’s dimensions if desired and click *Next*.

10. The Strategies page is displayed. Click the desired attributes depending on your manufacturing preferences. Click *Next*.

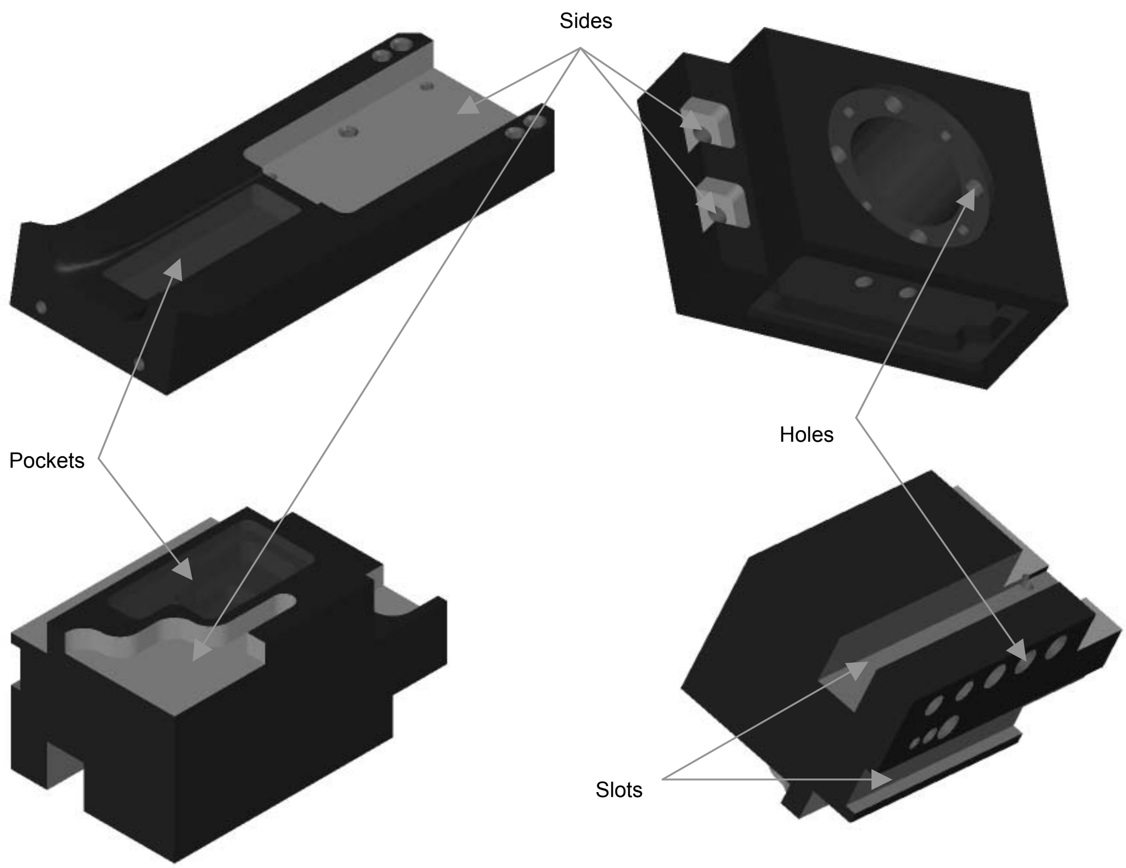
11. The Operations page is displayed. This page displays the operations that will be used to manufacture the feature along with the names of the selected tools and calculated feed and speed values. If these values are acceptable, click *Finish.* If you want to change the tooling or feeds and speeds click *Next* and follow the instructions on the screen.

Types of features that can be recognized

FeatureCAM can recognize the following feature types:

- Holes
- Slots
- Bosses
- Pockets
- Sides

Only these feature types will activate the *Extract feature from solid model* button in the Feature wizard.

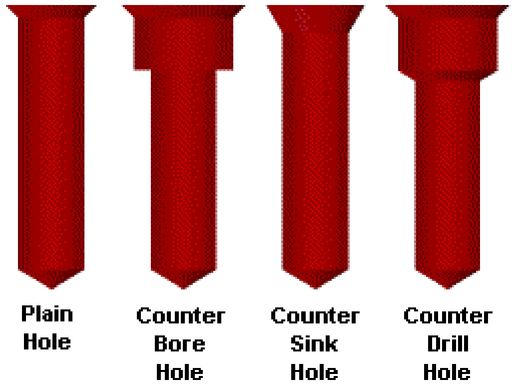


Hole recognition

Four types of holes can be recognized.

Hole type	Surfaces recognized from
Plain	Cylindrical surfaces with an optional 45° chamfer surface.
Counter bore	Two cylinders (larger diameter on top) with an optional 45°chamfer on the top.

Counter drill	A cylinder on top of a cone, on top of another cylindrical surface. A 45° chamfer is optional on top.
Counter sink	A cylinder with a cone on top that is not 45°.

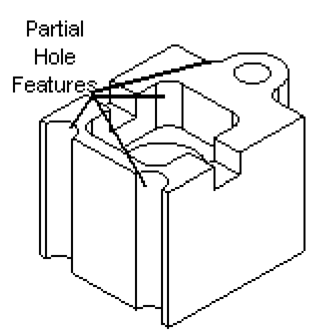


Note that the cylinders can be comprised of one or more surfaces.

To create a pattern from multiple hole surfaces use the *Make a pattern from feature* button on the New Feature dialog box. Note that the surfaces must result in identical holes. If the holes are of different types or dimensions, a pattern cannot be created.

How to recognize all holes in a setup

1. Import your solid model or IGES file.
2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z axis.
3. Click the Feature step  from the Steps tool bar or the Advanced toolbar.
4. Select the *Hole* feature type.
5. Click *Extract feature from solid model* and click *Next*.
6. Click *Recognize and construct multiple holes* and click *Next.*
7. Your next decision is whether to recognize holes that are not complete represented in the part.



   a. If you only want to recognize holes with 360 degree cross sections, leave the *Include partial feature* radio button unchecked. This option will typically recognize all the holes in your model.

   b. Partial hole features are recognized from pieces of cylinders. This includes holes that have been partially cut away or corners pockets. Click here for an example. If you would like to have holes recognized from these surfaces, click *Include partial feature*.

8. Next determine if you would like to ignore blind holes that are on the opposite of the part. Click *Ignore hidden holes* options to turn on this option. Note: this option is only available if your part model is solid model.

9. Patterns and/or groups can also be created.

   a. If holes with identical dimensions were recognized and you want to create patterns for these holes, click *Merge features of identical parameters into a pattern*. By creating a pattern, you will later be able to modify the entire pattern in a single dialog box.

   b. If you would like to create a group of all of the holes click *Add all new features into a group if there is more than one*.

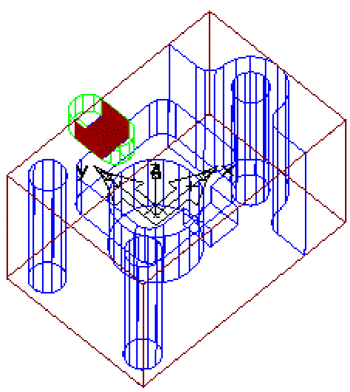
10. All of the recognized holes are highlighted in thick lines. You must select which of these holes you would like to keep.

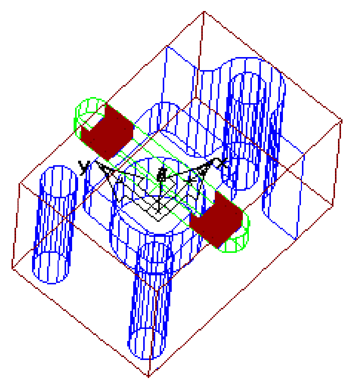
    a. If you want all of the recognized holes, click *Select all* and click *Finish*.

    b. If you want only some of the recognized holes, select the holes in the graphics window and click *Finish*.

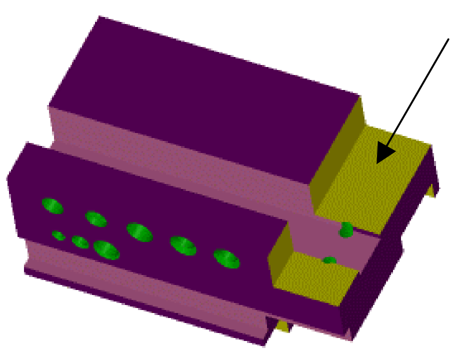
Slot recognition

Only straight slots can be recognized. They are recognized from straight, parallel side surfaces. You must select surfaces on both sides of the slot. The figure on the left is an example of a simple slot. For interrupted slots, select surfaces on each end. The right-hand figure is an example of an interrupted slot.





If the feature you are trying to recognize is a straight cut that extends off the side of the part, as shown below use a side feature. You cannot recognize such a feature as a slot, since there are not two opposing side walls.



Use a side instead of a slot for straight cuts like this with only one wall remaining in the solid model.

Note that chamfers, draft angles and bottom radii cannot be recognized, but you can add these parameters to the slot in the *Dimensions* page of the New Feature wizard.

Pocket recognition

Pockets are features that must be closed. This means that the cross-section of the feature must form a loop.

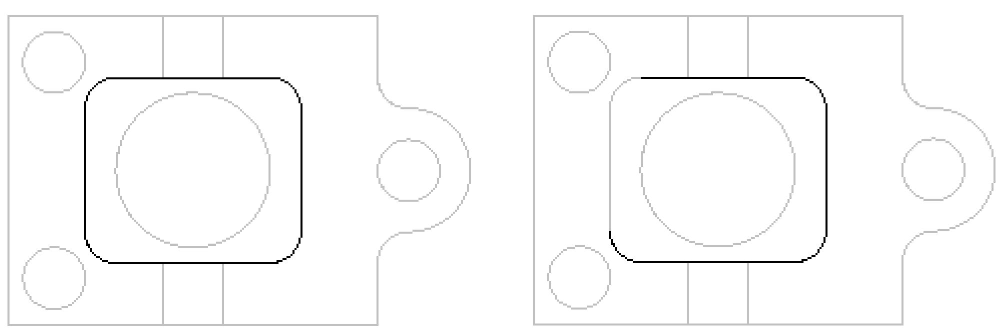
Pockets can be recognized using one of four methods.

1. Pockets with bottoms and walls all around can be automatically recognized from solid models. This is the only method that can recognize islands and bottom radaii.
2. Recognition from selected bottom surfaces.
3. Recognition from selected side surfaces
4. By extracting feature geometry and chaining curves. This is the only method that can be used to recognize drafted pockets

Pocket recognition from side surfaces

If you are recognizing the feature directly from surface data keep in mind:

1. It is often easiest to select the surfaces from the top view after clicking *Hide all nonvertical surfaces* from the Hide menu.
2. Bosses require a closed curve. Once you select your surfaces view them from the top. The selected surfaces should form a loop as shown on the right. If you don't see a loop as on the left, you must select additional surfaces to fully define the pocket.



3. For pockets there is no automatic island detection. Do not include island surfaces in your selection. Instead create your pocket without the island and then add the island separately by modifying the feature.
4. Chamfers, draft angles and bottom radii cannot be recognized, but you can add these parameters to the feature in the *Dimensions* page of the New Feature wizard.
5. If you select surfaces that define more than one cavity, a single pocket feature is created that contains multiple pocket cavities. The collection of features is then milled one z-level at a time.
6. You can recognize drafted pockets. See page 422 for more information.

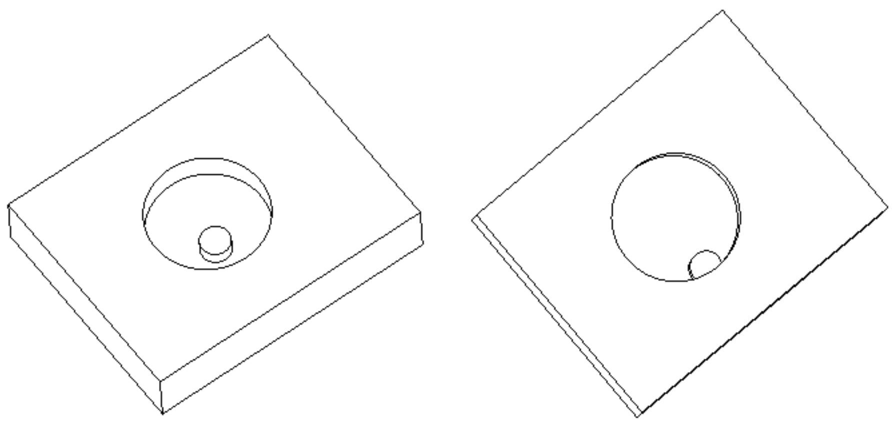
Automatic pocket recognition on solid models

If you are working with a solid model, you can recognize blind pockets automatically. For a cavity to be recognized as a pocket automatically the following conditions must be met:

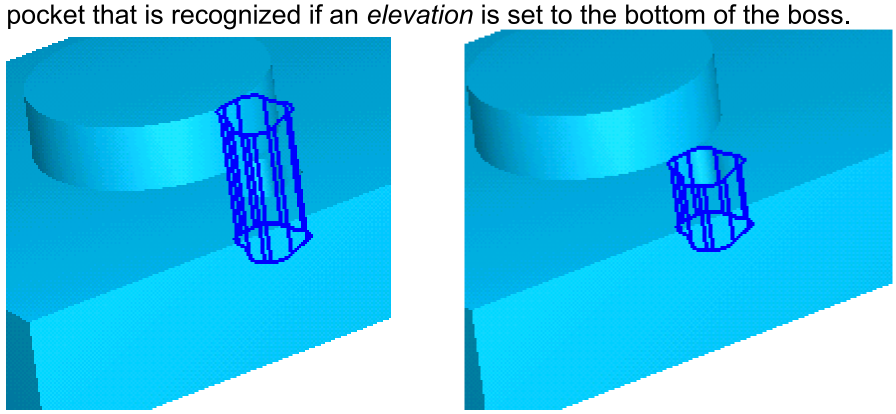
1. The pocket must have a flat floor. A cavity that passes all the way through the stock cannot be recognized.
2. The pocket must have walls all the way around it.

For cavities that meet these conditions the automatic recognition will find:

1. The pocket boundary and the island boundaries. Note that the islands must be distinct from the boundary, but they can be of different heights. The island of left-hand model will automatically be recognized, but the island of the right-hand model will not.



2. The pocket depth is determined by the tallest pocket wall. If you want to force all the features you are recognizing to start at the same Z coordinate, then click *Force same Z height* and then enter an *elevation.* If the pockets you are trying to recognize share a wall with a boss, you will want to control the depth in this way. The left-hand figure shows the pocket that is automatically recognized. The right-hand figure shows the pocket that is recognized if an *elevation* is set to the bottom of the boss.



3. If the pocket has a consistent bottom radius, it will be automatically recognized.

Chamfers are not recognized, but can be interactively added to a feature using the following procedure:

1. Turn off shading.
2. Make sure *snap to object* is set in the Snap modes dialog box.

3. Use interrogation to extract the Z distance from a line at the top of the pocket and a line at the bottom of the chamfer.

4. Enter this value as the Chamfer dimension of the part.

5. Add the Chamfer value to the feature depth.

6. Add the Chamfer value to the Z coordinate of the feature location.

How to recognize pockets and bosses from top or bottom surfaces

1. Import your solid model or IGES file.

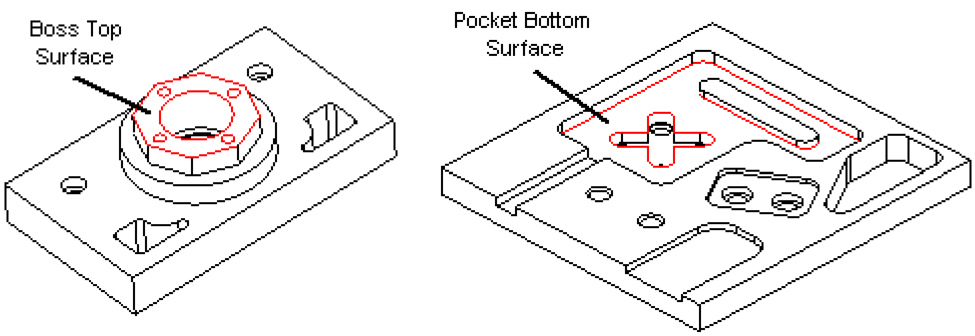
2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z axis.

3. Click the Feature step  from the Steps tool bar or the Advanced toolbar.

4. Select your either a *Boss* or *Pocket* feature type. Click *Extract with FeatureRECOGNITION* and click *Next*.

5. Click *Use horizontal surface* and click *Next*.

6. For a boss, select the flat top surface of the boss, for a pocket, select the flat bottom surface. Click *Next.*



7. The top and bottom page is displayed. Enter the top and bottom values or click the pick buttons and graphically pick the locations. Click *Next*.

8. Confirm the extracted dimensions of the feature. Modify the feature's dimensions if desired and click *Next*.

9. The Strategies page is displayed. Click the desired attributes depending on your manufacturing preferences. Click *Next*.

10. The Operations page is displayed. This page displays the operations that will be used to manufacture the feature along with the names of the selected tools and calculated feed and speed values. If these values are acceptable, click *Finish.* If you want to change the tooling or feeds and speeds click *Next* and follow the instructions on the screen.

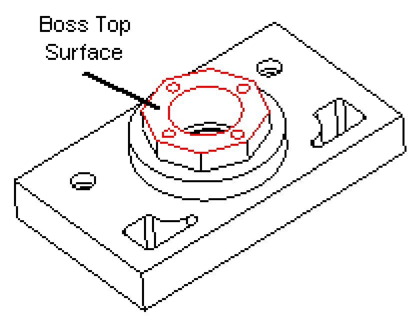
Boss recognition

Bosses are features that must be closed. This means that the cross-section of the feature must form a loop.

