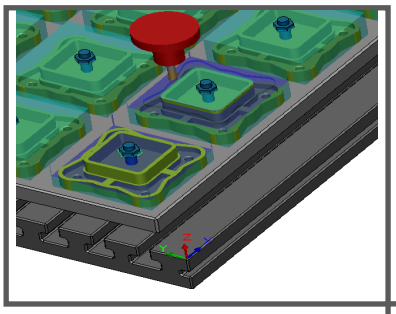
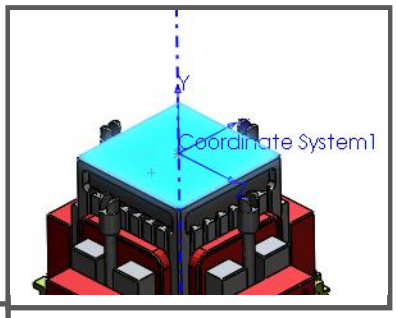
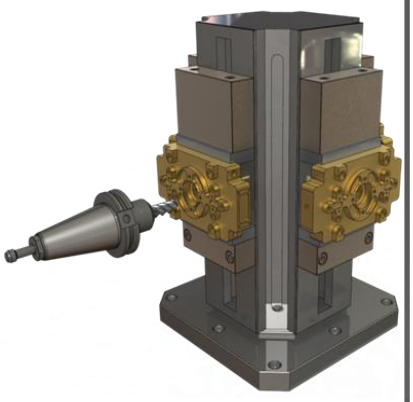
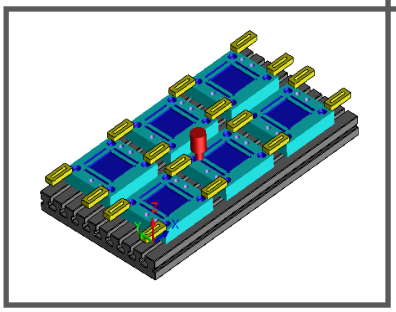
CAMWorks 2022

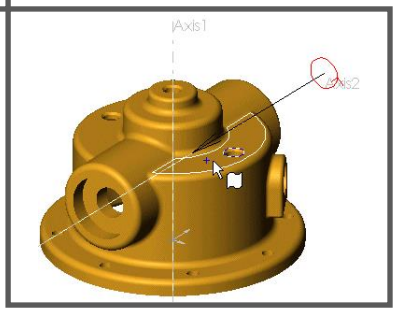
Mill Assemblies Tutorial











Disclaimer

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

Trademarks

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

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

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

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

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

Geometric
An HCL Technologies Company

TABLE OF CONTENTS

1. LEARNING ASSEMBLY MODE

This chapter provides an opportunity to learn to generate Mill toolpaths and NC code in Assembly mode for production machining. The information applies to running CAMWorks in SOLIDWORKS and CAMWorks Solids.

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

The exercise parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\Assemblies* folder. This folder is inside the CAMWorks data folder.

Typical location: *Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Assemblies*.

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.

We highly recommend that you go through the concepts explained in the **Mill Tutorial PDF** document before commencing with the tutorials explained in this document.

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 (part, clamps, vises, fixtures) and created the assembly document (.sldasm) in SOLIDWORKS/CAMWorks Solids.
2. Click on the CAMWorks Feature tree.
3. Define the Machine and Fixture Coordinate System (defines the default G-code origin, defines the XYZ machining directions and acts as a reference point, if subroutines are used).
4. Select the parts to be machined.
5. Define the Stock (separate or common).
6. Extract Machinable Features and interactively insert the features at Part Setup Level.
7. Generate the operation plan and adjust operations parameters.
8. Define G-code program zero location (Pat Setup Origin or Setup Origin).
9. Identify features and clamps.
10. Generate toolpaths.
11. Post process the toolpaths.

The following series of tutorials show you how to generate finish toolpaths on a SOLIDWORKS/CAMWorks Solids 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 assemblies are provided for the exercises in this chapter. When you install CAMWorks, these files are installed automatically.

Assembly 1

Topics covered in this tutorial:

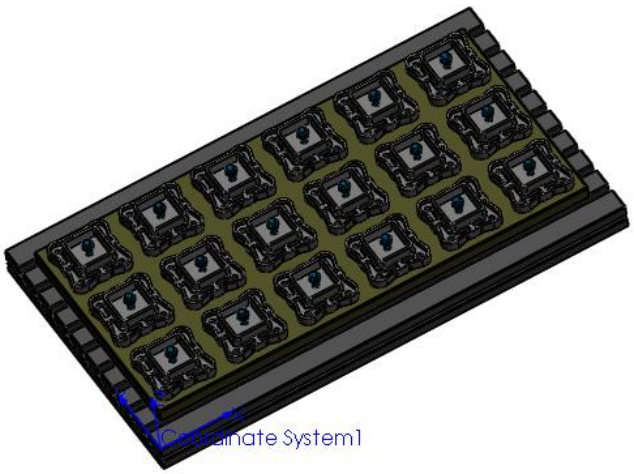
- Using the CAMWorks Command Manager or Workflow toolbar to execute commands
- Defining the Machine
- Selecting the Parts to be Machined
- Defining the Stock
- Changing Default Settings using Options dialog box
- Using the CAMWorks Feature Tree and Operation Tree
- Defining Machinable Features
- Sorting Part Instances to Determine Machining Order
- Generating Operation Plan
- Defining G-code Program Zero Location
- Identifying Fixtures and Clamps
- Generating Toolpaths and Sorting Operations
- Simulate Toolpaths

Step 1: Model Part in SOLIDWORKS/CAMWorks Solids or Import Part

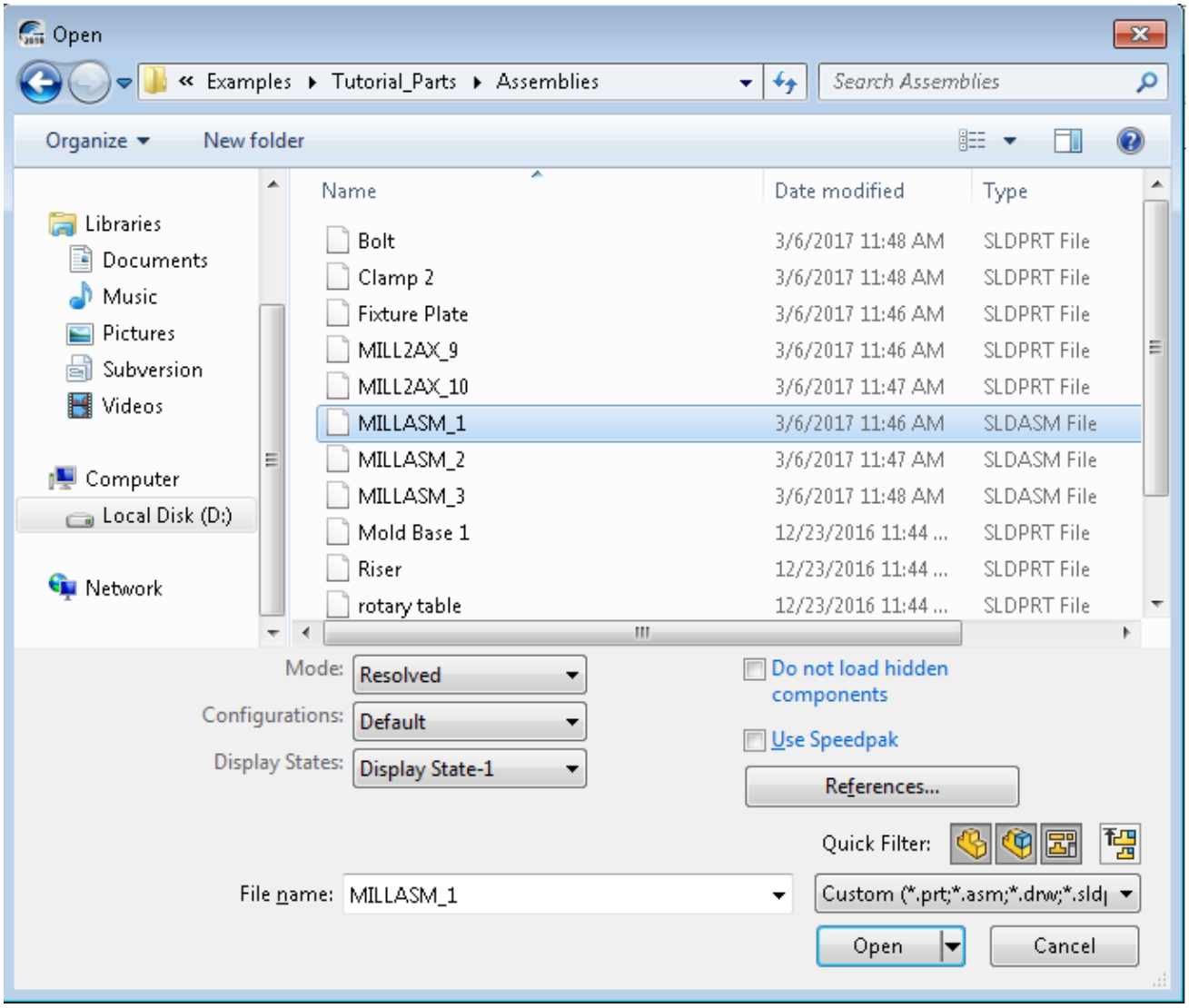
A part is a solid that is created with SOLIDWORKS/CAMWorks Solids or imported into SOLIDWORKS/CAMWorks Solids from another CAD system via an IGES, Parasolid, SAT file, etc. This tutorial uses an existing SOLIDWORKS part.

Open the assembly file **MILLASM_1.SLDASM** in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Assemblies



MILLASM_1.SLDASM



Opening the Solid Part file

Viewing the FeatureManager Design Tree

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 trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks trees. Click the *Pin* button to continuously view this Tree area.

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

Step 2: Change to CAMWorks Feature Tree

Click on the *CAMWorks Feature Tree* tab.

When the CAMWorks Feature tree is displayed, it initially lists the *NC Manager*, *Configurations, Part Manager, Machine* and *Recycle Bin* items.

The icon that displays for the 'Machine' is indicative of the machine which is currently selected. Step 3 of this tutorial explains how to select a turn machine (shown in the images on the next page).

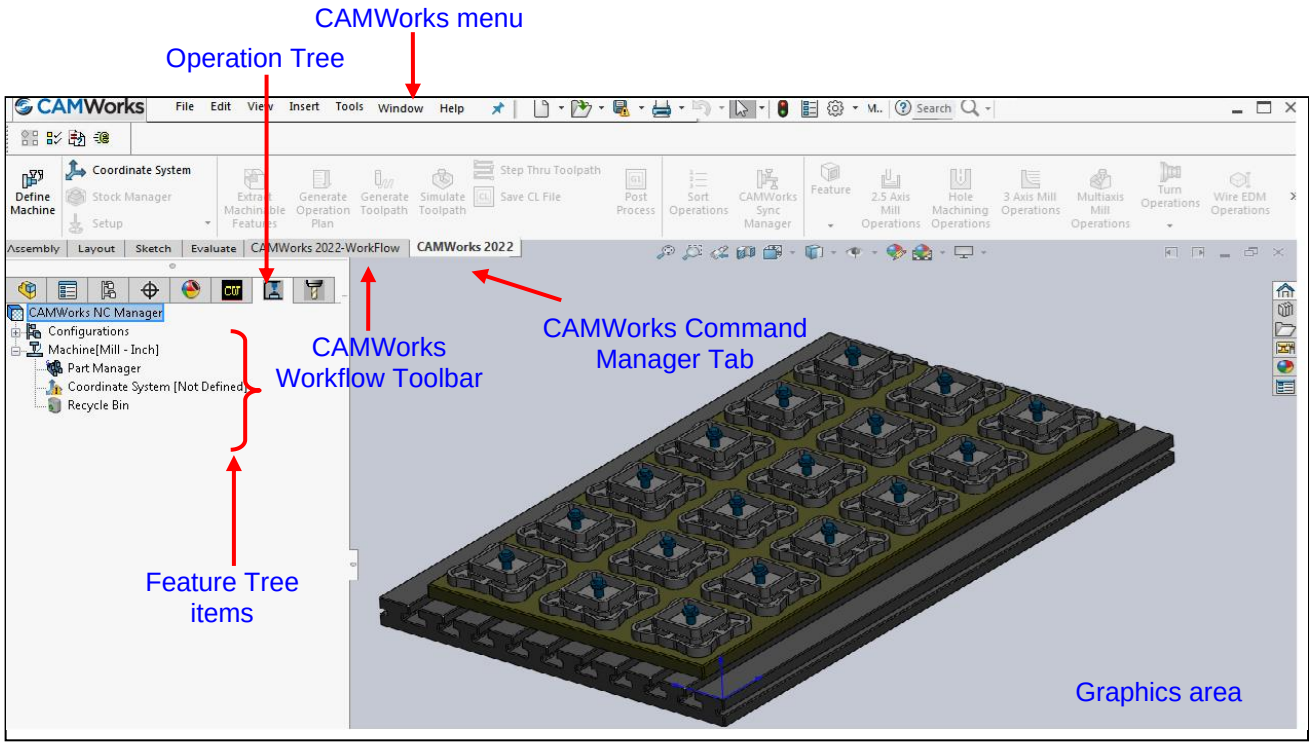
CAMWorks Machining Trees

The CAMWorks machining trees provide an outline view of the machining information for the model. Initially, the CAMWorks Feature tree shows only the CAMWorks NC Manager, Configurations, Part 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/CAMWorks Solids trees and the CAMWorks trees.

- **Configurations**

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



- **Machine**

  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. These machines are set up in the Technology Database.

- **Part Manager**

  This list displays the parts to be machined. You can pick the parts in the graphics area or from the SOLIDWORKS FeatureManager

  design tree. Each unique part is identified by its file name. Instances of the part are numbered incrementally and displayed under the file name.

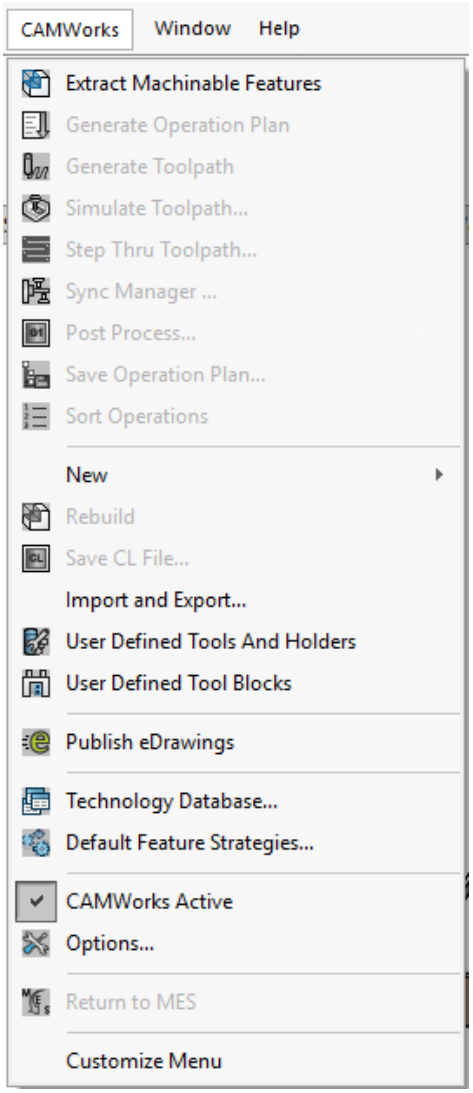
- **Recycle Bin**

  The Recycle Bin is used to store machinable features that you do not intend to machine.

CAMWorks Menu and CAMWorks NC Manager

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

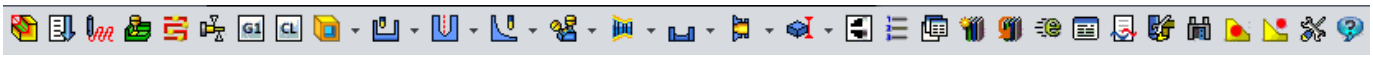
2. The CAMWorks NC Manager item is present in both the Feature tree and the Operation tree. Right click on the *CAMWorks NC Manager* in the tree. A list of executable commands is displayed on the context menu. These right-click context menus provide access to a variety of commands. The commands displayed on the context menu for *CAMWorks NC Manager* item in the Feature tree is different from that in the Operation tree.
3. The CAMWorks commands are explained in the CAMWorks context-based Help.



CAMWorks Menu

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. The CAMWorks Command Manager 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* from the RMB context menu. The

Customize dialog box will be displayed. Use the *Toolbars, Commands, Menus, Keyboard, Shortcut Bars, 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 Workflow Toolbar

The CAMWorks Workflow toolbar contains all the major commands of CAMWorks in a separate toolbar. When executed in sequential order, they define the workflow in CAMWorks. The workflow includes, setting up of the machine, stock and subsequent features, operations and toolpaths.

CAMWorks Options

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

   OR

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

Options button on the CAMWorks Command Manager

   The Options dialog box is displayed. This dialog box contains various tabs to customize settings and options related to Saving data, Feature recognition, Display, Simulation, Updation and Rebuild of CAMWorks data and File locations.

2. Click on each different tab in this dialog box and click the *Help* button. Each tab is explained in the context-based Help.
3.  To close the context-based Help, click the *Close* button in the upper right corner of the Help window to close the window.
4. Click *OK/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

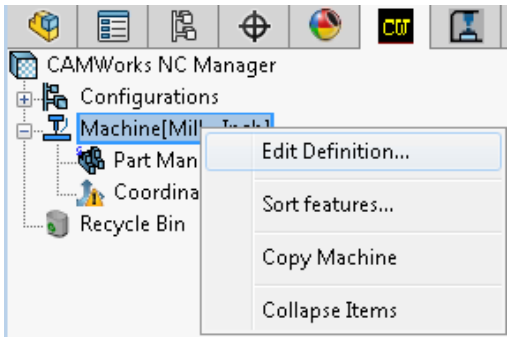
  An alternative machine can be selected at any time to output different G-code programs for alternative machine tools. If the machine type changes, then all features and operations will be deleted.
- **Tool crib:** A subset of tools from the tool library that are commonly loaded into or used with the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

Define the Machine

1. Right click the *Machine [Mill - Inch]* item in the CAMWorks Feature tree and select the Edit Definition from the context menu.

   OR

   Double click the *Machine [Mill - Inch]* item in the Feature tree to edit the machine definition.



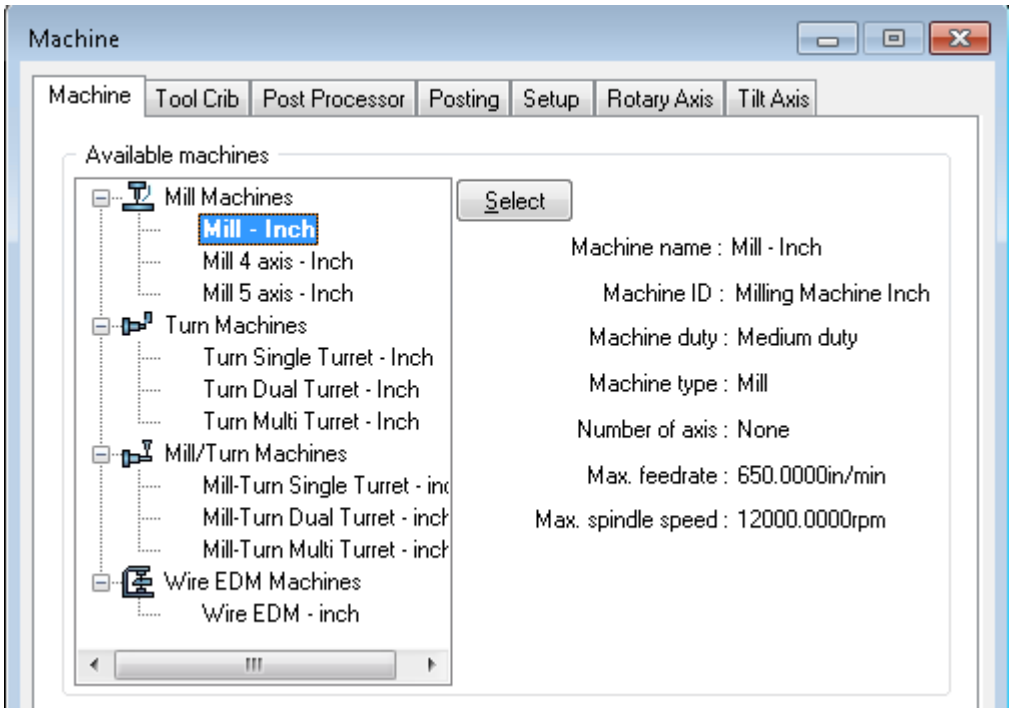
Select 'Edit Definition' on the context menu

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

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

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

2. On the Machine tab, select *Machine [Mill-inch]* in the list of Available machines and click the *Select* button.



Machine tab of Machine Dialog Box

Milling is the default machining type that is set when CAMWorks is first installed. The default machining type is specified in the Technology Database. This is the machine used for all the tutorials for Turn machining in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use.

