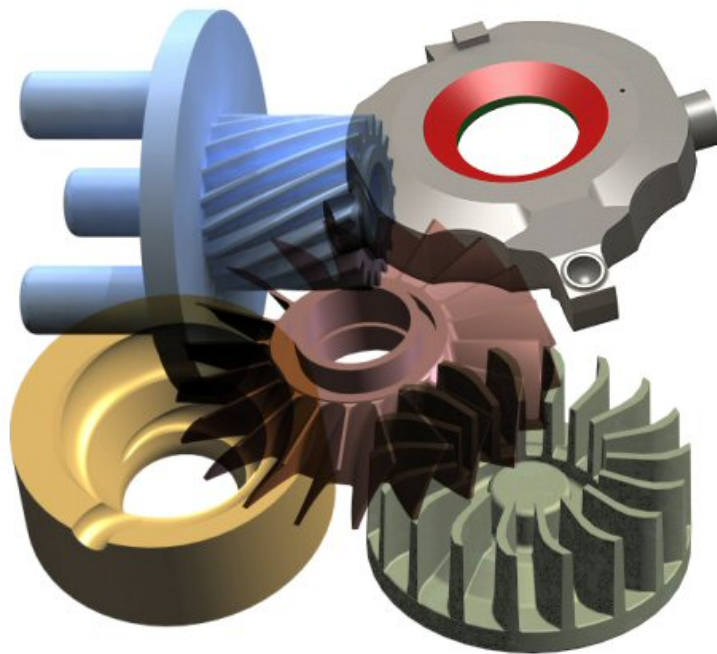




**camworks**  
powered by SolidWorks

# **CAMWorks 2007 Multiaxis Machining Tutorial**



**teksoft**  
Software Solutions  
Simple | Powerful | Productive

## ***License Agreement***

In receiving a software product described herein, you are granted a nonexclusive license to use that software on one computer. TekSoft, Inc. retains full title to and ownership of the product. Use of the product constitutes acceptance of this agreement. Duplication of the software product for any purpose, other than backup protection, including duplication for any commercial purpose, is prohibited by law.

TekSoft, 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.

## ***Warranty***

TekSoft, Inc. will replace defective software for a period of 90 days from date of delivery, at no charge. TekSoft, Inc. makes no other warranty, either express or implied, including, but not limited to, any implied warranties of merchantability or fitness for a particular purpose, regarding any product described herein. In no event shall TekSoft, Inc. be liable to anyone for special, collateral, incidental, or consequential damages in connection with or arising out of the purchase or use of these products. The sole and exclusive liability of TekSoft, Inc., regardless of form of action, shall not exceed the purchase price of the materials described herein.

## ***Trademarks***

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

TekSoft™ is a trademark of TekSoft, Inc.

CAMWorks™ is a trademark of Geometric Software Solutions Co. Ltd.

SolidWorks® is a registered trademark and FeatureManager™ is a trademark of SolidWorks Corporation.

Microsoft® Windows® is a registered trademark of Microsoft Corporation in the United States and/or other countries.

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

Copyright ©2006 TekSoft, Inc. All Rights Reserved.



**geometric**®  
Software Solutions  
*People Building Partnerships*

TekSoft is a subsidiary of Geometric Software Solutions Co. Ltd.

CW80c October, 2006

---

# Table of Contents

<b>CHAPTER 1 LEARNING MULTIAXIS MACHINING BASICS</b>	<b>1-1</b>
Multiaxis Machining Terminology .....	1-2
Steps to Generate Toolpaths and NC Code .....	1-3
Multiaxis 1 .....	1-4
What You'll Learn .....	1-4
Defining the Machine and Controller .....	1-4
Defining the Stock .....	1-6
Inserting a Part Setup and Defining Multi Surface Features .....	1-7
Generating an Operation Plan and Adjusting Operation Parameters...	1-8
Generating Toolpaths and Running Toolpath Simulation .....	1-10
Adjusting Machining Parameters .....	1-10
Defining the Tool Orientation .....	1-11
Post Processing Toolpaths .....	1-13
Multiaxis 2 .....	1-14
What You'll Learn .....	1-14
Using Collision Detection Options .....	1-15
Controlling the Start and End of the Toolpath .....	1-16
<b>CHAPTER 2 LEARNING MORE ABOUT MULTIAXIS</b>	<b>2-1</b>
Multiaxis 3 .....	2-2
What You'll Learn .....	2-2
Defining Tool Movement When Gaps are Encountered .....	2-3
Using Links with Gouge Checking .....	2-5
Multiaxis 4 .....	2-6
What You'll Learn .....	2-6
Machining Undercut Areas with 3 Axis Cutting .....	2-6
Multiaxis 5 .....	2-10
What You'll Learn .....	2-10
Changing Machining Parameters to Produce Different Results .....	2-10

## INDEX

## Table of Contents

---

# Chapter 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 *CAMWorks Multiaxis Machining Tutorial* contains more detailed information about using CAMWorks.

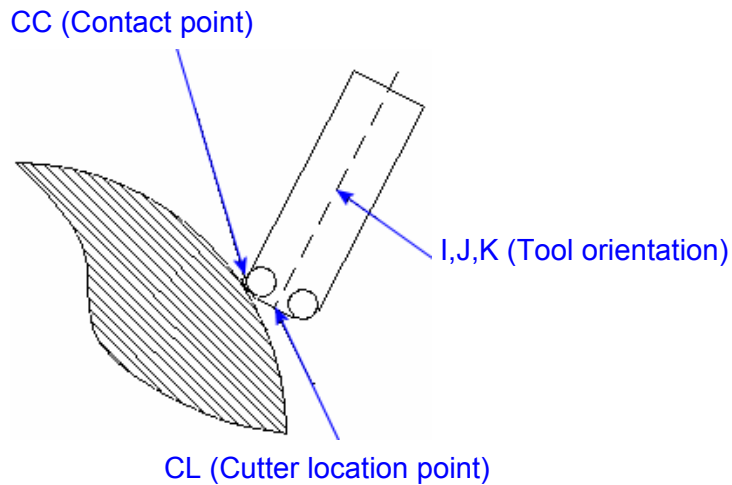
---

**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



CC = The point where the tool touches the material.

CL = The point that is represented by the coordinates of the machine program.

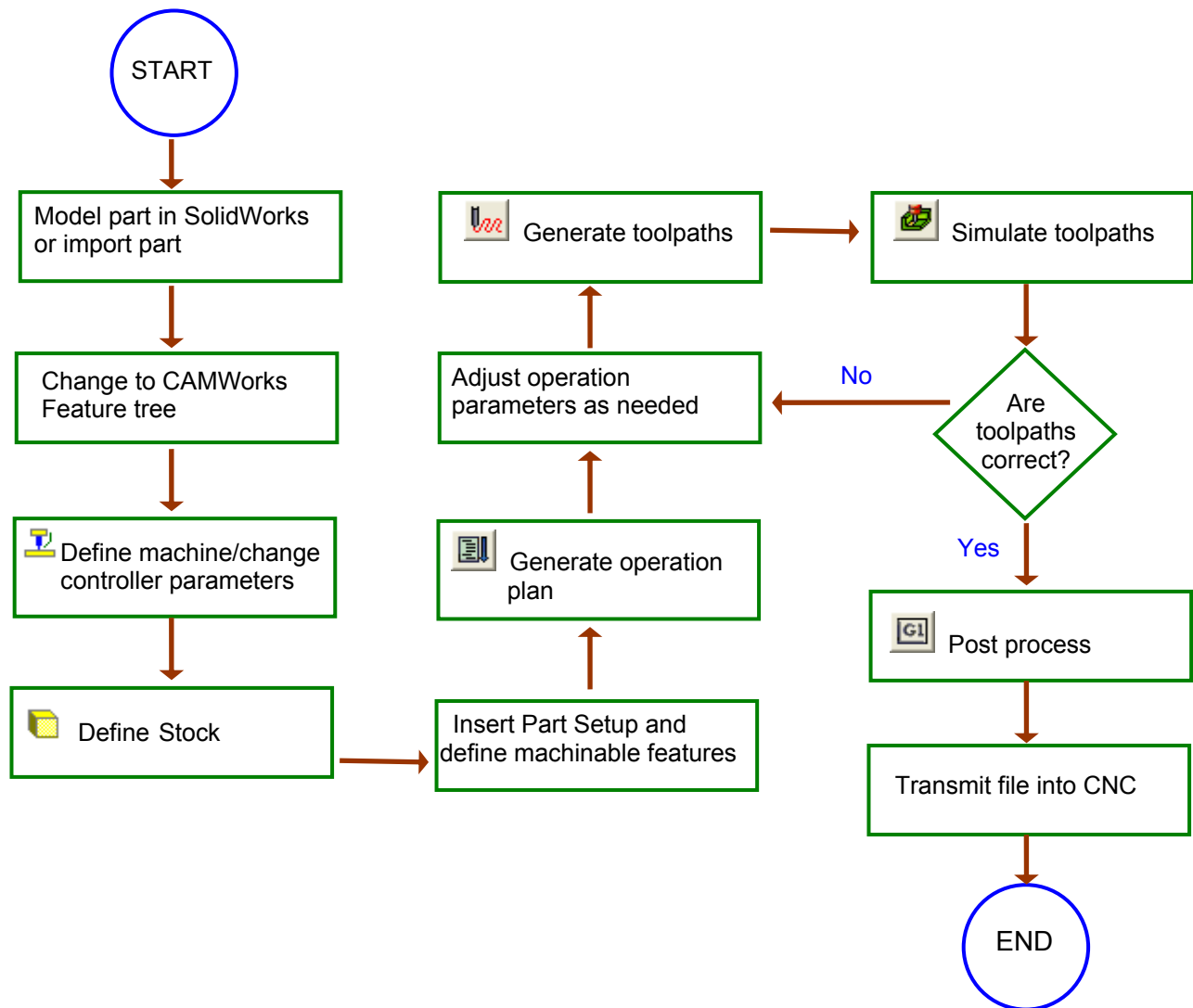
I,J,K = Values that represent the orientation of the tool axis

