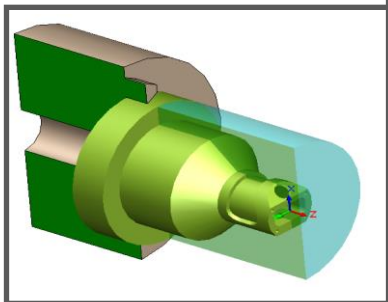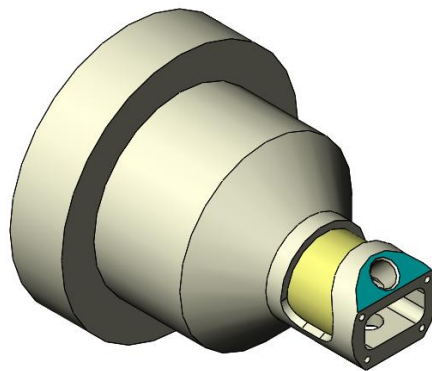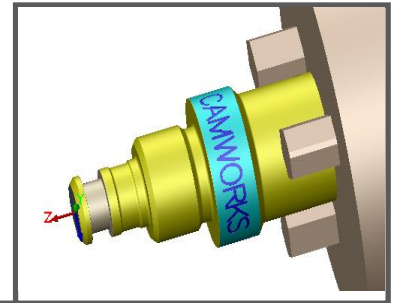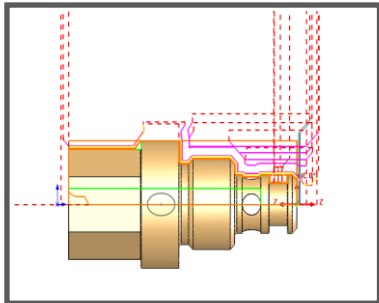# CAMWorks 2022

Mill-Turn Tutorial

Disclaimer

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

Trademarks

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

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

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

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

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

TABLE OF CONTENTS

1. LEARNING MILL-TURN BASICS

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

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

These exercises have been developed to provide supplemental information on Mill-Turn and assume you are familiar with CAMWorks Mill and Turn. The *CAMWorks Mill-Turn Tutorial* contains more detailed information about using CAMWorks.

The exercise parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\Mill-Turn* folder in the CAMWorks data folder.

Typical location: Drive:*CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn*

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

Mill-Turn 1

Steps to Generate Mill Toolpaths and NC Code in Part Mode

1. Model the part or open the part file in SOLIDWORKS/CAMWorks Solids.

2. Change to the CAMWorks Feature tree.

3. Define the Machine and modify the Post Processor parameters.

4. Define the Stock.

5. Define the machinable features.

6. Generate the operation plan and adjust operation parameters.

7. Generate toolpaths and run the simulation.

8. Adjust parameters if necessary.

9. Post process the toolpaths.

When you install CAMWorks, the exercise parts are installed in the *\Examples\Tutorial_Parts\Mill-Turn* folder in the CAMWorks data folder (e.g., *\CAMWorksData\CAMWorksxxxx\Examples\Tutorial_Parts\Mill-Turn*).
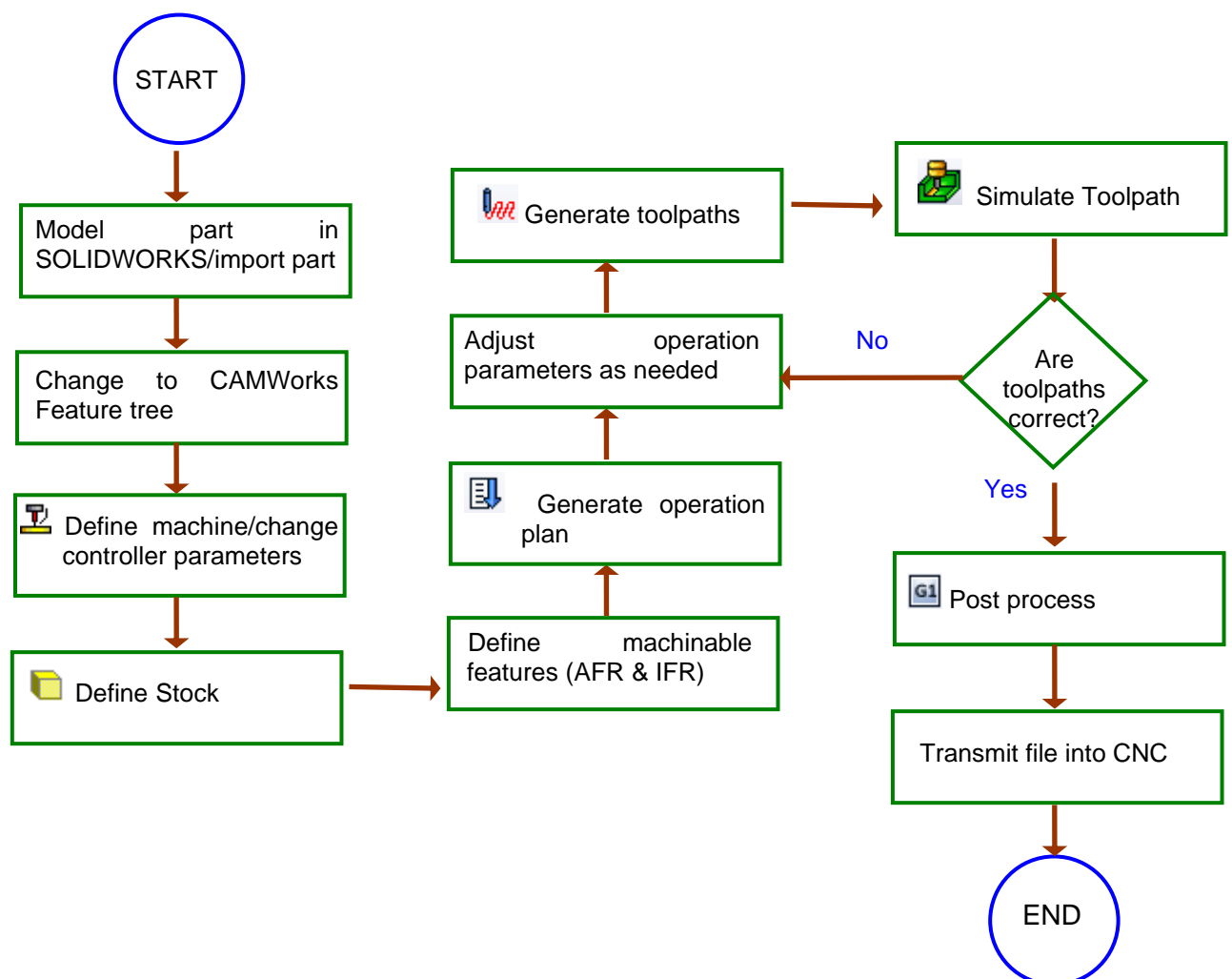
Step 1: Model Part in SOLIDWORKS/CAMWorks Solids or Import part

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

Open the part file **MT_1.SLDPRT** in the following folder*.*
Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn
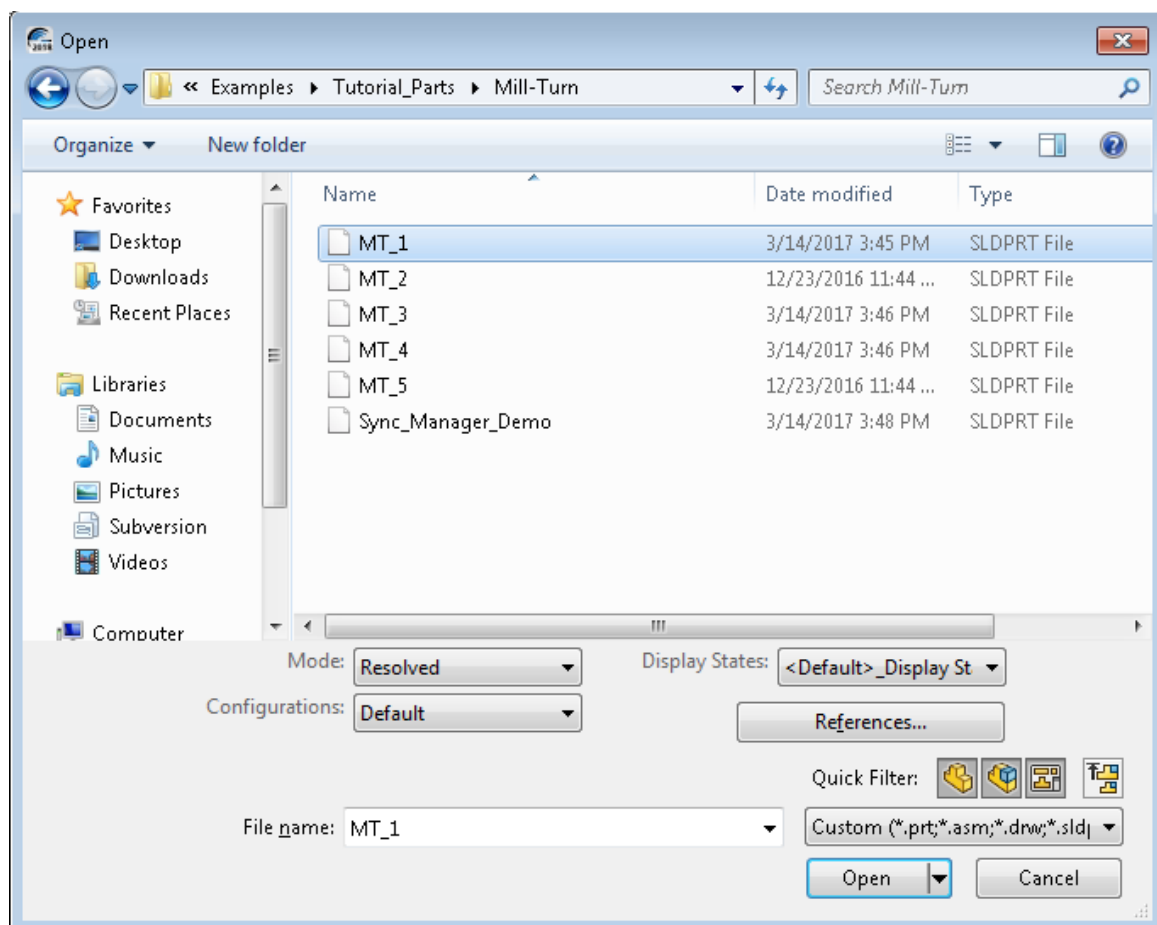
Opening the Solid Part file

Viewing the FeatureManager Design Trees

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

To use CAMWorks, you need to move between SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks

Solids trees and the CAMWorks Feature trees. Click the 📌 *Pin* button to continuously view this Tree area.

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

Tabs for SOLIDWORKS/
CAMWorks Solids Trees

CAMWorks Feature Tree tab

Line that divides the tree from the
graphics area

Graphics area
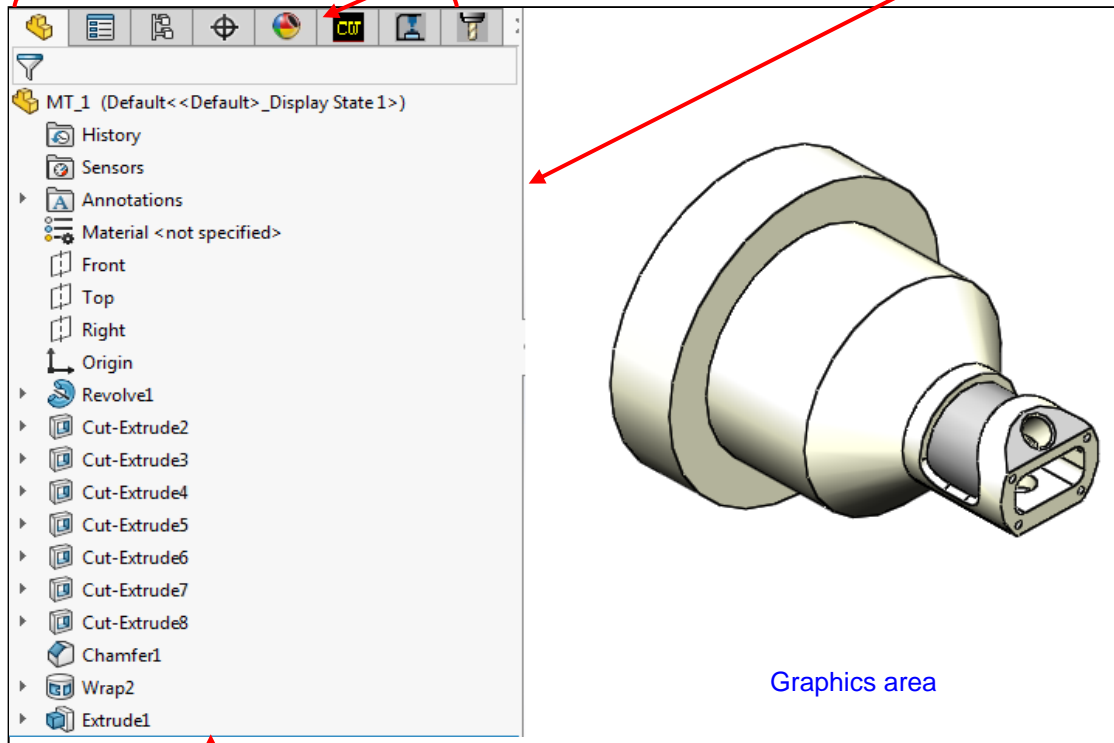
Items under the Feature Manager
Design tree

Step 2: Change to CAMWorks Feature Tree

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

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

CAMWorks Command Manager Area

CAMWorks menu

Items under CAMWorks Feature tree

Graphics area

Part Model

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, Configurations, 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.

- **Configurations**

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

- **Stock Manager**

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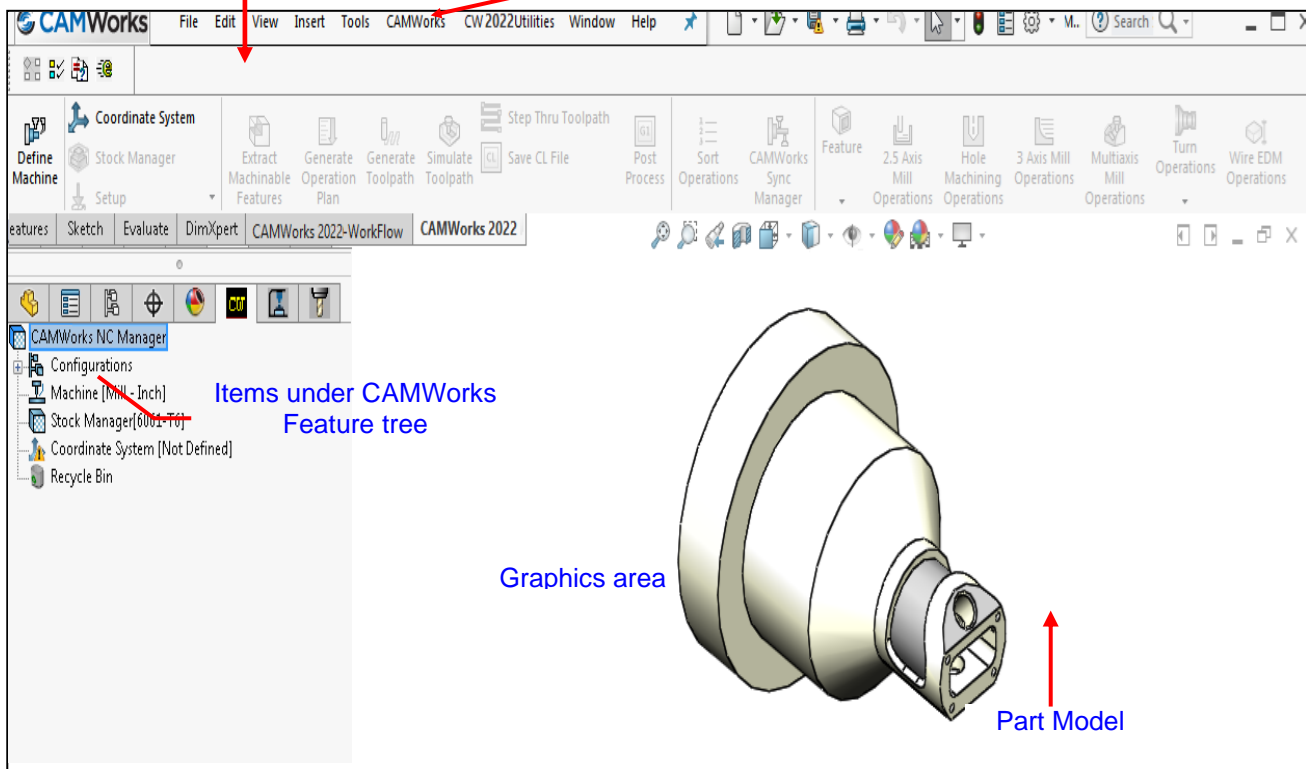
- **Machine**

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

- **Recycle Bin**

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

CAMWorks Command Manager

Click *Tools* on the SOLIDWORKS menu bar and select *CAMWorks* from the dropdown menu./Click *CAMWorks* on the *CAMWorks Solids* menu bar.