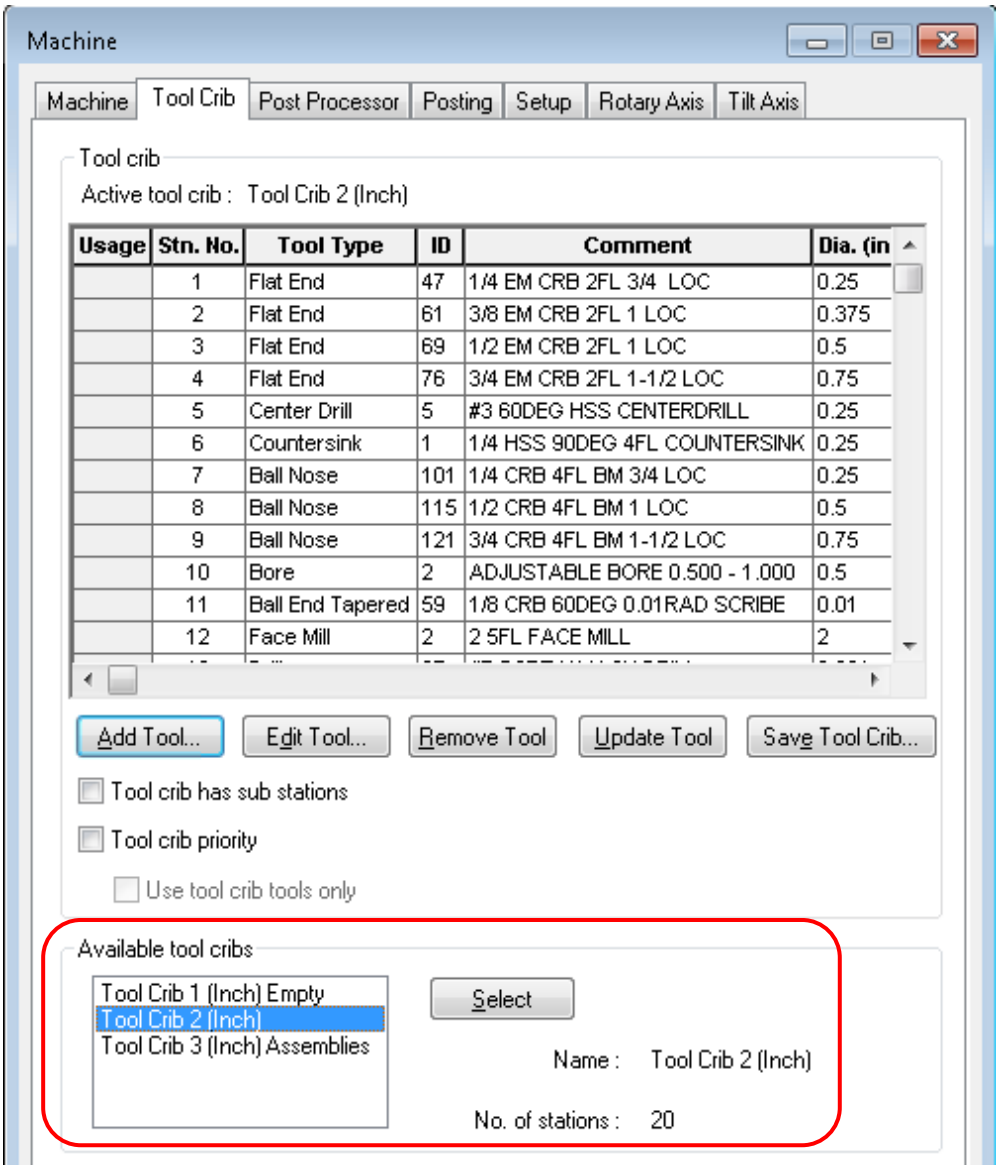
Machines and the Machine tools are set up in the TechDB. Before using CAMWorks to machine your parts, make sure you define the machine and the machine tools available in your facility within the TechDB.

3. Click the *Tool Crib* tab.
   a. Ensure that the *Tool crib priority* option is unchecked.
   b. Make sure *Tool Crib 2 (Inch)* is the Active tool crib.

   The name of the selected tool crib will be displayed below the *Select* button.

   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.

   Tool Crib 1 (Inch) is a default tool crib that has been set up for the sample mill assembly machine. When you define your machine tools in the Technology Database, you can set up your own tool cribs.



Selecting the Tool Crib

4. Click the *Post Processor* tab.

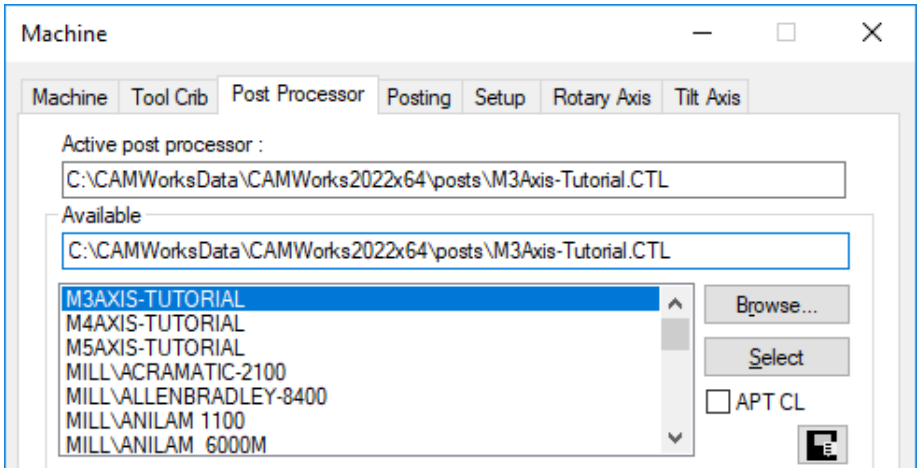
   This tab allows you to select the internal post processor or the APT CL option to output a CL file. The list that displays depends on the post processors that are installed on your system.

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

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

5. If *M3AXIS-TUTORIAL* (the tutorial post processor) is not the active post processor, highlight it in the list and click the *Select* button. This post processor is used for exercise in this manual. When you use CAMWorks to machine your own parts, you can select your machine tool controller or post processor.

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



Post Processor tab of Machine Dialog Box

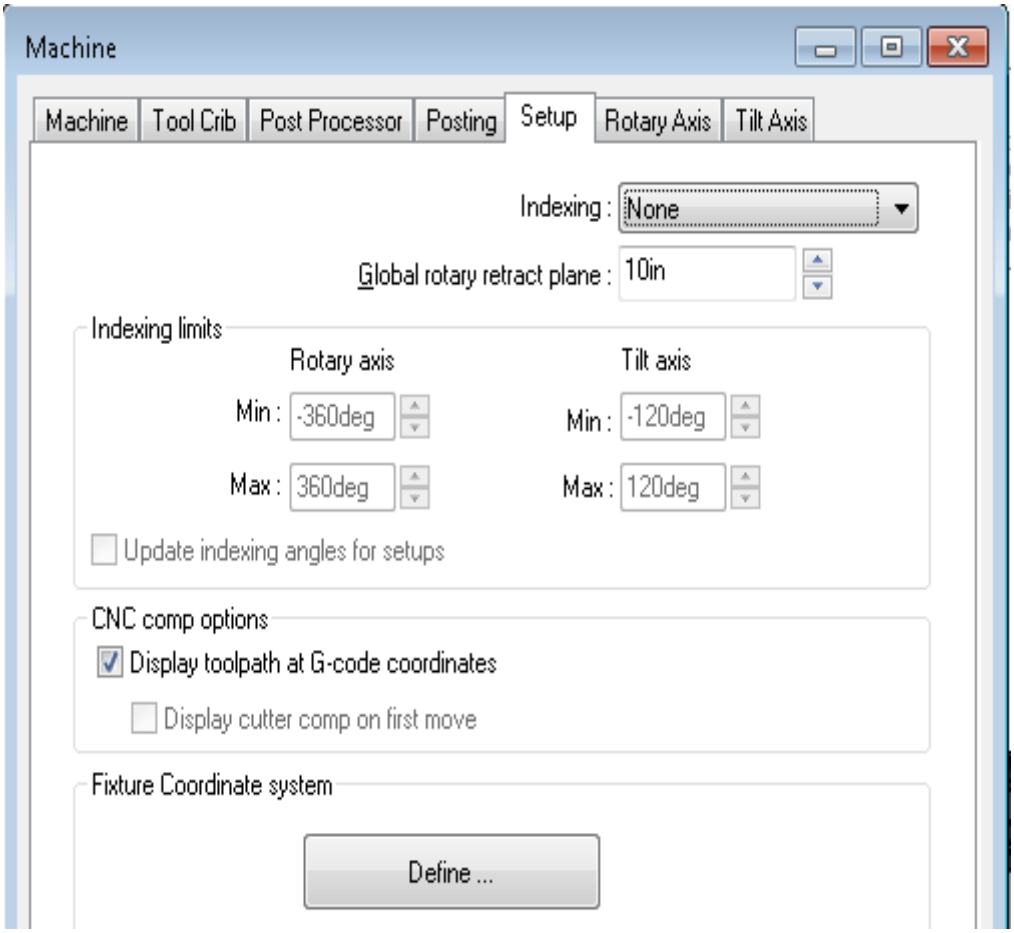
6. Click the *More* button of the Post Processor tab.

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

7. 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. The Fixture Coordinate System defines the default G-code origin, defines the XYZ machining directions and acts as a reference point, if subroutines are used.



Setup tab of Machine Dialog Box

When machinable features are extracted automatically on a part, it is likely that CAMWorks will create multiple mill part setups in order to machine all features on the part. However, if the machine does not support rotary indexing, only one machining direction is possible for the program. The Fixture Coordinate System is used as a filter to determine which of these mill part setups will be active. Therefore, the Fixture Coordinate System should be set first, before extracting machinable features.

8. In the Fixture Coordinate system Group box, Click on the Define button. The Fixture Coordinate system dialog box is displayed. In the Method group box select SOLIDWORKS coordinate system from the drop down list and click on Coordinate system1 in the Available Coordinate Systems group box. Coordinate System1 is automatically selected under the Coordinate Systems group box. Click on *OK* .
9. Click *OK* to close the Machine Dialog Box.

Step 4: 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.

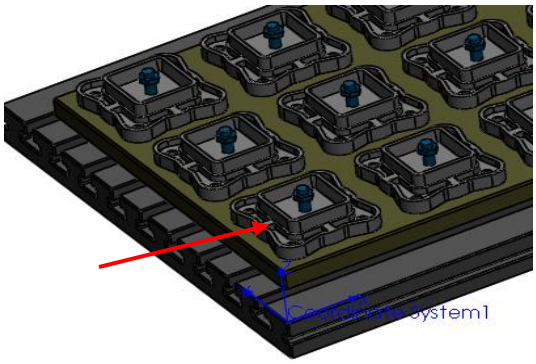
1. Double click *Part Manager* in the tree.

   The *Manage Parts* dialog box will be displayed.

2. In the graphics area, select the part in the lower left corner of the assembly as shown in the image.

   This action selects this part to the *Selected Parts* list within the *Manage Parts* dialog box.

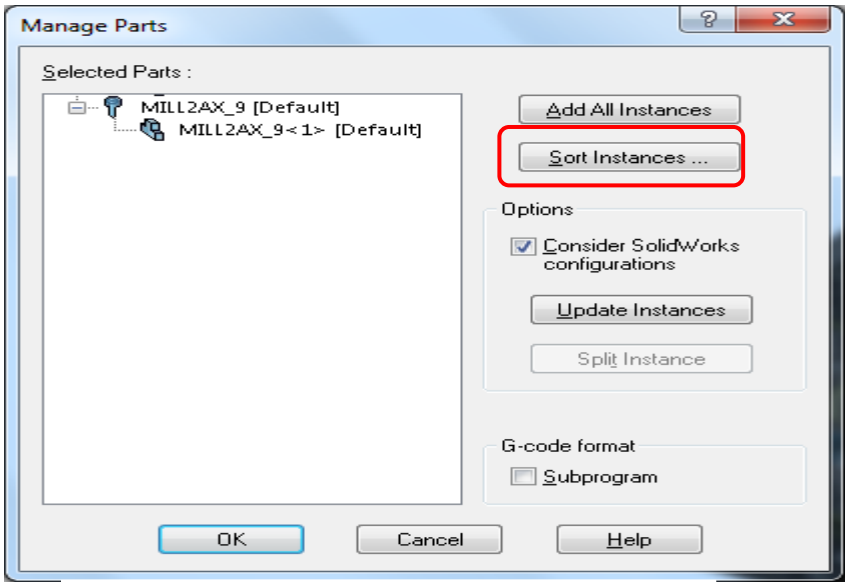
   For each unique part in the assembly, the first instance that you select is called the seed part. When an action is performed on the seed part, the same action is applied to every instance of that part in the assembly.



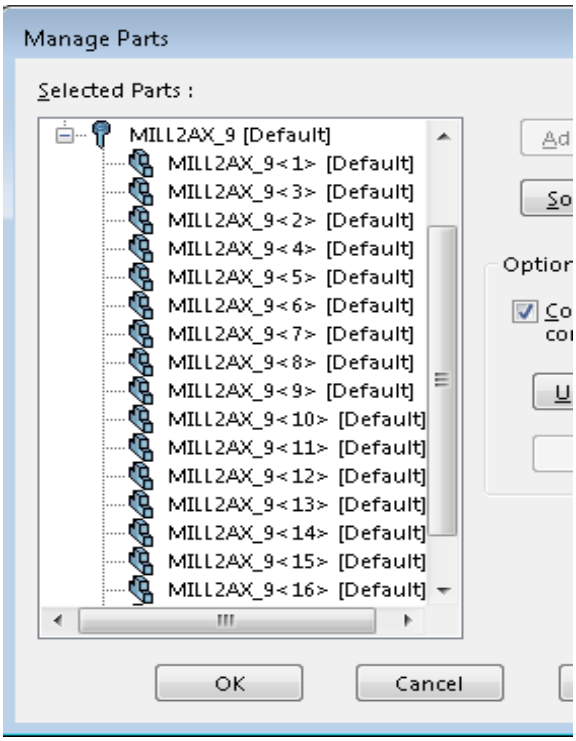
Select the lower left corner part of Assembly

3. Highlight the part *(MILL2AX_9.sldprt)* 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/CAMWorks Solids FeatureManager tree.



Manage Parts Dialog Box



List of Parts

Later in this tutorial, 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 *Manage Parts* 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 Mill Part Setups and machinable features associated to the seed part.
- For each unique part, all the instances are listed under the *Instances* item.

Step 5: Define 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.

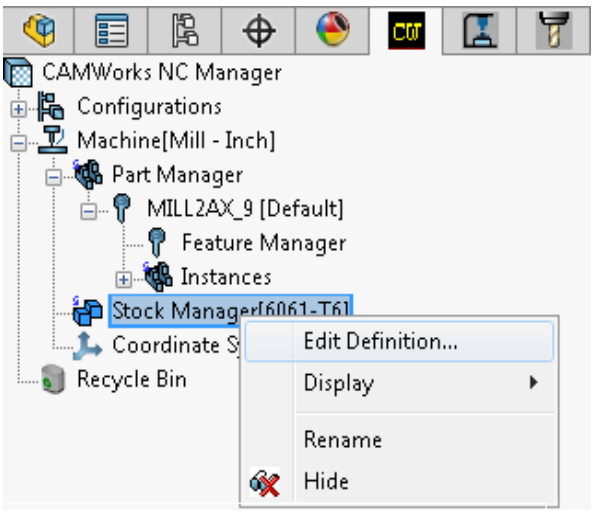
Following are the steps to define the Stock:

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

   OR

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

   The Stock Manager dialog box is displayed. This dialog box allows you to modify existing stock or create new stock for single parts and common stock for multiple parts.



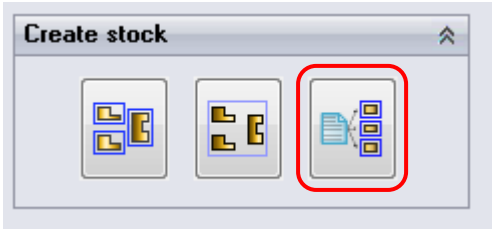
Command to open the Stock Manager dialog box

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. In the *Bounding Box Offset* group box, change the value for *Z+* to **0.1in.**

4. In the *Create Stock* group box, click the *Apply Current Stock Definition to All Parts* button. The change is applied to the stock for all part instances.



Click 'Apply Current Stock Definition to All Parts' button

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

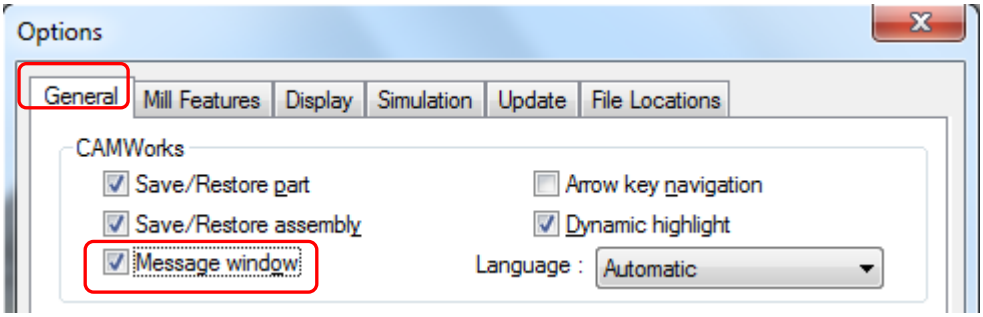
Step 6: CAMWorks Options dialog box to control CAMWorks Settings

Click on the *Options* button on the CAMWorks Command Manager. This action opens the *Options* dialog box.

To view the Options dialog box, you can also click CAMWorks on the menu bar OR right click CAMWorks NC Manager in the Feature tree and select the Options command.

General Tab

On the *General* tab of this dialog box, check the *Message Window* option. This is the setting to control whether the Message window displays temporarily or permanently. Checking this option keeps the Message Window in permanent display state whenever a command is executed within CAMWorks.



CAMWorks Options Dialog box - General Tab

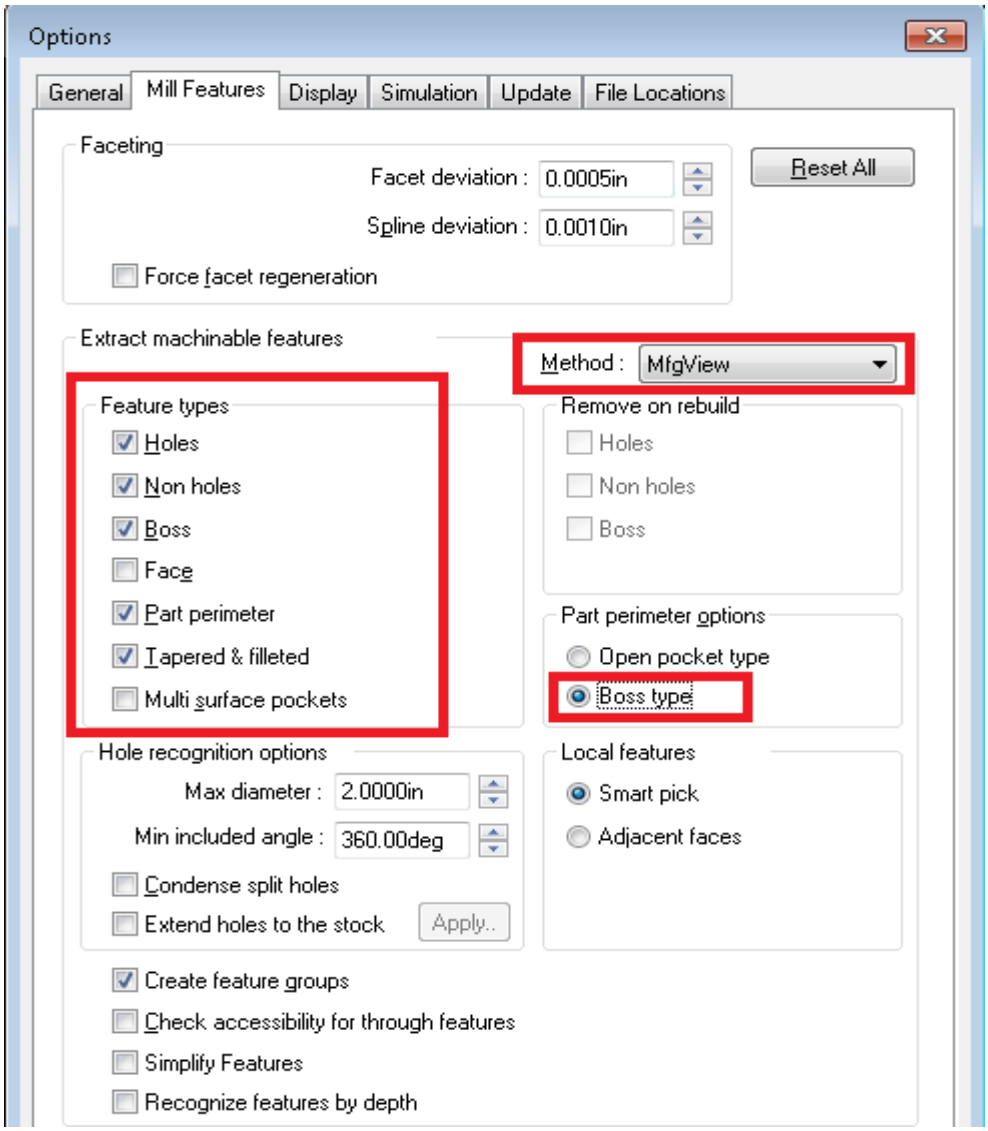
Mill Features Tab

On the *Mill Features* tab of this dialog box:

1. Ensure that in the *Method* dropdown list, *MfgView* is selected.
2. Under the *Feature types* group box, ensure that the following feature options are checked:
   - Holes
   - Non holes
   - Boss
   - Part perimeter
   - Tapered & filleted

   Only the features that are selected in this group box will be recognized automatically on executing the *Extract Machinable Features* command.
3. In the *Part perimeter options* group box, ensure that *Boss type* is selected.

   When this option is selected, if a part perimeter is recognized, then on executing the *Extract Machinable Features* command, a Boss feature based on the perimeter of the part will be created. When created, this feature will be automatically set to *Through*.
4. At the bottom of the tab, ensure that the *Create feature groups* option is checked. Selecting this option ensures that identical features are grouped together.
5. Click *OK* to apply the changes and close the *Options* dialog box.



CAMWorks Options Dialog box – Mill Features Tab

Step 7: Defining Machinable Features

You can now extract the machinable features. The features recognized will depend on the settings in the Mill Features tab of the *CAMWorks Options* dialog box as explained in the previous step. Machinable Features are added in the *Feature Manager* area of the tree.

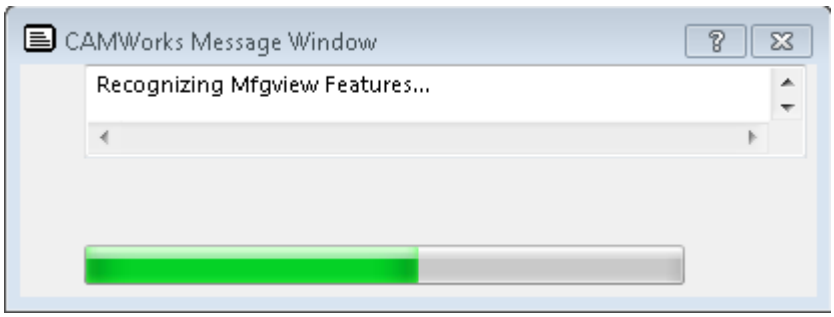
At the Mill Part Setup level, features can be inserted interactively using the *2.5 Axis* or *Multi Surface Feature* commands. CAMWorks automatically copies the features to every other instance of the part selected in the Part Manager. When machining multiple instances of the same part, if you only want to create one instance of the feature, you can use the Assembly Feature command on the feature context menu to declare the feature an Assembly Feature. By doing so, CAMWorks will not copy the feature to all instances of the part.

