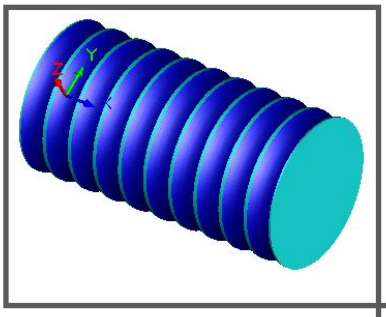
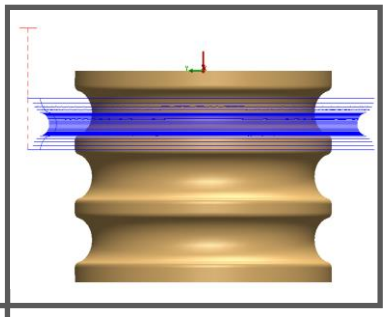
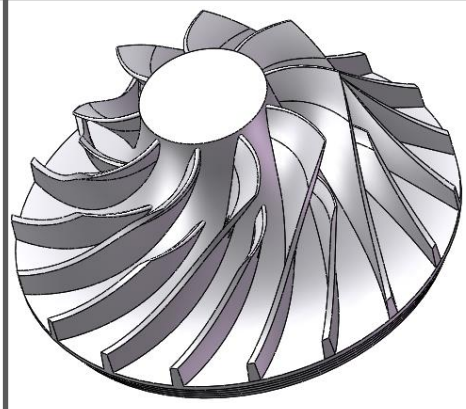
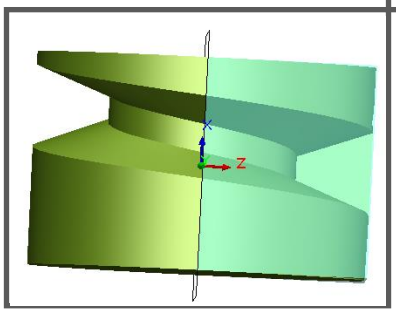
CAMWorks 2022

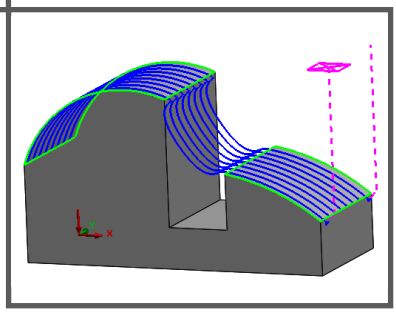
Multiaxis Tutorial











Disclaimer

Geometric Americas, Inc. makes no warranties, either expressed or implied with respect to this manual. Geometric Americas, Inc. reserves the right to revise and improve products as it sees fit, and to revise the specifications and information contained herein without prior notice. Due to continuing product development, specifications and capabilities described in this manual are subject to change without notice.

Trademarks

The following and other product names and corporate references herein are registered or proprietary trademarks of their respective owners.

CAMWorks® is a registered trademark of Geometric Americas, Inc.

SOLIDWORKS® is a registered trademark of Dassault Systèmes SOLIDWORKS Corp.

FeatureManager™ is a trademark owned jointly by DS SOLIDWORKS and Bentley Systems, Inc.

All other brands and names are the property of their respective owners.

Geometric
An HCL Technologies Company

TABLE OF CONTENTS

1. LEARNING MULTIAXIS MACHINING BASICS

CAMWorks Multiaxis Machining allows a wide variety of shops and manufacturing facilities to take advantage of 4/5-axis machines that provide greater productivity, equipment flexibility and quality. CAMWorks 4/5-axis simultaneous machining allows you to create toolpaths across complex shapes that could not be machined on 3-axis machines. This includes high-performance automotive port finishing, impellers, turbine blades, cutting tools, 5-axis trimming, and undercut machining in mold and die making.

This chapter provides an opportunity to learn CAMWorks Multiaxis Machining through a step-by-step hands-on tour of the features and functions.

The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

These exercises have been developed to provide supplemental information on Multi Axis Machining and assume you are familiar with CAMWorks Milling.

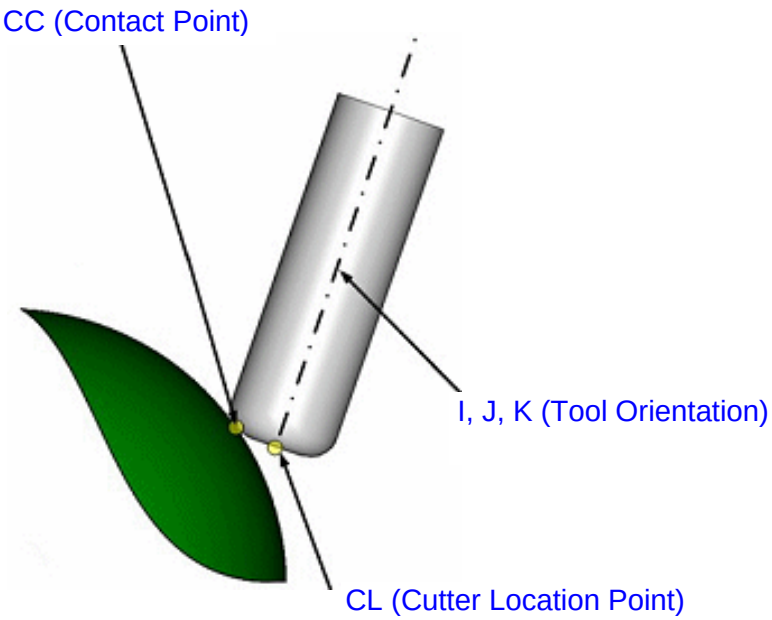
The exercise parts are installed when you install CAMWorks and are in the *Examples**Tutorial_Parts**4-5AxisMill* folder in the CAMWorks data folder.

Typical Drive: *CAMWorksData**CAMWorks202x**Examples**Tutorial_Parts**4-5AxisMill*.

IMPORTANT! **CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.**

Multiaxis Machining Terminology

When the tool tilts, it rotates around the contact point of the tool and the material. Note that the point on the material is fixed and the contact point of the tool changes according to the orientation of the tool axis (except for sharp corner tools).

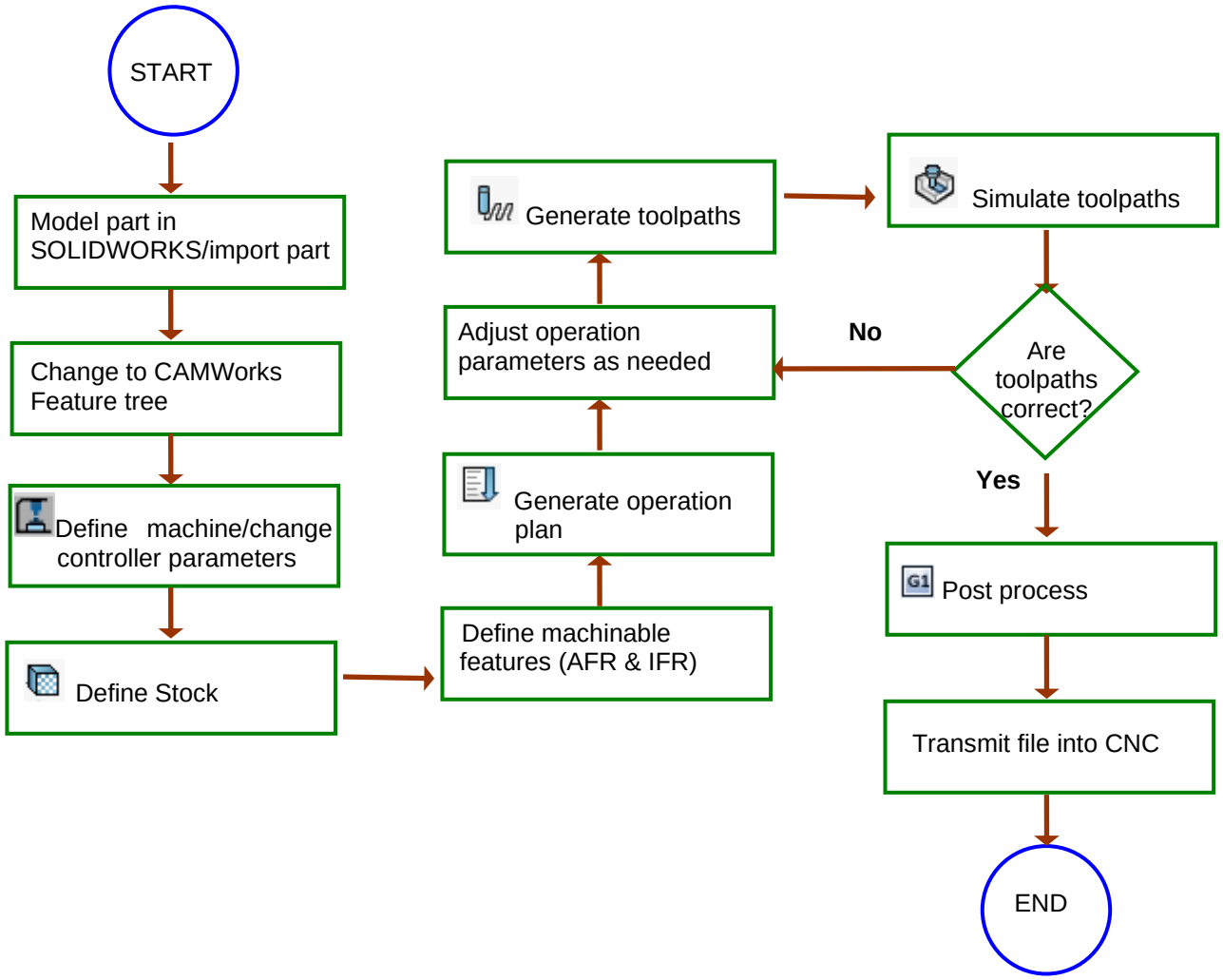


CC = The point where the tool touches the material.
CL = The point that is represented by the coordinates of the machine program.
I, J, K = Values that represent the orientation of the tool axis.

Steps to Generate Toolpaths and NC Code

The following steps are used to generate Multiaxis Mill toolpaths and NC code:

1. Model the part or open the part file in SOLIDWORKS/CAMWorks Solids.
2. Change to the CAMWorks Feature tree.
3. Define the Machine and modify the Post Processor parameters.
4. Define the Stock.
5. Insert a Mill Part Setup and define the machinable features.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths and run simulation
8. Adjust parameters if necessary.
9. Post process the toolpaths.



Multiaxis 1

Topics covered in this tutorial:

- Defining the Machine and Post Processor
- Defining the Stock
- Inserting a Mill Part Setup and Interactively Defining Multi Surface Features
- Generating an Operation Plan and Adjusting Parameters
- Generating Toolpaths and Running Toolpath Simulation
- Adjusting Machining Parameters and Defining the Tool Orientation
- Post Processing the Toolpaths

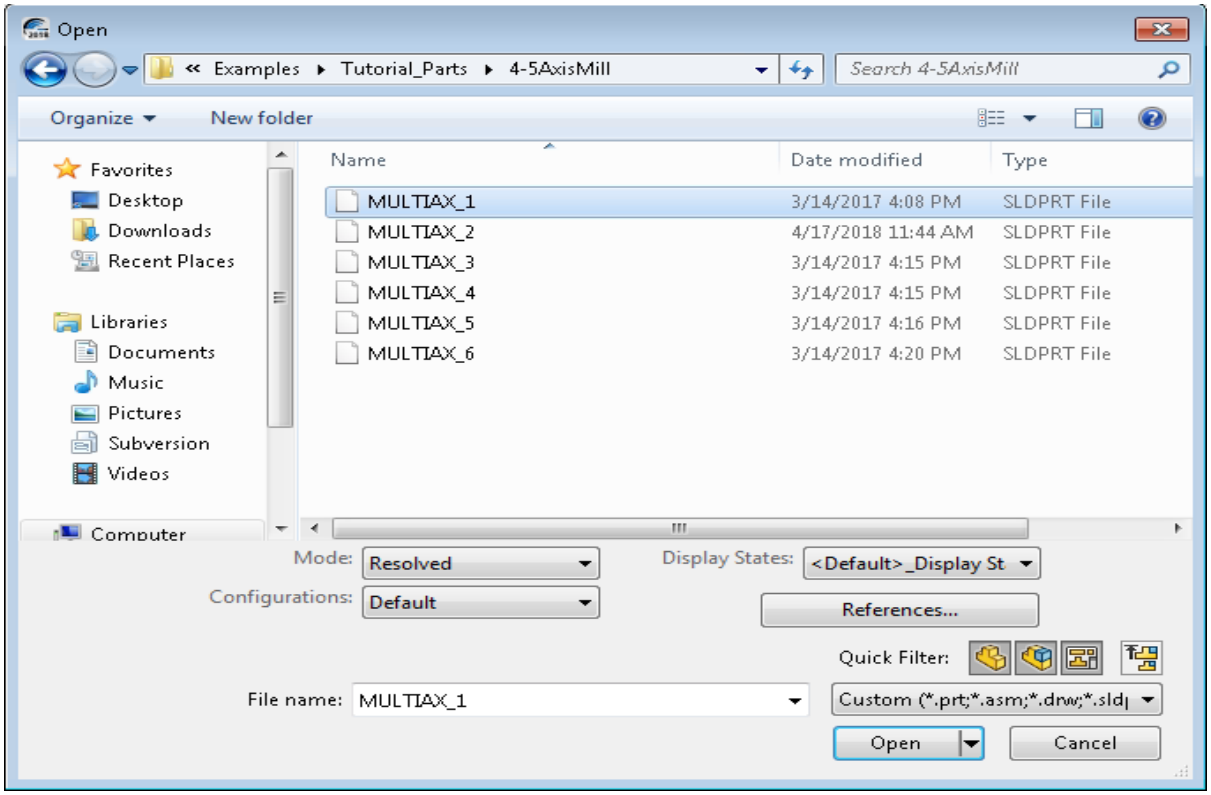
Model Part in SOLIDWORKS/CAMWorks Solids or Import Part

A part is a solid that is created with SOLIDWORKS/CAMWorks Solids or imported into SOLIDWORKS/CAMWorks Solids from another CAD system via an IGES, Parasolid, SAT file, etc. This exercise uses an existing SOLIDWORKS part.

Opening the Part

Open the part file **MULTIAX_1.sldprt** in the following folder.

Drive: \CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\4-5AxisMill



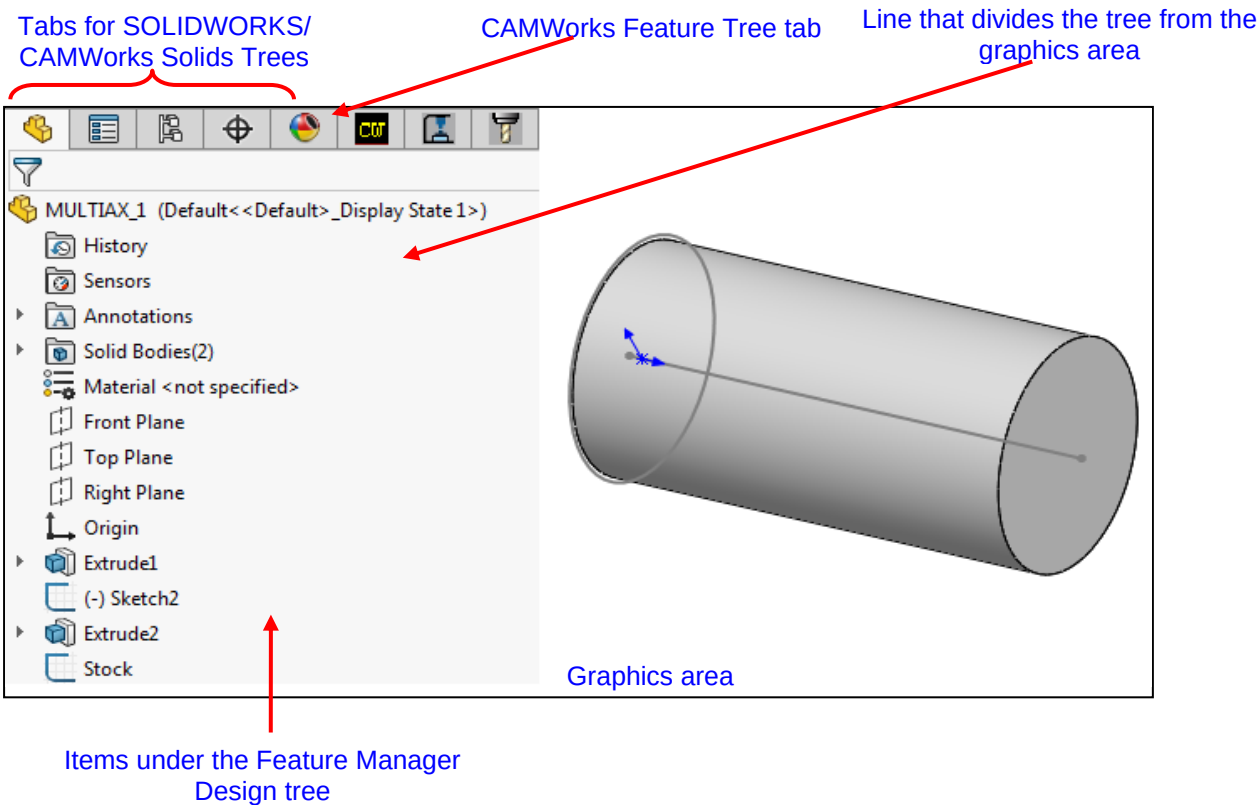
Opening the Solid Part file

Viewing the FeatureManager Design Trees

The FeatureManager design tree displays the list of the features, sketches, planes and axes related to the part.

To use CAMWorks, you need to move between SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Click the *Pin* button to continuously view this Tree area.

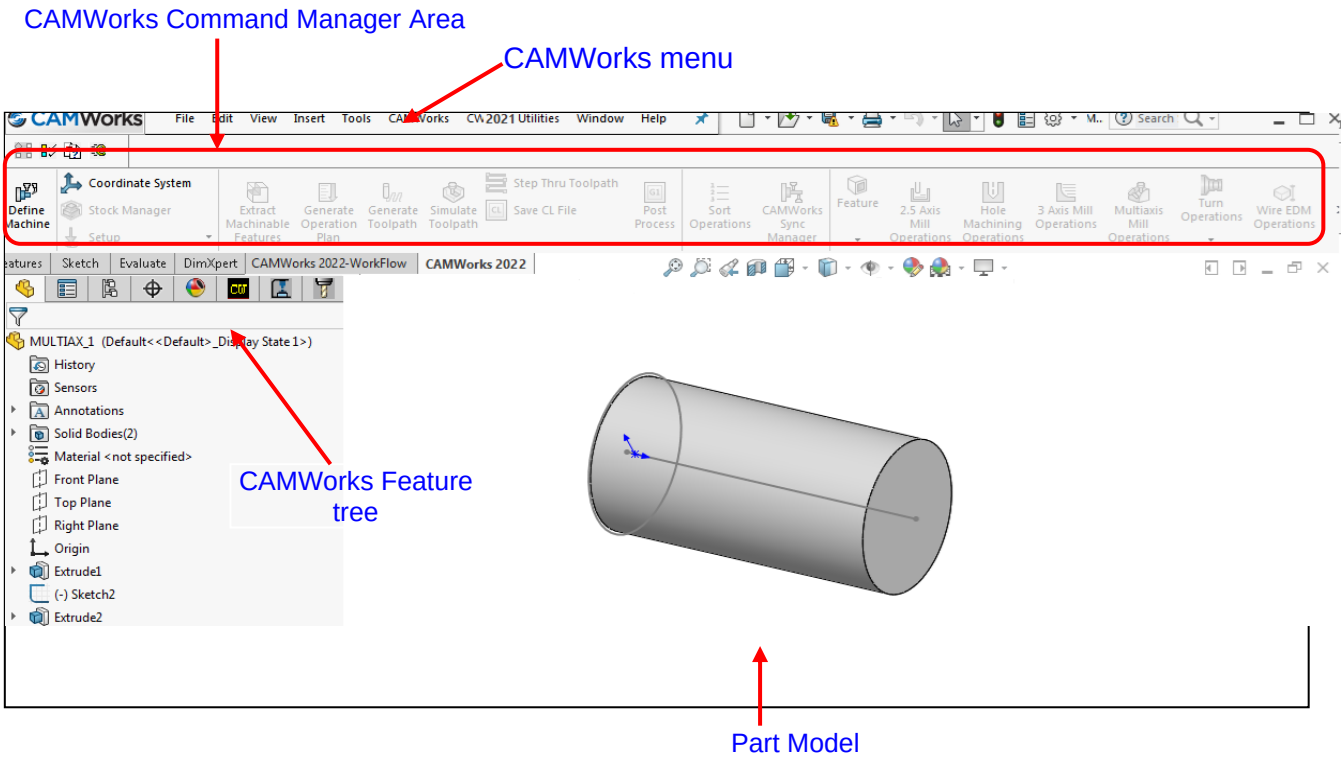
If the CAMWorks tabs [, ,] are not visible, you can expand the size of the tree. Position the cursor on the line that divides the tree area from the graphics area. When the cursor changes to a bar, drag the bar to the right until the tabs display.