This action displays the CAMWorks Command Manager. It provides access to the main CAMWorks commands. The commands are explained in the CAMWorks context-based Help.
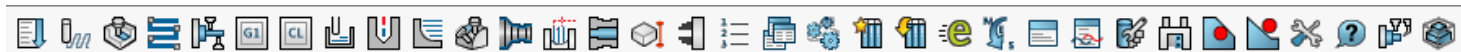
CAMWorks Command Manager

Customization of CAMWorks Command Manager

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

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

Alternative Access to CAMWorks Commands

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

CAMWorks Options

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

To open the *CAMWorks Options* dialog box:

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

    OR

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

    The *Options* dialog box is displayed.

2. In this dialog box, go to the *Mill Features* tab. Under *Extract Machinable Features* group box, ensure that the *Method* is set to *AFR*.

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

CAMWorks Context-based Help

In addition to tutorial documents, CAMWorks is provided with a context-based help. Every dialog box and interface within CAMWorks has an associated *Help* button. Click on the *Help* button

on the CAMWorks Command Manager to open the context-based Help. Every parameter and tab of each dialog box in explained in the context-based Help.

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 includes:

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

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

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

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

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

- **Post Processor:** The post processor identifies the format of the NC G-code output.

Define the Machine

1. Right click the *Machine* item in the CAMWorks Feature tree.

    OR

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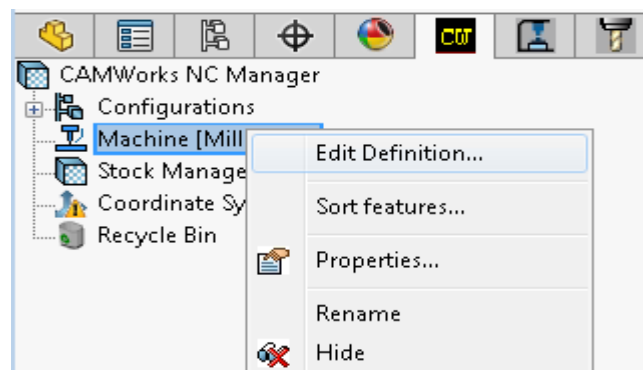

**Select 'Edit Definition' on
context menu**

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

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

2. In the Available machines list, highlight the *Mill-Turn Single Turret - Inch* and click on the *Select* button.
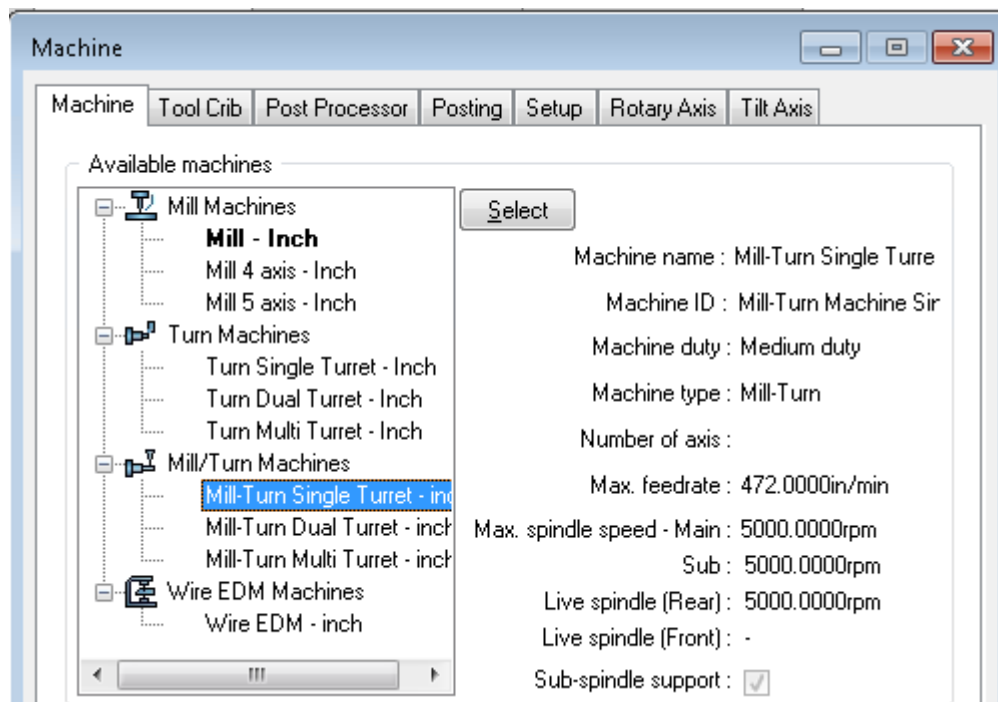
Machine tab of Machine Dialog Box

3. Click the *Tool crib* tab.

4. In the Available tool cribs field, highlight the *MT Tool Crib 2 (Rear)* option and click on the *Select* button to define the Active tool crib.

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

   When you define your machine tools in the Technology Database, you can set up your own tool cribs.

5. Make sure Tool crib priority option is unchecked.

Tool Crib tab of Machine Dialog Box

6. Click the *Post Processor* 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 tutorial post processors. Contact your CAMWorks Reseller for more information on obtaining and/or customizing post processors for your machine tool.

If the post processor do not display, use the *Browse* button to locate the folder containing the files (*.ctl).

If MT2AXIS-TUTORIAL (the tutorial Mill-Turn post processor) is not the Active post processor, highlight it in the list and click the *Select* button. This post processor is used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.

When you select this post processor, a short description displays in the window. This window contains information only if an optional file has been created for the post processor.
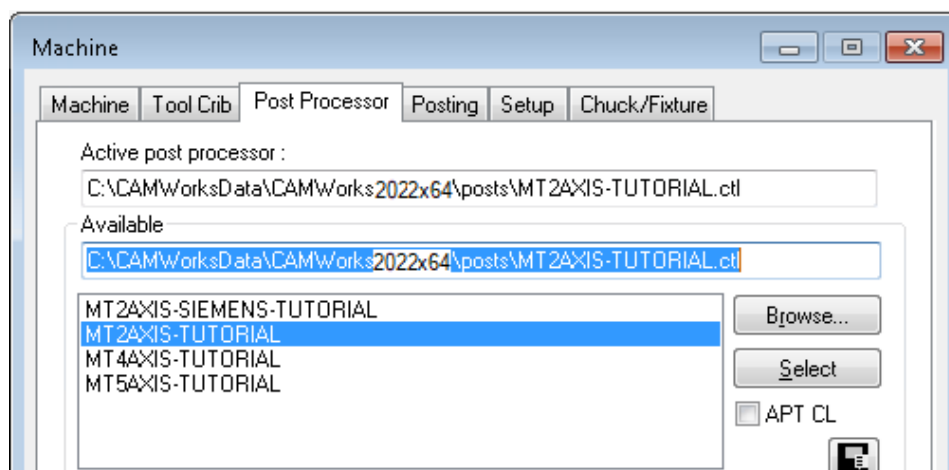
Post Processor tab of Machine Dialog Box

7. Click the *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. The parameters are included in the Setup Sheet.

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

Step 4: Define the Stock

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

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

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

   OR

   Right click *Stock Manager* and select *Edit Definition* on the context menu.
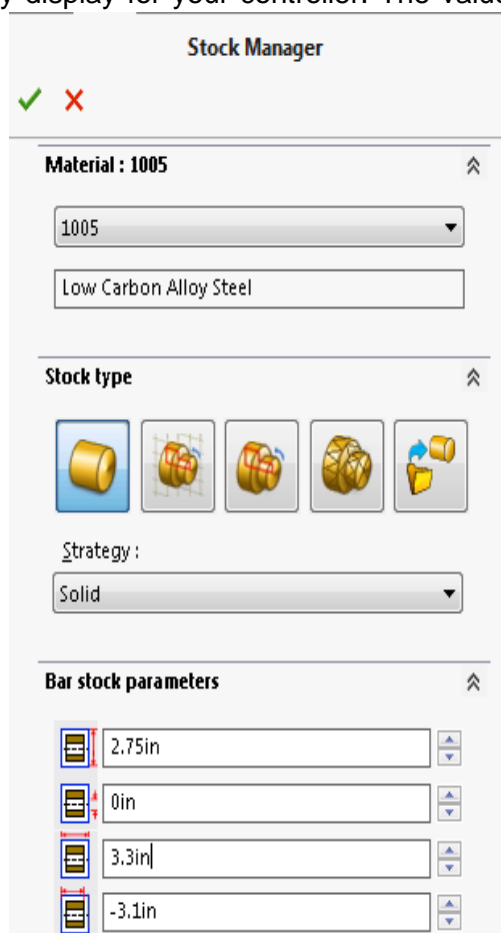
Stock Manager dialog box

The Stock Manager dialog box is displayed.

2. For the Stock type, use the default *Round Bar Stock* and set the following bar stock parameters:

 - Outside Diameter = **2.75in**

 - Inside Diameter = **0in**

 - Stock Length = **3.3in**

 - Back of Stock Absolute = **-3.1in**

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

 The Stock is updated in the graphics area.

Step 5: Define Machinable Features

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

- **Automatic Feature Recognition (AFR)**

 Automatic Feature Recognition analyzes the part shape and attempts to define most common mill and turn machinable features. Depending on the complexity of the part, AFR can save considerable time in defining features.

- **Interactively Created Features**

 If AFR does not recognize features you want to machine, you can use the *New Turn, New 2.5 Axis Feature, New Multi Surface Feature* and *New Wrapped Feature* commands to define the features.
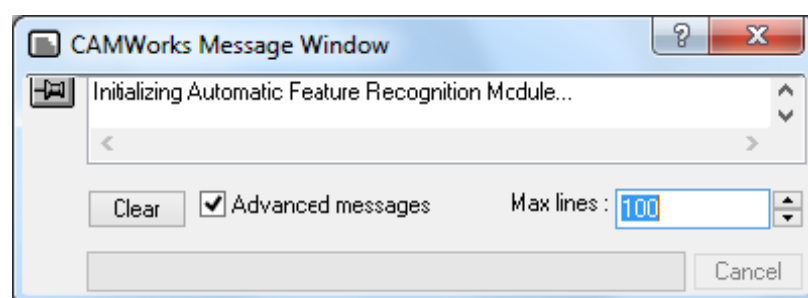
Using Automatic Feature Recognition (AFR)

Define Machinable Features Automatically

1. 🔲 Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

 The CAMWorks Message Window is displayed.

 The CAMWorks Message window displays automatically to report the progress and status of the current process.

CAMWorks Message Window

You can control whether this window displays temporarily or permanently.

2.   Click on the *CAMWorks Options* button on the CAMWorks Command Manager.

    The Options dialog box is displayed.

3. Check the *Message Window* option on the General tab in the Options dialog box.

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

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

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

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

    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.
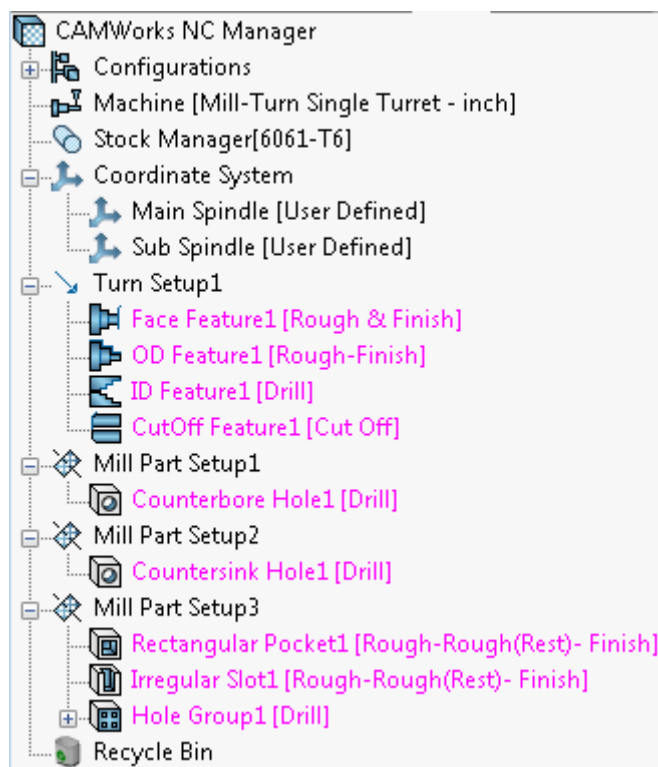
    Did you know: Features display in a different color when they have no operations generated. Once operations are successfully generated, they display in another color. If operations are not generated for certain features (this occurs when feature conditions have not been defined in the Technology Database), then they retain their original color indicating non-generation of operations. You can set the color on the Display tab in the Options dialog box.

6. Right click *Cutoff Feature1* under Turn Setup1 in the Feature tree, select *Delete* on the context menu.

7. Click *Yes* to confirm.

    The part will be held in a chuck so this feature is not needed.

8. Right click *Irregular Slot1* under *Mill Part Setup3* in the Feature tree, select *Delete* on the context menu.

9. Click *Yes* to confirm.

You are going to define this feature as a Corner Slot in Mill Part Setup2.
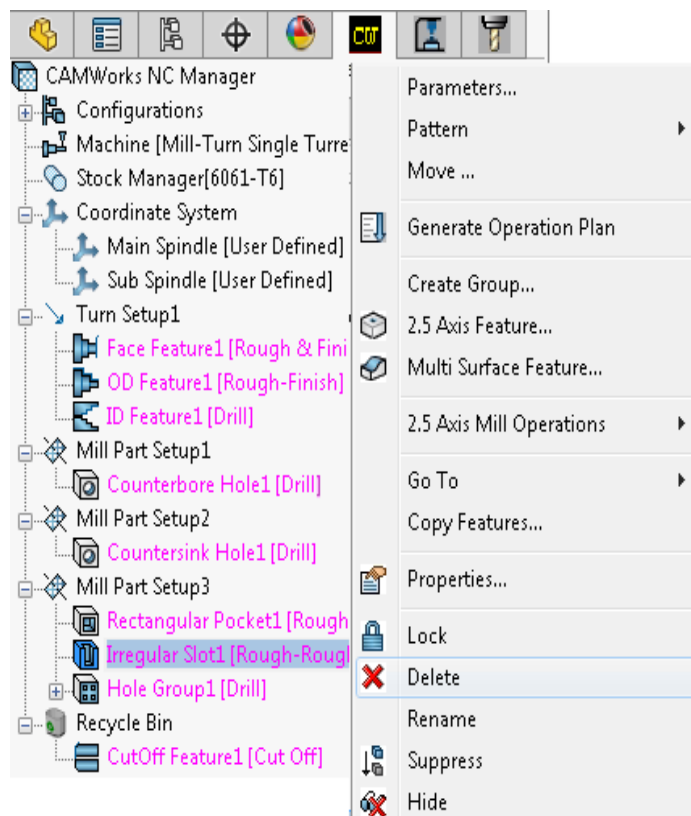
Deleting the features recognized by AFR

Defining Features Interactively

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

Insert 2.5 Axis Feature in Mill Part Setup1

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

The 2.5 Axis Feature: Select Entities dialog box is displayed.