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

   OR

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

The *Message Window* is displayed. This window is displayed automatically to report the progress of the current process. Generating Setups is always the last item during Automatic Feature Recognition. When you see the line *Generating Setups* in the message window, you can be sure that the process is almost complete.



CAMWorks Message Window

2. On execution of the *Extract Machinable Features* command, CAMWorks generates the Mill Part Setup and the machinable features. The items are displayed in the Feature tree.

Feature Tree: 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.
- Hide or show feature display in graphics area.
- Generate an Operation Plan and find the first operation for a feature.

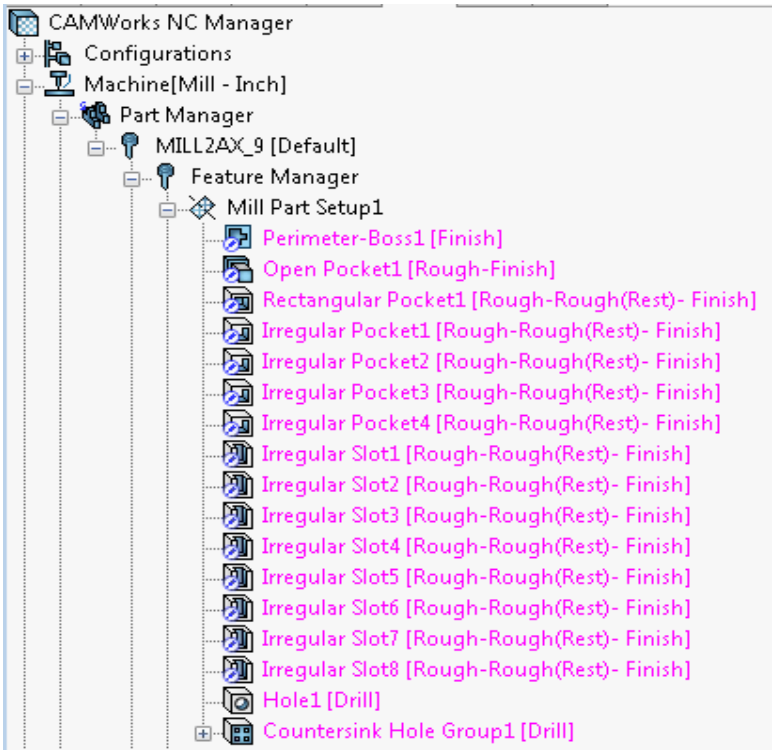
Did You Know: **When you recognize features by Automatic Feature Recognition (AFR) or Interactive Feature Recognition (IFR), the features listed in the Feature tree will display in different color Magenta color (by default) till you generate operations for these features. Once a valid operation is generated, the color of the corresponding feature item will change Black color (by default) indicating successful generation of the operation(s). If operations could not be generated for a feature (because the feature conditions have not been defined in the Technology Database for that particular feature type), then the feature will continue to display in the initial color (Magenta color), thus indicating that they have no operations defined.**

You can set these colors on the Display tab in the Options dialog box.

3.  Click the plus sign next to the *Feature Manager* in the Feature tree.

The Feature Manager displays all the Mill Part Setups and machinable features that were created by ***Automatic Feature Recognition (AFR)*** without consideration for how the machine is set up. In this case, AFR found and created two Mill Part Setups. Click on each Mill Part Setup and note the machining direction. Note that the machining direction for one of

the Mill Part Setups comes from the underside of the table and the other from the spindle side of the table. If there was no consideration for which side of the table the spindle is on, a through feature could be machined from either direction. Click several of the features to confirm that they can be machined from the Mill Part Setup that they belong to.



Generated Features listed in Feature Tree

4. For this part, the feature called *Hole1* is a through feature that we will assume has already exists and does not need to be machined. To avoid machining this hole, right-click on *Hole1* under *Mill Part Setup1* in the Feature tree and select *Delete* from the context menu.

5. For this part, the features in *Mill Part Setup2* can be processed based on how this machine is defined. In order to verify, expand *Setup1* at the bottom of the Feature tree by clicking on the '+' sign.

   As mentioned above, when you ran AFR, CAMWorks created two mill part setups without regard to the spindle direction. Because this machine is defined without Indexing support, there will be only a single machining Setup. Based on the -Z direction of the Fixture Coordinate System, CAMWorks found that *Mill Part Setup2* is in the same direction and created machining *Setup1* and placed all machinable features in this setup.

Setup 1 in the feature Tree

Step 8: 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 Feature tree, click the (⊞) plus sign next to several features.

   The order that the part instances are listed under each feature is the machining order for that feature. By default, for all features, the parts are in the order they appear in the Part Manager. You can change the order globally for all features or for individual features.

2.  Double click *Part Manager* in the Feature tree. The *Manage Parts* dialog box will be displayed.

3. Click the *Sort Instances* button in the *Manage Parts* dialog box.

   The *Sort Instances* dialog box will be displayed. This dialog box provides automatic or manual options for sorting the part instances for features in the Setup.

   - 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.
   - The *Feature instances* option allows you to manually reorder 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.
   - *Grid pattern* automatically sorts part instances for all features in the Setup based on the start corner, processing direction and process order.

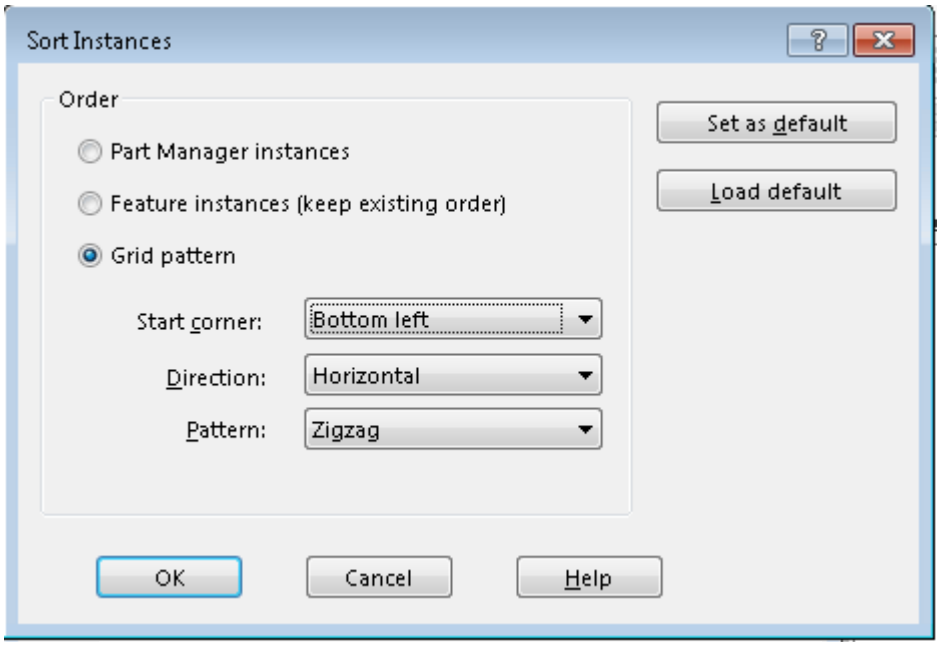
   Did You Know: **You can use one of the automatic methods, then if necessary, select the Feature instances option and make changes to the part order for individual features.**

4. Select the *Grid pattern* option.

   When you will select the *Grid pattern* option, the order will change for the part instances under every feature in the Setup.

5. Select the following grid options, then click *OK*:

   - Start corner = Bottom left
   - Direction = Horizontal
   - Pattern = Zigzag



Sort Instances Dialog Box

6. Click *OK* to apply the changes and close the *Manage Parts* dialog box.
7. Click the (⊞) plus sign next to a feature in the *Setup1* and click each part instance to view the machining order on the assembly in the graphics area.

Step 9: Generating the Operation Plan and Adjusting 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.

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager or right click the *CAMWorks NC Manager* of the Feature tree and select *Generate Operation Plan*.

   In the Operation tree, the Setup lists the operations.

   In the Operation tree, the generated operation is displayed in the Setup1 lists. The listed operations are displayed in **magenta** color (default color setting). This color indicates that toolpaths have not yet been generated for the operations.

   Operation Tree: 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.

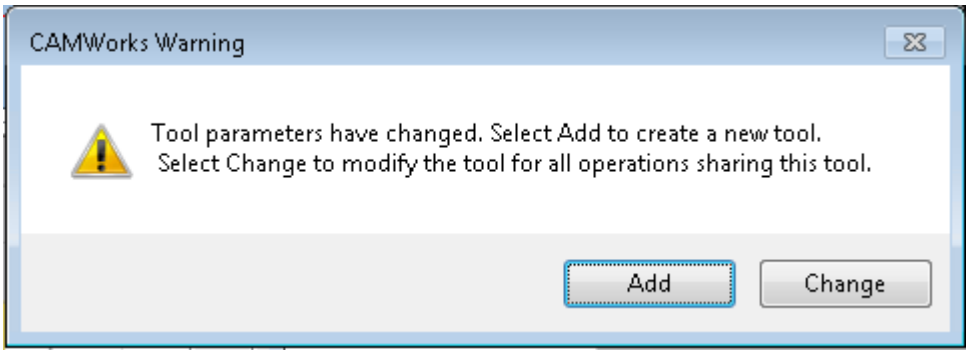
2. In the Operation tree, double click *Rough Mill3* operation. This operation is required to machine the *Rectangular Pocket1* feature. The default tool assigned by CAMWorks to this operation gouges the part. Hence, a suitable tool needs to be assigned from the Tool crib.
   The *Operation Parameters* dialog box will be displayed.
3. Click on the *Tool* tab and select *Tool Crib* page.
4. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
5. Click *Yes* to replace the corresponding holder too.
6. Click on the *Rouging* tab. In the *Rest machining* group box at the bottom of the tab, select *From WIP* in the *Machine* dropdown list.
7. Click *OK* to apply the changes and close the dialog box.
8. Right-click on the Contour Mill3 operation again and select *Generate Toolpath* command on the context menu.
9. Under *Setup1*, double-click on the *Contour Mill5* operation. The *Operation Parameters* dialog box will be displayed.
10. In the Operation tree, double click *Contour Mill5* operation.
    OR
    Right click *Contour Mill5* operation and select *Edit Definition* on the context menu.
    The *Operation Parameters* dialog box is displayed. *Contour Mill5* operation is used for machining the *Irregular Pocket1* feature of the part.
    Click on the *Tool* tab and select the *Mill Tool* page. It displays the parameters of the selected tool.
11. Observe the *Tool Usage* value. This value indicates the number of operations currently using this tool.

Generated Operations listed in Operation Tree

12. In the *Cut Diameter (D1)* field, set the diameter value to **0.625in**.
13. Click *OK* to apply the change.

    Since this tool is also shared by ten operations, making any changes will affect those operations too. Therefore, CAMWorks will display a warning message to this effect and prompt you to choose whether the changes are to be applied to the other operations or not.
14. Click *Change* on the message.

    - **Change:** When you click *Change* within this dialog, then the changes made to the tool parameters will affect all other operations sharing this tool.
    - **Add:** If you click *Add* within the warning message dialog, then CAMWorks creates a new tool with the changed tool parameters and lists this tool in the Active Tool crib. This action ensures that the changes made to the tool parameters will affect only the current operation and none of the other operations which share the same tool. Note that irrespective of which option you choose, the changes made to the tool parameters are applicable only for machining of the current part. The changes made to the tool parameters are not saved to the Technology Database.



CAMWorks Warning message

15. In the Operation tree, observe the ten Mill operations sharing this tool. Observe that the diameter of the Flat End tool given within the brackets now displays the edited values for all these three operations.

Step 10: 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. Double click *Setup1* in the Operation tree.

   The *Setup Parameters* dialog box will be displayed.
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 on the *Offset* tab.

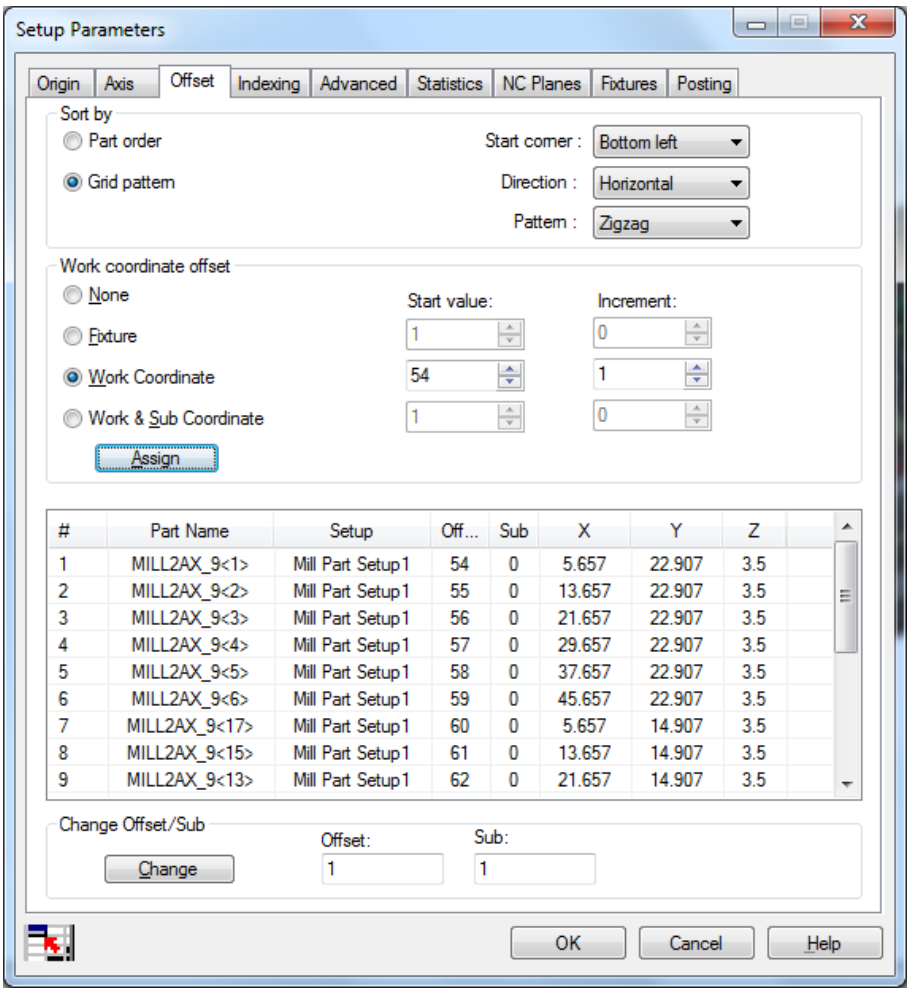
   The order of the parts on this page affects only the assignment of the offsets, not the machining order.
4. In the *Sort by* group box, select *Grid pattern*.

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 (Step 8-Point 5):
   - 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)

   Notice that the part order is updated in the table. You can specify a programmable coordinate offset and assign an offset to each part.
6. Set the *Work coordinate offset* to *Work Coordinate*. This option will output G54, G55, etc.
7. 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.
8. Click the *Assign* button of the *Work Coordinate offset* group box. The numbers update in the *Offset* and *Sub* columns in the table.
9. Click *OK* to apply the changes and close the *Setup Parameters* dialog box.
10. If any warning message is displayed, click *No* to continue.



Setup Parameters Dialog Box

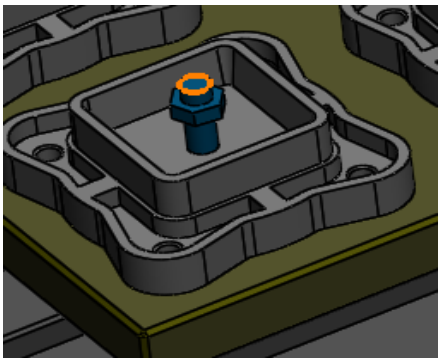
Did You Know: **Changing the machining order does not automatically change the offset assignments If you want the offset order to corresponding to the machining order, you need to sort the parts and reassign the offsets on the Offset tab.**

Step 11: Identifying Fixtures and Clamps

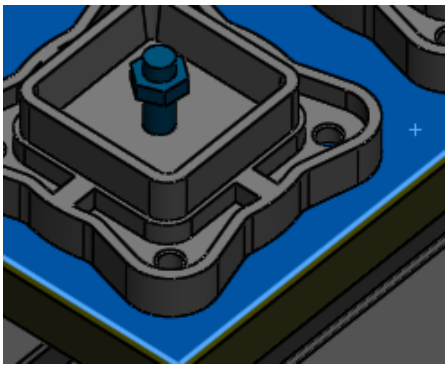
Clamps and fixture components are added on the *Fixtures* tab in the *Setup Parameters* dialog box. This dialog box allows you to define clamps, bolts, etc., so that machining toolpaths will avoid these areas and to specify the clamps and fixtures that you want displayed during simulation. Fixtures identified to avoid apply only to 2 Axis Rough and Contour toolpath calculations.

1. Double click *Setup1* in the Operation tree. The *Setup Parameters* dialog box will be displayed.
2. Click on the *Fixtures* tab in the *Setup Parameters* dialog box.
3. In the graphics area, pick the bolt holding the seed part.
4. In the graphics area, pick the fixture plate and the machining table one after another.

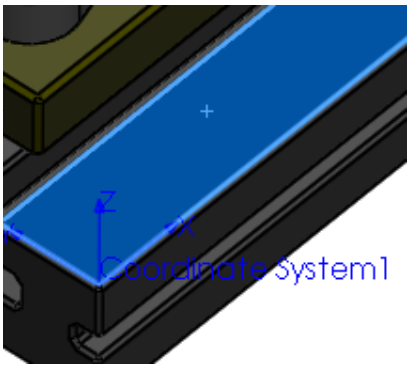
   The part names is displayed in the *Fixtures* list and will display during simulation.



Select 'Bolt' holding the Seed part



Select 'Fixture Plate'



Select 'Machining Table'

5. Highlight the bolt in the list and click the *Add All Instances* button.

   All the bolts in the assembly will be 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. Click on the *Avoid All* button to avoid all listed items.
8. Uncheck the *Avoid* check boxes for the fixture plate and the table.

   Although fixtures are 3 dimensional SOLIDWORKS/CAMWorks Solids parts, CAMWorks considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and contour operations. Therefore, parts such as vices, the machine table, or

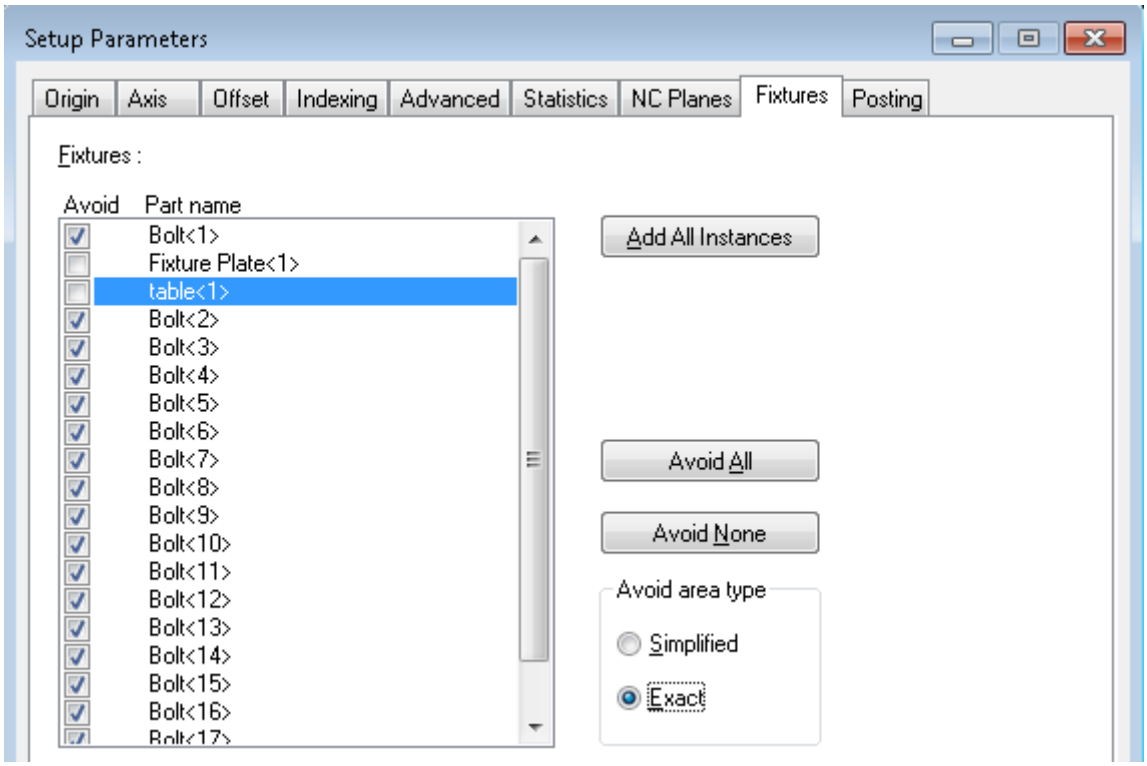
rotary fixtures whose silhouettes are larger than the part must not be selected to avoid, otherwise no toolpath will be generated.

9. 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.

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

11. If a warning message is displayed, click *No* to continue.



Fixtures tab of the Setup Parameters Dialog Box

Step 12: Generating Toolpaths and Sorting Operations

CAMWorks calculates toolpaths using the operation parameters to define how to machine each machinable feature. 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 Command Manager/Workflow toolbar. OR

   Right click *Setup1* in the Operation tree and select *Generate Toolpath* on the context menu.

   On executing the Generate Toolpath command, CAMWorks calculates the toolpaths for each operation in the Setup. The font color of all the listed operations in the Operation tree changes from **magenta** to **black**. This change in color indicates that toolpaths were successfully generated.

