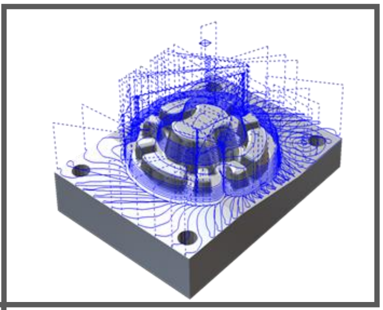
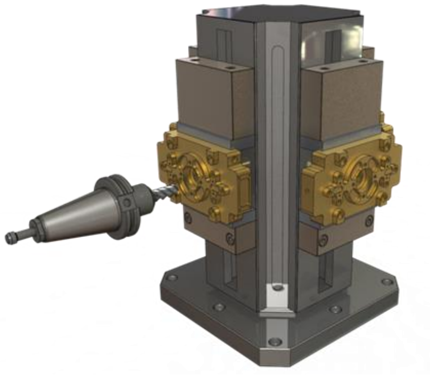
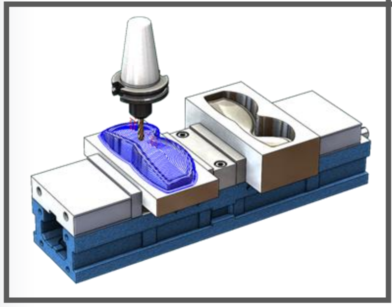
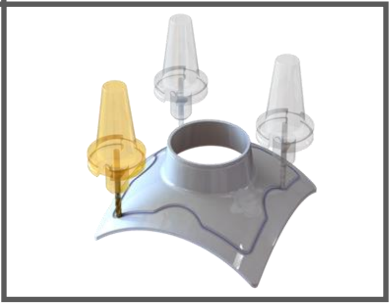
CAMWorks 2022

Mill Tutorial









Disclaimer

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

Trademarks

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

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

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

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

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

CERTIFIED
Gold
Product
SOLIDWORKS

Geometric
An HCL Technologies Company

Table of Contents

Chapter 1: Learning 2 Axis Mill Basics

This chapter provides an opportunity to learn CAMWorks 2 Axis Mill through a step by step hands-on tour of the features and functions.

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

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

(Typical Location: Drive: *\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill*).

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

The following series of exercises show you how to generate finish toolpaths on a SOLIDWORKS/CAMWorks Solids part model.

In order to give you a general understanding of how to use CAMWorks, you will work with a part that was previously modeled in SOLIDWORKS. When you define the operations and toolpaths, you will follow steps and instructions. These steps and instructions are brief in nature in order to show you the basics of generating toolpaths from start to finish without getting into the details at this time. In case you wish to have deeper understanding of the functionalities within CAMWorks, we recommend that you refer to **CAMWorks Context-based Help**. This context-sensitive Help is displayed when you click on the Help button of the various dialog boxes or User Interfaces within the application.

Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically. These sample parts are installed in the *Examples\ Tutorial_Parts* folder located within the CAMWorks installation folder. Sample parts for Mill are located in the *Mill* subfolder within the *Examples\ Tutorial_Parts* folder.

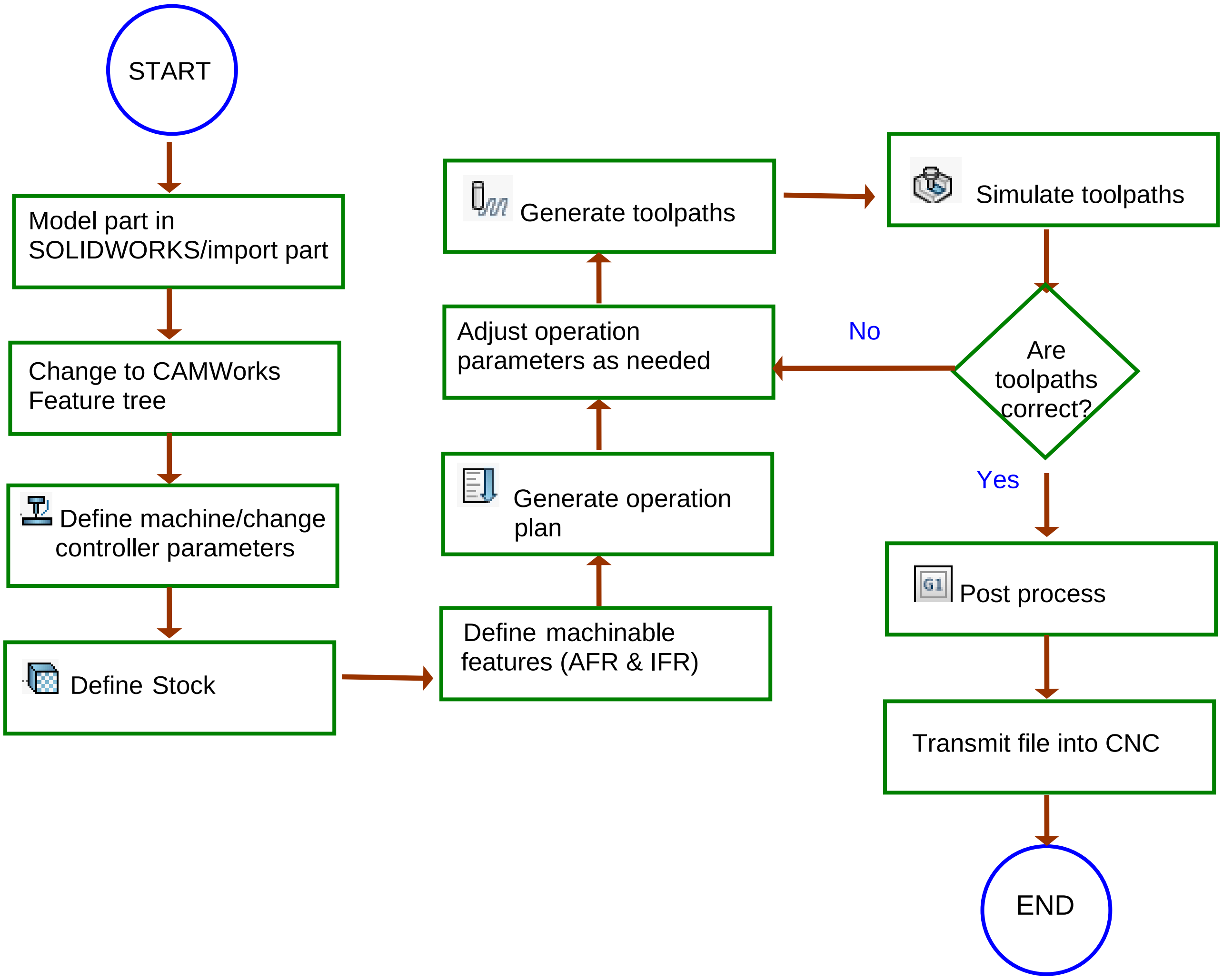
2 Axis Mill 1

Topics covered in this tutorial:

Steps to Generate Mill Toolpaths and NC Code in Part Mode

The following steps are used to generate Mill toolpaths and NC code:

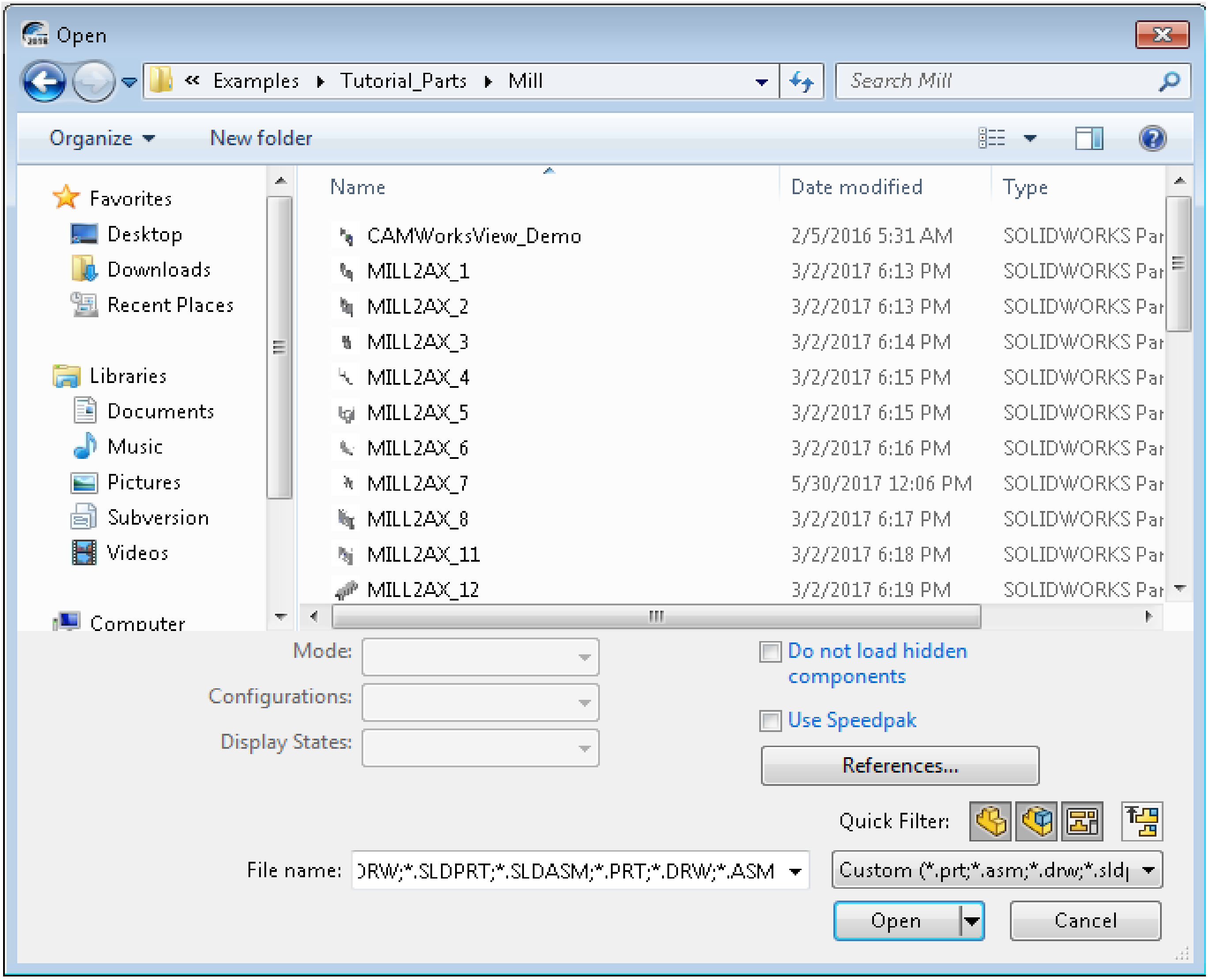
1. Model the part or open the part file in SOLIDWORKS/CAMWorks Solids.
2. Click on the CAMWorks Feature tree.
3. Define the Machine and modify the controller parameters.
4. Define the stock.
5. Define machinable features.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Simulate material removal.
9. Post process the toolpaths.


START
Model part in SOLIDWORKS/import part
Change to CAMWorks Feature tree
Define machine/change controller parameters
Define Stock
Define machinable features (AFR & IFR)
Generate operation plan
Adjust operation parameters as needed
Generate toolpaths
Simulate toolpaths
Are toolpaths correct?
No
Yes
Post process
Transmit file into CNC
END

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

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

Open the part file **MILL2AX_1.SLDPRT** in the following folder. *Drive:\CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill.*



Name	Date modified	Type
CAMWorksView_Demo	2/5/2016 5:31 AM	SOLIDWORKS Par
MILL2AX_1	3/2/2017 6:13 PM	SOLIDWORKS Par
MILL2AX_2	3/2/2017 6:13 PM	SOLIDWORKS Par
MILL2AX_3	3/2/2017 6:14 PM	SOLIDWORKS Par
MILL2AX_4	3/2/2017 6:15 PM	SOLIDWORKS Par
MILL2AX_5	3/2/2017 6:15 PM	SOLIDWORKS Par
MILL2AX_6	3/2/2017 6:16 PM	SOLIDWORKS Par
MILL2AX_7	5/30/2017 12:06 PM	SOLIDWORKS Par
MILL2AX_8	3/2/2017 6:17 PM	SOLIDWORKS Par
MILL2AX_11	3/2/2017 6:18 PM	SOLIDWORKS Par
MILL2AX_12	3/2/2017 6:19 PM	SOLIDWORKS Par

File name:)RW;*.SLDPRT;*.SLDASM;*.PRT;*.DRW;*.ASM

Custom (*.prt;*.asm;*.drw;*.sld|

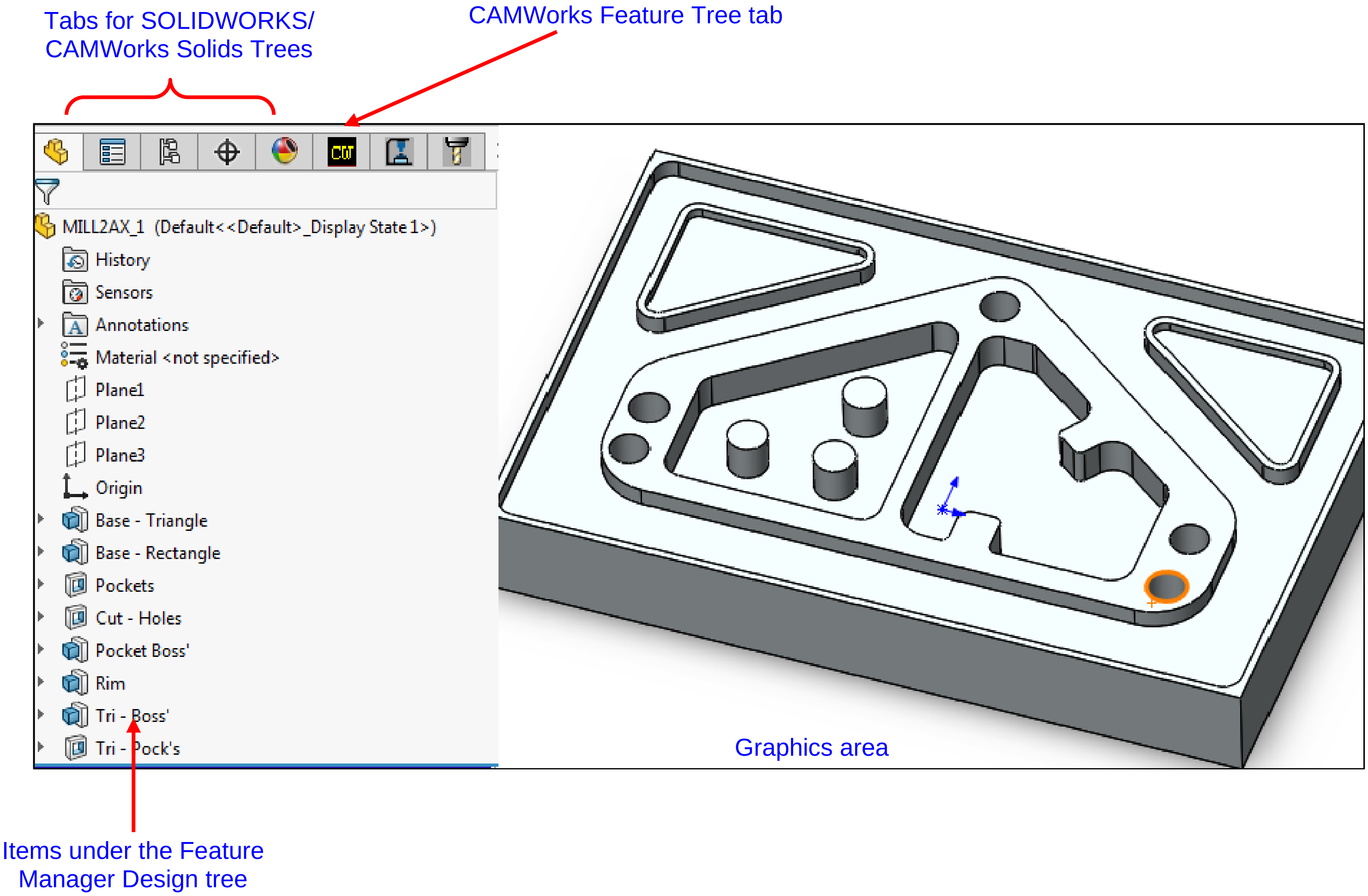
Opening the Solid Part file

Viewing the FeatureManager Design Trees

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

To use CAMWorks, you need to move between SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Click the *Pin* button to continuously view this Tree area.

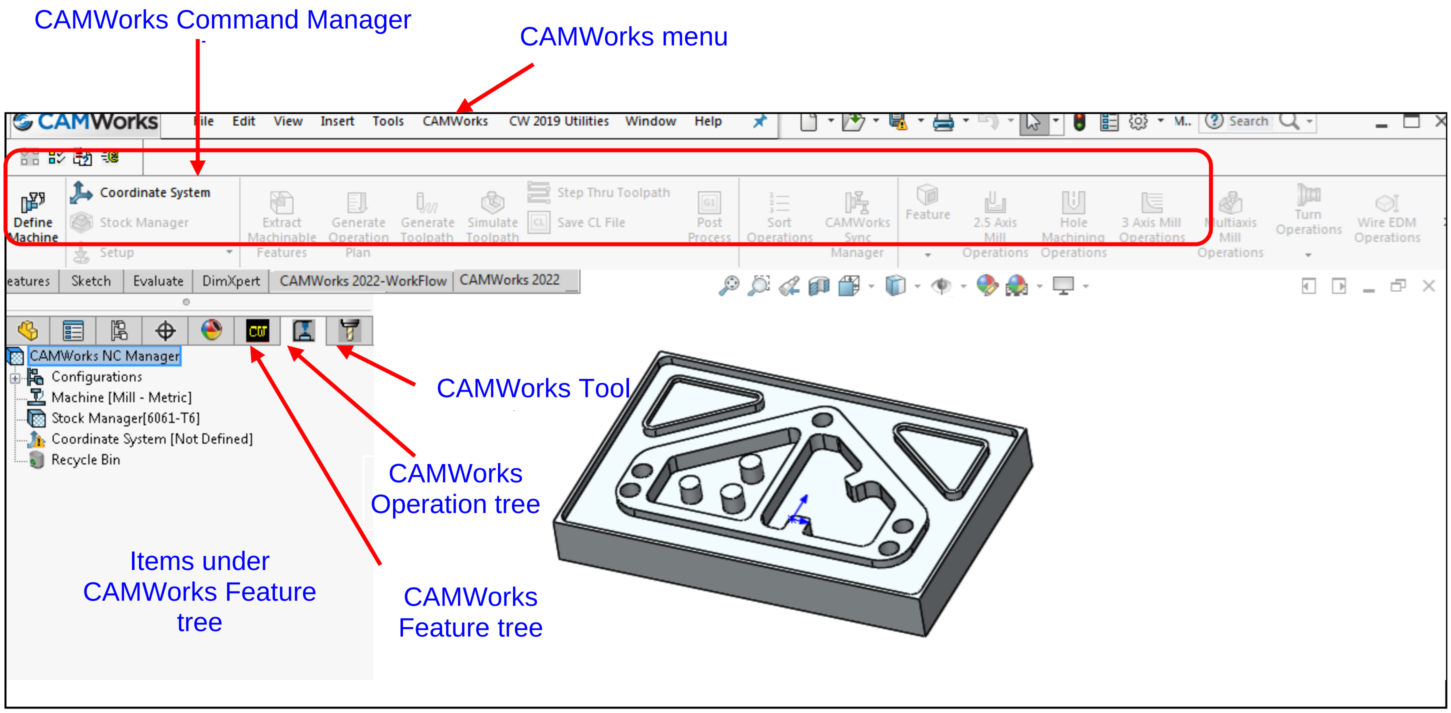
If the CAMWorks tabs [, ,] are not visible, you can expand the size of the tree. Position the cursor on the line that divides the tree area from the graphics area. When the cursor changes to a bar, drag the bar to the right until the tabs display.



Step 2: Change to CAMWorks Feature Tree

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

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



CAMWorks Machining Trees

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

- **Configurations**

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

- **Stock Manager**

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

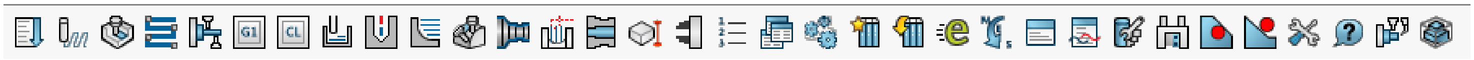
- **Machine**

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

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

CAMWorks Command Manager

Click *Tools* on the SOLIDWORKS menu bar and select *CAMWorks* from the dropdown menu./Click *CAMWorks* on the *CAMWorks Solids* menu bar. This action displays the CAMWorks Command Manager. It provides access to the main CAMWorks commands. The commands are explained in the CAMWorks context-based Help.



CAMWorks Command Manager

Customization of CAMWorks Command Manager

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

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

Alternative Access to CAMWorks Commands

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

CAMWorks Options

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

To open the *CAMWorks Options* dialog box:

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

OR

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

2. The *Options* dialog box will be displayed. In this dialog box, go to the *Mill Features* tab. Under *Extract Machinable Features* group box, ensure that the *Method* is set to *MfgView* (default setting).
3. Click *OK* to apply the changes and close the dialog box.

CAMWorks Context-based Help

In addition to tutorial documents, CAMWorks is provided with a context-based Help. Every dialog box and interface within CAMWorks has an associated *Help* button. Click on the *Help* button on the CAMWorks Command Manager to open the context-based Help. Every parameter and tab of each dialog box in explained in the context-based Help.

Step 3: Define the Machine

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

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

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

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

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

- **Tool crib:** A subset of tools from the tool library that are commonly loaded into or used with the current machine.
- **Post Processor:** The post processor identifies the format of the NC G-code output.

Define the Machine

1. Right click *Machine [Mill – Metric]* in the CAMWorks Feature tree and select *Edit Definition* or double click the item in the tree.

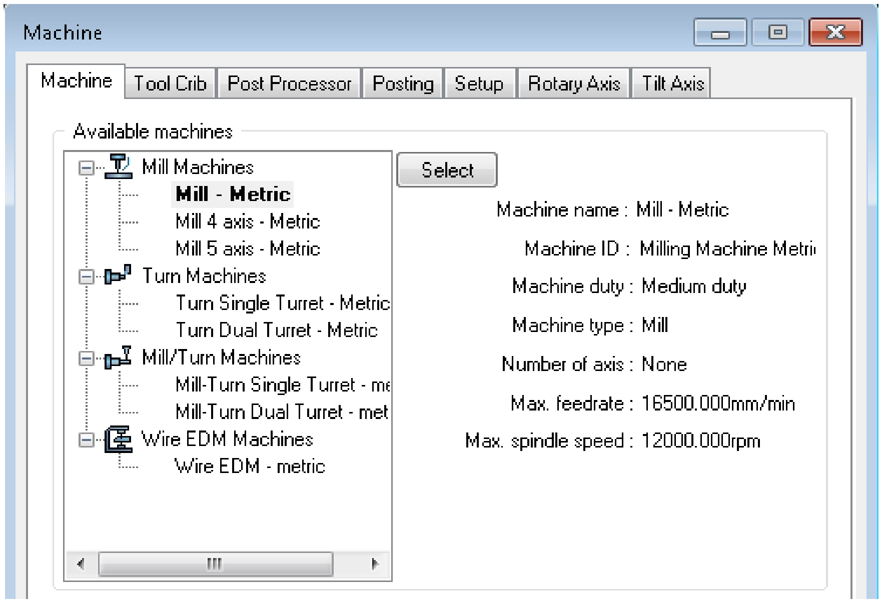
Did You Know ...

In the Feature and Operation trees, instead of right clicking items and selecting *Edit Definition*, you can double-click the item to open the dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.

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

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

2. In the Available machines list, highlight *Mill–Metric* and click the *Select* button.



Machine name : Mill - Metric
Machine ID : Milling Machine Metri
Machine duty : Medium duty
Machine type : Mill
Number of axis : None
Max. feedrate : 16500.000mm/min
Max. spindle speed : 12000.000rpm

Machine tab of Machine Dialog Box

3. Click the *Tool crib* tab and make sure *Tool Crib 2 (Metric)* is the Active tool crib.

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

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

4. Click the *Post Processor* tab.

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

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

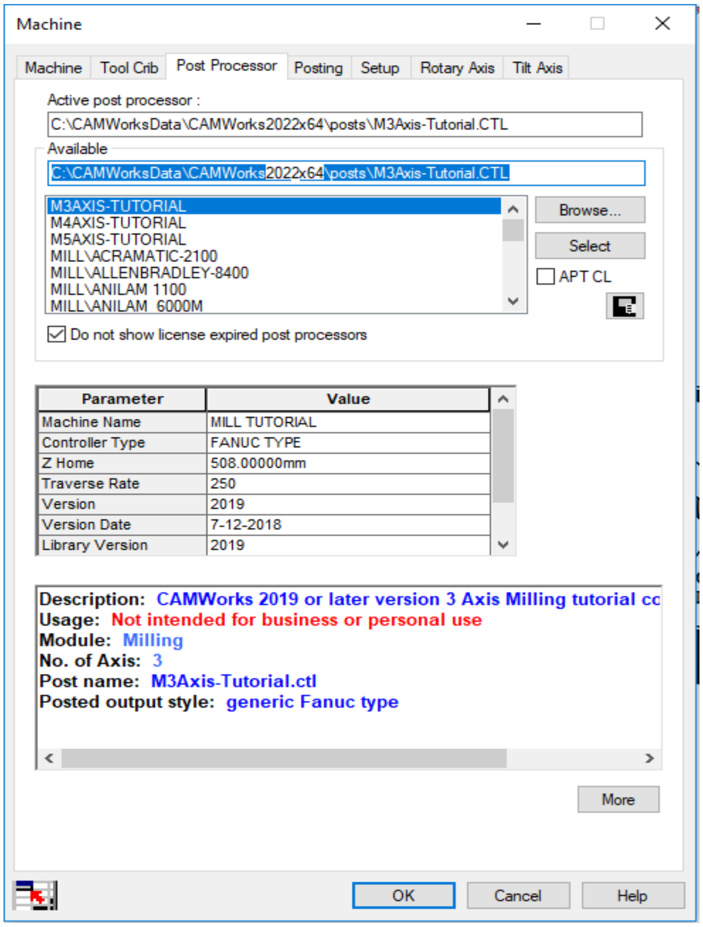
If the post processors do not display, use the *Browse* button to locate the folder containing the files (*.ctl).

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

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

6. Click the *More* button.

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



Active post processor : C:\CAMWorksData\CAMWorks2022x64\posts\M3Axis-Tutorial.CTL

Available: C:\CAMWorksData\CAMWorks2022x64\posts\M3Axis-Tutorial.CTL

M3AXIS-TUTORIAL
M4AXIS-TUTORIAL
M5AXIS-TUTORIAL
MILL\ACRAMATIC-2100
MILL\ALLENBRADLEY-8400
MILL\ANILAM 1100
MILL\ANILAM 6000M

☐ APT CL

☑ Do not show license expired post processors

Parameter	Value
Machine Name	MILL TUTORIAL
Controller Type	FANUC TYPE
Z Home	508.00000mm
Traverse Rate	250
Version	2019
Version Date	7-12-2018
Library Version	2019

Description: CAMWorks 2019 or later version 3 Axis Milling tutorial cc
Usage: Not intended for business or personal use
Module: Milling
No. of Axis: 3
Post name: M3Axis-Tutorial.ctl
Posted output style: generic Fanuc type

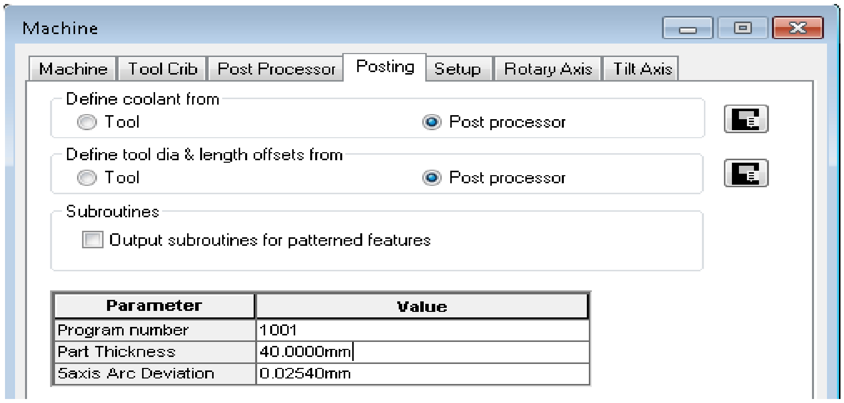
Post Processor tab of Machine Dialog Box

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

7. Click the *Posting* tab.

   The parameters on this page are used for the following:

   – To provide information required to generate the NC program. The parameters are machine-dependent and different parameters may display for your controller. The value for a parameter is output in the NC code if the machine requires it.



Define coolant from
○ Tool ● Post processor

Define tool dia & length offsets from
○ Tool ● Post processor

Subroutines
☐ Output subroutines for patterned features

Parameter	Value
Program number	1001
Part Thickness	40.0000mm
5axis Arc Deviation	0.02540mm

Posting tab of Machine Dialog Box

   – To provide information for the Setup Sheet, a file that is created when the NC program file is generated. All of the controller parameters are included in the Setup Sheet.

8. Type **1001** for the Program Number and press the down arrow on the keyboard to shift the focus to the Part Thickness field.
9. Type **40mm** for the Part Thickness.
10. Click *OK* to close the machine dialog box.

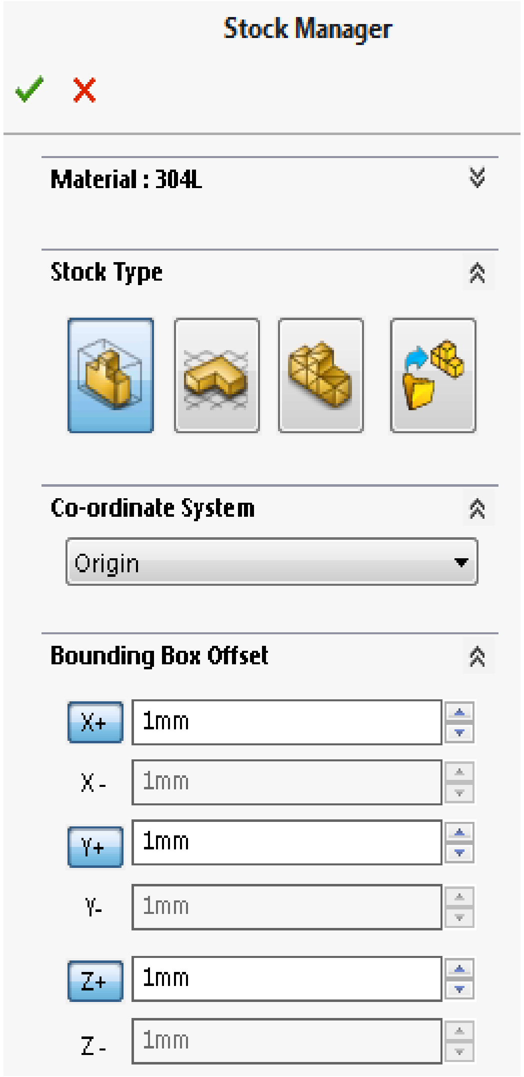
Step 4: Define the Stock

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

In this tutorial, you define the stock as a box offset *(bounding box)* from the part.

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

   The Stock Manager dialog box is displayed.
2. For the Bounding box offset, type **1mm** for X+ and click the X+ button (Uniform X).
3. Repeat step 2 for Y+ and Z+. To set these entered values as default values for future jobs, click on the *Set default* button. Use the *Get default* button to set the offset values to pre-defined default values.
4. Select ***304L*** as the material from the *Material* dropdown list.
5. ✓ Click *OK* to close the dialog box.



Stock Manager

Material : 304L

Stock Type

Co-ordinate System: Origin

Bounding Box Offset

X+ 1mm
X- 1mm
Y+ 1mm
Y- 1mm
Z+ 1mm
Z- 1mm

Stock Manager Dialog box

Step 5: Define Machinable Features

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

- **Automatic Feature Recognition (AFR)**
  Automatic Feature Recognition analyzes the part shape and attempts to define most common machinable features such as pockets, holes, slots and bosses. Depending on the complexity of the part, AFR can save considerable time in defining two-dimensional prismatic features.

  Did You Know ...

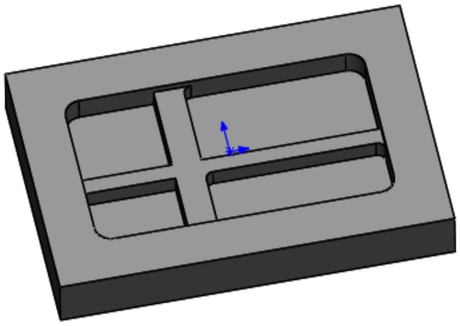
  You can select the type of 2.5 features that you want AFR to recognize. Open the Options dialog box. You can select the type of features to be recognized by selecting the desired features in the Feature Types group box on the Mill Features tab of this dialog box.

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

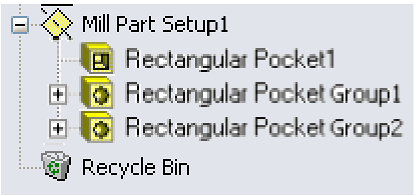
How AFR analyses a Solid Part

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

CAMWorks does not machine the SOLIDWORKS or CAMWorks Solids features directly. Instead, it creates a separate list of Machinable Features. This is because a single SOLIDWORKS feature may have several areas that need to be machined in different ways with different tools.



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



Mill Part Setup1
  Rectangular Pocket1
  Rectangular Pocket Group1
  Rectangular Pocket Group2
Recycle Bin

After AFR is run on this part, the CAMWorks Feature tree would look the one given on the right. There are 3 Machinable Features in the list: one for the large pocket on top, and 2 for the two different types of pockets around the ribs. This gives you more flexibility for machining. Using Automatic Feature Recognition (AFR)

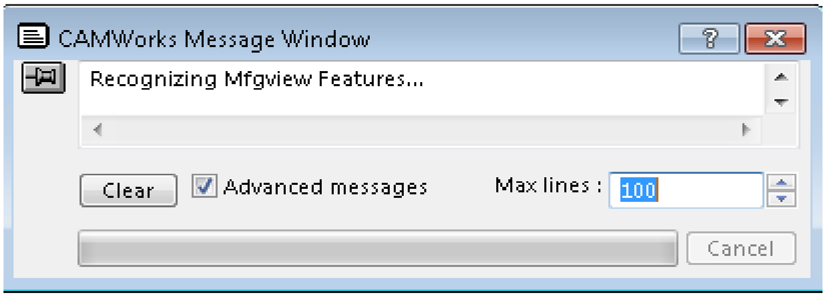
Defining machinable features automatically:

In the Feature tree, right click *CAMWorks NC Manager* and select *Extract Machinable Features* on the RMB context menu.

OR

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

The *CAMWorks Message Window* displays automatically to report the progress and status of the process.



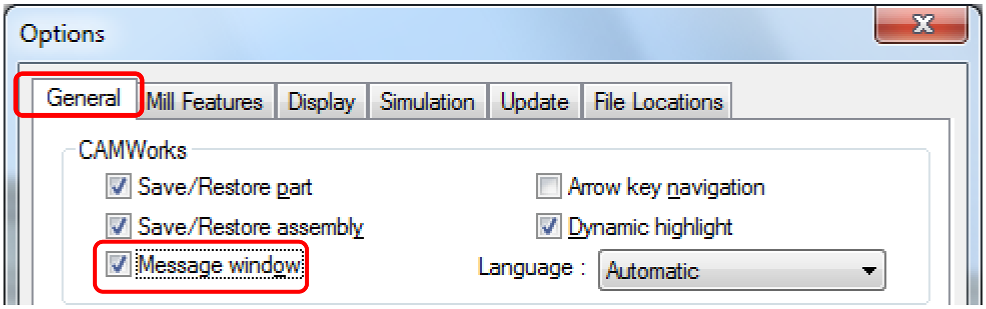
Recognizing Mfgview Features...

☑ Advanced messages

Max lines : 100

CAMWorks Message Window

You can control whether this window displays temporarily or permanently. Click on the *CAMWorks Options* button on the CAMWorks Command Manager. The *Options* dialog box is displayed. Check the *Message Window* option on the *General* tab of this dialog box to view this window continuously.



☑ Save/Restore part

☐ Arrow key navigation

☑ Save/Restore assembly

☑ Dynamic highlight

☑ Message window

Language : Automatic

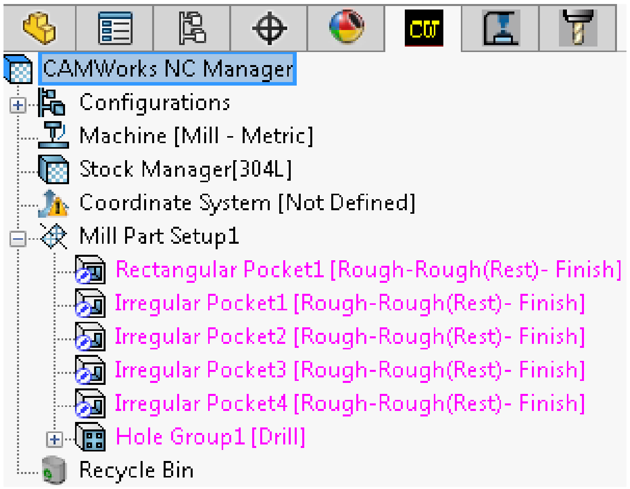
General Tab of CAMWorks Options dialog box

CAMWorks generates the Mill Part Setup and machinable features.

The items display in the *Feature* tree.

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

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



CAMWorks NC Manager
- Configurations
- Machine [Mill - Metric]
- Stock Manager[304L]
- Coordinate System [Not Defined]
- Mill Part Setup1
  - Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
  - Irregular Pocket1 [Rough-Rough(Rest)- Finish]
  - Irregular Pocket2 [Rough-Rough(Rest)- Finish]
  - Irregular Pocket3 [Rough-Rough(Rest)- Finish]
  - Irregular Pocket4 [Rough-Rough(Rest)- Finish]
  - Hole Group1 [Drill]
- Recycle Bin

Features recognized by AFR

The *Feature* tree allows you to:

- Copy, rename, suppress, delete and combine machinable features
- Change machinable feature parameters

- Change the order in which the features are machined
- Insert 2.5 Axis and Multi Surface features
- Search for a feature based on item name
- Hide or show feature display in graphics area

- Generate an Operation Plan and find the first operation for a feature.

Did You Know ...

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

M

Most feature parameters are fixed; however, some parameters can be changed using the *Parameters* command on the feature's RMB context menu.

1. Right click *Hole Group1* in the tree and select *Parameters* on the context menu.

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

2. Click the down arrow next to *Drill* to see the choices.
3. Click *Thread* in the list.

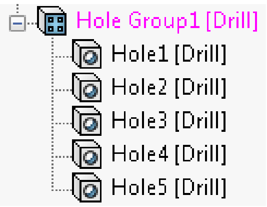
   The figure changes to reflect your choice and the Thread parameters are enabled.

4. Click the *Strategy* dropdown list and select *Drill* again.
5. Click *OK* to close the dialog box.
6. Click the ⊞ next to *Hole Group1* in the CAMWorks Feature tree.

   The tree expands to display each individual hole.

7. Click the ⊟ next to *Hole Group1* in the Feature tree.

   The tree collapses hiding the individual hole features.



Hole Group

8. Click the *Options* button the CAMWorks Command Manager.

   Note that you can also select the *Options* command from the RMB context menu of the CAMWorks NC Manager in the tree.

   - The *Options* dialog box is displayed.
   - On the *General* tab, make sure the *Save/Restore part* option is checked.
   - Click *OK* to apply the settings and close the dialog box.

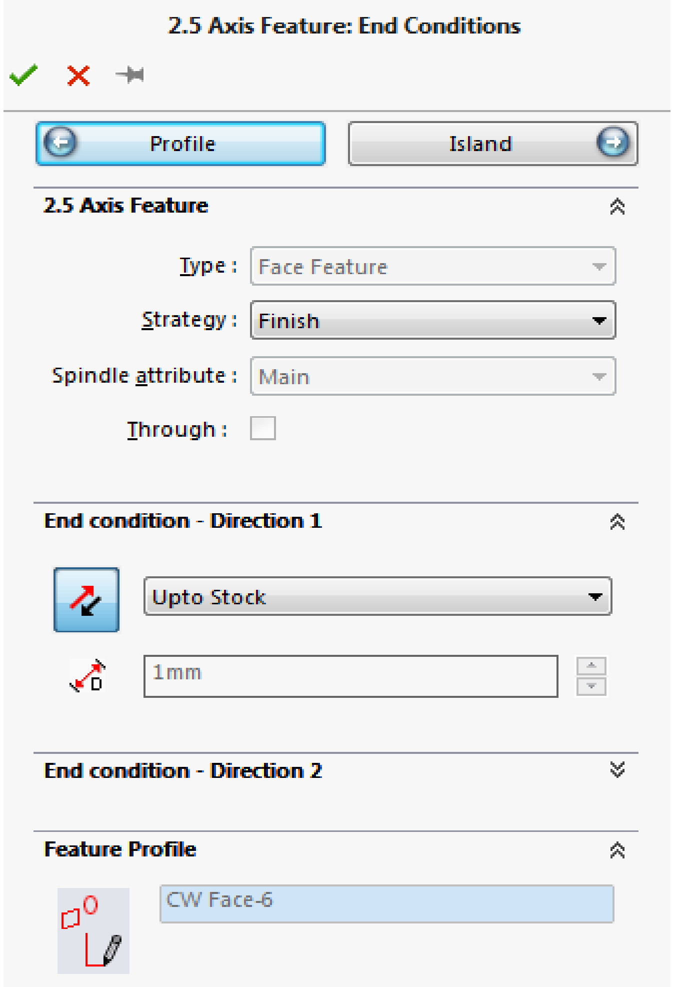
If this *Save/Restore part* option is checked, when you save and close a part document, the machining data is saved and restored with the part design information when the part document is reopened.

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

9. Select *Save As* on the File menu. Browse to the location where you wish to save the file.
10. In the *Save As* dialog box, assign the desired file name and click the *Save* button.

SAVE FREQUENTLY!

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



Type : Face Feature

Strategy : Finish

Spindle attribute : Main

Through : ☐

End condition - Direction 1: Upto Stock

1mm

Feature Profile: CW Face-6

2.5 Axis Feature: End conditions dialog box

Defining Features Interactively

Automatic Feature Recognition can save a significant amount of time; however, AFR does have limitations. AFR cannot recognize every feature on complex parts and does not recognize some types of features. To machine these areas, you can define machinable features interactively using the *New 2.5 Axis Feature* command.

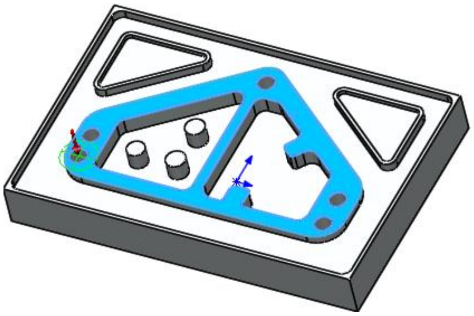
For some parts, Face Features can be recognized by AFR by selecting the Face option under the *Feature Types* group box on the *Mill Features* tab in the *Options* dialog box.

For learning purposes, in this tutorial, you will insert a Face Feature so that you can face the top of the part. In order to define a Face Feature, you select a face on the

SOLIDWORKS part that is at the depth you want to face the part to.

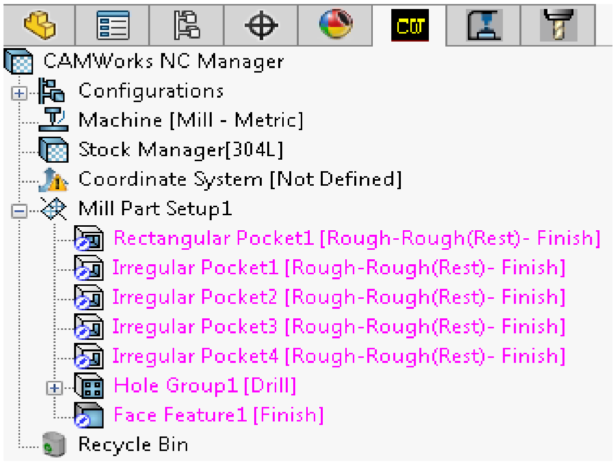
1. Right click *Mill Part Setup1* in the CAMWorks Feature tree and select *2.5 Axis Feature* on the context menu.

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



2. Click the down arrow next to the *Type* dropdown list box and select *Face Feature*.
3. Pick the top main face as shown on the right. The outline of the face will be highlighted on the part and *CW Face-6* will be displayed in the *Selected Entities* list.
4. Click the *End Condition* button at the top right of the dialog box.

   The *2.5 Axis Feature: End Conditions* dialog box will be displayed. This dialog box allows you to determine how CAMWorks calculates the depth of the feature and select a Strategy that defines a unique machining sequence.
5. In the *Strategy* dropdown list, select *Finish*.
6. In the *End Condition – Direction 1* group box, leave the *End condition Type* set to *Upto Stock*.
7. ✓ Click *OK* button to insert the feature and close the dialog box.
8. *Face Feature1* will be displayed in the CAMWorks Feature tree. You have now defined all the machinable features in this part and you are ready to generate the Operation Plan.

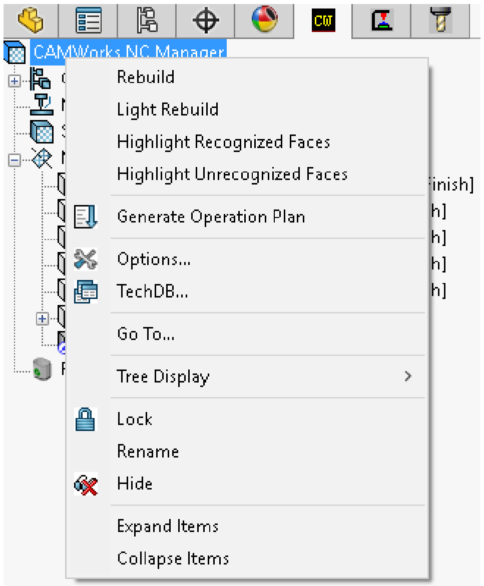

CAMWorks NC Manager
Configurations
Machine [Mill - Metric]
Stock Manager[304L]
Coordinate System [Not Defined]
Mill Part Setup1
Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Irregular Pocket4 [Rough-Rough(Rest)- Finish]
Hole Group1 [Drill]
Face Feature1 [Finish]
Recycle Bin

Interactively inserted Face Feature

Step 6: Generate Operation Plan and Toolpath

Generate Operation Plan Command

An Operation Plan contains information on how each machinable feature is to be machined and how the NC code will be output. When Generate Operation Plan is run, operations for each machinable feature are created automatically based on information in the Technology Database. In some situations, the operations defined for a feature in the Technology Database may not be sufficient and additional operations may be required. You can insert operations interactively using the *New 2 Axis Operation*, *New Hole Operation* and *New 3 Axis Operation* commands. These commands are explained in the CAMWorks context-based Help.



Generate Operation Plan command executed at CAMWorks NC Manager Level

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

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

OR

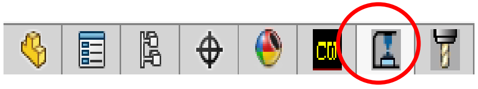
Right click *Mill Part Setup1* in the CAMWorks Feature tree and select *Generate Operation Plan* on the context menu.

OR

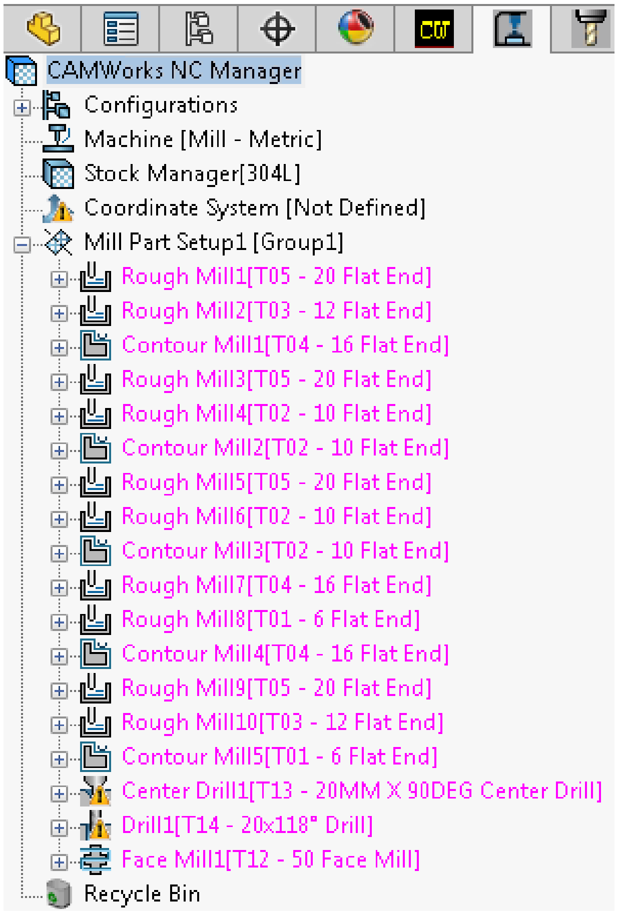
Right click *CAMWorks NC Manager* in the CAMWorks Feature tree and select *Generate Operation Plan* on the context menu.

Note:

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



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



CAMWorks NC Manager
Configurations
Machine [Mill - Metric]
Stock Manager[304L]
Coordinate System [Not Defined]
Mill Part Setup1 [Group1]
Rough Mill1[T05 - 20 Flat End]
Rough Mill2[T03 - 12 Flat End]
Contour Mill1[T04 - 16 Flat End]
Rough Mill3[T05 - 20 Flat End]
Rough Mill4[T02 - 10 Flat End]
Contour Mill2[T02 - 10 Flat End]
Rough Mill5[T05 - 20 Flat End]
Rough Mill6[T02 - 10 Flat End]
Contour Mill3[T02 - 10 Flat End]
Rough Mill7[T04 - 16 Flat End]
Rough Mill8[T01 - 6 Flat End]
Contour Mill4[T04 - 16 Flat End]
Rough Mill9[T05 - 20 Flat End]
Rough Mill10[T03 - 12 Flat End]
Contour Mill5[T01 - 6 Flat End]
Center Drill1[T13 - 20MM X 90DEG Center Drill]
Drill1[T14 - 20x118° Drill]
Face Mill1[T12 - 50 Face Mill]
Recycle Bin

Operations generated on executing GOP command

CAMWorks generates the operation plan for all the machinable features in *Mill Part Setup1*. The operations are listed in the CAMWorks Operation tree, which displays automatically. The Operation tree can also be accessed by clicking the CAMWorks Operation Tree tab.

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

The CAMWorks Operation tree allows you to:

- Insert, rename, suppress, and delete operations
- Change operation parameters
- Combine operations
- Sort operations
- Change the machining order

- Generate toolpaths
- Simulate toolpaths
- Post process the toolpaths
- Hide or show toolpath display
- Search based on item name

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

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

Exclamation mark symbol over certain operations

In the Operation tree, observe that *Center Drill1* and *Drill1* operations have an exclamation mark over them. The exclamation mark serves as a potential warning. In most cases, the exclamation mark indicates that no suitable tool was available in the active tool crib to machine that particular feature and hence a suitable tool was added from the tool library and assigned to the operation.

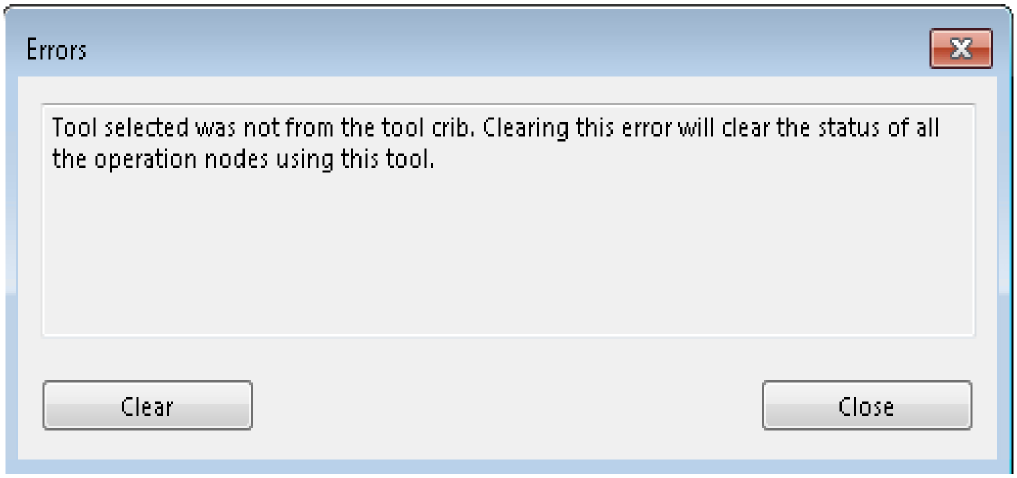
Following are the steps to get rid of the exclamation mark:

a. Right-click on the *Center Drill1* operation with an exclamation mark and select *What's wrong?* from the context menu.

b. A warning message indicating that tool selected was not from the active tool crib will be displayed.

c. Click on the *Clear* button in this dialog box to get rid of the exclamation mark.

d. Repeat above steps for all operations with the red exclamation mark. In this tutorial, repeat this step for *Drill1* operation.

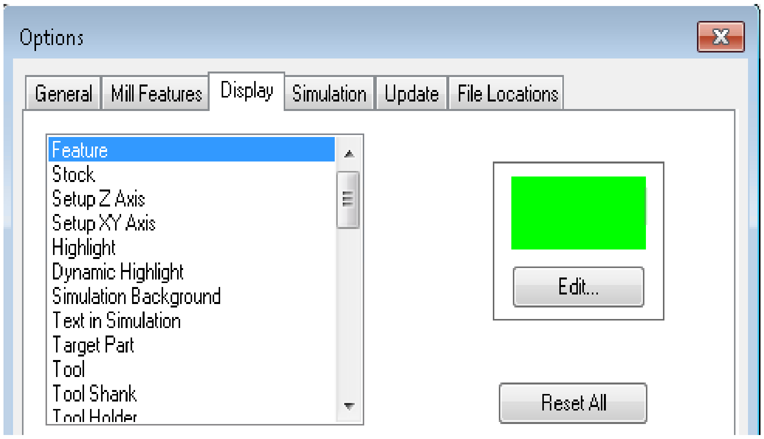


Tool selected was not from the tool crib. Clearing this error will clear the status of all the operation nodes using this tool.

Message indicating What's wrong with the selected Operation

Did You Know ...

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



Display tab of Options dialog box

Generate Toolpath Command

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

OR

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

CAMWorks calculates the toolpaths for each operation in the Mill Part Setup.

Note:

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

- Executing the *Generate Toolpath* command from the CAMWorks NC Manager level generates toolpaths for all the operations.
- Executing the *Generate Toolpath* command at Mill Part Setup level generates toolpaths only for operations listed under the given mill setup.
- You can also generate toolpaths for each operation individually by right-clicking on an operation and executing *Generate Toolpath* command on the context menu.

1. Highlight the first operation in the Operation tree, hold down the *Shift* key, then highlight the last operation.

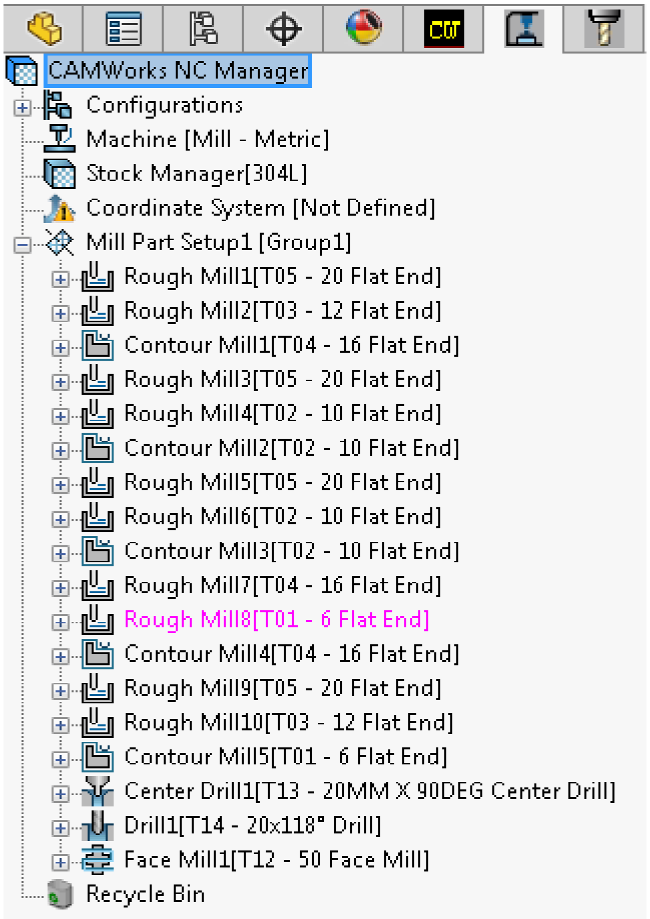
   This action displays the toolpaths for all the operations on the part (in the graphics area) showing the centerline of the toolpath.

2. Click an operation in the CAMWorks Operation tree.

   – The toolpath for that operation is displayed.

   – As you click each operation, the toolpath for that corresponding operation is displayed.

   Generated toolpath list is displayed in the Operation tree.



- CAMWorks NC Manager
  - Configurations
  - Machine [Mill - Metric]
  - Stock Manager[304L]
  - Coordinate System [Not Defined]
  - Mill Part Setup1 [Group1]
    - Rough Mill1[T05 - 20 Flat End]
    - Rough Mill2[T03 - 12 Flat End]
    - Contour Mill1[T04 - 16 Flat End]
    - Rough Mill3[T05 - 20 Flat End]
    - Rough Mill4[T02 - 10 Flat End]
    - Contour Mill2[T02 - 10 Flat End]
    - Rough Mill5[T05 - 20 Flat End]
    - Rough Mill6[T02 - 10 Flat End]
    - Contour Mill3[T02 - 10 Flat End]
    - Rough Mill7[T04 - 16 Flat End]
    - Rough Mill8[T01 - 6 Flat End]
    - Contour Mill4[T04 - 16 Flat End]
    - Rough Mill9[T05 - 20 Flat End]
    - Rough Mill10[T03 - 12 Flat End]
    - Contour Mill5[T01 - 6 Flat End]
    - Center Drill1[T13 - 20MM X 90DEG Center Drill]
    - Drill1[T14 - 20x118° Drill]
    - Face Mill1[T12 - 50 Face Mill]
  - Recycle Bin

Operation tree after generation of Toolpaths

Step 7: Adjusting Toolpath Parameters

CAMWorks calculates toolpaths using the operation parameters and the feature's size and shape. After toolpath is generated for a particular operation, then that operation is displayed in black-colored font in the Operation tree. These parameters can be edited after generating the toolpaths and before post processing the part.

Edit toolpath parameters:

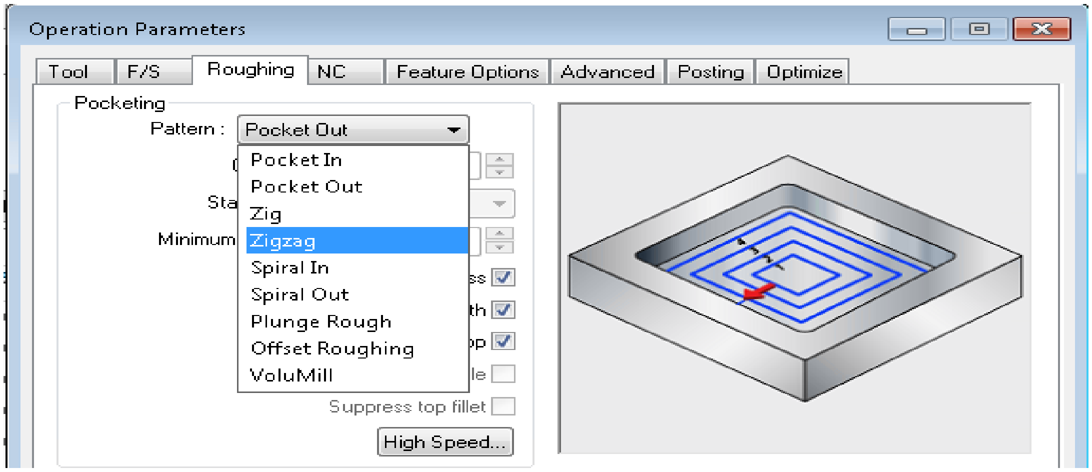
1. In the Operation tree, click on the ‘+’ sign next to *Rough Mill5* operation. The feature for which the operation was generated is listed below it. Observe that this operation was generated for *Irregular Pocket2* feature.
2. Double click *Rough Mill5* operation.

   OR

   Right click on *Rough Mill5* and select *Edit Definition* on the context menu.

   The *Operation Parameters* dialog box will be displayed whenever you double-click or execute *Edit Definition* command for an operation in the Operation tree. This dialog box gives you access to all the parameters used to define the toolpath.

   General parameters for the type of toolpath include the method of machining, depth of cut, step over, stock allowance, retract height, speeds and feeds. This dialog box also gives you access to the parameters for the tool you are using and allows you to select a different tool.
3. Click the *Roughing* tab and change the Pocketing Pattern to *Zigzag*.
4. Click the *Feature Options* tab. In the *Entry* group box, change the *Method* to *Entry Hole*.
5. Click *OK* to close the dialog box.



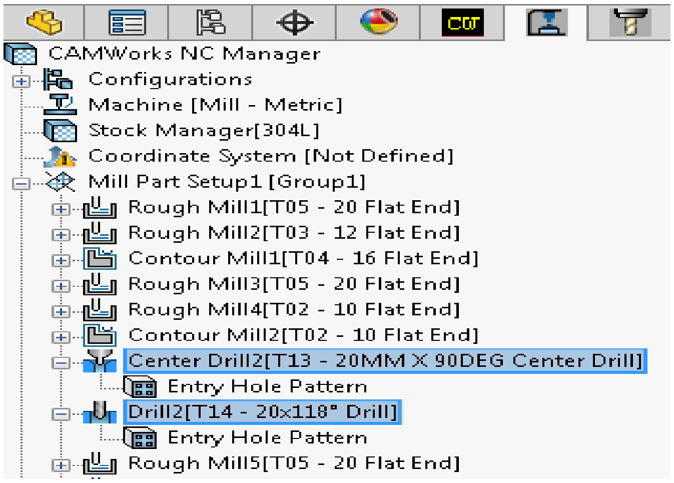
Pattern : Pocket Out

Roughing tab of Operation Parameters dialog box

6. Observe the CAMWorks Operation tree. Notice that the *Center Drill2* and *Drill2* operations were created automatically above the *Rough Mill5* operation since you set the *Entry Method* to *Entry Hole*.

   When using the Entry Hole method, control is provided to:

   - Specify any number of machining processes in the Technology Database to machine the entry hole (i.e. Center Drill and Drill).
   - Tool type selection is based on what each operation supports.
   - Tool diameter selection can be customized in the Technology Database.
   - Operation parameter defaults can be defined in the Technology Database.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Metric]
- Stock Manager[304L]
- Coordinate System [Not Defined]
- Mill Part Setup1 [Group1]
  - Rough Mill1[T05 - 20 Flat End]
  - Rough Mill2[T03 - 12 Flat End]
  - Contour Mill1[T04 - 16 Flat End]
  - Rough Mill3[T05 - 20 Flat End]
  - Rough Mill4[T02 - 10 Flat End]
  - Contour Mill2[T02 - 10 Flat End]
  - Center Drill2[T13 - 20MM X 90DEG Center Drill]
    - Entry Hole Pattern
  - Drill2[T14 - 20x118° Drill]
    - Entry Hole Pattern
  - Rough Mill5[T05 - 20 Flat End]

Generated Center drill and drill after inserting Entry Hole Method

Simulate Toolpath

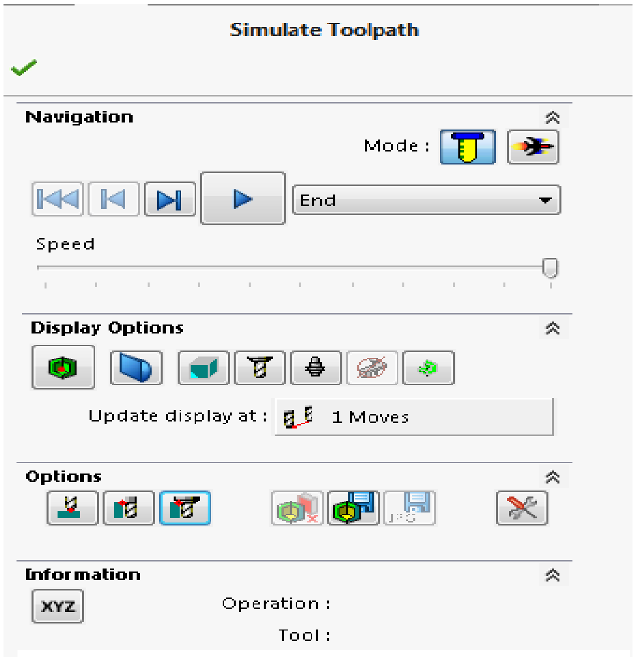
CAMWorks provides the ability to simulate the toolpath showing the tool movement and the resulting shape of the part.

1. Right click on *Mill Part Setup1* and select *Simulate Toolpath* from the context menu.

   The Toolpath Simulation toolbar is displayed.

   The toolbar controls allow you to:

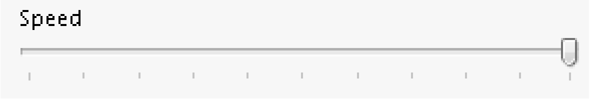
   - Run the simulation in *Tool* or *Turbo* mode.
   - Display the simulated part, the design part and a comparison of the two by clicking on the *Show Difference* button.
   - Customize the display of the stock, tool and tool holder (*wireframe, translucent, shaded, or no display*) in *Tool* mode.
     Note that these options are disabled in Turbo mode.
   - Run the simulation for all or selected operations.



Update display at : 1 Moves

Simulate Toolpath Dialog Box

- When simulating an operation, the simulation can be for the current operation or for all previous operations up to the selected operation.
- Pause the simulation using *Pause* button in both *Tool* and *Turbo* mode.
- Dynamically change the orientation of the part using zoom, pan, rotate, etc.
- Control the simulation speed by dragging the Simulation Speed Control slider

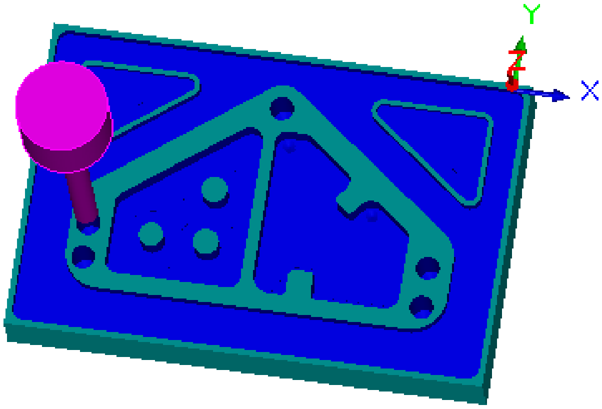


.

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

2. Set the simulation mode to *Tool mode* by clicking on the *Tool mode* button.
3. Set the tool display as *Shaded With Edges*.
4. Set the Tool Collision, Tool Shank Set the tool holder display as *Tool Cut Collision, Tool Shank Cut Collision and Tool Holder Cut Collision* .
5. Click the *Run* button.

   The simulation runs in *Tool mode* with the tool and holder displayed during simulation.
6. Click the *Pause* button to pause during simulation. Click the *Run* button to continue simulation.



Simulation Mode

Step Through Toolpath

CAMWorks also provides the ability to visualize the tool motion and verify tool positions using the Step Thru Toolpath command.

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

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

   OR

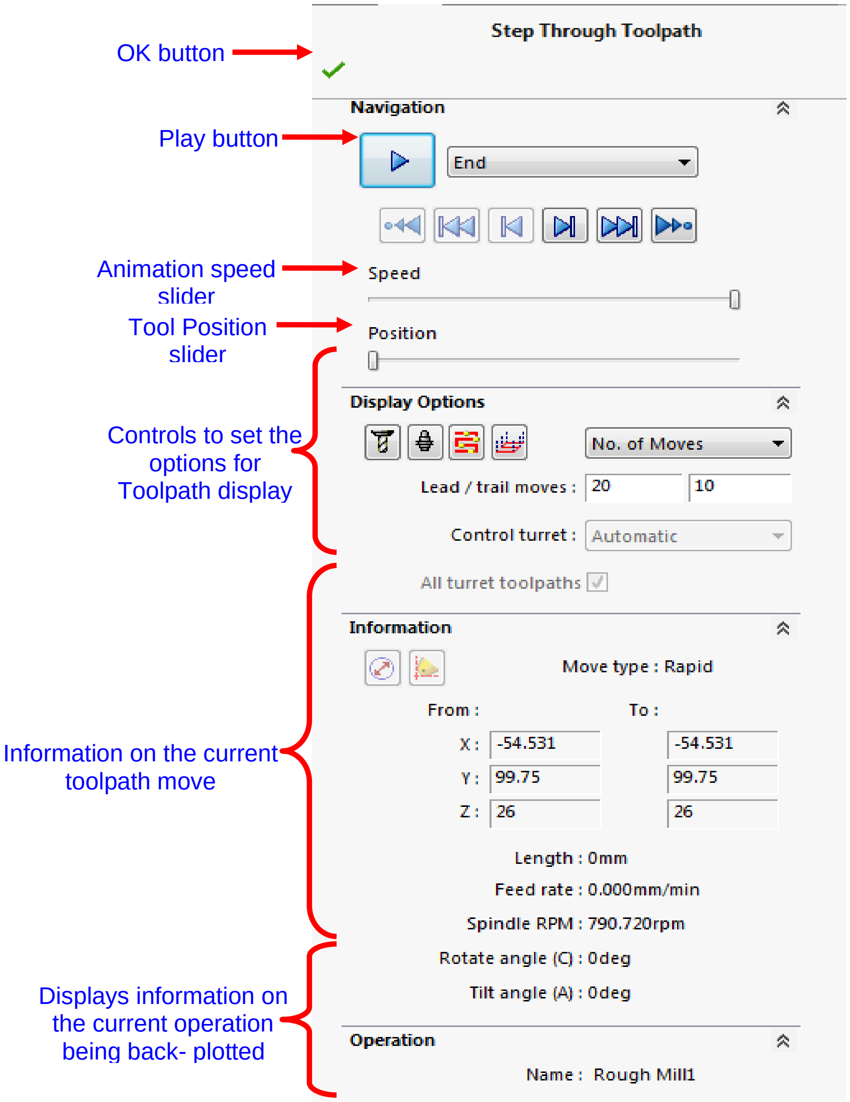
Select *Step Thru Toolpath* command on the Command Manager.

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

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

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

2. Use controls given in the *Display Options* group box to set the options as to how the toolpath will be displayed during the Step Through process.
3. In the *Play Tool Until* dropdown list, leave the end condition set to the default *End*.
4. Use the *Animation speed* slider to adjust the animation speed.
5. Click the *Play* button ▶ to continuously back-plot the tool till the end condition specified in the *Play Tool Until* field is reached.
6. During animation, the *Play* button display changes to *Pause* button ⏸. Click the *Pause* button to pause the animation while it is in progress.
7. When the animation has not yet begun or when it is paused, the *Forward Single Step*, *Goto End of Current Toolpath* and *Goto End* buttons are active.
   – Use the *Forward Single Step* button ⏭ to move one toolpath record



End

No. of Moves

Lead / trail moves : 20 10

Control turret : Automatic

All turret toolpaths ☑

Move type : Rapid

	From :	To :
X :	-54.531	-54.531
Y :	99.75	99.75
Z :	26	26

Length : 0mm

Feed rate : 0.000mm/min

Spindle RPM : 790.720rpm

Rotate angle (C) : 0deg

Tilt angle (A) : 0deg

Name : Rough Mill1

OK button

Play button

Animation speed slider

Tool Position slider

Controls to set the options for Toolpath display

Information on the current toolpath move

Displays information on the current operation being back- plotted

Step Through Toolpath dialog

forward each time the button is clicked.

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

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

8. When the animation has been completed or when it is paused, the *Reverse Single Step, Goto Start of current Toolpath* and *Goto Start* buttons are active.
   - Use the *Reverse Single Step* button ⏮ to move one toolpath record backwards each time the button is clicked.
   - Use the *Goto Start of Current Toolpath* button ⏮ to move the tool position to the first toolpath record of the current operation.
   - Use the *Goto Start* button ⏪ to move the tool position to the first toolpath record of the very first operation.
9. If you are unsure about the use of any parameter within this dialog box, click on the *Help* button ? at the upper right corner. This action will display the context-based help for Step Through Toolpath command.
10. ✓ Click the *OK* button in the upper left corner to close the dialog box.

Step 8: Post Process Toolpaths

Post processing is the final step in generating the NC program file. When you use a CAMWorks internal post processor, this step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the CAMWorks Operation tree. When you post process a part, CAMWorks creates two files: the NC program and the Setup Sheet. These are text files that you can read, edit and print using a word processor or text editor.

Did You Know ...

For more information on generating an enhanced APT CL file that can be used by external post processing programs, see the context-based Help.

