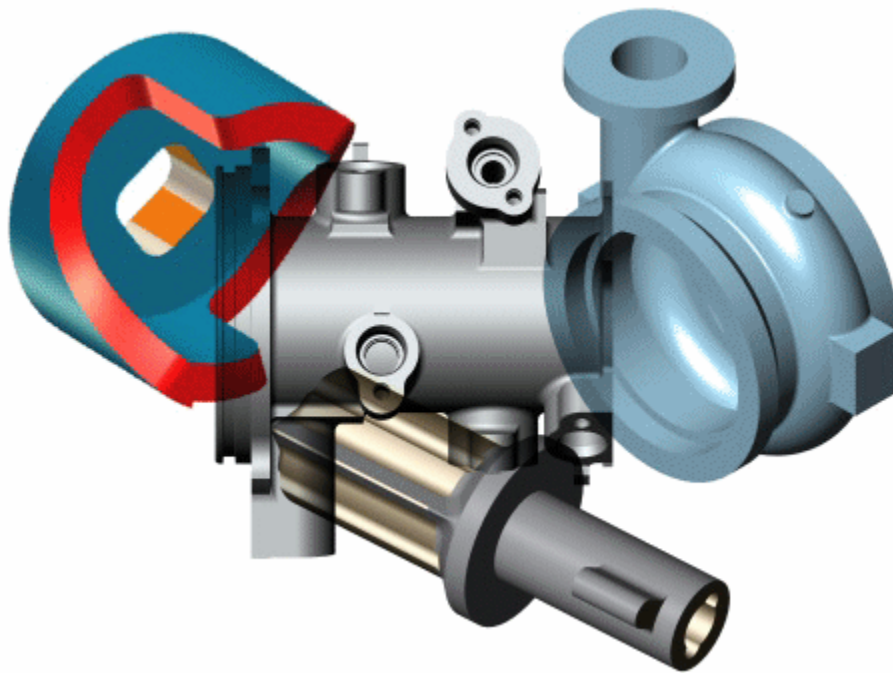




camworks
powered by SolidWorks

CAMWorks 2007 Mill/Turn 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.



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

CW70d July, 2006

Table of Contents

CHAPTER 1 LEARNING MILL-TURN BASICS	1-1
Mill-Turn 1	1-2
What You'll Learn.....	1-2
Steps to Generate Mill Toolpaths and NC Code in Part Mode.....	1-2
Step 1: Model Part in SolidWorks or Import Part.....	1-3
Step 2: Change to CAMWorks Feature Tree	1-3
Step 3: Define the Machine.....	1-4
Step 4: Define the Stock.....	1-6
Step 5: Define Machinable Features	1-7
Step 6: Generate Operation Plan / Adjust Operation Parameters	1-10
Step 7: Generate Toolpaths	1-11
Step 8: Post Process Toolpaths.....	1-13
Mill-Turn 2	1-14
What You'll Learn.....	1-14
Inserting Multiple Wrapped Features.....	1-15
Adjusting Operation Parameters / Machining Using C-Axis.....	1-15
Mill-Turn 3	1-17
What You'll Learn.....	1-17
Inserting a Wrapped Feature Using a 2D Sketch	1-17
Unwrapping a Cylindrical Face / Creating a Wrapped Feature	1-18
Adjusting Operation Parameters and Machining Using C-Axis.....	1-20
CHAPTER 2 LEARNING MORE ABOUT MILL-TURN	2-1
Mill-Turn 4	2-2
What You'll Learn.....	2-2
Inserting 2.5 Axis and Multi Surface Machinable Features.....	2-2
Inserting Contain and Avoid Areas for 3 Axis Operations	2-3
Adjusting Operation Parameters and Machining Using Y-Axis.....	2-4
Mill-Turn 5	2-5
What You'll Learn.....	2-5
Adjusting Entity Locations to Machine a Wrapped Engrave Feature ..	2-5
Inserting a Wrapped Engrave Feature to Machine Text	2-6
Inserting a Plane for Index or Rotational Angle for Mill Setups	2-7
Inserting a 2.5 Axis Wrapped Engrave Feature	2-8
Machining Using C-Axis	2-9

INDEX

Table of Contents

Chapter 1 Learning Mill-Turn Basics

This chapter provides an opportunity to learn CAMWorks Mill-Turn 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 Mill-Turn and assume you are familiar with CAMWorks Mill and Turn. The *CAMWorks Mill & Turn 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.

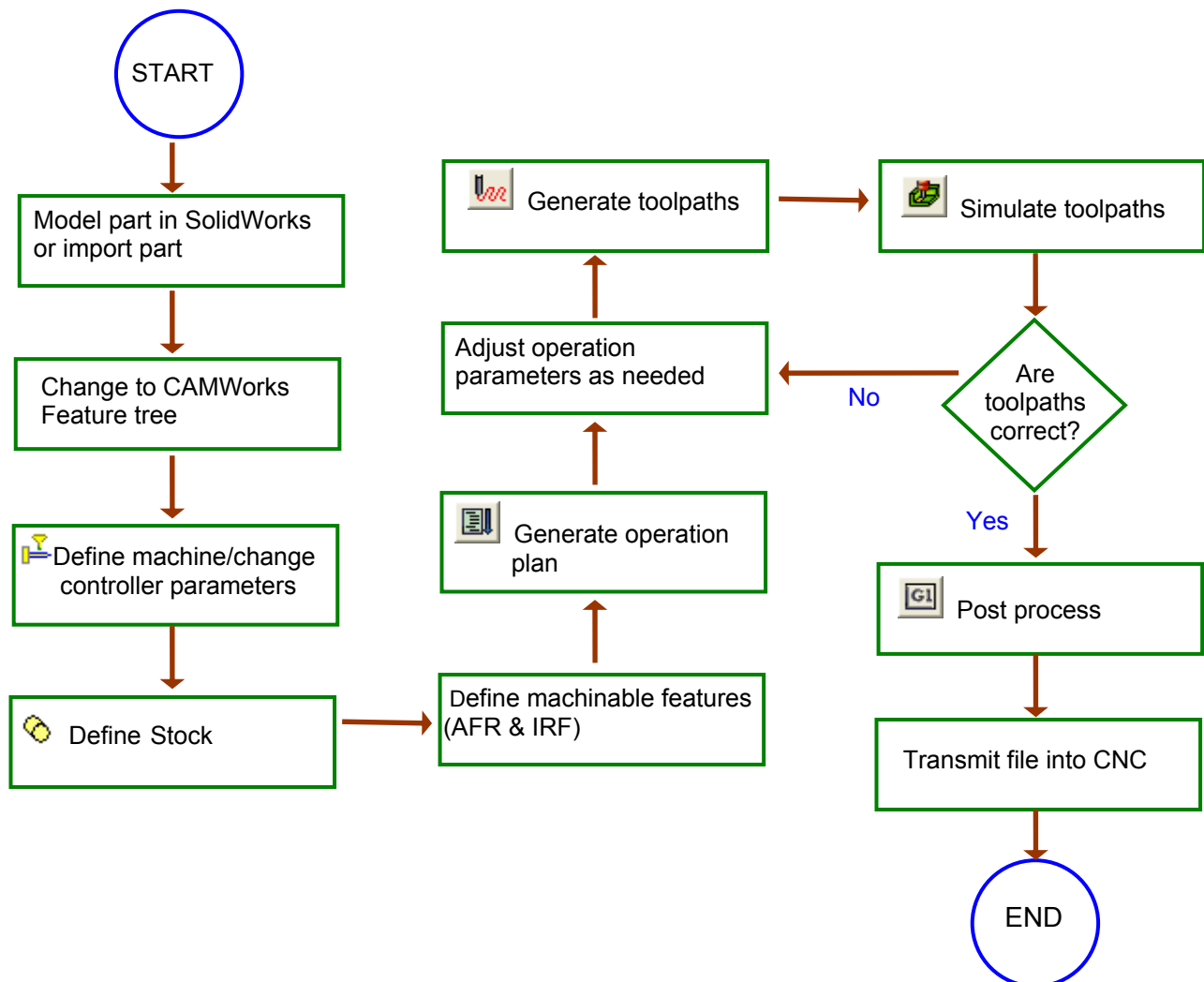
Mill-Turn 1

What You'll Learn

Steps to Generate Mill Toolpaths and NC Code in Part Mode

The following steps are used to generate Mill-Turn 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. 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.



Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill-Turn*).

Step 1: Model Part in SolidWorks or Import Part

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.

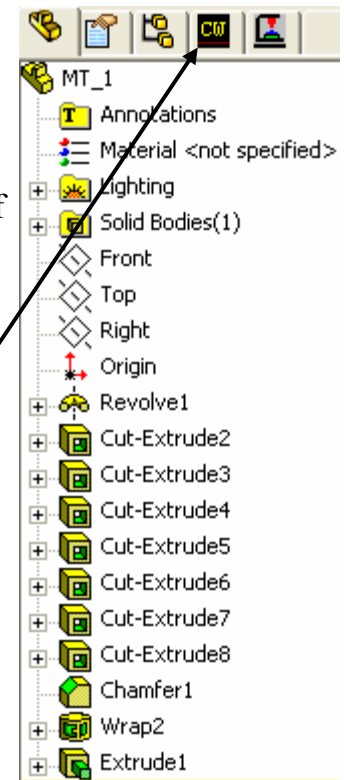
1. Open the part file **MT_1.SLDPRT** in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill-Turn*).

The FeatureManager design tree lists the features, sketches, planes and axes in the part.

The tabs at the top or bottom of the tree are for moving between the SolidWorks trees and the CAMWorks trees.

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 from the graphics area. When the cursor changes to a bar, drag the bar to the right until the tabs display.

CAMWorks Feature Tree tab



Step 2: Change to CAMWorks Feature Tree

1. Click the CAMWorks Feature Tree tab.

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


The CAMWorks machining trees provide an outline view of the machining information for the model. As you follow the steps to generate an NC program, the Feature tree expands to include Mill Part Setups and machinable features.