Bosses can be created by recognizing features directly from surfaces or by using curve chaining from the solid model.

1. If you are recognizing the feature directly from surface data keep in mind:
2. It is often easiest to select the surfaces from the top view after clicking *Hide all nonvertical surfaces* from the Hide menu.
3. Bosses require a closed curve. Once you select your surfaces view them from the top. The selected surfaces should form a loop. If you don't see a loop, you must select additional surfaces to fully define the pocket.
4. Chamfers, draft angles and bottom radii cannot be recognized, but you can add these parameters to the feature in the *Dimensions* page of the New Feature wizard.
5. If the part has multiple bosses at the same height, you should select all the surfaces of each boss and create a single boss feature. The collection of features is then milled at one z-level at a time. If you accidentally create more than one boss at the same height, the first boss will cut away the second boss.
6. You can recognize drafted bosses.

How to recognize pockets and bosses from top or bottom surfaces



1. Import your solid model or IGES file.
2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z axis.
3. Click the Feature step from the Steps tool bar or the Advanced toolbar.
4. Select your either a *Boss* or *Pocket* feature type. Click *Extract feature from solid model* and click *Next*.
5. Click *Use horizontal surface* and click *Next*.
6. For a boss, select the flat top surface of the boss, for a pocket, select the flat bottom surface. Click *Next.*
7. The top and bottom page is displayed. Enter the top and bottom values or click the pick buttons and graphically pick the locations. Click *Next*.
8. Confirm the extracted dimensions of the feature. Modify the feature's dimensions if desired and click *Next*.
9. The Strategies page is displayed. Click the desired attributes depending on your manufacturing preferences. Click *Next*.
10. The Operations page is displayed. This page displays the operations that will be used to manufacture the feature along with the names of the selected tools and calculated feed and speed values. If these values are acceptable, click *Finish.* If you want to change the tooling or feeds and speeds click *Next* and follow the instructions on the screen.

Side recognition

Sides are features can be closed or open, but in feature recognition it is assumed that you want to create an open side feature. Use a boss or pocket for closed features.

If you are recognizing the feature directly from surface data keep in mind:

1. It is often easiest to select the surfaces from the top view after clicking *Hide all nonvertical surfaces* from the Hide menu.

2. Chamfers, draft angles and bottom radii cannot be recognized, but you can add these parameters to the feature in the *Dimensions* page of the New Feature wizard.

How to recognize drafted features

1. Import your solid model or IGES file.

2. If necessary, transform the part so that the surfaces that represent the features are facing toward the Z axis.

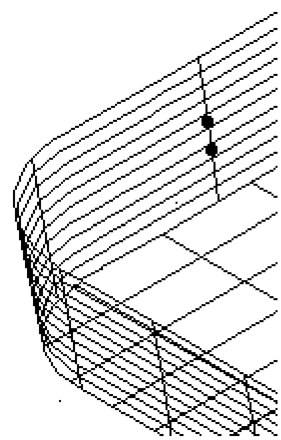
3. Click the Feature step from the Steps tool bar or the Advanced toolbar.

4. Select your feature type. Remember only pockets, bosses and sides can be recognized using curve chaining.

5. Click *Extract feature from solid model* and click *Next*.

6. Click *Chain curves*. Wall angle and elevation values are shown.

7. If you know the angle, type it. If not click on the *Draft angle* label. The dialog box will warp out of the way. Click two points on the same vertical isoline as shown in the figure. The dialog box returns.



8. For the *Elevation* enter the Z coordinate of the top of the feature or click on the *Elevation* label and click on the top of a wall of the drafted surface. Click *Next*.

9. The geometry for the features is projected onto the plane of the UCS. Chain the appropriate geometry.

10. The top and bottom page is displayed. Enter the top and bottom values or click the pick buttons and graphically pick the locations. Click *Next*. For more information see

11. Confirm the extracted dimensions of the feature. Modify the feature's dimensions if desired and click *Next*.

12. The Strategies page is displayed. Click the desired attributes depending on your manufacturing preferences. Click *Next*.

13. The Operations page is displayed. This page displays the operations that will be used to manufacture the feature along with the names of the selected tools and calculated feed and speed values. If these values are acceptable, click *Finish.* If you want to change the tooling or feeds and speeds click *Next* and follow the instructions on the screen.

How to recognized indexed or turn/mill features

You can recognize milling features for indexed parts or turn/mill documents using the folling steps:

1. On the Indexing tab of the stock properties dialog box, make sure that *4th axis indexing* is selected.

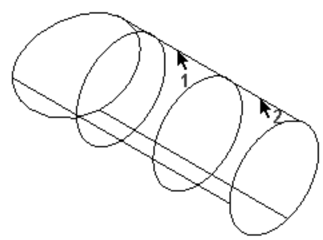
2. Click the Feature step  from the Steps tool bar or the Advanced toolbar.

3. Select your feature type. Remember only pockets, bosses and sides and holes can be recognized.

4. Click *Extract feature from solid model* and click *Next*.

5. On the New feature – feature alignment page you have two choices for the plane of the feature. Select *Along the setup Z axis* to recognize features inn the XY plane of the current setup axis.

6. Select *Around index axis* to recognize features that require indexing. You are then prompted to enter the index angle. The index angle is relative to the setup axis, just as it is with any feature alignment. The actual angle that is output in the code is relative to the stock axis. If you want to graphically enter this angle, click on the *Index angle* hyperlink label in the dialog box and then click twice on the feature you are trying to recognize. Click once near the bottom of the feature and then at the top.



7. Complete the rest of the new feature wizard.

Feature recognition surface requirements

The following requirements apply for the recognition of surfaces for milling features.

1. Only straight walled-surfaces are recognized. If a single surface also contains the bottom radius or a chamfer it will not be recognized.

2. Tapered surfaces are not recognized.

3. The current setup must be oriented so that the vertical walls of feature surfaces are parallel to the Z-axis.

4. You can include surfaces other than straight walls and they will be used to calculate the depth of the feature. These surfaces will not be used in determining the feature shape.

Chapter 24
Tombstone Machining

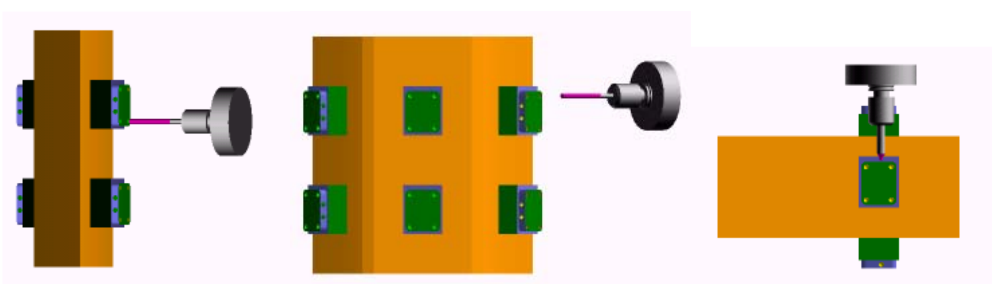
If your milling machine is equipped with a tombstone, FeatureCAM makes it easy to take advantage of this high production feature.

Note that you must license the *Tombstone Option* to use tombstone machining.

Overview of tombstone machining

The tombstone fixture document helps you arrange various FeatureCAM milling parts on a tombstone for production manufacturing. Your first step is to describe your tombstone. Tombstones for both vertical and horizontal milling machines are supported. Input the number of faces and the dimensions of a face and these dimensions are remembered for future parts. Next use the tombstone wizard to open FeatureCAM parts and locate them on the tombstone. Many options are available for fixture offsets. You can specify a unique fixture offset for each part setup so that each setup is located individually or place the parts relative to a global fixture offset to minimize the number of fixture offsets you use. You can cut multiple copies of a single part or mix different parts on the tombstone. Operations can be automatically ordered to minimize tool changes across all setups or to cut each setup completely before rotating.

The following figures show a horizontal tombstone with two faces, a horizontal tombstone with six faces and a vertical tombstone with 4 faces.



Creating a tombstone machined part

1. Select *New* from the File menu or select the *New*  button from the Standard toolbar.
2. Select *Tombstone Fixture* as the type.
3. Select the system units you'll use for the multiple part document. This is the unit you will use to specify the size of the tombstone and to position the parts on the faces of the tombstone. It specifies the system of units for all parts that you will place on the tombstone.
4. Click *OK*.
5. If it is the first time you have created a part for tombstone machining, your next step is to specify the dimensions of the tombstone.

6. If it is not your first tombstone machined part, your next step is to create a fixture coordinate system.

Specifying tombstone dimensions

It is important to accurately reflect the dimensions of the tombstone because these dimensions are used for simulating toolpaths and to calculate the required retract distances.

The tombstone dimension dialog box comes up automatically the first time you open a tombstone part. It is assumed that you will not have to change these dimensions again. If you need to alter the dimensions, click the *Tombstone* step in the Steps toolbox.

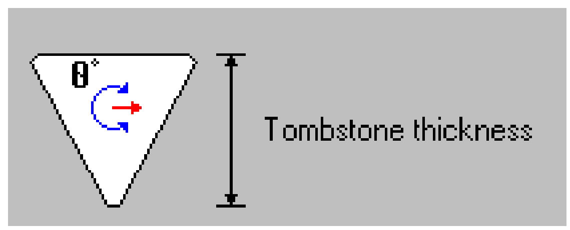
To set the dimensions of the tombstone:

1. Select the tombstone axis of rotation. For horizontal machining centers, this is normally the Y-axis. For vertical machining centers, this is normally the X-axis.

2. Specify the number of faces of the tombstone.

3. Specify the *Axis Length*. This is the dimension of the tombstone parallel with the tombstone axis.

4. Specify the *Width of a Face*. This is the dimension that is perpendicular to the *Axis Length*.



5. Specify the final dimension, the *Tombstone Thickness*. For a tombstone with an even number of faces, this is the distance between parallel faces. For a tombstone with an odd number of faces it is the perpendicular distance from a face to the joint of faces on the opposite side of the tombstone.

6. Click *OK* when finished.

Creating global fixture coordinate systems on the tombstone

The first step to placing parts on the tombstone is to locate a point on the tombstone to position the primary setups of the parts. These coordinate systems are created relative to the tombstone and are called global fixture coordinate systems.

1. Bring up the *Tombstone process plan* dialog box by clicking on the Properties button.

2. Click the *Fixtures …* button. The *Fixture offset locations* dialog box comes up.

3. Click the *Add fixture* button. The *Fixture location* dialog box comes up.

4. Click the *Create the fixture origin relative to one of the faces of the tombstone* and click *Next*.

5. The *Fixture ID* dialog box comes up. Specify the *fixture ID* to use for the first face.

6. If you want to set the origin of each face at the same point, click *Use this fixture ID on each face* and click *Next*.

7. Click *Increment the fixture ID for each face* if you want to use a different origin for each face and click *Next*.

8. The *Fixture zero location* dialog box comes up. Enter the offsets from the left edge, top edge and tombstone face. Note that these values are for simulation purposes only. When you set up the machine you can locate the origin anywhere on the face that you want.

9. Click the *Finish* button.

10. The fixture offsets you create are displayed on each face of the tombstone with the symbol.

11. The *Create fixture* dialog box is displayed showing the fixture IDs you created. The faces that they apply to are shown underneath the fixture IDs. Click *OK* to accept these fixture IDs.

Adding a part to the tombstone

1. It is usually a good idea to have the .fm file already opened in FeatureCAM before adding it to the tombstone document. This is not required, but it is often convenient to have the file (or files) open in case you need to edit the file since you cannot edit the contents of FeatureMILL or FeatureMILL3D files in a tombstone document.

2. Bring up the tombstone dialog box by clicking on the Properties button or click the *Parts* button in the Steps toolbox.

3. Click the *Add* button to place a part on one face of the tombstone and bring up the select part dialog box.

4. Click on the *Add to all* to position a part on all faces of the tombstone and bring up the select part dialog box.

5. Click *Next*. The *Select part* dialog box comes up.

Tombstone select part dialog box

In this dialog you select the part you will place on the tombstone.

1. All open part files are listed on the left-hand side of the dialog.

2. If you want to place one of the open parts, click on the part file and click *Next* to display the *Primary setup* dialog box.

3. If you want add a part from another file, click the *Browse* button and locate the part file and click *OK* and click *Next* to display the *Primary setup* dialog box.

Primary setup dialog box

1. All of the setups that are contained in your .fm file are listed.

2. Click on the one you want to place parallel to the tombstone face. This will be the primary setup for this part.

3. Click *Next*. The *Active setups* dialog box comes up.

Active setups dialog box

1. Uncheck any setups that you do not want to mill.

2. Click *Next* to display the *Select fixture* dialog box.

Select fixture dialog box

FeatureCAM suggests fixture IDs to use for each setup of your parts. It will try and use a global fixture ID to minimize the number of fixture IDs required to cut your part. If it cannot find a global fixture ID that is appropriate, it will suggest unique fixture IDs for a setup. This implies that you will locate each of these setups individually. If you want to modify fixture IDs for the setups follow one of the two procedures listed below.

To change a setup to be located relative to a global fixture:

1. Click on the setup in the list.

2. Select the global fixture ID from the drop down list.

3. Click on the *Use a global fixture ID* radio button.

To change assign a fixture ID to each setup:

1. Click *Create a new fixture zero at the setup origin.*

2. Enter a fixture ID number to use for the first setup.

Click *Next* to display the *Part location* dialog box.

Part location dialog box

If you are positioning the part relative to a global fixture ID, **these numbers must correspond to how the part is setup on the tombstone.**

In this dialog box you enter the offsets from the fixture location to the part's primary setup.

1. Enter the X, Y and Z offsets from the global fixture ID.

2. If you want to use this location as the fixture offset, enter zeros for each value.

3. Click *Next*.

If you are creating fixture IDs for each setup, then **these values are used for simulation only**.

1. Enter the offsets from the tombstone edge.

2. Click Next.

Preview dialog box

This dialog is the final of the tombstone add part wizard.

1. The parts are laid out on the tombstone in the graphics window.

2. If they like they are properly positioned, click the *Finish* button.

3. To edit some of your earlier entries, click the *Back* button.

4. To cancel the placement of this part on the tombstone, click the *Cancel* button.

Creating global fixture coordinate systems from setups on placed parts

If you want to use a setup of a part that you have placed on the tombstone to locate other part setups, use the following procedure:

1. Bring up the tombstone dialog box by clicking on the Properties button or clicking the *Parts* step in the Steps Toolbox.
2. Click the *Fixtures …* button. The *Fixture offset locations* dialog box comes up.
3. Click the *Add fixture* button. The *Fixture location* dialog box comes up.
4. Click the *Create the fixture origin at the origin of a setup* and click *Next*.
5. The *Select fixture* setup dialog box comes up. Select the setup whose origin you want to use and click *Next*.
6. The *Fixture ID* dialog box comes up. Specify the *fixture ID* to use for the first face. You will probably not have to change the default value since it was set when you placed the first part.
7. If you want to set the origin of each face at the same point, click *Use this fixture ID on each face*.
8. Click *Increment the fixture ID for each face* if you want to use a different origin for each face.
9. Click the *Finish* button.
10. The fixture offsets you create are displayed on each face of the tombstone with the  symbol.
11. The *Create fixture* dialog box is displayed showing the fixture IDs you created. The faces that they apply to are shown underneath the fixture IDs. Click *OK* to accept these fixture IDs.

Specifying ordering of tombstone operations

1. Bring up the tombstone dialog box by clicking on the Properties button.
2. If you want to minimize tool changes across all setups, click *Tool Dominant*. This ordering option also tries to minimize the rotations of the table. Note that you must also set the *Minimize tool changes* automatic ordering option for *Tool Dominant* to work correctly.
3. If you want to complete each setup before moving on to another setup, click *Setup Dominant*. Remember, this refers to the setups in the .fm files, not the fixture ID that you specify in the tombstone document. The setups are cut in the order that they are added to the tombstone. If you want to cut in a setup dominant fashion and want tight control in the setup order, you may want to add each part to the tombstone individually. The order that operations are performed within a setup are determined by the milling ordering attributes.

How the ordering specified in the Op list tab affects tombstone operation ordering

If you specify *Setup dominant* as the tombstone ordering option, the automatic ordering options (specified in the Op list tab) are applied to each setup. For example if you have *Setup dominant* set and set *Minimize tool changes*, then each setup is cut separately and tool changes are minimized within the setup.

If you specify *Tool dominant* as the tombstone ordering option, then the automatic ordering options are applied across all the faces of the tombstone. For example, specifying *Finish cuts last*, an endmill will cut all the roughing passes across the tombstone and then all the finish passes.

Tombstone delete button

This button allows you to delete a face of the tombstone or delete a part from a tombstone face.

To delete a face:

1. Click on a face name. Faces have names like *0 degrees, 90 degrees*.
2. Click the *Delete* button.

To delete a part from a face:

1. Click on a part file name.
2. Click the *Delete* button.

Tombstone reload button

This button allows you to reload a FeatureCAM file into your tombstone document. If your initial part file has changed since you created your tombstone part, you'll want to reload the part. If you have the .fm file loaded into FeatureCAM at the same time as your tombstone document, it is best to save the .fm file and then reload it into the tombstone document.

To reload a part

1. Click on a part filename.
2. Click the *Reload* button.

Tombstone edit button

The *Edit* button allows you to edit tombstone information.

To edit the location of a part on a face:

1. Select the filename.
2. Click *Edit*.
3. The Tombstone part location dialog box comes up for editing.

To edit a setup name or fixture ID:

1. Click on a setup.

2. Click the *Edit* button.

3. The Tombstone setup information dialog box comes up.

Tombstone setup information dialog box

This dialog box allows you to change the fixture ID or name of the setup. The fixture ID is automatically assigned in the adding a part to the tombstone process.

