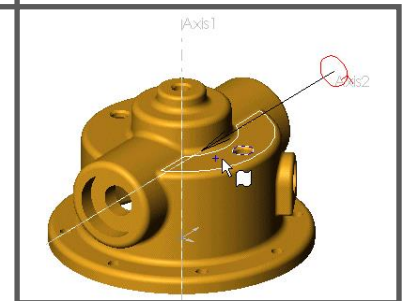
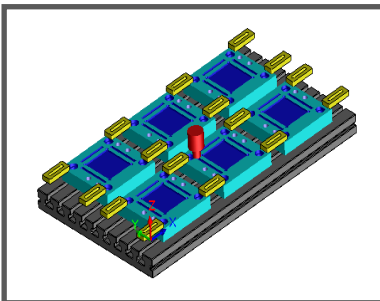
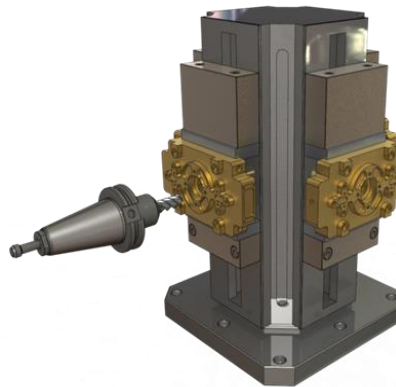
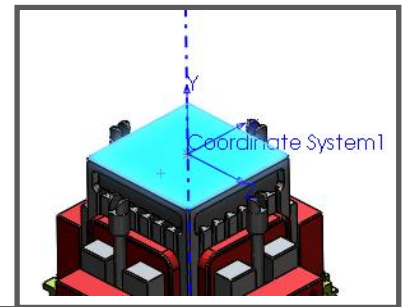
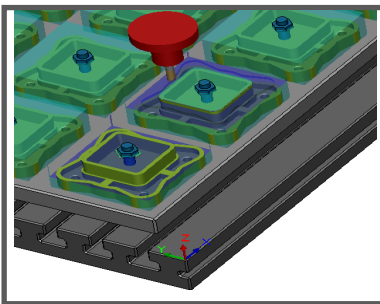


Mill Assemblies Tutorial



SOLIDWORKS CAM
2026

Copyright

© 1995-2025, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

Disclaimers

The information and the software application discussed in this document are subject to change without prior notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Trademarks

- SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.
- CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.
- FeatureWorks is a registered trademark of Geometric Americas, Inc.
- Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.
- Other brand or product names are trademarks or registered trademarks of their respective holders.



Product Name: SOLIDWORKS CAM 2026

TABLE OF CONTENTS

Assembly Machining with SOLIDWORKS CAM	5
Overview of Tutorials in this User guide	5
Basics: SOLIDWORKS CAM Machining Trees	6
The SOLIDWORKS CAM Machining Trees	6
Machining Tree Types	6
Expanding the Tree View Area	6
Items in the SOLIDWORKS CAM Feature Tree	7
Basics: Accessing SOLIDWORKS CAM Commands	9
SOLIDWORKS CAM Menu	9
SOLIDWORKS CAM Command Manager	10
SOLIDWORKS CAM NC Manager	10
Basics: SOLIDWORKS CAM Settings	11
SOLIDWORKS CAM Options Dialog Box	12
Basics: Generating NC Code in Assembly Mode	13
Steps involved in generating NC code using SOLIDWORKS CAM	13
Assembly Tutorial 1	14
Step 1: Open the Assembly file to be machined in SOLIDWORKS	14
Step 2: Switch to the SOLIDWORKS CAM Feature Tree	15
Step 3: Select the Machine, Tool Crib and Post Processor	15
Step 4: Assign the Fixture Coordinate System	18
Step 5: Selecting the Parts to be Machined	19
Step 6: Defining the Stock	21
Step 7: SOLIDWORKS CAM Options dialog box to control SOLIDWORKS CAM Settings	22
Step 8: Defining Machinable Features	24
Step 9: Sorting Part Instances to Determine Machining Order	27
Step 10: Generating the Operation Plan and Adjusting Operation Parameters	28
Step 11: Defining G-code Program Zero Location	31
Step 12: Identifying Fixtures and Clamps	33
Step 13: Generating Toolpaths and Sorting Operations	34
Step 14: Simulate Toolpaths	36
Assembly Tutorial 2	38
Multi-Plane Machining in Assembly Mode	38
Step 1: Open the Part	38
Step 2: Defining the Machine	39
Define the Machine	39
Editing the Tool Crib	39
Selecting the Post Processor	43
Setting the Setup and Rotary Axis	43
Step 3: Selecting the Parts to be Machined	44
Step 4: Define the Stock and Extract Machinable Features	45
Extracting Machinable Features	46
Step 5: Generating the Operation Plan	47
Step 6: Defining G-code Program Zero Location and Identify the Clamps and Fixtures	48
Step 7: Generate Toolpaths	50
Step 8: Simulate Toolpaths	51
Step 9: Change the Machining Sequence	52

Assembly Tutorial 3	54
Machining the Same Parts with Multiple Machine Tools	54
Step 1: Open the Part	54
Step 2: Defining the Machine	55
Step 3: Selecting the Parts to be Machined on the Rear table	56
Step 4: Define the Stock and Extract Machinable Features	56
Extracting Machinable Features	57
Step 5: Selecting the Parts to be Machined on the Front table	57
Step 6: Generating an Operation Plan and Toolpaths	59
Step 7: Simulating Toolpaths	61
Step 8: Post Processing Toolpaths	62
Assembly Tutorial 4	64
Simulating Castings	64
Step 1: Create the Assembly in SOLIDWORKS	64
Step 2: Define the Machine, Fixture Coordinates System and Axes	67
Step 3: Selecting the Parts to be Machined	69
Step 4: Define the Stock	69
Step 5: Extracting Machinable Features	69
Step 6: Generating an Operation Plan and Toolpaths	71
Step 7: Simulating Toolpaths	72
Legal Notices	74



ASSEMBLY MACHINING WITH SOLIDWORKS CAM

This User Guide document provides you with an opportunity to learn how to generate Mill toolpaths and NC code in Assembly mode for production machining using the SOLIDWORKS CAM application. The information applies to running SOLIDWORKS CAM in SOLIDWORKS.

Using SOLIDWORKS Assembly mode, SOLIDWORKS CAM allows you to:

- Position multiple copies of a part in an assembly document and machine the parts with SOLIDWORKS CAM.
- Generate long code or subroutine output to machine each part.
- Store the SOLIDWORKS CAM 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.

IMPORTANT!

SOLIDWORKS CAM 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.

Note: 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.

Overview of Tutorials in this User guide

The following series of tutorials in this User Guide document will show you how to generate toolpaths for a SOLIDWORKS assembly model. When you define the operations and toolpaths for these tutorial assemblies, you will follow steps which are explained only in brief. This is done to show you the basics of generating toolpaths from start to finish without getting into the details.

For detailed Help, refer the [SOLIDWORKS Context based Help file](#).

Location of Sample Assembly Files pre-installed with SOLIDWORKS CAM:

C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 202x\CAM Examples\Tutorial_Parts \Assemblies.

Note: In order to give you a general understanding of how to use SOLIDWORKS CAM, you will work sample assembly files which come pre-installed when you install SOLIDWORKS CAM. These tutorial example files are assembly files which were previously modeled in SOLIDWORKS.

The tutorials in this User Guide are intended to show you how to use SOLIDWORKS CAM and may not correspond to actual machining practices used at your facility.


Basics: SOLIDWORKS CAM Machining Trees

The SOLIDWORKS CAM Machining Trees

The SOLIDWORKS CAM Machining trees and associated SOLIDWORKS CAM commands will be displayed within SOLIDWORKS only when the following conditions are fulfilled:

- SOLIDWORKS CAM application is loaded as an active Add-In in SOLIDWORKS.
- The solid part or assembly file to be machined using SOLIDWORKS CAM is currently open in SOLIDWORKS.

In the *SOLIDWORKS* Tree view area on the left hand side adjacent to the graphics area, different tabs are provided to access the SOLIDWORKS trees and the SOLIDWORKS CAM trees.




For example, the SOLIDWORKS tree labelled FeatureManager design tree tab  displays the list of the features, sketches, planes and axes related to the part/assembly.

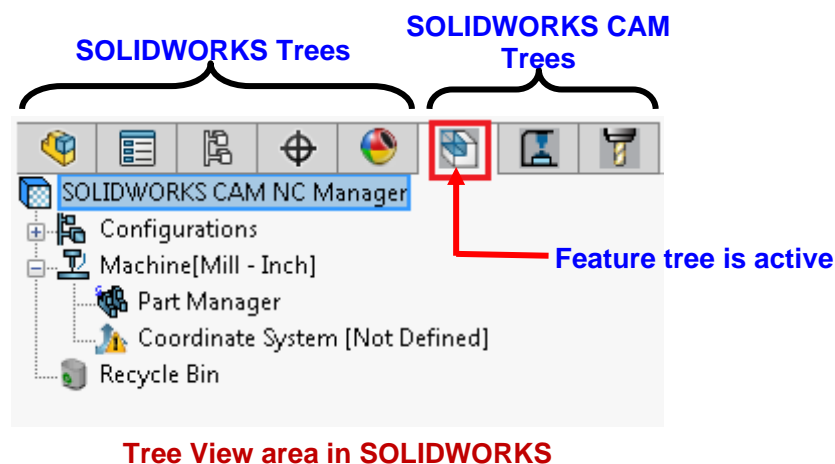
To use SOLIDWORKS CAM, you need to switch from the SOLIDWORKS trees to the SOLIDWORKS CAM trees.

Note: In case the Tree view area is not continuously visible, push the  *Pin* button to ensure continuous visibility of the Tree area.





Machining Tree Types

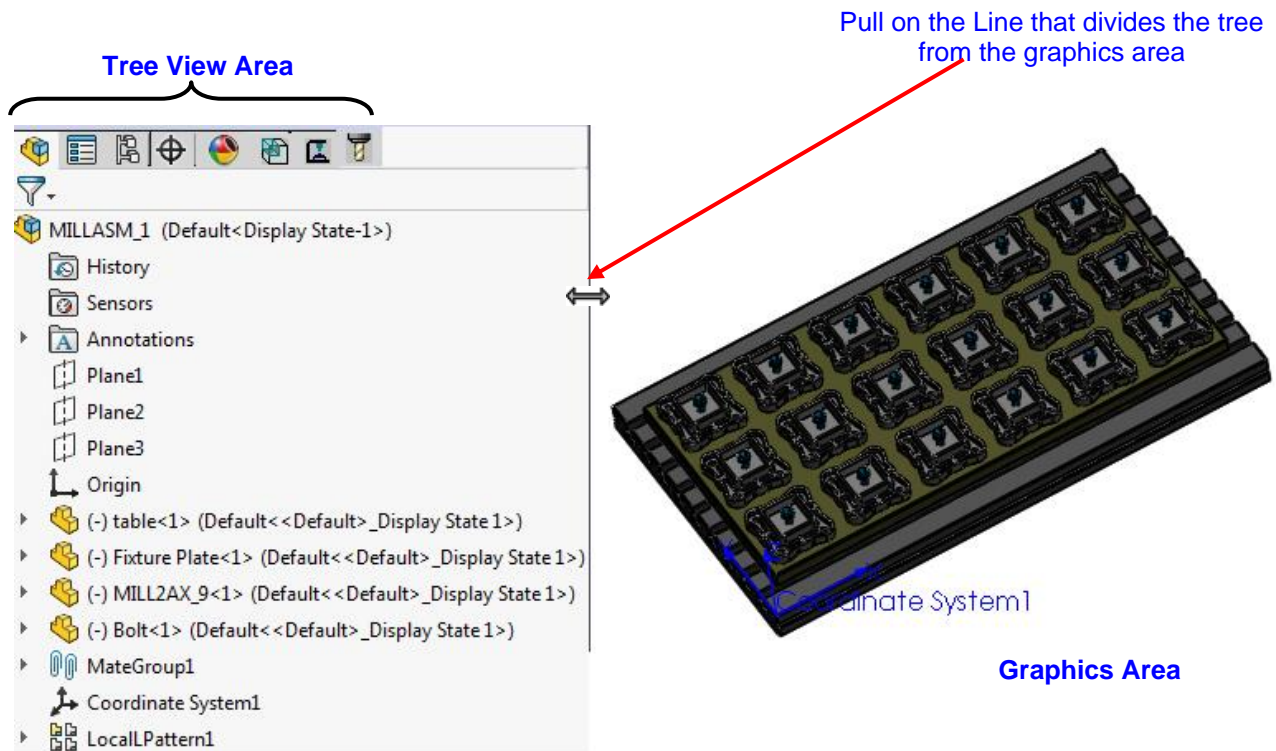
Following are the SOLIDWORKS CAM Trees available in the Tree View area:

- Feature Tree 
- Operation Tree 
- Tool tree 



Expanding the Tree View Area

If the SOLIDWORKS CAM tree tabs , ,  are not visible in the Tree View area, you can horizontally expand the size of the Tree area. 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 SOLIDWORKS CAM tree tabs display.



Drag on the Line that divides the Tree View Area from the Graphics Area

Items in the SOLIDWORKS CAM Feature Tree

The SOLIDWORKS CAM Machining trees provide an outline view of the machining information for the model.

When an assembly file is open within SOLIDWORKS, the SOLIDWORKS CAM Feature tree will initially display only the SOLIDWORKS CAM NC Manager, Configurations, Part Manager, Machine and Recycle Bin items. As you follow the [steps to generate an NC program](#), this tree will expand to include Mill Part Setups and machinable features.

- **Configurations**

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

- **Machine**

When you use SOLIDWORKS CAM to machine your own parts, the first step is to select the Machine tool that you want to use to machine the part. 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. Machine tools are set up in the Technology Database of SOLIDWORKS CAM. Before using SOLIDWORKS CAM to machine your parts, make sure you define the machine tools available in your facility in the Technology Database.



For information on how to define details of the Machine tools available at your facility in the Technology database of SOLIDWORKS CAM, refer the Technology Database Tutorial.

Important parameters of the machine definition include:

- **Machine type:** The machine type defines the machinable feature set that can be recognized automatically and defined interactively.

The icon display in the Feature tree/Operation tree identifies the current machine type:



Mill Machine



Turn Machine

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.



- **Part Manager**

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

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



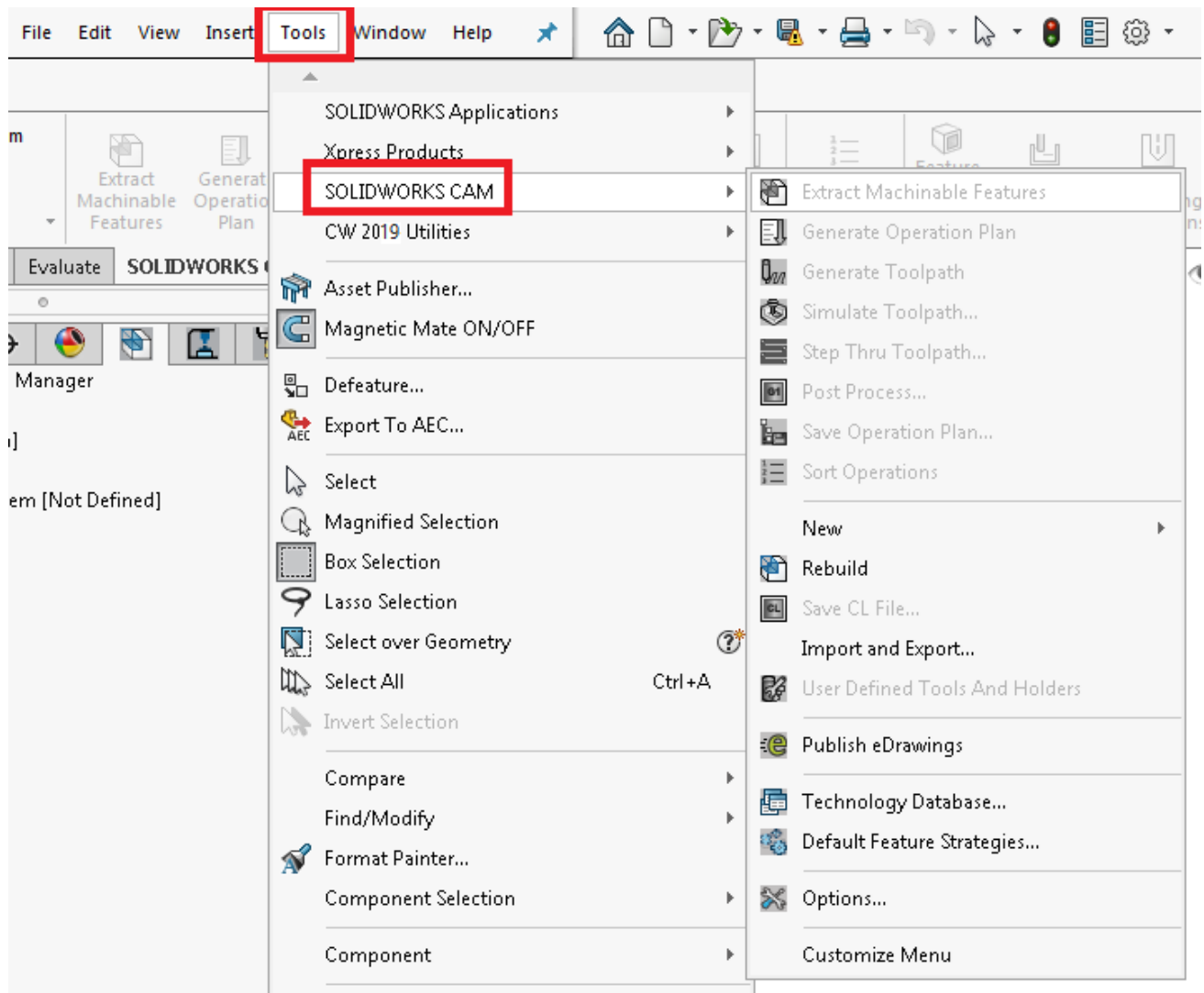
- **Recycle Bin**

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

Basics: Accessing SOLIDWORKS CAM Commands

SOLIDWORKS CAM Menu

Ensure that SOLIDWORKS CAM is loaded as an active Add-In within SOLIDWORKS. Click on the *Tools* menu on the SOLIDWORKS menu bar and select *SOLIDWORKS CAM* from the dropdown menu options. The cascading menu displayed lists all the major SOLIDWORKS CAM menu options.



Accessing the SOLIDWORKS CAM Menu

At the first glance, a few of these menu options might appear in a disabled state. The enabling of these command menu command options is dependent on the execution of specific commands as these commands are context-sensitive.

For example, the *Generate Toolpath* command will be enabled only after the *Generate Operation Plan* command is executed. Similarly, the *Simulate Toolpath*, *Step thru Toolpath* and *Post Process* commands will be enabled only after the *Generate Toolpath* command has been executed.

SOLIDWORKS CAM Command Manager

If **SOLIDWORKS CAM** is loaded as an active Add-In within **SOLIDWORKS**, then the **SOLIDWORKS CAM Command Manager** tab will be one of the Command Managers available in the Command Manager area of **SOLIDWORKS** when you create or open a solid part or assembly within the **SOLIDWORKS** application. This Command Manager provides access to the all the major **SOLIDWORKS CAM** commands. All the commands available in the [SOLIDWORKS CAM menu](#) will also be available in the **SOLIDWORKS CAM Command Manager**.

At the first glance, a few of the commands available on this *Command Manager* might appear in a disabled state. The enabling of these command menu command options is dependent on the execution of specific commands as these commands are context-sensitive.



Commands on the **SOLIDWORKS CAM Command Manager**

Note: Detailed context-based Help is provided for the **SOLIDWORKS CAM** commands and user interfaces. To view this context-based help documentation, click on the Help button within the specific **SOLIDWORKS CAM** user interface dialog box.

Clicking on the Help button  on the **SOLIDWORKS CAM Command Manager** displays the context-based Help.

SOLIDWORKS CAM NC Manager

The **SOLIDWORKS CAM NC Manager** item is present in both the *Feature tree* and the *Operation tree*. Right click on the **SOLIDWORKS CAM NC Manager** in the tree. A list of executable commands will be displayed on the context menu. The commands displayed within this menu depend on the context. These command options in the context menu provide access to a variety of commands.

All the commands executed from the [SOLIDWORKS CAM Command Manager](#) can also be alternatively accessed from the RMB context menu of the **SOLIDWORKS CAM NC Manager**.

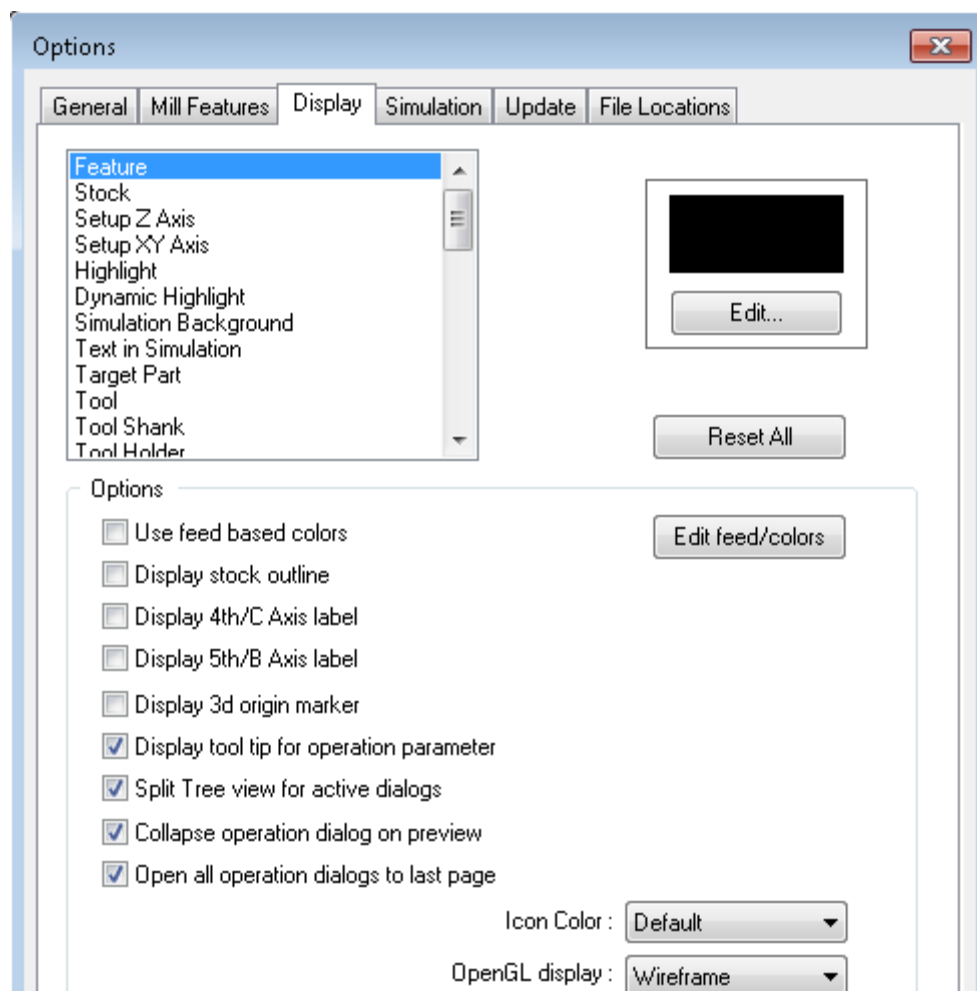
Note: Some of the commands options displayed in the context menu for **SOLIDWORKS CAM NC Manager** item in the *Feature tree* is different from those in the *Operation tree*.