2. Click the down arrow next to the Type list box and select *Corner Slot*.

3. Rotate the part and pick the flat face on the OD of the part *(highlighted in orange)*.
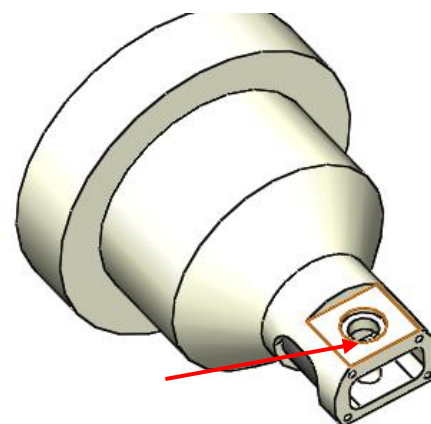
Flat face of OD feature

This will be a Y axis cut. The CW-Face is listed in the selected Entities field.

4. Make sure *Check for taper & fillets* is not checked.

    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 *End Condition* button.

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

6. Set the Strategy to *Rough-Finish.*

7. Leave the End condition Type set to *Blind.*

8. Set the Depth to **0.15in**.

9. Make sure the *Direction (indicated by red arrow)* is correct on the feature. If not, click the check box next to the Reverse direction option.

    The correct direction is shown in image.

10. ✔ Click *OK* button.

    Rectangular Corner Slot1 is displayed in the Mill Part Setup1 in CAMWorks Feature tree.
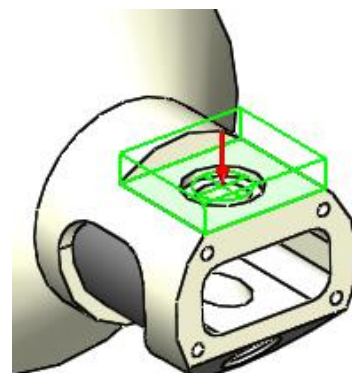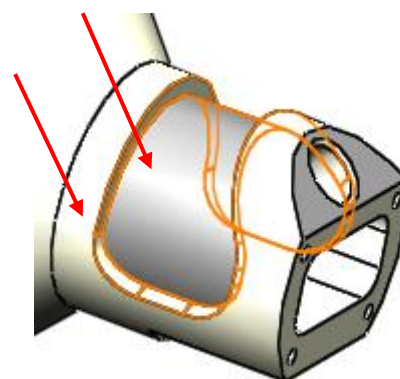
Insert Wrapped Feature in Mill Part Setup1

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

1. Right click *Mill Part Setup1* in the tree and select *Wrapped Feature* on the context menu.
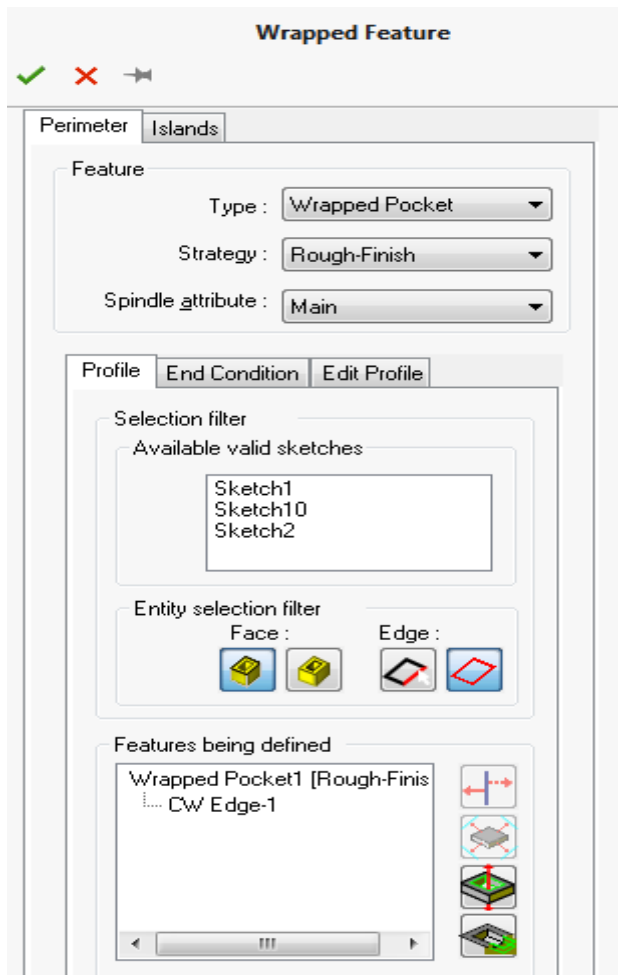
    This will be a C axis cut.

2. Set the Strategy to *Rough-Finish.*

3. Make sure Feature Type is set to *Wrapped Pocket.*

4. Pick anywhere on the *ID part (highlighted in orange color).*

5. Click on the *End Condition* tab.

6. Pick anywhere on the small OD of the part to set the depth.
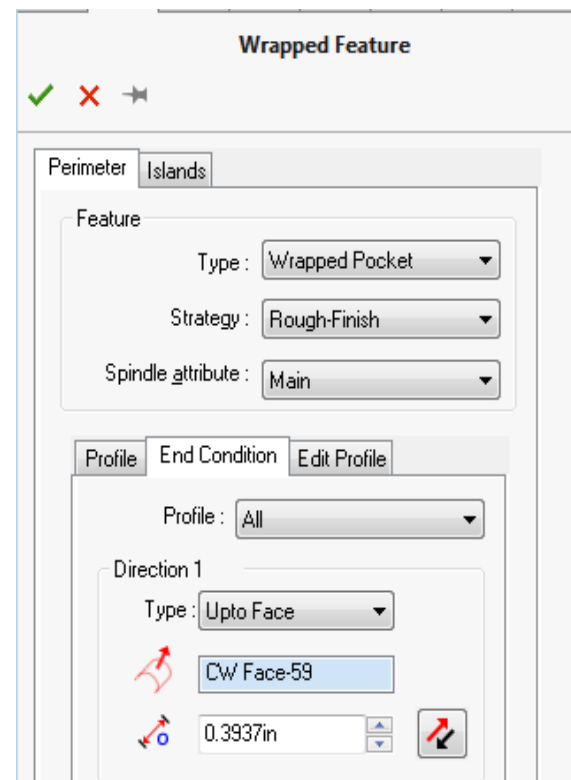
7. ✔ Click *OK* to insert the changes.

    Wrapped Pocket1 is listed in the Mill Part Setup1.

Cylindrical face on part

**Profile tab in New Wrapped Feature dialog box**

End Condition tab in New Wrapped Feature dialog box

Insert a 2.5 Axis Feature in Mill Part Setup2

1. Right click *Mill Part Setup2* in the tree and select *2.5 Axis Feature* on the context menu.

2. In the 2.5 Axis Feature: Select Entities dialog box, select *Corner Slot* for the Type.

3. Rotate the part, if necessary, and pick the angled face on the part *(highlighted in blue)*.

   This will be a B axis cut.

4. Click *End Condition* button.

5. Set the Strategy to *Rough-Finish.*

6. Leave the End condition Type set to *Blind*.

7. Set the Depth to **0.15in.**

8. Click *OK* button.

   Irregular Corner Slot1 is listed in the Mill Part Setup2
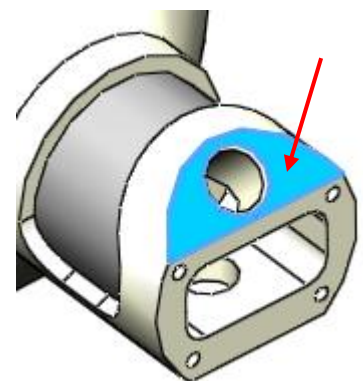
Angled face of the part

# 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. These commands are explained in the CAMWorks context-based Help.

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

OR

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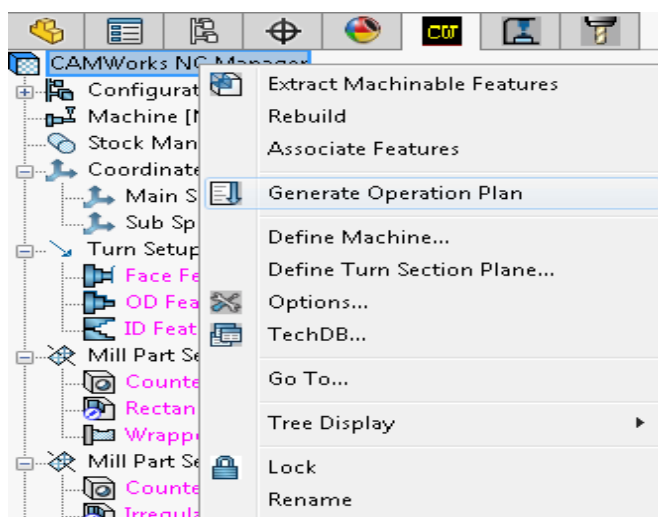
Generate Operation Plan command on CAMWorks NC Manager Level

Note:

- If you execute the *Generate Operation Plan* command from the Command Manager or the CAMWorks NC Manager level, then operations will be generated for all prismatic features in the tree, regardless of the active item in the tree.

- If you execute the *Generate Operation Plan* command from the Mill-Turn Part Setup level, then operations will be generated only for those prismatic features listed under the given Mill/Turn Setup.

- If you execute the *Generate Operation Plan* command at the feature level (by right-clicking on a feature listed in the Feature tree and executing *Generate Operation Plan* command from the RMB context menu), then operations will be generated only for the selected feature.

CAMWorks generates the operation plan for all the machinable features and the operations are listed in the CAMWorks Operation tree, 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 and Turn Part Setups in the same order as the machinable features in the Turn and Mill Part Setups. At the top of the tree is the CAMWorks NC Manager. The Stock Manager and Machine items are the same as in the CAMWorks Feature tree. You can change the stock size and shape and the post processor used by CAMWorks to produce G-code.

The CAMWorks Operation tree allows you to:

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

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

The operations that are generated by CAMWorks are based on information stored in the Technology Database. These operations are intended to be used as a starting point. Each operation contains parameters that affect how the toolpath is created and specific parameters that will be output to the NC program. These parameters can be edited before generating the toolpaths and post processing the part.

Operations generated on executing GOP command
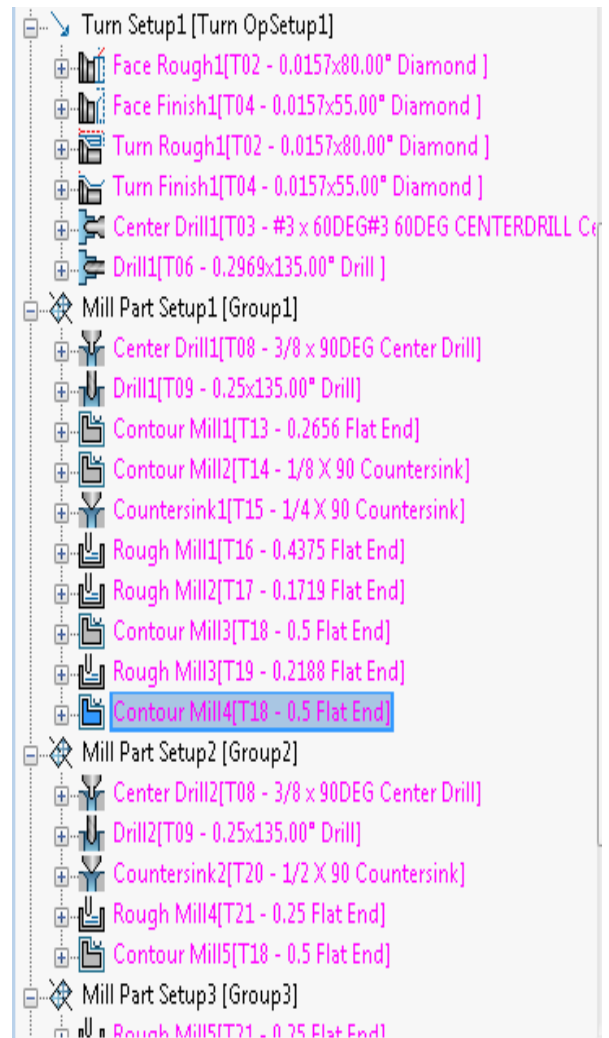
1. Double click *Contour Mill4* in Mill Part Setup1 in the Operation tree.

2. Click the *Tool Crib* page of Tool tab.

3. Highlight a *0.25in Flat End Mill* in the tool list and click the *Select* button.

4. Click *Yes* to replace the corresponding holder.

5. Click *OK* to apply the changes.

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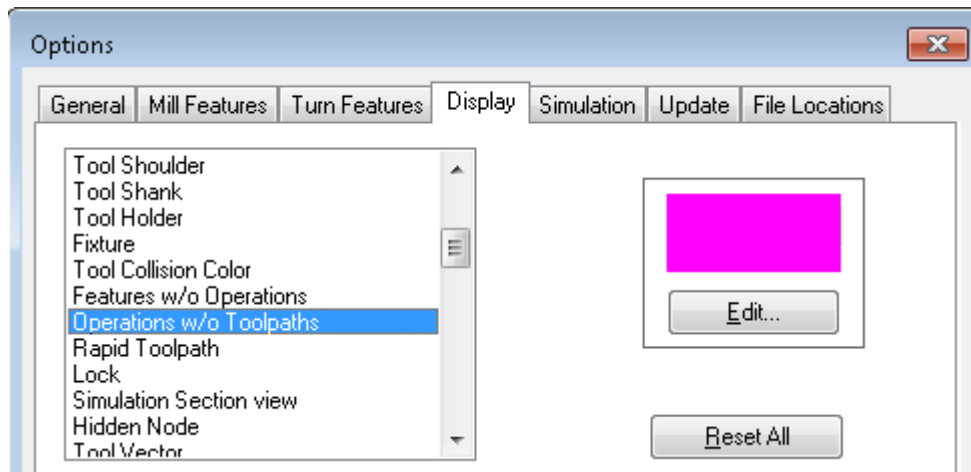
Display tab of Options dialog box

Step 7: Generate Toolpaths

CAMWorks calculates toolpaths using the operation parameters and the feature's size and shape. After toolpath is generated for a particular operation, then that operation is displayed in black-colored font in the Operation tree.

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

OR

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

Note:

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

- Executing the *Generate Toolpath* command from the CAMWorks NC Manager level generates toolpaths for all the operations.

- Executing the *Generate Toolpath* command at Mill/Turn Part Setup level generates toolpaths only for operations listed under the given Mill/Turn setup.

