UNIGRAPHICS

MILL MANUFACTURING PROCESS

STUDENT MANUAL

September 2002

MT11040 — Unigraphics NX

EDS Inc.

Proprietary & Restricted Rights Notices

Mill Manufacturing Process Student Manual Publication History:

Version 16.0 .... March 1999
Version 17.0 .... January 2001
Version 17.0 .... June 2001
Version 18.0 .... September 2001
Unigraphics NX .... September 2002

Table of Contents

(This Page Intentionally Left Blank)

Course Overview

Course Description

The Mill Manufacturing Process (MMP) course teaches the use of the Unigraphics NX Manufacturing application for creating 2-1/2 and 3-axis tool paths.

Intended Audience

This course is designed for Manufacturing Engineers, Process Planners and NC/CNC Programmers that have the basic knowledge of NC/CNC manual programming of 3-axis positioning and contouring equipment.

Prerequisites

- Practical Applications of Unigraphics course or CAST equivalent
- Basic understanding of the Master Model concept

Working knowledge of the following:

- Unigraphics User Interface
- Part file saving conventions
- Experience as an NC/CNC programmer

Objectives

After successfully completing this course, you should be able to perform the following activities in Unigraphics:

- Create operations and tool paths
- Create Parent Group objects that supply information to operations

- Utilize options and parameters that are common to various operation types
- Create drilling, planar milling and contour operations

Student Responsibilities

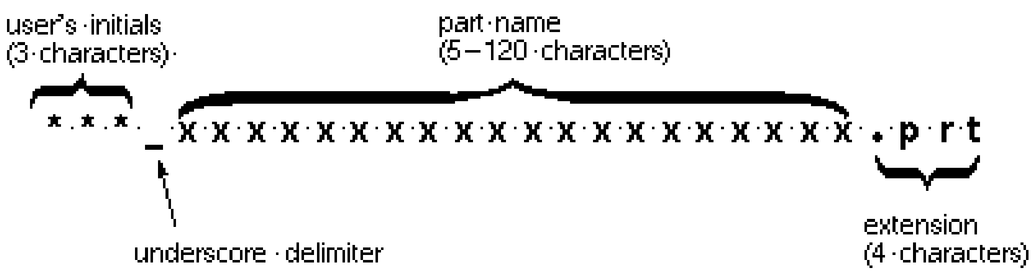
- Be on time
- Participate in class
- Focus on the subject matter
- Listen attentively and take notes
- Practice what you have learned
- Enjoy the class

Class Standards for Unigraphics Part Files

The following standards will be used in this class. Standardization allows users to work with others parts while being able to predict the organization of the part file. All work should be performed in accordance with these standards.

Class Part File Naming

This class utilizes the following file naming standard:



Where the student is requested to save a part file for later use, the initials of the student's given name, middle name, and surname replace the course identifier "***" in the new filename with the remainder of the filename matching the original. These files should reside in the student's personal directory.

TIP Currently up to 128 characters are valid for file names. A four character extension (.prt, for example) is automatically added to define the file type. This means the maximum number of user defined characters for the file name is actually 124.

Layers and Categories

The student will notice that there are standard layer assignments as well as standard category names in each of the part files as follows:

Layers 1–100, Model Geometry (Category: MODEL)

- Layers 1–14, Solid Geometry (Category: SOLIDS)
- Layers 15–20, Linked Objects (Category: LINKED OBJECTS)
- Layers 21–40, Sketch Geometry (Category: SKETCHES)
- Layers 41–60, Curve Geometry (Category: CURVES)
- Layers 61–80, Reference Geometry (Category: DATUMS)
- Layers 81–100, Sheet Bodies (Category: SHEETS)

Layers 101 – 120, Drafting Objects (Category: DRAFT)

Layers 101 – 110, Drawing Borders (Category: FORMATS)

Layers 121 – 130, Mechanism Tools (Category: MECH)

Layers 131 – 150, Finite Element Meshes and Engr. Tools (Category: CAE)

Layers 151 – 180, Manufacturing (Category: MFG)

Layers 181 – 190, Quality Tools (Category: QA)

Colors

The following colors are preset to indicate different object types:

Object	***Valid colors***
Bodies	
Solid	*Green*
Sheet	*Yellow*
Generating Curves (non-sketch)	
Lines and Arcs	*Orange*
Conics and Splines	*Blue*
Sketches	
Sketch Curves	*Cyan*
Reference Curves	*Gray*
Datum Features	*Aquamarine*
Points and Coordinate Systems	*White*
System Display Color	*Red*

Seed Part

Seed parts are an effective tool for establishing customer defaults or any settings that are *part*-dependent (saved with the part file). This may include non-geometric data such as:

- Sketch preferences
- Commonly used expressions
- Layer categories
- User-defined views and layouts
- Part attributes

TIP Once a seed part is established, it should be write-protected to avoid accidental modification of the seed part.

How to Use This Manual

It is important that you use the Student Guide in the sequence presented since later lessons assume you have learned concepts and techniques taught in an earlier lesson. If necessary, you can always refer to any previous activity where a method or technique was originally taught.

The format of the activities is consistent throughout this manual. Steps are labeled and specify what will be accomplished at any given point in the activity. Below each step are action boxes which emphasize the individual actions that must be taken to accomplish the step. As your knowledge of Unigraphics increases, the action boxes may seem redundant as the step text becomes all that is needed to accomplish a given task.

Step 1 This is an example of a step.

❑ This is an example of an action box.

The general format for lesson content is:

- presentation
- activity } One or more included in most lessons
- project }
- summary

While working through lesson activities, you will experience a higher degree of comprehension if you read the CUE and Status lines.

It is recommended that students who prefer more detail from an Instructor Led Course ask questions, confirm with restatement, and more importantly, attend and pay attention to the instruction as it is given.

Obviously, it is always necessary for students to consider the classroom situation and be considerate of other students who may have greater or lesser needs for instruction. Instructors cannot possibly meet the exact needs of every student.

At the start of each class day you will be expected to log onto your terminal and start Unigraphics, being ready to follow the instructor's curriculum. At the end of the day's class you should always quit Unigraphics and log off the terminal.

Workbook Overview

The workbook contains a project that requires you to apply the knowledge that you learned in the class and in the student activities. The projects do not contain detailed instructions as do the student activities.

It is the intent of this project to allow you to apply the skills taught in this course. However, the time constraint of this course is also a factor, at any point when progress is not being made, enlist the help of your instructor.

Classroom System Information

Your instructor will provide you with the following items for working in the classroom:

Student Login: **Username:** ____________________

Password: ____________________

Work Directory: ____________________

Parts Directory: ____________________

Instructor: ____________________

Date: ____________________

Getting Started

Lesson 1

1

PURPOSE

This lesson introduces several concepts that you will need to understand and apply in order to effectively use the Manufacturing application. In applying these concepts, prior to creating operations, you will save considerable time in creation of your program.

OBJECTIVES

Upon completion of this lesson, you will be able to:

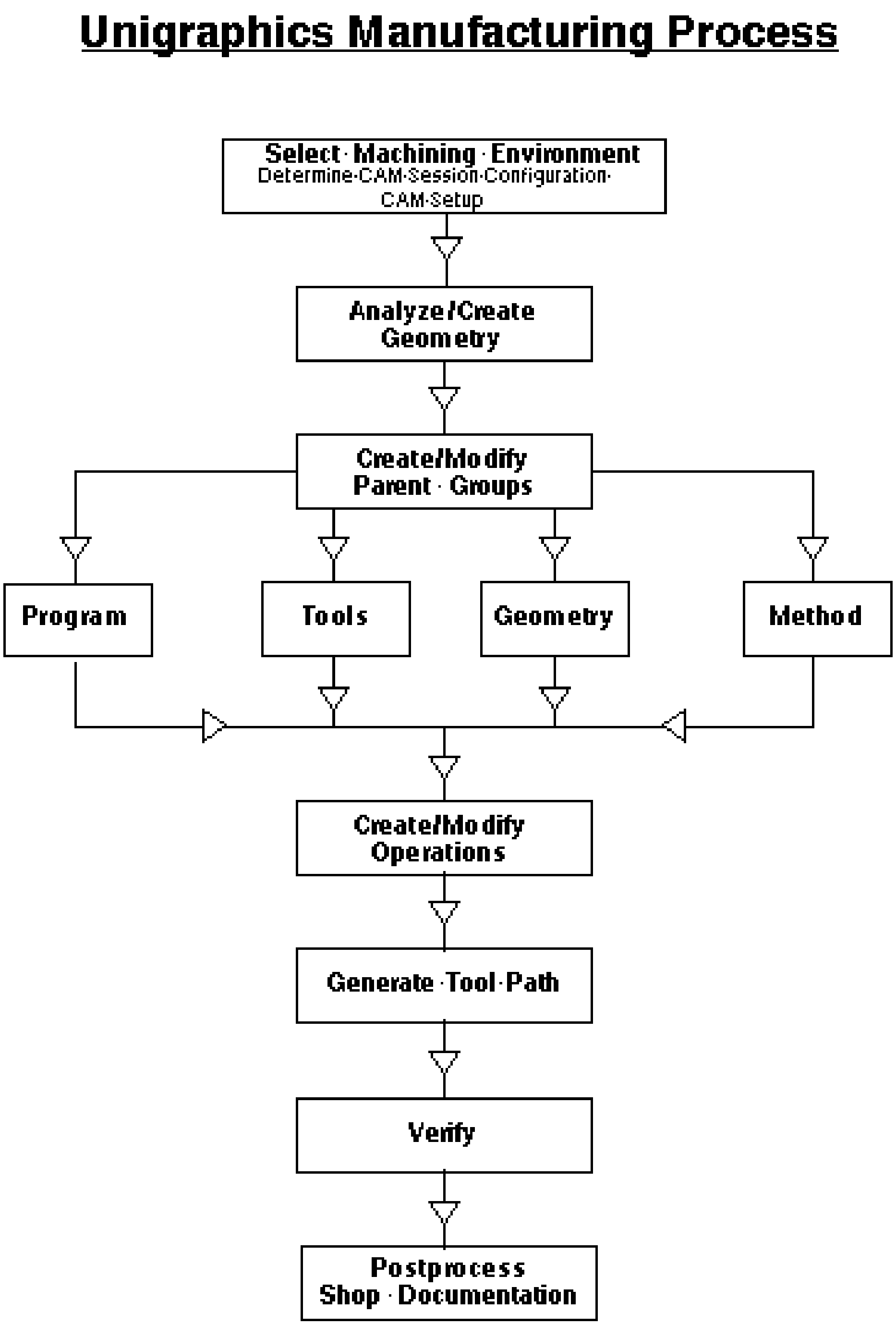
- Understand the concept and use of Parent Group objects
- Understand the Manufacturing Process
- Recognize and identify the differences in the creation of Operations, Programs, Tools, Geometry and Methods
- Understand the meaning of generating a tool path
- Recognize the Operation Navigator

The N/C Programming Sequence

The Manufacturing application programming sequence is to:

- Create the Manufacturing Assembly
- Select the proper Manufacturing Configuration
- Establish the Parent Group objects
- Create the operation(s)
- Verify the tool paths created
- Postprocess the tool paths
- Create Shop Documentation

This sequence is shown in the following Manufacturing Process flow chart.



Creating Parent Group Objects

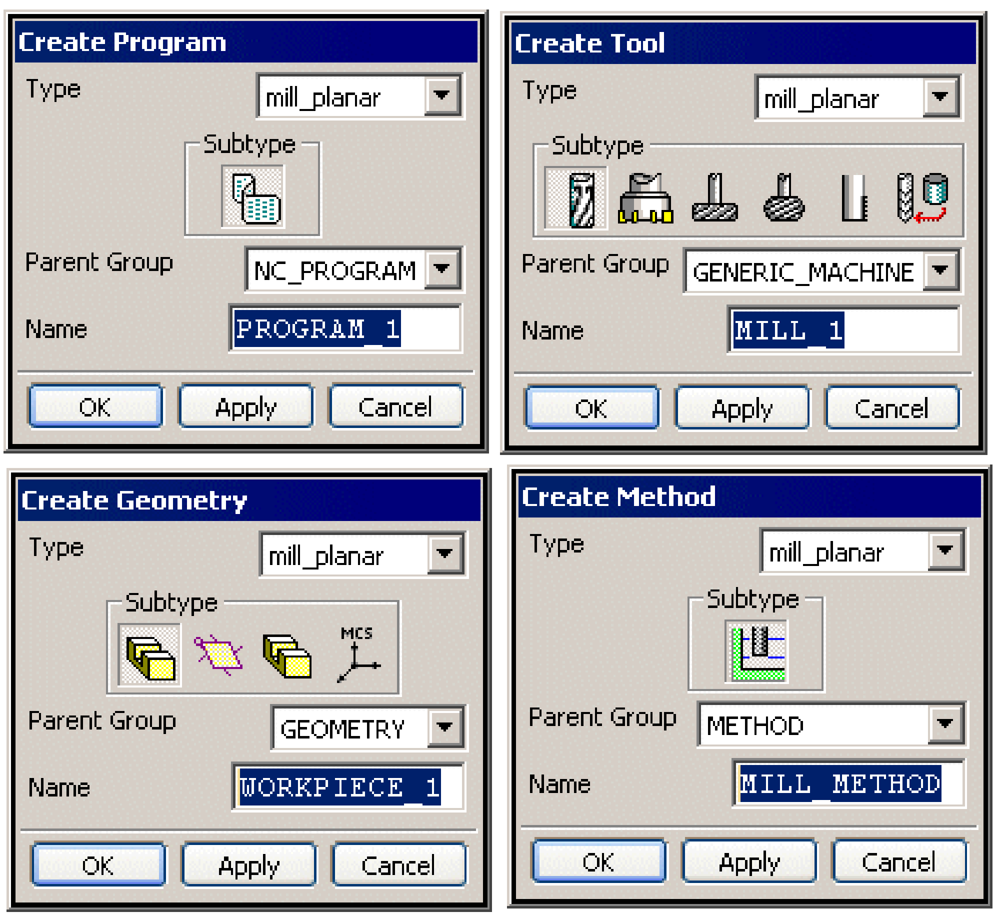
You can create Parent Group objects to store Manufacturing information such as tool data, feed rates, tolerances, etc.

The data specified in the Parent Group object is **inherited** by operations that are listed under that particular Parent Group object. This means that you do not need to specify the tool, and other data within an operation if it was specified in the Parent Group object above that particular operation.

You can create four types of Parent Group objects:

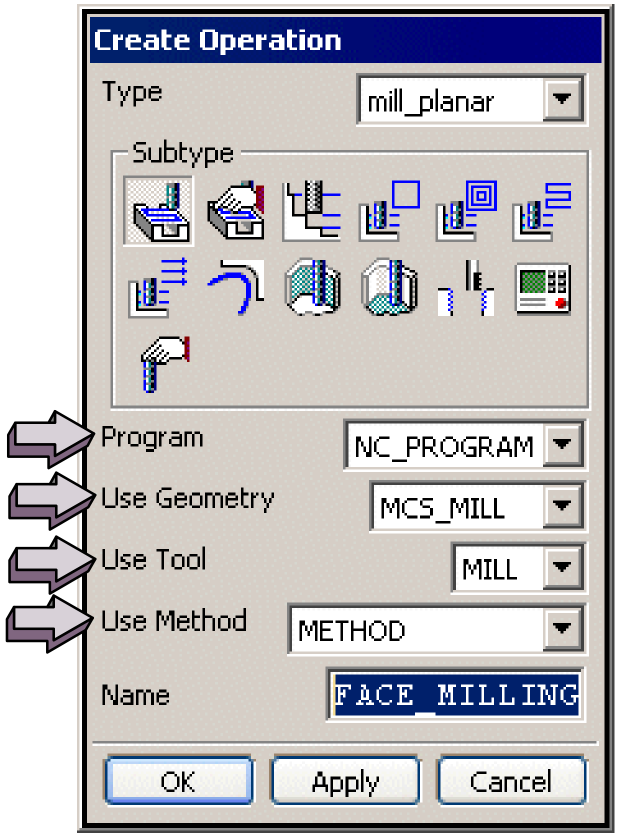
Parent Group	Data Contained
Geometry	Geometry data such as, part, blank, ID, OD, MCS, clearance plane, etc.
Method	Machining data such as feeds, speeds and tolerances
Program	The output order of operations
Tool	Tooling parameters

These dialogs are shown in the following diagram.



Creating an Operation

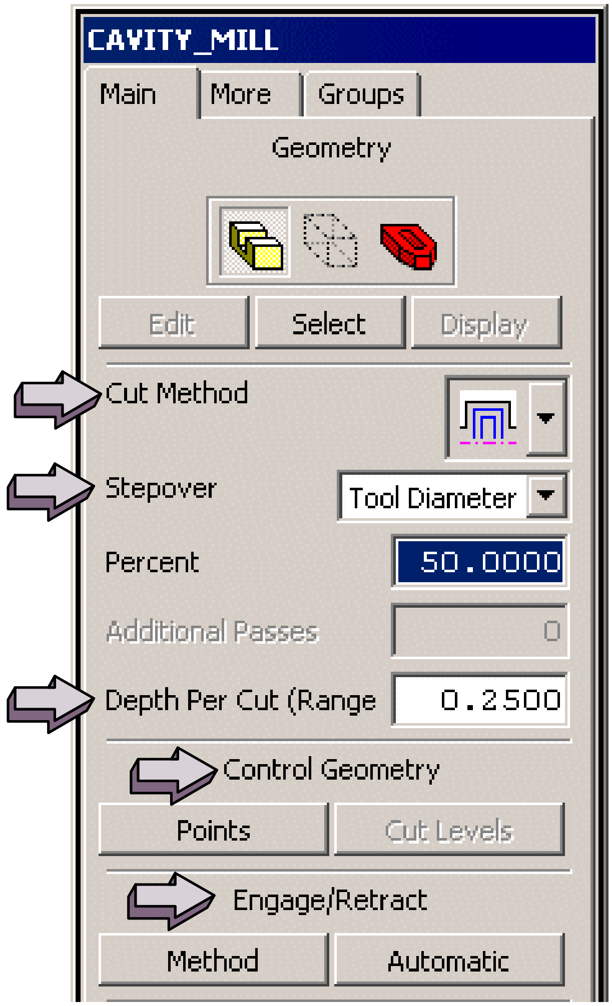
Before you create an operation, you may assign the Program, Geometry, Tool, and Method Parent Group to the operation.



Specifying Additional Operation Settings

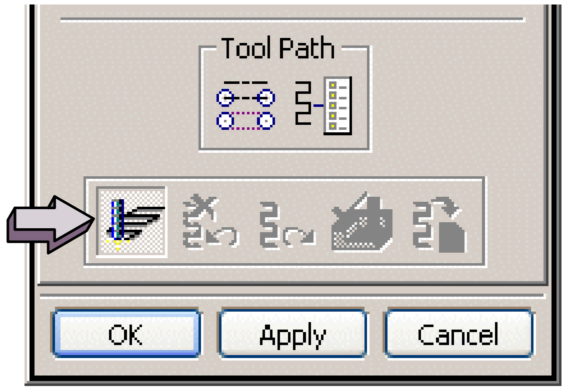
1

Once you create an operation, you have other options such as Cut Method, Stepover, Depth Per Cut, Control Geometry, etc. that will affect the tool path you are creating.



Generating the Tool Path

After you specify all the operation settings, you **Generate** the tool path. Generating the tool path causes Unigraphics to take all of the specified settings and options to calculate the tool path.



Verification, Postprocessing, and Creating Shop Documentation

After you are satisfied with the operations and the tool paths that you create, you use other Manufacturing application features to visually inspect the data.

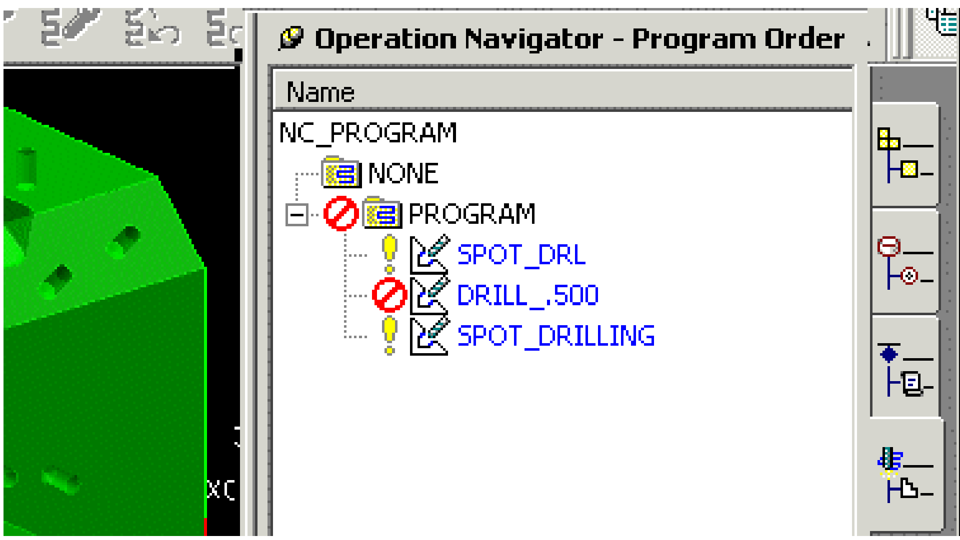
You can postprocess all of the tool paths so that the data is in the standard format used by CNC machine tool controllers.

Finally, you can create the Shop Documentation that describes the data to shop personnel.

1

The Operation Navigator

The Operation Navigator is a graphical user interface that allows you to manage operations and their parameters in the current part file. It allows you to specify groups of parameters that are shared among operations and uses a tree structure to illustrate the relationships between groups and operations. Parameters are passed or inherited from group to group and from group to operation based on the positional relationships in the Operation Navigator.The Operation Navigator is found in the Resource bar in the Manufacturing Application and is a primary tool used in the creation and/or modification of operations.



Manufacturing Application Capabilities.

The Manufacturing Application has extensive capabilities allowing you to generate from the simplest to the most complex of tool paths.

This class will introduce you to:

- The processes you will use to create milling/drilling operations
- How to create Parent Groups and use of the Operation Navigator
- Drilling, Planar Milling, Cavity Milling, and Fixed Contour operation types
- Visualization, Postprocessing, and Shop Documentation procedures

SUMMARY

This lesson is an introduction to the Operation Navigator and basic CAM concepts with respect to creating Parent Group objects and operations.

In this lesson, you learned that:

- Part files contain data (in Parent Groups) that are used by operations
- Operations use data contained in a Parent Group based on positional relationships
- You can view operation and Parent Group information through the Operation Navigator.

(This Page Intentionally Left Blank)

Cavity Mill and Parent Groups

Lesson 2

PURPOSE

This lesson introduces the fundamentals of Cavity Milling operations. Cavity Milling is used in roughing operations involving planar and contoured geometry. You will also create some of the tools required and incorporate there use through the Tool Parent Group object. You will also use the Operation Navigator to observe the method in which operations inherit information.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Understand the use of Cavity Milling
- Apply the use of Cavity Milling operations to rough geometry from an unfinished to semi-finished condition
- Create and use Tool Parent Group objects
- Create and modify various Parent Group objects in Cavity Milling operations

This lesson contains the following activities:

Cavity Milling Overview

Cavity Milling is designed for volume removal of material. It works best when used to remove excess amounts of material from blank stock to produce a near-net shape part. Cavity Milling is not designed for finishing operations. Operation types, such as Z-Level Profile, are better suited for finishing type operations.

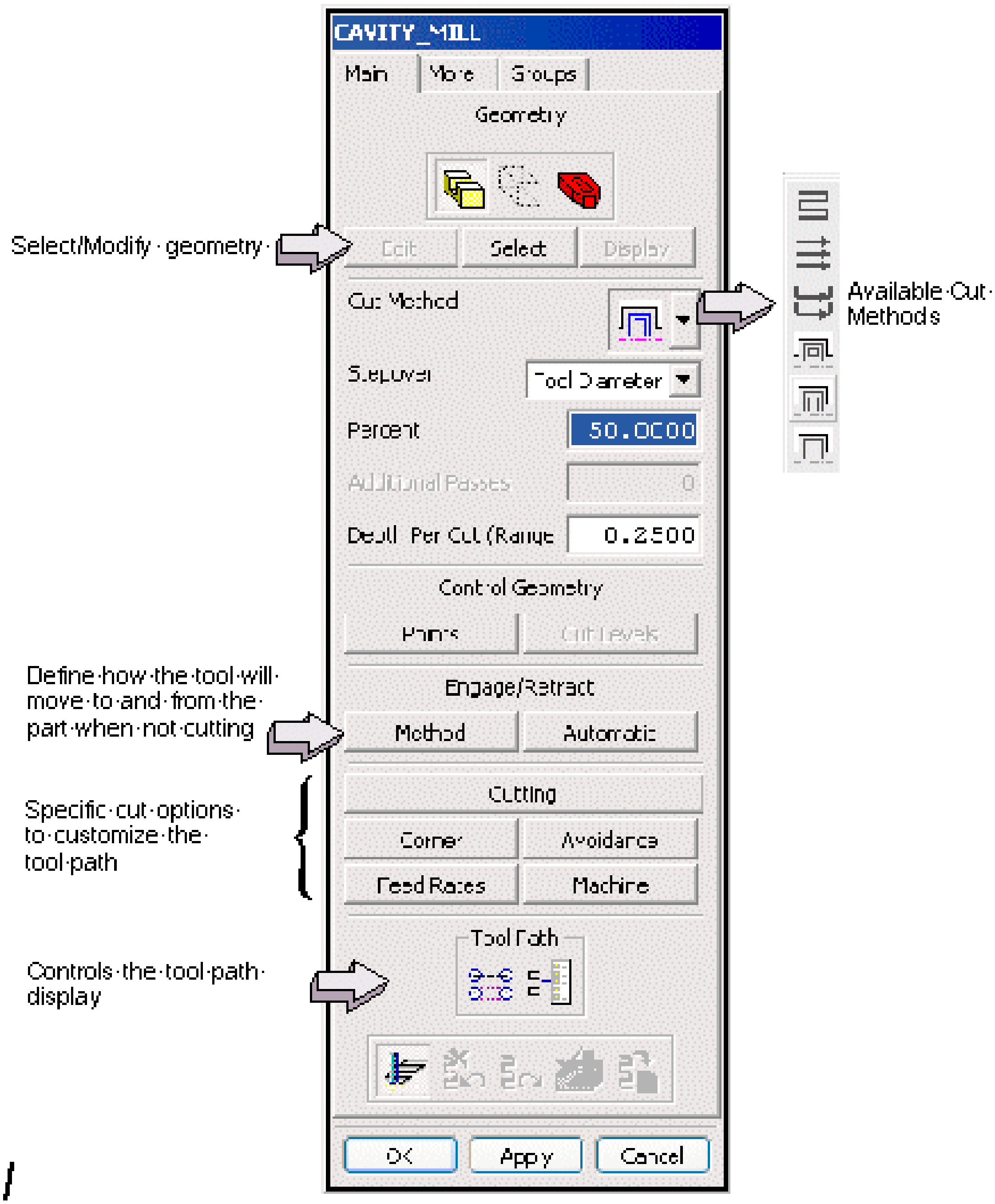
Cavity Milling works with planar and/or contoured geometry and uses a fixed tool axis for efficient roughing of geometry. Planar geometry is defined as either planar or perpendicular to the tool axis. Contoured geometry contains faces (surfaces) that are neither planar nor perpendicular.

Stock removal is done in levels or slices. At each new level the tool path follows the part contours at that level.

Cavity Milling can be used on sheet bodies, wireframe, and solid bodies. Solid bodies are easiest to use. Wireframe and sheet bodies usually require some manual intervention on the user's part to determine which side of the geometry should be machined.

Cavity Milling can be used in a variety of applications such as mold and die work, roughing of complex parts and roughing of material from castings or forgings.

Cavity Milling dialog showing the Main Property page

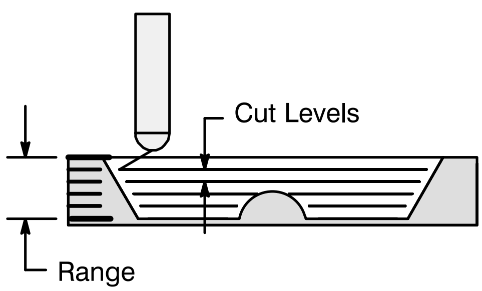


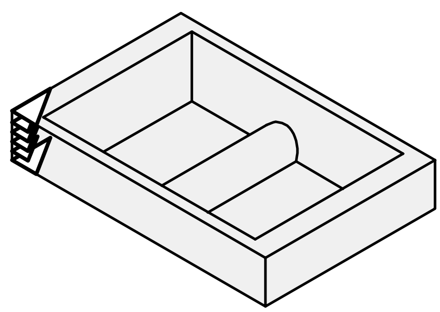
2

How Cavity Milling Creates Tool Paths

In order to efficiently use Cavity Mill, it is important to understand how Cavity Milling creates tool paths. The process is as follows:

- Select the Blank (or stock) material
- Select the Part geometry
- Cavity Milling then automatically sets the top and bottom of the Blank geometry as the highest and lowest level of cutting
- Based on the defined Cut Levels, a plane(s) is created that is perpendicular to the tool axis
- Intersection curves (referred to as traces) between the Cut Level planes and the geometry are created
- At each Cut Level, Cavity Milling creates a cut pattern to remove material from that level
- Finally, the different Cut Levels are combined with Engage and Retract moves





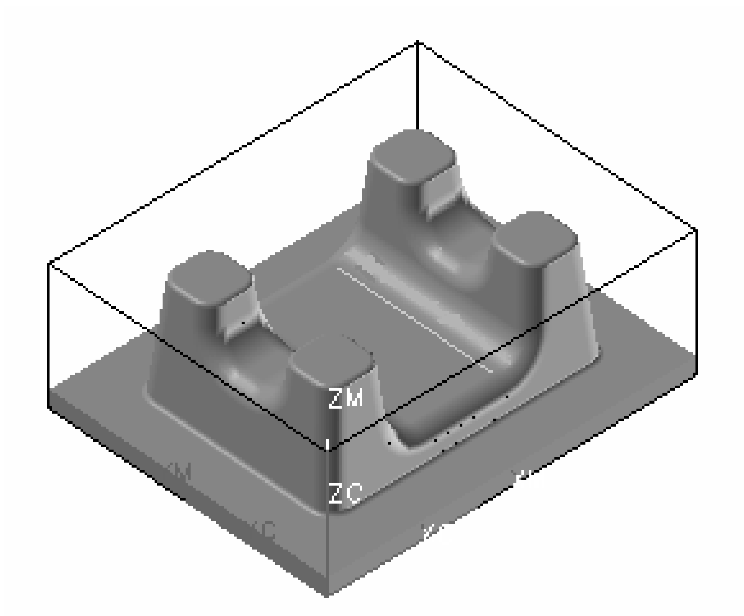
Plane symbols are used to display the Cut Ranges (the large symbols) and Levels (the small symbols).

Activity 2—1: Creating a Cavity Milling Operation

In this activity, you will create a Cavity Milling operation. You will change the Type, choose the cavity mill Subtype, define the Parent Groups and generate the tool path.

Step 1 Open and rename the Part File.

- ❑ Choose **File→Open** then choose **mmp_base_mfg_1.prt**.



This part is a representation of the core half of a forming die. Many detail components, such as clamping devices, are not added. The purpose of this activity is to demonstrate Cavity Milling.

- ❑ Rename the part using **File→Save As** ***_**base_mfg_1.prt** where *** represents your initials.

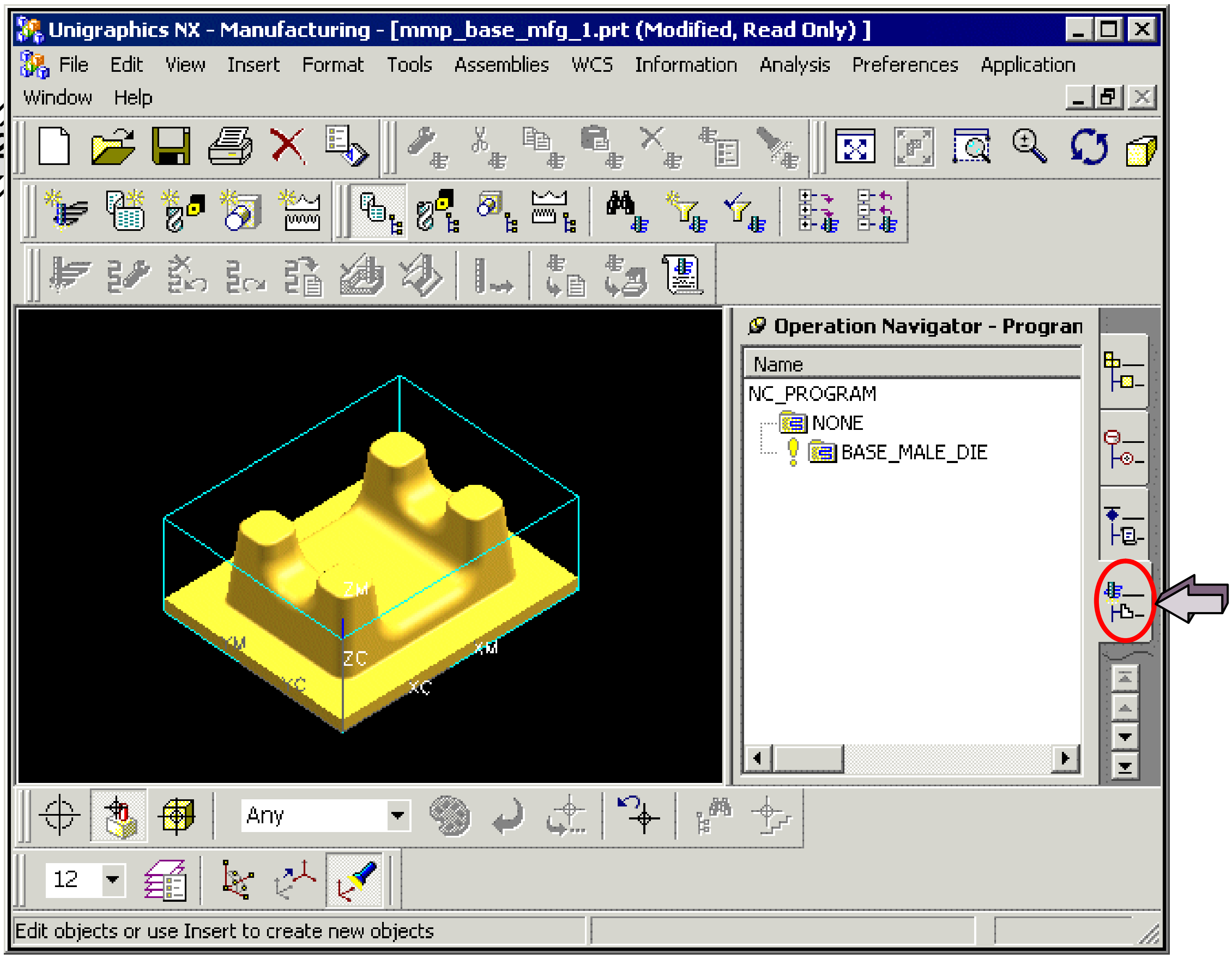
NOTE: Save the file in your home directory.

This part is an assembly. The part you just saved is the top-level component. This assembly contains two sub-components. Each sub-component is a separate part file. One part file is the base (the actual die), and the other is the base_stock (the blank material the die will be made from).

Step 2 Enter the Manufacturing application and display the Operation Navigator.

❑ Choose **Application→Manufacturing.**

❑ Choose the Operation Navigator tab from the resource bar.

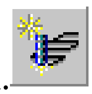


The Operation Navigator is displayed.

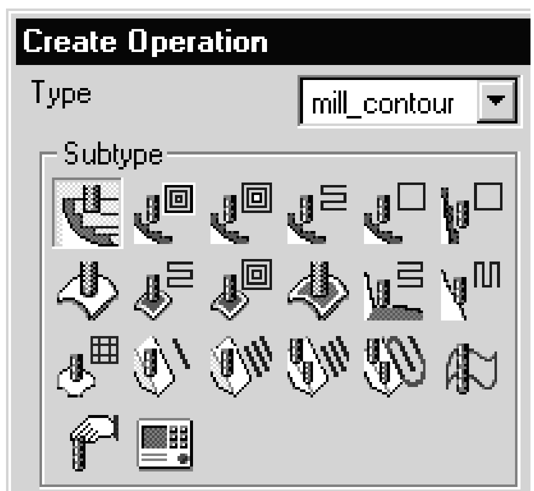
Step 3 Create a Cavity Milling operation.

Much of the background work necessary to successfully create a Cavity Milling tool path is already performed on this part. The Program Parent Group object in which the operation will reside is created. Likewise, the tool that the operation will use is also created. The Blank and Part geometry is specified as well.

To successfully create a Cavity Milling tool path, the operation needs specific parameters set such as the geometry to machine, the tool being used, the amount of stock to leave, and the amount of material to remove per pass. The next several steps will guide you through establishing those parameters.

❑ Choose the **Create Operation** icon. 

The Create Operation dialog is displayed.

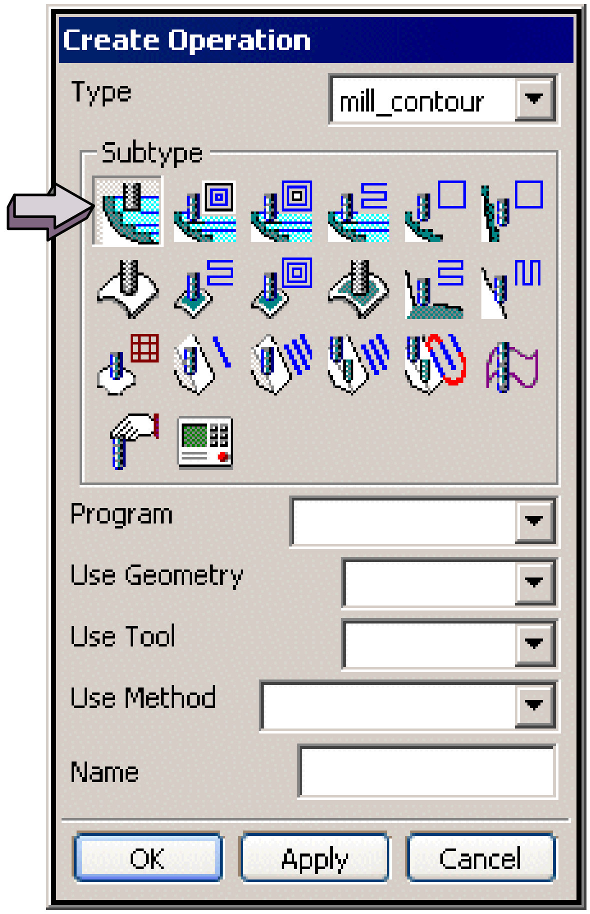


The Type label, at the top of the dialog, is very important. As the name suggests, Type controls what subtypes of operations that are available for creation. Cavity Milling is located under the mill_contour type, because it has the ability to machine geometry that is either contoured or planar.

❑ If necessary, set the Type to **mill_contour**.

Each of the icons in the Subtype area of the dialog represents a different operation type.

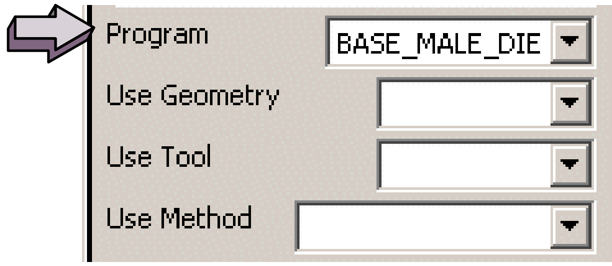
❑ Choose the **CAVITY_MILL** icon from the Subtype area of the Create Operation dialog.



You will now specify additional information necessary to create a successful tool path. Cavity Milling operations will inherit information from each of the Parent Group objects that are selected.

Begin by choosing the Program in which the operation will reside.

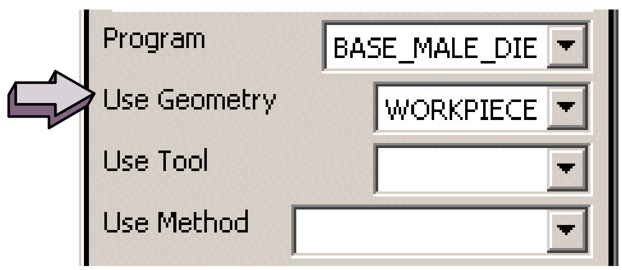
❑ At the Program label, choose **BASE_MALE_DIE**.



Cavity Milling will not work without knowing the blank (stock) or part geometry that it is to machine.

- **Blank –** the material that is the starting point of machining operations
- **Part –** the designed part that is the outcome of the machining operations

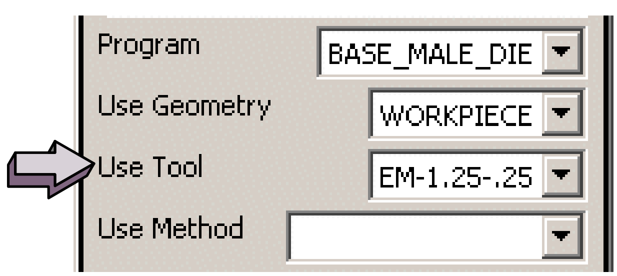
❑ Change the Use Geometry to **WORKPIECE.**



The WORKPIECE Geometry Parent Group contains information about the PART and BLANK geometry. Normally, you would specify this yourself, but for this first activity, this is done for you.

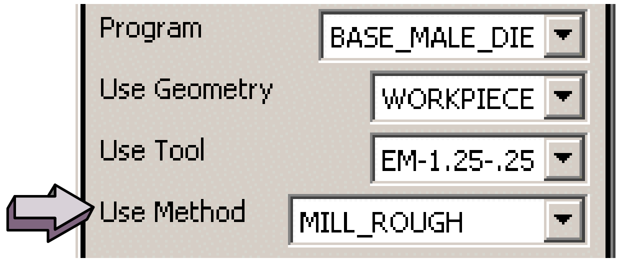
Next you will define the cutting tool.

❑ Change the Use Tool to **EM-1.25-.25.**



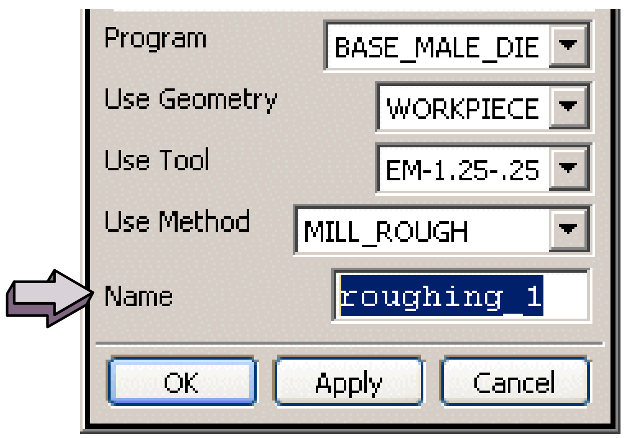
You will now specify the amount of stock that will be left on the part when this roughing operation is finished. This information is stored in the Method Parent Group.

❑ Change the Use Method to **MILL_ROUGH.**



It is a good idea to give operations a meaningful name, so that they can be easily identified to the cutting type of operation that is being performed.

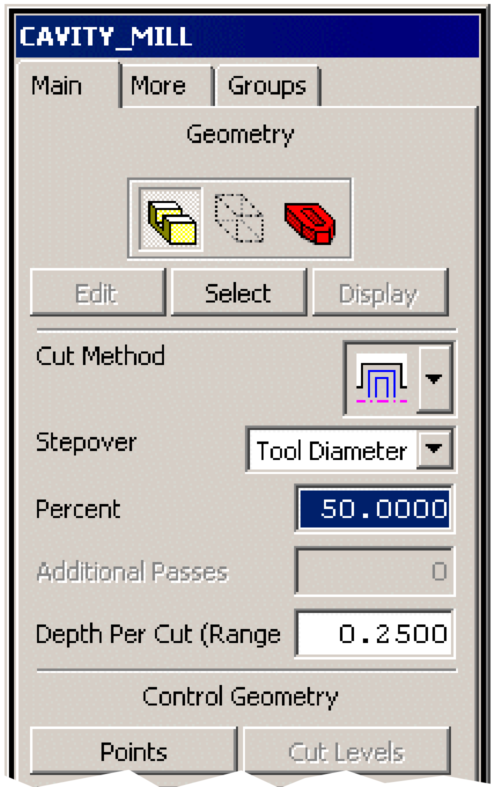
❑ At the bottom of the Create Operation dialog, in the Name field, enter **roughing_1**.

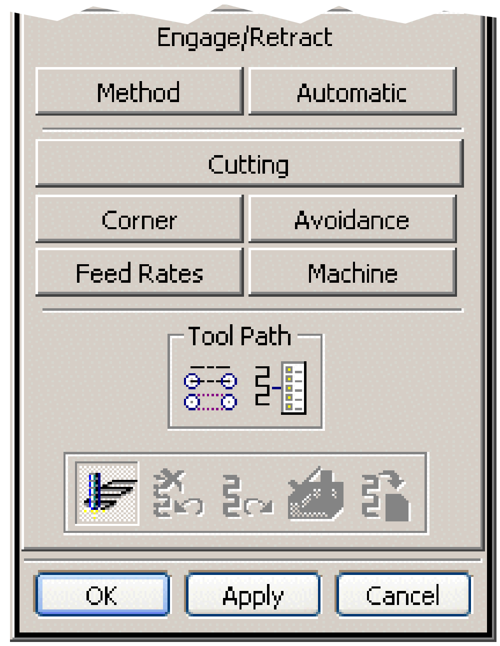


It does not matter whether you type the name in lowercase or uppercase. All characters are converted to uppercase. Spaces and special characters are not allowed and the name must begin with a alphabetic character.

❑ Choose **OK.**

The Cavity_Mill dialog is displayed.



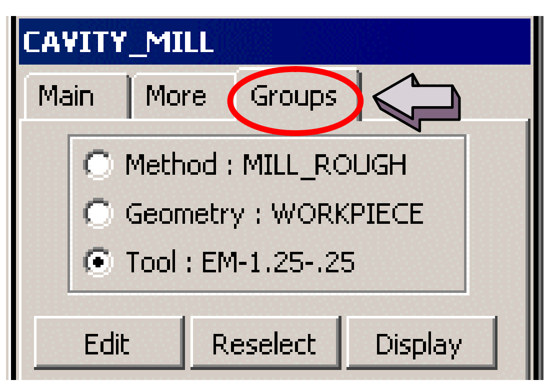


Notice the three tabs, referred to as Property pages designated as **Main**, **More** and **Groups**. By selecting any of these individual property pages, various options and parameters used in tool path generation, are available for selection.

Step 4 Examine the pre-selected geometry in the Geometry Parent Group (WORKPIECE).

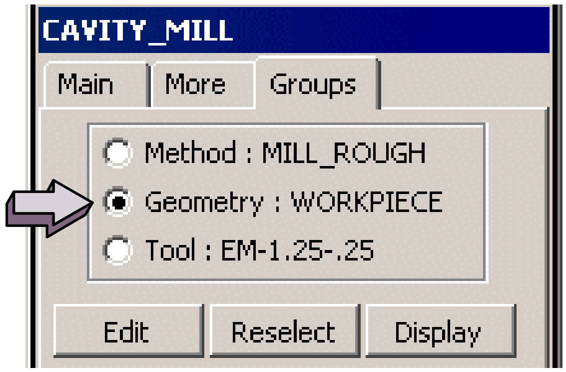
❑ Choose the Groups property page from the Cavity_Mill dialog.

The Parent Group, Method, Geometry and Tool are available for selection, modification or display.



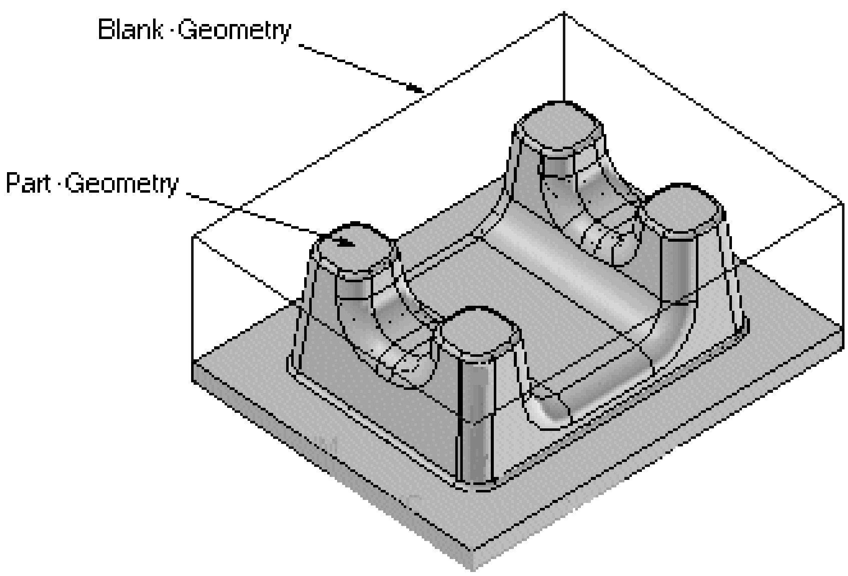
2

❑ Choose the **Geometry** button.



❑ Choose **Display**.

The Part and Blank geometry that are members of the WORKPIECE Geometry Parent Group are displayed.

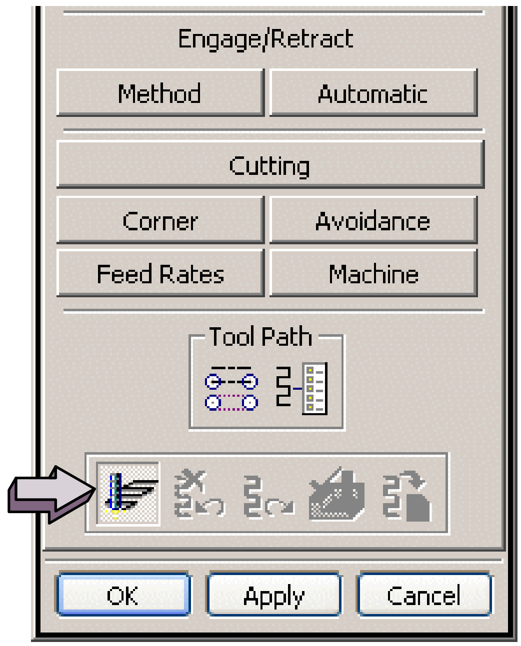


Step 5 Generate the operation.

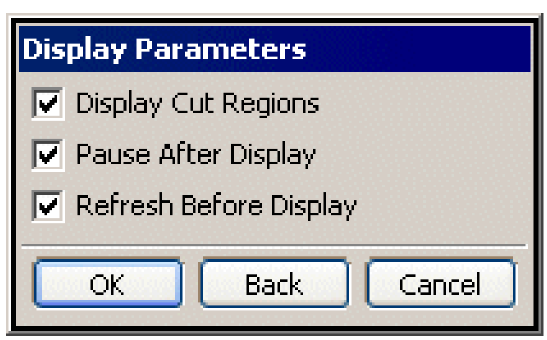
It is possible now to create, or generate the tool path since all the information necessary for creation is defined.

❑ Choose the **Main** property page (tab).

❑ Choose the **Generate** icon at the bottom of the Cavity Mill dialog.

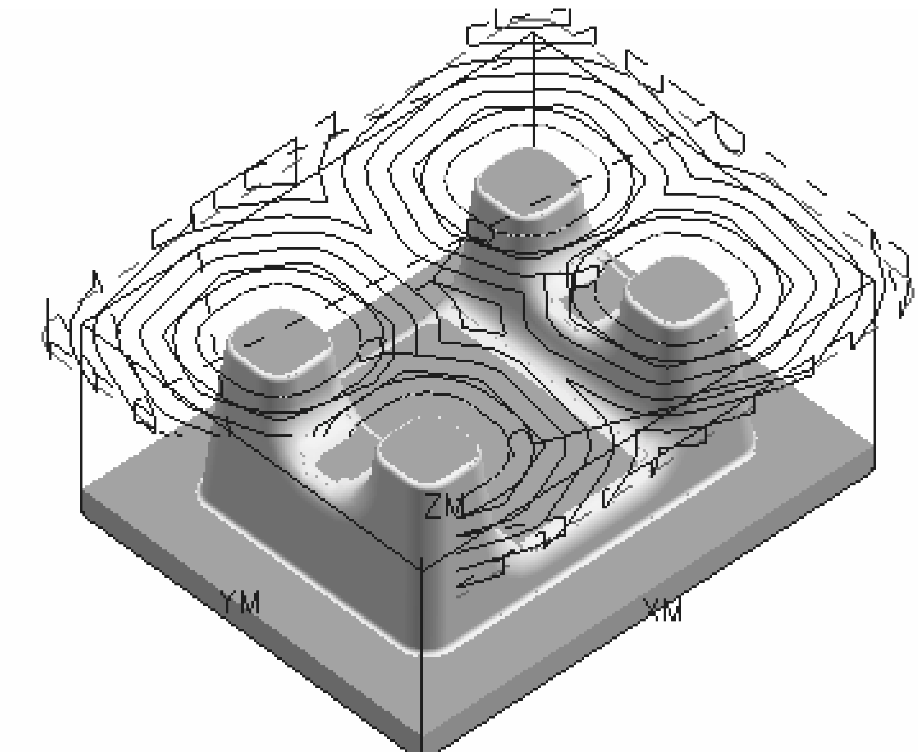


The Display Parameters dialog is displayed. This dialog is designed to allow you to step through each cut level. By default settings, a pause will occur after displaying each cut level to allow examination of the cut path.



The dashed magenta lines that you see, indicate the outside area that the center of the tool will cut on. The solid yellow lines indicate the inside area that the center of the tool will cut on.

❑ Choose **OK** to see the first cut level.



The cyan colored lines indicate the centerline of the cutting tool. There are several more cut levels to display.

❑ Continue choosing **OK** until all cut levels are generated, and a warning message is displayed.

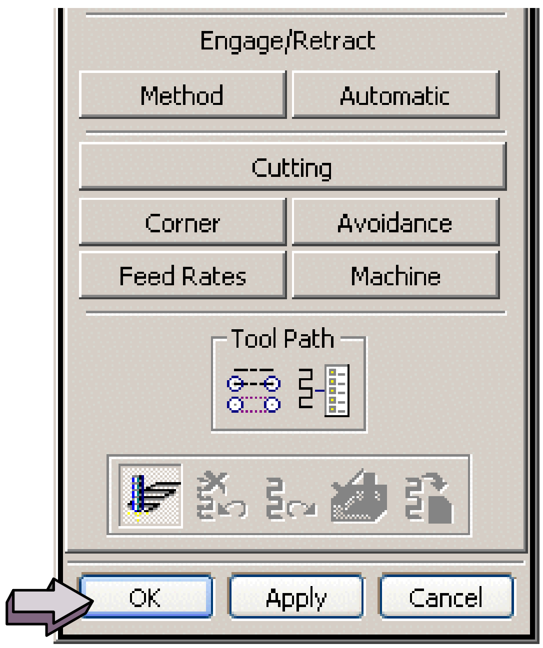
This warning message indicates that this operation is currently designed to cut lower than is physically possible, given the intersection of the PART geometry with the BLANK geometry and the cutter configuration. The tool path is still valid. Later, you will learn how to eliminate this warning message.

❑ Choose **OK** to the Warning message.

You have successfully created a Cavity Milling operation and used it to generate a tool path. It is very important to save your work as you create and generate operations.

To accept an operation along with the settings you have made, choose OK at the bottom of the operation dialog. This will accept the operation in the current form, and allow it to be displayed in the Operation Navigator. Do not choose Cancel unless you wish to discard the operation.

❑ Choose **OK** at the bottom of the Cavity_Mill dialog.

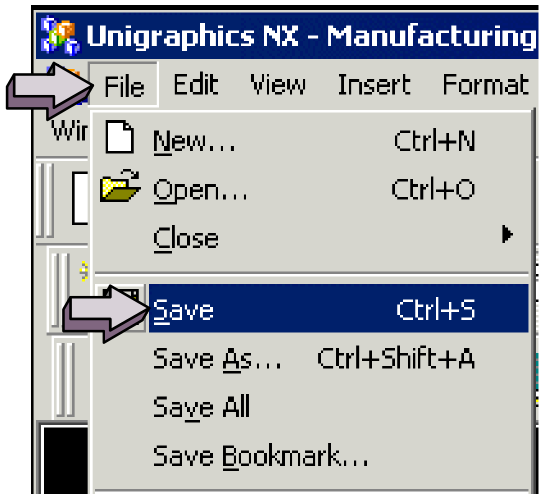


You must now save the part and the operation that you have just created. If you were to close this part file without first saving, you would loose all of the operation data and anything subsequent to the last part filing.

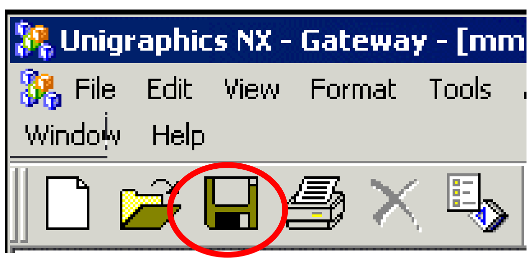
2

Step 6 Save the part file.

❑ From the menu bar choose **File** and then **Save.**



It is advisable to file your work often, to preserve the operations and data that you are creating. You can also save the file by selecting the File icon from the standard toolbar.



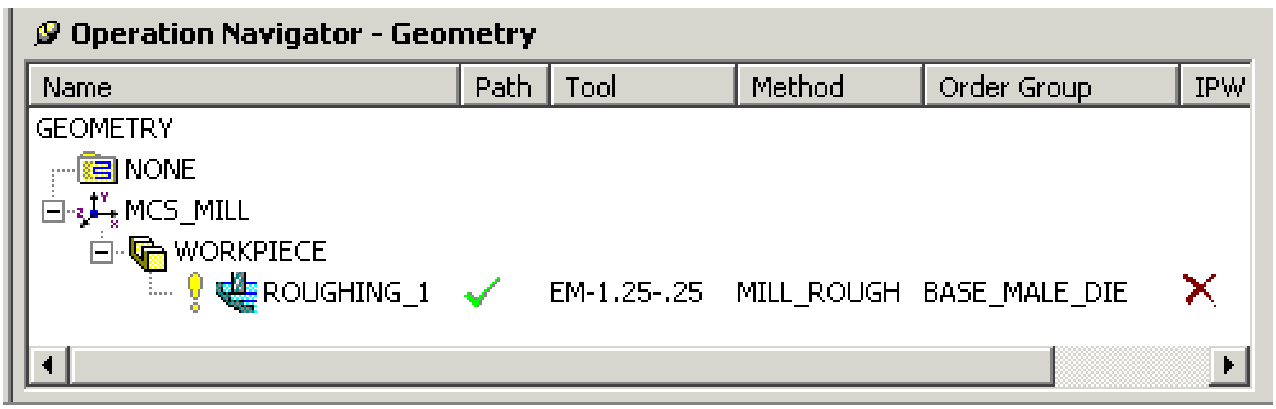
This completes this activity.

Modifying Parent Groups

The Operation Navigator — Geometry View

As mentioned earlier, Cavity Milling operations (for that matter, all operations) obtain or inherit some types of information from objects that exist outside the operation that is being created.

The ROUGHING_1 operation that you created is shown in the Geometry View, of the Operation Navigator, below.



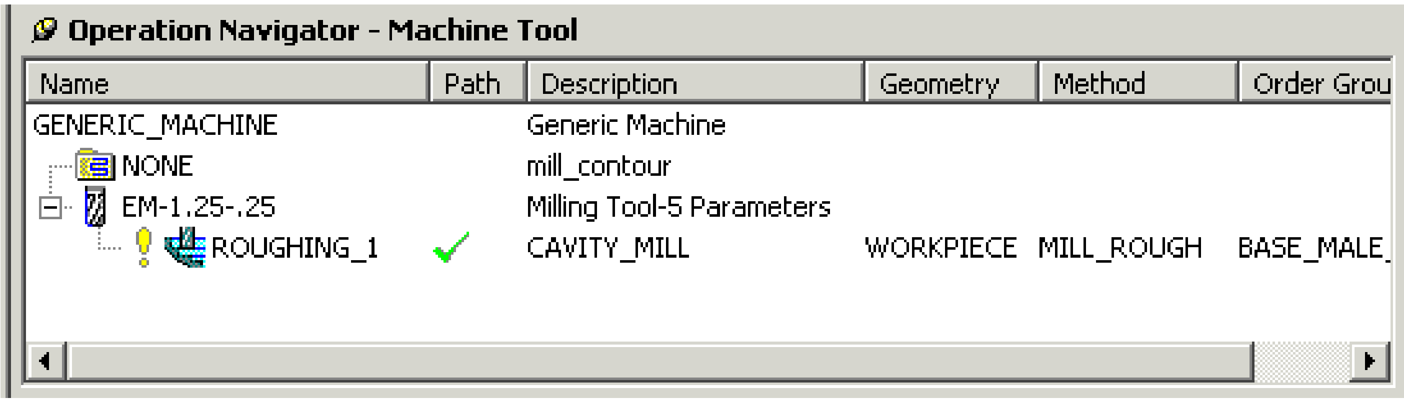
The operation, ROUGHING_1, obtains the PART and BLANK geometry from the Geometry Parent Group WORKPIECE.

Above the WORKPIECE Geometry Parent Group, is the MCS_MILL Parent Group. This Geometry Parent Group contains information about the location and orientation of the Machine Coordinate System and the Clearance Plane.

By following the hierarchy, of the parent groups and operations, you can see that the MCS_MILL parent is passing information to the WORKPIECE parent. The WORKPIECE parent then passes this information, plus any information that it contains to the ROUGHING_1 operation. By this method, the ROUGHING_1 operation inherits the geometry information, as well as other parameters needed to create a tool path.

The Operation Navigator — Machine Tool View

If the Operation Navigator were changed to the Machine Tool View, geometry objects would no longer display. Instead, tools that exist in the part would be display.



Once again, the Operation Navigator displays the ROUGHING_1 operation. However, this time, it displays the operation as it pertains to the cutting tool that is being used.

Inheritance is at work again. All information about the tool, including offset register and tool number, are passed to the operation, provided the user has turned ON **Update Post from Tool** in the Preferences → Manufacturing → Operation property page.

In the next activity, you will create a new tool. Then you will change the existing operation to use that new tool.

Activity 2—2: Modifying the Machine Tool Parent Group

In this activity, you will create a new cutting tool and change the inheritance of the ROUGHING_1 operation to use the new tool that was created.

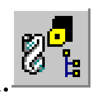
Step 1 Use the part file from the previous activity.

❑ Continue to use *****_base_mfg_1.prt**.

❑ If necessary, choose **Application→Manufacturing**.

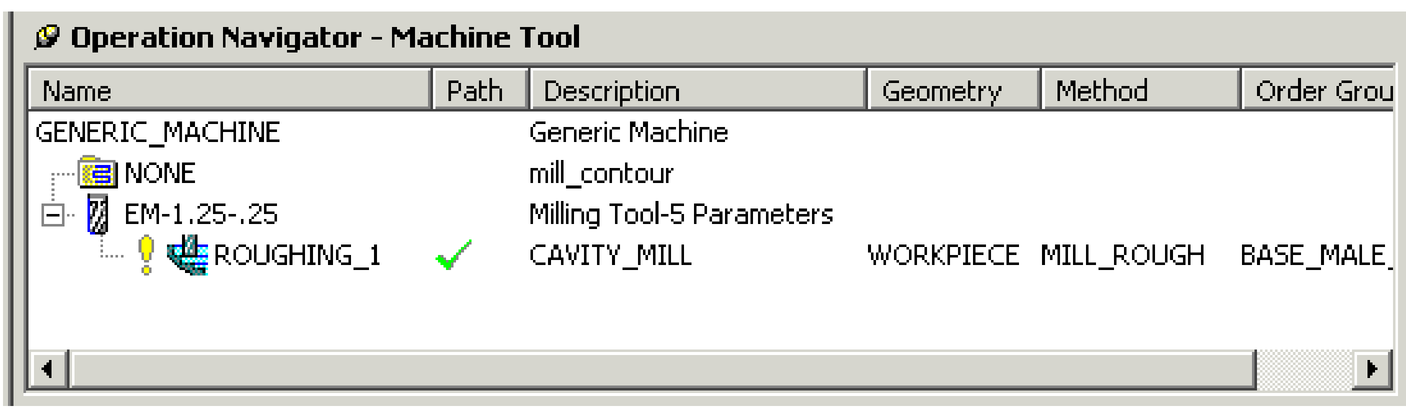
Step 2 Change the View of the Operation Navigator.

Before creating the cutting tool, change the Operation Navigator to the Machine Tool View.

❑ Choose the **Machine Tool View** icon. 

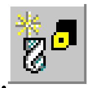
❑ In the Operation Navigator, choose the "+" sign next to the **EM-1.25-.25 tool**.

The ROUGHING_1 operation is shown below the tool.



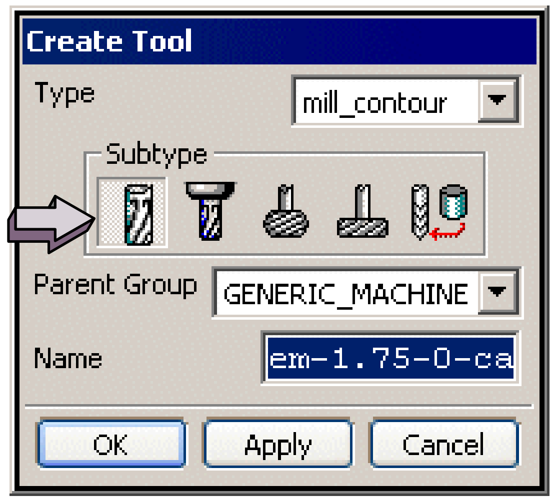
Step 3 Creating a new tool.

Now create a new tool instead of using the 1.25" end mill.

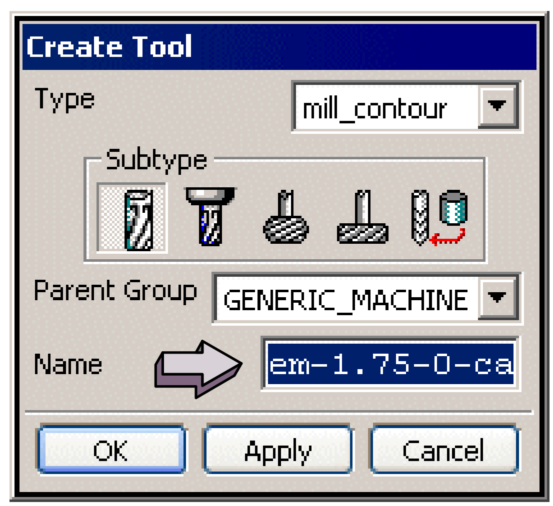
❑ Choose the **Create Tool** icon. 

❑ If necessary, change the Type to **mill_contour**.

❑ Choose the **MILL** icon.



❑ In the Name field, enter **em-1.75-0-carbide**.

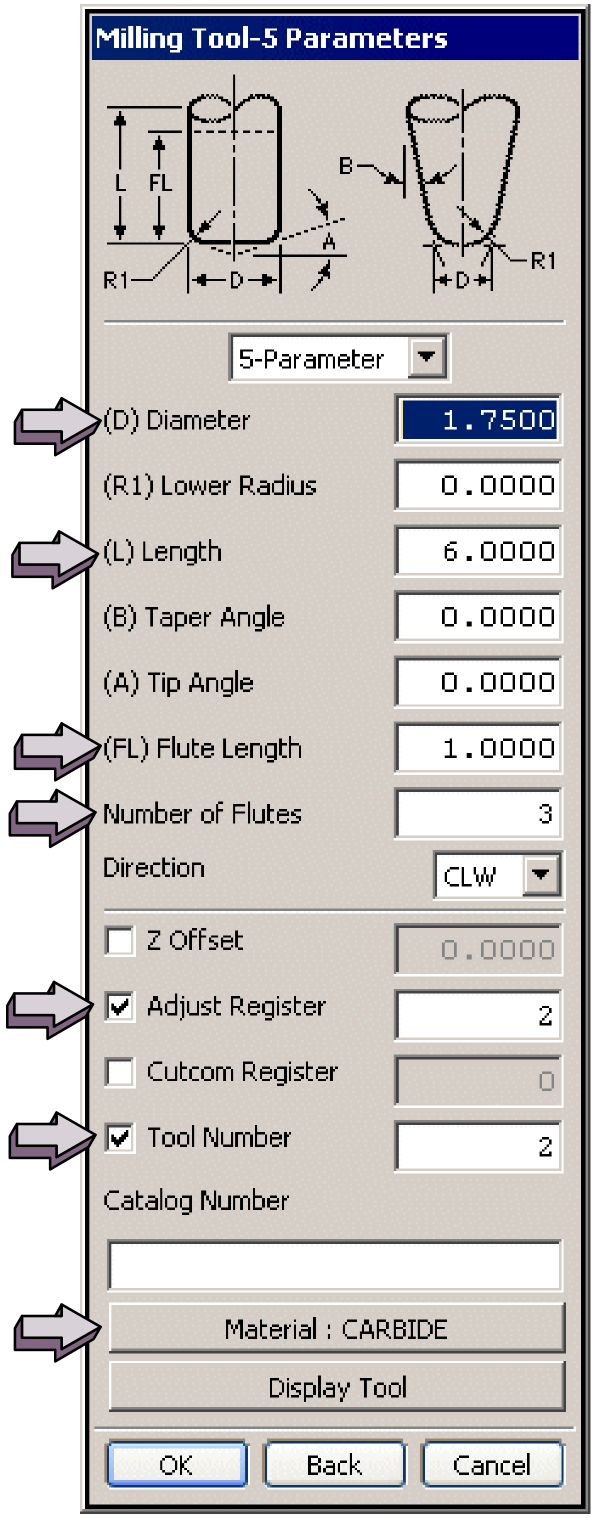


The new tool will be 1.75” in diameter, have zero corner radius, and be carbide-inserted.

❑ Choose **OK** from the Create Tool dialog.

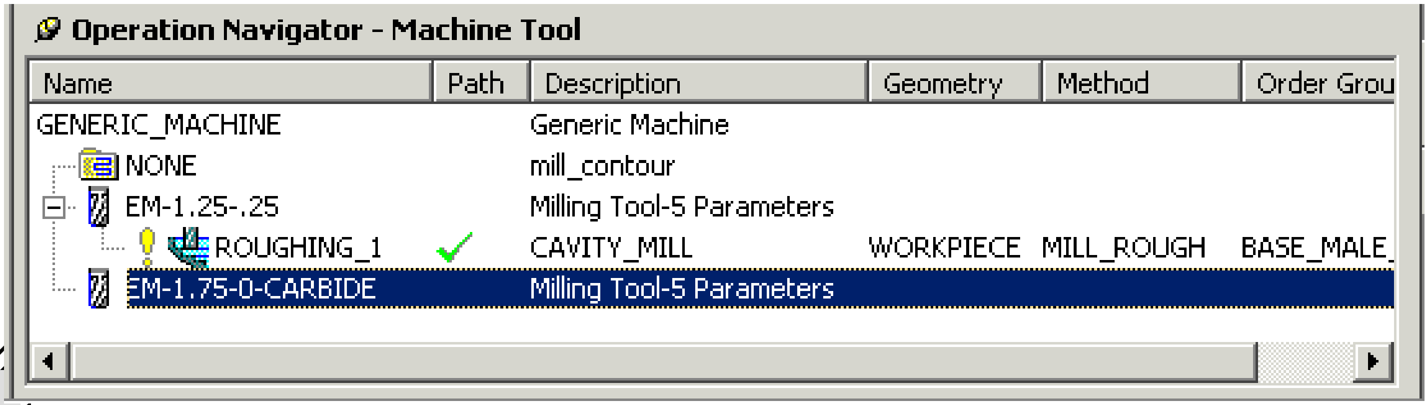
The Milling Tool – 5 Parameter dialog is displayed.

❑ Enter the parameters as shown.



❑ When finished, choose **OK**.

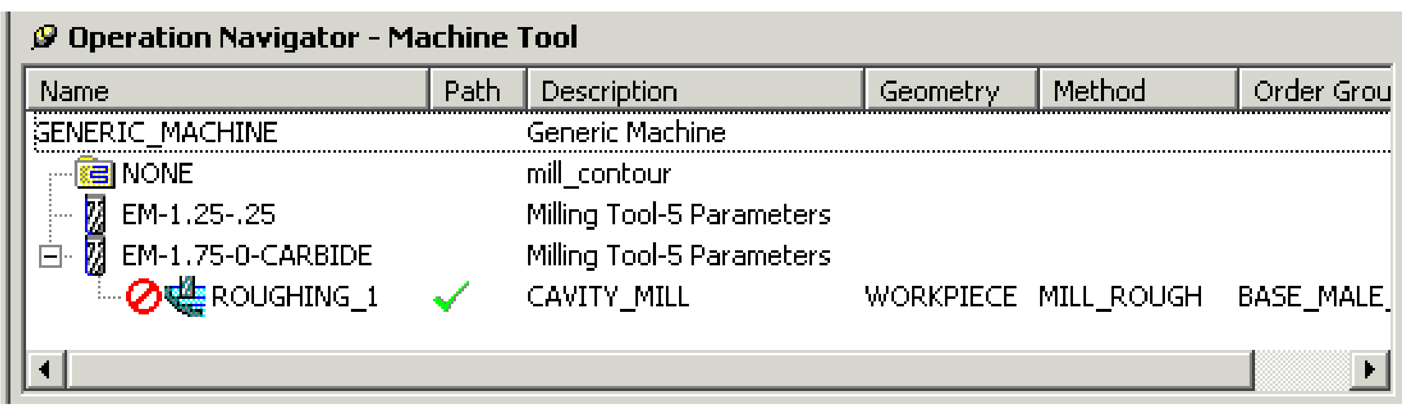
The new tool is displayed in the Operation Navigator.



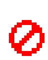
Step 4 Changing the tool for the ROUGHING_1 operation.

Now that the tool is created, the next step is to use it in the ROUGHING_1 operation.

❑ In the Operation Navigator, click and hold MB1 on the **ROUGHING_1** operation. Drag it downward and release it on the **EM-1.75-0-CARBIDE** tool.

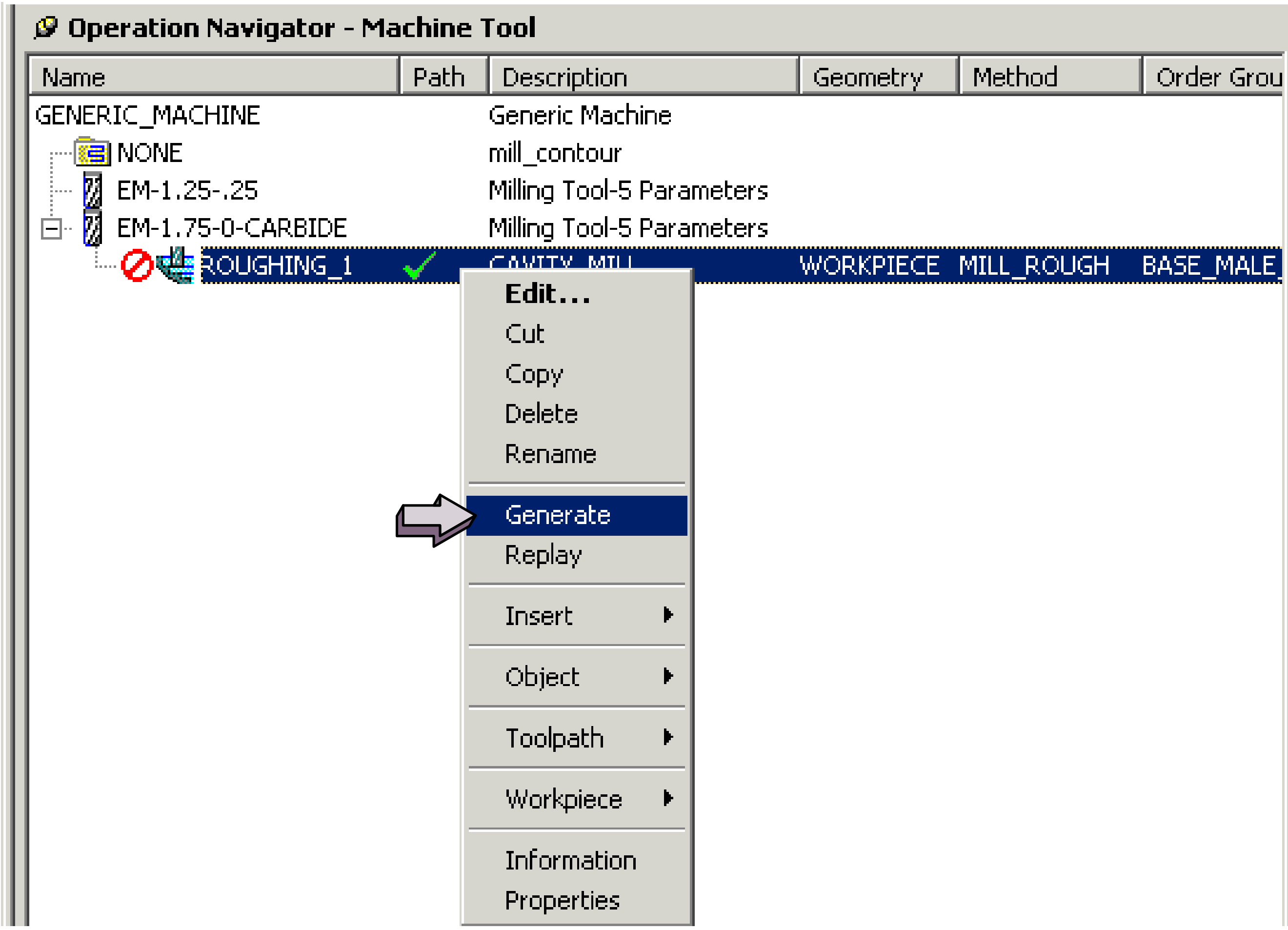


The operation is now assigned to the new tool. The last step is to re-generate the operation.

As soon as the operation was moved to the new tool, the operation needed to be regenerated. The symbol changed from  to . A discussion of the meaning of these symbols will follow later.

Step 5 Generate the operation.

❑ Highlight the **ROUGHING_1** operation, use MB3 and choose **Generate**.



The tool path is displayed in the graphics window.

❑ Choose **OK** to the warning message.

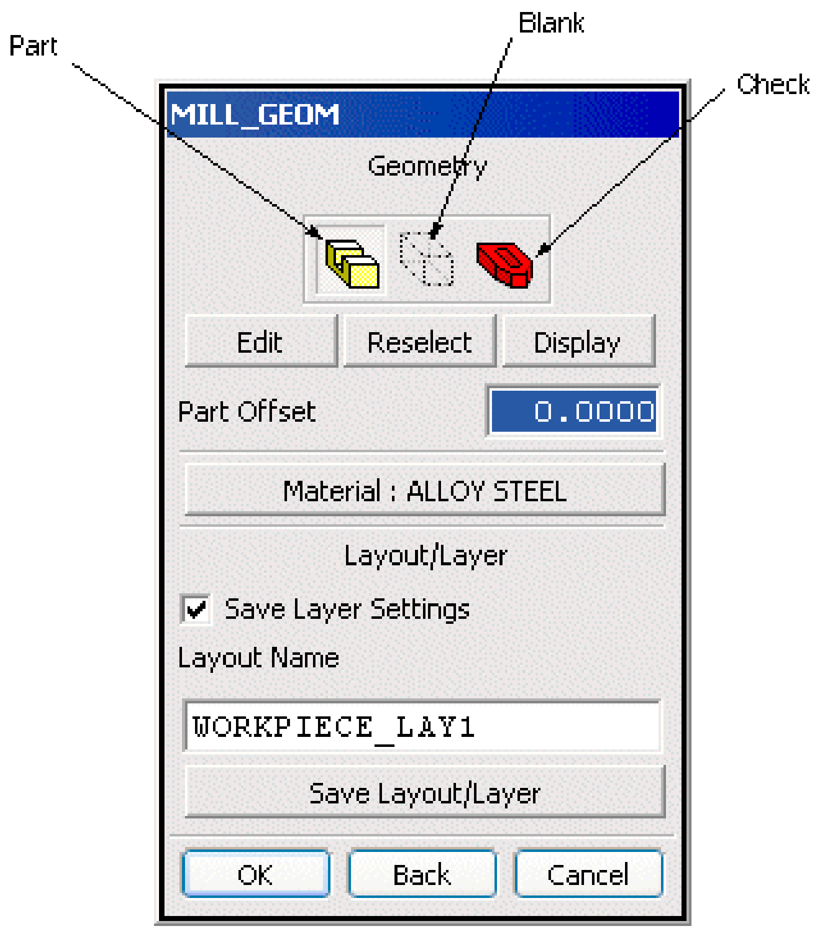
The operation is re-generated with the new tool.

❑ **Save** the part file.

2

Choosing the Blank and Part in the WORKPIECE

Defining the Part, Blank and Check geometry in the Geometry Parent Group will allow any operations that are located under it to inherit the geometry object.



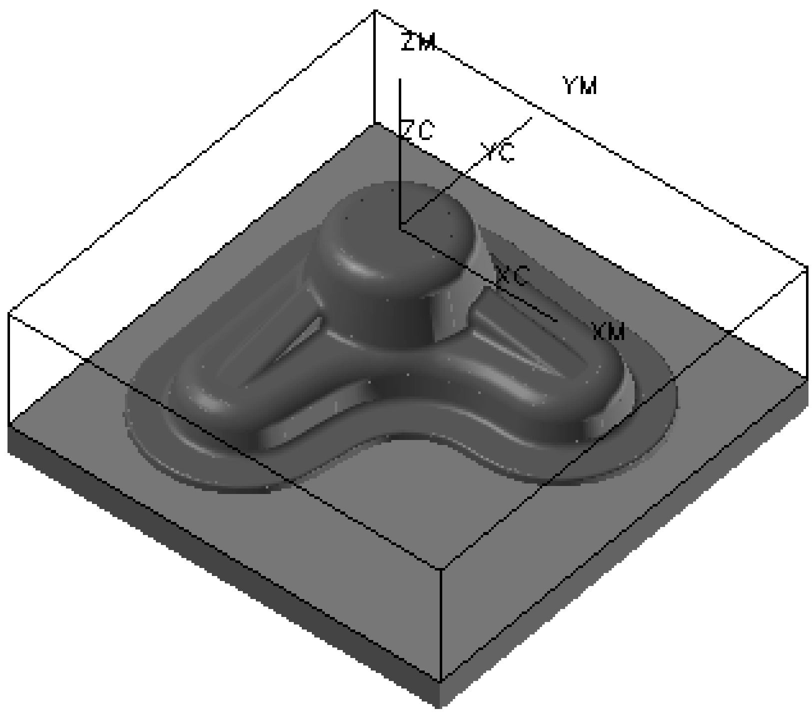
In the next activity, you will define the Blank and Part geometry in the Geometry Parent Group WORKPIECE.

Activity 2—3: Create/Use Parent Groups in an Operation

In this activity, you will create and then use a Machine Tool and Geometry Parent Group when creating the Cavity Milling operation to rough the part.

Step 1 Opening and renaming the part file.

❑ Choose **File → Open** then choose **mmp_male_cover_mfg_1.prt**.



This file is an assembly. It consists of the top level part, which will contain the manufacturing data, and the components:

- mmp_male_cover.prt — the designed part
- mmp_male_cover_mach_plate.prt — a plate to which the cover is bolted
- mmp_bolt_5.prt — two instances of this component hold the cover to the machining plate
- mmp_male_cover_stock.prt — the blank from which the part is made

Step 2 Save the part file and enter the Manufacturing application.

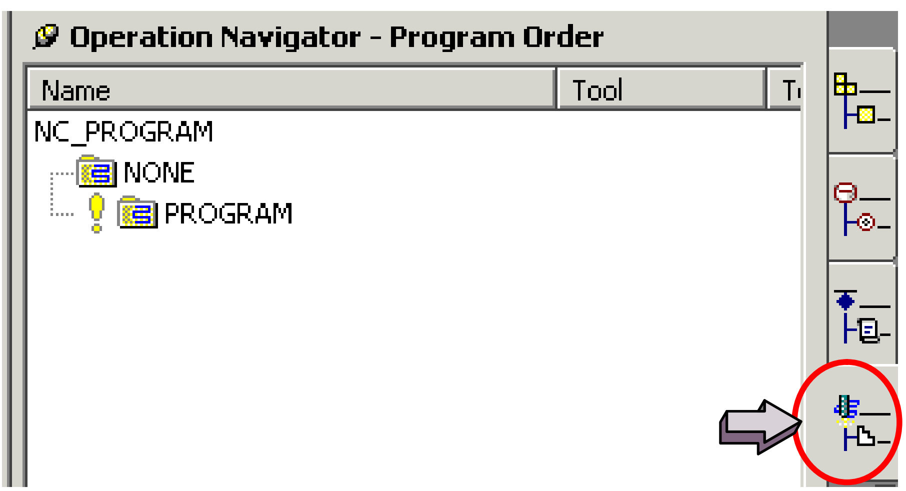
❑ Choose **File** → **Save As** and save the part file as *****_male_cover_mfg_1.prt**.

❑ Choose **Application**→**Manufacturing**.

This time, you will create/modify the:

- Program Parent Group
- Geometry Parent Group
- Machine Tool Parent Group
- Method Parent Group

❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.

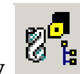


You can change views of the Operation Navigator by holding down MB3 in the Operation Navigator background and selecting the appropriate view.

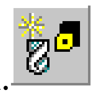
Step 3 Change the name of the existing Program Parent Group object.

❑ Highlight the **PROGRAM** Parent Group, use **MB3**, select **Rename**.

❑ Key in the text **male_cover**, then hit the return key.

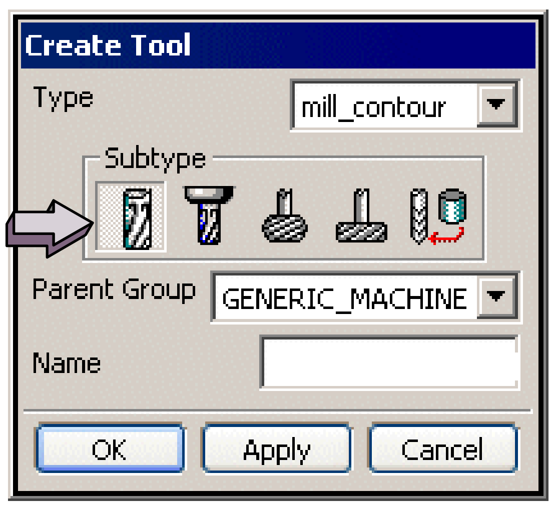
❑ Change to the **Machine Tool View**. 

There are no tools available to use, so it will be necessary to create one.

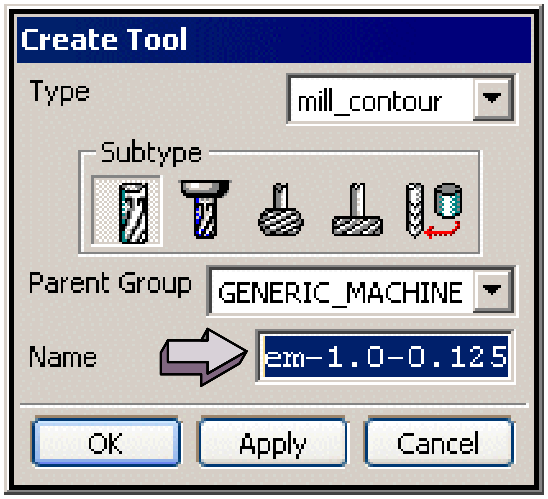
❑ Choose the **Create Tool** icon. 

❑ If necessary, set the Type to **mill_contour**.

❑ Choose the **MILL** icon.

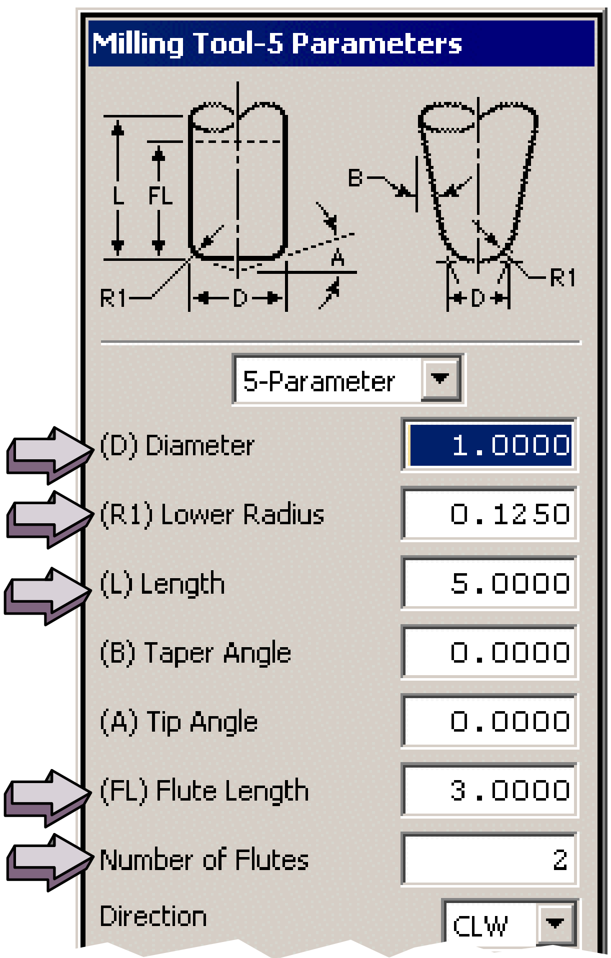


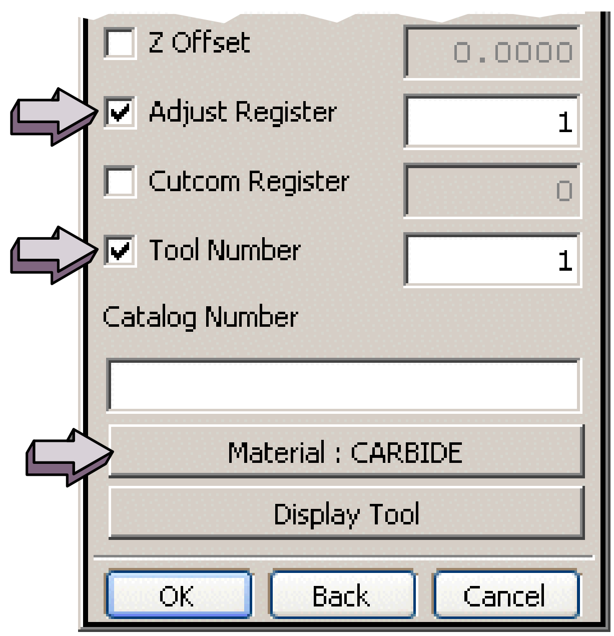
❑ In the Name field, enter **em-1.0-0.125-carbide**.



❑ Choose **OK.**

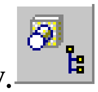
❑ Enter the parameters as shown:



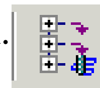


❑ When finished, choose **OK**.

The PROGRAM Parent Group is defined, and the cutting tool is created. You will now define the geometry to machine.



❑ Change the Operation Navigator to the **Geometry View**.

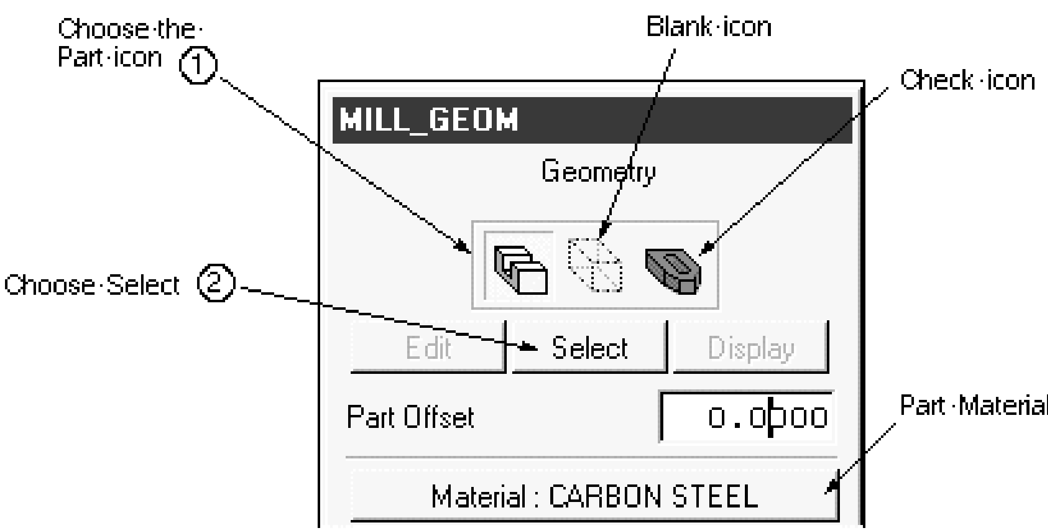
❑ Expand the Parent Group object MCS_MILL by clicking on the "+" adjacent to the Group object or select the Expand icon. 

❑ Double click the **WORKPIECE** Geometry Parent Group.

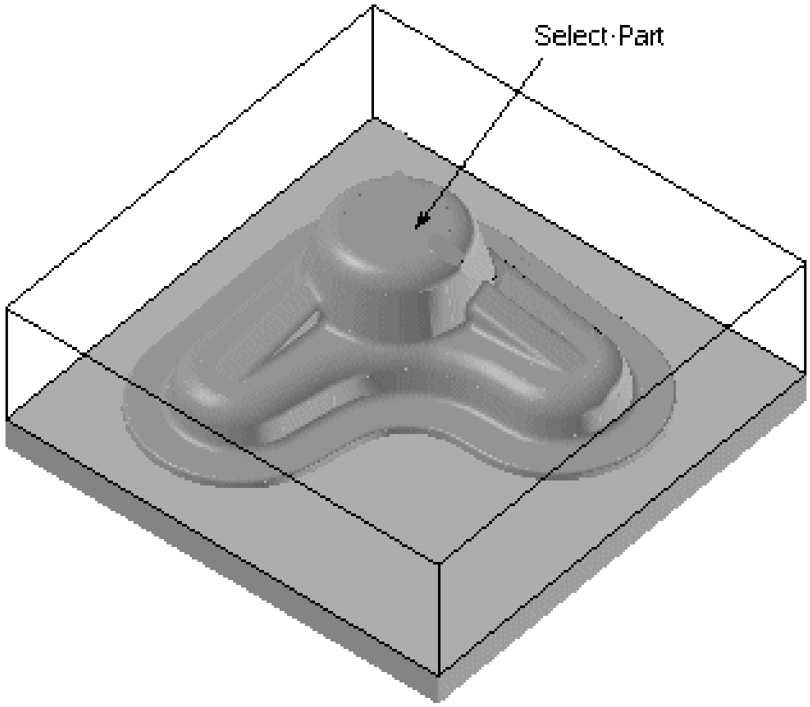
The MILL_GEOM dialog is displayed.

❑ Choose the **PART** icon.

❑ Choose the **Select** button.



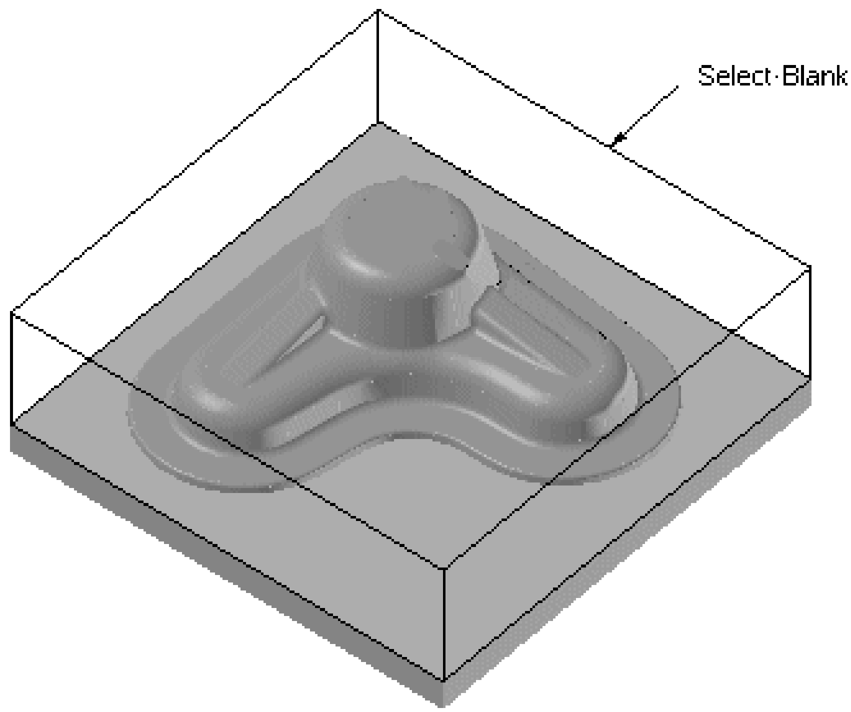
❑ Select the part, as shown below:



❑ Choose **OK**.

❑ In the MILL_GEOM dialog, choose the **Blank** icon, and then choose **Select**.

❑ Choose the Blank, as shown:



❑ Choose **OK**.

There is one more item to change before accepting these selections. The designed part is being made from aluminum. Changing the material specification to the proper material insures that the proper feeds and speeds are used in the machining process.

❑ Select the **Material** button from the MILL_GEOM dialog.

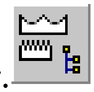
❑ Choose **ALUMINUM** from the list, and then choose **OK**.

To review, you just performed these tasks :

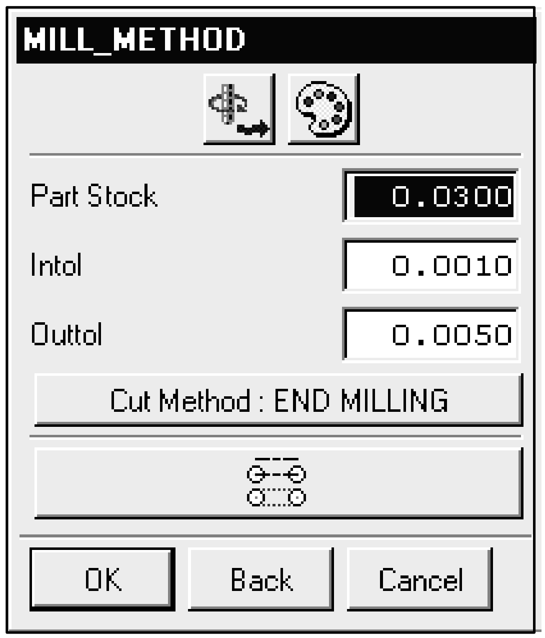
- You selected the Blank geometry, which is the material that the part will be cut from
- You selected the part geometry, which is the material that will remain
- You specified the part material (aluminum). which is used to determine the feeds and speeds

❑ Choose **OK** to accept the settings.

The last step before creating an operation is to define a Method for cutting. The Method Parent Group can specify the amount of stock remaining on the part as well as default feed rates and display options for the tool.

❑ Change to the **Machining Method View** of the Operation Navigator. 

❑ Double click on the **MILL_ROUGH** Method Parent Group.

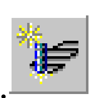


The Part Stock default is 0.030. You will want to leave an additional .020.

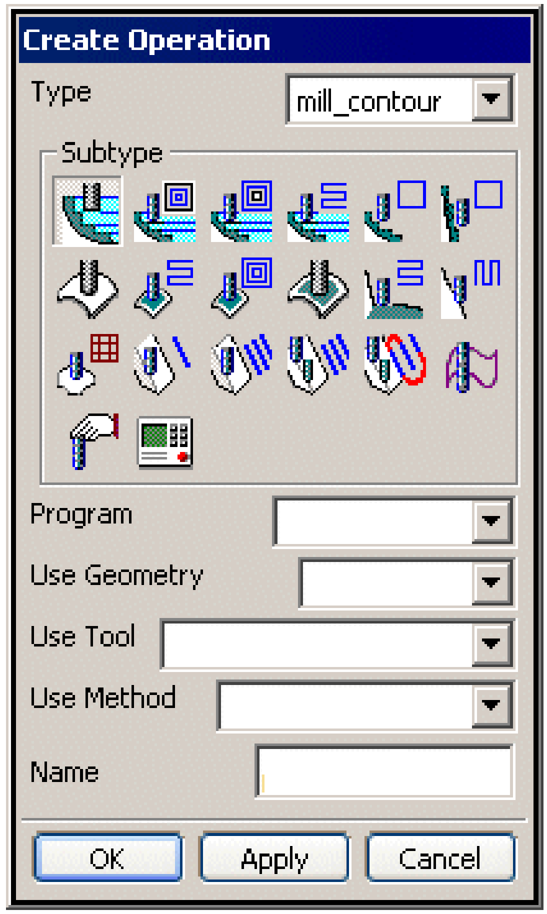
❑ Change the Part Stock value to **0.050**

❑ Choose **OK**.

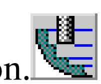
Now it is time to create the operation, using the Parent Groups that you have just created/modified to pass information to the operation.

❑ Choose the **Create Operation** icon. 

The Create Operation dialog is displayed.

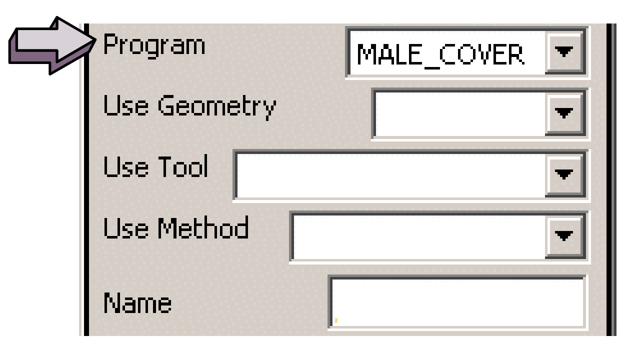


❑ If necessary, change the Type to **mill_contour**.

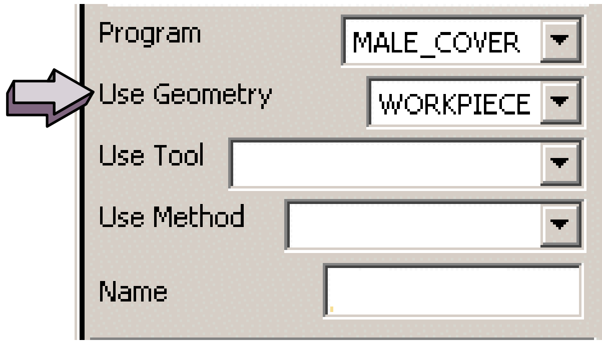
❑ Choose the **CAVITY_MILL** icon. 

Set the four Parent Groups to the items you created/modified.

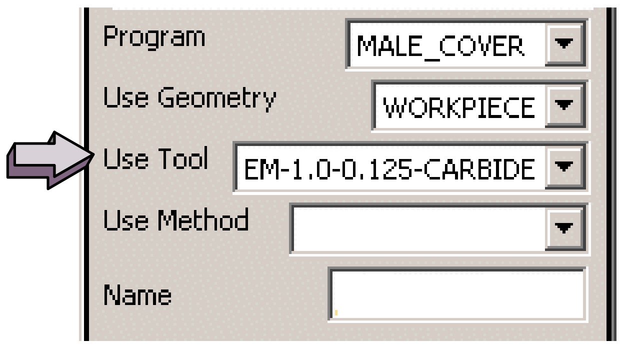
❑ Set the Program Parent Group to **MALE_COVER**.



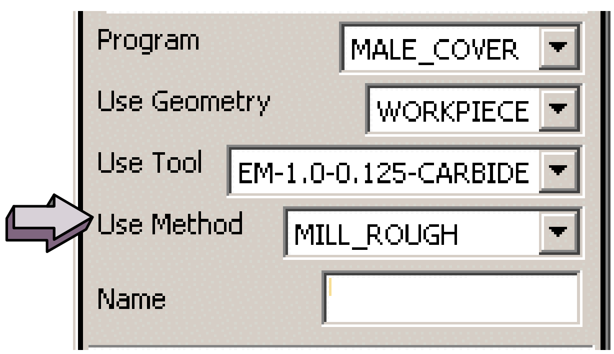
❑ Set the Use Geometry Parent Group to **WORKPIECE.**



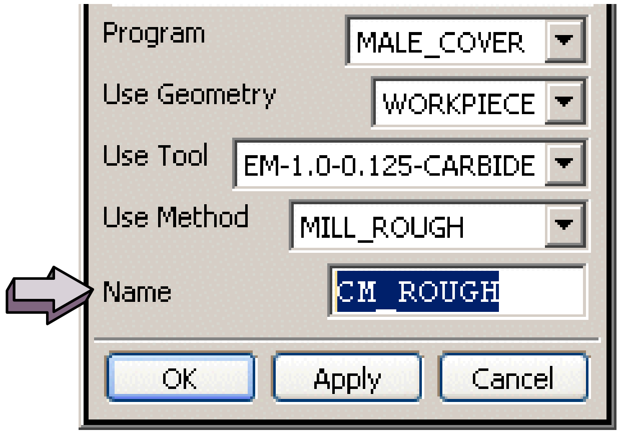
❑ Set the Use Tool Parent Group to **EM-1.0-.125-CARBIDE.**



❑ Set the Use Method Parent Group to **MILL_ROUGH.**

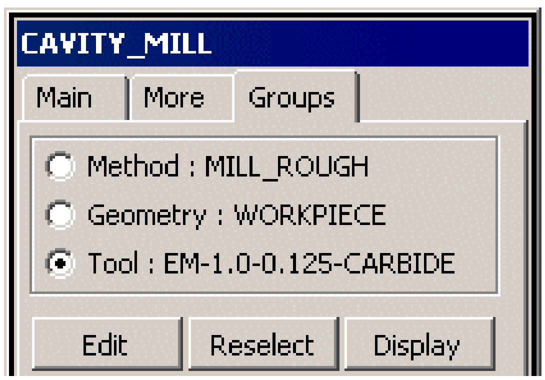


❑ In the Name field, enter **CM_ROUGH**.



❑ Choose **OK**.

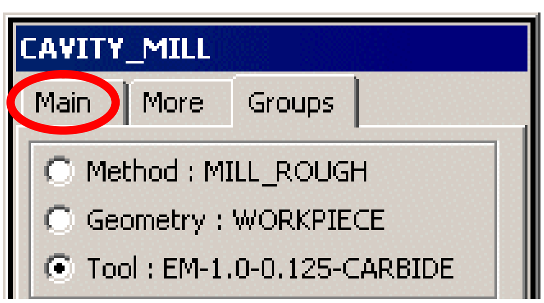
The newly created operation is displayed.



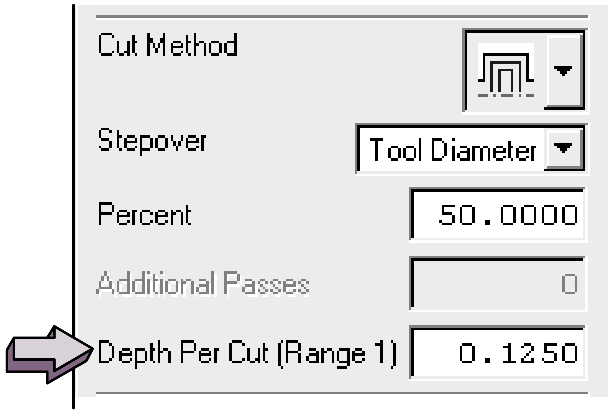
Examine the Group property page. It is displaying the Parent Group object names that have been selected/created at this point.

This operation could successfully be generated now, but you are going to make one more change, the depth per cut.

❑ Select the Main property page.

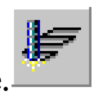


❑ In the middle of the dialog, for Depth of Cut (Range 1) change the value to **0.125**.

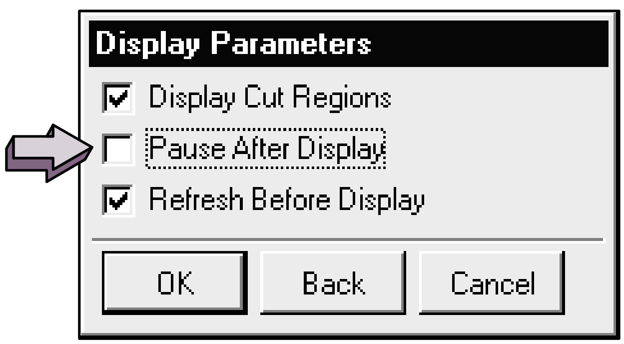


The Cut Levels will now be spaced .125 apart, instead of .25 as previously specified.

Now generate the operation.

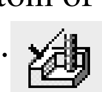
- ❑ Choose **Generate**. 

- ❑ Turn off **Pause After Display** in the Display Parameters dialog.



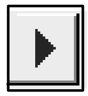
- ❑ Choose **OK**.

It is somewhat difficult to examine each cut level while generating tool paths. There are several analysis tools available for this purpose. You will use one of them now.

- ❑ At the bottom of the CAVITY_MILL dialog, choose the **Verify** icon. 

- ❑ Choose the **Dynamic** tab at the top of the dialog.

- ❑ At the bottom of the dialog, choose the **Play Forward** button.



It may take several moments, but Verify-Dynamic will display the results of the CM_ROUGH operation.

- ❑ Choose **OK** when finished viewing the verified operation.

- ❑ Choose **OK** in the CAVITY_MILL operation dialog to accept the tool path.

- ❑ **Save** and **Close** your part file.

This completes the activity and the lesson.

SUMMARY

Cavity Milling is used to remove large amounts of material in roughing operations. You can use Cavity Milling on planar or contoured geometry. Material removal is performed in levels using a fixed tool axis.

In this lesson you:

- learned how to create a Cavity Milling operation
- learned how information passes from Parent Group objects to operations through the concept of inheritance
- created a cutting tool
- modified Parent Group objects
- changed inheritance by moving operations in the Operation Navigator

Visualization

Lesson 3

PURPOSE

Visualization displays tool paths for your visual inspection.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Use Visualization to inspect single and multiple tool paths
- Display the tool as it removes material in one or more tool paths
- Change Tool Path display options

3

This lesson contains the following activities:

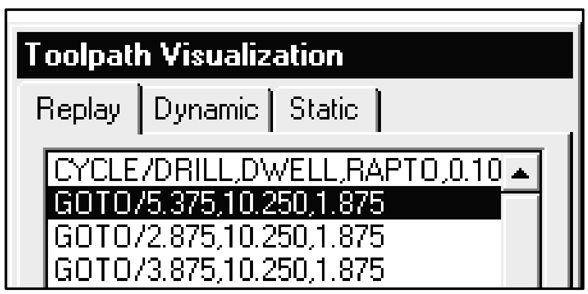
Tool Path Visualization

Tool Path Visualization provides graphical tool path display for all machining operations. Visualization also includes detecting unacceptable conditions such as gouging.

There are three methods of Verification:

- Replay – displays the tool or tool assembly, positioning to each GOTO to GOTO within the tool path
- Dynamic – displays the tool as it moves along the tool path(s), showing the path(s) with the material removed
- Static – displays the results *after* the tool path has removed the material

The Dynamic and Static verification methods require that you define the Blank in the WORKPIECE Parent Group object.



You can access this dialog by choosing:

- The Toolpath Verify icon from the toolbar
- Toolpath→Verify, from the Operation Navigator
- Tools→Operation Navigator→Toolpath→Verify, from the menu bar
- Toolpath Verify within an operation

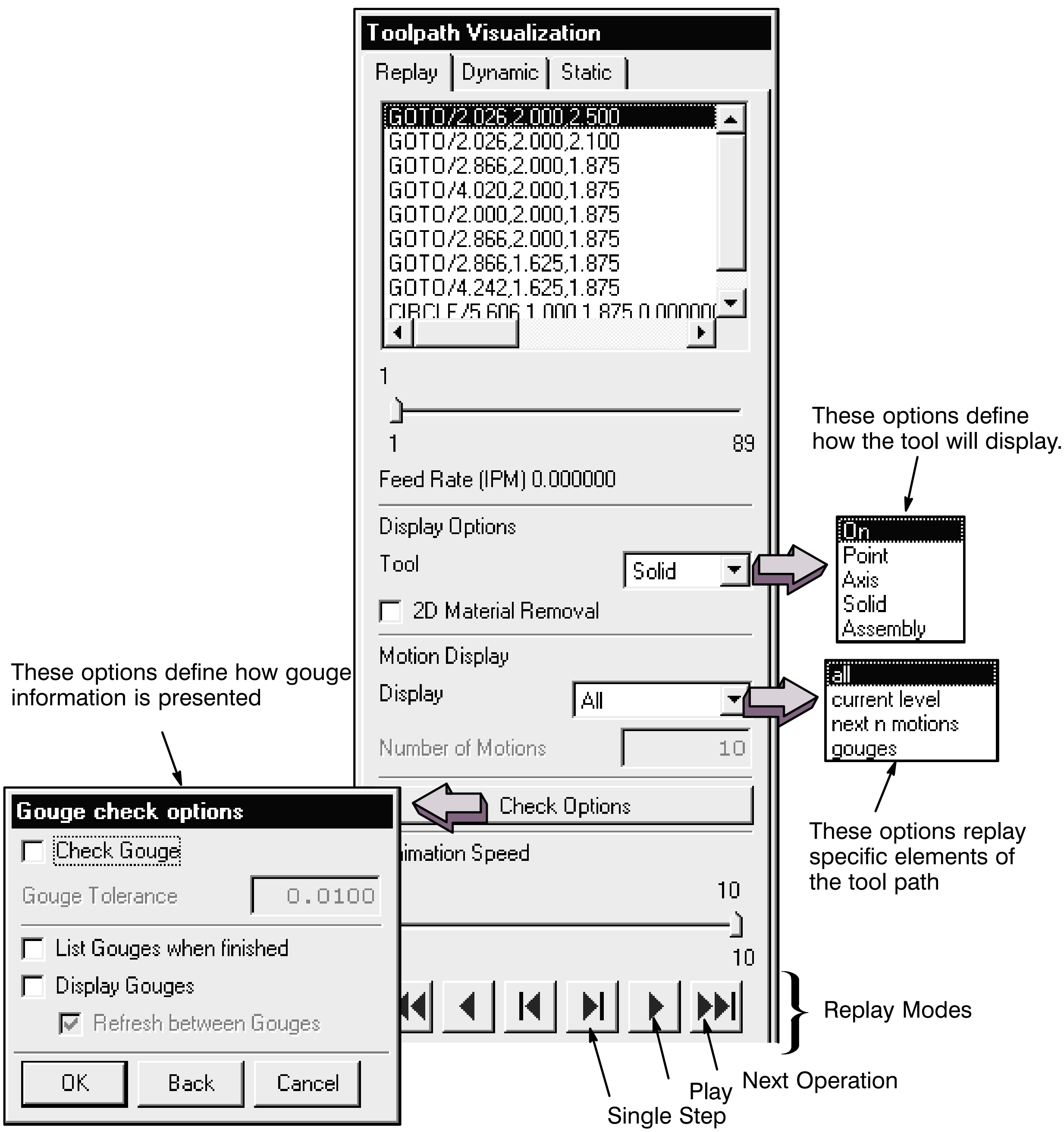
You also have the option to select a single tool path or a series of tool paths for visualization.

Replay

Replay is designed to provide a quick method of viewing the tool path by displaying the cutter location at each GOTO point. This is the quickest method of verification.

The Replay options allow you to:

- Replay single or multiple tool paths
- Replay by cut level
- Replay a specified number of tool motions
- Replay displaying the tool holder assembly

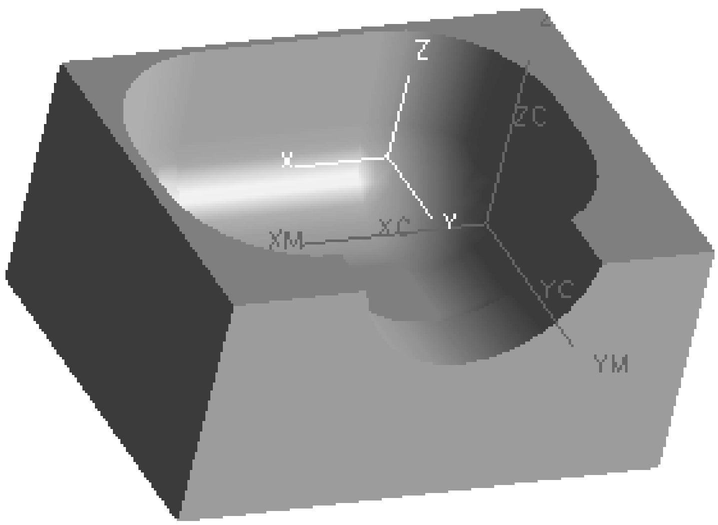


Activity 3–1: Using Replay Verification

In this activity, you will use the Replay option of Toolpath Visualization. You will also use the Single Step, Play, and Next Operation functions. These functions are common to all of the Visualization options.

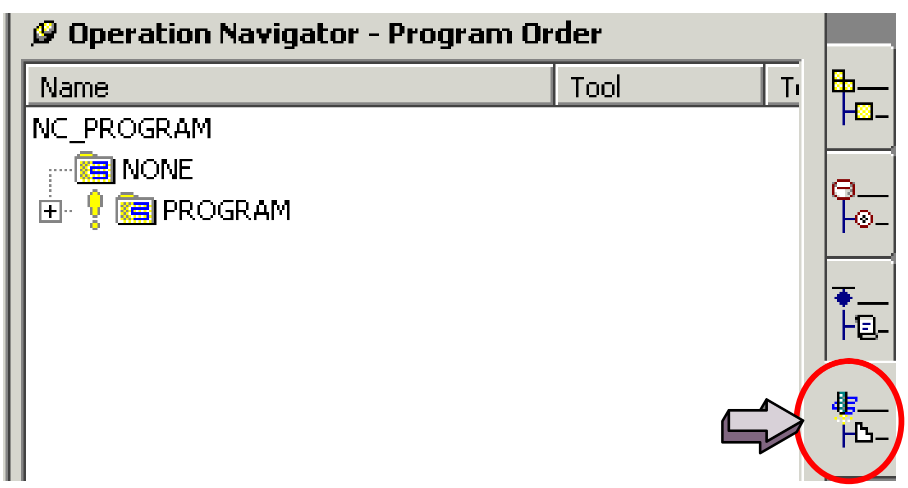
Step 1 Open the part file.

❑ Open the part file **mmp_die_insert_1.prt**.



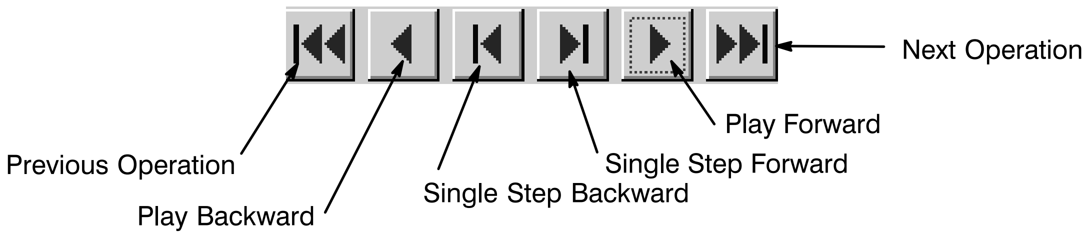
❑ Enter the **Manufacturing** application.

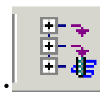
❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.



Step 2 Using the Replay option.

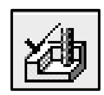
You are going to use the different replay buttons shown below.



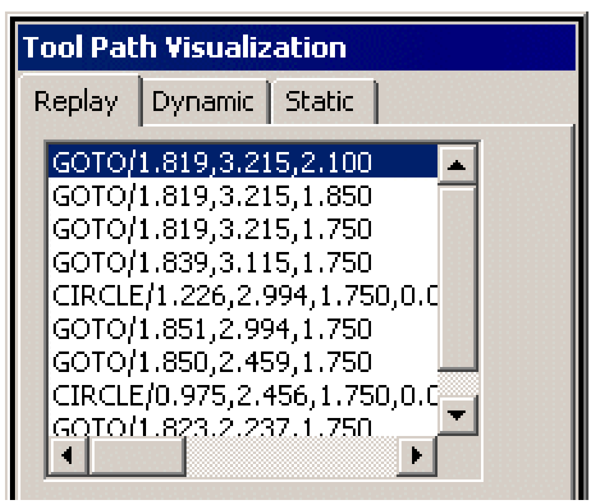
❑ Expand the Program Parent Group object . 

❑ Highlight the **Program** Parent Group object in the Operation Navigator.

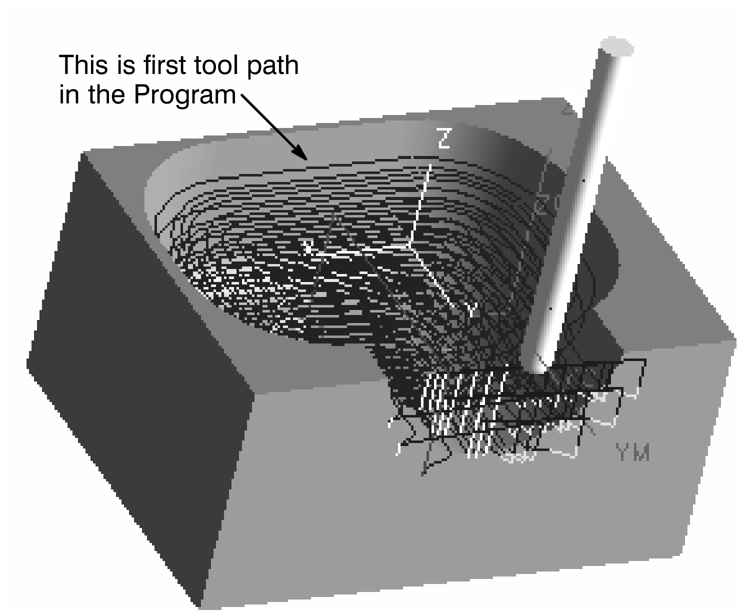
Note that there are a number of operations in this program.

❑ Choose the **Verify Toolpath** icon. 

The first GOTO of the operation, CAVITY_MILL (of the Parent Group object, PROGRAM) is listed in the listing window of the Tool Path Visualization dialog.

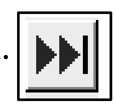


Notice that the operation, CAVITY_MILL, as listed in the Operation Navigator is accentuated in red since it is the operation being currently verified.



Step 3 Using the Next Operation option.

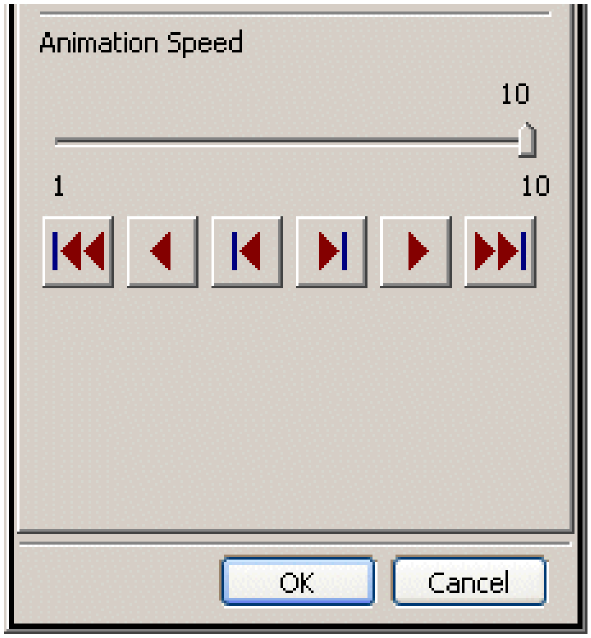
You are going to use the **Next Operation** button to sequentially advance the tool path display.

❑ Choose the **Next Operation** button. 

The next operation in the PROGRAM parent group is replayed quickly. Notice in the Operation Navigator that the second operation is displayed in red. The previous operation name is now displayed in blue.

Step 4 Slowing down the animation speed.

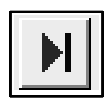
You can slow down the animation speed using the Animation Speed slider bar.



❑ Move the Animation Speed slider bar to approximately **8**.

Step 5 Using the Single Step Forward option.

You are going to use the **Single Step Forward** button to advance the animation step by step.

❑ Choose the **Single Step Forward** button. 

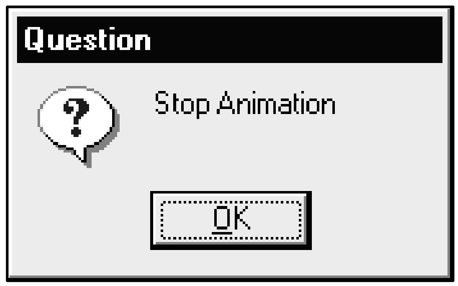
The animation advances one motion (step).

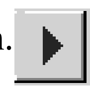
❑ Choose the **Single Step Forward** button several more times.

Step 6 Using the Play Forward button option.

You are going to use the **Play Forward** option to replay all of the operations in the Parent Group PROGRAM.

Before choosing this option, note the following method for halting animation. After choosing the Play Forward option, a Stop Animation message window is displayed; choosing the OK button will end the animation.



❑ Choose the **Play Forward** button. 

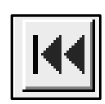
Each operation is replayed. Again, note that the operation displayed in red is the current operation.

❑ **Stop** the animation when you are ready.

Step 7 Using the Motion Display options.

These options determine how the tool paths are displayed.

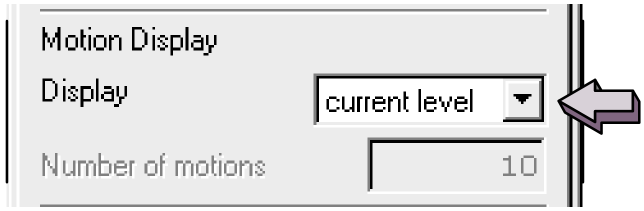
You are going to backtrack in the PROGRAM Parent Group object.

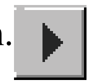
❑ Use the **Previous Operation** button  to return to the first operation, CAVITY_MILL.

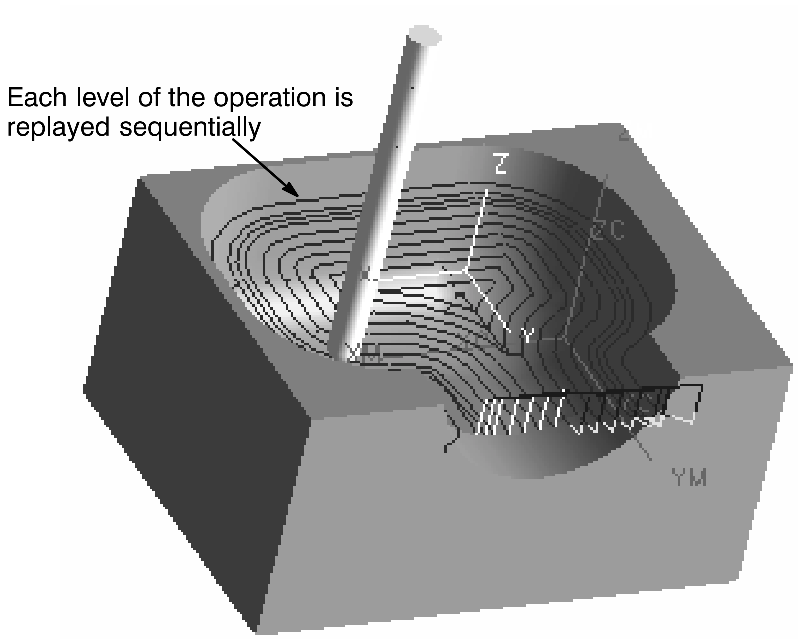
GOTO/1.819,3.215,2.100 should be highlighted and the Cavity_Mill operation should be displayed in red in the Operation Navigator.

You are now going to use the current level option. Current level replays the operation one level at a time.

❑ In the Motion Display area, change Display to **current level**.



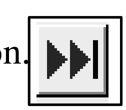
❑ Choose the **Play Forward** button.  Replay the operation for several levels and then **Stop** the animation.

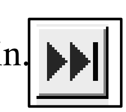


3

The first level of the operation is replayed. The screen refreshes and replays the second level and so on.

Now you are going to use the **Next Operation** option. This option will replay the first level of the current operation, then move to the first level of the next operation. This sequence is repeated until there are no more operations to verify.

❑ Choose the **Next Operation** button. 

❑ Choose the **Next Operation** button again. 

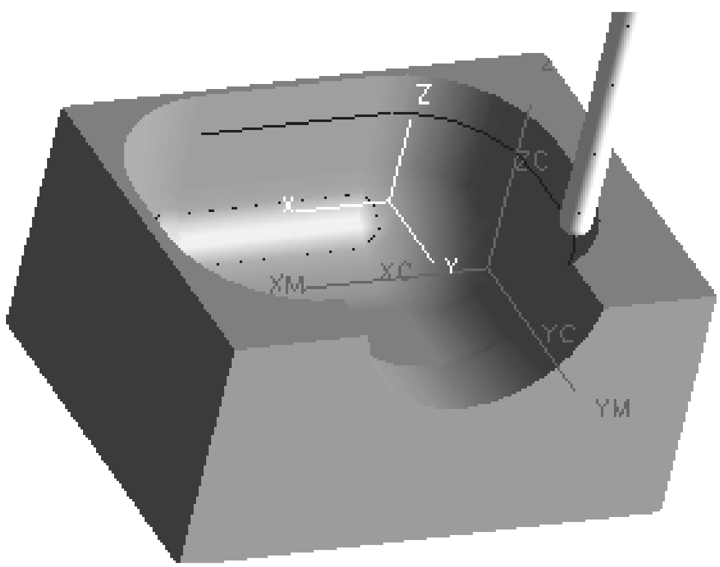
Step 8 Using the motions display option.

This option, **next n motions**, will replay only a specified number of GOTO's ahead of the current tool position.

❑ In the Motion Display area, change Display to **next n motions.**

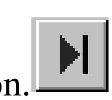
Note that the Number of motions value field is now available. Use the default of 10.

❑ Choose the **Play Forward** button. 



Note that the next 10 tool positions are displayed.

❑ **Stop** the animation.

❑ Choose the **Single Step Forward** button. 

The operation is replayed one motion at a time.

Note that the next 10 tool positions are also displayed.

❑ **Stop** the animation.

Step 9 Displaying the tool holder.

You are going to graphically display the tool holder while replaying an operation.

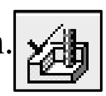
Note that the tool must have a tool holder defined. These tools can be retrieved from the Tool Library.

You are going to close and then reopen the Toolpath Visualization dialog.

❑ Choose **Cancel** to close the Toolpath Visualization dialog.

❑ Highlight the operation name **TOOL_HOLDER** on the Operation Navigator.

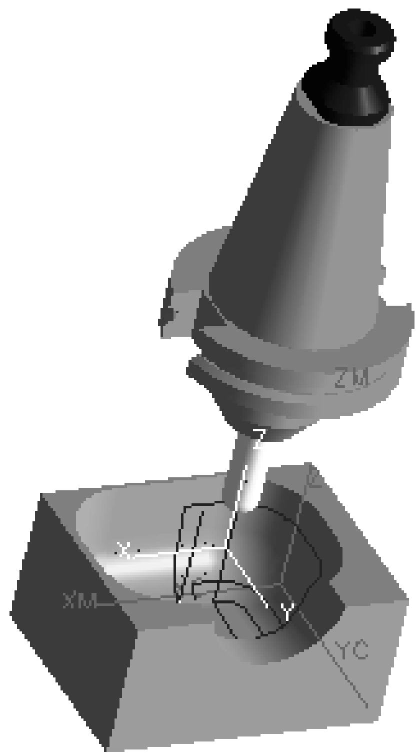
Note that the operation is displayed in blue. This indicates a tool change.

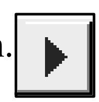
❑ Choose the **Verify Toolpath** icon. 

The Toolpath Visualization dialog is displayed.

❑ In the Display Options area, change Tool to **Assembly**.

The tool and tool holder are displayed.



- ❑ Choose the **Play Forward** button. 

  The operation is replayed.

- ❑ **Stop** the animation when you are ready.

- ❑ Choose **Cancel** from the Toolpath Verification dialog.

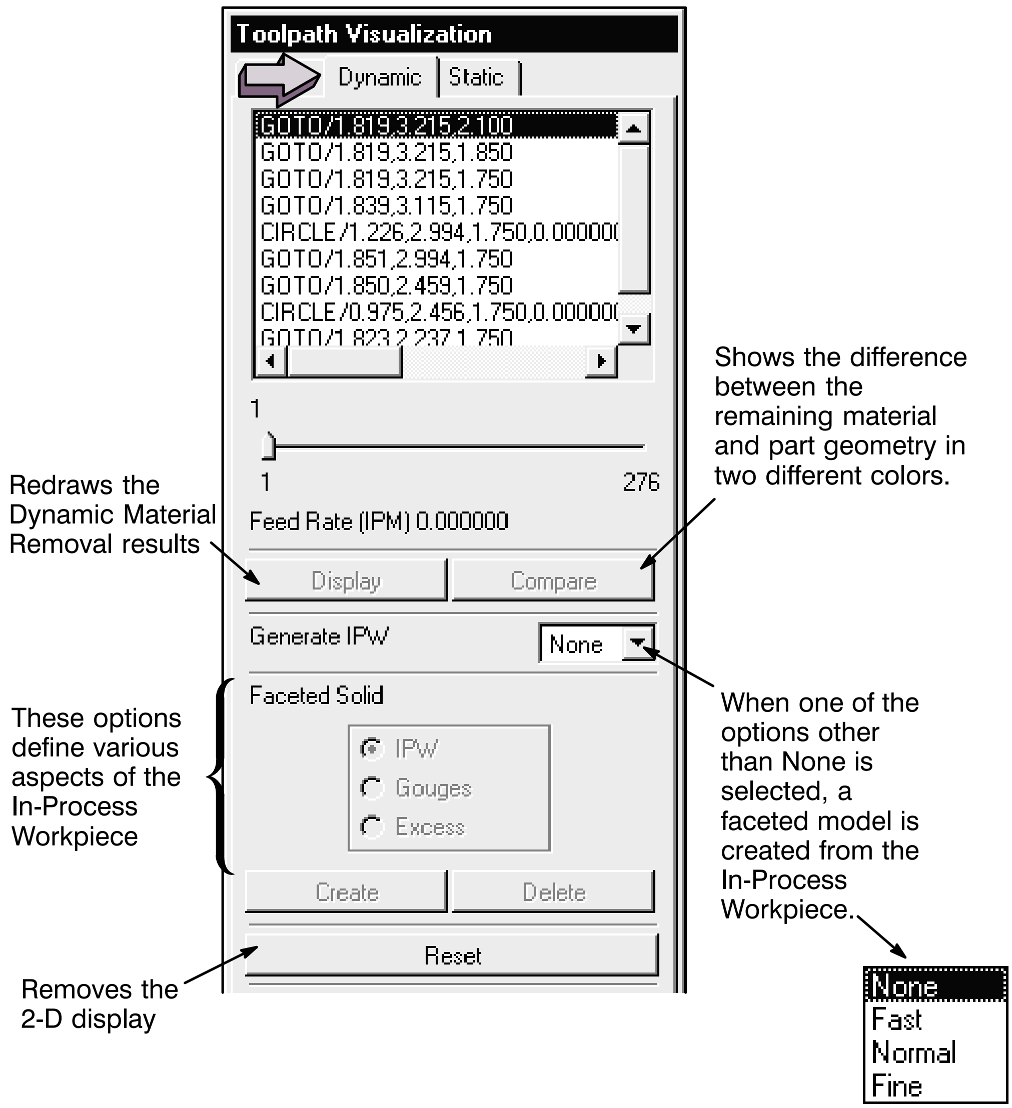
- ❑ **Close** the part.

This completes this activity.

Dynamic Tool Path display

Dynamic displays the cutter as it follows a tool path and removes material.

You can also use the Dynamic option to generate a faceted model from the In-Process Workpiece (IPW).



If material is encountered at RAPID moves, then these areas are highlighted in red as a warning.

If you are using multiple tools, each tool will display in a different color.

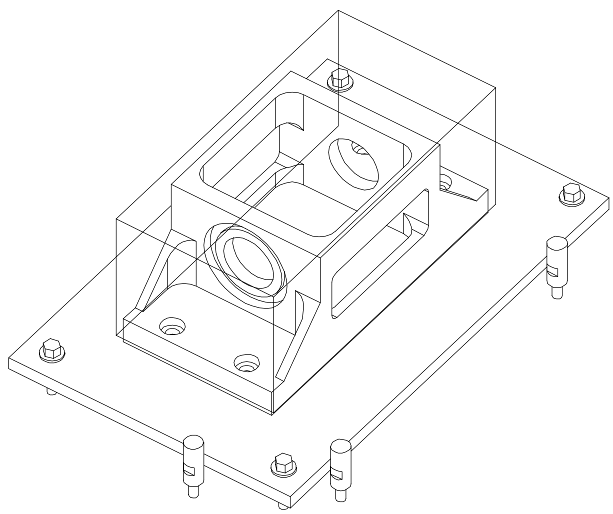
Dynamic Material is a 2D display. Rotate or Pan options are not available.

Activity 3–2: Using Dynamic Removal

You are going to use the Dynamic Removal method of visualization option to verify previously created tool paths.

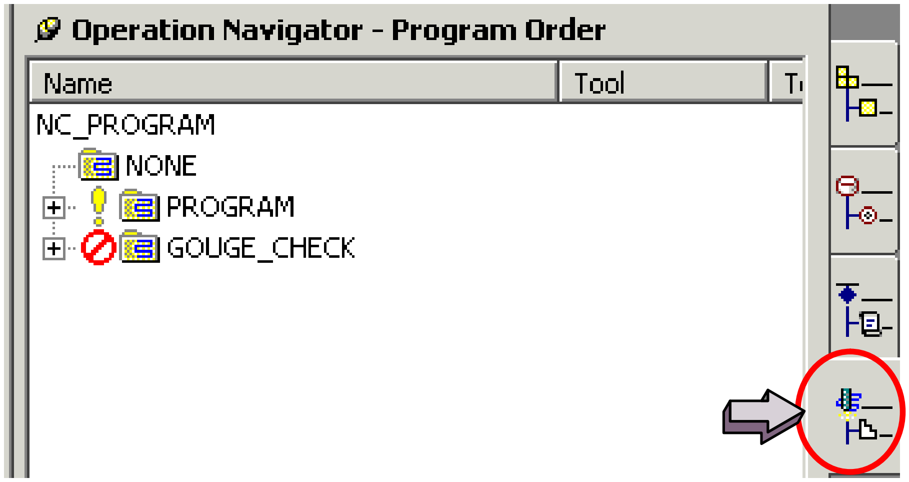
Step 1 Open the part file.

❑ Open the part file **mmp_bearing_case_mfg_3.prt**.



❑ Enter the **Manufacturing** application.

❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.



Step 2 Using the Dynamic Material Removal option.

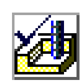
This option requires that you have Blank geometry selected. The Blank geometry is selected in the WORKPIECE Parent Group.

❑ Expand the **Program** Parent Group.

There are three operations in this program.

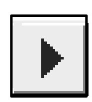
❑ Highlight the **PROGRAM** Parent Group on the Operation Navigator.

This will show the cutting motion, operation by operation, in the correct order.

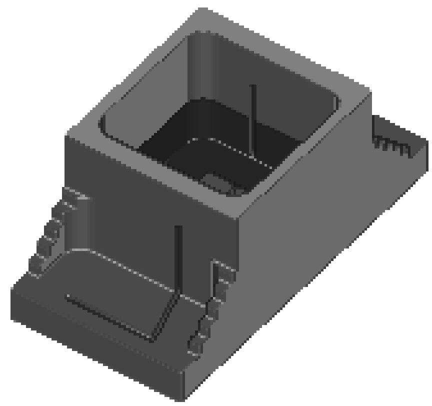
❑ Choose the **Verify Toolpath** icon from the tool bar. 

The Toolpath Visualization dialog is displayed.

❑ Choose the **Dynamic** tab on the Toolpath Visualization dialog.

❑ Choose the **Play Forward** button. 

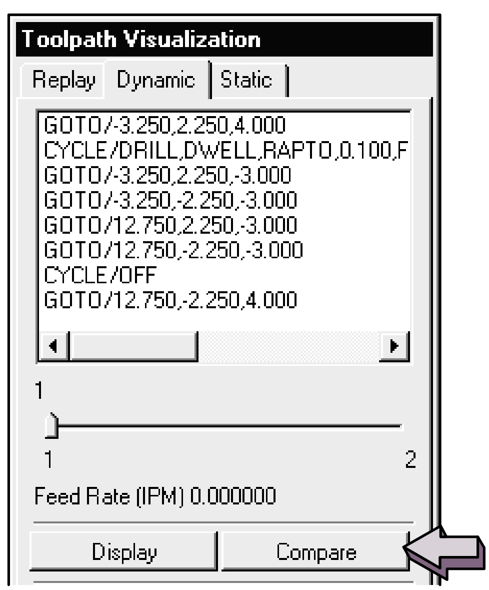
The blank material is graphically removed.



Note that:

- the center drill cuts through the part
- the center drill does not lift far enough above the blank surface between holes
- the end mill is to short; you can see this in a slower replay of the tool path
- the Blank is the light blue color
- each tool path is painted in a different color

❑ As shown below, choose **Compare**.