1. To change the name, enter a new *Setup name*.

2. To alter the *Fixture ID*, type in a new number.

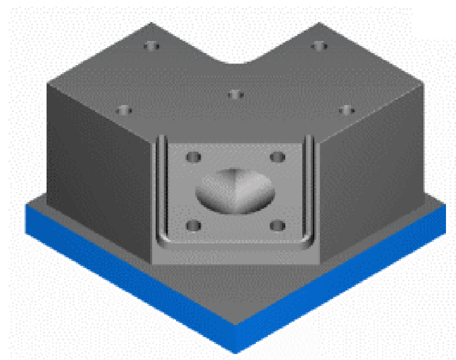
Chapter 25
Five Axis Positioning

If you have a 5-axis machine, FeatureCAM makes it easy minimize the fixturing of these parts.

Note that you must license the *5-axis positioning* option to use five axis positioning.

Overview of 5-axis positioning

5-axis positioning provides a convenient method of manufacturing parts that require milling on multiple faces by minimizing setups. The figure on the right shows an example of a part that requires milling from 4 different orientations. With 5-axis positioning, this entire part can be milled with a single program.



FeatureCAM provides two types of 5-axis positioning NC programs. The first method requires the operator to set only a single coordinate system. The entire program is then generated with regard to this coordinate system. The advantage of this method is that it minimizes the setup time. The disadvantage is that the resulting NC programs are harder to read since the coordinates for milling each face are rotated.

The second method uses standard fixture offsets to determine the coordinate system for each face. The advantage of this method is that the code for each face is easier to read. The disadvantage is that the operator must touch-off each face.

5-axis positioning using a single coordinate system

This method minimizes setup time by only requiring a single touch off point and is performed with the following steps:

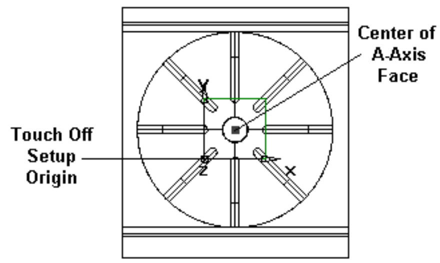
1. Create a part with multiple setups and either:

    - Click through to the third page of the stock wizard, the Multi-axis Positioning page. Click *5th Axis Positioning* and click *Next*; or

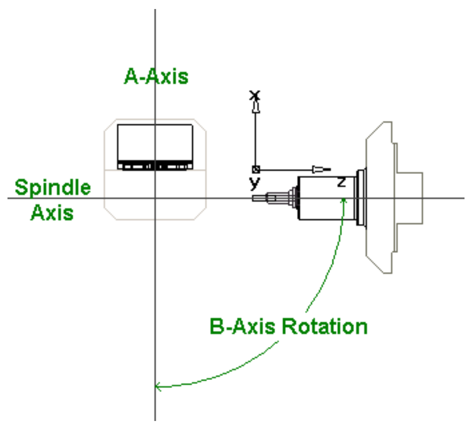
    - Double click on the stock and click on the *Indexing* tab and click *5th Axis Positioning*.

2. Designate one of your setups as the 5-Axis Touch-off Point. The Z-axis of this setup must be parallel with the A-axis.

3. For the *5-Axis Touch-off Setup Offset*, enter the distance, in each coordinate system direction, from the center of the A-axis face to the 5-Axis Touch-off Point. This distance will change for each part. In this example, the X and Y offsets are negative and the Z offset is positive.



4. For the *B-Axis Rotational Offset*, enter the angle (measured counter-clockwise) between the spindle and the A-axis when the B angle is set to 0. For example, the A-axis faces the spindle when B is set to 0, then enter 0. If it faces the door when B is set to 0, as shown in the figure below, then enter -90. This offset will be set the same for all parts machines on a specific machine.



5. If you want the ordering of operations to be tool dominant across all setups, click *Tool Dominant*. Note that you must also set the *Minimize tool changes* ordering attribute for Tool Dominant to work correctly.

6. If you want the ordering of operations to complete each setup before moving on to another setup, click *Setup Dominant*. The order that operations are performed within a setup are determined by the milling ordering attributes. Click *Generate Single Program* to create a single 5-axis indexed program.

7. Click *Finish* in the stock wizard or *OK* in the stock properties dialog.

8. Generate toolpaths. Don't forget to check *Minimize tool changes*, if you want Tool Dominant toolpaths.

9. The simulation works for both centerline and 3D simulation. In centerline simulation an arc is displayed for a 5-axis reorientation. In 3D simulation the part position and orientation stays fixed and the tool moves. The tool does not move smoothly between setups, it simply reappears in the new setup.

10. Select a 5-axis post processor and create NC code.

11. The NC code is generated with respect to the b-axis coordinate system. Only a single touch-off is required to locate the part.

12. The Rotary Center Offset values are contained in the post processor files. See page **Error! Bookmark not defined.** for more information.

5-axis positioning using fixture offsets

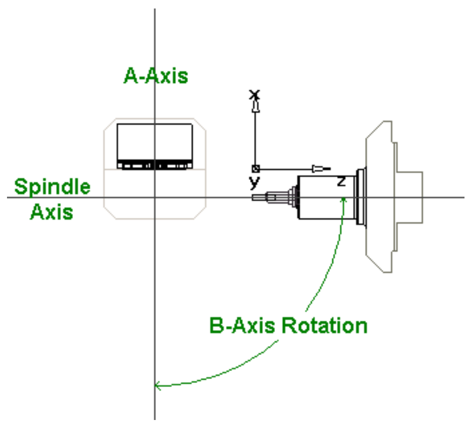
1. Create a part with multiple setups and either

   - Click through to the third page of the stock wizard, the Multi-axis Positioning page. Click *5th Axis Positioning* and click *Next*; or

   - Double click on the stock and click on the *Indexing* tab and click *5th Axis Positioning*.

2. Click *Use Fixture Offset*. The 5-Axis Touch-off Setup and 5-Axis Touch-off Setup Offset do not apply to this method.

3. For the *B-Axis Rotational Offset*, enter the angle (measured counter-clockwise) between the spindle and the A-axis when the B angle is set to 0. For example, the A-axis faces the spindle when B is set to 0, then enter 0. If it faces the door when B is set to 0, as shown in the figure below, then enter -90. This offset will be set the same for all parts machines on a specific machine.



4. If you want the ordering of operations to be tool dominant across all setups, click *Tool Dominant*. Note that you must also set the *Minimize tool changes* ordering attribute for Tool Dominant to work correctly. If you want the ordering of operations to complete each setup before moving on to another setup, click *Setup Dominant*. The order that operations are performed within a setup are determined by the milling ordering attributes. Click *Generate Single Program* to create a single 5-axis indexed program.

5. Click *Finish* in the stock wizard.

6. Generate toolpaths. Don't forget to check *Minimize tool changes*, if you want Tool Dominant toolpaths.

7. The simulation works for both centerline and 3D simulation. In centerline simulation an arc is displayed for a 5-axis reorientation. In 3D simulation the part position and orientation stays fixed and the tool moves. The tool does not move smoothly between setups, it simply reappears in the new setup.

8. Select a post processor that supports fixture offsets and create NC code.

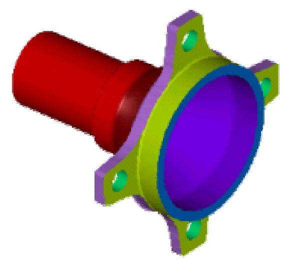
9. The NC code is generated with respect to the coordinate system of each setup. If it is desired to see the NC code for each setup specified in machining coordinates (Y up and X to the right), then each setup should be positioned so that the setup coordinate system will be aligned with the machine coordinates when that face is pointing toward the tool. The operator must touch off each setup individually.

Chapter 26
Turn/milling

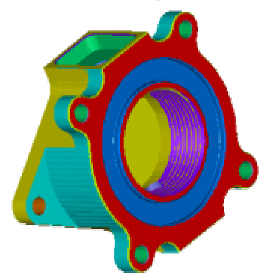
Turn/mill Overview

FeatureTURNMILL allows the combination of turning and milling features on lathes with powered rotary tools. FeatureTURNMILL supports the normal Z and X-axes of turning combined with the C and optional Y-axis.

FeatureTURN/MILL makes it easy to create parts on these lathes. Position and orient the features using custom wizards and FeatureTURN/MILL takes care of the details of creating the appropriate toolpaths and NC code.



The Yoke shown in the upper figure, could be manufactured with a C-axis lathe in two setups or using a C axis lathe with a subspindle. The lower figure is a piece that requires a Y-axis due to the flat pockets on the top and bottom of the piece. (If these pockets were wrapped, they would have a curved bottom and then they could be manufactured without a Y-axis.)



When creating features, you are given the choice of creating turning features or turn/mill features. Turning and turn/mill features can be mixed in a single setup. Turning features are identical to those on a 2-axis lathe and milling features are created the same as for a 3-axis mill, except that you are given new choices for positioning and orienting the features either on the OD or on the face of the part.

Tool selection in FeatureTURNMILL is very similar to FeatureMILL and FeatureTURN, except that rotary tools are automatically classified as either parallel to the X-axis or Z-axis. All forms of simulation (Centerline, 2D and 3D solid) are supported. In 3D simulation the rotation of the part is accurately simulated.

Turnmill parts require specially written posts. Two axis turning posts or milling posts will not work correctly.

Beginning a turnmill part

To begin a new turn/mill part:

1. Click the *New* button.
2. Select *Turnmill* as the type.

Features appropriate for turn/mill

Milling features can be performed in turnmill with the following considerations:

1. **Milled features on the Z face** - FeatureTURNMILL can make any feature on the face of a part by using only XZC moves, for machines that don't have a Y-axis. If you

want to use Y-axis on the face of a feature, you must check the *Cut using Y-axis coordinates* checkbox on the Dimensions page of the feature's Properties dialog box.

2. **Drilled features on the Z face or OD** can be created without any restrictions.

3. **Unwrapped milled features on the OD** - These features are output in X, Y and Z moves. If your machine does not have a Y axis, the only features you can cut on the OD (without wrapping) is a simple slot whose length is aligned with the Z axis. If your machine has Y-axis capabilities, you can cut the full set of milling features on the OD.

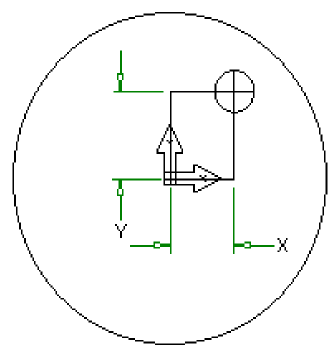
4. **Wrapped features** are supported (using a live tool), with known limitations -- the same limitations that FeatureCAM has with wrapped 4th-axis features. To invoke wrapping, you must check the *Wrap feature around Z-axis* checkbox on the Dimensions page of the feature's Properties dialog box

Creating turn/mill features

Create turn/mill features by using the Feature  Step in the Steps toolbar. The initial dialog that is displayed is the New Feature – Type dialog box. This dialog box only appears for turn/mill parts. Turnmill features are classified as either *Turnmill* or pure *Turning* features Turnmill features assume powered tools and turning features assume that the tool will not spin. For example, you can make a hole along the Z-axis with both feature types, but the milled one will use rotary tools and the turned hole, will use a drill that does not spin.

Once you complete this page the New Feature wizard continues as it normally would for a turning or milling feature.

New Feature- Location dialog box



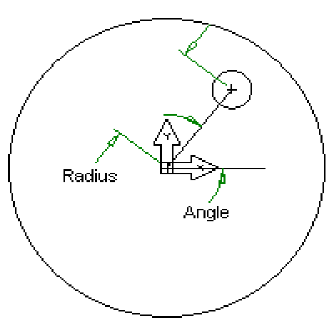
XYZ on the face

The Location page of the New Feature wizard is different for turn/mill features. The location of a turn/mill feature is specified as one of the three methods listed below

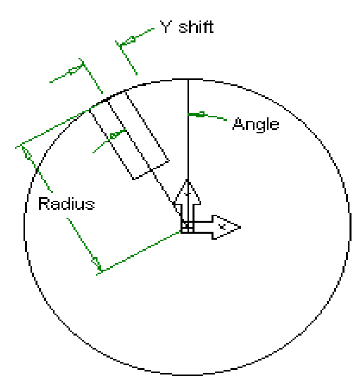
1. XYZ on the face: The feature is aligned so that its depth is aligned with the –Z direction of the setup. It is positioned by giving the X, Y and Z coordinates in the plane of the setup.

2. Polar on the face: The depth of each feature is aligned with the –Z direction of the setup. It is positioned by giving a radius and angle.

3. Radial about the X-axis OD: The *Radius* specifies the distance of the feature from the Z axis. The *Angle* is the counter clock-wise angle, in degrees, from the X-axis. The *Y distance* is distance the feature is translated from the radius. The Z coordinate is the distance the feature is translated in the Z direction. This option is only available for 4^{th} axis indexing or turn/mill documents.



Polar on the face



Radial about the X-

Tool selection for Turnmill features

Turn/mill features use the same tools as the normal milling features, but they are renamed with "-rotaryX" appended to the name to indicate that it is a powered rotary tool. For example, if a tool called "center_4" is selected for a turn/mill center drill operation, the tool is copied and the copy is called "center4-rotaryX". Rotary tools cannot be explicitly created, but if you manually select a tool for a turn/mill operation, it is copied and the copy is designated as a rotary tool. Rotary tools also cannot be used for turning operations

Feed rates for turn/mill features

All feed rates are specified in inches per minute (or millimeters per minute).

Polar interpolation in turn/mill

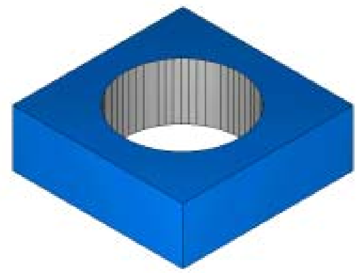
Polar interpolation can be performed by FeatureCAM or at the machine tool. This is controlled by the *Polar interpolation done by* variable in the post processor's *General* tab.

Chapter 27
Wire EDM

FeatureWIRE feature types

2-axis Die feature

The die feature requires one or more closed curves. It is assumed that the region(s) outside of the curve(s) is the part that you will keep. As a consequence, the wire travels on the inside of the curve(s).



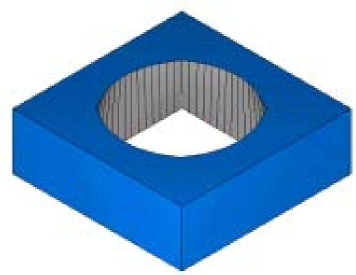
The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup.

A 2-axis die feature can create a number of cutting operations. These are specified on the Strategy tab.

4-axis Die feature (Wire)

The die feature requires two closed curves. It is assumed that the region outside of the curve is the part that you will keep. As a consequence, the wire travels on the inside of the curves.

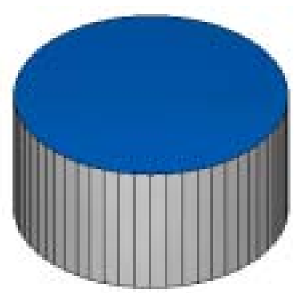


The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup.

2-axis Punch feature (Wire)

The punch feature requires one or more closed curves. The wire travels on the outside of the curve(s).

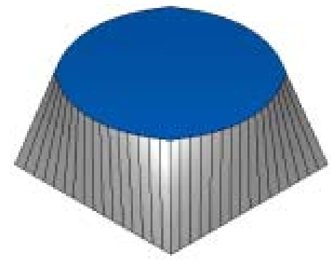


The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup.

A 2-axis punch feature can create a number of cutting operations. These are specified on the Strategy tab.

4-axis Punch feature (Wire)

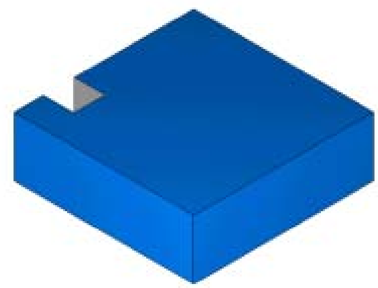


The punch feature requires two closed curves. The wire travels on the outside of the curves.

The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup.

2-axis Side feature (Wire)



The side feature can utilize one or more closed or open curves. The Punch or Die feature provides more cutting options for closed curves. You can optionally place the wire on either side of the curves. A side feature must have at least three arcs or lines in it including the lead moves.

The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup.

A 2-axis side feature can create a number of cutting operations. These are specified on the Strategy tab.

4-axis Side feature (Wire)

The side feature can utilize a closed or open curve. The Punch or Die feature provides more cutting options for closed curves. You can optionally place the wire on either side of the curve. A side feature must have at least three arcs or lines in it including the lead moves.

The *thickness* parameter is used to access the appropriate cutting data table.

The *A* parameter is used to rotate the feature around the Z-axis of the current setup

Side feature restrictions

Side features must have at least three moves in it including the lead moves. If it does not, the no feature will be displayed in the start page is displayed in the wizard. For example, if you are creating a feature from a single line, you must change the start point and end point so that the feature has three moves. If you add these moves and click the *Next* button in the wizard, the feature will display.

Multiple curves in a single 2-axis wire EDM feature

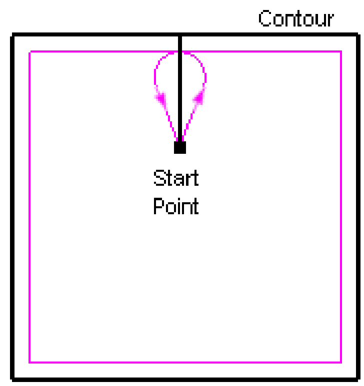
All wire EDM curves must lie in the XY plane (or a plane parallel to the XY plane.