- You can also generate toolpaths for each operation individually by right-clicking on an operation and executing *Generate Toolpath* command on the context menu.
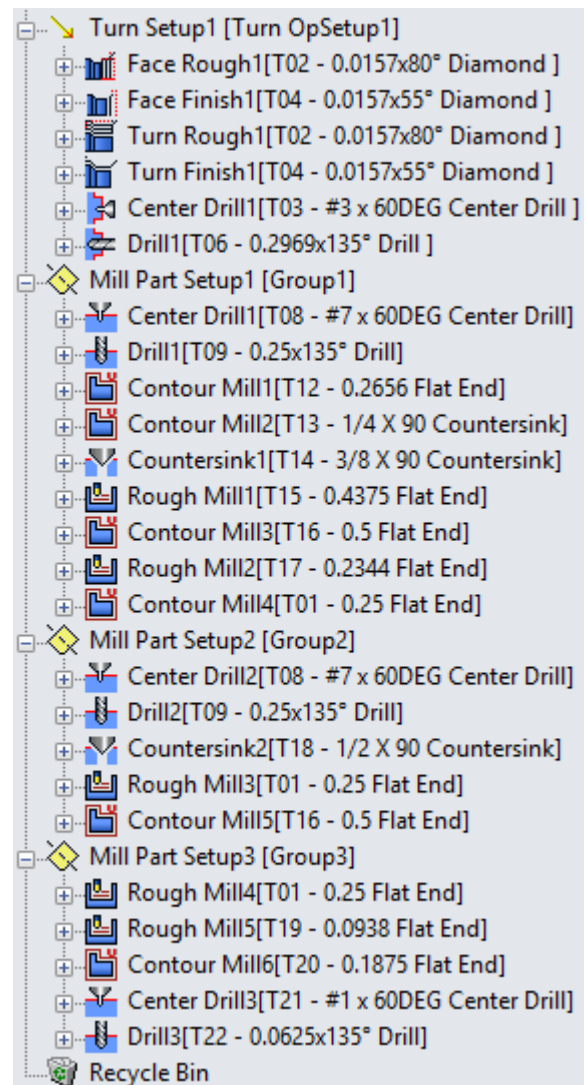
Operations tree after generating Toolpath

Learning Mill-Turn Basics

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

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

1. To set the proper view, right click *Turn Setup1*, select *Set View*.

2. Select *Turn ZX* from cascading context menu.

3. Double click *Turn Setup1* in the Operation tree.

   The Operation Setup Parameters dialog box is displayed.

4. Click the *Chuck/Fixture Location* tab.

5. In the Chuck Properties group box, change the Chuck defined from to *Setup* from dropdown list.

   This action will enable the *Edit* tab.

6. Click on the *Edit* tab.

   The Chuck Parameter: [Main Spindle] dialog box is displayed.

7. Select *4in_1Step_Chuck* for the configuration from Available Chucks dropdown list.

8. In the Jaw Parameter group box, set the Length and Width to **0.4in**.

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

10. In the Clamping diameter group box, click on *Select Entity for Clamping Diameter* field to set the focus.

11. Select the horizontal line for the chuck location as shown in the image.

12. In the Clamping diameter group box, click on *Select Entity for Z Clamping Location* field to set the focus.

13. Select the vertical line for the chuck location as shown in the image.

14. Click *OK* to apply the changes and close the dialog box.
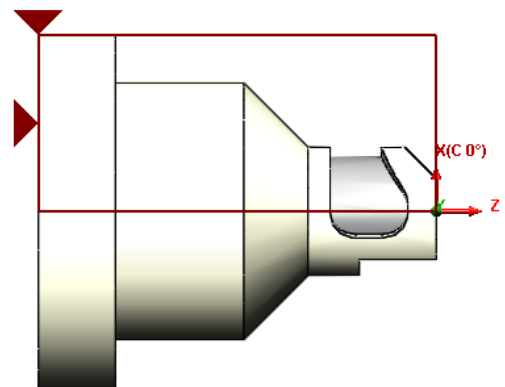
Location of Chuck on the part

Simulation Toolpath

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

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

   The Toolpath Simulation toolbar is displayed.
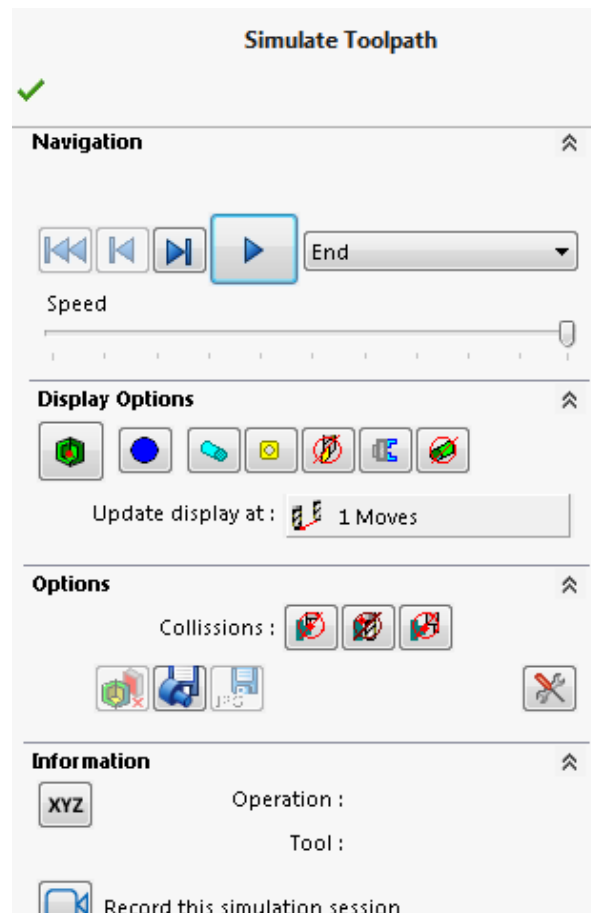
Simulate Toolpath Dialog Box

2. Set the End condition to *Next Setup.*

3. Click the *Run* button.

   The simulation runs through Turn Setup1, then pauses.

4. Click the *Run* button again.

5. When the simulation pauses, continue clicking the *Run* button to simulate each Mill Part Setup.

6. Click the ok button to cancel the simulation and return to the SOLIDWORKS display.
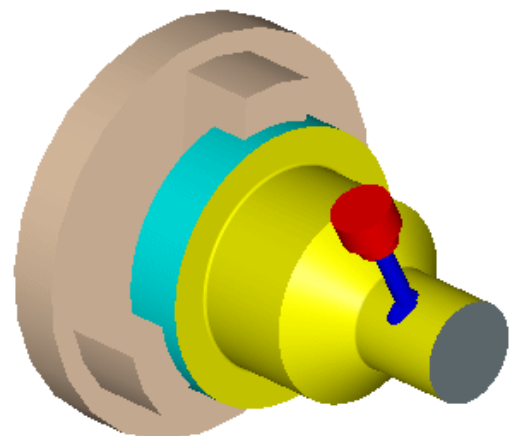
Simulation Toolpath

Step Through Toolpath

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

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

  OR

  Select *Step Thru Toolpath* command on the Command Manager.

- **Setup Level:** Right click on the *Mill Part Setup1* and select *Step Thru Toolpath* command in the context menu.

- **Operation level:** Right click on the desired operation in the Operation tree and select *Step Thru Toolpath* in the context menu.

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

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

OK button

Play button

Animation speed slider

Tool Position slider

Controls to set the options for Toolpath display

Information on the current toolpath move

Displays information on the current operation being back- plotted
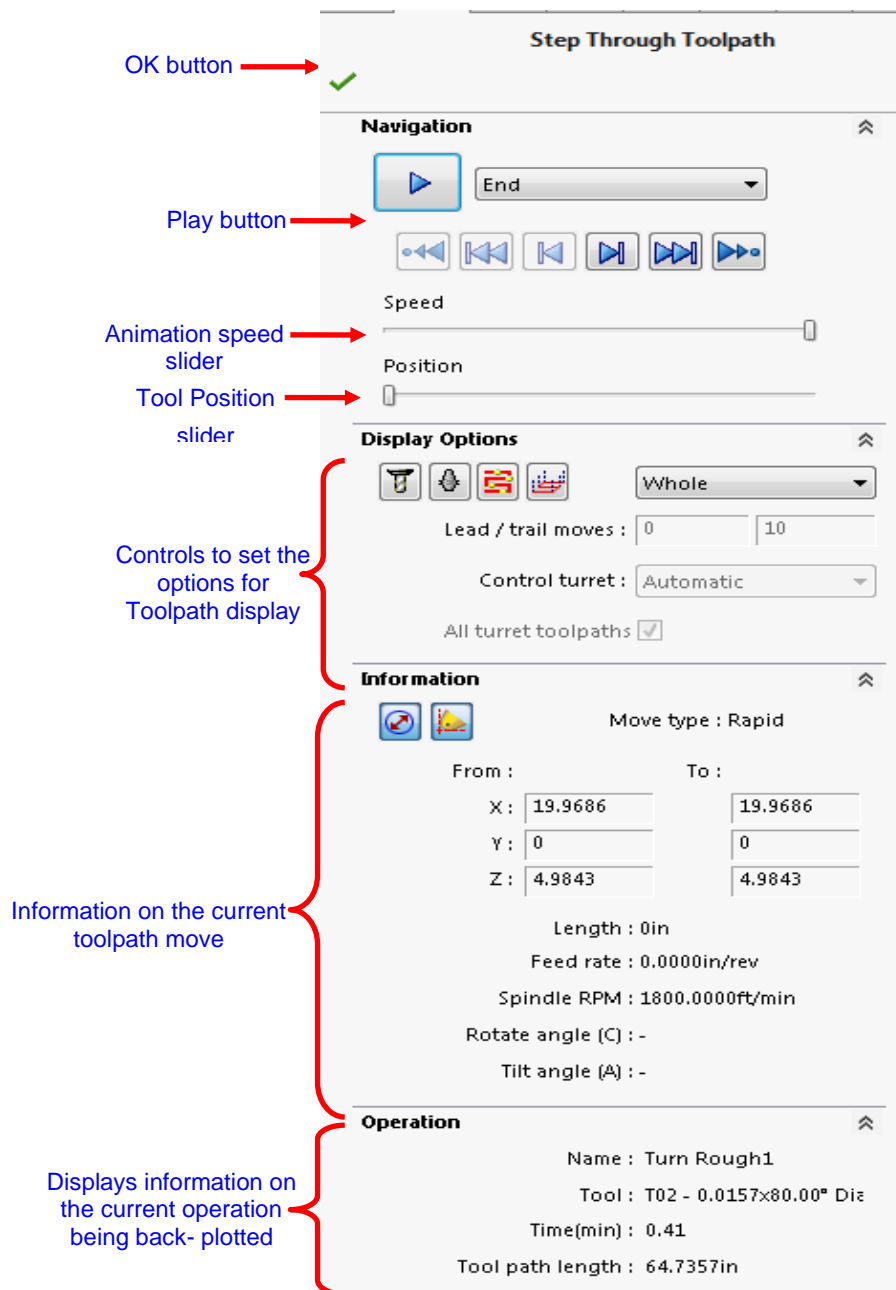
Step Through Toolpath dialog

3. In the *Play Tool Until* dropdown list, leave the end condition set to the default *End.*

4. Use the *Animation speed* slider to adjust the animation speed.

5. Click the *Play* button to continuously back-plot the tool till the end condition specified in the *Play Tool Until* field is reached.

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

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

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

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

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

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

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

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

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

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

9. If you are unsure about the use of any parameter within this dialog box, click on the *Help* button at the upper right corner. This action will display the context-based help for Step Through Toolpath command.

10. Click the *OK* button in the upper left corner to close the dialog box.

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

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

   OR

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

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

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

   > **Note:** **If the Post Process command is grayed out on the CAMWorks Command Manager or on any context menu, make sure that you have selected a post processor and generated the toolpaths.**
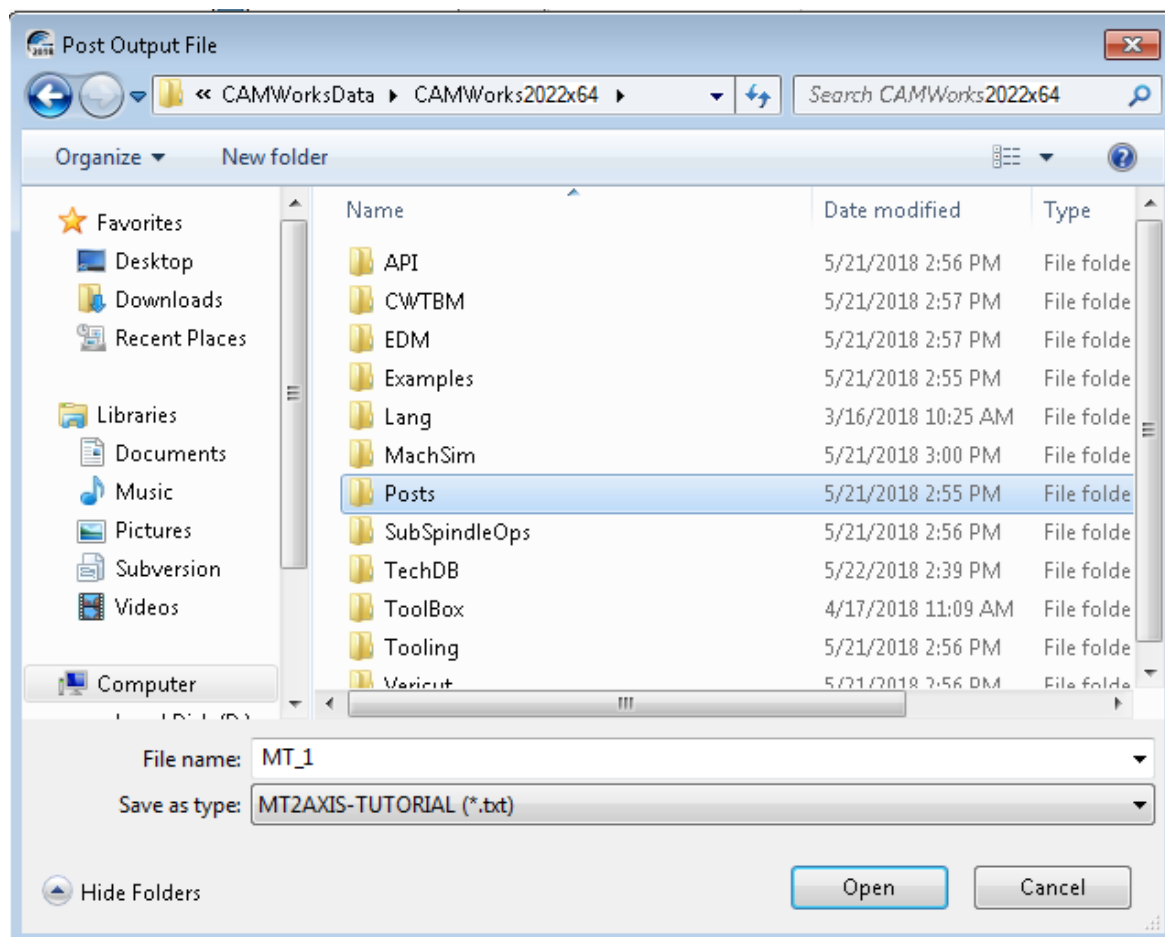
2. In the *Post Output File* dialog box, click the down arrow to the right of the *Save as type* box.

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

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

Post Output File dialog box

Note: **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* dialog box is displayed.

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 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 is displayed.

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

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

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

Mill-Turn 2

Topics covered in this tutorial:

- Inserting Multiple Wrapped Features
- Adjusting operations parameters and Machining using C-Axis

Open the part and Defining the Machine and Stock

1. Open the part file **MT_2.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn

   This part can be machined using C-axis.

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the Feature Tree.

   OR

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

   The Machine dialog box is displayed.

   - On the *Machine* tab, highlight *Mill-Turn Single Turret - inch* on the Machine tab and click *Select*.

   - Click on the *Tool Crib* tab and highlight *MT Tool Crib 2 Rear* and click Select to define the Active tool set.

   - Click on the *Post Processor* tab and make sure *MT2AXIS-TUTORIAL* is selected.

   - Click *OK* to close the *Machine* dialog box.

MT_2.SLDPRT

4. Double click *Stock Manager* in the tree.

   The Stock Manager dialog box is displayed.

5. Leave the stock is set as *Round Bar* and change the following parameters:

   - Outside Diameter = **5in**

   - Stock Length = **4.1in**

   - Back of Stock Absolute = **-4in**

   - Click *OK* to close the *Stock Manager* dialog box.
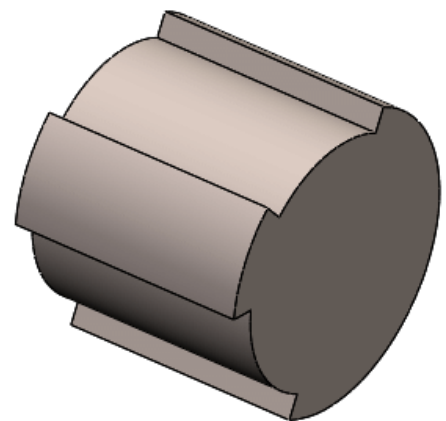
Extracting Machinable Feature

1. Click on the *CAMWorks Options* button on the CAMWorks Command Manager.

   OR

   Click on the *CAMWorks Options* option on the CAMWorks menu.