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

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

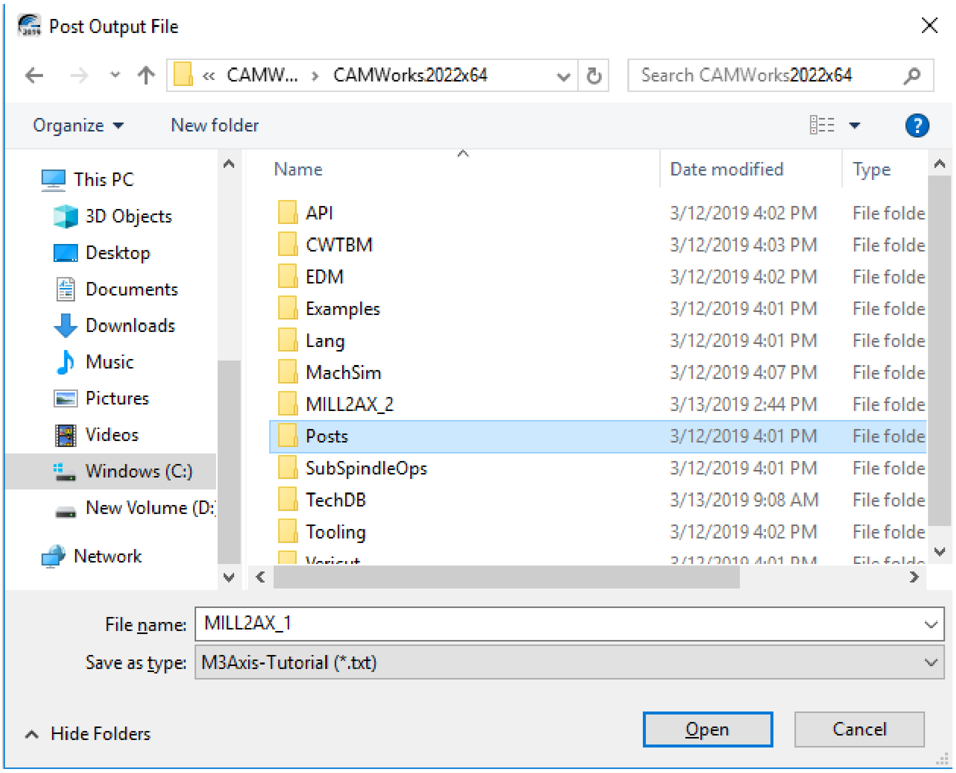
   Right click on the *CAMWorks NC Manager* in the Operation tree and select *Post Process* on the context menu.

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

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

   Note: If the Post Process command is grayed out on the CAMWorks Command Manager or on any context menu, make sure that you have selected a post processor and generated the toolpaths. Refer instruction 4 of Step 3 in this tutorial.

2. In the *Post Output File* dialog box, click the down arrow to the right of the *Save as type* box.
   CAMWorks provides a list of commonly used extensions that you can select. For this exercise, use the *.txt* extension.


Post Output File
« CAMW... › CAMWorks2022x64
Search CAMWorks2022x64
Organize
New folder
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
Windows (C:)
New Volume (D:
Network
Name
Date modified
Type
API 3/12/2019 4:02 PM File folde
CWTBM 3/12/2019 4:03 PM File folde
EDM 3/12/2019 4:02 PM File folde
Examples 3/12/2019 4:01 PM File folde
Lang 3/12/2019 4:01 PM File folde
MachSim 3/12/2019 4:07 PM File folde
MILL2AX_2 3/13/2019 2:44 PM File folde
Posts 3/12/2019 4:01 PM File folde
SubSpindleOps 3/12/2019 4:01 PM File folde
TechDB 3/13/2019 9:08 AM File folde
Tooling 3/12/2019 4:02 PM File folde
Hide Folders
Open
Cancel

File name: MILL2AX_1

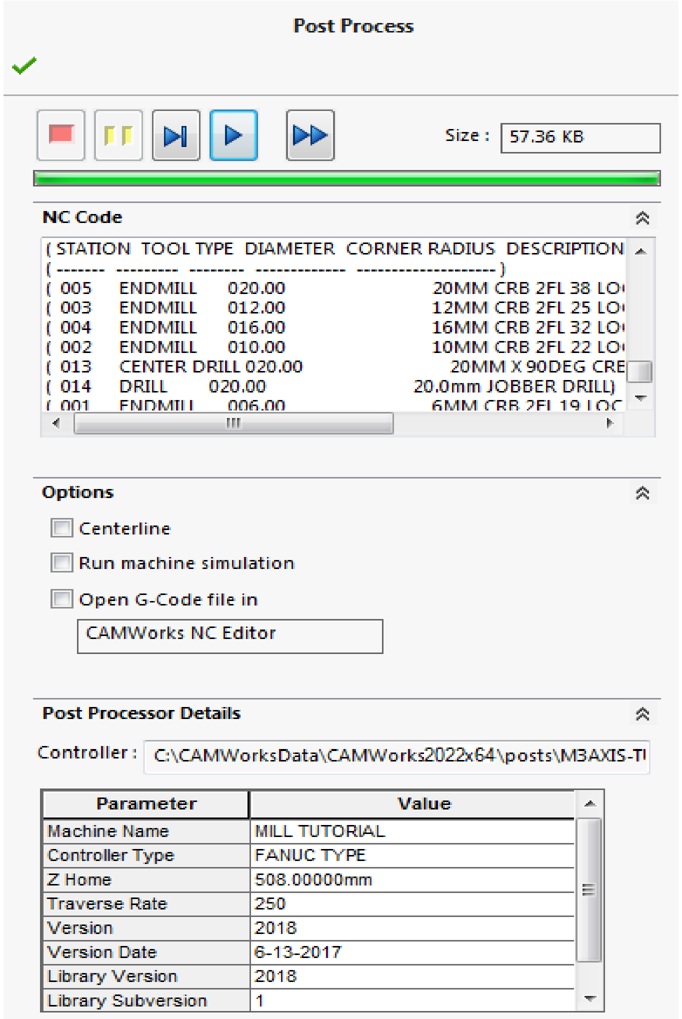
Save as type: M3Axis-Tutorial (*.txt)

Post Output File dialog box

Did You Know ...

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

3. In the *File name* textbox, type the suitable file name, and then click *Save* button.
4. The *Post Process Output* dialog box will be displayed. Click the *Step* button on the control bar at the top. CAMWorks starts to generate the NC program and the first line of NC code displays in the NC code output view box. The post processing mode is set to post process one line of code at a time (Step mode).
5. Click the *Step* button. The next line of NC code is displayed.
6. Click the *Run* button. Post processing continues until it is completed.
7. When the post processing is finished, view the code using the vertical scroll bar.
8. Click *OK* ✓ to close the dialog box.



```
( STATION  TOOL TYPE  DIAMETER  CORNER RADIUS  DESCRIPTION
( -------  ---------  --------  -------------  -------------------- )
( 005     ENDMILL    020.00                   20MM CRB 2FL 38 LO
( 003     ENDMILL    012.00                   12MM CRB 2FL 25 LO
( 004     ENDMILL    016.00                   16MM CRB 2FL 32 LO
( 002     ENDMILL    010.00                   10MM CRB 2FL 22 LO
( 013     CENTER DRILL 020.00                  20MM X 90DEG CRE
( 014     DRILL     020.00                  20.0mm JOBBER DRILL)
( 001     ENDMILL    006.00                   6MM CRB 2FL 19 LOC
```

☐ Centerline

☐ Run machine simulation

☐ Open G-Code file in

CAMWorks NC Editor

Controller : C:\CAMWorksData\CAMWorks2022x64\posts\M3AXIS-TI

Parameter	Value
Machine Name	MILL TUTORIAL
Controller Type	FANUC TYPE
Z Home	508.00000mm
Traverse Rate	250
Version	2018
Version Date	6-13-2017
Library Version	2018
Library Subversion	1

Post Process Dialog Box

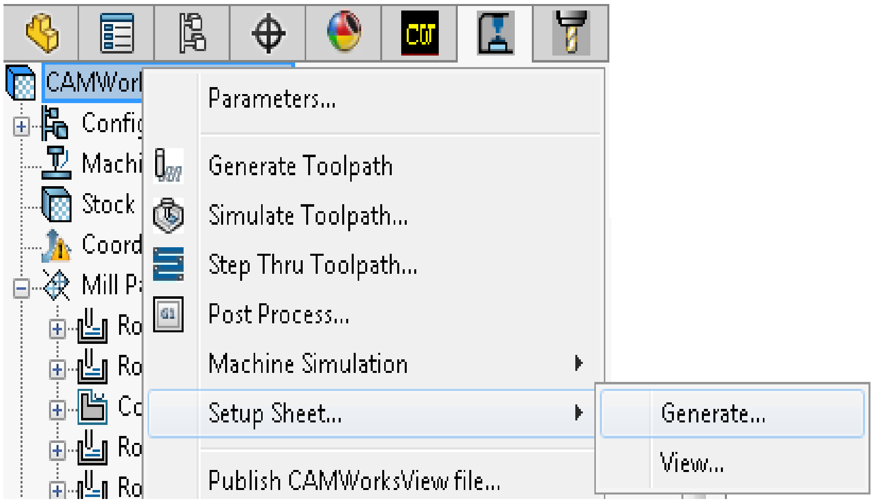
More About Setup Sheets

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

CAMWorks provides two methods for creating Setup Sheets:

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

- The Setup Sheet command on the CAMWorks NC Manager context menu and the Generate Setup Sheets command on the Setup context menu provide a choice of two formats: Access and XML. When you select Access, the Setup Sheet is based on an Access database report template and the information is stored in the Report Database to view at any time. CAMWorks supplies several report templates that can be used as is. You can also open the Report Database in Access and create customized reports based on these templates or design your own original reports. When you select XML, the information is saved in an XML file, which allows the Setup Sheet to be formatted with an HTML style sheet and displayed in a web browser.



Generating Setup Sheets

2 Axis Mill 2

Topics covered in this tutorial:

CAMWorks is a feature-based machining system. Feature-based machining provides numerous benefits because the definition of the feature enables a higher level of automation when creating machining operations and associated toolpaths. Adding to this benefit, CAMWorks provides two methods for extracting features:

- **AUTOMATIC FEATURE RECOGNITION**

  The automatic method of extracting features is called Automatic Feature Recognition (AFR). There are two methods for extracting machinable features automatically: *AFR* and *MfgView*. You can set the method for automatic extraction of machinable features to either *AFR* or *MfgView* on the *Mill Features* tab of *CAMWorks Options* dialog box. The default method is set to *MfgView*.

  The MfgView method has the following benefits:

  - CAMWorks uses an alternative method to generate features and finds additional feature types not found by AFR. Non-hole features can be edited the same as interactively inserted features.
  - A Recognize Features command is enabled on the Mill Part Setup context menu that provides the ability to recognize features from a user-defined direction only.

  We recommend that you use the *MfgView* method. The automatically recognized features in all the tutorials given in this manual are recognized using *MfgView* method.

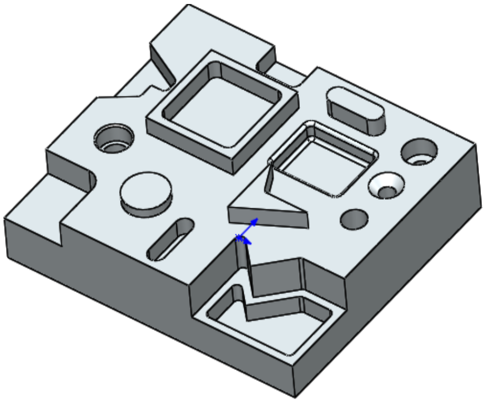
- **INTERACTIVE FEATURE RECOGNITION**

  An interactive method for inserting features using the *New 2.5 Axis Feature* or *New Multi Surface Feature* function.

  A CAMWorks file can contain both types of features – automatically recognized as well as interactively inserted features.

  The following tutorial demonstrates how some of the features are identified automatically using the *Extract Machinable Features* command.

Extracting and Working with Machinable Features



MILL2AX_2.SLDPRT

1. Open the part file **MILL2AX_2.SLDPRT** in the *Drive:\CAMWorksData\ CAMWorks202x\Examples\ Tutorial_Parts\Mill* folder.
2. Examine the part. You can recognize various sizes and shapes of holes, bosses, pockets and slots. It appears that CAMWorks extracts features based on SOLIDWORKS features used to create the part. As you will see, this is not necessarily so. Notice that the

design tree contains no design features. This part was imported into SOLIDWORKS/CAMWorks Solids.

In this tutorial, you will see how CAMWorks determines machinable features.

3. Click the *CAMWorks Feature Tree* tab.
4. Double click *Machine [Mill–Inch]* in the CAMWorks Feature tree.

Did You Know ...

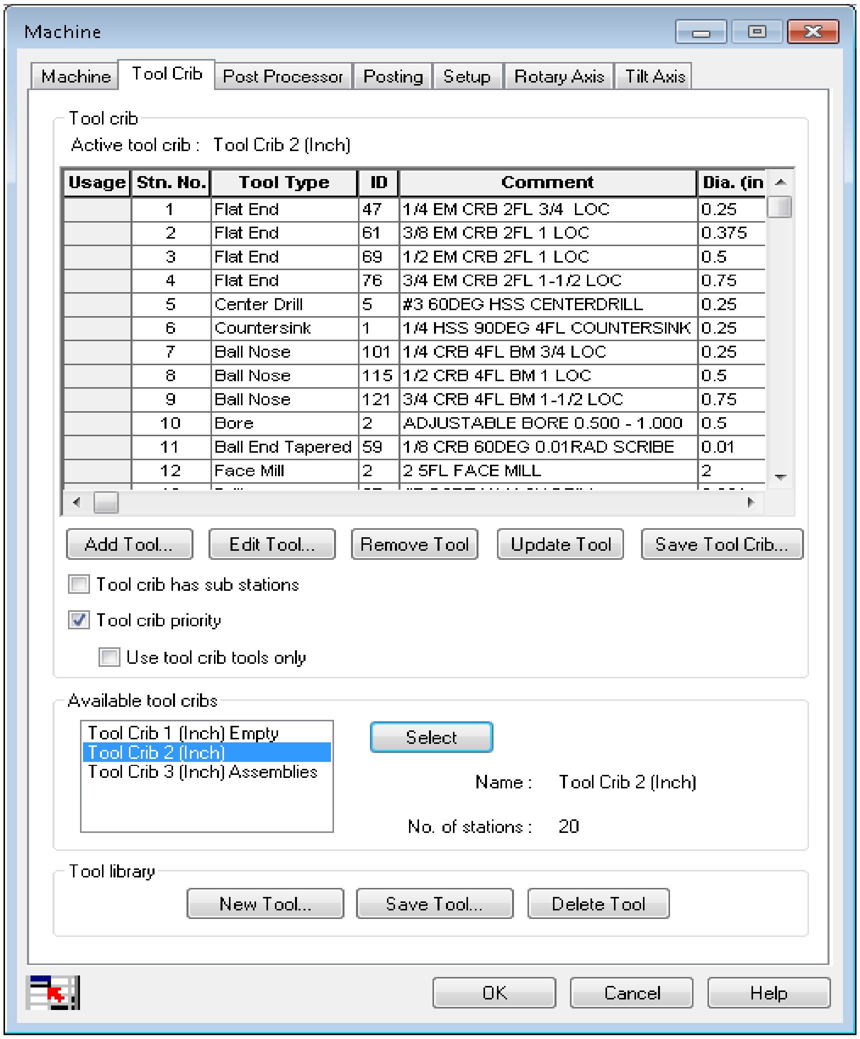
In the CAMWorks Feature and Operation trees, instead of right-clicking on the listed items and selecting Edit Definition from the context menu, you can double-click the item to open the dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.

The *Machine* dialog box will be displayed. This dialog box allows you to select a machine definition, tool crib and post processor. On the *Machine* tab of this dialog box, *Mill–Inch* is the active machine. This machine definition has been created for the CAMWorks tutorials.

Tool Crib Tab

When you use CAMWorks to machine your own parts, you need to select the machine tools that you want to use to machine the part.

Click on the *Tool Crib* tab. This tab allows you to choose a tool crib or set of tools that are used with the machine you have chosen. The Tool crib does not contain all the tools that are available; rather it contains a subset of tools that you can modify to represent the actual set of tools that is loaded on the machine.



Active tool crib : Tool Crib 2 (Inch)

Usage	Stn. No.	Tool Type	ID	Comment	Dia. (in
	1	Flat End	47	1/4 EM CRB 2FL 3/4 LOC	0.25
	2	Flat End	61	3/8 EM CRB 2FL 1 LOC	0.375
	3	Flat End	69	1/2 EM CRB 2FL 1 LOC	0.5
	4	Flat End	76	3/4 EM CRB 2FL 1-1/2 LOC	0.75
	5	Center Drill	5	#3 60DEG HSS CENTERDRILL	0.25
	6	Countersink	1	1/4 HSS 90DEG 4FL COUNTERSINK	0.25
	7	Ball Nose	101	1/4 CRB 4FL BM 3/4 LOC	0.25
	8	Ball Nose	115	1/2 CRB 4FL BM 1 LOC	0.5
	9	Ball Nose	121	3/4 CRB 4FL BM 1-1/2 LOC	0.75
	10	Bore	2	ADJUSTABLE BORE 0.500 - 1.000	0.5
	11	Ball End Tapered	59	1/8 CRB 60DEG 0.01RAD SCRIBE	0.01
	12	Face Mill	2	2 5FL FACE MILL	2

☐ Tool crib has sub stations

☑ Tool crib priority

☐ Use tool crib tools only

Name : Tool Crib 2 (Inch)

No. of stations : 20

Tool Crib tab of Machine dialog box

Tool Crib Priority

If you select the *Tool crib priority* option, tool selection rules are modified to place a higher priority on selecting tools from the tool crib. This results in fewer tools being added to the tool list from the tool library. Refer to the CAMWorks Context-based Help to understand the rules that are observed when this option is used. For this tutorial, leave the *Tool Crib priority* option check box checked.

Tool Crib has Sub Stations

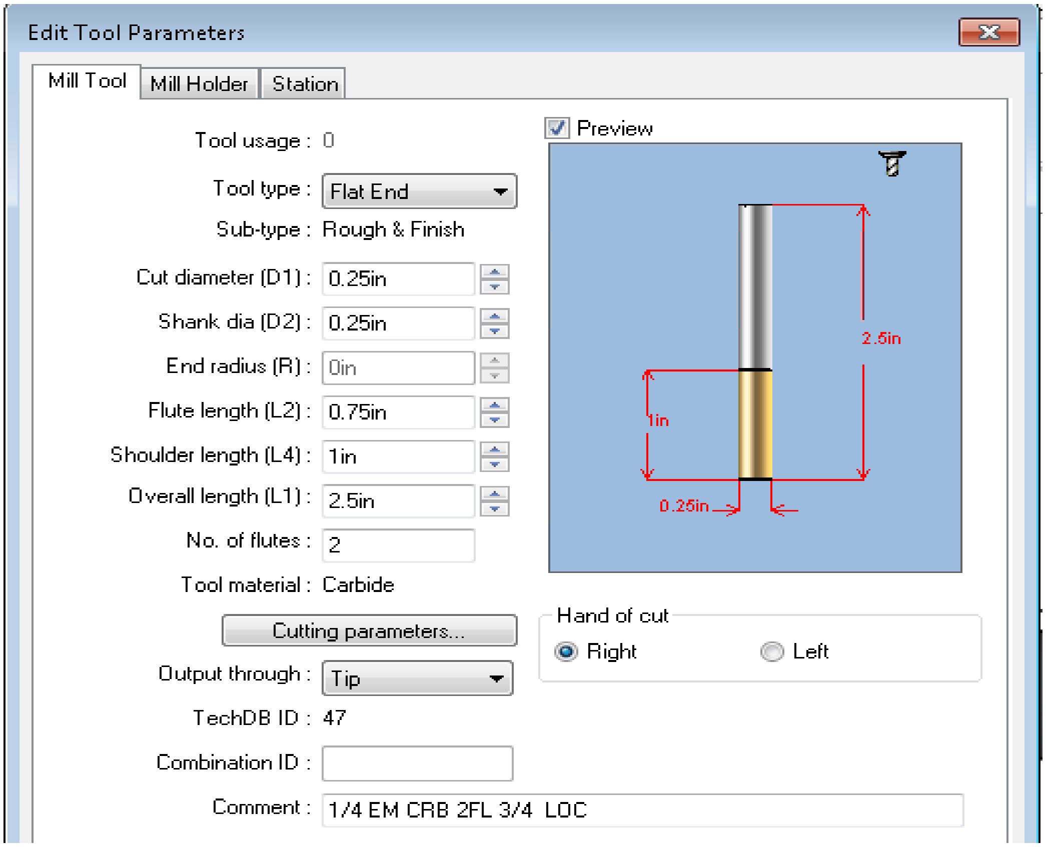
Select this option if your machine supports gang tooling. In this tutorial, this option will be left unchecked.

Editing the Tool Crib

You can add, remove and edit tools in the Active Tool Crib from the *Tool Crib* tab.

1. Make sure *Tool Crib 2 (Inch)* is the active tool crib.
2. Highlight a tool in the Active tool crib list and click *Edit Tool* button.

   The *Edit Tool Parameters* dialog box will be displayed.



Tool usage : 0
Tool type : Flat End
Sub-type : Rough & Finish
Cut diameter (D1) : 0.25in
Shank dia (D2) : 0.25in
End radius (R) : 0in
Flute length (L2) : 0.75in
Shoulder length (L4) : 1in
Overall length (L1) : 2.5in
No. of flutes : 2
Tool material : Carbide
Output through : Tip
TechDB ID : 47
Combination ID :
Comment : 1/4 EM CRB 2FL 3/4 LOC
☑ Preview
Hand of cut
● Right ○ Left

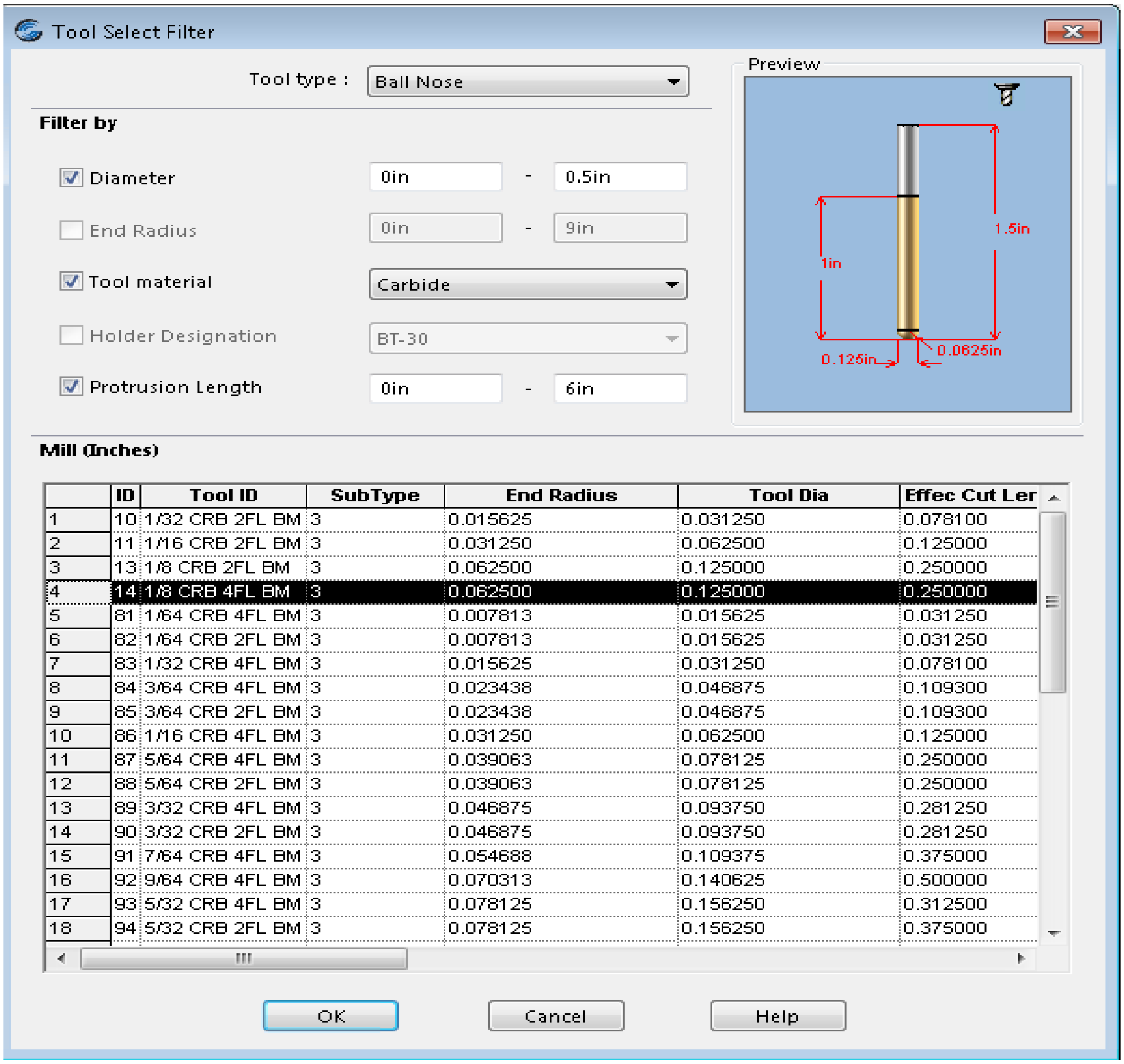
Edit Tool Parameter Dialog box

This dialog box allows you to change the parameters for the selected tool. The changes you make in this dialog box affect only the tool crib for the current part. To change the tool definition for all future jobs, click the *Save Tool Crib* button on the Tool Crib tab or edit the tool crib definition in the Technology Database.

3. Click *OK* to apply the changes and close the *Edit Tool Parameters* dialog box. Click *Cancel* to close this dialog box without saving the changes. In this tutorial, no changes have been done to the parameters of an tool in the Tool crib. Hence, click *Cancel*.
4. To add tools from the tool library to the active Tool crib, click the *Add Tool* button below the *Tool crib* tab in the *Machine* dialog box.

   The *Tool Select Filter* dialog box will be displayed. This dialog box allows you to set filters for displaying and selecting tools. At the bottom of this dialog box is a form listing all the existing tools in the Technology Database which meet the filter criteria.
5. Select the required filters and enter the corresponding range values in the provided fields. Click *OK*.
6. To add a tool from this list into the active tool crib, highlight the required tool and click *OK*.

   The tool will be added at the bottom of the active tool crib. To make this tool available in the active tool crib for all future jobs, click on the *Save Tool Crib* button.



Tool type : Ball Nose

Filter by

☑ Diameter 0in - 0.5in

☐ End Radius 0in - 9in

☑ Tool material Carbide

☐ Holder Designation BT-30

☑ Protrusion Length 0in - 6in

Mill (Inches)

	ID	Tool ID	SubType	End Radius	Tool Dia	Effec Cut Ler
1	10	1/32 CRB 2FL BM	3	0.015625	0.031250	0.078100
2	11	1/16 CRB 2FL BM	3	0.031250	0.062500	0.125000
3	13	1/8 CRB 2FL BM	3	0.062500	0.125000	0.250000
4	14	1/8 CRB 4FL BM	3	0.062500	0.125000	0.250000
5	81	1/64 CRB 4FL BM	3	0.007813	0.015625	0.031250
6	82	1/64 CRB 2FL BM	3	0.007813	0.015625	0.031250
7	83	1/32 CRB 4FL BM	3	0.015625	0.031250	0.078100
8	84	3/64 CRB 4FL BM	3	0.023438	0.046875	0.109300
9	85	3/64 CRB 2FL BM	3	0.023438	0.046875	0.109300
10	86	1/16 CRB 4FL BM	3	0.031250	0.062500	0.125000
11	87	5/64 CRB 4FL BM	3	0.039063	0.078125	0.250000
12	88	5/64 CRB 2FL BM	3	0.039063	0.078125	0.250000
13	89	3/32 CRB 4FL BM	3	0.046875	0.093750	0.281250
14	90	3/32 CRB 2FL BM	3	0.046875	0.093750	0.281250
15	91	7/64 CRB 4FL BM	3	0.054688	0.109375	0.375000
16	92	9/64 CRB 4FL BM	3	0.070313	0.140625	0.500000
17	93	5/32 CRB 4FL BM	3	0.078125	0.156250	0.312500
18	94	5/32 CRB 2FL BM	3	0.078125	0.156250	0.375000

Tool Select filter dialog box

Note: **You cannot use the Tool Select Filter dialog box to add new tools to the Technology Database. This dialog box can only be used to add tools to the active tool crib.**

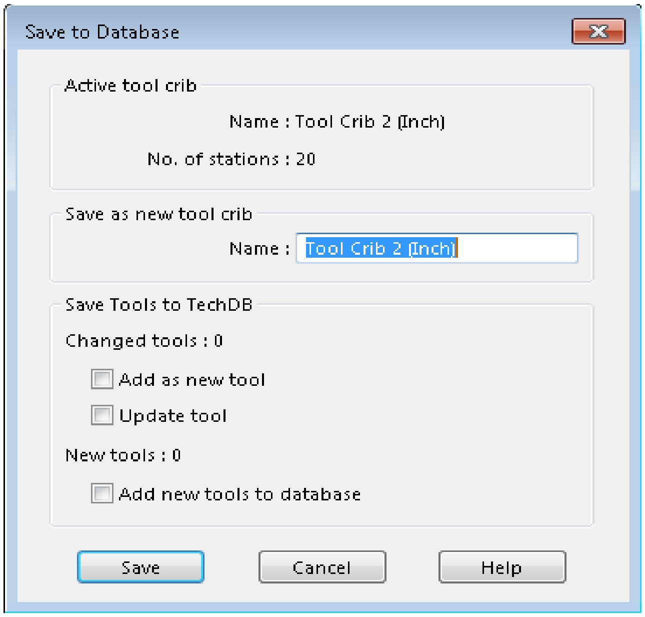
7. Click *Cancel* to close the *Tools Database* form without adding a tool to the active tool crib.

Modifying the Tools in the Technology Database

You can add, edit and delete tools present in the Technology Database using the Tool Library buttons provided in the *Tool Crib* tab of the *Machine* dialog box.

Adding a New Tool

To add a new tool to the Technology Database, click on the *New Tool* button in the *Tool Crib* tab. The *New Tool* dialog box is displayed. Assign tool parameters as desired. Click *OK* to close the *New Tool* dialog box. Observe that the new tool is added to the Active Tool crib list. The tool is also added to the Technology Database.



Active tool crib

Name : Tool Crib 2 (Inch)

No. of stations : 20

Save as new tool crib

Name : Tool Crib 2 (Inch)

Save Tools to TechDB

Changed tools : 0

☐ Add as new tool

☐ Update tool

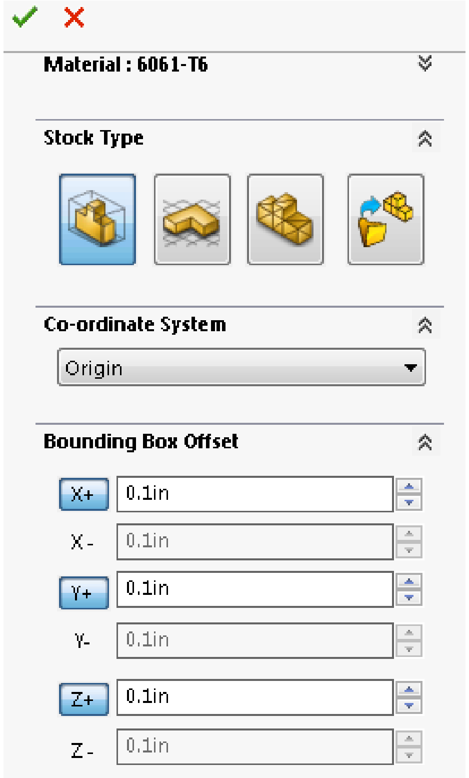
New tools : 0

☐ Add new tools to database

Save to Database dialog box displayed on clicking the 'Save Tool' button

Saving Changes Made to a Tool in the Active Tool Crib

If you edit a tool listed in the *Active tool crib* list, then the changes are applicable for the current part only. These changes can be made permanent (and thus available for future machining jobs) by saving them to the Technology Database. To do so, highlight the tool in the *Active tool crib* list and click on the *Save Tool* button. *Save to Database* dialog box. In the *Save Tool to TechDB* group box within this dialog box, you can choose to update the existing tool or save the tool as a new tool within the database. Select *Update tool* checkbox to save the changes to the existing tool. Select *Add new tools to database* to save the edited tool as a new tool.



Material : 6061-T6

Origin

X+	0.1in
X-	0.1in
Y+	0.1in
Y-	0.1in
Z+	0.1in
Z-	0.1in

Setting Bounding Box offsets in Stock Manager Dialog Box

Deleting a Tool

To permanently delete a tool from the Technology Database, highlight the tool in the *Active tool crib* list and click on the *Delete Tool* button. This action will remove the tool from the Technology Database. However, the tool will still be visible in the Active tool crib list for the current part. To remove it from the *Active tool crib* list, highlight the tool in the list and click on the *Remove tool* button.

Post Processor Tab and Posting Tab

1. Click the *Post Processor* tab in the Machine dialog box and make sure *M3AXIS-TUTORIAL* is selected.

   The *M3AXIS-TUTORIAL* post processor is used for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.

2. Click the *Posting* tab.

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

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

Defining the Stock

1. Double click *Stock Manager* in the Feature tree.
2. The *Stock Manager* dialog box is displayed. Leave the *Stock Type* set to default *Bounding box.*
3. Change the Bounding box offsets to a uniform **0.1in** for X, Y, and Z.
4. ✓ Click *OK* to apply the changes and close the dialog box.

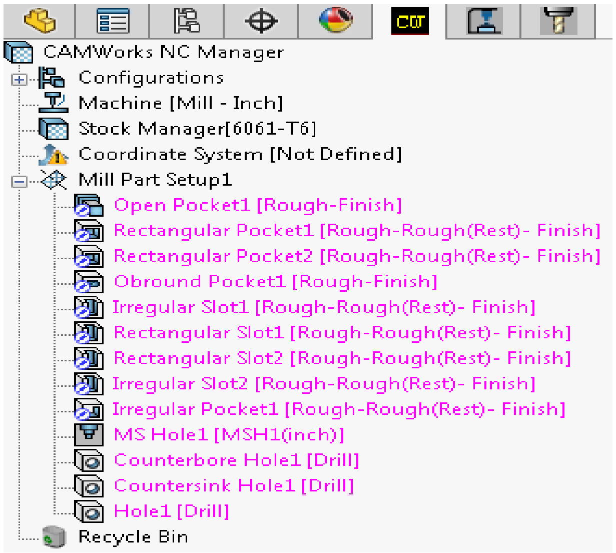
Extract Machinable Features

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

   OR

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

   The *CAMWorks Message Window* is displayed. This window reports the process status and shows the progress of the process.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Inch]
- Stock Manager[6061-T6]
- Coordinate System [Not Defined]
- Mill Part Setup1
  - Open Pocket1 [Rough-Finish]
  - Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
  - Rectangular Pocket2 [Rough-Rough(Rest)- Finish]
  - Obround Pocket1 [Rough-Finish]
  - Irregular Slot1 [Rough-Rough(Rest)- Finish]
  - Rectangular Slot1 [Rough-Rough(Rest)- Finish]
  - Rectangular Slot2 [Rough-Rough(Rest)- Finish]
  - Irregular Slot2 [Rough-Rough(Rest)- Finish]
  - Irregular Pocket1 [Rough-Rough(Rest)- Finish]
  - MS Hole1 [MSH1(inch)]
  - Counterbore Hole1 [Drill]
  - Countersink Hole1 [Drill]
  - Hole1 [Drill]
- Recycle Bin

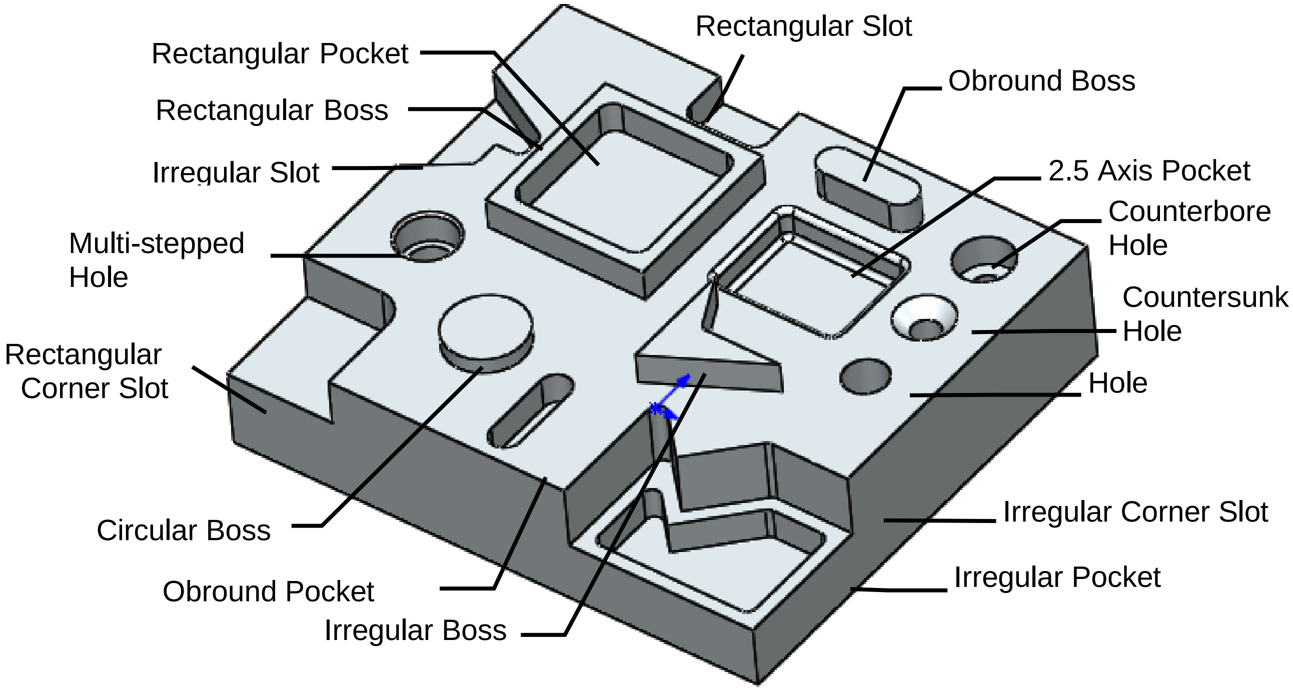
Features recognized automatically after executing EMF command

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

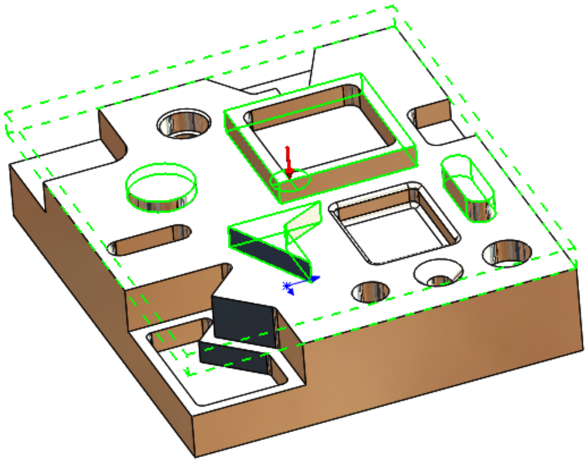
   From the tree, you can reorder features using drag and drop, change feature names and suppress features.

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

As you pick each machinable feature, the geometry for that feature highlights on the part.



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



Open Pocket Feature with boss features as islands

Setting Features Types to be recognized by AFR

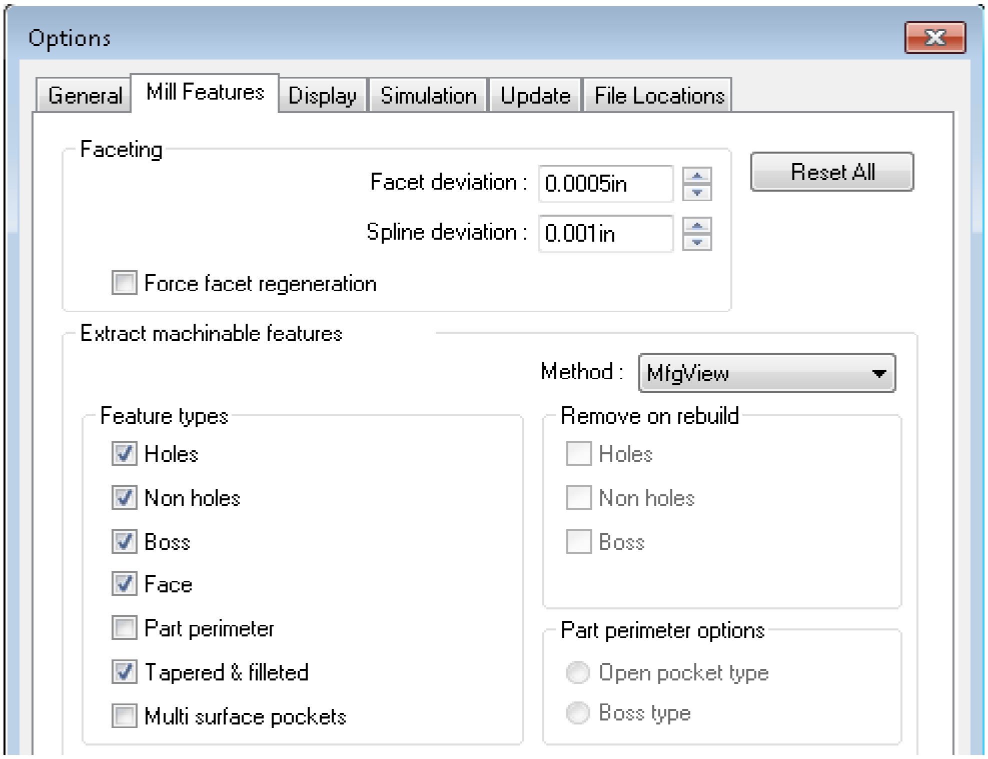
1. Click on the *Options* button in the CAMWorks Command Manager.

   The *Options* dialog box is displayed.

2. Click on the *Mill Features* tab. Notice that when the *Method* is set to *MfgView*, then the defaults in the *Feature types* group box are set for CAMWorks to recognize and create *Hole*s, *Non holes*, *Boss* and *Tapered and filleted* features. If the *Method* is set to *AFR*, then in the *Feature types* group box, the defaults are set to *Hole*s, *Non holes* and *Boss* features.

3. In the Feature types section, check the *Face* option.

   If the *Face* option is checked, then a Face Feature is created automatically when the *Extract Machinable Features* command is executed and is listed under the Feature tree under the Mill Part Setup if the top most face is parallel to the Setup and the Setup is parallel to one of the sides of the stock.



Faceting

Facet deviation : 0.0005in

Spline deviation : 0.001in

☐ Force facet regeneration

Extract machinable features

Method : MfgView

Feature types

☑ Holes

☑ Non holes

☑ Boss

☑ Face

☐ Part perimeter

☑ Tapered & filleted

☐ Multi surface pockets

Remove on rebuild

☐ Holes

☐ Non holes

☐ Boss

Part perimeter options

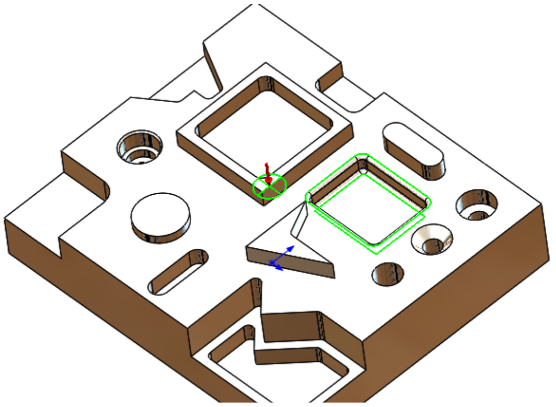
○ Open pocket type

○ Boss type

Selecting Feature Types to be recognized by CAMWorks when EMF command is executed

4. When CAMWorks extracted features, a 2.5 Axis pocket (*Rectangular Pocket2*) was found. This feature was recognized because the *Tapered & filleted* option in the *Feature Types* group box was checked. When the *Tapered & filleted* option is checked, CAMWorks can automatically create 2.5 axis features that meet the definition of filleted and tapered features which fulfill the following conditions:

   – The top and bottom of the feature are flat



Tapered & filleted rectangular pocket (highlighted in green)

and parallel to the Mill Part Setup direction.

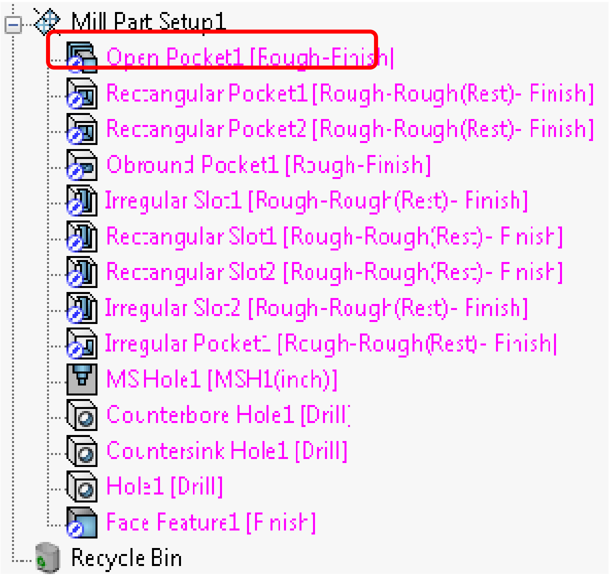
- The side wall can be vertical or a constant taper with a top or bottom fillet and/or a top of bottom chamfer.
- Corner fillets must be conical as defined by the taper angle of the feature.

Note: **Tapered and filleted features which do not fulfill the above conditions are not recognized automatically by AFR when the EMF command is executed. Such features can be created interactively using the New 2.5 Axis Feature command.**

Rebuild the Features

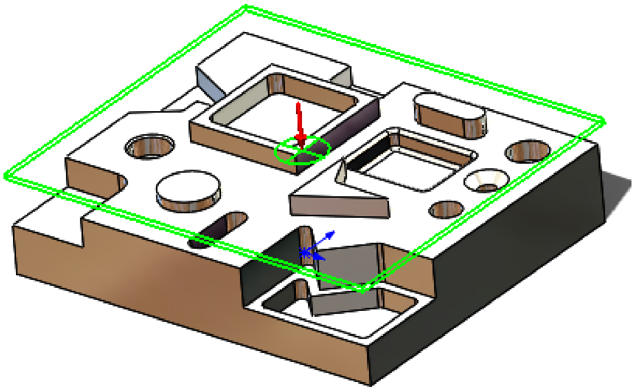
Now that the Face Feature option has been checked within the list of Features Types to be recognized, you need to execute the *Extract Machinable Features* command once again to rebuild the features.

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.
2. CAMWorks rebuilds the feature list. Observe the CAMWorks Feature tree. Notice that AFR has created a Face feature.



Mill Part Setup1
Open Pocket1 [Rough-Finish]
Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
Rectangular Pocket2 [Rough-Rough(Rest)- Finish]
Obround Pocket1 [Rough-Finish]
Irregular Slot1 [Rough-Rough(Rest)- Finish]
Rectangular Slot1 [Rough-Rough(Rest)- Finish]
Rectangular Slot2 [Rough-Rough(Rest)- Finish]
Irregular Slot2 [Rough-Rough(Rest)- Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
MS Hole1 [MSH1(inch)]
Counterbore Hole1 [Drill]
Countersink Hole1 [Drill]
Hole1 [Drill]
Face Feature1 [Finish]
Recycle Bin

Face Feature listed in Feature tree

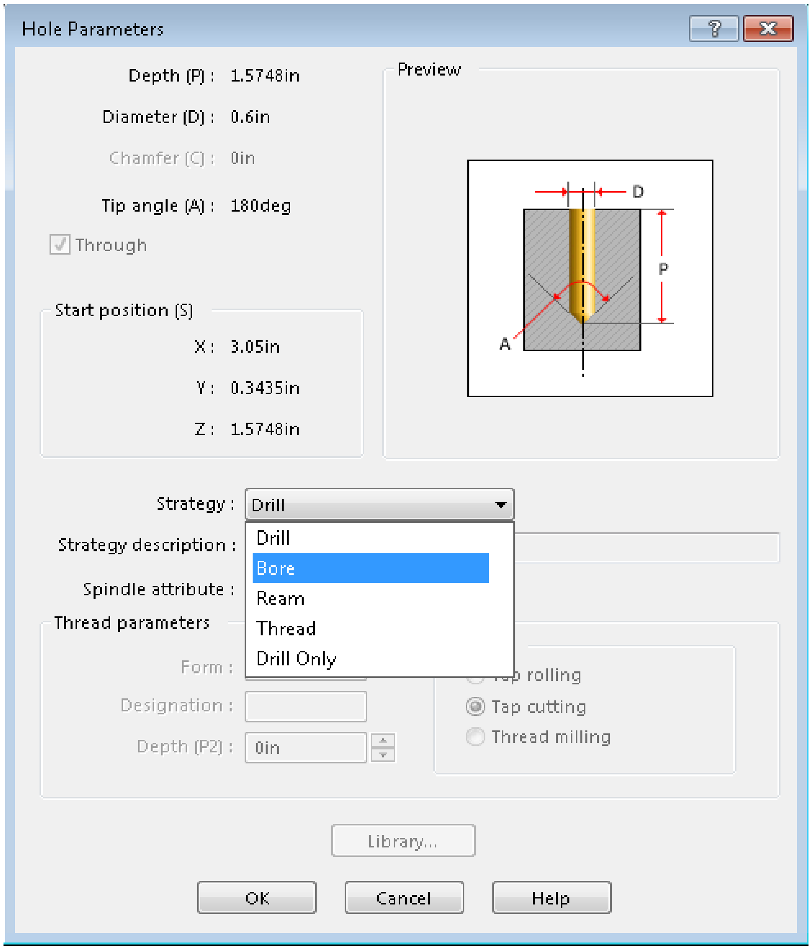


Face Feature (highlighted in green)

Setting Feature Strategy

Machinable features are recognized based on the part's geometry (the shape of a feature) and topology (how the features are related to one another). However, not all strategies of a feature can be determined from the geometry and topology.

For example, Automatic Feature Recognition (AFR) cannot determine whether a hole has been threaded, reamed or bored. CAMWorks allows you to specify an additional Strategy to a feature so that more appropriate operations can be can be created.



Hole Parameters

Depth (P) : 1.5748in

Diameter (D) : 0.6in

Chamfer (C) : 0in

Tip angle (A) : 180deg

☑ Through

Start position (S)

X : 3.05in

Y : 0.3435in

Z : 1.5748in

Strategy : Drill

Strategy description :

Spindle attribute :

Thread parameters

Form :

Designation :

Depth (P2) : 0in

○ p rolling

● Tap cutting

○ Thread milling

Hole Parameters dialog box

Following are the steps to specify the strategy**:**

1. Double click *Hole1* in the Feature tree.

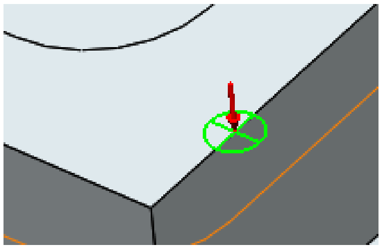
   The *Parameters* dialog box will be displayed. This dialog box provides a geometric and dimensional report on the feature and allows you to assign additional information to the feature. In the case of holes, you can identify whether the hole is to be drilled, bored, reamed or threaded, etc.

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

2. In the *Strategy* dropdown list, select *Bore*.
3. Click *OK* to apply the changes to Strategy and close the dialog box.
4. Review the parameters of other machinable features to understand more about the features.

Mill Part Setup

During Automatic Feature Recognition, the *Extract Machinable Features* command also created *Mill Part Setup1*. The Mill Part Setup defines the tool direction necessary to machine these features.



Marker displayed on part

1. Click *Mill Part Setup1* in the Feature tree.

   A marker displays on the part. The arrow indicates the machining direction. The cross in the center of the circle represents the machining X and

Y axes. The X machining direction can be changed if necessary. For more information, see the CAMWorks context-based Help.

The origin of the Mill Part Setup is program zero. By default, the origin is set to a vertex or middle of the top most features. In most situations, you will want to change this.

How to change the Mill Part Setup origin is explained in **Chapter 3** of this document.

Generate Operation Plan

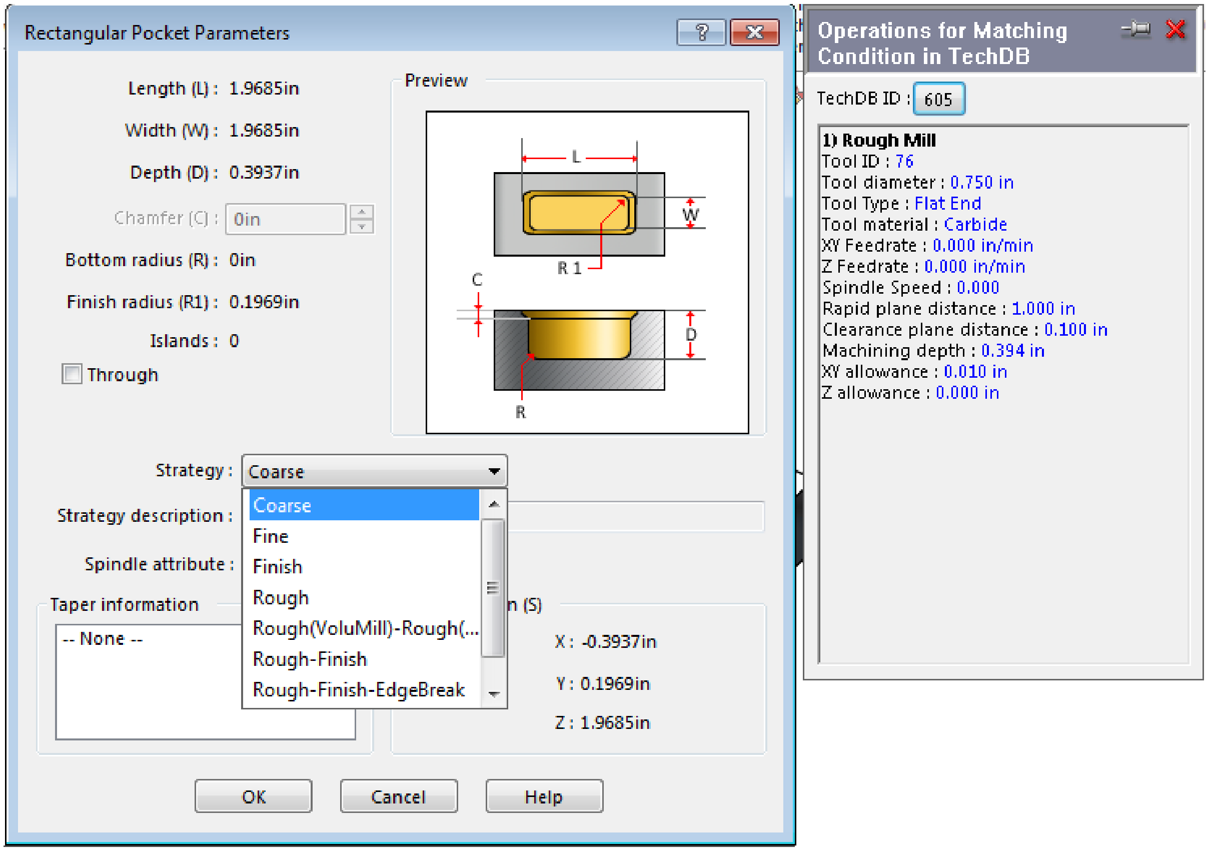
When you execute the *Generate Operation Plan* command, operations are created for every selected feature based on the rules defined in the Technology Database (TechDB). Operations are not generated for suppressed features and features in the Recycle Bin.

In this tutorial, you will first create an operation plan for an individual machinable feature and then for all machinable

features within the Mill Part Setup.

1. Right click on *Rectangular Pocket1* and select *Parameters* on the context menu.
2. The *Rectangular Pocket Parameters* dialog box is displayed. In the Strategy dropdown list, select *Coarse*.

   Coarse is a system Strategy that has been defined in the TechDB to generate a Rough Mill operation for a Rectangular Pocket feature. Details of the Rough Mill operation designated to this strategy will be displayed to the right hand side of the dialog box when you select the Strategy in the dropdown list. After you have learned how to use CAMWorks, you can modify system Strategies to correspond to your requirements.



Rectangular Pocket Parameters

Length (L) : 1.9685in
Width (W) : 1.9685in
Depth (D) : 0.3937in
Chamfer (C) : 0in
Bottom radius (R) : 0in
Finish radius (R1) : 0.1969in
Islands : 0
☐ Through

Strategy : Coarse
Strategy description :
Spindle attribute :

Taper information
-- None --

X : -0.3937in
Y : 0.1969in
Z : 1.9685in

Operations for Matching Condition in TechDB

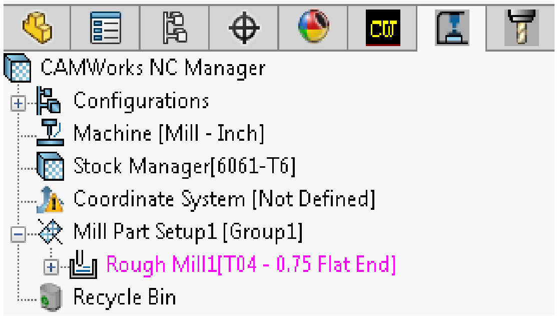
TechDB ID : 605

1) Rough Mill
Tool ID : 76
Tool diameter : 0.750 in
Tool Type : Flat End
Tool material : Carbide
XY Feedrate : 0.000 in/min
Z Feedrate : 0.000 in/min
Spindle Speed : 0.000
Rapid plane distance : 1.000 in
Clearance plane distance : 0.100 in
Machining depth : 0.394 in
XY allowance : 0.010 in
Z allowance : 0.000 in

Strategy set to *Coarse* for Rectangular Pocket feature in Parameters dialog box

3. Click *OK* to apply the changes and close the dialog box.
4. Right click *Rectangular Pocket1* in the Feature tree and select *Generate Operation Plan* command on the context menu.

   The Operation tree lists a *Rough Mill* operation for the rectangular pocket.



Rough Mill operation generated for Rectangular Pocket feature

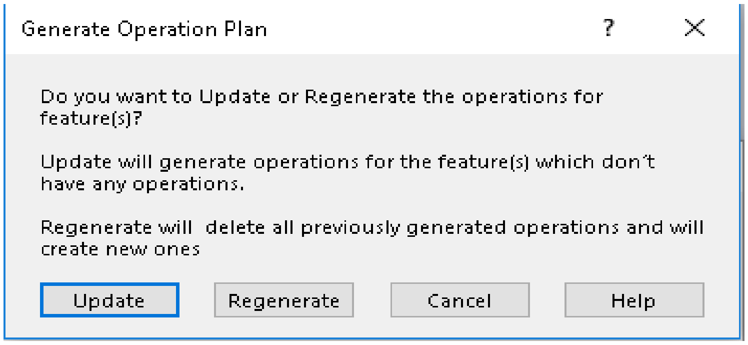
5. CW Click the *CAMWorks Feature Tree* tab to return to the Feature tree.
6. Right click *Rectangular Pocket1* in the tree and select *Parameters* in the context menu.
7. In the *Rectangular Pocket Parameters* dialog box, change the Strategy to *Rough-Finish*, and then click *OK*.

   This *Rough-Finish* strategy is one of the default strategies that have been set up in the TechDB. This strategy has been set up to generate Rough Mill and Contour Mill operations for a Rectangular Pocket feature. You can add your own user-defined strategies and modify or delete existing Strategies within TechDB to customize the TechDB. Refer ***'Technology Database Tutorial'*** for details.
8. Right click *Rectangular Pocket1* in the Feature tree and select *Generate Operation Plan* command.

   Since the *Generate Operation Plan* command has already been run once for this feature, when you execute the command again, CAMWorks displays the *Generate Operation Plan* dialog box which has the following choices:

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

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

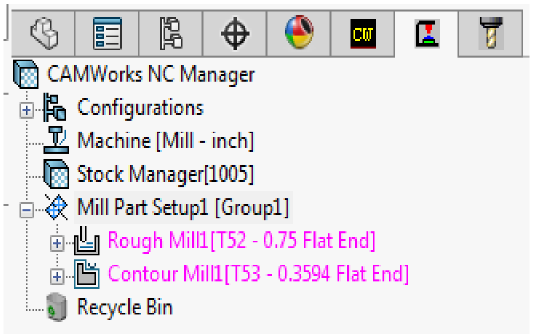


Do you want to Update or Regenerate the operations for feature(s)?

Update will generate operations for the feature(s) which don't have any operations.

Regenerate will delete all previously generated operations and will create new ones

Dialog box displayed when GOP command is executed for a feature with operation(s)


CW
CAMWorks NC Manager
Configurations
Machine [Mill - inch]
Stock Manager[1005]
Mill Part Setup1 [Group1]
Rough Mill1[T52 - 0.75 Flat End]
Contour Mill1[T53 - 0.3594 Flat End]
Recycle Bin

Operations generated when strategy is set to *Rough Finish*

Cancel: Click *Cancel* to revoke the *Generate Operation Plan* command.

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

   The Operation tree lists Rough Mill and Contour Mill operations for the rectangular pocket.

10. CW Click the *CAMWorks Feature Tree* tab to return to the Feature tree.
11. Right click *Mill Part Setup1* in the Feature tree and select *Generate Operation Plan* command on the context menu.
12. Since operations already exist for at least one feature listed in the Mill Part Setup (*Rectangular Pocket1* in this case), the *Generate Operation Plan* dialog box will be displayed. Select *Update* in this dialog box in order to generate operations only for those features for which do not have operations.

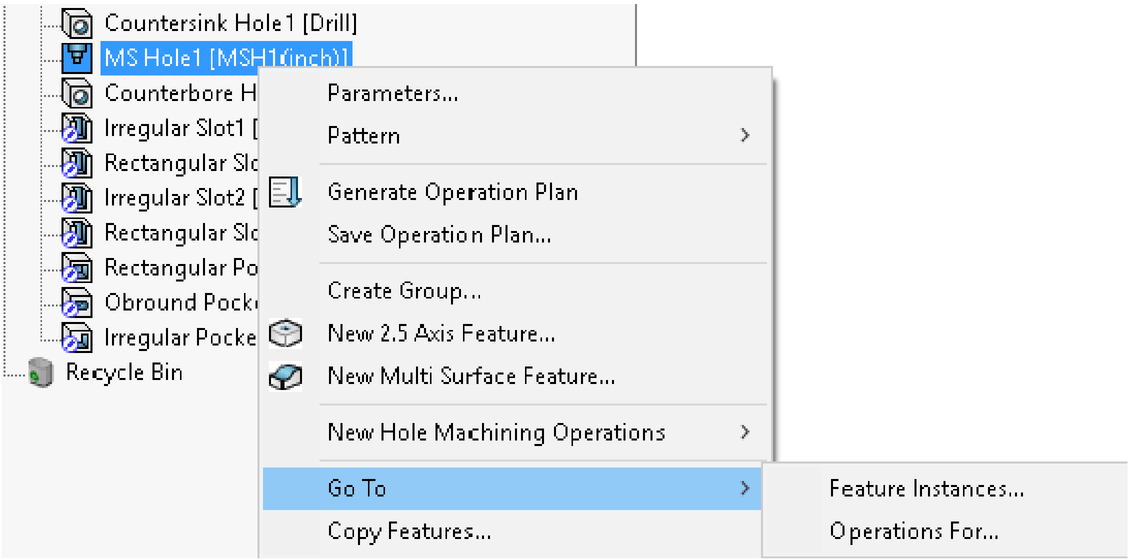
    You have not changed any operation parameters for the Rough Mill and Contour Mill operations that have already been generated for *Rectangular Pocket1*. Hence, you could select either *Regenerate* or *Update* because in both cases, the result would be the same. If you had modified these operations, you would want to select *Update* to generate an operation list only for the features that do not have operations.

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

Locating Operations for Selected Feature

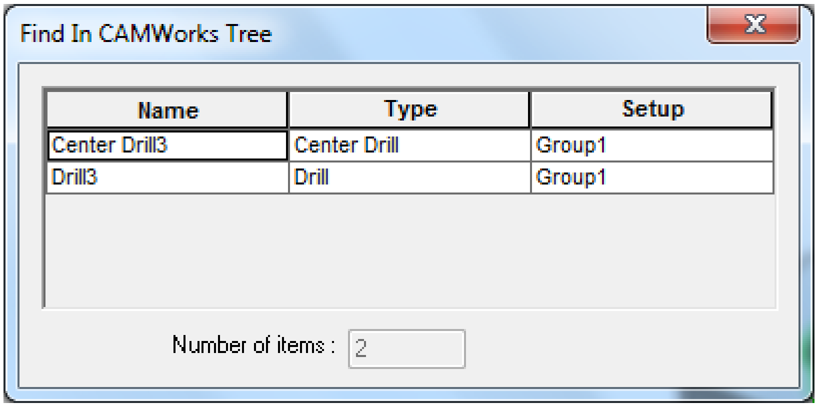
You can use the *Go To* command for locating operations for the desired feature.

1. CW Click the *CAMWorks Feature Tree* tab.
2. Right click on *MS Hole1 [MSH1(inch)]* in the Feature tree and select *Go To*, then select *Operations For…* on the context menu.



The *Operations For…* command in the context menu

This command allows you to quickly locate operations that were generated for a selected feature.



Name	Type	Setup
Center Drill3	Center Drill	Group1
Drill3	Drill	Group1

Number of items : 2

Find in CAMWorks Tree dialog box

The *Find in CAMWorks Tree* dialog box is displayed. The operation *Center Drill3* and *Drill3* is listed in this dialog box. A lone drill operation is not enough to machine a multi-step feature.

In the TechDB, the Strategy [MSH1(inch)] has been defined to generate a center drill cycle, 2 single drill cycles, 2 Rough Mill cycles and 2 Thread Mill cycles for multi-stepped hole features. Since multi-stepped holes can be any shape, you need to define machining sequences in the TechDB that are common for machining multi-stepped holes in your facility.

To generate the NC code for this part, you would adjust the operation parameters as required for the operations, insert a series of operations for the multi-stepped hole, generate toolpaths, and post process the part. These steps will be explored in further tutorials.

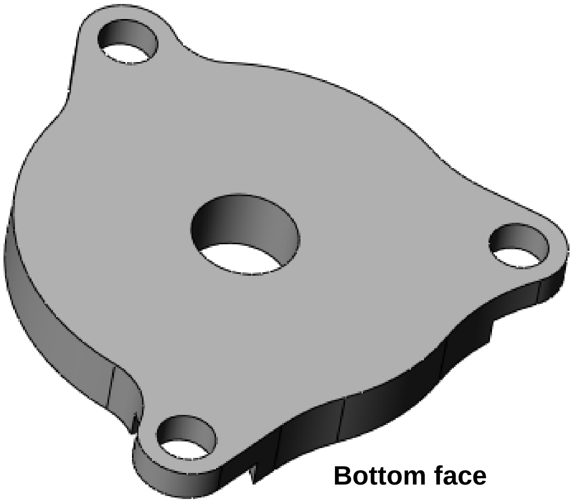
2 Axis Mill 3

Topics covered in this tutorial:

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

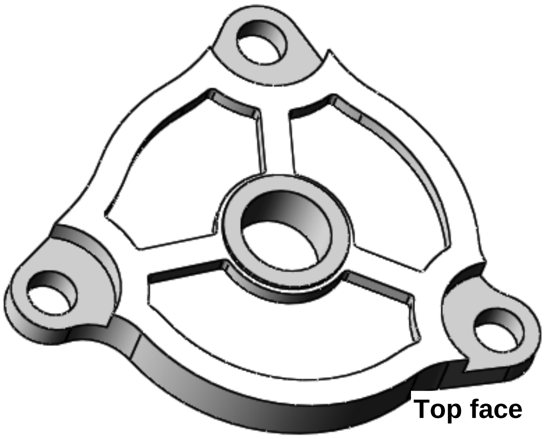
In this tutorial, assume that the part starts out as a casting and machining is required only for the holes and certain faces.

In the first clamping position or Mill Part Setup, you will face the bottom of the part and drill the large center hole and the three outside holes.


Bottom face

In the second clamping position or Mill Part Setup, you will machine off the top center face and the three irregular corner slots (shown in dark grey color).

Note that the irregular pockets (shown in light grey color) and the part perimeter will not be machined.


Top face

MILL2AX_3.SLDPRT: The part to be machined

Step 1: Opening the Part and Defining the Machine

1. Open part file **MILL2AX_3.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.
3. Double click on *Machine [Mill–Inch]* in the Feature tree. The Machine dialog box will be displayed.
   - On the *Machine* tab, ensure that *Mill–Inch* is the Active machine.

- Click on the *Tool Crib* tab and make sure *Tool Crib 2 (Inch)* is the Active tool set.
- Click on the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected; then click *OK* to close the *Machine* dialog box.

Step 2: Defining the Stock from a Bounding Box

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

The stock definition serves several purposes:

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

By default, the stock shape is a cube (bounding box) whose dimensions enclose the part. However, in reality, the size of the stock is usually larger than the size of the part.

For this tutorial, assume that the part is machined from a casting that has **0.1inch** material to be removed from the top, bottom and side surfaces. You will use the default Stock Type of *Bounding box* and offset the minimum stock size by 0.1inch.

1. Double click *Stock Manager* in the CAMWorks Feature tree.
2. The Stock Manager dialog box is displayed. Change the Bounding box offsets for X+, Y+, Z+, X-, Y- and Z- to a uniform **0.1in**.
3. Click *OK* to apply the offsets and close the dialog box.

Step 3: Extracting Machinable Features

Options Dialog box

Before CAMWorks Extracts Machinable Features, you can customize AFR by using the CAMWorks Options dialog box.

1. Click the *Options* button on the CAMWorks Command Manager.
2. The *Options* dialog box is displayed. Click the *Mill Features* tab in this dialog box.
3. Ensure that the *Method* is set to *MfgView*.
4. In the Feature types section, make sure the *Face* option is not checked.

   CAMWorks can recognize some Face features automatically when you select the *Face* option. However, for some parts, you may need to define a Face Feature interactively since the Face Feature recognized by AFR might not be suitable for your machining requirements. Interactive insertion of Face feature is explained in this tutorial.
5. Click *OK* to apply the changes and close the dialog box.

Executing Extract Machinable Features Command

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

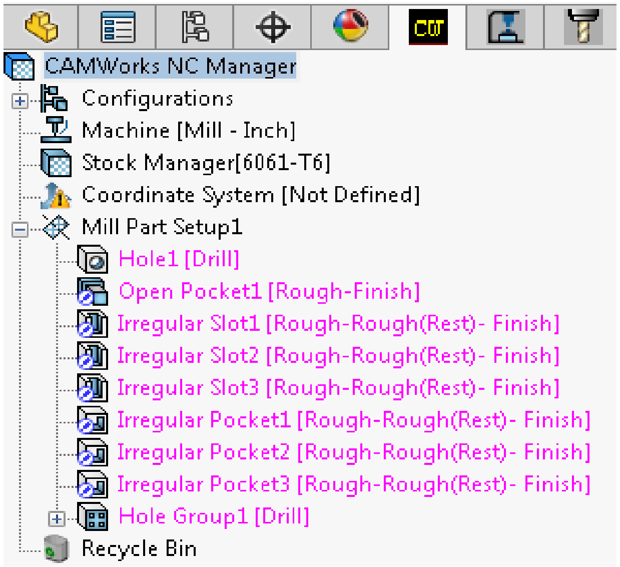
OR

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

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

Observe that the Face features weren’t recognized as this feature type was not selected in the types of features to be recognized using AFR in the Options dialog box.

In this tutorial, the face feature for the top and bottom face of the part will be inserted interactively.


CAMWorks NC Manager
Configurations
Machine [Mill - Inch]
Stock Manager[6061-T6]
Coordinate System [Not Defined]
Mill Part Setup1
Hole1 [Drill]
Open Pocket1 [Rough-Finish]
Irregular Slot1 [Rough-Rough(Rest)- Finish]
Irregular Slot2 [Rough-Rough(Rest)- Finish]
Irregular Slot3 [Rough-Rough(Rest)- Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Hole Group1 [Drill]
Recycle Bin

Features recognized through AFR

Step 4: Interactive Feature Recognition

Need for Interactive Feature Recognition

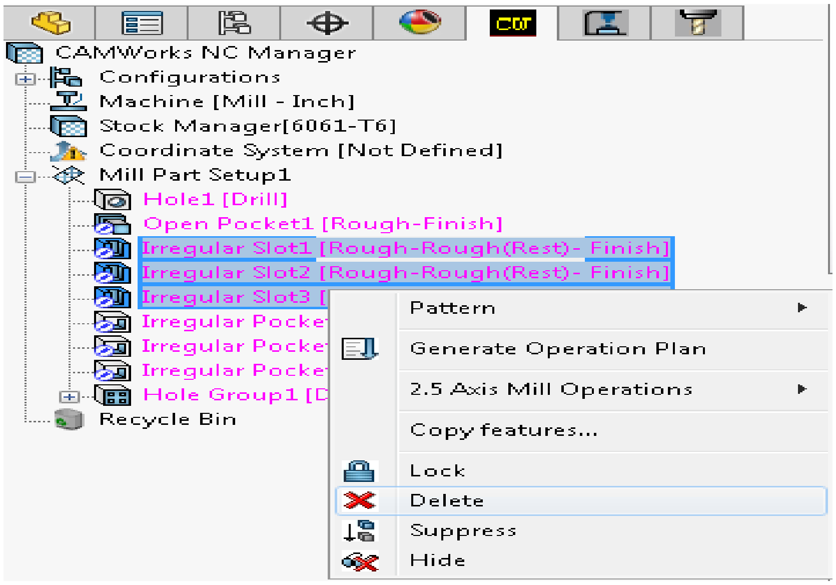
CAMWorks may not be able to find all features on a part. On complex parts, AFR may not recognize all possible features that you want to machine or AFR may recognize a feature that might not be suitable for the intended machining process. When this occurs, you can define these features interactively.

Deleting a Feature

For learning purposes, in this tutorial, you will have to delete the three irregular corner slot features and then insert them interactively.

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

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



Deleting features

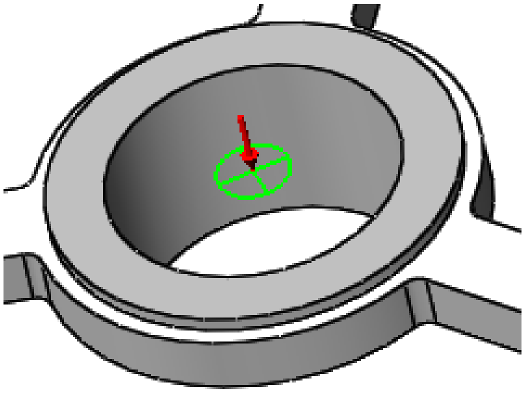
3. Click the minus sign ⊟ to the left of the Recycle Bin to collapse it.

Inserting Additional Mill Part Setups

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

Mill Part Setup1 defines the tool direction pointing down from the top of the part. In this tutorial, you need to face the underside of the part first.