When the tool tilts, it rotates around the CC point. Note that the point on the material is fixed and the point on the tool itself changes according to the orientation (except from sharp corner tools).

# 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.
2. Change to the CAMWorks Feature tree.
3. Define the Machine and modify the controller parameters.
4. Define the stock.
5. Insert a Part Setup and define 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

## What You'll Learn

- Defining the Machine and Controller
  - 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
1. Open the part file **MULTIAX\_1.SLDPRT** in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\4-5AxisMill*).  
 A part is a solid that is created with SolidWorks or imported into SolidWorks from another CAD system via an IGES, Parasolid, SAT file, etc. This exercise uses an existing SolidWorks part.  
 Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\4-5AxisMill*).
  2. Click the CAMWorks Feature Tree tab.

## Defining the Machine and Controller

1.  Right click the Example Mill-mm in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.



### **Did You Know ...**

In the Feature and Operation trees, instead of right clicking items and selecting Edit Definition, you can double-click the following items to open the dialog box for editing the items:

- Stock Manager
- Machine
- Setup
- Feature
- Operation



The Machine dialog box displays the Machine tab. The default machine is specified in the Technology Database. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

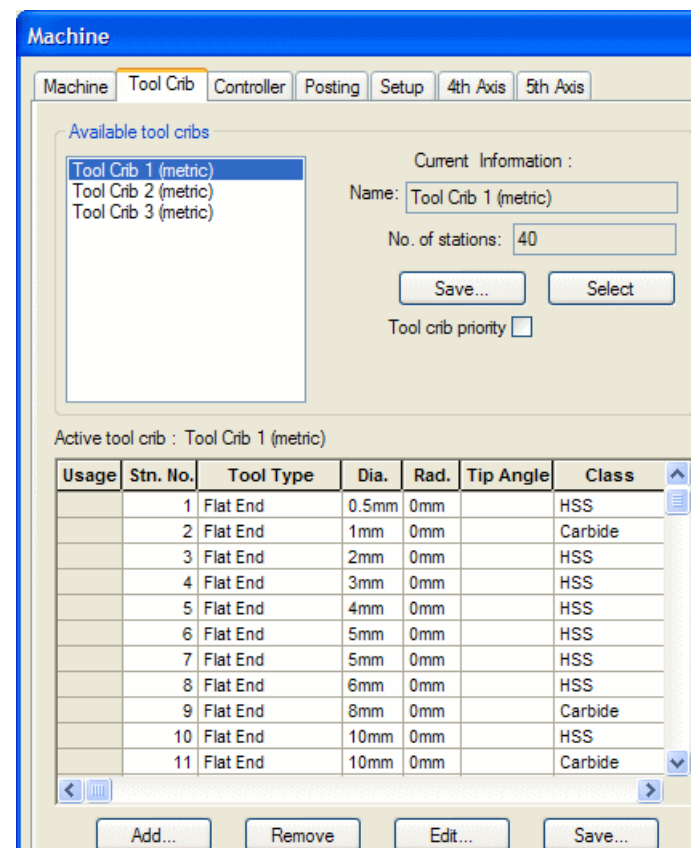
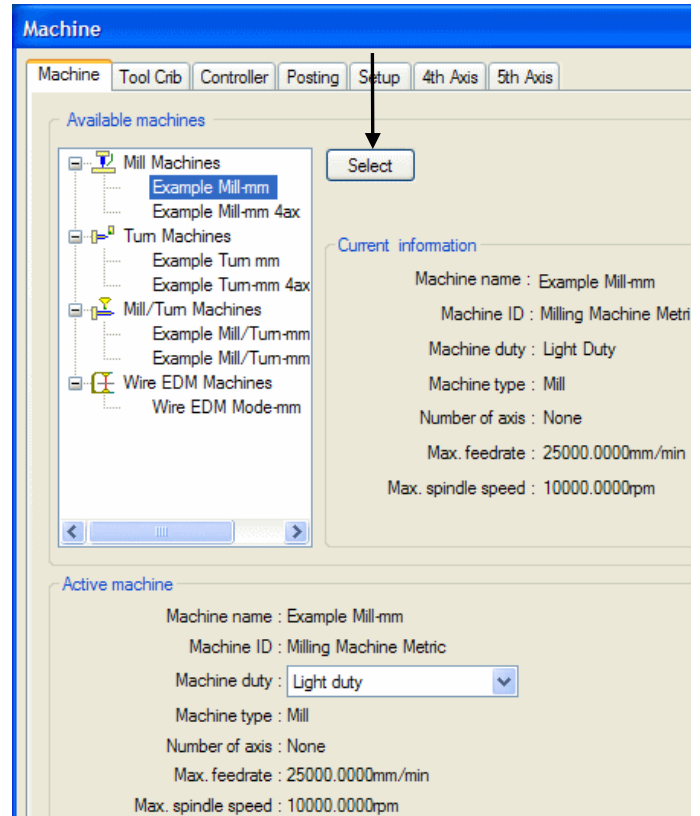
Machine tools are set up in the Technology Database. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility.

2. In the Available machines list, highlight Example Mill-mm and click the Select button.

3. Click the Tool crib tab and make sure Tool Crib 1 is selected.

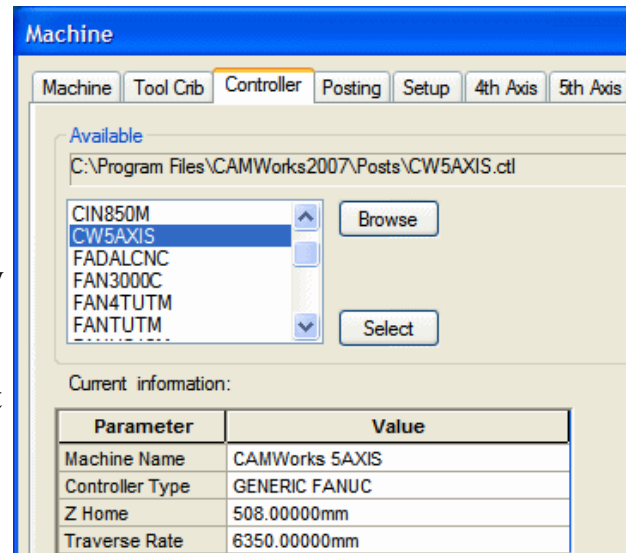
The Tool Crib page allows you to choose a Tool Crib, which is a set of tools or tool assemblies that are used with the machine you have chosen. These are not all the tools that are available, but a subset that you can modify to represent the actual set of tools that the machine has loaded.

Tool Crib 1 has been set up for the sample Mill-mm machine. When you define your machine tools in the Technology Database, you can set up your own tool cribs.

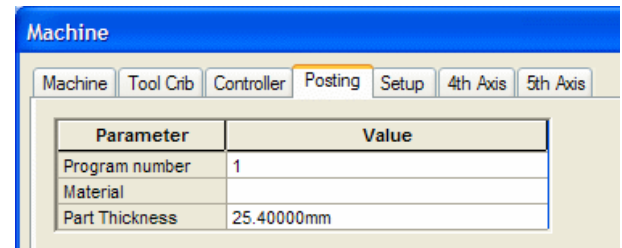


- Click the Controller tab.

The Controller tab allows you to select the post processor from a list of available controllers. The list that displays depends on the post processors that are installed on your system. By default, CAMWorks is supplied with several default post processors that may or may not be suitable for your needs. Contact your CAMWorks reseller for information on making changes to these post processors or for other post processors. If the controllers do not display, use the Browse button to locate the folder containing the controller files (\*.ctl).