   The Options dialog box is displayed.

2. Click the *Mill features* tab.

---

3. In the *Extract Machinable features* group box, set the Method to *AFR*.

4. Click *OK* button to close the dialog box.

5. ⬚ Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

   Right click on the *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context menu.

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

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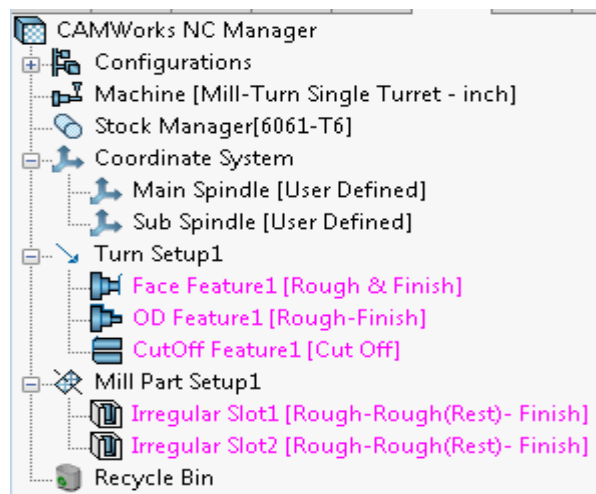
Features recognized by EMF

6. Right click *Mill Part Setup1* in the tree and select *Delete* on the context menu.

7. Click *Yes* to confirm the deletion.

8. Right click *Recycle Bin* in the tree, select *Empty* on the context menu.

9. Click *Yes* to confirm.

Inserting Multiple Wrapped Features

1. Right click *Turn Setup1* in the Feature tree and select *Mill Part Setup* on the context menu.

   The Mill Setup dialog box is dispalyed.

2. In the graphics area, double click or click on plus sign (⊞) next to the *MT_2* item to expand.

3. Pick the *Top* plane from the list of reference as shown in the image.

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

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

5. Right click *Mill Part Setup1* in the tree and select *Wrapped Feature* on the context menu.

   The Wrapped Feature dialog box is displayed.

6. Set the Feature Type to *Wrapped Slot*.
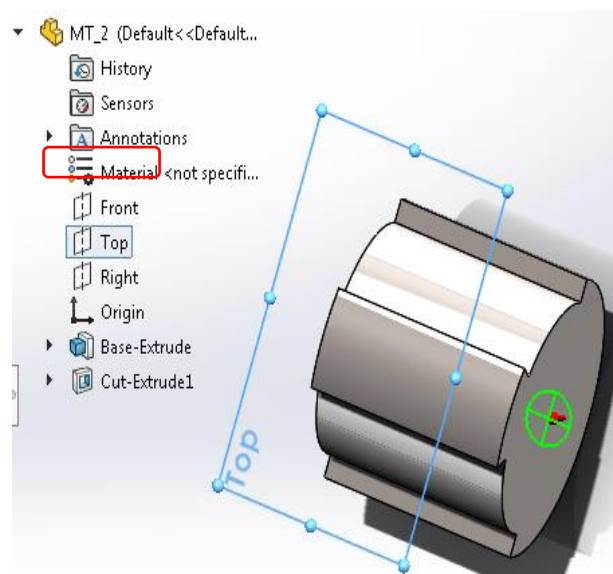
Pick the Top plane

The Profile tab is activated.

7. Set the Strategy to *Rough-Finish* from dropdown list.

8. On the part, pick the cylindrical faces of the both slots as shown in the image.

9. Click the *End Condition* tab.

10. Pick the *major OD* on the part *(highlighted in green)* as shown in the image.

Pick the Cylindrical faces
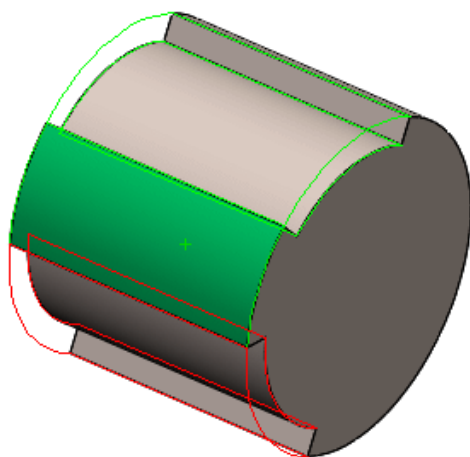
Pick the Major OD feature
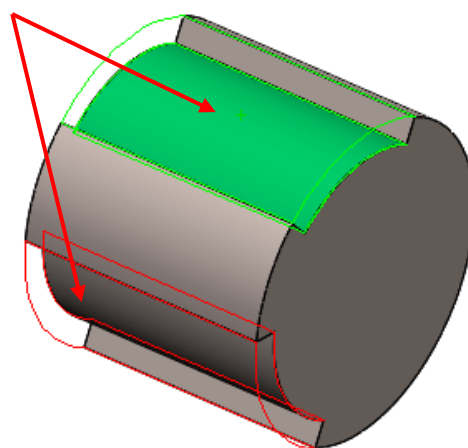
11. Click the *Edit Profile* tab.

    CAMWorks automatically calculates open air edges.

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

12. ✔ Click the *OK* button to insert the changes.

    The Wrapped Slot features are listed

    under Mill Part Setup1 in the tree.

    You want to machine the Cutoff

    feature after the other features, so

    you can create a Turn Setup and move the feature.

13. Right click *Mill Part Setup1* in the Feature

    tree and select *Turn Setup* on the context menu.

14. ✔ Click *OK* in the Turn Setup dialog box.

15. Turn Setup2 is listed in the CAMWorks Feature Tree.

16. Highlight *Cutoff Feature1* in the Turn Setup1

    and drag it to Turn Setup2.
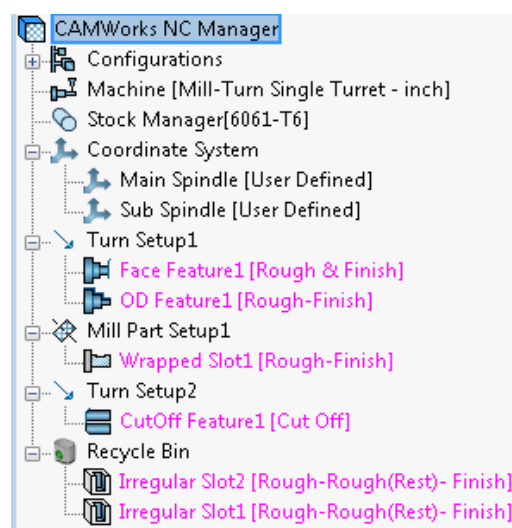
Drag the Cutoff Feature1 to Turn Setup 2

Adjusting Operation Parameters and Machining Using C-Axis

Generate Operation Plan and Toolpath

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

   OR

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

   The Operations are generated and listed in the Operation tree.

2. Double click *Rough Mill1* under Mill Part Setup1 in the Operation tree.

3. On the Roughing tab in the Pocketing group box,

   - Set pattern to *Zig* from dropdown list.

   - Check the *Automatic Angle* option.

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

4. Click *OK* to close the dialog box.
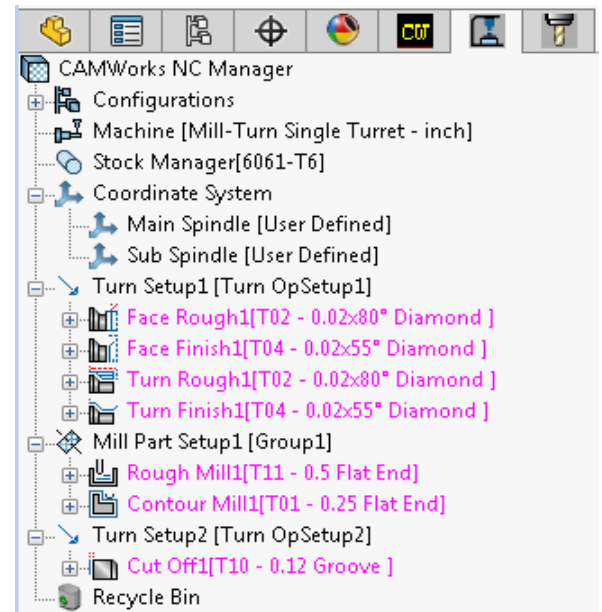
Generated operations in Operation tree

5. Double click *Rough Mill2* under Mill Part Setup1 in the Operation tree.

6. On the Roughing tab in the Pocketing group box,

   - Select pattern to *Zig* from dropdown list.

   - Check the *Automatic Angle* option.

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

8. Double click *Cut Off1* in the Feature tree.

9. Click on *Tool* tab.

10. On the Groove Insert page, set the Length to **2.95in**.

11. On the Holder page,

    - Set Orientation to *Down Left*

    - Protrusion to **2.65in**

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

13. ⬛ Click the *Generate Toolpath* button on the CAMWorks Command Manager.

    OR

    Right click on the *CAMWorks NC Manager* in the tree and select *Generate Toolpath* on the context menu

Simulation Toolpath

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

   OR

   Right click on the *CAMWorks NC Manager* in the tree and select *Simulate Toolpath* on the context menu.

2. Set the following display options:

   - 🔷 Stock: *Shaded Display*

   - 🔲 Tool: *Shaded Display*

   - 🧪 Tool Holder: *Shaded Display*

   - 🚫 Target part: *No Display*

3. ▶ Click the *Run* button on the toolbar to start the simulation.

4. Click *ok* ✔ *button* to close the Toolpath Simulation toolbar.
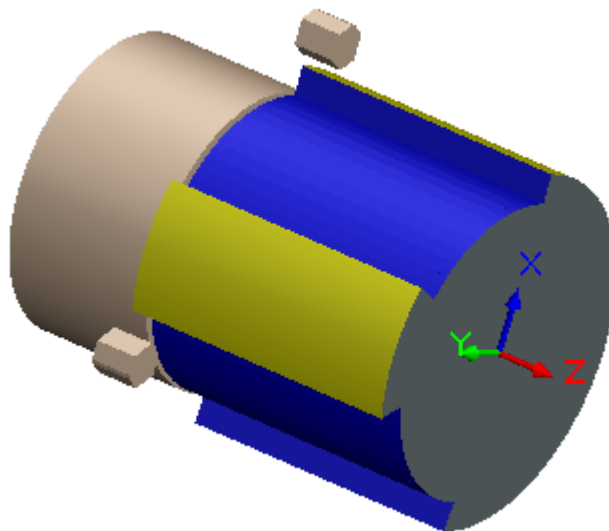
Simulation Toolpath

Mill-Turn 3

Topics covered in this tutorial:

- Creating a Wrapped Feature from a 2D sketch
- Unwrapping a solid model cylindrical face and creating a Wrapped feature
- Machining Using C-Axis

Open the Part and Defining the Machine and Stock

1. Open the part file **MT_3.SLDPRT** in the following folder.

   Drive:CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn

   This part can be machined using C-axis.

2. Click the *CAMWorks Feature Tree* tab.

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

   OR

   Double click *Machine* item in the Feature tree.

   The Machine dialog box is displayed.

   - On the *Machine* tab, highlight *Mill-Turn Single Turret - Inch* on the Machine tab and click *Select*.

   - Click on the *Tool Crib* tab and make sure *MT Tool Crib 2 Rear* is the Active tool set.

   - Make sure *Tool crib priority* options is not checked.

   - Click on the *Post Processor* tab and make sure *MT2AXIS-TUTORIAL* is selected.

   - Click *OK* to close the *Machine* dialog box.

4. Double click *Stock Manager* in the Feature tree.

   - In the Stock Manager dialog box,

   - Change the Stock Length to **10.25in**.

   - Click *OK* to close the dialog box and apply the changes.
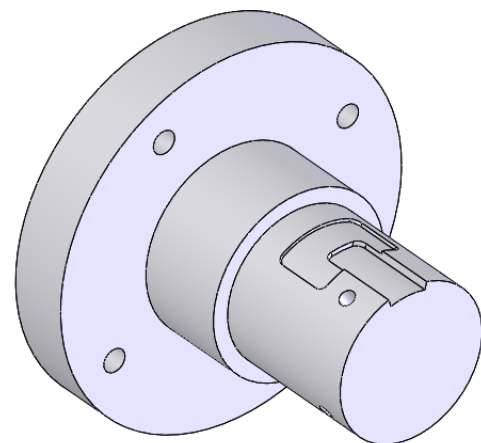
MT_3.SLDPRT

Extract Machinable Features

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

   Right click on the *CAMWorks NC Manager* in the Feature tree and select *Extract Machinable Features* on the context menu.

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

2. Right click *Cutoff Feature1* in Turn Setup1 and select *Delete* on the context menu.

3. Click *Yes* to confirm the deletion.

   The deleted feature is moved to Recycle Bin.
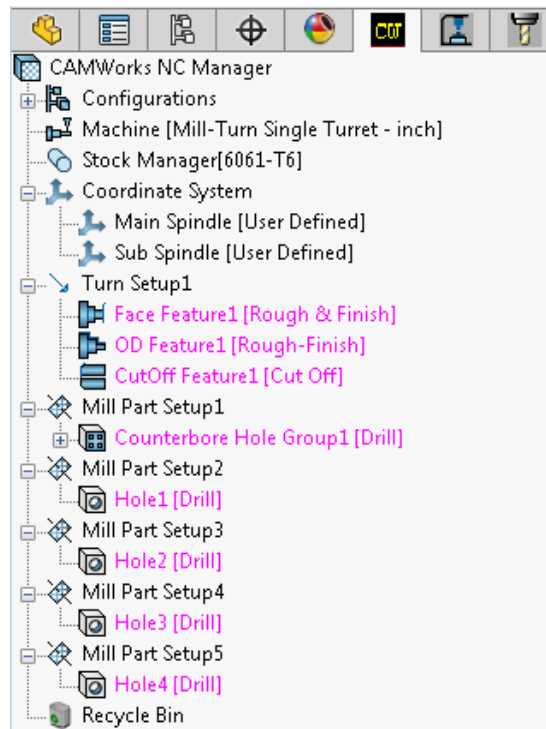
Features recognized by EMF

Inserting a Wrapped Feature Using a 2D Sketch

1. Right click *Mill Part Setup5* in the tree and select *Wrapped Feature* on the context menu.

   The Wrapped Feature dialog box is displayed.

2. In the Wrapped Feature dialog box, leave the Feature Type set to *Wrapped Pocket*.

3. Select the Strategy to *Rough-Finish* from dropdown list.

4. On the Profile tab in the Available valid sketches group box, pick *Wrapped 1* in the list.

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

5. On the Profile tab, click in the *Wrapping diameter entity* box to set the focus.

6. Pick the large OD on the part *(highlighted in green color)* as shown in the image.

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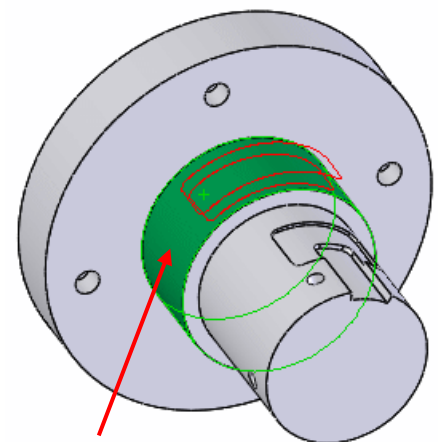
Pick the large OD feature

---

diameter or enter a value.

7. Click the *End Condition* tab.

8. ⬆ Click the *Reverse Direction* button and ↗D set the depth to **0.2in**.

9. ✔ Click the *OK* button to insert the changes.

The Wrapped Pocket1 feature is listed in the Feature tree under Mill Setup5.