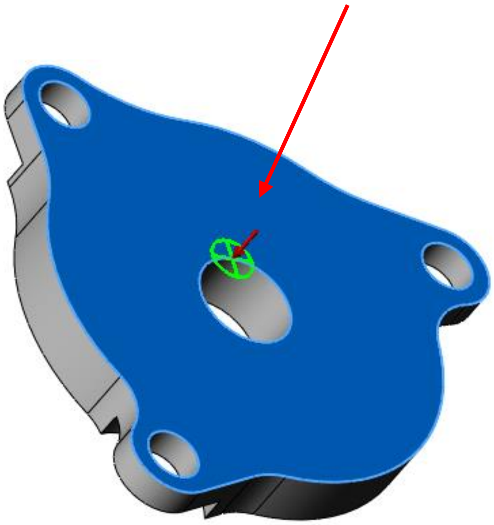
To machine the underside, you need to define a Mill Part Setup for that tool direction. New *Mill Part Setups* can be interactively inserted into the CAMWorks Feature tree by selecting a part face that is normal to the tool axis of the Mill Part Setup.



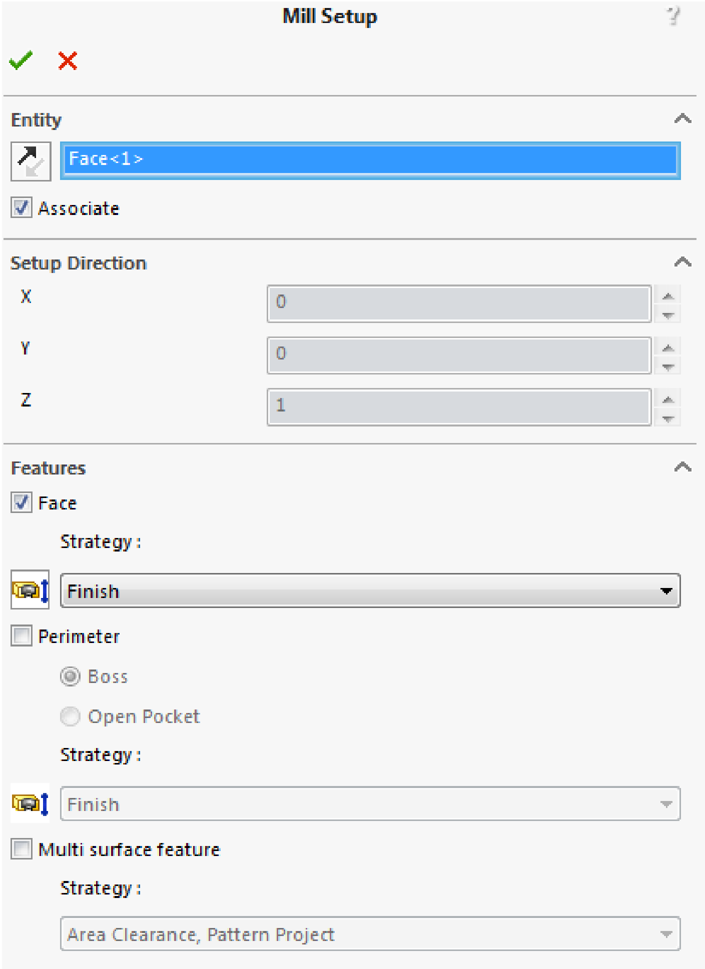
Machining direction indicated by triad

Steps to interactively insert a Mill Part Setup:

1. Rotate the part so that the underside surface is visible.
2. Right click *Mill Part Setup1* in the CAMWorks Feature tree and select *Mill Part Setup* from the context menu. The Mill Setup dialog box is displayed.
3. In the graphics area, pick the underside face. This face will be listed in the *Selected Entity* field.



Underside of the part



Mill Setup

Entity

Face<1>

☑ Associate

Setup Direction

X 0

Y 0

Z 1

Features

☑ Face

Strategy : Finish

☐ Perimeter

● Boss

○ Open Pocket

Strategy : Finish

☐ Multi surface feature

Strategy : Area Clearance, Pattern Project

Mill Setup Dialog box

4. In the *Features* group box, make sure the *Face* option is checked.

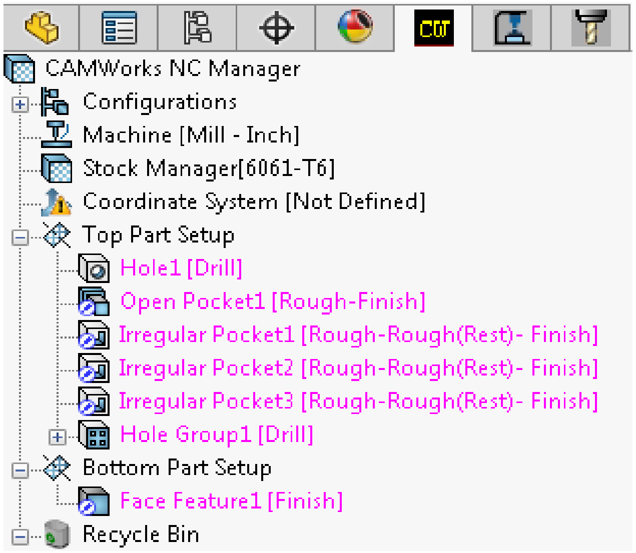
   If this option is checked, when CAMWorks creates a Mill Part Setup, a face feature is created automatically if the top most face is parallel to the Setup and the Setup is parallel to one of the sides of the stock.

5. Set the Strategy to *Finish*.

6. ✓ Click *OK* to close the *Mill Setup* dialog box.

   Mill Part Setup2 is created in CAMWorks Feature tree with the tool direction. The Face Feature is created automatically and listed under this Mill Part Setup.

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



CAMWorks NC Manager
Configurations
Machine [Mill - Inch]
Stock Manager[6061-T6]
Coordinate System [Not Defined]
Top Part Setup
Hole1 [Drill]
Open Pocket1 [Rough-Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Hole Group1 [Drill]
Bottom Part Setup
Face Feature1 [Finish]
Recycle Bin

Interactively inserted Mill Part Setup

7. Right click *Mill Part Setup1* and select *Rename* from the context menu.

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

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

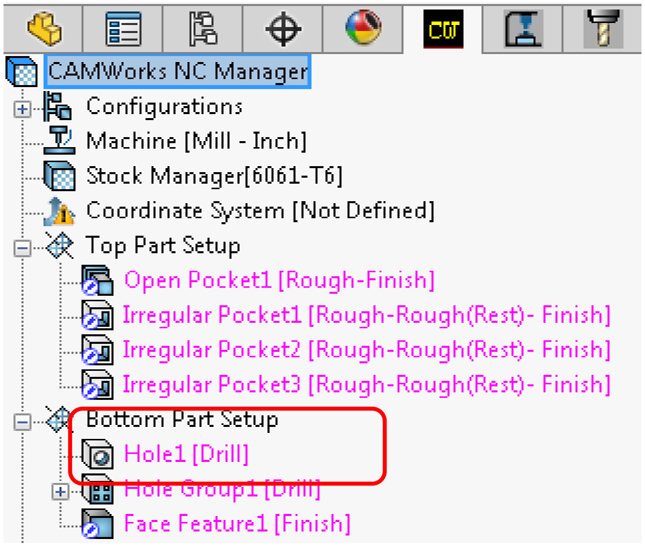
9. Right click *Mill Part Setup2* and select *Rename* from the context menu.

10. Change the name to **Bottom Part Setup**, and then press *Enter*.

Reorganizing Machinable Features

In addition to the face feature, you can also drill the hole features from the *Bottom Part Setup*. However, the hole features were recognized in the *Top Part Setup*. With the mouse, you can drag the features in the CAMWorks Feature tree from the *Top Part Setup* to the *Bottom Part Setup*.

1. Hold down the left mouse button and drag *Hole1* over Bottom Part Setup, then release the button.
   - Notice that as you drag the item, an arrow displays to indicate where the item will be positioned.



CAMWorks NC Manager
Configurations
Machine [Mill - Inch]
Stock Manager[6061-T6]
Coordinate System [Not Defined]
Top Part Setup
Open Pocket1 [Rough-Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Bottom Part Setup
Hole1 [Drill]
Hole Group1 [Drill]
Face Feature1 [Finish]

Reorganizing features using drag and drop function

– *Hole1* will be listed after *Bottom Part Setup*.

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

   Hole Group1 will be listed after *Hole1*.

Suppressing Machinable Features

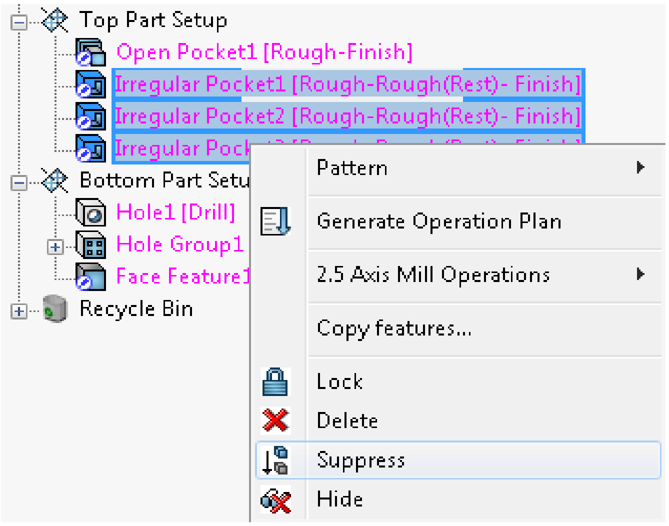
You will now define features to machine the top of the part. The only machinable features left in the Top Part Setup are the Open Pocket and the three irregular pockets. Since the irregular pocket features will not be machined in this tutorial, you can either delete or suppress them. The steps below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

1. Hold down the *Ctrl* key and select *Irregular Pocket1*, *Irregular Pocket2*, and *Irregular Pocket3* in the CAMWorks Feature tree.

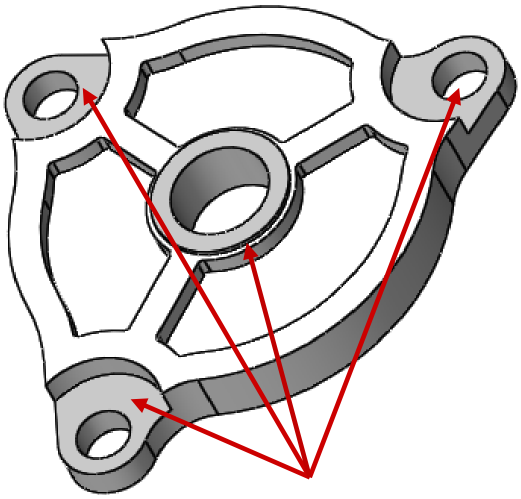
   Make sure you select all 3 features.

2. Right click and select *Suppress* from the context menu.

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



Suppressing features



Features to be interactively inserted

3. Click *Save As* on the *File* menu.
4. The *Save as* dialog box is displayed. In the *File name* field, type an appropriate file name and click the *Save* button.

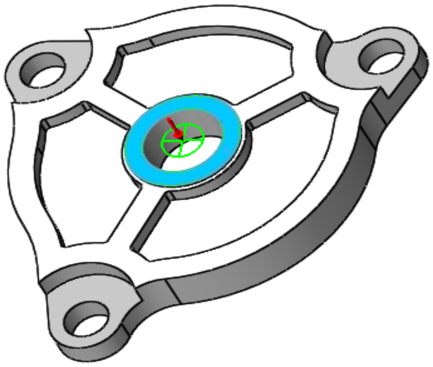
Inserting 2.5 Axis Features

As stated earlier, in this tutorial, the face feature and irregular slot features for this part will be inserted interactively. The face feature for the underside of the part was inserted along with the interactive insertion of Mill Part Setup *(Bottom Part Setup)*.

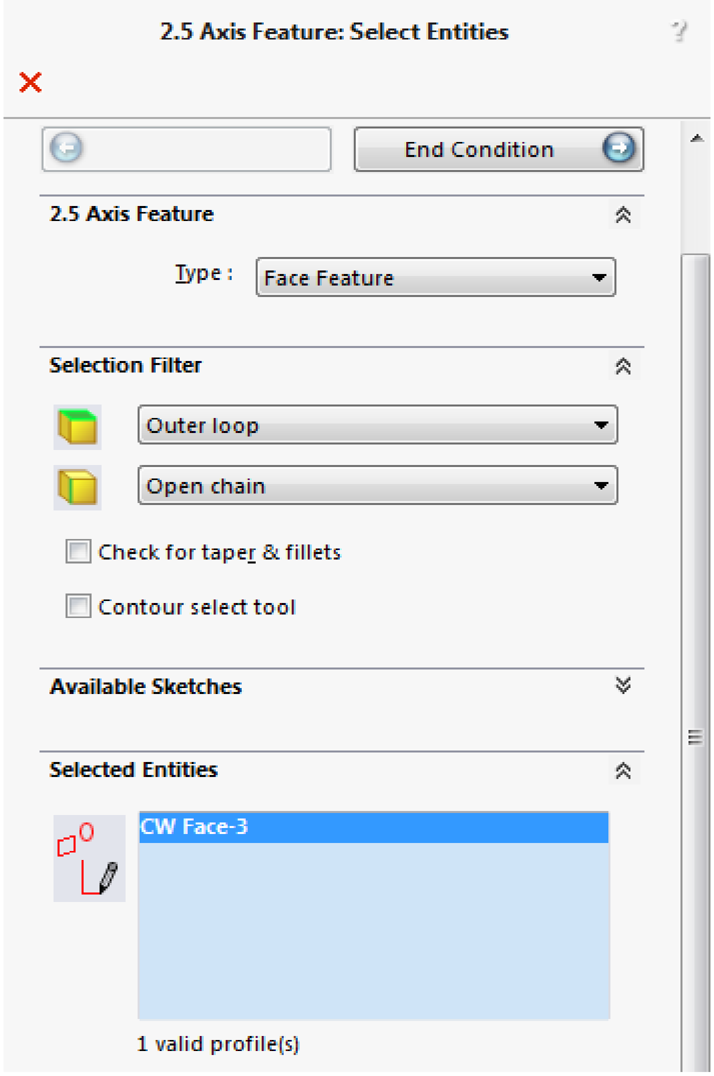
The top of the circular boss and the three irregular slots will be machined using the *New 2.5 Axis Feature* command to define them.

Steps to interactively insert Face Feature:

1. Rotate the part so the top face is visible.
2. Right click *Top Part Setup* in the CAMWorks Feature tree and select *2.5 Axis Feature* in the context menu.
3. The *2.5 Axis Feature: Select Entities* dialog box is displayed. Click the down arrow to display the *Feature Type* list and select *Face Feature*.
4. Pick the top center face from the top of the boss. On executing this action, it is highlighted in the graphics area.



Picking the face to interactively insert face feature



2.5 Axis Feature: Select Entities

2.5 Axis Feature

Type : Face Feature

Selection Filter

Outer loop

Open chain

☐ Check for taper & fillets

☐ Contour select tool

Available Sketches

Selected Entities

CW Face-3

1 valid profile(s)

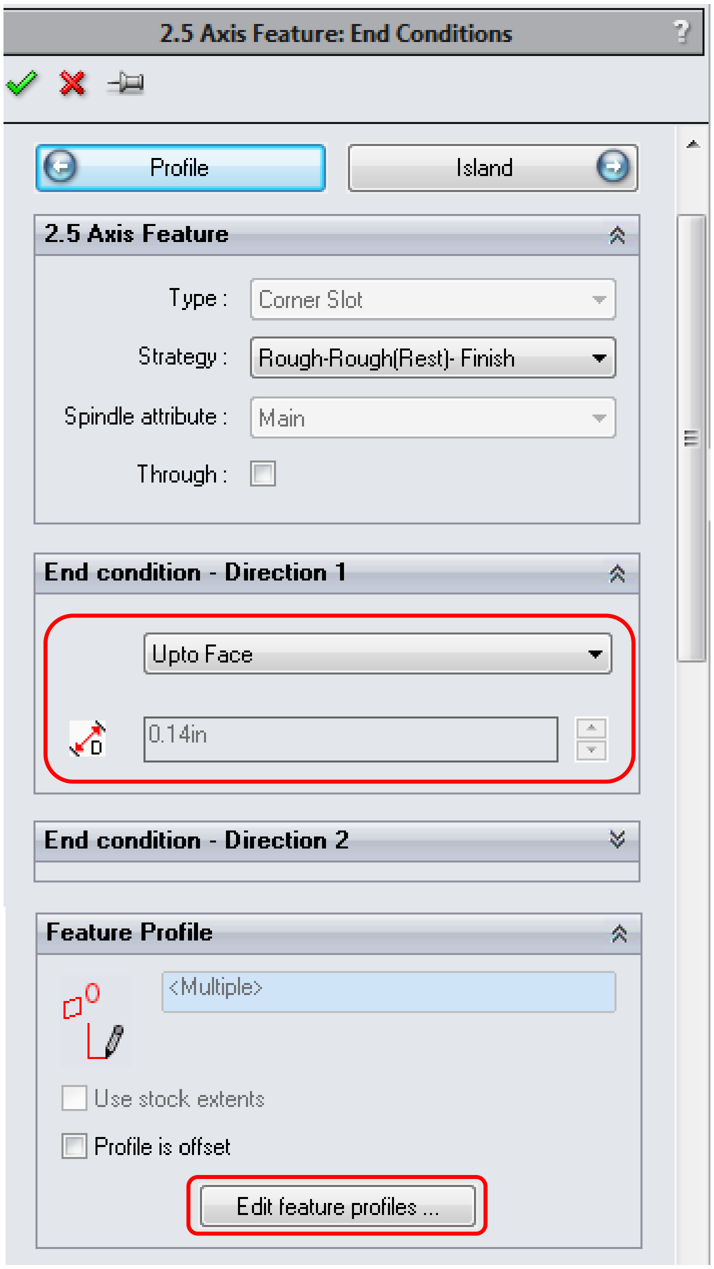
Interactively inserting the face feature

CW Face-3 is displayed in the *Selected Entities* field.

5. Click the *End Condition*button.

6. The *2.5 Axis Feature: End Conditions* dialog box will be displayed. Set the Strategy to *Finish*.
7. Leave the *End Condition Type* as *Upto Stock*.
8. ✓ Click *OK* to insert the feature and close the dialog box.

   Observe that *Face Feature2* is added at the bottom of the list of features under *Top Part Setup*.


2.5 Axis Feature: End Conditions
?
Profile
Island
2.5 Axis Feature
End condition - Direction 1
End condition - Direction 2
Feature Profile
Edit feature profiles ...

Type : Corner Slot

Strategy : Rough-Rough(Rest)- Finish

Spindle attribute : Main

Through : ☐

Upto Face

0.14in

<Multiple>

☐ Use stock extents

☐ Profile is offset

2.5 Axis Feature: End Conditions dialog box

Steps to interactively insert Corner Slot Feature:

After insertion of the Face feature, you will now use the *2.5 Axis Feature: Selected Entities* dialog box to define 2.5 Axis features for the three irregular corner slots.

1. Right-click on *Face Feature2* and select *2.5 Axis Feature* on the context menu.
2. In the *2.5 Axis Feature: Selected Entities* dialog box, click the down arrow to display the *Feature Type list* and select *Corner Slot*.

   These slots are defined as Corner Slots because more than one edge is open. By defining it as a corner slot, the tool will machine outside the slots on the open air edges.
3. In the graphics area, pick each of the three dark grey faces of the corner slots.

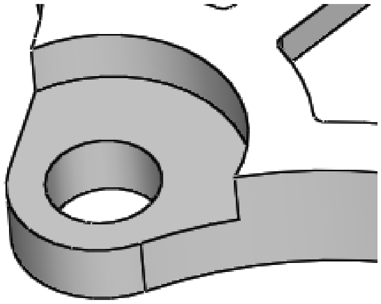
   The *Entities selected* list displays *CW Face-0*, *CW Face-1* and *CW Face-2*.
4. End Condition Click the *End Conditions* button. The *2.5 Axis Feature: End Conditions* dialog box will be displayed.

   The depth of material to leave for

this feature will be from the face of the corner slot to the face immediately above the corner slot. You can either pick an end condition from the *End Condition* dropdown list or select a face or vertex and have CAMWorks set the depth automatically.

5. Leave the *End condition* set to *Blind* and pick the dark grey colored face above the corner slot.

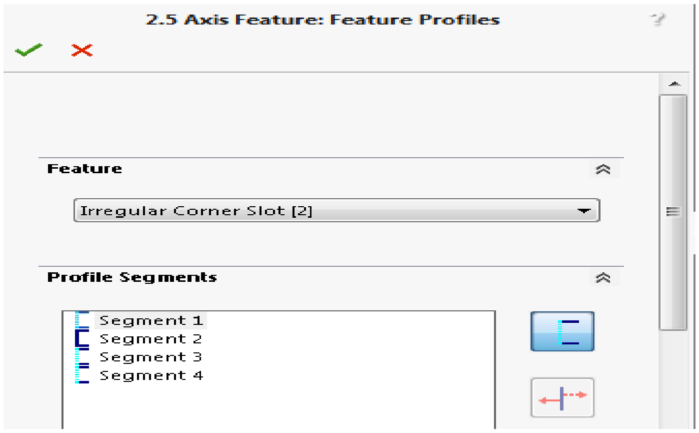
   On executing this action, the *End condition* is automatically set to *Upto Face* and the *Depth* parameter displays the distance as **0.14in**.



Pick the dark grey colored face

6. Click on the *Edit feature profiles* button.

   The *2.5 Axis Feature: Feature Profiles* dialog box will be displayed. When a Corner Slot is defined from a part face, CAMWorks determines the open air edges automatically. If necessary, you can change the open air edges.

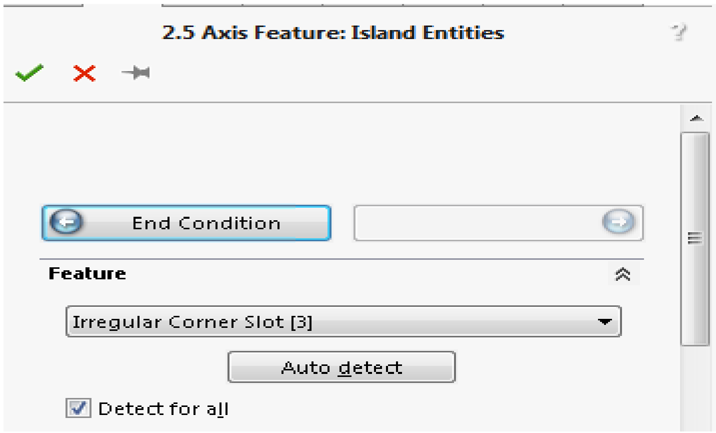


Irregular Corner Slot [2]

Segment 1
Segment 2
Segment 3
Segment 4

7. Select the *Feature* for which you wish to review the open air edges in the *Feature* dropdown list. Suppose *Irregular Corner Slot (2)* is selected.
8. Select any segment in the *Profile Segments* list.

   The corresponding edge is highlighted on *Irregular Corner Slot(2)*. If CAMWorks determined that the edge as an open air edge, then the Air Segment button is active.
9. For every segment listed in the *Profile segments* list, observe that the edges have been correctly identified by CAMWorks as Air Segment



Irregular Corner Slot [3]

☑ Detect for all

2.5 Axis Feature: Island Entities dialog box open/closed. You do not need to change the feature profile of any segments.

10. ✓ Click *OK*.

    The *2.5 Axis Feature: End Conditions* dialog box will be displayed. This dialog box is displayed when you define a Pocket, Slot, Corner Slot, or Face Feature.

11. Island ➜ Click the *Island* button.

    The *2.5 Axis Feature: Island Entities* dialog box will be displayed.

12. Make sure the *Detect for all* check box option is checked.

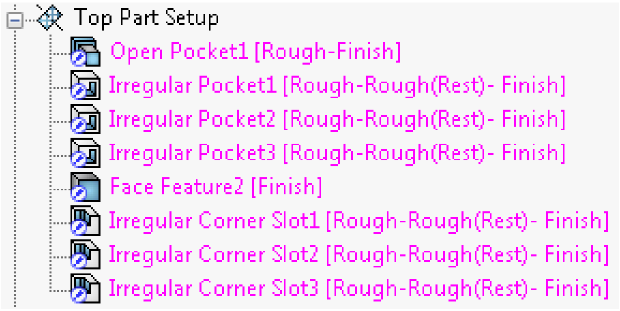
    When you define multiple features and select this option, CAMWorks attempts to find islands for all features. If this option is not checked, CAMWorks attempts to find islands only for the current feature.

13. Click *Auto detect*.

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

14. ✓ Click *OK* to insert the corner slot features and close the dialog box.

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



Top Part Setup
- Open Pocket1 [Rough-Finish]
- Irregular Pocket1 [Rough-Rough(Rest)- Finish]
- Irregular Pocket2 [Rough-Rough(Rest)- Finish]
- Irregular Pocket3 [Rough-Rough(Rest)- Finish]
- Face Feature2 [Finish]
- Irregular Corner Slot1 [Rough-Rough(Rest)- Finish]
- Irregular Corner Slot2 [Rough-Rough(Rest)- Finish]
- Irregular Corner Slot3 [Rough-Rough(Rest)- Finish]

Interactively inserted irregular slot features

Combining Machinable Features

Group Features

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

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

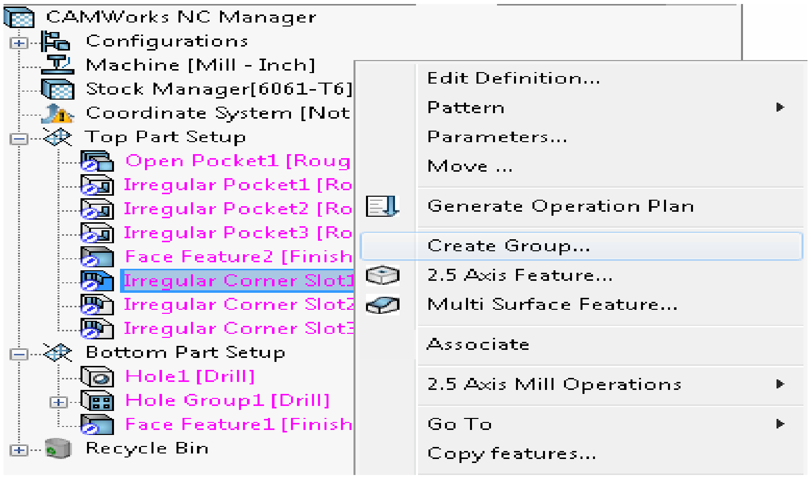
Steps to create a Group Feature:

In this tutorial, the irregular corner slot features will be grouped. Following are the steps to group the features:

1. Right click the first Irregular Corner Slot *(Irregular Corner Slot1)* in the CAMWorks Feature tree and select *Create Group* in the RMB context menu.

   The *Edit Feature Group* dialog box will be displayed.

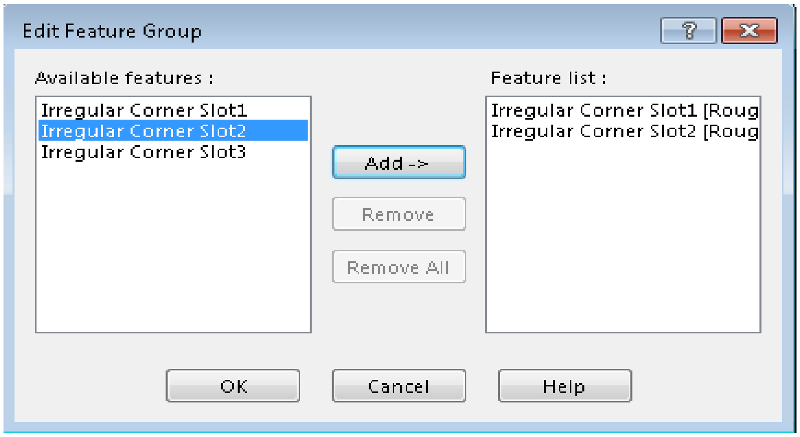
   The *Feature List* on the right displays the selected feature. The *Available Features* list on the left shows other features of the same type and depth that can be added to make the Group feature.



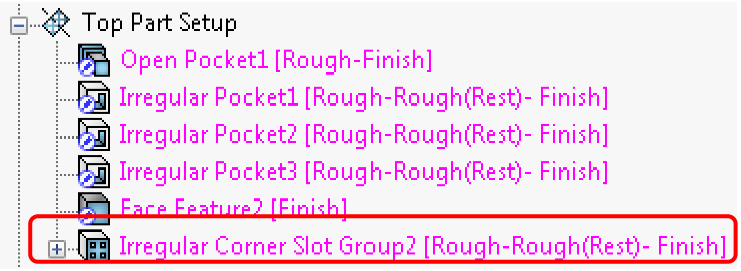
Create Group command on the RMB context menu

2. Pick *Irregular Corner Slot2*, click *Add*.
3. Observe that this feature is added to the *Feature List* on the right. Pick *Irregular Corner Slot3* and click *Add*.
4. Click *OK* to create the group and close the dialog box.

   The three pockets are combined into a single feature labeled *Irregular Corner Slot Group2* in the CAMWorks Feature tree.



Edit feature Group Dialog Box


Top Part Setup
Open Pocket1 [Rough-Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Face Feature2 [Finish]
Irregular Corner Slot Group2 [Rough-Rough(Rest)- Finish]

Group Feature added

Step 5: Generate Operation Plan and Toolpaths

Generate an Operation Plan for each Mill Part Setup

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

   OR

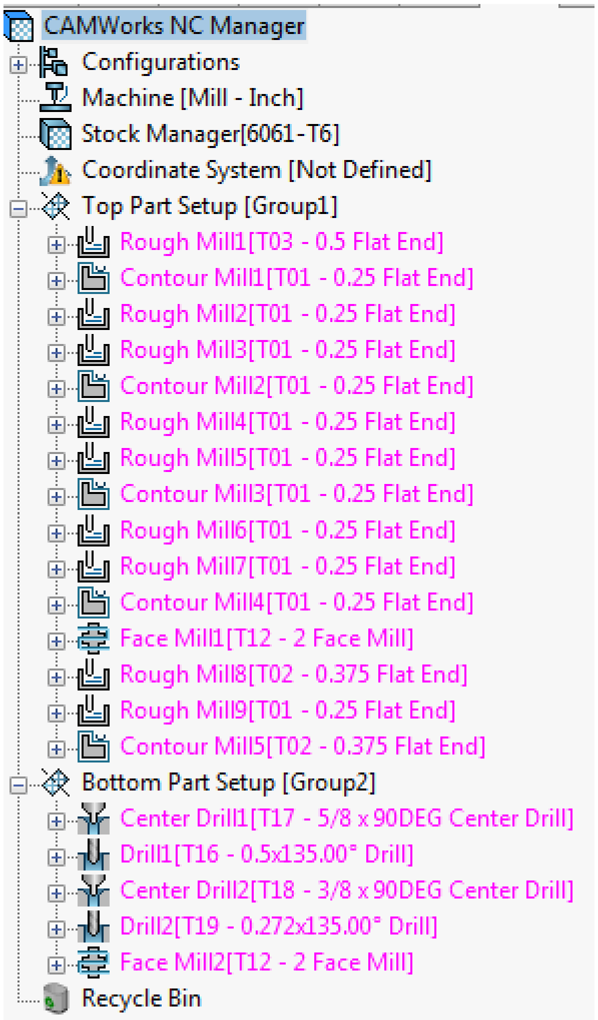
   Right click *CAMWorks NC Manager* in the CAMWorks Feature tree, select *Generate Operation Plan* from the context menu.

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

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

2. Observe that *Center Drill* operations and *Drill2* operations under *Bottom Part Setup* have a red exclamation mark because the tool assigned to these operations are not from the active tool crib.

   Right-click on each of these operations and select *What's Wrong?* from the context menu. In the Errors message box that is displayed, click the Clear button in order to get rid of the exclamation mark.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Inch]
- Stock Manager[6061-T6]
- Coordinate System [Not Defined]
- Top Part Setup [Group1]
  - Rough Mill1[T03 - 0.5 Flat End]
  - Contour Mill1[T01 - 0.25 Flat End]
  - Rough Mill2[T01 - 0.25 Flat End]
  - Rough Mill3[T01 - 0.25 Flat End]
  - Contour Mill2[T01 - 0.25 Flat End]
  - Rough Mill4[T01 - 0.25 Flat End]
  - Rough Mill5[T01 - 0.25 Flat End]
  - Contour Mill3[T01 - 0.25 Flat End]
  - Rough Mill6[T01 - 0.25 Flat End]
  - Rough Mill7[T01 - 0.25 Flat End]
  - Contour Mill4[T01 - 0.25 Flat End]
  - Face Mill1[T12 - 2 Face Mill]
  - Rough Mill8[T02 - 0.375 Flat End]
  - Rough Mill9[T01 - 0.25 Flat End]
  - Contour Mill5[T02 - 0.375 Flat End]
- Bottom Part Setup [Group2]
  - Center Drill1[T17 - 5/8 x 90DEG Center Drill]
  - Drill1[T16 - 0.5x135.00° Drill]
  - Center Drill2[T18 - 3/8 x 90DEG Center Drill]
  - Drill2[T19 - 0.272x135.00° Drill]
  - Face Mill2[T12 - 2 Face Mill]
- Recycle Bin

Operation tree after generation of features

3. If the Face Mill operation is not listed first in the *Top Part Setup* or *Bottom Part Setup*, you can drag and drop this operation to the first position.

   In the *Top Part Setup*, press and hold down the left mouse button with the pointer over *Face Mill1*.

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

   The Operation tree re-displays with the operation moved to the new location.

5. Repeat steps 2 for *Face Mill2* of *Bottom Part Setup* and drag the operation over the top of *Bottom Part Setup* and release the button. This action moves it first position under the *Bottom Part Setup*.

Generate Toolpaths and Post Process the Part

Verify Operation Parameters and Generating Toolpath

1. Double click each operation item in the tree to open the Operation Parameters dialog box and review the cutting parameters.
2. After you have verified that the parameters for each operation are set the way you want them, right click *Bottom Part Setup* in the CAMWorks Operation tree and select *Generate Toolpath* command in the RMB context menu.
3. Repeat step 2 for the *Top Part Setup*.
4. Experiment by changing parameters for the operations and regenerating the toolpaths.

CAMWorks Message Window

Check the CAMWorks Message Window.

- The Message Window provides information you may find helpful. For example, based on the information in this window, you may want to change a tool or insert a finish operation in order to cut a machinable feature completely.
  - If the Message Window is not displayed, select *Options* on the CAMWorks Command Manager and check the Message Window check box on the General tab.
- You can move the window and adjust the size. To change the size, point to a border or corner of the window (the pointer changes to a two-headed arrow). Drag the corner or border until the window is the size you want, and then release the mouse button.

Simulating the Toolpath

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

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

   Observe the simulated toolpath. In case of any errors or collisions, modify the Operation Parameters, regenerate toolpaths and simulate the toolpaths once again. Observe that the perimeter of the part is not simulated since no part perimeter feature was inserted in the list of features to be machined.

Step 6: Post Processing the Toolpaths

1. Right click *Bottom Part Setup* in the CAMWorks Operation tree and select *Post Process* in the RMB context menu.
2. In the *Save Post Output File* dialog box, click *Save* to accept the default file name. Change the file name if required.

Note that if you are running CAMWorks in Demonstration mode, this menu option and dialog box does not display.

3. The *Post Process Output* dialog box is displayed. In the *Display* group box of this dialog box, check the *Centerline* option so that the toolpath will be highlighted as each line of code is produced.
4. Click the *Play* button ▶.

   NC code is created for the *Bottom Part Setup*.
5. After viewing the code, click *OK* to close the

   dialog box.
6. Right click *Top Part Setup* in the CAMWorks

   Operation tree and select *Post Process* in the

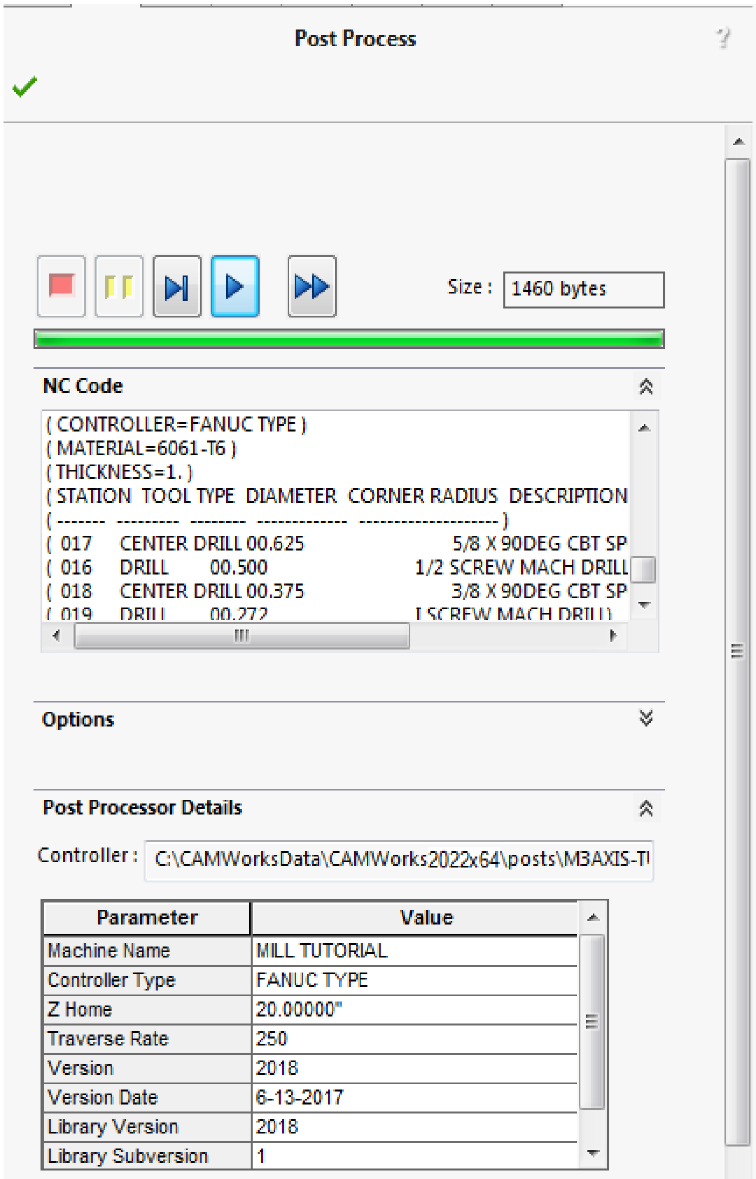
   RMB context menu.
7. In the *Save Post File* dialog box, type a suitable

   file name.
8. In the *Post Process Output* dialog box, click the

   Play button ▶

   NC code will be created for the *Top Part Setup*.
9. Click *OK* ✓ to close the dialog box.
10. Open the saved Post process files. Observe that

    the generated NC code has been saved.



Size : 1460 bytes

NC Code

```
( CONTROLLER=FANUC TYPE )
( MATERIAL=6061-T6 )
( THICKNESS=1. )
( STATION  TOOL TYPE  DIAMETER  CORNER RADIUS  DESCRIPTION
( -------  ---------  --------  -------------  -------------------- )
(  017     CENTER DRILL 00.625                    5/8 X 90DEG CBT SP
(  016     DRILL        00.500                  1/2 SCREW MACH DRILL
(  018     CENTER DRILL 00.375                    3/8 X 90DEG CBT SP
(  019     DRILL        00.272                  I SCREW MACH DRILL)
```

Post Processor Details

Controller : C:\CAMWorksData\CAMWorks2022x64\posts\M3AXIS-TI

Parameter	Value
Machine Name	MILL TUTORIAL
Controller Type	FANUC TYPE
Z Home	20.00000"
Traverse Rate	250
Version	2018
Version Date	6-13-2017
Library Version	2018
Library Subversion	1

Post Process Output Dialog box

2 Axis Mill 4

Topics covered in this tutorial:

- Defining the Stock from a Sketch
- Inserting Open Pocket Feature
- Adjusting Operation Parameters
- Deleting an Operation
- Inserting an Operation
- Defining Program Zero
- Sorting Operations

This tutorial guides you through the steps to machine the part shown below. The concepts given above (under What You'll Learn) will be explained during the execution of the steps required to machine this part.

You are expected to have an understanding of the concepts explained in the previous three tutorials before beginning with this tutorial.

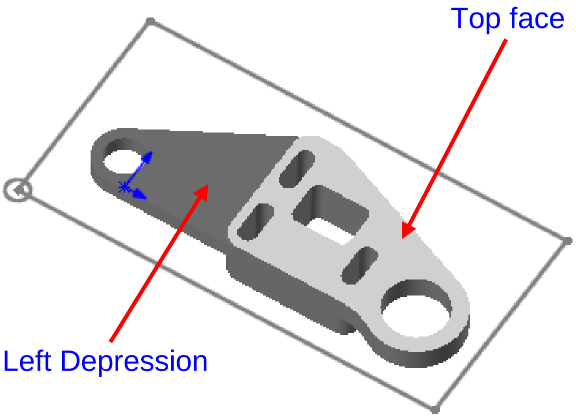
Sequence of machining:

Mill Part Setup1: To machine the top side of the part

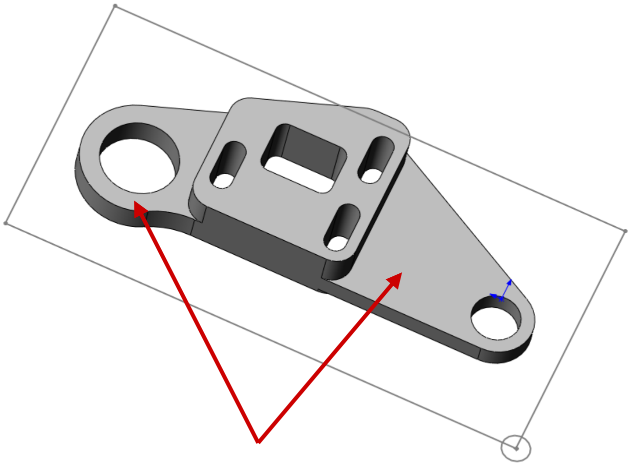
- Drill two holes on each end of part.
- Face top of part.
- Face depression on left side of part.
- Finish profiles the hole on each end of the part.
- Rough and finish rectangular pocket.
- Finish machine three obround holes.
- Rough and finish outside part shape from stock (open pocket).

Mill Part Setup2: To machine the bottom side of the part

- Face the bottom of the part.
- Rough and finish the two depressions on the bottom of the part.



Upper side of the part to be machined



Bottom Face depressions (irregular slots)

Step 1: Opening the Part and Defining the Machine

1. Open the part file **MILL2AX_4.SLDPRT** in the following folder.
   Drive:\CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill
2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill–Inch]* in the Feature tree. The *Machine* dialog box will be displayed.
   - On the Machine tab, *Mill–Inch* is the Active machine.
   - Click the *Tool Crib* tab and make sure *Tool Crib 2 (Inch)* is the active tool crib.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to apply these settings and close the *Machine* dialog box.

Step 2: Defining the Stock from a Sketch

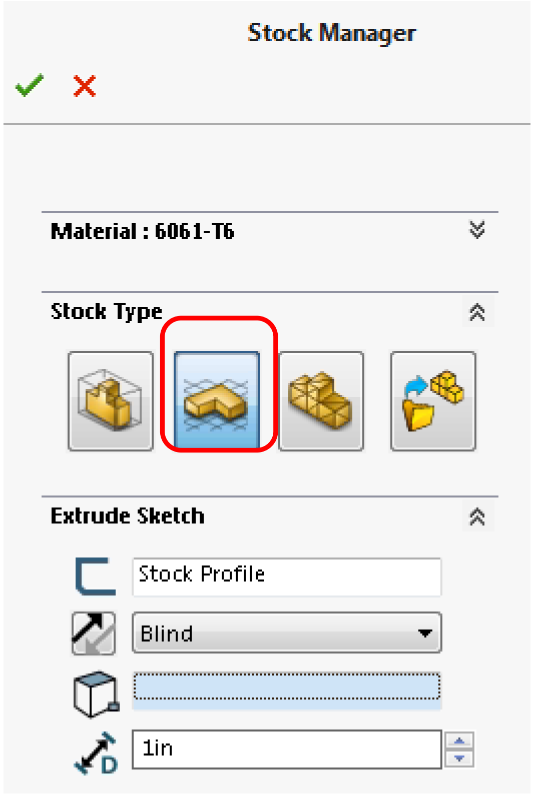
For this part, consider that the stock is a block of material that does not have the same amount of material on all sides. For such cases, CAMWorks has the provision to define a stock using a pre-defined sketch and user-input depth.

In this tutorial, you will define it using a rectangular sketch and a depth. A sketch named *Stock Profile* has already been created in this part. The sketch plane is positioned **0.05in** above the top of the part.

1. Double click *Stock Manager* in the CAMWorks Feature tree to open the *Stock* Manager dialog box.

   CAMWorks automatically calculates the stock size and shape based on the smallest block possible. You can change this definition by specifying a sketch and depth.
2. Under *Stock Type*, select *Extruded Sketch*.
3. Pick the rectangular sketch in the graphics area.

   The sketch will be highlighted.
4. Set the Depth to **1in**.



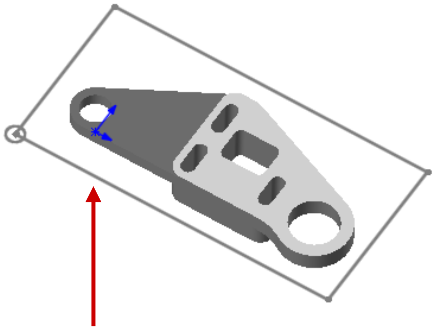
Material : 6061-T6

Stock Profile

Blind

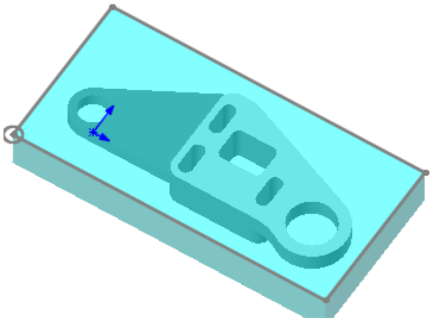
1in

Stock Manager Dialog Box



Select Sketch

5. If the stock doesn’t extrude in the correct direction, click the *Reverse Direction* button.
6. Click *OK* to apply the changes and close the dialog box.
7. Left click on the *Stock Manager* in the Feature tree to see an outline of the stock.

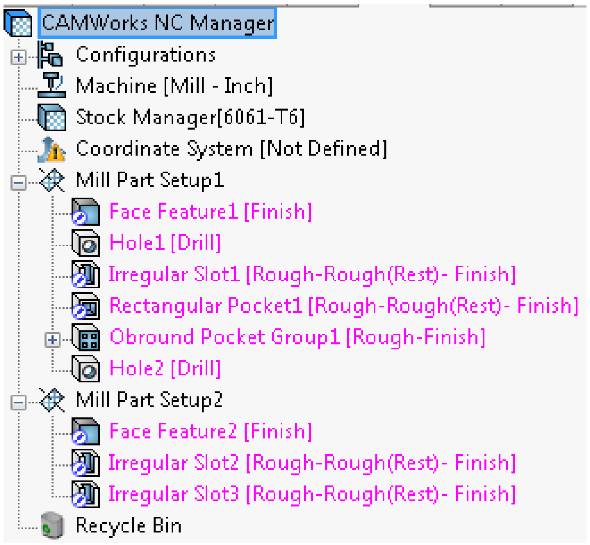


Stock extruded in correct direction

Step 3: Extracting Machinable Features

Settings for Recognizing Face Feature Automatically

1. Click the *Options* button on the CAMWorks Command Manager. The *Options* dialog box will be displayed.
2. Click the *Mill Features* tab in the *Options* dialog box.
3. Ensure that the *Extract Machinable Features Method* is set to *MfgView*.
4. Under *Feature Types*, check the *Face* option.
5. Click *OK* to apply the changes and close the dialog box.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Inch]
- Stock Manager[6061-T6]
- Coordinate System [Not Defined]
- Mill Part Setup1
  - Face Feature1 [Finish]
  - Hole1 [Drill]
  - Irregular Slot1 [Rough-Rough(Rest)- Finish]
  - Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
  - Obround Pocket Group1 [Rough-Finish]
  - Hole2 [Drill]
- Mill Part Setup2
  - Face Feature2 [Finish]
  - Irregular Slot2 [Rough-Rough(Rest)- Finish]
  - Irregular Slot3 [Rough-Rough(Rest)- Finish]
- Recycle Bin

Automatically Recognized Features

Automatic Feature Recognition

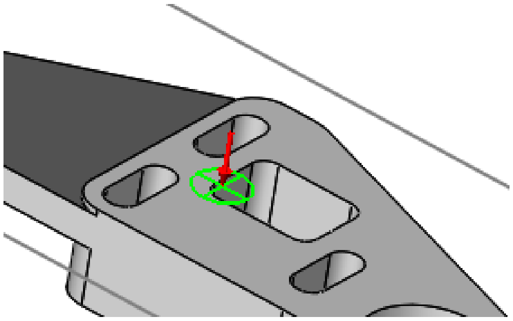
1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   CAMWorks creates the Mill Part Setups necessary to machine all of the machinable features that are recognized *(as defined under Feature Types list in the Mill*

Features Tab of the Options dialog box).

Under *Mill Part Setup1*, the recognized features include the face feature, two holes, the rectangular pocket and the three obround pockets, which are included in *Obround Pocket Group1*.

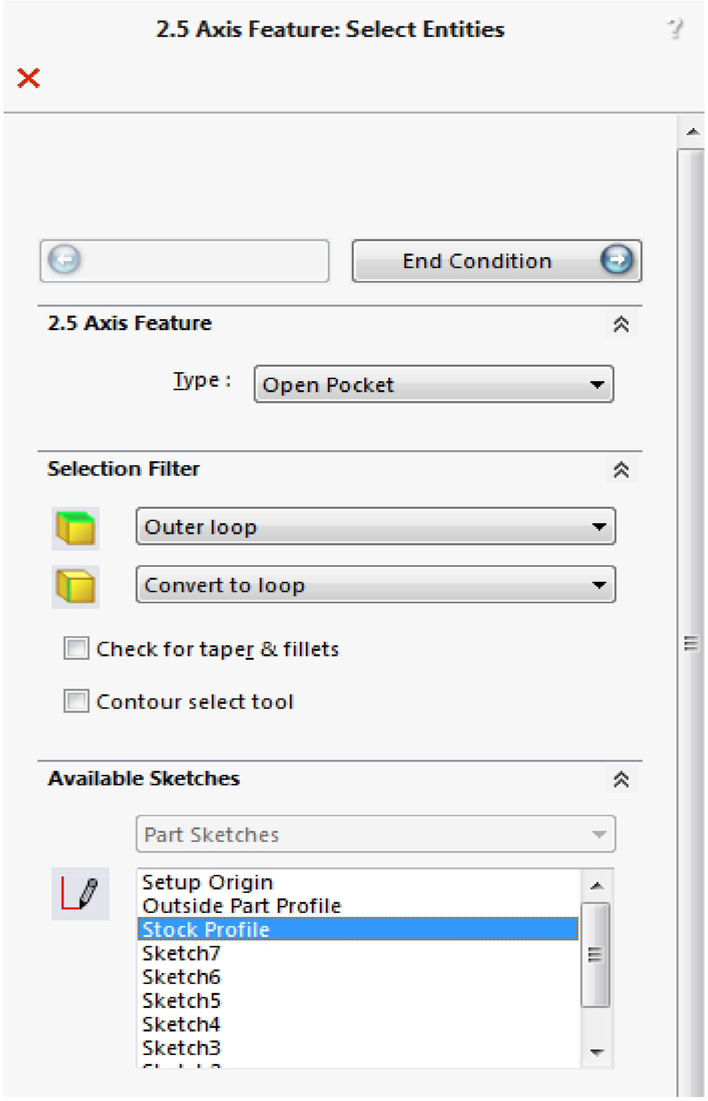
Mill Part Setup2 machines features from the bottom. Under *Mill Part Setup2*, the recognized features include the face feature and the two depressions which are recognized as Irregular slot features.



Automatic Correct direction for machining Hole & Pockets features

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

   All the hole and pocket features go through the part. Hence, AFR may determine the machining direction either from the top or the bottom of the part. If these features are recognized from the bottom direction instead of from the top as shown in the image of the previous page, drag and drop these features from *Mill Part Setup2* to *Mill Part Setup1*.



2.5 Axis Feature: Select Entities

2.5 Axis Feature

Type : Open Pocket

Selection Filter

Outer loop

Convert to loop

☐ Check for taper & fillets

☐ Contour select tool

Available Sketches

Part Sketches

Setup Origin
Outside Part Profile
Stock Profile
Sketch7
Sketch6
Sketch5
Sketch4
Sketch3

Interactively inserting a feature using a sketch

Step 4: Interactively Inserting Features

For this tutorial, you will interactively define an Open Pocket (feature that defines the outside profile of the part so it can be machined from the stock). This feature is created from a sketch.

Interactively inserting an Open Pocket Feature

Need for Open Pocket Feature

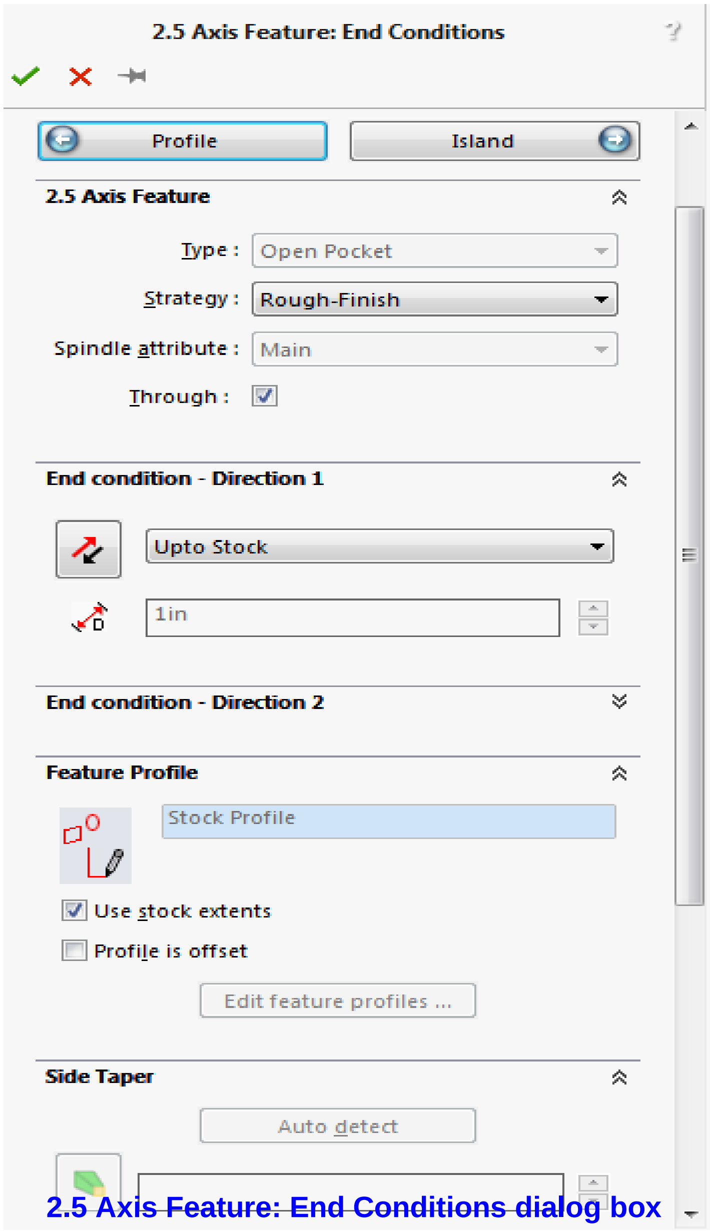
For this tutorial, you will rough the part from the stock, then finish machine the perimeter of the part. In order to do this, you define the stock perimeter as an Open Pocket and the part as an island in this pocket. The interactively inserted 2.5 Axis Open Pocket feature will be used to machine the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

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

Steps to interactively insert Open Pocket Feature:

1. In the Feature Tree, right click *Mill Part Setup1* and select *2.5 Axis Feature* command on the context menu.
2. The *2.5 Axis Feature: Select Entities* dialog box will be displayed. In this dialog box, select *Open Pocket* for the Feature Type.
3. In the *Available sketches* list, select the sketch labeled *Stock Profile*.
4. End Condition Click *End Condition* button.
5. The *2.5 Axis Feature: End Conditions* dialog box will be displayed. In the 2.5 Axis feature group box:
    - Set the *Strategy* set to *Rough-Finish*.
    - Check the *Through* checkbox option.

      With this option checked, the feature will be machined slightly deeper than the stock to remove all material.
6. In the *End condition – Direction 1* group box, click on the dropdown list and set the End condition to *Upto Stock*.
7. In the *Feature Profile* group box, leave the *Use Stock Extents* option checked.



2.5 Axis Feature: End Conditions

Profile
Island
2.5 Axis Feature
End condition - Direction 1
End condition - Direction 2
Feature Profile
Edit feature profiles ...
Side Taper
Auto detect

Type : Open Pocket

Strategy : Rough-Finish

Spindle attribute : Main

Through : ☑

Upto Stock

1in

Stock Profile

☑ Use stock extents

☐ Profile is offset

2.5 Axis Feature: End Conditions dialog box

8. Island ➔ Click *Island* button at the top of the dialog box.

9. The *2.5 Axis Feature: Island Entities* dialog box will be displayed.

   - Click the *Auto detect* button.

     CAMWorks considers a flat face that is normal to the Mill Part Setup as an island. In this part, the top face of the part and the left depression face are both flat and normal to the current Mill Part Setup (*Mill Part Setup1*) and are therefore identified as islands automatically.

     When necessary, you can define islands interactively by clicking the *Add* button in this dialog box and selecting faces, edges, loops and sketches.

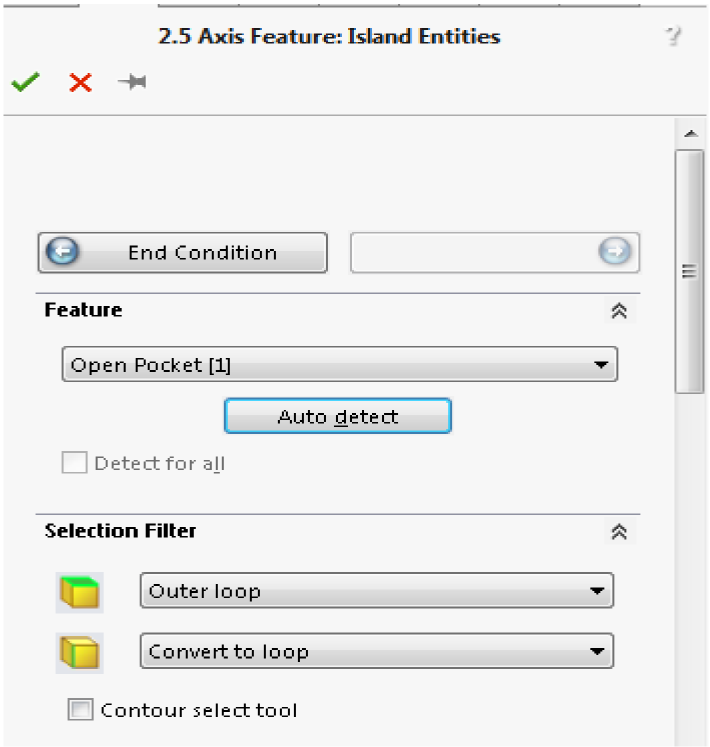
10. ✓ Click *OK* to insert the feature and close the dialog box.

11. ✗ Click *Close* to exit the dialog box.

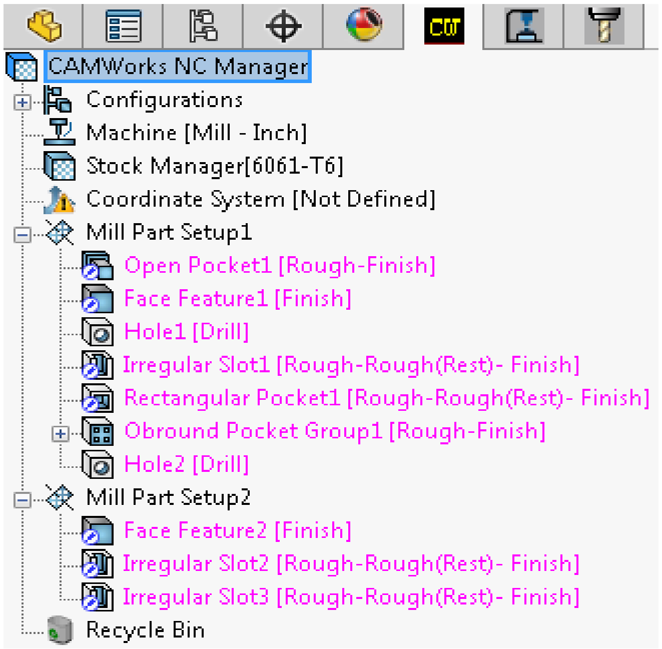
The Open Pocket Feature is now listed at the bottom of *Mill Part Setup1* in the CAMWorks Feature tree.

Use the **drag and drop** functionality to arrange the features in the sequence as shown in the image below.

Operations are generated for features in the same sequence as the features appear in a Mill Part Setup. Rearranging these features will ensure that when operations are generated for these features by executing the *Generate Operation Plan* command, then operations are sequenced in the same order as the features.



2.5 Axis Feature: Island Entities dialog box



CAMWorks NC Manager
Configurations
Machine [Mill - Inch]
Stock Manager[6061-T6]
Coordinate System [Not Defined]
Mill Part Setup1
Open Pocket1 [Rough-Finish]
Face Feature1 [Finish]
Hole1 [Drill]
Irregular Slot1 [Rough-Rough(Rest)- Finish]
Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
Obround Pocket Group1 [Rough-Finish]
Hole2 [Drill]
Mill Part Setup2
Face Feature2 [Finish]
Irregular Slot2 [Rough-Rough(Rest)- Finish]
Irregular Slot3 [Rough-Rough(Rest)- Finish]
Recycle Bin

Arrangement of features in Mill Part Setup 1

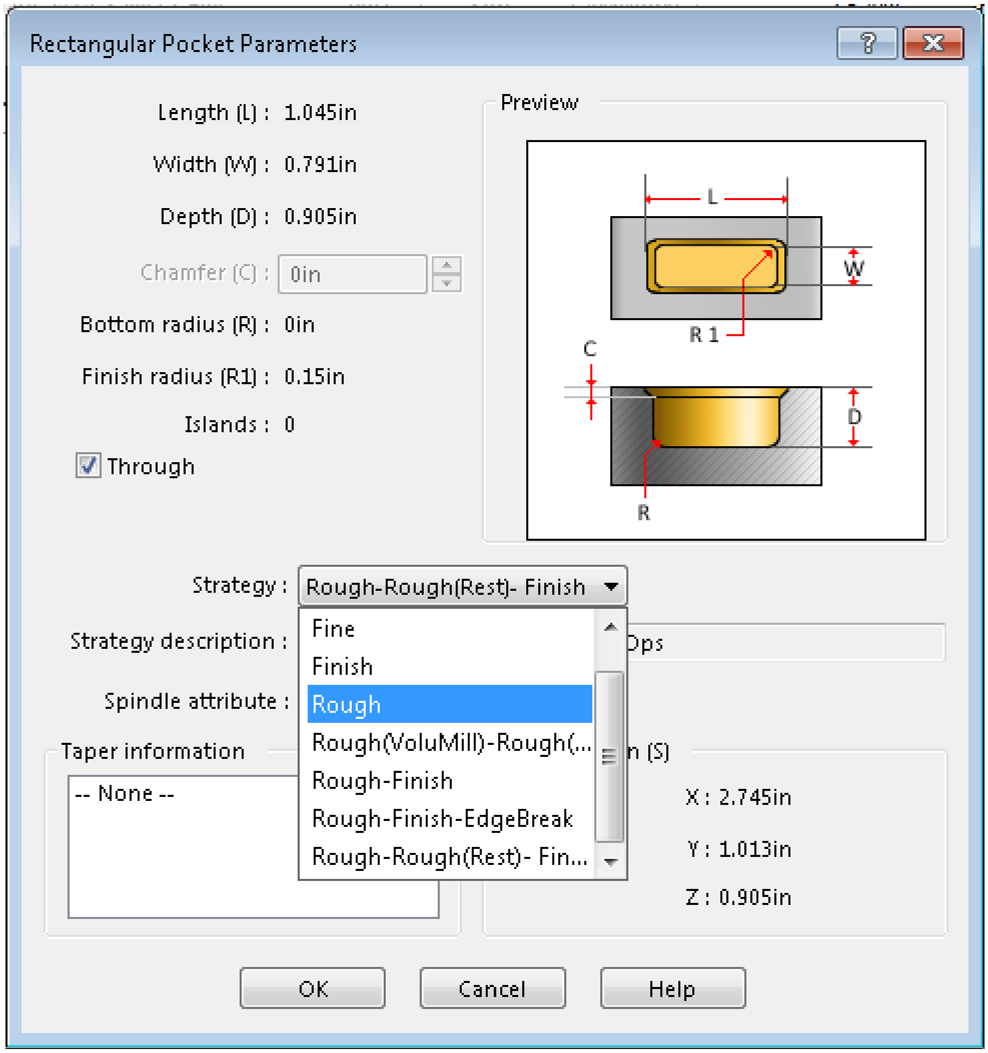
Step 5: Generating Operations

Changing Feature Strategy before Generating Operations

Before the *Operation Plan* is generated, strategies can be assigned to features so as to affect the operations that are created for each feature. In this tutorial, you will change the default strategy of *Rectangular Pocket1*.

1. In the Feature tree, right click *Rectangular Pocket1* and select *Parameters* on the context menu to open the *Rectangular Pocket Parameters* dialog box.
2. In this dialog box, set the Strategy to *Rough*.
3. Click *OK* to apply the changes and close the dialog box.

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


Rectangular Pocket Parameters

Length (L) : 1.045in

Width (W) : 0.791in

Depth (D) : 0.905in

Chamfer (C) : 0in

Bottom radius (R) : 0in

Finish radius (R1) : 0.15in

Islands : 0

☑ Through

Strategy : Rough-Rough(Rest)- Finish

Strategy description : …Ops

Spindle attribute :

Taper information: -- None --

X : 2.745in

Y : 1.013in

Z : 0.905in

Setting the Strategy to ‘Rough’ for Rectangular Pocket Feature

Executing Generate Operation Plan Command

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager. The operations are listed in the Operation tree for both *Mill Part Setup1* and *Mill Part Setup2*.

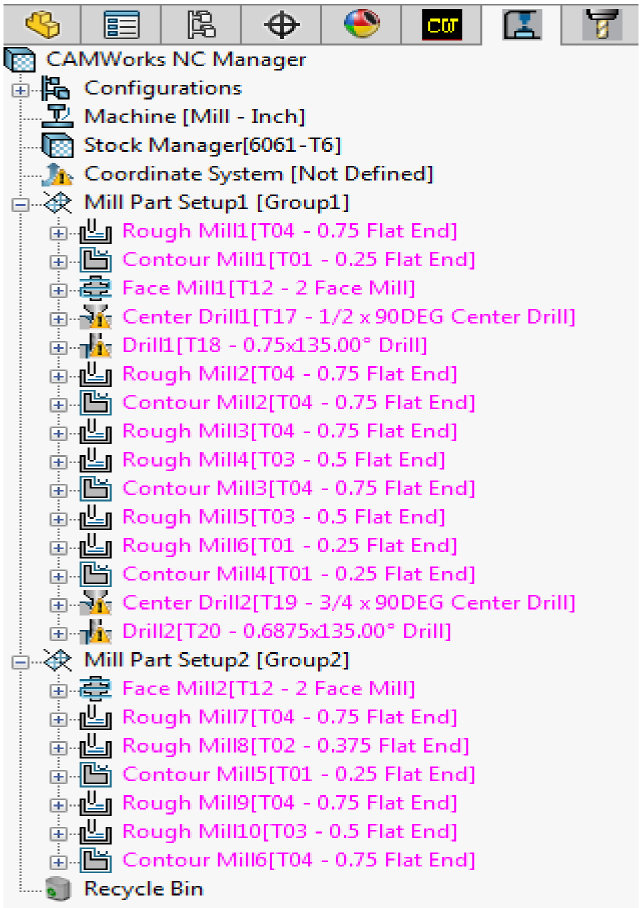
   OR

   Right click *Mill Part Setup1* in the tree and select *Generate Operation Plan* in the context menu.

   Operations will be generated for *Mill Part Setup1* in the CAMWorks Operation tree.

2. CW Switch back to the *CAMWorks Feature tree*.
3. Right click *Mill Part Setup2* and select *Generate Operation Plan* in the context menu.

   Operations will now be generated for *Mill Part Setup2* in the CAMWorks Operation tree.
4. Click on the plus symbol ⊞ next to each operation under *Mill Part Setup1*. This action expands the tree to identify the feature that is machined by the particular operation.
5. Observe that the operations are generated for the features in the same sequence as the features listed in the Feature tree. This is true for both *Mill Part Setup1* and *Mill Part Setup2*.
6. A yellow exclamation mark over certain operations indicates that the tools assigned to those operations were not assigned from the active tool crib. You can right-click on such operations and select *What's wrong?* on the context menu. Click on the *Clear* button on the displayed error message to get rid of the exclamation mark.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Inch]
- Stock Manager[6061-T6]
- Coordinate System [Not Defined]
- Mill Part Setup1 [Group1]
  - Rough Mill1[T04 - 0.75 Flat End]
  - Contour Mill1[T01 - 0.25 Flat End]
  - Face Mill1[T12 - 2 Face Mill]
  - Center Drill1[T17 - 1/2 x 90DEG Center Drill]
  - Drill1[T18 - 0.75x135.00° Drill]
  - Rough Mill2[T04 - 0.75 Flat End]
  - Contour Mill2[T04 - 0.75 Flat End]
  - Rough Mill3[T04 - 0.75 Flat End]
  - Rough Mill4[T03 - 0.5 Flat End]
  - Contour Mill3[T04 - 0.75 Flat End]
  - Rough Mill5[T03 - 0.5 Flat End]
  - Rough Mill6[T01 - 0.25 Flat End]
  - Contour Mill4[T01 - 0.25 Flat End]
  - Center Drill2[T19 - 3/4 x 90DEG Center Drill]
  - Drill2[T20 - 0.6875x135.00° Drill]
- Mill Part Setup2 [Group2]
  - Face Mill2[T12 - 2 Face Mill]
  - Rough Mill7[T04 - 0.75 Flat End]
  - Rough Mill8[T02 - 0.375 Flat End]
  - Contour Mill5[T01 - 0.25 Flat End]
  - Rough Mill9[T04 - 0.75 Flat End]
  - Rough Mill10[T03 - 0.5 Flat End]
  - Contour Mill6[T04 - 0.75 Flat End]
- Recycle Bin

Operations listed in the Feature Tree

Adjusting Operation Parameters

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

In this tutorial, you will edit the Operation Parameters of the following operations:

- For the Contour Mill Operation generated for Obround Pocket Group feature, the tool used to machine the operation will be changed.
- For the Rough Mill Operations generated for the Irregular Slot Features and Obround Pocket Group feature, the tool parameters will be changed.

Expanding and Collapsing Items in the Tree

1. Click the ⊞ symbol next to *Contour Mill4* in the CAMWorks Operation tree.

The tree expands to identify the *Obround Pocket Group1* as the feature for which the operation was generated for.

2. To collapse this item, click on the ⊟ symbol next to *Contour Mill4.*

   OR

   Right click *Contour Mill4* and select *Collapse Items* on the context menu.

   You can use either ⊞ and ⊟ symbols or the *Expand Items* and *Collapse Items* command on the context menus to view items in the trees.

Steps to change the Allowance for an Operation:

1. In the Operation tree, double click *Contour Mill4*.

   OR

   Right click *Contour Mill4* and select *Edit Definition* on the context menu.
2. The *Operation Parameters* dialog box will be displayed.
3. Click the *Contour* tab of the Operation Parameters dialog box.
4. In the *Side parameters* group box, ensure the *Allowance* is set to **0in**.

   Setting the *Allowance* to this value ensures that the pockets are cut to size.
5. Click *OK* to apply the changes and close the *Operation Parameters* dialog box.

Steps to change the tool parameters:

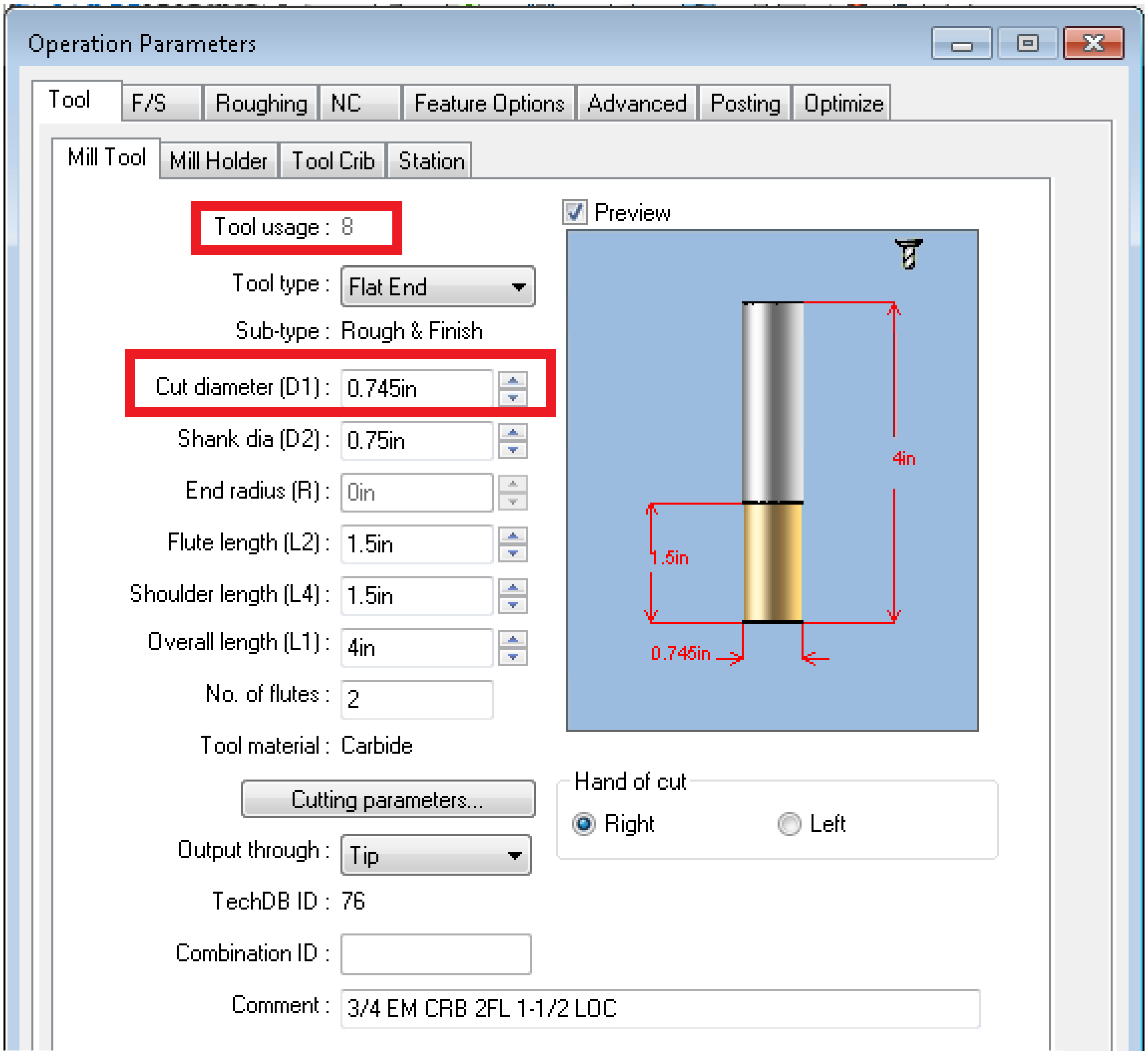
1. In the Operation tree, double click *Rough Mill3*.