- Highlight CW5AXIS (the tutorial Multiaxis Machining post processor) in the list and click the Select button. CW5AXIS is the controller used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.
- Click the Posting tab. The parameters on this page are used for the following:
  - 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.
- Click OK to close the dialog box.



## Defining the Stock

The stock is the material from which the part will be machined. Mill stock can be defined from a bounding box, an extruded sketch, an STL file or another SolidWorks part if working in assembly mode.

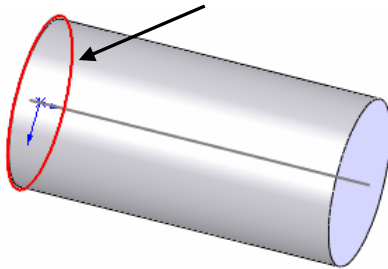
In this exercise, you define the stock as an extruded sketch.


- Double-click Stock Manager in the CAMWorks Feature tree. The Manage Stock dialog box displays.

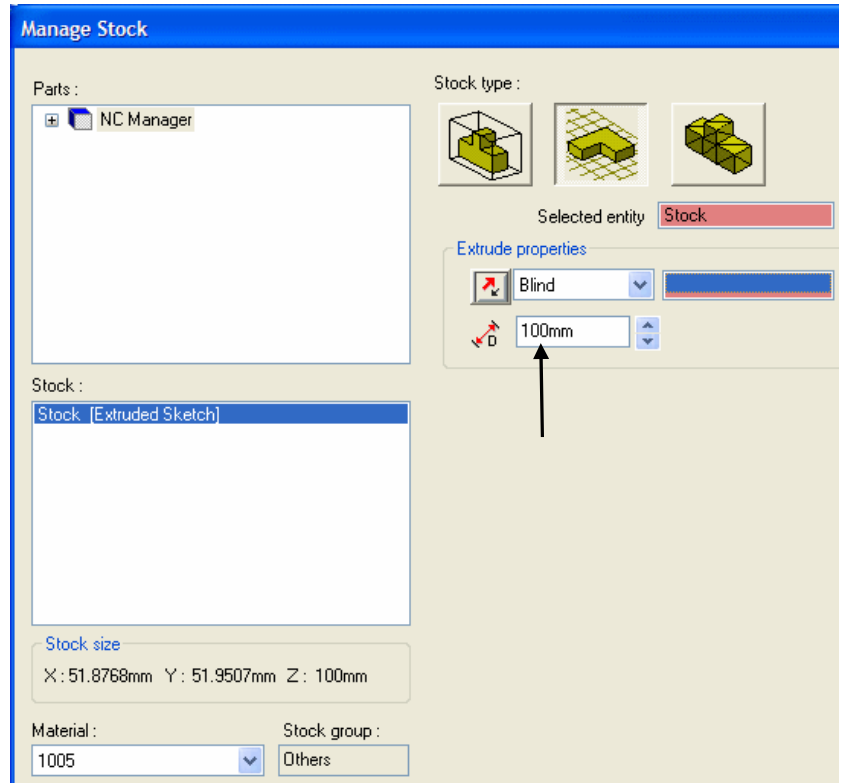


- Click the Extruded Stock button.

- For the stock, either pick the sketch named Stock in the SolidWorks FeatureManager design tree or pick the sketch on the part (shown in red in the figure below).



-  Set the depth to 100mm.  
The depth defines the distance the sketch is extruded.
- Click OK.



## 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.

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

- Right click the Stock Manager in the tree and select Insert Mill Part Setup on the shortcut menu.
- In the Mill Part Setup dialog box, pick the Top Plane, check the Reverse direction option and click OK.

The Part Setup is inserted with the direction normal to the selected face/plane. You should see a marker on the part. The large arrow indicates the machining direction.

- Right click Mill Part Setup1 in the CAMWorks Feature tree and select Insert Multi Surface Feature on the shortcut menu.

The Insert Multi Surface Feature dialog box displays.

- Pick only the cylinder face on the part model.



### **Did You Know ...**

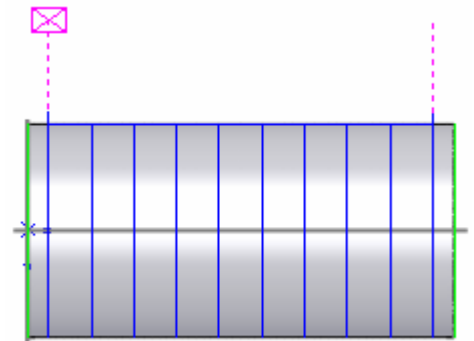
When picking faces in Multiaxis Machining, all faces will be machined in normal circumstances. You pick only the faces that need to be machined. Avoids are handled differently than 3 axis milling as you will learn in another exercise. In Multiaxis Machining, any faces that are selected to avoid in the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

5. Click the Show normal check box and notice a direction 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. Set the Attribute to 5 Axis.
7. Click Insert.
8. Click Close. Multi Surface Feature1 displays in the CAMWorks Feature tree.

## Generating an Operation Plan and Adjusting Operation Parameters


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

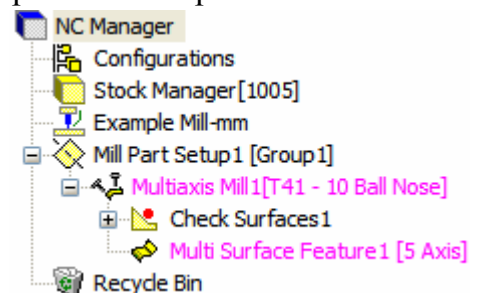
The Multiaxis Mill operations that are generated by CAMWorks are based on information stored in the Technology Database. These operations are intended to be used as a starting point and you can modify the machining parameters.



### 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 toolpaths 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 toolbar.  
CAMWorks generates the operation. The CAMWorks Operation tree provides an outline view of the operations for the machinable features.  
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. The Check Surfaces item is used to modify the surfaces used for collision detection.
2. Double click Multiaxis Mill1 in Mill Part Setup1.



3. Click the Tool Crib tab and scroll down the list of tools to locate a 20mm dia. Ball Nose.
  - If a tool is not found, click the Add button and select Ball Nose in the Tool Select Filter dialog box. In the Tools Database form, select a 20mm Ball Nose, then click OK.
  - On the Tool Crib tab, highlight the tool, click the <– Select button, then click Yes to replace the corresponding holder.
4. 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.

5. Select the Link to F/S Library option.

When this option is checked, the calculated feeds and speeds are updated automatically based on changes to the stock material, cutting conditions, or tool.

When this option is not checked, CAMWorks maintains any manually input feed and speed values even if the stock material, cutting conditions or tool change.

## Slice Pattern

1. Click the Pattern tab. This tab contains options to set the Pattern type, cutting order, cut limits and the surface quality.
2. For the Pattern Type, select Slice.

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

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

The machining strategy is parallel (slice) cuts. The angle on XY plane is 90 deg. 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.





### ***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.

4. Make sure the Direction is set to a Zigzag pattern and set the Max. stepover to 10mm.
5. Click the Entry/Retract tab and set the Leadin and Leadout method to None.
6. In the Clearance section set the following parameters:
  - Type = Plane in Z
  - Z = 25mm
7. In the Distances section set the following, then click OK.
  - Rapid length = 25mm
  - Feed length = 15mm


## Generating Toolpaths and Running Toolpath Simulation

1.  Click the Generate Toolpath button on the CAMWorks toolbar.

2.  Right click on Mill Part Setup1 and select Simulate Toolpath.  
The Simulate Toolpath toolbar displays.

3.  Click the Run button.

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

4.  Click the X button to cancel the simulation.


## Adjusting Machining Parameters

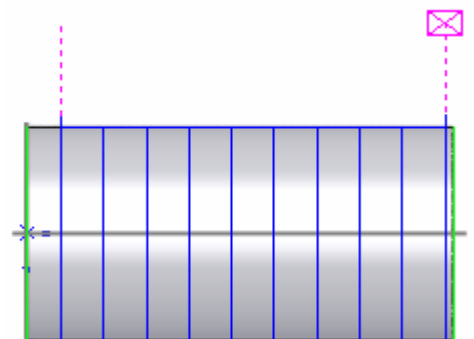
### Cuts Across Curve Pattern

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 the leading curve.

1. Double-click Multiaxis Mill1 in the tree.
2. On the Pattern tab, select Cuts Across Curve for the Pattern Type.
3. Click the Curve button.
4. Pick Sketch2 and click Finish.
5. Click the Preview button to generate and view the toolpath.

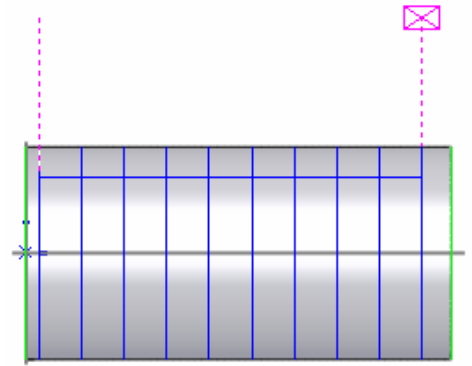
When you click this button, the toolpath for the current operation is calculated and displayed immediately. This allows you to preview the toolpath for the operation based on the current parameters without having to close the dialog box and select Generate Toolpath. When you preview a toolpath, CAMWorks temporarily collapses the dialog box to show only the title and menu bars.