Step 3: Define the Machine

The Machine includes information that identifies what to machine, how to machine it, and the format of the NC output. The icons that display in the tree identify the current Machine:

 Mill Machine  Turn Machine  Mill-Turn Machine  Wire EDM

Define the machine:

1.  Right click the Machine item in the CAMWorks Feature tree (e.g., Mill machine – inch).

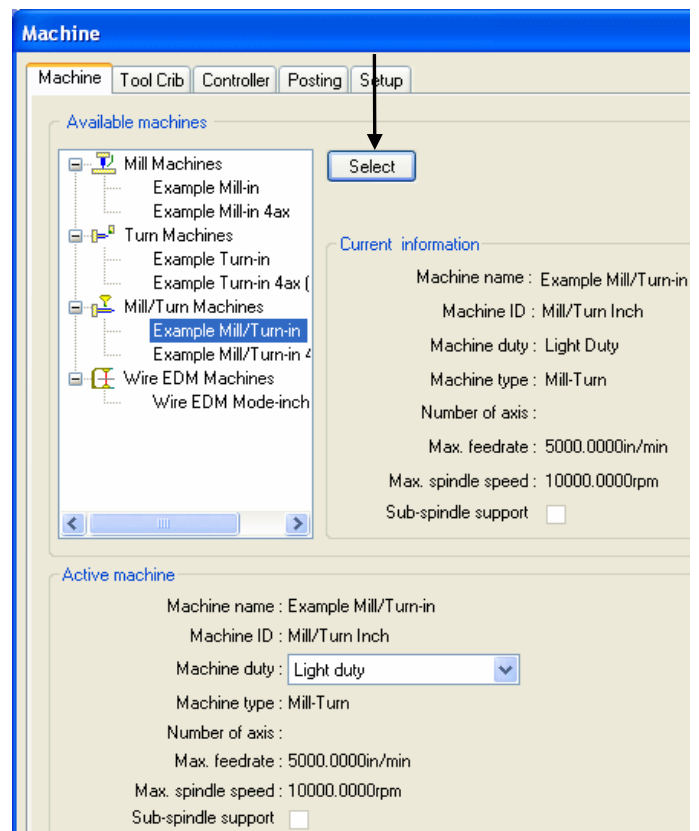
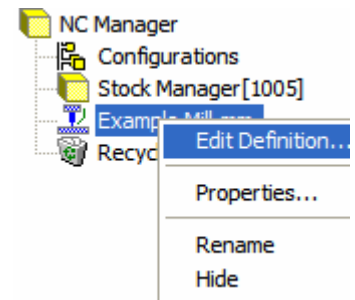
The shortcut menu displays. Right-click shortcut menus display commands that are appropriate for the item that is highlighted in the tree.

2. Select Edit Definition on the shortcut menu.

The Machine dialog box displays the Machine tab. The default machine is defined 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.

3. In the Available machines list, highlight Example Mill/Turn-in and click the Select button.



4. Click the Tool crib tab and make sure MT 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.

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

Machine

Machine | **Tool Crib** | Controller | Posting | Setup

Available tool cribs

- MT Crib 1
- MT Crib 2 (Fadal 4020)
- MT Crib 3 (Haas)
- MT Crib 4 (english)
- Inch MT Assembly

Current Information :

Name: MT Crib 1

No. of stations: 50

Save... Select

Tool crib priority ☐

Turret type: Rear Turret

Active tool crib : MT Crib 2 (Fadal 4020)

Usage	Stn. No.	Tool Type	Dia.	Rad.	Tip Angle	Class	Ct
	1	Flat End	0.0625in	0in		HSS	1/1
	2	Flat End	0.0937in	0in		HSS	3/3
	3	Flat End	0.125in	0in		HSS	1/8
	4	Flat End	0.1875in	0in		HSS	3/1
	5	Flat End	0.25in	0in		HSS	1/4
	6	Flat End	0.3125in	0in		HSS	5/1
	7	Flat End	0.375in	0in		HSS	3/8
	8	Flat End	0.218in	0in		HSS	7/3
	9	Flat End	0.25in	0in		HSS	1/4
	10	Flat End	0.3125in	0in		HSS	5/1
	11	Flat End	0.5in	0in		HSS	1/2

Add... Remove Edit... Save...

5. 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).

6. If FANTUTLM (the tutorial Mill-Turn post processor) is not highlighted, highlight it in the list and click the Select button. FANTUTLM 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.

Machine

Machine | Tool Crib | **Controller** | Posting | Setup

Available

C:\Program Files\CAMWorks2007\Posts\FANTUTLM.ctl

- FANTUTLM
- TUT4AXLM

Browse

Select

Current information:

Parameter	Value
Machine Name	TUTORIAL LATHE/MILL
Controller Type	GENERIC FANUC
Z Home	20.00000"
C Home	0.00000"
Traverse Rate	250

7. 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.

Machine	
Machine	Tool Crib
Controller	Posting
Setup	
Parameter	Value
Program number	1
Material	
Z Preset	10.00000"
X Preset	10.00000"
Maximum RPM	3000
Maximum Arc Deviation	0.00100"

8. Click OK to close the dialog box.

Step 4: Define the Stock

The stock is the material from which the part will be machined. The default stock is the smallest cylinder (bar stock) 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 length and/or diameter of the bar stock from the part or by defining the Stock from a closed sketch (for a casting or forging).

In this exercise, you define the stock as a cylinder (bar stock) box offset from the part in length.

1. Double click Stock Manager in the tree or right click Stock Manager and select Edit Definition on the shortcut menu.

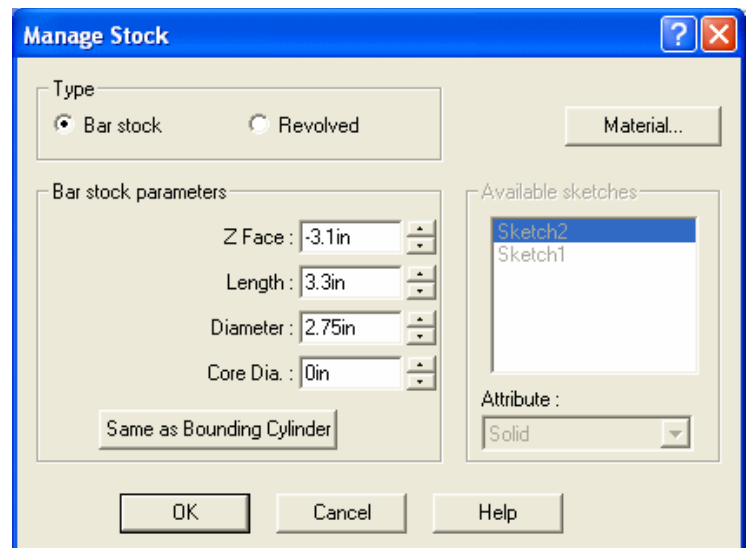
The Manage Stock dialog box displays.

2. For the Type, use the default Bar stock and set the following parameters:

- Z Face = -3.1in
- Length = 3.3in
- Diameter = 2.75in
- Core Dia. = 0in

3. Click OK to close the dialog box.

The Stock is updated in the graphics area.



Step 5: Define Machinable Features

In CAMWorks, machining can be done only on machinable features. You use the following two methods to define machinable features:

- Automatic Feature Recognition (AFR)
Automatic Feature Recognition analyzes the part shape and attempts to define most common mill and turn machinable features. Depending on the complexity of the part, AFR can save considerable time in defining features.
- Interactively created features
If AFR does not recognize features you want to machine, you can use the Insert Turn, Insert 2.5 Axis Feature, Insert Multi Surface Feature and Insert Wrapped Feature commands to define the features.

Using Automatic Feature Recognition (AFR)

Define machinable features automatically:

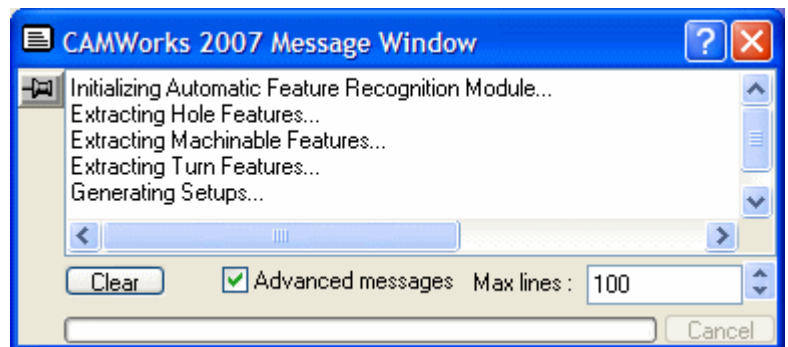


Click the Extract Machinable Features button on the CAMWorks toolbar.

The CAMWorks Message Window displays. This window displays automatically to report the progress and status of the current process. Generating Setups is the last item.

You can control whether this window displays temporarily or permanently by selecting the

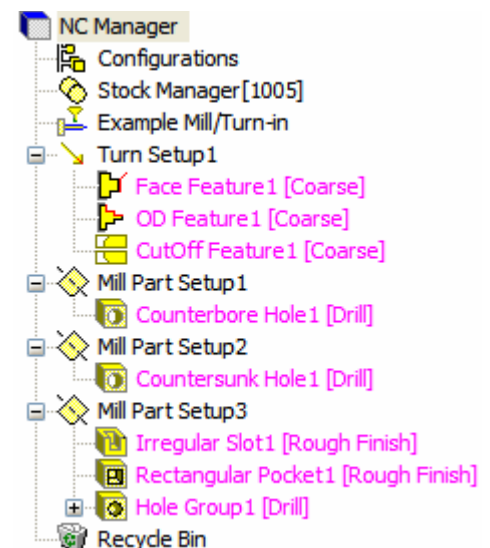
Options command on the CAMWorks menu and checking the Message Window option on the General tab in the Options dialog box.



CAMWorks generates the Turn Setups, Mill Part Setups and machinable features. The items display in the CAMWorks Feature tree.

