# CAMWorks 2006
## POWERED BY SolidWorks

# Mill & Turn
# Tutorial

SolidWorks
Certified Gold Product

TEKSOFT™
CAD/CAM SYSTEMS
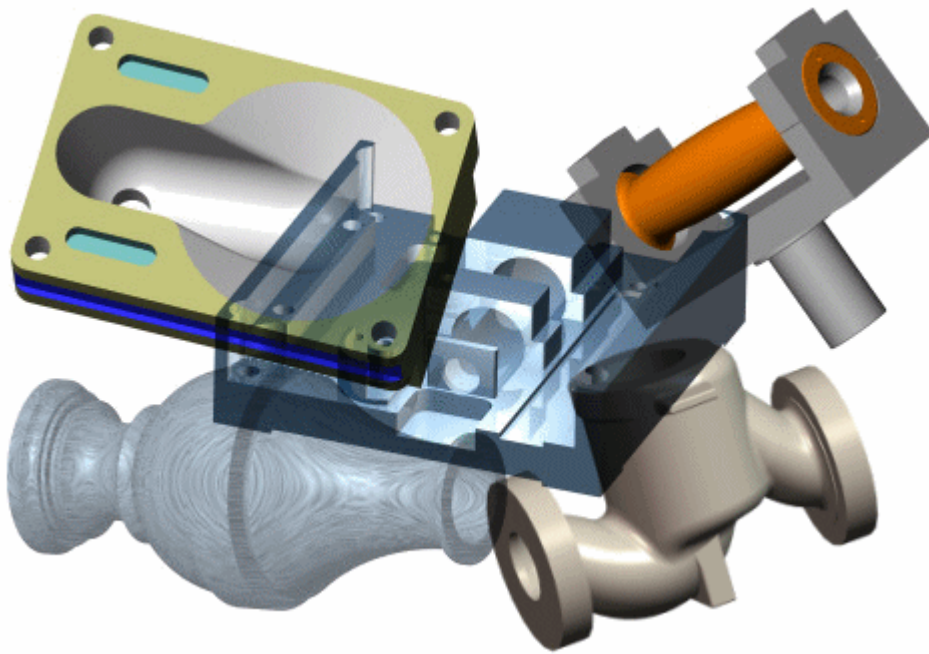
## *License Agreement*

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.

# Table of Contents

Table of Contents

**INDEX**

Table of Contents

# Chapter 1  Learning 2 Axis Mill Basics

This chapter provides an opportunity to learn CAMWorks 2 Axis Mill 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.

**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.

# 2 Axis Mill 1

## What You'll Learn

### *Steps to Generate Mill Toolpaths and NC Code in Part Mode*

The following steps are used to generate 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. Define machinable features.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Simulate material removal.
9. Post process the toolpaths.

The next series of exercises show you how to generate finish toolpaths on a SolidWorks part model. In order to give you a general understanding of how to use CAMWorks, you work with a part that was previously modeled in SolidWorks. When you define the operations and toolpaths, you will follow steps that are not explained in depth. This is done to show you the basics of generating toolpaths from start to finish without getting into the details at this time.

Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically.

## 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 **MILL2AX_1.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

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

The tabs at the bottom or top 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
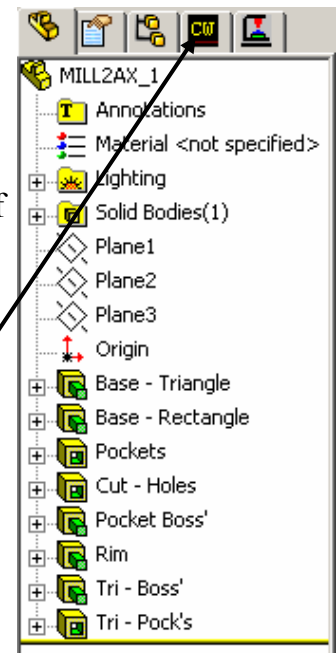
## Step 2: Change to CAMWorks Feature Tree

1. Click the CAMWorks Feature Tree tab.

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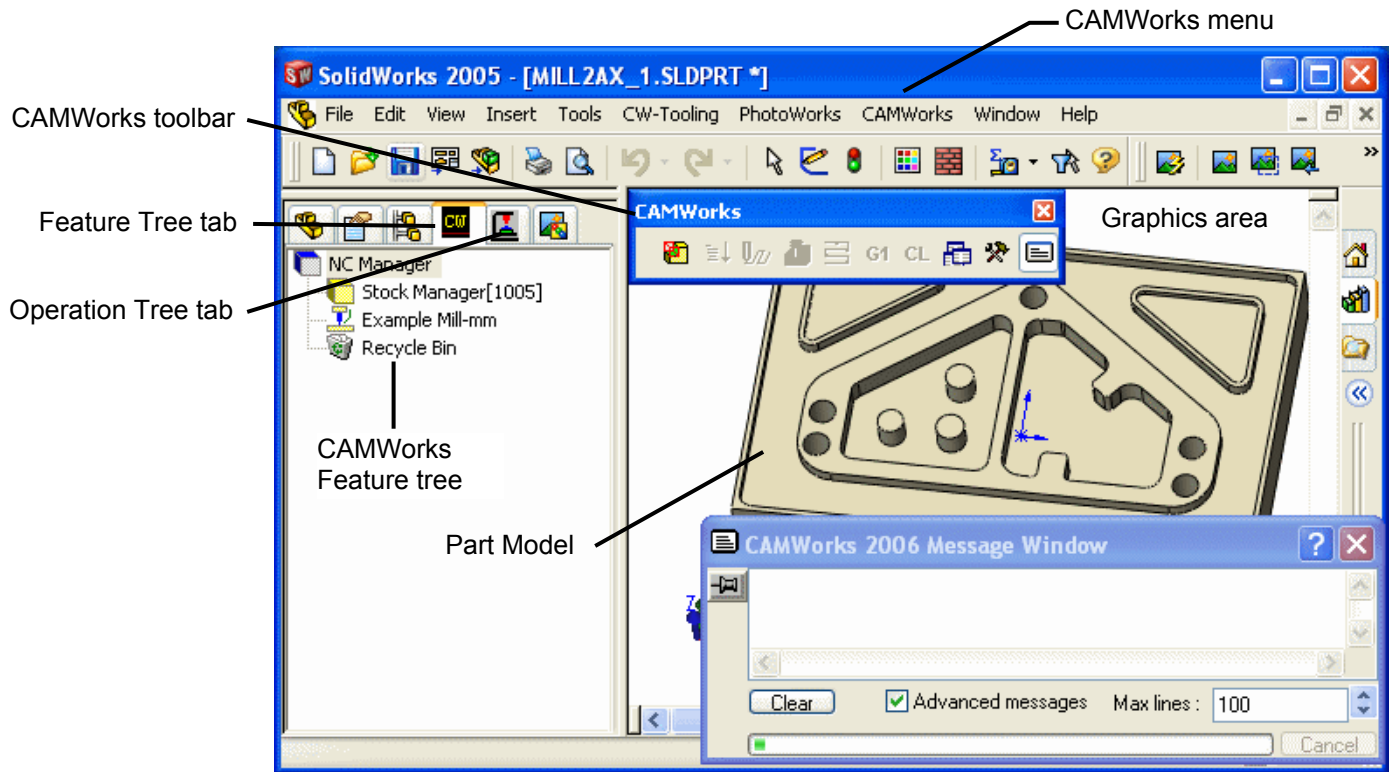
## CAMWorks Machining Trees

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

- **Stock Manager**

  The stock is the material from which the part will be machined. You can define the stock as a rectangular shape or as an extruded sketch. You can also specify the type of material.

- **Machine**

  The Machine item defines the machine tool that the part will be machined on. The machine definition includes the type of machine (e.g., mill, turn), tool definitions and the machine controller (post processor). The machines are set up in the Technology Database.

- **Recycle Bin**

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

## CAMWorks Menu

1. Click CAMWorks on the SolidWorks menu bar. The CAMWorks menu lists the CAMWorks commands. The commands are explained in the CAMWorks online Help.

2. Right click on the NC Manager in the tree. This is a shortcut menu. The right-click shortcut menus provide access to a variety of commands.

## CAMWorks Toolbar

The CAMWorks toolbar provides access to the main CAMWorks commands found on the CAMWorks menu. Clicking a toolbar button is the same as selecting a command from the NC Manager level, regardless of the active item in the tree.

1. Locate the CAMWorks toolbar and click the Options button.

2. In the Options dialog box, click each tab to view the options and settings that you can change in CAMWorks.

3. Click the Help button at the bottom of the dialog box. Each tab is explained in the online Help.

4. Click the Close button in the upper right corner of the Help window to close the window.

5. Click Cancel to close the Options dialog box.
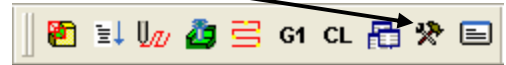
## Step 3: Define the Machine

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

- Machine type – mill, turn, mill-turn or Wire EDM:  The machine type defines the machinable feature set that can be recognized automatically and defined interactively. The icons that display in the tree identify the current machine:

  Mill Machine    Turn Machine    Mill-Turn Machine    Wire EDM

  An alternative machine can be selected at any time to output different G-code programs for alternative machine tools. If the machine type changes, then all features and operations will be deleted.

- Tool crib:  A subset of tools from the tool library that are commonly loaded into or used with the current machine.

- Controller: Also called a post processor. This post processor identifies the format of the NC G-code output.

Define the machine:

1. Right click Example Mill-mm in the CAMWorks Feature tree.

   The shortcut menu displays. Shortcut menus display commands that are appropriate for the highlighted item.

2. Select Edit Definition on the shortcut menu.
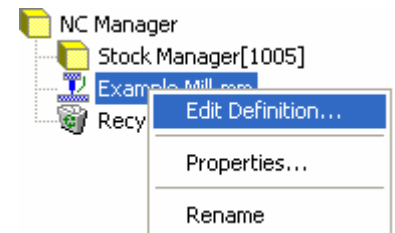
The Machine dialog box displays the Machine tab. The default selected machine is specified in the Technology Database. Example Mill–mm is the default machine used for the metric parts in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

Machine tools are set up in the Technology Database. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility. For more information, see Chapter 8.

3. In the Available machines list, highlight Example Mill–mm and click the Select button.
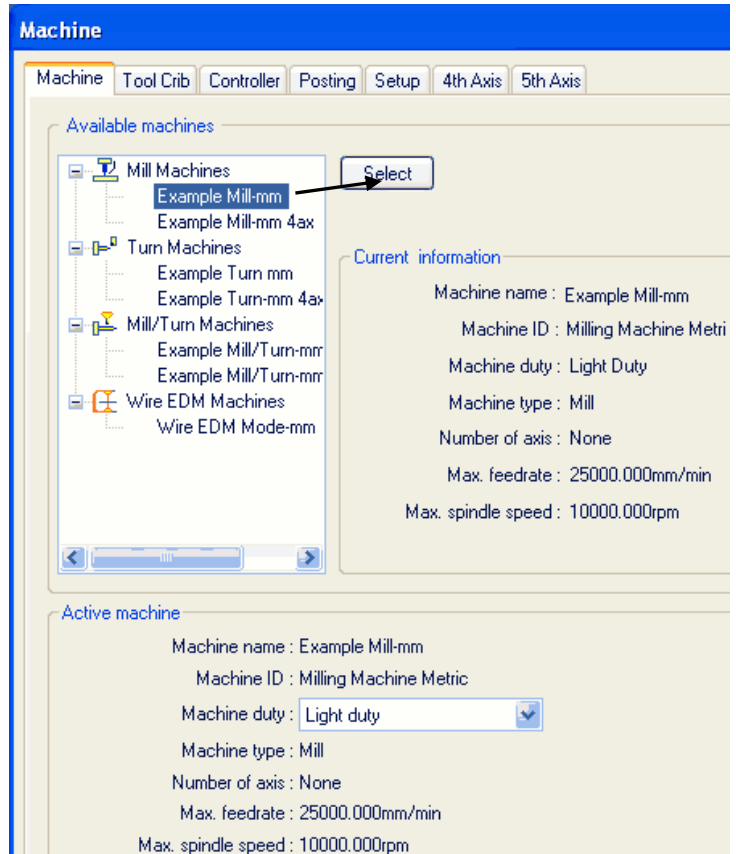
4. Click the Tool crib tab and make sure Tool Crib 1 (metric) is selected.

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 (metric) is a default tool crib that has been set up for the sample Mill machine. When you define your machine tools in the Technology Database, you can set up your own tool cribs.
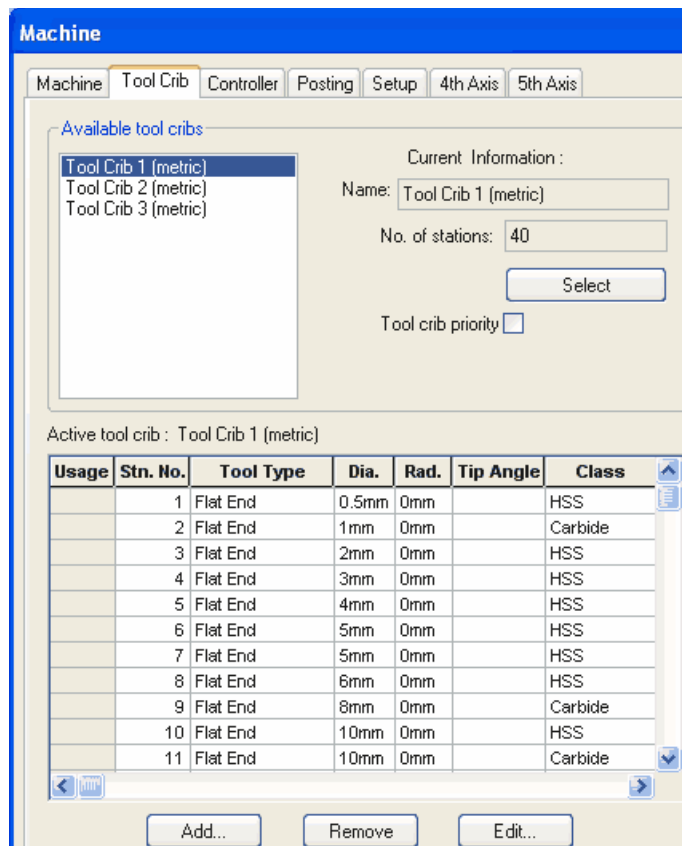
5. Click the Controller tab.

   This tab allows you to select the post processor. The list that displays depends on the post processors that are installed on your system. 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).

   If FANTUTM (the tutorial post processor) is not highlighted, highlight it in the list and click the Select button. FANTUTM 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.

   The Current information area displays information about the FANTUTM controller.

   A short description displays in the window below the Current information. This window contains information only if an optional file has been created for the post processor.

6. Click the More button.

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

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

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.
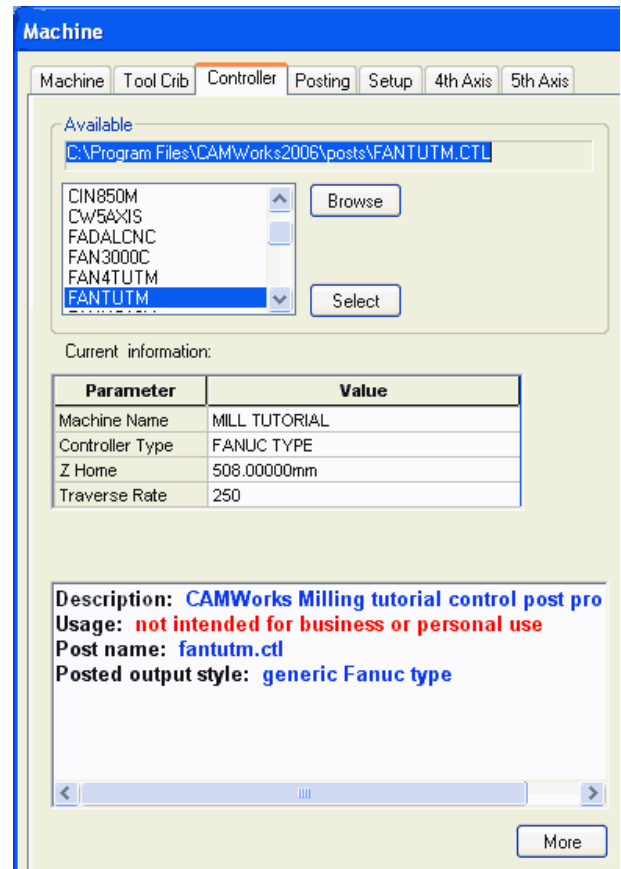
8. Type **1001** for the Program Number and press the down arrow on the keyboard.

9. Type **Stainless** for the Material and press the down arrow.

10. Type **40mm** for the Part Thickness and click OK.

**Machine**

| Parameter | Value |
|---|---|
| Program number | 1001 |
| Material | Stainless |
| Part Thickness | 40.00000mm |
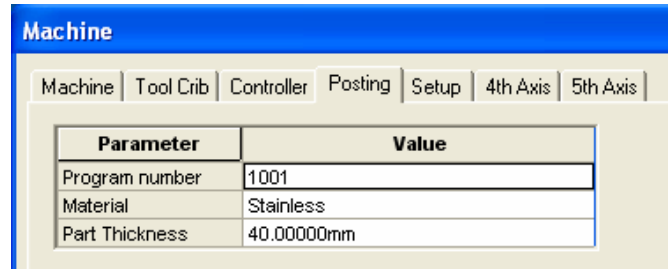
## Step 4: Define the Stock

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

In this exercise, you define the stock as a box offset from the part.

1. Right click Stock Manager in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
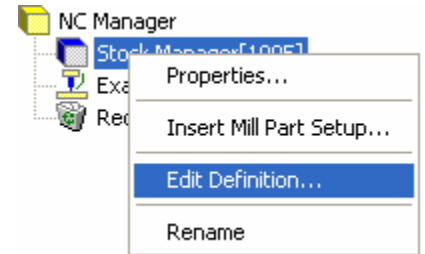
   The Manage Stock dialog box displays.

2. For the Bounding box offset, type **1** for X+ and click the X+ button (Uniform X).

3. Repeat step 2 for Y+ and Z+

4. Click the Material down arrow and select 304L.

5. Click Yes to continue.

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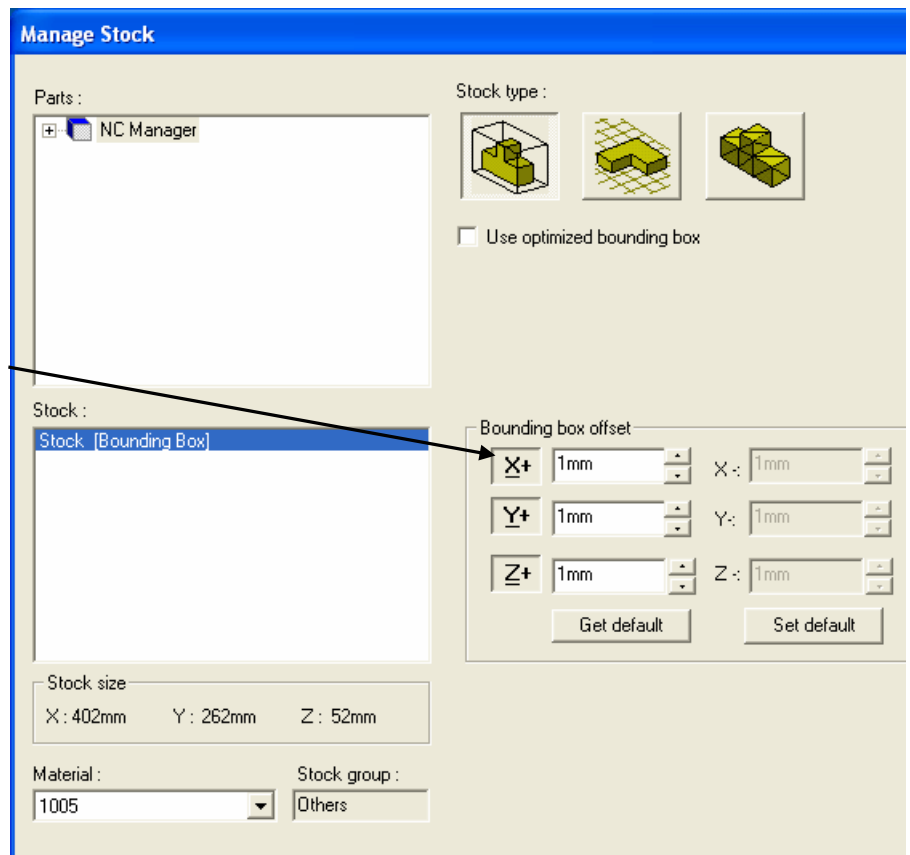
**Manage Stock**

Parts: NC Manager

Stock: Stock [Bounding Box]

Stock type:

Use optimized bounding box

Bounding box offset

| X+ | 1mm | X-: | 1mm |
| Y+ | 1mm | Y-: | 1mm |
| Z+ | 1mm | Z-: | 1mm |

Get default    Set default

Stock size
X: 402mm    Y: 262mm    Z: 52mm

Material: 1005

Stock group: Others

## 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 machinable features such as pockets, holes, slots and bosses. Depending on the complexity of the part, AFR can save considerable time in defining two-dimensional prismatic features.

- Interactively created features

  If AFR does not recognize a feature you want to machine, you need to define the feature using the Insert 2.5 Axis Feature command. If you have 3 Axis Milling, multi-surface features can be defined using the Insert Multi Surface Feature command.

The idea of AFR is to analyze the part for features that can be machined. This process is much the same as what you would do if you were to pick up a part that you had to machine. You would look it over, take measurements, and begin deciding how to define areas or features to machine and what machining processes you would need.

CAMWorks is not machining the SolidWorks features directly. It creates a separate list of Machinable Features instead. This is because a single SolidWorks feature may have several areas that need to be machined in different ways with different tools.

For example, SolidWorks would see the part on the right as having an extruded cut for the whole pocket and an extruded boss for the ribs. That works well for modeling parts in SolidWorks, but not for machining purposes. There are actually 5 pockets of 3 different types to machine here.

Look at the CAMWorks Feature tree. There are 3 Machinable Features in the list: one for the large pocket on top, and 2 for the two different types of pockets around the ribs. This gives you the most flexibility for machining.
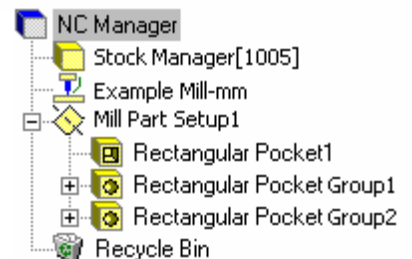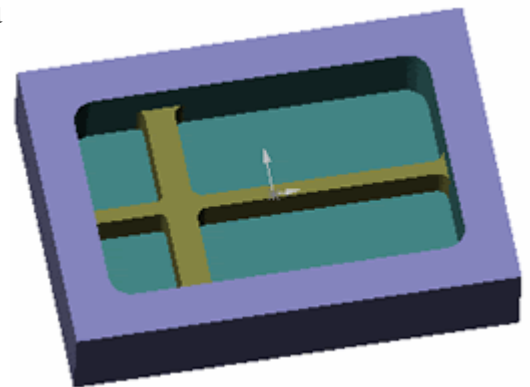
### Using Automatic Feature Recognition (AFR)

Define machinable features automatically:

1. Select one of the following methods to extract features:

   Right click NC Manager and select Extract Machinable Features on the shortcut menu.

   or

   Click the Extract Machinable Features button on the CAMWorks toolbar.
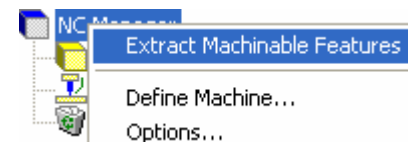
The CAMWorks Message Window displays. This window displays automatically to report the progress and status of the current process. Generating Mill Part Setups is the last item.
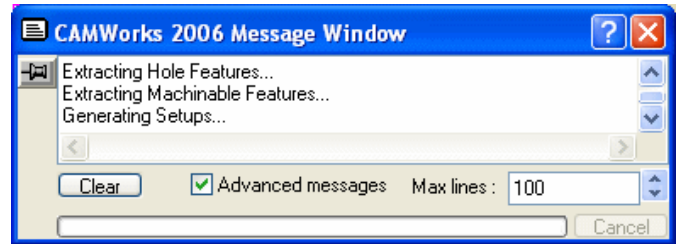
You can control whether this window displays temporarily or permanently by selecting the Options command on the CAMWorks menu and checking the Message Window option on the General tab in the Options dialog box.

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

The Mill Part Setup is the 2 axis plane that the tool movement will be based on. It has an origin location, and X, Y, and Z direction vectors. The Mill Part Setup is created automatically; however, you can move the origin, and change the direction and angles of the X and Y axes.

A Mill Part Setup is created for each different tool orientation. There is only one Mill Part Setup for this part because all features can be machined using a single tool orientation. For each Mill Part Setup, the machinable features are listed in the order in which they were recognized.

The Feature tree allows you to:
– Copy, rename, suppress, delete and combine machinable features
– Change machinable feature parameters
– Change the order in which the features are machined
– Insert 2.5 Axis and Multi Surface features
– Search for a feature based on item name
– Hide or show feature display in graphics area
– Generate an Operation Plan and find the first operation for a feature

**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.

Most feature parameters are fixed; however, some parameters can be changed using the Parameters command on the feature shortcut menu.

2. Right click Hole Group1 in the tree and select Parameters on the shortcut menu.

The Hole Parameters dialog box displays the number of components and the hole parameters. Since there is no physical information about the type of hole, CAMWorks allows you to define an attribute for the hole (Drill, Bore, Ream, Thread or a user-defined attribute).

3. Click the down arrow next to Drill to see the choices.

4. Click Thread in the list.

   The figure changes to reflect your choice and the Thread parameters are enabled.

5. Click the down arrow next to Thread and select Drill again, then click OK to close the dialog box.

6. Click the + next to Hole Group1 in the CAMWorks Feature tree.

   The tree expands to display each individual hole.

7. Click the - next to Hole Group1.

Save the part with the machinable feature data:

8.  Click the Options button the CAMWorks toolbar.

   Note that you can also click CAMWorks on the menu bar or right click NC Manager in the tree and select the Options command.

9. On the General tab, make sure the Save/Restore part option is checked, then click OK.

   If this option is checked, when you save and close a part document, the machining data is saved. When the part document is opened again, the machining data is restored along with the part design information.

   If this option is not checked, when you save and close a part document that contains at least one CAMWorks Setup, a message indicates that Save/Restore is disabled. If you click Yes, CAMWorks saves all CAMWorks data before closing the file. If you click No, CAMWorks closes the part and discards any new CAMWorks data since the last save.

10. Click Save As on the File menu.

11. In the Save As dialog box, type **cwex1** and click the Save button.

    **SAVE FREQUENTLY!**
    − When you open a file, you are actually working on a copy of the file. The original is still stored on disk. Periodically saving your file ensures that your latest work is retained.
    − CAMWorks provides an Auto save option on the General tab in the Options dialog box for automatically saving your CAMWorks data.
    − Frequent saves prevent having to redo a time-consuming model or CAM operation. If a power failure occurs, you will lose whatever you have been working on.

## 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. To machine these areas, you need to define machinable features interactively using the Insert 2.5 Axis Feature command.

In this exercise, you insert a Face Feature so that you can face the top of the part. In order to define a Face Feature, you select a face on the SolidWorks part that is at the depth you want to face the part to. In this example, the entire top of the part is the same height, so you can select any of the topmost faces.

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 Face Feature.

3. Click the main face.

   The outline of the face is highlighted on the part and Face <1> displays in the Entities selected list.
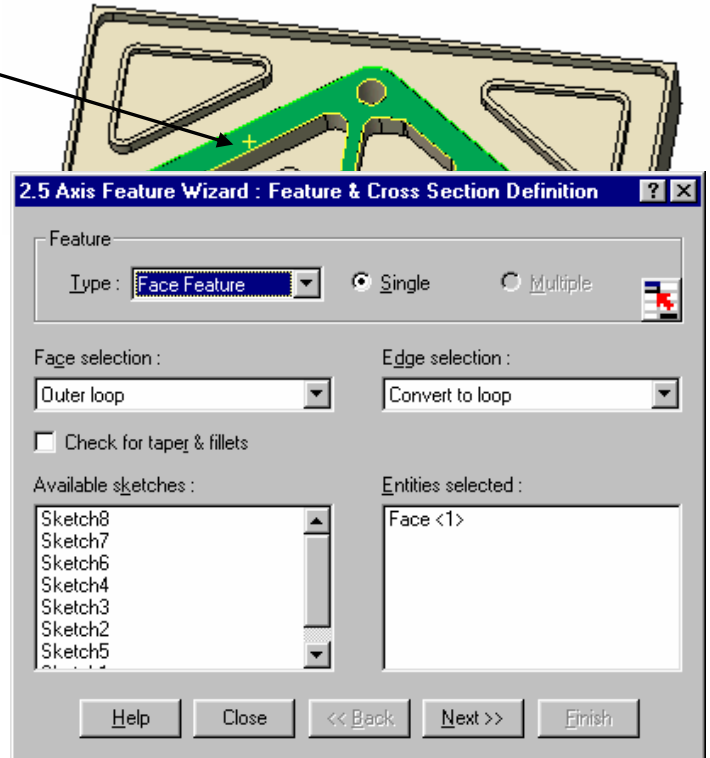
4. If the Check for taper & fillets option is checked, remove the checkmark.

   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.

5. Click the Next button.

   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.

6. Leave the End condition Type set to Upto Stock.

7. Leave the Attribute set to Coarse.

8. Click Finish.

9. Click Close to close the 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box. Face Feature1 displays in the CAMWorks Feature tree. You have now defined all the machinable features in this part and you are ready to generate the Operation Plan.
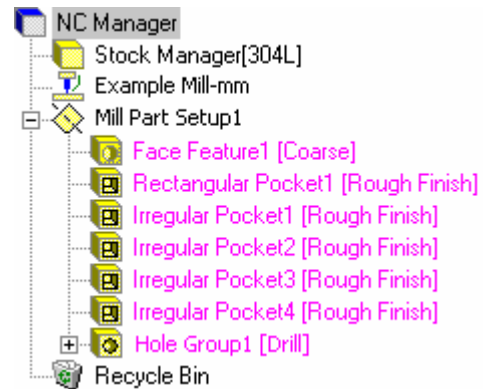
## Step 6:  Generate Operation Plan and Adjust Operation Parameters

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 using the Insert 2 Axis Operation, Insert Hole Operation and Insert 3 Axis Operation commands. These commands are explained in the CAMWorks online Help.

1. Select one of the following methods to generate an operation plan:

   Right click Mill Part Setup1 in the CAMWorks Feature tree and click Generate Operation Plan on the shortcut menu.

   or

   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 in Mill Part Setup1. The operations are listed in the CAMWorks Operation tree, which displays automatically. The Operation tree can also be accessed by clicking the CAMWorks Operation Tree tab.

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

   The CAMWorks Operation tree allows you to:
   – Insert, rename, suppress, and delete operations
   – Change operation parameters
   – Combine operations
   – Sort operations
   – Change the machining order
   – Generate toolpaths
   – Simulate toolpaths
   – Post process the toolpaths
   – Hide or show toolpath display
   – Search based on item name
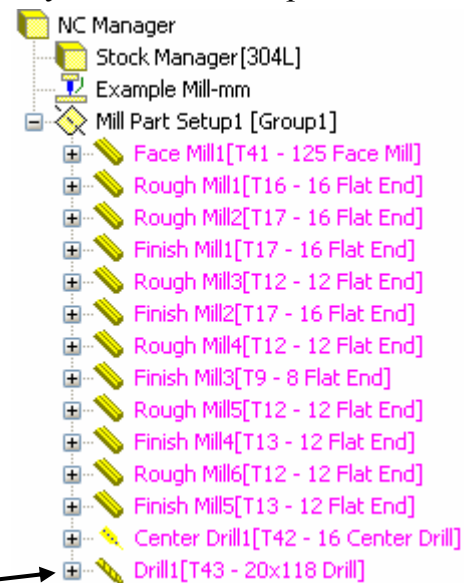
   To the left of each toolpath operation is a plus sign (+). Clicking a plus sign displays the name of the Machinable Feature that this operation is going to machine. These Machinable Feature items can be used to view geometric information and to modify the machining depth of the feature.

   **Did You Know ...**

   If an operation displays in a color instead of black, then toolpaths have not been generated. This occurs when you insert a new operation interactively, you insert a new feature interactively and generate operations for the new feature, or CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or a parameter is not correct. You can set the color on the Display tab in the Options dialog box.

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 operation 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.

Edit operation parameters using the Edit Definition command on the shortcut menu:

1. Right click on Rough Mill4 (the operation generated for Irregular Pocket2) in the CAMWorks Operation tree.

2. Select the Edit Definition command on the shortcut menu.

   The Operation Parameters dialog box displays. This dialog box gives you access to all the parameters used to define the toolpath.

   General parameters for the type of toolpath include the method of machining, depth of cut, step over, stock allowance, retract height, speeds and feeds. This dialog box also gives you access to all the parameters for the tool you are using and allows you to select a different tool.

3. Click the Roughing tab and change the Pocketing Pattern to Zigzag.

4. Click the Feature Options tab and change the Entry type to Entry Drill.

5. Click OK to close the dialog box.

## Step 7: Generate Toolpaths

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

1. Select one of the following options to generate toolpaths:

   Right click Mill Part Setup1 in the CAMWorks Operation tree and click Generate Toolpath on the shortcut menu.

   or

   Click the Generate Toolpath 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 calculates the toolpaths for each operation in the Mill Part Setup.

   You can also generate toolpaths for each operation individually by right clicking on an operation and clicking Generate Toolpath on the shortcut menu.

2. Highlight the first operation in the tree, hold down the Shift key, then highlight the last operation.

   The toolpaths for all the operations display on the part showing the centerline of the toolpath.

3. Click an operation in the CAMWorks Operation tree.

   − The toolpath for that operation displays.

   − As you click each operation, the toolpaths for that operation are displayed.

4. Click the Entry Drill operation.

– An Entry Drill operation was created automatically for Rough Mill4 because you set the Entry Method to Entry Drill.

– The Entry Drill displays as a black circle on the toolpath. The circle represents the diameter of the drill. By default, the diameter of the drill used in the Entry Drill is the same as the diameter of the mill used in the roughing cycle. If a tool of the same size is not available, a tool diameter of the next smallest size is used.

– The depth of the drill cycle for blind features is .02in or .5mm off the bottom of the feature. For through features, the depth of the drill is the same as the feature.

– Entry Drill operation parameters can be edited and the operation can be renamed, moved, suppressed, deleted, etc.
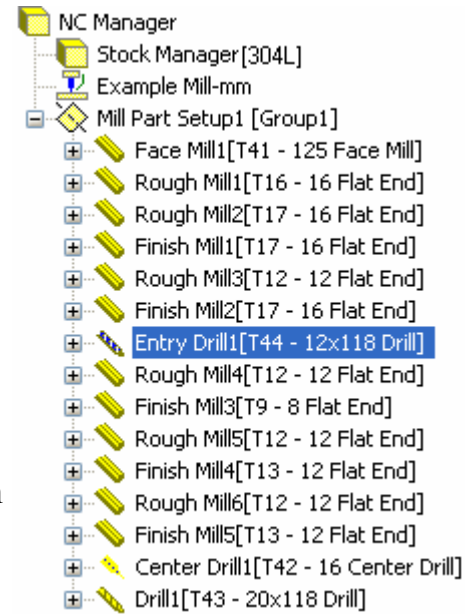
CAMWorks provides the ability to simulate the toolpath showing the tool movement and the resulting shape of the part.

5. Right click on Mill Part Setup1 and select Simulate Toolpath.

The Simulate Toolpath toolbar displays.
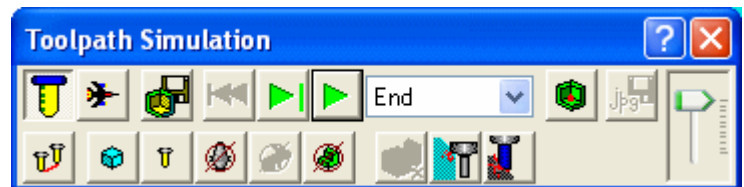
The toolbar controls allow you to:

– Run the simulation in Tool or Turbo mode.

– Display the simulated part, the design part and a comparison of the two.

– In Tool mode, customize the display of the stock, tool and tool holder (wireframe, translucent, shaded, or no display).

– Run the simulation for all or selected operations.

– When simulating an operation, the simulation can be for the current operation or for all previous operations up to the selected operation.

– Pause the simulation in either Tool or Turbo mode and dynamically change the orientation of the part using zoom, pan, rotate, etc.

– Control the simulation speed by dragging the Simulation Speed Control slider up or down.

If you want to simulate only the toolpath for a given operation, you can right click that operation, then select Simulate Toolpath.

6. ▶ Click the Run button.

The simulation runs in Tool mode with the tool displayed during simulation.

7. ✖ Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.
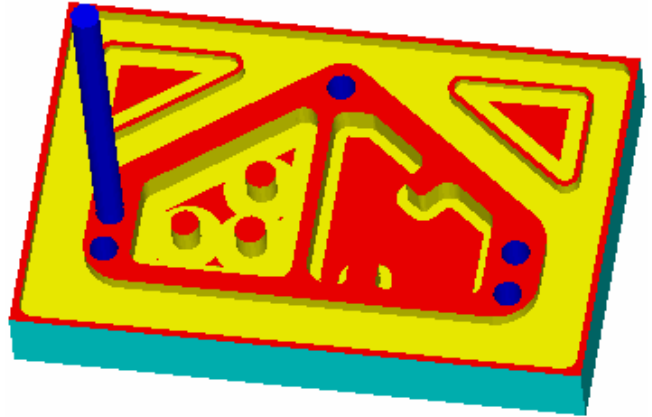
CAMWorks also provides the ability to step through the toolpath using the Step Thru Toolpath command.

8. Right click an operation in the tree and select Step Thru Toolpath.

The Step Thru Toolpath toolbar displays.

Multiple step increment

**Step Thru Toolpath**

Single step — Forward multiple steps — Number of toolpath segments to display
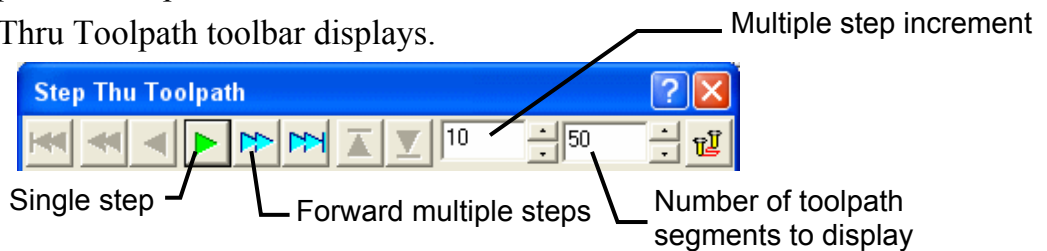
9. Click the Single Step button (single arrow) several times.

The active toolpath node is highlighted.

10. Set the Multiple step increment to 5 and click the Forward multiple steps button (double arrow) several times.

11. Click the Goto End button. The toolpath is completed.

12. ✖ 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.

   or

   Right click NC Manager in the Operation tree and click Post Process on the shortcut menu.

   The Post Output File dialog box displays so you can name the NC program file.

   Typically, the NC program and Setup Sheet files are stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder.

   If you are running CAMWorks in Demo mode, the Post Output File dialog box does not display because you cannot save NC code in Demo mode.

   If the Post Process command is grayed out on the CAMWorks menu or on a shortcut menu, make sure that you have selected a post processor and generated the toolpaths.

2. Click the down arrow to the right of the Save as type box.

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

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

3. If cwex1 is not in the File name text box, type **cwex1**, 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.
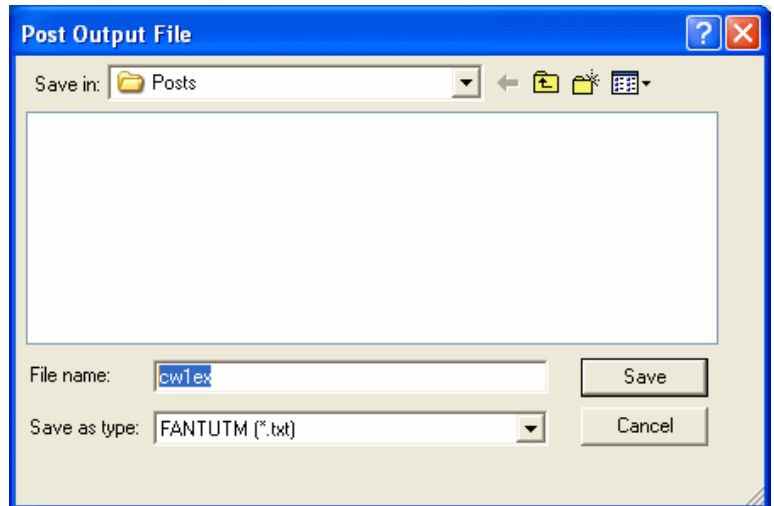
The Post Process Output dialog box displays.

4. Click the Step button on the control bar at the top.

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

5. Click the Step button. The next line of NC code displays.

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

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

8. Click OK to close the Post Process Output dialog box.

**More About Setup Sheets**

The Setup Sheet is a printable file that contains information the machine tool operator can use to set up the part and the tools required to produce a part. The information includes the machine, the controller, estimated machine time, the part material, and the tooling used to machine the part.

CAMWorks provides two methods for creating Setup Sheets:

− During post processing, CAMWorks automatically creates a simple text file with a .set extension.

− The Setup Sheet command on the NC Manager shortcut menu allows you to generate a Setup Sheet that is based on an Access database report template and store the information in the Report Database to view at any time. CAMWorks supplies several report templates that can be used as is. You can also open the Report Database in Access and create customized reports based on these templates or design your own original reports.

# 2 Axis Mill 2

## What You'll Learn

### *Extracting and Working With Machinable Features*

CAMWorks is a feature-based machining system. Feature-based machining provides numerous benefits because the definition of the feature enables a higher level of automation when creating machining operations and associated toolpaths. Adding to this benefit, CAMWorks provides both an automatic method of extracting features called Automatic Feature Recognition (AFR), and an interactive method using the Insert 2.5 Axis Feature function. A CAMWorks file can contain both types of features.

This exercise demonstrates some of the features that are identified automatically by the Extract Machinable Features command.

1. Open the part file **MILL2AX_2.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Examine the part.

   You can recognize various sizes and shapes of holes, bosses, pockets and slots.

   It would appear that CAMWorks extracts features based on SolidWorks features used to create the part. As you will see, this is not necessarily so. Notice that the design tree contains no design features. This part was imported into SolidWorks. In this exercise, you will see how CAMWorks determines machinable features.
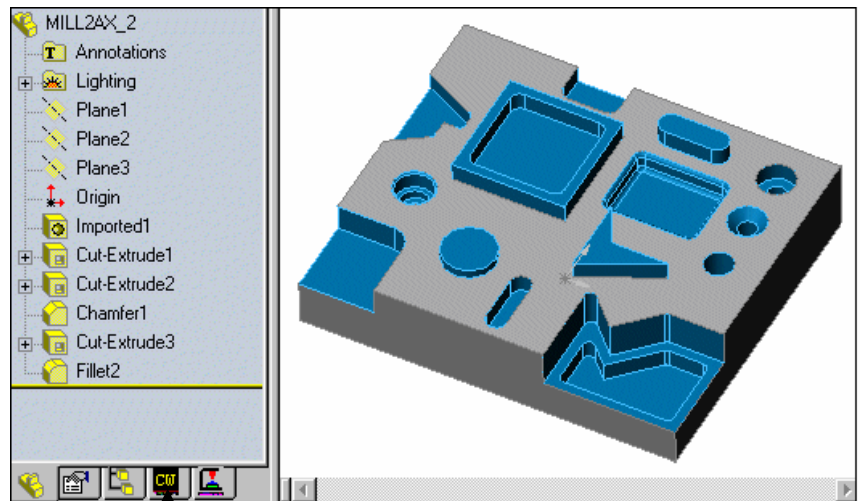
3. Click the CAMWorks Feature Tree tab.

4. Right click Example Mill–in in the CAMWorks Feature tree and select the Edit Definition command on the shortcut menu.

   The Machine dialog box displays the Machine tab. This dialog box allows you to select a machine definition, tool crib and controller.

   On the Machine tab, Example Mill–in is the Active machine. This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

5. Click the Tool Crib tab.

The Tool Crib page allows you to choose a tool crib or set of tools 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.

If you select the Tool crib priority option, tool selection rules are modified to place a higher priority on selecting tools from the tool crib. This results in fewer tools being added to the tool list from the tool library. The online help explains the rules that are observed when this option is used.

6.  Click the Select button to make sure Crib 1 is the Active tool set.

    From this page, you can add, remove and edit tools in the Tool Crib.
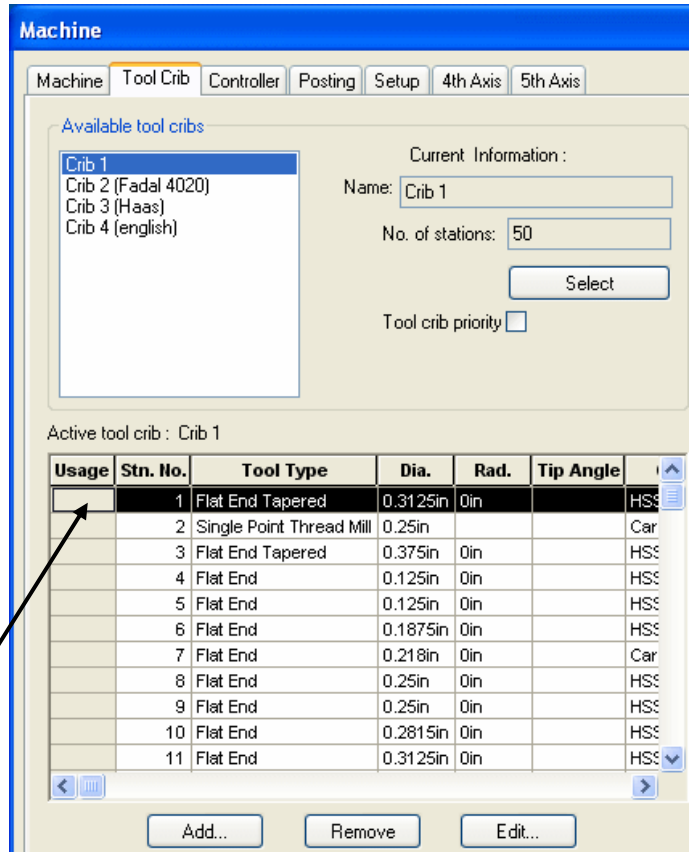
7.  Select a tool from the Active tool set list and click Edit.

    The Edit Tool Parameters dialog box displays.

    This dialog box allows you to change the parameters for the selected tool. The changes you make in this dialog box affect only the tool crib for the current part. To change the tool definition for all future jobs, edit the tool crib definition in the Technology Database.

8.  Click OK or Cancel to close the Edit Tool Parameters dialog box.
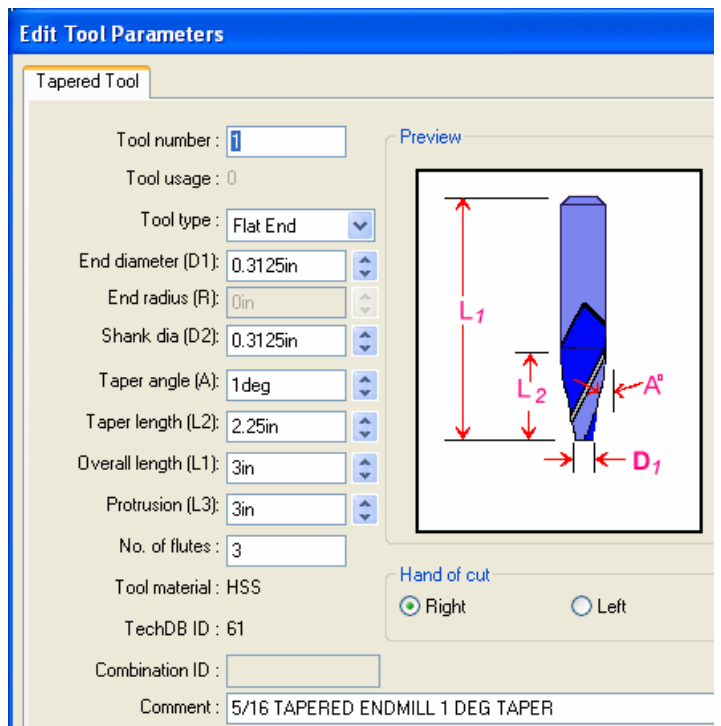
9. Click the Add button at the bottom of the Tool crib tab in the Machine dialog box.

The Tool Select Filter dialog box displays. This dialog box allows you to set filters for displaying and selecting tools.

**Tool Select Filter**

Tool type: Bore

Filter by
- ☐ Diameter   0in  .  9in
- ☐ Tool material   Carbide
- ☐ Holder designation   40 BT
- ☐ Protrusion length   0in  .  9in

OK    Cancel    Help

10. Click OK.

The Tools Database form displays. This form allows you to add an existing tool in the Technology Database to the Tool Crib for the current part. The list contains all the tools that have been entered into the Technology Database. Note that you cannot use the form to add new tools to the Technology Database.

**Tools Database - Bore (inch)**

| ID | Tool ID | Min Bore Dia | Max Radius Cut Depth | Radius Adjust Range | Effec Cut L |
|----|---------|--------------|----------------------|---------------------|-------------|
| 1  | BRS-001 | 0.593 | 0.04  | 0.125 | 0.52 |
| 2  | BRS-002 | 0.625 | 0.04  | 0.125 | 0.52 |
| 3  | BRS-003 | 0.656 | 0.07  | 0.125 | 0.59 |
| 4  | BRS-004 | 0.5   | 0.04  | 0.046 | 0.43 |
| 5  | BRS-005 | 0.531 | 0.04  | 0.046 | 0.54 |
| 6  | BRS-006 | 0.562 | 0.07  | 0.046 | 0.48 |
| 7  | BRS-007 | 0.625 | 0.06  | 0.156 | 0.56 |
| 8  | BRS-008 | 0.5   | 0.06  | 0.062 | 0.45 |
| 9  | BRS-009 | 0.718 | 0.06  | 0.156 | 0.56 |
| 10 | BRS-010 | 0.593 | 0.06  | 0.062 | 0.45 |
| 11 | BRS-011 | 0.718 | 0.05  | 0.156 | 0.56 |
| 12 | BRS-012 | 0.593 | 0.05  | 0.156 | 0.45 |
| 13 | BRS-013 | 0.953 | 0.065 | 0.218 | 0.71 |
| 14 | BRS-014 | 0.828 | 0.065 | 0.093 | 0.65 |
| 15 | BRS-015 | 0.953 | 0.08  | 0.218 | 0.78 |
| 16 | BRS-016 | 0.828 | 0.08  | 0.093 | 0.65 |

Record 1

OK    Cancel    Help

11. Click OK to close the Tools Database without adding a tool to the active tool set.

12. Click the Controller tab in the Machine dialog box and make sure FANTUTM (the tutorial post processor) is selected.

FANTUTM 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.

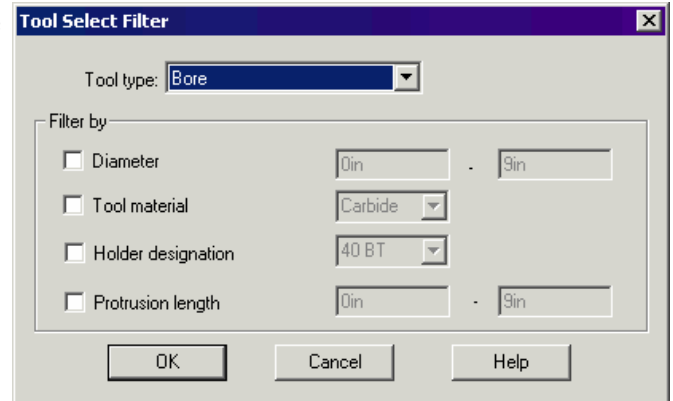13. Click the Parameters tab.

These parameters provide information required to generate the NC program. When you use CAMWorks to machine your own parts, change these parameters as required.

14. Click OK.

15. Right click the Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.

16. Change the Bounding box offsets to a uniform **.1in** for X, Y, and Z, then click OK.

Extract machinable features:

1.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.

   The CAMWorks Message Window displays. This window reports the process status and shows the progress of the process.

   When AFR is finished, the CAMWorks Feature tree displays the machinable features.

   The CAMWorks Feature tree is designed to work similar to the SolidWorks FeatureManager design tree. From the tree, you can reorder features using drag and drop, change feature names and suppress features.

2. Click each machinable feature in the CAMWorks Feature tree.

   As you pick each machinable feature, the geometry for that feature highlights on the part.
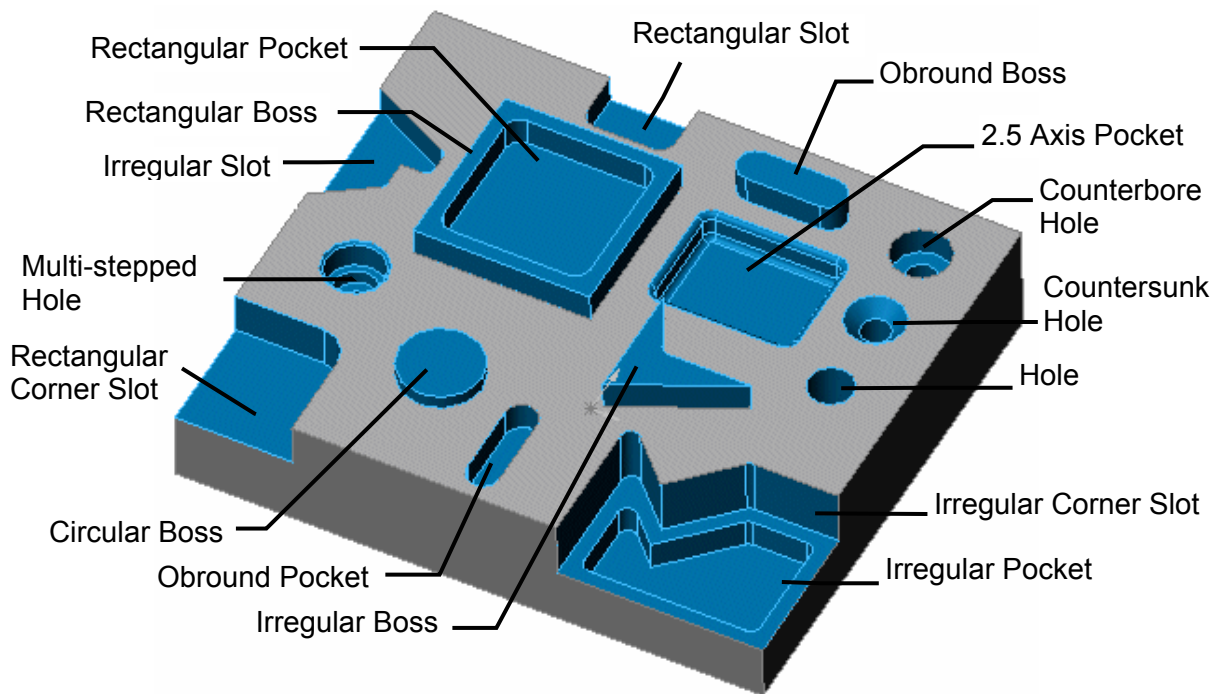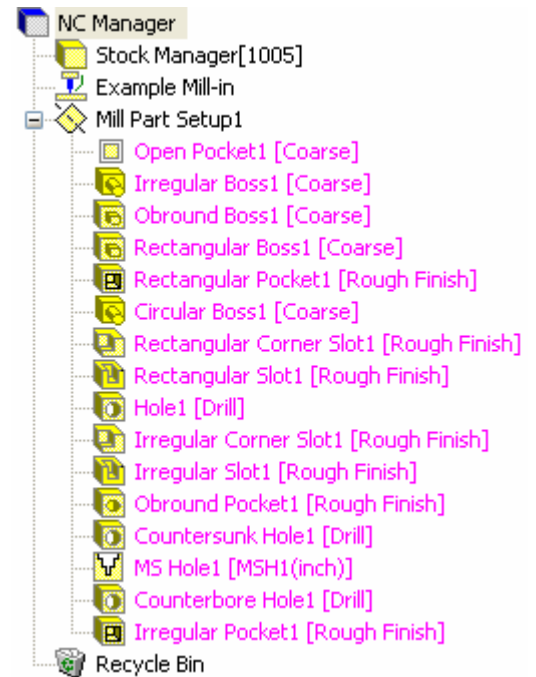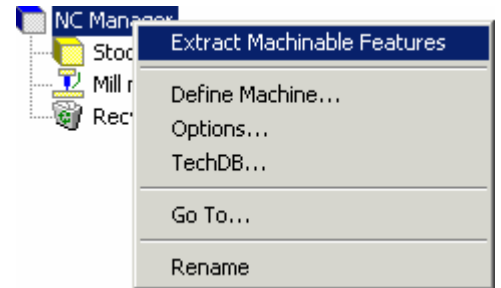
3. Notice that an Open Pocket feature was created.

When CAMWorks creates a Mill Part Setup, an Open Pocket feature is created automatically if AFR finds a boss feature and the Setup is parallel to one of the sides of the stock. The bottom of the Open Pocket is the bottom of the Boss feature and the boss becomes an island in the Open Pocket.

CAMWorks allows you to control the types of 2.5 axis and multi surface features that are created by AFR.

1. [icon] Click the Options button on the CAMWorks toolbar or select Options on the CAMWorks menu.

2. Click the Mill Features tab in the Options dialog box.

3. Notice that the defaults are set for CAMWorks to create hole, non-hole and boss features.

4. In the Feature types section, check the Face option.

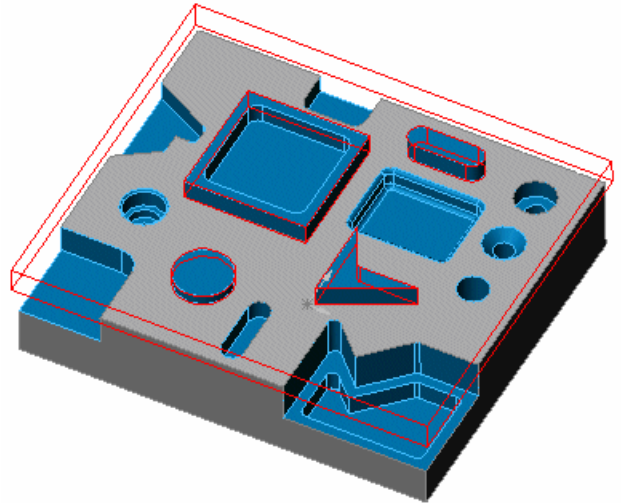If the Face option is checked, when CAMWorks creates a Mill Part Setup, a Face Feature is created automatically if the top most face is parallel to the Setup and the Setup is parallel to one of the sides of the stock.

5. Check the Tapered & filleted option.

When CAMWorks extracted features, the 2.5 Axis pocket was not found. If you check this option, CAMWorks can automatically create features that meet the definition of filleted and tapered features as created interactively using the Insert 2.5 Axis Feature command:

– The top and bottom of the feature are flat and parallel to the Mill Part Setup direction.
– The side wall can be vertical or a constant taper with a top or bottom fillet and/or a top of bottom chamfer.
– Corner fillets must be conical as defined by the taper angle of the feature.

6. Remove the check mark from the Boss option and check the Remove on Rebuild option, then click OK.

If you were machining this part, you would not want to generate operations for the Boss features that were found by AFR. You can delete the features or you can Rebuild the features without the bosses as shown in this exercise.

Rebuild the features:

7. [icon] Click the Extract Machinable Features button on the CAMWorks toolbar.

CAMWorks rebuilds the feature list and removes the boss features. Notice that AFR created a Face feature and a Rectangular Pocket feature for the tapered pocket.
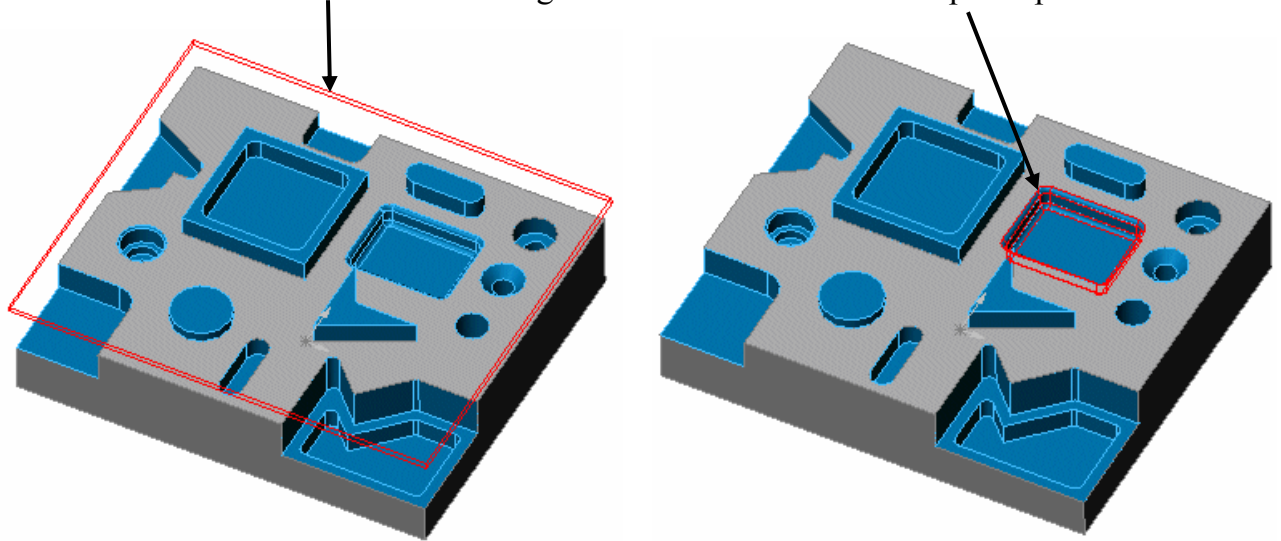
Machinable features are recognized based on the part's geometry (the shape of a feature) and topology (how the features are related to one another). However, not all attributes of a feature can be determined from the geometry and topology.

For example, AFR cannot determine that a hole has been threaded, reamed or bored. CAMWorks allows you to specify additional attributes to a feature so that more appropriate operations can be can be created.

1. Right click Hole1 in the tree and select Parameters.

The Parameters dialog box provides a geometric and dimensional report on the feature and allows you to assign additional information to the feature. In the case of holes, you can identify that the hole is to be drilled, bored, reamed or threaded, etc.
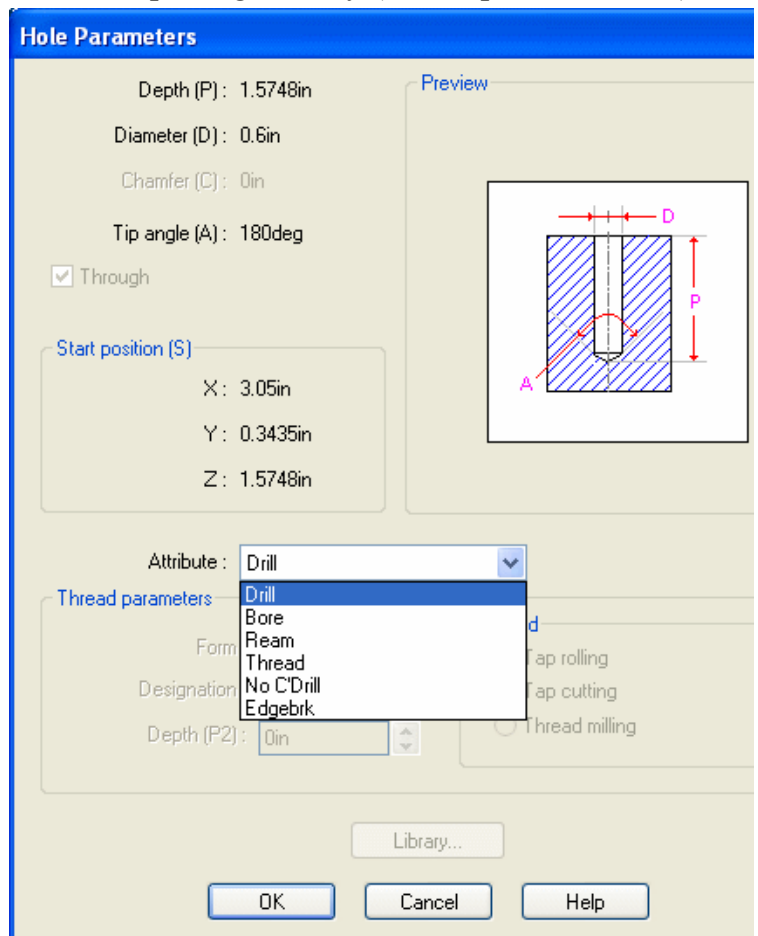
**Hole Parameters**

Depth (P) : 1.5748in
Diameter (D) : 0.6in
Chamfer (C) : 0in
Tip angle (A) : 180deg
☑ Through

Start position (S)
X : 3.05in
Y : 0.3435in
Z : 1.5748in

Attribute : Drill
- Drill
- Bore
- Ream
- Thread
- No C'Drill
- Edgebrk

Thread parameters
Form
Designation
Depth (P2) : 0in

Tap rolling
Tap cutting
Thread milling

Library...

OK    Cancel    Help

Depending on the attribute, a different machining process can be assigned. You can change how each feature is processed by defining the sequence in the Technology Database.

2. Click OK or Cancel to close the dialog box.

3. Review the parameters of other machinable features to understand more about the features.

Notice that AFR also created Mill Part Setup1. The Mill Part Setup defines the tool direction necessary to machine these features.

1. Click Mill Part Setup1 in the tree.

   A marker displays on the part. The large arrow indicates the machining direction. The cross in the center of the circle represents the machining X and Y axes. The X machining direction can be changed if necessary. For more information, see the CAMWorks online Help.
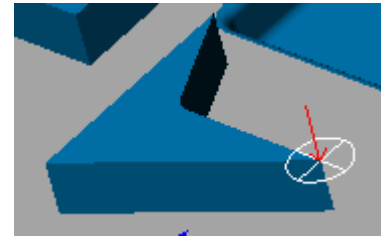
   The origin of the Mill Part Setup is program zero for the Mill Part Setup. By default, the origin is set to a vertex or middle of the top most feature. In most situations, you will want to change this. You learn how to change the Mill Part Setup origin in Chapter 3.

   If the triad does not display, click Redraw and select the Mill Part Setup again.

When you select Generate Operation Plan, operations are created for every selected feature based on the rules defined in the Technology database (TechDB). Operations are not generated for suppressed features and features in the Recycle Bin.

In this exercise, you create an operation plan for an individual machinable feature and then for all machinable features in the Mill Part Setup.

1. Right click Rectangular Pocket1 and select Parameters on the shortcut menu.

2. Click the down arrow to display the Attributes list and select Coarse.

   This is a system attribute that has been defined in the TechDB to generate a Rough Mill operation for a Rectangular Pocket feature. After you have learned how to use CAMWorks, you can modify system attributes to correspond to your requirements.

3. Click OK to close the dialog box.

4. Right click Rectangular Pocket1 in the tree, select Generate Operation Plan.
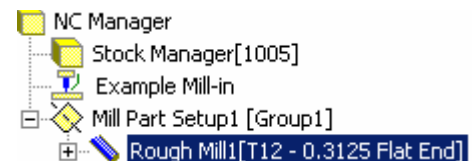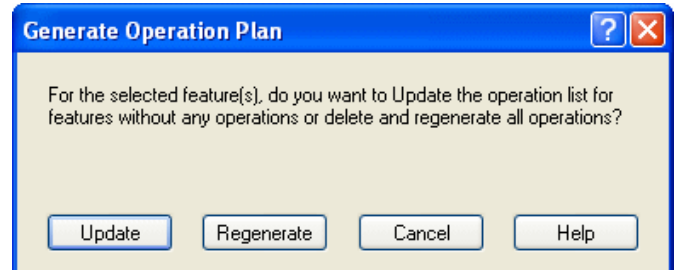
   The Operation tree lists a Rough Mill operation for the rectangular pocket.

5. Click the CAMWorks Feature Tree tab to return to the Feature tree.

6. Right click Rectangular Pocket1 and select Parameters on the shortcut menu.

7. Change the Attribute to Rough Finish in the Rectangular Pocket Parameters dialog box, then click OK.

This attribute is a user-defined attribute that has been set up in the TechDB to generate Rough Mill and Finish Mill operations for a Rectangular Pocket feature. You can modify or delete the user-defined attributes and you can define additional attributes to customize the TechDB.

8. Right click Rectangular Pocket1 in the tree, select Generate Operation Plan.

If Generate Operation Plan has been run once and you select the command again, CAMWorks displays a dialog box with the following choices:

Update: Select this option to generate operations for new features or for features that you deleted all previously generated operations. The operation list will not be updated for features that have previously generated operations.

Regenerate: If you select this option, all previously generated operations are deleted and new a new operation list is created. If you have modified any operations, these modifications will be lost.

9. Click Regenerate to delete the previous Rough Mill operation and generate a new operation list.

The Operation tree lists Rough Mill and Finish Mill operations for the rectangular pocket.
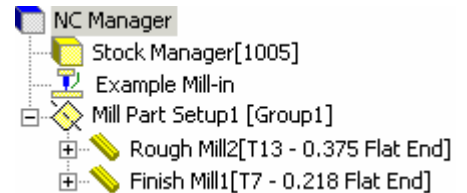
10. Click the CAMWorks Feature Tree tab to return to the Feature tree.

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

12. Click either Update or Regenerate in the Generate Operation Plan dialog box.

Since you have not changed any operation parameters for the Rough Mill and Finish Mill operations that have already been generated, you could select either Regenerate or Update. If you had modified these operations, you would want to select Update to generate an operation list only for the features that do not have operations.

The CAMWorks Operation tree now contains operations for all machinable features in the selected Mill Part Setup.

13. Click the CAMWorks Feature Tree tab.

14. Right click on MS Hole1 [MSH1(inch)] in the tree and select Go To First Operation on the shortcut menu.

This command allows you to quickly locate the first operation that was generated for a selected feature.

The Drill3 operation is highlighted in the Operation tree. In the TechDB, the attribute [MSH1(inch)] has been defined to generate a single drill cycle for multi-stepped hole features. Since multi-stepped holes can be any shape, you can define machining sequences in the TechDB that are common for machining multi-stepped holes in your facility.

To generate the NC code for this part, you would adjust the operation parameters as required for the operations, insert a series of operations for the multi-stepped hole, generate toolpaths, and post process the part.

# 2 Axis Mill 3

## What You'll Learn

- Defining the Stock from a Bounding Box
- Inserting Additional Mill Part Setups
- Creating a Face Feature
- Reorganizing Machinable Features
- Suppressing Machinable Features
- Inserting 2.5 Axis Features
- Combining Machinable Features

In this exercise, assume that the part starts out as a casting and machining is required only for the holes and certain faces.

In the first clamping position or Mill Part Setup, you will face the bottom of the part and drill the large center hole and the three outside holes. In the second clamping position or Mill Part Setup, you will machine off the top center face, and the three yellow ears.
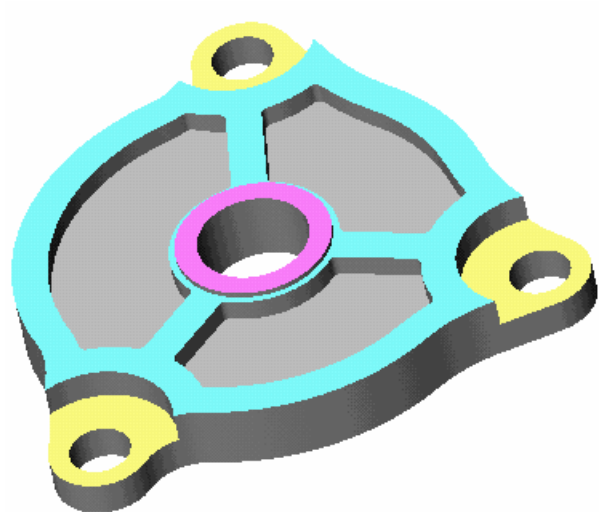
1. Open part file **MILL2AX_3.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Right click Example Mill–in in the Feature tree and select the Edit Definition command on the shortcut menu.

   – On the Machine tab, Example Mill–in is the Active machine.
   – Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   – Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

### Defining the Stock From a Bounding Box

The stock size, shape and material type is defined next.

The stock definition serves several purposes:

- Defines the stock size and shape for simulation.
- Specifies the material type, which is used by the TechDB for feed/speed calculations.

By default, the stock shape is a cube whose dimensions enclose the part. For this example, the part is machined from a casting that has .1in material to be removed from the top and bottom surfaces. You will use the default rectangle shape and offset the minimum stock size by .10in.

1.  Right click the Stock Manager in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.

2.  Change the Bounding box offsets for X, Y, and Z to a uniform **.1in**.

3.  Click OK.

## Extracting Machinable Features

Before CAMWorks extracts machinable features, you can customize AFR to eliminate Boss features:

1.  Click the Options button on the CAMWorks toolbar or select Options on the CAMWorks menu.

2.  Click the Mill Features tab in the Options dialog box.

3.  In the Feature types section, remove the check mark from Boss.

4.  Make sure the Face option is not checked.

    CAMWorks can recognize some Face features automatically when you select the Face option. However, for some parts you may need to define a Face Feature interactively as explained in this exercise.

5.  Click OK.

6.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.

    CAMWorks creates the Mill Part Setups necessary to machine all of the machinable features that are recognized.

    Other than drilling, the only additional operations needed to machine this part are facing cycles. CAMWorks may not be able to find all features on a part. On complex parts, AFR may not recognize all possible features that 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 define these features interactively.

    For learning purposes, in this exercise you delete the Irregular Corner Slot features and insert them interactively.

7.  Hold down the Shift key on the keyboard and click each Irregular Corner Slot in the tree to highlight the items, then select Delete on the shortcut menu.
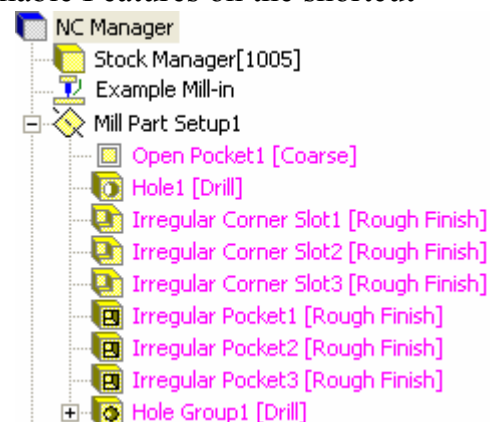
8.  Click Yes to confirm the deletion.

    The features are moved to the Recycle Bin. When a feature is deleted, it is automatically placed in the Recycle Bin, which is used to store machinable features that you do not intend to machine.

9.  Click the minus sign to the left of the Recycle Bin to collapse it.

10. Click Mill Part Setup1 in the tree and notice the machining direction indicated by the triad that displays in the graphics area.

### Inserting Additional Mill Part Setups

Mill Part Setup1 defines a tool direction pointing down from the top of the part. For this exercise, you want to face the underside of the part first. Before you can machine the underside, you must define a Mill Part Setup for that tool direction. New Mill Part Setups are inserted into the CAMWorks Feature tree by selecting a part face that is normal to the tool axis of the Mill Part Setup.

1.  Rotate the part so that the underside surface is visible.

2.  Right click Mill Part Setup1 in the CAMWorks Feature tree and select Insert Mill Part Setup. The Mill Part Setup dialog box displays.

3.  In the graphics area, pick the underside face.
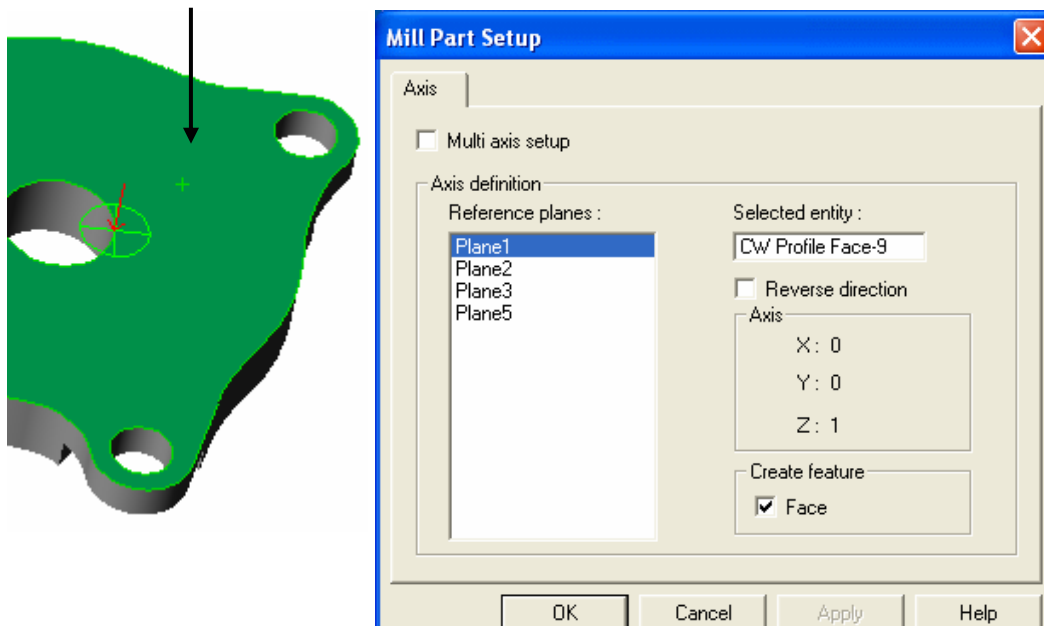


4.  Make sure the Face option is checked.

    If this option is checked, when CAMWorks creates a Mill Part Setup, a face feature is created automatically if the top most face is parallel to the Setup and the Setup is parallel to one of the sides of the stock.

5.  Click OK.

    Mill Part Setup2 is created with the tool direction as shown in the figure above. The Face Feature is created automatically and listed under the Mill Part Setup.

The names for Mill Part Setup1 and Mill Part Setup2 can be changed to Top and Bottom or some other recognized convention.

6. Right click Mill Part Setup1 and select Rename on the shortcut menu.

7. Change the name to **Top Part Setup**, then press Enter.

   Notice that when you click a Mill Part Setup in the tree, an axis displays on the part to indicate the tool direction.

8. Right click Mill Part Setup2 and select Rename.

9. Change the name to **Bottom Part Setup**, then press Enter.
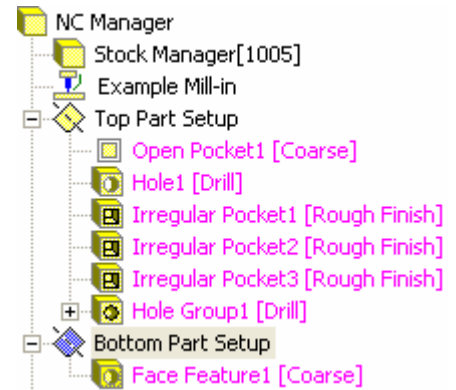
## *Reorganizing Machinable Features*

You also want to drill the holes in the Bottom Part Setup. However, the hole features were recognized in the Top Part Setup. With the mouse, you can drag the features in the CAMWorks Feature tree from the Top Part Setup to the Bottom Part Setup.

1. Hold down the left mouse button and drag Hole1 over Bottom Part Setup, then release the button.

   – Notice that as you drag the item, an arrow displays to indicate where the item will be positioned.

   – Hole1 is listed after Bottom Part Setup.

2. Hold down the left mouse button and drag Hole Group1 over Hole1, then release the button.

   Hole Group1 is listed after Hole1.
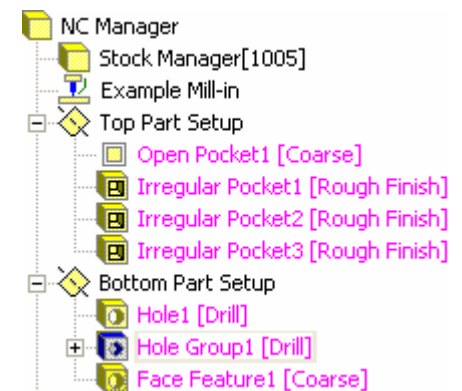
## *Suppressing Machinable Features*

You will now define features to machine the top of the part. The only machinable features left in the Top Part Setup are the Open Pocket and the three irregular pockets. Since these features will not be machined in this exercise, you can either delete or suppress them. The steps below show you how to suppress these features.

1. Hold down the CTRL key and select Open Pocket1, Irregular Pocket1, Irregular Pocket2, and Irregular Pocket3.

   Make sure you select all 4 features.

2. Right click and select Suppress from the shortcut menu.

   The icons for the features display in gray instead of yellow to indicate these features are suppressed.

3. Click Save As on the File menu.

4. Type **cwex3** and click the Save button.
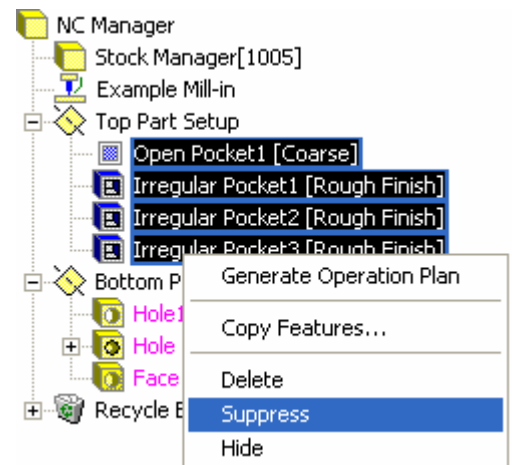
## *Inserting 2.5 Axis Features*

The top of the circular boss and the three ears need to be machined. You use the Insert 2.5 Axis Feature command to define them.

1. Rotate the part so the top face is visible.

2. Right click Top Part Setup in the CAMWorks Feature tree and select Insert 2.5 Axis Feature.

First, you will define a Face Feature for the top of the boss.

3. Click the down arrow to display the Feature Type list and select Face Feature.

4. Pick the top center face (magenta) for the top of the boss as shown in the figure.

   Face <1> displays in the Entities selected list.

5. Click the Next button.

6. In the 2.5 Axis Feature Wizard: End Conditions dialog box, make sure the Edit feature profiles option is not selected, then click Finish.

   The 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box displays again and you can continue to define 2.5 Axis features for the three ears.
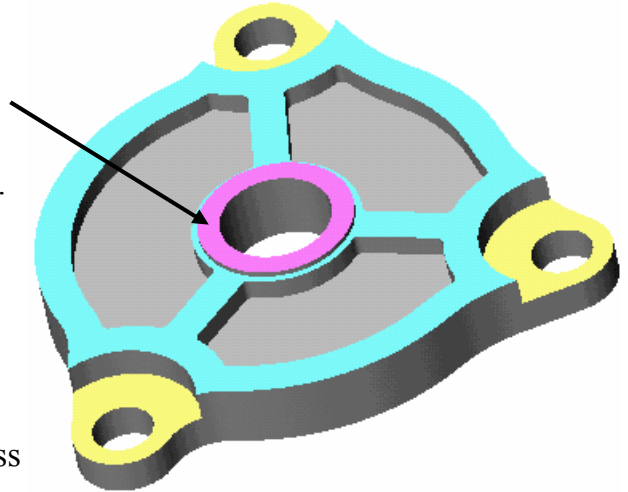
7. Click the down arrow to display the Feature Type list and select Corner Slot.

   The ears are defined as Corner Slots because more than one edge is open. By defining it as a corner slot, the tool will machine outside the ear on the open air edges.

8. Make sure the Multiple option is selected. This option allows you to define multiple similar features by selecting multiple faces, sketches and edges.