Blank and stocked areas are displayed in gray. The stock was determined by the Method Parent Group, MILL_ROUGH.

The Cavity Milling operation did not finish cutting the part.

The Planar Profile operation created a finish cut.

- ❑ Choose **Display.**

The results are displayed again.

- ❑ Choose **Cancel**.

You will now create an In-Process Workpiece, that can be used to show remaining material to be cut.

Step 3 Creating an In-Process Workpiece.

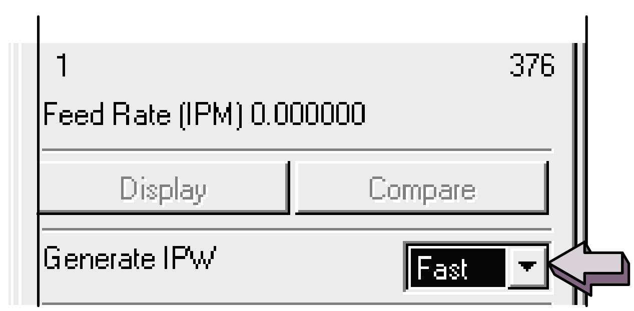
You are going to change the **Generate IPW** setting to create an In-Process Workpiece.

The results of this option are best viewed if you change the object color.

- ❑ Choose **Preferences→Object**.
- ❑ Change the color to **Pink**.
- ❑ Choose **OK**.
- ❑ Highlight the **PROGRAM** Parent Group object on the Operation Navigator.
- ❑ Choose the **Verify Toolpath** icon from the tool bar. 

The Toolpath Verification dialog is displayed.

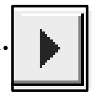
- ❑ Choose the **Dynamic** tab on the Toolpath Visualization dialog.
- ❑ Next to the **Generate IPW** label, change the setting to **Fast**.



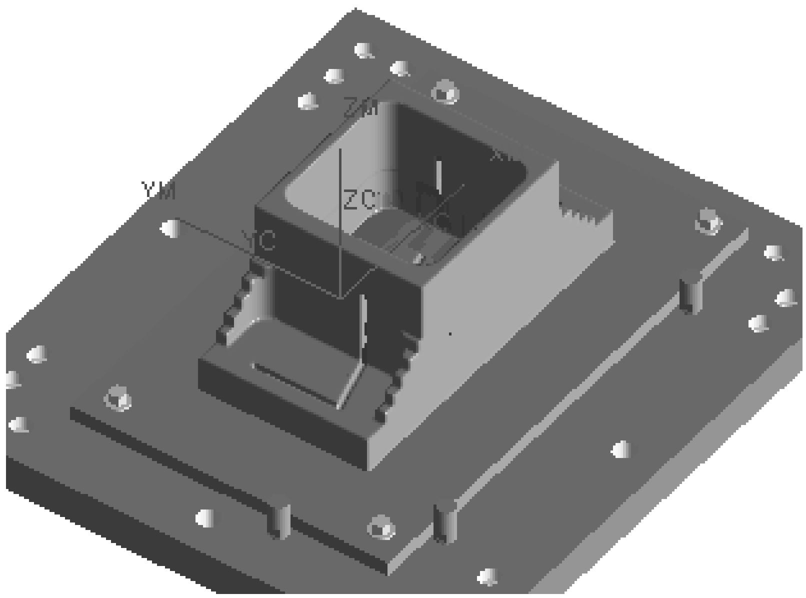
This will generate an In-process Workpiece.

The In-process Workpiece is created from the faceted model which was created when you used the Dynamic Material Removal option.

The three options; Fast, Normal, and Fine, refer to the quality of the faceted model generated as the In-process Workpiece.

❑ Choose the **Play Forward** button. 

❑ After the animation ends, choose **Create**.



The In-Process Workpiece is displayed.

You will not be using the IPW, therefore you will delete it.

❑ Choose **Delete.**

❑ Choose **Cancel** from the Toolpath Visualization dialog.

❑ Do not save the part file.

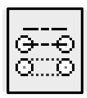
❑ **Close** the part file.

This completes the activity.

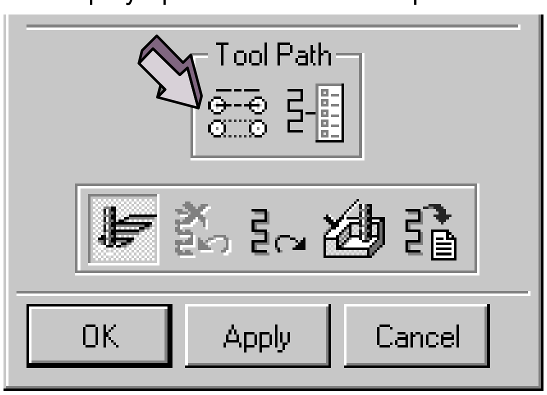
Setting Tool Path Display Options – Edit Display

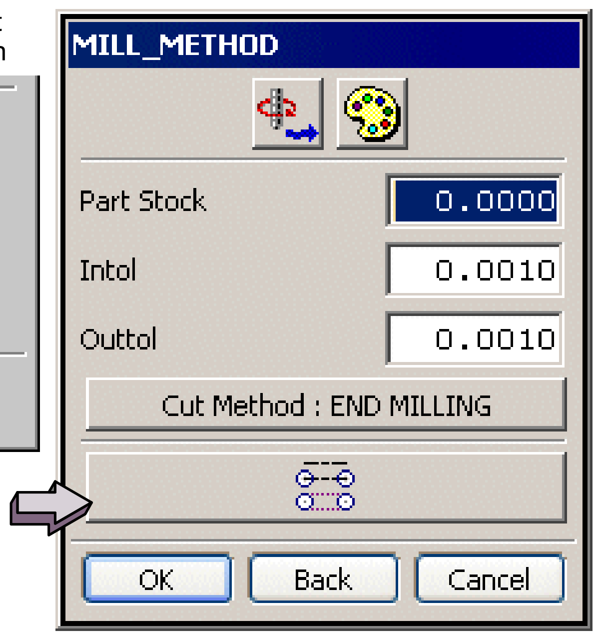
The tool path display options are used to control the display of the cutting tool and the tool path.

You can define the setting in the Method Parent Group or within an operation. Within the Method Parent Group, the settings will apply to all operations that are located below it (inherited). Within the operation, the settings apply to just that operation. Individual operation display settings override settings from the Parent Group.

To edit the Tool Path Display options within an operation, you use the Edit Display button. 

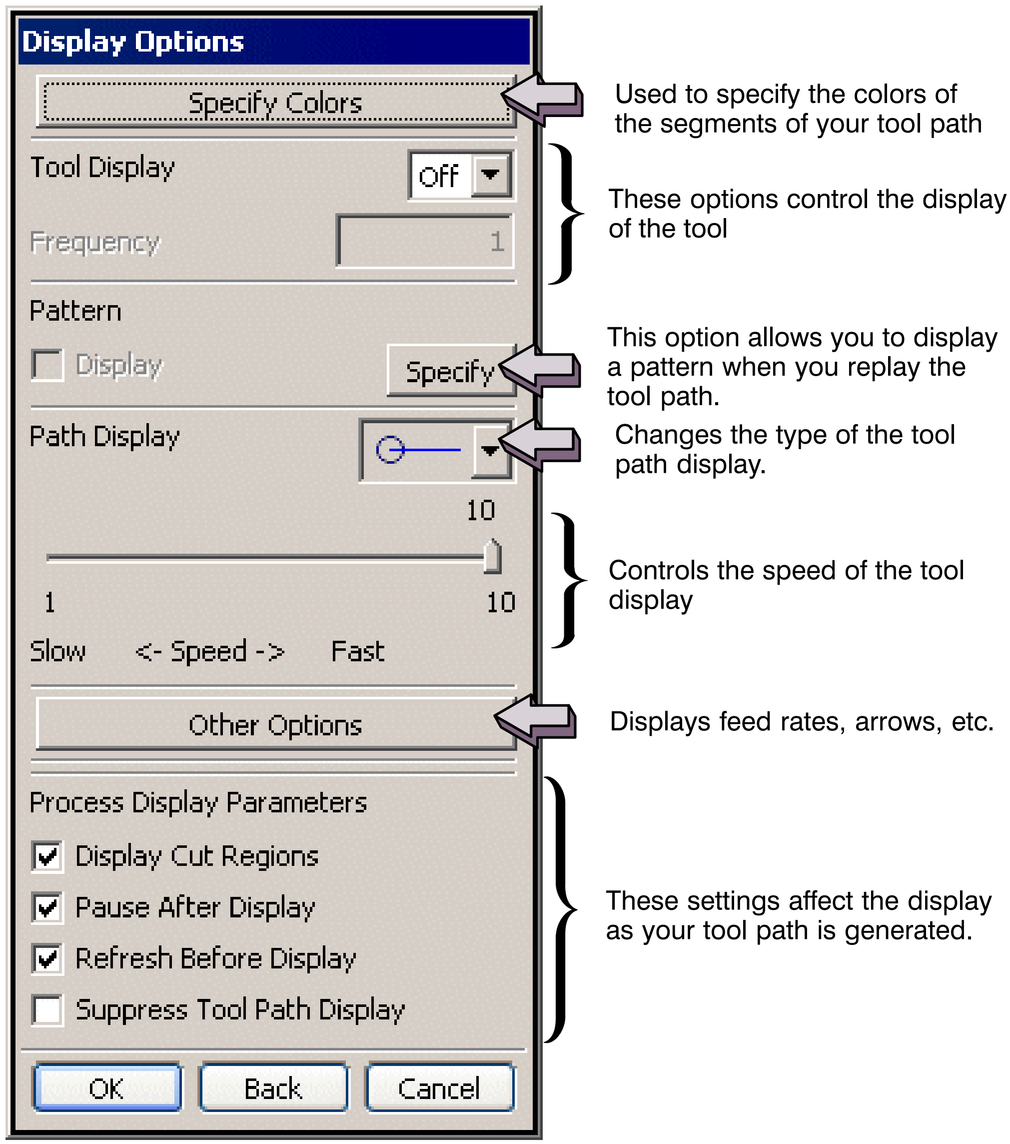
The Edit Display button allows you to edit the display option in the current operation





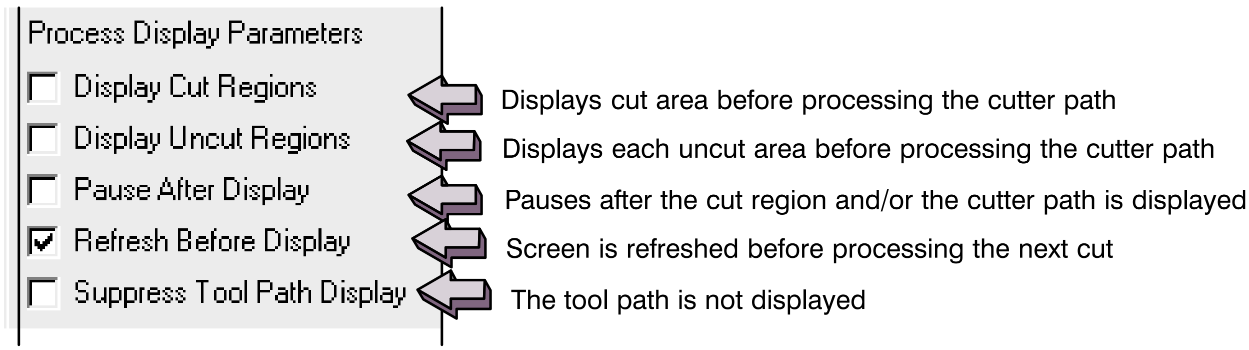
Setting the display options in the MILL_METHOD Parent Group allows the setting to be inherited by all tool paths placed under this Parent Group

The following is a summary of the different areas of the Display Options dialog:



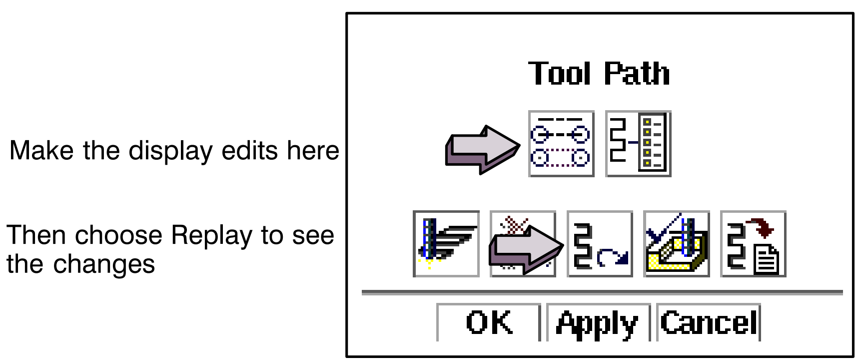
The Process Display Parameters

Take a few moments to look at the tool path Display Options.



You can toggle the options on and off as needed. For example, you may want to see the Cut Regions as you start to develop the tool path. Later, when you may only need to look at the cutter path, the cut region display can be turned off.

Also not that you can edit the tool path display of a generated tool path at any time without having to reject the tool path first. Make the display edits, then choose **Replay** from the dialog in order to see the changes. The settings are saved within the operation.

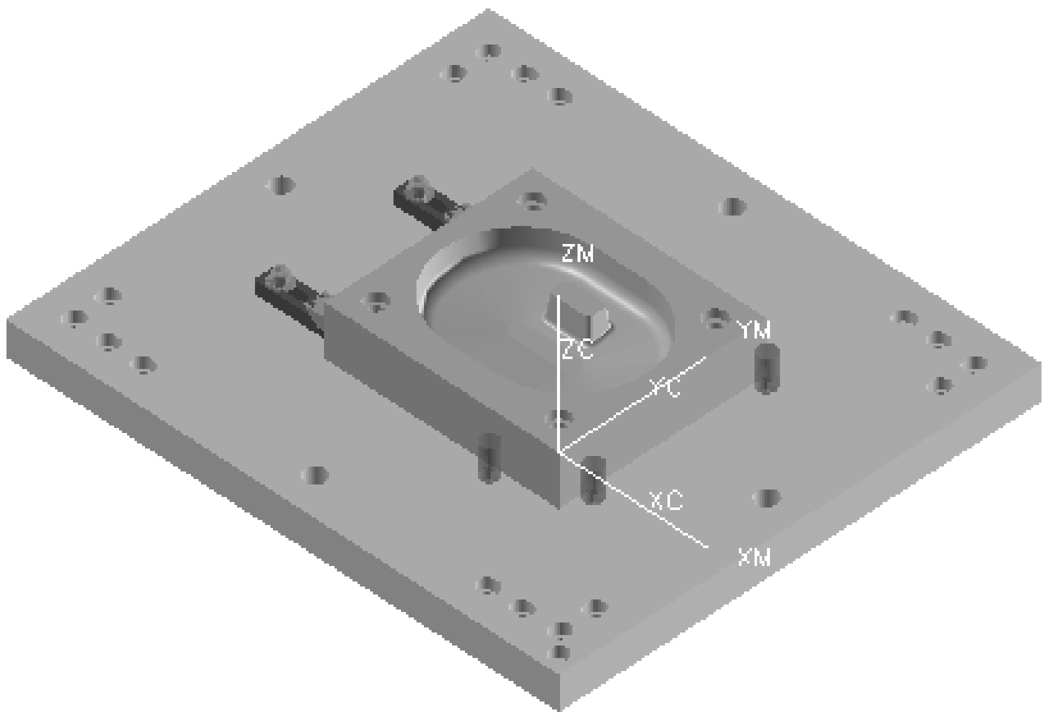


Activity 3—3: Using the Edit Display Options

In this activity, you will modify the Display Options by changing the tool ON/OFF display, change the color of the tool display and change the Replay speed.

Step 1 Open and rename the Part File.

❑ Choose **File→Open** then choose **mmp_cap_die_female_mfg_wp.prt**.



Step 2 Save the part file and enter the Manufacturing application.

❑ Choose **File→Save As**, and save the part as *****_cap_die_female_mfg_wp.prt.**

❑ Choose **Application→Manufacturing**.

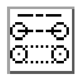
❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.

The Operation Navigator is displayed.

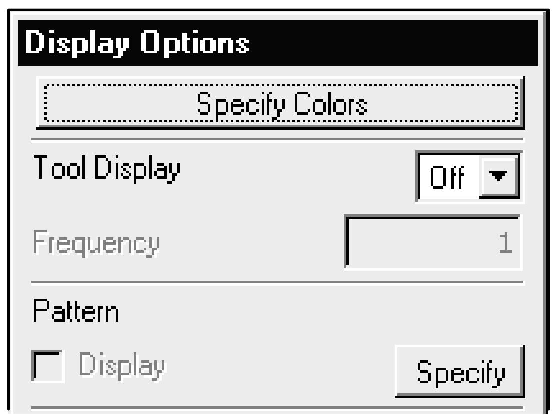
Step 3 Edit the operation and change the tool display.

❑ Expand the **FEMALE_DIE** Parent Group object.

❑ Double click on the operation SLAB_TOP.

❑ Under the Tool Path label, choose the **Edit Display** icon  .

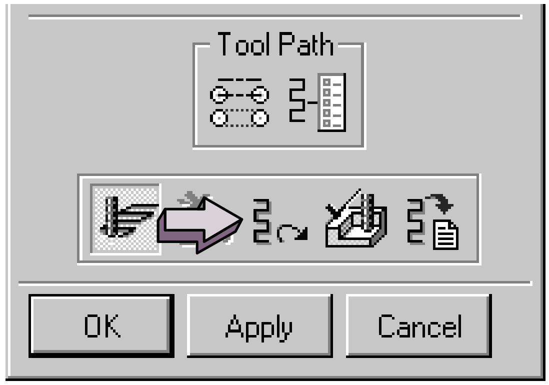
The Display Options dialog is displayed.



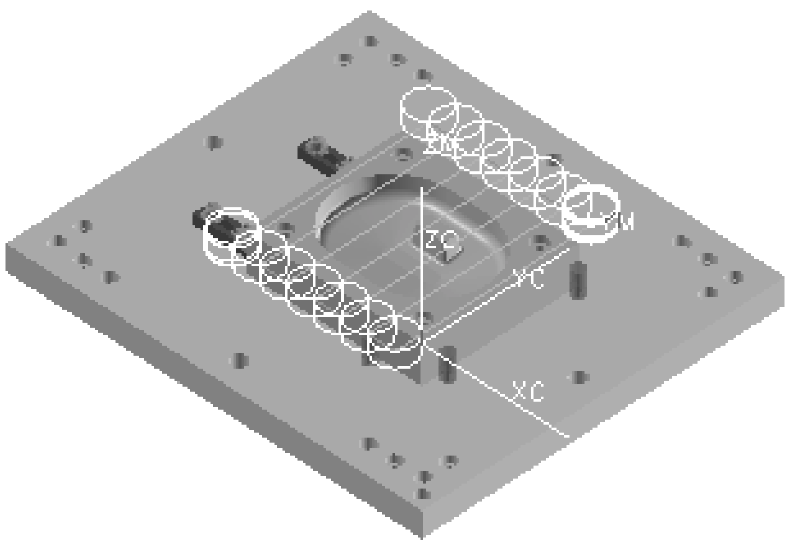
❑ Change the **Tool Display** setting to **3D.**

❑ Choose **OK.**

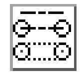
❑ Choose the **Replay** icon from the FACE_MILLING dialog.



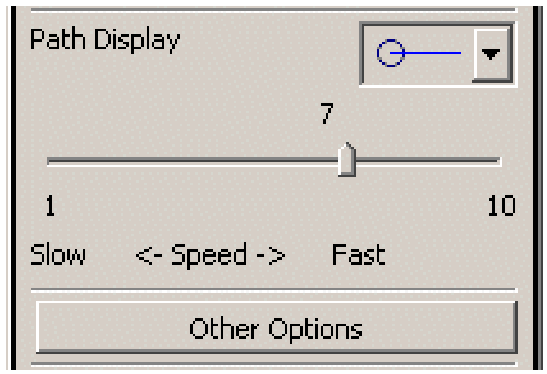
The tool path is replayed displaying the tool.

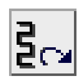


Step 4 Change the replay speed.

❑ Choose the **Edit Display** icon. 

❑ Using the slider bar, change the Frequency setting to **7,** then choose **OK**.



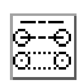
❑ Choose the **Replay** icon. 

The tool path is replayed at a slower rate.

Step 5 Change the display color.

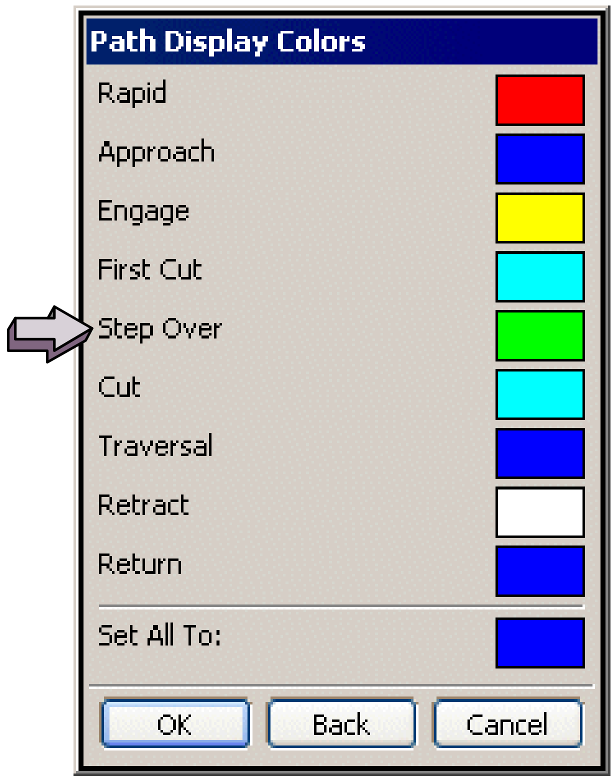
❑ Choose Replay from the FACE MILLING dialog.

Note the display colors of the tool path. The stepover color is green. You will change the color to pink.

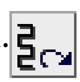
❑ Choose the **Edit Display** icon. 

❑ Choose the **Specify Colors** button.

The Path Display Colors dialog is displayed.

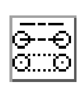


❑ Change the **Stepover** to **Pink,** then choose **OK**.

❑ Choose the **Replay** icon. 

The tool path is replayed showing the stepover in pink.

Step 6 Change some of the Process Display Parameters and examine the results.

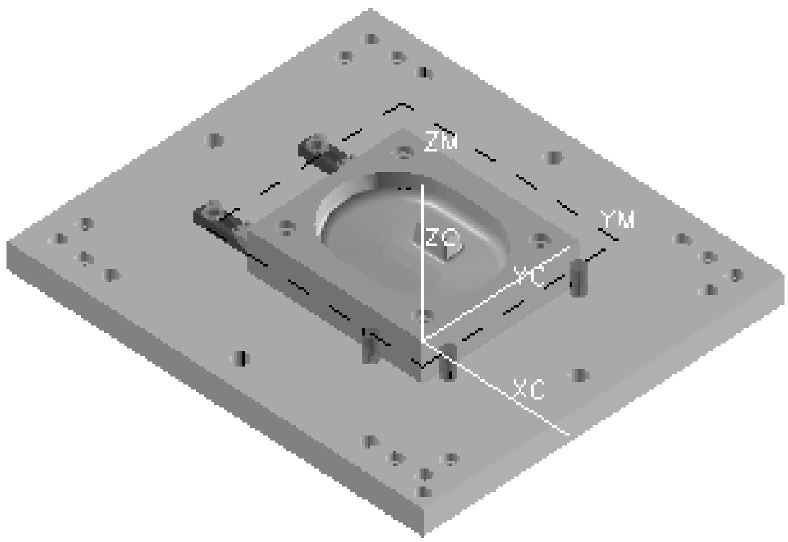
❑ Choose the **Edit Display** icon. 

❑ Under the label, Process Display Parameters, choose **Display Cut Region** to turn it on.

Notice the Pause After Display option is turned on.

❑ Choose **OK.**

❑ **Generate** the tool path.



The cut region is displayed and a pause occurs. This is the result of the Pause After Display option being turned on.

❑ Choose **OK** to continue generating the tool path**.**

❑ **Close** the part file.

The ends this activity and concludes the lesson.

SUMMARY

In this lesson, you learned how to verify operations and tool paths using the Replay and Dynamic options of the Verification feature.

In this lesson you:

- Replayed single and multiple operations
- Used the Dynamic Replay option to examine material removal in an operation
- Replayed operations step by step for visualization purposes
- Used the Edit Display features to change the tool display in an operation

EDS

Assembly Modeling for Manufacturing

Lesson 4

PURPOSE

Assemblies in Manufacturing utilize the Master Model concept. This concept is useful in protecting the design criteria of the part from corruption by other users. By creating a manufacturing assembly and adding a component, such as a fixture plate or clamp, application specific data can be generated in a separate part file which references the master geometry. This avoids duplication of model geometry and allows the concurrent use of the Master Model.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Understand and use the Master Model concept for Manufacturing applications
- Create a Manufacturing assembly
- Add components to a Manufacturing assembly

This lesson contains the following activities:

Review of Assembly Modeling and the Master Model Concept

A Unigraphics assembly is a part file containing stored links to other part files. These part files that are linked to the assembly are commonly referred to as piece parts. Geometry used to define the piece parts of the assembly reside in the original part file and are referenced but not duplicated in the assembly file.

The link in the assembly file is referred to as a component. The component stores pertinent information about the piece part, such as attributes, display parameters, location, orientation, permissions, and relationships to other parts.

The Master Model concept, as applied to Manufacturing, uses an assembly consisting of a one piece part. It is useful in protecting the design criteria of the part from corruption by other users, such as NC programmers, that need access to the design data. In this case the NC programmer will have write access to the assembly file but only read access to the model. The model is referenced for the manufacturing application work.

The Master Model in Manufacturing

Manufacturing personnel have the need to design fixtures, define and generate machining operations, create CNC/NC programs and shop documentation and then save this data within their models. By creating a manufacturing assembly and adding a component to it, application specific data can be generated in a separate part file that references the master geometry. This avoids duplication of model geometry and allows the simultaneous or concurrent use of the Master Model.

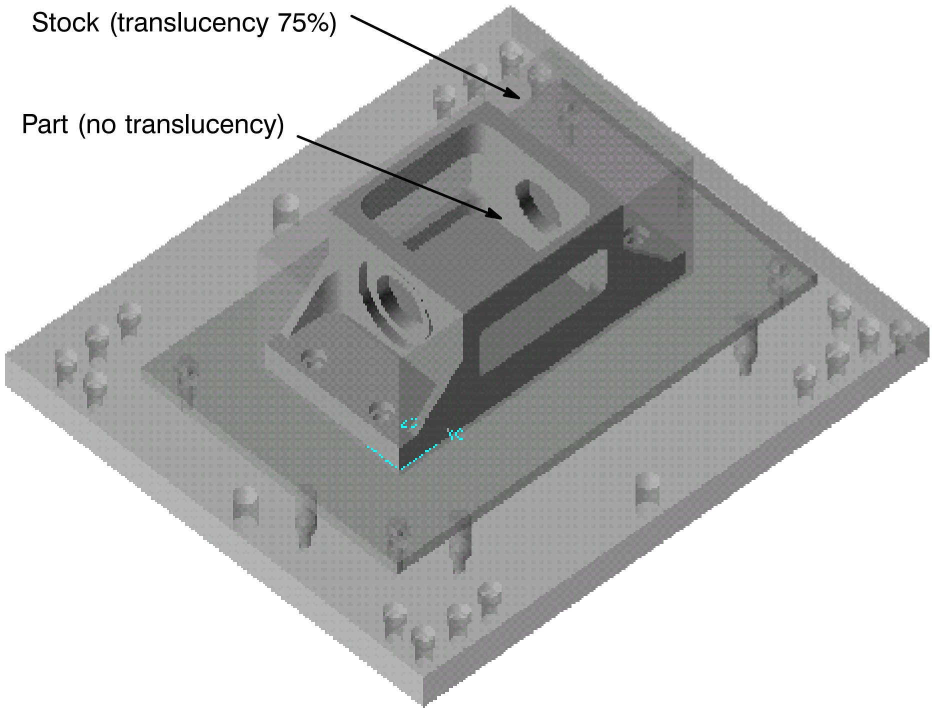
The Manufacturing Assembly

A typical assembly in manufacturing consists of the blank (billet, casting or forging), the part manufactured from the blank, various fixtures plates, mounting plates, tooling plates, clamps, bolts, hold downs and locators.

When creating the assembly, consideration should be given to the assembly's organization and appearance. For example, a typical manufacturing assembly consists of the following:

- the main assembly (all components together)
- the actual part
- blank material for the part
- the fixture comprised of bolts, locator pins, and numerous plates

Different colors should be used for all components to distinguish them from one another. Various degrees of translucency should be applied to components that lay on top of one another, such as part, blank material and fixture plates, to visually determine relationships with other component parts in the assembly.

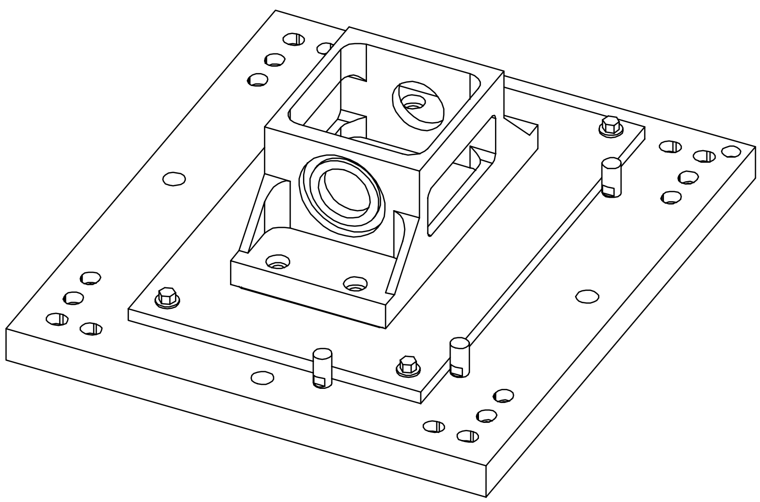


Activity 4—1: Master Model Concept for Manufacturing

This activity will demonstrate the use of a Master Model in a Manufacturing Assembly.

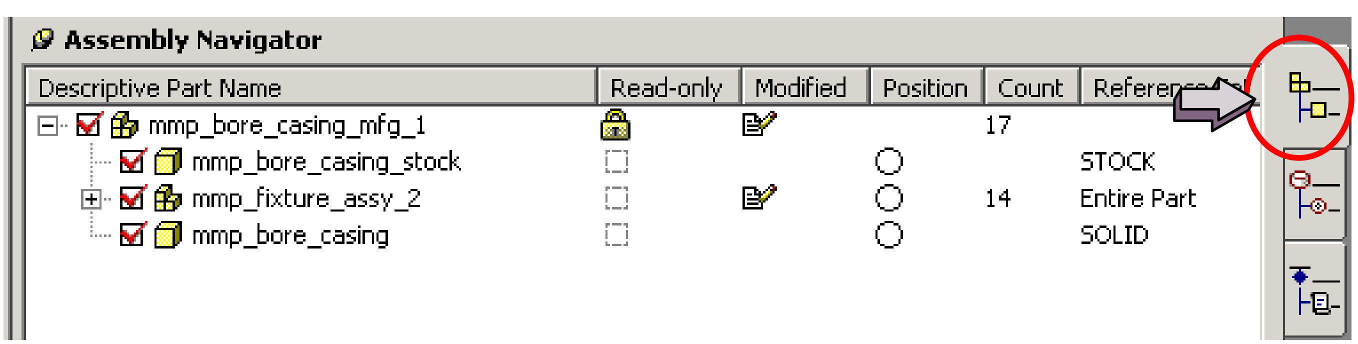
Step 1 Open the part file.

❑ Open the part file **mmp_bore_casing_mfg_1.prt.**

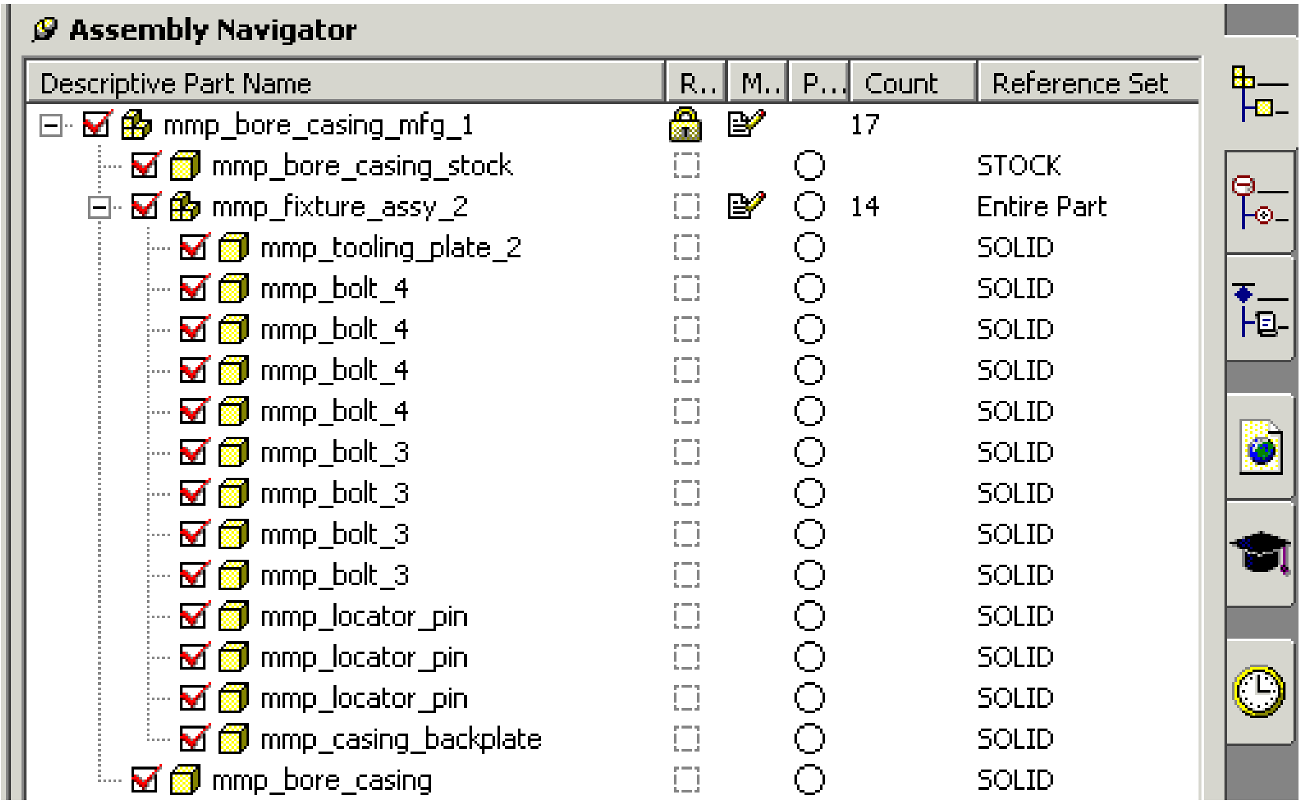


Step 2 Examine the model with the Assembly Navigator.

❑ If necessary, activate the Assembly Navigator by selecting the Assembly Navigator tab from the resource bar.



❑ **Expand** the Parent Groups in the Assembly Navigator.



❑ In the Assembly Navigator, highlight the different component names and notice that the component is displayed (in the system color) in the graphics window.

Notice the various translucencies associated with the components. Different degrees of translucencies assists in distinguishing components in an assembly when the model is shaded.

❑ In the Assembly Navigator, un-check one of the boxes containing the check mark.

Notice in the Assembly Navigator the component that you checked is blanked.

❑ Choose **File→Close→All Parts.**

This completes this activity.

You have just navigated through a Manufacturing assembly. You noted how various degrees of translucencies and shading aids in distinguishing different components in the assembly. You were also able to select component names in the Assembly Navigator and see the corresponding component displayed in the graphics window.

In the next activity, you will create a Manufacturing Assembly.

Activity 4—2: Creating the Manufacturing Assembly

In this activity you will create an assembly that contains the Master Model, the stocked part, and the fixture assembly. Due to time constraints of this class, the fixture assembly will include all but four components.

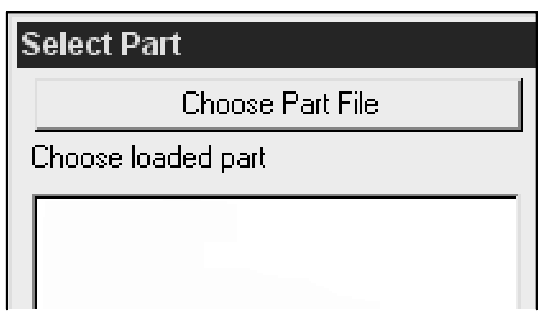
Step 1 Open and rename the part file.

- ❑ Open the part file **mmp_seedpart_in.prt.**
- ❑ Save the part as *****_bore_casing_mfg.prt** where ******* represents your initials.

Step 2 Add the part to the Manufacturing assembly file.

- ❑ Choose **Application→Assemblies** from the menu bar.
- ❑ Choose **Assemblies→Components→Add Existing** from the menu bar**.**

The Select Part dialog is displayed.

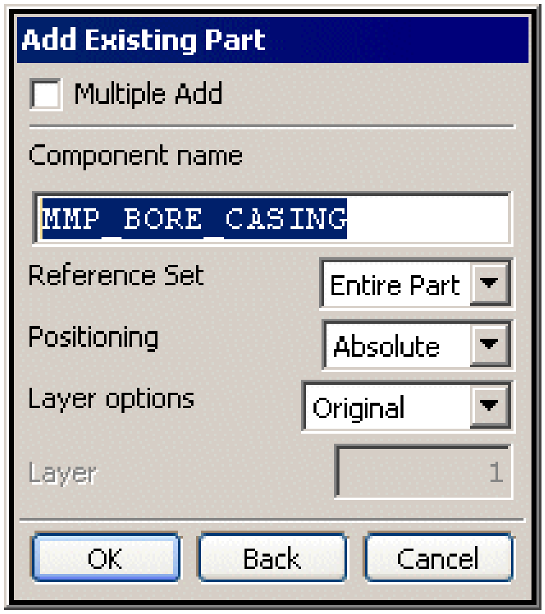


- ❑ Choose the **Choose Part File** button.

The Part Name dialog is displayed. The list includes the components that are needed to build the assembly.

- ❑ Choose **mmp_bore_casing**, then choose **OK**.

The Add Existing Part dialog is displayed.

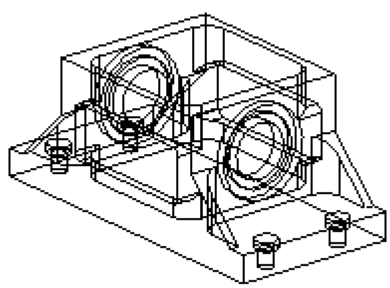


❑ Change the Layer options to **Original** and choose **OK**.

The Point Constructor dialog is displayed.

❑ Choose **Reset** in the Point Constructor, then choose **OK** to locate the part at zero.

❑ Replace the view with the **TFR-TRI** view.



Step 3 Adding the Blank to the Manufacturing assembly file.

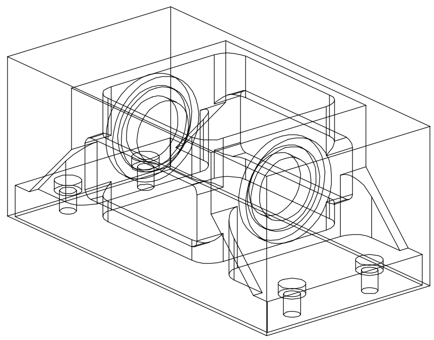
❑ Choose the **Choose Part File** button.

❑ Choose **mmp_bore_casing_stock,** then choose **OK**.

- ❑ Check that the Layer is set to **Original** and then choose **OK**.

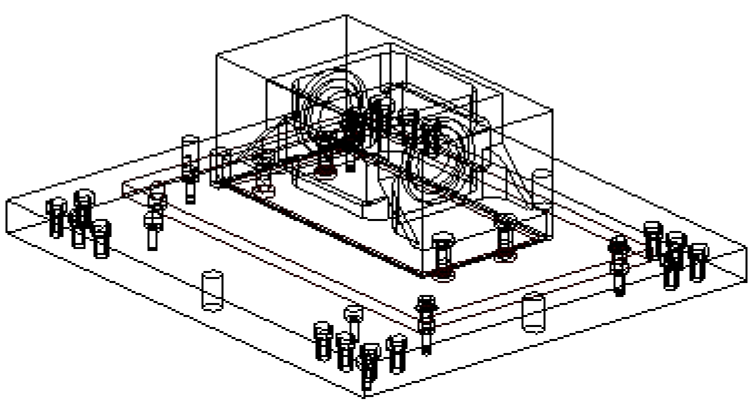
  The Point Constructor dialog is displayed.

- ❑ Choose **Reset** in the Point Constructor dialog, then choose **OK** to locate the Blank material at zero.



Step 4 Add the fixture assembly, less four component objects, to the Manufacturing assembly file.

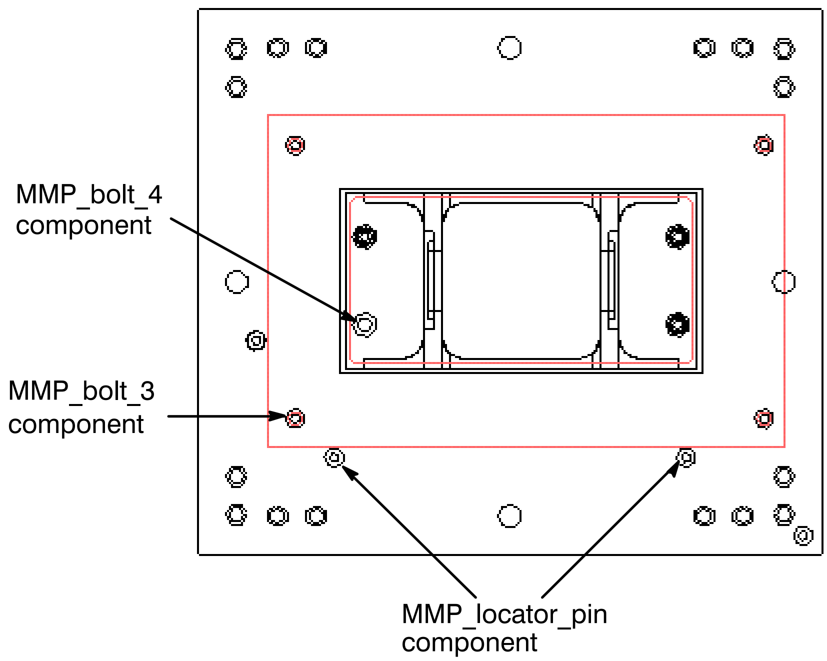
- ❑ Choose the **Choose Part File** button.
- ❑ Choose **mmp_fixture_asmb,** then choose **OK**.
- ❑ Check that the Layer is set to **Original** and then choose **OK**.
- ❑ Choose **Reset** in the Point Constructor dialog, then choose **OK** to locate the fixture assembly at zero.



- ❑ Choose **Cancel** to dismiss the dialog.

Step 5 Add component objects to the mmp_fixture_asmb part in the Manufacturing assembly file.

The fixture plate was added to the assembly less two locator pins (mmp_locator_pin), one bolt (mmp_bolt_4) that attaches the part stock to the casing backplate, and one bolt (mmp_bolt_3) that attaches the casing backplate to the tooling plate. You will now add these four components to the Manufacturing assembly.



- ❑ Choose **Assemblies→Context Control→Set Work Part**.
- ❑ Choose **mmp_fixture_asmb.**
- ❑ Choose **OK.**
- ❑ Choose **Assemblies→Components→Add Existing.**
- ❑ Choose **mmp_locator_pin** from the Choose loaded part list and then choose **OK**.
- ❑ Check that the Layer options is set to **Original** and then choose **OK**.

❑ Choose **Reset** in the Point Constructor dialog, then enter:

- XC = **-.5625**
- YC = **-4.5625**
- ZC = **-1.656**

❑ Choose **OK** to locate the 1st locator pin position.

You will now place the 2nd locator pin.

❑ Choose **mmp_locator_pin** from the Choose loaded part list and then choose **OK**.

❑ Check that the Layer options is set to **Original** and then choose **OK**.

❑ Choose **Reset** in the Point Constructor dialog, then enter:

- XC = **17.4375**
- YC = **-4.5625**
- ZC = **-1.656**

❑ Choose **OK** to locate the 2nd locator pin position.

You will now place the bolt that attaches the casing backplate to the tooling plate.

❑ Choose **mmp_bolt_3,** from the Choose loaded part list, followed by **OK.**

❑ Check that the Layer options is set to **Original** and then choose **OK**.

❑ Press **Reset** in the Point Constructor dialog, enter:

- XC = **-2.5625**
- YC = **-2.5625**
- ZC = **-2.750**

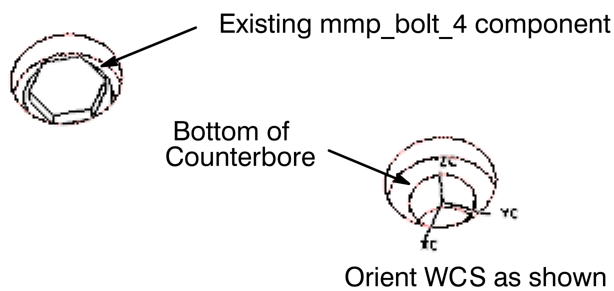
❑ Choose **OK.**

4

The bolt that attaches the casing backplate to the tooling plate is now placed in the proper position.

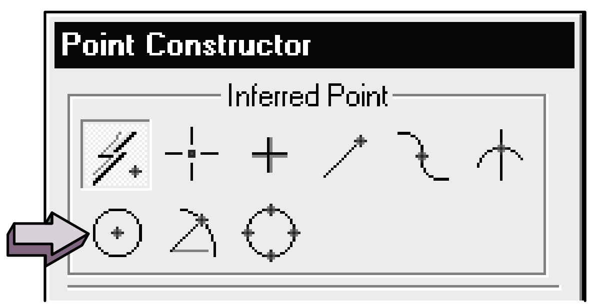
The next bolt for placement, **mmp_bolt_4**, attaches the part stock to the casing backplate. Three of these bolts have already been placed. Since the bolt attaches from the backside of the plate, you will deviate slightly from the way in which you have inserted the previous components.

One method of placement would be to locate the WCS to the backside of the plate, at the bottom of the counterbore, with ZC pointing away from the bottom of the counterbore. You will use this method.



❑ Choose **WCS→Origin** from the menu bar.

❑ Choose the **Arc/Ellipse/Sphere Center** icon.



❑ Select the bottom of the counterbore hole that **does not** contain the **mmp_bolt_4** component as previously shown.

❑ If necessary, orient the WCS with ZC pointing away from the bottom of the counterbore as previously shown.

❑ From the Choose loaded part list, choose **mmp_bolt_4** then choose **OK.**

❑ Check that the Layer options is set to Original and then choose **OK.**

❑ Choose the **Arc/Ellipse/Sphere Center** icon.

❑ Select the bottom of the counterbore (same hole and method as selected for locating the WCS).

❑ Choose **OK.**

The mmp_bolt_4 component is placed in the correct location with the proper orientation.

❑ Choose **Cancel.**

Step 6 Change the Work part in the assembly.

You are going to change the work part back to the top level of the assembly.

❑ Choose **Assemblies→Context Control→Set Work Part**.

❑ Choose *****_bore_casing_mfg.prt.**

The work part is now changed to your original part.

❑ Choose **File→Save.**

❑ **Close** the part file.

You have completed this activity and the lesson.

SUMMARY

Assembly modeling for manufacturing offers numerous benefits for the manufacturing community. With the creation of a manufacturing assembly, specific data can be generated that references master geometry. This simplifies the process of fixture creation, process stock creation, part and stock clamping, and tool path generation.

In this lesson you:

- Reviewed Assembly Modeling and the Master Model concept
- Explored an existing Manufacturing Assembly utilizing the Master Model concept
- Created a Manufacturing Assembly

Coordinate Systems

Lesson 5

PURPOSE

This lesson will explain the different coordinate systems that are used in the Manufacturing application, including the purpose and function of each one.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Understand the use and functionality of the various coordinate systems
- Create and move the Machine Coordinate System (MCS)

This lesson contains the following activities:

Coordinate Systems

A coordinate system can be viewed as a marker in 3-D model space to which other objects are referenced.

You can create a point that is X 2.0, Y 3.0, Z 1.0. That point can be placed with respect to the Absolute and Work Coordinate System. In the Manufacturing application you will be introduced to an additional coordinate system referred to as the Machine Coordinate System or MCS.

There are five coordinate systems that Manufacturing uses. They are the:

- Absolute Coordinate System
- Work Coordinate System
- Machine Coordinate System
- Reference Coordinate System
- Saved Coordinate System

Absolute Coordinate System

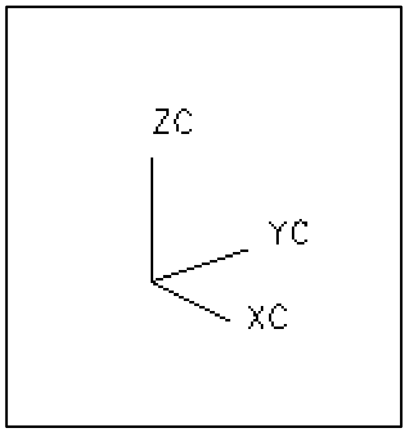
The Absolute Coordinate System is fixed in model space, and cannot be moved. This coordinate system is invisible to the user.

It is most useful as a reference for large assemblies. Frequently, users will design large scale machines composed of many different components. It is easy to find their relative position to being absolute since the components are located in reference to one another.

Work Coordinate System

The Work Coordinate System is abbreviated WCS. It has several modeling and some manufacturing functions as well. The WCS is a moveable coordinate system, which increases its usefulness. Not only can the WCS be moved in model space, but its orientation can change as well.

The WCS is a visible coordinate system, and it looks like the following:

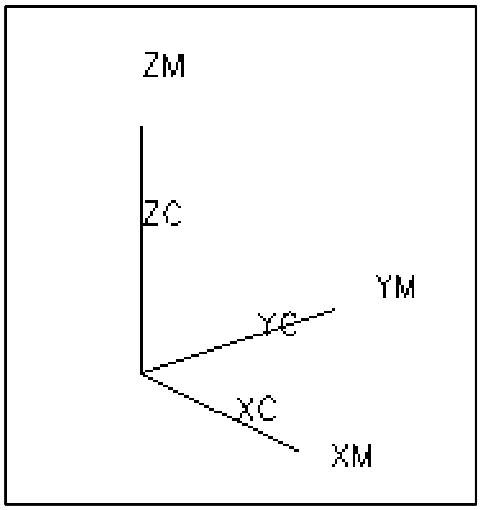


Notice that each leg of the WCS is followed by a “C”. That is a visual indication that it is the WCS. The WCS is used in creating geometry. Most important to manufacturing users, it is also used when establishing I,J,K vectors. I,J,K vectors are used to establish the tool axis, cut direction and geometric entities.

Machine Coordinate System

The Machine Coordinate System is abbreviated MCS. The MCS, like the WCS, is also a moveable coordinate system.

Notice that the MCS has an M at the end of each leg to distinguish it from the WCS. Also, the legs of the MCS are longer than the WCS. Below, both the WCS and MCS are shown together, to emphasize this difference.



The MCS references the program origin or simply 0,0,0, since the location of the MCS is the zero point of all tool path output. If the MCS moves, so does the zero point of all tool paths that use it.

Another important function of the MCS is contained in the Z-axis. The default tool axis is the same as the Z-axis of the MCS.

It is important to remember that if a tool axis is established using I,J,K vectors, that tool axis is actually based on the orientation of the WCS. If the default tool axis is used, it is based upon the orientation of the MCS.

The MCS is only visible when in the Manufacturing application

.

Reference Coordinate System

Another coordinate system that has some limited functionality is the Reference Coordinate System. It is abbreviated RCS.

Unigraphics NX stores every parameter set with a copy of the coordinate system that was active at the time it was created. This is called the RCS. This eliminates respecification of such parameters as engage vectors, clearance planes, etc., by allowing the retrieval and mapping of stored parameters. During subsequent operations, you can retrieve these parameters and reposition them to the current (moved) RCS.

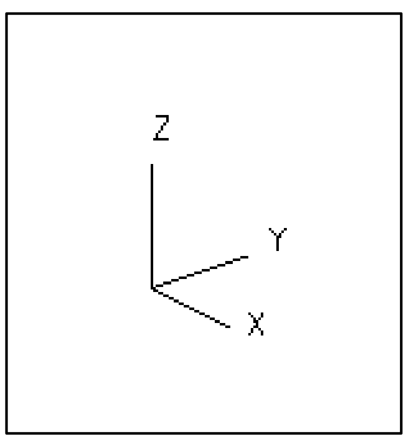
The default RCS is the Absolute Coordinate System.

Saved Coordinate System

An additional coordinate system that you sometimes refer to is known as the Saved Coordinate System. The Saved Coordinate System is not really a coordinate system at all. Rather, it is a marker, or place holder, to designate a position in model space. You can move the WCS or the MCS to this saved location.

The Saved Coordinate System is a modeling object and resides on a layer just like other modeling objects.

This is a Saved Coordinate System:



Summary

Absolute Coordinate System

- Fixed in model space
- Cannot be seen
- User may save coordinate system to mark location
- Useful as a reference in large assemblies

Work Coordinate System

- Referred to as the WCS
- Displayed in graphics area with **C** after each leg
- Moveable and changeable
- Used heavily in modeling
- I,J,K Vectors are based on orientation of WCS

Machine Coordinate System

- Referred to as the MCS
- Displayed in graphics area with **M** after each leg
- Moveable and changeable
- Zero (0,0,0) location for tool path output
- Default tool axis, Z-axis is referenced by this coordinate system

Reference Coordinate System

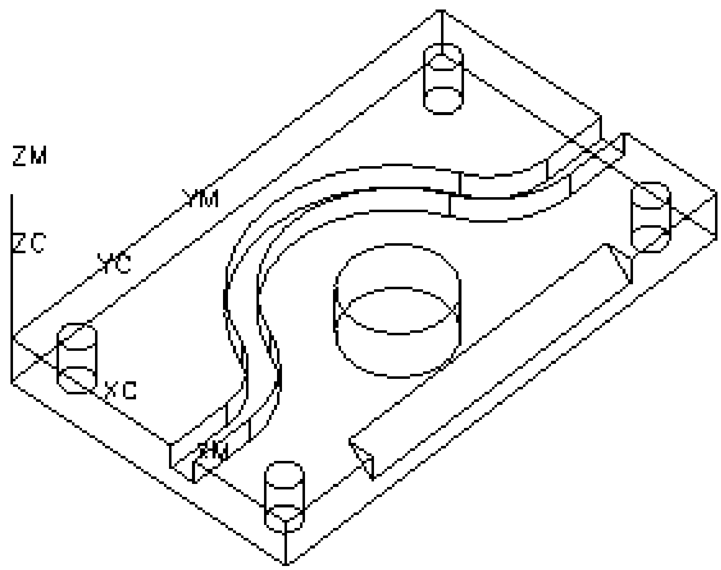
- Referred to as the RCS
- Limited functionality
- Eliminates re-specification of parameters by allowing the retrieval and mapping of stored parameters

Activity 5—1: Changing the MCS Position

The CNC/NC programmer who machined this part created the program with the Machine Coordinate System (MCS) in the wrong location. You will correct that mistake and then review the output.

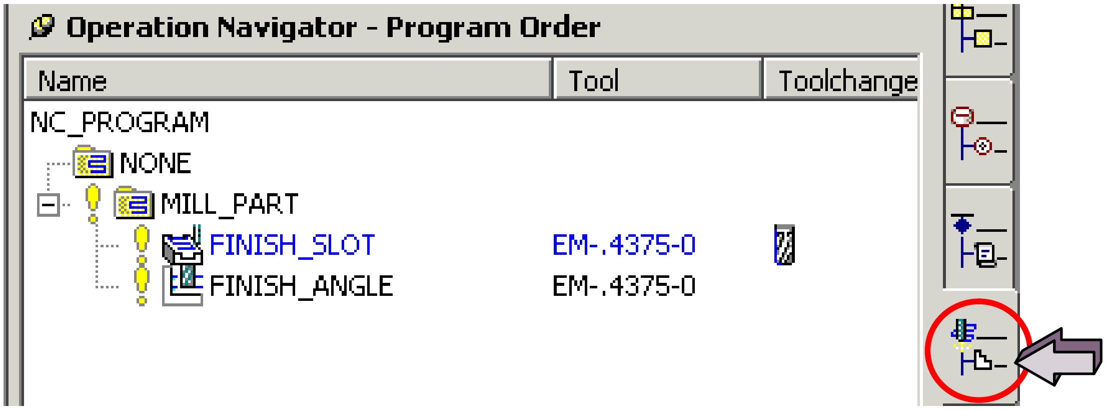
Step 1 Open the Part File and enter the Manufacturing application.

❑ Open the part file **mmp_half_channel_mfg_1.prt.**



❑ Choose **Application→Manufacturing**.

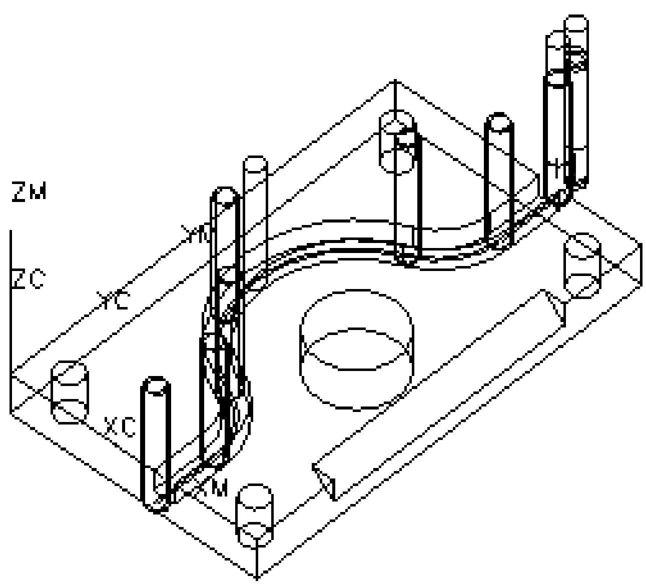
❑ Choose the Operation Navigator tab from the resource bar and expand the **MILL_PART** parent group.



Step 2 Replay the FINISH_SLOT operation.

❑ If necessary, change to the **Program Order View**.

❑ Highlight the **FINISH_SLOT** operation. use **MB3** and choose Replay.



The tool is slightly narrower than the slot, so two passes are necessary. Both the WCS and the MCS are located at the corner of the part. This is incorrect. The program 0,0,0 should be at the center of the large hole.

- ❑ Highlight the **FINISH_SLOT** operation, using **MB3** choose **Toolpath→List**.

```
TOOL PATH/FINISH_SLOT,TOOL,EM-.4375-0
TLDATA/MILL,0.4375,0.0000,3.0000,0.0000,0.0000
MSYS/0.0000,0.0000,0.0000,1.0000000,0.0000000,
PAINT/PATH
PAINT/SPEED,10
PAINT/TOOL,FULL,1
PAINT/COLOR,186
RAPID
GOTO/3.7187,10.7188,0.5000,0.0000000,0.0000000
RAPID
GOTO/3.7187,10.7188,-0.3000
PAINT/COLOR,211
RAPID
GOTO/3.7187,10.7188,-0.4000
```

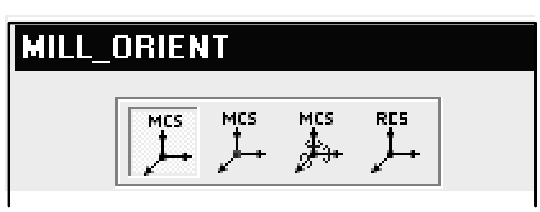
All of the “X” values (the first position numeric field in the GOTO statements) are all positive. That is because of the position of the MCS.

Step 3 Moving the MCS.

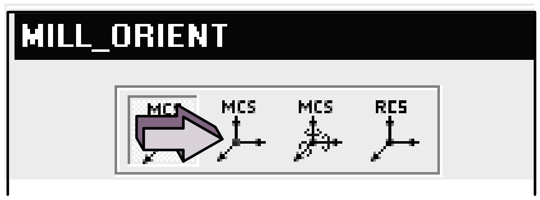
The MCS must be moved to change the program 0,0,0 point.

- ❑ Change to the **Geometry View** of the Operation Navigator.
- ❑ Double-click on the **MCS_MILL** Parent Group object.

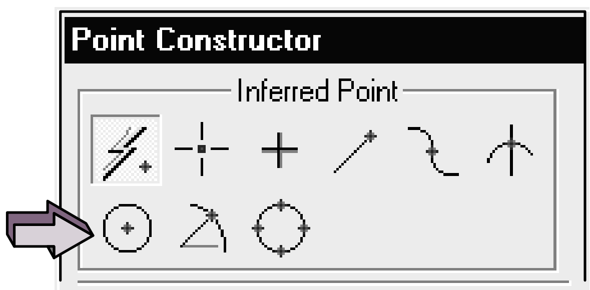
At the top of the dialog are four button. These buttons are used to change the orientation and location of the MCS.



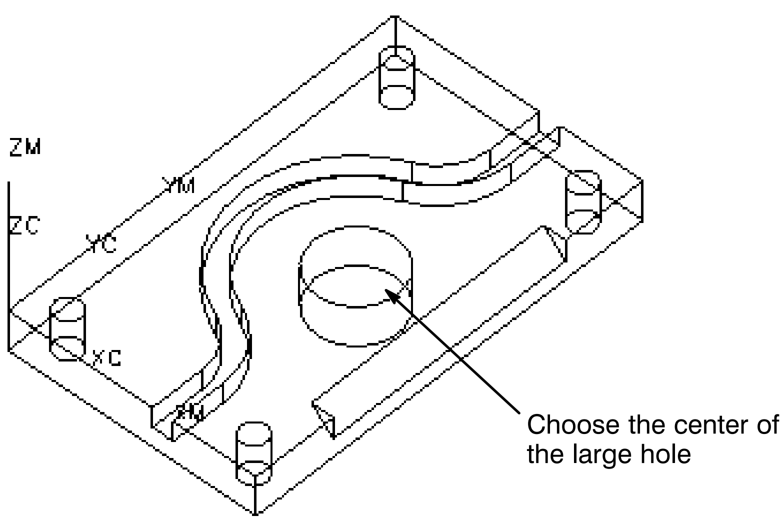
❑ Choose the **MCS Origin** button.



❑ At the top of the Point Constructor dialog, choose the **Arc/Ellipse/Sphere Center** button.



❑ Select the center of the large hole, as shown.



Note the new location of the MCS.

❑ Choose **OK**.

Step 4 List the tool path.

- ❑ Change to the **Program Order View**.

- ❑ Highlight the **FINISH_SLOT** operation, using **MB3** choose **Toolpath→List**.

```
TOOL PATH/FINISH_SLOT,TOOL,EM-.4375-0
TLDATA/MILL,0.4375,0.0000,3.0000,0.0000,0.0000
MSYS/3.7500,5.0000,0.0000,1.0000000,0.0000000,
PAINT/PATH
PAINT/SPEED,10
PAINT/TOOL,FULL,1
PAINT/COLOR,186
RAPID
GOTO/-0.0312,5.7188,0.5000,0.0000000,0.0000000
RAPID
GOTO/-0.0312,5.7188,-0.3000
PAINT/COLOR,211
RAPID
GOTO/-0.0312,5.7188,-0.4000
PAINT/COLOR,6
FEDRAT/IPM,37.7100
GOTO/-0.0312,5.7188,-0.5000
GOTO/-0.0312,5.2088,-0.5000
```

Notice the difference in the X and Y values. All locations are based on the origin of the MCS.

NOTE Note that it is not necessary to generate the operation to see the listing change. The listing file is created dynamically each time **Toolpath→List** is chosen.

NOTE Also note that it is not necessary for the WCS and the MCS to be at the same location or orientation.

- ❑ Do not close the part file, you will use it again in the next activity.

Additional Coordinate System Information

There is some additional information about coordinate systems that every user should know.

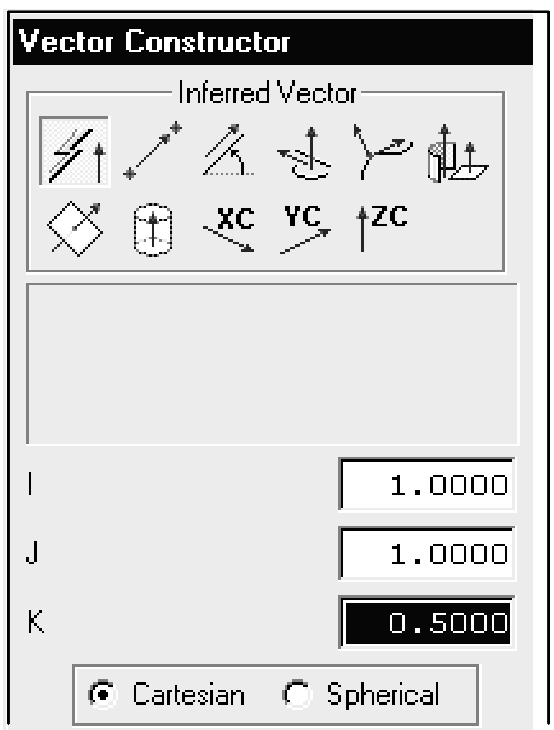
I,J,K Vectors

A vector can be thought of as a line between two points, has magnitude and direction. The first point of the vector is always assumed to be 0,0,0. The second point is the one you describe. A temporary line between these two points is created to establish a vector.

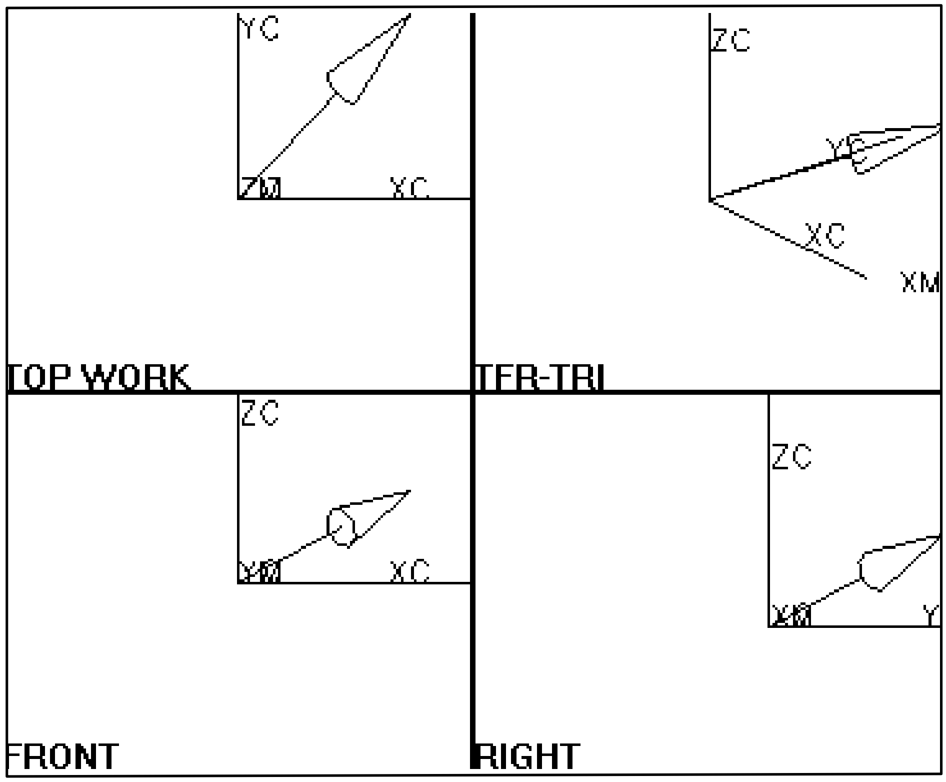
The letters I,J,K follow each other in the alphabet and correspond (parallel) to the X,Y,Z axes of the Work Coordinate System.

- I – relates to – X
- J – relates to – Y
- K – relates to – Z

When a value is given for each axis, the second point is computed, and thus the vector is created. The following is an example of the Vector Constructor dialog showing a vector of 1, 1, 0.5:



Below is a four-view layout showing the vector:



Rotary Vectors

When programming, it is not uncommon for some machine tools to support 4 and 5-axis work. There are rules for naming the rotary axes on machine tools. Usually, these rotary axes are given the letter designation A, B, or C.

The letter A designation usually designates a rotary axis that rotates about the linear X. To determine positive rotation, form your hand into a thumb's-up gesture (right hand rule), and point your thumb in the positive X direction. Your curled fingers show positive A axis rotation.

The letter B designation usually designates a rotary axis that rotates about the linear Y. To determine positive rotation, form your hand into a thumb's-up gesture and point your thumb in the positive Y direction. Your curled fingers show positive B axis rotation.

The letter C designation usually designates a rotary axis that rotates about the linear Z. To determine positive rotation, form your hand into a thumb's-up gesture and point your thumb in the positive Z direction. Your curled fingers show positive C axis rotation.

Tool Axis versus ZC Axis

It is commonly assumed that 3-axis machining can only be performed with the tool axis at the same orientation as the machine tool's Z-axis. This assumption is incorrect. On 4-axis machining centers, it is possible to perform 3-axis milling with the rotary axis at a different orientation than the MCS Z-axis.

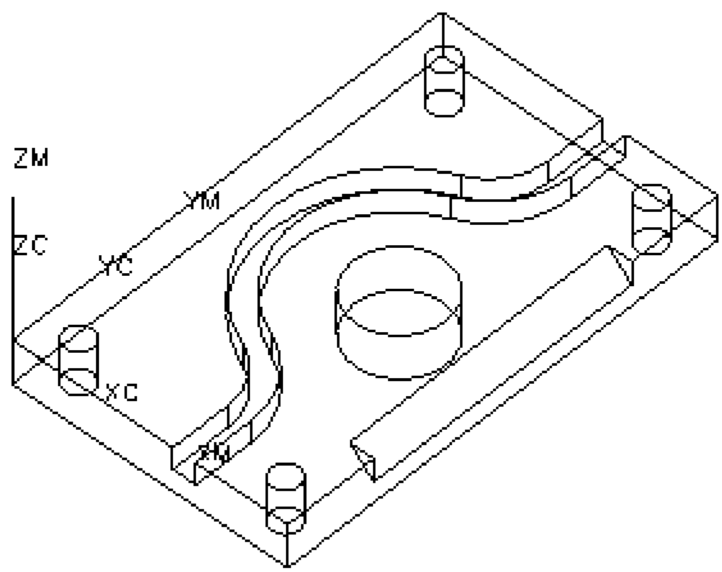
The next activity will demonstrate a simple way to perform 3-axis milling with the tool axis at a different angle than the MCS Z-axis.

Activity 5–2: Changing the Tool Axis

The previous programmer made another mistake. He machined the angled tab without changing the orientation of the tool. Remember, the system default for tool axis is to make it the same as the MCS Z axis. That must be changed for this operation to work correctly.

Step 1 Open the part file and enter the Manufacturing application.

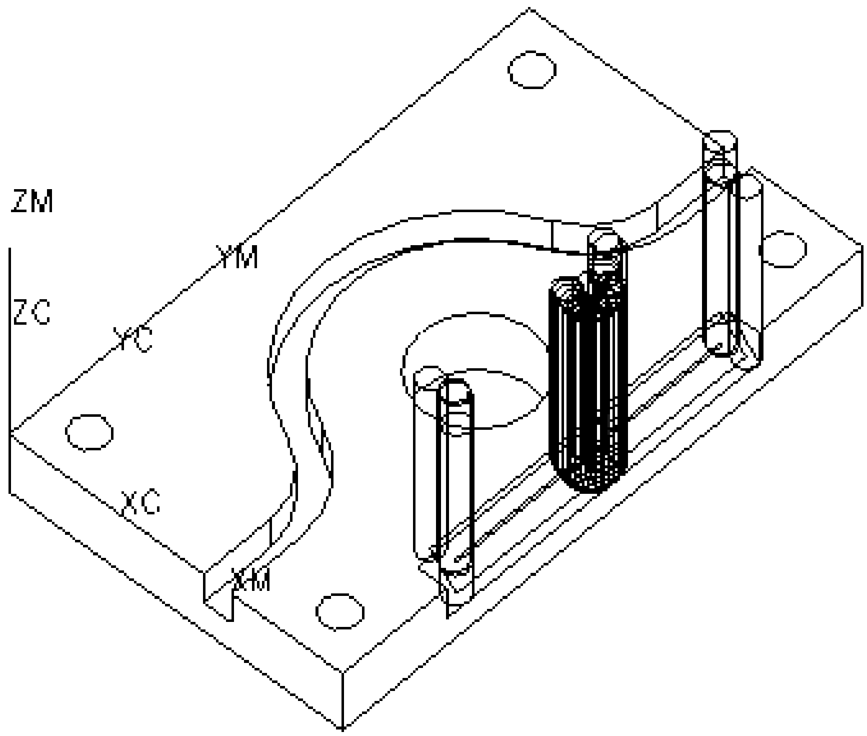
❑ Continue using the part file **mmp_half_channel_mfg_1.prt**



❑ Change to the **Program Order View** in the Operation Navigator.

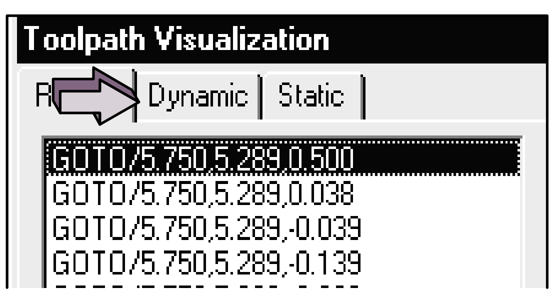
Step 2 Replay the current operation.

- ❑ If necessary, expand the **MILL_PART** Parent Group.
- ❑ Highlight the **FINISH_ANGLE** operation, using **MB3**, choose **Replay**.



The tool axis is the same as the Z-axis of the MCS. It results in an incomplete machined surface.

- ❑ Highlight the **FINISH_ANGLE** operation, using **MB3**, choose **Toolpath→Verify**.



- ❑ Choose the **Dynamic** tab and then the Play Forward button.

This is not the desired result. The machine tool has the capability to tilt the head in the B-axis.

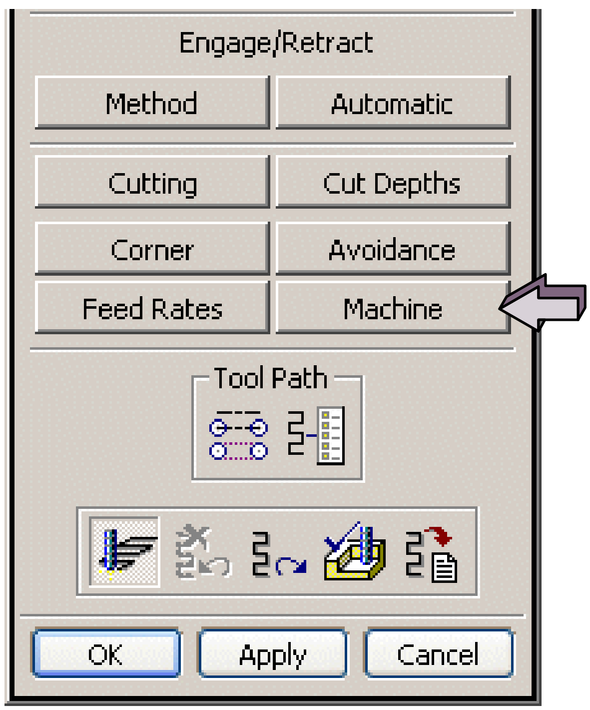
❑ Choose **Cancel** to dismiss the Toolpath Visualization dialog.

You will now edit the operation to respecify the tool axis.

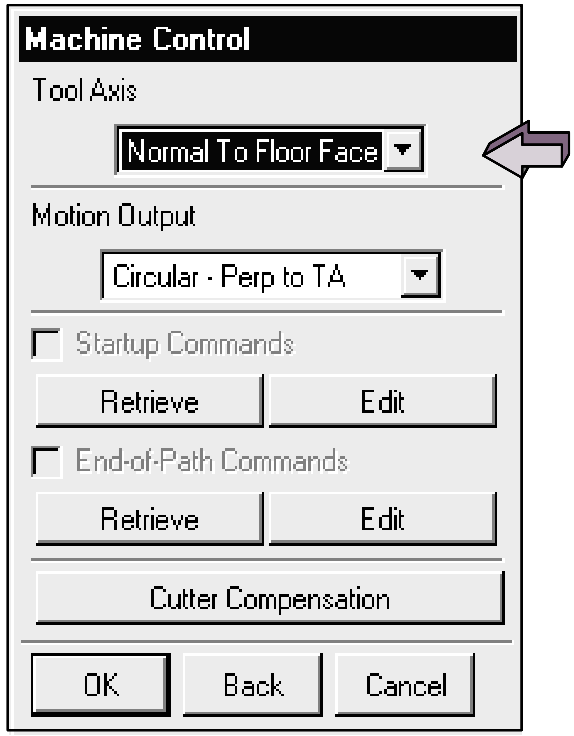
❑ Double-click on the **FINISH_ANGLE** operation.

This is a Planar Milling operation. The tool axis function is located under the **Machine** button.

❑ Choose the **Machine** button.



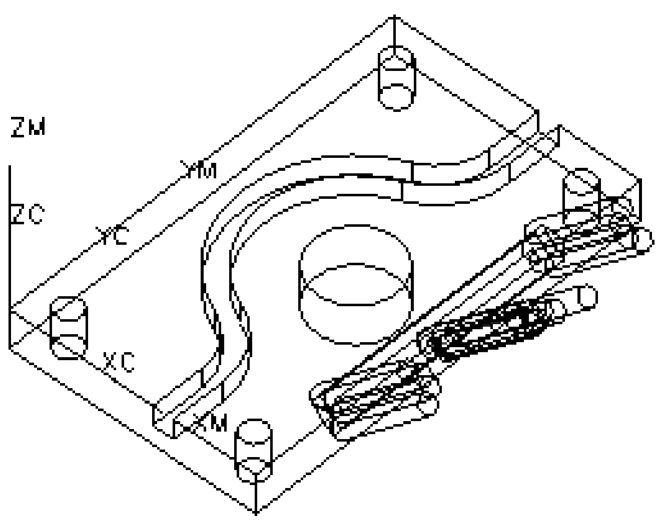
❑ In the Machine Control dialog, change the **Tool Axis** to **Normal To Floor Face**.



The floor face is the angled face at the bottom of the cut out area. The tool axis is now normal to the face.

❑ Choose **OK** to return to the Planar Milling dialog.

❑ Choose **Generate**.



The tool path is generated with the tool axis at the new angle.

- ❑ After viewing the results, choose **OK** to accept the tool path.
- ❑ **Close** the part file without saving.

This concludes the activity and the lesson.

SUMMARY

The ease of manipulation of the Machine Coordinate System affords the flexibility of performing various types of machining operations. The following functionality is used in defining coordinate systems and vectors that control machine tool output:

- When defining vectors, the values are entered with respect to the WCS
- Machine tool output coordinate output is controlled by the MCS
- The default tool axis is the same as the MCS Z-axis
- The tool axis is not required to have the same orientation as the MCS Z-axis

Smart Objects

Lesson 6

PURPOSE

Smart Objects are objects such as Clearance Planes and From Points that are linked to geometry. This lesson will teach you how to create and use Smart Objects.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Use the capabilities of Smart Objects
- Use a Smart Object when creating a Clearance Plane

This lesson contains the following activities:

Smart Objects

6

A Smart Object allows objects such as the MCS, RCS, Clearance Plane, From, Start or GOHOME points to be linked to the geometry that was used to define them. When the linked geometry is modified, the relationship between the Smart Object and the geometry is maintained.

The list of Smart Objects is not limited to the items mentioned above. The following table defines the type of Smart Objects that you can create and the operation types in which they are available:

All Operations	Drill	Planar Mill	Cavity Mill	Fixed Contour
Fixed Tool Axis Vector	Projection Vector	Initial Engage	Initial Engage	Projection Vectors
From Point	Material Side Vector	Final Retract	Final Retract	Non-Cutting: Direction
Start Point	Part Surface	Predrilled Engage Points	Predrilled Engage Points	
Return Point	Bottom Surface	Cut Region Start Points	Cut Region Start Points	
GOHOME Point		Floor Plane	Cut Levels	
Clearance Plane		Boundary Plane		
Lower Limit				
GOHOME point				
MCS				
RCS				

To create Smart Objects, you use the Point, Plane, Vector and CSYS *Constructor* dialogs.

Clearance Planes

A **Clearance Plane** defines a safe clearance distance (usually above the part) for the tool to move before and after cutting material or used for safe clearance for repositioning.

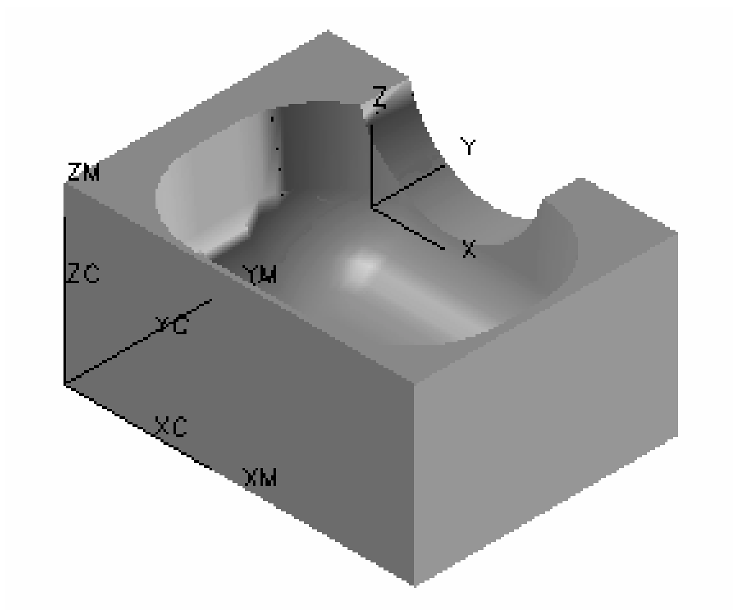
```
TOOL PATH/P1
TLDATA/MILL,0.7500,0.0600,4.0000,0.0000,0.0000
MSYS/0.0000000,0.0000000,3.0000000,1.0000000,0.00
00000,0.0000000,0.0000000,1.0000000,0.0000000
PAINT/PATH
FROM/4.6877,−0.7657,0.3000
RAPID
GOTO/4.9728,0.9943,0.2000 → (Start Point)
RAPID
GOTO/4.9728,0.9943,0.1000 → (Clearance Plane)
RAPID
GOTO/1.4043,2.8449,−2.7000 → (Engage Move)

GOTO/2.0936,3.8000,−2.8000
RAPID
GOTO/2.0936,3.8000,0.1000 → (Clearance Plane)
RAPID
GOTO/3.8879,1.5375,0.1000 → (Move At Clearance
Plane)
RAPID
GOTO/3.8879,1.5375,−2.7000

GOTO/5.9447,0.5927,−2.7500
RAPID
GOTO/5.9447,0.5927,0.1000 → (Clearance Plane)
RAPID
GOTO/5.0072,0.4101,0.2000 → (Return Point)
GOHOME/4.6877,−0.7657,0.3000 → (GOHOME)
END−OF−PATH
```

Activity 6–1: Creating a Smart Clearance Plane

In this activity, you will define a Clearance Plane in the MCS_MILL Parent Group, edit the part and review the results. You will then respecify the Clearance Plane, edit the part geometry and examine the results.

Step 1 Open and rename the Part file.

❑ Choose **File→Open** then choose **mmp_die_insert_2.prt**.



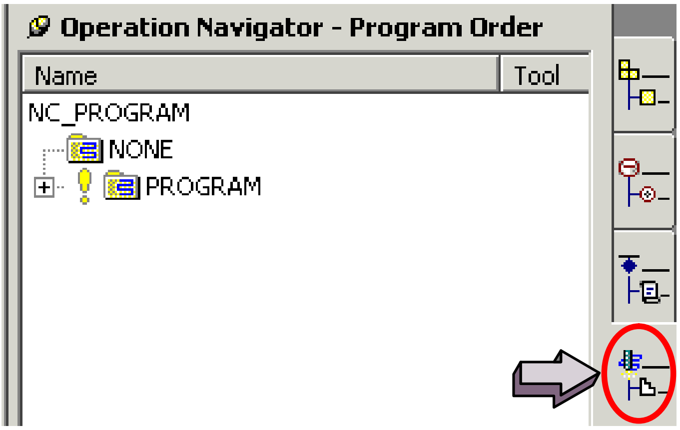
❑ Rename the part *****_die_insert_2.prt** using the **File→Save As** option on the menu bar, where *** represents you initials.

Step 2 Enter the Manufacturing application and display the Operation Navigator.

❑ Choose **Application→Manufacturing.**

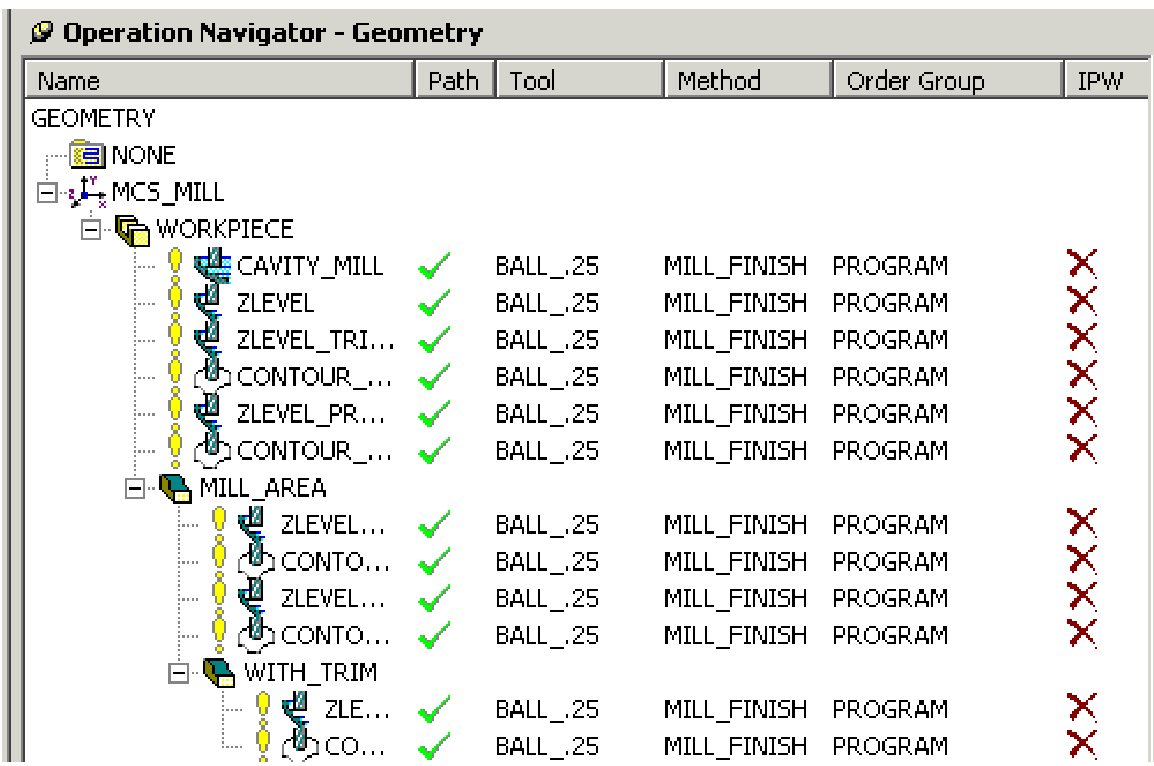
❑ Choose the Operation Navigator tab from the resource bar.

The Operation Navigator is displayed.



You can specify a Clearance Plane in the MCS_MILL Parent Group object. All of the operations placed under this parent group will inherit the Clearance Plane settings. You will change the Operation Navigator view to a view that is more conducive to working with geometry objects.

❑ Change the Operation Navigator to the **Geometry View** and expand all objects.

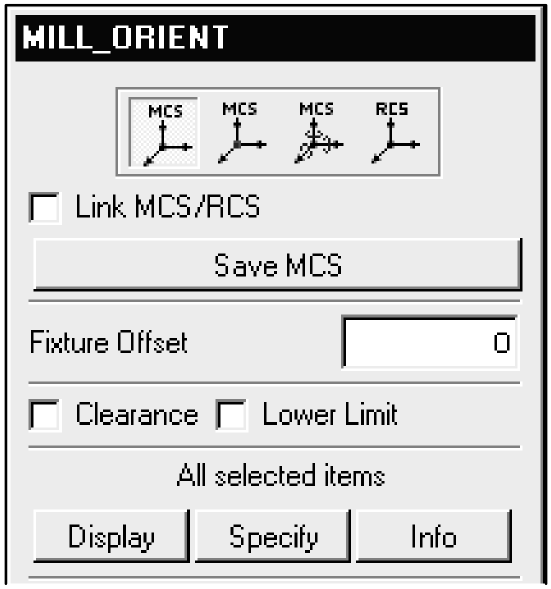


6

Step 3 Define a Clearance Plane by using the Principal Plane method.

You are going to create a general (non-smart) Clearance Plane, using the Principle Plane method.

❑ In the Operation Navigator, double click on **MCS_MILL.**

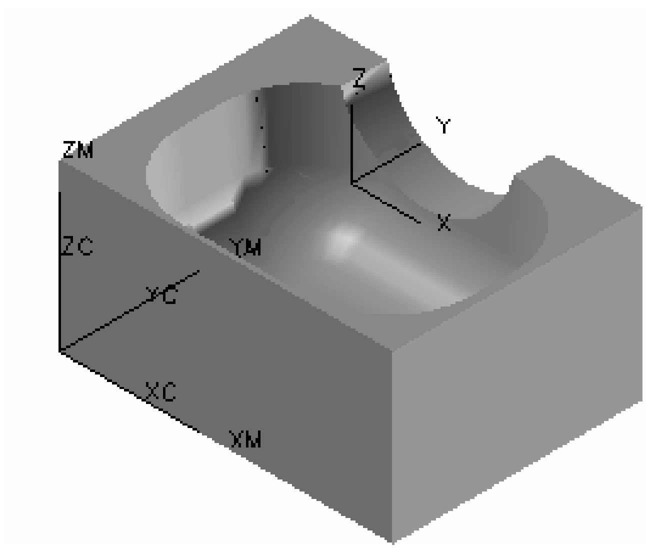


The MILL_ORIENT dialog is displayed.

Look at the MCS and WCS positions.

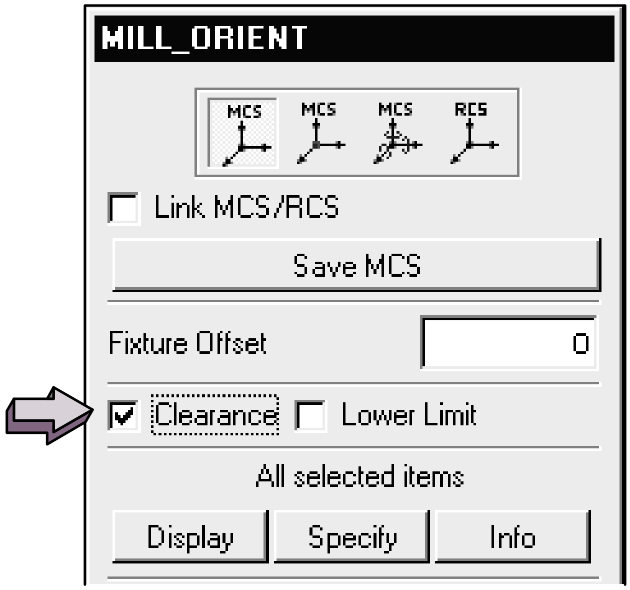
❑ Examine the position of the **MCS** and **WCS**.

Notice that they are in the same location.



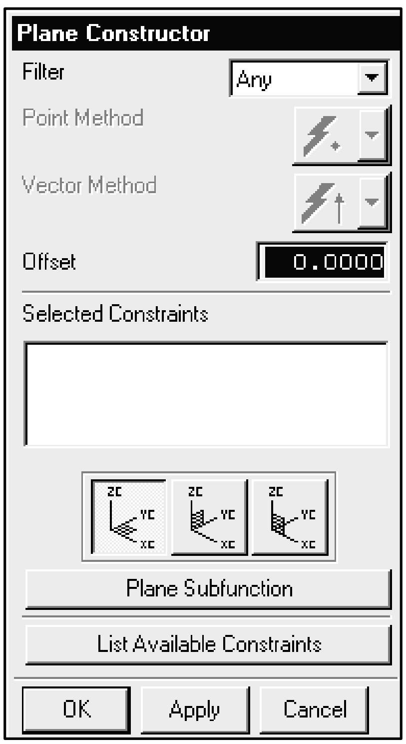
The Clearance Plane values are relative to the WCS.

❑ Choose the **Clearance** button.

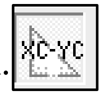


❑ Choose **Specify**.

The Plane Constructor dialog is displayed. You can define a Clearance Plane using any of options that are available.



You will use the principal plane, with respect to the XC-YC axis to define the Clearance Plane.

❑ Choose the **XC YC** icon. 

❑ Choose the **Plane Subfunction** button.

The Plane description dialog is displayed.

❑ Choose the **Principle Plane** button.

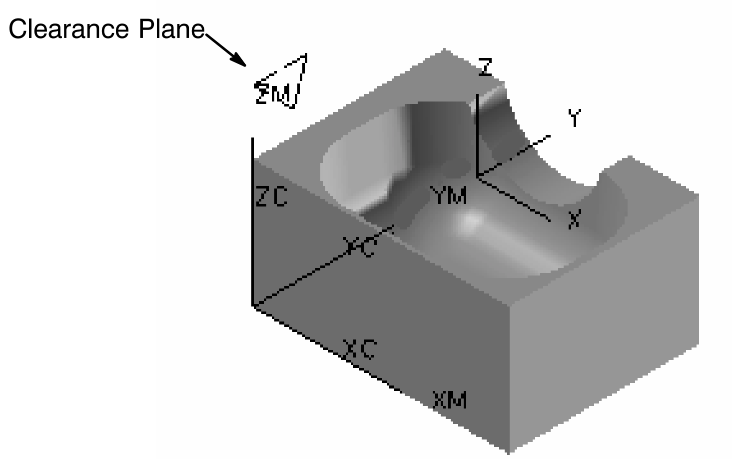
❑ Choose the **ZC Constant** button**.**

The dialog displayed allows you to enter a distance from the XC-YC plane. The XC-YC plane is located at the bottom of the part (at the WCS location). You will define the Clearance Plane 3 inches above the XC-YC plane, placing it 1 inch above the top of the part.

❑ Key in **3** in the value field.

❑ Choose **OK.**

The Clearance Plane is displayed in the graphic window.



❑ Choose **OK.**

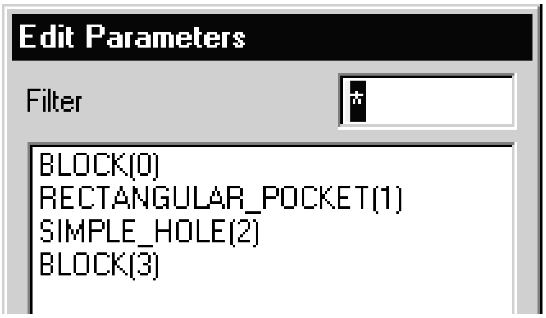
You will now increase the part height and observe what happens with the Clearance Plane.

Step 4 Changing the height of the part.

- ❑ Choose **Application→Modeling** from the menu bar.

- ❑ Choose **Edit→Feature→Parameters.**

  The Edit Parameter dialog lists the features in the part.



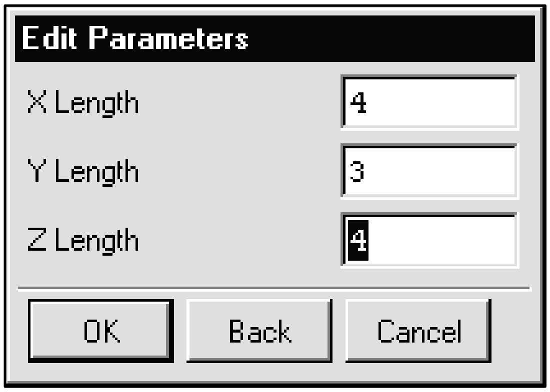
- ❑ Highlight the **Block(0)** feature and choose **OK.**

  The Edit Parameter dialog is displayed.

- ❑ Choose the **Feature Dialog** button.

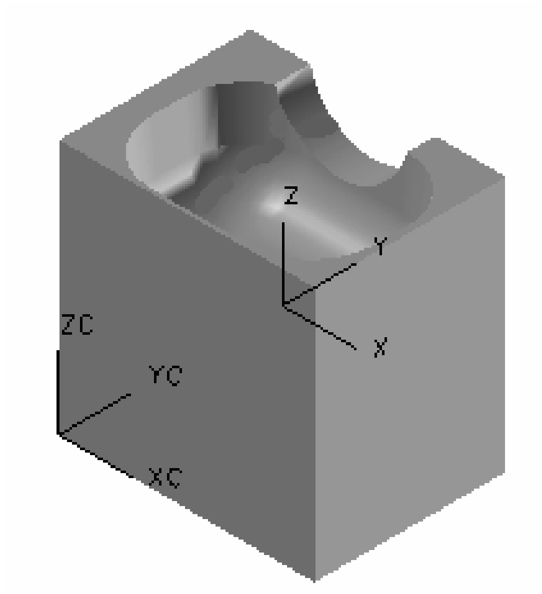
  Another Edit Parameter dialog is displayed, showing dimensions of the feature. Use this to change the dimension.

- ❑ Change the Z value to **4.**



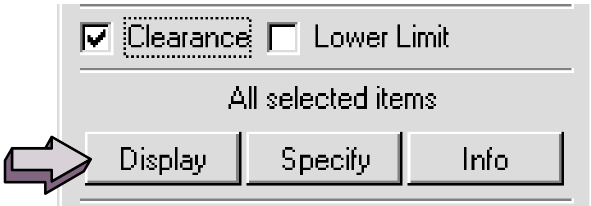
- ❑ Choose **OK** a total of three times, until the part updates with the new parameter.

The block resizes. You may need to fit the model to the graphics window. Now you will return to Manufacturing application.



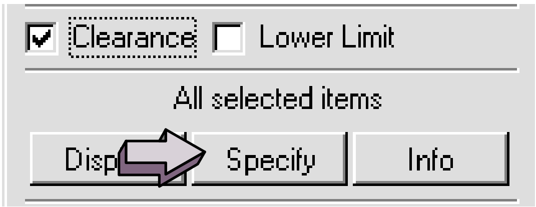
Step 5 Return to the Manufacturing application and reselect the clearance plane as a smart object.

- ❑ Choose **Application→Manufacturing** and then select the Operation Navigator icon from the resource bar**.**
- ❑ If necessary, select the Geometry View of the Operation Navigator.
- ❑ Double-click the **MCS_MILL** name in the Operation Navigator.
- ❑ Choose **Display** from the All Selected item area**.**



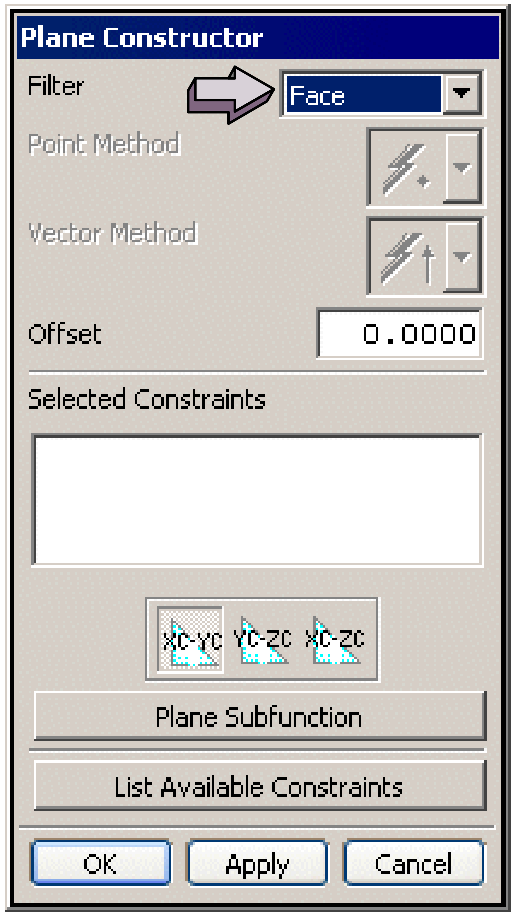
Notice that the Clearance Plane is below the part surface. Since this is a Clearance Plane that is referenced from the WCS and not the part geometry, it has no knowledge of the part height changing. You will now define the Clearance Plane with respect to the face of the part, which will allow the plane to change with any change made to the height of the part.

❑ Choose **Specify** from the MILL_ORIENT dialog**.**



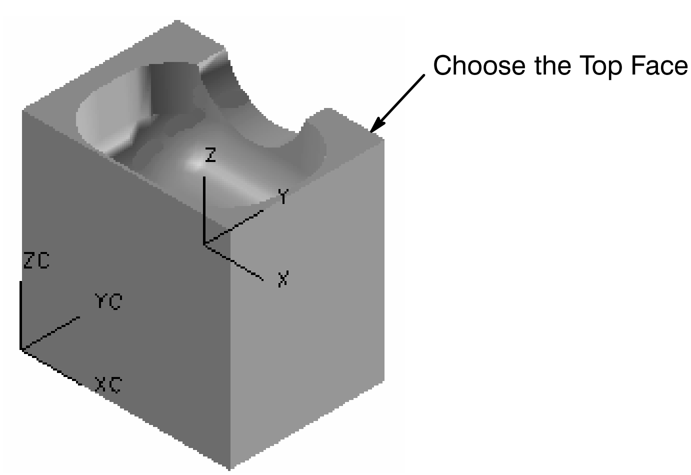
The Plane Constructor dialog is displayed.

❑ Change the filter to **Face**.



You will now offset the Clearance Plane .2 from the face that you will select.

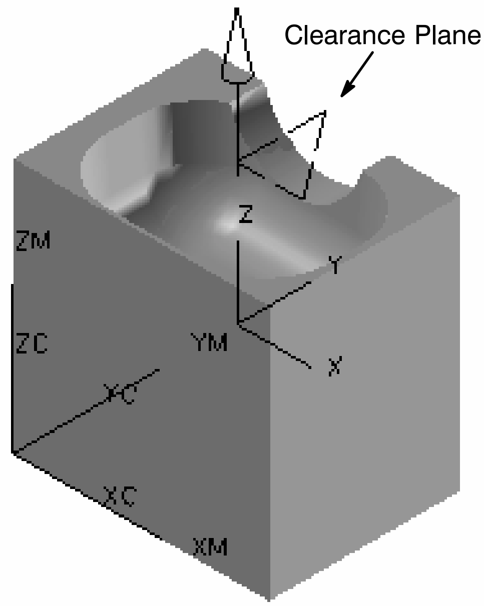
- ❑ Key in **.200** into the offset value field.
- ❑ Select the top face a shown.



A direction indicator is displayed indicating the direction of the offset for the Clearance Plane.

- ❑ Choose **OK.**

  The Clearance Plane is displayed .200 above the selected face.

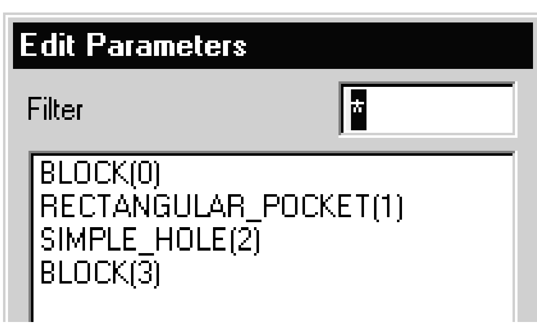


- ❑ Choose **OK.**

Step 6 Changing the overall height of the part.

- ❑ Choose **Application→Modeling** from the menu bar**.**
- ❑ Choose **Edit→Feature→Parameters.**

  The Edit Parameter dialog lists the features in the part.



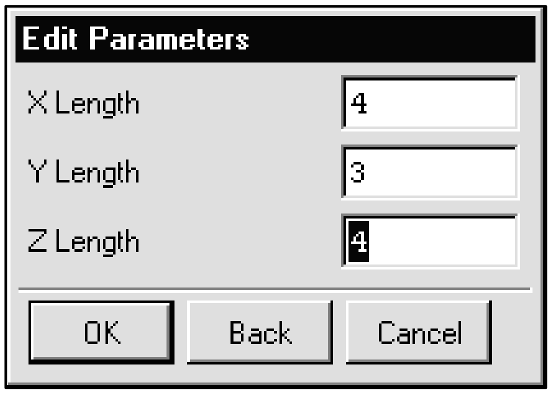
- ❑ Highlight the **Block(0)** feature and choose **OK.**

The Edit Parameter dialog is displayed.

- ❑ Choose the **Feature Dialog** button.

  Another Edit Parameter dialog is displayed, showing dimensions of the feature. Use this to change the dimension.

- ❑ Change the Z value to **2.**



- ❑ Choose **OK** a total of three times, until the part updates with the new parameter**.**

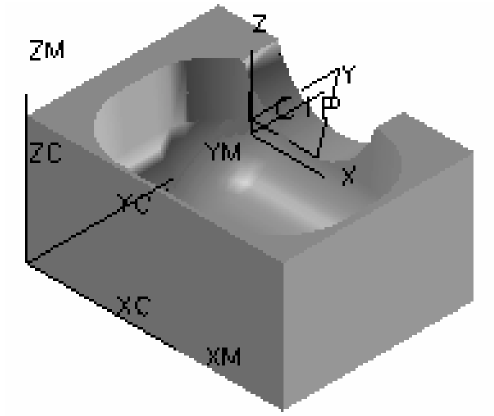
  The Block resizes.

Step 7 Examine the location of the Clearance Plane.

- ❑ Choose **Application→Manufacturing.**

- ❑ Double-click the **MCS_MILL** name in the Operation Navigator.

- ❑ Choose **Display.**



The Clearance Plane is in the correct position, .200 above the top face of the part.

❑ **Save** the part file.

SUMMARY

Smart Objects are objects that are linked to the geometry that was used to define them, so they are updated whenever the corresponding geometry is modified.

In this lesson you:

- learned the difference between Smart Objects and their counterparts
- created a Clearance Plane by conventional means that was not linked to associative geometry
- created a Smart Clearance Plane using the part geometry that is updated as the associative geometry is changed

The Machining Environment

Lesson 7

7

PURPOSE

This lesson introduces you to the concept of establishing a Machining Environment with the type of Manufacturing information that you will most likely need for a programming session.

OBJECTIVES

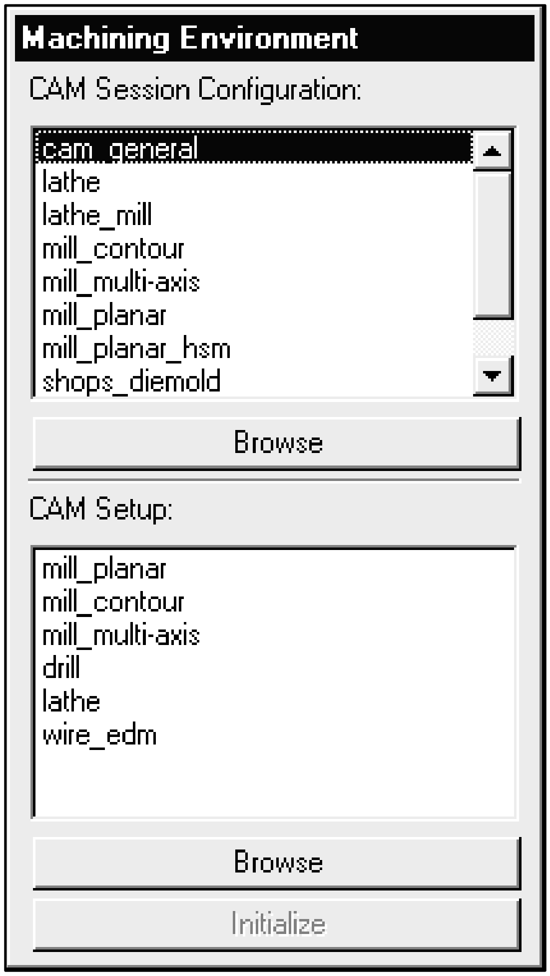
Upon completion of this lesson, you will be able to:

- Specify a Machining Environment Configuration and Setup
- Change the Machining Environment Type

This lesson contains the following activity:

The Machining Environment

The first time you choose the Manufacturing application within a part, you must choose a Machining Environment.



This is a basic choice for the type of operations you want to create, for example, a contour mill, planar or turning operation.

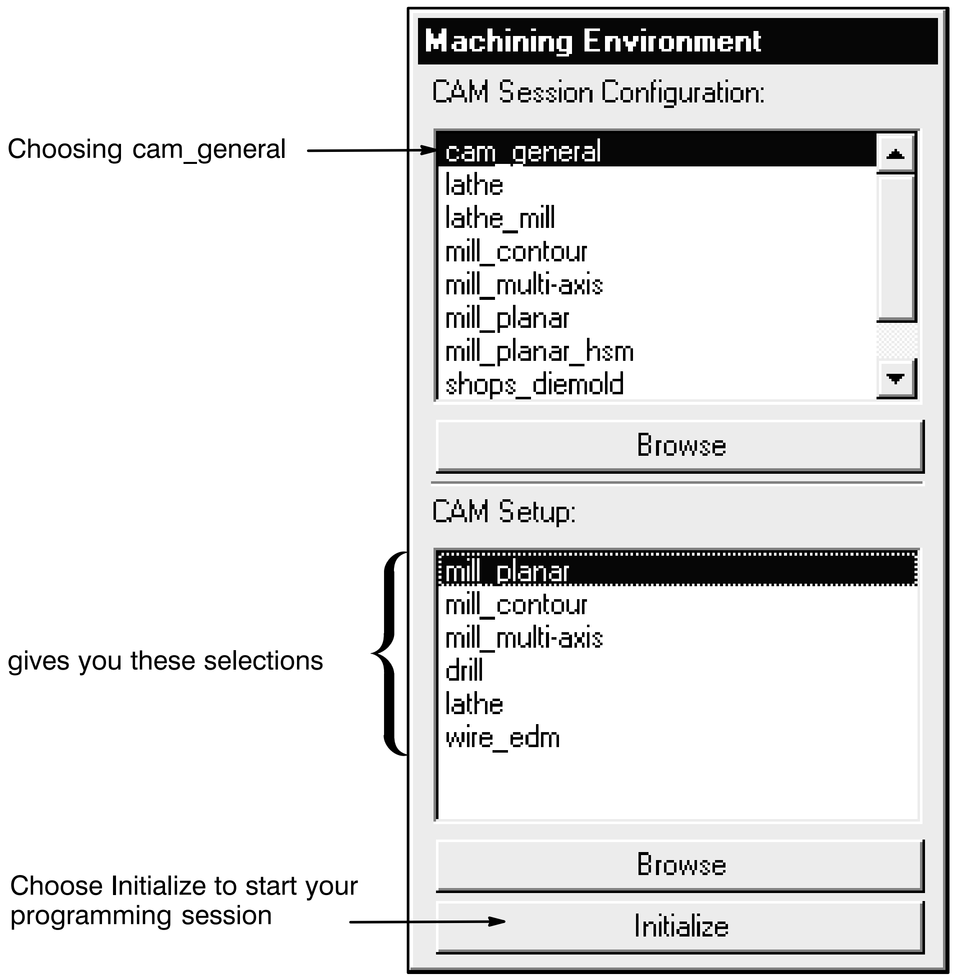
To specify the Machining Environment, you must specify a:

- Configuration
- Setup

Configuration

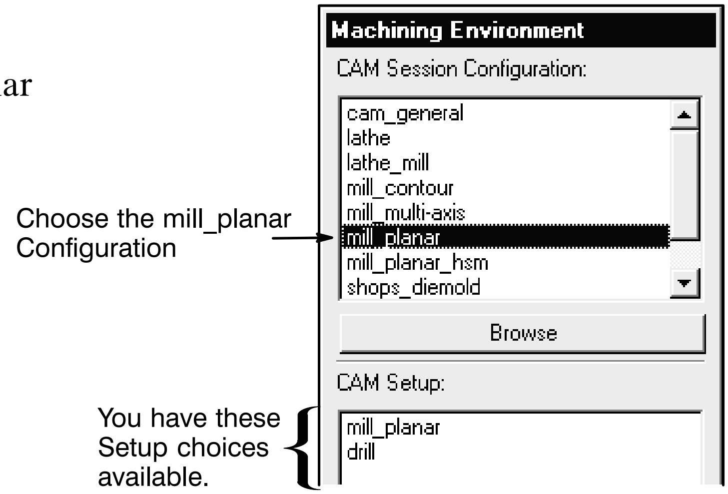
Your choice under the Configuration determines the types of operations that are listed under Setup.

The Configuration you choose determines the Setups that become available.



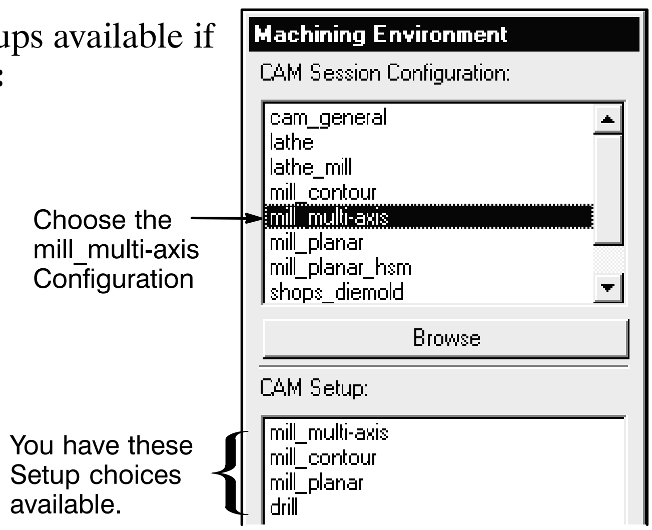
You have the following Setups available if you choose **mill_planar:**

- mill_planar
- drill



You have the following Setups available if you choose **mill_multi-axis:**

- mill_planar
- drill
- mill_contour
- mill_multi-axis



Your choice of Configuration determines the file output format and default location for:

- Shop Documentation
- Postprocessing
- CLS files
- Library files used to specify Tools, Machines, Cut Methods, part material, tool material and speeds and feeds

CAM Setup

After you specify the Configuration, you choose the CAM Setup to Initialize the current part.

Your choice under CAM Setup determines:

- The types of operations that become available
- The types of Programs, Tools, Methods, and Geometry that you can create
- What is created automatically the first time you enter the manufacturing application

Activity 7—1: Choosing the Machining Environment

In previous activities, you were not required to define a Machining Environment since operations already existed in the part file. In this activity you will open a part file that does not have any operations in it and will then define the Machining Environment.

Step 1 Create a New part file and enter the Manufacturing application.

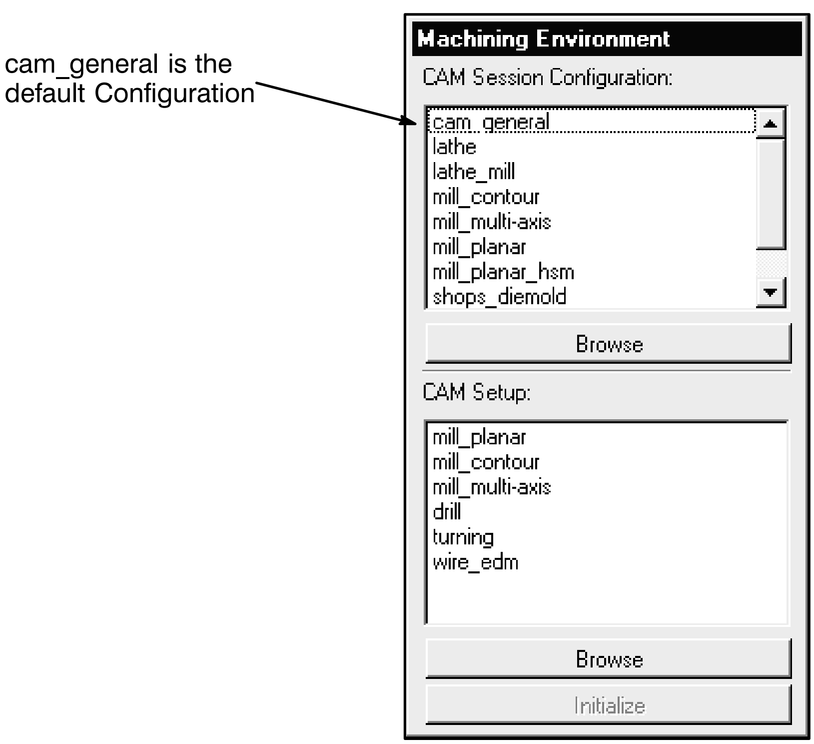
❑ Create a new part file and name it *****.prt.**

❑ Enter the **Manufacturing** application.

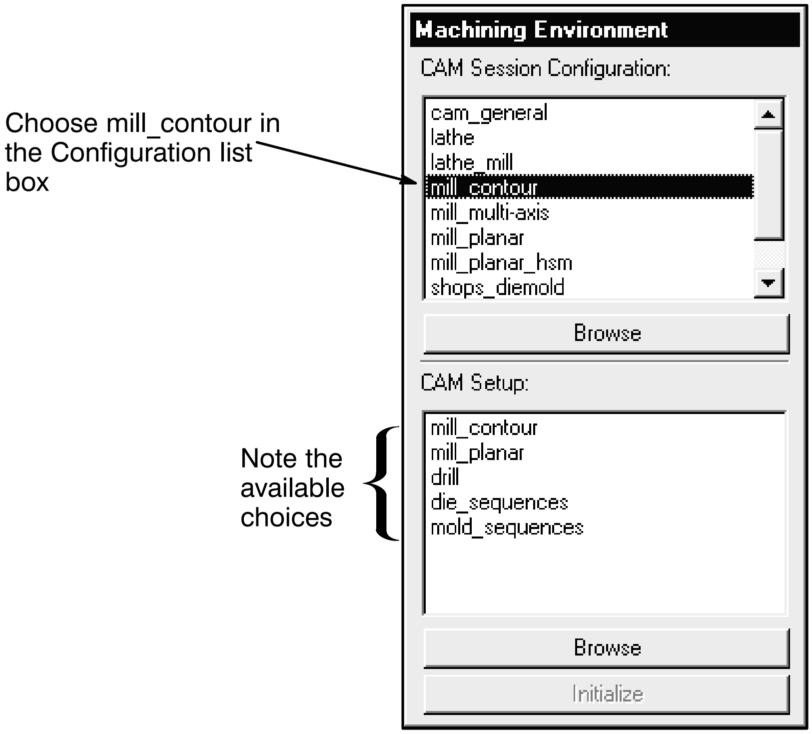
The Machining Environment dialog is displayed.

Step 2 Examine the different CAM Setup options that are available for the different Configuration settings.

The default Machining Environment configuration is currently **cam_general**. You are going to change the Configuration.

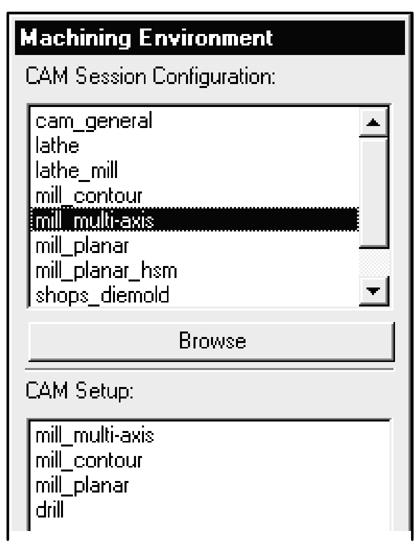


❑ Choose **mill_contour** in the Configuration list box.



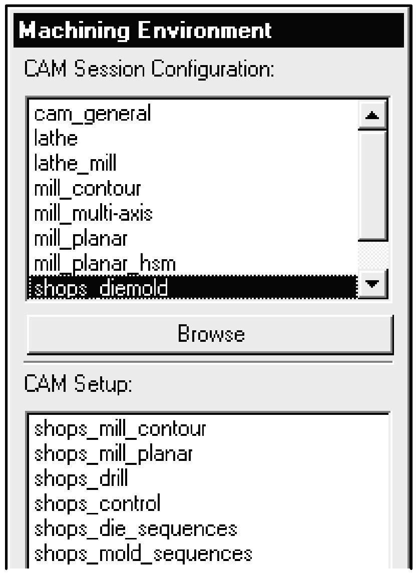
Note the choices you have in the CAM Setup window.

❑ Choose **mill_multi-axis** in the Configuration list box.



Note the different choices listed under CAM Setup.

❑ Choose **shops_diemold** in the Configuration list box.



❑ Choose the different Configuration selections and examine the CAM Setup choices.

Step 3 Defining a Machining Environment and Initializing the configuration.

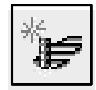
❑ Choose **mill_contour** in the Configuration list box.

❑ Choose **mill_contour** in the CAM Setup list box.

The mill_contour selection in the CAM Setup list box is going to determine the types of operations you will have available when you Initialize the configuration.

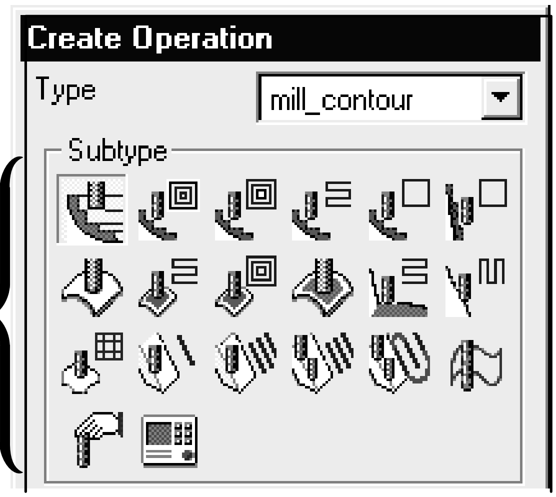
❑ Choose **Initialize.**

Step 4 Choose the Create Operation dialog.

- ❑ Choose the **Create Operation** icon  from the Manufacturing Create toolbar.

The Create Operation dialog is displayed.

The Configuration and Setup that you selected determines which Subtypes become available.



Notice the different Subtypes available. This was determined by the Configuration and Setup selections you made. The Configuration and Setup selections also determined the various default files, such as postprocessors, that are available to you.

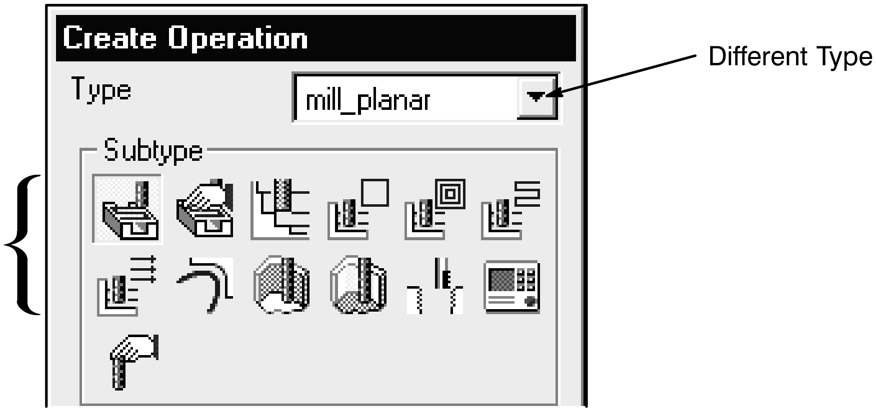
Step 5 Change the Type option on the Create Operation dialog.

- ❑ Change the **Type** and note the Subtype operations that become available.

Change the Type

Changing the Type is similar to, but not identical to, changing the Setup.

Notice the different Subtypes that you have available



❑ **Close** without saving the part file.

This completes the activity and the lesson.

SUMMARY

The Machining Environment is the basis for various operations and defaults which become available for machining applications.

In this lesson you:

- selected the Configuration which determines the CAM Setup choices
- selected the CAM Setup, which determined the types of operations you use within your program

(This Page Intentionally Left Blank)

The Operation Navigator

Lesson 8

PURPOSE

This lesson introduces you to the basic use and functionality of the Operation Navigator.

OBJECTIVES

Upon completion of this lesson, you will be able to:

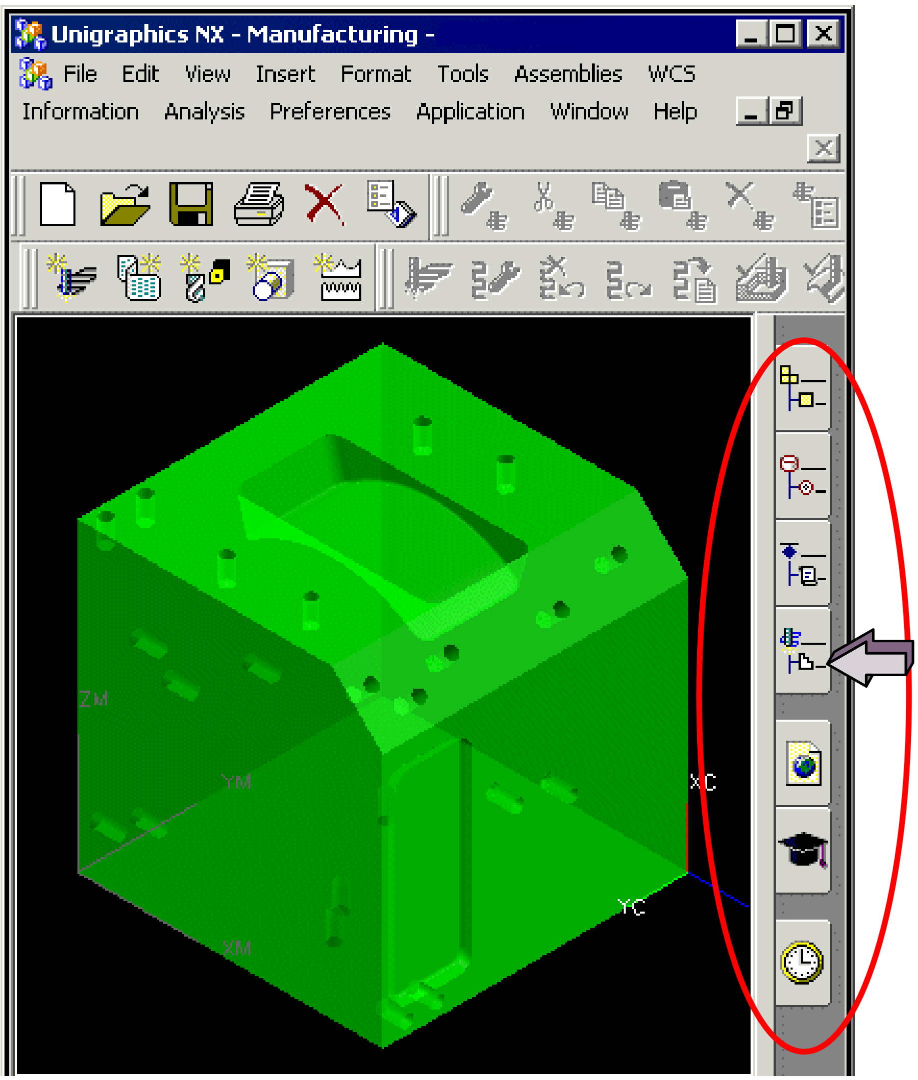
- Customize types of data displayed in the Operation Navigator
- Cut, paste, drag and drop data in the Operation Navigator
- Recognize the Operation Navigator symbols
- Rename an operation
- Manipulate the screen placement of the Operation Navigator
- Drag and drop multiple objects
- Activate the Operation Navigator from the resource bar
- Perform functions with the Dynamic Mouse Button 3 pop-up menu

This lesson contains the following activities:

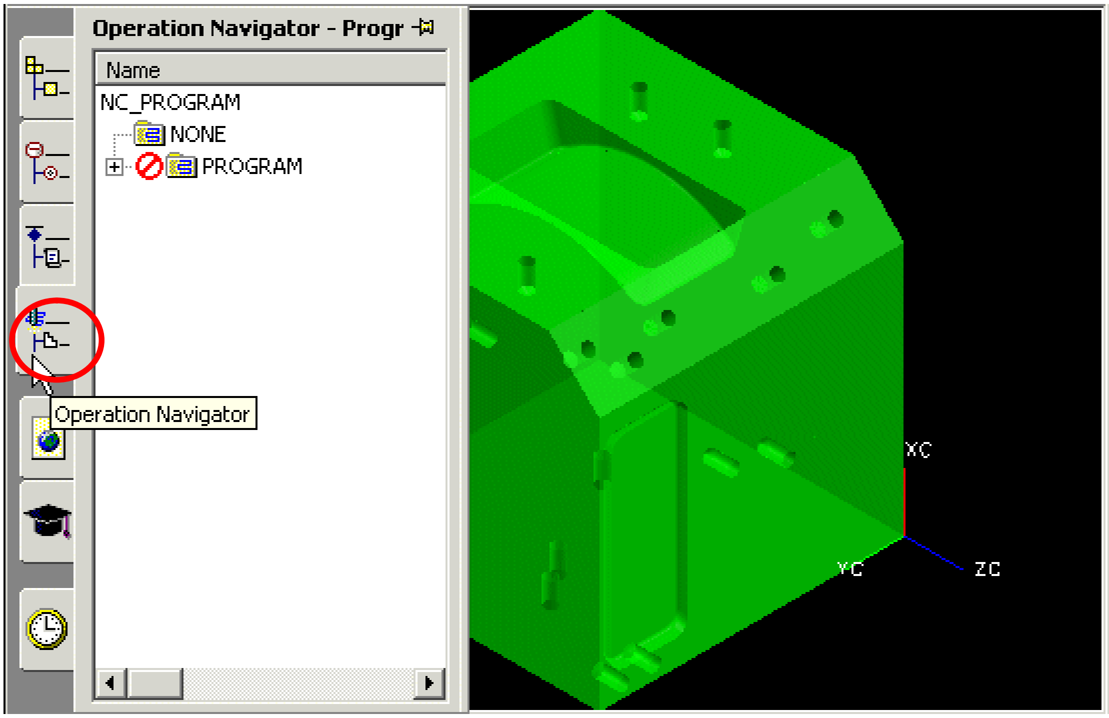
The Operation Navigator

The Operation Navigator is selected from the resource bar, normally located to the right of the screen, after you have initially entered the Manufacturing Application.

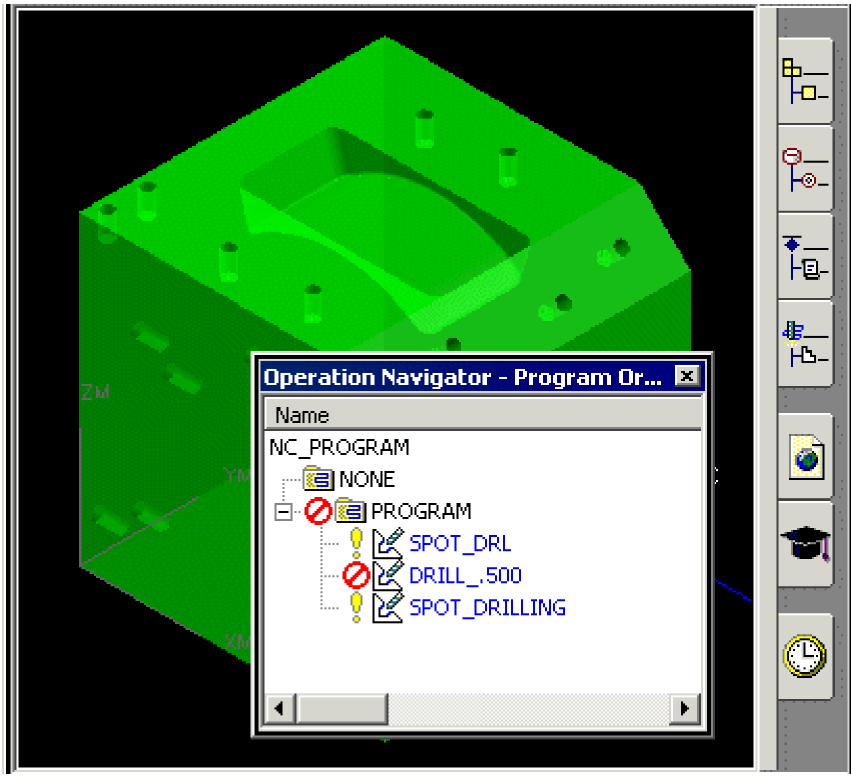
The resource bar contains numerous options designated by tabs with icons, based on preferences that you select and the application which is activated.



Choosing the Operation Navigator tab will bring up the Operation Navigator.



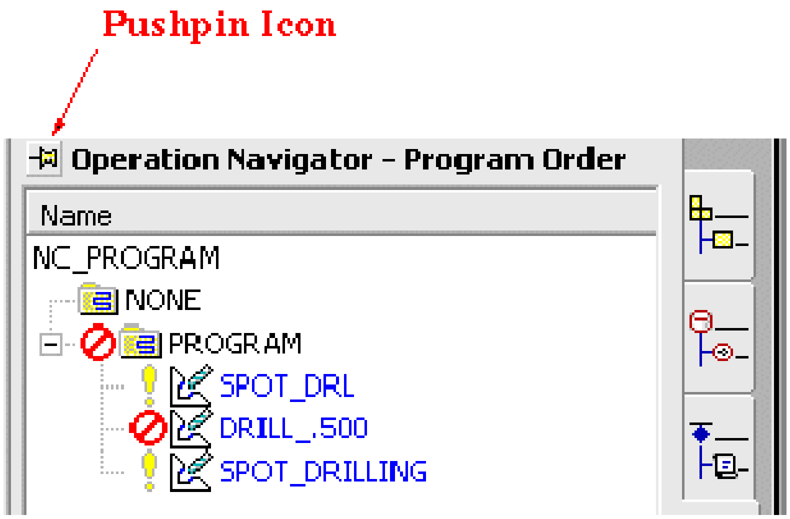
Double clicking the Operation Navigator tab allows the Operation Navigator to break away from the resource bar. The Operation Navigator can then be docked wherever you may choose to drag and drop it.



8

When you close the Operation Navigator window, it will return to the resource bar.

When the Operation Navigator is in the graphic window, a pushpin icon is found on the top left or right hand corner of the Navigator (based on User Interface Preferences).



Choosing the pushpin will cause the Operation Navigator to remain stationary. When you move your cursor outside the navigator window the navigator window will remain open and the graphics window will change size.

Choosing the pushpin a second time will release the Operation Navigator from its stationary position.

The objects (Parent Groups) that you create are displayed in the Operation Navigator in one of four different views:

- Program Order View
- Machine Tool View
- Geometry View
- Machining Method View

Each view displays *classes or groups* (Commonly referred to as Parent Groups) of information that is relevant to that particular view. For example, an end mill used for milling would fall within the Machine Tool view, because a machine tool uses the cutting tool. Check geometry would fall within the Geometry view because the check geometry is used when machining the part.

The Operation Navigator also allows you to:

- drag and drop objects
- cut and paste objects

- delete objects
- replay objects
- edit objects

Display of the Operation Navigator is also controlled by the Operation Navigator Toolbar which is located on the Manufacturing menu bar.

Choosing any of the icons that represent the various Operation Navigator views will determine the view that is displayed.

Operation Navigator Toolbar

Operation Navigator Views

8

Activity 8–1: Using the Operation Navigator

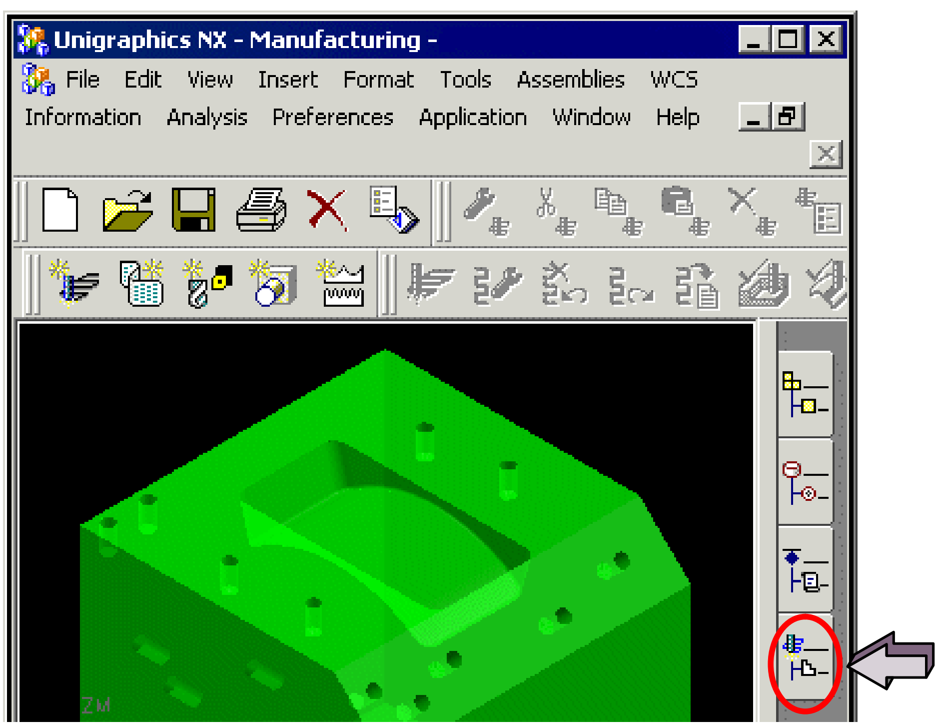
In this activity you will activate, drag and drop and manipulate the Operation Navigator.

Step 1 Open the part mmp_ui_nx and choose the Manufacturing application.

- ❑ From the menu bar, select **File**.
- ❑ Choose **Open**.
- ❑ Select the file **mmp_ui_nx** then choose **OK**.
- ❑ Choose **Application→Manufacturing**.

Step 2 Activate the Operation Navigator.

- ❑ Choose the **Operation Navigator** tab from the resource bar.

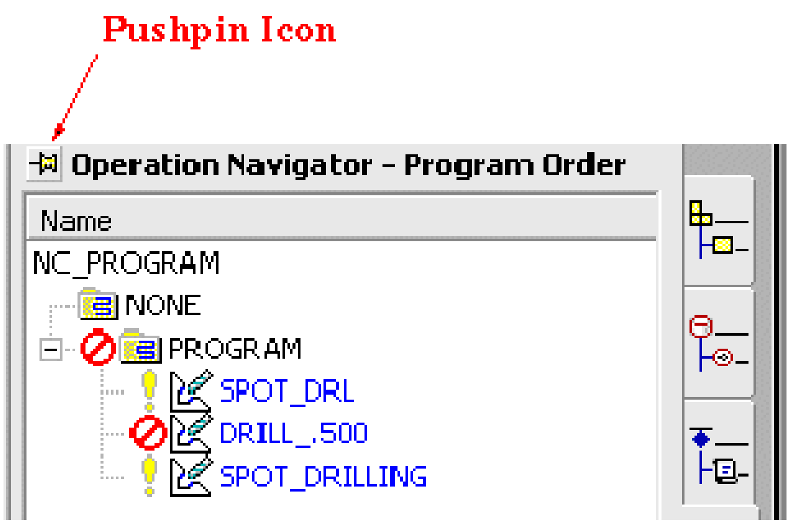


The Operation Navigator is displayed in the graphics window.

- ❑ Move the cursor around the graphics display window and notice that the Operational Navigator disappears.

- ❑ Selecting the Operation Navigator tab from the resource bar will display the Navigator.

You can keep the Operation Navigator displayed permanently by selection of the pushpin icon.



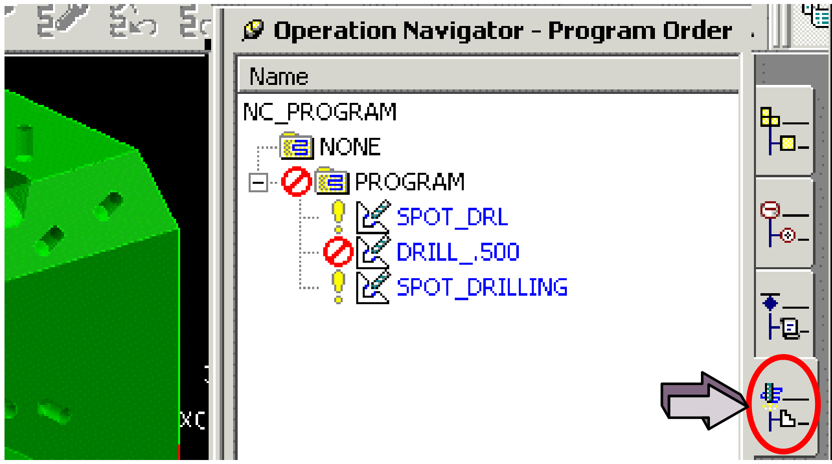
Step 3 Keeping the Operation Navigator display stationary.

- ❑ Choose the **pushpin icon** 

The Operation Navigator display is now stationary. Notice the change in the display of the **pushpin icon**.