Turn Setups and Mill Part Setups are 2 axis planes that the tool movement will be based on. They have an origin location and direction vectors. A Mill Part Setup is created for each different tool orientation. Turn Setups and Mill Part Setups are created automatically; however, you can move the origin and change the direction and angles of the axes.

For each Mill Part Setup and Turn Setup, the machinable features are listed in the order in which they were recognized.





Did You Know ...

Features that have no operations and features that could not produce operations because the feature conditions have not been defined in the Technology Database display in a different color. You can set the color on the Display tab in the Options dialog box.

4. Right click Cutoff Feature1 in Turn Setup1 in the tree, select Delete on the shortcut menu, then click Yes. The part will be held in a chuck so this feature is not needed.
5. Right click Irregular Slot1 under Mill Part Setup3 in the tree, select Delete on the shortcut menu, then click Yes. You are going to define this feature as a Corner Slot in Mill Part Setup1.

Defining Features Interactively

Automatic Feature Recognition can save a significant amount of time; however, AFR does have limitations. AFR cannot recognize every feature on complex parts and does not recognize some types of features, such as wrapped or rotary features. To machine these areas, you need to define machinable features interactively. In this exercise, you insert 2.5 Axis Features and a Wrapped Feature.

Insert 2.5 Axis Feature in Mill Part Setup1

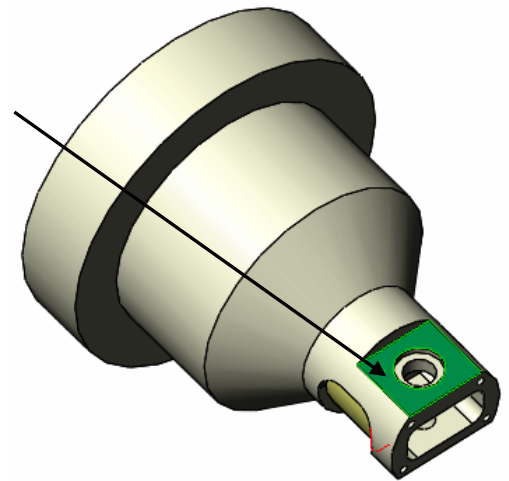
1. Right click Mill Part Setup1 in the CAMWorks Feature tree and select Insert 2.5 Axis Feature on the shortcut menu.

The 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box displays.

2. Click the down arrow next to the Type list box and select Corner Slot.
3. Make sure Single is selected.
4. Rotate the part and pick the flat face on the OD of the part.

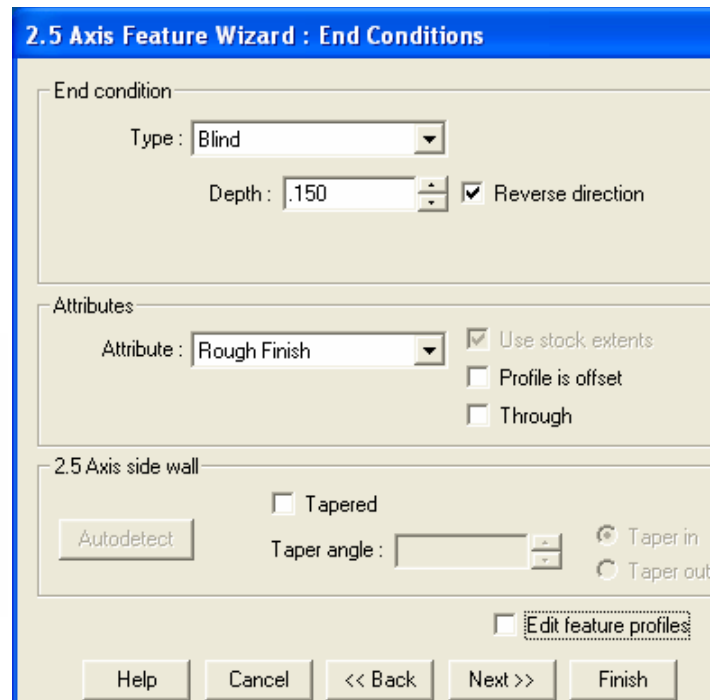
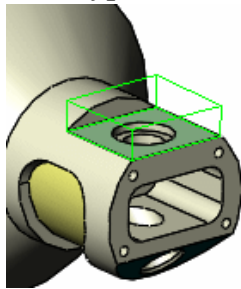
This will be a Y axis cut.

5. If the Check for taper & fillets option is checked, remove the checkmark.
If the feature you are defining does not contain any fillets at the base of the feature, you can disable Check for taper & fillets to improve the performance of the wizard.
6. Click Next.



The 2.5 Axis Feature Wizard: End Conditions dialog box displays. This dialog box allows you to determine how CAMWorks calculates the depth of the feature and select an attribute that defines a unique machining sequence.

7. Leave the End condition Type set to Blind.
8. Set the Depth to .150.
9. Make sure the Direction is correct on the feature. If not, click the check box next to the Reverse direction option.



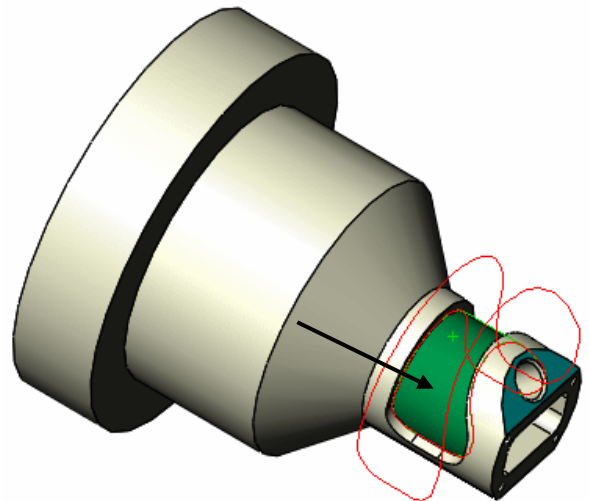
10. Leave the Attribute set to Rough Finish.
11. Remove the check mark from the Edit feature profiles option.
12. Click Finish.
13. Click Close to close the 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box.

Rectangular Corner Slot1 displays in the CAMWorks Feature tree.

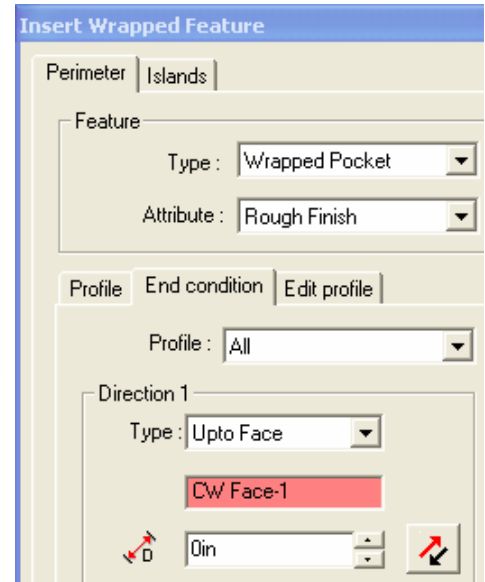
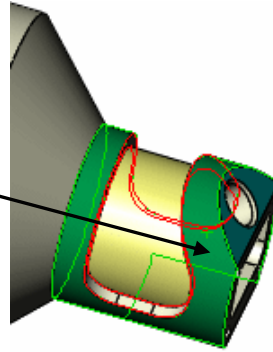
Insert Wrapped Feature in Mill Part Setup1

A Wrapped Feature is defined as a pocket, boss, slot, open profile, or engrave feature whose geometric shape lies on a constant radius cylinder, where the center line of the cylinder is the same as the turn axis of rotation. Wrapped features would be machined by C-axis motion.

1. Right click Mill Part Setup1 in the tree and select Insert Wrapped Feature on the shortcut menu.
2. Rotate the part and pick the cylindrical face on the part. This will be a C axis cut.

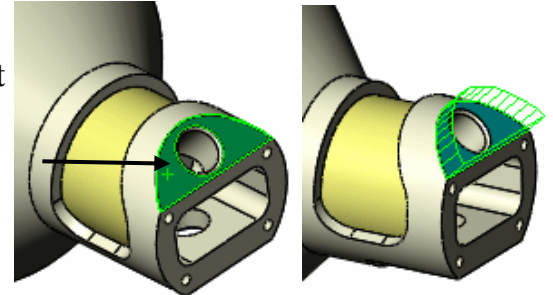


3. In the Insert Wrapped Feature dialog box, click the End Condition tab.
4. Pick anywhere on the small OD of the part to set the depth.
5. Click the Insert button, then click Close.




Insert a 2.5 Axis Feature in Mill Part Setup2

1. Right click Mill Part Setup2 in the tree and select Insert 2.5 Axis Feature on the shortcut menu.
2. In the 2.5 Axis Feature Wizard, select Corner Slot for the Type.
3. Rotate the part, if necessary, and pick the angled face on the part. This will be a B axis cut.
4. Click Next.
5. Leave the End condition Type set to Blind.
6. Set the Depth to .150.
7. If the Edit feature profiles option is checked, remove the check mark.
8. Click Finish, then click Close to exit the dialog box.



Step 6: Generate Operation Plan and Adjust Operation Parameters

An Operation Plan contains information on how each machinable feature is to be machined and how the NC code will be output. When Generate Operation Plan is run, operations for each machinable feature are created automatically based on information in the Technology Database. In some situations, the operations defined for a feature in the Technology Database may not be sufficient and additional operations may be required. You can insert operations manually. For more information, see the online help.

1.  Click the Generate Operation Plan button on the CAMWorks toolbar. Clicking this toolbar button is the same as selecting the command from the NC Manager level, regardless of the active item in the tree.