Basics: SOLIDWORKS CAM Settings

SOLIDWORKS CAM provides a mechanism to customize settings and options related to the following to suit your needs:

- Saving data
- Feature recognition
- Display settings
- Simulation
- Updation
- Rebuild of SOLIDWORKS CAM data
- File locations for various files generated



These customizations can be done in the SOLIDWORKS CAM Options dialog box.



SOLIDWORKS CAM Options Dialog Box


SOLIDWORKS CAM Options Dialog Box

To open the SOLIDWORKS CAM Options dialog box, execute any one of the following commands:

- Click the *Options* command button  on the SOLIDWORKS CAM Command Manager.
- On the SOLIDWORKS menu bar, click on the *Tools* menu and select *SOLIDWORKS CAM* from the dropdown menu. Select *Options* command  from the cascading menu.

Executing the Options command displays the *Options* dialog box. Use the various tabs within this dialog box to customize the settings to suit your requirements.

After making desired changes to the settings, click on the *OK* button of the dialog box to apply the changes and close the dialog box.

Clicking on the  or *Cancel* button of the dialog box will cause it to close without any of the changes being applied.

Note: For detailed understanding of the various settings available in these tabs, click on the corresponding *Help* button within those tabs. Each setting is explained in detail within the context-based Help.

Basics: Generating NC Code in Assembly Mode

SOLIDWORKS CAM 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 that comprises the assembly.

Steps involved in generating NC code using SOLIDWORKS CAM

The following steps are used to generate Mill toolpaths and NC code in Assembly mode for the Assembly file to be machined:

1. Ensure that SOLIDWORKS CAM is loaded as an active add-In in SOLIDWORKS.
2. Model the components (part, clamps, vises, fixtures) and create the assembly document (.sldasm) in SOLIDWORKS.
OR
Open the assembly file to be machined in SOLIDWORKS.
3. Click on the SOLIDWORKS CAM Feature tree.
4. 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).
5. Select the parts to be machined.
6. Define the Stock (separate or common).
7. Extract Machinable Features and interactively insert the features at Part Setup Level.
8. Generate the operation plan and adjust operations parameters.
9. Define G-code program zero location (Part Setup Origin or Setup Origin).
10. Identify features and clamps.
11. Generate toolpaths.
12. Post process the toolpaths.

IMPORTANT!

All the tutorials given in this User Guide document broadly follow the above format for illustrating the steps to generate toolpath and NC code.

Assembly Tutorial 1

Topics covered in this tutorial:

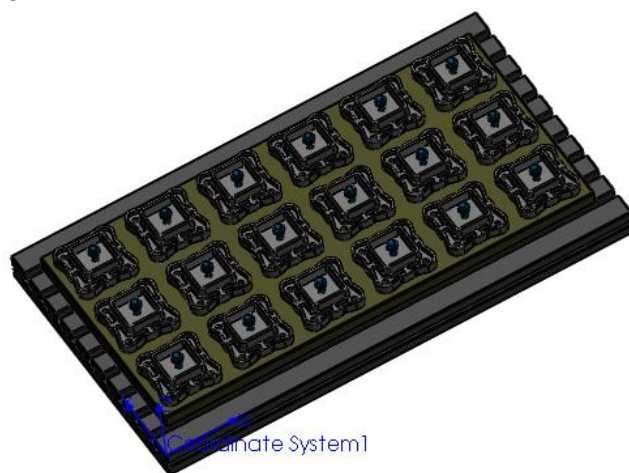
- [Defining the Machine](#)
- [Selecting the Parts to be Machined](#)
- [Defining the Stock](#)
- Changing Default Settings using [Options dialog box](#)
- Using the SOLIDWORKS CAM [Feature Tree](#) and [Operation Tree](#)
- [Defining Machinable Features](#)
- [Sorting Part Instances to Determine Machining Order](#)
- [Generating Operation Plan](#)
- [Defining G-code Program Zero Location](#)
- [Identifying Fixtures and Clamps](#)
- [Generating Toolpaths and Sorting Operations](#)
- [Simulate Toolpaths](#)

Step 1: Open the Assembly file to be machined in SOLIDWORKS

A part is a solid that is created with SOLIDWORKS or imported into SOLIDWORKS from another CAD system via an IGES, Parasolid, SAT file, etc. An assembly is a group of associated solid parts. This tutorial uses an existing sample SOLIDWORKS assembly file (*.sldasm) that come pre-installed with SOLIDWORKS CAM.


Open the assembly file named **MILLASM_1.SLDASM** located in the following folder:

*C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 202x\CAM Examples\
Tutorial_Parts \Assemblies*



MILLASM_1.SLDASM

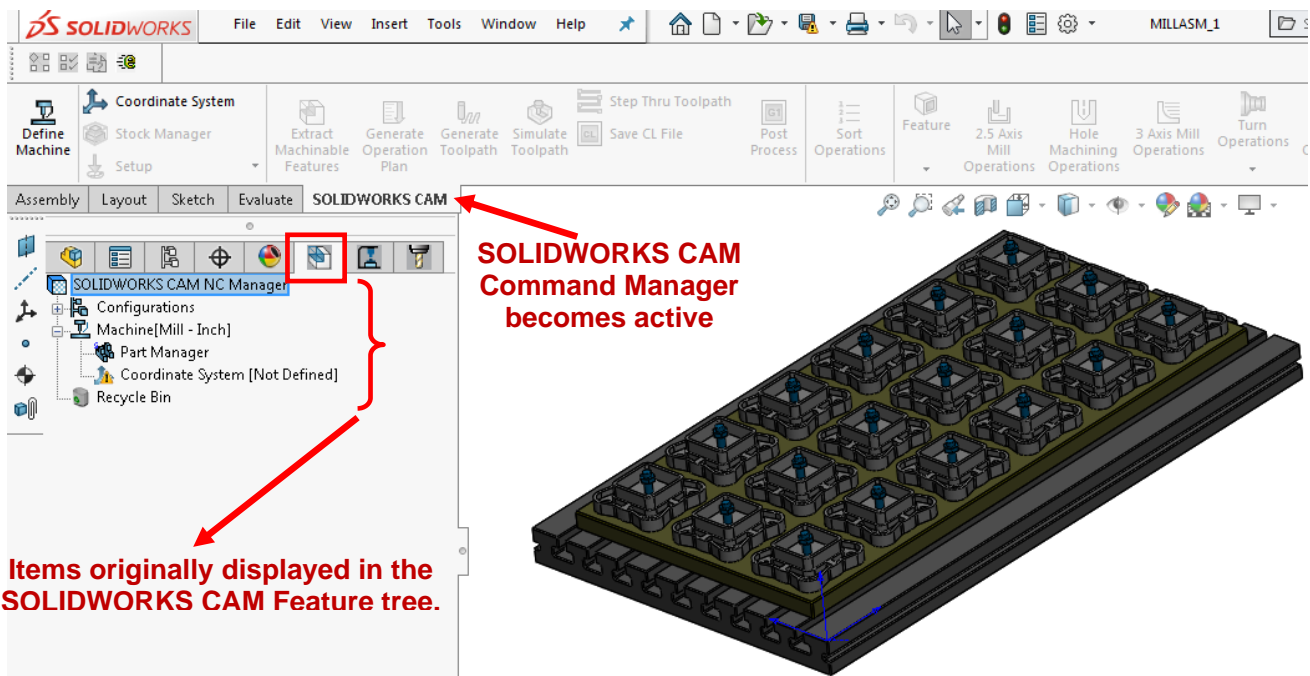
Step 2: Switch to the SOLIDWORKS CAM Feature Tree

In the tree View area of SOLIDWORKS application, click on the *SOLIDWORKS CAM Feature Tree* tab .



SOLIDWORKS CAM Feature Tree icon

When the *SOLIDWORKS CAM* Feature tree is displayed, it will initially list the [SOLIDWORKS CAM NC Manager](#), [Configurations](#), [Machine](#), [Part Manager](#), Coordinate System and [Recycle Bin](#) items.



User Interface when the SOLIDWORKS CAM Feature Tree is displayed

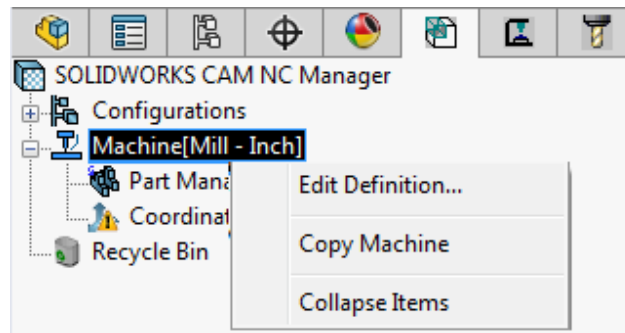
Step 3: Select the Machine, Tool Crib and Post Processor

In the *Feature tree*, the icon that represents the Machine is indicative of the machine currently selected. The associated Machine dialog box includes information that identifies what to machine, how to machine it, and the format of the NC output.

1. Right click the *Machine [Mill - Inch]* item in the SOLIDWORKS CAM 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.



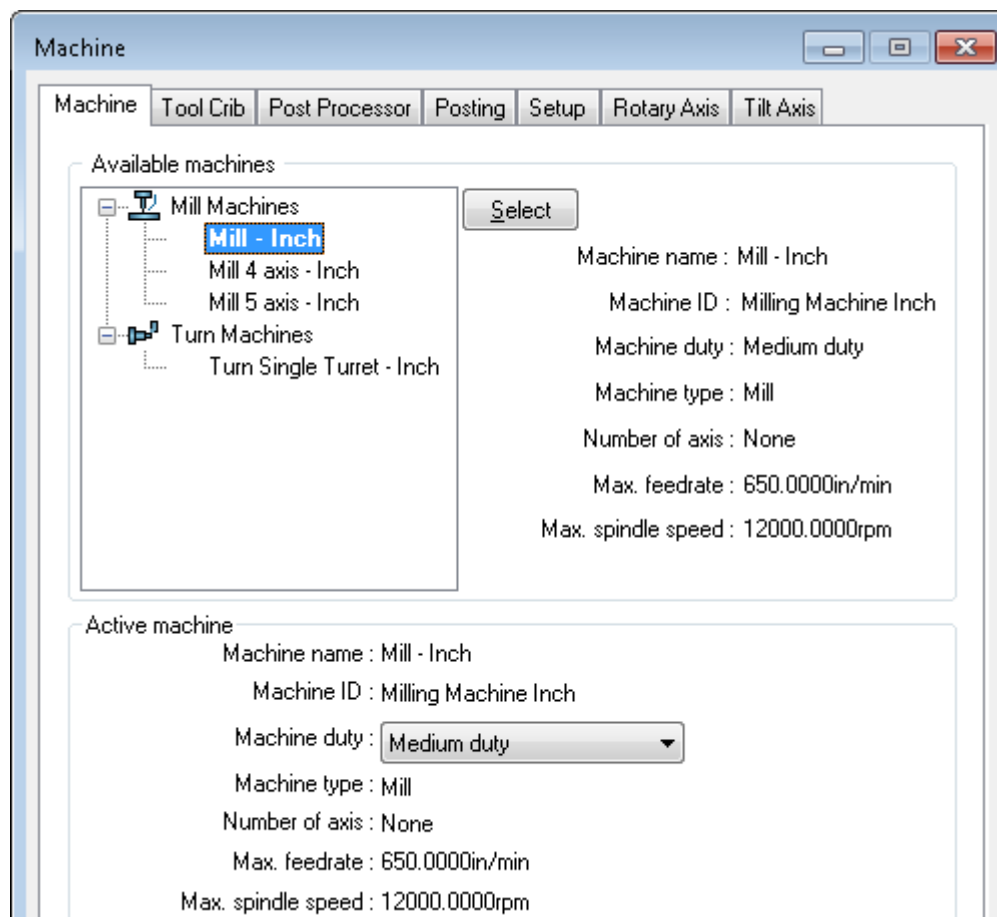
Select 'Edit Definition' on the context menu



In the Feature tree and Operation tree, instead of right-clicking items and selecting the 'Edit Definition' command on the context menu, you can double-click the item to open the corresponding dialog box. This applies to items such as the Part Manager, Machine, Setups, Features and Operations listed in these trees.

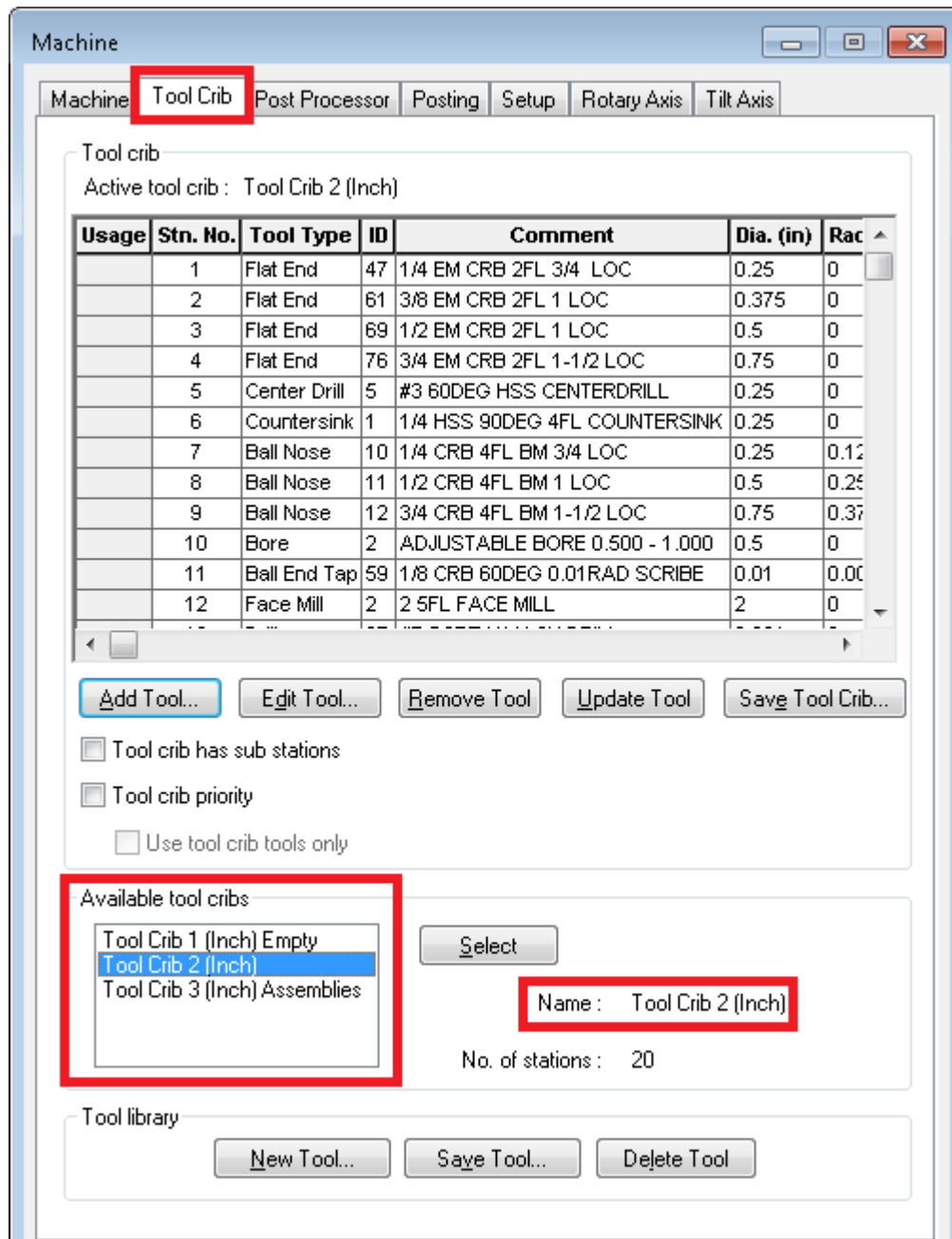
Executing this command displays the *Machine* dialog box with its *Machine* tab active. This tab lists the default machines defined in the Technology Database. *Machine [Mill - Inch]* is the default machine used for the inch parts in this manual.

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

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 currently active tool crib is displayed below the *Select* button in the *Available tool cribs* group box.]



Tool Crib tab of Machine Dialog Box

Tool Crib 2 (Inch) is the default tool crib that has been set up for the sample Mill machine. When you define your machine tools in the Technology Database, you can set up your own tool cribs.

Note: 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.

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.

Note: **SOLIDWORKS CAM is supplied with several tutorial post processors. Contact your SOLIDWORKS CAM 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. Ensure that *M3AXIS-TUTORIAL* (the tutorial post processor) is the active post processor in this tab. If *M3AXIS-TUTORIAL* (the tutorial post processor) is not the active post processor, highlight it in the list and click the *Select* button.

Note: **The *M3AXIS-TUTORIAL* is the post processor which will be used for all the tutorials in this User Guide. When you use SOLIDWORKS CAM to machine your own parts, you can select your machine tool controller or post processor.**

6. Click on the OK button to apply the changes and close the Machine dialog box.

Step 4: Assign the Fixture Coordinate System

Use the *Coordinate System* item in the Feature tree to define the “home point” or “main zero position” on the machine. This home point is known as the *Fixture Coordinate System (FCS)*.

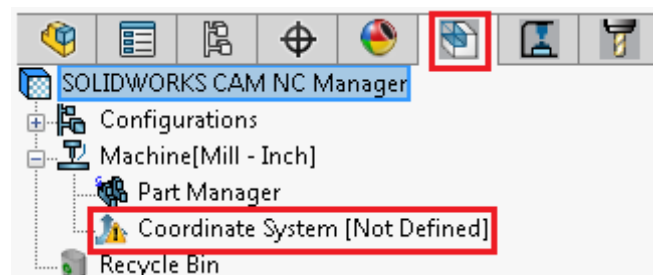
The *Fixture Coordinate System* defines the default G-code origin, defines the XYZ machining directions and acts as a reference point, if subroutines are used.

WHY ARE FIXTURE COORDINATE SYSTEMS IMPORTANT?

When machinable features are extracted automatically on a part, it is likely that SOLIDWORKS CAM 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 always be assigned first, before extracting the machinable features.

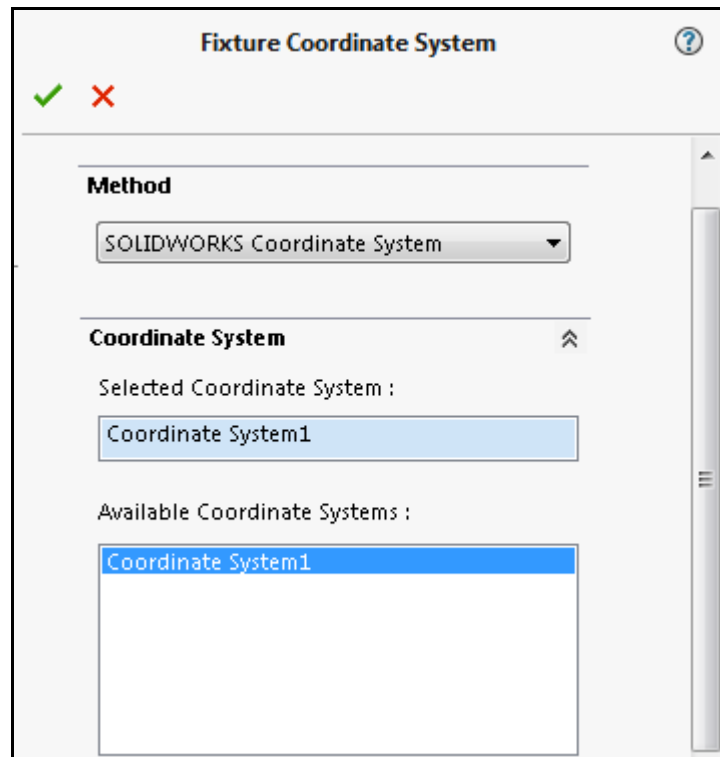
1. In the Feature tree, double-click on the *Coordinate System* item.




Double-click on the Coordinate System item in Feature tree

2. The *Fixture Coordinate System* dialog box will be displayed. This dialog box will be used to assign the FCS of the active machine. In this tutorial, the FCS will be assigned using a pre-defined SOLIDWORKS Coordinate System entity.

3. In the *Method* dropdown list, select *SOLIDWORKS Coordinate System*.
4. In the list of *Available Coordinate Systems* list box, highlight *Coordinate System1*. (This is a pre-defined Coordinate System for the tutorial part under consideration.)
5. The selected Coordinate System will be listed in the *Select Coordinate System* list box.



Fixture Coordinate System dialog box

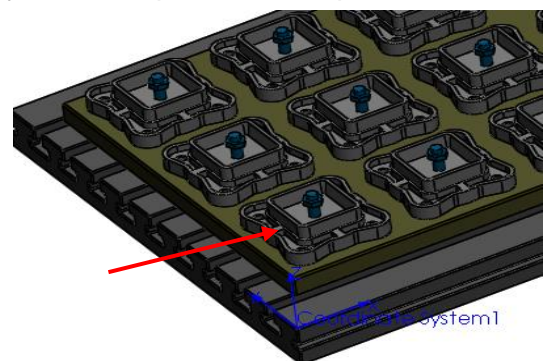
6. Click on the OK button  in this dialog box to apply the changes done and close this dialog box. The user interface will revert to the SOLIDWORKS CAM Feature tree.
7. Observe that the selected coordinate System is indicated in parentheses adjacent to the Coordinate System item in the feature tree.

Step 5: 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 SOLIDWORKS CAM 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 the *Part Manager* item in the Feature tree.

The *Manage Parts* dialog box will be displayed.



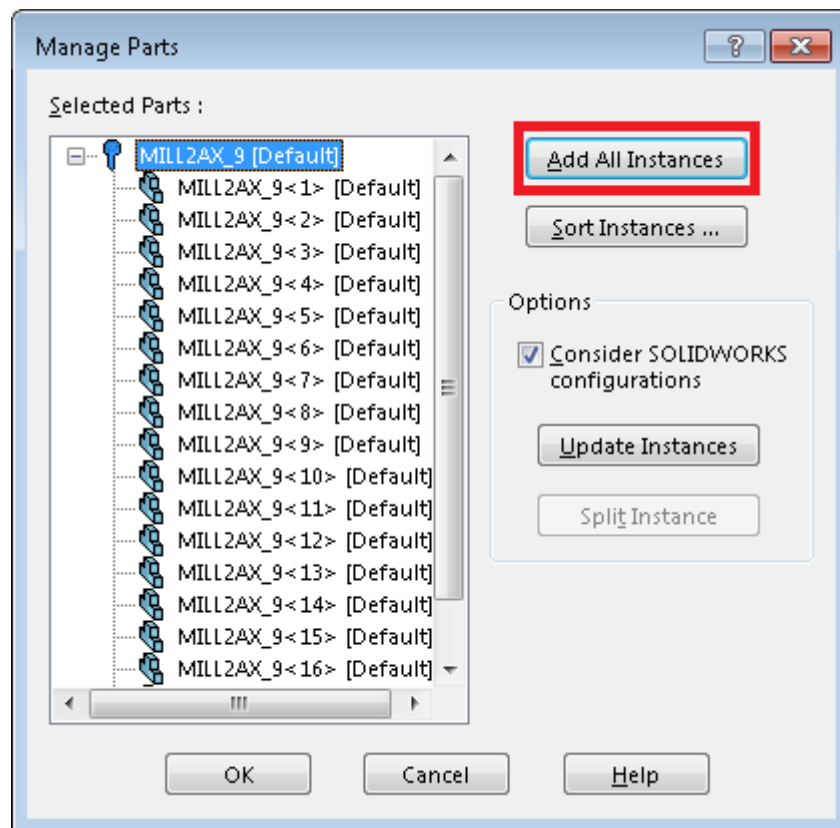
Select the lower left corner part of Assembly

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

This action will select 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.