   Did You Know: **If an operation displays in a magenta color instead of black, then toolpaths have not been generated. This might occur in one of the following situations:**

   i. When you insert a new operation interactively;

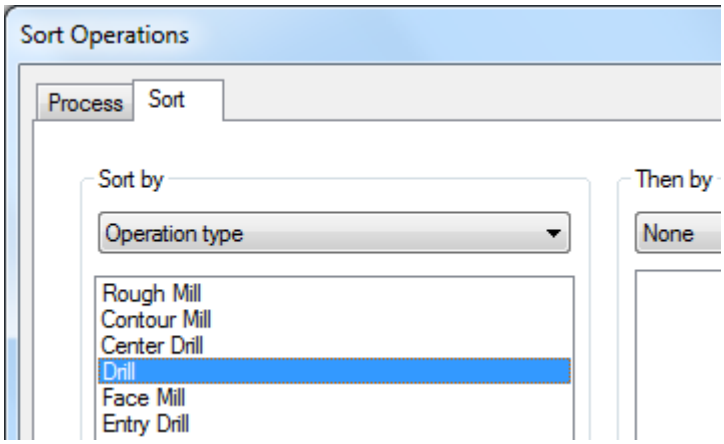
ii. **When you insert a new feature interactively and then generate operations for the new features;**

iii. **When CAMWorks cannot generate the toolpath for an operation because of error in the toolpath algorithm or a parameter is not correct.**

2. Right click *Setup1* in the Operation tree and select *Sort Operations* from the context menu.

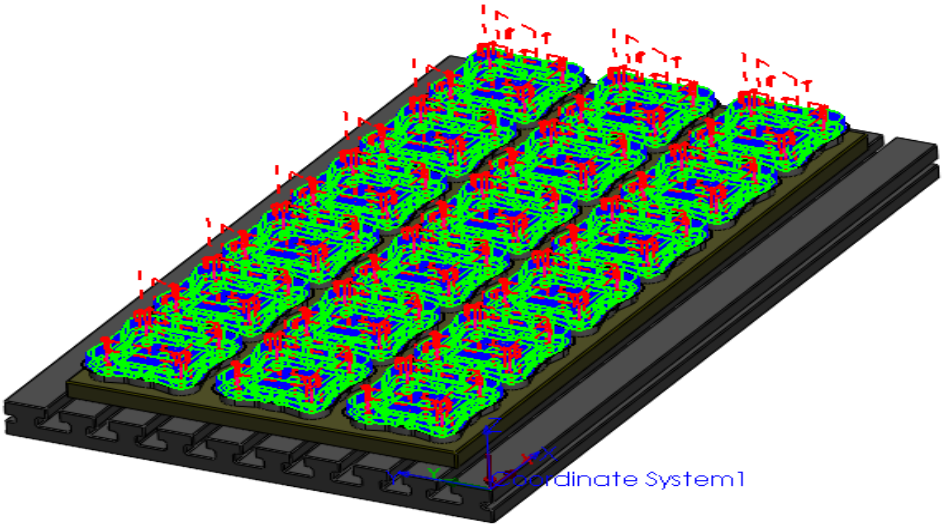
   The *Sort Operations* dialog box will be displayed.

3. On the *Process* tab, remove the check mark from the *Process complete feature* option.
4. Click on the *Sort* tab.
5. Drag and drop operations so that Rough Mill is at the top of the list, followed by Contour Mill, Center Drill, and Drill.



Drag and drop the operations

6. Click *Apply* and confirm that the tree view updates to sort the operations according to this order. If it sorts the operations as expected, then click *OK*.
7. The operations under *Setup1* are sorted based in the order on the *Sort* tab.
8. Left click any operation in the Operation tree. That operation will be highlighted in the Operation tree.
   - The toolpath for that highlighted operation will be displayed in the graphics area. As you highlight each operation in the tree, the toolpaths for that corresponding operation will be displayed.
   - Turning operation parameters can be edited and the operation can be renamed, moved, suppressed, deleted, etc. after toolpaths have been generated. These commands are available in the RMB context menu.
   - If you make any changes, the toolpaths must be updated by selecting *Generate Toolpath* command again at the Setup level.
9. Hold down the *Shift* key and select the first and last operation in the Operation tree. This action selects all the operations. The toolpaths for all the operations will be displayed on the part showing the centerline of the toolpath.



Toolpaths for all the operations displayed on the part when all the operations are selected in the Operation tree

Step 13: Simulate Toolpaths

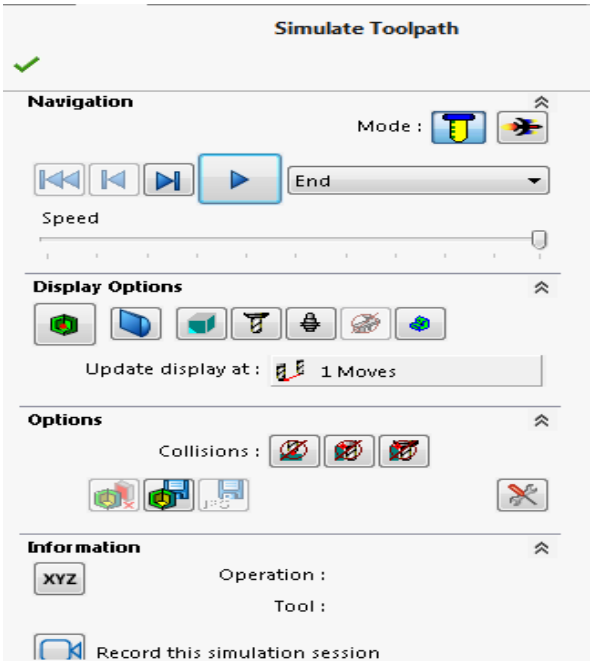
CAMWorks provides the ability to simulate the toolpaths showing the tool movement and the resulting shape of the part.

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

   OR

   Right click on *Turn Setup1* in the operation tree and select *Simulate Toolpath* on the context menu.

   On executing this command, the *Toolpath Simulation* toolbar will be displayed.



Toolpath Simulation Dialog Box

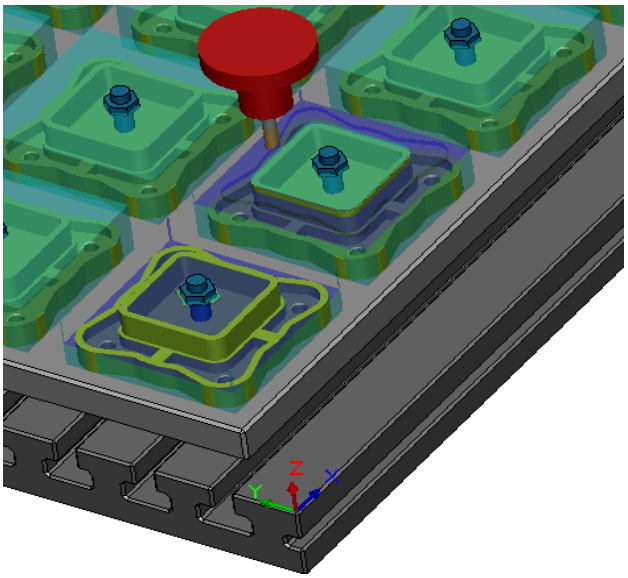
Some of the options you can select to customize the simulation include:

- 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 and then select Simulate Toolpath from the context menu.

2. When you click on the display control buttons of the Simulation toolbar, the available settings associated with that button are displayed in a dropdown list.
3.  Click the *Run* button.
4. The simulation is run with the tool displayed during simulation.
5. 

Use the *Simulation Speed Control* slider to control the speed of the Simulation.
6.  To pause the simulation while it is running, click on the *Pause* button. When you  click *Run* button again, the Simulation will continue from the point where it was paused.
7. Click *OK*  to exit the simulation mode and return to the SOLIDWORKS/CAMWorks Solids display.



Toolpath Simulation

Assembly 2

Topics covered in this tutorial:

- Editing a tool in the active tool crib
- Removing a tool from the active tool crib
- Adding a tool to the active tool crib
- Saving the changes made to the tools in the tool crib

Multi-Plane Machining in Assembly Mode

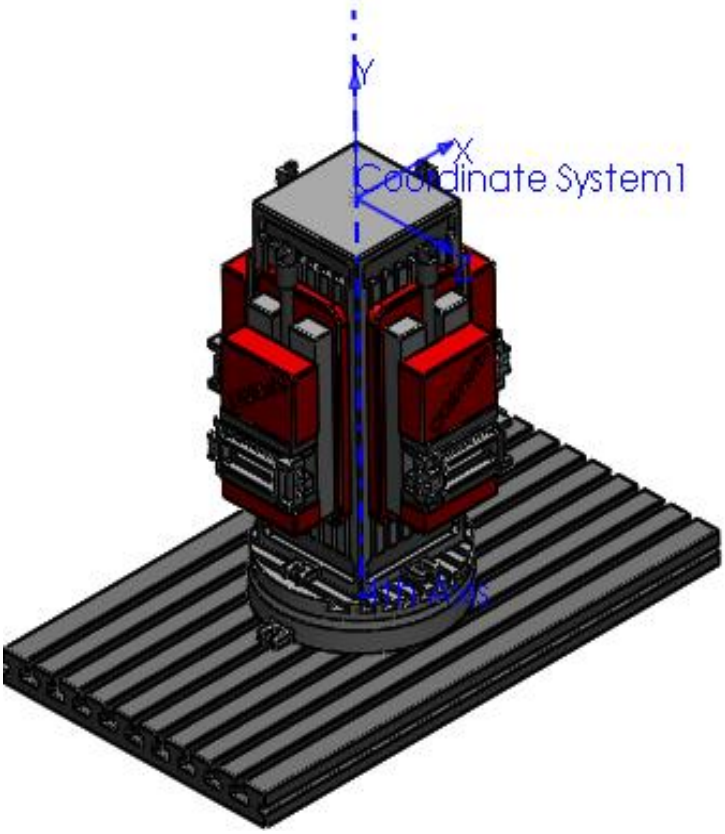
CAMWorks supports 4th and 5th axis rotary (prepositioning) output for milling. The 4th and 5th axis position angles can be user-defined or automatically calculated. Parts requiring multi-plane machining can be programmed in CAMWorks assembly mode.

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.

Step 1: Open the Part

Open the part file **MILLASM_2.SLDASM** located in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Assemblies.



MILLASM_2.SLDASM

Step 2: Defining the Machine

Define the Machine

1.  Click the CAMWorks Feature Tree tab.
2.  Double click the Machine [Mill-Inch] item in the Feature tree.

   OR

    Click on the Define Machine button on the CAMWorks Workflow toolbar.

   The Machine tab of the Machine dialog box is displayed.
3. Make sure Machine [Mill – Inch] is selected in the Available Machines list.

   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.

Editing the Tool Crib

1. Click the *Tool Crib* tab of the Machine dialog box.

   From this tab, you can add, remove and edit tools in the Tool Crib.
2. In the *Available tool cribs*, make sure *Tool Crib 2 (Inch)* is the Active tool crib.

   To select a particular tool crib as the Active tool crib, highlight it in the *Available tool cribs* list and then click the *Select* button.
3. Ensure that the Tool crib priority option is unchecked.

Tool Crib Tab of Machine dialog box

Editing a Tool

1. Select any tool from the Active tool crib list and click the *Edit Tool* button.

   To select a tool in the *Active tool crib* grid, click on any field in the row containing the tool.

2. The *Edit Tool Parameters* dialog box is displayed. This dialog box contains three tabs that allow you to change the parameters for the selected tool.

3. Click the tabs to view the tool and holder parameters. If you make any changes to the parameters, click *OK* to apply those changes and close the *Edit Tool Parameters* dialog box. 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 click on the *Save Tool Crib* button in the Tool crib tab else the changes will be applicable only for the current model part. Alternatively, you can edit the Tool Crib definition in the Technology Database.

Removing a Tool

To remove a tool from the Active tool crib, select the tool in the Active tool crib grid and then click the *Remove Tool* button.

Note that the tool removal from the tool crib is effective only for the current part. To make this change available for all future jobs, click on the *Save Tool Crib* button to permanently save the changes.

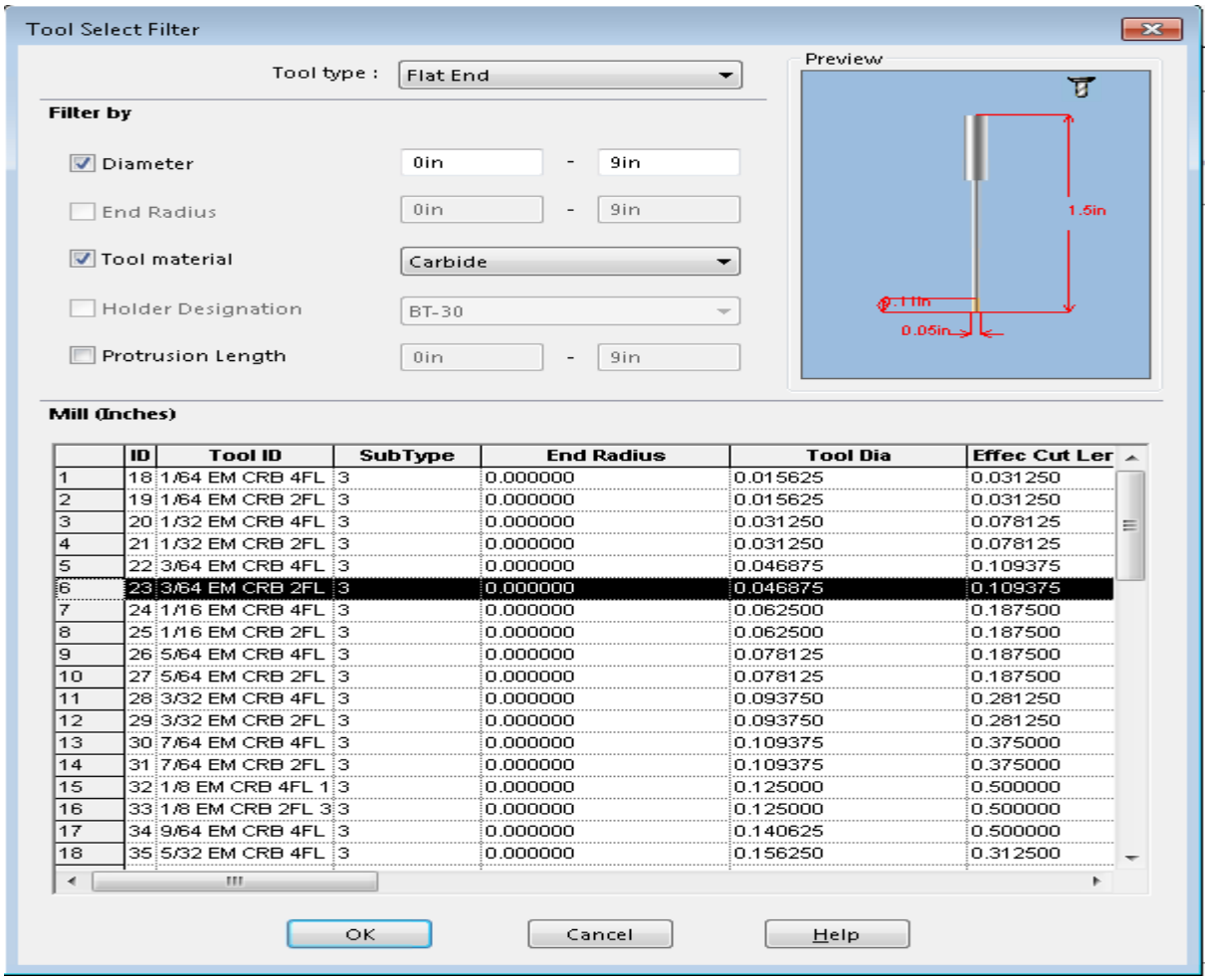
Adding a Tool

Use the Add Tool button to add a tool to the Active Tool crib.

1. Click the *Add Tool* button in the Tool Crib tab.
2. The *Tool Select Filter* dialog box is displayed. This dialog box allows you to set filters and display the list of tools to select.

   In this tutorial, you will insert a Bore to the active tool crib.
3. In the *Tool Select Filter* dialog box, select *Bore* from the dropdown list for the Tool type.

   The related Bore tools is displayed in the list. This list allows you to add an existing tool in the TechDB to your active Tool Crib. The list contains all the tools that have been entered into the TechDB. However, you cannot use this form to add new tools to the TechDB.
4. To add a tool from this list to the active tool crib, highlight the desired tool from the list.
5. The Preview window displays the 3D Model view of the selected tool.
6. Click the *OK* button. To exit without adding any tool, click on the *Cancel* button.
7. If you selected a tool from the list to add to active tool crib, then the new tool will be added to the bottom of the Active Tool crib grid.
8. The *Preview* window is also displayed on the right side of the dialog box. This *Preview* window contains the dynamic 3D model view of the tool and holder, thus enabling visual identification of the selected tool. The Tool Station number, Tool comment and labels of the Tool Parameters are displayed in the *Preview* window.



Tool Select Filter dialog box

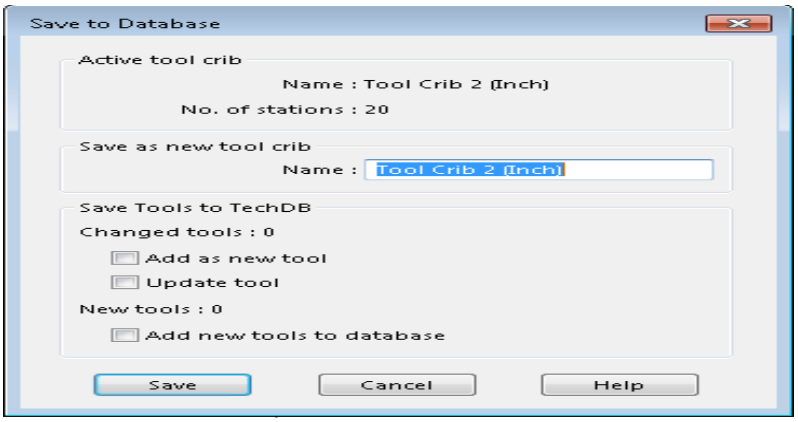
Note that the tool addition to the active tool crib is effective only for the current part. To make this tool addition available for all future jobs, click on the *Save Tool Crib* button to permanently save the changes.

Saving the changes made to a Tool

If you make changes to any tool in the active tool crib, the changes are effective only for the current part and not for any other part. The edits made to a tool to make available for all future jobs, highlight the tool in the *Active tool crib* grid and click on the *Save Tool Crib* button.

When you click *Save Tool Crib*, the Save to Database dialog box is displayed. This dialog box identifies the active tool crib and the number of stations. This dialog box prompts you to select whether you wish modification to the existing tool or not.

- If you select ***'Save'***, then the modifications made to the tool will overwrite the active tool crib in the database.
- You can also type a new name in the Save as input field and then click to *Save* button to create the new tool crib.



Message displayed when you try to Save a Tool

Selecting the Post Processor

1. Click the *Post Processor* tab in the Machine dialog box.
2. Make sure *M4AXIS-TUTORIAL* (the tutorial post processor) is selected as the active post processor.

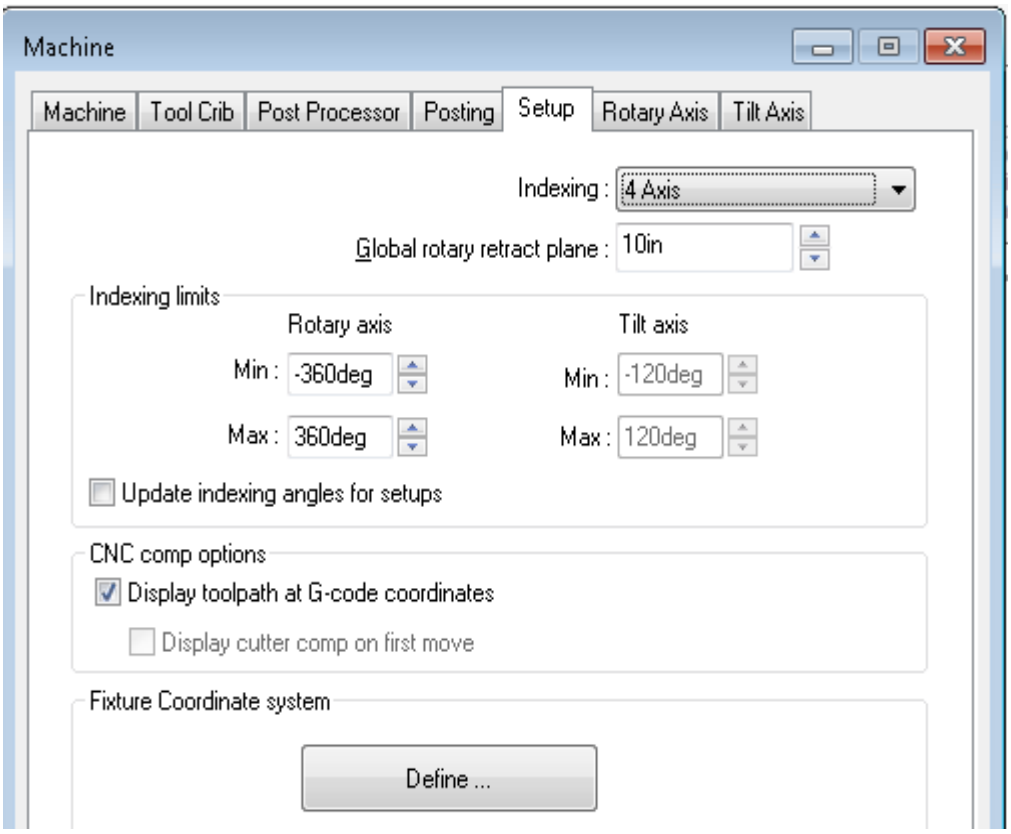
M4AXIS-TUTORIAL is used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.

Setting the Setup and Rotary Axis

1. Click the *Setup* tab.
2. Select ***4 Axis*** for the Indexing option from the dropdown list.

   This parameter defines to CAMWorks the allowable machining directions for the current CAMWorks machine.
3. In the Fixture Coordinate system Group box, Click on the Define button. The Fixture Coordinate system dialog box is displayed. In the Method group box select SOLIDWORKS coordinate system from the drop down list and click on Coordinate system1 in the Available Coordinate Systems group box. Coordinate System1 is automatically selected under the Coordinate Systems group box. Click on *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/CAMWorks Solids coordinate system entity.



Setup tab of Machine Dialog Box

4. Click the *Rotary Axis* tab.

   An axis is required in order to calculate the rotary angle. An axis can be defined by selecting a cylindrical face, a SOLIDWORKS/CAMWorks Solids axis entity or an axis relative to the Fixture Coordinate System.

5. In the Rotary axis is group box, select the *Y axis* option to define that the rotary axis is the same as the Y axis of the Fixture Coordinate System.