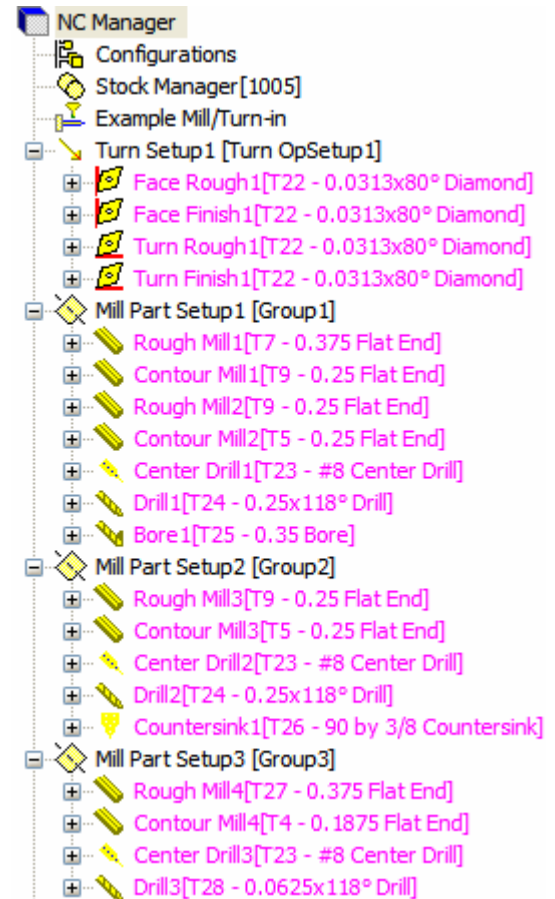
CAMWorks generates the operation plan for all the machinable features and the Operation tree displays.

The CAMWorks Operation tree provides an outline view of the operations for the machinable features. Operations are listed under the Mill and Turn Part Setups in the same order as the machinable features in the Turn and Mill Part Setups.

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.


The 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. Each operation contains parameters that affect how the toolpath is created and specific parameters that will be output to the NC program. These parameters can be edited before generating the toolpaths and post processing the part.

2. Right click Rough Mill4 in Mill Part Setup3 and select Edit Definition.
3. Click the Tool Crib tab.
4. Highlight a .25in Flat End Mill in the tool list and click the <-- Select button.
5. Click Yes to replace the corresponding holder, then click OK.



Step 7: Generate Toolpaths

CAMWorks calculates toolpaths using the operation parameters and the feature's size and shape.

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

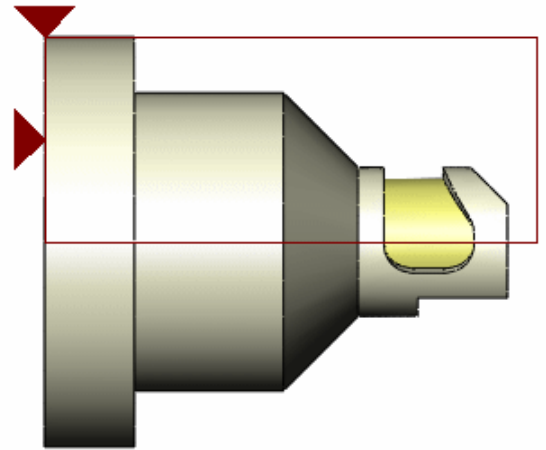
CAMWorks calculates the toolpaths for each operation in the Turn and Mill Part Setups.


CAMWorks provides the ability to simulate the toolpath showing the tool movement and the resulting shape of the part. You can define the chuck for display during simulation, including the size of the chuck base and the jaws.

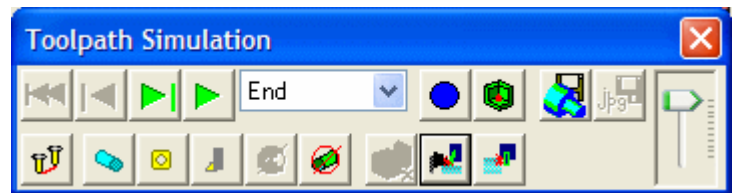
2. To set the proper view, right click Turn Setup1, select Set View, then select Turn ZX.
3. Right click Turn Setup1 in the tree and select Edit Definition on the shortcut menu.




The Operation Setup Parameters dialog box displays.

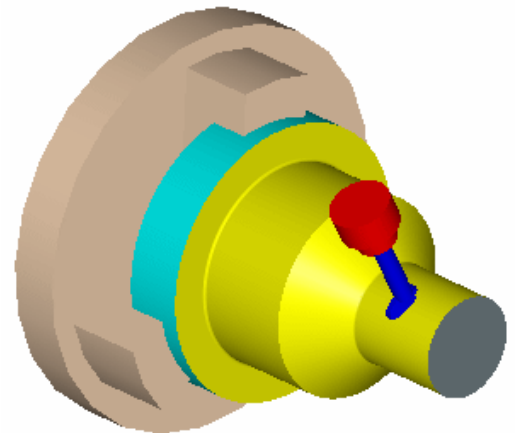
4. Click the Chuck Definition tab
5. Select 4in_1Step_Chuck for the configuration and set the Length and Width to .4in.
6. Click the Chuck Location tab.
7. Pick the two locations for the chuck shown in the figure.
8. Click OK.



9.  Click the Simulate Toolpath button on the CAMWorks toolbar. The Toolpath Simulation toolbar displays.
10. Set the End condition to Next Setup.

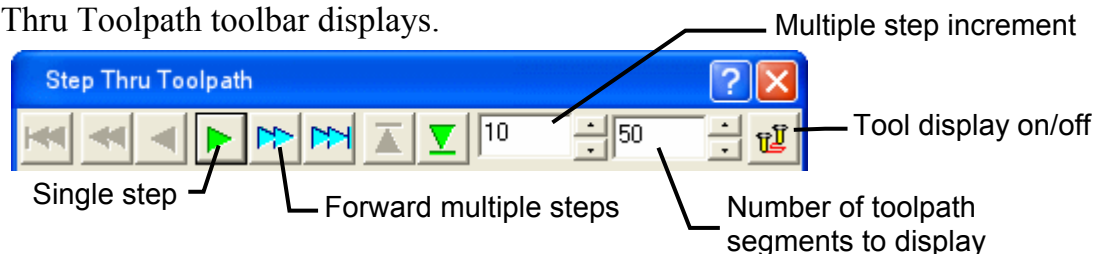


11.  Click the Run button. The simulation runs through Turn Setup1, then pauses.
12.  Click the Run button again.
13. When the simulation pauses, continue clicking the Run button to simulate each Mill Part Setup.
14.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.




CAMWorks also provides the ability to step through the toolpath using the Step Thru Toolpath command.

15. Right click an operation in the tree and select Step Thru Toolpath. The Step Thru Toolpath toolbar displays.



16. Click the Single Step button (single arrow) several times. The active toolpath node is highlighted.

17. Set the Multiple step increment to 5 and click the Forward multiple steps button (double arrow) several times.
18. Click the Goto End button. The toolpath is completed.
19.  Click the X button in the upper right corner to close the dialog box.




Did You Know ...

CAMWorks provides an option to display the XYZ position of the tool in the Message Window during Simulation and Step Thru Toolpath. To activate this option, select the Options command on the CAMWorks menu or the NC Manager shortcut menu. On the General tab, select the Message Window option. On the Simulation tab, select the Cutter Coordinates in Message Window option.

Step 8: Post Process 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.

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

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 MT_1 is not in the File name text box, type **MT_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.





Mill-Turn 2

What You'll Learn

- Inserting Multiple Wrapped Features
- Adjusting operation parameters and machining using C-axis

1. Open the part file **MT_2.SLDPRT** in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\Mill-Turn*).

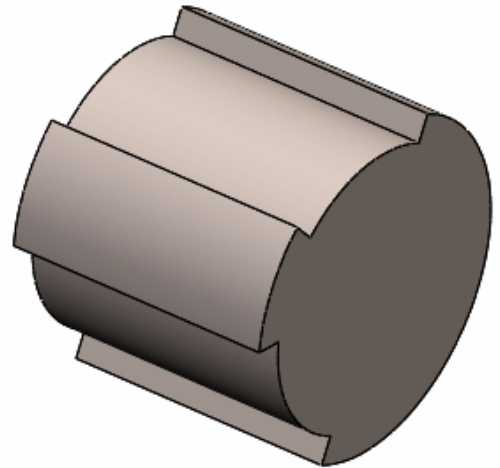
This part can be machined using C-axis.

2.  Click the CAMWorks Feature Tree tab.
3.  Right click Example Mill-in in the Feature tree and select Edit Definition.
4. Highlight Example Mill/Turn-in on the Machine tab and click Select.
5. Click OK to close the dialog box.
6.  Right click Stock Manager in the tree and select Edit Definition.
7. In the Manage Stock dialog box, set the following parameters, then click OK:
 - Z Face = -4in
 - Length = 4.1in
 - Diameter = 5in
8.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click on the NC Manager in the tree and select Extract Machinable Features.

CAMWorks generates a Turn Setup, a Mill Part Setup and machinable features. The items display in the CAMWorks Feature tree.

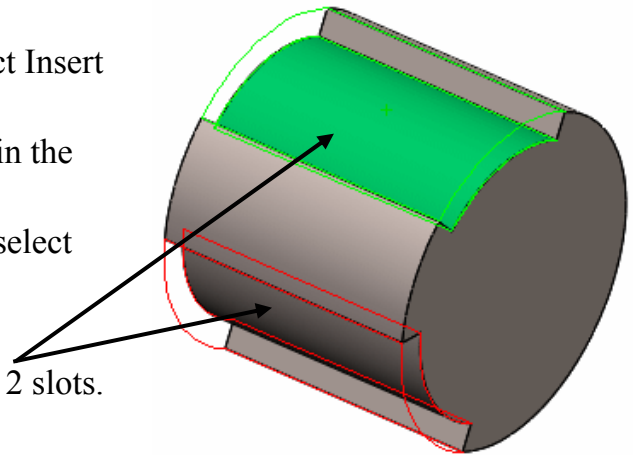
When AFR is run on some parts, CAMWorks may not recognize all the features you want to machine or AFR may recognize a feature that might not be suitable for the intended machining process. When this occurs, you can delete unwanted setups and features and define the setups and features interactively. In this exercise, the slot features are treated as OD features instead of face features, so a different Mill Part Setup and features are needed.