Change to CAMWorks Feature Tree

Click the *CAMWorks Feature Tree* tab to view the items under this tree.

Initially, the tree lists the CAMWorks NC Manager, Configurations, Machine, Stock Manager and Recycle Bin items.



CAMWorks Machining Trees

The CAMWorks machining trees provide an outline view of the machining information for the model. Initially, the CAMWorks Feature tree shows only the CAMWorks NC Manager, Configurations, Stock Manager, Machine and Recycle Bin items. As you follow the steps to generate an NC program, this tree expands to include Part Setups and machinable features.

- **Configurations**

  Multiple CAMWorks datasets are supported. Each dataset is called a configuration. You can use configurations to support multiple machines and SOLIDWORKS configurations.

- **Stock Manager**

  The stock is the material from which the part will be machined. If the Machine type chosen is Mill, you can define the stock as a rectangular shape (bounding box) or an extruded sketch or an STL file. You can also specify the type of material.

- **Machine**

  The Machine item defines the machine tool that the part will be machined on. The machine definition includes the type of machine (E.g.: Mill, Turn, Mill-Turn), Tool definitions and the Post processor. The machines are set up in the Technology Database.

- **Recycle Bin**

  The Recycle Bin in the CAMWorks Feature tree is used to store machinable features that you do not intend to machine.

CAMWorks Command Manager

Click *CAMWorks* on the SOLIDWORKS/CAMWorks Solids menu bar. This action displays the CAMWorks Command Manager. It provides access to the main CAMWorks commands. The commands are explained in the CAMWorks Context-based Help.

CAMWorks Command Manager

Customization of CAMWorks Command Manager

Command Manager is a context-sensitive toolbar that can be dynamically updates based on the toolbar you want to access. It provides access to the main CAMWorks commands found on the CAMWorks menu. By default, it has toolbars embedded in it.

The CAMWorks Command Manager can be customized. Right click anywhere on the CAMWorks Command Manager and select *Customize menu* from the RMB context menu. The *Customize* dialog box will be displayed. Use the *Toolbars, Commands, Menus, Keyboard shortcut, Mouse gesture and Options* tab of this dialog box to customize the Command Manager as per your requirements.

Alternative Access to CAMWorks Commands

All the commands executed from the CAMWorks Command Manager can also be alternatively accessed from the RMB context menu of the CAMWorks NC Manager. This is a context menu. To execute the command, right click on the CAMWorks NC Manager item in the tree and select the desired command from the RMB context menu. In addition to the Command Manager commands, this right click context menu also provides access to a variety of commands.

CAMWorks Options

Use the *CAMWorks Options* dialog box to changes the various settings you want apply in CAMWorks.

To open the *CAMWorks Options* dialog box:

1.  Click on the *CAMWorks Options* icon in the CAMWorks Command Manager.

   OR

2. Right click on the *CAMWorks NC Manager* item in the *CAMWorks Feature tree* and select *Options* from the RMB context menu.

   The *Options* dialog box will be displayed.

3. In this dialog box, go to the *Mill Features* tab. Under *Extract Machinable Features* group box, ensure that the *Method* is set to *MfgView* (default setting).

4. Click *OK* to apply the changes and close the dialog box.

CAMWorks Context-based Help

In addition to tutorial documents, CAMWorks is provided with a context-based help. Every dialog box and interface within CAMWorks has an associated *Help* button. Click on the *Help* button  on the CAMWorks Command Manager to open the context-based Help. Every parameter and tab of each dialog box in explained in the context-based Help.

Define the Machine

The machine includes information that identifies what to machine, how to machine it, and the format of the NC output. Important parameters of the machine definition include:

- **Machine type – Mill, Turn, Mill-Turn or Wire EDM**: The machine type defines the machinable feature set that can be recognized automatically and defined interactively.

  The icons that display in the tree identify the current machine:

  Mill Machine **Turn Machine** **Mill-Turn Machine** **Wire EDM**

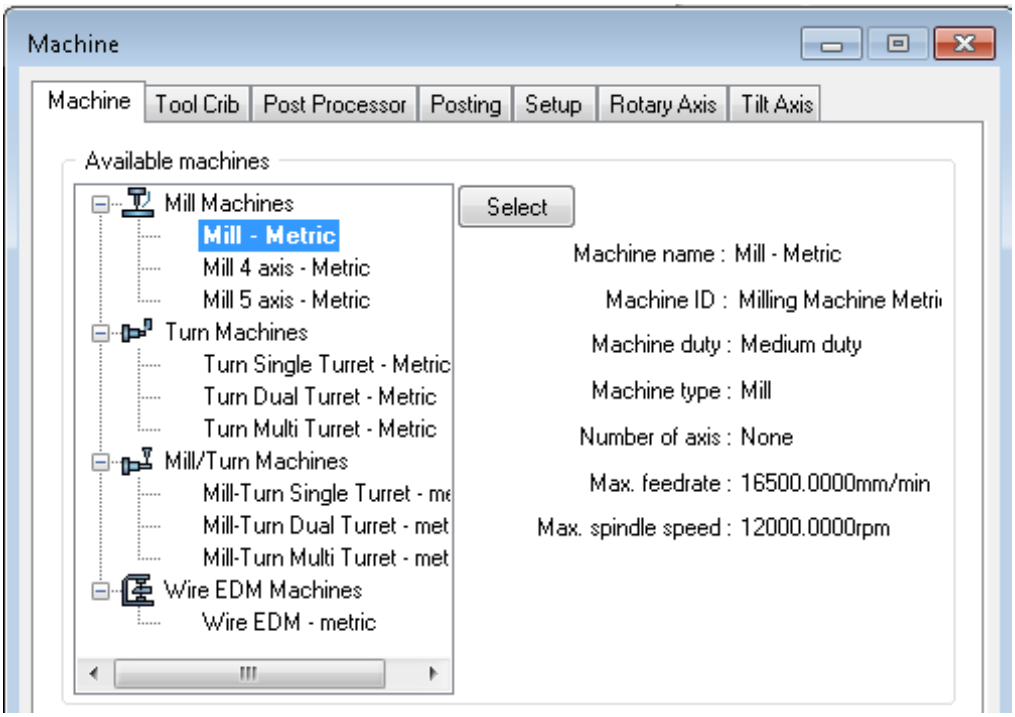
  An alternative machine can be selected at any time to output different G-code programs for alternative machine tools. If the machine type changes, then all features and operations will be deleted.
- **Tool crib:** A subset of tools from the tool library that are commonly loaded into or used with the current machine.
- **Post Processor:** The post processor identifies the format of the NC G-code output.

Following are the step to define the machine:

1. Right click *Machine [Mill–metric]* item in the CAMWorks Feature tree and select *Edit Definition* on the context menu.

   OR

   Double click the *Machine [Mill–metric]* item in the CAMWorks Feature tree.



Machine tab of Machine Dialog Box

Did You Know: **In the In the Feature and Operation trees, instead of right clicking items and selecting Edit Definition, you can double-click the item to open the dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.**

The Machine dialog box displays the Machine tab. The default machine is specified in the Technology Database. *Mill–metric* is the default machine used for the metric parts in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

Machine tools are set up in the Technology Database. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility. For more information, refer the PDF manual "***Technology Database Tutorial***".

2. In the Available machines list, highlight *Mill–metric* and click the *Select* button.
3. Click the *Tool crib* tab and highlight *Tool Crib 2 (Metric)* and click the *Select* button to define the Active tool crib.

   The Tool Crib page allows you to choose a Tool Crib, which is a set of tools or tool assemblies that are used with the machine you have chosen. These are not all the tools that

   are available, but a subset that you can modify to represent the actual set of tools that the machine has loaded.

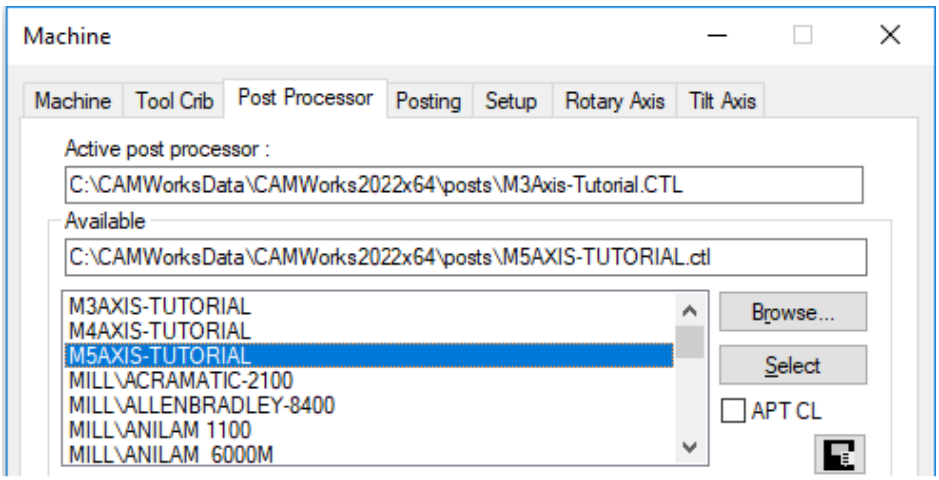
   When you define your machine tools in the Technology Database, you can set up your own tool cribs.
4. Make sure *Tool crib priority* option is unchecked.
5. Click the *Post Processor* tab.

   This tab allows you to select the internal post processor or the APT CL option to output a CL file. The list that displays depends on the post processors that are installed on your system.

   CAMWorks is supplied with several tutorial post processors. Contact your CAMWorks Reseller for more information on obtaining and/or customizing post processors for your machine tool.

   If the post processors do not display, use the *Browse* button to locate the folder containing the files (*.ctl).
6. If *M5AXIS-TUTORIAL* (the tutorial post processor) is not the Active post processor, highlight it in the list and click the *Select* button. This post processor is used for exercise in this manual. When you use CAMWorks to machine your own parts, you can select your machine tool controller or post processor.

   When you select this post processor, a short description displays in the window. This window contains information only if an optional file has been created for the post processor.



Post Processor tab of Machine Dialog Box

7. Click the *More* button.

   A longer description is displayed. The *More* button is activated only if a second optional file has been created. This information is intended for use in training or as a detailed description of post processor attributes that can be created.

   Information files are provided for the sample post processor that is used for the exercises in this manual. Your CAMWorks Reseller or your company manager may be able to supply these files if they are available for your post processor. If files are not available, you can create post information files as explained in the context-based Help.

8. Click the *Posting* tab.
   - The parameters on this page are used for the following:
   - To specify whether posting uses the coolant that is defined with the tool/insert or in the post processor.
   - To define whether the tool diameter and length offsets are set in the post processor or defined with the tool/insert.
   - To provide information required to generate the NC program. The parameters are machine-dependent and different parameters may display for your controller. The value for a parameter is output in the NC code if the machine requires it.
   - To provide information for the Setup Sheet, a file that is created when the NC program file is generated. All of the controller parameters are included in the Setup Sheet.
9. Click *OK* to close the Machine dialog box.

Define the Stock

The stock is the material from which the part will be machined. The default stock is the smallest cube *(bounding box)* that the part will fit into. Typically, this is not the size of the stock you will be using. You can change the stock definition either by offsetting the bounding box from the part or by defining the stock from a sketch and a depth *(extruded sketch)* or from an STL file.

1. Double click *Stock Manager* in the CAMWorks Feature tree.

   The Stock Manager dialog box is displayed.

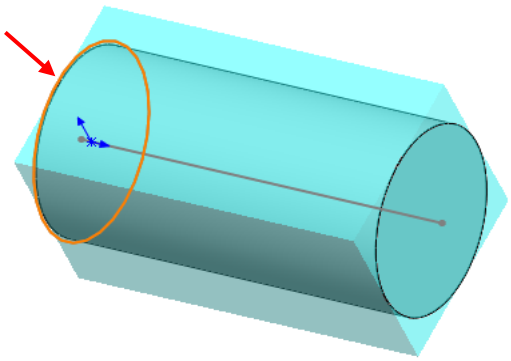
2. Under *Stock Type*, select *Extruded Sketch*.
3. Pick the circular sketch in the graphics area as shown in the image.

   The sketch is highlighted and the Stock is listed in the Selected entity field.

4. Set the Depth to **100mm**.

   The depth defines the distance the sketch is extruded.

5. Click *OK* to apply the changes and close the dialog box.



Pick the circular sketch

Inserting a Part Setup and Defining Multi Surface Features

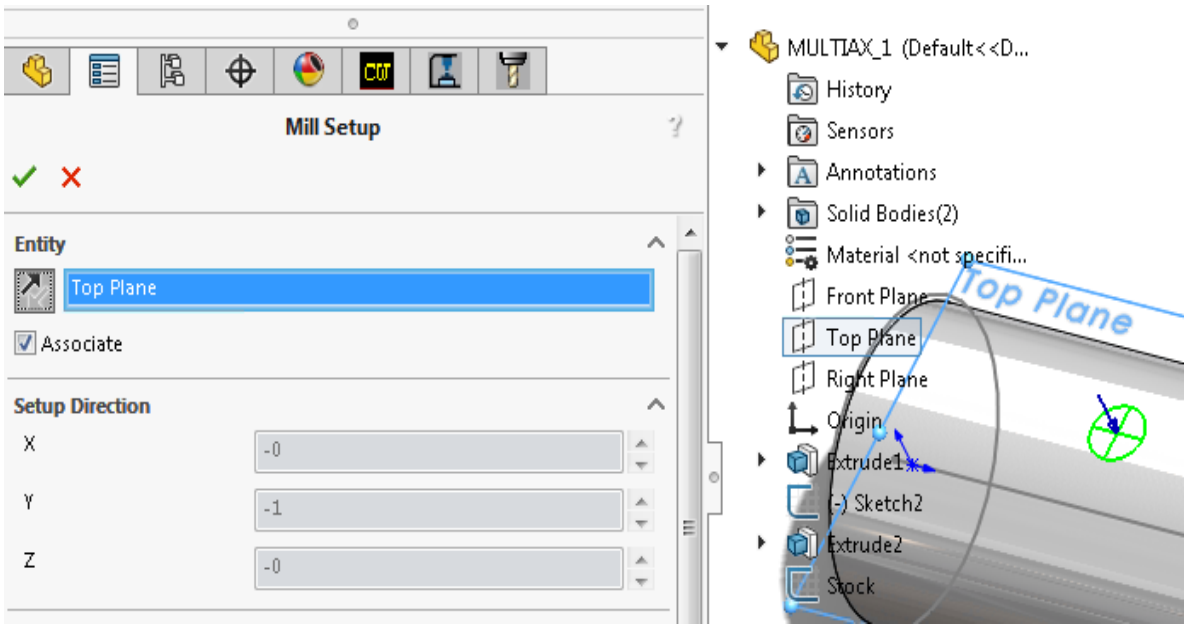
For parts machined with CAMWorks Multiaxis Machining, machinable features are defined interactively and it is necessary to insert a Mill Part Setup to machine these features from the required tool direction.

Inserting a Mill Part Setup

In this tutorial, you insert a Mill Part Setup and define a Multi Surface Feature.

1. Right click the *Stock Manager* in the Feature tree and select *Mill Part Setup* on the context menu.
2. In the graphics area, click on the (⊞) plus sign next to the *MULTIAX_1* to expand the tree.
3. Pick the *Top Plane* from references.
4.  Click the *Reverse Selected Entity* button in the Mill Setup dialog box.
5.  Click *OK* to insert the Mill Part Setup.

   The Mill Part Setup1 is inserted with the direction normal to the selected face/plane.
6. Click on the *Mill Part Setup1* in the Feature tree to view a marker on the part. The large arrow indicates the machining direction.



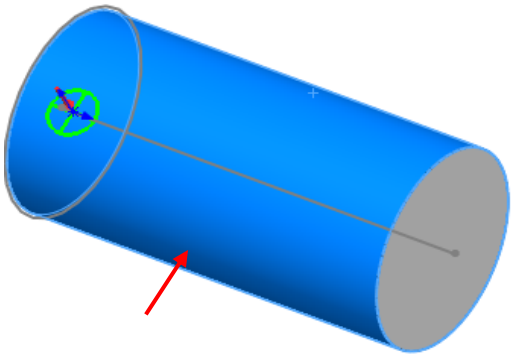
Inserting Mill Part Setup

Inserting a Multi Surface Feature

1. Right click *Mill Part Setup1* in the CAMWorks Feature tree and select *Multi Surface Feature* on the context menu.

   The Multi Surface Feature dialog box is displayed.

2. Set the Strategy to *5 Axis*.
3. Click in the *Selected Faces* list box to set the focus.
4. Pick the cylindrical face on the part model.



Pick the cylindrical face

Did You Know: **When picking faces in Multiaxis Machining, all faces will be machined in normal circumstances. You pick only the faces that need to be machined. Avoids are handled differently than 3 axis milling as you will learn in another exercise. In Multiaxis Machining, any faces that are selected to avoid in the Multi Surface Feature wizard are ignored.**

5. In the Selected Faces list box, check on the *Show normal*. Notice a direction of arrow displays on the cylinder.

   When generating 4 or 5 axis simultaneous toolpaths, the side of a surface to machine on is based on the surface normal. On faces from solid models, the surface normal is always pointing away from the solid body and the machining side will be correct. However, if machining a surface, the surface normal may not be pointing to the desired side to machine on. The Show normal option allows you to view and change the surface normal. The cylinder in this part is a solid model, so the machining side for the face of the cylinder is correct.

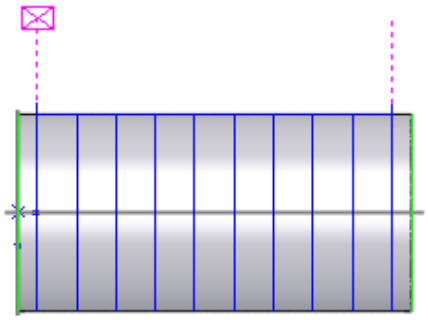
6.  Click *OK* to insert the multi surface feature.

   Multi Surface Feature1 is listed in the CAMWorks Feature tree.

Generating an Operation Plan

In the steps below, you are going to set up an operation to generate the toolpath with unidirectional (zigzag) passes that go around the cylinder with 10mm space between each toolpath (as shown in the image next page).

The Multiaxis Mill operations that are generated by CAMWorks are based on information stored in the Technology Database. These operations are intended to be used as a starting point and you can modify the machining parameters.



Unidirectional Zigzag passes

Did You Know: **The order of the tabs in the Operation Parameters dialog box is the recommended order for adjusting parameters. You may want to preview the toolpath after changing a parameter to see how the setting affects the toolpath. The Axis Control parameters can be set after you decide the toolpath is correct for your machining requirements.**

1.  Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

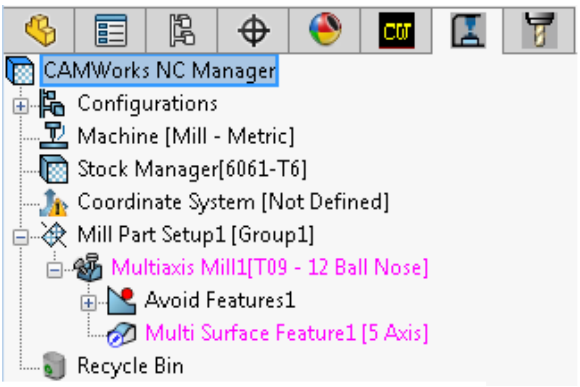
CAMWorks generates the operation.

OR

Right click on *CAMWorks NC Manager* in the Feature tree and select *Generate Operation Plan* on the context menu.

Note:

- If you execute the *Generate Operation Plan* command from the Command Manager or the CAMWorks NC Manager level, then operations will be generated for all prismatic features in the tree, regardless of the active item in the tree.
- If you execute the *Generate Operation Plan* command from the Mill Part Setup level, then operations will be generated only for those prismatic features listed under the given Setup.
- If you execute the *Generate Operation Plan* command at the feature level (by right-clicking on a feature listed in the Feature tree and executing *Generate Operation Plan* command from the RMB context menu), then operations will be generated only for the selected feature.



Operation Recognized after GOP

CAMWorks generates the operation plan for all the machinable features in Mill Part Setup1. The operations are listed in the CAMWorks Operation tree which displays automatically. The Operation tree can also be accessed by clicking the CAMWorks Operation Tree tab.