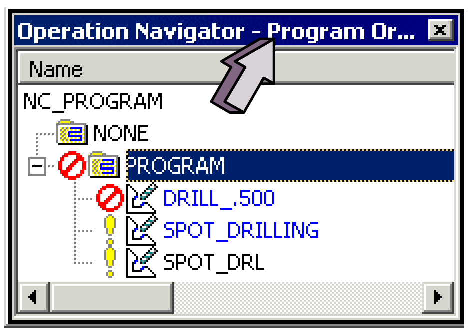
Step 4 Relocating the Operation Navigator display by drag and drop.

- ❑ **Double click** the Operation Navigator tab on the resource bar.



The Operation Navigator display is outside of the graphic window and can now be dragged and dropped by using MB1.

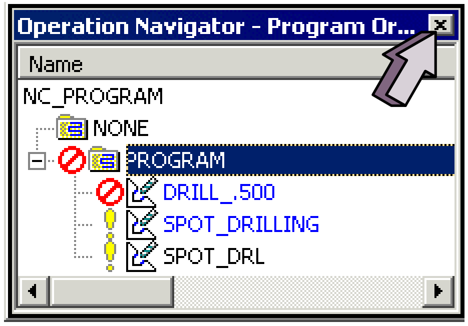
❑ With MB1, select the active title bar and drag the Operation Navigator to any open location on the screen, then release MB1.



This is a convenient method of locating the Operation Navigator to a viewing position that is desirable to you.

Step 5 Returning the Operation Navigator to the resource bar.

❑ Choose the close button in the upper right hand corner of the Operation Navigator window.



Do not save or close this part since you will use it in the next activity.

The Operation Navigator Views

The **Program Order** view is used to sequence the operations for output to the CLSF or postprocessor. It also shows which program parent group each operation belongs to. This is the *only* view in which the order of the operations is relevant.

Operation Navigator - Program Order

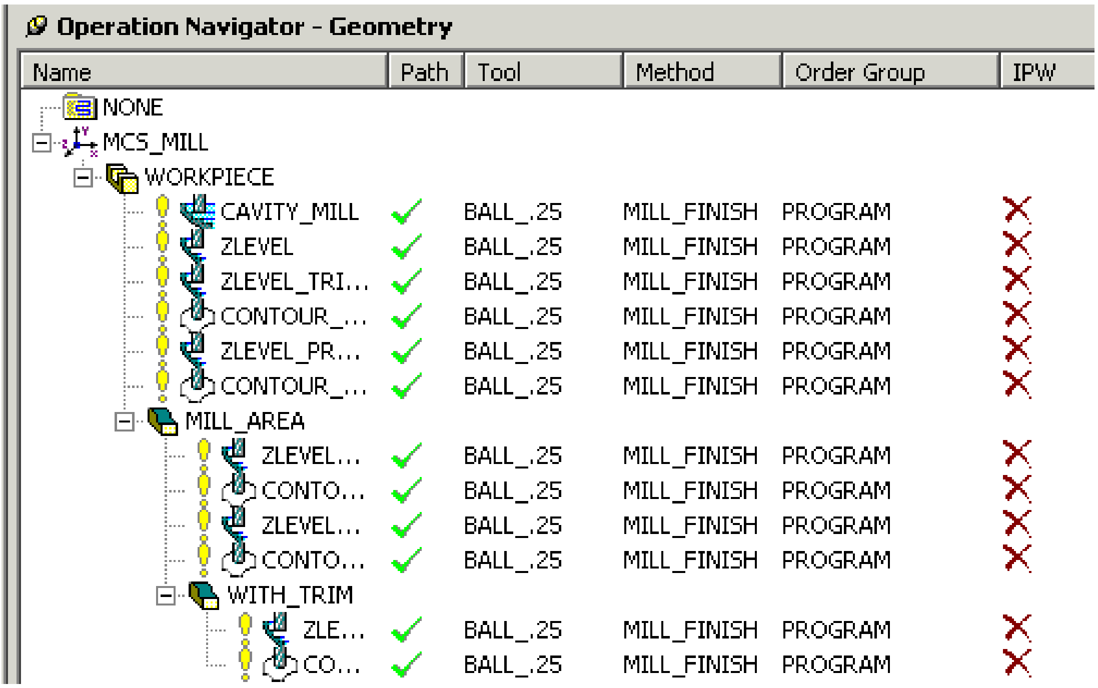
Name	Tool	Toolchange	Path	Tool Description	Geometry	Method
NC_PROGRAM						
NONE						
PROGRAM						
CAVITY_MILL	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
ZLEVEL	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
ZLEVEL_TRIM_SIL	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
CONTOUR_FOLLOW	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
ZLEVEL_PROFILE_STEE...	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
CONTOUR_AREA_NON...	BALL_.25		✓	Milling Tool-5 Parameters	WORKPIECE	MILL_FINISH
ZLEVEL_PROFILE	BALL_.25		✓	Milling Tool-5 Parameters	MILL_AREA	MILL_FINISH
CONTOUR_FOLLOW_1	BALL_.25		✓	Milling Tool-5 Parameters	MILL_AREA	MILL_FINISH
ZLEVEL_PROFILE_STEEP	BALL_.25		✓	Milling Tool-5 Parameters	MILL_AREA	MILL_FINISH
CONTOUR_AREA_NON...	BALL_.25		✓	Milling Tool-5 Parameters	MILL_AREA	MILL_FINISH
ZLEVEL_PROFILE_COPY	BALL_.25		✓	Milling Tool-5 Parameters	WITH_TRIM	MILL_FINISH
CONTOUR_FOLLOW_1...	BALL_.25		✓	Milling Tool-5 Parameters	WITH_TRIM	MILL_FINISH

The **Machine Tool** view arranges operations by cutting tools and can organize cutting tools by the type of tool. For lathes it can also organize tools by turrets.

Operation Navigator - Machine Tool

Name	Path	Description	Geometry	Method	Order Group
GENERIC_MACHINE		Generic Machine			
NONE		mill_contour			
BALL_.5		Milling Tool-5 Parameters			
BALL_.25		Milling Tool-5 Parameters			
CAVITY_MILL	✓	CAVITY_MILL	WORKPIECE	MILL_FINISH	PROGRAM
ZLEVEL_TRIM...	✓	ZLEVEL_PROFILE	WORKPIECE	MILL_FINISH	PROGRAM
ZLEVEL	✓	ZLEVEL_PROFILE	WORKPIECE	MILL_FINISH	PROGRAM
CONTOUR_F...	✓	CONTOUR_FOLLOW	WORKPIECE	MILL_FINISH	PROGRAM
ZLEVEL_PRO...	✓	ZLEVEL_PROFILE	MILL_AREA	MILL_FINISH	PROGRAM
CONTOUR_F...	✓	CONTOUR_FOLLOW	MILL_AREA	MILL_FINISH	PROGRAM
ZLEVEL_PRO...	✓	ZLEVEL_PROFILE	WITH_TRIM	MILL_FINISH	PROGRAM
CONTOUR_F...	✓	CONTOUR_FOLLOW	WITH_TRIM	MILL_FINISH	PROGRAM
ZLEVEL_PRO...	✓	ZLEVEL_PROFILE_STEEP	MILL_AREA	MILL_FINISH	PROGRAM
CONTOUR_A...	✓	CONTOUR_AREA_NON...	MILL_AREA	MILL_FINISH	PROGRAM
ZLEVEL_PRO...	✓	ZLEVEL_PROFILE_STEEP	WORKPIECE	MILL_FINISH	PROGRAM
CONTOUR_A...	✓	CONTOUR_AREA_NON...	WORKPIECE	MILL_FINISH	PROGRAM

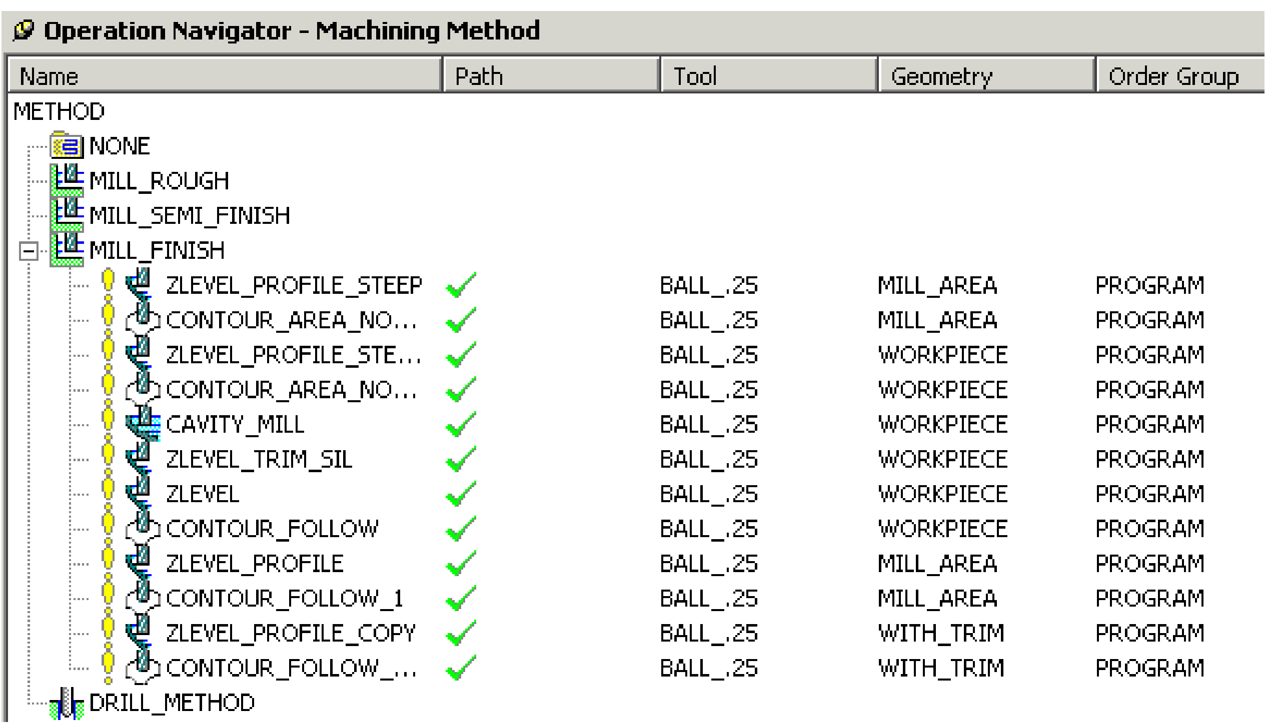
The **Geometry** view shows the MCS and machining geometry that operations and or geometry Parent Groups will use.



Operation Navigator - Geometry

Name	Path	Tool	Method	Order Group	IPW
NONE					
MCS_MILL					
WORKPIECE					
CAVITY_MILL	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
ZLEVEL	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
ZLEVEL_TRI...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
CONTOUR_...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
ZLEVEL_PR...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
CONTOUR_...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
MILL_AREA					
ZLEVEL...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
CONTO...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
ZLEVEL...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
CONTO...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
WITH_TRIM					
ZLE...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗
CO...	✓	BALL_.25	MILL_FINISH	PROGRAM	✗

The **Machining Method** view allows the organization of operations under machining disciplines that share common parameter values such as rough, semi-finish, and finish.



Operation Navigator - Machining Method

Name	Path	Tool	Geometry	Order Group
METHOD				
NONE				
MILL_ROUGH				
MILL_SEMI_FINISH				
MILL_FINISH				
ZLEVEL_PROFILE_STEEP	✓	BALL_.25	MILL_AREA	PROGRAM
CONTOUR_AREA_NO...	✓	BALL_.25	MILL_AREA	PROGRAM
ZLEVEL_PROFILE_STE...	✓	BALL_.25	WORKPIECE	PROGRAM
CONTOUR_AREA_NO...	✓	BALL_.25	WORKPIECE	PROGRAM
CAVITY_MILL	✓	BALL_.25	WORKPIECE	PROGRAM
ZLEVEL_TRIM_SIL	✓	BALL_.25	WORKPIECE	PROGRAM
ZLEVEL	✓	BALL_.25	WORKPIECE	PROGRAM
CONTOUR_FOLLOW	✓	BALL_.25	WORKPIECE	PROGRAM
ZLEVEL_PROFILE	✓	BALL_.25	MILL_AREA	PROGRAM
CONTOUR_FOLLOW_1	✓	BALL_.25	MILL_AREA	PROGRAM
ZLEVEL_PROFILE_COPY	✓	BALL_.25	WITH_TRIM	PROGRAM
CONTOUR_FOLLOW_...	✓	BALL_.25	WITH_TRIM	PROGRAM
DRILL_METHOD				

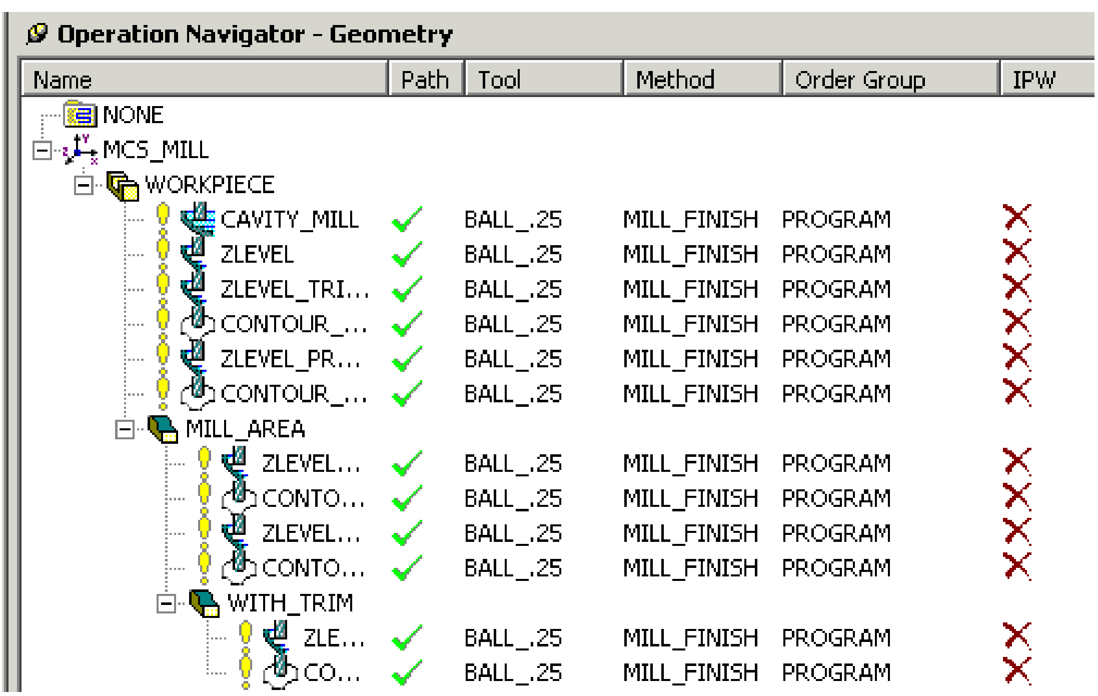
Parent Groups, Operations and Inheritance

As mentioned previously, objects (commonly referred to as Parent Groups) and operations that you create are displayed in the Operation Navigator in one of four different views. The Operation Navigator uses a tree structure to show the relationship between the various Parent Groups and operations.

Parent Groups may contain other Parent Groups and or operations. Information can be passed down (or inherited) to lower members contained within the group. This information might be a physical cutting tool, part or check geometry, MCS, tolerance or stock values. Any change you make to the Tool, Geometry or Method Parent Group, changes that setting in the operations or other Parent Group contained within that particular group.

For example, if you change a Method Parent Group cut feed rate, all operations contained in that Parent Group will have their cut feed rate changed. If you were to cut an operation from within a Parent Group and paste it under another Parent Group (in that same view) the operation will inherit the values of the new Parent Group. Most parameter or objects that can be inherited can be modified so that they are *not* inherited.

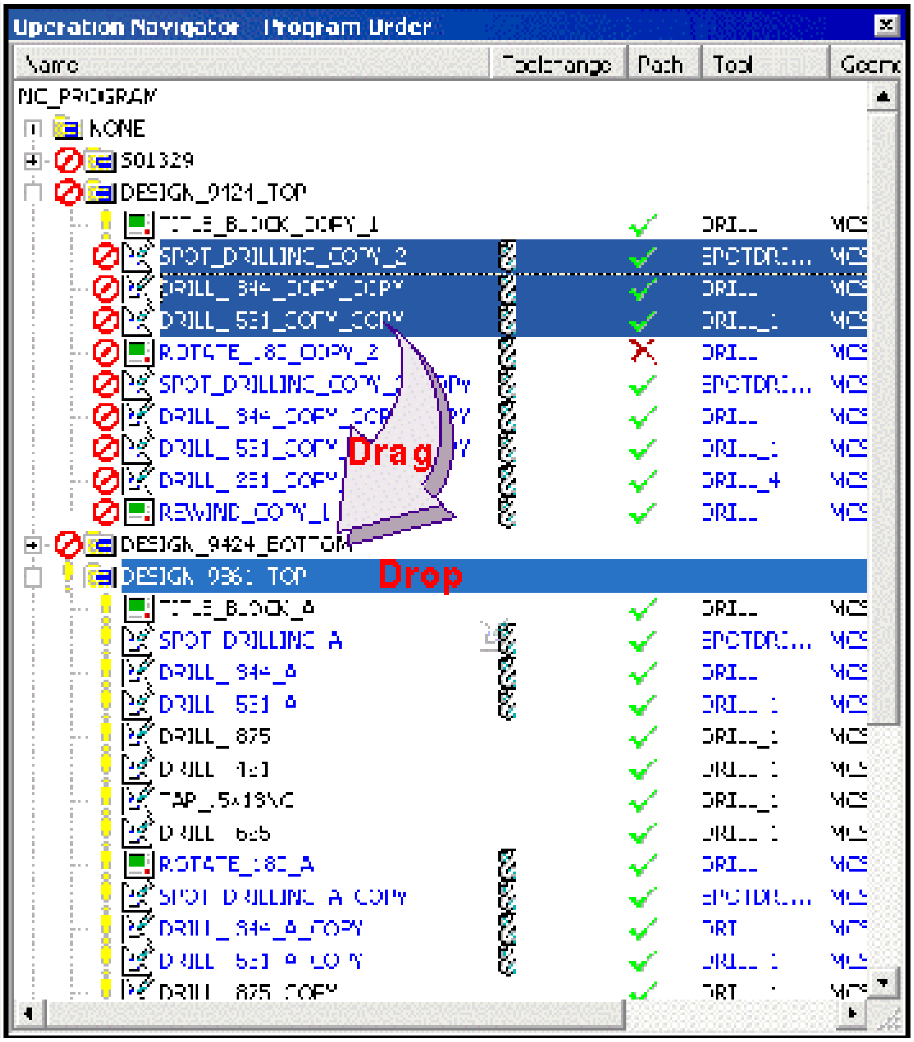
In the example that follows of the Geometry View, the Parent Groups are:



- **GEOMETRY –** Default Geometry Parent Group name containing all other Geometry parents.
- **NONE –** Default Geometry group name, sometimes used as a place holder.
- **MCS_MILL –** Parent Group containing the MCS. This MCS will be common to all group objects contained within MCS_MILL.
- **WORKPIECE –** Parent Group containing the Workpiece geometry.
- **MILL_AREA –** Parent Group containing geometry and the operations used to machine that geometry.
- **WITH_TRIM –** Parent Group containing trim geometry and the operations used to cut that geometry.

Dragging and Dropping Multiple Objects

You can drag and drop multiple objects in the Operation Navigator.



Select multiple objects by using the control key and MB1, then drag the objects to the desired location, and release MB1. The objects will appear below the selected location.

The Operation Navigator Appearance and Columns

Status Icons in Columns

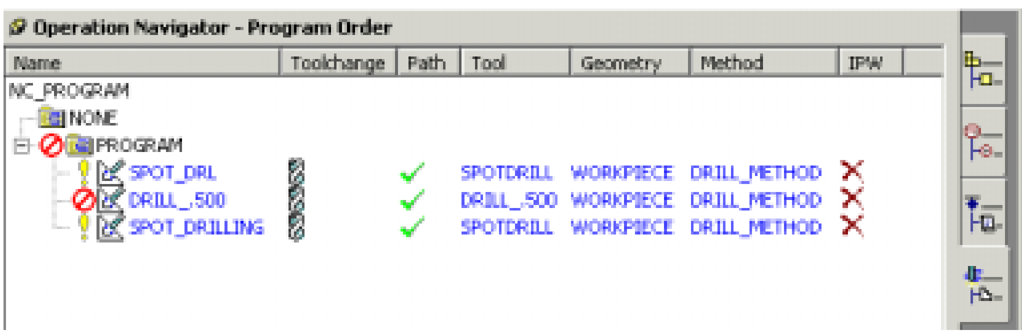
The Operation Navigator can display both icons and text, icons, or just text in the columns that are displayed.

The Name column uses a combination of icons and text to designate the name and status of the operations.

The columns for Toolchange, Path and In Process Workpiece shows icons only. This allows you to make these columns smaller allowing for the display of additional columns.

Moving the mouse over an icon, displays a small window which gives a description of what the icon represents.

In the Toolchange column, the description is the name of the tool used in the operation.



The Name column

In the Name column, operations are preceded by a status symbol icon which represents Complete, Regenerate, or Repost.

The icons and their representation are:

 Complete indicates the tool path has been generated and output has been created (postprocessed or CLS output). The path has not changed since the output was generated.

 Regenerate indicates the tool path has not been generated or the tool path is out of date. In the Operation Navigator, use **MB3, Objects → Update List** to display an Information window to see what has changed and is causing the tool path to be regenerated.

Repost indicates the tool path has never been output or the tool path has changed since it was last output. In the Operation Navigator, use **MB3 → Objects → Update List** to display an Information window to see what has changed and is causing the repost status. This information window displays the prompt **Need to Post**. Choose the Output CLSF icon in the tool bar to repost and update the status.

The Toolchange column

Displayed only in the Program View. The icons displayed are based on the type of the tool used. If a drill tool is being used, the icon for a drill tool is displayed.

The Path column

Represents the status of the tool path. The status can be:

Generated indicates that the tool path has been created. It may or may not contain actual tool movement.

None indicates the tool path has either been deleted or not generated.

Imported indicates that the tool path is a Cutter Location Source File (**Tools → CLSF → Import**). It can be replayed, postprocessed or edited with the graphical tool path editor (**MB3 → Tool path → Edit**).

Edited indicates the tool path has been changed with the graphical tool path editor (**MB3 → Tool path → Edit**).

Suspect indicates that questionable geometry was encountered when the tool path was generated. The path may or may not be valid and needs to be examined by using either **MB3 → Object →Display** or **MB3 → Object → Information** which will show diagnostics with a description of the condition encountered.

Transformed indicates the tool path is from a transformed operation (**MB3 → Object → Transform**).

The In-Process Workpiece (IPW) column

The IPW column indicates that an In-Process Workpiece has been saved by use of the Visualize Dynamic Material Removal option. The icons displayed are:

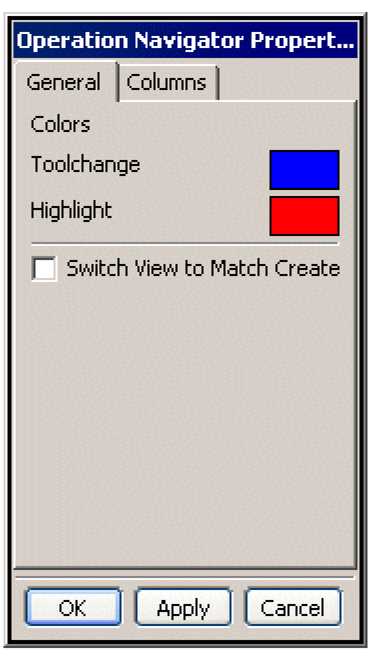
 Generated indicates the IPW has been generated and is current.

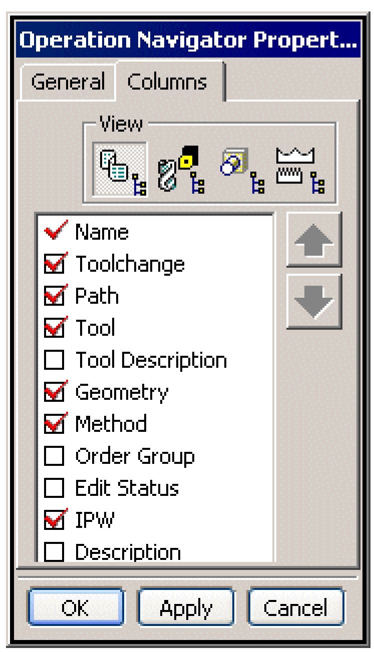
 None indicates that an IPW does not exist.

 Out of Date indicates the IPW is not current. It is updated if the next operation uses the IPW or if the operation is generated by the Dynamic Visualize option. This icon will appear in the IPW column for all the operations below the operation that is out of date.

Appearance of the Operation Navigator

Modifications to the appearance of the Operation Navigator can be accomplished by selecting **MB3** in the background of the Operation Navigator and then selecting Properties (**MB3 → Properties**). Selecting the General property page allows you to specify the color in which the text describing each operation is displayed for operations that contain a tool change. The Columns property page allows you to specify which columns display in the various views of the Operation Navigator.





- **Toolchange** allows you to specify a color for the row of information if a tool change is specified
- **Highlight** allows you to specify the color in which the various objects highlight, when using the option **Tools→Operation Navigator→Find**
- **Columns** allows you to specify which columns are displayed in each view

In the display of the Operation Navigator, the Program Order Group, Machining Method, Geometry and Machine Tool group column names indicate which groups are associated with each operation.

The Toolchange column is displayed only in the Program Order View and indicates if a tool change was required from the *previous* operation. If there is no tool change, the column is left blank.

The Path column displays:

- **Generated**, if the tool path is generated
- **None** if a tool path is not generated
- **Edited** when the tool path has been modified (**MB3, Tool Path→Edit**)
- **Imported** when the tool path has been imported (**Tools→CLSF→Import**)
- **Suspect** when there may be some type of problem encountered with geometry
- **Transformed** indicates that the operation has been transformed

The **IPW** column indicates that an In-Process Workpiece has been saved through the verification process. If **Generated** displays, the IPW has been generated; **None** indicates that no IPW has been generated and **Out of date** indicates that the IPW needs to be regenerated due to changes in the operation.

Activity 8–2: Using the Operation Navigator

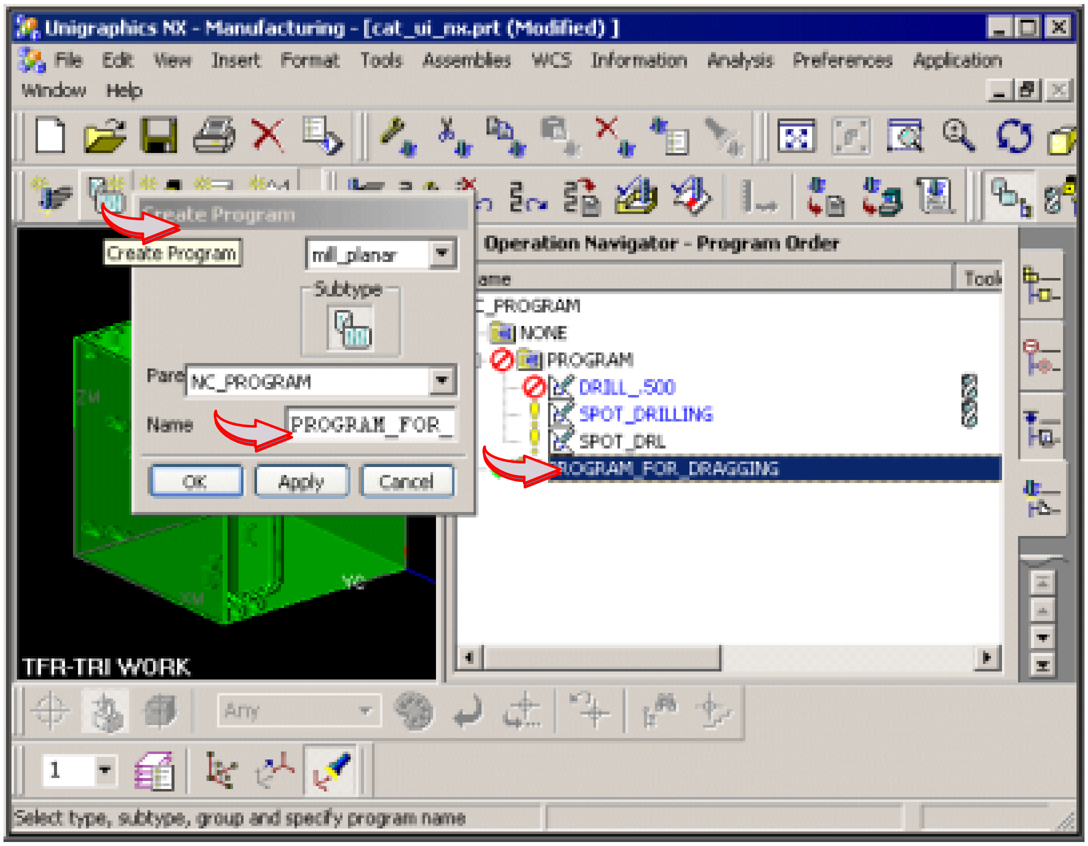
In this activity, you will drag and drop multiple operations using the Operation Navigator.

Step 1 Continue using the displayed part, mmp_ui_nx and verify that you are in the Contiguous Manufacturing application.

- ❑ If necessary, display the Operation Navigator.

Step 2 Creating a new Program parent group.

- ❑ From the Manufacturing Create toolbar, create a program named PROGRAM_FOR_DRAGGING.

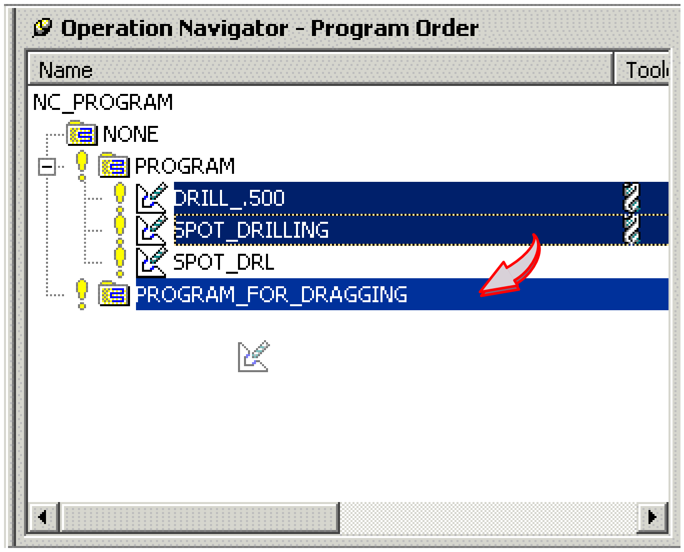


8

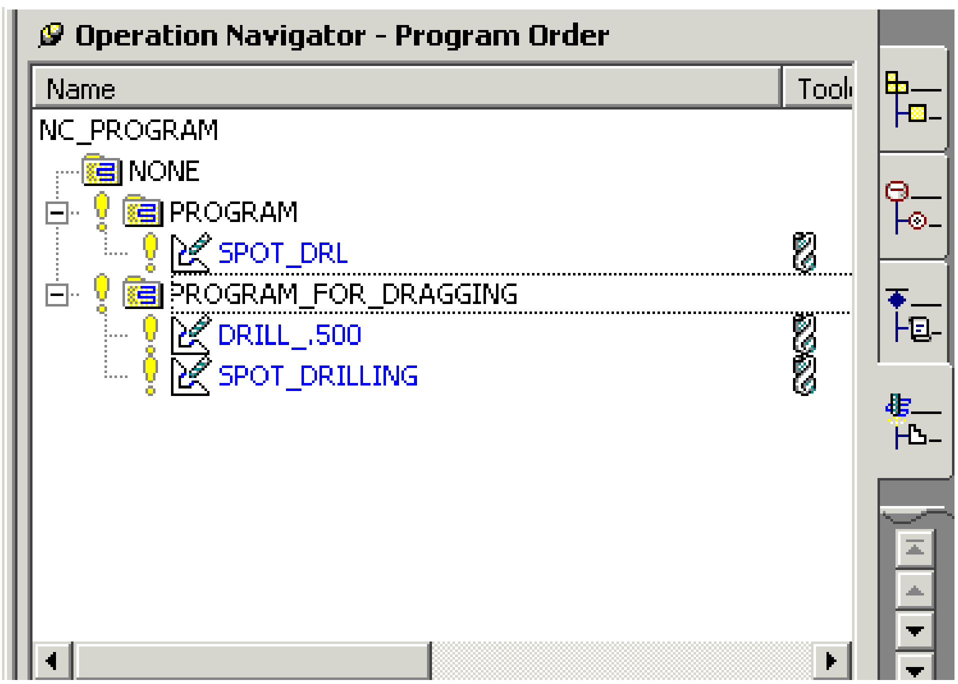
Step 3 Dragging and dropping operations to the new Program parent.

- ❑ Using MB1, select the **DRILL_.500** operation, then hold down the **SHIFT** key and select the **SPOT_DRILLING** operation.

- ❑ Drag the operations to the Program parent **PROGRAM_FOR_DRAGGING**, then release **MB1**.



The DRILL_500 and SPOT_DRILLING operations are moved to the PROGRAM parent PROGRAM_FOR_DRAGGING.

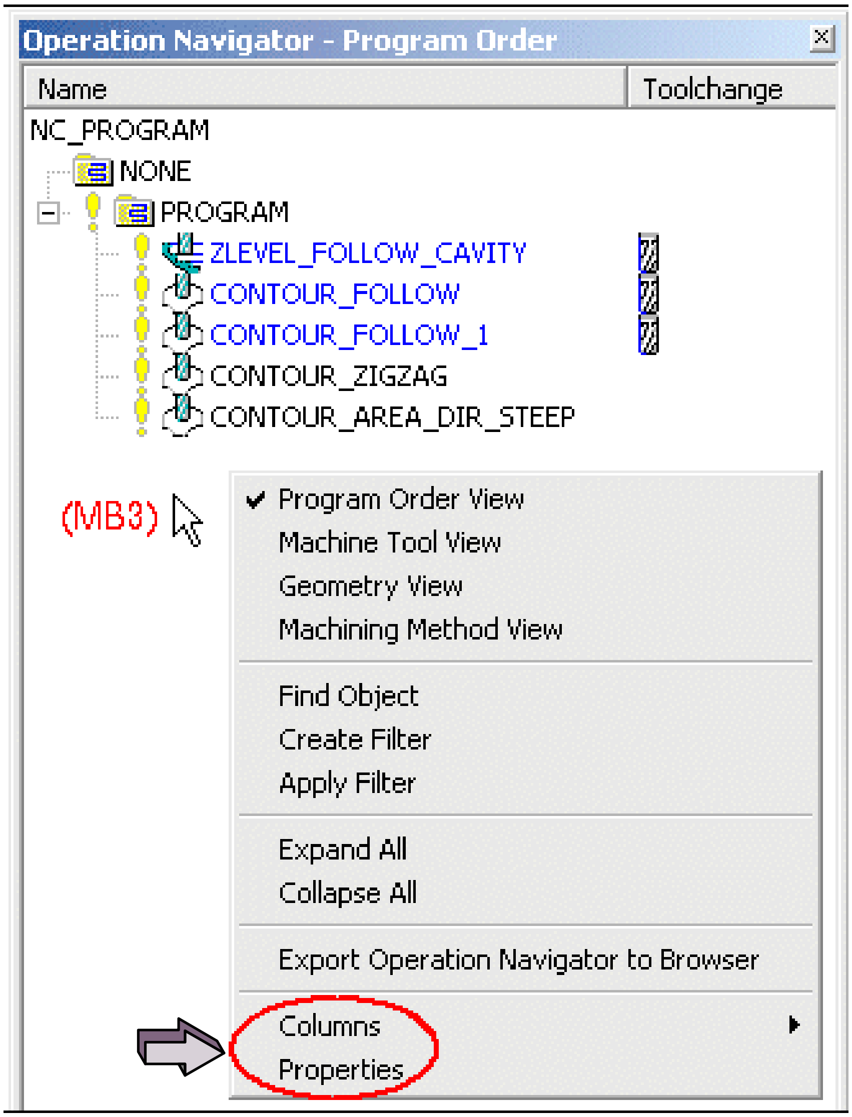


Do not save or close this part since you will use it in the next activity.

Third Mouse Button (MB3) options

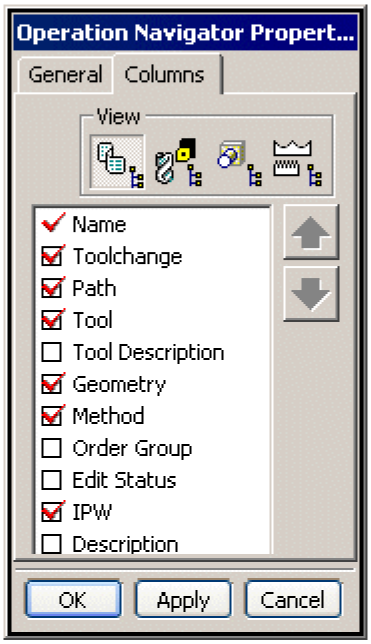
Columns and Properties

Columns and Properties options are now displayed in a pop-up menu by choosing **MB3** on the Operation Navigator background instead of being specified under **Preferences** → **Manufacturing** → **Operation Navigator**.



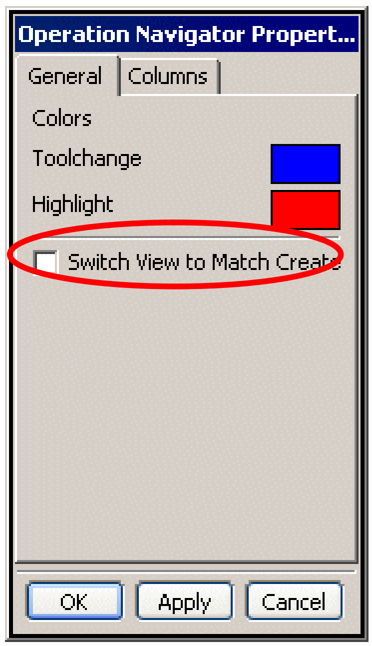
The Columns property page allows you to quickly enable or disable the display of columns in the current view of the Operation Navigator. This property page contains all columns that are available for the current view.

To configure the columns for a specific view, select the view icon at the top of the page. The list box will then show the available columns for the view. To switch an Operation Navigator column display on or off, enable or disable it. To reorder a column as it appears in the window, highlight the entry and choose Move Up or Move Down. You can also reorder a highlighted column using drag and drop functions with the cursor. The Name column must always be visible at the first position. You can not toggle it off or move it to another location.



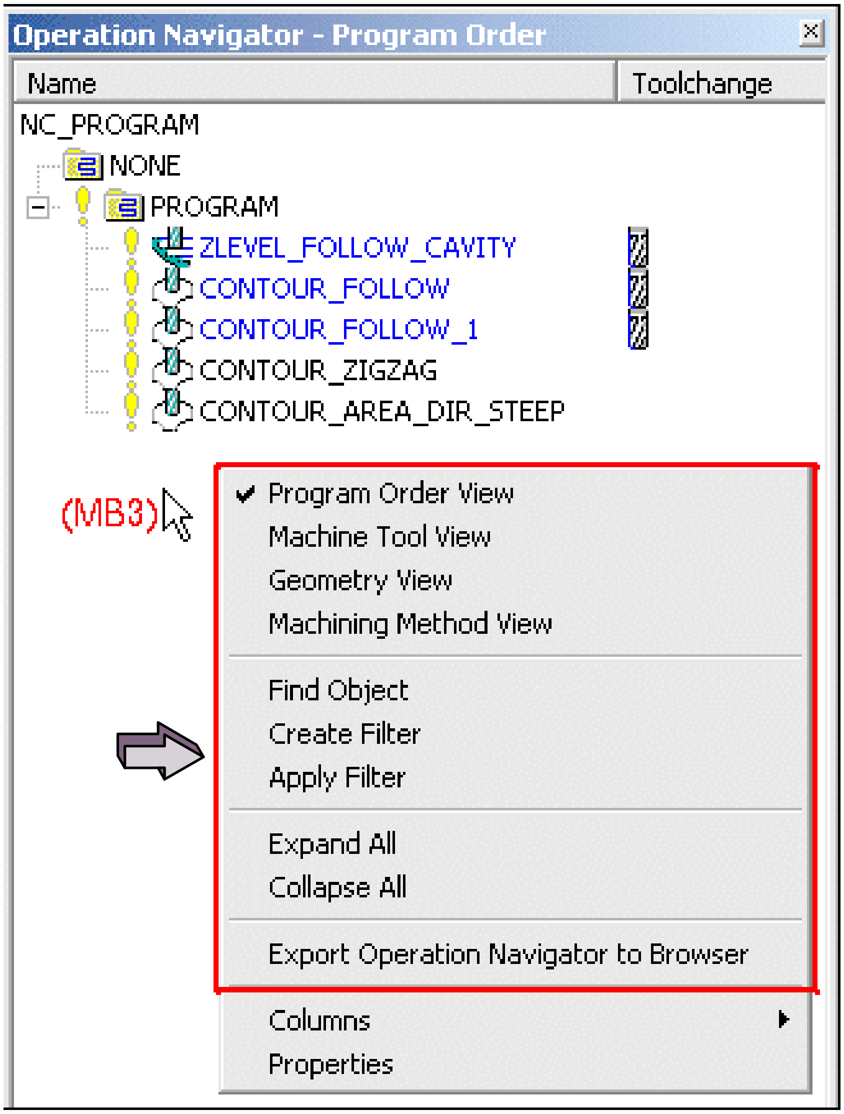
The General property page allows you to control the data displayed in the Operation Navigator. Toolchange allows you to specify the color in which the row of text describing each operation is displayed for operations where a tool change occurs. For each operation, where a tool change occurs, the text displayed in each column appears in the specified color. Highlight allows you to specify the color in which the row of the active operation highlights when using functions such as Visualize.

The option, **Switch View to Match Create**, when activated, will switch to the view of the group object that is being created. This allows you to use the Operation Navigator to select the parent group object as you create groups, and immediately see them in the tree structure.



Additional NX MB3 options

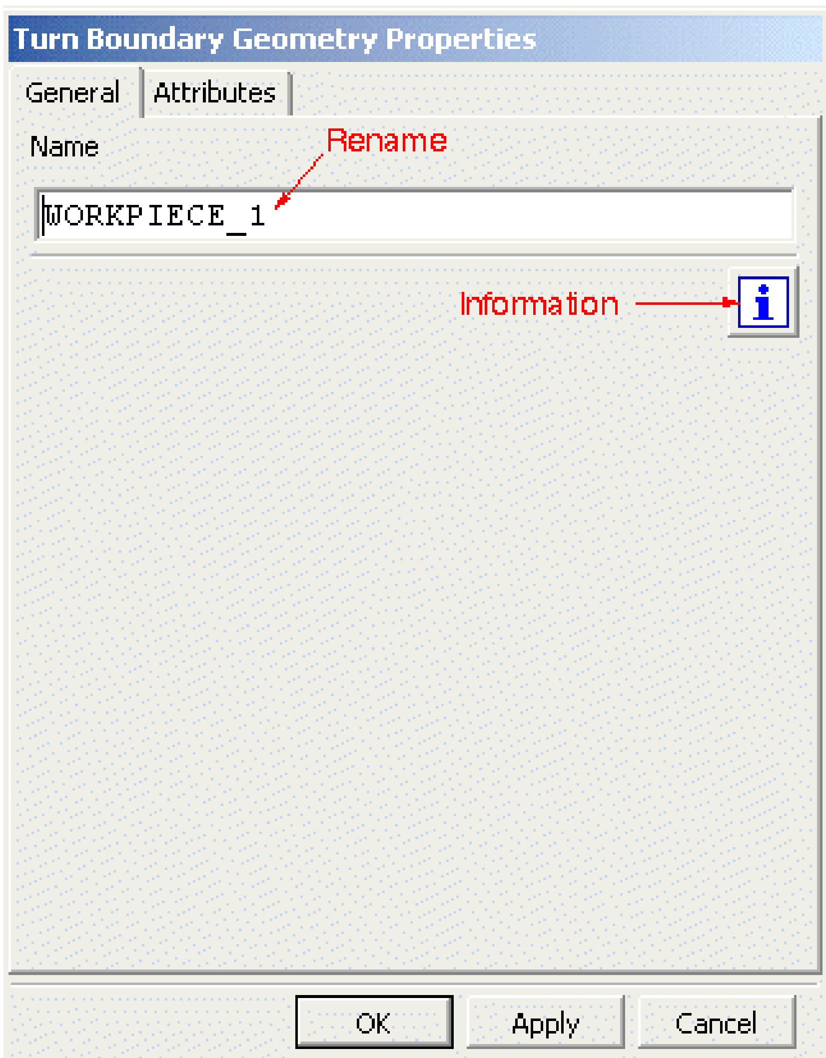
Options that were once available only in the toolbar are now also available in the MB3 pop-up menu.



Object Properties Dialog

Object properties are now displayed in page form. By choosing an object in the Operation Navigator and pressing **MB3 → Properties**, the page allows you to rename the object, retrieve information, and/or edit attributes. The previous method was to choose **Format → Attribute**.

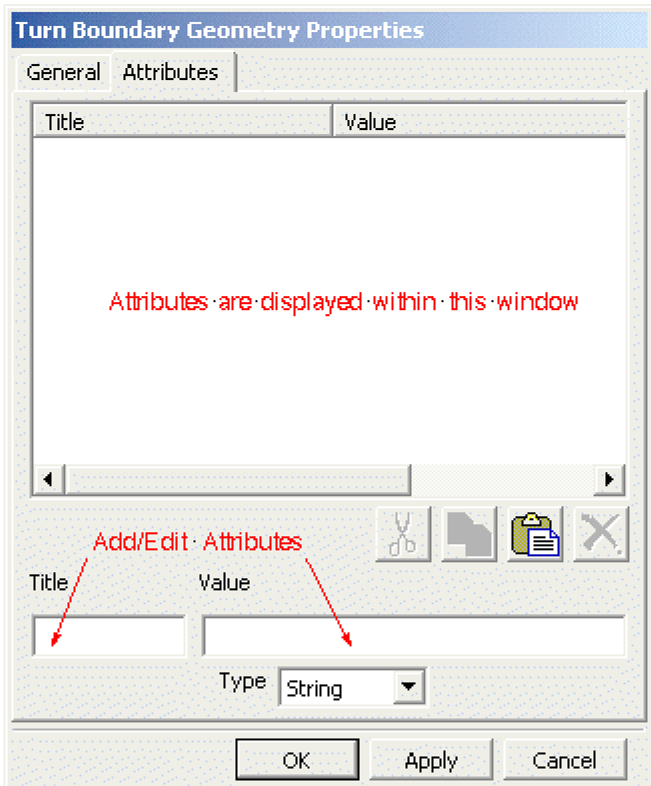
The General page contains an edit field. In the edit field you can rename the current operation name. Changing the name and selecting either OK or APPLY will change the name. Before the new name is applied, a check will be performed for validity. If the name is invalid, a message box will inform you.



Note that the edit field is disabled if the object properties dialog is invoked on a system object which cannot be renamed (for example NONE).

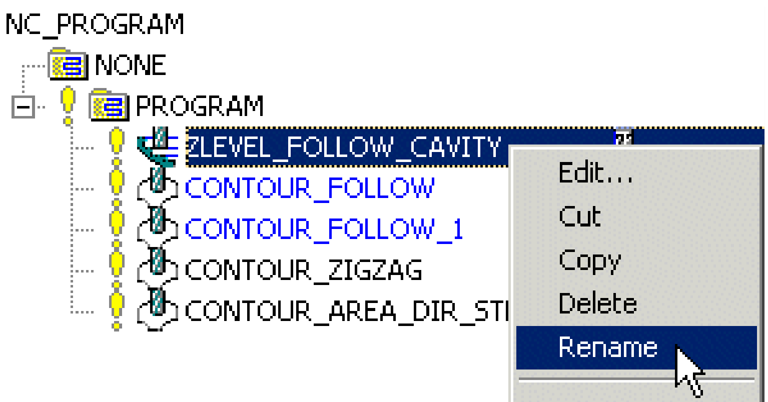
The information icon is also available on the General property page. This icon displays an information window on the currently selected operation or group object.

The Attributes page allows you to view attributes associated with the operation. It also allows you to edit, add, or delete attributes.



Renaming Objects

Objects can be renamed using the Rename option in the MB3 pop-up menu. This is somewhat easier than double-clicking slowly on the object name and then typing in the name text.



Activity 8–3: Use of MB3 with the Operation Navigator

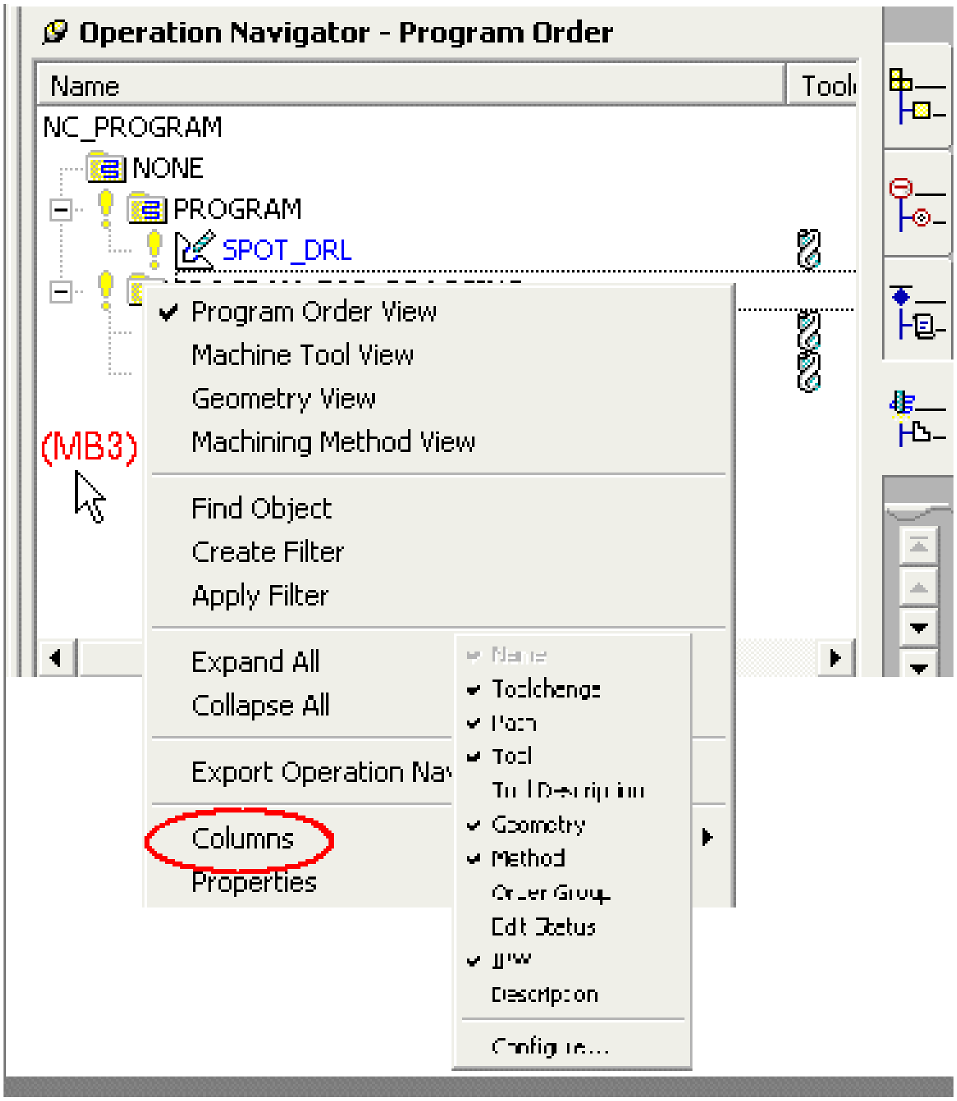
In this activity, you will explore the various functions of using MB3 with the Operation Navigator.

Step 1 Continue using the displayed part, mmp_ui_nx and verify that you are in the Manufacturing application.

❑ If necessary, display the Operation Navigator.

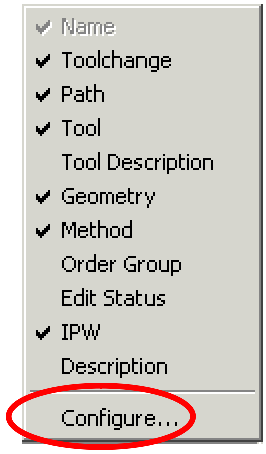
Step 2 Explore the Properties and Columns functionality of the Operation Navigator using MB3.

❑ With the cursor on the Operation Navigator background, select **MB3**, then select Columns from the pop-up menu.



A pop-up menu is displayed, listing the columns that are available for display in the Operation Navigator. Also note the various columns which are check marked. These are the columns which are displayed in the current view of the Operation Navigator.

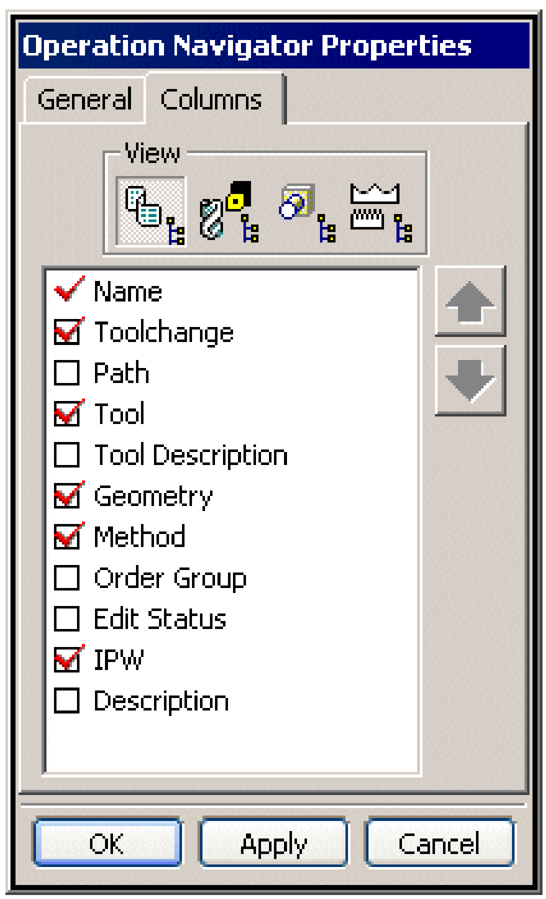
You may change the order in which the columns are displayed as well as have different columns displayed in different views of the Navigator by selecting the Configure option.



Step 3 Reconfigure the columns which are displayed in the Program view of the Operation Navigator. Display the Tool column prior to the Toolchange column and add the Path column to the display list.

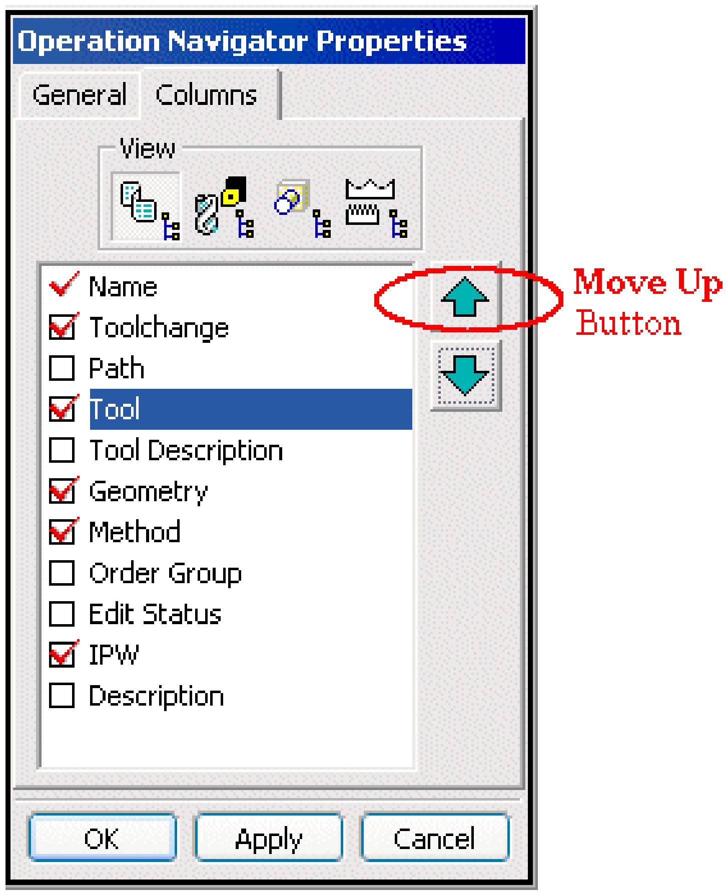
❑ Choose the Configure option from the pop-up menu.

The Operation Navigator Properties page is displayed.

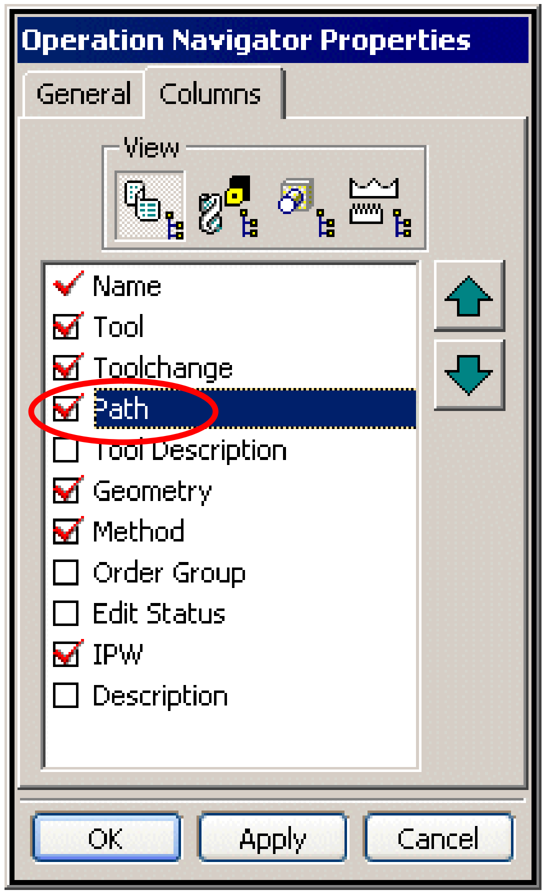


❑ If necessary, change to the Program Order view as shown above.

❑ Highlight the **Tool** column name, and using the **Move Up** button, move the **Tool** prior to the **Toolchange** column name.



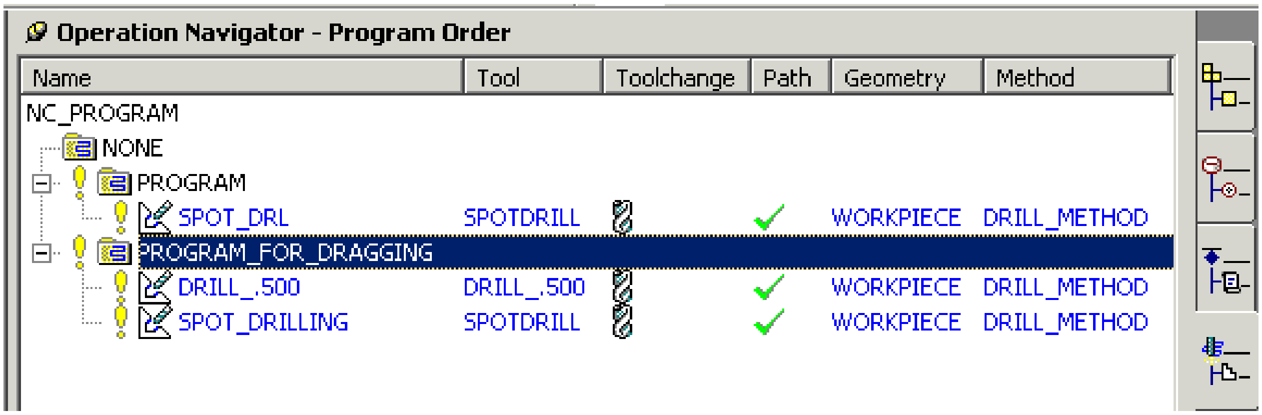
❑ Enable the **Path** column option.



8

❑ Choose **OK** from the Operation Navigator Columns properties page.

❑ Observe the changes to the Program Order view of the Operation Navigator.



Step 4 Rename the Program Order Parent object PROGRAM_FOR_DRAGGING to PROGRAM_2 using MB3.

❑ Highlight the Program Order Parent object **PROGRAM_FOR_DRAGGING**.

❑ Choose **MB3**, **Rename**.

❑ Type the text, **PROGRAM_2,** followed by the **Enter** or **Return** key.

The PROGRAM_FOR_DRAGGING Parent object is renamed to PROGRAM_2.

❑ Close the Part file.

This concludes the activity.

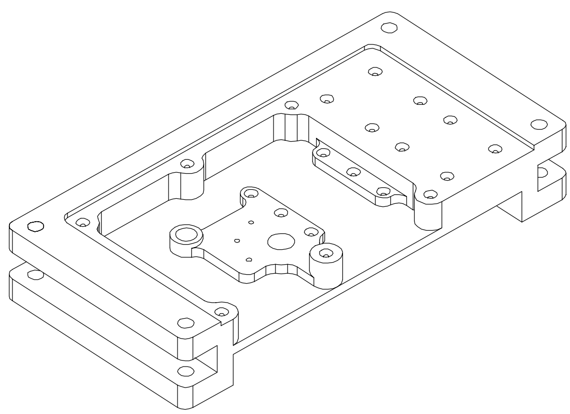
The following activity will be used to review what you have learned about the Operation Navigator.

Activity 8—4: Operation Navigator Review

In this activity, you will open an existing part file and create an operation that will display in the Operation Navigator. You will then perform various tasks that will familiarize you with the functions within the Operation Navigator. Although you have not been taken through the process of creating an operation, you will see the ease of creation in this activity.

Step 1 Open the part file and rename the part file.

- ❑ Choose **File→Open** then choose **mmp_tray_1.prt**.



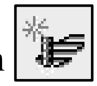
- ❑ Use the **Save As** option under **File** on the menu bar and rename the part to ***_tray_1.prt**, where ***** represents your initials.

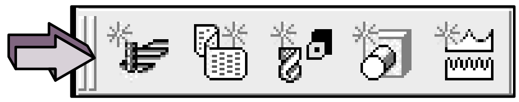
Step 2 Enter the Manufacturing application and display the Operation Navigator.

- ❑ Choose **Application→Manufacturing.**
- ❑ Choose the Operation Navigator tab from the resource bar.

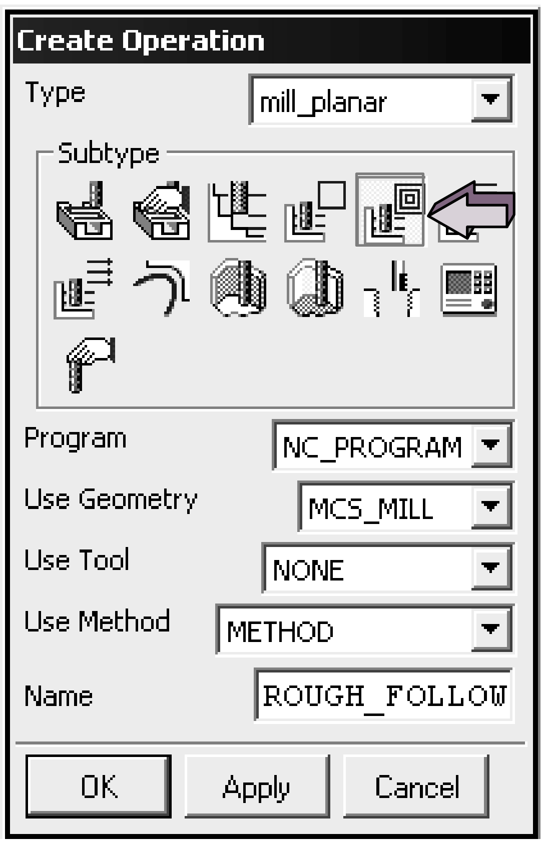
The Operation Navigator is displayed.

Step 3 Creating and Accepting an operation.

❑ Choose the **Create Operation** icon  on the Manufacturing Create toolbar.



❑ Choose the **ROUGH_FOLLOW** Subtype operation icon.



Notice that the default Name changes to ROUGH_FOLLOW.

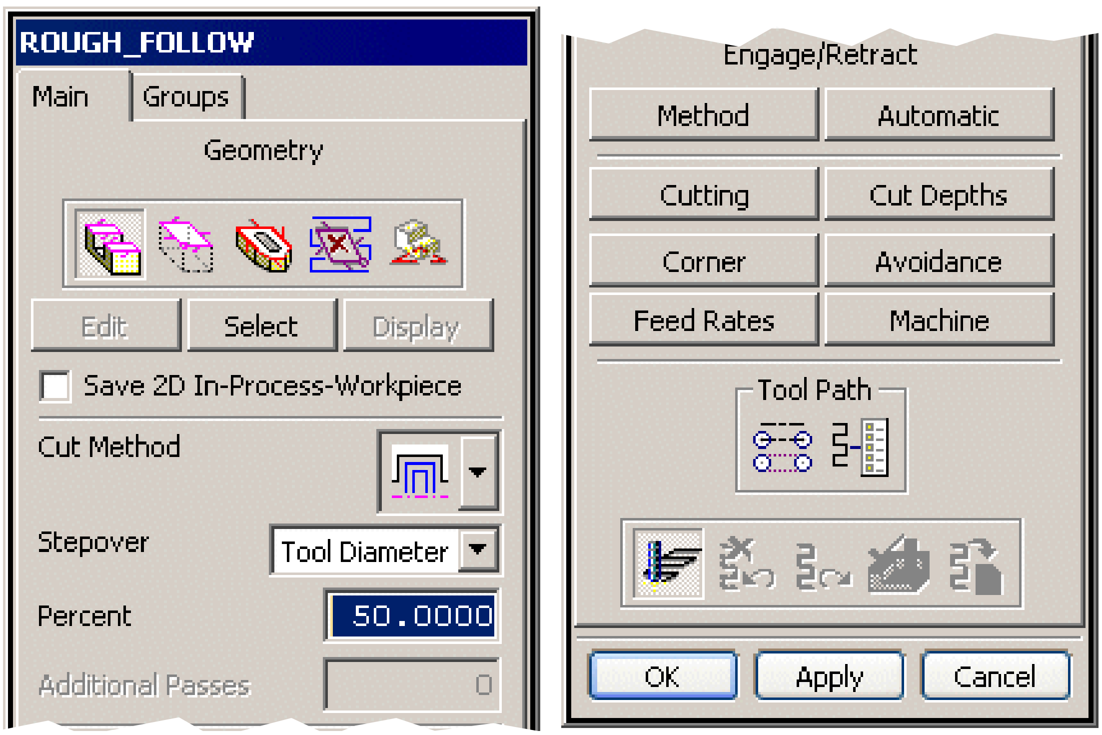
TIP When specifying the operation name, choose the Subtype icon before naming the operation. If you name the operation first and then choose the icon, the name is replaced with the default name associated with the icon.

You will now create the operation.

8

❑ Choose **OK**.

The ROUGH_FOLLOW operation dialog is displayed. Note the option settings used for roughing.

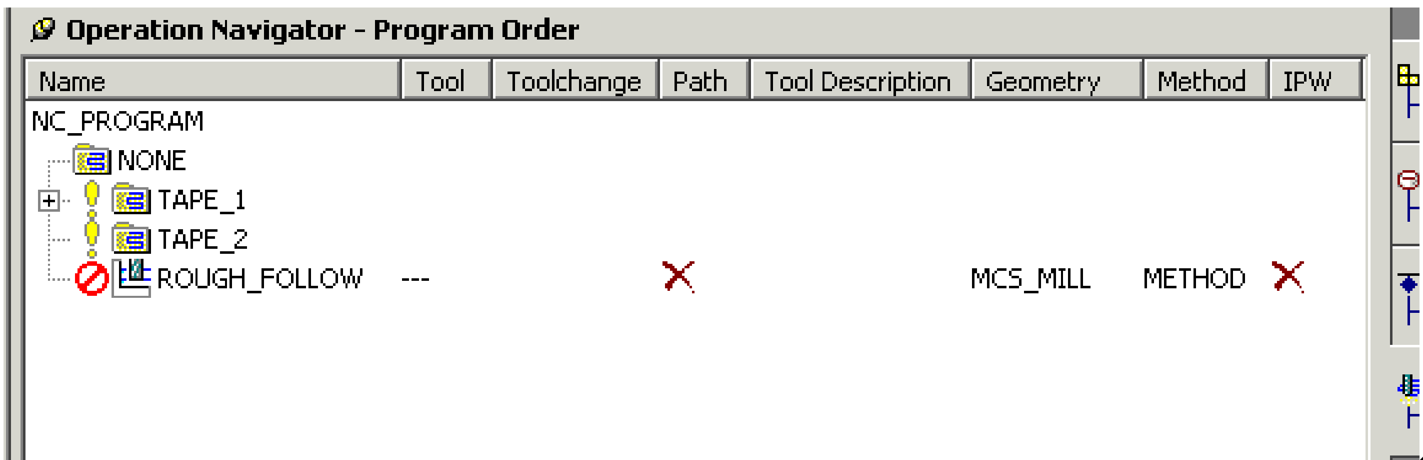


You will not create a tool path, but you will accept the operation settings.

Step 4 View the Operation listing.

❑ Choose **OK** from the bottom of the ROUGH_FOLLOW operation dialog.

Notice the ROUGH_FOLLOW operation is listed and the red symbol prior to the operation name. This indicates that the operation has not be generated.



You will now practice the drag and drop functions within the Operation Navigator.

❑ Expand the Program Parent Group **TAPE_1**.

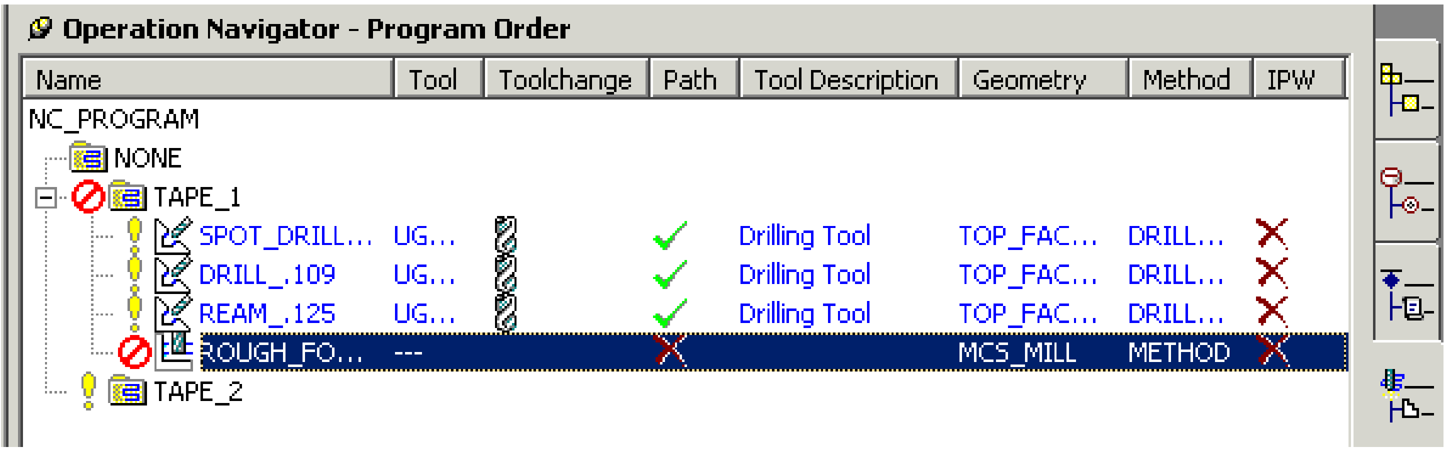
The operations contained within the Parent Group are listed.

Step 5 Move the ROUGH_FOLLOW operation previously created to the Program Parent Group TAPE_1.

❑ If necessary, use MB1 to highlight the **ROUGH_FOLLOW** operation.

❑ Using MB1, drag the operation to **TAPE_1,** then release MB1.

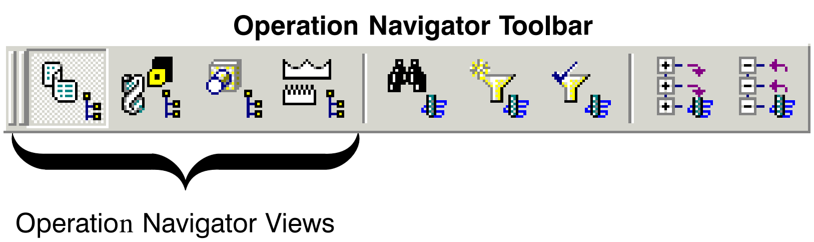
This will move the ROUGH_FOLLOW operation within the Program Order Parent Group TAPE_1.



You will now switch between the various Operation Navigator views.

Step 6 Change the various views that display in the Operation Navigator.

❑ Choose the various Operation Navigator view icons located on the Operation Navigator toolbar and notice the difference in the displayed data.

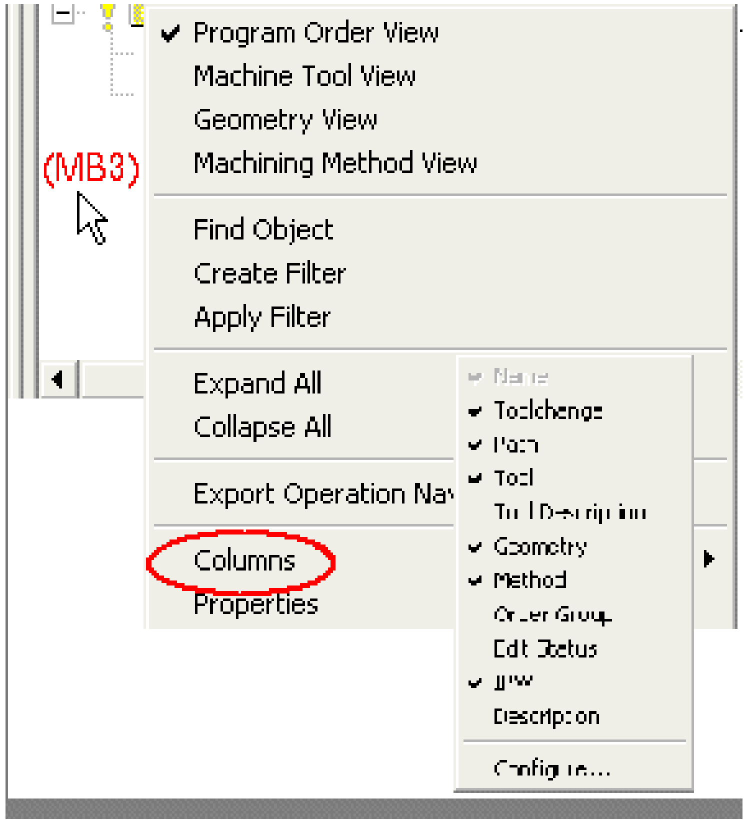


❑ Make sure that the Operation Navigator **Program Order View** is displayed.

You will now add columns to the Program Order view display.

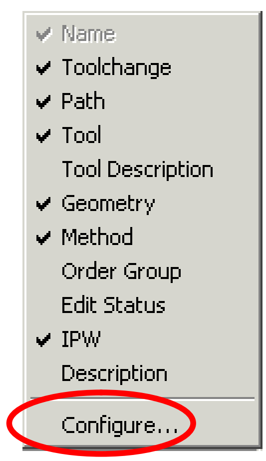
Step 7 Display all columns of data in the Program Order View.

❑ With the cursor on the Operation Navigator background, select **MB3**, then select Columns from the pop-up menu.

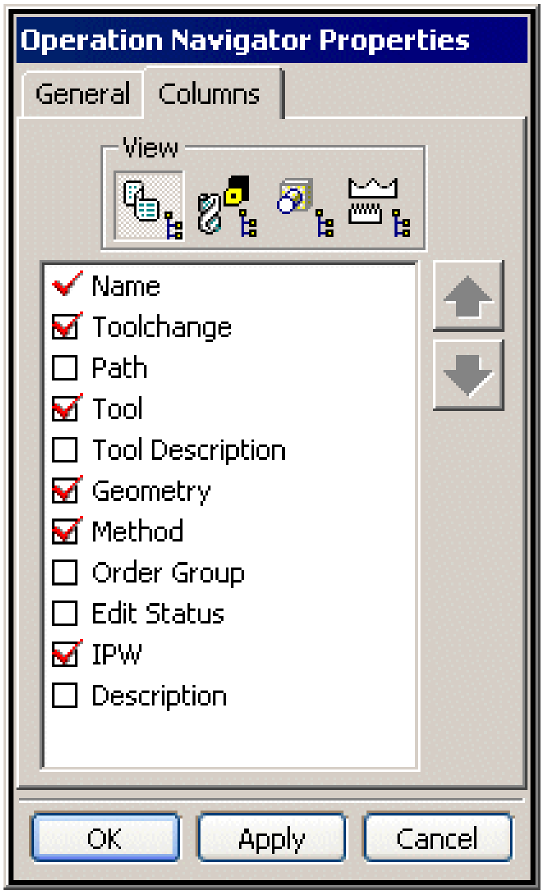


❑ Choose the Configure option from the pop-up menu.

8

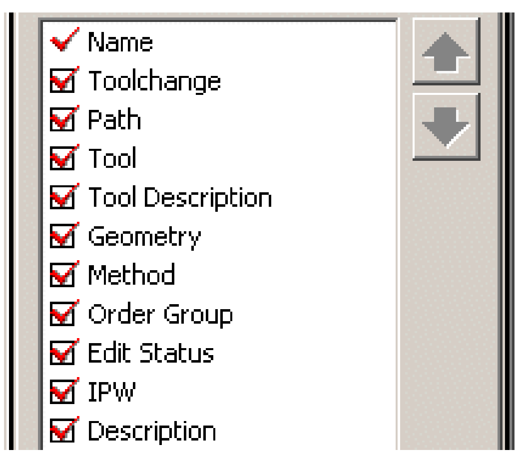


The Operation Navigator Properties page is displayed.



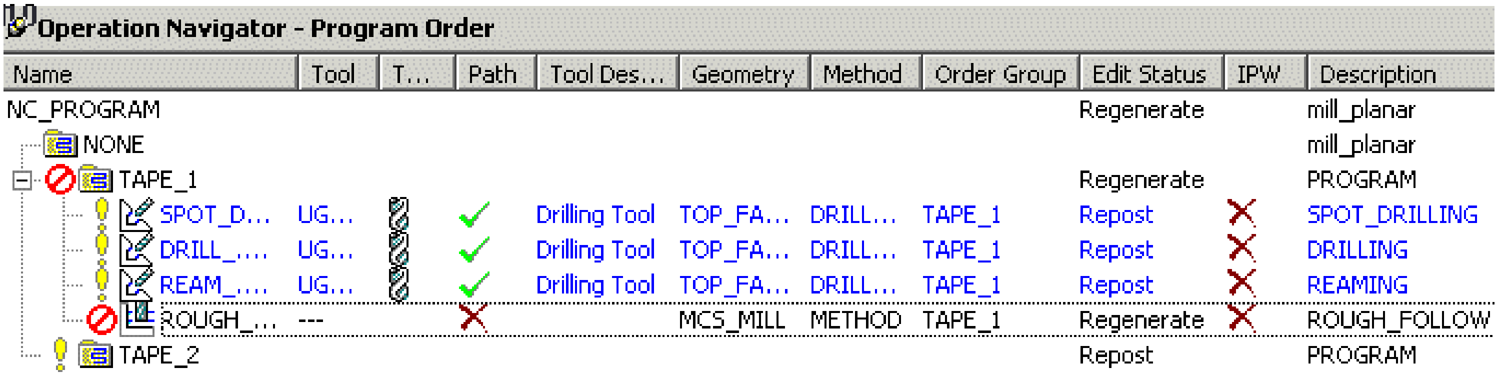
❑ If necessary, change to the Program Order view as shown above.

❑ Turn ON, all columns.



❑ Choose **OK**.

Notice the additional columns and information that is now displayed on the Operation Navigator.

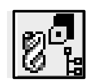


Step 8 Review the various MB3 options that are available.

- ❑ In the Operation Navigator, highlight an operation, and then choose **MB3**.

- ❑ Explore the options.

Step 9 Cut an existing operation and paste it to a new location.

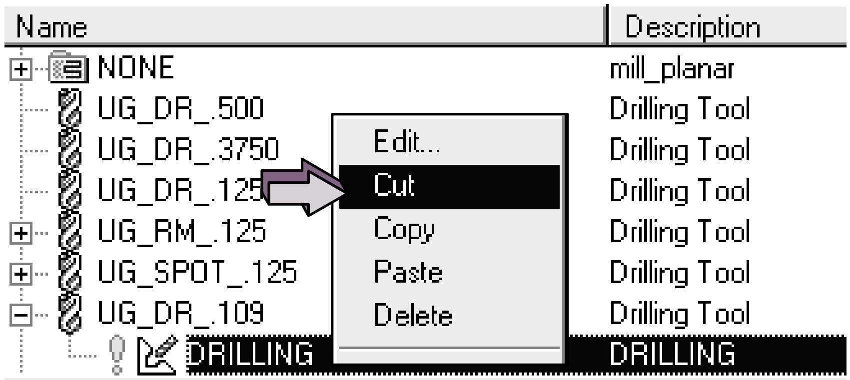
- ❑ Change the view on the Operation Navigator to the Machine Tool View by choosing the **Machine Tool View** icon  on the Operation Navigator toolbar.

  The Machine Tool view is displayed in the Operation Navigator.

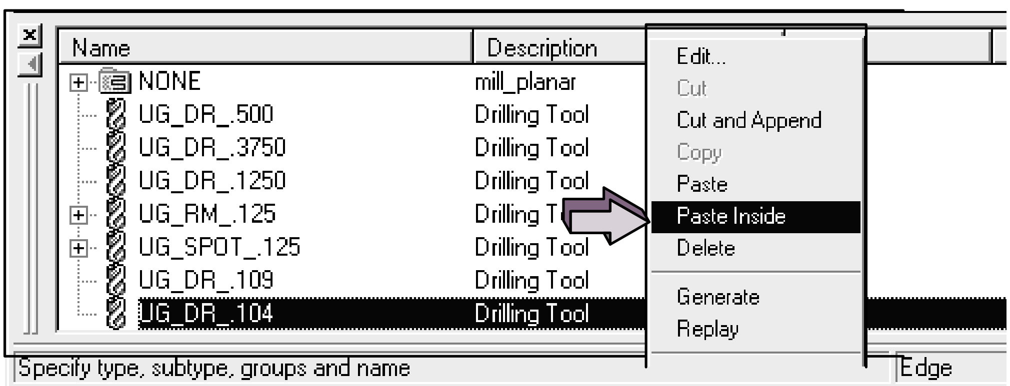
  Assume that the UG_DR_.109 drill did not leave enough material for reaming. Another tool, UG_DR_.104 is a better choice. You can move the operation, used for drilling, under the tool, UG_DR_.104. The operation will then use that tool.

- ❑ Expand **UG_DR_.109**.

- ❑ Highlight the operation **DRILL_.109** and use MB3 to **Cut** the operation.

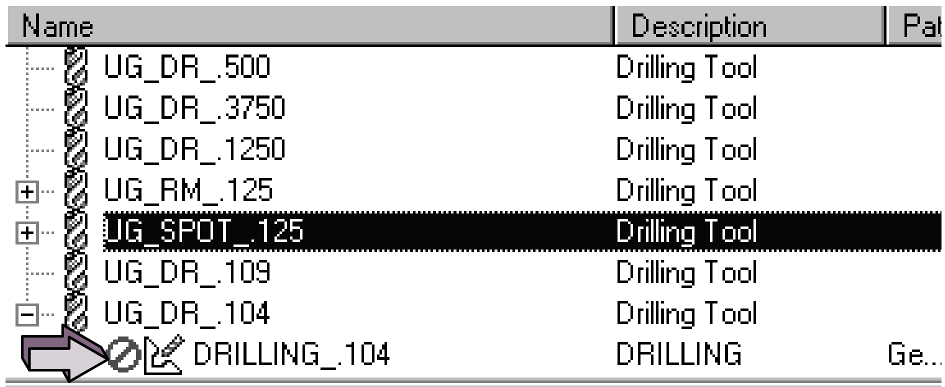


- ❑ Highlight the tool **UG_DR_.104** and use MB3 to **Paste Inside.**



TIP Moving this operation in the Machine Tool, Geometry or Machining Method view does not move the operation in the Program Order view. In this case it only moves the tool used in the operation.

The drilling operation will now use the UG_DR_.104 drill. You will need to re-generate the tool path as indicated by the red symbol next to the operation.



You will now rename an operation.

Step 10 Rename the DRILLING_.104 operation to DRILL_FOR_REAM.

❑ Change the name of the operation by highlighting the operation DRILL_.104, use MB3, select Rename, and key in DRILL_FOR REAM. Optionally, you can highlight the operation name and choose **Tools→ Operation Navigator→ Rename.**

❑ **Save** the part file.

This concludes this activity and also the lesson.

SUMMARY

In this lesson you:

- Became familiar with the functionality and views of the Operation Navigator
- Selectively changed the data displayed in columns on the Operation Navigator
- Used the cut, paste, drag and drop functions of the Operation Navigator
- Moved an operation in the Operation Navigator to use a different tool and inherit its values.
- Renamed an operation

Face Milling

Lesson 9

PURPOSE

This lesson is an introduction to Face Milling which is used to cut one or more planar faces.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Create Face Milling operations
- Cut a single face
- Cut multiple faces
- Avoid cutting open areas on a face
- Use different Cut Methods in face milling operations

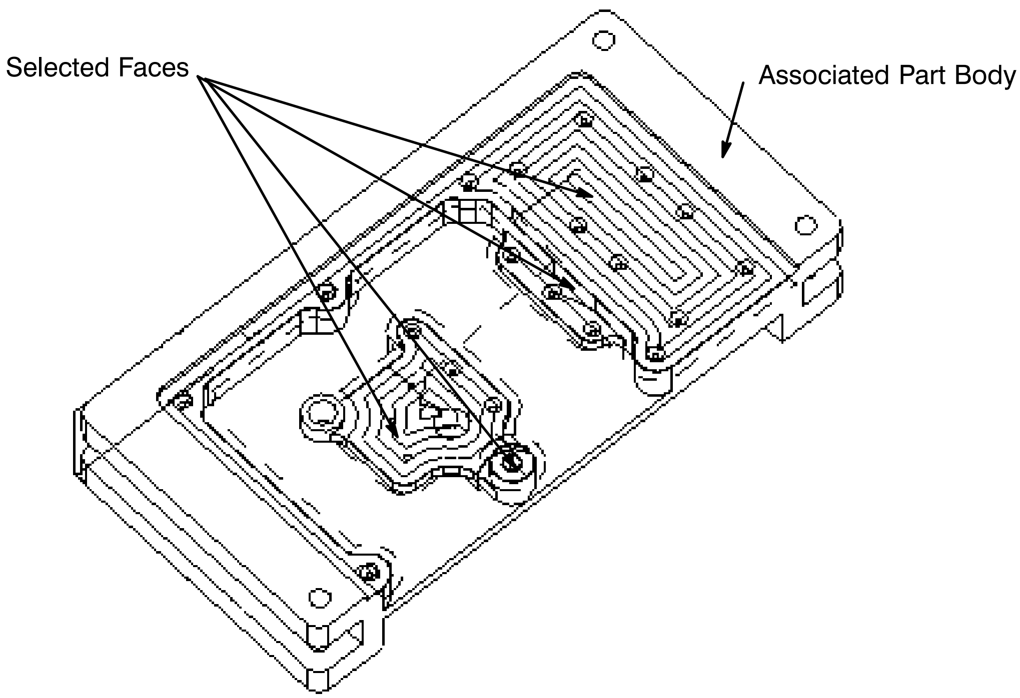
This lesson contains the following activities:

Face Milling

Face Milling is designed to help you quickly and easily create milling tool paths for planar faces. It uses boundaries to define the limits of the machining area, but knowledge of boundaries is not necessary for you to successfully create efficient tool paths.

Face Milling does require that you select the face or faces for machining. The faces must be both planar and perpendicular to the tool axis.

The tool axis is defined by the MCS as a Z-axis vector. Since Face Milling removes material in planar levels with respect to the tool axis, the normal of a face boundary plane must be perpendicular with the tool axis. If not, the face will be ignored during tool path generation.



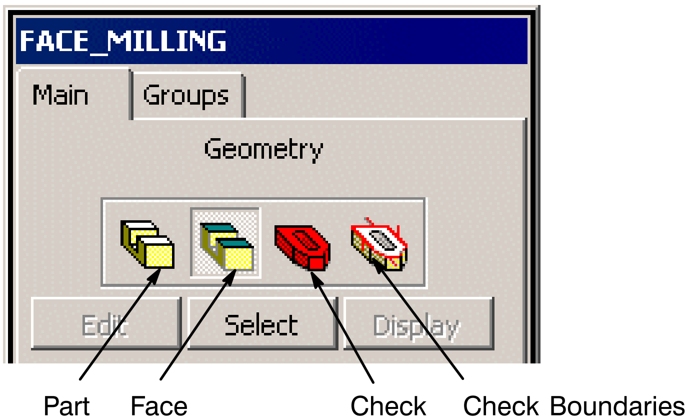
Face Milling requires geometry, a cutting tool and various parameters to generate a tool path. For each selected boundary to be faced, traces are created from geometry, regions are identified and then cut without gouging the part.

Many Face Milling parameters are similar to those that are used in Planar Milling.

Part, Face and Check Geometry

The Face Milling dialog allows you to select:

- Part geometry
- Face geometry
- Check geometry



Part geometry allows you to select bodies that represent the finished part.

Face geometry consists of closed boundaries with inside material indicating the areas to be cut. A face boundary can be created by selecting a:

- planar face
- curves and/or edges

When a face boundary is created from a face, the body associated to the selected face boundary is automatically used as part geometry to avoid any gouging of the part.

A face boundary created from curves, edges or points does not have this association.

All members of a face boundary have *tanto* tool positions. At least one face boundary must be selected to generate a tool path. The normal of a face boundary plane must be parallel with the tool axis.

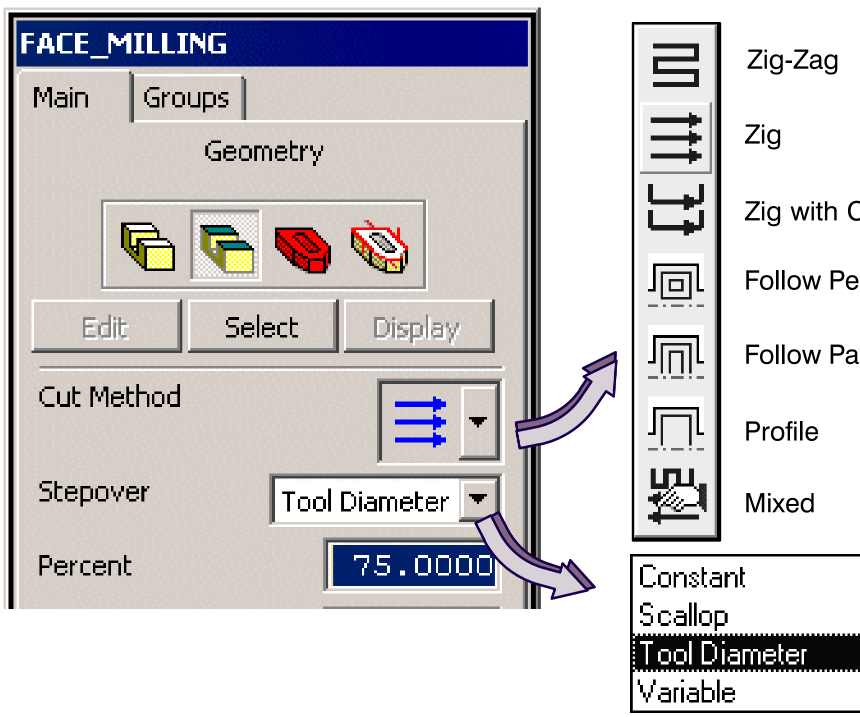
Check geometry consists of closed boundaries that represent clamps or fixtures. All members of a check boundary have tanto tool positions. Material side is defined with respect to the direction of the boundary. It indicates where the material is, either on the inside or outside (measured along the tool axis, starting from the plane of a boundary). The normal of a check boundary plane must be parallel with the tool axis.

Cut Method

Cut Method determines the tool path pattern used to machine cut regions. **Zig-Zag, Zig**, and **Zig with Contour** produce variations of parallel linear cutting moves.

Follow Periphery produces a sequence of concentric cutting passes that progress inward or outward.

Profile produces a single cutting pass that follows the cut region contour. Profile is designed to finish the walls of a part.

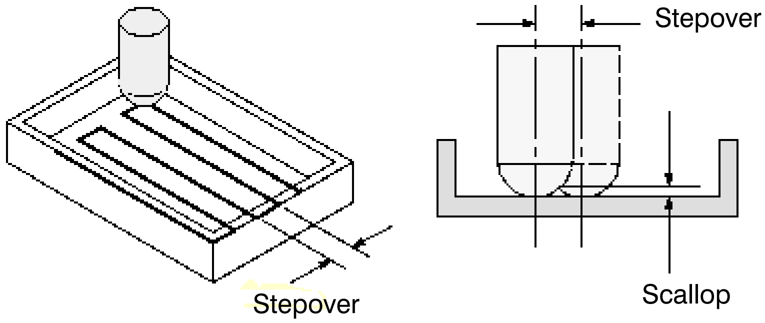


9

Stepover allows the specification of the distance between cut passes. The distance can be specified by entering:

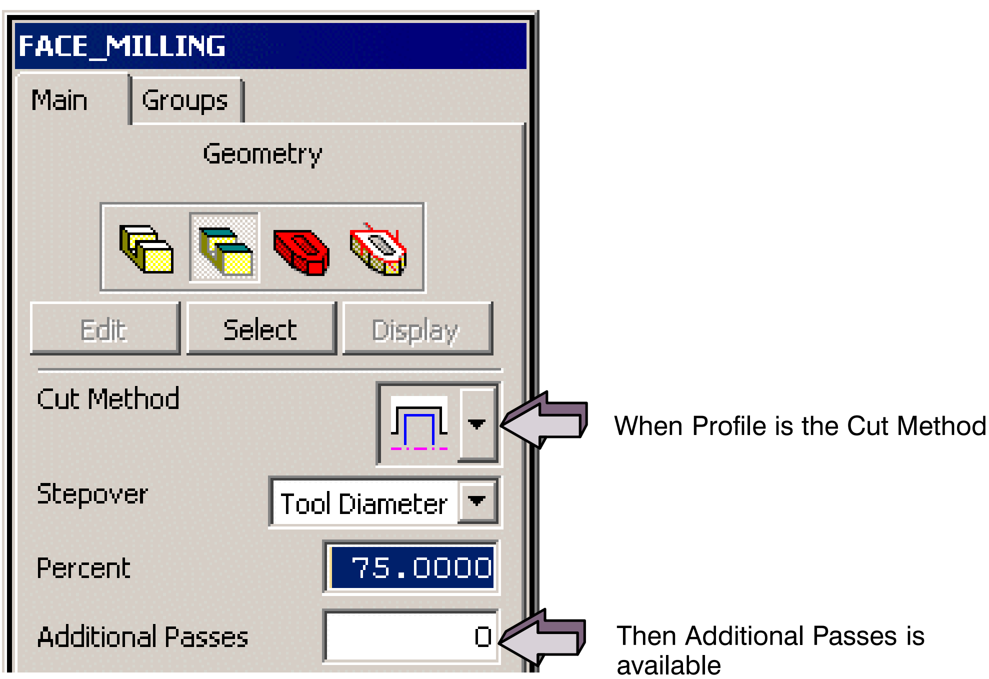
- a constant value
- a percentage of the tool diameter
- a scallop height
- variable stepovers

Variable Stepovers are defined by specifying an allowable range for use in determining the stepover size or by specifying stepover sizes and a corresponding numbers of passes.



Additional Passes

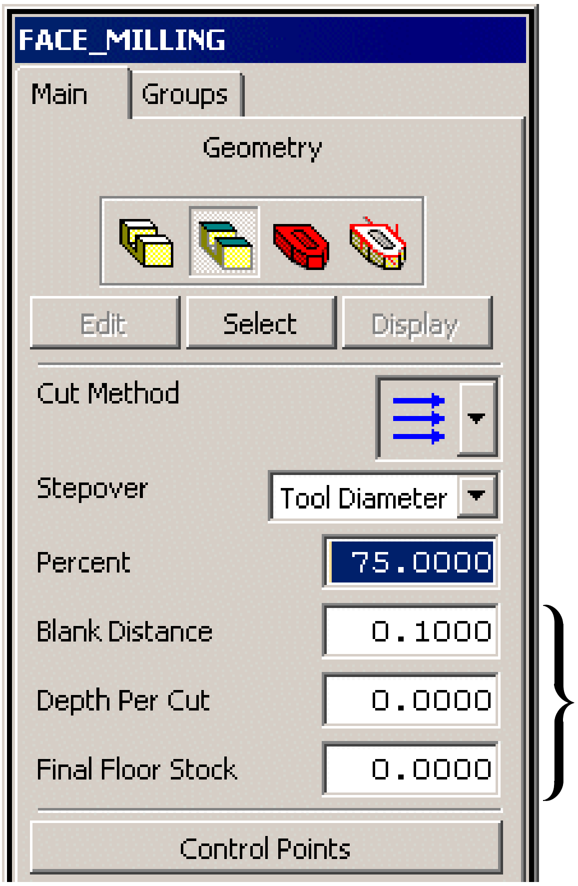
Additional Passes are available only when using a Profile cut method. It allows the removal of material in multiple passes. Additional Passes represents the number of passes in addition to the single pass along the boundary.



Note: For Unigraphics NX, Additional Passes is available only through the Customize Dialog option.

Blank Distance, Depth per Cut, Final Floor Stock

Blank Distance defines the total thickness of material to be removed and is measured above the plane of the selected face geometry along the tool axis.



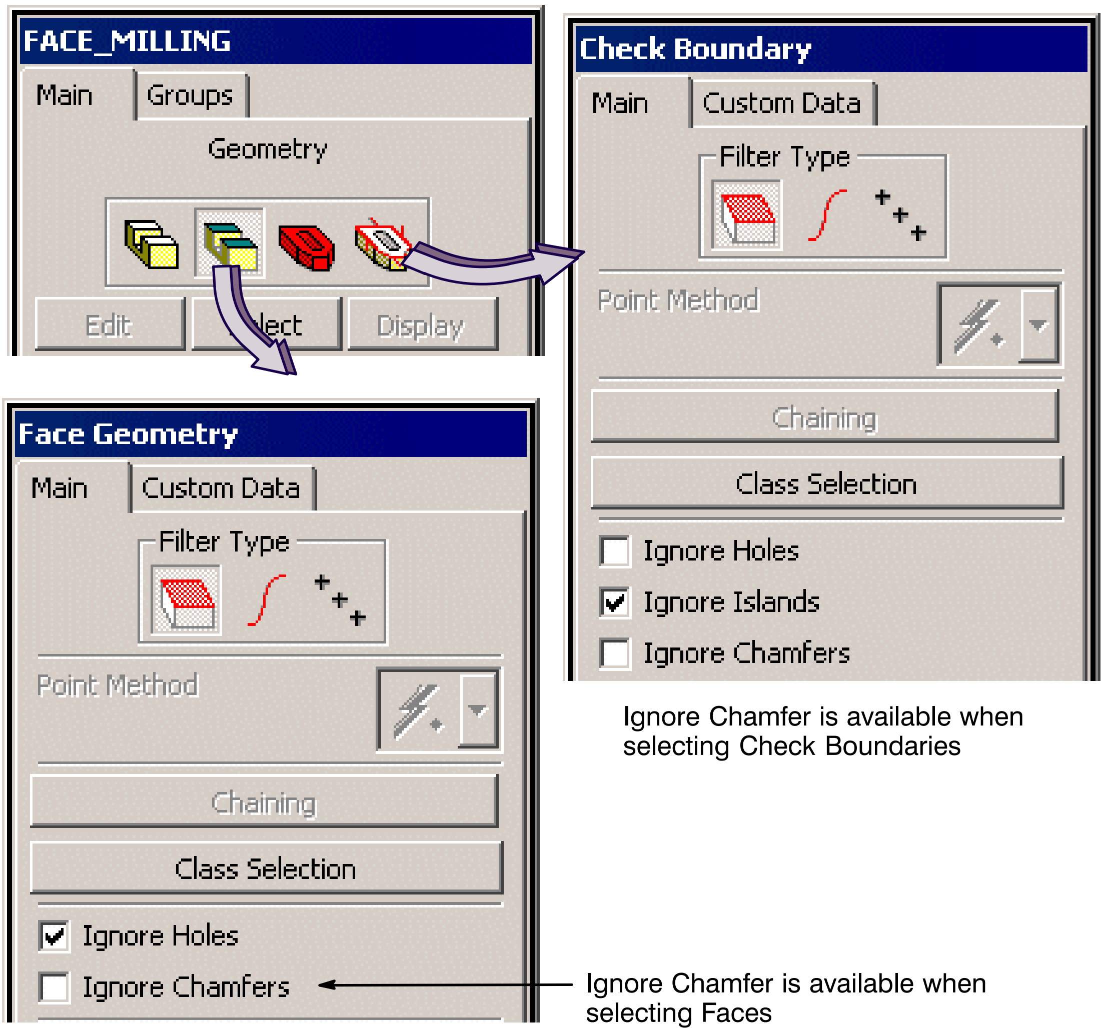
Defines material thickness

Blank Distance is used with **Final Floor Stock** and defines the thickness of material that is left uncut above the face geometry. The total thickness of material to be removed is the distance between the Blank Distance and the Final Floor Stock.

Depth Per Cut equally subdivides the total thickness of material to be removed into numerous levels.

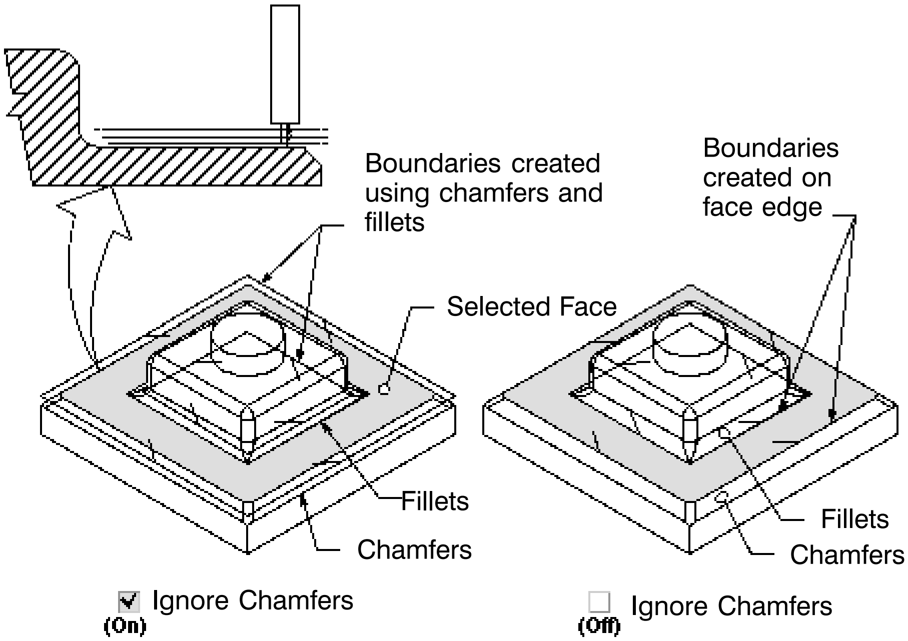
Boundary Construction from a Face and its Chamfers

Ignore Chamfers allows whether or not adjacent chamfers, fillets, and rounds will be recognized when creating boundaries from selected faces.



When Ignore Chamfers is toggled OFF, boundaries are created on the edges of the selected faces.

When toggled ON, boundaries are created to include chamfers, fillets, and rounds adjacent to selected faces.



If other objects are to inherit these boundaries, you would use Blank boundaries in a MILL_BND Geometry Parent Group.

Face Milling Review

Face Milling:

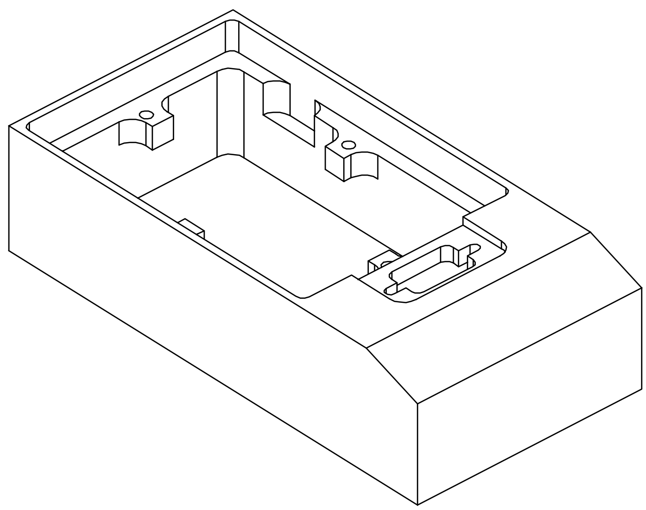
- is designed to easily create milling tool paths for planar faces
- uses boundaries to define the horizontal limits of the machining area, but user knowledge of boundaries is not necessary to successfully create efficient tool paths
- requires you to select the face or faces for machining
- faces can be selected either from within the operation or they can be inherited from a boundary geometry Parent Group
- the faces must be both planar, and perpendicular to the tool axis

Activity 9—1: Face Milling - Basics

This activity includes a demonstration of basic Face Milling functionality such as ignore holes, ignore chamfer, across void traverse cutting option and manual cut pattern mode.

Step 1 Open the part file and enter the Manufacturing application.

❑ Open the part file **mmp_frame_mfg_1.prt.**

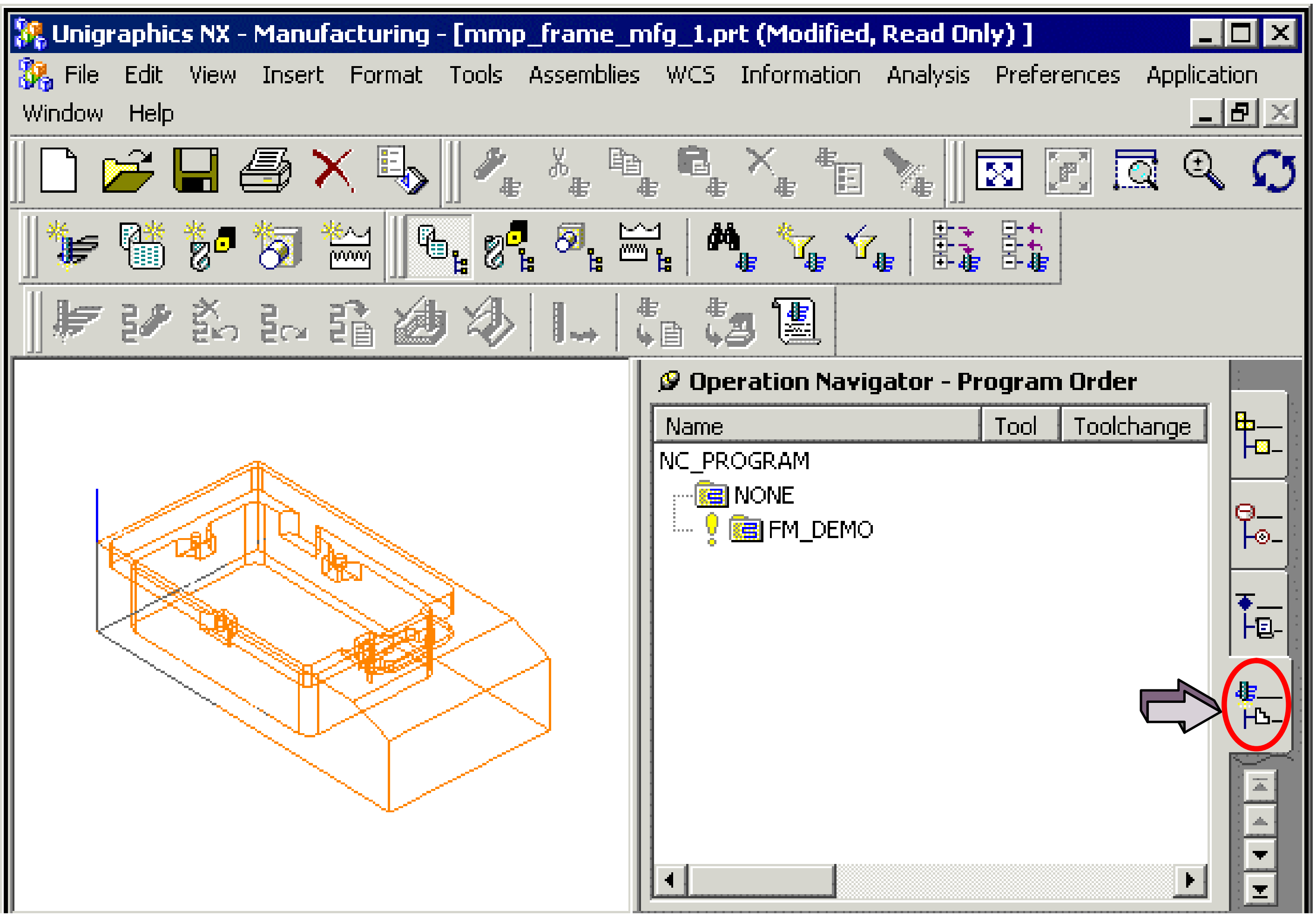


❑ Choose **File→ Save As**, and save the part as *****_frame_mfg_1.prt**, where *** represents your initials.

❑ Choose **Application→Manufacturing**.

Step 2 Activate the Operation Navigator.

❑ Choose the **Operation Navigator** tab from the resource bar.



The Operation Navigator is displayed in the graphics window.

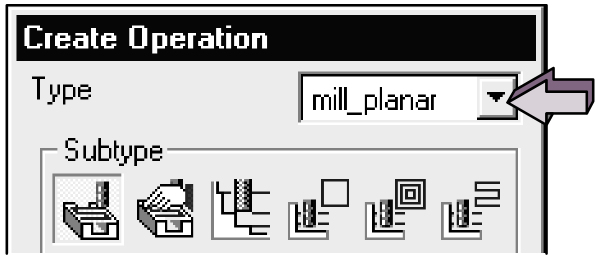
Step 3 Create a face milling operation to mill the top face of the model.

9

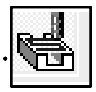
- ❑ Highlight the FM_DEMO Program Parent Group in the Operation Navigator.
- ❑ Using MB3, choose **Insert→Operation**.

The Create Operation dialog is displayed.

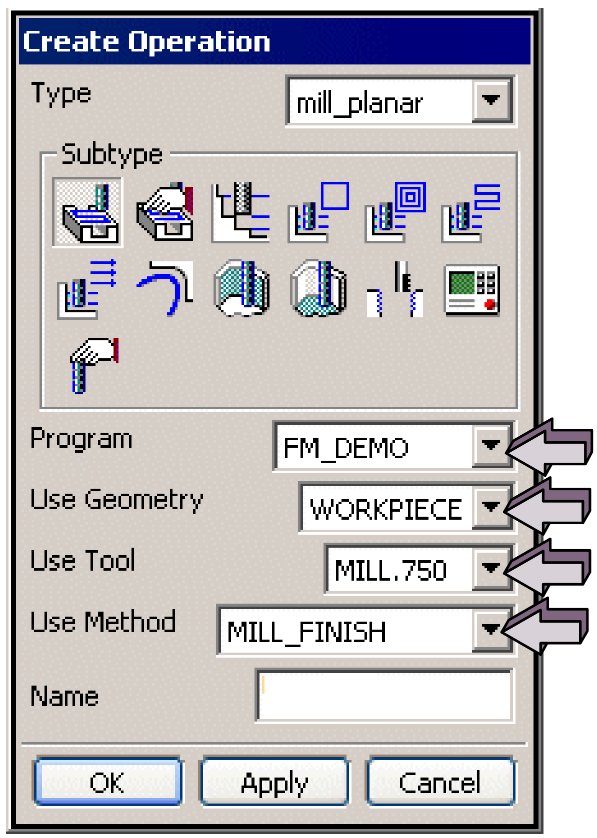
- ❑ If necessary, change the Type to **mill_planar**.



Change the type to mill_planar

❑ In the Create Operation dialog, choose the **FACE_MILLING** icon. 

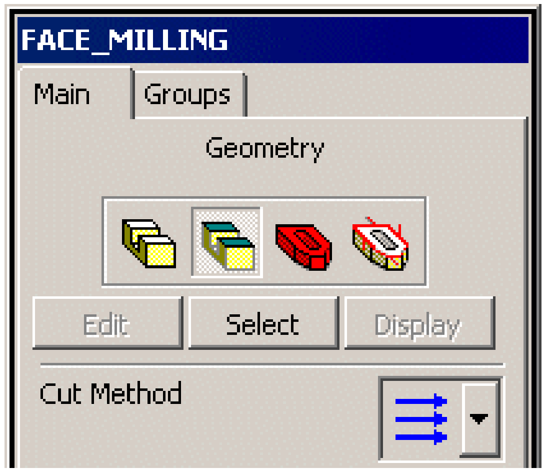
❑ From the Parent Group lists, set the Parent Groups to:



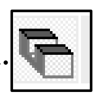
❑ In the Name field, enter **FACE_TOP**.

❑ Choose **OK**.

The FACE_MILLING dialog is displayed.

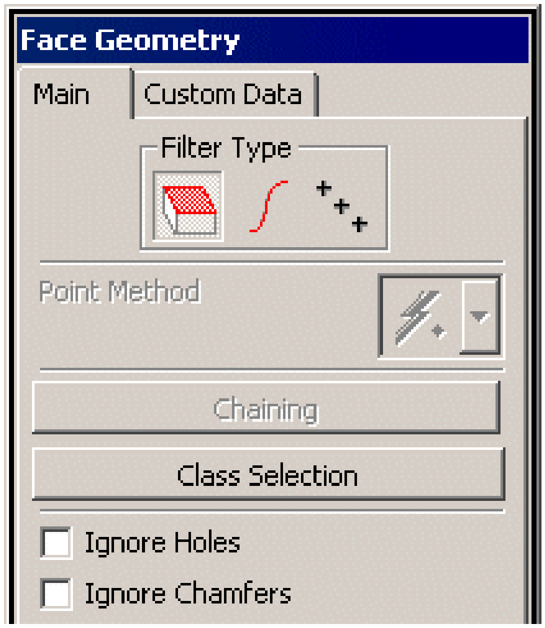


Step 4 Choose the Face Geometry.

❑ In the Geometry section of the dialog, choose the **Face** icon. 

❑ Choose the **Select** button.

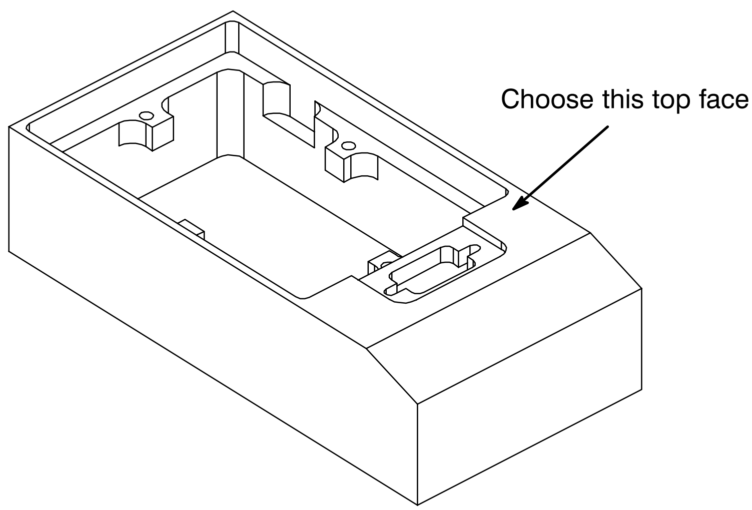
The Face Geometry dialog is displayed.



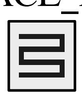
❑ Toggle OFF **Ignore Holes**.

When toggled ON, the system will ignore any voids in a face. Turning this option off instructs the system to create boundaries around any voids found in the selected face.

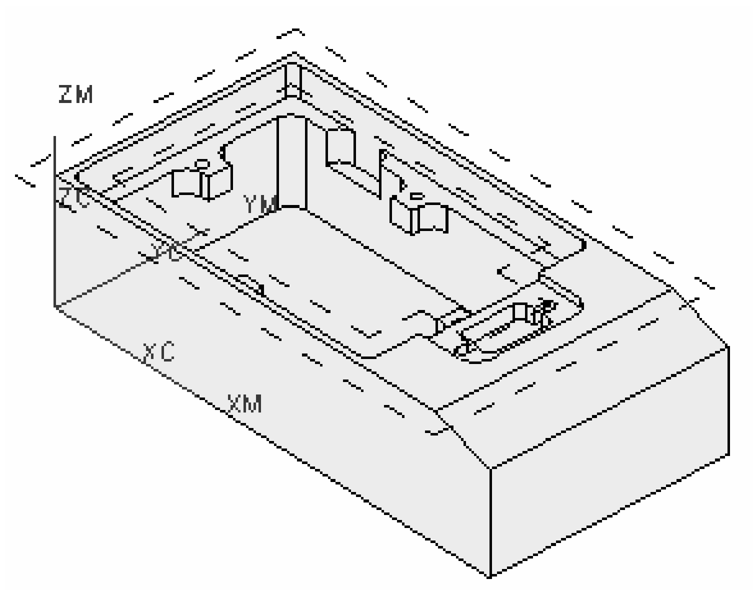
❑ Select the top face as shown.



❑ Choose **OK**.

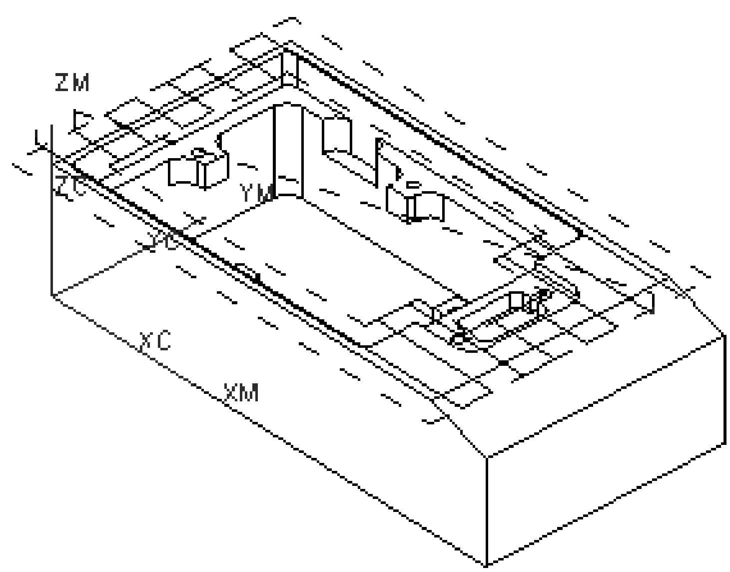
❑ In the FACE_MILLING dialog , set the Cut Method to **Zig-Zag**. 

❑ Choose **Generate.**



Two regions are displayed. The outer region is indicated by dashed magenta lines, and the inner region is indicated by dashed yellow lines. These lines indicate where the center of the cutter will machine to.

❑ Choose **OK.**



The tool path machines the top face, but does not machine over the pocket area or the large chamfer area on the right side of the part due to the Ignore Holes setting.

If this were the first cut on the part, it would be desirable to fully machine the top face. In that case, Ignore Holes would have been left ON. To demonstrate that, go back and re-select the top face again.

❑ In the Geometry section of the FACE_MILLING dialog, choose **Reselect**.

❑ Choose **OK** to the warning dialog.

❑ This time, toggle ON **Ignore Holes** and **Ignore Chamfers**.

❑ Choose the top face again.

Notice that the boundary includes the chamfer this time.

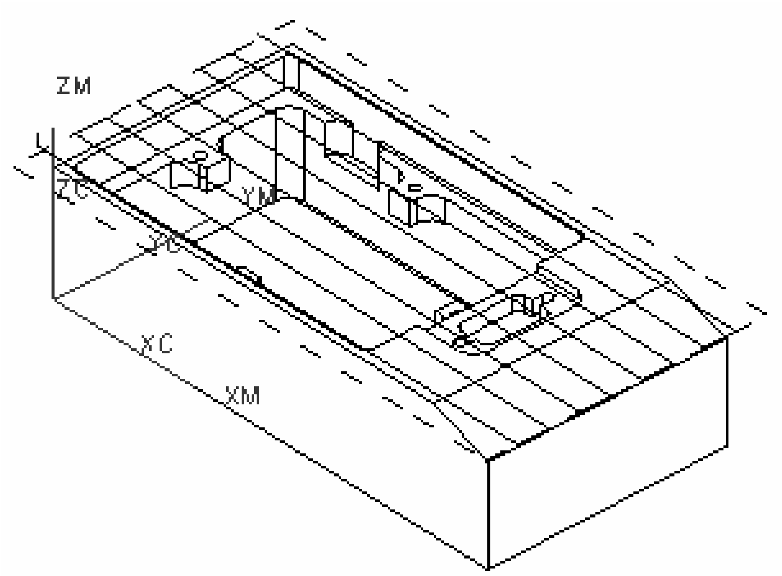
❑ Choose **OK**.

❑ Choose **Generate**.

There is only one region displayed.

❑ Choose **OK** to continue.

This time the entire face is fully machined.



❑ Choose **OK** to save the operation.

❑ **Save** the part.

This completes the activity.

Face Milling and Surrounding Geometry

You can machine faces inside of a part. The Face Mill option will determine which faces to cut or avoid without gouging other geometry.

The following activity will show some of the strengths of Face Milling's ability to determine the use of surrounding geometry.

Activity 9—2: Face Milling — Interior Geometry

In this activity, you will machine a face inside of the part. You will copy the previously created operation and then modify that copy to machine a different face of the part.

Step 1 Copying the previous operation.

- ❑ Continue using the part *****_frame_mfg_1.prt.**
- ❑ If necessary, switch to the **Program Order View** of the Operation Navigator.
- ❑ Highlight the **FACE_TOP** operation, using MB3, choose **Copy**.
- ❑ Highlight the **FACE_TOP o**peration, using MB3, choose **Paste**.

A duplicate operation to the original is created, with the name **FACE_TOP_COPY**.

Step 2 Modifying the new operation

For this new operation, you will:

- Change the name
- Change the tool
- Change the face geometry
- Modify the stepover values
- Generate the operation with the new settings

First, you will change the name.

- ❑ Highlight the **FACE_TOP** operation, using MB3, choose **Rename**.

9

❑ Key in the name **INSET_FACE**.

NOTE It does not matter whether uppercase or lowercase letters are entered. All characters are converted to uppercase. Special characters and/or spaces are not allowed. Also, while operation names may contain numeric characters, they must begin with alphabetic characters.

You will now edit the copied operation.

❑ Double click on the **INSET_FACE** operation.

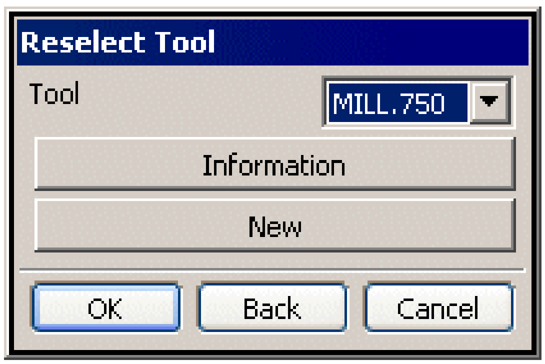
Step 3 Changing the Tool.

❑ Choose the Groups property page (tab) from the FACE_MILLING dialog.

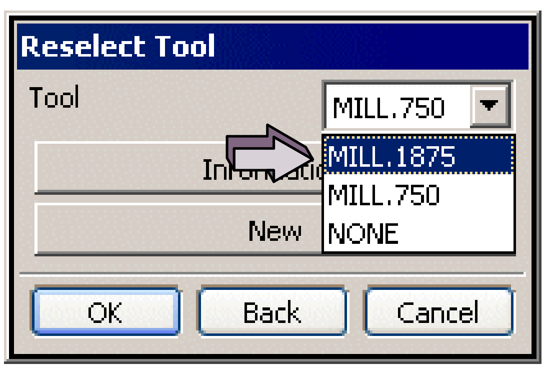
❑ If necessary, select the **Tool** Parent Group object.

❑ Choose **Reselect.**

The Reselect Tool dialog is displayed.



❑ Choose tool **MILL.1875** from the Reselect Tool dialog.



❑ Choose **OK.**

You will now reselect the face that is to be machined.

Step 4 Changing the Face to be machined.

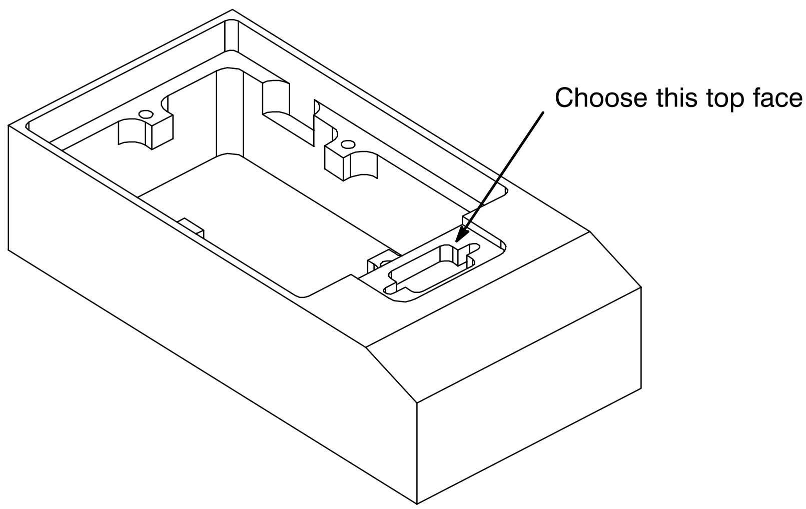
❑ Choose the Main property page (tab).

❑ In the Geometry area,of the dialog, select the **Face** icon, then choose **Reselect**.

❑ Choose **OK** to the warning dialog.

❑ Toggle Ignore Holes **OFF**.

❑ Select the face of the model.



❑ Choose **OK**.

Step 5 Changing the Stepover and generating the operation.

9

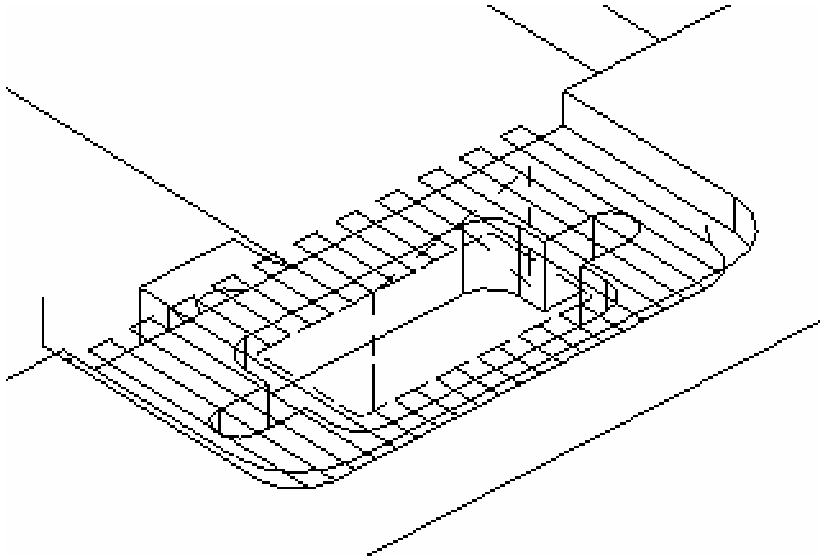
❑ Change the Percent field to **45.**

The default stepover is Tool Diameter. This means that each new zig-zag pass will occur at a value that is 45% of the tool's effective cutting diameter.

The effective cutting diameter is the full diameter of the tool minus the doubled value of the tool radius, if any.

For example, a 1" diameter tool with a .25" corner radius has an effective cutter diameter of 0.5" (1"– [.25" * 2] = 0.5"). Therefore, a 45% stepover for that tool is 0.225" (0.5" * .45 = 0.225"). Each new zig-zag pass will be 0.225" from the previous one.

❑ Choose **Generate.**



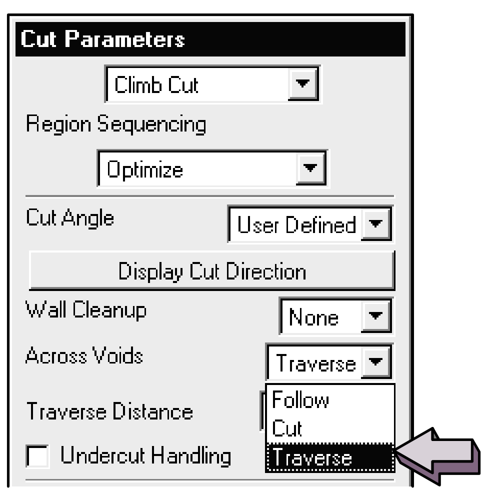
Notice that the tool path avoids machining the center void area.

Step 6 Transversing the void area.

One cut method that is available is to travel across voids at a faster feed rate. You will now change parameters to accomplish this.

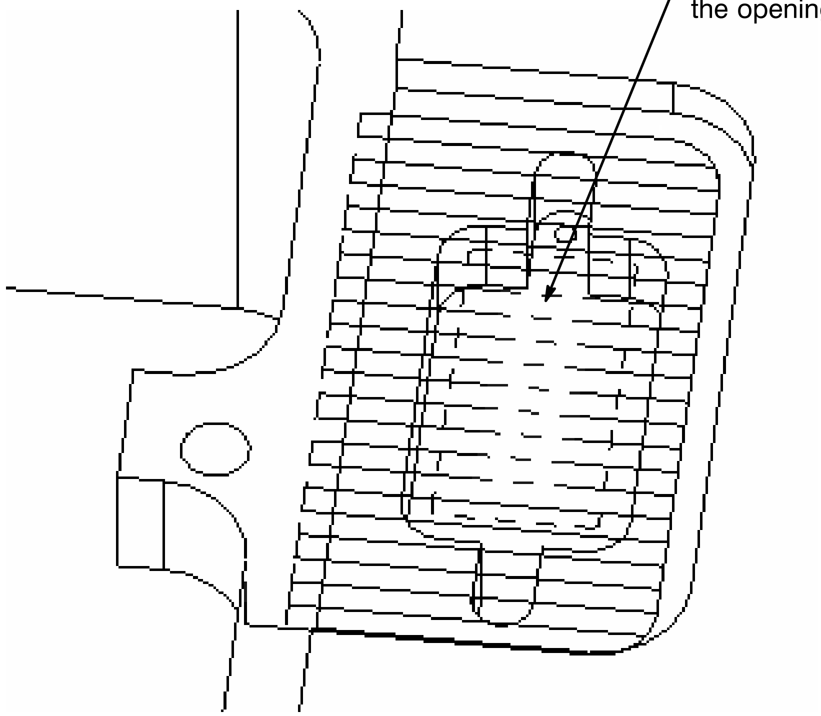
❑ In the FACE_MILLING operation dialog, choose **Cutting**.

❑ In the Cut Parameters dialog, choose **Traverse** from the Across Voids list.



❑ Choose **OK**.

❑ Choose **Generate.**



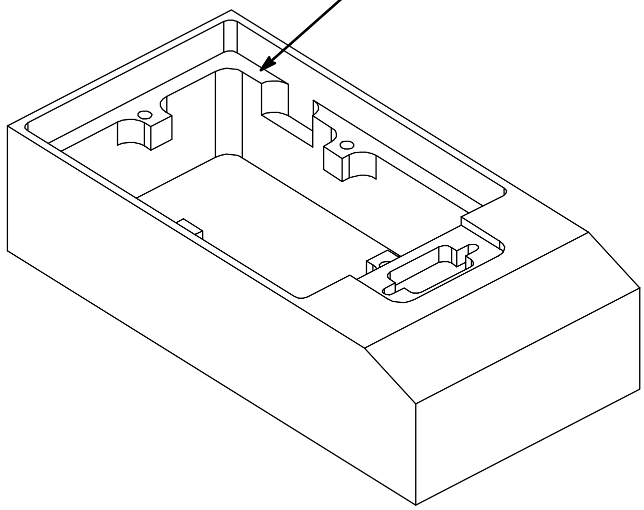
This setting results in the cutter cutting across the void area at the quicker traversal feed rate (displayed as dashed lines).

This operation is aware of the holes on the face with Ignore Holes toggled to **OFF**. You turned on the additional setting of Crossing Voids at a faster traversal feed rate.

Face Milling is ideal for generating a tool path on faces with varied shapes.

❑ Choose **OK** to accept the changes to the INSET_FACE operation.

❑ On your own, create another Face Mill operation and select the inner ledge face. Use the steps outlined previously as a guide.



❑ **Save** the part file.

This completes this activity.

Machining Multiple Faces in One Operation

In some situations it may be desirable to machine several faces in a single operation and to specify a separate cut pattern for each face. This can be accomplished by using the Mixed cut pattern.

Here are the steps required for using this option:

- Create a new Face Milling operation
- Select the Faces to be machined
- Set the Cut Method to Mixed
- Generate the tool paths

The Face Milling operation generation pauses at each region allowing you to set the cut pattern for each individual face.

Cut Parameters

The Cut Parameter, Region Sequencing, uses four methods of automatically and manually specifying the order in which cut regions are machined. The four methods are:

Standard allows the processor to determine the order in which cut regions are machined. The order in which cut regions are machined can be arbitrary and inefficient when using this option.

Optimize orders the machining of cut regions based on the most efficient machining time. The processor determines the order that requires less crossing back and forth between regions and a shorter overall distance when traversing from one region to another.

Follow Start Points and Follow Predrill Points orders the machining of cut regions based on the order in which Cut Region Start Points or Pre-Drill Engage Points were specified. These points must be Active for Region Sequencing to use them. If a point is specified for every region, the processor follows the exact order of point specification. If a point is not defined for every region, the processor finds a sequence of machinable regions that best follows a chain of line segments connecting the specified points. Using either the centroid of closed regions or start points of open regions, the processor projects each point to the chain.

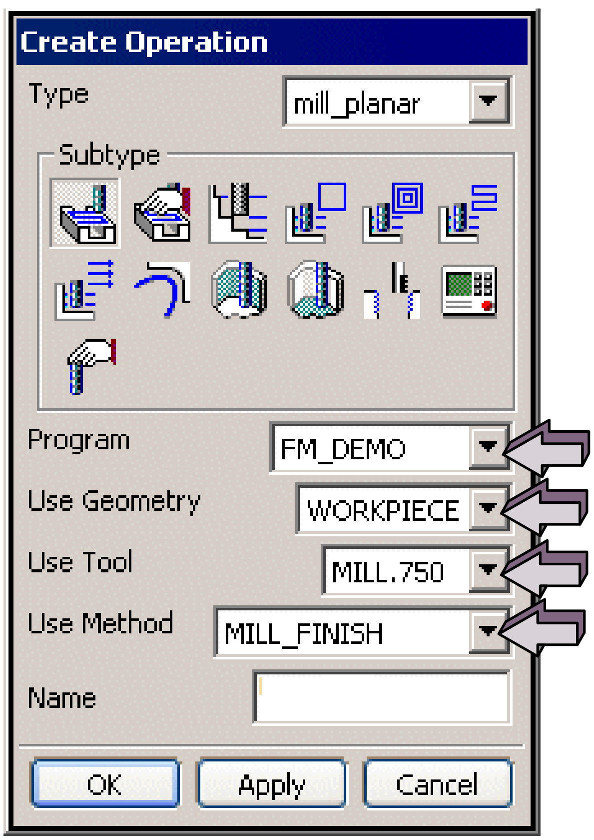
The following activity will guide you through the process of machining multiple faces in one operation.

Activity 9–3: Face Milling – Utilizing Mixed Cut Pattern

In this activity, you will use the Mixed Cut Pattern to vary the tool motion over different faces.

Step 1 Open the part.

- ❑ Continue using the part ***_frame_mfg_1.prt.**
- ❑ Choose the **Create Operation** icon.
- ❑ If necessary, set the Type to **mill_planar**.
- ❑ Select the **Face Milling** icon.
- ❑ Set the following Parent Groups:



- ❑ Enter **multiple_faces** in the Name field.

❑ Choose **OK**.

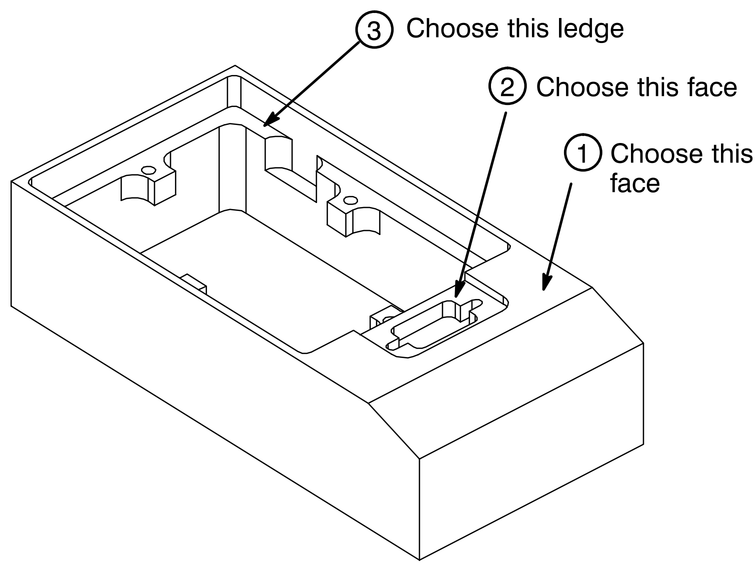
This time, instead of selecting a single face to machine, choose several. The only requirements are that the faces are planar, and parallel to each other.

❑ In the Geometry section, choose the **Face** icon.

❑ Choose **Select**.

❑ Turn OFF **Ignore Holes**.

❑ Select the three faces in the order shown.



❑ Choose **OK**.

9

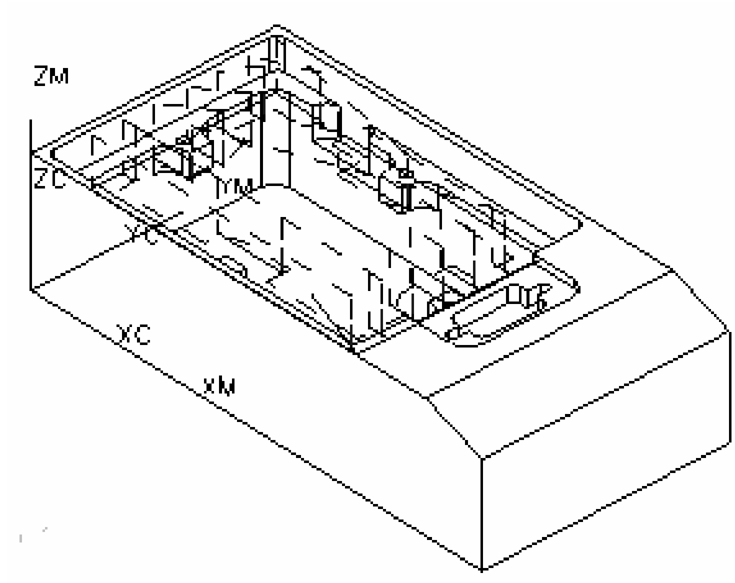
Note that in order for the sequence of tool paths to be the same as the sequence of face selection, the Cutting parameter, Region Sequencing, must be changed from **Optimized** to **Standard**.

❑ Choose the **Cutting** button.

❑ Change the Region Sequencing parameter to **Standard**.

❑ Choose **OK**.

❑ **Generate** the operation.

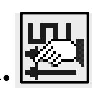


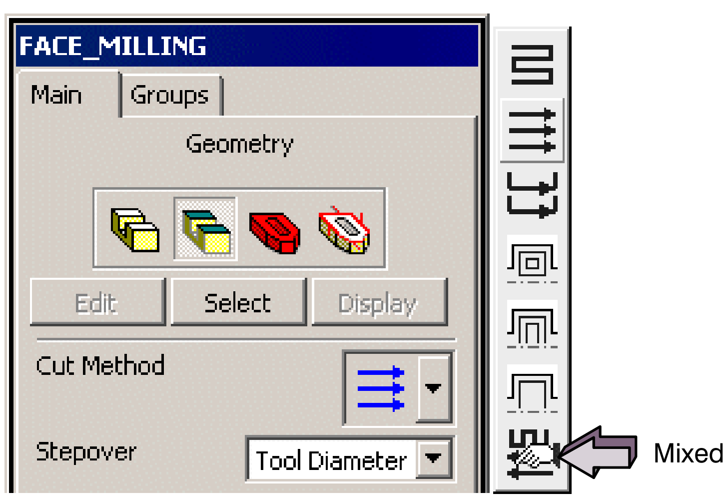
The tool path is generated on each face starting on the ledge (as shown), using the default Zig pattern.

❑ Choose **OK** to the Display Parameters dialog as necessary to generate the tool path.