6. In the 0 degree position group box, click *XY plane* if it is not already selected.

   Note: If defining 5 axis indexing, the rotary 4 axis 0 degree face will not be defined.

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

Step 3: Selecting the Parts to be Machined

As you can see, the assembly includes many different models. Only four represent the parts to machine. The other models define table and fixtures. This next step identifies to CAMWorks which of the models are the actual parts to machine.

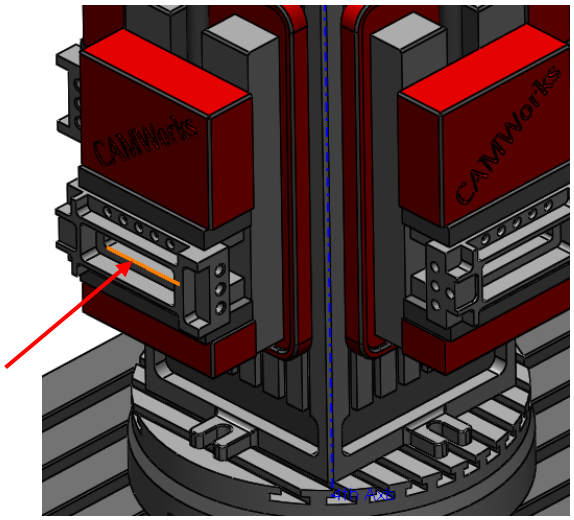
1. Double click *Part Manager* in the Feature tree.

   The *Manage Parts* dialog box will be displayed.

2. Select one of the part in the graphics area as shown in the image. This is a seed part.

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.



Select the part of Assembly in the graphics area

4. Click *OK* to exit the *Manage Parts* 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 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.

Step 4: Define the Stock and Extract Machinable Features

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.

Following are the steps to define the Stock:

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

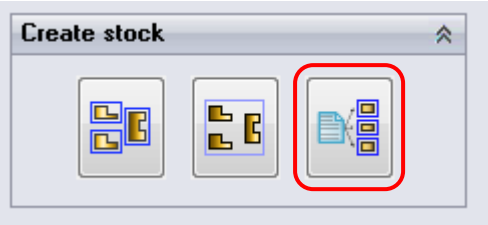
   OR

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

   The *Stock Manager* dialog box is displayed. This dialog box allows you to modify existing stock or create new stock for single parts and common stock for multiple parts.

   You want to add 0.1in material to the face of the part. This is done by entering a value for either positive or negative XYZ input boxes. Which input to change is determined by viewing the SOLIDWORKS/CAMWorks Solids main world coordinate system and the part that is highlighted with a wireframe bounding box.

   For example: If the face of the seed part that you selected is in the negative X direction, input 0.1 for the X- control and you will see the stock change. Once you have determined the correct direction and input 0.1in, click the *Apply Stock Definition to All Parts* button. This applies the 0.1in value to the stock definition of the 3 other part instances.



Click 'Apply Current Stock Definition to All Parts' button

2. Click *OK* to close the *Stock Manager* dialog box.

Extracting Machinable Features

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

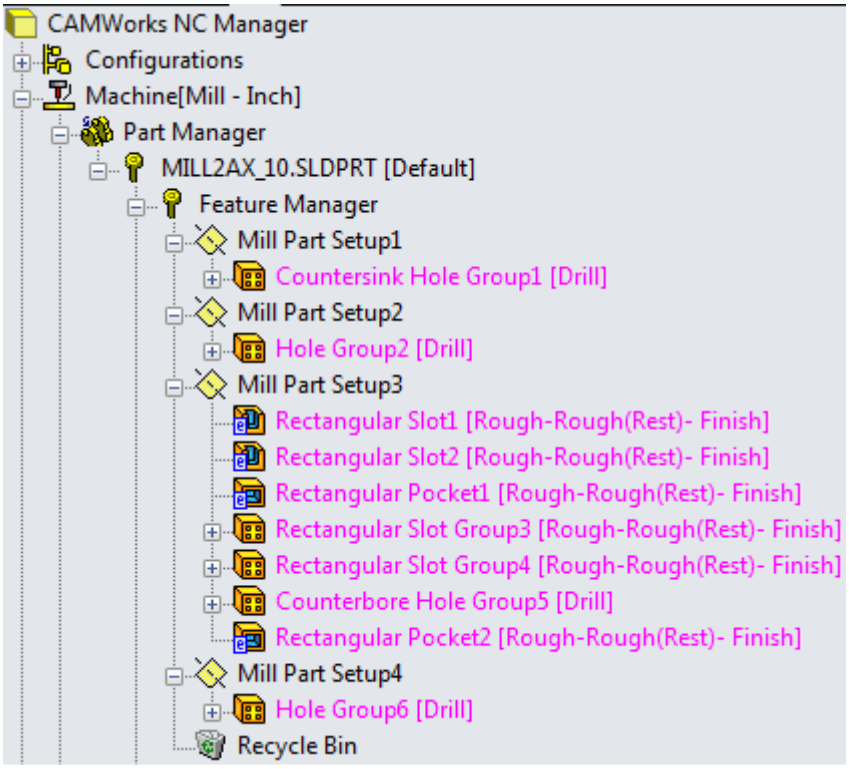
OR

Select the *Extract Machinable Features* command from the CAMWorks menu.

The CAMWorks Message Window is displayed. This window shows the progress of the process.

One of the powers of CAMWorks is that when you are machining multiple pieces of the same part, you insert features using AFR and/or IFR only once on the seed part and then features are automatically copied to the other part instances.

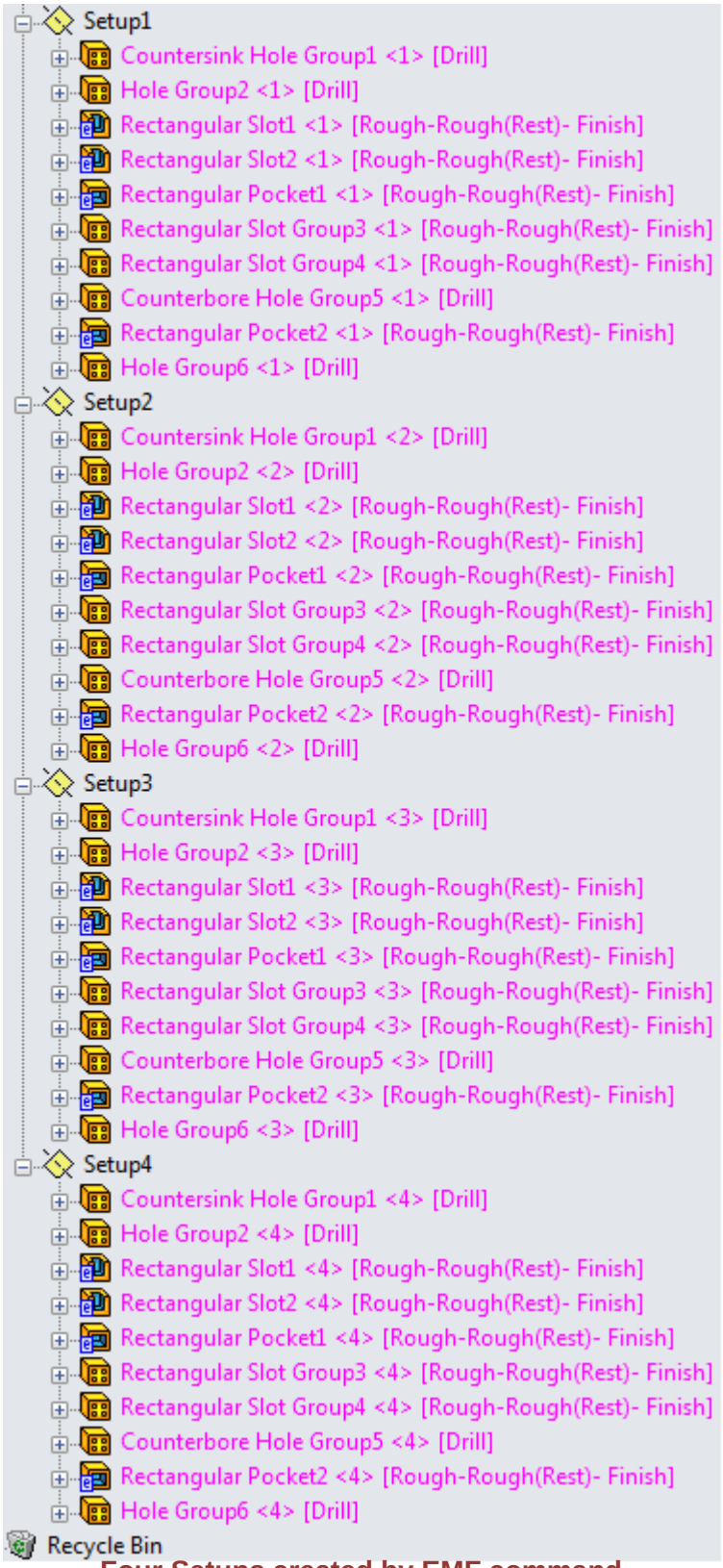
When AFR is run, features are recognized regardless of the machine's indexing capabilities and all Mill part setups and features are listed under the Feature Manager.



Mill Part Setups created by EMF command

Expand the Feature Manager item and observe that CAMWorks created 3 Mill Part Setups in order to machine the part.

Below the Stock Manager, CAMWorks creates Setups based on the parts in the Part Manager, the Mill Part Setups and features found for the seed part, and the indexing definition. In this tutorial, CAMWorks creates four Setups to machine the multiple instances of the part.

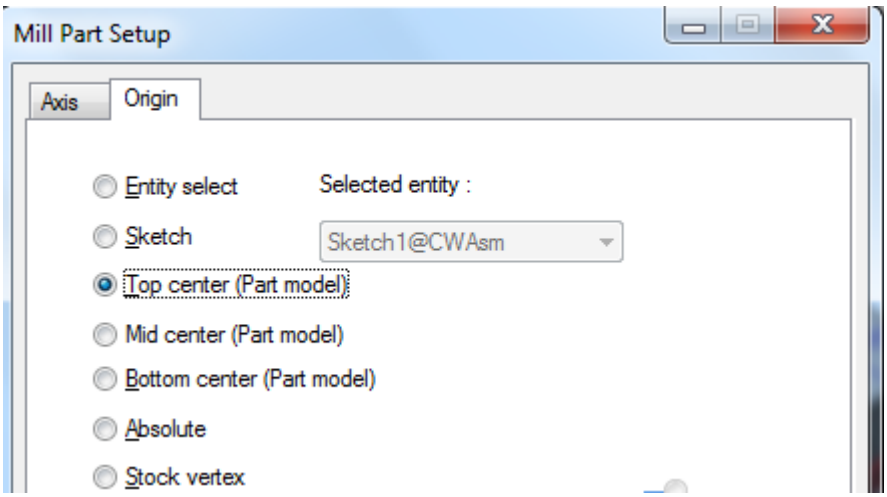


Four Setups created by EMF command

Step 5: Generating the Operation Plan

1. Click on the plus sign next to the *Feature Manager* to expand it.
2. Double click *Mill Part Setup1* in the tree.

   The *Mill Part Setup* dialog box will be displayed.
3. On the *Origin* tab, make sure the *Top center* is selected to set the location of the origin for this Mill Part Setup and then click *OK*.



Setting the origin for Mill Part Setup

4. Repeat steps **2** and **3** for *Mill Part Setup2* and *Mill Part Setup3* of the Feature Manager.
5. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   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.
6. Right click *Rough Mill1* under the *Setup1* of the Operation tree and select *Unlink Operation* on the context menu.

   The *Unlink Operations* dialog box lists the three other 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. You can also select an operation in the tree and unlink it from the parent.

   In this tutorial, the *Unlink* option won't be used.
7. Click *Cancel* to close the dialog box.

Step 6: Defining G-code Program Zero Location and Identify the Clamps and Fixtures

1. Double click *Setup1* in the Operation tree.

The *Setup Parameters* dialog box will be displayed.

2. On the *Origin* tab, make sure *Part Setup origin* is selected for the Output origin.

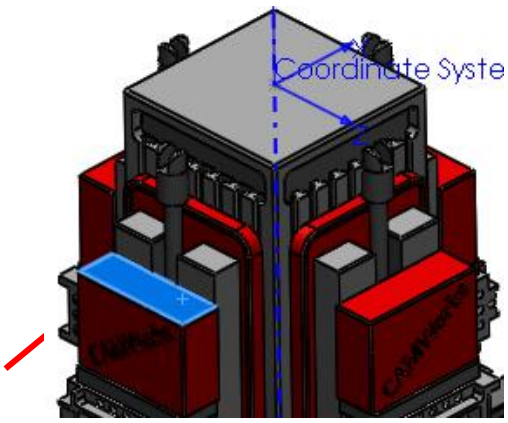
   This specifies that the origin for the G-code output is relative to the Part Setup origin instead of a global origin.

3. Click on the *Fixtures* tab.

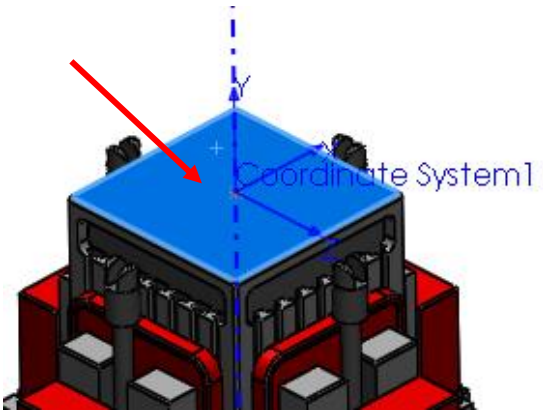
4. In the graphics area, pick the following:

   - Vise that holds the seed part
   - The tombstone
   - The rotary table
   - The machine table

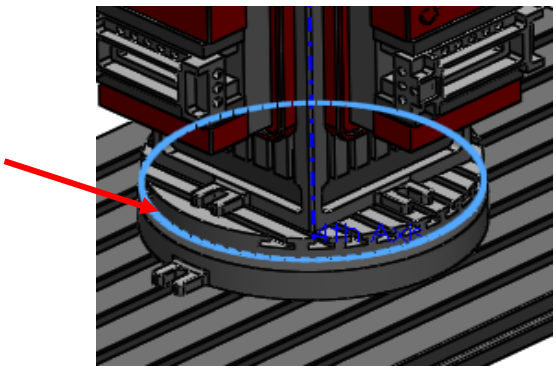
Did You Know: **For optimum performance, when modeling the fixtures, it is recommended that you omit any details that do not affect the machining process. Including details such as edge breaks, nuts, bolts and washers in the fixtures will consume a large amount of system memory and slow down the toolpath computation.**



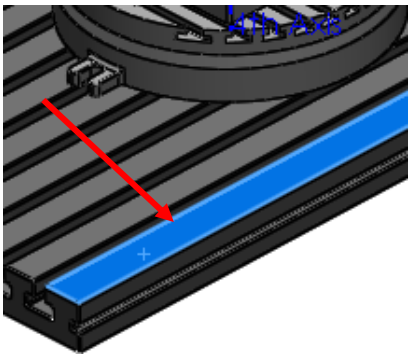
Select 'Vise Holding the Seed part'



Select 'Tombstone'



Select 'Rotary Table'



Select 'Machining Table'

5. In the dialog box, highlight the vise in the Feature list and click the *Add All Instances* button.

   All the vises in the assembly are listed and will display during simulation.

6. Do not check any of the Avoid check boxes.

   Although fixtures are 3 dimensional SOLIDWORKS/CAMWorks Solids parts, CAMWorks considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and contour 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. Set the *Avoid area type* to *Exact*.

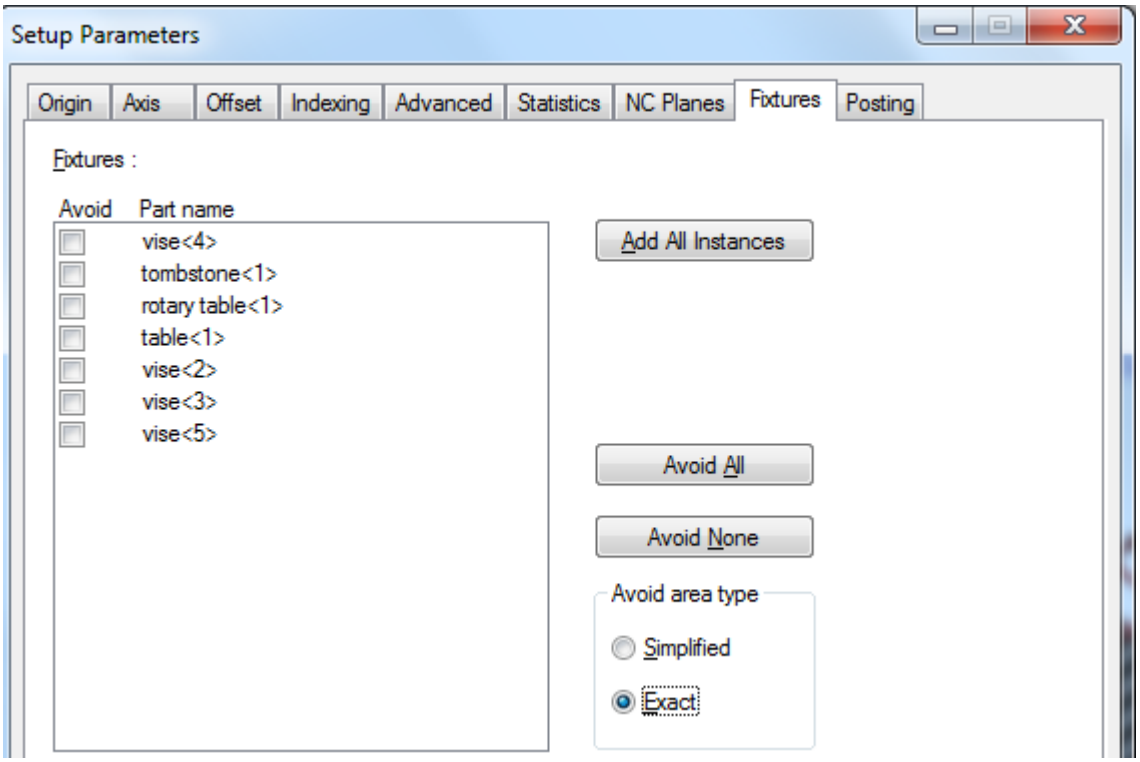
   When this option is selected, CAMWorks avoids the exact shape of the part.

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

9. If a warning message is displayed, click *No* to continue.

10. Double-click *Setup2* and change the Output Origin to *Part Setup Origin* in *Origin* tab of the *Setup Parameters* dialog box and click *OK* button.

11. Repeat above step for *Setup3* and *Setup4*.



Fixtures tab of the Setup Parameters Dialog Box

Step 7: Generate Toolpaths

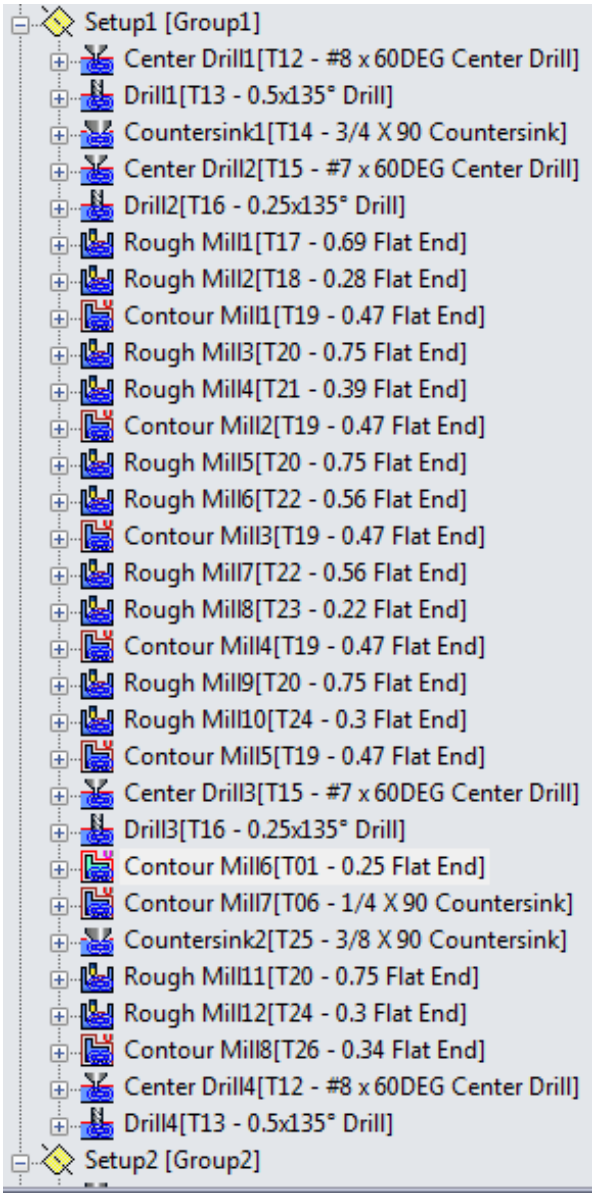
CAMWorks calculates toolpaths using the operation parameters to define how to machine each machinable feature. After generating toolpaths, you can sort the operations in a logical machining sequence and simulate the material removal.

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

2. Observe that toolpaths were generated for all operations.

CAMWorks was unable to compute a safe toolpath for this operation as the default tool selected for the operation gouges the part. A Flat End tool with suitable diameter needs to be selected from the tool crib.

3. Under *Setup1*, double-click on the *Contour Mill6* operation. The *Operation Parameters* dialog box will be displayed.
4. Click on the *Tool* tab and select *Tool Crib* page.
5. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
6. Click *Yes* to replace the corresponding holder too.
7. Click *OK* to apply the changes and close the dialog box.
8. Right-click on the Contour Mill operation again and select *Generate Toolpath* command on the context menu.
9. Observe that toolpaths were also generated for the Contour Mill toolpaths under *Setup2, Setup 3* and *Setup4* as those were linked operations.