6. 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.
7.  Click the Expand button at the top right of the collapsed dialog box to restore the dialog box.
8. On the Pattern tab, change the Limits to Start and End at Exact Surface Edge.






9. Check the Use start hint box and set Y to -25mm.
  10. 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.

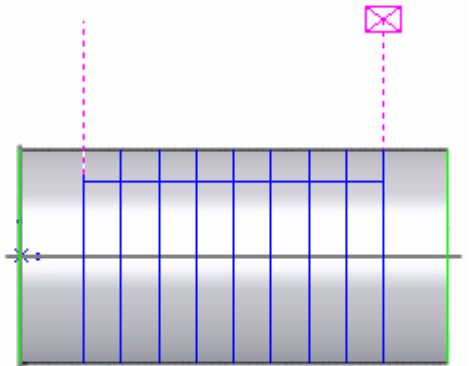


- Notice the new start point. The definition of a start point is only a hint for CAMWorks. CAMWorks tries to find the nearest possible position next to your point.

11.  Click the Expand button to restore the dialog box.

12. Change the Limits to Between Two Points.

13. Type 15mm for the X on the left (the toolpath start point) and 85mm for the X on the right (the end point).



14. 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 15mm from the right edge and the end to be 15mm from the left. Note that the distance between the passes is equally spaced (less than the 10mm specified) to ensure smooth results.

15.  Click the Expand button to restore the dialog box.




16. Click the Finish and Links tabs and look at the variety of parameters that can be set to control the toolpath. For this exercise, the defaults on these tabs 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.

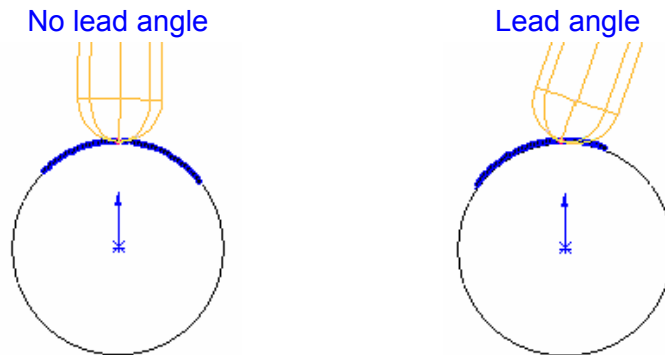
1. Click the Axis Control tab.
2. In the Cutting direction tilt options section, set the Tool axis to Normal to Surface and click OK.
3. Regenerate the toolpath.
4. Right click Multiaxis Mill1 in the tree and select Step Thru Toolpath.

The Step Thru Toolpath toolbar displays. You can step forward or backward through the toolpath one record at a time, in multiple step increments, go to the end or beginning of the toolpath, and set the number of CL records to display at a time.

5.  Click the Display Tool button to turn on the wireframe display of the tool.
6.  Click the 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.
7.  Click the Close button to cancel Step Thru Toolpath.
8. Double click the operation in the tree.
9. On the Axis Control tab, set the Tool axis to Tilted Relative To Cutting Direction.
10. Set the Lead/lag angle to 15.

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

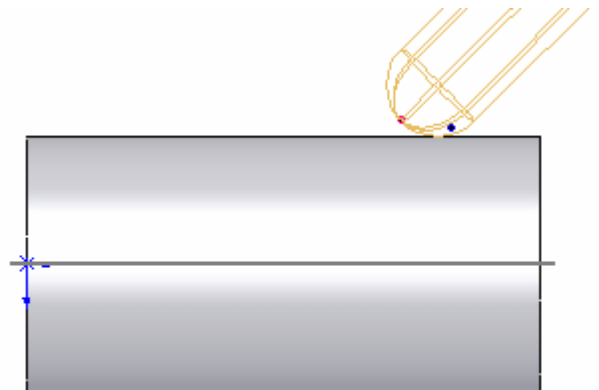


11. Click OK and regenerate the toolpaths.
12. Select Step thru Toolpath again and step through the toolpath.
13. Edit the operation and change the Tool axis to Tilted With Fixed Angle To Axis.



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.

14. Change the Tilt axis to X-Axis.
15. Set the Fixed tilt angle to 45deg.
16. Click OK and regenerate the toolpath.
17. Right click the operation in the tree and select Step Thru Toolpath.

18.  Click the Display Tool button to turn on the display of the tool.






19.  Click the Single Step button to step through the toolpath.  
The tool is now tilted towards X+ in a constant angle of 45 degrees 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.
20.  Click the Close button to cancel Step Thru Toolpath.



## ***Post Processing Toolpaths***

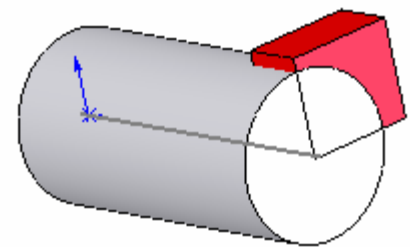
Post processing is the final step in generating the NC program file. This step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the CAMWorks Operation tree. When 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.

1.  Click the Post Process button on the CAMWorks toolbar.  
The Post Output File dialog box displays so you can name the NC program file.
2. If MULTIAX\_1 is not in the File name text box, type **MULTIAX\_1**, then click Save.  
You do not have to type the extension if you are using the default *.txt*. Naming the post output file the same as the part file is the most common way of saving parts and NC programs. Both files can have the same name because they have different extensions.
3. In the Post Process Output dialog box, click the Run button on the control bar at the top.
4. When the post processing is finished, view the code using the vertical scroll bar.
5. Click OK to close the Post Process Output dialog box.

# Multiaxis 2

## What You'll Learn

- Using Collision Detection options
  - Controlling the Start and End of the Toolpath
1. Open the part file **MULTIAX\_2.SLDPRT** in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\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 tree and notice that no faces have been selected to avoid.  
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 exercise. Any faces that are selected to avoid in the Multi Surface Feature dialog box are ignored.
  4. Set the Attribute to 5 Axis, then click Close.
  5.  Click the Generate Operation Plan button on the CAMWorks toolbar. CAMWorks generates the operation.
  6. Double-click Multiaxis Mill1 in the tree.
  7. Click the Tool Crib tab and click the Add button.
  8. Select Ball Nose in the Tool Select Filter dialog box and click OK.
  9. In the Tools Database form, highlight a 20mm tool and click OK.
  10. Highlight the tool in the Tool Crib, click the <- Select button, then click Yes to replace the holder.
  11. On the Pattern tab, set the Pattern to Cuts Across Curve and set the Max. stepover to 10mm.
  12. Click the Curve button.
  13. In the Curve Wizard: Select Geometry dialog box, select Sketch2 and click Finish.



## 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. On the Axis Control tab, set the Tool axis to Normal to Surface.
2. Click the Preview button and notice the toolpath ignores the red faces and cuts through them.

3.  Click the Expand button to restore the dialog box.

4. Click the Gouge Checking tab.

5. In the Gouge Checking section on the Group 1 tab, check the box next to Apply gouge checking to and 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.

6. In the Check against section, remove the check mark from the Feature surfaces option.
7. Select the Other surfaces option and click the  button.

The Avoid Surfaces dialog box displays and you can identify the surfaces to check for collisions. There are two methods for adding avoid surfaces 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.

8. Click the Create Features button. The Insert Multi Surface Feature dialog box displays.
9. Pick each face on the red box.
10. Click Insert, then click Close.

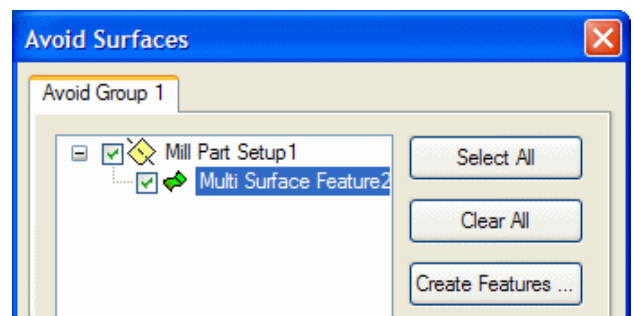
Multi Surface Feature2 is now listed in the Avoid Surfaces dialog box.




11. Make sure the box next to the feature is checked and click OK.
12. In the Operation Parameters dialog box, set the Gouge check 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.

13. 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.