Step 2 You will now change the cut pattern and regenerate the tool path.

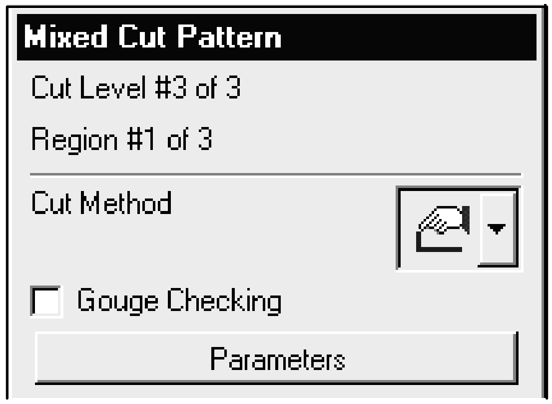
Change the Cut Pattern to Mixed.

❑ **Reject** and change the Cut Method from Zig to **Mixed.** 



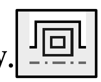
❑ Choose **Generate**.

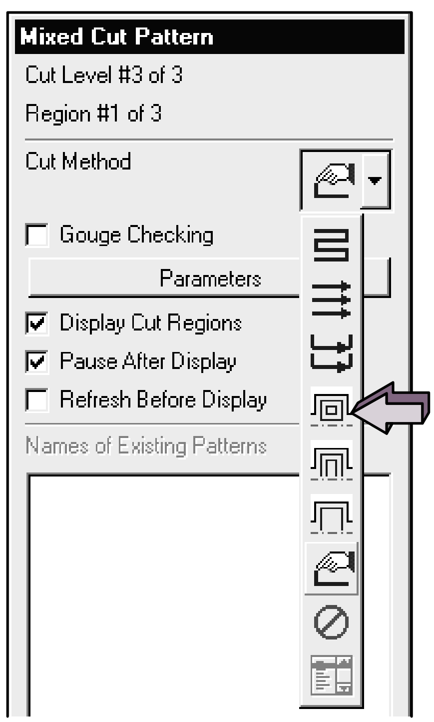
The Mixed Cut Pattern dialog is displayed.



The number of Regions displayed is #1 of #3. You selected three faces and there are three regions.

The number of Cut Levels is also three. You did not specify any Cut levels within a Region. The tool will cut the entire region in one cut, then move to the next region, and perform the cut.

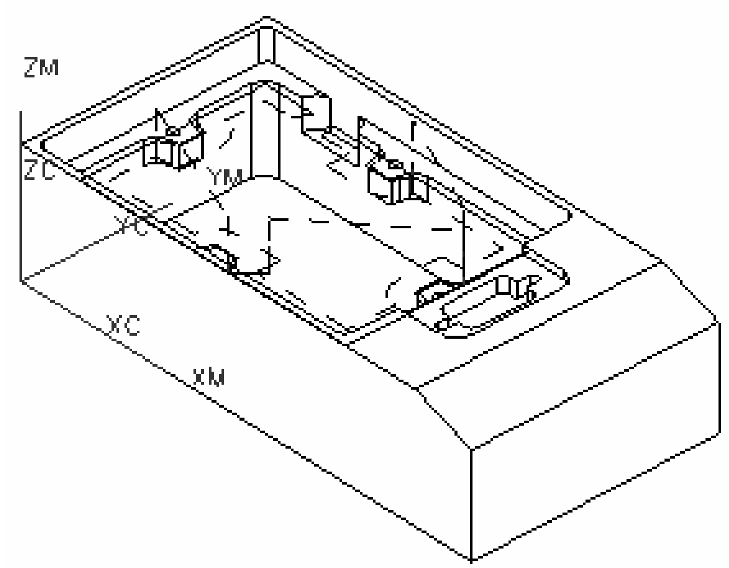
❑ Change the Cut Method to **Follow Periphery**. 



This will force the currently displayed region to be cut using the Follow Periphery cut pattern.

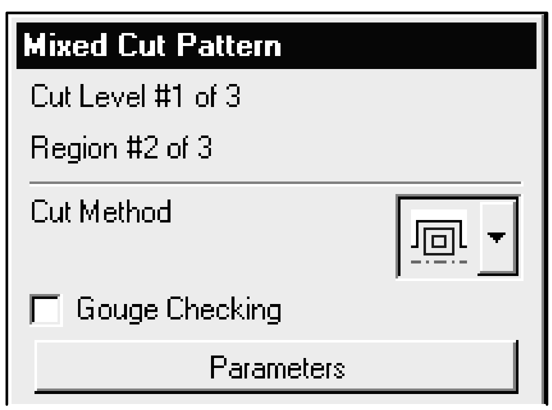
❑ Choose the **Generate** icon at the bottom of the Mixed Cut Pattern dialog.

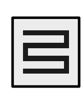
The Follow Periphery cut pattern is generated at the current region.



❑ Choose **OK** to accept the cut**.**

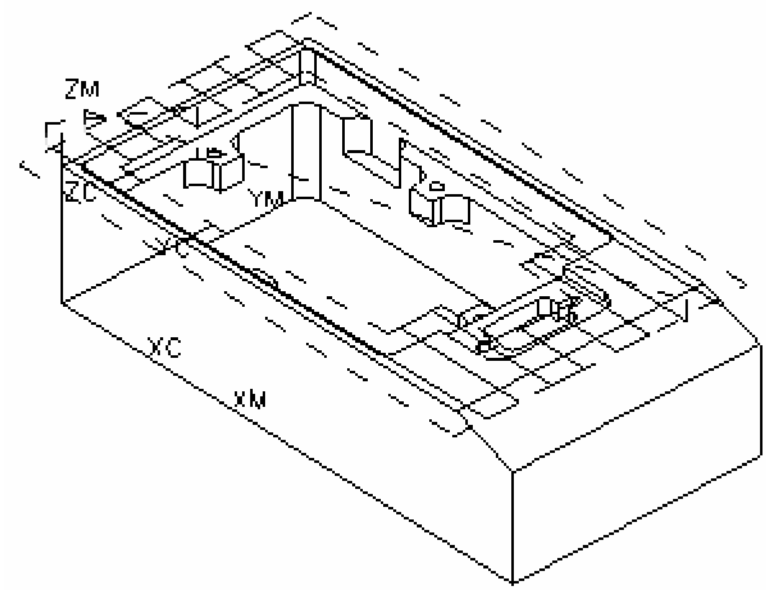
The first cut pattern is placed in the list, and the cut pattern for the next region will be created.



❑ Change the Cut Method to **Zig-Zag**. 

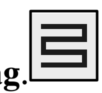
❑ Choose **Generate**.

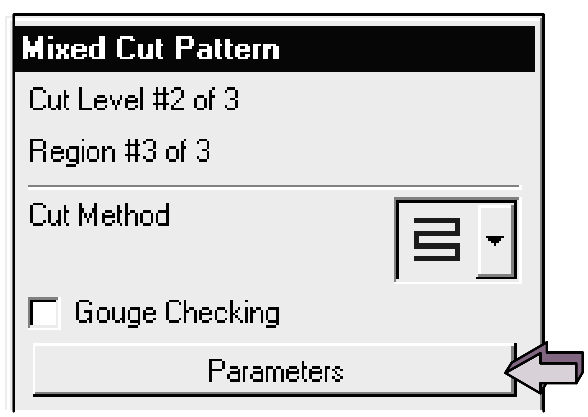
The Zig-Zag cut pattern is generated at the current region.



- ❑ Choose **OK** to accept the cut.

For the last region, change some of the parameters.

- ❑ If necessary, set the Cut Method to **Zig-Zag**. 
- ❑ Choose the **Parameters** button.



- ❑ Change the Percent field to **25**.
- ❑ Change the Cut Angle to **User Defined**, even if it already is set to User Defined.

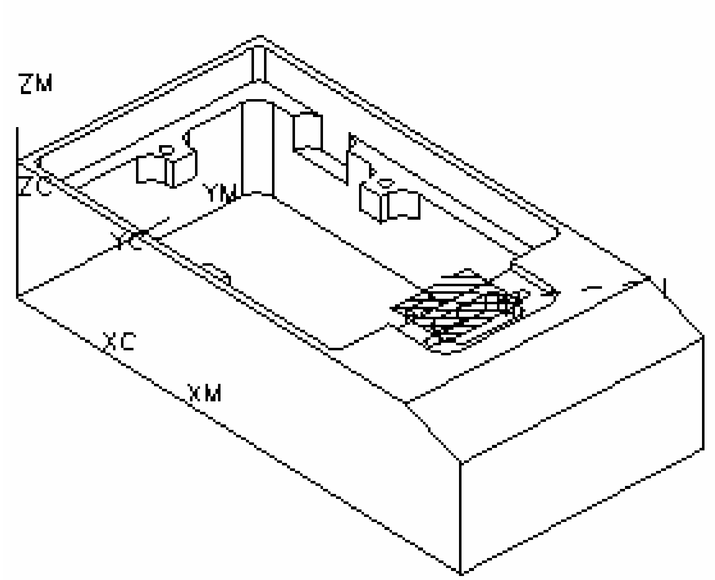
❑ Key in **45**, in the Degrees field.

❑ Choose **OK** to accept the new cut angle.

❑ Choose **OK** to accept the changes to the Parameters dialog.

❑ Choose **Generate**.

All levels have been generated. The new cut pattern is displayed on the current region.



❑ Choose **OK**.

You are returned to the FACE_MILL dialog. Replay the tool path that you just created.

❑ Choose **Replay.**

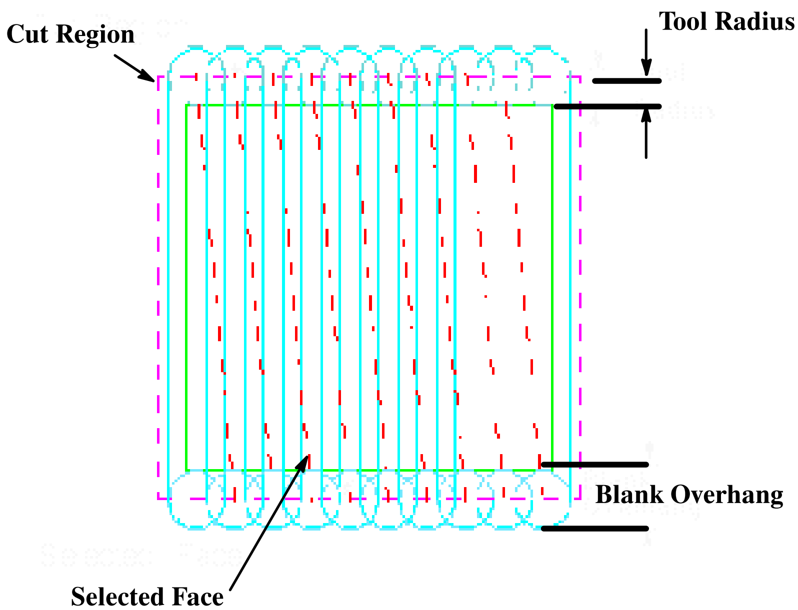
The tool path is replayed cutting each level with the specified cut pattern and the cut parameters that you previously defined.

❑ Choose **OK** to save the operation.

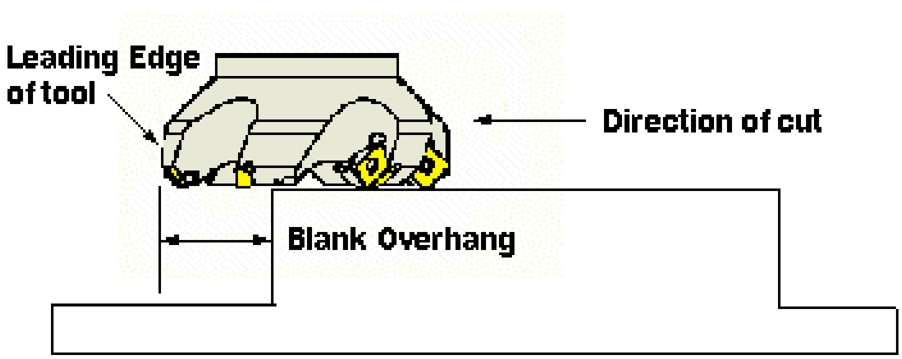
❑ **Save** the part file.

Face Milling – Blank Overhang

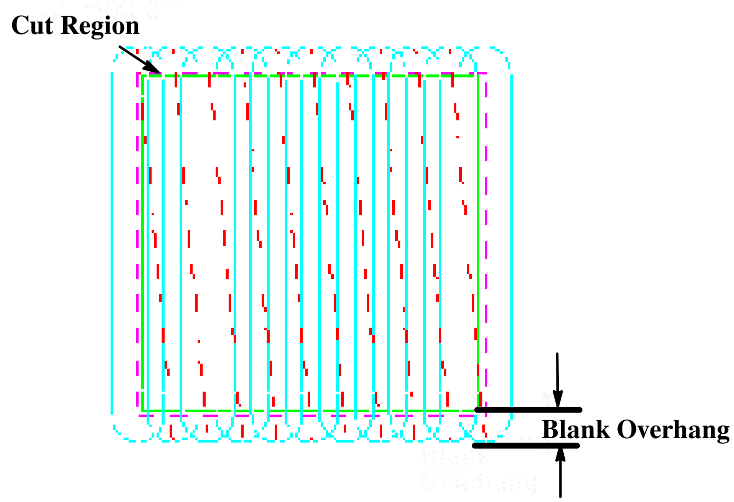
Blank Overhang allows you to control the distance that the cutting tool will travel beyond the edge of a face.



Blank Overhang is the distance from the leading edge of the cutter to the edge of the face that is being cut.

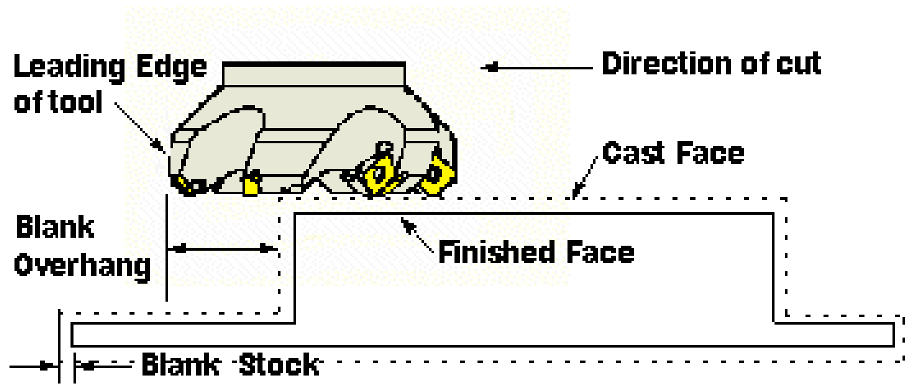


Setting the Blank Overhang parameter to a value smaller than the cutter diameter minimizes tool motion.



Blank Overhang is not the same as Blank Stock. There is a distinct difference between the two parameters.

Blank Stock applies additional stock to a finished face while Blank Overhang is the distance that the leading edge of the cutter extends beyond the edge of the face.



Use of these parameters, either individually or in combination, can greatly minimize the amount of time that is spent cutting air.

Activity 9—4: Face Milling — Utilizing Blank Overhang

In this activity, you will replay a Face Milling operation that generates a tool path with considerable time cutting air. You will then edit the operation, setting the Blank Overhang parameter, minimizing the distance the cutter will travel beyond the face. This will lead to a substantial decrease in the amount of time which is spent in cutting air.

Step 1 Open the part mmp_fm_blank_overhang.prt and enter the Manufacturing application.

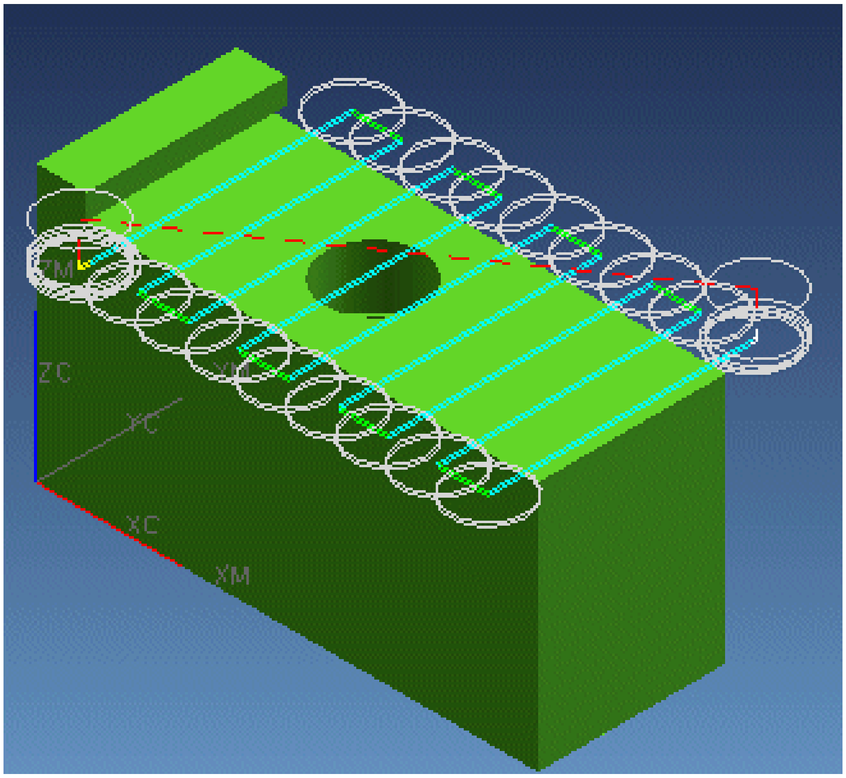
- ❑ From the menu bar, select **File**.
- ❑ Choose Open.
- ❑ Select the file **mmp_fm_blank_overhang.prt**, then choose **OK**.
- ❑ Choose **Application→Manufacturing**.

Step 2 Activate the Operation Navigator.

- ❑ Choose the **Operation Navigator** tab from the resource bar.
- ❑ If necessary display the Program Order View, using **MB3** on the Operation Navigator.

Step 3 Replay the FACE_MILL operation.

- ❑ Highlight the **FACE_MILL** operation, use **MB3** and **Replay** the operation.



Notice the cut region area and how it extends beyond the part.

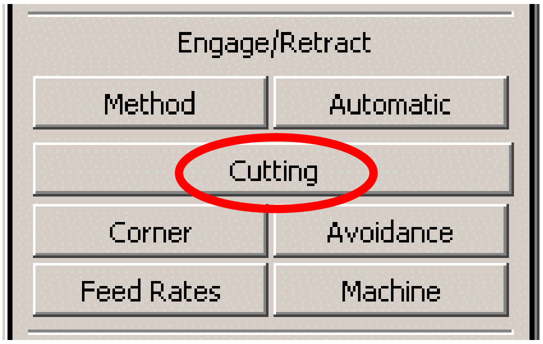
Blank Overhang represents the distance from the leading edge of the cutter to the edge of the face being cut and is expressed as a percentage of the cutter diameter.

Step 4 Set the Blank Overhang as being 25% of the cutting tool's diameter.

❑ Highlight the **FACE_MILLING** operation, use **MB3** and choose **Edit**.

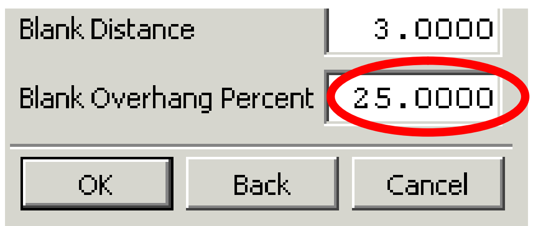
The FACE_MILLING dialog is displayed.

❑ Choose **Cutting** from the FACE_MILLING dialog.



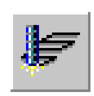
The Cut Parameters dialog is displayed.

- ❑ Key in the 25.0 as the value for the Blank Overhang Percent.

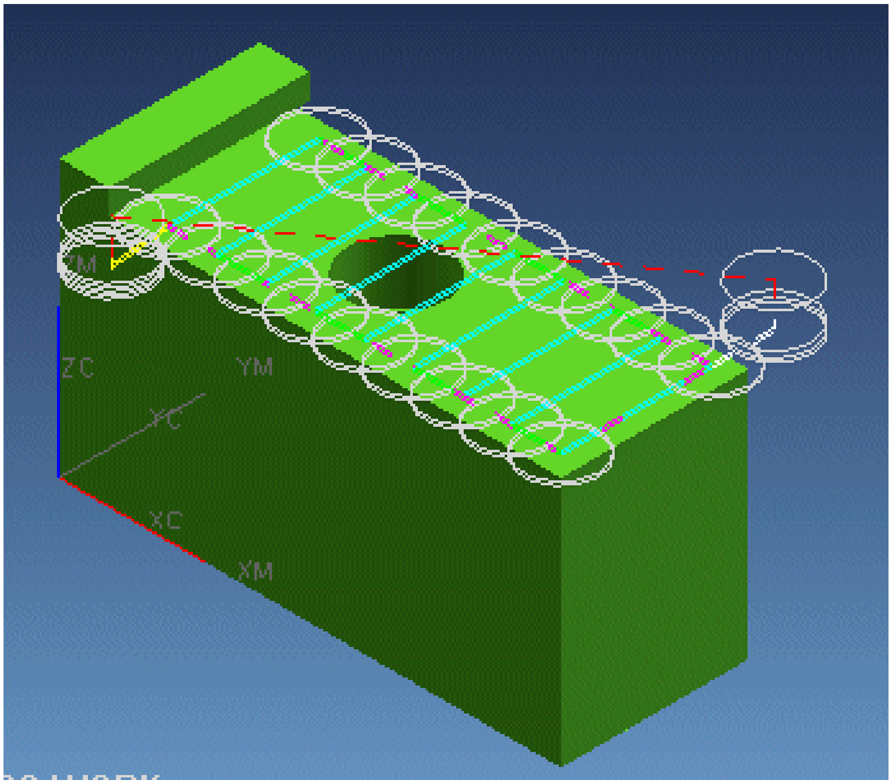


- ❑ Choose **OK** to accept the Blank Overhang Percentage.

Step 5 Generate the tool path.

- ❑ Choose the **Generate** tool path button. 

9



Note the location of the tool at the end of each pass.

❑ Choose OK to accept the operation.

❑ **Save** the part file.

Do not close the part since you will be using this part in the next activity.

Face Milling – Tool Run-Off

Some applications require a Zig or Zig-Zag cut pattern, to either retract the tool completely off the part or allow the tool to stay on the part, after each pass. This will minimize cutting tool travel time. The Tool Run-Off parameter addresses each of these situations.

Tool Run-Off applies only to operations which utilize Zig or Zig-Zag cut patterns. For Zig cut patterns, Internal Retract method must be set to Automatic. For Zig-Zag cut patterns, the Final Retract method must be set to Automatic. When Automatic Retract is employed and Tool Run-Off is ON, the tool will automatically retract off the part after each cutting pass by a distance that is equal to the Horizontal Clearance. If Tool Run-Off is set to OFF, the tool will not retract off of the part after each cutting pass.

Based on the amount of Blank Overhang, the tool may or may not remain on the part.

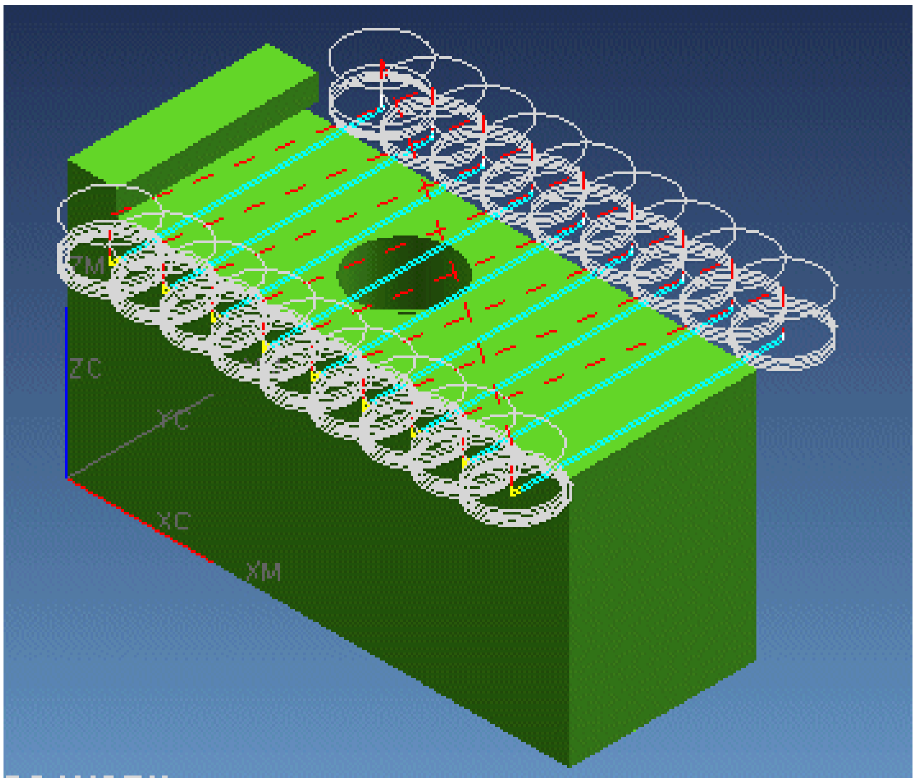
The Tool Run-Off function is currently applicable for face, planar and cavity milling operations.

Activity 9–5: Face Milling – Using Run-Off

You will replay the existing FACE_MILL _RUN_OFF operation and observe how the cutter cuts the top face of the part with the Run-Off parameter turned OFF. You will then activate the Run-Off parameter and observe the difference in tool motion.

Step 1 Replaying and reviewing an existing operation.

- ❑ Continue using the **mmp_fm_blank_overhang** part.
- ❑ Highlight the **FACE_MILL _RUN_OFF** operation, use **MB3** and **Replay** the operation.

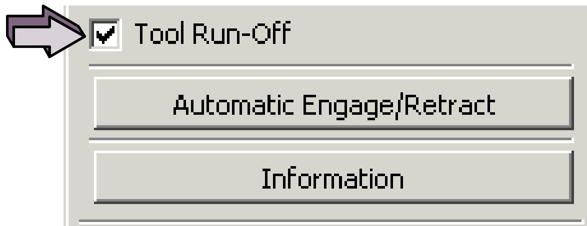


Note how the tool retracts from the part surface.

Step 2 You will now activate the Tool Run-Off parameter.

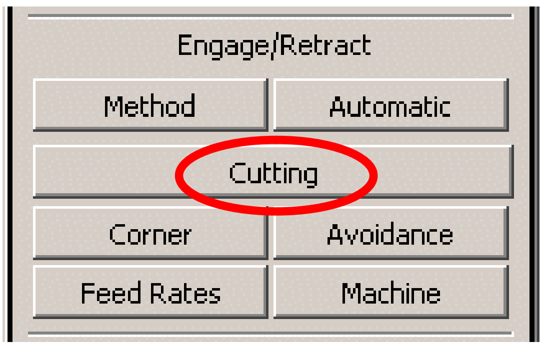
- ❑ Choose Method under Engage/Retract.

❑ Turn the Tool Run-Off button ON.

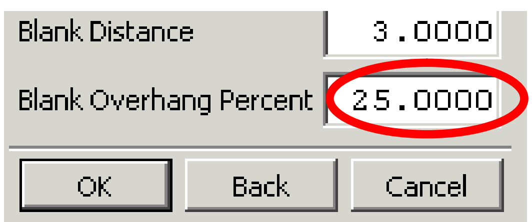


❑ Choose **OK** to accept the Engage/Retract dialog.

❑ Choose **Cutting** from the FACE_MILLING dialog.

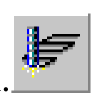


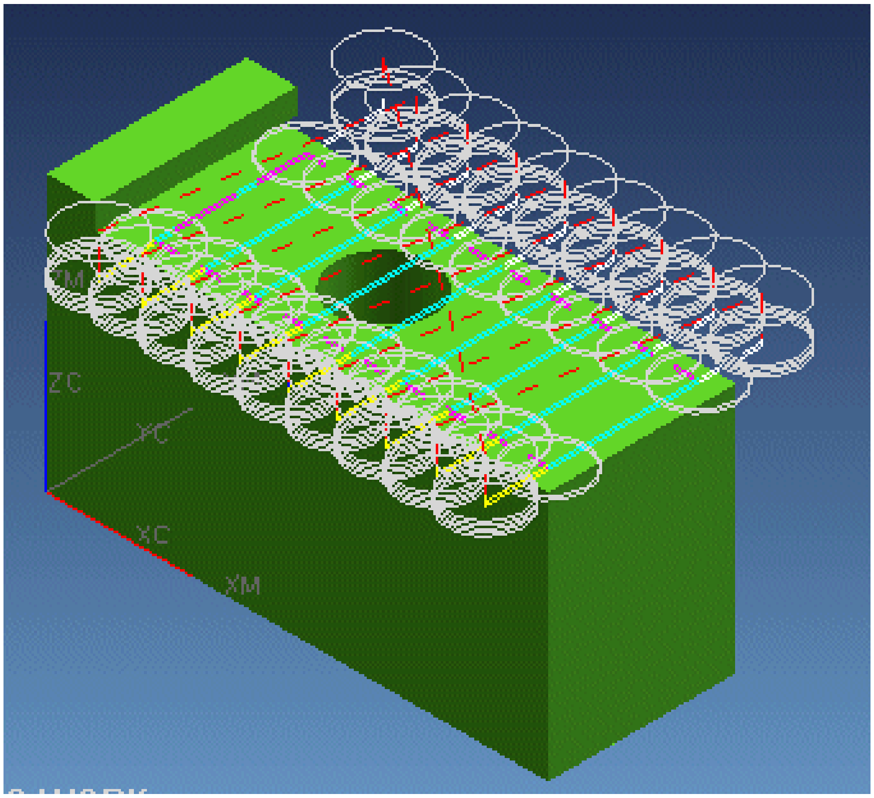
❑ Key in **25.0** as the value for the Blank Overhang Percent.



❑ Choose **OK** to accept the Blank Overhang Percentage.

Step 3 Generate the tool path.

❑ Choose the **Generate** tool path button. 



Note the tool movements and retractions from the part material.

- ❑ Choose **OK** to accept the operation.
- ❑ **Close** the part file.

You have completed this activity.

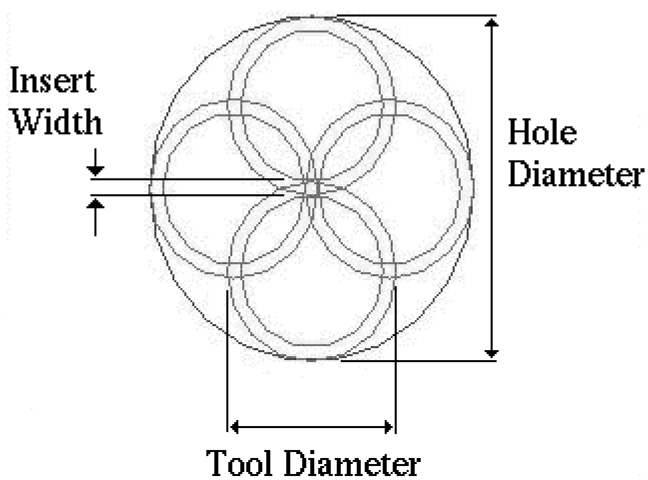
Face Milling — Controlling automatic engages directly into material.

With the development of high performance cutting tools, the need to drill relief or start holes has been virtually eliminated and tool engagement must be controlled to prevent cutting tool damage.

The most effective way of performing this engagement is through the milling of a starting hole using helical interpolation. Associated with this method of ramping is the parameter **Helical Diameter % —** which describes the maximum diameter path used by the tool for a Helical engage and is used for the Helical ramp type only.

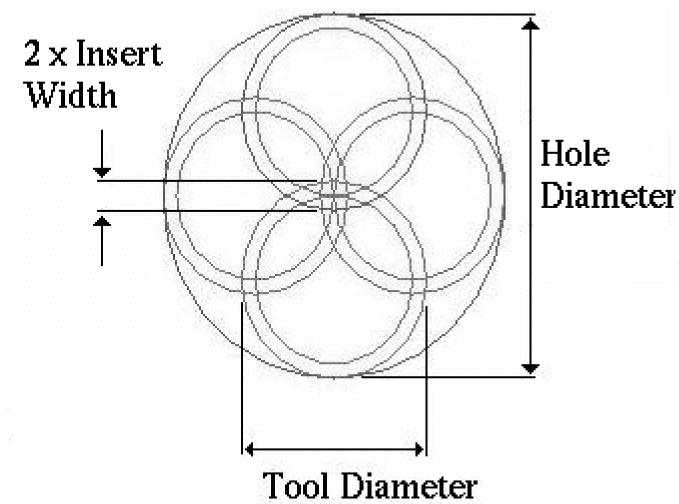
When using the helical engage method, there is a minimum and maximum hole diameter that can be utilized, with the result that no center or cone material remains at the bottom of the hole. The maximum hole diameter is equal to:

2 x Tool Diameter — Insert Width

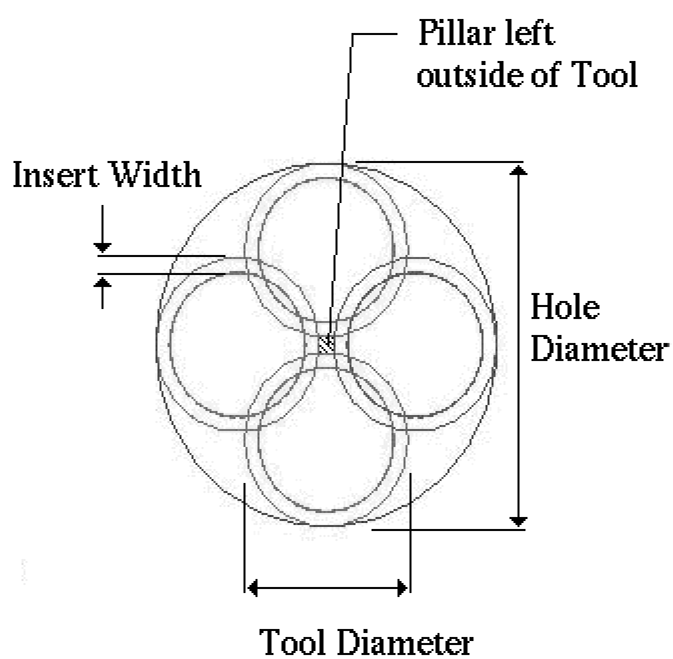


The minimum hole diameter is equal to:

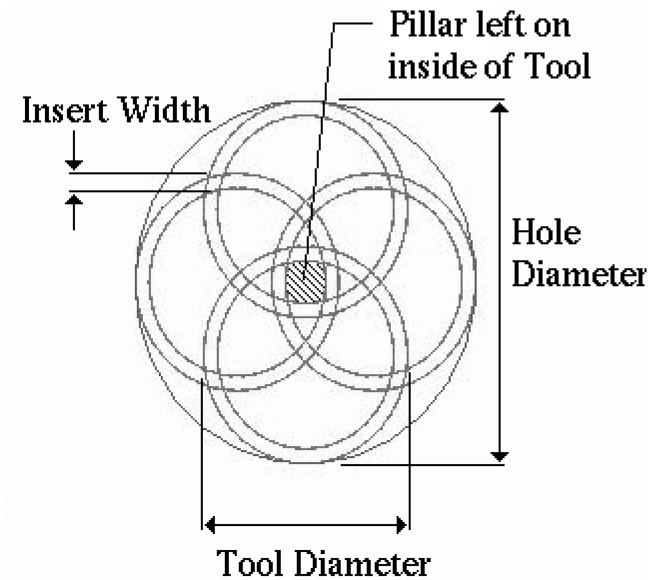
2 x Tool Diameter – 2 x Insert Width



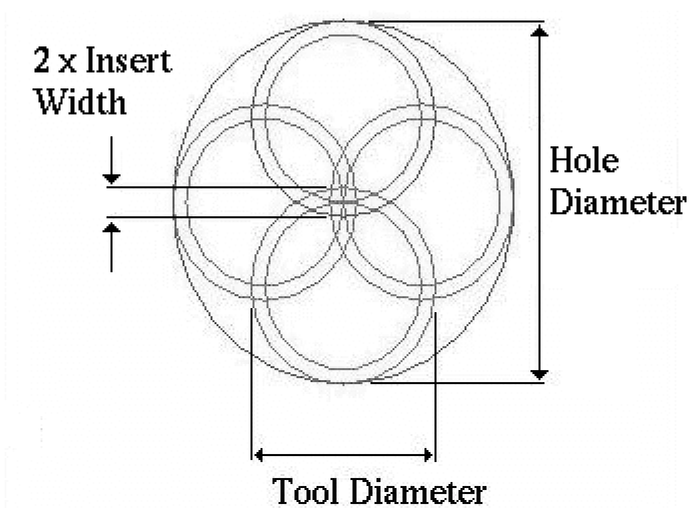
If the hole diameter is greater than the calculated maximum, material remains in the center of the hole but has no effect on the tool since the material is on the outside of the cutter.



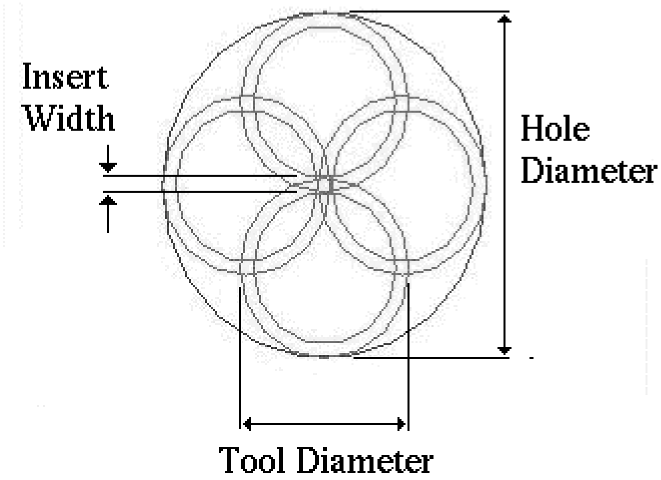
If the hole diameter is less than the calculated minimum, material remains in the center of the hole, causing the tool to break.



If the hole diameter is equal to the calculated minimum, no material remains in the hole and the cutting tool will cut normally.



If the hole diameter is equal to the calculated maximum, the bottom of the hole is completely flat. This is the preferred method of helical interpolation.

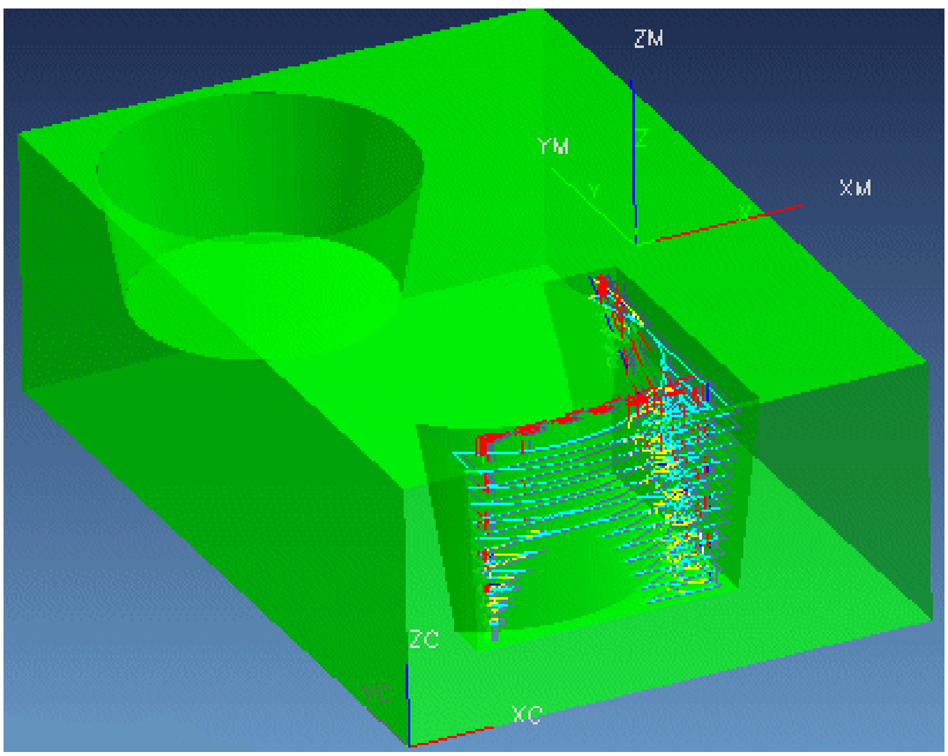


Activity 9–6: Face Milling – Utilizing Helical Engagement

You will replay the existing Helical engagement operation and observe how the cutter engages the part. You will then edit the operation to engage the part without leaving stock that could potentially damage the cutter.

Step 1 Open the part file, enter the Manufacturing application and replay and review an existing operation.

- ❑ From the menu bar, select **File**.
- ❑ Choose **Open**.
- ❑ Select the file **mmp_helical_engage**, then choose **OK**.
- ❑ Choose **Application→Manufacturing**.
- ❑ Highlight the **ZLEVEL_FOLLOW_CAVITY_HELICAL** operation, and then **Replay** the operation using **MB3**.



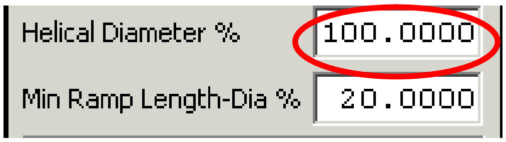
Notice how the tool engages the work piece.

Step 2 Setting the Helical Engage Parameters.

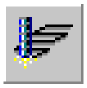
❑ Double-click the **ZLEVEL_FOLLOW_CAVITY_HELICAL** operation.

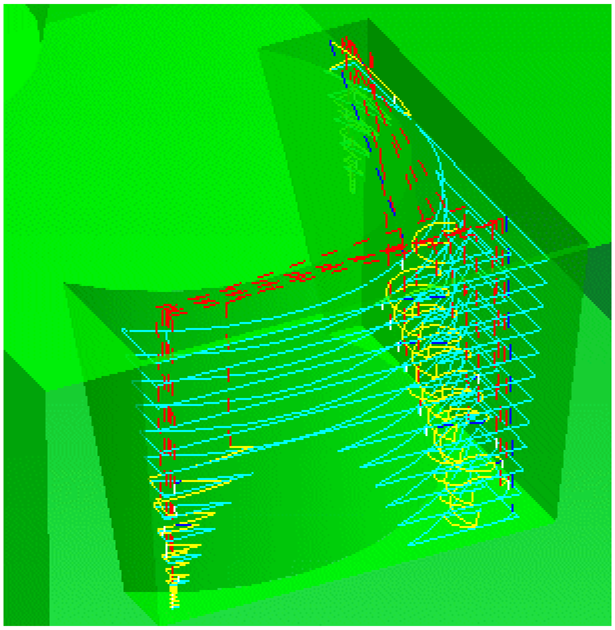
❑ Choose **Automatic** under Engage/Retract.

❑ Key in the value **100.** for the **Helical Diameter %**.



❑ Choose **OK** to accept the Automatic Engage/Retract dialog.

❑ Choose **Generate** to generate the tool path. 



Notice the difference in the Helical engage.

- ❑ Choose **Ok** to accept the operation.

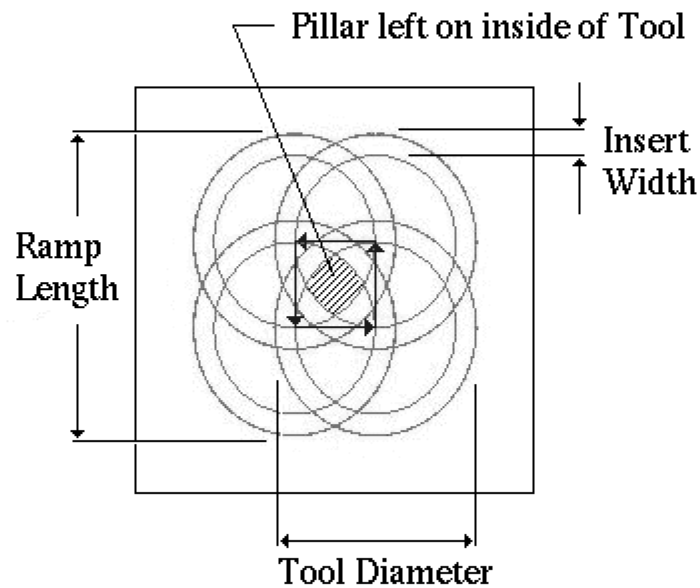
Do not close this part, you will be using it in the next activity.

Ramp type engagement

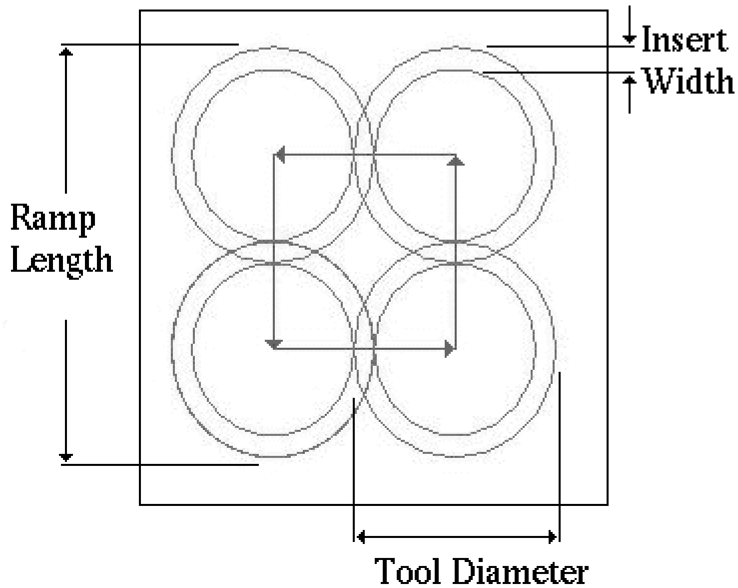
In cases where helical interpolation may be impractical (i.e. long narrow cut areas) the method for engagement is to ramp into the part. When using an inserted cutting tool, the ramping motion must be long enough to eliminate any uncut material. Associated with the ramp method is the parameter **Min Ramp Length-Dia% –** which is used by all three Ramp Types (On Lines, On Shape, and Helical). For On Shape and On Lines, the **Minimum Ramp Length-Dia%** represents the minimum path distance used by the tool from the top to the bottom of the ramp. For Helical Ramp Type, the parameter represents the minimum diameter path used by the tool. The minimum ramp length can be calculated as follows:

2 x Tool diameter – 2 x Insert Width

If the ramp length is less than the calculated minimum, material remains which results in tool breakage.



If the ramp length is greater than or equal to the calculated minimum, no material remains and the tool will cut normally.



When using the Helical Ramp Type, the calculations are made with the attempt to generate a helical motion using the Helical Diameter. If the area is not large enough to allow the Helical Diameter, the diameter of the path will be reduced, and an attempt will be made to perform the engage again.

This process will continue until either the Helical Engage is successful or the path diameter becomes less than the Minimum Ramp Length. If the area is not large enough to allow the path diameter that is equal to the Minimum Ramp length, calculations will be performed that will attempt to ramp into the area.

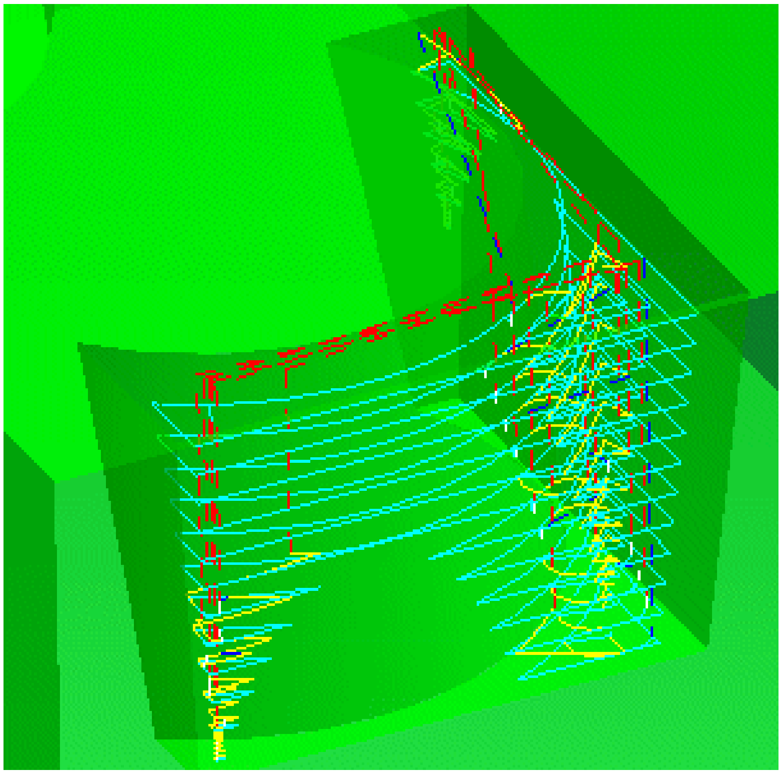
When ramping into a part, a check is made to verify that at least some portion of the region is long enough to allow ramping motion using the Minimum Ramp Length. If this is impossible, a warning message will be displayed.

Activity 9–7: Face Milling – Helical Ramp Engagement

In this activity, you will replay and review an existing operation, observing how the cutter engages the part. You will then alter the **Min Ramp Length-Dia %** parameter to avoid cutting the small triangular regions that you observed in the replay of the operation.

Step 1 Continue using the part file, mmp_helical_engage, replay and review an existing operation.

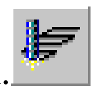
- ❑ If necessary, display the Program Order View in the Operation Navigator.
- ❑ Highlight the **ZLEVEL_FOLLOW_CAVITY_RAMP** operation, using **MB3**, **Replay** the operation.



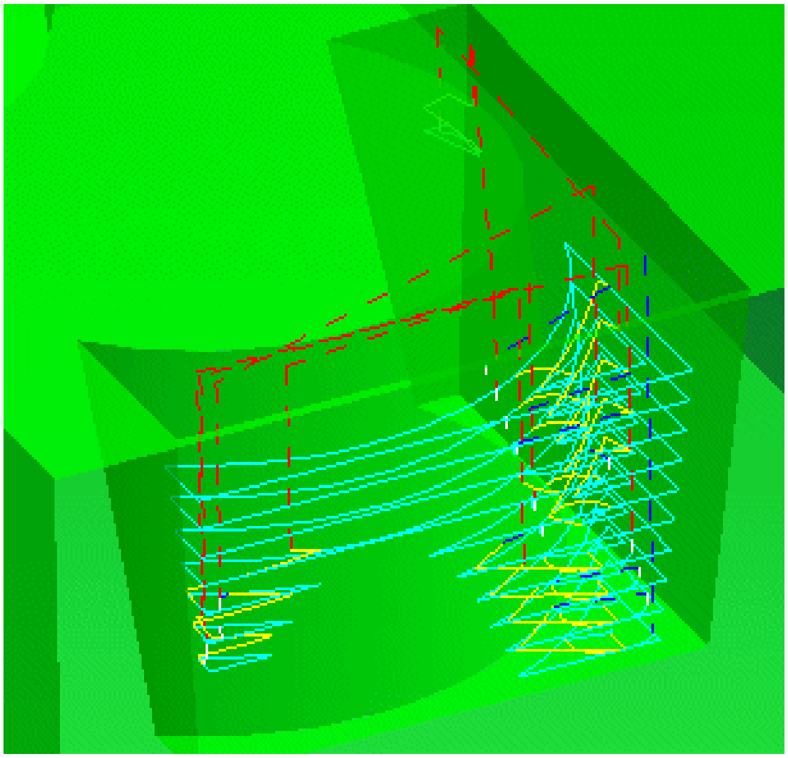
Notice how the tool engages the work piece. You will notice that there are numerous small triangular regions that are being cut. These are the areas that you want to avoid.

Step 2 Set the Minimum Ramp Length.

- ❑ Choose **Automatic** under Engage/Retract.
- ❑ Key in **80.0** as the value for the Min Ramp Length-Dia%.

- ❑ Choose **OK** to accept the Engage/Retract dialog.
- ❑ Choose **Generate** to generate the tool path. 
- ❑ Choose **OK** to the following warning dialog:

Notice how the cutter engages the part.



❑ **Close** the part file.

This concludes this activity and the lesson.

SUMMARY

This lesson was an introduction to Face Milling. Face Millings flexibility and ease of creating specific operations for milling faces of a part affords you increased productivity and efficiencies in the machining of your parts.

In this lesson you:

- Cut single and multiple faces
- Used the Traverse option to move the tool more quickly over a void areas of a face
- Used the ignore holes option to minimize the time cutting air
- Used different cut patterns when cutting multiple faces on the part
- Used the Blank Overhang option to minimize cutter travel
- Used the Run–Off and various helical engagement options to maximize tool life by controlling engagement into the part

9

Creating Drilling Tools and Operations

Lesson 10

PURPOSE

Drill Operation types are used to create tool paths for drilling, tapping, boring, counterboring and reaming operations. Numerous parameters are used to control depths and features that are associated with various types of holes.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Create cycles and cycle parameter sets
- Create drilling tools
- Create operations for Spot Drilling, Drilling, and Reaming
- Use various options to control drilling operations

This lesson contains the following activities:

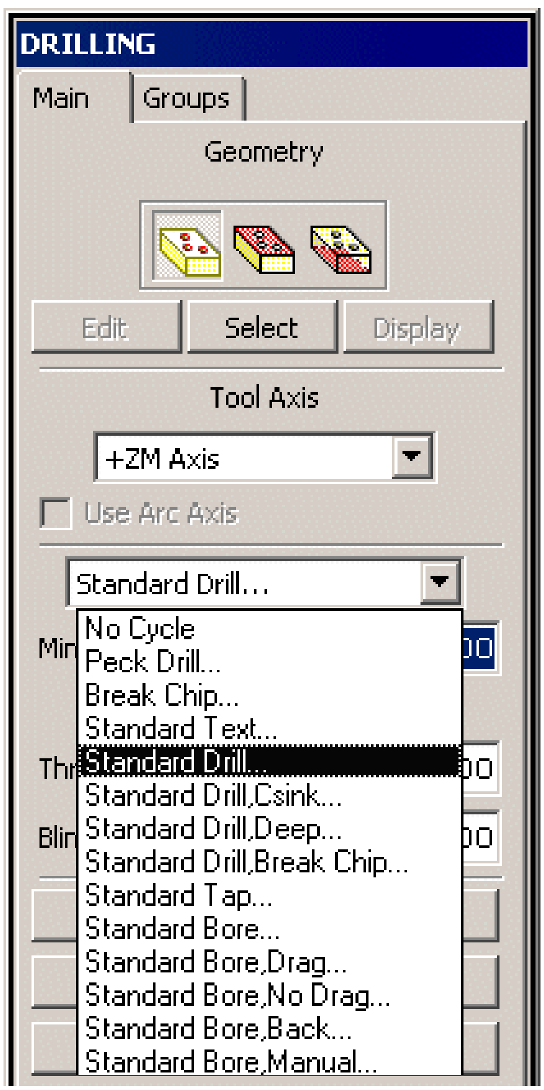
What is a Cycle?

A cycle describes the machine tool movements necessary to perform a machining function, for example; drilling, reaming, or boring.

Through postprocessing, cycle statements are normally output as *canned* cycle codes.

However, some machines do not have *canned* cycles. In those cases, only GOTO points are output.

The list of available cycles are shown in the following Drilling dialog:



Comments about Cycle Types

- The cycle type of No Cycle, Peck Drill, and Break Chip do not output CYCLE/ commands; the motion is simulated with GOTO points
- Standard Cycle options will output a CYCLE/ command at each of the specified CL points
- 'Drill', 'Drill,Deep' and 'Drill, Break Chip' output canned cycles and are the equivalent of 'No Cycle', 'Peck Drill' and 'Break Chip' which output simulated motion

The CYCLE/ command statements generated by the Unigraphics drill operations are compatible with the Unigraphics UGPOST postprocessing module. Depending upon the target postprocessor, controller and machine tool, the *Cycle* type you choose causes the postprocessor to:

- generate code for all the tool motions in a CYCLE/ operation to be executed by the machine tool, or
- generate GOTO/ command statements to define each of the tool motions and machine functions which simulate the desired cycle

After you choose one of the cycle options other than No Cycle, Peck Drill and Break Chip, you must specify how many parameter sets you intend to define for that cycle operation.

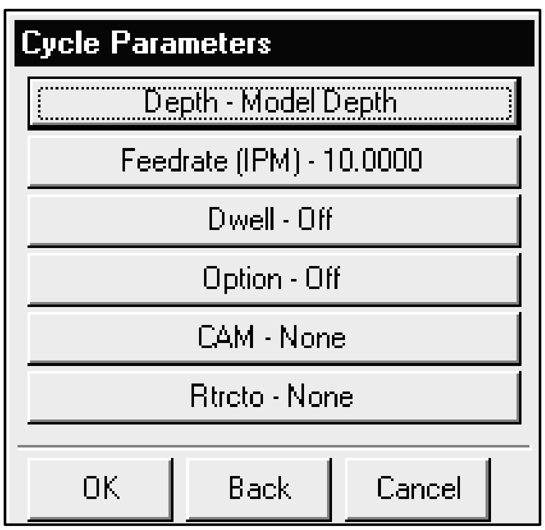
Cycle Parameter Sets

Cycle Parameter Sets are machining parameters such as depth, feed rate, dwell times and cutting increments, which are used to drill a series of holes.

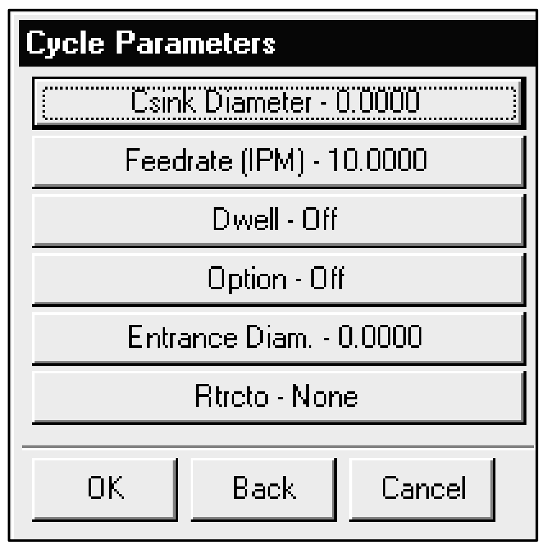
If all the holes in a tool path have the same cycle parameter values, you will use one Cycle Parameter Set. If you want to vary any of the cycle parameter values, for example the depth, you will create a Cycle Parameter Set for each hole or group of holes with different depths. You can have up to five parameter sets per cycle.

After you specify a Cycle Parameter Set, a dialog is displayed containing the options available for that cycle. Many of the options are the same from cycle type to cycle type, but there are some differences. The illustration below shows a comparison of the Standard - Drill cycle and the Standard - Drill,Csink cycle.

Standard – Drill Cycle



Standard – Drill,Csink Cycle



10

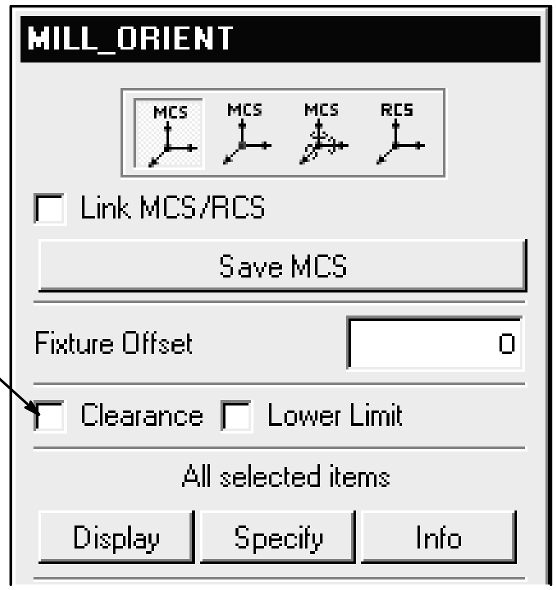
The following is a summary of the different cycle parameter options:

- **CAM –** is a number that specifies a preset CAM stop position for tool depth for machine tools with no programmable Z axis
- **Csink Diameter –** is the diameter of a countersunk hole
- **Depth –** is the depth of cut
- **Dwell –** is the delay of tool at the depth of cut
- **Entrance Diameter –** is the outside diameter of an existing hole that is to be enlarged by a countersink operation
- **Feedrate –** is the cutting feed rate
- **Increment –** is the dimensional value of one of a series of regular consecutive cuts to progressive depths used in Peck and Break Chip drilling operations
- **Option –** is used to activate machining characteristics that are unique to a particular machine and is usually postprocessor dependent (this function includes the word OPTION in the CYCLE statement)
- **RTRCTO –** is the cycle retract distance
- **Step Values –** is the dimensional value of one of a series of regular consecutive cuts to progressive depths used in Standard – Drill, Deep and Standard – Drill, Brkchp operations

Setting Clearance Planes in a Drill Operation

Clearance Planes are recognized in drilling operations. Drilling operations will also inherit a Clearance Plane that is defined within an MCS Parent Group.

Define the Clearance Plane in the MCS_MILL Parent Group. This will allow any operation placed under it to inherent the Clearance Plane settings.



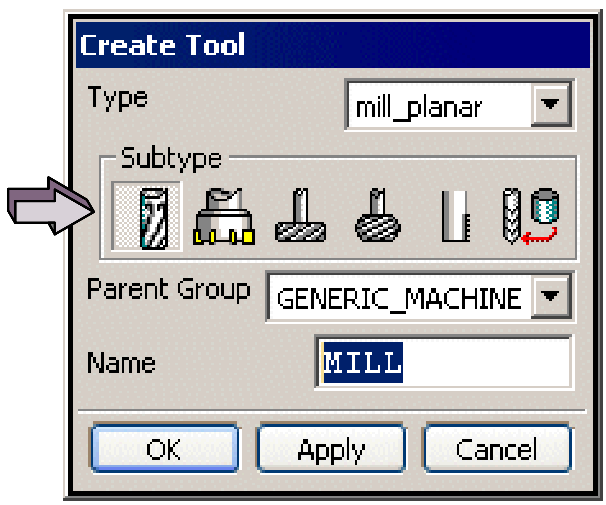
Tools Created and Used in Drilling

The tools that you can create are initially determined by the Machining Environment **Setup**. You will learn more about this later.

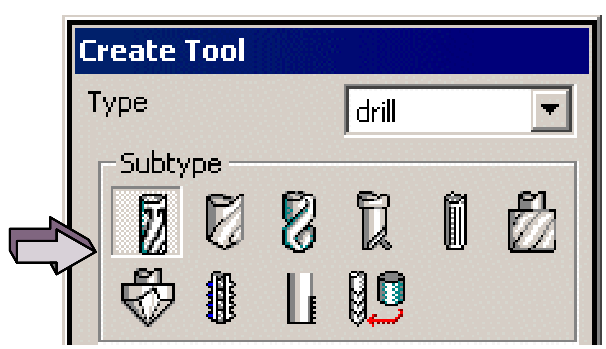
Types of Tools

The Setup that you specify becomes the **Type** shown in the Create dialog. You will also learn about this functionality later.

In the Create Tool dialog below, the Type is mill_planar; therefore, the tools that you create are milling tools (i.e., Mill, Face Mill, T-cutter, and Barrel type).



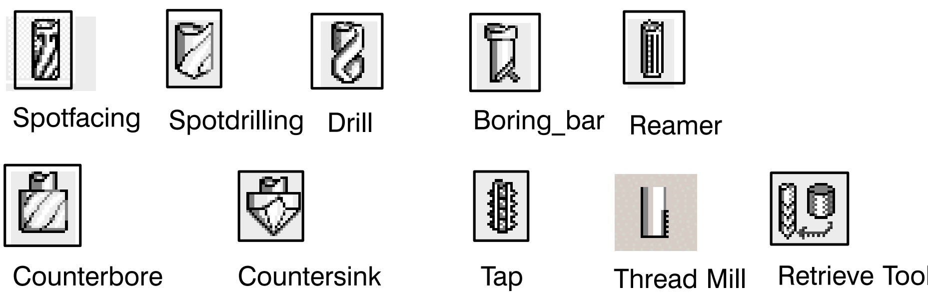
You can change the Type setting and then create different kinds of tools. As shown below, the type has been set to Drill. Notice the drill types that you can create.



Creating a Drill Tool

Tools can be created within a Parent Group or within an operation. The preferred method is to use the Parent Group.

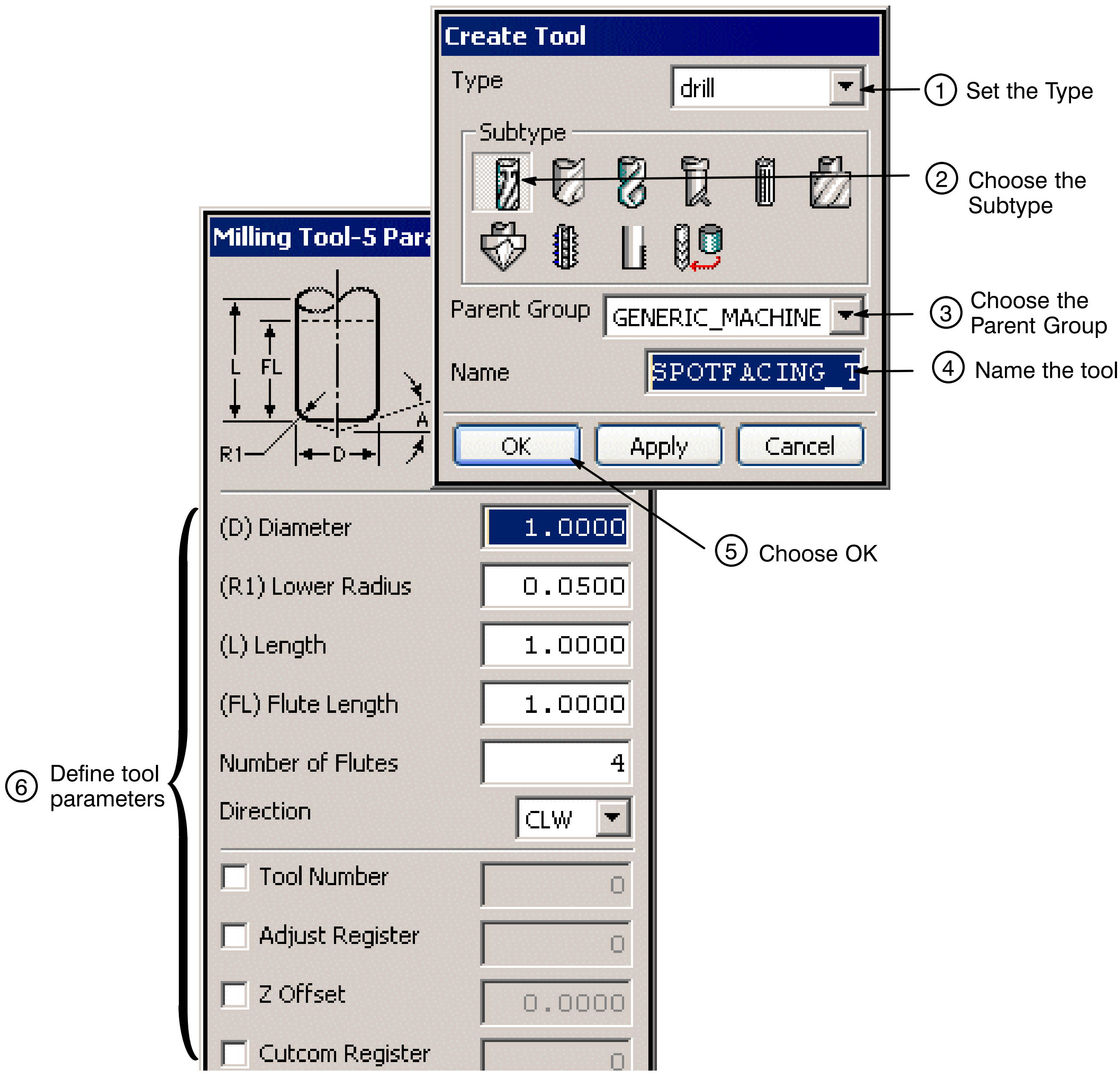
The Drill types you can create are represented in the Subtype area as icons. There are a total of nine drill types. The last icon in the Subtype area is the Retrieve Tool icon.



How to Create a Tool

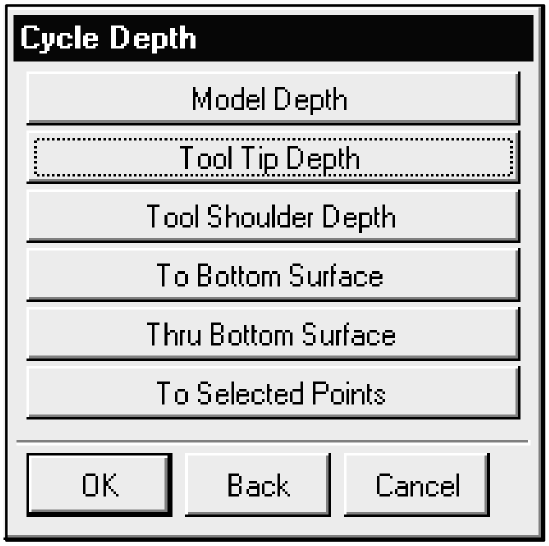
The steps taken to define a tool in the Parent Group are as follows:

- Choose the Create Tool icon
- Choose the Type (i.e., drill, planar_mill)
- Choose the Subtype (i.e., spotdrill, endmill)
- Choose the Parent Group
- Name the tool
- Choose OK
- Define the tool parameters

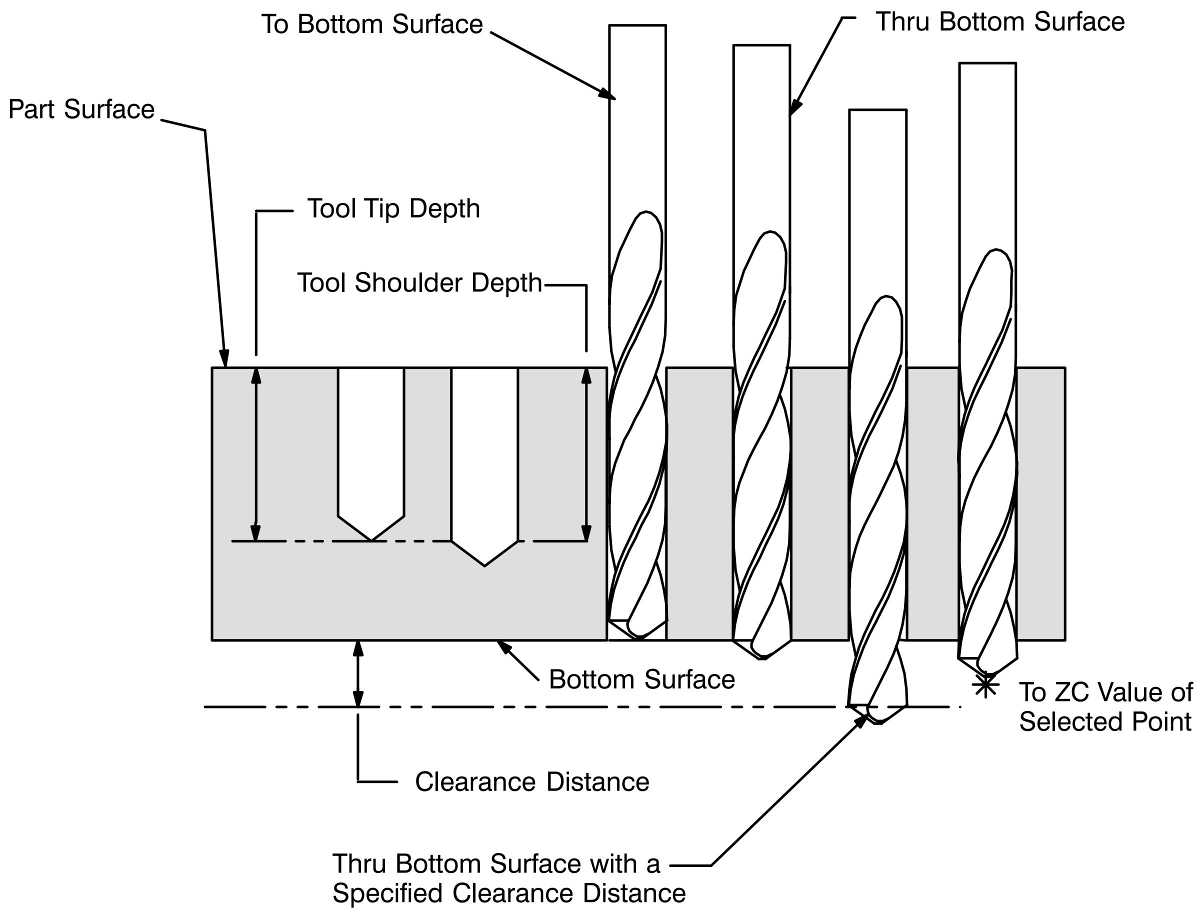

Create Tool
Type
drill
Subtype
Parent Group
GENERIC_MACHINE
Name
SPOTFACING_T
OK
Apply
Cancel
Milling Tool-5 Para
L
FL
R1
D
A
(D) Diameter
1.0000
(R1) Lower Radius
0.0500
(L) Length
1.0000
(FL) Flute Length
1.0000
Number of Flutes
4
Direction
CLW
Tool Number
0
Adjust Register
0
Z Offset
0.0000
Cutcom Register
0
1 Set the Type
2 Choose the Subtype
3 Choose the Parent Group
4 Name the tool
5 Choose OK
6 Define tool parameters

Tool Depth

One of the cycle options that you set is the tool depth. The different methods are shown and explained as follows:

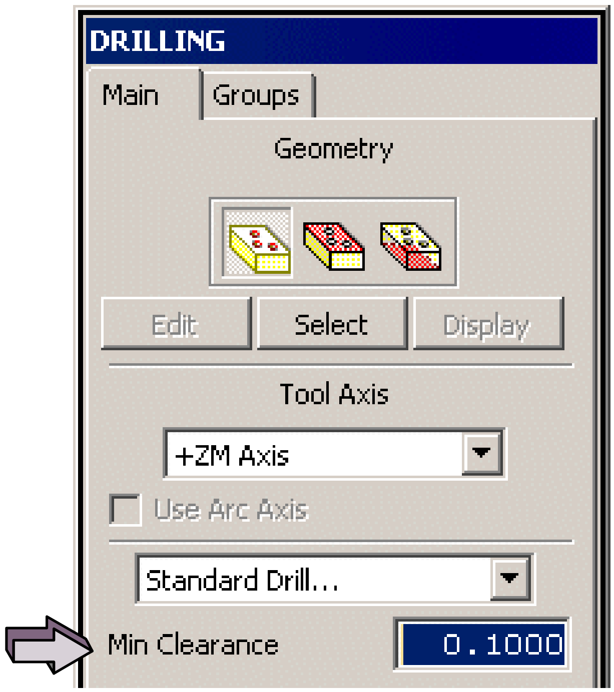


- **Model Depth –** automatically computes the depth for each hole in a solid
- **Tool Tip Depth –** the tool tip feeds to the depth specified
- **Tool Shoulder Depth –** moves the shoulder of the tool to the specified depth
- **To Bottom Surface –** the tool tip feeds to the bottom surface
- **Thru Bottom Surface –** moves the shoulder of the tool to the bottom surface (or past it if a clearance distance is specified)
- **To Selected Points –** feeds the tip of the tool to the Z depth of the specified points

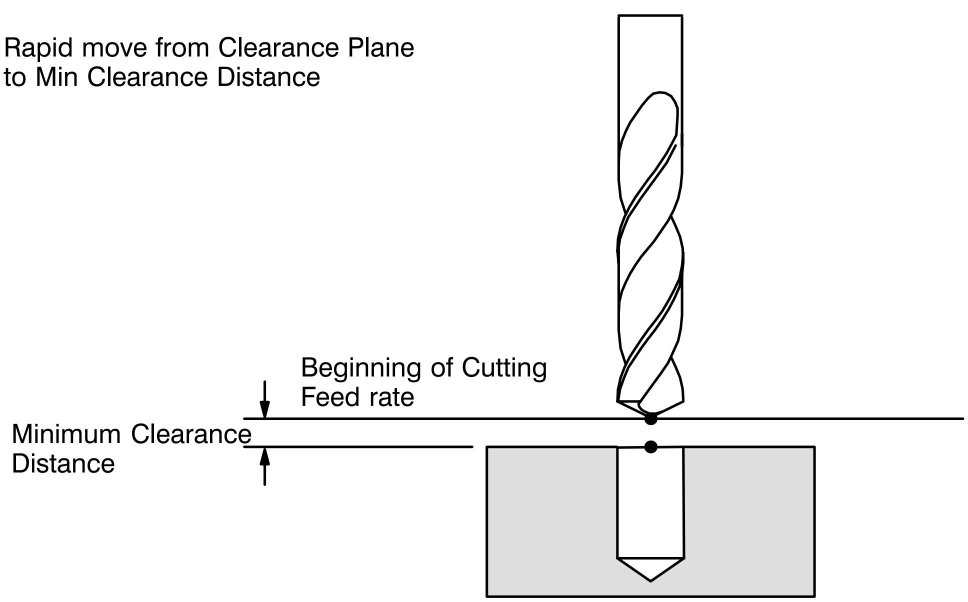

To Bottom Surface
Thru Bottom Surface
Part Surface
Tool Tip Depth
Tool Shoulder Depth
Bottom Surface
To ZC Value of
Selected Point
Clearance Distance
Thru Bottom Surface with a
Specified Clearance Distance

Minimum Clearance

The Minimum Clearance Distance determines how the tool is positioned before entering the material.



If a Clearance Plane has not been set, the tool will position to the next hole at the rapid feed rate directly to the specified Minimum Clearance distance above the part surface. If a Clearance Plane is specified, the tool will move at the Rapid feed rate from the Clearance Plane to the specified Minimum Clearance.

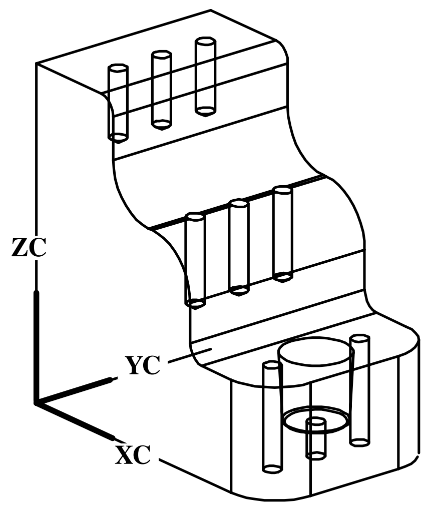


Activity 10—1: Creating a Spot Drilling Operation

In this activity, you will create a spot drilling operation. You will use the Create Operation dialog to define a cycle parameter used in the drilling operation.

Step 1 Open the part and enter the Manufacturing application.

❑ Open the part file **mmp_register_1.prt**.

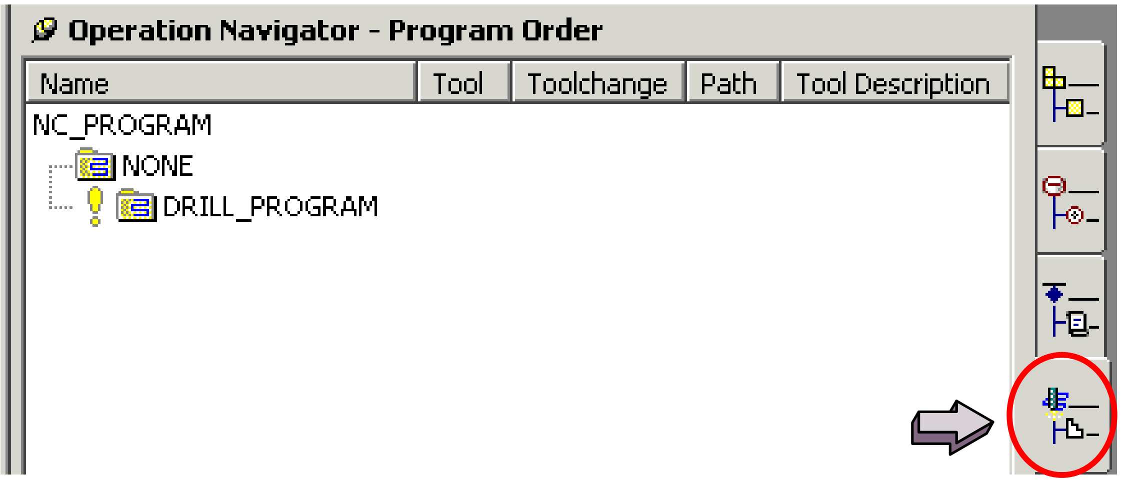


❑ Rename the part *****_register_1.prt** using the **File→Save As** option on the menu bar.

In this part some of the holes are instances. The counterbore is actually a tapered cylindrical pocket. Some of the holes are actually ellipses and qualify as valid cut geometry.

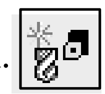
❑ Choose **Application→Manufacturing.**

❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.



The Operation Navigator is displayed.

Step 2 Creating the spot drill.

❑ Choose the **Create Tool** icon. 

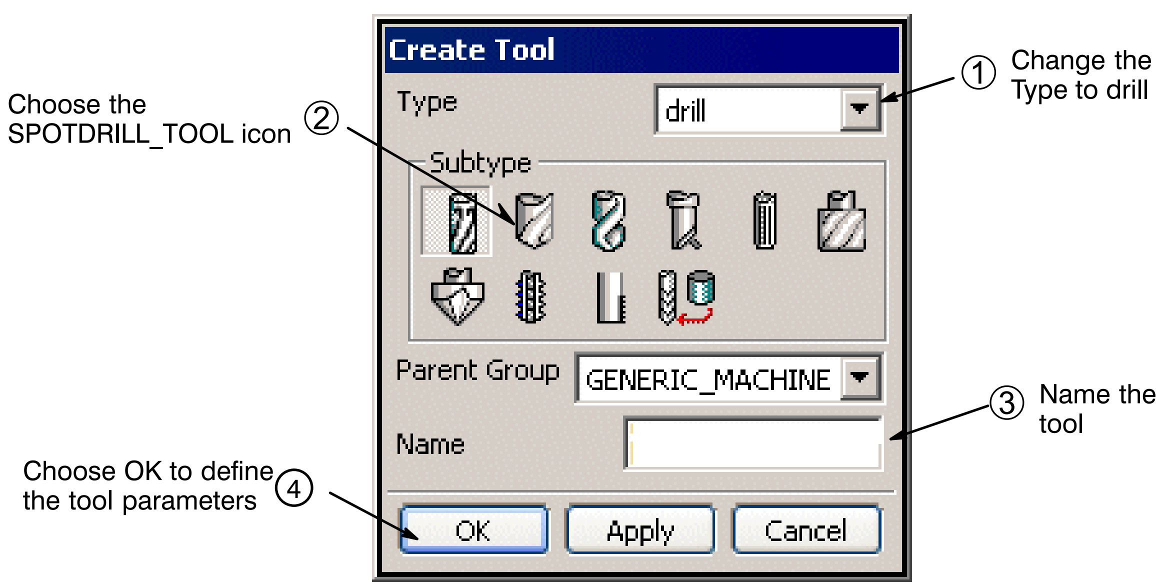
The Create Tool dialog is displayed.

You need to make sure the Type is drill.

❑ Change the Type to **drill,** if necessary.

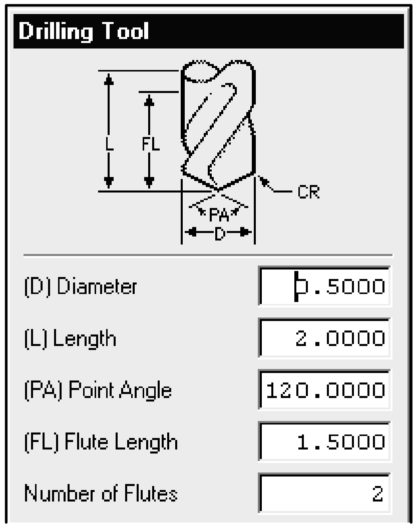
❑ Choose the **SPOTDRILLING_TOOL** icon.

❑ Name the spot drilling tool, **ug_spot_.5_90.**

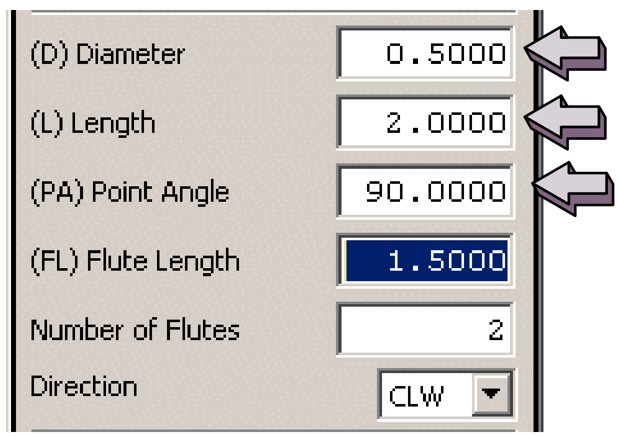


❑ Choose **OK.**

The Drilling Tool dialog is displayed.



❑ Enter the following parameters:



❑ Choose **OK.**

❑ Change to the **Machine Tool View** of the Operation Navigator.

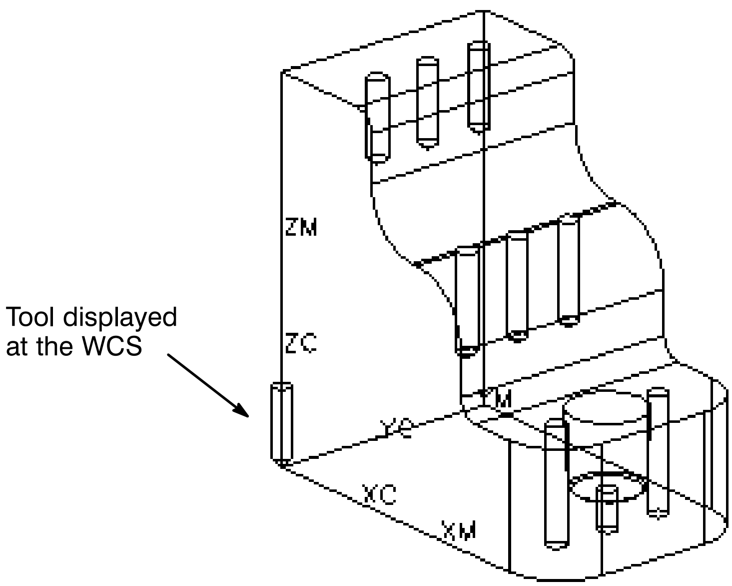
The spot drill that you created is listed in the window.

❑ In the Operation Navigator, highlight the tool **ug_spot_.5_90.**

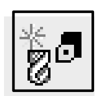
❑ Using MB3, choose **Object→Display.**

10

The tool is displayed at the WCS.

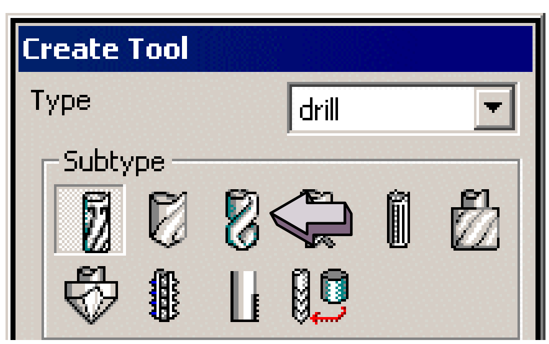


Step 3 Creating the drill.

❑ Choose the **Create Tool** icon. 

The Create Tool dialog is displayed.

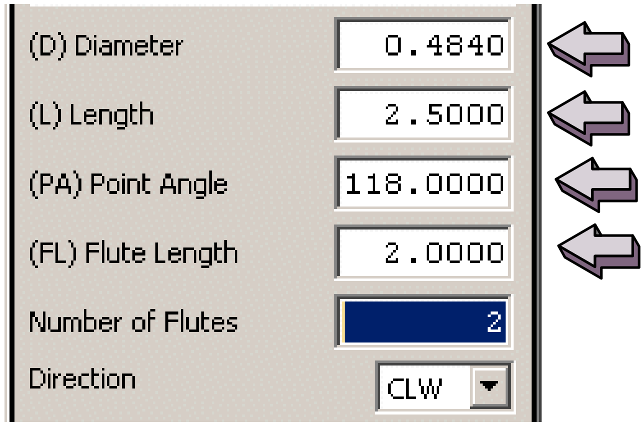
❑ Choose the **Drill** icon.



❑ Enter the tool name **ug_drill_.484**

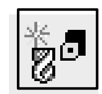
❑ Choose **OK.**

❑ Enter the following tool parameters:



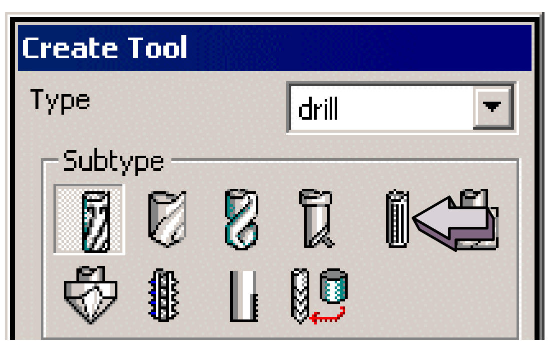
❑ Choose **OK.**

Step 4 Creating a reamer.

❑ Choose the **Create Tool** icon. 

The Create Tool dialog is displayed.

❑ Choose the **Reamer** icon.

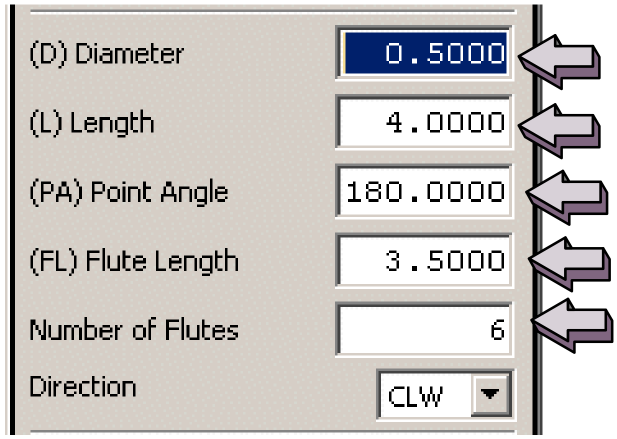


❑ Enter the tool name **ug_reamer_.5_180.**

❑ Choose **OK.**

❑ Enter the following tool parameters:

10



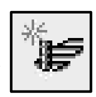
❑ Choose **OK.**

You are now ready to create the operations.

Step 5 Creating a new operation.

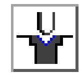
You now need to select the Tool, Geometry, Method and Program Parent Groups required for this operation.

The geometry was selected in the Geometry Parent Group DRILL_GEOM.

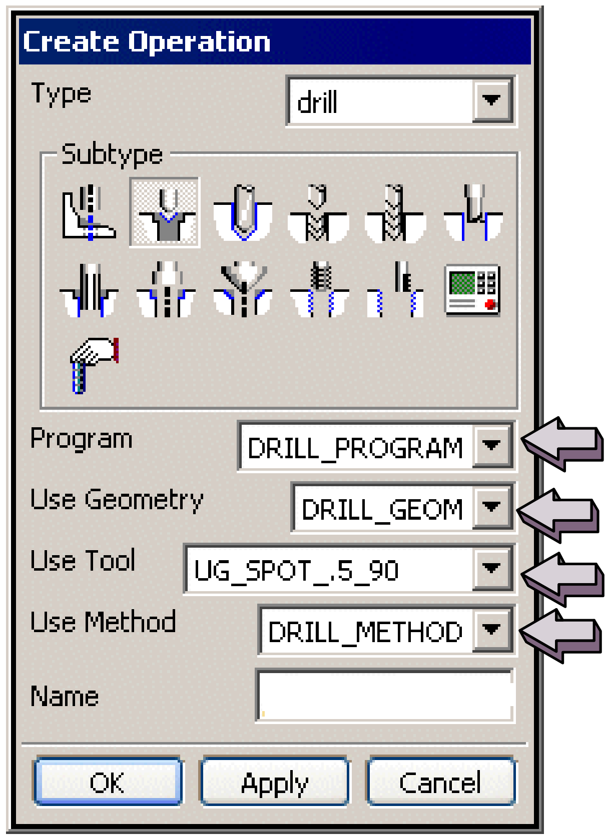
❑ Choose the **Create Operation** icon .

❑ Change the Type to **drill,** if necessary.

❑ Change the Operation Navigator to the **Program Order View**, if necessary.

❑ Choose the **SPOT_DRILLING** icon  .

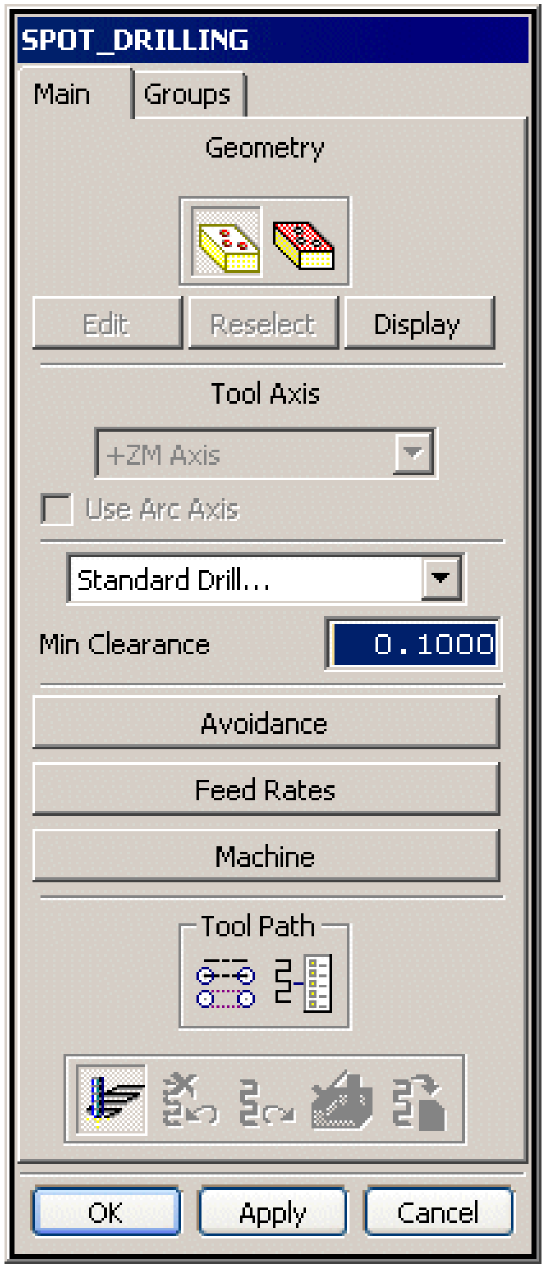
❑ Set the following Parent Groups:



❑ Enter **SPOT_DRILL_1** in the **Name** field and choose **OK.**

10

The SPOT_DRILLING dialog is displayed.



10

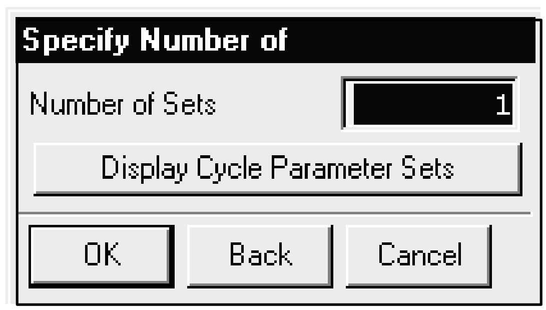
Step 6 Defining a Standard – Drill cycle containing one cycle parameter set.

❑ Click on the **Standard Drill** option.

The option dialog lists all the available cycle types.

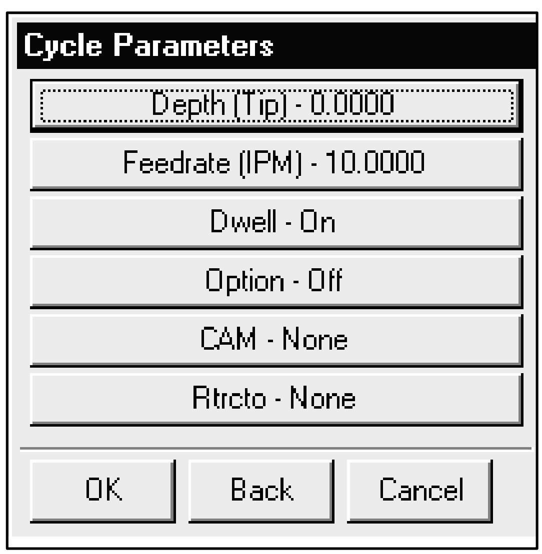
❑ Choose **Standard Drill** from the list of cycle types.

A new dialog is displayed and the Cue line prompts you to specify how many Cycle Parameter Sets you will use for this operation.

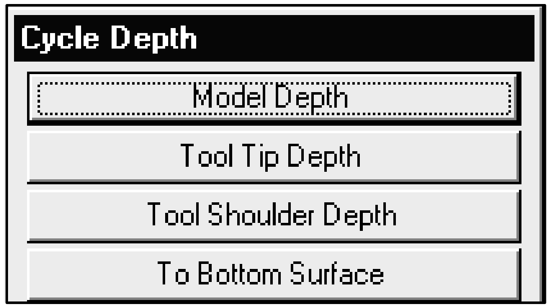


❑ Choose **OK** to accept the default of **1** Cycle Parameter Set.

The Cycle Parameters dialog is displayed. It shows the parameter settings for this cycle (the Cue line indicates that this is Cycle Parameter Set 1).



❑ Choose **Depth** from the dialog.



The dialog is displayed showing the different methods to specify the depth.

You are going to use Tool Tip Depth to feed to a specified depth.

❑ Choose **Tool Tip Depth**.

The depth is going to be half of the diameter of the tool, which is the typical rule used in spot drilling.

❑ Enter **.5/2** as the depth and choose **OK**.

Step 7 Setting the Dwell.

❑ Choose **Dwell** from the dialog.

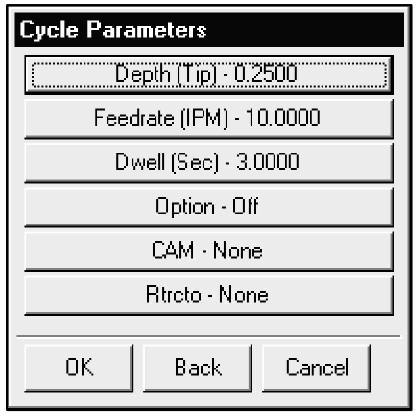
❑ Choose **Seconds**.

❑ In the Seconds field, enter **3.**

❑ Choose **OK**.

The tool will dwell for 3 seconds when it reaches depth**.**

You return to the Cycle Parameters menu.

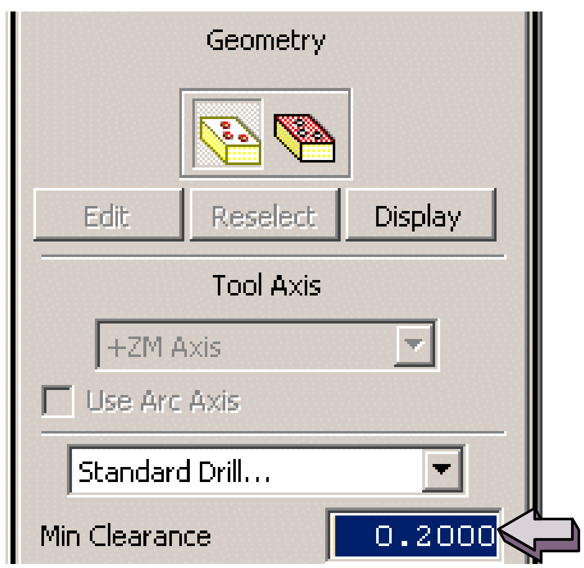


❑ Choose **OK** to return to the SPOT_DRILL dialog.

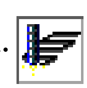
Step 8 Defining a Minimum Clearance Distance.

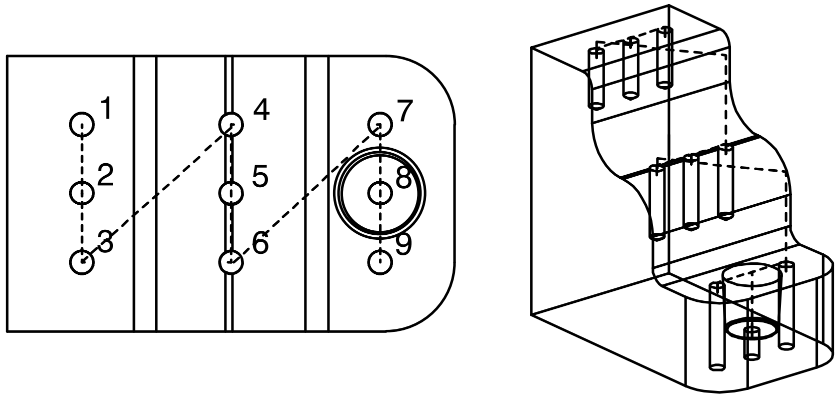
The Minimum Clearance Distance determines where the tool is positioned before entering the material.

❑ Enter **.200** in the **Min Clearance** field.



Step 9 Generating the tool path.

❑ Choose the **Generate** icon. 



Your tool path should look similar to the above.

The tool moves:

- along the minimum clearance distance to the hole
- at the cut feed rate from the top of the minimum clearance distance to the specified cut depth
- then retracts to the minimum clearance distance
- repositions to the next hole

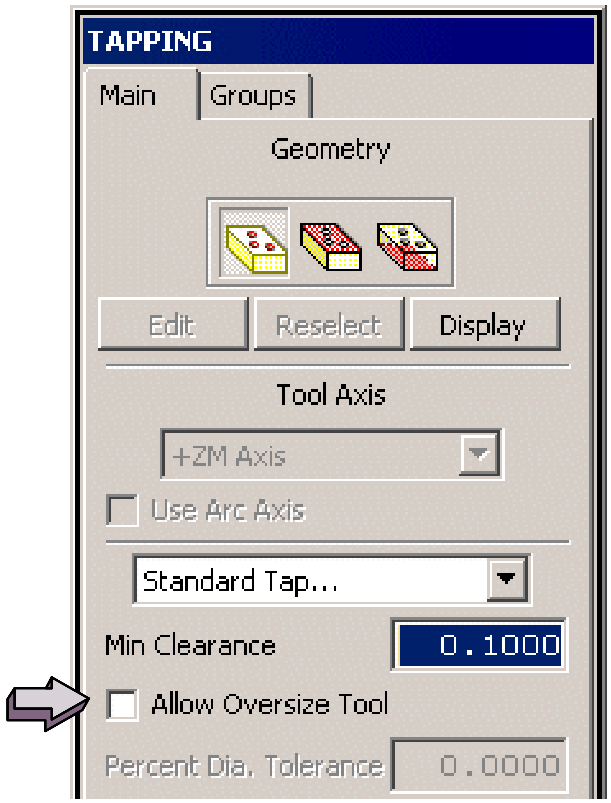
❑ Choose **OK** to accept the tool path.

❑ **Save** the part file.

This completes this activity. Do not close the part file since you will be using it in the next activity.

Drilling Over Sized Holes

Allow Oversize Tool option allows you to use a tool (typically a drill) that is larger than the modeled hole. This allows the use of various cycles on holes that have been modeled using the hole's nominal diameter. To use this feature, you must use Modeled Depth as the depth parameter (for some types of operations, such as reaming, Allow Oversize Tool, must be added through the use of Custom Dialogs).



When toggled *ON*, the Percent Dia. Tolerance becomes available, allowing you to specify an oversize tool tolerance in terms of percentage of the hole diameter. The Percent Dia. Tolerance is designed as a check that eliminates a tool that is too large to make the hole.

Any tool can be used that is less than the hole diameter plus the specified tolerance.

Depth Offset

The Depth Offset options are used in conjunction with the depth you set in the Cycle Parameter Sets.

Here is a summary of the Depth Offset options:

- **Blind Hole –** Defines the amount of material that will remain above the bottom of a blind hole using the *tip* of the tool
- **Thru Hole –** Defines the distance that the drill will move *past* the break out of a thru hole

Activity 10–2: Creating a Drilling Operation

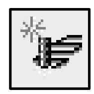
In this activity, you will create a new operation to drill the holes you previously spot-drilled. You will specify a different tool and generate the tool path.

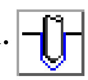
Step 1 Open the part file.

❑ Continue to use the ***_**register_1.prt.**

Step 2 Create a new operation to drill.

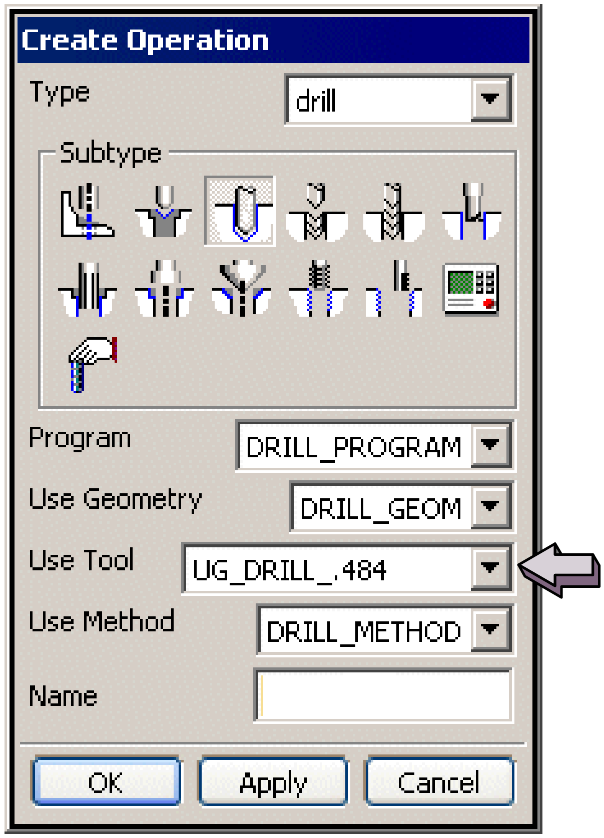
❑ Change the Operation Navigator to the **Program View,** if necessary.

❑ Choose the **Create Operation** icon. 

❑ Choose the **DRILLING** icon. 

You need to change the tool for this operation.

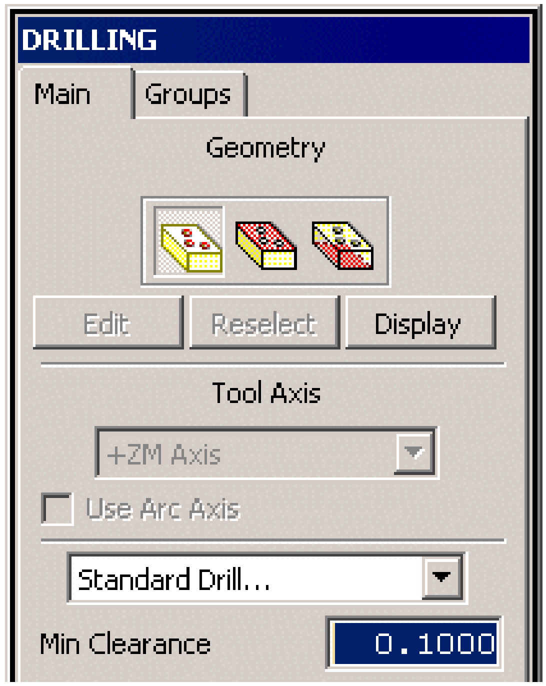
- ❑ Set the **Use Tool** Parent Group to UG_DRILL_.484.



- ❑ Enter **DRILL_1** in the **Name** field and choose **OK**.

10

The DRILLING option dialog is displayed.



Step 3 Creating the cycle and defining the cycle parameters.

You will define the Standard – Drill cycle.

❑ Choose **Standard Drill** from the list of cycle types.

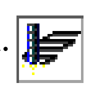
The Specify Number of dialog is displayed.

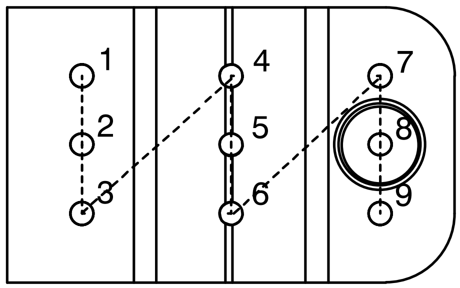
❑ Choose **OK** to accept the cycle parameter set.

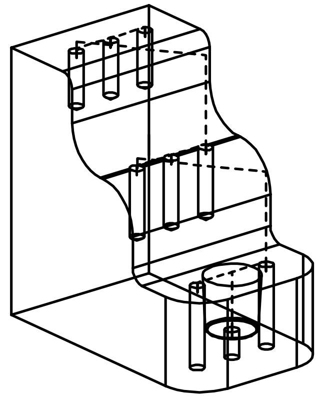
The Cycle Parameters dialog is displayed.

Notice that the Depth is set to **Model Depth**.

❑ Choose **OK** to accept the Cycle Parameter Set.

❑ Choose the **Generate** icon. 





The tool cuts to the bottom of each hole.

❑ Choose **OK** to accept the operation.

The operation and tool path are accepted.

❑ **Save** the part.

This completes this activity. Do not close the part since you will be using it in the next activity.

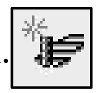
Activity 10–3: Creating a Reaming Operation

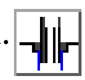
In this activity, you will ream the holes that were previously drilled. You only need to redefine the tool for this operation. You will add a depth offset to this operation to prevent the reamer from gouging at the bottom of the hole.

Step 1 Open the part file.

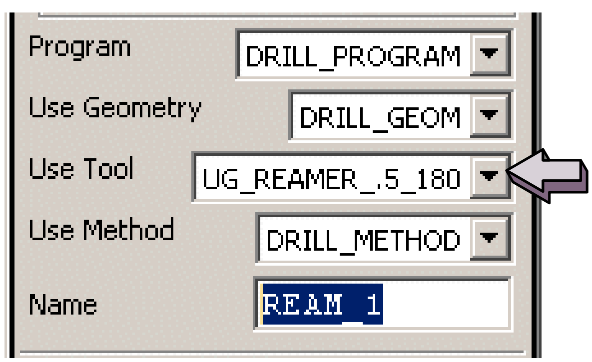
❑ Continue to use the ***_**register_1.prt.**

Step 2 Create a new operation.

❑ Choose the **Create Operation** icon. 

❑ Choose the **REAMING** icon. 

❑ Set the **Use Tool** Parent Group to UG_REAMER_.5_180.



❑ Enter **REAM_1** in the **Name** field and choose **OK**.

The REAMING dialog is displayed.

Step 3 Create the Cycle and define the cycle parameters.

You will define the “Standard – Bore” cycle for the reaming operation.

❑ Choose **Standard Bore** from the list of cycle types.

The Number of Sets dialog is displayed.

❑ Choose **OK**.

The Cycle Parameters dialog is displayed.

❑ The **Depth** should be **Model Depth**.

❑ Choose **OK** to accept the Cycle Parameters.

Step 4 Defining a Depth Offset.

Remember, the Depth Offset options are used with the depth you have set in the Cycle Parameter Sets. In this case, the offset is applied to the model depth.

You will use an equation in the value field. Specify an offset (.100) plus the drill size (.484) multiplied by the tip depth (.3).

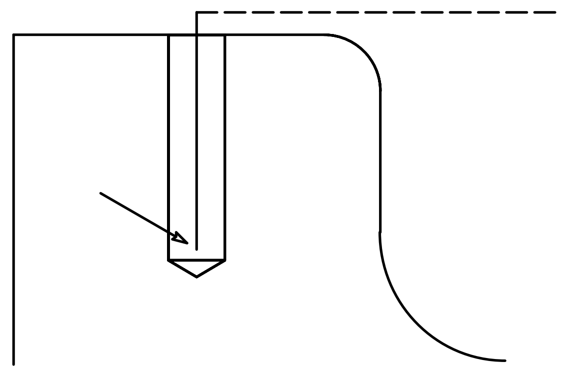
❑ In the **Blind Hole** field, enter **.100+.484*.3**.

You are ready to generate the tool path. All of the remaining options are the same and no changes are required.

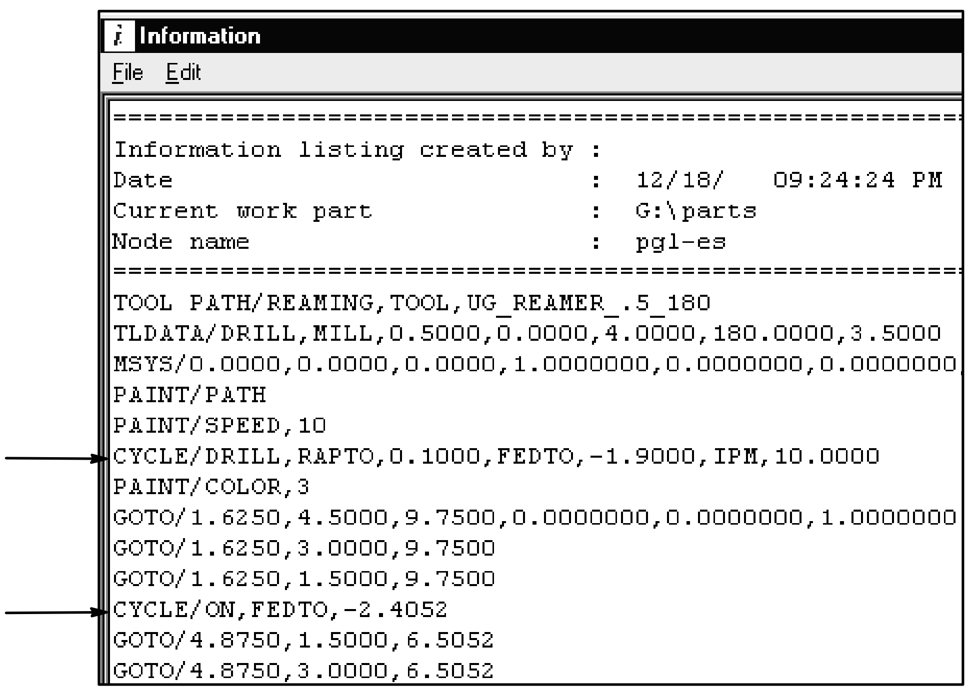
Step 5 Generate the tool path.

- ❑ Choose the **Generate** icon.

Notice that the tool stays away from the bottom of the holes the specified Depth Offset distance.



- ❑ Choose **List**.




```
============================================================
Information listing created by :
Date                              :  12/18/   09:24:24 PM
Current work part                 :  G:\parts
Node name                         :  pgl-es
============================================================
TOOL PATH/REAMING,TOOL,UG_REAMER_.5_180
TLDATA/DRILL,MILL,0.5000,0.0000,4.0000,180.0000,3.5000
MSYS/0.0000,0.0000,0.0000,1.0000000,0.0000000,0.0000000
PAINT/PATH
PAINT/SPEED,10
CYCLE/DRILL,RAPTO,0.1000,FEDTO,-1.9000,IPM,10.0000
PAINT/COLOR,3
GOTO/1.6250,4.5000,9.7500,0.0000000,0.0000000,1.0000000
GOTO/1.6250,3.0000,9.7500
GOTO/1.6250,1.5000,9.7500
CYCLE/ON,FEDTO,-2.4052
GOTO/4.8750,1.5000,6.5052
GOTO/4.8750,3.0000,6.5052
```

Compare the output of this operation to the output of the DRILL operation. Note the difference in the drill depths.

- ❑ **Close** the listing window.
- ❑ Choose **OK** to accept the operation.

Step 6 **Save** and close your part file.

This completes this activity and also the lesson.

SUMMARY

Drilling operation types allows for tool path creation for various types of holes. Numerous options which are available to you allow the control of depths, type of cycles generated and quality of the hole that is being created.

In this lesson, you learned how to:

- Create the necessary tools used for spot drill, drilling, and reaming operations
- Specify the options to define Cycle Parameter sets
- Set the tool depth and defined the tool offset to prevent the tool from gouging the bottom of a blind hole
- Set the Minimum Clearance Distance for tool positioning

(This Page Intentionally Left Blank)

Drilling Geometry Parent Groups

Lesson 11

PURPOSE

This lesson will show you how to create Drill Geometry Parent Groups. This Geometry Parent Group contains the geometry you will use to create various drilling type operations.

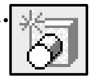
OBJECTIVES

Upon completion of this lesson, you will be able to:

- Create a Drill Geometry Parent Group
- Specify the drill geometry
- Specify and edit drill geometry from within an operation
- Optimize a drilling tool path
- use the Hole Making Module

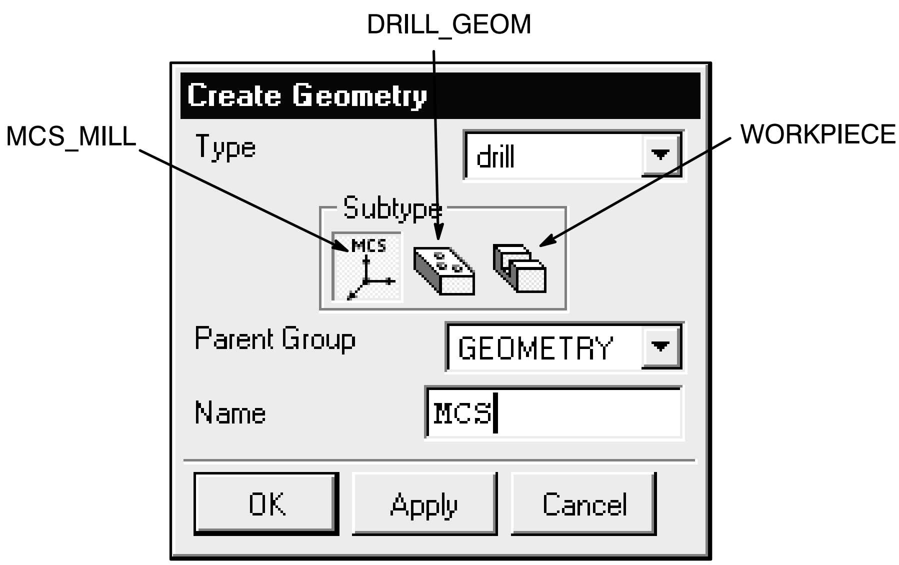
This lesson contains the following activities:

The Drill Geometry Parent Groups

Geometry Parent Groups contain the geometry that is used in an operation(s). To create a Geometry Parent Group you choose the Create Geometry icon. 

The Geometry Parent Groups that are used for Drill operation types are:

- MCS_MILL
- WORKPIECE
- DRILL_GEOM



The MCS_MILL geometry Parent is used to define the origin for subsequent tool path data based on the Machine Coordinate system.

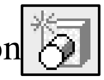
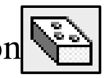
The WORKPIECE geometry Parent is typically used to assign Part and Blank material. It is also used in tool path verification.

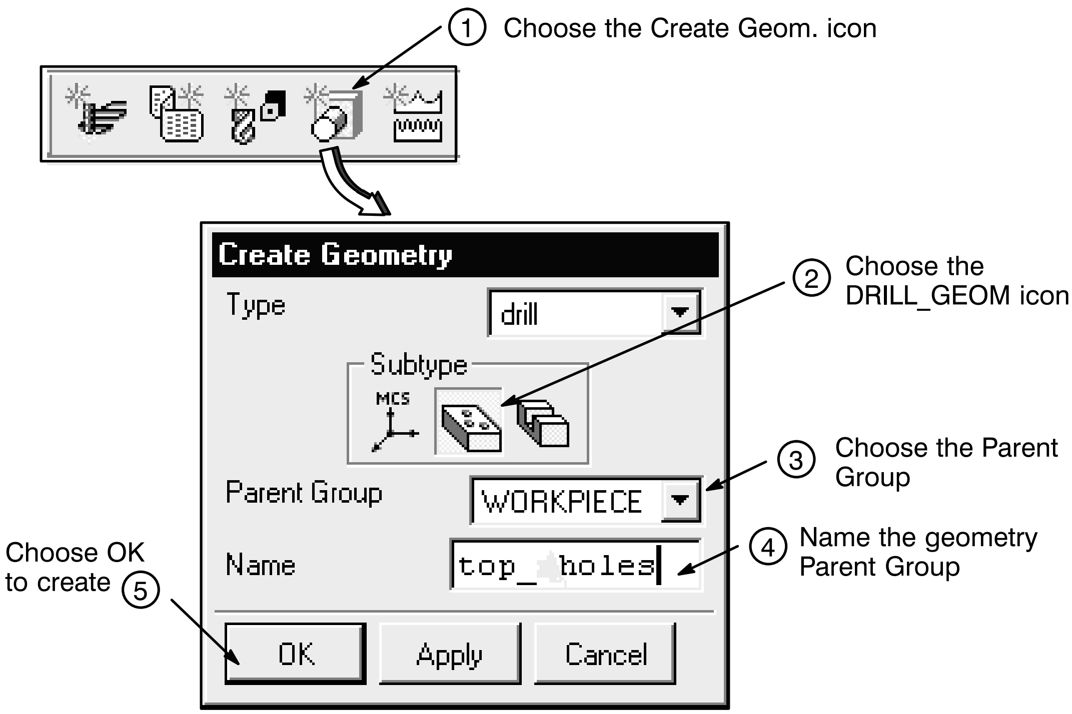
The DRILL_GEOM geometry Parent is used to define hole geometry used in drilling operations.

Creating the DRILL_GEOM Parent Group

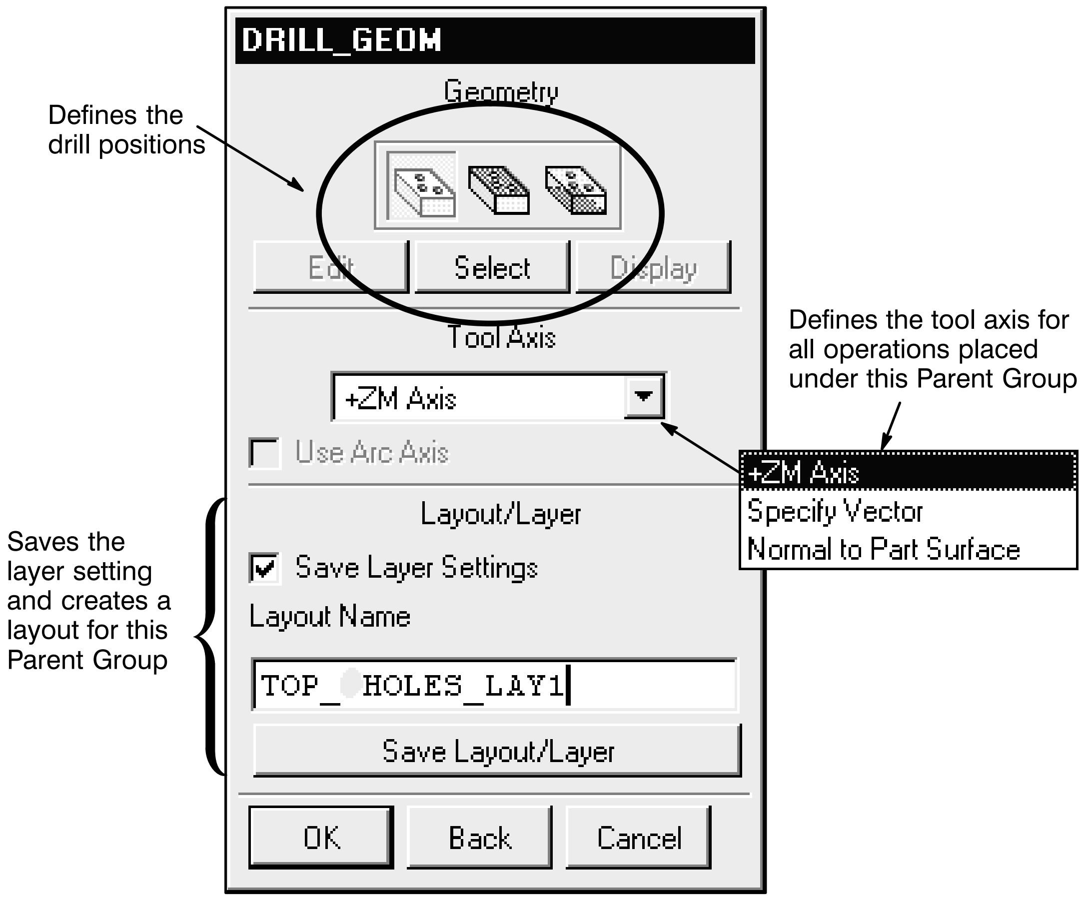
The DRILL_GEOM Parent Group is typically used to group geometry for specific types of drilling operations, such as all spot drilled or all holes that are drilled to a specific diameter.

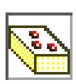
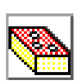
Use the following steps to create a DRILL_GEOM Parent:

- Choose the Create Geometry icon 
- Choose the DRILL_GEOM icon 
- Define the Parent Group where the part is defined (typically the WORKPIECE Parent Group)
- Name the geometry Parent Group used for drilling operations

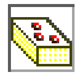


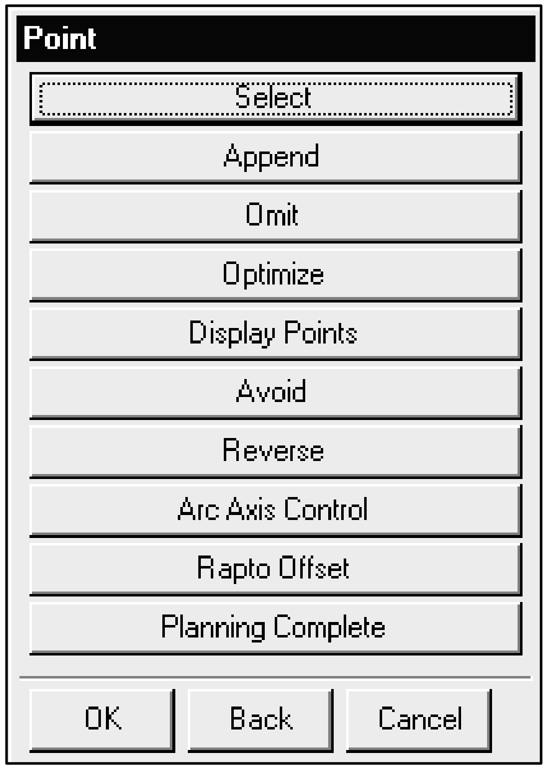
The DRILL_GEOM Dialog



- Holes  – provides different methods for hole selection
- Part Surface  – defines a plane to where all points are projected
- Bottom Surface  – determines the hole depth used if holes are just points or arcs defined in the model and can be an existing face or a generic plane

The Holes Icon

When you choose the Holes icon  the following dialog is displayed.

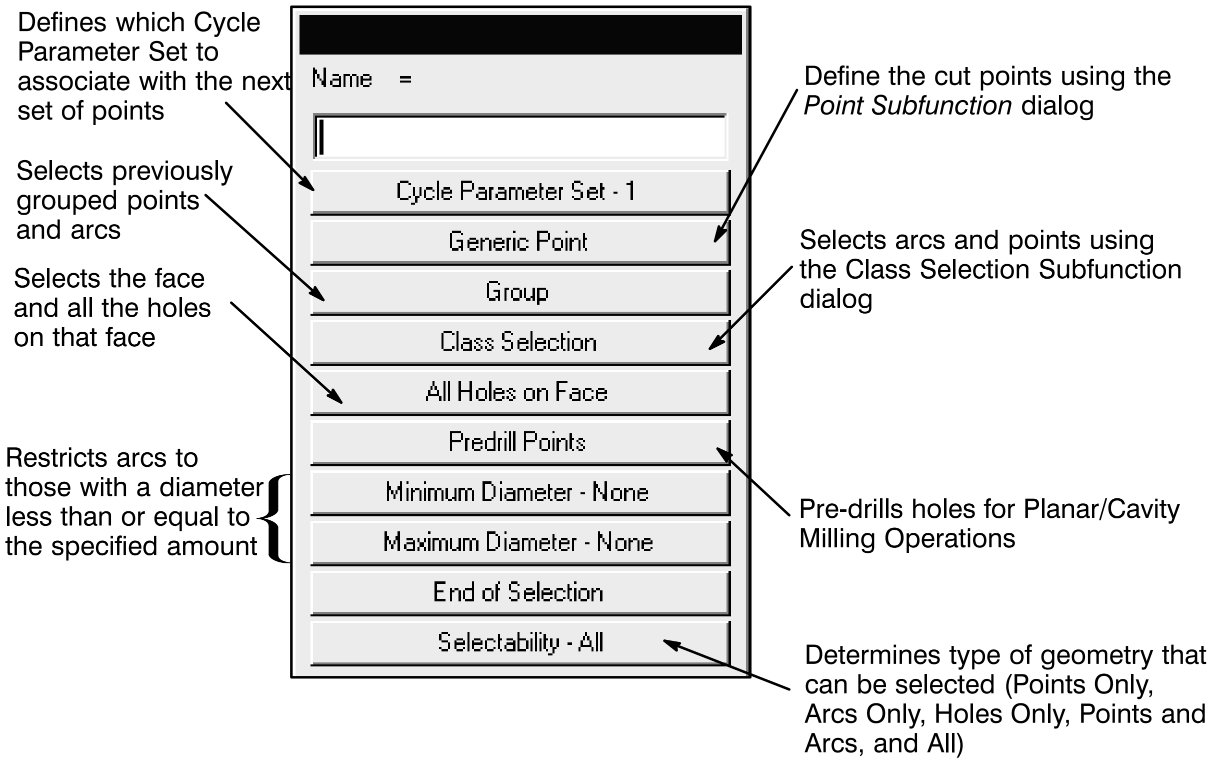


Only the first option, *Select*, is used when defining the geometry. The remaining options allow you to edit the already selected geometry.

You may use the following types of geometry to represent a hole in Drilling operations:

- Hole in a sheet body
- Hole in a solid body
- Point, arc, circles, or ellipses

Choose the Select option on the dialog when you are ready to define the geometry. There are many methods of selection available.



Selection Methods

- All Holes on Face
- Cursor Method
- Pre-Drilled Points

Minimum/Maximum Diameters

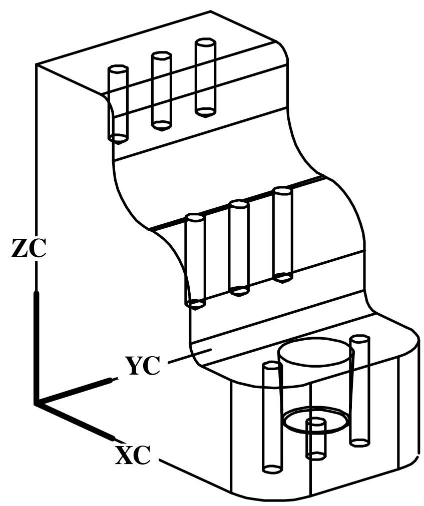
These options narrow the selection by entering a minimum and/or maximum hole diameter size. Then, only the holes within the diameter ranges are selectable.

Activity 11—1: Creating Drill Geometry Parent Groups

In this activity, you will create a Drill Geometry Parent Group containing drill geometry which can be used in several operations.

Step 1 Open the part file, rename it and enter the Manufacturing application.

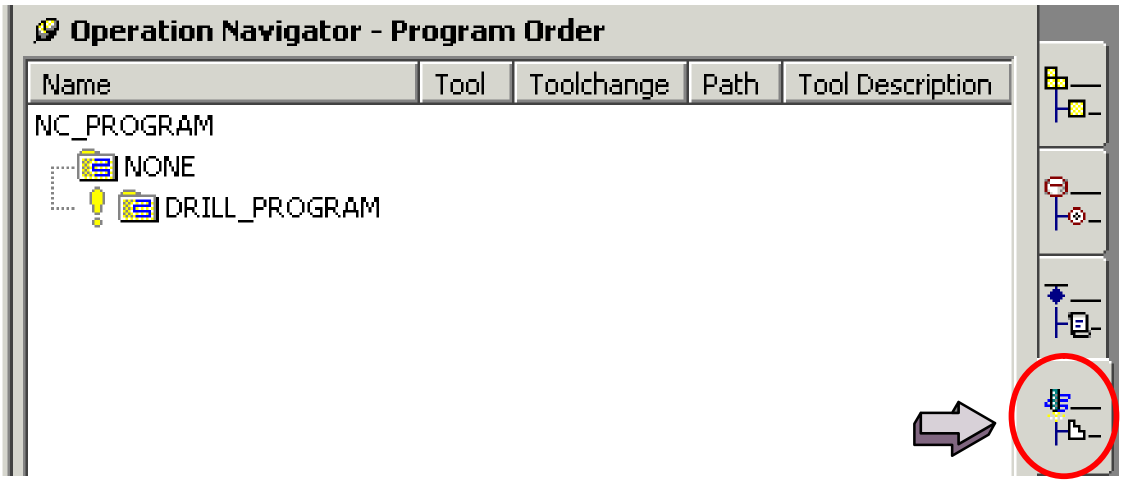
❑ Open the part file **mmp_register_2.prt**.



❑ Rename the part *****_register_2.prt** using the **File→Save As** option on the menu bar.

❑ If necessary, enter the **Manufacturing** application.

❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.

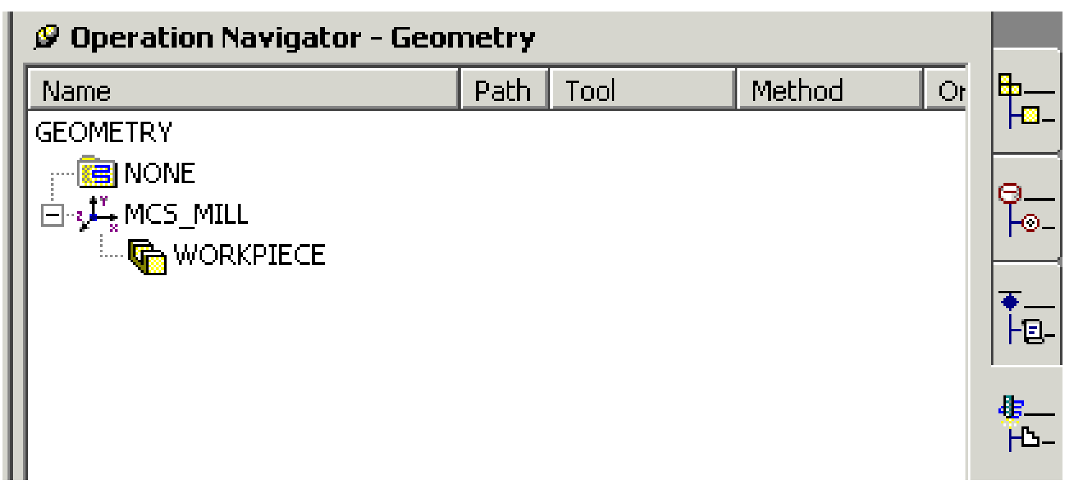


The Operation Navigator is displayed.

Step 2 Examine the WORKPIECE Parent Group.

❑ Change the Operation Navigator view to the **Geometry View.**

❑ **Expand** the Geometry Parent Group MCS_MILL.



Examine the geometry that is already selected in the WORKPIECE Parent Group.

❑ Double click on the **WORKPIECE** Parent Group.

The MILL_GEOM dialog is displayed.

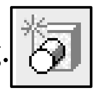
❑ Choose the **Display** icon.

The entire part is highlighted in the system color in the graphics window.

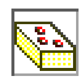
❑ Choose **Cancel.**

Step 3 Selecting the drill geometry.

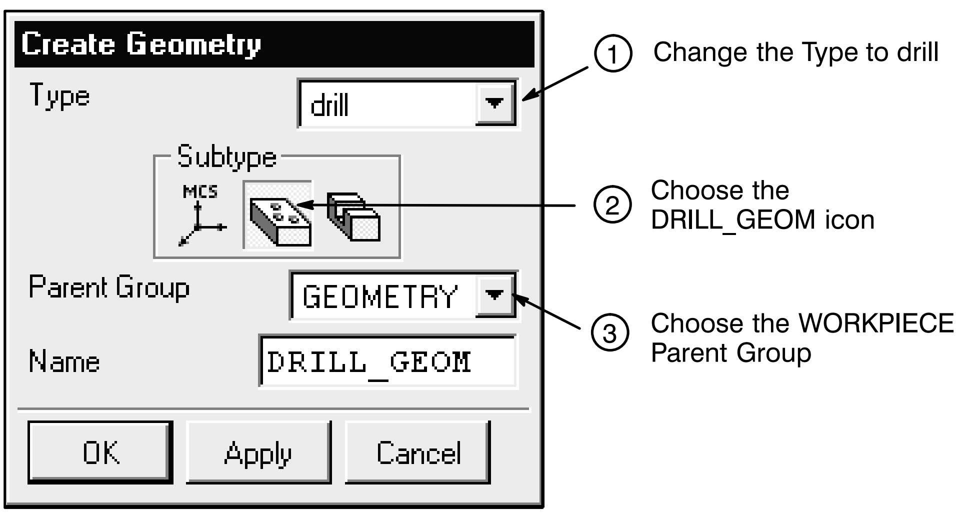
You will assign the geometry, required for drilling, to a DRILL_GEOM Parent Group.

❑ Choose the **Create Geometry** icon from the Manufacturing Create dialog. 

❑ If necessary, change the Type to **drill.**

❑ If necessary, choose the **DRILL_GEOM** icon  from the Create Geometry dialog**.**

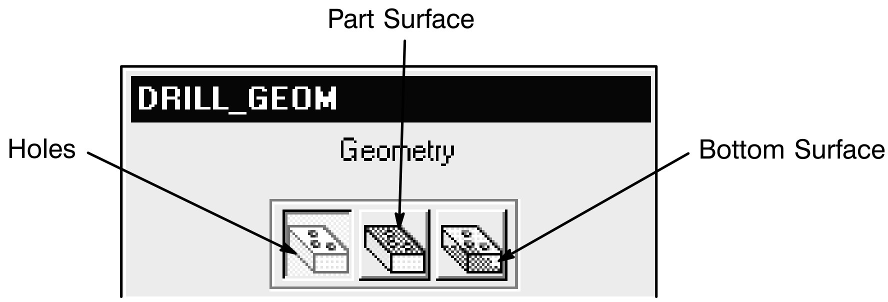
❑ Choose the Parent Group **WORKPIECE**.



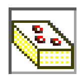
❑ Choose **OK.**

The DRILL_GEOM dialog is displayed. Notice the geometry types that are available for selection.

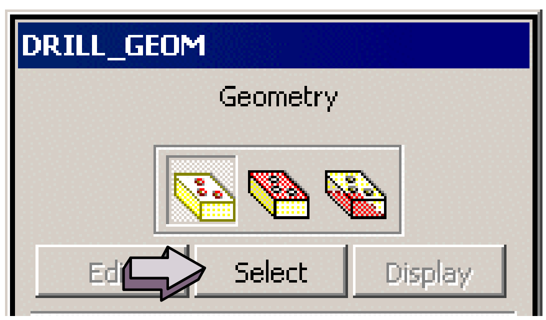
❑ Move the mouse over the three different geometry icons.



The choices are Holes, Part Surface and Bottom Surface.

❑ Choose the **Holes** icon. 

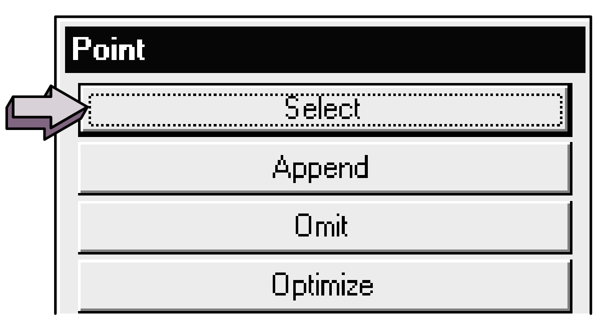
❑ Choose **Select** from the DRILL_GEOM dialog.



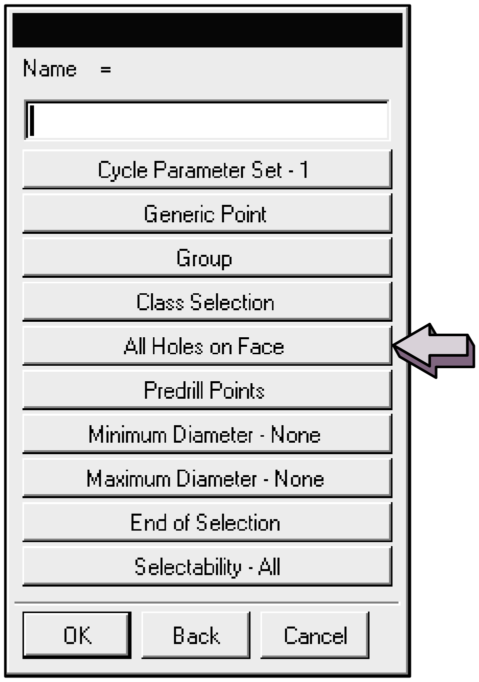
Step 4 Select the holes as indicated in the following steps.

You will use several different hole geometry selection methods.

❑ Choose **Select** from the Point dialog.