   OR

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

   Rough Mill3 operation is used to rough the Irregular Slot feature on the left side of the top face of the part.
2. The *Operation Parameters* dialog box will be displayed. Click on the *Tool* tab.
3. Click on the *Mill Tool* page. It displays the parameters of the selected tool. Observe the *Tool Usage* value. This value indicates the number of operations currently using this tool.

   The Rough Mill operation for Hole Feature under *Mill Part Setup1*, Rough Mill and Contour Mill operations for *Irregular Slot2* feature and Rough Mill operation for *Irregular Slot3* feature under *Mill Part Setup2* also use this tool.
4. In the *Cut Diameter (D1)* field, change the diameter to **0.745in**.
5. Click the *OK* button to apply the changes.



Operation Parameters

Tool usage : 8

Tool type : Flat End

Sub-type : Rough & Finish

Cut diameter (D1) : 0.745in

Shank dia (D2) : 0.75in

End radius (R) : 0in

Flute length (L2) : 1.5in

Shoulder length (L4) : 1.5in

Overall length (L1) : 4in

No. of flutes : 2

Tool material : Carbide

Output through : Tip

TechDB ID : 76

Combination ID :

Comment : 3/4 EM CRB 2FL 1-1/2 LOC

☑ Preview

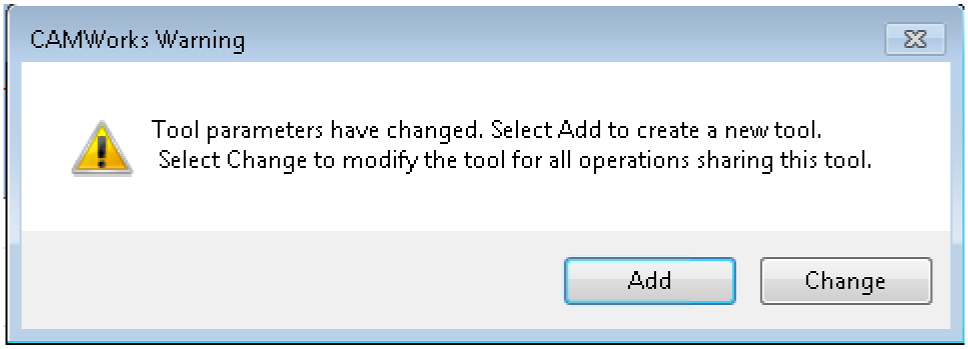
Hand of cut: ● Right ○ Left

Changing Mill Tool Parameters

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

7. Click *Change* on the message.

   Change: When you click *Change* within this dialog box, then the changes made to the tool parameters



CAMWorks Warning

Tool parameters have changed. Select Add to create a new tool.
Select Change to modify the tool for all operations sharing this tool.

CAMWorks Warning Message

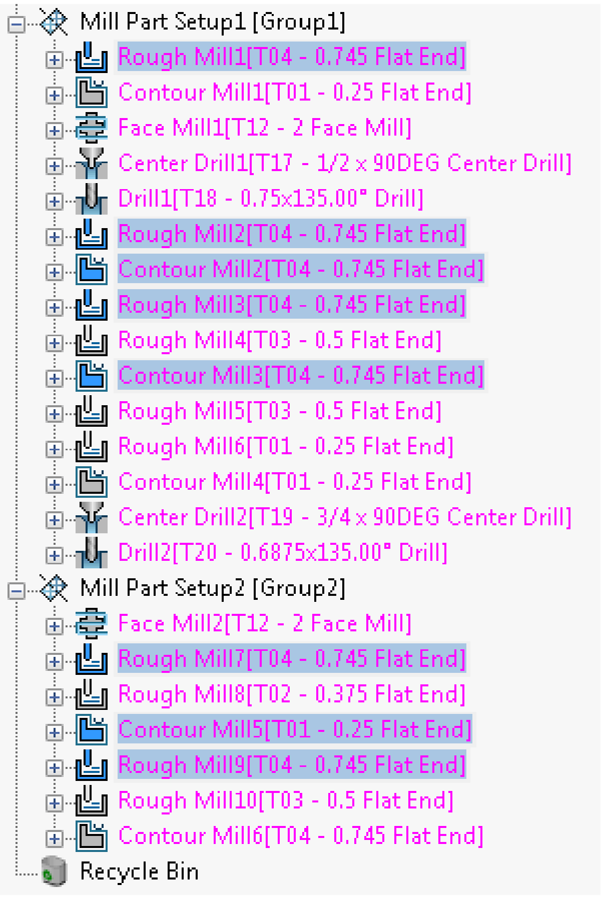
will affect all other operations sharing this tool.

Add: If you click *Add* within the warning message dialog box, then CAMWorks creates a new tool with the changed tool parameters and lists this tool in the Active Tool crib. This action ensures that the changes made to the tool parameters will affect only the current operation and none of the other operations which share the same tool.

Note that irrespective of which option you choose, the changes made to the tool parameters are applicable only for machining of the current part. The changes made to the tool parameters are not saved to the Technology Database.

To save the changes made to the tool parameters in the Technology Database, you have to execute the *Save Operation Plan* command. This functionality is explained in a later tutorial.

8. In the Operation tree, observe the Rough and Contour 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 operations.



Mill Part Setup1 [Group1]
Rough Mill1[T04 - 0.745 Flat End]
Contour Mill1[T01 - 0.25 Flat End]
Face Mill1[T12 - 2 Face Mill]
Center Drill1[T17 - 1/2 x 90DEG Center Drill]
Drill1[T18 - 0.75x135.00° Drill]
Rough Mill2[T04 - 0.745 Flat End]
Contour Mill2[T04 - 0.745 Flat End]
Rough Mill3[T04 - 0.745 Flat End]
Rough Mill4[T03 - 0.5 Flat End]
Contour Mill3[T04 - 0.745 Flat End]
Rough Mill5[T03 - 0.5 Flat End]
Rough Mill6[T01 - 0.25 Flat End]
Contour Mill4[T01 - 0.25 Flat End]
Center Drill2[T19 - 3/4 x 90DEG Center Drill]
Drill2[T20 - 0.6875x135.00° Drill]
Mill Part Setup2 [Group2]
Face Mill2[T12 - 2 Face Mill]
Rough Mill7[T04 - 0.745 Flat End]
Rough Mill8[T02 - 0.375 Flat End]
Contour Mill5[T01 - 0.25 Flat End]
Rough Mill9[T04 - 0.745 Flat End]
Rough Mill10[T03 - 0.5 Flat End]
Contour Mill6[T04 - 0.745 Flat End]
Recycle Bin

Operations sharing the tool

Deleting an Operation

In some cases, it may be desirable to delete operations that are created automatically or to replace automatically created operations with interactively inserted operations. In this tutorial, two operations were created for the *Obround Pocket Group1* feature - Rough Mill operation (*Rough Mill6)* and Contour Mill operation (*Contour Mill4*). The tool that is selected in the Contour Mill operation for this feature can machine the feature completely. Hence, the Rough Mill operation can be deleted.

Steps to delete an operation:

1. In the CAMWorks Operation tree, right click *Rough Mill6* (the roughing operation for the *Obround Pocket Group1*) and select *Delete* on the context menu.
2. CAMWorks will display a warning message asking whether you are sure that you want to delete the operation and all dependent items. Click *Yes* to confirm to deletion.

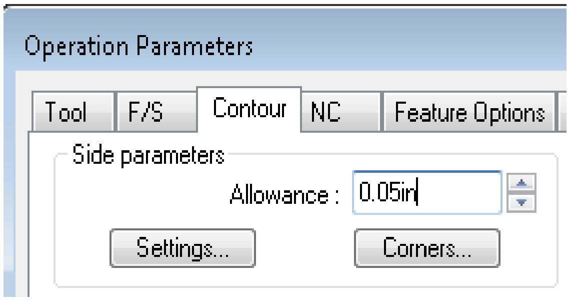
   The operation will be removed from the Operation tree and placed in the Recycle bin .

Inserting an Operation

The obround pocket group feature is now machined using a single Contour Mill operation. To give a better finish, you can interactively insert an additional Contour Mill operation the makes use of a tool with smaller cut diameter which will finish machine the obround pocket features.

In this tutorial, you will interactively insert an additional Contour Mill operation for *Obround Pocket Group1*. This operation will be inserted after the *Contour Mill4* operation.

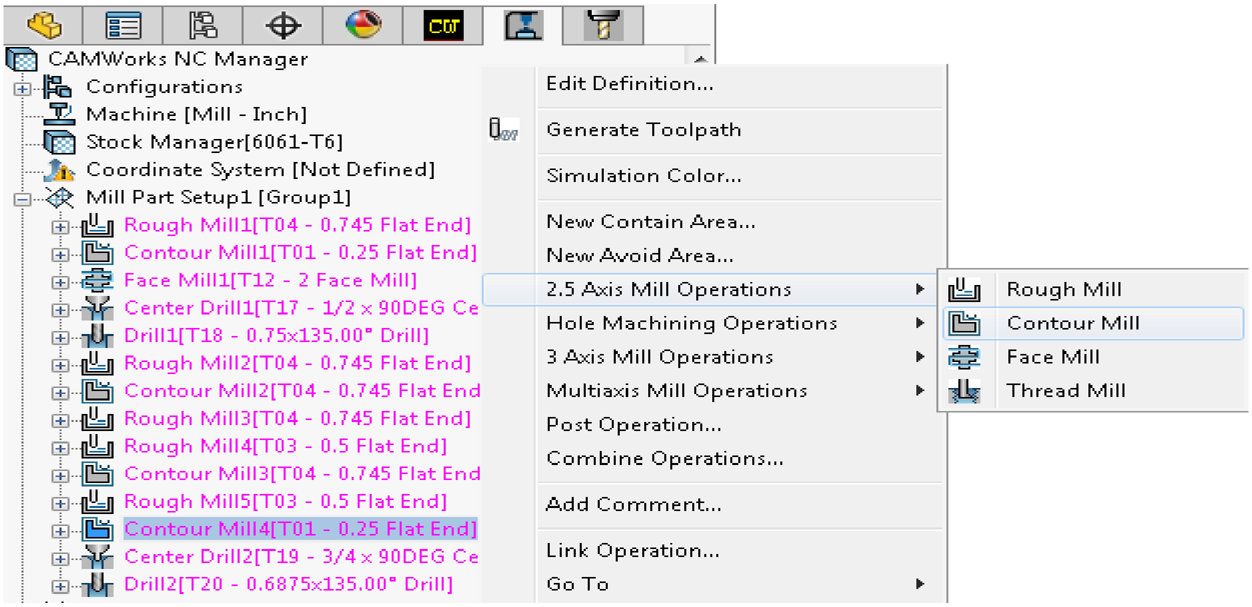
Steps to interactively insert an operation:



Allowance : 0.05in

Setting Allowance value

1. In the Operation tree, right click *Contour Mill4* operation and select *Edit Definition* to open the *Operation Parameters* dialog box.
2. Click on the *Contour* Tab and in the *Side Parameters* group box, set the allowance to **0.05in**.
3. Click *OK* to apply the changes and close the dialog box.
4. The new Contour Mill operation is to be inserted after *Contour Mill4* operation. Right click *Contour Mill4* in the CAMWorks Operation tree and select *New 2.5 Axis Mill Operations >>Contour Mill*.

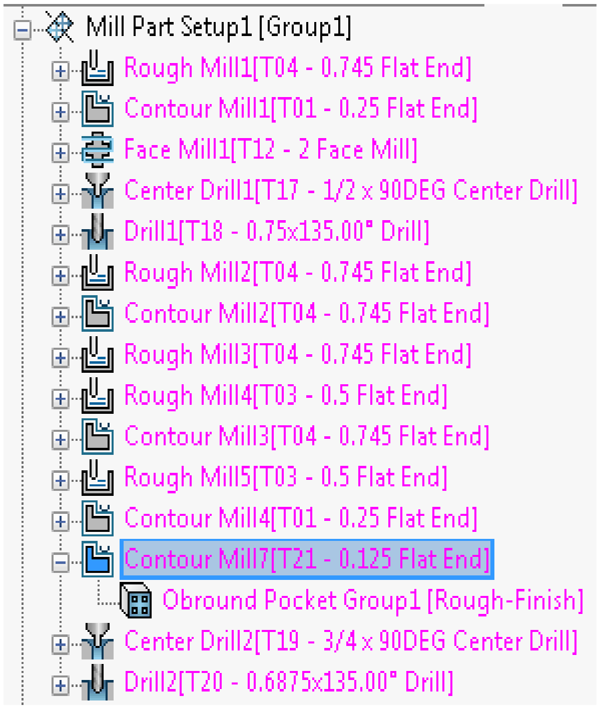

CAMWorks NC Manager
Configurations
Machine [Mill - Inch]
Stock Manager[6061-T6]
Coordinate System [Not Defined]
Mill Part Setup1 [Group1]
Rough Mill1[T04 - 0.745 Flat End]
Contour Mill1[T01 - 0.25 Flat End]
Face Mill1[T12 - 2 Face Mill]
Center Drill1[T17 - 1/2 x 90DEG Ce
Drill1[T18 - 0.75x135.00° Drill]
Rough Mill2[T04 - 0.745 Flat End]
Contour Mill2[T04 - 0.745 Flat End
Rough Mill3[T04 - 0.745 Flat End]
Rough Mill4[T03 - 0.5 Flat End]
Contour Mill3[T04 - 0.745 Flat End
Rough Mill5[T03 - 0.5 Flat End]
Contour Mill4[T01 - 0.25 Flat End]
Center Drill2[T19 - 3/4 x 90DEG Ce
Drill2[T20 - 0.6875x135.00° Drill]
Edit Definition...
Generate Toolpath
Simulation Color...
New Contain Area...
New Avoid Area...
2.5 Axis Mill Operations
Hole Machining Operations
3 Axis Mill Operations
Multiaxis Mill Operations
Post Operation...
Combine Operations...
Add Comment...
Link Operation...
Go To
Rough Mill
Contour Mill
Face Mill
Thread Mill

Selecting command to New a 2.5 Axis Operation interactively

The *New Operation: Contour mill dialog box* will be displayed.

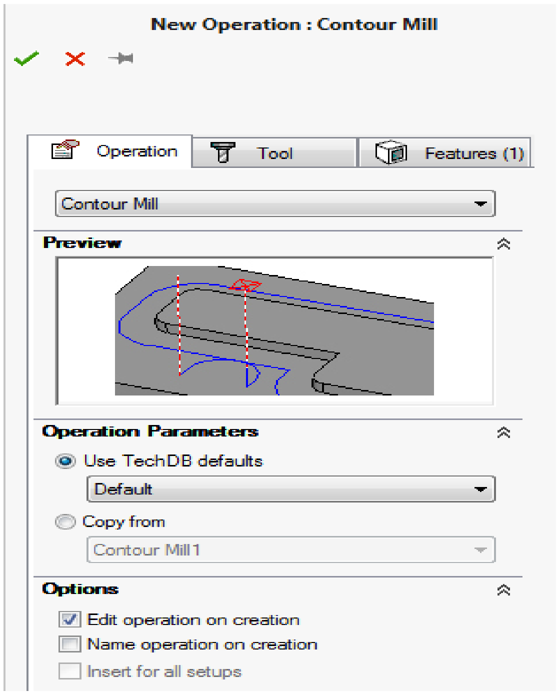
5. Select *Obround Pocket Group1* in the Features list located at the center of the dialog box.

6. In the *Tool* group box, observe that a **0.25in** Flat End mill tool has been assigned for this operation. Since this operation finish machines the feature, a tool with lower diameter needs to be selected. In this tutorial, we will assign a **0.125in** tool.
   - In the Tool dropdown list, select *Tool crib*.
   - In the Tool list, observe that there is no Flat End tool with **0.125in** tool diameter.
   - Click on the *Add New* button.
   - The Tool Select Filter dialog box will be displayed.
     - Select *Flat End* for Tool Type.
     - Select the *Diameter* checkbox in the *Filter by* group box.
     - Enter the lower diameter ranger as **0in** and higher range as **0.25in**.
     - From the list of displayed tools, highlight a tool with **0.125in** diameter and click the *OK* button.
7. In the *Options* group box at the bottom of the dialog box, make sure the *Edit operation on creation* option is checked.
8. ✓ Click *OK* to insert the operation. The *Operation Parameters* dialog box for the newly inserted operation will be displayed.



Mill Part Setup1 [Group1]
Rough Mill1[T04 - 0.745 Flat End]
Contour Mill1[T01 - 0.25 Flat End]
Face Mill1[T12 - 2 Face Mill]
Center Drill1[T17 - 1/2 x 90DEG Center Drill]
Drill1[T18 - 0.75x135.00° Drill]
Rough Mill2[T04 - 0.745 Flat End]
Contour Mill2[T04 - 0.745 Flat End]
Rough Mill3[T04 - 0.745 Flat End]
Rough Mill4[T03 - 0.5 Flat End]
Contour Mill3[T04 - 0.745 Flat End]
Rough Mill5[T03 - 0.5 Flat End]
Contour Mill4[T01 - 0.25 Flat End]
Contour Mill7[T21 - 0.125 Flat End]
Obround Pocket Group1 [Rough-Finish]
Center Drill2[T19 - 3/4 x 90DEG Center Drill]
Drill2[T20 - 0.6875x135.00° Drill]

Contour Mill Operation inserted



New Operation : Contour Mill

Contour Mill

Operation Parameters

● Use TechDB defaults

Default

○ Copy from

Contour Mill1

Options

☑ Edit operation on creation

☐ Name operation on creation

☐ Insert for all setups

Interactive insertion of Operation

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

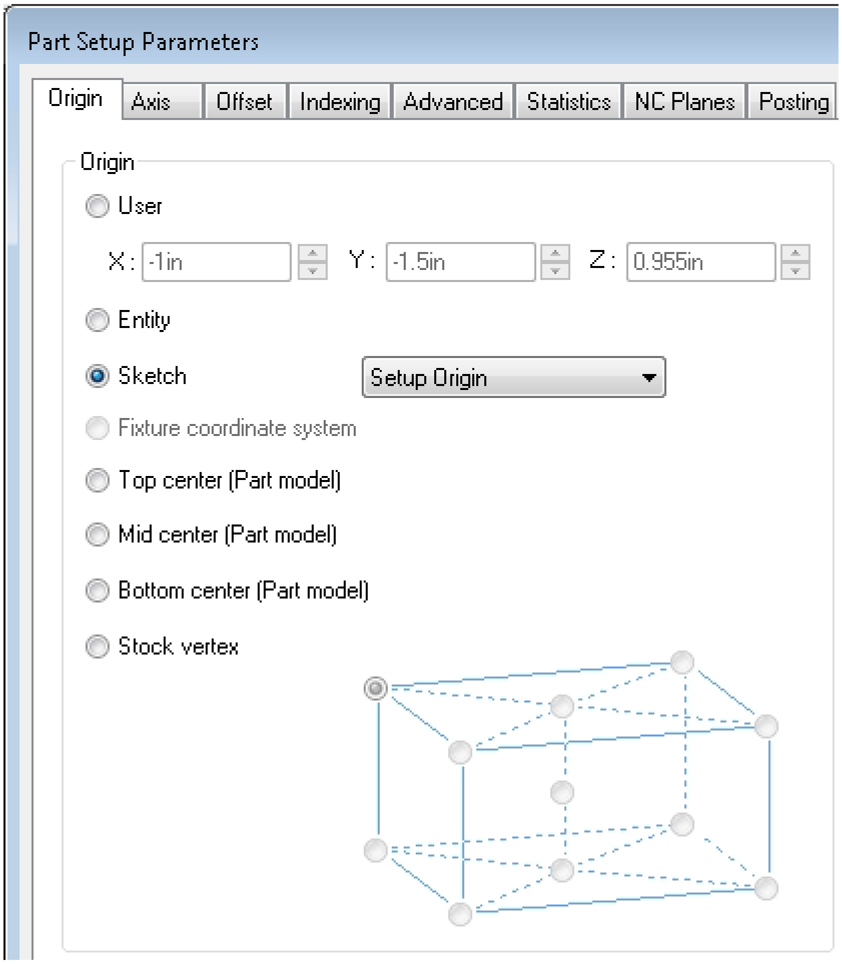
   Under *Mill Part Setup1*, a *Contour Mill17* operation is added after *Contour Mill4*.

Step 6: Defining Program Zero

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

In this tutorial, you will change the default Program Zero and use a sketch containing a circle to assign the new Program Zero.

1. Double-click *Mill Part Setup1* in the Operation tree to open the *Part Setup Parameters* dialog box.



Part Setup Parameters

Origin

○ User

X: -1in Y: -1.5in Z: 0.955in

○ Entity

● Sketch Setup Origin

○ Fixture coordinate system

○ Top center (Part model)

○ Mid center (Part model)

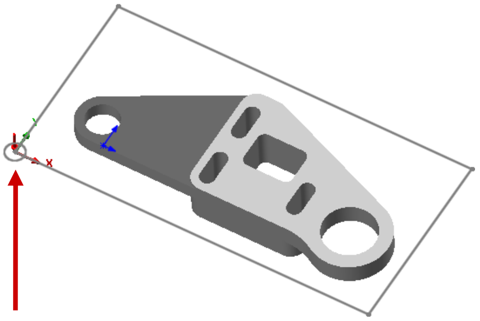
○ Bottom center (Part model)

○ Stock vertex

Using a sketch containing a single circle to assign Program Zero

2. On the *Origin* tab, select the *Sketch* option and then select *Setup Origin* for the sketch dropdown list.
3. Click *OK* to apply the changes made and close the dialog box.

   The circle in the lower left corner of the stock is now the Mill Part Setup origin.



Centre of circle assigned as Program Zero

Did You Know ...

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

Step 7: Generate and Simulate Toolpaths

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

   OR

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

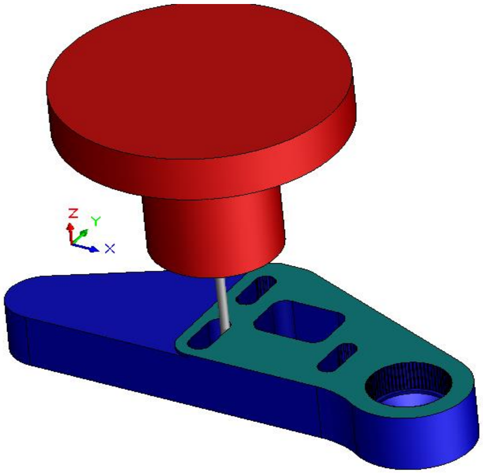
   The font color of all the operations listed in the Operation tree changes to black indicating that toolpaths that were generated for all operations.

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

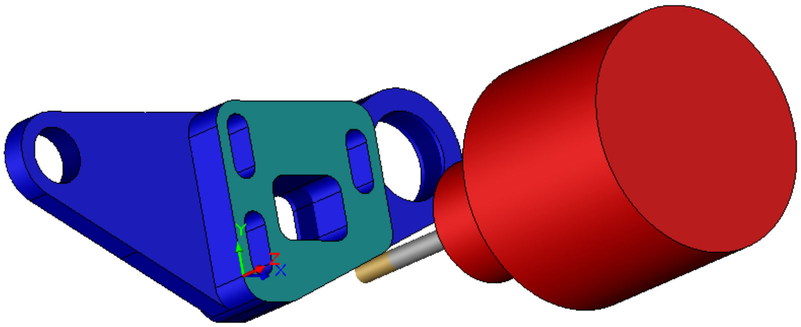
   OR

   Right click *Mill Part Setup1* in the CAMWorks Operation tree and select *Simulate Toolpath* on the context menu.

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



Toolpath Simulation of Mill Part Setup1 operations



Toolpath Simulation of Mill Part Setup2 operations

Step 8: Sorting Operations

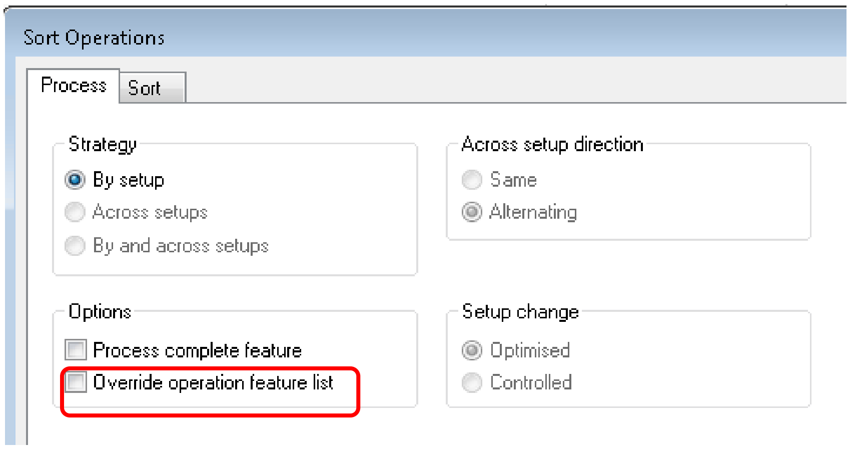
When the *Generate Operation Plan* command is executed, operations are created for each feature in the Feature tree in the order listed. As shown above, operations can also be inserted manually and deleted. After making these changes, it is possible that the machining order is not in the most efficient sequence. For example, you may want all of the roughing done before the contouring, and the hole making operations to be done last. CAMWorks allows automatic and interactive control over changing the machining sequence. Using drag and drop, operations can be reordered into an optimized sequence. However, this could become time consuming for a part that includes dozens of operations. To make the ordering task easier, CAMWorks provides the *Sort Operations* command that sorts operations based on their type, depth and tool. The *Sort Operations* command provides an automatic approach.

1. Right click *Mill Part Setup1* and select *Sort Operations* from the context menu.

   The *Sort Operations* dialog box will be displayed.

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

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



Strategy
- ● By setup
- ○ Across setups
- ○ By and across setups

Across setup direction
- ○ Same
- ● Alternating

Options
- ☐ Process complete feature
- ☐ Override operation feature list

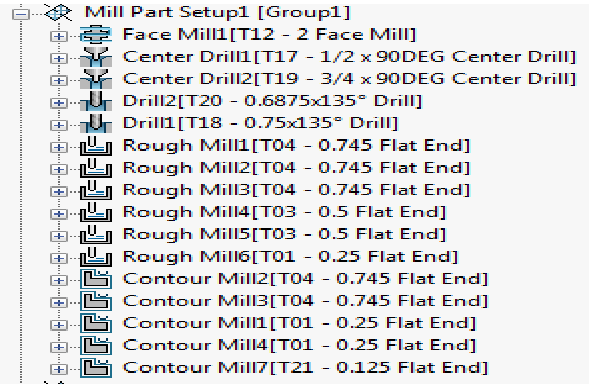
Setup change
- ● Optimised
- ○ Controlled

Process Tab of Sort Operation dialog box

3. Click on the *Sort* tab.

   This tab shows the order in which the operations will be sorted. Operations can be sorted based either on the *Operation Type or Tool*. More information on sorting operations can be found in the CAMWorks context-based help.

4. In the *Sort by* dropdown list, select *Operation Type*. The operation listed in the Operation tree will be machined in order of their types. You can use the drag and drop



Mill Part Setup1 [Group1]
- Face Mill1[T12 - 2 Face Mill]
- Center Drill1[T17 - 1/2 x 90DEG Center Drill]
- Center Drill2[T19 - 3/4 x 90DEG Center Drill]
- Drill2[T20 - 0.6875x135° Drill]
- Drill1[T18 - 0.75x135° Drill]
- Rough Mill1[T04 - 0.745 Flat End]
- Rough Mill2[T04 - 0.745 Flat End]
- Rough Mill3[T04 - 0.745 Flat End]
- Rough Mill4[T03 - 0.5 Flat End]
- Rough Mill5[T03 - 0.5 Flat End]
- Rough Mill6[T01 - 0.25 Flat End]
- Contour Mill2[T04 - 0.745 Flat End]
- Contour Mill3[T04 - 0.745 Flat End]
- Contour Mill1[T01 - 0.25 Flat End]
- Contour Mill4[T01 - 0.25 Flat End]
- Contour Mill7[T21 - 0.125 Flat End]

Operations of Mill Part Setup1 after sorting

function to sort the order of operation types listed in the *Sort by* group box.

5. In the *Then by* dropdown list, select *Tool*. All the tools used by the various operations in the selected Mill Part Setup are listed.

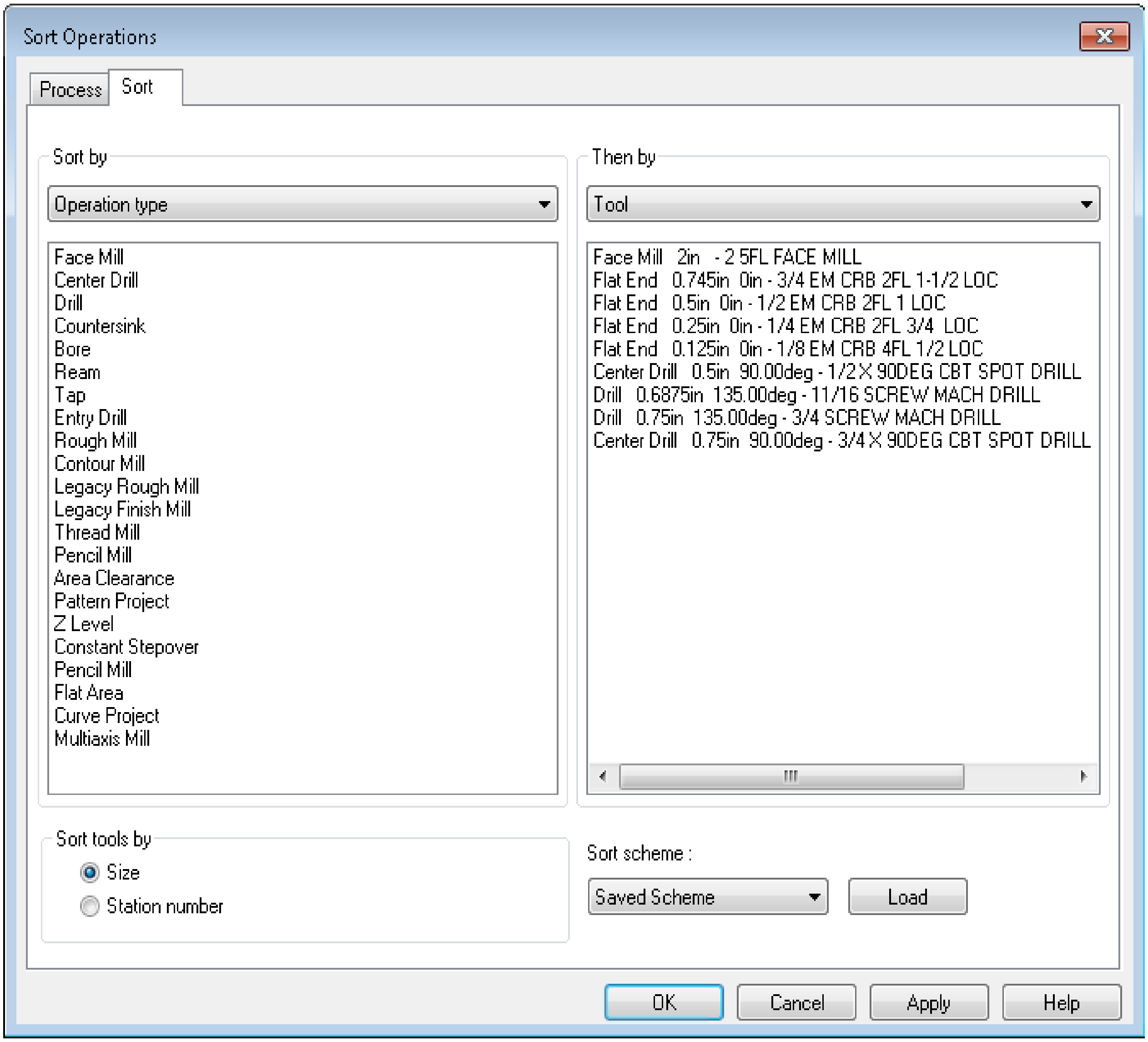
   When multiple operations of the same operation type have to machined, you can further sort the operations based on the tool used by the operations. Tools can be sorted either based on *Size* or *Station number*.

6. In the *Sort Tools by* group box, select the *Size* option.

   When operations are sorted by Tool size, the Mill operations are ordered on the basis of their tool size from largest to smallest. For single point operations, the tool order is from the smallest to the largest diameter.

7. Click the *Apply* button.

   This action will sort the operations in the Operation tree according the specified operation sequence. Within each type of operation, the operations with the highest Z face will be processed first. If you do not like the order, you can click the *Undo* button and change the order or any of the options, then click *Apply* again to see the results.



Sort by: Operation type

Face Mill
Center Drill
Drill
Countersink
Bore
Ream
Tap
Entry Drill
Rough Mill
Contour Mill
Legacy Rough Mill
Legacy Finish Mill
Thread Mill
Pencil Mill
Area Clearance
Pattern Project
Z Level
Constant Stepover
Pencil Mill
Flat Area
Curve Project
Multiaxis Mill

Then by: Tool

Face Mill 2in - 2 5FL FACE MILL
Flat End 0.745in 0in - 3/4 EM CRB 2FL 1-1/2 LOC
Flat End 0.5in 0in - 1/2 EM CRB 2FL 1 LOC
Flat End 0.25in 0in - 1/4 EM CRB 2FL 3/4 LOC
Flat End 0.125in 0in - 1/8 EM CRB 4FL 1/2 LOC
Center Drill 0.5in 90.00deg - 1/2 X 90DEG CBT SPOT DRILL
Drill 0.6875in 135.00deg - 11/16 SCREW MACH DRILL
Drill 0.75in 135.00deg - 3/4 SCREW MACH DRILL
Center Drill 0.75in 90.00deg - 3/4 X 90DEG CBT SPOT DRILL

Sort tools by
● Size
○ Station number

Sort scheme : Saved Scheme

Sort Tab of Sort Operations dialog box

8. Click the *OK* button to commit the changes and close the dialog box.

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

Step 9: Simulate Toolpath and Post Process

1. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
2. Run the simulation and note the changes.
3. Save the part.

4. Right click *Mill Part Setup1* and select *Post Process* on the context menu.
   - The *Post Output File* dialog box will be displayed.
   - If you are running CAMWorks in Demo mode, this dialog box will not be displayed. Post processing functions works only when you have a valid license for CAMWorks.
5. Click *Save* to use the default name for the NC program file.

   The *Post Process Output* dialog box will be displayed.
6. ▶ Click the *Run* button.

   CAMWorks generates the NC program.
7. ✓ Click *OK* to close the *Post Process Output* dialog box.

Chapter 2: Learning More 2 Axis Mill

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

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

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

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

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

2 Axis Mill 5

Topics covered in this tutorial:

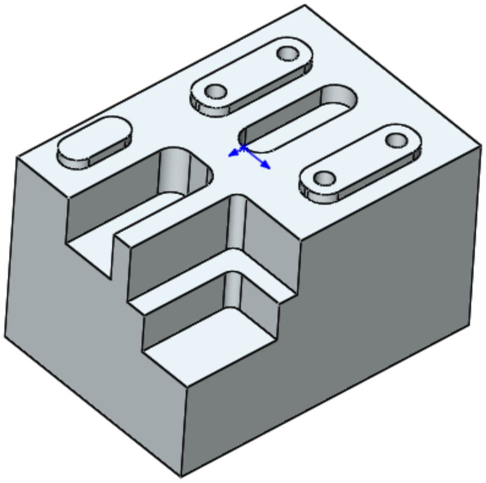
- Setting Strategies and Generating Operations
- Sorting and Sequencing Operations
- Machining Island Tops
- Changing the Mill Part Setup Origin
- Combining Operations

Opening the Part and Extracting Machinable Features by AFR

1. Open the part file **MILL2AX_5.SLDPRT** in the following folder.

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

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



MILL2AX_5.SLDPRT

2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill–Metric]* in the Feature tree to open the *Machine* dialog box.
   - On the Machine tab, ensure *Mill–Metric* is the Active machine.
   - Click the *Tool Crib* tab and make sure *Tool Crib 2 (Metric)* is the Active tool set.
   - Click the *Post Processor* tab; make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.

Did You Know ...

In both the Feature and Operation trees, instead of right clicking items and selecting Edit Definition, you can double-click the item to open the dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.

4. Double click *Stock Manager* in the tree to open the *Stock Manager* dialog box.
   - Change the *Bounding box offsets* for X, Y and Z to a uniform **1mm** in all directions.
   - Leave the *Material* set to default.
   - Click *OK* to apply the changes and close the dialog box.
5. Click the *Options* button on the CAMWorks Command Manager.

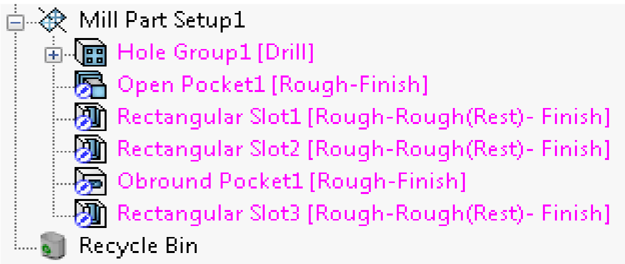
- Click the *Mill Features* tab.
- Ensure that the *Extract Machinable Features Method* is set to *MfgView*.
- In the *Feature types* section, remove the check mark from the *Boss* option.
- Make sure the *Face* option is **<u>not</u>** checked.
- Click *OK* to apply the changes and close the dialog box.

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

   OR

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

   Automatic Feature Recognition analyzes the model and extracts the features to machine. The Feature tree lists the features that were found.



Mill Part Setup1
Hole Group1 [Drill]
Open Pocket1 [Rough-Finish]
Rectangular Slot1 [Rough-Rough(Rest)- Finish]
Rectangular Slot2 [Rough-Rough(Rest)- Finish]
Obround Pocket1 [Rough-Finish]
Rectangular Slot3 [Rough-Rough(Rest)- Finish]
Recycle Bin

Features recognized by AFR

- An Open Pocket feature was created with the three Obround Bosses as islands.
- The obround pocket on the top face was recognized.
- The four holes in the boss features were extracted as a group feature since they were identical in size.
- The three Rectangular Corner Slots were not extracted as a Group Feature because they are of different sizes.

Did You Know ...

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

Setting Strategies and Generating Operations

1. Double click *Hole Group1* in the CAMWorks Feature tree.
2. In the displayed *Hole Parameters* dialog box, change the Strategy to *Ream*.
3. Click *OK* to apply the changes and close the dialog box.
4. Double-click *Open Pocket1* in the CAMWorks Feature tree.
5. In the displayed *2.5 Axis Feature: Select Entities* dialog box, click on the *Edit Feature* button. This activates the *End Condition* button.
6. End Condition Click on the *End Condition* button.
7. The *2.5 Axis Feature: End Condition* dialog box will be displayed. In the *Strategy*.

8. dropdown list within the *2.5 Axis Feature* group box, select the *Rough-Finish* Strategy.

   This strategy has been programmed to generated a Rough and Contour Mill operation for the selected feature.

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

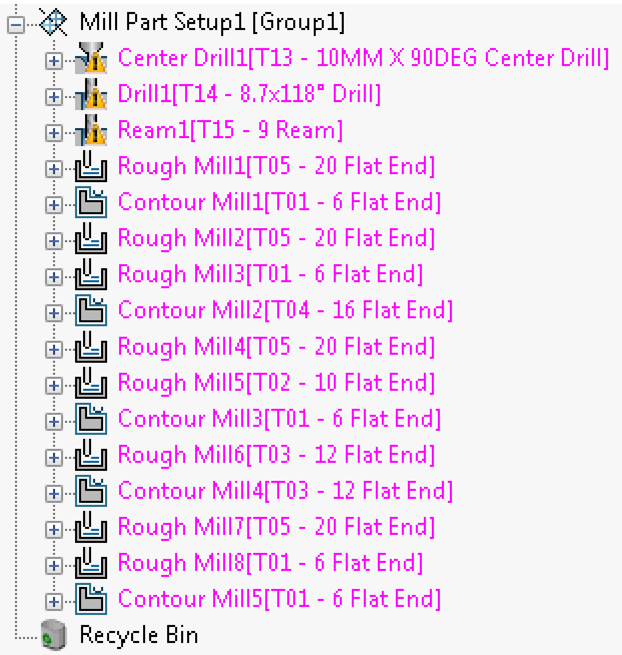
10. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

    OR

    Right click *Mill Part Setup1* in the Feature tree and select *Generate Operation Plan* command on the context menu.

    CAMWorks creates the list of operations to machine the part. The operations for the features are sequenced in order the same order as the features listed in the Feature tree.

11. Some of the operations will have a yellow exclamation mark over them indicating that the tool assigned to those operations are not from the Tool crib. RMB on such operation and select *What's Wrong?* on the context menu. Click the *Clear* button in the displayed *Error* message to get rid of the exclamation mark.



Mill Part Setup1 [Group1]
- Center Drill1[T13 - 10MM X 90DEG Center Drill]
- Drill1[T14 - 8.7x118° Drill]
- Ream1[T15 - 9 Ream]
- Rough Mill1[T05 - 20 Flat End]
- Contour Mill1[T01 - 6 Flat End]
- Rough Mill2[T05 - 20 Flat End]
- Rough Mill3[T01 - 6 Flat End]
- Contour Mill2[T04 - 16 Flat End]
- Rough Mill4[T05 - 20 Flat End]
- Rough Mill5[T02 - 10 Flat End]
- Contour Mill3[T01 - 6 Flat End]
- Rough Mill6[T03 - 12 Flat End]
- Contour Mill4[T03 - 12 Flat End]
- Rough Mill7[T05 - 20 Flat End]
- Rough Mill8[T01 - 6 Flat End]
- Contour Mill5[T01 - 6 Flat End]
- Recycle Bin

Operations generated using GOP command

Sorting and Sequencing Operations

The machining sequence is based on the order of the operations in the tree. The Rough Mill and Contour Mill operations for the Open Pocket are listed after the hole operations. You may not want this order as it isn't very efficient.

In this tutorial, you will sequence the operations such that all of the roughing is done before the contouring, and the hole making operations to be done last.

Using the drag and drop functionality to reorder the operations is suitable only when a few changes need to be made. However, when making numerous changes to the order of the tree, the *Sort Operations* command provides an automatic approach.

In this tutorial, the machining sequence will be changed using the *Sort Operations* command.

Steps to reorder operations using Sort Operations Command:

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

   The *Sort Operations* dialog box will be displayed.

2. In the *Process* tab, remove the check mark in the *Process complete feature* checkbox in the *Options* group box.

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

3. Click on the *Sort* tab. Select Sort by *Operation Type* in the dropdown list.

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

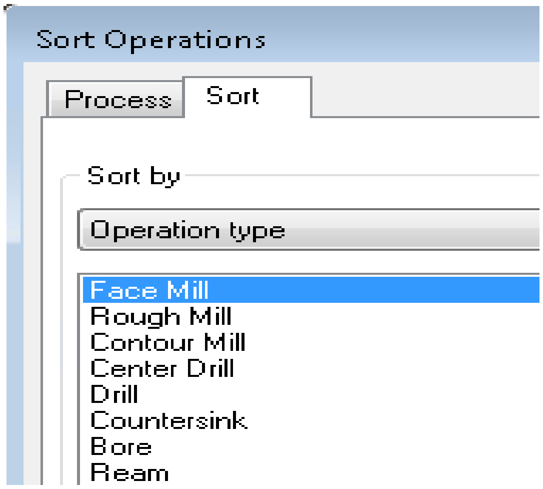
4. Click the *Apply* button to see the result of the current sorting rules.

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

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

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

   - Select *Rough Mill* in the list and drag it below *Face Mill*. Since there is no *Face Mill* operation in the operation tree for this part, positioning the *Rough Mill* cycle below *Face Mill* does not matter.
   - Select *Contour Mill* and drag and drop it below *Rough Mill.*
   - Ensure that *Center Drill*, *Drill*, *Countersink*, *Bore* and *Ream* operations are listed below Contour Mill operation necessarily in the same order as it is given here.



Sort Operations

Process | Sort

Sort by

Operation type

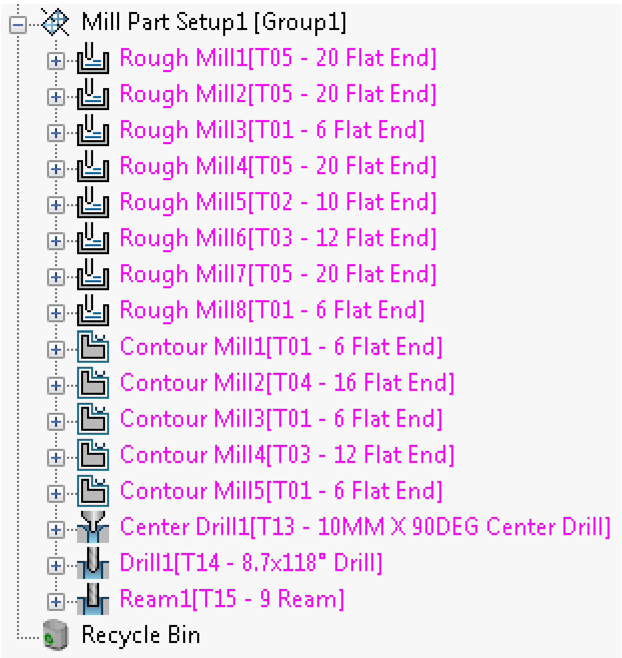
Face Mill
Rough Mill
Contour Mill
Center Drill
Drill
Countersink
Bore
Ream

New Sorting Sequence

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

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

   The order of the operations in the Operation tree can be further modified using drag and drop functionality if necessary.



Mill Part Setup1 [Group1]
Rough Mill1[T05 - 20 Flat End]
Rough Mill2[T05 - 20 Flat End]
Rough Mill3[T01 - 6 Flat End]
Rough Mill4[T05 - 20 Flat End]
Rough Mill5[T02 - 10 Flat End]
Rough Mill6[T03 - 12 Flat End]
Rough Mill7[T05 - 20 Flat End]
Rough Mill8[T01 - 6 Flat End]
Contour Mill1[T01 - 6 Flat End]
Contour Mill2[T04 - 16 Flat End]
Contour Mill3[T01 - 6 Flat End]
Contour Mill4[T03 - 12 Flat End]
Contour Mill5[T01 - 6 Flat End]
Center Drill1[T13 - 10MM X 90DEG Center Drill]
Drill1[T14 - 8.7x118" Drill]
Ream1[T15 - 9 Ream]
Recycle Bin

Operations after sorting

CAUTION!

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

Machining Island Tops

To recap what was explained in the previous tutorials, the *Operation Parameters* dialog box allows you access to all the parameters that will be used to machine a feature. You can change parameters to customize the toolpath that CAMWorks generates to suit your machining requirements.

Did You Know ...

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

In this tutorial, you will change the operation parameters for the Rough Mill operation used to machine the Open Pocket feature so that the following occur:

- The Pocketing pattern creates parallel lace cuts with the cuts alternating in direction. Between each lace cut, the tool stays down in the material. (Zigzag pattern)
- The islands within this Open Pocket are faced off.

Steps to activate option for Machining Island Tops:

1. Double click *Rough Mill1* (the operation for Open Pocket) in the Operation tree to open its corresponding *Operation Parameters* dialog box.
2. Click the *Roughing* tab in this dialog box.
3. Change the Pocketing Pattern to *Zigzag*.

   This option creates parallel lace cuts with the cuts alternating direction. Between each lace cut, the tool stays down in the material.
4. In the *Pocketing* group box, ensure that the checkbox option *Machine island top* is enabled so that the islands on the top of the part are faced off.
5. Click *OK* to apply the changes and close the dialog box.

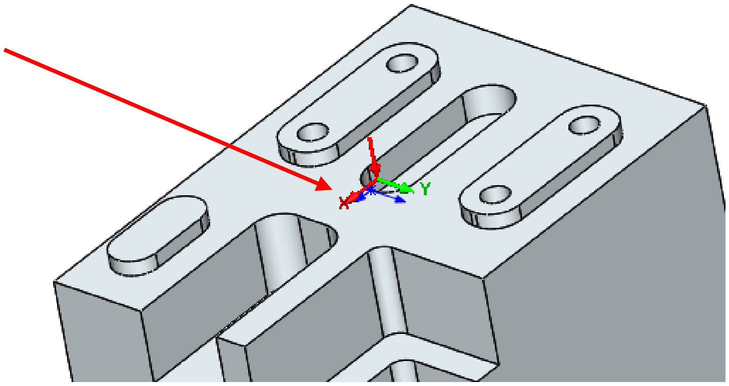
Changing the Mill Part Setup Origin

CAMWorks creates Mill Part Setups automatically during *Automatic Feature Recognition* with the *Mill Part Setup depth* set to the highest face on the recognized features and the *Mill Part Setup origin* is set on the edge of the first recognized feature on the top of the part.

This *Mill Part Setup origin* defines the G-code program zero location using the machine tool's fixture location, Work Coordinates and/or Sub Coordinate offsets. The Mill Part Setup origin position is based on the SOLIDWORKS or CAMWorks Solids origin. You can move the Mill Part Setup origin to a more useful location for machining.

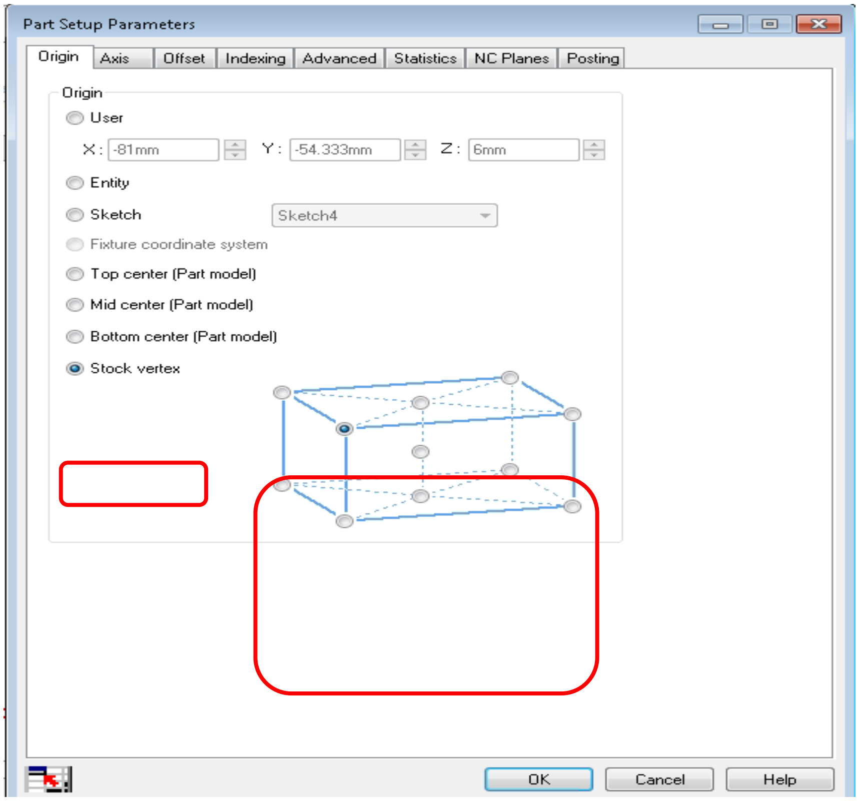
1. Click on *Mill Part Setup1* in the Operation tree.

   Notice where the origin is located on the center of the top face of the part. It is in sync with the SOLIDWORKS origin.



Locating the Mill Part Setup Origin

2. Double click *Mill Part Setup1* in the Operation tree in order to open the *Part Setup Parameters* dialog box.
3. In the *Origin* tab, select the *Stock vertex* option.
4. Pick the corner as shown in the below image.



Origin

○ User

X : -81mm Y : -54.333mm Z : 6mm

○ Entity

○ Sketch Sketch4

○ Fixture coordinate system

○ Top center (Part model)

○ Mid center (Part model)

○ Bottom center (Part model)

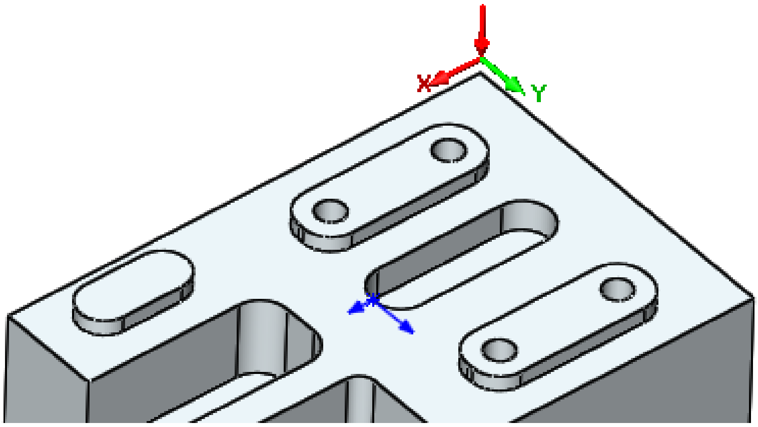
● Stock vertex

Changing Mill Part Setup Origin to a stock vertex

5. Click *OK* to apply the changes and close the dialog box.
6. Click on the *Mill Part Setup1* and observe the part in the graphics area.

   The origin of the Mill Part Setup is moved on the part to the corner you selected.

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



New Mill Part Setup Origin at selected stock vertex

Combining Operations

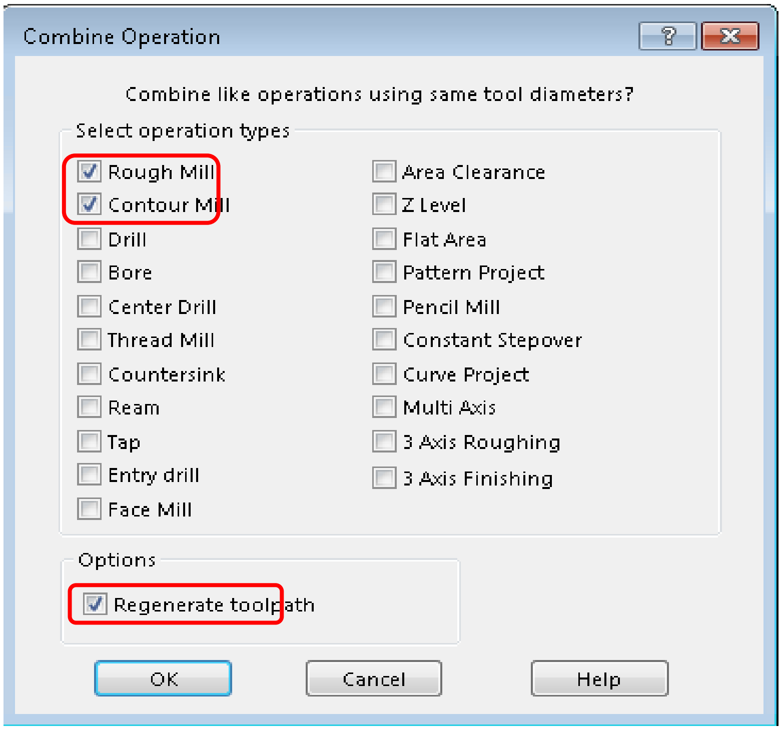
Notice that several roughing and contouring operations use the same tool. Similar operation types that use the same tool can be combined into one operation in order to improve efficiency.

Steps to combine operations:

1. In the Operation tree, right-click on *Mill Part Setup1* in the tree and select *Combine Operations* command from the context menu.

   The Combine Operation dialog box will be displayed.
2. Select *Rough Mill* and *Contour Mill* from the list.
3. Ensure that the *Regenerate Toolpath* option is checked.
4. Click *OK* to apply the changes and close the dialog box.

   CAMWorks combines similar Rough Mill operations that use the same tool. Same applies to the Contour Mill operations too.
5. Click the ⊞ next to the *Rough Mill1*, *Rough Mill3*, *Rough Mill5* and *Contour Mill1* operations in the Operation tree to expand these items.



Combine like operations using same tool diameters?

Select operation types

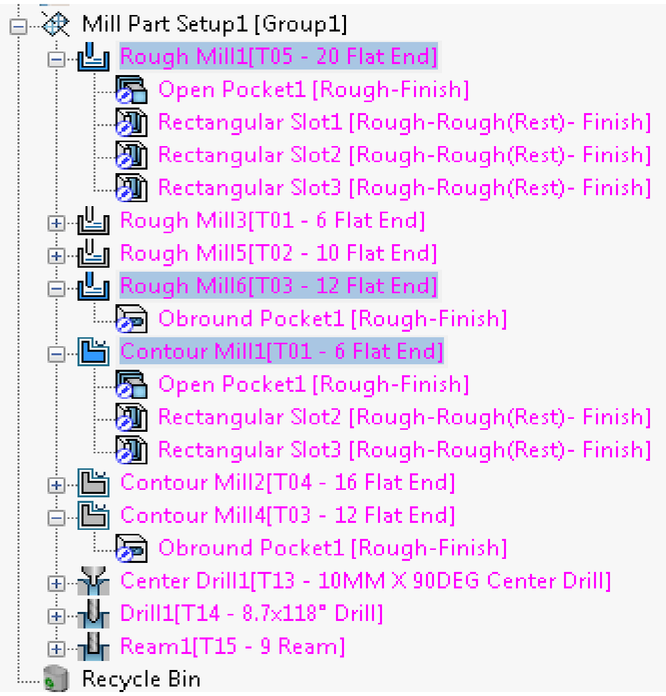
☑ Rough Mill
☑ Contour Mill
☐ Drill
☐ Bore
☐ Center Drill
☐ Thread Mill
☐ Countersink
☐ Ream
☐ Tap
☐ Entry drill
☐ Face Mill
☐ Area Clearance
☐ Z Level
☐ Flat Area
☐ Pattern Project
☐ Pencil Mill
☐ Constant Stepover
☐ Curve Project
☐ Multi Axis
☐ 3 Axis Roughing
☐ 3 Axis Finishing

Options

☑ Regenerate toolpath

Combine Operation dialog box

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



- Mill Part Setup1 [Group1]
  - Rough Mill1[T05 - 20 Flat End]
    - Open Pocket1 [Rough-Finish]
    - Rectangular Slot1 [Rough-Rough(Rest)- Finish]
    - Rectangular Slot2 [Rough-Rough(Rest)- Finish]
    - Rectangular Slot3 [Rough-Rough(Rest)- Finish]
  - Rough Mill3[T01 - 6 Flat End]
  - Rough Mill5[T02 - 10 Flat End]
  - Rough Mill6[T03 - 12 Flat End]
    - Obround Pocket1 [Rough-Finish]
  - Contour Mill1[T01 - 6 Flat End]
    - Open Pocket1 [Rough-Finish]
    - Rectangular Slot2 [Rough-Rough(Rest)- Finish]
    - Rectangular Slot3 [Rough-Rough(Rest)- Finish]
  - Contour Mill2[T04 - 16 Flat End]
  - Contour Mill4[T03 - 12 Flat End]
    - Obround Pocket1 [Rough-Finish]
  - Center Drill1[T13 - 10MM X 90DEG Center Drill]
  - Drill1[T14 - 8.7x118° Drill]
  - Ream1[T15 - 9 Ream]
- Recycle Bin

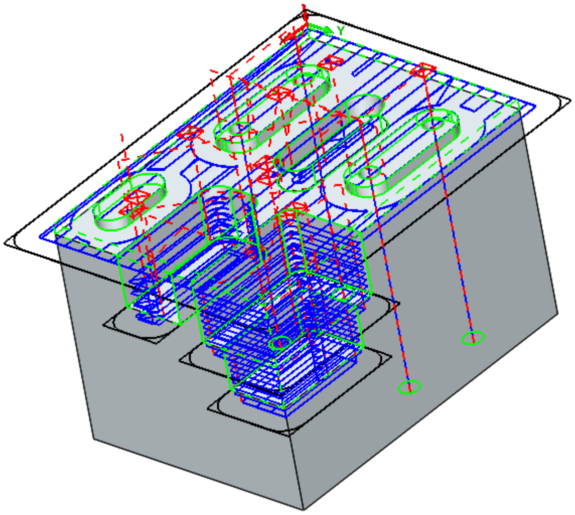
Did You Know ...

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

Changing Tool Parameters and Generating Toolpath

1. Double click *Drill1* in the Operation tree.
2. Click on the *Tool* tab.
3. On the Drill Tool page, change the *Overall length* to **150mm.**
4. On the Mill Holder page, change the *Protrusion* to **130mm**,
5. Click *OK* to apply the changes and close the dialog box.
6. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR



Toolpaths generated for all operations

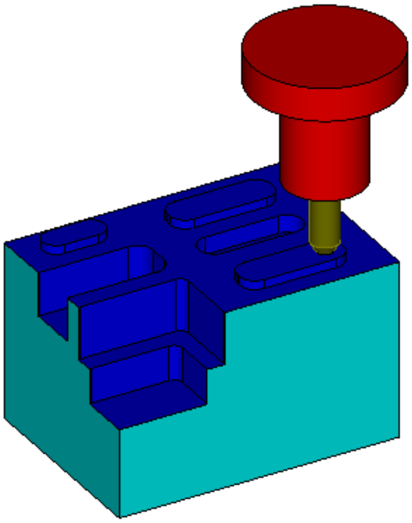
Right click the *Mill Part Setup1* in the Operation tree and select *Generate Toolpath* in the context menu.

7. Highlight the first operation in the tree, hold down the *Shift* key, then highlight the last operation. This action selects all the operations and highlights all the toolpaths on the part in the graphics area.

   Your model should look similar to the figure on the right.

Simulate Toolpath and Post Process

1. Click on the *Simulate Toolpath* button on the CAMWorks Command Manager.
2. ▶ Click the *Run* button on the toolbar to start the simulation.
3. ⏮ Click the *Return* to start button.
4. Click the *Turbo mode* button, and then click *Run*.
5. The simulation does not update the stock until the simulation is completed. This is the fastest method for simulation.
6. ✓ Click the Ok button in the upper left corner of the Simulate Toolpath dialog box to exit simulation mode.
7. Right click on the *Mill Part Setup1* and select *Post Process* on the context menu.
8. The Post Output File dialog box is displayed so you can name the NC program file.
9. Use either the default file name or type a different name and save the file.
10. Click the ▶ *Run* button in the Post Process Output dialog box to output the G-code file.
11. Click *OK* ✓ to close the dialog box.



Simulate Toolpath

2 Axis Mill 6

Topics covered in this tutorial:

- Interactively inserting 2.5 Axis Features
- Avoiding Clamps
- Changing Operation Parameters and Toolpaths
- Customizing Toolpaths

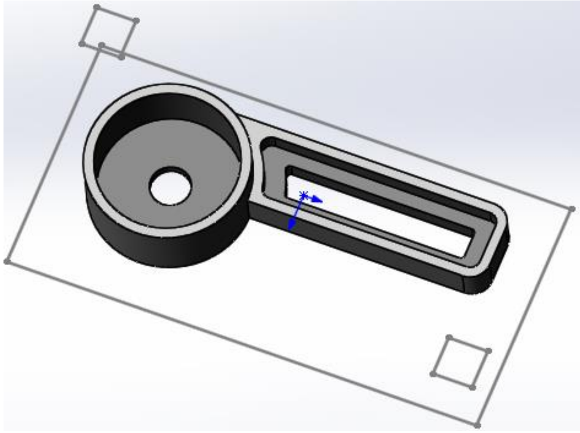
This tutorial involves the machining of a non-rectangular solid part that requires several features to be interactively inserted.

Opening the Part and Extracting Machinable Features by AFR

1. Open the part file **MILL2AX_6.SLDPRT** in the following folder.

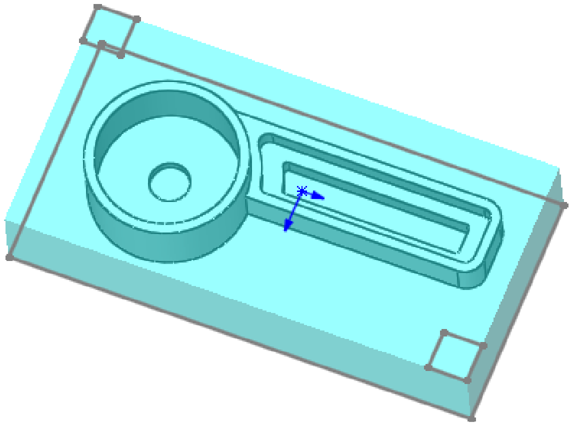
   Drive:\CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill

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



MILL2AX_6.SLDPRT

2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill–Metric]* in the Feature tree to open the *Machine* dialog box.
   - On the *Machine* tab, set *Mill–Metric* as the Active machine.
   - Click the *Tool Crib* tab and make sure *Tool Crib 2 (Metric)* is the Active tool crib.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.
4. Double click *Stock Manager* in the Feature tree. The *Stock* Manager dialog box will be displayed.
   - Click on the *Extruded Sketch* button.
   - In the graphics area, pick the sketch representing the stock.



Defining Stock using Extruded Sketch

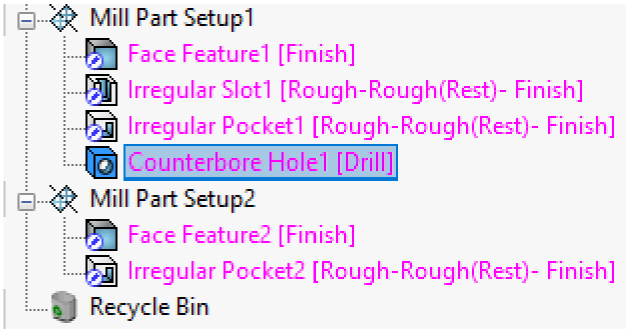
The *Stock* sketch will be listed in the Sketch field.

- Set the Depth to **45mm**.
- If the stock doesn't extrude in the correct direction, click the *Reverse Direction* button.
- Click *OK* to apply the changes and close the dialog box.

5. Click the *Options* button on the CAMWorks Command Manager to open the Options dialog box.
   - On the *Mill Features* tab.
   - Ensure that the *Extract Machinable Features* Method is set to *MfgView*.
   - Under the *Feature Types* group box, check the *Face* option.
   - Make sure that the Max diameter of Hole recognition options group box is set to 1000mm in the Mill features of CAMWorks Options dialog box.
   - Click *OK* to apply the changes and close the dialog box.
6. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

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



Mill Part Setup1
Face Feature1 [Finish]
Irregular Slot1 [Rough-Rough(Rest)- Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Counterbore Hole1 [Drill]
Mill Part Setup2
Face Feature2 [Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Recycle Bin

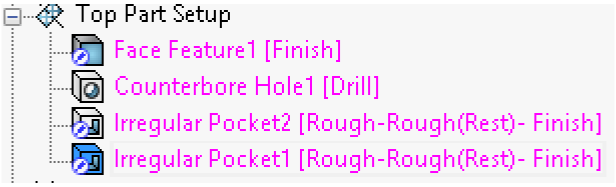
Features recognized using EMF command

Rename Setups and Drag and Drop Features

The Face Feature (for the bottom face of the part) and one Irregular Pocket Feature are recognized under *Mill Part Se*

tup2.

1. Right-click *Mill Part Setup1* and select Rename from the context menu. Rename it to *Top Part Setup*.
2. Right-click *Mill Part Setup2* and select Rename from the context menu. Rename it to *Bottom Part Setup*.
3. In this tutorial, the *Irregular Pocket1* feature and *Counterbore Hole1* feature will be machined from *Top Part Setup*. Use the drag and drop option to move *Irregular Pocket2* to *Top Part Setup*. In this tutorial,



Top Part Setup
Face Feature1 [Finish]
Counterbore Hole1 [Drill]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]

Moving Irregular Pocket2 feature to Top Part Setup

we will interactively insert *Bottom Part Setup* as well as the Face Feature for the bottom face of the part. Hence, right click *Bottom Part Setup* and select *Delete* from the RMB context menu. Click *Yes* to confirm.

4. Right click *Recycle Bin* and select *Empty* from the RMB context menu. Click *Yes* to confirm.

Inserting 2.5 Axis Features Interactively

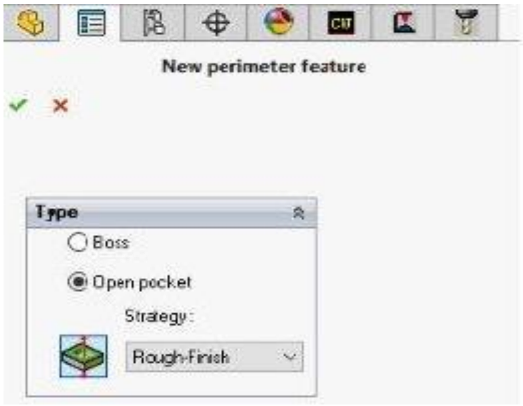
Next, you need to insert a 2.5 Axis Feature to cut the outside shape of the part from the rectangular stock.

Following are the steps:

1. Right click *Top Part Setup* and select *Part Perimeter Feature*.
2. Set the Feature type to *Open Pocket*.
3. Set the Strategy to *Rough-Finish*.

   Selecting this option allows the area between the part and the stock to be roughed.
4. ✓ Click *OK* to insert the changes.

   The action inserts the Perimeter feature to the CAMWorks Feature tree under *Top Part Setup* at the bottom of the list.
5. Drag and drop the features so that they are in the order as shown on the right side image.


New perimeter feature

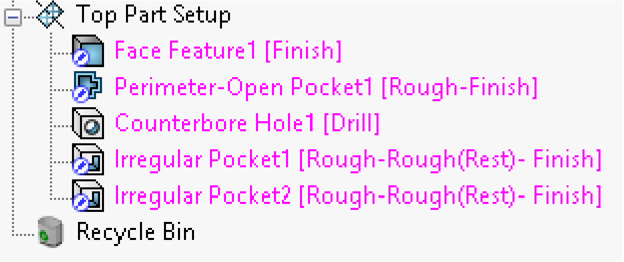
Type

○ Boss

● Open pocket

Strategy: Rough-Finish

New Part Perimeter Feature dialog box


Top Part Setup
- Face Feature1 [Finish]
- Perimeter-Open Pocket1 [Rough-Finish]
- Counterbore Hole1 [Drill]
- Irregular Pocket1 [Rough-Rough(Rest)- Finish]
- Irregular Pocket2 [Rough-Rough(Rest)- Finish]

Recycle Bin

Part Perimeter Feature added to Mill Part Setup

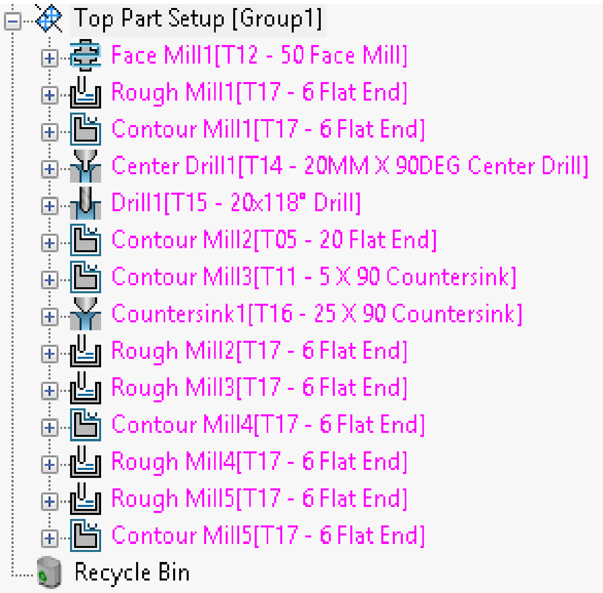
Generating Operations

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

OR

Right click on *Top Part Setup* and select *Generate Operation Plan* from the RMB context menu.

The Operations list should look similar to the image on the right.


Top Part Setup [Group1]
- Face Mill1[T12 - 50 Face Mill]
- Rough Mill1[T17 - 6 Flat End]
- Contour Mill1[T17 - 6 Flat End]
- Center Drill1[T14 - 20MM X 90DEG Center Drill]
- Drill1[T15 - 20x118° Drill]
- Contour Mill2[T05 - 20 Flat End]
- Contour Mill3[T11 - 5 X 90 Countersink]
- Countersink1[T16 - 25 X 90 Countersink]
- Rough Mill2[T17 - 6 Flat End]
- Rough Mill3[T17 - 6 Flat End]
- Contour Mill4[T17 - 6 Flat End]
- Rough Mill4[T17 - 6 Flat End]
- Rough Mill5[T17 - 6 Flat End]
- Contour Mill5[T17 - 6 Flat End]

Recycle Bin

Generated Operations

Defining Avoid Areas for Clamps

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

In this tutorial, you will define avoid areas for the Face Feature and the Perimeter Open Pocket feature to avoid machining the clamps.