14.  Click the Expand button to restore the dialog box.
15. Change the Strategy to Remove Gouge Positions and set the Allowance to 2mm.
16. Click Preview and view the results.  
The avoid surfaces are bypassed with some rectangular motions.
17. Change the orientation to Right view to see the offset distance from the avoid surfaces.
18.  Click the Expand button.
19. On the Pattern tab, check the box next to Reverse stepover and change the Limits to Start and End at Exact Surface Edge.
20. On the Axis Control tab, set the tool axis to Tilted With Fixed Angle to Axis.
21. Set the Tilt axis to X-Axis and set the Fixed tilt angle to 45deg.
22. On the Gouge Checking tab, set the Strategy to Move Tool Away.
23. Set the Retract tool to Along YZ Plane.
24. 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 are done on the YZ plane, maintaining a constant X value.
25.  Click the Expand button.
26. On the Gouge Checking tab, set the Strategy to Stop Toolpath Calculation and click Preview.  
CAMWorks generates a partial toolpath. The calculation stopped when a gouge was detected. Only the non-gouging motions calculated before the gouge are generated.


### ***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 Expand button.
2. On the Gouge Checking tab, set the Strategy to Remove Gouged Positions.
3. Click the Entry/Retract tab and set the following options:
  - Clearance = Cylinder About X (the clearance shape is a 40mm radius cylinder with the center passing through the origin point)
  - Radius = 40mm
  - 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

4. 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).

5.  Click the Expand button.
6. Click the Pattern tab and change the Direction pattern to Zig.
7. Check the box next to Start hint and set the Rotate next cut by option to 10deg.
8. Click the Entry/Retract tab.
9. Set the Leadin move to Use Leadin and the Leadout move to Use Leadout.
10. On the Leadin and Leadout tabs, set the following:
  - Type = Tangent arc
  - Parameters = Arc
  - Arc angle = 90deg
  - % Tool diameter = 50
  - Height = 0.001mm
11. 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.



---

## Chapter 2 Learning More About Multiaxis

---

This chapter provides an opportunity to learn more about CAMWorks Multiaxis Machining. The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

---

**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

### What You'll Learn

- 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.

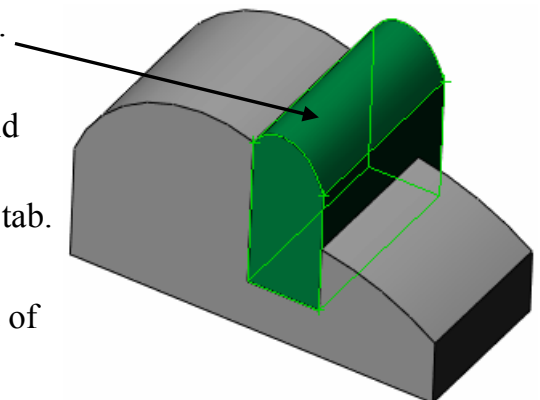
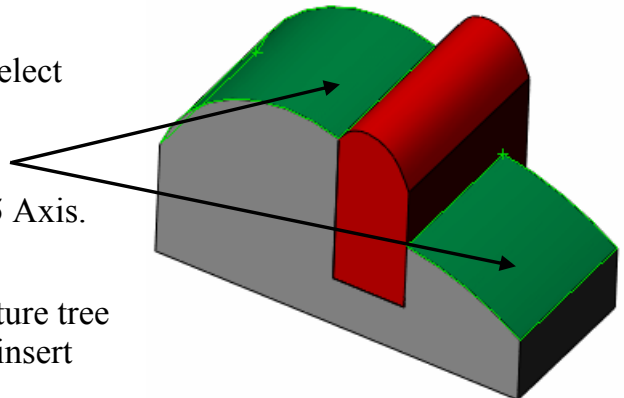
1. Open the part file **MULTIAX\_3.SLDPRT** in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\4-5AxisMill*).

2.  Click the CAMWorks Feature Tree tab.
3.  Double click Example Mill-in in the Feature tree.
4. Highlight Example Mill-in on the Machine tab and click Select.
5. Click the Controller tab, highlight CW5AXIS and click Select.
6. Click OK.
7. Right click Stock Manager in the tree and select Insert Mill Part Setup.
8. Select Top for the Reference plane and click the Reverse direction option.
9. Click OK.
10. Right click Mill Part Setup1 in the tree and select Insert Multi Surface Feature.
11. Pick the two top gray faces.
12. Click the arrow next to Attribute and select 5 Axis.
13. Click Insert.


Multi Surface Feature1 is inserted in the Feature tree and the dialog box remains open so you can insert additional features.

14. Pick the five faces on the red solid and click Insert.
15. Click Close.
16. Rename Multi Surface Feature2 in the tree to Avoid Feature.
17. Click the SolidWorks FeatureManager design tree tab.
18. Right click Extrude2 and select Hide.

You will enable collision detection in the next part of this exercise.



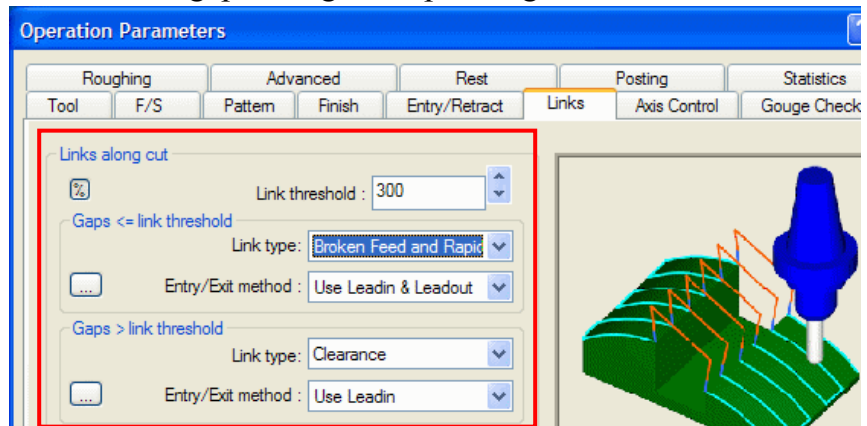


19.  Click the CAMWorks Feature Tree tab.
20. Right click Multi Surface Feature1 in the tree and select Generate Operation Plan on the shortcut menu.

### **Defining Tool Movement When Gaps are Encountered**

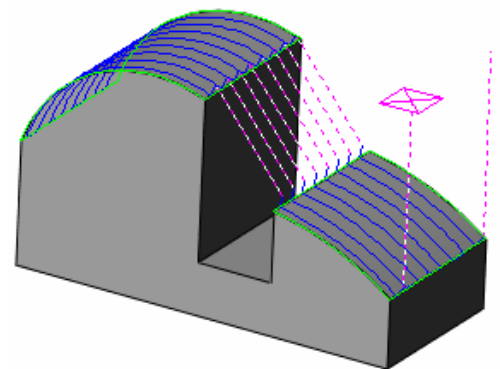
1. Double click Multiaxis Mill1 in the Operation tree.
2. On the Pattern tab, set the following:
  - Pattern = Slice
  - Cut angle (XY) = -180deg
  - Cut angle (Z) = -90deg
  - Max stepover = .15in
3. On the Entry/Retract tab, set the Clearance Type to Plane in Z and set Z to 2in.
4. On the Links tab, for Links along cut, set the Link threshold to 300.

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 settings for Gaps > link threshold.




5. For Gaps <= link threshold, set the following:
  - Link type = Broken Feed and Rapid
  - Entry/Exit method = Use Leadin & Leadout
6. For Gaps > link threshold, set the Link type to Clearance.
7. Click the Preview button.

With the Broken Feed and Rapid 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.



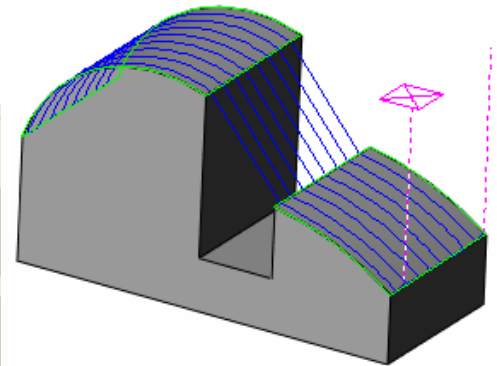
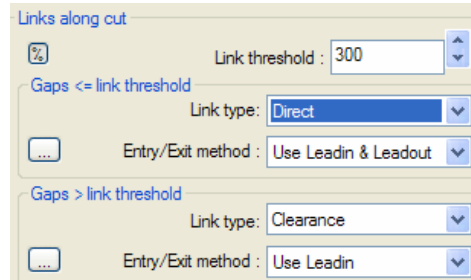
**Broken Feed and Rapid**

8.  Click the Expand button at the top right of the collapsed dialog box to restore the dialog box.



9. For Links along cut, change the Link type for Gaps  $\leq$  link threshold to Direct and click the Preview button.

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.