Toolpaths for Setup1

Step 8: Simulate Toolpaths

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

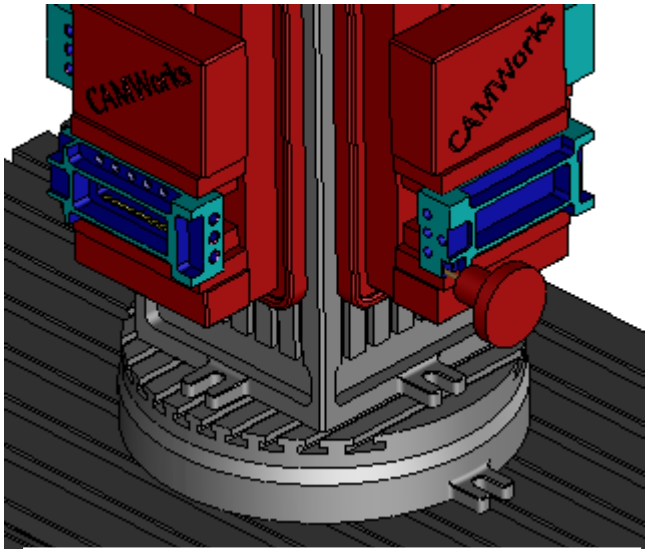
   OR

   Right click on *Machine [Mill-Inch]* in the Operation tree and select *Simulate Toolpath* on the context menu.

2. On the *Simulate* Toolbar, make sure the *Tool Mode* button is selected.
3. Set the following display options:
   - Stock: Shaded with Edges
   - Tool: Shaded with Edges
   - Tool Holder: Shaded with Edges

- Target part: No Display

4. Click the *Run* button to start the simulation.



Toolpath Simulation

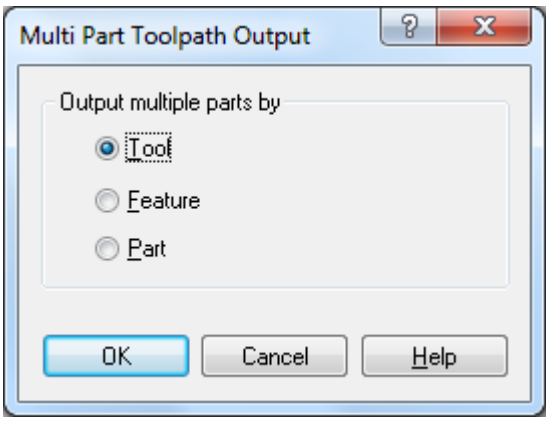
5. Click the *OK* button to exit the simulation mode and return to the SOLIDWORKS/CAMWorks Solids display.

Step 9: Change the Machining Sequence

1. Right click *Machine [Mill – Inch]* in the Operation tree and *select Multi Part Toolpath Output* from the context menu.

   The *Multi Part Toolpath Output* dialog box allows you to set the toolpath output order in Assembly mode based on the following:

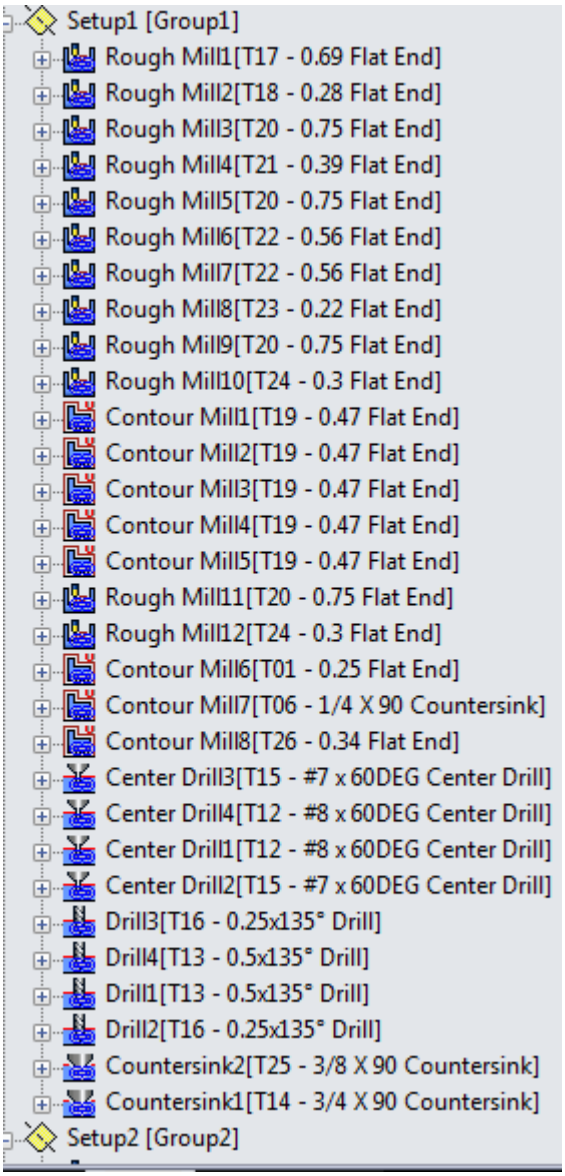
   - **Tool:** Starting with the first feature, for consecutive operations using the same tool, all toolpaths machined by the tool are processed (posted/simulated) on one part and the same sequence is repeated for other part instances.
   - **Feature:** Toolpath on each instance of a feature is processed before moving on to the next feature.
   - **Part:** All the toolpaths on a part are processed and then the next part is processed.



Multi Part Toolpath Output dialog box

2. For this tutorial, the Tool option will be retained. Click *OK* to close this dialog box.
3. Right click *Machine [Mill – Inch]* again and select *Sort Operations* from the context menu.
4. The options on the *Process* tab in the *Sort Operations* dialog box allow you to establish the rules for sorting operations. You can click the Help button to read an explanation of the options.

5. For this tutorial, in the *Sort* tab, select *Sort by Operation type*.
6. In the list of operations, drag and drop the operation types listed such that Rough Mill, Contour Mill, Center Drill and Drill operations top the list in that order.
7. Click *Apply* button and then click *OK*.
8. Run the simulation again to observe the changes.



Sorted Toolpaths for Setup1

Assembly 3

Topics covered in this tutorial:

- Selecting the Parts to be Machined on the Rear table
- Selecting the Parts to be Machined on the Front table
- Post Processing Toolpaths

Machining the Same Parts with Multiple Machine Tools

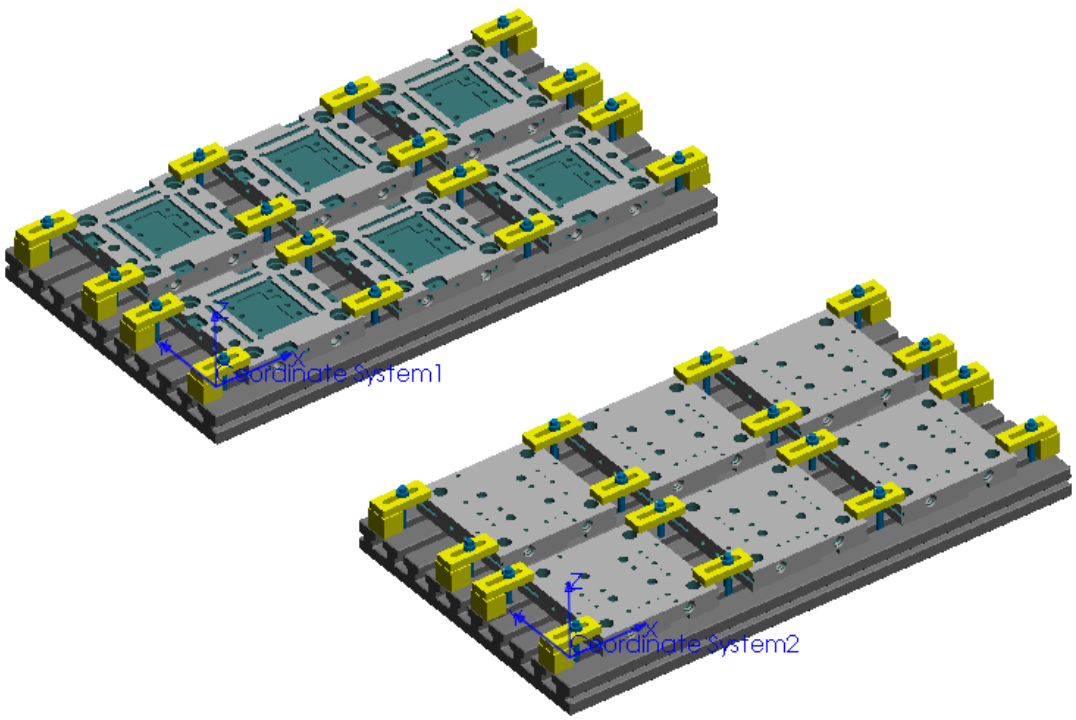
Using CAMWorks Assembly mode, the same parts can be machined using multiple machine tools. In this tutorial, you set up the first machine to cut the parts on the rear table, then copy the machine and program the parts on the front table.

Step 1: Open the Part

Open the part file **MILLASM_3.SLDASM** located in the following folder.

Drive:*CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Assemblies*.

The parts in this assembly document have been positioned on two tables to be machined using two machine tools.



MILLASM_3.SLDASM

Step 2: Defining the Machine

1. Click the *CAMWorks Feature Tree* tab.

   Set up the first machine tool to cut the parts on the rear table.

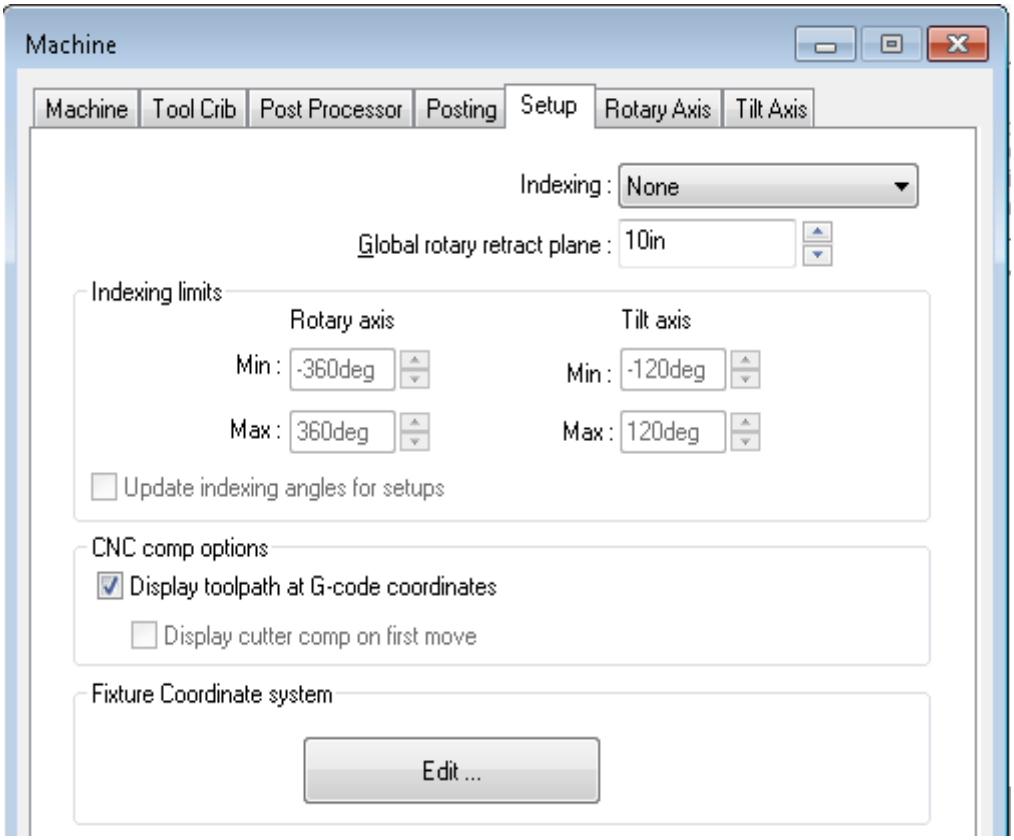
2. Double click the *Machine [Mill - Inch]* item in the Feature tree.

   The Machine tab of the Machine dialog box will be displayed.

3. On the Machine tab, *Mill - Inch* is the Active machine.
4. Click the *Tool Crib* tab of the Machine dialog box.
5. Ensure that the *Tool crib priority* option is unchecked.
6. In the Available tool cribs, make sure *Tool Crib 2(Inch)* is the Active tool crib.

   To select a particular tool crib as the Active tool crib, highlight it in the Available tool cribs list and then click the Select button.

7. Click the *Post Processor* tab in the Machine dialog box.
8. Make sure *M3AXIS-TUTORIAL* is selected as the Active post processor.
9. Click the *Setup* tab.
10. Select *None* for the Indexing option from the dropdown list.
11. Select *Coordinate System 1* in the Fixture Coordinate System list.
12. Click *OK* to close the dialog box.



Setup tab of Machine Dialog Box

Step 3: Selecting the Parts to be Machined on the Rear table

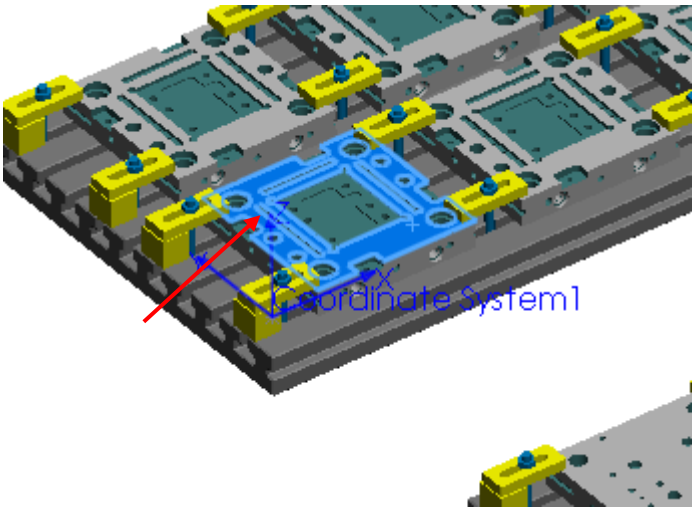
1. Double click *Part Manager* in the Feature tree.

   The *Manage Parts* dialog box will be displayed.

2. Pick the six parts on the rear table *(Coordinate System 1)* in order you want them to be machined.

   The first part that you pick is referred to as the seed part.

3. Click *OK* to exit the Manage Parts dialog box.



Select all the six part in the graphics area

Step 4: Define the Stock and Extract Machinable Features

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.

Following are the steps to define the Stock:

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

   OR

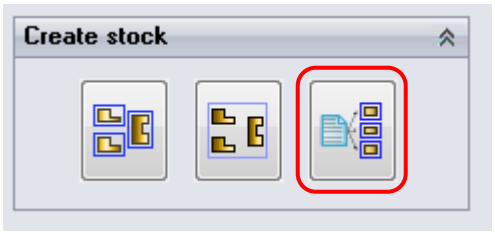
   Right click *Stock Manager* item in the Feature tree and select *Edit Definition* on the context menu.

   The *Stock Manager* dialog box will be displayed.

2. Change the +Z to **0.1in.**
3. In the Create Stock group box, select the *Apply Stock Definition to All Parts* button.

   The change is applied to the stock for all part instances.

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



Click 'Apply Current Stock Definition to All Parts' button

Extracting Machinable Features

Click the *Extract Machinable Features* button on the CAMWorks Command Manager/Workflow toolbar.

OR

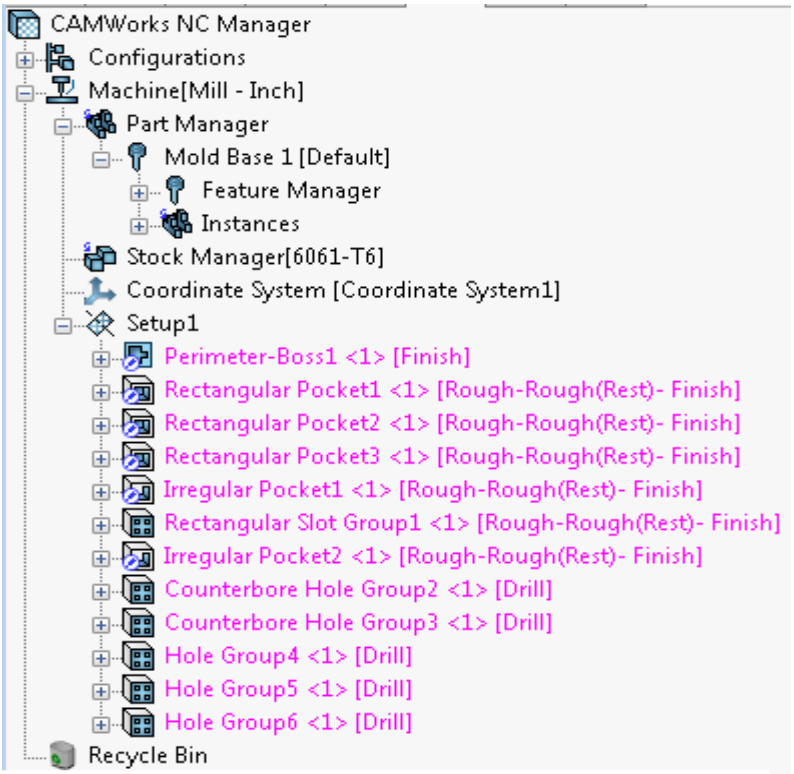
Select the *Extract Machinable Features* command from the CAMWorks menu.

OR

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

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.



Features recognized after executing the EMF

Step 5: Selecting the Parts to be Machined on the Front table

Set up the parts on the front table to be cut by a different machine tool.

1. Right click *Machine [Mill – Inch]* in the Feature tree and select *Copy Machine* on the context menu.

   A second machine will be listed at the bottom of the Feature tree.

2. Double click this new *Machine[Mill – Inch] item* in the Feature tree.

   The Machine dialog box will be displayed.

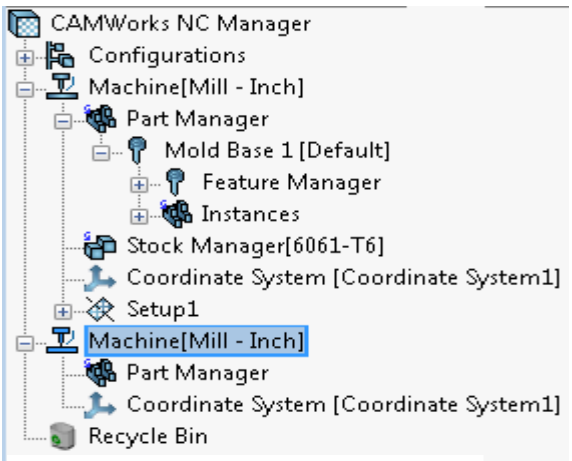
3. Click the *Setup* tab and select *None* for the Indexing option.

4. Select *Coordinate System 2* in the Fixture Coordinate System list.

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

6. Right click the *Part Manager* under the second machine in the Feature tree and select *Manage Parts* on the context menu.

   OR



Second Machine created

Double click the *Part Manager* under the second machine.

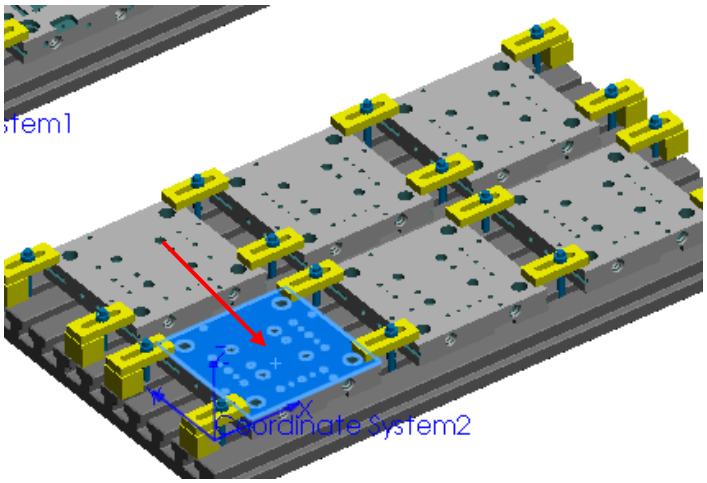
The *Manage Parts* dialog box will be displayed.

7. 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.

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

   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.



Select all the six part in the graphics area

Step 6: Generating an Operation Plan and Toolpaths

1. 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.

   The generated operations for *Setup1* and *Setup2* are listed in the Operation tree.

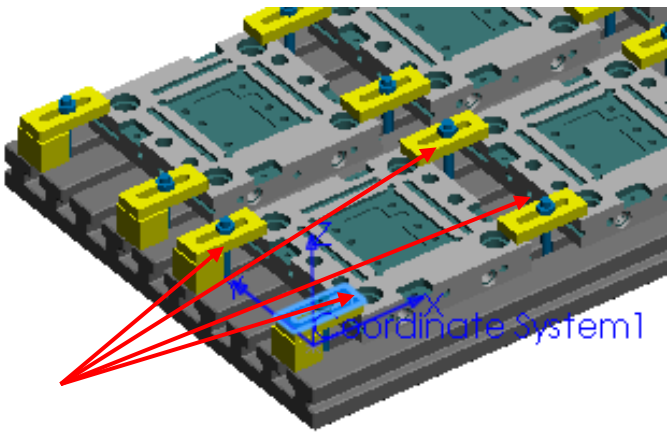
2. Double click *Setup1* in the Operation tree.

   The *Setup Parameters* dialog box is displayed.

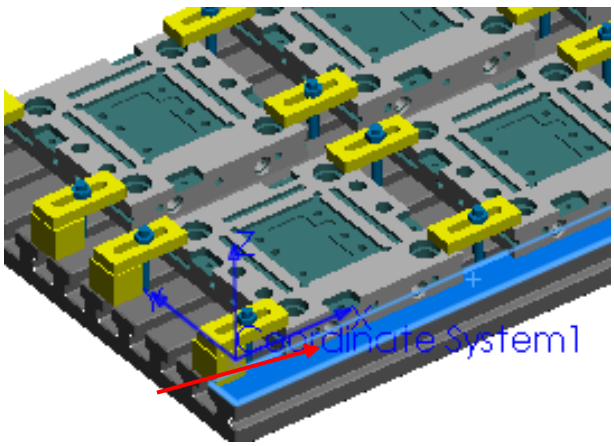
3. Click on the *Fixtures* tab.

4. In the graphics area, pick the four clamps holding the seed part on the rear machine.

   The part names will display in the *Fixtures* list of the tab.