   A sketch with geometry representing multiple features can also be selected. This option can be selected for Pocket, Slot, Corner Slot, Boss, Hole, and Open Profile feature types.
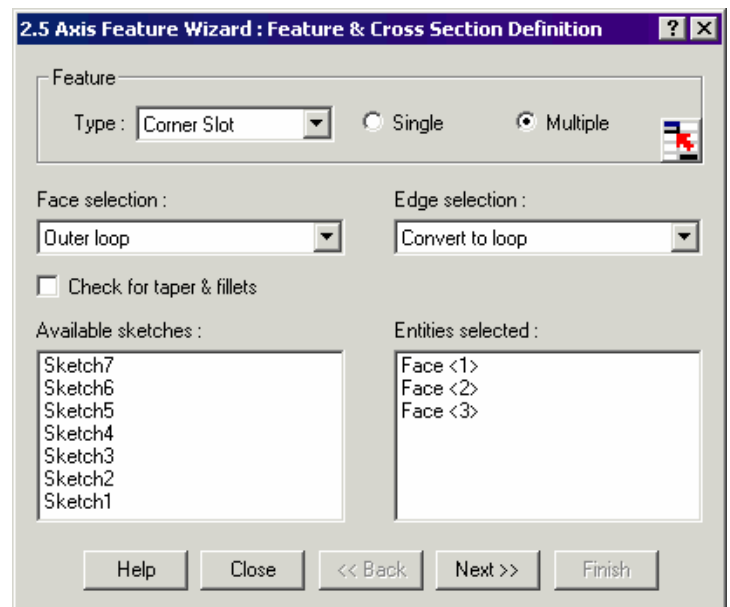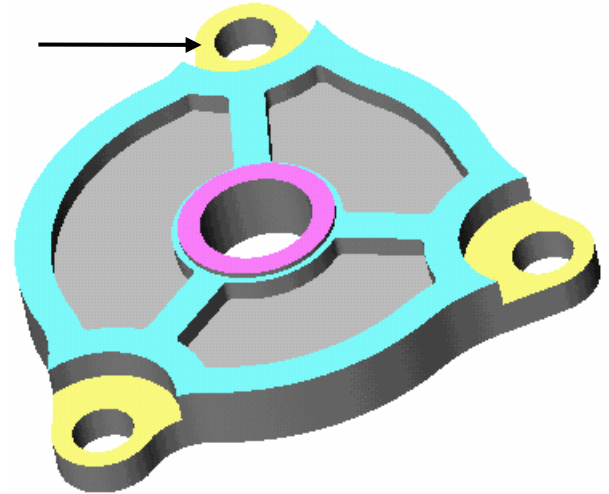
9. Pick each of the three ears on the part.

   The Entities selected list shows Face <1>, Face <2> and Face <3>.

10. Click Next.

    The 2.5 Axis Feature Wizard: End Conditions dialog box displays.

The depth of material to leave for this feature will be from the face of the ear to the cyan face immediately above the ear. Like SolidWorks, the depth can be specified based on faces or vertex points on the part. You can either pick an end condition from the End Condition pulldown or select a face or vertex and have CAMWorks set the depth automatically.

11. Leave the End condition set to Blind and pick the cyan colored face above the ear.

    The End condition is automatically set to Upto Face and the Depth parameter displays the distance as 0.14in.

12. Make sure the Edit feature profiles option is checked.

13. Click Next.

    The Edit Feature Profiles dialog box displays. When a Corner Slot is defined from a part face, CAMWorks determines the open air edges automatically. If necessary, you can change the open air edges.

14. Select a segment in the Profile Segments list.

    The corresponding edge is highlighted on Irregular Corner Slot(1). If CAMWorks determined that the edge is an open air edge, the Open air edge option is checked.

15. Select several segments in the Profile segments list. You do not need to change any segments.
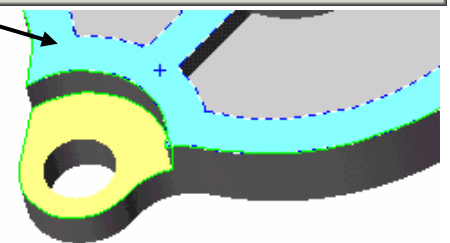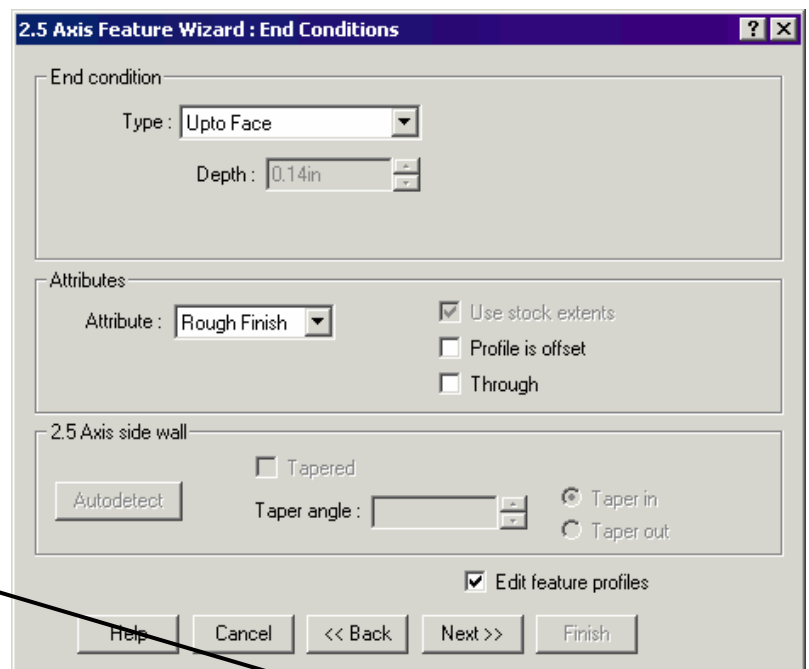
16. Click Next.

The 2.5 Axis Wizard: Island End Conditions dialog box displays. This dialog box displays when you define a Pocket, Slot, Corner Slot, or Face Feature.

17. Make sure the Detect for all option is checked.

When you define multiple features and select this option, CAMWorks attempts to find islands for all features. If this option is not checked, CAMWorks attempts to find islands only for the current feature.
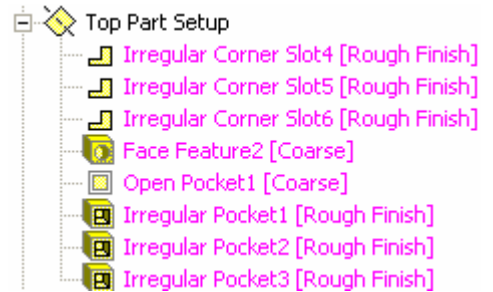
18. Click Autodetect.

CAMWorks does not find any islands for the corner slots. If necessary, you could click the Add button and interactively insert islands.

19. Click Finish.

The Irregular Corner Slot features are listed in the tree. All machinable features are now defined.

20. Click Close to exit the wizard.

## Combining Machinable Features

Three Irregular Corner Slots were defined for the ears. If you were to generate an operation plan now, separate operations would be created for each feature. For example, if the Technology Database defines that an Irregular Corner Slot feature will be machined with a rough and finish cycle, Generate Operation Plan would create three Rough Mill operations and three Finish Mill operations for these features.

To make it easier to manage the operation parameters for these features, you can make a single Group feature that contains all three features. As a Group feature, the three Irregular Corner Slot features will be machined with one set of operations.

1. Right click the first Irregular Corner Slot in the CAMWorks Feature tree and select Insert Group.

The Edit Feature Group dialog box displays.
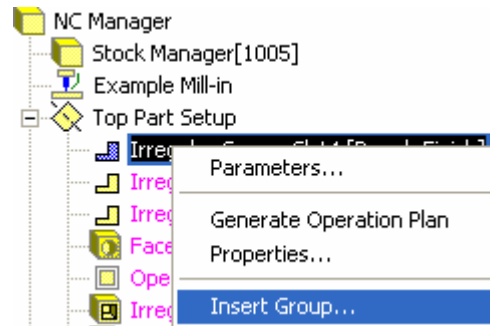
The Feature List on the right displays the selected feature. The Available Features list on the left shows other features of the same type and depth that can be added to make the Group feature.

2. Pick Irregular Corner Slot5, then hold down the CTRL key and pick Irregular Corner Slot6.

3. Click Add.

   The two features are added to the Feature List on the right.

4. Click OK.

   The three pockets are combined into a single feature labeled Irregular Corner Slot Group2 in the CAMWorks Feature tree.
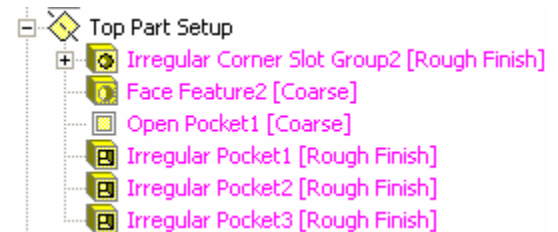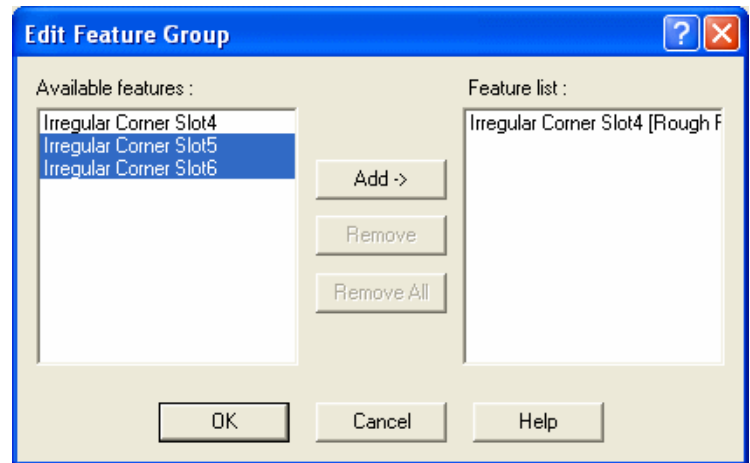
Generate an operation plan for each Mill Part Setup:

1. [≡↓] Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the CAMWorks Feature tree, select Generate Operation Plan.

   Operations are created for the machinable features in the Bottom and Top Part Setup.

   The operations are in listed in the Operation tree in the same order as their associated machinable features. You can easily change the order.

2. Press and hold down the left mouse button with the pointer over Face Mill1.

3. Drag the operation over the top of Top Part Setup and release the button.

   The Operation tree redisplays with the operation moved to the new location.

4. Repeat steps 2 and 3 for Face Mill2.
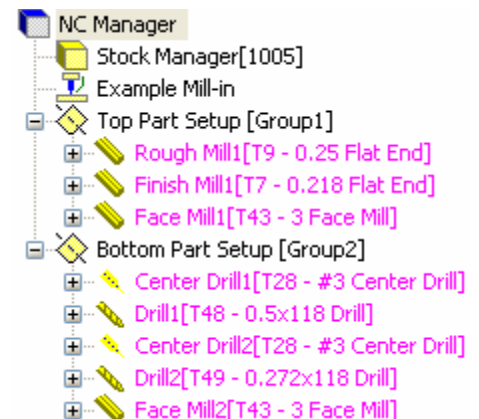
Generate toolpaths and post process the part:

1. Right click each operation, select Edit Definition, and review the cutting parameters.

2. After you have verified that the parameters for each operation are set the way you want them, right click Bottom Part Setup in the CAMWorks Operation tree and select Generate Toolpath.

3. Repeat step 2 for the operations in Top Part Setup.

4. Check the CAMWorks Message Window.

   – The Message Window provides information you may find helpful. For example, based on the information in this window, you may want to change a tool or insert a finish operation in order to cut a machinable feature completely.

   – If the Message Window is not displayed, select Options on the CAMWorks menu and check the Message Window check box on the General tab.

   – You can move the window and adjust the size. To change the size, point to a border or corner of the window (the pointer changes to a two-headed arrow). Drag the corner or border until the window is the size you want, then release the mouse button.

5. Experiment by changing parameters for the operations and regenerating the toolpaths.

CAMWorks provides the ability to simulate toolpaths showing the tool movement and the resulting shape of the part.

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

7. Click the Run button.

8. Click the X button in the upper right corner of the Simulation toolbar to exit Simulation mode.

9. Right click Bottom Part Setup in the CAMWorks Operation tree and select Post Process.

10. In the Save Post File dialog box, click Save to accept the default file name (cwex3.txt).

   Note that if you are running CAMWorks in Demo mode, this dialog box does not display.

11. In the Post Process Output dialog box, check the Display Centerline option so that the toolpath will be highlighted as each line of code is produced.

12. Click the Run button (the double arrow).

   NC code is created for the Bottom Part Setup.

13. After viewing the code, click Cancel.

14. Right click Top Part Setup in the CAMWorks Operation tree and select Post Process.

15. In the Save Post File dialog box, type **cwex3top**.

16. In the Post Process Output dialog box, click the Run button (the double arrow).

# 2 Axis Mill 4

## What You'll Learn

- Defining the Stock from a Sketch
- Defining Additional Machinable Features
- Adjusting Operation Parameters
- Deleting an Operation
- Inserting an Operation
- Defining Program Zero
- Sorting Operations

This lesson guides you through the steps to machine the part shown below. The bottom depressions of the part will not be machined in this exercise.

Sequence of machining:

Drill two holes on each end of part.
Face top of part.
Face depression on left side of part.
Finish profile the hole on the left.
Rough and finish rectangular pocket.
Finish machine three obround holes.
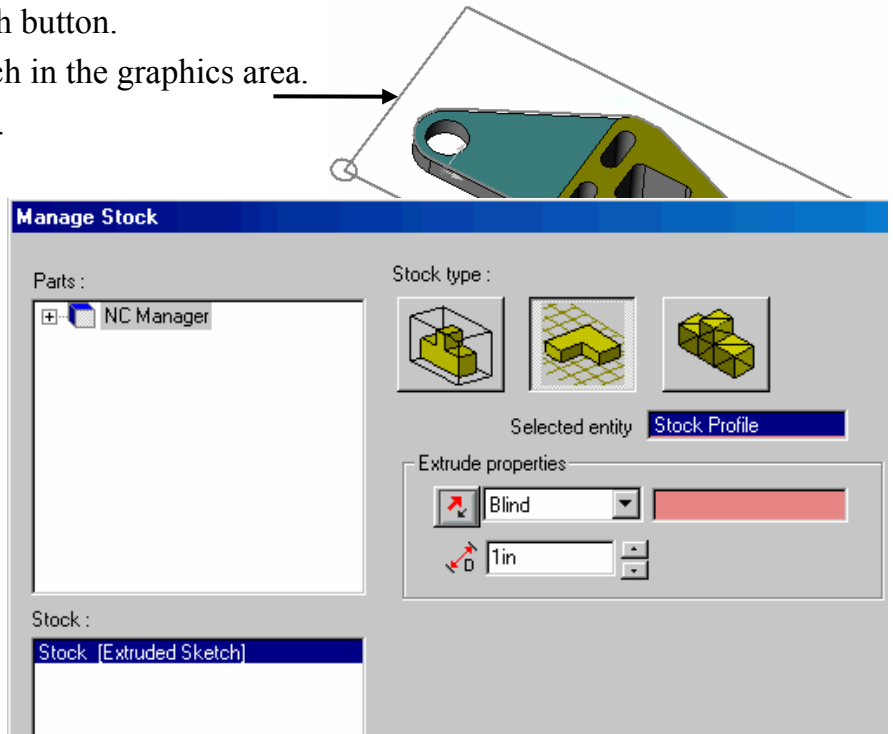Rough and finish outside part shape
from stock.
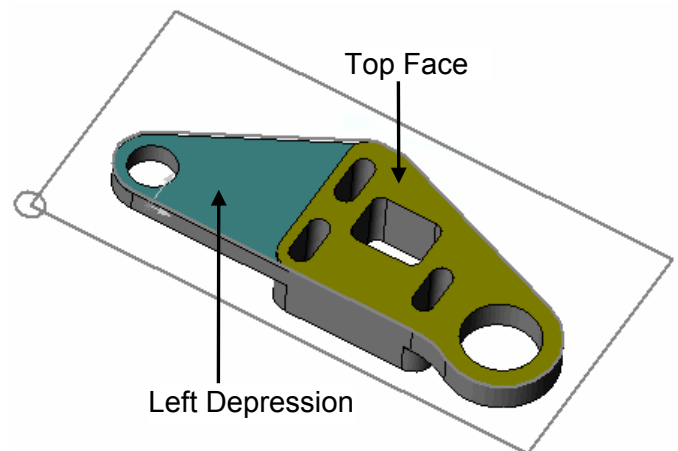


1. Open the part file **MILL2AX_4.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Right click Example Mill–in in the Feature tree and select Edit Definition.

   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

### Defining the Stock from a Sketch

In this example, the stock is a block of material that does not have the same amount of material on all sides. Therefore, you will define it using a rectangular sketch and a depth. A sketch named 'Stock Profile' has already been created in this part. The sketch plane is positioned .05in above the top of the part.

1. 🗔 Right click Stock Manager in the CAMWorks Feature tree and select Edit Definition.

   CAMWorks automatically calculates the stock size and shape based on the smallest block possible. You can change this definition by specifying a sketch and depth.

2. Click the Extruded Sketch button.

3. Pick the rectangular sketch in the graphics area.

   The sketch is highlighted.

4. ⬦ Set the Depth to **1in**.

5. ⬈ If necessary, click the Reverse Direction button to extrude the stock in the correct direction.

6. Click OK.

7. Left click Stock Manager in the tree to see an outline of the stock.

**Manage Stock**

Parts :
☐ NC Manager

Stock type :

Selected entity  Stock Profile

Extrude properties
⬈ Blind ▼
⬦ 1in

Stock :
Stock [Extruded Sketch]

## *Extracting Machinable Features*

Define machinable features automatically:

1. ⚒ Click the Options button on the CAMWorks toolbar.

2. Click the Mill Features tab in the Options dialog box.

3. Check the Face option in the Feature types section and click OK.

4. 🖼 Click the Extract Machinable Features button on the CAMWorks toolbar.

   The recognized features include the face feature, two holes, the rectangular pocket and the three obround pockets, which are included in Obround Pocket Group1.

   NC Manager
   Stock Manager[1005]
   Example Mill-in
   ☐ Mill Part Setup1
       Face Feature1 [Coarse]
       Rectangular Pocket1 [Rough Finish]
       Hole1 [Drill]
       ☐ Obround Pocket Group1 [Rough Finish]
       Hole2 [Drill]
   Recycle Bin

5. Click Mill Part Setup1 in the Feature tree and check the machining direction that displays on the part.

   Because all the features go through the part, AFR may determine the machining direction is from the bottom. If the direction is from the bottom instead of from the top as shown on the right, right click Mill Part Setup1 and select Edit Definition. Click the Reverse Direction button, then click OK.
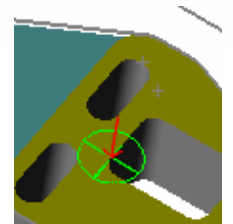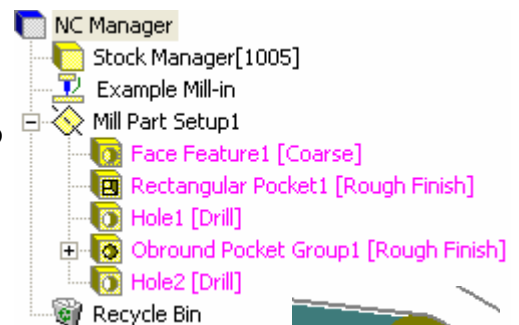
Correct direction

## *Defining Additional Features*

For this example, you interactively define two machinable features.

− Irregular Corner Slot (to machine the depression on the left side of the part)
− Open Pocket (defines the outside profile of the part so it can be machined from the stock)

One of the features is created from part faces and the second from sketches.

First, you create a 2.5 Axis feature for the depression on the left side of the part. To completely remove the material along the open edges, you need to define the feature as an Irregular Corner Slot. By definition, an Irregular Corner Slot has two or more adjacent edges that have open air edges.

1.  Right click Mill Part Setup1 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.  Select Corner Slot for the Feature Type and pick the top face of the depression on the left of the part.

3.  Click Next. The 2.5 Axis Feature Wizard: End Conditions dialog box displays.

4.  Pick the topmost face on the right.
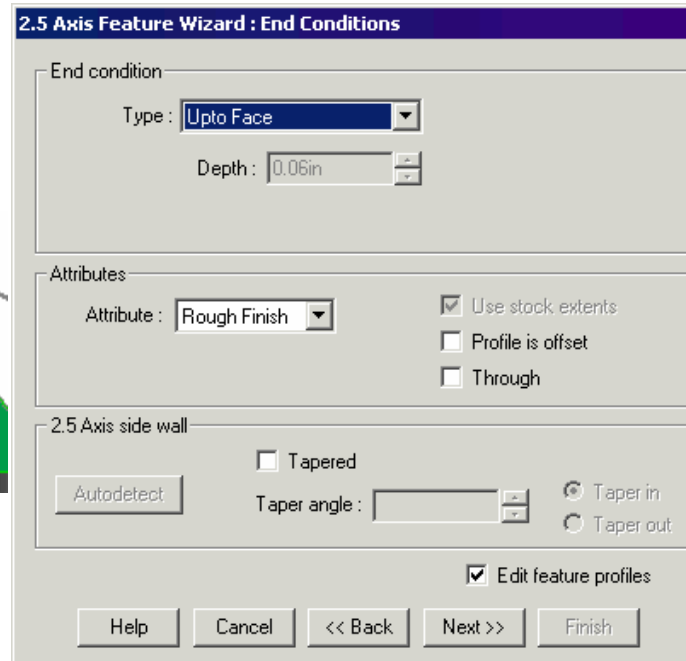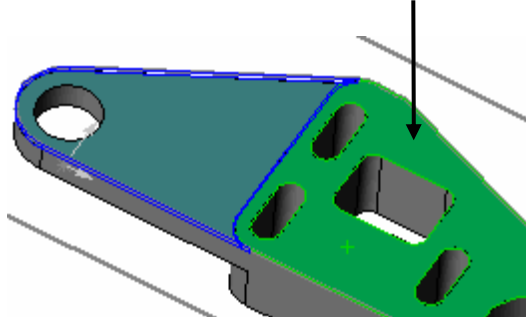
    CAMWorks sets the End condition to Upto Face and the Depth to 0.0600in.

5.  Make sure the Edit feature profiles option is checked, then click Next.
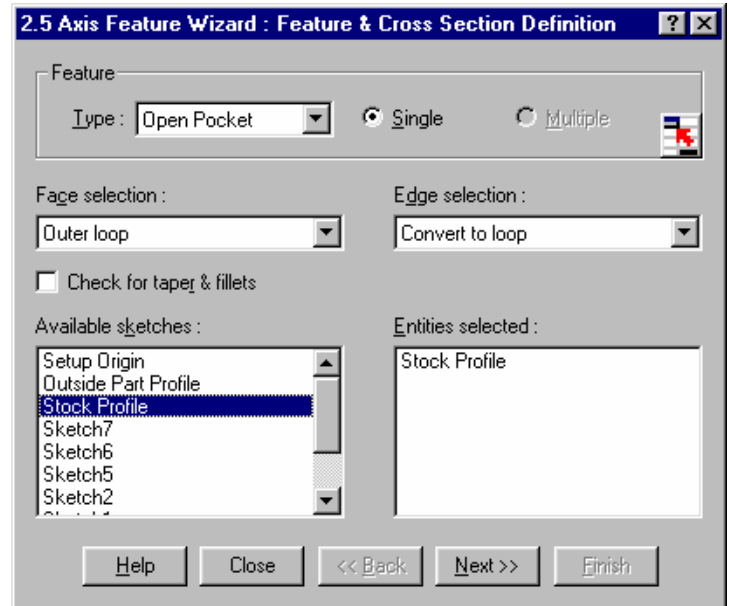
    The 2.5 Axis Feature Wizard: Edit Feature Profiles dialog box displays. When a Corner Slot or Slot feature is defined from a part face, CAMWorks automatically determines the open air edges. If necessary, you can change the open air edges.

6.  Click the segments in the Profile Segments list to view the open air edges selected by CAMWorks. You do not need to change any segments.

7.  Click Finish. Irregular Corner Slot1 is created and added to the CAMWorks Feature tree.

The second 2.5 Axis feature will be used to machine the perimeter of the part. For this example, you will rough the part from the stock, then finish machine the perimeter of the part. In order to do this, you define the stock perimeter as an Open Pocket and the part as an island in this pocket.

By defining this 2.5 Axis feature as an Open Pocket, the toolpath will extend beyond the stock perimeter so that all of the material will be removed. In comparison, if the stock perimeter were defined as a simple Pocket, the roughing toolpath would be constrained within the perimeter of the stock leaving unmachined areas in the corners.

1. In the 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box, select Open Pocket for the Type.

2. In the Available sketches list, select the sketch labeled Stock Profile and click Next.

3. In the 2.5 Axis Feature Wizard: End Conditions dialog box, set the End condition Type to Upto Stock.

4. Select the Through option.

   With this option checked, the feature will be machined slightly deeper than the stock to remove all material.

5. Click Next.

   The 2.5 Axis Feature Wizard: Island End Conditions dialog box displays.
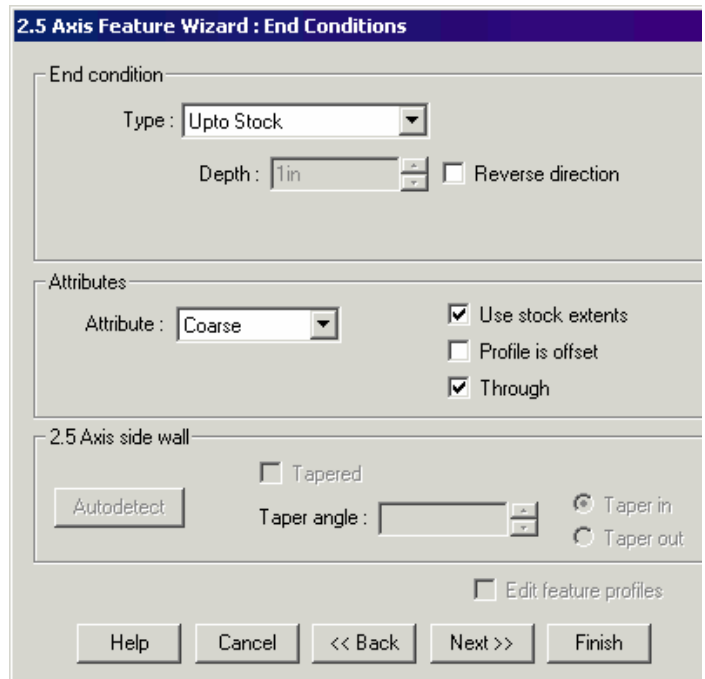
6. Click the Autodetect button.

   CAMWorks considers a flat face that is normal to the Mill Part Setup as an island. In this part, the top face of the part and the left depression face are both flat and normal to the current Mill Part Setup and will be identified as islands automatically.
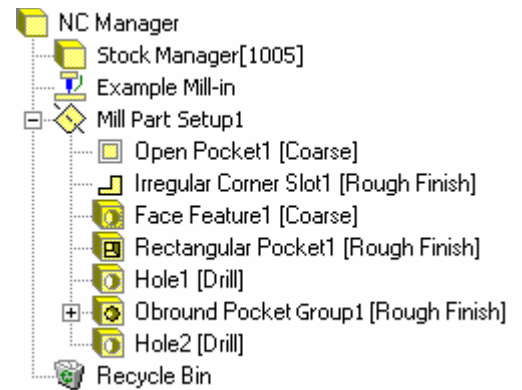
   If necessary, you can define islands interactively by clicking the Add button and selecting faces, edges, loops and sketches.
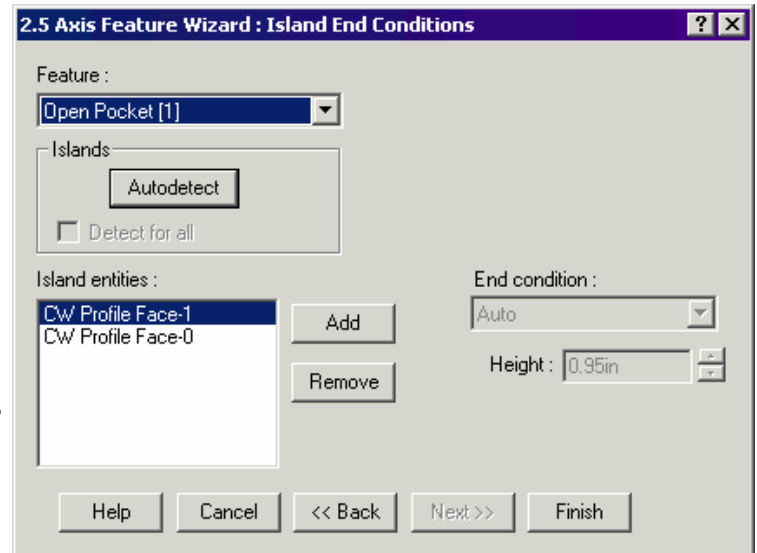
7. Click Finish.

   The last 2.5 Axis feature is now complete and all machinable features are listed in the CAMWorks Feature tree.

8. Click Close to exit the Wizard.

## Generating Operations

Before the operation plan is generated, attributes can be assigned to features to affect the operations that are created for each feature.

1. Right click Rectangular Pocket1 and select Parameters.

2. In the Rectangular Pocket Parameters dialog box, set the Attribute to Coarse, then click OK.

   This system attribute has been set up in the TechDB to generate a Rough Mill operation for a Rectangular Pocket feature.

3. Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 in the tree and select Generate Operation Plan.

   The operations are listed in the Operation tree.
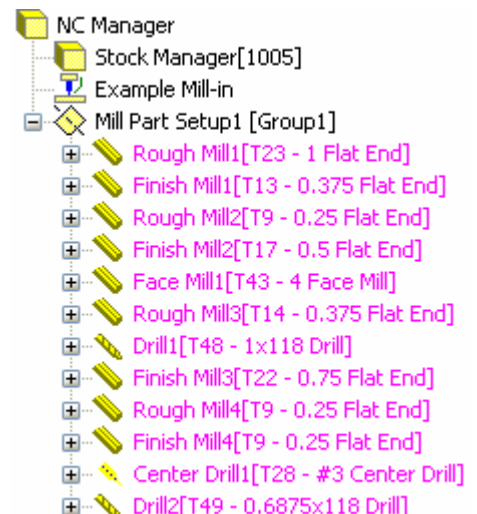
## *Adjusting Operation Parameters*

The operations that are generated by CAMWorks are based on information stored in the Technology Database. 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.

1. Click the + next to Finish Mill4 in the CAMWorks Operation tree.

   The tree expands to identify the Obround Pocket Group feature the operation was generated for.

2. Right click Finish Mill4 and select Collapse Items on the shortcut menu.

   You can use either the + and – symbols or the Expand and Collapse Items commands on the shortcut menus to view items in the trees.

3. Right click Finish Mill4 again and select Edit Definition.

4. In the Operation Parameters dialog box, click the Tool Crib tab.

5. Select a **.1875in** diameter flat end mill.

   If Flat End tools are not listed, clear the check mark to the left of the Filter button.

6. Click the <-- Select button, then click Yes to replace the corresponding holder.

7. Click the Finish tab.

8. In the Side parameters section, make sure the Allowance is set to **0in**.

   This will cut the pockets to size.

9. Click OK to close the Operation Parameters dialog box.
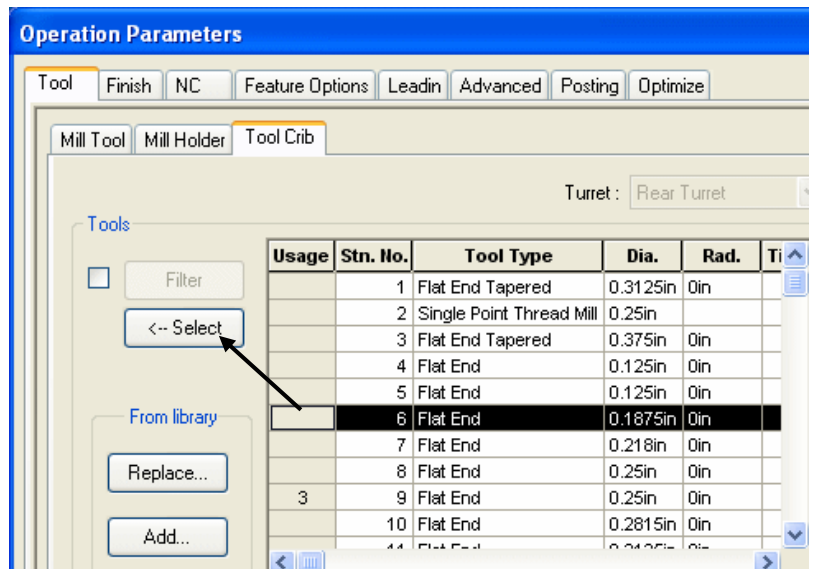
## *Deleting an Operation*

In some cases, it may be desirable to delete operations that are created automatically. In this example, Rough Mill and Finish Mill operations were created for the Obround Pocket Group1 machinable feature. Based on the tool that is going to be used for the obround hole, a Finish Mill operation is all that is needed. You can delete the Rough Mill operation.

1. In the CAMWorks Operation tree, right click Rough Mill4 (the roughing operation for the obround slots) and select Delete on the shortcut menu.

2. Click Yes to confirm you want to delete the operation. The operation is removed from the tree.

## *Inserting an Operation*

The larger hole on the right has been drilled. In this example, you also want to finish mill around the inside of the hole. To do this, you need to add a Finish Mill operation.

1. Right click Drill1 in the CAMWorks Operation tree and select Insert Hole Operation.

   The Insert hole operation dialog box displays.
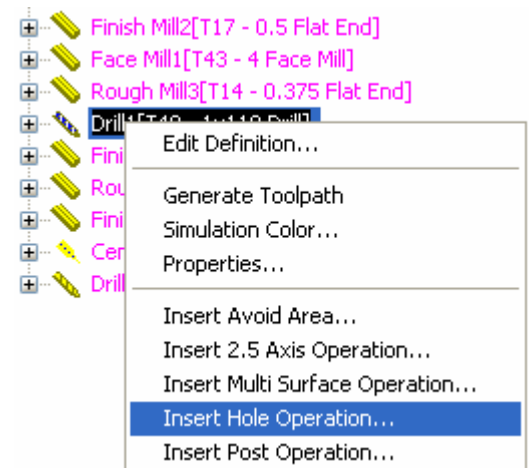
2. Set the Type of operation to Finish Mill.

3. Select Hole1 in the Features list on the left and click Define.
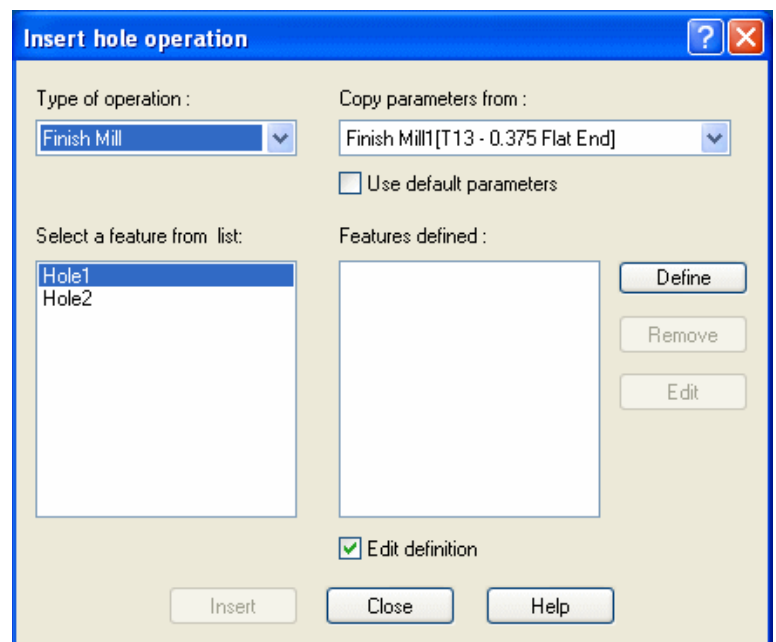
The Hole Feature Definition dialog box displays. This dialog box allows you to define the top and bottom of the hole and the hole diameter. Since this hole contains only a single diameter, the start and end location of the hole diameter are set automatically to the correct values.

4. Click OK.

   The Insert Hole Operation dialog box displays again.

5. Make sure the Edit definition option is checked, then click Insert.

   The Operation Parameters dialog box displays.

6. If a **.5in** flat end mill is not selected on the Mill Tool page, click the Tool Crib tab and select it from the Tools list, then click the <-- Select button.

7. Click Yes to replace the holder, then click OK to close the Operation Parameters dialog box.

   A finish operation is added after the Drill1 operation in the tree.

## Defining Program Zero

The Mill Part Setup defines the tool direction as well as program zero. Program zero can be changed in the Mill Part Setup by selecting a vertex or arc edge, typing in the origin relative to SolidWorks zero, identifying a sketch that contains a single circle, or by picking a stock corner. When selecting a sketch, the center of the circle in the sketch will be the origin of the Mill Part Setup and program zero. Each Mill Part Setup can have a different origin.

1. Right click Mill Part Setup1 and select Edit Definition.

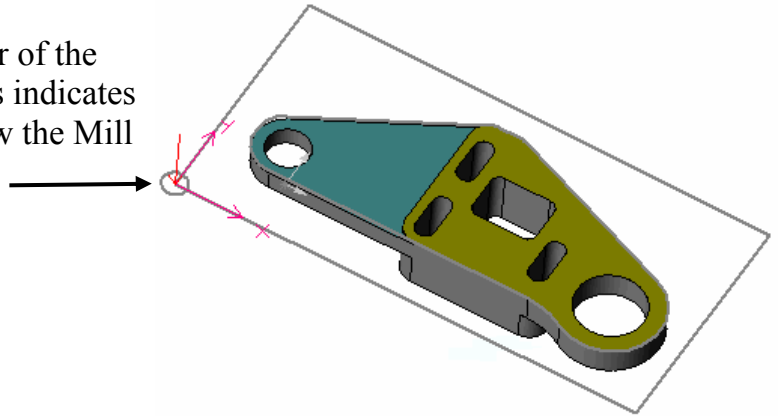   The Part Setup Parameters dialog box displays.
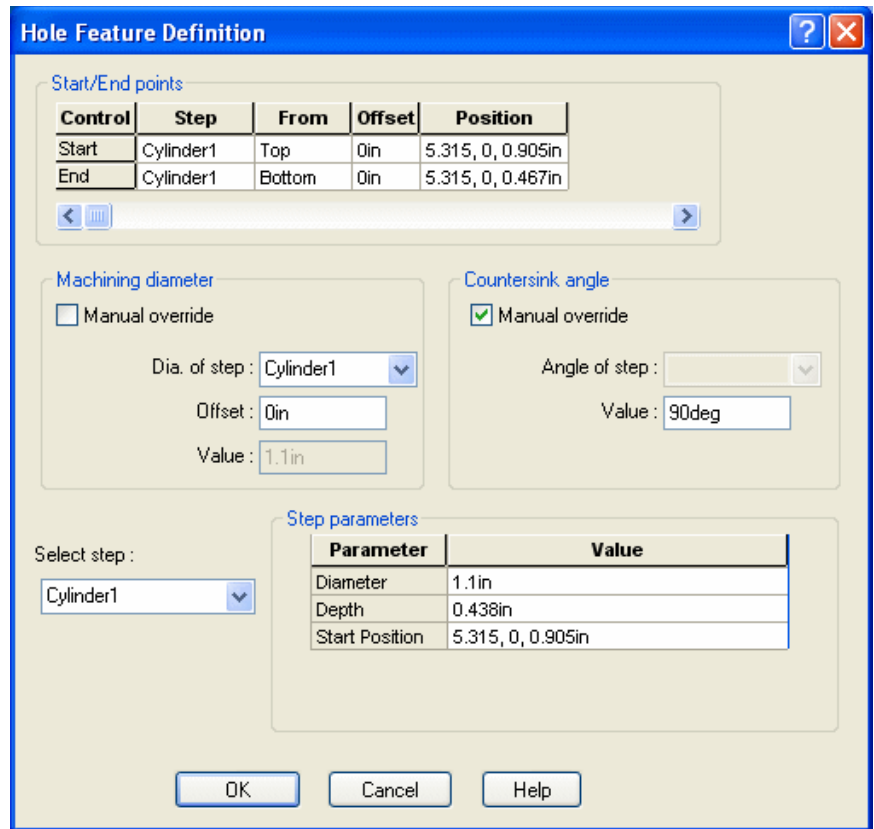
2. On the Origin tab, select the Sketch option and select Setup Origin for the sketch.
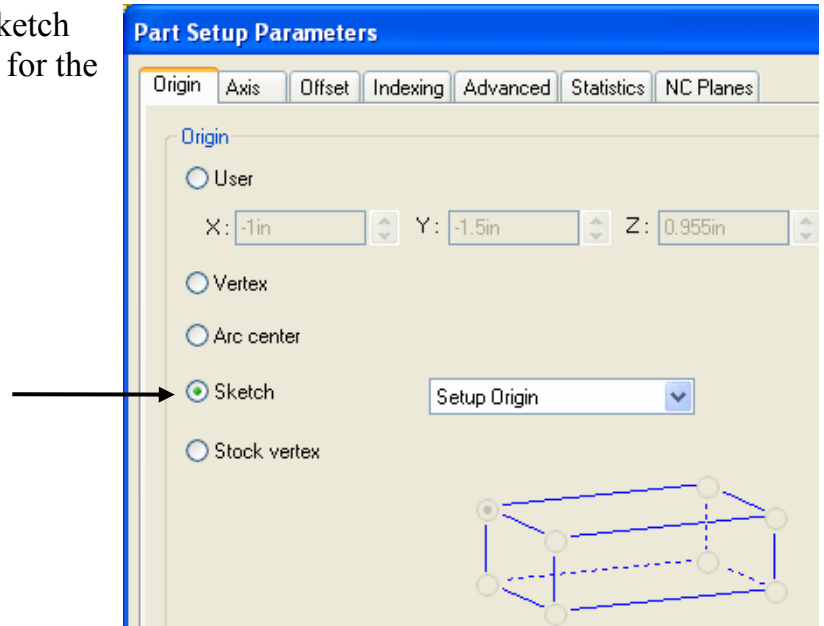


The circle in the lower left corner of the stock should be highlighted. This indicates that the center of the circle is now the Mill Part Setup origin.

3. Click OK.



**Did You Know ...**

If you change the Mill Part Setup origin after generating toolpaths, always regenerate all toolpaths. Otherwise, the post processed output will not be relative to the origin.

Generate toolpaths:

1. ⬚ Click the Generate Toolpath button on the CAMWorks toolbar or right click Mill Part Setup1 in the CAMWorks Operation tree and select Generate Toolpath.

   The toolpaths that were generated for all operations display.

2. Right click Mill Part Setup1 and select Simulate Toolpath.

3. ▶ Click the Run button to simulate the toolpaths.

4. ❌ Click the Close button to exit the simulation.
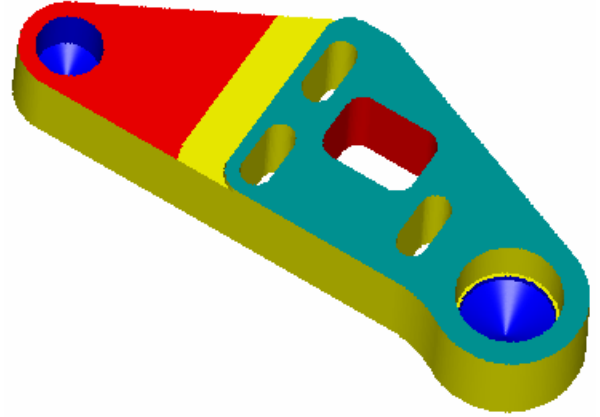
## *Sorting Operations*

When the Generate Operation Plan command is run, operations are created for each feature in the Feature tree in the order listed. As shown above, operations can also be inserted manually and deleted. After making these changes, it is possible that the machining order is not in the most efficient sequence. Using drag and drop, operations can be reordered into an optimized sequence; however, this could become time consuming for a part that includes dozens of operations. To make the ordering task easier, CAMWorks provides a Sort command that sorts operations based on their type, depth and tool.

1. Right click Mill Part Setup1 and select Sort Operations.

   The Sort Operations dialog box displays.

2. On the Process tab, remove the check mark from the Process complete feature option.

   When this option is checked, all operations for a given feature will be sorted and grouped together so that the entire feature is completed before processing other feature operations. After all feature operations are grouped, the order the operations are processed will be from highest Z face to lowest Z face.

3. Select the Sort tab.

   This page shows the order in which the operations will be sorted. More information on sorting operations can be found in the CAMWorks online help.

4. Click the Apply button.

   Note that the operations have been sorted according the specified operation sequence. Within each type of operation, the operations with the highest Z face will be processed first. If you do not like the order, you can click the Undo button and change the order or any of the options, then click Apply again to see the results.

5. Click the OK button.

   If required, the operations can still be reordered manually using drag and drop.

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

7. Run the simulation and note the changes.

8. Save the part.

9. Right click Mill Part Setup1 and select Post Process.

   – The Post Output File dialog box displays.

   – If you are running CAMWorks in Demo mode, this dialog box does not display.

10. Click Save to use the default name for the NC program file.

    The Post Process Output dialog box displays.

11. Click the Run button.

    CAMWorks generates the NC program.

12. Use the vertical scroll bar to view the code.

13. Click Cancel to close the Post Process Output dialog box.

# Chapter 2  Learning More 2 Axis Mill

This chapter provides an opportunity to learn more about CAMWorks 2 Axis Mill.

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.

# 2 Axis Mill 5

## What You'll Learn

- Setting Attributes and Generating Operations
- Sorting and Sequencing Operations
- Machining Island Tops
- Changing the Mill Part Setup Origin
- Combining Operations

1. Open the part file **MILL2AX_5.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

   Since this part is rectangular, you do not have to machine the outside of the part. AFR will find the holes, bosses, slot, corner slots and open pocket features.

2. ⊞ Click the CAMWorks Feature Tree tab.

3. 🗗 Right click Example Mill–mm in the Feature tree and select Edit Definition.

   - On the Machine tab, Example Mill–mm is the Active machine.
   - Click the Tool Crib tab and make sure Tool Crib 1 (metric) is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the dialog box.

4. 🗀 Right click on the Stock Manager in the CAMWorks Feature tree and select Edit Definition on the shortcut menu. The Manage Stock dialog box displays.

5. Change the Bounding box offset for X, Y and Z to a uniform **1mm**.

6. Select a Material in the pulldown list, then click OK.

7. ⚒ Click the Options button on the CAMWorks toolbar or select Options on the CAMWorks menu.

8. Click the Mill Features tab in the Options dialog box.

9. In the Feature types section, make sure the Face option is not checked and click OK.

10. 🗗 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.
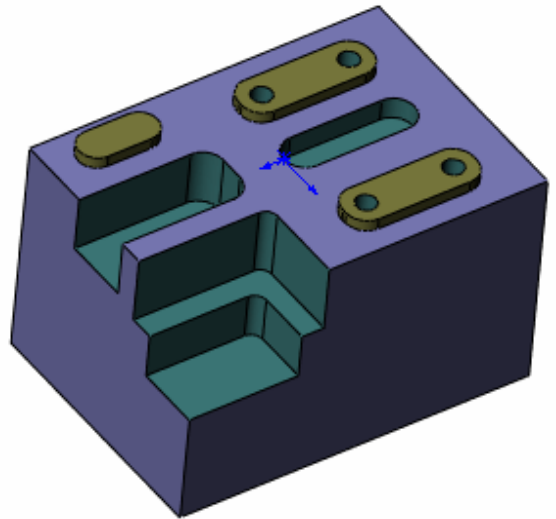
Automatic Feature Recognition analyzes the model and extracts the features to machine. The tree displays the list of features that were found.

- − An Open Pocket feature was created with the Obround Boss and the two Rectangular Bosses as islands.
- − The two rectangular bosses were extracted as a Rectangular Boss Group.
- − The Rectangular Corner Slots were not extracted as a Group Feature because they are different sizes.
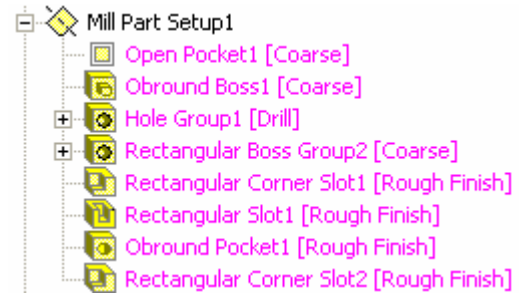
*Did You Know ...*

The default machining sequence is based on the order that the machinable features appear in the list. Using drag and drop, you could rearrange the features so that they are in the order that you would like them to be machined. However, the CAMWorks Operation tree allows operations to be automatically sorted and interactively sequenced (as explained below) so it is unnecessary to reorder features.

## Setting Attributes and Generating Operations

1. Right click on Hole Group1 in the CAMWorks Feature tree and select Parameters.

2. In the Hole Parameters dialog box, change the Attribute to Ream, then click OK.

3. Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 in the Feature tree and select Generate Operation Plan.

   CAMWorks creates the list of operations to machine the part. All the operations for a given Feature are listed in order.

## Sorting and Sequencing Operations

The machining sequence is based on the order of the operations in the tree. The Rough Mill and Finish Mill operations for the Rectangular Corner Slot are listed at the bottom. You may not want this order. For example, you may want all of the roughing done before the finishing, and the hole making operations to be done last. CAMWorks allows automatic and interactive control over changing the machining sequence. Using drag and drop is suitable when only several changes need to be made. However, when making numerous changes to the order of the tree, the Sort Operations command provides an automatic approach.

1. Right click Mill Part Setup1 in the Operation tree and select Sort Operations.

   The Sort Operations dialog box displays.

2. Remove the check mark in the Process complete feature checkbox.

   When checked, the operation sequence will be based on machining each feature completely before machining the next feature. The features will be sorted from highest Z to lowest Z.

3. Change to the Sort tab.

   The Sort page allows operations to be sorted based on either one or two criteria. The Sort by list shows the current order for sorting operations. The default order is defined in the Technology Database and can be modified as required.

4. Click the Apply button to see the result of the current sorting rules.

   The operations in the tree are sorted according to the order in the list.

5. Click the Undo button to return to the original order.

   In this example, you want to sort by operation type with Rough Mill cycles first, Finish Mill cycles second, then hole making cycles. To do this, you can use drag and drop to reorder the operation list.

6. Select Rough Mill in the list and drag it to the top. Note that the operation will be inserted before the operation it is dropped on.

7. Select Finish Mill and drag and drop it on Center Drill.

8. Click the Apply button. The order in the tree changes to reflect the current operation list.

9. Click OK.

   The operation order in the tree can be further modified using drag and drop if necessary.

   **CAUTION!** Be careful when dragging and dropping the operations. Make sure you do not change the order so that the tools will cause a crash. For example, you could drag the ream operation in front of the drill operation. CAMWorks assumes that you know what you are doing and will process the toolpaths exactly the way you have them ordered.

## *Machining Island Tops*

As in the previous exercises, the Operation Parameters dialog allows you access to all the parameters that will be used to machine a Feature. You can change parameters to customize the toolpath that CAMWorks generates to suit your machining requirements.

 *Did You Know ...*
The parameters that you change in the Operation Parameters dialog box are used only for the current operation. Other operations and other parts will revert to the default parameters that are defined in the Technology Database.

1. Right click the Rough Mill1 in the Operation tree and select Edit Definition.

2. Click the Roughing tab in the Operation Parameters dialog box.

3. Check the Machine island top option so that the islands on the top of the part are faced off.

4. Change the Pocketing Pattern to Zigzag.

   This option creates parallel lace cuts with the cuts alternating direction. Between each lace cut, the tool stays down in the material.

5. Click OK.

## *Changing the Mill Part Setup Origin*

CAMWorks creates Mill Part Setups automatically with the Mill Part Setup depth set to the highest face on the recognized features and the Mill Part Setup origin on the edge of the first recognized feature on the top of the part. The Mill Part Setup origin defines the G-code program zero location using the machine tool's fixture location, Work Coordinates and/or Sub Coordinate offsets. The Mill Part Setup origin position is based on the SolidWorks origin. You can move the Mill Part Setup origin to a more useful location for machining.

1. Left mouse click on Mill Part Setup1 in the tree.

   Notice where the origin is located on the part.

2. Right click on Mill Part Setup1 and select Edit Definition.

   The Part Setup Parameters dialog box displays.

3. For the Origin, select the Stock vertex option.

4. Pick the corner shown in the figure, then click OK.

   The origin location is moved on the part to the corner you selected.
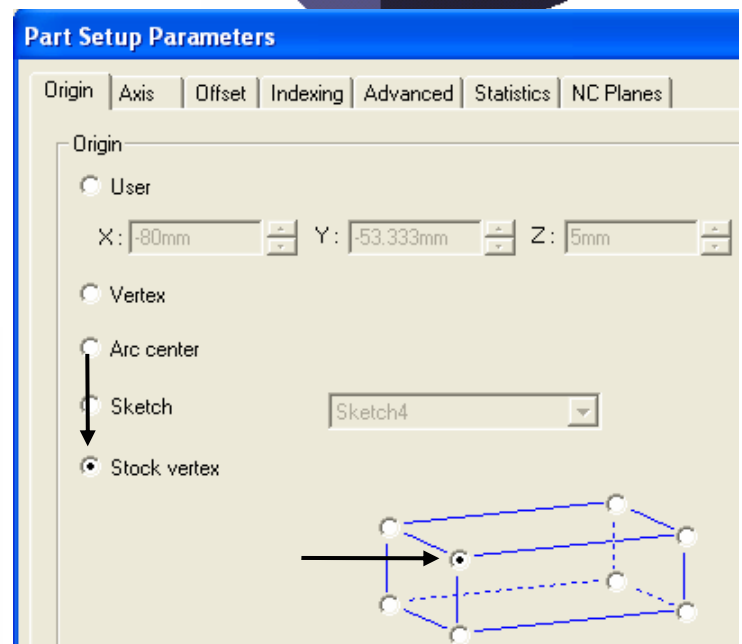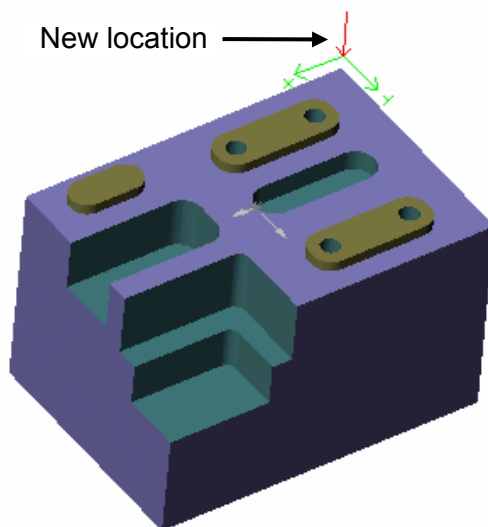
Based on the orientation of the part, selecting a corner in the dialog box may not appear to be the same corner on the part. This is because the model can be rotated to any position in space.

## *Combining Operations*

Notice that several roughing and finishing operations use the same tool. Similar operation types that use the same tool can be combined into one operation.

1. Right click on Mill Part Setup1 in the tree and select Combine Operations.

2. Remove the check mark from the Regenerate toolpath option.

3. Select Finish Mill, then click OK.

   CAMWorks combines similar Finish Mill operations that use the same tool.

4. Click the + next to the Finish Mill operations in the tree to expand the items.

   The operations that use the same size tool were combined. Notice that the order of the operations was maintained. The rough operations are first, followed by the finish operations, and finally the hole operations.

**Before Combining**

NC Manager
- Stock Manager[1005]
- Example Mill-mm
- Mill Part Setup1 [Group2]
  - Rough Mill1[T18 - 18 Flat End]
  - Rough Mill2[T8 - 6 Flat End]
  - Rough Mill3[T8 - 6 Flat End]
  - Rough Mill4[T8 - 6 Flat End]
  - Rough Mill5[T8 - 6 Flat End]
  - Finish Mill1[T12 - 12 Flat End]
  - Finish Mill2[T41 - 12 Flat End]
  - Finish Mill3[T41 - 12 Flat End]
  - Finish Mill4[T5 - 4 Flat End]
  - Finish Mill5[T9 - 8 Flat End]
  - Finish Mill6[T8 - 6 Flat End]
  - Finish Mill7[T5 - 4 Flat End]
  - Center Drill1[T42 - 16 Center Drill]
  - Drill1[T43 - 8.9x118 Drill]
  - Ream1[T44 - 8.9 Ream]

**After Combining**

NC Manager
- Stock Manager[1005]
- Example Mill-mm
- Mill Part Setup1 [Group2]
  - Rough Mill1[T18 - 18 Flat End]
  - Rough Mill2[T8 - 6 Flat End]
  - Rough Mill3[T8 - 6 Flat End]
  - Rough Mill4[T8 - 6 Flat End]
  - Rough Mill5[T8 - 6 Flat End]
  - Finish Mill1[T12 - 12 Flat End]
  - Finish Mill2[T41 - 12 Flat End]
    - Obround Boss1 [Coarse]
    - Rectangular Boss Group2 [Coarse]
  - Finish Mill4[T5 - 4 Flat End]
    - Rectangular Corner Slot1 [Rough Finish]
    - Rectangular Corner Slot2 [Rough Finish]
  - Finish Mill5[T9 - 8 Flat End]
  - Finish Mill6[T8 - 6 Flat End]
  - Center Drill1[T42 - 16 Center Drill]
  - Drill1[T43 - 8.9x118 Drill]
  - Ream1[T44 - 8.9 Ream]

### *Did You Know ...*

After toolpaths have been generated, if you combine operations and disable the option to Regenerate toolpaths option in the Combine Operations dialog box, you must generate toolpaths again.

5. Right click FinishMill4 and select Edit Definition.

6. On the Mill Tool tab, change the Overall length to 100mm and the Protrusion to 60mm, then click OK.

7. Right click Drill1 and select Edit Definition.

8. On the Drill Tool tab, change the Overall length to 150mm and the Protrusion to 130mm, then click OK.

9.    Click the Generate Toolpath button on the CAMWorks toolbar or right click the Mill Part Setup in the tree and select Generate Toolpath.

10. Highlight the first operation in the tree, hold down the Shift key, then highlight the last operation.

    Your model should look similar to the figure on the right.
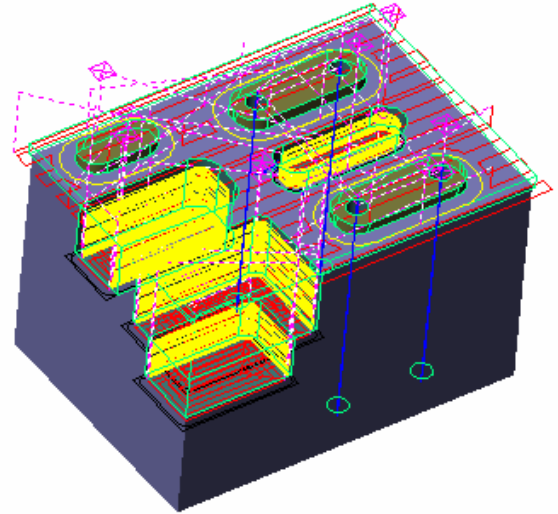
11.    Click on the Simulate Toolpath button on the CAMWorks toolbar.

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

13.    Click the Return to start button.

14.    Click the Turbo mode button, then click Run.

    The simulation does not update the stock until the simulation is completed. This is the fastest method for simulation.

15.    Click the X button in the upper right corner of the Simulate Toolpath toolbar to exit simulation mode.

16. Right click on the Mill Part Setup and select Post Process.

    The Post Output File dialog box displays so you can name the NC program file.

17. Use either the default file name or type a different name and save the file.

18. Click the Run button in the Post Process Output dialog box to output the G-code file.

19. Click OK to close the dialog box.
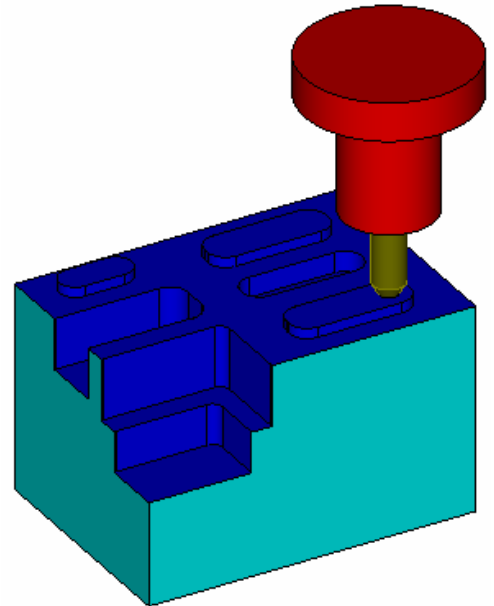
# 2 Axis Mill 6

## What You'll Learn

- Interactively Inserting 2.5 Axis Features
- Avoiding Clamps
- Changing Operation Parameters and Tools
- Inserting a Mill Part Setup and Face Feature to Cut the Bottom of the Part
- Customizing Toolpaths

This exercise involves a non-rectangular part that requires several interactive features.

1. Open the part file **MILL2AX_6.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

   Since this part is non-rectangular, a feature to machine the outside of the part has to be inserted. In addition, a feature will be added to cut the top of the pockets. All machining will be from the top of the part.

2. ![CW icon] Click the CAMWorks Feature Tree tab.

3. ![icon] Right click Example Mill–mm in the Feature tree and select Edit Definition.
   - On the Machine tab, Example Mill–mm is the Active machine.
   - Click the Tool Crib tab and make sure Tool Crib 1 (metric) is the Active tool set.
   - Click the Controller and make sure FANTUTM is selected, then click OK.
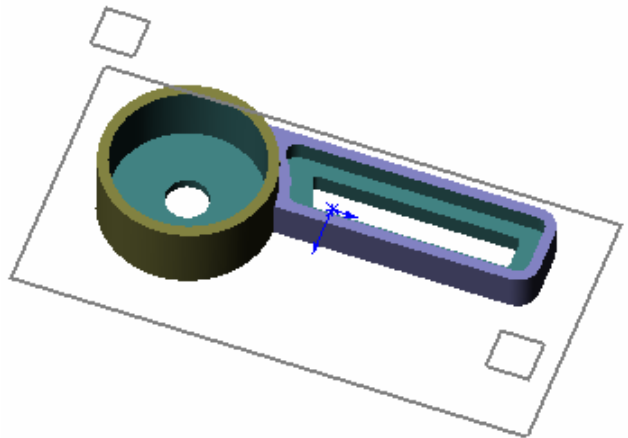
4. ![icon] Right click on the Stock Manager and select Edit Definition.

5. ![icon] Click the Extruded Sketch button and pick the sketch representing the stock

6. ![icon] Set the Depth to **45mm**.

7. Click OK to close the dialog box.

8. ![icon] Click the Options button on the CAMWorks toolbar.

9. On the Mill Features tab, check the Face option in the Feature types section and click OK.
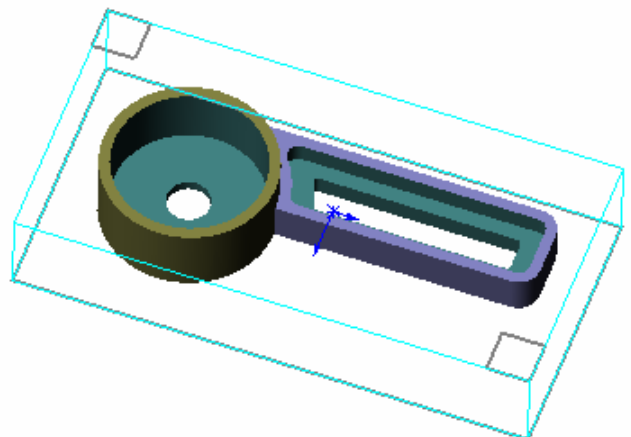
10.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click the NC Manager in the tree and select Extract Machinable Features.

Automatic Feature Recognition analyzes the model for features to machine and the tree displays the list of Machinable Features that CAMWorks has found. The face, large hole and the two internal pockets have been found. Additional features will be defined interactively to completely machine the part.
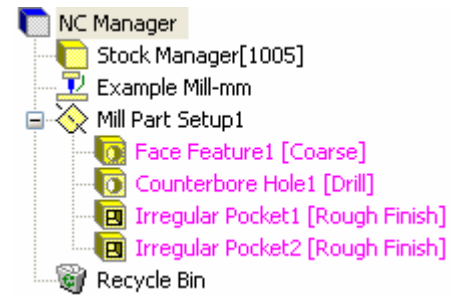
## Inserting 2.5 Axis Features

You need to insert 2.5 Axis Features to cut the face on the top of the pockets and the outside of the part.

Insert the first 2.5 Axis Feature to machine the top of the pockets.

1. Right click on Face Feature1 and select Insert 2.5 Axis Feature. The 2.5 Axis Feature Wizard starts and the Feature & Cross Section Definition dialog box displays.

2. Select Open Pocket for the Feature Type.

3. Pick the top face of the part at the top of the pockets.

   The Entities selected list shows Face<1>.

4. Click Next.

   The 2.5 Axis Feature Wizard: End Conditions dialog box displays.

5. Make sure Use Stock extents is checked so that the entire top of the material is faced off.

6. Click Finish to complete the feature.

   The Feature & Cross Section Definition dialog box displays again so you can continue inserting features.
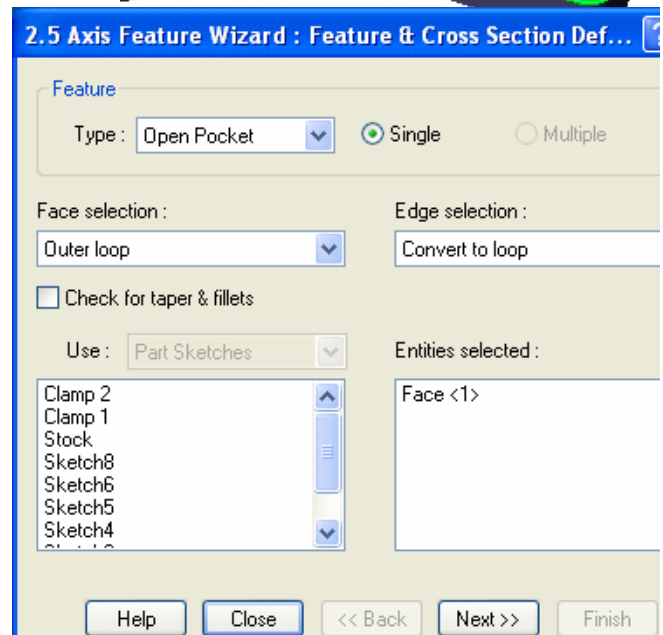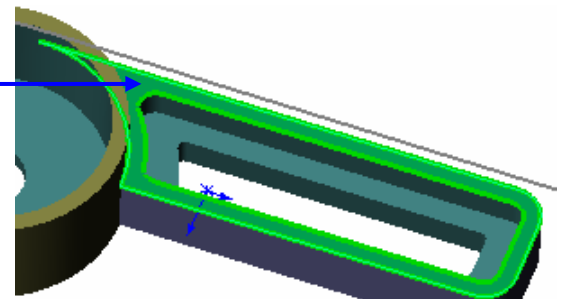
The second 2.5 Axis Feature will be the area around the outside of the part. The stock was defined so that the bottom of the part is 5mm off the bottom of the stock. If the feature is defined as having a bottom depth equal to the bottom of the part, then a 5mm skin will remain to hold the part to the clamp.

1. Rotate the part so that you can see the bottom face.

2. Make sure the Feature Type is set to Open Pocket, then pick the bottom face as the entity to be machined.

3. Click Next.

4. Rotate the part back to view the top.

5. Pick the top face of the part at the top of the hole.

   In the End Conditions dialog box, the End condition Type changes to Upto Face and the Depth is calculated automatically.

6. Make sure that the Use stock extents option is checked, then click Finish.

7. Click Close to exit the Feature Wizard.
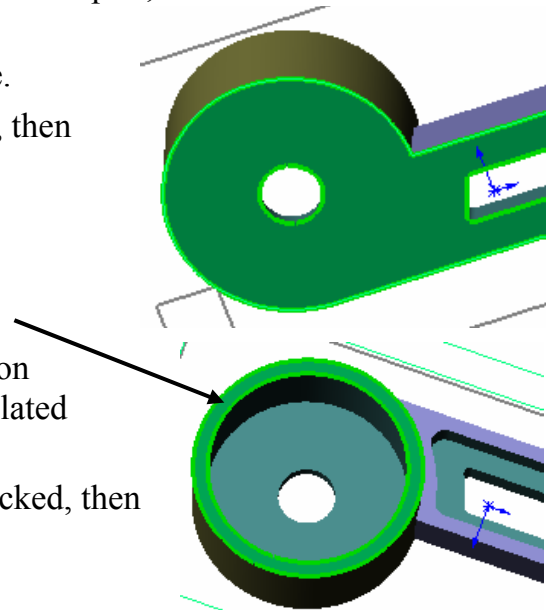
   The features are listed in the CAMWorks Feature tree in the order you added them.

8. Drag and drop the features so that they are in the order shown on the right.
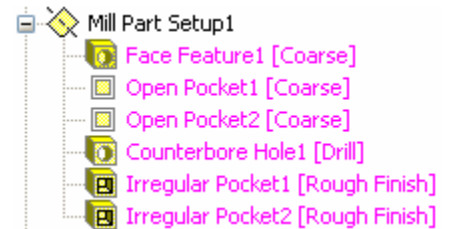
9.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click on Mill Part Setup1 and select Generate Operation Plan.

   The Operations list should look similar to the example on the right.
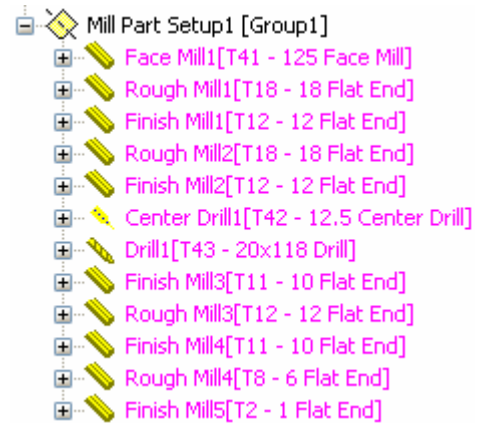
## *Defining Avoid Areas for Clamps*

Rough, Finish and Face Mill operations for 2 and 2.5 Axis features can be modified by specifying Avoid Areas. Avoid areas can be used as a definition for clamps and large holes in the feature where machining is to be avoided. Avoid areas can be defined from planar or non-planar model edges and faces, as well as from sketches. Avoid Areas are considered to be islands within the perimeter and will be machined around leaving material equal to the sum of the Side allowance and Avoid allowance parameters.

In this exercise, you define avoid areas for the Face Feature and the Open Pockets to avoid machining the clamps.

1. Right click FaceMill1 in the Operation tree and select Insert Avoid Area on the shortcut menu.

   The 2.5 Axis Wizard dialog box displays.

2. Select Clamp 1 in the sketches list and click Finish.

3. Click the plus sign next to FaceMill1 in the Operation tree.

   The Avoid Area is listed under the operation.
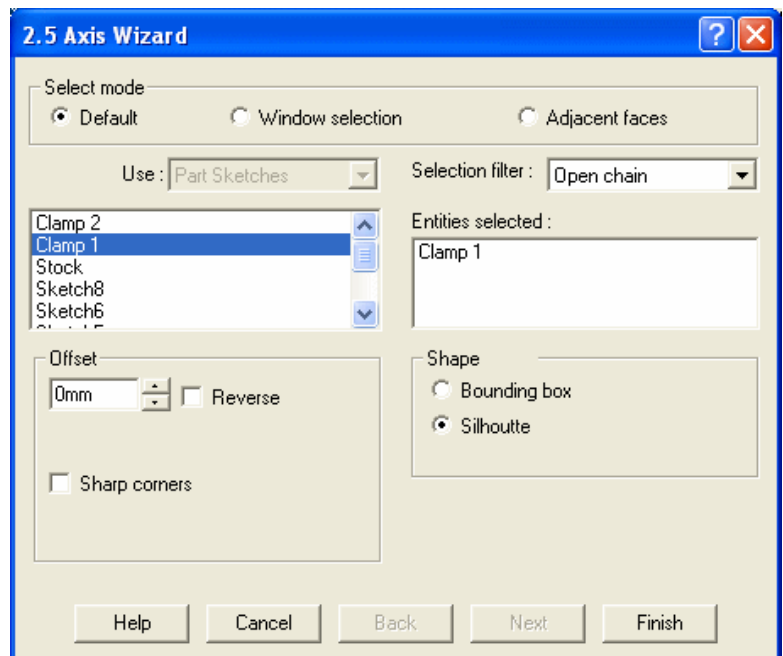


4. Right click Face Mill1 and select Insert Avoid Area on the shortcut menu.

5. In the 2.5 Axis Wizard dialog box, pick Clamp 2 and click Finish.

6. Repeat these steps to define avoid areas for Rough Mill1 and RoughMill2.

7. Drag and drop the Finish Mill operations for the Open Pocket feature to the positions shown on the right.
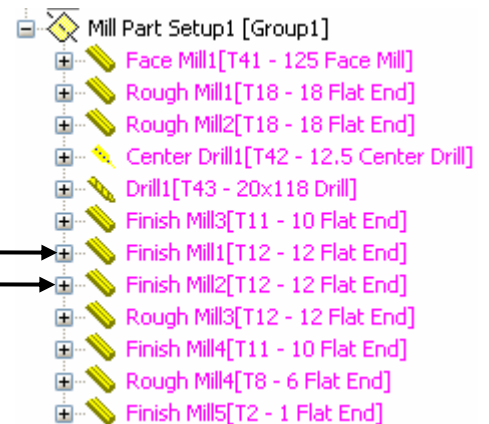
## *Changing Operation Parameters and Generating Toolpaths*

Before generating toolpaths, we will modify some of the operation parameters. Typically, choosing parameters for machining is based on personal preference.

1.  Right click Rough Mill1 in the tree and select Edit Definition.
2.  On the Roughing tab, make sure the Cleanup pass option is *not* checked, then click OK.
3.  Repeat steps 1 and 2 for Rough Mill2.

    If the general rule is to not generate a cleanup pass, the default could be set to off in the Technology Database.

4.  Right click Finish Mill1 and select Edit Definition.
5.  Click the Finish tab in the Operation Parameters dialog box.

    By default, the depths of cut will produce multiple finish cuts.

6.  Change the value for First cut amt. to **50mm**.
7.  For Depth processing, select the To Depth by region option, then click OK.
8.  Repeat steps 4 - 6 for Finish Mill2.

The default values for milling the counterbore are small and will produce more cuts than are needed. You can edit the operation and change these values.

9.  Right click Finish Mill3 and select Edit Definition.
10. Click the Finish tab and change the following parameters:

    Side Parameters: Cut amt. = **10mm**.
    Depth Parameters: First cut amt. = **10mm** and Max cut amt. = **10mm**

11. Click the Tool tab, then click the Tool Crib tab.

12. In the Tool Crib, scroll through the tools to find a 20mm flat end mill.

    If Flat End tools are not listed, clear the check mark to the left of the Filter button.

13. Click in a cell on the 20mm tool line to select the tool, then click the <- Select button.

14. Click Yes to replace the corresponding holder.

15. Click OK to close the Operation Parameters dialog box.
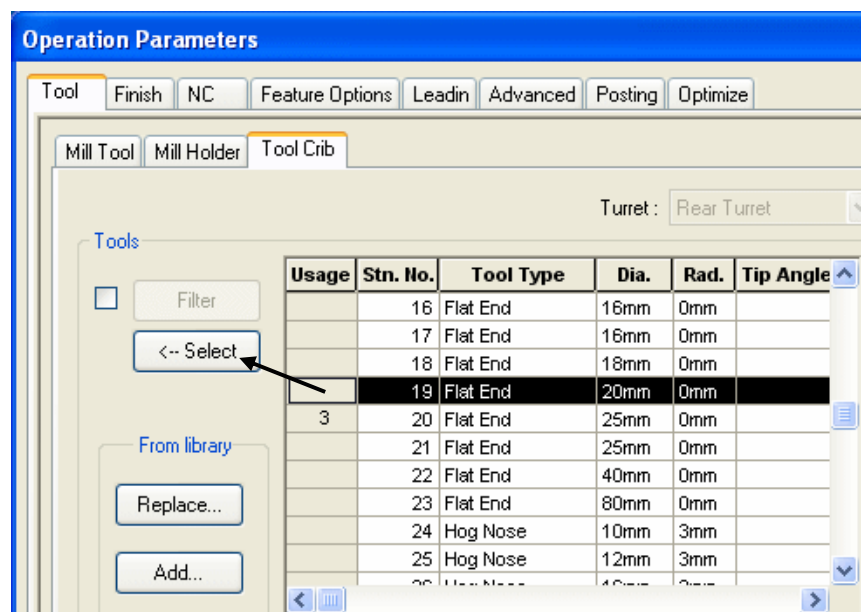
16. Right click Finish Mill5 in the tree and select Edit Definition.

17. On the Tool Crib page, click in a cell on the 2mm Flat End tool line and click Select.

18. Click Yes to replace the holder

19. ⬛ Click the Generate Toolpath button on the CAMWorks toolbar or right click the Mill Part Setup and select Generate Toolpath.

    CAMWorks generates the toolpaths for all operations.

20. Hold down the Shift key and select the first and last operations in the tree.

    The result should look like the example. Notice how the toolpath has avoided the clamp areas.

21. Change to the Top view and notice that the toolpath has not gone to the bottom of the material.

22. Right click on the Mill Part Setup and select Simulate Toolpath.

23. ▶ Run the tool path simulation.

    The result should be similar to the example.

24. ❌ Close the Simulation to return to the model.
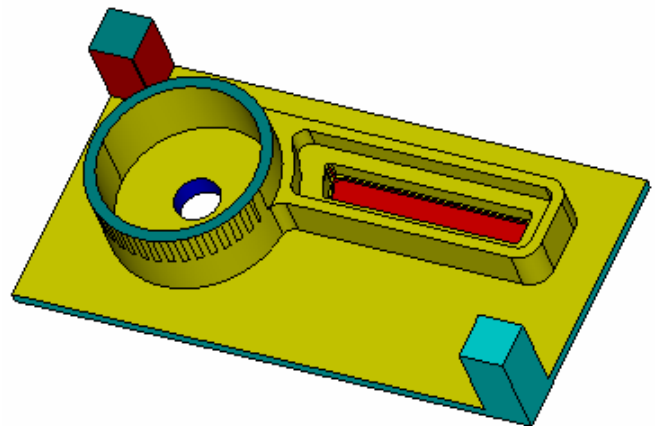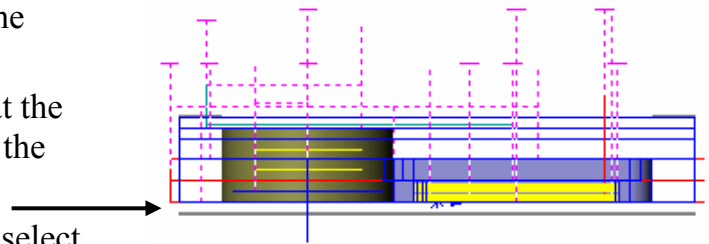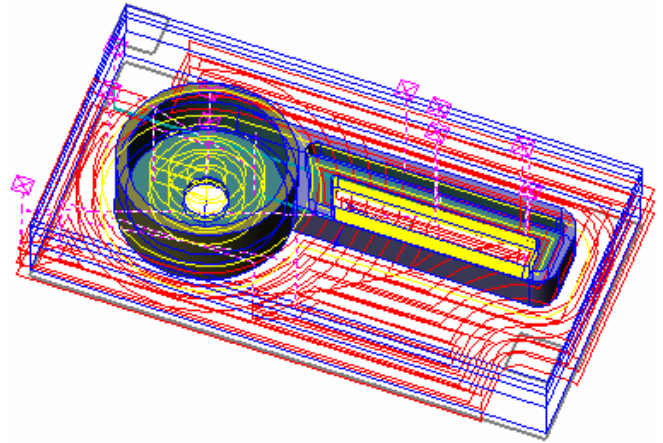
## *Inserting a Mill Part Setup and Adding a Facing Cut*

This series of steps shows you how to add a facing cut for the bottom side of the part. For setup purposes, you are going to flip the part over, hold it by the machined rectangular section, and machine the remainder of the material off with a facing cut.

You begin by creating a Mill Part Setup that is opposite from the existing Mill Part Setup. From this Mill Part Setup, you will then create a face feature.

1. Rotate the part so that you can see the bottom face.

2. Click the CAMWorks Feature Tree tab.

3. Right click on Mill Part Setup1 and select Insert Mill Part Setup.

4. Pick the bottom face of the part in the graphics area.

   CW Face5 is shown as the Selected entity. When you insert a Mill Part Setup, you can select either a face or one of the planes listed in the Mill Part Setup dialog box.

5. Make sure the Face option is checked.

   If this option is checked, CAMWorks automatically creates a Face feature when the Mill Part Setup is created if the topmost face is parallel to the Mill Part Setup and the Mill Part Setup is parallel to one of the sides of the stock.
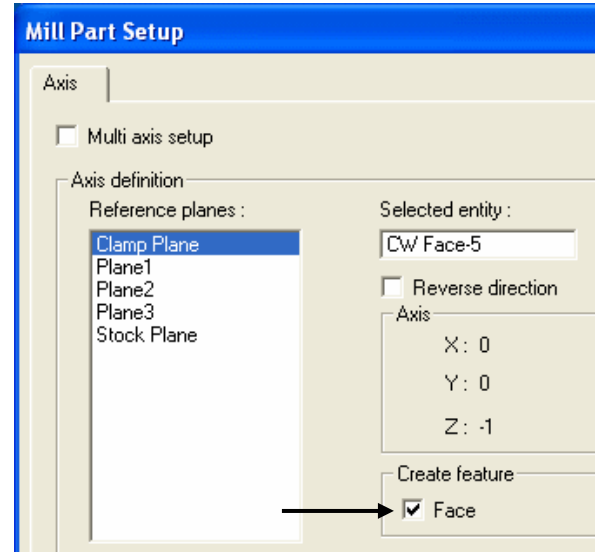
6. Click OK.

   A new Mill Part Setup is inserted on the bottom of the part. Mill Part Setup2 and Face Feature2 are listed in the tree.

7. Right click Mill Part Setup2 and select Generate Operation Plan.
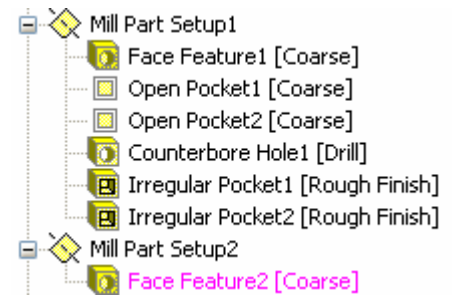
8. In the Operation tree, right click Mill Part Setup2 and select Generate Toolpath.

   The resulting tool path should look similar to the figure on the right.

9. Rotate the part back to view the top.

**Mill Part Setup**

Axis

☐ Multi axis setup

Axis definition

Reference planes :
Clamp Plane
Plane1
Plane2
Plane3
Stock Plane

Selected entity :
CW Face-5

☐ Reverse direction

Axis
X : 0
Y : 0
Z : -1

Create feature
☑ Face

Mill Part Setup1
    Face Feature1 [Coarse]
    Open Pocket1 [Coarse]
    Open Pocket2 [Coarse]
    Counterbore Hole1 [Drill]
    Irregular Pocket1 [Rough Finish]
    Irregular Pocket2 [Rough Finish]
Mill Part Setup2
    Face Feature2 [Coarse]

10. Right click on the NC Manager in the CAMWorks Operation tree and select Simulate Toolpath.

11. ▶ Click the Run button.