9. Right click Mill Part Setup1 in the tree and select Delete on the shortcut menu.
10. Click Yes to confirm the deletion.
11. Right click Recycle Bin in the tree, select Empty, then click Yes.



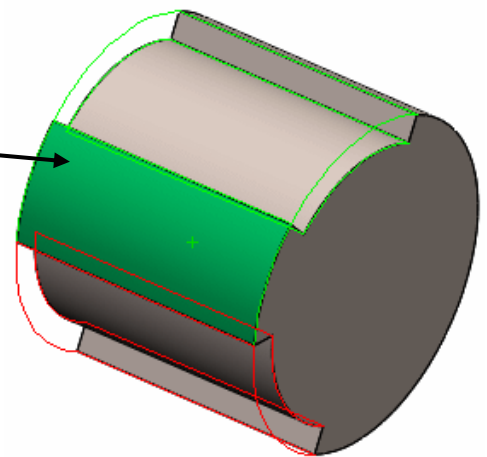
Inserting Multiple Wrapped Features

1. Right click Turn Setup1 in the tree and select Insert Mill Part Setup on the shortcut menu.
2. In the Mill Part Setup dialog box, pick Top in the Reference planes list and click OK.
3. Right click Mill Part Setup2 in the tree and select Insert Wrapped Feature.
4. Set the Feature Type to Wrapped Slot.
5. On the part, pick the cylindrical faces of the 2 slots.



6. Click the End Condition tab.
7. Pick the major OD.
8. Click the Edit Profile tab.

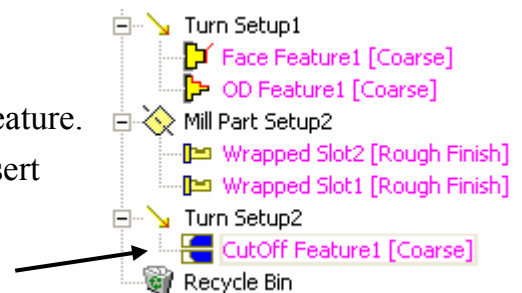
CAMWorks automatically calculates open air edges. By definition, a Slot contains edges that are open so that the resulting toolpath is machined past these open edges. This page allows you to modify these selections, if necessary. The segments are listed for the selected feature and an icon identifies the segment attribute (air or material). The segments are correct for these features and no changes are necessary.




9. Click the Insert button, then click Close. The Wrapped Slot features are listed in the tree.








You want to machine the Cutoff feature after the other features, so you can create a Turn Setup and move the feature.

10. Right click Mill Part Setup2 in the tree and select Insert Turn Setup.
11. Click OK in the Turn Setup dialog box.
12. Highlight Cutoff Feature1 in the tree and drag it to Turn Setup2.



Adjusting Operation Parameters and Machining Using C-Axis

1.  Click the Generate Operation Plan button on the CAMWorks toolbar.
2. Right click Rough Mill1 in the tree and select Edit Definition.





3. On the Roughing tab, select the Automatic angle option.
When you use this option, CAMWorks picks the longest segment and runs the toolpath parallel to that segment.
4. Click OK.
5. Right click Rough Mill2 in the tree and select Edit Definition.
6. On the Roughing tab, select the Automatic angle option, then click OK.
7. Right click Cut Off1 in the tree and select Edit Definition.
8. On the Groove Insert tab, set the Length to 2.95in.
9. On the Holder tab, set Orientation to Down Left and the Protrusion to 2.65 and click OK.
10.  Click the Generate Toolpath button on the CAMWorks toolbar.
11.  Click on the Simulate Toolpath button on the CAMWorks toolbar.
12. Set the following display options:
 -  Stock Shaded Display
 -  Tool Shaded Display
 -  Holder Shaded Display
13.  Target No Display
14.  Click the Run button on the toolbar to start the simulation.

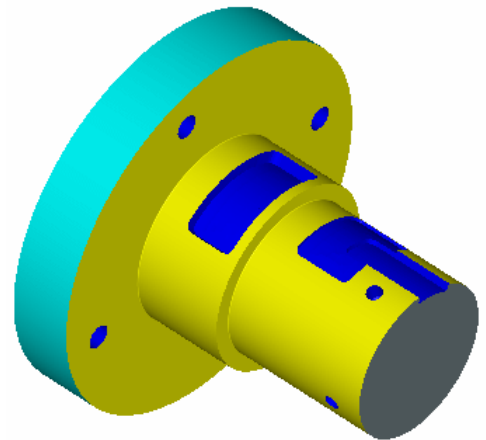
Mill-Turn 3

What You'll Learn

- Creating a Wrapped Feature from a 2D sketch
 - Unwrapping a solid model cylindrical face and creating a Wrapped Feature
 - Machining Using C-axis
1. Open the part file **MT_3.SLDPRT** in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\Mill-Turn*).

This part can be machined using C-axis.



2.  Click the CAMWorks Feature Tree tab.
 3.  Right click Example Mill-in in the Feature tree and select Edit Definition.
 4. Highlight Example Mill/Turn-in on the Machine tab and click Select.
 5. Click OK to close the dialog box.
 6.  Right click Stock Manager in the tree and select Edit Definition.
 7. In the Manage Stock dialog box, change the Length to 10.25in, then click OK.
 8.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click on the NC Manager in the tree and select Extract Machinable Features.
- CAMWorks generates a Turn Setup, Mill Part Setups and machinable features. The items display in the CAMWorks Feature tree.
9. Right click Cutoff Feature1 in Turn Setup1 and select Delete.
 10. Click Yes to confirm the deletion.

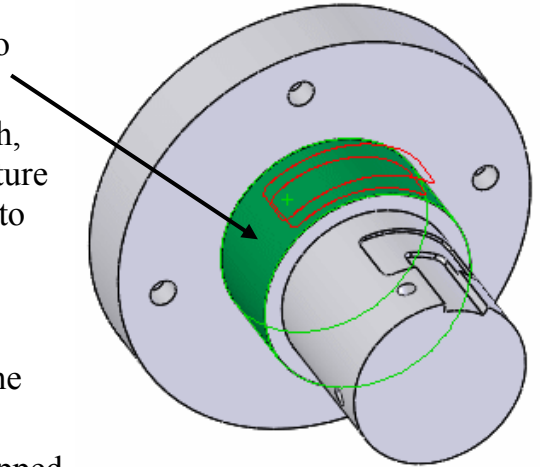


Inserting a Wrapped Feature Using a 2D Sketch

1. Right click Mill Part Setup5 in the tree and select Insert Wrapped Feature on the shortcut menu.
2. In the Insert Wrapped Feature dialog box, leave the Feature Type set to Wrapped Pocket and pick Wrapped 1 in the list.


2D sketches that lie on a sketch plane that is parallel to and passes through the turn axis. These sketches provide an alternative method of defining a wrapped feature. In most cases, wrapped features will be created from edges and surfaces on cylindrical faces. However, if the model does not have cylindrical edges or faces that can be used to create the feature, a 2D sketch can then be used to create a feature.

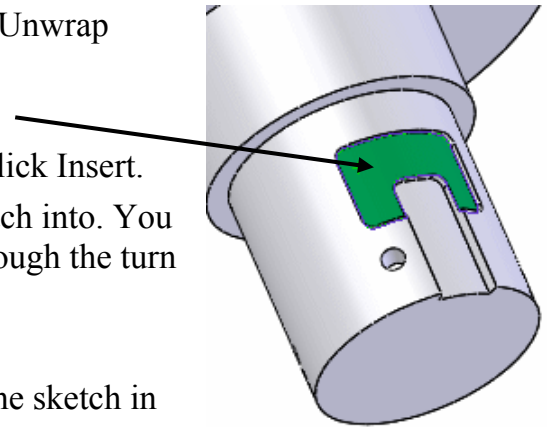
3.  Click in the Wrapping diameter entity box to set the focus, then pick the large OD on the part.
When creating a wrapped feature from a 2D sketch, you must specify a diameter value to wrap the feature about. You can either pick an entity on the model to define the diameter or enter a value.
4. Click the End Condition tab.
5.  Click the Reverse Direction button and set the depth to .2in.
6. Click the Insert button, then click Close. The Wrapped Pocket feature is listed in the tree.





Unwrapping a Solid Model Cylindrical Face and Creating a Wrapped Feature

When the solid model geometry does not provide enough definition for a feature, you can use a CAMWorks function to unwrap a solid model cylindrical face or edge to a 2D sketch. Then, you can modify the sketch geometry and use the sketch to create the Wrapped Feature. In this exercise, to optimize the pocket cutting strategy, it would be beneficial to cut the feature highlighted in the figure as a rectangular wrapped feature.

1. Right click Mill Part Setup5 in the tree and select Unwrap Faces/Edges to Sketch on the shortcut menu.
2. Pick the cylindrical face indicated in the figure.
3. In the Reference plane list, pick Top Plane, then click Insert.
The Reference plane is the plane to unroll the sketch into. You should pick a plane that is parallel to and goes through the turn axis.
4. Click the SolidWorks FeatureManager tab.
5. Click Sketch10 in the tree and zoom out to view the sketch in the graphics area.
6. Right click Sketch10 in the tree and select Edit Sketch on the shortcut menu.
7.  Change the Orientation to Normal To.



8. Zoom up and window pick the sketch as shown in Figure 1.
9. Press the Delete key to delete the entities (Figure 2).
10.  Use the SolidWorks Extend command to connect the open entities (Figure 3).
The sketch is now a fully closed loop that CAMWorks can wrap (Figure 4).