3. Highlight the part (*MILL2AX_9.sldprt*) in the *Selected Parts* list and click the *Add All Instances* button.



Manage Parts dialog box

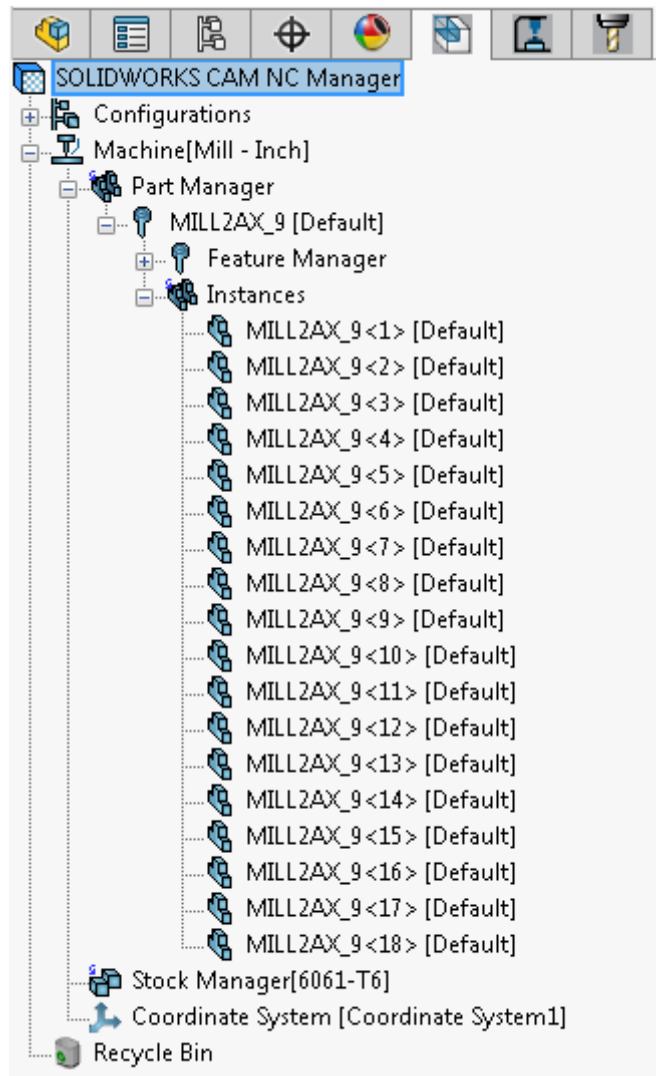
The parts listed will be in the order they appear within the assembly file. You can also pick the parts individually in the graphics area or from the SOLIDWORKS Feature Manager Design tree.

Later in this tutorial, you use the *Sort Instances* function to change the machining order.

Note: 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 apply the changes and exit the *Manage Parts* dialog box.
5. In the Feature tree, observe the following:

- The part name is listed under the *Part Manager* in the SOLIDWORKS CAM 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. Expand the *Instances* item by clicking on its "+" symbol adjacent to it in order to view all those instances.



Step 6: Defining the Stock

When you add parts in the *Manage Parts* dialog box, a default Stock is created for each part based on a 0.00 bounding box offset. The Stock Manager item in the feature tree allows you to customize the stock associated to the parts.

Following are the steps to define the Stock:

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

OR

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

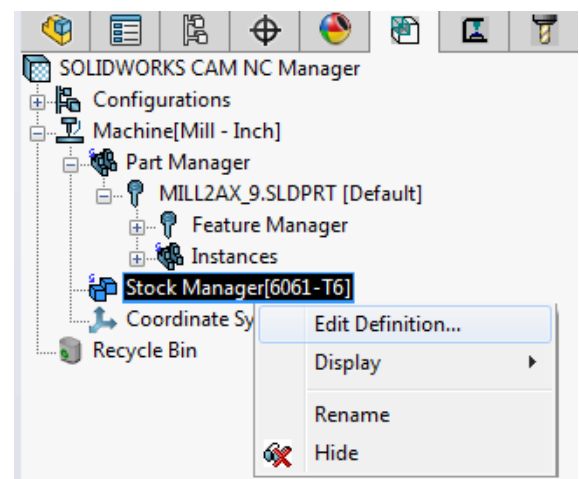
The *Stock Manager* dialog box will be displayed. This dialog box allows you to modify existing stock or create new stock for single parts and common stock for multiple parts.

2. Scroll down to the Number of Stocks list box. Click the first part in the *Parts* list box. This is the seed part.

The associated stock will be highlighted in the Stock list box and also in the graphics area. The current settings for the Bounding box offset will be displayed.

3. Scroll up to the *Bounding Box Offset* group box and change the value for Z+ to **0.1in.**

Command to open the Stock Manager dialog box



Command to open the Stock Manager dialog box

4. Scroll down to the *Create Stock* group box. Click the *Apply Current Stock Definition to All Parts* button. The action will apply the changes made to the stock of the first part instance to the stock for all other part instances.
5. Click *OK* to close the Stock Manager dialog box.

Step 7: SOLIDWORKS CAM Options dialog box to control SOLIDWORKS CAM Settings

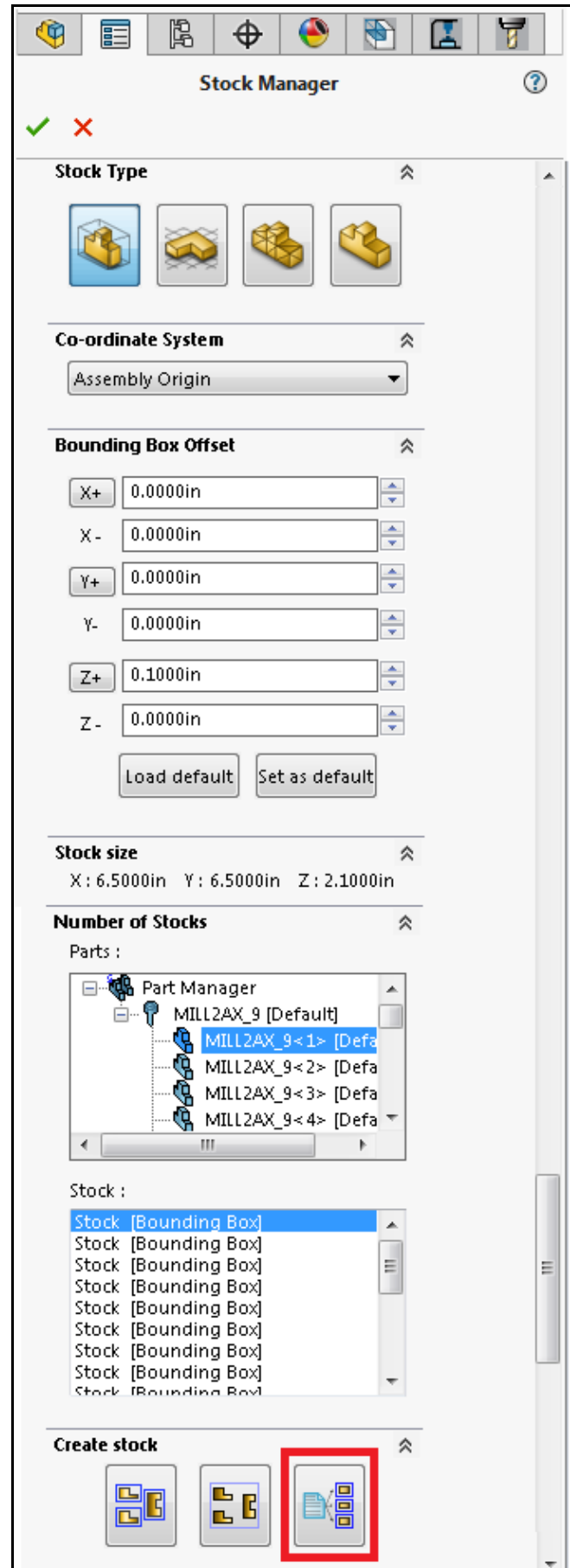
Click on the *Options* button on the SOLIDWORKS CAM Command Manager. This action opens the *Options* dialog box.



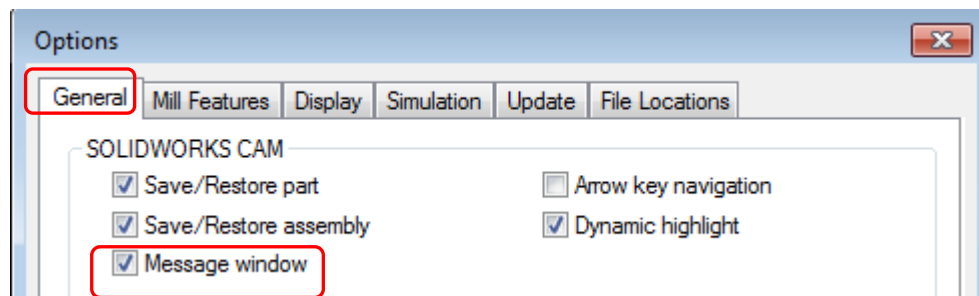
To view the Options dialog box, you can also click **SOLIDWORKS CAM** on the menu bar OR right click **SOLIDWORKS CAM NC Manager** in the Feature tree and select the **Options** command.

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 SOLIDWORKS CAM.



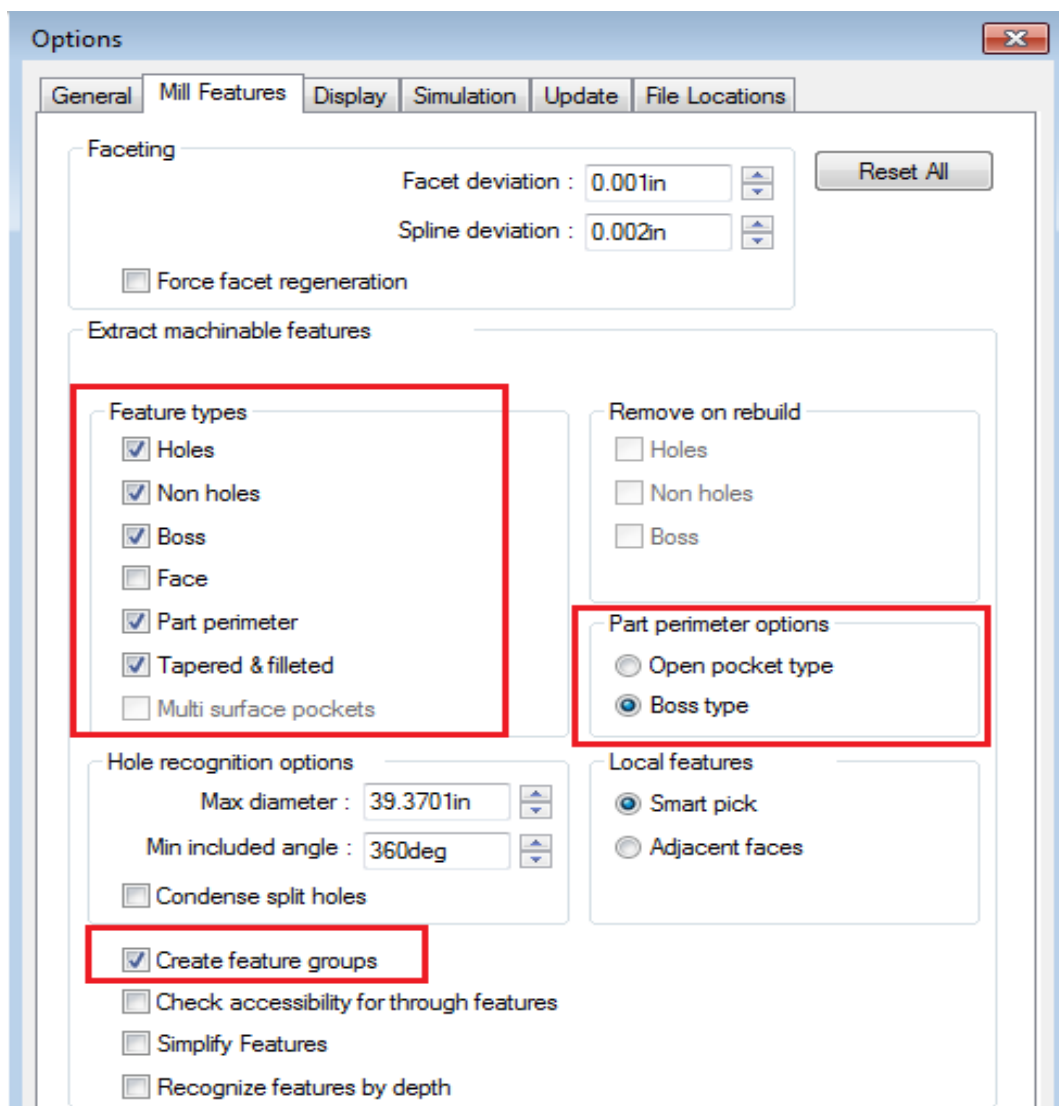
Stock Manager dialog box



SOLIDWORKS CAM Options Dialog box - General Tab

Mill Features Tab

Click on the *Mill Features* tab of this dialog box:



SOLIDWORKS CAM Options Dialog box – Mill Features Tab

1. 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.

2. 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*.

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

Step 8: 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 *SOLIDWORKS CAM Options* dialog box as explained in the previous step. Machinable Features are added in the *Feature Manager* area of the tree.

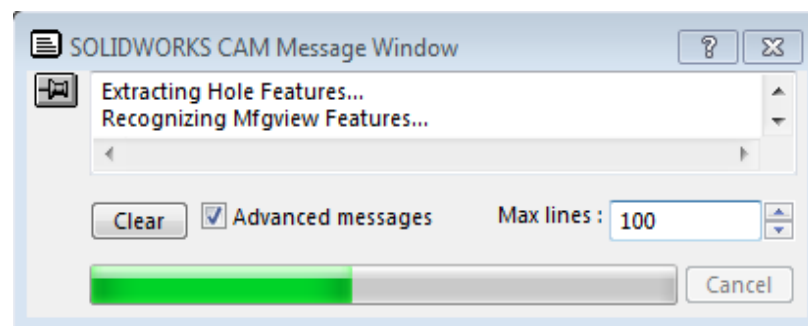
At the Mill Part Setup level, features can be inserted interactively using the *New 2.5 Axis* or *New Multi Surface Feature* commands. SOLIDWORKS CAM automatically copies the features to every other instance of the part selected in the Part Manager. When machining multiple instances of the same part, 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, SOLIDWORKS CAM will not copy the feature to all instances of the part.

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

OR

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

The *Message Window* will be 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.



SOLIDWORKS CAM Message Window

- On execution of the *Extract Machinable Features* command, SOLIDWORKS CAM 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.

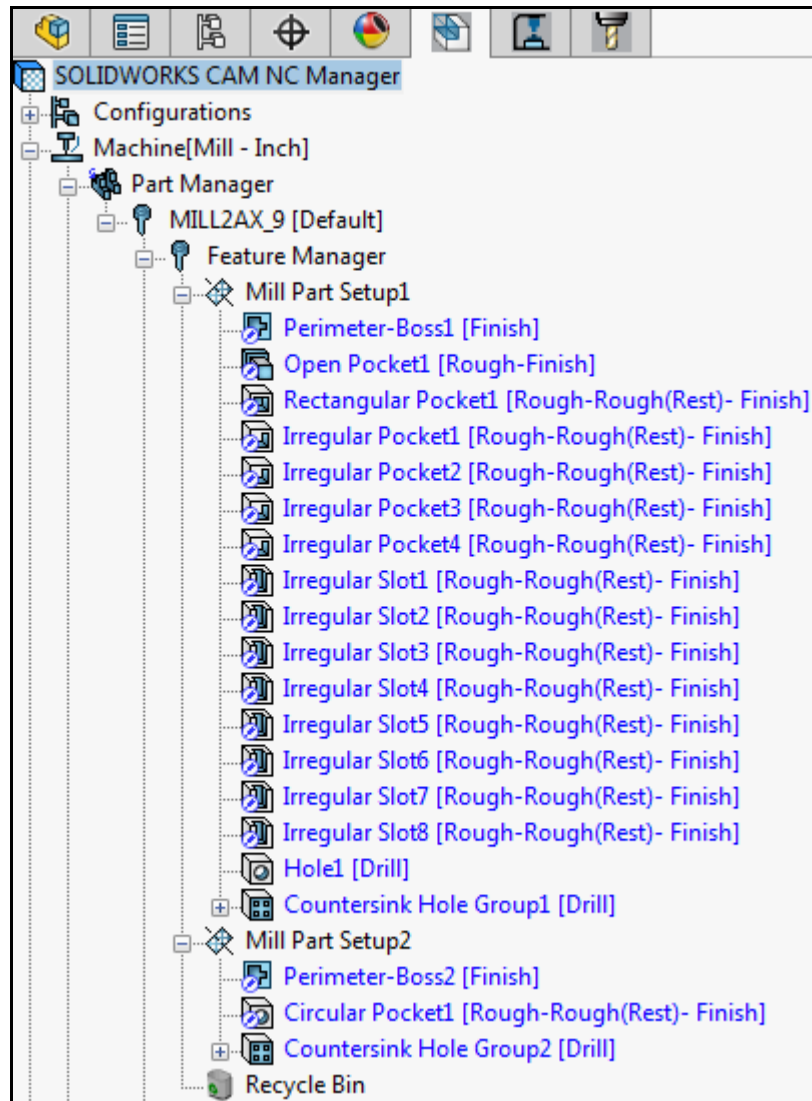
Did You Know:

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.

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

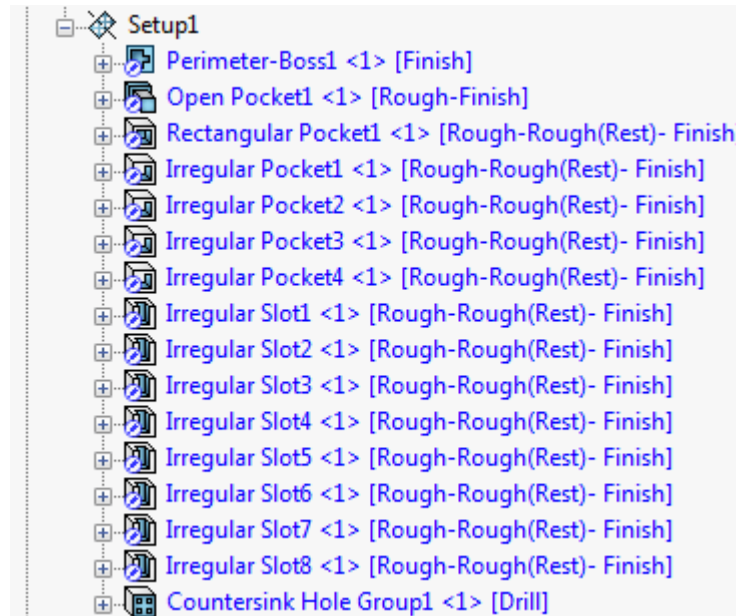
The Feature Manager displays all the Mill Part Setups and machinable features that were created by **Automatic Feature Recognition (AFR)** without consideration for how the machine is set up. In this case, AFR found and created two Mill Part Setups. Click on each Mill Part Setup and note the machining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



Extract Features are listed in the 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, SOLIDWORKS CAM 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, SOLIDWORKS CAM 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 9: 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. SOLIDWORKS CAM 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:

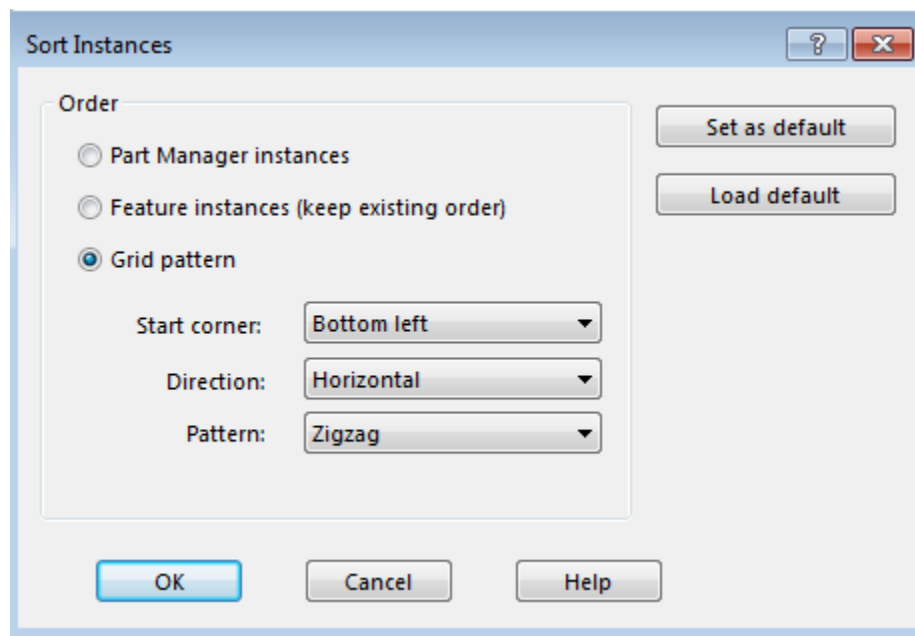
You can use one of the automatic methods, then if necessary, select the Feature instances option and make changes to the part order for individual features.

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 10: 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 SOLIDWORKS CAM Command Manager or right click the *SOLIDWORKS CAM 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 SOLIDWORKS CAM to this operation gouges the part. Hence, a suitable tool needs to be assigned from the Tool crib.

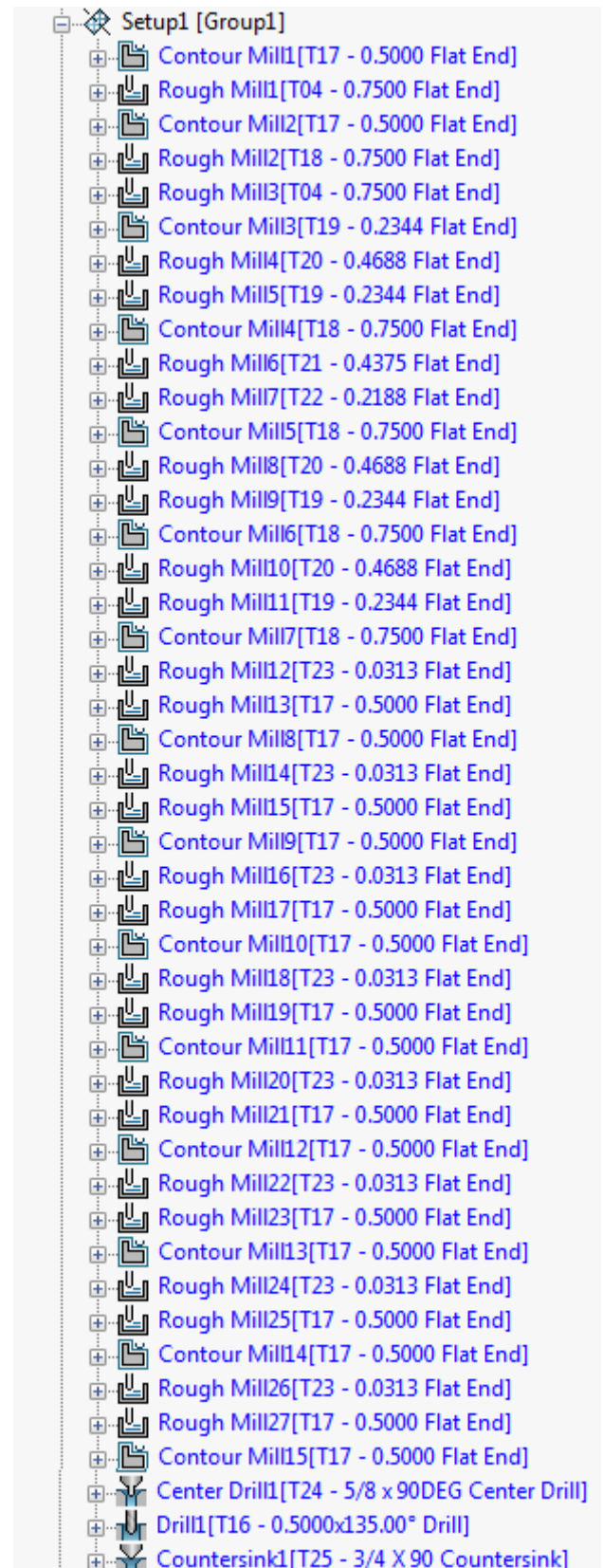
The *Operation Parameters* dialog box will be displayed.