The resulting simulation should appear as shown. The remainder of the material has been removed, and the part has been machined completely.

12. ✖ Click the exit button.
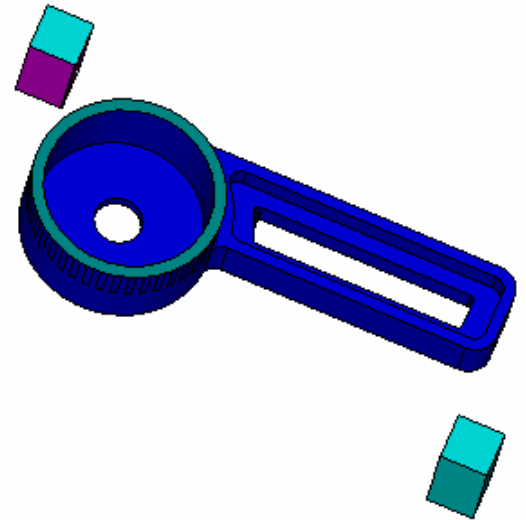
## *Customizing Toolpaths*

In the final part of this exercise, you change some of the Mill Part Setup and operation parameters to customize the toolpaths.

1. Change the orientation to Back view and zoom up on the toolpath.
2. Right click on Finish Mill3 in the tree and select Edit Definition.
3. On the Tool Crib tab, highlight a 10mm Flat End and click the Select button.
4. Click Yes to replace the holder.
5. Click the Leadin tab and change the Leadin Type to Perpendicular.
6. Set the Leadin Amt. to **5mm**.
7. Leave the Leadin/out point set to Mid-point and click OK.

   The pierce point displays as a square marker in the middle of the edge because the Mid-point option is selected.
8. Right click on the same Finish Mill operation and select Generate Toolpath.
9. Rotate the part and notice the leadin line at the start of the Finish Mill tool path. The angle is perpendicular to the start angle at the beginning of the arc.
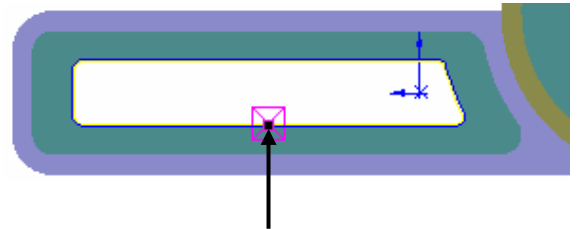
   You can see the SolidWorks origin. Next, you will use that as the start of the leadin.
10. Right click on the Finish Mill3 operation and select Edit Definition again.
11. Click the Leadin tab and set the Leadin Type to None.
12. Click the Feature Options tab.
13. For the Entry Type, use the default None.
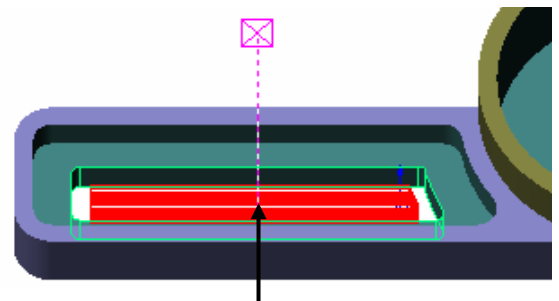
   With this option, the end mill feeds down in Z at the specified entry point.

14. Rotate the part, if necessary, and pick the SolidWorks origin with your mouse.

    The entity displays in the Entities selected list.

15. Click OK.

16. Right click on the same Finish Mill operation and select Generate Toolpath.

    The start of the leadin changes to the origin point that was selected. This acts as a leadin even though the Leadin Type was set to None.
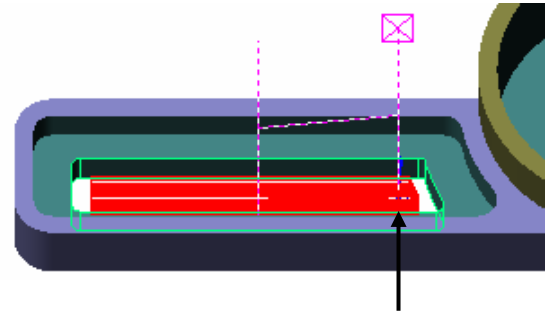
17. Right click Mill Part Setup1 in the tree and select Post Process.

18. In the Post Output File dialog box, use either the default file name or type a different name and save the file.

19. Click the Run button in the Post Process Output dialog box to output the G-code file.

20. After reviewing the code, click OK to close the dialog box.

# 2 Axis Mill 7

## What You'll Learn

- Interactively Inserting 2.5 Axis Features
- Inserting Engrave Features
- Generating an Operation Plan and Adjusting Operation Parameters
- Updating Toolpaths for Design Changes

1. Open the part file **MILL2AX_7.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. [CW] Click the CAMWorks Feature Tree tab.

3. Use the default Example Mill–in, controller and active tool crib (as defined in the Machine dialog box).

4. Right click on the Stock Manager and select Edit Definition.

   The default bounding box displays on the part.

5. Click OK to use this stock.

6. Click the Extract Machinable Features button on the CAMWorks toolbar.

   AFR extracts features to machine and generates the Mill Part Setups needed to machine these features.

### *Defining Features Interactively*

In the following exercises, you edit the Open Pocket feature to machine the top of the flange and insert features to machine the perimeter of the part and the four corner pockets.

Edit the Open Pocket feature to machine only the top of the flange.

1. Right click Open Pocket1 in the Feature tree and select Edit Definition on the shortcut menu.

2. In the 2.5 Axis Feature Wizard: End Conditions dialog box, remove the check mark from the Use Stock extents option.

   By default, CAMWorks uses the extents of the stock volume to define open pocket features. In this feature, you want to machine only the top of the flange.

3. Click Finish.
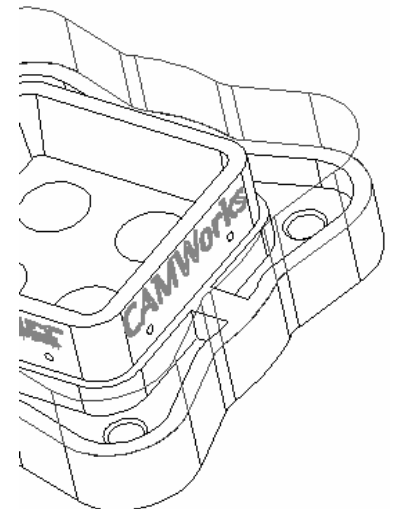
Insert a Boss feature to machine the perimeter of the part.

1. Right click Open Pocket1 and select Insert 2.5 Axis Feature.

2. Change the Feature Type to Boss.

3. Make sure the Edge selection is set to Convert to loop.

   With this option selected, when you pick a part edge, CAMWorks automatically chain selects all other edges connected to that edge at the depth. This allows you to pick one edge on the inside of a face and identify the entire inside loop.
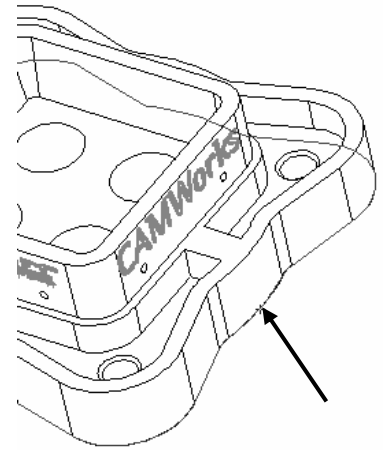
4. Pick the bottom edge of the part.

   The entire face of the part is highlighted and loop<1> is added to the Entities selected list.

5. Click Next.

6. Pick the top of the flange for the end condition.

   CAMWorks automatically changes the End condition Type to Upto Face and calculates the Depth.

7. Click Finish.
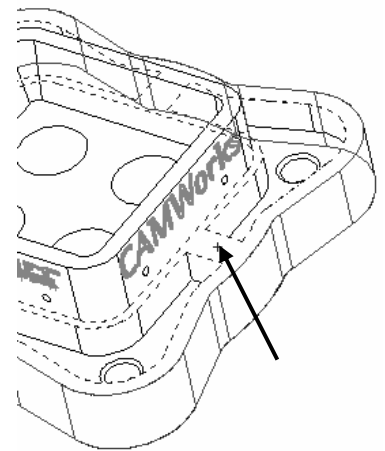
Insert Pocket features to machine the bottom faces of the four corner pockets.

1. Change the Feature Type to Pocket.

2. Make sure the Multiple option is selected.

   This option allows you to define multiple similar features by selecting multiple faces, sketches and edges.

3. Pick the faces on the bottom of the 4 corner pockets and click Next.

4. Pick the face on the top of the flange for the end condition.

5. Click Finish, then click Close to exit the wizard.

   The features are listed in the tree in the order you inserted them. This is the order they will be machined when you generate an operation plan.

6. Use drag and drop to reorder the features in Mill Part Setup2 so that Open Pocket1 is first and Irregular Boss1 is last.
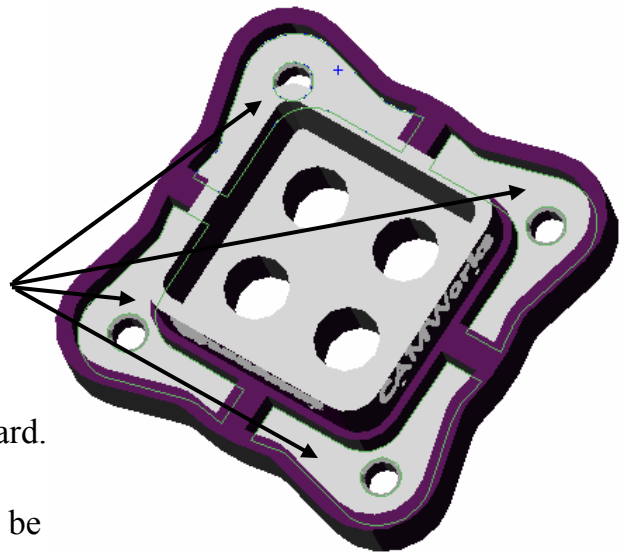
## *Inserting Engrave Features*

Add two Engrave features to machine the text on the part.

1. Right click Mill Part Setup5 (or the Mill Part Setup on the same face as the TekSoft logo) in the tree and select Insert 2.5 Axis Feature.

2. In the 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box, select Engrave Feature for the Feature Type.

3. In the Available Sketches list, select Teksoft logo.

4. When Teksoft logo displays in the Entities selected list, click Next.

5. In the 2.5 Axis Feature Wizard: End Conditions dialog box, change the Depth to .015, then click Finish.

6. Click Close to exit the wizard.

   Engrave Feature1 is listed in the tree.

7. Right click Mill Part Setup6 (or the Mill Part Setup on the same face as the CAMWorks logo) and select Insert 2.5 Axis Feature.

8. In the 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box, select Engrave Feature for the Feature Type.

9. In the Available Sketches list, select CWlogo.

10. When CWlogo displays in the Entities selected list, click Next.

11. In the 2.5 Axis Feature Wizard: End Conditions dialog box, change the Depth to .015, then click Finish.

12. Click Close to exit the wizard.

    You have defined all the features and are now ready to generate an operation plan.

## *Generating an Operation Plan and Adjusting Operation Parameters*

1. Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the tree and select Generation Operation Plan.

   The tree lists the operations that CAMWorks generated for the features in all the Mill Part Setups.

Adjust tools before generating the toolpaths:

2. Right click Finish Mill2 (the operation for Irregular Pocket1) in the tree and select Combine Operations.

   This command allows you to combine and remove duplicate operations.

3. In the Combine Operations dialog box, select the Combine operations option.

   When this option is checked, the features machined by operations in the Features or operations list are machined by the current operation and the original individual operations for those features are deleted.

4. Make sure the Operations list displays in the left list box, then hold down the Shift key and select the three operations in the list box.

5. Click the Add button.

   The operations are added to the Features or operations list on the right.

6. Click OK.

   Finish Mill3, Finish Mill4 and Finish Mill5 are removed from the tree and the Irregular Pocket features are listed under Finish Mill2.

7. Right click Finish Mill2 in the tree and select Edit Definition.

   By combining operations for the Irregular Pockets, you have to change the parameters only once instead of for each operation.

8. On the Tool tab, make sure the current tool is a .38in flat end mill.

   If a different tool is selected, click the Turret tab and select a .38in flat end mill, click the <- Replace button, then click Yes to replace the corresponding holder.

9. Click OK to close the Operation Parameters dialog box.

10. Look at each Center Drill operation in the tree and check the size of the tool selected by the TechDB.

    The tool for each Center Drill operation should be a #3 60 degree center drill with a .11in diameter. If a different tool is listed in the tree, right click the operation and select Edit Definition. Then, click the Turret tab and select the correct tool in the Tool Crib.

11. Right click on Finish Mill8 in the tree (the Finish Mill operation for the Irregular Boss) and select Edit Definition.

12. On the Mill Tool tab, make sure the current tool is a .5in flat end mill.

    If it is not, click the Turret tab, select a .5in flat end mill in the Tool Crib, click the <- Select button, then click Yes to replace the corresponding holder.

13. Click OK.

14.  Click the Generate Toolpath button on the CAMWorks toolbar or right click NC Manager in the tree and select Generate Toolpath.

    CAMWorks generates toolpaths based on the information in the Technology Database.

## Associating Machining Information After Design Changes

One of the unique features of CAMWorks is the ability to re-apply, or associate, machining information to a part after the size or shape of the part has been modified. This associative action is done through the CAMWorks Rebuild process. Maintaining this associativity is a complex process and may require user interaction to help reconcile unresolved associations.

In this exercise, you change the size of the four 1.00in diameter holes in the center of the part.

1. Click the FeatureManager design tree tab.

2. Right click on one of the 1.00in diameter holes and select Edit Sketch.

3. Change the 1.00in dimension to 1.25in.

4. 🚦 Rebuild the model to accept the change.

5. 📷 Click the CAMWorks Feature Tree tab.

   CAMWorks displays the message: Part has changed. Select Full to rebuild all CAMWorks data or light if changes do not affect CAMWorks data.
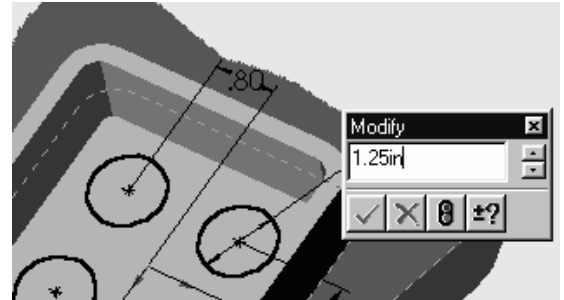
6. Click Full.

   If the machinable features from the original part shape cannot be associated automatically to machinable features created after a model change, both the current and new features display in the Associate Features dialog box. You can interactively resolve which new features are associated to current features.

7. Click Hole Group1 in the Current features list.

   When a Current feature is selected, operations used to machine the feature are listed under the Operation List for Current Feature. For visual confirmation, when a Current feature is selected in the list, a red outline of the feature displays on the part.

8. Click on Hole Group8 in the New features list.

   The Associate button is enabled.

9. Click the Associate button.

   When you click the button, CAMWorks replaces the selected Current feature with the selected New feature in every operation that contains the Current feature. The New feature is removed from the list and the Current feature icon changes from a red exclamation mark to a green resolved marker.
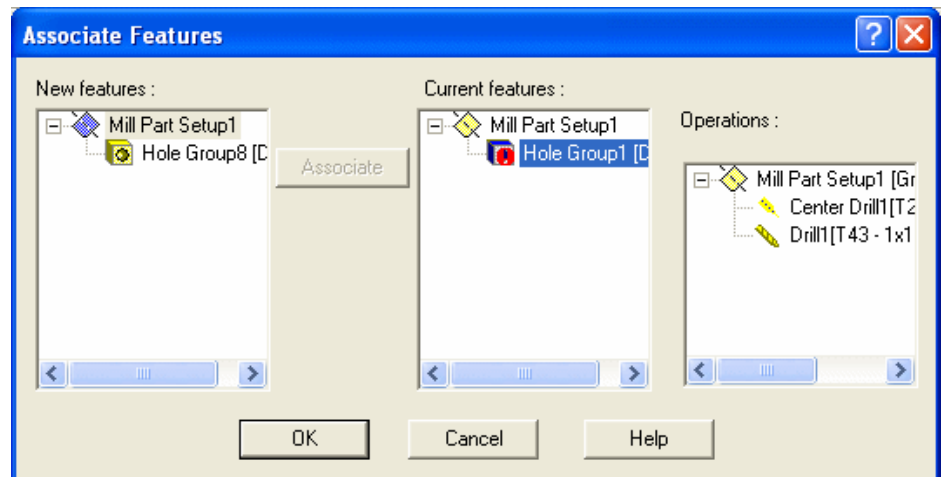
10. Click OK.

    When the dialog box is closed, all Current features that are resolved are removed from the CAMWorks Feature tree.

11. In the Feature tree, right click Hole Group1 and select Generate Operation Plan.

12. Click the Regenerate button to generate new operations for the feature.

    Based on information in the TechDB for the 1.25in diameter hole, a Finish Mill operation is applied to the feature in addition to a Drill operation.

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

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

15.  On the Simulate Toolbar, make sure the Tool Mode button is selected, then set the following display options:

    – Tool and Holder Display = Shaded, Translucent, or Wireframe

    –  Target Display = No Display

16.  Click the Run button to start the simulation.

17.  Click the Goto Start button.

18.  Click the Show Holder collision button to activate this function.

    When you select this button, if the holder interferes with the stock, the simulation shows material being removed by the holder.

19.  Run the simulation again and notice that the drill holders gouge the part. You can select a longer drill tool to eliminate the gouges.

**Did You Know ...**
You can modify the simulation color per operation. Select an operation or multiple operations in the Operation tree, right click and select Simulation Color on the shortcut menu. In the Simulation Color dialog box, click Edit and choose a different color.

# 2 Axis Mill 8

## What You'll Learn

- Extracting Machinable Features and Generating Operations
- Specifying a Tool Overlap on Open Air Segments
- Using an Avoid Area
- Defining Rapid and Clearance Planes

1. Open the part file **MILL2AX_8.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ▣ Click the CAMWorks Feature Tree tab.

3. 🔁 Use the default Example Mill–in.

### Extracting  Machinable Features and Generating Operations

1. 🔲 Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

   Automatic Feature Recognition creates the Mill Part Setups necessary to machine all of the machinable features that are recognized. In order to machine the open profile on the right side of the part with a keyway cutter, you would interactively insert the features as explained in exercise TechDB9 in Chapter 8.

2. Right click Mill Part Setup1 in the tree and select Generate Operation Plan.

   Note that if you click the Generate Operation Plan button on the CAMWorks toolbar, operations will be generated for all Mill Part Setups. For this exercise, you only need the operations in Mill Part Setup1. The operations are listed in the CAMWorks Operation tree.

### Specifying a Tool Overlap on Open Air Segments

When machining an Open Pocket, Slot, Corner Slot or Face Feature, the Feature Options tab in the Operation Parameters dialog box allows you to specify how much to overlap the air segment edges for 2 Axis rough and finish operations. The amount is specified as a percent of the tool diameter up to 150%.

1. Left click Rough Mill1 in the tree.

   This should be the operation for the Rectangular Slot.

2. Change the orientation to Front and zoom up on the toolpath to see the open slot toolpath. The toolpath was generated with the overlap set at 75% of the tool diameter.

3. Right click Rough Mill1 and select Edit Definition.
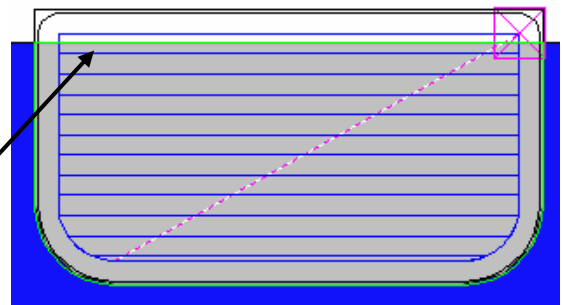
4. Click the Feature Options tab.

5. In the Air segment offset, change the % Tool diameter to 100 and click OK.

6. Generate the toolpath.

   Notice the overlap on the open side of the slot.

7. Zoom to fit the complete part in the graphics area.
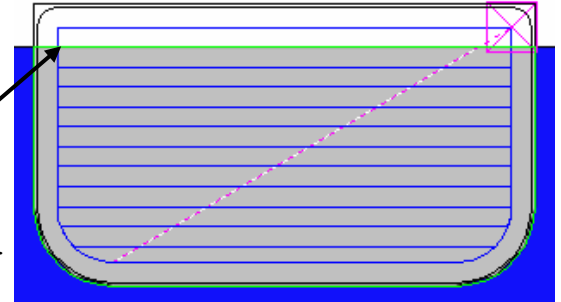
## Using an Avoid Area

After rough and finish toolpaths have been generated, you can add avoid areas. Avoid areas can be used as a definition for clamps and large holes in the feature where machining is to be avoided.

1. Left click Rough Mill2 or the applicable operation for the large pocket and notice the toolpath.

2. Right click Rough Mill2 and select Insert Avoid Area.
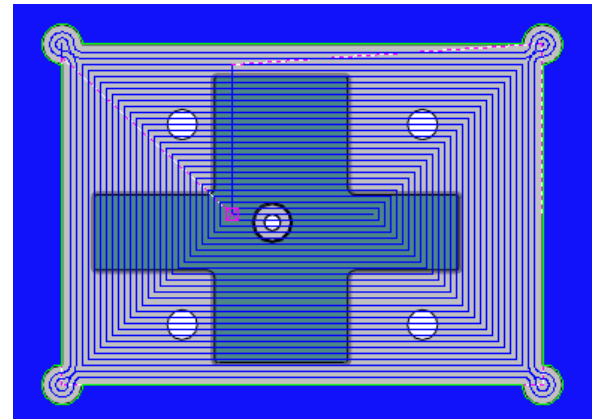
3. In the 2.5 Axis Wizard dialog box, select Avoid Area in the Available sketches list. This sketch represents the perimeter of the drafted pocket.

   Optionally, the Offset option allows you to offset the avoid area.

4. Click Finish.

5. Click the plus sign (+) next to Rough Mill2.

   The avoid area is identified as Avoid Area1. If you wanted to delete an avoid area, you would highlight it in the tree and press the Delete key on the keyboard.
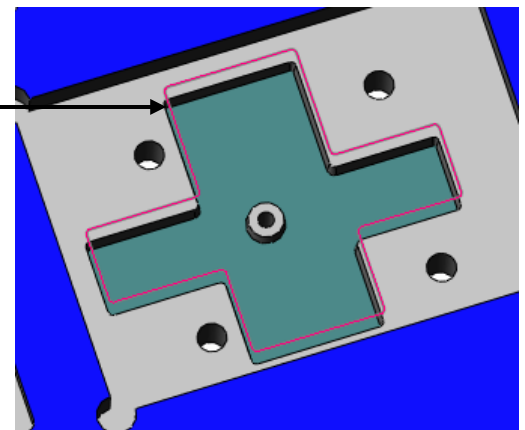
6. Right click Rough Mill2 and select Generate toolpath.

   Notice that the toolpath avoids the selected sketch.
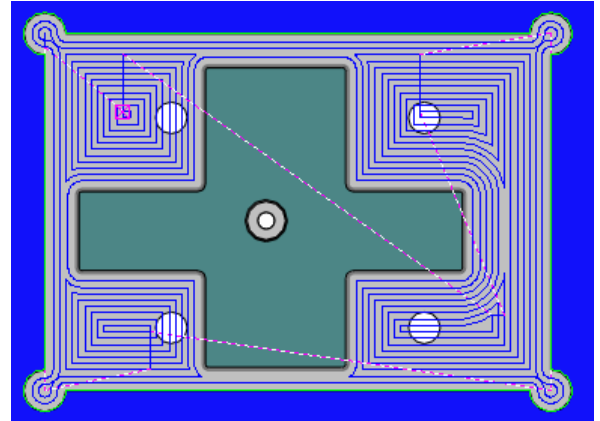


## Defining Rapid and Clearance Planes

For 2 Axis mill operations, the definition of Rapid and Clearance planes can be done locally within each operation or globally from the Mill Part Setup level.

1. Right click Rough Mill1 in the tree and select Edit Definition.

2. Click the NC tab and notice that the Use Setup definition options are not checked.

   The operations generated for this part all use the default settings that were defined for this operation in the Technology Database. When the Use Stock definition option is not checked, you can select a different method to define the Rapid and Clearance plane Z depth.

3. In the Rapid plane section, check the Use Setup definition option.

   The method and depth change to the current setting for the Setup and the options are grayed out.

4. Click OK.

5. Right click Finish Mill1 in the tree and select Edit Definition.

6. On the NC tab, check the Use Part Setup definition in the Rapid plane section, then click OK.

7. Right click Mill Part Setup1 in the tree and select Edit Definition.

8. Click the NC Planes tab.

   The options on this tab provide methods for identifying the Rapid and Clearance plane locations for all operations generated for 2 and 2.5 Axis features in a Mill Part Setup.

9. Click the arrow in the Rapid plane is above section and select Top of Stock in the list.

10. Set the Distance to 1.00in and click OK.

    This value is the distance above the specified location that defines the Z value of the Rapid plane.

    The message displays: The Rapid or Clearance Plane value for an operation is higher than the value for the Mill Part Setup. Change the Mill Part Setup definition to the highest operation value?

11. Click Yes.

A Warning message displays indicating that the toolpaths need to be regenerated because the origin or machining direction has changed.

12. Click Yes.

The toolpaths are regenerated based on the changed setting for the Rapid plane.

# 2 Axis Mill 9

## What You'll Learn

- Defining Features Interactively
- Inserting Linear Pattern Features
- Inserting Sketch Driven Pattern Features

1. Open the part file **MILL2AX_15.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Use the default Example Mill–mm.

### *Defining Features Interactively*

1. Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

   AFR found an Irregular Corner Slot and a Hole feature. The Pocket features need to be defined interactively. You can simplify the creation of multiple features by defining one feature and then inserting a Pattern feature to array it to other positions. The toolpath generated for Pattern features is generated only once on the original feature and copied to each position, which reduces the calculation time.

2. Right click Hole1 in the tree and select Insert 2.5 Axis Feature.

3. Set the Feature Type to Pocket.

4. Pick Sketch2 in the list. The sketch is highlighted on the part.

5. Click Next.

6. Pick Upto Stock for the end condition. CAMWorks automatically calculates the Depth.

7. Click Finish, then click Close.

8. Right click Irregular Pocket1 in the tree and select Generate Operation Plan.

   The tree lists the operations that CAMWorks generated for this feature.

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

   When you insert the Pattern feature, the toolpaths generated for this feature will be automatically copied to each position. You can generate the operations and toolpaths either before or after you insert a Pattern feature.
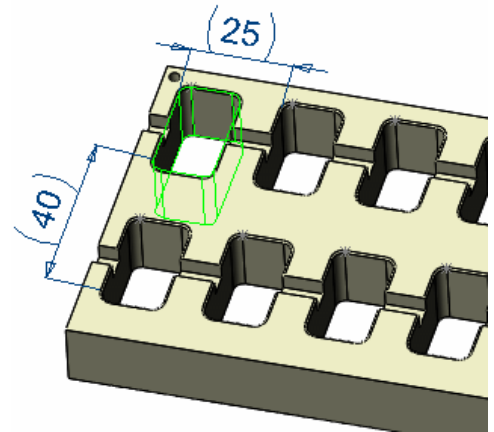
## *Creating a Pattern Feature*

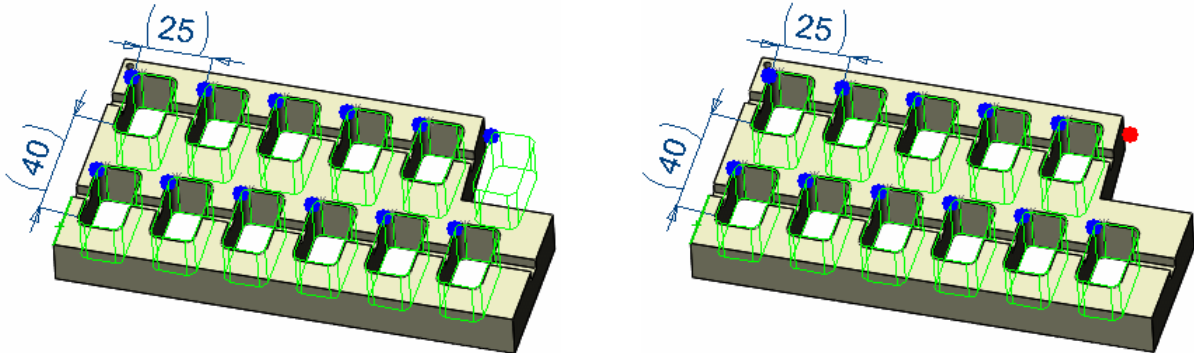In this exercise, you insert the Pattern feature using the Linear and Sketch Driven Patterns.

1.  Click the CAMWorks Feature Tree tab.

2.  Right click Irregular Pocket1 in the tree and select Pattern, then select Insert Pattern.

3.  In the Insert Pattern dialog box, select Linear Pattern for the Type.

4.  Pick one of the horizontal edges of the part for the X direction (see figure).

5.  Click the Reverse Direction button, if necessary, to point the arrow in the correct direction (see figure).

6.  Type **25** for the incremental distance between features instances.

7.  Type **6** for the Number of instances. The feature instances in the X direction display on the part as shown below.



Edge for X Direction          Direction Arrow
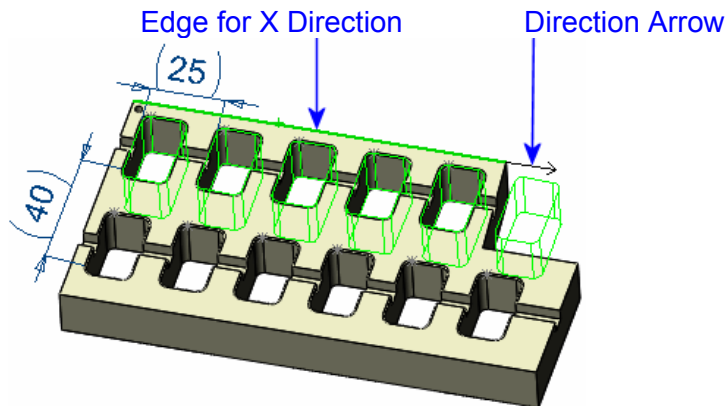
8.  In the Direction 2 section, click in the direction box to set the focus, then pick one of the vertical edges of the part for the Y direction.

9.  Click the Reverse Direction button, if necessary, to point the arrow in the correct direction.

10. Type **40** for the incremental distance between features instances.

11. Type **2** for the Number of instances. The feature instances in the Y direction display on the part as shown below.



Edge for Y Direction

Direction Arrow
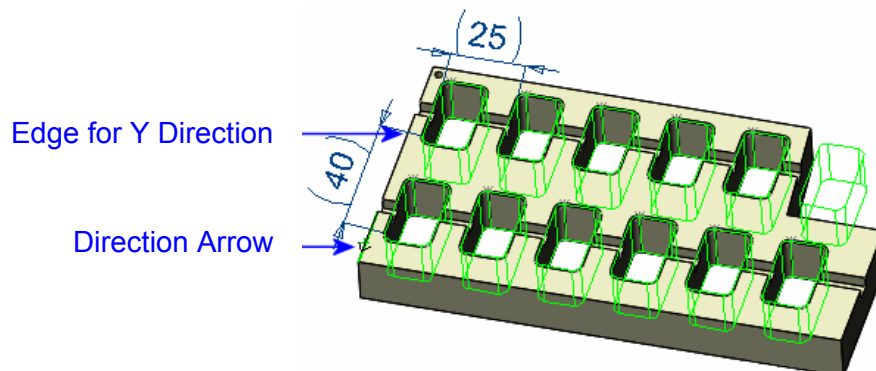
12. Click in the Instances to skip box to set the focus. Blue points display on each instance as shown in the figure on the left.

13. Move the pointer over the point for the instance on the top right of the part until it turns white, then click. The instance is removed and the point changes to red.

14. Click OK. Pattern1-Irregular Pocket1 is listed in the Feature tree.

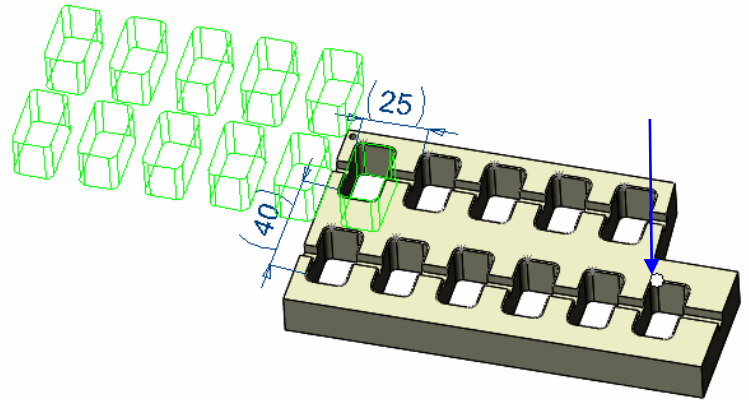15. Click each operation in the Operation tree. Notice the toolpaths have been copied to each instance.

You can also use a sketch to insert a Pattern feature.

1.  ![CW] Click the CAMWorks Feature Tree tab.

2.  Right click Pattern1-Irregular Pocket1 in the tree, select Pattern, then select Remove Pattern.

3.  Click Yes to confirm you want to remove the Pattern feature.

    The Pattern feature is removed. The original feature, Irregular Pocket1, and associated operations and toolpaths are not deleted.

4.  Right click Irregular Pocket1 in the tree and select Pattern, then select Insert Pattern.

5.  In the Insert Pattern dialog box, select Sketch Driven Pattern for the Type.

6.  Pick Sketch Pattern in the list of Available valid sketches.

    This box lists 2D sketches that contain arcs, circles or point entities. A 2D or 3D sketch can also be graphically selected. While the sketch can contain any entity type, only the center point of circles, arcs or point entities will be used to define the pattern feature locations. The Sketch Pattern sketch consists of points on each of the pockets.
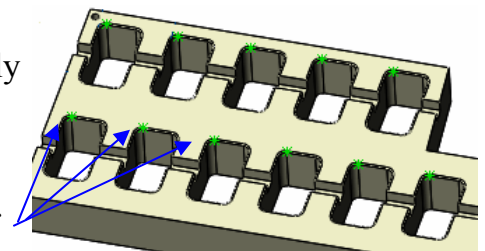
7. Notice that the pattern is not in the correct position on the part. The white point on the pocket at the bottom right identifies the point on the sketch that is the reference point for the feature. The default is the first point in the sketch. When this point is not correct, you can change it.

8. ⬅ ➡ Click the Move to previous and Move to next buttons to select the sketch point that positions the pattern correctly.

9. Click OK to insert the Pattern feature.

10. 🔨 Click the CAMWorks Operation Tree tab. Notice the toolpaths have been copied to each instance.
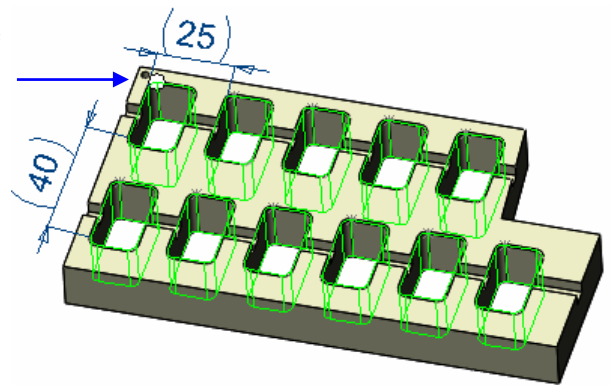
# 2 Axis Mill 10

## What You'll Learn

- Defining Features Interactively
- Inserting Circular Pattern Features
- Inserting Sketch Driven Pattern Features

1. Open the part file **MILL2AX_16.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. [CW] Click the CAMWorks Feature Tree tab.

3. Use the default Example Mill–in.

### *Defining Features Interactively*

1. Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

   AFR found a Hole Group feature. The Pocket features need to be defined interactively. You can simplify the creation of multiple features by defining one feature and then inserting a Pattern feature to array it to other positions. The toolpath generated for Pattern features is generated only once on the original feature and copied to each position, which reduces the calculation time.

2. Right click Hole Group1 in the tree and select Insert 2.5 Axis Feature.

3. Set the Feature Type to Pocket.

4. Make sure the Check for taper & fillets option is checked.

5. Pick the cyan face at the bottom of the pocket on the right side of the part.

6. Click Next.

7. Click the top face of the part to set the end condition to Upto Face.

8. Click the Autodetect button in the 2.5 Axis side wall section.

9. Click Finish.
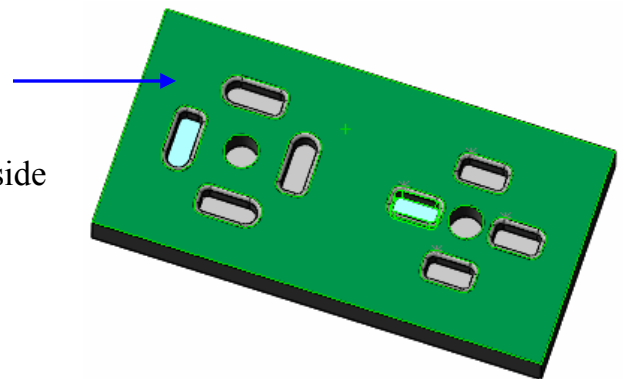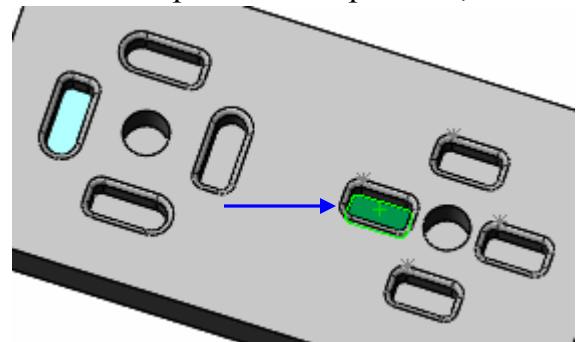
Define the 2.5 Axis feature for the circular pattern.

10. Use the same settings in the dialog box and pick the cyan face at the bottom of the pocket on the left side of the part.

11. Click Next.

12. Click the top face of the part to set the end condition to Upto Face.

13. Click the Autodetect button in the 2.5 Axis side wall section.

14. Click Finish, then click Close.
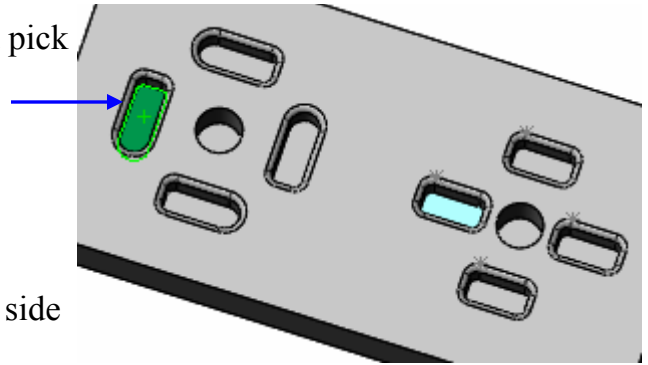
## *Creating a Pattern Feature*

In this exercise, you insert the Pattern feature using the Circular and Sketch Driven Patterns.

1. Right click Irregular Pocket1 in the tree and select Pattern, then select Insert Pattern.

2. In the Insert Pattern dialog box, select Circular Pattern for the Type.

3. Pick the top edge of the hole to define the rotation axis.

   By default, four instances of the pocket are rotated at 360 degrees around the axis.

4. Click OK.

5. Right click Rectangular Pocket1 in the tree and select Pattern, then select Insert Pattern.

6. In the Insert Pattern dialog box, select Sketch Driven Pattern for the Type.

7. Select Circular Sketch in the list of Available valid sketches.
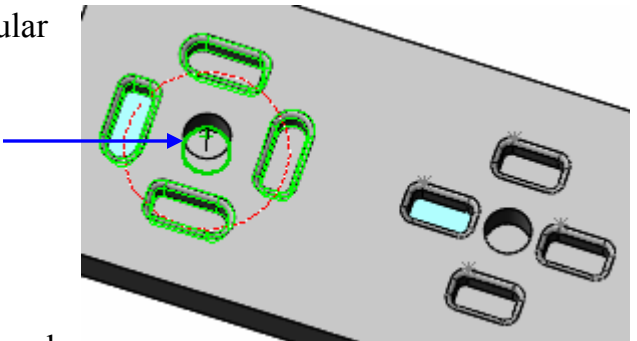
   This box lists 2D sketches that contain arcs, circles or point entities. A 2D or 3D sketch can also be graphically selected. While the sketch can contain any entity type, only the center point of circles, arcs or point entities will be used to define the pattern feature locations. The Circular Sketch consists of points on each of the pockets.

8. Notice that the pattern is not in the correct position on the part. The white point on the pocket at the bottom identifies the point on the sketch that is the reference point for the feature. The default is the first point in the sketch. When this point is not correct, you can change it.
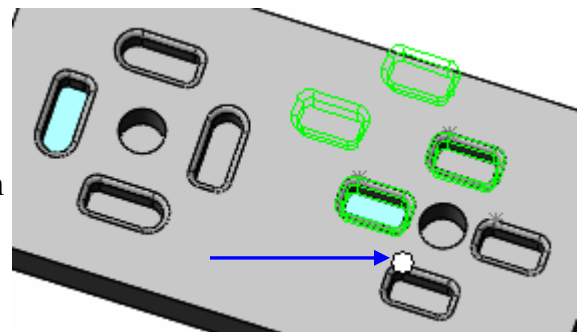
9.  Click the Move to previous button to select the sketch point that positions the pattern correctly.

10. Click OK to insert the Pattern feature.

11.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the tree and select Generation Operation Plan.
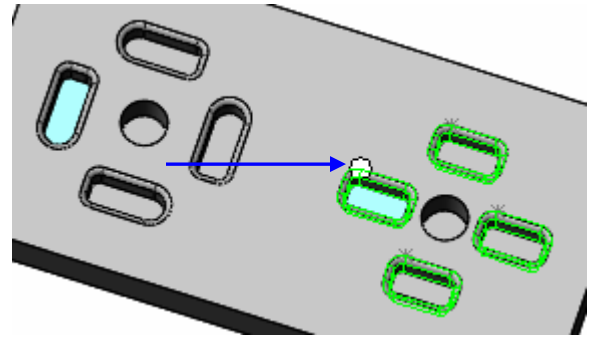
The tree lists the operations that CAMWorks generated for these features.

12. ⊞ Click the plus sign next to each operation in the tree and notice the operations have been generated only for the original features, not each for instance in the Pattern features.

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

14. Click the operations to view the toolpaths.

When you insert the Pattern feature, the toolpaths generated for this feature will be automatically copied to each position. You can generate the operations and toolpaths either before or after you insert a Pattern feature.

# For More Practice

The following exercises provide a chance for you to practice using CAMWorks. Instead of the detailed steps in the previous exercises, only a brief description of the procedure is included.

## Part1

Since this part is rectangular, you do not have to machine the outside of the part. AFR will find the holes and the pocket.

- Open the part file **MILLP1.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

- Change to the CAMWorks Feature tree.

- Use the default Example Mill–mm, controller and active Tool Crib.

- You can use the default bounding box for the stock.

- Click the Extract Machinable Features button on the CAMWorks toolbar.

- Right click on the Hole Group1 feature and select Parameters. Change the Attribute to Ream.

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

- Click the Generate Toolpath button on the CAMWorks toolbar.

- Right click on the Mill Part Setup and select Simulate Toolpath. Run the simulation.

- Click the Post Process button on the CAMWorks toolbar and generate the NC code.
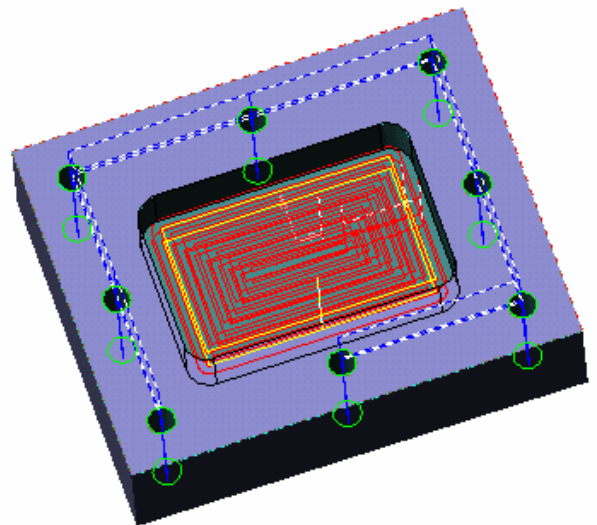
## Part2

Since this part is rectangular, you do not have to machine the outside of the part. AFR will find the holes, bosses, slots, pocket and corner slots.

- Open the part file **MILLP2.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

- Change to the CAMWorks Feature tree.

- Use the default Example Mill–mm machine, controller and active Tool Crib.

- Right click on the Stock Manager and select Edit Definition. Change the Bounding box offset for X, Y and Z to a uniform **1mm**.

- Click the Extract Machinable Features button on the CAMWorks toolbar.
- Right click on the Hole Group2 feature and select Parameters. Change the Attribute to Ream.
- Click the Generate Operation Plan button on the CAMWorks toolbar.
- Using the Sort Operations command, reorder to the operations so that the Rough Mill operations are first, followed by the Finish Mill operations and finally the hole operations.
- Right click on the Mill Part Setup and select Combine Operations to merge operations that use the same tool.

  In the Combine Operations dialog box, select the Rough Mill, Finish Mill and Center Drill options.

- Click the Generate Toolpath button on the CAMWorks toolbar.
- Click the Simulate Toolpath button on the CAMWorks toolbar. Make sure the Shoe Holder Collision button is selected, then run the simulation. Notice the holder collides with the part.
- Right click Rough Mill1 in the tree and select Edit Definition. On the Mill Tool tab, change the Protrusion to 65mm.

  This parameter defines the distance from the tip of the tool to the bottom of the holder. This value is used to determine how much of the tool protrudes from the mill holder during simulation. The protrusion length must be greater than the Effective Cut Length and less than the Overall Length.

- Simulate the toolpaths again to see the difference. This time the holder clears the part.
- Click the Post Process button on the CAMWorks toolbar and generate the NC code.
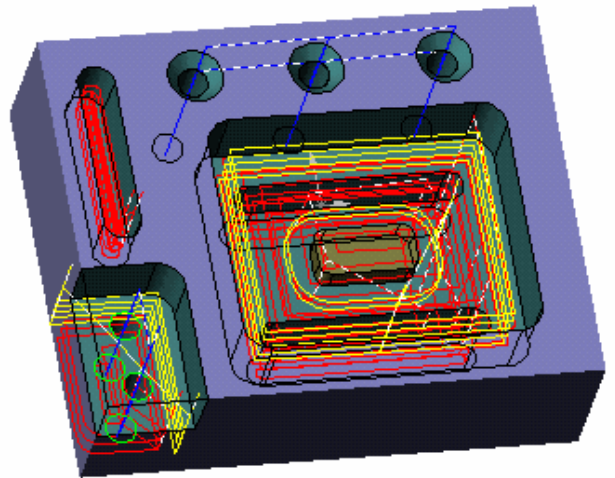
For More Practice

# Chapter 3  Learning 3 Axis Mill

This chapter provides an opportunity to learn 3 Axis Mill Level I through a step by step hands-on tour of the features and functions.

We recommend that you go through the exercises in Chapter 1 and Chapter 2 first. In addition to learning how to generate 2 axis mill toolpaths, these exercises explain basic functions and procedures that are used to generate both 2 and 3 axis mill toolpaths and NC code.

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

**IMPORTANT!**  The exercises in this chapter are intended to be used with the default Technology Database that is supplied with this version of CAMWorks. If you have modified the default Technology Database or imported customized data, the machining sequences and operations data in your database may be different.

If necessary, you can temporarily link to the default Technology Database using one of the following methods:

- Download the default CAMWorks Technology Database from the TekSoft web site (www.teksoft.com). Save it in the examples folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples*). Then run the Link Database Wizard, browse to the examples folder and select techdb.mdb. When you are finished with the exercises, run the Link Database Wizard and link to the TechDB that contains the customized data.

    or

- Reinstall CAMWorks. This reinstalls the default Technology Database. During installation, a backup is made of the techdb.mdb file that contains the customized data. When finished, move the backup to the active techdb.mdb location.

# 3 Axis Mill 1

This exercise introduces you to generating toolpaths and NC code for 3 axis machining.

## What You'll Learn

### *Defining the Machine, Stock and Machining Direction*

1. Open the part file **MILL3AX_1.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

Define the machine:

3. Right click Example Mill–in in the Feature tree and select Edit Definition.

   The Machine dialog box displays.

   On the Machine tab, Example Mill–in is the Active machine.

   This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

4. Click the Tool Crib tab and make sure Crib 1 is the Active tool crib.

5. Click the Controller tab and make sure FANTUTM is selected.

   FANTUTM 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.

6. Click OK to close the Machine dialog box.

Define the stock size and shape:

7. Right click the Stock Manager and select Edit Definition on the shortcut menu.

8. In the Manage Stock dialog box, click the Extruded Sketch button.

9. Pick the rectangular sketch representing the perimeter of the stock.

10. Click the Reverse Direction button.

11. Set the Depth to 1.5in, then click OK.
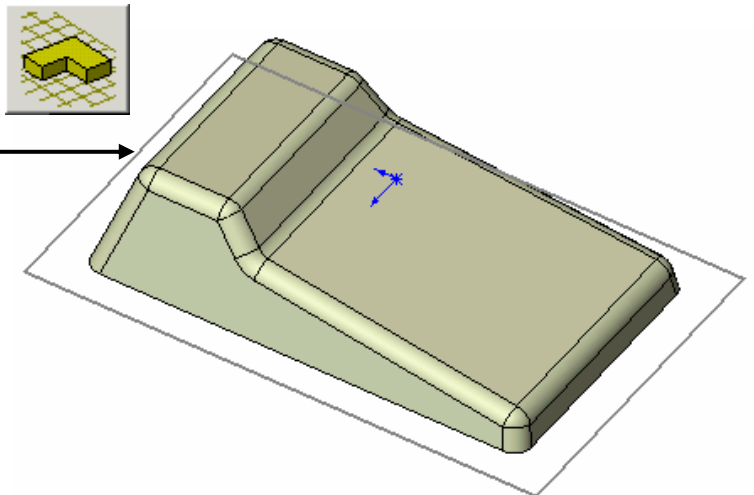
Define the machining direction, which will be normal to the face indicated below:

12. Right click Stock Manager in the Feature tree and select Insert Mill Part Setup on the shortcut menu.

13. Pick the top face.

14. Make sure the Face option is not checked, then click OK.

Mill Part Setup1 is added to the tree.
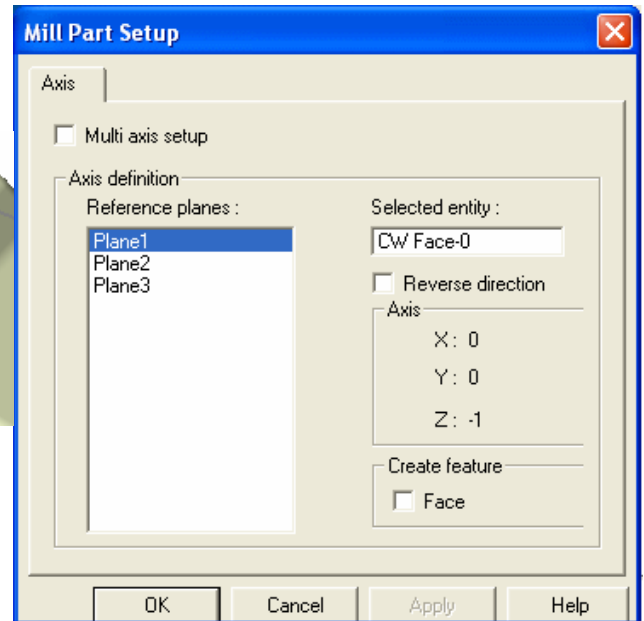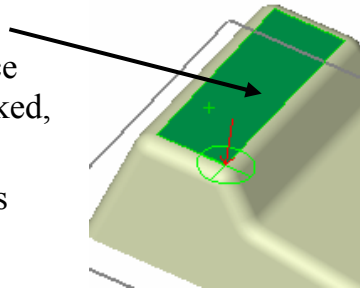
## Creating a Multi Surface Feature

In this exercise, AFR is not used since there are no 2.5 axis features on the part for AFR to find. You add the Multi Surface feature interactively.

Define a Multi Surface feature that includes the faces on the part to cut.

1. Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

The Insert Multi Surface Feature dialog box displays.

2. Click the Select all and associate option.

The Faces to cut box lists the selected faces.

Set the Attribute to Rgh-Lace, Fin-Slice. This attribute is defined in the Technology Database to generate an Adv. 3 Axis Rough operation with a Lace Pattern and an Adv. 3 Axis Pattern Project operation with a Slice Pattern.

### Did You Know ...
When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.
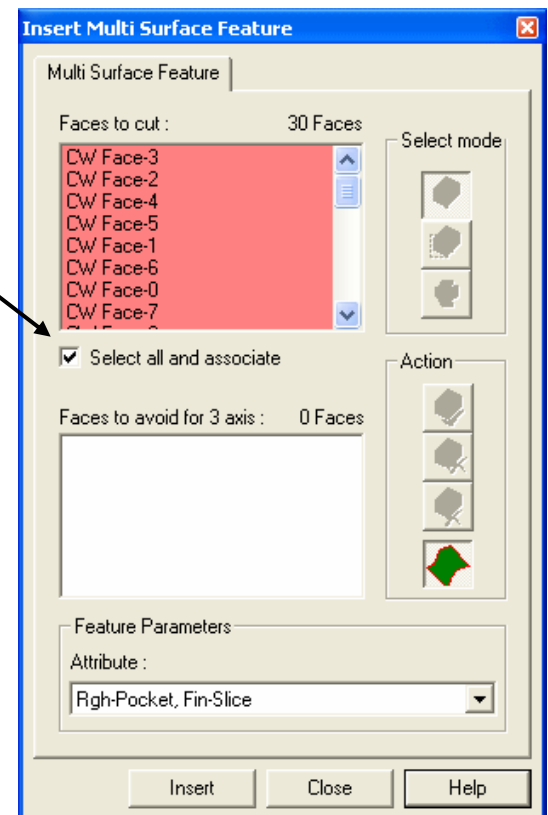
3. Click Insert, then click Close. The Multi Surface feature is listed in the Feature tree.

   New features, interactively inserted 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.
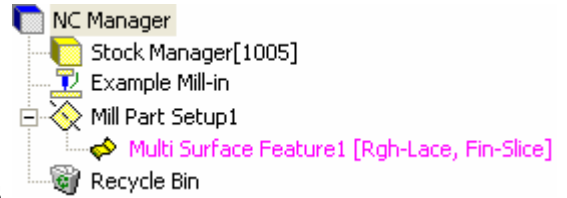
## Generating an Operation Plan and Modifying Operation Parameters

1. Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 in the Feature tree and select Generate Operation Plan.

   The Operation tree lists Adv 3 Axis Rough and Pattern Project operations.

2. Right click Adv 3 Axis Rough Mill1 in the tree and select Edit Definition.

3. Click the Tool Crib tab.

4. Click the check box to the left of the Filter button, then click the Filter button.
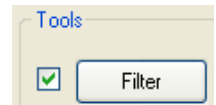
   The Tool Select Filter dialog box displays. This dialog box allows you to set filters for displaying and selecting tools.

5. Click the check box for the Filter by option to the right of the Type list box.

6. Select Flat End for the Tool type and click OK.

   The Tool Crib list changes to display only Flat End tools.

   When you set a tool filter for an operation, the setting is stored and becomes the default for that type of operation. You can change the filter settings as required or turn off the filter by clearing the check box.

7. Select a 0.5in diameter Flat End, then click the < - Select button.
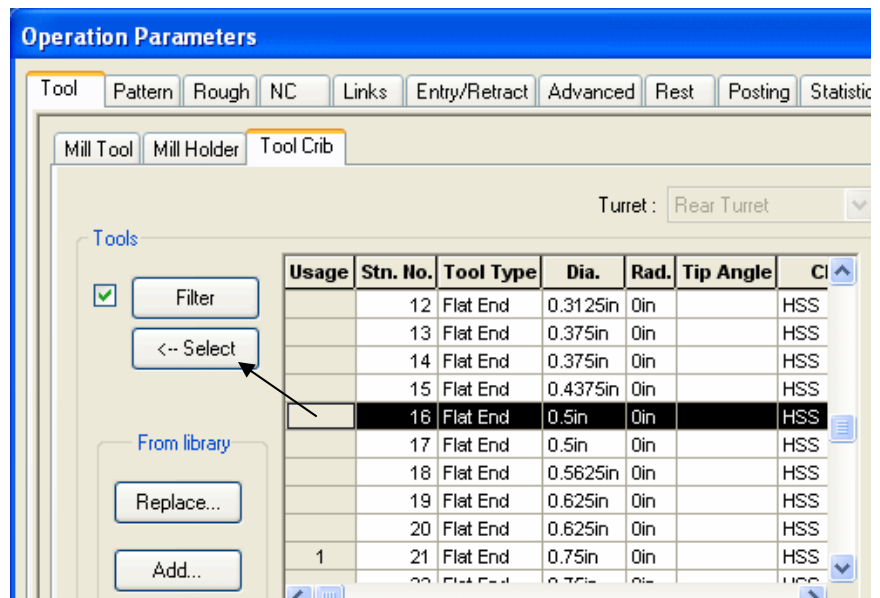
8. Click Yes to replace the corresponding holder.

9. Click the Rough tab.

10. Set the Mach. deviation to .005in.

This parameter defines a tolerance that is used to condense or shorten the 3 axis toolpath. The smaller the value, the greater the number of linear cuts and the smoother the surface finish.

**Operation Parameters**

| Tool | Pattern | Rough | NC | Links | Entry/Retract | Advanced | Rest | Posting | Statistics |

Surface finish

XY Allowance : 0.025in

Z allowance : 0.025in

Mach. deviation : 0.005in

Depth parameters

Method : Variable & Flats

Cut amount : 0.25in

Min. cut amount : 0.125in

Max. XY stepover : 0.1in

Level accuracy : 0.005in

Direction control

Milling type : ⦿ Climb
○ Conventional
☑ Zigzag
☐ Reverse direction
☐ Reverse stepover

Limits

First cut from : Top of Stock

Offset. : 0in

Last cut at : Bottom of Stock

Offset : 0in

11. For the Depth parameters Method, select Variable & Flats.

12. Leave the Cut amount set to 0.25in.

13. Set the Min. cut amount to 0.125in and the Max XY stepover to 0.1in, then click OK.
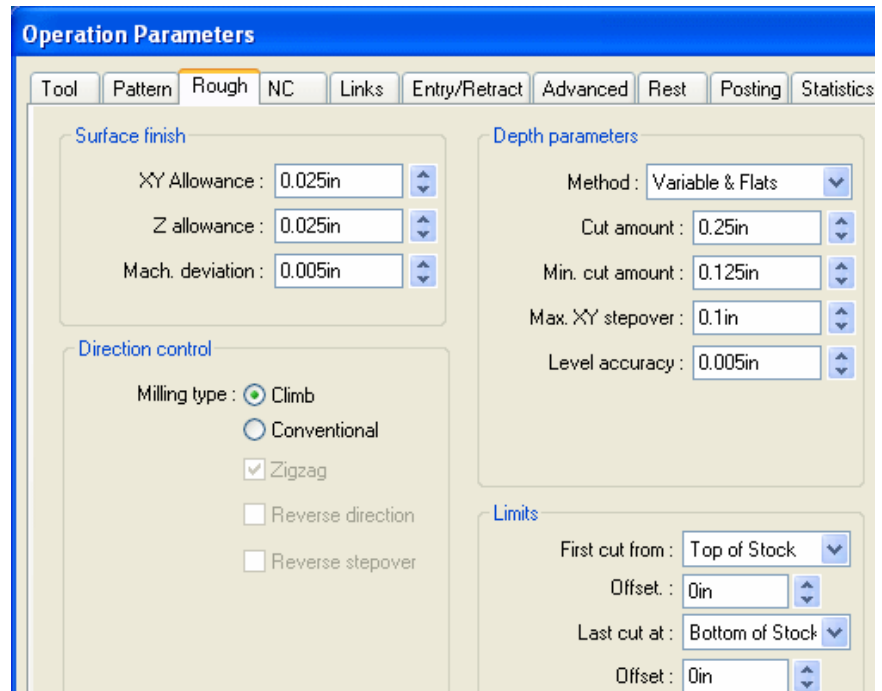
14. Right click Adv 3 Axis Pattern Project1 in the tree and select Edit Definition.

15. On the Pattern tab, notice that the Pattern is set to Slice as defined for the Rgh-Lace, Fin-Slice attribute in Technology Database. This pattern generates a series of linear parallel cuts across the part and is appropriate for semi-finishing almost any part shape and finishing non-vertical areas.

16. Set the following parameters on the Pattern tab:
   – Cross machining = Standard (This option allows you to automatically generate a second perpendicular toolpath on areas that are not machined or on poorly finished faces. The second toolpath is calculated by rotating the previous toolpath 90 degrees. The Standard option generates a perpendicular toolpath after the Slice toolpath.
   – Extension = 0.1in (Specifies the length a pass extends past the edge of the boundary to allow the tool to move into the cut at machining feedrate rather than rapid feedrate.)
   – Staydown = checked (This parameter affects how the tool transitions within contain areas and around avoid areas. When checked, tool retracts are minimized, however, additional feed motion is possible.)

17. On the Finish tab, make sure the XY and Z allowances are 0in. This value represents the amount of material left on the part after generating the toolpath.

18. Set the Mach. deviation to .005in.

19. On the Entry/Retract tab, use Skim for the Retract Method.

With this method, the tool rapids vertically to the minimum Z height needed to clear the faces in the feature, moves horizontally, then feeds vertically down to the start of the next pass. The minimum height of the retract is determined by the Skim clearance parameter.

20. Click OK.

## *Generating Toolpaths and Post Processing*

Generate toolpaths, simulate material removal and post process:

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

2. Click each operation in the tree to view the toolpath.

3. Right click Mill Part Setup1 and select Simulate Toolpath.

4. ⊁ Click the Turbo mode button on the Simulate Toolpath toolbar.
   Turbo mode provides ultra-fast simulation of toolpaths on a single Mill Part Setup.

5. ▶ Click Run.

6. ✖ Click the X button to exit the simulation.

7. G1 If you were generating the NC program for the toolpaths, the last step would be to click the Post Process button on the CAMWorks toolbar and post process the part.

# 3 Axis Mill 2

## What You'll Learn

- Inserting a Mill Part Setup
- Creating a Multi Surface Feature
- Generating Adv. 3 Axis Rough and Adv. 3 Axis Z Level toolpaths
- Using Contain and Avoid areas and user-defined limits to selectively machine within specified areas
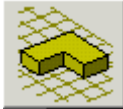
1. Open the part file **MILL3AX_2.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. [CW] Click the CAMWorks Feature Tree tab.

Define the machine:

3. Right click Example Mill-in in the Feature tree and select Edit Definition.

   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

Define the stock:

4. Right click the Stock Manager in the tree and select Edit Definition on the shortcut menu.

5. In the Manage Stock dialog box, click the Extruded Sketch button.

6. Pick the rectangular sketch that represents the stock.

7. Click the Reverse Direction button.

8. Set the Depth to **1.3in**, then click OK.
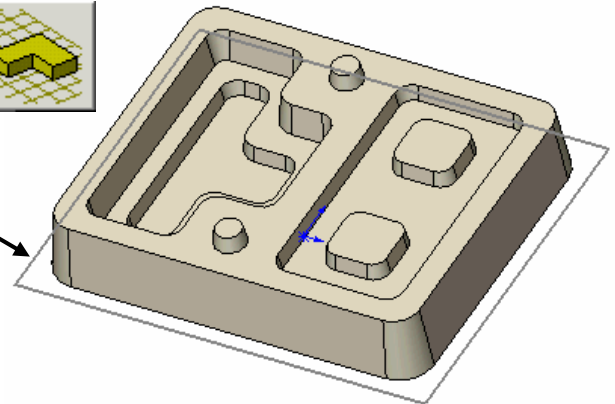
## *Inserting a Mill Part Setup and Creating a Multi Surface Feature*

Insert the Mill Part Setup and create a Multi Surface Feature:

1. Right click the Stock Manager and select Insert Mill Part Setup.

2. Pick the top of one of the rounds cone as shown in the figure.

3. Make sure the Face option is not checked, then click OK.

4. Right click the Mill Part Setup in the tree and select Insert Multi Surface Feature.

5. Click the Select all and associate option to select it.

6. For the Attribute, select Coarse.

7. Click Insert, then click Close.
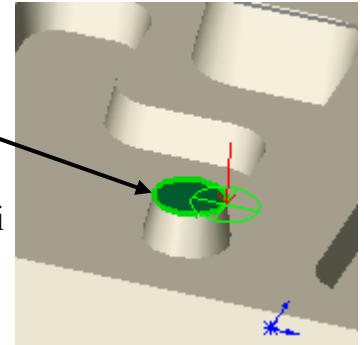
> **Did You Know ...**
>
> When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

## *Generating Adv. 3 Axis Rough Toolpaths*

Generate operations and toolpaths:

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

2. Right click Adv 3 Axis Rough1 in the Operation tree and select Edit Definition.

3. Click the Tool Crib tab, highlight a .375in Flat End mill and click the Select button.

4. Click Yes to replace the corresponding holder.

5. On the Pattern tab, set the Pattern to Lace.

6. On the Rough tab, use Constant for the Method and set the Cut amount to 0.2in.

7. On the Links tab, pick To depth by region for the Depth processing, use Round Corners and set the Max radius and Part deviation to 0.0375in.

8. On the Entry/Retract tab, use Skim for the Retract Method and set the Max. Ramp angle to 20deg.

9. Click OK.

10. Right click Adv 3 Axis Rough1 in the tree and select Generate Toolpath.
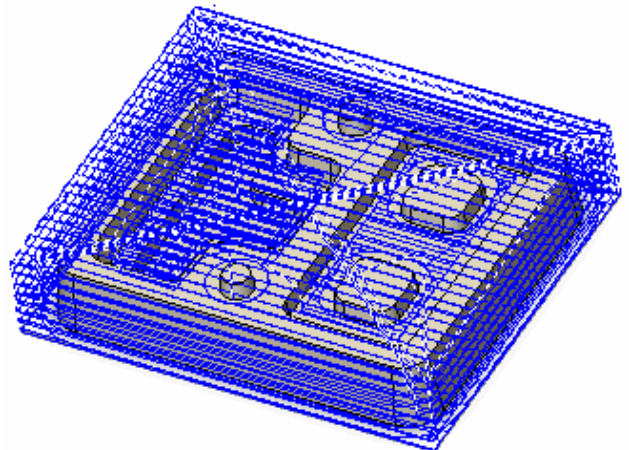
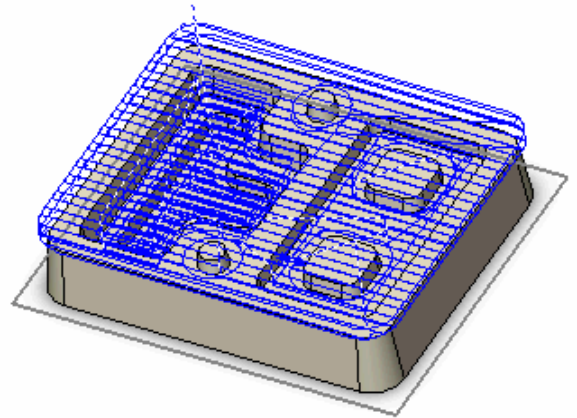## *Inserting Contain Areas to Selectively Machine Areas*

Depending on the machining requirements, you may want to insert a contain area to rough only the top of the part:

1. Right click Adv 3 Axis Rough1 in the tree and select Edit Definition.

2. On the Advanced tab, change the Automatic contain area Method to Outer Silhouette.

   With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature.

3. Click OK.

   When you click OK, the *Update 3 axis toolpaths when parameters change* option on the Update tab in the Options dialog box controls if 3 axis toolpaths are updated automatically. If this option is set to Prompt, a message displays asking if you want to regenerate toolpaths. Click Yes. If set to Always, CAMWorks automatically regenerates the toolpaths. If set to Never, right click the operation in the tree and select Generate Toolpath on the shortcut menu.

   ### 🔔 *Did You Know ...*
   For Advanced 3 Axis operations, the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either automatically on the Advanced tab in the Operation Parameters dialog box or interactively using the Insert Contain Area command.
   The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or not defined, then an automatically generated XY contain area will be used.

You may want to rough only the cavities:

4. Right click Adv 3 Axis Rough1 in the tree and select Insert Contain Area.

5. In the 3 Axis Wizard: Select Geometry dialog box, set the Selection filter to Convert to loop.

6. Set the Shape to Multiple.

7. Pick a top edge on each cavity, then click Finish. CAMWorks automatically chain selects all other edges connected to that edge at a constant depth.

8. Right click Adv 3 Axis Rough1 in the tree and select Edit Definition.

9. On the Pattern tab, change the Pattern to Pocket Out.

10. On the Links tab, set the Offset deviation to 0.0375in, then click OK.
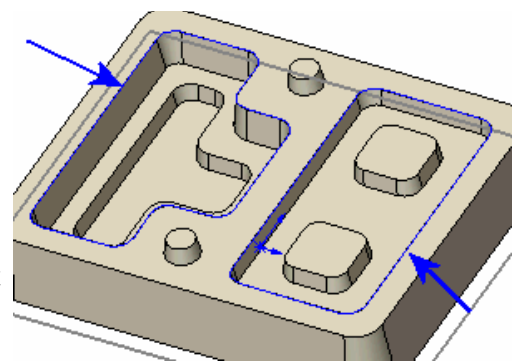
When you click OK, the *Update 3 axis toolpaths when parameters change* option on the Update tab in the Options dialog box controls if 3 axis toolpaths are updated automatically. If this option is set to Prompt, a message displays asking if you want to regenerate toolpaths. Click Yes. If set to Always, CAMWorks automatically regenerates the toolpaths. If set to Never, right click the operation in the tree and select Generate Toolpath on the shortcut menu.
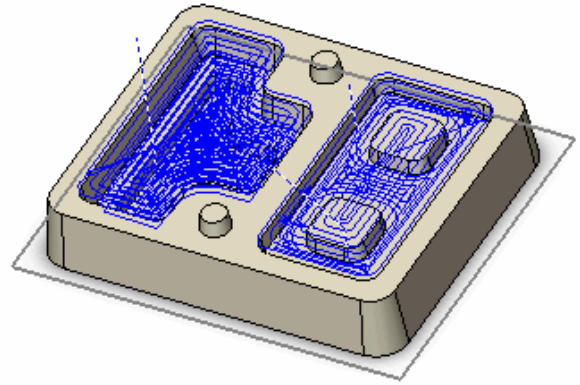
11. Right click Mill Part Setup1 in the tree and select Set View, then pick Setup XZ.

12. Click the operation in the tree and notice the amount of material at the bottom of the cavities.
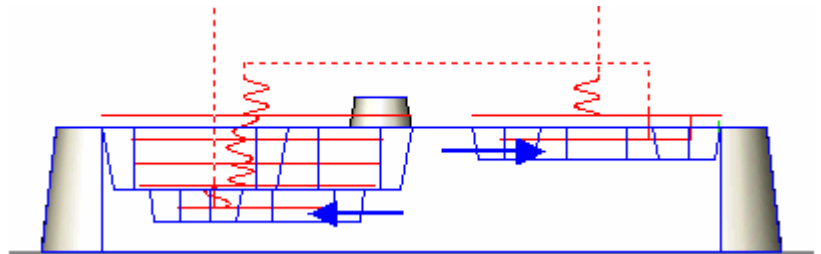
CAMWorks provides a Variable & Flats method, which inserts additional cuts in the toolpath at a Z location equal to the Allowance above any flat face between the top and bottom of the surface. These flat faces may define ledges or island tops. When this option is selected, the distance between tool passes varies based on the settings for Min. cut amount, Max. XY stepover, and Level accuracy.

13. Right click 3 Axis Rough1 in the tree and select Edit Definition.

14. On the Rough tab, change the Method to Variable & Flats, set the Min. cut amount to 0.05in, then click OK.

15. Generate the toolpath and notice the additional levels.
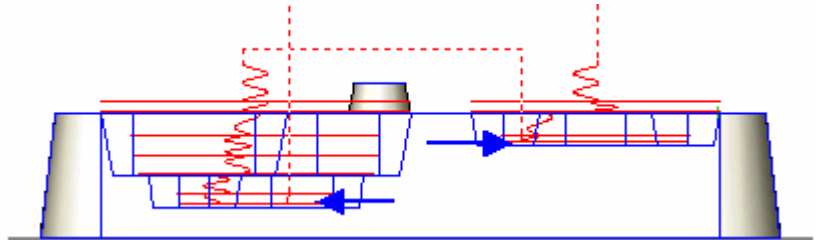
## Generating Adv. 3 Axis Z Level Toolpaths

The Advanced 3 Axis Z-Level cycle is a surface finish contouring cycle that removes material by making a series of horizontal planar cuts. The cuts follow the contour of the surface at different levels. Cutting starts from the highest location on the model and works down. Z-Level has unique characteristics that make it appropriate for steep walled areas. This cycle is generally not used on parts with flat or near flat faces.

Insert an Advanced 3 Axis Z-level operation:

1. Right click Adv 3 Axis Rough1 in the tree and select Insert Multi Surface Operation.

2. Set the Type of operation to Adv 3 Axis Z Level.

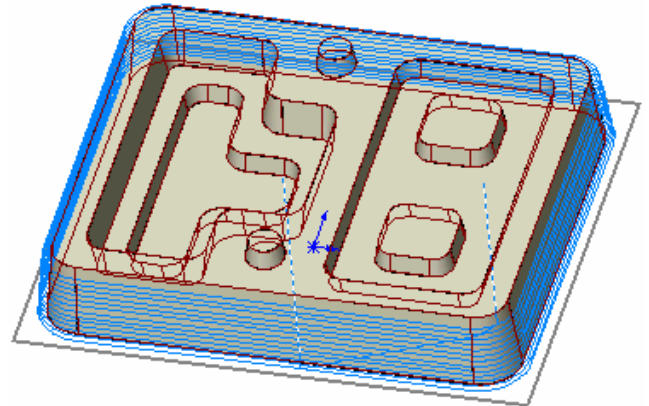3. Highlight Multi Surface Feature1 and click the Add button.

4. Make sure the Edit definition option is checked, then click Insert.

5. Click the Tool Crib tab, highlight a .25in Ball Nose tool, then click Select.

6. Click Yes to replace the corresponding holder.

7. On the Z Level tab, set the Cut amount to 0.08in.

8. On the Links tab, make sure To depth by region is selected for Depth processing.

9. On the Entry/Retract tab, use Skim for the Retract Method and set the Max. ramp angle to 20deg.

10. Click OK.

11. Right click Adv 3 Axis Z Level1 in the tree and select Generate Toolpath.
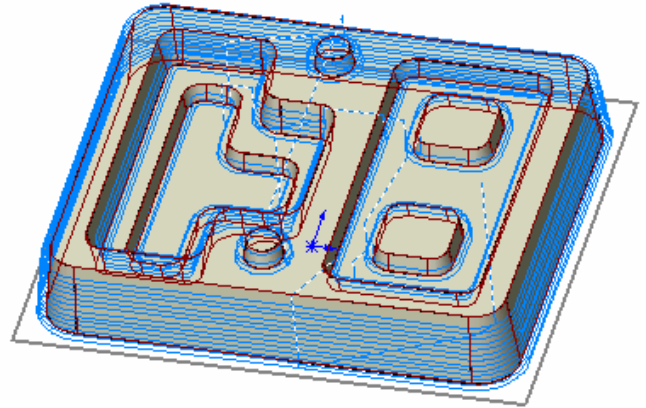
## Inserting an Avoid Area to Selectively Machine Areas

Depending on machining requirements, you may want to cut around only the outside of the part.

1. Right click Adv 3 Axis Z Level1 in the tree and select Insert Avoid Area.

2. Pick the top face of the part and click Finish.

3. Right click Adv 3 Axis Z Level1 in the tree and select Generate Toolpath.

## Using User-defined Limits to Machine Specific Areas

Insert an Advanced 3 Axis Z Level operation and limit the toolpath to cut around the cones with a tapered tool:

1. Right click Adv 3 Axis Z Level1 in the tree and select Insert Multi Surface Operation.

2. Set the Type of operation to Adv 3 Axis Z Level.

3. Remove the check mark from the Use default parameters option to copy the parameters from the Adv 3 Axis Z Level1 operation.

4. Highlight Multi Surface Feature1 and click the Add button.

5. Make sure the Edit definition option is checked, then click Insert.

6. Click the Tool Crib tab and select a .25in Flat End Tapered tool.

7. Click Yes to replace the corresponding holder.

8. On the Tapered Tool tab, make sure the Taper angle to 10deg.

9. Click the Z Level tab.

10. In the Limits section, select User Defined for the First cut from option.

11.  Click the Pick Point button.

12. Pick the top edge of one of the cones, then click OK to close the dialog box.

13. Select User Defined for the Last Cut at option.

14.  Click the Pick Point button.

15. Pick the bottom edge of the cone and click OK.

16. Click OK to close the dialog box.
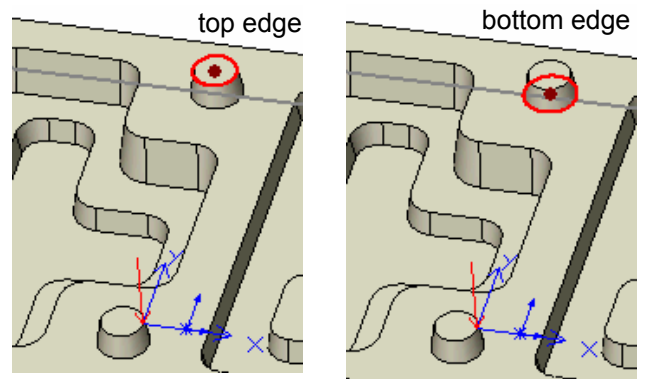
> You do not need to change any other parameters since you copied the parameters from the previous operation.

17. Left click the + next to Adv 3 Axis Z Level2 in the tree.

18. Right click Avoid Area2, select Delete on the shortcut menu, then click Yes.

> When you insert an operation and copy parameters from another operation, contain and avoid areas associated to that operation are also copied.

19. Right click Adv 3 Axis Z Level2 in the tree and select Generate Toolpath. Only the area around the cones is cut.

20. Right click Adv 3 Axis Z Level2 in the tree and select Simulate Toolpath.

21.  Click Run. The tapered tool cuts around the cones.

22.  Click the X button to exit the simulation.
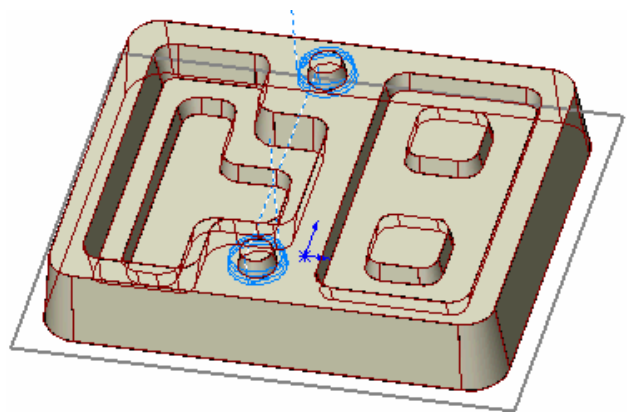
## Finishing the Flat Areas

If you have a Mill 3AX Level II license, you can use an Advanced 3 Axis Flat Area operation to machine the flat areas on this part. The Advanced 3 Axis Flat Area cycle uses a roughing pattern to remove material on the z-levels of flat faces between the top and bottom of the surface. CAMWorks generates toolpaths only on completely flat areas.

Note that if you do not have a Level II license, you can interactively define 2.5 Axis features for the flat areas, then generate 2 Axis operations.

Insert an Advanced 3 Axis Flat Area operation:

1. Right click Adv 3 Axis Z Level2 in the tree and select Insert Multi Surface Operation.

2. Set the Type of operation to Adv 3 Flat Area.

3. Highlight Multi Surface Feature1 and click the Add button.

4. Make sure the Edit definition option is checked, then click Insert.

5. Click the Tool Crib tab and select a .375in Flat End tool.

6. Click Yes to replace the corresponding holder.

7. On the Links tab, make sure To depth by region is selected for the Depth processing.

8. On the Entry/Retract tab, use Skim for the Retract Method, set the Max. ramp angle to 20deg, then click OK.

9. Right click Adv 3 Axis Flat Area1 in the tree and select Generate Toolpath.

   CAMWorks generates toolpaths on the flat areas.
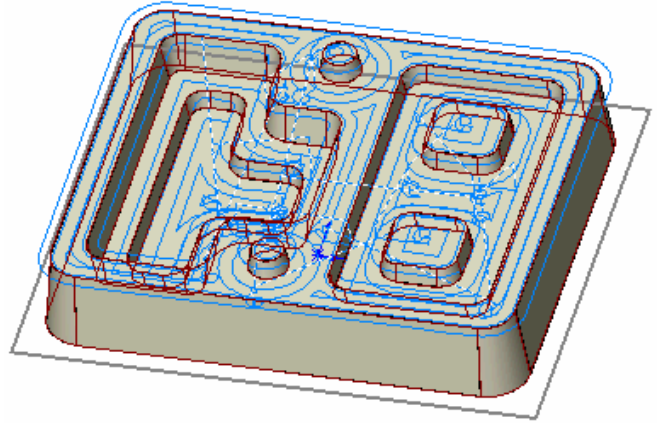
# 3 Axis Mill 3

CAMWorks allows parts to be machined using any combination of 2 and 3 Axis machining operations.

## What You'll Learn

- Creating 2.5 Axis and Multi Surface features and operations for the same part.
- Inserting a contain area to control the area machined on the Multi Surface feature.
- Generating Advanced 3 Axis Pattern Project toolpaths using the Cross machining option.
- Inserting an Advanced 3 Axis Pencil Mill operation.

### Using AFR to Identify 2.5 Axis Features

1. Open the part **MILL3AX_3.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ▣ Click the CAMWorks Feature Tree tab.

   This part needs to be machined with a combination of both 2 axis and 3 axis machining operations. The 2.5 axis features include the counterbored holes and the two pockets on either end of the main cavity. The cavity itself has tapered walls as well as a tapered bottom, which necessitates a 3 axis machining operation.
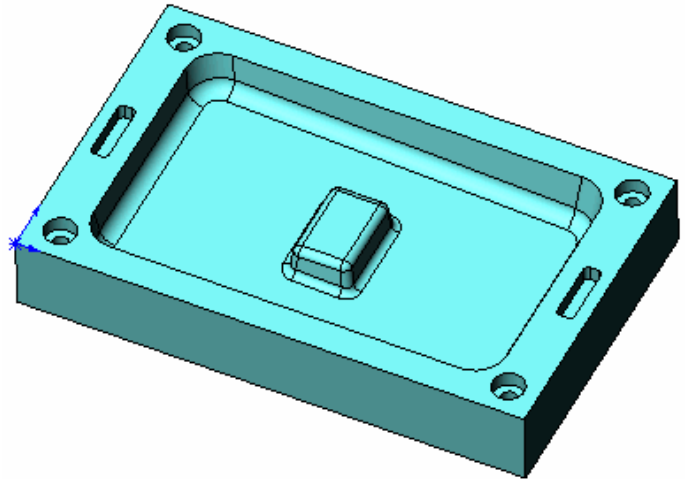
3. ⚏ Right click Example Mill–in in the Feature tree and select Edit Definition.