Pick up the four clamps holding the seed part



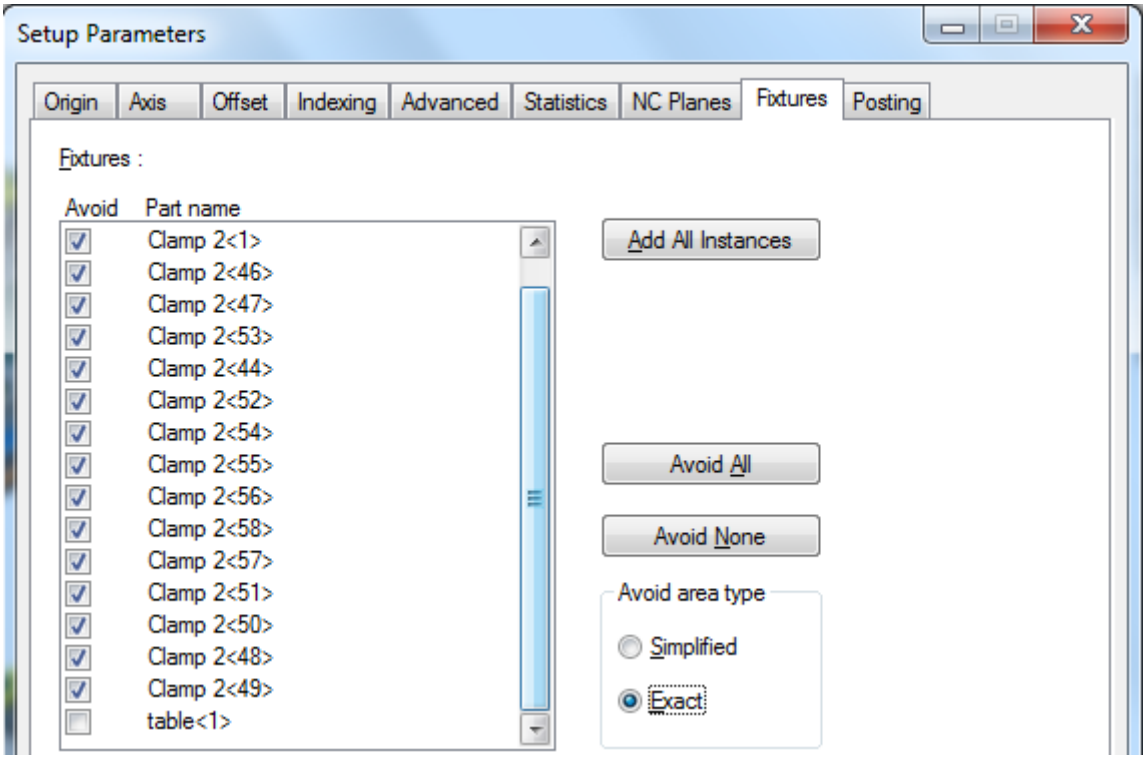
Select 'Machining table'

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. Click on the *Avoid All* button to check all the listed clamps.
8. On the rear machine, pick the table from the graphics area. Make sure the *Avoid* check box is *not* checked for the table.
9. 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.
10. Click *OK* to close the dialog box.



Fixtures tab of the Setup Parameters Dialog Box

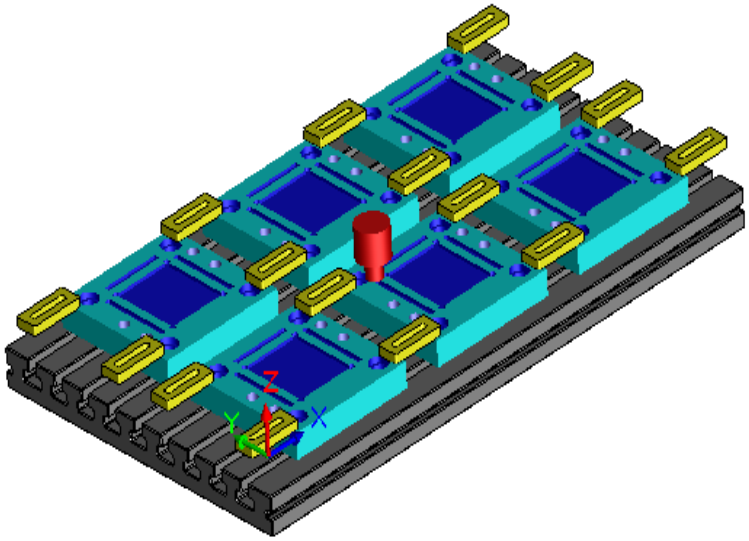
11. If a warning message is displayed, click *No* to continue.
12. Double click *Setup2* (for front machine) in the Operation tree.
13. Repeat the above procedure (Step 3 to Step 10) to pick the table and clamps for the front machine.
14. Click the *Generate Toolpath* button on the CAMWorks Command Manager.
15. Observe that under *Setup1*, toolpaths were not generated for *Contour Mill5 and Contour Mill6* for *Counterbore Hole Group1* feature and *Contour Mill1* operation for *Irregular Pocket1* feature.

    CAMWorks was unable to compute a safe toolpath for these operation as the default tool selected for the operation gouges the part. A Flat End tool with suitable diameter needs to be selected from the tool crib.

16. Under *Setup1*, double-click on the *Contour Mill1* operation for which toolpath wasn't generated. The *Operation Parameters* dialog box will be displayed.
17. Click on the *Tool* tab and select *Tool Crib* page.
18. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
19. Click *Yes* to replace the corresponding holder too.
20. Click *OK* to apply the changes and close the dialog box.
21. Right-click on the Contour Mill operation again and select *Generate Toolpath* command on the context menu. Observe that the toolpath is now generated.
22. Under *Setup1*, double-click on the *Rough Mill2*. The *Operation Parameters* dialog box will be displayed.
23. Click on the *Tool* tab and select *Tool Crib* page.
24. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
25. Click *Yes* to replace the corresponding holder too.
26. Click *OK* to apply the changes and close the dialog box.
27. Under *Setup1*, double-click on the *Contour Mill5* operation for which toolpath wasn't generated. The *Operation Parameters* dialog box will be displayed.
28. Repeat above sub-steps 17 to 21.
29. Similarly, assign the Flat End tool at Station No. 1 for the *Contour Mill13* operation under *Setup2* for which toolpaths weren't initially generated and then execute the command to generate toolpaths.
30. For *Contour Mill16* operation (for *Counterbore Hole Group8* feature) under *Setup2*, the Flat End Tool with the required diameter is not available in the active tool crib and will hence need to be added to the Tool Crib. Following are the steps:
    a. Double click on *Contour Mill16* operation to open the *Operation Parameters* dialog box.
    b. Click on the *Tool* tab and select the *Tool crib* page.
    c. Click on the *Add* button. The *Tool Select Filter* dialog box will be displayed.
    d. In this dialog box:
        i. Select *Flat End* for Tool type.
        ii. In the Filter by group box, check the *Diameter* option.
        iii. Leave the lower diameter range to **0in**.
        iv. Assign **0.25in** as the higher diameter range.
        v. Click the tab button. The list of tools will be updated.
        vi. Highlight the tool with **ID 33** and click the *OK* button.
    e. The selected tool will be added at the bottom of the active tool crib. In the Tool crib tab, highlight this tool and click the *Select* button.
    f. Click *Yes* to replace the corresponding holder too.
    g. Click *OK* to apply the changes and close the dialog box.
31. Right-click on *Contour Mill16* operation and select *Generate Toolpath command from the context menu.*

Step 7: Simulating Toolpaths

1. Right click *Setup 1* in the Operation tree and select *Simulate Toolpath* on the context 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 Toolpath Simulation toolbar.
4. Click the *OK* button to exit Simulation mode.



Running Toolpath Simulation for Setup1

5. Right click *Setup 2* in the Operation tree and select *Simulate Toolpath* on the context menu.
6.  Click the *Run* button on the *Toolpath Simulation* toolbar.
7. Click the *OK*  button to exit Simulation mode.

Step 8: Post Processing Toolpaths

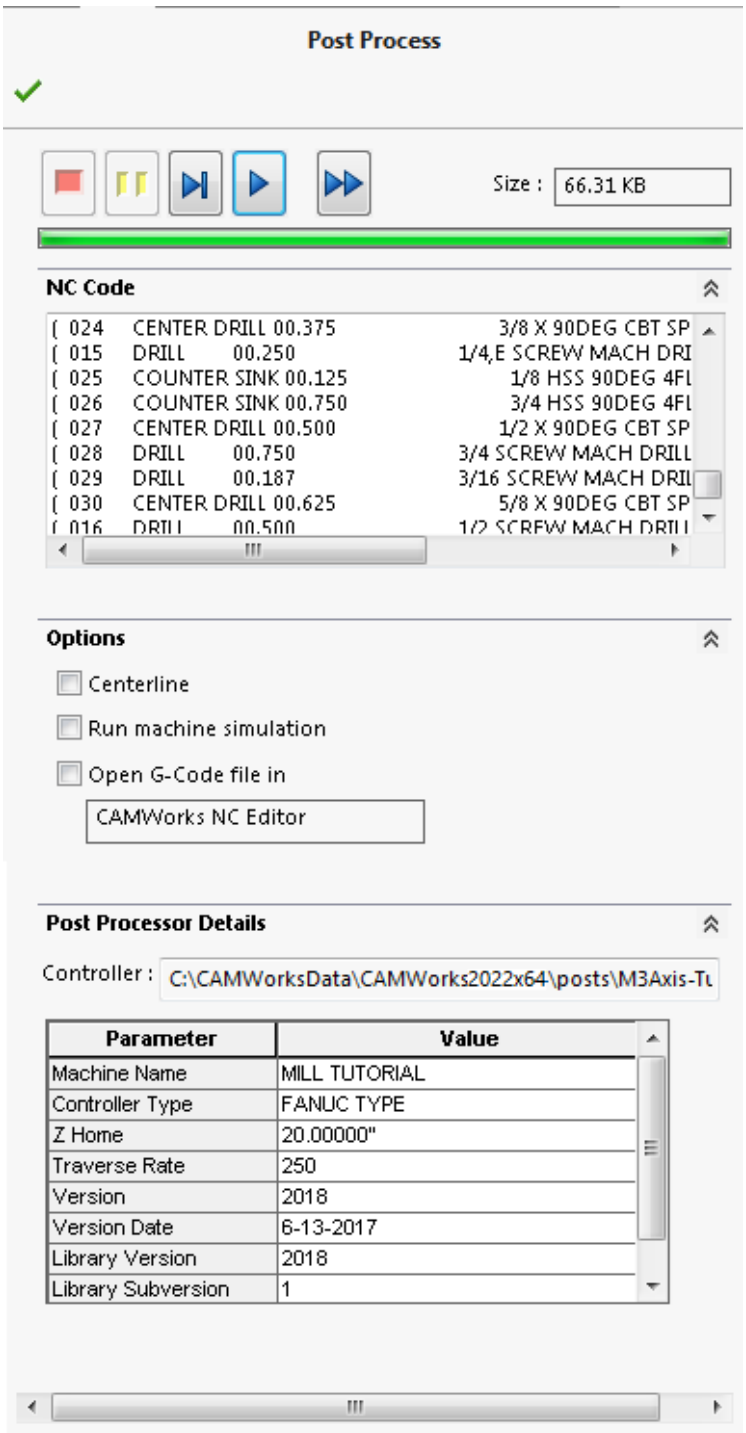
1. Right click the first Machine item in the Operation tree and select *Post Process* on the context menu.
2. The *Post Output File* dialog box is displayed. Browse to the folder where you wish to save the file.

    **If you are running CAMWorks in Demo mode, the Post Process functionality will be disabled.**
3. Type *Rear Machine* and click *Save* to save the file.

   CAMWorks generates the NC code for the parts on the rear table. The *Post Process* dialog box is displayed.
4.  Click on the *Step* button to view the NC code one by one.

5.  Click the *Play* button.

   This command generates the NC code. The generated NC code can be viewed in the *NC code* area of the dialog box.



Post Process dialog box (for Setup1)

6. After viewing the code,  click *OK* to close the dialog box.
7. Right click the second *Machine* item in the Operation tree and select *Post Process* on the context menu.

8. In the Post Output File dialog box, type *Front Machine* and click *Save*.

   The Post Process dialog box is displayed.

9.  Click the *Play* button to generate the code for the parts on the front table.

10. Click *OK* when the NC Code is completed.

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.

Assembly 4

Topics covered in this tutorial:

- Create the Assembly in SOLIDWORKS/CAMWorks Solid Part

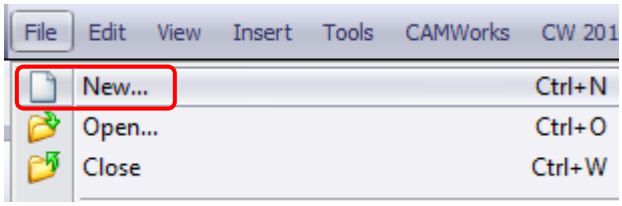
Simulating Castings

CAMWorks supports irregular shaped stock, such as castings. To define the stock as a casting in Assembly Mode, you can select a SOLIDWORKS/CAMWorks Solids part either graphically or from the SOLIDWORKS/CAMWorks Solids FeatureManager design tree. Any SOLIDWORKS/CAMWorks Solids part can be selected, even a part that is a part to machine.

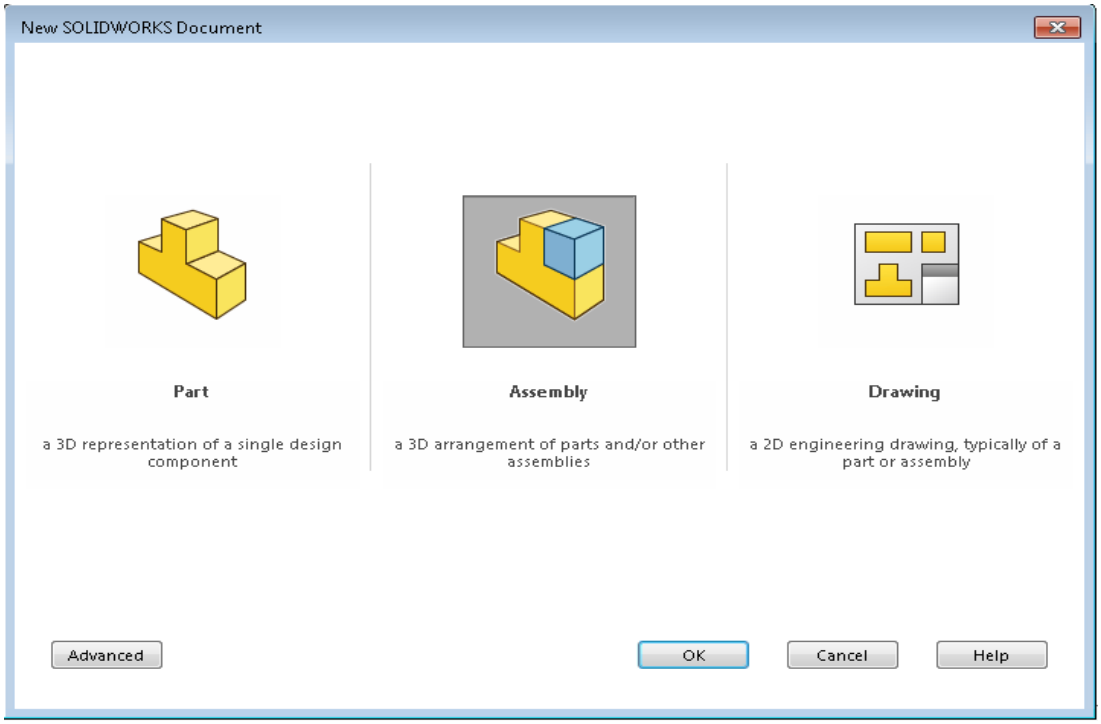
Step 1: Create the Assembly in SOLIDWORKS/CAMWorks Solid

1. Select the Inch Pound Second (IPS) Unit system in SOLIDWORKS/ CAMWorks Solids.
2. Click on *File* menu on the SOLIDWORKS menu bar and select *New* from the dropdown menu.

   The New SOLIDWORKS Document dialog box will be displayed.
3. Click on *Assembly* option and click *OK* to close the dialog box.



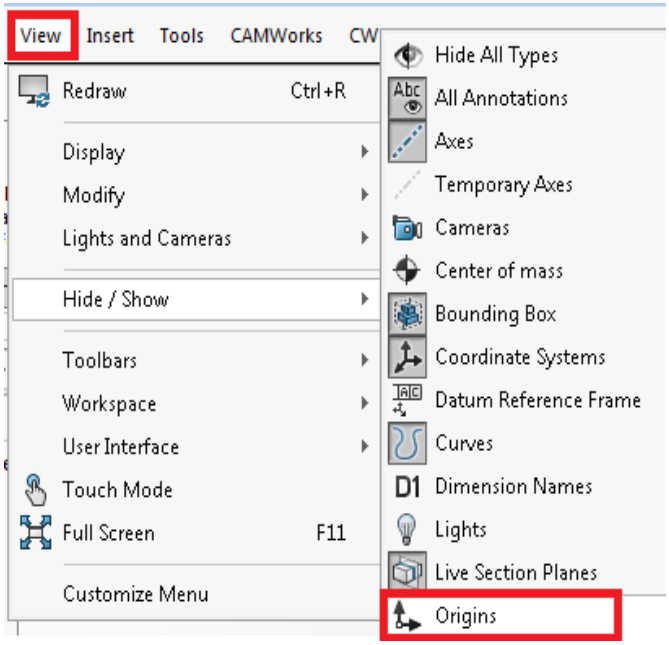
Select 'New' on the File menu



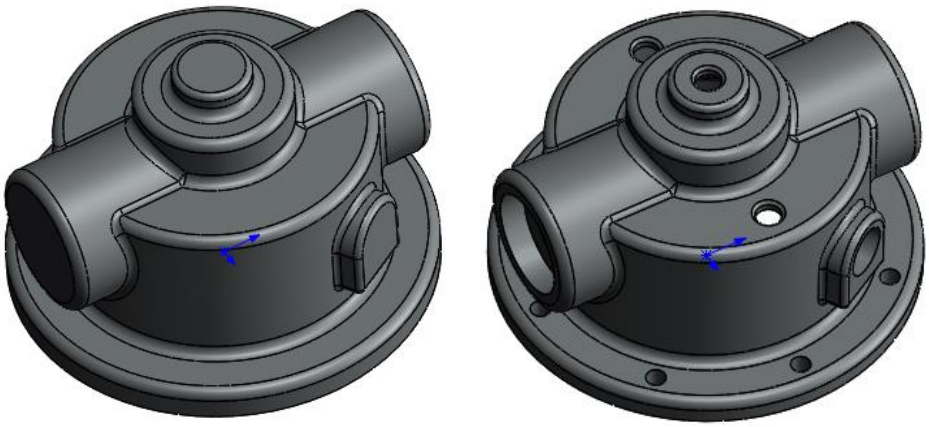
New SOLIDWORKS Document dialog box

4. The *Begin Assembly* dialog box will be displayed.
5. Click the *Keep Visible pin* in the *Begin Assembly* dialog box to keep it open.
6. On the *View* menu of the SOLIDWORKS/CAMWorks Solids menu bar, click *Origins* so the assembly origin can be seen. If needed, zoom out to see the assembly origin in the graphics area.
7. Click *Browse* button in the *Begin Assembly* dialog box and open **MILL2AX_14 As-Cast.SLDPRT** from the following folder location.

   Drive:*CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill*.
8. Position the mouse pointer on the origin and left click to position the part on the origin.

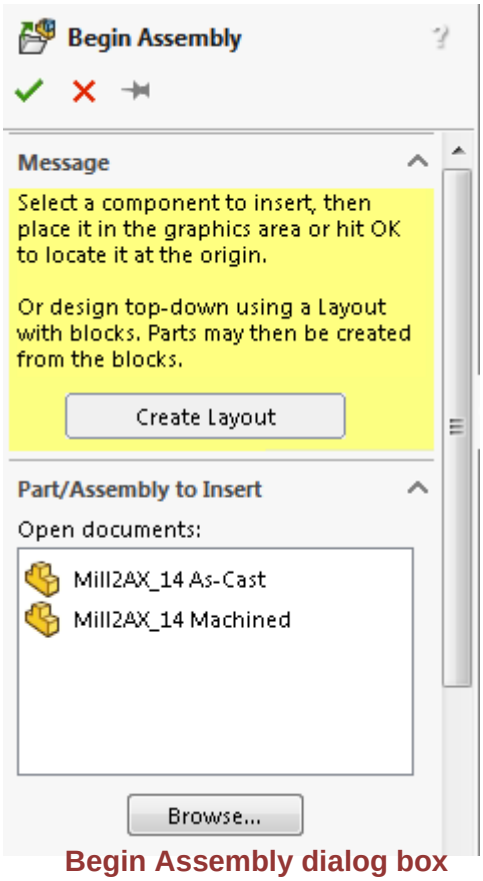


Select 'Origins' on View menu



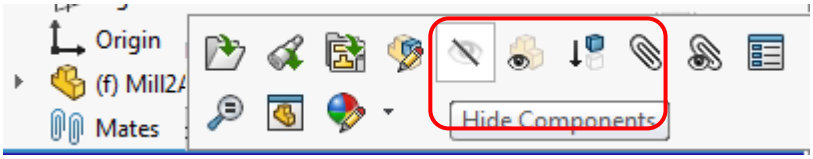
MILL2AX_14 As-Cast.sldprt MILL2AX_14 Machined.sldprt

9. Again click on the *Browse* and select **MILL2AX_14 Machined.SLDPRT**.
10. In the *Part/ Assembly to insert* group box of the *Begin Assembly* dialog box, highlight **MILL2AX_14 Machined.SLDPRT**. Position the pointer on the origin and left click to position the part on the same origin so that it is on top of the first part completely overlapping it.
11. Click *OK* to complete adding parts to the assembly.



Begin Assembly dialog box

12. Click the *FeatureManager design tree* tab.
13. Right click the first *Mill2AX_14 As-Cast* item in the tree and select *Hide components* on the context menu.



Select 'Hide Components' on the context menu

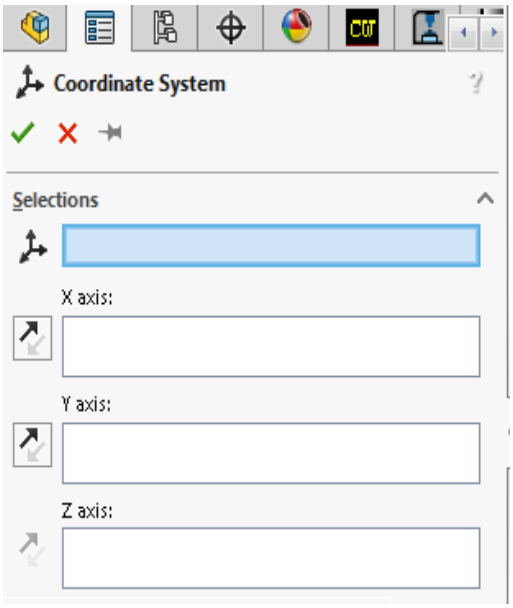
14. Click *Insert* on the SOLIDWORKS/CAMWorks Solids menu bar and select *Reference Geometry*, then *Coordinate System*.