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 *Roughing* tab. In the *Rest Machining* group, 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 Mill3* operation.

OR

Right click *Contour Mill3* operation and select *Edit Definition* on the context menu.

The *Operation Parameters* dialog box is displayed. *Contour Mill3* operation is used



Generated Operations listed in Operation Tree

for machining the *Irregular Pocket1* feature of the part.

11. Click on the *Tool* tab and select the *Mill Tool* page.

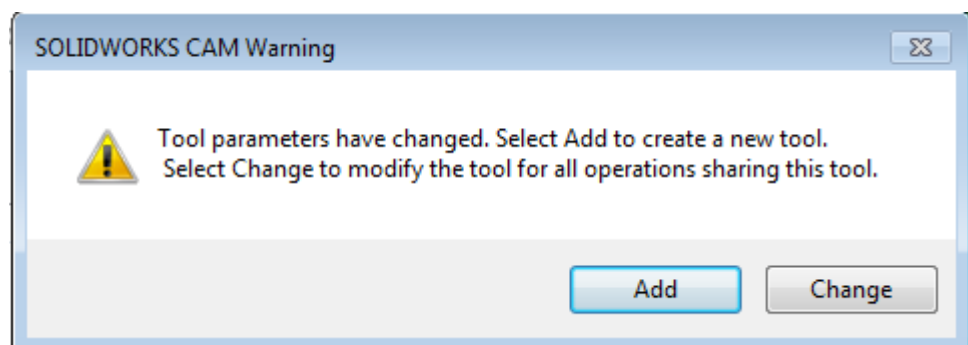
It displays the parameters of the selected tool.

12. Observe the *Tool Usage* value. This value indicates the number of operations currently using this tool.
13. In the *Cut Diameter (D1)* field, set the diameter value to **0.625in.**
14. Click *OK* to apply the change.

Since this tool is also shared by three other operations, making any changes will affect those operations too. Therefore, SOLIDWORKS CAM 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.

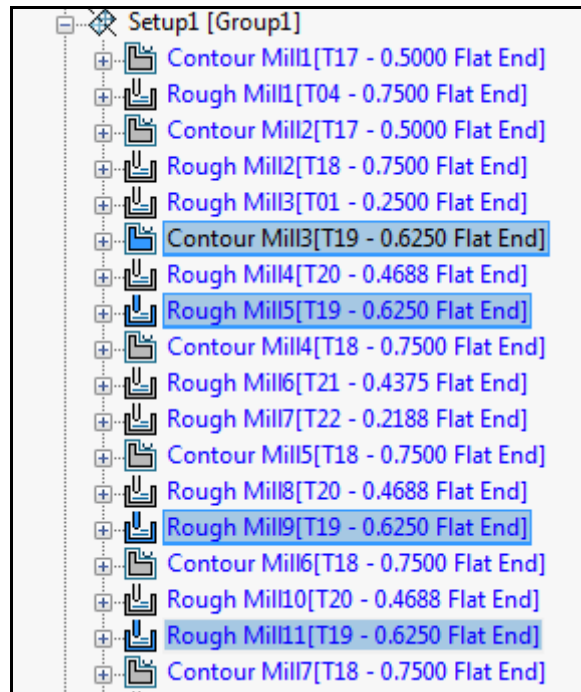
15. Click *Change* on the message.

- **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 SOLIDWORKS CAM 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.



SOLIDWORKS CAM Warning message


16. In the Operation tree, observe the four 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 four operations.



Operations sharing the same Flat End Mill tool

Step 11: 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.

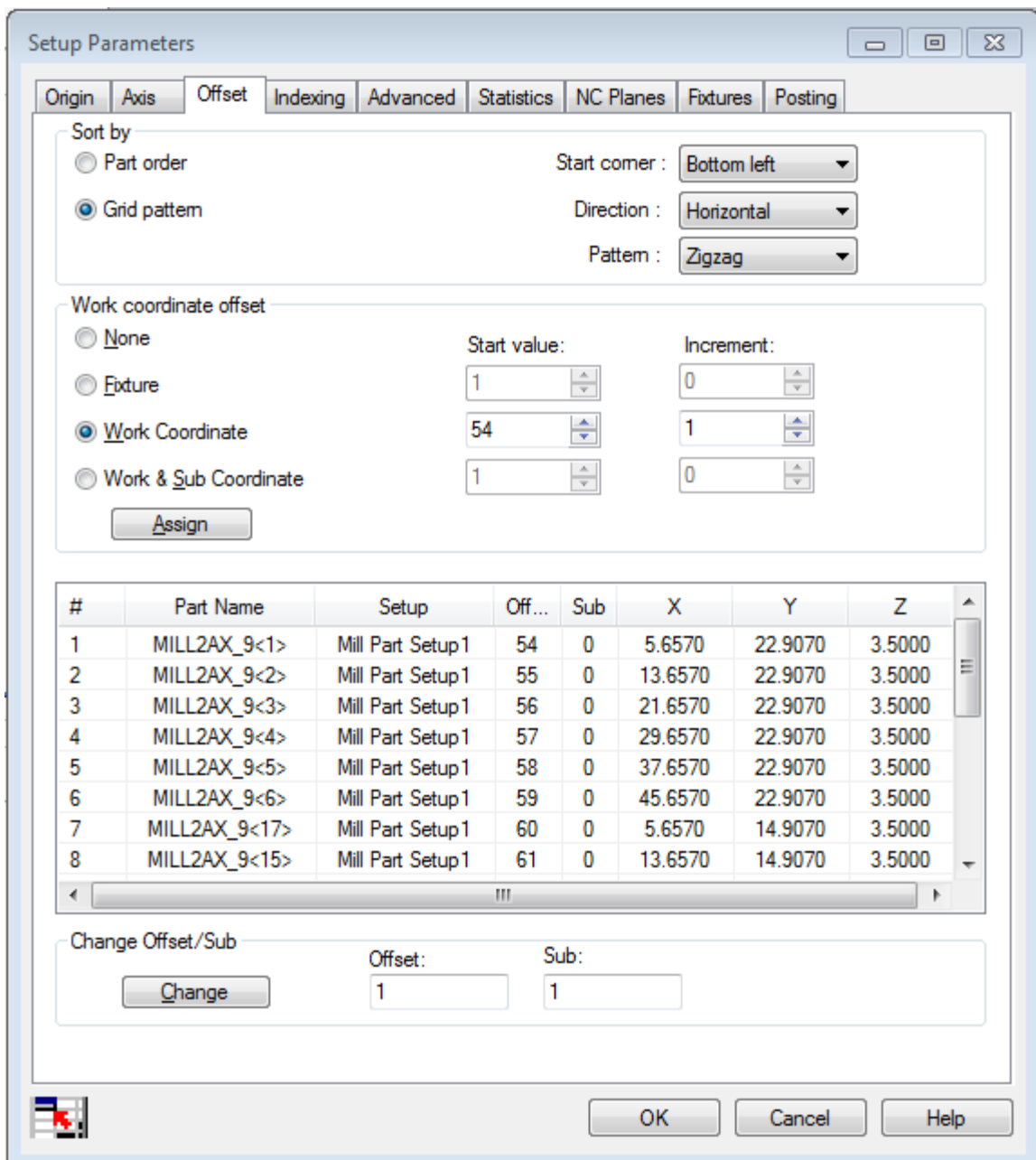
1.  Double click *Setup1* in the Operation tree.
The *Setup Parameters* dialog box will be displayed.
2. On the *Origin* tab, make sure *Part Setup origin* is selected for the Output origin.
Note that when *Setup origin* is selected, you can specify the origin using several methods.
3. Click on the *Offset* 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 9-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

Origin | **Axis** | **Offset** | Indexing | Advanced | Statistics | NC Planes | Fixtures | Posting

Sort by:
☐ Part order
☒ Grid pattern

Start corner: Bottom left
 Direction: Horizontal
 Pattern: Zigzag

Work coordinate offset:
☐ None
☐ Fixture
☒ **Work Coordinate**
☐ Work & Sub Coordinate

Start value: 54
 Increment: 1

Assign

#	Part Name	Setup	Off...	Sub	X	Y	Z
1	MILL2AX_9<1>	Mill Part Setup1	54	0	5.6570	22.9070	3.5000
2	MILL2AX_9<2>	Mill Part Setup1	55	0	13.6570	22.9070	3.5000
3	MILL2AX_9<3>	Mill Part Setup1	56	0	21.6570	22.9070	3.5000
4	MILL2AX_9<4>	Mill Part Setup1	57	0	29.6570	22.9070	3.5000
5	MILL2AX_9<5>	Mill Part Setup1	58	0	37.6570	22.9070	3.5000
6	MILL2AX_9<6>	Mill Part Setup1	59	0	45.6570	22.9070	3.5000
7	MILL2AX_9<17>	Mill Part Setup1	60	0	5.6570	14.9070	3.5000
8	MILL2AX_9<15>	Mill Part Setup1	61	0	13.6570	14.9070	3.5000

Change Offset/Sub
 Offset: 1
 Sub: 1
Change

OK Cancel Help


Setup Parameters Dialog Box

Did You Know:

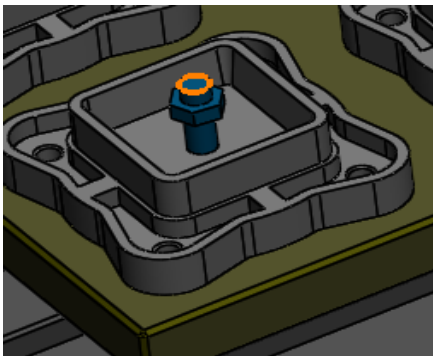
Changing the machining order does not automatically change the offset assignments. If you want the offset order to corresponding to the machining order, you need to sort the parts and reassign the offsets on the Offset tab.

Step 12: Identifying Fixtures and Clamps

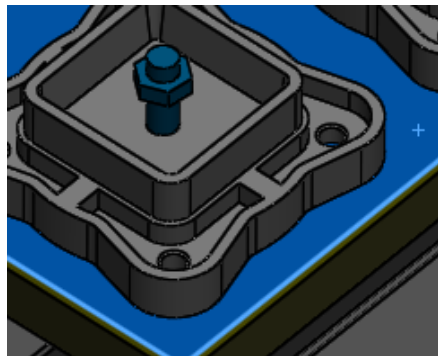
Clamps and fixture components are added on the *Fixtures* tab in the *Setup Parameters* dialog box. This dialog box allows you to define clamps, bolts, etc., so that machining toolpaths will avoid these areas and to specify the clamps and fixtures that you want displayed during simulation. Fixtures identified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

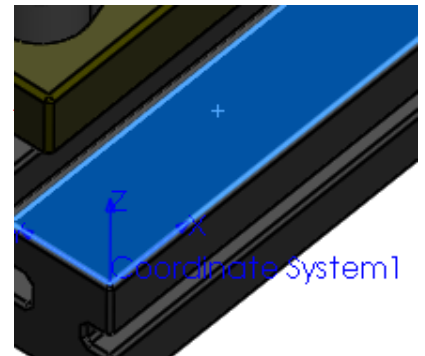
The part names are displayed in the *Fixtures* list and will display during simulation.



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, SOLIDWORKS CAM 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.

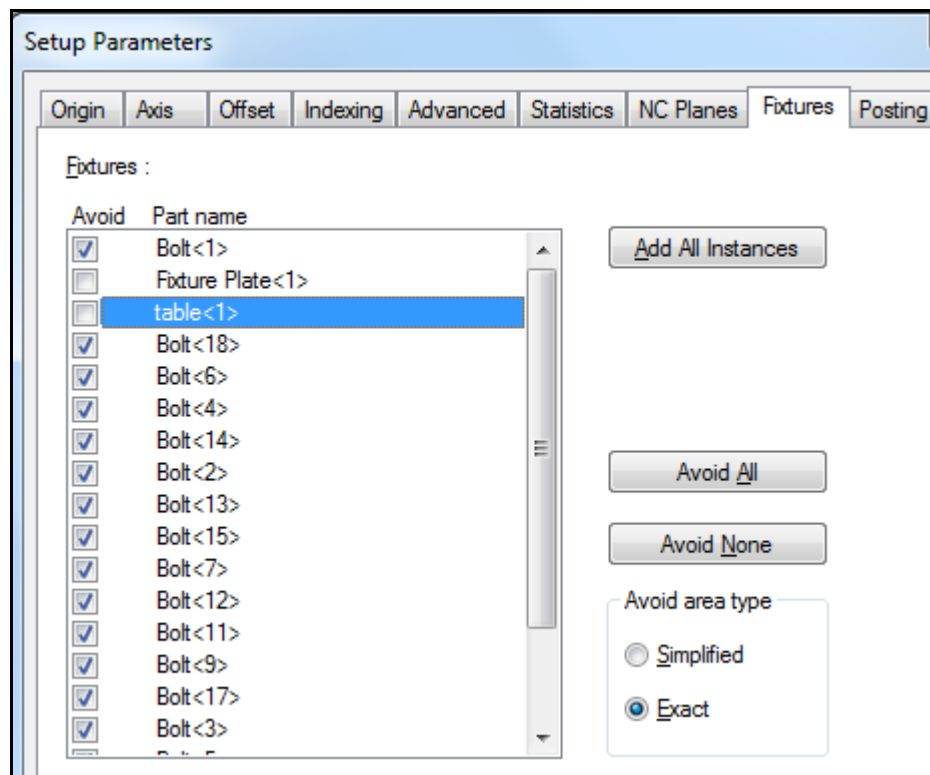
Although fixtures are 3 dimensional SOLIDWORKS parts, SOLIDWORKS CAM considers the outside silhouette or XY bounding box of avoid fixtures as islands to avoid in 2 Axis rough and contour operations. Therefore, parts such as vises, the machine table, or rotary fixtures whose silhouettes are larger than 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, SOLIDWORKS CAM 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 13: Generating Toolpaths and Sorting Operations

SOLIDWORKS CAM 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 SOLIDWORKS CAM Command Manager.
OR

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


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

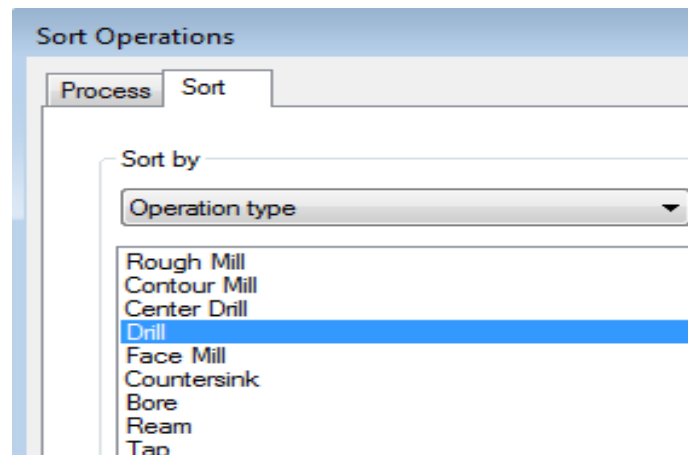
Did You Know:

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;**

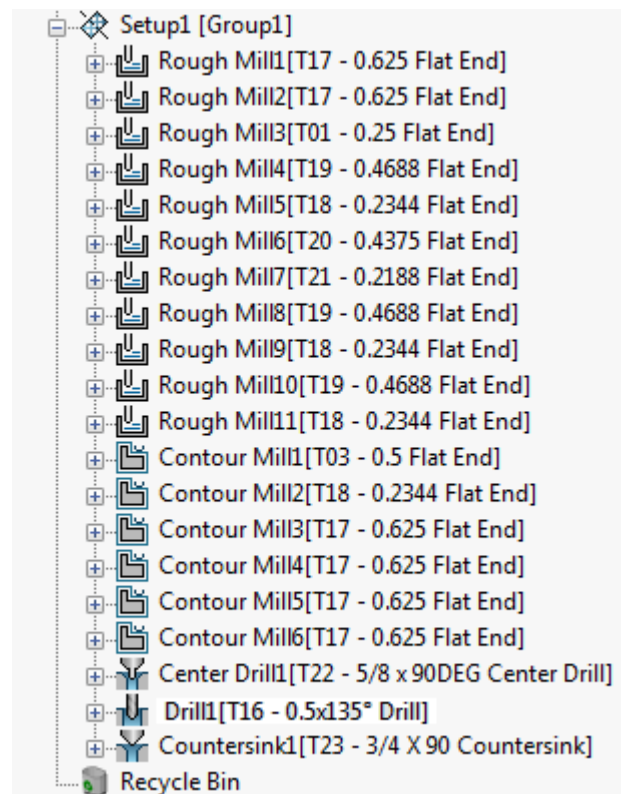
- ii. When you insert a new feature interactively and then generate operations for the new features;
- iii. When SOLIDWORKS CAM cannot generate the toolpath for an operation because of error in the toolpath algorithm or a parameter is not correct.

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

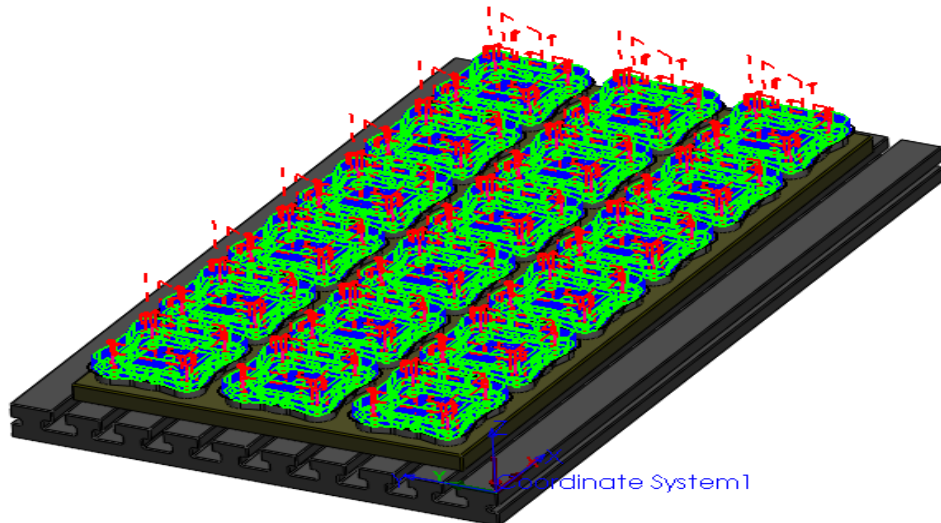
6. Click *Apply* and confirm that the tree view updates to sort the operations according to this order. If it sorts the operations as expected, then click *OK*.
7. The operations under *Setup1* are sorted based in the order on the *Sort* tab.
8. Left click any operation in the Operation tree. That operation 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*



Updated list of operations after executing the Sort Command

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 14: Simulate Toolpaths

SOLIDWORKS CAM 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 SOLIDWORKS CAM Command Manager.

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.

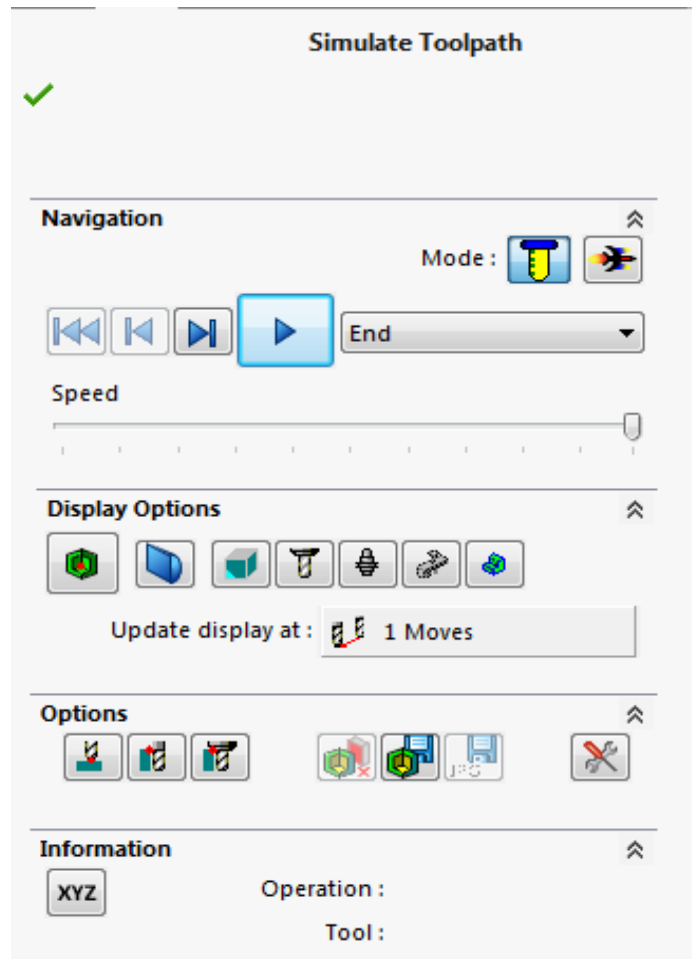
Some of the options you can select to customize the simulation include:

- 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.


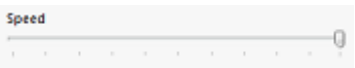





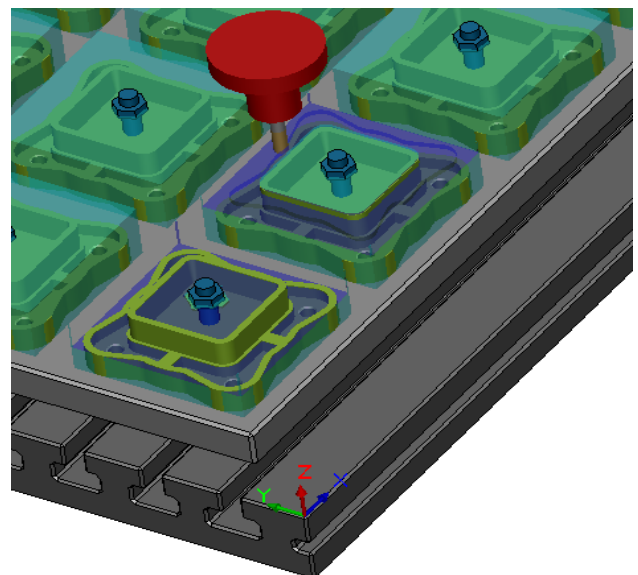
If you want to simulate only the toolpath for a given operation, you can right click that operation and then select *Simulate Toolpath* from the context menu.