   − On the Machine tab, Example Mill–in is the Active machine.
   − Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   − Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. ▢ Right click Stock Manager in the tree and select Edit Definition.

   The default bounding box displays on the part.

5. Click OK to use the default stock for this exercise.

When a part contains 2.5 axis features, normally the next step is to allow CAMWorks to identify features automatically using Automatic Feature Recognition. Then, if AFR does not recognize features that you are certain could be machined with 2 axis cutting cycles, you can define the features interactively with the Insert 2.5 Axis Features command.

6.      Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

CAMWorks finds two features: a Rectangular Pocket Group that includes the two rectangles at the ends of the cavity and a Counterbore Hole Group that includes the four counterbore holes.



## Creating Multi Surface Features

Add the Multi Surface feature interactively:

1.   Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

   The Insert Multi Surface Feature dialog box displays. A Multi Surface feature consists of a list of faces to cut and optionally a list of faces to avoid.

2.   Click the Select all and associate option.

   The Faces to cut box lists the selected faces. The Select all and associate option selects all faces that can be machined from any machining direction.

   *Did You Know ...*
   When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

3.   Set the Attribute to Rgh-Lace, Fin-Slice, then click Insert and Close.

   A Multi Surface feature is created and Multi Surface Feature1 is added to the Feature tree.

4.      Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 and select Generate Operation Plan.

## *Modifying Operation Parameters and Inserting Contain Areas*

Before generating toolpaths for the Multi Surface feature, you can change machining parameters and create a contain area to machine only the cavity:

1. Right click Adv Pattern Project1 in the Operation tree and select Insert Contain Area on the shortcut menu.

2. In the 3 Axis Wizard: Select Geometry dialog box, change the Selection filter to Convert to loop.

3. Pick an edge around the top of the cavity.

   The Entities selected list displays Loop <1>.

4. Click Finish.

5. Right click Rough Mill1 in the tree and select Edit Definition on the shortcut menu.

6. Click the Tool Crib tab and select a .1875in Flat End mill.

7. Click Yes to replace the corresponding holder.
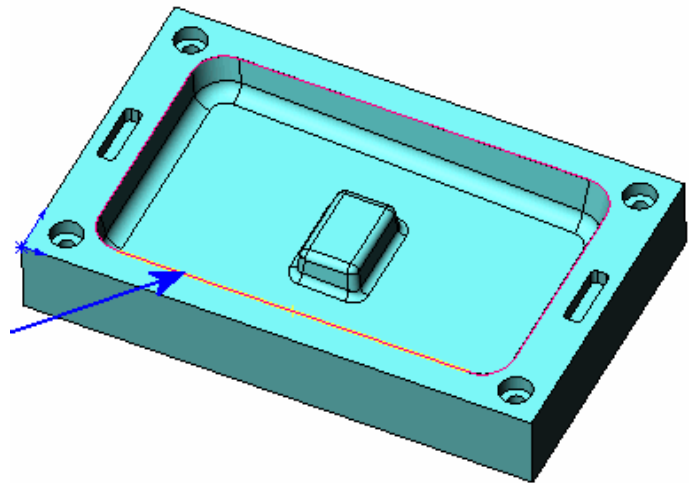
8. Click OK to close the dialog box.

9. ⬛ Click the Generate Toolpath button on the CAMWorks toolbar.

   Toolpaths are generated for all the operations.

10. ⬛ Click the Simulate Toolpath button on the CAMWorks toolbar.

11. Set the End condition to Next Operation.

12. ⬛ ▶ Click the Turbo button, then click Run to view each toolpath simulation.

13. ⬛ Click the Show Difference button and notice the amount of material left on the tapered areas of the cavity and around the boss.
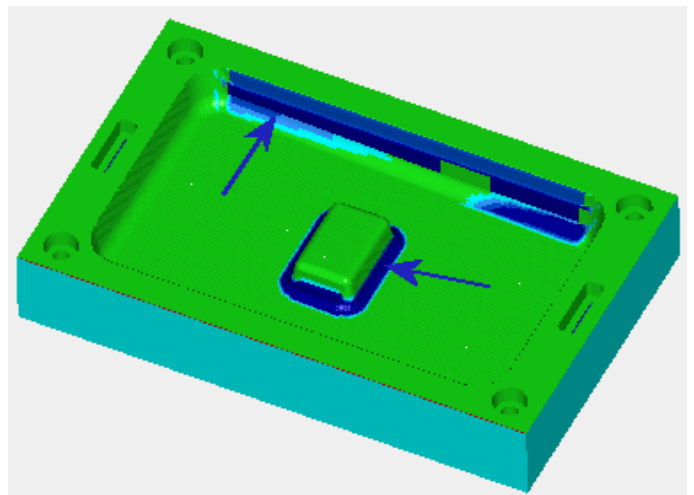
14. ❌ Exit the simulation.

## *Generating Toolpaths Using Cross Machining*

1. Right click Adv 3 Axis Pattern Project1 in the tree and select Edit Definition.

2. On the Pattern tab, change the Cross machining parameter to Standard.

   When Slice is selected, Cross machining allows you to automatically generate a second perpendicular toolpath on areas that are not machined or on poorly finished faces. The second toolpath is calculated by rotating the previous toolpath 90 degrees. When Standard is selected, a perpendicular toolpath is generated after the Slice toolpath.

3. Click the Staydown option to select it.

   This option affects how the tool transitions within contain areas and around avoid areas. With Staydown selected, tool retracts are minimized but additional feed motion is possible.

4. Click OK.

   When you click OK, the *Update 3 axis toolpaths when parameters change* option on the Update tab in the Options dialog box controls if 3 axis toolpaths are updated automatically: If this option is set to Prompt, a message displays asking if you want to regenerate toolpaths. Click Yes. If set to Always, CAMWorks automatically regenerates the toolpaths. If set to Never, right click the operation in the tree and select Generate Toolpath on the shortcut menu.
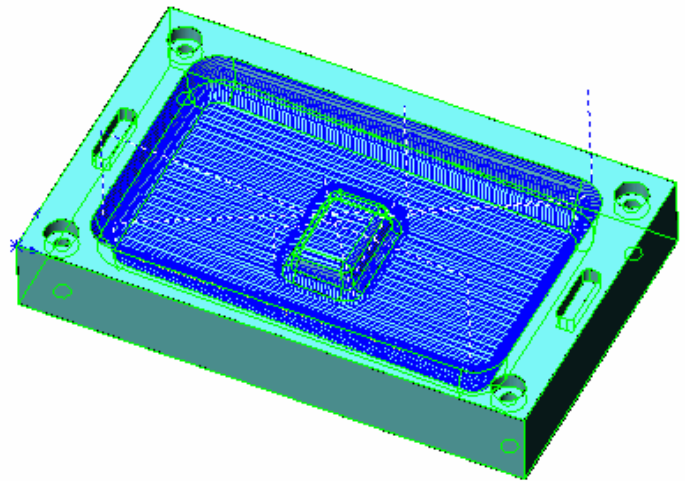
   The tapered areas of the cavity are machined by the cross machining toolpath.

## *Inserting an Adv. 3 Axis Pencil Mill Operation*

1. Right click Adv 3 Axis Pattern Project1 in the tree and select Insert Multi Surface Operation.

2. For the Type of operation, select Adv 3 Axis Pencil Mill.

3. Pick the Multi Surface Feature1 and click Add.

4. Click Insert.

5. On the Mill Tool tab, make sure the tool is a .25in Ball Nose.

6. On the Pattern tab, set the Pattern to Parallel Lace

7. On the Links tab, set the Max. link distance to .25in, then click OK.

8. Right click Adv 3 Axis Pencil Mill1 and select Insert Contain Area.

9. In the 3 Axis Wizard: Select Geometry dialog box, change the Selection filter to Convert to loop.

10. Pick an edge around the top of the cavity, then click Finish.

11. Right click Adv 3 Axis Pencil Mill1 and select Generate Toolpath.
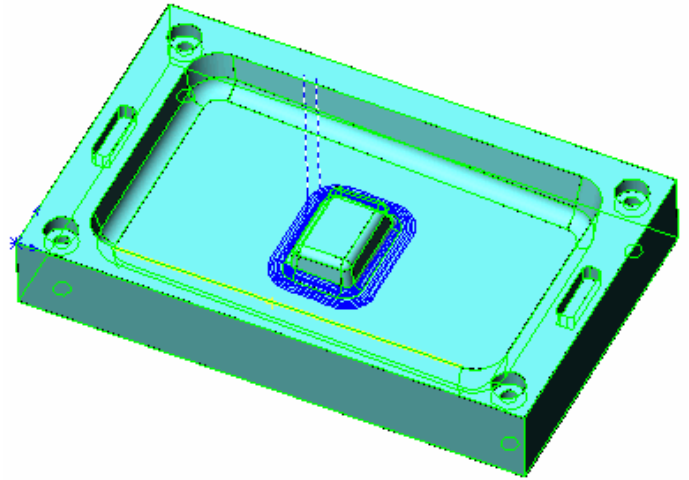
    The toolpath cuts the area around the boss.

12. ![Simulate] Click the Simulate Toolpath button on the CAMWorks toolbar.

13. Set the End condition to Next Operation.

14. ![Turbo] ![Run] Click the Turbo button, then click Run to view each toolpath simulation.

15. ![Show Difference] Click the Show Difference button and notice that all the material is removed.

16. ![Exit] Exit the simulation.

# 3 Axis Mill 4

## What You'll Learn

- Inserting a Mill Part Setup
- Creating a Multi Surface Feature
- Defining program zero
- Adjusting operation parameters and generating toolpaths
- Generating Adv. 3 Axis Stepover, Adv. 3 Axis Pattern Project and Adv. 3 Axis Pencil Mill toolpaths

### Defining the Machine and Machining Direction

1. Open the part file **MILL3AX_4.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

   This part is an imported surface model. The rectangular shape of the part creates a bounding box material block that works well for machining the part.

2. [CW] Click the CAMWorks Feature Tree tab.

3. 🔁 Right click Example Mill–in in the tree and select Edit Definition.

   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. 📁 Right click Stock Manager in the tree and select Edit Definition.

   The default bounding box displays on the part.

5. Click OK to use the default stock for this exercise.

6. Right click the Stock Manager and select Insert Mill Part Setup.

   The Mill Part Setup dialog box displays.

7. Select Plane4 from the list of Reference planes.
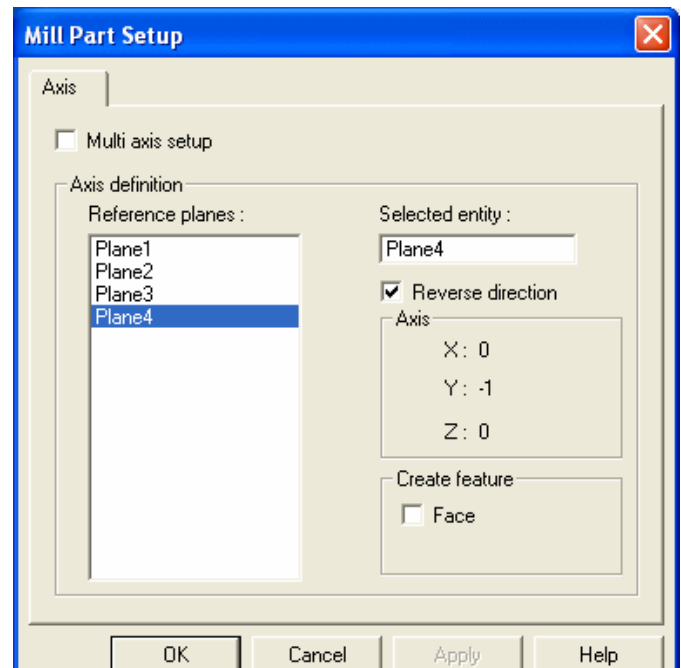
8. Select the Reverse direction option to flip the Z axis up.

9. Make sure the Face option is not checked, then click OK.

   The new Mill Part Setup is listed in the Feature tree.
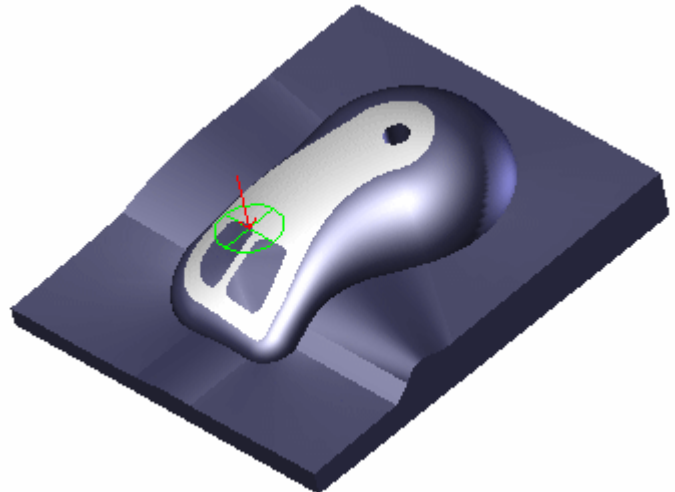
## Creating a Multi Surface Feature

In this exercise, AFR will not be used since there are no 2.5 axis features on the part for AFR to find. You add the Multi Surface feature interactively.

1. Right click Mill Part Setup1 in the tree and select Insert Multi Surface Feature.

   The Insert Multi Surface Feature dialog displays.

2. Click the Select all and associate option.

   **Did You Know ...**
   When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

3. Set the Attribute to Rgh-Lace, Fin-3D Stepover, Pen-Lace.

   This attribute has been defined in the TechDB to generate an Adv. 3 Axis Rough operation with a Lace Pattern, an Adv. 3 Axis Stepover operation and an Adv. 3 Axis Pencil Mill operation.

4. Click Insert, then click Close.

   The new Multi Surface feature is listed under the Mill Part Setup in the Feature tree.

## Defining Program Zero

1. Click the Generate Operation Plan button on the CAMWorks toolbar or right click the Multi Surface feature and select Generate Operation Plan.

2. Right click on the Mill Part Setup in the Operation tree and select Edit Definition.
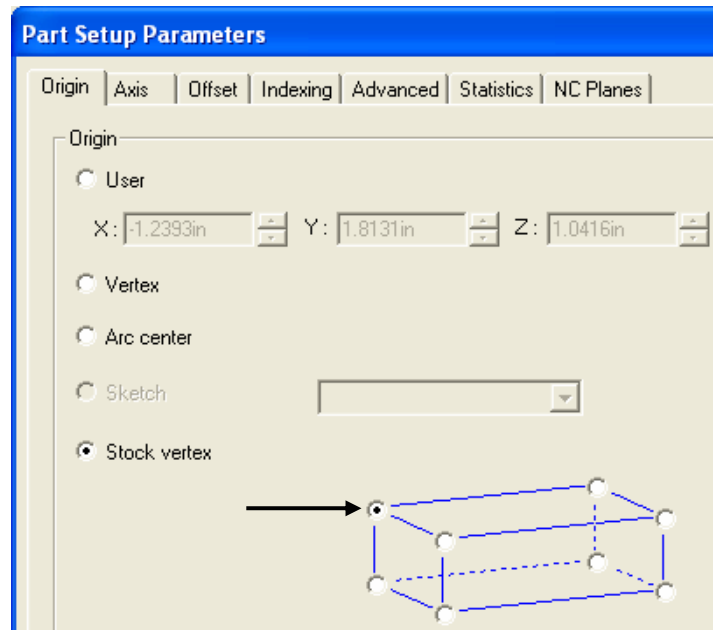
The Part Setup Parameters dialog box displays. This dialog box allows you to change the parameters of the Mill Part Setup you inserted earlier.

3. Select Stock vertex and leave the corner as shown.

The XYZ zero location is now set at the theoretical top corner of the material and the Mill Part Setup indicator moves to that position in the graphics area.

This is good practice and will assure that your default rapid moves will not contact the part.

4. Click OK.

## *Adjusting Operation Parameters and Generating Toolpaths*

1. Right click Adv 3 Axis Rough1 in the tree and select Edit Definition.

The Operation Parameters dialog box displays.

2. On the Pattern tab, set the Cut angle to 90deg.

3. On the Rough tab, set the following parameters:
   - XY allowance = 0.01in
   - Z allowance = 0.01in
   - Method = Variable & Flats
   - Cut amount = .25in
   - Min. cut amount = 0.1in
   - Max. XY stepover = 0.1in

4. On the Entry/Retract tab, use Skim for the Retract Method and set the Ramping Method to Ramp, then click OK.

5. Right click Adv 3 Axis Rough1 in the tree and select Generate Toolpath.

## *Editing the Adv. 3 Axis Stepover Operation to Cut the Mouse*

The Advanced 3 Axis Stepover operation removes material by maintaining a constant user-defined stepover.

1. Right click on Adv 3 Axis Stepover1 in the tree and select Edit Definition.

2. On the Stepover tab, if you have 3 Axis Mill Level II, the Method is set to 3D Stepover. Otherwise the Method is grayed out and 2D Stepover will be used.

When 2D Stepover is selected, the distance between each pass will be constant and equal to the Horizontal cut amount, as viewed from the XY direction. This method produces a consistent surface finish on feature faces that do not have a steep slope angle. Because the stepover is constant in the XY plane, the surface finish on feature faces that are vertical or have a steep slope angle will not be as good.

The 3D Stepover method generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope of the feature faces. In shallow areas the stepover will be equal to the Horizontal cut amount and in steeper areas the stepover will be equal to the Vertical cut amount. This method produces the most consistent surface finish on features whose faces have either shallow or steep slope angles. A Mill 3Ax Level II license is required for the 3D Stepover method.

3. Set the Horizontal and Vertical cut amounts to 0.05in, then click OK.

4. Right click Adv 3 Axis Stepover1 in the tree and select Insert Contain Area on the shortcut menu.

5. Pick each edge around the mouse.

6. Set the Offset to 0.1875in.

   This value is the radius of the tool and allows the tool to machine to the bottom of the mouse. Notice the offset curve that is generated from the selected edges. The toolpath will be created inside of this bounding curve.
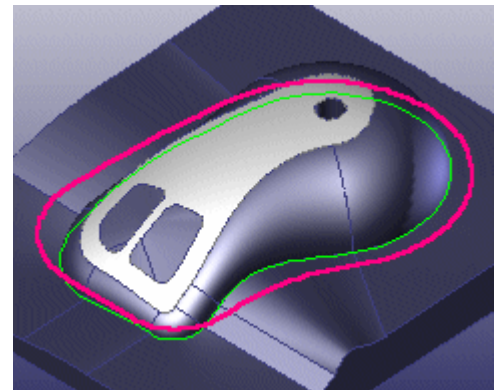
7. Click the Finish button to close the dialog box.

8. Right click Adv 3 Axis Stepover1 in the tree and select Insert Avoid Area on the shortcut menu.

9. Pick the edge around the top of the hole.

10. Check the Machine over option, then click Finish.

    For this exercise, you can assume the hole will be drilled later. This option controls whether CAMWorks machines over or around the avoid area.
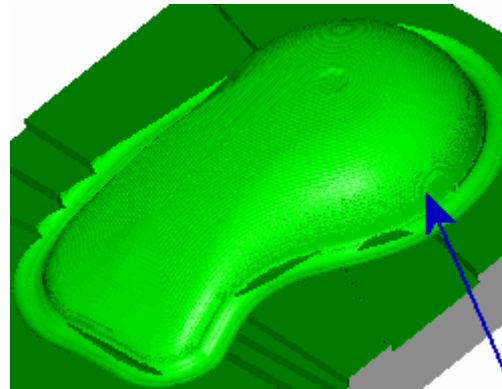
11. Right click 3 Axis Stepover1 in the tree and select Generate Toolpath.

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

13. Set the End condition to Next Operation.
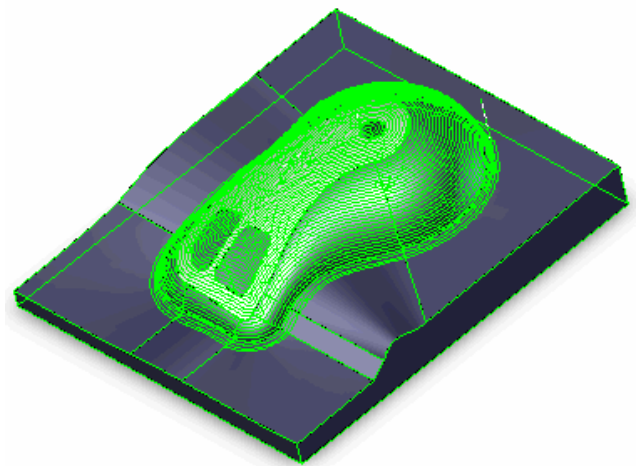
14.   Click the Turbo button, then click Run to view each toolpath simulation.
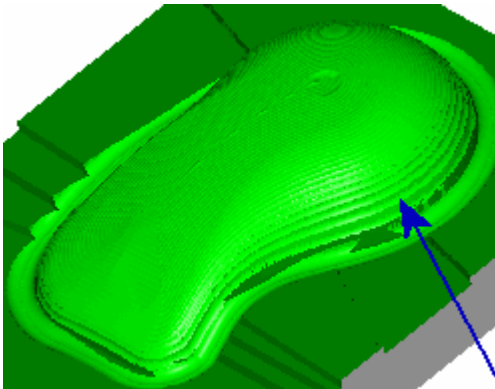


| 2D Stepover | 3D Stepover |

15. In the above figures notice that with the 2D Stepover method the surface finish on the vertical feature faces is not as good as the finish with the 3D Stepover method.

16.  Click the X button in the upper right corner of the simulation toolbar or press the Escape key on the keyboard to exit the simulation.

## *Inserting an Adv. 3 Axis Pattern Project Operation to Cut the Base*

1. Right click on Adv 3 Axis Stepover1 in the tree and select Insert Multi Surface Operation.

2. For the Type of operation, select Adv 3 Axis Pattern Project.

3. Pick Multi Surface Feature1 in the Features list and click Add.

4. Leave the Edit Definition option checked, and click Insert.

5. On the Pattern tab, use Slice for the Pattern and check the Staydown option.

6. On the Finish tab, make sure the Direction control is set to Zigzag.

7. On the Links tab, set the Max link distance to .25in, then click OK.

Insert a Contain Area to cut only the top of the base:

8. Right click Adv 3 Axis Pattern Project1 in the tree and select Insert Contain Area.

9. In the 3 Axis Wizard: Select Geometry dialog box, pick Sketch1, set the Tool condition to On and click Finish.

Insert an Avoid Area to avoid machining the mouse:

10. Right click Adv 3 Axis Pattern Project1 in the tree and select Insert Avoid Area on the shortcut menu.

11. Pick each edge around the mouse.

12. Set the Tool condition to Upto and click Finish.

13. Right click Adv 3 Axis Pattern Project1 in the tree and select Generate Toolpath.
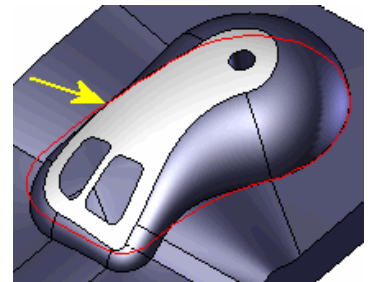
CAMWorks generates the toolpath on the base and avoids the mouse.

14. 🔧 Click the Simulate Toolpath button on the CAMWorks toolbar.

15. Set the End condition to Next Operation.

16. ⚹ ▶ Click the Turbo button, then click Run to view each toolpath simulation.

17. 🔲 When the simulation is finished, click the Show Difference button.

This function shows the difference between the machined model and the SolidWorks design model. Note the blue area at the parting line showing material left over by the radius of the ball tool.

18. ❌ Exit the simulation.
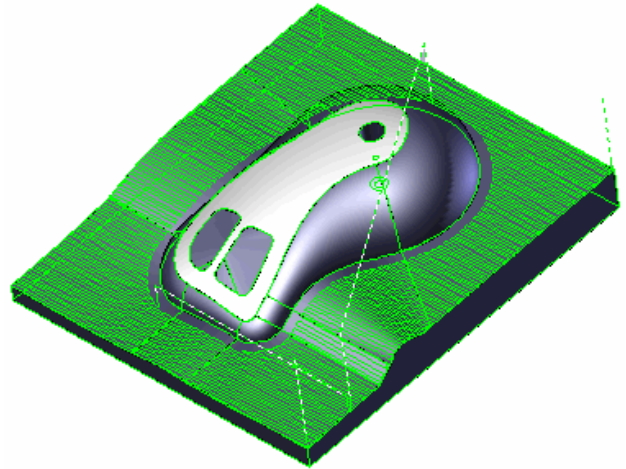
## Editing the Adv. 3 Axis Pencil Mill Operation to Cut Parting Line

1. Right click on Adv 3 Axis Pencil Mill1 in the tree and select Edit Definition.

2. On the Mill Tool tab, make sure the Cut diameter is .125in.

3. On the Pattern tab, the Pattern is set to Parallel Lace.
   This method generates multiple parallel passes to remove the material. The distance between passes is equal to the Cut amount.

4. Set the following parameters:
   - Bitangency angle = 30deg.
   - Method = 3D Stepover (in shallow areas the stepover will be equal to the Horizontal cut amt. and in steeper areas the stepover will be equal to the Vertical cut amt.)
   - Number of Cuts = Checked with 5 cuts
   - Horizontal & Vertical cut amt. = 0.04in

5. Click OK.

6. Right click Adv 3 Axis Pencil Mill1 in the tree and select Insert Avoid.

7. Pick the top edge of the hole and click Finish.

8. Right click Adv 3 Axis Pencil Mill1 in the tree and select Generate Toolpath.
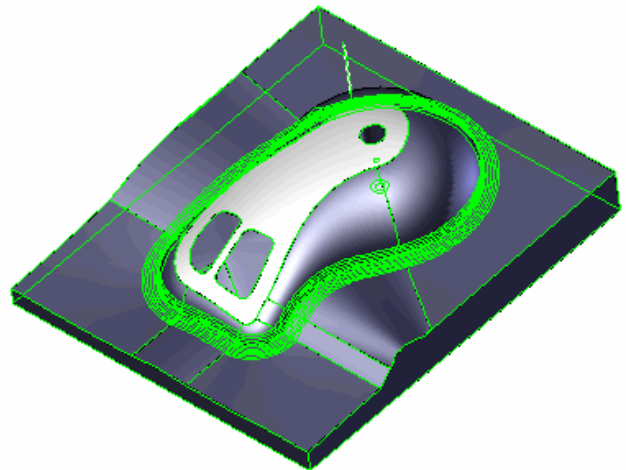
# 3 Axis Mill 5

For an Advanced 3 Axis Pattern Project operation, the Flowline pattern has unique characteristics that make it appropriate for machining areas that are defined using SolidWorks loft or sweep features.

The Flowline pattern produces toolpaths based on two user-specified curves. The toolpath can be generated parallel to (Along) or between (Across) the two curves. The curves can be either open or closed.

## What You'll Learn

- Generating Flowline machining toolpaths using curves to direct the flow of the toolpath
- Using contain and avoid areas to modify toolpaths

Define the Mill Part Setup and insert the Multi Surface feature:

1. Open the part file **MILL3AX_5.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ▣ Click the CAMWorks Feature Tree tab.

3. ⊥ Right click Example Mill–in in the tree and select Edit Definition.
   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. 📁 Right click the Stock Manager and select Edit Definition.

5. Click OK to use the default Stock for this exercise.

6. Right click the Stock Manager and select Insert Mill Part Setup.

   The Mill Part Setup dialog displays.

7. Select Plane2 from the list of Reference planes.

8. Select the Reverse direction option.

9. Make sure the Face option is not checked, then click OK.

10. Right click Mill Part Setup1 in the tree and select Insert Multi Surface Feature.

11. In the Insert Multi Surface Feature dialog box, click the Select all and associate option.

12. Set the Attribute to Fine, then click Insert and Close.

    This attribute has been set up in the TechDB to generate one Advanced 3 Axis Pattern Project operation.
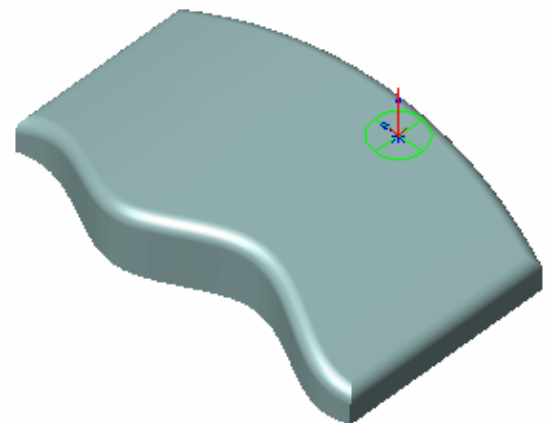
## Generating Toolpaths Using the Flowline Pattern

Generate an Operation Plan, modify parameters and generate toolpaths:

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

2. Right click Adv 3 Axis Pattern Project1 in the tree and select Edit Definition.

3. On the Pattern tab, set the Pattern to Flowline and select the Staydown option.

4. In the Curve section, set the Cut option to Along.

5. Click the Curve 1 button.

   The Curve 1 and Curve 2 buttons in this section are used to choose the sketch curves to direct the toolpath. Each of the 2 curves must be a unique sketch in SolidWorks.

6. In the 3 Axis Wizard: Select Geometry dialog box, select Flow 1 in the Available sketches list.

   Flow 1 displays in the Entities selected list and the curve is highlighted on the part.

   Optionally, you can add an offset to the curve if needed. For this exercise, an offset is not needed.
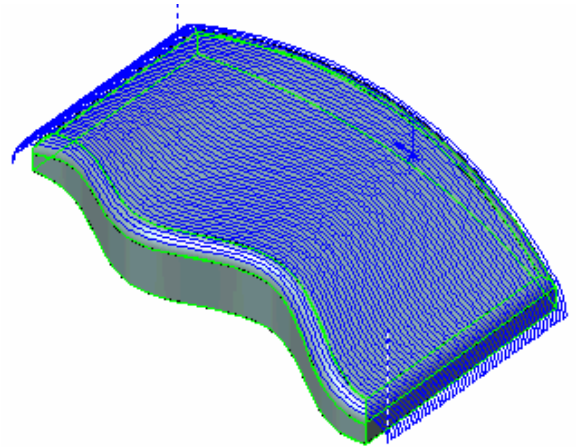
7. Click Finish.

8. Click the Curve 2 button.

9. In the 3 Axis Wizard: Select Geometry dialog box, select Flow 2 in the Available sketches list, then click Finish.
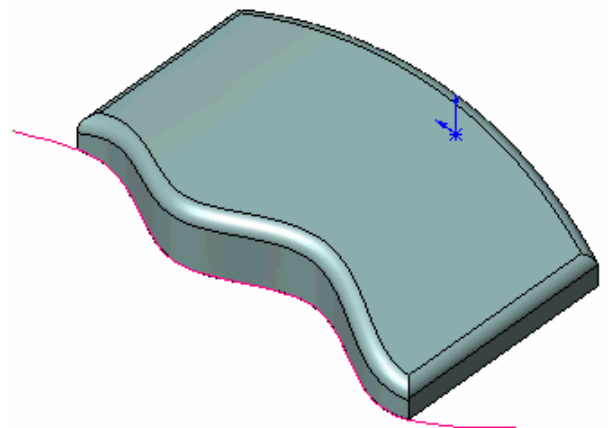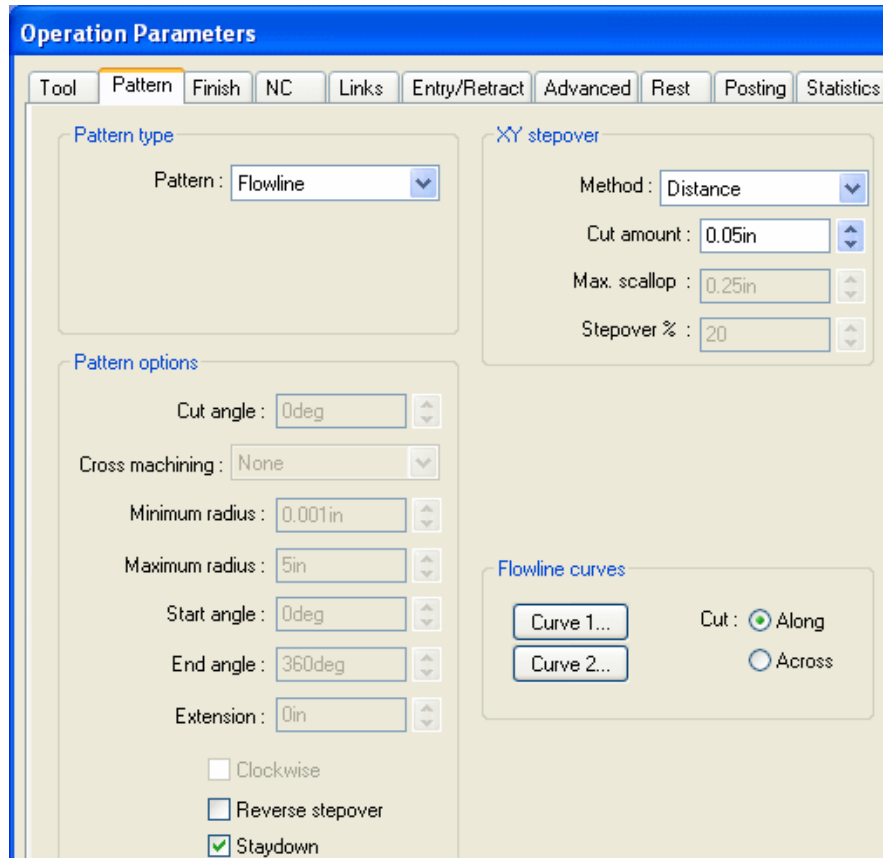
10. On the Finish tab, set the Mach. Deviation to .001in.

11. On the Links tab, set the Type to Spline Curve and the Max. link distance to 2in, then click OK.

12. ⬚ Click the Generate Toolpath button on the CAMWorks toolbar.

    The toolpath starts in the direction of the first curve, then translates to the shape of the second curve as it progresses.

## *Defining Contain and Avoid Areas to Modify Toolpaths*

Finish toolpaths for Multi Surface features can be modified to stay contained within an area or to avoid a specified area. Contain and avoid areas can be used to selectively machine within specified areas for re-machining or other purposes.

1. Right click Adv. 3 Axis Pattern Project1 in the tree and select Insert Contain Area.

2. In the 3 Axis Wizard: Select Geometry dialog box, select Contain in the Available sketches list, then click Finish.

3. Left click the plus sign (+) to the left of 3 Axis Finish Mill1 in the tree.

   The Contain Area item displays. If you needed to delete the contain area later, you would highlight and delete this item.

4. ⬚ Click the Generate Toolpath button on the CAMWorks toolbar.

   The toolpath is contained within the rectangular sketch that was chosen for the Contain Area. The toolpath is trimmed to the contain geometry. The radius of the tool would pass beyond this boundary. Use the Offset value when choosing the curve if this is not desirable.

5. Right click Adv. 3 Axis Pattern Project1 in the tree and select Insert Avoid Area.
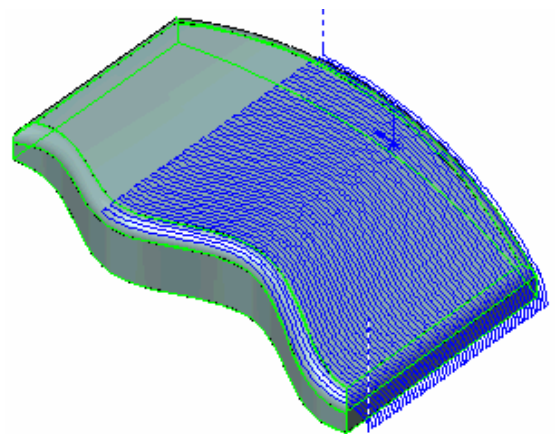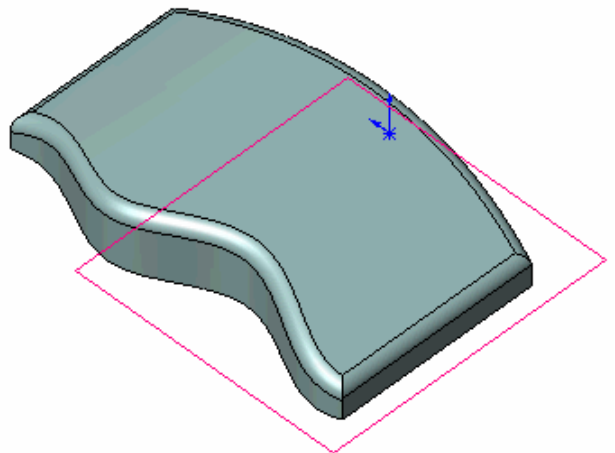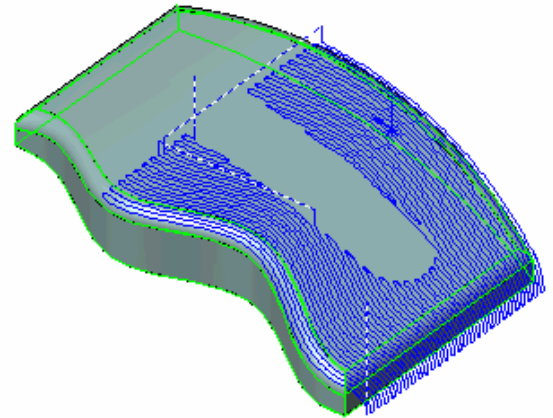
6. In the 3 Axis Wizard: Select Geometry dialog box, select Avoid in the Available sketches list, then click Finish.

7. Click the plus sign (+) to the left of 3 Axis Finish Mill1 in the tree.

   The Contain Area and Avoid Area items display.

8. ▯ Click the Generate Toolpath button on the CAMWorks toolbar.

   The toolpath is generated based on the contain and avoid areas.

# 3 Axis Mill 6

CAMWorks provides numerous patterns that can be used to finish or semi-finish a Multi Surface feature. Each pattern has unique characteristics. While you may not use these for general machining, occasionally you will machine a part that is appropriate for one of these styles. The following exercises explain how to generate radial, spiral and 2D stepover toolpaths..

## What You'll Learn

- Inserting operations.
- Generating spiral and radial finish toolpaths using the Adv. 3 Axis Pattern Project operation.
- Generating 2D stepover toolpaths using the Adv. 3 Axis Stepover operation.

Define the Mill Part Setup and insert the Multi Surface feature:

1. Open the Open the part file **MILL3AX_6.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Right click Example Mill–in in the tree and select Edit Definition.
   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. Right click the Stock Manager and select Edit Definition.

5. Click OK to use the default stock for this exercise.

6. Right click the Stock Manager and select Insert Mill Part Setup.

   The Mill Part Setup dialog box displays.

7. Select Plane1 from the list of Reference planes.

8. Make sure the Face option is not checked, then click OK.

9. Right click Mill Part Setup1 in the tree and select Insert Multi Surface Feature.

10. Click the Select all and associate option.

11. Set the Attribute to Fine, then click Insert and Close.
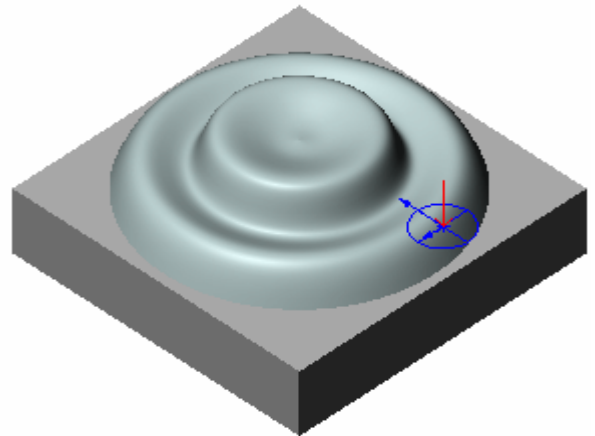
Generate the operation and change the Mill Part Setup origin:

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

2. Right click Mill Part Setup1 in the Operation tree and select Edit Definition.

3. On the Origin tab, change the Origin to Stock vertex and change the corner to the front bottom.

4. Click OK.
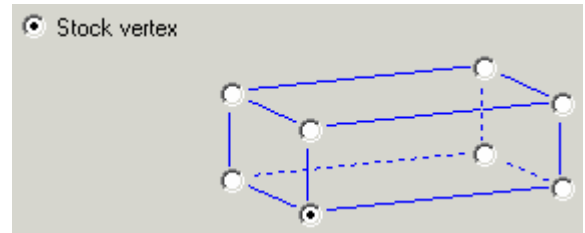
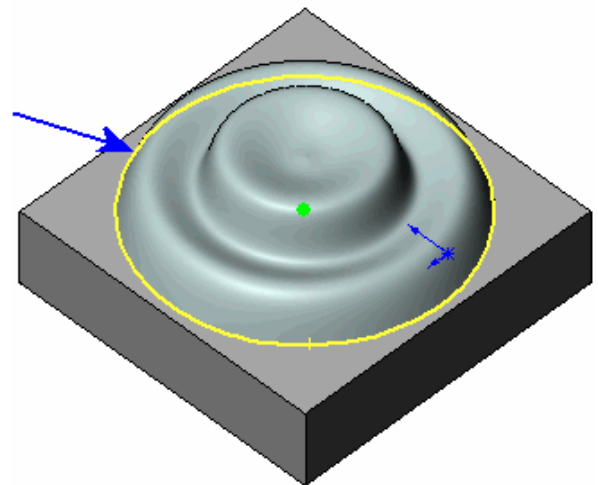## *Generating Radial Toolpaths Using Adv. 3 Axis Pattern Project*

1. Right click Adv 3 Axis Pattern Project1 in the tree and select Edit Definition.

2. Click the Tool Crib tab and select a .25in Ball Nose.

3. Click Yes to replace the corresponding holder.

4. On the Pattern tab, change the Pattern to Radial.

5. Check the Reverse stepover option to start the toolpath in the center.

   This option allows you to specify whether the tool will enter the material at the center or on the diameter of the radial pattern.

6. Change the XY Stepover Cut amount to .15in.

7. 🖱 Click the Pick Point button in the Center Point section.

   The Define Point dialog box displays. This dialog box allows you to define the center of the radial toolpath. You can define the applicable point relative and associative to a vertex, planar arc edge or arc segment defined in a sketch on the part.

8. Pick the bottom edge of the revolved boss.

   CAMWorks calculates the center point at the arc center.

9. Click OK.

10. On the Finish tab, change the Mach. deviation to .001in, then click OK.

11. Right click Adv 3 Axis Pattern Project1 in the tree and select Insert Contain Area.

12. Pick the bottom edge of the boss.

13. In the 3 Axis Wizard; Select Geometry dialog box, set the Tool condition to Past, then click Finish.

    This parameter defines how CAMWorks internally offsets the area/curve.

14. Right click Adv 3 Axis Pattern Project1 in the tree and select Generate Toolpath.

15. Right click the operation in tree and select Step Thru Toolpath.

16. ▶▶ Click the Forward Multiple Steps button several times and notice that the toolpath radiates from the center out.

17. ✕ Click the Close button to exit.
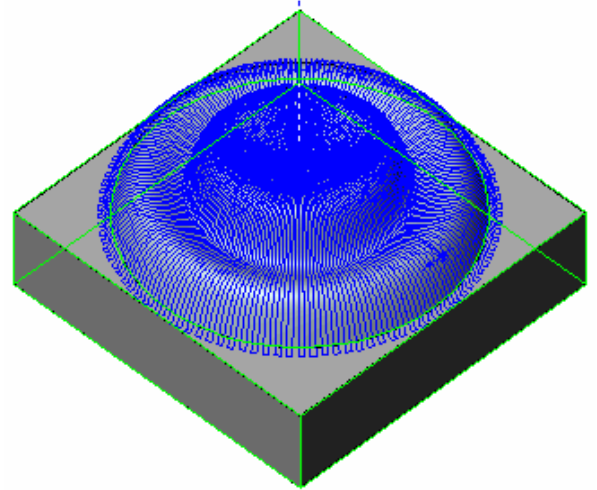
## *Generating Spiral Toolpaths Using Adv. 3 Axis Pattern Project*

Insert an Adv. 3 Axis Pattern Project operation to generate a Spiral toolpath:

1. Right click Adv 3 Axis Pattern Project1 in the tree and select Insert Multi Surface Operation.

2. In the Insert Operation dialog box, select Adv 3 Axis Pattern Project for the Type of operation.

3. Select Multi Surface Feature1 in the list box on the left and click the Add button.

4. Make sure the Edit definition option is checked, then click Insert.

5. On the Tool Crib tab, select a .25in Ball Nose.

6. Click Yes to replace the corresponding holder.

7. On the Pattern tab, set the following parameters:
   – Pattern = Spiral
   – Minimum radius = 0.01in (to start the spiral 0.01in from the center point)
   – Maximum radius = 2in (the overall size of the circle)
   – Reverse stepover = checked

8. 🔲 Click the Pick Point button in the Center Point section.

   The Define Point dialog box displays.

9. Pick the bottom edge of the revolved boss, then click OK.

   CAMWorks calculates the center point at the center of the circular boss.

10. On the Finish tab, make sure the Mach. deviation is .001in.

11. Set the Direction control to Zig.

12. Check the Reverse direction option to start the toolpath in the center, then click OK.

   This option allows you to specify whether the tool will enter the material at the center or on the diameter of the radial pattern.

13. Right click 3 Axis Pattern Project2 in the tree and select Insert Contain Area.

    The 3 Axis Wizard; Select Geometry dialog box displays.

14. Pick the bottom edge of the boss, set the Tool condition to Past, then click Finish.

15. Right click Adv 3 Axis Pattern Project2 in the tree and select Generate Toolpath.

    CAMWorks generates a spiraling circular toolpath starting from the center of the part. You can set the Minimum and Maximum radius in the Operation Parameters dialog box to limit the extents of the toolpath.
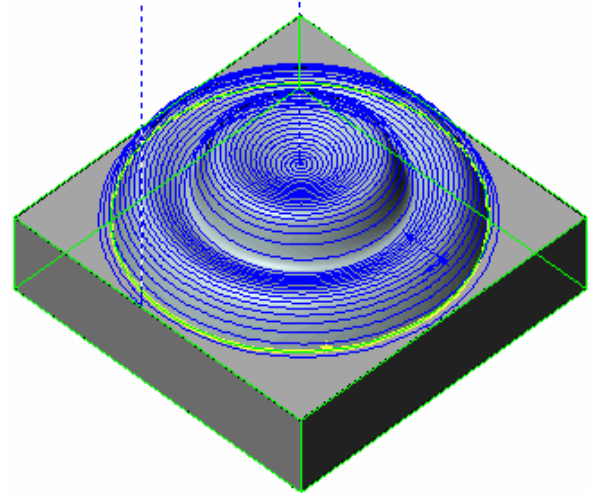
16. Right click 3 Axis Pattern Project2 in the tree and select Edit Definition.

17. On the Pattern tab, change the Minimum radius to .5in and the Maximum radius to 1.5in and click OK.

18. Regenerate the toolpath.

    CAMWorks generates the toolpath within the specified radii.

## *Generating 2D Stepover Toolpaths Using Adv. 3 Axis Stepover*

Insert an Advanced 3 Axis Stepover operation and generate the toolpath:

1. Right click Adv 3 Axis Pattern Project2 in the tree and select Insert Multi Surface Operation.

2. In the Insert Operation dialog box, select 3 Axis Stepover for the Type of operation.

3. Select Multi Surface Feature1 in the list box on the left and click the Add button.

4. Make sure the Edit definition option is checked, then click Insert.

5. On the Tool Crib tab, select a .25in Ball Nose.

6. Click Yes to replace the corresponding holder.

7. On the Stepover tab, if you have 3 Axis Mill Level II, set the Method to 2D Stepover. If you do not have 3 Axis Mill Level II, the Method is grayed out and 2D Stepover will be used.

    When 2D Stepover is selected, the distance between each pass will be constant and equal to the Horizontal cut amount, as viewed from the XY direction. This method produces a consistent surface finish on feature faces that do not have a steep slope angle. Because the stepover is constant in the XY plane, the surface finish on feature faces that are vertical or have a steep slope angle will not be as good.

8. Check the Number of cuts option and set the number to 12.

    This option allows you to specify whether the toolpath will continue until the entire part is machined or will be generated a specific number of passes.

9. Set the Horizontal and Vertical cut amt to 0.05in, then click OK.

10. Right click 3 Axis Stepover1 in the tree and select Insert Contain Area.

11. Pick the bottom edge of the revolved boss, set the Tool condition to Past, then click Finish.

12. Right click Adv 3 Axis Stepover1 and select Generate Toolpath.

CAMWorks generates the toolpath. Because the Number of cuts is set to 12 passes, the revolved surface is not machined completely.

If you had selected the 3D Stepover method, the toolpath would look like the figure on the right. This method generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope of the feature faces. In shallow areas the stepover will be equal to the Horizontal cut amount and in steeper areas the stepover will be equal to the Vertical cut amount. This method produces the most consistent surface finish on features whose faces have either shallow or steep slope angles. A Mill 3Ax Level II license is required for the 3D Stepover method.
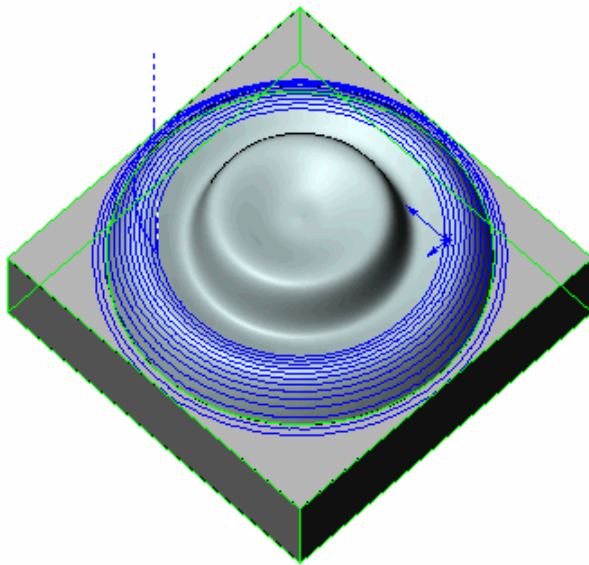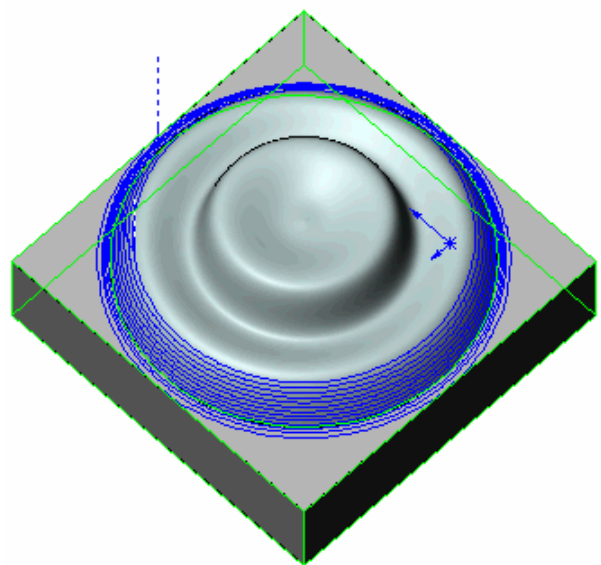
| | |
|---|---|
| 2D Stepover Toolpath | 3D Stepover Toolpath |

# Chapter 4  Learning More 3 Axis Mill

This chapter provides an opportunity to learn more about 3 Axis Mill. Stepover (3D Method), Curve Project, Flat Area and Rest Machining are Level II cycles. Your license must be programmed to support Mill 3x Level II.

We recommend that you go through the exercises in the previous chapters first. In addition to learning how to generate 2 and 3 axis mill toolpaths, these exercises explain basic functions and procedures that are used to generate both all toolpaths and NC code.

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

---

**IMPORTANT!**  The exercises in this chapter are intended to be used with the default Technology Database that is supplied with this version of CAMWorks. If you have modified the default Technology Database or imported customized data, the machining sequences and operations data in your database may be different.

If necessary, you can temporarily link to the default Technology Database using one of the following methods:

- Download the default CAMWorks Technology Database from the TekSoft web site (www.teksoft.com). Save it in the examples folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples*). Then run the Link Database Wizard, browse to the examples folder and select techdb.mdb. When you are finished with the exercises, run the Link Database Wizard and link to the TechDB that contains the customized data.

    or

- Reinstall CAMWorks. This reinstalls the default Technology Database. During installation, a backup is made of the techdb.mdb file that contains the customized data. When finished, move the backup to the active techdb.mdb location.

---

3 Axis Mill 7

# 3 Axis Mill 7

This exercise explains how to generate toolpaths for Adv. 3 Axis Rough rest machining, Adv. 3 Axis Z Level and Adv. 3 Axis Flat Area operations.

## What You'll Learn

- Generating Adv. 3 Axis Rough operations and toolpaths for rest machining.
- Generating Adv. 3 Axis Z Level and Adv. 3 Axis Flat Area operations and toolpaths for finish machining.

1. Open the part file **MILL3AXII_1.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ![icon] Click the CAMWorks Feature Tree tab.

3. ![icon] Right click Example Mill-in in the Feature tree and select Edit Definition.
   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. ![icon] Right click the Stock Manager and select Edit Definition on the shortcut menu.

   The default bounding box displays on the part.

5. Click OK to use the default stock for this exercise.

6. Right click Stock Manager in the Feature tree and select Insert Mill Part Setup on the shortcut menu.

7. Pick the top face on the model.

8. Make sure the Face option is not checked, then click OK.
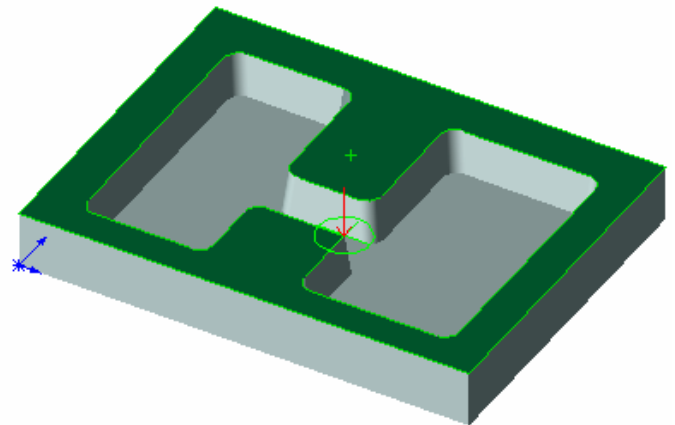
Define a Multi Surface feature:

1. Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

2. In the Insert Multi Surface Feature dialog box, click the Select all and associate button under the Action area.

   The Faces to cut box lists the selected faces.

3. Set the Attribute to Rgh-Lace, Fin-Zlevel, Fin – Flat Area. This attribute is defined in the Technology Database to generate Advanced 3 Axis Rough, Z Level and Flat Area operations.

4. Click Insert and Close. The Multi Surface feature is listed in the Feature tree.

4-2  Learning More 3 Axis Mill

> 💡 **Did You Know ...**
> When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

Generate the operation for the Multi Surface feature:

1. ▣↓ Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 in the Feature tree and select Generate Operation Plan.

2. Right click Adv. 3 Axis Rough1 in the Operation tree and select Edit Definition.

3. On the Pattern tab, use Lace for the Pattern and set the Cut angle to 90deg.

4. On the Rough tab, set the following parameters:
   - XY allowance = 0.15in
   - Z allowance = 0.15in
   - Depth parameter Method = Constant
   - Cut amt = .25in

5. On the Links tab, set the following parameters:
   - Links Type = Spline Curve
   - Corners Type = Round
   - Max. radius = .05in
   - Part deviation = .02in
   - Depth processing = By level

6. Click OK.

## Inserting an Adv. 3 Axis Rough Operation for Rest Machining

Insert an Advanced 3 Axis Rough operation to remove remaining material.

1. Right click Adv 3 Axis Rough1 in the tree and select Insert Multi Surface Operation.

2. For the Type of operation, select Adv 3 Axis Rough.

3. Pick Multi Surface Feature1 in the Features list and click Add.

4. Leave the Edit Definition option checked, and click Insert.

5. Click the Tool Crib tab and select a .5in diameter Flat End mill.

6. Click Yes to replace the corresponding holder.

7. On the Pattern tab, use Pocket Out for the Pattern.

8. On the Rough tab, set the following parameters:
   - XY allowance / Z allowance= 0.05in
   - Depth parameter Method = Constant
   - Cut amt = .125in

9. On the Links tab, set the following parameters:
   - Links Type = Spline Curve
   - Corners Type = Round
   - Max. radius = .05in
   - Part deviation = .02in
   - Offset deviation = .02in
   - Depth processing = By level

10. On the Entry/Retract tab, use Spiral for the Ramping Method and set the Max. ramp angle to 20deg and the Spiral radius to .15in.

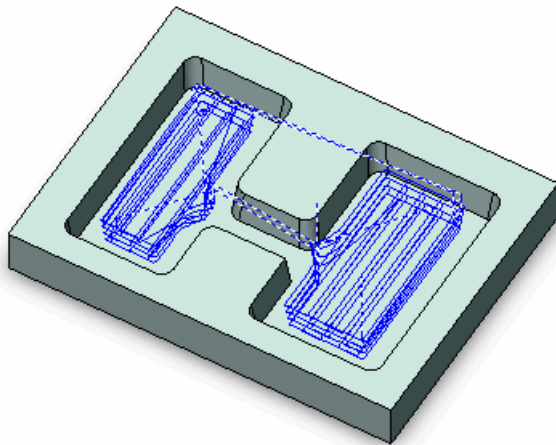11. On the Rest tab, set the Rest machining Method to From WIP, then click OK.

   When you select this method, CAMWorks assumes that toolpaths should be generated only where material is left by previous operations. CAMWorks creates a work in process stock model by starting with the original stock definition and removing material for the selected operations that occur before the rest machining operation. The work in process is then used to calculate the areas where the rest machining toolpaths should be generated. This method gives good results when used for roughing, where accuracy is not as important as speed.

## *Generating Adv. 3 Axis Roughing Toolpaths*

Generate toolpaths and simulate material removal:

1. Right click Adv 3 Axis Rough1 in the tree and select Generate Toolpath.
2. Right click Adv 3 Axis Rough2 in the tree and select Generate Toolpath.
3. Notice the rest machining toolpath removes material left by the previous operation.

| Adv. 3 Axis Rough1 Toolpath | Adv. 3 Axis Rough2 Rest Machining Toolpath |
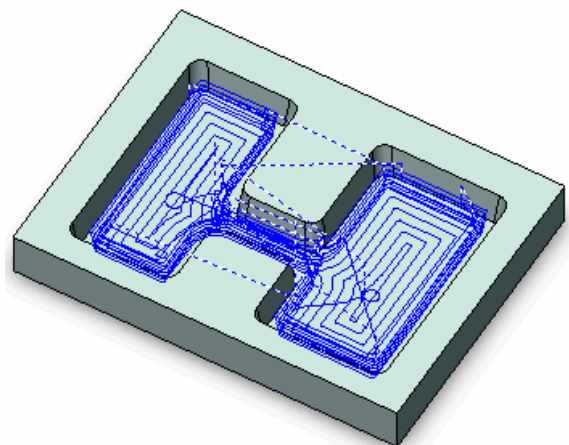


4. Click the Simulate Toolpath button on the CAMWorks toolbar.
5. Make sure Tool mode is selected.

6. Set the End condition to Next Operation.

7. ▶ Click Run to simulate the first operation.

8. ▶ Click Run again to simulate the rest machining operation.

9. ☒ Click the X button to exit the simulation.

## *Using an Automatic Contain Area*

Edit the Adv. 3 Axis Z Level operation and generate toolpaths to machine the sides of the pockets:

1. Right click Adv 3 Axis ZLevel1 in the tree and select Edit Definition.

2. Click the Tool Crib tab and select a Flat End Tapered tool with a .3125in diameter.

3. Click Yes to replace the corresponding holder.

4. On the Tapered Tool tab, change the Taper angle to 4deg.

5. On the Links tab, set the Depth processing to By level.

6. On the Advanced tab, set the Automatic contain area Method to Outer Silhouette.

With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature. Note that faces to avoid in the Multi Surface feature are not used when calculating the silhouette.

7. Click OK.

8. Right click Adv 3 Axis Z Level1 in the tree and select Generate Toolpath on the shortcut menu.

CAMWorks generates the Z level toolpaths.
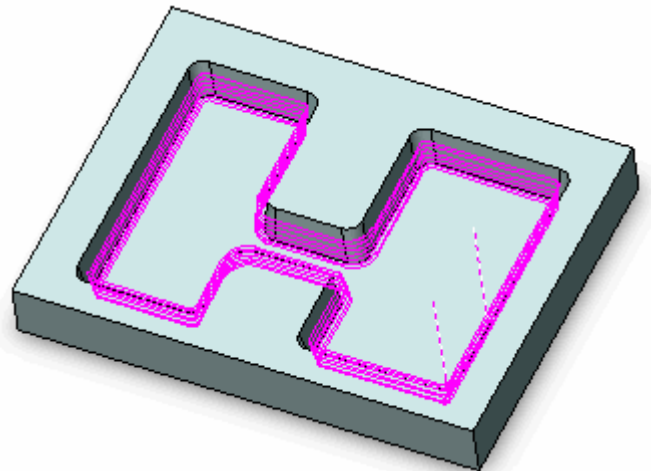
> 💡 *Did You Know ...*
>    For Advanced 3 Axis operations, the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either automatically on the Advanced tab in the Operation Parameters dialog box or interactively using the Insert Contain Area command.
>    The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or not defined, then an automatically generated XY contain area will be used.

## *Removing Material on Flat Areas*

The Advanced 3 Axis Flat Area cycle uses a roughing pattern to remove material on the z-levels of flat faces between the top and bottom of the surface. CAMWorks generates toolpaths only on completely flat areas. If a surface has even a small gradient, CAMWorks will not generate a toolpath. This cycle can be used for finishing where excess material has already been cleared.

Edit the Adv. 3 Axis Flat Area operation and generate toolpaths to machine the bottom of the pockets:

1.  Right click Adv 3 Axis Flat Area1 in the tree and select Edit Definition.

2.  On the Flat Area tab, set the First cut from to Top of stock with a -0.05in Offset.

3.  On the Links tab, set the following parameters:

    – Links Type = Spline Curve
    – Corners Type = Round
    – Part deviation = .02in
    – Offset deviation = .02in
    – Depth processing = By level

4.  On the Entry/Retract tab, set the Max. ramp angle to 20deg and click OK.

5.  Right click Adv 3 Axis Flat Area1 in the tree and select Generate Toolpath on the shortcut menu.
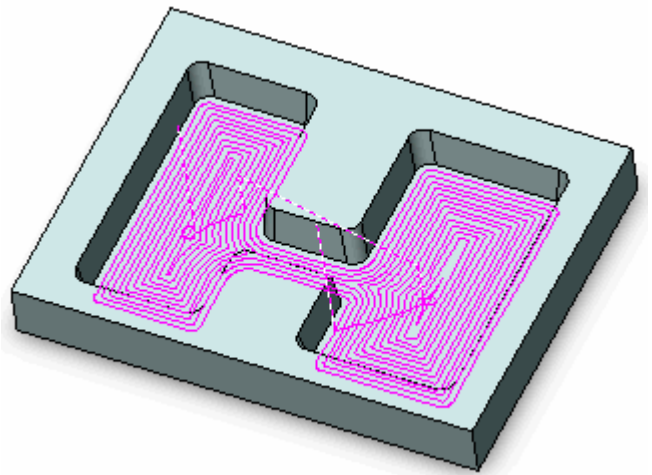
Run the simulation:

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

2.  Make sure Tool mode is selected.

3.  Set the End condition to Next Operation.

4.  Click Run to simulate the first operation.

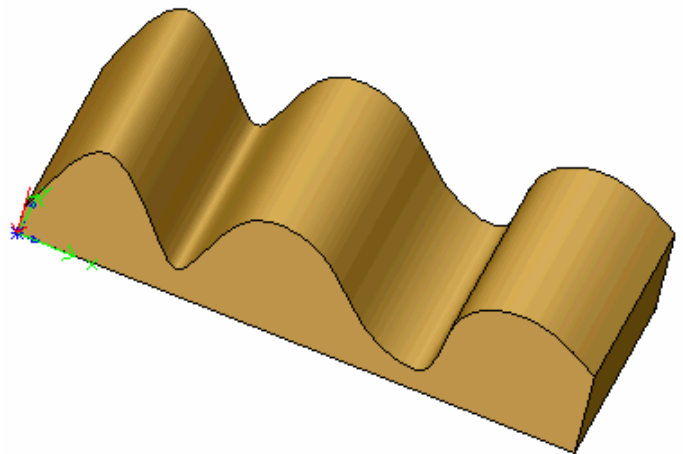5.  Click Run again to simulate each operation.
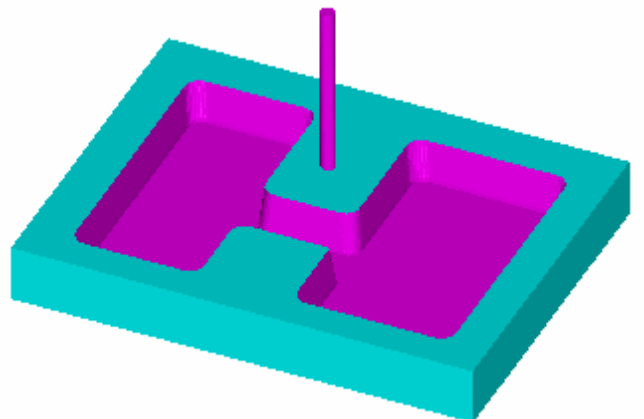
# 3 Axis Mill 8

This exercise introduces you to finish rest machining using an STL file as the stock model. The STL file can be created from an external CAD system or a simulated stock model as explained in this exercise.

## What You'll Learn

- Generating Advanced 3 Axis Pattern Project toolpaths
- Creating and Using an STL file as the stock model for rest machining
- Generating Advanced 3 Axis Pattern Project toolpaths for rest machining

1. Open the part file **MILL3AXII_2.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Right click Example Mill-in in the Feature tree and select Edit Definition.

   – On the Machine tab, Example Mill–in is the Active machine.
   – Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   – Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. Right click the Stock Manager and select Edit Definition on the shortcut menu.

   The default bounding box displays on the part.

5. Click OK to use the default stock for this exercise.

6. Right click Stock Manager in the Feature tree and select Insert Mill Part Setup on the shortcut menu.

7. Pick the Front plane and select Reverse direction.

8. Make sure the Face option is not checked, then click OK.

   Mill Part Setup1 is added to the tree.

Define a Multi Surface feature:

9. Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

10. In the Insert Multi Surface Feature dialog box, click the Select all and associate option.

The Faces to cut box lists the selected faces.

💡 *Did You Know ...*

When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

11. Set the Attribute to Fine. This attribute is defined in the Technology Database to generate an Advanced 3 Axis Pattern Project finish operation.

12. Click Insert and Close.

The Multi Surface feature is listed in the Feature tree.

## *Generating an Operation Plan and Modifying Parameters*

Generate machining operations for the Multi Surface feature:

1. [icon] Click the Generate Operation Plan button on the CAMWorks toolbar or right click Mill Part Setup1 in the Feature tree and select Generate Operation Plan.

Modify operation parameters:

2. Right click Adv 3 Axis Pattern Project1 in the Operation tree and select Edit Definition.

3. Click the Tool Crib tab.

4. Select a 1in diameter Flat End mill, then click the < - Select button.

5. Click Yes to replace the corresponding holder.

6. On the Pattern tab, use Slice for the Pattern and set the Cut amt to .15in.

This pattern, which generates a series of linear parallel cuts across the part, is appropriate for semi-finishing almost any part shape and finishing non-vertical areas.

7. On the Finish tab, make sure the XY and Z allowances are 0in.

8. On the Links tab, set the Type to Spline Curve.

9. On the Advanced tab, set the Automatic contain area Method to Outer Silhouette.

For this exercise, you use this contain area method to machine only the top of the feature.

10. Click OK.

## Generating Toolpaths

Generate toolpaths, simulate material removal and post process:

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

2. ⬚ Click the Simulate Toolpath button on the CAMWorks toolbar.

3. ⬚ Click the Turbo mode button on the Simulate Toolpath toolbar.

   Turbo mode provides ultra-fast simulation of toolpaths on a single Mill Part Setup.
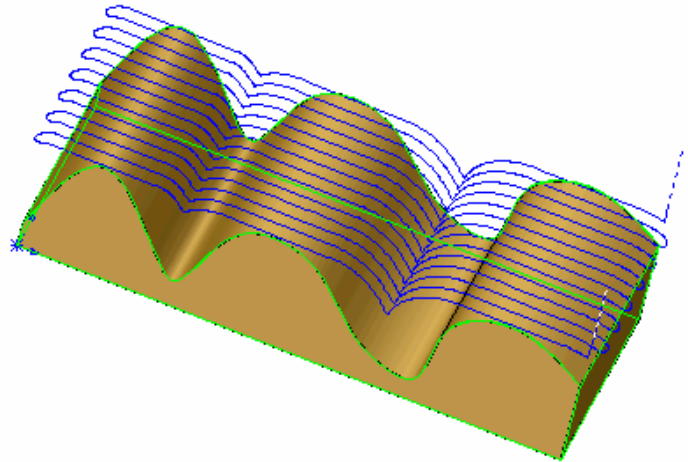
4. ⬚ Click Run.

5. ⬚ Click the Save STL button on the Simulate Toolpath toolbar.

6. In the Save As dialog box, select Inches for the Units, then Click OK.

   The WIP is saved in an STL file that you will use for rest machining.

7. ⬚ Click the X button to exit the simulation.

   💡 *Did You Know ...*
   You can also save an STL file using the Save WIP Model command on the Stock Manager shortcut menu in the Operation tree. With this method, you select the operations to be used in calculating the WIP from a list, then define the file name and applicable options. The Toolpath Simulation runs in the background to generate the STL file.

## Inserting an Operation for Rest Machining

1. Right click the operation in the tree and select Insert Multi Surface Operation on the shortcut menu.

2. For the Type of operation, click the down arrow and select Adv 3 Axis Pattern Project.

3. Highlight Multi Surface Feature1 in the Feature list and click the Add button.

4. Leave the Edit Definition option checked and click Insert.

5. Click the Tool Crib tab.

6. Select a .25in diameter Ball Nose, then click the < - Select button.

7. Click Yes to replace the corresponding holder.

8. On the Pattern tab, use Slice for the Pattern with a .05in Cut amt and make sure the Staydown option is checked.

9. On the Finish tab, make sure the XY and Z allowances are 0in and the Avoid allowance is 0.025in.

10. On the NC tab, change the Rapid plane to .25in.

11. On the Links tab, set the Type to Spline Curve and the Max. link distance to .1in.

12. On the Entry/Retract tab, set the Ramp clearance to .1in.

13. On the Advanced tab, set the Automatic contain area Method to Outer Silhouette.

14. Click the Rest tab.

15. For the Method, click the down arrow and select From STL File.

16. ⬚ Click the Browse button.

17. In the Open dialog box, locate the STL file you created, select Inches for the Units, then click Open.

18. Click OK to close the Operation Parameters dialog box.

## *Viewing the STL/WIP Model*

To view the shape and position of the STL stock model, you can:

- Click the Stock Manager in the tree to view a wireframe bounding box indicating the extents of the stock model.
- Click the Simulate Toolpath button on the CAMWorks toolbar.
- Add the STL file as a Graphics Body to the SolidWorks tree as explained below.

1. Click the SolidWorks FeatureManager design tree button.

2. Select Insert on the SolidWorks menu bar, select Features, then pick Imported on the cascading menu.

3. Set the file type to .stl.

4. Locate and select the STL file you are using for the WIP model.

5. Click the Options button.

6. In the Import as group box, select Graphics Body and set the Unit to Inches, then click OK.

7. Click Open.

   The STL file is added to the SolidWorks FeatureManager design tree as a graphics body. This body can be hidden, made transparent or a different color, etc.
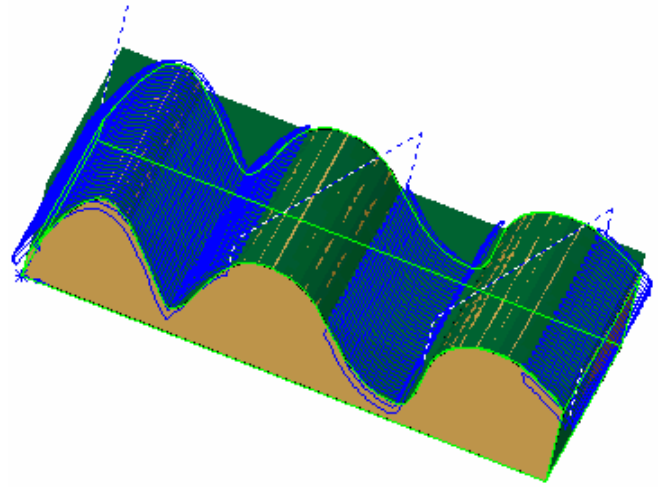
8. Click STL Graphics1 in the design tree and rotate the part to see the stock model.

9. ⬚ Click the CAMWorks Operation Tree tab.
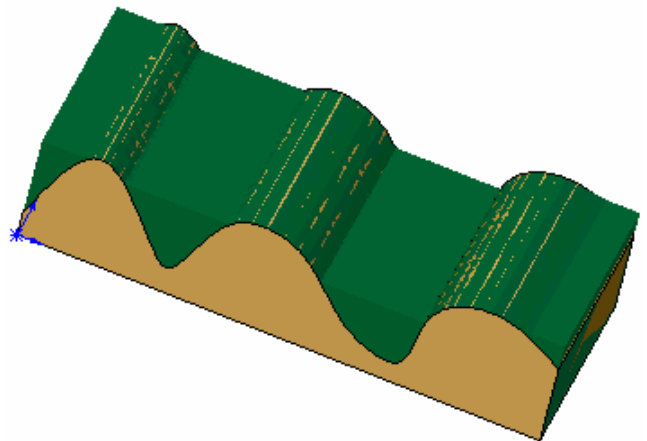
## *Generating the Rest Machining Toolpaths*

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

2.  CAMWorks generates toolpaths only where material is left on the stock model in the STL file.

# 3 Axis Mill 9

This exercise demonstrates how to engrave using an Advanced 3 Axis Curve Project cycle. This cycle removes material by projecting selected 2.5 Axis Engrave and/or Curve features onto the surface of a Multi Surface feature and generating toolpaths along the projected entities.

## What You'll Learn

- Defining a Multi Surface Feature and a 2.5 Axis Engrave Feature
- Inserting a Curve Project operation and adding an Engrave Feature
- Generating Advanced 3 Axis Curve Project toolpaths for engraving

1. Open the part file **MILL3AXII_3.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

Define the machine:

3. Right click Example Mill-in in the Feature tree and select Edit Definition.
   - On the Machine tab, Example Mill–in is the Active machine.
   - Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   - Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

Define the stock size and shape:

4. Right click the Stock Manager and select Edit Definition on the shortcut menu.
   The default bounding box displays on the part.

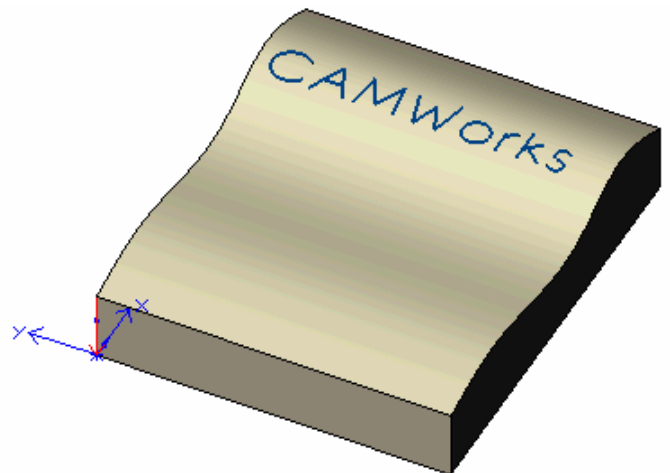5. Click OK to use the default stock for this exercise.

Define the machining direction:

6. Right click Stock Manager in the tree and select Insert Mill Part Setup on the shortcut menu.

7. Pick the Plane2 and select Reverse direction.

8. Make sure the Face option is not checked, then click OK.
   Mill Part Setup1 is added to the tree.

Define a Multi Surface feature:

9. Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

10. In the Insert Multi Surface Feature dialog box, click the Select all and associate option.

    The Faces to cut box lists the selected faces.

    > 💡 ***Did You Know ...***
    > When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

11. Set the Attribute to Fine. This attribute is defined in the Technology Database to generate an Advanced 3 Axis Pattern Project finish operation.

12. Click Insert and Close. The Multi Surface feature is listed in the Feature tree.

13. Right click Mill Part Setup1 and select Insert 2.5 Axis Feature on the shortcut menu.

14. Click the down arrow next to the Feature Type and select Engrave Feature.

15. Select Sketch2 and click Next.

16. In the 2.5 Axis Feature Wizard: End Conditions dialog box, set the Depth to .1 and click Finish.

17. Click Close.

## *Generating an Operation Plan and Modifying Parameters*

Generate machining operations for the Multi Surface feature:

1. Right click Multi Surface Feature1 in the tree and select Generate Operation Plan on the shortcut menu.

2. Right click Adv Pattern Project1 in the Operation tree and select Insert Contain Area on the shortcut menu.

3. Pick the top face on the model and click Finish.

Modify operation parameters:

4. Right click the operation again and select Edit Definition on the shortcut menu.

5. Click the Tool Crib tab and select a .25in Ball Nose.

6. Click Yes to replace the corresponding holder.

7. On the Pattern tab, set the following parameters:
   − Pattern = Slice
   − Cut angle = 90 deg
   − Cut amount = 0.1in.

8.  On the Finish tab, set the set the following parameters:
    - XY allowance = 0in
    - Z allowance = 0in
    - First cut from = Top of stock with 0in Offset

9.  On the Links tab, set the Max link distance to 1in.

10. On the Advanced tab, set the Automatic contain area Method to Outer Silhouette.

11. With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature. Note that faces to avoid in the Multi Surface feature are not used when calculating the silhouette.

12. Click OK.

## *Inserting an Adv. 3 Axis Curve Project Operation to Cut the Text*

1.  Right click the operation in the tree and select Insert Multi Surface Operation on the shortcut menu.

2.  For the Type of operation, click the down arrow and select Adv 3 Axis Curve Project.

3.  Highlight Multi Surface Feature1 in the Features list and click the Add button.

4.  Leave the Edit Definition option checked and click Insert.

5.  On the Mill Tool tab, change the Cut diameter to 0.063in.

6.  On the Curve Project tab, set the following parameters:
    - XY allowance = 0in
    - Z allowance = -0.05in
    - Number of cuts = 1
    - Axial offset = .1in

7.  On the Links tab, set the Max. link distance to 0in.

8.  On the Entry/Retract tab, set the Retract Method to Full.

    With this Retract method, the tool moves vertically up to the clearance plane, moves horizontally along this level, and drops down vertically to the start of the next pass.

9.  For the Vertical and Horizontal leadin and leadout, set all the radius values to 0in.

10. Click OK to close the Operation Parameters dialog box.

11. Right click Adv 3 Axis Curve Project1 in the tree and select Add Engrave/Curve Feature.

12. In the Engrave/Curve Feature dialog box, pick Engrave Feature1 in the Available features list and click the Add button.

13. Click OK.

## *Generating Toolpaths and Running Toolpath Simulation*

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

    CAMWorks generates toolpaths for the Multi Surface feature and the text.
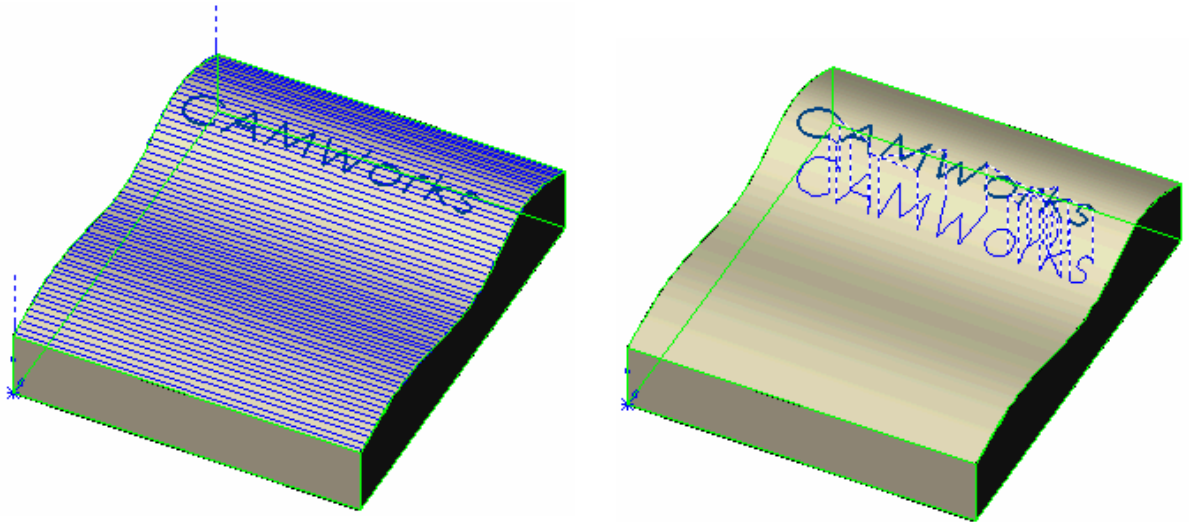
2.  Click the Simulate Toolpath button on the CAMWorks toolbar.

3.  Make sure Tool mode is selected.

4.  Click Run.
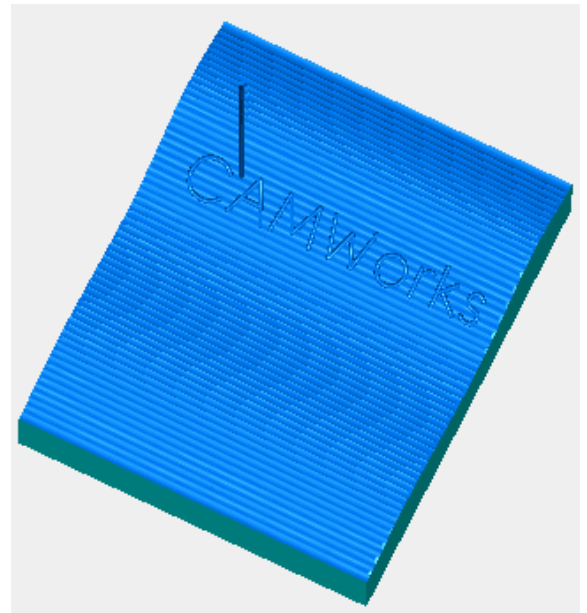
# 3 Axis Mill 10

This exercise uses an Advanced 3 Axis Stepover operation to remove material by maintaining a constant user-defined stepover relative to the surface. This cycle generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope being machined.

## What You'll Learn

- Defining a Multi Surface Feature
- Inserting an Advanced 3 Axis Stepover operation
- Adding contain and avoid areas
- Generating toolpaths

1. Open the part file **MILL3AXII_4.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ![CW] Click the CAMWorks Feature Tree tab.

3. ![icon] Right click Example Mill-mm in the Feature tree and select Edit Definition.

   – On the Machine tab, Example Mill–mm is the Active machine.
   – Click the Tool Crib tab and make sure Crib 1 is the Active tool set.
   – Click the Controller tab and make sure FANTUTM is selected, then click OK to close the Machine dialog box.

4. ![icon] Right click the Stock Manager and select Edit Definition on the shortcut menu.

   The default bounding box displays on the part.

5. Click OK to use the default stock for this exercise.

6. Right click Stock Manager in the Feature tree and select Insert Mill Part Setup on the shortcut menu.

7. Pick the top face on the part model.

8. Make sure the Face option is not checked, then click OK.

   Mill Part Setup1 is added to the tree.