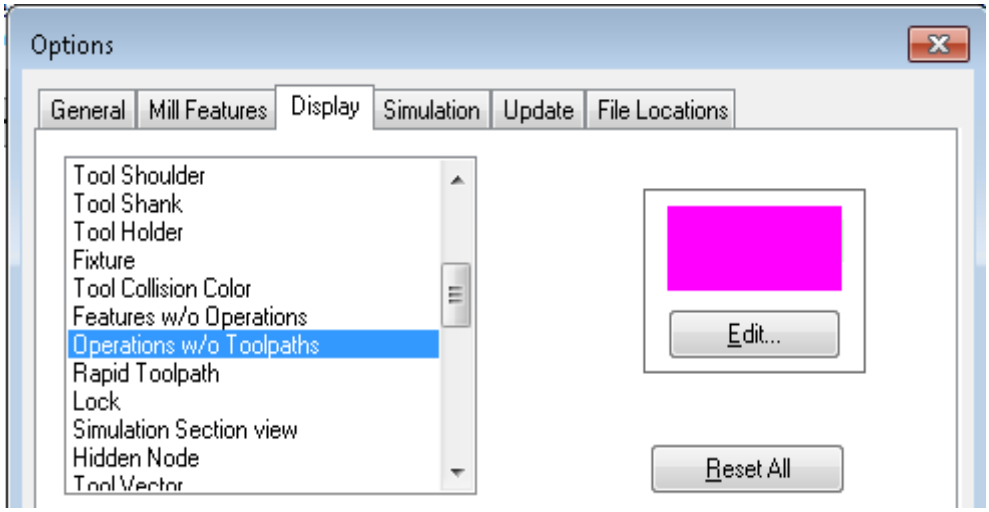
The CAMWorks Operation tree provides an outline view of the operations for the machinable features. Operations are listed under the Part Setup in the same order as the machinable features. At the top of the tree is the CAMWorks NC Manager. The Stock Manager and Machine items are the same as in the CAMWorks Feature tree. You can change the stock size and shape and the post processor used by CAMWorks to produce G-code.

The CAMWorks Operation tree allows you to:

- Insert, rename, suppress, and delete operations
- Change operation parameters
- Combine operations
- Sort operations
- Change the machining order
- Generate toolpaths
- Simulate toolpaths
- Post process the toolpaths
- Hide or show toolpath display
- Search based on item name

To the left of each toolpath operation is a plus sign (⊞). Clicking a plus sign displays the name of the Machinable Feature that this operation is going to machine. These Machinable Feature items can be used to view geometric information and to modify the machining depth of the feature.

Did You Know: **If an operation displays in a color other than black, then it indicates that toolpaths have not been generated for that particular operation. This occurs when you insert a new operation interactively, you insert a new feature interactively and generate operations for the new feature, or CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or a parameter is not correct. You can set the color for operations without toolpaths on the Display tab in the Options dialog box.**



Display tab of Options dialog box

2. Double click *Multiaxis Mill1* in the Mill Part Setup1.

   The Operation Parameters dialog box is displayed.

3. Click the *Tool Crib* page under *Tool* tab.
4. In the *Tool Crib* page, click the *Add* button to select a new tool from the Tool library.

   The Tool Select Filter dialog box is displayed.

5. Leave the Tool type set to *Ball Nose*.

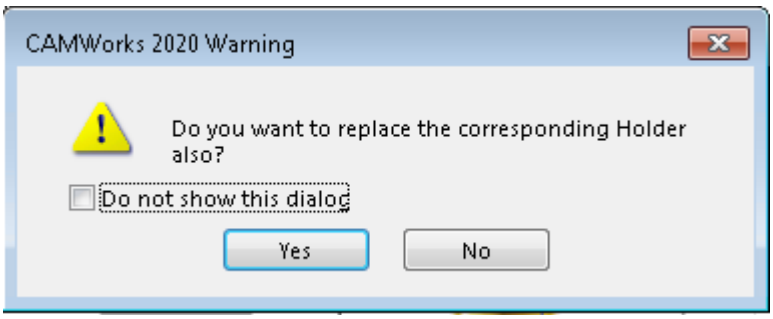
   The Mill (metric) list displays the tool list based on the selected tool type.

6. Select a **20mm** diameter tool on the list.
7. The Preview window is activated and displays the 3D view of the selected tool.
8. Click *OK* button.

   This action closes the dialog box and adds the selected tool to the active Tool crib. The new tool is listed at the bottom of the Tool Crib grid.

9. On the Tool Crib tab, highlight the **20mm** Ball Nose tool at bottom of the Tool Crib.
10. A Preview window is displayed at right side of the Operation Parameters dialog box. This Preview window contains the dynamic 3D view of the tool and holder. The tool Station Number, Tool Comment and labels of the Tool parameters are displayed in the Preview window.

11. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.



CAMWorks Warning Message

12. Click *Yes* to replace the corresponding holder.
13. Click the *F/S* tab.

    The F/S tab in the Multiaxis Mill Operation Parameters dialog box lists the parameters that affect the feeds and speeds that are output in the code.

14. Set the Defined by option to *Library* from dropdown list.

    When this option is selected, CAMWorks uses the values defined in the Feed/Speed library. The calculated feeds and speeds are updated automatically based on changes to the stock material, cutting conditions, or tool. When Operation is selected, CAMWorks uses the values defined on this tab and maintains any manually input feed and speed values even if the stock material, cutting conditions or tool change. When Tool is selected, CAMWorks uses the current values in the Cutting Parameters dialog box.

Slice Pattern

1. Click the *Pattern* tab in the opened Operation Parameters dialog box.

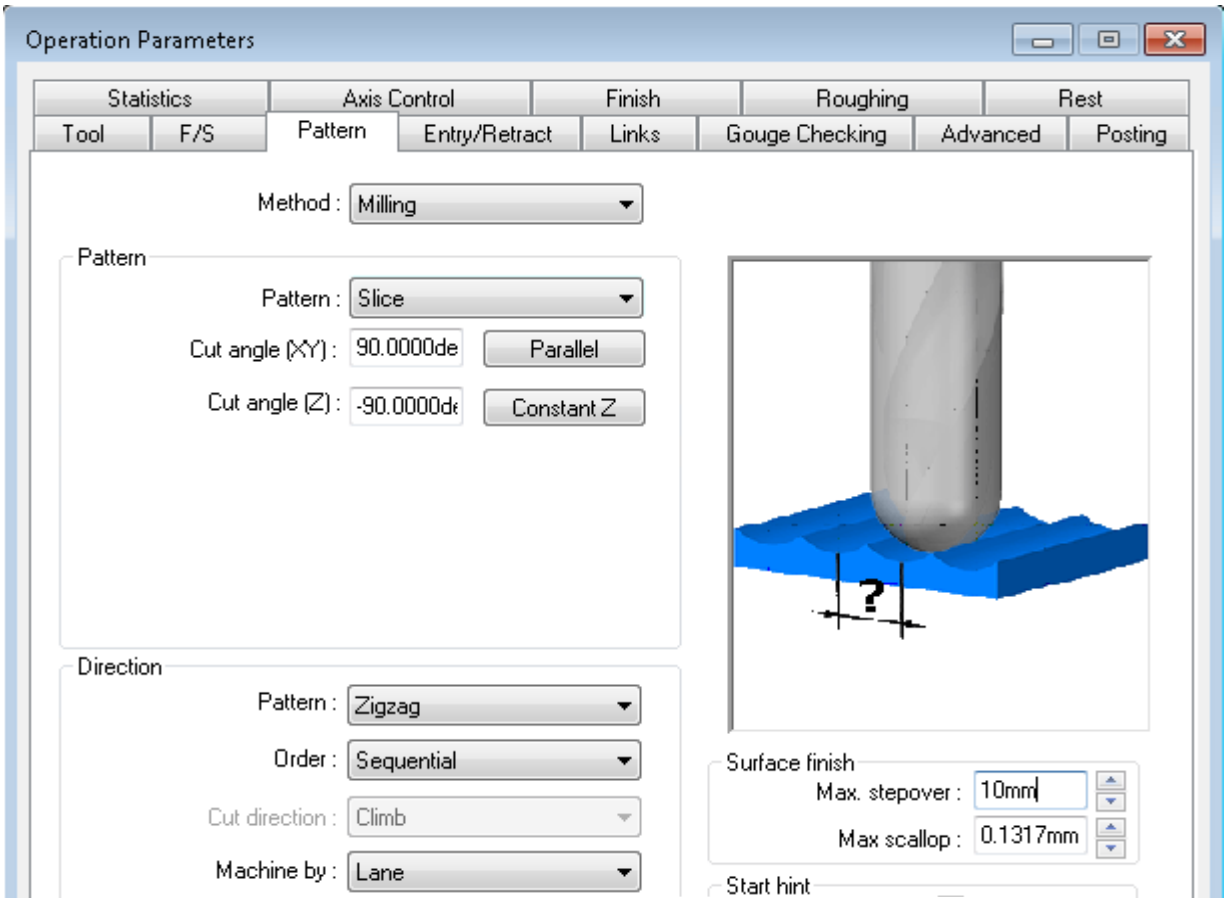
   This tab contains options to set the Pattern type, cutting order, cut limits and the surface quality.

2. Make sure Method is set to *Milling*.
3. Set the Pattern Type to *Slice*.

   The Slice pattern creates toolpaths that are parallel to each other. The direction of the cuts is defined by two Cut angles.

4. Set the Cut angle (XY) to **90deg** and the Cut angle (Z) to **-90deg**.

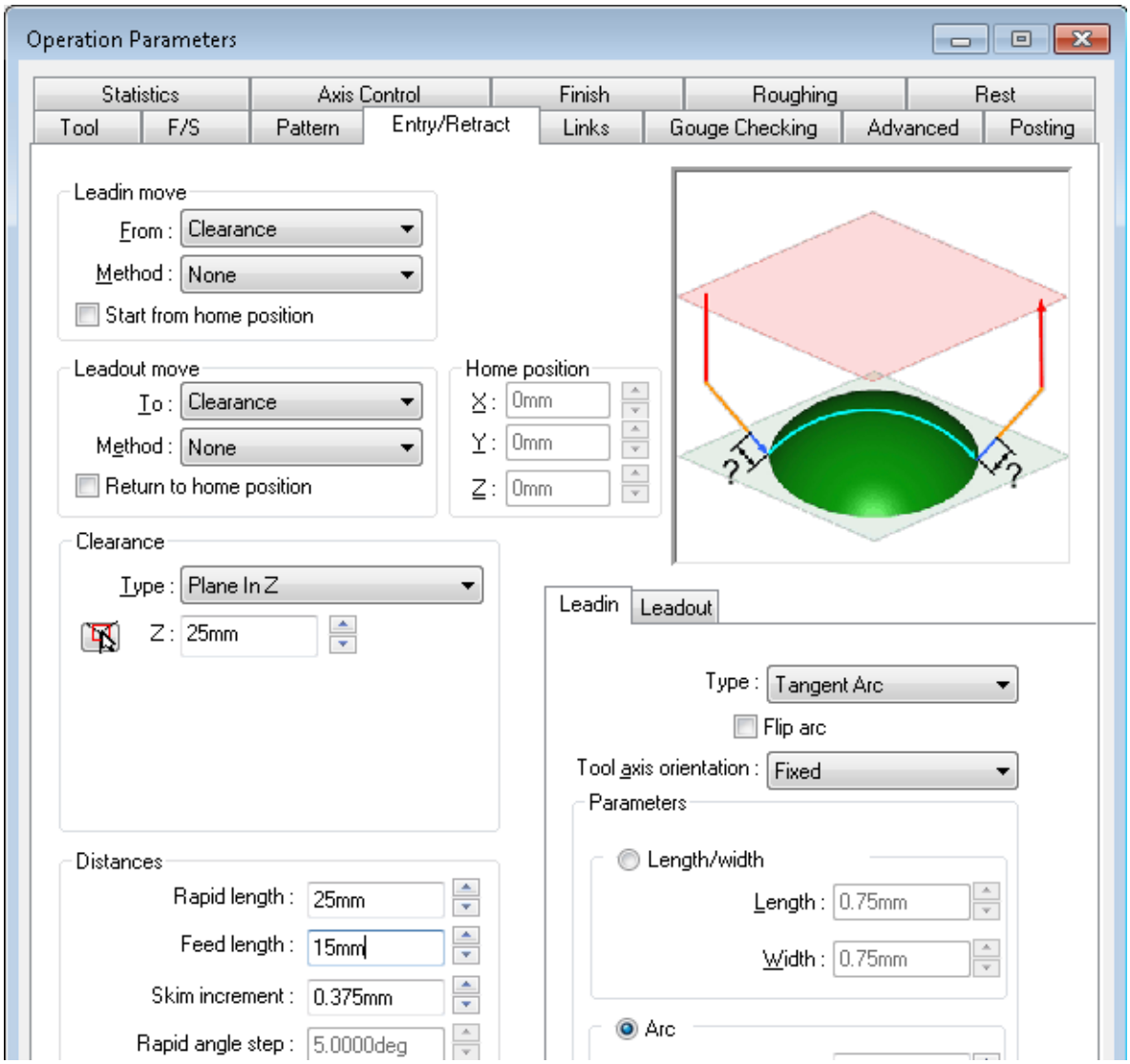
   The machining strategy is parallel (slice) cuts. The angle on XY plane is 90deg. The Angle in Z in this case is the angle in ZX plane. The setting of -90 deg means that motions will start at the right side and advance to the left. Generally, this angle will be done on a plane normal to the above set XY plane.



Pattern Tab on Operation Parameters dialog box

Did You Know: **When you place the cursor in the input box for each parameter, the graphic in the dialog box shows the effect of the parameter on the toolpath.**

5. Make sure the Direction is set to a *Zigzag* pattern.
6. In the Surface Finish group box, set the Max. stepover to **10mm**.
7. Click the *Entry/Retract* tab.
8. In the Leadin move and Leadout move group box, set the Leadin and Leadout Method to *None*.
9. In the Clearance group box, set the following parameters:
   - Type = **Plane in Z**
   - Z = **25mm**
10. In the Distances group box, set the following parameters:
    - Rapid length = **25mm**
    - Feed length = **15mm**



Entry/Retract tab on Operation Parameters dialog box

11. Click the *Links* tab.
12. In the Links between cut group box, set the Entry/Exit method options to *None.*
13. Click *OK* to apply the changes and close the dialog box.

Generating Toolpaths and Running Toolpath Simulation

1. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click on the *CAMWorks NC Manager* or *Mill Part Setup1* on the Feature tree and select *Generate Toolpath* on the context menu.

   Note:

   - Just like the Generate Operation Plan command, the scope of the Generate Toolpath command too depends from which level the command is executed.

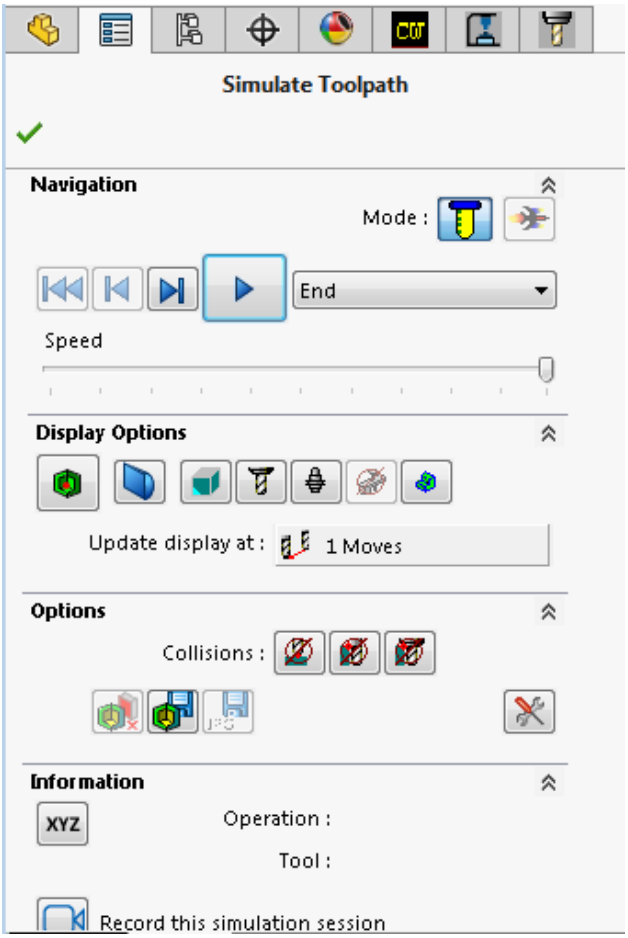
- Executing the *Generate Toolpath* command from the CAMWorks NC Manager level generates toolpaths for all the operations.
- Executing the *Generate Toolpath* command at Mill Part Setup level generates toolpaths only for operations listed under the given mill setup.
- You can also generate toolpaths for each operation individually by right-clicking on an operation and executing *Generate Toolpath* command on the context menu.

2. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

OR

Right click on the *CAMWorks NC Manager* or *Mill Part Setup1* on the Feature tree and select *Simulate Toolpath* on the context menu.

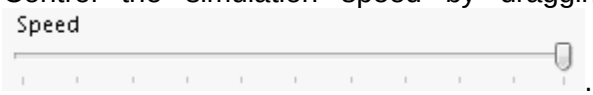
The Toolpath Simulation toolbar is displayed.



Simulate Toolpath Dialog Box

The toolbar controls allow you to:

- Run the simulation in *Tool* or *Turbo* mode.
- Display the simulated part, the design part and a comparison of the two by clicking on the *Show Difference* button.

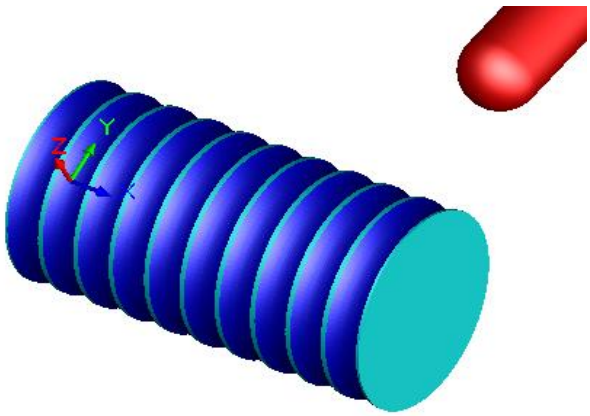
- Customize the display of the stock, tool and tool holder (*wireframe, translucent, shaded, or no display*) in *Tool* mode.
- Note that these options are disabled in Turbo mode.
- Run the simulation for all or selected operations.
- When simulating an operation, the simulation can be for the current operation or for all previous operations up to the selected operation.
- Pause the simulation using *Pause* button in both *Tool* and *Turbo* mode.
- Dynamically change the orientation of the part using zoom, pan, rotate, etc.
- Control the simulation speed by dragging the Simulation Speed Control slider 

.

If you want to simulate only the toolpath for a given operation, you can right click on that operation and select *Simulate Toolpath* in the context menu.

3. Set the simulation mode to *Tool mode* by clicking on the *Tool mode* button .
4. Click the *Run* button.

   Note that the material left between each pass is due to the large stepover used for this tutorial.



Simulation Mode

5. Click the *Pause* button to pause during simulation. Click the *Run* button to continue simulation.
6. Click the *OK* button to exit the simulation.

Adjusting Machining Parameters

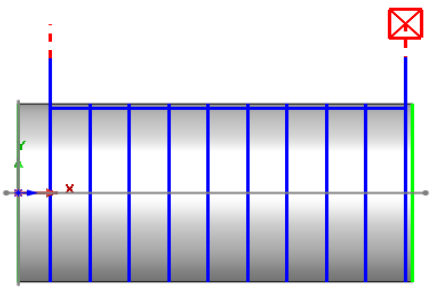
Cuts Across Curve Pattern

There is another Pattern type you can use to get the desired results without having to calculate the angles needed as in the previous example. Using this method, the cuts are done normal to a leading curve.

1. Double click *Multiaxis Mill1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
2. On the Pattern tab, select *Cuts Across Curve* for the Pattern Type.
3. Click the *Curve* button.

   The Curve Wizard dialog box is displayed.
4. Pick *Sketch2* from the Allowed Sketches list.



Generated toolpath after picking Sketch2

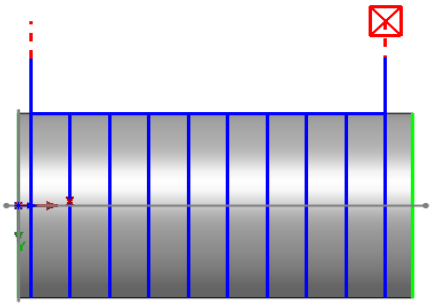
This action will display the selected Sketch2 in the Selected Contours list box.

5. Click *OK* to close the wizard dialog box.
6. In Limits group box, make sure Method is set to *Avoid Cuts at Exact Edges*.
7. Click the *Preview* button to generate and view the toolpath.

   When you click this button, the toolpath for the current operation is calculated and displayed immediately. This allows you to preview the toolpath for the operation based on the current parameters without having to close the dialog box and select Generate Toolpath. When you preview a toolpath, CAMWorks temporarily collapses the dialog box to show only the title and menu bars. Notice the gaps between the first and last toolpath passes and the cylinder edges. Also notice the start point that was randomly selected near the top end.