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.



Simulate Toolpath dialog box

3.  Click the *Run* button.
The simulation is run with the tool displayed during simulation.
4.  Use the *Simulation Speed Control* slider to control the speed of the Simulation.
5. To pause the simulation while it is running, click on the *Run* button  again. The button will change to *Pause* button . When you click this *Pause* button, the simulation will once again begin running, continuing from the point where it was paused. The *Pause* button will change to *Run* button.
6.  Click the *Close* button in the upper right corner of the Simulation toolbar to exit the simulation mode and return to the SOLIDWORKS display.



Toolpath Simulation

Assembly Tutorial 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

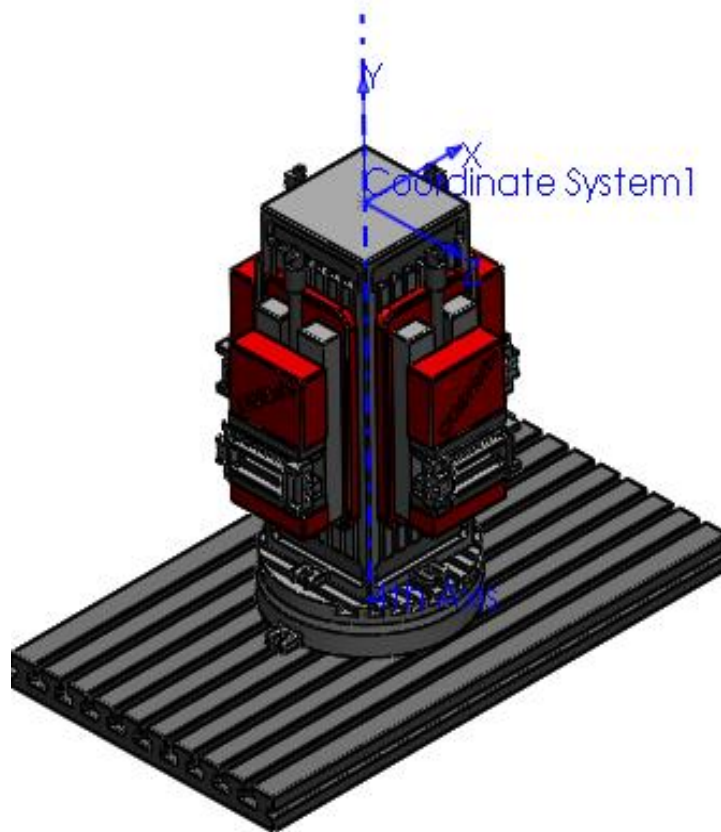
SOLIDWORKS CAM supports 4th and 5th axis rotary (prepositioning) output for milling. The 4th and 5th axis position angles can be user-defined or automatically calculated. Parts requiring multi-plane machining can be programmed in SOLIDWORKS CAM assembly mode.

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.




*C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 202x\CAM
Examples\Tutorial_Parts \Assemblies*



MILLASM_2.SLDASM

Step 2: Defining the Machine

Define the Machine

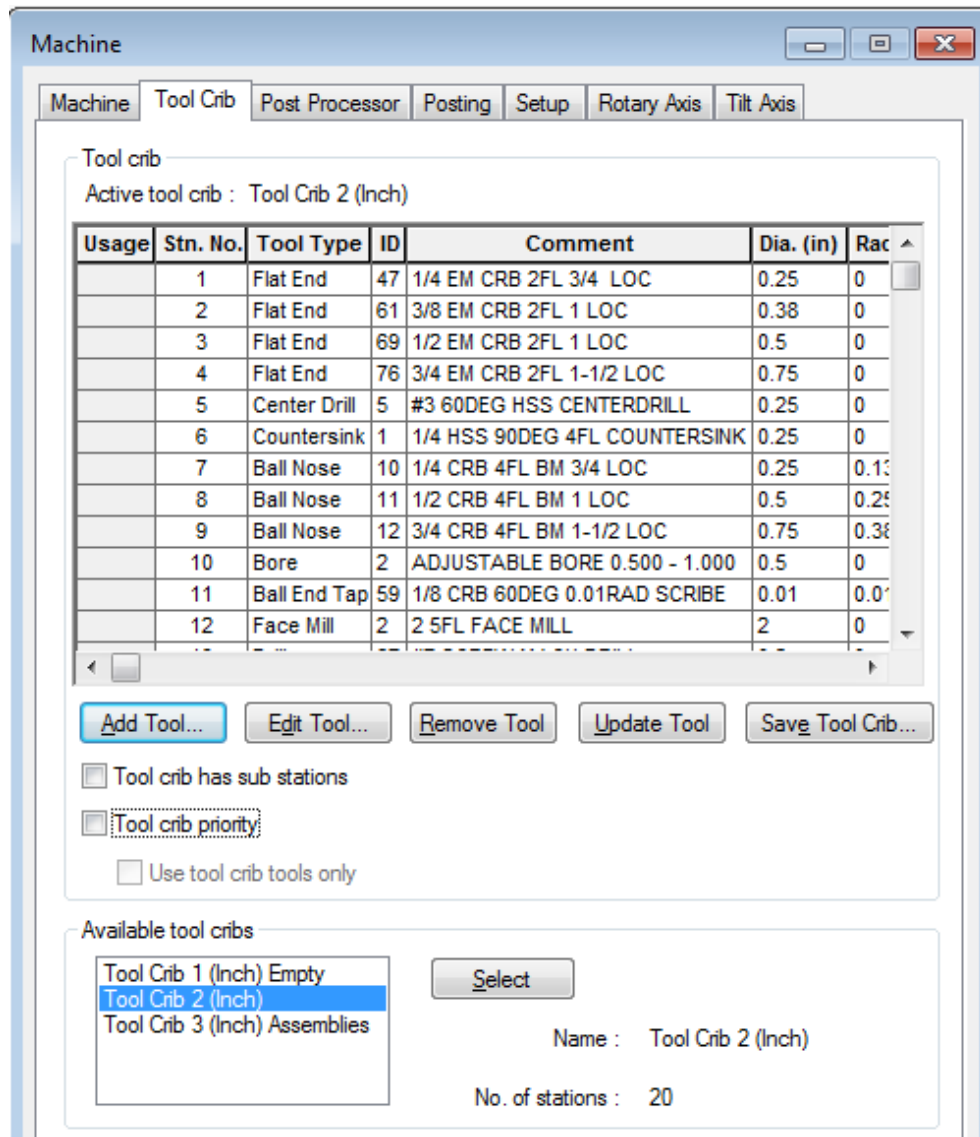
1.  Click the SOLIDWORKS CAM Feature Tree tab.
2.  Double click the Machine [Mill-Inch] item in the Feature tree.
OR
 Click on the Define Machine button on the SOLIDWORKS CAM Command Manager.
The Machine tab of the Machine dialog box is displayed.
3. Make sure Machine [Mill – Inch] is selected in the Available Machines list.



This machine definition has been created for the SOLIDWORKS CAM exercises. When you use SOLIDWORKS CAM 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.

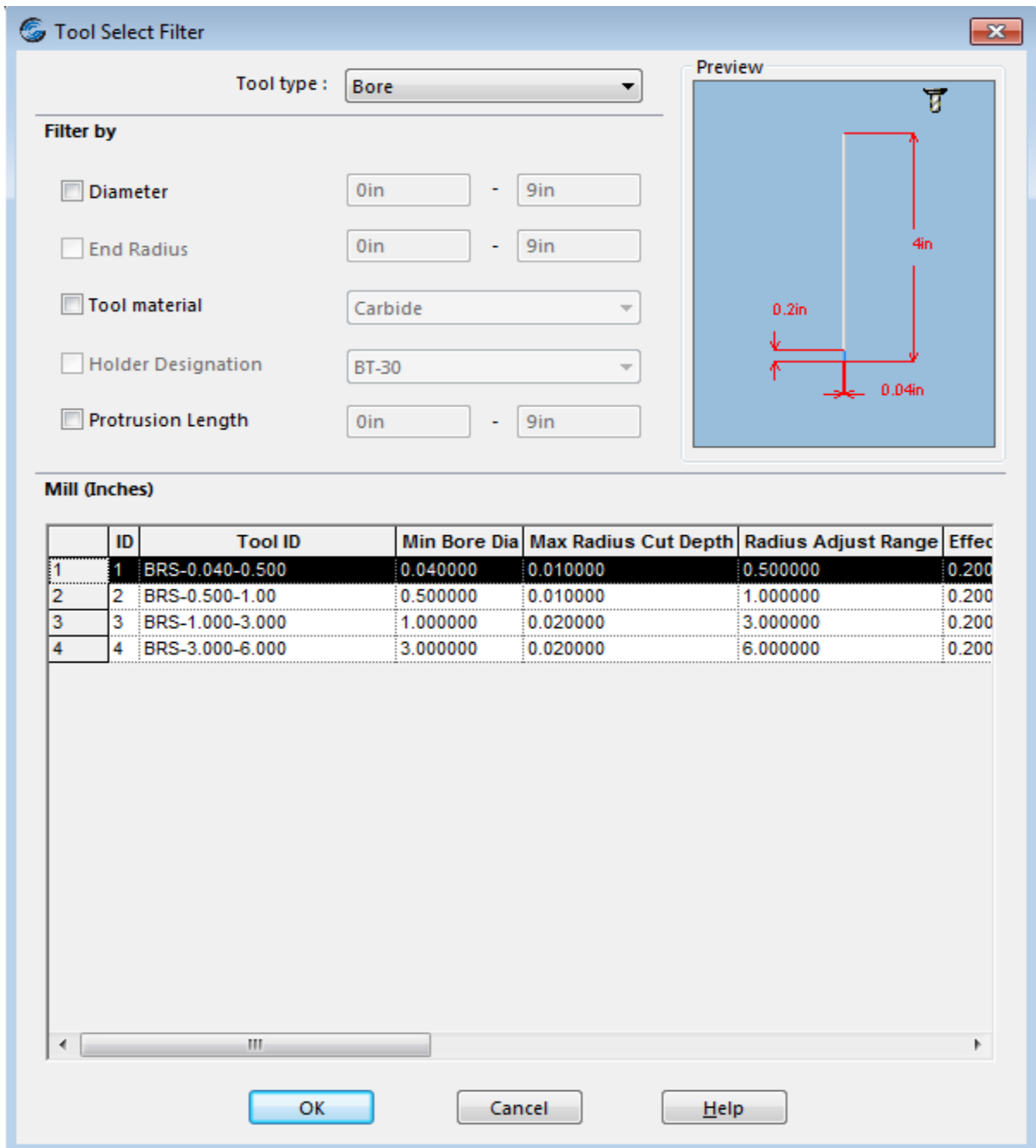
1. Click the *Add Tool* button in the Tool Crib tab.
2. The *Tool Select Filter* dialog box is displayed. This dialog box allows you to set filters and display the list of tools to select.

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



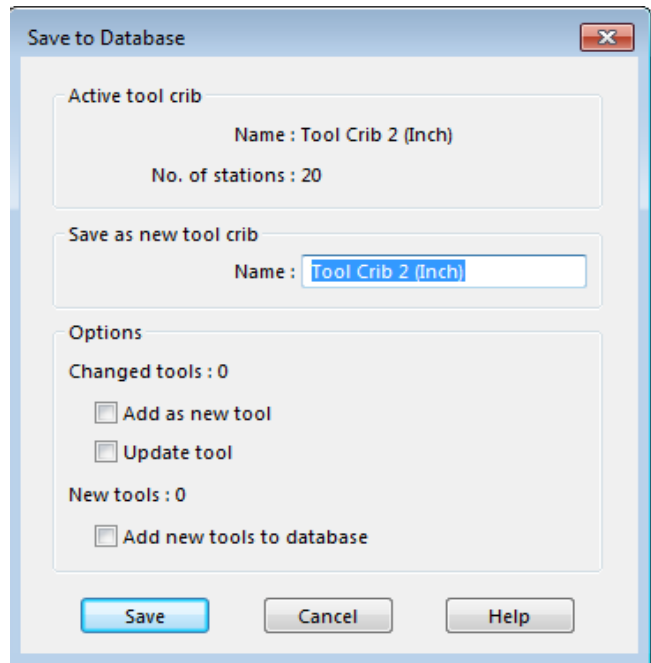
Note that the tool addition to the active tool crib is effective only for the current part. To make this tool addition available for all future jobs, click on the **Save Tool Crib** button to permanently save the changes.

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 SOLIDWORKS CAM 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 SOLIDWORKS CAM the allowable machining directions for the current SOLIDWORKS CAM machine.

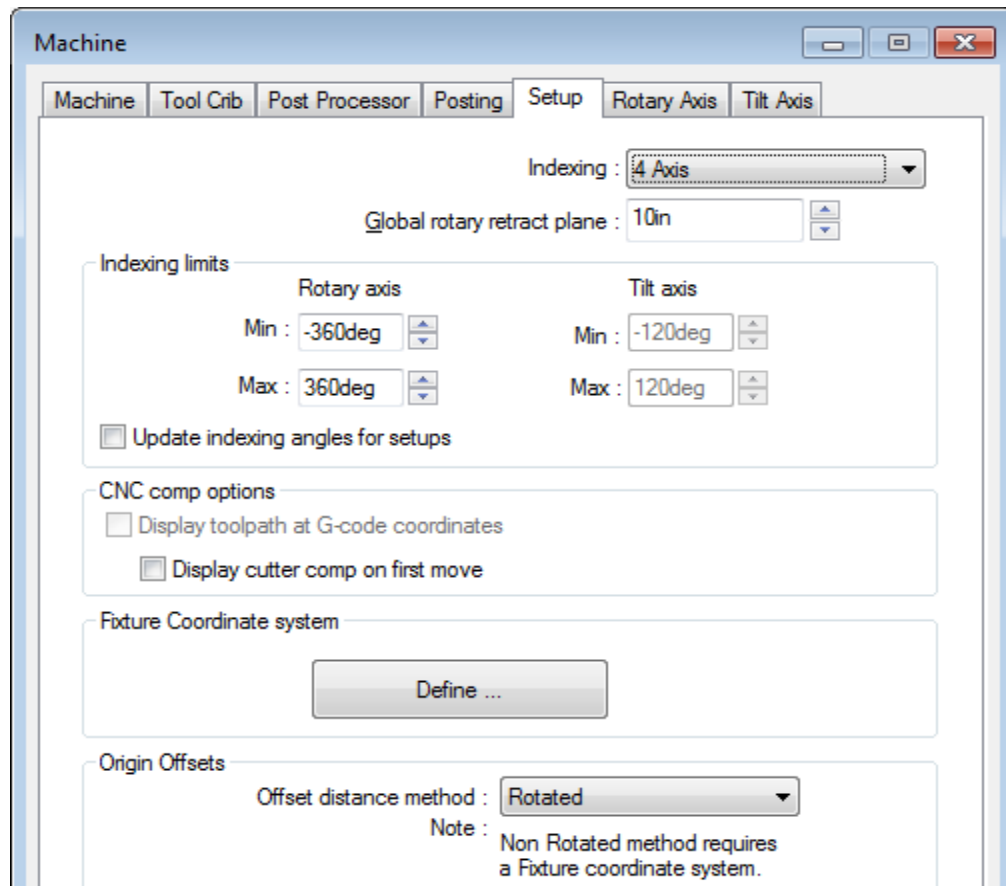
3. In this tutorial, the Fixture Coordinate System (FCS) will be defined from a SOLIDWORKS coordinate system entity.

FCS 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.

Following are the steps:

- a. Click the *Define* button in the *Fixture Coordinate System* group box. The *Fixture Coordinate System* Dialog box will be displayed.
- b. In the *Method* dropdown group box, select *SOLIDWORKS Coordinate System* from the dropdown list.
- c. In the *Available Coordinate Systems* list box, select *Coordinate System1*. This action will display *Coordinate System1* in the *Selected Coordinate System* field.

- d. Click the  **Ok** button to apply the changes and close this dialog box. The user interface will revert to the *Setup* tab.



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 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.

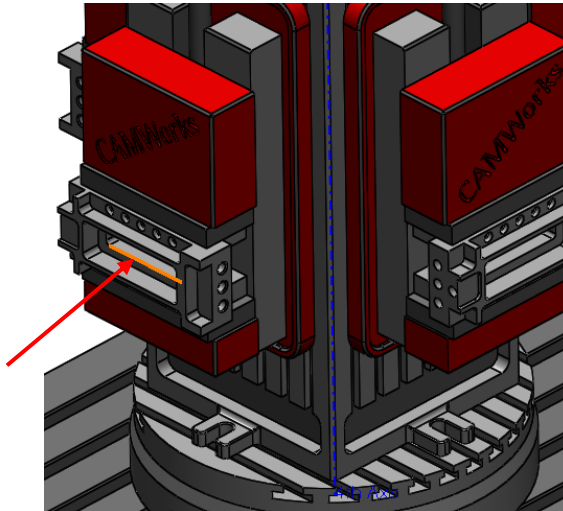
Step 3: Selecting the Parts to be Machined

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 SOLIDWORKS CAM 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 SOLIDWORKS CAM 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.

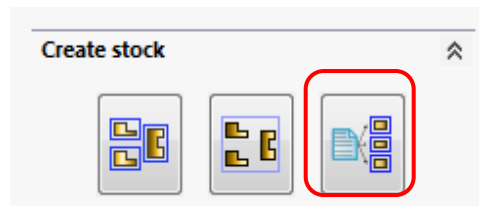
OR

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 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 Current 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

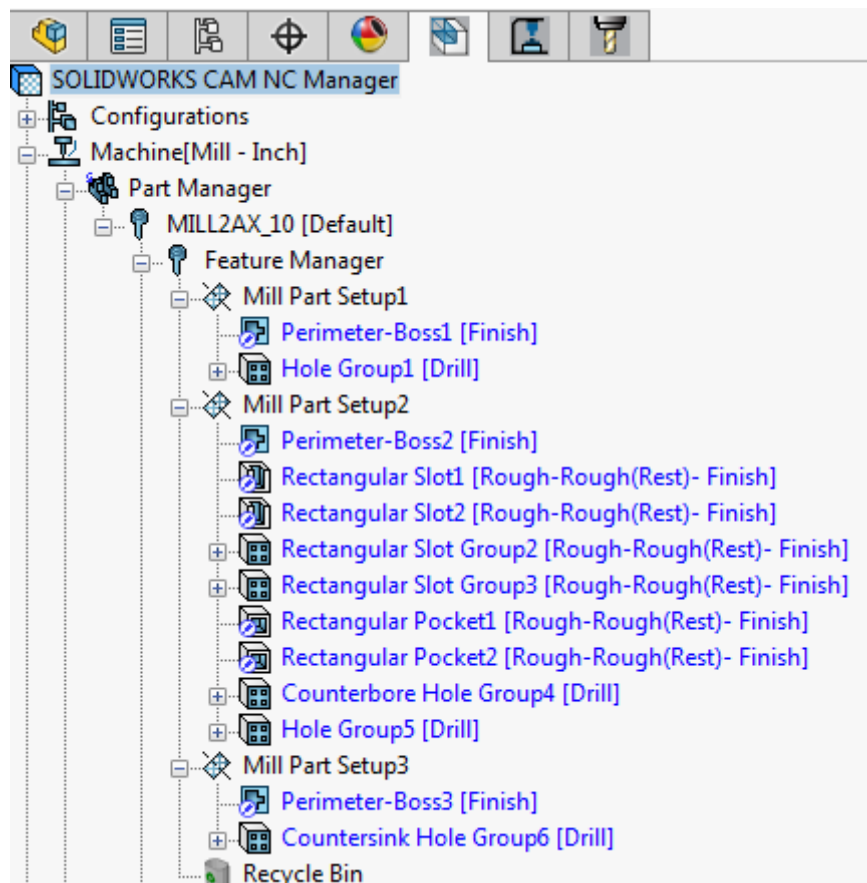
2.  Click OK to close the *Stock Manager* dialog box.

Extracting Machinable Features

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

OR

Select the *Extract Machinable Features* command from the SOLIDWORKS CAM menu.



Recognized Features listed in the feature tree

- The SOLIDWORKS CAM Message Window is displayed. This window shows the progress of the process.

Note:

One of the powers of SOLIDWORKS CAM 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.

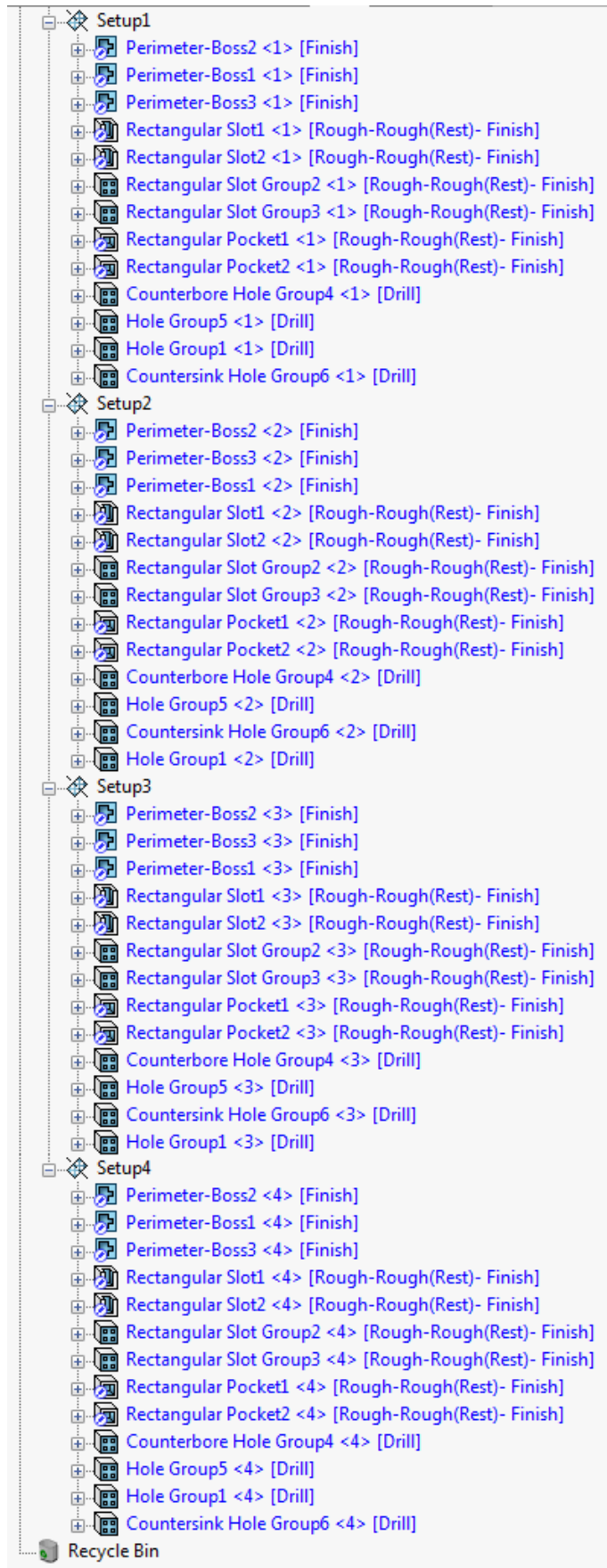
- Expand the Feature Manager item and observe that SOLIDWORKS CAM created 3 Mill Part Setups in order to machine the part.
- Below the Stock Manager, SOLIDWORKS CAM creates Setups based on the parts in the Part Manager, the Mill Part Setups and features found for the seed part, and the indexing definition. In this tutorial, SOLIDWORKS CAM creates four Setups to machine the multiple instances of the part.