9. Right click Mill Part Setup1 and select Insert Multi Surface Feature on the shortcut menu.

10. In the Insert Multi Surface Feature dialog box, click the Select all and associate option. The Faces to cut box lists the selected faces.
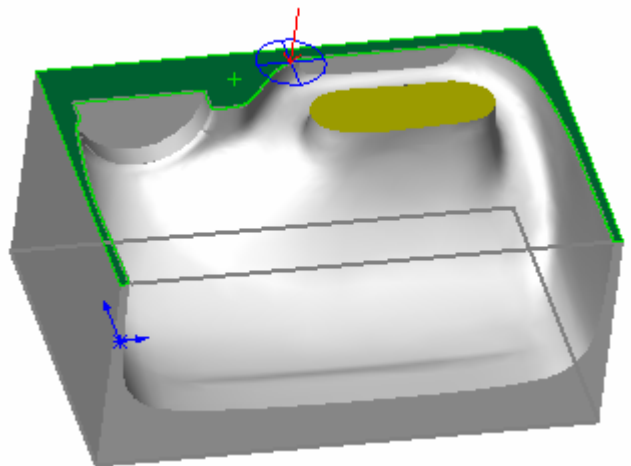
11. Click Insert and Close.

The Multi Surface feature is listed in the Feature tree.

💡 **Did You Know ...**

When you define Multi Surface features for Advanced 3 Axis operations, the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the Insert Contain and Insert Avoid commands at the operation level.

## Inserting an Advanced 3 Axis Stepover Operation

Insert an Advanced 3 Axis Stepover operation:

1.  🔳 Click the CAMWorks Operation Tree tab.

2.  Right click Mill Part Setup1 in the tree and select Insert Multi Surface Operation on the shortcut menu.

3.  For the Type of operation, click the down arrow and select Adv 3 Axis Stepover.

4.  Highlight Multi Surface Feature1 in the Features list and click the Add button.

5.  Leave the Edit Definition option checked and click Insert.

6.  On the Stepover tab, set the following parameters:
    - Method = 3D
    - Horizontal cut amt. = 2mm
    - Vertical cut amt. = 2mm
    - Inside to out = checked
    - First cut from = Top of Stock with a 2.5mm Offset

7.  Click OK.

## Inserting Contain and Avoid Areas

1.  Right click Adv Stepover1 in the tree and select Insert Contain Area on the shortcut menu.

2.  In the 3 Axis Wizard: Select Geometry dialog box, pick ContainArea, set the Tool condition to On and click Finish.

3.  Right click Adv Stepover1 in the tree again and select Insert Avoid Area on the shortcut menu.

4.  Pick the gold face on the part, set the Tool condition to On and click Finish.

5.  Click the + sign next to Adv Stepover1 in the tree. Contain Area1 and Avoid Area1 are listed under the operation.

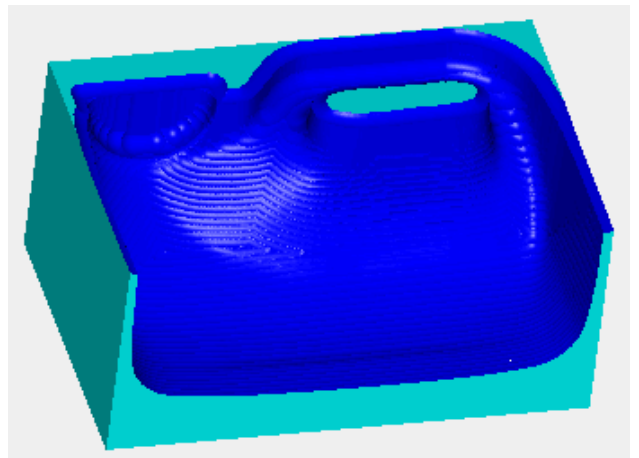Pick face to avoid
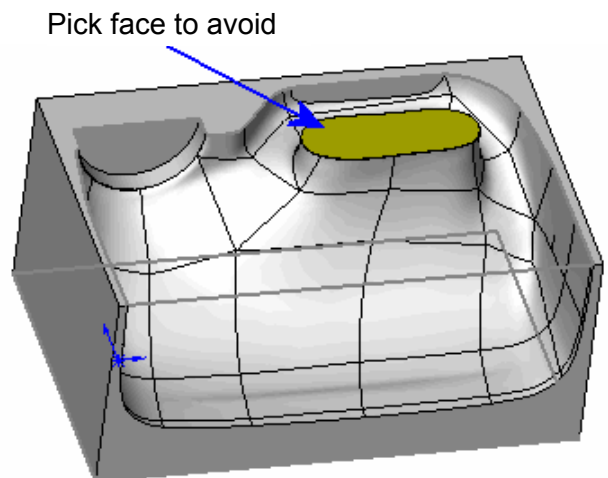
Generate and simulate the toolpath:

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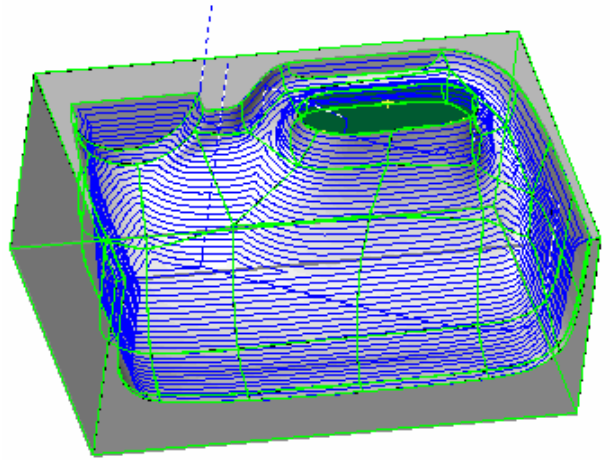


2.  Click the Simulate Toolpath button on the CAMWorks toolbar.

3.  Click Turbo mode.

4.  Click Run.

# 3 Axis Mill 11

For Advanced 3 Axis operations, the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either interactively using the Insert Contain Area command or automatically on the Advanced tab in the Operation Parameters dialog box as shown in this exercise.

The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or if none have been defined, then an automatically generated XY contain area will be used.

## What You'll Learn

### *Using Automatic Contain Areas*

1. Open the part file **MILL3AXII_5.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ![icon] Click the CAMWorks Operation Tree tab at the bottom of the split FeatureManager design tree.

   An Adv. 3 Axis Z Level operation has already been generated for the Multi Surface feature.

**Stock Method**

3. Right click Adv 3 Axis Z Level1 in the tree and select Edit Definition.

4. Click the Advanced tab and notice that the Automatic contain area Method is set to Stock.

   When you use this method, the XY shape of the contain area is based on the original stock shape. If the stock shape is defined from a sketch and the current Setup is normal to the sketch, the contain area will be the same as the sketch. Otherwise, the contain area will be based on the minimum XY bounding box of the stock.

5. Click OK.

6. Right click the operation in the tree and select Generate Toolpath.

7. In the upper portion of the split tree, scroll down in the FeatureManager design tree and position the pointer over the Stock sketch item.

   The red sketch that displays represents the stock shape.
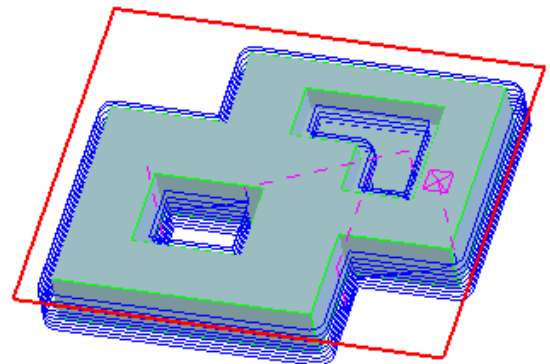
## Bounding Box Method

Edit the operation and change the Automatic contain area method to Bounding Box:

1. Right click Adv 3 Axis Z Level1 in the tree and select Edit Definition.

2. On the Advanced tab, change the Automatic contain area Method to Bounding Box.
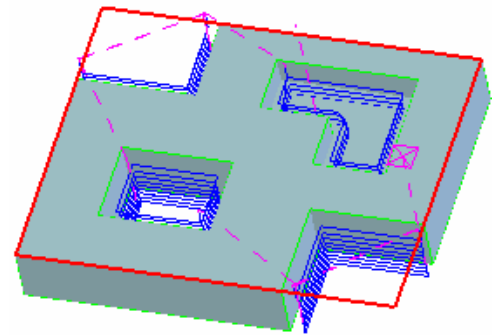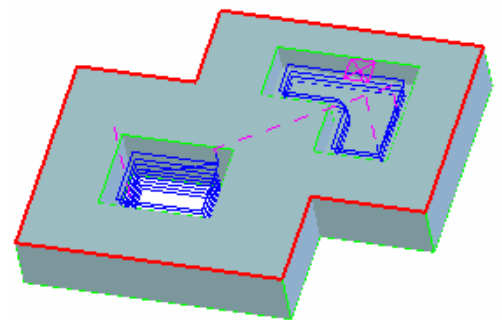
   With this method, the XY shape of the contain area is based on the minimum XY bounding box of the faces in the Multi Surface feature. The bounding box is based only on the Faces to Cut in the Multi Surface feature.

3. Click OK.

   When you click OK, the *Update 3 axis toolpaths when parameters change* option on the Update tab in the Options dialog box controls if 3 axis toolpaths are updated automatically. If this option is set to Prompt, a message displays asking if you want to regenerate toolpaths. Click Yes. If set to Always, CAMWorks automatically regenerates the toolpaths. If set to Never, right click the operation in the tree and select Generate Toolpath on the shortcut menu.

4. In the upper portion of the split tree, scroll down and position the pointer over the Bounding Box sketch item.

   The red sketch that displays represents the bounding box.

## Outer Silhouette Method

Edit the operation and change the Automatic contain area method to Outer Silhouette:

1. Right click Adv 3 Axis Z Level1 in the tree and select Edit Definition.

2. On the Advanced tab, change the Automatic contain area Method to Outer Silhouette, then click OK.

   With this method, the XY shape of the contain area is based only on an outer XY silhouette of the Faces to Cut in the Multi Surface feature.

3. Generate the toolpath.

4. In the upper portion of the split tree, scroll down and position the pointer over the Outer Silhouette sketch item.

   The red sketch that displays represents the outer silhouette.

## All Silhouettes Method

Edit the operation and change the Automatic contain area method to All Silhouettes:

1. Right click Adv 3 Axis Z Level1 in the tree and select Edit Definition.

2. On the Advanced tab, change the Automatic contain area Method to All Silhouettes, then click OK.

   With this method, the XY shape of the contain area is based on an XY silhouette of Faces to Cut in the Multi Surface feature. The silhouette will include the outer shape of the faces as well as any internal shape based on through features. The internal silhouette areas will be treated automatically as avoid areas.

3. Generate the toolpath.

4. In the upper portion of the split tree, scroll down and position the pointer over the All Silhouettes sketch item.

   The red sketch that displays represents the outer silhouette of the Faces to Cut in the Multi Surface feature and the internal silhouette for the through feature.
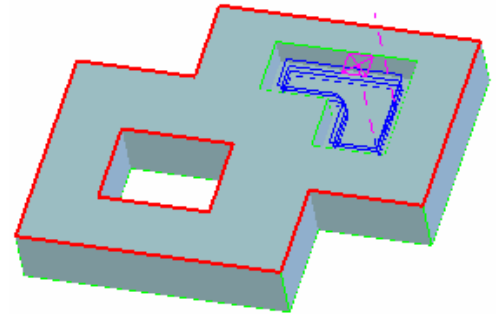
# Chapter 5  Learning 2 Axis Turn

This chapter provides an opportunity to learn CAMWorks 2 Axis 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.

**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 exactly 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.

# Turn 1

## What You'll Learn

### *Steps to Generate Turn Toolpaths and NC Code*

The following steps are used to generate 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. Edit the Stock definition.
5. Define machinable features and adjust feature parameters.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Post process the toolpaths.

The next series of exercises show you how to generate finish toolpaths on a SolidWorks part model. In order to give you a general understanding of how to use CAMWorks, you work with a part that was previously modeled in SolidWorks. When you define the operations and toolpaths, you will follow steps that are not explained in depth. This is done to show you the basics of generating toolpaths from start to finish without getting into the details at this time.

Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically.

## *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. This exercise uses an existing SolidWorks part.

1. Open the part file **TURN2AX_1.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

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

   The tabs at the bottom or top of the tree are for moving between the SolidWorks trees and the CAMWorks trees.
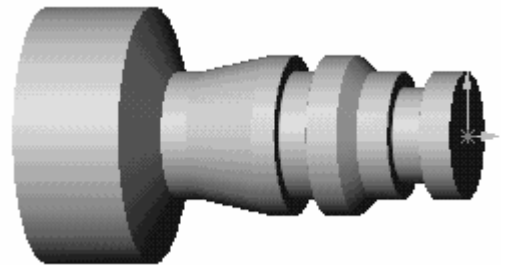
## *Step 2: Change to CAMWorks Feature Tree*

1. Click the CAMWorks Feature Tree tab.

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

   👉 The icons that display for the Stock Manager and Machine may indicate that a mill machine is currently selected. Step 3 on page 4-5 explains how to change the machine to a turn machine.

   If you work mainly with turn parts, you can set the default to Turn in the TechDB so that CAMWorks will select a Turn machine and stock when you open part files. To change the default, click the TechDB button on the CAMWorks toolbar. Under the navigation tree in the Technology Database – Main Menu window, change the Appln default to Turn and click Quit. When you open a part file, CAMWorks will select a Turn machine.
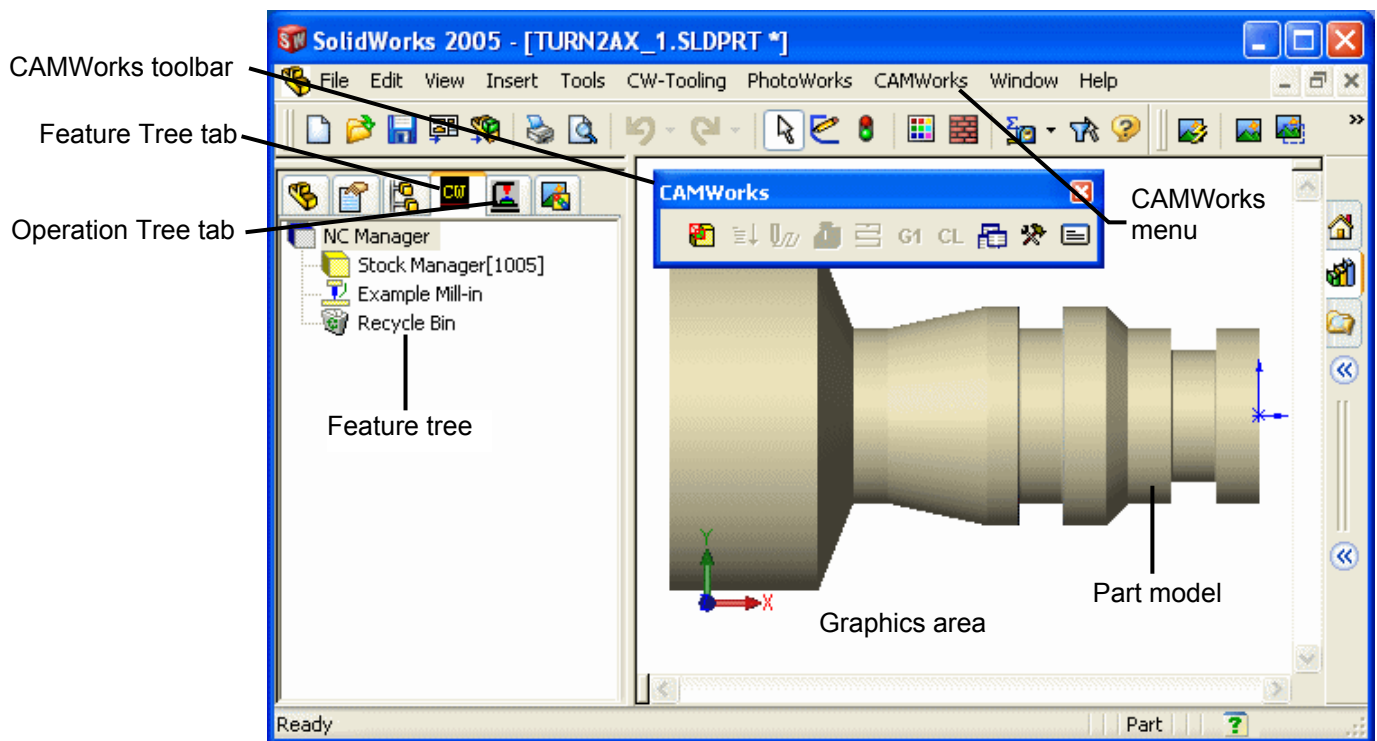
## CAMWorks Machining Trees

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

- ◇ **Turn Stock Manager**     🟨 **Mill Stock Manager**

  The turn stock is the material from which the part will be machined. You can define the stock as a cylinder (for bar stock) or as a closed sketch (for a forging or casting) and specify the type of material.

- ▫ **Turn**     🔩 **Mill**     ▫ **Mill-Turn**     🔲 **Wire EDM**

  The Machine item defines the machine tool that will be used to machine the part. The machine definition includes tool definitions and the post processor. The machines are set up in the Technology Database.

- 🗑 **Recycle Bin**

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

## CAMWorks Menu

1. Click CAMWorks on the SolidWorks menu bar. The CAMWorks menu lists the CAMWorks commands. The commands are explained in the CAMWorks online Help.

2. Right click on the NC Manager in the tree. A menu of commands displays. This is a shortcut menu. The right-click shortcut menus provide access to a variety of commands.

## CAMWorks Toolbar

The CAMWorks toolbar provides access to the main CAMWorks commands found on the CAMWorks menu. Clicking a toolbar button is the same as selecting a command from the NC Manager level, regardless of the active item in the tree.

1. Locate the CAMWorks toolbar and click the Options button.

2. In the Options dialog box, click each tab to view the options and settings that you can change in CAMWorks.

3. Click the Help button. Each tab is explained in the online Help.

4. Click the Close button in the upper right corner of the Help window to close the window.

5. Click Cancel to close the Options dialog box.

## Step 3: *Define the Machine*

The machine definition specifies the type of machining that will be done for the current model (i.e., mill or turn) and the associated machine tool control (post processor) for proper generation of the NC program. Based on the selected machine, the Extract Machinable Features command will recognize either mill or turn features. The machine can be changed at a later time to produce NC code for an alternate machine tool of the same type. The icons that display in the tree identify the machine:

Turn      Mill      Mill-Turn      Wire EDM

1. Right click the Machine item in the CAMWorks Feature tree.

   The shortcut menu displays. Right-mouse shortcut menus display commands that are appropriate for the item that is highlighted in the tree.
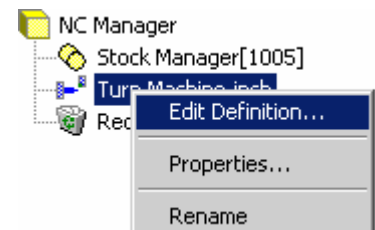
2. Select Edit Definition on the shortcut menu.

   The Machine dialog box displays.

3. Select Example Turn–in in the list of Available machines and click the Select button.

   This is the machine used for some of the exercises in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

   Machine tools are set up in the Technology Database. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility. For more information, see the CAMWorks online Help.
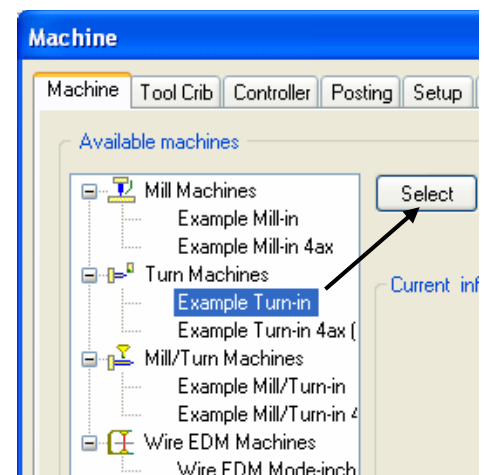
4. Click the Tool Crib tab and make sure Inch Turret 1 is selected.

   The Tool Crib page allows you to choose the tool crib (set of tools) that is 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.

   Inch Turret 1 is a default tool crib that has been set up for the sample turn machine. When you define your machine tools in the Technology Database, you can set up your own tool cribs.

5. Click the Controller tab.

   The Controller tab allows you to select the post processor. The list that displays depends on the post processors installed on your system. CAMWorks supplies 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).

   If FANTUTL (the tutorial post processor) is not highlighted, highlight it in the list and click the Select button. FANTUTL is the controller used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your post processor.
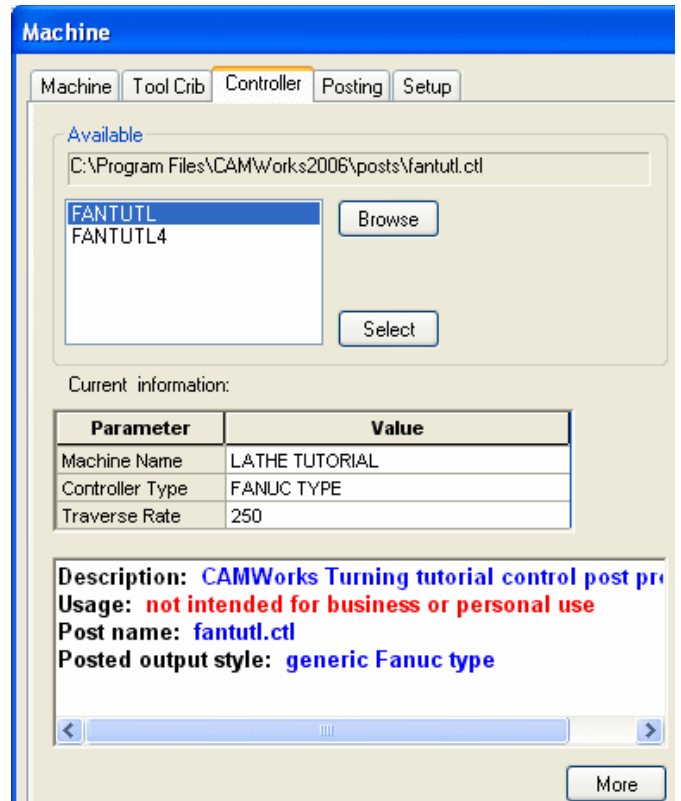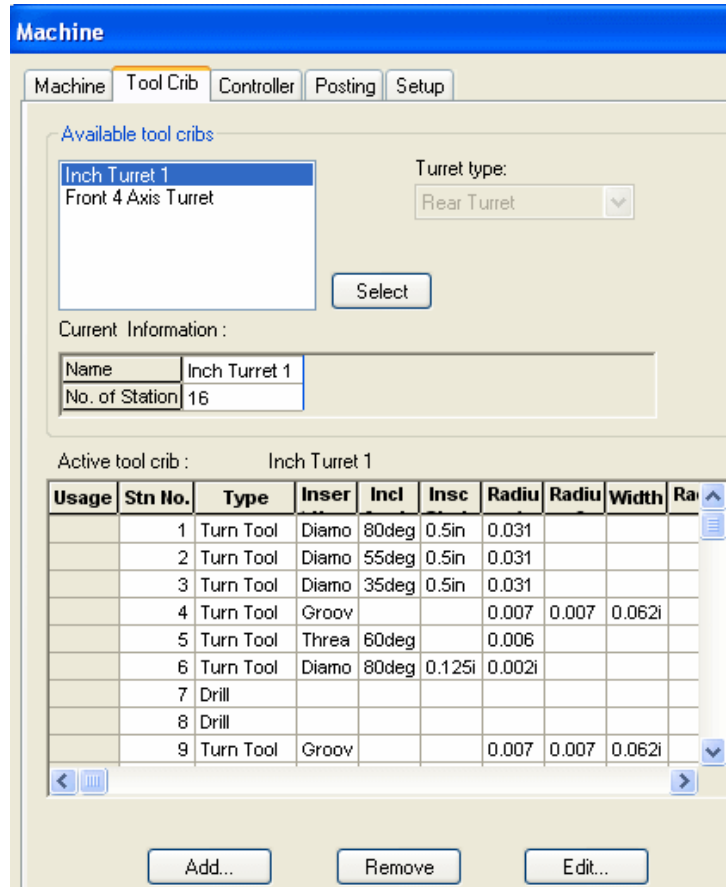
The Current Information area displays information about the FANTUTL controller. A short description displays in the window below the Current information. This window contains information only if an optional file has been created for the post processor.
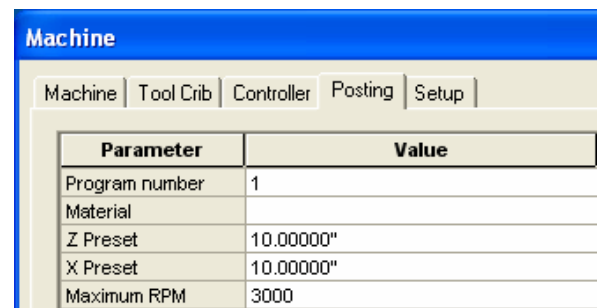
6. Click the More button.

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

Information files are provided for the sample FANTUTL post processor. Your TekSoft dealer or your company manager may be able to supply these files if they are available for your post processor. If files are not available, you can create post information files as explained in the online Help.

7. Click the Posting tab.

These parameters 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.



If you have installed the Feed/Speed Library, you can specify a material on this tab or when you define the Stock. If the Feed/Speed Library is not installed, you can specify the material only on this tab.

The Maximum RPM defines the maximum RPM that your machine allows when running Constant Surface Feed per Minute (CSFM). This prevents an overspeed spindle alarm as the tool moves to centerline. If necessary, you can set a lower maximum RPM for individual operations.

The Z and X Preset values are used by the system for a return position. These values are absolute numbers from X0,Z0. If your machine requires an absolute preset to be output in the program, these values are used. If you always change tools at the same position, setting these values saves you time later.

8. Type 1001 for the Program Number, then click OK to close the Machine dialog box.

## Step 4: Edit the Stock Definition

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). Currently, the sketch has to be in the same plane as the Stock in terms of the X and Z plane and must be a closed profile. No revolve line is needed for the geometry.

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

1. Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
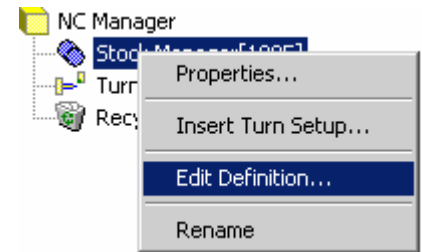
   The Manage Stock dialog box displays.

2. Leave the Type set to Bar stock.

3. Change the Length dimension to **6.85in** and press Enter.

   You can leave the Diameter set to 4.00in.

4. Click the Material button.

5. In the Stock Material dialog box, click the down arrow button and select 304L for the Common name, then click OK.

6. Click OK to close the dialog box.

   The Stock is updated in the graphics area.

   Notice the Stock line moved out from the face by .100 (in the +Z-axis direction).
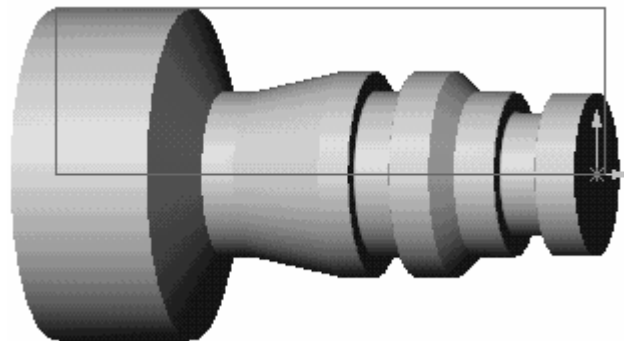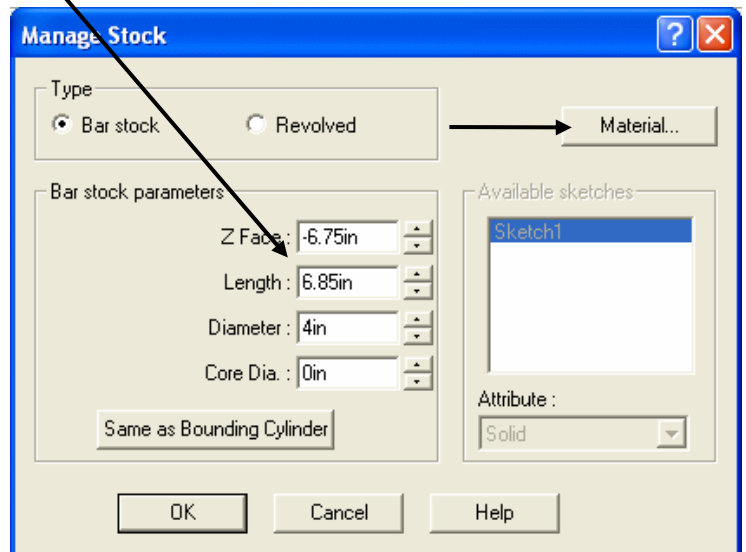
## Step 5: Define Machinable Features

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 machinable features such as grooves, outside and inside profile features, and face features. Depending on the complexity of the part, AFR can save considerable time in defining turn machinable features.

- Interactively created features

  If AFR does not recognize a feature you want to machine, you can define the feature interactively using the Insert Turn Feature command.

## Using Automatic Feature Recognition (AFR)

Automatic Feature Recognition is one of CAMWorks' most powerful features. The idea of AFR is to scan the part for features that can be machined. This process is much the same as what you would do if you were to pick up a part that you had to machine. You would look it over, take measurements, and begin deciding what machining processes you would need.

CAMWorks is not machining the SolidWorks features directly. It creates a separate list of Machinable Features instead. This is because a single SolidWorks feature may have several areas that need to be machined in different ways with different tools.

The part in this exercise contains the following machinable features, which are defined automatically by CAMWorks:

- Face Feature
- OD Profile Feature
- Rectangular Groove Features
- Cutoff Feature

Define machinable features automatically:

1. ![icon] Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.

   The Message Window displays. This window displays automatically to report the progress of the current process. Generating Setups is always the last item in AFR. When you see it, you know the process is almost complete.
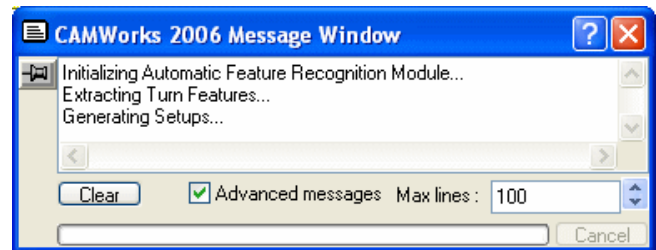
   

   You can control whether this window displays temporarily or permanently by selecting the Options command on the NC Manager shortcut menu or the CAMWorks menu and checking the Message Window option on the General tab in the Options dialog box.

   

   CAMWorks generates the Turn Setup and the machinable features. The items display in the Feature tree.
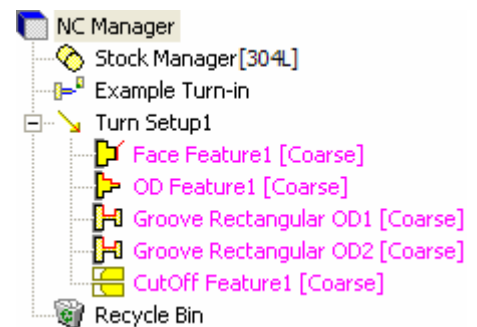
   The Turn Setup is the 2 axis (X and Z) plane that the tool movement will be based on. It has an origin location, and X, Y, and Z direction vectors. The Turn Setup is created automatically; however, you can move the origin and change the direction and angles of the axis. Most turning applications work in the X and Z plane, which is relative to the SolidWorks Y and X axis plane respectively.

   When you define the Stock as bar stock, only one Turn Setup (X and Z) is created and all features will be machined using tool orientations that are relevant for a rear turret machine configuration.

The Feature tree allows you to:

- Copy, rename, suppress, delete and combine machinable features
- Change machinable feature parameters
- Change the order in which the features are machined
- Insert Turn features
- Search for a feature based on item name
- Hide or show feature display in graphics area
- Generate an Operation Plan and find the first operation for a feature

**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.

Most feature parameters are fixed; however, some parameters can be changed using the Parameters command on the feature shortcut menu.

2. Right click OD Feature1 in the tree and select Parameters on the shortcut menu.

The OD Profile parameters include the length, maximum and minimum diameter and maximum back angle. Since there is no physical information about the type of profile, CAMWorks allows you to define an attribute for the profile.

3. Click the down arrow next to Coarse to see the choices.

Coarse, Taper, Reverse Taper and Thread are system attributes. User-defined attributes are also listed. For example, RF80-80 is a shortcut name for roughing and finishing the profile using 2 tools (80 degree diamond tools).

4. Leave the Attribute set to Coarse and click OK or Cancel to close the dialog box.



OD Profile Parameters

Length (L) : 6.75in
Maximum dia. (D1) : 4in
Minimum dia. (D2) : 2in
Finish radius (R) : 0in
Max. back angle (A) : -12.5288deg
Designation :
Thread type :
Type :
Browse...
Attribute : Coarse
Spindle attribute : Main
Thread Parameters
Method
  Tap rolling
  Tap cutting
  Threading
Library ...
ID :
Major dia (M1) :
Thread depth :
Pitch (P) :
Minor dia (M2) :
OK   Cancel   Help
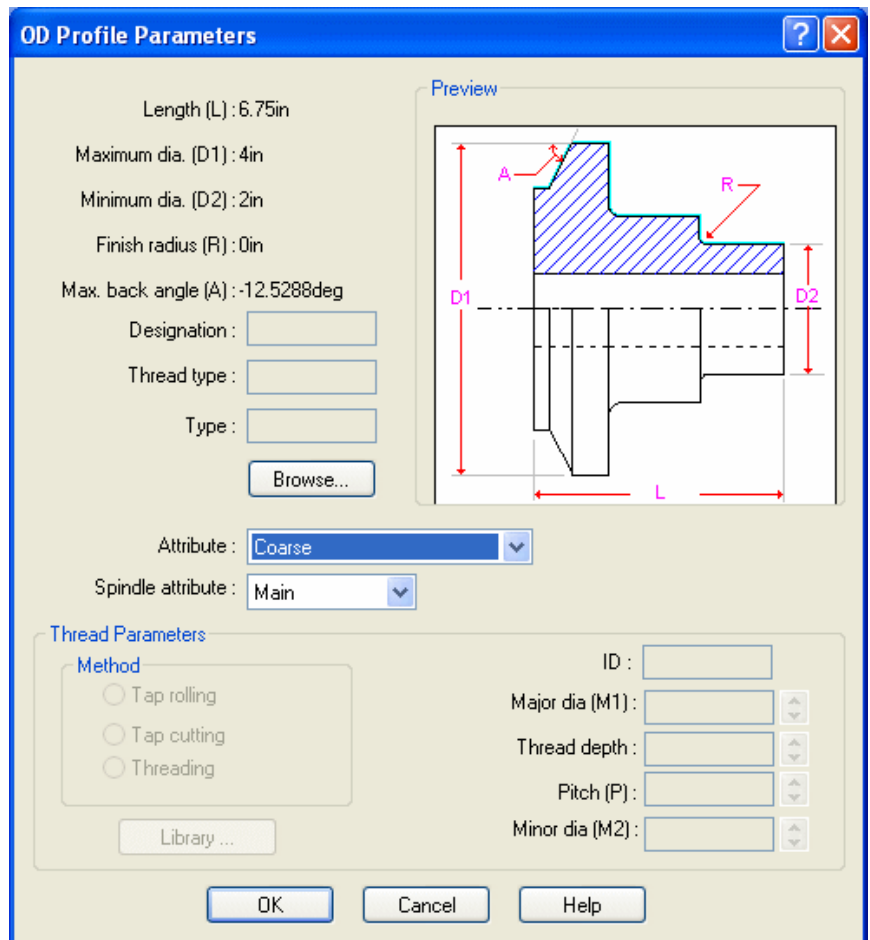
Save the part with the machinable feature data:

5.  Click the Options button the CAMWorks toolbar.

   Note that you can also click CAMWorks on the menu bar or right click NC Manager in the tree and select the Options command.

6. Make sure the Save/Restore part option on the General tab is checked, then click OK.

   If this option is checked, when you save and close a part document, the machining data is saved. When the part document is opened again, the machining data is restored along with the part design information.
   If this option is not checked, when you save and close a part document that contains at least one CAMWorks Setup, a message indicates that Save/Restore is disabled. If you click Yes, CAMWorks saves all CAMWorks data before closing the file. If you click No, CAMWorks closes the part and discards any new CAMWorks data since the last save.

7. Click Save As on the File menu.

8. In the Save As dialog box, type **cwturn1** and click the Save button.

   Make sure you save frequently. When you open a file, you are actually working on a copy of the file. The original is still stored on disk. Periodically saving your file ensures that your latest work is retained.

## 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 features such as certain types of grooves. To machine these areas, you need to define machinable features interactively using the Insert Turn Feature command.

CAMWorks also provides the flexibility to edit machinable features to accommodate machining requirements. For example, if you want the OD machined in segments to take into account possible machining rigidity, you can define features for the segments in the OD Feature as shown in the following exercise.

In this exercise, you insert an OD Feature so that you can rough turn the first OD step of the part. This OD segment will be finished in the Turn Finish operation for OD Feature1.

1. Right click OD Feature1 in the Feature tree and select Insert Turn Feature on the shortcut menu.

   The Insert Turn Feature dialog box displays. The profile of the part is shown. If necessary, move the dialog box to view the part.

2. Pick the first OD profile segment to the right of the groove closest to the face to select it.

   The single segment is highlighted.

3. Click the down arrow next to the Feature info list box.

   The turn machinable features that can be inserted are listed.

4. Select OD Feature.

5. Click on Extend 2 in the Selected entities list, then select Along X in the Extend group.
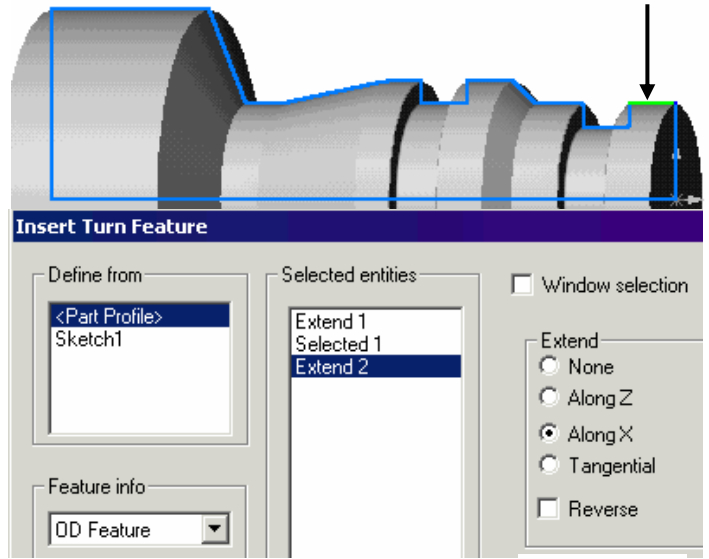
   This terminates the feature at the end of the segment and up to the Stock major OD.

6. Click OK to close the Insert Turn Feature dialog box.

   OD Feature2 displays in the Feature tree. You have now defined all the machinable features in this part.
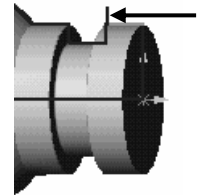
7. Drag OD Feature2 above OD Feature1 in the tree.

   You need to reorder these features because you want to rough OD Feature2 before cutting the entire OD. If you generated operations and toolpaths without reordering, CAMWorks would generate toolpaths for OD Feature1 first, which includes the segment in OD Feature2. Toolpaths would not be generated for OD Feature2 since there would be no material left to remove.

## 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 TechDB.

### *Did You Know ...*

In some situations, the operations defined for a feature in the TechDB may not be sufficient and additional operations may be required. You can insert operations manually using the Insert Operation command. This command is explained in the CAMWorks online Help.

1. Click the Generate Operation Plan button on the CAMWorks toolbar or right click Turn Setup1 in the Feature tree and click Generate Operation Plan on the shortcut menu.

   CAMWorks generates the operation plan for all the machinable features in Turn Setup1.

   The Operation tree displays and lists the operations that were generated. You can also click the Operation Tree tab to display the Operation tree.

The Operation tree provides an outline view of the operations for the machinable features. Operations are listed under the Turn Setup in the same order as the machinable features. At the top of the tree is the NC Manager. The Stock Manager and Machine items function the same as in the Feature tree. You can change the definition of your material block and the controller definition used by CAMWorks to produce G-Code.

The Operation tree allows you to:

– Insert, rename, suppress and delete operations
– Change operation parameters
– Edit the feature list
– Change the machining order
– Generate toolpaths
– Simulate toolpaths
– Hide or show toolpath display
– Post process the toolpaths
– Search based on an item name

2. Click the plus sign (+) next to Turn Finish1.
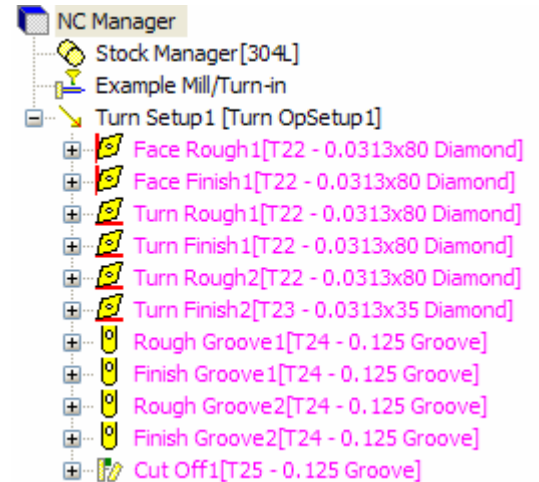
Clicking a plus sign displays the name of the machinable feature that this operation is going to machine.

**Did You Know ...**

If an operation displays in a color instead of black, then toolpaths have not been generated. This occurs when you insert a new operation interactively, you insert a new feature interactively and generate operations for the new feature, or CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or a parameter is not correct. You can set the color on the Display tab in the Options dialog box.

3. Right click Turn Finish1, select Delete on the shortcut menu and click Yes to confirm the deletion.

– Based on the information in the TechDB, rough and finish operations were generated for OD Feature2. In this exercise, you want to finish turn this OD segment in the finish operation for the entire OD (Turn Finish2) so you can delete the finish operation.

– If you typically machine the OD in segments, you can customize the TechDB and set up an Attribute to generate only a rough turn operation for this type of OD feature.

The operations that are generated by CAMWorks are based on information stored in the Technology Database. 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.

4. Right click on Turn Rough1 (OD Feature2) and select Edit Definition on the shortcut menu.

The Operation Parameters dialog box displays.

This dialog box gives you access to all the parameters used to define the toolpath. This dialog box also provides access to the parameters for the tool you are using and allows you to select a different tool.

5. Click the Rough Turn tab and change the First cut amount to **.15in**.

6. Click the other tabs and view the parameters. When you cut your own parts, you can adjust these values for your machining requirements.

7. Click OK to close the dialog box.
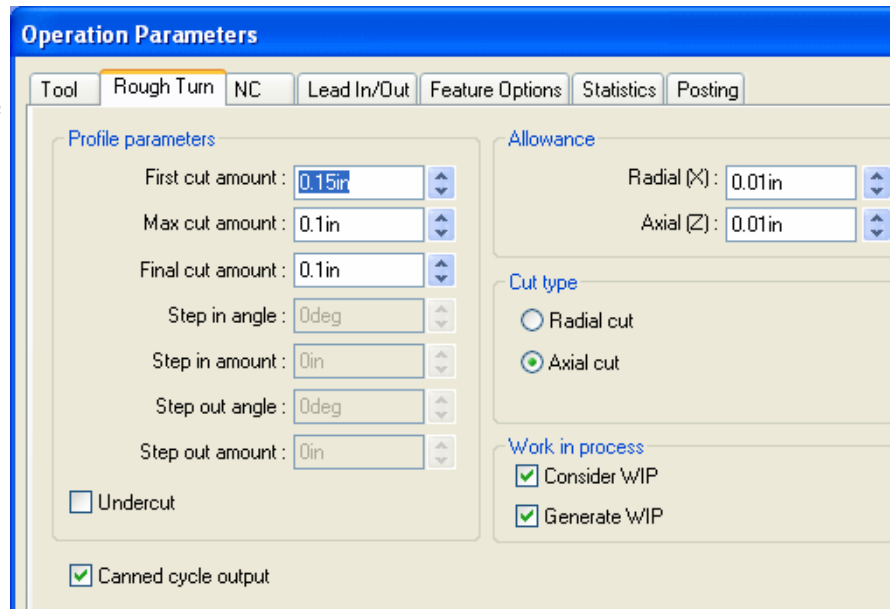
## Step 7: Generate Toolpaths

CAMWorks calculates toolpaths using the operation parameters to define how to machine each machinable feature.

1.  Click the Generate Toolpath button on the CAMWorks toolbar or right click Turn Setup1 in the Operation tree and select Generate Toolpath on the shortcut menu.

    CAMWorks calculates the toolpaths for each operation in the Setup.

2. Hold down the Shift key and select the first and last operation in the tree.

    The toolpaths for all the operations display on the part showing the centerline of the toolpath.

3. Left click an operation in the Operation tree.

    − The toolpath for that operation displays. As you click each operation, the toolpaths for that operation display.

    − Turning operation parameters can be edited and the operation can be renamed, moved, suppressed, deleted, etc. after toolpaths have been generated. If you make any changes, the toolpaths must be updated by selecting Generate Toolpath again at the Setup level.

### Simulate Toolpaths

CAMWorks provides the ability to simulate the toolpaths showing the tool movement and the resulting shape of the part.

1.  Click the Simulate button on the CAMWorks toolbar or right click on Turn Setup1 and select Simulate Toolpath.

   The Turning Simulation toolbar displays. Some of the options you can select to customize the simulation include:

   

   − Update the Stock after each cut or show the completed part at the end of the simulation.
   − Change the display of the stock, tool, tool holder, and target part (wireframe, translucent, shaded, or no display).
   − Run the simulation to the end or advance by single step or by feature.
   − Compare the design part and the simulated part during simulation.
   − Show a cross section of the material removal.
   − Show holder and fixture collisions.
   − Control the simulation speed by dragging the Simulation Speed Control slider.

   If you want to simulate only the toolpath for a given operation, you can right click that operation, then select Simulate Toolpath.

2.  Click the Run button.

   The simulation runs in Tool mode with the tool displayed during simulation.

3.  Click the X button in the upper right corner to exit the simulation and return to the SolidWorks display.

   🔅 *Did You Know ...*
   CAMWorks provides an option to display the XZ position of the tool in the Message Window during simulation. 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 Display Coordinates option.

## Change the Machining Order

Operations are generated in the same order that the machinable features are listed in the tree. This may not cut the part in the order you think is most efficient or practical for your machining requirements. You can drag and drop operations in the tree to reorder them.



1. Drag and drop the Groove operations in the order shown in the tree on the right.

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

   If you change the order after you generate toolpaths, you <u>must</u> re-generate the toolpaths so that the work in process can be updated.
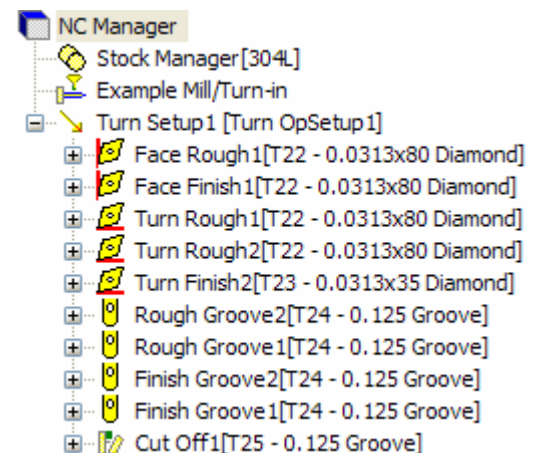
3. Right click Turn Setup1 and select Simulate Toolpath.

4. ▶ Run the simulation and notice that the groove closest to the face is cut first and both grooves are rough cut, then finished.

## 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 toolpath operation appears in the 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. **G1** Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and select Post Process on the shortcut menu.

The Post Output File dialog box displays so you can name the NC program file.

If you are running CAMWorks in Demo mode, the Post Output File dialog box does not display because you cannot save NC code in Demo mode.
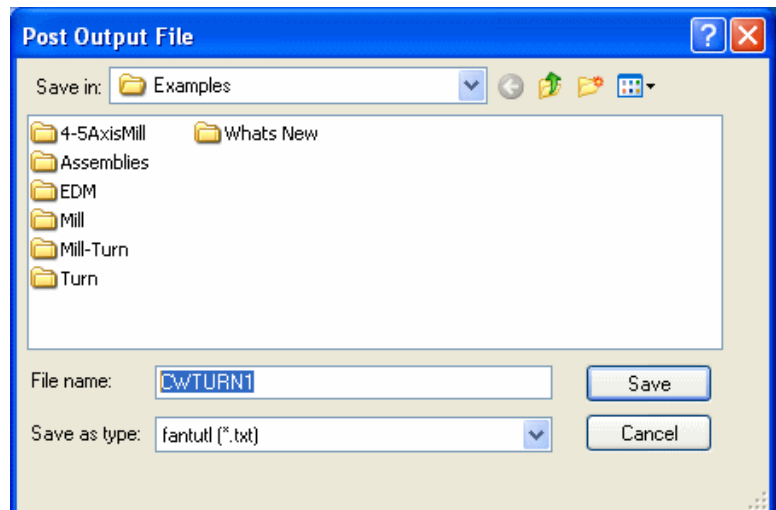
Typically, the NC program and Setup Sheet files are stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder.

If Post Process is grayed out on a menu or the CAMWorks toolbar, make sure that you have selected a post processor and generated the toolpaths.

2. Click the down arrow to the right of the Save as type box.

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

### 💡 Did You Know ...

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

3. If cwturn1 is not in the File name text box, type **CWTURN1**, 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.

   The Post Process Output dialog box displays.

4. Click the Step button at the top of the Post Process Output dialog box.

   CAMWorks starts to generate the NC program and the first line of NC code displays in the NC code output view box. When you click the Step button, CAMWorks generates one line of code.

5. Click the Step button again.

   The next line of NC code displays.

6. Click the Run button.

   Post processing continues until it is completed.

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

7. Click OK to close the Post Process Output dialog box.

### More About Setup Sheets

The Setup Sheet is a printable file that contains information the machine tool operator can use to set up the part and the tools required to produce a part. The information includes the machine, the controller, estimated machine time, the part material, and the tooling used to machine the part.

CAMWorks provides two methods for creating Setup Sheets:

− During post processing, CAMWorks automatically creates a simple text file with a .set extension.

− The Setup Sheet command on the NC Manager shortcut menu allows you to generate a Setup Sheet that is based on an Access database report template and store the information in the Report Database to view at any time. CAMWorks supplies several report templates that can be used as is. You can also open the Report Database in Access and create customized reports based on these templates or design your own original reports.

# Turn 2

## What You'll Learn

### *Extracting and Working With Machinable Features*

CAMWorks is a feature-based machining system. Feature-based machining provides numerous benefits because the definition of the feature enables a higher level of automation when creating machining operations and associated toolpaths. Adding to this benefit, CAMWorks provides both an automatic method of extracting features called Automatic Feature Recognition (AFR), and an interactive method using the Insert Turn Feature function. A CAMWorks file can contain both types of features.

The following feature types are currently supported:

- Face Feature

  A Face feature exists on the outermost portion of the part.

- OD Profile Feature

  An OD profile feature covers the entire outer surface of the part.

- ID Profile Feature

  An ID profile feature covers the entire inner surface of the part.

- Groove Feature with vertical walls

  A Groove is a feature that is totally closed and below the surrounding features.

  – Grooves are further broken down into one of two categories: rectangular and outbound.

  – Three types of grooves are supported: OD Groove, ID Groove, and Face Groove.

- Cutoff Feature

  A Cutoff feature is generated on the opposite side of the Face feature when the Stock is defined as bar stock. A Cutoff feature is similar to a face and can be converted to a Face feature for two-step turning operations.

This exercise demonstrates the types of features identified automatically by the Extract Machinable Features command.

1. Open the part file **TURN2AX_3.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. Examine the part.

   You can recognize various sizes and shapes of faces, profiles and grooves. It would appear that CAMWorks extracts features based on SolidWorks features used to create the part. As you will see, this is not necessarily so. Notice that the design tree contains no design features. This part was imported into SolidWorks.
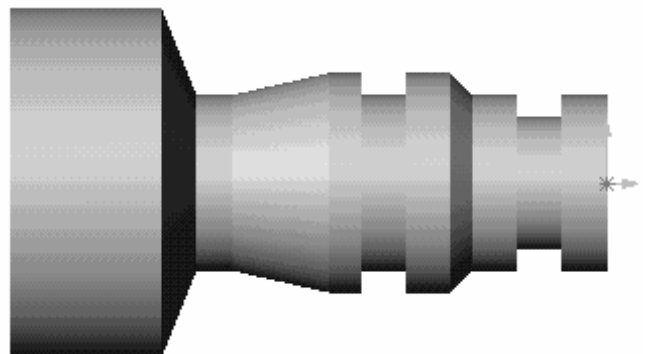
In this exercise, you will see how CAMWorks determines machinable features.

3. Click the CAMWorks Feature Tree tab.

4. ☞ Right click the Machine in the Feature tree and select the Edit Definition command on the shortcut menu.

5. Highlight Example Turn–in and click the Select button.

   This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

6. Click the Tool Crib tab.

7. Make sure Inch Turret 1 is the Active tool crib.

   From this page, you can add, remove and edit tools in the Tool Crib.

8. Select a tool from the Active tool crib list and click Edit.

**Machine**

Machine | Tool Crib | Controller | Posting | Setup

Available tool cribs

Inch Turret 1
Front 4 Axis Turret

Turret type: Rear Turret

[Select]

Current Information:

| Name | Inch Turret 1 |
| No. of Station | 16 |

Active tool crib: Inch Turret 1

| Usage | Stn No. | Type | Inser | Incl | Insc | Radiu | Radiu | Width | Ra |
|---|---|---|---|---|---|---|---|---|---|
| | 1 | Turn Tool | Diamo | 80deg | 0.5in | 0.031 | | | |
| | 2 | Turn Tool | Diamo | 55deg | 0.5in | 0.031 | | | |
| | 3 | Turn Tool | Diamo | 35deg | 0.5in | 0.031 | | | |
| | 4 | Turn Tool | Groov | | | 0.007 | 0.007 | 0.062i | |
| | 5 | Turn Tool | Threa | 60deg | | 0.006 | | | |
| | 6 | Turn Tool | Diamo | 80deg | 0.125i | 0.002i | | | |
| | 7 | Drill | | | | | | | |
| | 8 | Drill | | | | | | | |
| | 9 | Turn Tool | Groov | | | 0.007 | 0.007 | 0.062i | |

[Add...] [Remove] [Edit...]

The Operation Parameters dialog box displays three tabs that allow you to change the parameters for the selected tool. The changes you make in this dialog box affect only the tool crib for the current part. To change the tool definition for all future jobs, you need to edit the Tool Crib definition in the Technology Database.

**Operation Parameters**

Parameters | Diamond Insert | Holder

Tool number: 3
Tool usage: 1
TechDB ID: 168
Comment: DVJNL-164C VNMG-432
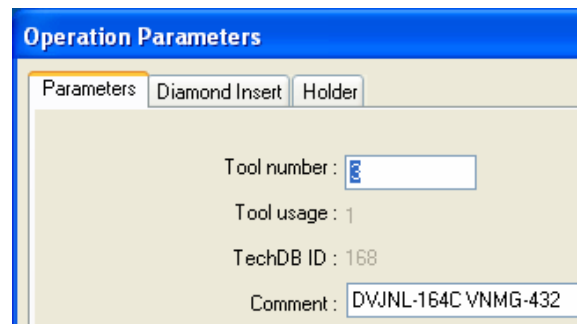
9. Click the tabs to view the tool parameters, then click OK to close the Operation Parameters dialog box.

10. Click the Add button in the Machine dialog box.

    The Tool Select Filter dialog box displays. This dialog box allows you to set filters for displaying and selecting tools.

11. Click the down arrow next to the Tool type, select Turn Tool and click OK.

    The Tools Database form for Turn Tools displays.

    This form allows you to add an existing tool in the TechDB to your Tool Crib. The list contains all the tools that have been entered into the TechDB. You cannot use this form to add new tools to the TechDB.

| | ID | ON | Insert Type | InsertID | InsertSummary | InsertTabRecSrc |
|---|---|---|---|---|---|---|
| ▶ | 6 | ☒ | 2 | 1 | 0.10 Width Groove Insert | TurnGrooveInsert_in |
| | 7 | ☒ | 1 | 13 | CNMG-432 | TurnDiamondInsert_in |
| | 8 | ☒ | 1 | 251 | DNMG-432 | TurnDiamondInsert_in |
| | 9 | ☒ | 1 | 369 | VNMG-432 | TurnDiamondInsert_in |
| | 10 | ☒ | 3 | 22 | NT-2RK all 60 | TurnThreadInsert_in |
| | 11 | ☒ | 1 | 1 | CNMG-431 | TurnDiamondInsert_in |
| | 13 | ☒ | 1 | 13 | CNMG-432 | TurnDiamondInsert_in |
| | 14 | ☒ | 1 | 13 | CNMG-432 | TurnDiamondInsert_in |
| | 15 | ☒ | 1 | 251 | DNMG-432 | TurnDiamondInsert_in |
| | 16 | ☒ | 1 | 251 | DNMG-432 | TurnDiamondInsert_in |
| | 17 | ☒ | 1 | 251 | DNMG-432 | TurnDiamondInsert_in |

Tools Database - Turn Tool (inches)

12. Click OK or Cancel to close the Tools Database.

13. Click the Controller tab in the Machine dialog box and make sure FANTUTL (the tutorial post processor) is selected.

    FANTUTL 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.

14. Click the Posting tab.

    These parameters provide information required to generate the NC program. When you use CAMWorks to machine your own parts, change these parameters as required.

15. Click OK.

16. Right click the Stock Manager in the Feature tree and select Edit Definition.

17. Change the Length of the bar stock to **6.85in** and press Enter, then click OK.

Extract machinable features:

1. [icon] Click the Extract Machinable Features button on the CAMWorks toolbar.

    The CAMWorks Message Window displays. This window shows the progress of the process.

    When AFR is finished, the Feature tree displays the machinable features. Five machinable features were found by AFR. The Feature tree is designed to work similar to the SolidWorks FeatureManager design tree. From the tree, you can reorder features (using drag and drop), change feature names and suppress features.

    NC Manager
    - Stock Manager[1018]
    - Example Turn-in
    - Turn Setup1
        - Face Feature1 [Coarse]
        - OD Feature1 [Coarse]
        - Groove Rectangular OD1 [Coarse]
        - Groove Rectangular OD2 [Coarse]
        - CutOff Feature1 [Coarse]
    - Recycle Bin
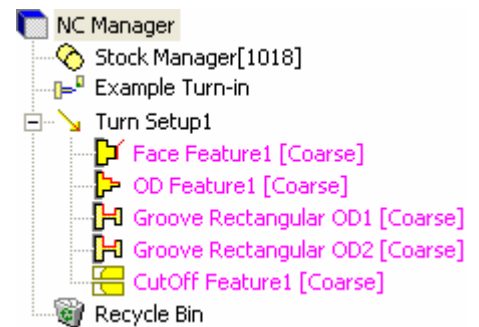
Feature recognition also created Turn Setup1. The Setup defines the tool direction necessary to machine these features. The origin of the Setup is program zero for the Setup. By default, the origin is set to the front of the part. In some situations, you will want to change this. Exercise Turn 4 explains how to change the origin.
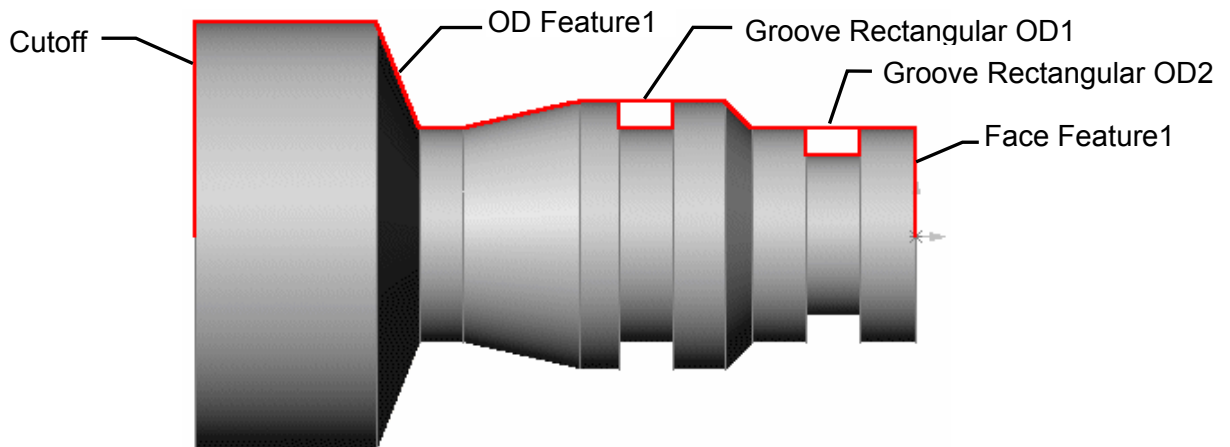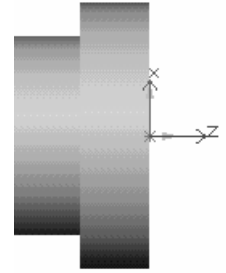


2. Click each machinable feature in the Feature tree.

   As you pick each machinable feature, the geometry for that feature highlights on the part.

Machinable features are recognized based on the part's geometry (the shape of a feature) and topology (how the features are related to one another). However, not all attributes of a feature can be determined from the geometry and topography. For example, AFR cannot determine that a ID profile has been threaded, reamed or bored. CAMWorks allows you to specify additional attributes to a feature so that more accurate operations can be created.

3. Right click OD Feature1 and select Parameters.

The Parameters dialog box provides a geometric and dimensional report on the feature and allows you to assign additional information to the feature. Depending on the Attribute, a different machining process can be assigned. You can change how each feature is processed by defining the sequence in the Technology Database.



4. Click the down arrow to the right of the Attribute box.

   The attributes in the list are user-defined attributes that have been set up in the TechDB for other operation sequences. After you have learned how to use CAMWorks, you can modify or delete these user-defined attributes and you can define additional attributes to customize the TechDB.
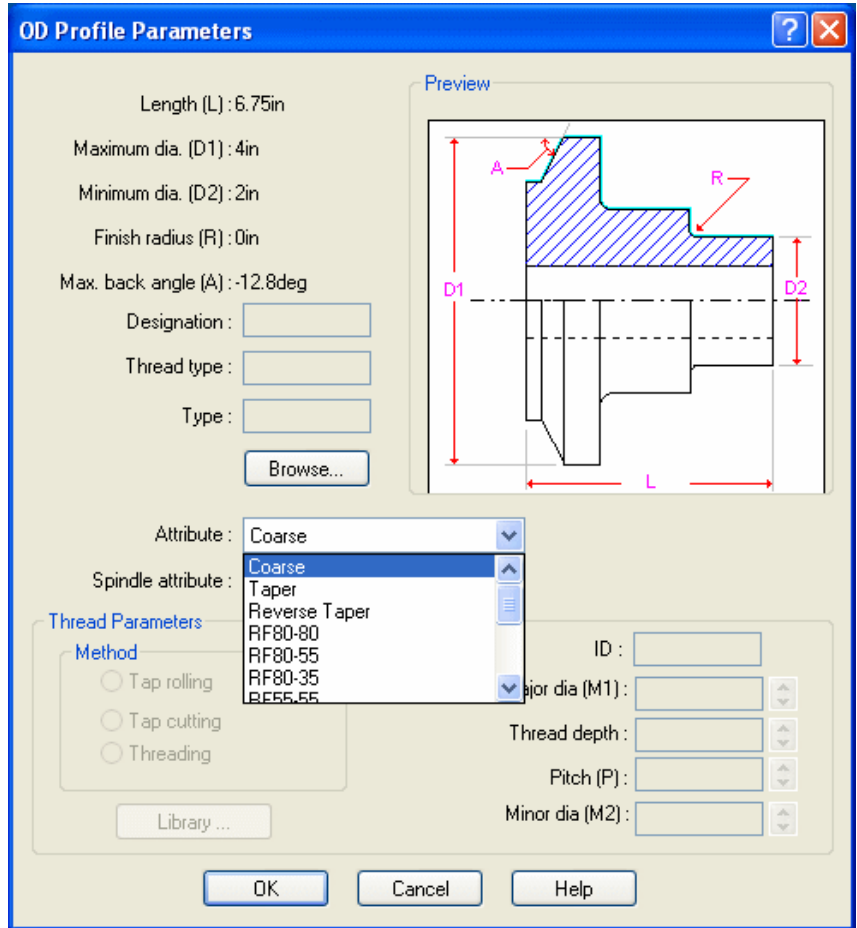
5. Leave the Attribute set to Coarse.

   This attribute has been defined to generate Rough Turn and Rough Finish operations using an 80-degree diamond insert.

6. Click OK to close the dialog box.

7. Review the parameters of other machinable features to understand more about the features.

Generate the operation plan for OD Feature1:

1. Right click OD Feature1 in the tree and select Generate Operation Plan.

   The Operation tree lists Turn Rough and Turn Finish operations for the OD feature.



2. Right click Turn Setup1 and select Generate Toolpath.
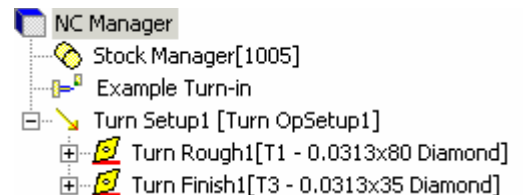
Notice the amount of material remaining after the Turn Rough operation. The back angle of the insert does not allow the tool to machine the entire feature.
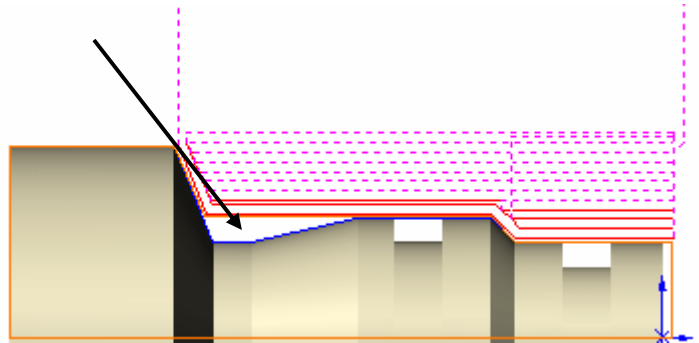
3.  Click the CAMWorks Feature Tree tab to return to the Feature tree.

4. Right click OD Feature1 and select Parameters on the shortcut menu.

5. Set the Attribute to RF80-GR, then click OK.

   This attribute has been defined in the TechDB to generate rough turn, finish turn rough groove and finish groove operations for an OD Profile feature.

6. Right click OD Feature1 and select Generate Operation Plan.

   If Generate Operation Plan has been run once, CAMWorks displays a dialog box with choices:
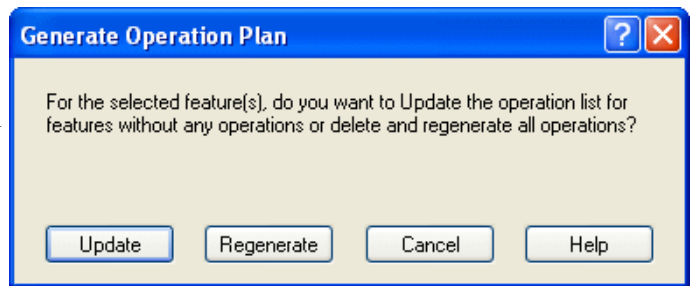
   Update: Select this option to generate operations for new features or if you have deleted previously generated operations for existing features. The operation list will not be updated for features that have previously generated operations.

   Regenerate: If you select this option, all previously generated operations are deleted and a new operation list is created. If you have modified any operations, these modifications will be lost.

7. Click Regenerate to delete the previous operations and generate a new operation list.

   The Operation tree lists four operations for the OD feature.
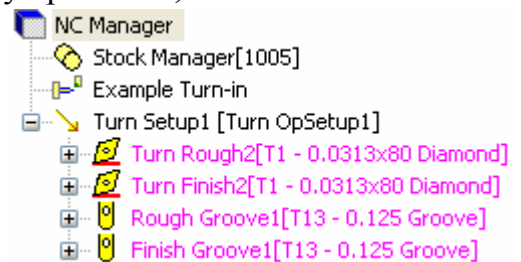
8. Right click Turn Setup1 and select Generate Toolpath.

   Notice the toolpaths generated by the groove operations.
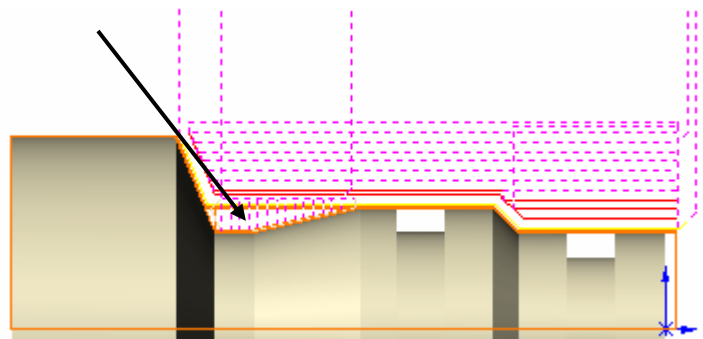
9.  Click the Generate Operation Plan button on the CAMWorks toolbar.

The Generate Operation Plan message window displays again.

10. Click either Update or Regenerate.

Since you have not changed any machining parameters for the operations that have already been generated, you could select either Regenerate or Update. If you had modified these operations, you would want to select Update to generate an operation list only for the features that do not have operations.

The Operation tree now contains operations for all machinable features in the selected Turn Setup.

To generate the NC code for this part, you would:
- Adjust the operation parameters as required
- Reorder the operations in the tree if necessary
- Generate toolpaths
- Run the Toolpath Simulation
- Readjust operation parameters if necessary
- Post process the part

*Did You Know ...*
Depending on your machining preferences, you could delete the OD feature and the two rectangular groove features found by AFR, then interactively insert an OD feature that includes the grooves. This feature could be machined completely with the operations generated by the RF80-GR attribute. The next exercise explains how to insert Turn features.

# Turn 3

## What You'll Learn

- Inserting an OD Profile for Threading
- Reorganizing Machinable Features
- Suppressing Machinable Features
- Generating an Operation Plan and Toolpaths
- Post Processing Toolpaths

In this exercise, assume that the part starts out as a casting and machining is required only for the features up to the larger diameter.

In Turn Setup1, you face the part, rough and finish the OD profile, rough and finish the rectangular grooves and cut the part off.

1.  Open part file **TURN2AX_1.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2.  [CW] Click the CAMWorks Feature Tree tab.

3.  Right click the Machine in the Feature tree and select Edit Definition on the shortcut menu.

    – Highlight Example Turn–in and click Select.
    – Click the Tool Crib tab and make sure Inch Turret 1 is the Active tool crib.
    – Click the Controller tab and make sure FANTUTL is selected, then click OK to exit the Machine dialog box.

4.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
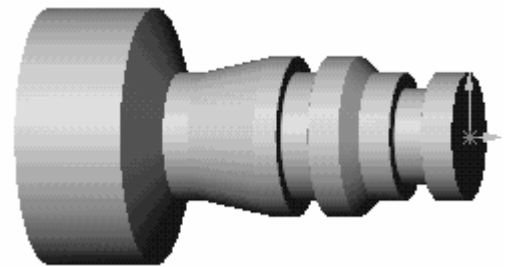
    The Manage Stock dialog box displays.

    By default, the Stock shape is a cylinder whose dimensions enclose the part. For this example, the part is machined from bar stock that has .10in material to be removed from the face. Assume that the largest OD of the part is to size and is not machined in this Setup. Since CAMWorks currently defines the Stock as round bar stock, you will use the default round shape and offset the minimum Stock size by .10in in length.

5.  Change the Length of the bar stock to **6.85in** and press Enter.

6.  Click OK.

7.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

   Turn Setup1 is created and AFR recognizes five features.
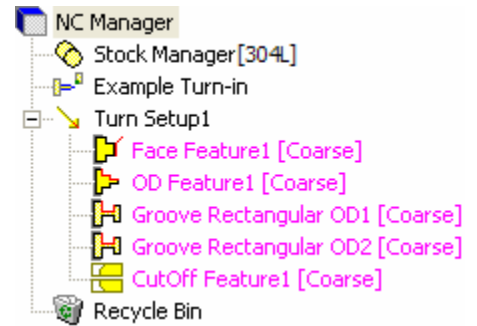
8. Left click Turn Setup1 in the tree and notice the machining direction indicated by the datum that displays on the part model.

   You can change the machining direction using the Edit Definition command on the Turn Setup shortcut menu.

   For this exercise, you can use the current machining direction.
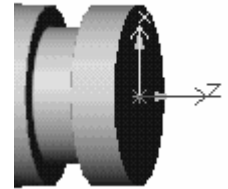
## Inserting an OD Profile for Threading

Now that a Turn Setup has been defined, you can create an OD profile feature to thread the part. The OD Feature is created using the Insert Turn Feature command.

1. Right click OD Feature1 in the Feature tree and select Insert Turn Feature.

   The Insert Turn Feature dialog box displays.

2. Pick the OD profile on the right at the part face.

   Notice that as you move your cursor over the profile, the entities change color and are highlighted.

   Turn features are defined by an outside shape and optionally inside shapes to represent profiles. These shapes can be sketches or part profiles found by AFR that are normal to the current Setup.

   If the dialog box obstructs your view of the part, drag the title bar of the window to a new location.

3. Make sure OD Feature is selected in the Feature info list.

4. Click the down arrow next to the Attribute list box and select Thread.

   The Thread attribute has been defined in the TechDB to generate a thread operation.
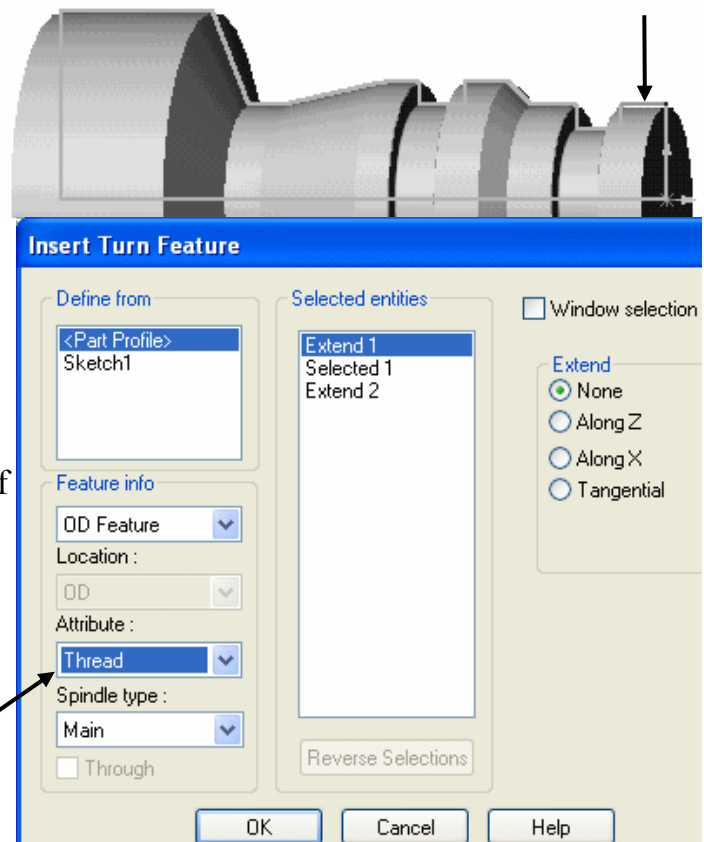
5. For Extend 1 and Extend 2, make sure None is selected in the Extend group.

   If CAMWorks is calculating the Work In Process (WIP), ID and OD Thread features should not be extended in either Z or X. If you extend Thread features, CAMWorks determines the feature length relative to the WIP and may extend the feature to start threading closer to the face than you want. Typically, this would occur when the thread feature is not located at the face of the part.

6. Click OK.

   A feature labeled OD Feature 2 is listed under OD Feature1 in the Feature tree.

## *Reorganizing Machinable Features*

To change the machining order, you can reorder either the machinable features or the operations that are generated for the features. You can drag a feature in the Feature tree from the current location to a position above or below another feature in the same Setup.

1. Click Groove Rectangular OD1, then click Groove Rectangular OD2 and notice the order the grooves are highlighted on the model.

   This may not be the order you want to machine these features.

2. Hold down the left mouse button and drag Groove Rectangular OD1 over Groove Rectangular OD2, then release the button.

   Notice that as you drag the item, an arrow displays to indicate where the item will be positioned. Rectangular OD2 is now listed before Rectangular OD1 and will be machined first.
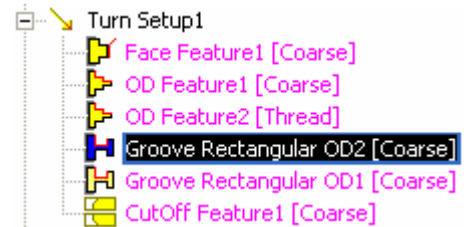


## *Suppressing Machinable Features*

You can suppress a machinable feature and machine the feature later. Suppressing a feature prevents operations (and associated toolpaths) from being created for that feature. For example, if there is not enough tool capacity to do threading on the machine you are using, you can suppress OD Feature2 and output the thread operation separately.

1. Right click OD Feature 2 in the Feature tree and select Suppress on the shortcut menu.

   The OD Feature2 icon is grayed out in the tree.

2. Click Save As on the File menu.

3. Type **cwturn3** and click the Save button.

## *Generating an Operation Plan and Toolpaths*

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the Feature tree and select Generate Operation Plan.

   Operations are created for the machinable features in the Turn Setup.

The operations are in listed in the Operation tree.
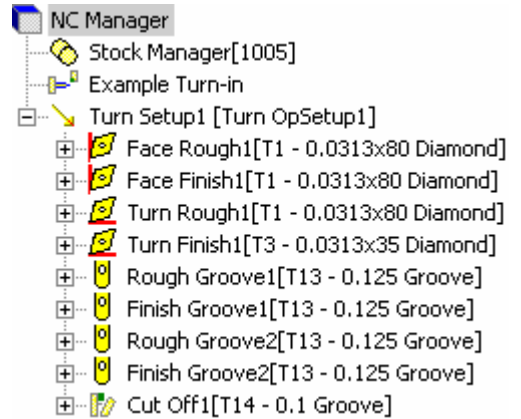
2.  Click the Generate Toolpath button on the CAMWorks toolbar or right click Turn Setup1 in the Operation tree and select Generate Toolpath.

3. Check the CAMWorks Message Window.

   − The Message Window provides information you may find helpful. For example, based on the information in this window, you may want to change a tool or insert a finish operation in order to cut a machinable feature completely.

   − If the Message Window is not displayed, select Options on the NC Manager shortcut menu or the CAMWorks menu and check the Message Window check box on the General tab.

   − You can move and resize the window. To change the size, point to a border or corner of the window (the pointer changes to a two-headed arrow). Drag the corner or border until the window is the size you want, then release the mouse button.

4. Experiment by changing parameters for the operations and regenerating the toolpaths for all operations in the Setup.

   **Did You Know ...**
   When you change operation parameters after generating toolpaths, we recommend that you regenerate toolpaths at the Setup level so the work in process can be updated correctly.

5. Right click NC Manager and select Simulate Toolpath.

6.  Click the Section View button and select Half.

7. Select the following display options:

    Stock Translucent Display

    Tool Shaded Display

    Target Shaded Display

8. Rotate the part, if necessary, so you can see the simulation.

9.  Click the Run button.

10.  Click the X button in the upper right corner of the Simulation toolbar to exit Simulation mode.
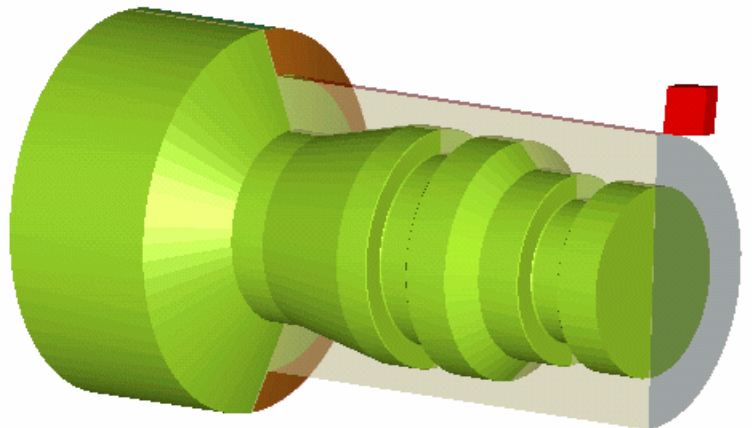
## *Post Processing Toolpaths*

1. G1 Click the Post Process button on the CAMWorks toolbar or right click Turn Setup1 in the Operation tree and select Post Process.

2. In the Post Output File dialog box, click Save to accept the default file name (cwturn3.txt).

   Note that if you are running CAMWorks in Demo mode, this dialog box does not display.

3. In the Post Process Output dialog box, check the Display Centerline option so that the toolpath will be highlighted as each line of code is produced.

4. Click the Run button (the double arrow).

   NC code is generated.

5. After viewing the code, click OK.


*Did You Know ...*

You should save the part frequently. If you want the CAM information saved with the part, make sure that the Save/Restore part option is checked on the General tab in the Options dialog box before you save. When you open the part, make sure that Save/Restore is checked or the CAM information will not be restored.

The General tab in the Options dialog box also has an Auto save option for automatically saving your CAMWorks data.

# Turn 4

## What You'll Learn

- Defining the Stock as a Forging or Casting
- Defining Additional Turn Features
- Deleting an Operation
- Adjusting Operation Parameters
- Defining Program Zero

This lesson guides you through the steps for the sequence of machining the part shown below. The bottom depressions of the part will not be machined in this exercise. The machining sequence is:

    – Rough and finish the face of the part
    – Rough and finish the OD profile
    – Rough and finish the groove

1. Open the part **TURN2AX_4.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. ![CW] Click the CAMWorks Feature Tree tab.

3. ![icon] Right click the Machine and select Edit Definition.

    – On the Machine tab, make sure that Example Turn-in is the active machine.
    – Click the Tool Crib tab and make sure Inch Turret1 is the Active tool crib.
    – Click the Controller tab and make sure FANTUTL is selected, then click OK.
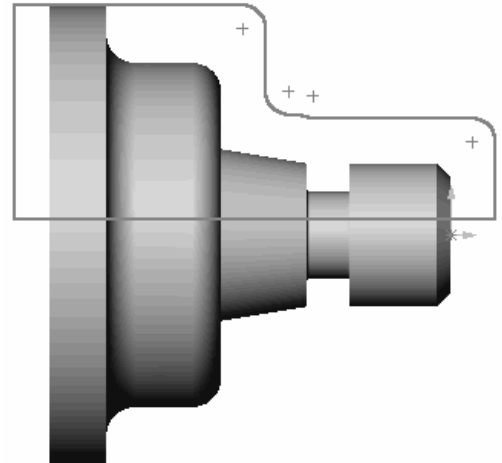
### *Defining the Stock as a Forging or Casting*

In this example, the Stock is a forging. A sketch named Stock Profile has already been created in this part to represent the forging.

1. Right click the Stock Manager and select Edit Definition.

CAMWorks automatically calculates the Stock size based on the smallest cylinder possible. You can change this definition by specifying a sketch to use.