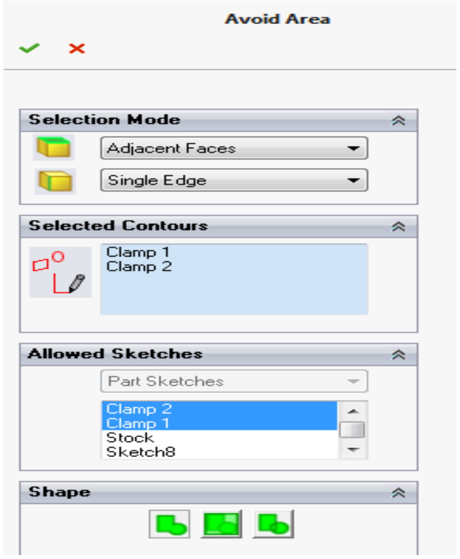
Following are the steps:

1. Right click *FaceMill1* in the Operation tree and select *New Avoid Area* on the RMB context menu.

   The Avoid Area dialog box is displayed.

2. In the Selection Mode group box:

   - Select *Adjacent Faces* in the Face Selection dropdown list.
   - Select *Single Edge* in the Edge Selection dropdown list.

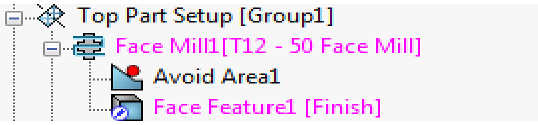

Avoid Area
✓ ✗
Selection Mode
Adjacent Faces
Single Edge
Selected Contours
Clamp 1
Clamp 2
Allowed Sketches
Part Sketches
Clamp 2
Clamp 1
Stock
Sketch8
Shape

Avoid Area dialog box

3. In the *Allowed Sketches* list, select *Clamp 1* and *Clamp 2* from the list. This action selects these sketches to the *Selected Contours* list box.

4. In the Shape section, select *Multiple*. This allows you to create multiple avoid areas from the selected entities.

5. ✓ Click *OK* to close the dialog box and insert the Avoid Area.

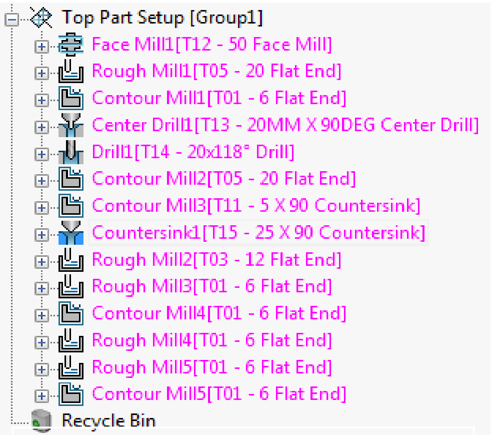

Top Part Setup [Group1]
Face Mill1[T12 - 50 Face Mill]
Avoid Area1
Face Feature1 [Finish]

Avoid Area for Face Mill Operation

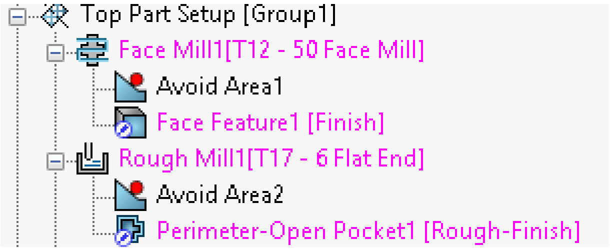
6. Click the (⊞) *plus sign* next to *Face Mill1* in the Operation tree. The Avoid Area is listed under the Face Mill operation.

   You can copy avoids to other operations in the same Mill Part Setup. You can copy the inserted Avoid Area as the Avoid Area for the Rough Mill operation of the Part Perimeter feature.

7. Click *Avoid Area1*, hold down the *Ctrl* key and drag the Avoid on top of Rough Mill1.


Top Part Setup [Group1]
Face Mill1[T12 - 50 Face Mill]
Rough Mill1[T05 - 20 Flat End]
Contour Mill1[T01 - 6 Flat End]
Center Drill1[T13 - 20MM X 90DEG Center Drill]
Drill1[T14 - 20x118° Drill]
Contour Mill2[T05 - 20 Flat End]
Contour Mill3[T11 - 5 X 90 Countersink]
Countersink1[T15 - 25 X 90 Countersink]
Rough Mill2[T03 - 12 Flat End]
Rough Mill3[T01 - 6 Flat End]
Contour Mill4[T01 - 6 Flat End]
Rough Mill4[T01 - 6 Flat End]
Rough Mill5[T01 - 6 Flat End]
Contour Mill5[T01 - 6 Flat End]
Recycle Bin

Operation list



Top Part Setup [Group1]
Face Mill1[T12 - 50 Face Mill]
Avoid Area1
Face Feature1 [Finish]
Rough Mill1[T17 - 6 Flat End]
Avoid Area2
Perimeter-Open Pocket1 [Rough-Finish]

Avoid Areas

8. Drag and drop the operations to match the list shown on the right.

Changing Operation Parameters and Generating Toolpaths

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

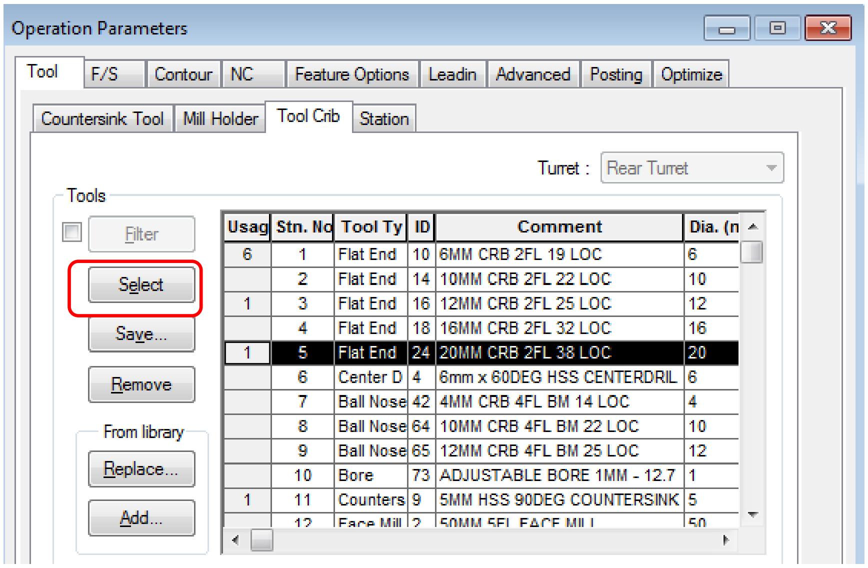
1. Double click *Face Mill1* in the Operation tree to open the *Operation Parameters* dialog box.
2. Under the *Tool Crib* page of the *Tool* tab, click the *Add* button to select a new tool from the Tool library.
3. The Tool Select Filter dialog box is displayed. Leave the Tool type set to *Face Mill* and click *OK*.
4. The Tool Database (metric) form is displayed. Select a **100mm** diameter tool on the Tools Database form and click *OK*.

   This action closes the form and adds the selected tool to the active Tool crib. The new tool is listed at the bottom of the Tool Crib grid.
5. On the Tool Crib tab, scroll down the list of tools and highlight the **100mm** Face Mill tool and click *Select*.
6. Click *Yes* to replace the corresponding holder.
7. Click *OK* to apply the changes and close the Operation Parameters dialog box.
8. Double click *Rough Mill1* in the tree. This is the Rough Mill operation for the Part Perimeter feature.
9. The Operation Parameters dialog box for this operation is displayed. On the Tool tab, click on the *Mill Tool* page. Change the Flute Length to **20mm.**
10. On the Roughing tab, make sure that in the *Pocketing* group box, the *Cleanup pass* option is not checked.
11. CAMWorks Warning message is displayed. Click on Add to create a new tool.
12. In the Depth Parameters group box, change the Bottom allowance to **4.5mm**, then click *OK* to apply the changes and close the Operation Parameters dialog box.

    The stock was defined in such a way that the bottom of the part is 5mm off the bottom of the stock. When machining the Perimeter feature, if the Bottom allowance is 5mm or slightly less, then a skin will remain to hold the part to the clamp.

13. Double click *Contour Mill1* in the tree. This is the Contour Mill operation for the Part Perimeter feature.
14. The Operation Parameters dialog box for this operation is displayed. Click the *Contour* tab in this dialog box.

    By default, the depths of cut will produce multiple contour cuts.
15. In the Depth Parameters group box, change the value for First cut amt. to **50mm** and the Allowance to **4.5mm**. (Similar to Rough Mill1, this Allowance allows the part to remain attached to the stock.)
16. For Depth processing, select the *To Depth by region* option in the Depth Processing group box.
17. Click *OK* to apply the changes and close the Operation Parameters dialog box.
18. The default values for milling the Counterbore are small and will produce more cuts than are needed. You can edit the operation and change the values.
19. Double click *Contour Mill3*. This is the Milling operation for the Counterbore hole.
20. The Operation Parameters dialog box is displayed. Click on the *Contour* tab and change the following parameters:
    - In the Depth Parameters group box: First cut amt. = **10mm** and Max cut amt. = **10mm**
    - In the Side parameters section, click the *Settings…* button. The *Side Parameters* dialog box is displayed. Change the *Cut amt* to **10mm** and click *OK*.
21. Click the *Tool* tab, and then click on the *Tool Crib* page.
22. In the Tool Crib, scroll through the listed tools to find a **20mm** Flat End mill.



Usag	Stn. No	Tool Ty	ID	Comment	Dia. (n
6	1	Flat End	10	6MM CRB 2FL 19 LOC	6
	2	Flat End	14	10MM CRB 2FL 22 LOC	10
1	3	Flat End	16	12MM CRB 2FL 25 LOC	12
	4	Flat End	18	16MM CRB 2FL 32 LOC	16
1	5	Flat End	24	20MM CRB 2FL 38 LOC	20
	6	Center D	4	6mm x 60DEG HSS CENTERDRIL	6
	7	Ball Nose	42	4MM CRB 4FL BM 14 LOC	4
	8	Ball Nose	64	10MM CRB 4FL BM 22 LOC	10
	9	Ball Nose	65	12MM CRB 4FL BM 25 LOC	12
	10	Bore	73	ADJUSTABLE BORE 1MM - 12.7	1
1	11	Counters	9	5MM HSS 90DEG COUNTERSINK	5
	12	Face Mill	2	50MM 5FL FACE MILL	50

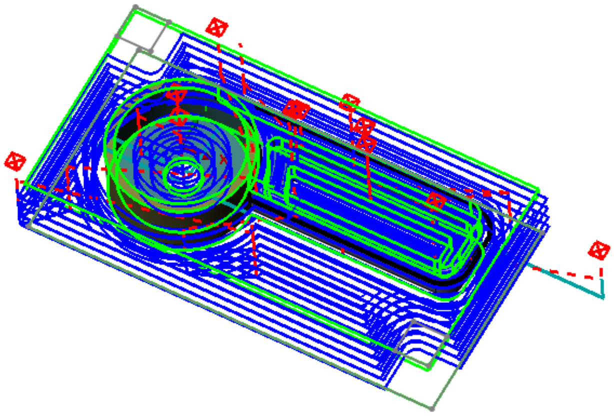
Selecting a 20mm Flat End Tool

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

23. Click in a cell on the **20mm** tool line to select the tool, then click the *Select* button.
24. Click *Yes* to replace the corresponding holder.
25. Click *OK* to apply the changes and close the Operation Parameters dialog box.
26. Double click *Contour Mill5* in the tree. This is the Contour Mill operation for the Irregular Pocket2 feature.

27. In the Operation Parameters dialog box, go to the Tool Crib page of the Tool tab. Scroll the list of tools in the Active Tool crib and select a **20mm** Flat End and click *Select* button.
28. Click *Yes* to replace the holder, then click *OK* to apply the changes and close the dialog box.

Generating Toolpaths



All toolpaths avoid the clamp area

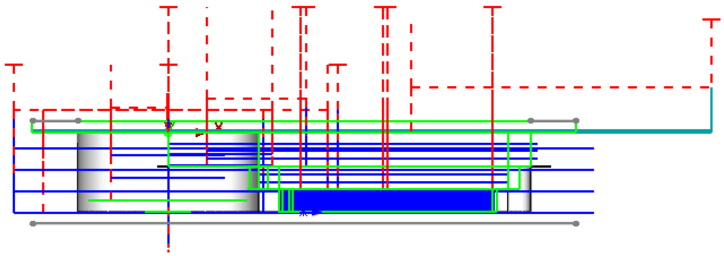
1. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click the *Mill Part Setup1* and select *Generate Toolpath* from the RMB context menu.

   CAMWorks generates the toolpaths for all operations.
2. Hold down the *Shift* key and select the first and last operations in the tree.

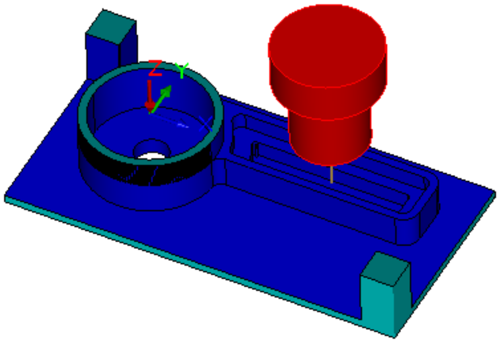
   Notice how the toolpath has avoided the clamp areas.



Toolpaths do not go upto the bottom of the material

3. Click on the *View Orientation* icon and change to the *Top view* and notice that the toolpath has not gone to the bottom of the material because a *Bottom Allowance* of **4.5mm** was set for the Rough and Contour Mill operations of the Part Perimeter feature.

Simulating Toolpaths



Simulating the Toolpath

1. Right click on the *Mill Part Setup* and select *Simulate Toolpath* from the RMB context menu.
2. Click ▶ *Run* to simulate the tool path.

   The result should be similar to the example.
3. ✓ Click *OK* to close the simulation and return to the model.

Inserting a Mill Part Setup and Adding a Facing Cut

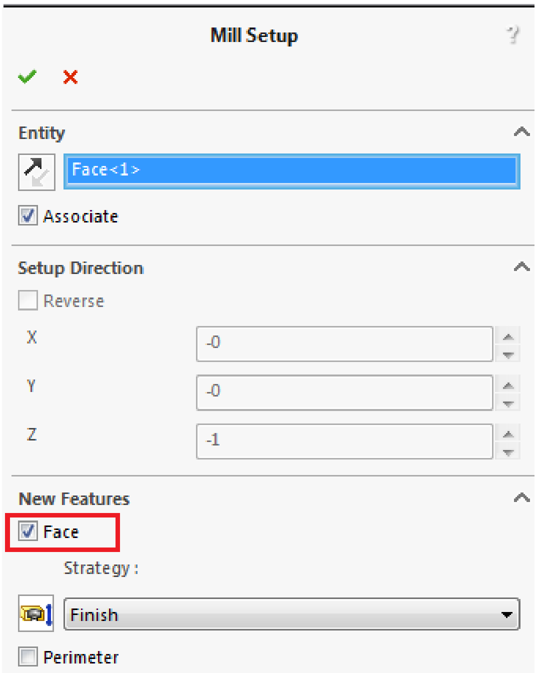
We will now see the steps one must follow to interactively insert a Mill Setup and a Face Feature.

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

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

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

1. Rotate the part so that you can see the bottom face.
2. Click the *CAMWorks Feature Tree* tab.
3. Right click on *Mill Part Setup1* and select *New Mill Part Setup* from the RMB context menu.
4. To insert a Mill Part Setup, you can select either a face or one of the planes listed in the Mill Part Setup dialog box. In this tutorial, we will insert the Mill Part Setup by selecting a face.
5. Pick the bottom face of the part in the graphics area. This action selects the bottom face as the *Selected Entity* within the dialog box.


Mill Setup

Entity

Face<1>

☑ Associate

Setup Direction

☐ Reverse

X -0

Y -0

Z -1

New Features

☑ Face

Strategy : Finish

☐ Perimeter

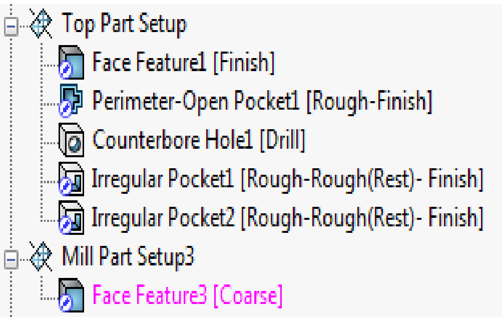
Interactively Inserting a Mill Part Setup

In the *New Feature* group box, make sure the *Face* option is checked. Leave the Strategy set to *Coarse*.

CAMWorks provides the option of automatically adding a Face, Perimeter or Multi Surface Feature at the time of creation of the Mill Part Setup. If the *Face* option is checked, CAMWorks automatically creates a Face feature when the Mill Part Setup is created. To insert this feature, the following two conditions must be fulfilled:

i. The topmost face should be parallel to the Mill Part Setup

ii. The Mill Part Setup should be parallel to one of the sides of the stock.

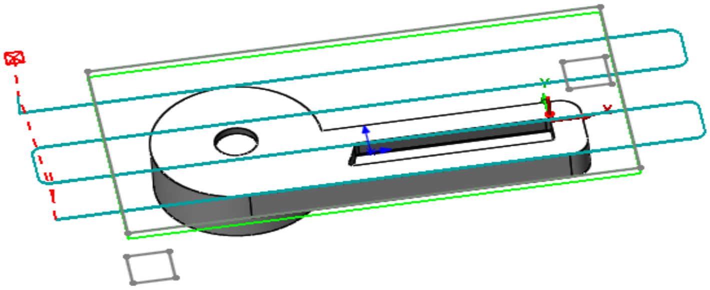

Top Part Setup
- Face Feature1 [Finish]
- Perimeter-Open Pocket1 [Rough-Finish]
- Counterbore Hole1 [Drill]
- Irregular Pocket1 [Rough-Rough(Rest)- Finish]
- Irregular Pocket2 [Rough-Rough(Rest)- Finish]

Mill Part Setup3
- Face Feature3 [Coarse]

Interactively insert Mill Setup

6. Click *OK* to close the dialog box and insert the Mill Setup.

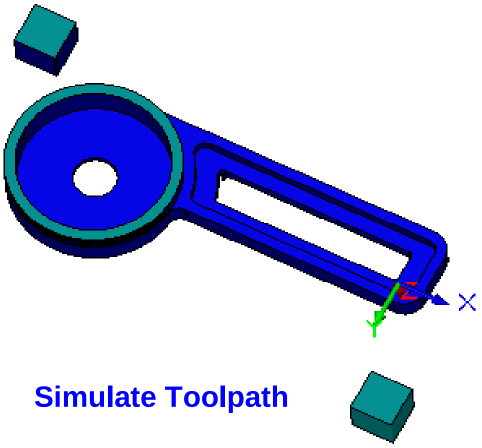
   A new Mill Part Setup is inserted on the bottom of the part. *Mill Part Setup3* and *Face Feature3* are listed in the tree.
7. Right click *Mill Part Setup3* and select *Generate Operation Plan* from the RMB context menu.
8. Rotate the part back to view the top.



Toolpath for Face Feature

9. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
10. Click the *Run* button.

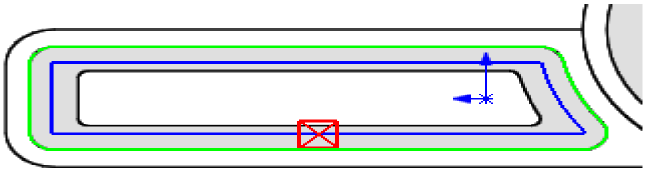
    The resulting simulation should appear as shown. The remainder of the material has been removed, and the part has been machined completely.



Simulate Toolpath

Customizing Toolpaths

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

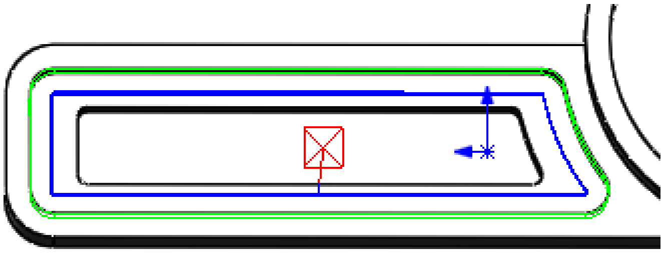


Contour Mill Toolpath

1. Click the *CAMWorks Options* button on the CAMWorks Command Manager.
2. The Options dialog box is displayed. Click the *Update* tab.
3. Set the *Update toolpaths when parameters change* option to *Never*.
4. Click *OK* to apply the changes and close the dialog box.
5. Double click *Contour Mill4* in the Operation tree to open its Operation Parameters dialog box.
6. Click the *Leadin tab*.
7. In the Leadin/out point group box, leave the option set to *Mid-point.*
8. Change the Leadin Type to *Perpendicular*.
9. Set the Leadin amount to **5mm.**

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

    The pierce point displays as a square marker in the middle of the edge because the Mid-point option is selected.



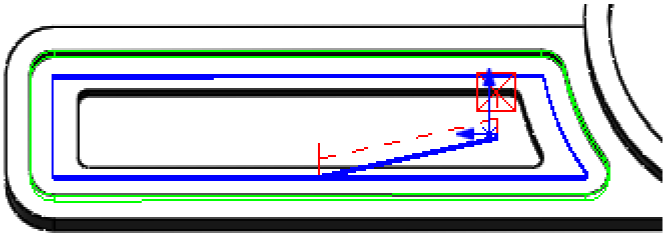
Pierce point marker at the middle of the edge

11. Right click on the same *Contour Mill4* operation and select *Generate Toolpath* from the RMB context menu.
12. Rotate the part and notice the leadin is perpendicular at the mid-point of the edge.

    Notice the SOLIDWORKS/CAMWorks Solids origin. You will use that as the start of the leadin.
13. Double click *Contour Mill4* again.
14. In the Operation Parameters dialog box, click the *Leadin* tab and set the Leadin Type to *None*.
15. Click the *Feature Options* tab.
16. For the Entry Type, use the default Method of *None*.

    With this option, the end mill feeds down in Z at the specified entry point.
17. In the Entry points group box, click in the Entities selected box to set the focus.
18. Rotate the part, if necessary, and pick the SOLIDWORKS/CAMWorks Solids origin with your mouse.
19. *Point* displays in the Entities selected list and the X,Y coordinates are specified.
20. Click *OK* to apply the changes and close the dialog box.
21. Right click on the same *Contour Mill4* operation in the Operation tree and select *Generate Toolpath* on the RMB context menu.

    The start of the leadin changes to the origin point that was selected. This acts as a leadin even though the Leadin Type was set to *None*.



Leadin is perpendicular at the midpoint of the edge

22. Right click *Mill Part Setup1* in the tree and select *Post Process* from the RMB context menu.
23. In the Post Output File dialog box, use either the default file name or type a different name and save the file.
24. ▶ Click the *Run* button in the Post Process Output dialog box to output the G-code file.
25. After reviewing the code, click *OK* ✓ to close the dialog box.

2 Axis Mill 7

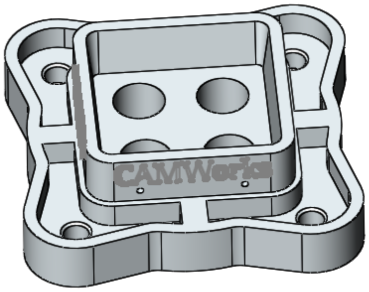
Topics covered in this tutorial:

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

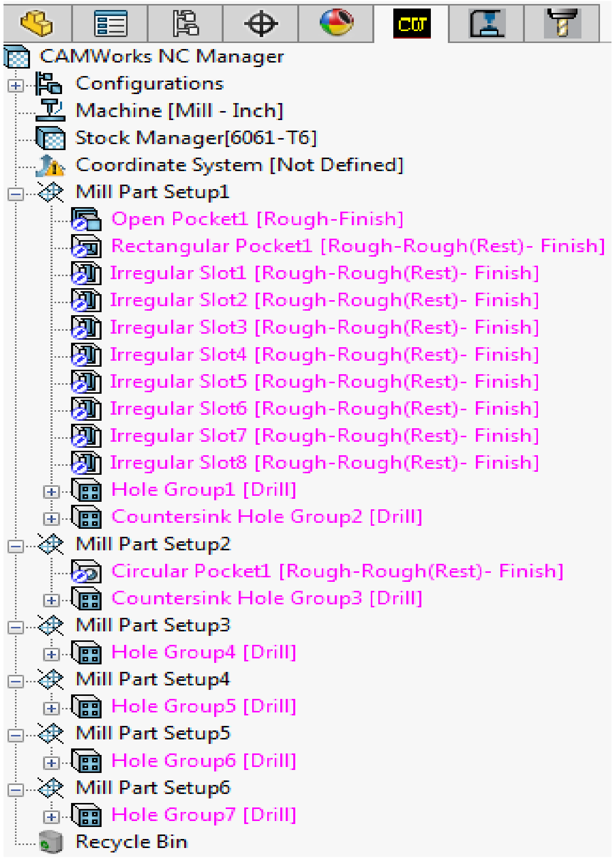
Opening the Part and Recognizing Features Automatically

1. Open the part file **MILL2AX_7.SLDPRT** in the following folder.

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



2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill–inch]* in the Feature tree to open the *Machine* dialog box.
   - On the *Machine* tab, set *Mill–inch* as the Active machine.
   - Click the *Tool Crib* tab and make sure Tool *Crib 2* is the Active tool set.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.
4. Double click *Stock Manager* in the Feature tree. The *Stock Manager* dialog box is displayed.
   - Leave the *Stock Type* set to default *Bounding box.*
   - Click *OK* to use this stock and close the dialog box.
5. Click on *Options* in the Command Manager to open the Options dialog box. On the *Mill Features* Tab, ensure that the Method is set to *MfgView*.



CAMWorks NC Manager
- Configurations
- Machine [Mill - Inch]
- Stock Manager[6061-T6]
- Coordinate System [Not Defined]
- Mill Part Setup1
  - Open Pocket1 [Rough-Finish]
  - Rectangular Pocket1 [Rough-Rough(Rest)- Finish]
  - Irregular Slot1 [Rough-Rough(Rest)- Finish]
  - Irregular Slot2 [Rough-Rough(Rest)- Finish]
  - Irregular Slot3 [Rough-Rough(Rest)- Finish]
  - Irregular Slot4 [Rough-Rough(Rest)- Finish]
  - Irregular Slot5 [Rough-Rough(Rest)- Finish]
  - Irregular Slot6 [Rough-Rough(Rest)- Finish]
  - Irregular Slot7 [Rough-Rough(Rest)- Finish]
  - Irregular Slot8 [Rough-Rough(Rest)- Finish]
  - Hole Group1 [Drill]
  - Countersink Hole Group2 [Drill]
- Mill Part Setup2
  - Circular Pocket1 [Rough-Rough(Rest)- Finish]
  - Countersink Hole Group3 [Drill]
- Mill Part Setup3
  - Hole Group4 [Drill]
- Mill Part Setup4
  - Hole Group5 [Drill]
- Mill Part Setup5
  - Hole Group6 [Drill]
- Mill Part Setup6
  - Hole Group7 [Drill]
- Recycle Bin

Features Recognized by EMF Command

6. In the *Feature types* group box, ensure that the *Face* option is not selected.
7. Click *OK* to close the dialog box.
8. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

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

Did You Know ...

Automatic Feature Recognition (AFR) can extract perimeter features automatically. The Mill Features tab in the CAMWorks Options dialog box has an option to automatically recognize a Part perimeter feature as a boss or open pocket.

Defining Features Interactively

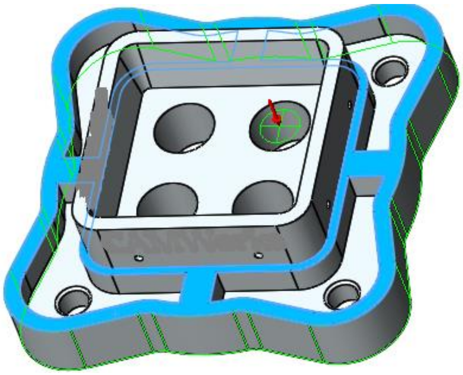
In the tutorial, you will edit the Open Pocket feature to machine the top of the flange and insert features to machine the perimeter of the part and the four corner pockets.

Step 1: Edit the Open Pocket feature to machine only the top of the flange

1. Double click *Open Pocket1* in the tree.

   The 2.5 Axis Feature: Select Entities dialog box is displayed.
2. Click on the *Edit Features* button to activate the dialog box to edit the parameters.
3. End Condition Click the *End Condition* button.
4. In the *2.5 Axis Feature: End Conditions* dialog box, ensure that the *Use Stock extents* option is not checked.

   When the *Use Stock Extents* option is checked, CAMWorks uses the extents of the stock volume to define open pocket features. In this feature, you want to machine only the top of the flange. Hence, this option should be remain unchecked.
5. ✓ Click *OK*.



Inserting Boss feature

Step 2: Insert a Boss feature to machine the perimeter of the part

1. Right click *Open Pocket1* and select *2.5 Axis Feature* from the RMB context menu.
2. Select *Boss* for the Feature Type from the dropdown list.
3. Make sure the *Edge selection* is set to *Convert to loop*.

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

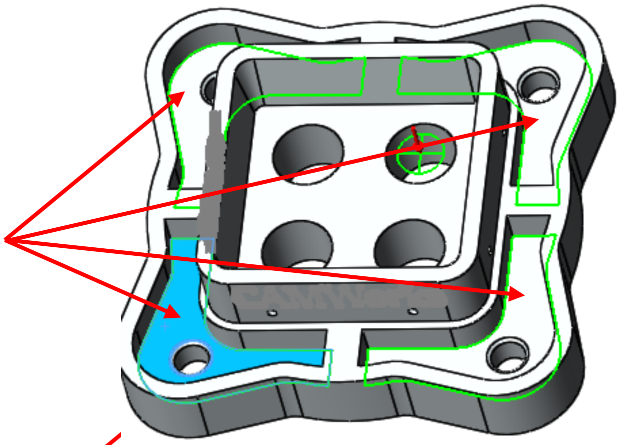
4. Pick the bottom edge of the part. Loop<1> is added to the Entities selected list.
5. End Condition ➔ Click the *End Condition* button. If dialog is pinned then after insertion of feature dialog will remain open.
6. Pick the top face of a flange for the end condition (*highlighted in blue*).

   CAMWorks automatically changes the End condition Type to *Upto Face* and calculates the *Depth*.
7. ✓ Click *OK* to insert the feature.

   Observe that *Irregular Boss1* feature is added below *Open Pocket1* in the Feature tree.

Step 3: Insert Pocket features to machine the bottom faces of the four corner pockets.

Automatic Feature Recognition does not recognize the four irregular corner pockets automatically. They will be inserted interactively.



Irregular Corner Pockets

1. The 2.5 Axis Feature dialog box is still open. Change the Feature Type to *Pocket*.
2. Pick the faces on the bottom of the 4 corner pockets.

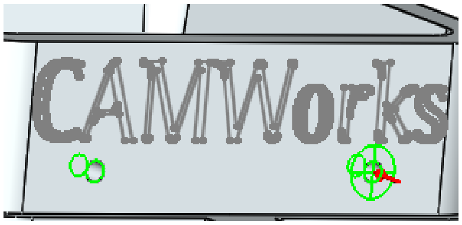
   Face<1>, Face <2>, Face <3> and Face <4> are added to the Selected Entities list.
3. End Condition ➔ Click the *End Condition* button.
4. Pick the face on the top of the flange for the end condition.

   CAMWorks automatically changes the End condition Type to *Upto Face* and calculates the *Depth*.
5. ✓ Click *OK* to insert the pocket features.
6. ✗ Click *Close* to exit the dialog box.

   The four Irregular pocket features are listed in the tree below the Irregular Boss1 feature. The features will be machined in the order they are listed in the feature tree when you generate an operation plan.
7. Drag and drop to reorder the features in *Mill Part Setup1* so that *Open Pocket1* is first and *Rectangular Boss1* is last.

Inserting Engrave Features

Engrave features are not recognized by Automatic Feature Recognition. Engrave features have to inserted interactively. In this tutorial, you will add Engrave features to machine the text on the part.



Engrave Feature on part

1. Right click *Mill Part Setup6* (or the Mill Part Setup on the same face as the CAMWorks text) and select *2.5 Axis Feature* from the RMB context menu.
2. In the 2.5 Axis Feature: Select entities dialog box, select *Engrave Feature* for the Feature Type.
3. In the Available Sketches list, select *CWlogo*.
4. When *CWlogo* is displayed in the *Selected Entities* list, End Condition Click the *End Condition* button.
5. In the 2.5 Axis Feature: End Conditions dialog box, change the Depth to **0.015**, then click *OK*.
6. Click *Close* to exit the dialog box.

   Engrave Feature1 is listed in the tree under *Mill Part Setup6*.

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

Generating an Operation Plan and Adjusting Operation Parameters

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

   OR

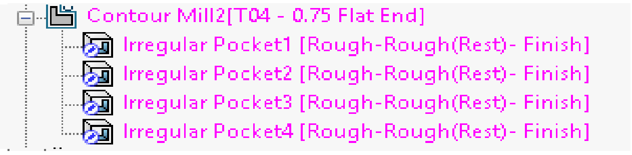
   Right click *CAMWorks NC Manager* in the tree and select *Generation Operation Plan*.

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

   Adjust tools before generating the toolpaths.
2. Right click *Contour Mill2* (the operation for *Irregular Pocket1*) in the tree and select *Combine Operations* from the RMB context menu.

   This command allows you to combine and remove duplicate operations.
3. In the *Combine Operations* dialog box, select the *Combine operations* option.

   When this option is checked, the features machined by operations in the Features or operations list are machined by the current operation and the original individual operations for those features are deleted.
4. Hold down the *Ctrl* key and select the first three operations in the Operations list box . The selected Contour Mill operations are



Contour Mill2[T04 - 0.75 Flat End]
Irregular Pocket1 [Rough-Rough(Rest)- Finish]
Irregular Pocket2 [Rough-Rough(Rest)- Finish]
Irregular Pocket3 [Rough-Rough(Rest)- Finish]
Irregular Pocket4 [Rough-Rough(Rest)- Finish]

Combined Operations

associated with Irregular Pocket features.

5. Click the *Add* button.

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

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

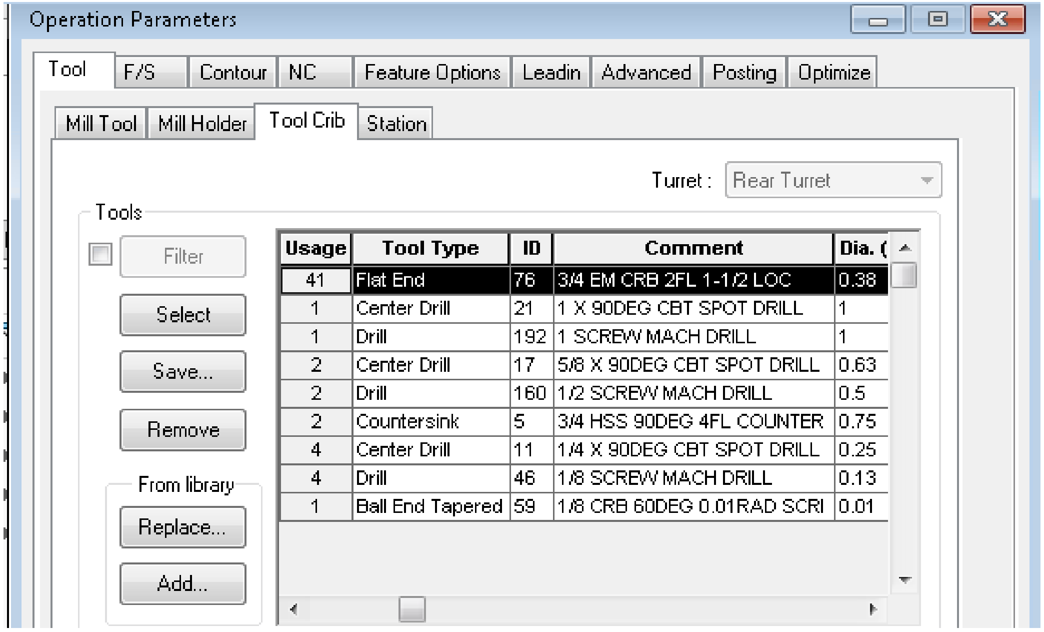
   Contour Mill3, *Contour Mill4* and *Contour Mill5* are removed from the tree and the Irregular Pocket features are listed under *Contour Mill2*.

7. Double click *Contour Mill2* in the Operation tree to open the Operation Parameters dialog box.

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

8. Click on the *Tool* tab. On the *Mill Tool* page under this tab, make sure the current tool is a **0.38in** Flat End mill.

   If a different tool is selected, click the *Tool Crib* page and select a **0.38in** Flat End mill, click the *Select* button, then click *Yes* to replace the corresponding holder.



Turret : Rear Turret

☐ Filter

Usage	Tool Type	ID	Comment	Dia. (
41	Flat End	76	3/4 EM CRB 2FL 1-1/2 LOC	0.38
1	Center Drill	21	1 X 90DEG CBT SPOT DRILL	1
1	Drill	192	1 SCREW MACH DRILL	1
2	Center Drill	17	5/8 X 90DEG CBT SPOT DRILL	0.63
2	Drill	160	1/2 SCREW MACH DRILL	0.5
2	Countersink	5	3/4 HSS 90DEG 4FL COUNTER	0.75
4	Center Drill	11	1/4 X 90DEG CBT SPOT DRILL	0.25
4	Drill	46	1/8 SCREW MACH DRILL	0.13
1	Ball End Tapered	59	1/8 CRB 60DEG 0.01RAD SCRI	0.01

Selecting a 0.38in Flat End tool from the Tool Crib for an operation

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

10. Double click *Contour Mill6* of the Mill Part Setup1 in the Operation tree to open the Operation Parameters dialog box.

11. Click on the *Tool* tab. On the *Mill Tool* page under this tab, make sure the current tool is a **0.5in** Flat End mill.

    If a different tool is selected, click the *Tool Crib* page and select a **0.5in** Flat End mill, click the *Select* button, then click *Yes* to replace the corresponding holder.

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

13. Double click *Contour Mill8* of the Mill Part Setup2 in the Operation tree to open the Operation Parameters dialog box.

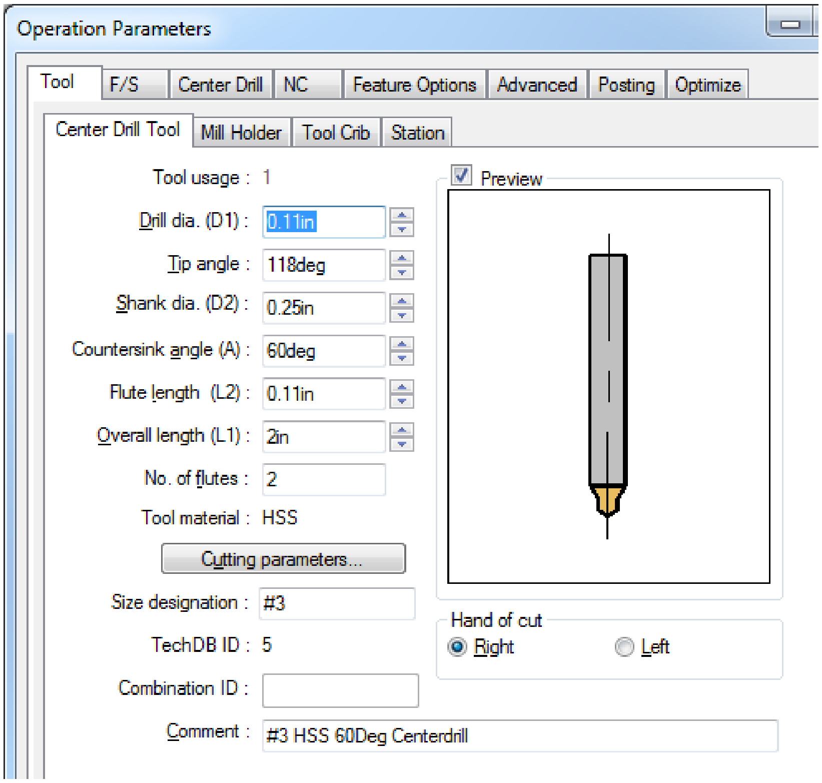
14. Click on the *Tool* tab. On the *Mill Tool* page under this tab, make sure the current tool is a **0.56in** Flat End mill.

    If a different tool is selected, click the *Tool Crib* page and select a **0.56in** Flat End mill, click the *Select* button, then click *Yes* to replace the corresponding holder.

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

16. Look at each *Center Drill* operation in the trees and check the size of the tool selected by the TechDB by viewing the *Center Drill Tool* page under the *Tool* tab of its *Operation Parameters* dialog box.

    The tool for each Center Drill operation should be a **#3 HSS 60Deg Center drill** with a **0.11in** diameter. If a different tool is listed, then, click the *Tool Crib* page and select the correct tool in the Tool Crib.



Tool usage : 1

Drill dia. (D1) : 0.11in

Tip angle : 118deg

Shank dia. (D2) : 0.25in

Countersink angle (A) : 60deg

Flute length (L2) : 0.11in

Overall length (L1) : 2in

No. of flutes : 2

Tool material : HSS

Size designation : #3

TechDB ID : 5

Combination ID :

Comment : #3 HSS 60Deg Centerdrill

☑ Preview

Hand of cut: ● Right ○ Left

Selecting a 0.11in Center drill tool from the Tool Crib for an operation

17. Under Mill Part Setup1 in the Operation tree, double click *Contour Mill7* (the Contour Mill operation for the Irregular Boss1 feature).

18. The Operation Parameters dialog box is displayed. Under the *Tool* tab, click on the *Mill Tool* page. Make sure the current tool is a **0.5in** Flat End mill.

    If it is not, click the *Tool Crib* tab, select a **0.5in** Flat End mill in the Tool Crib, click the *Select* button, then click *Yes* to replace the corresponding holder.

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

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

    OR

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

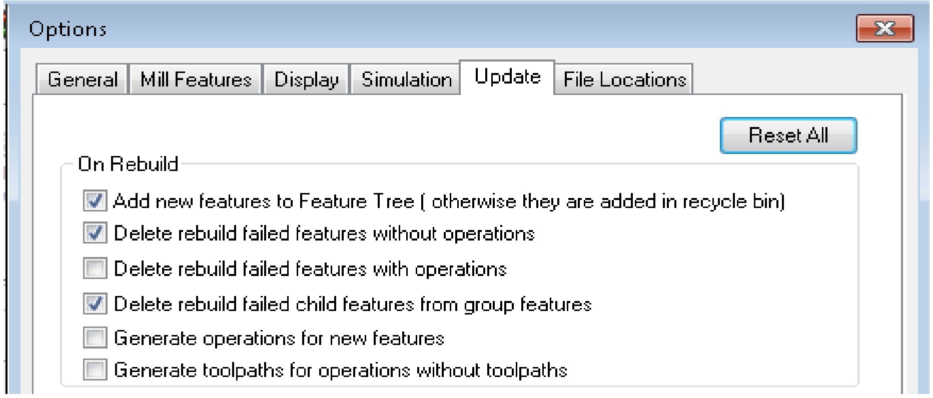
Associating Machining Information after Design Changes

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

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

1. Click the *Options* button on the CAMWorks Command Manager.
2. Click the *Update* tab and make sure the *Generate operations for new features* option is not checked.

   When this option is disabled, you can associate the current operations for the hole feature to the modified hole geometry when CAMWorks rebuilds the machining data.



Options

On Rebuild

☑ Add new features to Feature Tree (otherwise they are added in recycle bin)
☑ Delete rebuild failed features without operations
☐ Delete rebuild failed features with operations
☑ Delete rebuild failed child features from group features
☐ Generate operations for new features
☐ Generate toolpaths for operations without toolpaths

Update Tab on the Options dialog box

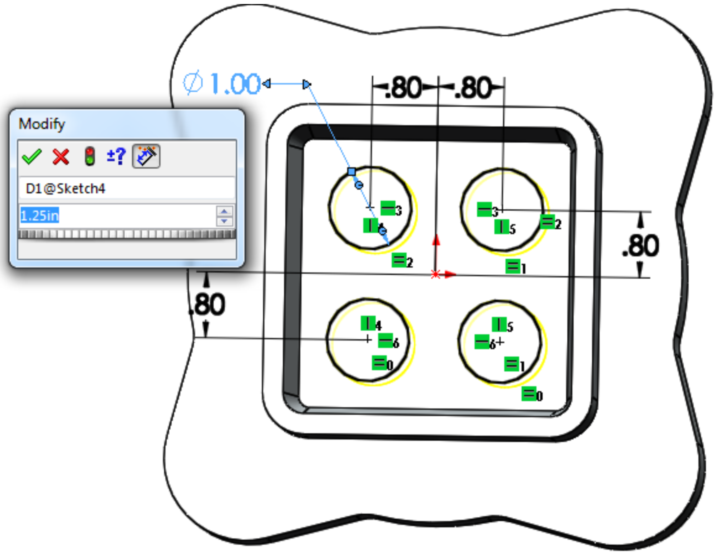
3. Make sure the *Delete rebuild failed features with operations* option is not checked.
4. Click *OK* to apply the changes and close the dialog box.

   When this option is disabled, all features without operations that could not be automatically associated will display in the *Associate Features* dialog box.

5. Click the *FeatureManager Design Tree* tab.
6. In the graphics area, right click on one of the **1.00in** diameter holes and select *Edit Sketch* on the context menu.

   You can view the dimension of holes.

7. Double click on the *1.00in* dimension in the graphics area and change **1.00in** dimension to **1.25in.**



Editing the diameter of the hole

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

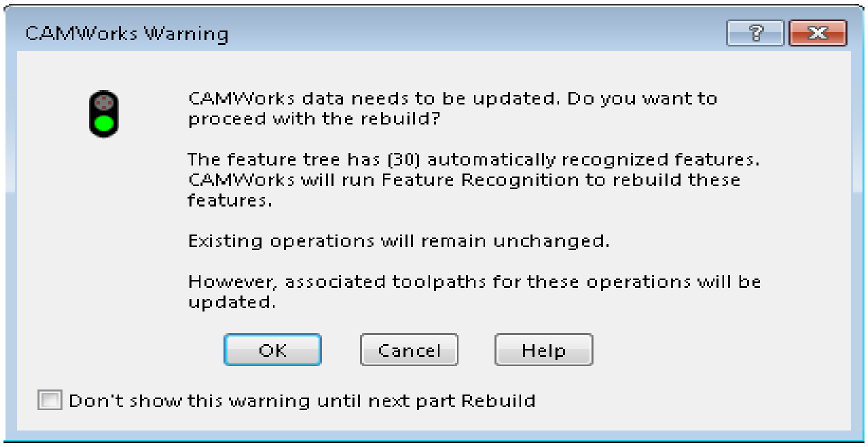
9. Click *Close* dialog box button to close the Dimension dialog box.

10. In the graphics area, right click on the part and select *Exit Sketch* on the context menu.

11. Click the *CAMWorks Feature Tree* tab.

    CAMWorks will display a warning message which display the message that CAMWorks data needs to be updated. Do you want to proceed with the rebuild.

    The Message window is displayed and completing the rebuild process. After completion of the process, the Associate Features dialog box is displayed automatically.



CAMWorks data needs to be updated. Do you want to proceed with the rebuild?

The feature tree has (30) automatically recognized features. CAMWorks will run Feature Recognition to rebuild these features.

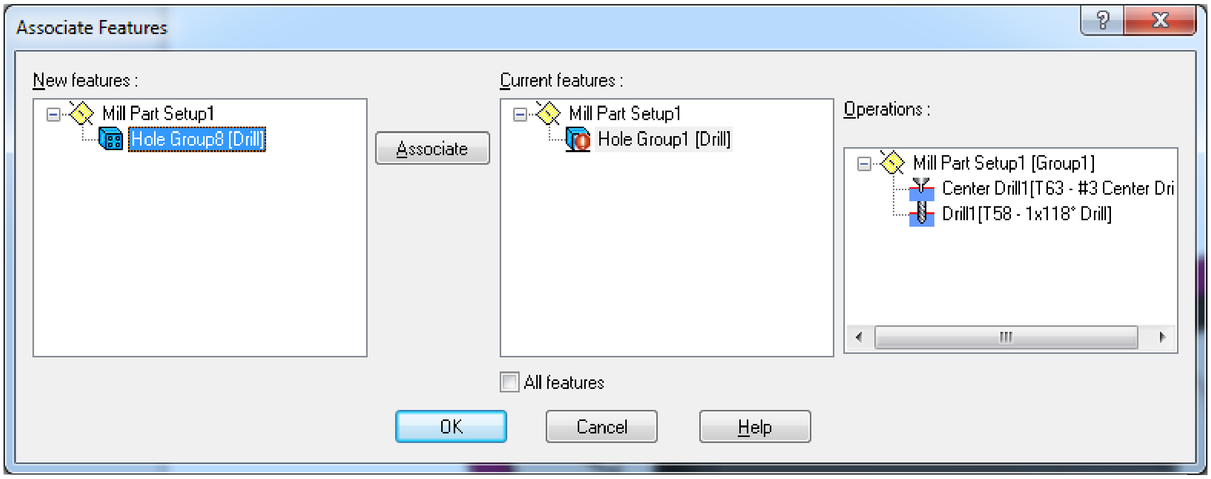
Existing operations will remain unchanged.

However, associated toolpaths for these operations will be updated.

Warning Message dialog box

12. Click *Hole Group1* in the Current features list of Associate Features dialog box.

    When a Current feature is selected, operations used to machine the feature are listed. For visual confirmation, when a Current feature is selected in the list, a red outline of the feature displays on the part in the graphics area.



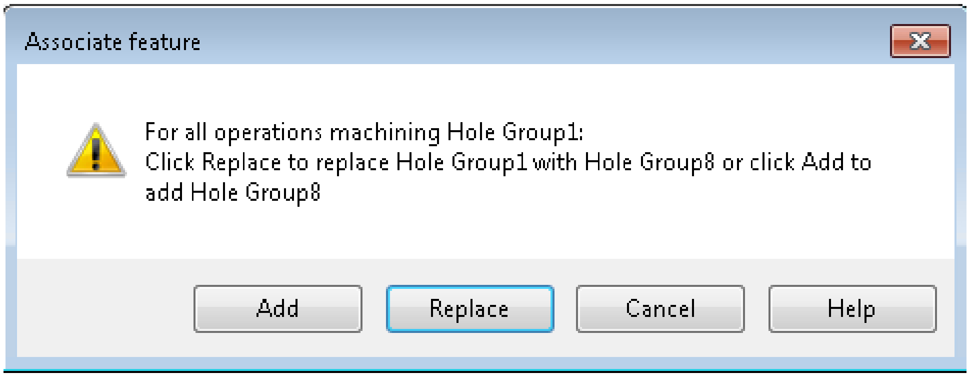
Associate Feature dialog box - Displayed after Rebuild process

13. Click on *Hole Group8* in the New features list. The *Associate* button is enabled.

14. Click the *Associate* button.

    CAMWorks will display a message which prompts you to add, cancel or replace the existing features.

15. Click the *Replace* button on the Associate Features message dialog box.



For all operations machining Hole Group1:
Click Replace to replace Hole Group1 with Hole Group8 or click Add to add Hole Group8

Warning Message

16. Click *OK* to close the Associate Features dialog box.

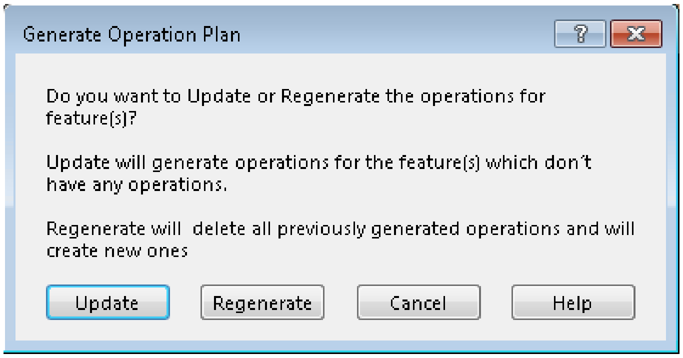
    CAMWorks replaces the selected Current feature (Hole Group1) with the selected new feature (Hole Group8) in every operation that contains the Current feature and deletes Hole Group1 from the Feature tree.

Did You Know ...

If you click the Add button in the message dialog box, CAMWorks adds the New feature to all operations for the selected Current feature. The Current feature is not replaced. Using Add is another method to simplify machining similar features.

17. In the CAMWorks Feature tree, right click *Hole Group8* in the Mill Part Setup1and select *Generate Operation Plan* on the context menu.

    CAMWorks will display a message which prompts you to update or regenerate the operations.



Do you want to Update or Regenerate the operations for feature(s)?

Update will generate operations for the feature(s) which don't have any operations.

Regenerate will delete all previously generated operations and will create new ones

Warning Message

18. Click the *Update* button in the message dialog box to generate new operations for the feature.

    Based on information in the TechDB for the **1.25in** diameter hole, operations *will be* applied to the feature in addition to a Drill operation.

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

    OR

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

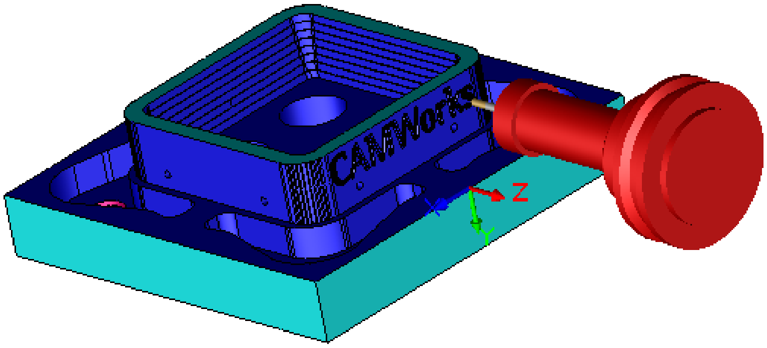
20. If necessary, adjust the tool for the Drill and Contour Mill operations for Hole Group8 and regenerate the toolpaths.

Simulation Toolpath

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

   OR

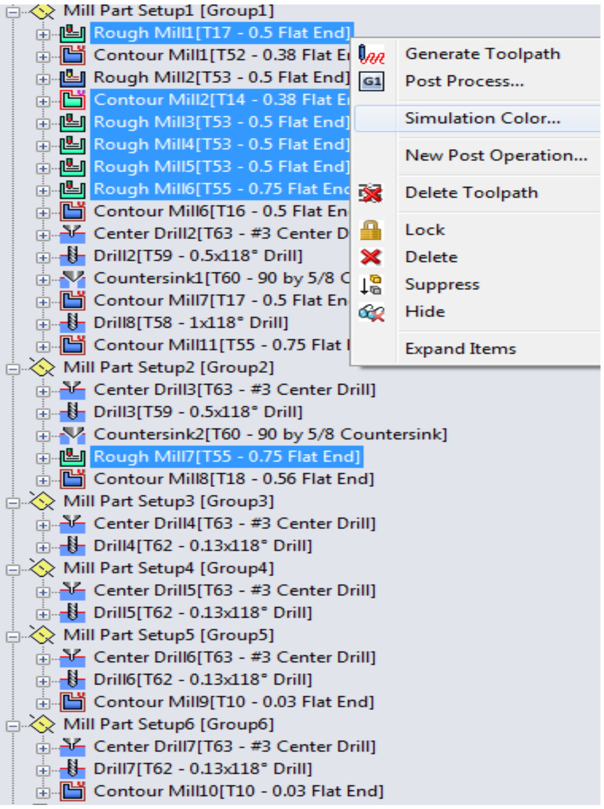
   Right click *CAMWorks NC Manager* in the tree and select *Simulate Toolpath* in the RMB context menu.



Simulation Toolpath

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*
   - Tool: *Ignore Collision*
   - Tool Shank: *Ignore Collision*
   - Tool Holder: *Ignore Collision*
4. Click the *Run* button to start the simulation.
5. Click the *Goto Start* button.
6. Change the *Tool Holder* button to *Tool Holder to Cut Collision* to activate this function.

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



Mill Part Setup1 [Group1]
- Rough Mill1[T17 - 0.5 Flat End]
- Contour Mill1[T52 - 0.38 Flat E
- Rough Mill2[T53 - 0.5 Flat End]
- Contour Mill2[T14 - 0.38 Flat E
- Rough Mill3[T53 - 0.5 Flat End]
- Rough Mill4[T53 - 0.5 Flat End]
- Rough Mill5[T53 - 0.5 Flat End]
- Rough Mill6[T55 - 0.75 Flat En
- Contour Mill6[T16 - 0.5 Flat En
- Center Drill2[T63 - #3 Center D
- Drill2[T59 - 0.5x118° Drill]
- Countersink1[T60 - 90 by 5/8 C
- Contour Mill7[T17 - 0.5 Flat En
- Drill8[T58 - 1x118° Drill]
- Contour Mill11[T55 - 0.75 Flat

Mill Part Setup2 [Group2]
- Center Drill3[T63 - #3 Center Drill]
- Drill3[T59 - 0.5x118° Drill]
- Countersink2[T60 - 90 by 5/8 Countersink]
- Rough Mill7[T55 - 0.75 Flat End]
- Contour Mill8[T18 - 0.56 Flat End]

Mill Part Setup3 [Group3]
- Center Drill4[T63 - #3 Center Drill]
- Drill4[T62 - 0.13x118° Drill]

Mill Part Setup4 [Group4]
- Center Drill5[T63 - #3 Center Drill]
- Drill5[T62 - 0.13x118° Drill]

Mill Part Setup5 [Group5]
- Center Drill6[T63 - #3 Center Drill]
- Drill6[T62 - 0.13x118° Drill]
- Contour Mill9[T10 - 0.03 Flat End]

Mill Part Setup6 [Group6]
- Center Drill7[T63 - #3 Center Drill]
- Drill7[T62 - 0.13x118° Drill]
- Contour Mill10[T10 - 0.03 Flat End]

Selected Rough Mill operation

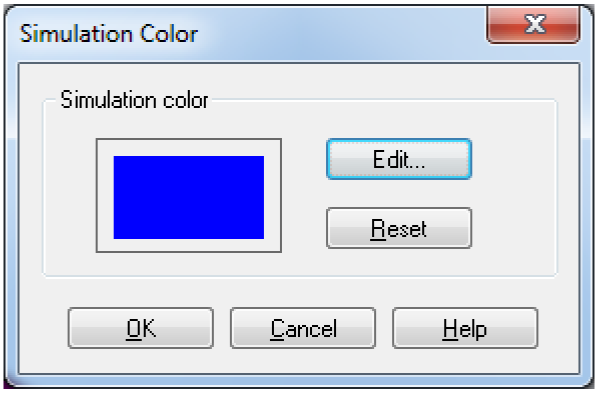
7. Run the simulation again and notice that the drill holders gouge the part.

8. You can select a longer drill tool to eliminate the gouges.

Changing the color of Simulation Toolpath operations

You can modify the simulation color per operation.

1. Select an operation or multiple operations in the Operation tree by using *Ctrl* key.
2. Right click on any one operation and select *Simulation Color* on the context menu.
3. In the Simulation Color dialog box, click *Edit* and choose a different color for selected operation.
4. Click *OK* to apply the changes and close the dialog box.



Simulation Color dialog box

2 Axis Mill 8

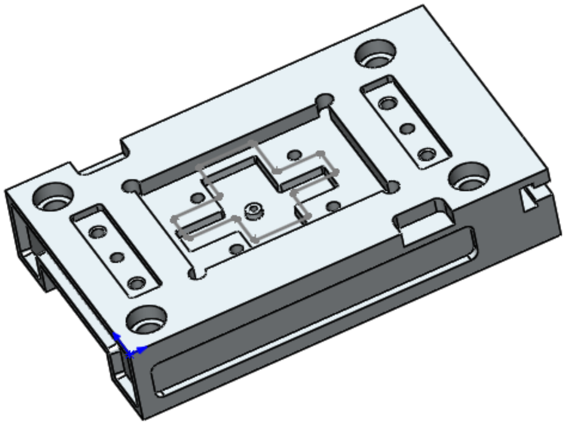
Topics covered in this tutorial:

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

Step 1: Opening the Part and Defining the Machine and Stock

1. Open the part file **MILL2AX_8.SLDPRT** in the following folder.

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



MILL2AX_8.SLDPRT

2. Click the *CAMWorks Feature Tree* tab.
3. Double click the *Machine [Mill-inch]* in the *Feature* tree to open the *Machine* dialog box.
   - On the *Machine* tab, highlight *Mill–inch* as the Active machine.
   - Click the *Tool Crib* tab and make sure *Crib 2* is the Active tool set.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to apply the changes and close the dialog box.
4. Double click Stock Manager in the Feature tree. The *Stock Manager* dialog box is displayed.
   - Leave the *Stock Type* set to default *Bounding box.*
   - Click *OK* to use this stock and close the dialog box.

Step 2: Extracting Machinable Features and Generating Operations

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

   OR

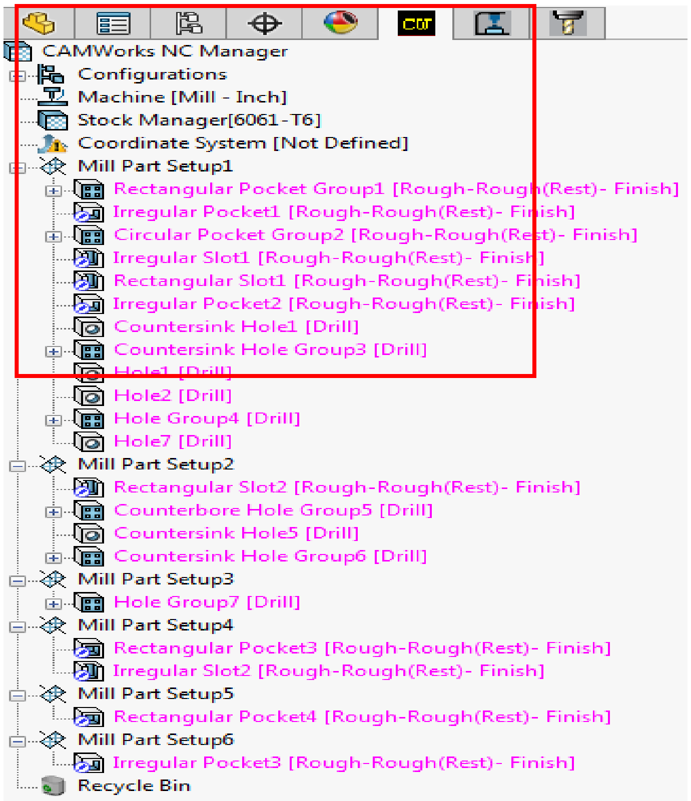
   Right click *CAMWorks NC Manager* in the Feature tree and select *Extract Machinable Features* from the context menu.

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

2. Right click *Mill Part Setup1* in the tree and select *Generate Operation Plan* from the context menu.

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

3. Right click *Mill Part Setup1* in the Operation tree and select *Generate Toolpath* from the context menu.



- CAMWorks NC Manager
  - Configurations
  - Machine [Mill - Inch]
  - Stock Manager[6061-T6]
  - Coordinate System [Not Defined]
  - Mill Part Setup1
    - Rectangular Pocket Group1 [Rough-Rough(Rest)- Finish]
    - Irregular Pocket1 [Rough-Rough(Rest)- Finish]
    - Circular Pocket Group2 [Rough-Rough(Rest)- Finish]
    - Irregular Slot1 [Rough-Rough(Rest)- Finish]
    - Rectangular Slot1 [Rough-Rough(Rest)- Finish]
    - Irregular Pocket2 [Rough-Rough(Rest)- Finish]
    - Countersink Hole1 [Drill]
    - Countersink Hole Group3 [Drill]
    - Hole1 [Drill]
    - Hole2 [Drill]
    - Hole Group4 [Drill]
    - Hole7 [Drill]
  - Mill Part Setup2
    - Rectangular Slot2 [Rough-Rough(Rest)- Finish]
    - Counterbore Hole Group5 [Drill]
    - Countersink Hole5 [Drill]
    - Countersink Hole Group6 [Drill]
  - Mill Part Setup3
    - Hole Group7 [Drill]
  - Mill Part Setup4
    - Rectangular Pocket3 [Rough-Rough(Rest)- Finish]
    - Irregular Slot2 [Rough-Rough(Rest)- Finish]
  - Mill Part Setup5
    - Rectangular Pocket4 [Rough-Rough(Rest)- Finish]
  - Mill Part Setup6
    - Irregular Pocket3 [Rough-Rough(Rest)- Finish]
  - Recycle Bin

Features Recognized by EMF Command

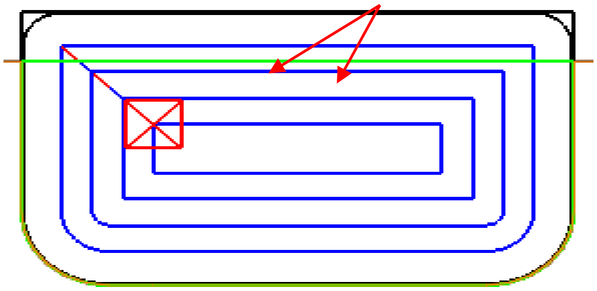
Step 3: Specifying a Tool Overlap on Open Air Segments

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

1. Right click *Mill Part Setup1* in the Operation tree, select *Set View*, and then select *Setup XY* from cascading menu.
2. Left click *Rough Mill3* in the Operation tree and zoom in on the toolpath to see the open slot.

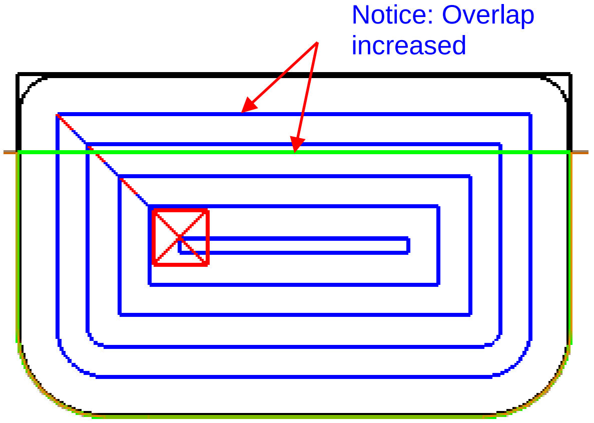
   The toolpath for the Rectangular Slot was generated with the overlap set at 75% of the tool diameter.

3. Double click *Rough Mill3* to open the Operation Parameters dialog box.



Overlap at 75% of tool diameter

4. Click the *Feature Options* tab.
5. In the *Air segment offset* group box, change the *% Tool diameter* to **100.**
6. Click *OK* to apply the changes and close the dialog box.
7. Right click *Rough Mill3* in the Operation tree and select *Generate Toolpath* from the context menu and notice the on the open side of the slot.
8. Zoom to fit the complete part in the graphics area.



Overlap at 100% of tool diameter

Step 4: Using an Avoid Area

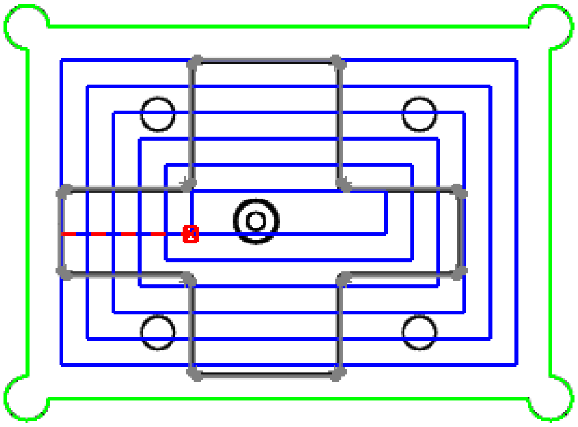
After rough and contour toolpaths have been generated, you can add avoid areas. Avoid areas can be used as a definition for clamps and large holes in the feature where machining is to be avoided.

1. Click on the *View Orientation* and change the view to *Front View* .
2. Left click *Rough Mill4* or the applicable operation for the large pocket in the center of the part and notice the toolpath.
3. Right click *Rough Mill4* and select *New Avoid Area* from the context menu.
4. In the Avoid Area dialog box, select *Avoid Area* in the Available sketches list. This sketch represents the perimeter of the drafted pocket.

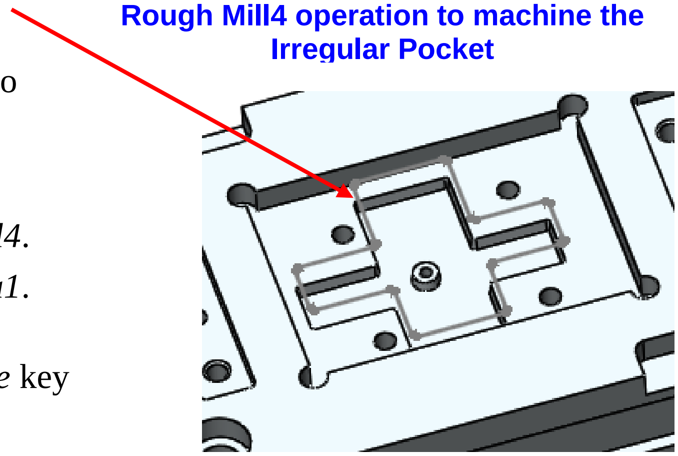
   Optionally, the Offset option allows you to offset the avoid area.
5. Click *OK to* close the dialog box.
6. Click the *plus sign* (⊞) next to *Rough Mill4*.

   The avoid area is identified as *Avoid Area1*.

   If you want to delete an avoid area, highlight it in the tree and press the *Delete* key on the keyboard.

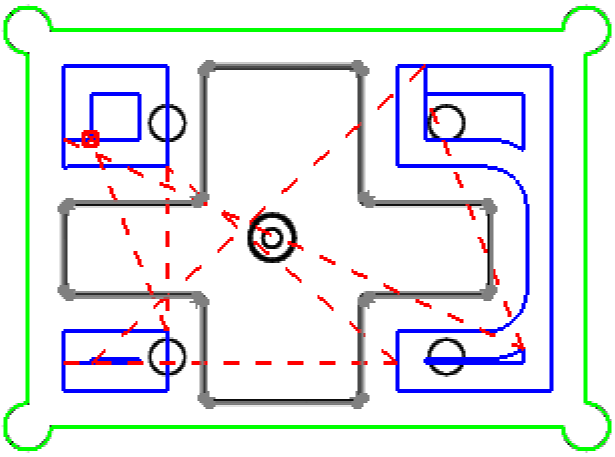


Rough Mill4 operation to machine the Irregular Pocket



7. Right click *Rough Mill4* and select *Generate toolpath* from the context menu.

   Note that the toolpath now avoids the selected sketch.



Toolpaths avoid the selected sketch

Step 5: Defining Rapid and Clearance Planes

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

In this tutorial, you will define the Rapid plane at a global level.

1. Double click *Rough Mill1* in the tree.
2. Click the *NC tab,* make sure in the Rapid Plane and Clearance Plane group boxes, the *Use Setup Definition* options are not checked.

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

   The method and depth change to the current setting for the Setup and the options are grayed out.
4. Click *OK* to close the dialog box.
5. Right click *Rough Mill1* and select *Generate toolpath* from the context menu to regenerate the toolpaths.
6. Double click *Contour Mill1* to open the Operation Parameters dialog box.
7. On the NC tab, check the *Use Setup Definition* in the Rapid plane section, and then click *OK*.
8. Right click *Contour Mill1* and select *Generate toolpath* from the context menu to regenerate the toolpaths.

9. Double click *Mill Part Setup1* in the Operation tree. The Part Setup Parameters dialog box is displayed.

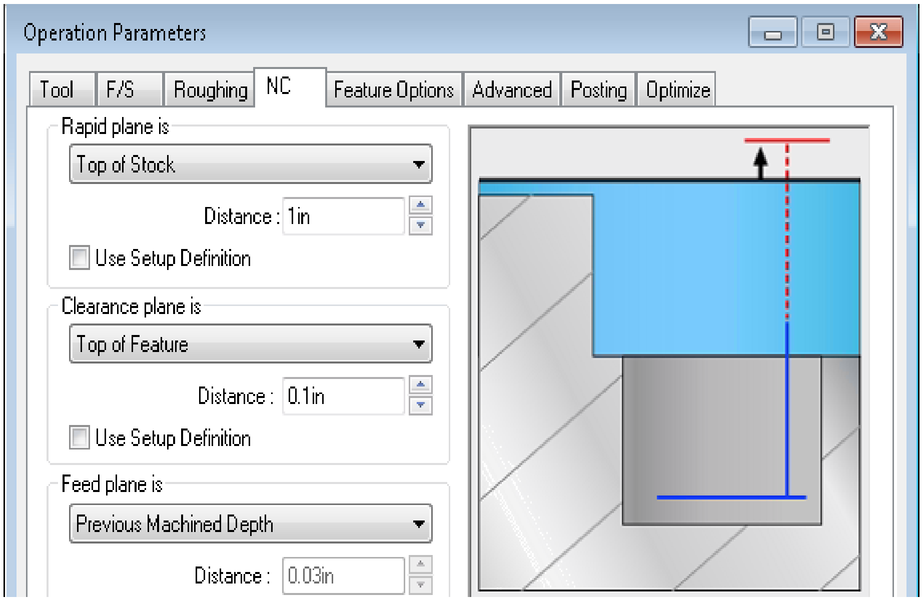
10. Click on the *NC Planes* tab.

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

11. In the Rapid Plane is above dropdown list, select *Top of Stock* in the list.

12. Set the Distance to **1in** and click *OK*.

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



Rapid plane is
Top of Stock
Distance : 1in
☐ Use Setup Definition

Clearance plane is
Top of Feature
Distance : 0.1in
☐ Use Setup Definition

Feed plane is
Previous Machined Depth
Distance : 0.03in

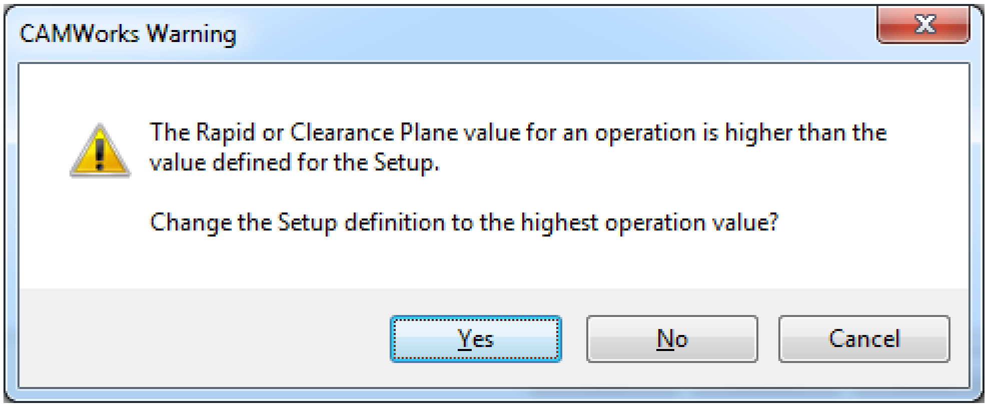
Part Setup Parameters dialog box

13. The warning message is displayed which ask for permission to change the setup definition.

14. Click *Yes.*

15. Right click *Mill Part Setup1* and select *Generate Toolpath* from the context menu.

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



The Rapid or Clearance Plane value for an operation is higher than the value defined for the Setup.