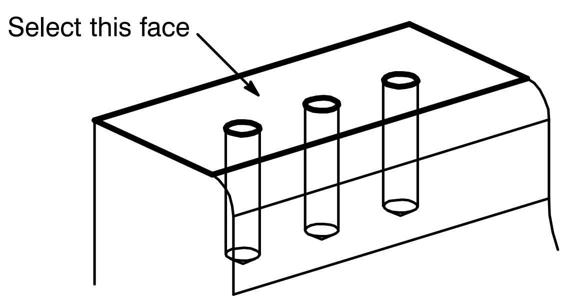


The following dialog is displayed.

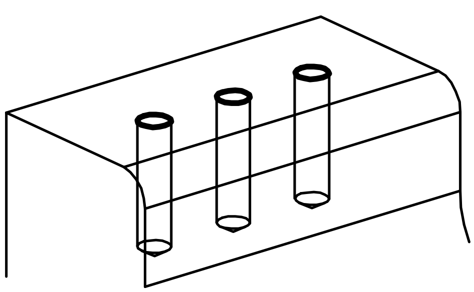


❑ Choose **All Holes on Face**.

❑ Select the face as indicated below.



The three holes on the face are selected and highlighted.

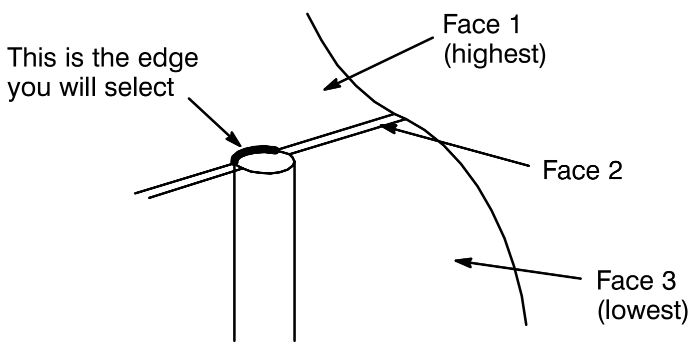


❑ Choose **OK** to accept these holes and return to the previous dialog.

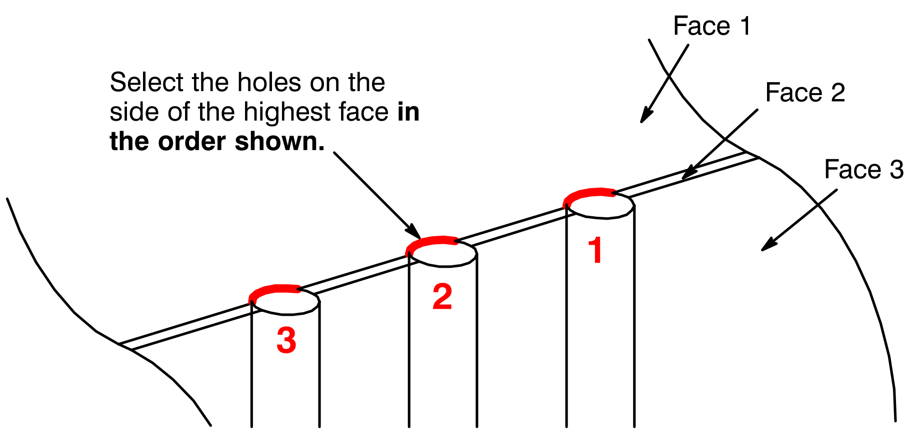
Step 5 Use the cursor to select the next three holes.

You will now select the holes in the middle of the part.

The tops of these holes are on three different faces, with the edges of these holes at different heights. You must be careful in the selection of the correct hole geometry.



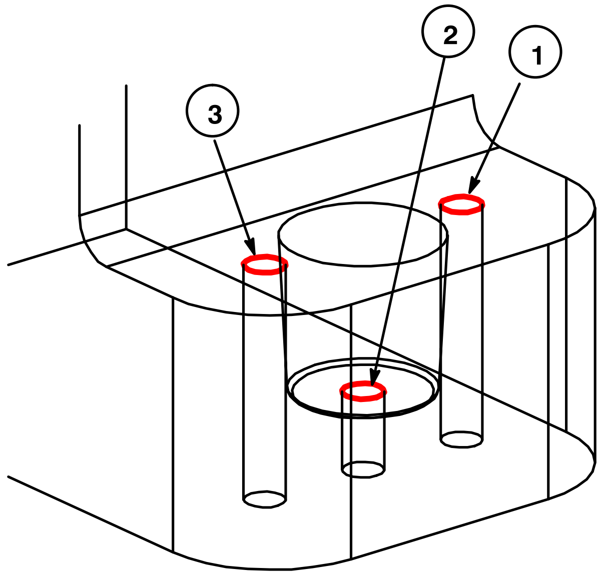
❑ Select the three holes in the order shown, with the cursor on the side of the hole that is on Face 1.



Notice that when you select the hole, only the portion of the hole that resides on the face is highlighted.

As you select the geometry, you may receive an error message saying “NOT A HOLE”. This means you have selected an invalid edge. Choose **OK** and try again.

❑ Select the remaining three holes.



❑ Choose **End of Selection** to indicate that you are through selecting geometry.

This completes the geometry selection.

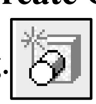
❑ Choose **OK**.

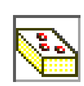
The DRILL_GEOM dialog is displayed.

❑ Choose **OK** from the DRILL_GEOM dialog.

Now you are going to create a second Geometry Parent Group containing drill geometry. This will contain the geometry to complete the drilling of the tapered cylinder.

Step 6 Creating the second Geometry Parent Group for drilling.

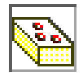
❑ Choose the **Create Geometry** icon from the Manufacturing Create dialog. 

❑ If necessary, choose the **DRILL_GEOM** icon  and then choose **OK.**

❑ If necessary, choose the Parent Group **WORKPIECE**.

❑ Name the Parent Group, **tapered_geom** and then choose **OK**.

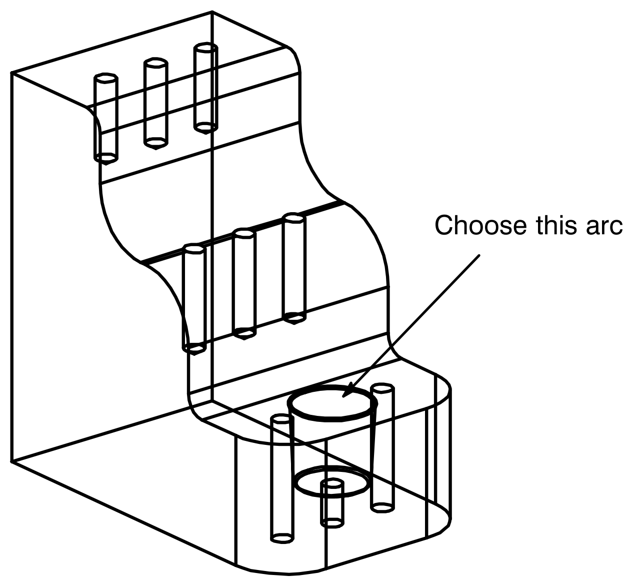
The DRILL_GEOM dialog is displayed.

❑ Choose the **Holes** icon,  then choose **Select** from the DRILL_GEOM dialog.

The Point dialog is displayed.

❑ Choose **Select** from the Point dialog.

❑ Using the cursor method, choose the geometry as shown.



❑ Choose **OK** when the selection is complete.

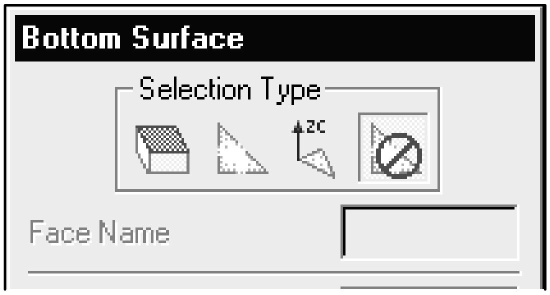
❑ Choose **OK** to return to the DRILL_GEOM dialog.

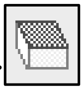
You are going to define a bottom surface. This will define the depth that can be drilled to.

Step 7 Defining the bottom surface.

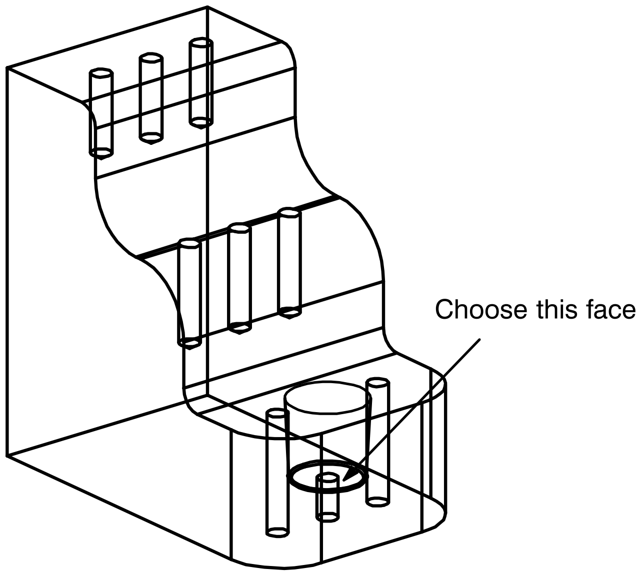
❑ Choose the Bottom Surface icon  and then choose the **Select**.

The Bottom Surface dialog is displayed.



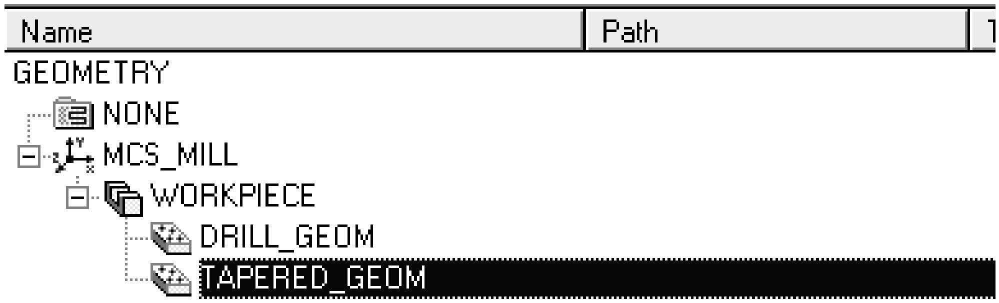
❑ Choose the **Face** icon. 

❑ Using the cursor method, select the geometry as shown.



❑ Choose **OK** until you return to the Operation Navigator.

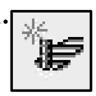
Examine the Operation Navigator. The two Drill Geometry Parent Groups are listed under the WORKPIECE Parent.



You are ready to start programming the part. In the interest of time you will not spot drill the holes, you will only create operations necessary to drill the holes.

Step 8 Creating the drilling operation.

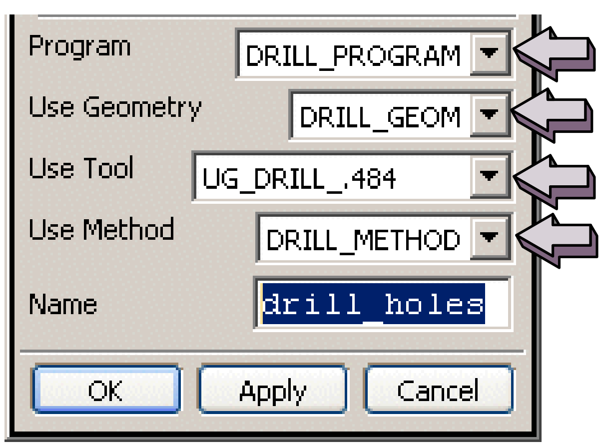
❑ Change the Operation Navigator to the **Program Order View**.

❑ Choose the **Create Operation** icon. 

❑ Choose the DRILLING Subtype.

❑ Name the operation **drill_holes.**

❑ Set the following Parent Groups:

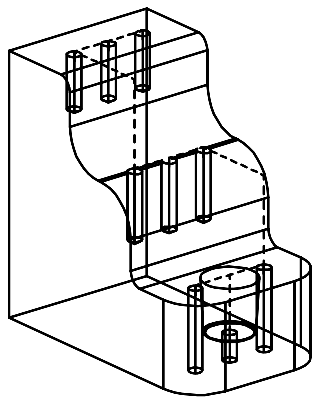


❑ Choose **OK.**

You do not need to specify the Standard Drill cycle. You will use the default setting.

Step 9 Generating the tool path.

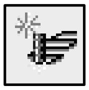
- ❑ **Generate** the tool path.

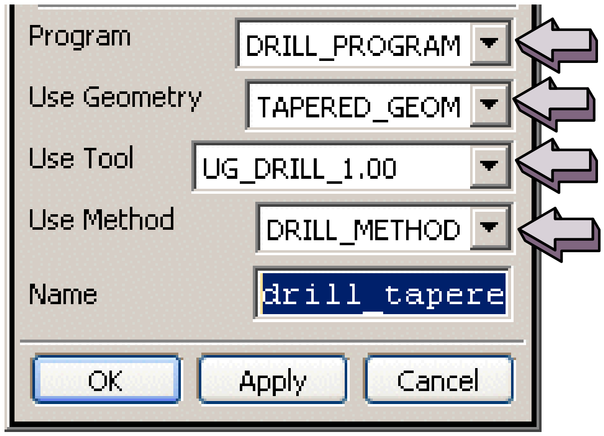


The tool path is generated and drills the holes in the order selected. Later you will optimize the tool path and reorder the drill pattern.

- ❑ Choose **OK** to accept the tool path.

Step 10 Creating the second operation to drill the tapered cylinder.

- ❑ Choose the **Create Operation** icon. 
- ❑ Choose the DRILLING Subtype.
- ❑ Name the operation **drill_tapered_cylinder**.
- ❑ Set the following Parent Groups:

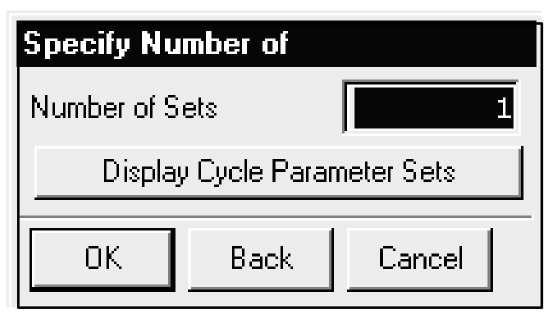


❑ Choose **OK.**

❑ Click on **Standard Drill** on the Main property page from the Drilling dialog.

❑ Choose **Standard Drill** from the list of cycle types.

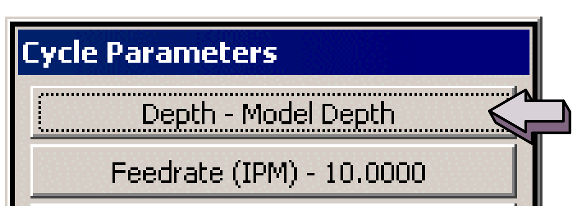
The Specify Number of dialog is displayed.



❑ Choose **OK**.

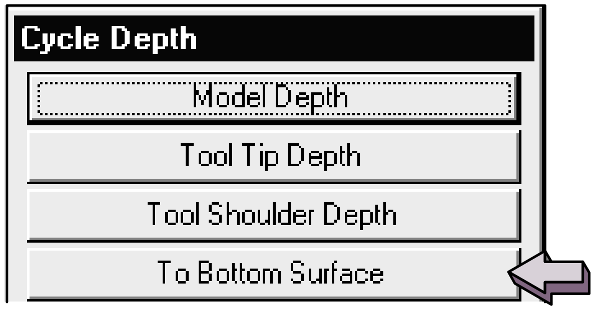
The Cycle Parameters dialog is displayed.

❑ Choose the Depth-Model Depth button.



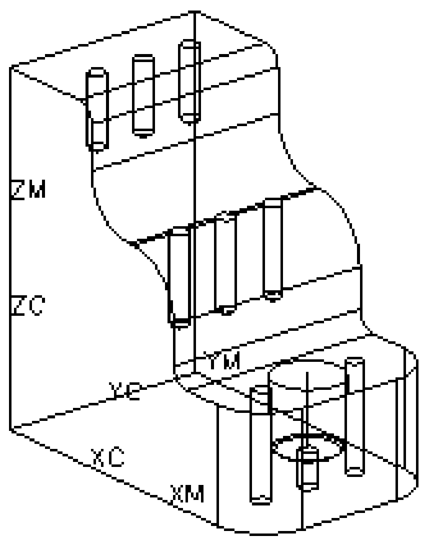
The Cycle Depth dialog is displayed.

❑ Change the **Depth** to **To Bottom Surface**.



❑ Choose **OK.**

❑ **Generate** the tool path.



The tool path is generated. The tapered cylinder is drilled using a larger drill.

❑ Choose **OK** to accept the tool path.

❑ **Save** the part file.

❑ Do not close the part file since you will be using it in the next activity.

Optimizing the Tool Path

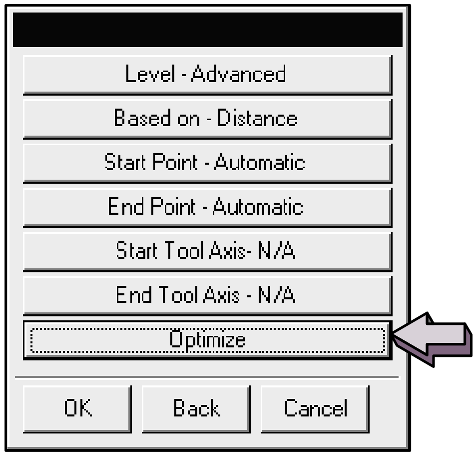
In some cases the tool path that is generated may not be the most efficient tool path in terms of motion. Optimize allows you to rearrange the tool moves into a more efficient order.

Here is a summary of the Optimize dialog:

- **Shortest Path –** arranges the points in the order required to minimize total machining time
- **Horizontal Bands** and **Vertical Bands –** are used for confining the tool path; these bands are used for other machining constraints, such as clamp locations, machine travel limits, table size, etc.
- **Repaint Points –** repaints all the points after each optimization if toggled to Yes

Shortest Path Options

When you choose the Shortest Path option the following dialog is displayed.



Here is a summary of the Optimization Parameters dialog:

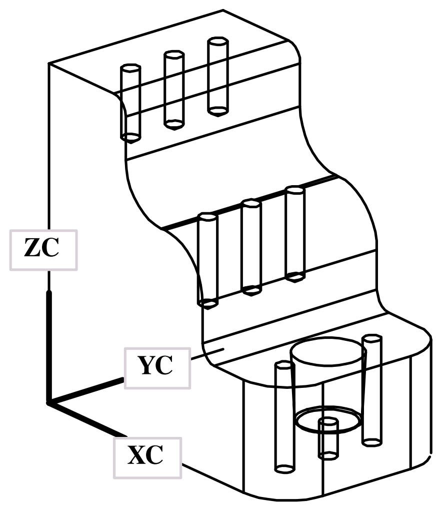
- **Level** *Standard* or *Advanced* - Refers to the process of analysis that you want to use in determining the shortest tool path; advanced increases machine time efficiency the greatest amount
- **Based On –** *Distance* is the only option for a fixed axis tool path; variable axis tool paths can take the tool axis into account when determining machining efficiency
- **Start Point –** controls the start point of the tool path
- **End Point –** controls the end point of the tool path
- **Start Tool Axis –** is for variable axis tool paths only and controls the tool axis at the beginning of the cutting motion
- **End Tool Axis –** is for variable axis tool paths only and controls the tool axis at the end of the cutting motion
- **Optimize –** initiates the optimization process

Activity 11–2: Optimizing A Tool Path

In this activity, you will edit a drilling operation and optimize the tool path using the shortest path method.

Step 1 Open the part file and enter Manufacturing.

❑ Continue using the part file *****_register_2.prt.**



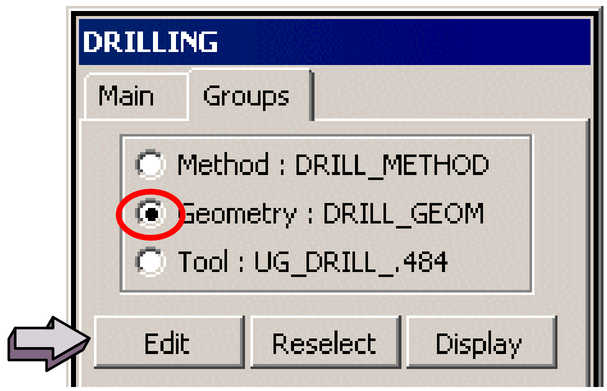
Step 2 Optimizing the tool path.

You are going to change the order that the holes are drilled and edit the Geometry Parent Group from within the operation.

❑ In the Operation Navigator, double click the **drill_holes** operation.

❑ Choose the **Geometry** button located at the top of the DRILLING dialog Groups property page.

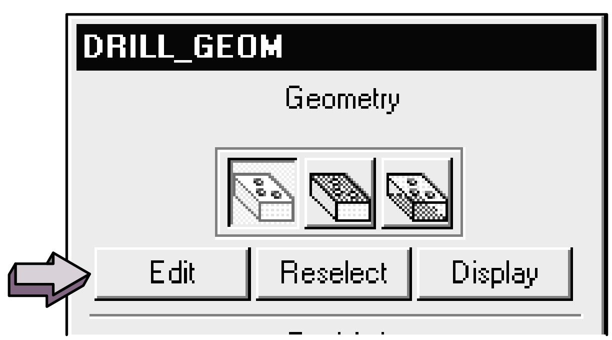
❑ Choose **Edit**.



❑ Choose **OK** to the Group Editing message.

The DRILL_GEOM dialog is displayed.

❑ Choose **Edit**.



The Point dialog is displayed.

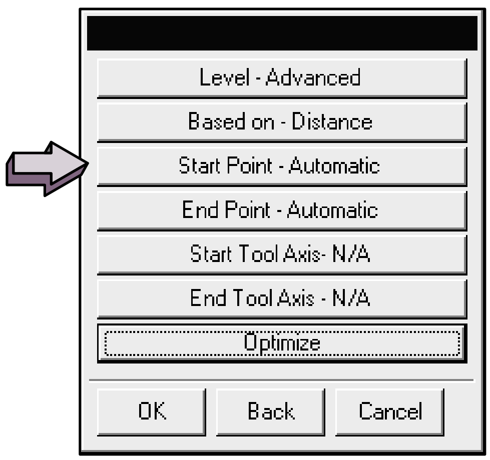
❑ Choose **Optimize** from the list.

The Optimize option dialog is displayed.

❑ Choose **Shortest Path**.

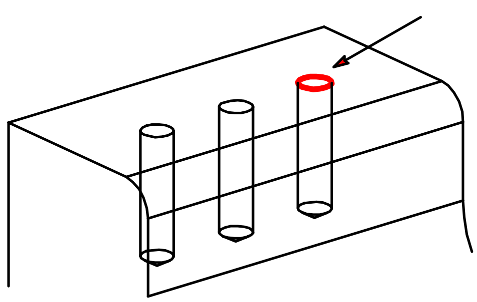
The Optimization options dialog is displayed.

❑ Choose **Start Point – Automatic**.



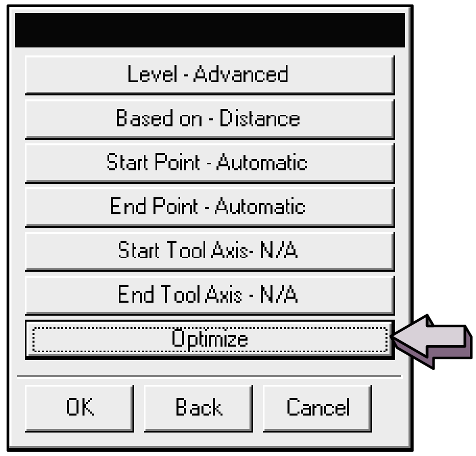
The Select Start Point dialog is displayed.

❑ Select the hole as shown below.

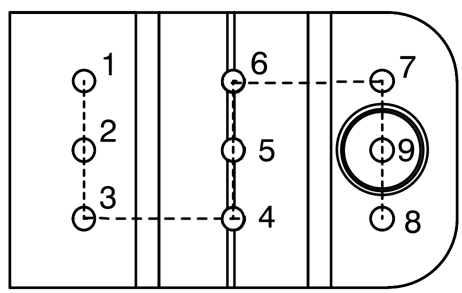


The option dialog is displayed.

❑ Choose **Optimize**.



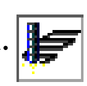
The points are renumbered and a menu is displayed. By comparing the *Before Optimization* and *After Optimization* values, you can see that the tool path was shortened by approximately 1.3 inches.



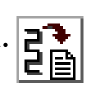
Tool path after Optimization

❑ Choose **Accept**.

❑ Choose **Planning Complete** and then **OK**.

❑ Choose the **Generate** icon. 

List the tool path.

❑ Choose the **List** icon. 

The tool path is listed in the listing window. Notice the CYCLE statement and the parameter settings determined by the Cycle Parameter Set.

- ❑ **Close** the listing window.
- ❑ Choose **OK** to accept the tool path.
- ❑ **Save** the part file.

This completes this activity.

You will now have a brief introduction to the new Hole Making Module which can greatly reduce the effort required in the generation of tool paths used in hole making operations. Accompanying the introduction will be a brief activity to show the functionality of this module.

Introduction to Hole Making

The Hole Making module creates operations and subsequent tool paths for the machining of drilled, tapped, counter-bored, counter-sunk, and reamed holes. The module is highly automated and works with features that are characteristic of various hole types.

Currently, the Hole Making module supports feature and attribute identification used in the process to create operations. Feature identification uses User Defined (commonly referred to as UDF's) and standard features for hole detection and operation creation. Attribute identification allows non-featured based geometry such as points, edges, arcs and cylindrical faces to have CAM related information assigned that is recognized by the Hole Making module. These attributes are assigned through a specific tagging function that is a standard feature of the Hole Making Module.

The Machining Environment Required for Hole Making

The CAM Setup that you initially choose determines the templates that will be used for machining. For Hole Making, the CAM Session Configuration that is required is **hole_making.** This template is a part file that contains predefined manufacturing features, tools, operations and rules that are used to automate the Hole Making process. The rules are most critical to the success of the hole making process and in addition to describing how the features will be machined, determine which groups, operations and tools that will be used.

Analyzing existing features

If you display the properties of the various hole types you will normally see the name of the feature that matches the feature defined in the Hole Making template. This correlation allows the recognition of features that need to be machined in the part. If the feature names do not match the names in the templates, the features will not be machined.

Analyzing existing objects

If you display the properties of geometric elements (created by the tagging function), such as points or arcs, you will see that the name of the attribute associated with this object matches the object in the template. This correlation allows the recognition of attributed objects for machining.

The template part file used by the Hole Making module

The template part file, **hole_making.prt**, contains predefined manufacturing features (i.e. counterbored hole), feature groups, tools, operations, machining rules and adopted operations.

Feature groups contain features that are recognized by the Hole Making module to create operations.

The generic template, provided with the Hole Making module, recognizes the following features: simple hole, counterbored hole, countersunk hole and symbolic thread; User Defined Features (UDF's): cap screw counterbore, cap screw countersink, fit hole, standard thread; and the following combination features: thread simple hole, thread counterbore hole, thread countersink hole. This file can be customized to your own method of making holes, but requires the knowledge and application of Knowledge Fusion

Adopted operations are operations that are made known to the Knowledge Fusion processor, which then applies machining rules. The Knowledge Fusion processor is the rules based engine that is used by the Hole Making module to define, modify and process the rules used for determining the various types of holes that are to be processed.

The Geometry View of the Operation Navigator associates feature groups to manufacturing features. Feature groups organize operations based on feature types. The feature group, named SIMPLE_HOLE, for example, contains all operations that applicable to machining a simple hole feature.

Feature groups associate operations with adopted operations in Knowledge Fusion. The Knowledge Fusion Navigator displays these adopted operations. Through the matching of the adopted operations from the template with the machining features of the part, the Hole Making module identifies the appropriate feature groups and then copies the adopted operation into the part file.

The machining rules that are part of the adopted operations determine which operations to use based on the properties and attributes of the features that are contained in the part. The operations that are ignored due to rule violations are suppressed. Note that these machining rules have been predefined and can be customized. Customization involves knowledge of the Knowledge Fusion language, a subset of Intent, which will not be covered in this session.

Creating Hole Making Operations

Generally speaking, creating holes, using the Hole Making module, is simple if you use the following guide lines:

- Make sure that you have tools defined that can be used to create all holes required. You can always edit an operation to add a tool, but hole generation is more complete if all tools have been previously defined.
- Choose the proper tool axis to identify holes to be machined. This is done in the Geometry parent group MCS. If you select all axes, then all holes, regardless of location in the object will be selected. Normally all axes is the selected option when machining on a four or five axis machining center.
- Define the object or body containing the holes (geometry), ALWAYS, as the parent group WORKPIECE. Defining geometry under any other parent group will result in a failed operation.
- When creating holes, based on features, i.e. SIMPLE_HOLE, verify the Geometry parent group as being the WORKPIECE. If not, change to the WORKPIECE parent.

Optimizing tool paths generated by the Hole Making module

Optimization uses Machining Feature Templates (as mentioned previously, these templates are used to identify a hole feature and then assign a machining method for processing) to determine the most efficient method and order of operations used to machine the hole.

Optimization reorders operations to eliminate redundant tool changes and sequences features to minimize the distance that a tool will travel. There are three different procedures that are use to obtain optimized tool paths. They are:

- through selecting a minimum number of tools based on tool query parameters of each operation (these tools are then retrieved from the tool library)
- operations that use the same tool are grouped together to minimize tool changes
- positioning is based on tool and feature optimization

Optimization is performed in the Program Order view by selecting a program parent or program group. Some general rules that optimization follows are:

- Selected operations within the same Program in consecutive order are reordered together
- Operations selected from different Program subgroups are reordered separately
- If the Program contains a subprogram, the optimization is applied to the subprogram separately without breaking up the subprogram

Activity 11—3: Utilizing the Hole Making Module

In this activity, you will use the Hole Making Module to automate the drilling, tapping, counterboring and deburring of numerous holes in a part.

Step 1 Open the part file, rename it and enter the Manufacturing application.

❑ Open the part file **mmp_plate_hole_making.prt**.

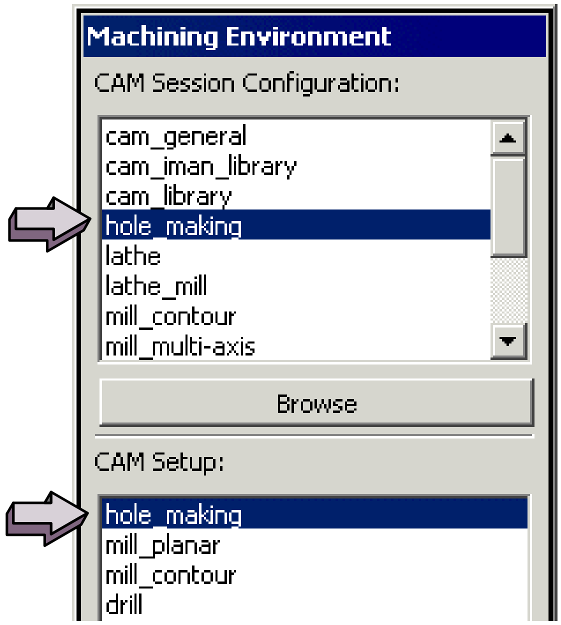
❑ Choose **Application → Manufacturing**.

The Machining Environment dialog is displayed.

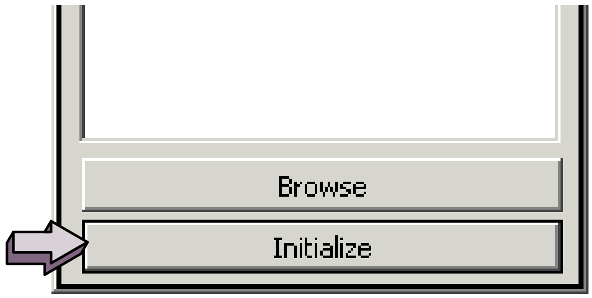
Step 2 Defining the Manufacturing Environment.

The Hole Making Module has its own configuration. You will choose this from the list of available configurations.

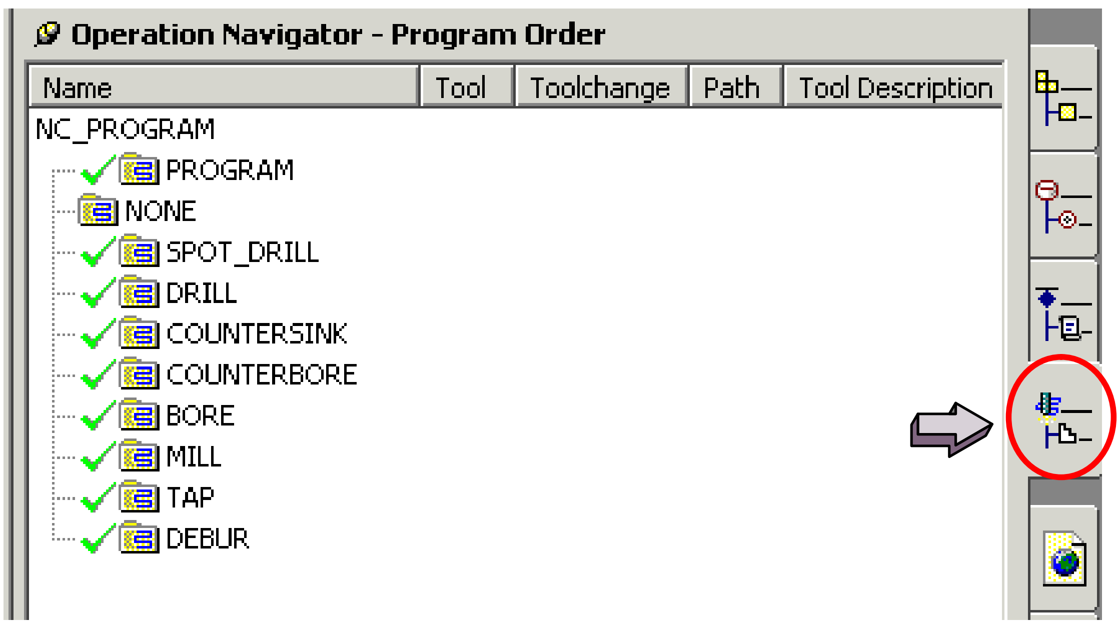
❑ Choose **hole_making** as the Configuration and then **hole_making** as the CAM Setup.



❑ Choose Initialize.



❑ If necessary, activate the Operation Navigator by selecting the Operation Navigator tab from the resource bar and display the **Program Order View**.



The Operation Navigator is displayed. You will notice that in the Program Order View, several sub-program groups have been created. Hole Making organizes the operations into groups, based upon the type of machining that will be required to generate the hole.

Do you remember the earlier guidelines for successful hole making? To refresh your memory:

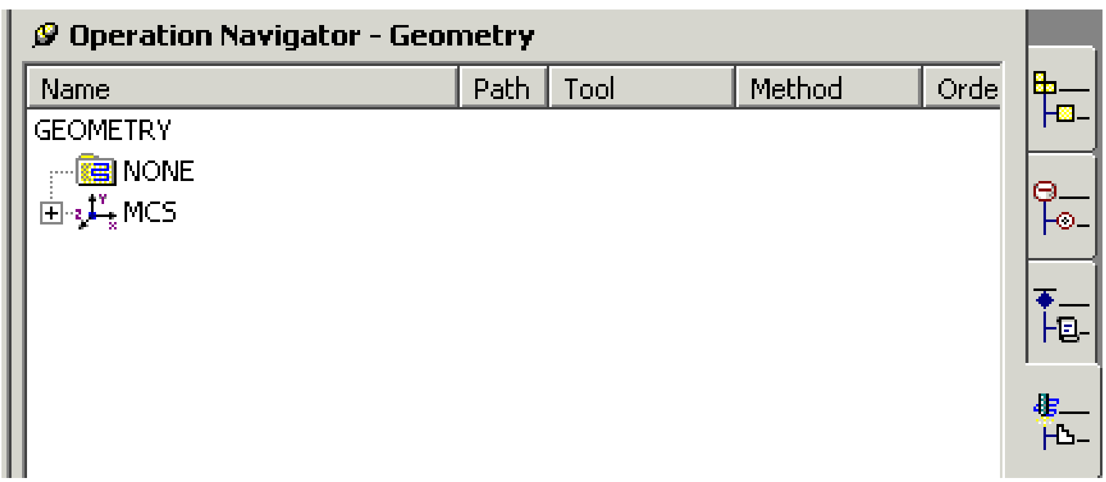
- Make sure that you have tools defined that can be used to create all holes required. You can always edit an operation to add a tool, but hole generation is more complete if all tools have been previously defined.

- Choose the proper tool axis to identify holes to be machined. This is done in the Geometry parent group MCS. If you select all axes, then all holes, regardless of location in the object will be selected. Normally all axes is the selected option when machining on a four or five axis machining center.
- Define the object or body containing the holes (geometry), ALWAYS, as the parent group WORKPIECE. Defining geometry under any other parent group will result in a failed operation.
- When creating holes, based on features, i.e. SIMPLE_HOLE, verify the Geometry parent group as being the WORKPIECE. If not, change to the WORKPIECE parent.

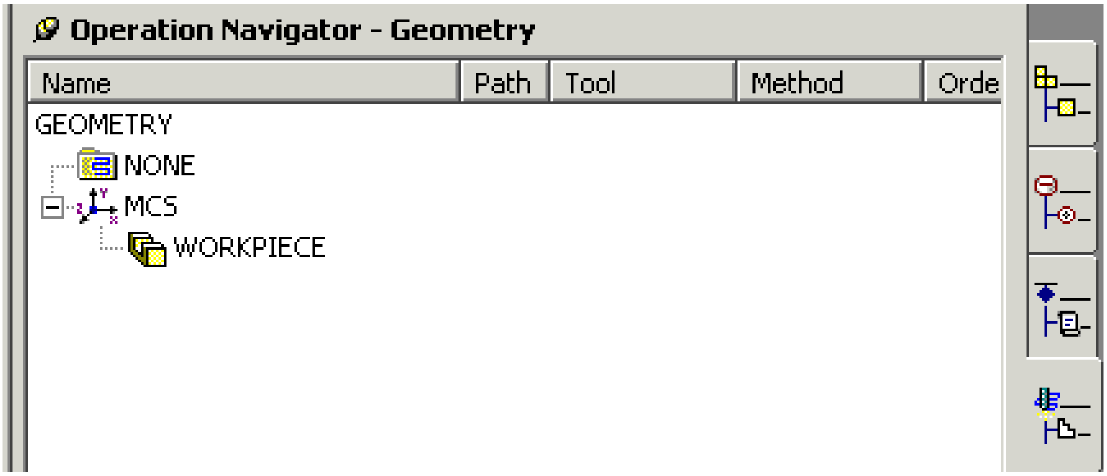
The third and fourth bullet item refers to defining geometry under the WORKPIECE parent group. You will now establish the WORKPIECE parent.

Step 3 Establishing the WORKPIECE Parent Group Object.

❑ Change to the Geometry View of the Operation Navigator.



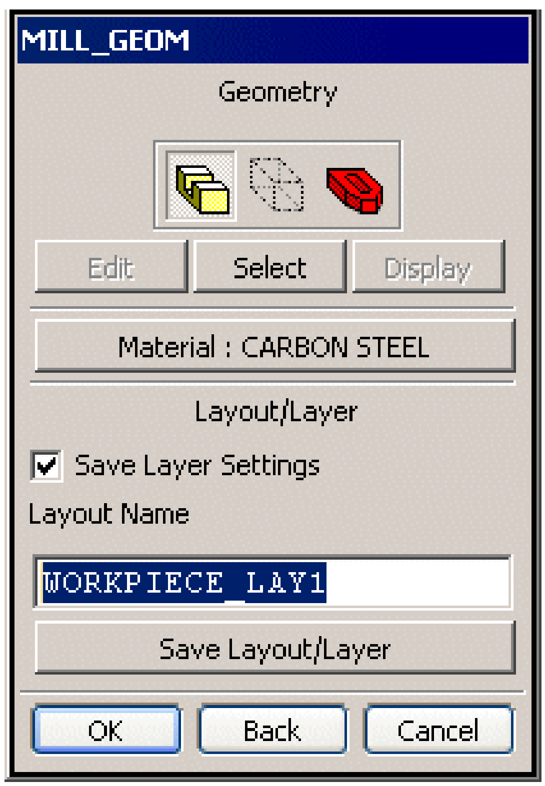
❑ Expand the MCS parent group by clicking on the "+" next to the group name.



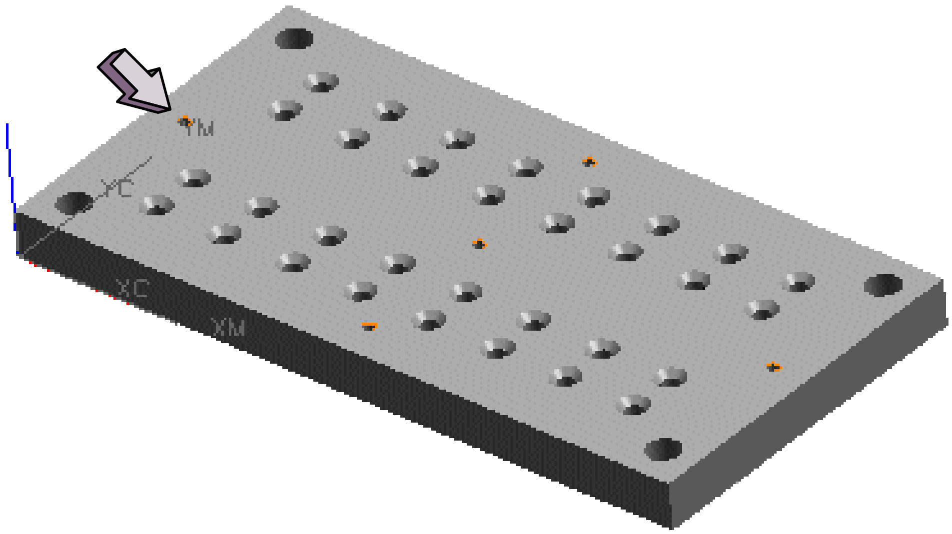
❑ Double-click on the WORKPIECE parent object.

The MILL_GEOM dialog is displayed.

❑ Select the **Part** icon, then choose the **Select** button.



❑ Select the plate from the graphics window.



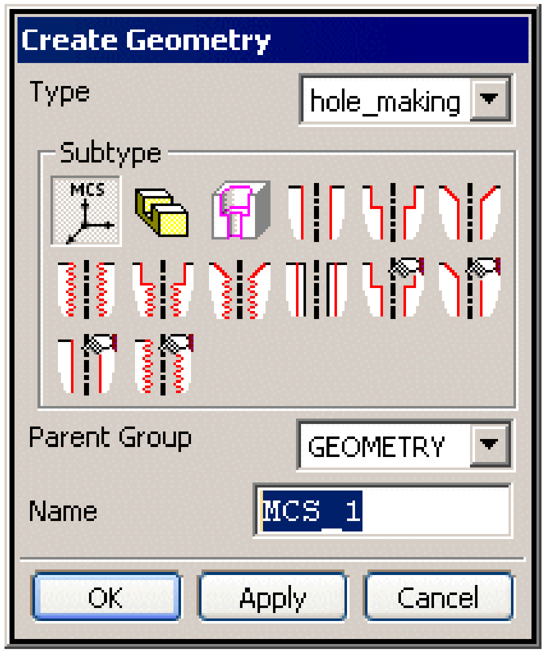
❑ Choose **OK** twice.

You have completed all of the preliminary work required to set-up Hole Making. You are now ready to create the operations.

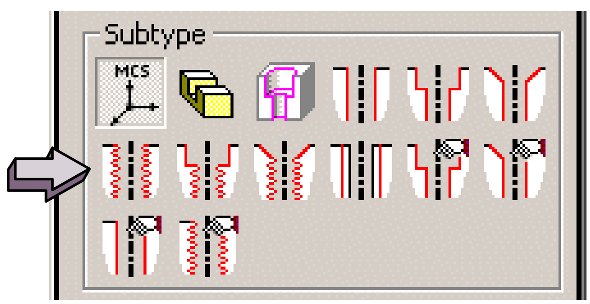
Step 4 Creating Hole Making operations.

❑ Choose the Create Geometry icon. 

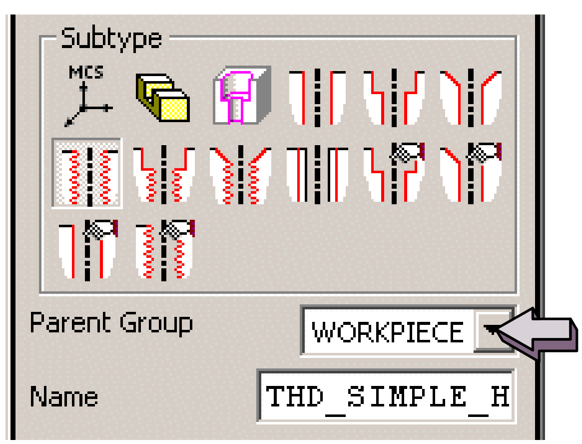
The Create Geometry dialog is displayed.



❑ Choose the **THD_SIMPLE_HOLE** subtype.



❑ Set the Parent Group to WORKPIECE

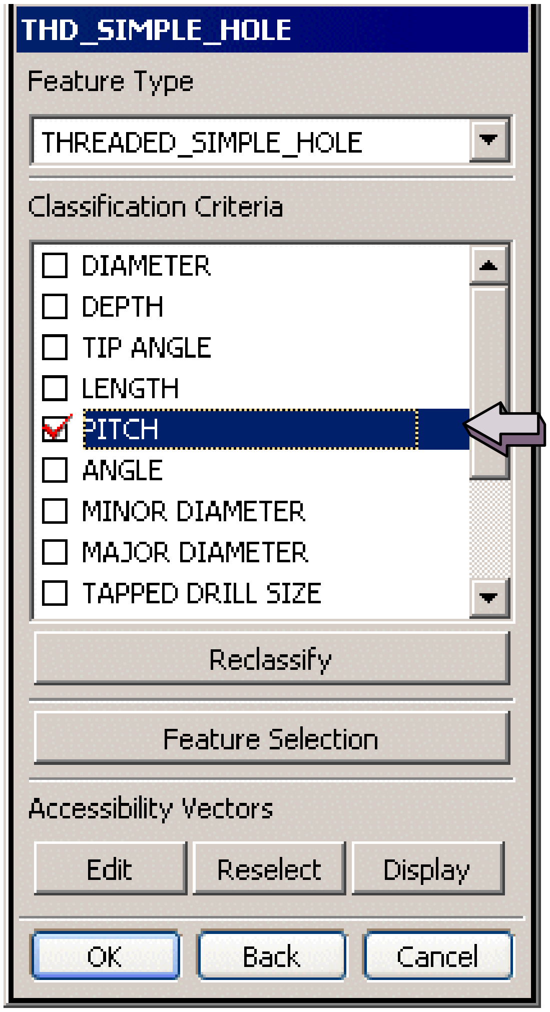


❑ Choose **OK.**

The Hole Making module compares all of the model's features to determine which holes qualify as threaded simple holes. After the comparison is complete, operations are created and tools are selected (if available) to perform the required operation(s). An Information window is displayed, showing features found, groups and operations created.

❑ Examine the Information window to observe the actions performed on this plate by the Hole Making module.

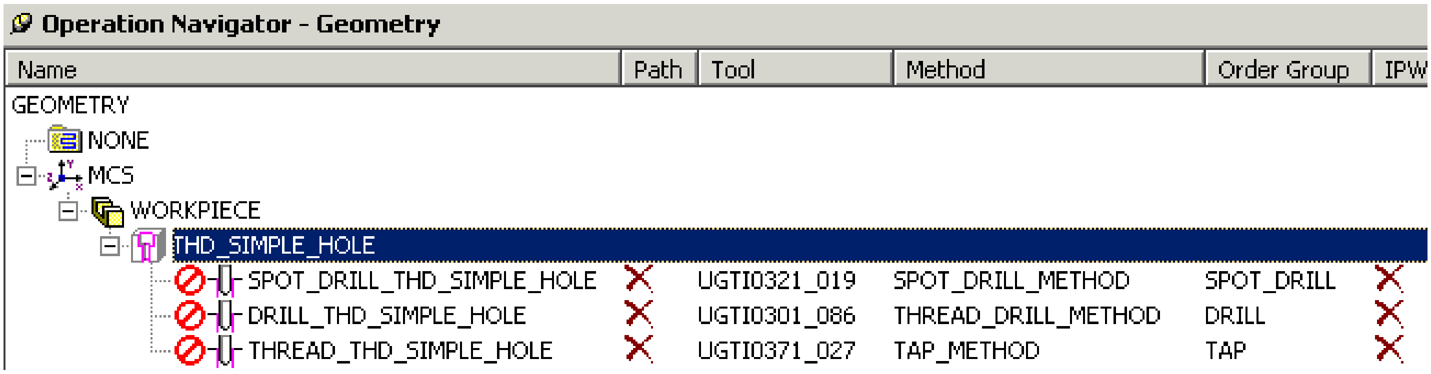
Along with the Information window, a THD_SIMPLE_HOLE dialog is displayed, which lists the criteria used for classification of the threaded holes. In this particular situation, Classification Criteria will be based on the thread pitch only.



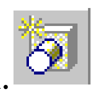
❑ Choose **OK** in the THD_SIMPLE_HOLE dialog to accept Feature Type and Classification Criteria.

❑ Close the Information window.

❑ Expand the **THD_SIMPLE_HOLE** parent group, in the Operation Navigator.

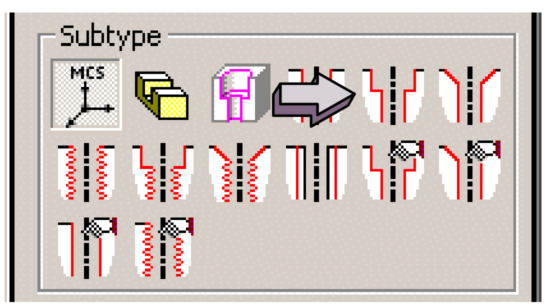


Three operations have been created. You will now create the operations necessary for a counterbored hole.

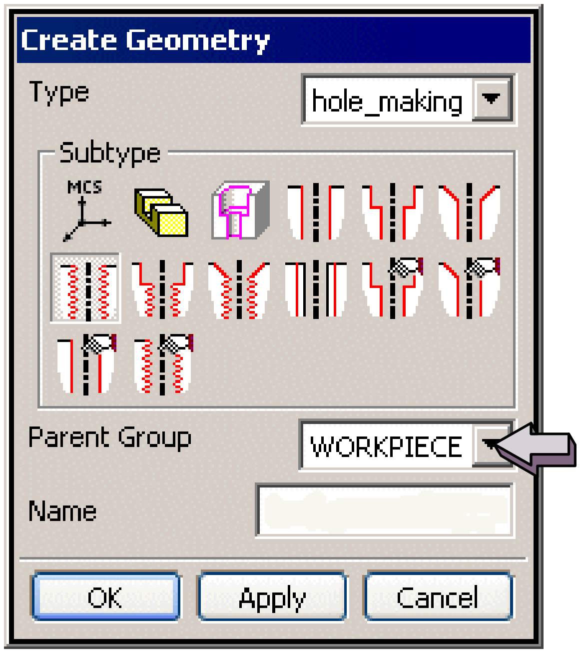
❑ Choose the Create Geometry icon. 

The Create Geometry dialog is displayed.

❑ Choose the **CB_HOLE** subtype.



❑ If necessary, set the Parent Group to WORKPIECE



❑ Choose **OK.**

All features are searched to find those that match the counterbored hole criteria; operations are created and cutting tools are selected, if found, based on the selection criteria from the library.

❑ Dismiss the Information window.

❑ Choose **OK** in the CB_HOLE dialog.

❑ Expand the CB_Hole parent group.

Note that four operations, spot drilling, drilling, counterboring and deburring have been created. Notice the CBORE_CB_HOLE operation. This operation does not have a tool assigned since the standard library does not contain an appropriate sized counterbore tool. You will create this tool in the next step.

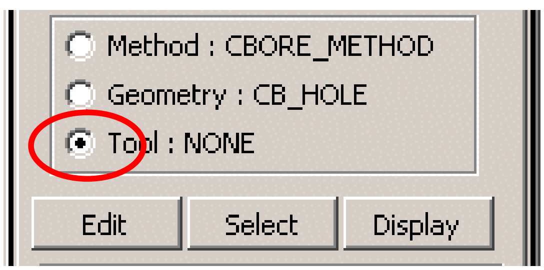
Step 5 Creating tools when appropriate tools are not found in the tool library.

❑ Double-click on the **CBORE_CB_HOLE** operation.

Notice that this is a regular drilling operation, comparable to one that would be created manually. You will now define the tool for this operation.

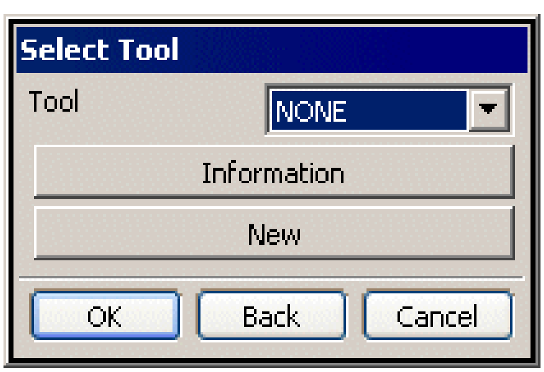
❑ Choose the Groups Property page (tab) from the HOLE_MAKING dialog.

❑ In necessary, activate the Tool radio button.



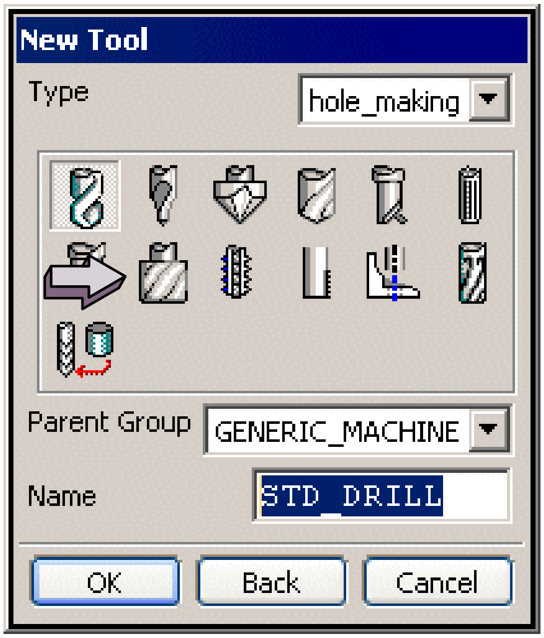
❑ Choose **Select**.

The Select Tool dialog is displayed.



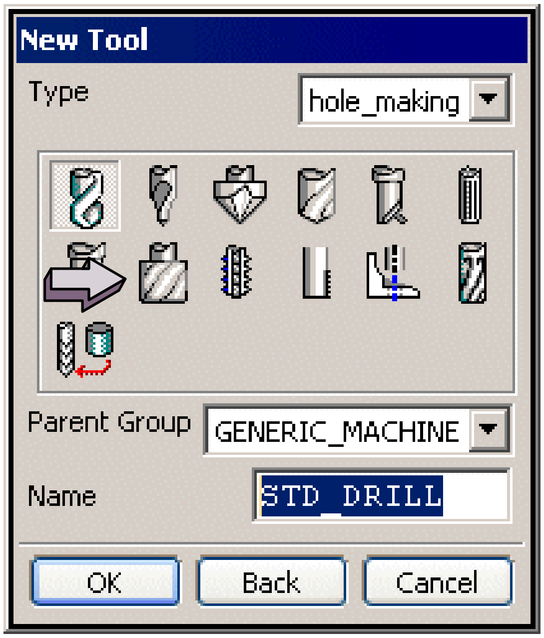
❑ Choose the **New** button.

The New Tool dialog is displayed.



❑ If necessary, change the **Type** to hole_making.

❑ Choose the COUNTER_BORE tool.



❑ Choose **OK.**

The Counterbore dialog is displayed. Before the tool can be created, you must determine the diameter of the counterbored hole.

❑ Choose **Information → Object.**

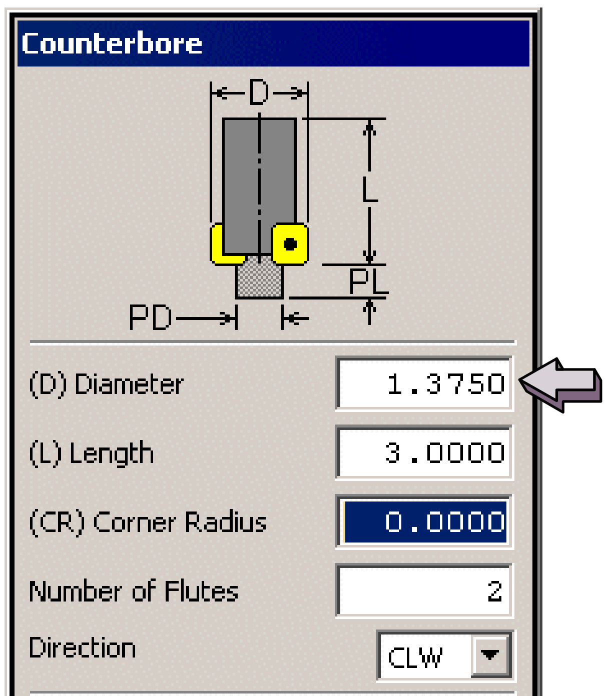
❑ Select the top of one of the four counterbored holes.

❑ Choose **OK.**

Upon examination of the Information window, you will see that the counterbore diameter is 1.375”.

❑ Close the Information window.

❑ In the Counterbore dialog, change the Diameter to **1.375**.



❑ Choose **OK** on the Counterbore dialog.

The required tool is now created.

❑ Choose **OK** on the Hole_Making dialog.

Step 6 Generate the operation.

❑ Change to the Program Order View of the Operation Navigator.

- ❑ Highlight the NC_PROGRAM parent group, use **MB3** and choose **Generate**.
- ❑ Choose **OK** until all the operations have been generated.

Step 7 Time permitting.......

- ❑ If you finish ahead of the class, and time permitting, attempt to use the Hole Making module to machine the countersunk holes. It may be necessary to create the appropriate sized countersink tool to complete the operation.

This completes the activity and the lesson.

SUMMARY

Geometry parent groups provide an efficient and robust method of creating and utilization geometry used in the manufacturing process of creating holes. In addition, the Hole Making module simplifies the process with high degrees of automation to the hole making process.

The following are used in the process of manufacturing holes:

- DRILL_GEOM Parent Groups
- Tools retrieved from the standard tool library
- Specific hole making operations
- Optimization functionality
- Hole Making Module

Advanced Cavity Milling Topics

Lesson 12

12

PURPOSE

This lesson teaches you how to use additional Cavity Milling options to create tool paths. You will also use Geometry Parent Groups to efficiently machine Cavity Milling geometry.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Utilize the advanced Cavity Milling options
- Create and modify Geometry parent groups for Cavity Milling
- Create and modify Cut Levels
- Utilize the In-Process Workpiece

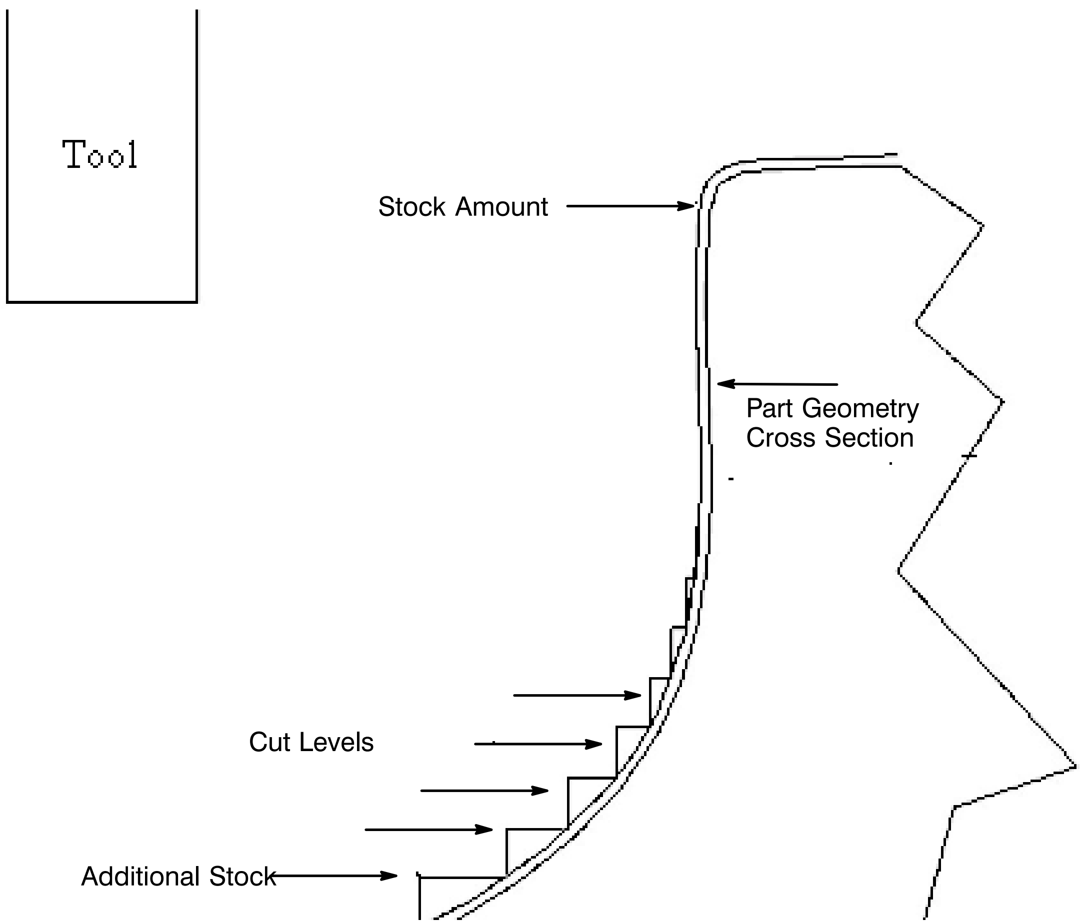
This lesson contains the following activities:

Cut Levels

Cavity Milling cuts geometry in planes or levels.

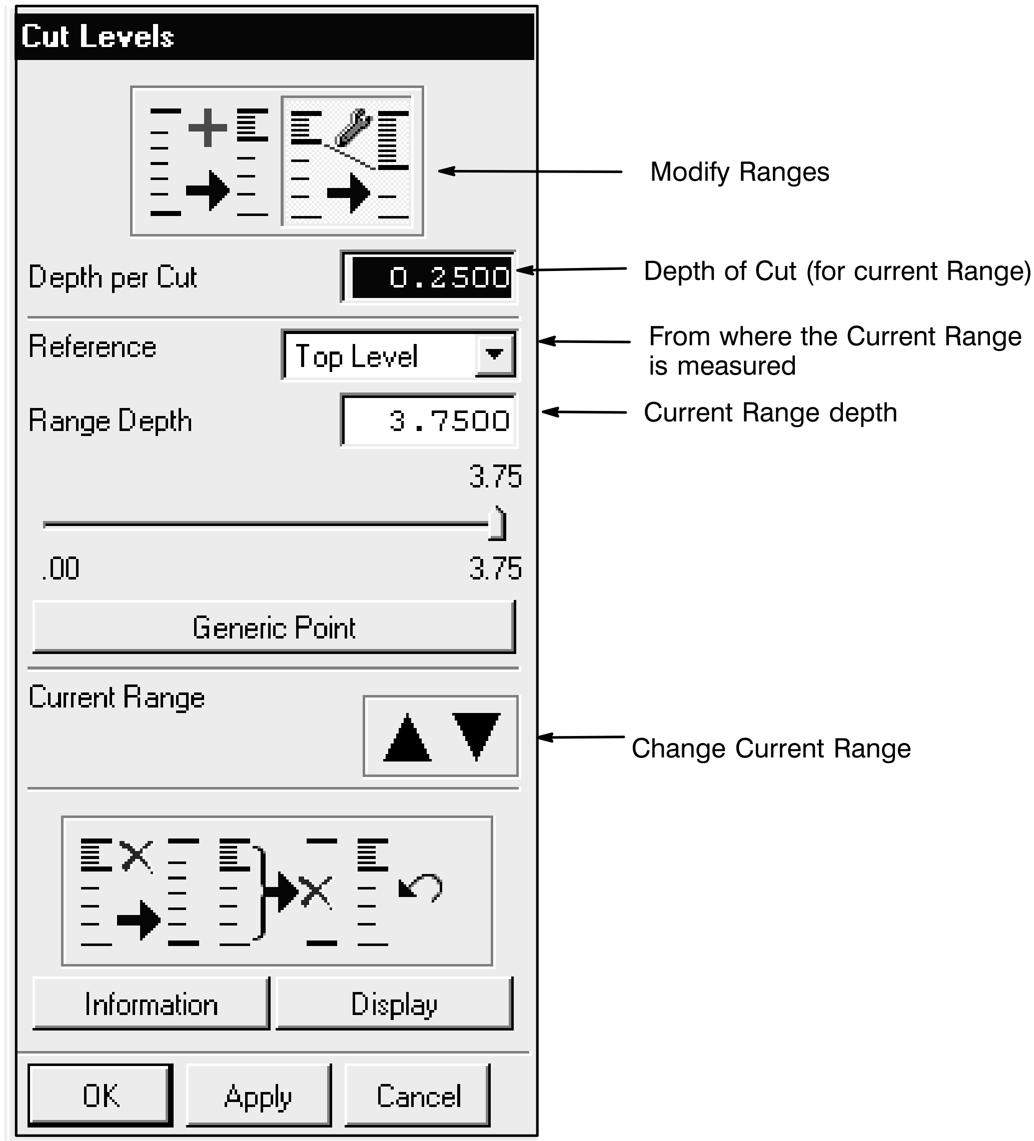
The advantage to this approach is that tool paths remain relatively short, due to minimum tool path movement, which is performed in layers.

The disadvantage is that when machining geometry that is close to horizontal the system may leave more stock than desired. See the diagram below.



The closer the geometry approaches being horizontal, the more stock that remains. Through the use of Cut Level parameters, you can reduce the amount of stock that remains.

The Cut Levels dialog is located under the Cut Levels button in the Cavity Mill dialog.



The Cut Levels dialog serves two primary functions:

- Create and modify Ranges
- Modify Cut Levels within Ranges

To reduce the amount of additional stock, a new range can be added. The Depth of Cut in that Range only is modified.

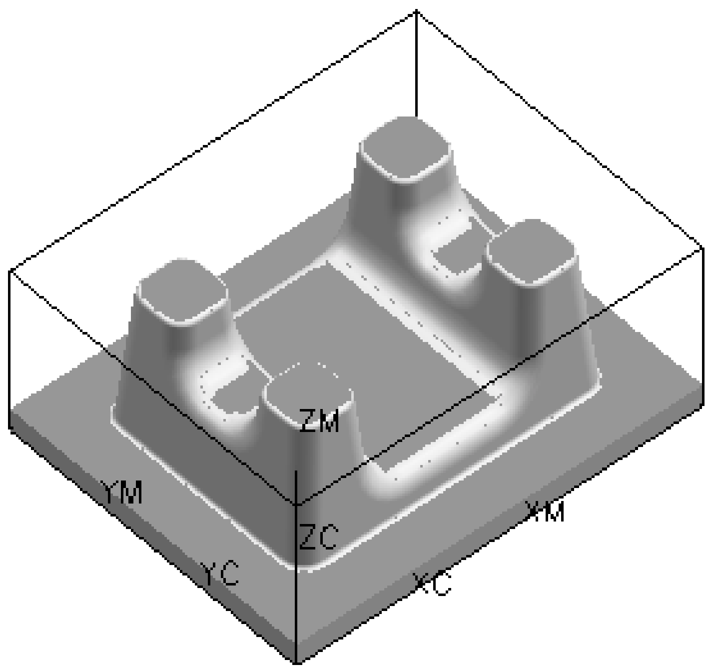
In the next activity, you will use the various Cut Level parameters.

Activity 12—1: Using Cut Levels Parameters

In this activity, you will replay an operation and review the various Cut Levels. You will then add two new Ranges and modify the Cut Levels within the new Ranges.

Step 1 Open, rename the part file, and enter the Manufacturing application..

❑ Open the part file **mmp_base_mfg_2.prt.**

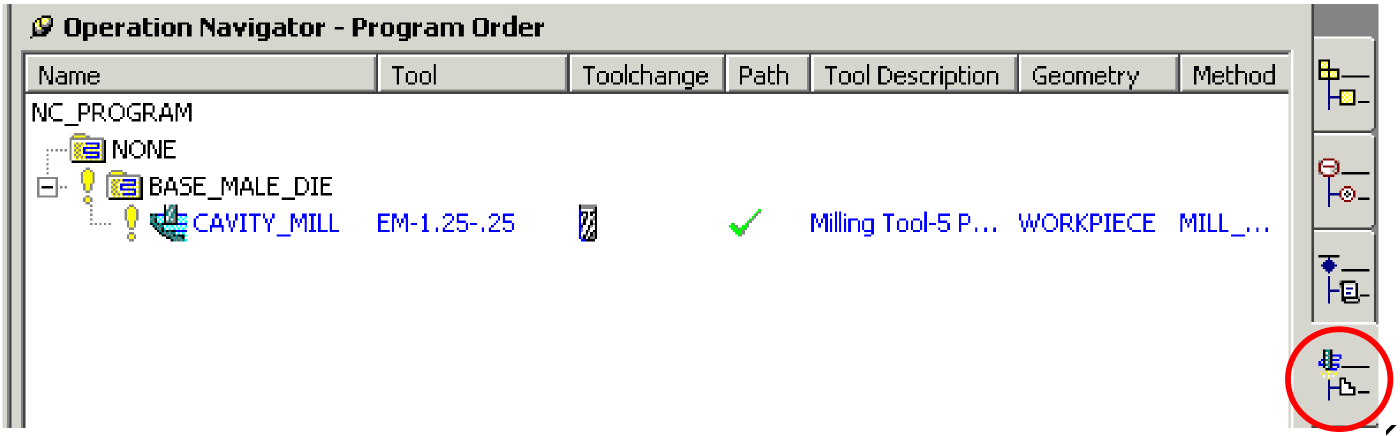


❑ Rename the part *****_base_mfg_2.prt** using the **File→ Save As** option on the menu bar.

❑ Choose **Application → Manufacturing.**

Step 2 Activate the Operation Navigator.

❑ Choose the **Operation Navigator** tab from the resource bar and expand the BASE_MALE_DIE parent group.

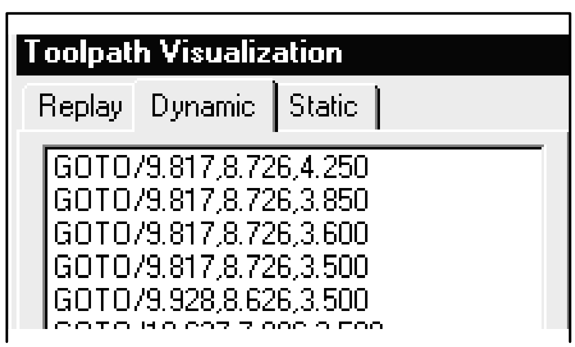


❑ In the Operation Navigator, make sure the **Program Order View** is active.

Step 3 Using Verify to examine the operation.

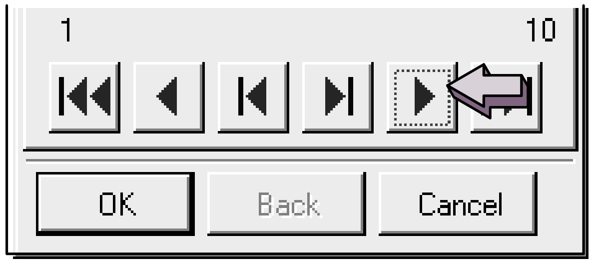
When working with an NC program that someone else has started, it is common practice to examine the existing work. You will use the Verify option to replay an existing operation.

❑ Highlight the **CAVITY_MILL** operation, using MB3, choose **Toolpath → Verify.**



❑ In the Toolpath Visualization dialog, choose the **Dynamic** property page (tab).

❑ As shown, choose the **Play Forward** icon.



TIP To speed up the Dynamic Replay mode, make the image smaller. In the graphics window, choose MB3→Zoom In/Out and zoom the object out. Unfortunately, this option is unavailable when the Toolpath Visualization dialog is active.

This operation has several items that need your attention:

(1) Too much stock is left on the top of the square bosses.

(2) Too much stock is left between the square bosses.

(3) The operation tries to machine too low on the part.

You will correct item #3 first.

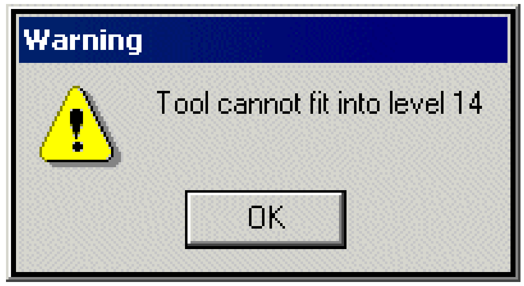
❑ Choose **Cancel** in the Toolpath Visualization dialog.

Step 4 Editing the Bottom of Range #1.

The first edit is to remove the warning from this operation by changing the cut range.

❑ Double click on the **CAVITY_MILL** operation.

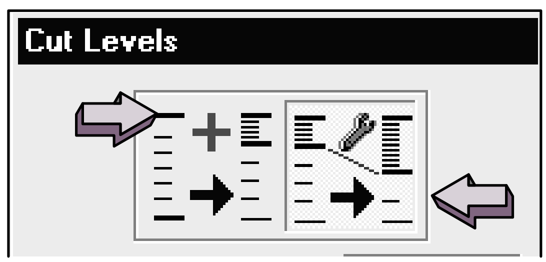
❑ **Generate** the operation.



This operation generates the warning " Tool cannot fit into level 14". At this level, the part and blank geometry are exactly the same, the trace generated by the module for the part and blank geometry are the same, therefore no geometry is available for machining. You will now alter the cut levels to eliminate the warning message.

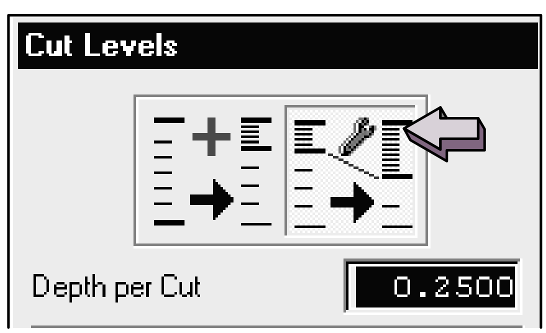
- ❑ Choose **OK.**
- ❑ Choose **Cut Levels** from the CAVITY_MILL dialog.

At the very top of the dialog, there are two buttons. One allows the editing of existing Ranges, while the other one allows the creation of new Ranges.



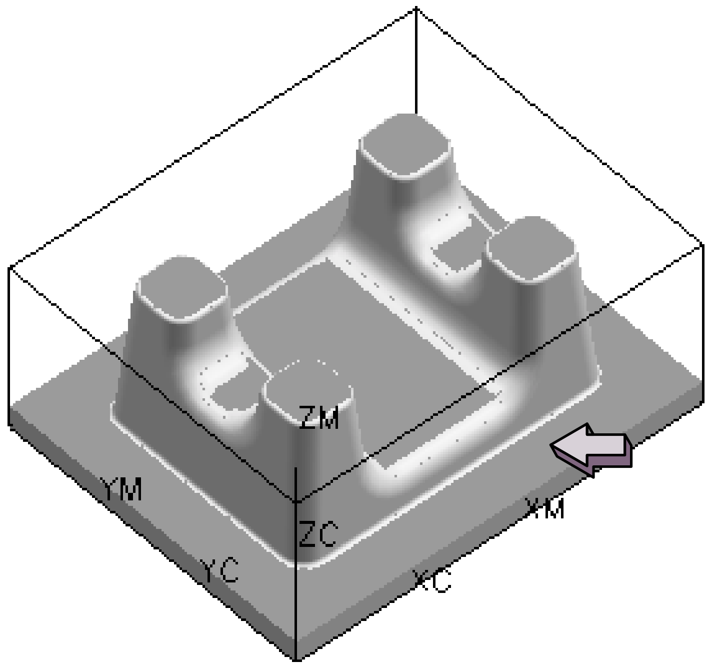
Examining the Status Line, you will find that there are currently 15 Cut Levels in this operation.

- ❑ At the top of the Cut Levels dialog, choose the **Modify Ranges** icon.



At this point, choosing a face will modify the bottom of Range #1.

❑ Select the face of the part as shown.



The Status Line now shows 13 Cut Levels.

❑ Choose **OK** in the Cut Levels dialog.

❑ **Generate** the operation.

The operation now generates without any warning messages.

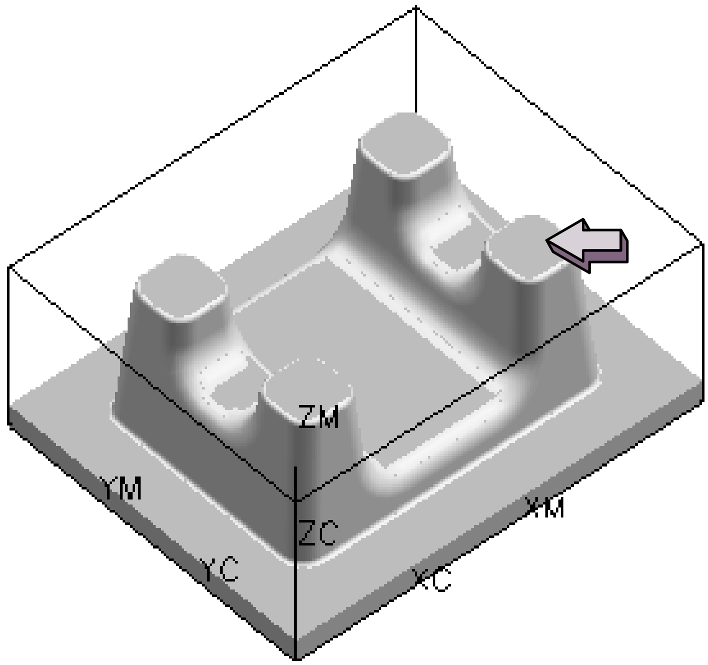
Step 5 Adding a Cut Level at the top of the Square Boss.

The only way to specify a specific Cut Level is to establish a new Range at that level.

❑ Choose the **Cut Levels** button.

❑ Choose the **Add Ranges** icon.

❑ Select the top of one of the square bosses, as shown below.



You have just added a new range.

❑ Choose either the Upward or Downward button and watch the Status Line and the graphics window.

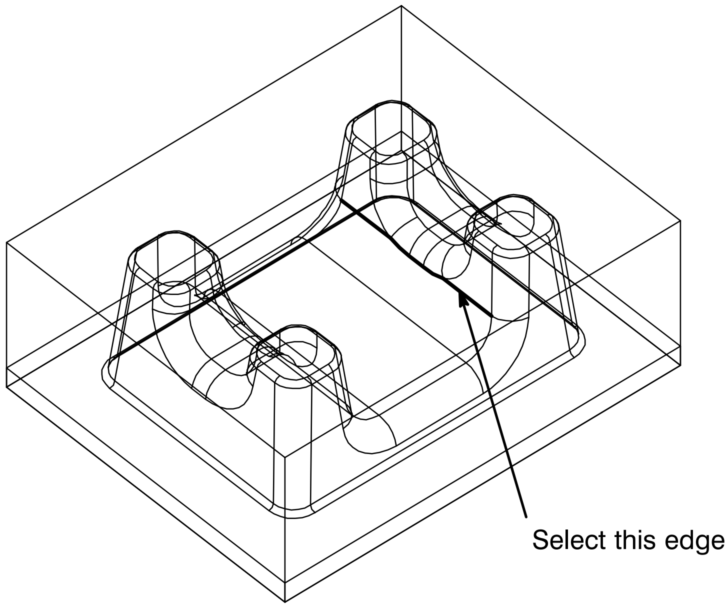
In the graphics window, you will see the current range highlights in the system color. In the Status Line, you will see the current Range #, and number of Cut Levels in that Range.

Step 6 Adding another Range.

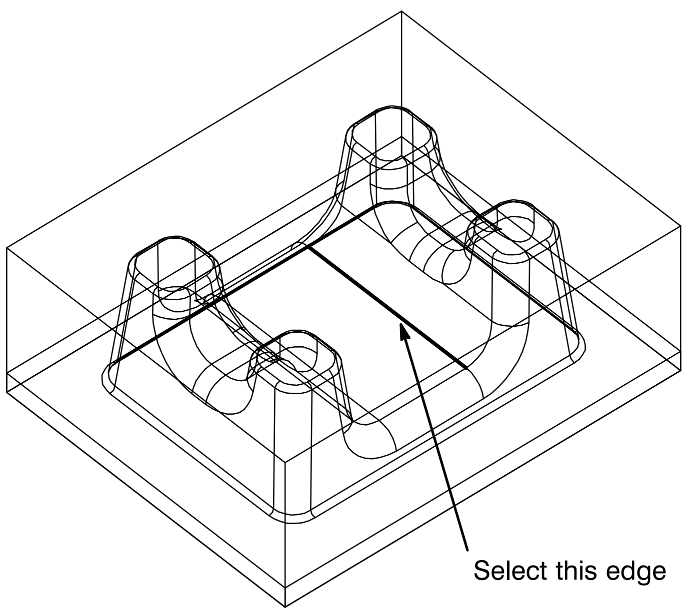
❑ Change the display to wireframe mode for easier viewing.

❑ In the Cut Levels dialog, choose **Generic Point**.

❑ Select the edge of the face as shown below.



❑ Now select the edge of the face as shown below.



❑ Choose **OK.**

❑ Cycle through the Ranges using the **Upward/Downward arrows**.

❑ Stop at Range #3.

Now you will change the Depth Per Cut for Range #3 to .125

- ❑ Change the **Depth Per Cut** to **0.125**.
- ❑ Choose **OK** in the Cut Levels dialog.
- ❑ Choose **Generate.**

The operation has generated. Now you will verify the results.

- ❑ Choose the **Verify** icon.
- ❑ Choose the **Dynamic** tab.
- ❑ Choose the **Forward** button.

The Verify operation shows the changes that were just made.

- ❑ Choose **OK** until the operation is saved and you are returned to the Operation Navigator.
- ❑ **Save** the part file.

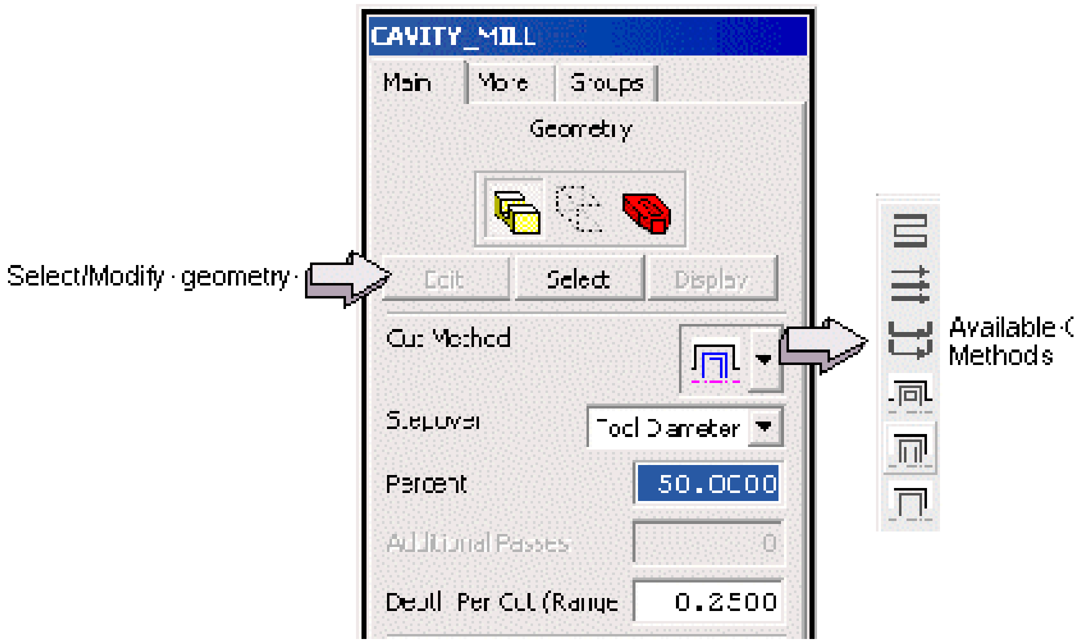
This completes this activity.

12

Cut Patterns

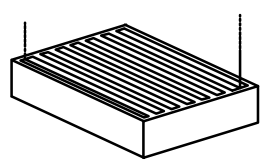
The Cut Method determines the cut pattern used for cutting.

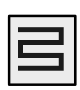
Cavity Milling dialog showing the Main Property page



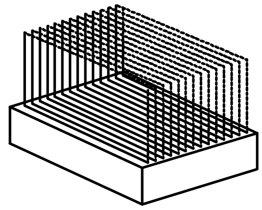
The methods are as follows:

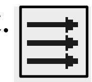
Zig-Zag



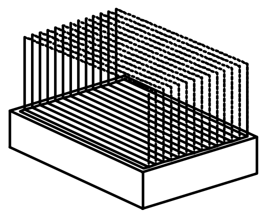
Zig-Zag machines in a series of parallel straight line passes. Climb or conventional cut directions are not maintained since the cut direction changes from one pass to the next. 

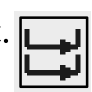
Zig



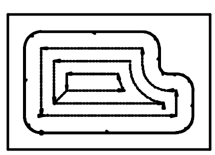
Zig always cuts in one direction. The tool retracts at the end of each cut, then positions to the start of the next cut. 

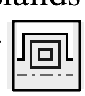
Zig With Contour



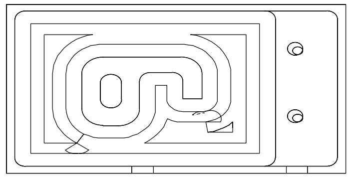
Zig with Contour also machines with cuts going in one direction. However, contouring of the boundary is added between passes, before and after the cut motion. The tool then retracts and re-engages at the start of the contouring move for the next cut. 

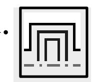
Follow Periphery



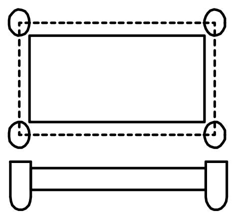
Follow Periphery offsets the tool from the outermost edge that is defined by Part or Blank geometry. Internal islands and cavities will require Island Cleanup or a clean up Profile pass. 

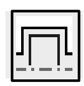
Follow Part



Follow Part creates concentric offsets from all specified Part geometry. The outermost edge and all interior islands and cavities are used to compute the tool path. Climb (or Conventional) cutting is maintained. 

Profile



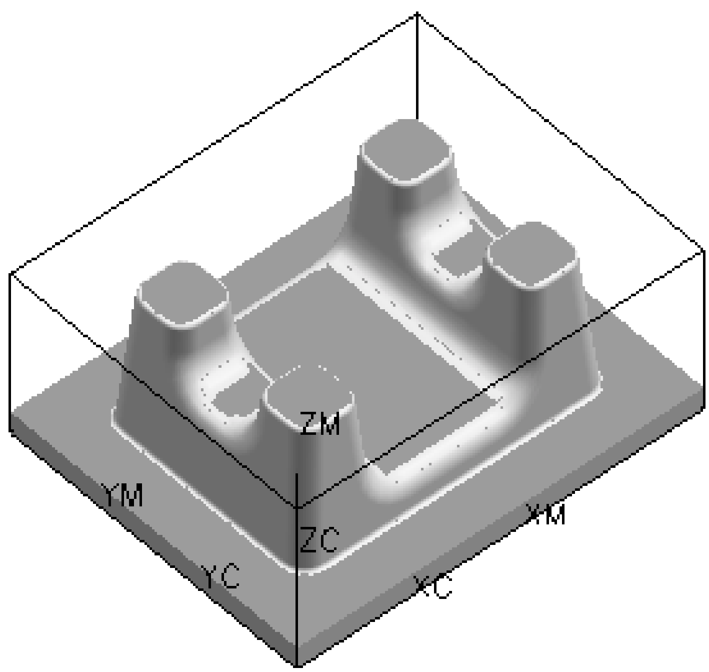
Profile follows a boundary using the side of the tool. For this method, the tool follows the direction of the boundary. 

Activity 12–2: Zig-Zag Cut Pattern

In this activity, you will gain experience with the Zig-Zag cut pattern.

Step 1 Open the part file, and enter the Manufacturing application.

- ❑ Continue using the part from the previous activity, *****_base_mfg_2.prt.**

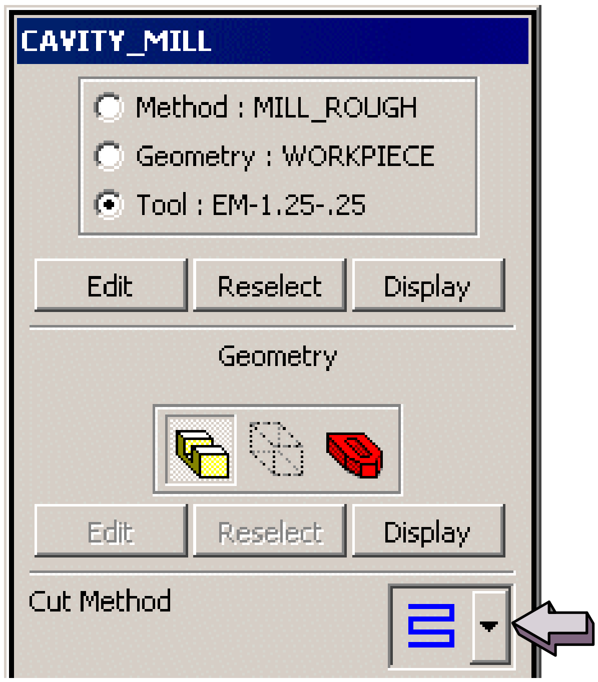


- ❑ If necessary, choose **Application→Manufacturing.**

Step 2 Edit an existing operation to change the Cut Pattern.

❑ Double click on the **CAVITY_MILL** operation.

❑ Change the Cut Method to **Zig-Zag**.



❑ From the CAVITY_MILL dialog, choose the **Zig-Zag** Cut Method.

Step 3 Generating the operation.

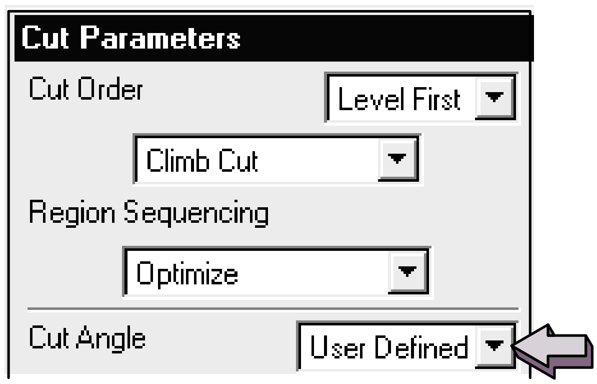
❑ Choose the **Generate** icon to generate the operation.

The tool path is generated.

Step 4 Changing the Cutting options.

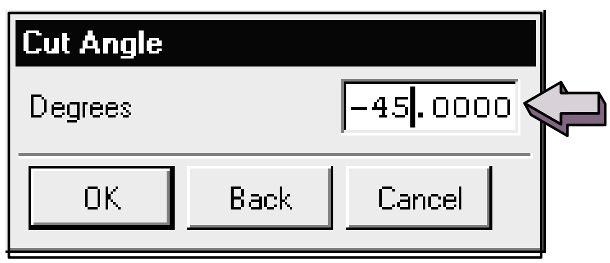
- ❑ Choose the **Cutting** button from the CAVITY_MILL dialog.

  The Cut Parameters dialog is displayed. Options available are based on the selected Cut Method.



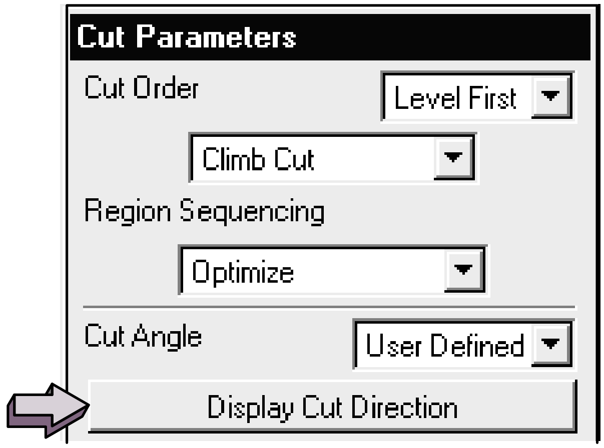
- ❑ Choose the **User Defined** option in the Cut Angle field, then choose **User Defined** again to bring up the Cut Angle dialog.

- ❑ Key in **-45.0** the Degrees field.



- ❑ Choose **OK** in the Cut Angle dialog.

❑ Choose the **Display Cut Direction** button.



In the graphics area, a vector indicator indicates the applied Cut Angle.

❑ Choose **OK** in the Cut Parameters dialog.

Step 5 Generating the operation.

❑ Choose the **Generate** icon to generate the operation.

❑ Use **Dynamic Verification** to analyze the results.

The Zig-Zag cut pattern does not have a stepover on every pass, causing a less than desirable tool path.

❑ **Cancel** the Toolpath Visualization dialog.

❑ Change the Cut Method to **Zig with Contour**.

Step 6 Generate the operation.

❑ Choose the **Generate** icon to generate the operation.

❑ Verify the toolpath, using the **Dynamic** option.

This time the tool path does a better job of cleaning up the corners.

- ❑ **Save** the part.

This completes the activity.

In-Process work piece for Cavity Milling

To make the various Cavity Milling operations as efficient as possible, you must determine what has been machined in each operation. Variables such as cutting tool lengths and diameters, draft angles and undercuts, fixture and tool clearances, will affect the amount of material that each operation may leave. The material that remains after each operation is executed is referred to as the In-Process work piece or IPW.

Generally speaking, the remaining material (IPW) can be used for input into a subsequent operation which may be used for additional roughing. The end result is a semi-finished part that has most of the rough material or stock completely removed.

To use the IPW, certain conditions must be adhered to. Tool path generation must be done sequentially, from the first operation to the last, within a certain geometry group. The tool path must be successfully generated and accepted in all previous operations in the sequence before the IPW can be used for the next operation of the sequence.

Activity 12—3: Using the In-Process Work Piece (IPW)

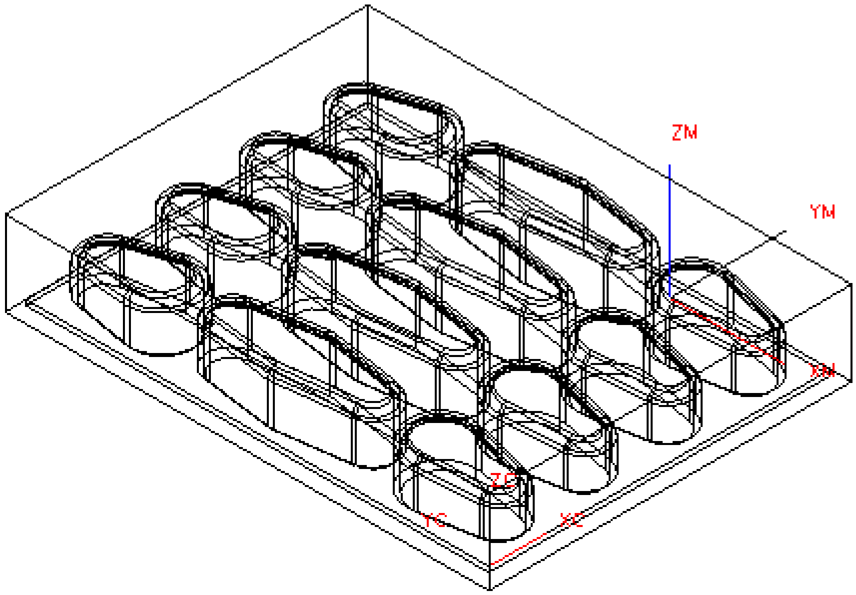
In this activity, you will machine the core block for an ATM key pad using three different cutter sizes. You will define the BLANK in the MILL_GEOM parent group, activate the use of the 3D IPW and generate the operation. You will then use the subsequent IPW as the blank for the next operation and will then use the IPW created from that operation to finish the keypad.

Step 1 Open the part mmp_ipw.prt and choose the Manufacturing application.

- ❑ From the menu bar, select **File**.
- ❑ Choose **Open**.
- ❑ Select the file **mmp_ipw**, then choose **OK**.
- ❑ Choose **Application→Manufacturing**.

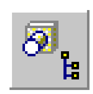
Step 2 Activate the Operation Navigator.

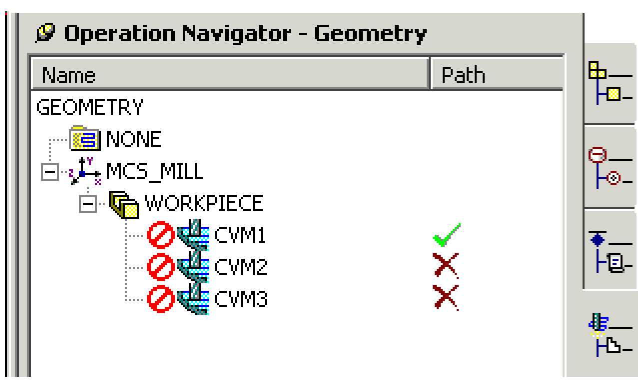
- ❑ Choose the **Operation Navigator** tab from the resource bar.



12

Step 3 Display the Geometry View in the Operation Navigator and expand the objects.

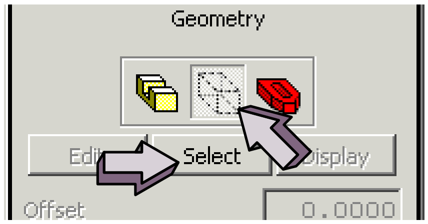
❑ Choose the **Geometry View** button  from the Operation Navigator tool bar, then expand the **MCS_MILL** and **WORKPIECE** parent groups.



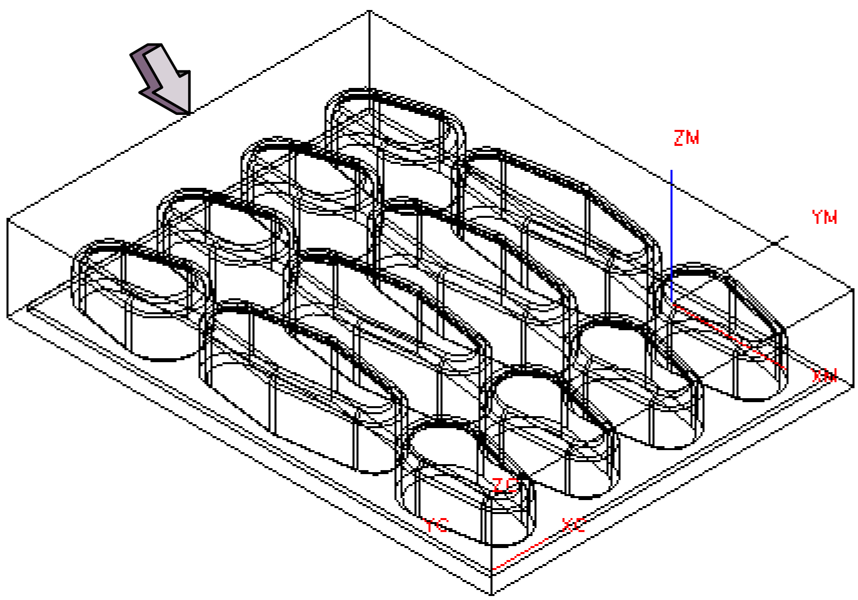
Step 4 Selecting the Blank geometry, in the WORKPIECE parent group, that will be used to generate the first IPW.

❑ Double-click the **WORKPIECE** parent group.

❑ Choose **Blank** and then **Select** from the MILL_GEOM dialog.



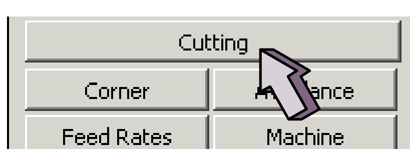
❑ Select the rectangular body as the Blank material.



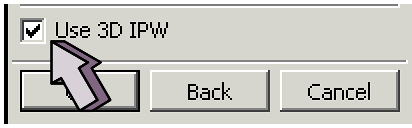
❑ Choose **OK** twice to accept the Blank material.

❑ Double-click the **CVM1** operation in the Operation Navigator.

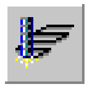
❑ Choose **Cutting** from the Cavity-Mill dialog.

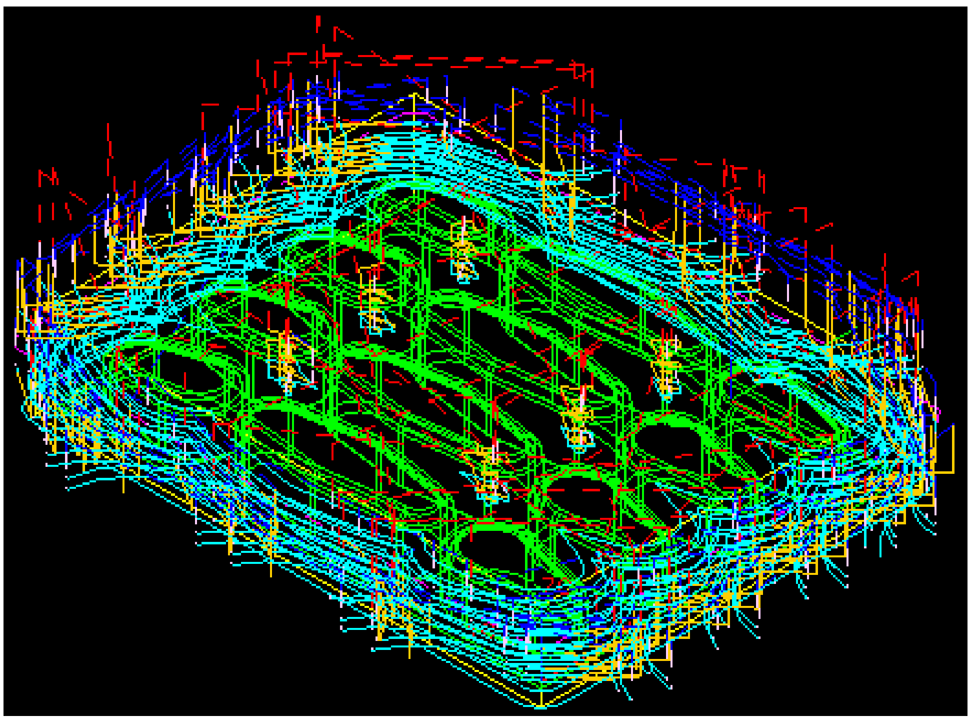


❑ Turn the **Use 3D IPW** button **ON**.



❑ Choose **OK** to accept the Cut Parameters.

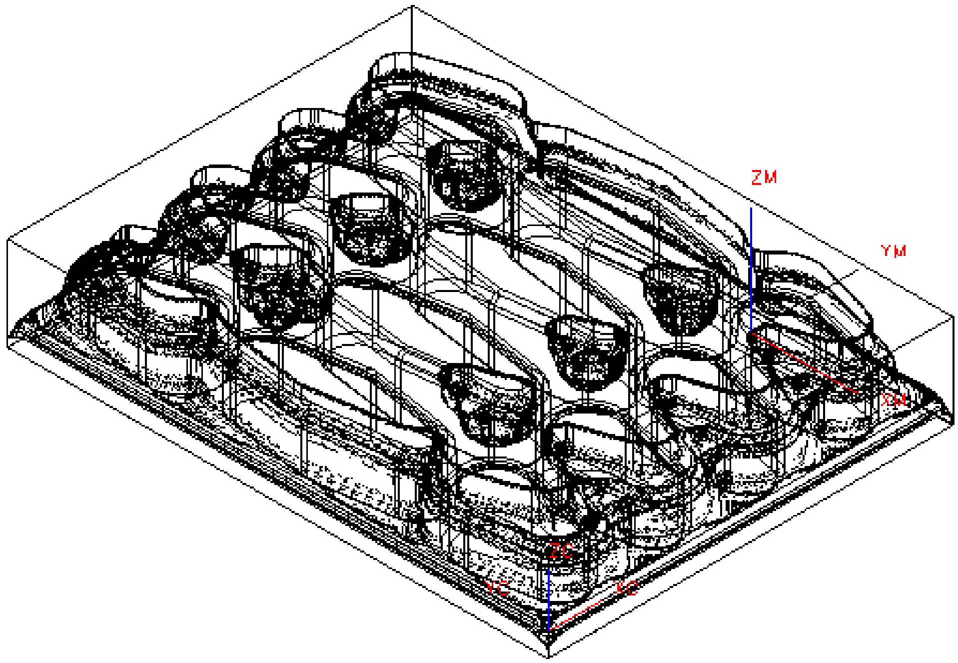
❑ Choose **Generate**  to generate the tool path.



The first Cavity Milling tool path is displayed. You will want to display the amount of stock that remains and that will become the blank for the next operation.

❑ Choose the **Display Output IPW** icon.

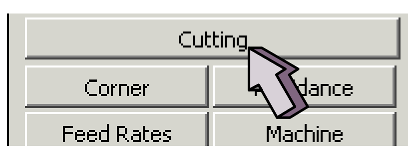
The resultant IPW is displayed.



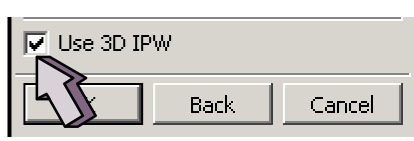
This IPW will be used as the Blank for the next operation, CVM2.

You saw that the initial IPW was defined as the Blank in the WORKPIECE geometry parent group. You generated the operation, using the initial IPW, and set options needed to create the IPW for a subsequent operation. You will use this IPW as the Blank for the operation, CVM2.

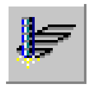
- ❑ Choose **OK** to accept the previous operation.
- ❑ Double-click the **CVM2** operation in the Operation Navigator.
- ❑ Choose **Cutting** from the Cavity_Mill dialog.

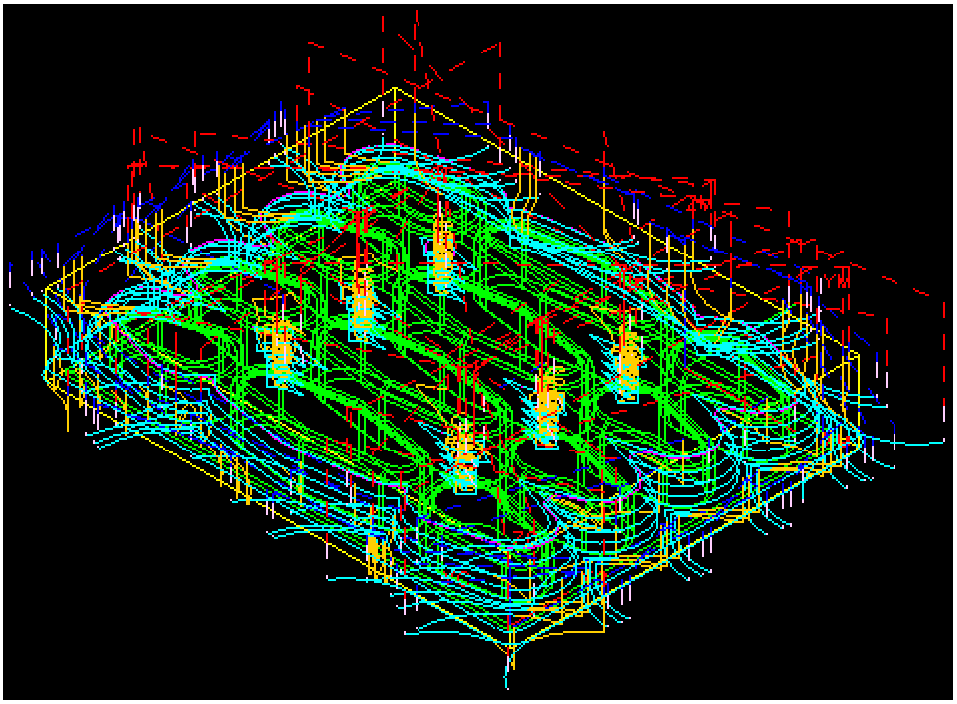


- ❑ Turn the **Use 3D IPW** button **ON**.



- ❑ Choose **OK** to accept the Cut Parameters.

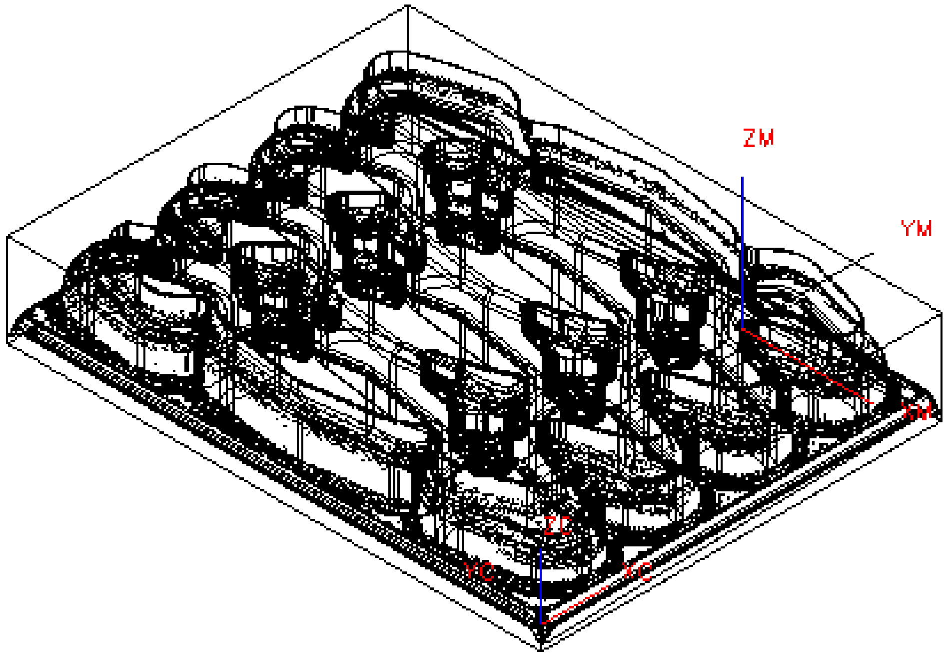
❑ Choose **Generate**  to generate the tool path.



The second Cavity Milling tool path is displayed. You will now display the IPW to show the remaining material.

❑ Choose the **Display Output IPW** icon.

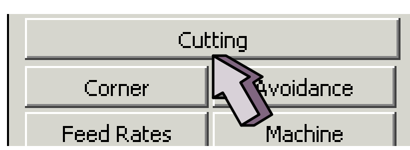
The resultant IPW is displayed.



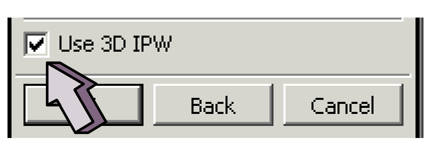
This IPW will be used as the Blank for the next operation.

You will use the current IPW for the final Cavity Milling operation.

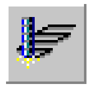
- ❑ **OK** to accept the previous operation.
- ❑ Double-click the **CVM3** operation in the Operation Navigator.
- ❑ Choose **Cutting** from the Cavity_Mill dialog.

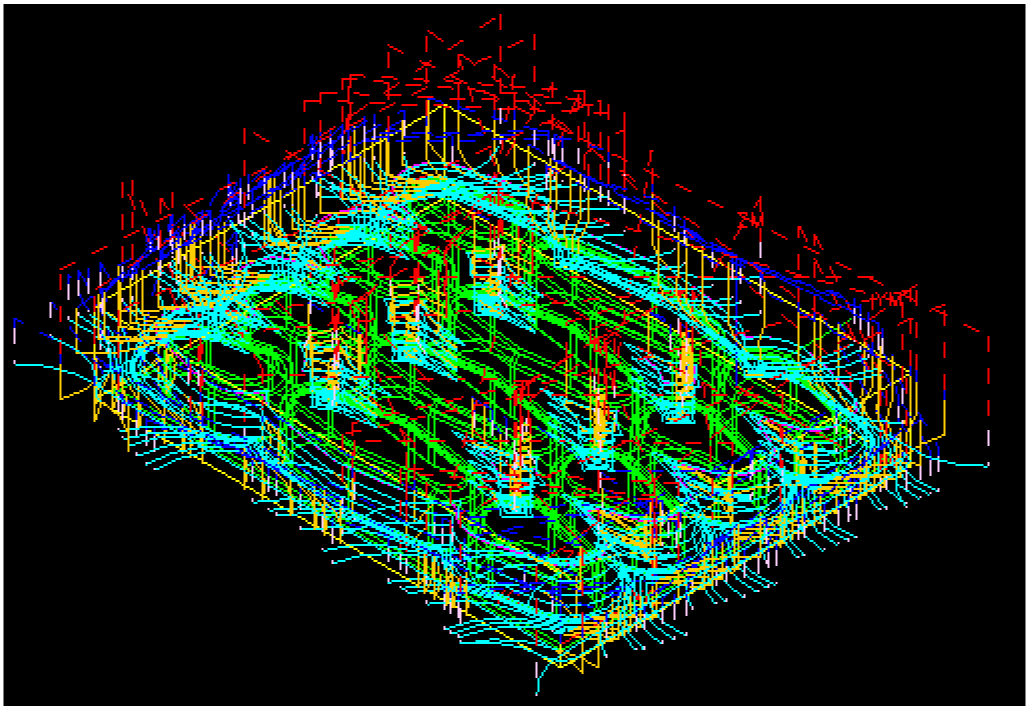


- ❑ Turn the **Use 3D IPW** button **ON**.



- ❑ Choose **OK** to accept the Cut Parameters.

❑ Choose **Generate**  to generate the tool path.



❑ Close the part file.

This concludes this activity and the lesson.

SUMMARY

The Cavity Milling module provides efficient and robust capabilities of removing large amounts of stock, primarily in cavity and core type applications.

The following functions are available in Cavity Milling:

- Use of the In-Process work piece for accurate removal of material using different size cutting tools
- Cut levels to precisely control depths of cut
- Cut patterns to control direction and method of removing stock

(This Page Intentionally Left Blank)

Planar Milling - Basics

Lesson 13

PURPOSE

This lesson introduces you to the basic interaction and usage of Planar Milling. You use Planar Milling for roughing and finishing operations using boundary geometry.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Determine when to use Planar Mill and Cavity Milling for creating operations
- Create Part and Blank boundaries in an operation
- Create Part and Blank boundaries in a MILL_BND Parent Group
- Create Planar Mill operations

This lesson contains the following activities:

Planar Milling Philosophy

Planar Milling is a machining method that allows you to define an area to be machined where the geometry is suited for 3-axis milling and the Z-axis remains fixed. For example, if a part has a pocket to be machined, where the walls are perpendicular to the floor, Planar Milling is the method of choice.

The question of whether to use Planar Mill or Cavity Milling to machine a part can be best answered by examining the part to be machined. For example:

- If the part has tapered walls that need to be finished, Planar Milling would ***not*** the choice
- If you are rough machining a part and want to break the machining up into small planar regions, in which the Z-axis is fixed, Planar Milling can be very useful

Planar Milling Guidelines:

- Planar Milling can be used to rough and/or finish
- Planar Milling requires geometry that is either parallel or perpendicular to the tool axis
- Planar Milling does not look at solid geometry to determine cut regions, instead it uses boundaries, created from the edges of faces
- Planar Milling can perform single and multi-level cutting

Planar Milling vs. Cavity Milling

Planar Mill is intended to cut planar geometry.

Use Planar Mill to create tool paths for:

- Planar geometry
- Roughing and finishing
- Profile pass

Cavity Milling is intended to rough cut contoured geometry, removing and keeping track of material removal by volume.

The tool can follow contoured geometry at each Cut Level. The tool remains fixed in the Z-axis while cutting and follows irregular shapes in the X and Y-axis.

You use Cavity Milling to create tool paths for:

- Roughing contoured cavity and core type parts
- Roughing large volumes of material

Boundaries

Planar Milling is controlled by boundaries. The tool will cut inside or outside of the boundary depending upon whether you are cutting pockets or islands.

Boundaries are created by selecting a face or a series of edges, curves, or points to define a profile. Boundaries can be created:

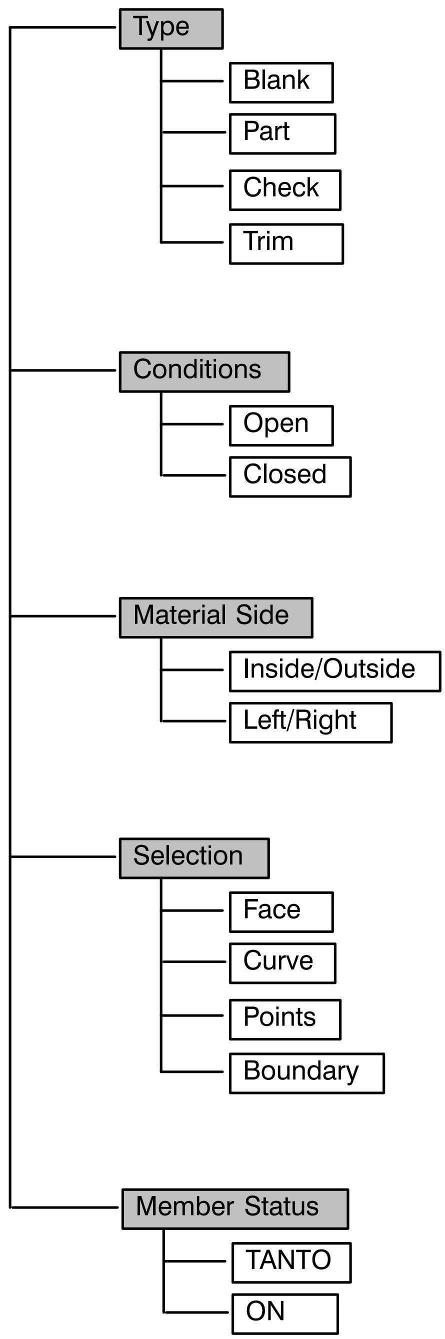
- in a MILL_BND geometry Parent Group
- within an operation

The MILL_BND geometry Parent Group is used when a boundary is to be used by several Planar Milling operations. This affords defining the geometry once, but using it for several operations.

Creating boundaries within an operation is preferred when custom data is to be applied to the boundary. This concept is discussed later.

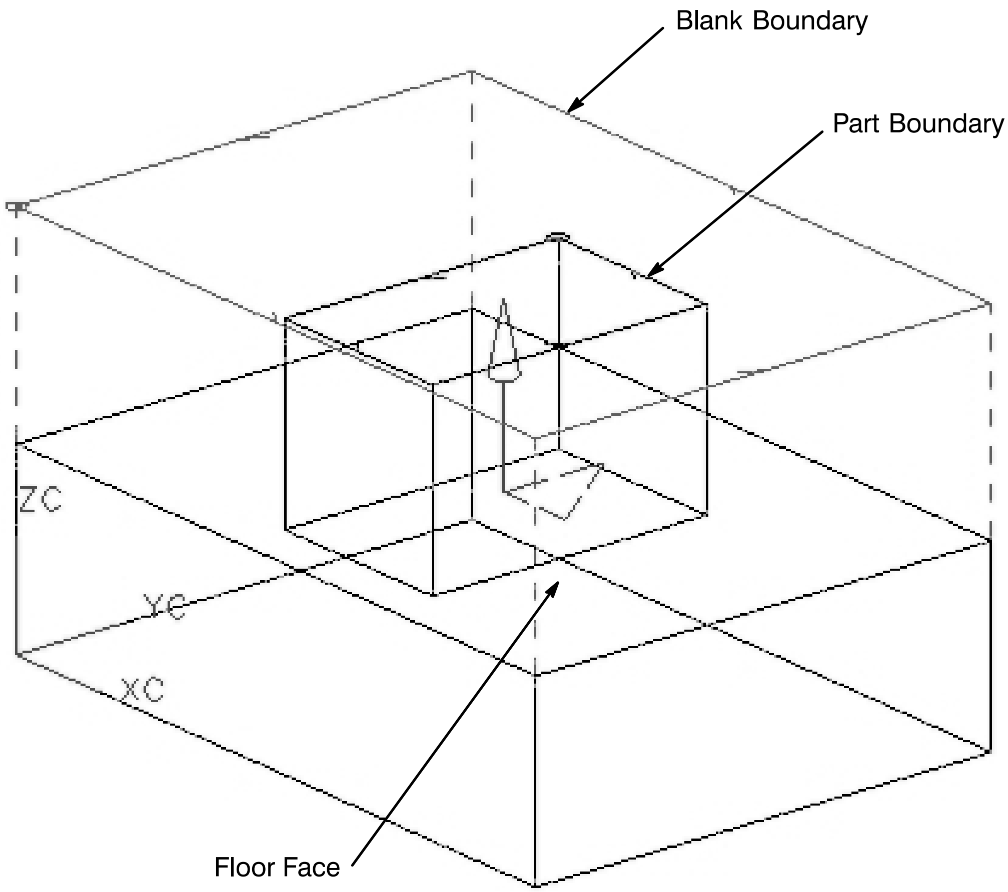
Boundary Flow Chart

The following is a flow chart outlining the characteristics of boundaries.



Blank Boundaries

Blank boundaries are used to define the material within which the part is located. This boundary type is very useful for parts that have protruding features that are above the overall topology of the part. For example, if a boss is located on the top of the part and you need to machine all the material around it, you will need to define the area for machining by selecting a Blank boundary.



As shown in the figure above, the blank boundary defines the excess stock to remove. The tool will enter from outside the blank boundary and remove material until it encounters the part boundary.

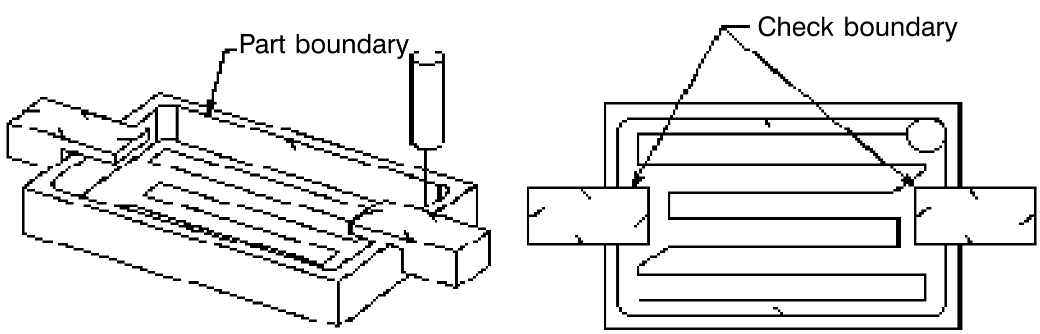
Part Boundaries

Part boundaries are used to define the part.

Check Boundaries

Check Boundaries are used to identify areas you do not want the tool to violate.

An example of check geometry would be clamps, fixture components or specific areas of a part that does not get cut.



Trim Boundaries

Trim boundaries are used to control specific areas for cutting/non cutting purposes.

Selection Modes

- Face – most associative, preferred
- Curve – very powerful
- Points – quick and easy
- Boundary – not recommended

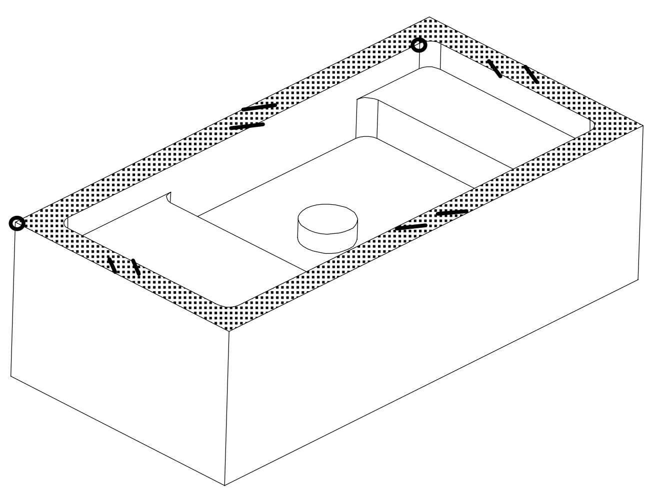
When creating boundaries you can select planar faces, edges, curves and points.

Select edges, curves and points when you cannot use a planar face.

Using Faces to Create Boundaries

Selecting the faces of solid models to create boundaries is the easiest way to identify areas to be machined in Planar Milling operations. It is sometimes easier to use this method (Face) and remove any unwanted boundaries than it is to select curves or edges.

In the example below you can see that selecting the crosshatched face gives you the inside and outside closed boundary elements.



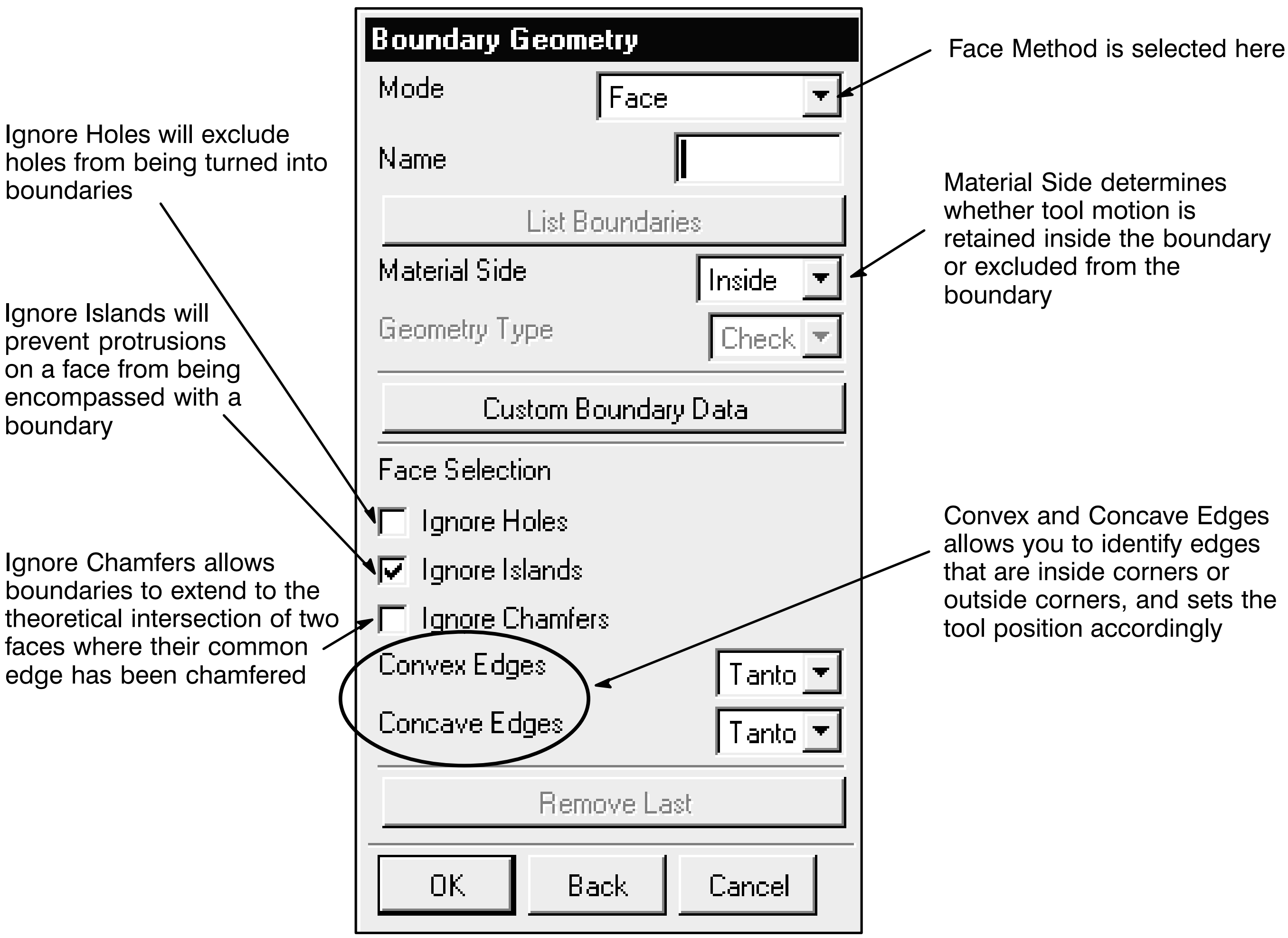
If you were only interested in machining the pocket, you would remove the outside boundary.

Removing a boundary is accomplished by editing the boundary in the MILL_BND Parent Group or in the operation.

You do have some control over which edges are used to create boundaries when using the Face mode. There are several toggles on the boundary dialog that can be set to influence the boundary results.

Creating Boundaries within an Operation

Boundaries created within a Planar Milling operation use the Boundary Geometry dialog.

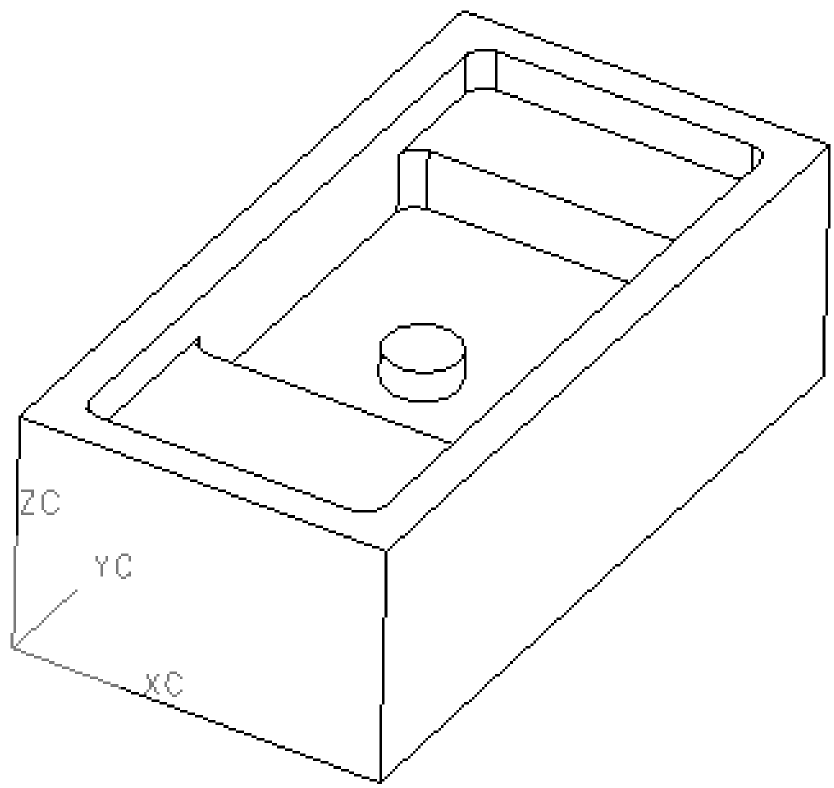


Activity 13–1: Use of Blank boundaries

In this activity, you will create a Blank boundary to control material removal and a Floor Plane to control cutter depth. You will use these boundaries to mill the top of the part.

Step 1 Open the part file, rename and enter the Manufacturing application.

❑ Open the part file **mmp_simple_box_mfg_1.prt.**

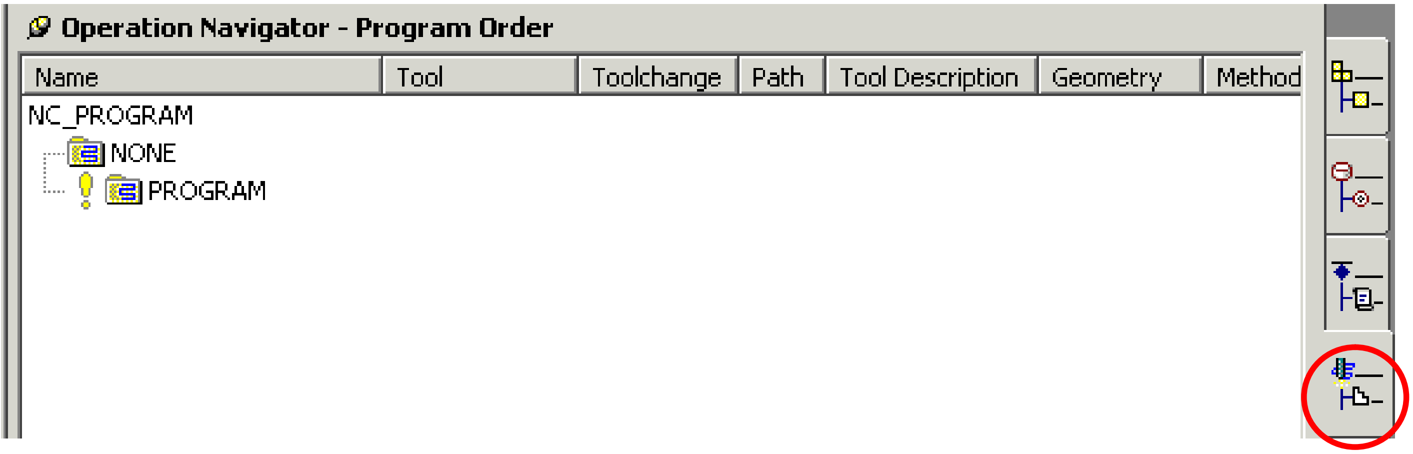


❑ Rename the part *****_simple_box_mfg_1.prt** using the **File→Save As** option.

❑ Choose **Application → Manufacturing**.

Step 2 Activate the Operation Navigator.

❑ Choose the **Operation Navigator** tab from the resource bar.



Step 3 Examine the Parent Groups.

❑ Explore the Parent Groups using the different views of the **Operation Navigator**.

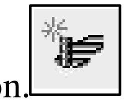
The WORKPIECE geometry Parent Group and two tools are defined.

13

Step 4 Creating a new Planar Milling operation to mill the top of the part.

For this newly created operation, you will:

- Choose the Type and Subtype
- Choose the Tool Parent Group
- Choose the Geometry Parent Group
- Choose the Method Parent Group
- Name the operation
- Select a boundary within the operation
- Generate the operation and observe the results
- Modify some settings and regenerate the operation

❑ Choose the **Create Operation** icon. 

❑ Make sure **Type** is set to **mill_planar.**

❑ Choose the **ROUGH_ZIGZAG** icon from the Create Operation dialog.

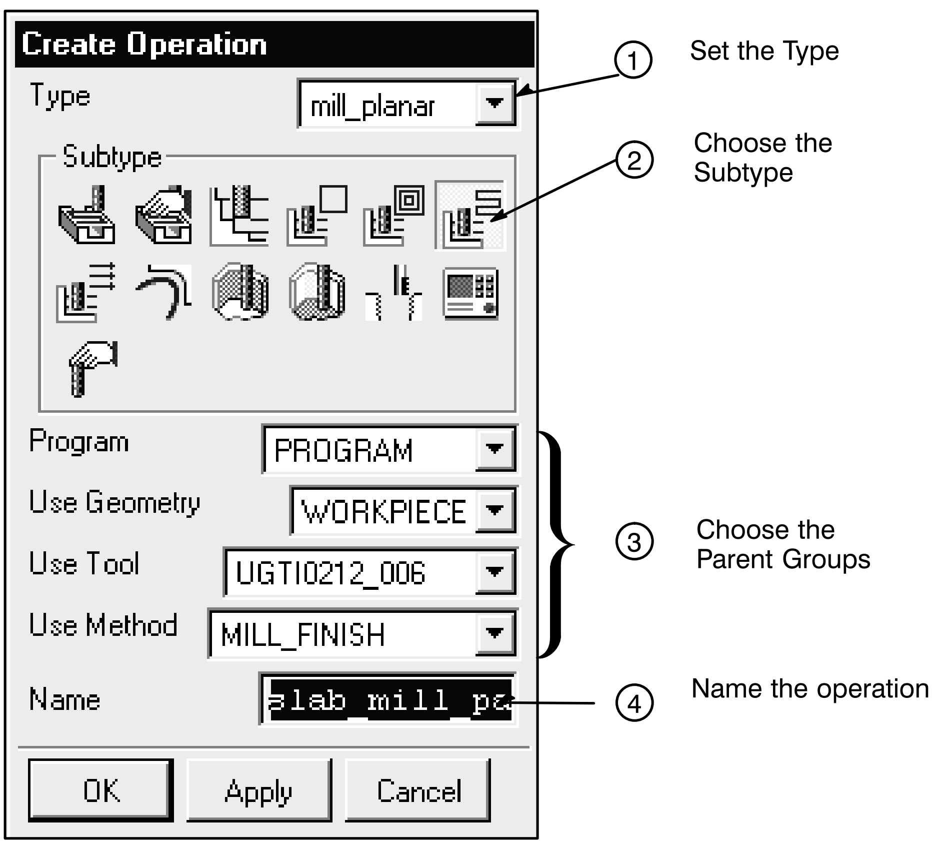
❑ Set the Program to **PROGRAM**.

❑ Set the Use Geometry to **WORKPIECE**.

❑ Set the Use Tool to **UGTI0212_006.**

❑ Set the Use Method to **MILL FINISH**.

❑ Name the operation **slab_mill_part**.



❑ Choose **OK**.

The ROUGH_ZIGZAG dialog is displayed.

You will now create the required boundaries within the operation.

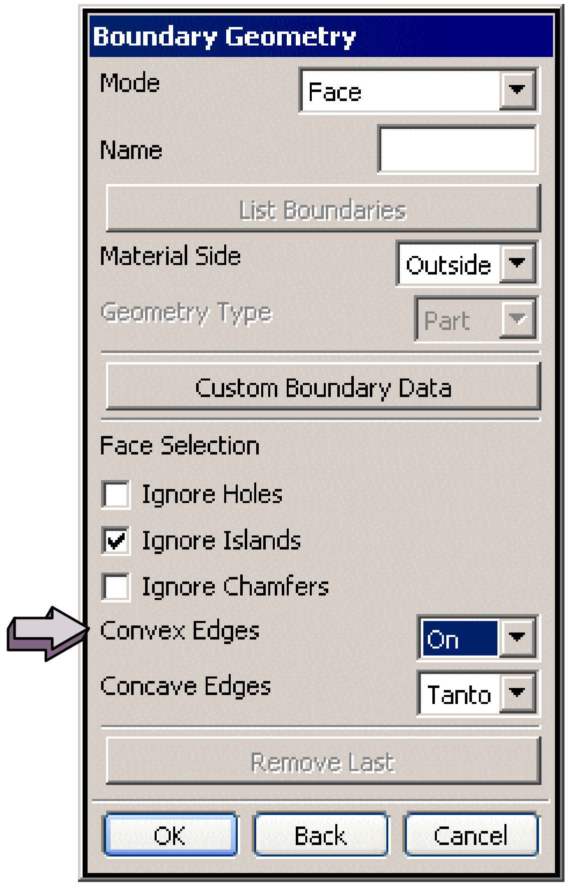
❑ Under the Geometry label, choose the **Part** icon.

❑ Choose the **Select** button.

❑ Change the Material Side to **Outside**.

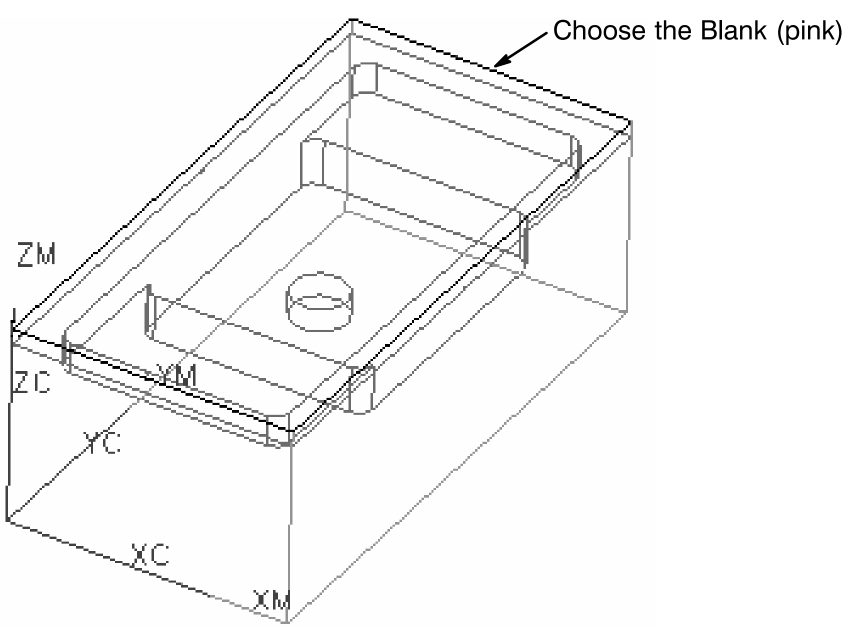
❑ Turn Convex Edges **ON.**

❑ Turn on **Ignore Holes.**

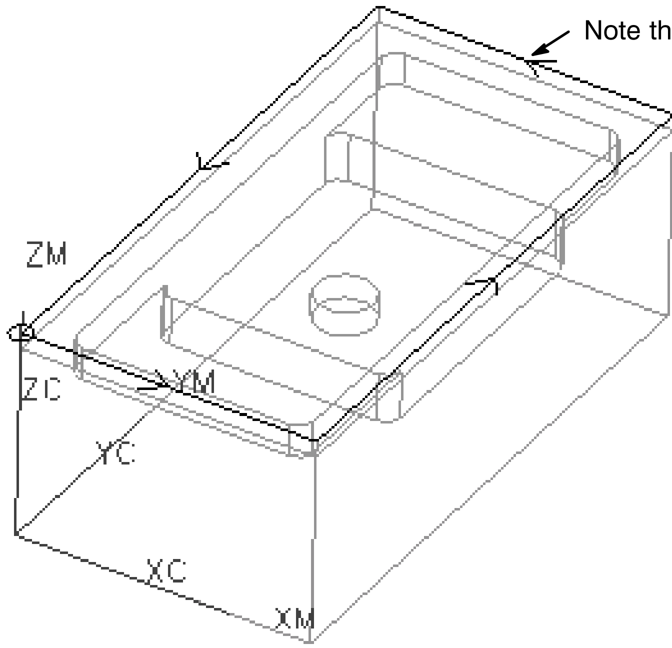


You will create a boundary on the top of the part.