2. Select Revolved for the Type.

3. In the Available sketches list, select Stock Profile.

The profile sketch representing the shape of the stock is highlighted on the part model.
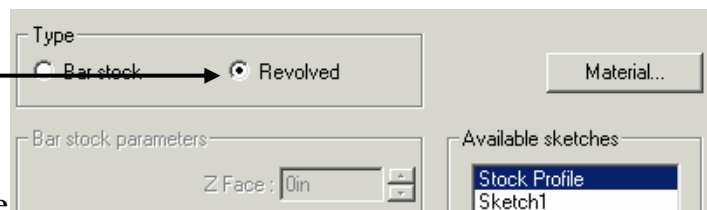
4. Click OK. An outline of the stock displays in the graphics area.

5. 🔳 Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

The recognized features in Turn Setup1 are the Face Feature, an OD Feature and a Rectangular OD groove. CAMWorks also created a separate setup for the second side of the part, which includes a Face feature. When the Stock is defined from a sketch, CAMWorks creates a second setup with a Face feature.

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 features and define features interactively. For learning purposes, in this exercise you delete the rectangular OD groove and insert the feature interactively.

6. Right click Groove Rectangular OD1 in the tree and select Delete on the shortcut menu.

7. Click Yes to confirm the deletion.
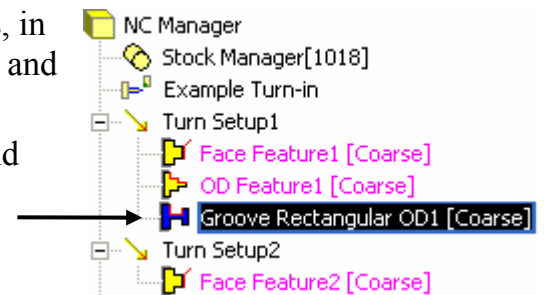
The feature is moved to the Recycle Bin. Deleted features are automatically placed in the Recycle Bin, which is used to store machinable features that you do not intend to machine.

8. Click the minus sign to the left of the Recycle Bin to collapse it.

## Defining Additional Turn Features

You can now define a rectangular OD groove feature to machine the undercut.

1. Right click OD Feature1 and select Insert Turn Feature.

2. Select <Part Profile> in the Define from list.

All profiles in the current section plane are highlighted on the model.

3. Change the Feature info to Groove Rectangular.

4. Make sure the Attribute is Coarse.

5. With the Window selection check box checked, select the geometry for the groove by drawing a window around the groove entities as shown in the figure.

The Selected entities list should contain 5 selected entities and 2 extend entities.
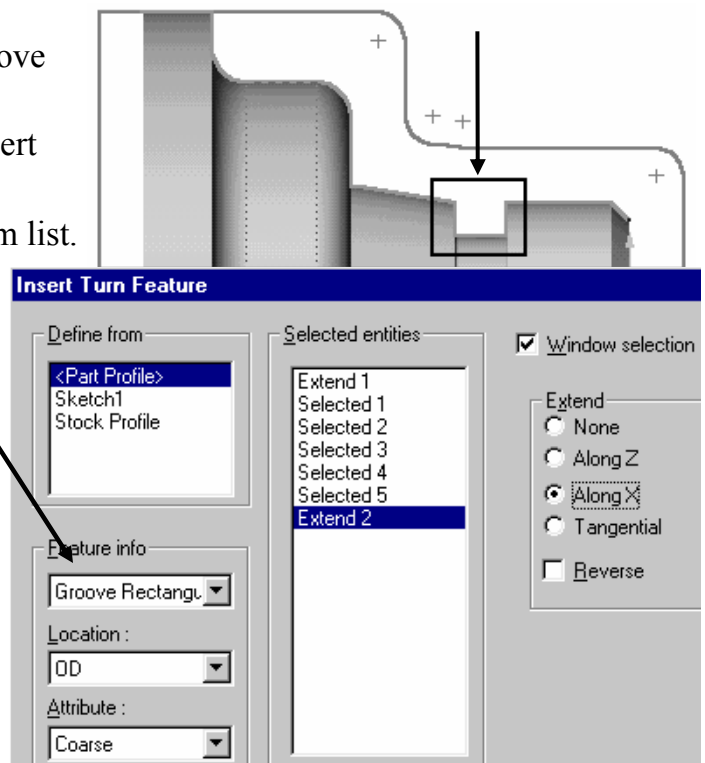
6. Pick Extend 1 and set the Extend to Along X.

7. Pick Extend 2 and set the Extend to Along X.

   The extends display on the model.

8. Click OK.
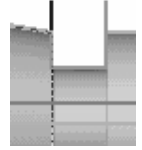
   Groove Rectangular OD2 is added to the tree.

9. ▣↓ Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager and select Generate Operation Plan.

   The operations for Turn Setup1 and Turn Setup2 are listed in the Operation tree.
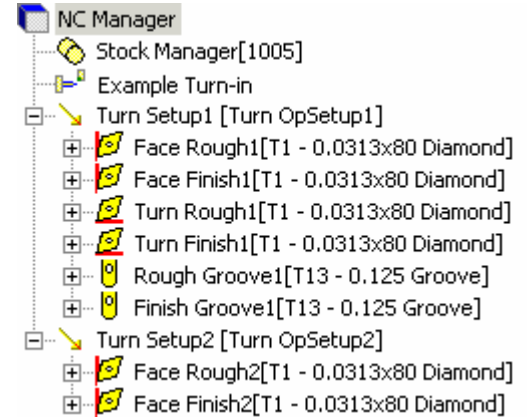
## Deleting an Operation

In some cases, it may be desirable to delete operations that are created automatically. In this example, all of the operations that were generated for Turn Setup1 are needed to machine the part. If the stock is a casting or forging and you use a sketch to define the Stock, CAMWorks generates a second setup for the other side of the part. Depending on whether or not a second setup is needed for this machining session, you can delete or suppress the operations that were generated for Turn Setup2. In this example, you delete the second setup and all the operations under it.

1. In the Operation tree, right click Turn Setup2 and select Delete on the shortcut menu.

2. Click Yes to confirm you want to delete the Setup and all dependant operations under it.

   The Setup and associated operations are removed from the tree.

## Adjusting Operation Parameters

Each operation contains machining 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.

1. Click the + next to Face Rough1 in the Operation tree.

   The tree expands to identify the machinable feature the operation was generated to machine (Face Feature1).

2. Click the − next to Face Rough1.

3. Right click Face Rough1 and select Edit Definition.

   The Operation Parameters dialog box displays.
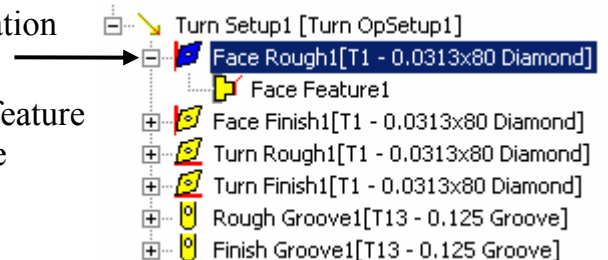
4. Click the NC tab and change the Clearance Plane to **.15in**.

   This parameter specifies the distance away from the part to start the cycle in the X and Z axis. This is where the tool rapids to at the beginning of the cycle and the distance the tool retracts from the part after each cutting pass.

5. Click OK.

6. Right click Face Finish1 and select Edit Definition.

7. Click the Tool Crib tab on the Tool page.

8. Select the Diamond 55deg tool, then click Select.

   The current tool is replaced with the tool you selected.

   If the tool you want is not listed in the Tool Crib, you can click Add and select a tool on the Tools Database form.
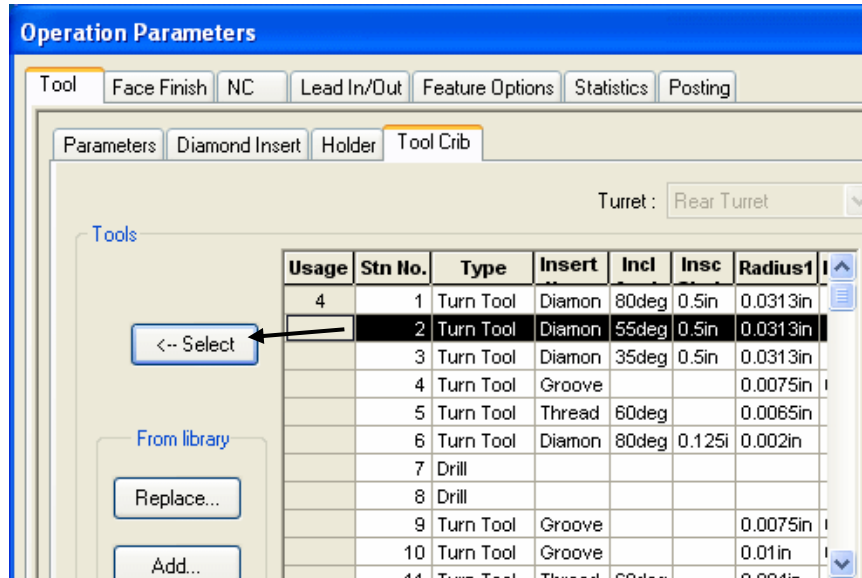


9. Click OK to close the dialog box.

## Defining Program Zero

The Turn Setup defines the tool direction and program zero. Program zero can be changed in the Setup by picking one of the following that is closest to the desired work Z0 face: an OD, angle or ID face that is concentric to the part. Each Setup has a different origin.

1. Right click Turn Setup1 and select Edit Definition.

   The Operation Setup Parameters dialog box displays.

2. On the Origin tab, select Automatic in the Defined from list box and check the Other end option to move the origin to the left side of the part.

   The Chuck Location and Chuck Definition tabs are explained in the next lesson.
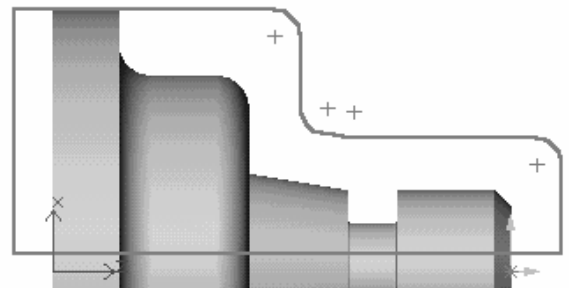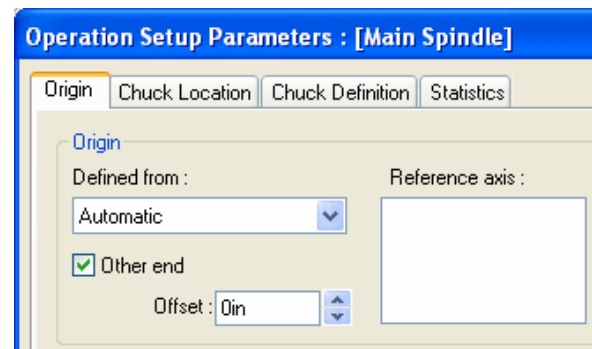
3. Click OK.

4. ⬚ Click the Generate Toolpath button on the CAMWorks toolbar.

   Toolpaths are generated for all operations.

5. Click each operation in the tree to view the toolpaths.

6. Change the orientation to Isometric view.

7. Right click Turn Setup1 and select Simulate Toolpath.

8. ⬚ Click the Section View button and select Half.

9. Select the following display options:

   ⬚ Stock Shaded Display

   ⬚ Tool Shaded Display

   ⬚ Target Shaded Display

10. ⬚ Click the Run button.

    Since you deleted Turn Setup2 and the Face Feature, material is left on the locating face. This could be machined in another setup.

11. ⬚ Click the X to exit the simulation.

12. Using drag and drop, reorder the operations to cut the part in the order you think is most efficient.

13. Save the part as **CWTURN4**.

14. ⬚ Click the Post Process button on the CAMWorks toolbar or right click Turn Setup1 and select Post Process.

15. In the Post Output File dialog box, click Save to use cwturn4.txt for the NC program file name.

    If you are running CAMWorks in Demo mode, this dialog box does not display.

16. Click the Run button in the Post Process Output dialog box.

    CAMWorks generates the NC code.

17. After viewing the code, click Cancel to close the dialog box.
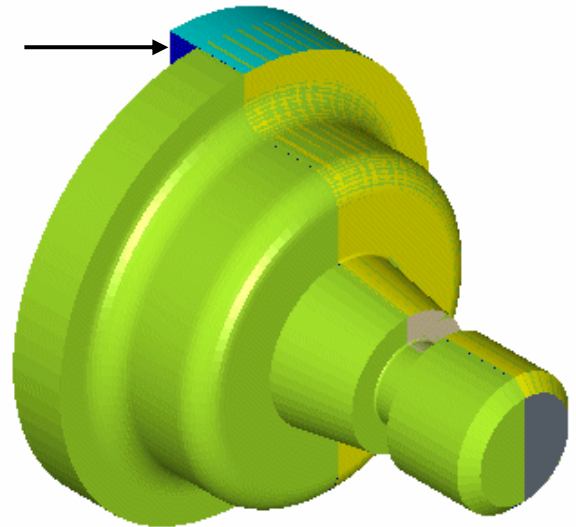
# Turn 5

## What You'll Learn

- Adding an ID Groove Feature
- Adding OD Thread and ID Thread Features
- Generating Operations and Toolpaths
- Displaying Locating Symbols
- Defining the Chuck and Viewing in Toolpath Simulation

1. Open the part file **TURN2AX_5.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. Change the orientation to Front view.

3. Click the CAMWorks Feature Tree tab.

4. Right click the Machine and select Edit Definition.

   - On the Machine tab, make sure that Example Turn-in is the active machine.
   - Click the Tool Crib tab and make sure Inch Turret 1 is the Active tool crib.
   - Click the Controller tab and make sure FANTUTL is selected, then click OK to close the dialog box.

5. Right click on the Stock Manager and select Edit Definition on the shortcut menu.

6. In the Manage Stock dialog box, leave the Type set to Bar stock, change the Length to **2.05in** and press Enter.

7. Click the Material button in the top right corner of the dialog box.

8. In the Stock Material dialog box, select a material in the pulldown list, then click OK.

9. Click OK to close the Manage Stock dialog box.

10. Click the Extract Machinable features button on the CAMWorks toolbar.

    AFR analyzes the model for features to machine. When AFR is completed, the tree displays Turn Setup1, which establishes the workplane for turning, and the machinable features that were recognized.

    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 features and define features interactively.
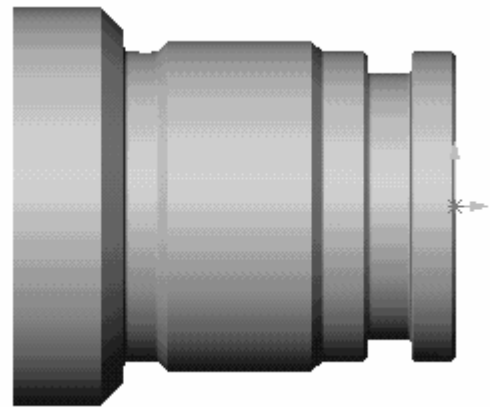
11. Right click the Groove Rectangular ID2 feature and select Delete key on the shortcut menu.

    For learning purposes, in this exercise you delete the Groove Rectangular ID2 feature and insert it interactively.

12. Since you are chucking, delete Cutoff Feature1.
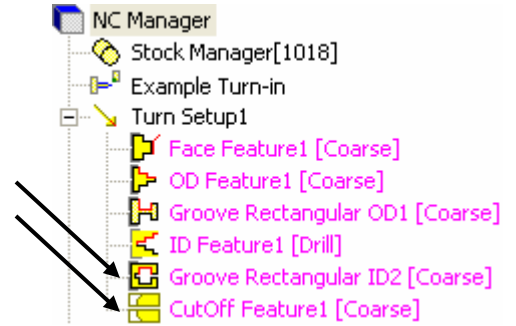
## *Adding an ID Groove Feature*

You are going to add three features interactively. The first feature is the groove that you deleted. CAMWorks understands the shape of a groove and knows how it is to be machined. The typical groove configuration is defined as a rectangular shape consisting of two equal length walls, which may or may not have fillet radii and may or may not have corner breaks. When you insert a Groove feature for this undercut area, CAMWorks knows how to machine the area using the standard Rectangular Groove machining techniques.

1. Right click on ID Feature1 and select Insert Turn feature.

    The Insert Turn feature dialog box displays.

2. In the Feature info selection box, select Groove Rectangular and set the Location to ID.

3. Leave the Attribute set to Coarse.

4. Make sure the Window selection check box is checked.

5. Hold down the left mouse button, draw a window around the groove area on the ID profile, then release the button.

    The Selected entities list is updated to contain 2 Extend entities and a total of 5 Selected entities (Selected 1-5).
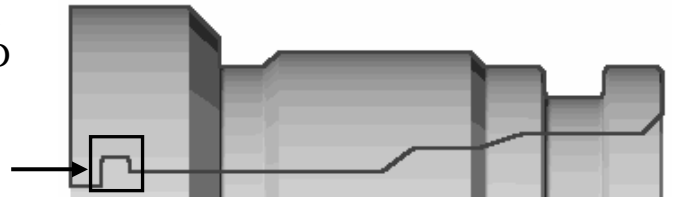
6. Select the Extend 1 in the list and set the Extend option to Tangential

7. Click Extend 2 in the list and set the Extend option to Tangential, then click OK.
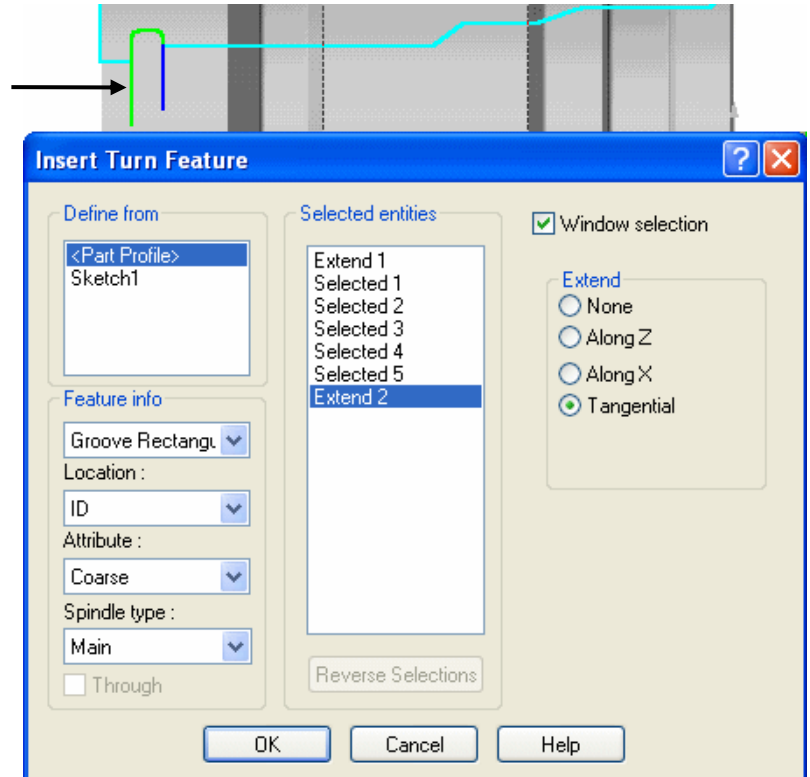


The tree lists Groove Rectangular ID3.
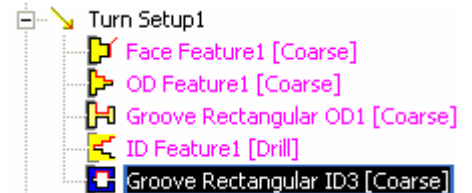
## Adding OD Thread and ID Thread Features

Thread information comes in many different formats and has many parameters. For a CAD/CAM system, it is not practical to model threads on a solid. Even though this can be done, it is very difficult for software to interpret a helical cut in the OD of a part and decipher what type of thread it is. In CAMWorks, you use attributes to define a thread on a feature.

In this exercise, you define OD and ID features to be threaded.

1. Right click OD Feature1 and select Insert Turn feature.

2. Leave the Feature info selection box set to OD Feature.

3. Set the Attribute to Thread.

4. Pick the OD segment where the thread will be machined (indicated by the arrow in the figure).

5. For Extend 1 and Extend 2, make sure None is selected in the Extend group.
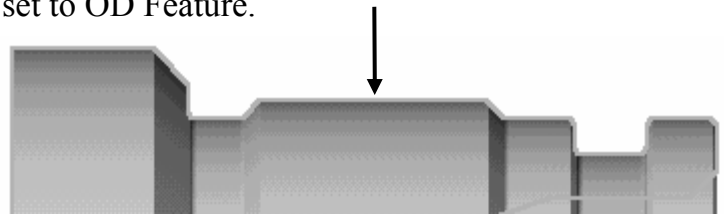
If CAMWorks is calculating the WIP (work in process), ID and OD Thread features should not be extended in either Z or X. If you extend Thread features, CAMWorks determines the feature length relative to the WIP and may extend the feature to start threading closer to the face than you want. Typically, this would occur on thread features such as this one where the feature is not located at the face of the part.

To adjust the machining length for a Thread feature, you can
- Specify a Start length in the Thread operation machining parameters.
- Set up the TechDB to define an amount that you want the tool to go beyond the thread length into a thread relief area.

6. Click OK. OD Feature2 is listed in the tree.

7. Right click OD Feature2 and select Parameters.

The OD Profile Parameters dialog box displays.

The Maximum diameter for this feature is 1.5in. In order to generate a Thread operation, the corresponding thread condition must be selected from the TechDB. The major diameter in the TechDB must match the feature maximum diameter.

Initially, the Thread Parameters display the values in the first record in the TechDB; however, this record has not been selected.

8. Click the Library button. The Tools Database - Thread Condition form displays.
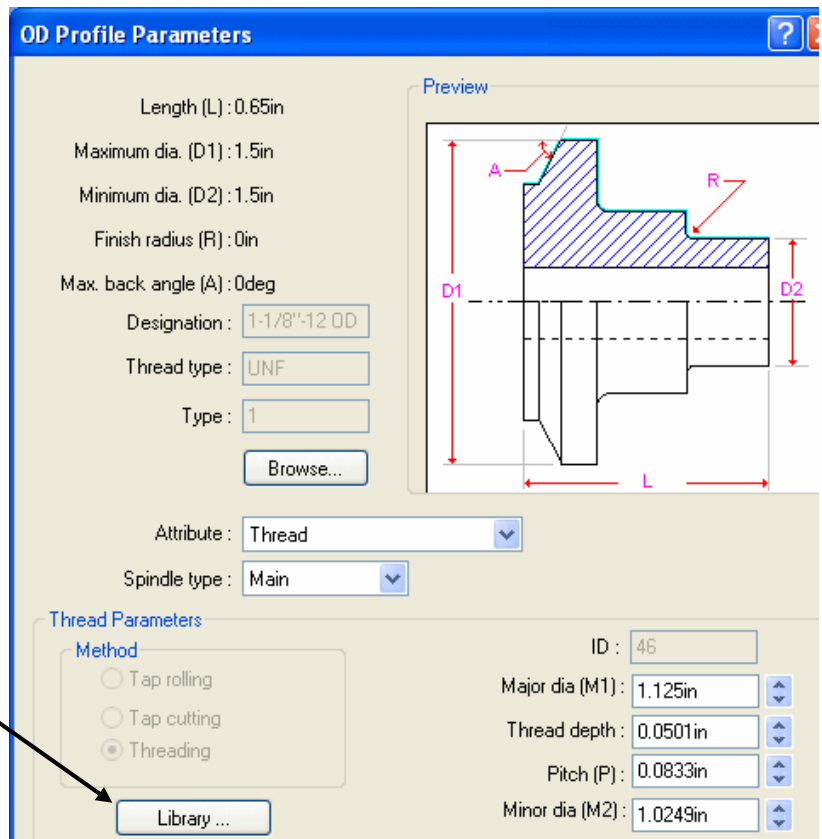
Several thread conditions have been set up in the TechDB. When you machine your own parts, you will add data for additional threads.

9. Select OD 3, which has a 1.5 Major Dia, then click OK.

**Tools Database - Thread Condition (inches)**

| ID | Type | Designation | MajorDia | MinorDia | Pitch | EndPitch | DepthOfThread |
|----|------|-------------|----------|----------|-------|----------|---------------|
| 3  | BSF  | OD Thread   | 1.5      | 1.42     | 0.0625 | 0       | 0.04          |
| 4  | BSF  | OD Thread 1 | 3.3631   | 3.2831   | 0.0625 | 0       | 0.04          |
| 6  | UNF  | 0-80 OD     | 0.06     | 0.0467   | 0.0125 | 0       | 0.00665       |

10. Click OK to close the OD Profile Parameters dialog box.

11. Right click ID Feature1 in the tree and select Insert Turn feature.

    The Insert Turn Feature dialog box displays.

12. For the Feature info, select ID feature.
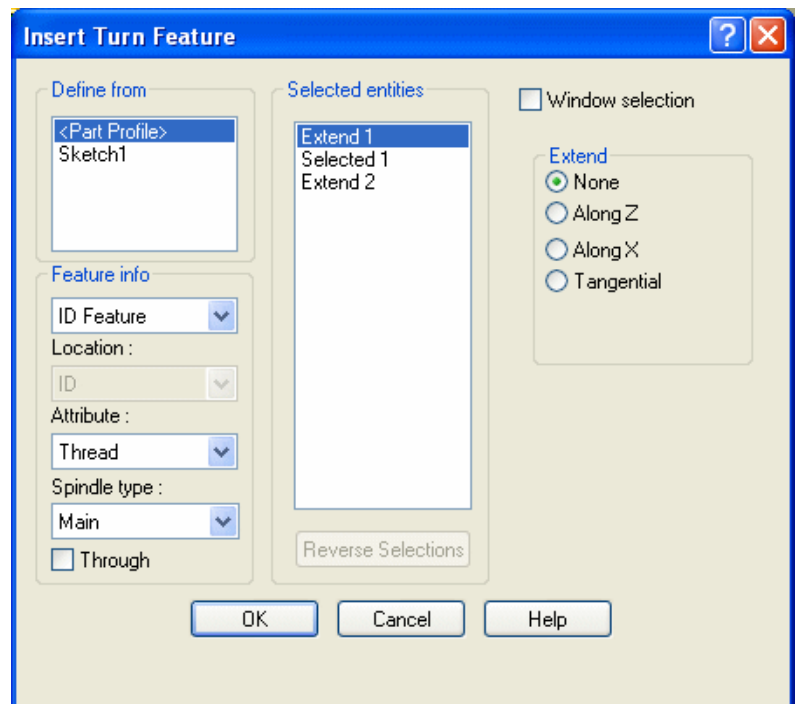
13. Set the Attribute to Thread.

14. Select the ID segment at the front of the part (indicated by the arrow on the figure).

15. For Extend 1 and Extend 2, make sure None is selected in the Extend group.

16. Click OK.

    ID Feature2 is listed in the tree.

17. Right click ID Feature2 and select Parameters.

    The ID Profile Parameters dialog box displays. You need to select the thread condition for ID Feature2.
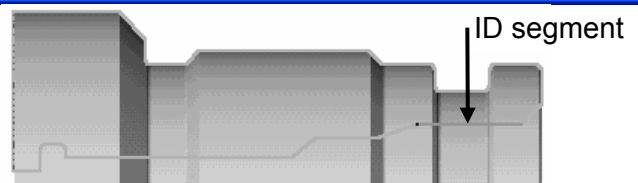
18. Click the Library button.

19. On the Tools Database - Thread Condition form, select ID 5, which has a .95 Major Dia, then click OK.

20. Click OK to close the ID Profile Parameters dialog box.

**Insert Turn Feature**

Define from: `<Part Profile>`, Sketch1

Selected entities: Extend 1, Selected 1, Extend 2

Window selection

Extend: ⊙ None  ○ Along Z  ○ Along X  ○ Tangential

Feature info: ID Feature

Location: ID

Attribute: Thread

Spindle type: Main

☐ Through

Reverse Selections

OK    Cancel    Help

ID segment

## *Generating Operations and Modifying Operation Parameters*

Before generating operations, you can change feature parameters and rename features so that they are more descriptive.

1. Right click ID Feature1 in the tree and select Parameters.

2. Change the Attribute to Coarse and click OK.

3. Right click OD Feature2 in the tree and select Rename.
   A text box displays around the item. ▶ OD Feature2

4. Type **OD Thread** and press Enter.

5. Right click ID Feature2 and select Rename.

6. Type **ID Thread** and press Enter.

7. 📥 Click the Generate Operation Plan button on the CAMWorks toolbar or right click Turn Setup1 in the tree and select Generate Operation Plan.

   You can modify operation parameters before or after generating toolpaths. Choosing parameters for machining depends on your machining practices. The following examples show you how to make changes.

8. Right click Thread1 in the tree and select Edit Definition.

9. Change the following values on the Thread tab, then click OK.
   – Depth per cut = .03in
   – Final cut amount = .009in
   – Start length = .15in
     The start length is measured from the start of the OD Thread feature that you created and is a distance in front or to the right of the feature. Since this is an OD feature and the tool is on a rear turret, the tool orientation is down and left.
   – Pitch = .065in

10. Right click Rough Groove1 and select Edit Definition.

11. On the Rough Groove tab, change the Radial (X) and Axial (Z) allowances to 0.0.

12. Click OK.

13. Right click Finish Groove1 in the tree, select Delete and click Yes to confirm the deletion.
    By setting the Rough Groove allowances to 0.0, you can rough and finish the groove in one operation.

14. Right click Rough Groove2 and select Edit Definition.
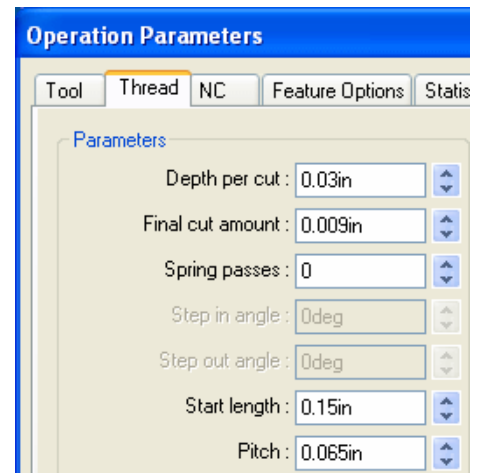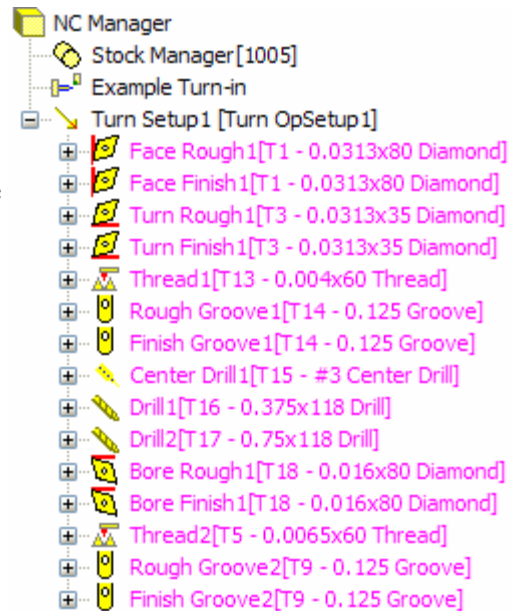
On the Rough Groove tab, change the Radial (X) and Axial (Z) allowances to 0.0, then click OK.

15. Right click Finish Groove2, select Delete and click Yes to confirm the deletion.

## *Defining the Machining Sequence, Adjusting Parameters, Generating Toolpaths*

The order that the operations are listed is relative to the feature order. Both the Feature tree and the Operation tree can be reordered to achieve the specific machining sequences that are needed in terms of tool order and material removal.

1. In the tree, drag Rough Groove2 above Bore Finish1.

2. ⬛ Click the Generate Toolpath button on the CAMWorks toolbar or right click on Turn Setup1 and select Generate Toolpath.

3. Click on each operation to view the toolpath.

4. ⬛ Click the Generate Toolpath button on the CAMWorks toolbar to update the toolpaths.

## *Displaying Locating Symbols and Defining the Chuck*

You can position two triangles to provide a graphical representation of where the part will be clamped in the chuck or collet.

1. Left click on Turn Setup1. A triangle displays at the part zero location on the part. This locating symbol can be used to show where the chuck or collet is located.

2. Right click Turn Setup1 and select Edit Definition.

The Origin tab displays in the Operation Setup Parameters dialog box.

3. Select Automatic in the Defined from list.

This is where the origin or piece part zero is located.
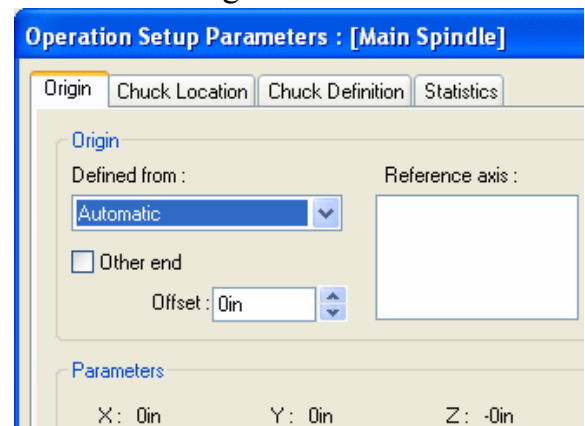
4. Click the Chuck Definition tab.

   This dialog box allows you to define the size of the chuck base and the jaws for display during toolpath simulation. CAMWorks allows multiple configurations of chuck parameters to be saved in the TechDB and reused. To define additional configurations, set the parameters on this tab, then type a name in the input box and click the Save button. The configuration is saved in the TechDB and the new name is added to the drop down list. You do not need to save the parameters for this exercise.
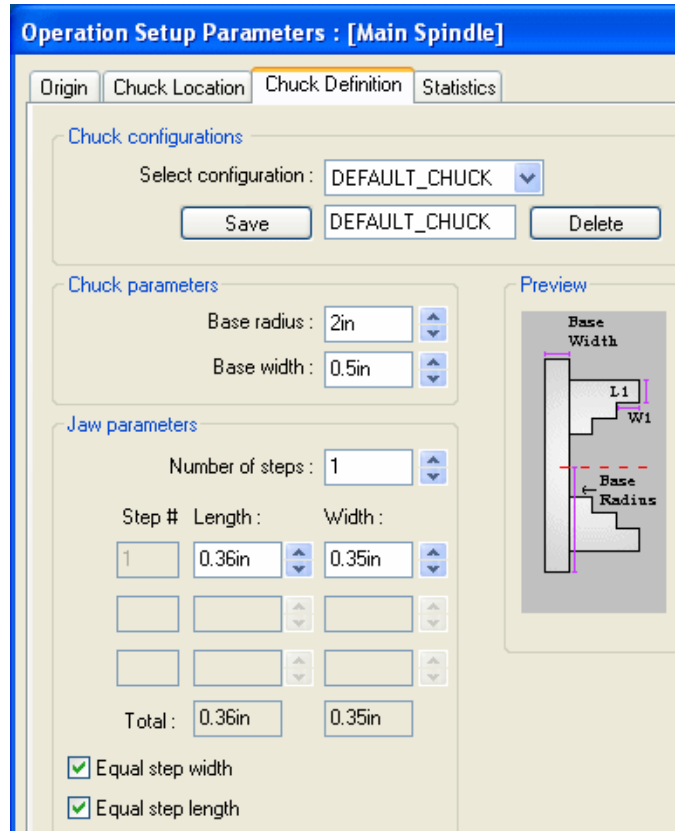
5. Change the following values:
   - Base radius = 2in
   - Base width = .5in
   - Step Length = .36in
   - Step Width = .35in

   The Equal step width and length options apply only when there are multiple steps.

6. Click the Chuck Location tab.

   The Stock boundary displays. The two triangles display at part zero. This is the default position for the locating symbols.

   You can position the two triangles to reflect where the chuck or Collet will be located.

   You can also change the size of the triangles.

7. Pick the OD line and the Stock boundary to position the triangles as shown in the figure.

   A graphic of the part can be added to the setup sheet to give the shop floor a better understanding of the setup.
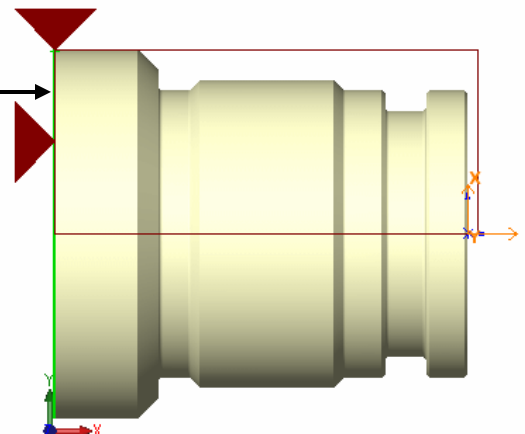
8. Click OK to close the dialog box.

   **Did You Know ...**
   If you change the Chuck Definition, you need to pick the chuck location again on the Chuck Location tab.

9. Change the view orientation to Isometric.

10. Right click Thread2 in the tree and select Suppress.

    You do not need to simulate this operation.

11. Right click Turn Setup1 and select Simulate Toolpath.

    ⬛ The Fixture Shaded Display button is enabled and the chuck that you just defined displays.

12. ⬛ Click the Section View button and select Half.

13. ⬛ Click the Target button and select Translucent Display.

14. ▶ Run the simulation.

15. ❌ Exit the simulation.

16. Right click on Turn Setup1 in the Operation tree and select Edit Definition.

17. Click the Chuck Definition tab.

18. Try different settings: change the number of steps and the size of the steps and the base, then click OK.

19. Run the Toolpath Simulation again and view the changes.
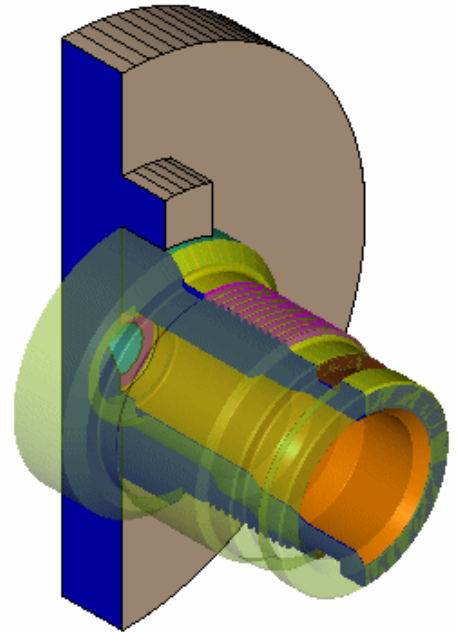
# Turn 6

Double chucking is a method that allows the programmer to machine a Stock on both sides of the part. Double chucking operations are supported in CAMWorks turning. The method for doing a double chucking operation is explained below.

## What You'll Learn

- Defining the Stock from a sketch for double chucking
- Establishing Part Zero
- Defining Machinable Features
- Generating an Operation Plan
- Changing the origin machining direction and generating toolpaths
- Defining the chuck location and simulating toolpaths

In this exercise, assume that Side 1 and Side 2 are established as shown below. Side 1 will be machined first and you will be chucking on side 2. Since the material is a solid billet, you will be locating on the billet OD on side 2.

1. Open part file **TURN2AX_6.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. Click the CAMWorks Feature Tree tab.

3. Right click the Machine in the Feature tree and select Edit Definition on the shortcut menu.

   – Highlight Example Turn–in and click Select.
   – Click the Tool Crib tab and make sure Inch Turret 1 is the Active tool crib.
   – Click the Controller tab and make sure FANTUTL is selected.

4. Do not close the Machine dialog box.
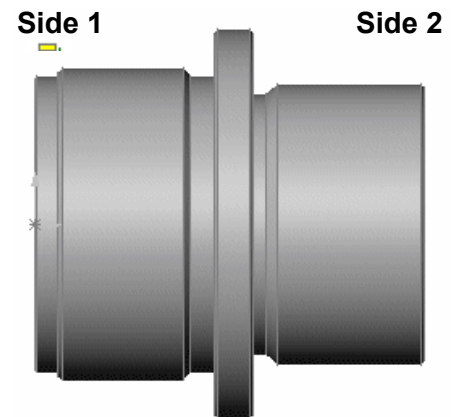
### *Establishing Part Zero*

Since this part is a cylinder, CAMWorks already knows where the centerline is. CAMWorks automatically sets the origin to a default setting and establishes a default centerline. For a single chucking operation, the default origin may be acceptable. Since you are doing a double-chucking operation, however, you need change the default to set the system origin for side 1.

1. Click the Setup tab in the Machine dialog box.

   The origin X and Z symbol displays on the part in the default location (at the face of the part) for Side 2.

2. On the Setup tab, select Coordinate System for the Method.

3. Click the FeatureManager design tree tab and pick Side 1 Origin in the tree.

   The origin marker moves to Side 1.

5. Click OK to close the dialog box.

6. ![CW] Click the CAMWorks Feature Tree tab.
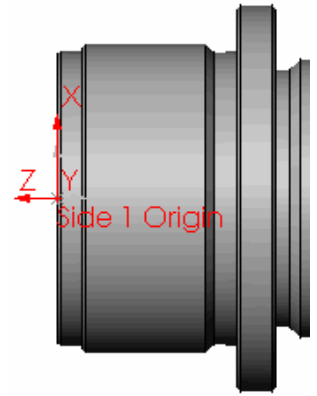
## *Defining the Stock from a Sketch for Double Chucking*

The part is being made from a solid Stock. In order for CAMWorks to define 2 Setups in the tree, a sketch must be used for the Stock. The sketch in this figure is drawn longer than the part on both sides.

1. Right click Stock Manager in the Feature tree and select Edit Definition.

2. In the Manage Stock dialog box, select Revolved for the Type.

3. In the Available sketches list, select Sketch6.

   Selecting a sketch establishes 2 distinct faces (or ends of the part). Because bar stock has an infinite side length and one face, you cannot select Bar stock for double chucking. The sketch representing the shape of the stock is highlighted on the part model.

4. Click OK.

## *Defining Machinable Features*

1. ![icon] Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

   Two Turn Setups are created and AFR recognizes the features that can be machined in each Setup.

   AFR found an OD Feature and an ID Feature in Setup 1 that each extend from the front face to the back face. However, to machine this part in a double-chucking environment with 2 Setups, you need an ID Feature and an OD Feature in each Setup. In order to accomplish this, you edit the feature definitions in Setup 1 and create new features in Setup 2.

2. Right click OD Feature1 in the tree and select Edit Definition.

3. In the Edit Turn Feature dialog box, make sure the Window selection option is checked.

4. Window pick the segments of the part as shown in the figure.

   The segments are removed from the Selected entities list.
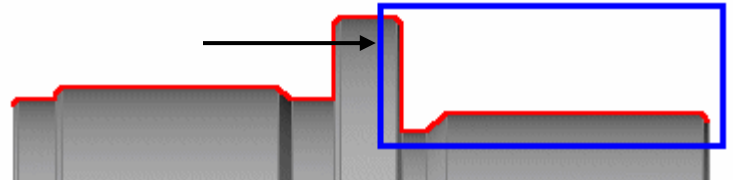
5. Click OK.

6. Right click Face Feature2 in Setup2 and select Insert Turn Feature.

7. In the Insert Turn Feature dialog box, make sure the Window selection option is checked.

8. Window pick the same segments that you removed from OD Feature1 as shown in the figure above.

9. Click OK.

   OD Feature2 is added to Setup2.

10. Right click ID Feature1 in the tree and select Edit Definition.

11. In the Edit Turn Feature dialog box, make sure the Window selection option is checked.

12. Window pick the segments of the part as shown in the figure.

    The segments are removed from the Selected entities list.
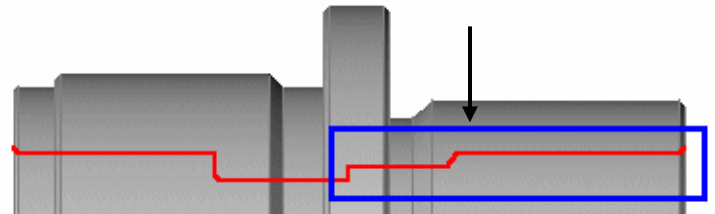
13. Click OK.

14. Right click OD Feature2 in Setup2 and select Insert Turn Feature.

15. In the Insert Turn Feature dialog box, make sure the Window selection option is checked.

16. For the Feature info, select ID Feature.

17. Window pick the same segments that you removed from ID Feature1 as shown in the figure above.

18. Click OK.

    The OD and ID features in Setup2 should look like the features in the figure on the right.
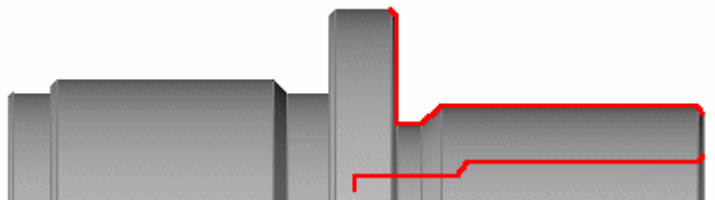
## Changing the Origin Machining Direction and Generating Toolpaths

1. ⬇ Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the Feature tree and select Generate Operation Plan.

   Operations are created for the machinable features in the Turn Setups and are listed in the Operation tree.

2. Left click each Turn Setup in the tree and notice the machining direction indicated by the datum that displays on the part model.

3. The origin is at the back of the part for both Setups. Turn Setup1 is correct; however, the origin needs to be changed for Setup2.

4. Right click Turn Setup2 in the Operation tree and select Edit Definition.

5. On the Origin tab, select Automatic for the Defined from option, then click OK.

   The origin moves to the front face.

6. Right click Drill1 in the tree and select Edit Definition.

7. On the Feature Options tab, change the Machining length to 5.2in.

   Increasing this length will drill the part all the way through.

8. Click the Add tip length check box.

   With this option selected, CAMWorks takes into account the drill point angle when generating the toolpath.
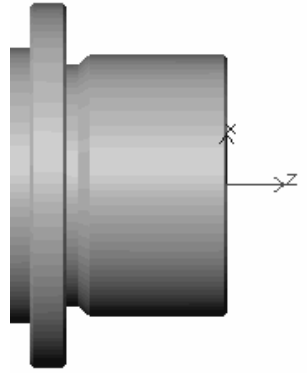
9. Click OK.

10. Right click Drill2 in the tree and select Edit Definition.

11. On the Feature Options tab, change the Machining length to 5.2in and click the Add tip length check box.

12. Click OK.

13. In Turn Setup2, hold down the Shift key and click the Center Drill2, Drill3 and Drill4 operations in the tree.

14. Press the Delete key.

15. Click Yes to confirm the deletion.

    As a result of the changes to the drill operations in Setup1, you can delete these operations because there will no longer be stock on the face.

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

17. Click each operation in the tree to view the toolpaths.

## *Defining the Chuck Location and Simulating Toolpaths*

You are now ready to define the chuck location for each Setup and run the Toolpath Simulation.

1. Right click Setup1 in the tree and select Edit Definition.
2. Click the Chuck Definition tab and select 6in_2Step_Chuck for the configuration.
3. Change the Length and Width to .5in.
4. Click the Chuck Location tab.
5. Pick the horizontal line representing the OD of the Stock, then pick the vertical line representing the face of the part.

   The triangles move as shown in the figure.
6. Click OK.
7. Right click Turn Setup2 in the tree and select Edit Definition.
8. Click the Chuck Definition tab and use the same configuration and parameters that you used for Setup1.
9. Click the Chuck Location tab.
10. Locate the triangles by picking the small OD on the left side and the vertical edge for the chuck face.

    Note that you may need to zoom in to this area in order to pick these segments easily.
11. Click the Reverse direction option.
12. Click OK.
13. ![button] Click the Simulate Toolpath button on the CAMWorks toolbar.

    ![button] The Fixture Shaded Display button is enabled and the chuck defined for Turn Setup1 displays.
14. Click the down arrow next to End and select Next Setup.
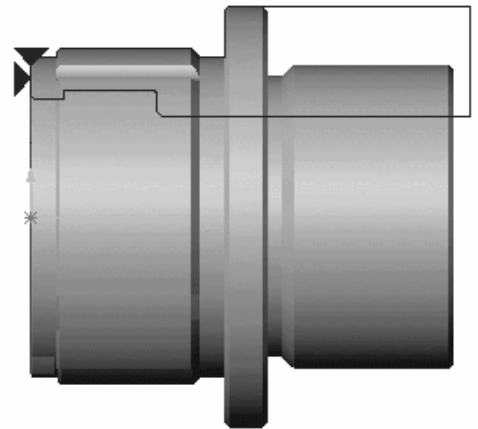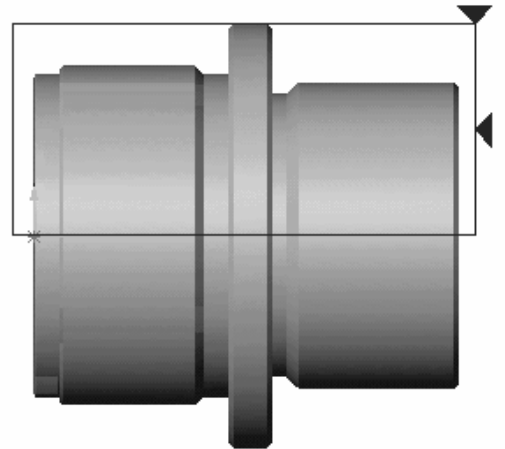
15. Select the following display options:

    [icon] Stock Translucent Display

    [icon] Target Shaded Display

16. Rotate the part so you can see the part being machined in Setup1.

17. [icon] Click Run.

    The simulation runs through the operations in Turn Setup1, then the chuck display changes to Turn Setup2.

18. Rotate the part so you can see the part being machined in Setup2.

19. [icon] Click Run

    The toolpaths for the operations in Turn Setup2 are simulated.
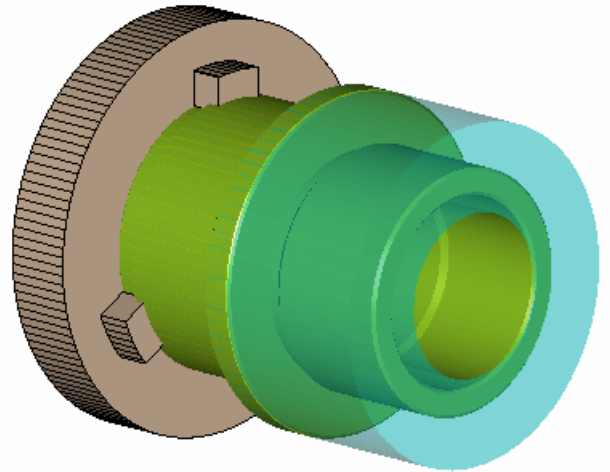
# Turn 7

Turn features that are recognized automatically and created interactively are based on the spindle centerline and turn section plane. CAMWorks provides two methods for AFR. Generation 2 uses a more advanced method of recognizing OD features and in most cases is capable of calculating the minimum rotary silhouette of the OD of the part. Generation 2 is the preferred method. However, if Generation 2 does not find features correctly, you can use Generation 1.

When Generation 1 is used, CAMWorks sections the part through the centerline and along the X axis. If this section plane does not cross through all areas of the part, the resulting features will not be correct. The Define Turn Section Plane command allows you to define the section plane so the features will be correct.

## What You'll Learn

The model in this exercise contains protrusions or ears on the OD of the part. If Generation 2 were used, the OD feature would include the shape of the protrusion. However, in order to learn how to define a section plane, you use Generation 1.

1.  Open part file **TURN2AX_7.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

    Notice the protrusion on the OD of the large diameter of the part.

2.  ![icon] Click the CAMWorks Feature Tree tab.

3.  ![icon] Right click the Machine in the Feature tree and select Edit Definition.

4.  Highlight Example Turn–in and click Select.

5.  Click OK.

6.  ![icon] Click the Options button on the CAMWorks toolbar.

7.  On the Turn Features tab, set the Method to Generation 1 and click OK.

8.  ![icon] Click the Extract Machinable Features button on the CAMWorks toolbar.

9.  Change the view orientation to Front.

10. Click OD Feature1 in the tree and notice that the OD feature does not include the shape of the protrusion.
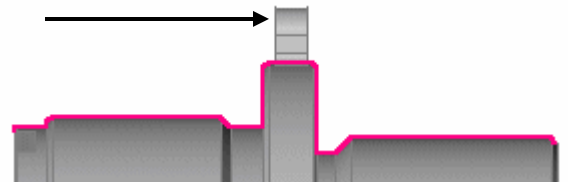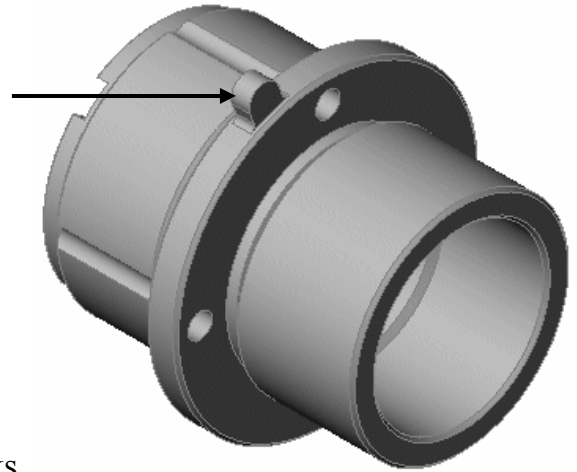
    You need to define a section plane that cuts through the protrusion.

## *Using the Generation 1 Method to Extract Machinable Features*

1. Change the View to Right.

2. Right click NC Manager in the tree and select Define Turn Section Plane.

   The spindle centerline and origin are automatically determined properly for most turning parts. By default, CAMWorks will position the machining origin at one end of the part and set the positive Z machining direction to be in the same direction as the positive SolidWorks X, Y, or Z axis. For this part, the origin is on the front right side of the part. The Turn Section Plane command is used, when necessary, to modify the automatically determined origin, spindle centerline, machining direction and AFR section plane (for Generation 1).

   On the Setup tab in the Machine dialog box, the Setup definition options define the machining origin and the location of the spindle centerline for the part. The origin specifies program zero and the Z axis identifies the location and direction of the spindle axis, which determines the features that will be recognized automatically and created interactively. If the part contains more than one centerline direction, you can define the desired centerline by selecting a SolidWorks coordinate system. In this example, there is only one logical centerline direction and the Automatic setting finds it properly.

   When recognizing features using Generation 1, CAMWorks sections the part through the centerline and along the CAMWorks X axis. If this section plane does not cross through all areas of the part, the resulting recognized features will not be correct.

   By default, the X axis, which defines the section plane, is vertical and will not cut through the protrusion, so the protrusion would not be included in the OD feature.

   The center of the protrusion is 20 degrees counterclockwise off the vertical, so you need to rotate the section plane to cut through the protrusion.

3. For the Turn feature section plane, type 20 for the Angle.

   Notice that the X axis points through the protrusion.

4. Press Tab and notice that the X axis is vertical again. The focus is now in the Turn feature display plane section of the dialog box, which is currently set to 0 degrees. The Turn feature display plane is the plane that the features are displayed parallel to. This plane can be set separately from the Turn section plane. The display plane is provided for viewing purposes only and has no affect on AFR.

5. To graphically verify that the rebuilt OD feature includes the protrusion, type 20 for the display plane Angle and click OK.

6. Click Yes when the message displays asking if you want to rebuild.
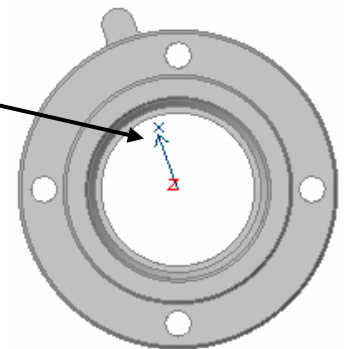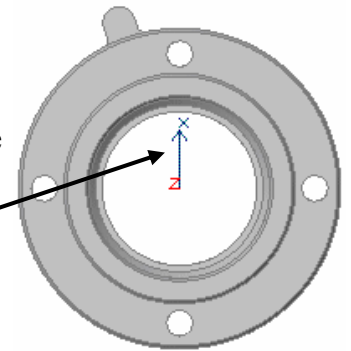
7. Right click Turn Setup1, select Set View on the shortcut menu, then select Turn ZX.

   CAMWorks displays turn features from a single view that is normal to the Turn feature display plane.
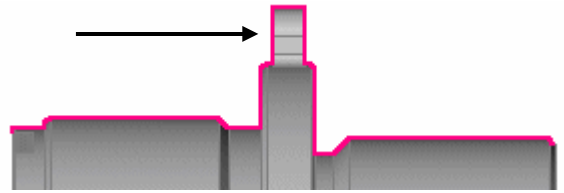
8. Click on the OD feature in the tree to verify that the protrusion is included in the feature.

   In many cases, you may not have a SolidWorks orientation that is normal to the Turn section plane and you may want to view the features relative to a standard orientation. To do this, you can set the Turn feature display plane to be parallel to a standard orientation.

9. Right click Example Turn–in in the tree and select Edit Definition.
   This is another way to access the Setup tab.

10. On the Setup tab, set the Turn feature display plane angle back 0 degrees and click OK.

11. Change the view orientation to Front.

12. Click the OD feature in the tree and notice the difference in the display.

# Turn 8

In this exercise, you define a thread feature and operations to cut double and triple lead threads.

## What You'll Learn

- Defining a Thread Feature
- Cutting Multiple Start Threads

### *Defining a Thread Feature*

1. Open part file **TURN2AX_8.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. Right click Revolve1 in the tree and select Edit Sketch.

3. Notice that separate line segments were created.

   The length is 3 inches. In this exercise you are threading only the first inch at the front of the part. Creating a separate line segment is one method you can use to define the feature being machined for the length of the thread. You could also use one long line and edit the operation's feature length to cut only the first inch.
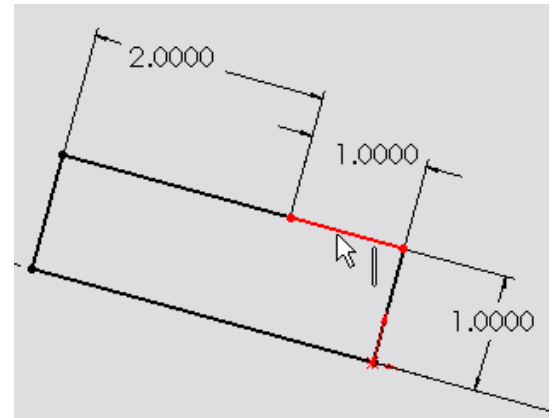
4. Exit the sketch.

5. Click the CAMWorks Feature Tree tab.

6. Right click the Machine in the Feature tree and select Edit Definition.

7. Highlight Example Turn–in and click Select.

8. Click OK.

First, you are going to cut 8 threads per inch double lead with a pitch of .125in. To do this, you create 2 operations set for 4 threads per inch. Since 4 TPI equals a pitch setting of .25in, each operation will have a pitch of .25in.

9. Click the Extract Machinable Features button on the CAMWorks toolbar.

   AFR finds a Face Feature, an OD Feature and a Cutoff Feature. You are not going to use these features, so you can delete them.

10. Select the three features in the tree, then right click and select Delete on the shortcut menu.

11. Click Yes to confirm the deletion.

12. Right click the Recycle Bin and select Empty. Click Yes to confirm.

13. Right click Turn Setup1 in the tree and select Insert Turn Feature.

14. Highlight Sketch1 in the Define from list.

15. Make sure the Feature info is set to OD Feature.

16. Set the Attribute to Thread.

17. Rotate the part so you can see the sketch.

18. Move the pointer over the sketch and pick the 1 inch line segment.

19. Click OK.

20. Right click OD Feature2 in the tree and select Parameters on the shortcut menu.

21. In the OD Profile Parameters dialog box, set the following:
    − Major diameter = 2in
    − Depth = .085in
    − Pitch = .25in

22. Click OK.
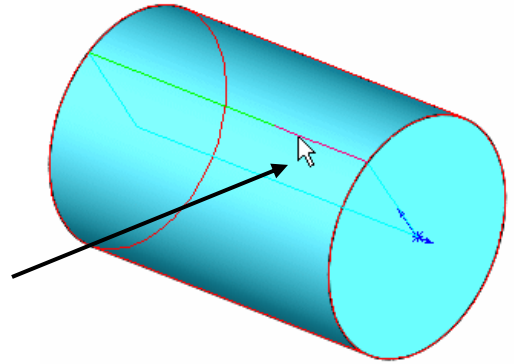
## *Defining Operations for a Double Lead Thread*

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

2. Right click Thread1 in the Operation tree and select Edit Definition.

3. On the Thread tab, change the Depth per cut to .1in.

   **Did You Know ...**
   This depth per cut is greater than the actual thread depth and is used only for viewing toolpaths in this exercise. When programming and cutting parts, you need to consult your Machinist Handbook to use the proper Thread Cuts per pass.

4. Set the Final cut amount to 0.0in.

5. Notice that the Start length is .25in.

   This first operation will have a leadin of .25in. The second operation will have a leadin of .375in. This is necessary because you have to add one Pitch distance to the leadin on each operation in order to cut the other thread leads.

   Therefore: .25in leadin + .125 Pitch of 8 TPI = .375 for the second operation leadin.

6. Click the Preview button to generate and view the toolpath.

7. Click the Expand button to expand the dialog box, then click OK to accept the parameters and the toolpath.
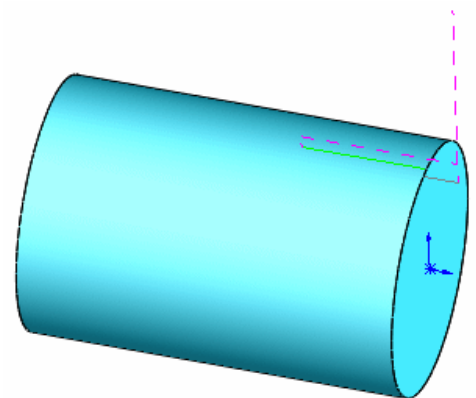
For the second operation, you insert an operation and copy the parameters from Thread1.

8. Right click Thread1 in the tree and select Insert Operation.

9. In the Insert Operation dialog box, select Thread for the Type of operation.

   Thread1 is automatically selected in the Copy parameters from list box.

10. Pick OD Feature2 in the list and click Add.

11. Make sure the Edit definition option is checked and click Insert.

12. On the Thread tab, change the Start length to .375in (see the explanation on the previous page).

13. Click the Preview button to generate and view the toolpath.

14. ⊡ Click the Expand button to expand the dialog box, then click OK to accept the parameters and the toolpath.

15. 🛠 Click the Simulate Toolpath button on the CAMWorks toolbar.

16. Right click Turn Setup1 in the tree, select Set View, then select Turn ZX.

17. Zoom up on the front of the part.

18. ▶ Click Run.

19. Notice there are 8 threads per inch with 2 leads at 4 threads per inch.
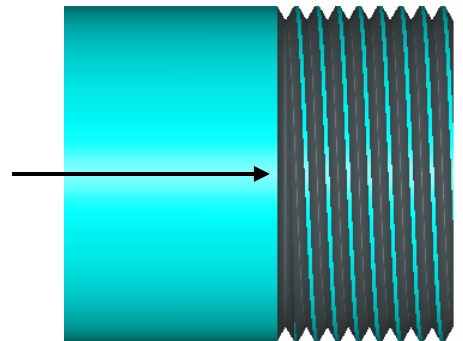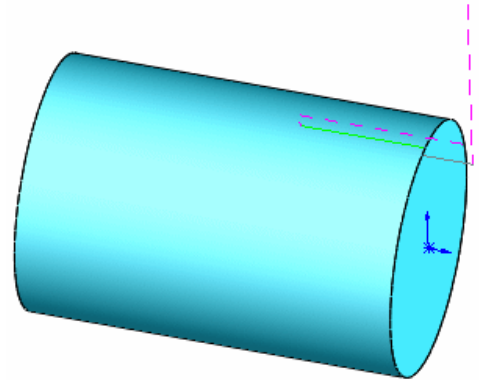
## Defining Operations for a Triple Lead Thread

In this section, you are going to cut 12 threads per inch triple lead with a pitch of .0833in (1 divided by 12 = .0833). To do this, you create 3 operations set for 4 threads per inch. Since 4 TPI equals a pitch setting of .25in, each operation will have a pitch of .25in and the following Start length:

   Operation 1 = .25
   Operation 2 = .25 + .0833 = .3333
   Operation 3 = .25 + .0833 + .0833 = .4166

1. Select the 2 operations in the tree, right click and select Suppress on the shortcut menu.

   You suppress these operations so they will not be simulated when you simulate toolpaths for the triple lead thread operations.

2. 🖳 Click the CAMWorks Feature Tree tab.

3. Right click the OD feature in the tree and select Copy Features.

4. In the Copy Feature dialog box, make sure the Also copy operations option is checked and click OK.

5. Right click OD Feature3 in the tree and select Parameters.

6. Change the Thread depth to .075 and click OK.

7. Right click the feature again, select Generate Operation Plan and click the Update button.

8. Right click Thread3 in the Operation tree and select Edit Definition.

9. On the Thread tab, make sure the Start length is .25in and click OK.

10. Right click Thread4 in the tree and select Edit Definition.

11. Set the Start length to .3333 in (see the explanation on the previous page) and click OK.

12. Right click Thread4 in the tree and select Insert Operation.

13. In the Insert Operation dialog box, select Thread for the Type of operation.

    Thread1 is automatically selected in the Copy parameters from list box. You can copy the parameters from this operation; however all the Thread operations are similar and could be selected.

14. Pick OD Feature3 in the list and click Add.

15. Make sure the Edit definition option is checked and click Insert.

16. On the Thread tab, set the Start length to .4166 and click OK.

17. ▣ Click the Generate Toolpath button on the CAMWorks toolbar.

18. Click Thread3, Thread4 and Thread5 in the tree to view the toolpaths.

19. ▣ Click the Simulate Toolpath button on the CAMWorks toolbar.

20. ▶ Click Run.

    Notice there are twelve threads per inch with three leads at 4 threads per inch.
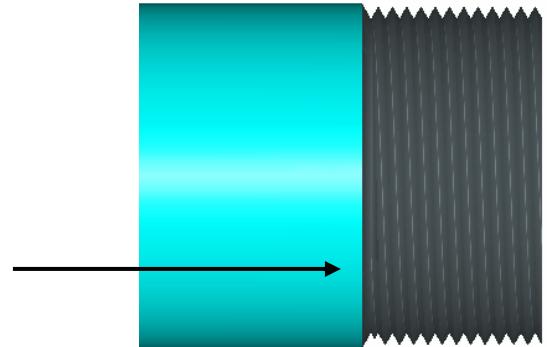
# Chapter 6  Learning 4 Axis Turn

CAMWorks uses the Rear turret for 2 Axis Turn machining and as the default for 4 Axis Turn machining. All tool motion and output is displayed and output from a Rear turret perspective. When the machine has 4 Axis capability, you can define the machine in the Technology Database to use both Rear and Front turrets. Then, you can choose to have the display and output for any of the operations in the Operation tree generated from the Rear or Front turret (providing there is a tool available on the Front turret).

This chapter provides an opportunity to learn CAMWorks 4 Axis Turn. The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

# 4 Axis Turn 1

CAMWorks provides support for machines with front and rear turret configurations allowing 2+2 axis machining. For machines that are defined as 4 axis in the Technology Database, an option is provided to select either Front or Rear turret for each operation. Sync codes are manually provided by the user.

## What You'll Learn

* Defining a 4 Axis machine
* Defining machinable features
* Editing features
* Generating operations and adjusting parameters for front turret
* Generating toolpaths

There are 5 grooves on the part in this exercise: 2 face grooves, 2 OD grooves and 1 ID groove. There is also an OD thread that will be machined.

1. Open part file **TURN4AX_1.SLDPRT** in the *\Examples\Turn* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Turn*).

2. ![CW] Click the CAMWorks Feature Tree tab.

3. ☞ Right click the Machine in the tree and select the Edit Definition command on the shortcut menu.

4. Highlight Example Turn-in 4ax (2+2) and click Select.

   This machine has been defined as a 4 axis machine in the TechDB. Before using CAMWorks to machine your parts, make sure you define the machine tools available in your facility.

5. Click the Tool Crib tab.

6. Click the down arrow next to the Turret type.

   When you select a 4 axis machine, you can choose a tool set to use with each turret. For this part, select Inch Turret 1 as the Active tool set for both Rear and Front turrets.

7. Click the Controller tab and select FANTUTL4, then click OK to close the Machine dialog box.

   This tutorial post processor supports 2+2 axis output. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.

8. Right click the Stock Manager in the tree and select Edit Definition.

9. Leave the Type set to Bar stock.

10. Set the Z Face to -4.05in, the Length to 4.1in and the Diameter to 4.25in, then click OK.
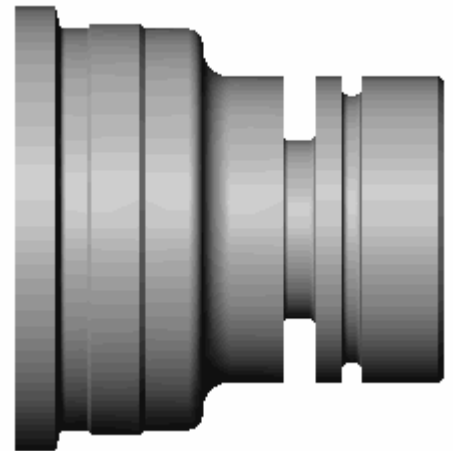
## *Defining Machinable Features Automatically and Interactively*

In order to define the machinable features on this part, you run the Extract Machinable Features command and insert the OD Thread feature.

1.  ▣ Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

    Two Turn Setups are created and AFR recognizes the features that can be machined in each Setup.

2.  Since you are chucking, delete Cutoff Feature1.

3.  Right click on Groove Rectangular OD3 in the tree and select Insert Turn Feature.

    The Insert Turn Feature dialog box displays.

4.  For the Feature info, select OD Feature.

5.  For the Attribute, select Thread from the list.

6.  Pick the horizontal segment shown in the figure.

    The selected entity and two extend entities display in the Selected Entities list. You do not need to extend either end of the segment.

7.  Click OK. OD Feature2 is listed in the tree.

8.  Right click on OD Feature2 and select Parameters.

9.  In the OD Profile Parameters dialog box, set the following thread parameters:
    - Major dia = 3.85in
    - Thread depth = 0.05in
    - Pitch = 0.05in
    - Minor dia = 3.75in

10. Click OK to close the OD Profile Parameters dialog box.
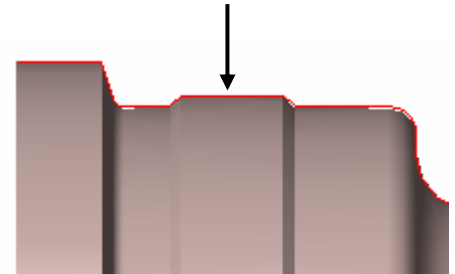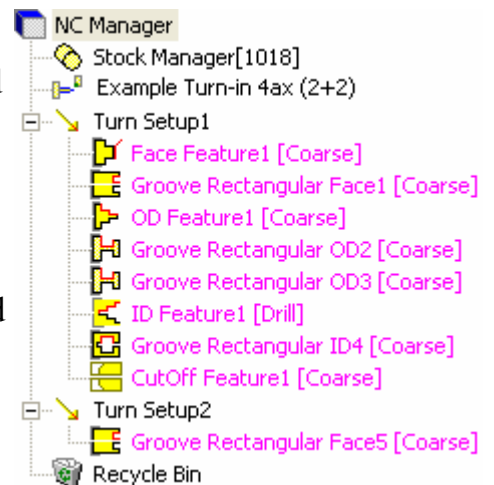
## *Editing Machinable Features*

When you machine a part, the length of the stock may change. Because CAMWorks uses WIP (work in process), you can edit the definition of the OD and ID profiles to make sure these features are cut completely if the stock changes.

1.  Right click OD Feature1 in the tree and select Edit Definition.

2.  Pick the segment shown in the figure to remove it from the feature.

3.  Pick Extend 1 and select Along Z.
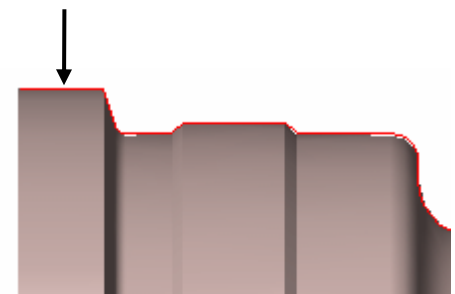
4. Pick Extend 2 in the Selected entities list and select Along X for the Extend option, then click OK.
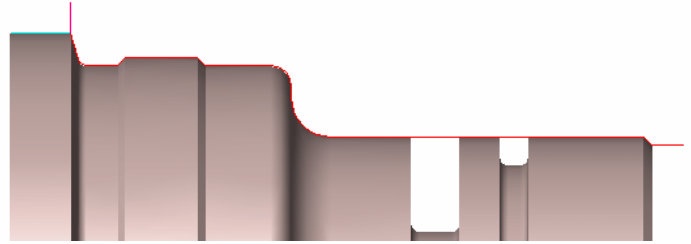
5. Right click ID Feature1 and select Edit Definition.

6. Pick Extend 1 in the Selected entities list and select Along X.

7. Click the Reverse option to point the line down.
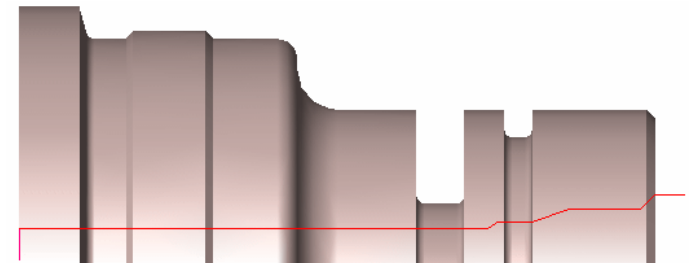
8. Pick Extend 2 in the Selected entities list and select Along Z, then click OK.

9. Right click Turn Setup 2 and select Insert Turn Feature.

10. For the Feature info, select Face.

11. Pick the vertical edge shown in the figure, then click OK.

12. Right click Face Feature2 and select Insert Turn Feature.

13. Leave the Feature info set to OD Feature and the Attribute set to Coarse.

14. Pick the horizontal edge shown in the figure.
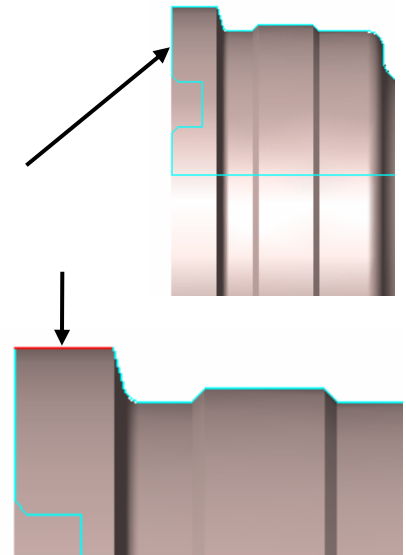
15. Click OK.

## *Generating Operations and Adjusting Parameters for Front Turret*

1. ⬇ Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager in the Feature tree, select Generate Operation Plan.

   Operations are created for the machinable features in the Turn Setups.

2. Right click Turn Rough1 in the Operation tree and select Edit Definition.

3. On the Rough 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. When checked, the toolpath follows the entire toolpath observing the back angle.

4. Right click Turn Finish1 and select Edit Definition.

5. On the Finish Turn tab, check the Undercut option and click OK.

6. Right click on Rough Groove2 (Groove Rectangular OD2) and select Edit Definition.

7. On the Rough Groove tab, make the following changes:

   − Change the Groove peck type to Constant. With this option selected, the tool feeds in the First peck amount, then continues feeding in the Sub peck amount until the depth is reached.

   − Set the First peck amt. and the Sub peck amt to .05in.

   − Make sure the Groove style is set to Normal. This option generates a toolpath where the tool plunges perpendicular to the bottom of the groove. Multiple parallel plunge cuts will be produced when the width of the groove is wider than the tool. The sequence of the cuts is based on the specified order. Using the optional Deep groove cycle, the tool can be retracted between each depth of cut to allow chip removal.

   − Check the Deep groove cycle option. With this option selected, the tool pecks at a specific depth (much like in the Cutoff cycle). This cycle is used for machining grooves that are unusually deep and require using a thin insert that would break if you could not peck the groove.

8. Click the Tool tab.

9. On the Tool Crib page, select Front for the Turret option.

10. Highlight the tool in Stn No. 4 and click Select.

11. On the Groove Insert page, set the Length to 1in.

12. Click the Holder page and click OK when this message displays: The current insert/holder is not allowed for the Main Spindle while using the Front Turret. CAMWorks has selected a valid orientation. Please review this selection on the Holder page and change if required.

13. On the Holder page, make sure the Orientation is set to Up right or Up left, set the Protrusion to .65in and click OK.

14. Right click Finish Groove2 in the tree, select Edit Definition.

15. On the Tool Crib page, select Front for the Turret option.

16. Highlight the tool in Stn No. 4 and click Select.

17. Click the Holder tab and click OK when the holder orientation message displays.

18. Make sure the Orientation is set to either Up right or Up left and click OK.

19. Right click Rough Groove4 (Groove Rectangular ID4) and select Edit Definition.

20. On the Rough Groove tab, change the Radial (X) and Axial (Z) allowances to 0.0 and click OK.

21. Right click Finish Groove4 in the tree, select Delete and click Yes to confirm the deletion.

22. Right click Drill1 and select Edit Definition.

23. On the Feature Options tab, change the Machining length to 4.5in, check the Add tip length option and click OK. Increasing this length ensures that the full diameter of the drill will pass through the back of the part.

24. Click the Generate Toolpath button on the CAMWorks toolbar or right click NC Manager and select Generate Toolpath.

## *Defining the Chuck Location for Setup1*

Define chuck to use for toolpath simulation:

1. Right click Setup1 in the tree and select Edit Definition.
2. Click the Chuck Definition tab.
3. Click the down arrow to the right of the Select configuration input box and select 8in_2Step_Chuck.

   This is one of several chuck configurations that has been saved in the TechDB. You can define additional configurations by setting the parameters on this tab, then typing a name in the input box and clicking the Save button.

4. Click the Chuck Location tab.
5. Pick the OD line on the work-in-process (WIP) sketch, then pick the vertical line representing the back (locating) face.

   The triangles move as shown in the figure.

6. Click OK.
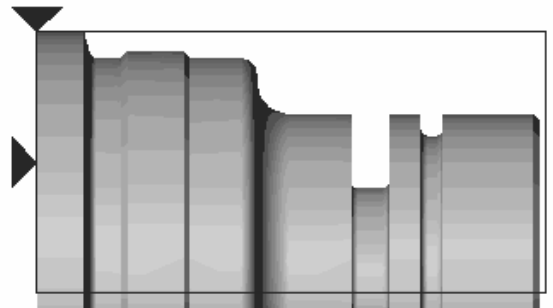
## *Changing the Origin and Defining the Chuck Location for Setup2*

1. Right click Turn Setup2 in the tree and select Edit Definition.
2. On the Origin tab, change the Origin Defined from option to Automatic.
3. Click the Other end option.

   The origin marker displays as shown in the figure.

4. Click the Chuck Definition tab and select 8in_2Step_Chuck.
5. Click the Chuck Location tab.
6. Locate the triangles by picking the OD line on the work-in-process (WIP) sketch near the small OD on the right side and the WIP sketch at the back vertical edge of the small diameter for the chuck face.

   Note that you may need to zoom in to this area in order to pick easily.

7. Click the Reverse direction option.
8. Click OK.
9. ⬜ Click the Generate Toolpath button on the CAMWorks toolbar to regenerate toolpaths for both Setups.

   ### 💡 *Did You Know ...*
   If you change the Chuck Definition, you need to pick the chuck location again on the Chuck Location tab.
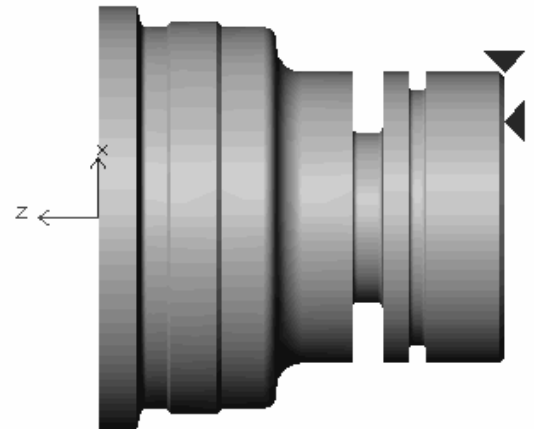
## *Simulating the Toolpaths for Turn Setup1 and Setup2*

1. Right click Thread1 in the tree and select Suppress.

   This toolpath does not need to be simulated for this exercise and suppressing it speeds up the simulation.

2. 🔩 Click the Simulate Toolpath button on the CAMWorks toolbar.

   🔲 The Fixture Display button is enabled and the chuck defined for Turn Setup1 displays.

3. Click the down arrow next to End and select Next Setup.

4. Select the following display options:

   🔹 Stock Translucent Display

   🔲 Tool Shaded Display

   🔳 Holder Shaded Display

   🔷 Target Shaded Display

5. ▶️ Click Run.

   The simulation runs through the operations in Turn Setup1, then the chuck display changes to Turn Setup2.

6. Rotate the part so you can see the face with the groove.

7. ▶️ Click Run.

   The operations in Turn Setup2 are simulated.
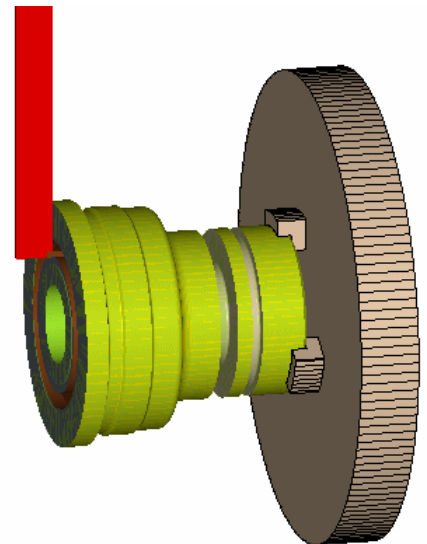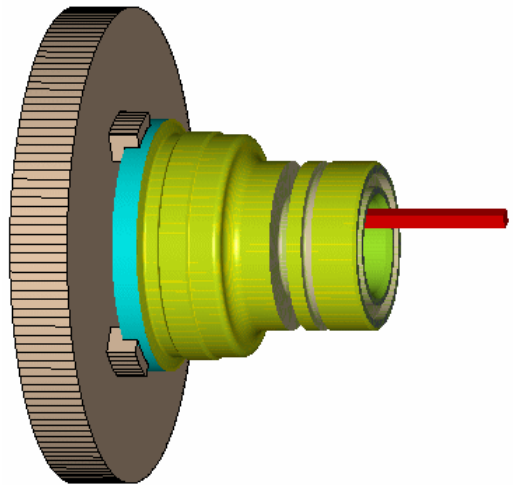
4 Axis Turn 1

# Chapter 7  Learning Assembly Mode

This chapter provides an opportunity to learn to generate Mill toolpaths and NC code in Assembly mode for production machining.

Using SolidWorks Assembly mode, CAMWorks allows you to:

- Position multiple copies of a part in an assembly document and machine the parts with CAMWorks.

- Generate long code or subroutine output to machine each part.

- Store the CAMWorks data within the assembly document. This is particularly valuable for facilities that are ISO 9000 compliant and cannot have non-design data stored with the model.

- Design and layout machine components, parts, stock, clamps, and fixtures to provide a realistic representation of the machining environment.

- Display clamps during simulation with the option to display collisions between the tool and the clamps.

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.

---

# Generating NC Code in Assembly Mode

CAMWorks Assembly mode allows you to position multiple parts and/or multiple copies of a part in an assembly document and generate long code or subroutine output to machine each part.