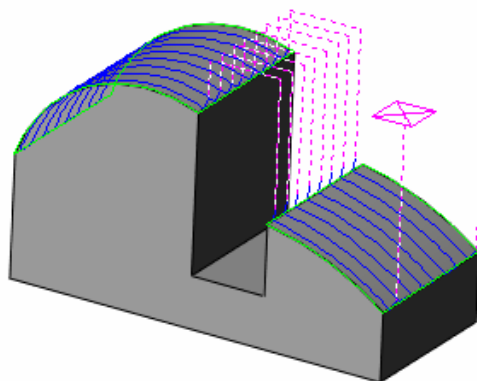


Direct

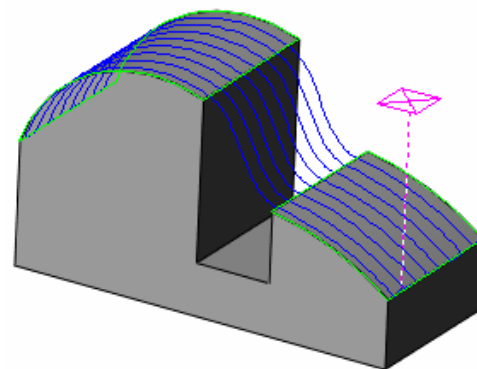
10.  Click Expand and change the Link type to Clearance.
11. Click the Preview button and notice that the toolpath retracts to the Clearance area. The tool rapids back to the Clearance plane. Only the return to the surface has machining speed.
12.  Click Expand, change the Link type for Gaps  $\leq$  link threshold to Blend Spline and click Preview.

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.

13.  Click Expand.




Clearance

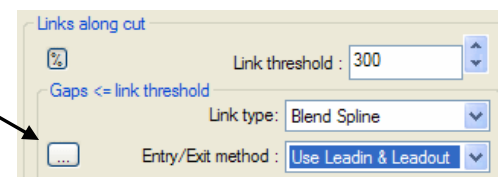


Blend Spline

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

1. Click the  button to the left of the Entry/Exit method for Gaps  $\leq$  link threshold.



The Operation Parameters dialog box displays the Leadin and Leadout parameters that can be set.

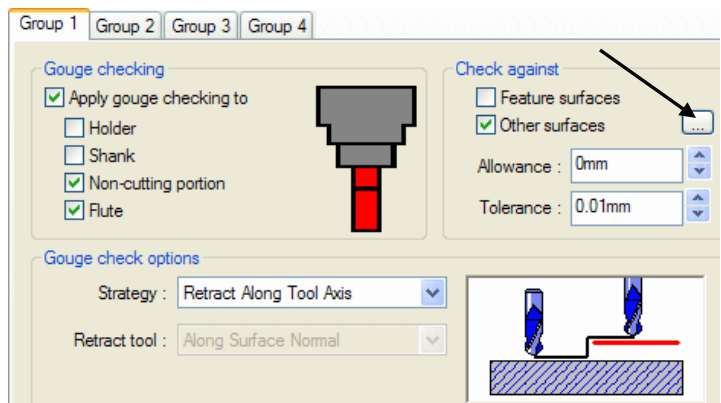
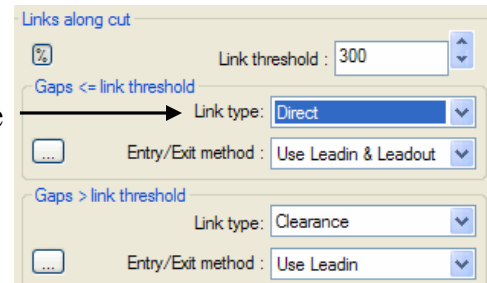


2. Select a Leadin and Leadout Type and the parameters, then click OK.

3. Click OK to close the Operation Parameters dialog box and regenerate the toolpath.
4. Notice the difference in the leadin and leadout.
5. 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.
3.  Click the Operation tree tab.
4. Double click Multiaxis Mill1.
5. On the Links tab, in the Links along cut section, set the Link type to Direct for Gaps <= link threshold.
6. On the Gouge Checking tab, select the Apply gouge checking to option on the Group 1 tab.
7. In the Gouge check options, set the Strategy to Retract Along Tool Axis.
8. In the Check against section, remove the check mark from the Feature surfaces option.
9. Select the Other surfaces option and click the  button.
10. Click the Select All button and click OK.
11. Click the Preview button and notice that the toolpath avoids the red surface.





# Multiaxis 4

## What You'll Learn

### 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.

1. Open the part file **MULTIAX\_4.SLDPRT** in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\4-5AxisMill*).
2.  Click the CAMWorks Feature Tree tab.
3.  Double click Example Mill-in in the Feature tree.
4. Highlight Example Mill-in on the Machine tab and click Select.
5. Click the Controller tab, highlight CW5AXIS and click Select.
6. Click OK.
7. Right click Stock Manager in the tree and select Insert Mill Part Setup.
8. Select Top for the Reference plane and click the Reverse direction option.
9. Click OK.

Insert the Multi Surface feature:

1. Right click Mill Part Setup1 in the tree and select Insert Multi Surface Feature.
2. Pick all the blue faces.
3. Set the Attribute to Undercut - 5 Axis.
4. Click Insert, then click Close.

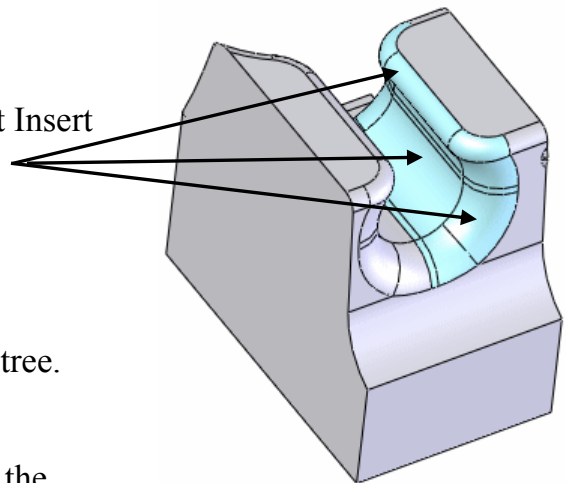
Multi Surface Feature1 is inserted in the Feature tree.

Generate the operation and adjust parameters:

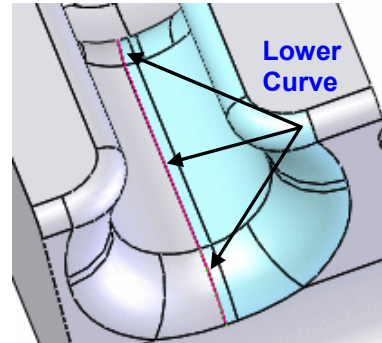
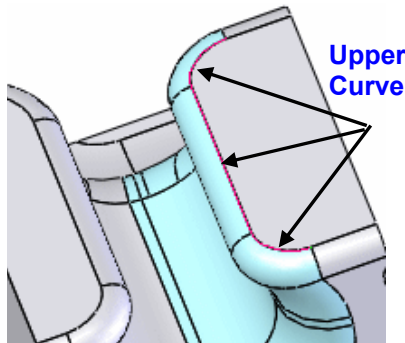
1.  Click the Generate Operation Plan button on the CAMWorks toolbar.

CAMWorks generates the operation.

2. Double click Multiaxis Mill1 in the Operation tree.
3. Click the Tool Crib tab and click the Add button.
4. Select Lollipop in the Tool Select Filter dialog box and click OK.




5. In the Tools Database form, highlight the lollipop tool and click OK.
6. Scroll down and highlight the tool in the Tool Crib, click the <- Select button, then click Yes to replace the holder.
7. On the Lollipop Tool tab, change the Shank diameter (D2) to .15in and the Diameter (D1) to .4in.
8. 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.
9. Click the Upper button.
10. Pick the three edges at the top of the feature and click Finish.
11. Click the Lower button.
12. Pick the three edges at the bottom of the feature and click Finish.