Unwrapping a Solid Model Cylindrical Face and Creating a Wrapped Feature

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

1. Right click *Mill Part Setup5* in the Feature tree and select *Unwrap Faces/Edges to Sketch* on the context menu.

   The Unwarp Faces/Edges to Sketch dialog box is displayed.

2. Rotate the part and zoom in to pick the cylindrical face as shown in the image.
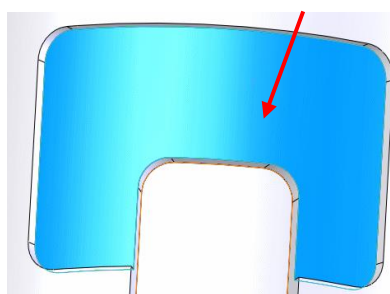
Pick the cylindrical face feature

3. In the Reference plane group box, pick *Top Plane* from the list.

4. ✔ Click *OK* to insert the changes.

   The Reference plane is the plane to unroll the sketch into. You should pick a plane that is parallel to and goes through the turn axis.

5. 📙 Click the *FeatureManager design tree* tab.

6. ✏ Right click *Sketch10* in the tree and select *Edit Sketch* on the context menu.

7. ↕ Right click *Sketch10* in the tree and change the Orientation to *Normal To*.

   OR
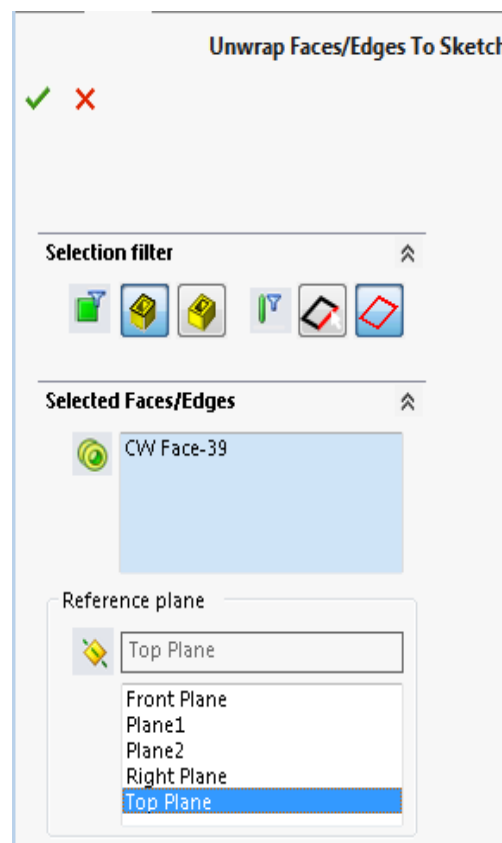
Unwrap Faces/Edge to Sketch dialog box

⬚ ▾ Click *View Orientation* button and select ⬍ *Normal to*.

8. Zoom in and window-pick the sketch as shown in **Figure 1**.

9. The Properties dialog box is displayed with listed selected entities.

10. Press the *Delete* key on the keyboard to delete the entities as shown in the **Figure 2**.

11. Hold the left button of mouse on upper point (shown in Figure2) and extend the line till lower point to connect the open entities as shown in the **Figure 3**.

    OR

    ✀ ▾ Click on arrow next to *Trim Entities* to display the dropdown list in the Sketch Ribbon bar and select ┬ *Extent Entities* from the list.

12. Click on upper point and extend the line to connect the open entities.

13. The sketch is now a fully closed loop that CAMWorks can wrap (**Figure 4**).
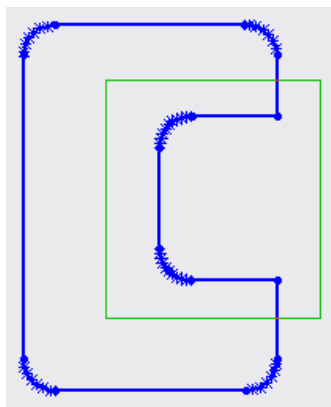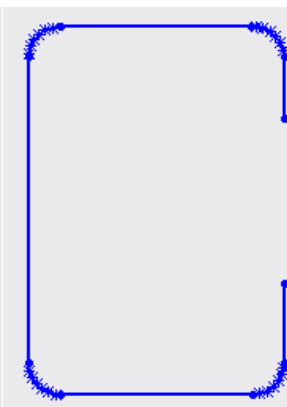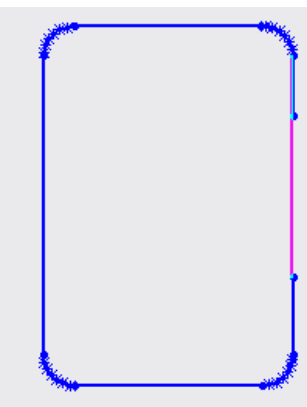
Figure 1	**Figure 2**	**Figure 3**	**Figure 4**
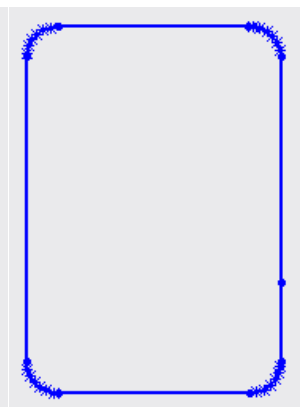
14. 🔍 Click on *Zoom to Area* button in the graphics area and select bottom right side corner of the sketch as shown in the image.

    The extended line is displayed. You can remove the line by trimming the line from the sketch.

15. ✀ ▾ Click on *Trim Entities* button on the *Sketch Ribbon bar* and select the point on extended line.

16. Click on the extended line which is present on the sketch. This action with change the color of extended line.

17. Then click on the spline curve as shown in the image below.

    The extended line is removed from the sketch.
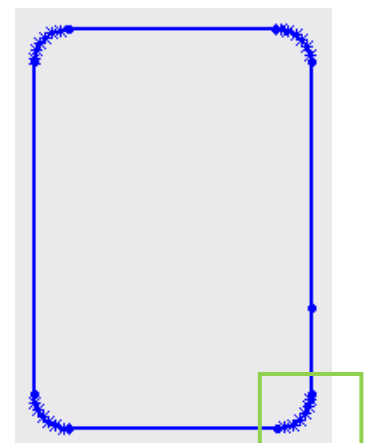
Zoom the bottom right side area

Select the extended line on zoom sketch
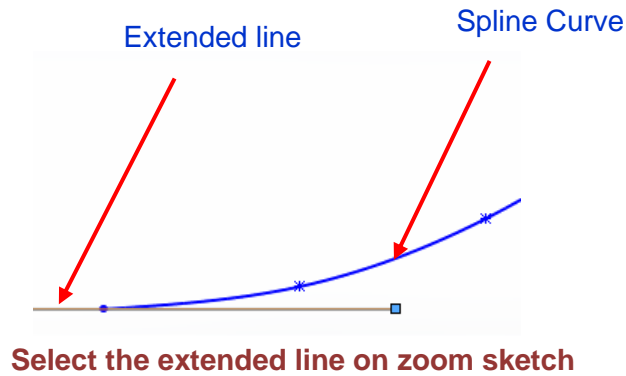
18. ✔ Click *Close* dialog box to close the Trim dialog box.

OR

Click on *Trim Entities* button to close the dialog box.

19. Click *Exit Sketch* button to exit Sketch mode.

20. Click the *CAMWorks Feature Tree* tab.

21. If CAMWorks displays a warning stating that CAMWorks data needs to be updated. Do you want to proceed with the rebuild.Click on Ok button to proceed with the rebuild.

22. In the graphics area, click *Zoom to Fit* to display the part.

23. Right click *Wrapped Pocket1* in the tree under Mill Part Setup5 and select *New Wrapped Feature* on the context menu.

24. Leave the Feature Type set to *Wrapped Pocket.*

25. Select the Strategy to *Rough-Finish* from dropdown list.

26. Pick *Sketch10* in the Available valid sketches list.

Pick the cylindrical bottom face

27. Click in the *Wrapping diameter entity* box to set the focus.

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

28. Rotate the part and pick the bottom face of the pocket as shown in the image.

29. Click the *End Condition* tab.

30. Pick the smaller OD face feature on the part as shown in the image.

31. ✔ Click *OK* to insert the changes.

32. Right click *Wrapped Pocket2* in the Feature tree under

Pick the smaller OD face
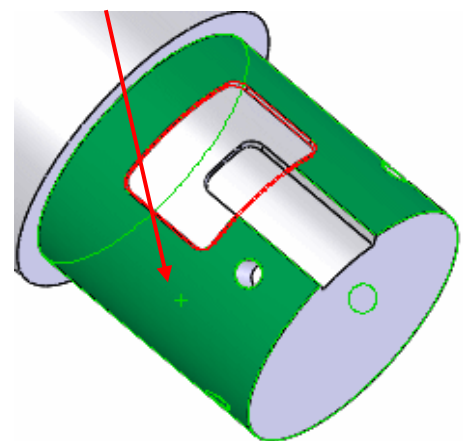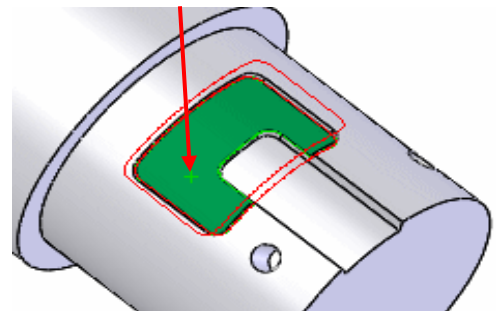
Mill Part Setup5 and select *New Wrapped Feature* on the context menu.

33. Change the Feature Type to *Wrapped Slot*.

34. Select the Strategy to *Rough-Finish* from dropdown list.

35. Pick the slot feature on the face as shown in the image.
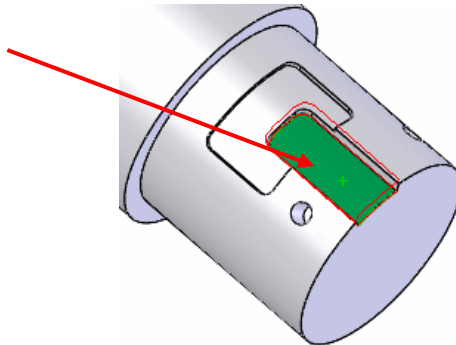
Pick the slot feature

36. Click the *End Condition* tab.

37. Pick the smaller OD on the part.

38. ✅ Click *OK* to insert the changes.

39. In the Feature tree, make sure Wrapped Slot1 is placed below Wrapped Pocket5 to change the machining order.
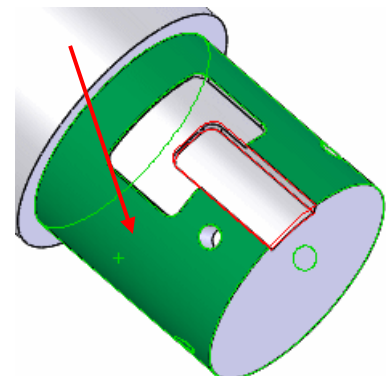
Pick the smaller OD face
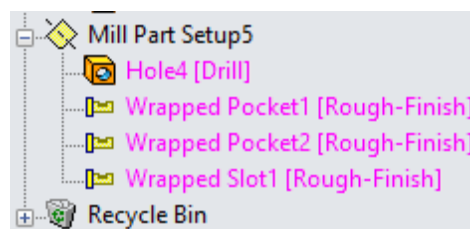
Rearranged order of features

Adjusting Operation Parameters and Machining Using C-Axis

Generate Operation Plan and Toolpath

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

    OR

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

2. 🔧 Click the *Generate Toolpath* button on the CAMWorks Command Manager.

    OR

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

---

3. Click *RoughMill3 (Wrapped Slot1)* in Mill Part Setup5 and notice the angle of the toolpath.

   You can edit the operation and have CAMWorks automatically pick the most efficient angle.

4. Double click *Rough Mill3* in the Operation tree.

5. On the Roughing tab, set the Pattern to *Zigzag* and check the *Automatic angle* option.

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

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

7. Right click *Rough Mill3* and select *Generate Toolpath* on the context menu.

8. Notice the difference in the toolpath.
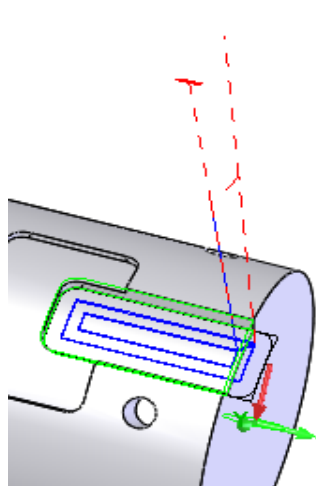
Angle of Toolpath before changing the Pattern   **Angle of Toolpath after changing the Pattern**
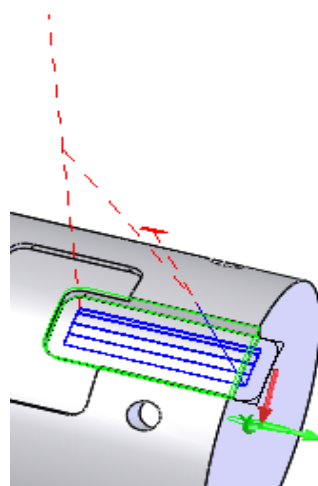
Simulation Toolpath

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

   OR

   Right click on the *CAMWorks NC Manager* in the tree and select *Simulate Toolpath* on the context menu.

2. Set the following display options:

   - [icon] Stock: *Shaded Display*

   - [icon] Tool: *Shaded Display*

   - [icon] Tool Holder: *Shaded Display*

   - [icon] Target part: *No Display*

3. [icon] Click the *Run* button on the toolbar to start the simulation.

4. Click *OK* [icon] button to close the toolpath simulation toolbar.
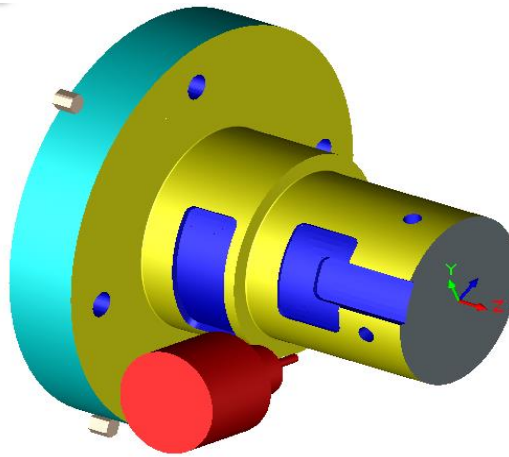
Simulation Toolpath

2. LEARNING MORE ABOUT MILL-TURN

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

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

The tutorial parts are installed when you install CAMWorks and are in the *Examples\ Tutorial_Parts\Mill –Turn* folder on your computer. This folder is inside the CAMWorks data folder (Drive:*CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn*).

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

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

Mill-Turn 4

Topics covered in this tutorial:

- Interactively inserting 2.5 Axis and Multi Surface features
- Inserting Contain and Avoid areas for 3 Axis operations
- Adjusting operation parameters and machining using Y-Axis

Open the Part and Defining the Machine and Stock

1. Open the part file **MT_4.SLDPRT** in the following folder.

    Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn
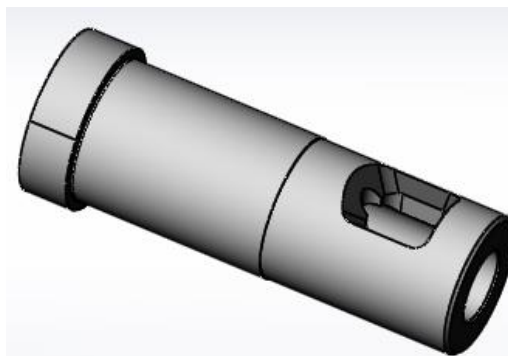
    MT_4.SLDPRT

    This part can be machined using Y-axis.

2. Click the *CAMWorks Feature Tree* tab.

3. Double click *Machine* item in the Feature tree.

    The Machine dialog box is displayed.

    - On the *Machine* tab, highlight *Mill-Turn Single Turret - inch* and click on the *Select* button.