Change the Setup definition to the highest operation value?

Warning message dialog box

2 Axis Mill 9

Topics covered in this tutorial:

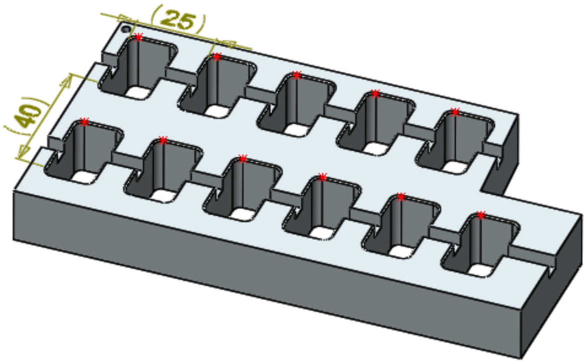
- Defining Features Interactively
- Inserting Linear Pattern Feature
- Inserting Sketch-Driven Pattern Feature

Open the Part and Defining the Machine and Stock

1. Open the part file **MILL2AX_15.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature tree* tab.



MILL2AX_15.SLDPRT

3. Double click on the *Machine [Mill–metric]* in the Feature tree to open the Machine dialog box.
   - On the Machine tab, ensure *Mill–metric* is selected as the Active machine.
   - Click the *Tool Crib* tab and make sure *Tool Crib 2(metric)* is the Active tool set.
   - Click the *Post Processor* tab and select *M3AXIS-TUTORIAL* as the active post processor.
   - Click *OK* to apply the changes and close the Machine dialog box.
4. Double click on the *Stock Manager* item in the *CAMWorks Feature tree*.

   The *Stock Manager* dialog box is displayed.
   - Leave the stock type set to default Bounding box.
   - Click *OK* to use the default stock.

Defining Features Interactively

In this tutorial, you will learn the process of how you can simplify the creation of multiple features by defining one feature and then inserting a Pattern feature to array it to other positions. The toolpath generated for Pattern features is generated only once on the original feature and copied to each position, which reduces the calculation time.

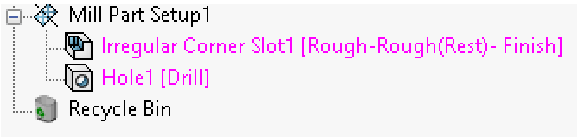
Since the *MfgView* method of Automatic feature Recognition recognizes most 2.5 axis features, you will use the *AFR* method of automatic feature recognition for this tutorial.

1. Click on the *Options* button on the CAMWorks Command Manager.
2. Click on the *Mill Features* tab of the Options dialog box.

3. In the Extract machinable features group box, set the Method to *AFR*.
4. Click *OK* to apply the changes and close the dialog box.
5. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

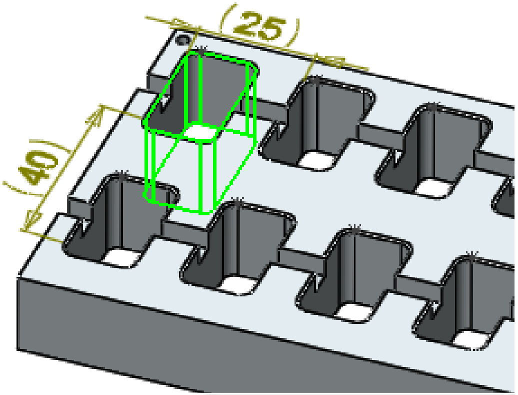
   Right click *CAMWorks NC Manager* and select *Extract Machinable Features* from the context menu.

   Mill Part Setup1
   Irregular Corner Slot1 [Rough-Rough(Rest)- Finish]
   Hole1 [Drill]
   Recycle Bin

   Features Recognized by EMF command

   AFR found an Irregular Corner Slot and a Hole feature. The Pocket features need to be defined interactively. One pocket feature will be defined and it will be duplicated using the Pattern feature.

   Interactively inserting pocket feature

6. Right click *Hole1* in the Feature tree and select *New 2.5 Axis Feature* from the context menu.
7. In the 2.5 Axis Feature: Select Entities dialog box, set the Feature Type to *Pocket*.
8. Pick *Sketch2* in the list. The sketch is highlighted on the part in the graphics area.
9. Pick *Upto Stock* for the End condition. CAMWorks automatically calculates the Depth.
10. Click *OK*, and then click *Close* to close the dialog box.

    The *Irregular Pocket1* feature is added to the Feature tree.

11. Right click *Irregular Pocket1* in the tree and select *Generate Operation Plan* from the context menu.

    The Operation tree lists the Rough Mill and Contour Mill operations that CAMWorks generated for this feature.

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

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

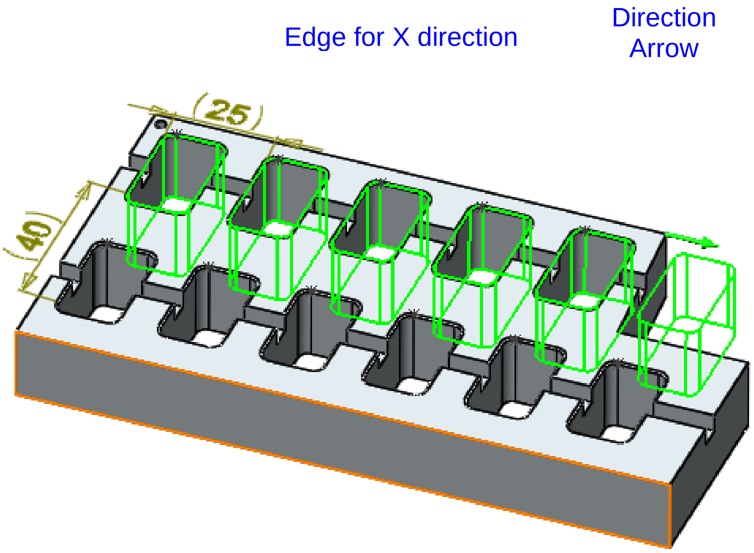
Creating a Pattern Feature

In this tutorial, you will insert the Pattern feature using the *Linear Pattern* and *Sketch Driven Patterns*.

Inserting a Pattern Feature (Linear Pattern)

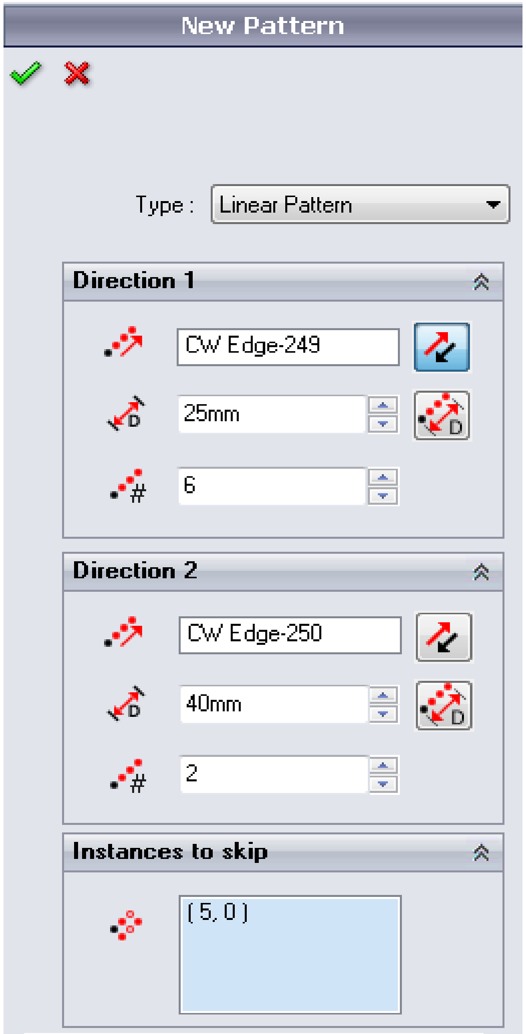
1. Click the *CAMWorks Feature Tree* tab.
2. Right click *Irregular Pocket1* in the tree and select *Pattern*, and then select *Create Pattern*.

3. The *New Pattern* dialog box is displayed. In this dialog box, select *Linear Pattern* for the Type.
4. Pick one of the horizontal edges of the part for the X direction as shown in image below.
5. Click the *Reverse Direction* button, if necessary, to point the arrow in the correct direction as shown in the image below.



Defining Pattern in Horizontal Direction

6. Type **25mm** for the incremental distance between features instances.
7. Type **6** for the Number of instances. The feature instances in the X direction display on the part as shown in right side image.
8. In the *Direction 2* section, click in the direction box to set the *focus*, then pick one of the vertical edges of the part for the Y direction as shown in the image below.
9. Click the *Reverse Direction* button, if necessary, to point the arrow in the correct direction.


New Pattern

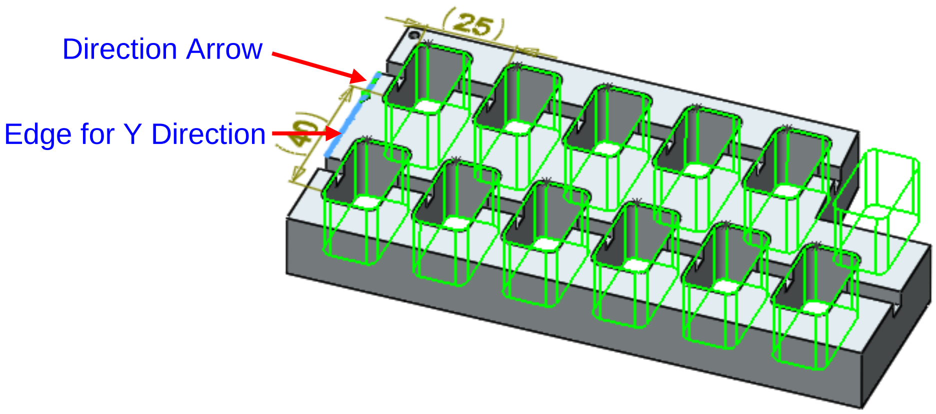
Type : Linear Pattern

Direction 1
CW Edge-249
25mm
6

Direction 2
CW Edge-250
40mm
2

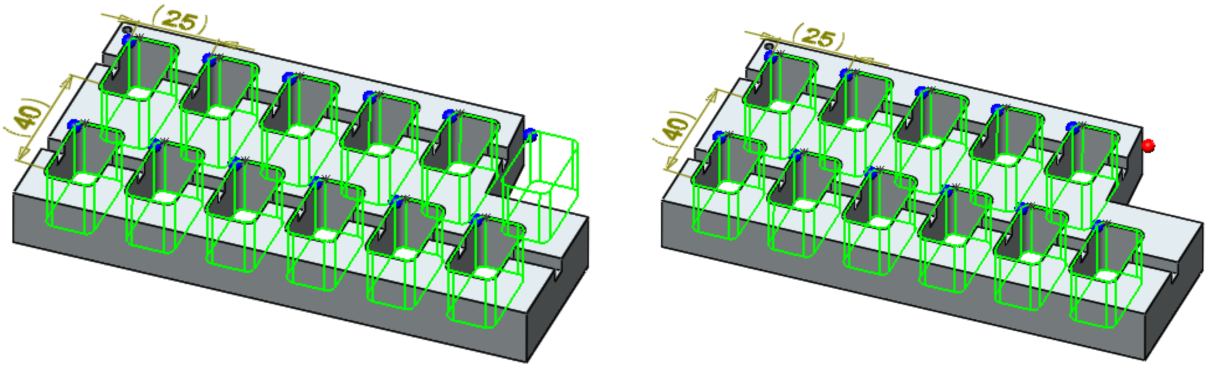
Instances to skip
(5, 0)

New Pattern Dialog Box



Defining Pattern Feature in vertical direction

10. Type **40mm** for the incremental distance between features instances.
11. Type **2** for the Number of instances. The feature instances in the Y direction display on the part as shown below.
12. Click in the *Instances to skip* box to set the focus. Blue points display on each instance as shown in the below image.
13. Move the pointer over the point for the instance on the top right of the part until it turns white, then click. The instance is removed and the point changes to red.



Selecting the instance within the Pattern Feature which is to be skipped

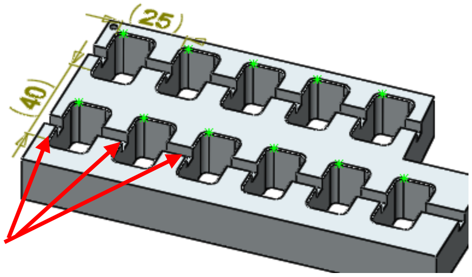
14. Click *OK* to close the dialog box. *Pattern1-Irregular Pocket1* is listed in the Feature tree.
15. Click each operation in the Operation tree. Notice the toolpaths have been copied to each instance.

Inserting a Pattern Feature using a sketch (Sketch-driven Pattern)

1. Click the *CAMWorks Feature Tree* tab.
2. Right click *Pattern1-Irregular Pocket1* in the tree, select *Pattern*, then select *Remove Pattern*.
3. Click *Yes* to confirm you want to remove the Pattern feature.

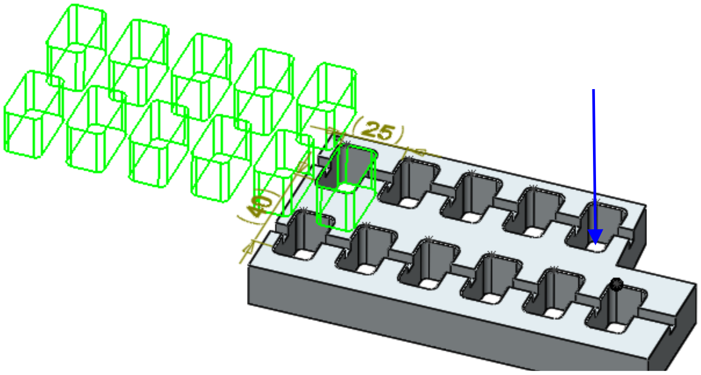
   The Pattern feature is removed. The original feature, *Irregular Pocket1*, and associated operations and toolpaths are not deleted.
4. Right click *Irregular Pocket1* in the Feature tree and select *Pattern*, and then select *Create Pattern*.
5. In the *New Pattern* dialog box, select *Sketch Driven Pattern* for the Type.
6. Pick *Sketch Pattern* in the list of *Available valid sketches*.

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



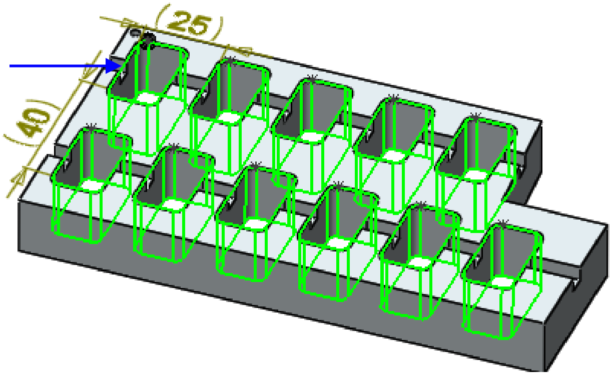
Sketch pattern consists of points on each of the pockets

7. Notice that the pattern is not in the correct position on the part. The point on the pocket at the bottom right identifies the point on the sketch that is the reference point for the feature. The default is the first point in the sketch. When this point is not correct, you can change it.



Sketch-driven Pattern at incorrect position

8. Click the *Move to previous* and *Move to next* buttons to select the sketch point that positions the pattern correctly.
9. Click *OK* to insert the Pattern feature.
10. Click the *CAMWorks Operation Tree* tab. Notice the toolpaths have been copied to each instance.



Sketch-driven Pattern Feature

2 Axis Mill 10

Topics covered in this tutorial:

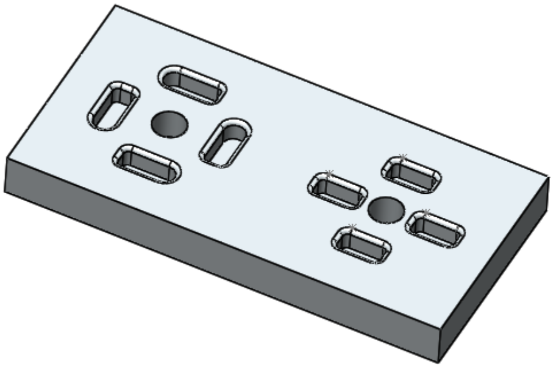
- Defining Features Interactively
- Inserting Circular Pattern Feature
- Inserting Sketch Driven Pattern Feature

Open the Part and Defining the Machine and Stock

1. Open the part file **MILL2AX_16.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.



MILL2AX_16.SLDPRT

3. Double click *Machine [Mill-inch]* in the Feature tree to open Machine dialog box.
   - On the Machine tab, ensure *Mill–inch* is the active machine.
   - Click the *Tool Crib* tab and make sure *Crib 1* is the active tool set.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected as the active post processor.
   - Click *OK* to apply the changes and close the Machine dialog box.

4. Double click on the *Stock Manager* item in the CAMWorks Feature tree.

   The *Stock Manager* dialog box is displayed.
   - Leave the stock type set to default Bounding box.
   - ✓ Click *OK* to use the default stock.

Defining Features Interactively

In this tutorial, you will learn the process of how you can simplify the creation of multiple features by defining one feature and then inserting a Pattern feature to array it to other positions. The toolpath generated for Pattern features is generated only once on the original feature and copied to each position, which reduces the calculation time.

Since the *MfgView* method of Automatic feature Recognition recognizes most 2.5 axis features, you will use the method of automatic feature recognition for this tutorial.

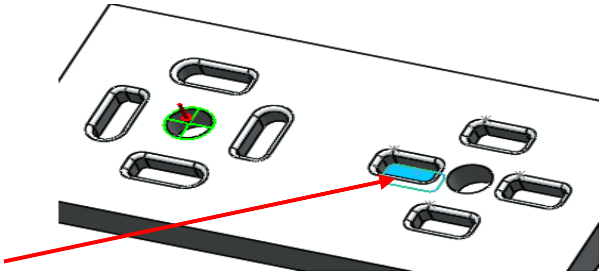
1. Click on the *Options* button on the CAMWorks Command Manager.
2. Click on the *Mill Features* tab of the *Options* dialog box.
3. In the *Extract machinable features* group box, set the Method to *AFR*.

4. Click *OK* to apply the changes and close the dialog box.
5. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   OR

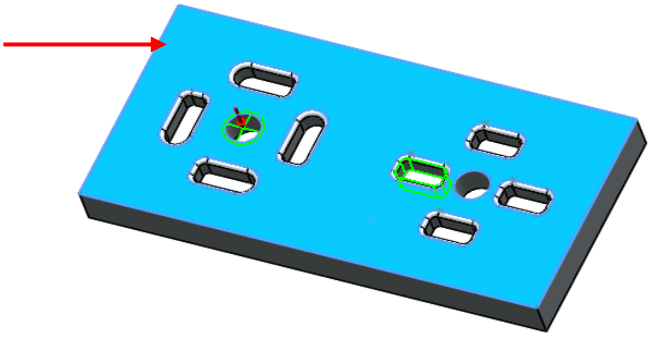
   Right click *CAMWorks NC Manager* and select *Extract Machinable Features* from the context menu.

   AFR has found a Hole Group feature (*Hole Group1*). The Pocket features will be defined interactively. One pocket feature, along with its associated operations and toolpaths, will be defined and then a Pattern Feature will be inserted to array it to other positions.
6. Right click *Hole Group1* in the Feature tree and select *New 2.5 Axis Feature* from the context menu.
7. Set the Feature Type to *Pocket*.



Pick the highlighted blue colored bottom face of the Pocket

8. Make sure the *Check for taper & fillets* option is checked.
9. Pick the highlighted blue colored face at the bottom of the pocket on the right side of the part.
10. End Condition Click the *End Condition* button.
11. Click the top face of the part to set the End condition to *Upto Face*.
12. Click the *Auto detect* button in the 2.5 Axis side wall group box.
13. Click *OK*.



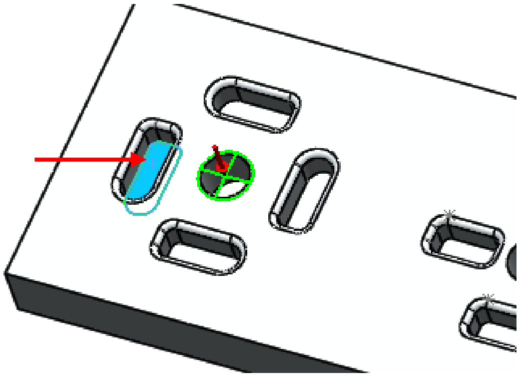
Pick the top face to set the depth

14. Click *Close* to close the 2.5 Axis Feature dialog box.

    Observe that *Rectangular Pocket1* is inserted in the feature tree.

Define the 2.5 Axis feature for the circular pattern

1. Right click *Rectangular Pocket1* in the Feature tree and select *New 2.5 Axis Feature* from the context menu. The *2.5 Axis Feature: Select Entities* dialog box is displayed.



Pick the highlighted blue colored bottom face of the pocket

2. Set the Feature Type to *Pocket*.
3. Make sure the *Check for taper & fillets* option is checked.
4. Pick the highlighted blue colored face at the bottom of the pocket on the left side of the part. This action selects the face in the *Selected Entities* list.
5. Click the top face of the part to set the End condition to *Upto Face*.
6. Click the *Auto detect* button in the *2.5 Axis side wall* group box. This action detects the tapers and fillets.
7. Click *OK*, and then click *Close* to close the dialog box.

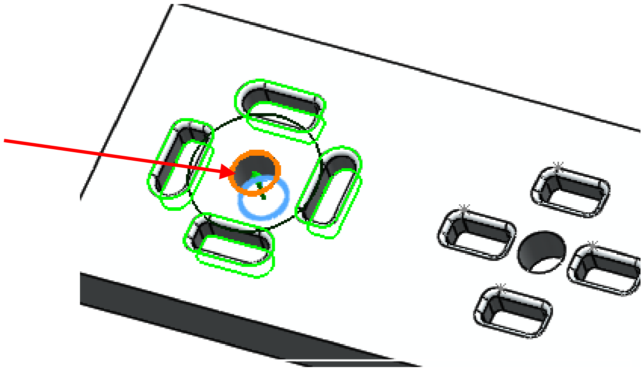
   Observe that *Irregular Pocket1* is listed in the Feature tree.

Creating a Pattern Feature

In this tutorial, you will insert the Pattern feature using the *Circular Pattern* and *Sketch Driven Patterns.*

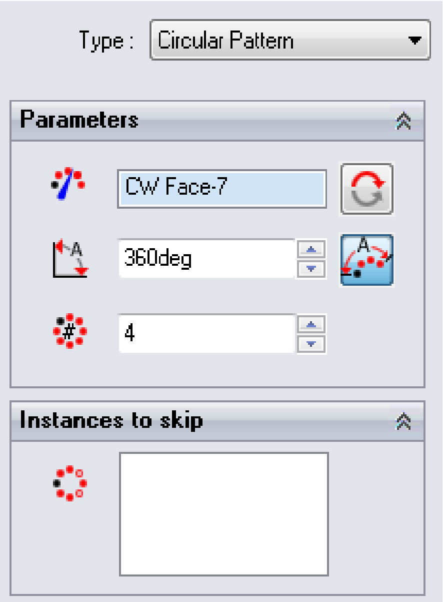
Inserting a Circular Pattern Feature

1. Right click *Irregular Pocket1* in the tree and select *Pattern*, then select *Create Pattern.*
2. In the *New Pattern* dialog box, select *Circular Pattern* for the Type.
3. Pick the top edge of the hole to define the rotation axis.



Defining Pattern Feature in vertical

4. By default, four instances of the pocket are rotated at **360 degrees** around the axis.
5. Ensure that the number of instances is set to **4**.
6. Click *OK* to close the dialog box.



Type : Circular Pattern

Parameters

CW Face-7

360deg

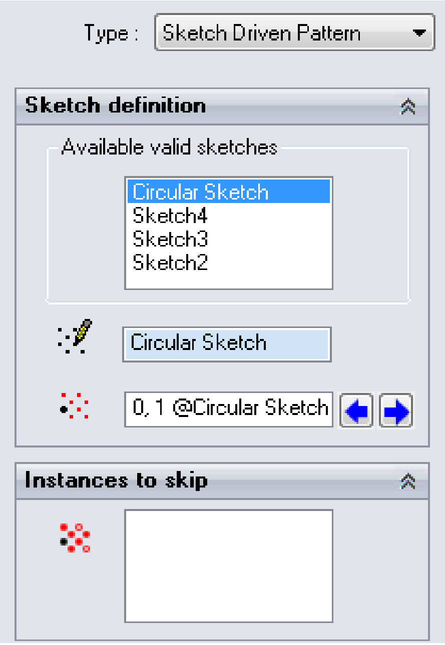
4

Instances to skip

New Pattern dialog box (Irregular Pocket)

Inserting a Sketch-Driven Pattern Feature

1. Right click *Rectangular Pocket1* in the tree and select *Pattern*, then select *Create Pattern* from the context menu.
2. In the *New Pattern* dialog box, select *Sketch Driven Pattern* for the Type.
3. Select *Circular Sketch* in the list of *Available valid sketches*.
4. This box lists 2D sketches that contain arcs, circles or point entities. A 2D or 3D sketch can also be graphically selected. While the sketch can contain any entity type, only the center point of circles, arcs or point entities will be used to define the pattern feature locations. In this tutorial, the 2D sketch named *Circular Sketch* consists of points on each of the pockets.
5. Notice that the pattern is not in the correct position on the part. The point on the pocket at the bottom identifies the point on the sketch that is the reference point for the feature. The default is the first point in the sketch. When this point is not correct, you can change it.
6. Click the *Move to previous* button to select the sketch point that positions the pattern correctly.


Type : Sketch Driven Pattern

Sketch definition

Available valid sketches

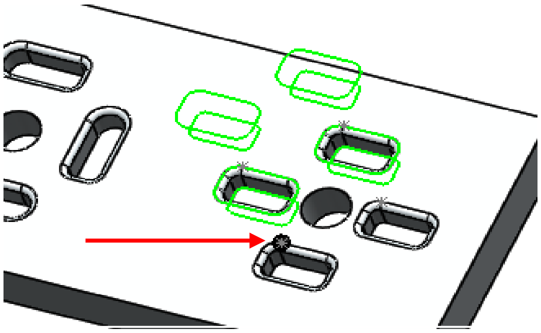
Circular Sketch
Sketch4
Sketch3
Sketch2

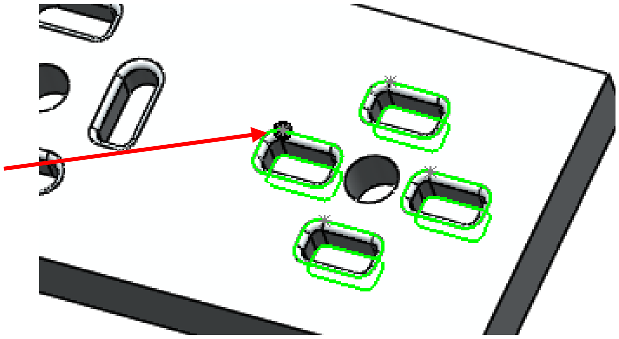
Circular Sketch

0, 1 @Circular Sketch

Instances to skip

New Pattern dialog box (Rectangular Pocket)


Incorrect location of Sketch-Driven pattern


Correct location of Sketch-Driven pattern

7. ✓ Click *OK* to insert the Pattern feature. *Pattern2-Rectangular Pocket1*feature is added to the tree.
8. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

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

   The tree lists the operations that CAMWorks generated for these features.

9. ⊞ Click the plus sign next to each operation in the tree and notice the operations have been generated only for the original features, not each for instance in the Pattern features.
10. Click the *Generate Toolpath* button on the CAMWorks Command Manager.
11. Click the operations to view the toolpaths.

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

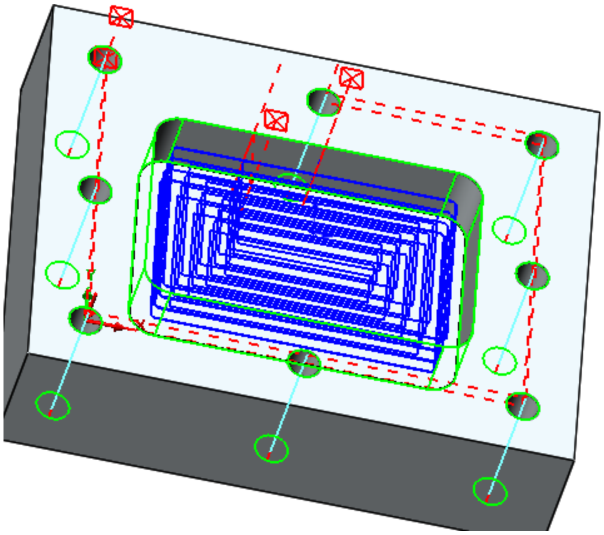
For More Practice

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

Part1 for Practice

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

- Open the part file **MILLP1.SLDPRT** in the *Examples*\ *Tutorial_Parts**Mill* folder.
- Change to the *CAMWorks Feature* tree.
- Use the default ‘Machine [*Mill–metric]’*, post processor and active Tool Crib.
- You can use the default bounding box for the stock.
- Click the *Extract Machinable Features* button on the CAMWorks Command Manager.
- Double click the *Hole Group1* feature. Change the Strategy to *Ream*.
- Click the *Generate Operation Plan* button on the CAMWorks Command Manager.
- Click the *Generate Toolpath* button on the CAMWorks Command Manager.
- Right click on the *Mill Part Setup* and select *Simulate Toolpath* from the context menu. Run the simulation.
- Click the *Post Process* button on the CAMWorks Command Manager and generate the NC code.



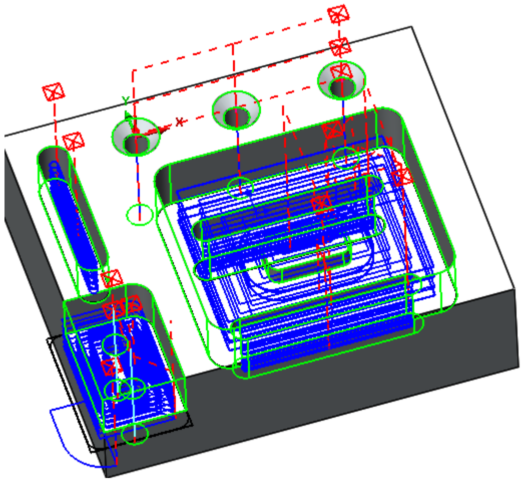
MILLP1.SLDPRT

Part2 for Practice

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

- Open the part file **MILLP2.SLDPRT** in the *Examples*\ *Tutorial_Parts**Mill* folder.
- Change to the *CAMWorks Feature tree*.
- Use the default ‘Machine [*Mill–metric]’* machine, post processor and active Tool Crib.

- Double click *Stock Manager* in the tree. Change the Bounding box offset for X, Y and Z to a uniform **1mm**.
- Click the *Extract Machinable Features* button on the CAMWorks Command Manager.
- Double click *Hole Group2* in the tree. Change the Strategy to *Ream*.
- Click the *Generate Operation Plan* button on the CAMWorks Command Manager
- Using the *Sort Operations* command, reorder to the operations so that the Rough Mill operations are first, followed by the Contour Mill operations and finally the hole operations.



MILLP2.SLDPRT

- Right click on the *Mill Part Setup* and select *Combine Operations* to merge operations that use the same tool.

  In the Combine Operations dialog box, select the *Rough Mill, Contour Mill* and *Center Drill* options.
- Click the *Generate Toolpath* button on the CAMWorks Command Manager.
- Click the *Simulate Toolpath* button on the CAMWorks Command Manager. Make sure the Tool Holder Cut Collision button is selected, then run the simulation. Notice the holder collides with the part.
- Double click *Rough Mill1* in the Operation tree. On the Mill Holder tab, change the Protrusion to **65mm**.

  This parameter defines the distance from the tip of the tool to the bottom of the holder. This value is used to determine how much of the tool protrudes from the mill holder during simulation. The protrusion length must be greater than the Flute length and less than the Overall length.
- Simulate the toolpaths again to see the difference. This time the holder clears the part.
- Click the *Post Process* button on the CAMWorks Command Manager and generate the *NC code*.

Chapter 3: Learning 3 Axis Mill

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

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

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

The tutorials in this chapter are intended to show you how to use CAMWorks for 3 Axis Mill operations and may not correspond to actual machining practices.

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

3 Axis Mill 1

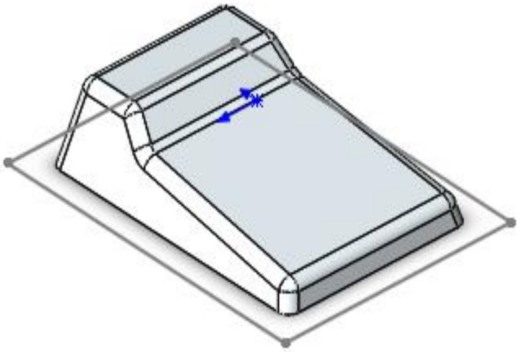
Topics covered in this tutorial

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

Defining the Machine, Stock and Machining Direction

1. Open the part file **MILL3AX_1.SLDPRT** in the following folder.
   Drive:\CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill
2. Click the *CAMWorks Feature Tree* tab.

Define the Machine



MILL3AX_1.SLDPRT

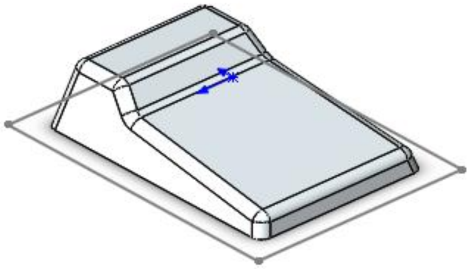
1. Double click *Machine [Mill–inch]* in the Feature tree.
   The Machine dialog box is displayed.
2. On the Machine tab, *Mill–inch* is the Active machine. This machine definition has been created for the CAMWorks exercises. When you use CAMWorks to machine your own parts, select the machine tool you want to use to machine the part.

   Did You Know ...

   In the Feature and Operation trees, instead of right clicking items and selecting Edit Definition, you can double-click the item to open the dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.
3. Click the *Tool Crib* tab and make sure *Crib1* is the Active tool crib.
4. Click the *Post Processor* tab and make sure M3AXIS-TUTORIAL is selected.
   M3AXIS-TUTORIAL is the post processor used for the tutorials in this manual. When you use CAMWorks to machine your own parts, select your machine tool controller or post processor.
5. Click *OK* to close the *Machine* dialog box.

Define the Stock Size and Shape

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



Pick the rectangular sketch

2. In the *Stock Manager* dialog box, click the *Extruded Sketch* button.
3. In the graphics area, pick the rectangular sketch representing the perimeter of the stock.
4. Click the *Reverse Direction* button.
5. Set the Depth to **1.5in.**
6. ✓ Click *OK* to apply the changes and close the dialog box.

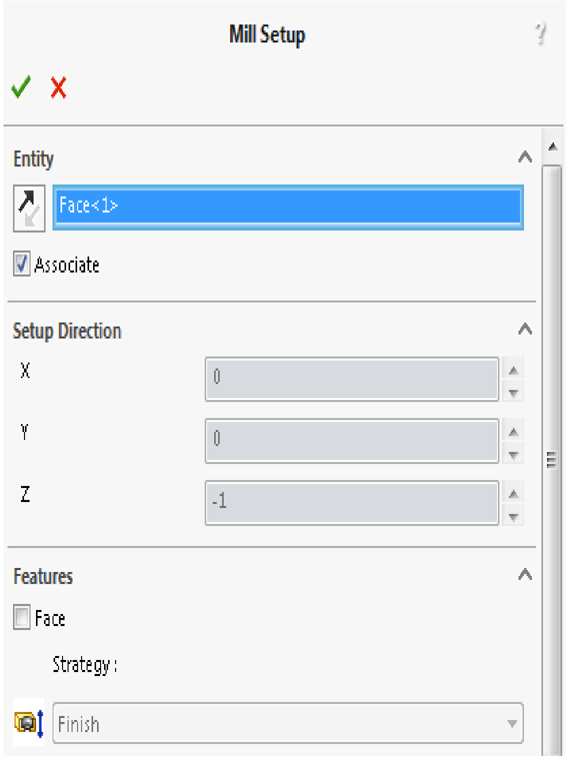
Define the Machining Direction

The Machining direction will be normal to the face indicated below:

1. Right click *Stock Manager* in the Feature tree and select *New Mill Part Setup* on the context menu.

   The *Mill Part Setup* dialog box is displayed.
2. In the graphics area, pick the top face of the part. This action selects the top face as the Selected entity.
3. In the New Features group box, make sure the *Face* option is not checked,
4. ✓ Click *OK* to insert the setup and close the dialog box.

   Mill Part Setup1 is added to the tree.



Mill Setup

Entity

Face<1>

☑ Associate

Setup Direction

X 0

Y 0

Z -1

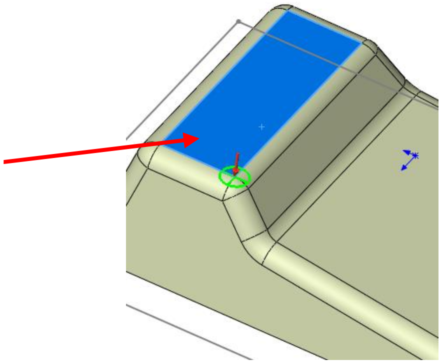
Features

☐ Face

Strategy:

Finish

Interactively inserting a Mill Part Setup



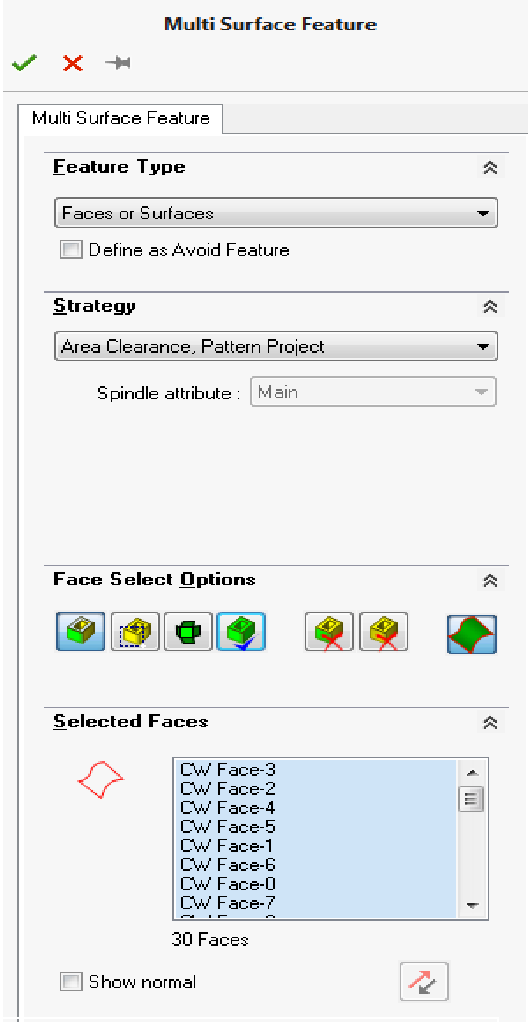
Pick the top face of the part

Creating a Multi Surface Feature

Automatic Feature Recognition (AFR) is used to automatically find 2.5 axis features on the part. In this tutorial, there are no 2.5 axis features on the part. Hence, Automatic Feature

Recognition (AFR) by executing the *Extract Machinable Features* command will not be used. Instead, you will add the Multi Surface feature interactively.

Following are the steps to define a Multi Surface feature that includes the faces on the part to cut.



Faces or Surfaces

☐ Define as Avoid Feature

Area Clearance, Pattern Project

Spindle attribute : Main

CW Face-3
CW Face-2
CW Face-4
CW Face-5
CW Face-1
CW Face-6
CW Face-0
CW Face-7

30 Faces

☐ Show normal

Interactively inserting a Multi Surface Feature

1. Right click *Mill Part Setup1* and select *Multi Surface Feature* on the context menu.

   The *New Multi Surface Feature* dialog box is displayed.

2. In the Face Select Options group box, click the *Select all faces* button.

   This action lists all the selected face in the *Selected Faces* list box.

3. In the Strategy group box, use the dropdown list to set the Strategy to *Area Clearance, Pattern Project*.

   This Strategy is defined in the Technology Database to generate an Area Clearance operation with a Lace Pattern and a Pattern Project operation with a Slice Pattern.

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

   Did You Know ...

   When you define Multi Surface features for 3 Axis operations (except Rough and Finish), the most reliable results are usually obtained by picking all the model faces as Selected Faces. If you need to restrict machining to specific areas, use the In New Contain and New Avoid commands at the operation level. The previously available Faces to Avoid function (also used to restrict machining areas) is no longer supported from CAMWorks 2013 onwards.

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

   The Multi Surface feature is listed in the Feature tree.

   Did You Know ...

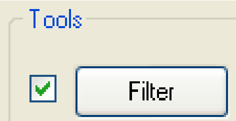
   New features, interactively inserted features that have no operations and features that could not produce operations because the feature conditions have not been defined in the Technology Database display in a different color. You can set the color on the Display tab in the Options dialog box.

Generating an Operation Plan and Modifying Operation Parameters

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

   OR

   Right click *Mill Part Setup1* in the Feature tree and select *Generate Operation Plan* from the context menu

   Tools

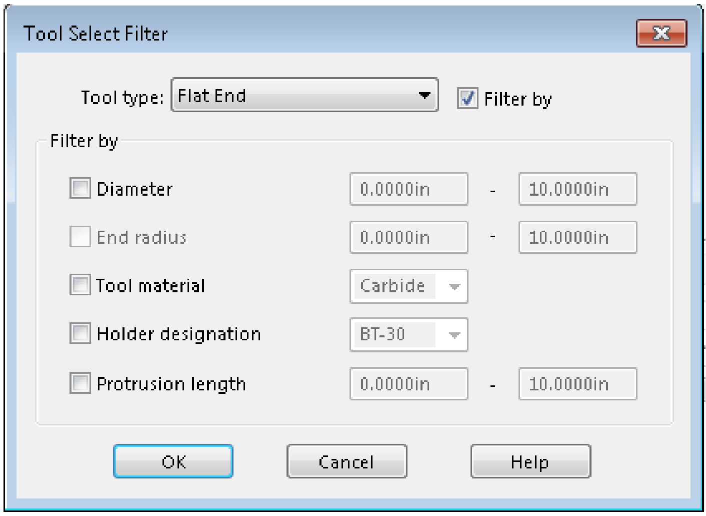
   ☑ Filter

   The Operation tree lists *Area Clearance* and *Pattern Project* operations.
2. Double click *Area Clearance1* in the Operation tree.
3. Click the *Tool tab* and then click on the *Tool Crib* page.
4. Click the check box to the left of the *Filter* button, and then click the *Filter* button.

   The *Tool Select Filter* dialog box is displayed. This dialog box allows you to set filters for displaying and selecting tools.
5. Click the check box for the *Filter* by option to the right of the Tool Type list box.
6. Select *Flat End* for the Tool type and click *OK*.

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

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



Tool Select Filter

Tool type: Flat End ☑ Filter by

Filter by

☐ Diameter 0.0000in - 10.0000in

☐ End radius 0.0000in - 10.0000in

☐ Tool material Carbide

☐ Holder designation BT-30

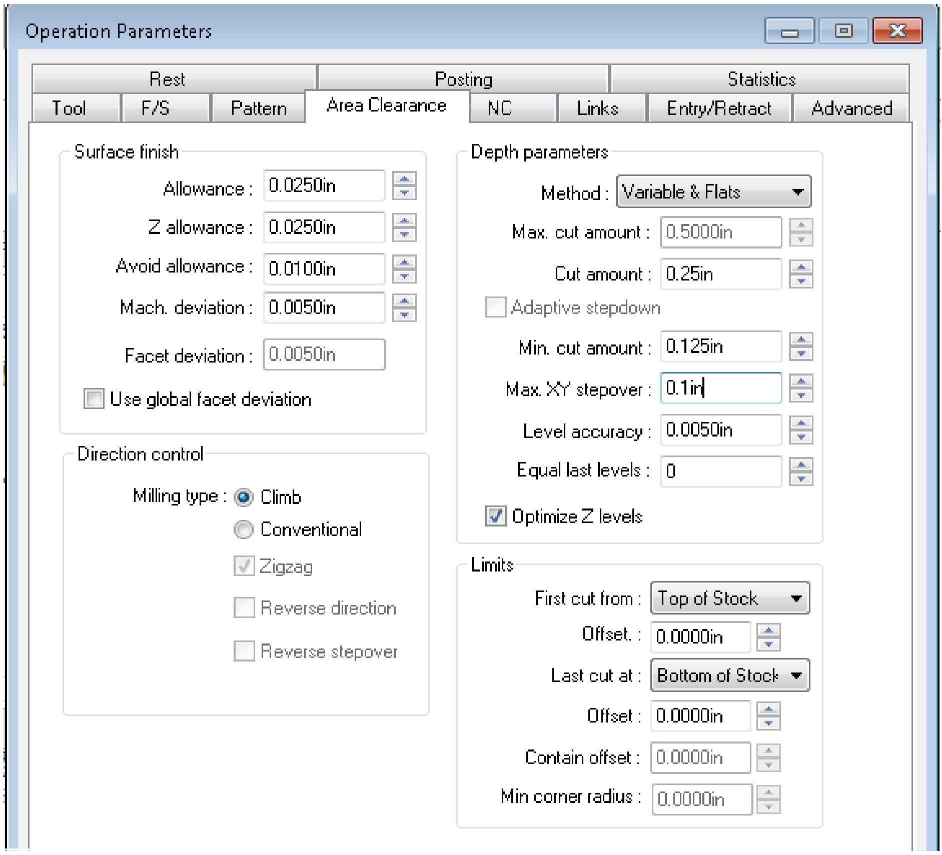
☐ Protrusion length 0.0000in - 10.0000in

Tool Select Filter Dialog box

7. Select a **0.5in** diameter Flat End, and then click the *Select* button.
8. Click *Yes* to replace the corresponding holder.
9. Click the *Area Clearance* tab in the Operation parameters dialog box.
10. In the Surface Finish group box, set the *Mach. deviation* to **0.005in**.

    This parameter defines a tolerance that is used to condense or shorten the toolpath. The smaller the value, the greater the number of linear cuts and the smoother the surface finish.
11. For the Depth parameters Method, select *Scallop*.
12. Leave the *Cut amount* set to **0.25in**.
13. Set the *Min. cut amount* to **0.125in** and the *Max XY stepover* to **0.1in.**

14. Click *OK* to apply the changes and to close the dialog box.
15. If prompted, click *Yes* to regenerate toolpaths.
16. Double click *Pattern Project1* in the Operation tree.
17. On the *Pattern* tab, notice that the Pattern is set to *Slice* as defined for the *Area Clearance, Pattern Project* Strategy in Technology Database. This pattern generates a series of linear parallel cuts across the part and is appropriate for semi-finishing almost any part shape and finishing non-vertical areas.
18. Set the following parameters on the Pattern tab:
    - Cross machining = *Standard* (This option allows you to automatically generate a second perpendicular toolpath on areas that are not machined or on poorly finished faces. The second toolpath is calculated by rotating the previous toolpath 90 degrees. The Standard option generates a perpendicular toolpath after the Slice toolpath.)
    - Extension = **0.1in** (Specifies the length a pass extends past the edge of the boundary to allow the tool to move into the cut at machining feedrate rather than rapid feedrate.)
    - Staydown = Checked (This parameter affects how the tool transitions within contain areas and around avoid areas. When checked, tool retracts are minimized, however, additional feed motion is possible.)
19. On the Finish tab, make sure the XY allowance and Z allowances are set to **0in**. This value represents the amount of material left on the part after generating the toolpath.



Surface finish

Allowance : 0.0250in

Z allowance : 0.0250in

Avoid allowance : 0.0100in

Mach. deviation : 0.0050in

Facet deviation : 0.0050in

☐ Use global facet deviation

Direction control

Milling type : ● Climb

○ Conventional

☑ Zigzag

☐ Reverse direction

☐ Reverse stepover

Depth parameters

Method : Variable & Flats

Max. cut amount : 0.5000in

Cut amount : 0.25in

☐ Adaptive stepdown

Min. cut amount : 0.125in

Max. XY stepover : 0.1in

Level accuracy : 0.0050in

Equal last levels : 0

☑ Optimize Z levels

Limits

First cut from : Top of Stock

Offset. : 0.0000in

Last cut at : Bottom of Stock

Offset : 0.0000in

Contain offset : 0.0000in

Min corner radius : 0.0000in

Setting Area Clearance Parameters

20. Set the *Mach. deviation* to **0.005in.**
21. On the *Entry/Retract* tab, use *Skim* for the Retract Method.

    With this method, the tool rapids vertically to the minimum Z height needed to clear the faces in the feature, moves horizontally, then feeds vertically down to the start of the next pass. The minimum height of Retract is determined by the Skim clearance parameter.
22. Click *OK* to apply the changes and close the dialog box.

23. If prompted, click *Yes* to regenerate toolpaths

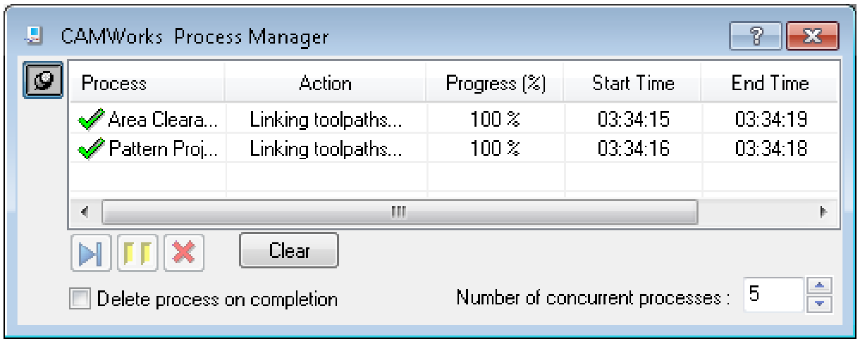
Generating Toolpaths and Post Processing

Generate toolpaths, simulate material removal and post process:

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

   The Process Manager window displays automatically when Generate Toolpath command is executed. The Process Manager provides the ability to generate 3 and 4/5 axis simultaneous toolpaths as separate threads and in separate processes. Running as a separate process allows you to continue to use CAMWorks while toolpaths are generating. Running in a separate process reduces the amount of memory needed to run inside SOLIDWORKS/CAMWorks Solids.

   If the Number of concurrent processes option is set to 2 or more, notice that both operations are processed simultaneously. The window provides feedback on each process and green checkmarks indicate the processes have completed.



CAMWorks Process Manager

Process	Action	Progress (%)	Start Time	End Time
✓ Area Cleara...	Linking toolpaths...	100 %	03:34:15	03:34:19
✓ Pattern Proj...	Linking toolpaths...	100 %	03:34:16	03:34:18

☐ Delete process on completion

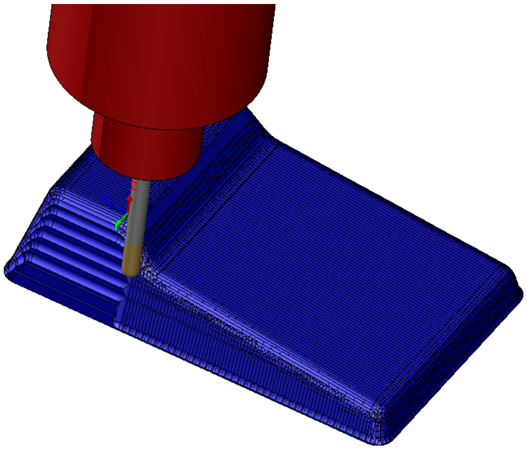
Number of concurrent processes : 5

CAMWorks Process Manager

2. Click each operation in the tree to view the toolpath.
3. Right click *Mill Part Setup1* and select *Simulate Toolpath* from the context menu.
4. Click the *Turbo mode* button on the *Simulate Toolpath* toolbar.

   Turbo mode provides ultra-fast simulation of toolpaths on a single Mill Part Setup.

5. Click *Run*.
6. Click the *Ok* button to exit the simulation.
7. If you were generating the NC program for the toolpaths, the last step would be to click the *Post Process* button on the CAMWorks Command Manager and post process the part.



Simulate Toolpath

3 Axis Mill 2

Topics covered in this tutorial:

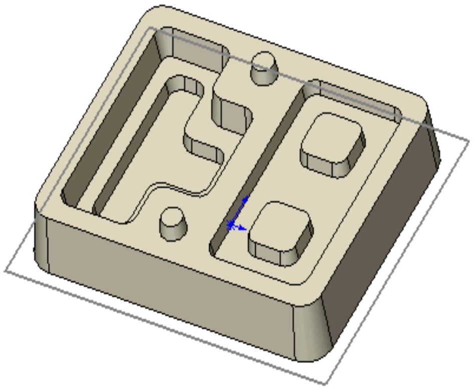
- Inserting a Mill Part Setup & creating a Multi Surface Feature
- Generating Area Clearance toolpaths
- Using Contain area to selectively machine within specified areas
- Generating and Z Level toolpaths
- Using Avoid Area to selectively machine within specified areas
- Defining user-defined limits to selectively machine specific areas
- Generating Flat Area toolpaths

Open the Part and Defining the Machine and Stock

1. Open the part file **MILL3AX_2.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.



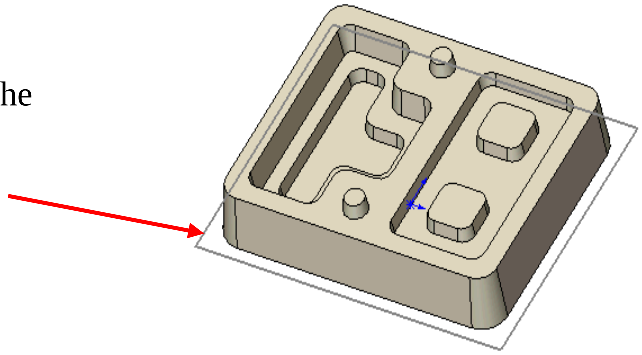
MILL3AX_2.SLDPRT

Define the Machine

1. Double click *Machine [Mill-inch]* in the Feature tree to open the Machine dialog box.
2. On the Machine tab, *Mill–inch* is the Active machine.
3. Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.
4. Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
5. Click *OK* to close the Machine dialog box.

Define the Stock

1. Double click *Stock Manager* in the tree.
2. In the Stock Manager dialog box, click the *Extruded Sketch* button.
3. In the graphics area, pick the rectangular sketch that represents the stock. This action selects the sketch to the *Selected Entity* field.



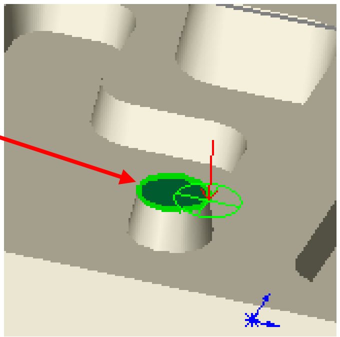
4. Click the *Reverse Direction* button.
5. Set the Depth to **1.3in**, and then click *OK* to apply the changes and close the dialog box.

Inserting a Mill Part Setup and Creating a Multi Surface Feature

Steps to insert the Mill Part Setup and create a Multi Surface Feature:

1. Right click the *Stock Manager* and select *Mill Part Setup* from the context menu.

   The *Mill Setup* dialog box is dispalyed.



Pick the top face of the cone

2. In the graphics area, pick the top of one of the rounds cone as shown in the figure.

   This action selects the face to the *Selected entity* field.
3. Make sure the *Face option* under Features group box is not checked.
4. ✓ Click *OK* to close the dialog box.

   The Mill Part Setup1 is inserted in the Feature tree.
5. Right click the *Mill Part Setup1* in the tree and select *Multi Surface Feature* from the context menu.

   The *Multi Surface Feature* dialog box will be displayed.
6. In the Strategy dropdown list, select *Area Clearance, Z Level*.
7. Under Face Select Options, click the *Select all faces* button.
8. ✓ Click *OK* to insert the feature.

   The *Multi Surface Feature1* is now listed in the Feature tree.

   Did You Know ...

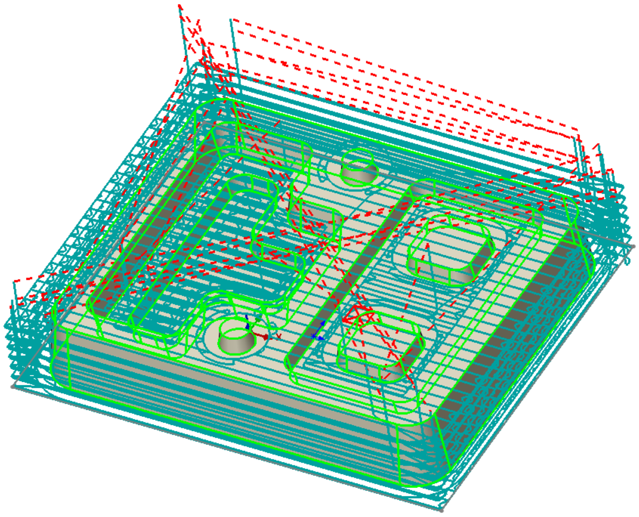
   When you define Multi Surface features for 3 Axis operations (except 3 Axis Rough and Finish), the most reliable results are usually obtained by picking all model faces as Faces to Cut. Using Faces to Avoid is not recommended. If you need to restrict machining to specific areas, use the New Contain and New Avoid commands at the operation level.

Generating Area Clearance Toolpaths

Generate Operations and Toolpaths

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.
2. Double click *Area Clearance1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
3. Click the *Tool* tab, then select the *Tool crib* page.
4. Highlight a **0.375in** Flat End mill and click the *Select* button.
5. Click *Yes* to replace the corresponding holder.
6. On the Pattern tab, set the Pattern type to *Lace*.
7. On the Area Clearance tab, within the Depth Parameters group box:

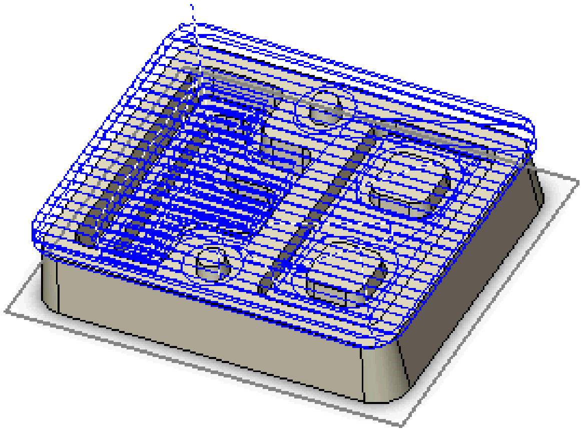


Toolpath generated for Area Clearance operation

  - Set the Method to *Constant*
  - Set the Cut amount to **0.2in**.
8. On the Links tab:
   - In the Depth Processing group box, select the option *To depth by region* for the Depth processing.
   - In the Corners group box, select *Round* for Type and set the Max radius and Part deviation to **0.0375in**.
9. On the Entry/Retract tab:
   - Select *Skim* for the Retract Method
   - Set the Max. Ramp angle to **20deg** in the Ramping group box.
10. Click *OK* to apply the changes and close the Operation Parameters dialog box.
11. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

Inserting Contain Areas to Selectively Machine Areas

Depending on the machining requirements, you may want to insert a contain area to rough only the top of the part:



Contain area to rough only the top of the part

1. Double click *Area Clearance1* in the tree.
2. On the Advanced tab, change the Automatic contain area Method to *Outer Silhouette*.

   With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature.
3. Click *OK* to apply the changes and close the dialog box.

   Be default, the *Update 3 axis toolpaths when parameters change* option on the Update tab in the Options dialog box is set to *Prompt*. When this option is set to Prompt, if you click the *OK* button of any Operation Parameters dialog box after changing its parameters, then a message is displayed asking if you want to regenerate toolpaths. Click *Yes*. If the option in the Options dialog box is set to *Always*, CAMWorks automatically regenerates the toolpaths. If set to *Never*, then you need to manually execute the *Generate Toolpath* command on the context menu.
4. Click *Yes* when prompted to regenerate the toolpaths. If you click no, the toolpaths will be generated when you execute the *Generate Toolpath* command .

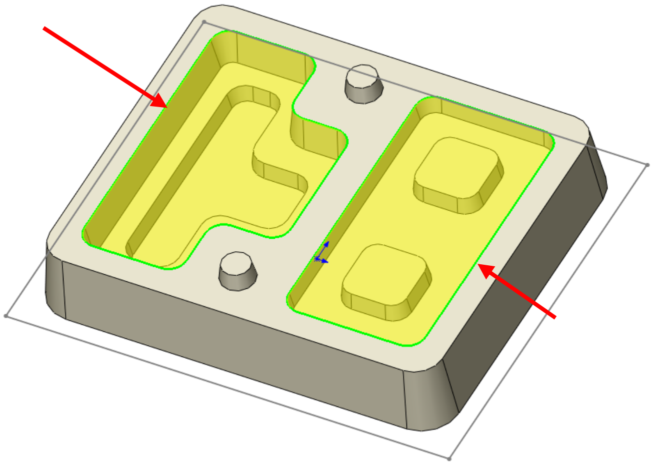
Did You Know ...

For 3 Axis operations (except 3 Axis Rough and Finish), the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either automatically on the Advanced tab in the Operation Parameters dialog box or interactively using the New Contain Area command.

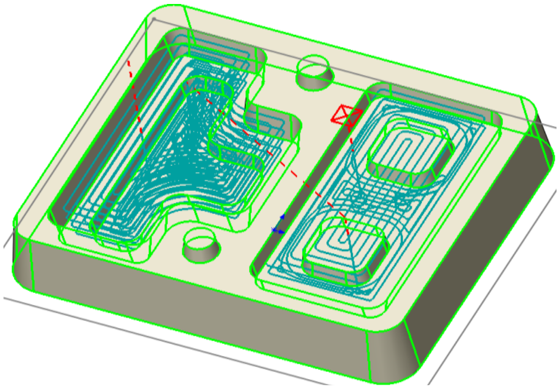
The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or not defined, then an automatically generated XY contain area will be used.

Steps to rough machine only the cavities of the part:

You may want to rough only the cavities. Following are the steps:



Selecting Edges of the cavities

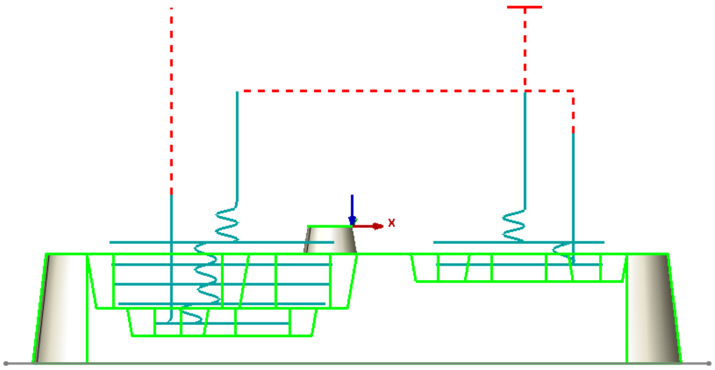


Contain area toolpaths to machine only the cavities of the part

1. Right click *Area Clearance1* in the tree and select *New Contain Area* from the context menu.
2. The Contain Area is displayed. In the Selection Mode group box, set the Edge Selection to *Constant Depth Loop*.
3. Click in the *Selected Contours* list box to set the focus.
4. In the graphics area, pick a top edge on each cavity as shown in the image on right.

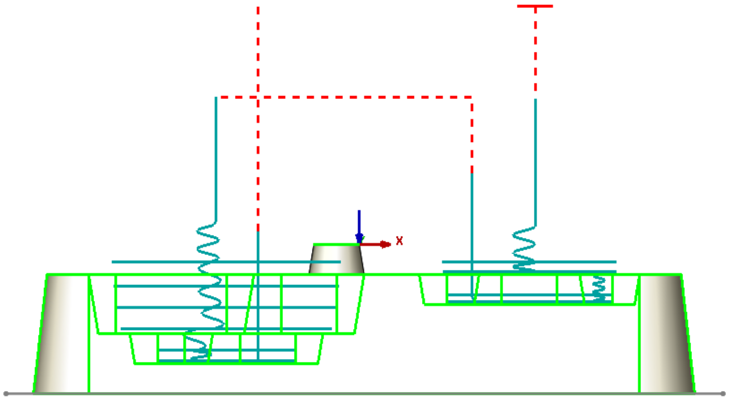
   Loop<1> and *Loop<2>* are displayed in the list box. CAMWorks automatically chain selects all other edges connected to that edge at a constant depth.
5. Click *Multiple* to set the Shape.
6. Click *OK* to insert the Contain area.
7. Click *Yes*, if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
8. Double click *Area Clearance 1* in the tree. The Operation Parameters dialog box is displayed.
9. On the Pattern tab, change the Pattern Type to *Pocket Out*.
10. On the Links tab, in the Corners group box, set the Offset deviation to **0.0375in**.
11. Click *OK* to apply the changes and close the Operation Parameters dialog box.
12. Click *Yes*, if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

13. Right click *Mill Part Setup1* in the tree and select *Set View*, then pick *Setup XZ* in the cascading menu.
14. The view of the part in the graphics area changes. Click the *Area Clearance1* operation in the tree and notice the amount of material at the bottom of the cavities.
15. Double click *Area Clearance1* in the Operation Tree.



Amount of material left in the cavities

16. On the Area Clearance tab, change the Depth Parameters Method to *Scallop*, set the Min. cut amount to **0.05in**.
17. Click *OK* to apply the changes and close the Operation Parameters dialog box.
18. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
19. Notice the additional levels with respect to the previous settings.



Additional levels of cuts in the toolpath using the Scallop method

Generating Z Level Toolpaths

The Z-Level cycle is a surface finish contouring cycle that removes material by making a series of horizontal planar cuts. The cuts follow the contour of the surface at different levels. Cutting starts from the highest location on the model and works down. Z-Level has unique characteristics that make it appropriate for steep walled areas. This cycle is generally not used on parts with flat or near flat faces.

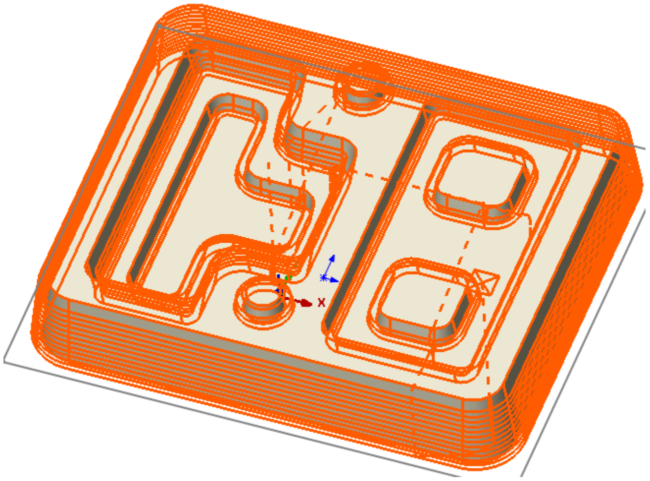
Steps to insert a Z-level operation:

1. Right click *Area Clearance1* in the tree and select *New 3 Axis Mill Operation* from the context menu. Select *Z Level* from the cascading menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.
3. Make sure the *Edit Operation* on creation option is checked.
4. ✓ Click *OK* to insert the operation.

   Since the *Edit operation on creation* option was checked, the Operation Parameters dialog box for the newly inserted *Z Level* operation is displayed.
5. Click the *Tool* tab and then on the *Tool Crib* page. Highlight a **0.25in** Ball Nose tool, then click the *Select* button.

If a Ball Nose tool of this diameter is not available, click on the *Add* button and use the Tools Database form to add the requisite tool to the Tool Crib.

6. Click *Yes* to replace the corresponding holder.
7. On the Z Level tab, in the Depth Parameters group box, set the Cut amount to **0.08in**.
8. On the Links tab, make sure that the *Depth processing* option is set to *To depth by region*.
9. On the Entry/Retract tab, use *Skim* for the *Retract Method*. In the *Ramping* group box, and set the *Max. ramp angle* to **20deg**.



Toolpaths for Z Level operation

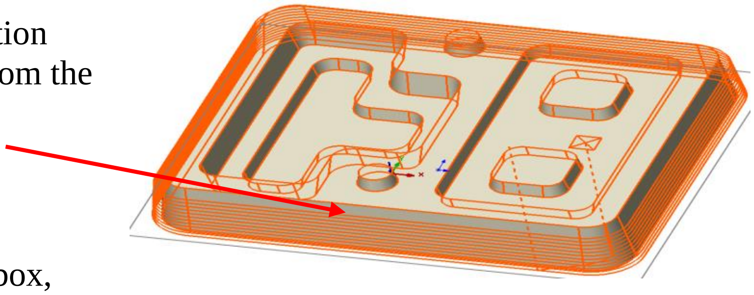
10. Click *OK* to apply the changes and close the dialog box.
11. ✖ Click *Cancel* to close the *New Operation* dialog box.
12. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
13. If CAMWorks didn’t display a message prompting you to regenerate the toolpaths (Step 12), then right click *Z Level1* in the Operation tree and select *Generate Toolpath* from the context menu .

Inserting an Avoid Area to Selectively Machine Areas

Depending on machining requirements, you may want to cut around only the outside of the part.

1. Right click *Z Level1* in the Operation tree and select *New Avoid Area* from the context menu.

   The *Avoid Area* dialog box is displayed.
2. In the *Selection Mode* group box, set the *Face Selection* to *Single Face*.



Toolpath for Z Level operation with Avoid Area defined

3. Click in the *Selected Contours* list box to set the focus. Pick the top face of the part.

   This action lists the top face in the *Selected Contours* list box.
4. ✓ Click *OK* to insert the Avoid Area.

5. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
6. If such a warning is not displayed, the right click *Z Level1* in the Operation tree and select *Generate Toolpath* from the context menu .

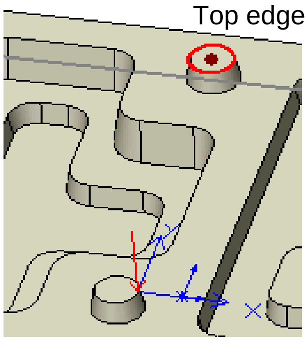
Using User-defined Limits to Machine Specific Areas

Following are the steps to insert a Z Level operation and limit the toolpath to cut around the cones with a tapered tool:

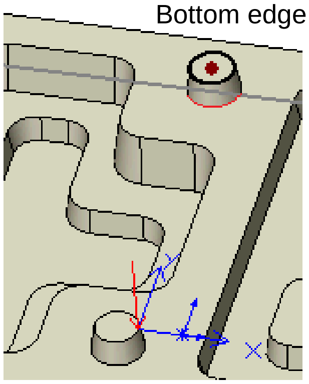
1. Right click *Z Level1* in the tree and select *New 3 Axis Mill Operation* from the context menu. Select *Z Level* from the cascading menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.
3. Remove the check mark from the *Use TechDB Defaults* option. In the Copy from dropdown list, select *Z Level1*.
4. Make sure the *Edit Operation on creation* option is checked.
5. ✓ Click *OK* to insert the operation.

   The Operation Parameters dialog box for the newly inserted operation is displayed.
6. Click on the *Tool* tab and then click on the *Tool Crib* page.
7. Pick a **0.25in** Flat End Tapered tool and click the *Select* button.
8. Click *Yes* to replace the corresponding holder.

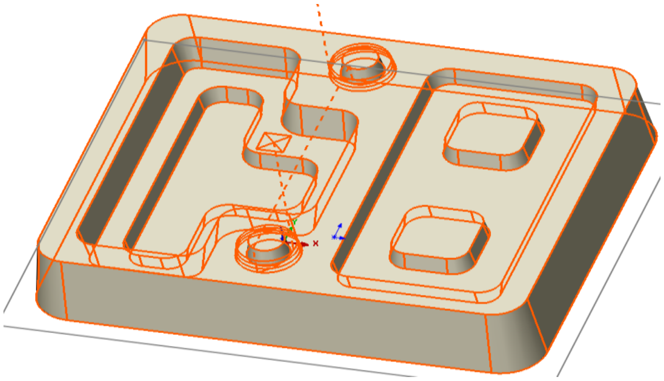
   If a Flat End Tapered tool of this diameter is not available, click on the *Add* button and use the Tools Database form to add the requisite tool to the Tool Crib.
9. On the Tapered Tool page, make sure the Taper angle is **10deg**.
10. Click the *Z Level* tab.
11. In the Limits group box, for the First cut from option, select *User Defined*.
12. Click the *Pick Point* button.
13. The Define Point dialog box is displayed. Pick the top edge of one of the cones, then click *OK* to close the dialog box.
14. In the Limits group box, for the Last Cut at option, select *User Defined*.
15. Click the *Pick Point* button.
16. The *Define Point* dialog box is displayed. Pick the bottom edge of the cone and then click *OK* to close the dialog box.
17. Click *OK* to close the *Operation Parameters* dialog box.



Pick the top edge of the cone



Pick the bottom edge of the cone



Z level toolpath to cut around the cones with a tapered tool

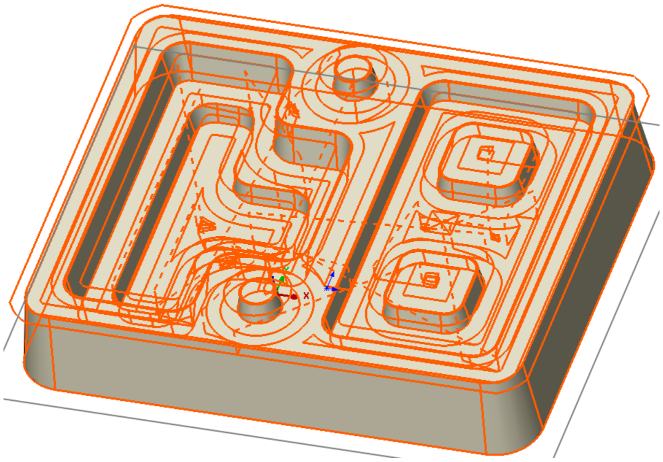
You do not need to change any other parameters since you copied the parameters from the previous operation.