❑ Select the top face of the blank (the blank is pink in color).



A boundary is created at the top of the part.



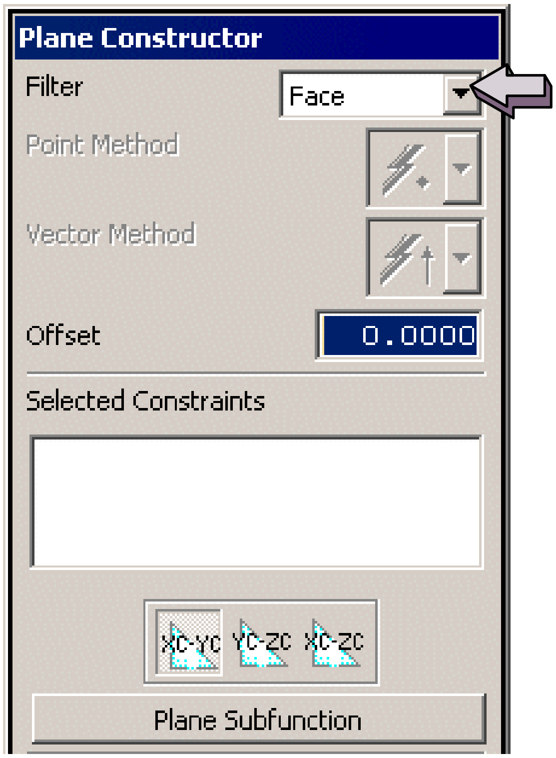
❑ Choose **OK**.

This concludes the boundary selection. You will now define the Floor Plane.

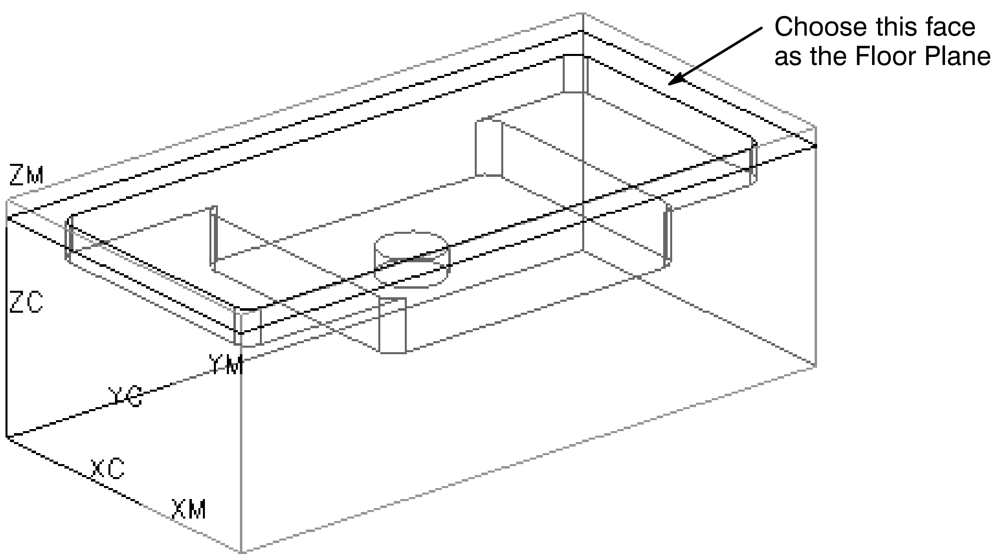
Step 5 Selecting the Floor Plane.

❑ In the Geometry area, of the ROUGH_ZIGZAG dialog, choose the **Floor Plane** icon, then choose **Select.**

The Plane Constructor dialog is displayed.



❑ Change the Filter to **Face,** then select the top of the part as the Floor Plane.

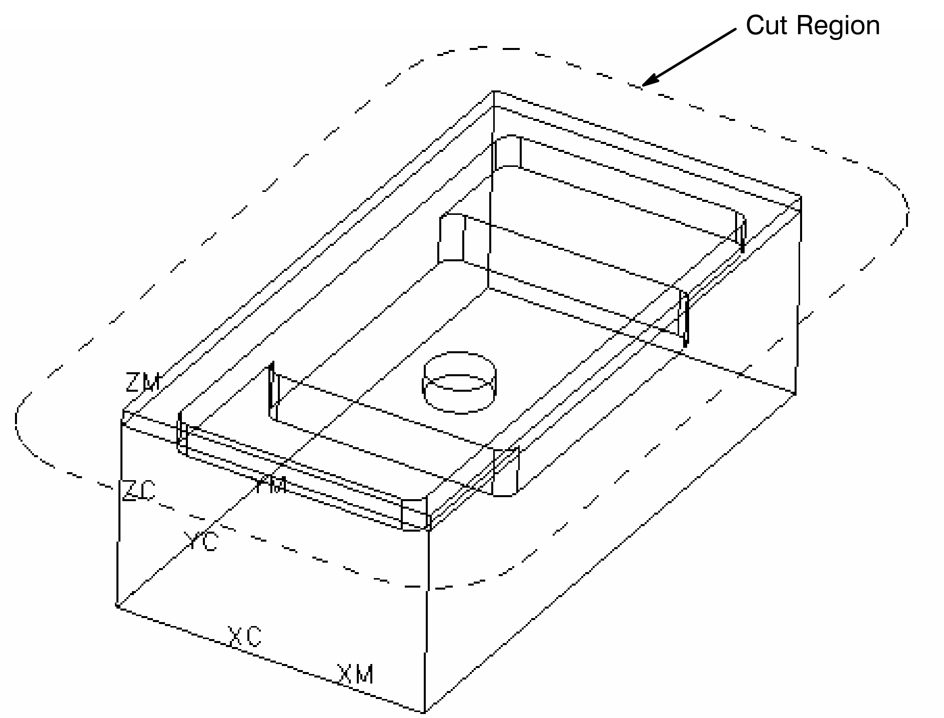


❑ Choose **OK**.

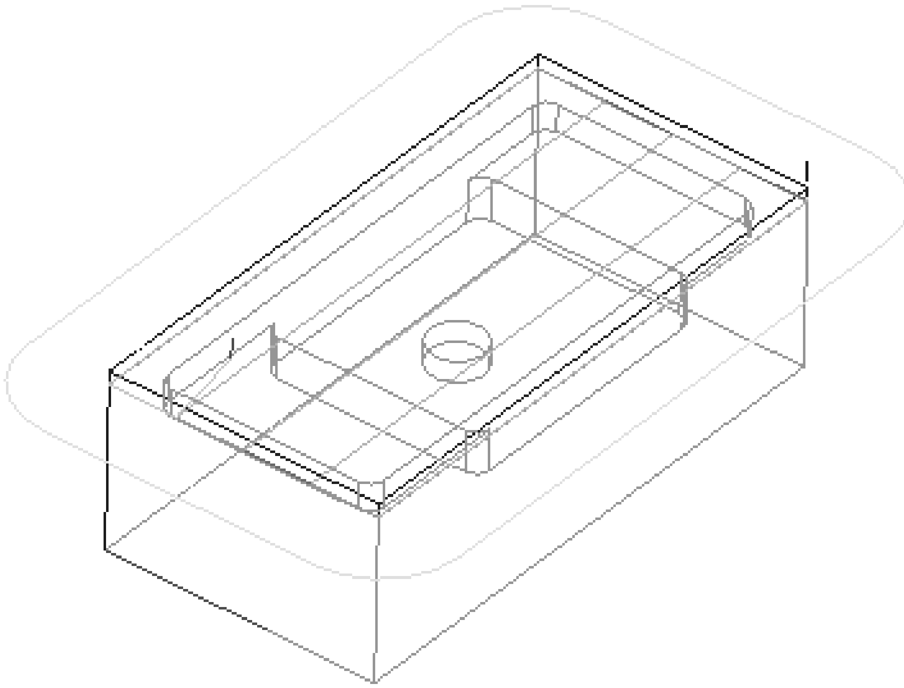
Step 6 Generating the tool path.

❑ Choose the **Generate** icon.

The Display Parameters dialog is displayed. Notice the Cut Region displayed.



❑ Choose **OK** to continue generating the tool path.



❑ Choose **OK** to accept the operation.

❑ **Save** and **Close** the part.

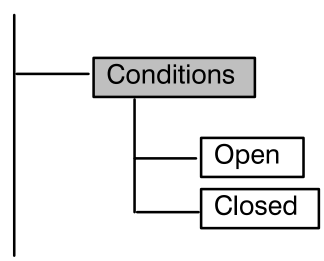
This completes this activity.

Rules for Using Boundaries

Some basic rules for using boundaries in Planar Milling operations are:

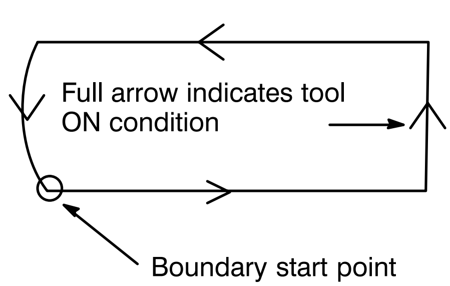
- Closed boundaries can keep the tool inside or outside of the boundary. The material side selected when creating the boundary determines the following:
  - Boundaries with material retained inside are called islands; the tool will not cut inside the island
  - Boundaries with material retained outside are called pockets; the tool is fenced in by a pocket boundary
- The location of the boundary in relationship to the Floor Plane determines whether the Planar Milling operation can perform multiple level cutting. If the boundary is defined at the Floor Plane you will get one cut level. If the boundary is located above the Floor Plane, the tool will approach the part, enter the material and machine at that level. Depending on depth of cut settings, the tool will make several cuts to reach the Floor Plane.
- It is a good practice to create the boundary at the level of the geometry
- Island boundaries must have material defined around them

Boundary Conditions

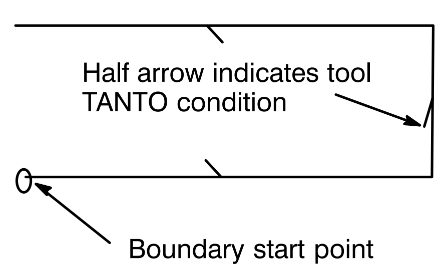


- Open – Single pass, can clean-up walls; uses Standard Drive and Profile cut methods
- Closed – Volume removal applications; uses Follow Pocket, Follow Periphery, Zig-Zag cut methods

Closed Boundaries



Open Boundaries

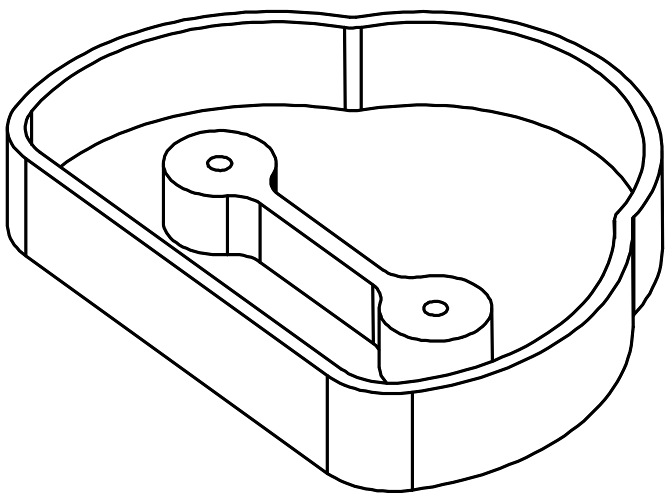


Activity 13–2: Selecting a boundary within an operation

In this activity, you will create a Planar Milling operation to create the pocket by selecting the boundary from within the operation.

Step 1 Open and rename the part and enter the Manufacturing application.

- ❑ Open the part **mmp_cover_housing_mfg_1.prt.**



- ❑ Choose **File** →**Save As** and save the part as *****_cover_housing_mfg_1.prt.**

- ❑ If necessary, enter the **Manufacturing** application.

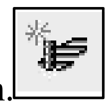
- ❑ Examine the Parent Groups using the different views of the **Operation Navigator.**

Notice that the Parent Group WORKPIECE and one tool is defined.

Step 2 Creating a new Planar Milling operation.

In this new operation, you will:

- Choose the operation Type and Subtype
- Choose the Tool Parent Group
- Choose the Geometry Parent Group
- Choose the Method Parent Group
- Name the operation
- Select faces to create boundaries within the operation
- Select a Floor Plane
- Generate the operation

❑ Choose the **Create Operation** icon. 

❑ Make sure the Type is set to **mill_planar.**

❑ Choose the **ROUGH_FOLLOW** icon from the Create Operation dialog.

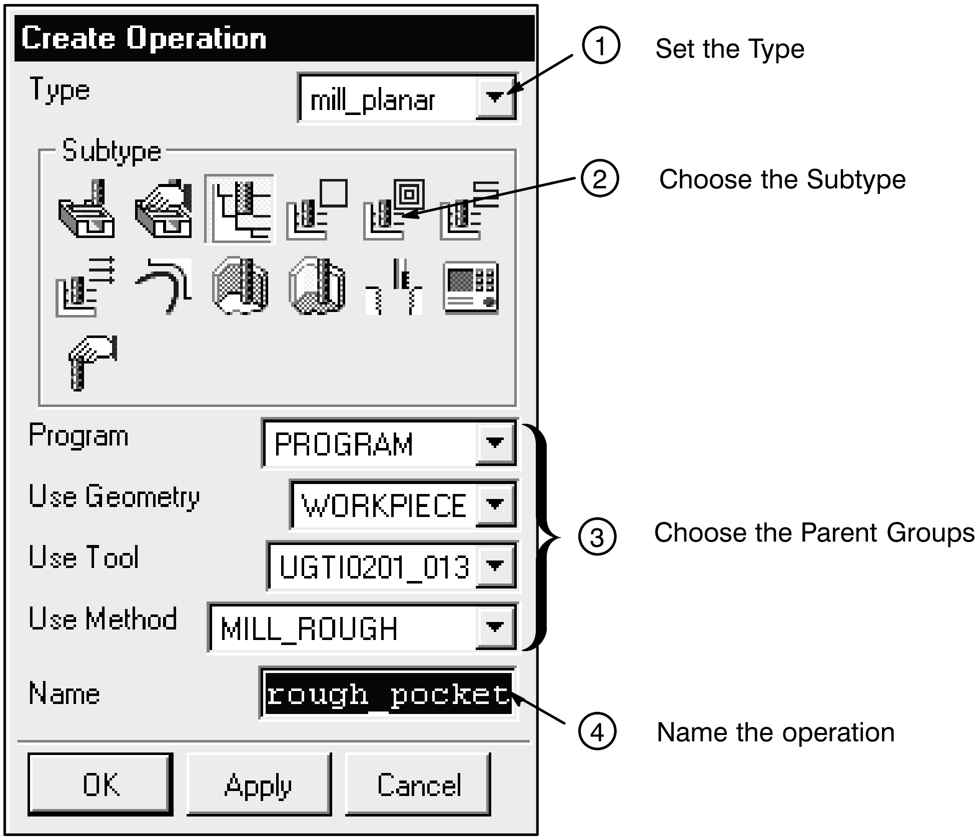
❑ Set the Program to **PROGRAM**.

❑ Set the Use Geometry to **WORKPIECE**.

❑ Set the Use Tool to **UGTI0201_013.**

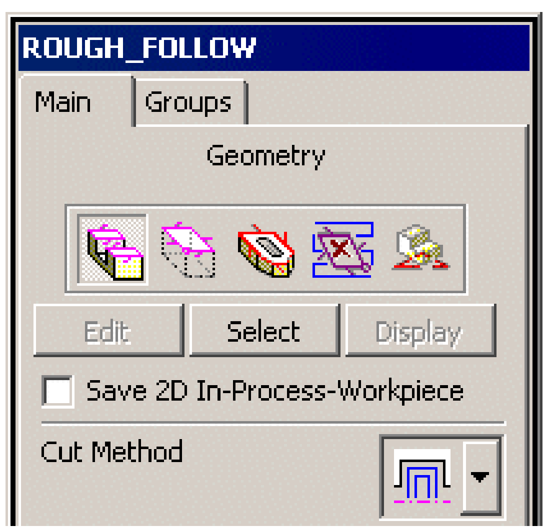
❑ Set the Use Method to **MILL_ROUGH**.

❑ Name the operation **rough_pocket**.



❑ Choose **OK**.

The ROUGH_FOLLOW dialog is displayed.



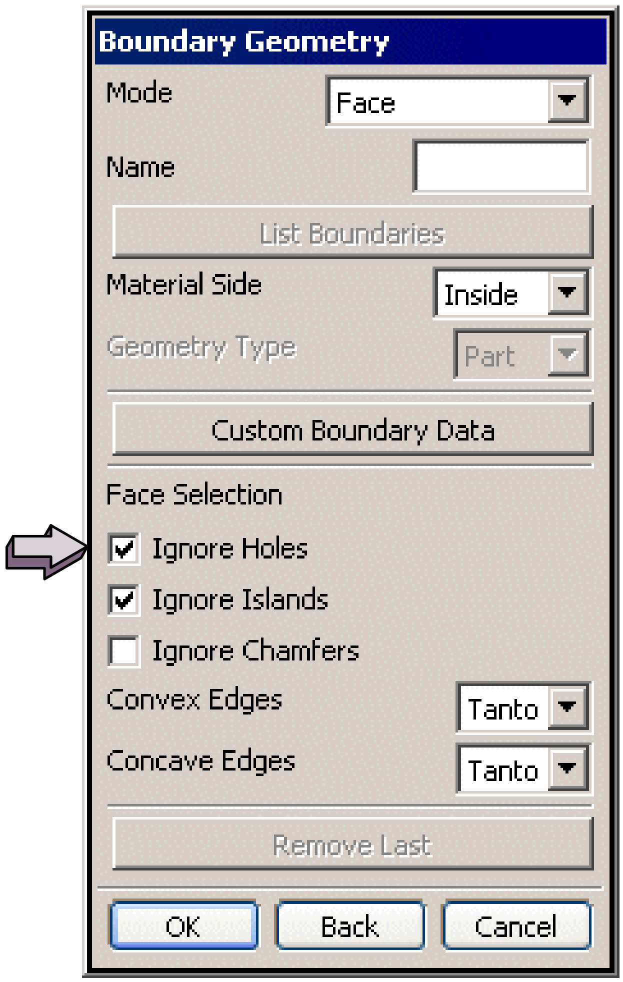
Step 3 Defining the part boundaries.

❑ Choose the **Part** icon and then **Select**.

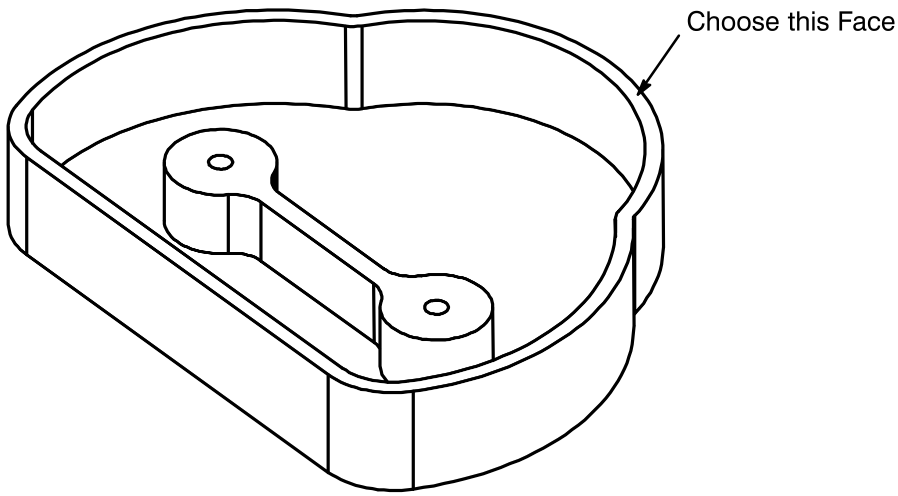
The Boundary Geometry dialog is displayed.

❑ Turn ON, **IGNORE HOLES**.

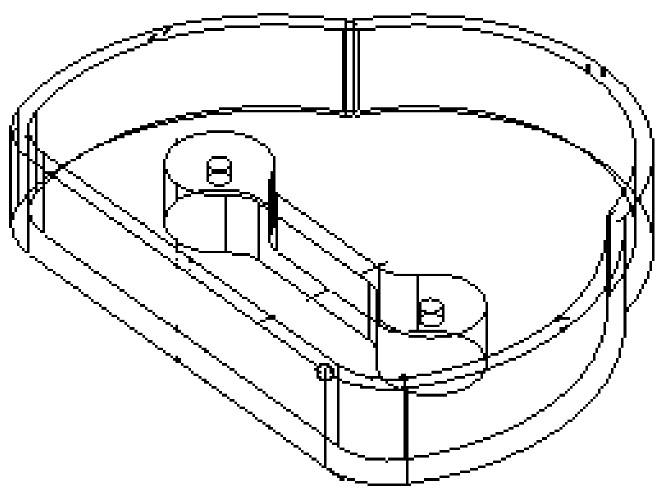
This option will keep the cutter from plunging into the hole.



❑ Select the top face of the pocket wall.

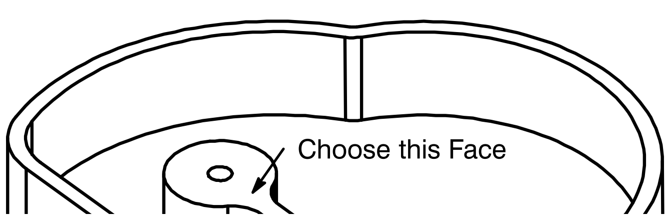


Two boundaries are created, one on the inside and one on the outside of the pocket wall.



You will now create the boundaries that will be used to machine the dog bone area of the part.

❑ Choose the top face of the dog bone area.



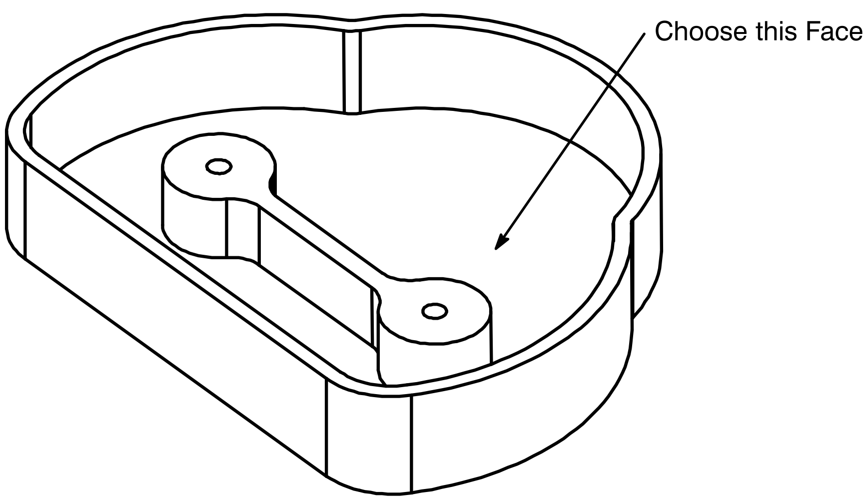
❑ Choose **OK** to return to the ROUGH_FOLLOW dialog.

Step 4 Selecting the Floor Plane.

❑ In the Geometry area, choose the **Floor** icon.

❑ Choose the **Select** button.

❑ Select the bottom of the pocket as the Floor Plane.

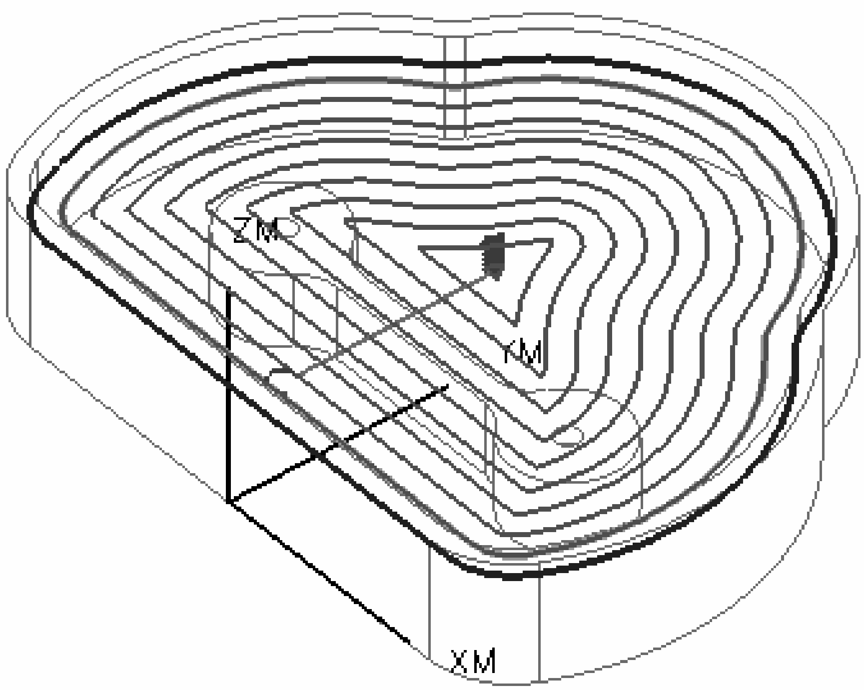


❑ Choose **OK**.

Step 5 Generate the operation and examine the tool path results.

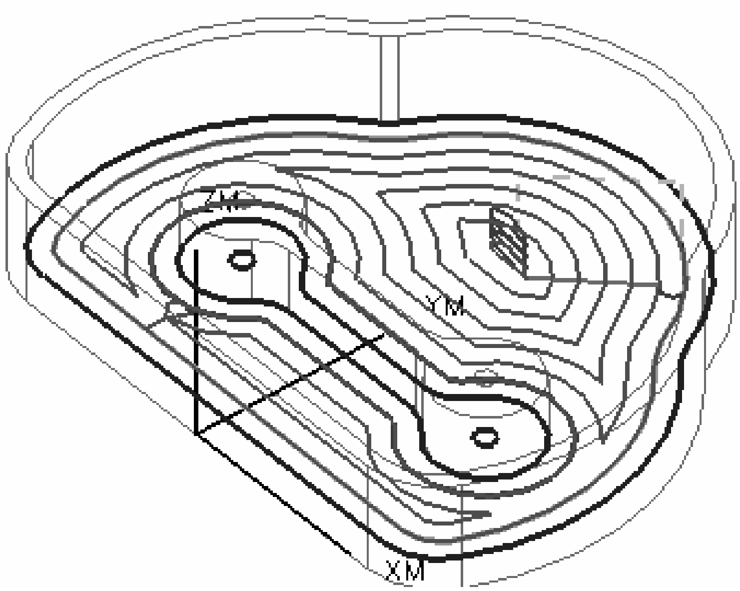
❑ Choose the **Generate** icon.

❑ Choose **OK** to continue generating the tool path.



The first level is displayed.

❑ As necessary, choose **OK** to continue generating the tool path.



The tool cuts in levels from the top to the bottom of the part, allowing for the dog bone protrusion.

❑ Choose **OK.**

❑ **Save** the part.

This completes this activity. The next activity will finish machine the floor of the part.

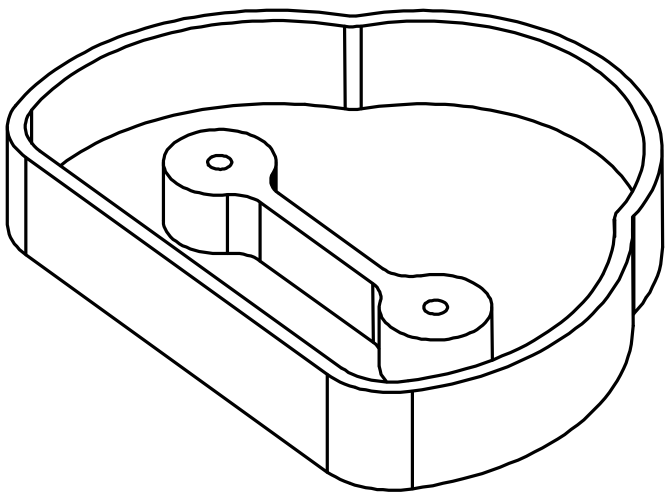
13

Activity 13–3: Finishing the floor of the part

In this activity, you will create an operation to finish the floor of the pocket that you just rough machined.

Step 1 Open the part file.

❑ Continue using the part ***__cover_housing_mfg_1.prt.__**



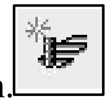
❑ If necessary, enter the **Manufacturing** application.

Step 2 Create a new Planar Milling operation to finish the floor.

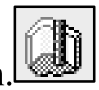
For this operation, you will:

- Choose the operation Subtype
- Choose the Tool Parent Group
- Choose the Geometry Parent Group
- Choose the Method Parent Group
- Name the operation
- Create a new operation

- Select faces to create boundaries within the operation
- Generate the operation and observe the results

❑ Choose the **Create Operation** icon. 

❑ Make sure the Type is set to **mill_planar**.

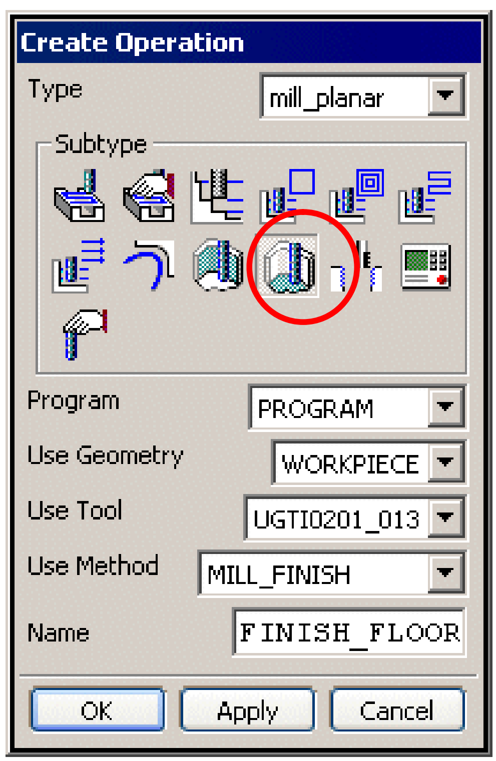
❑ Choose the **FINISH_FLOOR** icon. 

❑ Set the Program to **Program**.

❑ Set Use geometry to **WORKPIECE**.

❑ Set Use Tool to **UGTI0201_013**.

❑ Set Use Method to **MILL_FINISH**.

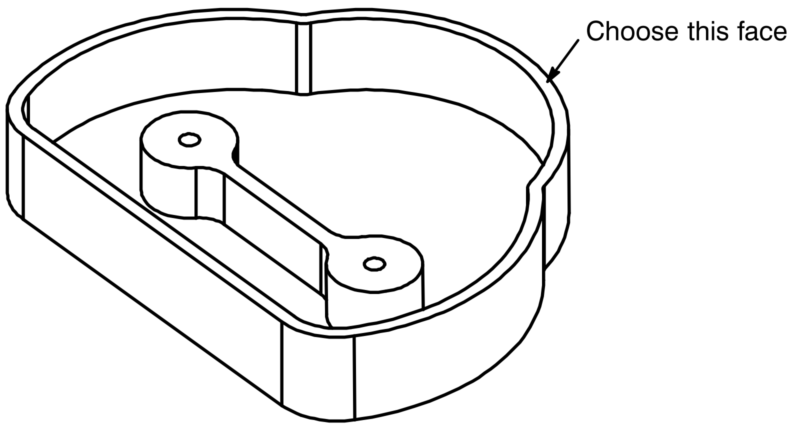


❑ Choose **OK**.

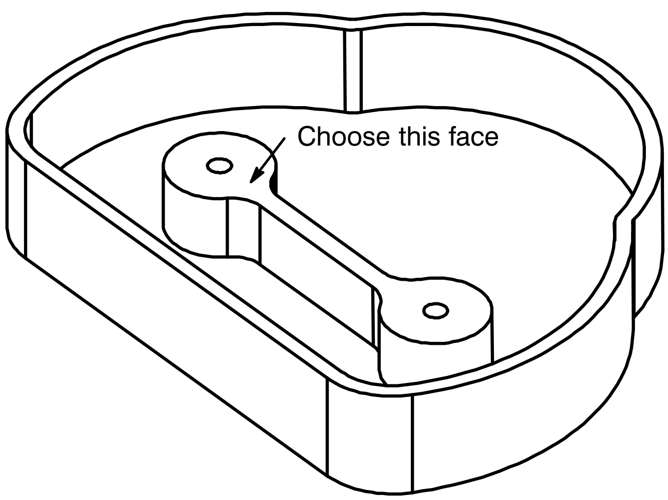
Step 3 Defining the part boundaries.

❑ Choose the **Part** icon and then **Select**.

❑ Select the top face of the pocket wall.



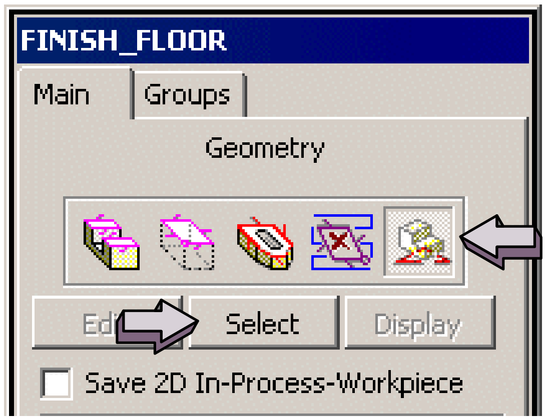
❑ Choose the top face of the dog bone area.



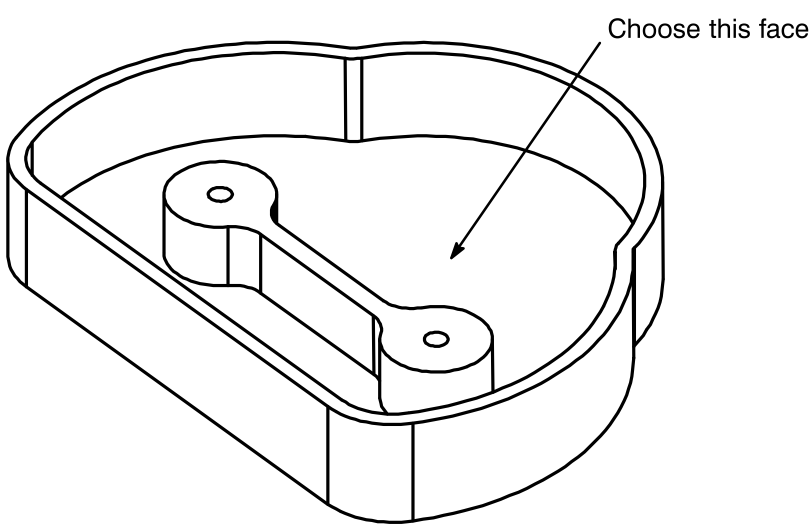
❑ Choose **OK** to return to the FINISH_FLOOR dialog.

Step 4 Selecting the Floor Plane.

❑ In the Geometry area of the dialog, choose the **Floor** icon and then **Select.**



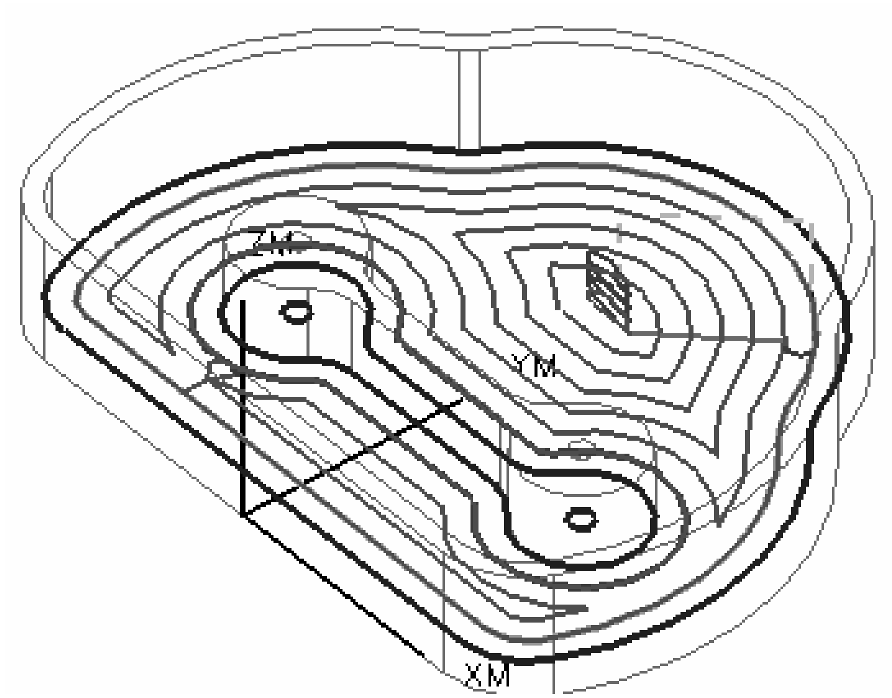
❑ Select the bottom of the pocket as the Floor Plane.



❑ Choose **OK**.

Step 5 Generating the operation.

❑ **Generate** the operation.



The tool finishes the floor of the part.

❑ Choose **OK.**

❑ **Save** and **Close** the part.

This completes the activity.

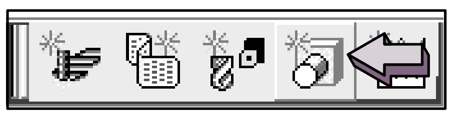
MILL_BND Geometry Parent Groups

When an area of a part is machined using many different operations, it may be desirable to create Geometry Parent Groups. This allows you to select geometry once that can be used in many different operations.

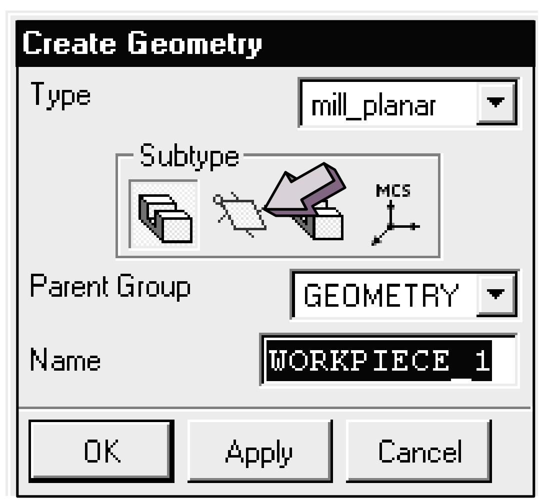
MILL_BND is one type of Geometry Parent Group that is used in Planar Milling operations. It allows you to define boundary geometry once and reuse it many times.

Creating MILL_BND Parent Groups

To create a Geometry Parent Group, choose the Create Geometry icon from the main menu bar.



Choose the MILL_BND icon, as shown below, to create a boundary.



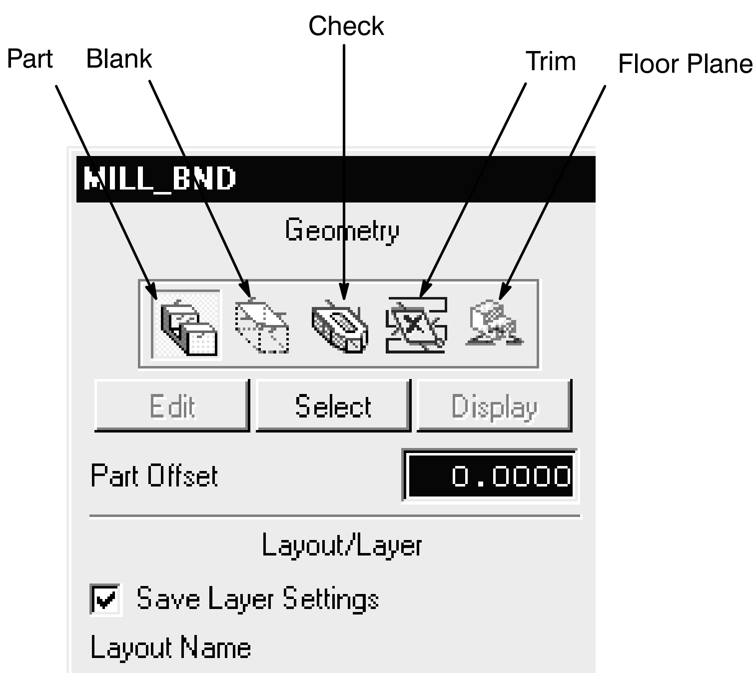
MILL_BND Geometry Parent can be used by the following operation types:

- PLANAR_MILL
- PLANAR_PROFILE
- ROUGH_FOLLOW
- ROUGH_ZIGZAG
- ROUGH_ZIG
- CLEANUP_CORNERS
- FINISH_WALLS
- FINISH_FLOORS

When creating MILL_BND Parent Groups, some boundary settings are not available, such as Convex/Concave Edges and some of the Custom Boundary Data options.

At times it may become necessary to make edits and or modifications to the boundary when used in a particular operation. These operation specific edits or modifications should be performed within the specific operation only (that way you can still use the MILL_BND parent in other operations without affecting the geometry of those operations).

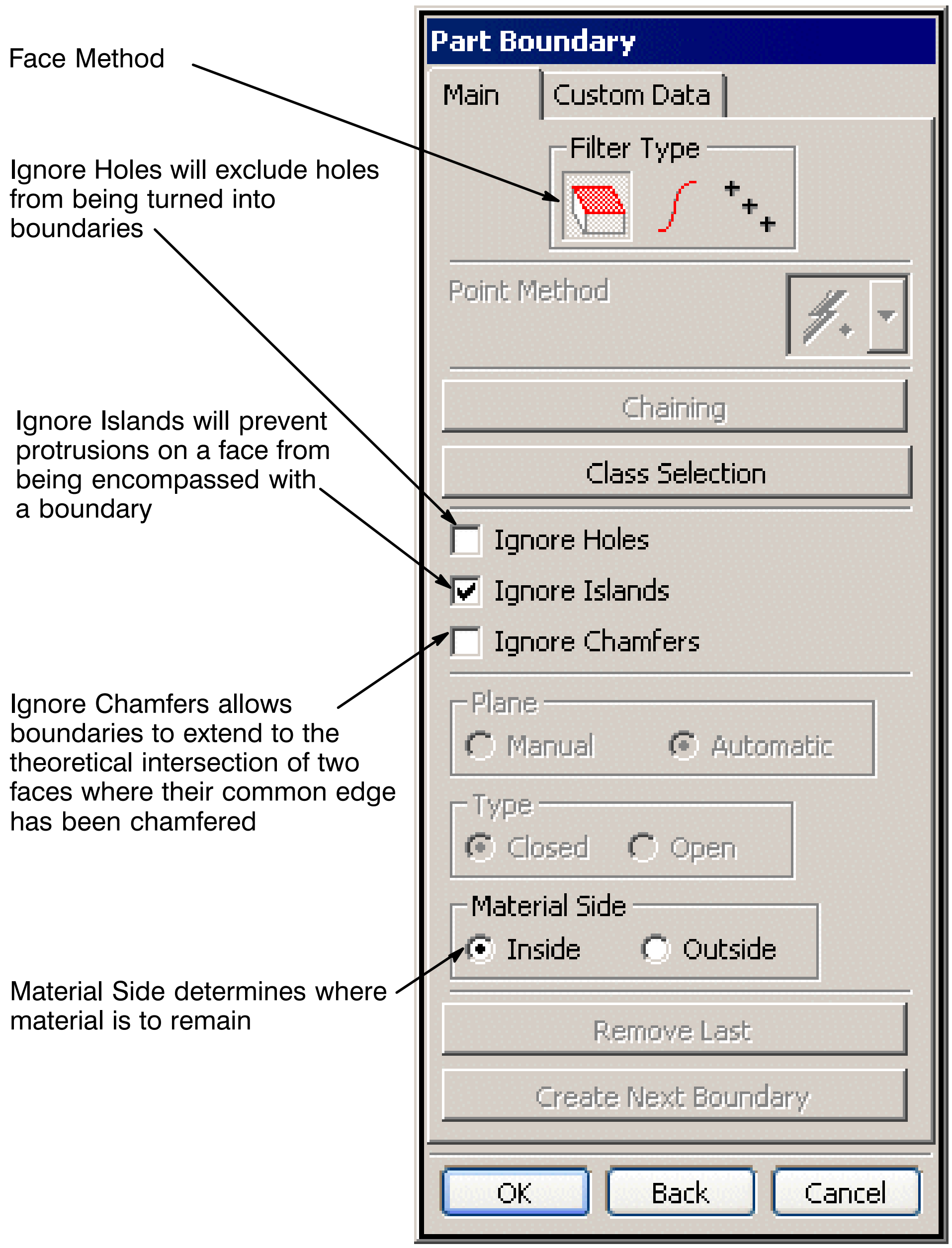
The MILL_BND dialog below shows the different boundary types and the Floor Plane option that can be specified for a Planar Milling operation. You will focus on the Part, and Floor Plane.



Selection of Geometry for the MILL_BND Parent Group

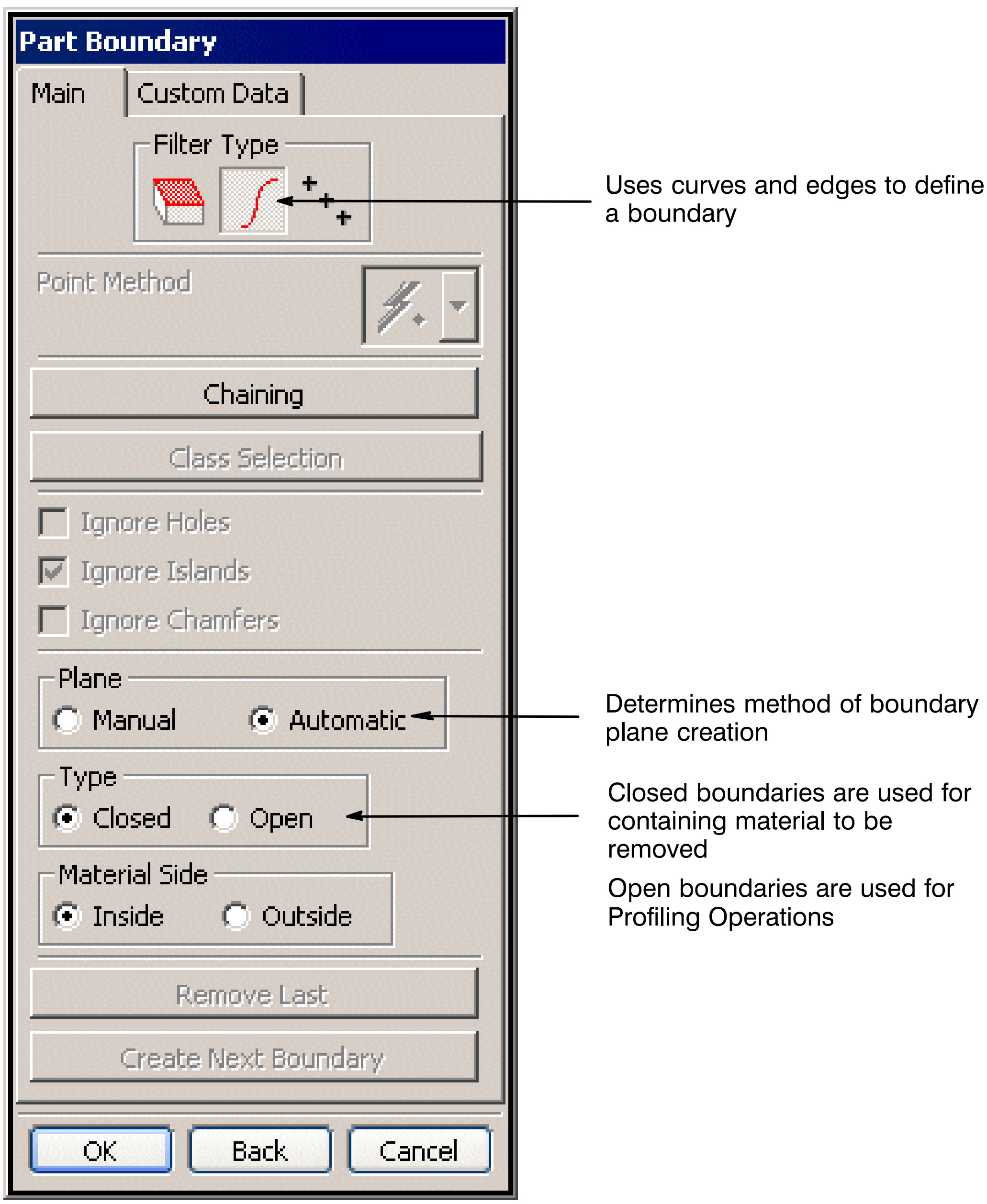
Face Selection Method

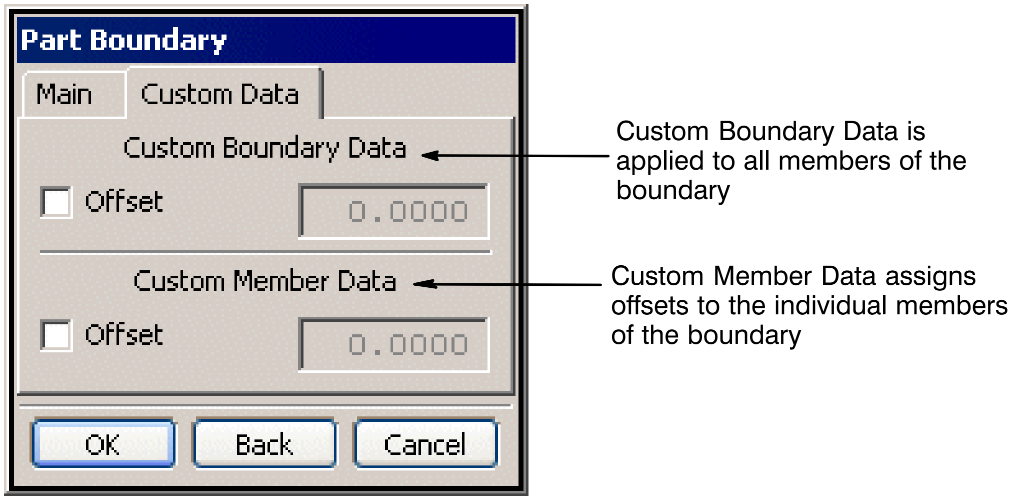
Boundaries created within MILL_BND Parent Groups using faces are created using Part, Blank, or Check Boundary Geometry. Below, the Part Boundary dialog is displayed.



Curves and Edges Selection Method

When you use Curves and Edges to define boundaries, you will have the following options available on the dialogs:





Creating Boundaries by selecting faces is one of the quickest methods to define a boundary. There are times when it is necessary to use curves and/or edges to define a boundary.

Rules for Creating Boundaries Using Curves

Some basic rules to follow when creating a boundary using curves:

13

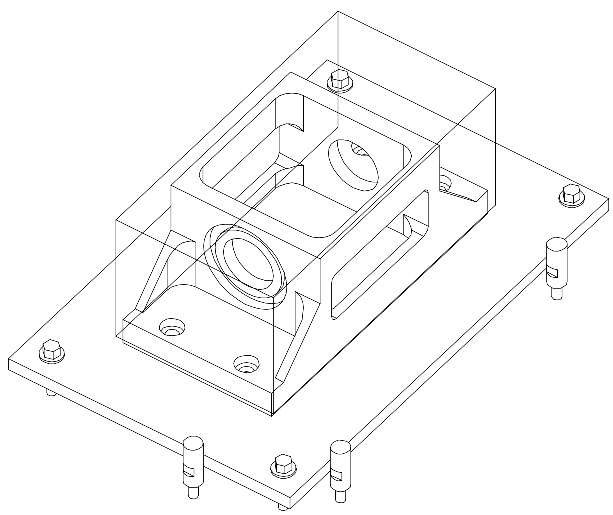
- Select the curves and edges in sequential order as if you are driving the tool around the boundary
- Curves and edges do not have to meet to form a boundary, but they must intersect theoretically; you can select curves and leave gaps, and the curves will extend to meet each other
- The Automatic Boundary Plane is determined by the first two entities selected
- The first curve selected is the start location of the boundary
- When creating open boundaries you must indicate which side of the boundary the material will remain (imagine you are in a car driving along the path of your boundary from the start location to the end; if the material would be to your right, select right; and left, if it is to the left)

Activity 13—4: Creating and Using Geo. Parent Groups

In this activity you will create and use Geometry Parent Groups in Planar Milling operations.

Step 1 Open and rename the part file and enter the Manufacturing application.

- ❑ Open the part file **mmp_bearing_case_mfg_2.prt.**



- ❑ Save the part file as *****_bearing_case_mfg_2.prt.**
- ❑ Enter the **Manufacturing** application.

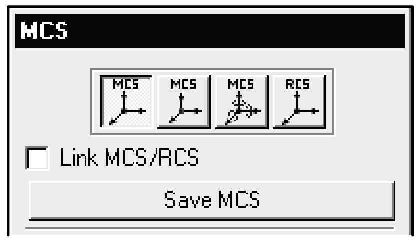
The tools required are already defined in this part.

Step 2 Changing the MCS.

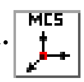
The MCS in this part is not the one you want to use when creating the operations used in this activity. You are going to edit the existing MCS.

❑ In the **Geometry View** of the Operation Navigator, double click on **MCS_MILL.**

The Mill_Orient dialog is displayed.



You are going to move the MCS to the WCS coordinates.

❑ Choose the **MCS Origin** icon. 

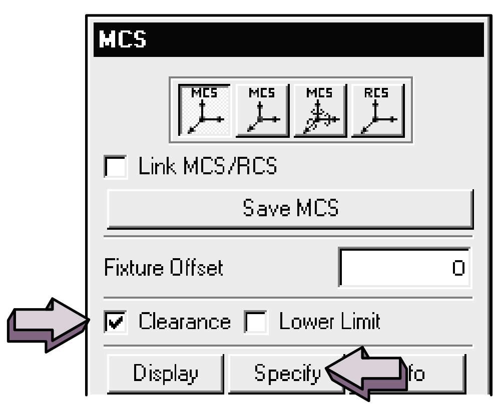
The Point Constructor dialog is displayed.

❑ Choose **Reset** to set all three coordinate fields to **Zero** and choose **OK**.

The MCS is moved to the WCS position. Now each time you create an operation the output will be in reference to this MCS.

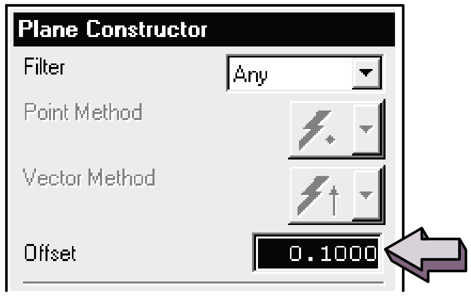
Now you are going to create a Clearance Plane using the Mill_Orient option on this dialog.

❑ Click the Clearance option **ON** and then choose **Specify.**



The Plane Constructor dialog is displayed.

❑ Set the Clearance Plane **Offset** to **.100.**



❑ Select the top face of the Blank.

❑ Choose **OK** until you return to the Operation Navigator.

Step 3 Selecting the geometry for the WORKPIECE.

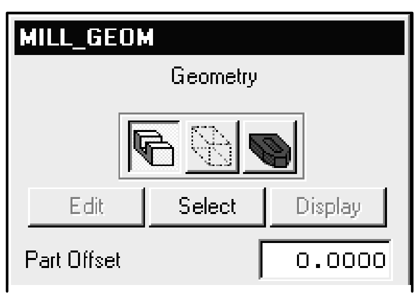
You are going to select the Part and Blank geometry for the WORKPIECE Parent Group.

13

❑ In the Geometry View of the Operation Navigator, expand the MCS_MILL Group object.

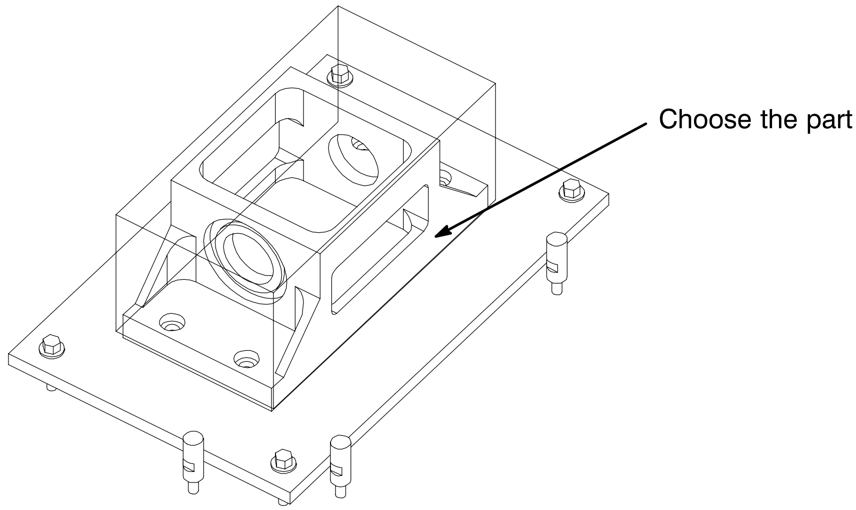
❑ Double click on the Parent **WORKPIECE.**

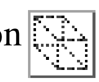
The MILL_GEOM dialog is displayed.



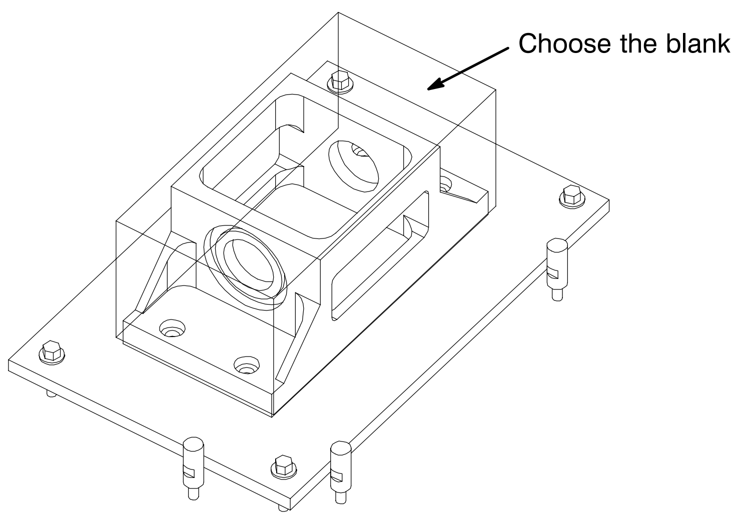
❑ Choose the **Part** icon  and then choose **Select**.

❑ Choose the part as shown and then choose **OK**.



❑ Choose the **Blank** icon  and then choose **Select**.

❑ Choose the Blank object in the graphics window and then choose **OK**.



❑ Choose **OK** and return to the Operation Navigator.

Step 4 Selecting the boundaries for the MILL_BND Parent Group.

You are going to create a MILL_BND Parent Group and create several boundaries for that group object. Before you get started, you are going to blank the Blank geometry for easier selection.

❑ Choose **Edit** → **Blank** → **Blank.**

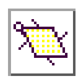
❑ Select the Blank geometry and then choose **OK.**

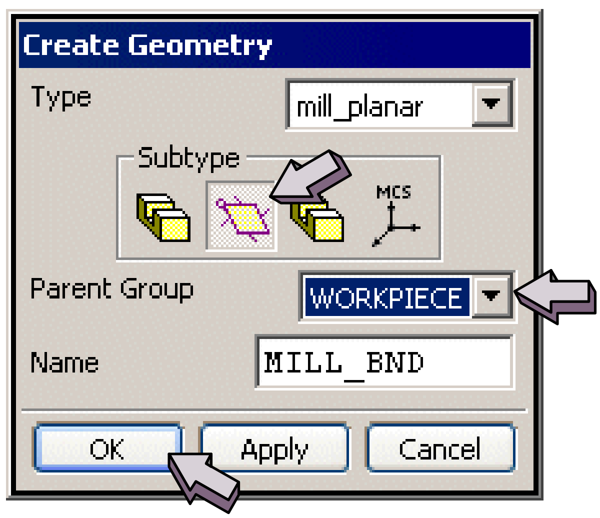
NOTE You could have Blanked the Blank using the Assembly Navigator.

You want WORKPIECE to be the Parent Group for the MILL_BND Group object.

❑ Choose the **Create Geometry** icon.

❑ In the Create Geometry dialog, choose **WORKPIECE** as the Parent Group.

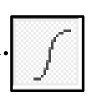
❑ Choose the **MILL_BND** icon  and then choose **OK**.



The MILL_BND dialog is displayed. You are going to define the boundaries necessary to cut the pocket in the middle of the part.

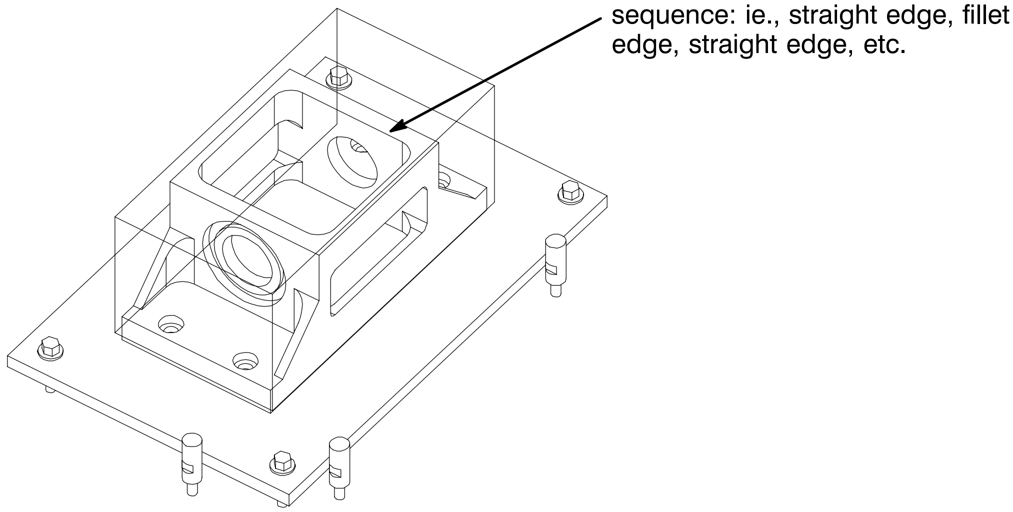
❑ Choose the **Part** icon  and then choose **Select**.

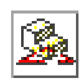
The Part Boundary dialog is displayed.

❑ Choose the **Curve** Boundary icon. 

❑ Change the Material Side to **Outside**.

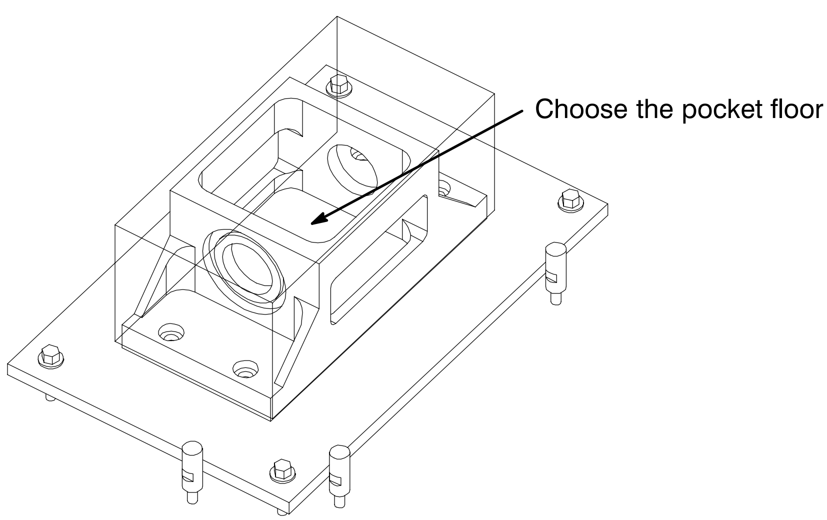
❑ Choose the top edges of the pocket geometry as shown and then choose **OK**.



❑ Choose the **Floor** icon  and then choose **Select.**

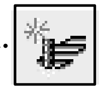
❑ Change Filter to **Face.**

❑ Select the floor of the pocket**.**

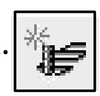
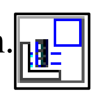


❑ Choose **OK** until you return to the Operation Navigator.

Step 5 Creating a roughing operation using the Geometry Parent Groups and pre-selected geometry.

- ❑ Choose the **Create Operation** icon. 
- ❑ Change the Type to **mill_planar.**
- ❑ Choose the **ROUGH_FOLLOW** icon.
- ❑ Set the Program to **PROGRAM.**
- ❑ Set Use Geometry to **MILL_BND**.
- ❑ Set Use Tool to **UGTI0201_024.**
- ❑ Set Use Method to **MILL_ROUGH.**
- ❑ Choose **OK**.
- ❑ Choose **OK** to return to the Operation Navigator.

Step 6 You will create another Planar Milling operation to Profile the pocket wall.

- ❑ Choose the **Create Operation** icon. 
- ❑ Choose the **PLANAR_PROFILE** icon. 
- ❑ Set Use Geometry to **MILL_BND**.
- ❑ Set Use Tool to **UGTI0201_023.**
- ❑ Set Use Method to **MILL_FINISH.**
- ❑ Choose **OK**.
- ❑ Choose **OK** to return to the Operation Navigator.

You are now ready to generate these operations.

Step 7 Generate the tool paths.

❑ Change the Operation Navigator to the **Program Order View**.

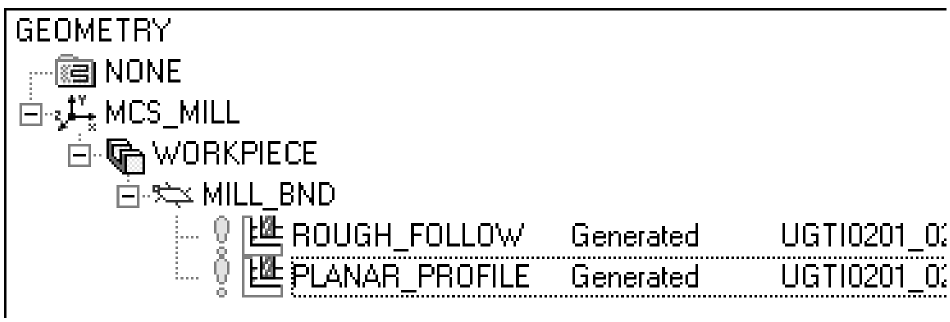
You can generate the tool paths sequentially using the Operation Navigator.

❑ Highlight **Program** on the Operation Navigator.

❑ Using **MB3**, choose **Generate**.

❑ Accept the paths as they are generated.

Look at the tree structure on the Geometry View of the Operation Navigator. You can see that each operation is placed under the appropriate Parent Group.



❑ **Save** and **Close** the Part.

This concludes the activity and the lesson.

SUMMARY

Planar Milling operations offer diverse methods of roughing and finishing planar or 2D geometry. The flexibility of these operations allows for roughing, semi-finishing and finishing, using a multitude of options to achieve the desired results.

In this lesson you:

- used boundaries to contain the tool path
- used boundaries to mill the top face of a part
- defined boundary geometry inside of an operation.
- created an operation to rough and finish a pocket
- selected and used boundary geometry in a MILL_BND Geometry Parent Group
- edited a MILL_BND Geometry Parent Group

EDS

Planar Milling - Intermediate

Lesson 14

PURPOSE

This lesson shows you some of the intermediate functions of Planar Milling. You will learn how to create multi-level cutting tool paths, modify custom boundary data and utilize 2D Contact Contour machining.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Create a Planar Milling tool path that cuts in multiple levels
- Modify and edit data that pertains directly to boundaries and their members
- Utilize 2D Contact Contour machining

This lesson contains the following activities:

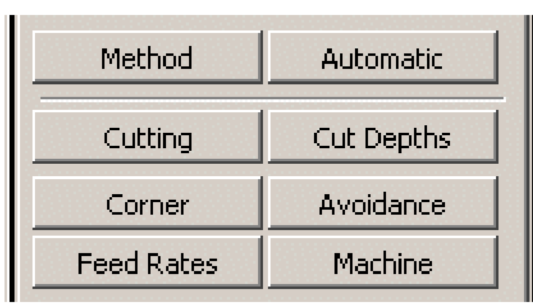
Multi-level Cutting

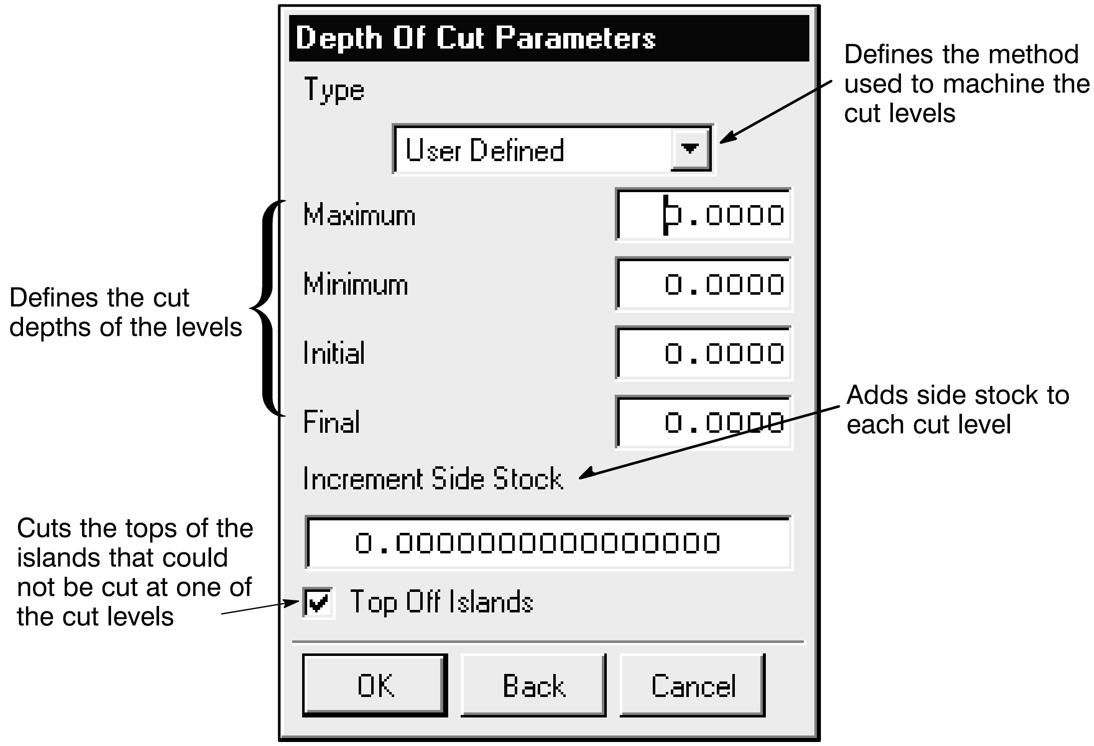
To successfully perform multi-level cutting in Planar Milling, some rules need to be followed. These rules pertain to best practices for boundary creation, and setting the depth per pass. They are:

- Planar Milling ignores all boundaries until the tool is below the level of that boundary; each boundary needs to be at the top of the geometry that it represents
- The part can contain an unlimited number of boundaries
- You can cut sides and tops of islands
- Sometimes a Blank boundary is not necessary; in those cases, the highest part boundary describes the material available for milling, while all other part boundaries describe the actual part geometry
- You can specify the depths of cut or the maximum and minimum cut depths
- If you have islands within the pocket, cut levels are created at the top of these islands; if a level cannot be generated at the top of an island and remain within the cut depth constraint, you can specify an optional pass to cut the top face of the islands

Depth of Cut

You define the cut levels for your multi-level tool path using the Depth of Cut Parameters dialog which is activated by the Cut Depths button from the Main PLANAR_MILLING (or like type operation) dialog.



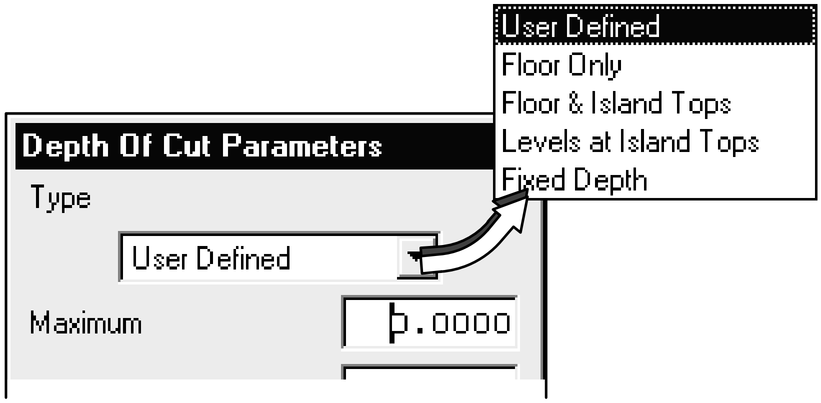


NOTE When there are islands in the part, and the maximum and minimum depths of cut are defined, the tops of the islands will be one of the cut depths. This is subject to the number of islands defined and the specified maximum and minimum cut depths.

NOTE Use the Top Off Islands option to ensure that they are cut even if the Minimum depth value bypasses an island top.

14

There are five types of Cut Levels available under the Type label:



- **User Defined –** You can set the Maximum, Minimum, Initial, and Last depths of cut
- **Floor Only –** Generates a single cut level at the Floor plane
- **Floor & Island Tops –** Generates one cut level at the Floor plane and then generates a cleanup cut at the top of each island
- **Levels at Island Tops –** Generates a cut level at the top of each island
- **Fixed Depth –** Generates cut levels at a constant depth, using the Maximum field

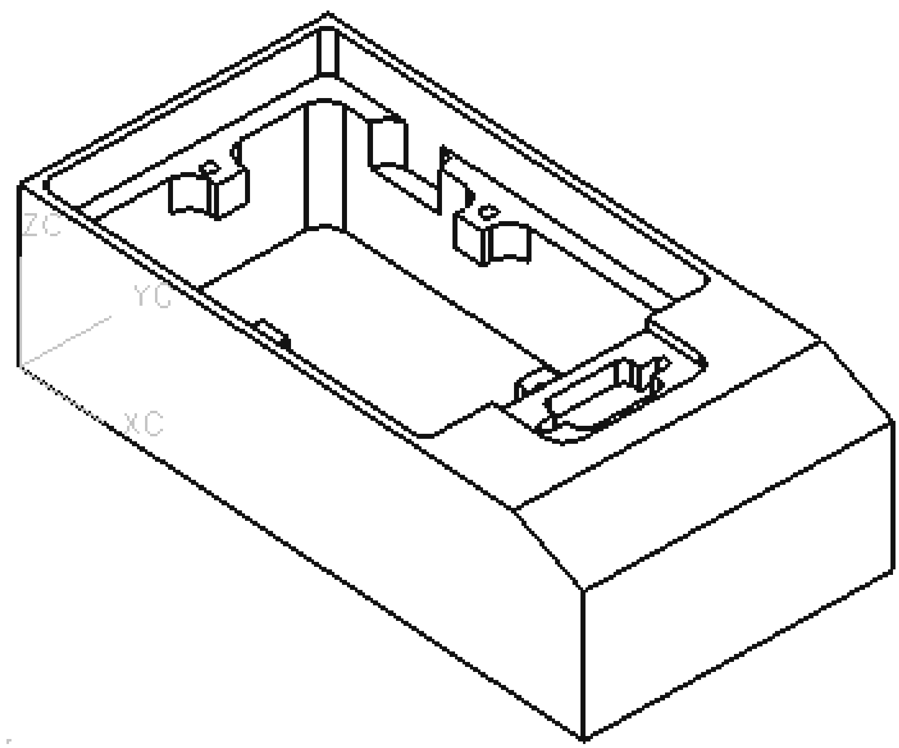
14

Activity 14–1: Planar Milling – Multi Level Cutting

In this activity, you will use Planar Milling to remove material in multiple levels.

Step 1 Open the part file, rename and enter the Manufacturing application.

❑ Open the part file **mmp_frame_mfg_2.prt**.



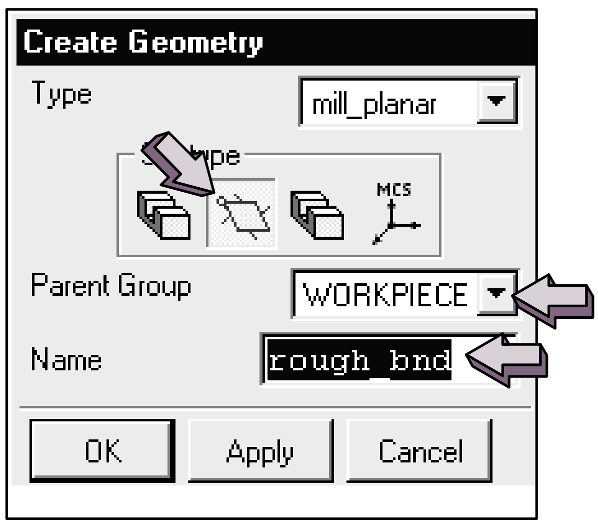
❑ Rename the part *****_frame_mfg_2.prt** using the **File→Save As** option on the menu bar.

❑ Choose **Application → Manufacturing**.

14

Step 2 Creating the MILL_BND Geometry Parent Group.

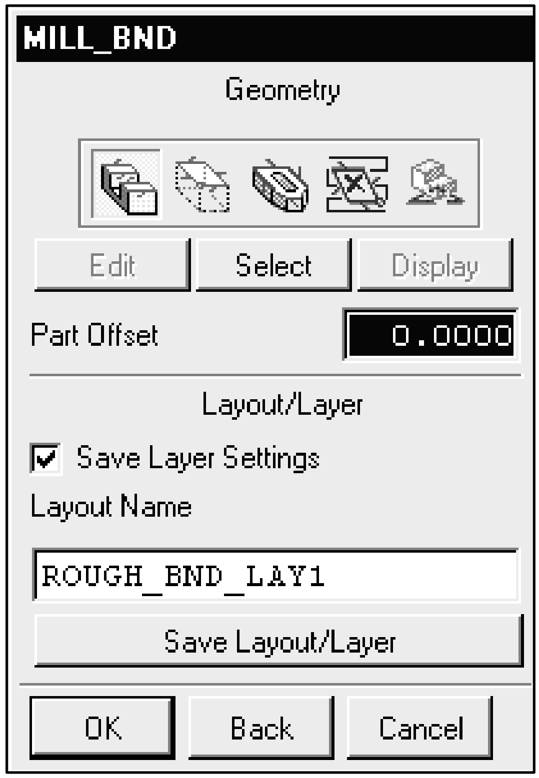
- ❑ Choose the Create Geometry icon from the menu bar.
- ❑ In the Create Geometry dialog, set the Type to **mill_planar**.
- ❑ Choose the **MILL_BND** icon.
- ❑ Change the Parent Group to **WORKPIECE**.
- ❑ In the Name field, type in **rough_bnd.**



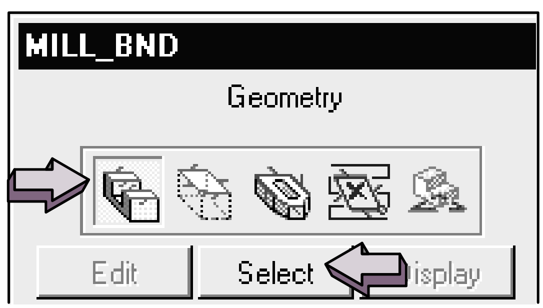
- ❑ Choose **OK**.

14

The MILL_BND dialog is displayed.

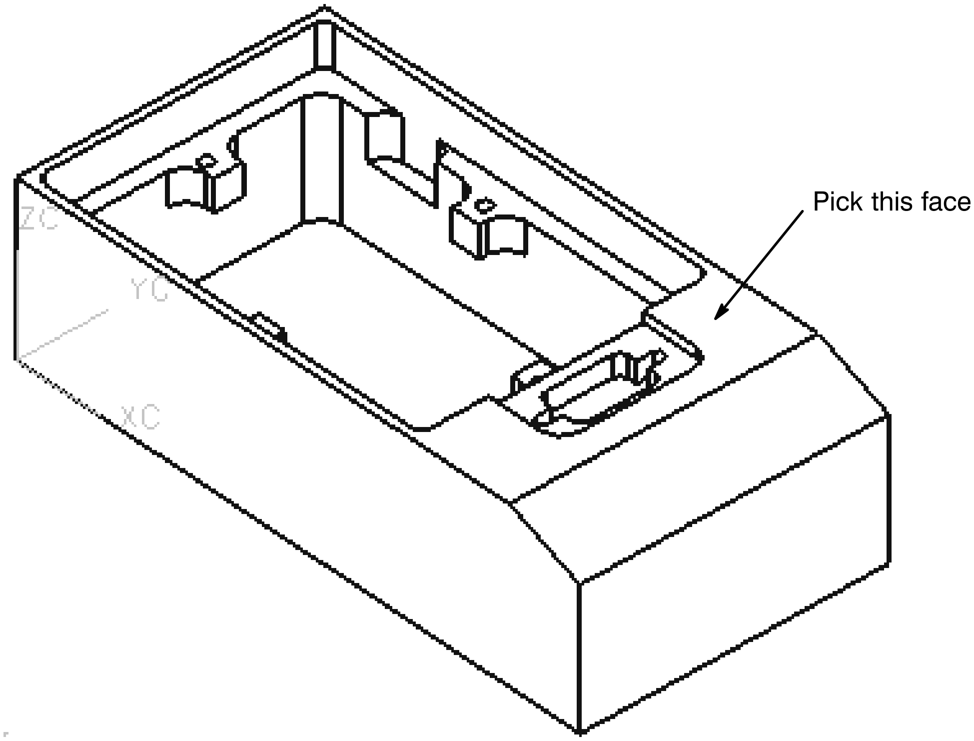


❑ Choose the **Part** icon and choose **Select.**

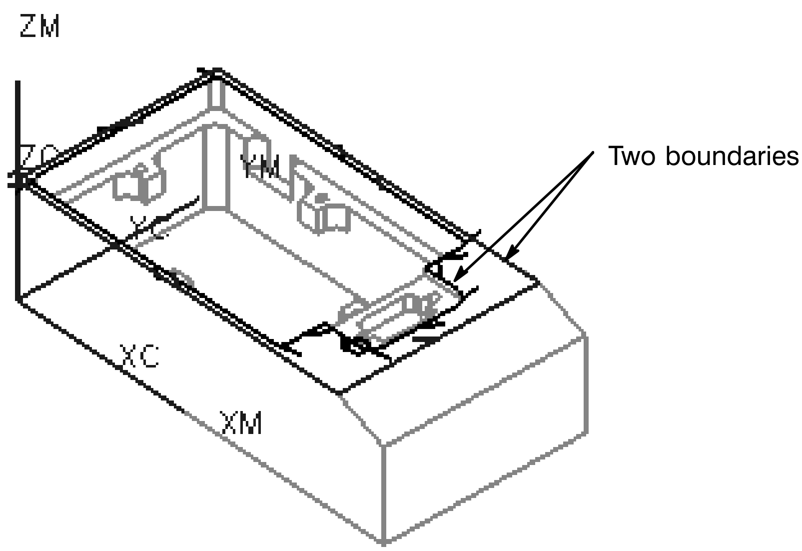


You will use the Face mode to choose the boundary.

❑ Choose the top face of the part, as shown.

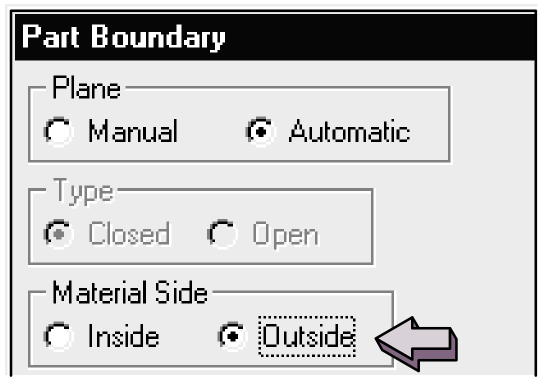


❑ Choose **OK.**



❑ In the MILL_BND dialog, with the Part icon still depressed, choose the **Edit** button.

❑ Change the Material Side to **Outside.**

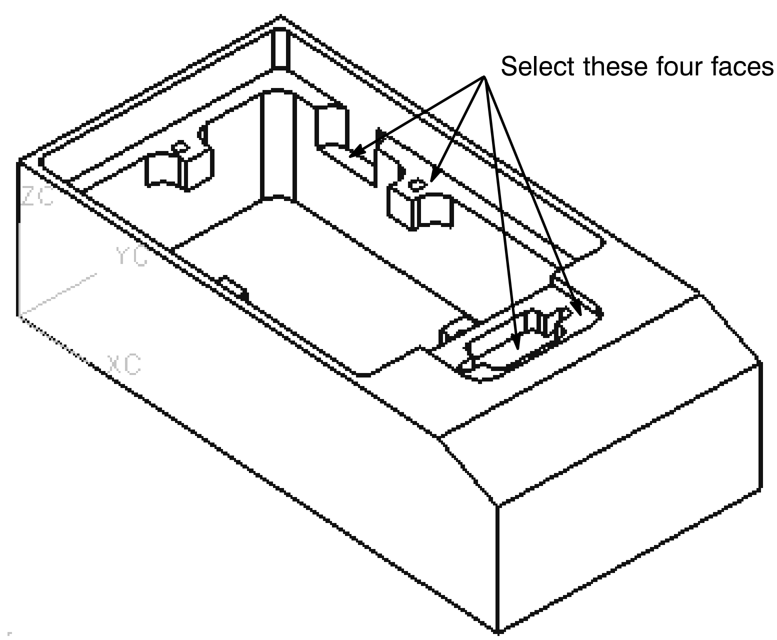


The Part boundary that describes the area to be machined has now been created. Next, you will create the Part boundaries that represent other features that require machining.

❑ On the Part Boundary dialog, choose **Append**.

Append allows you to add new boundaries to those that have already been selected.

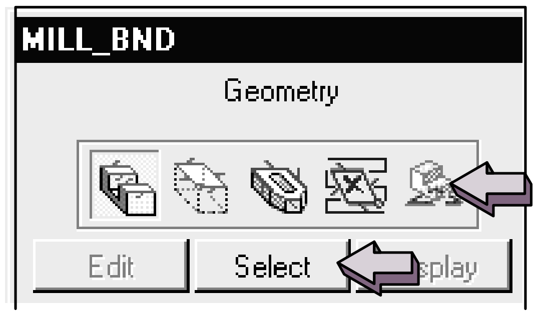
❑ Choose the four faces highlighted below.



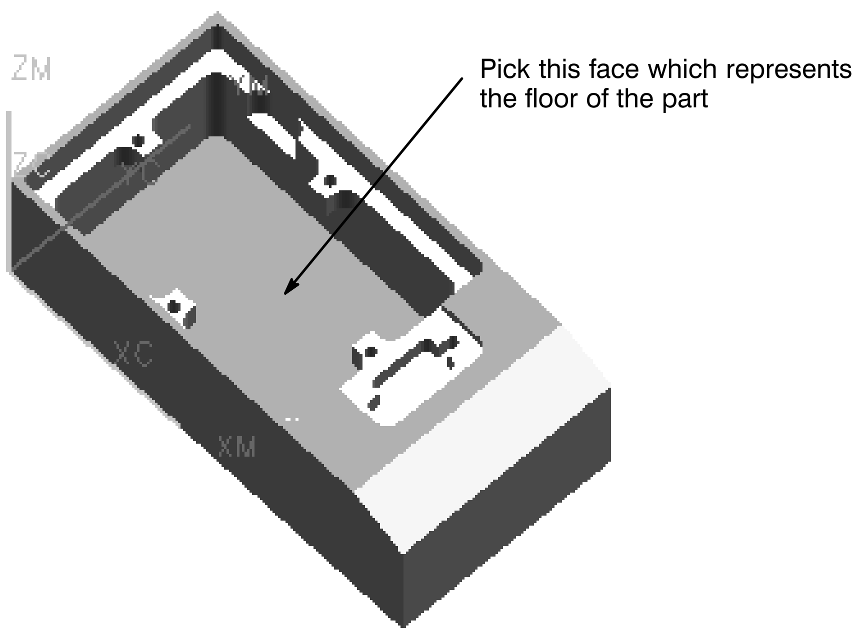
❑ Choose **OK** until the MILL_BND dialog is displayed.

You will now choose the Floor Plane. The Floor Plane represents the lowest level that the cutter will cut to. The Floor Plane function is similar to that of the Bottom Range in Cavity Milling.

❑ Choose the **Floor** icon, and then choose **Select**.



❑ Choose the floor of the part, as shown.



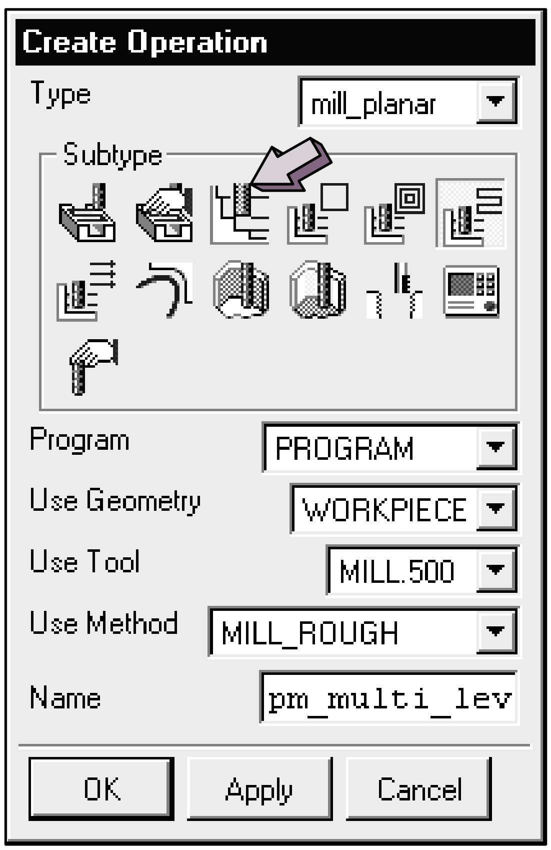
❑ Choose **OK** in the Plane Constructor dialog.

❑ Choose **OK** in the MILL_BND dialog.

The Geometry Parent is created. You will now generate the operation.

Step 3 Creating a Planar Milling operation.

- ❑ Choose the **Create Operation** icon.
- ❑ Set the operation Subtype to **Planar Mill**.
- ❑ For the Program parent, choose **ROUGHING**.
- ❑ For the Geometry parent, choose **ROUGH_BND**.
- ❑ Change the Use Tool to **MILL.500**.
- ❑ Choose **MILL_FINISH** as the Method**.**
- ❑ In the Name field, type in **pm_multi_level**.



- ❑ Choose **OK**.

14

The Planar Mill dialog is displayed. The only parameter left to set is the Cut Depth. You will set it to half of the tool diameter.

❑ Change the Cut Depth maximum value to **0.375**.

❑ Choose **Generate**.

❑ After the operation is generated, use the **Verify → Dynamic** option to show the results of your work.

❑ **Save** and **Close** the part.

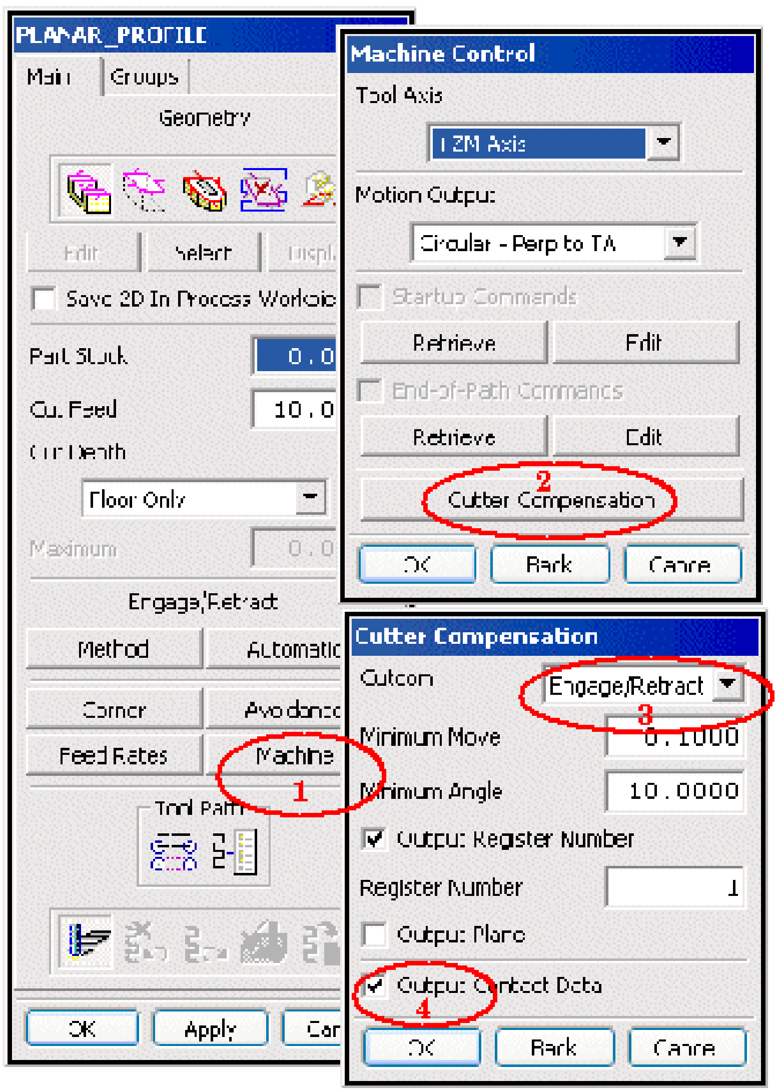
You have completed this activity.

2D Contact Contour Machining

2D Contact Contour machining outputs tool contact data rather than tool end locations and is applicable only for planar profile operations. This allows the operator to effectively use any size cutter to machine the part.

This method is sometimes commonly referred to as piece part or part print programming.

2D Contact Contour Machining can be used only in planar profile operations and is activated by selecting **Machine → Cutter Compensation → Output Contact Data** (note that Engage/Retract option for Cutcom must be selected).



14

Activity 14—2: Using 2D Contact Contour Machining

In this activity you will edit and replay an existing operation, observing ON and TANTO boundary members and cutter path output. You will then activate 2D Contact Contour machining and regenerate the tool path.

Step 1 Open the part mmp_gmill_2d_contour and choose the Manufacturing application.

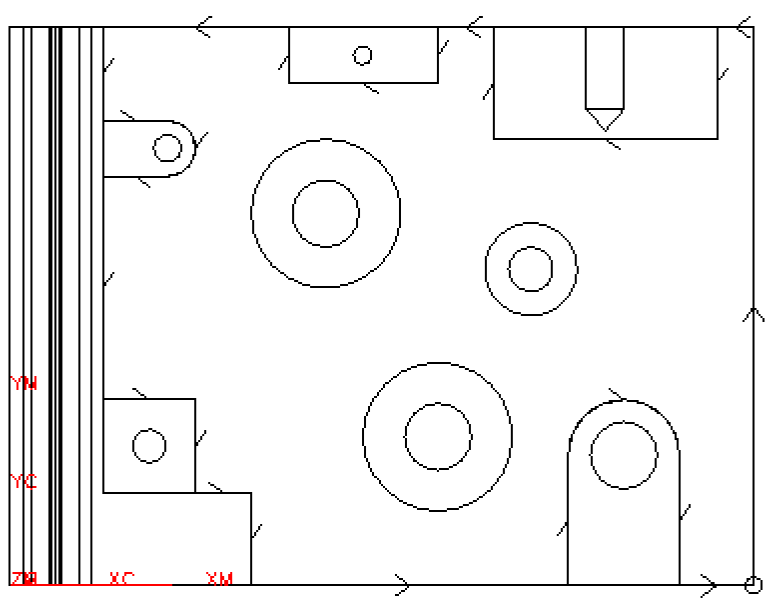
- ❑ From the menu bar, select **File**.
- ❑ Choose **Open**.
- ❑ Select the file **mmp_gmill_2d_contour**, then choose **OK**.
- ❑ Choose **Application→Manufacturing**.

Step 2 Activate the Operation Navigator.

- ❑ Choose the **Operation Navigator** tab from the resource bar.

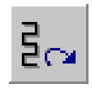
Step 3 Edit the existing operation, FINISH_PROFILE, display the boundary members and replay the tool path.

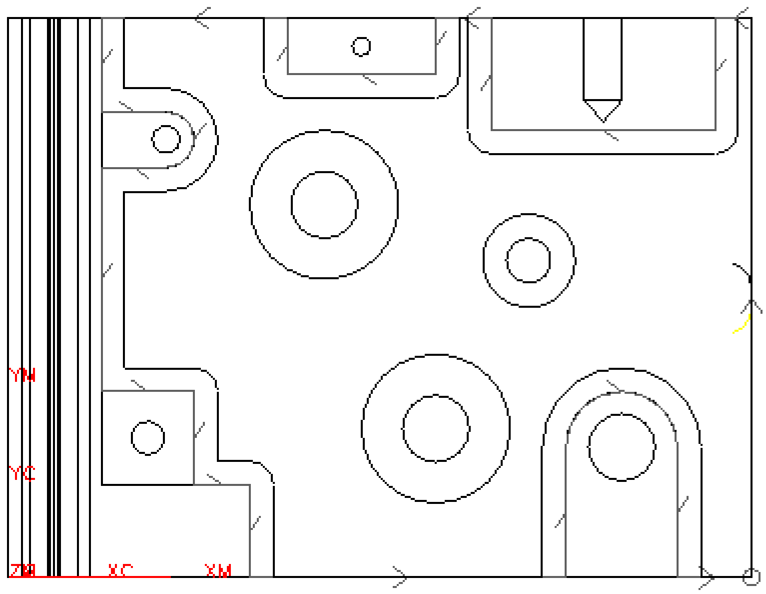
- ❑ Highlight the operation, **FINISH_PROFILE**, choose **MB3, Edit**, then choose **DISPLAY**.



14

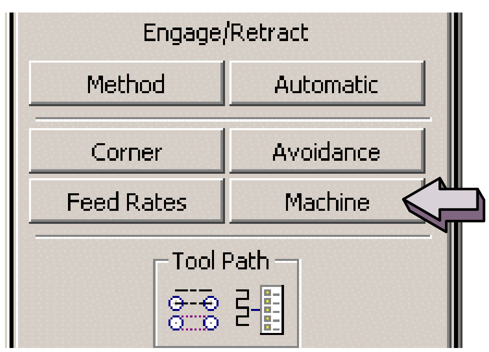
Note the ON and TANTO condition of the various boundary members.

- ❑ Choose the **REPLAY** button  and observe the tool path.



Step 4 Activate the 2D Contact Contour machining option.

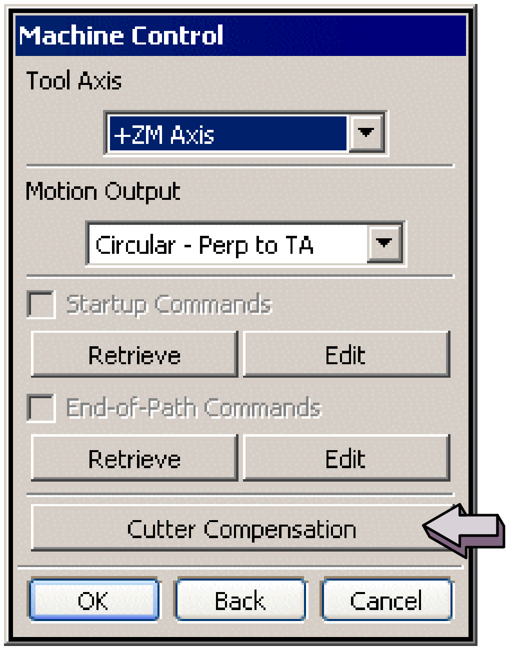
- ❑ Choose Machine from the **PLANAR_PROFILE** property page.



The Machine Control dialog is displayed.

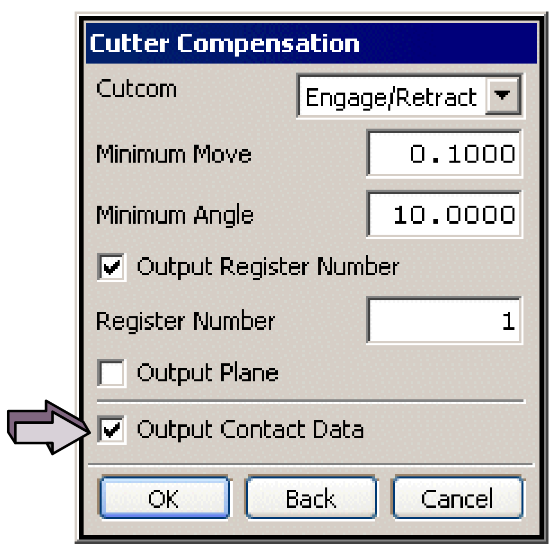
14

❑ Choose the **Cutter Compensation** button from the **Machine Control** dialog.



The Cutter Compensation dialog is displayed.

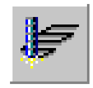
❑ Choose Output Contact Data ON.

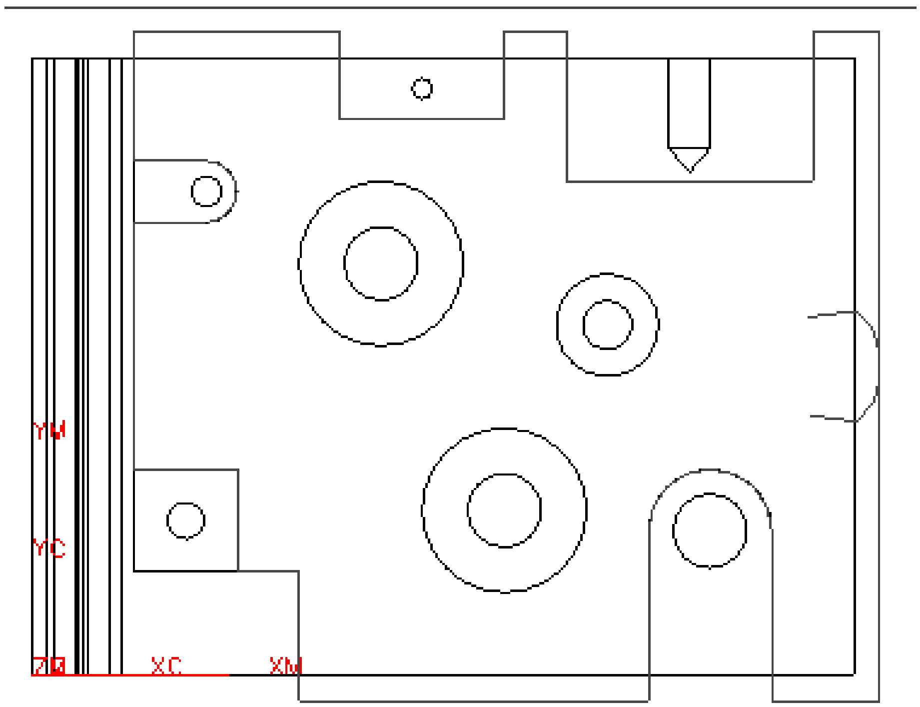


14

❑ Choose **OK** twice until you return to the **PLANAR_PROFILE** property page.

Step 5 Generate the tool path.

- ❑ Choose the **Generate** tool path button. 



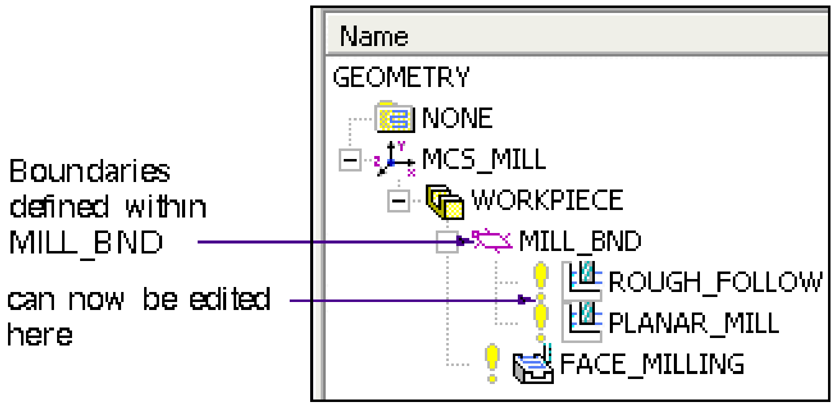
Notice the offset for the boundary segments that were set to ON. Also note that 2D Contact Contour is only available when Cutter Compensation is set to Engage/Retract and when Engage/Retract is set to Automatic.

- ❑ Close the Part file.

This concludes this activity.

Custom Boundary Data for Milling

Boundaries that are defined in geometry parent groups can now be edited from within the milling operations that inherit them. Boundary stock, cut feed rates, tool position, and in-path machine control commands for boundaries and boundary members can now be accessed through this method.



The following operation and boundary types can be edited within the operation:

- Face Milling (Blank and Check boundaries)
- Planar Milling (Part, Blank, Check, Trim boundaries)
- Cavity Milling (Part, Blank, Check Trim boundaries)
- Zlevel Milling (Trim boundaries)
- Surface Contouring-Area Milling (Trim boundaries)

Activity 14–3: Custom Boundary data

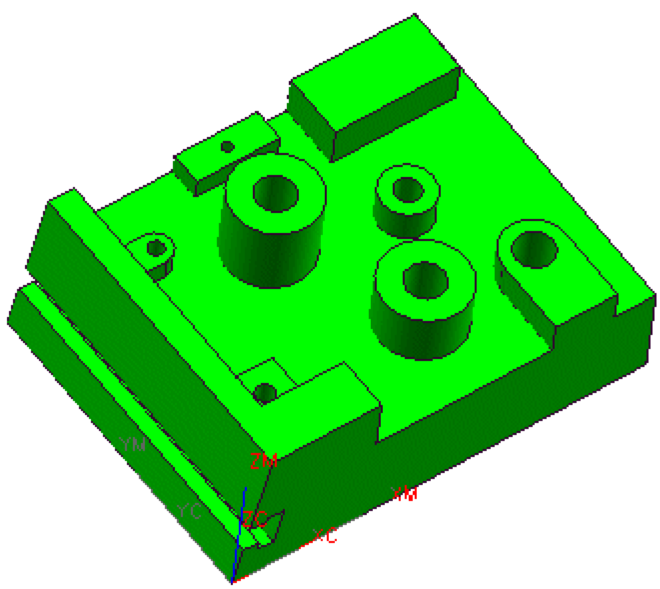
In this activity, you will edit boundary elements within an operation, by adding custom stock to boundary members. The boundary which is edited, is inherited from a Geometry parent group.

Step 1 Open the part mmp_custom_bnd_data and choose the Manufacturing application.

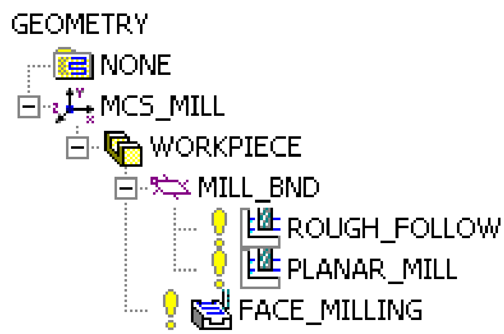
- ❑ From the menu bar, select **File**.
- ❑ Choose **Open**.
- ❑ Select the file **mmp_custom_bnd_data**, then choose **OK**.
- ❑ Choose **Application→Manufacturing**.

Step 2 Activate the Operation Navigator.

- ❑ Choose the **Operation Navigator** tab from the resource bar.



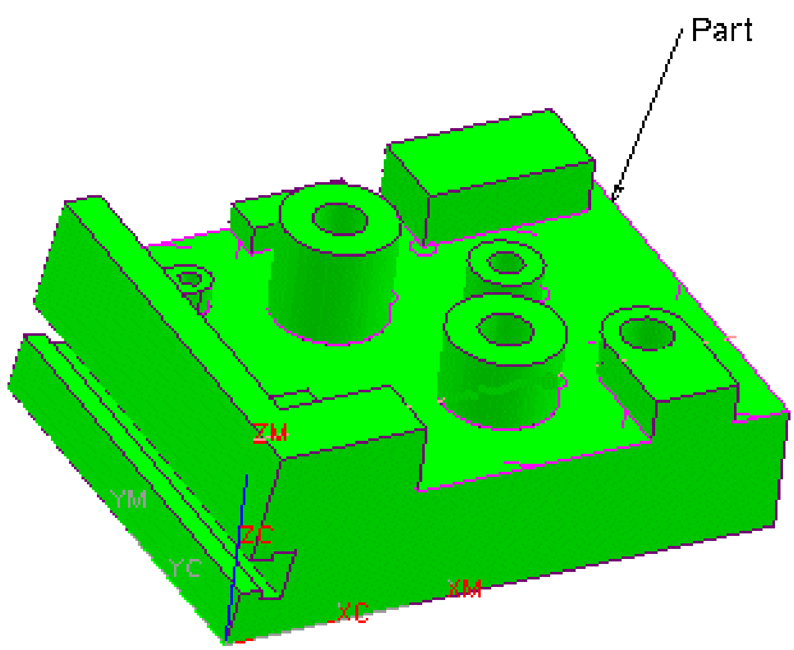
Step 3 Display the Geometry View in the Operation Navigator and expand the objects.



Step 4 Display the part boundaries which have been previously defined in the MILL_BND parent group.

- ❑ Double-click the **MILL_BND** parent group in the Operation Navigator.

- ❑ Choose the **Part** geometry and then **Display**.



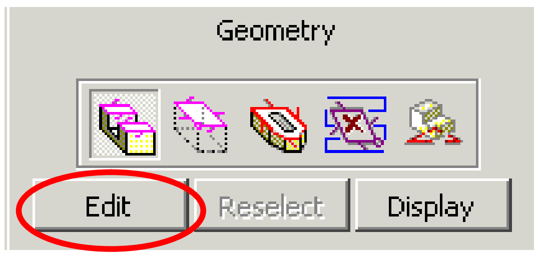
This boundary is inherited by the ROUGH_FOLLOW and PLANAR_MILL operations.

❑ **Cancel** the **MILL_BND** dialog.

Step 5 You will edit the part boundary within the ROUGH_FOLLOW operation by the addition of stock.

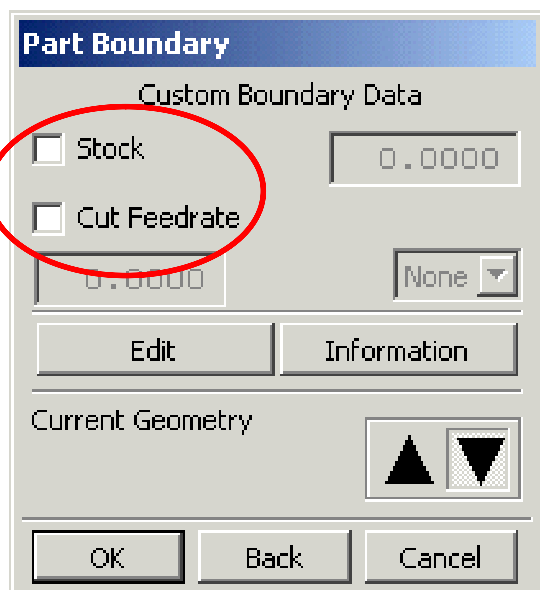
❑ Double-click the **ROUGH_FOLLOW** operation in the Operation Navigator.

The boundary data can be edited, but the inherited boundary geometry cannot be selected from within the operation.



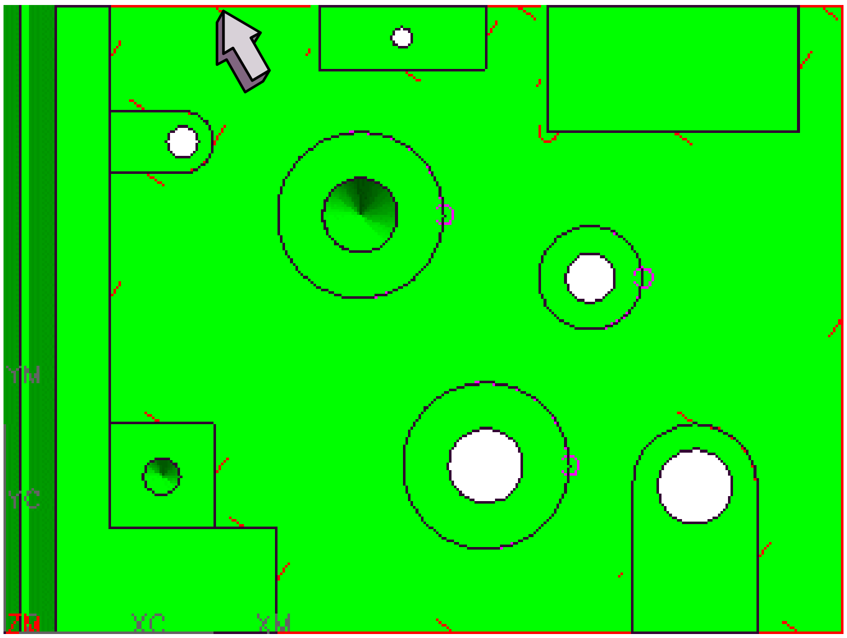
❑ Choose **Part** geometry and then **Edit**.

The Part Boundary dialog contains options that allow you to add stock and specify a cut feed rate for the specified boundary that was inherited from the MILL_BND parent.

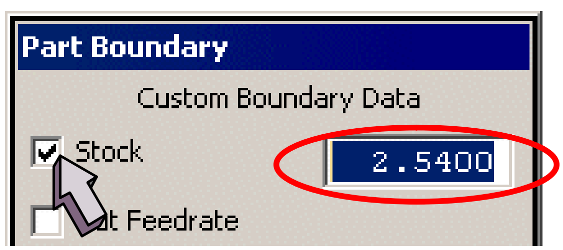


Boundary stock is added to and does not replace an offset defined within the geometry group.

❑ Select the boundary highlighted as illustrated below.



❑ Turn the **Stock** option **ON** and key in 2.54.



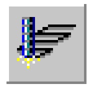
❑ Choose **OK** to accept the Custom Boundary Data.

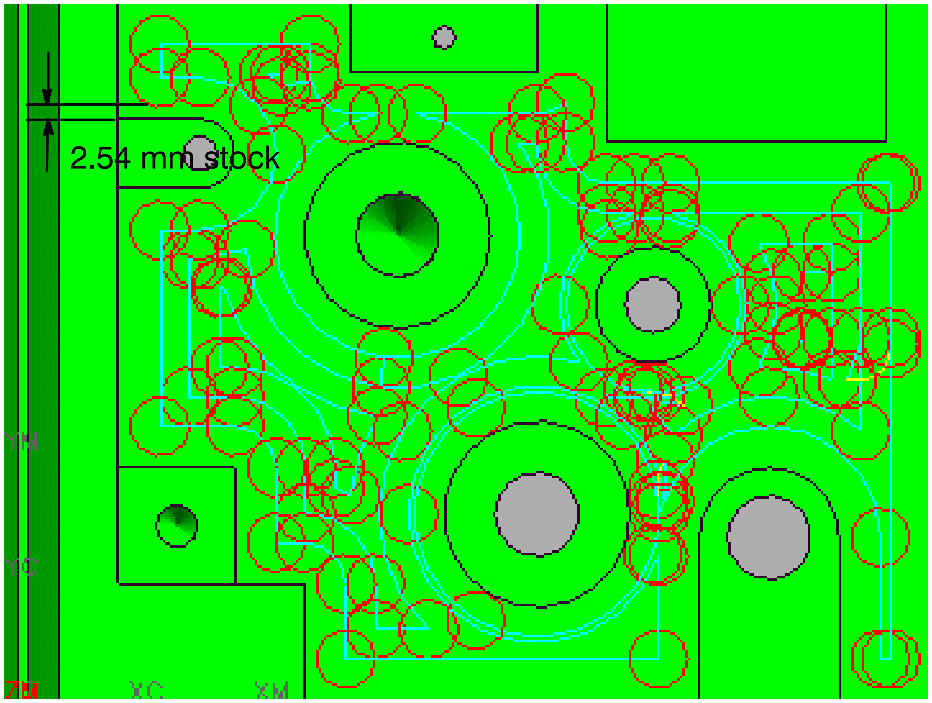
The 2.54 mm of stock will apply to the ROUGH_FOLLOW operation only.

Step 6 You will generate the tool path and observe how the stock is applied to the boundary.

❑ Choose **MB3→Replace View→Top**.

❑ Set the **Tool Display** option to **2–D.**

❑ Choose **Generate**  to generate the tool path.



❑ Choose **OK** to complete the operation.

❑ Close the part file.

This concludes this activity and the lesson.

SUMMARY

You have been introduced to some of the more intermediate features of Planar Milling operations that greatly enhance the flexibility of tool path generation.

In this lesson you:

- Created a Planar Milling operation to remove material in multiple cuts
- edited boundary members and data within an operation
- used 2D Contact Contour machining to generate simple tool paths

14

Planar Milling - Advanced

Lesson 15

PURPOSE

This lesson will teach you how to create a Planar Mill Profile operation. You will also learn how to create operations that automatically create boundaries from uncut material that can be used in clean-up operations.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Use the Profile cut pattern in an operation
- Create boundaries and operations from uncut material

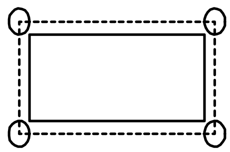
This lesson contains the following activities:

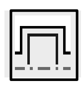
Introduction to Profiling

This portion of the lesson introduces Profiling.

Follow Periphery, Follow Part, and Zig-Zag are designed for milling areas. Generally, they use closed boundaries. Profiling typically uses an open boundary.

Profile Cut example



Profile follows a boundary using the side of the tool. For this method, the tool follows the direction of the boundary. 

The following activity will demonstrate the creation of an open boundary followed by the creation of an operation using the Profile cut method.

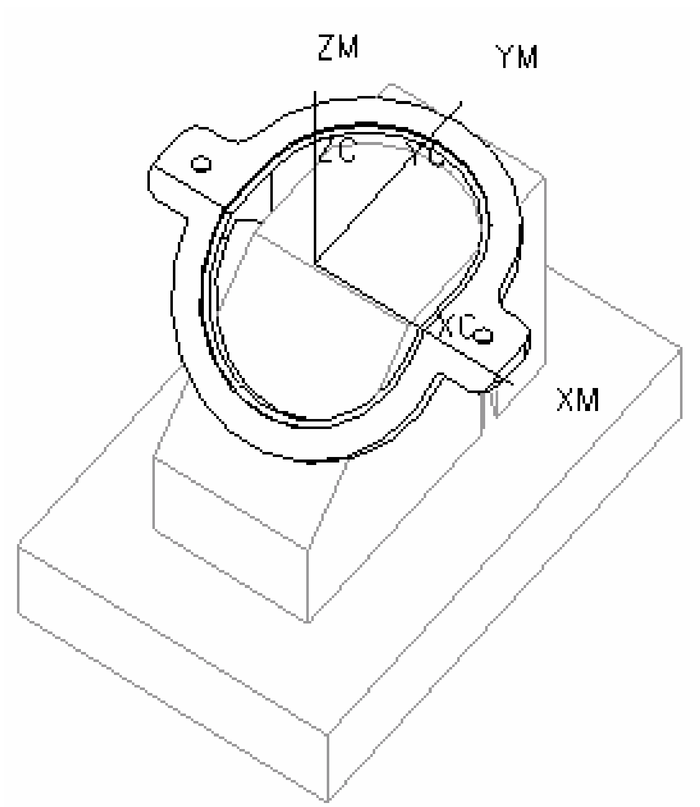
Activity 15–1: Profile Cut Pattern

In this activity, you will create a tool path using the Profile cut pattern. You will machine the back side of the casting and the slot around the inner lip.

Step 1 Open the part file, rename and enter the Manufacturing application.

❑ Open part file **mmp_lens_cover_mfg_1.prt**.

❑ Rename the part *****_lens_cover_mfg_1.prt** using the **File→Save As** option on the menu bar.

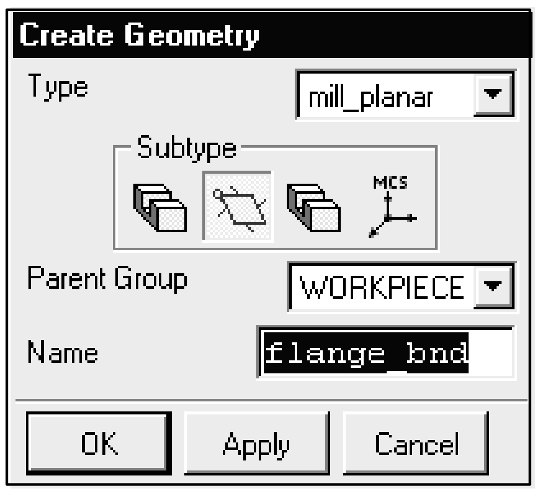


The holding fixture is light gray.

❑ Choose **Application→Manufacturing**.

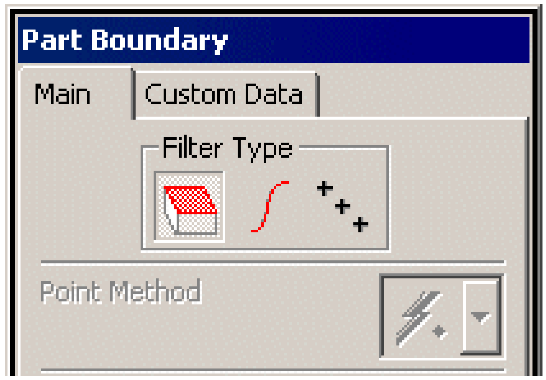
Step 2 Create an Open Boundary.

- ❑ Change the Operation Navigator to the **Geometry View.**
- ❑ Choose the **Create Geometry** icon.
- ❑ If necessary, set the Type to **mill_planar.**
- ❑ Choose **MILL_BND** as the Subtype.
- ❑ Choose **WORKPIECE** as the Parent Group.
- ❑ In the Name field, enter **flange_bnd**.

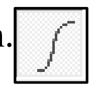


- ❑ Choose **OK**.
- ❑ In the MILL_BND dialog, choose the **Part** icon.
- ❑ Choose the **Select** button.

The Part Boundary dialog is displayed.

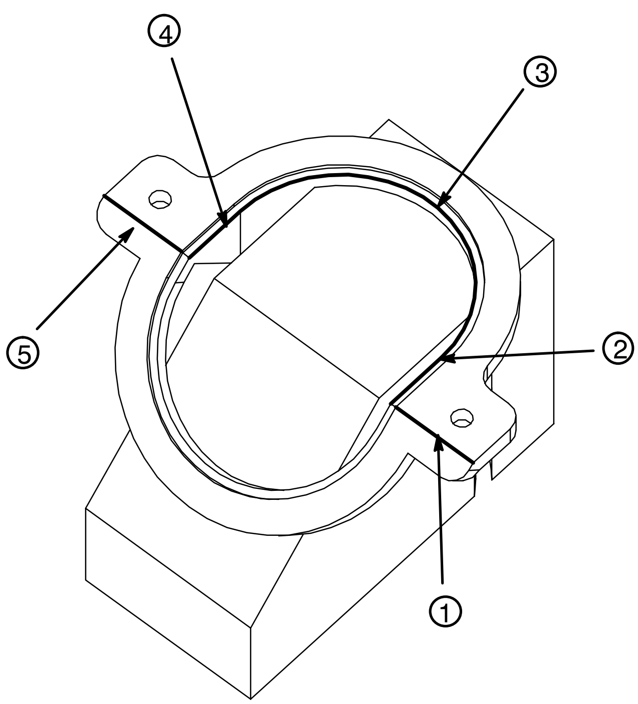


Note the filter setting at the top of the dialog. The Face Mode is inappropriate for the boundary, since Face Mode creates closed boundaries. For Profile cut operations, an open boundary is desired.

- ❑ Choose the **Curve Boundary** icon. 

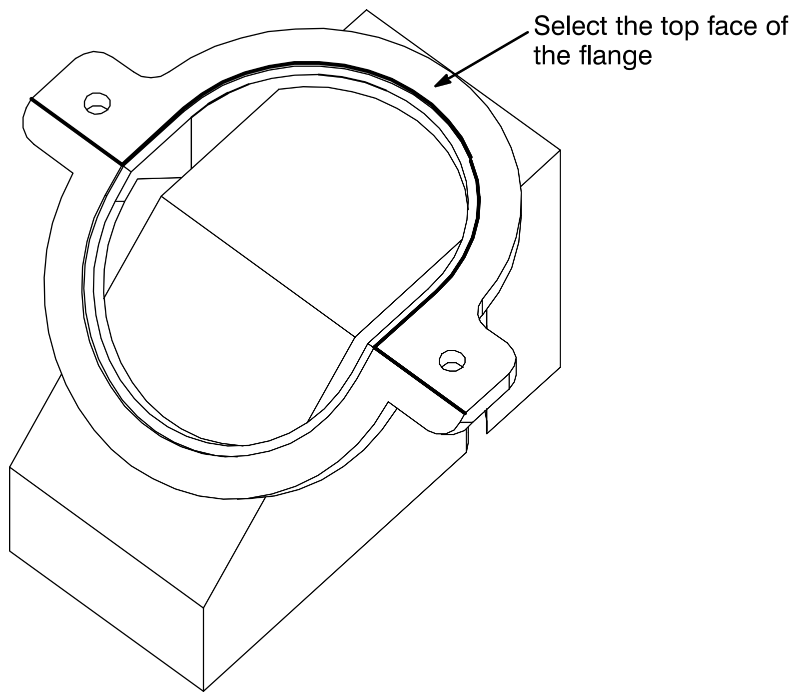
- ❑ Set the Type to **Open.**

- ❑ Select the edges in the order shown.



- ❑ Choose **OK**.

- ❑ On the MILL_BND dialog, choose the **Floor** icon.
- ❑ Choose the **Select** button.
- ❑ Change the Filter to **Face**.
- ❑ Select the top face of the flange.

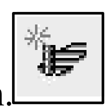
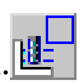


- ❑ Choose **OK**.

You have just created an Open Boundary and the Floor Plane.

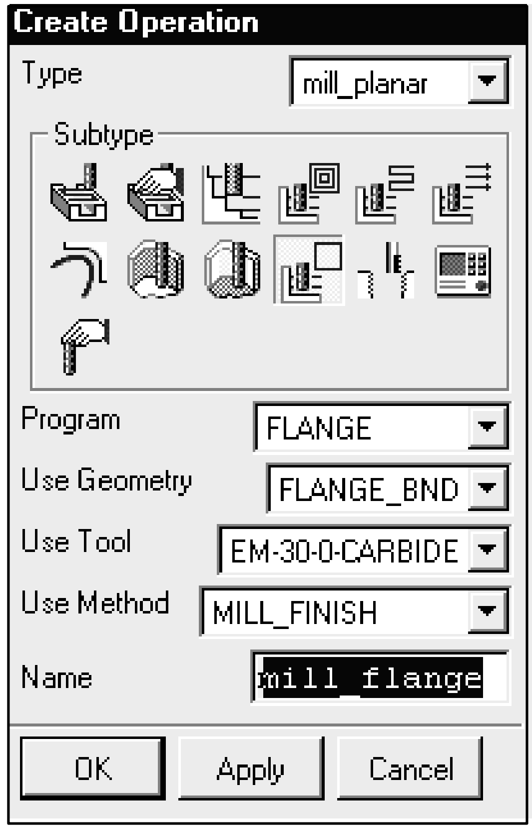
- ❑ Choose **OK** in the MILL_BND dialog.

Step 3 Creating a Planar Profile operation.

- ❑ Choose the **Create Operation** icon. 
- ❑ Choose the **PLANAR_PROFILE** icon. 

15

❑ Set the Parent Groups as shown:



❑ Name the operation **mill_flange**.

❑ Choose **OK**.

The Planar Profile dialog is displayed.

Step 4 Changing the Display Options.

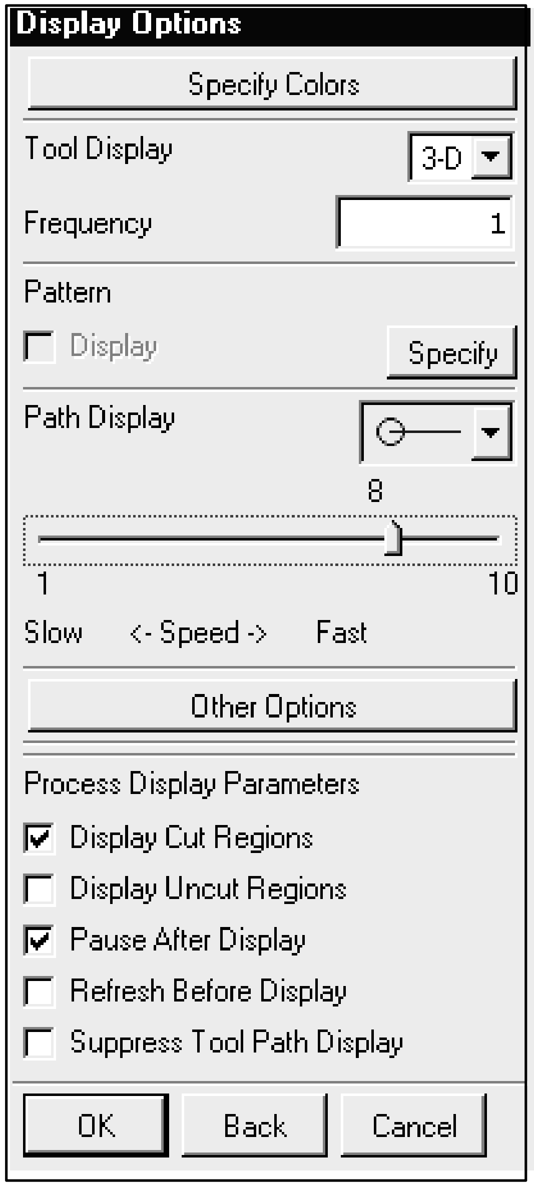
As a visual aid, you will slow down the speed of the tool display and show it in 3-D.

❑ Choose the **Edit Display** icon.

❑ Change the Tool Display to **3-D**.

❑ Move the Path Display slider bar to **8**.

15

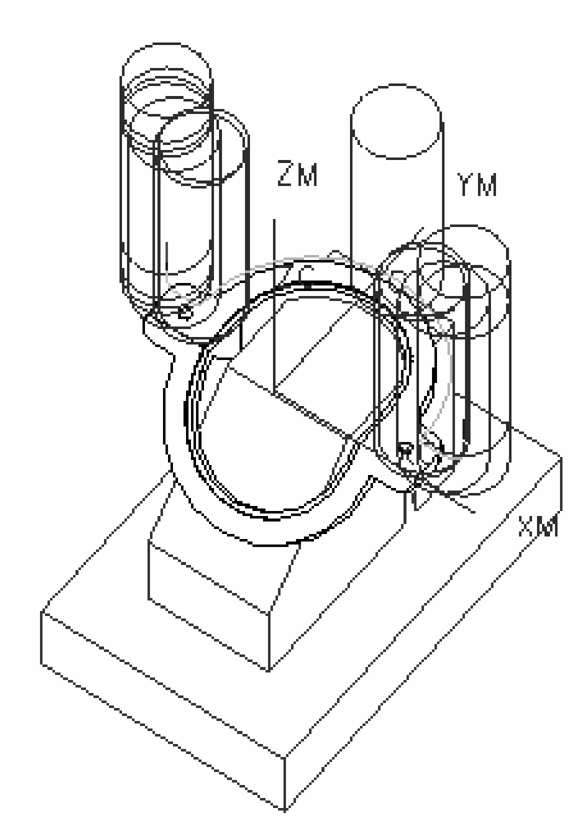


❑ Choose **OK.**

Step 5 Generate the tool path.

❑ Choose the **Generate** icon.

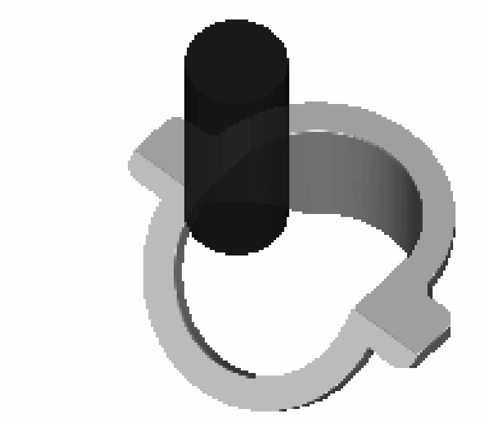
❑ Choose **OK** in the Display Parameters dialog.



Verify the tool path.

Step 6 Verify the Operation.

- ❑ Choose the **Verify** icon on the Planar Profile dialog.
- ❑ Choose the **Dynamic** tab.
- ❑ Choose the **Play Forward** icon.
- ❑ Slow down the **Animation Speed**.



The face of the flange is not completely machined. The boundary you created has a TANTO condition. The radius of the tool prevents it from machining the square corner.

❑ You will now modify the boundary.

Step 7 Modify the Boundary Offset.

❑ **Cancel** the Toolpath Visualization dialog.

❑ At the top of the PLANAR_PROFILE dialog, choose the **Geometry** radio button.

❑ Choose **Edit.**

The MILL_BND Geometry Group Parent that you created earlier is displayed.

❑ Choose the **Part** icon.

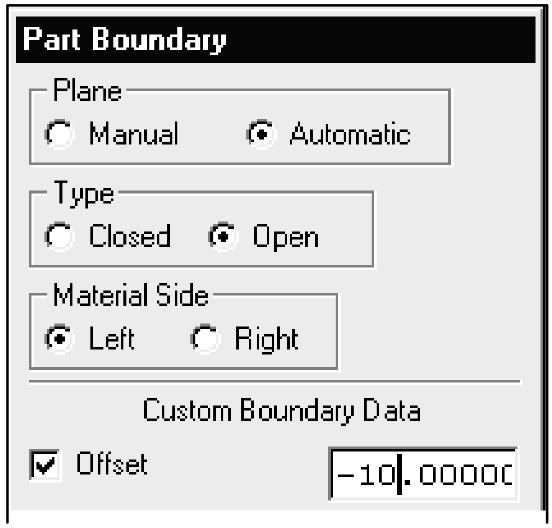
❑ Choose **Edit**.

There is only one boundary in this Geometry Parent Group, it is highlighted. All edits to the Part Boundary dialog will affect that boundary.

You will apply negative offset to force the tool to drive over the top of the boundary.

❑ Select the **Offset** toggle to turn it ON.

❑ Enter **-10** into the Offset value field.



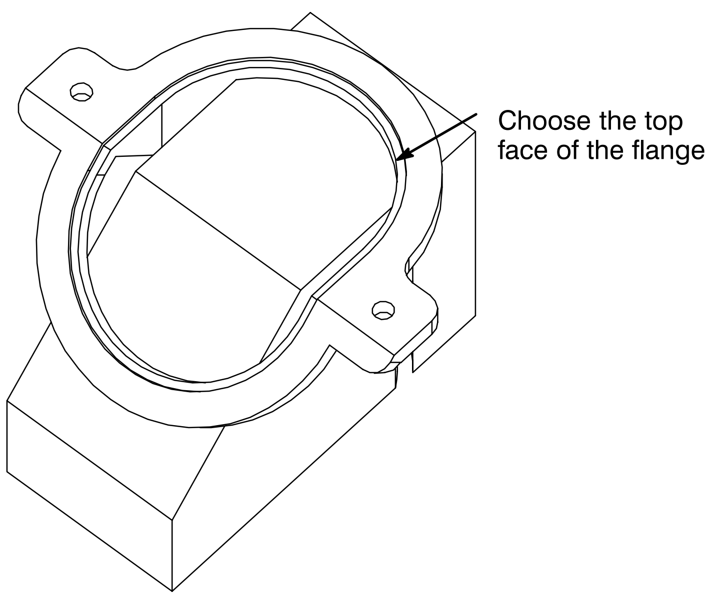
❑ Choose **OK** twice.

❑ Choose **Generate**.

The face is fully machined.

Step 8 Creating an additional Profile operation (optional).

If you wish, create a Planar Profile operation to machine the small step on the inside of the flange, as shown below:



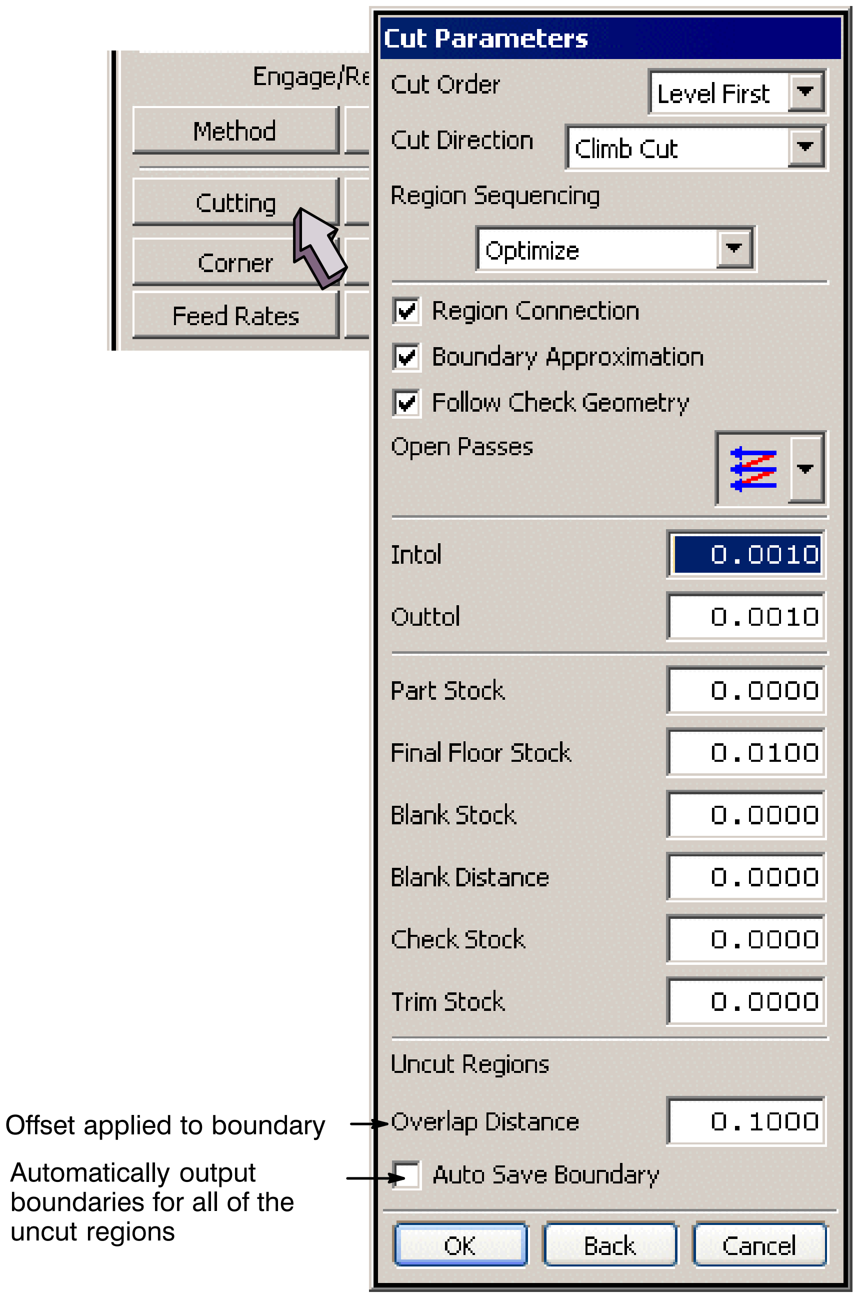
❑ **Save** and close the part file.

This completes this activity.

Uncut Regions

Many times when developing a tool path, you may have areas of uncut material. This may be caused by a number of different conditions such as a tool that is too big or stock left on the part walls or corners.

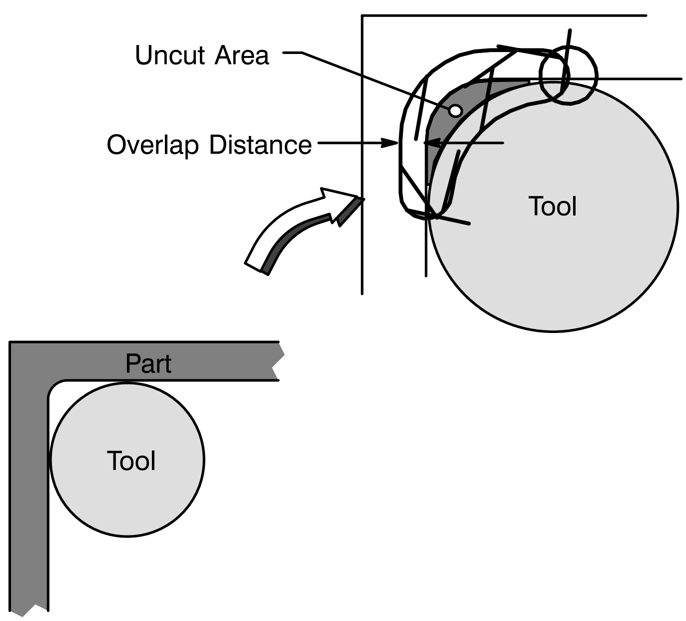
The Uncut Regions option can automatically output boundaries for these uncut regions. The Uncut Regions option is located under Cutting options, Cut Parameters.