11.  Click Close to exit Sketch mode.

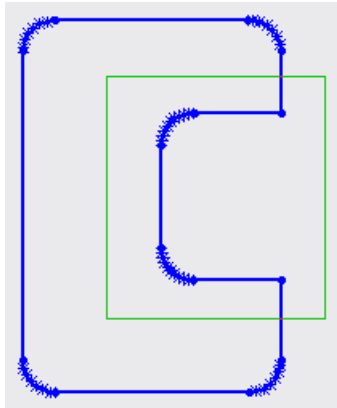


Figure 1

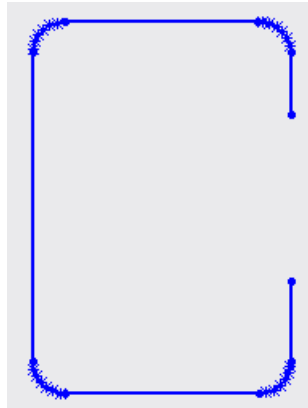


Figure 2

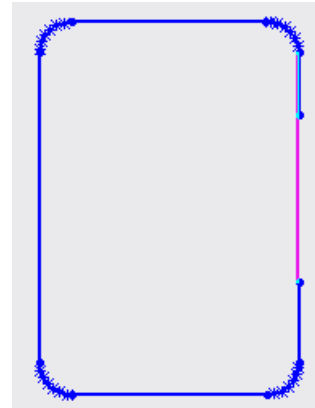


Figure 3

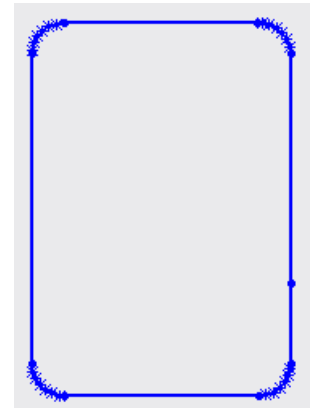


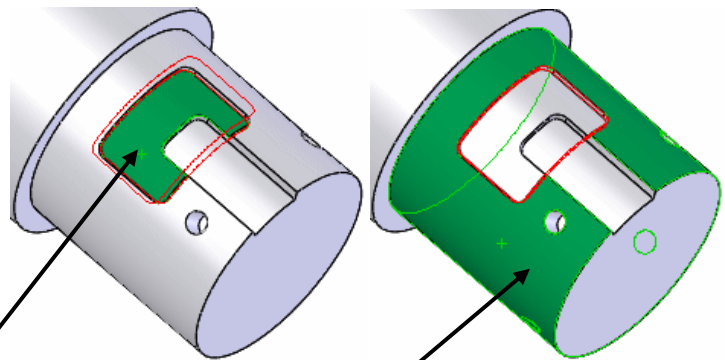


Figure 4

12.  Click the CAMWorks Feature Tree tab.
13. Click the Light button in the CAMWorks Warning dialog box.
The Light Rebuild option is provided for situations where a Full Rebuild is not necessary because you know that the only changes are to sketches or that any model changes do not affect automatically recognized features. The Light Rebuild option performs all actions of the Full Rebuild except running Automatic feature recognition.
14. Right click in the graphics area and click Zoom to Fit to display the part.
15. Right click Wrapped Pocket1 in the tree and select Insert Wrapped Feature.
16. Leave the Feature Type set to Wrapped Pocket and pick Sketch10 in the list.

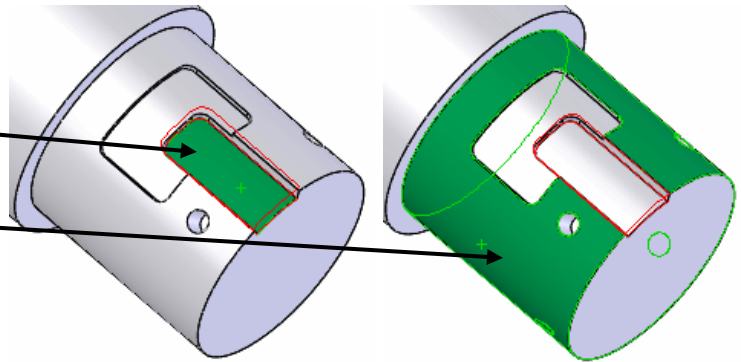
17.  Click in the Wrapping diameter entity box to set the focus.
When creating a wrapped feature from a 2D sketch, you must specify a diameter value to wrap the feature about. You can either pick an entity on the model to define the diameter or enter a value.





18. Rotate the part and pick the bottom face of the pocket.
19. Click the End Condition tab and pick the smaller OD on the part, then click Insert.

You can leave the dialog box open and insert the final feature, which is the wrapped slot.


1. Change the Feature Type to Wrapped Slot.
2. Pick the face of the slot.
3. Click the End Condition tab.
4. Pick the smaller OD on the part.
5. Click Insert, then click Close.
6. In the Feature tree, highlight Wrapped Slot1 and drag it below Wrapped Pocket2 to change the machining order.



Adjusting Operation Parameters and Machining Using C-Axis

1.  Click the Generate Operation Plan button on the CAMWorks toolbar.
2.  Click the Generate Toolpath button on the CAMWorks toolbar.
3. Click RoughMill4 in Mill Part Setup5 and notice the angle of the toolpath. You can edit the operation and have CAMWorks automatically pick the most efficient angle.
4. Right click Rough Mill4 in the tree and select Edit Definition.
5. On the Roughing tab, make sure the Pattern is Zigzag and select the Automatic angle option.

When you use this option, CAMWorks picks the longest segment and runs the toolpath parallel to that segment.

6. Click OK.
7. Right click Rough Mill4 and select Generate Toolpath.
8. Notice the difference in the toolpath.
9.  Click on the Simulate Toolpath button on the CAMWorks toolbar.
10. Set the following display options:



Stock Shaded Display




Tool Shaded Display



Holder Shaded Display

11.  Target No Display

12.  Click the Run button on the toolbar to start the simulation.

Chapter 2 Learning More About Mill-Turn





This chapter provides an opportunity to learn more about CAMWorks Mill-Turn.

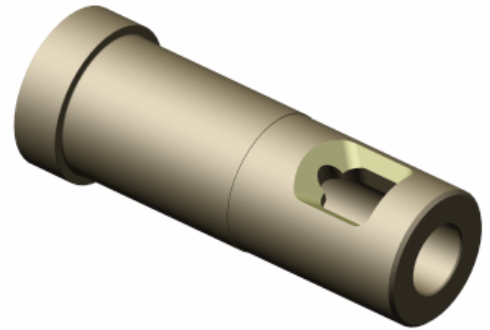
The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

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.

Mill-Turn 4



What You'll Learn

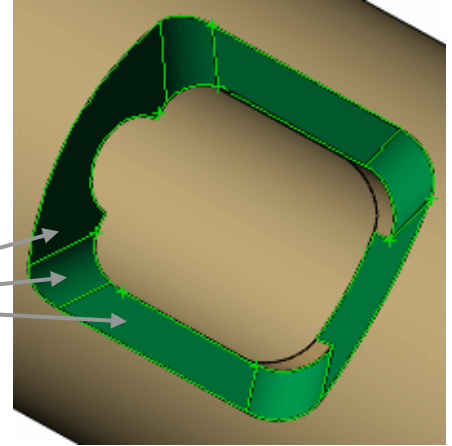
- Interactively inserting 2.5 Axis and Multi Surface features
 - Inserting contain and avoid areas for 3 Axis operations
 - Adjusting operation parameters and machining using Y-axis
1. Open the part file **MT_4.SLDPRT** in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill-Turn*).
This part can be machined using Y-axis.
 2.  Click the CAMWorks Feature Tree tab.
 3.  Right click Example Mill-in in the Feature tree and select Edit Definition.
 4. Highlight Example Mill/Turn-in on the Machine tab and click Select, then click OK.
 5.  Right click Stock Manager in the tree and select Edit Definition.
 6. In the Manage Stock dialog box, change the Length to 16in, then click OK.
 7.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click on the NC Manager in the tree and select Extract Machinable Features.
CAMWorks generates a Turn Setup and machinable features. The items display in the CAMWorks Feature tree.
 8. Right click Cutoff Feature1 in Turn Setup1, select Delete and click Yes to confirm the deletion.



Inserting 2.5 Axis and Multi Surface Machinable Features

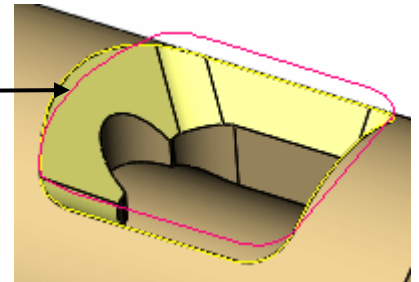
1. Right click Turn Setup1 in the tree and select Insert Mill Part Setup on the shortcut menu.
2. Select the top plane and click the Reverse direction option.
3. Click OK.
4. Right click Mill Part Setup1 in the tree and select Insert 2.5 Axis Feature on the shortcut menu.
5. In the 2.5 Axis Feature Wizard dialog box, set the Feature Type to Pocket.
6. Pick 2x Pocket in the list of sketches and click Next.
7. Leave the End condition type set to Blind.
8. Change the Depth to 2in and click the Reverse direction option.

9. Change the Attribute to Coarse to generate only a Roughing operation, then click Finish.
10. Click Close to exit the Wizard.
11. Right click Irregular Pocket1 in the tree and select Insert Multi Surface Feature.
12.  Make sure the Select Face button is selected.
13. Pick the yellow faces around the pocket.
14. Set the Attribute to Rgh-Pocket, Fin-3D Stepover.
15. Click Insert, then click Close.
16.  Click the Generate Operation Plan button on the CAMWorks toolbar.



Inserting Contain and Avoid Areas for 3 Axis Operations




1. Right click Area Clearance1 in the tree and select Insert Contain Area on the shortcut menu.
2. Pick the top edges of the pocket. A silhouette displays when all the edges are selected.
3. Click Finish.
4. Right click Area Clearance1 in the tree and select Insert Avoid Area on the shortcut menu.
5. Pick 2x Pocket in the list of sketches.



The pocket is being machined by Rough Mill1, the 2 Axis operation that was generated for Irregular Pocket1.




6. Set the Tool Condition to On and click Finish.







You want to contain the Constant Stepover toolpath. Instead of defining a Contain Area for this operation, you can copy the Contain you inserted for the Area Clearance operation.