8. Click the *Close* button at the top right of the collapsed dialog box to restore the dialog box.
9. On the Pattern tab, change the Limits Method to *Start and End at Exact Surface Edge* in the Limit group box.
10. In the Start hint group box, check the *Use start hint* checkbox and set Y to **-25mm**.
11. Click the *Preview* button to view the toolpath.

    With the Limits set to Start and End at Exact Surface Edge, the toolpath is generated on the entire surface and exactly up to the surface edge or to the nearest possible position. In this case, the gap that remains on the left side is probably because of tolerance reasons, as the next setup has to be on the edge.

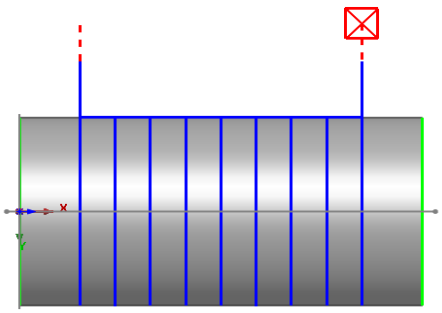


Generated toolpath after selecting Start and End at Extract Surface Edge limit method

    Notice the new start point. The definition of a start point is only a hint for CAMWorks. CAMWorks tries to find the nearest possible position next to your point.

12. Click the *Close* button to restore the dialog box.
13. On the Pattern tab, change the Limits Method to *Between Two Points* in the Limit group box.
14. Set the **-35mm** for the X on the left (the toolpath start point) and **35mm** for the X on the right (the end point).
15. Click the *Preview* button to view the toolpath.

    This option allows you to limit the machining between one or two points. Setting the 2 limit points caused the start to be -35mm from the right edge and the end to be 15mm from the left as coordinate system is default set to Top center. Note that the distance between the passes is equally spaced to ensure smooth results.



Generated toolpath after selecting Between Two Points limit method

16. Click the *Close* button to restore the dialog box.
17. Notice the variety of parameters that can be set to control the toolpath. For this exercise, the defaults on this tab can be used.

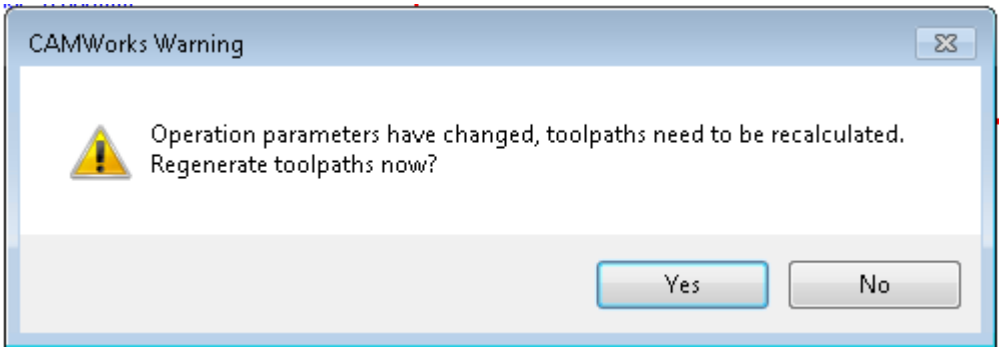
Defining the Tool Orientation

The Axis Control tab allows you to define the tool orientation relative to the surface normal. To get a good finish with a 5 axis machine, the tool should have a specific angle to the surface. This is called the tilt angle. The tilt angle can be a fixed angle or a variable angle based on the current normal vector of the surface.

1. Click the *Axis Control* tab in the opened Operation Parameters dialog box.
2. In the Cutting direction tilt options group box, set the Tool axis will be to *Normal to Surface.*
3. Click *OK* to apply the changes and close the dialog box.
4. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

   Toolpaths will be regenerated.

5. If any warning message is not displayed, click the *Generate Toolpath* button on the CAMWorks Command Manager.



Warning message to regenerate toolpath

Step Through Toolpath

CAMWorks also provides the ability to visualize the tool motion and verify tool positions using the Step Thru Toolpath command.

6. **Executing the Step Thru Toolpath command:** Both the *Simulate Toolpath* and *Step Thru Toolpath* commands can be executed at the global, setup and operation levels.

   - **Global level:** To step through the toolpaths of all the operations in the Operation tree, right click on the *CAMWorks NC Manager* in the Operation tree and select *Step Thru Toolpath* command in the context menu.

     OR

     Select *Step Thru Toolpath* command on the Command Manager.
   - **Setup Level:** Right-click on the *Mill Part Setup1* and select *Step Thru Toolpath* command in the context menu.
   - **Operation level:** Right-click on the desired operation in the Operation tree and select *Step Thru Toolpath* in the context menu.

   On execution of the Step Thru Toolpath command using one of the above methods, the *Step Thru Toolpath* dialog box is displayed.

7. Use controls given in the *Display Options* group box to set the options as to how the toolpath will be displayed during the Step Through process.

8.  Set the Tool to *Wireframe Display* in the Display Options group box.

9.  Click the *Forward Single Step* button to step through the toolpath, then hold down the space bar for continuous movement.

   Notice that the tool axis is not tilted. The tool axis direction equals the face normal at touch point.

10. Alternatively, click the *Play* button  to continuously back-plot the tool till the end condition specified in the *Play Tool Until* field is reached.

11. During animation, the *Play* button display changes to *Pause* button  . Click the *Pause* button to pause the animation while it is in progress.

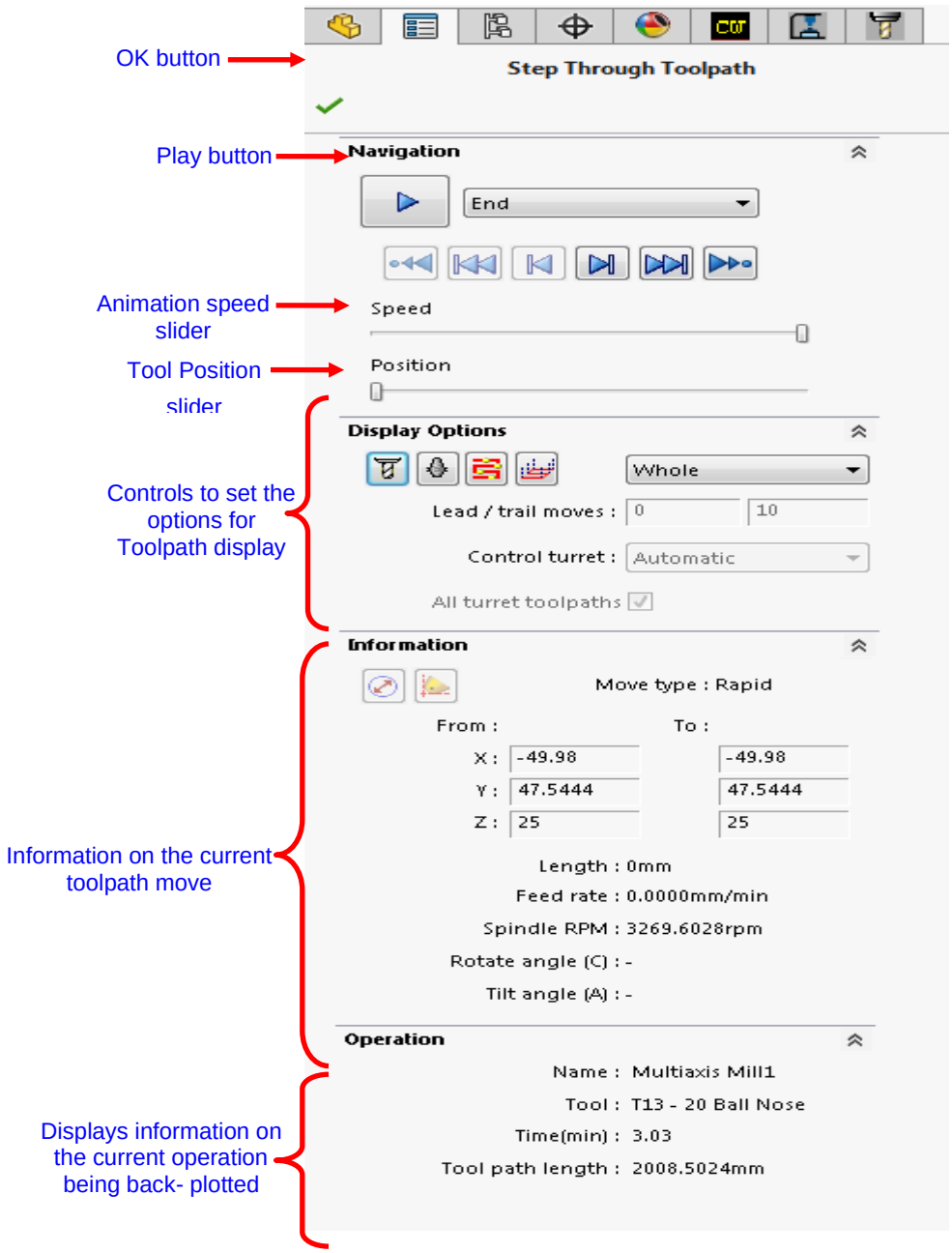
12. When the animation has not yet begun or when it is paused, the *Forward Single Step, Goto End of Current Toolpath* and *Goto End* buttons are active.

    - Use the *Forward Single Step* button  to move one toolpath record forward each time the button is clicked.

    - Use the *Goto End of Current Toolpath* button  to move the tool position to the last toolpath record of the current operation.

    - Use the *Goto End* button  to move the tool position to the last toolpath record of the very last operation.

Note: **The last operation varies depending on which level the Step Through Toolpath command is executed from. For example, if this command was executed from the Mill Setup level, then the last operation would be the last operation under that particular setup.**



Step Through Toolpath dialog

13. When the animation has been completed or when it is paused, the *Reverse Single Step, Goto Start of current Toolpath* and *Goto Start* buttons are active.

    - Use the *Reverse Single Step* button  to move one toolpath record backwards each time the button is clicked.

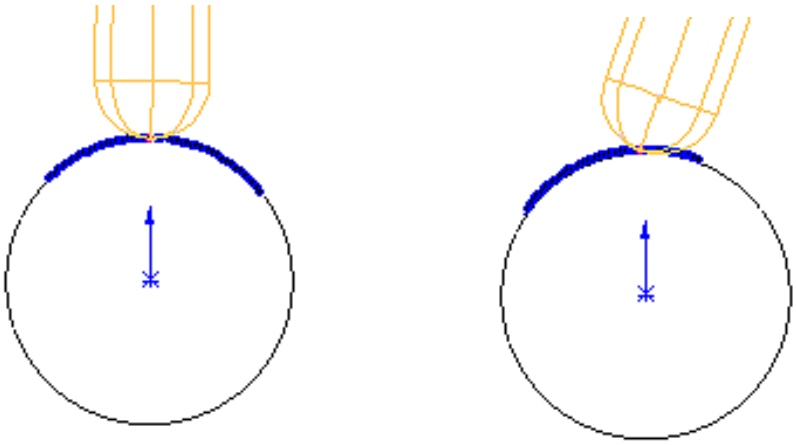
    - Use the Goto Start of Current Toolpath button  to move the tool position to the first toolpath record of the current operation.

    - Use the *Goto Start* button  to move the tool position to the first toolpath record of the very first operation.

14. If you are unsure about the use of any parameter within this dialog box, click on the *Help* button at the upper right corner. This action will display the context-based help for Step Through Toolpath command.
15. Click the *OK* button in the upper left corner to close the dialog box.
16. Double click *Multiaxis Mill1* in the tree.
17. On the Axis Control tab, set the Tool axis will be to *Tilted Relative To Cutting Direction*.
18. Set the Lead/lag angle to **15deg**.

    With this option, you can define a Lead/lag angle to the cutting direction.

    - Positive value = Tool leans to the movement direction
    - Negative value = Tool leans away from the movement direction



No Lead angle **Lead angle**

19. Click *OK* to apply the changes and close the dialog box.
20. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

    Toolpaths will be regenerated.
21. Right click the *Multiaxis Mill1* in the tree and select *Step thru Toolpath* again on the context menu.

    Observe the changes and step through the toolpath.
22. Double click *Multiaxis Mill1* again in the tree.
23. On the Axis Control tab, set the Tool axis will be to *Tilted With Fixed Angle To Axis*.

    With this option, the tool axis is tilted in a fixed angle relative to the chosen axis. The tilt axis can be to the X, Y and Z axis or to any line created in the geometry. Tilt axis and surface normal build a plane in which the tool tilts.
24. Change the Tilt axis to *X-Axis*.
25. Set the Fixed tilt angle to **45deg**.
26. Click *OK* to apply the changes and close the dialog box.
27. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

    Toolpaths will be regenerated.

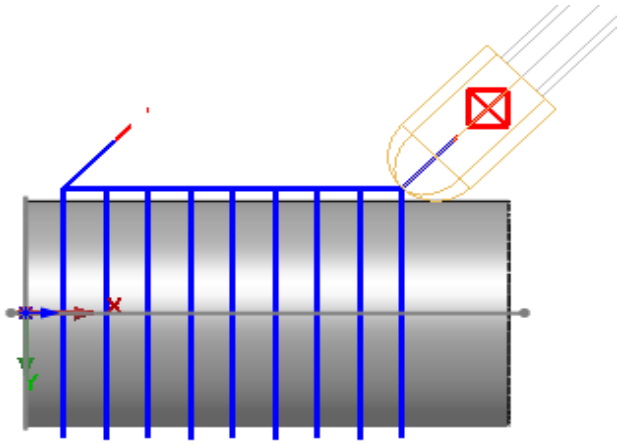
28. Right click the *Multiaxis Mill1* in the tree and select *Step thru Toolpath* again on the context menu.

29.  Set the Tool to *Wireframe Display* in the Display Options group box.

30.  Click the *Forward Single Step* button to step through the toolpath.

    The tool is now tilted towards X+ in a constant angle of 45degrees all along the TP. Notice that the CC (touch) point on the surface is maintained and one of the toolpaths is moved due to the tilt. The CL point is now moved.

31.  Click the *Close* button to cancel Step Thru Toolpath.



Generated toolpath

Post Processing Toolpaths

Post processing is the final step in generating the NC program file. When you use a CAMWorks internal post processor, this step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the CAMWorks Operation tree. When you post process a part, CAMWorks creates two files: the NC program and the Setup Sheet. These are text files that you can read, edit and print using a word processor or text editor.

Did You Know: **For more information on generating an enhanced APT CL file that can be used by external post processing programs, see the Context-based Help.**

In this tutorial, you will post process all the operations and generate the NC program:

1.  Click the *Post Process* button on the CAMWorks Command Manager.

   OR

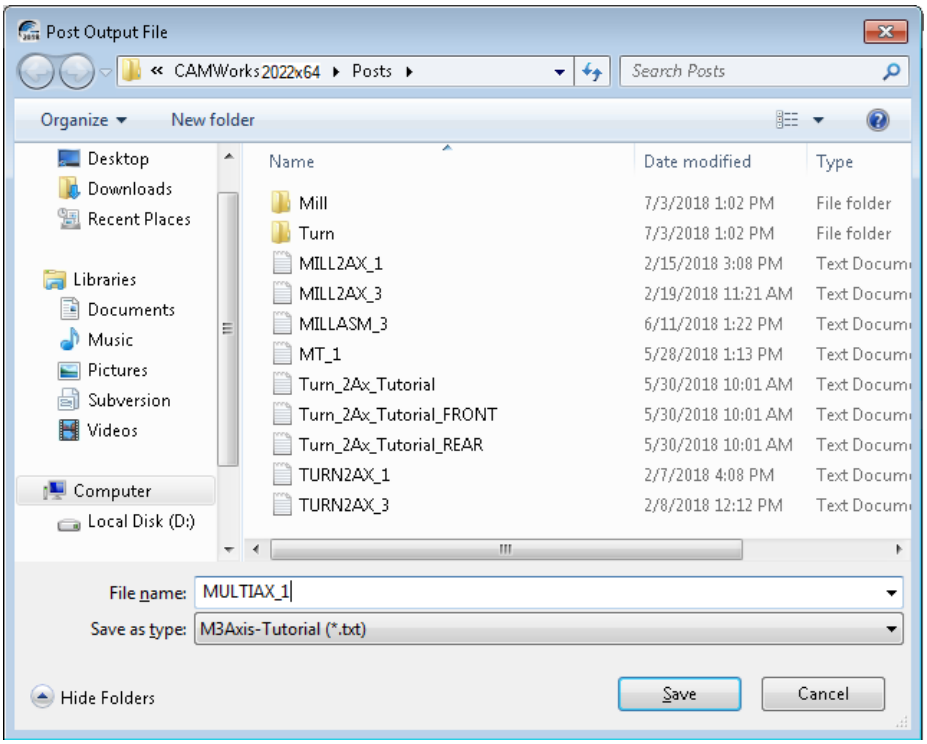
   Right click on the *CAMWorks NC Manager* in the Operation tree and select *Post Process* on the context menu.

   The Post Output File dialog box is displayed so you can name the NC program file. Typically, the NC program and Setup Sheet files are stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder location.

Note: **If the Post Process command is grayed out on the CAMWorks Command Manager or on any context menu, make sure that you have selected a post processor and generated the toolpaths.**

2. In the *Post Output File* dialog box, click the down arrow to the right of the *Save as type* box.

   CAMWorks provides a list of commonly used extensions that you can select. For this exercise, use the *.txt* extension.



Post Output File dialog box

Note: **If you want change the default extension from .txt to one of the ones in the list or if you want a different file name extension for NC program files, you can edit or create a .pinf file and specify the new extension. For more information on making these changes, see the context-based Help.**

3. In the *File name* textbox, type the suitable file name, and then click *Save* button.

4. The *Post Process* dialog box is displayed. Click the *Step* button  on the control bar at the top.

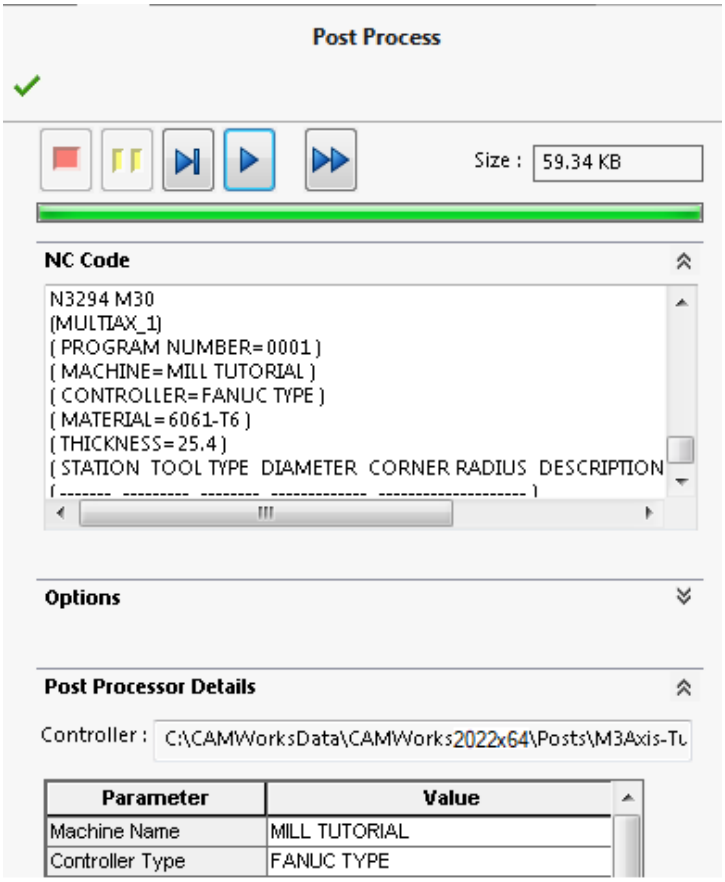
   CAMWorks starts to generate the NC program and the first line of NC code displays in the NC code view box. The post processing mode is set to post process on line of code at a time (Step mode).

5.  Click the *Step* button. The next line of NC code is displayed.

6.  Click the *Run* button. Post processing continues until it is completed.

7. When the post processing is finished, view the code using the vertical scroll bar.

8. Click *OK* to close the dialog box.



Post Process dialog box

Multiaxis 2

Topics covered in this tutorial:

- Using Collision Detection options
- Controlling the Start and End of the Toolpath

Opening the Part and Adjusting the Parameters

1. Open the part file **MULTIAX_2.sldprt** in the following folder.

   Drive:\CAMWorksData/CAMWorks202x\Examples\Tutorial_Parts\4-5AxisMill

2. Click the *CAMWorks Feature Tree* tab.

   A Multi Surface feature has already been defined.