18. ✕ Click *Cancel* to close the *New Operation* dialog box.
19. Observe that the toolpath was not generated for the *Z Level2* operation (indicated by the still magenta color of the *Z Level2* item in the operation tree). This is because when the parameters of *Z Level1* operation was copied to *Z Level2*, the contain and avoid areas associated with that operation were also copied.
20. Left click the *plus sign* ⊞ next to Z Level2 in the tree.
21. Right click *Avoid Area2*, select *Delete* on the context menu. Click *Yes* to confirm.
22. Right click *Z Level2* in the Operation tree and select *Generate Toolpath* from the context menu. The toolpath for this operation is successfully generated. Only the area around the cones is cut.
23. Right click *Z Level2* in the tree and select *Simulate Toolpath* on the context menu.
24. ▶ Click *Run*. The tapered tool cuts around the cones.
25. ✓ Click the *Ok* button to exit the simulation.

Generating Flat Area Toolpaths

You can use a Flat Area operation to machine the flat areas on this part. The Flat Area cycle uses a roughing pattern to remove material on the Z-levels of flat faces between the top and bottom of the surface. CAMWorks generates toolpaths on completely flat areas.

Note that you could also interactively define 2.5 Axis features for the flat areas, then generate 2 Axis operations.



Toolpath for Flat area operation

Steps to insert a Flat Area operation:

1. Right click *Z Level2* in the Operation tree and select *New 3 Axis Mill Operation* from the context menu. Select *Flat Area* from the cascading menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.
3. Make sure the *Edit Operation* on creation option is checked.
4. ✓ Click *OK* to insert the operation.
5. The Operation Parameters dialog box for the newly inserted operation is displayed. Click the *Tool* tab in this dialog box and then select the *Tool Crib* page.
6. Highlight **0.375in** Flat End tool in the Tool crib and click the *Select* button.
7. Click *Yes* to replace the corresponding holder.
8. On the Links tab, for the Depth processing, make sure that the *To depth by region* option is selected.
9. On the Entry/Retract tab, use *Skim* for the Retract Method, use *Ramp* for the Ramping Method, set the Max. Ramp angle to **20deg**.
10. Click *OK* to apply the changes and close the Operation Parameters dialog box.
11. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
12. ✗ Click *Cancel* to close the *New Operation* dialog box.
13. If CAMWorks didn't display a message prompting you to regenerate the toolpaths (Step 12), then right click *Flat Area1* in the Operation tree and select *Generate Toolpath* from the context menu . CAMWorks generates toolpaths on the flat areas.

3 Axis Mill 3

CAMWorks allows parts to be machined using any combination of 2 and 3 Axis machining operations.

Topics covered in this tutorial:

- Recognizing 2.5 Axis Features
- Creating Multi Surface features and operations for the same part
- Inserting a contain area to control the area machined on the Multi Surface Feature
- Generating Pattern Project toolpaths using the Cross machining option
- Inserting a Pencil Mill operation

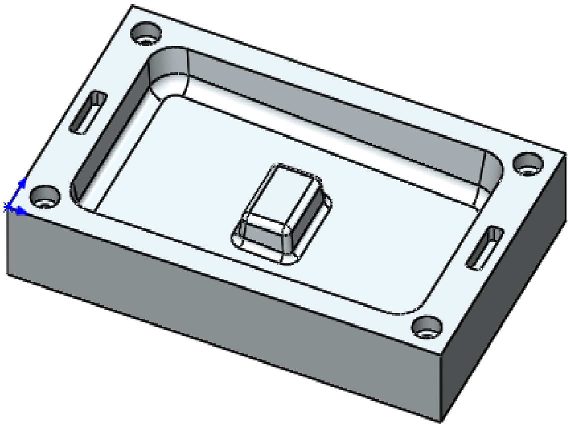
Opening the Part, Defining the Machine and Stock

1. Open the part **MILL3AX_3.SLDPRT** in the following folder.

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

2. Click on the *CAMWorks Feature Tree* tab.

   This part needs to be machined with a combination of both 2 axis and 3 axis machining operations. The 2.5 axis features include the counterbored holes and the two pockets on either end of the main cavity. The cavity itself has tapered walls as well as a tapered bottom, which necessitates a 3 axis machining operation.



MILL3AX_3.SLDPRT

3. Double click *Machine [Mill–inch]* in the Feature tree to open the *Machine* dialog box.
   - On the *Machine* tab, ensure *Mill–inch* is the Active machine.
   - Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected, then click *OK* to close the *Machine* dialog box.
4. Double click *Stock Manager* in the tree. The Stock Manager dialog box is displayed.
   - Leave the Stock Type set to the default Bounding box.
   - Click *OK* to use the default stock for this tutorial.

Automatically Recognizing 2.5 Axis Features

When a part contains 2.5 axis features, normally the next step is to allow CAMWorks to identify features automatically using Automatic Feature Recognition. Then, if AFR does not recognize features that you are certain could be machined with 2 axis cutting cycles, you can define the features interactively with the *New* 2.5 Axis Features command.

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

OR

Right click *CAMWorks NC Manager* and select *Extract Machinable Features* from the context menu.

CAMWorks finds two features: a Counterbore Hole Group that includes the four counterbore holes and a Rectangular Pocket Group that includes the two rectangles at the ends of the cavity.

Creating Multi Surface Feature and Generating Operations

The cavity has tapered walls as well as a tapered bottom, which necessitates a 3 axis machining. Following are the steps to interactively insert a Multi Surface feature for the cavity:

1. In the Feature tree, right click *Mill Part Setup1* and select *New Multi Surface Feature* on the context menu.

   The *New Multi Surface Feature* dialog box is displayed. The set of faces to cut for a Multi Surface Feature should be selected to the Selected Faces list box. Optionally, a Multi Surface Feature being defined can be defined as an avoid feature.
2. For the Feature Type, select *All Displayed* from the dropdown list.
3. Set the Strategy to *Area Clearance, Pattern Project.*

   Did You Know ...

   The Faces to Avoid function is no longer supported from CAMWorks 2013. Instead, you can use the Define as Avoid Feature option to classify such features. If you need to restrict machining to specific areas, use the New Contain area and New Avoid Area commands at the operation level.
4. ✓ Click *OK* to insert the feature and then click ✗ to close the *New Multi Surface Feature* dialog box.

   A Multi Surface feature is created and *Multi Surface Feature1* is added to the Feature tree.
5. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *Mill Part Setup1* and select *Generate Operation Plan* from the context menu.

Modifying Operation Parameters and Inserting Contain Areas

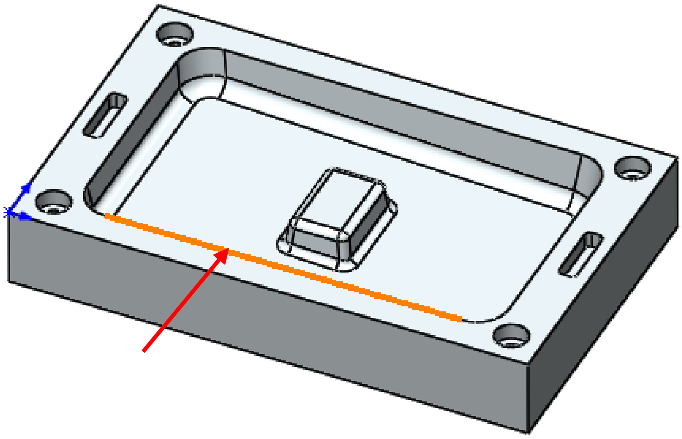
Before generating toolpaths for the Multi Surface feature, you must change the machining parameters for this feature by creating a contain area to machine only the cavity:

1. Right click *Pattern Project1* in the Operation tree and select *New Contain Area* on the context menu.

   The Contain Area dialog box is displayed.

2. In the *Selection Mode* group box, set the *Edge Selection* to *Constant Depth Loop*.
3. Pick an edge around the top of the cavity.

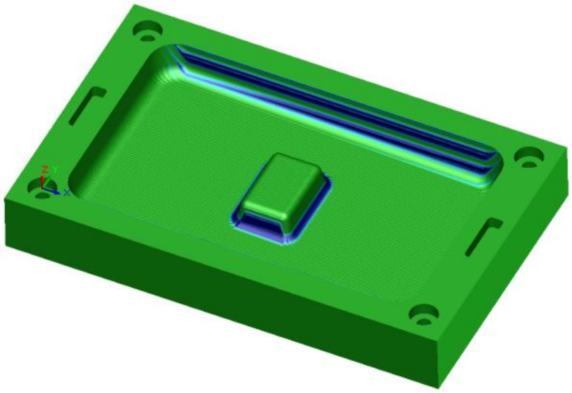
   The *Selected Contours* list displays *Loop <1>*.



Defining a Constant Depth Loop by selecting an edge

4. ✓ Click the *OK* to insert the contain area and close the dialog box.
5. Double click *Area Clearance1* in the Operation tree to open its *Operation Parameters* dialog box.
6. On the *Tool* tab, select the *Tool Crib* page. Highlight a **0.1875in** Flat End mill in the *Tool crib* and click the *Select* button.
7. Click *Yes* to replace the corresponding holder. Click *OK* to apply the changes and close the *Operation Parameters* dialog box. Click *Yes* if prompted to regenerate the toolpaths.
8. Double click *Rough Mill1* (operation for Rectangular Pocket Group) to open its Operation Parameters dialog box.
8. On the *Tool* tab, select the *Tool Crib* page. Highlight a **0.1875in** Flat End mill in the *Tool crib* and click the *Select* button.
9. Click *Yes* to replace the corresponding holder. Click *OK* to apply the changes and close the *Operation Parameters* dialog box.
10. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   Toolpaths are generated for all the operations.



Amount of Material leftover indicated during Toolpath Simulation

Simulating Toolpaths

1. Click the *Simulate Toolpath* button on

the CAMWorks Command Manager.

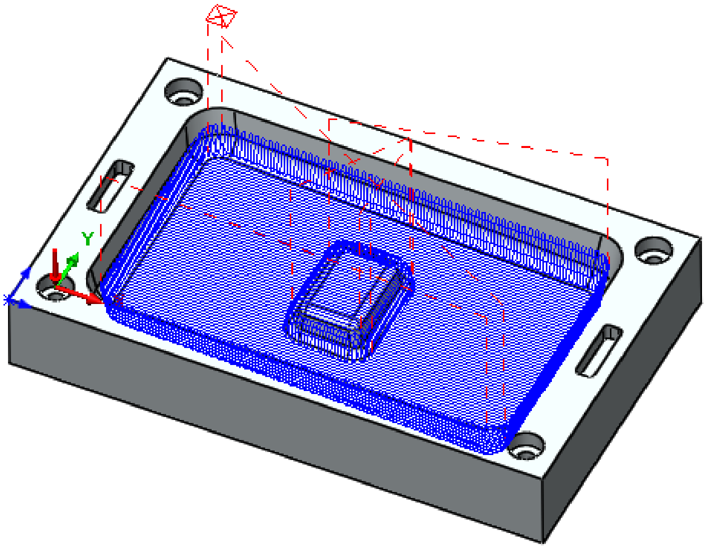
2. Set the End condition to *Next Operation*.
3. Click the *Turbo* button and then ▶ click Run to view each toolpath simulation.
4. Click the *Show Difference* button and notice the amount of material left on the tapered areas of the cavity and around the boss.
5. Click ✓ *Ok* button and Exit the simulation.

Generating Toolpaths Using Cross Machining

1. Double click *Pattern Project1* in the Operation tree.
2. On the Pattern tab of the Operation Parameters dialog box, in the *Pattern Options* group box, set the *Cross machining* parameter to *Standard* from the dropdown list.

   When *Slice* is selected for the Pattern Type, Cross machining allows you to automatically generate a second perpendicular toolpath on areas that are not machined or on poorly finished faces. The second toolpath is calculated by rotating the previous toolpath 90 degrees. When *Standard* is selected as the Cross Machining parameter, a perpendicular toolpath is generated after the *Slice* toolpath.
3. In the *Pattern options* group box, place a check in the *Staydown* option to select it.

   This option affects how the tool transitions within contain areas and around avoid areas. When *Staydown* is selected, tool retracts are minimized but additional feed motion is possible.
4. Click *OK*.
5. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths



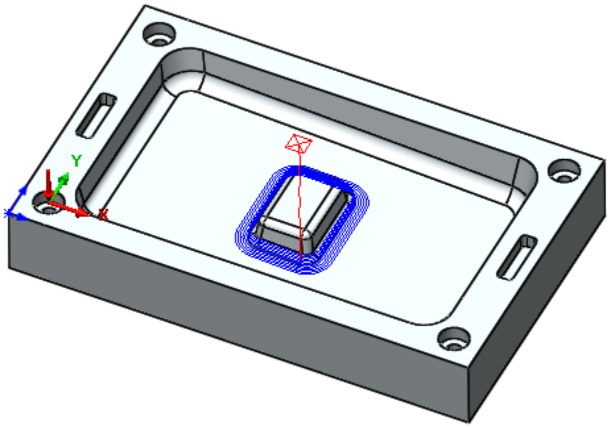
'Slice' type Pattern Project Toolpath using Standard Cross Machining option

6. If such a warning is not displayed, right click *Pattern Project1*and select *Generate Toolpath* from the context menu.

   The tapered areas of the cavity are machined by the cross machining toolpath.

Inserting an Pencil Mill Operation

1. Right click *Pattern Project1* in the tree and select *New 3 Axis Mill Operations*. Select *Pencil Mill* from the cascading menu. The *New Operation* dialog box is displayed.
2. Highlight *Multi Surface Feature1* in the Features list.
3. Ensure that the *Edit operation on creation* option is checked.
4. ✓ Click *OK* to insert the operation. The Operation Parameters dialog box is displayed.
5. Click on the *Tool* tab.
6. On the *Tool Crib* page, select the **0.25in** Ball Nose tool from the Tool Crib. Click *Yes* to change the corresponding holder.
7. On the Pattern tab, set the Pencil mill type Pattern to *Parallel Lace*.
8. On the Links tab, set the *Max. link distance* to **0.25in**, then click *OK* to apply the changes and close the dialog box.
9. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
10. Right click *Pencil Mill1* in the Operation tree and select *New Contain Area* from the context menu.
11. In the Contain Area dialog box, within the *Selection Mode* group box, set the Edge Selection to *Constant Depth Loop*.
12. Pick a bottom face of the cavity. *The faces* will be displayed in the *Selected Contours* list box.



Pencil Mill Toolpath

13. ✓ Click *OK* to insert the contain area.
14. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths
15. If such a warning is not displayed, right click *Pencil Mill1* and select *Generate Toolpath* from the context menu.

    The toolpath cuts the area around the boss.
16. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.
17. Set the End condition to *Next Operation*.
18. Click the *Turbo* button, then ▶ click *Run* to view each toolpath simulation.
19. Click the *Show Difference* button and notice that all the material is removed.
20. Exit the simulation.

3 Axis Mill 4

Topics covered in this tutorial:

- Using the CAMWorks Workflow Toolbar
- Defining the Machining Direction
- Creating a Multi Surface Feature
- Defining program zero
- Editing the Constant Stepover operation to define a contain area
- Generating Pattern Project containing contain and avoid areas
- Generating Pencil Mill toolpaths containing avoid areas

The CAMWorks Workflow Toolbar

The CAMWorks Workflow toolbar provides access to all the major commands of CAMWorks found on the CAMWorks menu. This toolbar is active by default. Click on the *CAMWorks 20xx-WorkFlow* tab in the graphics area to use this toolbar. When executed in sequential order, the commands in this toolbar define the workflow of CAMWorks, irrespective of the machine definition.

Using this comprehensive toolbar which contains the most frequently used commands eliminates the need to use context menus.

In this tutorial, a majority of the commands will be executed from the CAMWorks Workflow toolbar and CAMWorks Command Manager.

Defining the Machine and Machining Direction

1. Open the part file **MILL3AX_4.SLDPRT** in the following folder.

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

   This part is an imported surface model. The rectangular shape of the part creates a bounding box material block that works well for machining the part.

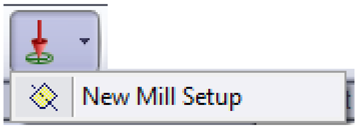
2. Click the *CAMWorks Feature Tree* tab.

3. Click on the *CAMWorks 201x Workflow* toolbar. Click on the *Define Machine* button. The Machine dialog box is displayed.
   - On the Machine tab, ensure *Mill–inch* is the Active machine.
   - Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.
   - Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.
   - Click *OK* to close the Machine dialog box.

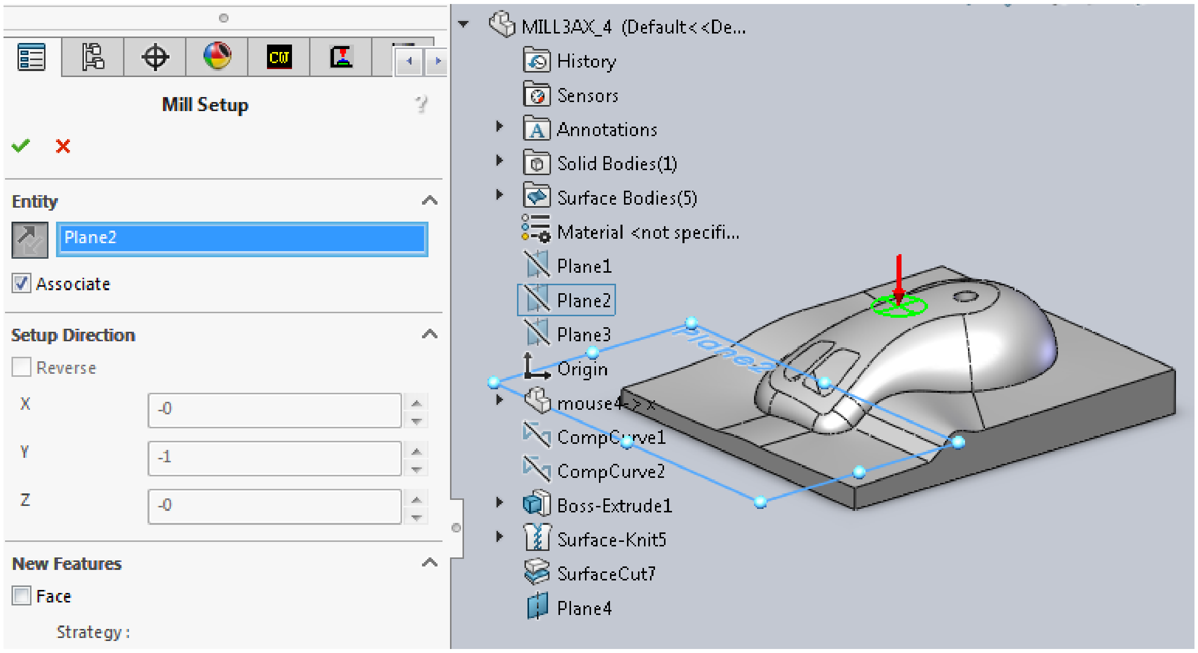
4. Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   The *Stock Manage*r dialog box is displayed.

- Leave the Stock type set as default Bounding box.
- ✓ Click *OK* to use the default stock.



5. Click the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown list.


Mill Setup

Entity: Plane2

☑ Associate

Setup Direction

☐ Reverse

X: -0

Y: -1

Z: -0

New Features

☐ Face

Strategy :

Interactively Inserting a Mill Part Setup

- In the graphics area, click on the Mill3AX_4 to expand the tree and select *Plane4*.
- Click the *Reverse Selected Entity* button to flip the Z axis up.
- ✓ Click *OK* to insert the Mill Setup

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

Creating a Multi Surface Feature

In this tutorial, AFR using the *Extract Machinable Features* command will not be used since there are no 2.5 axis features on the part for AFR to find.

In the absence of any 2.5 axis features, you will add the Multi Surface feature interactively. Following are the steps to interactively insert a Multi Surface Feature:

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

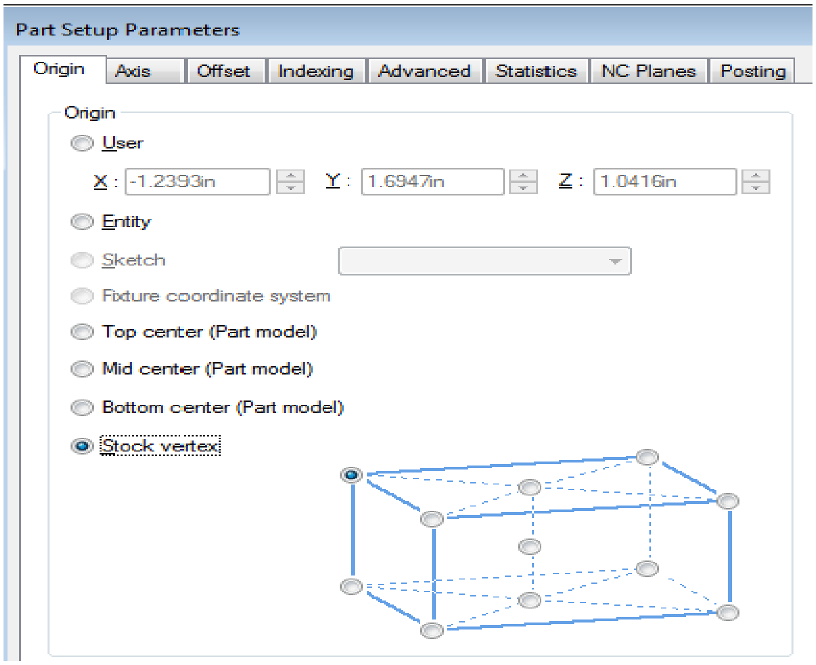
   The *New Multi Surface Feature* dialog box is displayed.
2. Set the Feature type to *All Displayed*.

   Did You Know ...
   The Select All and Associate option in the New Multi Surface Feature dialog box which was available till the CAMWorks 2012 version has now been replaced with the All Displayed option available in the Feature Type list.
3. Set the Strategy to *Area Clearance, Constant Stepover*.

   This Strategy has been defined in the TechDB to generate an Area Clearance operation and a Constant Stepover operation.
4. ✓ Click *OK* to insert the operation.
5. ✗ Click *Cancel* to close the *New Multi Surface Feature* dialog box.

   The new Multi Surface feature is listed under the Mill Part Setup in the Feature tree.

Defining Program Zero



Part Setup Parameters

Origin

○ User

X: -1.2393in Y: 1.6947in Z: 1.0416in

○ Entity

○ Sketch

○ Fixture coordinate system

○ Top center (Part model)

○ Mid center (Part model)

○ Bottom center (Part model)

● Stock vertex

Changing the Mill Setup Origin

1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar.

   OR

   Right click the *Multi Surface Feature1* in the Feature tree and select *Generate Operation Plan* from the context menu.
2. Double click the *Mill Part Setup* in the Operation tree.

   The *Part Setup Parameters* dialog box is displayed. This dialog box allows you to change the parameters of the Mill Part Setup you inserted earlier.
3. Select *Stock vertex* and leave the selected corner as shown.

   The XYZ zero location is now set at the theoretical top corner of the material and the Mill Part Setup indicator moves to that position in the graphics area.

   This is good practice and will assure that your default rapid moves will not contact the part.
4. Click *OK* to close the *Part Setup Parameters* dialog box.

Adjusting Operation Parameters and Generating Toolpaths

1. Double click *Area Clearance1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
2. On the *Pattern* tab, set the *Pattern type* to *Lace* and the *Cut angle* to **90deg**.
3. On the *Area Clearance* tab, set the following parameters:
   - Allowance = **0.01in**
   - Z allowance = **0.01in**
   - Method = **Scallop**
   - Cut amount = **0.25in**
   - Min. cut amount = **0.1in**
   - Max. XY stepover = **0.1in**
4. On the *Entry/Retract* tab, use *Skim* for the *Retract Method* and set the *Ramping Method* to *Ramp*, then click *OK* to close the *Operation Parameters* dialog box.
5. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
6. If such a warning is not displayed, right click *Area Clearance1* in the tree and select *Generate Toolpath* from the context menu.

Editing the Constant Stepover Operation to Cut the Mouse

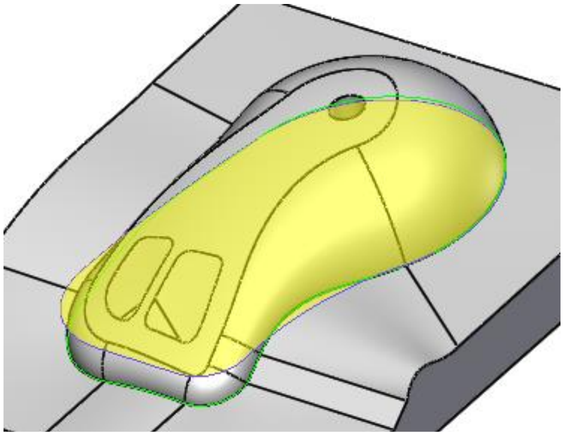
The Constant Stepover operation removes material by maintaining a constant user-defined stepover.

1. Double click *Constant Stepover1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
2. On the *Constant Stepover* tab, make sure the *Pass options Method* is set to *3D Stepover*.

   When 2D Stepover is selected, the distance between each pass will be constant and equal to the Horizontal cut amount, as viewed from the XY direction. This method produces a consistent surface finish on feature faces that do not have a steep slope angle. Because the stepover is constant in the XY plane, the surface finish on feature faces that are vertical or have a steep slope angle will not be as good.

   The 3D Stepover method generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope of the feature faces. In shallow areas the stepover will be equal to the Horizontal cut amount and in steeper areas the stepover will be equal to the

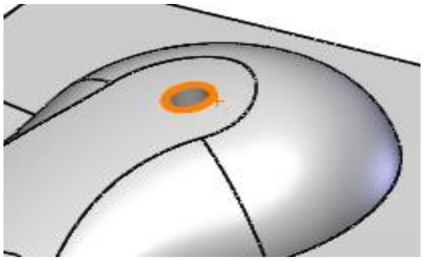


Selecting edges around the mouse

Vertical cut amount. This method produces the most consistent surface finish on features whose faces have either shallow or steep slope angles.

3. Set the Horizontal and Vertical cut amounts to **0.05in**, then click *OK* to close the dialog box.
4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
5. Right click *Constant Stepover1* in the tree and select *New Contain Area* on the context menu.
6. The *Contain Area* dialog box is displayed. In the Selection Mode group box, set the *Edge Selection* to *Single Edge*.
7. In the graphics area, pick each edge around the mouse (shown in green highlights in the image given to the right). Rotate the part if necessary. The selected seven edges will be displayed in the *Selected Contours* list box.
8. Set the Offset to **0.1875in**.

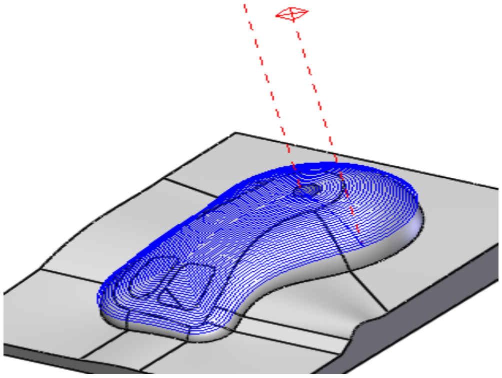
   This value is the radius of the tool and allows the tool to machine to the bottom of the mouse. Notice the offset curve that is generated from the selected edges. The toolpath will be created inside of this bounding curve.
9. ✓ Click the *OK* button to insert the Contain area and to close the dialog box.
10. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.



Selecting the top edge of the hole

11. Right click *Constant Stepover1* in the tree and select *New Avoid Area* on the context menu.
12. The *Avoid Area* dialog box is displayed. In the Selection Mode group box, set the *Edge Selection* to *Single Edge*.
13. Pick the edge around the top of the hole (shown in orange highlight in the image on the right). *Edge<1>* will be selected to the *Selected Contours* list box.
14. In the *Avoid Areas* group box, click on the *Machine over* button to select this option. When this option is selected, CAMWorks machines over the Avoid area. When the *Machine Around* button is clicked, CAMWorks machines around the avoid area.
15. ✓ Click the *OK* button to insert the Avoid area and to close the dialog box.

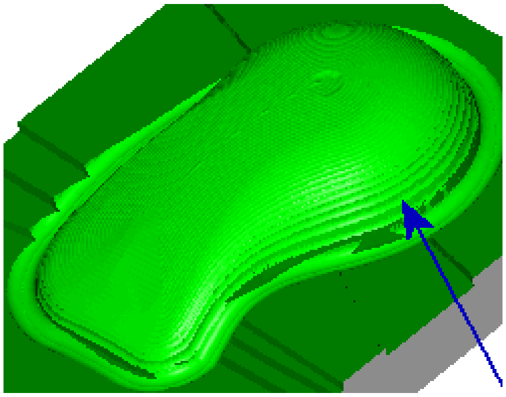
    For this tutorial, you can assume the hole will be drilled later.
16. Click *Yes* if CAMWorks displays a warning



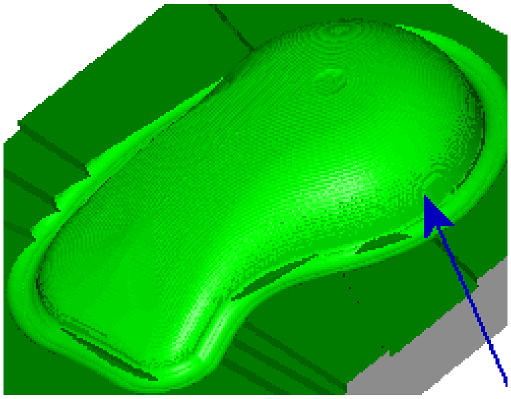
Constant Stepover

message asking if you want to regenerate the toolpaths.

17. If such a warning is not displayed, right click *Constant Stepover1* in the tree and select *Generate Toolpath* from the context *menu*.
18. Click the *Simulate Toolpath button* on the CAMWorks Workflow toolbar.
19. The *Toolpath Simulation* toolbar is displayed. Set the End condition to *Next Operation*.
20. Click the *Turbo* button, then click *Run* to view each toolpath simulation.
21. The *Constant Stepover* toolpath simulation observed had its *Pass options Method* is set to *3D Stepover*. Exit the Toolpath Simulation mode. In the Operations tree,
    - Double click on the *Constant Stepover* Operation to open its Operation Parameters dialog box.
    - On the Constant Stepover tab, set the *Pass options Method* to *2D Stepover*.
    - Click *OK* to apply the changes and close the dialog box.
    - Click *Yes* if CAMWorks prompts you to regenerate the toolpaths.
22. Run the Toolpath Simulation once again and observe the *Constant Stepover* toolpath simulation.



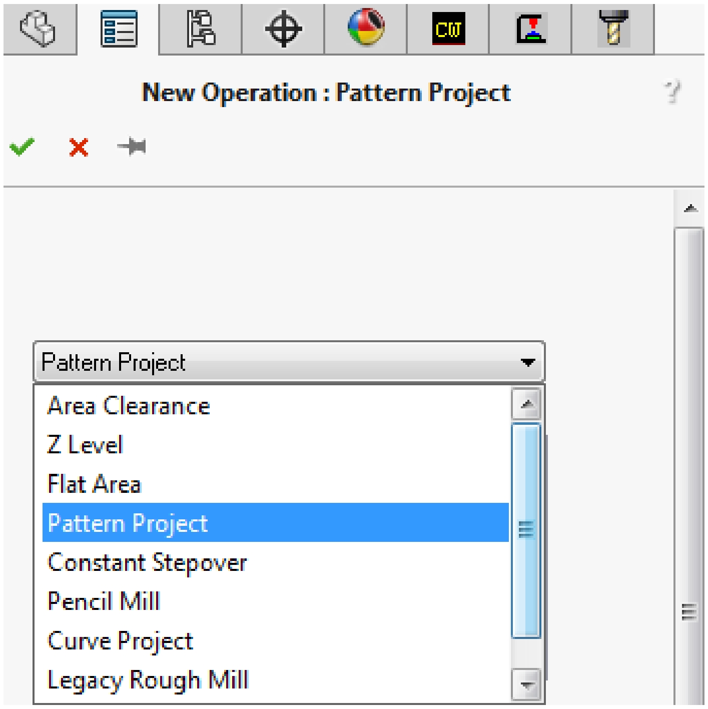
2D stepover



3D Stepover

23. In the above figures notice that with the 2D Stepover method the surface finish on the vertical feature faces is not as good as the finish with the 3D Stepover method.
24. Double click on the *Constant Stepover* Operation to open its Operation Parameters dialog box.
25. On the Constant Stepover tab, set the *Pass options Method* to *3D Stepover again*.
26. Click *OK* to apply the changes and close the dialog box.
27. Click *Yes* if CAMWorks prompts you to regenerate the toolpaths.
28. Click ✓ *Ok* to close the simulation toolbar OR press the *Escape* key on the keyboard to exit the simulation mode.

Inserting a Pattern Project Operation to Cut the Base



Selecting Pattern project Operation

1. Highlight *Constant Stepover1* in the Operation tree. In the CAMWorks Workflow toolbar, select *New 3 Axis Mill Operation*. Select *Pattern Project* from the dropdown menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.
3. End Condition Click the *End Condition* button. Leave the *Edit Operation on creation* option checked.
4. ✓ Click *OK* to insert the operation.
5. The *Operation Parameters* dialog box for the newly inserted Pattern Project operation is displayed. On the *Pattern* tab, ensure that the Pattern type is set to the *Slice*. In the *Pattern Options* group box, check the *Staydown* option.
6. On the *Finish* tab, make sure the *Direction control* is set to *Zigzag*.
7. On the *Links* tab, set the *Max link distance* to **0.25in.**
8. Click *OK* to apply the changes and to close the *Operation Parameters* dialog box.
9. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

Inserting a Contain Area to cut only the top of the base

1. Highlight the newly inserted *Pattern Project1* operation in the tree and click on the *New Contain Area* button on the CAMWorks Command Manager.

   OR

   Right click *Pattern Project1* in the Operation tree and select *New Contain Area* from the context menu.

   The *New Contain Area* dialog box is displayed.
2. In the *Selection Mode* group box, set the *Edge Selection* to *Constant Depth Loop*.
3. In the *Allowed Sketches* list box, pick *Sketch1*. The sketch will be listed in the *Selected Contours* list box.

4. In the Offset group box, the default *Tool Condition* is *Tool Upto Curve* ⊡. Click on the *Tool condition* button to view the dropdown menu options. Select *Tool on Curve* ⊡ from the dropdown menu.
5. ✓ Click *OK* to insert the Contain Area.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

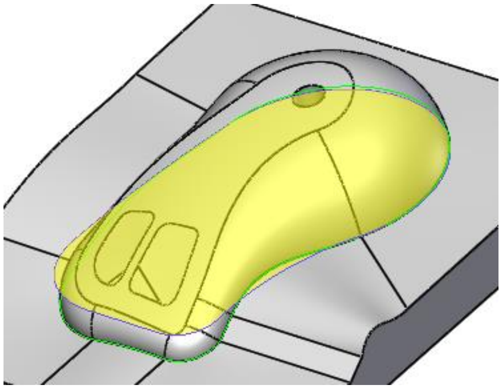
Inserting an Avoid Area to avoid machining the mouse

1. Highlight the *Pattern Project1* operation in the tree and click on the *New Avoid Area* button ⊡ on the CAMWorks Command Manager.

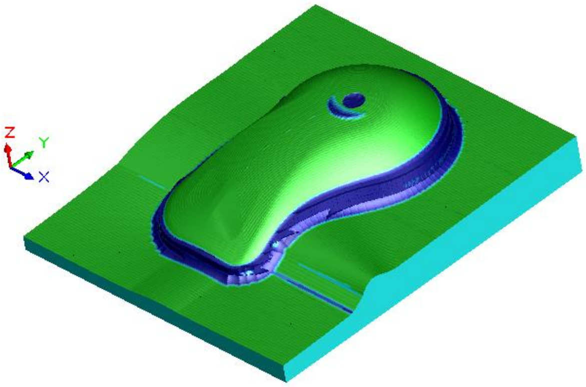
   OR

   Right click *Pattern Project1* in the Operation tree and select *New Avoid Area* on the context menu.
2. The Avoid Area dialog box is displayed. In the *Selection Mode* group box, set the ⊡ *Edge Selection* to *Single Edge*.
3. In the graphics area, pick each edge around the base of the mouse. Rotate the part if necessary. The selected seven edges will be displayed in the *Selected Contours* list box.
4. In the Offset group box, click on the *Tool condition* button to view the dropdown menu options. Select *Tool Upto Curve* ⊡ from the dropdown menu.
5. ✓ Click *OK* to insert the Avoid Area.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
7. If such a warning is not displayed, right click *Pattern Project1* in the tree and select *Generate Toolpath* from the context menu.

   CAMWorks alters the Pattern Project toolpath to machine the top of the base and avoid the mouse.



Inserting Avoid area to void the mouse



Blue material indicates leftover material

8. Click the *Simulate Toolpath* button either on the CAMWorks Workflow toolbar OR on the CAMWorks Command Manager.
9. The *Toolpath Simulation* toolbar is displayed. Set the End condition to *Next Operation*.
10. Click the *Turbo* button, then click *Run* button to view each toolpath simulation.
11. When the simulation is finished, click the *Show Difference* button.

    This function shows the difference between the machined model and the SOLIDWORKS/CAMWorks Solids design model. Note the blue area at the parting line showing material left over by the radius of the ball tool.
12. Click ✓ *OK* to exit the simulation.

Inserting a Pencil Mill Operation to Cut Parting Line

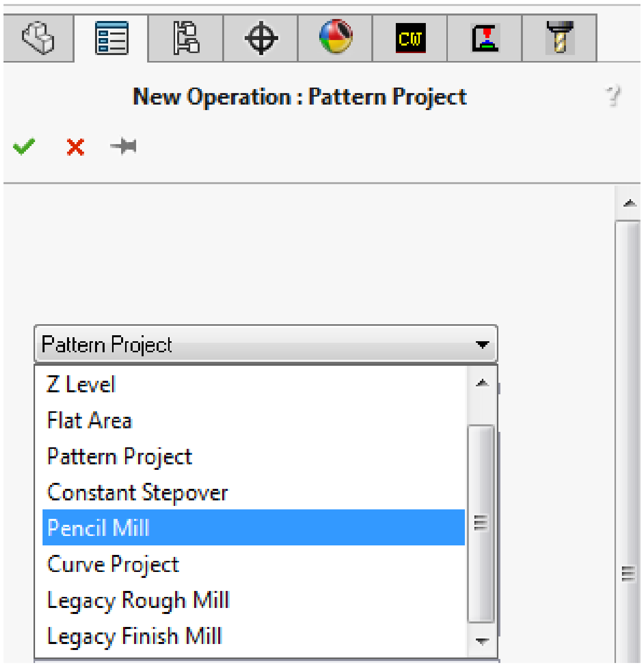
1. On the CAMWorks Workflow Toolbar OR CAMWorks Command Manager, click on the *New 3 Axis Mill Operation* button. Select *Pencil Mill* from the dropdown menu.

   OR

   Right-click *Pattern Project1* operation in the Operations tree and *New 3 Axis Mill Operations>>Pencil Mill* from the cascading menu.
2. The *New Operation* dialog box is displayed.

   Highlight *Multi Surface Feature1* in the Features list.

   End Condition Click the *End Condition* button. Leave the *Edit Operation on creation* option checked.



Selecting Pencil Mill Operation from dropdown menu

3. Click *OK* to insert the operation.

   The *Operation Parameters* dialog box for the newly inserted *Pencil Mill1* operation is displayed.
4. Click on the *Tool* tab and select the *Mill Tool* page; make sure the Cut diameter is **0.125in**.

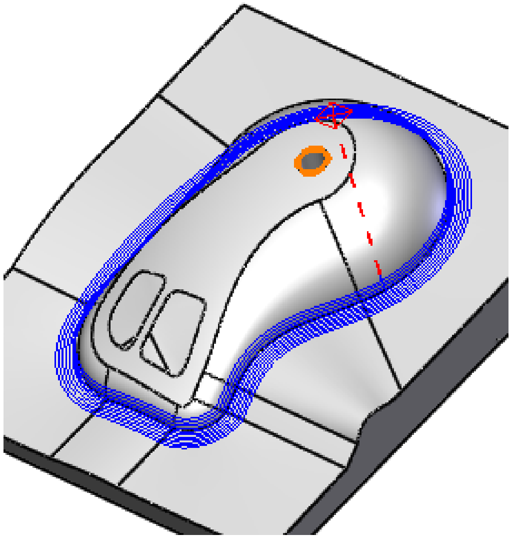
   In case it isn't, click on the *Tool Crib* page; highlight a **0.125in** Ball Nose Tool and click the *Select* button. Click *Yes* to replace the corresponding holder. Observe that the *Mill Tool* page now displays a Ball nose tool with a cut diameter of **0.125in**.
5. On the *Pattern* tab, ensure that the Pattern is set to *Parallel Lace*.

This method generates multiple parallel passes to remove the material. The distance between passes is equal to the Cut amount.

6. Set the following parameters in the *Pattern* tab:
   - Bitangency angle = **30deg**
   - Pass Options Method = **3D Stepover** (in shallow areas the stepover will be equal to the Horizontal cut amt. and in steeper areas the stepover will be equal to the Vertical cut amt.)
   - Number of Cuts = **Checked with 5 cuts**
   - Horizontal & Vertical cut amt. = **0.04in**
7. Click *OK* to apply the changes and close the *Operation Parameters* dialog box.
8. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
9. Highlight *Pencil Mill1* in the tree and click on the *New Avoid Area* button on the CAMWorks Command Manager.

   OR

   Right click *Pencil Mill1* in the tree and select *New Avoid Area* from the context menu.
10. The *Avoid Area* dialog box is displayed. In the Selection Mode group box, set the *Edge Selection* to *Single Edge*.
11. Pick the edge around the top of the hole. *Edge<1>* will be selected to the *Selected Contours* list box.
12. In the *Avoid Areas* group box, click on the *Machine around* button to select this option.
13. ✓ Click the *OK* button insert the Avoid area and also to close the dialog box.
14. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
15. If such a warning is not displayed, right click *Pencil Mill1* in the tree and select *Generate Toolpath* from the context menu.
16. View the toolpaths in Simulation mode once again. Observe that the edges around the mouse are now cut neatly.



Pencil Mill Toolpath with Avoid Area

3 Axis Mill 5

For a Pattern Project operation, the Flowline pattern has a unique characteristic that makes it appropriate for machining areas which are defined using SOLIDWORKS/CAMWorks Solids loft or sweep features.

The Flowline pattern produces toolpaths based on two user-specified curves. The toolpath can be generated parallel to *(Along)* or between *(Across)* the two curves. The curves can be either open or closed.

Contain and avoid areas can be defined for all 3 Axis Mill operations (except 3 Axis Legacy Rough Mill operations). In this tutorial, contain and avoid areas will be defined for the generated Pattern project, which leads to alteration the Pattern project toolpath.

Topics covered in this tutorial:

- Generating Flowline machining toolpaths using curves to direct the flow of the toolpath
- Using contain and avoid areas to modify toolpath

Step 1: Defining the Mill Part Setup

1. Open the part file **MILL3AX_5.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.

3. Double click *Machine [Mill–inch]* in the tree.

   OR

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

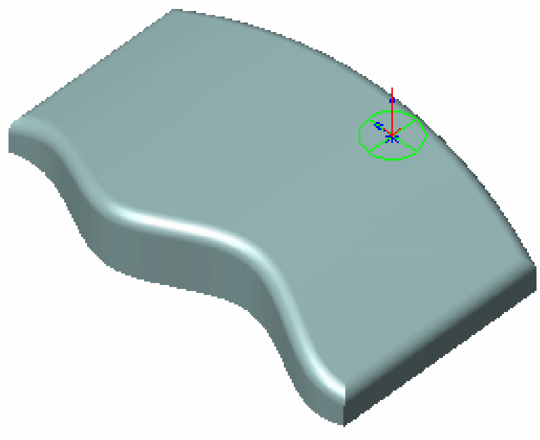
   The *Machine* dialog box is displayed.

   – On the *Machine* tab, ensure that *Mill–inch* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

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



MILL3AX_5.SLDPRT

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

   OR

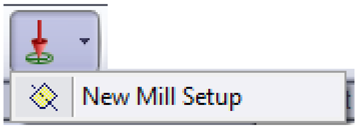
   Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   The *Stock Manager* dialog box is displayed.

   – Click *OK* to use the default Stock type (Bounding box) for this tutorial.

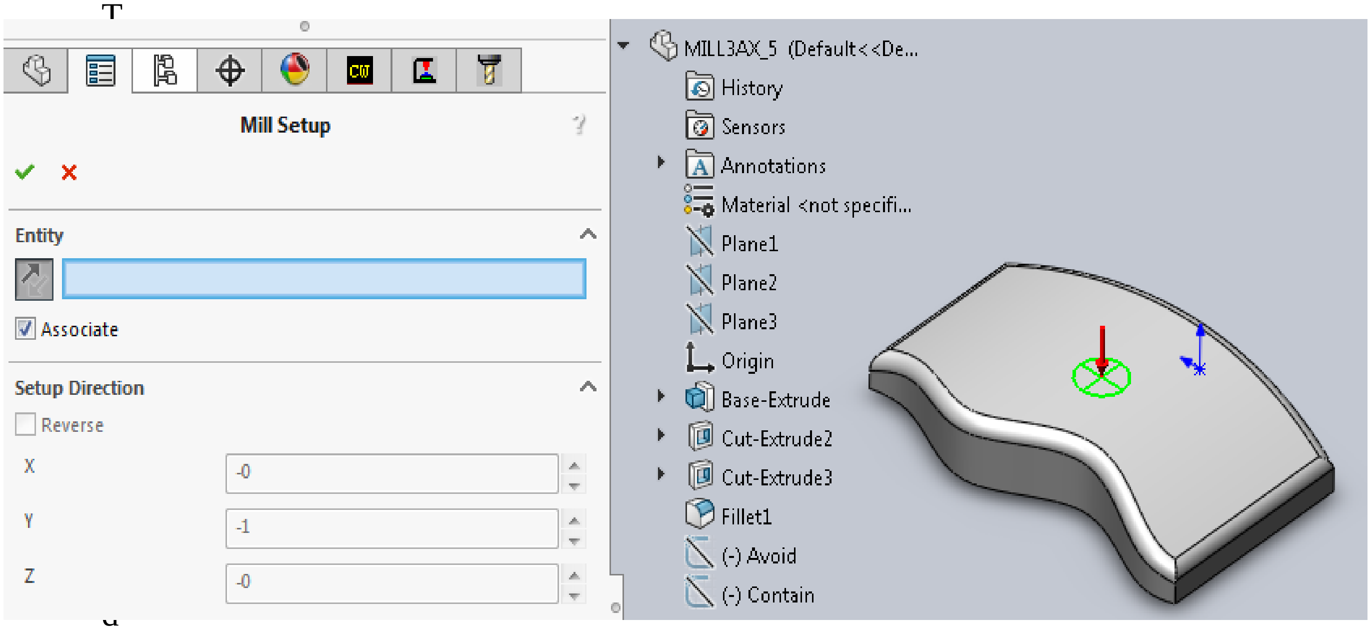
5. Right click the *Stock Manager* item and select *New Mill Part Setup* from the context menu.

   OR

   Click on the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu.

   T



☑ Associate

☐ Reverse

X -0

Y -1

Z -0

u

.

Inserting the Mill Part Setup

u

log box is displayed.

– In the graphics area, click the MILL3AX_5 to expand the tree and select *Plane2*.

– In the *Mill Setup* dialog box, click the *Reverse Selected Entity* button.

– Make sure the Face option is not checked.

– ✓ Click OK to insert the setup.

  Mill Part Setup1 is inserted in the Feature tree.

Step 2: Inserting the Multi Surface Feature

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

   The New Multi Surface Feature dialog box is displayed.

2. In the Feature Type group box, select the *All Displayed* option from the dropdown list.
3. Set the Strategy to *Area Clearance, Pattern Project*.

This Strategy has been set up in the TechDB to generate one Area Clearance operation and one Pattern Project operation.

4. Click *OK* to insert the feature.

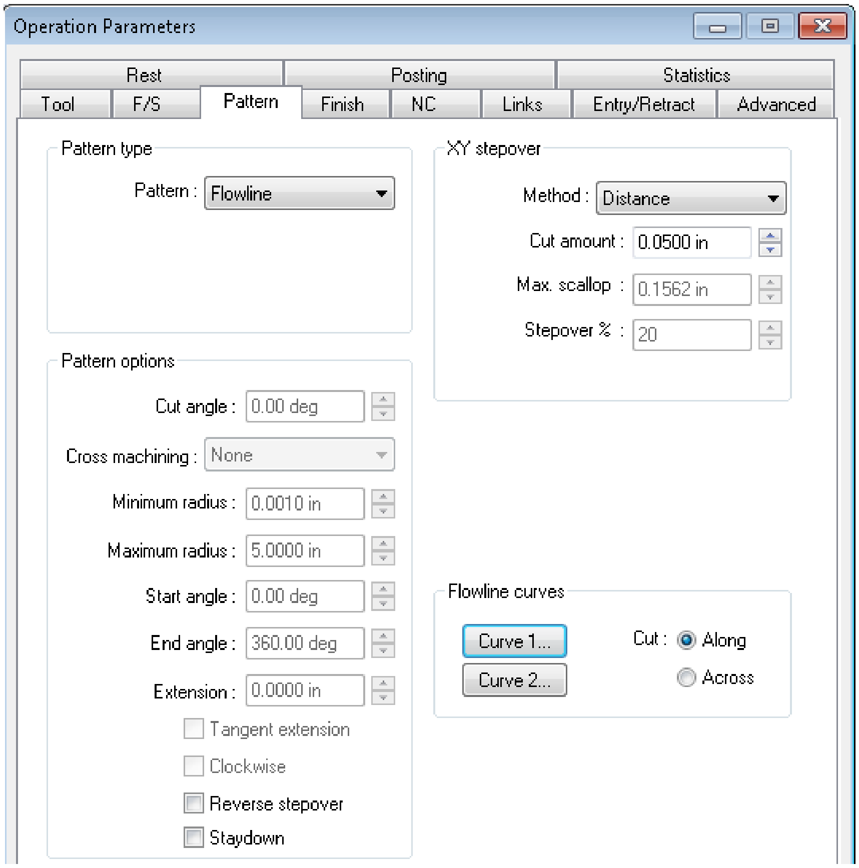
Step 3: Generating Toolpaths Using the Flowline Pattern

Generate an Operation Plan, modify parameters and generate toolpaths:

1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar OR Command Manager.
2. A Pattern Project operation is generated in the Operation tree. Double click *Pattern Project1* in the tree.
3. The *Operation Parameters* dialog box is displayed. On the *Pattern* tab.
   - In the *Pattern Type* group box, set the Pattern to *Flowline*
   - In the *Pattern Options* group box, check the *Staydown* option.
   - In the *Flowline Curves* group box, set the *Cut* option to *Along*.
4. In the Flowline group box, click the *Curve 1* button.

   The *Curve 1* and *Curve 2* buttons in this section are used to choose the sketch curves to direct the toolpath. Each of the 2 curves must be a unique sketch in SOLIDWORKS/CAMWorks Solids.
5. The Curve Wizard dialog box is displayed. In this dialog box, select *Flow 1* in the *Allowed Sketches* list.

   Flow 1 is then displayed in the *Selected Contours* list box and the curve is highlighted on the part.



Pattern : Flowline

Method : Distance

Cut amount : 0.0500 in

Max. scallop : 0.1562 in

Stepover % : 20

Cut angle : 0.00 deg

Cross machining : None

Minimum radius : 0.0010 in

Maximum radius : 5.0000 in

Start angle : 0.00 deg

End angle : 360.00 deg

Extension : 0.0000 in

☐ Tangent extension

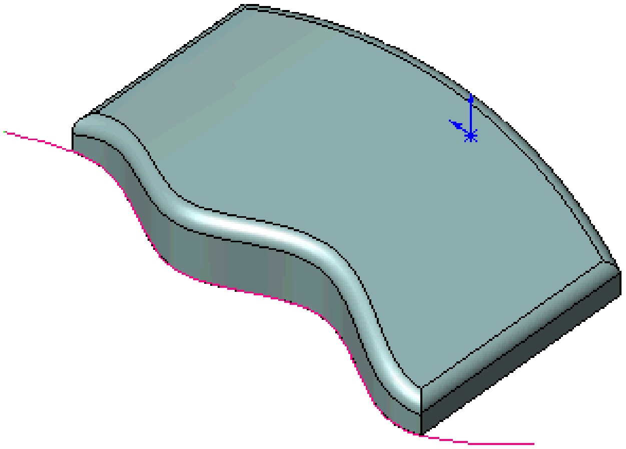
☐ Clockwise

☐ Reverse stepover

☐ Staydown

Cut : ● Along ○ Across

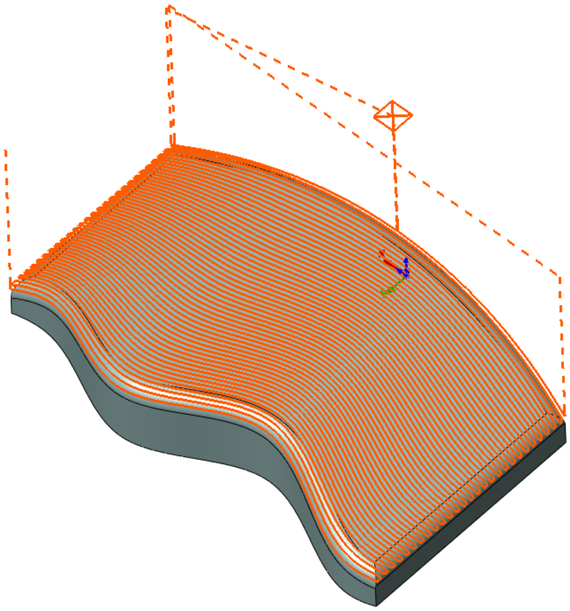
Pattern Project Flowline Pattern



Flow1 curve highlighted on the part

Optionally, you can add an offset to the curve if needed. For this exercise, an offset is not needed.

6. ✓ Click *OK* to insert the flowline curve.
7. In the Flowline group box, click the *Curve 2* button.
8. The Curve Wizard dialog box is once again displayed. Select *Flow 2* in the *Allowed Sketches* list.
9. Click *OK* ✓ to insert the flowline curve.
10. On the *Finish* tab, set the *Mach. Deviation* to **0.001in**.
11. On the *Links* tab:
    - Set the Type to *Spline Curve*.
    - *Set the Max. link distance* to **2in.**
12. Click *OK* to close the Operation Parameters dialog box.
13. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
14. If such a warning is not displayed, click the *Generate Toolpath* button on the CAMWorks Workflow toolbar/ Command Manager.



Pattern Project Flowline Toolpath

The toolpath starts in the direction of the first curve, and then translates to the shape of the second curve as it progresses.

Step 4: Defining Contain and Avoid Areas to Modify Toolpaths

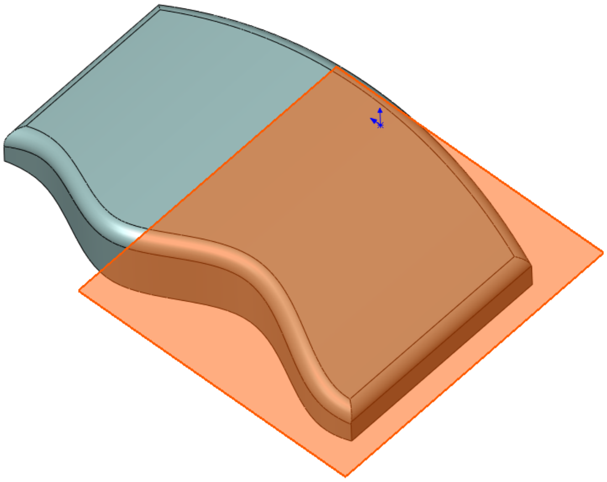
Finish toolpaths for Multi Surface features can be modified to stay contained within an area or to avoid a specified area. Contain and avoid areas can be used to selectively machine within specified areas for re-machining, clamping or other purposes.

The Contain and avoid areas defined for the toolpath are illustrated for learning purpose.

Defining a Contain Area for the Pattern Project toolpath

1. Right click *Pattern Project1* in the Operation tree and select *New Contain Area* from the context menu.

   OR



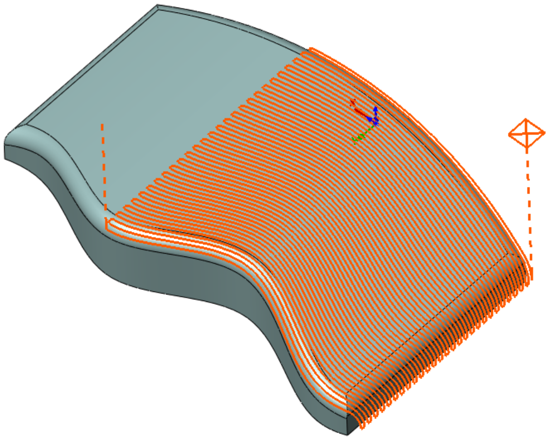
Defining Contain Area using a sketch

Highlight *Pattern Project1* in the Operation tree and click the *New Contain Area* button in the CAMWorks Command Manager.

The Contain Area dialog box is displayed.

2. Select the sketch named *Contain* in the *Allowed Sketches* list. This sketch will then be listed in the *Selected Contours* list box.
3. ✓ Click *OK* to insert the Contain area.
4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
5. If such a warning is not displayed, click the *Generate Toolpath* button on the CAMWorks Workflow toolbar/Command Manager.
6. Left click the plus sign ⊞ to the left of *Pattern Project1* in the Operation tree.

   The Contain Area item is displayed under *Pattern Project1*. If you need to delete any contain area later, you can highlight and delete that item.
7. Highlight *Pattern Project1* in the tree. Observe that the toolpath is contained within the rectangular sketch that was chosen for the Contain Area. The toolpath is trimmed to the contain geometry. The radius of the tool would pass beyond this boundary. Use the Offset value when choosing the curve if this is not desirable.



Pattern Project toolpath confined to Contain area

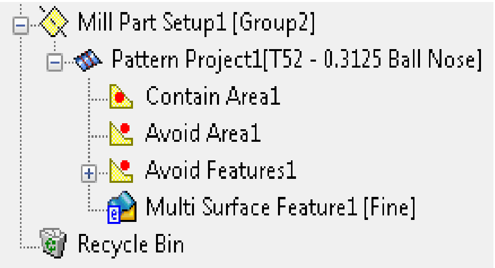
Defining an Avoid Area for the Pattern Project Toolpath

1. Right click *Pattern Project1* in the tree and select *New Avoid Area* from the context menu.

   OR

   Highlight *Pattern Project1* in the Operation tree and click the *New Avoid Area* button in the CAMWorks Command Manager.

   The Avoid Area dialog box is displayed.
2. Select the sketch named *Avoid* in the *Allowed Sketches* list. This sketch will then be listed in the *Selected Contours* list box.
3. ✓ Click *OK* to insert the Avoid area.

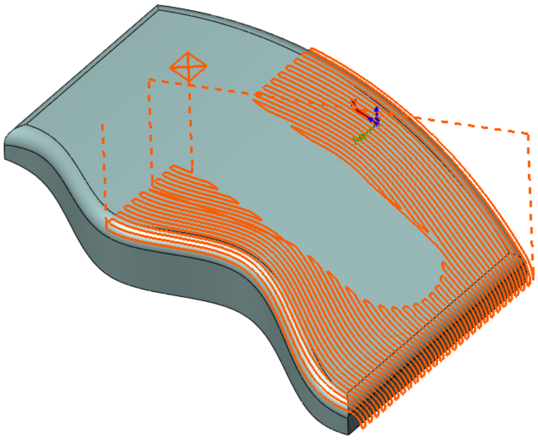


```
⊟ Mill Part Setup1 [Group2]
   ⊟ Pattern Project1[T52 - 0.3125 Ball Nose]
        Contain Area1
        Avoid Area1
      ⊞ Avoid Features1
        Multi Surface Feature1 [Fine]
  Recycle Bin
```

Contain & Avoid Area items in the Operation tree

4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
5. If such a warning is not displayed, click the *Generate Toolpath* button on the CAMWorks Workflow toolbar/Command Manager.
6. Click the *plus sign* ⊞ to the left of *Pattern Project1* in the tree.

   The Contain Area and Avoid Area items are displayed in the tree under the *Pattern Project1* item.
7. Highlight *Pattern Project1* in the tree. Observe that the toolpath is contained within the Contain Area and avoid the area defined as Avoid area. The toolpath is trimmed to contain and avoid geometries.



Pattern Project toolpath confined to contain area and avoiding the avoid area

3 Axis Mill 6

CAMWorks provides numerous patterns that can be used to finish or semi-finish a Multi Surface feature. Each pattern has unique characteristics. While you may not use these for general machining, occasionally you will machine a part that is appropriate for one of these styles.

This tutorial explains how to generate radial, spiral and 2D stepover toolpaths.

Topics covered in this tutorial:

- Generating radial finish toolpaths using the Pattern Project operation.
- Using Step Through Toolpath
- Generating spiral finish toolpaths using the Pattern Project operation.
- Generating 2D stepover toolpaths using the Constant Stepover operation.

Step 1: Defining the Mill Part Setup

1. Open the Open the part file **MILL3AX_6.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.

3. Double click *Machine [Mill–inch]* in the tree.

   OR

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

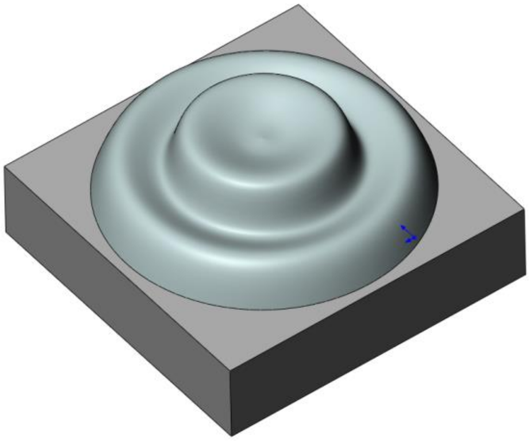
   The *Machine* dialog box is displayed.

   – On the *Machine* tab, ensure that *Mill–inch* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

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



MILL3AX_6.SLDPRT

4. Double click *Stock Manager* in the tree

   OR

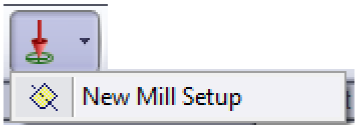
   Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   The *Stock Manager* dialog box is displayed.

   – Click *OK* to use the default Stock (Bounding box) for this tutorial.

5. Right click the Stock Manager item and select *New Mill Part Setup* from the context menu

   OR

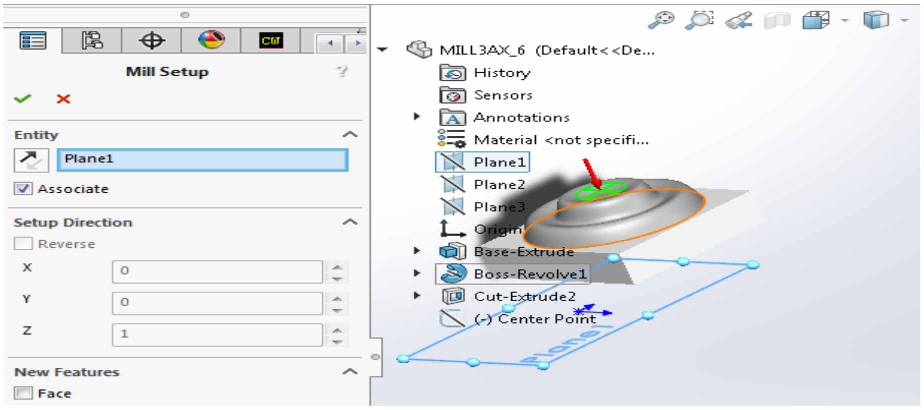


   Click on the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu.

   The *Mill Setup* dialog box is displayed.

   – In the graphics area, click on the *MILL3AX_6* to expand the tree and select *Plane1*.

   – Make sure the *Face* option is not checked.


Mill Setup

Entity: Plane1

☑ Associate

Setup Direction

☐ Reverse

X: 0

Y: 0

Z: 1

New Features

☐ Face

Inserting the Mill Part Setup

   ✓ Click OK to insert the setup. *Mill Part Setup1* is inserted in the tree.

Step 2: Inserting Multi Surface Feature

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

   The New Multi Surface Feature dialog box is displayed.

2. In the Feature Type group box, select the *All Displayed* option from the dropdown list.

3. Set the Strategy to *Area Clearance, Pattern Project*.

   This Strategy has been set up in the TechDB to generate one Pattern Project operation.

4. ✓ Click OK to insert the feature.

   Multi Surface Feature1 is listed in the Feature tree.

Step 3: Generating Operations and Changing the Mill Part Setup origin

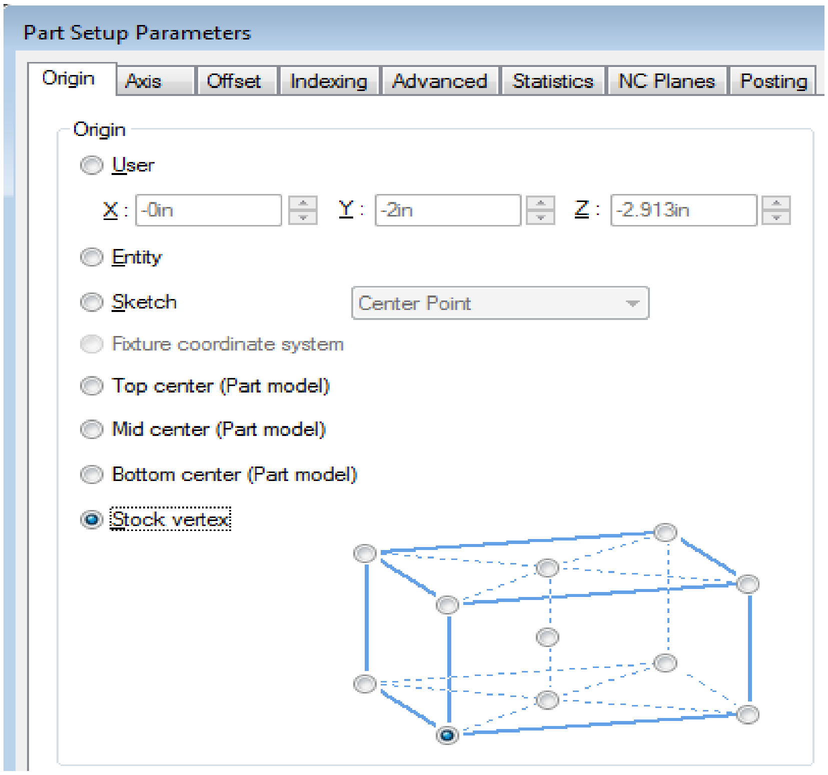
1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar/Command Manager.

   The Pattern Project operation *(Pattern project1)* generated is listed in the Operation tree.

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

   The *Part Setup Parameters* dialog box is displayed.

3. On the *Origin* tab, change the Origin to *Stock vertex*.
4. Change the corner to the front bottom as shown in the image given at the right.
5. Click *OK* to apply the changes and close the dialog box.



Origin

○ User

X : -0in Y : -2in Z : -2.913in

○ Entity

○ Sketch Center Point

○ Fixture coordinate system

○ Top center (Part model)

○ Mid center (Part model)

○ Bottom center (Part model)

● Stock vertex

Changing the Mill Part Setup Origin

Step 4: Generating Radial Toolpaths Using Pattern Project

1. Double click *Pattern Project1* in the Operation tree.

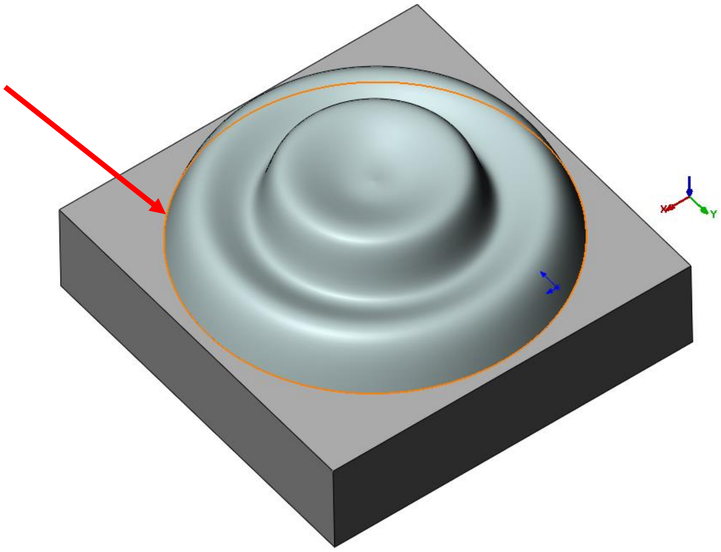
   The Operation Parameters dialog box is displayed.

2. Click on the Tool tab.
   - Select Tool Crib page and select a **0.25in** Ball Nose tool.
   - Click *Yes* to replace the corresponding holder.
3. Click on the Pattern tab.
   - In the *Pattern type* group box, change the Pattern to *Radial*.
   - In the *Pattern options* group box, check the *Reverse stepover* option to start the toolpath in the center.
     This option allows you to specify whether the tool will enter the material at the center or on the diameter of the radial pattern.
   - In the *XY Stepover* group box, change the *Cut amount* to **0.15in**.
   - Click the *Pick Point* button in the *Center Point* group box.
   - The *Define Point* dialog box is displayed. This dialog box allows you to define the center of the radial toolpath. You can define the applicable point relative and associative to a vertex, planar arc edge or arc segment defined in a sketch on the part.

- In the graphics area, pick the bottom edge of the revolved boss feature.

  CAMWorks calculates the center point at the arc center.
- Click OK to close the dialog box.

4. Click on the *Finish* tab.
   - In the *Surface Finish* group box, change the *Mach. deviation* to **0.001in**.
5. Click *OK* to close the Operation Parameters dialog box.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.



Selecting the bottom edge of Revolved Boss feature

7. If such a warning is not displayed, click on the *Generate Toolpath* button on the CAMWorks Ribbon bar.
8. Click on the *Pattern Project1* operation in the Operation tree to view the toolpath.

Step 5: Defining Contain Area for Pattern Project

1. Right click *Pattern Project1* in the tree and select *New Contain Area* from the context menu.

   OR

   Highlight *Pattern Project1* in the Operation tree and click the *New Contain Area* button in the CAMWorks Command Manager.

   The Contain Area dialog box is displayed.
2. In the *Selection Mode* group box, set the *Edge Selection* to *Single Edge*.
3. Pick the bottom edge of the boss. *Edge<1>* will then be displayed in the *Selected Contours* list box.
4. In the *Offset* group box, click on the Tool Condition button. Set the Tool condition to *Tool Past Curve* .

   This parameter defines how CAMWorks internally offsets the area/curve.
5. Click OK to insert the contain area.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

7. If such a warning is not displayed, right click *Pattern Project1* in the tree and select *Generate Toolpath* from the context menu.

Step 6: Step Through Toolpath

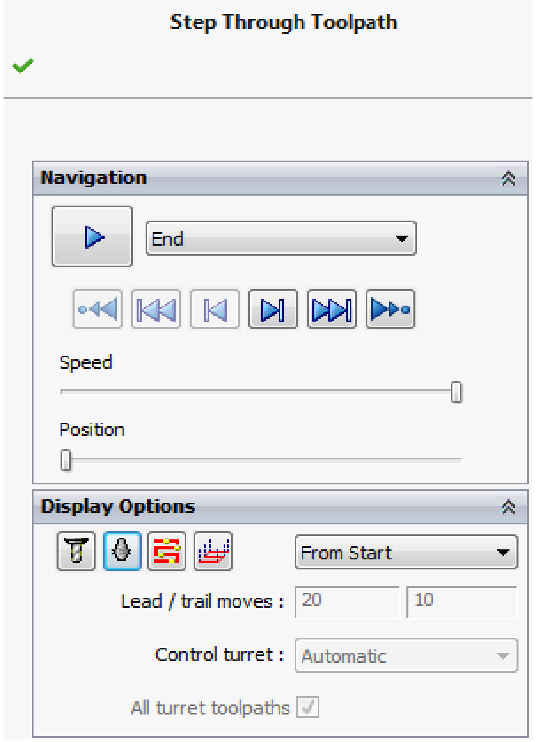
1. Right click the *Pattern Project1* operation in tree and select *Step Thru Toolpath* from the context menu.

   OR

   Click on the *Step Thru Toolpath* button in the CAMWorks Workflow Toolbar/ Command Manager.

   The Step Through Toolpath dialog box is displayed.
2. In the *Display Options* group box of this dialog box,
   - Click on the Tool display button. Set the display to *Shaded display* .
   - Click on the Tool Holder Display button. Set the display to *Tool Holder Shaded With Edges* .
   - Set the *Display Number of Toolpath Records* to *From Start*.

     When this option is selected, the toolpath centerline will display from the first toolpath record to the current toolpath record.
3. In the Navigation group box, click on the Play button . Observe the Play Motion in the graphics area. Notice that the toolpath radiates from centre out.
4. Click the *OK* button to close the *Step Through Toolpath* dialog box.



End

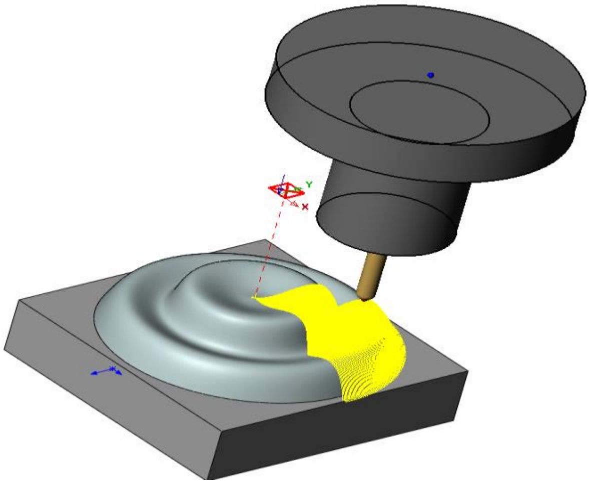
From Start

Lead / trail moves : 20 10

Control turret : Automatic

All turret toolpaths ☑

Step Through Toolpath dialog box



Play Motion in Step Through Toolpath mode

Step 7: Generating Spiral Toolpaths Using Pattern Project

You will now insert a Pattern Project operation to generate a Spiral toolpath:

1. Right click *Pattern Project1* in the tree and select *New 3 Axis Mill Operation>> Pattern Project* from the context menu

   OR

   Highlight *Pattern Project1* in the tree. In the CAMWorks Workflow toolbar, click on the *New 3 Axis Mill Operation* button . The *New Operation* dialog box is displayed.

   select *Pattern project* from the dropdown menu.