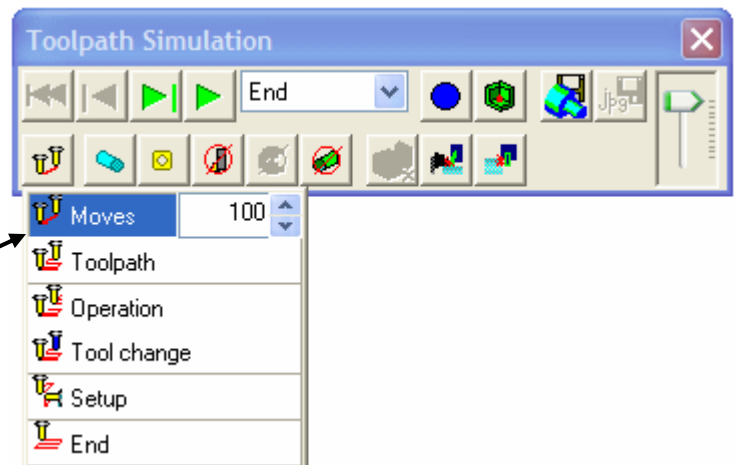
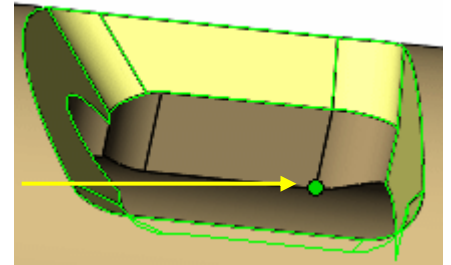
7. Click the  next to Area Clearance1 in the tree to expand the item. The Contain and Avoid areas are listed in the tree.
8. Click Contain Area1 to highlight the item, hold down the Ctrl key and drag the item on top of Constant Stepover1, then release the mouse button.
9.  The cursor changes to an arrow to indicate copy mode.
10. Click the  next to Constant Stepover1. Contain Area2 is listed.

You can copy features, contains and avoids to other operations in the same Mill Part Setup.

Adjusting Operation Parameters and Machining Using Y-Axis

1. Right click Turn Finish1 in the tree and select Edit Definition.
2. On the Finish Turn tab, check the Undercut option and click OK.
This option defines whether the toolpath can create an undercut up to the back angle of the tool. If checked, the toolpath follows the entire toolpath observing the back angle.
3. Right click Area Clearance1 in the tree and select Edit Definition.
4. On the Area Clearance tab in the Limits section, leave the First cut from option set to Top of Stock, change the Offset to -.25in. and click OK.
5. Right click Constant Stepover1 in the tree and select Edit Definition.
6. On the Constant Stepover tab, set the Direction control to Zigzag.
7. In the Limits section, set the Last cut at option to User Defined.
8.  Click the Pick Point button.
9. Pick the vertex shown in the figure.
10. Set the Z Offset to -.1in. Since the bottom edge of the feature is curved, applying the offset in Z will generate the toolpath to cut all the material.
11. Click OK to close the Define Point dialog box, then click OK to close the Operation Parameters dialog box.
12.  Click the Generate Toolpath button on the CAMWorks toolbar.
13.  Click on the Simulate Toolpath button on the CAMWorks toolbar.
14. Set the following display options:

 Stock Shaded Display	 Tool Shaded Display
 Holder No Display	 Target No Display
15.  Click the Run button on the toolbar to start the simulation.
 During the simulation, you can click the Show Animation button to adjust the speed of the simulation. Increasing the number of moves speeds up the simulation.



Mill-Turn 5


What You'll Learn

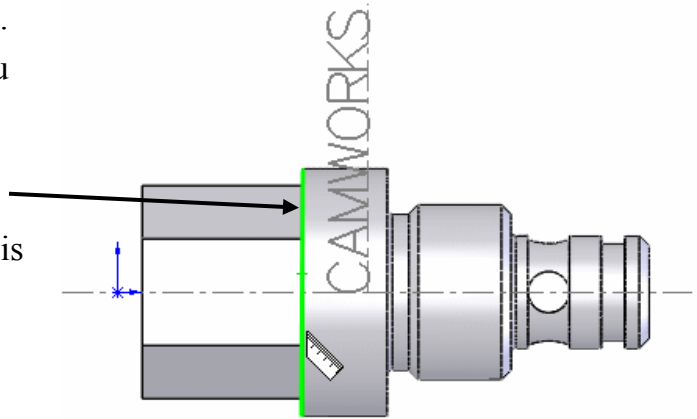
- Adjusting text or entity locations to machine a Wrapped Engrave Feature
- Interactively inserting Wrapped Engrave Features
- Adjusting Starting Angles and C-Axis Rotation Direction
- Creating Planes to control index angles and Setups
- Machining using C-Axis

Adjusting Entity Locations to Machine a Wrapped Engrave Feature

1. Open the part file **MT_5.SLDPRT** in the *\Examples\Mill-Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\Mill-Turn*).

This part can be machined using C-axis.

2.  Click Tools on the SolidWorks menu bar and select Measure.
3. Pick the vertical line that represents the edge of the 1.5in diameter surface.
4. Notice that the Length (circumference) is 4.7124in.
5. Click Close to exit the Measure utility.
6. Right click TEXT-SKETCH in the SolidWorks FeatureManager design tree and select Edit Sketch.



Notice the vertical dimension of 1.75in. If you were to cut this 1.5in diameter surface and lay it out flat, you would be able to machine up to a 4.7124in surface around this part. The letters of the CAMWorks engraving will cover only 1.75in. The letter C of the text is located at a zero start location because the text sketch was started on the Front Plane at the edge of the center of the rotational axis of the part. If you needed to start this at a different location, you would need to move the text up or down the vertical line offset from the rotational axis.


In this exercise, you are going to start this text at 45 degrees instead of zero degrees, so you need to calculate how far to move this text to start machining at 45 degrees.

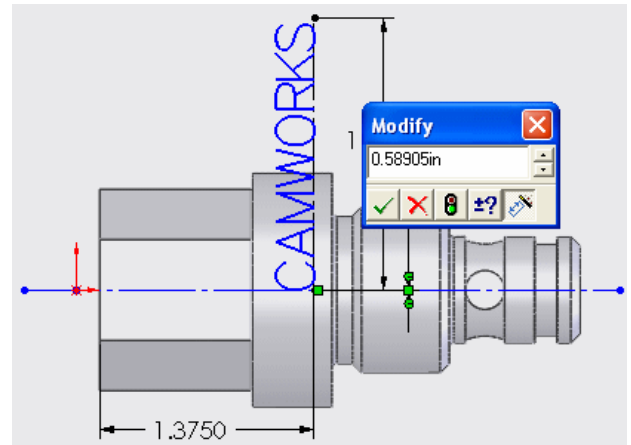
To calculate this:

4.7124 (circumference distance) divided by 360 degrees = .01309 per one degree of rotation





45 degrees x .01309 = .58905 distance

Therefore, you need to move the text a distance of .58905in up or down vertically. This will shift the text and start angle to start machining at 45 degrees off the center of the rotational axis.

7. Select and double-click the .0001 vertical dimension on the part, change it to .58905, then click the green check mark to save the change (Figure 1).
8.  Click Close to exit Edit Sketch mode.



Inserting a Wrapped Engrave Feature to Machine Text

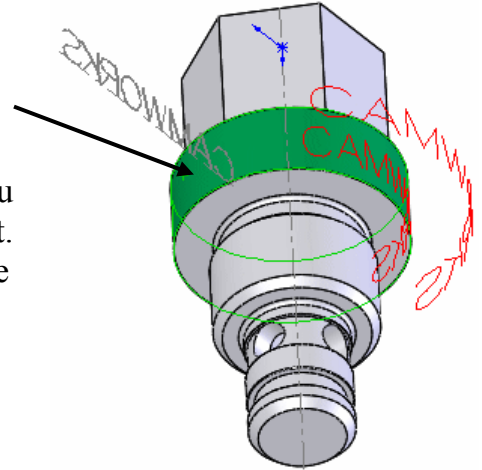
1.  Click the CAMWorks Feature Tree tab.
2.  Right click Example Mill-in in the Feature tree and select Edit Definition.
3. Highlight Example Mill/Turn-in on the Machine tab and click Select, then click OK.
4.  Right click Stock Manager in the tree and select Edit Definition.
5. In the Manage Stock dialog box, change the Length to 3.1in, then click OK.
6.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click on the NC Manager in the tree and select Extract Machinable Features.
CAMWorks generates a Turn Setup and Mill Setups with machinable features. The items display in the CAMWorks Feature tree.
7. Right click Cutoff Feature1 in Turn Setup1, select Delete and click Yes to confirm the deletion.
8. Select the Mill Part Setups, right click, select Delete and click Yes.
You are not machining the side holes in this exercise, so you can delete the setups and features.
9. Right click the Recycle Bin, select Empty and click Yes.
10. Right click Turn Setup1 in the tree and select Insert Mill Part Setup on the shortcut menu.
11. In the Mill Part Setup dialog box, select Front Plane and click the Reverse direction option.
12. Click OK.
13. Right click the Mill Part Setup in the tree and select Insert Wrapped Feature on the shortcut menu.
14. In the Insert Wrapped Feature dialog box, select Wrapped Engrave for the Feature Type..
15. Set the Attribute to Fine. This attribute has been set up in the TechDB to generate one Contour Mill operation.


16. Pick TEXT-SKETCH in the available valid sketches list.

This list shows the 2D sketches that lie on a sketch plane that is parallel to and passes through the turn axis. These sketches provide an alternative method of defining a wrapped feature. In most cases, wrapped features will be created from edges and surfaces on cylindrical faces. However, if the model does not have cylindrical edges or faces that can be used to create the feature, a 2D sketch can be used to create a feature.

17. Rotate the part to see how the feature is trying to wrap around the surface.
18. Click in the Wrapping Diameter entity box and select the 1.5in diameter face.

When creating a wrapped feature from a 2D sketch, you must specify a diameter value to wrap the feature about. You can either pick an entity on the model to define the diameter or enter a value.