A single 2-axis die, punch or side feature can contain multiple curves. In this case you will have the option of setting separate start points and variable taper values for each curve, but all other settings will apply to all of the curves.

If the feature has multiple operations, such as a retract followed by a cutoff, each operation will be performed on each curve before moving on to the next operation. If you wish to change this ordering, you must manually reorder the operations in the Operations List.

Overview of EDM operations

FEATUREWIRE offers five different machining cycles for different purposes. Every operation type has a whole range of parameters and options.



Contouring operation

Available for both 2-axis and 4-axis features.

This cycle is the generally applicable cycle for cutting contours.

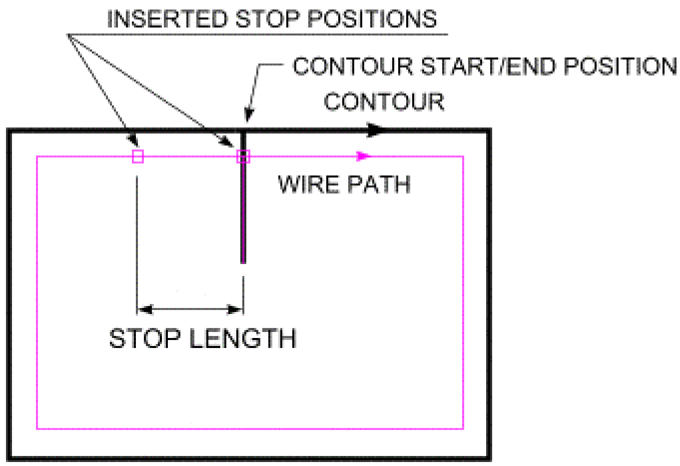
General Rules

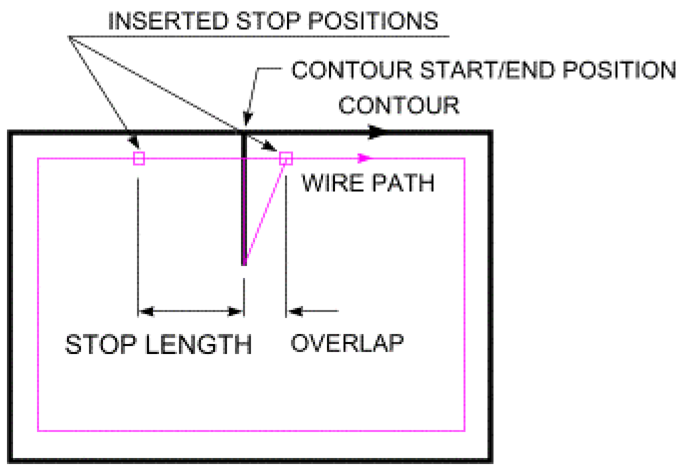
The curve can be open or closed. A contour operation traces the entire length of the curve. The number of passes is determined by Contour passes. A contour operation can be optionally added to a retract, stop, pocket, zig-zag or cutoff operation by checking the *contour* checkbox in the strategy tab.

Stop operation

Available only for 2-axis features.

A stop command is inserted into the wire path together with a further stop at the end of the contour. The distance of the new stop position from the contour end point is defined in the **Stop Length** field. If you wish to overcut the contour end position you can also define an **Overlap**. In centerline simulation, the locations where the wire stops are shown as small squares. A stop operation will have two different locations where the wire will stop.

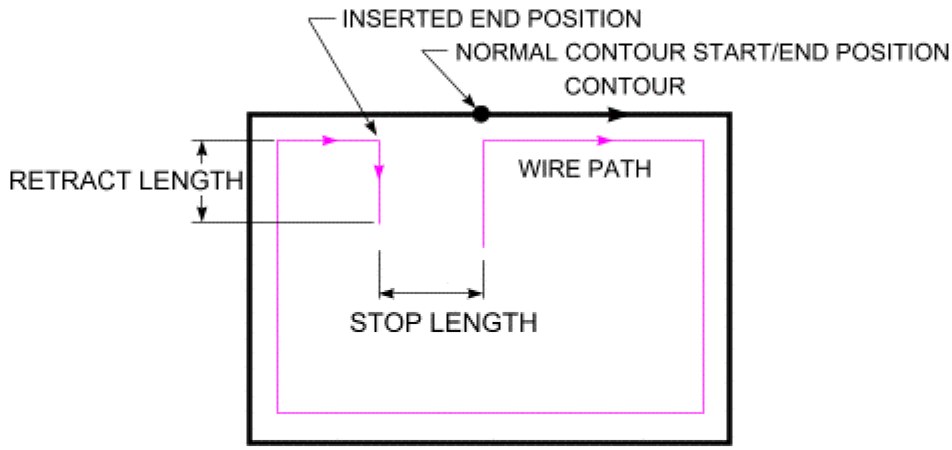




Retract operation

Available only for 2-axis features.

The contour is not cut completely but is stopped shortly before the contour end point. The distance of this position from the contour end point is set in the **Stop Length** field. This option is generally used when cutting multiple contours whereby the cut part should not be separated from the work piece - e.g. when the program should run automatically overnight..



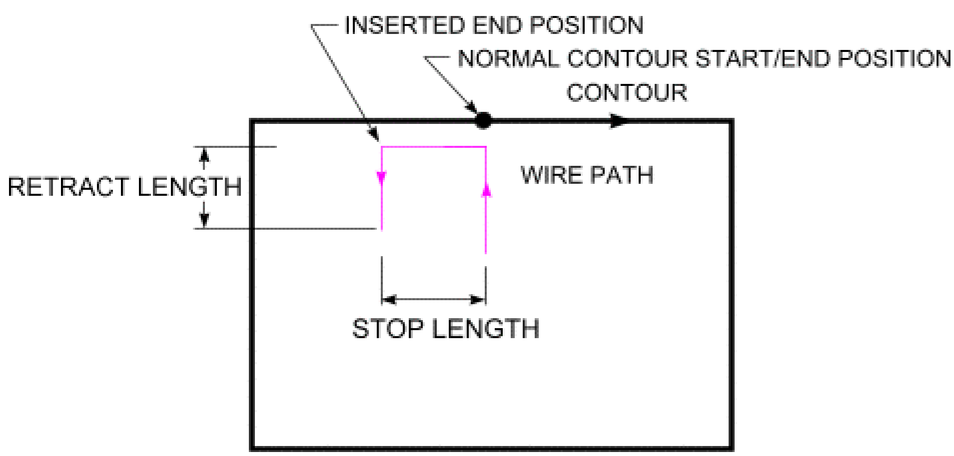
A cutoff operation is normally used after this operation to cut the part of the contour that is not cut by the Retract operation. **Cut Off** machines the left over part in the opposite direction to the machining curve.

Cutoff operation

Available only for 2-axis features.

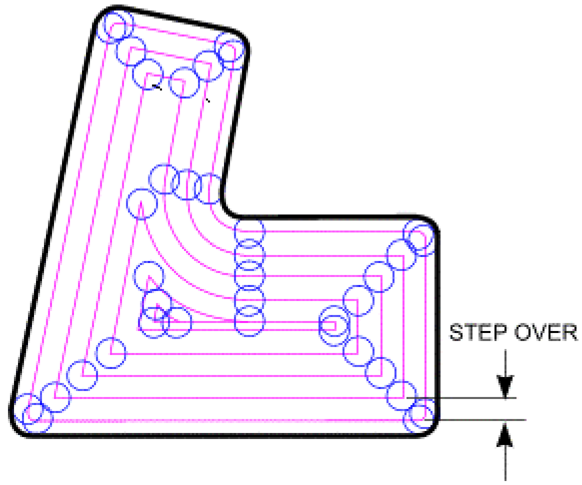
This option is normally used after a contour has been cut with the Auto Path option **Retract**.

Cut Off cuts the part of the contour that is left by the **Retract** option. The machining takes place in the reverse direction to that of the machining curve i.e. from the contour end point to the stop position. The wire will stop prior to pulling away from the part. This stop location will be shown as a small square.



Pocketing Operation

This cycle enables contour parallel area clearance of a closed curve. Available only for 2-axis features.



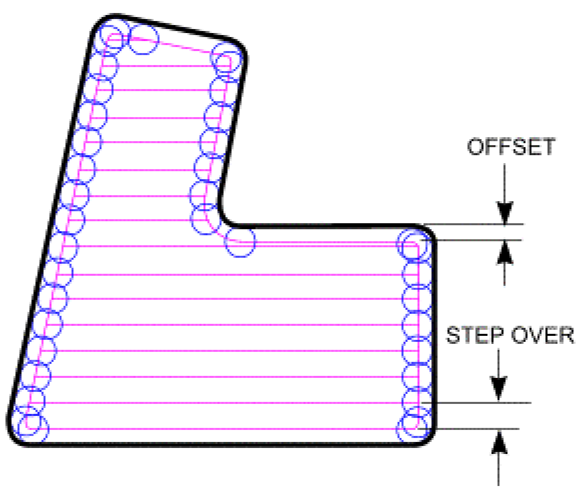
General rules

The machining curve must define a closed pocket

- The start position of the cut is set automatically to a point on the boundary. The wire then feeds to the center of the pocket and then cuts from the inside toward the boundary.
- If you wish to ensure that the wire starts in a particular position (to thread the wire for example) you should define a start point. The toolpath will then start at the start point and then feed to the center of the pocket.
- Note that no attempt is made to check if the stepover value is larger or smaller than the wire diameter.
- Cutter compensation does not apply to a pocketing operation because it outputs a wire path that is already corrected (i.e. no compensation is entered on the machine).

Zig-Zag Operation - Overview

This option defines a zig-zag area clearance cycle for a closed curve. Available only for 2-axis features.



General Rules

- The machining curve must define a closed pocket
- The start position of the cut is set automatically to a point on the boundary. The wire then feeds to the beginning of the zig-zag pattern.
- If you wish to ensure that the wire starts in a particular position (to thread the wire for example) you should define a start point. The toolpath will then start at the start point and then feed to the beginning of the zig-zag pattern. .
- By the definition of the cut Angle you can control the direction of the wire path and also the start point of the cycle.
- No attempt is made to check if the stepover is larger or smaller than the wire diameter.
- Cutter compensation does not apply to a pocketing operation because it outputs a wire path that is already corrected (i.e. no compensation is entered on the machine).

Start point

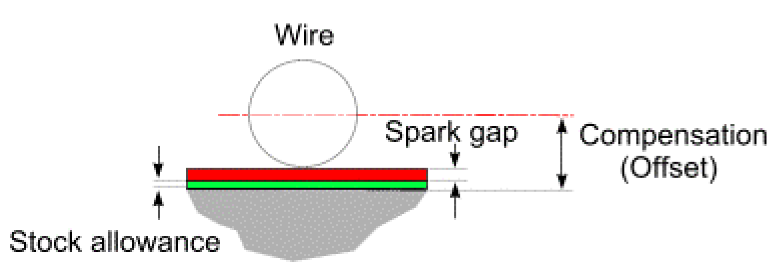
The start position that is automatically calculated by the software is directly dependent upon the Angle of the Wire path as set on the **Cycle Data** tab on the Segment Data dialog.

Angle = 0° Start point of the cycle ➔ bottom left
Angle = 90° Start point of the cycle ➔ bottom right
Angle = 180° Start point of the cycle ➔ top right
Angle = 270° Start point of the cycle ➔ top left

The chosen angle can naturally lie between the examples shown above. In this case the start point will be set to the nearerst of the positions shown. On extremely complex contours it may be necessary to experiment with the cut angle to find the optimum start point .

Working with wire radius compensation

To cut a workpiece to the finished size the Wire radius compensation function on the NC-machine is normally used. Activating this function with a particular value causes the machine to calculate a new path for the center of the wire. The compensation value is normally composed of the wire radius plus the spark gap allowance plus any finishing allowance that may be required. The compensation value is normally entered in a **Compensation Register** on the machine



If the erosion of a workpiece needs to be made in several cuts (roughing and finishing), each cut normally uses a different compensation value or compensation register. The values for the compensation are often given in a table supplied by the machine builder or automatically entered in the compensation registers via the technology tables built into the controller.

In every case you should ensure that the appropriate linear lead-in and lead-out moves are contained within the program to enable the compensation to be switched on and off.

The output of the commands to activate and deactivate the compensation is automatically carried out by the software on the first and last moves.

Wire Radius compensation on the Machine

When using the Wire Compensation command, the center path of the wire will be calculated and corrected directly by the nc-machine. The compensation amount is normally entered in a Compensation Register on the machine controller and activated by an appropriate command within the nc-program. The format of the command to activate the compensation and to control the compensation direction is dependent upon the nc-machine type. The FEATUREWIRE software supports the output of these commands both for single and multiple cuts (backwards/forwards cutting or Main/Sub-Programs)

The following parameters control the use of wire compensation on the machine:

Total passes, leave allowance , contour passes, uni-dir (uses macro).

Wire radius compensation via the Software

If the wire radius compensation is carried out by the software, the appropriate compensation value and direction will be automatically used by the software to produce a wire path that is already corrected. The path cannot be altered by changing the offset register of the nc-machine. This may be necessary for example, when cutting a contour which contains elements or arcs which are smaller than the required compensation amount and thus cannot be cut using the machine registers

The following parameters control the use of wire compensation in the software:

Total stock, leave allowance, stepover, contour stock.

Post options FeatureWIRE

Block start sets the starting line number for your CNC programs.

Block increment sets the increment between line numbers in your CNC programs.

Block maxiumum is the maximum block number for the CNC program.

Agie Control should be set if you are posting to a machine with a control from Agie. If this radio box is checked, the lines of NC code that correspond to cutter compensation are output in a different order.

FeatureWIRE feature level attributes

Strategy tab attributes (FeatureWIRE)

Operations (FeatureWIRE)

The operations that you select control the types of strategy attributes that are displayed on Strategy tab. More than one operation can be created by selecting the primary operation from the drop-down menu and then checking the *cutoff* or *contour* checkboxes that appear to the right of the drop-down menu.

Retract

Stop

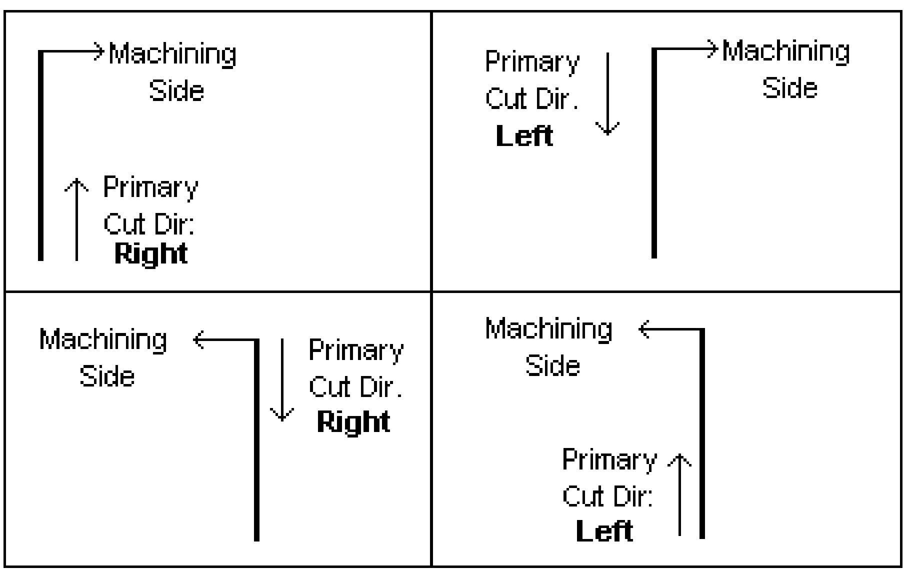
Pocketing

Zig-zag

Cutoff

Primary cut direction

This attribute controls the direction of a cut. For closed curves the options are clockwise (*CW*) or counter-clockwise (*CCW*). For open curves the choices are *right* or *left*. These settings are relative to the *machining side* setting.

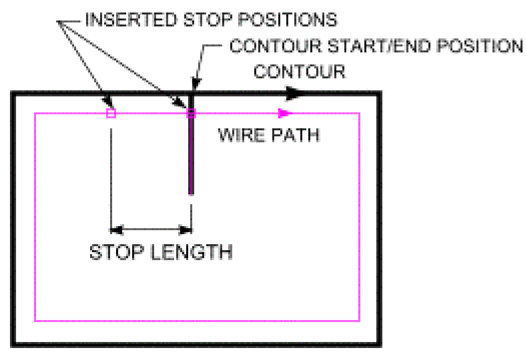


Stop length

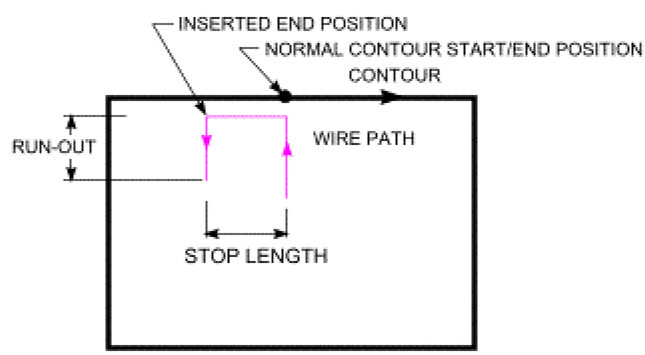
This parameter is used for Retract, Stop or Cutoff operations.