The boundaries will be output to the level of the part boundary that results in the uncut region. If more than one region level exists, the uncut boundary will be output to the lowest level part boundary.

Overlap Distance

The Overlap Distance is an offset that is applied to boundaries. Offsets extend the boundary normal to the cut region, except where it will cause the tool to gouge the part.

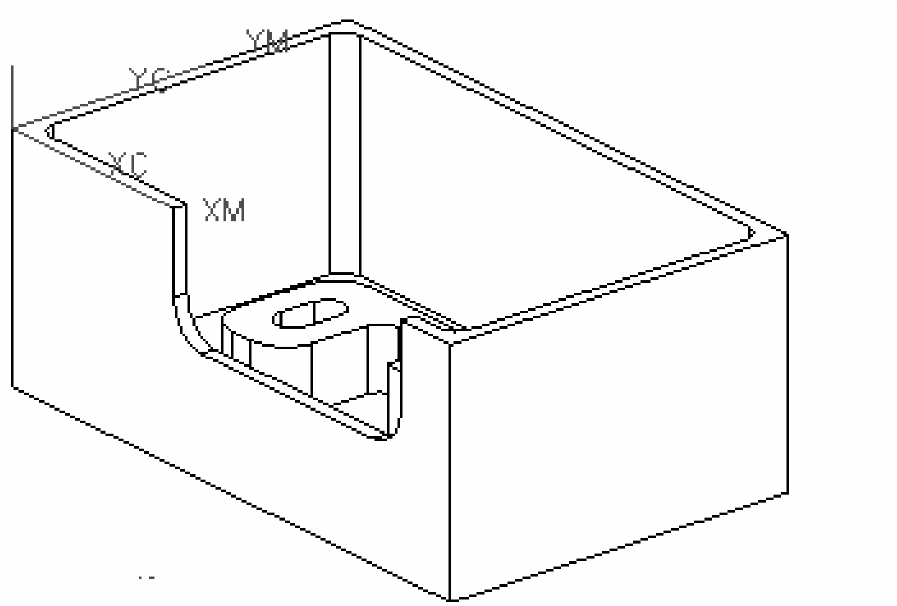


Activity 15–2: Creating Auto Save Boundaries

You will create a tool path using a 1.00 inch diameter tool. This tool is larger than the corner radii and will leave excess material uncut. You will save the uncut regions as Auto Save Boundaries that can be machined in an later operation.

Step 1 Open a part file, rename it, and enter the Manufacturing application.

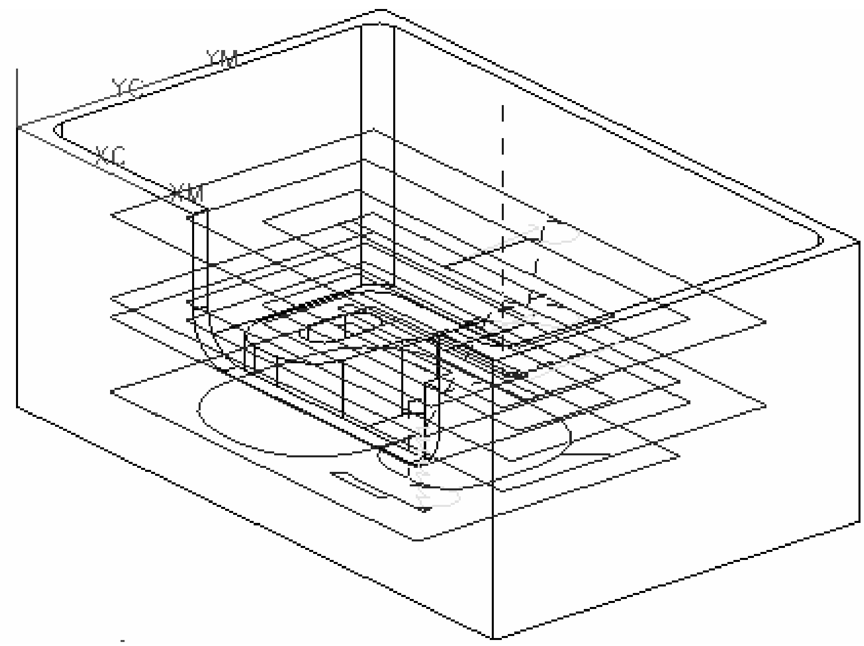
❑ Open the **mmp_tape_cover_1.prt** part file and save as ***_tape_cover_1.prt.



❑ Choose **Application** → **Manufacturing**.

Step 2 Replaying the tool path.

❑ In the Operation Navigator Geometry View, expand all Parent Groups and use MB3 to **Replay** the operation **UCUT.**

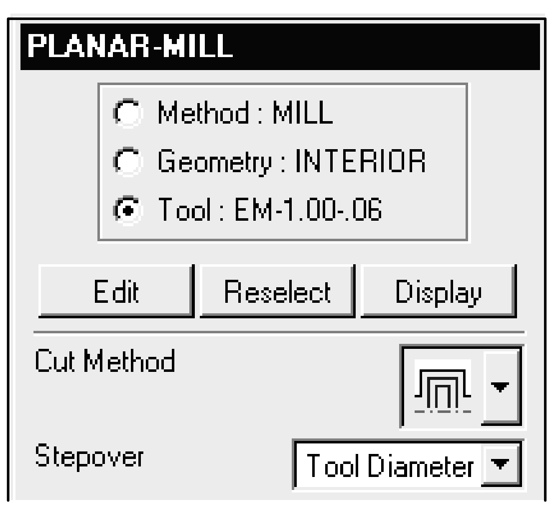


Note that this operation rough machines the interior of the part.

Step 3 Edit the Operation.

❑ Double click on the **UCUT** operation.

The Planar Milling dialog is displayed.

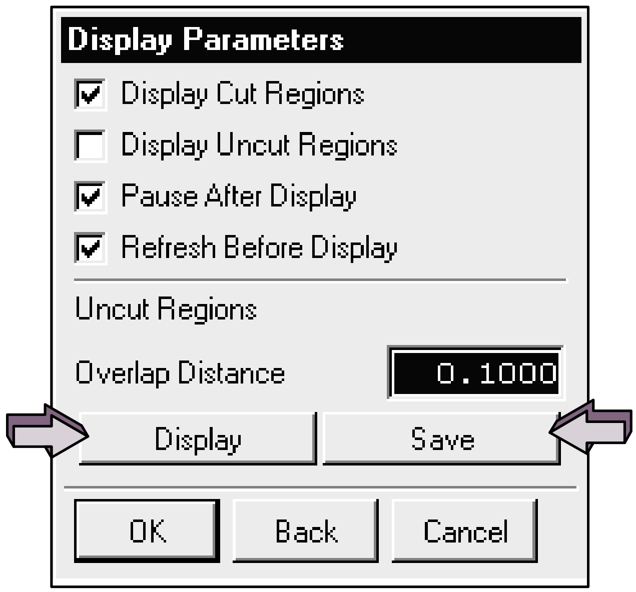


Step 4 Generate the tool path.

While the tool path is generating, you can display and save uncut regions as boundaries to be machined later.

❑ **Generate** the tool path.

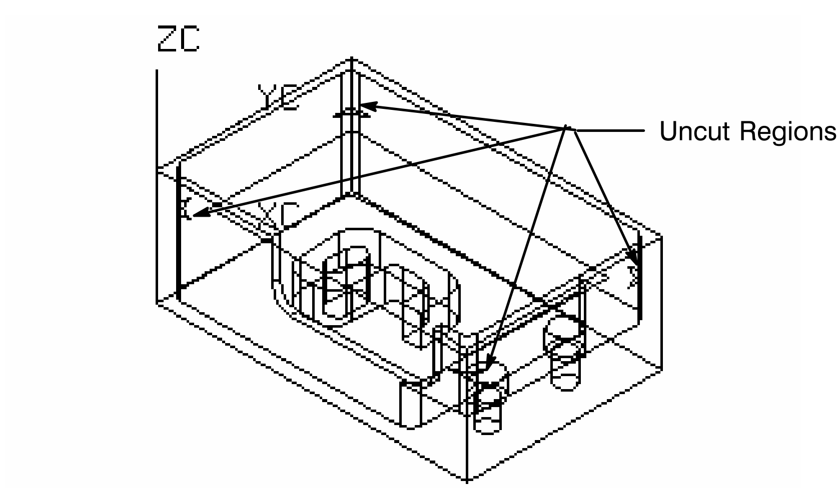
The first cut level is displayed along with the Display Parameters dialog.



At the bottom of the Display Parameters dialog you can save and display the uncut regions as boundaries.

You are going to display the regions.

❑ Choose **Display**.

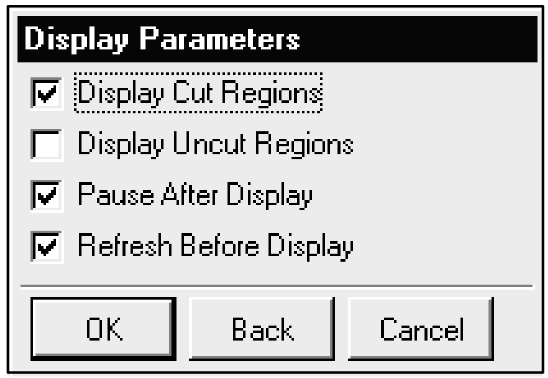


The boundaries, displayed in white, represent the material left in the corners by the tool.

You could select the Save option, but you will wait for the last cut level before saving the uncut regions.

❑ Choose **OK** to continue processing the tool path.

The Display Parameters dialog is displayed.



Notice that you do not have the option of saving the uncut regions at this time.

❑ Choose **OK,** again, to continue processing the tool path.

The second cut level is displayed along with the Display Parameters dialog. Once again, you have the option to Save the uncut regions, but you will not.

You will Cancel this operation and create the uncut boundaries using the Auto Save Boundary method.

❑ Choose **Cancel** from the bottom Display Parameters dialog.

A message window is displayed.

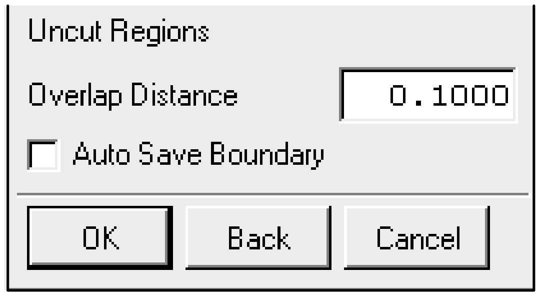
❑ Choose **OK** to dismiss the message window.

Now you will use the Automatic method of saving the uncut regions.

Step 5 Using the Auto Save Boundary method.

❑ Choose **Cutting** from the Machining Parameters area.

The Cut Parameters dialog is displayed. Notice the Uncut Regions options available. This is where you automatically save the boundaries.

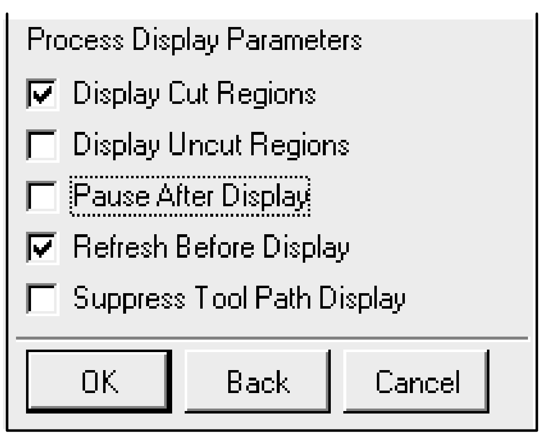


- ❑ Check **Auto Save Boundary.**
- ❑ Choose **OK**.

Step 6 Turn off the Pause After Display option.

You do not have to interact with the dialogs this time when you generate the tool path.

- ❑ Under the Tool Path label, choose the **Edit Display** button.
- ❑ Turn **OFF** the **Pause After Display.**
- ❑ If necessary, turn **ON** the **Refresh Before Display.**



- ❑ **OK** the settings and dismiss the dialog.

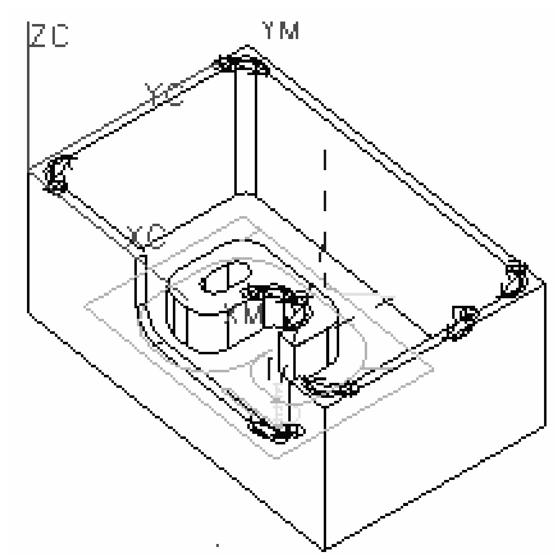
You are ready to generate the tool path.

Step 7 Generating the tool path.

While the tool path is generating, you will also be displaying and saving the uncut region as boundaries.

❑ **Generate** the tool path**.**

The boundaries are created at each cut level as the tool path is generated.



❑ Choose **OK** to accept the tool path.

Do not save this part. You will use it in the next activity.

Using the Auto Save boundaries in an operation

The uncut region boundaries are saved on the work layer in the part file. Once you save the boundaries, you can use them in another operation. You can try it for yourself in the next activity.

Activity 15—3: Create an Operation Using Uncut Regions

You will create a tool path using the boundaries that were created from the previous operation. You will copy the previous operation and make your boundary and tool changes to that copy.

Step 1 Open the part file.

❑ Continue to use the same part ***__tape_cover_1.prt.**

Step 2 Copy the operation.

❑ Change to the **Geometry View** of the Operation Navigator.

❑ In the Operation Navigator, expand the Geometry Parents as necessary until the operation named **UCUT** is displayed.

❑ Highlight the operation **UCUT**, using **MB3**, select copy**.**

❑ Using **MB3**, select **Paste**.

The operation is pasted under the UCUT name.

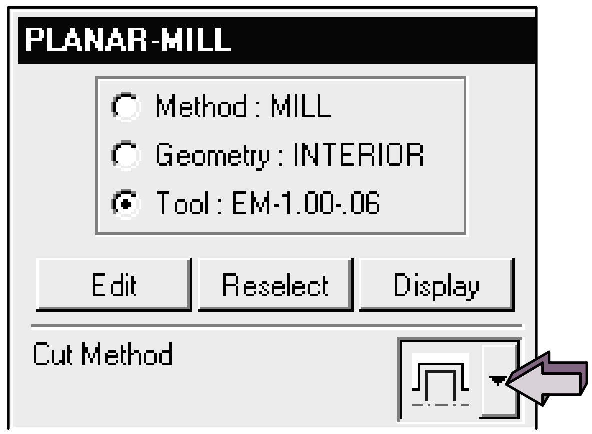
❑ Double click on the copied operation.

The Planar Mill dialog is displayed.

15

Step 3 Changing the Cut Method

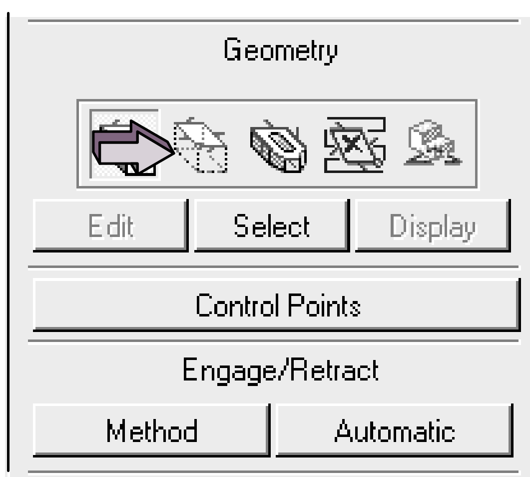
❑ Change the Cut Method to **Profile**.



Step 4 Creating new boundaries.

You are going to define the new uncut region boundaries as geometry.

❑ As shown, under the **Geometry** label, choose **Blank.**



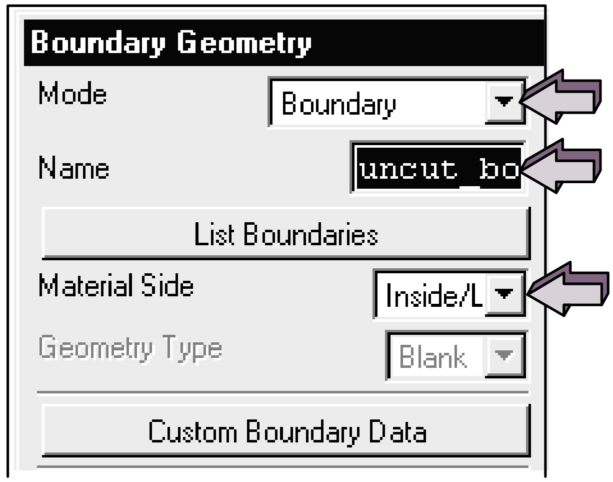
❑ Choose **Select.**

The Boundary Geometry dialog is displayed.

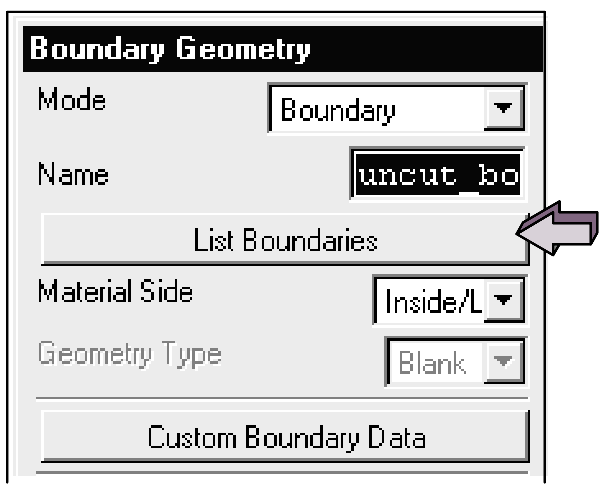
❑ Choose **Boundary** for the Mode.

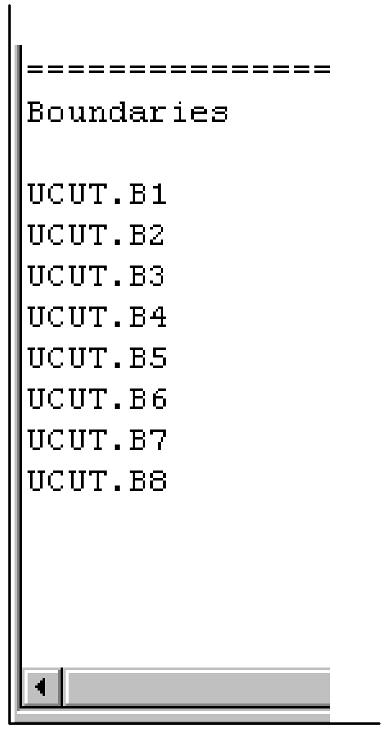
❑ In the Name field, enter **uncut_boundaries**.

❑ The Material Side should be **Inside/Left.**



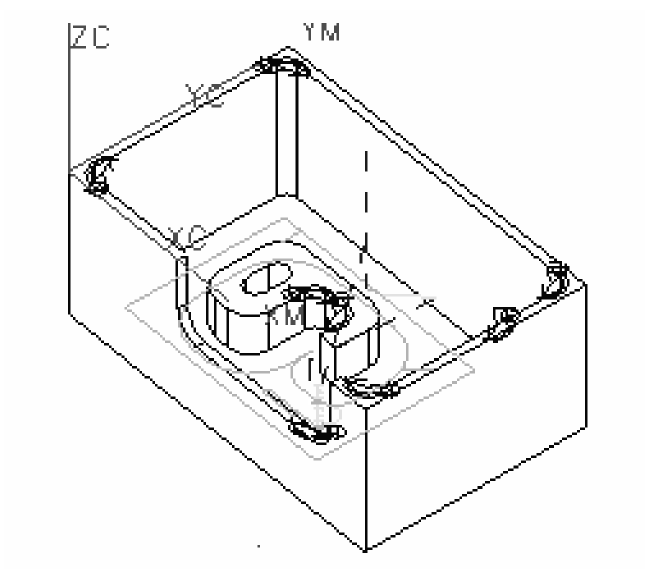
❑ Choose **List Boundaries**





Note that there are eight boundaries listed.

- ❑ In the graphics window, choose the eight boundaries.



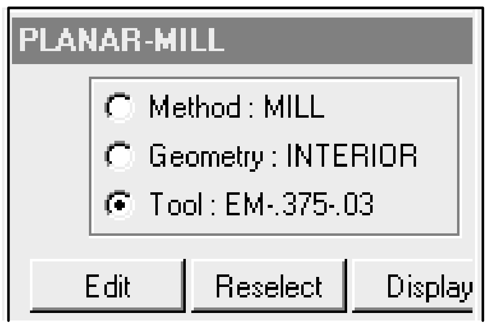
- ❑ Choose **OK**.

The Planar Mill dialog is displayed.

Step 5 Using a smaller tool for a cleanup operation.

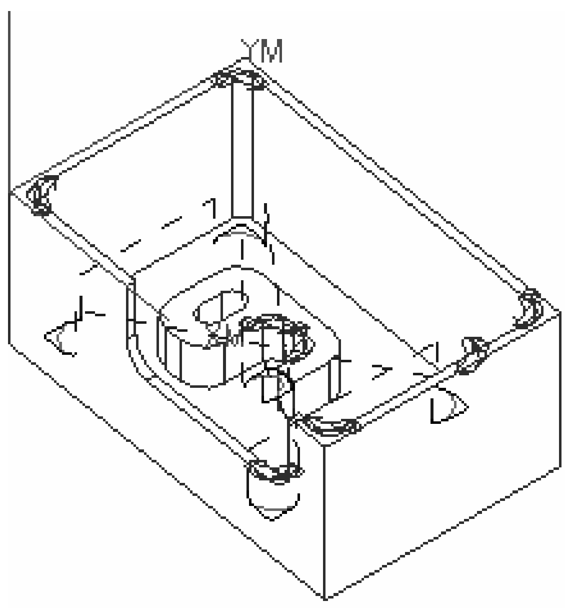
- ❑ On the Planar Mill dialog, choose **Tool**.
- ❑ Choose **Reselect.**

  The Reselect Tool dialog is displayed.
- ❑ Click on the down arrow and choose **EM–.375–.03.**
- ❑ Choose **OK**.



Step 6 Generating the tool path.

- ❑ **Generate** the tool path**.**



The tool cuts all of the boundaries at each level then moves down to the next level to cut all of the boundaries at that level. This continues until all boundaries are cut to the Floor Plane. You could specify that the tool cut one boundary to the finish depth before moving to the next boundary.

Step 7 Save and close your part file.

- ❑ Choose **OK** to accept the operation.
- ❑ Choose **File→Save**.

This completes the activity and the lesson.

SUMMARY

The robustness and flexibility of the Planar Milling module allows the automatic creation of boundaries in uncut areas of material for cleanup in a subsequent operation.

In this lesson you:

- Created a Profile operation
- Created a Planar Milling operation and saved the uncut material as boundaries
- Copied a previously created operation containing uncut material as boundaries, used the uncut boundaries as blank boundaries and removed the uncut material

(This Page Intentionally Left Blank)

Z-Level Milling

Lesson 16

PURPOSE

This lesson is an introduction to the Z-Level operation type, which is useful when profiling steep areas. You can also isolate specific areas that you want to cut or avoid cutting within a Z-Level operation.

OBJECTIVES

Upon completion of this lesson, you will be able to:

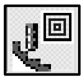
- Understand the uses of Z-Level Milling
- Create milling operations using the Z-Level operation type

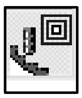
This lesson contains the following activities:

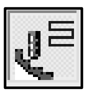
Z-Level Milling

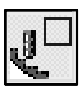
Z-Level Milling is designed to profile bodies or faces at multiple depths. It will cut steep areas (the steepness of the part at any given area is defined by the angle between the tool axis and the normal of the face) or the entire part.

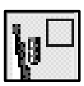
The following Z-Level operation types are available:

-  **ZLEVEL_FOLLOW_CAVITY –** uses the Follow Part Cut Method; ideal for “cavity” type parts

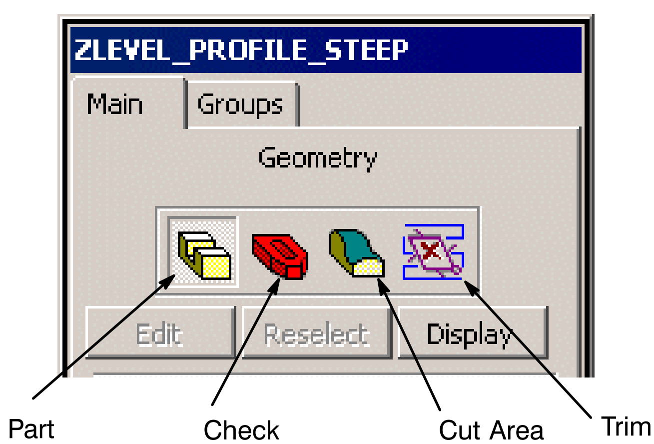
-  **ZLEVEL_FOLLOW_CORE –** uses the Follow Part Cut Method; ideal for “core” type parts

-  **ZLEVEL_ZIGZAG –** uses the ZIGZAG Cut Method

-  **ZLEVEL_PROFILE –** uses the Profile Cut Method without the Steep Angle being set

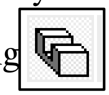
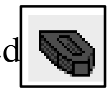
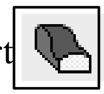
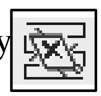
-  **ZLEVEL_PROFILE_STEEP –** uses the Profile Cut Method with the Steep Angle set to 65 degrees

Part geometry and Cut Area geometry can be specified to limit the area to be cut. If cut area geometry is not defined, then the entire part is used as the cut area.



Many of the option settings found in Z-Level Milling are the same as in Cavity Milling. A description of some of these options are as follows:

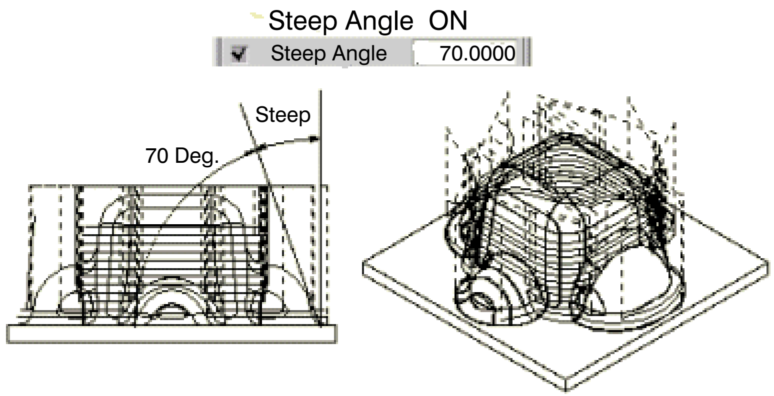
Geometry

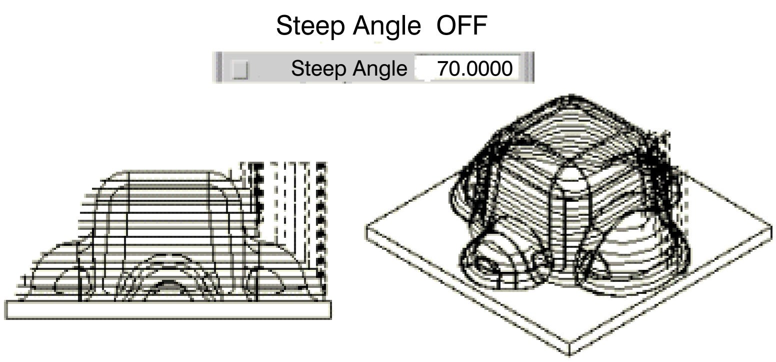
- **Part** geometry consists of bodies and faces which represents the Part after cutting 
- **Check** geometry consists of bodies and faces which represent clamps or obstructions that are *not* to be machined 
- **Cut Area** geometry represents the areas on the Part to be machined; it can be some or all of the part 
- **Trim** geometry consists of closed boundaries which indicate where material will be left or removed; all Trim boundaries have tool positions ON only 

During tool path generation, the geometry is traced, steep areas and trace shapes are determined, cut areas are identified and a tool path is generated for all cut depths specified.

Steep Angle

The steepness of the part at any given area is defined by the angle between the tool axis and the normal of the face. The steep area is the area where the steepness of the part is greater than the specified Steep Angle. When the Steep Angle is toggled ON, areas of the part with a steepness greater than or equal to the specified Steep Angle are cut. When the Steep Angle is toggled OFF, the part as defined by the part geometry and any limiting cut area geometry is cut.



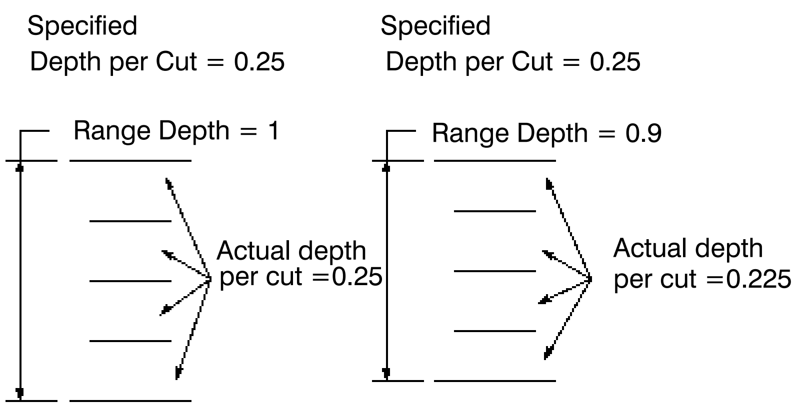


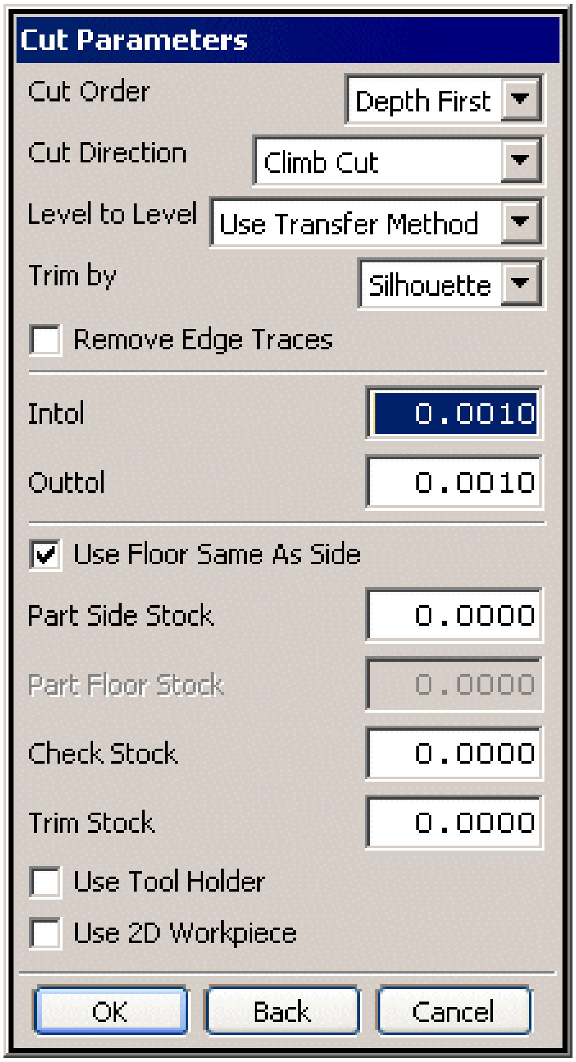
Minimum Cut Length

Minimum Cut Length enables the elimination of short tool path segments that may occur in isolated areas of the part. Moves shorter than this value are not generated.

Depth Per Cut

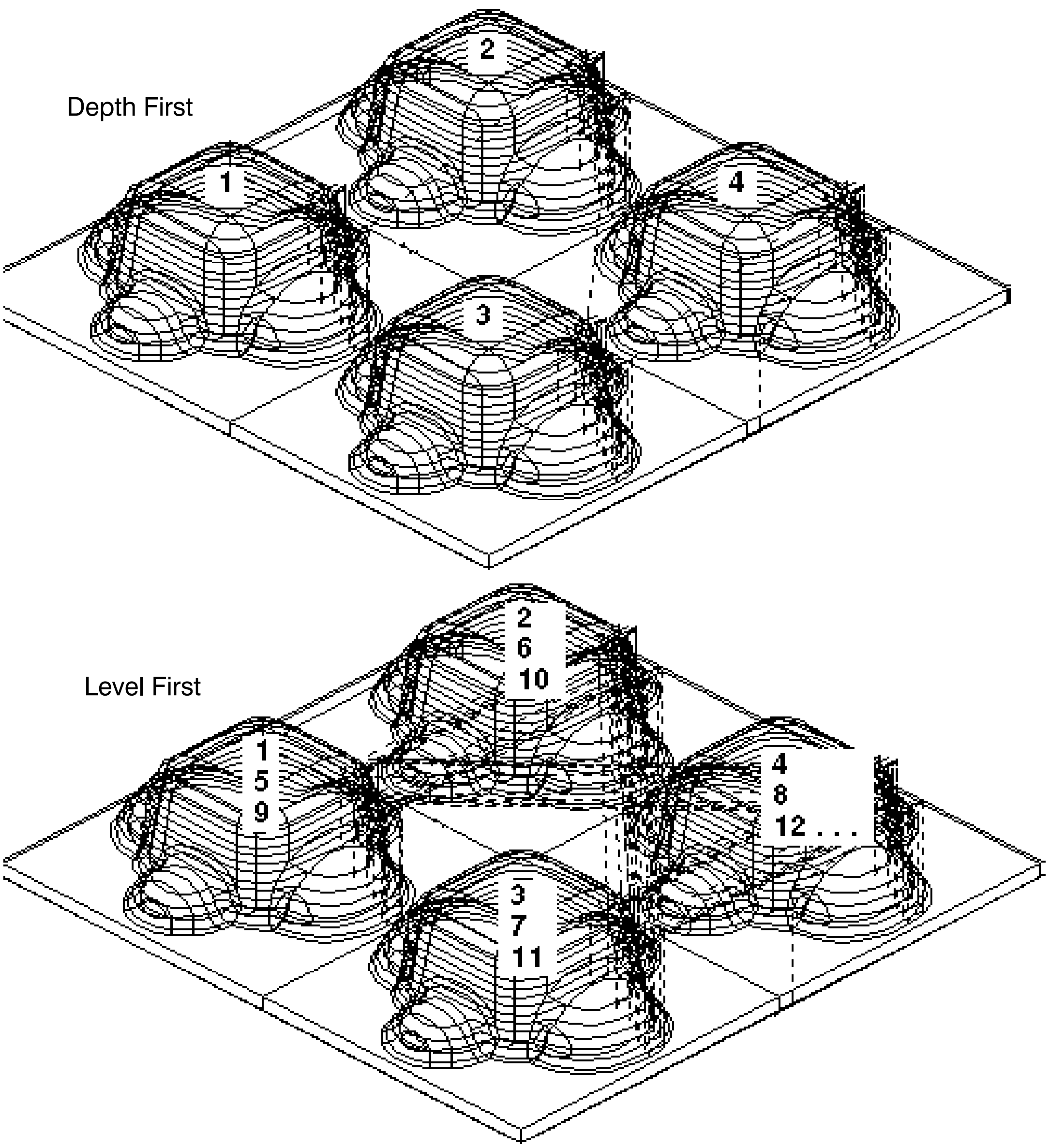
Depth Per Cut allows the specification of the maximum depth per cut in a range. Cut depths are calculated that are equal and do not exceed the specified Depth Per Cut value.





Cut Order

Z-Level Milling determines cut traces by shape. Shapes can be profiled by Depth First in which case each shape is completely profiled before beginning to profile the next shape. Shapes can also be profiled by Level First in which case all shapes are profiled at a particular level before cutting each shape at the next level.



Control Geometry

Control Geometry allows the specification of Control Points to determine where the tool engages the part and the floor plane.

Trim by

Trim by is used to prevent the tool from rolling around corners.

The Silhouette option uses the outline of the part geometry, as viewed down the tool axis, to generate a trace. The tool is positioned along the silhouette of the part geometry. The trace is then offset to the outside by the tool radius distance. The silhouette can be thought of as the shadow of the part projected along the tool axis.

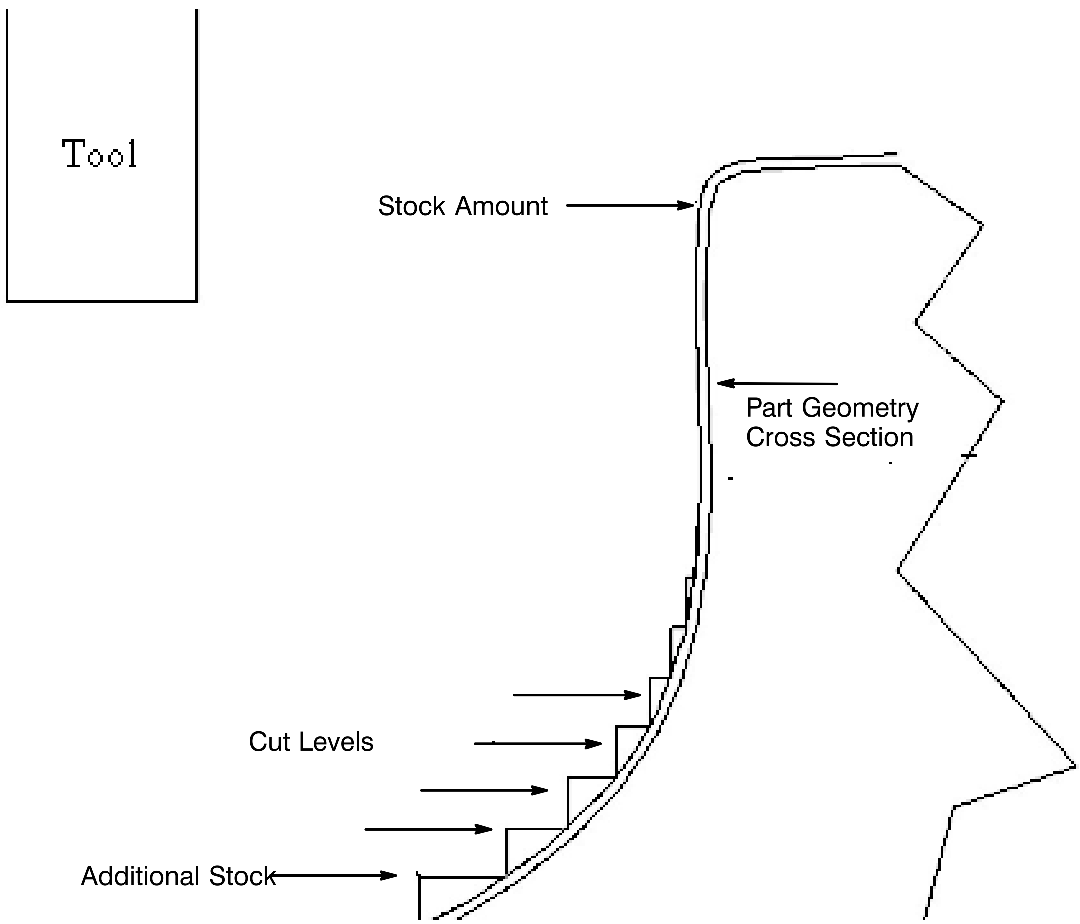
When using Trim by Silhouette, the traces at the bottom of the part geometry are used as trim shapes. These shapes will be projected along the tool axis to each cut level and will be used in the process of generating the machinable regions as trim shapes.

Remove Edge Traces

Edge tracing (edge roll) is usually an undesirable condition that can occur when the Drive Path extends beyond the edge of the part geometry. The tool rolls over the edge of the part geometry potentially gouging the part. The Remove Edge Traces option allows the control of whether or not edge tracing occurs.

Machining Steep Geometry

As shown in the diagram below, the closer the geometry approaches horizontal, the more stock that is left.

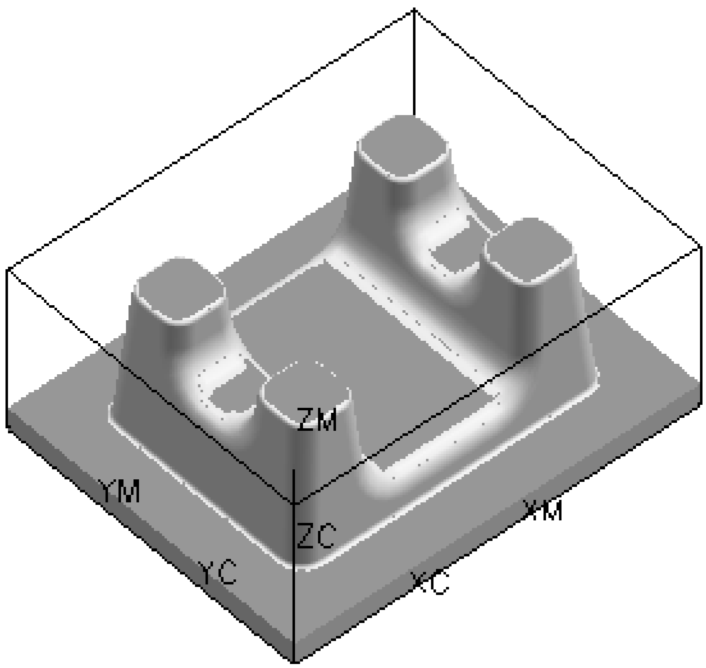


Activity 16–1: Z-Level Milling

In this activity, you will generate tool paths using Z-Level Milling. Z-Level is designed to profile an entire part or steep areas that were previously left by the Area Milling Drive Method.

Step 1 Opening the part file and enter the Manufacturing application.

❑ Open the part **mmp_base_mfg_3.prt.**



❑ Enter the **Manufacturing** application.

The Operation Navigator is displayed.

❑ Change the view of the Operation Navigator to the **Geometry View**.

The MCS_MILL Parent Group is displayed in the Operation Navigator.

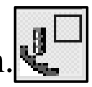
❑ Expand the **MCS_MILL** and **WORKPIECE** Geometry Parent Groups.

The ROUGHING_1 operation is listed in the Operation Navigator.

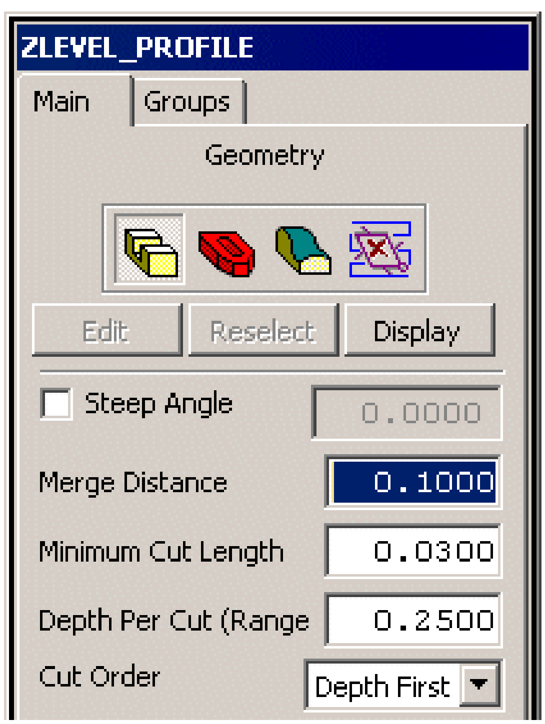
16

Step 2 Creating a Z-Level operation.

- ❑ Choose the **Create Operation** icon on the Manufacturing Create toolbar.

- ❑ Make sure the Type is set to **mill_contour.**
- ❑ Choose the **ZLEVEL_PROFILE** icon. 
- ❑ Set the Program to **BASE_MALE_DIE**.
- ❑ Set the Use Geometry to **WORKPIECE**.
- ❑ Set the Use Tool to **EM_1.25_.25.**
- ❑ Set the Use Method to **MILL_FINISH**.
- ❑ Name the operation **zlevel_finish**.
- ❑ Choose **OK.**

The Z–LEVEL_PROFILE dialog is displayed.



Step 3 Changing the Depth of Cut.

You will change the depth of cut.

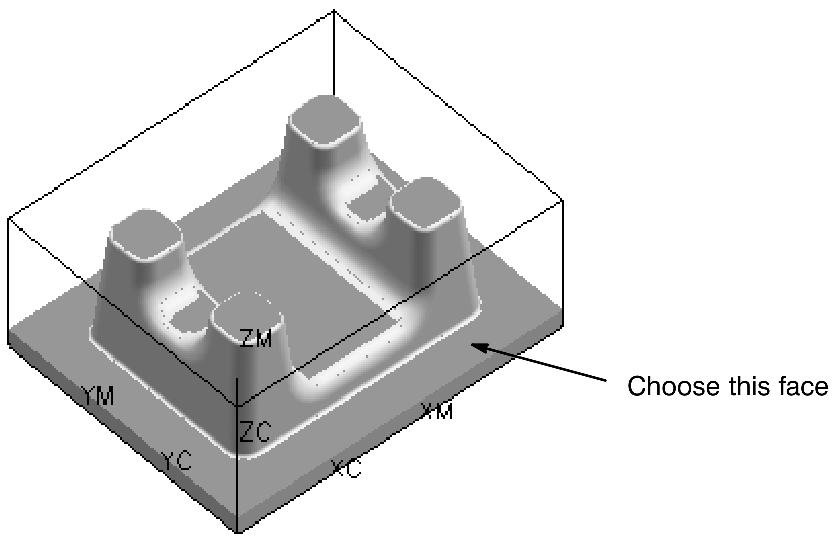
❑ Next to the Depth Per Cut (Range 1) label, enter **0.100**.

Change the cut levels. You will stop cutting material at the top of the bottom face. The default is the bottom face.

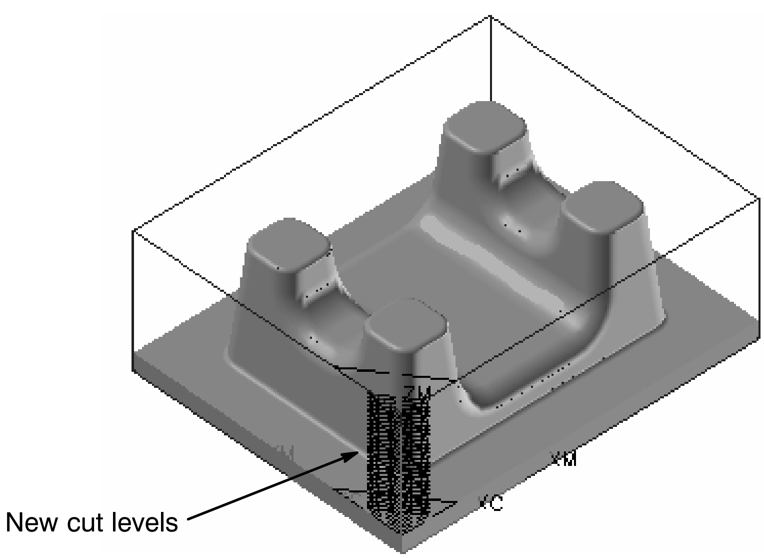
❑ Choose the **Cut Levels** button.

The Cut Levels dialog is displayed.

❑ Choose the top of the bottom face as shown below.



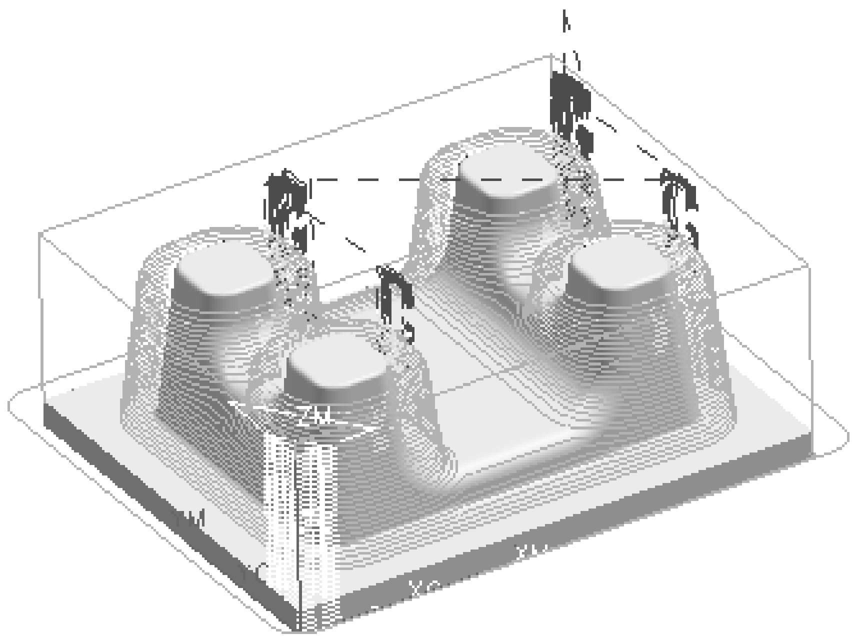
The new cut levels are displayed.



❑ Choose **OK.**

Step 4 Generate the tool path.

- ❑ Choose the **Generate** icon and generate the tool path.
- ❑ Choose **OK in** the Display Parameters dialog to continue generating the tool path.



- ❑ Choose **OK** to accept the operation.

Step 5 Verify the Program that you have created.

- ❑ Use **Toolpath Verification** to examine the tool path results.
- ❑ Close the part file.

This completes the activity and the lesson.

SUMMARY

This lesson was an introduction to Z-Level Milling, which is used when profiling steep areas (the steepness of the part at any given area is defined by the angle between the tool axis and the normal of the face). This operation type is useful in minimizing the amount of scallop or cusps that remains on the part.

In this lesson you:

- Created an operation using Z-Level Profile operation types

(This Page Intentionally Left Blank)

MILL_AREA Geometry Parent Groups

Lesson 17

17

PURPOSE

This lesson introduces you to the MILL_AREA Geometry Parent Groups, which are useful in limiting the cut area.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Create and use MILL_AREA Geometry Parent Groups
- Create and modify Trim Boundaries
- Recognize the type of geometry MILL_AREA Parent Groups use

This lesson contains the following activities:

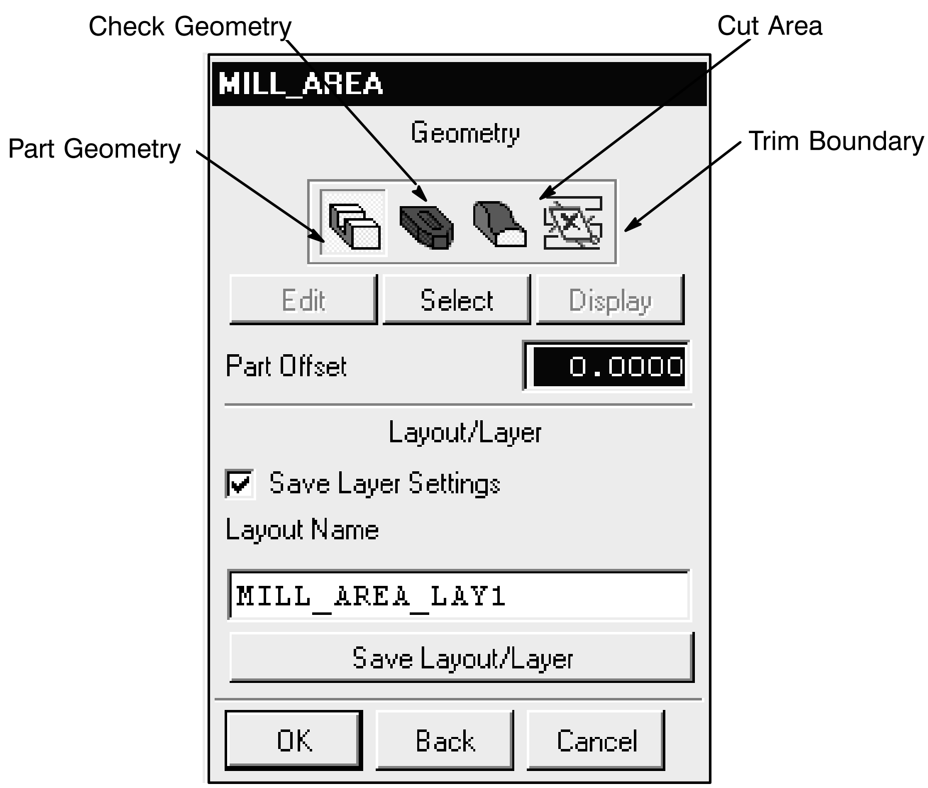
MILL_AREA Geometry Overview

Occasionally, when machining large or complex parts, it is desirable to limit the area that an operation machines. The MILL_AREA Geometry Parent Group is designed for that purpose.

The MILL_AREA Geometry Parent Group allows the user to select a small portion of a part to machine. This area is based on the faces of the part which you select. This group of faces to machine is called a **Cut Area**.

The area to machine can be further limited by use of a **Trim Boundary**.

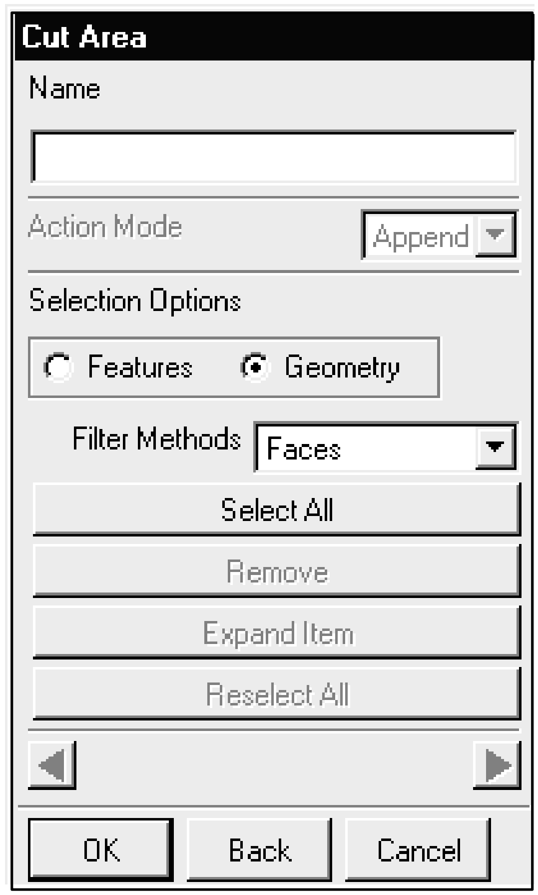
Below is the MILL_AREA Geometry Parent Group dialog.



- **Part Geometry –** is typically specified in the WORKPIECE Geometry Parent Group and represent the material to be cut
- **Check Geometry –** represents clamps, vises, locator pins, and other items that are not cut
- **Cut Area –** represents the specific geometry to be machined
- **Trim Boundary –** allows you to define trim boundaries that limit the cutting area

Cut Area

When choosing the Cut Area icon, the Cut Area dialog is displayed.



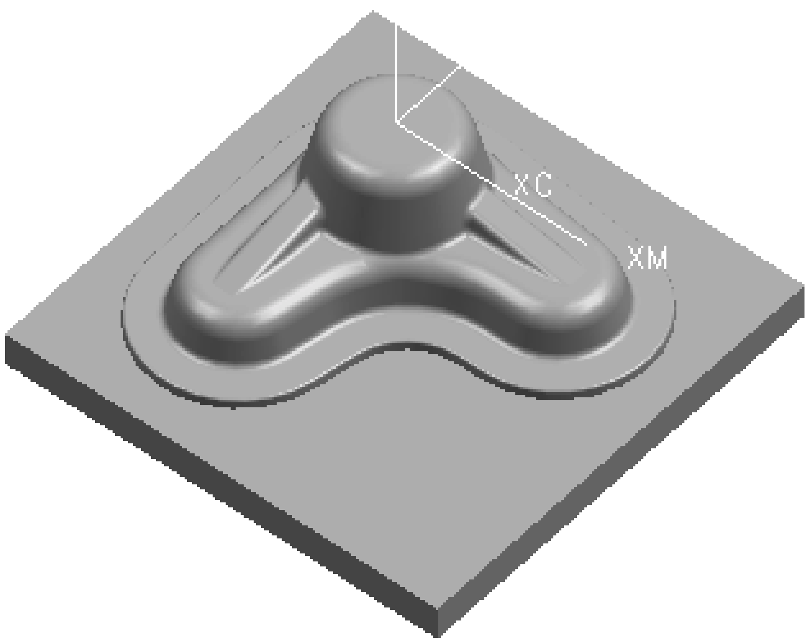
Only faces and sheet bodies can be selected for Cut Area geometry. The Features option allows surface regions (groups of faces or sheet bodies) for selection purposes.

Activity 17–1: MILL_AREA Geom. Parent Groups

This activity will demonstrate how to create and use a MILL_AREA geometry Parent Group in an operation. You will Replay and examine the results of an existing operation. You will then create a MILL_AREA geometry Parent Group consisting of faces and will modify the inheritance of the operation to use the MILL_AREA parent.

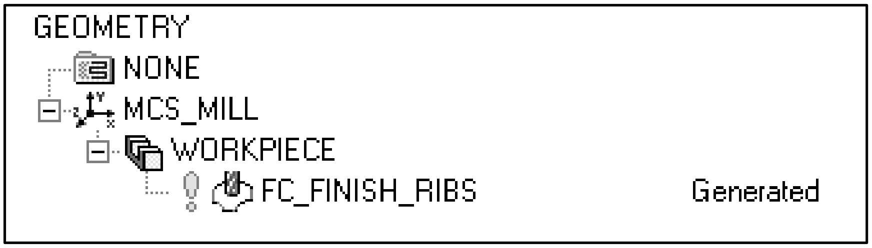
Step 1 Open the part file, rename it, and enter the Manufacturing application.

- ❑ Open part file **mmp_male_cover_mfg_2. prt**.



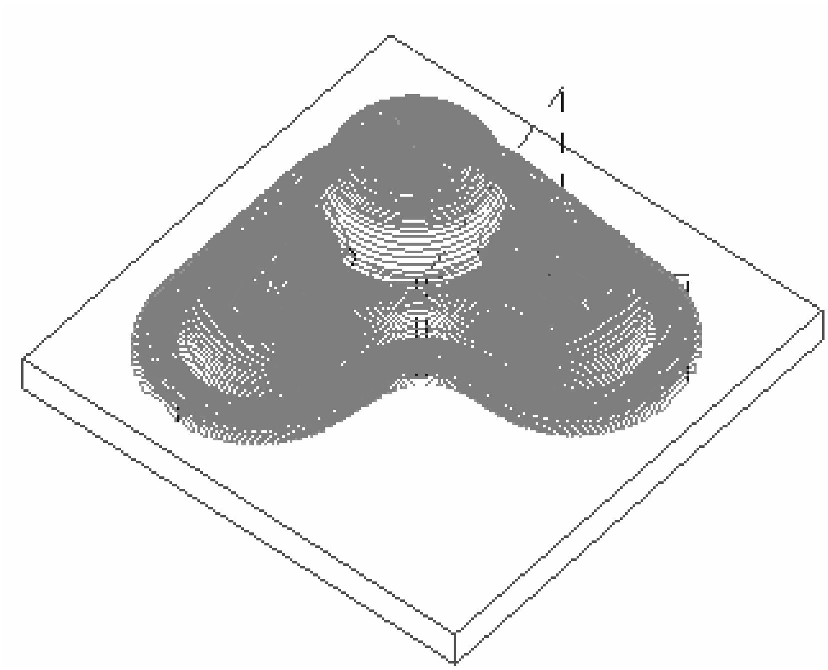
- ❑ Rename the part *****_male_cover_mfg_2.prt** using the **File→Save As** option on the menu bar.
- ❑ Choose **Application→Manufacturing**.
- ❑ Change the Operation Navigator to the **Geometry View**.

❑ Expand the MCS_MILL and WORKPIECE Geometry Parent Groups.



Step 2 Replaying the current operation.

❑ Highlight the FC_FINISH_RIBS operation, using **MB3** select, **Replay.**



This Fixed Contour operation machines the entire part. This is not the desired result.

In the next steps, you will create a MILL_AREA geometry Parent Group to limit the machining to just the two ribs protruding from the part.

❑ **Refresh** the graphics screen.

TIP There are at least three ways to refresh the screen:

1) MB3→Refresh
2) Press the F5 button
3) From the top menu bar choose View→Refresh.

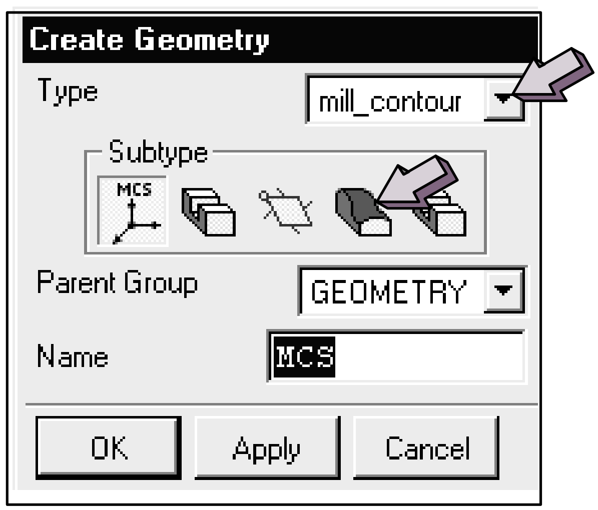
17

Step 3 Creating the MILL_AREA Geometry Parent Group.

❑ Choose the **Create Geometry** icon.

❑ If necessary, change the Type to **mill_contou**r.

❑ Choose the Subtype **MILL_AREA**.



❑ Change the Parent Group to **WORKPIECE**.

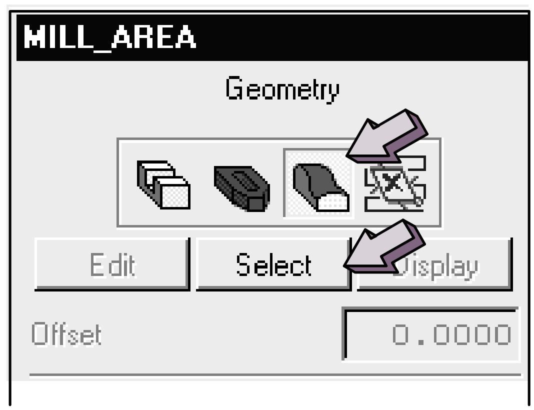
❑ In the Name field, enter **two_ribs**.

❑ Choose **OK.**

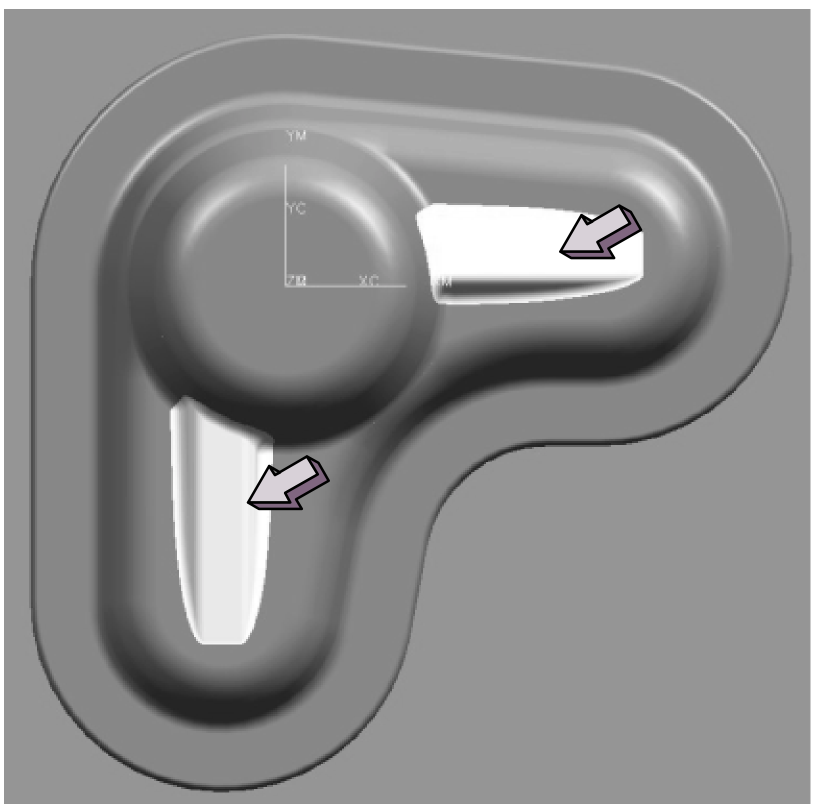
The MILL_AREA dialog is displayed.

Step 4 Defining the Cut Area geometry.

- ❑ Choose the **Cut Area** icon.
- ❑ Choose the **Select** button.



- ❑ Choose the faces of the ribs, as shown.



- ❑ When finished selecting the faces, choose **OK**.

17

❑ Choose **OK** again to accept the dialog.

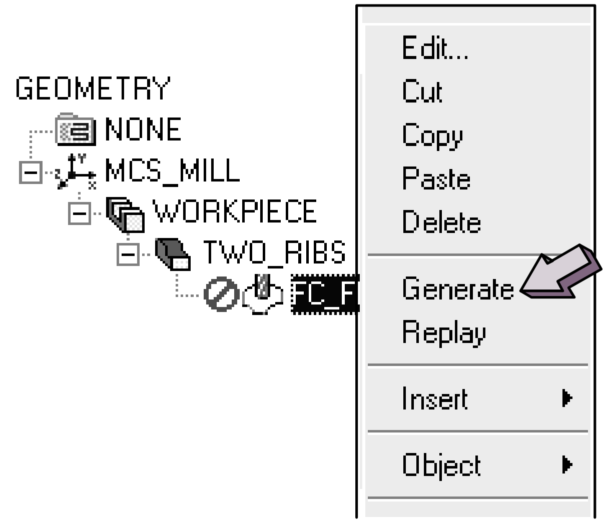
Step 5 Changing the inheritance of the operation.

You will move the **FC_FINISH_RIBS** operation, so that the operation will machine only the faces specified.

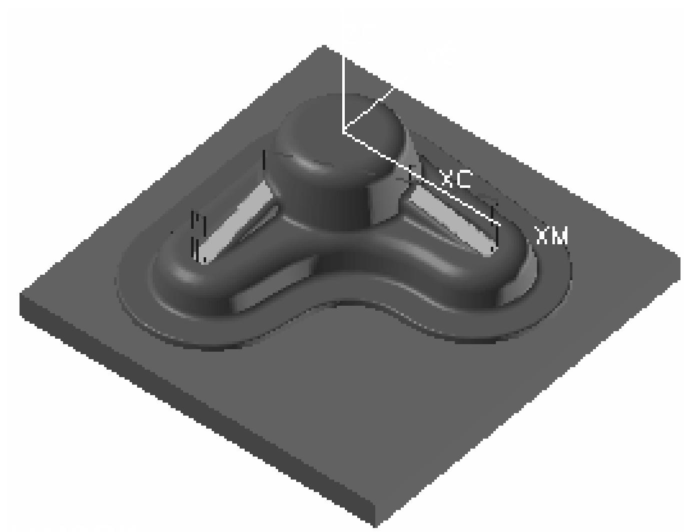
Currently, the Geometry View of the Operation Navigator looks like this:

❑ Using MB1, click and drag the **FC_FINISH_RIBS** operation so that it resides under the **TWO_RIBS** Parent Group, then release MB1

❑ Highlight the **FC_FINISH_RIBS** operation, using MB3, select **Generate** from the pop-up menu.



The tool path is generated and cuts the faces described in the MILL_AREA Parent Group.



❑ Choose **OK** to accept the tool path.

❑ **Save** the part file.

17

17

Trim Boundary

A Trim Boundary is the same as any other boundary except that any tool path that falls within the area described by the boundary will be trimmed away.

When you choose the Trim Boundary icon, the standard boundary dialog is displayed.

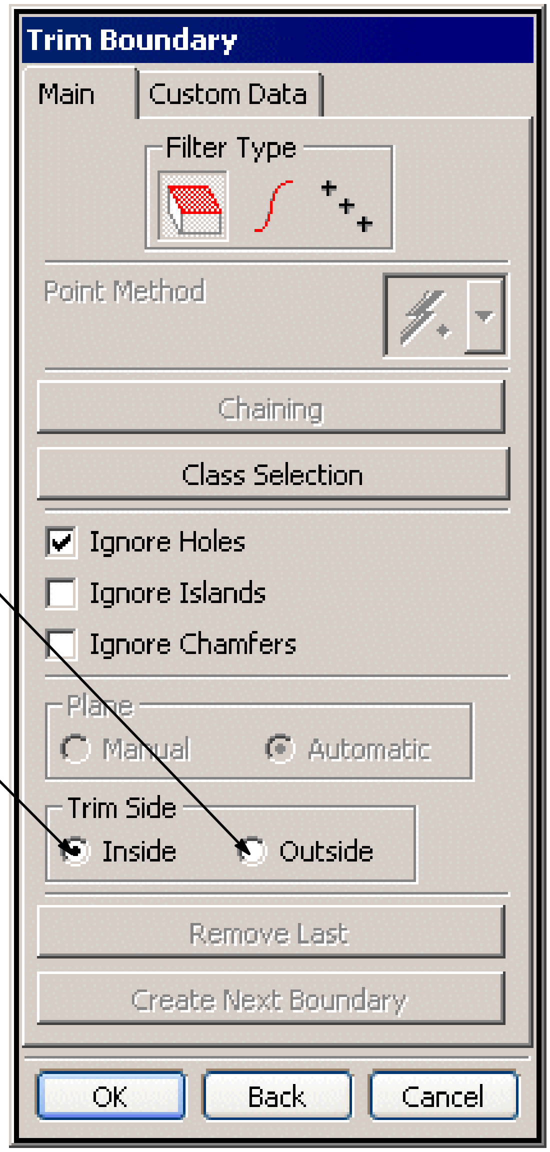
Trim Side:

Outside:

The tool path will only be created inside the boundary

Inside:

The tool path will only be created outside the boundary

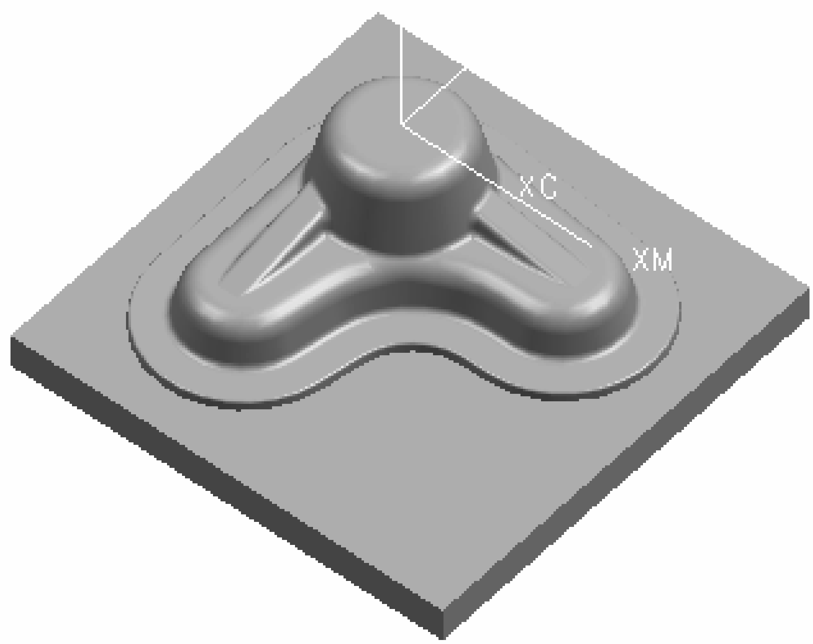


Activity 17–2: Using Trim Boundaries

In this activity, you will create a trim boundary inside of a MILL_AREA Parent Group and will then generate the corresponding operation.

Step 1 Continue using the part file.

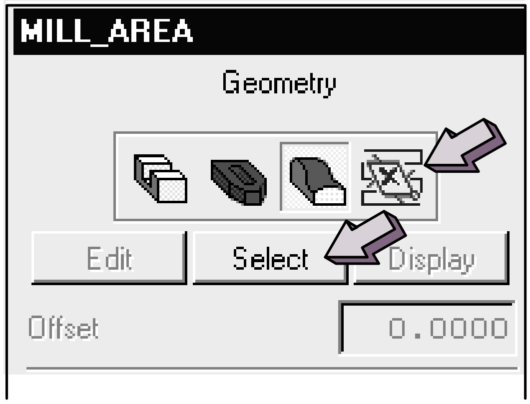
- ❑ Continue using the *****_male_cover_mfg_2.prt**.



Step 2 Creating a Trim Boundary.

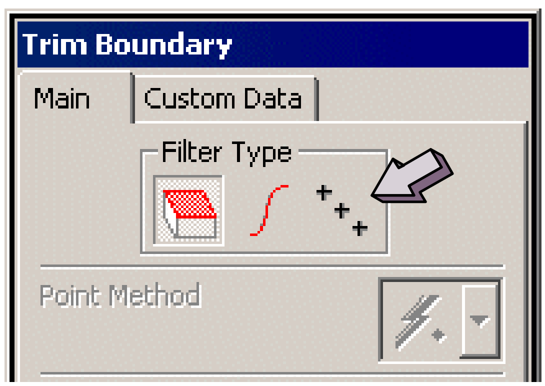
- ❑ Change the View to the **TOP** view.
- ❑ Change the Operation Navigator to the **Geometry View**.

  You will now edit the operation.
- ❑ Double-click on the **TWO_RIBS** operation.
- ❑ Choose the **TRIM** icon, and then choose **Select.**

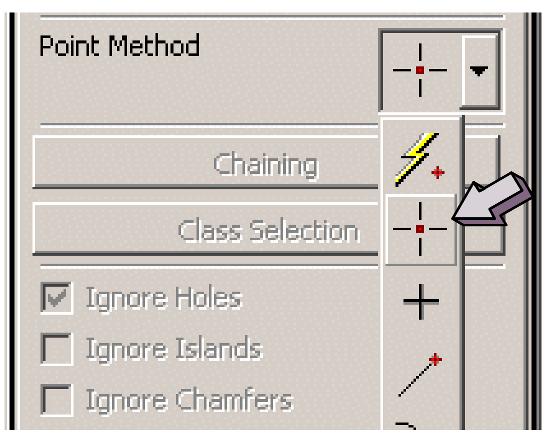


The boundary you will create will be a free-hand design using cursor location points.

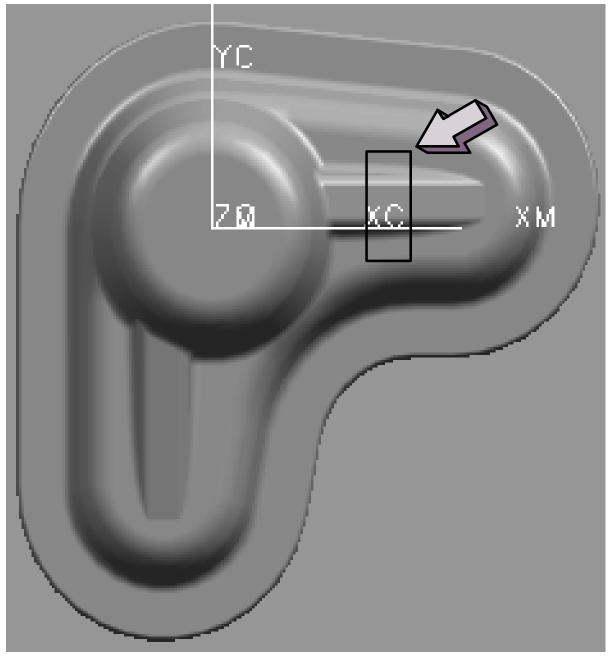
❑ Choose the **Point Boundary** icon.



❑ Change the Point Method to **Cursor Location.**



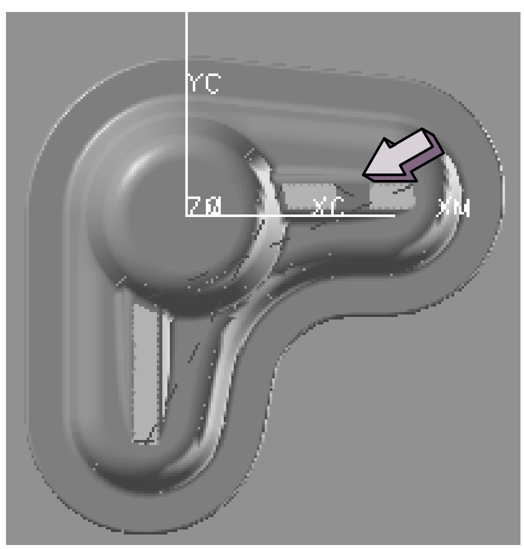
❑ Create a trim boundary similar to the one shown below.



❑ Choose **OK** to return to the main dialog.

Step 3 Re-generate the tool path.

❑ Generate the **FC_FINISH_RIBS** operation and examine the results.



Any tool path that falls within the Trim boundary is removed.

❑ **Save** the part file.

You have finished the activity and also the lesson.

SUMMARY

The MILL_AREA geometry group allows additional flexibility in determining exact areas for cutting purposes. The use of this geometry group and Trim Boundaries gives you the flexibility of isolating specific areas of geometry used in the machining process.

In this lesson you:

- Created MILL_AREA geometry machine specific areas
- Used Trim boundaries in conjunction with the MILL_AREA parent group, to further isolate specific areas for machining

Fixed Contour Operation Types

Lesson 18

PURPOSE

This lesson will show you how to create a Fixed Contour operation using several of the options and concepts that are unique to Fixed Contour machining.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Use the Fixed Contour Area Milling and Flow Cut Drive methods to create tool paths
- Use Non-cutting moves

This lesson contains the following activities:

Fixed Contour Terminology

The following are terms that are used in Fixed Contour applications:

- **Drive Geometry –** used to generate Drive Points, either on or projected onto, the part geometry
- **Drive Method –** these are alternative methods of creating tool paths based upon the type of geometry to be cut or special cutting requirements
- **Part Geometry –** is the resultant geometry that is to remain after machining
- **Non-Cutting Moves –** are cutter positioning moves that do not cut any material

Fixed Contour Overview

Fixed Contour operations use a fixed tool axis for accurately finishing contoured geometry. Fixed Contour can effectively clean up ridges and scallops left by other tool paths.

Fixed Contour is the better choice for finish machining for several other reasons:

- In addition to Part geometry, *Drive* geometry can control tool movement
- Numerous *Drive Methods* are available for specialized machining
- Uncut areas left after semi-finishing or finishing passes can be easily removed

Drive Methods

The Drive Method that you select is dependent upon the geometry that you are to machine. You will learn about the following Drive Methods:

- Area Milling
- Flowcut

Area Milling

This method examines the part geometry for tool path accuracy and collision avoidance. With Area Milling you may select individual faces for milling or mill the entire part.

Flowcut

This method examines the part geometry for those areas that the cutter could not machine in prior operations. You can then use smaller tools to cut those areas in other Flow Cut operation types.

Flow Cutting will not occur in areas where the curvature of the surface is larger than the corner radius of the tool since the tool must always contact part geometry at two points.

Other Drive Methods

Other Fixed contour Drive Methods are:

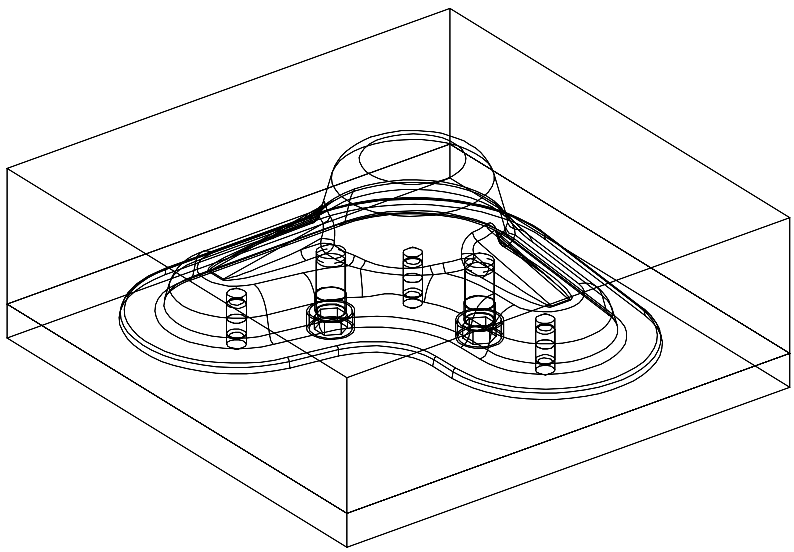
- Curve/Point
- Spiral
- Boundary
- Surface Area
- Tool Path
- Radial Cut
- User Function

Activity 18–1: Creating Fixed Contour operations

This activity creates Fixed Contour rough and finish operations.

Step 1 Open the part file, rename and enter the Manufacturing application.

❑ Open the part **mmp_male_cover_mfg_3.prt**.



❑ **Save As** ***_**male_cover_mfg_3.prt.**

❑ Enter the **Manufacturing** application and display the Operation Navigator.

Step 2 Review the Cavity Milling roughing operation.

This part file contains a Cavity Milling operation that rough cuts the part.

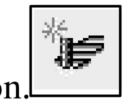
❑ Highlight the **ROUGH_CM** operation**,** using **MB3**, choose **REPLAY.**

Note that a number of .250 steps were left in the material as a result of the specified Cut Level. Also, .050 Floor and Side Stock were specified in the operation.

❑ **Refresh** the display.

You will create a Fixed Contour operation to finish machine the part.

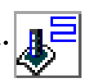
Step 3 Creating a Fixed Axis operation.

❑ Choose the **Create Operation** icon. 

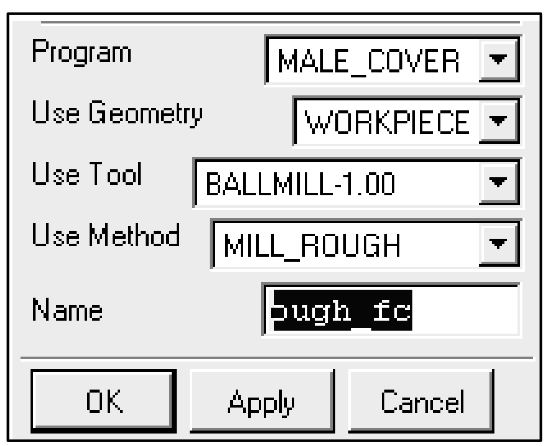
❑ If necessary, change the Type to **mill_contour**.

❑ In the Create Operation dialog, set:

- Program **MALE_COVER**
- Use Geometry **WORKPIECE**
- Use Tool **BALLMILL–1.00**
- Use Method **MILL_ROUGH**

❑ Choose the **CONTOUR_ZIGZAG** icon. 

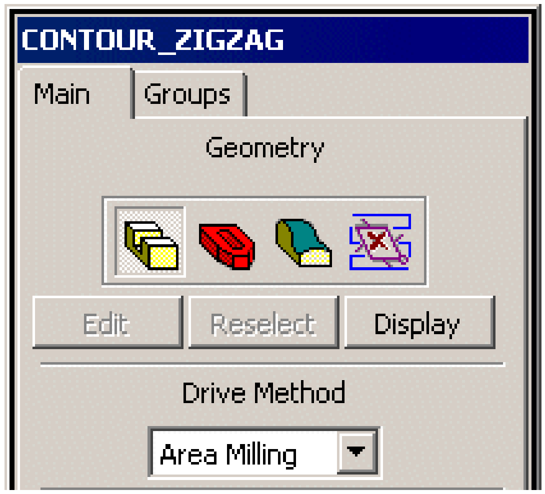
❑ Enter the Name as **rough_fc.**



❑ Choose **OK.**

The CONTOUR_ZIGZAG dialog is displayed.

18



❑ Under the Geometry label, choose **Display** for the **Part** and **Check** geometry.

You will use most of the default settings of the Area Milling Method to create a roughing tool path.

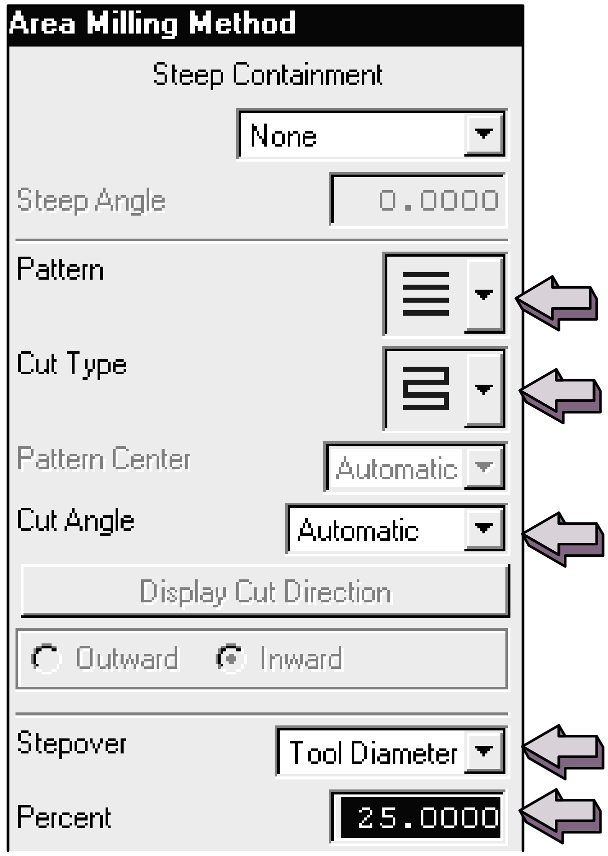
18

❑ Under the Drive Method label, choose **Area Milling.**

The Area Milling Method dialog is displayed.

❑ Set the following options:

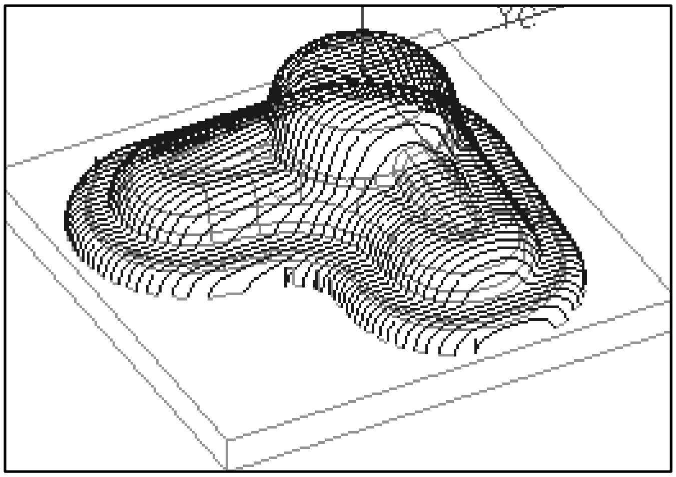
- Pattern **Parallel Lines**
- Cut Type **Zig Zag**
- Cut Angle **Automatic**
- Stepover **Tool Diameter**
- Percent **25**



❑ Choose **OK**.

❑ **Generate** the tool path and expect a warning.

❑ Choose **OK.**



❑ Choose the **List** icon to list the tool path.

Note that the listing contains many warnings of Interference between the cutter and the Check geometry.

You will see messages similar to the one shown below.

Information

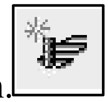
File Edit

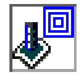
```
TOOL PATH/ROUGH_FC,TOOL,BALLMILL-1.00
TLDATA/MILL,1.0000,0.5000,5.0000,0.0000,0.00
MSYS/-5.2500,-9.2500,-4.5000,1.0000000,0.000
PAINT/PATH
PAINT/SPEED,10
PAINT/TOOL,FULL,20
PAINT/COLOR,4
RAPID
GOTO/0.9043,2.9442,4.6000,0.0000000,0.000000
PAINT/COLOR,1
RAPID
GOTO/0.9043,2.9442,1.1020
$$ START OF CHECK GEOMETRY INTERFERRENCE
PAINT/COLOR,3
FEDRAT/IPM,10.0000
GOTO/0.9043,2.9746,0.6338
```

❑ Close the listing window.

❑ Choose **OK** to accept the operation.

Step 4 Creating a Fixed Axis finishing operation using the Contour Follow operation type.

❑ Choose the **Create Operation** icon. 

❑ Choose the **CONTOUR_FOLLOW** icon. 

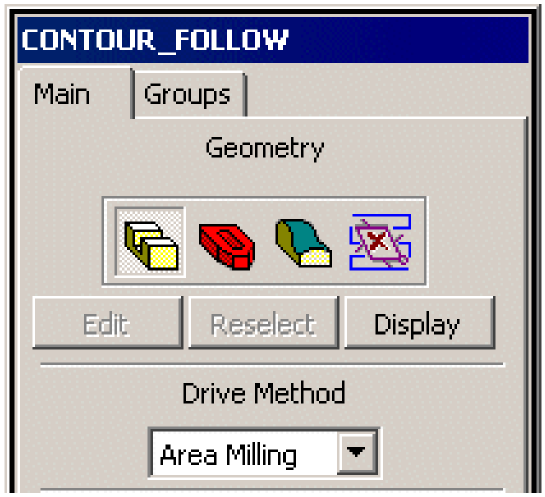
❑ In the Create Operation dialog, set:

- Program **MALE_COVER**
- Use Geometry **WORKPIECE**
- Use Tool **BALLMILL–1.00**
- Use Method **MILL_FINISH**

❑ Enter the Name as **finish_fc.**

❑ Choose **OK.**

The CONTOUR_FOLLOW dialog is displayed.



❑ Under the Geometry label, choose **Display** for the **Part** and **Check** geometry.

Note that the part geometry as well as the check geometry representing pins, bolts and the surface plate are displayed.

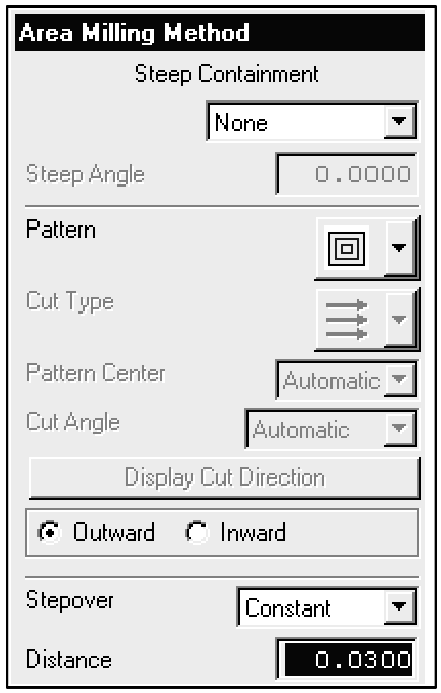
18

❑ Under the Dive Method label, choose **Area Milling.**

The Area Milling Method dialog is displayed.

❑ Set the following options:

- Pattern **Follow Periphery**
- Tool motion **Outward**
- Stepover **Constant**
- Distance **.030**



❑ Choose **OK**.

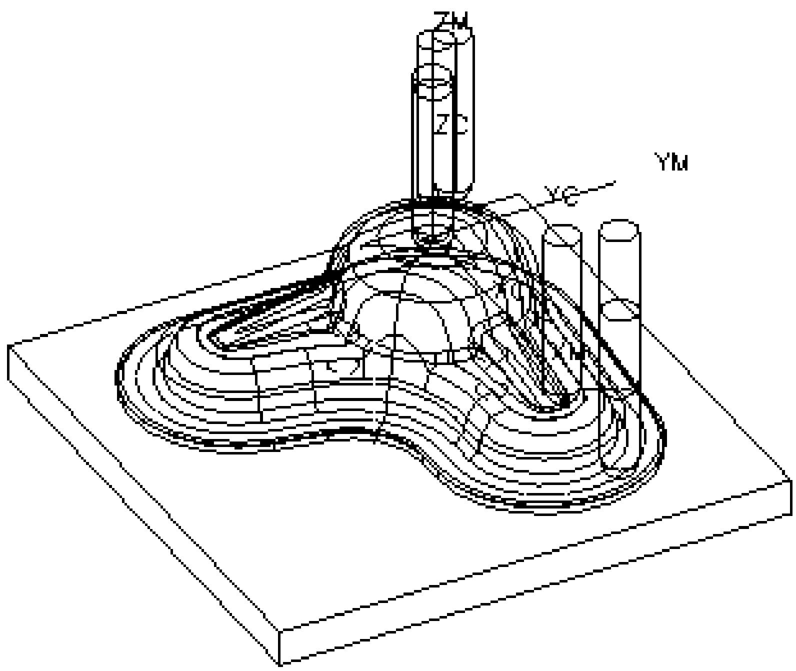
The next action will prevent the Warning message from appearing.

❑ Under the Machining Parameters label, choose **Cutting.**

❑ Change the When Gouging option to **Retract**.

❑ Choose **OK.**

❑ **Generate** the tool path.



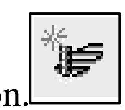
Your tool path should look similar to the above. Note that Warnings were not generated and the tool path follows the contour of the part. In the illustration above, the Stepover distance was increased for better visualization.

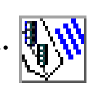
18

❑ Choose **OK** to accept the tool path.

Step 5 Creating a Flow Cut finishing operation.

The tool could not fit into some areas of the part geometry because of tool size. You will use a Flow Cut operation and a smaller tool to remove uncut areas.

❑ Choose the **Create Operation** icon. 

❑ Choose the **FLOWCUT_REF_TOOL** icon. 

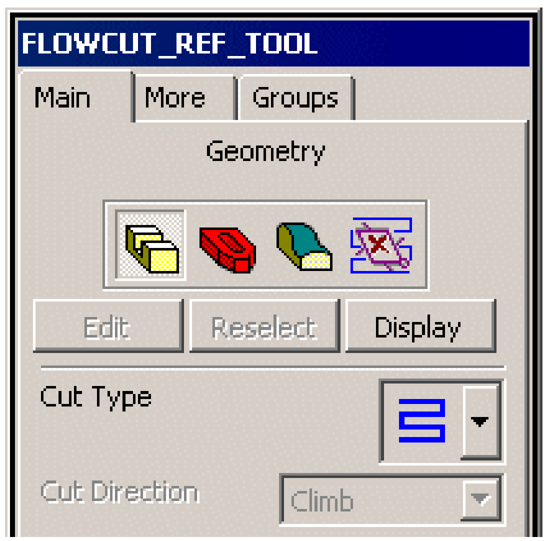
❑ In the Create Operation dialog, set:

- Program **MALE_COVER**
- Use Geometry **WORKPIECE**
- Use Tool **BALLMILL–0.500**
- Use Method **MILL_FINISH**

❑ Enter the Name as **flow_fc.**

❑ Choose **OK.**

The FLOWCUT_REF_TOOL dialog is displayed.



❑ Under the Geometry label, **Display** the **Part** and **Check** geometry.

Note that on the dialog there is no Drive Method label since Flow Cut is the Drive Method.

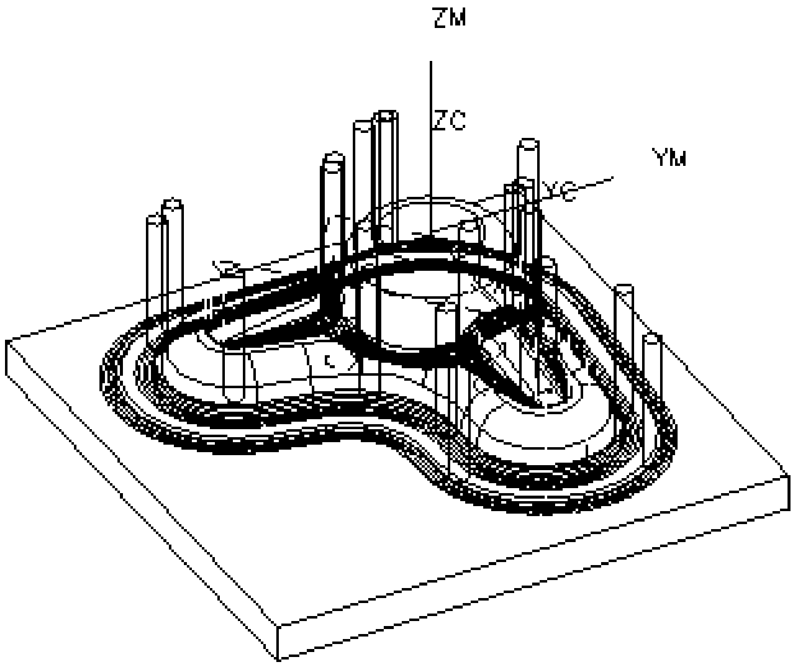
Step 6 Changing the Reference Tool setting.

You will change the Reference Tool setting. The previous tool used was a 1.00 diameter tool.

❑ Enter **1.00** in the Ref. Tool Diameter value field.

Step 7 Generating the tool path.

❑ Choose the **Generate** icon.

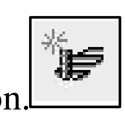


Note that the area being cut is in reference to the 1.000 Reference Tool diameter.

❑ Choose **OK**.

Step 8 Creating a finish Planar Milling Profile pass.

You have finish machined the core part except for the tapered outer edge. The geometry is planar and requires a finish cut; therefore, you will use a Planar Milling operation to generate the tool path. The MILL_BND geometry parent group, which contains the geometry needed for the profile pass, has already been created for you.

❑ Choose the **Create Operation** icon. 

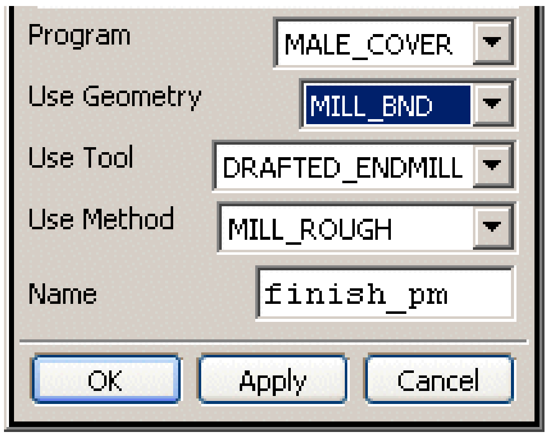
❑ Choose **mill_planar** as the Type.

❑ Choose the **PLANAR_PROFILE** icon.

❑ Set the following:

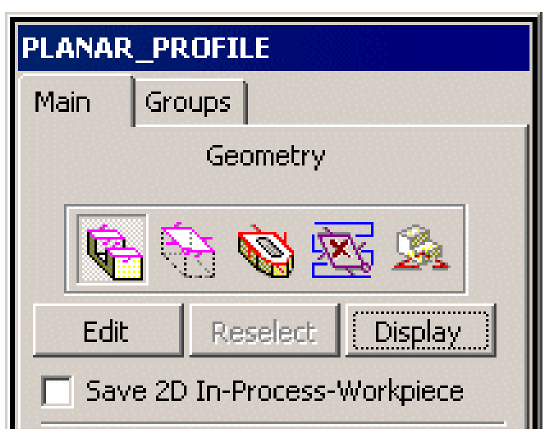
- Program **MALE_COVER**
- Use Geometry **MILL_BND**
- Use Tool **DRAFTED_ENDMILL**
- Use Method **MILL_FINISH**

❑ Enter the Name as **finish_pm.**



❑ Choose **OK.**

The PLANAR_PROFILE dialog is displayed.



❑ Choose the **Display** button.

This Parent Group (MILL_BND) contains the outer edge **Part** boundary and the part **Floor**.

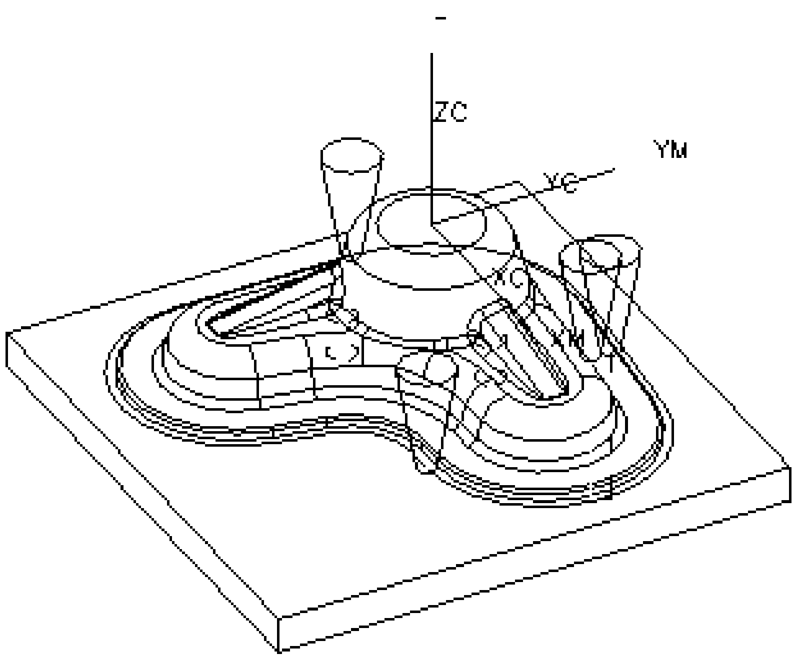
- Remember, this is a Planar Milling operation, which uses boundary geometry
- You generally use a MILL_BND Parent Group for Planar Milling operations

❑ If necessary, set the Cut Depth to **Floor Only** for a single depth of cut.

The other default PLANAR_PROFILE settings will be used to demonstrate this operation.

❑ **Generate** the tool path.

Note that the tool cuts the outer boundary and forms the tapered wall joining the part to the plate.



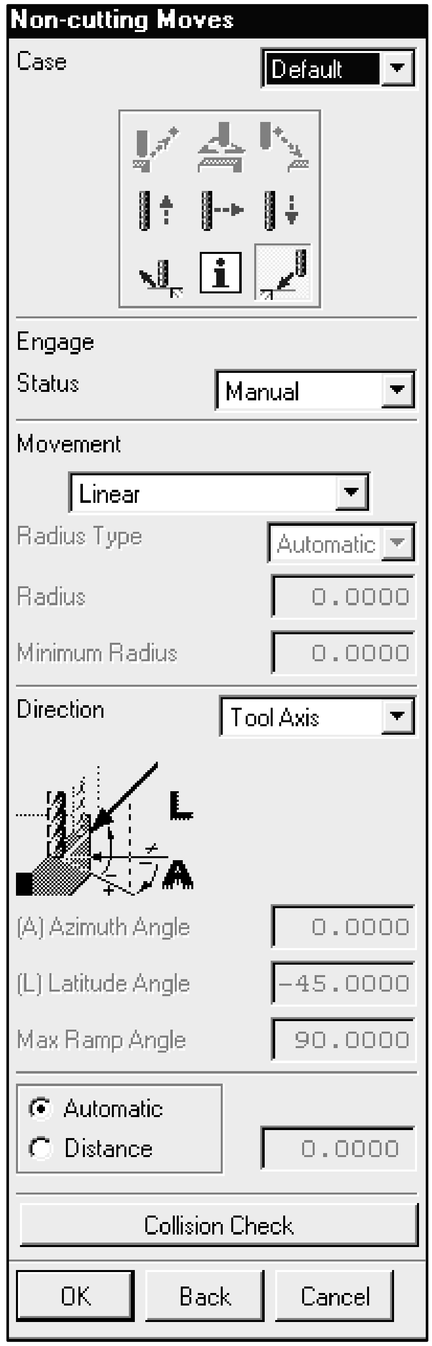
❑ Choose **OK** to accept the operation

❑ **Save** the part file.

This completes this activity.

Non-Cutting Moves

Fixed Contour operations uses Non-Cutting Moves for control of the tool when not physically cutting metal.

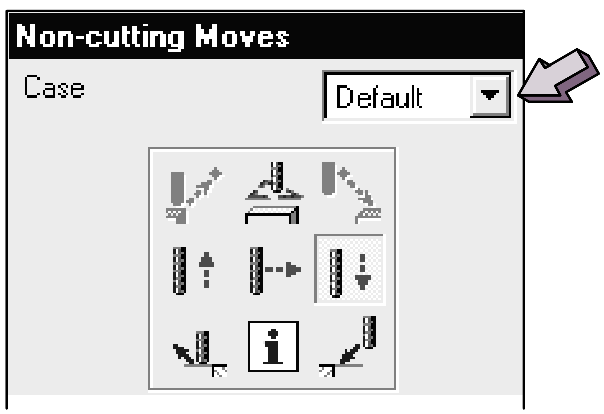


18

There are five individual cases when the tool is not physically cutting metal. They are:

- **Initial Case –** At the beginning of the operation, controls how the tool moves from it's present position to cutting metal
- **Final Case –** At the end of the operation, controls how the tool moves from it's last cutting move to a safe position above the work piece
- **Check Case –** When the encountering check geometry, determines how the tool retracts from the work piece and moves to a new cutting position
- **Reposition Case –** controls how the tool retracts and re-engages the work piece when there are gaps in the part geometry.
- **Local Case –** When the tool has to leave the part surface to complete the stepover for the next pass, this determines what action will be taken

The Case is specified at the top of the Non-Cutting Moves dialog.



Each Case has up to five moves that can be specified. The Moves are:

- **Retract Move –** controls how the tool disengages from the work piece
- **Departure Move –** Once the tool has retracted, controls how the tool moves to a safe clearance area
- **Traverse –** move from the current position to a safe area above the next engage position
- **Approach Move –** controls movement into position for engage motion
- **Engage Move –** controls how the tool engages into the work piece

To avoid having you manually set all moves for all cases, the Default Case was created. This case has all the moves that the other cases have. Each move has been pre-defined for the most common machining situation. Additionally, all other cases have been assigned to use the Default case.

To use Non-Cutting Moves:

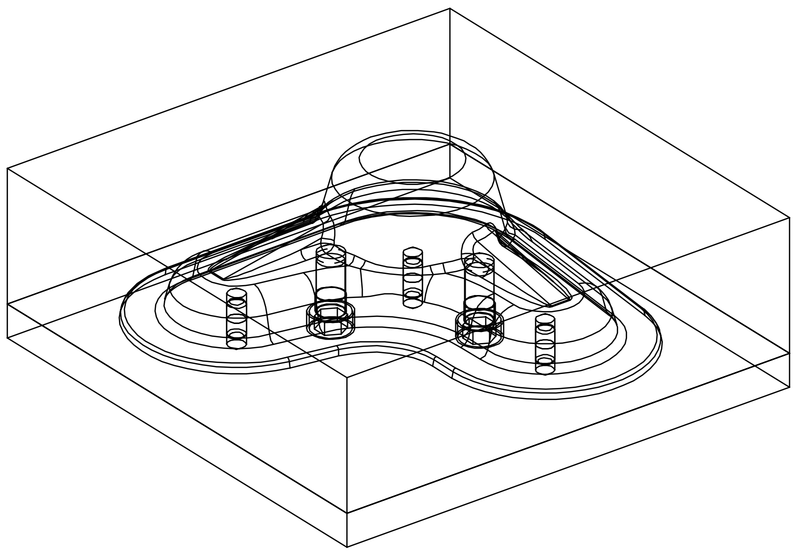
- Create a Fixed Contour operation
- Set all Cutting Parameters necessary (Drive Method, stepover, etc.)
- Generate the operation
- Examine the default Non-Cutting moves
- If necessary, edit the Non-Cutting moves and change only the affected moves

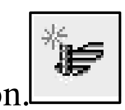
Activity 18—2: Using Non-Cutting Moves

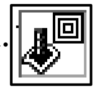
This activity teaches you how to use the various Non-Cutting Moves options.

Step 1 Continue using the part file.

- ❑ Continue using the *****_male_cover_mfg_3.prt.**



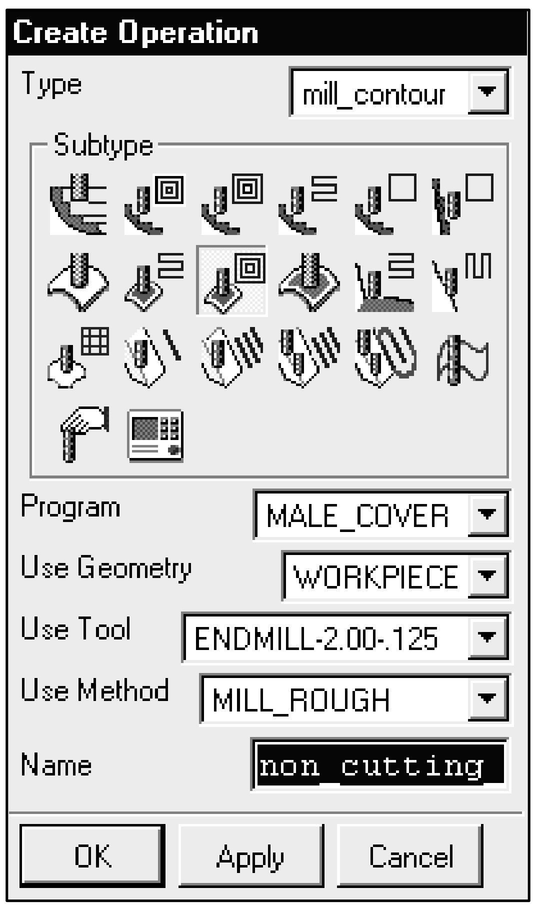
- ❑ Choose the **Create Operation** icon. 
- ❑ The Type should be set to **mill_contour**.

❑ Choose the **CONTOUR_FOLLOW** icon. 

❑ Set:

- Program **MALE_COVER**
- Use Geometry **WORKPIECE**
- Use Tool **ENDMILL–2.00–.125**
- Use Method **MILL_ROUGH**

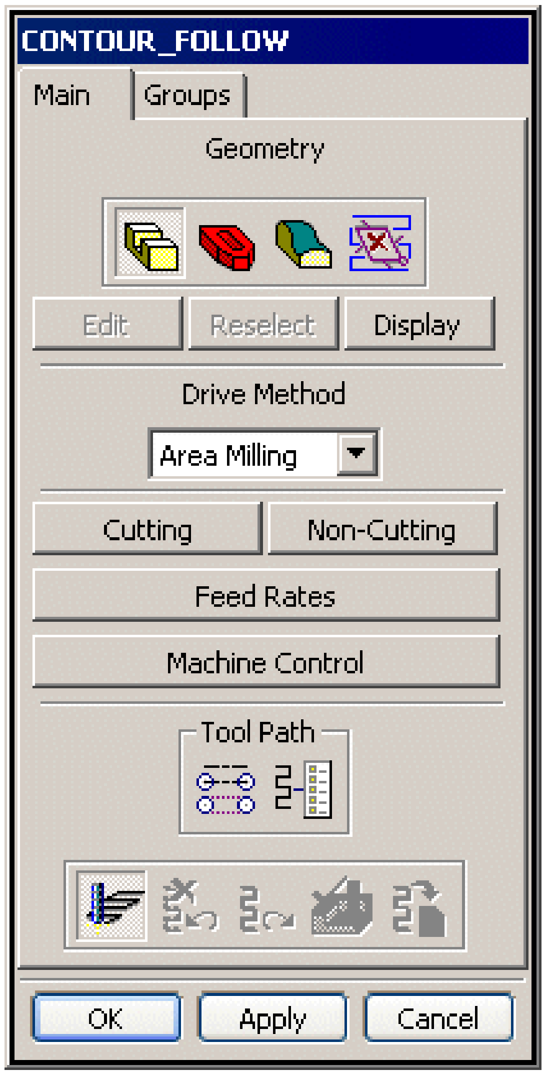
❑ Enter the Name as **non_cutting_fc.**



❑ Choose **OK.**

Step 2 Generate the default tool path.

The CONTOUR_FOLLOW dialog is displayed. The Drive Method is Area Milling.

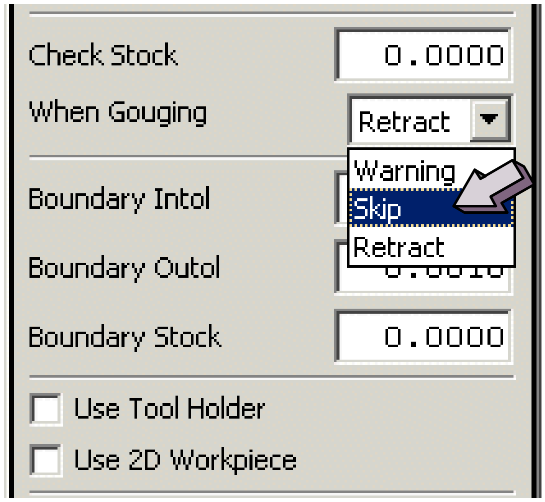


Also note, the Geometry that you specified is WORKPIECE, which is a MILL_GEOM Parent Group for contour geometry.

❑ Under the Geometry label, **Display** the Part, and Check geometry to verify the geometry selections.

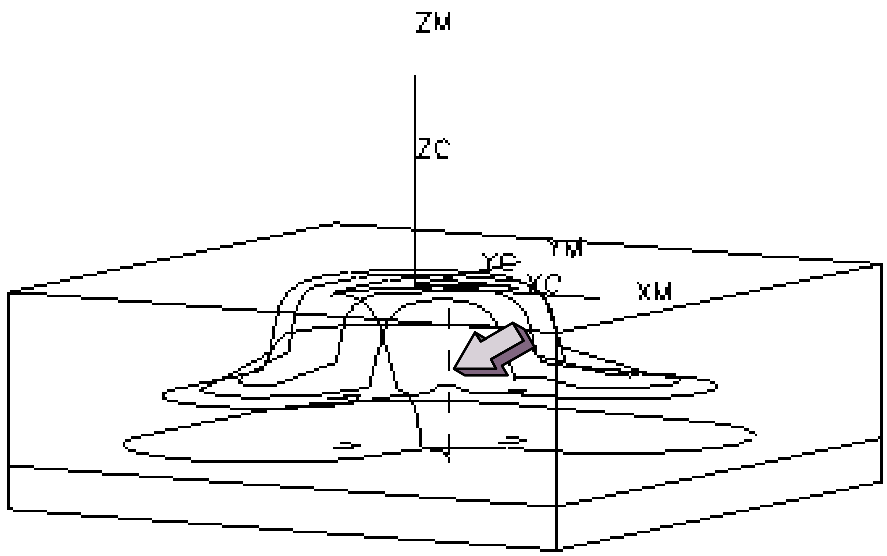
❑ For easier visualization, set the Tool Display to **Off**.

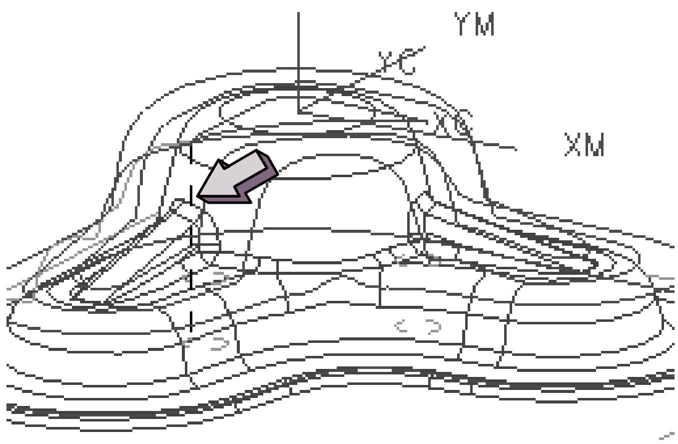
❑ Choose **Cutting** → **When Gouging** → **Skip.**



❑ Choose **OK** for the Cut Parameters dialog**.**

❑ **Generate** the tool path.

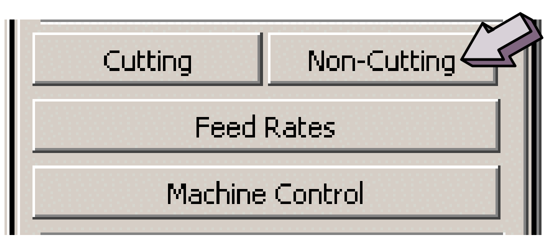




Note the Non-Cutting move to the Automatic Clearance Plane. The move is represented as the dashed vertical line.

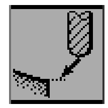
Step 3 Specifying Non-Cutting Moves.

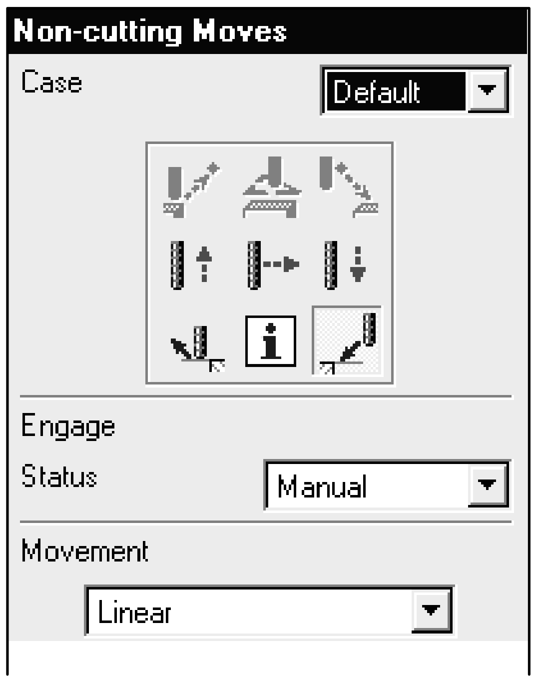
❑ On the CONTOUR_FOLLOW dialog, choose the **Non-Cutting** option.

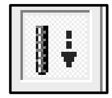


Note that the Case is set to **Default**.

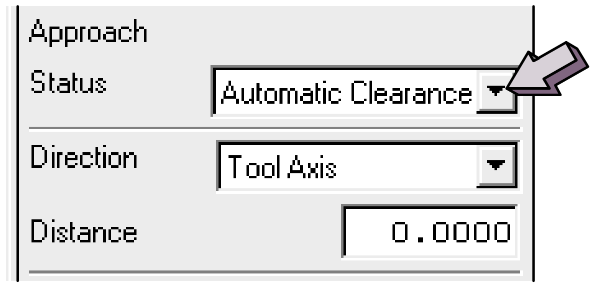
Also Note:

- The **Engage** icon  is highlighted by default
- The Engage Status is **Manual**
- The movement is **Linear**



❑ Choose the **Approach** icon. 

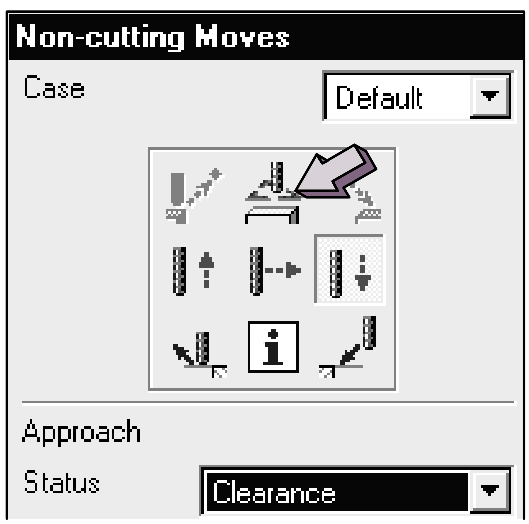
A default **Automatic Clearance** plane is created at a safe distance above the highest area of the Part and Check geometry.



Next you will change the default setting from **Automatic Clearance** to **Clearance.**

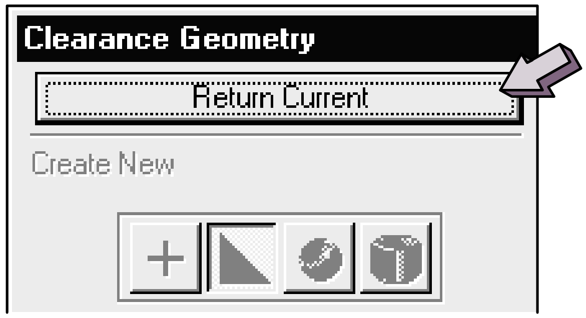
❑ Choose **Clearance** from the Approach Status pull down menu.

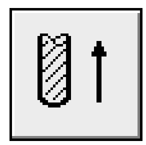
❑ Choose the **Clearance** icon.



Notice that the Plane icon on the Clearance geometry dialog is selected.

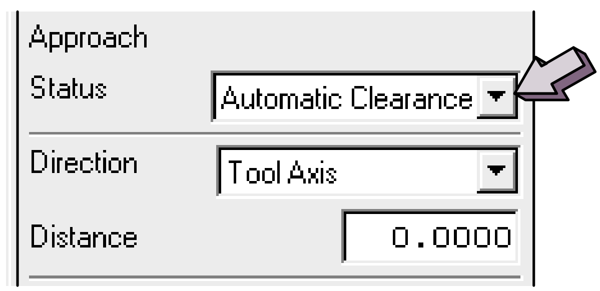
❑ Choose the **Return Current** button from the Clearance geometry dialog. This will activate the Clearance Plane.



❑ Choose the Departure icon 

Again, a default **Automatic Clearance** plane is created at a safe distance above the highest area of the Part and Check geometry.

18



Next you will change the default setting from **Automatic Clearance** to **Clearance.**

❑ Choose **Clearance** from the Approach Status pull down menu.

Note that the Clearance icon becomes available.

❑ Choose the **Clearance** icon.

Notice that the Plane icon on the Clearance geometry dialog is selected.

❑ Choose the **Return Current** button from the Clearance geometry dialog.

❑ Choose **OK.**

❑ **Generate** and review the tool path to verify that the clearance plane moves are correct.

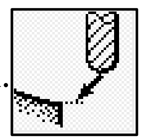
Note that the tool path engages the part in a linear motion. The preferred method of engagement is a circular ramping motion.

❑ Choose the **Reject** icon.

Step 4 Changing the Engage move.

❑ Choose the **Non-Cutting** button**.**

You are still setting options for the Default case.

❑ Choose the **Engage** icon. 

The Engage Status should be **Manual** by default.

❑ Change Movement to **Arc Tangent to Approach.**

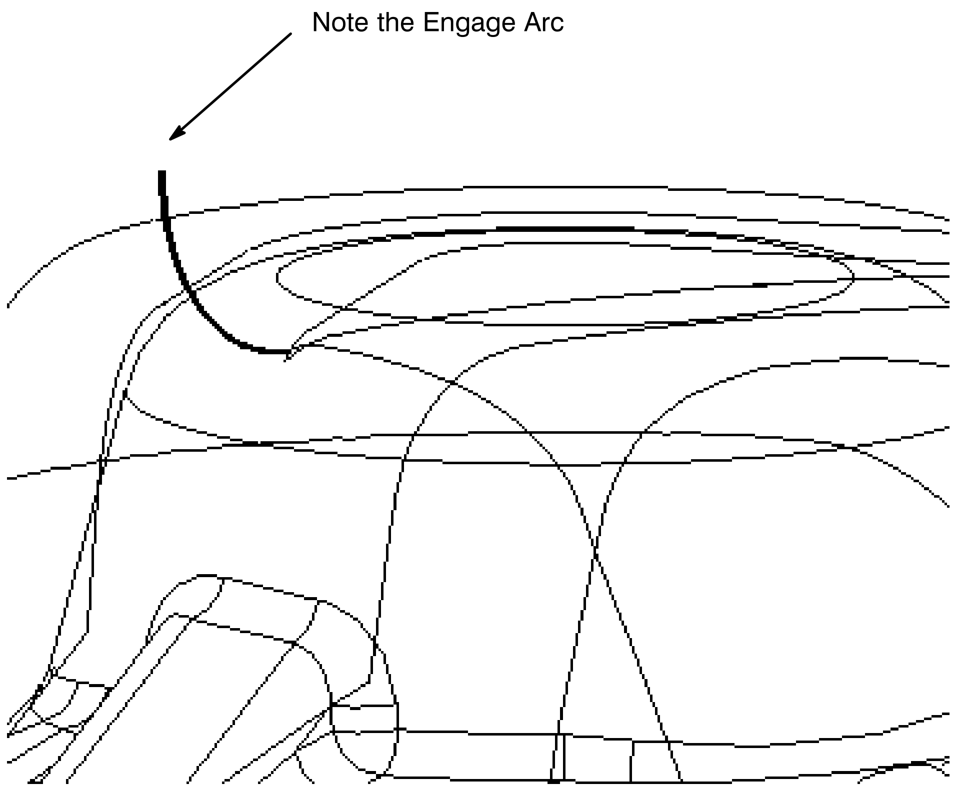
❑ Change Radius Type to **Radius.**

❑ Enter **.500** for the Radius.

The Retract Status setting will be set to **Use Engage** by default. This can be verified by choosing the **Retract** icon.

❑ Choose **OK** and return to the CONTOUR_FOLLOW dialog.

❑ **Generate** the tool path.



Step 5 Changing other Non-Cutting options.

❑ On your own, change other Non-Cutting options and generate tool paths to see the effects.

❑ **Save** and close the part file.

This completes the activity and the lesson.

SUMMARY

This lesson introduced you to Fixed Contour operations that gives you the ability to machine complex contour geometry with numerous options.

In this lesson you:

- created Area Milling and Flow Cut operations
- made extensive use of the MILL_GEOM and MILL_BND parent group
- created non-cutting moves to control cutter movements to and from the part during the machining process

Tool Path Information Output

Lesson 19

PURPOSE

This lesson introduces several ways that tool path data can be output as data and text. You will learn how to use UGPOST to Postprocess your program. Once you successfully post the program you can then create the shop documentation that can be used on the shop floor.

OBJECTIVES

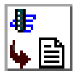
Upon completion of this lesson, you will be able to:

- Postprocess with UGPOST
- Create Shop Documentation
- Output a CLSF

This lesson contains the following activities:

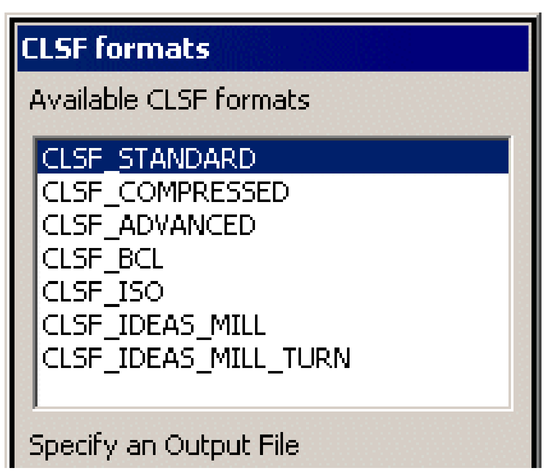
Output CLSF

Cutter Location Source Files, also referred to as CLSF, are output formats that can be used for input into numerous postprocessing systems, including legacy GPM and other 3rd party systems.

The icon  **Output CLSF** – is used to create a CLSF.

Types of output are:

- **CLSF_STANDARD** – standard APT type output, with GOTO and postprocessors statements
- **CLSF_COMPRESSED –** outputs only the START and END-OF-PATH statements
- **CLSF_ADVANCED –** automatically generates Spindle and Load Tool commands based on operation data
- **CLSF_BCL –** represents Binary Coded Language which is a specific controller language developed in conjunction with the US Navy
- **CLSF_ISO –** represents a cutter location source file based on ISO standards
- **CLSF_IDEAS_MILL –** represents an IDEAS compatible cutter location source file
- **CLSF_IDEAS_TURN**



If you have legacy data that you need to post using the GPM, you would use the CLSF_Standard format.

Postprocessing

You use the Manufacturing application to generate tool paths for machining parts. The tool path consists of GOTO points and other information that controls the movement of a tool with respect to the part. This unmodified tool path usually needs to be specifically formatted for a particular machine tool/controller combination. Some of these differences are based on character formats, tool change requirements, type of machine, number of controlled axis of motion, etc.

The tool path must be formatted to match the unique characteristics of the machine tool/controller combination. The procedure of modifying this generic tool path to a form that can be understood and used by the machine tool controller is called postprocessing.

Two elements are required for postprocessing. They are:

- **tool path** - A Unigraphics internal tool path
- **postprocessor** - this is a program that reads, converts and reformats tool path information for a particular machine tool/controller combination

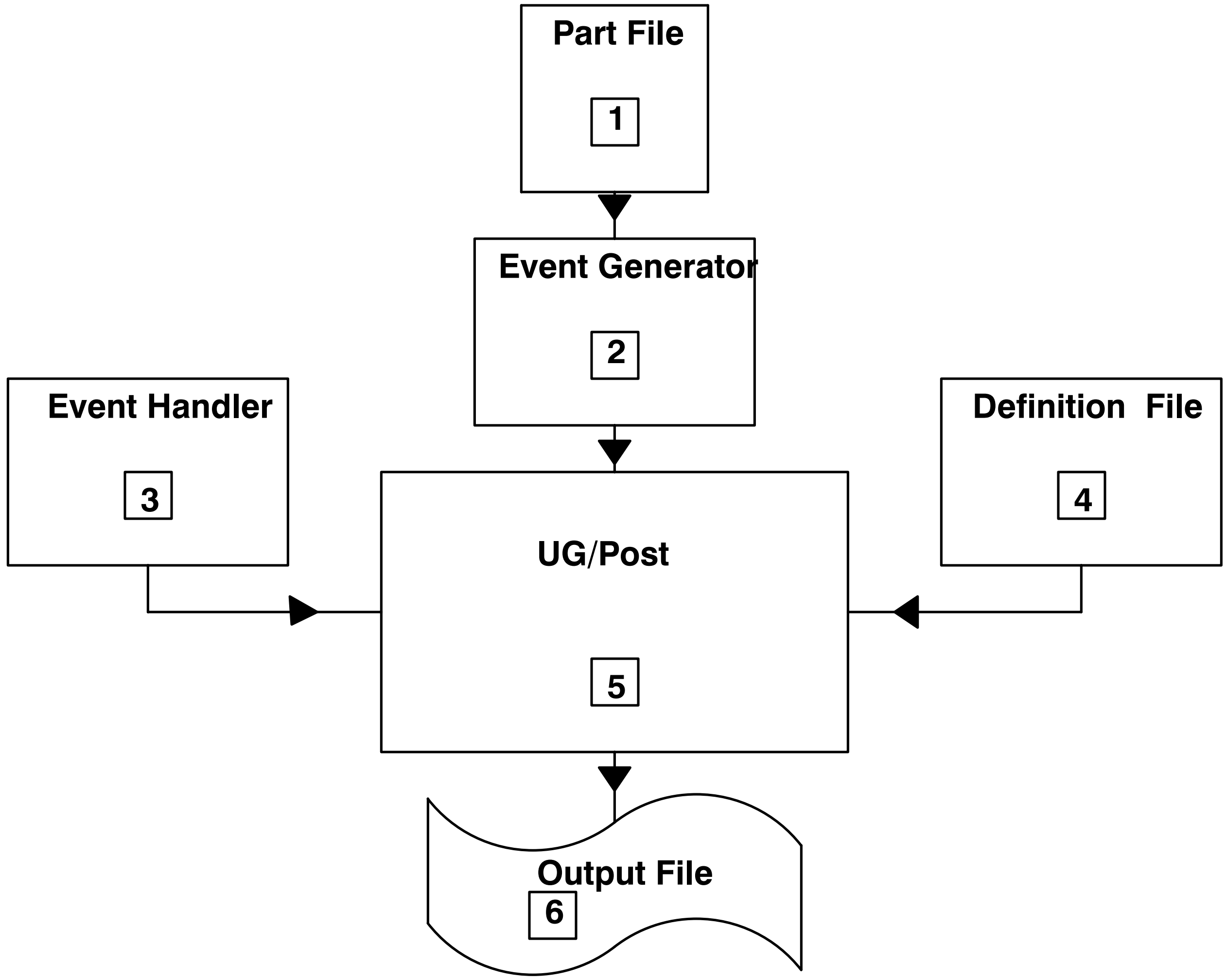
UGPOST Execute

Unigraphics provides a postprocessor, UGPOST, which utilizes Unigraphics tool path data as input and outputs machine controller readable NC/CNC code.

UGPOST is customizable through the use of user created Event Handler and Definition files. These files, in conjunction with UGPOST, are used to generate output for the simplest to the very complex of machine tool/controller combinations.

The UGPOST processor is highly scalable and can be used to generate output for simple milling machines and lathes to ultra complex multi-axis (4+ axis) machining and production centers (a production center is considered to be a milling/turning type machine). The flexibility of UGPOST is achieved through the scripting language Tcl and the use of the Unigraphics concept of Definition files.

The following flowchart illustrates the steps required to process (postprocess) tool path data in an acceptable format for a machine tool/controller using the UGPOST postprocessor.



1 **Part File**

2 **EVENT GENERATOR extracts information from the part and sends information to the postprocessor**

3 **EVENT HANDLER processes each EVENT according to instructions**

4 **EVENTS processed by the EVENT HANDLER are formated according to the Definition File**

5 **UGPOST Postprocessor controls the entire sequence**

6 **Postprocessed machine control instructions are written to an Output file**

19

The UGPOST execute module consists of the following components:

- **Event Generator -** sends Events to UGPOST when you postprocess; an Event is a collection of data which is processed by UGPOST, creating data which causes a specific action(s) by the machine tool/controller
- **Event Handler -** is a file containing a specific set of instructions, written in the Tcl scripting language, dictating how each event type is to be processed
- **Definition file -** is a file containing specific information about machine tool/controller format
- **Output file -** is a file generated by UGPOST, passed to the machine tool/controller, that executes specific instructions

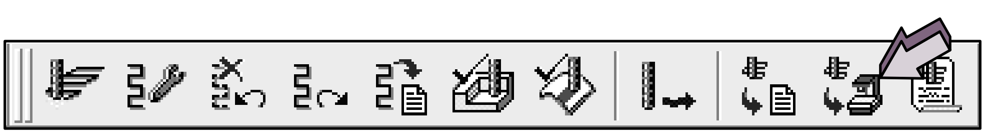
The Event Generator, Event Handler, and the Definition file are interdependent and together convert the internal tool path into a set of instructions that can be read and executed by the specific machine tool/controller combination.

Manufacturing Output Manager (MOM)

The Manufacturing Output Manager, commonly referred to as MOM, is a utility program used by UGPOST for generating output based upon data that is stored within the internal tool path. Functionally, UGPOST uses the Manufacturing Output Manager to start, add data and specify functions to the interpreter, and to load Event Handlers and Definition files.

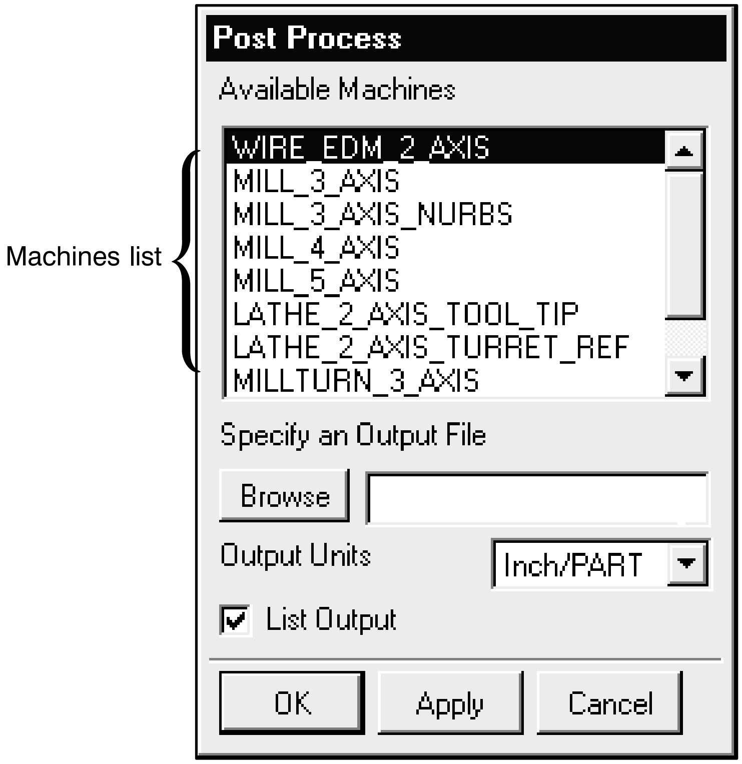
Postprocessing dialogs using UGPOST

The UGPOST postprocessor is activated by selecting the UGPOST postprocessor icon from the Manufacturing Operations toolbar.



Once the icon is select, the Post Process dialog is displayed.

The Post Process dialog list the available postprocessors for postprocessing.



The postprocessor is added to the Post Process dialog by modifying the *template_post.dat* file located in the *\mach\resource\postprocessor* directory. This file specifies the location of the definition and event handler files used for your particular postprocessor.

Specify an Output File allows you to specify where you want the posted output to go.

UGPOST Builder

The UGPOST mechanism as mentioned earlier, uses Tcl (Tool Command Language) scripts and numerous files to postprocess information. These files are used to extract information from the part file, process this information according to defined rules, format the information for output and then output the data to a file which is later used by the machine control for machining a part. These files are highly customizable and require the knowledge of the Tcl scripting language.

To ease the process of creating these files and knowing Tcl, the UGPOST Builder was developed.

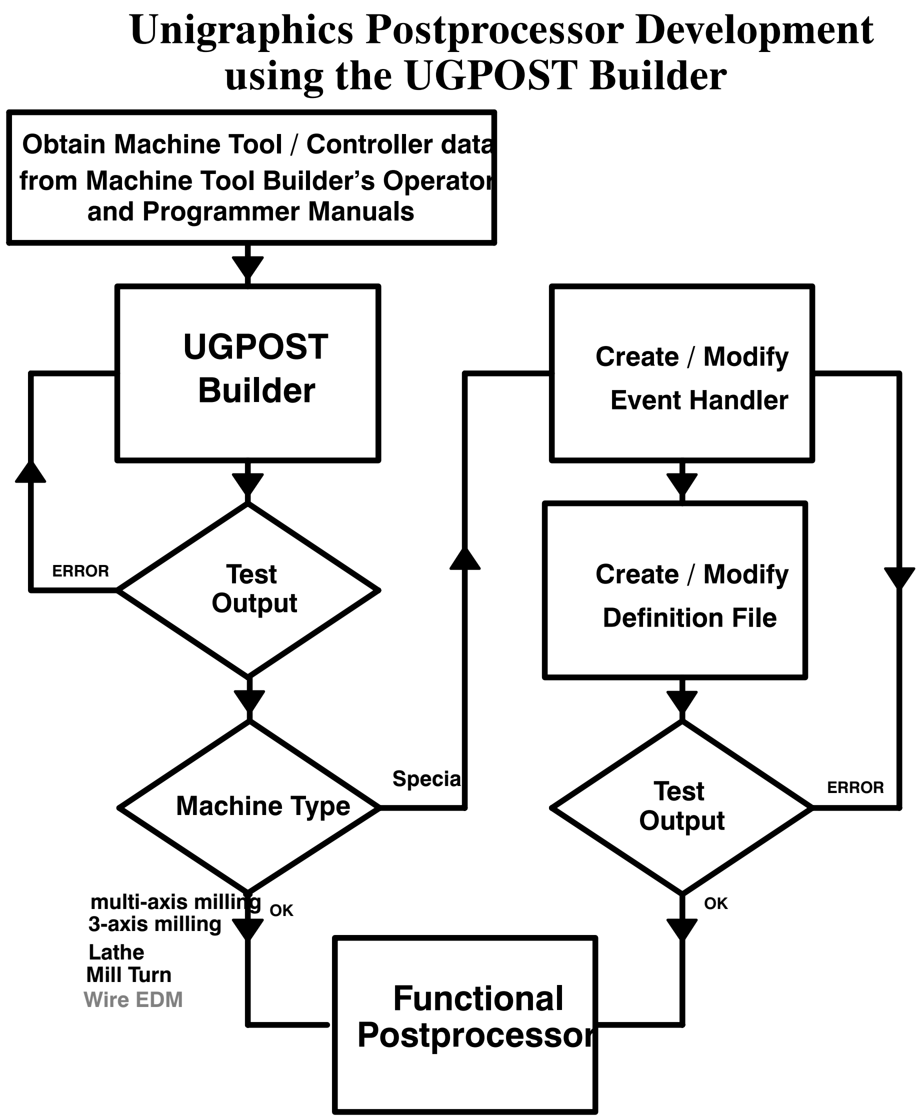
UGPOST Builder provides an interactive graphical User Interface for building postprocessors. The design intent of the UGPOST Builder is to create all the necessary files needed for postprocessing without the knowledge of file structure or Tcl.

The UGPOST Builder is very flexible and allows for the definition of various types of output blocks and word addresses. Sequence of output in the NC output file is very easy to control for blocks involving the start of program, start of operation, end of operation, end of program, tool changes and canned cycles.

UGPOST Builder currently configures postprocessors for the following:

- 3-axis milling machines
- 4-axis milling machines with a rotary table or a rotary head
- 5-axis milling machines with dual rotary heads or dual rotary tables
- 5-axis milling machines with rotary head and rotary table
- 2-axis lathes
- Mill-Turn centers (lathes with a live milling spindle)

The following flowchart illustrates the process of building a postprocessor using the UGPOST Builder.



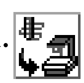
Activity 19—1: Post Processing Using UGPOST

In this activity, you are going to postprocess tool paths using the UGPOST postprocessor.