3. Double click *Multi Surface Feature1* in the Feature tree.

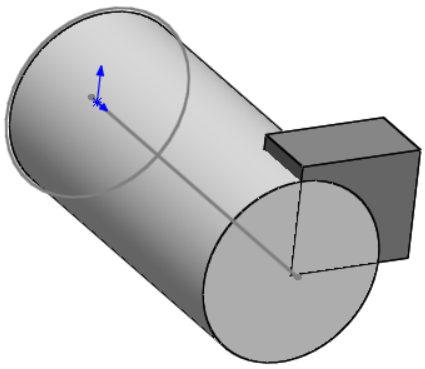
   OR

   Right click on *the Multi Surface Feature1* in the tree and select *Edit Definition* in the context menu.

   The Multi Surface Feature dialog box is displayed.

4. In the Feature type group box, make sure the *Define as Avoid Features* option is unchecked.

   For Multiaxis Milling operations, you pick only the faces that need to be machined. Avoids are handled differently than 3 axis milling as you will learn in this tutorial. Any faces that are selected to avoid in the Multi Surface Feature dialog box are ignored.



MULTIAX_2.sldprt

5. Set the Strategy to *5 Axis*.

6. Click *OK* to close the dialog box.

Generating Operation Plan and Adjusting Operation Parameters

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click on the *Mill Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

   The generated operation is displayed in the Operation tree.

2. Double click *Multiaxis Mill1* in the Operation tree.

3. Click the *Tool Crib* page under Tool tab.

4. In the *Tool Crib* page, click the *Add* button to select a new tool from the Tool library.

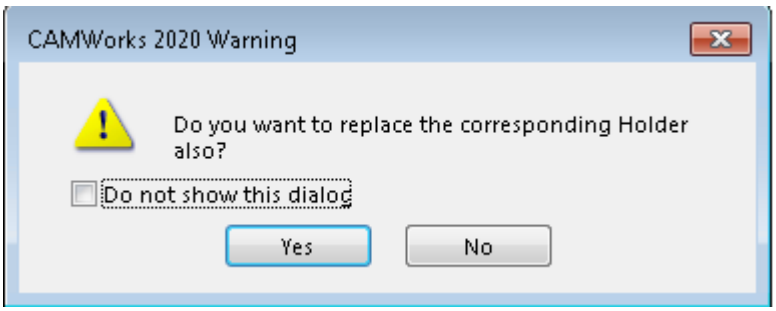
   The Tool Select Filter dialog box is displayed.

5. Leave the Tool type set to *Ball Nose*.

The Mill (metric) list displays the tool list based on the selected tool type.

6. Select a **20mm** diameter tool on the list.
7. The Preview window is activated and displays the 3D view of the selected tool.
8. Click *OK* button.

   This action closes the dialog box and adds the selected tool to the active Tool crib. The new tool is listed at the bottom of the Tool Crib grid.
9. On the Tool Crib tab, highlight the **20mm** Ball Nose tool at bottom of the Tool Crib.
10. A Preview window is displayed at right side of the Operation Parameters dialog box. This Preview window contains the dynamic 3D view of the tool and holder. The tool Station Number, Tool Comment and labels of the Tool parameters are displayed in the Preview window.
11. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.



CAMWorks Warning Message

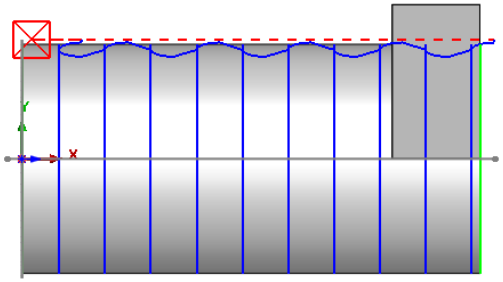
12. Click *Yes* to replace the corresponding holder.
13. Click on the *Pattern* tab.
14. Set the Pattern to *Cuts Across Curve* in the Pattern type group box.
15. Click the *Curve* button. The Curve Wizard dialog box is displayed.
16. Pick *Sketch2* from the Allowed Sketches list.

    This action will display the selected *Sketch2* in the Selected Contours list box.
17. Click *OK* to close the Curve Wizard dialog box.
18. In the Surface Finish group box, set the Max. stepover to **10mm**.

Using Collision Detection Options

The Gouge Checking tab contains options to prevent the tool from gouging the surfaces to cut and selected surfaces that are not going to be machined by the current operation. Gouge checking looks at the generated toolpath and the surfaces to decide whether the tool components are gouging the surfaces.

1. Click on the *Axis Control* tab.
2. In the Cutting Direction Tilt Options group box, set the Tool axis will be to *Normal to Surface*.



Generated toolpath has ignored the dark gray face

3. Click the *Close* button in the collapsed dialog box to restore the dialog box.
4. Click the *Gouge Checking* tab.
5. In the Group1 page of Gouge Checking group box, check the box next to *Apply gouge checking to.*

   This action will enable the other check boxes on the Group1 page.
6. Make sure the *Non-cutting portion* and *Flute* options are checked.

   This option enables the first of four different groups of options for gouge checking that can be set up to check for different collision conditions.
7. In the Check against group box, remove the check mark from the *Feature surfaces* option.
8. Select the *Other surfaces* option and click the *Browse* button.

   The Avoid Features dialog box is displayed. You can identify the features to check for collisions. There are two methods for adding avoid features in a Multiaxis Mill operation.

   - You can create an avoid feature before you generate operations. In this case, the feature would be in the list box.

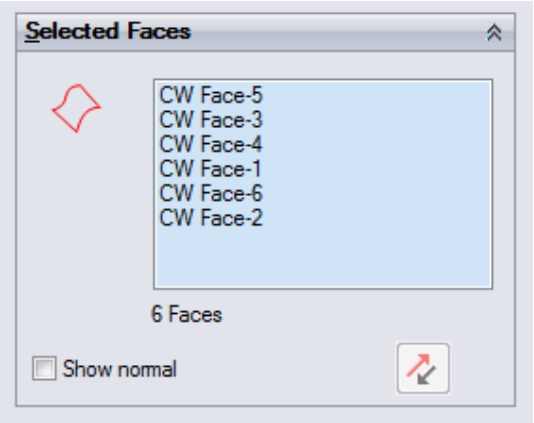
     OR

   - You can click the *Create Features* button in this dialog box and pick the faces required to create the feature.
9. Click the *Create Features* button.

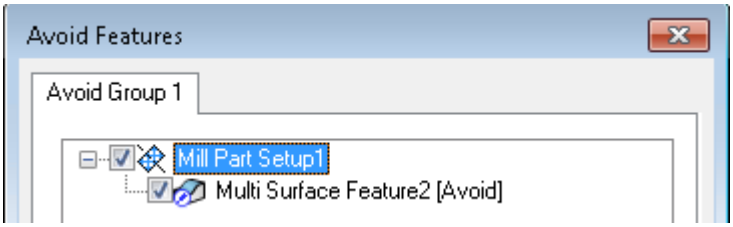
   The Multi Surface Feature dialog box is displayed.
10. Pick each face on the dark gray box. If required, rotate the part to pick the faces.

    The selected 6 faces are listed in the Selected Faces list box.
11. Click *OK* to insert the changes.

    Multi Surface Feature2 is now listed in the Avoid Features dialog box.
12. Make sure the check box next to the feature is checked and click *OK*.



Selected 6 faces



Multi Surface Feature is listed in Avoid Features dialog box

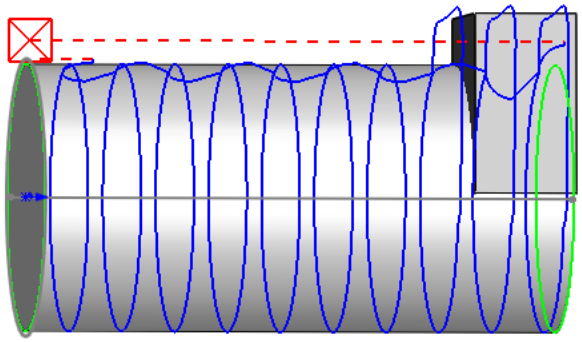
The Operation Parameters dialog box is again displayed.

13. In the Gouge check options group box of Gouge Checking tab, set the Strategy to *Retract Along Tool Axis*.

With this option selected, when a gouge is detected, the tool will retract along its axis to a position where it no longer gouges.

14. Click the *Preview* button and view the results.

    The avoid faces were machined. At all times, the tool orientation is kept normal to the cylinder.

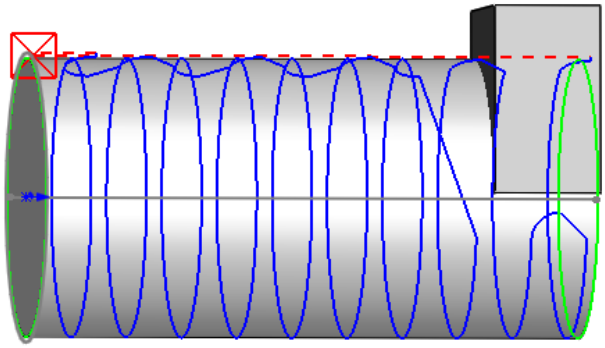


Machine the avoid faces

15. Click the *Close* button to restore the dialog box.
16. In the Gouge check options group box, change the Strategy to *Remove Gouged Positions.*
17. In the Check against group box, set the Allowance to **2mm**.
18. Click *Preview* and view the results.

    The avoid faces are bypassed with some rectangular motions.

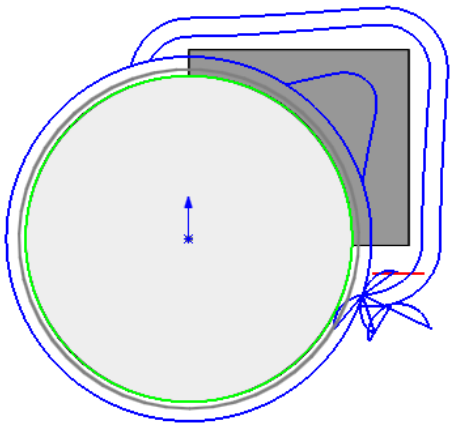
19. Rotate the part to view the offset distance from the avoid surfaces.
20. Click the *Close* button to restore the dialog box.
21. Click on the *Pattern* tab.
22. In the Direction group box, check the *Reverse stepover* option.



Bypass the avoid faces

23. In the Limits group box, set the Method to *Start and End at Exact Surface Edge*.
24. Click on the *Axis Control* tab.
25. In the Cutting Direction tilt options group box, set the Tool axis will be to *Tilted With Fixed Angle to Axis*.
26. Set the Tilt axis to *X-Axis* and set the Fixed tilt angle to **45deg**.
27. Click on the *Gouge Checking* tab.
28. In the Gouge check options group box, set the Strategy to *Move Tool Away*.
29. Set the Retract tool to *Retract tool opt in YZ plane*.
30. Click the *Preview* button and view the results.

    Notice that the machining starts from the other side. In order to prevent the gouging of the avoid faces, the tool is moving away from the geometry. All the movements

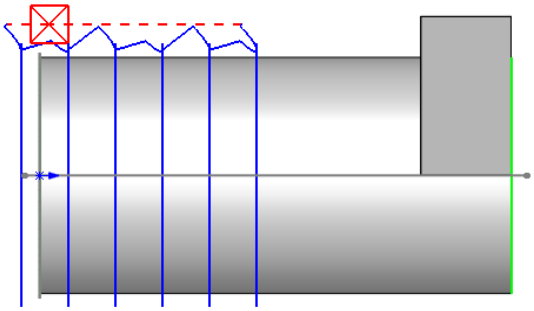


Prevent the gouging

are done on the YZ plane, maintaining a constant X value.

31. Click the *Close* button to restore the dialog box.
32. In the Gouge check options group box of Gouge Checking tab, set the Strategy to *Stop Toolpath Calculation.*
33. Click *Preview* button and view the result.

    CAMWorks generates a partial toolpath. The calculation stopped when a gouge was detected. Only the non-gouging motions calculated before the gouge are generated.



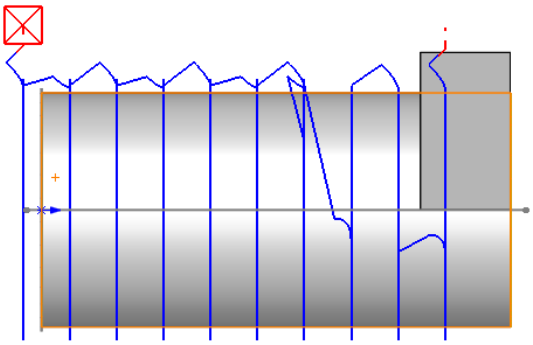
Generated partial toolpath

Controlling the Start and End of the Toolpath

The Entry/Retract tab options control how the toolpath is started and ended (Leadin and Leadout).

1. Click the *Close* button to restore the dialog box.
2. In the Gouge check options group box of the Gouge Checking tab, set the Strategy to *Remove Gouged Positions*.
3. Click the *Entry/Retract* tab and set the following parameters in the Clearance group box:
   - Clearance = *Cylinder About X* (the clearance shape is a 40mm radius cylinder with the center passing though the origin point)
   - Radius = **40mm**
4. In the Distances group box,
   - Rapid length = **5mm** (the last rapid motion before the first feed motion)
   - Feed length = **3mm** (the last approach motion toward the material)
   - Skim increment = **10mm**
5. Click *Preview* and view the results.

   The rapid motions bypassing the avoid surfaces have a cylindrical shape. Notice that the created cylindrical motions are done in a greater radius than defined. CAMWorks calculates the minimal safe distance needed in order to avoid gouging any geometry. The search for a new safe height is done in steps of 10mm (the Skim increment).

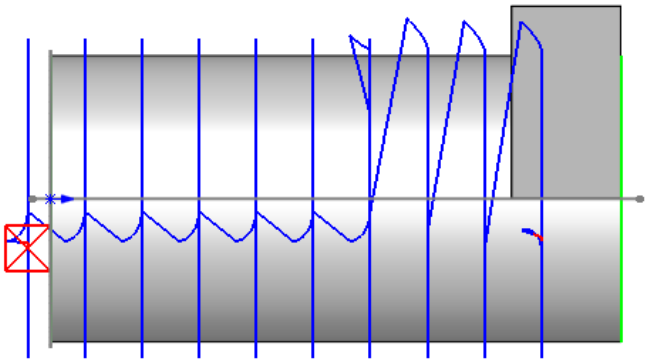


Bypassing the avoid faces and generated cylindrical motion

6. Click the *Close* button to restore the dialog box.

7. Click the *Pattern* tab and change the Direction pattern to *Zig.*
8. Check the *Use Start hint* in the Start hint group box and set the Rotate next cut by option to **10deg**.
9. Click the *Entry/Retract* tab.
10. Set the Leadin move Method to *Use Leadin* and the Leadout move Method to *Use Leadout*.
11. In the both Leadin and Leadout pages, set the following same parameters:
    - Type = *Tangent arc*
    - Parameters = **Arc**
    - Arc angle = **90deg**
    - % Tool diameter = **50**
    - Height = **0.001mm**
12. Click the *Preview* button and view the results.

    Notice the tangent approach and retract. The shifted start point is the result of rotating the next cut by 10deg on the Pattern tab.



Tangent approach

13. Click *OK* to apply the changes and close the Operation Parameters dialog box.
14. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

    Toolpaths will be regenerated.

2. LEARNING MORE ABOUT MULTIAXIS

This chapter provides an opportunity to learn more about CAMWorks Multiaxis Machining.

The tutorials in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

The tutorial parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\4-5AxisMill* folder on your computer. This folder is inside the CAMWorks data folder.

(Drive:*CAMWorksData**CAMWorks202x**Examples**Tutorial_Parts**4-5AxisMill*)

Before you begin with the tutorials given in this chapter, ensure that you are through with the concepts explained in Chapter 1.

IMPORTANT! **CAMWorks uses a set of knowledge based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.**

Multiaxis 3

Topics covered in this tutorial:

- Defining Tool Movement when Gaps are Encountered
- Using Links with Gouge Checking

CAMWorks provides numerous settings on the Links tab that allow you to define the tool movement when gaps are encountered along cuts, between cuts and between multiple passes. This exercise uses the options for gaps along cuts.

Opening the Part and Defining the Machine

1. Open the part file **MULTIAX_3.sldprt** in the following folder.

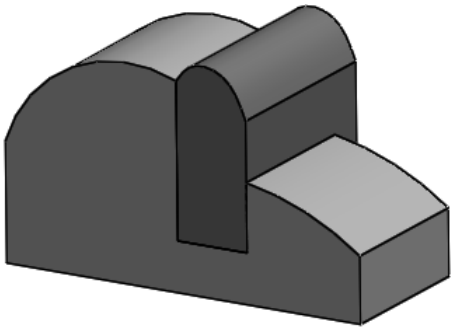
   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\4-5AxisMill

2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine* item in the Feature tree.

   OR

   Right click the *Machine* in the Feature tree and select *Edit Definition* on the context menu.

   - On the Machine tab, highlight *Mill–inch* and click on *Select* button.
   - Click the *Tool Crib* tab and highlight *Tool Crib 2 (Inch)* and click Select button to define he Active tool set.
   - Make sure *Tool crib priority* option is unchecked.
   - Click the *Post Processor* tab; select *M5AXIS-TUTORIAL* as Active post processor.
   - Click *OK* to apply the changes and close the dialog box.



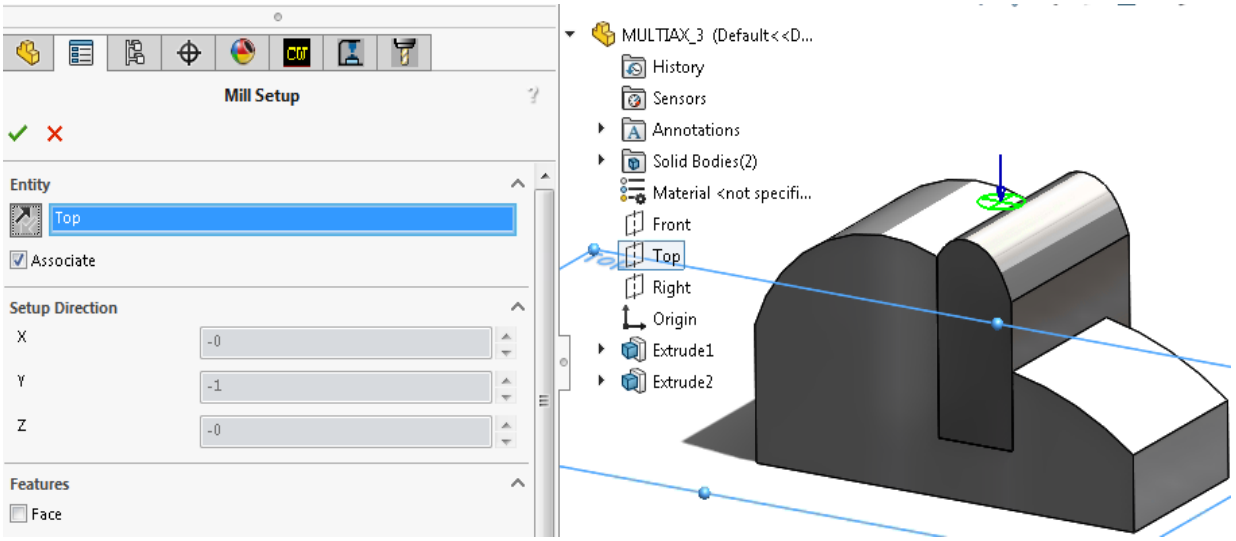
MULTIAX_3.sldprt

4. Double click *Stock Manager* in the Feature tree to open the Stock Manager dialog box.
   - Leave the Stock type set to *Bounding box.*
   - Leave *Material* set to default.
   - Click *OK* to close the dialog box.

Inserting Mill Part Setup

1. Right click *Stock Manager* in the tree and select *Mill Part Setup* on the context menu.
2. In the graphics area, click on the (⊞) plus sign next to the *MULTIAX_3* to expand the tree.
3. Pick the *Top Plane* from the references.
4. Click the *Reverse Selected Entity* button in the Mill Setup.
5. Click *OK* to insert the Mill Part Setup.

The Mill Part Setup1 is listed in the tree.

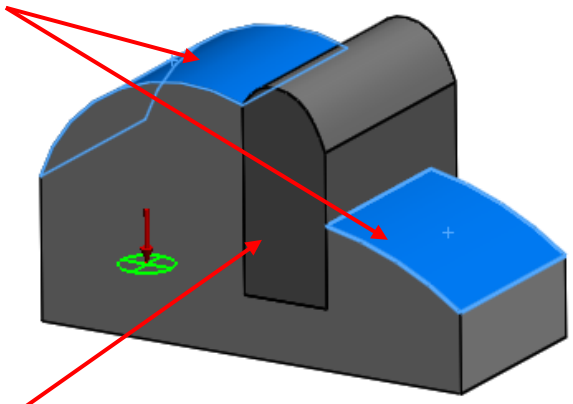


Pick the Top Plane

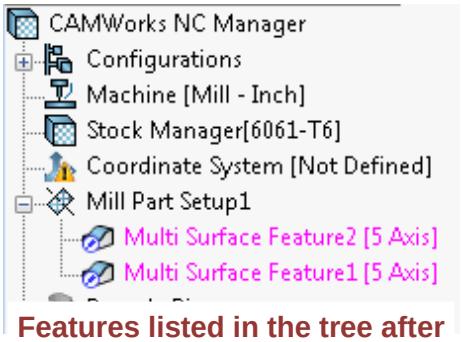
6. Right click *Mill Part Setup1* in the Feature tree and select *Multi Surface Feature* on the context menu.
7. Pick the two top gray faces as shown in the image (*highlighted in blue*).