It defines the distance from the normal contour end position to the inserted stop or end position

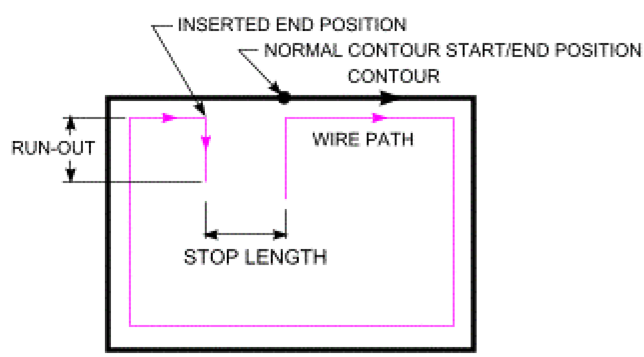
STOP OPERATION



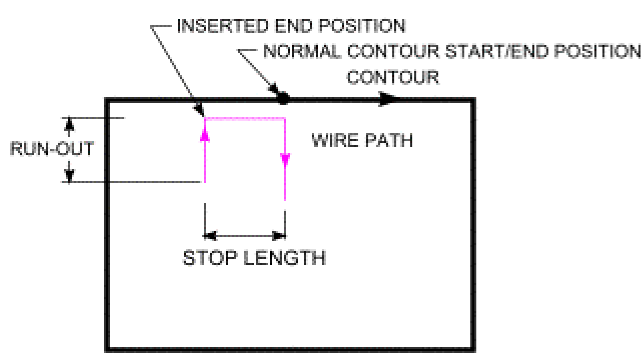
CUTOFF OPERATION



RETRACT OPERATION



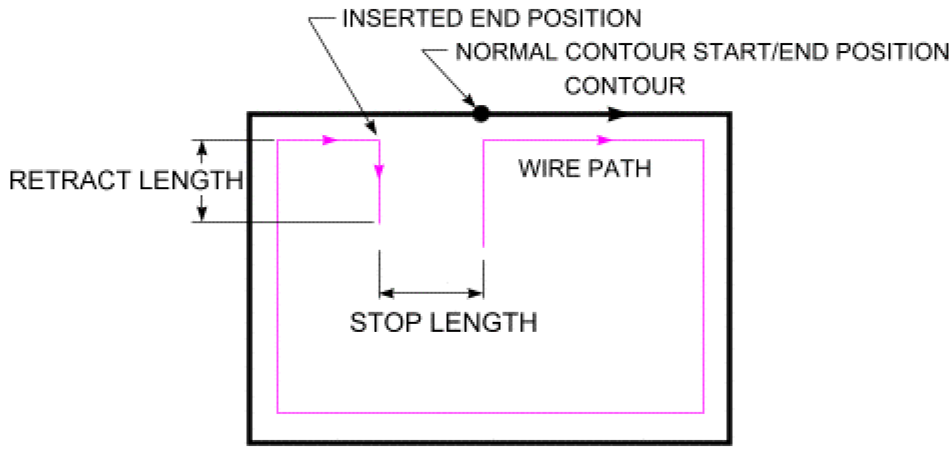
CUTOFF OPERATION (CCW)



Retract length

This parameter is used for Retract, Stop or Cutoff operations.

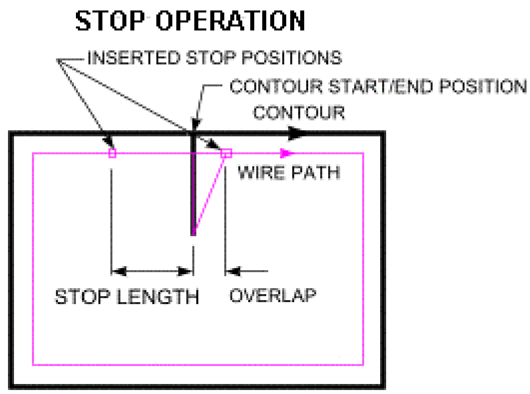
It defines the distance that the wire retracts from the part at the conclusion of an operation.

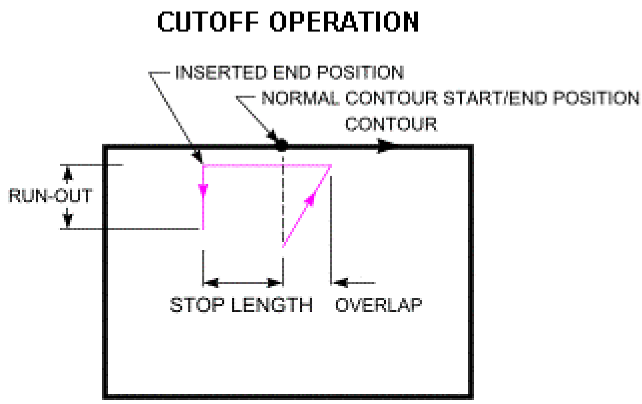


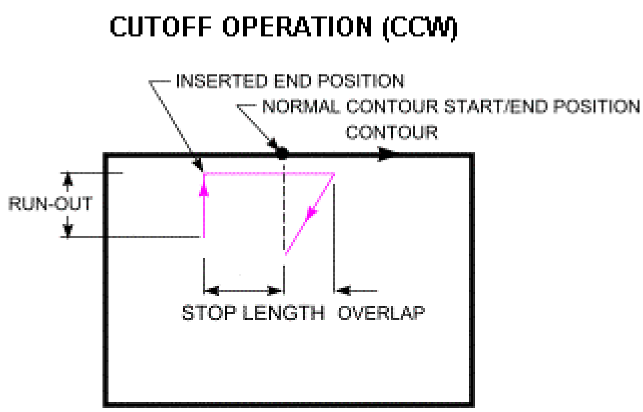
Overlap

This parameter is used for Stop or Cutoff operations.

It defines the distance by which the normal contour end position will be overcut.







Note: The run-off back to the end position of the contour will be at an angle. On some machines (e.g. Agie), an angled run-off may not be allowable. Note also that if the overlap is too large, a triangular piece of material will be left, which may fall and halt the machine.

Offset method

This radio box controls whether the offsetting of the wire path is performed on the machine using cutter compensation or by FeatureCAM. Select *Cutter comp* to perform the offsetting on the machine, or *Offset toolpath* to have FeatureCAM perform the offsetting.

Total passes

The total number of passes to take to cut the feature. If a feature has a Retract, Stop or Cutoff operation these operations will each be performed *Total passes – Contour passes*. If a feature has a contour operation, Total passes must be at least 2. *Total passes* must be between 1 and 9. Leave allowance

Uni-dir (Uses Macro)

Applies to contour, stop, retract and cutoff operations that use the cutter comp offset method.

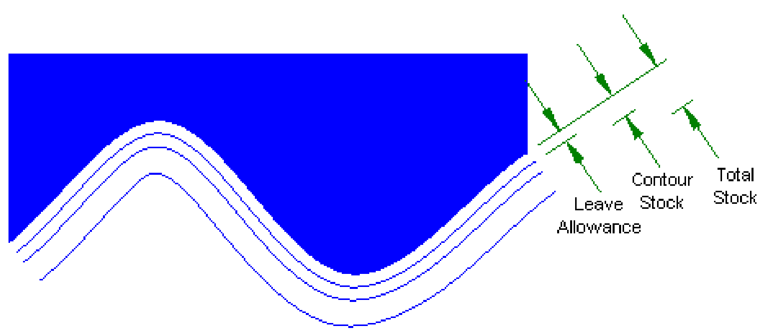
This option activates the automatic creation of sub-programs for the machining or operations with more than one pass. In this case the cutting direction for each following pass is **NOT** reversed and all passes take place in the defined direction. At the end of each pass the wire is cut and the machine re-positions to the start point for the next pass. The format and output of sub-programs must be defined in the post processor. The use of sub-programs is particularly useful when producing chain programs. In this case each machining contour will be written to a separate sub-program. The main calling program will then contain only the movements required to move to the next start point.

Total Stock

Applies to a contour operation.

This parameter sets the amount of material that should be removed from the contour when using the offset toolpath method. If this value is set to 0 only one cutting pass will be made.

Note: The calculated wire path represents the **center of the wire**. i.e. the **actual amount that will be left on the curve** is dependant upon the values set for cutter compensation.



Step Over

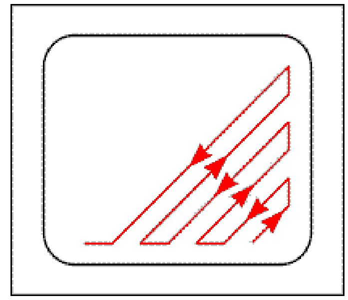
This parameter defines the step over between passes for the **Pocketing** and **Zig-Zag** cycles

Contour stock

Contour stock is the amount of material to leave for the contour operation.

Contour passes

The number of passes to take for the contour operation.

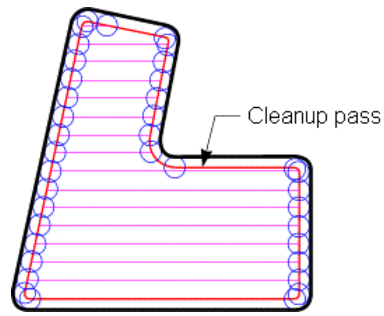


Cut Angle

This parameter sets the cutting angle for a Zig-Zag cycle. The angle is defined from the X-positive axis of the current UCS.

Cleanup pass

This option is used to create a finishing cut at the end of a Zig-Zag operation. The contour will be cut with a contour parallel finishing path to remove any rouh edges left by the step-over between passes.



Finish allowance

This is the amount of material left after a zig-zag pass. Even if a cleanup pass is used the finish allowance still remains.

Leave allowance

Leave allowance is the amount of material to leave after the contour pass.

Step over

This parameter defines the step over between passes for cutoff, stop, retract and contour operations. It applies when the offset toolpath option is selected.

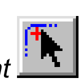
Start tab attributes (FeatureWIRE)

Start/end point (FeatureWIRE)

When this page is displayed in the wizard, the feature is displayed by showing a ruled surface. If the feature is not displayed, click here.

Die and Punch Features

For 2 and 4-axis die or punch features, a start point is automatically calculated that is off of the curve. The start point for a die or punch feature can be changed by using the following procedure:

1. If the feature has multiple curves, select the appropriate curve from the *Curve* drop down list.
2. Click the *Pick location then segment*  button.
3. Click on the new Start point. (Note that you must pick on the appropriate side for the feature type. For die features, pick on the inside. For punch features, pick on the outside. For side features pick on the machining side. If you pick a point in the incorrect side, the approach moves will gouge the part.)
4. Select the segment of the curve to connect the start point to by moving the mouse

over the feature until the segment of the curve you want is highlighted and then click the mouse.

5. The X and Y coordinates of the new start point is displayed in the dialog box.

If you want to force the wire to start on the curve, you cannot simply double-click. You must move your second pick (the one that indicates the curve segment) slightly so that you are not picking the same point twice.

If you are creating a 4-axis feature, follow the above procedure for both the upper and lower curves.

Side Features

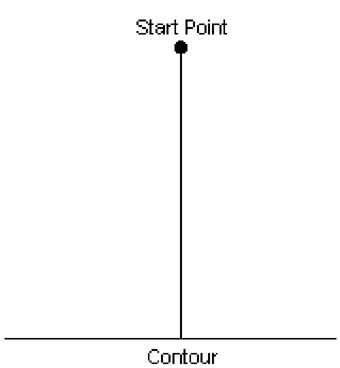
For 2 and 4-axis side features, the default is to start at the first point of the curve. To add new linear moves at the start or end of a curve:

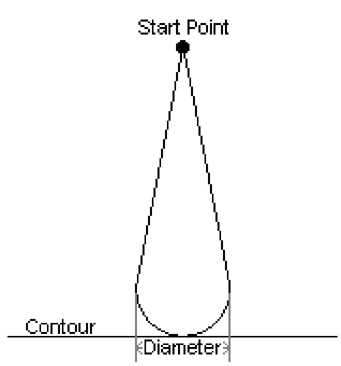
1. If the feature has multiple curves, select the appropriate curve from the *Curve* drop down list.
2. Click the *Pick point* {bmct btn-pick-xyz.bmp} button for either the start or end point.
3. A line will rubber-band from the current start of end point.
4. Click a point at the new location.
5. Alternatively, you can simple enter the new end point coordinates.

Lead style (FeatureWIRE)

There are two choices for the type of moves for leading in and out of a contour operation

Direct will move straight from the start point to the contour. Ramp will move in a straight line and then arc onto the contour. The diameter of the arc is specified as a percentage of the tool diameter. The same diameter arc is used to ramp off the contour and then the wire returns to the start point.





Misc tab (FeatureWIRE)

Wire cutting/threading

Wire cutting/threading offers four settings to control the output of wire threading or wire cutting commands.

Off	No wire threading or wire cutting commands are output in the NC-program.
Both	The commands to thread the wire and cut the wire are output automatically at the start and end of each operation within the feature.
Cut	The wire cutting command is output at the end of each operation within the feature (but no wire thread command at the start). When viewing the toolpath in center line simulation, the Cut location is denoted with a small circle.
Thread	The wire threading command is output at the start of each operation within the feature (but no cut wire command at the end). When viewing the toolpath in center line simulation, the thread location is shown as a small plus sign.

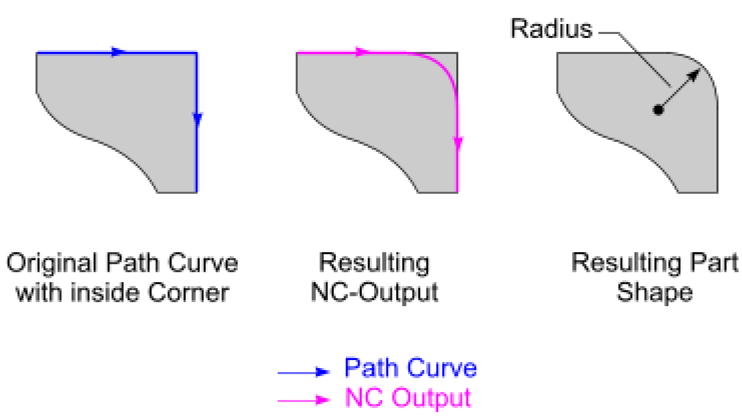
Corner round options

Theses options control how sharp corners are cut.

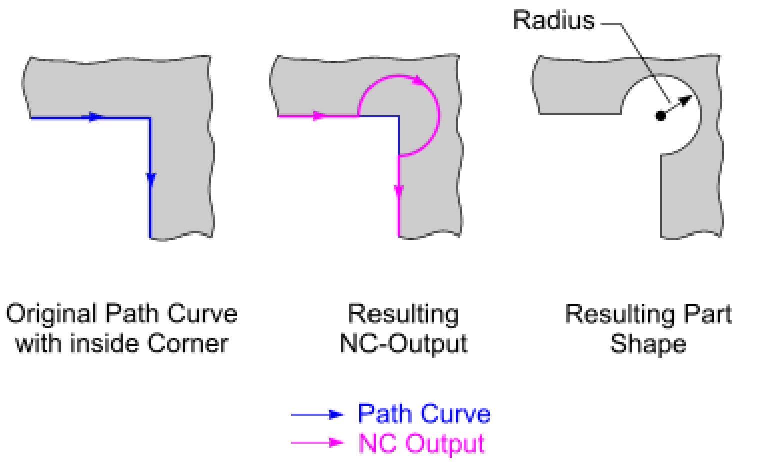
The following options are available for Retract, cutoff, stop and contour operations:

None: The machine will cut to the intersection point of the two lines

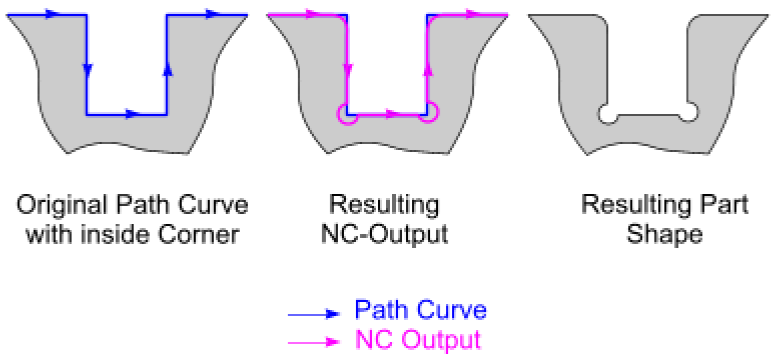
Modify outside corners: A radius will be inserted into the wire path at each outside sharp corner. This can be useful for reducing unnecessary movements and for producing cleaner corners.



Modify inside corners: A circle is inserted in the corners. The center of the circle lies on the corner point.

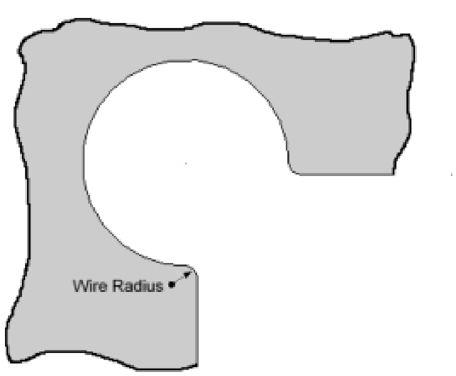


Modify both: Both inside and outside corners are modified as shown below.



Auto round

If Auto round is set, arcs are inserted at all sharp corners. This is applicable only if cutter comp. is set and you have a leave allowance or if cutter comp is not set. If you already have modify outside corners set, it will not perform any further rounding on these corners. It will round inside corners (even if modify inside corners is set) by inserting an arc before and after the circular corner as shown below: The radius of the inserted arcs is equal to the radius of the wire.



Wire EDM Tab (Default machining attributes)

Die operations – Sets the default operations for a die feature.

Punch/side operations – Sets the default operations for a punch or side feature.

Die/punch primary cut dir – The default value for primary cut direction variable for die or punch features.

Side primary offset dir - The default value for primary cut direction variable for side features

Retract/cutoff/stop stop length – The default value for Stop Length feature variable

Overlap – The default value for the overlap feature variable.