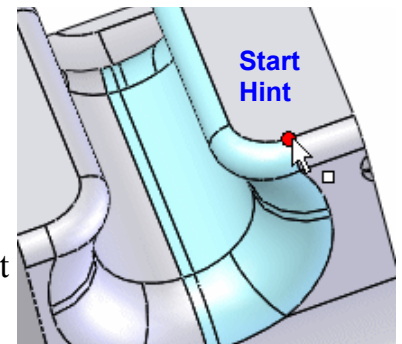


13. On the Pattern tab, set the following:
  - Limits Method = Start and End at Exact Surface Edges
  - Max stepover = .02in

14. Check the Use start hint option.

15.  Click the Define Point button and pick the vertex on the part shown in the figure on the right and click OK.

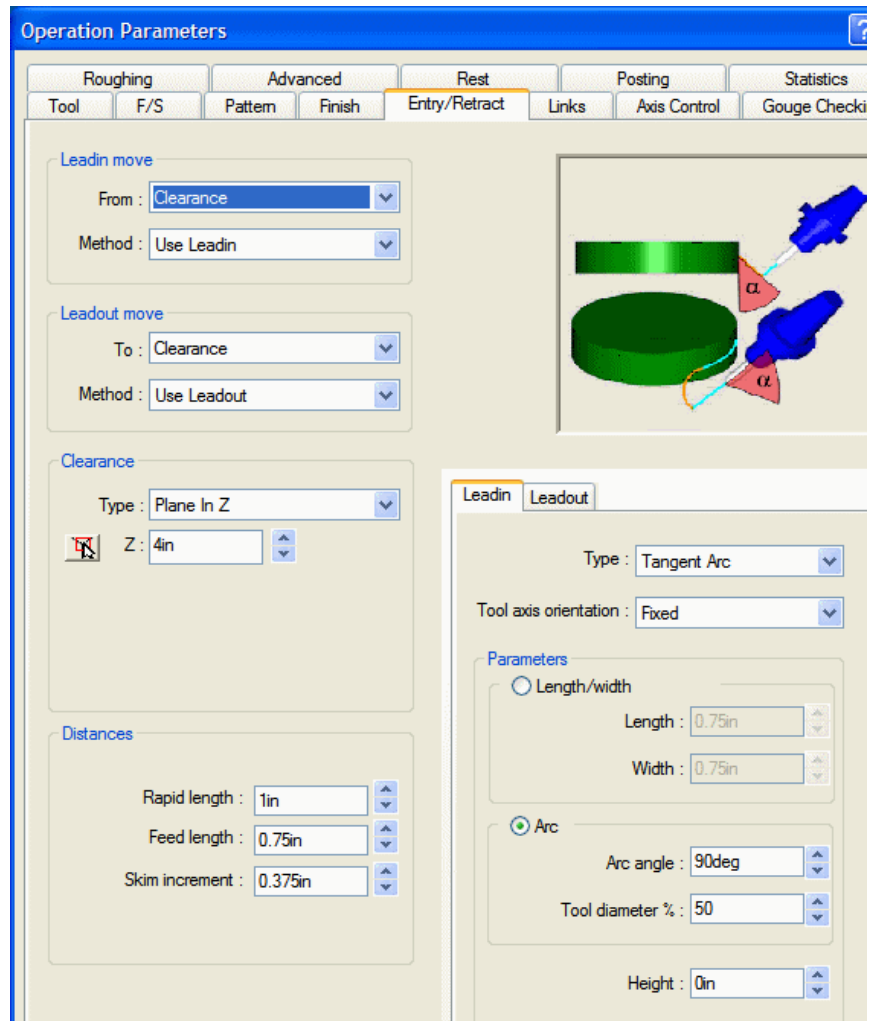
The Start hint option provides more control over the XY entry position of the tool. Selecting an entry position does not mean that the tool will actually enter at the position, but the entry location of the tool will be as close as possible to the start hint location.



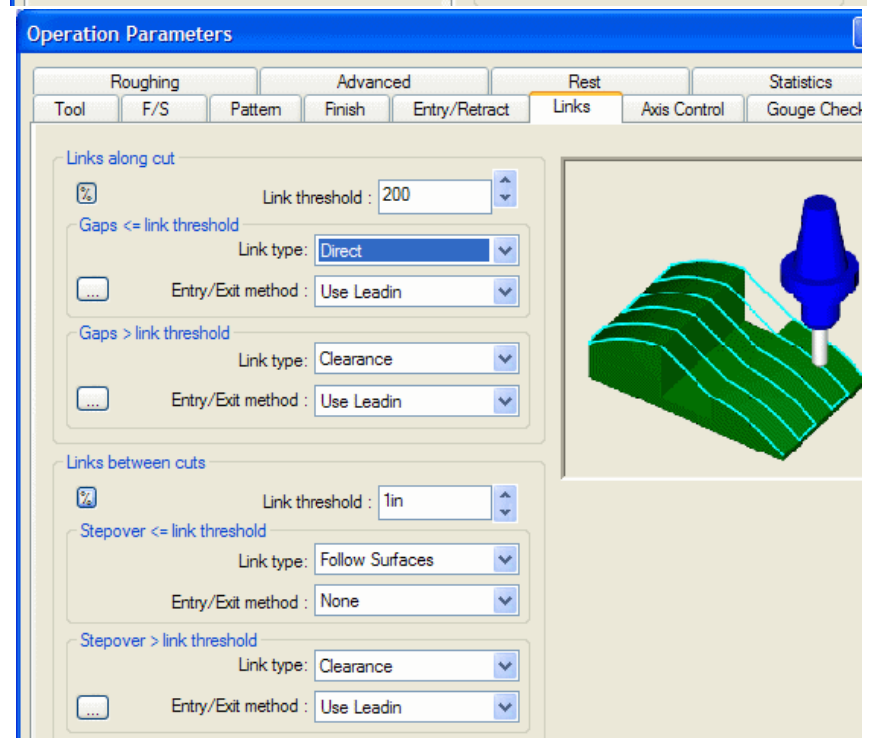
16. Click the Finish tab and set the Chaining tol. to .1in.


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.

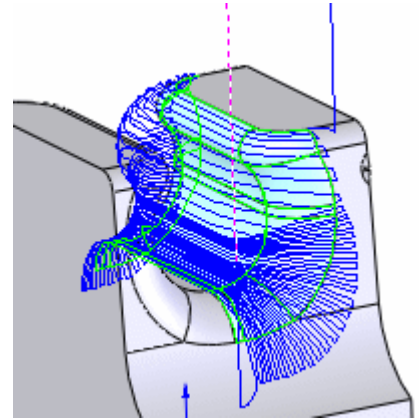
17. Click the Entry/Retract tab and set the parameters as shown in the figure on the right. Set the parameters on the Leadout tab the same as the Leadin.



18. Click the Links tab and set the parameters as shown in the figure on the right.



19. Click the Axis Control tab and set the following:
  - Number of axis = 3 Axis
  - Tool contact point = Auto
20. Click OK to close the Operation Parameters dialog box.
21.  Click Generate Toolpath on the CAMWorks toolbar.
22. Run Toolpath Simulation and Step Thru Toolpath.







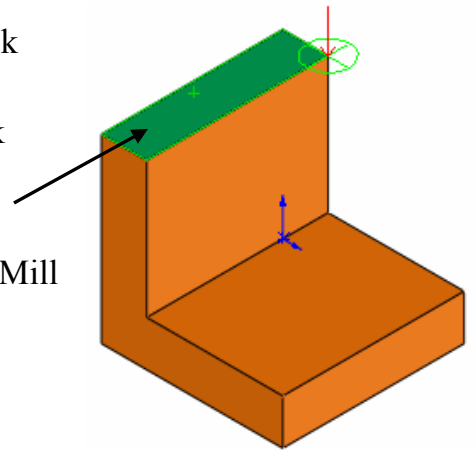
# Multiaxis 5

## What You'll Learn

### Changing Machining Parameters to Produce Different Results

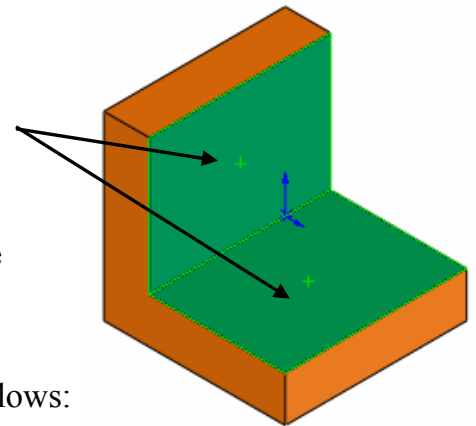
Different results occur depending on how the parameters are set up. It is possible to machine this part using different approaches.

1. Open the part file **MULTIAX\_5.SLDPRT** in the *\Examples\4-5AxisMill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\4-5AxisMill*).
2.  Click the CAMWorks Feature Tree tab.
3.  Double click Example Mill-in in the Feature tree.
4. Highlight Example Mill-in on the Machine tab and click Select.
5. Click the Controller tab, highlight CW5AXIS and click Select.
6. Click OK.
7. Right click Stock Manager in the tree and select Insert Mill Part Setup.
8. Pick the top face as shown and click OK.



Insert the Multi Surface feature:

1. Right click Mill Part Setup1 and select Insert Multi Surface Feature.
2. Select the two surfaces as shown.
3. For the Attribute, select 5Axis.
4. Click Insert and then click Close.