   The Faces-1 and Face-2 are listed in the Selected Faces list box.
8. Set the Strategy to *5 Axis*.
9. Click *OK* to insert the feature.
10. Right click *Mill Part Setup1* in the Feature tree and select *Multi Surface Feature* on the context menu.
11. Pick the 6 faces on the dark gray solid part as shown in the image. Rotate the part if required to pick all the faces.
12. Set the Strategy to *5 Axis*.
13. Click *OK* to insert the multi surface feature.

    Multi Surface Feature1 and Multi Surface Feature2 are listed under the Mill Part Setup1 in the Feature tree.
14. Right click on the *Multi Surface Feature2* in the tree and select the *Rename* on the context menu.
15. In the place of Multi Surface Feature2, rename to *Avoid Feature* as shown in the image.
16. Click the *SOLIDWORKS FeatureManager design tree* tab.

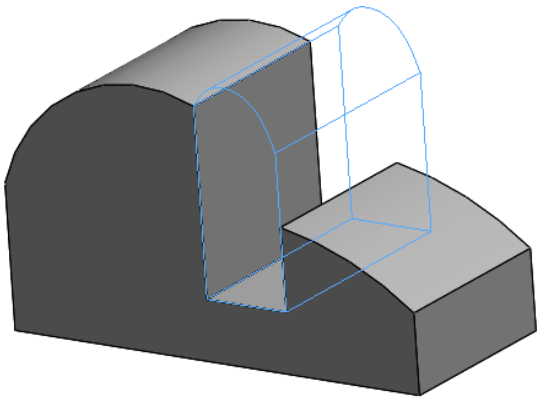


Pick the two Top faces



Features listed in the tree after Renaming

17. Right click *Extrude2* and select the *Hide* command from the context menu.
18. In the graphics area, observe that the dark gray solid part is removed from the model part. You will enable collision detection in the next part of this tutorial.



Hide the Extrude2 Sketch

19. Click the *CAMWorks Feature Tree* tab.
20. Right click *Multi Surface Feature1* in the Feature tree and select *Generate Operation Plan* on the context menu.

Defining Tool Movement When Gaps are Encountered

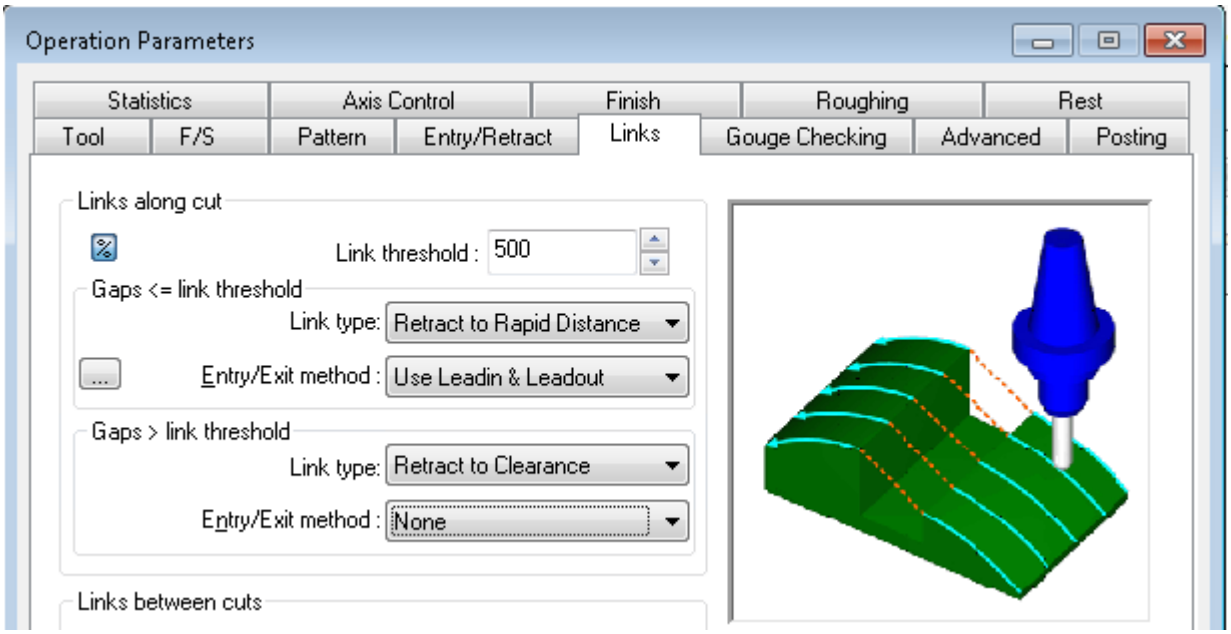
1. Double click *Multiaxis Mill1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
2. Make sure Method is set as *Milling*.
3. Click on the *Pattern* tab and set the following parameters:
   - Pattern = **Slice**
   - Cut angle (XY) = **-180deg**
   - Cut angle (Z) = **-90deg**
   - Max stepover = **0.15in** in the Surface finish group box.
4. Click on the *Entry/Retract* tab.
5. In the Clearance group box, set the Clearance Type to *Plane in Z* and set Z to **2in**.
6. In the Distances group box, set the following parameters:
   - Rapid length = **0.15in**
   - Feed length = **0.05in**
7. In the Parameters group box,
   - Check the *Arc* option in both Leadin and Leadout page.
8. Click on the *Links* tab.
9. In the Links along cut group box, set the Link threshold to **500**.

   The Link threshold sets the threshold for gaps along a toolpath segment as an absolute value or as a percent of the tool diameter. Gaps along the toolpath segment that are smaller than or equal to this value are processed according to the settings for Gaps <= link

threshold. Gaps that are larger than this value are processed according to the Gaps > link threshold.

10. For Gaps <= link threshold, set the following parameters:
    - Link type = *Retract to Rapid Distance*
    - Entry/Exit method = *Use Leadin & Leadout*
11. For Gaps > link threshold, set the following parameters:
    - Link type = *Retract to Clearance*
    - Entry/Exit method = *None*

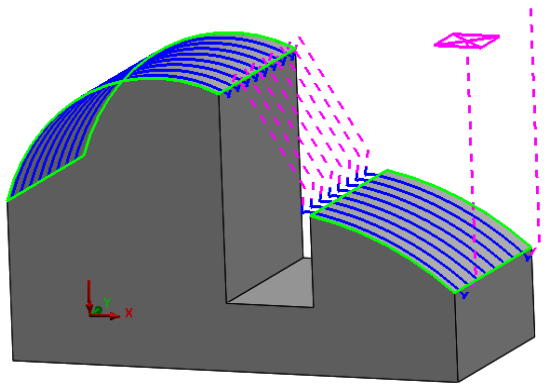


Links tab on the Operation Parameters dialog box

12. Click the *Preview* button.

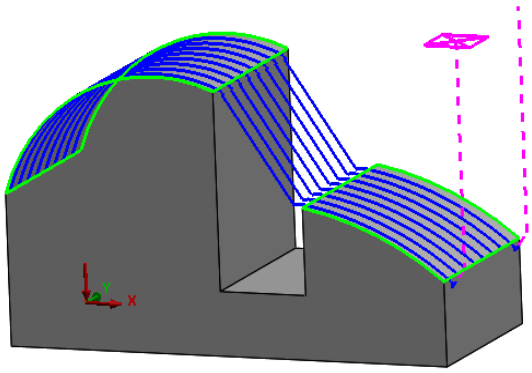
    With the Retract to Rapid Distance option, when a gap is detected, the tool retracts to the rapid distance. The retracting direction is the tool axis. The tool rapids from the surface and moves over to the next toolpath point with machining speed.

13. Click the *Close* button at the top right to restore the collapsed dialog box.
14. In the Links along cut, change the Link type for Gaps <= link threshold to *Direct*.
15. Click the *Preview* button.



Retract to Rapid Distance Link Type

With this option, the tool uses the shortest path to the other side of the gap without any retracting movements. The toolpath in the gap is a straight line and the tool moves in machining speed.

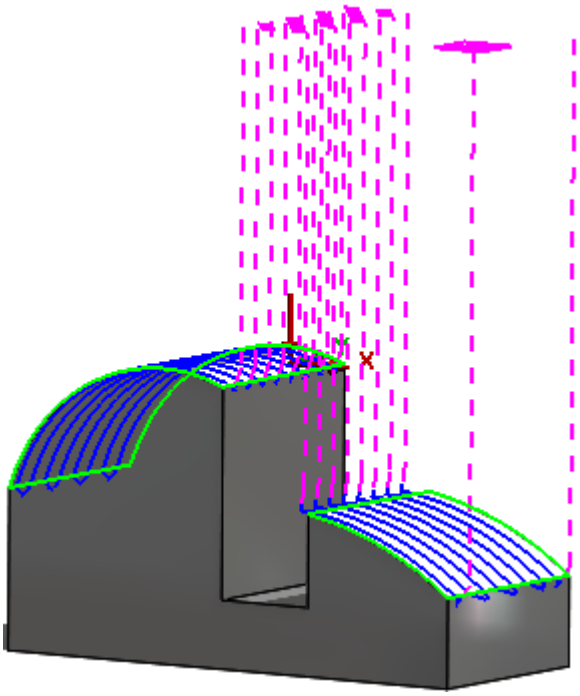


Direct Link Type

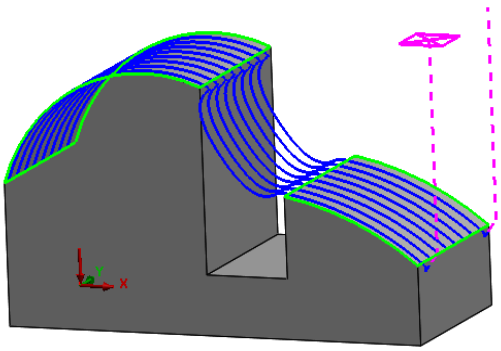
16. Click *Close* to restore the dialog box.
17. In the Links along cut, change the Link type for Gaps <= link threshold to *Retract to Clearance*.
18. Click the *Preview* button.

    Notice that the toolpath retracts to the Clearance area. The tool rapids back to the Clearance plane. Only the return to the surface has machining speed.
19. Click *Close* to restore the dialog box.
20. Now, in the Links along cut, change the Link type for Gaps <= link threshold to *Blend Spline*.
21. Click *Preview* button.

    The Blend spline option connects the surfaces with a toolpath that leaves and enters the surfaces tangentially. The result is a very smooth connection even on edgy gaps.
22. Click *Close* to restore the dialog box.



Retract to Clearance Link Type



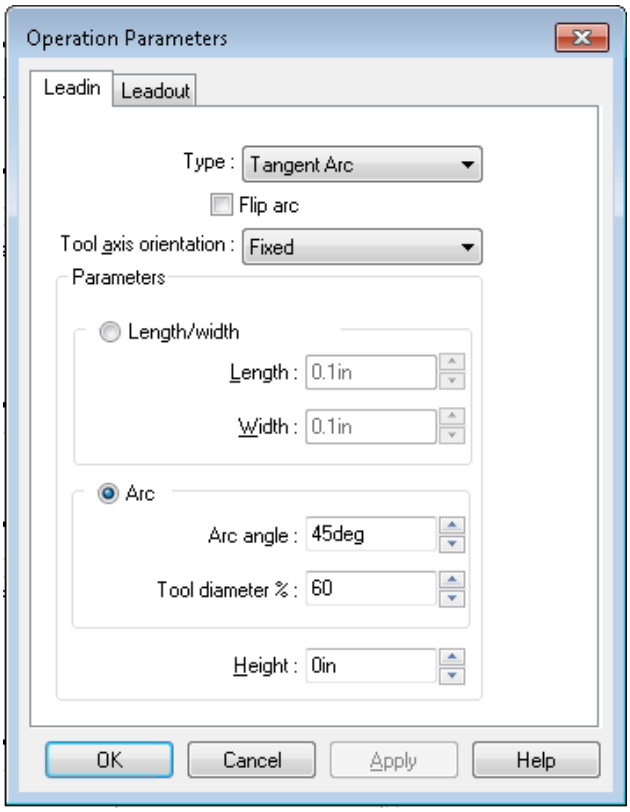
Blend Spline Link Type

In this tutorial, you will set the Entry/Exit method to Use Leadin & Leadout. The type of leadin and leadout can be changed.

23. In the opened Operation Parameters dialog box, click the *Browse* button to the left of the Entry/Exit method for Gaps <= link threshold.

    The Operation Parameters dialog box is displayed to set the Leadin and Leadout parameters.

24. Select a *Leadin* and *Leadout* Type as you desired and set the parameters.
25. Click *OK* to close the dialog box.



Operation Parameters dialog box

26. Click *OK* to close the Operation Parameters dialog box and regenerate the toolpath.

    Notice the difference in the leadin and leadout.

27. Double click *Multiaxis Mill1* in the tree and repeat these steps selecting a different type and view the difference in the toolpath.

Using Links with Gouge Checking

1. Click the *SOLIDWORKS FeatureManager Design* tree tab.
2. Right click *Extrude2* and select *Show* icon.
3. Click the *Operation tree* tab.
4. Double click *Multiaxis Mill1* in the Operation tree to open the Operation Parameters dialog box.
5. On the *Links* tab, in the Links along cut group box, set the Link type to *Follow Surfaces* for Gaps <= link threshold.

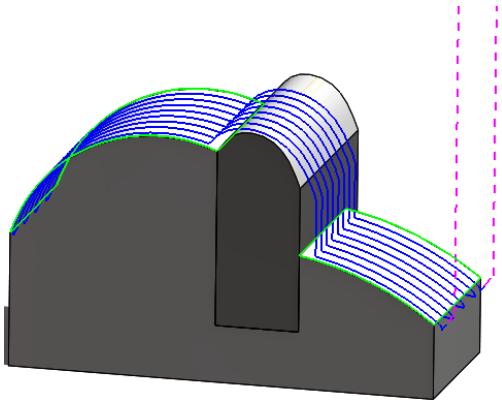
6. On the *Gouge Checking* tab, select the *Apply gouge checking to* option on the Group 1 page.
7. Check *Non-cutting portion* and *Flute* options under the Gouge checking group box.
8. In the Gouge check options group box, set the Strategy to *Retract Along Tool Axis*.
9. In the Check against group box, remove the check mark from the *Feature surfaces* option.
10. Select the *Other surfaces* option and click the *Browse* button .

    The Avoid Features dialog box is displayed.
11. In the dialog box, click *Create Features*.

    This button allows you to define additional Multi Surface features if necessary.
12. In the *New Multi Surface Feature* dialog box, pick all the faces of the dark gray color part. Rotate the part if required to pick the faces.

    All the selected faces of the part are added to the selected faces list box.
13. Click *OK* to insert the feature.

    The Avoid Features dialog box is displayed again.
14. In the dialog box, click *Select All.*
15. Click *OK* to close the dialog box.
16. Click the *Preview* button and notice that the toolpath avoids the dark gray surface.



Toolpath avoids the dark gray surface

Multiaxis 4

Topic covered in this tutorial:

- Machining Undercut Areas with 3 Axis Cutting

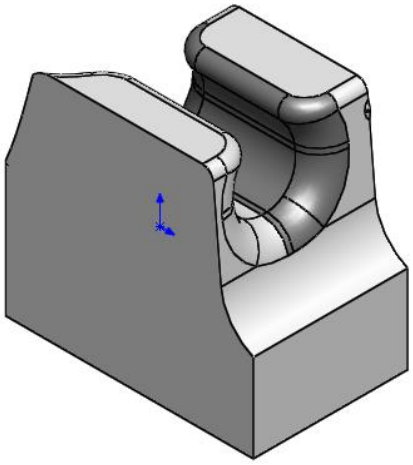
CAMWorks allows you to take advantage of keyway and lollipop cutters to machine complex forms and access any undercut areas without the need for 4- and 5-axis indexing or tool tilting.

Opening the Part and Defining the Machine and Stock

1. Open the part file **MULTIAX_4.sldprt** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\4-5AxisMill

2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine* item in the Feature tree to open the *Machine* dialog box.
   - On the Machine tab, highlight *Mill–inch* and click *Select* button.
   - Click the *Tool Crib* tab and select the *Tool Crib 2 (Inch)* as Active tool set.
   - Make sure *Tool crib priority* option is unchecked.
   - Click the *Post Processor* tab; make sure *M5AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.
4. Double click *Stock Manager* in the tree to open the *Stock Manager* Dialog box.
   - Leave the Stock type set to *Bounding box.*
   - Leave the *Material* set to default.
   - Click *OK* to apply the changes and close the dialog box.



MULTIAX_4.sldprt

Inserting Mill Part Setup and Multi Surface Feature

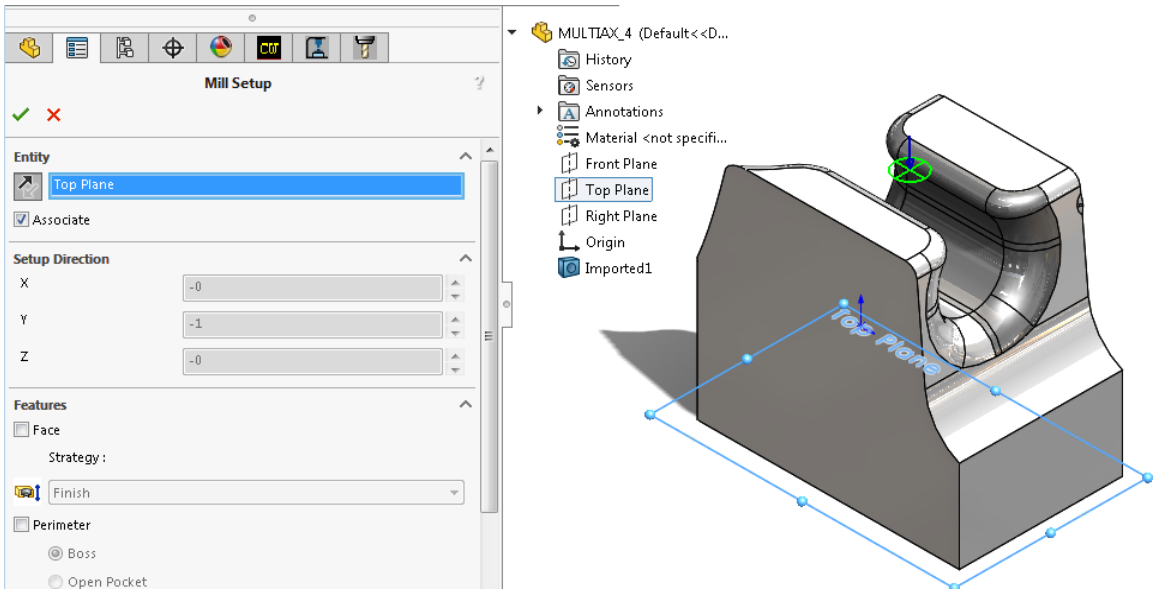
Insert Mill Part Setup

1. Right click *Stock Manager* in the tree and select *Mill Part Setup* on the context menu.

   The Mill Setup dialog box is displayed.
2. In the graphics area, click on the plus sign next to the *MULTIAX_4* to expand.
3. Pick the *Top Plane* from the references.
4. Click the *Reverse Selected Entity* button.

5. Click *OK* to insert the Mill Part Setup.

   The Mill Part Setup 1 is listed in the Feature tree.

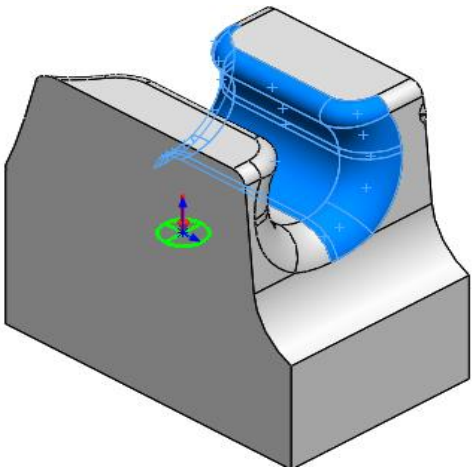


Pick the Top Plane

Insert Multi Surface Feature

1. Right click *Mill Part Setup1* in the tree and select *New Multi Surface Feature* on the context menu.
2. Set the Strategy to *5 Axis*.
3. Pick all the dark gray faces on the model part (highlighted in dark blue color after selection).
4. Click *OK* to insert the multi surface feature.