Retract/cutoff retract length – The default value for retract length feature variable.

Pocket/zig-zag finish allowance – The default finish allowance for pocketing or zig-zag operations.

Stepover – The default stepover for for pocketing or zig-zag operations. This default machining attribute is specified as a percentage of the wire diameter.

Zig-zag cut angle – The default value for the zig-zag cut angle featue attribute.

Cleanup pass – Check this box to include a cleanup pass on zig-zag or pocketing operations.

Offsetting Tab (Default machining attributes)

Offset method – The default offset method for wire features.

Total passes – The default value for the total passes for a retract, stop or contour operation.

Contour passes – The default value for the number of coutour passes for a retract, stop or contour operation.

Uni-dir – The default state of the Uni-dir feature attribute.

Total stock – The default for the total stock feature attribute.

Contour stock – The default for the contour stock attribute.

Stepover – The default machining attribute for the stepover feature attribute. This default machining attribute is specified as a percentage of the wire diameter.

Leave allowance – The default value for leave allowance for retract, cutoff, stop and contour operations.

Misc Tab (Default machining attributes)

Die/punch lead length – This is the default distance for the automatically calculated lead move of die or punch features. The initial point of the toolpath by this lead length perpendicular from the start point of the curve. The start point of the toolpath can be changed on the start page of the wire feature.

Wire cutting/threading – The default value for the cutting/threading drop down list.

Contour lead style – The default setting for lead style.

Corner options – The default value for corner options.

Auto round – The default value for auto rounding.

4-axis match curve

The feature should be displayed when this dialog box comes up. If not click here.

FeatureCAM attempts to match up the points of the upper and lower curves of a 4-axis feature. If the curves are not matched as you would like, then you can manually match the two curves.

Before starting to manually match your points, first make sure that the start points of each curve are close to each other when viewed from the top. Use the start page of feature wizard or the start tabof the properties dialog box.

To match two curves:

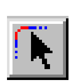
1. Click the pick curve point {bmct btn-pick-4axis-points.bmp} button.
2. All of the endpoints of the two curves are displayed as green rectangles and their connections are shown as thick green lines.
3. Click on a point of the lower curve.
4. Click on the corresponding point of the upper curve. These points will now be connected with a green line.
5. Repeat steps 3 and 4 until you have matched all the points you wish to change.
6. If you want to see how the feature will look with your new matches, click the Next button. If you want to make further changes, click the Back button and repeat steps 1-4.

If you want to connect a point on the bottom curve to multiple points on the top curve, hold down the SHIFT key when you select the bottom point and each top point.

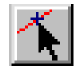
Note that you cannot change the start points of the curves by adjusting the matching points. Use the start tab to change the start points.

Sometimes you need to insert extra points or delete points from a curve In order to properly align the curves.

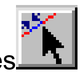
Adding points uniformly to a 4-axis wire curve

1. Enter the number of pieces. A piece is the segment between the points. The number of points added is *number of pieces* –1.
2. Click the *Pick curve piece*  button.
3. Click the piece of the curve.

Adding a single point to a 4-axis wire curve piece

1. Click the *Pick curve piece then location*  button.
2. Select the curve piece.
3. Slide the cursor along the pieces and click the location.
4. A new point is inserted at this location.

Removing curve pieces from a 4-axis wire curve

1. Click the Merge broken pieces  button.
2. Click the first curve piece.
3. Click the last curve piece. The portion of the curve is highlighted as you move the cursor to preview the selected region of the curve.
4. All internal points within these pieces are removed.

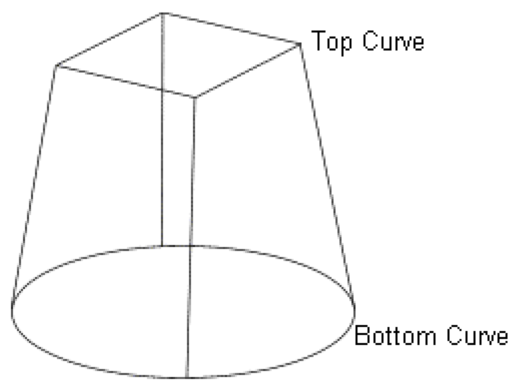
Note that you cannot remove the segment that contains the start point.

New feature - Upper curve

This dialog box is where you specify the curve that will dictate the upper shape of your feature.

To complete this page:

1. If you still need to chain your geometry into a curve, click the *Curve chaining* button and chain the curve. See Overview of Chaining for more information.
2. Click the *Pick curve*  button. The dialog box warps into a small button. Click the geometry or curve in the graphics window.   The curve must be in the XY plane of the current setup. The Z value of the upper curve is used as the default value for the top of the feature.
3. When you have selected the curve click *Next*.

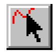


New feature - Lower curve

This dialog box is where you specify the curve that will dictate the lower shape of your feature.

To complete this page:

1. If you still need to chain your geometry into a curve, click the *Curve chaining* button and chain the curve. See Overview of Chaining for more information.

2. Click the *Pick curve*  button. The dialog box warps into a small button. Click the geometry or curve in the graphics window. The curve must be in the XY plane of the current setup. The Z value of the lower curve does not matter. It is later transformed to the plane of the upper curve and the translated back down by the *Thickness*.

3. When you have selected the curve click *Next*.

Feed, water and cutter comp registers

Most modern nc-machines today have an integrated technology database with which the machine uses to set up the optimum cutting conditions for the workpiece. In this way the cutting will be accurate and will use the full power of the machine. These settings for the cutting conditions are usually stored in Registers in the controller and are activated by particular codes in the nc-program. The FeatureWIRE software enables you to define these codes for up to nine cuts (either backwards/forwards cutting or with sub-programs). You can also load pre-defined settings from a database which you have prepared yourself.

Feed

This field is used to select the generator setting on the wire machine. The generator setting controls the cutting speed of the machine by setting parameters such as strength, pulse time and pause time between pulses of the electrical current used to produce the spark. These parameters will vary with the workpiece material, height and so on.

Water

This field is used to select the machine register that defines the water flow during cutting. The parameters that are controlled include the water pressure, flow rate etc.

Comp. Reg.

This field sets the number of the Compensation Register of the NC-Machine which is used for wire radius compensation. The value held within this register is the amount by which the wire will be corrected to the left or right of the defined wire path when the function for wire correction on the machine is switched on (normally G41 or G42)

Comp. Val

This field sets the wire radius correction value for the given offset register on the machine. The value is normally the sum of the wire radius + spark gap + any finishing allowance required. For most machines the compensation value is referenced through the compensation register, so there is no need to set this value.

Cut data dialog box

The cut data tab box provides the Feed, water and cutter compensation registers for each pass of the operation. The usage of all these settings is dependant upon the machine and controller type. These values can also be set automatically from the current material database, or they can be filled in manually. To display this tab:

1. Double-click on a FeatureWIRE feature to bring up the properties dialog.

2. Click on an operation in the tree view.

Creating a text file for wire material databases

A material database can be imported as an ASCII (readable text) file that can be created using any convenient text editor (Notepad - Wordpad etc.). When the file has been written it should be saved as plain ASCII text (if your editor offers more than one option) with a name that has the extension dxx e.g Copper.dxx. This file can be imported into the Feed/Speed and Cutting Data Tables dialog.

Format of the File:

2.0	This value is a version number. It has no special function but should always be entered to maintain the file format
COPPER 10.0 0.25 3	This line is the first line of a new material definition. The fields have the following functions: COPPER - the name of the material 10.0 - The material thickness 0.25 - The wire diameter 3 - the number of cuts
1 1 1 0.1	This line specifies the generator and water settings etc. for the first cut. The fields are as follows: 1 - the generator setting register number 1 - the water setting register number 1 - the compensation register number 0.1 - The value of the compensation
2 2 2 0.2	The parameters as above but for the second cut
3 3 3 0.3	The parameters as above but for the third cut
COPPER 20.0 0.25 3	New Material
11 11 11 0.11	
12 12 12 0.12	
13 13 13 0.13	
COPPER 30.0 0.25 3	
21 21 21 0.183	
22 22 22 0.173	
23 23 23 0.163	
COPPER 40.0 0.25 3	
31 31 31 0.184	
32 32 32 0.174	

33 33 33 0.164

STEEL 20.0 0.25 4

41 41 41 0.185

42 42 42 0.175

43 43 43 0.165

44 45 46 0.155

Wire EDM cut data

FeatureWIRE has material databases that describe the cutting conditions for various materials. Each database entry is uniquely identified by the material type, material thickness, wire type, wire diameter, and machine type. The values for each entry are the feed, water and cutter compensation registers on the machine.   The database is used to fill in the values in the cutdata dialog box, if the current cutting conditions match. The material thickness does not have to be an exact match. The values for the closest thickness will be used. If there is not a match of in the database, the cutdata for the wire operation.

New cutting condition

Add a new wire EDM cutting condition by filling in the following values:

Material – Pick an existing material or add a new material name by typing it.

Thickness – Pick an existing thickness from the drop down list or enter a new thickness. If you do not specify a unit then it defaults to the unit of the part. For millimeter enter *mm* (i.e. 5 mm). For inches enter *in.* (i.e. 0.25 in.). Do not forget the period.

Wire – Pick an existing wire type or type a new name.

Wire Dia. - Pick an existing wire diameter or enter a new one. The unit of the diameter is handled the same as the *wire thickness.*

Machine – Select an existing machine name from the drop down list or type a new machine name.

After you enter the values, click *OK*.

Cutting condition tables can contain values for cutting anywhere from 1 to 9 passes. Select the number passes and click the *New* button to add the feed, water and cutter compensation registers.

Condition dialog box

The material, wire type, wire diameter and machine type for an EDM part is specified using the stock properties dialog box.

To enter these values:

1. Bring up the stock Properties dialog box by double clicking on the stock.

2. Click the *Condition…* button.

3. Select each field using the drop down lists.

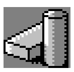
4. If you want to use a different value than is listed in the drop down list, click the *New condition…* button and define new fields. These new values will then be available in the Condition dialog box.

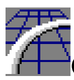
5. Click *OK*.

Note that if you are not using a cutdata database to automatically determine feed, water and cutter compensation registers , then the only value in this dialog box that is important is the wire diameter. This value is used in the 3D simulation for the actual width of the cut.

Introduction to FeatureWIRE for FeatureMILL users

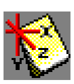
FeatureWIRE is structured similarly to all other FeatureCAM products and the steps are used to walk you through the process.

 A model of the stock is displayed in the graphics window.

Geometry and curves are used to describe the shape of the part.

 The geometry and curves are included in features and toolpaths are derived from these features.

Toolpaths are simulated in the graphics window using line drawings or 3D shaded graphics. 3D simulation also allows the simulation of slug removal.

 Toolpaths are translated in to machine specific code using the appropriate post processor.

Differences between FeatureWIRE and other FeatureCAM products include:

1. The diameter and type of wire you are using are specified as part of the Stock wizard instead of on individual operations.

2. FeatureWIRE provides cutting condition tables instead of feed/speed databases. These cutting condition tables list registers that contain power settings, water (coolant) settings and sometimes cutter compensation. This information is machine specific and is supplied by the EDM machine tool vendor.

3. For some machine types, FeatureWIRE provides a single text file that can control the entire process. For other controls, FeatureWIRE provides a file that you must reference from a command file that you create at the control.

Types of wire EDM tapers

Left: The taper is performed to the left of the curve relative to the Primary Cut Direction. The arcs or the curve result in conical corners.

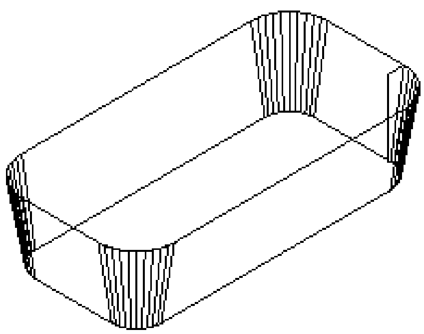
Right: The taper is performed to the right of the curve relative to the Primary Cut Direction. The arcs or the curve result in conical corners.

ISO-Left: The taper is performed to the left of the curve relative to the Primary Cut Direction. The arcs or the curve result in cylindrical corners.

ISO-Right: The taper is performed to the right of the curve relative to the Primary Cut Direction. The arcs or the curve result in cylindrical corners.

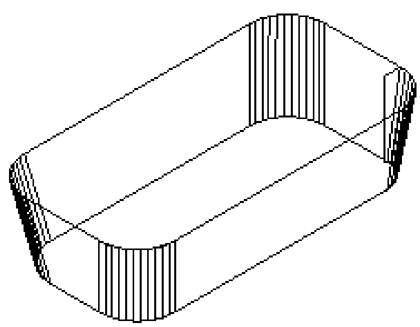
Default conical corner

The normal taper has corners that are shaped like a cone. The arc of the top curve is different from the arc of the bottom curve.



ISO cylindrical corner

An ISO taper has cylindrical corners. The arcs of the top and bottom curves have the same radius.



Wire EDM Taper

None - With this option, the feature has no taper.

Constant - With this option, the feature has a constant taper angle all the way around the feature. The taper is specified by an angle, specified in degrees, and taper type.

Variable – If you want to specify a different taper value for each line and arc of your curve, click the *Variable* radio button and then click the *Variable* button to display the variable taper table.

Variable taper table

If the feature has multiple curves, select the appropriate curve from the *Curve* drop down list. The curve is broken down into arc and line segments and each segment is represented by a row in the table. As you click on a row of the table, the segment of the curve is highlighted in the graphics window. If you wish to find the row of the table that corresponds to a segment, click the *Pick curve piece* {bmct btn-pick-segment.bmp} button and select the segment. The row of the table will then be selected.

The columns of each table are:

Status – Double click on box in the status column to display a drop down list of Types of wire EDM tapers.

First Angle – This is the taper angle, specified in degrees, for the first end point of the segment.

Angle – This is the taper angle, specified in degrees, for the second end point of the segment.

Taper reg – This is the register on the machine containing taper information for this segment.

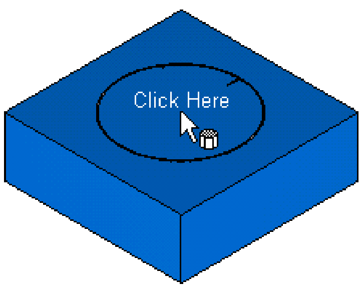
Note that if you want smooth walls of your part, the *angle* of one segment must match the *first angle* of the next segment.

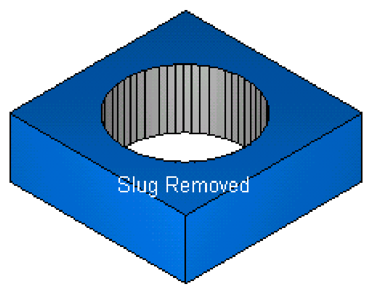
Simulated slug removal

In FeatureWIRE, you can graphically remove a slug after the 3D simulation is finished. The region to remove must be totally separated from the rest of the part.

To remove the slugs:

1. Perform a 3D solid toolpath simulation.
2. Toggle the viewing mode in the viewing flyout menu to the up position.
3. Click on the slug in the graphics window. The slug is removed.
4. You can click as many slugs as you wish to remove.
5. Click the Stop button in the simulation toolbar to clear the screen.





Chapter 28
Saving Your Work

Saving a part file

You can't save a FeatureCAM file unless you have a dongle. To save an FeatureCAM part file the first time

1. Select Save or *Save As* from the *File* menu.
2. Select the directory where you want to store your file.
3. Name your part file.

You can give the part file any name of any size limited only by your Windows version. FeatureCAM automatically appends a fm suffix to your part file name—this identifies file as an FeatureCAM file.

The *Save As* dialog box includes these options:

File Name enters a new filename to save a FeatureCAM part file with a different name. A filename can contain up to eight characters and an extension of up to three characters.

Save File as Type lets you save the current part file as a fm file, or as a file with any name and extension you choose, but the file is still in FeatureCAM format.

Drives selects the drive for storing the part file.

Directories selects the directory for storing the part file.

Network connects to a network location, assigning it a new drive letter.

Save Options

In FeatureCAM, you have the option of saving a number of previous versions of your part as you work. To activate this feature,

1. Click Save Options from the File menu.
2. Check Create backup copies.
3. Enter the Number of Copies to Keep.

Now when you save your work, the previous version(s) will be saved to disk under a name that begins with "Backup". The latest version of your work will always be saved using the name of your FeatureCAM part.

Save computed toolpath

For certain parts, generating toolpaths can be time consuming and you may want to save them for the next time you open a part. The setting of *Save computed toolpath* controls the default behavior for saving toolpaths. Select from *Never save*, *Always save*, or *Ask me* to be prompted each time you save a part. Note that this is different than saving the NC text file that the NC machine reads. Instead you are saving the FeatureCAM internal toolpath representation.

Saving an NC part program to disk

Generate NC code and save it to disk. The actions here require a dongle.

1. Open the file.

2. Generate the toolpaths by clicking Show Centerline.

4. Click NC tab in the Manufacturing Results window.

The NC part program appears in the Manufacturing Feedback window. It has not been saved to disk yet.

To save NC part programs and manufacturing documentation to disk:

1. Select Save NC from File menu.

2. Change the directory if you wish.

3. If you want the NC code file to have a different name that the part name, enter a different name for the NC file name. If you omit the file extension, it will default to .txt.

4. If you want to save the NC code for only the current setup, click *Current Setup*. Click *All Setups* if you want to save separate NC programs for each setup.

5. Choose what you would like to save to disk from among: *Operations list, Tool list, NC code, Tool data, F/S data*.

6. Click *Create subfolder* if you would like to create an additional folder underneath the *NC Output Directory*. This folder will have the same name as your part.