The following steps are used to generate Mill toolpaths and NC code in Assembly mode:

1. Model the components (parts, clamps, vises, fixtures) and create the assembly document (.sldasm) in SolidWorks.
2. Change to the CAMWorks Feature tree.
3. Define the Machine and Fixture Coordinate System (fixture origin and X,Y,Z axis machining directions).
4. Select parts to be machined.
5. Define the stock (separate or common).
6. Extract machinable features and interactively insert features at Part Setup level.
7. Generate the operation plan and adjust operation parameters.
8. Define G-code program zero location (Part Setup origin or Setup origin).
9. Identify fixtures and clamps.
10. Generate toolpaths.
11. Post process the toolpaths.

Sample assemblies are provided for the exercises in this chapter. When you install CAMWorks, these files are installed automatically.

# Assembly 1

## What You'll Learn

### Defining the Machine and Fixture Coordinate System

1. Open the assembly file **MILLASM_1.SLDASM** in the *\Examples\Assemblies* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Assemblies*).

2. [CW] Click the CAMWorks Feature Tree tab.

3. [icon] Right click the Example Mill–in in the Feature tree and select Edit Definition.

   On the Machine tab, Example Mill–in is the Active machine. This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

4. Click the Tool Crib tab and make sure Crib 1 is the Active tool crib.

5. Click the Controller tab and make sure FANTUTM is selected.

   FANTUTM 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.

6. Click the Setup tab.

   The Setup tab allows you to set a Fixture Coordinate System. This refers to the "home point" or main zero position on the machine. While G-code output can be based on this point, it is meant to be used as a reference point. Setting the position does more than set the location; it also sets the positive X, Y, and Z directions to be used for all moves on this machine. When the code is output in subprogram format, an alternative part setup

   origin is referenced; however, the X, Y, and Z directions set by the Fixture Coordinate System are still used. The Fixture Coordinate System should be set first, before extracting machinable features.

7. Select Coordinate System 1 in the Fixture Coordinate System list.

   The Fixture Coordinate System is defined from a SolidWorks coordinate system entity. For more information on defining a SolidWorks coordinate system, see the SolidWorks documentation.

8. Click OK.
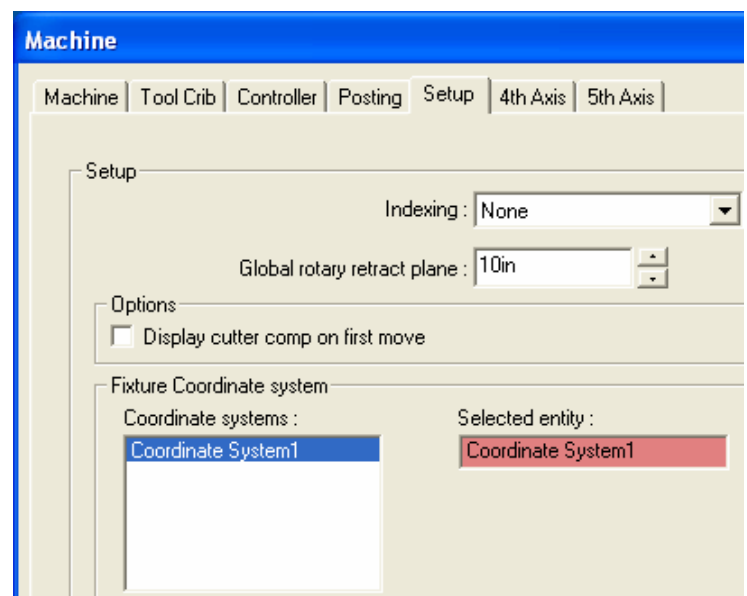
## *Selecting the Parts to be Machined*

The Assembly mode document can contain different part model documents. In addition to the parts that are going to be machined, the document can also include clamps to avoid and other fixture and machine components that are included to assist in the layout of the parts and shop documentation. The parts that are to be machined must be identified to CAMWorks by adding them to the Part Manager. When machining multiple instances of the same part, you must add all instances to the Part Manager.

1. 🎲 Right click the Part Manager in the tree and select Manage Parts on the shortcut menu.

2. Select the part in the lower left corner of the assembly.

   For each unique part in the assembly, the first instance that you select is called the seed part. When an action is performed on the seed part, the same action is applied to every instance of that part in the assembly.

3. Highlight the part in the Selected Parts list and click the Add All Instances button.
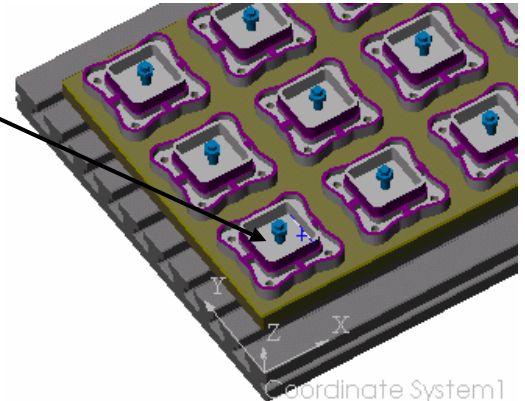
   The parts are listed in the order they are in the file. You can also pick the parts individually in the graphics area or in the SolidWorks FeatureManager tree. Later in this exercise, you use the Sort Instances function to change the machining order.

   Part instances can be added at any time. You can select only one instance of a part (the seed part) to work on first and then add other instances later. Any features, operations and toolpaths that have been generated for the seed part are automatically transferred to instances of the same part when they are added in the Manage Parts dialog box.

4. Click OK to exit the Part Manager dialog box.

   🎲 The part name is listed under the Part Manager in the CAMWorks Feature tree.

   ♟ A Feature Manager, which is created for each part, is used to define the Part Setups and machinable features associated to the seed part.

   🎲 For each unique part, all the instances are listed under the Instances item.

## *Defining the Stock*

When you add parts in the Manage Parts dialog box, a default Stock is created for each part based on a 0.00 bounding box offset. The Stock Manager allows you to customize the stock associated to the parts.

1. 🎲 Right click on the Stock Manager in the tree and select Edit Definition on the shortcut menu.

   The Manage Stock dialog box displays. This dialog box allows you to modify existing stock or create new stock for single parts and common stock for multiple parts.

2. Click the first part in the Parts list box. This is the seed part.

The associated stock is highlighted in the Stock list and in the graphics area. The current settings for the Bounding box offset display.

3. Change the +Z to **.1in** and click the Apply Current Stock Definition to All Parts button.

   The change is applied to the stock for all part instances.

4. Click OK to close the dialog box.

## *Defining Machinable Features*

You can now extract the machinable features. For this exercise, you insert one feature to complete the part. Machinable Features are added in the Feature Manager area of the tree. At the Part Setup level, features can be inserted interactively using the Insert 2.5 Axis or Insert Multi Surface Feature commands. The feature is inserted on the first instance of the part (called the seed part) and on every occurrence of that part in the assembly or you can use the Assembly Feature command on the shortcut menu to insert the feature only one time at its defined position.

1. 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.

2. Click the plus sign next to the Feature Manager in the tree.

   The Feature Manager displays all the Mill Part Setups and machinable features that can possibly be machined on this part regardless of the machine setup.
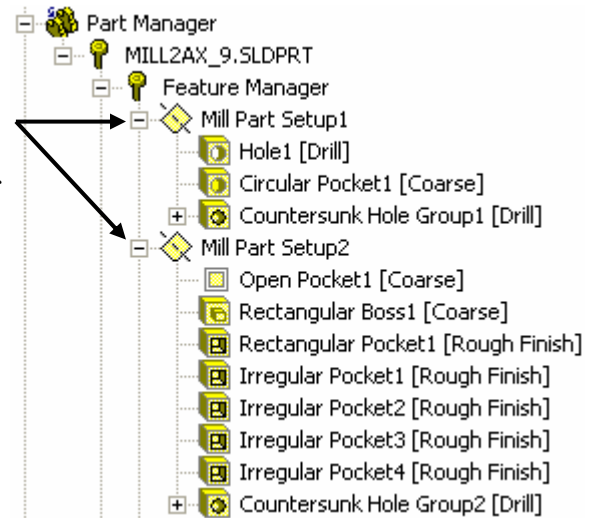
   If the Hole1 feature is listed in Mill Part Setup2, drag it to Mill Part Setup1. This exercise assumes the hole has already been drilled.

3. Right click Mill Part Setup2 under the Feature Manager and select Insert 2.5 Axis Feature on the shortcut menu.

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

4. Change the Feature type to Boss.

5. Make sure the Edge selection is set to Convert to loop.

6. Zoom up on the seed part (the first part you picked when you picked the parts to machine) and pick the bottom edge of the part.

7. Click the Next button.

8. Pick the top face of the lower portion of the seed part.

9. Click the Finish button.

10. When the Wizard returns to the start, click Close.

    The Irregular Boss feature is added to the seed part and to every instance of the part.
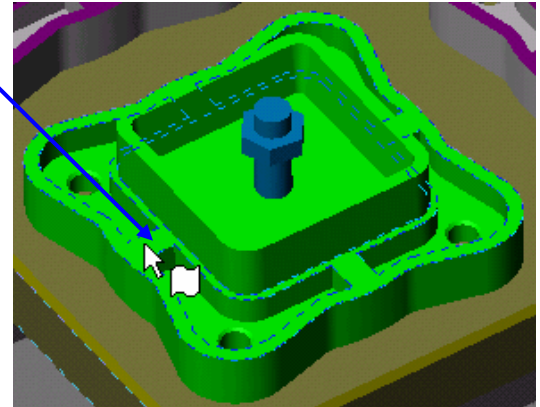
11. Expand Setup1 at the bottom of the tree.

    When you ran AFR, CAMWorks created the Setups that define the machining directions that will actually be approached for the current CAMWorks machine. For 3 axis machines, the maximum number of Setups is one.

    Setup1 all the features that can be machined perpendicular to the Z axis that was specified in the coordinate system.

## Sorting Part Instances to Determine Machining Order

When you add part instances individually or using the Add All Instances button, the instances may not be listed in the best machining order. CAMWorks provides options for sorting part instances to be processed in a more efficient order.

1. Under Setup1 in the tree, click the + symbol next to several features.

   The order that the part instances are listed under each feature is the processing order for that feature. For all features, the parts are currently listed in the order the parts are in the file. You can change the order globally for all features or for individual features.

2. Right click the Part Manager in the tree and select Manage Parts.

3. Click the Sort Instances button in the Part Manager dialog box.

   The Sort Instances dialog box displays options for sorting the part instances for features in the Setup either manually or automatically.

   – The Part Manager instances option automatically sorts part instances for all features in the Setup based on the user-defined order of instances listed in the tree under the Part Manager. To set the order using this option, expand the Part Manager and Instances items, then drag and drop the part instances.

   – Grid pattern automatically sorts part instances for all features in the Setup based on the start corner, processing direction and process order.

   – The Feature instances option allows you to manually sort the part instances listed under each feature in the Setup. To set the order using this option, expand a feature in the Setup, then use drag and drop to move the part instances.

> 💡 *Did You Know ...*
> You can use one of the automatic methods, then if necessary, select the Feature instances option and make changes to the part order for individual features.

4. Select the Grid pattern option.

   When you use the Grid pattern option, the order is changed for the part instances under every feature in the Setup.

5. Select the following grid options, then click OK:
   - Start corner = Bottom left
   - Processing direction = Horizontal
   - Process order = Zigzag

6. Click OK to close the Manage Parts dialog box.

7. Click the + symbol next to a feature in the Setup and click each part instance to view the machining order on the assembly in the graphics area.

8. Try to drag and drop a part instance under the feature to a different position in the list and notice that the order does not change.

   In order to change the order manually for an individual feature, you need to open the Sort Instances dialog box and select Feature instances, then you can drag and drop the instances under the feature. For this exercise, you can use the grid pattern order.

## *Generating the Operation Plan*

1. ◆ Right click Part Setup2 in the tree and select Edit Definition on the shortcut menu.

   The Part Setup dialog box displays.

2. On the Origin tab, select Top center for the location of the origin for all parts, then click OK.

   The origin marker displays at the top center of the seed part.

3. ▤↓ Click the Generate Operation Plan button on the CAMWorks toolbar or right click the NC Manager at the top of the tree and select Generate Operation Plan.

   In the Operation tree, the Setup lists the operations.
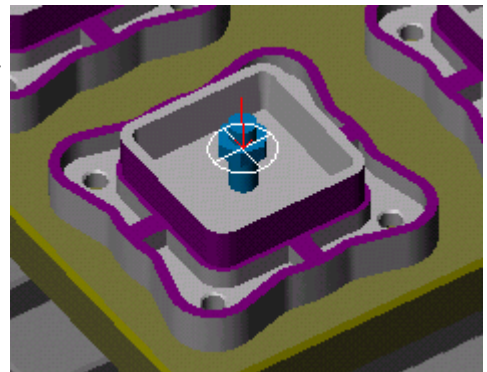
## *Defining G-code Program Zero Location*

Toolpaths can be output relative to the Part Setup origin or a global Setup origin. In this exercise, you use the Part Setup origin. The Part Setup origin specifies only the toolpath zero point, not the X,Y,Z machining direction. The machining direction is based on the Fixture Coordinate System. When machining multiple instances of the same part, the origin is defined relative to the first (seed) part and referenced for all other instances of the same part.

1. ◈ Right click Setup1 in the tree and select Edit Definition.

   The Setup Parameters dialog box displays.

2. On the Origin tab, make sure Part Setup origin is selected for the Output origin.

   Note that when Setup origin is selected, you can specify the origin using several methods.

3. Click the Offset tab.

   The order of the parts on this page affects only the assignment of the offsets, not the machining order.

4. In the Sort by section, select Grid pattern.

   When this option is selected, the parts are sorted in a grid pattern. When you pick this option, the parts in the table are automatically reordered based on the current settings for Start corner, Direction and Pattern.

5. Set the Grid pattern parameters to the same settings you used when sorting part instances for the machining order (page 6-7):
   – Start corner = Bottom left (specifies which part, based on a grid layout, will be assigned the register equal to the Start Value)
   – Direction = Horizontal (relative to the Start corner part, the Direction defines which part will be assigned the next offset register value)
   – Pattern = Zigzag (defines the order the offsets are assigned)

6. Notice that the part order is updated in the table.

   You can specify a programmable coordinate offset and assign an offset to each part.

7. Set the Work coordinate offset to Work Coordinate. This option will output G54, G55, etc.

8. Set the Start value to 54 and the Increment to 1.

   For the Start value, specify only the numerical value of the offset and not the G-code prefix.

9. Click the Assign button. The numbers update in the Offset and Sub columns in the table.

10. Click OK to close the Setup Parameters dialog box.

    If a warning message displays, you can click No to continue.

    ### 💡 *Did You Know ...*
    Changing the machining order does not automatically change the offset assignments. If you want the offset order to correspond to the machining order, you need to sort the parts and reassign the offsets on the Offset tab.

## *Identifying Fixtures and Clamps*

Clamps and fixture components are added on the Fixtures tab in the Setup Parameters dialog box. This dialog box allows you to define clamps, bolts, etc., so that machining toolpaths will avoid these areas and to specify the clamps and fixtures that you want displayed during simulation. Fixtures identified to Avoid apply only to 2 Axis Rough and Finish toolpath calculations.

1. ◇ Right click Setup1 in the tree and select Edit Definition again.

2. Click the Fixtures tab in the Setup Parameters dialog box.

3. In the graphics area, pick the bolt holding the seed part.

4. Pick the fixture plate and the machine table.

   The part names display in the Fixtures list and will display during simulation.

5. Highlight the bolt in the list and click the Add All Instances button.

   All the bolts in the assembly are listed and will display during simulation.

6. Click the Avoid check box for the first bolt you picked that holds the seed part.

   You need to check the Avoid check boxes only for clamps, bolts, etc., that touch the seed part. For all other instances of the part, CAMWorks automatically avoids whatever you select to avoid for the seed part.

7. Make sure the Avoid check boxes are *not* checked for the fixture plate and the table.

   Although fixtures are 3 dimensional SolidWorks parts, CAMWorks considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and finish operations. Therefore, parts such as vices, the machine table, or rotary fixtures whose silhouettes are larger than the part must not be selected to avoid, otherwise no toolpath will be generated.

8. Set the Avoid area type to Exact.

   When this option is selected, CAMWorks avoids the exact shape of the part. The Simplified option creates a bounding box around the part that will be avoided.

9. Click OK to close the dialog box.

   If a warning message displays, you can click No to continue.

## *Generating Toolpaths*

After generating toolpaths, you can sort the operations in a logical machining sequence and simulate the material removal.

1. 🔲 Click the Generate Toolpath button on the CAMWorks toolbar or right click Setup1 in the tree and select Generate Toolpath.

2. ◇ Right click Setup1 in the tree and select Sort Operations.

3. On the Process tab, remove the check mark from the Process complete feature option.

4. On the Sort tab, drag and drop operations so that Rough Mill is at the top of the list, followed by Finish Mill, Center Drill, and Drill.

5. Click Apply, then click OK.

   The operations under Setup1 are sorted based in the order on the Sort tab.

6. Right click Setup1 in the tree and select Simulate Toolpath on the shortcut menu.

7. Optionally, click the buttons in the second row of the Simulate Toolpath toolbar and change the display of the stock, tool, tool holder and fixtures (shaded, wireframe, translucent, no display).

8. Click the Run button on the Simulate Toolpath toolbar.

9. When you are finished, click the X button to exit the simulation.

# Assembly 2

## What You'll Learn

### *Multi-Plane Machining in Assembly Mode*

CAMWorks supports 4th and 5th axis rotary (prepositioning) output for milling. The 4th and 5th axis position angles can be user-defined or automatically calculated. Parts requiring multi-plane machining can be programmed in CAMWorks assembly mode.

The following exercise shows you how to use assembly mode to generate the code for a part that will be mounted on a rotary table for machining.

1. Open the assembly file **MILLASM_2.SLDASM** in *\Examples\Assemblies* inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Assemblies*).

2. Click the CAMWorks Feature Tree tab.

Define the machine, the Fixture Coordinate System and the axis that will be used to calculate the 4th axis rotary angle:

3. Right click Example Mill–in in the CAMWorks Feature tree and select Edit Definition.

4. On the Machine tab, Example Mill–in is the Active machine.

5. Click the Tool Crib tab and make sure Crib 1 is the Active tool set.

6. Click the Controller tab and make sure FANTUTM is selected.

7. Click the Setup tab.

8. Select 4 Axis for the Indexing option.

   This parameter defines to CAMWorks the allowable machining directions for the current CAMWorks machine.

9. Select Coordinate System 1 in the Fixture Coordinate System list.

   This refers to the "home point" or main zero position on the machine. While G-code output can be based on this point, it is meant to be used as a reference point. This parameter also sets the positive X, Y, and Z directions to be used for all moves on this machine. The Fixture Coordinate System is defined from a SolidWorks coordinate system entity. For more information on defining a SolidWorks coordinate system, see the SolidWorks documentation.
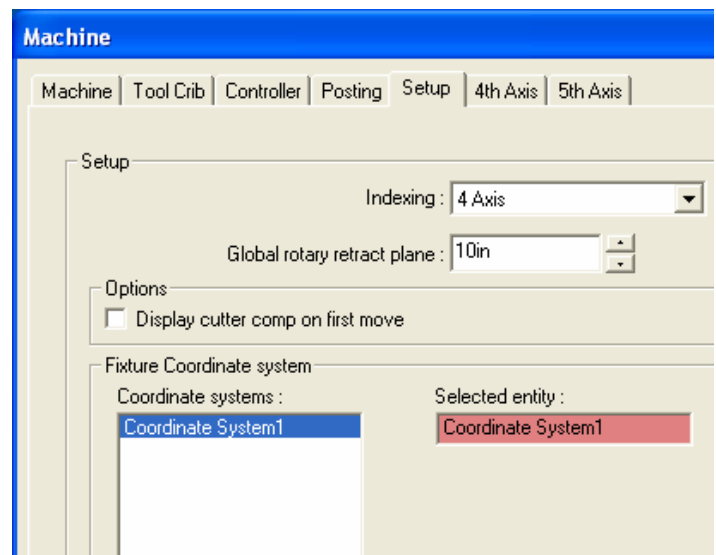
10. Click the 4th Axis tab.

An axis is required in order to calculate the rotary angle. An axis can be defined by selecting a cylindrical face or a SolidWorks axis entity.

11. Click in the Axis entity box and pick the SolidWorks axis in the graphics area.

12. Click in the 0 degree face/plane box and pick a face on the model that is perpendicular to the axis.

This identifies the 0 degree position about the selected axis.

13. Click OK.

Select the parts:

1. 🐞 Right click the Part Manager in the tree and select Manage Parts on the shortcut menu.

2. Pick one of the parts in the graphics area. This is the seed part.

3. Highlight the part in the Selected Parts list and click Add All Instances.

The other part instances are added to the list.

4. Click OK.

🐞 The part name is listed under the Part Manager in the CAMWorks Feature tree.

🔑 A Feature Manager, which is created for each unique part, is used to define the Mill Part Setups and machinable features associated to the seed part.

🐞 For each unique part, all the instances are listed under the Instances item. You can re-order and/or delete the part instances in the tree.
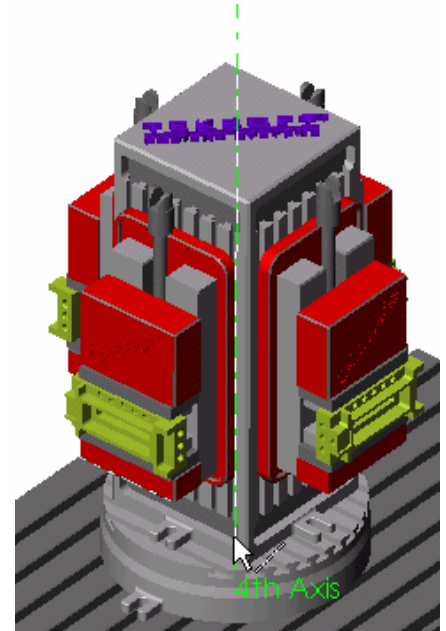
Define the Stock and Extract machinable features:

1. 🐞 Right click the Stock Manager in the tree and select Edit Definition on the shortcut menu.

2. Change the value in the Z- box to **.1** and click the Apply Stock Definition to All Parts button to All button.

3. Click OK.

4. 🖼 Click the Extract Machinable Features button on the CAMWorks toolbar.

When Automatic Feature Recognition is complete, all the Part Setups and machinable features that can possibly be machined on this part regardless of the machine setup are listed under the Feature Manager.

At the bottom of the tree, AFR also creates Setups for the 4 parts on the tombstone. Under each Setup are all of the features that can be machined from that tool direction.

Generate the Operation Plan:

1.  ◆ Expand the Feature Manager, then right click Mill Part Setup1 in the tree and select Edit Definition.

2.  On the Origin tab, select the Top center to set the location of the origin for this Mill Part Setup and click OK.

3.  Repeat steps 1 and 2 for Mill Part Setup2 and Mill Part Setup3.

4.  ▣↓ Click the Generate Operation Plan button on the CAMWorks toolbar.

    The operations that were generated are listed under each Setup in the Operation tree. Notice that the operations have a blue or black link icon. When machining the same feature on different parts at different orientations in Assembly Mode, CAMWorks automatically links the operations for these features so that the feature is machined the same on all parts. The first linked operation in the tree is designated as the "parent" of the set of linked operations and has a blue link.

5.  Right click Rough Mill1 in the Operation tree and select Unlink Operation on the shortcut menu.

    The Delink Operations dialog box lists the three operations that are linked to this parent operation. These are the operations for same feature on the other parts around the tombstone. You can unlink single or multiple operations in this dialog box.
    ⊞ ⬙ Rough Mill1[T9 - 0.25 Flat End]

6.  Click Cancel to close the dialog box.

    You can also select an operation in the tree and unlink it from the parent.

7.  Right click Rough Mill7 in the tree and select Unlink Operation on the shortcut menu.
    ⊞ ⬙ Rough Mill7[T9 - 0.25 Flat End]

    The parent operation is listed in the Selected operations list.

8.  Click Cancel to close the dialog box.

Define the G-code program zero location and identify the clamps and fixtures:

1.  ◆ Right click Setup1 in the Operation tree and select Edit Definition on the shortcut menu.

2.  On the Origin tab, make sure the Output origin is set to Part setup origin.

    This specifies that the origin for the G-code output is relative to the Part Setup origin instead of a global origin.

3.  Click the Fixtures tab.

4.  In the graphics area, pick the tombstone, the rotary table, the machine table and the vise that holds the seed part.

5.  In the dialog box, highlight the vise in the Fixtures list and click the Add All Instances button.

6.  Do not check any of the Avoid check boxes.

Although fixtures are 3 dimensional SolidWorks parts, CAMWorks considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and finish operations. Therefore, do not check the Avoid option for parts such as the machine table, rotary fixtures or vises whose silhouettes are larger than the part. Otherwise, no toolpaths will be generated.

7. Select Exact for the Avoid area type option, then click OK.

If a warning message displays, you can click No to continue.

8. Change the Output origin to Part setup origin for the other Setups (see steps 1 and 2).

Note that the fixture components you picked in Setup1 will display during simulation for all Setups. If a warning message displays, you can click No.

Generate toolpaths and simulate material removal:

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

2. Right click the Machine in the tree and select Simulate Toolpath on the shortcut menu.

3. Optionally, change the stock, tool, tool holder and fixture displays.

4. Click the Run button on the Simulate Toolpath toolbar.

5. When you are finished, click the X button to exit the simulation.

# Assembly 3

## What You'll Learn

### *Machining the Same Parts with Multiple Machine Tools*

Using CAMWorks Assembly mode, the same parts can be machined using multiple machine tools. In this exercise, you set up the first machine to cut the parts on the rear table, then copy the machine and program the parts on the front table.

1. Open the assembly file **MILLASM_3.SLDASM** in *\Examples\Assemblies* inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Assemblies*).

   The parts in this assembly document have been positioned on two tables to be machined using two machine tools.

2. ⬛ Click the CAMWorks Feature Tree tab.

Set up the first machine tool to cut the parts on the rear table:

3. ⬛ Right click Example Mill–in in the CAMWorks Feature tree and select Edit Definition.

   On the Machine tab, Example Mill–in is the Active machine.

4. Click the Tool Crib tab and make sure Crib 1 is the Active tool crib.

5. Click the Controller tab and make sure FANTUTM is selected.

6. Click the Setup tab.

7. Select None for the Indexing option.

8. Select Coordinate System 1 in the Fixture Coordinate System list, then click OK.

   This refers to the "home point" or main zero position on the machine. While G-code output can be based on this point, it is meant to be used as a reference point.
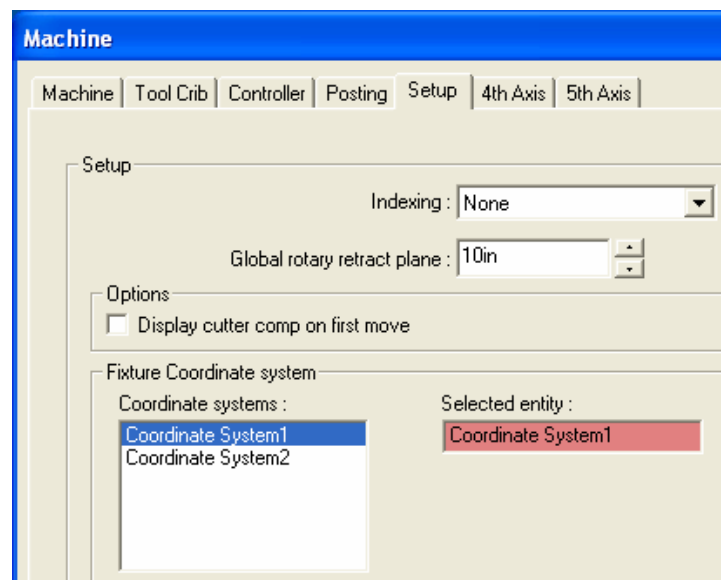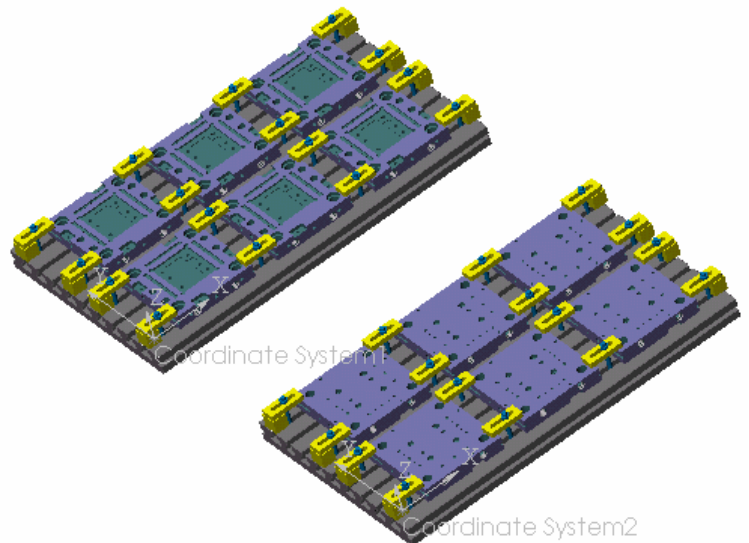
This parameter also sets the positive X, Y, and Z directions to be used for all moves on this machine. The Fixture Coordinate System is defined from a SolidWorks coordinate system entity. For more information on defining a SolidWorks coordinate system, see the SolidWorks documentation.

9.  Right click the Part Manager in the tree and select Manage Parts on the shortcut menu.

10. Pick the six parts on the rear table in the order you want them to be machined.

    The first part that you pick is referred to as the seed part.

11. Click OK.

12.  Right click on the Stock Manager in the tree and select Edit Definition on the shortcut menu.

    The Manage Stock dialog box displays. This dialog box allows you to modify existing stock or create new stock for single parts and common stock for multiple parts.

13. Change the +Z to **.1in**, then click the Apply Stock Definition to All Parts button.

    The change is applied to the stock for all part instances.

14. Click OK to close the dialog box.

15.  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.

    AFR creates the Setups that define the machining directions that will actually be approached for the current CAMWorks machine. For 3 axis machines, one Setup is created.

    Under Setup1 at the bottom of the tree are all the features that can be machined perpendicular to the Z axis that was specified in the coordinate system.

Set up the parts on the front table to be cut by a different machine tool:

1.  Right click Example Mill-in in the tree and select Copy Machine on the shortcut menu.

    A second machine is listed at the bottom of the tree.
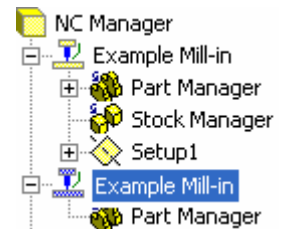
2.  Right click the new Machine item and select Edit Definition on the shortcut menu.

3.  Click the Setup tab and select None for the Indexing option.

4.  Select Coordinate System 2 in the Fixture Coordinate System list, then click OK.

5.  Right click the Part Manager under the second machine in the tree and select Manage Parts on the shortcut menu.

6.  Pick the six parts on the front table in the order you want them to be machined.

    The first part that you pick is referred to as the seed part.

7. Click OK.

   Notice that the Feature tree lists all the Machinable Features for the parts on the front table. Once you have run AFR, all the features for all parts on all sides are available. When you add the parts in the Parts Manager, CAMWorks displays the features automatically.
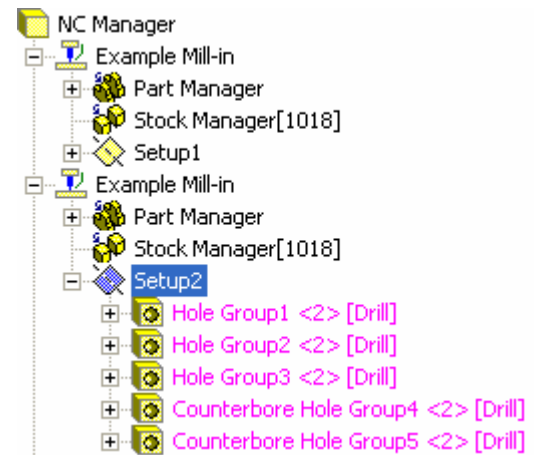
Generate the operation plan and toolpaths for all the parts:

1. [≣↓] Click the Generate Operation Plan button on the CAMWorks toolbar or right click the NC Manager at the top of the tree and select Generate Operation Plan.

   In the Operation tree, Setup1 and Setup2 list the operations.

2. Right click Setup1 in the tree and select Edit Definition on the shortcut menu.

3. Click the Fixtures tab in the Setup Parameters dialog box.

4. In the graphics area, pick the four clamps holding the seed part on the rear machine.

   The part names display in the Fixtures list.

5. Click the Avoid check box next to the four clamp parts in the Fixtures list.

   You need to check the Avoid check boxes only for clamps, bolts, etc., that touch the seed part. For all other instances of the part, CAMWorks will automatically avoid whatever you select to avoid for the seed part.

6. In the graphics area, pick the rest of the clamps and the machine table on the rear machine.

7. Make sure the Avoid check box is *not* checked for the table.

   Although fixtures are 3 dimensional SolidWorks parts, CAMWorks considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and finish operations. Therefore, parts such as vices, the machine table, or rotary fixtures whose silhouettes are larger than the part must not be selected to avoid, otherwise no toolpath will be generated.

8. Set the Avoid area type to Exact.

   When this option is selected, CAMWorks avoids the exact shape of the part. The Simplified option creates a bounding box around the part that will be avoided.

9. Click OK to close the dialog box.

   If a warning message displays, you can click No to continue.

10. Right click Setup2 in the tree and repeat the above procedure to pick the table and clamps for the front machine.

11.  Click the Generate Toolpath button on the CAMWorks toolbar or right click NC Manager in the tree and select Generate Toolpath.

    CAMWorks generates toolpaths for all the operations.

Simulate the material removal and post process the parts:

1. Right click Setup1 in the tree and select Simulate Toolpath on the shortcut menu.

2. Optionally, change the display of the stock, tool, tool holder and fixtures (shaded, wireframe, translucent, no display).

3.  Click the Run button on the Simulate Toolpath toolbar.

4.  When you are finished, click the X button to exit the simulation.

5. Right click Setup2 in the tree and select Simulate Toolpath on the shortcut menu.

6.  Click the Run button on the Simulate Toolpath toolbar.

7.  When you are finished, click the X button to exit the simulation.

8. Right click the first machine in the tree and select Post Process on the shortcut menu.

9. In the Post Output File dialog box, type Rear Machine and click Save.

10. Click the Run button in the Post Process Output dialog box.

    CAMWorks generates the NC code for the parts on the rear table.

11. Click OK when the code is finished.

12. Right click the second machine in the tree and select Post Process on the shortcut menu.

13. In the Post Output File dialog box, type Front Machine and click Save.

14. Click the Run button in the Post Process Output dialog box.

    CAMWorks generates the code for the parts on the front table.

15. Click OK when the code is finished.

# Assembly 4

## What You'll Learn

### *Simulating Castings*

CAMWorks supports irregular shaped stock, such as castings. To define the stock as a casting in Assembly Mode, you can select a SolidWorks part either graphically or from the SolidWorks FeatureManager design tree. Any SolidWorks part can be selected, even a part that is a part to machine.

Create the assembly in SolidWorks:

1.  Start a New assembly file in Inch units.

2.  If the Insert Component PropertyManager is not open, click Insert on the SolidWorks menu bar, then select Component and Existing Part/Assembly.

3.  In the PropertyManager, click Keep Visible to add more than one component.

4.  Click Browse and open **MILL2AX_14 As-Cast.SLDPRT** in *\Examples\Mill* inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

5.  In the graphics area, position the part on the origin and left click to insert it.

6.  Click the Browse button again and open **MILL2AX_14 Machined.SLDPRT.**

7.  Position the part on the origin so that it is on top of the first part and left click to insert it.

8.  In the PropertyManager, click the OK button.
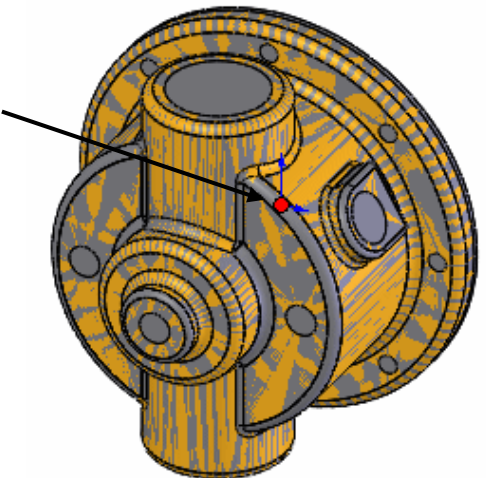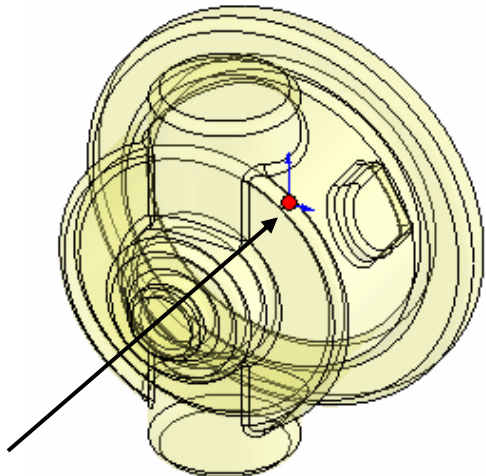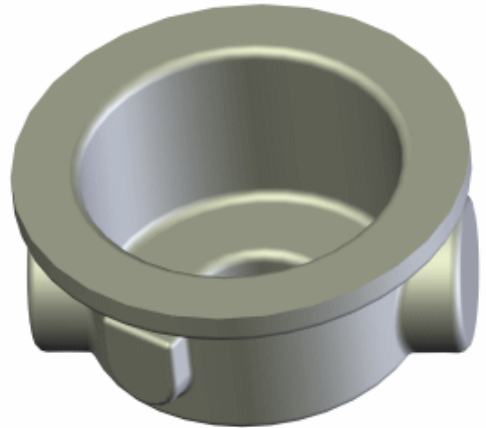
9. In the Assembly window, rotate the assembly so that the orientation is similar to figure on the right.
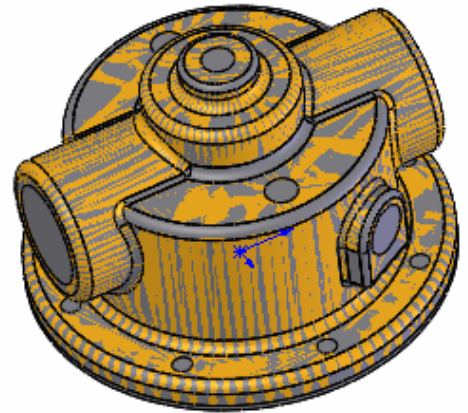
10. ⬛ Click the FeatureManager design tree tab.

11. Right click the Mill2AX_14 As-Cast item in the tree and select Hide on the shortcut menu.

12. Click Insert on the SolidWorks menu bar and select Reference Geometry, then Coordinate System. The Coordinate System PropertyManager displays.

    You need to create a SolidWorks coordinate system that will be used to define the CAMWorks Fixture Coordinate System.

13. Click OK. If no selection is made, the origin of the Coordinate System is the SolidWorks assembly origin. In this example, the default origin works well.

    Next, you are going to create the 4$^{th}$ and 5$^{th}$ axis rotation axes.

14. Pick the face shown in the figure below for the 4$^{th}$ axis.

15. Click Insert on the SolidWorks menu bar and select Reference Geometry, then Axis.

    In the Reference Axis PropertyManager, the Cylindrical/Conical Face option is selected and the list box displays the face you selected.
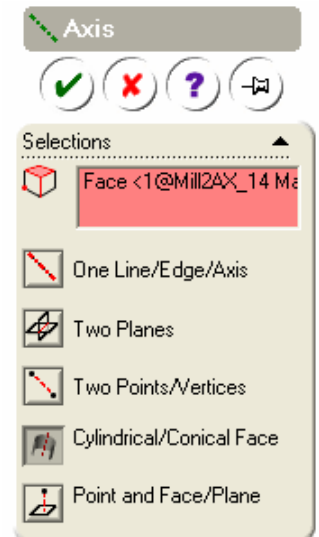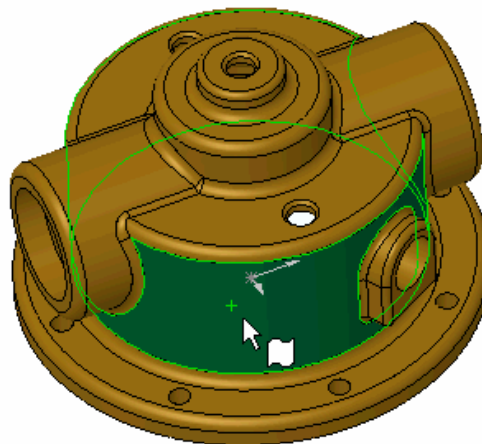
16. Click OK.

    Axis1 displays.

17. Pick the second cylindrical surface shown in the figure on the right for the 5$^{th}$ axis.

18. Select the Axis command again.

    In the Reference Axis dialog, the Cylindrical/Conical Face option is selected and the list box displays the face you selected.

19. Click OK.

    Axis2 displays.

20. Click View on the SolidWorks menu bar and select Axes.
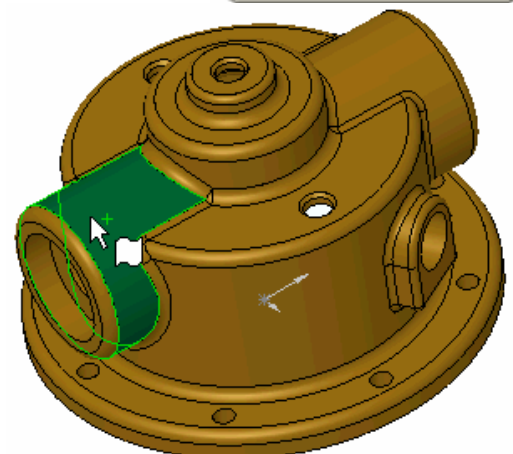
Selecting this command turns on the display of the axes that you just created so they can be selected as the centers of rotation for the 4th and 5th axes of the machine tool.

21. Extend the axis lines to make them easier to select for the CAMWorks machining axes.

In CAMWorks, define the machine, Fixture Coordinate System and axes:

1. **CW** Click the CAMWorks Feature Tree tab.

2. Right click the Example Mill–in item in the tree and select Edit Definition.

3. Click the Setup tab in the Machine dialog box.

4. Set the Indexing option to 5 Axis.

5. Select Coordinate System1 for the Fixture Coordinate System.

**Machine**

Machine | Tool Crib | Controller | Posting | Setup | 4th Axis | 5th Axis

Setup

Indexing : 5 Axis

Global rotary retract plane : 10in

Options
☐ Display cutter comp on first move

Fixture Coordinate system
Coordinate systems :        Selected entity :
Coordinate System1           Coordinate System1
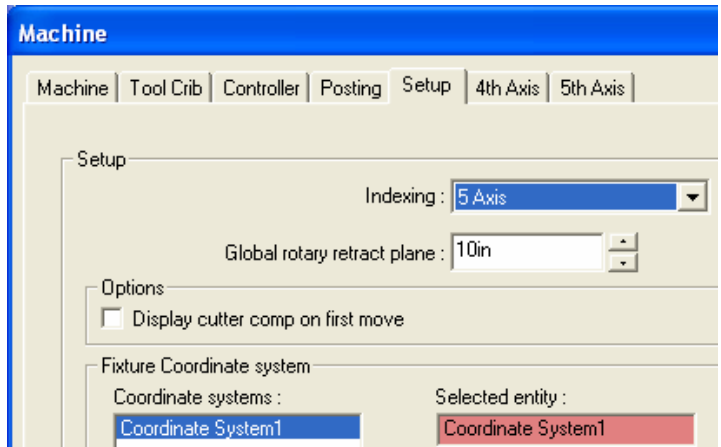
6. Click the 4th Axis tab.

7. Click in the Axis entity box and pick Axis1 in the graphics area or return to the SolidWorks FeatureManager design tree and pick the Axis1 item in the tree.

**Machine**

Machine | Tool Crib | Controller | Posting | Setup | 4th Axis | 5th Axis

4th axis setup

Axis entity :
Axis1@CWAsm          ☐ Reverse direction

0 degree face/plane :
CW ASM Face-0

Rotation direction :
CW
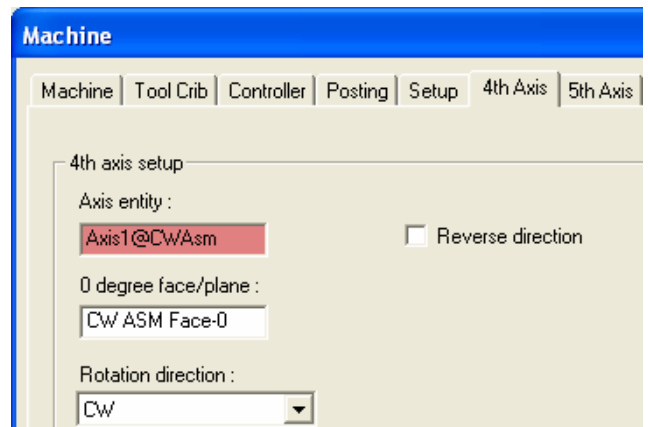
8. Click in the 0 degree face/plane box and pick the face on the part shown in the figure on the right.
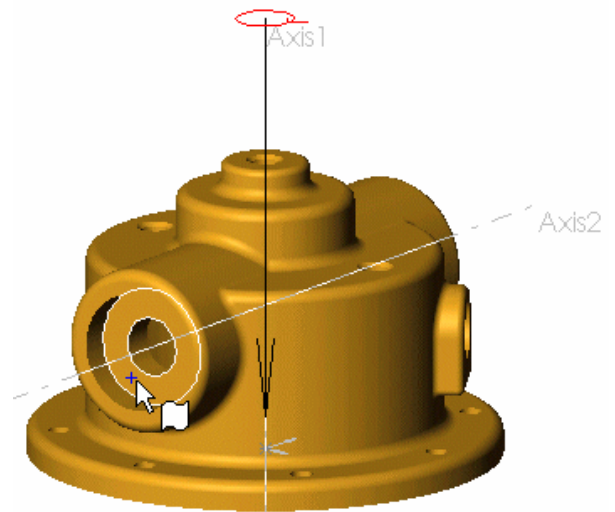
9. Click the 5th Axis tab.

10. Click in the Axis entity box and pick Axis2 in the graphics area or return to the SolidWorks FeatureManager design tree and pick the Axis2 item in the tree.

11. Click in the 0 degree face/plane box and pick the face on the part shown in the figure on the right.

12. Click OK.
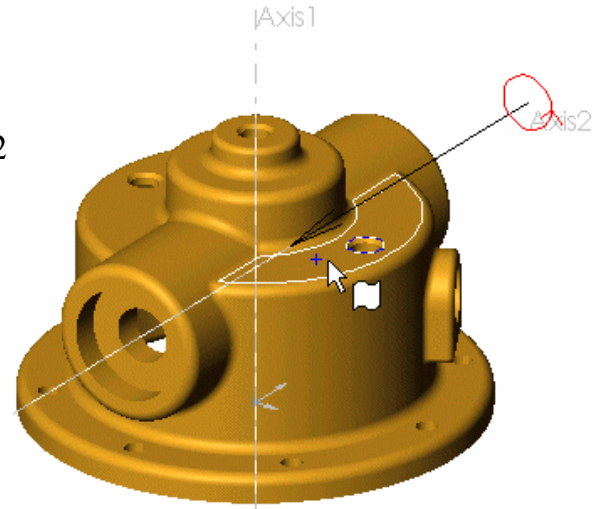
Select the part, define the stock and define Machinable Features:

1. 🐝 Right click the Part Manager item in the Feature tree and select Manage Parts.

   The Manage Parts dialog box displays.
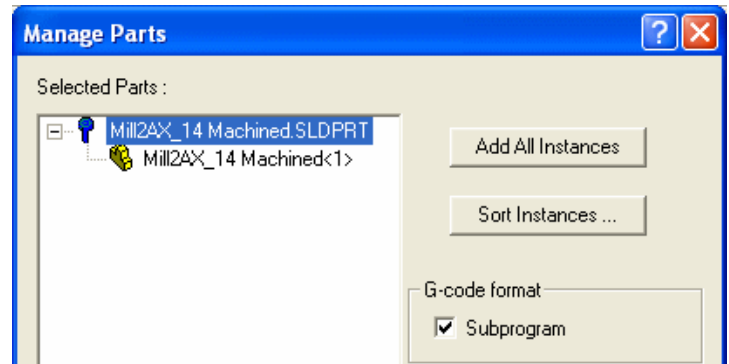
2. Pick the part model Mill2AX_14 Machined in the graphics area.

   The part name displays in the Selected Parts list.

3. Click OK.

4. Right click Stock Manager in the tree and select Edit Definition.

5. For the Stock type, click the SolidWorks part button.

6. 🟩 Click the FeatureManager design tree tab.

7. Pick the Mill2AX_14 As-Cast part in the tree.

8. Click OK to close the dialog box.

9. 🟦 Click the CAMWorks Feature Tree tab.

10. 🟧 Click the Extract Machinable Features button on the CAMWorks toolbar.

11. Expand the Feature Manager item to show all the Mill Part Setups and features found by AFR.

12. Right click MS Hole feature under Mill Part Setup4, and select Delete.

   This feature is inside the casting and will not be machined.

13. Click Yes to confirm the deletion.
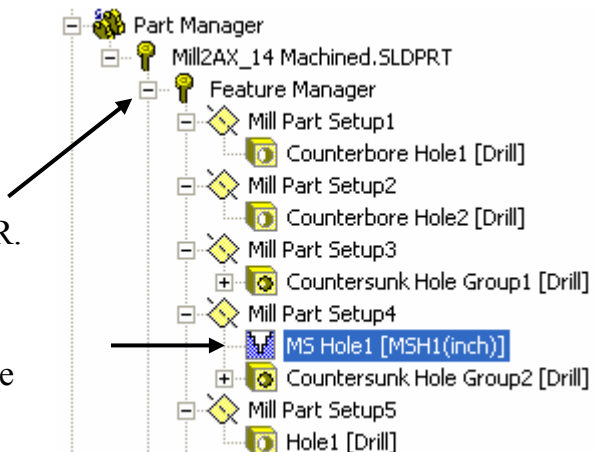
14. ◈ Right click Mill Part Setup4 in the tree and select Insert 2.5 Axis Feature.

15. Rotate the part so you can see the bottom.

16. Change the Feature Type to Face Feature.

17. Pick the bottom face of the part (the face with the circular hole pattern).

18. Click Next.
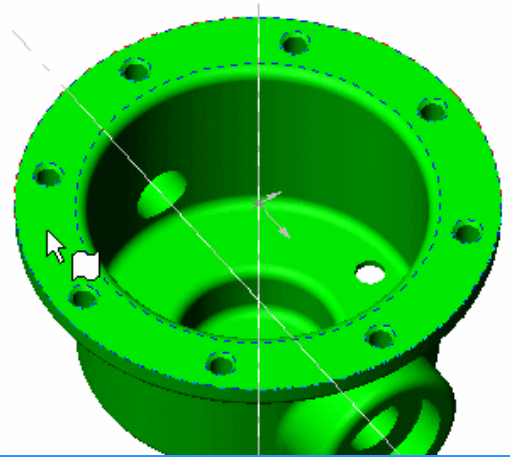
19. Set the End Condition to Blind.

20. Set the Depth to 0.1in and remove the check mark from the Reverse direction option.

21. Leave the Attribute set to Coarse.

22. Remove the check mark from the Use stock extents option so only the circular face is machined.

23. Click Finish to complete the addition of the new Face Feature.

24. Click Close to exit the wizard.

**2.5 Axis Feature Wizard : End Conditions**

End condition

Type : Blind

Depth : 0.1in    ☑ Reverse direction

Attributes

Attribute : Coarse    ☐ Use stock extents
                      ☐ Profile is offset
                      ☐ Through

2.5 Axis side wall

☐ Tapered

Autodetect    Taper angle :    ⦿ Taper in
                                ○ Taper out

☐ Edit feature profiles

Help    Cancel    << Back    Next >>    Finish
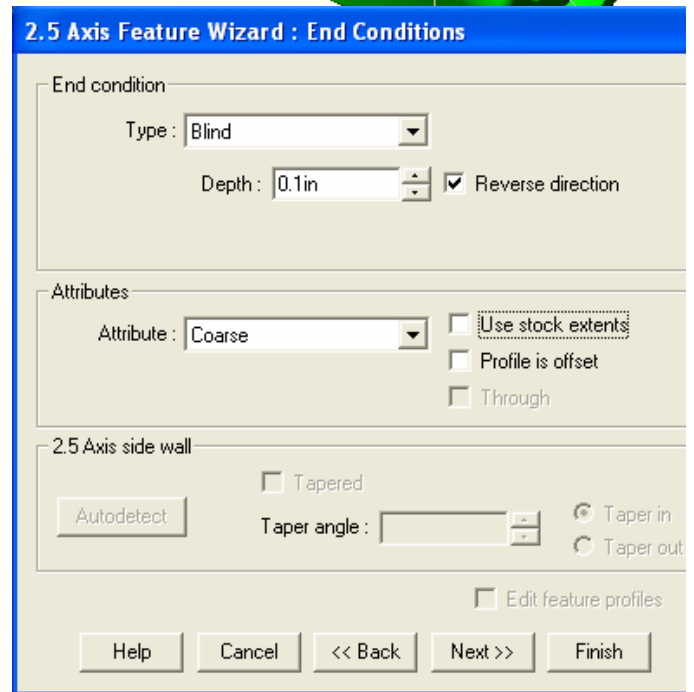
25. In the lower section of the Feature tree, drag and drop the Face Feature onto Setup4 so that it is the first feature machined on that Setup.
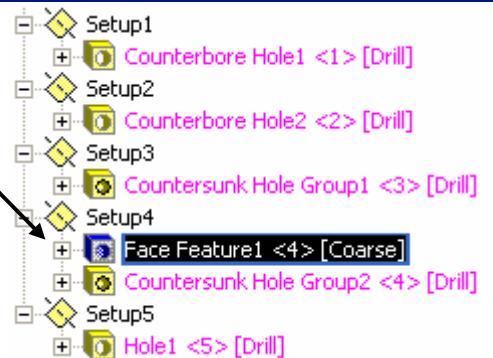
Generate operations and toolpaths, then run the simulation:

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

2. 🧩 Click the SolidWorks FeatureManager design tree tab.

3. Right click the Mill2AX-14 As-Cast item in the tree and select Show.

4. 🔧 Click the CAMWorks Operation Tree tab.

5. ◈ Right click Setup1 in the Operation tree and select Edit Definition.

Setup1
  Counterbore Hole1 <1> [Drill]
Setup2
  Counterbore Hole2 <2> [Drill]
Setup3
  Countersunk Hole Group1 <3> [Drill]
Setup4
  Face Feature1 <4> [Coarse]
  Countersunk Hole Group2 <4> [Drill]
Setup5
  Hole1 <5> [Drill]

6. On the Origin tab, change the Output origin to Part setup origin and click OK.

   This message displays: The Rapid or Clearance Plane value for an operation is higher than the value defined for the Setup. Change the Setup definition to the highest operation value?
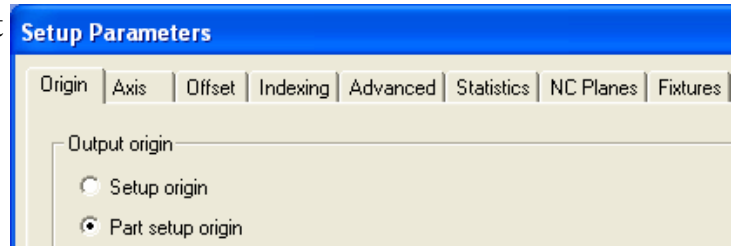
7. Click No.

8. Set the Origin to Part setup origin for the rest of the Setups in the same manner.

9. Right click the Center Drill operation under Setup5 in the tree and select Edit Definition on the shortcut menu.

10. Click the NC tab.

11. Set the Rapid plane and Clearance plane to Top of Feature, then click OK.

   By default, the Rapid and Clearance planes are relative to the stock. However, the Top of Stock setting applies only to Setups that are normal to the sides of the stock. Setup5 is not normal to the stock, so the Top of Stock setting will not produce correct Rapid and Clearance planes.
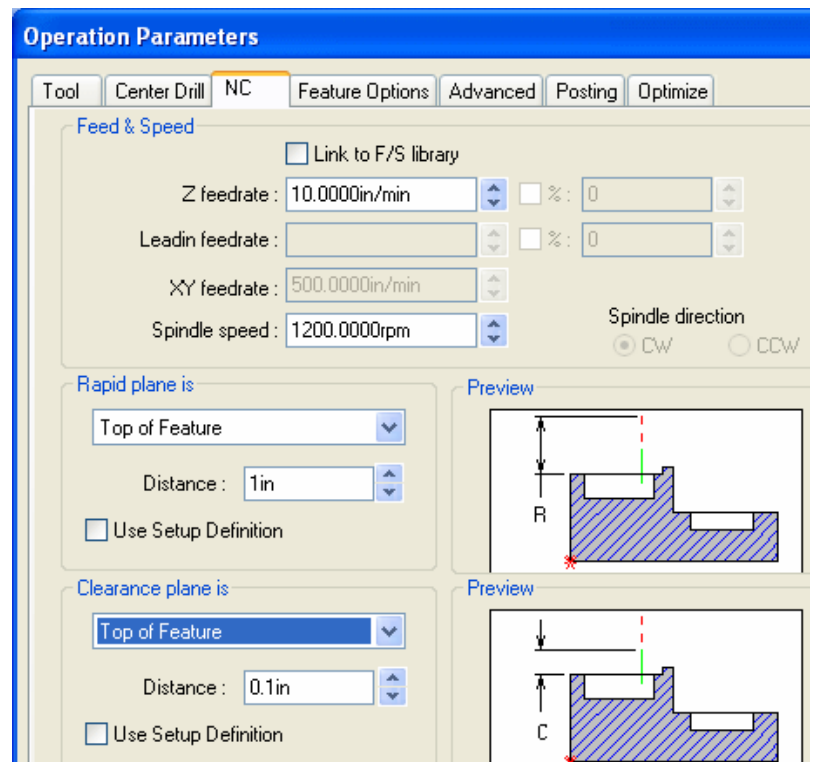
12. Right click the Drill operation under Setup5 in the tree and select Edit Definition.

13. On the NC tab, set the Rapid and Clearance planes to Top of Feature, then click OK.

14. Click Generate Toolpath on the CAMWorks toolbar.

15. Click Simulate Toolpath on the CAMWorks toolbar.

16. Make sure the Tool Mode button is highlighted.

17. Set the Target Part to No Display.

18. Click the Run button.

# Chapter 8  Learning the TechDB

The intelligence behind CAMWorks machining automation is TekSoft's proprietary Technology Database (TechDB™). The Technology Database is shipped with data that is considered generally applicable to most machining environments. In order to gain full advantage of CAMWorks, you will want to modify this data and input additional knowledge that represents your expertise and your facility's capabilities and machining methods. This knowledge can then be recalled and applied in CAMWorks to help automate and promote consistency in NC programming and machining.

Even without this individual tailoring, CAMWorks can be used as a productive tool. With the exception of tooling and machine information, any method or machining procedure that is defined in the Technology Database can be applied interactively and modified in CAMWorks.

The TechDB is responsible for the following:

- definition and selection of machine tools
- definition and selection of tool sets
- definition and selection of cutting tools and holders
- determination of machining methods
- definition of speeds and feeds conditions

# TechDB 1

## What You'll Learn

### *Adding and Editing Mill Machines*

One of the first steps when using CAMWorks is to define the machine that the part will be programmed for. The machine tool used for the manufacturing of a part will influence the methods used to complete the task. CAMWorks has automated this by allowing your facility to create a database of the existing milling centers.

When you set up a mill machine in the TechDB, you can:

- Define the CNC machine tools in your facility.

- Assign a name or machine number to each machine tool.

- Assign a Post Processor to each individual machine.

- Assign Tool Cribs to each individual machine.

- Define the machine as a light duty or heavy duty machine, which if used, will directly influence the speeds and feeds output by the MES Speed and Feed library.

- Set X & Y travel limits of the machine tool.

- Define the number of axes that the machine tool is capable of machining to.

The following exercise explains how to use the Machine Parameters form to add additional machines.

1. Open a mill part file and click the TechDB button on the CAMWorks toolbar.

   Microsoft Access starts and the TechDB Main Menu displays.

   The TechDB, which is based on the Microsoft Access database engine, contains a navigation tree, menus and forms that are used to modify and enter data.

2. For this exercise, you can use either Inches or Metric for the Units option below the navigation tree.

   The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing: metric or inch.

3. Expand the Mill item in the navigation tree, then click the Machine item to select it.
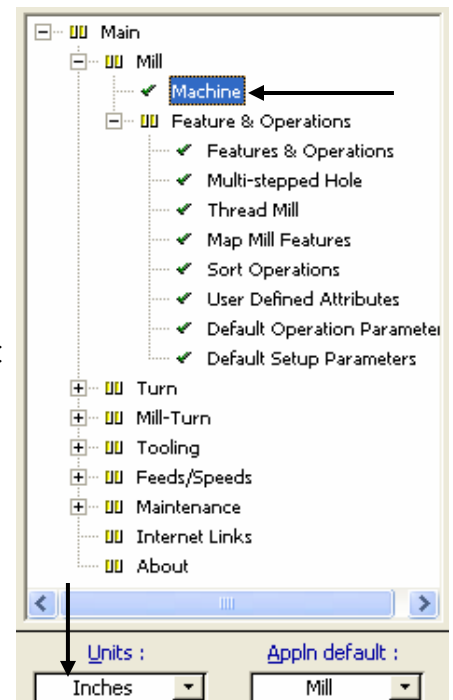
The Machine Parameters form displays the machine information in the first record. All data in the TechDB is considered a record. The Machine Parameters form is divided into three sections: General, Specifications and Tooling.



## Machine name, Machine ID, Machine description

These are user description fields. The Machine name is the name that appears in the list of machines in CAMWorks. Machine ID is used to reflect a number or name the user can assign to the machine. Machine description allows for a description that is shown in CAMWorks along with the name. These fields do not affect code output.

## Default machine

Only one machine can be the default. This machine is the one chosen automatically when you enter CAMWorks. Normally, you would make the default machine the one you program most. If you program on different machines, you can change the machine easily in CAMWorks.

## Controller

This parameter defines the CAMWorks post processor file to use for the selected machine.

## Machine duty

This option allows you to specify the kind of work this machine does. Changing the setting changes the speeds and feeds coming out of the database. A different speed and feed can be set for each material depending on this setting. This allows you to set speeds and feeds according to the particular machine's capabilities. For instance, a machine with a 20 hp spindle motor may be set as Heavy Duty, while another machine with a 10 hp spindle motor may be set as Medium or Light Duty.



The Specifications section allows you to set some operating parameters for the machine. Most are self-explanatory such as Horsepower, Max spindle speed and Max feed rate. These values are available during posting.

The **Indexing** can be set to None, 4, or 5. The default setting displays for the Indexing parameter on the Setup tab in the Machine dialog box. This option defines to CAMWorks whether single direction or rotary indexing is active and in Assembly mode controls the allowable machining directions. AFR and interactively inserted features can be defined for any machining direction. If set to None, you can define a single machining direction. If set to 4 or 5, the number of directions is unlimited.

**Table travel** sets the maximum table travel in X, Y, and Z.
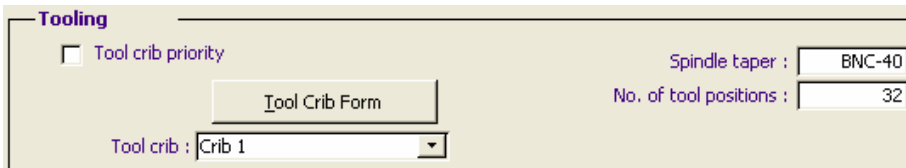
The Tooling section allows you to set the standard tools for this machine.

The **Tool crib** pulldown lets you select the standard set of tools this machine uses by default.

The **Tool crib priority** option tells CAMWorks to override some settings for choosing a tool if the tool exists in the tool crib. For example, a 20mm cobalt end mill chosen because of the material being machined would be overridden if another 20mm end mill exists in the tool crib. When this option is selected, CAMWorks changes the icon of an operation in the Operation tree to indicate a tool has been chosen that did not come from the Tool crib.

For more information on the Tool Crib and how to use it, see the TechDB 2 and TechDB 3 exercises.

4. Locate the Record buttons at the lower left of the form and click the Next button.

The Record buttons allow you to navigate through the list of machines. All forms in the CAMWorks TechDB have these buttons. You can move to the beginning and end of the list, as well as move through each record one at a time. The total number of records is displayed here. CAMWorks supplies sample machines. You can modify or delete these machine definitions as required.

5. Click the Go To Last button.

The form displays an empty record that you would use to add an additional machine.

6. Click the Close button.

When you finish entering information for machines, there are no additional commands needed to save the data. The information is saved when you click Close.

# TechDB 2

## What You'll Learn

### *Adding Tools to the Tools Library*

The Tools Library command on the Technology Database Main Menu provides access to various commands for adding, deleting, and editing mill tools, holders, tool assemblies, and tool cribs. Tool types supported by CAMWorks include drills, taps, end mills, user defined tools and many others. While the Technology Database is supplied with many different tool types and sizes, it will not likely include all various tools used within your facility. Ideally, to take the most advantage of CAMWorks, it is recommended that you add or modify the tool libraries to include the tools used in your machining practices.

Each tool is defined with both geometric and non-geometric characteristics. Some of these parameters are used directly by CAMWorks in the calculation of toolpaths and feeds/speeds and for display during toolpath simulation. Other parameters are provided for the purpose of user identification and tool management.

Optionally, holders can be defined and mated to tools for increased tool reference information and management as well as enhancing toolpath simulation and validation.

The following exercise shows you how to add a tool.

1. Open a mill part file and click the TechDB button on the CAMWorks toolbar.

   Microsoft Access starts and the TechDB Main Menu displays.

2. For this exercise, you can use either Inches or Metric for the Units option.

   The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing: metric or inch.

3. Expand the Tooling item in the navigation tree and click Drill under Point-to-Point Tools to select it.

   The Tools Database – Drill form displays. The tool library contains all of the standard drill sizes. The tool expressions for hole features are based on the standard drill sizes so that CAMWorks will automatically find the proper size tool for a hole feature. All of the drills entered in the database are cobalt.

The field name is listed across the top of the form above each column. Most of these fields are self-explanatory as they pertain to the size and configuration of the tool. There are 3 fields for user customization.

The **ON** field allows you to control whether CAMWorks can select the tool. When this option is checked, CAMWorks can select the tool when an operation plan is generated. When it is not checked, CAMWorks will not select the tool. For example, if a tool that is normally used is temporarily not available, the check mark can be removed so that the tool will not be selected when Generate Operation Plan is selected.

**Tool ID** can be used to identify tools. For example, if you have a tool crib identification system in use, this field could reflect it or you could use the stock # or order # for the specified tool.

The **Comment** field at the far right can be used to customize the comment shown for each tool when displayed in CAMWorks.

## Sorting



Below the list of tools is an area with search and show options. These options allow you to:
- Type a tool diameter and click the Find button to display only tools with a specified diameter.
- Click the Show All button to reset the list to show all tools.
- Select the Active or Inactive radio button to display only active tools (those tools with the ON option checked) or only inactive tools (those tools with the ON option not selected).

1. Type .5 in the Enter diameter text box and click the Find button.

   The list displays only tools with a .5in diameter and the total number of tools that were found displays at the bottom of the form.

2. Click the Show All button to display the entire list again.

You can also use the Sort command to sort records by field:

3. Click in a cell in the Eff. Cut length column.

4. Select Records on the menu bar at the top of the Access window and select Sort, then select Sort Ascending.
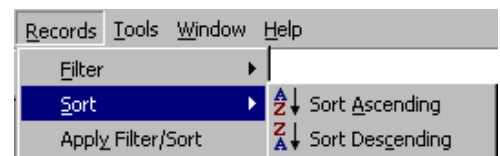
   

   The tools are sorted based on the cut length value.

5. Click the Show All button to return the list to the original order.

## Navigating Through the Tool List and Adding Tools

At the bottom of the form are buttons and options that can be used to navigate through the list of tools. The total number of records is also displayed. In the figure below, there are 367 drills in the database.

Use the Record buttons to navigate to the beginning and end of the list or move through the list one record at a time. All forms in the TechDB have these buttons.



1. Click the Go To End button.

   The end of the list displays with an empty record at the end to allow you to add new records. Every list in the TechDB has an empty record at the end.

   Note that you can also use the scroll bar on the right side of the form to move through the list.



2. If you were entering a tool, you would start typing values into the fields.

   **IMPORTANT!** As soon as you start typing in an empty record, a new record will display below the one you are editing. It can be confusing to determine which record you are working on since the record you are editing is no longer the last record.

3. Click the Close button.

   When you finish adding new tools, there are no additional commands needed to save the data. The information is saved when you click Close.

# TechDB 3

## What You'll Learn

### *Using Tool Cribs*

After you have entered all the tools into the Tool Library to meet your manufacturing needs, you need to build a Tool Crib for each milling center or build one Tool Crib for use with all of your machining centers. A tool crib is a subset of all the tools in the library and usually represents the tools that are commonly loaded into the machine's turret or are commonly used. A Tool Crib is then associated to a Machine definition so that when the machine is selected in CAMWorks, these common tools are made available during the programming of the part. There is no limit to the number of tools in a Tool Crib or the number of tool cribs you can create.

CAMWorks is supplied with several example tool cribs that are intended to be used with the CAMWorks example files. These tool cribs can be modified or deleted as required.

1. 🖳 Open a mill part file and click the TechDB button on the CAMWorks toolbar.

    Microsoft Access starts and the TechDB Main Menu displays.

2. For this exercise, you can use either Inches or Metric for the Units option.

    The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing.

3. Expand the Tooling item in the navigation tree.

4. Expand Mill Tooling and click Tool Crib to select it.

    The Tool Crib form displays.

    Using standard Access commands, you can perform the following functions in the Tool Crib form: create new Tool Cribs with an unlimited number of tools in each one; modify the tools in an existing tool crib; and delete a tool crib or selected tools in a tool crib.

    When assigning a Tool Crib to the machine that you are programming for, you are indicating that only these tools should be available for use.
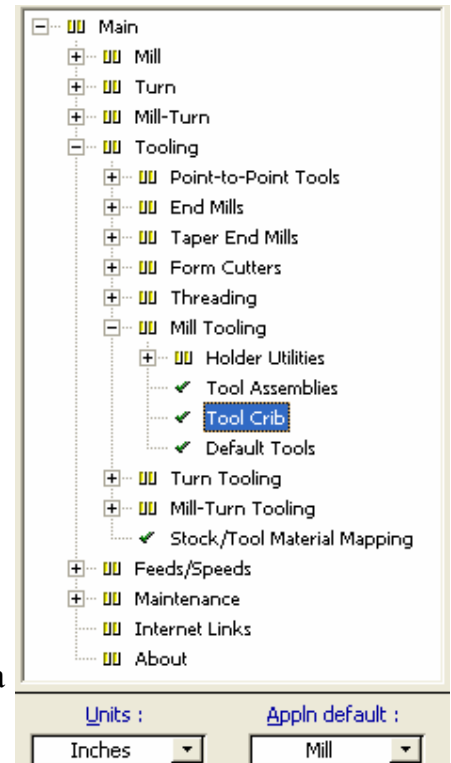
At the top of the Tool Crib form is the name of the crib and the number of stations the Tool crib has. You cannot have more tools in the list than the number in Total stations. For example, you may have a machine tool with a 30-tool position magazine and there are an additional 20 tools that are readily available. You could define a Tool Crib of 50 tools. When programming a part in CAMWorks, you can easily select the tools to use from this Tool Crib. There is no limit to the number of tools in a Tool Crib or the number of tool cribs you can create.

The Define using Tool Assemblies option determines if the tool crib will be defined individually from tools and holders (unchecked) or using tool and holder assemblies (checked). Crib 1 was defined individually from tools and holders.

There are three main fields on the Tool Crib form:

The **Stn no.** refers to the particular tool turret station that the tool will be loaded in, and will be the tool number output in the G-Code.

The **Tool type** reflects the type of tool, such as Flat End Mill, Ball Mill, Drill, etc.

The **Key parameters** field is filled automatically after the tool is selected.

5. Under Tool type, click the down-arrow button to the right of the tool type of the first tool.

   If you were adding a tool, you would choose the tool type from the list of available types.

6. Leave the current tool type selected.

7. To the right of the Tool ID column is an edit button to select the particular tool for this station. Click the button to display the selection form.

8. The Tool Database form for the type of tool selected under the Tool Type column appears.
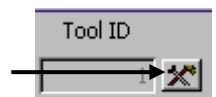
   If you were adding a tool to the crib or replacing the existing tool in this station, you would locate the tool in the list, select it, then click OK.

9. Click Cancel to keep the existing tool and close the Tools Database form.

10. Click the edit button to the right of the Holder ID column to open the Mill Holder Library form.

    Each tool in the Tool Crib has a tool holder associated with it. You can display the holder during toolpath simulation and check for collision between holder and stock.

11. Click Cancel to keep the existing holder, then click Close to close the Tool Crib form.

12. Expand the Mill item in the tree and click the Machine item to display the Machine Parameters form.



13. In the Tooling section at the bottom of the form is a pulldown list for selecting the Tool Crib to associate to the machine. When you define a machine, you would select the desired Tool Crib for the machine.

The Tool Crib Form button is another method for opening the Tool Crib form.

When the Tool Crib Priority option is checked, CAMWorks changes the icon of an operation in the Operation tree to indicate a tool has been chosen that did not come from the Tool crib.

# Tech DB 4

## What You'll Learn

### *Feature and Operation Basics*

The Feature & Operations area of the database is essentially the brain of the TechDB. The operations generated in CAMWorks and all the parameters they contain come from Feature & Operations. This example explains how to define:

- Defaults for the Setup Clearance and Retract planes.
- Operation defaults that display when you define new operations in the TechDB.
- Default sort sequences for operations when you use the Sort Operation command in CAMWorks.
- New attributes.

1. 🔲 Open a mill part file and click the TechDB button on the CAMWorks toolbar. Microsoft Access starts and the TechDB Main Menu displays.

2. You can use either Inches or Metric for the Units option at the bottom of the tree.

   The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing: metric or inch.

### Default Setup Retract

3. Expand the Mill item in the tree, then expand Feature & Operations.

4. Click the Default Setup Parameters item to select it.

   The Default Setup Parameters form allows you to set the default Rapid and Clearance settings and Upper and Lower Z Limits.

   The Rapid plane is the safe Z retract position that the tool enters from and returns to before and after each operation.

   The Clearance plane is a Z position that the tool retracts to between Z cutting depths within the operation.

5. Click the down arrow next to the Rapid plane is and Clearance Plane is parameters to view the choices, then click Close.
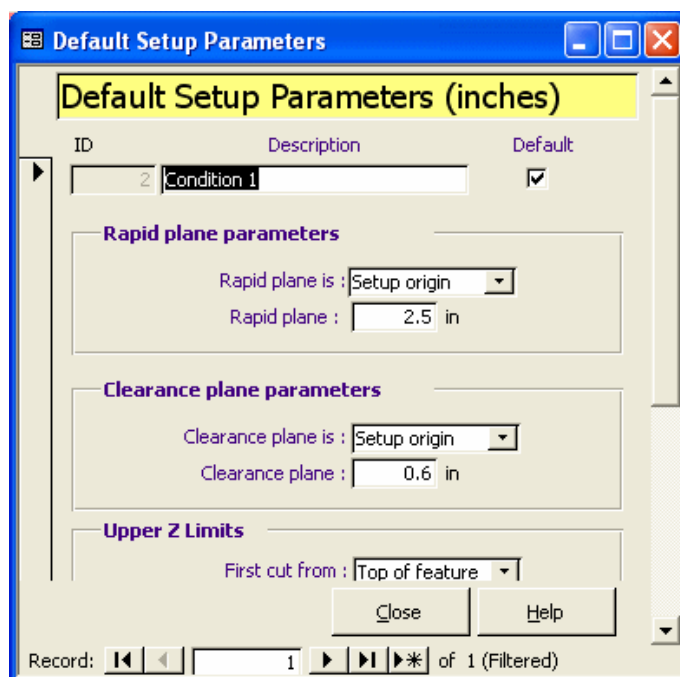
## Default Operation Parameters

6. Click the Default Operation Parameters item under Feature & Operations in the navigation tree.

The Default Operation Parameters form displays. This form controls the defaults that display when you create new operations in the TechDB database.

For example, if you were to add a new Rough Mill operation in the database, the fields on the form would display default values. This is where those parameters come from. Setting your preferences beforehand can save you a significant amount of time later.

Additionally, when using the Insert Operation and Insert Hole Operation command to insert operations manually in CAMWorks, if the option Use default parameters is active, the default parameters for the inserted operation are also derived from this form.

For increased flexibility, this form allows you to define different sets of defaults, where each set should contain only one of each operation type. The Default check box on the form defines which set is active.
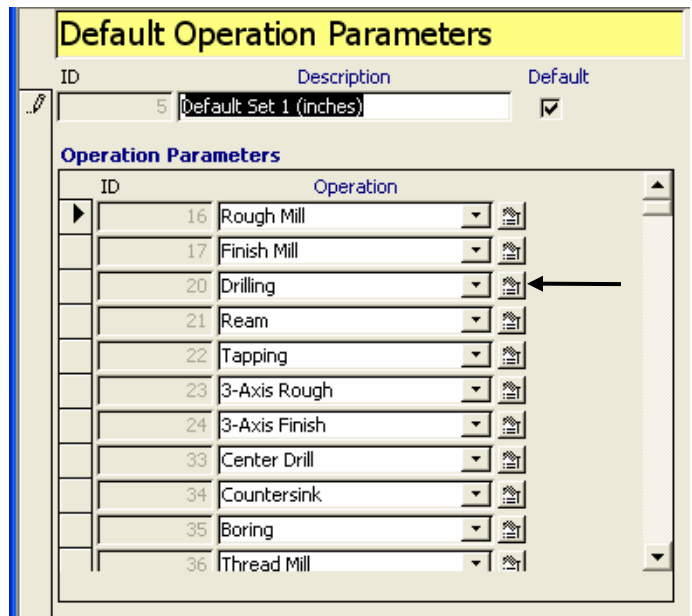
7. Click the edit button next to the Drilling operation.

The Drill Operation Parameters form displays.

The settings in this form are the defaults for new drill operations.
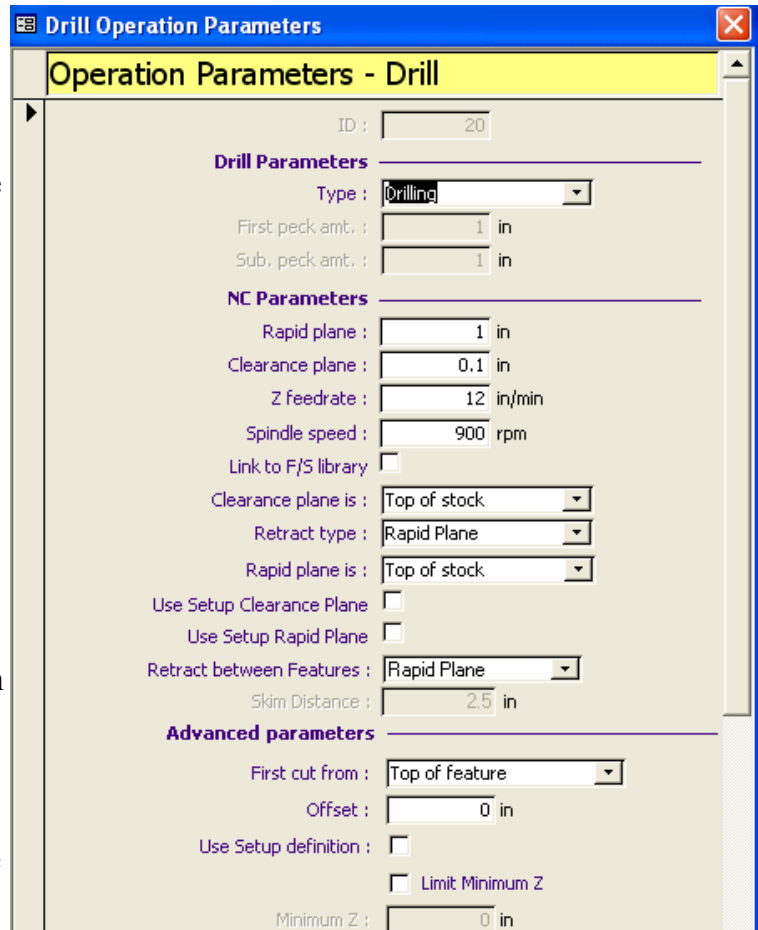
Remember, this is for new operations, not for existing ones.

8. Click Close to return to the Operation Default Parameters form

9. Click the edit button for the other operation types and view the default parameters.

10. When you are finished, click Close to exit the form.

## Sort Operations Default

11. Click the Sort Operations item under Feature & Operations in the TechDB navigation tree.

    The Sort Operations form displays. The operations can be sorted manually by dragging and dropping in the Operation tree or automatically using the Sort Operations command. This form controls the default order for how the operations would be sorted using the command.

    Note that hole operations are listed first followed by Rough Milling, and then Finish Milling.

    To change the order of the operations in the form, highlight the operation to be moved and use the Decrease or Increase buttons to change the position.
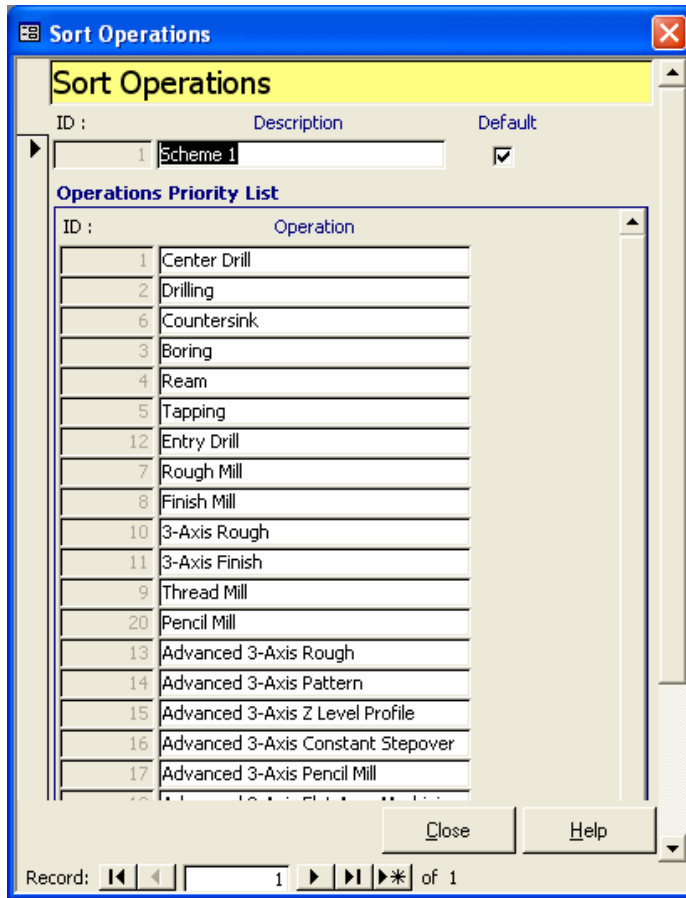
12. Click Close to exit the form.