   Multi Surface Feature1 is inserted in the Feature tree.



Pick all the faces

Generate Operation Plan and Adjusting the Parameters

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   CAMWorks generates the operation and listed under the Operation tree.
2. Double click *Multiaxis Mill1* in the Operation tree.
3. Click the *Tool Crib* page on the Tool tab and click the *Add* button.

The Tool Select Filter dialog box is displayed.

4. Select the Tool type to *Lollipop*.

   The Mill (Inches) list will update and display the tool list based on the selected tool type.

5. Select a tool from the list with **ID no. 1** (0.125in diameter).
6. The Preview window is activated and displays the 3D view of the selected tool.
7. Click *OK* button.
8. On the Tool Crib tab, highlight the lollipop tool at bottom of the Active Tool Crib.
9. A Preview window is displayed at right side of the Operation Parameters dialog box. This Preview window contains the dynamic 3D view of the tool and holder.
10. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.
11. Click *Yes* to replace the corresponding holder.
12. On the Lollipop Tool page on the Tool tab, change the Shank diameter (D2) to **0.15in** and the Diameter (D1) to **0.4in**.
13. Click the *Pattern* tab and set the Pattern to *Flowline Between Curves*.

    This pattern creates swarf cuts between two curves and can be used to machine steep areas for mold making.

14. Click the *Upper* button.

    The Curve Wizard dialog box is displayed.

15. Make sure the Selection mode is set to *Single Face* and *Single Edge*; then pick the top edges of the feature.
16.  Click *OK* to apply the changes.

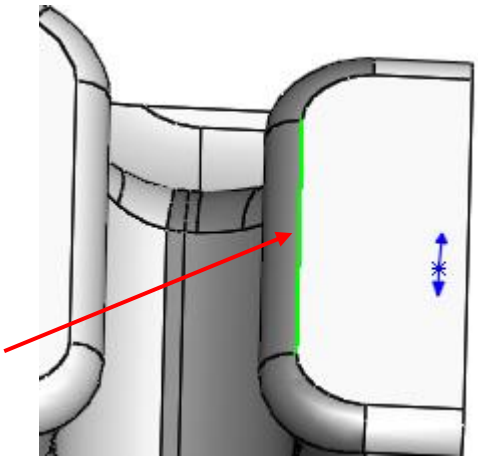
    Edge<1> is listed in the Selected Contour field and the Operation Parameters dialog box redisplays so you can insert additional features.

17. Click the *Lower* button.

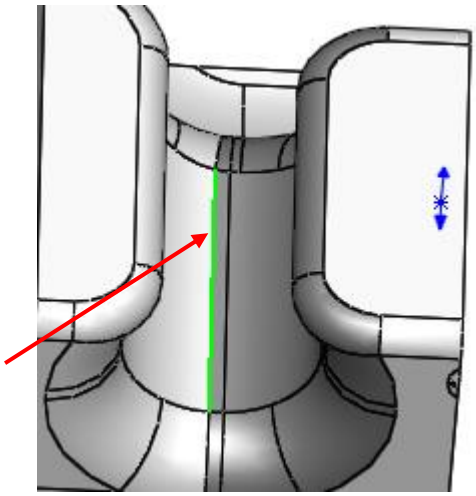
    The Curve Wizard dialog box is displayed.

18. Make sure the Selection mode is set to *Single Face* and *Single Edge*, then pick the bottom edges of the feature.
19.  Click *OK* to apply the changes.

    Edge<1> is listed in the Selected Contour field.



Pick the Top edge

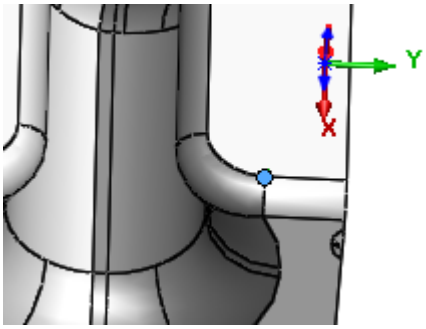


Pick the Bottom edge

20. On the Pattern tab, set the following parameters:
    - In the Limits group box, set the Method to *Start and End at Exact Surface Edges*.
    - In the Surface Finish group box, set the Max stepover to **0.02in.**
    - In the Start Hint group box, check the *Use start hint* option.
    -  Click the *Define Point* button.

      The Define Point dialog box is displayed.
    - Pick the vertex on the part shown in the image.
    - Click *OK* to close the dialog box.

      The Start hint option provides more control over the XY entry position of the tool. Selecting an entry position does not mean that the tool will actually enter at the position, but the entry location of the tool will be as close as possible to the start hint location.



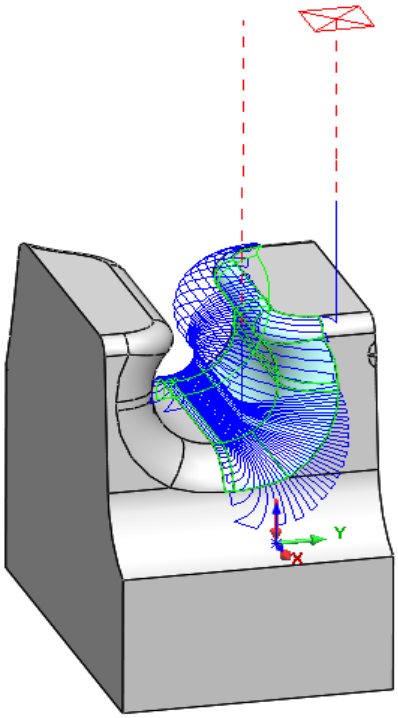
Pick the vertex point

21. Click the *Finish* tab and set the following parameters.
    - In the Surface Finish group box, set the Chaining tol. to **0.1in.**

      This value is used in combination with the Mach deviation for toolpath calculation and should be set to 1 to 10 times the Mach deviation. Note that using higher values for this option increases the calculation speed; however, inaccuracies in the toolpath may occur.
22. Click the *Entry/Retract* tab and set the following parameters:
    - In the Leadin move group box, set Method to *Use Leadin* and From to *Clearance*.
    - In the Leadout move group box, set the Method to *Use Leadout* and To *Clearance*.
    - In the Clearance group box, set the type to *Plane in Z* and Z to **4in.**
    - In the Distance group box, set the Rapid length to **1in**
    - In the Leadin page, click on the *Arc*. Set the same parameters from the Leadout tab.
23. Click the *Links* tab and set the following parameters:
    - In the Links along cut group box, set the Link threshold to **200**.

- Set the Gaps <= link threshold link type to *Follow Surfaces*.
- In the Gaps > link threshold, set the link type to *Retract to Clearance* and Entry/Exit method to *Use Leadout*.
- In the Links between cuts group box, click on the percentage sign to change into inch format, if necessary.
- Set the link threshold to **1in**.
- Set the Stepover <= link threshold link type to *Follow Surfaces*.
- In the Stepover > link threshold, set the link type to *Retract to Clearance* and Entry/Exit method to *Use Leadin*.
- In the Surface edge merging group box, click on the percentage sign to change into inch format, if necessary.
- Set the Distance to **0.0039in**.

24. Click the *Axis Control tab* and set the following parameters:
    - Number of axis to *3 Axis*
    - Contact point to *Auto*
25. Click *OK* to apply the changes and close the Operation Parameters dialog box.
26. Click the *Generate Toolpath* button on the CAMWorks Command Manager.
27. Click on the *Multiaxis Mill1* in the Operation tree and observe the toolpath.



Obtained Toolpath

Simulate Toolpath

1. Click on the *Simulate Toolpath* button on the CAMWorks Command Manager.

2.  Click the *Run* button on the toolbar to start the simulation.
3. Click the *OK*  button to exit simulation mode.

Multiaxis 5

Topic covered in this tutorial:

- Changing Machining Parameters to Produce Different Results

Different results occur depending on how the parameters are set up. It is possible to machine this part using different approaches.

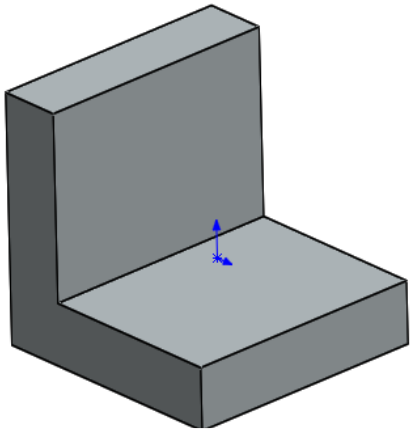
Opening the Part and Defining the Machine and Stock

1. Open the part file **MULTIAX_5.sldprt** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\4-5AxisMill

2. Click the *CAMWorks Feature Tree* tab.

3. Double click *Machine* item in the Feature tree to open the Machine dialog box.
   - On the Machine tab, highlight *Mill–inch* and click on *Select* button.
   - Click the *Tool Crib* tab and make sure *Tool Crib 2 (Inch)* is selected as Active tool set.
   - Make sure *Tool crib priority* option is unchecked.
   - Click the *Post Processor* tab; make sure *M5AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.

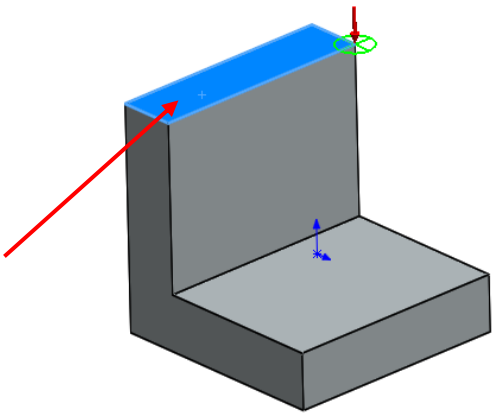


MULTIAX_5.sldprt

4. Double click *Stock Manager* in the Feature tree to open the Stock Manager dialog box.
   - Leave the Stock type set to *Bounding box.*
   - Leave the *Material* set to default.
   - Click *OK* to apply the changes and close the dialog box.

Inserting the Multi Surface Feature

1. Right click *Stock Manager* in the tree and select *New Mill Part Setup* on the context menu.
2. In the graphic area, click on the plus sign next to the MULTIAX_5 to expand.
3. Pick the *Top face* as shown in the image.
4. Click on the *Reverse Selected Entity* button to reverse the machining direction.
5. Click *OK* to insert the mill part setup.



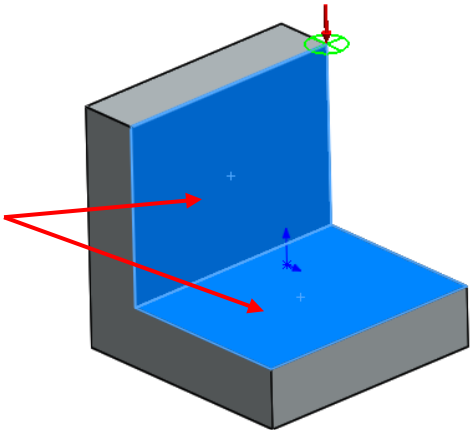
Pick the Top face

The Mill Part Setup1 is listed in the Feature tree.

6. Right click *Mill Part Setup1* and select *New Multi Surface Feature* on the context menu.
7. Pick the two surfaces as shown in the image.

   The Face 1 and Face 2 are listed in the selected faces list box.
8. For the Strategy, select *5Axis*.
9. Click *OK* to insert the multi surface feature.

   The Multi Surface Feature1 is listed in the Feature tree.



Pick the two surfaces

Generate Operation Plan and Adjusting the Parameters

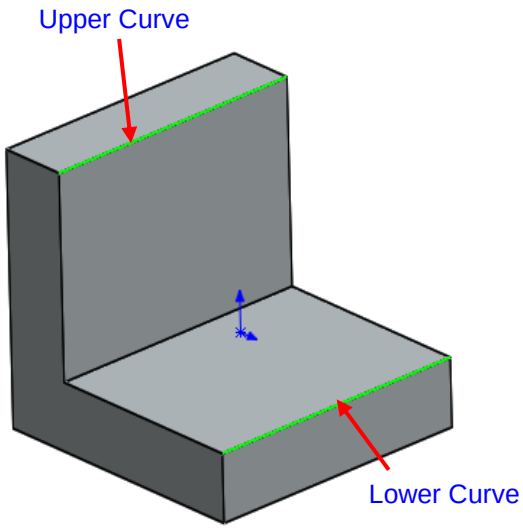
1. Right click *Multi Surface Feature1* and select *Generate Operation Plan* on the context menu.

   The Multiaxis Mill1 is listed in the Operation tree.
2. Double click *Multiaxis Mill1* in the Operation tree to open the Operation Parameters dialog box.
3. Click the *Mill Holder* page on the Tool tab and change the settings as follows:
   - Top diameter (D1) = **4in**
   - Bottom diameter (D2) = **2in**
   - Overall length (L1) = **4in**
   - Bottom length (L2) = **2in**
   - Protrusion = **1.375in**
4. Click the *Pattern* tab.
5. For the Pattern type, set the *Flowline Between Curves*.
6. Click the *Upper* button.

   The Curve Wizard dialog box is displayed.
7. Pick the green edge on the upper curve as shown in the image.

   The Edge<1> is listed in the Selected Contour field.
8. Click *OK* to apply the changes.

   The Operation Parameters dialog box is displayed again.
9. Click the *Lower* button and the Curve Wizard dialog box is displayed.



Pick the top and bottom edges

10. Pick the green edge on the lower curve as shown in the image.

    The Edge<1> is listed in the Selected Contour field.

11. Click *OK* to apply the changes.

    The Operation Parameters dialog box is displayed.

12. In the Surface finish group box, set the Max. stepover to **0.1in**.
13. Click the *Entry/Retract* tab.
    - In the Leadout move, set the Method to *None*.
    - In the Clearance group box, set the type to *Plane in Z* and set Z to **1in.**
    - In the Distances group box, set the Rapid length to **0.5in** and the Feed length to **0.1in**.
14. Click the *Links* tab.
    - In the Links along cuts, set the Gaps <= link threshold to *Direct* and Gaps > link threshold to *Retract to Clearance*.

      The surfaces defining the workpiece can have gaps and holes. If gaps are detected along a toolpath segment, you have multiple options of how to transition the gap. Some options include a direct move from the start to the end of the gap and retracting to the clearance plane. The limit for ignoring the gap is defined as a percent of the tool diameter.
15. Click *OK* to close the Operation Parameters dialog box.

Generate Toolpath and Simulate Toolpath

1. Right click on the *Multiaxis Mill1* in the tree and select *Generate Toolpath* on the context menu.

   OR

   Click the *Generate Toolpath* in the CAMWorks Command Manager.

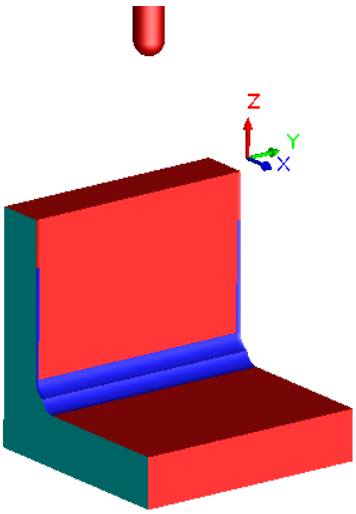
2. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click on the *Multiaxis Mill1* in the Operation tree and select *Simulate Toolpath* on the context menu.

3. Set the following parameters:
   - Tool: *Cut Collision*
   - Tool Shank: *Cut Collision*
   - Tool Holder: *Cut Collision*
4. Click the *Run* button.

   Notice that the part is gouged by half the diameter of the tool when approaching the walls of the part, as well as by the holder.



Simulate Toolpath

5. Click the *OK* button to cancel the simulation.

Make Adjustments to Eliminate Gouging

1. Double click *Multiaxis Mill1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

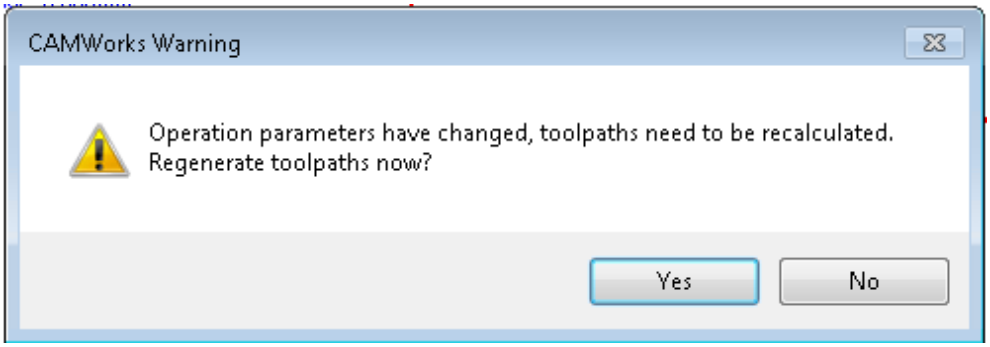
2. Click the *Finish* tab and check the *Blend surfaces by tool radius* option in the Options group box.

   When this option is checked, CAMWorks finds small radius areas and inner sharp edges in the surface model and does not generate toolpath in these areas. Inside corners, where the radius is less than the tool radius may result in motion that looks similar to a fish tail. These fish tails are removed using this option. In most cases, this option is used with a ball cutter, lollipop cutter or a conical cutter with ball tip. If swarf machining is applied (side cutting), then this option can also be used with cylinder and torus cutters.

3. Click *OK* to apply the changes.

4. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

   Toolpaths will be regenerated.



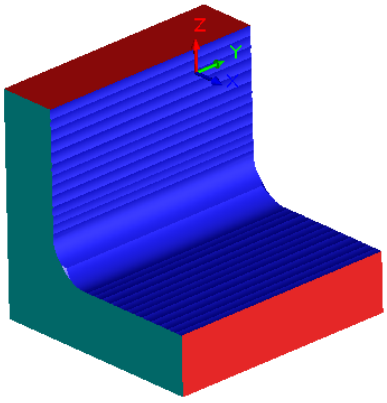
Warning message to regenerate toolpath

5. If any warning message is not displayed, click the *Generate Toolpath* button on the CAMWorks Command Manager.

6. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

7. Click the *Run* button.

   Notice that the only gouges that are left are caused by the tool holder.



Simulate Toolpath

8. Click the *OK* button to cancel the simulation.

9. Double click *Multiaxis Mill1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

10. Click the *Axis Control* tab.

    - In the Cutting direction tilt options group box, set the Side tilt angle to **45deg**.

11. Click *OK* to apply the changes.

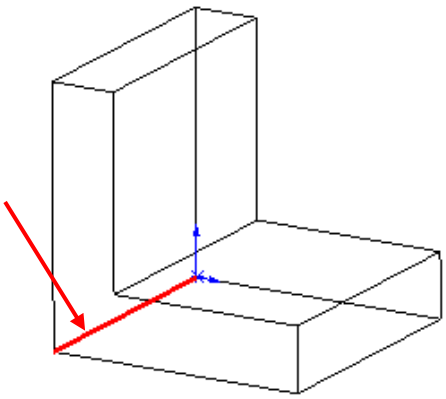
12. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

Toolpaths will be regenerated.

13. If any warning message is not displayed, click the *Generate Toolpath* button on the CAMWorks Command Manager.
14. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
15. Click the *Run* button.

    Notice the toolpath looks good until it starts to cut into the lower section. The tool needs to stay at the 45degree angle throughout the toolpath.
16. Click the *OK* button to cancel the simulation.
17. Double click *Multiaxis Mill1*in the Operation tree.
18. Click the *Axis Control* tab.
    - In the Cutting direction tilt options group box, select *Tilted From Curve* for the Tool axis.
    - Set the Curve tilt strategy to *From Start to End for Each Contour*.
    - Click on the *Display style* in the graphics area and select the *Wireframe* from the list.
    - Click the *Tilt Curve* button.

      The Curve Wizard dialog box is displayed.
    - In the Selection mode, make sure the *Single Edge* is selected.
    - Pick the red bottom edge as shown in the image.
    - Click *OK* to close the dialog box.



Pick the bottom edge

19. Click *OK* to apply the changes.
20. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

    Toolpaths will be regenerated.
21. If any warning message is not displayed, click the *Generate Toolpath* button on the CAMWorks Command Manager.
22. Click on the *Display style* in the graphics area and change the display back to Shaded or Shaded with edges.
23. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
24. Click the *Run* button.
25. Click the button to cancel the simulation.