Step 5: Generating the Operation Plan

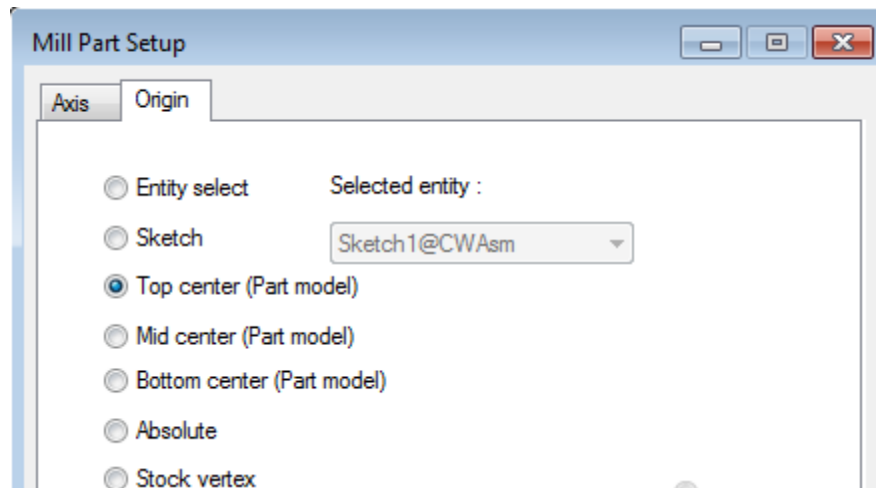
- Click on the plus sign next to the *Feature Manager* to expand it.
- Double click *Mill Part Setup1* in the tree.

The *Mill Part Setup* dialog box will be displayed.


- 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*.



Four Setups created on executing EMF Command



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 SOLIDWORKS CAM 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, SOLIDWORKS CAM 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.

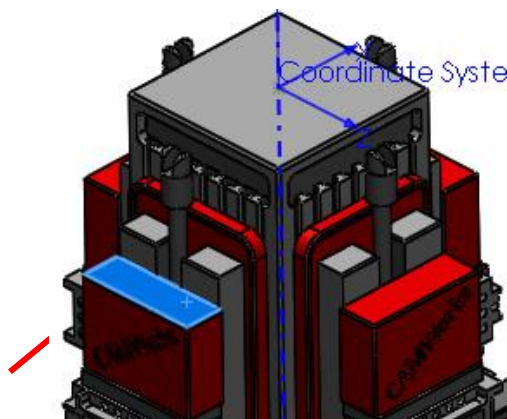
The *Setup Parameters* dialog box will be displayed.
2. On the *Origin* tab, make sure *Part Setup origin* is selected for the Output origin.

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

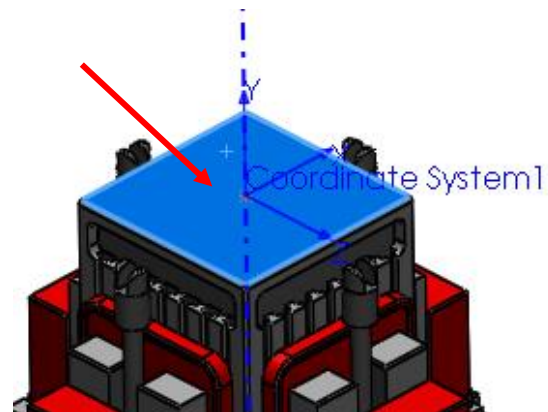
- The tombstone
- The rotary table
- The machine table

Did You Know:

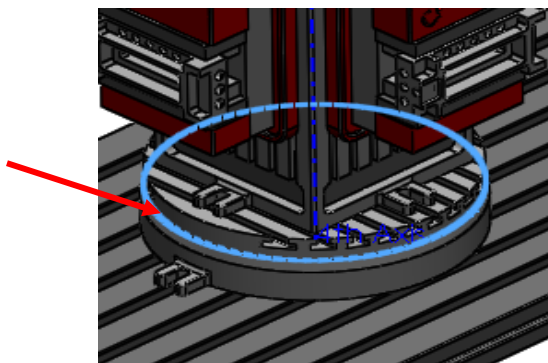
For optimum performance, when modeling the fixtures, it is recommended that you omit any details that do not affect the machining process. Including details such as edge breaks, nuts, bolts and washers in the fixtures will consume a large amount of system memory and slow down the toolpath computation.



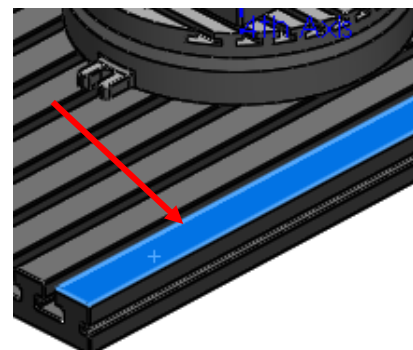
Select 'Vise Holding the Seed part'



Select 'Tombstone'



Select 'Rotary Table'



Select 'Machining Table'

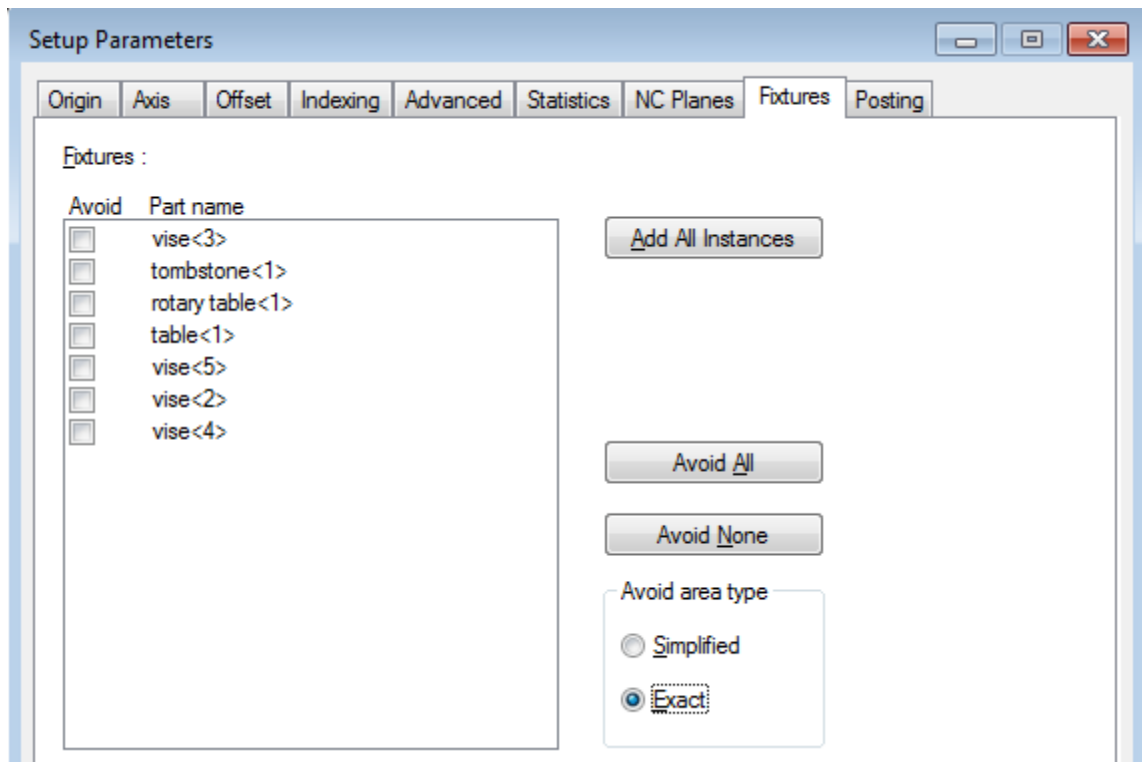
5. In the dialog box, highlight the vise in the Feature list and click the *Add All Instances* button. All the vises in the assembly are listed and will display during simulation.
6. Do not check any of the *Avoid* check boxes.

Although fixtures are 3 dimensional SOLIDWORKS parts, SOLIDWORKS CAM 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, SOLIDWORKS CAM 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

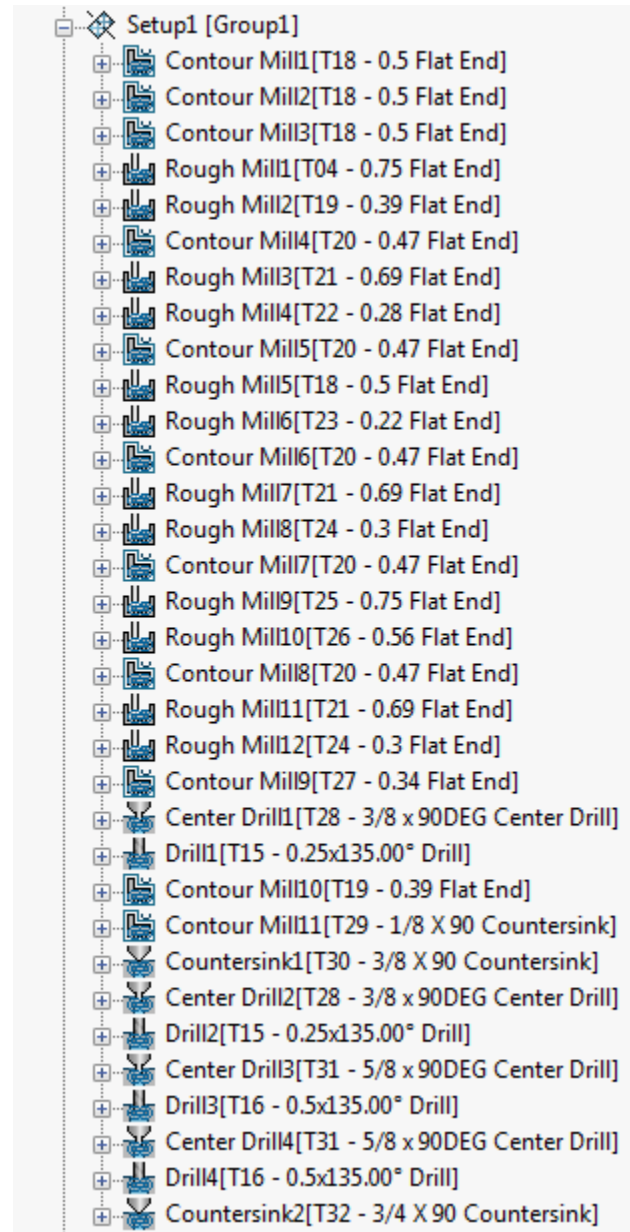
SOLIDWORKS CAM calculates toolpaths using the operation parameters to define how to machine each machinable feature. After generating toolpaths, you can sort the operations in a logical machining sequence and simulate the material removal.

1.  Click the *Generate Toolpath* button on the SOLIDWORKS CAM Command Manager.
2. Observe that toolpaths were generated for all operations.
SOLIDWORKS CAM 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.

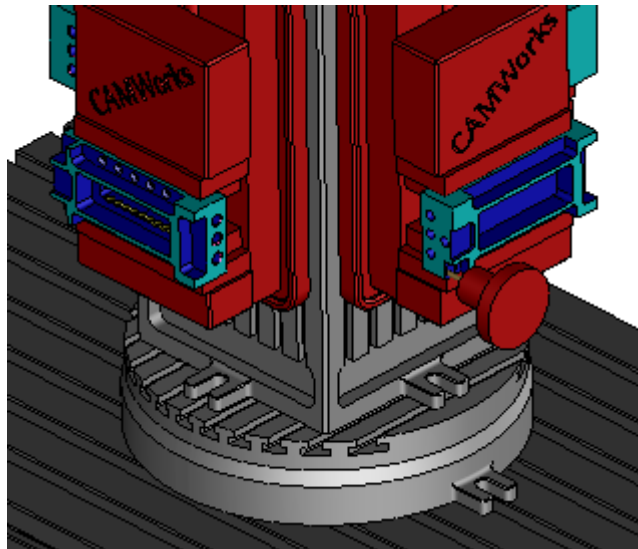
8. Right-click on the Contour Mill6 operation again and select *Generate Toolpath* command on the context menu.
9. Observe that toolpaths were also generated for the Contour Mill toolpaths under *Setup2*, *Setup 3* and *Setup4* as those were linked operations.

Step 8: Simulate Toolpaths

1.  Click the *Simulate Toolpath* button on the SOLIDWORKS CAM Command Manager.
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.
5. Click the *Close* button in the upper right corner of the Toolpath Simulation toolbar to exit the simulation mode and return to the SOLIDWORKS display.



Operations listed under Setup1



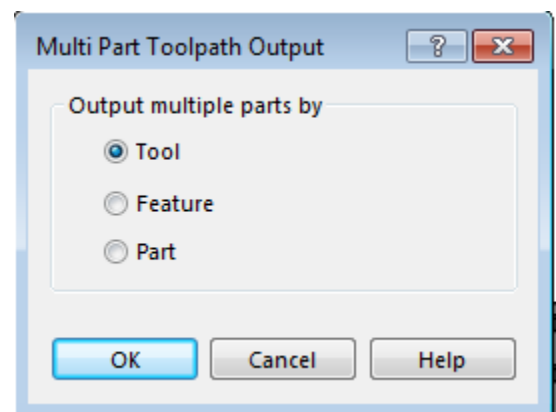
Toolpath Simulation

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.

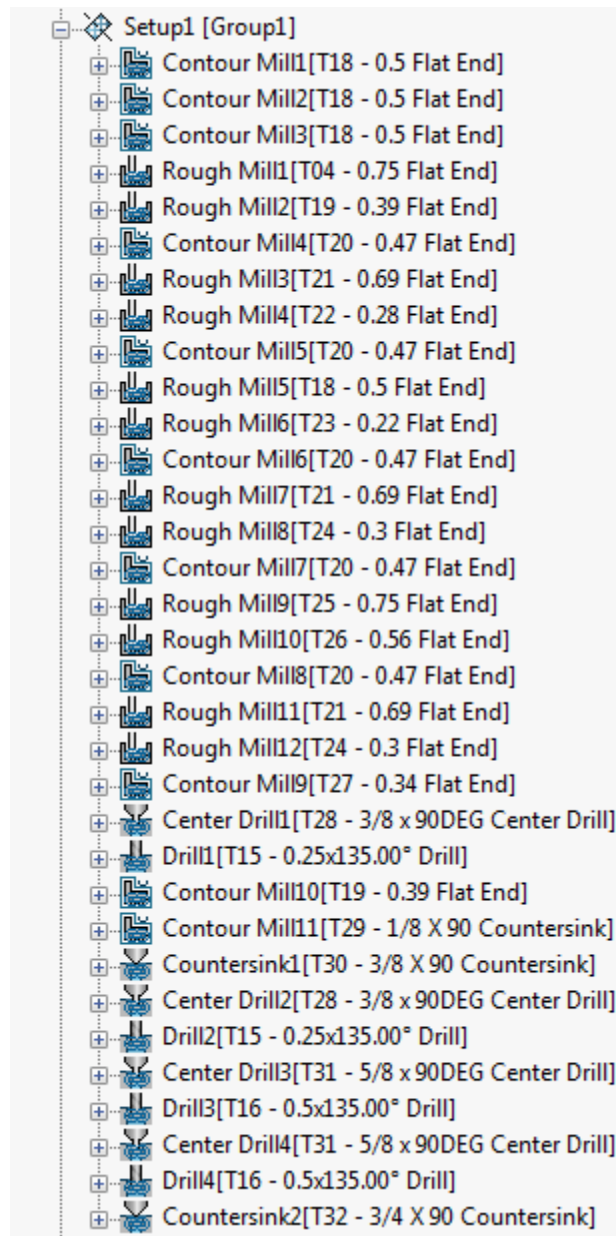
The *Multi Part Toolpath Output* dialog box allows you to set the toolpath output order in Assembly mode based on the following:

- **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*.



Operations listed under Setup1

8. Run the simulation again to observe the changes in the same manner as mentioned in the [previous tutorial](#).

Assembly Tutorial 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

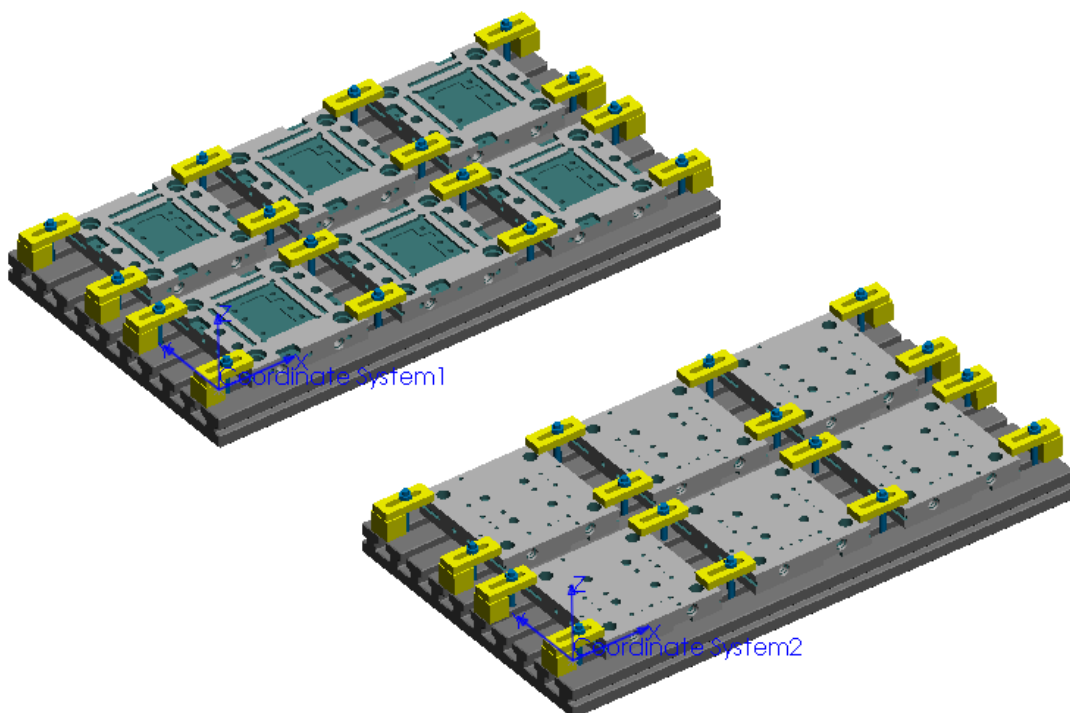
Using SOLIDWORKS CAM Assembly mode, the same parts can be machined using multiple machine tools. In this tutorial, you set up the first machine to cut the parts on the rear table, then copy the machine and program the parts on the front table.

Step 1: Open the Part

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



*C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 202x\CAM
Examples\Tutorial_Parts \Assemblies*

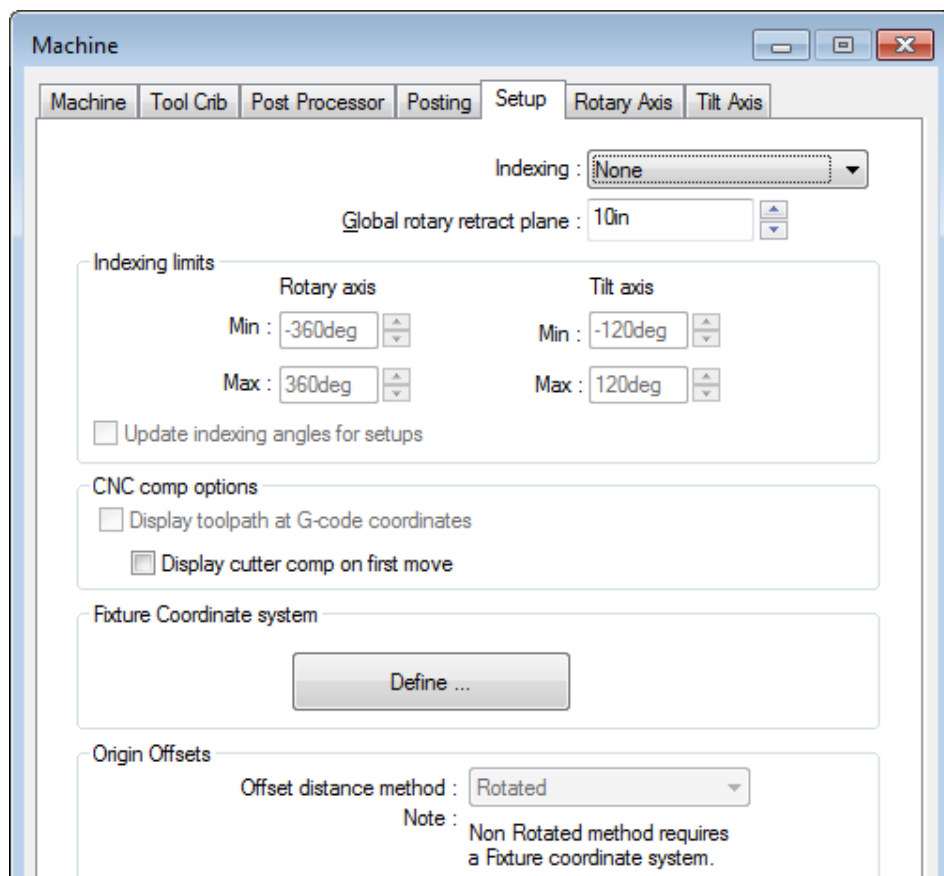
The parts in this assembly document have been positioned on two tables to be machined using two machine tools.



MILLASM_3.SLDASM

Step 2: Defining the Machine


1.  Click the *SOLIDWORKS CAM 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.



Setup tab of Machine Dialog Box

10. Select *None* for the Indexing option from the dropdown list.
11. In this tutorial, the Fixture Coordinate System (FCS) will be defined from a SOLIDWORKS coordinate system entity.

Following are the steps:

- Click the *Define* button in the *Fixture Coordinate System* group box. The *Fixture Coordinate System* Dialog box will be displayed.
- In the *Method* dropdown group box, select *SOLIDWORKS Coordinate System* from the dropdown list.
- In the *Available Coordinate Systems* list box, select *Coordinate System1*. This action will display *Coordinate System1* in the *Selected Coordinate System* field.
- Click the  *Ok* button to apply the changes and close this dialog box. The user interface will revert to the *Setup* tab.

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

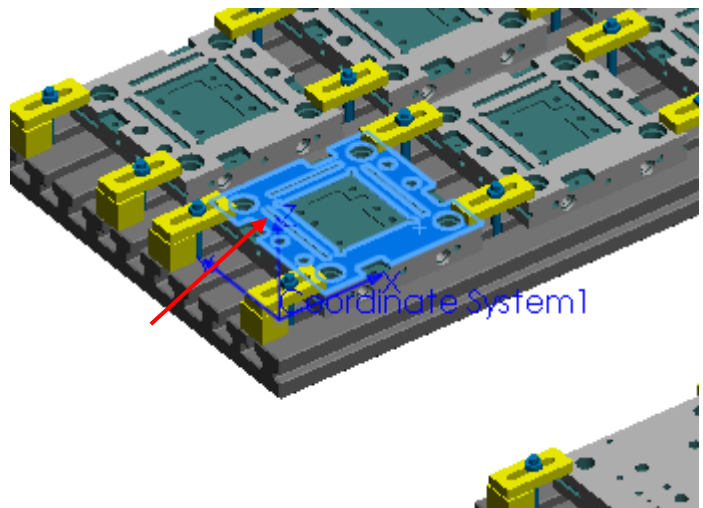
- Double click *Part Manager* in the Feature tree.

The *Manage Parts* dialog box will be displayed.

- 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.