## User-defined Attributes

13. Click the User Defined Attributes item under Feature & Operations in the TechDB navigation tree.

    The Feature & Attributes form displays. Every feature in CAMWorks, (pocket, hole, slot, multi surface feature, etc.) has attributes assigned to it. Attributes allow you to set up multiple methods for machining the same type of feature. The hole feature shown here is a good example. You may want to drill, ream, bore or thread a hole. The attribute allows you to choose the method for each feature in CAMWorks before generating the operation plan.
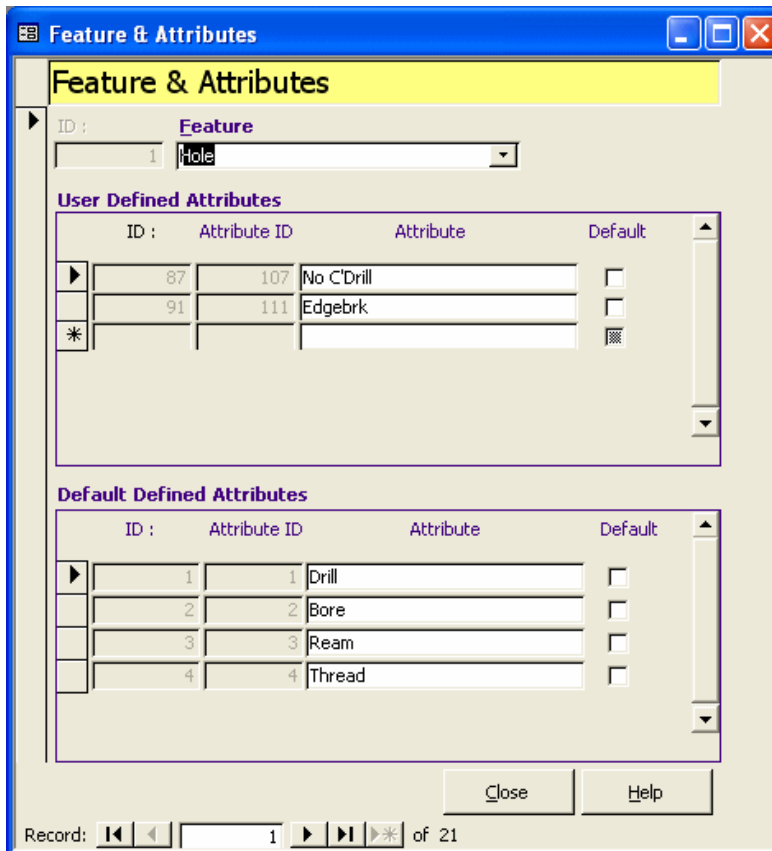
The Feature & Attributes form allows you to define the names of the attributes. That is all you do on this form. The parameters for the given attribute are set elsewhere.

The User Defined Attributes section at the top allows you to add and change names. After adding a new attribute in this form, you set up the machining method for the new attribute in a different form (Feature & Operations).

Default Defined Attribute names cannot be changed. For example, the Drill, Bore, Ream, and Thread attribute names for hole features cannot be changed; however, the parameters that those attributes call can be changed in the Feature & Operations form.

14. Click Close to close the form.

15. On the navigation tree, click the Features & Operations item under Feature & Operation.

The Feature & Operations form displays the conditions and operations defined for a hole feature.

16. Locate the Attribute column in the Feature Conditions section.

17. Click the down arrow to the right of the Drill attribute in the first record and notice that the names of the attributes in the list are the same as the names in the Feature & Attributes form. The next exercise explains how to set up the machining sequences associated to an attribute.
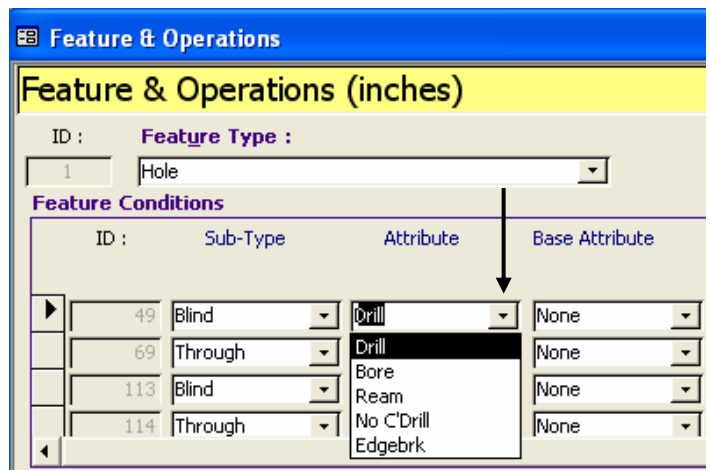
# TechDB 5

## What You'll Learn

- Creating an attribute and associating a machining sequence
- Applying the attribute in CAMWorks

Creating new attributes allows you to customize the TechDB so that CAMWorks will machine features using your exact methods. In the following exercise, you create a new attribute and associate a machining sequence in the TechDB, then apply the attribute to a feature and generate the operations in CAMWorks.

### *Creating an Attribute and Associating a Machining Sequence*

1. Open the part file **MILL2AX_11.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ▣ Click the TechDB button on the CAMWorks toolbar.

3. Make sure the Units selection box is set to **Inches**.

   The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing: metric or inch.

4. Click the User Defined Attributes item under Feature & Operations in the TechDB navigation tree.

   The Feature & Attributes form displays.

   There are five attributes listed in the User Defined Attributes section. An empty record is under the Drill Entry attribute.

5. Click the down arrow below the Feature and select Rectangular Pocket.

6. Click in the empty record text box and type Example.

7. Click Close to exit the form.

   The attribute is saved automatically when you close the form.
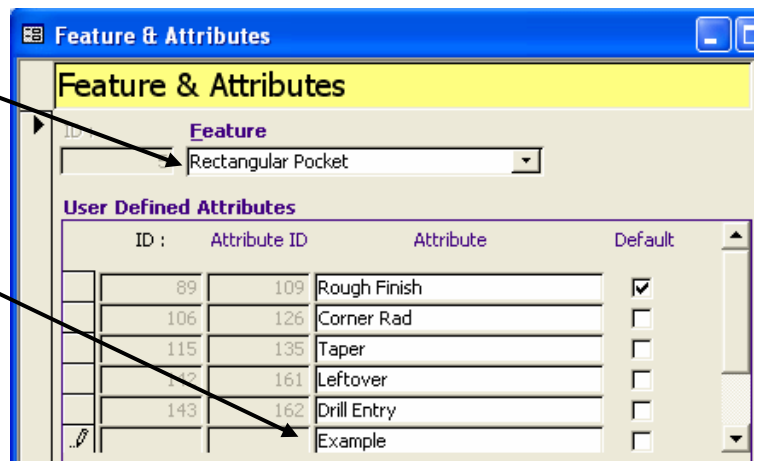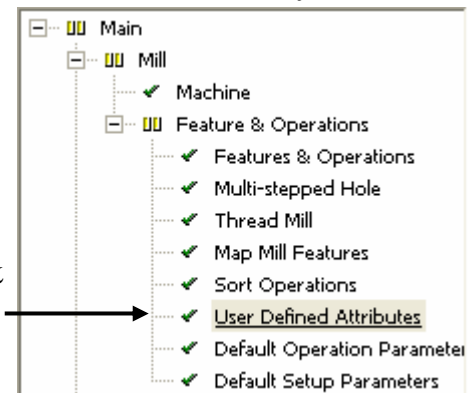
## Associate a Machining Sequence to the Attribute

1. In the TechDB navigation tree, click the Features & Operations item under Feature & Operations.

   The Feature & Operations form displays. This form is where all of the machining operations and their parameters are contained and where most TechDB editing is done.

   This form has three sections:

   − Feature Type is the type of feature, such as Hole, Rectangular Slot, Irregular Pocket, etc.

   − Feature Conditions sets a particular condition that the feature will fit into based on its size and attribute.

   − Operations for Matching Conditions is the list of machining operations and associated tools that CAMWorks will generate when a feature is found that matches the feature type and conditions.
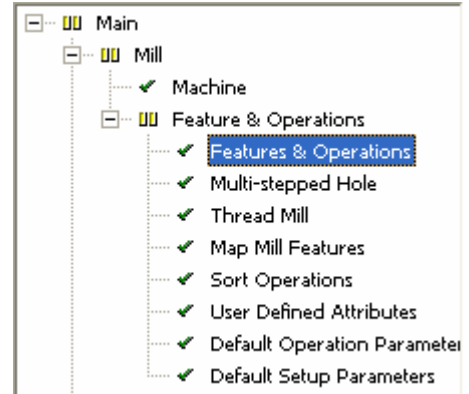
2. Under Feature Type, click the down arrow and select Rectangular Pocket.

3. In the Feature Conditions section, scroll the list to the end using the scroll bar on the right. The last record does not have an ID number. This is an empty record that you can use to add your new attribute.
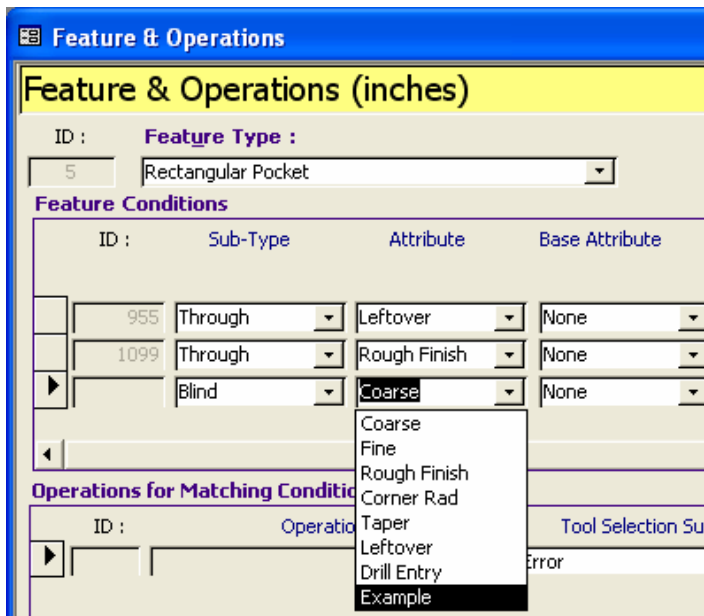
4. In the empty record, click the down arrow next to the Coarse attribute and select the Example attribute.

   This is the attribute you defined in the first part of this exercise.

5.  Leave the Base Attribute set to None.

    The Base Attribute is a feature condition that is used to distinguish features with a fillet between the bottom and the sides of the feature.

    None indicates that there is no fillet. Bottom Radius indicates that there is a fillet. Features with bottom radius fillets can be defined interactively using the Insert 2.5 Axis Feature command.

6.  Under the > Width column, enter a value of 1.

7.  Under <= Width, enter a value of 10.

    Reading across the record, you have defined a blind rectangular pocket, using the Example attribute, no base attribute, with a feature size ranging from greater than 1" to less than or equal to 10".

    The last step is to define the operations you want generated when this pocket is found. Notice that under Operations for Matching Conditions, the Operation field is grayed out. Operations cannot be selected until this field is activated.
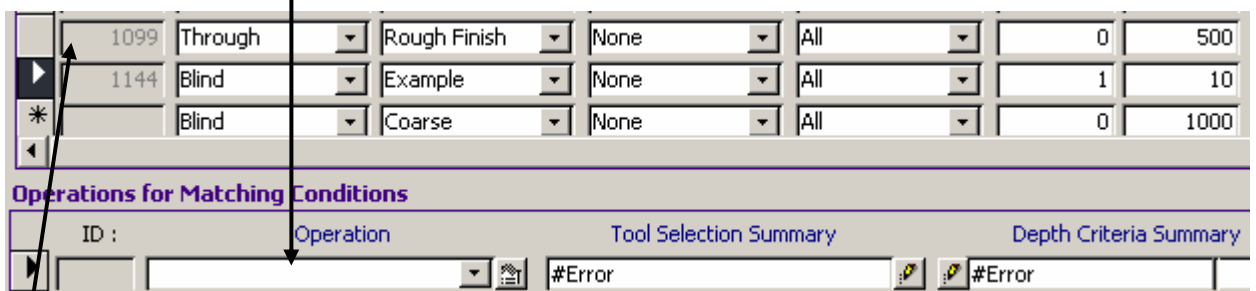


8.  In the list of Feature Conditions, select the record above the new record for the Example attribute, and then select the Example record again.

    The Operation field is now active.

    The Operations for Matching Conditions area has three fields:

    –   Operation selects the type of Operation, such as Rough Mill, Drilling, Reaming, etc.
    –   Tool Selection Summary allows you to pick the tool to be used.
    –   Depth Criteria Summary sets the depth of cut.

    In this exercise, you define operations for roughing and finishing a pocket. You set up two roughing operations, the first with a large tool and a second with a smaller tool cutting only where the large tool could not. There are two finishing operations, the first for cutting the bottom of the pocket and the second for cutting the side walls.

9.  Click the down-arrow button under Operation and select Rough Mill.

    A new empty operation displays below the first one.

10. Select Rough Mill for the second operation.
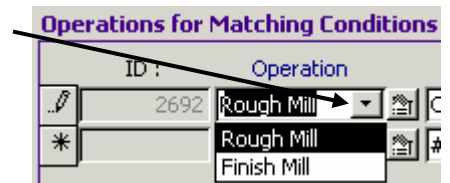
11. Select Finish Mill for the third and fourth operations.

    When complete, four operations are listed as shown here.

    To the right of each operation is an edit button to access all the machining parameters.

| ID : | Operation |
|------|-----------|
| 2690 | Rough Mill |
| 2691 | Rough Mill |
| 2692 | Finish Mill |
| 2693 | Finish Mill |

12. Click the edit button next to the first Rough Mill operation.

13. In the Rough Mill Operation Parameters, make sure the Generate leftover option is checked.

14. Scroll down to the Depth Parameters and set the Bottom allowance to .05.

15. Under the Depth Parameters, set the First cut amt. and Sub cut amt. to 1.

**Depth Parameters**

| | |
|---|---|
| Bottom allowance : | 0.05 in |
| Island allowance : | 0 in |
| First cut amount : | 1 in |
| Max./Sub cut amt. : | 1 in |
| Final cut amount : | 0 in |
| Minimum cut amount : | 0 in |
| Method : | Equal |
| Depth processing : | To depth by region |

16. Click Close.

17. Click the edit button for the second Rough Mill operation.

18. In the Operation Parameters – Rough Mill form:

    Set the Pattern to Pocket In.
    Set the Machine leftover option to Previous Leftover.
    Under Depth Parameters, set the Bottom Allowance to 0.05.
    Under Depth Parameters, set the First cut amount and Sub cut amt. to 0.5.

19. Click Close.

20. Click the edit button for the first Finish operation.

21. In the Operation Parameters – Finish Mill form, set the Side Parameters Allowance to 0.05.

22. Under the Depth Parameters, set the First cut amt. to 2.

23. Scroll to the Flat Areas section at the bottom of the form and select the Bottom finish option.

24. Set the Stepover % to 50.

25. Click Close.

26. Click the edit button for the second Finish operation.

**Flat Areas**

| | |
|---|---|
| Bottom finish | ☑ |
| Perimeter pass : | None |
| Pattern : | Pocket Out |
| Stepover : | 0 |
| Stepover % : | 50 |

27. In the Operation Parameters – Finish Mill form under Depth Parameters, set the Ramp to Angle and the Ramp angle to 3.

| | |
|---|---|
| Ramp : | Angle |
| Ramp angle : | 3 degrees |
| Ramp cleanup pass | ☑ |

28. Check the Ramp cleanup pass option, then click Close.

## Select Tools for Each Operation

The next step is to select the tools for each operation.

1. For the first Rough operation, click the edit button under Tool Selection Summary.

   The Tool Selection form displays.

2. Leave the Tool type set to Flat End Mill.

3. In the Tool Selection section, click the Use constant radio button.

   With this method, you are specifying only a constant diameter, not a particular tool. CAMWorks uses other criteria, such as the material being cut, to choose the tool.

   There are two other methods for choosing a tool:

   Use expression allows you to give a tool diameter range. If you use a range, when an Operation Plan is generated, CAMWorks searches the Tools database for a tool diameter that is within the specified upper and lower diameter. If a tool cannot be found, the CAMWorks Message Window will inform you that a tool of the required size could not be found.

   With the Select tool option, you can click the Select Tool button and choose a particular tool from the tools database.

4. Set the Constant value to 1. This sets the basic tool diameter to 1in. The exact tool chosen will be determined by other factors, such as the material being machined.

5. Click Close.

6. Edit the Tool Selection Summary for the second Rough Mill operation.

7. In the Tool Selection form, click the Use constant radio button and set the Constant value to .25.

8. Click Close.

9. Edit the Tool Selection Summary for the first Finish Operation.

10. Click the Select Tool radio button, then click the Select Tool button.
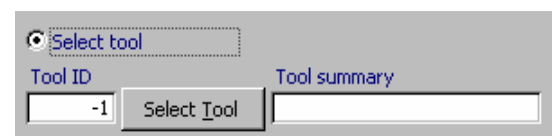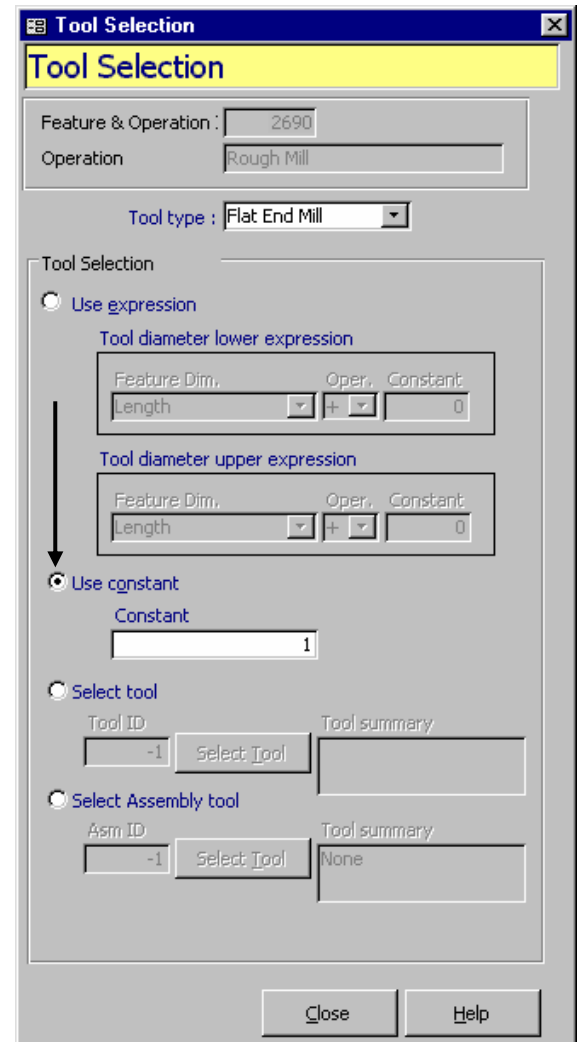
11. In the Tools Database - Mill form, scroll down and select ID number 683, which is a ¼", 4 flute carbide end mill.

| ID | ON | Tool ID | Sub-type | Diameter (D1) | End radius (R) | Shank dia. (D2) | Eff. cut length |
|---|---|---|---|---|---|---|---|
| 681 | ☑ | 1/4 Em4H | Rough & Fi ▾ | 0.25 | 0 | 0.25 | 0.625 |
| 682 | ☑ | 1/4 Em2C | Rough & Fi ▾ | 0.25 | 0 | 0.25 | 0.625 |
| 683 | ☑ | 1/4 Em4C | Rough & Fi ▾ | 0.25 | 0 | 0.25 | 0.625 |
| 684 | ☑ | 5/16 Em2H | Rough & Fi ▾ | 0.3125 | 0 | 0.3125 | 0.75 |

12. Click OK.

   The Tool ID and summary display in the Tool Selection form.

13. Close the Tool Selection form.

14. 🖉 Edit the Tool Selection Summary for the second Finish Mill operation.

15. Click the Select Tool radio button.

16. Click the Select Tool button.

17. In the Tools Database Mill form, select ID number 683 again and click OK.

18. Close the Tool Selection form.

19. 🖉 Click the edit button under Depth Criteria Summary for the first Rough Mill operation.

20. In the Depth Evaluation form, select Use Expression.

   Note that the default expression is Feature Depth + zero (0). This tells CAMWorks to machine to the bottom of the feature.

21. Close the form.

22. Repeat these steps for the remaining three operations.

   When you are finished, the Operations for Matching Conditions section of the Feature & Operations form should look like the figure below.

| ID : | Operation | Tool Selection Summary | Depth Criteria Summary |
|---|---|---|---|
| 2690 | Rough Mill ▾ | Const: 1 | (Feat Depth + 0) |
| 2691 | Rough Mill ▾ | Const: 0.25 | (Feat Depth + 0) |
| 2692 | Finish Mill ▾ | Tool Selected | (Feat Depth + 0) |
| 2693 | Finish Mill ▾ | Tool Selected | (Feat Depth + 0) |

23. Click Close to exit the Feature & Operations form.

24. Click Exit to close the TechDB and return to CAMWorks.

## *Applying the Attribute in CAMWorks*

1. **[CW icon]** Click the CAMWorks Feature Tree tab.

2. **[icon]** Click the Extract Machinable Features button on the CAMWorks toolbar.

   Automatic Feature Recognition finds a rectangular pocket group feature.
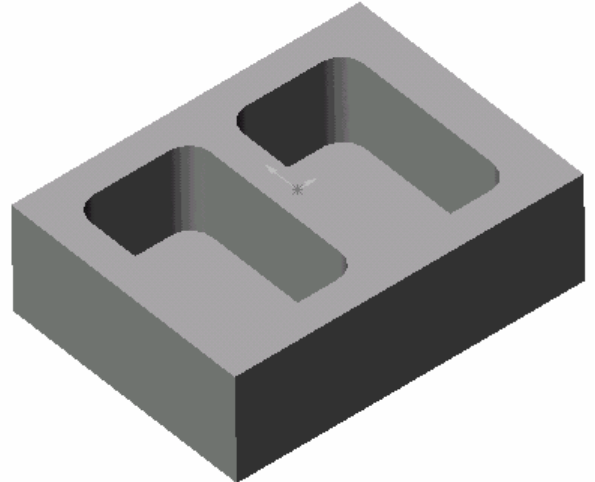
3. Right click Rectangular Pocket Group1 in the tree and select Parameters on the shortcut menu.

4. In the Rectangular Pocket Parameters dialog box, click the down arrow to the right of the Attribute selection box and select Example.

5. Click OK.

**Rectangular Pocket Parameters - Pattern of 2 Features**

Length (L) : 4.5in
Width (W) : 2.75in
Depth (D) : 2in
Chamfer (C) : 0in
Bottom radius (R) : 0in
Finish radius (R1) : 0.5in
Taper angle : 0deg
☑ Tapered inward
Islands : 0
☐ Through

Attribute : Rough Finish

Coarse
Corner Rad
Drill Entry
Example
Fine
Leftover
Rough Finish
Taper

Start position (S)
X : -1in          Z : 0in

Help
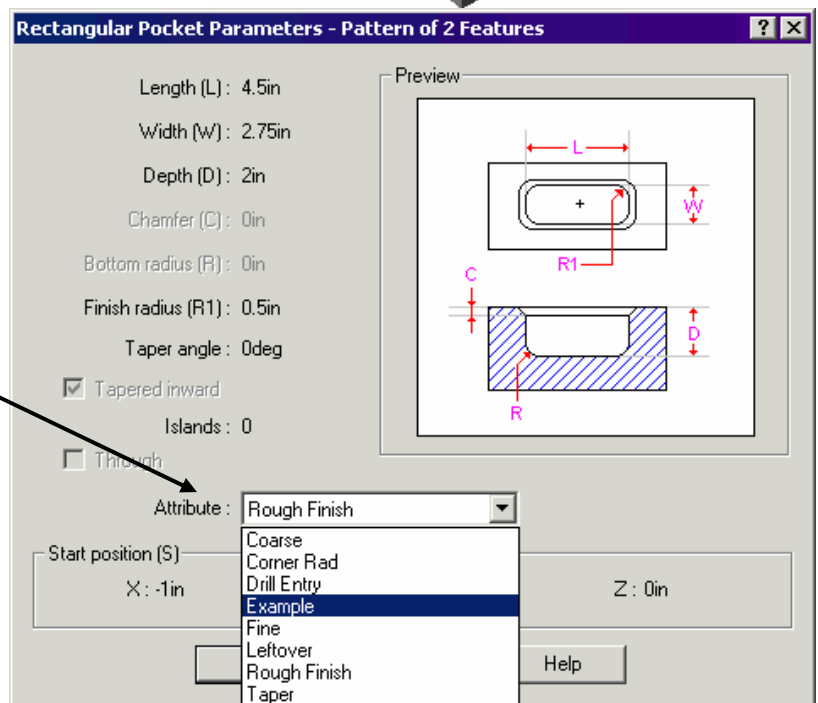
6. **[icon]** Click the Generate Operation Plan button on the CAMWorks toolbar.

   The Operation tree lists the four operations. This is the machining sequence that you defined in the TechDB for the Example attribute.

7. For each operation, right click on the operation and select Edit Definition on the shortcut menu. In the Operation Parameters dialog box, notice that the parameters are set to the values you specified in the TechDB, then click OK.

NC Manager
  Stock Manager[1005]
  Mill machine - Inch
  Mill Part Setup1 [Group1]
    Rough Mill1[T24 - 1 Flat End]
    Rough Mill2[T9 - 0.25 Flat End]
    Finish Mill1[T43 - 0.25 Flat End]
    Finish Mill2[T43 - 0.25 Flat End]

8.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Mill Part Setup in the tree and select Generate Toolpath.

9. Right click on the Mill Part Setup in the tree and select Simulate Toolpath.

   The Simulate Toolpath toolbar displays.

10.  Click the Run button to start the simulation.

11. Close the Simulate Toolpath toolbar when you are finished.

# TechDB 6

## What You'll Learn
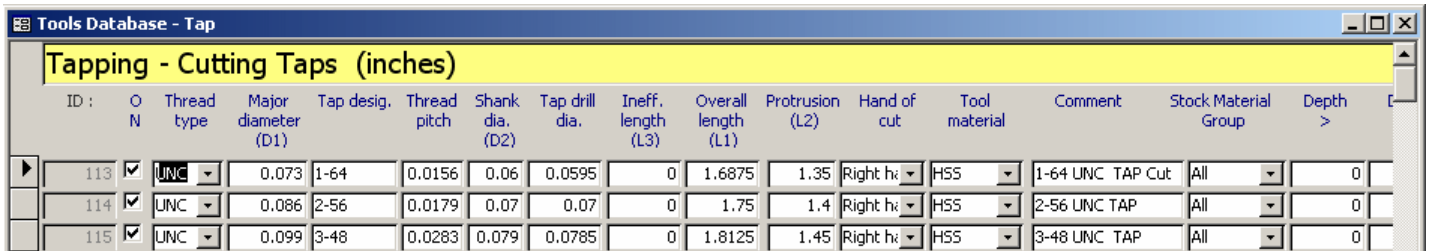
### *Understanding Threading Parameters*

The thread portion of the TechDB is a departure from the method the database normally uses. In all other cases, the definition of the tool, the feature, and the operations to machine the feature are in different places in the TechDB. Threads are different in that all the parameters are on the same form.

### Defining Taps and Associated Operations

1. ⊞ Click the TechDB button on the CAMWorks toolbar.

   Microsoft Access starts and the TechDB Main Menu displays.

2. For this exercise, you can use either Inches or Metric for the Units option.

   The information in the database is stored in two sections: metric and inch. If your SolidWorks part is metric, CAMWorks will access the metric information in the database. If the part is inch, CAMWorks will access the inch information. When you enter information in the TechDB, you need to specify which section of the database you are editing.

3. In the TechDB navigation tree, expand Tooling, expand Threading, then click the Tap – Cutting item.

   The Tools Database – Tap form displays. The form is divided into two sections.



The top section lists the taps and associated parameters. This section is similar to the forms for other tool types. There are fields for the physical size aspects of the tap, as well as fields for material, hand of cut, and comments.

The Operations for Tapped Holes section is a tabbed list of the Mill and Turn Tap operations that will be generated for a threaded hole using the selected tap in the top section of the form. Next to each operation is an edit button to customize the operation parameters.

4. Click the edit button to the right of the Center Drill operation.

**Operations for Tapped Hole**

Mill Tap Operations | Turn Tap Operations

| | ID : | Operation | | | Summary of Tool Selection | | Summary of Depth Criteria | |
|---|---|---|---|---|---|---|---|---|
| ▶ | 828 | Center Drill | ▾ | 🔲 | Tool selected | 🖉 | Const: 0.1 | 🖉 |
| | 829 | Drilling | ▾ | 🔲 | Const: 0.0595 | 🖉 | (Depth - 0) | 🖉 |
| | 1208 | Countersink | ▾ | 🔲 | Tool selected | 🖉 | C'sink Dia: 0.1 | 🖉 |
| | 1408 | Thread Mill | ▾ | 🔲 | Const: 0.073 | 🖉 | (Depth - 0) | 🖉 |
| | 1770 | Thread Mill | ▾ | 🔲 | Const: 0 | 🖉 | (Depth - 0) | 🖉 |
| ✳ | | | ▾ | 🔲 | Const: 0 | 🖉 | Const: 0 | 🖉 |
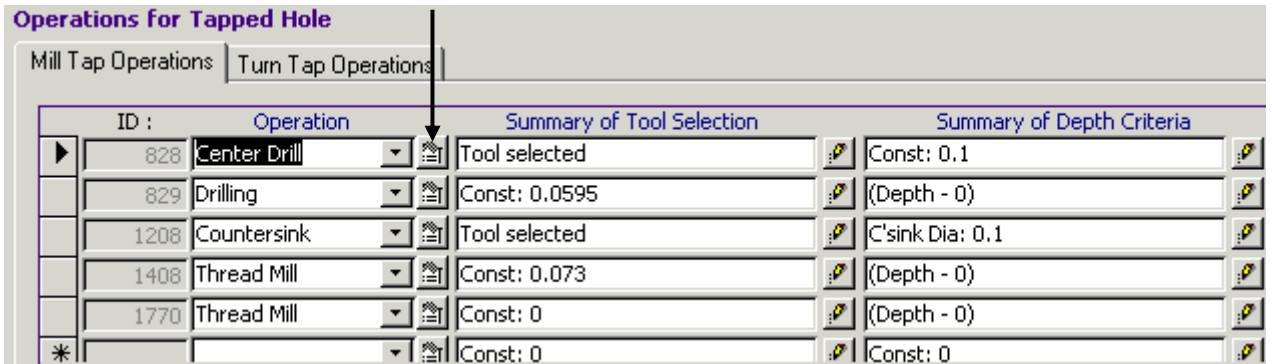
The Operation Parameters - Center Drill form displays.

Any changes to the parameters in this form apply to operations generated for the particular tapped hole size and do not affect operations for other hole sizes.

5. Click Close to exit the Operation Parameters form.

**Center Drill Operation Parameters**

**Operation Parameters - Center Drill**

ID : 2112

**Center Drill Parameters**
Type : Drilling
First peck amt. : 0.15 in
Sub. peck amt. : 0.15 in

**NC Parameters**
Rapid plane : 1 in
Clearance plane : 0.1 in
Z feedrate : 10 in/min
Spindle speed : 2000 rpm
Link to F/S library : ☐
Clearance plane is : Top of stock
Spindle direction : CW
Retract type : Rapid Plane
Rapid plane is : Top of stock
Use Setup Clearance Plane : ☐
Use Setup Rapid Plane : ☐
Retract between Features : Rapid Plane
Skim Distance : 0 in

**Advanced parameters**
First cut from : Top of feature
Offset : 0 in
Use Setup definition : ☐
☐ Limit Minimum Z
Minimum Z : 0 in

6. For the same Center Drill operation, click the edit button to the right of the tool under Summary of Tool Selection.

| ID : | Operation | | | Summary of Tool Selection | |
|---|---|---|---|---|---|
| 828 | Center Drill | ▾ | 🔲 | Tool selected | 🖉 |

The Tool Selection form displays.

The Tool Selection form allows you to choose the particular tool to be used for this Center Drill operation.

The tool can be chosen using two methods: Use constant or Select Tool.

With Use Constant, you are not specifying a particular tool, only a constant diameter. CAMWorks uses other criteria, such as the material being cut, to choose the tool.

With the Select Tool option, you can click the Select Tool button and choose a particular tool from the database.

7. Click Close to exit the Tool Selection form.

8. For the same Center Drill operation, click the edit button to the right of Summary of Depth Criteria.

Because this is a drilling operation, there are three depth setting sections in the form. The first is for blind/flat bottom holes, the second for through holes, and the third for blind/drill point holes.

In each of the depth sections, there are three methods for determining the depth the tool reaches.

− Use constant specifies a constant Z depth for the tool to drill to.

− Use expression allows you to use a mathematical expression to determine the depth. For instance, you can specify a depth that is more or less than the depth of the feature.

− Use countersink diameter allows you to specify the diameter of the countersunk hole as opposed to a depth setting. In this case, CAMWorks would figure the depth necessary to obtain the desired diameter.

9. Click Close to exit the Machining Depth form.

10. Once all the operations and their respective parameters have been set, the tapped hole settings are complete and you can click Close to exit the Tools Database – Tap form.

# TechDB 7

## What You'll Learn

### *Defining and Using Custom Tools*

CAMWorks allows you to create and display your own custom tools and tool holders. It is important to be able to use the actual tool shape during simulation, so that you can see an accurate representation of the tool cutting the part.

### Create Tools in SolidWorks

CAMWorks provides a method for geometrically defining a special shaped mill tool that can be used in 2 axis rough and finish milling and single point operations (drill, ream and bore). A special shaped tool is any tool shape not defined by the current CAMWorks tool types. For certain types of manufacturing, it is common to use special shaped tools for numerous milling operations including roughing, finishing, drilling, etc. These special shaped tools are routinely used in roughing and finishing wood-based products as well as for complex holes for ports.

Creation of the shapes is done in SolidWorks by drawing a cross section of the shape and a center line, and then creating a revolved solid. The sketch in the figure below shows the geometry used to create the tool used in this exercise.

The geometry of the tool is defined from a SolidWorks sketch and must conform to the following:

- The sketch must be made from the SolidWorks default plane, Plane1.

- The geometry can contain only lines and arcs. Splines are not supported.

- The center tip of the tool must start at the SolidWorks origin point (0,0).

- All geometry must be in positive XY space relative to the origin (i.e., the geometry must be to the right and above the SolidWorks origin).

- The axis of the tool must be vertical and pass through SolidWorks 0,0.

- The geometry must be closed and observe the rules required to make a SolidWorks solid of revolution.

- The Y value of the current end point must be greater than or equal to the previous end point.

In the following exercise, you use the CW-Tooling utility to save a SolidWorks sketch as a tool, add the tool to the Tool Library in the TechDB, then use the tool to cut a part.

## Use CW-Tooling Utility to Save Sketch as a Tool

1.  Open the part file **MillTool_SideGroove.sldprt** in *\Tooling\MTools* inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Tooling\MTools*).

    This tool was created by drawing a sketch that conforms to the rules on the previous page and then creating a revolved solid.

    Before using this tool in CAMWorks, you need to use the CW-Tooling utility to save the sketch as a form tool. CAMWorks uses the file generated by CW-Tooling, not a standard SolidWorks part document. However, if you save the model as a SolidWorks document, you can modify the model easily, if necessary.

2.  Select CW-Tooling on the SolidWorks menu bar and select Create Form Tool/Holder.
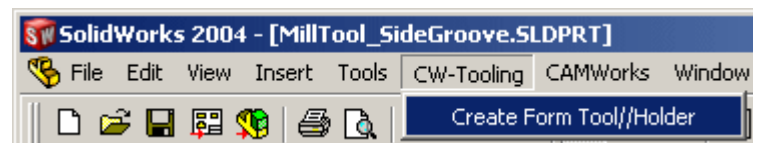
    If CW-Tooling is not displayed on the menu bar, select Add-Ins on the SolidWorks Tools menu, check the CW Tool and Holder Utility, then click OK.

3.  In the CAMWorks Form Tools and Holders dialog box, select Mill form tool and click the Save button.

4.  In the Save As dialog box, open the *\Tooling\MTools* folder inside the CAMWorks folder.

    This folder is provided as a default location for custom tools; however, you can organize tool and holder files in a folder structure that best suits your requirements. For example, you can save a tool file in the same folder as the associated part file or you can set up shared folders for tools and holders.

5.  Use **MillTool_SideGroove** for the file name and click Save.

    The file can have the same name as the SolidWorks document or a different name. Mill form files are saved automatically with a .mt extension.

    The program validates the geometry, saves the file and displays a 2D wire frame picture of the tool or holder in the Preview area.

6.  Click Close.
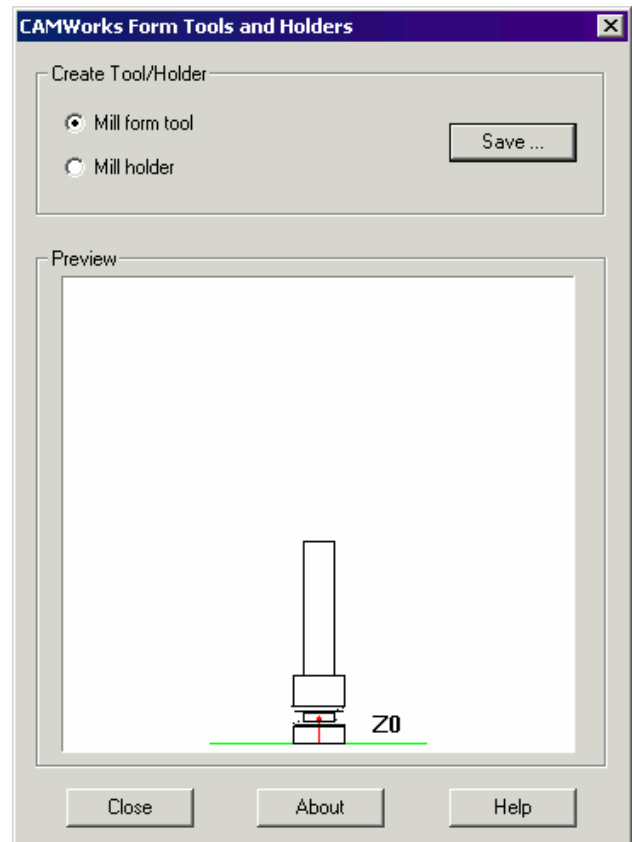
## Add the Tool to the TechDB Tools Library

1.  Click the TechDB button on the CAMWorks toolbar.
2.  Make sure the Units selection box is set to **Inches**.
3.  Expand the Tooling item in the tree.
4.  Expand Form Cutters, then click User Defined Tool.
    The Tools Database – Mill User Defined Tool form displays.

5.  Define the tool using the following information:
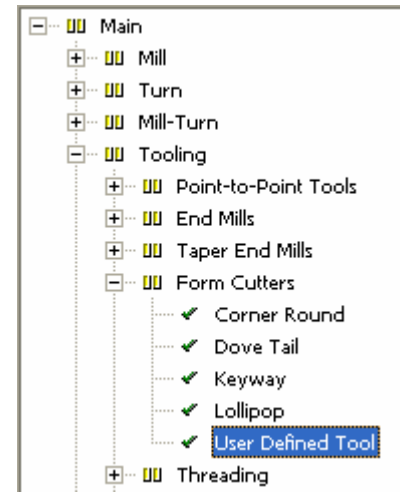
| Field | Value | Comments |
| --- | --- | --- |
| ON | Checked | When the option is checked, CAMWorks can select the tool when an operation plan is generated. |
| Tool ID | User1 | This is a user-defined tool identification field. |
| Designation | Side groove | The text entered in this field displays in the tool description section of the operation name in the Operation tree. |
| Cut diameter | 1 | When generating roughing and finishing toolpaths, CAMWorks bases the toolpath offset calculation on this parameter. |
| Shank diameter | 0.625 | Defines the diameter of the tool shank. In this release, this field is used for reference only. |
| Cut length | 1.25 | Length of the tool that can be used for cutting. |
| Overall length | 3.7 | Overall length of the tool. |
| Protrusion | 3 | Defines the distance from the tip of the tool to the bottom of the holder. This value is used to determine how much of the tool protrudes from the mill holder during simulation. |
| Tip offset | 0 | Defines an offset for the Z program point. |
| Hand of cut | Right | Determines if your spindle will rotate CCW or CW. In the case of a right hand tool, the spindle will rotate CW. |
| No. of flutes | 4 | Number of flutes on the tool. Feed and Speed calculations are partially based on the number of flutes. |
| Tool Material | HSS | The material is used for feed/speed calculations. |
| Comment | 1" Mill Side Groove | The comment is output in the posted G-code. |
| Tool name & path | Click to locate file | Defines the folder and name of the user-defined tool file. Use the Browse button to locate the \Tooling\MTools folder and select the MillTool_SideGroove.mt file. |

6.  When you are finished, click Close.
    The information is saved automatically.
7.  Click Exit to close the TechDB.
8.  Close **MillTool_SideGroove.sldprt.** You are finished with this file.

## Use the Tool in CAMWorks

You are now ready to use the tool to cut a part.

1.  Open part file **MILL2AX_12.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2.  [icon] Click the CAMWorks Feature Tree tab.

3.  You can use the default machine and stock.

4.  [icon] Click the Extract Machinable Features button on the CAMWorks toolbar.

5.  Click Mill Part Setup1 in the tree.

    The features you are going to machine in this exercise cannot be machined from this direction, so you can delete this setup and insert a new setup.

6.  Right click Mill Part Setup1 and select Delete.

7.  Click Yes to confirm the deletion.

8.  Right click Stock Manager and select Insert Mill Part Setup on shortcut menu.

    The Mill Part Setup dialog box displays.

9.  Pick the top face of the part.

10. Make sure the Face option is not checked and click OK.

    Mill Part Setup1 is listed in the tree.

11. Right click Mill Part Setup1 and select Insert 2.5 Axis Feature on the shortcut menu.

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

12. Select Open Profile for the Feature Type.

13. Pick the lower inside edge of the groove as shown in the figure.

14. Rotate the part and pick the corresponding edge on the other side.

15. Click the Next button.

16. Rotate the part back to the original view.

17. For the End Condition, pick the vertex on the top of the part as shown in the figure.

    The Type changes to Upto Vertex and the Depth of the feature is 1".

18. Make sure the Edit feature profiles option is checked and click Next.
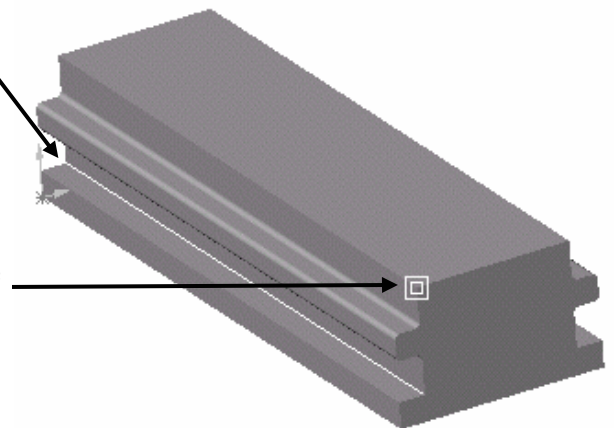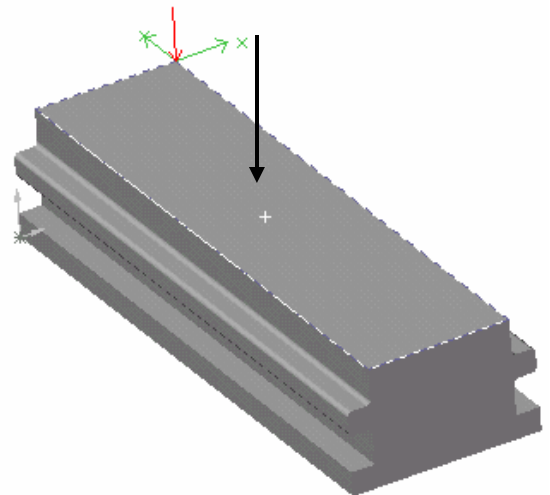
The 2.5 Axis Feature Wizard: Edit Feature Profiles dialog box displays. Notice that the direction of cut that displays on the part for the Open Profile(1) feature is correct.

19. Click the down arrow in the Feature selection box and select Open Profile(2).

The direction of cut that displays on the part for the Open Profile(2) feature is not correct.

20. Select the Flip direction of cut option.

The arrow should change to the opposite direction as shown in the figure.
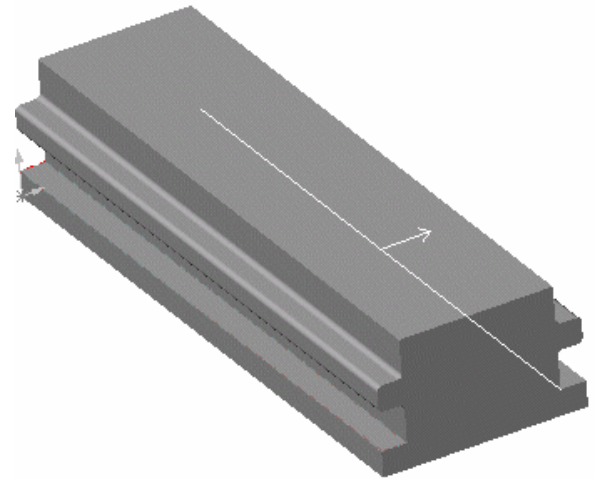
21. Click Finish, then click Close to exit the Wizard.

The tree lists the two Open Profile Features.

22. ▣↓ Click the Generate Operation Plan button on the CAMWorks toolbar.

Two Finish operations are listed in the Operation tree. Since you are using the same cutting conditions to machine the two features, you can combine the features into one operation.
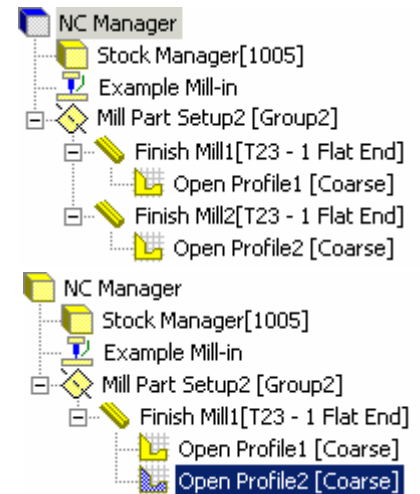
23. Click the plus signs next to the each finish operation to expand the items.

24. Drag Open Profile2 over Open Profile1 and release the mouse button.

Finish Mill2 operation is deleted and the two features can be machined with the same operation.

25. Click the minus sign next to Finish Mill1 to condense the item.

Before you can use the form tool, you need to add it to the Tool Crib. If you add a tool to the Tool Crib in the TechDB, the tool will be available to use in other parts. If a tool is used occasionally or only for a specific part, you can add the tool to the Tool Crib in the Machine dialog box or to the Tool Crib on the Tool Crib tab in the Operation Parameters dialog box so that it is available only for the current part. In this exercise, you add the tool in the Operation Parameters dialog box for the Finish Mill1 operation.

1. Right click Finish Mill1 in the tree and select Edit Definition on the shortcut menu.

2. Click the Tool Crib tab.

3. Click the Add button in the From library section.

4. In the Select Tool Filter dialog box, select User defined tool in the list and click OK.

The Tool Database – User Defined Tools form displays.

5. Select the Side Groove tool and click OK.

The tool is added to the Tool Crib.

6. Scroll down the list and select the User defined tool.

7. Click the <- Select button.

8. Click Yes to replace the corresponding holder.

   CAMWorks provides a default holder for any tool that does not have an associated holder. You can associate holders to tools individually when you add a tool to the Tool Crib in the TechDB or using the Default Tools form in the TechDB.

| | Mill Tool | Mill Holder | Tool Crib | | | | |
|---|---|---|---|---|---|---|

**Tools**

☐ Filter

<- Select

**From library**

Replace...

Add...

| | Usage | Stn. No. | Tool Type | Dia. | Rad. | Ti |
|---|---|---|---|---|---|---|
| | | 39 | Countersink | 0.5in | | 9C |
| | | 40 | Countersink | 0.625in | | 9C |
| | | 41 | Countersink | 0.25in | | 9C |
| | | 42 | Flat End Tapered | 0.1875in | 0in | |
| | 2 | 43 | Flat End | 0.032in | 0in | |
| | | 44 | Flat End Tapered | 0.25in | 0in | |
| | | 45 | Flat End | 0.0625in | 0in | |
| | | 46 | Flat End | 0.0937in | 0in | |
| | | 47 | Bore | 0.593in | | |
| | | 48 | User defined tool | 1in | 0in | |
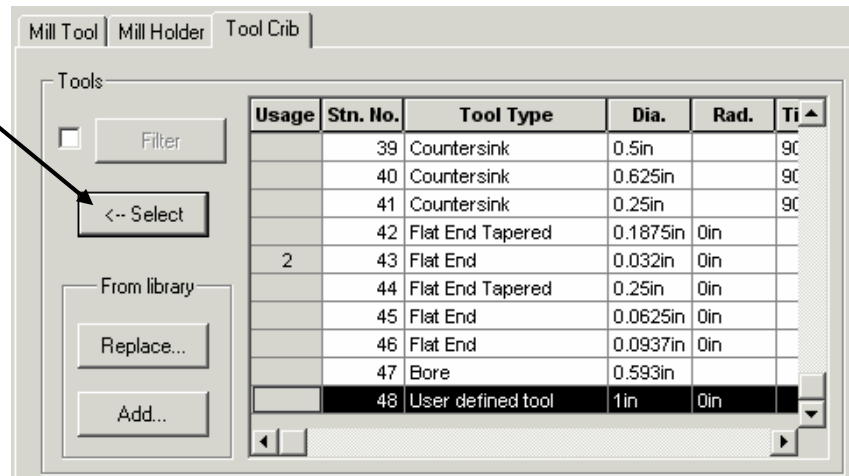
9. Click the Mill User Defined Tool tab.

   The Side Groove tool information is listed and a wire frame picture of the tool displays in the Preview area.

10. Click the Finish tab.

11. In the Depth Parameters section, set the First cut amt to 1in.

    Since the feature depth is 1.00in, this setting cuts the feature to depth in one pass.

12. In the Side Parameters section, set the Cut amt to 0.0625 and the Prev. allowance to 0.1875in.

    These two parameters determine the number of side passes. These values will generate three side passes.

13. Click the Feature Options tab.

    Notice that the Feature list currently shows Open Profile1. When machining multiple features with the same operation, most of the cutting conditions are the same and the settings in the Operation Parameters dialog box apply automatically to all features. To allow greater flexibility in machining individual features, cutting conditions that are feature-based are not applied automatically to all features. For some feature-based parameters, an Apply to All button is provided to set the parameters the same. For others, the parameters must be set for each feature.

14. Click the down arrow to the right of Open Profile1 and select Open Profile2.

15. Click the Finish tab.

16. Notice that your changes are retained for the depth cut amount and the side cut amount; however, the Prev. allowance value is the original default. This parameter must be set for each feature.

17. Change the Prev. allowance to .1875in.

18. Click the Leadin tab.

    You need to define a leadin and leadout in order to avoid gouging the part.

19. Select a Parallel leadin with a Leadin amount of 0.5in.

20. Use the same for the Leadout.

21. Click the Apply to All button to apply these settings to both features.

22. Click OK to close the dialog box.

23. ⬚ Click the Generate Toolpath button on the CAMWorks toolbar.

    The toolpaths are generated on both sides of the part.

24. Highlight the operation in the tree to view the toolpaths.

25. ⬚ Click the Simulate Toolpath button on the CAMWorks toolbar.

26. ▶ Click the Single Step button several times to step through the simulation.

    Note that Turbo mode does not support user-defined tools.
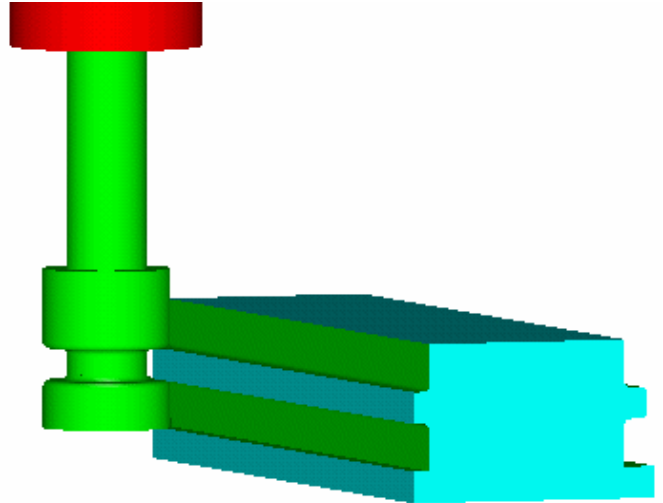
# TechDB 8

## What You'll Learn

### *Creating and Using User-defined Mill Tool Holders*

CAMWorks includes the capability to define mill tool holders and create a Holder library in the TechDB. The Holder library is a database of all holders available within a machining facility. When a holder is mated to a mill tool, the holder can be displayed during simulation. Displaying mill tool holders in simulation provides valuable feedback to determine whether a collision will occur during machining.

CAMWorks supports two types of mill holders. The first type, called Basic, has the shape of a larger diameter cylinder on top of a smaller diameter cylinder. The Basic holders are defined directly in the Holder library by specifying the Top diameter (D1), Bottom diameter (D2), Overall length (L1), and Bottom length (L2) as shown in the figure on the right. For many applications, the Basic mill holder shape may be adequate. However, in cases where additional detail in the holder is desired, a User defined holder can be used. The shape of the user defined holder is based on a SolidWorks part model. The exercise below describes how to create a user defined holder, add it to the Holder library and use it in CAMWorks.
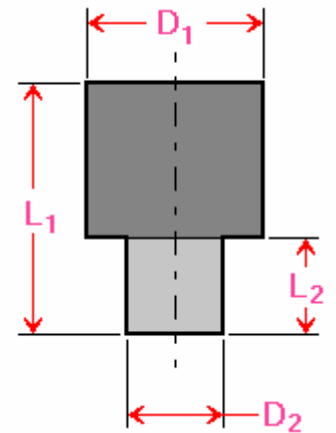
**Basic Holder Parameters**

### Create Holder in SolidWorks

The method for geometrically defining a mill holder is the same as for a user-defined tool. Creation of the shape is done in SolidWorks by drawing a cross section of the shape and a center line, and then creating a revolved solid as explained on page 7-26. The sketch in the figure shows the geometry used to create the holder used in this exercise.

**User-defined Holder Sketch**

2. Make sure the Units selection box below the navigation tree is set to **Metric**.

   When you enter information in the TechDB, you need to specify which section of the database you are editing.

3. Expand the Tooling item in the navigation tree, then expand Mill Tooling.

4. Expand Holder Utilities and click Holders.

   The Mill Holder Library (Metric) form displays.
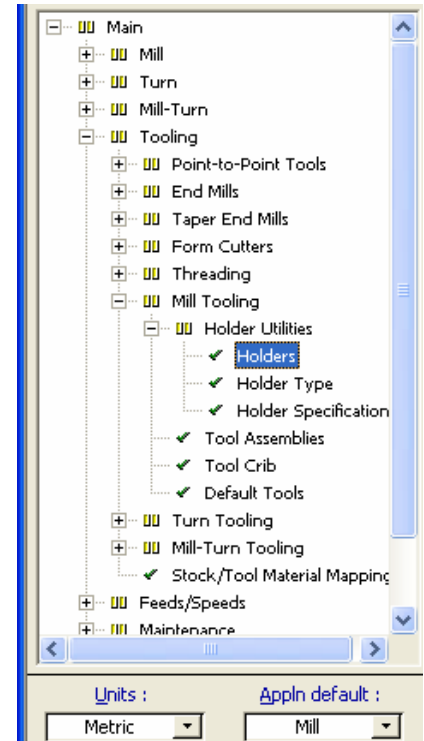
5. Define the holder using the following information:

| Field | Value | Comments |
|---|---|---|
| Holder number | Holder1 | User defined alphanumeric string. |
| Holder shape | User Defined | Specifies whether the holder shape is defined from parameter values or user-defined geometry: |
| | | Basic – The shape is defined parametrically from: Top diameter, Bottom diameter, Overall length, and Bottom length. |
| | | User Defined – The shape is defined from user defined geometry. |
| Holder type | Flange Adapter | This list is determined by the types on the Holder Types form. |
| Holder specification | CAT 45V | This list is determined by the specifications on the Holder Specification form. |
| User defined name & path | 📁 Click to locate file | Defines the folder and name of the user-defined holder file. Use the Browse button to locate the *\Tooling\MHolders* folder and select the MillHolder_CAT45-VFlange.mh file. |
| Comment | CAT 45V | User supplied comment. |

6. When you are finished, click Close. The information is saved automatically.

CAMWorks provides two methods of assigning holders to tools:

- In the less formal approach, a holder is mated to a tool on the Tool crib form. When creating the tool crib, do not select the Define using Tool Assemblies option. In this approach, tool management is focused more on the tool itself.

- In environments where a more formal strategy of managing both tool and holder assemblies is required, each tool and holder combination can be mated permanently using the Tool Assemblies form.

The Mill Tool tab in the Operation Parameters dialog box has a Holder page that allows you to change the holder for an individual operation, if necessary.

In this exercise, you use the Mill Tool Crib form to associate the holder to a tool.

7. Click the Tool Crib item under Tooling in the TechDB navigation tree.

   The Tool Crib (Metric) form displays.
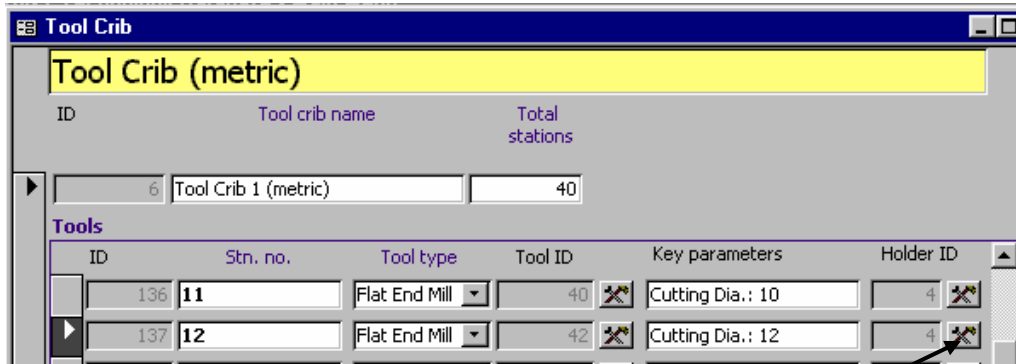


8. Scroll down the list and select Stn. no. 12.

9. Click the edit button next to the Holder ID.

10. In the Mill Holder Library form, select Holder1 and click Close.

11. Click Exit to close the TechDB.

12. Close **MillHolder_CAT45-VFlange.sldprt.** You are finished with this file.

## Use the Holder in CAMWorks

1. Open part file **MILL2AX_13.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. Click the CAMWorks Feature Tree tab.

3. Click the Options button on the CAMWorks toolbar.

4. On the Mill Features tab, make sure the Boss and Face options are not checked, then click OK.

5. You can use the default machine and stock.

6. Click the Extract Machinable Features button on the CAMWorks toolbar.

   AFR finds an Open Pocket and a Rectangular Pocket.

7. Right click Rectangular Pocket1 in the tree and select Parameters on the shortcut menu.

8. In the Parameters dialog box, select Coarse for the Attribute, then click OK.

   This attribute has been set up in the TechDB to generate a Rough Mill operation for a Rectangular Pocket feature.

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

10. ⬛ Click the Generate Toolpath button on the CAMWorks toolbar.

    The toolpaths are generated for the three operations.

11. ⬛ Click the Simulate Toolpath button on the CAMWorks toolbar.

12. On the Simulate Toolpath toolbar, make sure the Stock, Tool and Holder options are set to Shaded Display.

13. ⬛ Click the Show Holder Collision button to turn it on.

14. ▶ Click the Run button.

    Notice that the tool holder gouges the boss.

15. ❌ Click the Cancel button to exit the simulation.

16. Right click Rough Mill2 in the Operation tree and select Edit Definition on the shortcut menu.

17. Click the Tool Crib tab.

18. Scroll down the tool list and select the tool in Stn. No. 12.

    You associated the user-defined holder to this tool in the TechDB.

19. Click the <- Select button.

20. Click Yes to replace the corresponding holder.

21. Click the Mill Holder tab.

    The information for Holder1 displays.

22. Click the Mill Tool tab and change the Protrusion to 65mm.

    This parameter defines the distance from the tip of the tool to the bottom of the holder.

23. Click OK to close the dialog box.

24. Right click Rough Mill2 in the tree and select Generate Toolpath on the shortcut menu.

    You need to regenerate the toolpath because you selected a different diameter tool.

25. ⬛ Click the Simulate Toolpath button on the CAMWorks toolbar.

26. ▶ Click the Run button.

    This holder does not gouge the boss.
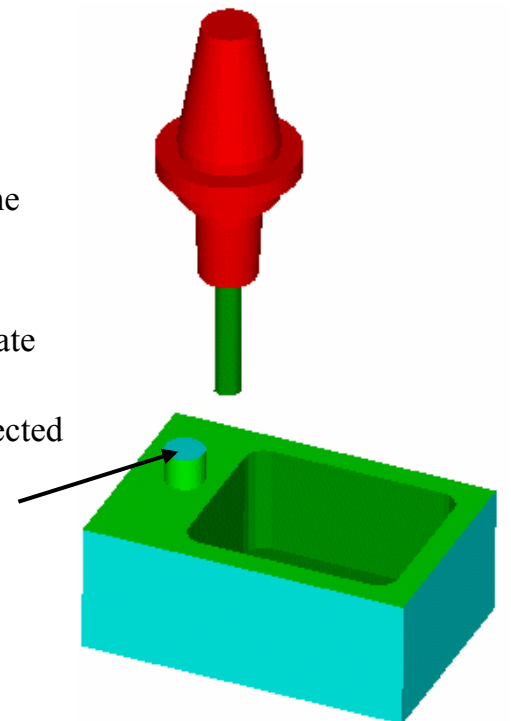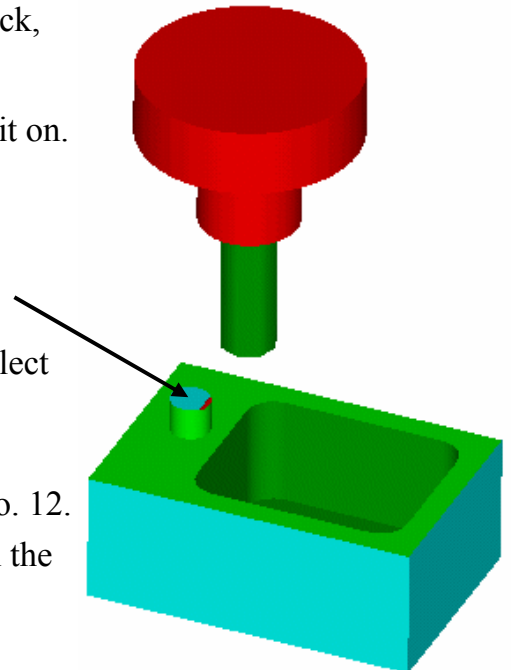
# TechDB 9

CAMWorks supports keyway and dovetail cutting tools. In this exercise, you create a user-defined T-slot attribute in the TechDB using a keyway cutter and apply it to a machinable feature.

## What You'll Learn

- Defining a keyway cutter in the Technology Database
- Creating a user-defined T-slot attribute in the Technology Database
- Inserting a 2 Axis Feature and applying the TechDB T-slot attribute
- Generating Operations and Toolpaths

1. Open the part file **MILL2AX_8.SLDPRT** in the *\Examples\Mill* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\Mill*).

2. ![CW] Click the CAMWorks Feature Tree tab.

3. ![icon] Use Example Mill–in in the Feature tree.

4. ![icon] Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager and select Extract Machinable Features.

### Defining an Attribute in the Technology Database

First, you need to define the keyway cutter in the TechDB.

1. ![icon] Click the TechDB button on the CAMWorks toolbar or select Technology Database on the CAMWorks menu.

   Microsoft Access starts and the TechDB Main Menu displays.

2. Make sure the Units selection box is set to **Inches**.

3. Expand the Tooling item in the navigation tree, then expand Form Cutters.

4. Click Keyway.

5. On the Tools Database – Keyway (inches) form, go to the last record (0 in the ID field) and use the following information and define the tool:

   - Tool ID = 1010          Diameter = 1.75
   - Shank = .75              Bottom radius = .05
   - Top radius = .05         Eff. cut length = .5
   - Overall length = 3       Protrusion = 2
   - Hand of cut = Right hand No of flutes = 8
   - Tool class = carbide     Comment = 1.75" Keyway
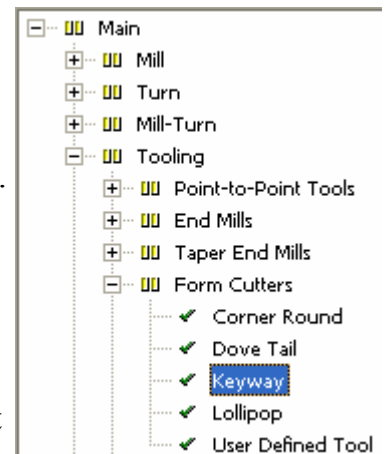
6. After verifying the tool is defined on the form, click Close.

Next, you create a user-defined attribute to apply a machining process in the TechDB.

7. Expand Mill in the navigation tree and expand Features and Operations

8. Click User Defined Attributes.
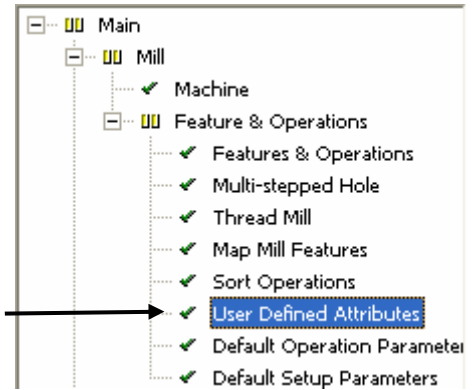
   The Feature & Attributes form displays.

9. For the Feature, select Open Profile on the drop down list.

10. In the User Defined Attributes section, click in the empty cell below Taper and type T-slot.

11. Click Close.
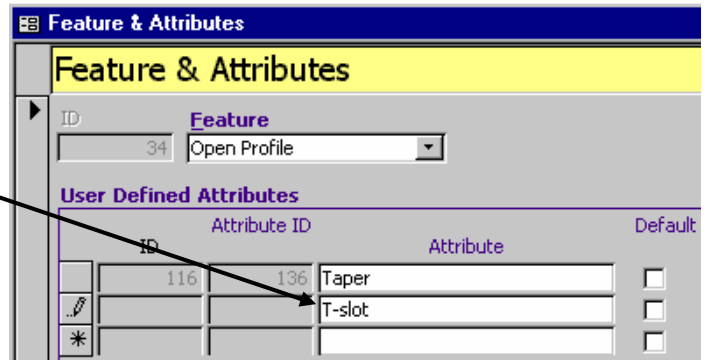
12. Click Features & Operations under Feature & Operations in the tree.

    The Feature & Operations form displays.

13. For the Feature Type, select Open Profile on the drop down list.

14. In the Feature Conditions section, scroll down to the last record, which is identified with an asterisk (*), and click the asterisk.

15. In the Attribute column, click the down arrow and select T-slot from the list.

16. Click in the Feature Depth <= cell and type 1.
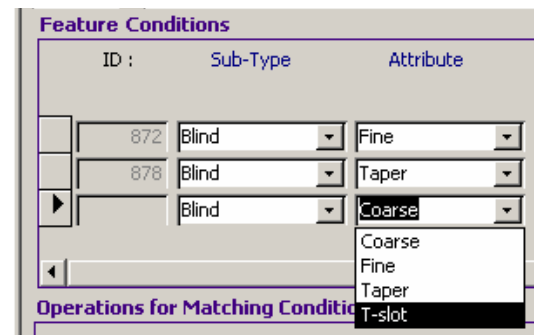
Now, you can specify the tool to use and the feature conditions to apply to this attribute.

17. In the Feature Conditions section, click the record above the T-slot record you just entered, then click the T-slot record to activate the Operations for Match Conditions section.

18. In the Operations for Matching Conditions section, click the first record.

19. In the Operation column, click the down arrow and select Finish Mill.
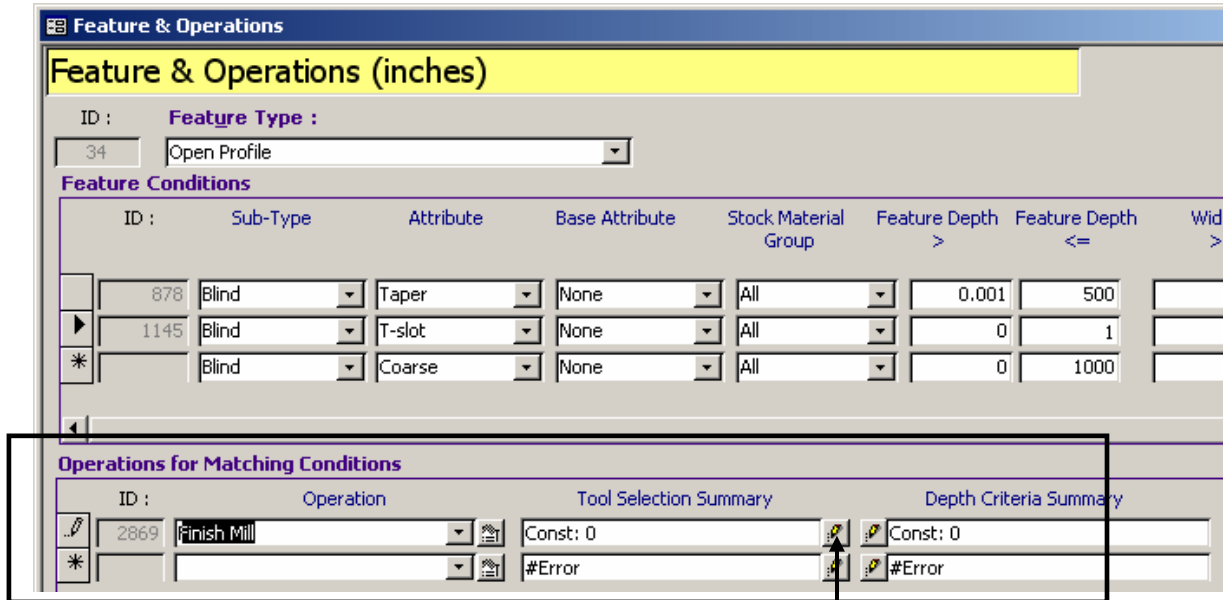


20. Click the Pencil button next to the Tool Selection Summary cell.

The Tool Selection form displays.
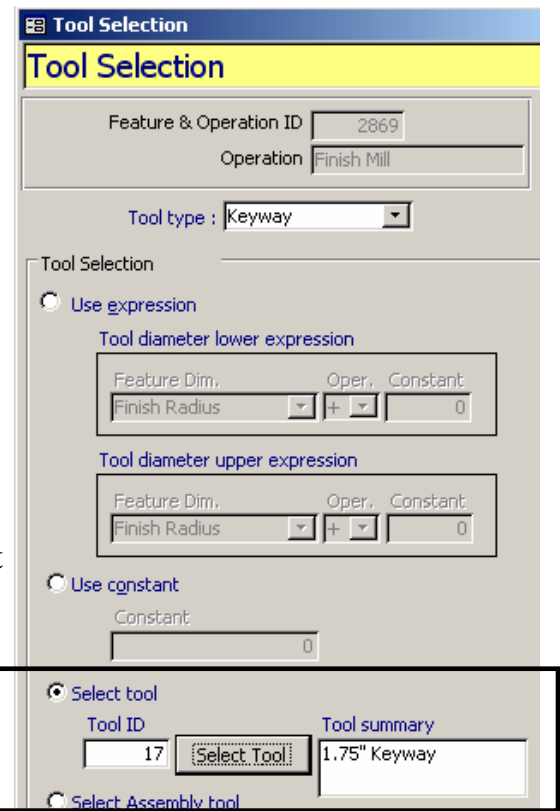
21. Select Keyway for the Tool Type.

22. Click the Select tool option and click the Select Tool button.

The Tools Database – Keyway form displays.

23. Select the 1.75" keyway tool defined earlier in this exercise.

24. Click OK.

The tool information displays on the Tool Select form.

Click Close.

25. Click the Pencil button next to the cell in the Depth Criteria Summary column.
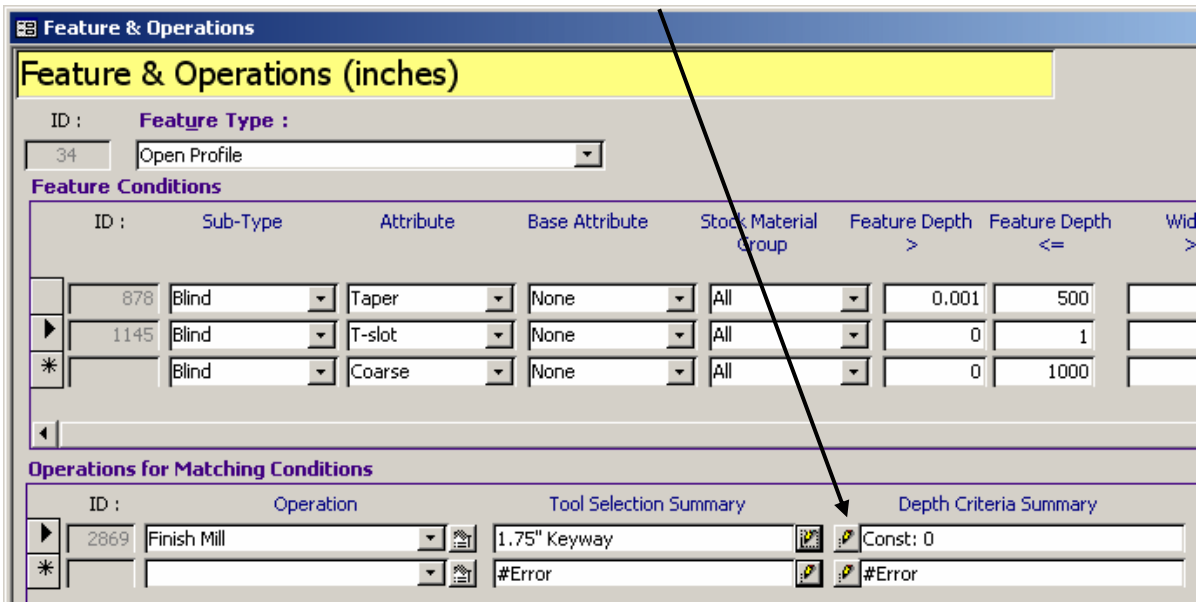


26. In the Depth Evaluation form, click the Use expression option and use the defaults in the Matching depth expression section.

27. Click Close.

The Feature & Operations form should look like the figure below.

28. Click Close.

29. On the main menu, click Exit to close the TechDB.
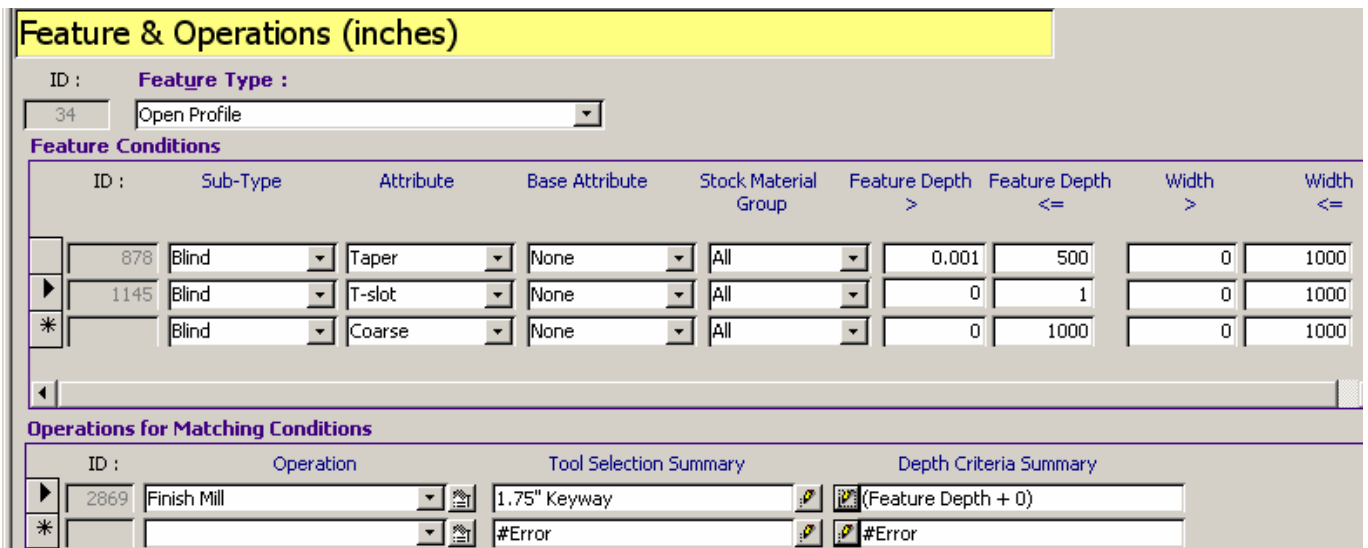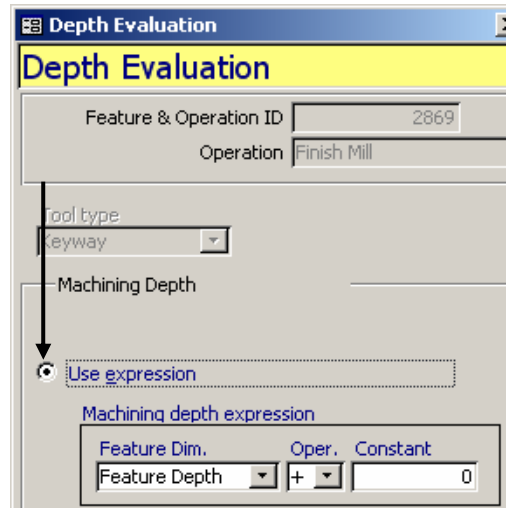
## *Inserting a 2 Axis Feature and Applying the T-slot Attribute*

Define two machinable features and apply the new TechDB information to one of the features.

1. Close **MILL2AX_8.SLDPRT** and reopen it to make sure your changes to the TechDB will be used.

2. ![CW icon] Click the CAMWorks Feature Tree tab.

3. Right click NC Manager in the tree and select Collapse items.

   This makes it easier to work with the items in the tree.

4. Click the plus (+) symbol next to Mill Part Setup3 to expand it.

5. Right click Mill Part Setup3 and select Insert 2.5 Axis Feature.

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

6. Select Open Profile for the Feature Type.

7. Rotate the part and pick the edge shown in the figure.

   You may find it easier to pick edges and faces if you the change the part display from solid to Hidden lines removed.

8. Click Next.

   The 2.5 Axis Wizard: End Conditions dialog box displays.
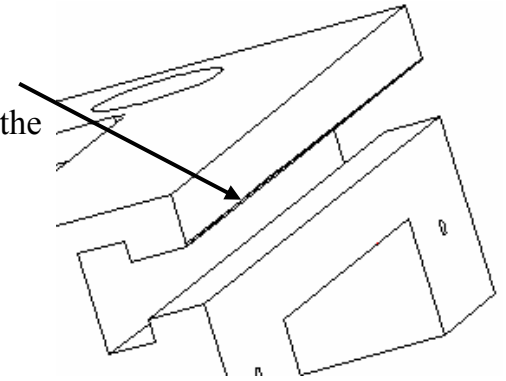
9. Pick the face shown in the figure.

   Upto Face displays as the End Condition Type and the depth is calculated automatically.

10. Make sure the Edit feature profiles option is checked.

11. Click Next.

    Notice that the direction of the arrow that displays on the part indicates the incorrect side to cut.
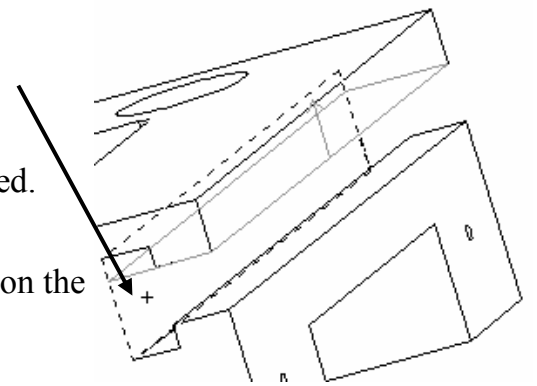
12. In the 2.5 Axis Feature Wizard: Edit Feature Profiles dialog box, select the Flip direction to cut option.

    The arrow direction changes as shown in the figure.

13. Click Finish.

    The 2.5 Axis Feature Wizard: Feature & Cross Section Definition dialog box displays again.
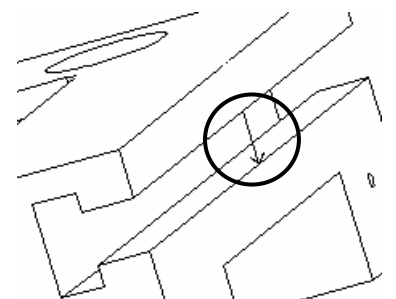
Define the open profile for the keyway cutter.

14. Make sure Open Profile is selected for the listed for the Feature Type.

15. Pick the bottom inside edge of the t-slot.

16. Click Next.

17. For the Attribute, select T-slot.
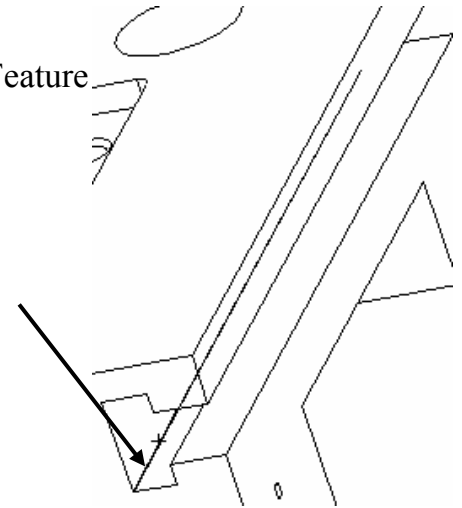
18. For the End Condition, rotate the part and pick the underside of the keyway.

    Upto Face displays as the End Condition Type and the depth is calculated automatically.

19. Remove the check mark from the Edit feature profiles option and click Finish.

20. Click Close to exit the wizard.
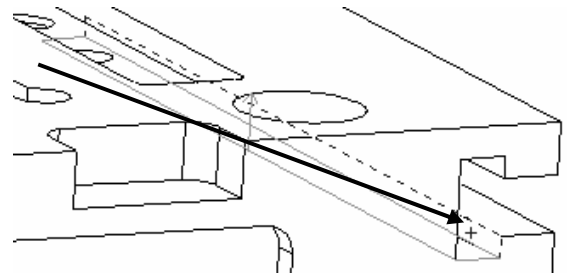
    The two Open Profile Features are listed in the tree.

21. Hold down the left mouse button and drag Open Profile1 over Open Profile2.

    This is the order you would want to machine this area. Note that you can also change the machining order in the CAMWorks Operation tree after operations have been generated.

## *Generating Operations and Toolpaths*

1. Right click Mill Part Setup3 and select Generate the Operation Plan.

   Two Finish Mill operations have been generated for the Open Profile features.

2. Right click the first Finish Mill operation in the tree and select Edit Definition.

3. On the Leadin tab, select Parallel for the Leadin type.

4. Set the Leadin amount to 1in and the Leadin overlap to 0.1in, then click OK.

5. Right click the second Finish Mill operation and select Edit Definition.

6. Make the same changes on the Leadin tab, then click OK.

7. On the Finish tab, set the First cut amount and Max cut amount to 1in, then click OK.

   This value is greater than the depth of the profile and will generate only one pass with the keyway cutter.

8. Right click Mill Part Setup3 and select Generate Toolpath.

9. Right click Mill Part Setup3 and select Simulate Toolpath.

10. Set the End condition to Next Operation.

11. ▶ Click the Run button.

12. ▶| When the first operation has completed, click the Single Step button to see the keyway tool cut the T-slot.

# Index