7. If you know you would like to overwrite all existing folders with the same names, click *Overwrite Existing Files.* Otherwise you will be asked to confirm before overwriting any file.

8. Click OK.

9. Confirm the save operation.

Five different types of files are saved for your part.

- *.OP is the Manufacturing Operations list and is the same information shown in the Manufacturing Feedback window when you click OP.

- *.TL is the Manufacturing Tool Detail Sheet and is the same information shown when you click TL.

- *.TXT is a text file containing the NC code for the particular part file. This is the same information shown when you click NC.

- *.TDB is a FeatureCAM tooling database that contains just the tools you used to create the part.

- *.FDB is a FeatureCAM material database that contains the feed and speed tables used for the part.

- *.CDB is a FeatureCAM machine configuration database that contains the settings for default machining attributes.

The filename of all files will be the same as the part name. If you have a part called "part", the files created would be: part.op, part.tl, part.txt, part.tdb, part.fdb. When saving the NC code you are given the opportunity to change the NC file name. If you enter a different NC file name, the default file extension is .txt.

If a part has multiple setups, the setup ID number is appended to the part name. For example, if you had a part called "plate" with three setups, three different sets of files are created called "plate", "plate2", and "plate3". An. OP, .TL, .TXT, .TDB, .FDB and .CDB file would be created for each setup for a total of eighteen files.

NC program names

The NC program name defaults to be the same as the FeatureCAM part file name. This name is used in three different places:

- The part name in the NC file comment
- If the part has macros, the names of the macros are derived from the program name. A two digit number is appended to the program name to form the macro name. For example, if the part is named, *plate*, the first macro would be named *plate01*.
- The name of the NC text file and documentation files (setup sheet and tooling list).

The program name can be changed in the Setup dialog box. Even though the name is changed in the Setup dialog box, this name is the same for all setups. A number is appended to subsequent setups. Changing the program name changes the name of the NC file, tooling list and operations sheet that is later generated.

Note for Fanuc control users: you will want to use a numeric value for the NC program name. This will give you a numeric NC file name and appropriately named macros.

Saving your settings

All program options are stored in the *ezfm.ini* file located in the same directory that Windows is installed in. That file includes the settings you choose for Toolbars, the *Viewing, Simulation, Machining Attributes*, and *Post Process* dialog boxes. Three settings from the *Options* menu affect the communication with the *ezfm.ini* file. *Save Now* writes the current settings to the file. *Reload* reads the settings contained in the file into the program. *Save on Exit* saves the current settings when ever you exit the program. If this last option is not selected, the settings for your current session won't be saved to the *ezfm.ini* file when you exit.

Exit command

Use this command to end your FeatureCAM session. You can also use the Close command on the application Control menu. FeatureCAM prompts you to save documents with unsaved changes.

Shortcuts

Mouse: Double-click the application's Control menu button.
Keys: ALT+F4

Chapter 29
How Do I Get the File to the Machine?

Loading a part program to an NC machine

Communicate between your PC and your milling with the *EZ-UTILS* software (for past users of EZ-CAM), the Microsoft Windows Terminal program or other communications programs. Windows 95 has no comparable utility. If you are using Windows 95, you may need to purchase a communications utility, if you don't already have one, capable of communicating along a serial port connection.

Communications between your PC that is running FeatureCAM and your milling machine can be accomplished with the EZ-UTILS software, the Microsoft Windows Terminal program or other communications programs.

Configuring HyperTerminal

HyperTerminal is easy to use to send and receive NC code from the machine. The first time you use HyperTerminal, set up an icon for communication with the machine tool. All the parameters for machine communication are linked to the icon link you created. In later sessions, you only have to double click the icon instead of re-entering the communications settings.

- Launch Hyperterminal from the Start menu.
- Double click the Hypertrm.exe icon. You may or may not see the .exe extension depending on your computer configuration. In later sessions with the machine
- You may be prompted to install a modem. If you don't have one, click No and proceed with setting up a communications icon.
- Enter a name for the icon and pick an icon from the group.
- Click OK.
- Set the list box at the bottom of the screen to Direct to COM1 (or whichever port you will communicate through. Click OK and a communications properties box appears.
- Your machine tool should have recommended communication settings. If so, use those settings here.
- Click OK. HyperTerminal is configured to communicate with your machine.

You may also need to review how to send and receive files from the machine.

Sending files to the machine

Set up the machine to receive NC programs. The specific actions to perform at the machine vary from manufacturer to manufacturer.

- At the computer, from within HyperTerminal, follow these steps
- Select Send Text in the Transfer menu. Do NOT select Send File, only Send Text works.
- In the Browse dialog box, select the file to send.
- Click OK. HyperTerminal transfers the information.

Receiving files at the computer

At the computer, from within HyperTerminal, follow these steps:

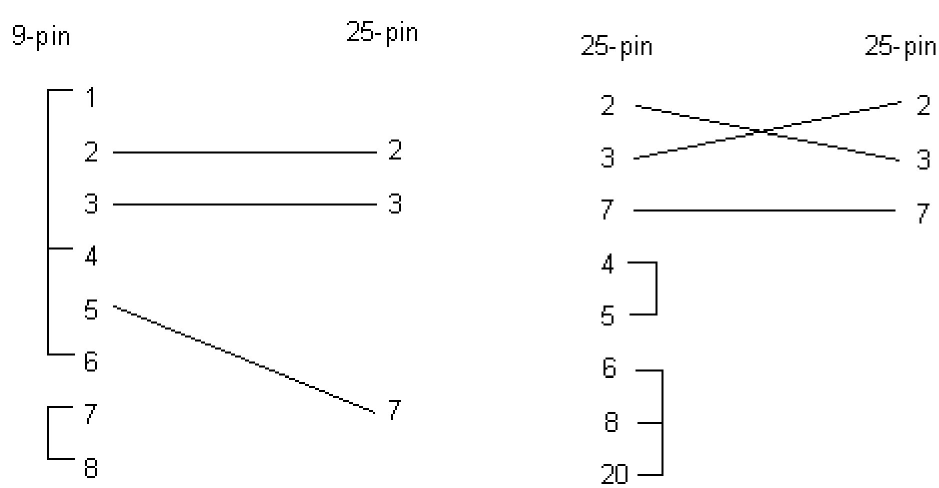
- Select Capture Text in the Transfer menu. Do NOT select Receive File.
- Name the file to be received.
- Click OK.
- Send the file from the machine.

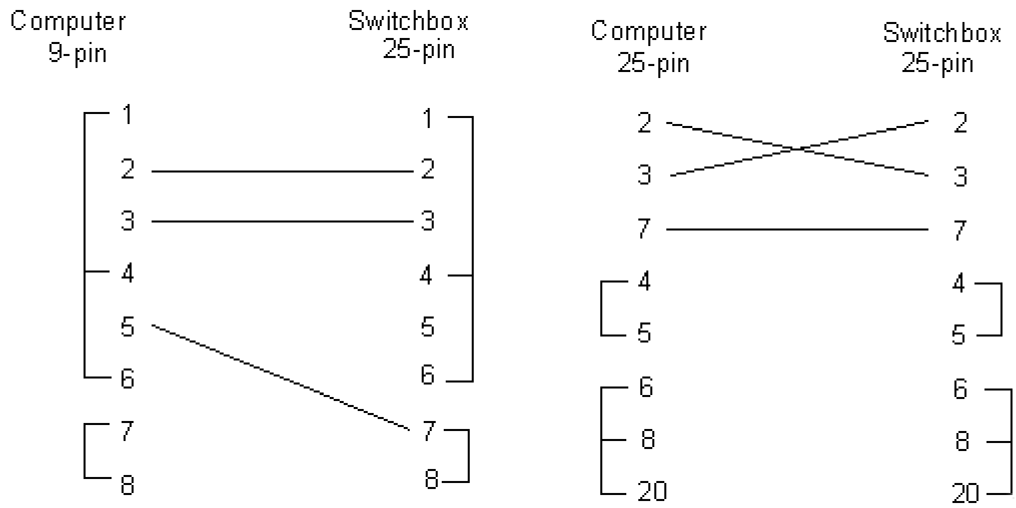
Cable

You'll need an RS-232 adapter cable to connect the computer to the communications cable leading to the CNC machine. The adapter cable may also be used to connect the computer to a serial port expander (ABC switch box). The adapter cable can be plugged into the *COM1* or *COM2* port, at the back of the main unit. Note that the cable can be a 9-pin or 25 pin connector at the computer end. The manual contains an illustration of the required wiring pattern for the cable.

Serial port pinouts

The necessary communications cables for connecting the computer to any device or CNC control can be made with parts available from most electronics supply stores. The pin functions for a standard 25-pin RS-232 port and a standard 9-pin RS-232 port are shown below. Check your computer owner's manual for the correct information.





EZ-UTILS

To run the EZ-UTLS program, double click the EZ-UTLS icon. When the EZ-UTLS main screen appears, there are three menus at the top of the screen: are To CNC, From CNC, and Other. These menus contain commands to communicate with various CNC controls.

To CNC menu

The To CNC menu shows five commands. These are all used to allow a file to be sent from the FeatureCAM system to the CNC control via a hard-wired cable which is attached to both the CNC control and the FeatureCAM computer.

From CNC menu

The From CNC menu shows three commands. These are all used to accept a file sent from a CNC control to the FeatureCAM system via a hard-wired cable which is attached to both the CNC control and the FeatureCAM computer.

Other menu

The Other menu in EZ-Utils contains three menu choices. These are used only before sending a part program to the CNC.

Resequence lets you renumber the lines in a part program without re-posting the program. When the Resequence command is selected from the Other menu, a dialog box is shown asking for the Start Number and Increment values. When OK is selected, the screen displays a dialog box that allows the operator to select a *.TXT file from a disk volume.

Once a file is selected, enter a filename at the prompt so the resequenced file can be saved under a new name. The name can also be the same as the file selected. Once a file name is selected, the file is renumbered. The first line in the program is numbered with the Start Number and the line number on every succeeding line in the part program file is incremented by the Increment value.

A message is shown on the screen indicating that the operation was completed successfully. Click OK to return to the EZ-Utils menu level.

Serial port changes two basic communications settings. The Baud Rate is the speed at which the computer sends data to the CNC. This speed must be set the same at both the CNC and FeatureCAM computer.

The Port setting determines which serial port the computer uses to communicate to the CNC. The communications cable must be attached to the selected port.

Set directory changes the locations where incoming files from a CNC are stored. This command should be set before attempting to download any file, or before establishing communications using the EZ-Link utility.

Communicating with specific controllers

The following topics address connecting to controllers using EZ-UTLS.

Bridgeport SX and DX controls, and BOSS 8-10

The *EZ-Link* utility allows communications in both directions even though it appears in the To CNC menu. Running this utility displays a dialog box with the message *Data Transfer in Progress!*. Communications can then be controlled from the CNC control via the *REMOTE* and *DNCLINK* utilities. Procedures for using the EZ-Link utility are listed below.

Connecting to the SX15 or DX32

When linking EZ-UTLS to the Bridgeport SX15 and DX32 controls, connect the Bridgeport communications cable to the RS-232 serial port on the machine and the adapter cable connected to the RS-232 serial port (*COM 1* or *COM 2*) on the computer.

Sending from EZ-UTLS to the CNC

At the EZ-UTLS system:

1. Start EZ-UTLS and select the *EZ-Link* utility from the To CNC menu.

At the CNC:

2. Select the *Load Remote* command on the numeric keypad.
3. Press the *4 RECV EZLINK* key on the keyboard.
4. Press the *F3 LOAD* key. A list of files is shown. Press the + key to enter a new file name for receiving. Be sure to type the *.TXT* file extension.

The file should be sent automatically to the control.

Sending from the CNC to EZ-UTLS

At the EZ-UTLS system:

1. Start the EZ-UTLS program and select *EZ-Link* from the To CNC menu.

At the CNC:

2. Select the *4 Load Remote* command on the numeric keypad.

3. Press the *3 SEND EZLINK* key on the keyboard.

4. Press the *F3 LOAD* key. A list of files is shown. Use the cursor keys to select the file to be sent to EZ-UTLS. Press *Enter* when the file name is highlighted.

The file should be sent automatically to EZ-UTLS.

Connecting the EZ-UTLS system to BOSS 8, 9, or 10

When linking EZ-UTLS to BOSS 8, 9, or 10, connect the Bridgeport communications cable to the BOSS 8 adapter cable, then plug the round end of this cable into the port on the machine. Connect the adapter cable to the RS-232 serial port (COM 1 or COM 2) on the computer. The BOSS 8 adapter cable should be connected to port *B* on the CNC.

NOTE: Use the REMOTE load utility to upload and download programs less than 12,000 characters long (V2E3, R2E3) and less than 200,000 characters long for the R2E4. Use DNCLINK (Direct Numerical Control) when the part programs are too long to reside in the CNC memory.

Sending from EZ-UTLS to the CNC

To transfer a program from disk into the memory of the BOSS 8/9/10:

At the EZ-UTLS system:

1. Select *Set Directory* in the Other menu.

2. Choose the directory from the dialog box.

3. Click *Open*.

4. Select *EZ-Link* from the To CNC menu.

At the CNC:

5. Under *OPTIONS*, set the baud rate to *4800*.

6. Press the *LOAD/CLEAR/EDIT* button once.

7. Press 0 (REMOTE), then EXECUTE.

8. Press 0 (FROM REMOTE), then EXECUTE

9. Enter the file name and press *EXECUTE* (if the file name is numeric). If the file name is NOT numeric, just press *EXECUTE*, and the EZ-UTLS system prompts for the file name. Enter the file name, then press *RETURN.*

Sending from the CNC to EZ-UTLS

To transfer a program to disk from the memory of the BOSS 8/9/10:

At the EZ-UTLS system:

1. Select *Set Directory* in the Other menu.

2. Choose the directory from the dialog box.

3. Click *Open*.

4. Select *EZ-Link* in the To CNC menu.

At the CNC:

5. Under *OPTIONS*, set the baud rate to *4800*.
6. Press the *LOAD/CLEAR/EDIT* button once.
7. Press *0 (REMOTE)*, then *EXECUTE*.d) Press *1 (TO REMOTE)*, then *EXECUTE*.
8. Enter the file name and press *EXECUTE* (if the file name is numeric). If the file name is NOT numeric, just press *EXECUTE*, and the EZ-UTLS system prompts for the file name. Enter the file name, then press *RETURN*.

Connecting to the EZ-Trak SX

When linking EZ-UTLS to an EZ-Trak SX, connect the Bridgeport communications cable to the RS-232 serial port on the machine and the EZ-UTLS adapter cable connected to the RS-232 serial port (COM 1 or COM 2) on the computer.

Sending from EZ-UTLS to the CNC

To transfer a program from EZ-UTLS into the memory of the EZ-Trak control:

At the EZ-UTLS system:

1. Select *Set Directory* in the Other menu.
2. Choose the directory from the dialog box.
3. Click *Open*.
4. Select *EZ-Link* in the To CNC menu.

At the CNC:

5. Press *UTILS*.
6. Press *7 Send or Receive Files*.
7. Select the *EZ-Link* communications protocol, by pressing *1*.
8. Select either *3* or *4*, receive an EZ-TRAK (.PGM) file or other (.TXT).
9. Enter the numeric name of the file to be sent from EZ-UTLS.

Sending from the CNC to EZ-UTLS

To transfer a program to EZ-UTLS from the memory of the EZ-TRAK:

At the EZ-UTLS system:

1. Select *Set Directory* in the Other menu.
2. Choose the directory from the dialog box.
3. Click *Open*.
4. Select *EZ-Link* from the To CNC menu.

At the CNC:

5. Press *UTILS*.
6. Press *7 Send or Receive Files*.
7. Select the *EZ-Link* communications protocol, by pressing *1*.
8. Select either *1* or *2*, send an *EZ-TRAK (.PGM)* file or other *(.TXT)*.
9. Enter the numeric name of the file to be sent to the EZ-UTLS system.

Heidenhain controls

Use the *Heidenhain* utility to transfer an entire file from EZ-UTLS to the Heidenhain control. The first set of procedures describes communications between EZ-UTLS and a TNC 145 control; the second set describes communications for a TNC 150 control.

Connecting to Interact I and II

When linking EZ-UTLS to Interact I or II or to Bridgeport R2C3 Series I or II with the TNC 145 control, connect the Bridgeport communications cable to the RS-232 serial port on the machine and the adapter cable to the RS-232 serial port (*COM 1* or *2*) on the computer.

Sending from EZ-UTLS to the CNC

If downloading from EZ-UTLS, make sure that the text file is in the proper Heidenhain format.

See the *Heidenhain TNC 145 Operation Manual* for details.

At the EZ-UTLS system:

1. Select *Serial Port* from the Other menu.
2. Set the *Baud Rate* to *2400* and click *OK*.
3. Select *Heidenhain* from the To CNC menu.
4. Select the desired directory and file name from the dialog box.

At the TNC 145:

5. Select the *MANUAL* mode.
6. Click *EXT*.
7. Enter *2400* for the baud rate and press *ENT*.
8. Now choose the *PROGRAMMING* and *EDITING* mode.
9. Press the *EXT* key. The CRT displays the message: *EXTERNAL DATA INPUT*.

At the EZ-UTLS system:

10. On the screen, the message *Data Transfer in Progress* appears.

Sending from the CNC to EZ-UTLS

At the EZ-UTLS system:

1. Select *Serial Port* from the Other menu.
2. Set the *Baud Rate* to *2400* and click *OK*.
3. Select *Heidenhain* from the From CNC menu.
4. Select the desired directory and enter a new file name in the dialog box.
5. Click *Open*.