- 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:

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

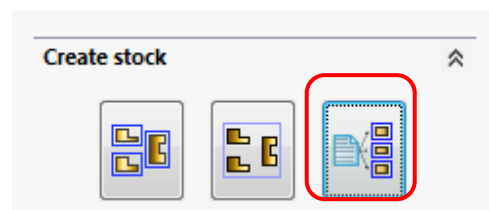
OR

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.

- Change the +Z to **0.1in.**
- In the *Create Stock* group box, select 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

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

Extracting Machinable Features

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

OR

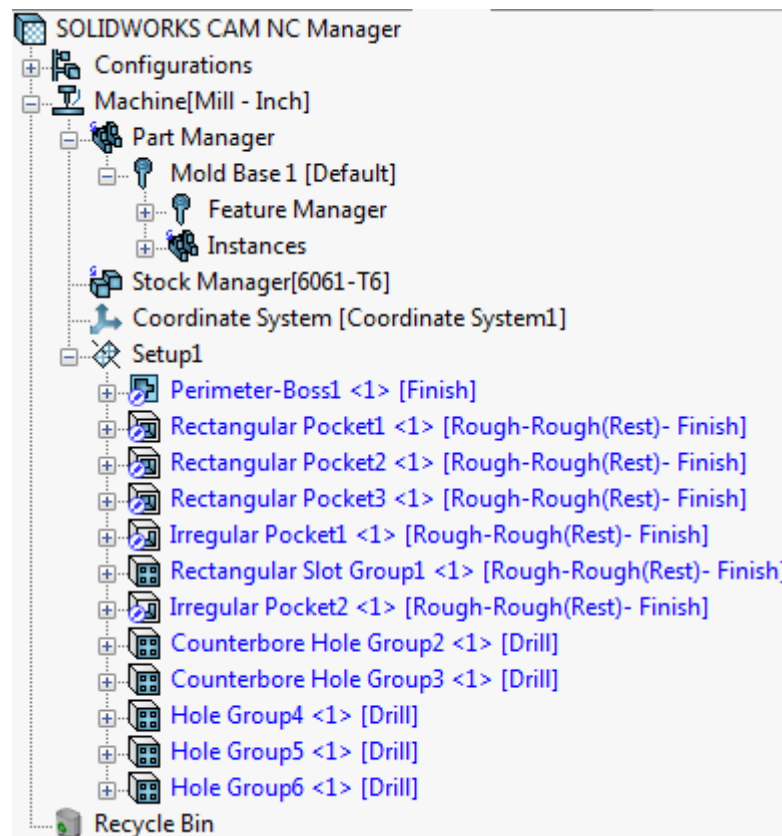
Select the *Extract Machinable Features* command from the SOLIDWORKS CAM menu.

OR

Right-click on the *SOLIDWORKS CAM 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 SOLIDWORKS CAM 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. To define the Fixture Coordinate System, following are the steps:

- a. Click the *Edit* button in the *Fixture Coordinate System* group box. The *Fixture Coordinate System* Dialog box will be displayed.
- b. In the *Method* dropdown group box, select *SOLIDWORKS Coordinate System* from the dropdown list.
- c. In the *Available Coordinate Systems* list box, select *Coordinate System2*. This action will display *Coordinate System2* in the *Selected Coordinate System* field.
- d. Click the  *Ok* button to apply the changes and close this dialog box. The user interface will revert to the *Setup* tab.

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

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

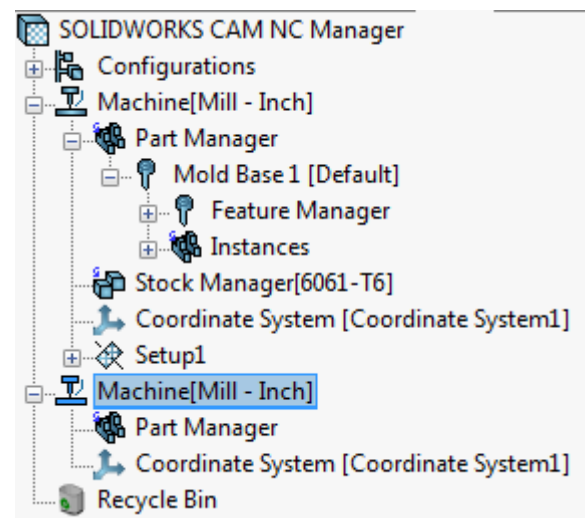
The *Manage Parts* dialog box will be displayed.

7. Pick the six parts on the front table in the order you want them to be machined.

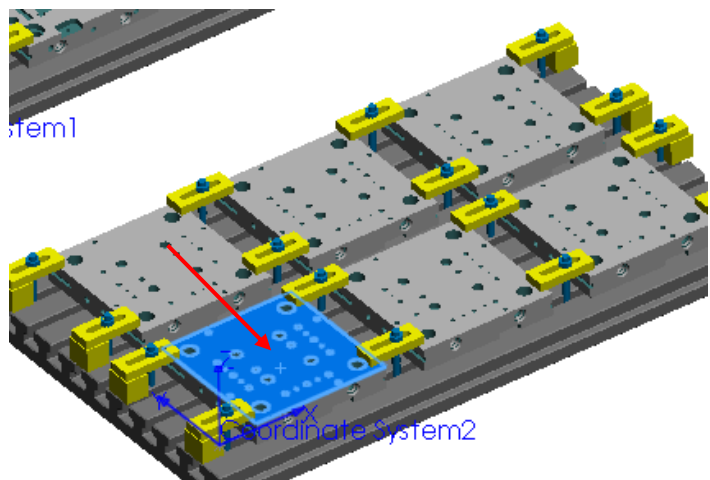
The first part that you pick is referred to as the seed part.

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, SOLIDWORKS CAM displays the features automatically.




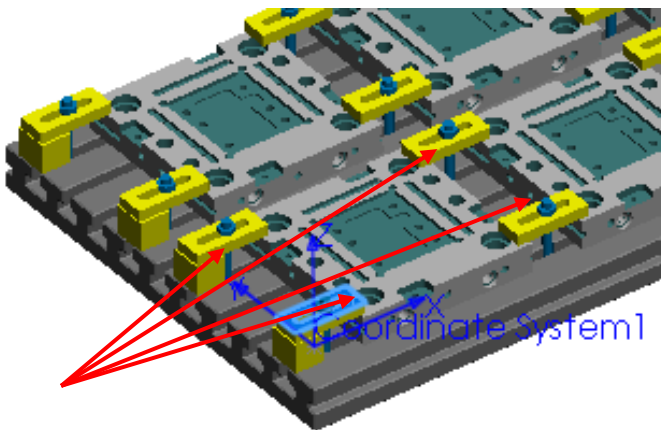
Second Machine created



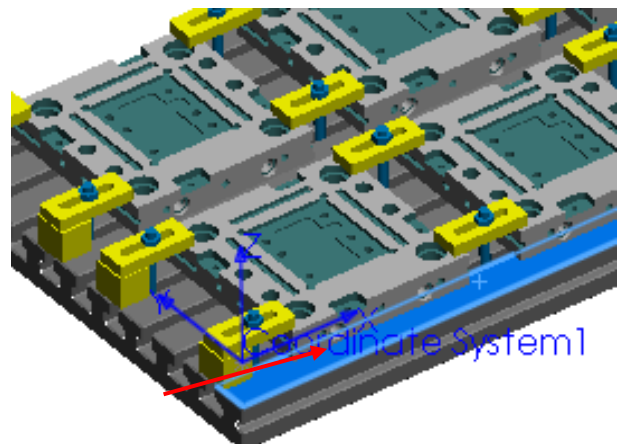
Select all the six part in the graphics area

Step 6: Generating an Operation Plan and Toolpaths

1.  Click the *Generate Operation Plan* button on the SOLIDWORKS CAM Command Manager.
OR
Right click *SOLIDWORKS CAM 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.

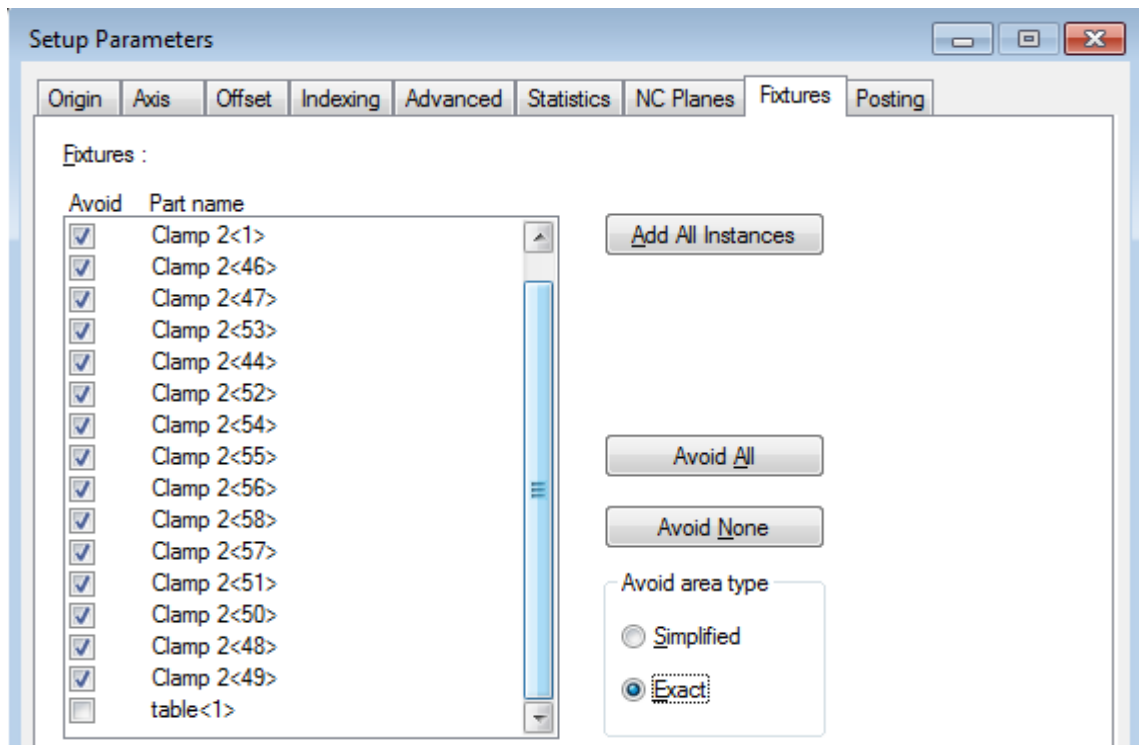


Pick up the four clamps holding the seed part




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, SOLIDWORKS CAM 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, SOLIDWORKS CAM 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.





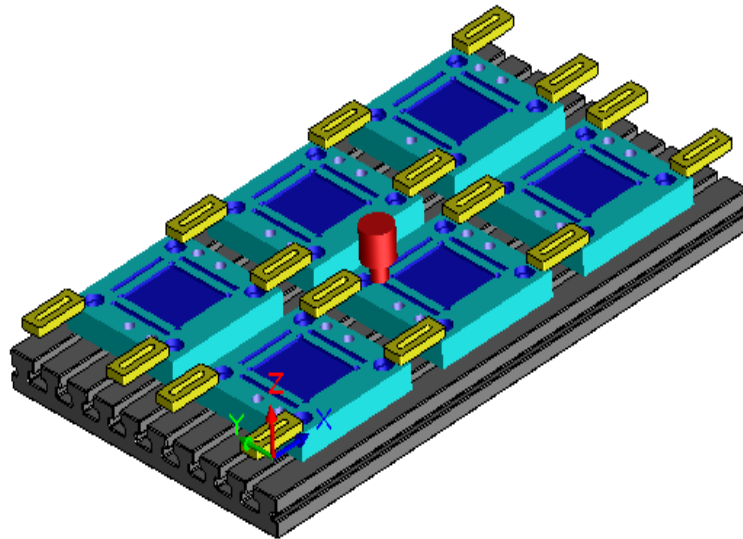
Fixtures tab of the Setup Parameters Dialog Box

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 SOLIDWORKS CAM Command Manager.\
15. Observe that under *Setup1*, toolpaths were not generated for *Contour Mill5* and *Contour Mill10* for *Counterbore Hole Group3* feature and *Contour Mill11* for *Counterbore Hole Group3* feature.
SOLIDWORKS CAM was unable to compute a safe toolpath for these operations 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 Mill11* 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 Mill11* 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 30** 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 close button in the upper right corner of the *Toolpath Simulation* toolbar 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 *Close* button in the upper right corner of the *Toolpath Simulation* toolbar to exit Simulation mode.


Step 8: Post Processing Toolpaths

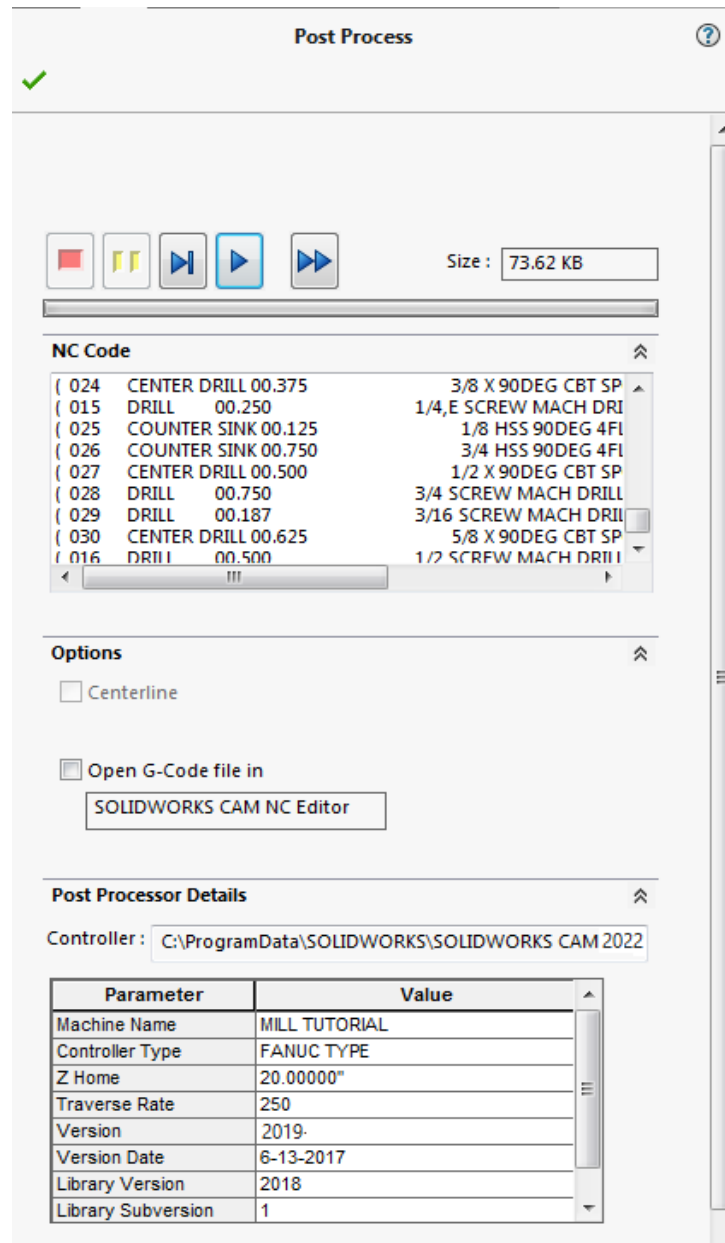
1. Right click the first *Machine* item in the Operation tree and select *Post Process* on the context menu.
 2. The *Post Output File* dialog box is displayed. Browse to the folder where you wish to save the file.
 3. Type *Rear Machine* and click *Save* to save the file.
- SOLIDWORKS CAM generates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

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.



Post Process dialog box (for Setup1)

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/assembly file 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 **SOLIDWORKS CAM** data.

Assembly Tutorial 4

Topics covered in this tutorial:

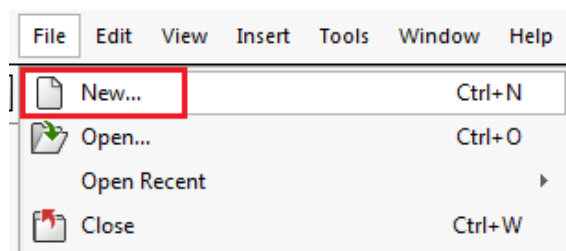
- [Creating the Assembly file from SOLIDWORKS Part files](#)

Simulating Castings

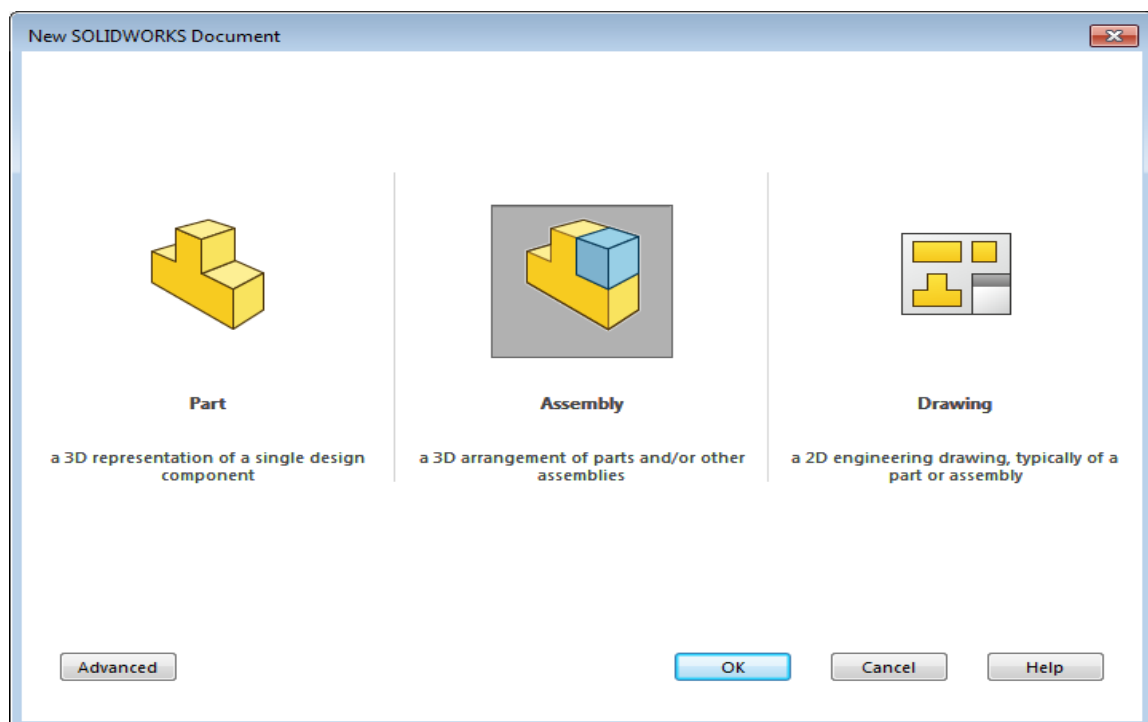
SOLIDWORKS CAM supports irregular shaped stock, such as castings. To define the stock as a casting in Assembly Mode, you can select a SOLIDWORKS part either graphically or from the SOLIDWORKS FeatureManager design tree. Any SOLIDWORKS part can be selected, even a part that is a part to machine.

Step 1: Create the Assembly in SOLIDWORKS



1. Select the Inch Pound Second (IPS) Unit system in SOLIDWORKS.
2. Click on *File* menu on the SOLIDWORKS menu bar and select *New* from the menu bar and select *New* from the menu. Select 'New' on the File menu dropdown menu. The New SOLIDWORKS Document dialog box will be displayed.
3. Click on *Assembly* option and click *OK* to close the dialog box.

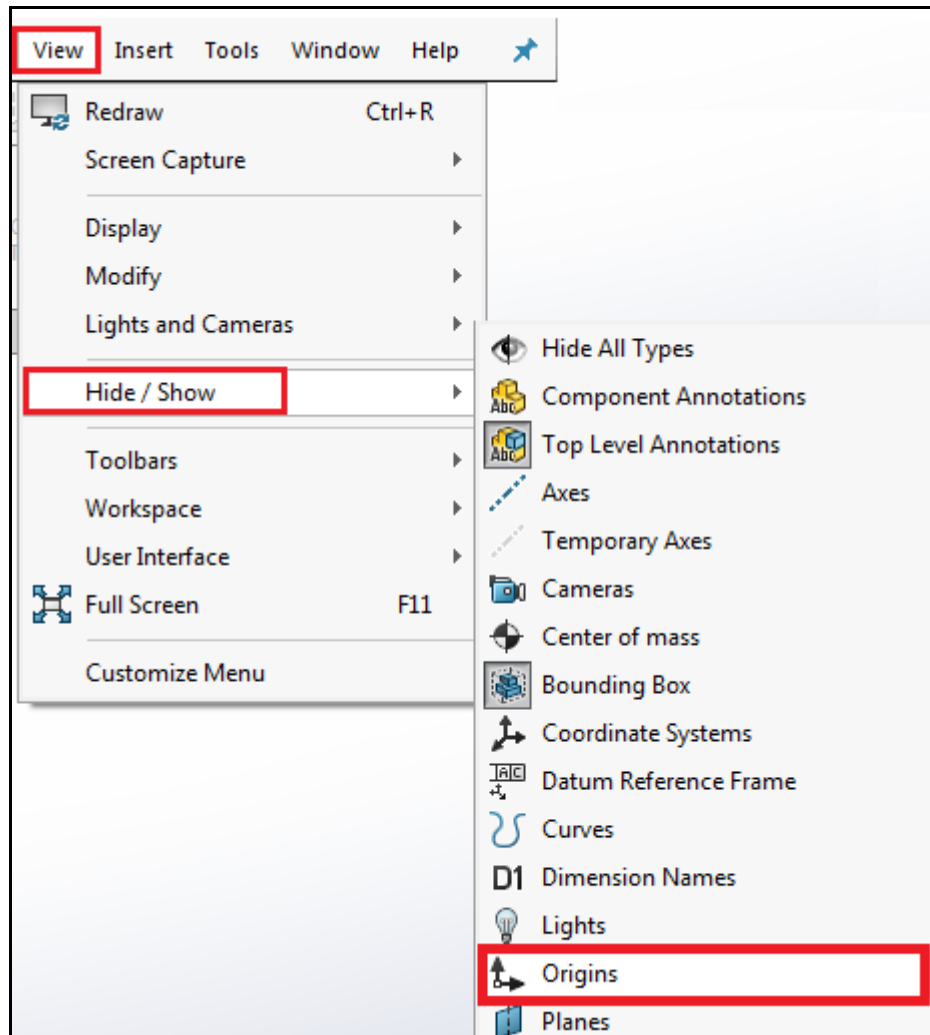


Select 'New' on the File menu



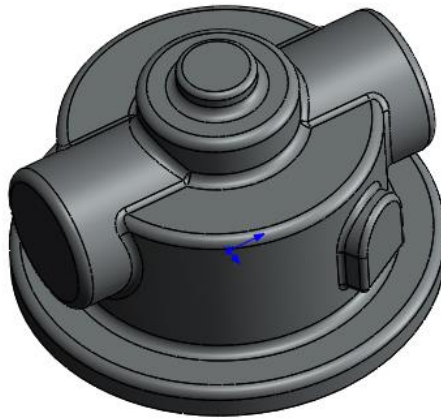
New SOLIDWORKS Document dialog box

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 menu bar, click Hide/show>>  *Origins* so the assembly origin can be seen. If needed, zoom out to see the assembly origin in the graphics area.