    The *Coordinate System* dialog box will be displayed.

    You need to create a SOLIDWORKS/CAMWorks Solids coordinate system that will be used to define the CAMWorks Fixture Coordinate System.

15. Click *OK*.

    If no selection is made, the origin of the Coordinate System is the SOLIDWORKS/CAMWorks Solids assembly origin. In this tutorial, the default origin works well.



Select 'Hide Components' on the context menu

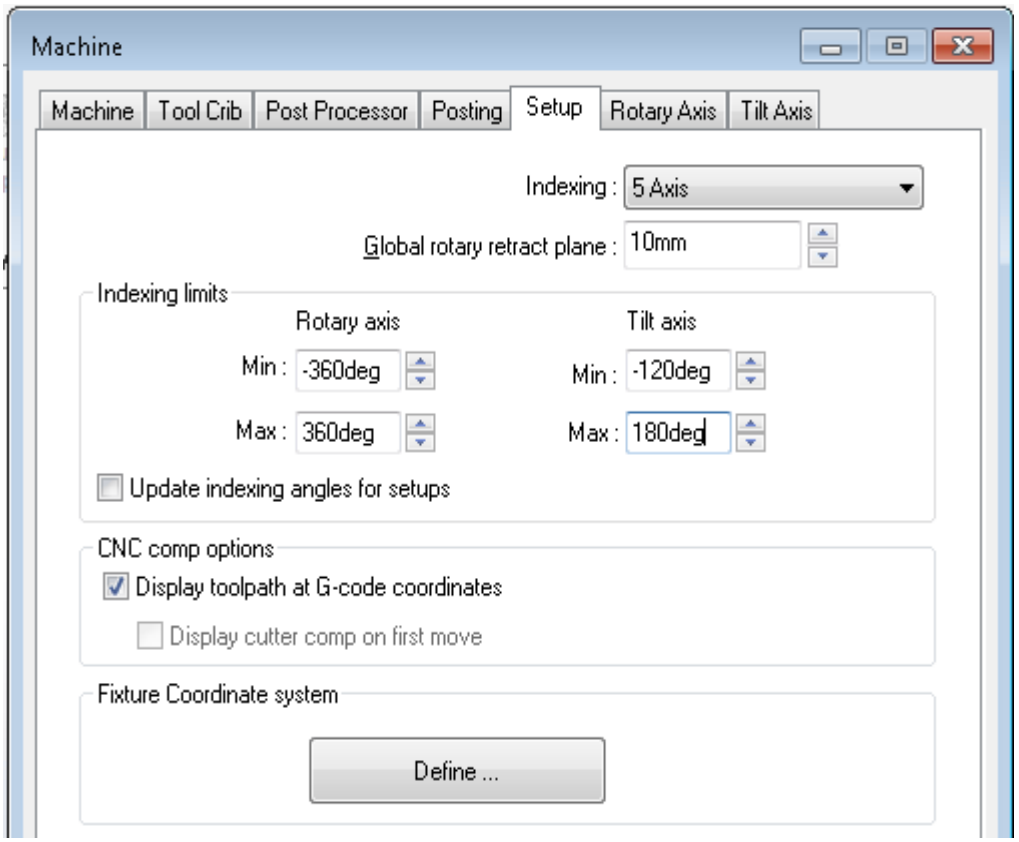
Step 2: Define the Machine, Fixture Coordinates System and Axes

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

   The Machine dialog box will be displayed.
3. On the Machine tab, highlight *Mill-Inch* and click *Select* button.
4. Click the *Tool Crib* tab of the Machine dialog box.
5. Uncheck the *Tool crib priority* option.
6. In the Available tool cribs, make sure *Tool Crib 2(Inch)* is the active tool crib.

   To select a particular tool crib as the active tool crib, highlight it in the Available tool cribs list and then click the Select button.
7. Click the *Post Processor* tab of the Machine dialog box.
8. Make sure *M5AXIS-TUTORIAL* is selected as the active post processor.
9. Click the *Setup tab* in the Machine dialog box.
   - Set the Indexing option to *5 Axis*.
   - In Indexing limits group box, set the Tilt Axis Max. limit to **180degrees**.

- Click on the Define button on the Fixture Coordinate System Group Box. Fixture Coordinate System Dialog Box will be displayed. In the Method Group box, select SOLIDWORKS Coordinate system from the drop down list. Coordinate System1 will be displayed in the Available Coordinate Systems group box. Select Coordinate System1 from the Available Coordinate Systems group box. This action will display Coordinate System1 in the selected coordinate system group box. Click on ok button.



Setup tab of Machine Dialog Box

10. Click the *Rotary Axis* tab.
11. In the *Rotary axis is* group box, select the *Z axis* option to define that the rotary axis is the same as the Z axis of the Fixture Coordinate System.

    Note: **When 5 axis indexing is selected, a 0 degree position is not required.**
12. Click the *Tilt Axis* tab.
13. In the *Tilt axis is* group box, select *Y axis* to define the tilt axis.
14. In the *0 degree position* group box, click *XY plane*, if not already selected.
15. Click *OK* to apply the changes and close the *Machine* dialog box.

Step 3: Selecting the Parts to be Machined

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

The *Manage Parts* dialog box will be displayed.

2. Pick the part model *Mill2AX_14 Machined* in the graphics area.

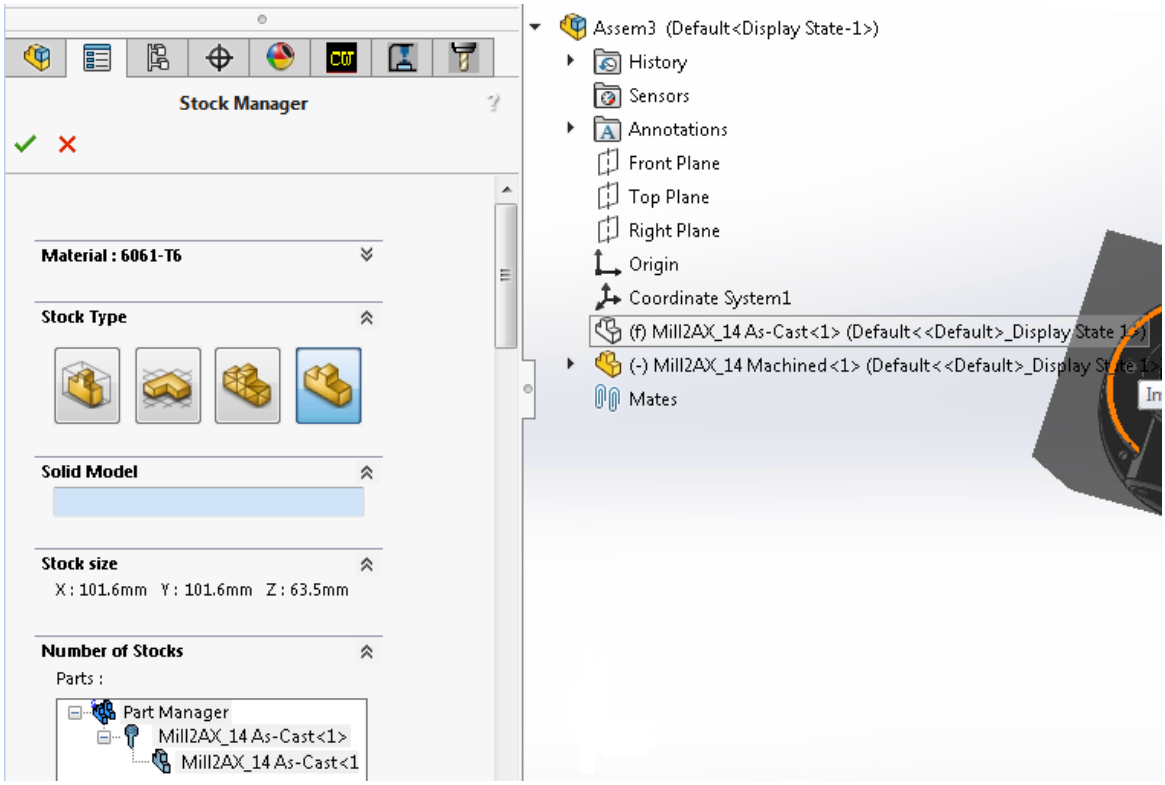
   The part name displays in the *Selected Parts* list.

3. Click *OK* to exit the *Manage Parts* dialog box.

Step 4: Define the Stock

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

   The *Stock Manager* dialog box will be displayed.



Select 'Mill2AX_14 As-Cast' in Graphics area

2. For the Stock type, select *SOLIDWORKS Part* as a Stock type.
3. Click on the plus sign next to the *FeatureManager Design tree* in the graphics area.
4. Select the *Mill2AX_14 As-Cast* part. This action will display the part name in the Solid Model field of the Stock Manager dialog box.
5. Click *OK* to close the Stock Manager dialog box.

Step 5: Extracting Machinable Features

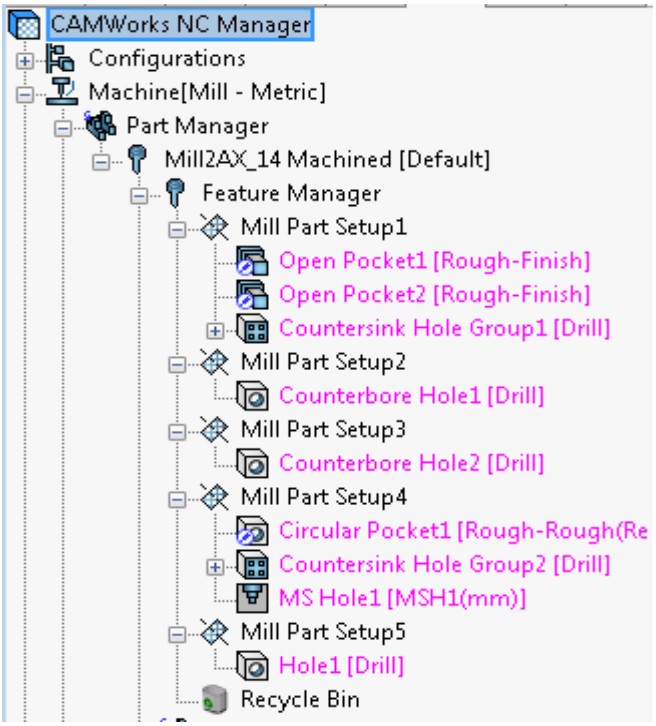
1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

2. Expand the *Feature Manager* item to show all the Mill Part Setups and features found by AFR.
3. Right click *Open Pocket1* feature under *Mill Part Setup1* and select *Delete* on the context menu. Similarly delete *Open Pocket2* feature.

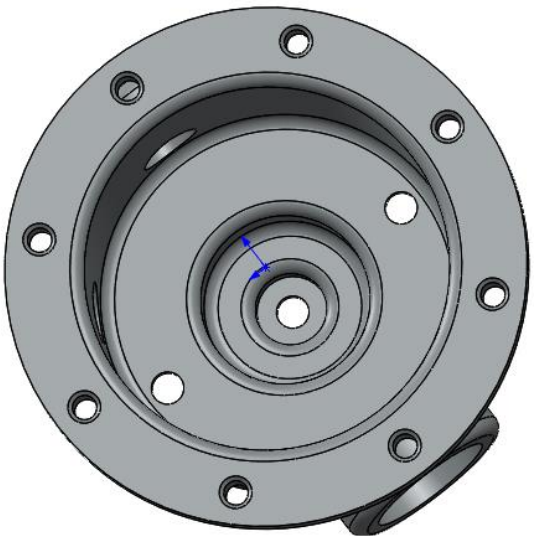
   These features are already inside the casting and hence need not be machined.
4. Right click *MS Hole* feature under *Mill Part Setup4* and select *Delete* on the context menu.
5. This feature is inside the casting and will not be machined.
6. Click *Yes* to confirm the deletion.
7. In the graphics area, rotate the part so you can see the bottom face of the part.
8. Right click *Mill Part Setup4* in the Feature tree and select *2.5 Axis Feature* on context menu.

   The *2.5 Axis Feature: Select Entities* dialog box will be displayed. In this dialog box:

   i. Change the Feature Type to *Face Feature*.
   ii. Pick the bottom face of the part (the face with the circular holes pattern). *CW ASM Face-18* will be displayed in the *Selected Entities* group box.
   iii. Click *End Condition*.
   iv. Set the Strategy set to *Finish*.
   v. Set the End Condition to *Up to Stock*.
   vi. Remove the check mark from the *Use stock extents* option so that only the circular face is machined.
9. Click *OK* to complete the addition of the new Face Feature.

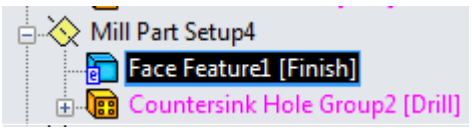


Features recognized after executing the EMF



Bottom face of the part

10. In the lower section of the Feature tree, drag and drop the *Face Feature* onto the top of the *Mill Part Setup 4* so that it is the first feature machined on that Setup.



Drag and Drop Face Feature on top of Setup4

Step 6: Generating an Operation Plan and Toolpaths

1. 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.

   The generated operations for *Setup* are listed in the Operation tree.

2. Click the *SOLIDWORKS/CAMWorks Solids FeatureManager design tree* tab.

3. Right click *Mill2AX-14 As-Cast* in the tree and select *Show components* on the context menu.

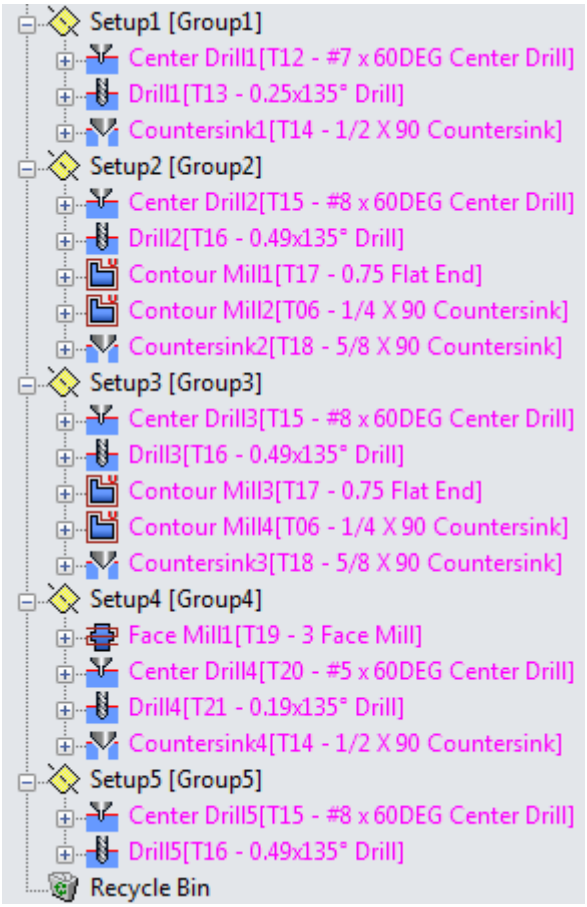
4. Right click *Mill2AX-14 As-Cast* again and select *Change transparency* on the context menu.

5. Click the *CAMWorks Operation Tree* tab.

6. Click *Generate Toolpath* on the CAMWorks Command Manager/Workflow toolbar.

   OR

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



Generated operations

7. Under *Setup2*, double-click on the *Contour Mill1* operation for which toolpath wasn't generated. The *Operation Parameters* dialog box will be displayed.
8. Click on the *Tool* tab and select *Tool Crib* page.
9. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
10. Click *Yes* to replace the corresponding holder too.
11. Click *OK* to apply the changes and close the dialog box.
12. Right-click on the *Contour Mill* operation again and select *Generate Toolpath* command on the context menu.
13. Under *Setup3*, double-click on the *Contour Mill3* operation for which toolpath wasn't generated.
14. On the NC tab, change the Rapid and Clearance planes to *Top of Feature.*
15. Click on the *Tool* tab and select *Tool Crib* page.
16. Highlight the *Flat End* tool at Station No. 1 which has a diameter of 0.25 inch and click the *Select* button.
17. Click *Yes* to replace the corresponding holder too.

18. Click *OK* to apply the changes and close the dialog box.
19. Right-click on the Contour Mill operation again and select *Generate Toolpath* command on the context menu. Observe that the toolpath is now generated.

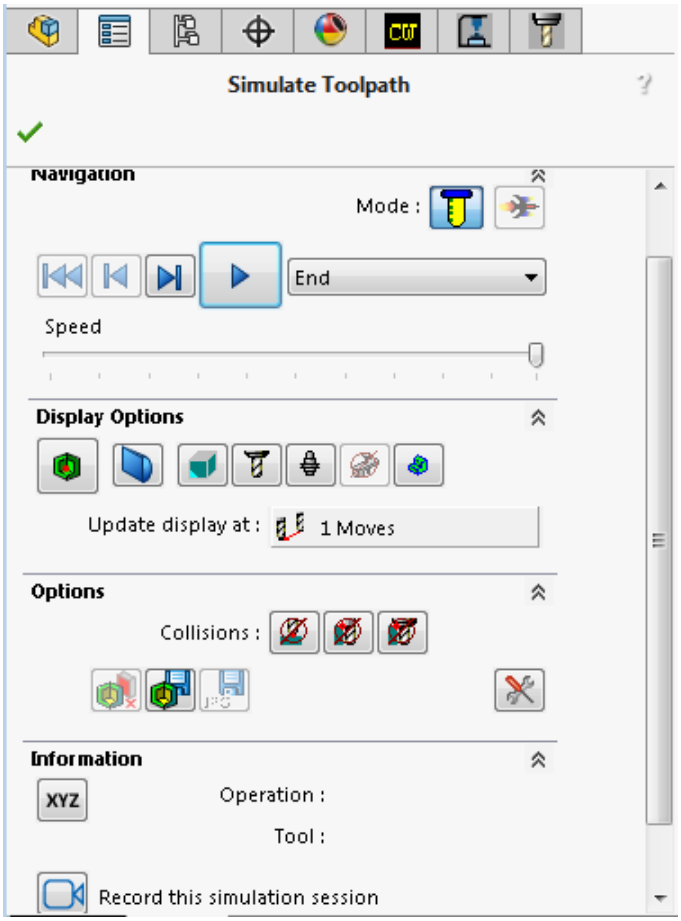
Step 7: Simulating Toolpaths

1. Click *Simulate Toolpath* on the CAMWorks Command Manager/Workflow toolbar.

   OR

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

   The *Toolpath Simulation* toolbar will be displayed.



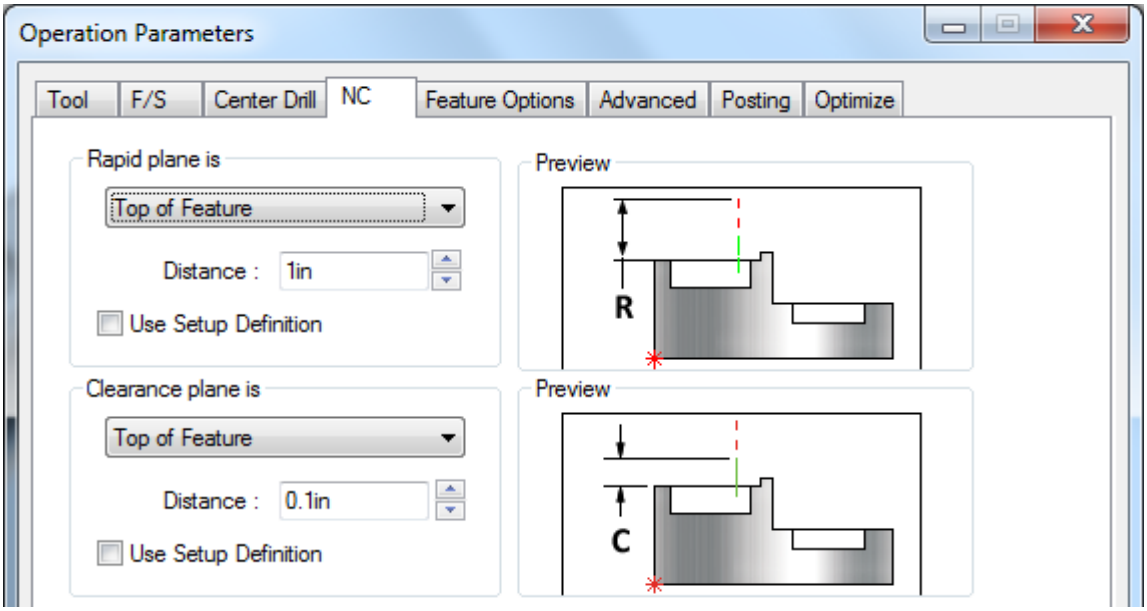
Toolpath Simulation Dialog Box

2. Optionally, change the display of the stock, tool, tool holder and fixtures (shaded, wireframe, translucent, no display).
3.  Click the *Single Step* button and observe the tool motion.

   When the Center Drill and a Drill operations in *Setup5* are simulated, notice that the tool begins to feed into the part from a long distance away from the part itself. 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 efficient Rapid and Clearance planes.

4. Click the *OK* button to exit Simulation mode.
5. Double click the *Center Drill5* operation under *Setup 5*.
6. On the NC tab, set the Rapid and Clearance plane to *Top of Feature*, then click *OK*.



NC tab on the Operation Parameters dialog box

7. Click *Generate Toolpath* on the CAMWorks Command Manager to regenerate the toolpath.
8. Double click the *Drill* operation under *Setup5*.
9. On the NC tab, change the Rapid and Clearance planes to *Top of Feature.*
10. Click *OK* to close the dialog box.
11. Click *Generate Toolpath* on the CAMWorks Command Manager to regenerate the toolpath.
12. Right click *Setup5* and select *Step Thru Toolpath* on the context menu.

    The *Step through Toolpath* dialog box will be displayed.
13. Click the *Single Step* button and confirm that the tool now rapids to within a reasonable distance from the part before feeding into the hole.
14. Click *OK* to close the dialog box.