    - Click on the *Tool Crib* tab and make sure *MT Tool Crib 2 (Rear)* is the Active tool set.

    - Click on the *Post Processor* tab and make sure *MT2AXIS-TUTORIAL* is selected.

    - Click *OK* to close the *Machine* dialog box.

4. Right click *Stock Manager* in the Feature tree and select *Edit Definition* on the context menu.

    The Stock Manager dialog box is displayed.

    - In the Stock Manager dialog box, change the Stock Length to **16in.**

    - Click *OK* to close the dialog box.

Extract Machinable Features

1. [icon] Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    OR

    Right click on the *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context menu.

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

2. Right click *Cutoff Feature1* in Turn Setup1 and select *Delete* on the context menu.

3. Click *Yes* to confirm the deletion.
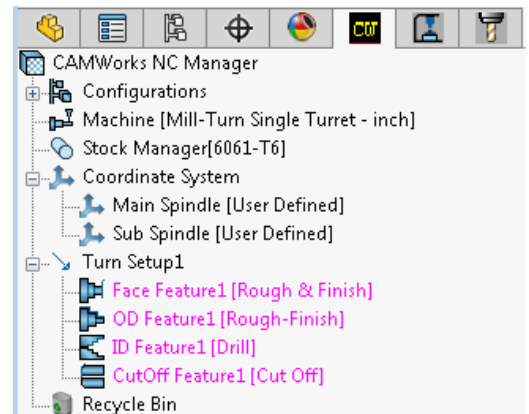
Features recognized by EFM

Inserting 2.5 Axis and Multi Surface Machinable Features

1. Right click *Turn Setup1* in the tree and select *Mill Part Setup* on the context menu.

    The Mill Setup dialog box is displayed.

2. In the graphics area, double click on MT_4 item to expand, if necessary.

    OR

    Click on the (⊞) plus sign next to *MT_4* to expand the tree in the graphics area.
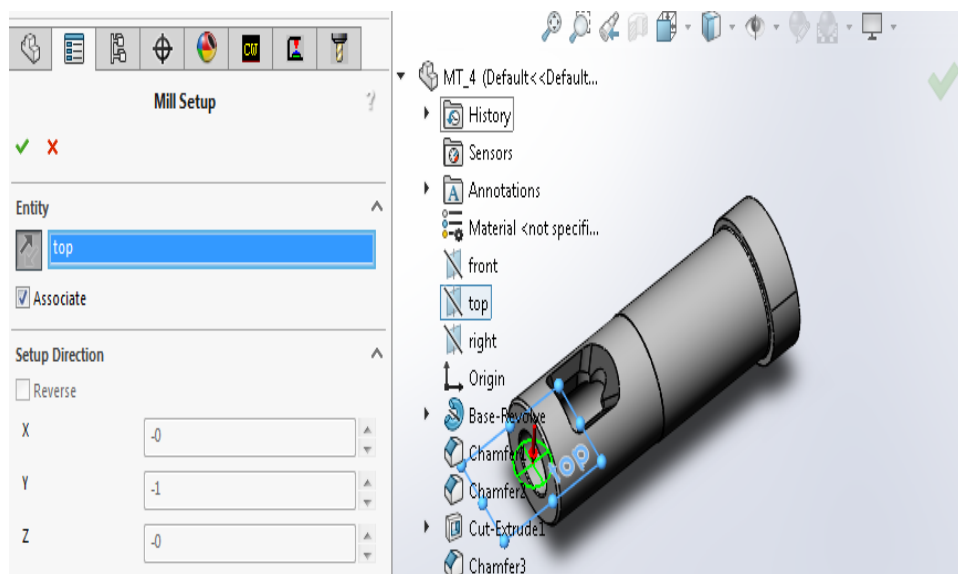
Select the Top plane in graphics area

3. Pick the *Top plane* from references as shown in the image.

4. Click *the Reverse Selected Entity* button and make sure the machining direction is from top (as shown in above image).

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

   The Mill Part Setup2 is listed in the Feature tree.

6. Right click *Mill Part Setup2* in the tree and select *New 2.5 Axis Feature* on the context menu.

7. In the 2.5 Axis Feature: Select Entities dialog box, set the Feature Type to *Pocket*.

8. Pick *2x Pocket* in the list of sketches.

9. Click *End Condition* button.

10. Leave the End condition type set to *Blind*.

11. Change the Depth to **2in.**

12. Change the Strategy to *Coarse* to generate only a Roughing operation.

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

   The Irregular Pocket1 is listed in the Feature tree under Mill Part Setup2.

14. Right click *Irregular Pocket1* in the tree and select *New Multi Surface Feature* on the context menu.

   The New Multi Surface Feature dialog box is displayed.

15. Set the Strategy to *Area Clearance, Constant Stepover*.

16. Make sure the *Select Face* button is selected.

17. Pick all the dark gray faces around the pocket as shown in the image.

18. Click *OK* to insert the changes.

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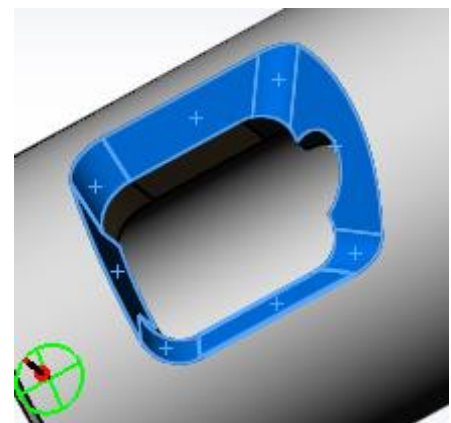
   OR

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

Pick all the dark gray faces

Inserting Contain and Avoid Areas for 3 Axis Operations

1. In the CAMWorks Operation tree, right click *Area Clearance1* and select *New Contain Area* on the context menu.

   The Contain Area dialog box is displayed.

2. Make sure the Selection mode is set to Single Face and Single Edge (*Default*).

3. Pick all the top edges of the pocket.
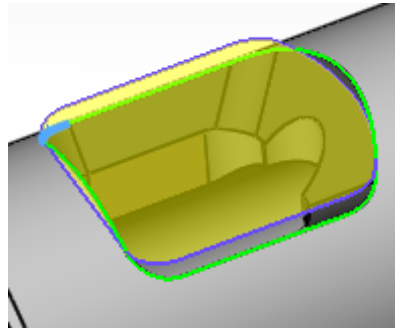
**Silhouette displays after
picking all the top edges**

A silhouette displays on the pocket when all the edges are selected. All the selected edges are listed in the Selected Contour list box.

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

5. Right click *Area Clearance1* in the tree and select *New Avoid Area* on the context menu.

6. Pick *2x Pocket* in the list of Allowed sketches.

   This action will display the selected sketch (2x Pocket) in the Selected Contour list box.

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

7. ↑ Set the Tool Condition to *Tool On Curve*.

8. ✔ Click *OK* to insert the changes and close the dialog box.

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

1. Click the (➕) plus sign next to Area Clearance1 in the Operation tree to expand the item. The Contain and Avoid areas are listed in the tree.

2. Click *Contain Area1* to highlight the item, hold down the Ctrl key and drag the item on top of Constant Stepover1, then release the mouse button.

   The cursor changes to an arrow to indicate copy mode.

3. Click the (➕) plus sign next to Constant Stepover1. Contain Area2 is listed in the tree.

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

Adjusting Operation Parameters and Machining Using Y-Axis

1. Double click *Turn Finish1* under Turn Setup1 in the CAMWorks Operation tree.

   The Operation Parameters dialog box is displayed.

2. On the Turn Finish tab, make sure the *Undercut* option in the Profile Parameters group box is checked.

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

This option defines whether the toolpath can create an undercut up to the back angle of the tool. If checked, the toolpath follows the entire toolpath observing the back angle.

4. Double click *Area Clearance1* in the tree under Mill Part Setup2.

   The Operation Parameters dialog box is displayed.

5. On the Area Clearance tab in the Limits group box,

   - Leave the First cut from option set to *Top of Stock*

   - Change the Offset to **-0.25in**.

   - Click *OK* to apply the changes.

6. Double click *Constant Stepover1* in the tree.

   The Operation Parameters dialog box is displayed.

7. On the Constant Stepover tab,

   - In the Surface Finish group box, set the Mach. Deviation to **0.008in.**

   - In the Direction control group box, set the option to *Zigzag*.

   - In the Limits group box, set the Last cut at option to *User Defined.*

   - Click the *Pick Point* button.

     The Define Point dialog box is displayed.

   - Zoom on the pocket, mouse over on the point and pick the vertex point as shown in the image on right.

   - In the Offset group box, set the Z Offset to **-0.1in.**

     Since the bottom edge of the feature is curved, applying the offset in Z will generate the toolpath to cut all the material.

   - Click *OK* to close the Define Point dialog box.
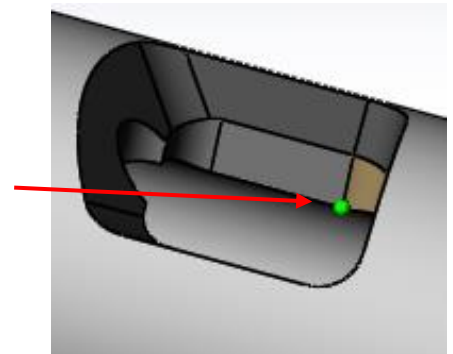
Select the Vertex point
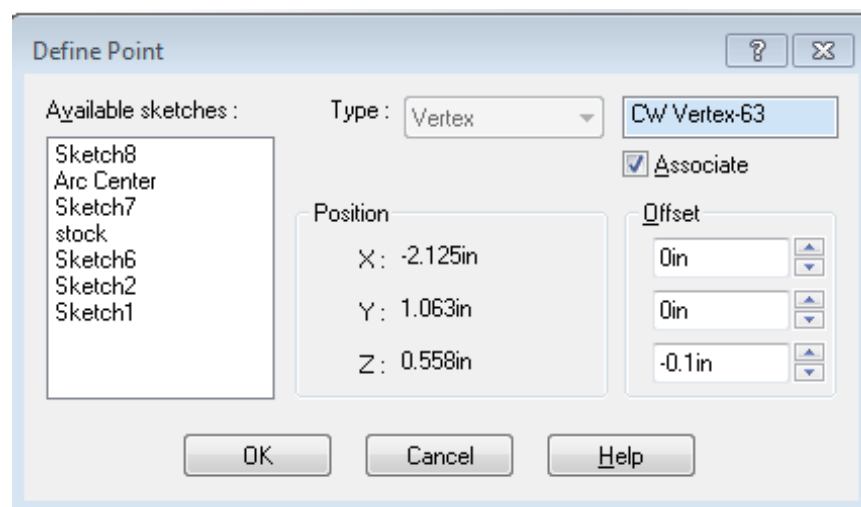
Define Point dialog box

8. Click *OK* to close the Operation Parameters dialog box and apply the changes.

---

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

OR

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

Simulate Toolpath

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

2. Set the following display options:

   - Stock: *Shaded Display*

   - Tool: *Shaded Display*

   - Tool Holder: *No Display*

   - Target: *No Display*

3. Click the *Run* button on the toolbar to start the simulation.

4. During the simulation, you can click the *Show Animation* button to adjust the speed of the simulation. Increasing the number of moves speeds up the simulation.
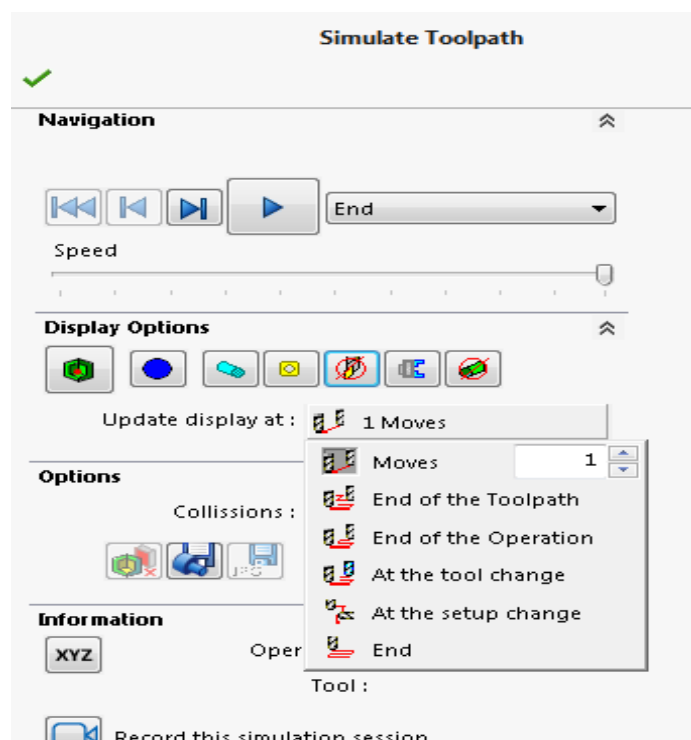
Simulate Toolpath Dialog Box

Mill-Turn 5

Topics covered in this tutorial:

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

Adjusting Entity Locations to Machine a Wrapped Engrave Feature

1. Open the part file **MT_5.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill-Turn

   This part can be machined using C-axis.

2. Click *Tools* on the SOLIDWORKS/CAMWorks Solids menu bar and select *Measure* on the context menu.

3. Pick the vertical line that represents the edge of the **1.5in** diameter surface (*highlighted in green*).
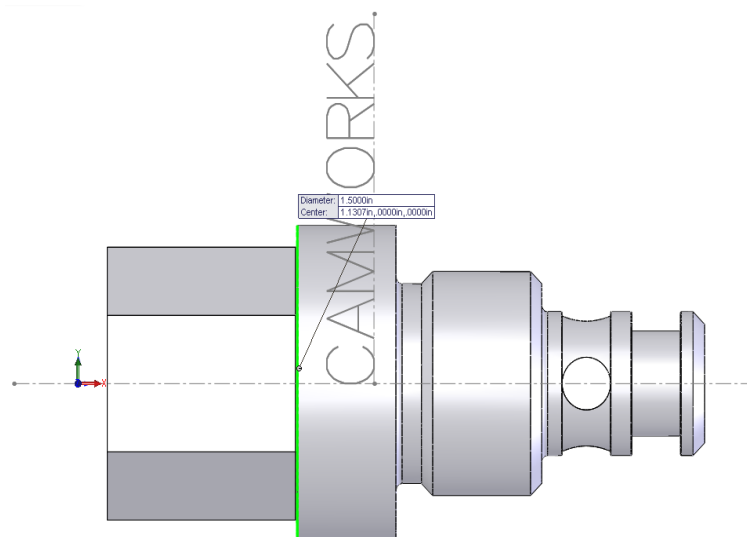
MT_5.SLDPRT part with highlighted vertical line

4. Click *Expand* button on the Measure – MT_5.SLDPRT to expand the dialog box and notice that the Length (circumference) is **4.7124in**.