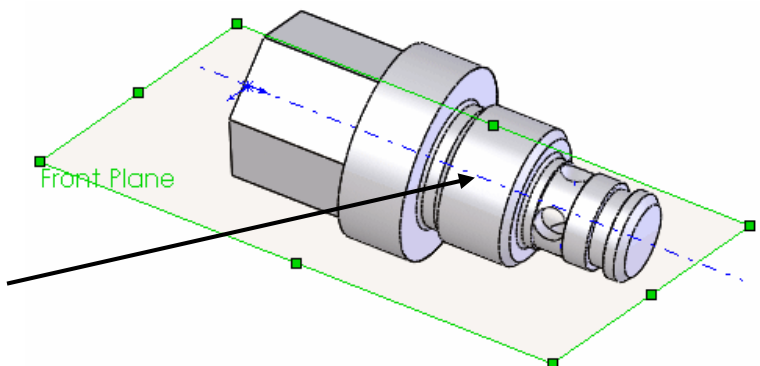






19. Click the End Condition tab.
20. Leave the Type set to Blind.
21. Change the Depth to .005in and click the Reverse Direction button.
22. Click Insert, then click Close.
23. Rotate the part to see the Wrapped Feature on the outside diameter.
24.  Click the SolidWorks FeatureManager design tree tab.
25. Right click TEXT-SKETCH and select Hide. You no longer need to see the SolidWorks sketch.

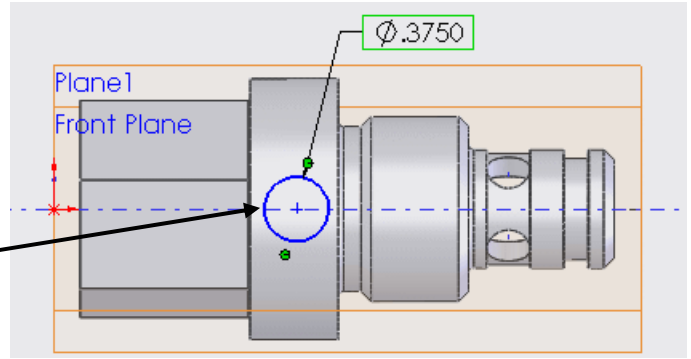
Inserting a Plane to Control the Index or Rotational Angle for Mill Setups

In CAMWorks Mill-Turn, you can use Planes to control indexing angles or to create Mill Setups.




1. Right click on Front Plane in the FeatureManager design tree and select Show.
2. Click View on the SolidWorks menu bar and select Temporary Axis to turn it on.
3. Rotate the part as shown in the figure.
4. Select Front Plane in the tree to highlight the plane in the graphics area.
5. Click Insert on the SolidWorks menu bar and select Reference Geometry, then select Plane.
6. Pick the Temporary Axis running through the center of the part.

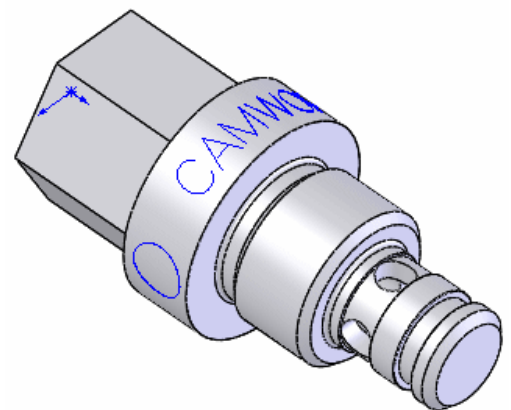
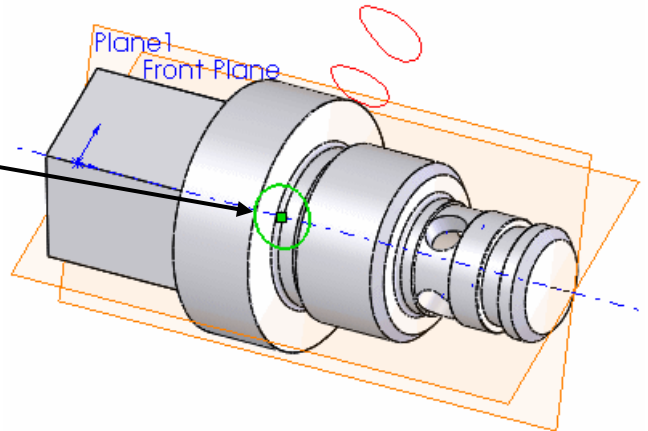


7. Set the Angle to 45 degrees and click OK.
8.  With Plane1 highlighted, change the orientation to Normal to.
9.  Click the Sketch button.
10.  Insert a .375in circle close to the center.
11.  Click Close to exit Edit Sketch mode.











Inserting a 2.5 Axis Wrapped Engrave Feature

1.  Click the CAMWorks Feature Tree tab.
2. Right click the Mill Part Setup and select Insert Mill Part Setup.
3. In the Mill Part Setup dialog box, select Plane1, check the Reverse option and click OK.
4. Right click the new Mill Part Setup in the tree and select Insert Wrapped Feature.
5. In the Insert Wrapped Feature dialog box, set the Feature Type to Wrapped Engrave and set the Attribute to Fine.
6. Pick the .375in circle geometry on the part.
7. Click in the Wrapping diameter entity box and pick the 1.5in diameter face on the part.
8. Click the End Condition tab.
9. Leave the Type set to Blind.
10. Change the Depth to .005in and click the Reverse Direction button.
11. Click Insert, then click Close.
12.  Click the SolidWorks FeatureManager design tree tab.
13. Select Front Plane, Plane 1 and Sketch6, then right click and select Hide. You no longer need to display these items.
14. Click View on the SolidWorks menu bar and select Temporary Axis to turn it off.
15.  Click the CAMWorks Feature Tree tab.
16. Rotate the part and view the features.



Machining Using C-Axis

1.  Click the Generate Operation Plan button on the CAMWorks toolbar.
2.  Click the Generate Toolpath button on the CAMWorks toolbar.
3.  Click on the Simulate Toolpath button on the CAMWorks toolbar.
4. Set the following display options:

 Stock Shaded Display	 Tool Shaded Display
 Holder No Display	 Target No Display
5.  Click the Run button on the toolbar to start the simulation.

Index

2.5 Axis Feature wizard 1-8, 1-10, 2-2

2.5 Axis feature, inserting 1-8, 1-10, 2-2

A

Automatic Feature Recognition 1-7, 1-14, 1-17, 2-2, 2-6

avoid areas, inserting for Adv 3 Axis operations 2-3

C

CAMWorks Message Window 1-7

chuck, defining for simulation 1-11

contain areas, inserting for Adv 3 Axis operations 2-3

E / F / G

Extract Machinable Features command 1-7, 1-14, 1-17, 2-2, 2-6

FANTUTLM 1-5

file

NC program 1-13

setup sheet 1-13

Generate Operation Plan command 1-10, 1-15, 1-20, 2-3, 2-9

Generate Toolpath command 1-11, 1-16, 1-20, 2-4, 2-9

I

Insert 2.5 Axis Feature command 1-8, 1-10, 2-2

Insert Avoid Area 2-3

Insert Contain Area 2-3

Insert Multi Surface Feature command 2-3

Insert Wrapped Feature command 1-9, 1-15, 1-17, 1-19, 2-6, 2-8

Insert Wrapped Feature dialog box 1-10, 1-15, 2-6, 2-8

M

machinable features

creating Corner Slot feature 1-8, 1-10

creating Multi Surface feature 2-3

creating Wrapped Engrave feature 2-6, 2-8

creating Wrapped feature 1-9, 1-15, 1-17, 1-19

defining automatically 1-7, 1-14, 1-17, 2-2, 2-6

defining interactively 1-8, 1-9, 1-10, 1-15, 1-17, 1-19, 2-2, 2-6, 2-8

features recognized automatically 1-7, 1-14, 1-17, 2-2, 2-6

inserting 2.5 Axis feature 1-10, 2-2

Machine 1-4

Machine dialog box

Controller tab 1-5

Machine tab 1-4, 1-14, 1-17, 2-2, 2-6

Posting tab 1-5

Tool Crib tab 1-5

Manage Stock dialog box 1-6, 1-14, 1-17, 2-2, 2-6

Mill Part Setup, inserting plane to control angle 2-7

Multi Surface feature 2-3

N

NC program

file 1-13

generating 1-13

steps to generate 1-2

O

Operation Setup Parameters dialog box

Chuck Definition tab 1-12

Chuck Location tab 1-11

Operation tree 1-11

operations

generating 1-10, 1-15, 1-20, 2-3, 2-9

generating toolpaths 1-11, 1-16, 1-20, 2-4, 2-9

inserting avoid areas 2-3

inserting contain areas 2-3

post processing 1-13

simulating toolpaths 1-12, 1-16, 1-20, 2-4, 2-9

Options dialog box, General tab 1-13

P

parts used in exercises

mt_1.sldprt 1-3

mt_2.sldprt 1-14

mt_3.sldprt 1-17

mt_4.sldprt 2-2

mt_5.sldprt 2-5

Post Output File dialog box 1-13

Post Process command 1-13

post processor, parameters 1-5

S

setup sheet 1-6

Simulate Toolpath command 1-12, 1-16, 1-20, 2-4, 2-9

Simulate Toolpath toolbar 1-12

Step Thru Toolpath command 1-12

Index

steps to generate NC program in Part mode 1-2

Stock 1-6

T

text, using Wrapped Engrave Feature to engrave 2-5

tool, selecting tool crib 1-5

toolpaths

- generating 1-11, 1-16, 1-20, 2-4, 2-9

- inserting avoid areas 2-3

- inserting contain areas 2-3

- post processing 1-13

- simulating 1-12, 1-16, 1-20, 2-4, 2-9

- stepping through 1-12

U

Unwrap Faces/Edges to Sketch command 1-18

unwrapping solid model cylindrical face 1-18

W

Wrapped Engrave feature 2-6, 2-8

Wrapped feature 1-9, 1-15

- using 2D sketch 1-17, 1-19

- Wrapping diameter 1-18, 1-19