At the TNC 145:

6. Select the *MANUAL* mode.
7. Press the *EXT*ernal key.
8. Enter *2400* for the baud rate and press *RETURN*.
9. Select either *PROGRAM RUN* modes (SINGLE BLOCK or FULL SEQUENCE).
10. Enter the starting block number of the program, then press the *GO TO* key.
11. Press *EXT*ernal key. The CRT displays the message *EXTERNAL DATA OUTPUT*.

At the EZ-UTLS system:

12. The message *Data Transfer in Progress* appears on the screen.

Connecting to Series II Interact 4

When linking EZ-UTLS to Series II Interact 4 with the TNC 150 control, connect the Bridgeport communications cable to the RS-232 serial port on the machine and the adapter cable connected to the RS-232 serial port (*COM 1* or *COM 2*) on the computer.

Chapter 30
Support information

Technical support

Technical support is available through:

1. Your reseller
2. EGS WEB site, *www.featurecam.com.*
3. Engineering Geometry Systems. Send mail to *sales@featurecam.com* or call 1-888-393-6455 for pricing information.

What does this warning mean?

General information on manufacturing errors

Since only one stage is displayed at a time, an error may occur in another stage in the document, but will not appear in the operation sheet currently displayed. Note also that FeatureCAM will not switch stages to take you to the first error.

A single problem will often result in multiple errors. When multiple errors occur, it is recommended to fix the tooling errors and regenerate the toolpaths. In many cases, the other errors will be resolved.

Warnings are also shown in the operation sheet, but do not prevent FeatureCAM from continuing to generate tool paths. FeatureCAM does not draw attention to the warnings it gives since the tool path generation was successful.

Manufacturing errors

Manufacturing errors occur when FeatureCAM is unable to complete the tool path generation for a part. When an error occurs, the user is unable to run any simulation, post to NC code or save NC code. Errors appear between lines of asterisks (*) in the operation sheet listed after the operation in which the error occurred. Errors are also displayed in the Op List tab with an exclamation point  icon in the left margin. Warnings are also tagged using the yield sign  icon.

If an error is detected during toolpath generation, the *Code generation failed* dialog box is displayed.

If you click *Yes* in this dialog, the first error in the operations list will be highlighted. If you click *No*, the errors will still appear in the operations list, but you must explicitly ask to step through the errors by clicking *Next Error*.

If an error occurs during toolpath generation, three error buttons appear in a separate toolbar in the left-hand corner of the window to enable you to read and fix the errors. The buttons perform the following functions:

 Selects the next error in the operations list.

 Selects the previous error in the operations list.

 Provides options for fixing the error that is selected.

If you click *Hint* while an error is selected in *the Manufacturing Operations Sheet,* a series of dialog boxes appears to help you fix the error

How the error hints dialog box works

Every *Error Hints* dialog box contains an error code, an error category, and a description of the error. See "Warnings and Error Codes" for a complete list of FeatureCAM error codes. Each dialog box also provides an option for fixing the error. The following buttons appear at the bottom of the dialog box to help you fix the error:

Prev Display the previous option for fixing the error. This button is only for navigating through the errors. Pressing it does not commit you to accepting this approach to fixing the error.

Next displays the next option for fixing the error. This button is only for navigating through the errors. Pressing it does not commit you to accepting this approach to fixing the error.

Action accepts the proposed method of fixing the problem. The specific types of actions proposed include: *Add Tool, Manager*, and *Override*. If you select this option, FeatureCAM will step you through fixing the error.

Cancel the error fixing process without attempting any fix.

Warnings and error codes

Here are some general rules about FeatureCAM's errors and warnings:

- Codes that begin with TS are related to tool selection. A tool selection error occurs when a tool of a specific size is not found in the currently selected tool crib.
- Codes that begin with FS are related to feed and speed calculation. A feed and speed error occurs if no feed/speed table exists for the combination of stock material type and hardness, tool material, and tool finish, or if there is no tool.
- Codes that begin with DC are related to decomposition of features. A decomposition error occurs if FeatureCAM is unable to decompose a feature correctly. This will happen if a tool is not found for the feature in order to determine the number of operations necessary to cut the entire depth.
- Codes that end with W are warnings. Warnings are given if FeatureCAM changed the way in which a feature was cut or if the user needs to complete some action before the feature can be cut.

Warning codes

Code	Cause	Suggested Action
TSD02W	Could not find a spiral tool, substituting with a gun style.	1. Accept Copy tool from another crib. 2. Create a tool. 3. Override with a different tool 4. Select a different crib.

		5. Modify feature.
TSD12W	Could not find a plug tool, substituting a bottoming tap	See TSD02W.
TSK12W	Chamfer tool can't cut chamfer.	1. Override with a different tool. 2. Copy tool from another crib. 3. Create a tool. 4. Select a different crib. 5. Modify feature.
TSK22W	Counter sink tool can't cut chamfer	See TSK12W
TSK03W	Tool can't cut round.	See TSK12W
TSJ11W	Tool must be ground before use, zigzag operation	Grind tool.
TSH11W	Tool must be ground before use, slotting operation.	Grind tool.
TSK11W	Tool must be ground before use, profile milling operation.	Grind tool.
TSI02W	This tool must be adjusted by operator for correct diameter--. boring operation	Adjust diameter prior to running part.
TPL02W	Improperly defined stock curve	Stock curve is a point or is undefined somehow
TPL03W	profile lies outside stock boundary	No feature curve defined or feature curve does not overlap with stock curve.
TPL04W	Ignoring improperly specified start point.	Change start point.
TPL05W	Ignoring improperly specified end point.	Change end point.
TPL06W	Ignoring Tool Nose Radius Compensation. Disabled in post	Cutter comp is turned on for the feature but off in the post options
TPL07W	Undercut detected	Unable to completely rough feature with this tool.
TPL08W	Undercut detected. Unable to completely finish feature with this tool.	Operation did not completely cut the feature. some material remains
DCT02W	Bottoming tap required for this feature.	1. Accept. 2. Modify feature.

Error codes

Code	Cause	Suggested Action
TSA01	Tool not found for standard drilling operation	1. Copy a tool from drilling operation another crib. 2. Create a tool. 3. Override with another tool. 4. Select a different crib. 5. Modify feature.
TSA02	Tool not found for drilling operation before ream.	See TSA01 Action
TSB01	Tool not found for spotdrilling operation.	See TSA01 action
TSC01	Tool not found for reaming operation.	See TSA01 action
TSD01	Tool not found for tapping	See TSA01 action

	operation.	
TSE01	Countersink tool not found for chamfer operation.	See TSA01 action
TSF01	Tool not found for countersink operation.	See TSA01 action
TSG01	Tool not found for counterbore operation.	See TSA01 action
TSH01	Tool not found for slotting operation.	See TSA01 action
TSH02	Selected tool not valid for slotting operation.	See TSA01 action
TSI01	Tool not found and unable to create a custom tool for boring operation.	See TSA01 action
TSJ01	Tool not found for zigzag operation.	See TSA01 action
TSK01	Endmill tool not found for profile milling operation.	See TSA01 action
TSK02	Selected tool not valid for profile chamfering operation.	See TSA01 action
TSK03	Tool not found for profile rounding operation.	See TSA01 action
TSM01	Tool not found for profile milling operation during decomposition of the feature.	See TSA01 action
FS001	Feed/speed table not found for the given operation.	1. Add a new feed/speed table. 2. Override the feeds and speeds. 3. Select a different tool.
FS002	Feed/Speed table not found for the given operation. This error indicates that a BRIGHT tool material table could have been used but was unable to find it.	See FS001 action
FST01	Feeds and speeds were not computed because a tool was not available.	Fix tool selection error
TPD03	Successive moves on the same line	Check feature curve.
TPD04	Feature curve cannot end inside stock boundary	Look at boundaries or feature curve.
TPD14	Tool insert is too large to cut curve cleanly.	Change selected tool
TPD18	No Cutting area determined	Check feature or stock curve.
TPD19	Feature curve start or end point in material.	Change start or end point.
TPD20	Feature curve start or end point in material.	Change start or end point.
TPD21	Unable to determine area to be machined.	Check feature curve or stock curve.
TPD22	Unable to cut in specified direction.	Cut in a different direction.
TPD52	Illegal value for cut depth.	Check depth feature parameter.
TPD54	First infeed is too small	Check threading parameters.
TPD56	Second infeed is too small	Check threading parameters.
TPD58	Memory overflow	Contact distributor.
TPD60	Depth is smaller than infeed	Check threading parameters.

TPD61	Check retract/engage angles and depth	Change parameters
TPD64	Tool undefined	Change tool.
TPD65	Illegal value of engage or withdraw angle	Change parameters.
TPD66	Illegal tool shape	Adjust tool.
TPD67	Tool wider than slot	Change tool.
TPD68	Slot deeper than max tool depth	Change tool or slot parameters.
TPD69	Unable to verify, check path	Check feature curve.
TPD70	Check tool dia value	Check tooling parameters
TPD71	Check zstep value	Check feature parameters.
DCT01	Feature was unable to be decomposed.	Fix tool selection error

Security keys (Dongles)

For most users, all you need to know about the security keys is mentioned in Chapter 1 of *Getting Started with FeatureCAM* guide. These sections are included in case you must perform further installation procedures to get your key to function properly.

Windows 95/98

Windows 95 Driver Installation Procedure

1. Make a backup copy of the diskette.

2. Start Windows 95. Select "Run" from the Taskbar and run the file SENTW95.EXE in the \WIN_95 subdirectory on the driver diskette. There are two command line options:

- /q Quiet mode. Normal dialogs described below are not displayed. Error messages are displayed.

- /pxxx Path, where xxx is the path of files to be installed. Specify the path of files to be installed. Otherwise, files will be copied from the default directory.

3. Select "Install Sentinel Driver" from the "Functions" menu.

4. Click "OK" when the "Driver installed! Restart your system." message appears. Restart Windows 95.

The following files have been created on your hard disk:

WINDOWS\SYSTEM\SENTINEL.VXD

WINDOWS\SYSTEM\RNBOSENT\SENTW95.EXE

WINDOWS\SYSTEM\RNBOSENT\SENTW95.DLL

WINDOWS\SYSTEM\RNBOSENT\SENTW95.HLP

WINDOWS\SYSTEM\RNBOSENT\SENTINEL.SAV

Windows 95 Driver Configuration

1. Start Windows 95. Select "Run" from the Taskbar and run the file SENTW95.EXE in the WINDOWS\SYSTEM\RNBOSENT subdirectory.
2. Select "Configure Sentinel Driver" from the "Functions" menu.
3. Click the "Edit" button to edit an existing parallel port setting or click the "Add" button to add a new parallel port setting. Select "OK" after you finish the port configuration.
4. Restart Windows 95 for the changes to take effect.

The information about the dongle is based on the Sentinel System Drivers Version 5.17,

Manual Installation of Sentinel System Driver for Windows 95

We highly recommend that you install the Sentinel System Driver for Windows 95 with our installer. If you decide to install it manually, you may do so by performing the following steps:

1. Make a backup copy of the diskette.
2. If your application is a Win32 application, go to step 7.
3. Run Registry Editor (REGEDIT.EXE in Windows 95 root directory).
4. Select HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\RunServices. (Clicking on the expansion box next to the item name expands the branch)
5. With RunServices highlighted, click on "Edit" menu and select "New", then select "String Value" from its submenu. Registry Editor adds an entry "New Key #1" to the end of the list. Rename it to "RNBOStart". (To rename a key, click it with right mouse button, select Rename, and type the new name) Double-click on it to bring up "Edit String" dialog box. Type "%system_root%\system\rnbosent\sentstrt.exe" and click OK, where %system_root% is the name of the Windows 95 root directory.
6. Alternatively, the file sentstrt.exe can be copied to the %system_root%\startm~1\programs\startup subdirectory.
7. Copy the file "SENTINEL.VXD" from the "WIN_95\" directory on the Sentinel Driver diskette to the %system_root%\system directory. Create the subdirectory %system_root%\system\rnbosent. Copy all other files from the "WIN_95\" directory to the %system_root%\system\rnbosent subdirectory. Also copy "SENTINEL.VXD" to %system_root%\system\rnbosent as "SENTINEL.SAV", this is your back-up file to the system driver.
8. The installation is now complete. To use the driver with Win32 applications, start the application. For all other applications, restart Windows 95.

The information about the dongle is based on the Sentinel System Drivers Version 5.17,

Windows 95 Driver Un-install

1. Start Windows 95. Select "Run" from the Taskbar and run the file SENTW95.EXE in the WINDOWS\SYSTEM\RNBOSENT subdirectory (or from the original distribution media).

The driver can be removed via the command-line options or the pull-down menu.

2. Command-line options: SENTW95 /q /u - Quietly removes the existing driver.
3. Pull-down menu: Select "Remove Sentinel Driver" from the "Function" menu.
4. When complete, a dialog box with the message "Sentinel Driver Removed" is displayed.
5. Click "OK" to continue.

The information about the dongle is based on the Sentinel System Drivers Version 5.17, Copyright (c) 1991-1996 Rainbow Technologies, Inc. All Rights Reserved.

Windows NT

Windows NT Driver Installation Procedure

1. Make a backup copy of the diskette.
2. Under the Microsoft Windows NT Main group, double click on "Command Prompt".
3. Change drive to the floppy drive contains the Portable Driver Diskette. In case of the floppy diskette distribution, change the current directory to \WIN_NT. In case of the CD-ROM distribution, change the current directory to \PRODUCT\DRIVERS\WIN_NT.
4. Type "INSTALL.BAT" at the command prompt.
5. There are two command line options:

- /q Quiet mode. Normal dialogs described below are not displayed. Error messages are displayed.
- /pxxx Path, where xxx is the path of files to be installed. Specify the path of files to be installed. Otherwise, files will be copied from the default directory.

6. A window with the title bar "Sentinel Driver Setup Program" is displayed.
7. Select "Functions" and then "Install Sentinel Driver" from the menu bar.
8. A dialog box with the default path for the NT driver is displayed. Change the drive letter if necessary and click "OK".
9. The Sentinel Driver and associated files are copied to the hard disk.
10. One of the DLLs, SNTI386.DLL, SNTMIPS.DLL, SNTALPHA.DLL, or SNTPPC.DLL and SENTTEMP.HLP are copied to \%SYSTEMROOT%\SYSTEM32. SENTTEMP.SYS is copied to the file \%SYSTEMROOT%\SYSTEM32\DRIVERS\SENTINEL.SYS.
11. %SYSTEMROOT% is the directory where Microsoft Windows NT has been installed.
12. If the driver installation is successful, a dialog box with the message "Sentinel Driver Files Copied Successfully" is displayed.
13. When complete, a dialog box with the message "Driver Installed! Restart your system" is displayed.
14. Click "OK" to continue.
15. Restart your computer.

16. If this doesn't work for you, try the manual installation. You might also try the DOS installation.

The information about the dongle is based on the Sentinel System Drivers Version 5.17, Copyright (c) 1991-1996 Rainbow Technologies, Inc. All Rights Reserved.

Windows NT Driver Configuration Procedure

If your system setting has been changed and you would like to reconfigure the Sentinel Driver, perform the following steps:

1. Open the Microsoft Windows NT "Control Panel".
2. In the Control Panel double click on "Drivers".
3. Select "Sentinel for i386 System" in the installed driver list box.
4. Click the "Setup" button.
5. Click the "Edit" button to edit an existing parallel port setting or click the "Add" button to add a new parallel port setting.
6. Select "OK" after you finish the port configuration.
7. A dialog box with the message "Your driver setting has changed" is displayed. You will need to exit and restart Windows NT so that the new setting can take effect."
8. If you have any other application running at the background with unsaved data, choose "Don't Restart Now". Quit all the application and restart Windows NT, Otherwise select "Restart Now".

The information about the dongle is based on the Sentinel System Drivers Version 5.17, Copyright (c) 1991-1996 Rainbow Technologies, Inc. All Rights Reserved.

Manual Installation of Sentinel System Driver for Windows NT

We highly recommend that you install the Sentinel System Driver for Windows NT with our installer. If you decide to install it manually or wish to integrate our driver installation into your application setup, review the install.bat file in the WIN_NT directory for an example of how invoke our Windows NT install program.

The information about the dongle is based on the Sentinel System Drivers Version 5.17, Copyright (c) 1991-1996 Rainbow Technologies, Inc. All Rights Reserved.

Windows NT - DOS Device Driver

It is possible, though unlikely, that on some systems, applications using the Watcom C/C++ with Rational DOS/4G and Microsoft Visual C/C++ with Phar Lap TNT DOS Extender may be unable to communicate with our Windows NT Device Driver. In the unlikely event that you experience problems running your application under Windows NT, a DOS Device Driver is provided which allows your application to communicate with the Windows NT Device Driver on systems where it otherwise could not.

To install this device driver, do the following:

1. Copy the file SENTDOS.SYS to the target system's hard disk.

2. Add the following statement to the custom Config file used by your application's PIF file, or if your application does not use a custom Config file, to the system's CONFIG.NT file.
   `device=%path%\sentdos.sys`
   Where %path% is the actual path where SENTDOS.SYS resides.

The information about the dongle is based on the Sentinel System Drivers Version 5.17,

Windows NT Driver Un-install

1. Open a Microsoft Windows NT "Command Prompt".
2. Change to the change the current directory to \WIN_NT. In case of the CD-ROM distribution, change the current directory to \PRODUCT\DRIVERS\WIN_NT.
3. Run "INSTALL.BAT" from the command prompt.
4. Use Command-line options: (See the INSTALL.BAT file for examples.) /q /u - Quietly removes the existing driver.
5. Via Pull-down menu - Run INSTALL.BAT and select "Remove Sentinel Driver" from the “Function" menu.
6. When complete, a dialog box with the message "Sentinel Driver Removed" is displayed.
7. Click "OK" to continue.

Note: Some files may not be removed until you restart your computer.

The information about the dongle is based on the Sentinel System Drivers Version 5.17,

Index

D

R

S

T