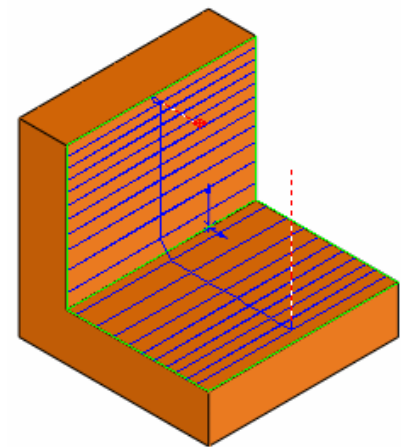
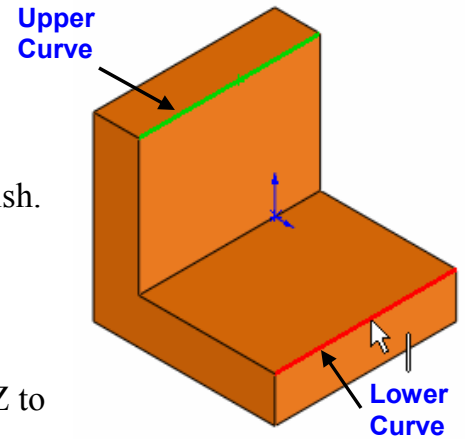


Generate the operation and adjust parameters:

1. Right click Multi Surface Feature 1 and select Generate Operation Plan.
2. In the Operation tree, double click Multiaxis Mill.
3. Click the Mill Holder tab and change the settings as follows:
  - Top diameter (D1) = 4in
  - Bottom diameter (D2) = 2in
  - Overall length (L1) = 4in
  - Bottom length (L2) = 2in
  - Protrusion = 1.5in



4. Click the Pattern tab.
5. For the Pattern, pick Flowline Between Curves.
6. Click the Upper button, pick the green edge and click Finish.
7. Click the Lower button, pick the red edge and click Finish.
8. Set the Max. stepover to .1in.
9. Click the Entry/Retract tab.
10. For the Leadout move, set the Method to None.
11. In the Clearance section, set the type to Plane in Z and Z to 10in.
12. In the Distances section, set the Rapid length to .5in and the Feed length to .1in.
13. Click the Preview button.







Notice the toolpath starts and finishes at the center of the part.


14. Expand the dialog box and click the Links tab.
15. Under Links along cuts, notice both the Gaps  $\leq$  and Gaps  $>$  link threshold are set to Direct.

The surfaces defining the workpiece can have gaps and holes. If gaps are detected along a toolpath segment, you have a choice of ignoring the gap and moving the tool to connect the two sides of the gap or retracting the tool back to the rapid plane and skipping the gap, then coming back from the rapid plane to the other side of the gap and continuing machining. The limit for ignoring the gap is defined as a percent of the tool diameter.


16. Change the Gaps  $>$  link threshold to Clearance.
17. Click OK and generate the toolpath.

18.  Click the Simulate Toolpath button on the CAMWorks toolbar.

19.    Set the Collision buttons for the Tool, Tool Shank and Tool Holder to Cut Collision.

20.  Click the Run button.

Notice that all of the gouged areas within the part are due to the tool holder and the direction of cut.

21.  Click the X button to cancel the simulation.

Make adjustments to eliminate gouging:


1. Double click Multiaxis Mill1.
2. Click the Finish tab and check the Blend surfaces by tool radius option.

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 cause so called "fish tails" in toolpaths. These fish tails are removed using this option. This option can also be considered as a fillet generator. The surface model is rounded (filleted) in the direction of toolpath slices with a radius to avoid small radii and inner sharp corners. The applied radius is the main tool radius plus the current stock to leave value. The fillet generation is independent of tool type and shape. 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 and regenerate the toolpath.


4.  Simulate the toolpath.

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

5.  Close the simulation.
6. Double click Multiaxis Mill1.
7. Click the Axis Control tab.
8. Under Cutting direction tilt options, set the Side tilt angle to 45 deg.
9. Click OK and regenerate the toolpath.

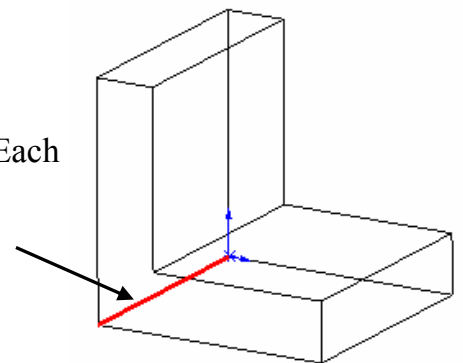
10.  Simulate the toolpath.

Notice the toolpath looks good until it starts to cut into the lower section. The tool needs to stay at the 45 degree angle throughout the toolpath.


11.  Close the simulation.
12. Double click Multiaxis Mill1.
13. Click the Axis Control tab.
14. For the Cutting direction tilt options, select Tilted From Curve for the tool axis.
15. For the Curve tilt strategy, select From Start to End for Each Contour.
16. Click the Tilt Curve button, pick the red edge and click Finish.
17. Click OK and regenerate the toolpath.

18.  Simulate the toolpath.


The toolpath looks good.

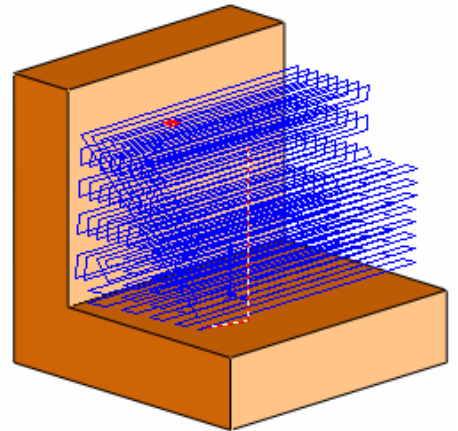


You can add some roughing passes to include the leftover:

1.  Close the simulation.
2. Double click Multiaxis Mill1.
3. Click the Roughing tab.
4. Check the Multi passes option.
5. For 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.

6. In the Roughing section, enter 8 for the Number (the number of roughing passes to be generated) and 0.1in for Spacing (the distance between each rough pass).
7. Click OK and regenerate the toolpath.
8.  Simulate the toolpath.





# Index

3 Axis cutting for undercut areas 2-6

## A

avoid features, defining 1-15

Avoid Surfaces dialog box 1-15

## C

collision detection options 1-15, 2-5

controlling toolpath start and end 1-16

CW5AXIS 1-6

## F

file

NC program 1-13

setup sheet 1-13

## G

Generate Operation plan command 1-8, 1-14, 2-6

Generate Toolpath command 1-10

## I

Insert Mill Part Setup command 1-7, 2-2, 2-6, 2-10

Insert Multi Surface Feature command 1-7, 2-2, 2-6

Insert Multi Surface feature dialog box 1-15

## M

machinable features

creating Multi Surface feature 1-7, 2-2, 2-6

defining interactively 1-7, 2-2, 2-6

Machine 1-4

Machine dialog box

Controller tab 1-6

Machine tab 1-5, 2-2, 2-6, 2-10

Posting tab 1-6

Tool Crib tab 1-5

Manage Stock dialog box 1-6

Mill Part Setup

inserting 1-7, 2-2, 2-6, 2-10

Multi Surface feature, inserting 1-7, 2-2, 2-6

Multiaxis Mill operation

Axis Control tab 1-11, 2-9, 2-12

changing parameters 2-10

collision detection options 1-15, 2-5

Cuts Across Curve Pattern 1-10

defining tool orientation 1-11

Entry/Retract tab 1-16, 2-8, 2-11

F/S tab 1-9

Finish tab 2-7, 2-12

Flowline Between Curves Pattern 2-7, 2-11

Gouge Checking tab 1-15, 2-5

handling gaps 2-3

identify avoid surfaces 1-15

Links tab 2-3, 2-8, 2-11

Pattern tab 1-9, 1-10, 2-3, 2-7

Roughing tab 2-13

Slice Pattern 1-9, 2-3

Tool Crib tab 1-9

toolpath start and end 1-16

multiaxis milling terminology 1-2

## N

NC program

file 1-13

generating 1-13

steps to generate 1-3

## O

Operation tree 1-8

operations

generating operation plan 1-8, 1-14, 2-6

generating toolpaths 1-10

post processing 1-13

simulating toolpaths 1-10, 2-11, 2-12

## P

parts used in exercises

multiax\_1.sldprt 1-4

multiax\_2.sldprt 1-14

multiax\_3.sldprt 2-2

multiax\_4.sldprt 2-6

multiax\_5.sldprt 2-10

Post Output File dialog box 1-13

Post Process command 1-13

post processor, parameters 1-6

## S

setup sheet 1-6

Simulate Toolpath command 1-10, 2-11, 2-12

Step Thru Toolpath command 1-11

steps to generate NC program in multiaxis milling 1-3

Stock, defining 1-6

## T

tool

adding from library 1-9, 1-14, 2-6

defining orientation 1-11

selecting tool crib 1-5

using lollipop for undercut machining 2-6

## Index

### toolpaths

- controlling start and end 1-16

- generating 1-10

- post processing 1-13

- simulating 1-10, 2-11, 2-12

- stepping through 1-11

### **U**

- undercut areas, machining 2-6

- Use start hint option 1-11, 2-7