5. Click *Close* to exit the Measure utility.
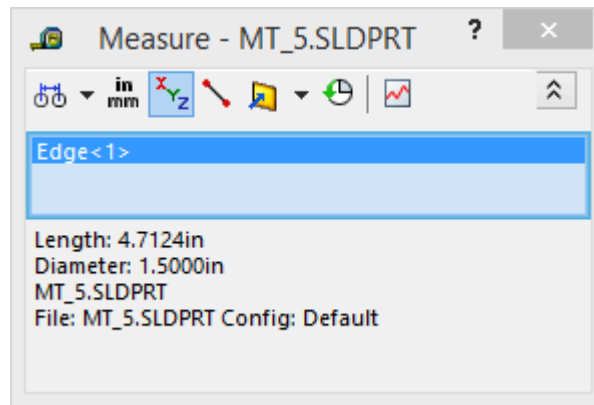
Measure dialog box

6. In the FeatureManager design tree, right click *TEXT-SKETCH* and select *Edit Sketch* on the context menu.

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

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

    To calculate this:

    4.7124 (circumference distance) divided by 360degrees = 0.01309 per one degree of rotation
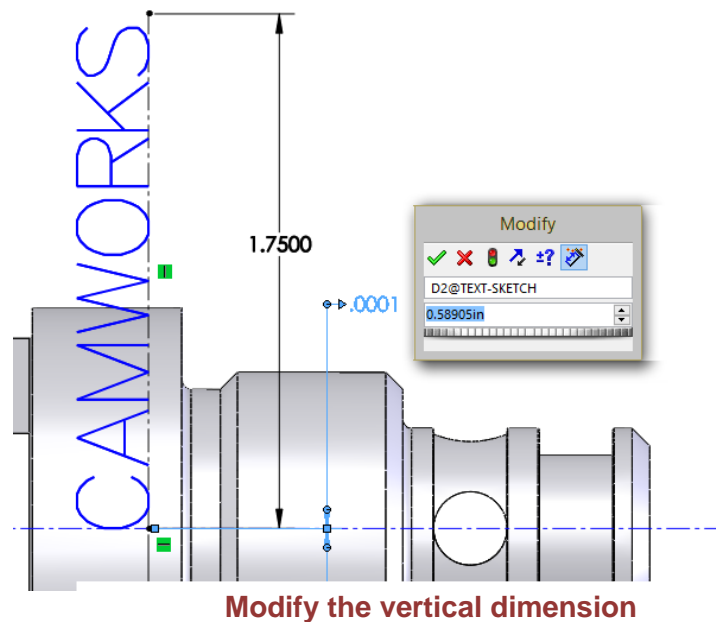
    45degrees x 0.01309 = 0.58905 distance

    Therefore, you need to move the text a distance of **0.58905in** up or down vertically. This will shift the text and start angle to start machining at 45degrees off the center of the rotational axis.

7. Select and double click on the **0.0001in** vertical dimension on the part, and change it to **0.58905in**.

8. Click *the Save the current value and exit the dialog* button to close the dialog.

9. Click *Exit Sketch* to exit the Edit Sketch mode.

Modify the vertical dimension

Inserting a Wrapped Engrave Feature to Machine Text

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

2. Double click *Machine* item in the Feature tree.

   OR

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

   - The Machine dialog box is displayed.

   - On the Machine tab, highlight *Mill-Turn Single Turret - inch* on the Machine tab and click Select.

   - Click on the Tool Crib tab and make sure *MT Tool Crib 2 (Rear)* is the Active tool set.

   - Make sure *Tool crib priority* option is unchecked.

   - Click on the Post Processor tab and make sure *MT2AXIS-TUTORIAL* is selected.

   - Click *OK* to close the Machine dialog box.

3. Double click *Stock Manager* in the Feature tree.

   - In the Stock Manager dialog box, change the Stock Length to **3.1in.**

   - Click *OK* to close the dialog box.

Inserting a Wrapped Engrave Feature to Machine Text

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

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

2. Right click *Cutoff Feature1* in Turn Setup and select *Delete* on the context menu.

3. Click *Yes* to confirm the deletion.

4. Click on the *Mill Part Setup 1*, then press the *Ctrl* key and select remaining *Mill Part Setup*.

5. Right click on the any *Mill Part Setup* and select *Delete* on the context menu.

6. Click *Yes* to confirm the deletion.

   You are not machining the side holes in this exercise, so you can delete the setups and features.

7. Right click the *Recycle Bin* and select *Empty* on the context menu.

8. Click *Yes* to confirm.

9. Right click *Turn Setup1* in the Feature tree and select *New Mill Part Setup* on the context menu.

10. In the graphics area, click on the (⊞) plus sign next to MT_5 to expand, if necessary.

11. Pick the *Front plane* from the references as shown in the image.

12. Click the *Reverse Selected Entity* button in the Mill Setup dialog box.

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

14. Right click the *Mill Part Setup6* in the Feature tree and select *New Wrapped Feature* on the context menu.

   The New Wrapped feature dialog box is displayed.

15. In the Perimeter group box, select *Wrapped Engrave* for the Feature Type.

16. Set the Strategy to *Fine*.

    This Strategy has been set up in the TechDB to generate one Contour Mill operation.

17. Pick *TEXT-SKETCH* in the Available valid sketches list.

    This list shows the 2D sketches that lie on a sketch plane that is parallel to and passes through the turn axis. These sketches provide an alternative method of defining a wrapped feature. In most cases, wrapped features will be created from edges and surfaces on
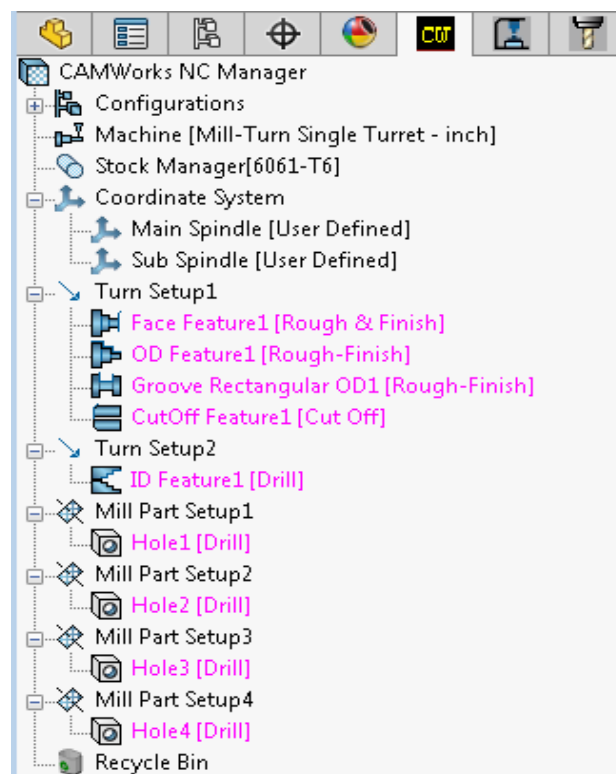
Feature recognized by EMF
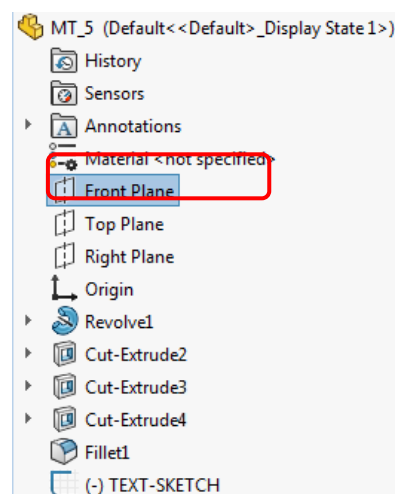
Pick the Front Plane

cylindrical faces. However, if the model does not have cylindrical edges or faces that can be used to create the feature, a 2D sketch can be used to create a feature.

18. Rotate the part to see how the feature is trying to wrap around the surface.

19. Click in the *Wrapping Diameter entity* box and select the **1.5in** diameter face (highlighted in green).

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

20. Click the *End Condition* tab.

21. Leave the Type set to *Blind*.

22. Change the Depth to **0.005in.**

23. Click the *Reverse Direction* button.

24. Click *OK* to insert the changes.

Pick the wrapped feature

25. Rotate the part to see the Wrapped Feature on the outside diameter.

26. Click the *CAMWorks FeatureManager design tree* tab.

27. Right click *TEXT-SKETCH* and select *Hide* button on the context menu.

    You no longer need to view the sketch.
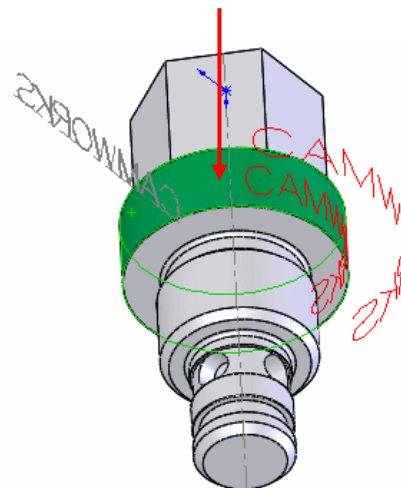
Inserting a Plane to Control the Index or Rotational Angle for Mill Setups

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

1. Highlight the *Front Plane* in the FeatureManager design tree.

2. Click *View* menu on the SOLIDWORKS/CAMWorks Solids menu bar and select *Temporary Axes* to turn the same on.

3. Rotate the part.

4. Select *Front Plane* in the FeatureManager design tree to highlight the plane in the graphics area.

5. Click *Insert* menu on the SOLIDWORKS/CAMWorks Solids menu bar and select *Plane* on the cascading menu under *Reference Geometry*.
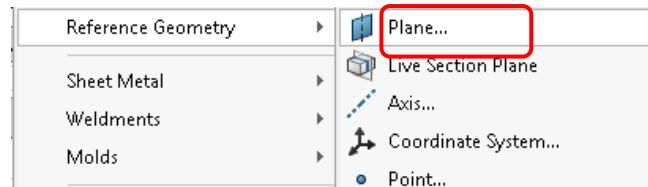
Select Plane on Reference Geometry cascading menu

The Plane dialog box is displayed.

6. Select the *Front Plane* as First reference.

7. Pick the *Temporary Axes* running through the center of the part to select it as the Second Reference as shown in the image.
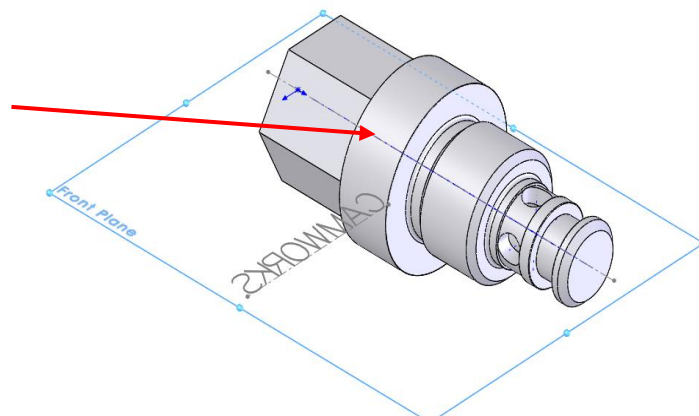
Pick the Temporary Axes

8. Ensure that the Angle is set to **45degrees.**

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

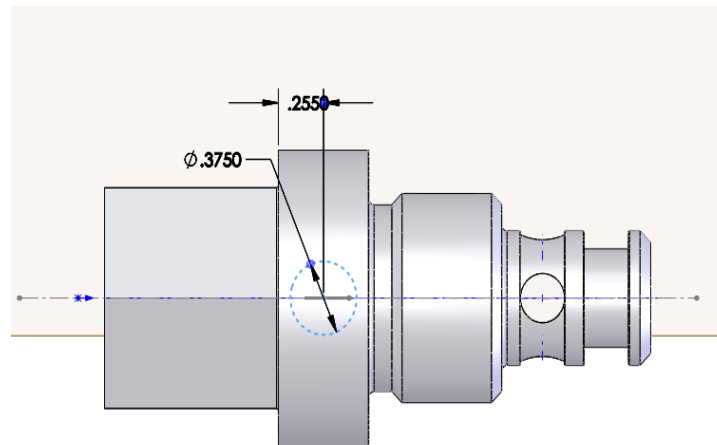The Plane 1 is listed in the CAMWorks FeatureManager design tree.

10. Highlight the Plane1 in the CAMWorks FeatureManager design tree.

11. Click on *View Orientation* in the graphics area and select Normal to.

12. Click the *Sketch* button under the Sketch Ribbon bar.

13. Insert a **0.375in** circle close to the center.

14. Click *Close* to exit Edit Sketch mode.

Insert a Circle

Inserting a 2.5 Axis Wrapped Engrave Feature

1. Click the *CAMWorks Feature Tree* tab.

2. Right click the *Mill Part Setup6* and select *New Mill Part Setup* on the context menu.

   The Mill Setup dialog box is displayed.

3. In the graphics area, click on the (⊞) plus sigh next to the *MT_5 item* to expand and *Plane1* from the references.

4. Click the *Reverse Selected Entity* button in the Mill Setup dialog box.

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

6. Right click the new inserted *Mill Part Setup7* in the Feature tree and select *New Wrapped Feature* on the context menu.

   The New Wrapped Feature dialog box is displayed.

7. Set the Feature Type to *Wrapped Engrave* and set the Strategy to *Fine*.

8. Rotate the part to get the clear view of the circle.

9. Pick the **0.375in** circle geometry on the part as shown in the image.

10. Click in the *Wrapping diameter entity* box to set the focus and pick the **1.5in** diameter face on the part.

11. Click the *End Condition* tab.

12. Leave the Type set to *Blind*.

13. Change the Depth to **0.005in.**

14. Click the *Reverse Direction* button.

15. Click *OK* to insert the changes.
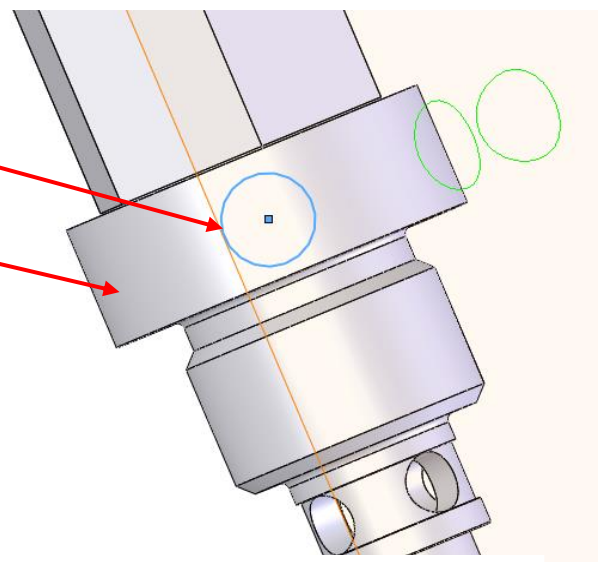
Pick the Circle and Face on the part

---

16. Click the *CAMWorks FeatureManager design tree* tab.

17. Select *Front Plane*, *Plane 1* and *Sketch6* and right click on the item respectively and select *Hide* button on the context menu.

    You no longer need to display these items.

18. Click *View* menu on the SOLIDWORKS/CAMWorks Solids menu bar and select *Temporary Axis* to turn it off.
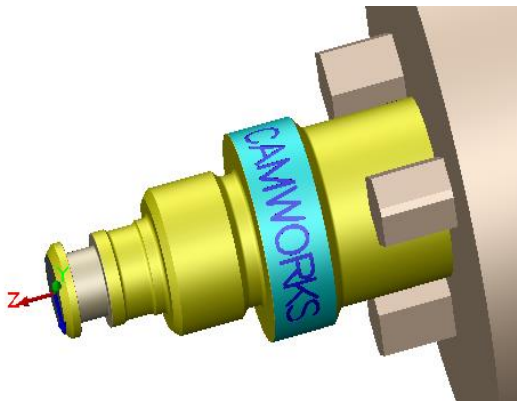
Machining Using C-Axis

1. Click the *CAMWorks Feature Tree* tab.

2. CAMWorks displays a warning stating that CAMWorks data needs to be updated. Do you want to proceed with the rebuild.Click on Ok button to proceed with the rebuild.

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

    OR

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

4. Double click *Groove Rough1* operation of the Turn Setup1 in the tree.

    The Operation Parameters dialog box is displayed.

5. In the Holder page of Tool tab, make sure the *Holder Edge* is set to *Side.*

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

7. Double click *Groove Finish1* operation of the Turn Setup1 in the tree.

8. In the Holder page of Tool tab, make sure the *Holder Edge* is set to *Side*.

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

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

    OR

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

Simulate Toolpath

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

    OR

    Right click on *CAMWorks NC Manager* in the Feature tree and select *Simulate Toolpath* on the context menu.

2. Set the following display options:

- Stock: *Shaded Display*

- Tool: *Shaded Display*

- Tool Holder: *No Display*

- Target part: *No Display*

3. Click the *Run* button on the toolbar to start the simulation.

4. Click *OK* *button* to exit the Toolpath simulation toolbar.

Simulation Toolpath