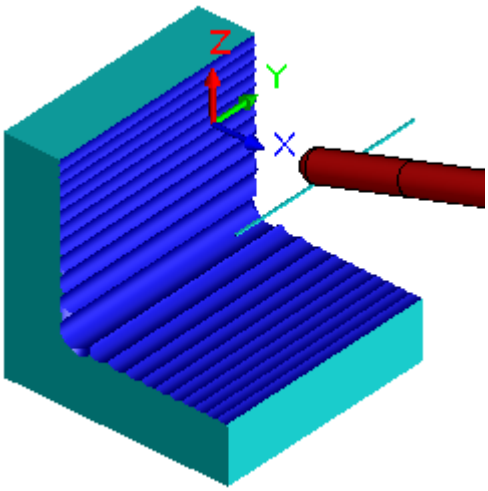
Remove the Bulk Material

Roughing passes can be added to remove the bulk of the material above the faces you are machining.

1. Double click *Multiaxis Mill1* in the Operation tree.
2. Click the *Roughing* tab.
   - Check the *Multi passes* option.
   - In the Depth processing, select *By Level*.

     When this option is selected, roughing is completed for all areas of a given feature at a given Z depth before machining the next Z depth.
   - In the Roughing group box, enter **8** for the Number (the number of roughing passes to be generated) and **0.1in** for Spacing (the distance between each rough pass).
3. Click *OK* to apply the changes.
4. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click *Yes* to regenerate the toolpaths.

   Toolpaths will be regenerated.
5. If any warning message is not displayed, click the *Generate Toolpath* button on the CAMWorks Command Manager.
6. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
7. Set the following parameters:
   - Tool: *Ignore Collision*
   - Tool Shank: *Ignore Collision*
   - Tool Holder: *Ignore Collision*
8. Click the *Run* button.
9. Click the *OK* button to cancel the simulation.



Simulate Toolpath

Multiaxis 6

Topic covered in this tutorial:

- Using Surfaces and Gouge Checking to Simplify Machining

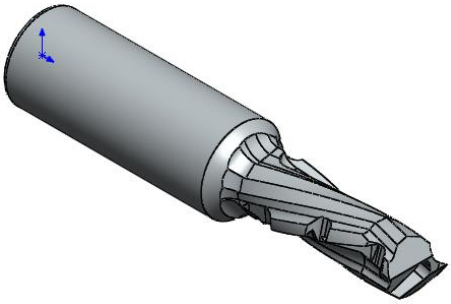
This is a model of a milling tool body and the objective is to demonstrate how to machine the flute area of the model. For the purposes of this tutorial, it is desired to drive the tool radially around the axis of model. Since the shape of the model is relatively complex, defining the tool axis to be perpendicular to the axis running through the center of the model is not possible from the part geometry itself. One of the techniques for defining and creating a multiaxis toolpath is to machine a simple shape, in this case a cylinder, then apply gouge avoidance methods to modify the toolpath to the actual shape of the part.

Opening the Part and Defining the Machine

1. Open the part file **MULTIAX_6.sldprt** in the following folder.

   Drive:\CAMWorksData\CAMWorks201x\Examples\Tutorial_Parts\4-5AxisMill

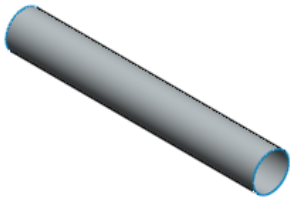
   Note that this part resembles a mill tool body, which is relatively complex. Controlling the tool axis from the faces on the actual part model will be difficult. To demonstrate how to use surfaces and gouge checking to simplify machining this part, a cylindrical surface fully inside the part model has been created.



MULTIAX_6.sldprt

2. To view the cylindrical surface, right click on *Solid Bodies(1)* in the FeatureManager Design tree and select the *Hide* icon.
3. Right click *Surface-Bodies(1)* and select the *Show* icon.

   In the graphics area, observe that the cylindrical surface is displayed.



Cylindrical surface

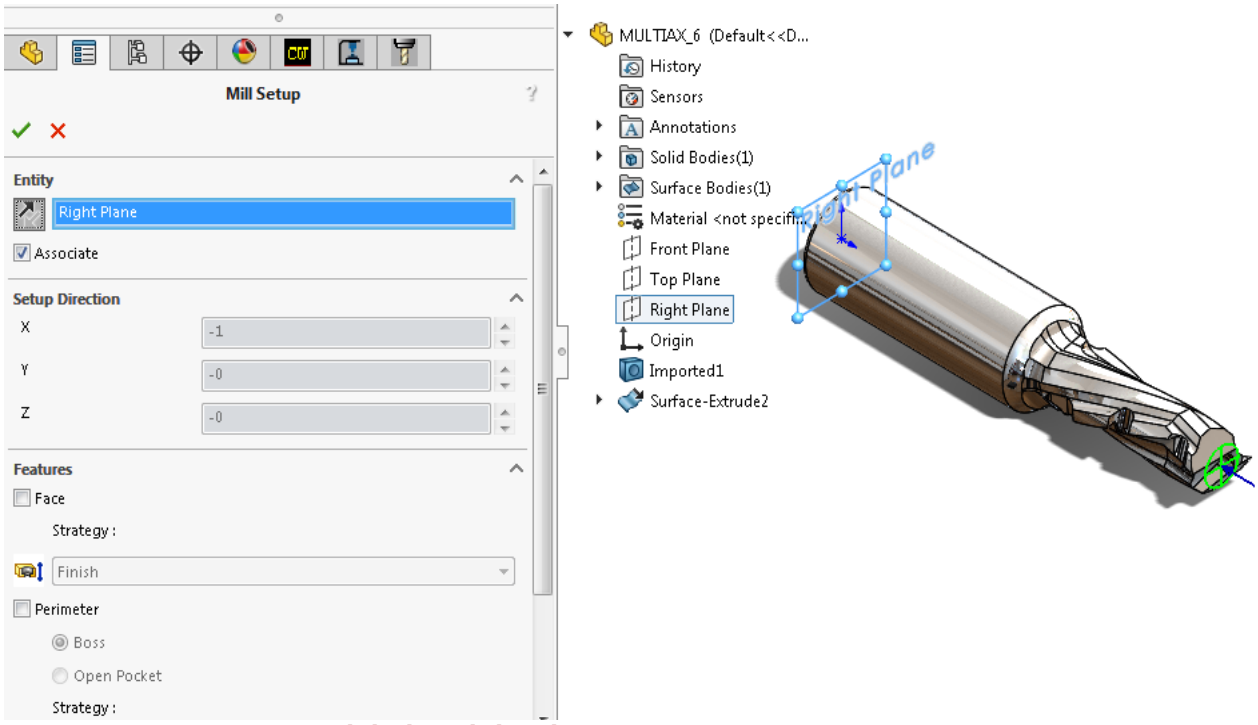
4. Click the *CAMWorks Feature Tree* tab.
5. Double click *Machine* item in the Feature tree to open the Machine dialog box.
6. In the Machine dialog box, highlight *Mill-inch* and click *Select* button to set as Active machine.
7. Click on the *Tool Crib* tab, highlight *Tool Crib 2 (Inch)* and click *Select* button to define as Active tool crib.
8. Make sure Tool crib priority option is unchecked.
9. Click the *Post Processor* tab, highlight *M5AXIS-Tutorial* and then click *Select.*
10. Click *OK* to close the dialog box.

Inserting Mill Part Setup

1. Right click *Stock Manager* in the tree and select *Mill Part Setup* on the context menu.

   The Mill Setup dialog box is displayed.
2. In the graphics area, click the (⊞) plus sign next to the *MULTIAX_6* to expand.
3. Pick the *Right Plane* from the references.
4.  Click the *Reverse Selected Entity* button in the Mill Setup dialog box.
5.  Click *OK* to insert the mill part setup.

   The Mill Part Setup1 is listed in the Feature tree.



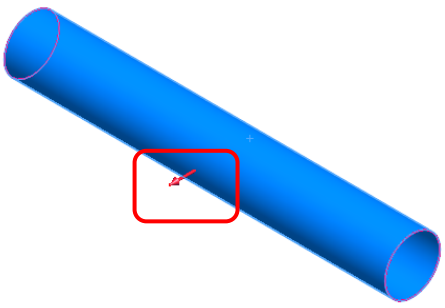
Pick the Right Plane

Inserting Multi Surface Feature

In this tutorial, you will interactively insert two Multi Surface Features. The first will be defined from the cylindrical surface and the second will be defined from the actual geometry.

1. Right click *Mill Part Setup1* and select *Multi Surface Feature* on the context menu.
2. Select *5 Axis* for the Strategy.
3.  In the Face Select Options group box, click the *Select All Faces* button.

   CW Face-131 is listed in the Selected Faces list box. This action graphically selects the cylindrical surface.
4. In the Select Faces list box, select the *CW Face-131*.



Machining direction

5. Check the *Show Normal* option. The surface normal indicates which side of the surface will be machined.
6.  Click the *Reverse Direction* button and make sure the arrow is pointing away from the surface as shown in the image.

   Now the tool will machine to the outside of the surface.
7.  Click *OK* to insert the multi surface feature.

   The Multi Surface Feature1 is listed in the CAMWorks Feature tree.

Generate Operation Plan

1.  Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click the *Mill Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

   The Multiaxis Mill1 is listed in the Operation tree.

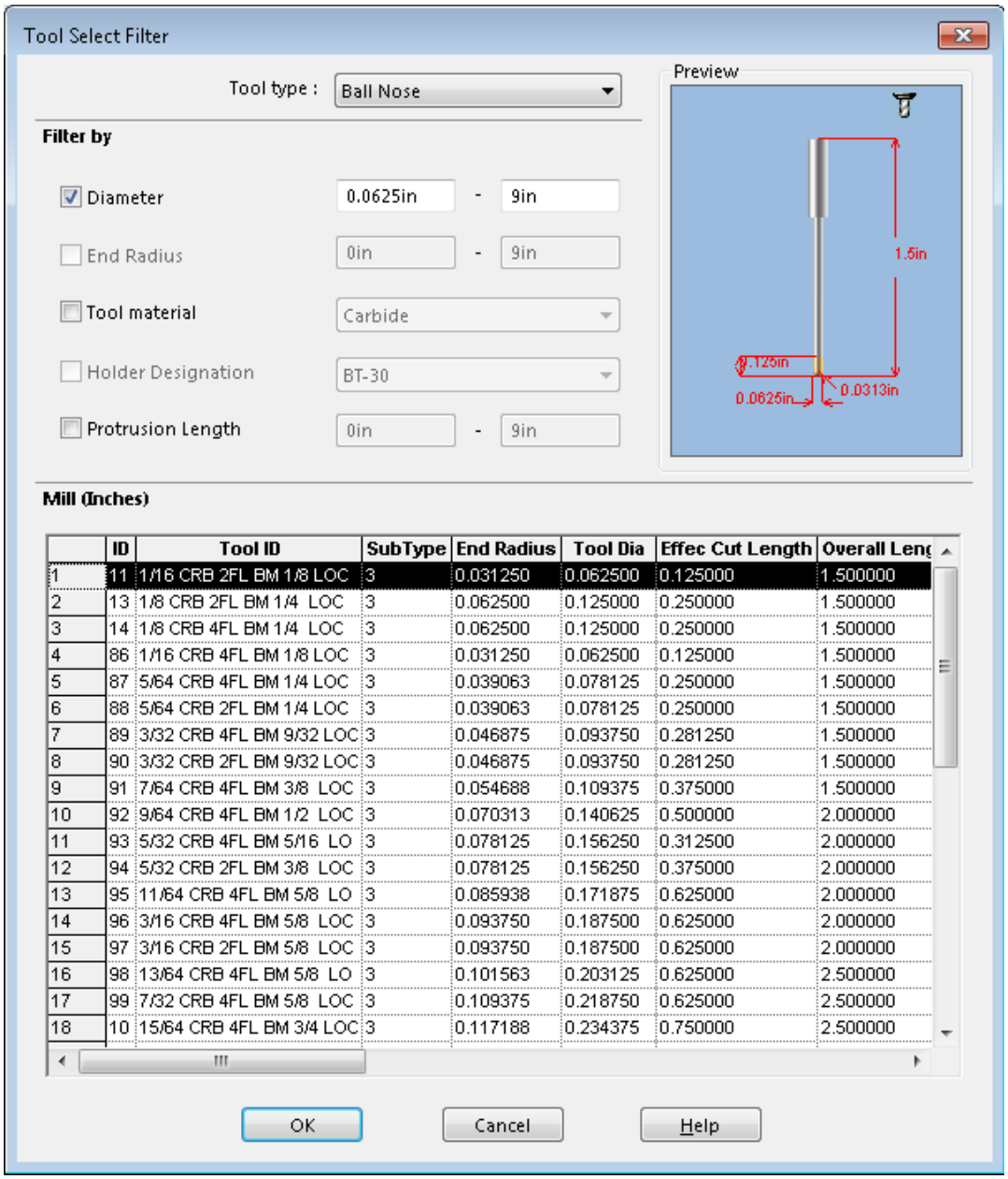
Adjusting the Operation Parameters and Generating Toolpath

1. Double click *Multiaxis Mill1* in the tree to Operation Parameters dialog box.
2. Click the *Tool Crib* page on the Tool tab and click the *Add* button.

   The Tool Select Filter dialog box is displayed.
3. Leave the Tool type set to *Ball Nose*.
4. Check the *Diameter* option and enter the **0.0625in** value in the first input box to the right of Diameter.

   CAMWorks will update the tool list based on the selected tool type and defined diameter.
5. Select a **0.0625in** diameter Ball nose tool from the updated list.
6. The Preview window is activated and displays the 3D view of the selected tool.
7. Click *OK* button.

   This action closes the dialog box and adds the selected tool to the active Tool crib. The new tool is listed at the bottom of the Tool Crib grid.



	ID	Tool ID	SubType	End Radius	Tool Dia	Effec Cut Length	Overall Leng
1	11	1/16 CRB 2FL BM 1/8 LOC	3	0.031250	0.062500	0.125000	1.500000
2	13	1/8 CRB 2FL BM 1/4 LOC	3	0.062500	0.125000	0.250000	1.500000
3	14	1/8 CRB 4FL BM 1/4 LOC	3	0.062500	0.125000	0.250000	1.500000
4	86	1/16 CRB 4FL BM 1/8 LOC	3	0.031250	0.062500	0.125000	1.500000
5	87	5/64 CRB 4FL BM 1/4 LOC	3	0.039063	0.078125	0.250000	1.500000
6	88	5/64 CRB 2FL BM 1/4 LOC	3	0.039063	0.078125	0.250000	1.500000
7	89	3/32 CRB 4FL BM 9/32 LOC	3	0.046875	0.093750	0.281250	1.500000
8	90	3/32 CRB 2FL BM 9/32 LOC	3	0.046875	0.093750	0.281250	1.500000
9	91	7/64 CRB 4FL BM 3/8 LOC	3	0.054688	0.109375	0.375000	1.500000
10	92	9/64 CRB 4FL BM 1/2 LOC	3	0.070313	0.140625	0.500000	2.000000
11	93	5/32 CRB 4FL BM 5/16 LO	3	0.078125	0.156250	0.312500	2.000000
12	94	5/32 CRB 2FL BM 3/8 LOC	3	0.078125	0.156250	0.375000	2.000000
13	95	11/64 CRB 4FL BM 5/8 LO	3	0.085938	0.171875	0.625000	2.000000
14	96	3/16 CRB 4FL BM 5/8 LOC	3	0.093750	0.187500	0.625000	2.000000
15	97	3/16 CRB 2FL BM 5/8 LOC	3	0.093750	0.187500	0.625000	2.000000
16	98	13/64 CRB 4FL BM 5/8 LO	3	0.101563	0.203125	0.625000	2.500000
17	99	7/32 CRB 4FL BM 5/8 LOC	3	0.109375	0.218750	0.625000	2.500000
18	10	15/64 CRB 4FL BM 3/4 LOC	3	0.117188	0.234375	0.750000	2.500000

Tool Select Filter Dialog Box

8. On the Tool Crib tab, highlight the **0.0625in** Ball Nose tool at bottom of the Tool Crib.
9. A Preview window is displayed at right side of the Operation Parameters dialog box. This Preview window contains the dynamic 3D view of the tool and holder.
10. Click the *Select* button. This action will assign the highlighted tool as the tool to be used for machining this operation.
11. Click *Yes* to replace the corresponding holder.
12. Click the *Pattern* tab and set the following parameters:
    - Pattern type = *Slice*
    - Click the *Constant Z* button to define the Cut angle (XY) and Cut angle Z values
    - Direction Pattern = *Zig*

- Select the cut direction with CW or CCW.
- Check the *Maintain cutting direction* in the Direction group box
- Max stepover = **0.03in**

13. Click the *Entry/Retract* tab.
    - In the Leadin move group box, select *None* for the Method.
    - In the Leadout move group box, select *None* for the Method.
    - In the Clearance group box, select *Cylinder About Z* and set the Radius to **2in**.
    - In the Distances group box, set the Rapid length to **0.05in** and the Feed length to **0.05in**.
14. Click the *Links* tab.
    - Set every Entry/Exit method to None.
15. Click the *Axis Control* tab.
    - Set the Number of axis to *4 Axis.*
    - Set the Rotate about to *Z Axis*.
    - In the Cutting direction tilt options group box, set the Tool axis will be to *Tilted Relative to Cutting Direction.*
16. Click *OK* to close the Operation Parameters dialog box.
17. Right click *Multiaxis Mill1* in the tree and select *Generate Toolpath* on the context menu.
18. Right click *Multiaxis Mill1* and select *Step Thru Toolpath* on the context menu.

    Step the tool for a few passes to verify the pattern and the tool axis vector.

You now have the intended pattern and tool axis vector, but the toolpath will machine a cylinder and not the actual part shape. Using gouge protection against the actual part shape will result in the needed toolpath.

1.  Click the *FeatureManager Design Tree* tab.
2.  Right click on *Solid Bodies(1)* and select the *Show* icon.
3.  Right click on *Surface Bodies(1)* and select the *Hide* icon.
4.  Click the *Operation Tree* tab.
5. Double click *Multiaxis Mill1* in the Operation tree.
6. Click the *Gouge Checking* tab.
7. In Group 1 page, check the *Apply gouge checking to* check box.

   Gouge checking will be applied to the actual part shape and not the cylindrical surface.
8. In the Check against group box, remove the check from the *Feature surfaces* option and make sure the *Other surfaces* option is checked.
9.  Click the *Browse* button to the right of the Other surfaces option so that you can define a Multi Surface feature that contains the faces of the actual part model.

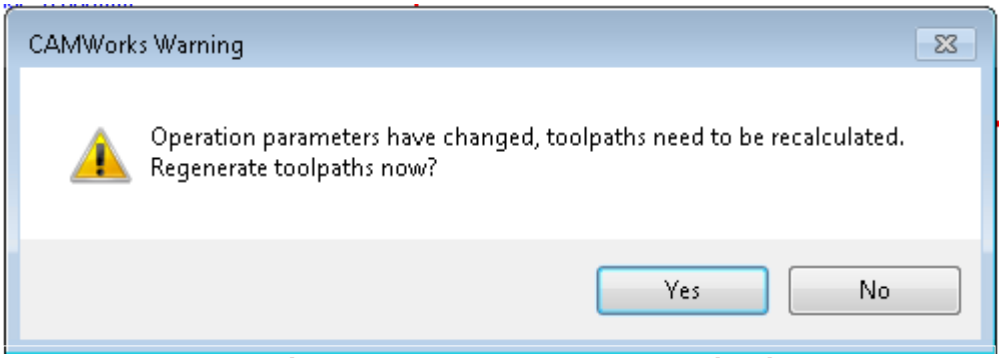
   The Avoid Features dialog box is displayed.
10. In the dialog box, click *Create Features*.

This button allows you to define additional Multi Surface features if necessary.

11. In the displayed New Multi Surface Feature dialog box, click *Select all faces*. All the faces of the part model are added to the new Multi Surface feature.
12. Click *OK* to insert the feature.

    The Avoid Features dialog box is displayed again.
13. In the dialog box, click *Select All*.
14. Click *OK* to close the dialog box.
15. In the Gouge check options group box, select *Move Tool Away* for the Strategy and then click *OK*.
16. CAMWorks displays a warning indicating that toolpaths need to be regenerated owing to change in machining direction. Click Yes to regenerate the toolpaths.

    Toolpaths will be regenerated.

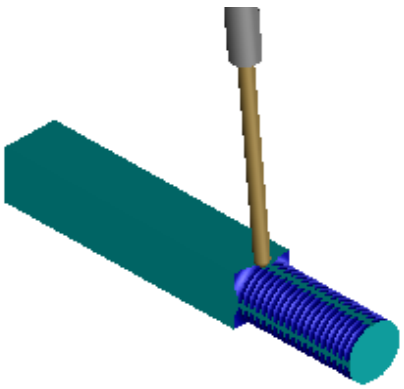


Warning message to regenerate toolpath

17. If such warning message is not displayed, then click Generate Toolpath button on the CAMWorks Command Manager.
18. Click the Simulate Toolpath button on the CAMWorks Command Manager.

    OR

    Right click Multiaxis Mill1on the in the tree and select Simulate Toolpath on the context menu.
19. Click Run button and observe the toolpath.



Simulation Toolpath