   The *New Operation* dialog box is displayed.
2. Highlight *Multi Surface Feature1* in the Features list.
3. End Condition Click the *End Condition* button. Make sure the *Edit Operation on creation* option is checked.
4. ✓ Click *OK* to insert the operation.

   The *Operation Parameters* dialog box for the new Pattern Project operation is displayed.
5. Click on the *Tool* tab and select the *Tool Crib* page.
6. Highlight a **0.25in** Ball Nose tool in the tool crib and click the *Select* button.
7. Click *Yes* to replace the corresponding holder.
8. Click on the *Pattern* tab. In the Pattern type group box, set the type to *Spiral*.
9. In the Pattern Options group box:
   - Set the Minimum radius to **0.01in** (to start the spiral 0.01in from the center point)
   - Set the Maximum radius to **2in** (the overall size of the circle)
   - Check the Reverse stepover option
10. In the Center Point group box, click the *Pick Point* button.

    The *Define Point* dialog box is displayed.
11. In the graphics area, pick the bottom edge of the revolved boss. This action selects this edge as the *Selected Entity* within the dialog box.

    CAMWorks calculates the center point at the center of the circular boss.
12. Click *OK* to close the dialog box.
13. On the Finish tab, set the following parameters:
    - In the Surface Finish group box, make sure the Mach. deviation is **0.001in**.
    - Set the Direction control to *Zig*.
    - Check the *Reverse direction* option to start the toolpath in the center.

      This option allows you to specify whether the tool will enter the material at the center or on the diameter of the radial pattern.
14. Click *OK* to close the *Operation Parameters* dialog box.

15. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
16. ✖ Click to close the *New Operation* dialog box.

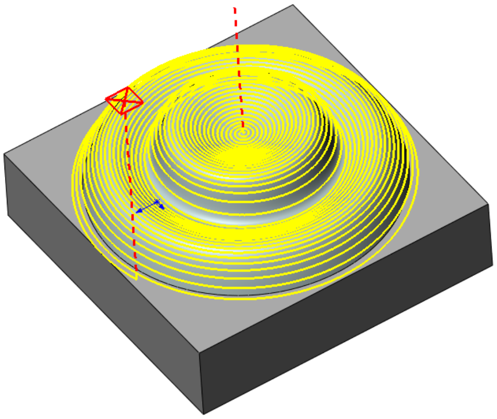
    The Operation *Pattern Project2* is inserted in the tree.
17. Right click *Pattern Project2* in the tree and select *New Contain Area* from the context menu.

    OR

    Highlight *Pattern Project2* in the Operation tree and click the *New Contain Area* button in the CAMWorks Command Manager.

    The Contain Area dialog box is displayed.
18. In the *Selection Mode* group box, set the *Edge Selection* to *Single Edge*.
19. In the graphics area, pick the bottom edge of the boss.

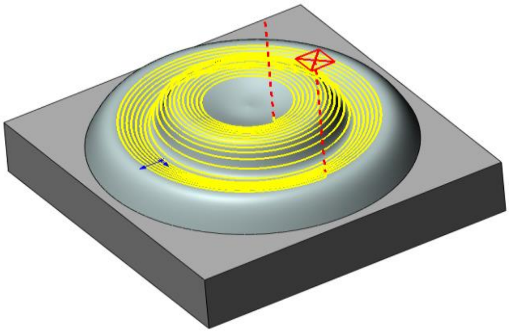
    *Edge<1>*will then be displayed in the *Selected Contours* list box.
20. In the Offset group box, click on the *Tool Condition* button.
21. Set the Tool Condition to *Tool Past Curve*.
22. ✓ Click OK to insert the contain area and close the dialog box.
23. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.



Pattern Project Toolpath with spiraling circular motion starting from center of part

24. If such a warning is not displayed, right click *Pattern Project2* in the tree and select *Generate Toolpath* from the context menu.
25. Alternatively, you may click on the *Generate Toolpath* button in the CAMWorks Workflow toolbar/Command Manager.

    CAMWorks generates a spiraling circular toolpath starting from the center of the part. You can set the Minimum and Maximum radius in the Operation Parameters dialog box to limit the extents of the toolpath.
26. Double click *Pattern Project2* in the tree. The *Operation Parameters* dialog box is displayed.
27. On the Pattern tab, change the Minimum radius to **0.5in** and the Maximum radius to **1.5in**.
28. Click *OK* to apply the changes and close the dialog box.



Pattern Project Toolpath with spiraling circular motion within specified radii

29. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
30. If such a warning is not displayed, right click *Pattern Project2* in the tree and select the *Generate Toolpath* command from the context menu.
31. Click on *Pattern Project2* operation in the Operations tree.

    Observe that CAMWorks has generated the toolpath within the specified radii.

Step 8: Generating 2D Stepover Toolpaths Using Constant Stepover

Steps to insert a Constant Stepover operation:

1. Right click *Pattern Project2* in the Operation tree and select *New 3 Axis Mill Operation>>Constant Stepover* from the cascading menu.

   OR

   Highlight *Pattern Project1* in the tree. In the CAMWorks Workflow toolbar, click on the *New 3 Axis Mill Operation* button .The *New Operation* dialog box is displayed select *Constant Stepover* from the dropdown menu.

   The *New Operation* dialog box is displayed.
2. Highlight *Multi Surface Feature1* in the Features list.
3. End Condition Click the *End Conditions* button. Make sure the *Edit operation on creation* option is checked.
4. ✓ Click *OK* to insert the dialog box.

   The Operation Parameters dialog box for the newly inserted operation is displayed.
5. Click on the *Tool* tab and then click on the *Tool Crib* page.
6. Highlight a **0.25in** Ball Nose tool in the tool crib and click the *Select* button.
7. Click *Yes* to replace the corresponding holder.
8. Click on the *Constant Stepover* tab.
9. In the Pass Options group box, set the Method to *2D Stepover*.

   When 2D Stepover is selected, the distance between each pass will be constant and equal to the Horizontal cut amount, as viewed from the XY direction. This method produces a consistent surface finish on feature faces that do not have a steep slope angle. Because the stepover is constant in the XY plane, the surface finish on feature faces that are vertical or have a steep slope angle will not be as good.
10. Check the *Number of cuts* option. For the *Number of cuts* option, set the number to **12**.

    The *Number of cuts* option allows you to specify whether the toolpath will continue until the entire part is machined or will be generated a specific number of passes.
11. Set the Horizontal cut amt and Vertical cut amt. to **0.05in**.

12. Click *OK* to close the Operation Parameters dialog box.
13. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

    The Constant Stepover operation *(Constant Stepover1)* is listed in the Operation tree.

Steps to define a Contain area for the Constant Stepover operation:

1. Right click *Constant Stepover1* in the tree and select *New Contain Area* from the context menu.

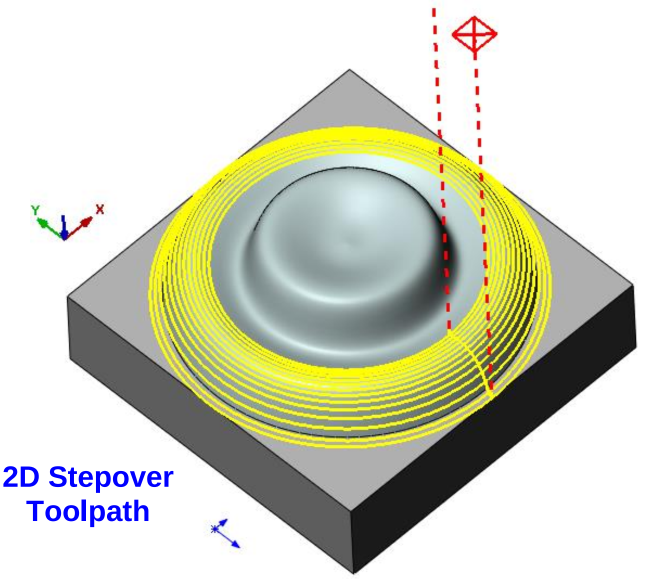
   OR

   Highlight *Pattern Project2* in the Operation tree and click the *New Contain Area* button in the CAMWorks Command Manager.

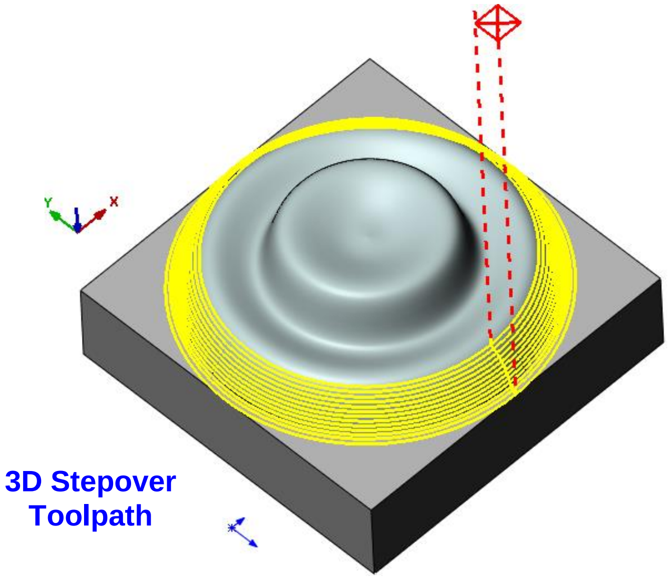
   The Contain Area dialog box is displayed.
2. In the *Selection Mode* group box, set the *Edge Selection* to *Single Edge*.
3. In the graphics area, pick the bottom edge of the boss.

   *Edge<1>*will then be displayed in the *Selected Contours* list box.
4. In the Offset group box, click on the *Tool Condition* button.
5. Set the Tool Condition to *Tool Past Curve*.
6. ✓ Click *OK* to insert the contain area and close the dialog box.
7. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
8. If such a warning is not displayed, right click *Constant Stepover1* and select *Generate Toolpath* from the context menu.

   CAMWorks generates the toolpath. Because the *Number of cuts* is set to **12** passes, the revolved surface is not machined completely.



2D Stepover Toolpath



3D Stepover Toolpath

If you had selected the *3D Stepover* method, the toolpath would look like the above figures. This method generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope of the feature faces. In shallow areas the stepover will be equal to the Horizontal cut amount and in steeper areas the stepover will be equal to the Vertical cut amount. This method produces the most consistent surface finish on features whose faces have either shallow or steep slope angles.

Chapter 4: Learning More 3 Axis Mill

This chapter provides more information on 3 Axis Mill operations such as Constant Stepover (3D Method), Curve Project, Flat Area and Rest Machining, thus enabling you to learn to utilize 3 Axis Mill Operations more effectively.

We recommend that you go through the tutorials in the previous chapters first. A thorough understanding of the concepts explained in Chapter1, Chapter 2 and Chapter 3 is necessary before you proceed with this chapter.

In addition to learning how to generate 2 and 3 axis mill toolpaths, the tutorials in this chapter explain basic functions and procedures that are used to generate both all toolpaths and NC code.

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

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

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

3 Axis Mill 7

This tutorial explains how to generate toolpaths for Area Clearance rest machining, Z Level and Flat Area operations.

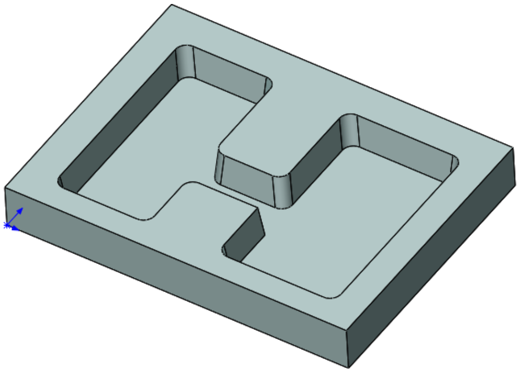
Topics covered in this tutorial:

- Generating Area Clearance operations and toolpaths for rest machining.
- Generating Z Level and Flat Area operations and toolpaths for finish machining.

Defining the Mill Part Setup and Inserting the Multi Surface feature

1. Open the part file **MILL3AXII_1.SLDPRT** in the following folder.

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



MILL3AXII_1.SLDPRT

2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill-inch]* in the tree.

   OR

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

   The *Machine* dialog box is displayed.

   – On the *Machine* tab, ensure *Mill–inch* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

   – Click *OK* to close the Machine dialog box.
4. Double click Stock Manager in the tree.

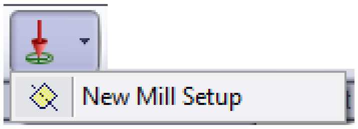
   OR

   Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   The *Stock Manager* dialog box is displayed.

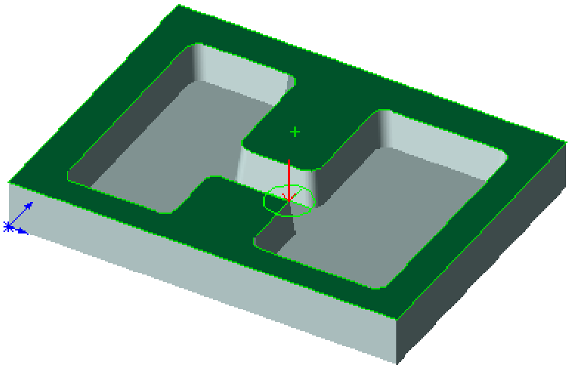
   – ✓ Click *OK* to use the default Stock (Bounding box) for this tutorial.
5. Right click the *Stock Manager* item in the Feature tree and select *New Mill Part Setup* on the context menu.

   OR



Click on the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu.

The *Mill Setup* dialog box is displayed.



Pick the top face of the part

6. In the graphics area, pick the top face on the model.
7. Make sure the *Face* option is not checked.
8. ✓ Click *OK* to insert the setup.

   Mill Part Setup1 is inserted in the tree.

Inserting a Multi Surface Feature

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

   The New Multi Surface Feature dialog box is displayed.
2. In the Feature Type group box, select the *All Displayed* option from the dropdown list.
3. Set the Strategy to *Area Clearance, Z Level*.

   This Strategy has been set up in the TechDB to generate Area Clearance and Z Level operations.
4. ✓ Click *OK* to insert the feature.
5. ✗ Click *Cancel* to close the dialog box.

   Multi Surface Feature1 is listed in the Feature tree.

Generate Operation for the Multi Surface feature

1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar/Command Manager.
2. Double click *Area Clearance1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
3. Click on the *Pattern* tab.
   - In the *Pattern Type* group box, use *Lace* for the Pattern .
   - In the *Pattern options* group box, set the *Cut angle* to **90deg**.
4. Click on the *Area Clearance* tab.
   - In the *Surface Finish* group box, set the *XY Allowance* to **0.15in** and the *Z allowance* to **0.15in**.
   - In the *Depth Parameters* group box, set the *Cut amount* to **0.25in**

5. Click on the *Links* tab.
    - In the *Links* group box, set the Type to *Spline Curve*.
    - In the *Corners* group box, set the Type to *Round*; *Max. radius* to **0.05in** and *Part deviation* to **0.02in**.
    - Ensure that the *Depth processing* is set to *By level*.
6. Click *OK* to close the Operation Parameters dialog box.
7. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
8. If such a warning is not displayed, right click *Area Clearance1* in the tree and select *Generate Toolpath* from the context menu.

Inserting an Area Clearance Operation for Rest Machining

Following are the steps to insert an Area Clearance operation to remove remaining material:

1. Right click *Area Clearance1* in the Feature tree and select *New 3 Axis Mill Operation* from the context menu. Select *Area Clearance* from the cascading menu.

    OR

    Highlight *Area Clearance1* in the tree. In the CAMWorks Workflow toolbar/Command Manager, click on the *New 3 Axis Mill Operation* button . Select *Area Clearance* from the dropdown menu.

    The *New Operation* dialog box is displayed.
2. Highlight *Multi Surface Feature1* in the Features list.
3. End Condition Click the *End Condition* button. Make sure the *Edit operation on creation* option is checked.
4. ✓ Click *OK* to insert the dialog box.

    The Operation Parameters dialog box for the newly inserted operation is displayed.
5. Click on the *Tool* tab.
    - Click on the *Tool Crib* page.
    - Highlight a **0.5in** Flat End Mill tool in the tool crib.
    - Click the *Select* button.
    - Click *Yes* to replace the corresponding holder.
6. Click on the *Pattern* tab.
    - In the *Pattern Type* group box, use *Pocket Out* for the Pattern.
7. Click on the *Area Clearance* tab.
    - In the Surface Finish group box, set the Allowance and the Z allowance to **0.05in**.
    - In the *Depth Parameters* group box, set the *Method* to *Constant* and *Cut Amount* to **0.125in**.
    - Click on the *Links* tab. In the Links group box, set the Type to *Spline Curve*.

- In the Corners group box, set the Type to *Round*; Max. radius to **0.05in**, Part deviation and Offset Deviation to **0.02in**.
- Ensure that the *Depth processing* is set to *By level*.

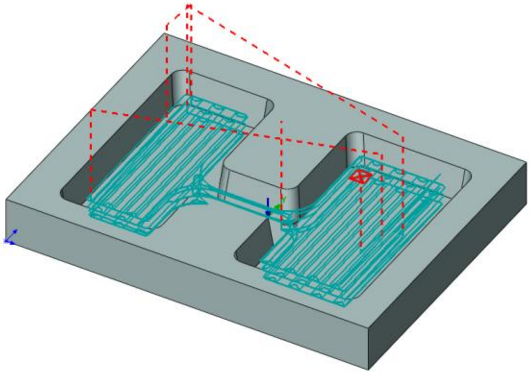
8. Click on the *Entry Retract* tab.
    - In the *Ramping* group box, set the Method to *Spiral*, Max. ramp angle to **20deg** and Spiral radius to **0.15in**.
9. Click on the *Rest* tab.
    - In the *Rest Machining* group box, set the Method to *From WIP*.
      When you select this method, CAMWorks assumes that toolpaths should be generated only where material is left by previous operations. CAMWorks creates a work in process stock model by starting with the original stock definition and removing material for the selected operations that occur before the rest machining operation. The work in process is then used to calculate the areas where the rest machining toolpaths should be generated. This method gives good results when used for roughing, where accuracy is not as important as speed.
10. Click *OK* to close the Operation Parameters dialog box.
11. ✖ Click *Cancel* to close the *New Operation* dialog box.
12. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
13. If such a warning is not displayed, right click *Area Clearance2* in the tree and select *Generate Toolpath* from the context menu.

    The Area Clearance operation *(Area Clearance2)* is listed in the Operation tree.

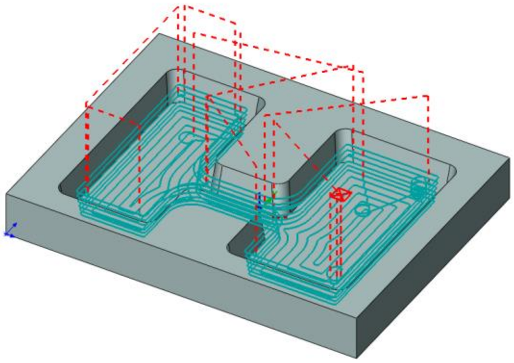
Simulating Material Removal

1. Highlight *Area Clearance1* in the tree and observe the toolpath in the graphics area.
2. Highlight *Area Clearance2* in the tree and observe the toolpath in the graphics area.

    Notice the rest machining toolpath removes material left by the previous operation.



Area Clearance1 Toolpath



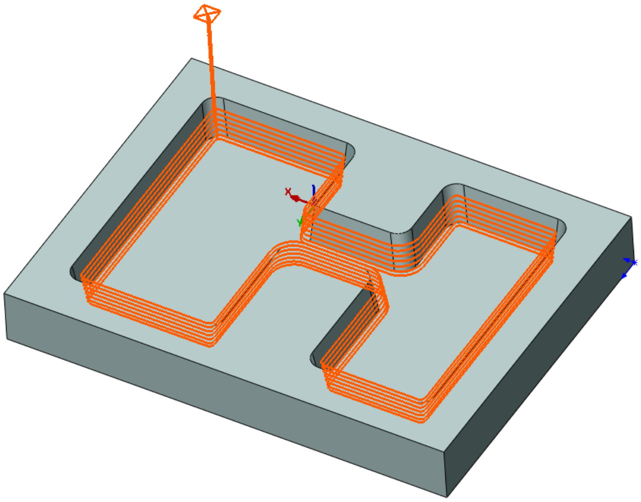
Area Clearance2 Toolpath

3. Click the *Simulate Toolpath button* on the CAMWorks Workflow toolbar/Command Manager.
4. The *Toolpath Simulation* toolbar is displayed. Set the End condition to *Next Operation*.
5. Click on the *Tool Mode* button to activate Tool mode.
6. ▶ Click *Run* to view each toolpath simulation.
7. ▶ Click *Run* again to simulate the rest machining operation.
8. Click ok ✓ exit the simulation mode.

Using an Automatic Contain Area

Edit the Z Level operation and generate toolpaths to machine the sides of the pockets:

1. Double click *Z Level1* in the operation tree.
2. Click on the *Tool* tab.
   - Select *Tool Crib* page and select a **0.3125in** Flat End Tapered tool.
   - Click *Yes* to replace the corresponding holder. The *Tapered Tool* page is displayed.
   - On the *Tapered Tool* page, change the *Taper angle* to **6deg**.
     The side walls of the part have a taper angle of 6deg. Hence, using a tapered tool with the same taper angle of 6deg gives a better finish toolpath.
3. Click on the *Links* tab.
   - Set the Depth processing to *By level*.
4. Click on the *Advanced* tab.
   - In the *Automatic contain area* group box, set the Method to *Outer Silhouette*.
     With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature. Note that faces to avoid in the Multi Surface feature are not used when calculating the silhouette.
   - Set the Tool Condition to *Upto*.
     With this condition, when the toolpath is generated, the XY extents of the toolpath are limited so that the full diameter of the tool is within the automatic contain area shape. If the Tool condition is set to *Past*, then the toolpath will not be generated as the



Z Level Toolpath

   full diameter of the tool falls outside the defined contain area shape of *Outer Silhouette*.

5. Click *OK*.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
7. If such a warning is not displayed, right click *Z Level1* in the tree and select *Generate Toolpath* on the context menu.

   CAMWorks generates the Z level toolpaths.

   Did You Know ...

   For 3 Axis operations (except 3 Axis Rough and Mill), the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either automatically on the Advanced tab in the Operation Parameters dialog box or interactively using the New Contain Area command.

   The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or not defined, then an automatically generated XY contain area will be used.

Interactively Inserting a Flat Area and Removing Material on Flat Areas

The Flat Area cycle uses a roughing pattern to remove material on the Z-levels of flat faces between the top and bottom of the surface. CAMWorks generates toolpaths only on completely flat areas. If a surface has even a small gradient, CAMWorks will not generate a toolpath. This cycle can be used for finishing where excess material has already been cleared.

Steps to insert and edit a Flat Area operation and generate toolpaths to machine the bottom of the pockets:

1. Right click *Z Level1* in the tree and select *New 3 Axis Mill Operation>> Flat Area* from the cascading context menu.

   OR

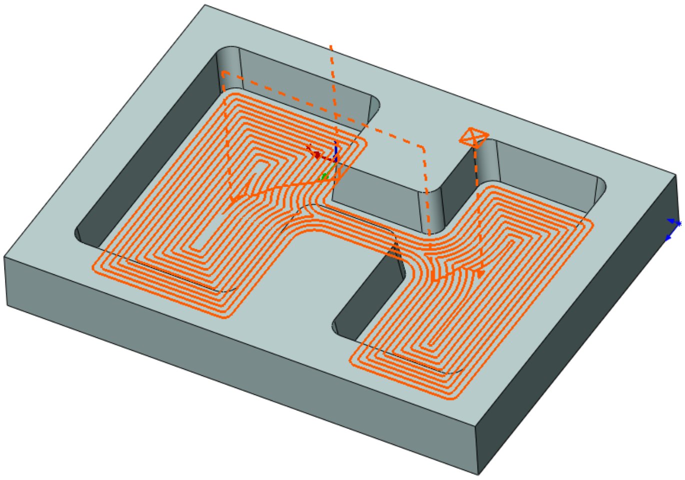
   Highlight *Z Level1* in the tree. In the CAMWorks Workflow toolbar/Command Manager, click on the *New 3 Axis Mill Operation* button . Select *Flat Area* from the dropdown menu.

   The New Operation dialog box is displayed.
2. Highlight Multi Surface Feature1 in the Features list.
3. End Condition Click the *End Condition* button. Make sure the *Edit Operation on creation* option is checked.
4. ✓ Click *OK* to insert the operation.

   The *Operation Parameters* dialog box is displayed.
5. Click on the *Flat Area* tab.

- In the Limits group box, set the First cut from to *Top of stock* with a **-0.05in** Offset.

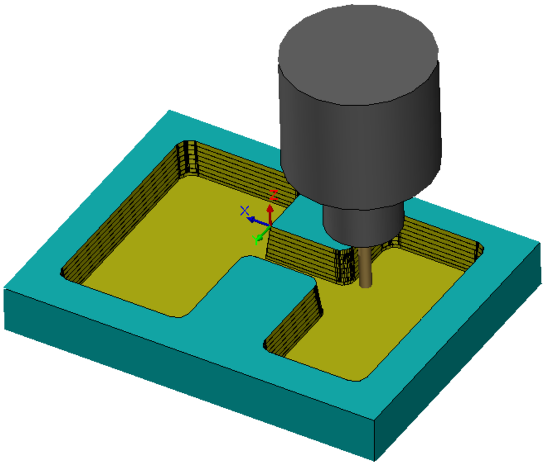
6. Click on the *Links* tab.
   - In the Links group box, set the type to *Spline Curve*.
   - In the Corners group box, set the Type to *Round*.
   - In the Corners group box, set the Part Deviation and Offset deviation to **0.02in.**
   - In the *Depth Processing* group box, set the Depth Processing to *By level*.
7. Click on the *Entry/Retract* tab.
   - In the Ramping group box, set the Method to *Ramp*.
   - In the Max. ramp angle field set to **20deg.**
8. Click *OK* to close the Operation Parameters dialog box.
9. Click *Cancel* to close the *New Operation* dialog box.
10. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
11. If such a warning is not displayed, right click *Flat Area1* in the tree and select *Generate Toolpath* on the context menu.



Flat Area Toolpath

Run the simulation

1. Click the *Simulate Toolpath button* on the CAMWorks Workflow toolbar/Command Manager.
2. The *Toolpath Simulation* toolbar is displayed. Set the End condition to *Next Operation*.
3. Click on the *Tool Mode* button to activate Tool mode.
4. Click *Run* to simulate the first operation.
5. Click *Run* again to simulate each operation.



Simulation Mode

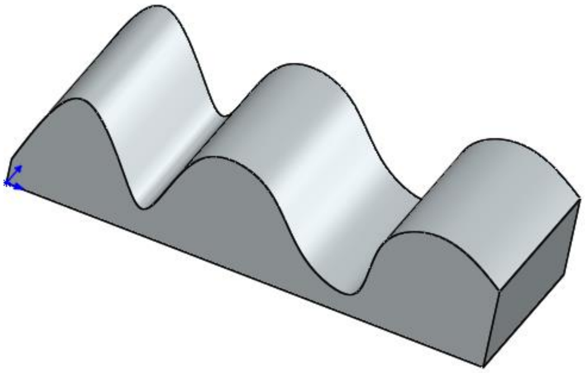
3 Axis Mill 8

This tutorial introduces you to finish rest machining using an STL file as the stock model. The STL file can be created from an external CAD system or a simulated stock model as explained in this tutorial.

Topics covered in this tutorial:

- Creating and Using an STL file as the stock model for rest machining
- Viewing the STL/WIP model
- Generating Pattern Project toolpaths for rest machining

Defining the Machine, Machining Direction and Multi Surface Feature



MILL3AXII_2.SLDPRT

1. Open the part file **MILL3AXII_2.SLDPRT** in the following folder.

   Drive: *\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\Mill*
2. Click the *CAMWorks Feature Tree* tab.
3. Double click *Machine [Mill-inch]* in the tree.

   OR

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

   The Machine dialog box is displayed.

   – On the Machine tab, ensure *Mill–inch* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

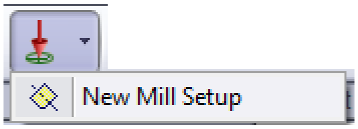
   – Click *OK* to close the Machine dialog box.
4. Double click Stock Manager in the tree.

   OR

   Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   The *Stock Manager* dialog box is displayed.
5. ✓ Click *OK* to use the default Stock (Bounding box) for this tutorial.

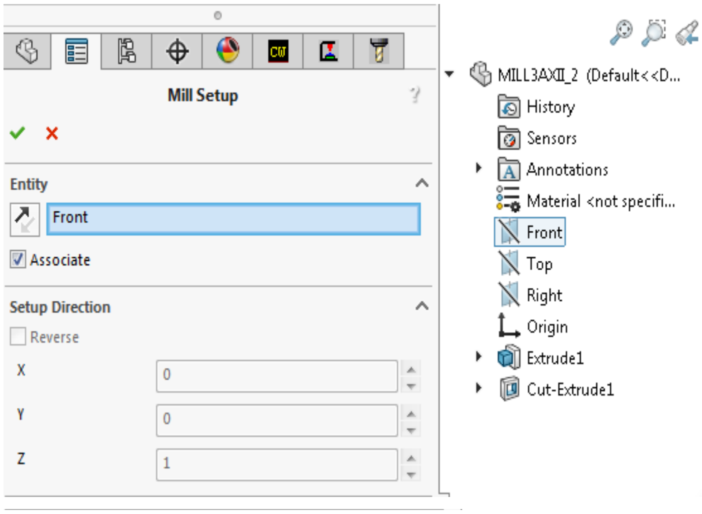
Defining the Machining Direction by inserting a Mil Part Setup



1. Click the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu.

   OR

   Right click *Stock Manager* item in the Feature tree and select *New Mill Part Setup* on the context menu.



Entity: Front

☑ Associate

Setup Direction

☐ Reverse

X: 0

Y: 0

Z: 1

Interactively Inserting a Mill Part Setup

2. The Mill Setup dialog box is displayed.
   - Click on the *Mill3AXII_2* to expand the tree in the graphics area and select *Front*.
   - Click the *Reverse Selected Entity* button to flip the Z axis up.
   - Make sure the *Face* option is not checked.
   - Click *OK* to insert the Mill Setup

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

Define a Multi Surface Feature

1. Right click *Mill Part Setup1* in the tree and select *New Multi Surface Feature* from the context menu.

   The *New Multi Surface Feature* dialog box is displayed.
2. Set the Feature type to *All Displayed*.

Did You Know ...

The Select All and Associate option in the New Multi Surface Feature dialog box which was available till the CAMWorks 2012 version has now been replaced with the All Displayed option available in the Feature Type list.

3. Set the Strategy to *Area Clearance, Pattern Project*.

   This Strategy is defined in the Technology Database to generate Area Clearance and Pattern Project operations.

4. Click *OK* to insert the operation.

   The new Multi Surface feature is listed under the Mill Part Setup in the Feature tree.

Generating an Operation Plan and Modifying Parameters

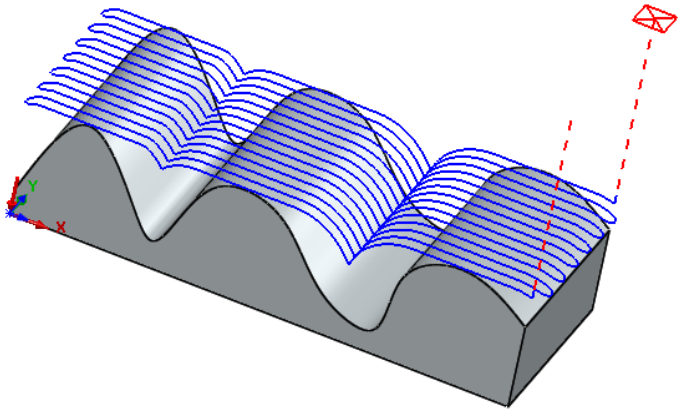
1. Click the *Generate Operation Plan* button on the CAMWorks Workflow toolbar.

   OR

   Right click the *Multi Surface Feature1* in the Feature tree and select *Generate Operation Plan* from the context menu.

   A Pattern Project operation is generated for the feature and listed in the Operation tree.
2. Double click *Pattern Project1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
3. Click on the *Tool* tab.
   - Select *Tool Crib* page and select a **1in** diameter Flat End Mill.
   - Click *Yes* to replace the corresponding holder.
4. Click on the *Pattern* tab.
   - In the *Pattern type* group box, set the Pattern to *Slice*.
     This pattern, which generates a series of linear parallel cuts across the part, is appropriate for semi-finishing almost any part shape and finishing non-vertical areas.
   - In the *XY Stepover* group box, set the Cut amt. to **0.15in**.
5. Click on the *Finish* tab.
   - Make sure the *XY Allowance* and *Z Allowance* are set to **0in**.
6. Click on the *Links* tab.
   - Set the Type to *Spline Curve*.
7. Click on the *Advanced* tab.
   - In the *Automatic contain area* group box, set the Method to *Outer Silhouette*.
     For this tutorial, you will use this contain area method to machine only the top of the feature.
   - Set the Tool Condition to *On center*.
8. Click *OK* to close the Operation Parameters dialog box.



Pattern Project Toolpath

9. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

   OR

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

Toolpath Simulation & Saving the WIP as an STL file

1. Click the *Simulate Toolpath* button either the CAMWorks Workflow toolbar OR the CAMWorks Command Manager.
2. The *Toolpath Simulation* toolbar is displayed. Leave the End condition set to *End*.
3. Click the *Turbo* button.

   Turbo mode provides ultra-fast simulation of toolpaths on a single Mill Part Setup.
4. Click *Run* button to view each toolpath simulation.
5. Click the *Save WIP as STL* button on the *Simulate Toolpath* toolbar.
6. In the *Save As* dialog box,
   - Browse to the folder where you wish to save the file.
   - Assign a desired file name
   - Select *Inches* for the Units
   - Click the *Save* button
7. The WIP is saved in an STL file that you will use for rest machining.
8. Click *OK* *button to* Exit the simulation.

Did You Know ...

You can also save an STL file using the Save WIP Model command on the Stock Manager context menu in the Operation tree. With this method, you select the operations to be used in calculating the WIP from a list, then define the file name and applicable options. The Toolpath Simulation runs in the background to generate the STL file.

Inserting a Pattern Project Operation for Rest Machining

1. Highlight *Pattern Project1* in the Operation tree. In the CAMWorks Workflow toolbar, select *New 3 Axis Mill Operation* . Select *Pattern Project* from the dropdown menu.

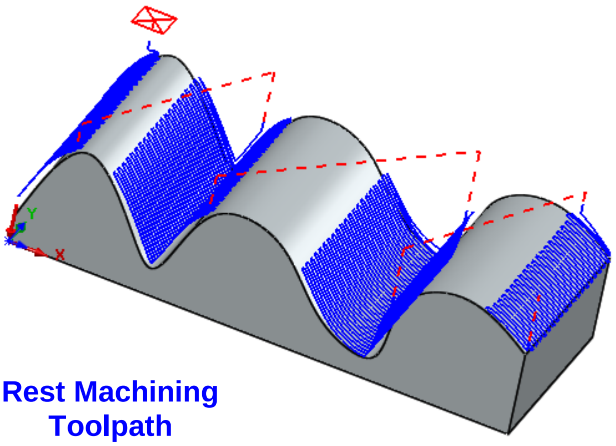
   OR

   Right click the *Pattern Project* operation in the tree and select *New 3 Axis Mill operations>>Pattern Project* from the context menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.

3. End Condition ➔ Click the *End Condition* button. Leave the *Edit Operation on creation* option checked.
4. ✓ Click *OK* to insert the operation.

   The *Operation Parameters* dialog box for the newly inserted Pattern Project operation is displayed.
5. Click on the *Tool* tab.
   - Select *Tool Crib* page and select a **0.25in** diameter Ball Nose tool.
   - Click *Yes* to replace the corresponding holder.
6. Click on the *Pattern* Tab.
   - Select *Slice* for the Pattern type.
   - In the *Pattern Options* group box, check the *Staydown* option.
7. Click on the *Finish* tab.
   - Make sure the *XY allowance* and *Z allowance* are set to **0in**.
   - Ensure that *Avoid allowance* is set to **0.025in**.
8. Click on the *NC* tab.
   - In the *Rapid Plane is* group box, set the *Distance* to **0.25in**.
9. Click on the *Links* tab.
   - Set the Type to *Spline Curve*.
   - Set the *Max. link distance* to **0.1in**.
10. Click on the *Entry/Retract* tab.
    - In the Ramping group box, set the Method to *Ramp*.
    - Set the Z offset to **0.1in.**

      This action enables the Z offset field in this group box.
11. On the *Advanced* tab, set the following.
    - In the *Automatic contain area* group box, set the Method to *Outer Silhouette*.
    - Ensure that the *Tool Condition* in this group box is set to *On center*.
12. Click on the *Rest* tab.
    - In the Rest Machining group box, select *From STL File* for the Method.
    - ... Click the *Browse* button.
    - In the *Open* dialog box, locate the STL file you created in the previous step, select *Inches* for the Units and then click *Open* button.



Rest Machining Toolpath

  - The location path of the selected STL file will be displayed *STL path* field within the group box.

13. Click *OK* to apply the changes and to close the Operation Parameters dialog box.
14. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
15. If such a warning is not displayed, right click the *Pattern Project2* and select the Generate Toolpath command on the context menu.

Viewing the STL/WIP Model

To view the shape and position of the STL stock model, you can do one of the following:

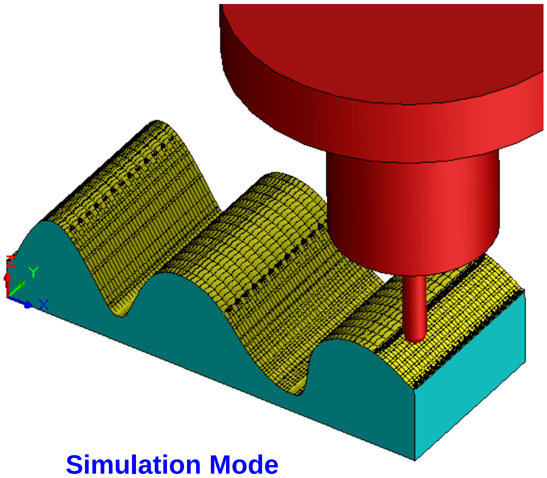
- Click the Stock Manager in the tree to view a bounding box indicating the extents of the stock model. The display may be either a wireframe, translucent or shaded model depending on the Display options. You can set the Display options in the RMB context menu of the Stock Manager item.
- Click the *Simulate Toolpath* button on the CAMWorks toolbar.

Generating the Rest Machining Toolpaths

1. Right click *Pattern Project1* operation in the Operation tree and select *Suppress* from the context menu.
2. Double click the *Stock Manager* item in the tree. The *Stock Manager* dialog box is displayed.
3. Select *STL File* for Stock type.
4. Click on the Browse button next to the *STL Path* field.
5. Locate and select the STL file you are using as WIP model and which you created in the previous step.
6. In the Open dialog box, ensure that the Units is set to *Inches*.
7. Click *Open*.
8. Click *OK* to close the Manage Stock dialog box.

Stimulating the Toolpath

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



Simulation Mode

2. Observe that STL stock is displayed in the graphics area.
3. ▶ Click *Run* and view the results.

   CAMWorks generates toolpaths only where material is left on the stock model in the STL file.

3 Axis Mill 9

This tutorial demonstrates how to engrave using a Curve Project cycle. This cycle removes material by projecting selected 2.5 Axis Engrave and/or Curve features onto the surface of a Multi Surface feature and generating toolpaths along the projected entities.

Topics covered in this tutorial:

- Defining a Multi Surface Feature and a 2.5 Axis Engrave Feature
- Inserting a Curve Project operation and adding an Engrave Feature

Defining the Machine and Machining Direction

1. Open the part file **MILL3AXII_3.SLDPRT** located in the following folder.

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

2. Click the *CAMWorks Feature Tree* tab.

MILL3AXII_3.SLDPRT

3. Double click the *Machine [Mill-inch]* item.

   OR

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

   The Machine dialog box is displayed.

   – On the Machine tab, ensure *Mill–inch* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

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

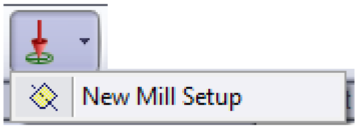
4. Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

   OR

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

   The *Stock Manager* dialog box is displayed.

5. The default bounding box displays on the part. ✓ Click *OK* to use the default stock.

6. 

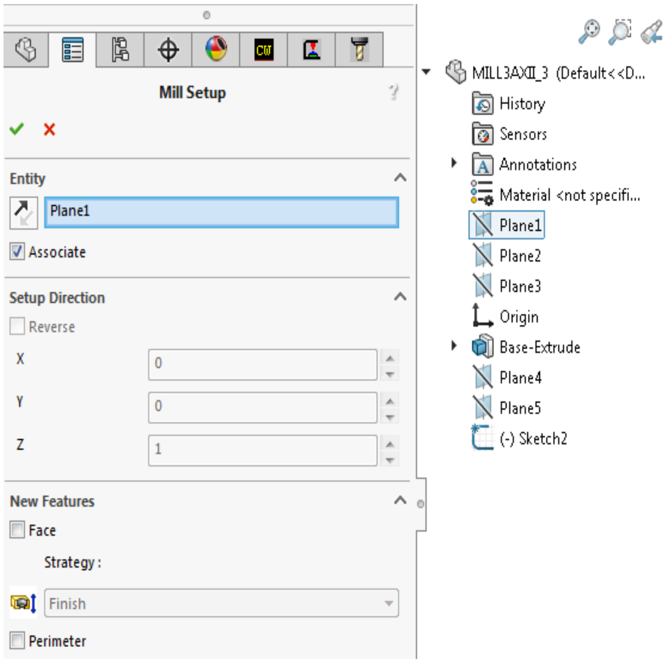
Click the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu

OR

Right click *Stock Manager* item in the tree and select *New Mill Part Setup* on the context menu.

7. The Mill Setup dialog box is displayed.
   - – Click on the *Mill3AXII_3* to expand the tree in the graphics area and select *Plane2*.
   - – Click the *Reverse Selected Entity* button to flip the Z axis up.
   - – Make sure the *Face* option is not
   - – checked.
   - – ✓ Click *OK* to insert the Mill Setup

   The new Mill Part Setup *(Mill Part Setup1)* is listed in the Feature tree.



Interactively Inserting a Mill Part Setup

Creating a Multi Surface Feature and Engrave Feature

Following are the steps to interactively insert a multi surface feature:

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

   The *New Multi Surface Feature* dialog box is displayed.

2. Set the Feature type to *All Displayed*.

   Did You Know ...

   The Select All and Associate option in the New Multi Surface Feature dialog box which was available till the CAMWorks 2012 version has now been replaced with the All Displayed option available in the Feature Type list.

3. Set the Strategy to *Area Clearance, Pattern Project*.

   This Strategy is defined in the Technology Database to generate Area Clearance and Pattern Project operations.

4. ✓ Click *OK* to insert the operation.

5. ✗ Click *Cancel* to close the *New Multi Surface Feature* dialog box.

The new Multi Surface feature is listed under the Mill Part Setup in the Feature tree.

6. Right click *Mill Part Setup1* and select *New 2.5 Axis Feature* on the context menu.

   The *2.5 Axis Feature: Select Entities* dialog box is displayed.
7. Select *Engrave Feature* for the Feature type.
8. Select *Sketch2* in the Part Sketches list. This sketch is then listed in the Selected Entities list box.
9. End Condition Click the *End Condition* button.

   The *2.5 Axis Feature: End Conditions* dialog box is displayed.
10. In the End condition group box, set the Depth to **0.1in**.
11. Click *OK*.
12. Click *Close* to apply the changes and to close the dialog box.

    The Engrave Feature is listed under the Mill Part Setup in the Feature tree.

Generating an Operation Plan and Modifying Parameters

Inserting Contain Area

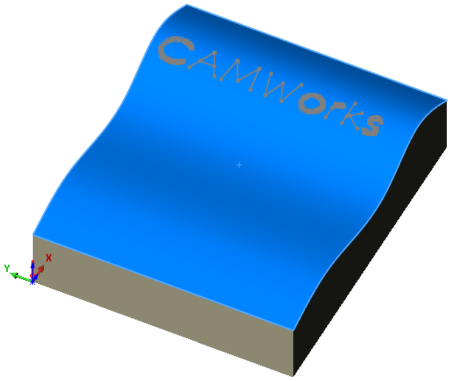
1. In this tutorial, we will automatically generate operations only for the Multi Surface Feature. The operation to machine the Engrave feature will be inserted interactively.

   Hence, right click the *Multi Surface Feature1* in the Feature tree and select *Generate Operation Plan* from the context menu. A Pattern Project operation is generated for the Multi Surface Feature.
2. Highlight the newly inserted *Pattern Project1* operation in the tree and click on the *New Contain Area* button on the CAMWorks Command Manager.

   OR

   Right click *Pattern Project1* in the Operation tree and select *New Contain Area* from the context menu.

   The Contain Area dialog box is displayed.



Pick the top face

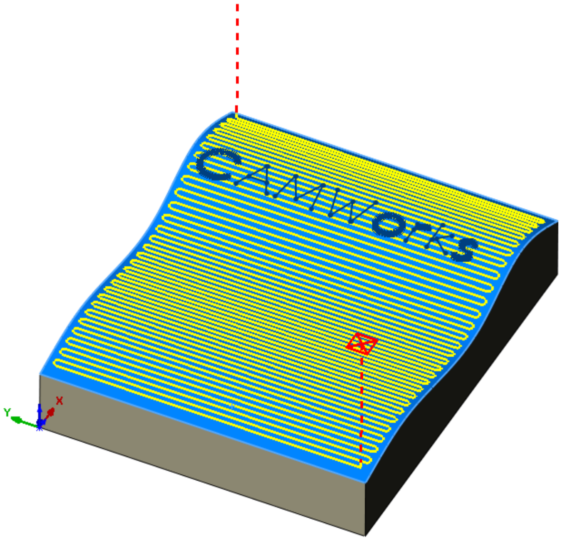
3. In the Selection Mode group box, ensure that the *Face Selection* is set to *Single Face*.
4. Pick the top face of the model part (*highlighted in blue*). The selected face will be listed in the Selected Contours list box.
5. In the Offset group box, set the Offset to **0.02in**.
6. ✓ Click *OK* to insert the Contain Area.

Modifying Operation Parameters

1. Double click *Pattern Project1* in the Operation tree.

   The Operation Parameters dialog box is displayed.
2. Click the *Tool* tab.
   - Select *Tool Crib* page and select a **0.375in** diameter Ball Nose tool.
   - Click *Yes* to replace the corresponding holder.
3. Click on the *Pattern* tab.
   - Select *Slice* for the Pattern type.
   - In the *Pattern Options* group box, set the *Cut angle* to **90deg**.
   - In the *XY Stepover* group box, set the *Cut amount* to **0.1in**.
4. On the *Finish* tab.
   - In the Surface Finish group box, ensure that the *XY Allowance* and *Z Allowance* are set to **0in**.
   - In the Limits group box, set the *First cut from* to *Top of stock* with **0in** as *Offset*.
5. Click on the *Links* tab.
   - Set the *Max link distance* to **1in**.
6. Click on the *Advanced* tab.
   - In *Automatic contain area* group box, set the Method to *Outer Silhouette*.

     With this method, the XY shape of the contain area is based only on an outer XY silhouette of the faces to cut in the Multi Surface feature. Note that faces to avoid in the Multi Surface feature are not used when calculating the silhouette.
   - Ensure that the Tool Condition is set to either *On center* or *Upto*.
7. Click *OK* to close the Operation Parameters dialog box.
8. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.



Pattern Project Toolpath

Inserting a Curve Project Operation to Cut the Text

1. Right click the *Pattern Project1* operation in the tree and select *New 3 Axis Mill Operations>>Curve Project* from the context menu.
2. The *New Operation* dialog box is displayed. Highlight *Multi Surface Feature1* in the Features list.
3. Click *Next*. Leave the *Edit Operation on creation* option checked.

4. ✓ Click *OK* to insert the operation.

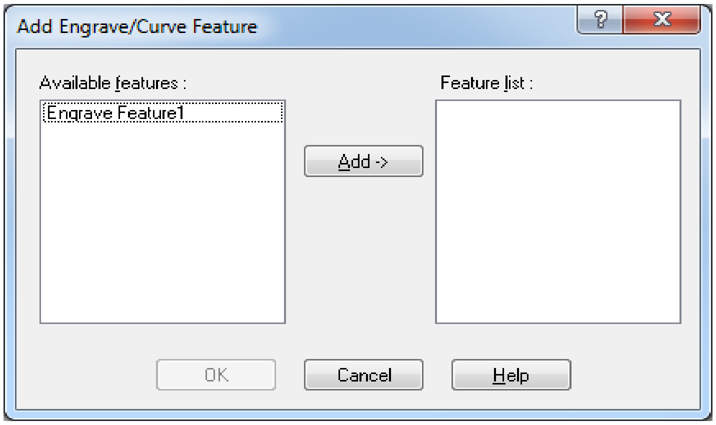
   The *Operation Parameters* dialog box for the newly inserted Pattern Project operation is displayed.

5. Click on the *Tool* tab.
   - On the Mill Tool page, change the Cut diameter to **0.063in**.
   - Select *Tool Crib* page and select a **0.0625in** diameter Ball Nose tool.
     In case a Ball Nose Tool of this diameter is not available in the Tool Crib, use the Tool Library to add a Ball Nose tool with this diameter to the Tool Crib and then select this tool for the operation.
   - Click *Yes* to replace the corresponding holder.
6. Click on the *Curve Project* Tab.
   - Make sure the *XY allowance* is set to **0in**
   - Set the *Z allowance* are set to **-0.05in**.
   - In the *Axial offsetting* group box, ensure that the *Number of cuts* is set to **1** and the *Axial offset* is set to **0.1in**.
7. Click on the *Links* tab.
   - Set the *Max. link distance* to set to **0in**.
8. Click on the *Entry/Retract* tab.
   - Ensure that the *Retract Method* is set to *Full*.
     With this Retract method, the tool moves vertically up to the clearance plane, moves horizontally along this level, and drops down vertically to the start of the next pass.
   - In the *Vertical leadin/out* group box, set both the Leadin radius and Leadout radius to **0in.**
   - In the *Horizontal leadin/out* group box, set both the Leadin radius and Leadout radius to **0in.**
9. Click *OK* to close the Operation Parameters dialog box.

   The *Curve Project1* operation is listed in the Operation tree.

10. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
11. Right click *Curve Project1* in the tree and select *Add Engrave/Curve Feature* from the context menu.

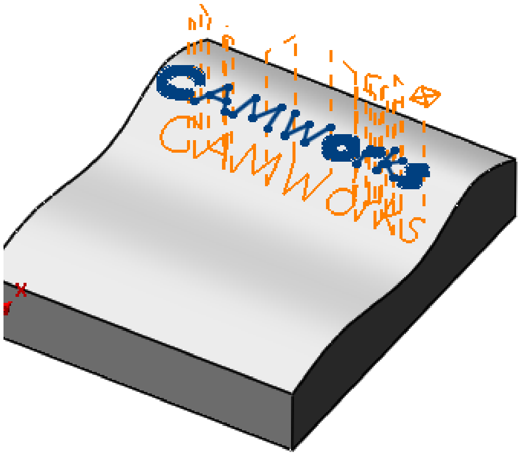
    The Add Engrave/Curve Feature dialog box is displayed.



Add Engrave/Curve Feature dialog box

12. In the Engrave/Curve Feature dialog box, pick *Engrave Feature1* in the *Available features* list.
13. Click the *Add* button.
14. Click *OK* to close this dialog box.
15. Right click *Mill Part Setup1* in the Operation tree and select *Generate Toolpath* from the context menu.

    CAMWorks generates toolpaths for the Multi Surface Feature and the text.

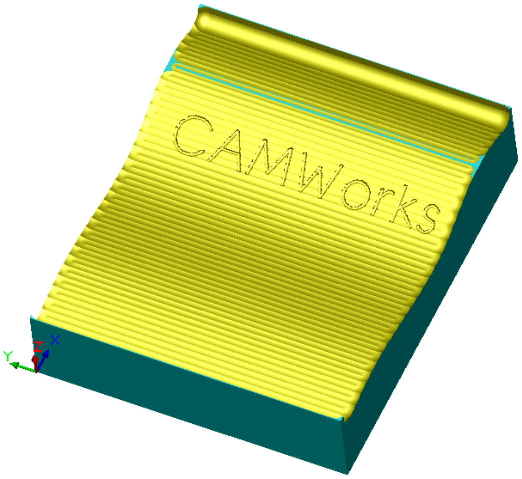


Curve Project Toolpath

Running Toolpath Simulation

1. Click the *Simulate Toolpath* button either the CAMWorks Workflow toolbar or the CAMWorks Command Manager.

   The *Toolpath Simulation* toolbar is displayed.
2. Set the End condition set to *Next Operation*.
3. Click the *Tool Mode* button.
4. Click on the *Tool* button. Set the display to *Shaded With Edges*.
5. Click on the *Holder* button. Set the display to *Shaded With Edges*.
6. Click *Run* button to view each toolpath simulation.



Simulation Mode

3 Axis Mill 10

This tutorial uses a Constant Stepover operation to remove material by maintaining a constant user-defined stepover relative to the surface. This cycle generates a semi-finish or finish toolpath with a true constant stepover regardless of the slope being machined.

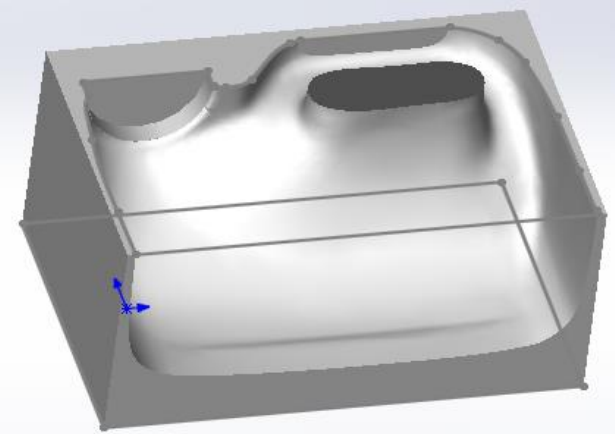
Topics covered in this tutorial:

- Defining a Multi Surface Feature
- Adding Contain and Avoid areas to Constant Stepover Operation

Defining the Machine and Machining Direction

1. Open the part file **MILL3AXII_4.SLDPRT** in the following folder.

Dr*ive:\CAMWorksData\CAMWorks202x\Examples\ Tutorial_Parts\Mill*


MILL3AXII_4.SLDPRT

2. Click the *CAMWorks Feature Tree* tab.
3. Double click on the *Machine [Mill–metric]* item in the tree.

   OR

   Click on the *Define Machine* button on the CAMWorks Workflow toolbar. The Machine dialog box is displayed.

   – On the Machine tab, ensure *Mill–metric* is the Active machine.

   – Click the *Tool Crib* tab and make sure *Crib 1* is the Active tool set.

   – Click the *Post Processor* tab and make sure *M3AXIS-TUTORIAL* is selected.

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

4. Click on the *Stock Manager* button on the CAMWorks Workflow toolbar.

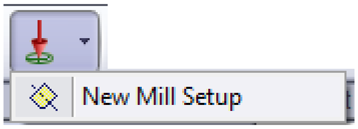
   OR

   Double click *Stock Manager* item in the tree.

   The *Stock Manager* dialog box is displayed. The default bounding box displays on the part.

   – Click *OK* to use the default stock.

Defining the Machining Direction by inserting a Mil Part Setup



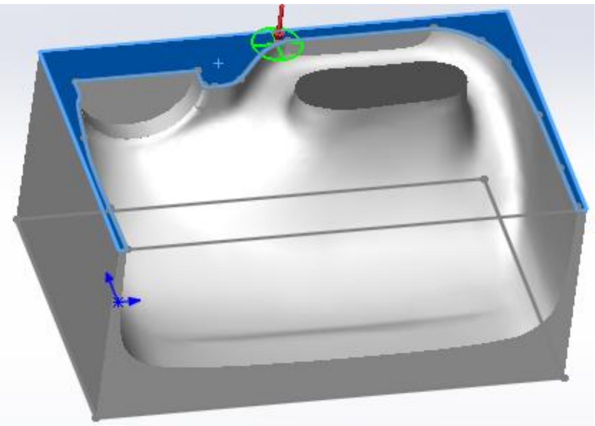
1. Click the *New Setup* button on the CAMWorks Workflow toolbar. Select *New Mill Setup* from the dropdown menu.

   OR

   Right click *Stock Manager* item in the Feature tree and select *New Mill Part Setup* on the context menu.

   The Mill Setup dialog box is displayed.
2. Pick the top face of the part model (highlighted in blue color).
3. Make sure the *Face* option is not checked.
4. ✓ Click *OK* to insert the Mill Setup

   The new Mill Part Setup *(Mill Part Setup1)* is listed in the Feature tree.



Pick the top face

Creating a Multi Surface Feature

Following are the steps to interactively insert a multi surface feature:

1. Right click *Mill Part Setup1* in the Feature tree and select *New Multi Surface Feature* from the context menu.

   The *New Multi Surface Feature* dialog box is displayed.
2. Set the Feature type to *All Displayed*.

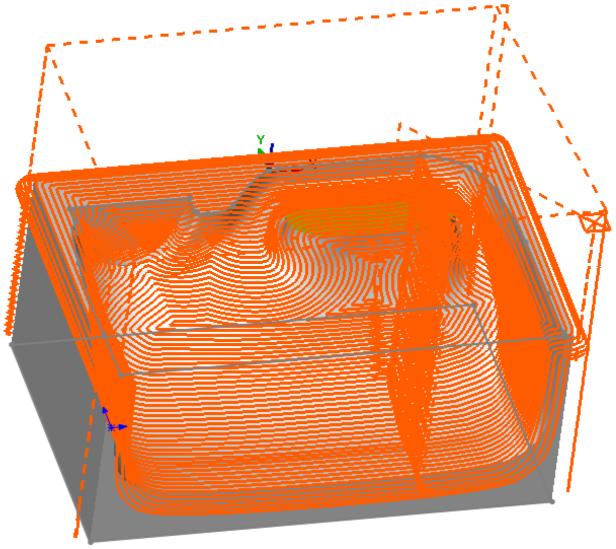
   Did You Know ...
   The Select All and Associate option in the New Multi Surface Feature dialog box which was available till the CAMWorks 2012 version has now been replaced with the All Displayed option available in the Feature Type list.
3. Set the Strategy to *Area Clearance, Constant Stepover*.

   This Strategy is defined in the Technology Database to generate a Constant Stepover operation.
4. ✓ Click *OK* to insert the operation.
5. Click the *Generate Operation Plan butt*on on the CAMWorks Command Manager.
6. Click the Generate Toolpath button on the CAMWorks Command Manager.

Modifying the Operation Parameters of the Constant Stepover Operation

1. Double click on the *Constant Stepover* operation in the Operation tree.

   The *Operation Parameters* dialog box is displayed.
2. Click on the *Mill Tool* page on the Tool tab.
3. Make sure the Flute Length set to **55mm**.
4. Click on the *Constant Stepover* tab. Set the following parameters:
   - In the *Direction control* group box, set the control to *Zigzag*.
   - In the *Pass options* group box,
     - Set the Method to *3D Stepover*.
     - Set the *Horizontal cut amt.* to **2mm**.
     - Set the *Vertical cut amt.* to **2mm**.
     - Ensure that the *Reverse order* checkbox is checked.
   - In the Limits group box,
     - Set the First cut from to *Top of Stock*.
     - Set the *Offset* to **2.5mm**.
5. Click *OK* to close the Operation Parameters dialog box.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.



Constant Stepover Toolpath

Inserting Contain and Avoid Areas

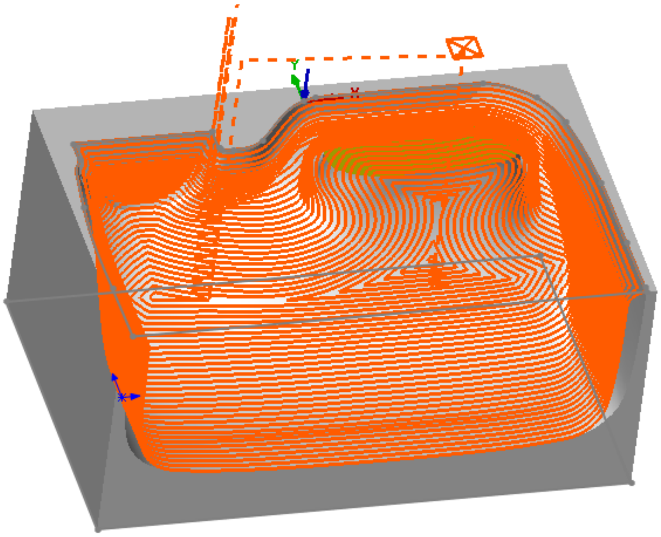
Inserting Contain Area

1. Highlight the newly inserted *Constant Stepover1* operation in the tree and click on the *New Contain Area* button on the CAMWorks Command Manager.

   OR

   Right click *Constant Stepover1* in the Operation tree and select *New Contain Area* from the context menu.

   The Contain Area dialog box is displayed.



Constant Stepover Toolpath contained to a defined area

2. In the *Allowed Sketches* list, pick *Contain Area*. This sketch is then listed in the *Selected Contours* list box.
3. In the Offset group box, click on the *Tool Condition* button. Set the Tool Condition to *Tool on Curve*.
4. In the Offset group box, set the Offset to **0.5mm**.
5. ✓ Click *OK* to insert the Contain Area.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

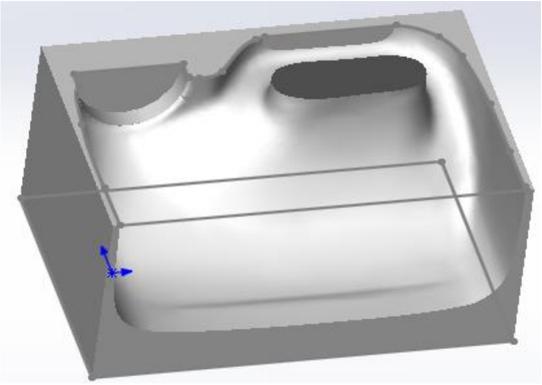
Inserting Avoid Area

1. Highlight the newly inserted *Constant Stepover1* operation in the tree and click on the *New Avoid Area* button on the CAMWorks Command Manager.

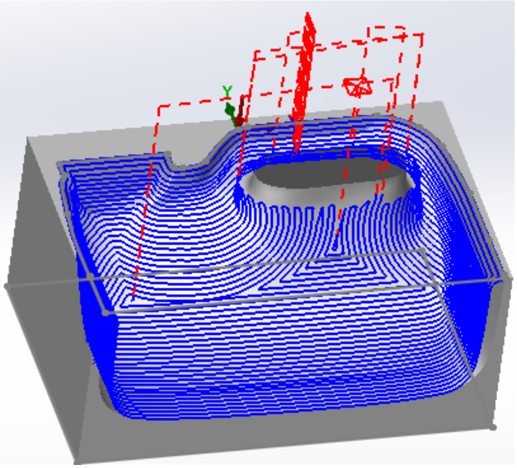
   OR

   Right click *Constant Stepover1* in the Operation tree and select *New Avoid Area* from the context menu.

   The Avoid Area dialog box is displayed.
2. In the *Selection Mode* group box, ensure that Face Selection is set to Single Face.
3. Pick the dark grey colored face on the part. The face is then listed in the *Selected Contours* list box.
4. In the Offset group box, click on the *Tool Condition* button. Set the Tool Condition to *Tool on Curve*.
5. ✓ Click *OK* to insert the Avoid Area.
6. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.
7. If such a warning is not displayed, click on the *Generate Toolpath* command on the CAMWorks Workflow toolbar/Command Manager.
8. Click the ⊞ sign next to *Constant Stepover1* in the tree. *Contain Area1* and *Avoid Area1* are listed under the operation.

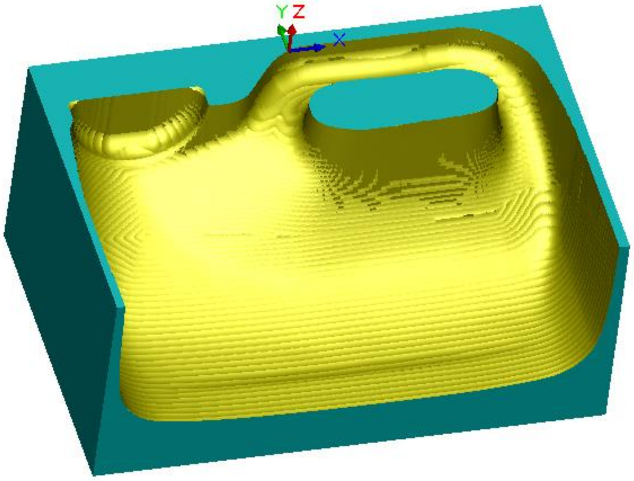


Pick the dark grey colored face



Constant Stepover Toolpath after insertion of Contain and Avoid Area

Simulate the Toolpath



Toolpath Simulation

1. Click the *Simulate Toolpath* button on the CAMWorks toolbar.
2. Click *Turbo mode* button.
3. Click *Run*.

   Observe the resultant toolpath.

3 Axis Mill 11

For 3 Axis operations, the XY extents will be within a contain area while machining over and around the faces and surfaces in the Multi Surface feature. The contain area can be defined either interactively using the New Contain Area command or automatically on the Advanced tab in the Operation Parameters dialog box as shown in this tutorial.

The priority is to observe the XY shape of any interactively defined contain areas. If interactively defined contain areas are suppressed or if none have been defined, then an automatically generated XY contain area will be used.

Topics covered in this tutorial:

- Understanding the differences between the various Automatic Contain Area methods

Open the Part

1. Open the part file **MILL3AXII_5.SLDPRT** in the following folder.

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

2. Click the *CAMWorks Operation Tree* tab.

   An Area Clearance operation and Z Level operation have already been generated for the Multi Surface feature.

Using Automatic Contain Areas

Stock Method

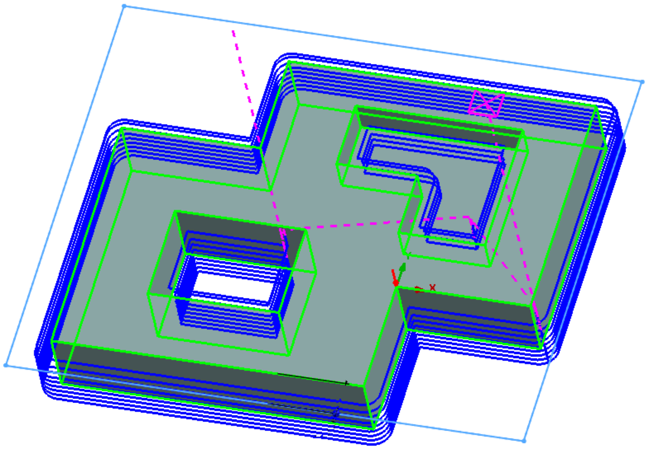
1. Double click *Z Level1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

2. Click on the *Advanced* tab. In the *Automatic contain area* group box, notice that the Method is set to *Stock*.

   When you use this method, the XY shape of the contain area is based on the original stock shape. If the stock shape is defined from a sketch and the current Setup is normal to the sketch, the contain area will be the same as the sketch. Otherwise, the contain area will be based on the minimum XY bounding box of the stock.

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



Automatic Contain Area set to 'Stock'

4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

   OR

   Right click the *Z Level1* operation in the tree and select Generate Toolpath from the context menu.

   OR

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

5. In the FeatureManager design tree, position the pointer over the *Stock* sketch item.

   The Stock shape is represented by the rectangular colored sketch.

Bounding Box Method

1. Double click *Z Level1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

2. Click on the *Advanced* tab. In the *Automatic contain area* group box, notice that the Method is set to *Bounding Box*.

   With this method, the XY shape of the contain area is based on the minimum XY bounding box of the faces in the Multi Surface feature. The bounding box is based only on the *Faces to Cut* in the Multi Surface feature.

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

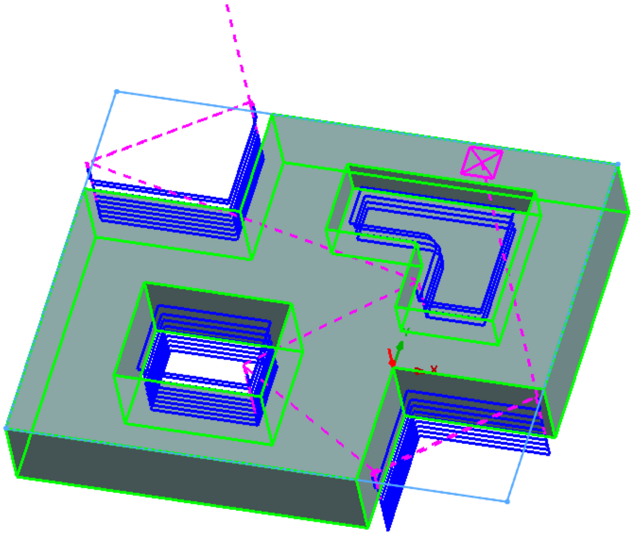
4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

   OR

   Right click the *Z Level1* operation in the tree and select *Generate Toolpath* from the context menu.

   OR

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



Automatic Contain Area set to ‘Bounding Box’

5. In the FeatureManager design tree, position the pointer over the *Bounding Box* sketch item.

   The Bounding Box is represented by the rectangular colored sketch.

Outer Silhouette Method

1. Double click *Z Level1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

2. Click on the *Advanced* tab. In the *Automatic contain area* group box, notice that the Method is set to *Outer Silhouette*.

   With this method, the XY shape of the contain area is based only on an outer XY silhouette of the *Faces to Cut* in the Multi Surface feature.

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

4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

   OR

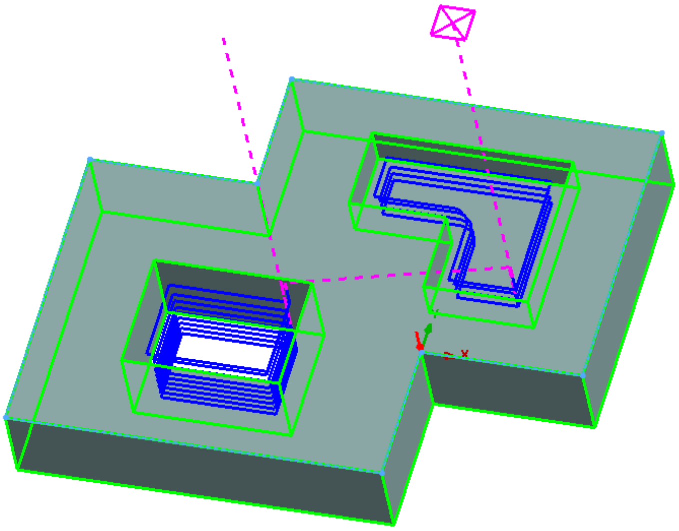
   Right click the *Z Level1* operation in the tree and select *Generate Toolpath* from the context menu.

   OR

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

5. In the FeatureManager design tree, position the pointer over the Outer Silhouette sketch item.

   The sketch is displayed on the perimeter of the top face represents the Outer Silhouette.



Automatic Contain Area set to 'Outer Silhouette'

All Silhouettes Method

1. Double click *Z Level1* in the Operation tree.

   The Operation Parameters dialog box is displayed.

2. Click on the *Advanced* tab. In the *Automatic contain area* group box, notice that the Method is set to *All Silhouettes*.

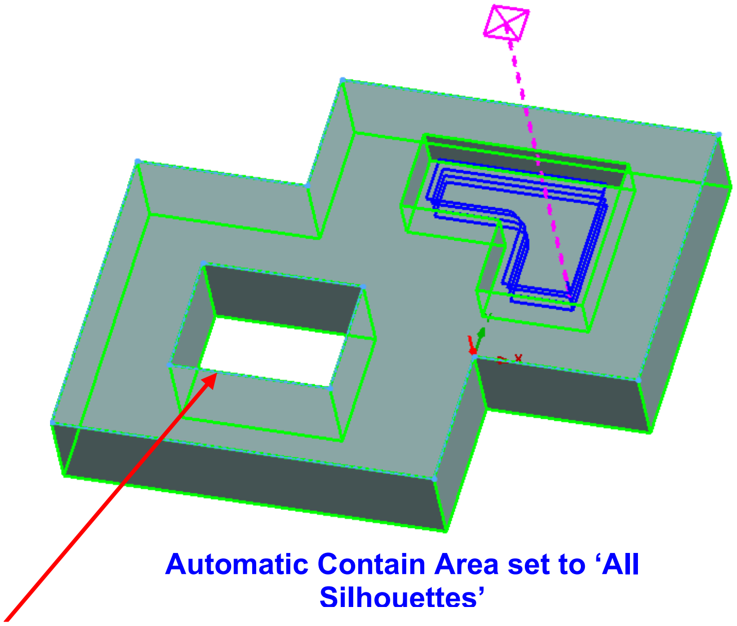
   With this method, the XY shape of the contain area is based on an XY silhouette of Faces to Cut in the Multi Surface feature. The silhouette will include the outer shape of the faces as well as any internal shape based on through features. The internal silhouette areas will be treated automatically as avoid areas.

3. Click *OK* to close the Operation Parameters dialog box.
4. Click *Yes* if CAMWorks displays a warning message asking if you want to regenerate the toolpaths.

   OR

   Right click the *Z Level1* operation in the tree and select *Generate Toolpath* from the context menu.
5. In the Feature Manager design tree, position the pointer over the *All Silhouettes* sketch item.

   The displayed sketch represents the outer silhouette of the *Selected Faces* in the Multi Surface Feature and the internal silhouette for the through feature.



Automatic Contain Area set to 'All Silhouettes'

Internal Silhouette for the through feature