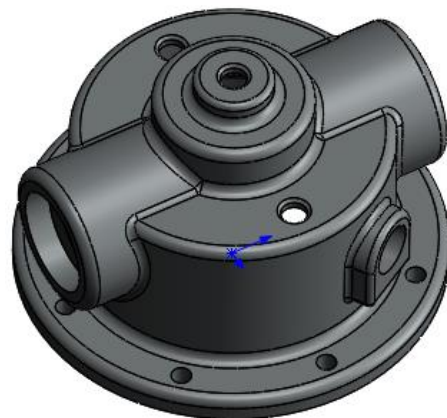


Select 'Origins' on View menu


7. Click *Browse* button in the *Begin Assembly* dialog box and open **MILL2AX_14 As-Cast.SLDPRT** from the following folder location.
C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 202x\CAM Examples \Tutorial_Parts\Mill
8. Position the mouse pointer on the origin and left click to position the part on the origin.

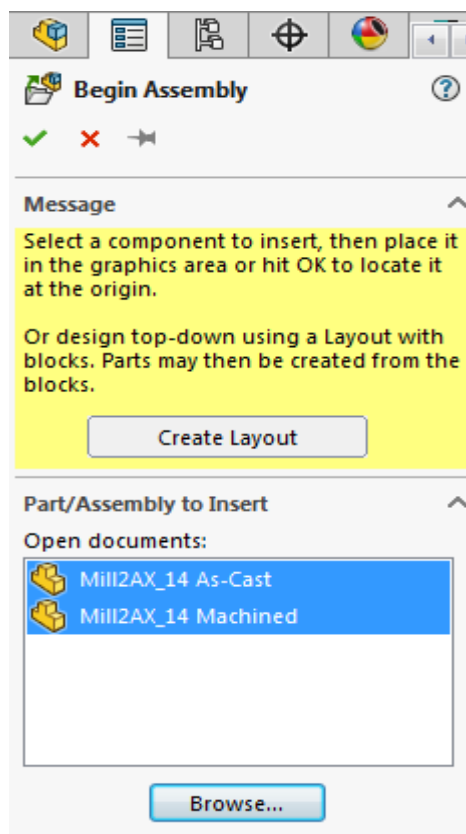


MILL2AX_14 As-Cast.sldprt



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

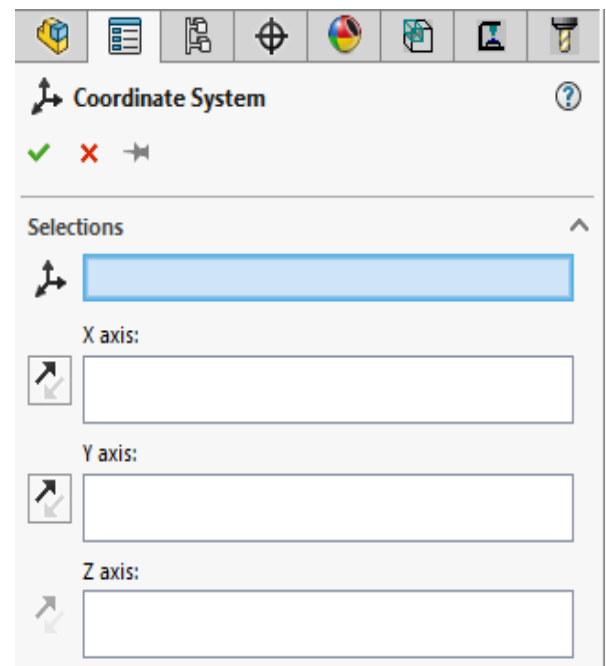
14. Click *Insert* on the SOLIDWORKS menu bar and select *Reference Geometry*, then *Coordinate System*.

The *Coordinate System* dialog box will be displayed.

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

15. Click OK.

If no selection is made, the origin of the Coordinate System is the SOLIDWORKS 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

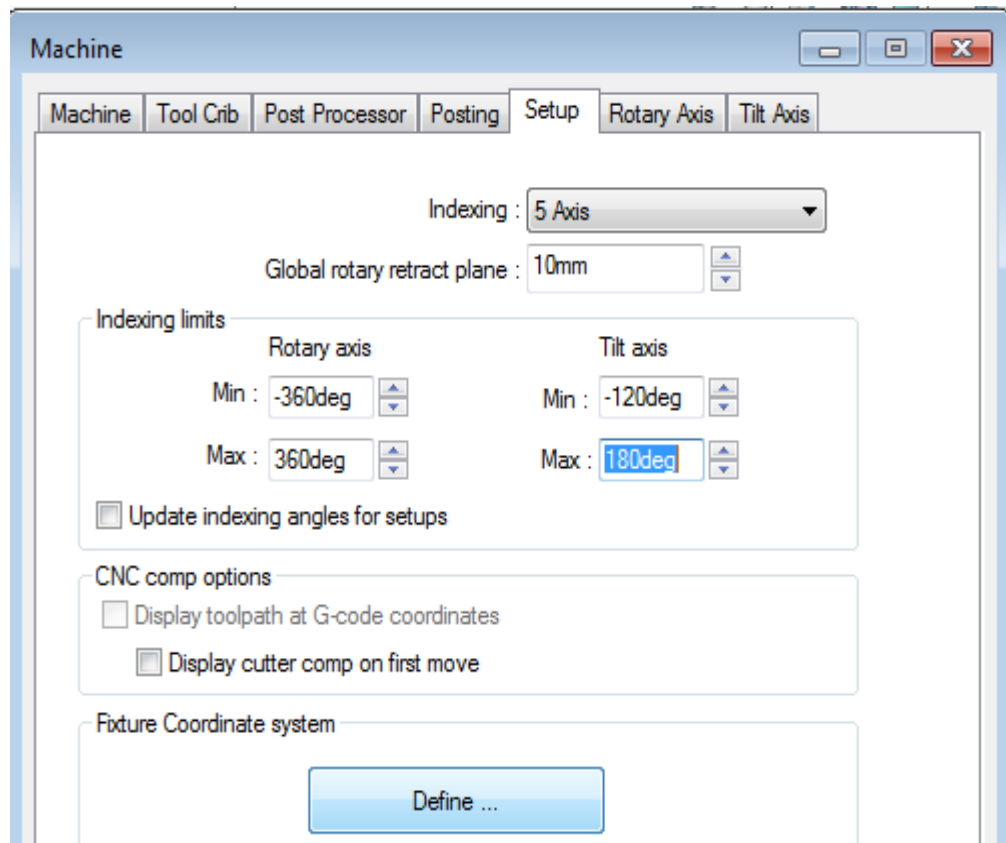
1. Click the *SOLIDWORKS CAM Feature Tree* tab.
2. Double click the *Machine* item in the Feature tree.

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 Define button on Fixture Coordinate System group box. Fixture Coordinate System dialog box is displayed. In the Method group box, select SOLIDWORKS Coordinate System from the drop down list, and highlight Coordinate System 1 in the Available Coordinate Systems. Click OK.



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.

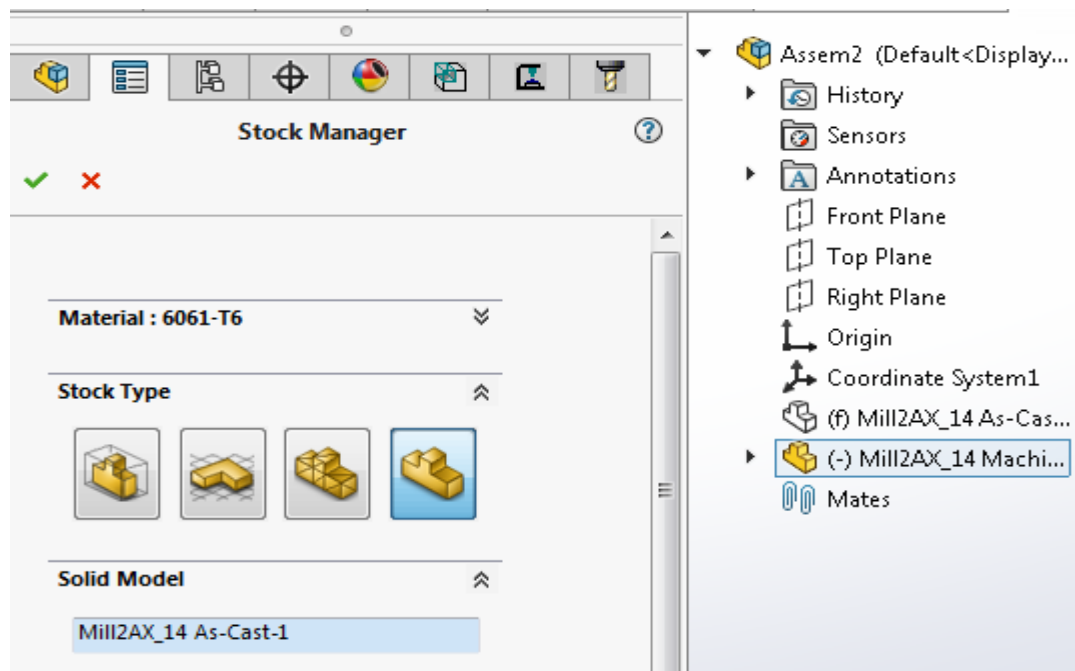
12. Click the *Tilt Axis* tab.
13. In the *Tilt axis is* group box, select *Y axis* to define the tilt axis.
14. In the *0 degree position* group box, click *XY plane*, if not already selected.
15. Click *OK* to apply the changes and close the *Machine* dialog box.

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.




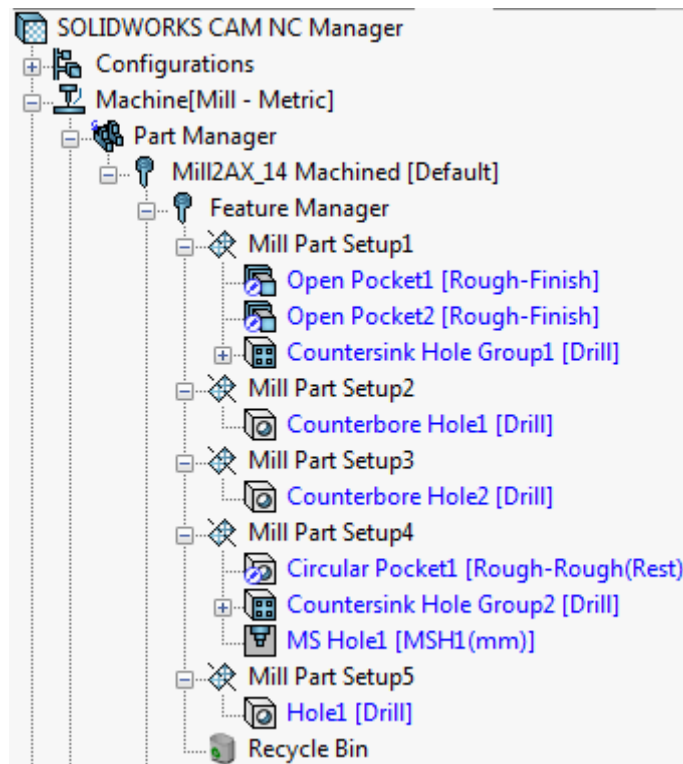
Select 'Mill2AX_14 As-Cast' in Graphics area

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 SOLIDWORKS CAM Command Manager.

2.  Expand the *Feature Manager* item to show all the Mill Part Setups and features found by AFR.



Features recognized after executing the EMF Command

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.

This feature is inside the casting and will not be machined.

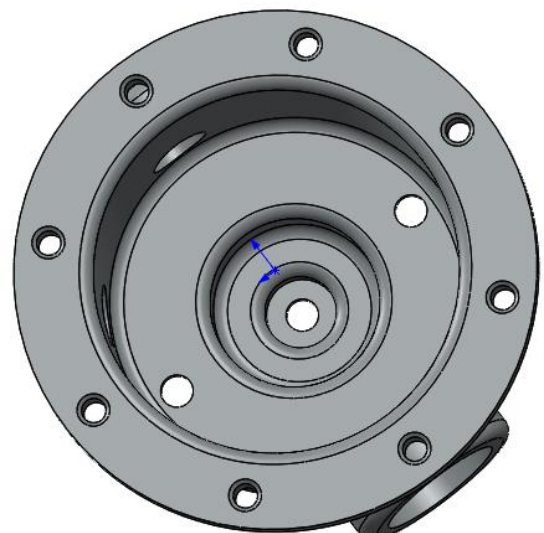
5. Click *Yes* to confirm the deletion.

6. In the graphics area, rotate the part so you can see the bottom face of the part.

7.  Right click *Mill Part Setup4* in the Feature tree and select *2.5 Axis Feature* on context menu.

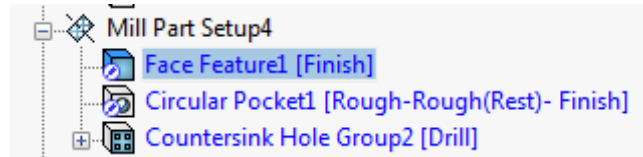
The *2.5 Axis Feature: Select Entities* dialog box will be displayed. In this dialog box:

- 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*.



Bottom face of the part

- 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.
8. Click *OK* to complete the addition of the new Face Feature.
 9. 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 to top of Setup4

Step 6: Generating an Operation Plan and Toolpaths

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

OR

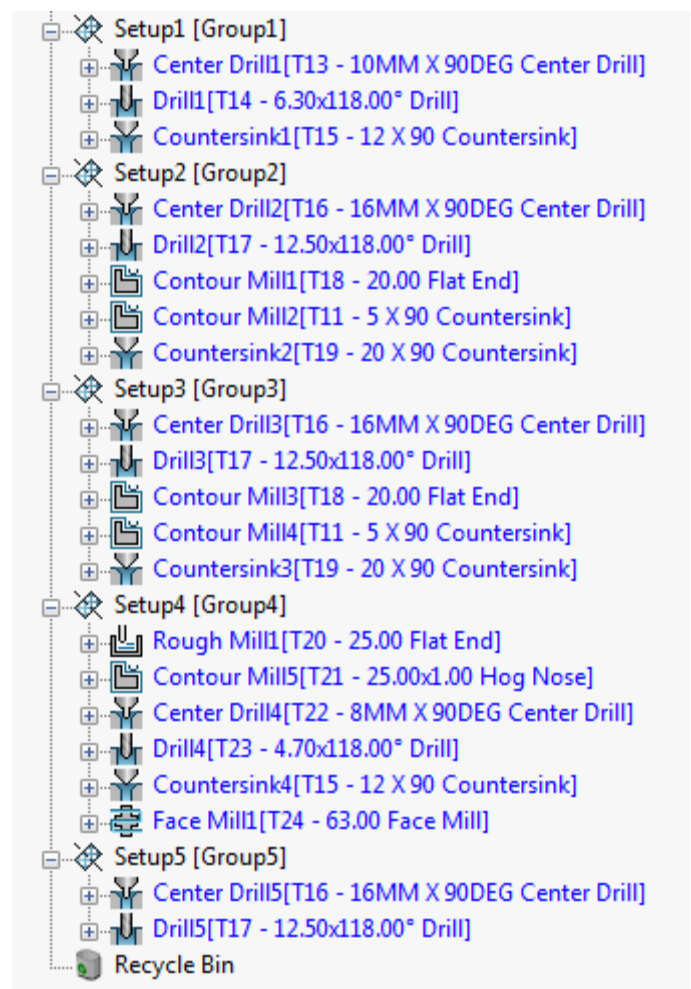
Right click *SOLIDWORKS CAM NC Manager* in the Feature tree and select *Generate Operation Plan* on the context menu.

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

2. Click the *SOLIDWORKS Feature Manager 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 *SOLIDWORKS CAM Operation Tree* tab.
6. Click *Generate Toolpath* on the SOLIDWORKS CAM Command Manager.

OR

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



Generated operations

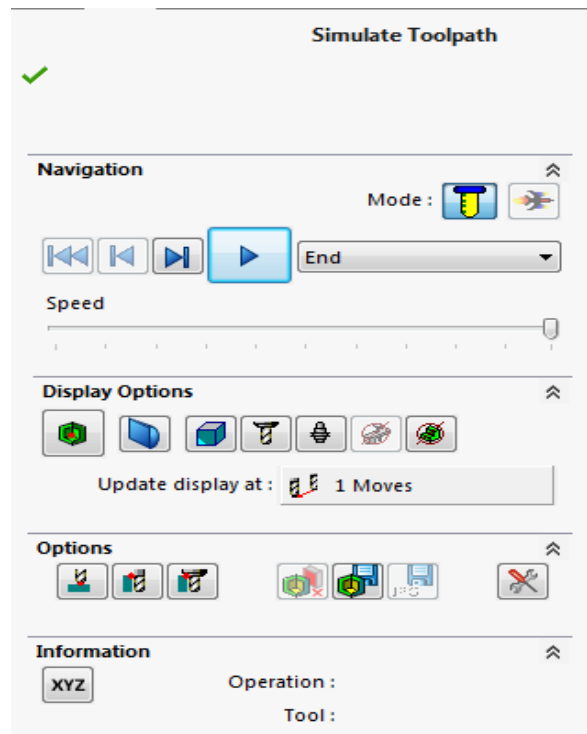
Step 7: Simulating Toolpaths

1. Click *Simulate Toolpath* on the SOLIDWORKS CAM Command Manager.

OR

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

The *Toolpath Simulation* toolbar will be displayed.




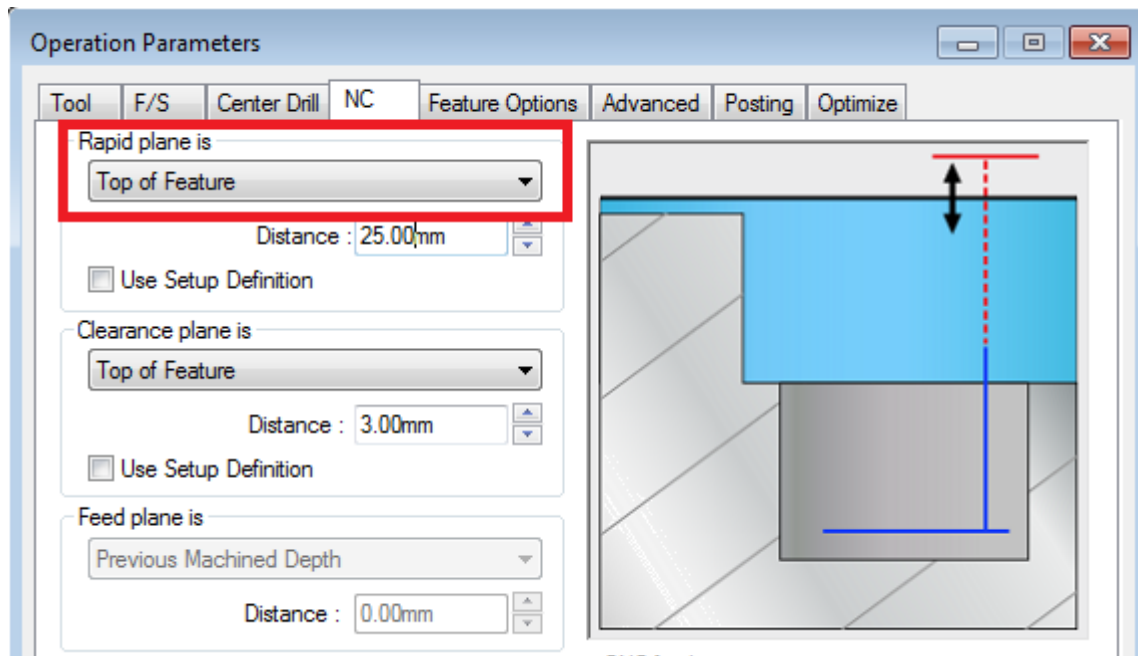
Simulate Toolpath

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 close button in the upper right corner of the *Toolpath Simulation* toolbar 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 SOLIDWORKS CAM 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 SOLIDWORKS CAM 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.

LEGAL NOTICES

© 1995-2025, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Patent Notices

- SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents
- 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072; 9,262,863; 9,465,894; 9,646,412; 9,870,436; 10,055,083; 10,073,600 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).
- eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.
- U.S. and foreign patents pending.

Trademarks and Product Names for SOLIDWORKS Products and Services

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of HCL Technologies Ltd.

SOLIDWORKS 2025, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, SOLIDWORKS Flow Simulation, SOLIDWORKS CAM, SOLIDWORKS Manage, eDrawings Viewer, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic Professional, SOLIDWORKS Electrical 3D, SOLIDWORKS Electrical Professional, CircuitWorks, SOLIDWORKS Composer, SOLIDWORKS Inspection, SOLIDWORKS MBD, SOLIDWORKS PCB powered by Altium, SOLIDWORKS PCB Connector powered by Altium, and SOLIDWORKS Visualize are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective Holders.

COMMERCIAL COMPUTER SOFTWARE – PROPRIETARY

The Software is a “commercial item” as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of “commercial computer software” and “commercial software documentation” as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government (a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R. 227.7202-1 (JUN 1995) and 227.7202-4 (JUN 1995).

In the event that you receive a request from any agency of the U.S. Government to provide Software with rights beyond those set forth above, you will notify DS SolidWorks of the scope of the request and DS SolidWorks will have five (5) business days to, in its sole discretion, accept or reject such request. Contractor/Manufacturer: Dassault Systemes SolidWorks Corporation, 175 Wyman Street, Waltham, Massachusetts 02451 USA.

Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products

- Portions of this software © 1986-2025 Siemens Product Lifecycle Management Software Inc. All rights reserved.
- This work contains the following software owned by Siemens Industry Software Limited:
 - ▀ D-Cubed® 2D DCM © 2025. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® 3D DCM © 2025. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® PGM © 2025. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® CDM © 2025. Siemens Industry Software Limited. All Rights Reserved.
 - ▀ D-Cubed® AEM © 2025. Siemens Industry Software Limited. All Rights Reserved.
- Portions of this software © 1998-2025 HCL Technologies Ltd.
- Portions of this software incorporate PhysX™ by NVIDIA 2006-2010.
- Portions of this software © 2001-2018 Luxology, LLC. All rights reserved, patents pending.
- Portions of this software © 2007-2018 DriveWorks Ltd.
- © 2011, Microsoft Corporation. All rights reserved.
- Includes Adobe® PDF Library technology
Copyright 1984-2016 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382;
- Patents Pending.

Copyright Notices for SOLIDWORKS Simulation Products

- Portions of this software © 2008 Solversoft Corporation.
- PCGLSS © 1992-2020 Computational Applications and System Integration, Inc. All rights reserved.

Copyright Notices for SOLIDWORKS PDM Professional Product

- Outside In® Viewer Technology, © 1992-2012 Oracle
- © 2011, Microsoft Corporation. All rights reserved.

Copyright Notices for eDrawings Products

- Portions of this software © 2000-2014 Tech Soft 3D.

- Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.
- Portions of this software © 1998-2001 3Dconnexion.
- Portions of this software © 1998-2017 Open Design Alliance. All rights reserved.
The eDrawings[®] for Windows[®] software is based in part on the work of the Independent JPEG Group.
- Portions of eDrawings[®] for iPad[®] copyright © 1996-1999 Silicon Graphics Systems, Inc.
- Portions of eDrawings[®] for iPad[®] copyright © 2003 – 2005 Apple Computer Inc.

Copyright Notices for SOLIDWORKS PCB Products

- Portions of this software © 2017-2018 Altium Limited.

Copyright Notices for SOLIDWORKS Visualize Products

- NVIDIA GameWorks™ Technology provided under license from NVIDIA Corporation. Copyright © 2002-2015 NVIDIA Corporation. All rights reserved