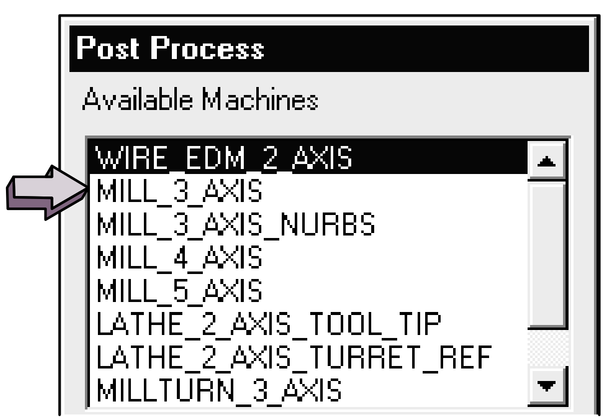
Step 1 Open the part file, rename and enter the Manufacturing application.

- ❑ Open the part file **mmp_bearing_case_mfg_3.prt**.
- ❑ **Save** the part as *****_bearing_case_mfg_3.prt.**
- ❑ Enter the **Manufacturing** application, if necessary.
- ❑ Change the Operation Navigator to the **Program Order View**, if necessary.

Step 2 Using the UGPOST Postprocessor.

- ❑ Highlight the **PROGRAM** Parent Group object on the Operation Navigator.
- ❑ Choose the **Postprocessing** icon. 

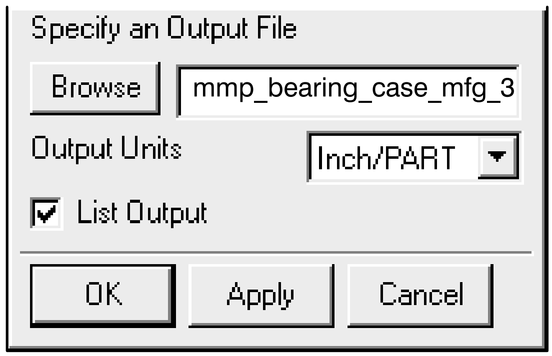
The Post Process dialog is displayed. You are going to use the MILL_3_AXIS post.



- ❑ Choose **MILL_3_AXIS** from the Available Machines list.

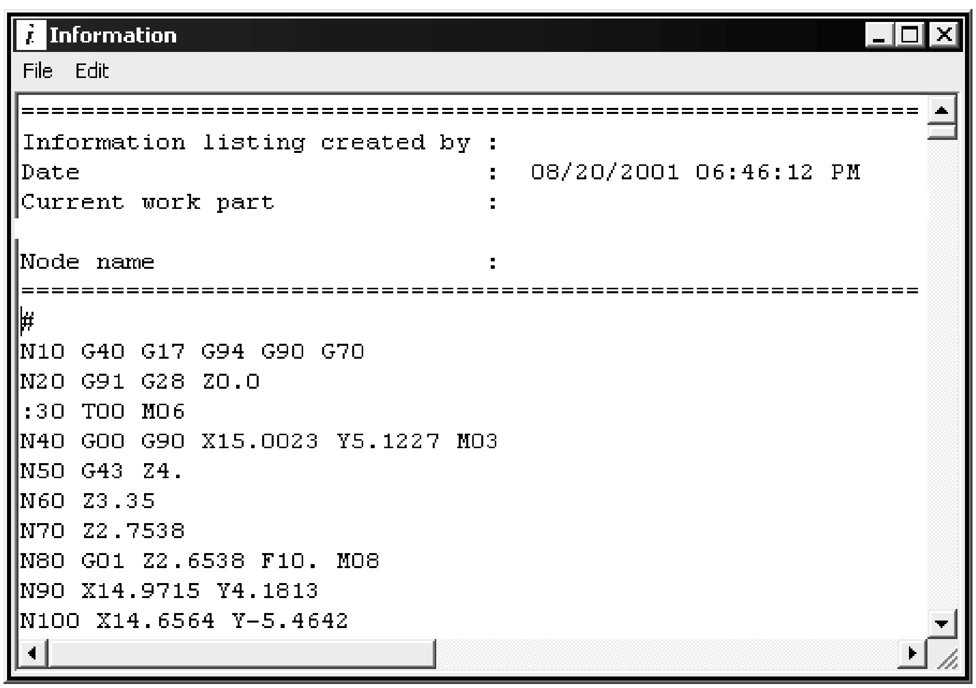
You need to specify a name and location for the posted output file that you are about to create. The default location is your current directory.

❑ Accept the default name in the **Specify an Output File** field.



❑ Choose **OK**.

The Information listing window is displayed containing the postprocessed output for the three tool paths selected.



❑ **Close** the Information window.

❑ Do not save or close the part file.

This completes this activity. You will be using this part file in the next activity.

Shop Documentation

Shop Documentation allows you to generate customized information in the form of reports that can aid in the manufacturing processes of creating a part.

This information includes data concerning:

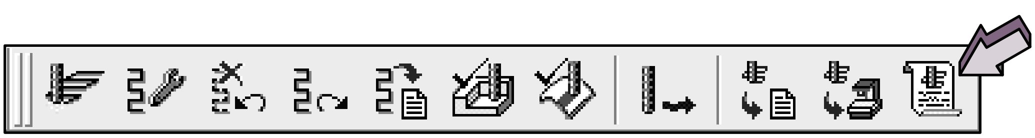
- tools and material
- control geometry
- machining parameters
- post commands
- tool parameters
- tool path information

Output can be either ASCII text or HTML format.

Generic templates are provided or customized templates can be created and used to create very detailed output that can be incorporated into the manufacturing environment.

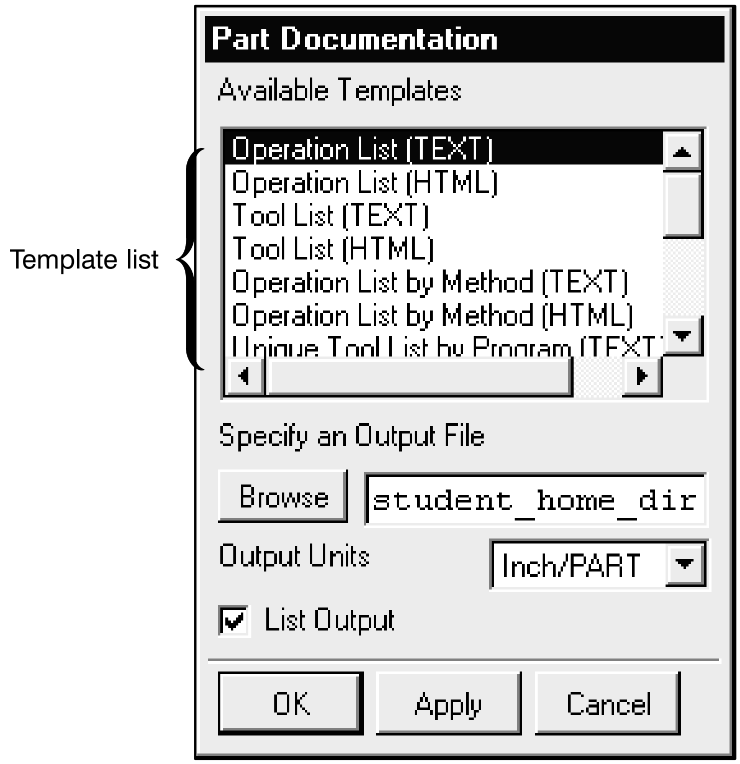
Shop Documentation Dialogs

Shop Documentation is activated by selecting the Shop Documentation icon from the Manufacturing Operations toolbar.



Once the icon is selected, the Part Documentation dialog is displayed.

The Part Documentation dialog lists the Available Templates which are used to format the output.



These customized templates create various formatted output in both ASCII text and HTML for:

- operation and tool list
- operation list by method
- tool list by program
- advanced operation list
- web page listings

You can add your own custom templates to the dialog by modifying the *shop_doc.dat* file located in the *\mach\resource\shop_doc* directory. This file specifies the location of the template and event handler files used for your shop documentation.

Specify an Output File allows you to specify where you want the shop output to go.

Activity 19—2: Creating Shop Documentation

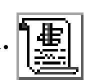
In this activity, you are going to use the Shop Documentation option to create documentation.

Step 1 Open the part file.

- ❑ Continue to use the part file *****_bearing_case_mfg_3.prt**.
- ❑ Enter the **Manufacturing** application, if necessary.
- ❑ Change the Operation Navigator to the Program Order View, if necessary.

  The Operation Navigator is displayed.

Step 2 Using the Shop Documentation Operation List (TEXT) option.

- ❑ Highlight the **PROGRAM** parent group on the Operation Navigator.
- ❑ Choose the Shop Documentation icon. 

  The Shop Documentation dialog is displayed.

  You must specify a name and location for the documentation data file you are about to create. The default location is the current directory which should be your home directory.

  Note that you will accept the default name in the Specify an Output File field (or change to your home directory path).

- ❑ Highlight **Operation List (TEXT)** in the Available Templates list box.
- ❑ Use the remaining default settings for Output Units and List Output.

❑ Choose **OK**.

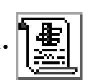
The Operation List is displayed in the Information window.

```
--------------------------------------------------------------------------------
                        OPERATION  LIST BY PROGRAM
                        *********  **** ** *******
PROGRAM NAME : NC_PROGRAM
--------------------------------------------------------------------------------
OPERATION NAME                 OPERATION DESCRIPTION               TOOL NAME
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------

PROGRAM NAME : PROGRAM
--------------------------------------------------------------------------------
OPERATION NAME                 OPERATION DESCRIPTION               TOOL NAME
--------------------------------------------------------------------------------
CAVITY_MILL                    mill_contour/CAVITY_MILL            UGTI0202_019
PLANAR_PROFILE                 mill_planar/PLANAR_PROFILE          UGTI0201_023
SPOT_DRILLING                  drill/SPOT_DRILLING                 UGTI0322_010
--------------------------------------------------------------------------------

PROGRAM NAME : GOUGE_CHECK
--------------------------------------------------------------------------------
OPERATION NAME                 OPERATION DESCRIPTION               TOOL NAME
--------------------------------------------------------------------------------
PLANAR_PROFILE_GOUGE           mill_planar/PLANAR_PROFILE          UGTI0201_023
--------------------------------------------------------------------------------
```

❑ Examine the information that has been listed.

❑ **Exit** the Information window.

Step 3 Using the Shop Documentation Tool List (TEXT) option.

❑ Choose the Shop Documentation icon. 

You do not need to specify the output file because this option is modal.

❑ Highlight **Tool List (TEXT)** in the Available Templates list box.

❑ Choose **OK**.

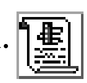
The Tool List is displayed in the Information window.

```
                              TOOLING  LIST
                              *******  ****

DRILLING TOOLS
----------------------------------------------------------------------------------------------
TOOL NAME           DESCRIPTION                    DIAMETER   TIP ANG    FLUTE LEN  ADJ REG
----------------------------------------------------------------------------------------------
UGTI0322_006        Centre Drill Type A            0.1094     120.0000   0.1220     0
UGTI0322_007        Centre Drill Type A            0.1250     120.0000   0.1535     0
UGTI0322_009        Centre Drill Type A            0.1875     120.0000   0.2480     0
UGTI0322_010        Centre Drill Type A            0.2500     120.0000   0.3150     0
----------------------------------------------------------------------------------------------

MILLING TOOLS
----------------------------------------------------------------------------------------------
TOOL NAME           DESCRIPTION                    DIAMETER   COR RAD    FLUTE LEN  ADJ REG
----------------------------------------------------------------------------------------------
UGTI0203_008        Ball End 2"                    2.0000     1.0000     2.5000     0
UGTI0203_013        Ball End 1 1/2"                1.5000     0.7500     2.0000     0
UGTI0203_014        Ball End 1"                    1.0000     0.5000     1.5000     0
UGTI0203_015        Ball End 3/4"                  0.7500     0.3750     1.5000     0
UGTI0203_016        Ball End 1/2"                  0.5000     0.2500     1.0236     0
UGTI0201_013        End Mill 1"                    1.0000     0.0000     1.7716     0
UGTI0201_024        End Mill 1 1/2"                1.5000     0.0000     2.0000     0
```

❑ Examine the information that has been listed.

❑ **Exit** the Information window.

Step 4 Using the Shop Documentation Tool List (HTML) option.

❑ Choose the Shop Documentation icon. 

You do not need to specify the output file because this option is modal.

❑ Highlight **Tool List (HTML)** in the Available Templates listing window.

❑ Choose **OK**.

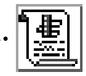
❑ Open the HTML browser (Internet Explorer or Netscape).

❑ On the browser select the HTML file you just created.

❑ Examine the output.

- ❑ Return to Unigraphics.

Step 5 Using the Shop Documentation Advanced Operation List (HTML) option.

- ❑ Choose the Shop Documentation icon. 
- ❑ Highlight **Advanced Operation List (HTML)** in the Available Templates listing window and choose **OK.**
- ❑ Choose **OK**.
- ❑ Examine the output.
- ❑ **Close** the part file.

You have completed this activity as well as the lesson.

SUMMARY

The flexibility of the postprocessing and shop documentation options that are available allows you to postprocess data and generate customized shop documentation for your specific needs.

In this lesson you:

- used the UGPOST postprocessor to create output for a 3-axis milling machine
- created Shop Documentation for providing information for set-ups and manufacturing processes in both ASCII text and HTML formats

(This Page Intentionally Left Blank)

Libraries

Appendix A

A

PURPOSE

This chapter introduces you to the concept of libraries as they pertain to the Manufacturing Application. Libraries are used to access data for cutting tools, machine tools, part materials, tool materials, cut methods and speeds and feeds.

Libraries contain predefined entries, such as cutting tools and part materials, and can be modified with user defined entries. Pre-V16 tool libraries contained in part files can also be converted and utilized.

OBJECTIVES

Upon completion of this lesson, you will be able to:

- Understand the concept and functionality of CAM libraries and data files
- Add entries to existing libraries
- Convert existing tool part file libraries to V18 tool libraries

This lesson contains the following activities:

Overview of CAM Libraries

Libraries are a convenient and easy tool that are used to access reference data. Currently libraries can access information related to:

- Cutting tools
- Machine tools
- Part materials
- Cutting tool materials
- Cut methods
- Speeds and feeds

The configuration selected at the beginning of your CAM session (i.e. **mill_contour**) defines the *location* of the various external libraries. Each external library is represented by one line of information which identifies library type and points to the Event Handler (file with **.tcl** extension) and Definition file (file with **.def** extension) that are used for the Data Base Connection (DBC). The Definition file is used to establish a class hierarchy (the way tools are organized) and associated mechanism for queries, establishes dialog layout definitions, attribute mappings, option menu definitions, library reference names and delimiters.

Sample Configuration file (mill_contour.dat)

```
TEMPLATE_OPERATION,${UGII_CAM_TEMPLATE_SET_DIR}mill_contour.opt
TEMPLATE_DOCUMENTATION,${UGII_CAM_SHOP_DOC_DIR}shop_doc.dat
TEMPLATE_POST,${UGII_CAM_POST_DIR}template_post.dat
USER_DEFINED_EVENTS,${UGII_CAM_USER_DEF_EVENT_DIR}ude.cdl,$
        {UGII_CAM_USER_DEF_EVENT_DIR}ude.tcl
TEMPLATE_CLSF,${UGII_CAM_TOOL_PATH_DIR}template_clsf.dat
LISTING_FORMAT,${UGII_CAM_TOOL_PATH_DIR}clsf.def,${UGII_CAM_TOOL_PATH_DIR}clsf_listing.tcl
LIBRARY_TOOL,${UGII_CAM_LIBRARY_TOOL_ASCII_DIR}dbc_tool_ascii.def,$
        {UGII_CAM_LIBRARY_TOOL_ASCII_DIR}dbc_tool_ascii.tcl
LIBRARY_MACHINE,${UGII_CAM_LIBRARY_MACHINE_ASCII_DIR}dbc_machine_ascii.def,$
        {UGII_CAM_LIBRARY_MACHINE_ASCII_DIR}dbc_machine_ascii.tcl
LIBRARY_FEEDS_SPEEDS,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}feeds_speeds.def,$
        {UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}feeds_speeds.tcl
LIBRARY_PART_MATERIAL,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}part_materials.def,$
        {UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}part_materials.tcl
LIBRARY_TOOL_MATERIAL,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}tool_materials.def,$
        {UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}tool_materials.tcl
LIBRARY_CUT_METHOD,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}cut_methods.def,$
        {UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}cut_methods.tcl
WIZARD,${UGII_CAM_WIZARD_DIR}wizard_mill_contour.tcl
```

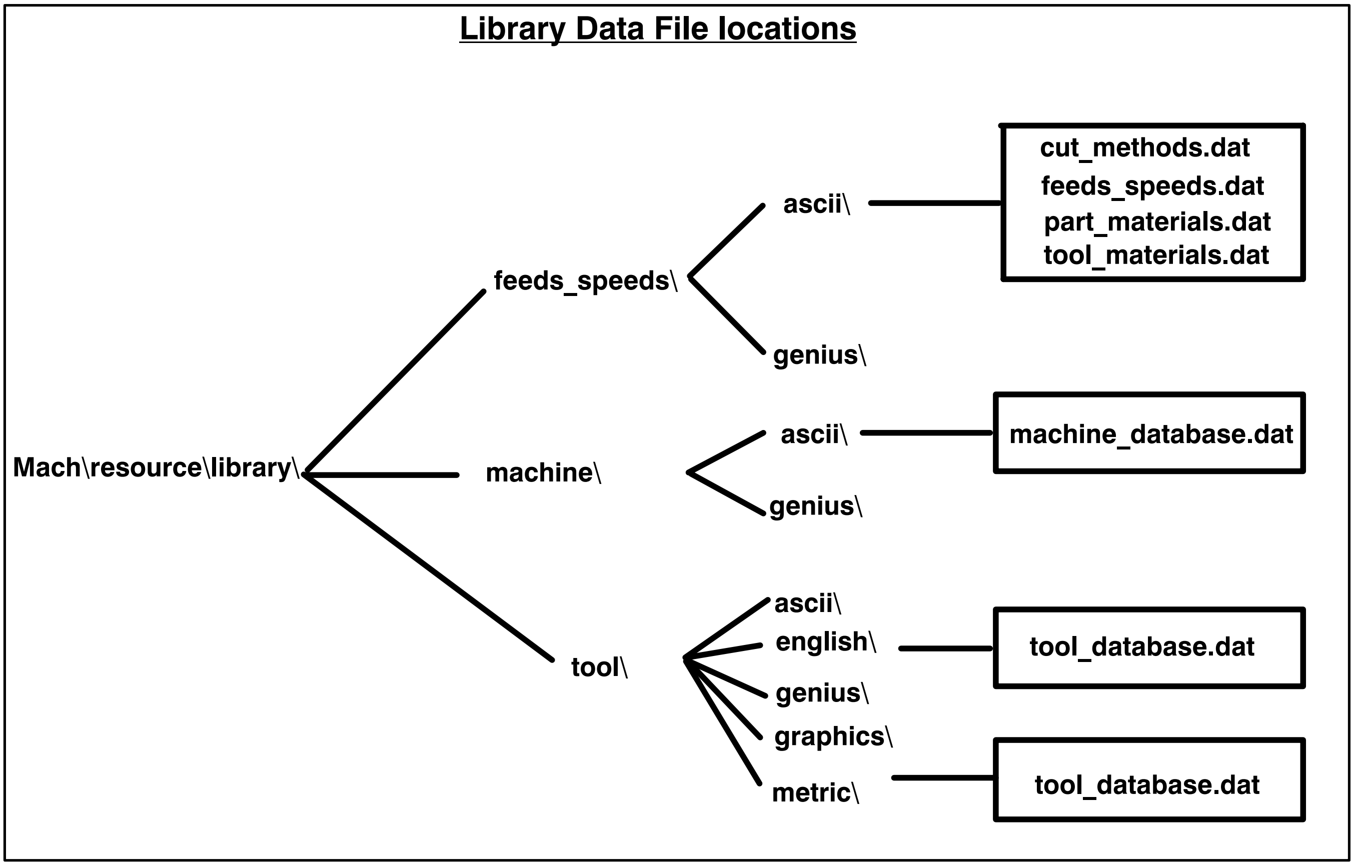
All library files are located in the *Mach\resource\library* directory. Subdirectories, under this directory, are categorized for feeds and speeds, machine and cutting tools. Each subdirectory also contains additional subdirectories of ASCII or GENIUS/4000 files.

Included in each release of Unigraphicsare the access mechanisms to the ASCII text files and optional GENIUS/4000 database files. Sample ASCII based libraries are provided. GENIUS/4000 libraries are optional.

For this lesson we will only be using the ASCII libraries provided with the system.

If your tool requirements include hundreds or thousands of tools, consideration should be given to the optional Genius/4000 Tooling Database Manager.

A



The following pertains to library files in general:

- Some library files are opened and read by the system only the first time they are read for performance reasons.
- When you edit library files, keep the *library references* (library references are names given to every entry in the library) unique.
- After editing library files, reset the configuration to force any changes to be read.
- Not all the information located in the library files are retrieved into the part file. Extra fields are used to aid in selection by POST and Shop Documentation.

Activity A–1: Preparation for modifying CAM Libraries

A

In order to modify libraries, you will need read/write access to the library directory structure. Due to the number of students in this class and the need to customize library files, it is more conducive for each student to have a copy of the library files in their home directory. In this activity you will make a copy of the *mach\resource* directory structure to your home directory and modify the directories for read/write access. Instructions are presented for Windows NT and Unix separately.

Windows NT:

Step 1 Copying the *Mach\resource* directory.

- ❑ On the Unigraphics menu bar, select **Help→UG Log File** to verify the *Mach* directory being used by looking for the environment variable UGII_CAM_BASE_DIR.
- ❑ Open a Windows NT Explorer window and locate the directory from the previous action item.
- ❑ Highlight the *Mach\resource* directory, right click on *Mach\resource* directory and select **Copy**.
- ❑ Highlight your home directory, right click on your home directory and select **Paste.**

Step 2 Copying the UG environment file, *ugii_env.dat* to your home directory.

- ❑ From the Explorer window locate the *ugii_env.dat* file in the *\xxx\ugii* directory (where *xxx* represents the NX base directory).
- ❑ Highlight the *ugii_env.dat* file, right click on the *ugii_env.dat* file and select **Copy.**
- ❑ Highlight your home directory, right click on your home directory and select **Paste.**

A

Step 3 Edit the *ugii_env.dat* file to redefine your *Mach\resource* directory location.

- ❑ Highlight the *ugii_env.dat* file from your home directory, right click on the *ugii_env.dat* file and select **Open.**

- ❑ Scroll down the file until you find the following line: **UGII_CAM_RESOURCE_DIR=${UGII_CAM_BASE_DIR} resource**\ and change the line to **UGII_CAM_RESOURCE_DIR=${Homedrive}resource**\ where Homedrive is the letter designator of the disk drive where your home directory is located.

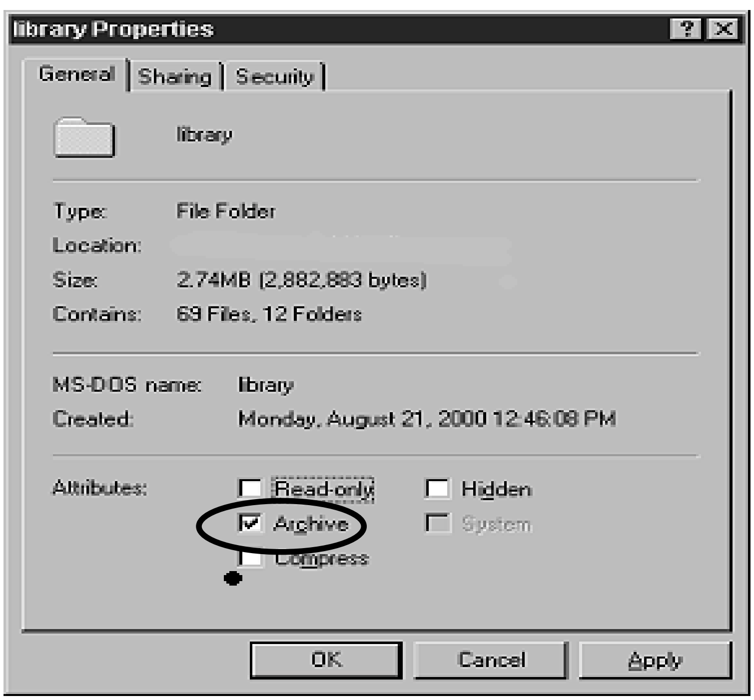
- ❑ Save the file and exit from Notepad.

Step 4 If necessary, change the Read-only protection on your just created local \Mach\resource\library directory to Archive.

- ❑ With the Windows Explorer, locate your home \mach\resource\library directory.

- ❑ Highlight the directory and with **MB3** select **Properties**.

- ❑ Uncheck **Read-Only** and then Check **Archive**.



Step 5 Restart Unigraphics

❑ Exit Unigraphics and then restart Unigraphics.

❑ On the menu bar, select **Help→UG Log File** to verify that ***your*** resource directory is being used.

Unix:

Step 1 Copying the *mach/resource* directory.

❑ On the menu bar, select **Help→UG Log File** to verify the *mach* directory being used by looking for the environment variable UGII_CAM_BASE_DIR.

❑ Open a terminal window making sure that your default directory is set to your home directory.

❑ Copy the *mach/resource* directory to your home directory. The path for the file will be the value obtained for UGII_CAM_BASE_DIR. The format will be similar to the following:

cp /usr/xxx/mach/resource (where xxx represents the NX base directory)**.**

Step 2 Copying the UG environment file, *.ugii_env* to your home directory.

❑ Copy the *.ugii_env* file from */usr/xxx/ugii* directory to your home directory. The format will be similar to the following:

cp /usr/ugii/.ugii_env .

Step 3 Edit the *.ugii_env* file to redefine your *mach/resource* directory location.

❑ Edit the *.ugii_env* file from your home directory using the vi or other Unix editor.

❑ Find the following line in the *.ugii_env* file **UGII_CAM_BASE_RESOURCE=${UGII_CAM_BASE_DIR}resource**/ and change the line to **UGII_CAM_BASE_RESOURCE=${Home}/resource**/.

A

- ❑ Save the changes and exit from editor.

Step 4 If necessary, change the Read-only protection on your just created local /mach/resource directory to rwed.

- ❑ Change the directory protection by typing the following command.: **chmod 777 ${HOME}/mach/resource**

Step 5 Restart Unigraphics

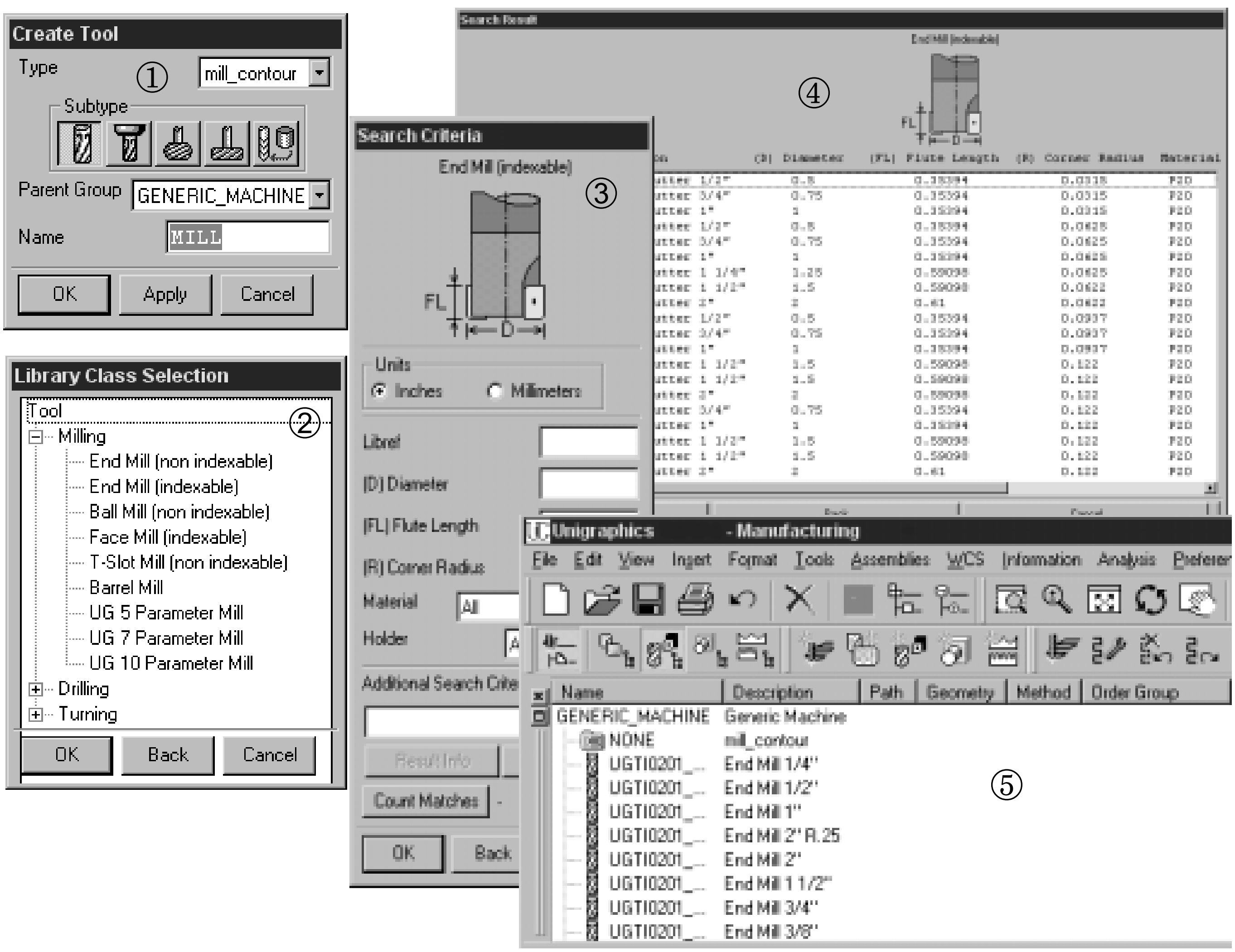
- ❑ Exit Unigraphics and then restart UG.

- ❑ On the menu bar, select **Help→UG Log File** to verify that ***your*** resource directory (based from your home directory) is being used.

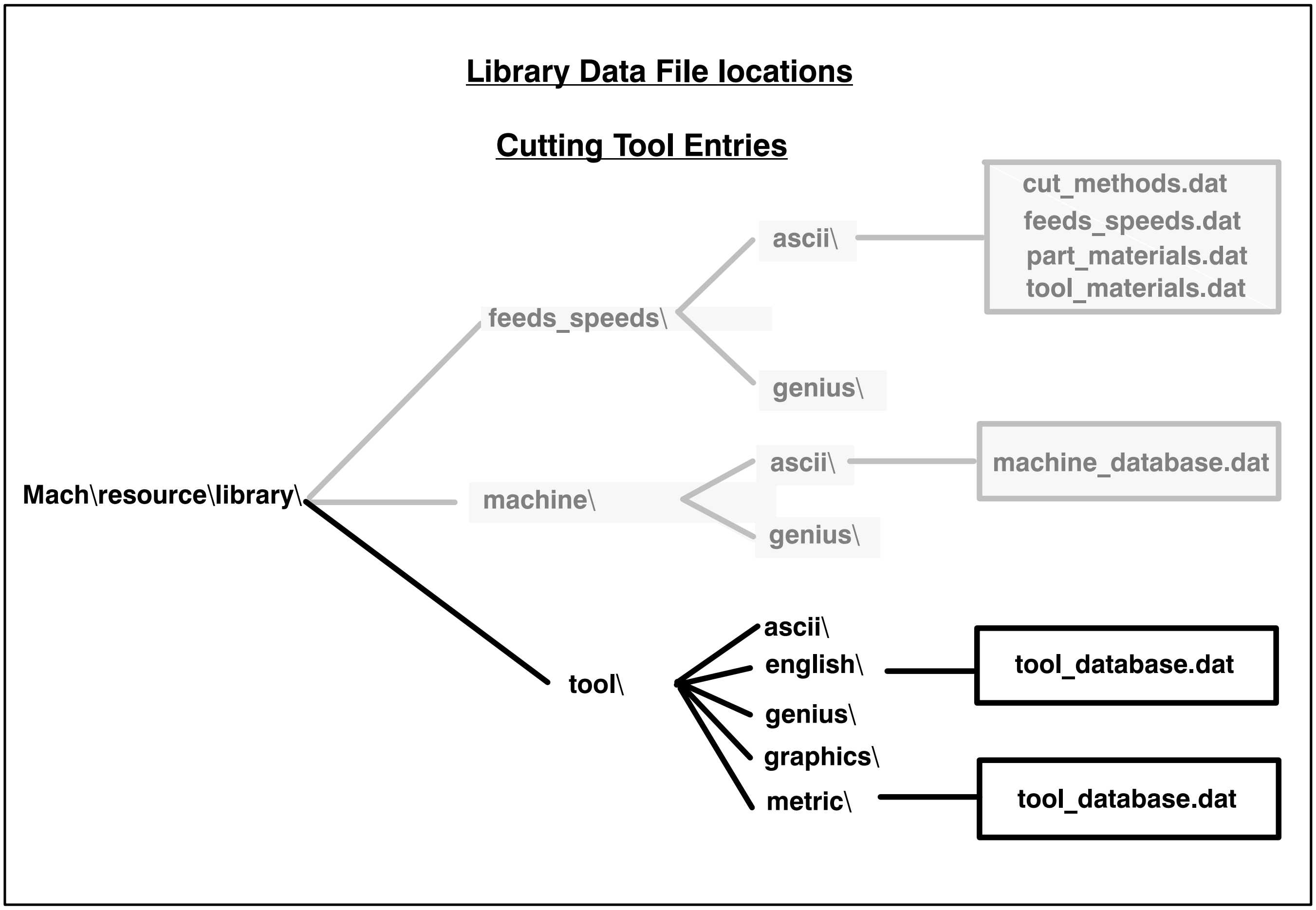
Cutting Tool Libraries

Cutting Tool Libraries contain information related to all cutting tools used in the generation of a tool path from an operation.

To access information in the cutting tool library, from the *Create Tool* dialog box, selection of the *Retrieve Tool* button (①), displays the *Library Class Selection* menu for selection of the particular type of tool required for the Operation which you are about to create (milling, drilling, turning). Once the type of tool is selected (②), the *Search Criteria* (③)dialog is displayed, which allows you to search for tools based on certain parameters. The search is then performed based on information contained within the *tool_database.dat* file (modification of this file, for the addition of your own tool entries, will be explained later in this lesson) and a listing of the *Search Results* (④) is then displayed. Tools can then be selected for retrieval into your part file for later use (⑤).



Cutting tool data is located in the **Mach\resource\library\tool** directory.



This directory contains the following five subdirectories:

- ascii -----contains Definition and Event Handler files for ASCII text databases. These files are used for the Data Base Connection and usually are ***not*** modified by the user.
- english--contains the ASCII text database file (tool_database.dat) which contains all the data records used for *English* tool descriptions. ***This file is edited by the user when adding or modifying tool data entries to the library.***
- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.
- graphics-contains part files of tool assemblies (10 examples) used for advanced replays with a solid tool. These assemblies are provided with

the CAM release.

- metric---contains the ASCII text database file (tool_database.dat) which contains all the data records used for *metric* tool descriptions. ***This file is edited by the user when adding or modifying tool data entries to the library.***

The **tool_database.dat** files contains a list of tool data *records* that defines parameters used for tool definition. This is the only file that needs to be modified when you want to enter new tools.

A data record consists of a *record type* and associated *parameters.*

There are three record_types represented in this file. They are:

- **#** which indicates a comment, the record is ignored.
- **FORMAT** which describes the type of parameters of subsequent DATA records.
- **DATA** which consists of parameters which describe the tool.

For example:

FORMAT LIBRF T ST DESCR MATREF MATDES HLD HLDDES DIA FLEN FN HEI

describes the following data record:

DATA |ugt0201_001|02|01|End Mill|TMC0_00006|HSS−Co5−TiN | 320 | Steep Taper 20 | 10.5 | 35.3| 4| 55

Tool entries can be added to the Tool Libraries by two different methods. Data records, as described previously, can be added to the tool_database.dat file by simply editing the file. Or existing tools from current or legacy part files can be extracted by activating the **Shop Documentation** Icon (**Information −> Documentation**) and selecting **Export Tool Library to ASCII datafile** from the dialog box. This will export all tooling data to the file that you designate. You then will *cut* the tool data record(s) from the designated file and *paste* it into the **tool_database.dat** file.

The following is an example of the various attributes used with the Tool Library formats. All fields are documented within the file. Note that they are *not* the same for each tool.

Tool Library Format

 # LIBRF – Library Reference
 # T – Tool Type
 # ST – Tool SubType
 # DESCR – Description
 # MATREF – Reference to cutter material table
 # MATDES – Cutter material description
 # HLD – Holding system (Type of Machine Adapter)
 # HLDDES – Holding system description
 # DIA – Diameter
 # FLEN – Flute Length
 # FN – Number of Flutes
 # HEI – Height
 # HLD –Tool Holding System
 # HLDDES – Tool Holding System Description
 # DIA – Tool Diameter
 # FN – Tool Flutes Number
 # HEI – Tool Length (Height)
 # ZOFF – Tool Z Offset
 # DROT – Tool Direction (3=clockwise, 4=counterclockwise)
 # FLEN – Tool Flute Length (Cutting Depth)
 # TAPA – Tool Taper Angle
 # COR1 – Tool Corner1 Radius
 # HDIA – Tool Holder Diameter
 # HLEN – Tool Holder Length
 # HTAP – Tool Holder Taper
 # HOFF – Tool Holder Offset

```
FORMAT LIBRF       T  ST DESCR          MATREF     MATDES      HLD HLDDES
DIA   FN HEI     ZOFF     DROT FLEN     TAPA COR1  HDIA    HLEN    HTAP HOFF
DATA   |ugti0201_011|02|01|End  Mill  1/4”     |TMC0_00006|HSS–Co5–TiN|300|Steep  Taper
SKG30|.25  |2 |.90551 |–2.87402|3   |.51181 |    |0.0  |.5315  |1.21654|31. |0.0
```

Note that the **MATREF** attribute is a reference to the cutter material located in the file **\MACH\resource\library\feeds_speeds\ascii\tool_materials.dat**.

Activity A–2: Inserting Pre-existing Tools

Cutting tools within part files can be extracted and inserted into tool libraries. This procedure will work with pre-V16 as well as NX part files. The following activity will take you through the process of inserting tools from a part file into a tool library.

Step 1 Open the pre-existing part file containing tool entries.

- ❑ Open the part file **mmp_lib_tools.prt**

Step 2 Enter the Manufacturing Application.

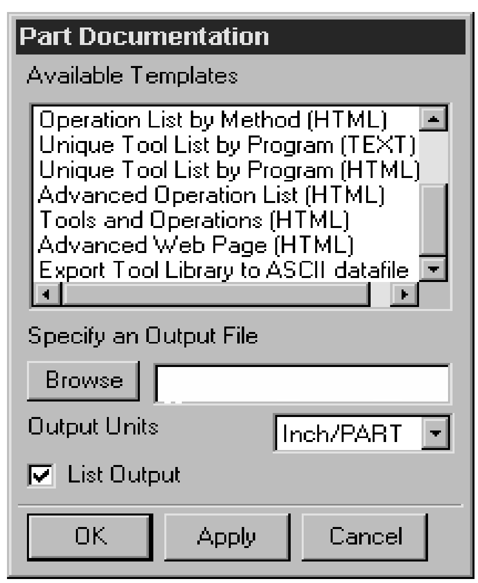
- ❑ Choose **Application** → **Manufacturing**

The Operation Navigator and the Create Operation dialogs are displayed.

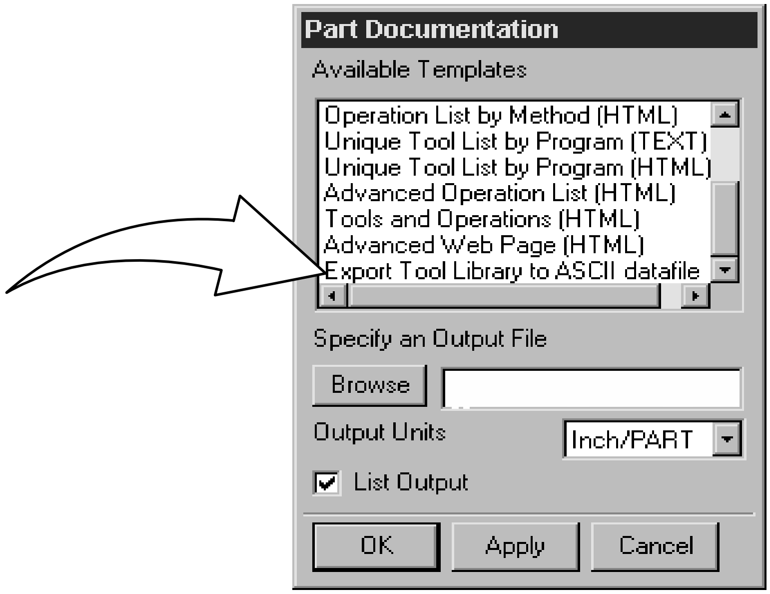
Step 3 Export the existing tool entries to an ASCII data file.

- ❑ Choose **Information** → **Shop Documentation** (or select the Shop Documentation Icon).

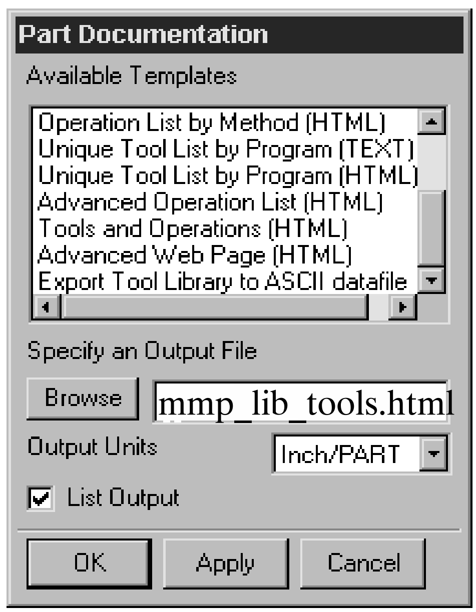
The Part Documentation Dialog is displayed.



- ❑ Select "Export Tool Library to ASCII datafile".



❑ Accept the default name.



Two files are created with a “.html” and “.dat” extension. The “.html” file is displayed in the information window when you select **OK**, the “.dat” file contains entries that you will use to insert into the **tool_database.dat** file.

❑ Choose **OK**.

Examine the listing window to see what tools have been exported to the data file. Note at the end of the listing window the location and name of the “.dat” file.

Step 4 Importing the tool data into the "tool_database.dat" file.

- ❑ Open the file, **tool_database.dat**, in your home **MACH\resource\library\tool\english** directory (Use the Notepad editor).

- ❑ Open the data file, **mmp_lib_tools.dat**, created from **Step 3,** and scroll to the area that begins with **FORMAT LIBRF**.

```
#=====================================================================
#   ASCII Database File   : lib_tools.dat
#   Creation date         : Thu May 4 14:08:00 2001
#   Unit                  : English
#   Created from Part file : lib_tools.prt
#=====================================================================
# UG 5 Parameter Mill
# This type is for legacy tools which were converted from old
# partfile tool libraries
#  LIBRF   – Tool Library Reference
#  T       – Tool Type
#  ST      – Tool SubType
#  DESCR   – Description
#  MATREF  – Tool Material Code
#  MATDES  – Tool Material Description


FORMAT LIBRF     T  ST DESCR            MATREF MATDES DIA  FN HEI ZOFF DROT FLEN TAPA
COR1 [illegible]IA HLEN HTAP HOFF
#
——+——[illegible]+—+—+————————————————+————+————+———+—+——+———+———+–
——+———+———+[illegible]——+———+———
DATA  |EM–1.250–12|02|90|Milling Tool–5 Parameters|    |    |1.25|6 |3.5|0.0 |1   |3.  |0.0 |.12 |0.0 |0.0 |0.0 |0.0
DATA  |EM–.750–06 |02|90|Milling Tool–5 Parameters|    |    |.75 |4 |3.5|0.0 |1   |3.  |0.0 |.06 |0.0 |0.0 |0.0 |0.0
DATA  |EM–.500–06 |02|90|Milling Tool–5 Parameters|    |    |.5  |4 |3. |0.0 |1   |2.75|0.0 |.06 |0.0 |0.0 |0.0 |0.0
DATA  |EM–.375–03 |02|90|Milling Tool–5 Parameters|    |    |.375|4 |2.5|0.0 |1   |2.  |0.0 |.03 |0.0 |0.0 |0.0 |0.0
DATA  |EM–1.00–50 |02|90|Milling Tool–5 Parameters|    |    |1.  |2 |2.5|0.0 |1   |2.  |0.0 |.5  |0.0 |0.0 |0.0 |0.0
DATA  |EM–1.00–06 |02|90|Milling Tool–5 Parameters|    |    |1.  |4 |3.5|0.0 |1   |3.  |0.0 |.06 |0.0 |0.0 |0.0 |0.0
```

- ❑ Select the lines beginning with **FORMAT LIBRF** and ending with **DATA |EM–1.00–06** (hold down **MB1** and drag the mouse through the lines).

- ❑ Select **MB3,** then Copy.

- ❑ Select the Notepad Window containing the contents of the file **tool_database.dat** (this file was previously opened).

- ❑ Select **MB3**, then Paste the contents into the file **tool_database.dat** (You can paste anywhere in the file, but it is suggested that you paste prior to the first **FORMAT** line).

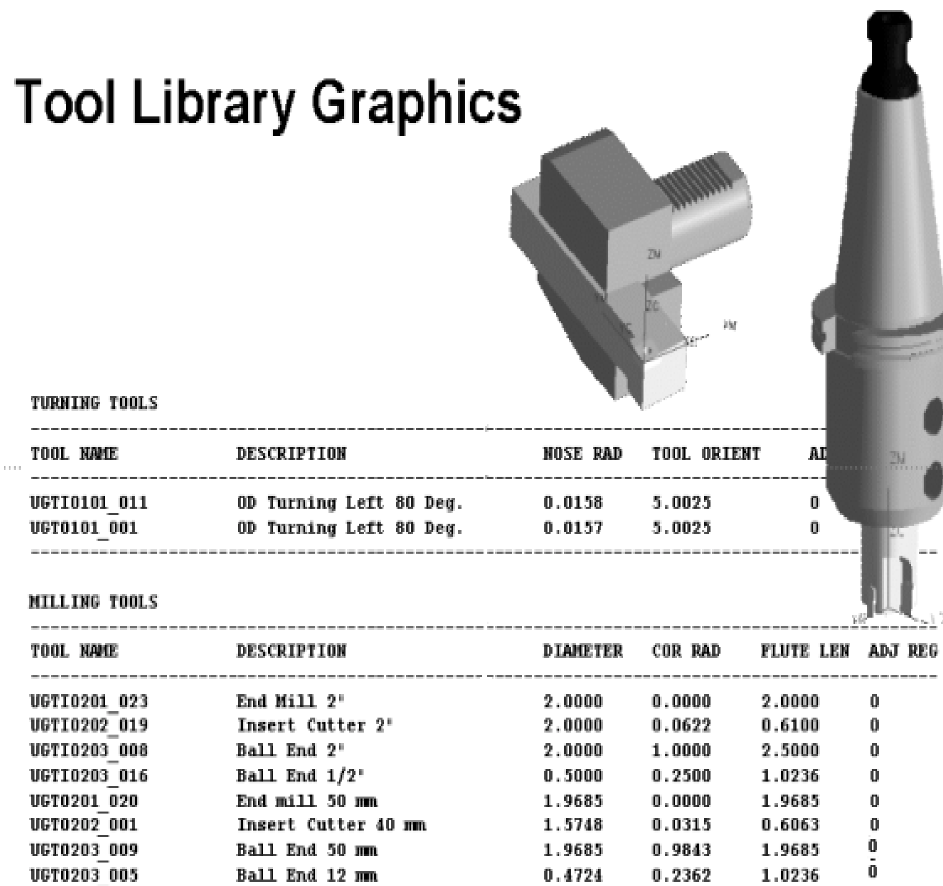
- ❑ Save the contents of the **tool_database.dat** file.

Tool Graphics Library

The Tool Graphics Library contains information related to the display of tool assemblies configured for CAM.

Ten tool assemblies, which are part file assemblies, are provided in the Advanced replay (Solid Tool) option. The Advanced replay mode displays the Tool Path Replay dialog box when an operation is Replayed. The options allow you to set preferences such as display, tool path animation and material removal. The Advanced replay option can be found in **Preferences →Operation** *Path Replay Options, Replay Mode*.

Two of the provided assemblies are for turning, the remainder apply to milling. These files are stored in the *graphics* subdirectory.



Tool Library Graphics

TURNING TOOLS

TOOL NAME	DESCRIPTION	NOSE RAD	TOOL ORIENT	AD
UGTI0101_011	OD Turning Left 80 Deg.	0.0158	5.0025	0
UGT0101_001	OD Turning Left 80 Deg.	0.0157	5.0025	0

MILLING TOOLS

TOOL NAME	DESCRIPTION	DIAMETER	COR RAD	FLUTE LEN	ADJ REG
UGTI0201_023	End Mill 2"	2.0000	0.0000	2.0000	0
UGTI0202_019	Insert Cutter 2"	2.0000	0.0622	0.6100	0
UGTI0203_008	Ball End 2"	2.0000	1.0000	2.5000	0
UGTI0203_016	Ball End 1/2"	0.5000	0.2500	1.0236	0
UGT0201_020	End mill 50 mm	1.9685	0.0000	1.9685	0
UGT0202_001	Insert Cutter 40 mm	1.5748	0.0315	0.6063	0
UGT0203_009	Ball End 50 mm	1.9685	0.9843	1.9685	0
UGT0203_005	Ball End 12 mm	0.4724	0.2362	1.0236	0

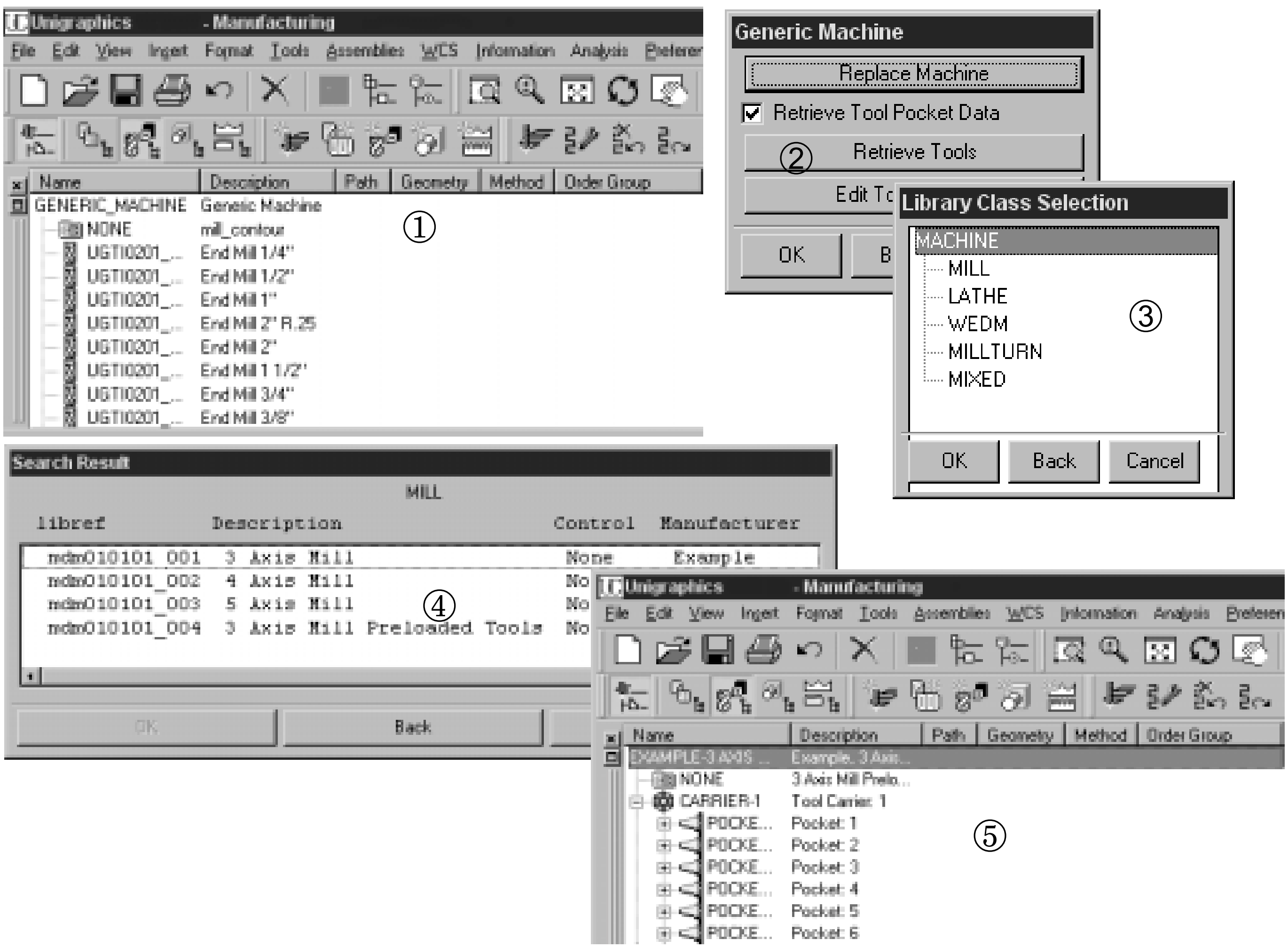
For drilling and milling the tool axis is oriented in the X+ direction. For lathe the holder is oriented in the X+ direction, the tool in the XY plane.

Machine Tool Libraries

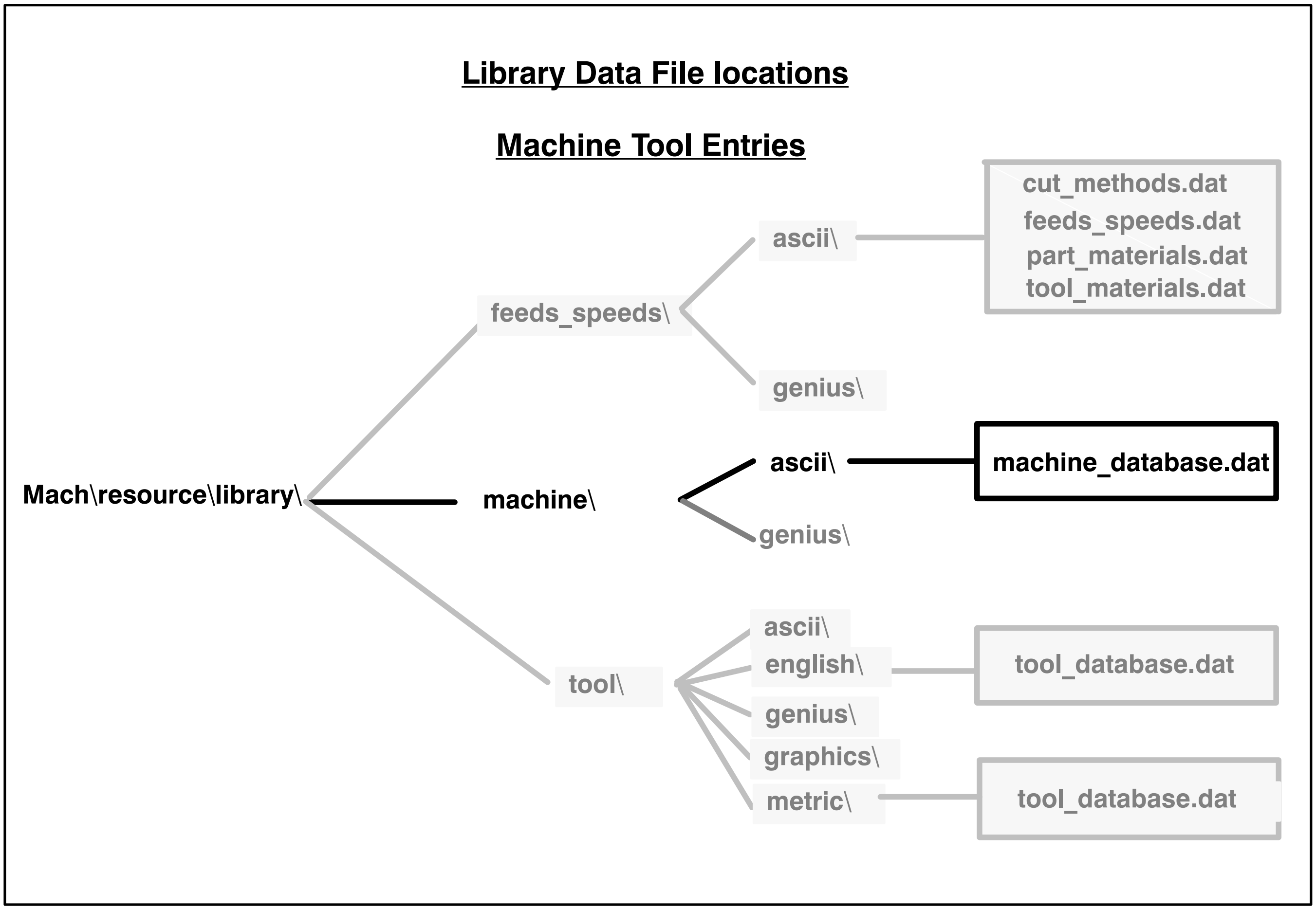
A

Machine Tool libraries contain information related to various machine tools configured for CAM. This information is used for postprocessing.

To access information from the Machine Tool libraries, from the **Machine Tool** View of the Operation Navigator, editing the Generic_Machine object (①), displays the *Generic Machine Selection* dialog box. Selection of the *Replace Machine* button (②) from this dialog box, presents the *Library Class Selection* menu (③). Selection of the Machine type creates a query to the *machine_database.dat* file (modification of this file, for the addition of your own machine entries will be explained later in this lesson) with a listing of the *Search Results* (④). Machine Tools can then be selected for later use (⑤).



Machine tool data is located in the **Mach\resource\library\machine** directory.



This directory contains the following two subdirectories:

- ascii-----contains Definition and Event Handler files for ASCII text databases. These files are associated with the Data Base Connection (DBC) and usually are ***not*** modified by the user. It also contains the **machine_database.dat** file. This file describes the various machine tools configured for CAM. Attributes within this file reference machine type, machine tool manufacturer, machine description, machine controller and postprocessor which is displayed on the Post Process Dialog.

- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.

You can add or modify machine tool information by editing the data records in the *machine_database.dat* file. Data records consist of library reference (LIBRF), machine type (T), machine tool manufacturer (MNF), description of the machine tool (DESCR), machine controller (CNTR) and the postprocessor data file. Specifying the postprocessor data file specifies a specific post for that machine tool.

Machine Tool Library Format

```
## The following key words for Attribute ids are defined
## LIBRF   – unique record identifier
## T       – Machine type – 1–Mill machines
##                        – 2–Lathe machines
##                        – 3–Wedm machines
##                        – 9–Mixed machines
## MNF     – Manufacturer
## DESCR   – Short description ( for example 3 Axis Mill)
## CNTR    – indicating the controller of the machine
## POST    – the configuration file name with the list of postprocessors for this machine
##          (The path will be found from the search path environment variable)
#####################################################################################

FORMAT LIBRF          T   MNF     DESCR                 CNTR   POST
DATA|mdm010101_001|1|Example|3AxisMill|None|${UGII_CAM_POST_DIR}mill_3_axis.dat
DATA|mdm010101_002|1|Example|4AxisMill|None|${UGII_CAM_POST_DIR}mill_4_axis.dat
DATA|mdm010101_003|1|Example|5AxisMill|None|${UGII_CAM_POST_DIR}mill_5_axis.dat
DATA|mdm010101_004|1|Example|3AxisMillPreloaded-
Tools|None|${UGII_CAM_POST_DIR}mill3ax_preloaded.dat
DATA|mdm010102_001|2|Example|2AxisLathe|None|${UGII_CAM_POST_DIR}lathe_2_axis.dat
DATA|mdm010103_001|3|Example|2AxisWireEDM|None|${UGII_CAM_POST_DIR}wedm.dat
DATA|mdm010109_001|9|Example|GenericMachine|None|${UGII_CAM_POST_DIR}tem-
plate_post.dat
```

Activity A—3: Machine Tool Libraries

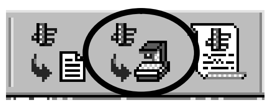
In this activity you will become familiar with the procedure to access Machine Tool data from CAM Libraries. You will see how this library is used in conjunction with the Post Process dialog by replacing the Available Machines with a 5-axis postprocessor.

Step 1 Open the part file.

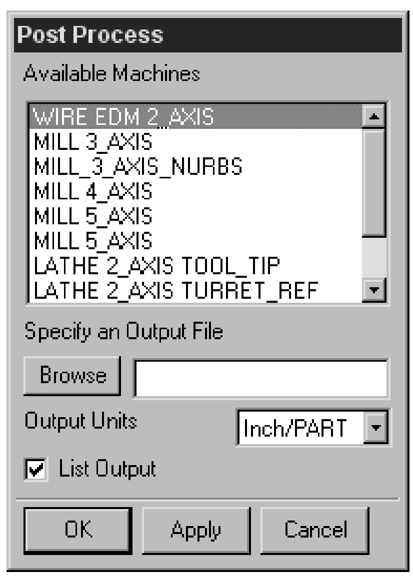
- ❑ Open the part file, **mmp_lib_function.prt.**
- ❑ If necessary, enter the **Manufacturing Application.**

Step 2 Review available machines in the Post Process Dialog.

- ❑ Select the Post Process Icon**.**



The Post Process Dialog is displayed.



- ❑ Choose **Cancel** or **Ok.**

A

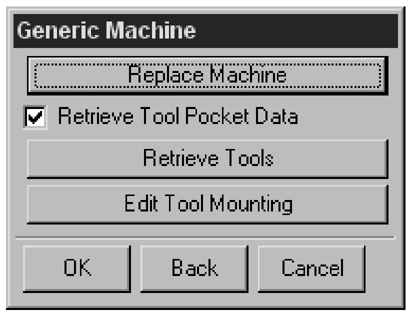
Step 3 Change the Post Process Dialog to show a 5-axis machine tool only.

❑ If necessary change the Operation Navigator to **Machine Tool** view.

❑ Highlight the GENERIC_MACHINE, then **MB3,** and select edit.

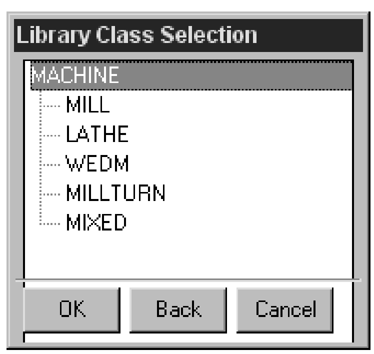
The Generic Machine dialog is displayed.

❑



❑ Select the Replace Machine button.

The Library Class Selection Menu is displayed.

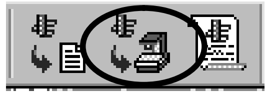


❑ Highlight MILL, then select **OK**.

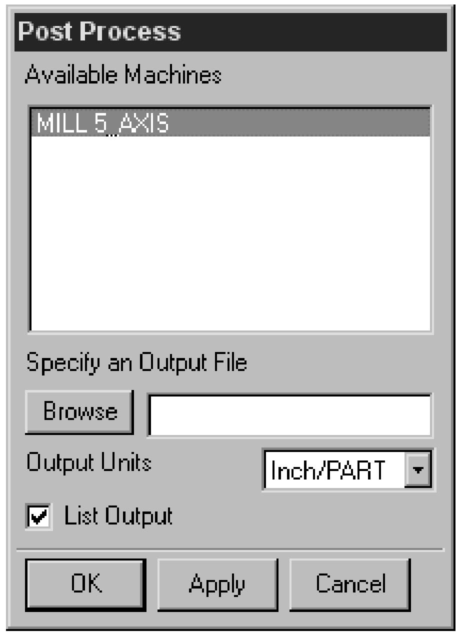
The Search Result listing is displayed.

❑ Highlight **mdm010101_003 5-Axis Mill**, then select **OK** until you are back to the Create dialog.

❑ Select the Post Process Icon**.**



The Post Process Dialog is displayed.



Notice that only the MILL_5_AXIS machine is displayed in the Post Process dialog..

❑ Choose the **Cancel** button**.**

You are finished with this activity and will be using this part file in the next activity.

Notice that only the MILL_5_AXIS machine is displayed in the Post Process dialog..

❑ Choose the **Cancel** button**.**

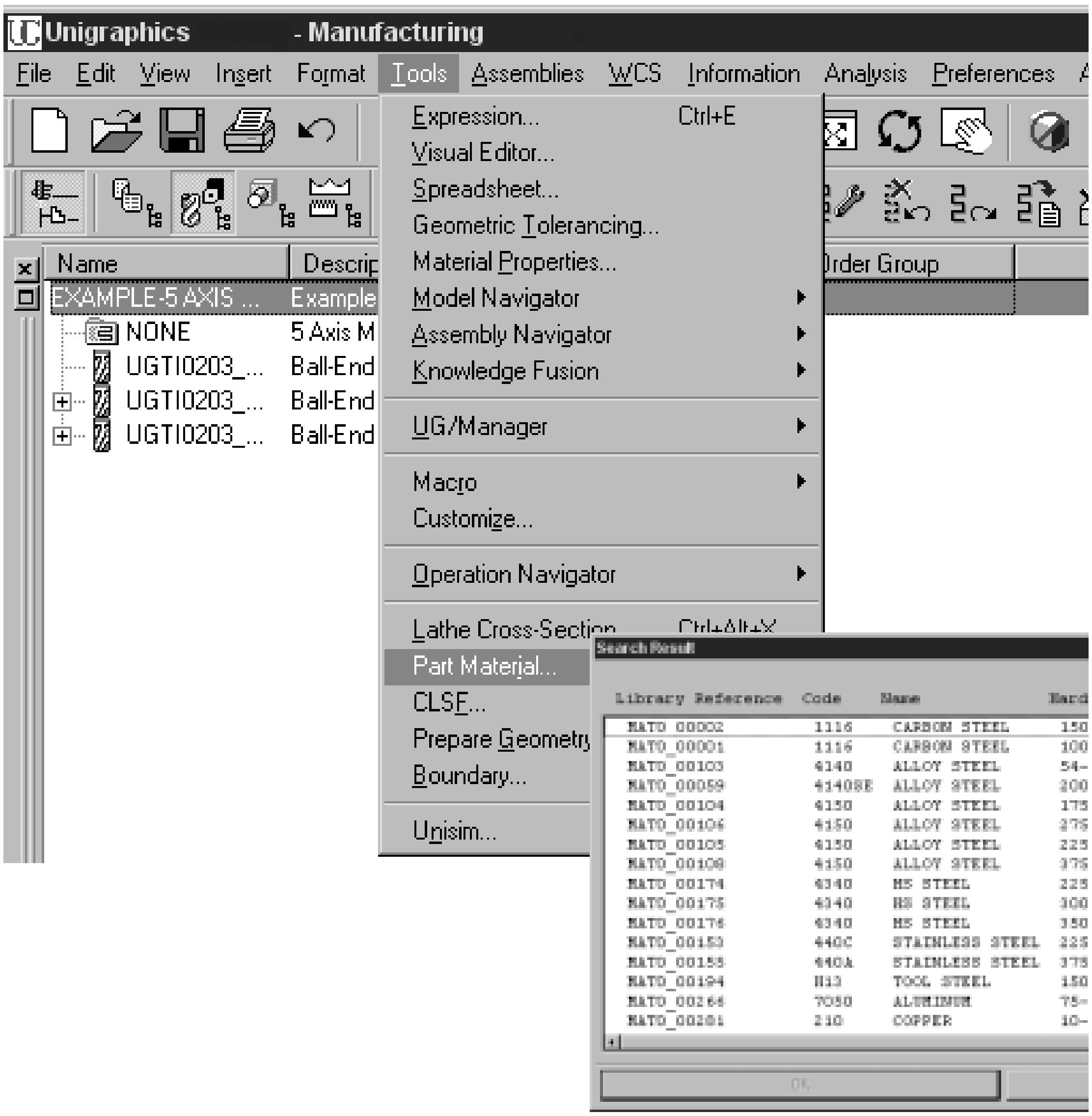
You are finished with this activity and will be using this part file in the next activity.

Part Material Libraries

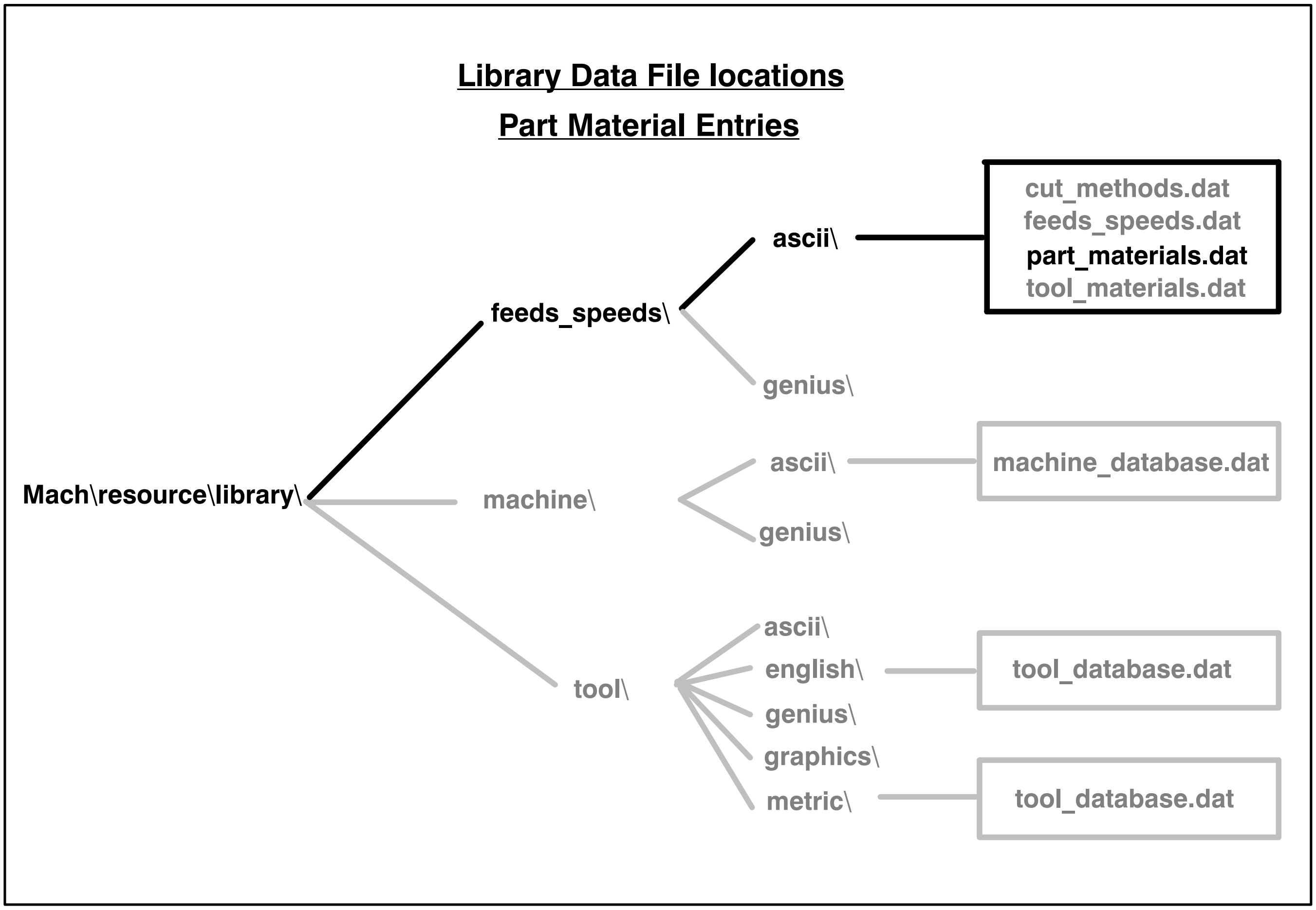
A

Part Material libraries contains information which is used in the calculation of feeds and speeds. This is *not* the same as Part Material used in Modeling.

To select the Part Material for a Set-up, select from the menu bar **Tools→Part Material**. Select the desired Part Material from the *Search Results* list.



Part Material can also be selected for individual Geometry Groups. Part Material data is located in the **Mach\resource\library\feeds_speeds** \ directory.



This directory contains the following two subdirectories:

- ascii-----contains Definition and Event Handler files for ASCII text databases. These files are associated with the Data Base Connection (DBC) and usually are ***not*** modified by the user. It also contains the **part_materials.dat** file. This file defines part material used in the calculation of feeds and speeds.
- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.

You can add or modify Part Material information by editing the data records in the *part_materials.dat* file. Data records consist of material code (MATCODE), material name (MATNAME), material description (PARTMAT), material hardness (HARDNESS) and part material library reference (LIBREF).

A

Part Materials Library Format

```
## The following key words for Attribute ids are defined
##  MATCODE   material_code         – Material Code
##  MATNAME   material_name         – Material Name (appears on the label)
##  PARTMAT   material_description  – Material Description
##  HARDNESS  material_hardness     – Material Hardness
##  LIBREF    partmaterial_libref   – Unique record identifier
##                                    (Library Reference)
#—————————————+——————————————————————————————————+—————————————

FORMAT MATCODE MATNAME PARTMAT HARDNESS LIBRF

#—————————————+——————————————————————————————————+—————————————
DATA|1116|CARBON STEEL|FREE MACHINING CARBON STEELS,WROUGHT – Low Carbon Resulfurized|150–200|MAT0_00002
DATA|1116|CARBON STEEL|FREE MACHINING CARBON STEELS,WROUGHT– Low Carbon Resulfurized|100–150|MAT0_00001
DATA|4140|ALLOY STEEL|ALLOY STEELS,WROUGHT – Medium Carbon|54–56|MAT0_00103
DATA|4140SE|ALLOY STEEL|FREE MACHINING ALLOY STEELS, WROUGHT – Medium Carbon Resulfurized|200–250 |MAT0_00059
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT – Medium Carbon|175–225|MAT0_00104
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT – Medium Carbon|275–325|MAT0_00106
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT – Medium Carbon|225–275|MAT0_00105
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT – Medium Carbon|375–425|MAT0_00108
DATA|4340|HS STEEL|HIGH STRENGTH STEELS, WROUGHT –|225–300|MAT0_00174
DATA|4340|HS STEEL|HIGH STRENGTH STEELS, WROUGHT –|300–350|MAT0_00175
DATA|4340|HS STEEL|HIGH STRENGTH STEELS, WROUGHT –|350–400|MAT0_00176
DATA|440C|STAINLESS STEEL|STAINLESS STEELS, WROUGHT – Martensitic|225–275 HB|MAT0_00153
DATA|440A|STAINLESS STEEL|STAINLESS STEELS, WROUGHT – Martensitic|375–425 HB|MAT0_00155
DATA|H13|TOOL STEEL|TOOL STEELS, WROUGHT – Hot Work|150–200 HB |MAT0_00194
DATA|7050|ALUMINUM|ALUMINUM ALLOYS, WROUGHT –|75–150 HB |MAT0_00266
DATA|210|COPPER|COPPER ALLOYS|10–70  HRB |MAT0_00281
```

Activity A–4: Part Materials Libraries

In this activity you will become familiar with accessing Part Material from the Library.

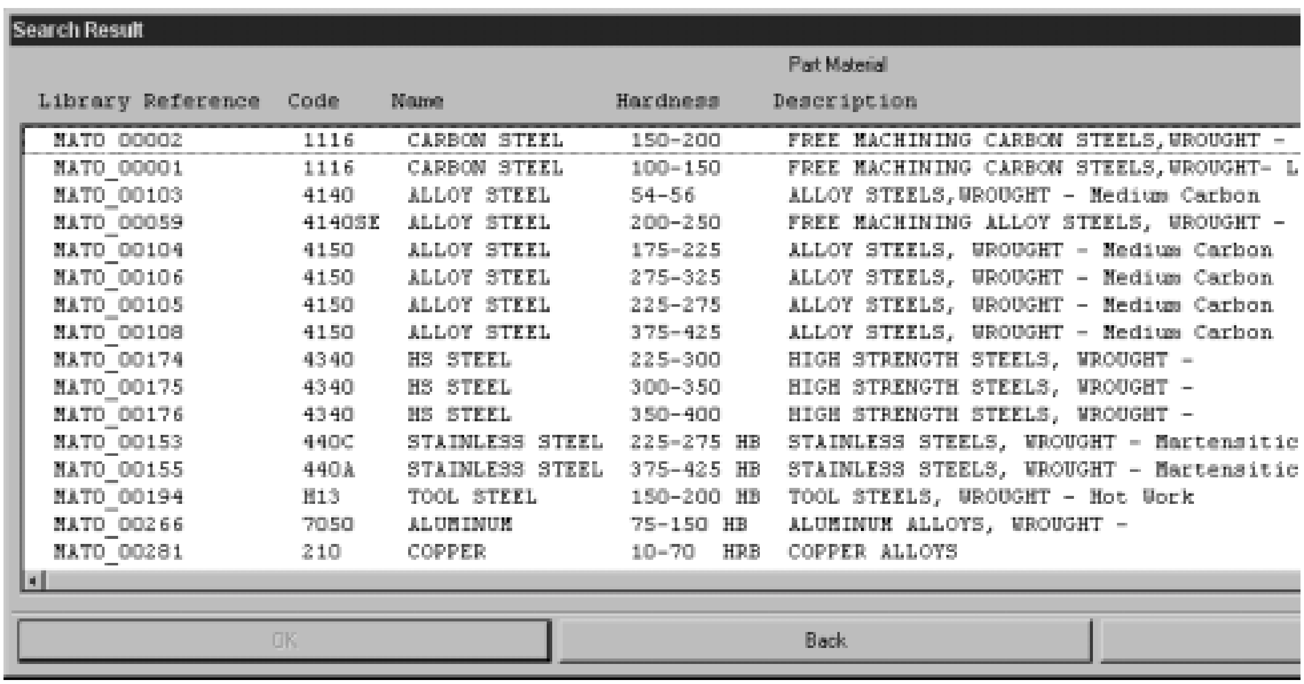
Step 1 List all part materials which are available for selection.

❑ Continue with the opened part, **mmp_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

❑ Select **Tools** → **Part Material** from the menu bar.

The Search Results List is displayed.


Search Result

Part Material

Library Reference	Code	Name	Hardness	Description
MAT0_00002	1116	CARBON STEEL	150-200	FREE MACHINING CARBON STEELS,WROUGHT -
MAT0_00001	1116	CARBON STEEL	100-150	FREE MACHINING CARBON STEELS,WROUGHT- L
MAT0_00103	4140	ALLOY STEEL	54-56	ALLOY STEELS,WROUGHT - Medium Carbon
MAT0_00059	4140SE	ALLOY STEEL	200-250	FREE MACHINING ALLOY STEELS, WROUGHT -
MAT0_00104	4150	ALLOY STEEL	175-225	ALLOY STEELS, WROUGHT - Medium Carbon
MAT0_00106	4150	ALLOY STEEL	275-325	ALLOY STEELS, WROUGHT - Medium Carbon
MAT0_00105	4150	ALLOY STEEL	225-275	ALLOY STEELS, WROUGHT - Medium Carbon
MAT0_00108	4150	ALLOY STEEL	375-425	ALLOY STEELS, WROUGHT - Medium Carbon
MAT0_00174	4340	HS STEEL	225-300	HIGH STRENGTH STEELS, WROUGHT -
MAT0_00175	4340	HS STEEL	300-350	HIGH STRENGTH STEELS, WROUGHT -
MAT0_00176	4340	HS STEEL	350-400	HIGH STRENGTH STEELS, WROUGHT -
MAT0_00153	440C	STAINLESS STEEL	225-275 HB	STAINLESS STEELS, WROUGHT - Martensitic
MAT0_00155	440A	STAINLESS STEEL	375-425 HB	STAINLESS STEELS, WROUGHT - Martensitic
MAT0_00194	H13	TOOL STEEL	150-200 HB	TOOL STEELS, WROUGHT - Hot Work
MAT0_00266	7050	ALUMINUM	75-150 HB	ALUMINUM ALLOYS, WROUGHT -
MAT0_00281	210	COPPER	10-70 HRB	COPPER ALLOYS

Step 2 Select the desired part material.

❑ Select **MAT0_00105** from the Search Results list

❑ Choose **OK.**

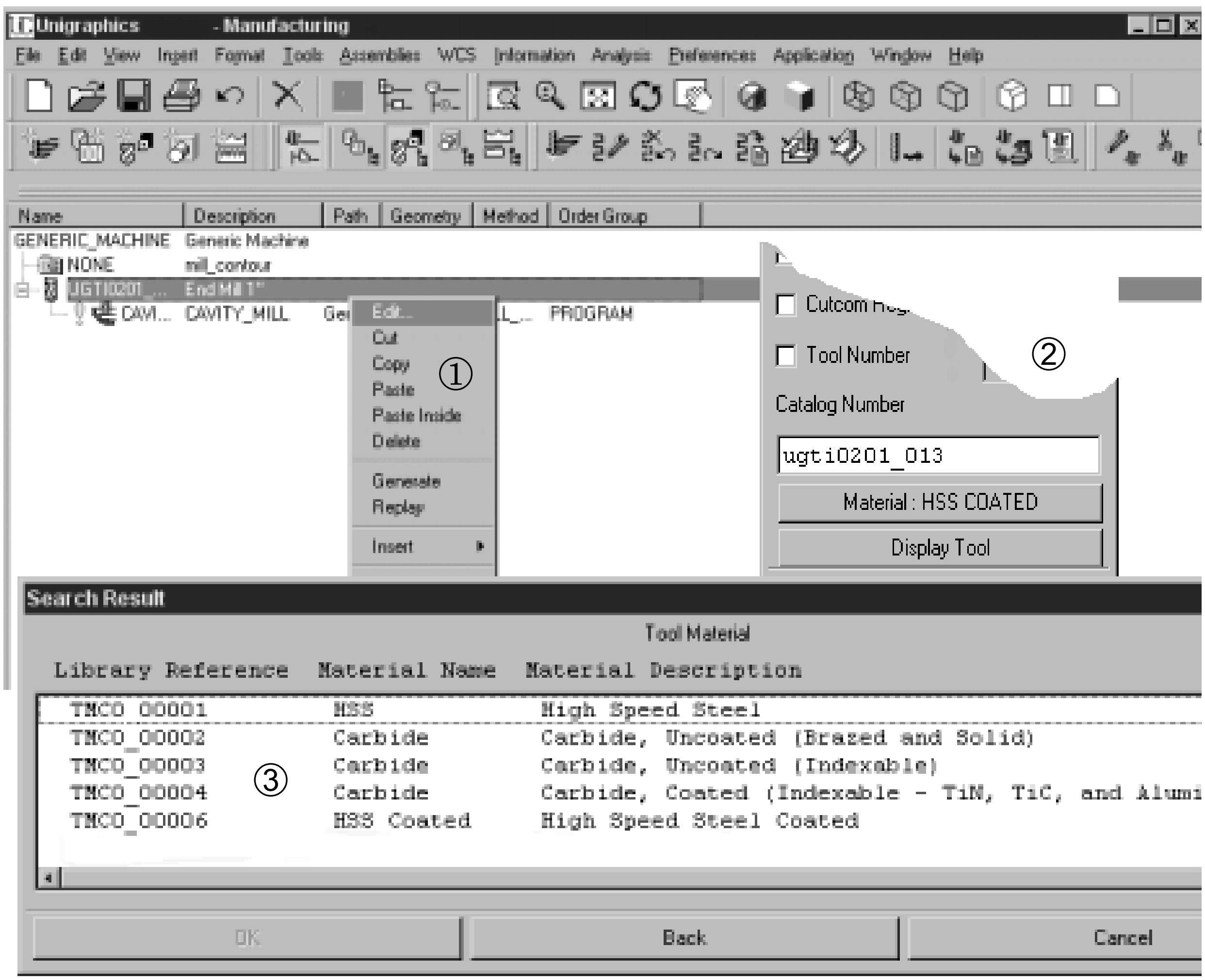
You are finished with this activity and will be using this part file in the next activity.

Cutting Tool Material Libraries

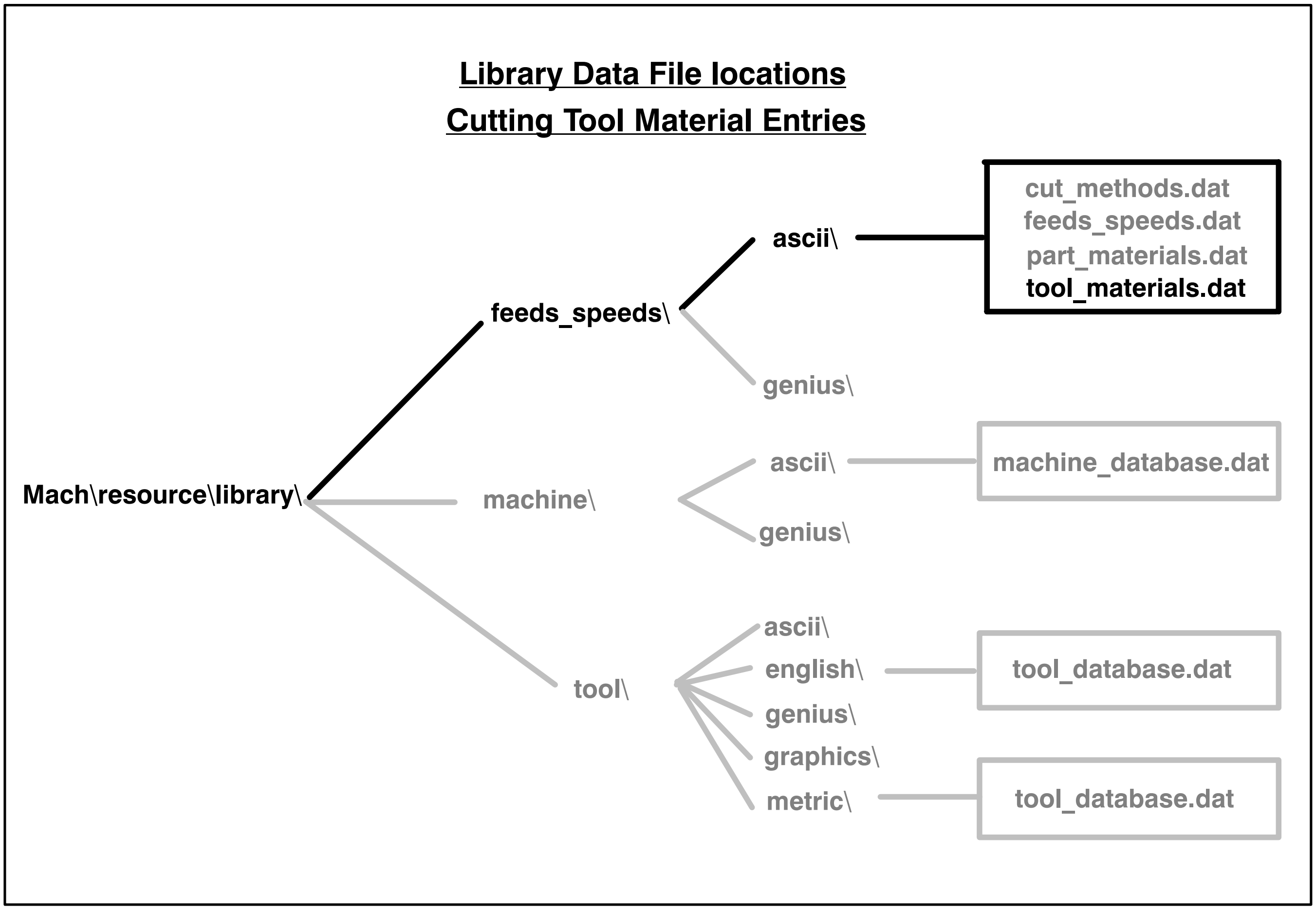
A

Cutting Tool Material libraries contains information which pertains to the cutting tool material type which is used in the calculation of feeds and speeds.

To access information from the Cutting Tool libraries, in the **Machine Tool** View of the Operation Navigator, editing any tool object (①) displays the *Tool Parameter* dialog box. Selection of the *Material:* button (②) from this dialog box, creates a query of the *tool_materials.dat* file (modification of this file, for the addition of your own tool materials entries will be explained later in this lesson) with a listing of the *Search Results*. Cutting Tool Material can then be selected from this list (③).



Cutting Tool Material data is located in the **Mach\resource\library \feeds_speeds** directory.



This directory contains the following two subdirectories:

- ascii-----contains Definition and Event Handler files for ASCII text databases. These files are associated with the Data Base Connection (DBC) and usually are ***not*** modified by the user. It also contains the **tool_materials.dat** file. This file describes cutting tool material which is used in feed and speed calculations. This file is used in conjunction with the **tool_database.dat** file which is used for the definition of cutting tools.

- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.

You can add or modify Cutting Tool Material information by editing the data records in the *tool_materials.dat* file. Data records consist of cutting material code (LIBREF), material name (MATNAM) and material description (MATDESC).

A

Tool Materials Library Format

```
##  The following key words for Attribute ids are defined
##
##  LIBREF   material_code        – Unique record identifier
##                                  (Library Reference)
## MATNAM   material_name      – Material Name  (appears on the label)
## MATDESC material_description – Material Description
###############################################################################
#————————————+——————————————————————+——————————
FORMAT LIBRF MATNAM MATDESC
#————————————+——————————————————————+——————————
DATA|TMC0_00001|HSS|High Speed Steel
DATA|TMC0_00002|Carbide|Carbide, Uncoated (Brazed and Solid)
DATA|TMC0_00003|Carbide|Carbide, Uncoated (Indexable)
DATA|TMC0_00004|Carbide|Carbide, Coated (Indexable – TiN, TiC, and Aluminum Oxide)
DATA|TMC0_00006|HSS Coated|High Speed Steel Coated
```

Activity A—5: Cutting Tool Materials Libraries

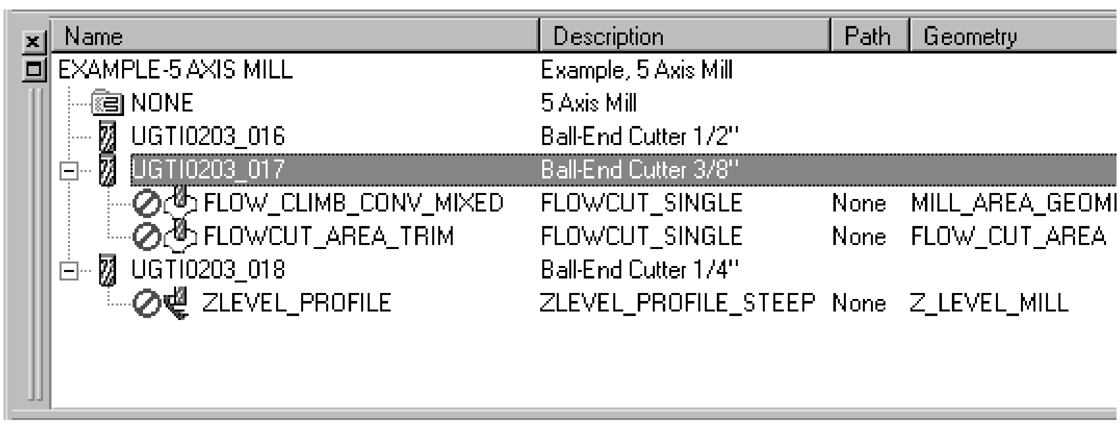
In this activity, you will become familiar with the Cutting Tool Materials data. Cutting Tool Materials are used in the calculation of feeds and speeds.

Step 1 Accessing the Cutting Tool Material library.

❑ Continue with the opened part, **mmp_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

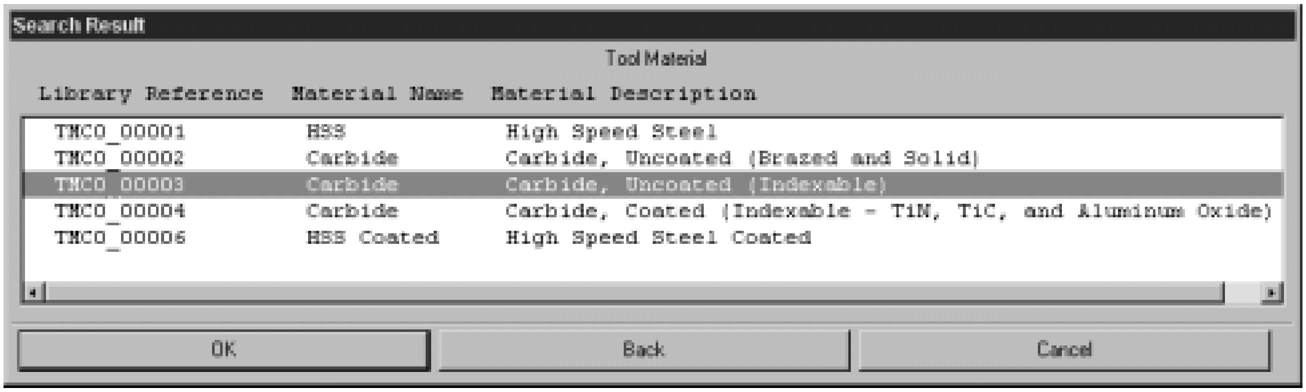
❑ If necessary, change the view of the Operation Navigator to the **Machine Tool** view.



❑ Highlight **UGTI0203_017**, select **MB3**, then Edit.

The Cutting Tool Parameter List is displayed.

❑ Select the **Material** button.



The Cutting Tool Material Search List is displayed.

❑ Select the **TMC0_00003 Carbide, Uncoated (Indexable)** as the material type.

❑ Choose **OK** until your are returned to the Create Operation Menu.

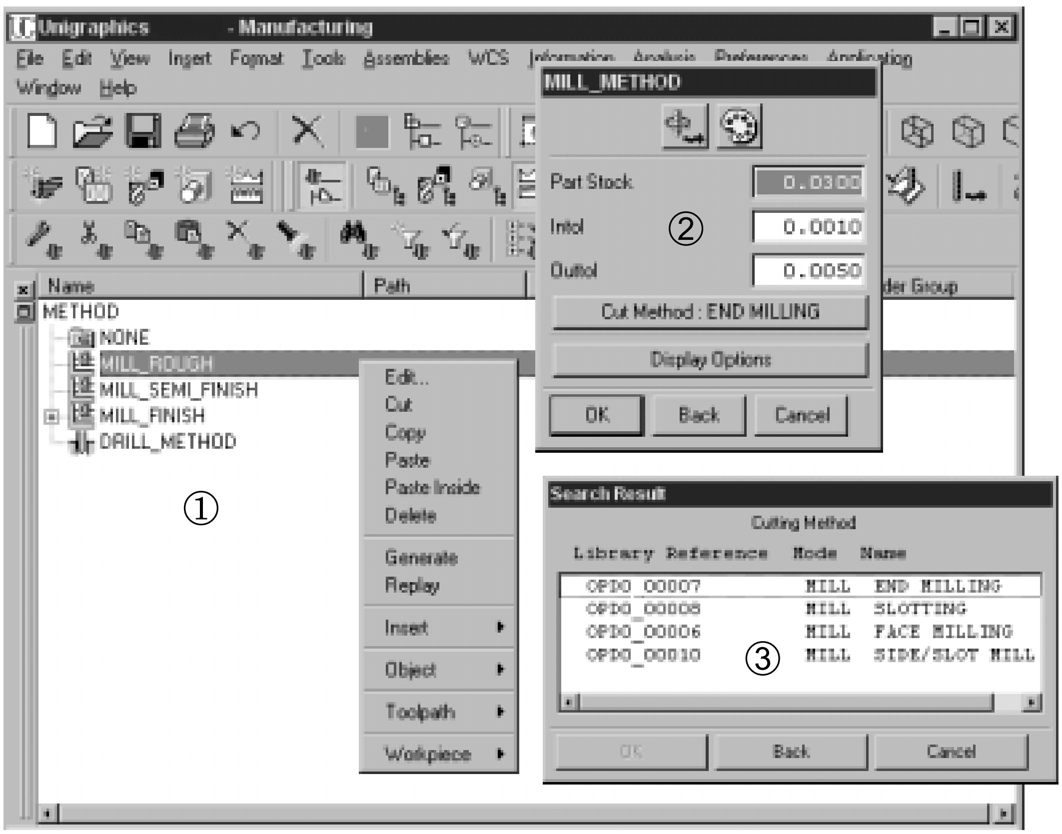
You are finished with this activity and will be using this part file in the next activity.

Cut Method Libraries

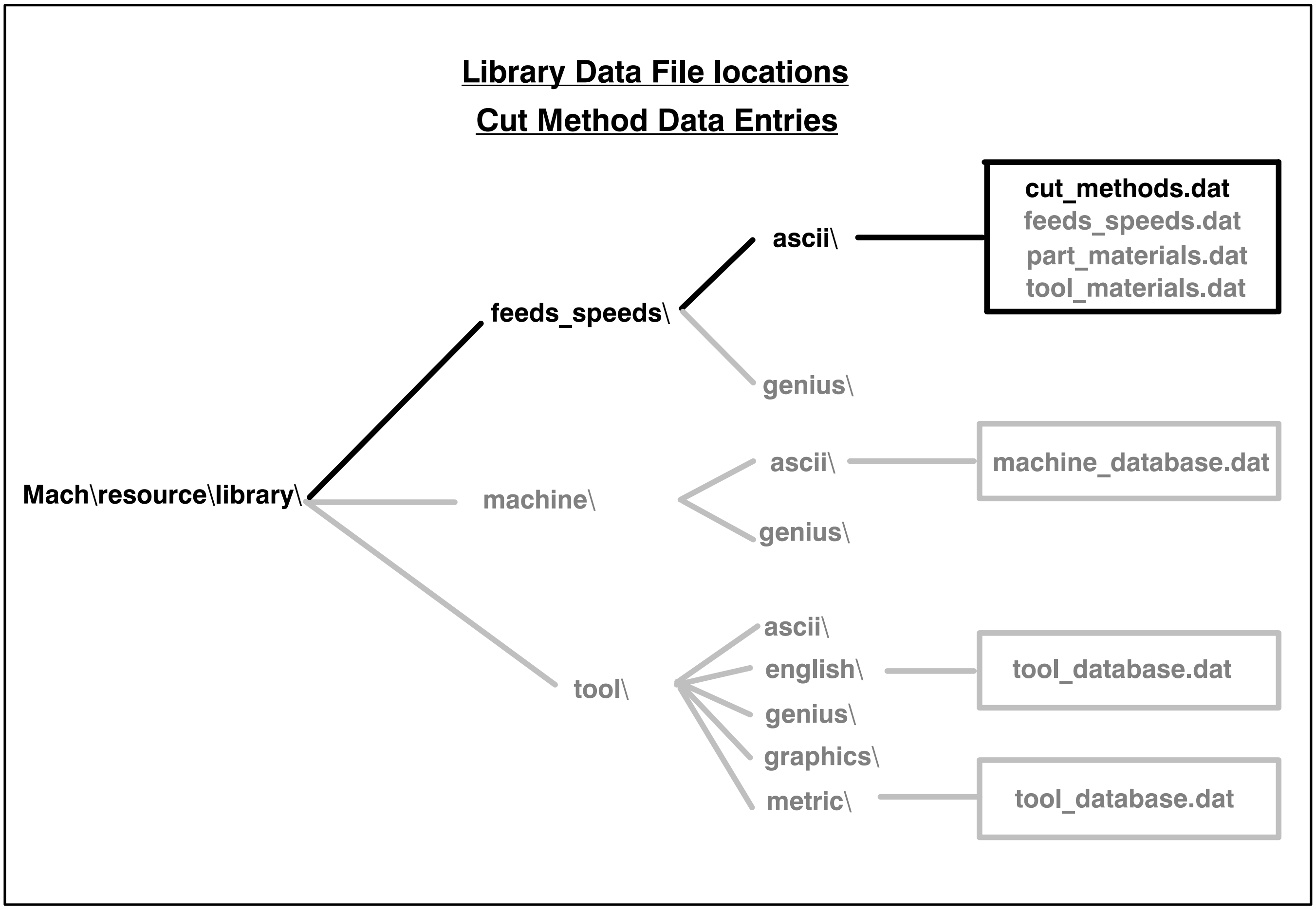
A

Cut Method libraries contain information which pertains to the Cut Method type and is used in the calculation of speeds and feeds.

To access information from the Cut Method libraries, from the **Machining Method** View of the Operation Navigator, editing any of the method objects (①) displays the *Method* dialog box. Selection of the *Cut Method:* button (②) from this dialog box, creates a query of the *cut_methods.dat* file with a listing of the *Search Results.* A Cut Method can then be selected from this list (③).



Cut Method data is located in the **Mach\resource\library \feeds_speeds** directory.



This directory contains the following two subdirectories:

- ascii-----contains Definition and Event Handler files for ASCII text databases. These files are used for the Data Base Connection (DBC) and usually are ***not*** modified by the user. It also contains the **cut_methods.dat** file. This file describes the “Cut Method” used for the Machining Method in UG/CAM. The library reference is used for feed and speed calculations.
- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.

Cut Methods Library Format

 # ————–+————————————————————+——————————————
FORMAT LIBRF MODE NAME DESCRIPTION
 # —————+————————————————————+——————————————
DATA|OPD0_00002|DRILL|BORING|0
DATA|OPD0_00011|DRILL|DRILLING|0
DATA|OPD0_00007|MILL|END MILLING|0
DATA|OPD0_00008|MILL|SLOTTING|0
DATA|OPD0_00006|MILL|FACE MILLING|0
DATA|OPD0_00010|MILL|SIDE/SLOT MILL|0
DATA|OPD0_00003|LATHE|TURN, CUTOFF|0
DATA|OPD0_00001|LATHE|TURN, POINT|0

A

Activity A–6: Cut Methods Libraries

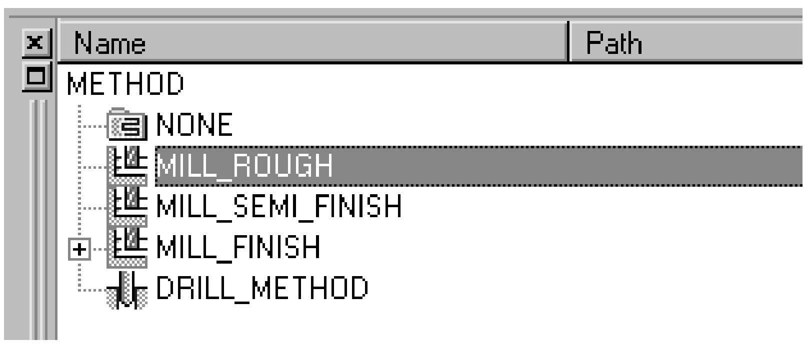
In this activity you will become familiar with the Cut Methods library.

Step 1 Accessing the Cut Method library.

❑ Continue with the opened part, **mpp_lib_function.prt.**

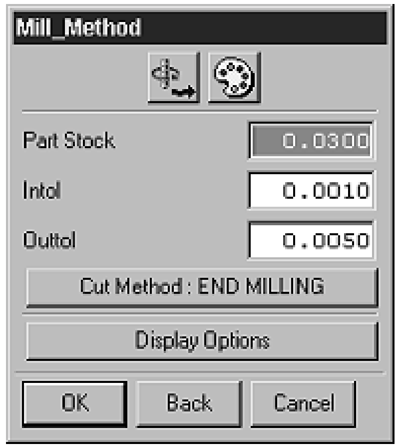
❑ If necessary, enter the **Manufacturing Application.**

❑ If necessary, change the view of the Operation Navigator to the **Machine Method** view.

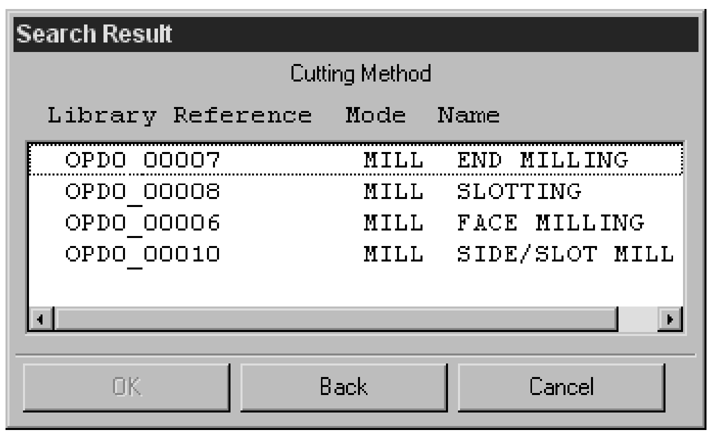


❑ Highlight **MILL_ROUGH**, select **MB3**, then Edit.

The Mill_Method dialog is displayed.



❑ Select the **Cut Method** button.



The Cutting Method Search List is displayed.

❑ Select the **OPD0_00010 MILL SIDE/SLOT MILL method.**

❑ Choose **OK** and notice the label on the **Cut Method** button.

❑ Close the part.

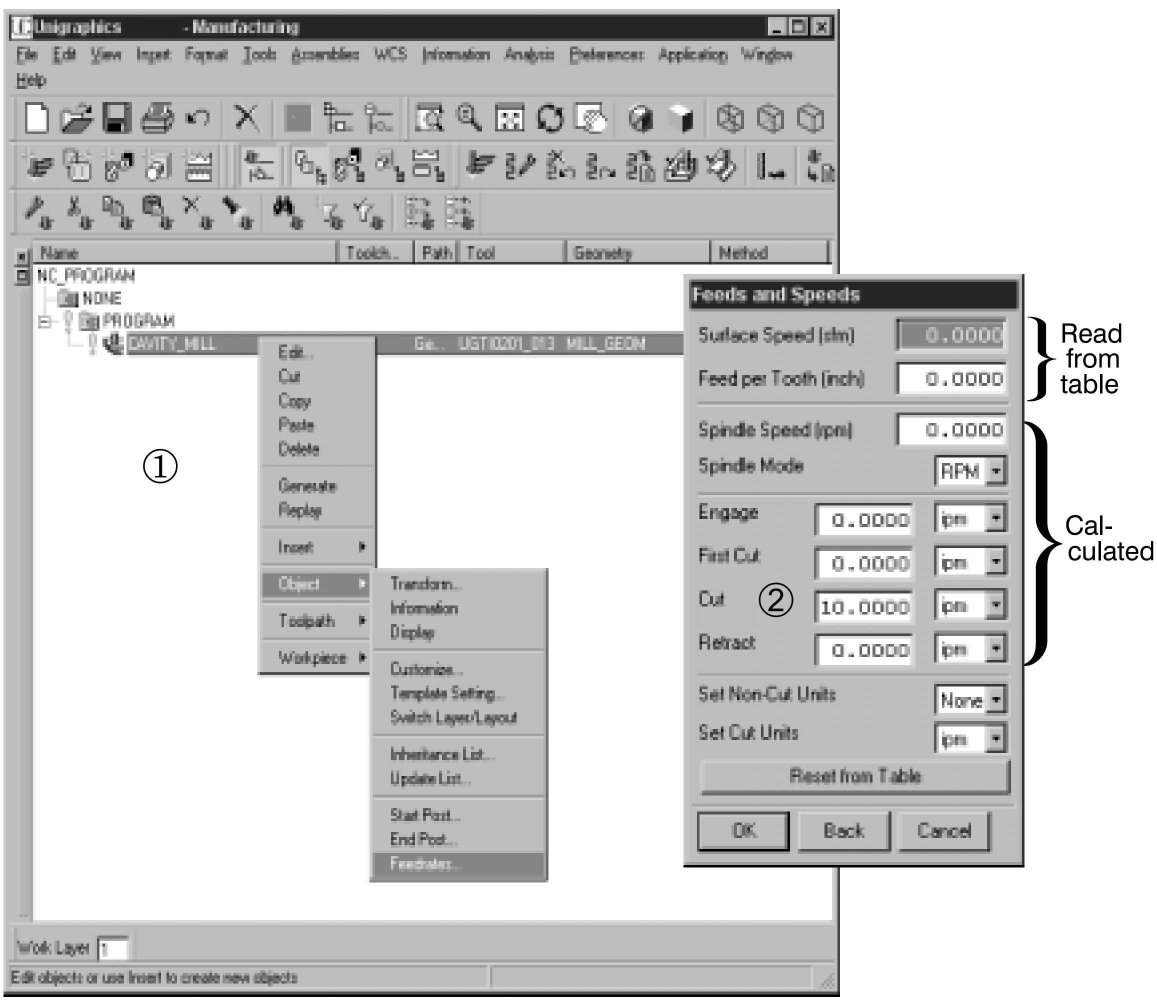
You are finished with this activity.

Feeds and Speeds Libraries

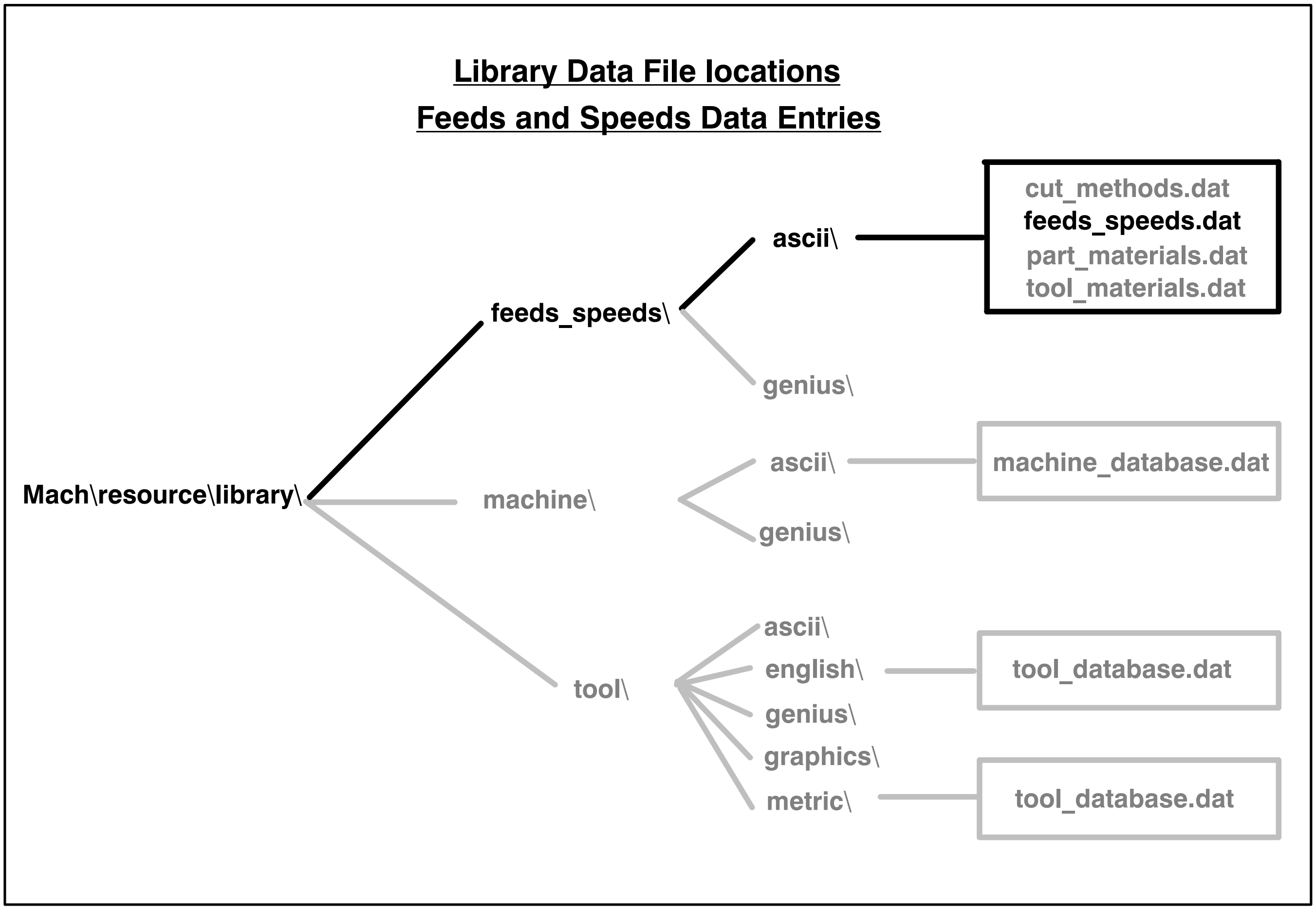
A

Feeds and Speeds libraries contains information which pertains to feeds and speeds used in the generation of an Operation.

Feeds and speeds information can be accessed while editing an Operation or from any of the Operation Navigator views. Select or highlight the Operation, use **MB3** and choose **Object** → **Feedrates** (①). The *Feeds and Speeds* dialog box is displayed. Selection of the *Reset from Table* button (②) from this dialog box, will calculate the feeds and speeds based on data obtained from the *feeds_speeds.dat* file, part material, tool material, cut method chosen for the operation (modification of this file will be explained later in this lesson) and Depth of Cut.



Feeds and Speeds data are located in the **Mach\resource\library \feeds_speeds** directory.



This directory contains the following two subdirectories:

- ascii-----contains Definition and Event Handler files for ASCII text databases. These files are associated with the Data Base Connection (DBC) and usually are ***not*** modified by the user. It also contains the **feeds_speeds.dat** file. This file is used for defining feeds and speeds in an Operation.

- genius---contains the Definition and Event Handler files for Genius databases. These files are ***not*** modified by the user.

A

You can add or modify Feeds and Speeds information by editing the data records in the *feeds_speeds.dat* file. Data records consist of (LIBREF), Cut Method Library reference (OPERTYPE), Part Material Library reference (PARTMAT), Tool Material Library reference (TOOLMAT), Depth of Cut (DPT_CUT_IN or DPT_CUT_MM), Surface Speed (SURF_SPEED_FPM or SURF_SPEED_MPM) and Feed per Tooth (FEED_IPT or FEED_MMPT).

When adding entries for Feeds and Speeds, be sure that the Library Reference for the Part Material (located in part_materials.dat), Tool Material (located in tool_materials.dat) and Cut Method (located in cut_methods.dat) exist, are unique and of the correct type.

Be sure to reset the configuration to force the update of the files that have been modified.

Feeds and Speeds Library Format

```
# LIBRF          – Unique record identifier
#                  (Library Reference)
# OPERTYPE    – cutmthd_libref              Cut Method Library Reference
# PARTMAT     – part_material_libref Part   Part Material Library Reference
# TOOLMAT     – tool_material_libref        Tool Material Library Reference
# DPT_CUT_IN  – dpth_of_cut                 Depth_of_cut(inch)
# DPT_CUT_MM – dpth_of_cut                  Depth_of_cut(mm)
# SURF_SPEED_FPM  – surface_speed           Suface Speed(FPM)
# SURF_SPEED_MPM  – surface_speed           Suface Speed(MPM)
# FEED_IPT       – feed_per_tooth           Feed per Tooth(IPT)
# FEED_MMPT    – feed_per_tooth             Feed per Tooth(MMPT)
#———————————————————————————————————————————————

FORMAT LIBRF OPERTYPE PARTMAT TOOLMAT DPT_CUT_IN DPT_CUT_MM IN-
DEX1 INDEX2 SURF_SPEED_FPM SURF_SPEED_MPM FEED_IPT FEED_MMPT

#———————————————————————————————————————————————
DATA|FSDO_00001|OPD0_00001|MAT0_00001|TMC0_00001|.040|1.|||200.|60.8|.007|0.1778
DATA|FSDO_00002|OPD0_00001|MAT0_00001|TMC0_00001|.150|4.|||150.|45.6|.015|0.381
```

Activity A–7: Feeds and Speeds

In this activity, you will set the options necessary for system generated feeds and speeds. You will then change some of these settings to see how they affect feeds and speeds which are calculated by the system.

Step 1 Opening the part file.

❑ Open the part file **mmp_lib_act_feeds_speeds.prt** and then rename it to (***)**_lib_act_feeds_speeds.prt**.

❑ Enter the **Manufacturing Application.**

The Operation Navigator is displayed.

Step 2 Defining the Part Material.

You only need to define the Part material once.

❑ Change the Operation Navigator to the **Geometry View**.

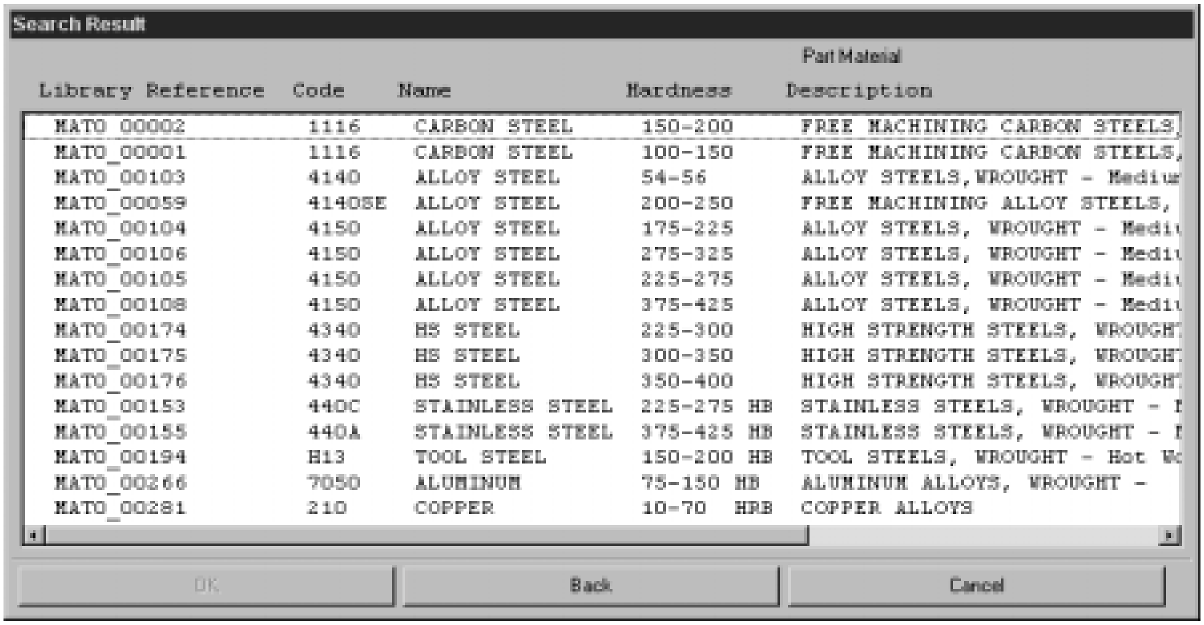
❑ Highlight **Workpiece**, then **MB3,** Edit.

The Mill_Geom dialog is displayed.

❑ Choose the **Material** button.

The Search Result window is displayed. You will select the material type from here.

A



Library Reference	Code	Name	Hardness	Description
MATO_00002	1116	CARBON STEEL	150-200	FREE MACHINING CARBON STEELS,
MATO_00001	1116	CARBON STEEL	100-150	FREE MACHINING CARBON STEELS,
MATO_00103	4140	ALLOY STEEL	54-56	ALLOY STEELS,WROUGHT - Mediur
MATO_00059	4140SE	ALLOY STEEL	200-250	FREE MACHINING ALLOY STEELS,
MATO_00104	4150	ALLOY STEEL	175-225	ALLOY STEELS, WROUGHT - Mediu
MATO_00106	4150	ALLOY STEEL	275-325	ALLOY STEELS, WROUGHT - Mediu
MATO_00105	4150	ALLOY STEEL	225-275	ALLOY STEELS, WROUGHT - Mediu
MATO_00108	4150	ALLOY STEEL	375-425	ALLOY STEELS, WROUGHT - Mediu
MATO_00174	4340	HS STEEL	225-300	HIGH STRENGTH STEELS, WROUGHT
MATO_00175	4340	HS STEEL	300-350	HIGH STRENGTH STEELS, WROUGHT
MATO_00176	4340	HS STEEL	350-400	HIGH STRENGTH STEELS, WROUGHT
MATO_00153	440C	STAINLESS STEEL	225-275 HB	STAINLESS STEELS, WROUGHT - I
MATO_00155	440A	STAINLESS STEEL	375-425 HB	STAINLESS STEELS, WROUGHT - I
MATO_00194	H13	TOOL STEEL	150-200 HB	TOOL STEELS, WROUGHT - Hot Wo
MATO_00266	7050	ALUMINUM	75-150 HB	ALUMINUM ALLOYS, WROUGHT -
MATO_00281	210	COPPER	10-70 HRB	COPPER ALLOYS

❑ Select **Aluminum** from the list and then choose **OK** until the Create Operation dialog is displayed.

Step 3 Defining the Cut Method.

You will now define the Cut Method by editing the Mill Rough Method.

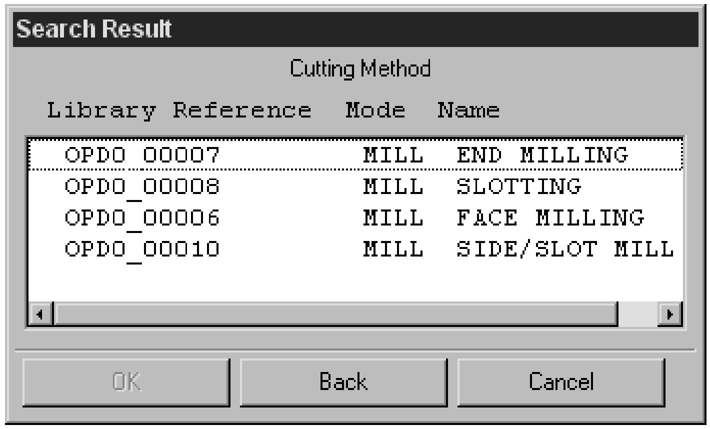
❑ Change the Operation Navigator to the **Method View**.

❑ Highlight **Workpiece**, then **MB3,** Edit.

The Mill Method dialog is displayed.

❑ Choose the **Cut Method** button.

The Search Result dialog is displayed.



From this dialog, you can choose the type of cutting.

❑ Select **End Milling** and then choose **OK** until you return to the **Create** dialog.

Step 4 Defining the Tool Material.

You can define the tool material when you first create a tool or when editing an existing tool.

In this case, you are going to edit an existing tool.

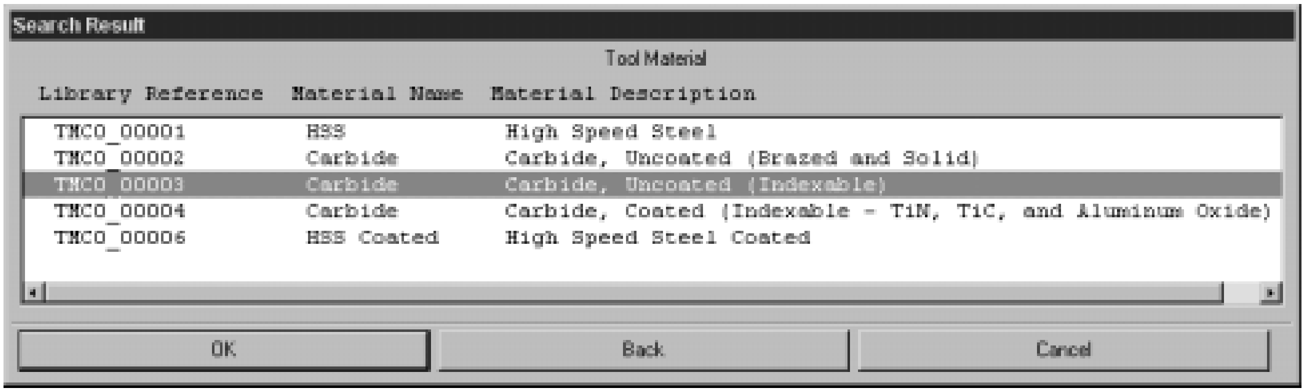
❑ Change the Operation Navigator to the **Tool View**.

❑ Highlight the **UGTI0201_013** tool name, then **MB3,** edit.

The Milling Tool - 5 Parameters dialog is displayed. Note the lower portion of the dialog. This is where you define the Tool Material. Right now the material type is HSS COATED.

❑ Choose the Material button.

The available tool material types are displayed.



❑ Select **TMCO_00002** Carbide on the list, then choose **OK** until you return to the Create menu.

Remember, you could have also changed the Material type from within the Operation by editing the tool description.

Step 5 Defining the Cut Depth.

This option is set from within the Operation and is used in the calculation of feeds and speeds.

You are now going to edit an existing Operation.

❑ Change the Operation Navigator to the **Program View**.

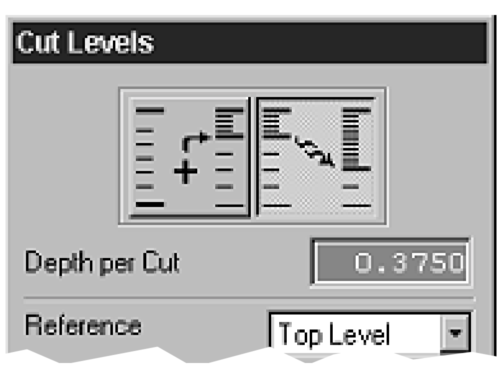
❑ Highlight the Operation named **Cavity Mill,**then **MB3,** edit.

The Cavity_Mill dialog is displayed.

❑ Select the **Cut Levels** button under Control Geometry.

The Cut Levels dialog is displayed.

❑ Under the **Depth per Cut** box, key in **.375**.



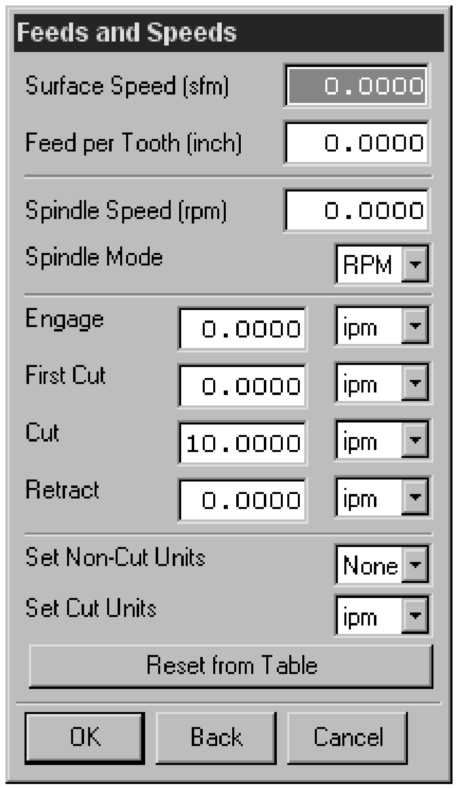
❑ Choose **OK** until the Create Operation dialog is displayed.

Step 6 Setting the Feeds and Speeds.

❑ Highlight the **Cavity_Mill** operation.

❑ Use MB3 and choose **Object→Feedrates**.

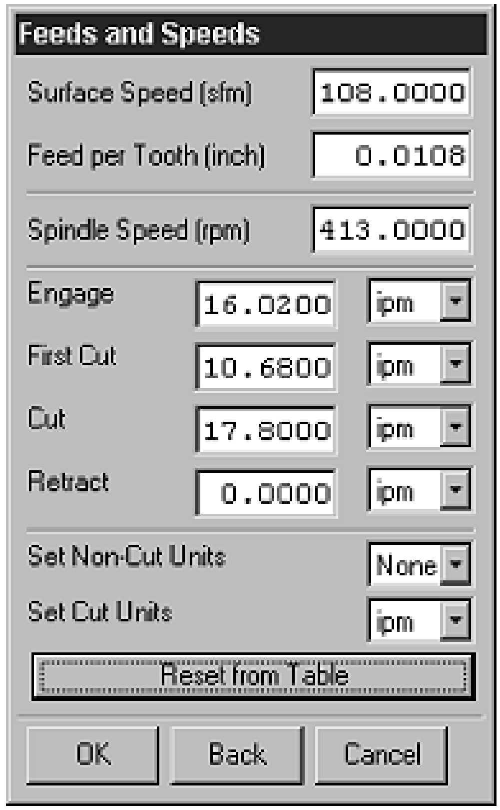
The Feeds and Speeds dialog is displayed.



❑ Choose the **Reset from Table** button.

A

The feeds and speeds parameters are calculated and displayed in the value fields.



❑ Choose **OK** to return to the Create Operation dialog.

NOTE Resetting the speeds and feeds turns off the inheritance of feed rates from the method parent.

Step 7 Changing the tool material and adjusting the Feeds and Speeds.

You are going to change the tool material then recalculate the Feeds and Speeds for the operation.

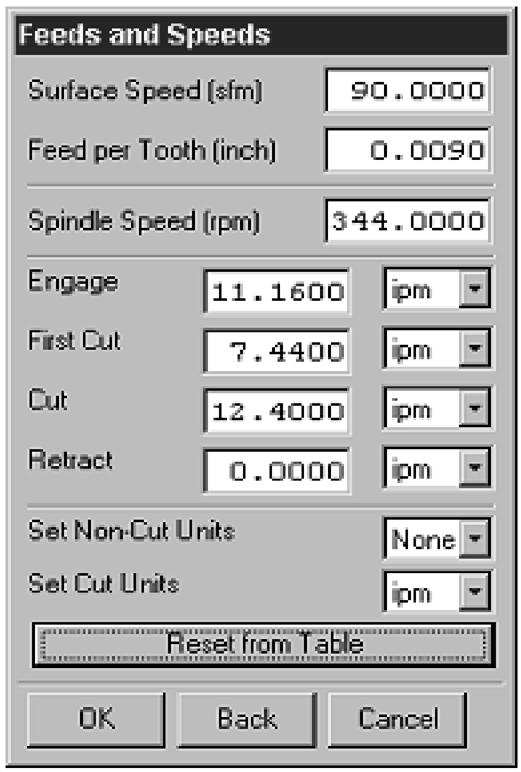
❑ Change the Operation Navigator to the **Machine Tool View**.

❑ Highlight the **UGTI0201_013** tool name, then **MB3,** edit.

❑ Choose the **Material** button.

The Search Result dialog is displayed.

- ❑ Select **TMCO_00001** HSS on the list, then choose **OK** until you return to the Create menu.
- ❑ Change the Operation Navigator to the **Program View**.
- ❑ Highlight the **Cavity_Mill** operation.
- ❑ Choose **Object→Feedrates**.
- ❑ Choose **Reset from Table.**



Notice the change in the speeds and feeds

- ❑ **Save** and **close** the part file.

SUMMARY

Libraries are used for numerous applications in the Manufacturing Application. Libraries are convenient and easy tools that can be used to access reference data with respect to cutting tool, machine tool, part material, cutting tool material, cut method and feeds and speeds.

In this lesson you:

- Reviewed Cutting Tool Libraries.
- Inserted pre-existing cutting tools into libraries.
- Reviewed the Tool Graphics Library.
- Reviewed the Machine Tool, Part Material, Cutting Tool Material and Cut Method Libraries.
- Changed various option settings to show their effect on feeds and speeds.

Glossary

ABS – Absolute coordinate system.

Absolute Coordinate System – Coordinate system in which all geometry is located from a fixed or absolute zero point.

active view – One of up to 49 views per layout in which you can directly work.

angle – In Unigraphics, an angle measured on the X-Y plane of a coordinate system is positive if the direction that it is swept is counterclockwise as viewed from the positive Z axis side of the X-Y plane. An angle swept in the opposite direction is said to be negative.

arc – An incomplete circle; sometimes used interchangeably with the term "circle."

ASCII – American Standard Code for Information Interchange. It is a set of 8-bit binary numbers representing the alphabet, punctuation, numerals, and other special symbols used in text representation and communications protocol.

aspect ratio – The ratio of length to height which represents the change in size of a symbol from its original.

assembly – A collection of piece parts and sub-assemblies representing a product. In Unigraphics, an assembly is a part file which contains components.

assembly part – A Unigraphics part file which is a user-defined, structured combination of sub-assemblies, components and/or objects.

associativity – The ability to tie together (link) separate pieces of information to aid in automating the design, drafting, and manufacture of parts in Unigraphics.

attributes – Pieces of information that can be associated with Unigraphics geometry and parts such as assigning a name to an object.

block font – A Unigraphics character font which is the default font used for creating text in drafting objects and dimensions.

body – Class of objects containing sheets and solids (see solid body and sheet body).

bottom-up modeling – Modeling technique where component parts are designed and edited in isolation of their usage within some higher level assembly. All assemblies using the component are automatically updated when opened to reflect the geometric edits made at the piece part level.

boundary – A set of geometric objects that describes the containment of a part from a vantage point.

CAD/CAM – Computer Aided Design/Computer Aided Manufacturing.

category, layer – A name assigned to a layer, or group of layers. A category, if descriptive of the type of data found on the layers to which it is assigned, will assist the user in identifying and managing data in a part file.

chaining – A method of selecting a sequence of curves which are joined end-to-end.

circle – A complete and closed arc, sometimes used interchangeably with the term "arc."

component – A collection of objects, similar to a group, in an assembly part. A component may be a sub-assembly consisting of other, lower level components.

component part – The part file or "master" pointed to by a component within an assembly. The actual geometry is stored in the component part and referenced, not copied, by the assembly. A separate Unigraphics part file that the system associates with a component object in the assembly part.

cone direction – Defines the cone direction using the Vector Subfunction.

cone origin – Defines the base origin using the Point Subfunction.

half angle – The half vertex angle defines the angle formed by the axis of the cone and its side.

constraints – Refer to the methods you can use to refine and limit your sketch. The methods of constraining a sketch are geometric and dimensional.

construction points – Points used to create a spline. Construction points may be used as poles (control vertices), defining points, or data points. See POLES, DEFINING POINTS, and DATA POINTS.

control point – Represents a specific location on an existing object. A line has three control points: both end points and the midpoint of the line. The control point for a closed circle is its center, while the control points for an open arc are its end and midpoints. A spline has a control point at each knot point. A control point is a position on existing geometry. Any of the following points: 1. Existing Points 2. Endpoints of conics 3. Endpoints and midpoints of open arcs 4. Center points of circles 5. Midpoints and endpoints of lines 6. Endpoints of splines.

convert curve – A method of creating a b-curve in which curves (lines, arcs, conics or splines) may be selected for conversion into a b-curve.

Coordinate System – A system of axes used in specifying positions (CSYS).

counterclockwise – The right-hand rule determines the counter- clockwise direction. If the thumb is aligned with the ZC axis and pointing in the positive direction, counterclockwise is defined as the direction the fingers would move from the positive XC axis to the positive YC axis.

current layout – The layout currently displayed on the screen. Layout data is kept in an intermediate storage area until it is saved.

curve – A curve in Unigraphics is any line, arc, conic, spline or b-curve. A geometric object; this may refer to a line, an arc, a conic, or a spline.

defaults – Assumed values when they are not specifically defined.

defining points – Spline construction points. Splines created using defining points are forced to pass through the points. These points are guaranteed to be on the spline.

degree-of-freedom arrows – Arrow-like indicators that show areas that require more information to fully constrain a sketch.

design in context – The ability to directly edit component geometry as it is displayed in the assembly. Geometry from other components can be selected to aid in the modeling. Also referred to as edit in place.

dimensional constraint – This is a scalar value or expression which limits the measure of some geometric object such as the length of a line, the radius of an arc, or the distance between two points.

directory – A hierarchical file organization structure which contains a list of filenames together with information for locating those files.

displayed part – The part currently displayed in the graphics window.

edit in place – See design in context.

emphasize work part – A color coding option which helps distinguish geometry in the work part from geometry in other parts within the same assembly.

endpoint – An endpoint of a curve or an existing point.

expression – An arithmetic or conditional statement that has a value. Expressions are used to control dimensions and the relationships between dimensions of a model.

face – A region on the outside of a body enclosed by edges.

feature – An all-encompassing term which refers to all solids, bodies, and primitives.

file – A group or unit of logically related data which is labeled or "named" and associated with a specified space. In Unigraphics, parts, and patterns are a few types of files.

filtering – See object filtering.

font box – A rectangle or "box" composed of dashed line objects. The font box defines the size, width and spacing of characters belonging to a particular font.

font, character – A set of characters designed at a certain size, width and spacing.

font, line – Various styles of lines and curves, such as solid, dashed, etc.

free form feature – A body of zero thickness. (see body and sheet body)

generator curve – A contiguous set of curves, either open or closed, that can be swept or revolved to create a body.

geometric constraint – A relationship between one or more geometric objects that forces a limitation. For example, two lines that are perpendicular or parallel specifies a geometric constraint.

grid – A rectangular array of implied points used to accurately align locations which are entered by using the "screen position" option.

guide curve – A set of contiguous curves that define a path for a sweep operation.

virtual intersection – Intersection formed by extending two line segments that do not touch to the position that they cross. The line segments must be non-parallel and coplanar.

inflection – A point on a spline where the curve changes from concave to convex, or vice versa.

interactive step – An individual menu in a sequence of menus used in performing a Unigraphics function.

isometric view (Tfr-ISO) – Isometric view orientation – one where equal distances along the coordinate axes are also equal to the view plane. One of the axes is vertical.

knot points – The defining points of a spline. Points along a B-spline, representing the endpoints of each spline segment.

layer – A layer is a partition of a part. Layers are analogous to the transparent material used by conventional designers. For example, the user may create all geometry on one layer, all text and dimensions on a second, and tool paths on a third.

layout – A collection of viewports or window areas, in which views are displayed. The standard layouts in Unigraphics include one, two, four or six viewports.

layouts – Standard layouts are available to the user. These include:
L1 – Single View,
L2 – Two Views,
L3 – Two Views,
L4 – Four Views,
L6 – Six Views.

Information window – The window used in listing operations, such as **Info**.

loaded part – Any part currently opened and in memory. Parts are loaded explicitly using the *File→Open* option and implicitly when they are used in an assembly being opened.

menu – A list of options from which the user makes a selection.

model space – The coordinate system of a newly created part. This is also referred to as the "absolute coordinate system." Any other coordinate system may be thought of as a rotation and/or translation of the absolute coordinate system.

name, expression – – The name of an expression is the single variable on the left hand side of the expression. All expression names must be unique in a part file. Each expression can have only one name. See expression.

objects – All geometry within the Unigraphics environment.

offset face – A Unigraphics surface type created by projecting (offsetting) points along all the normals of a selected surface at a specified distance to create a "true" offset.

options – A number of various alternatives (functions, modes, parameters, etc.) from among which the user can choose.

origin – The point X = 0, Y = 0, Z = 0 for any particular coordinate system.

parametric design – Concept used to define and control the relationships between the features of a model. Concept where the features of the model are defined by parameters.

part – A Unigraphics file containing a .prt extension. It may be a piece part containing model geometry, a sub-assembly, or a top-level assembly.

part or model – A collection of Unigraphics objects which together may represent some object or structure.

partially loaded part – A component part which, for performance reasons, has not been fully loaded. Only those portions of the component part necessary to render the higher level assembly are initially loaded (the reference set).

point set – A distribution of points on a curve between two bounding points on that curve.

Point Subfunction Menu – A list of options (methods) by which positions can be specified in Unigraphics.

read-only part – A part for which the user does not have write access privilege.

real time dynamics – Produces smooth pan, zoom, and rotation of a part, though placing great demand on the CPU.

Refresh – A function which causes the system to refresh the display list on the viewing screen. This removes temporary display items and fills in holes left by ***Blank*** or ***Delete.***

right-hand rule, conventional – The right-hand rule is used to determine the orientation of a coordinate system. If the origin of the coordinate system is in the palm of the right fist, with the back of the hand lying on a table, the outward extension of the index finger corresponds to the positive Y axis, the upward extension of the middle finger corresponds to the positive Z axis, and the outward extension of the thumb corresponds to the positive X axis.

right-hand rule for rotation – The right-hand rule for rotation is used to associate vectors with directions of rotation. When the thumb is extended and aligned with a given vector, the curled fingers determine the associated direction of rotation. Conversely, when the curled fingers are held so as to indicate a given direction of rotation, the extended thumb determines the associated vector.

screen cursor (cursor) – A marker on the screen which the user moves around using some position indicator device. Used for indicating positions, selecting objects, etc. Takes the form of a full-screen cross.

sheet – A object consisting of one or more faces not enclosing a volume. A body of zero-thickness. Also called sheet body.)

sketch – A collection of geometric objects that closely approximates the outline of a particular design. You refine your sketch with dimensional and geometric constraints until you achieve a precise representation of your design. The sketch can then be extruded or revolved to obtain a 3D object or feature.

Sketch Coordinate System (SCS) – The SCS is a coordinate system which corresponds to the plane of the sketch. When a sketch is created the WCS is changed to the SCS of the new sketch.

solid body – An enclosed volume. A type of body (see Body).

spline – A smooth free-form curve.

stored layout – The last saved version of a layout.

stored view – The last saved version of a view.

string – A contiguous series of lines and/or arcs connected at their end points.

sub-assembly – A part which both contains components and is itself used as a component in higher-level assemblies.

surface – The underlying geometry used to define a face on a sheet body. A surface is always a sheet but a sheet is not necessarily a surface (see sheet body). The underlying geometry used to define the shape of a face on a sheet.

system – The Unigraphics System.

temporary part – An empty part which is optionally created for any component parts which cannot be found in the process of opening an assembly.

top-down modeling – Modeling technique where component parts can be created and edited while working at the assembly level. Geometric changes made at the assembly level are automatically reflected in the individual component part when saved.

trim – To shorten or extend a curve.

trimetric view (Tfr-Tri) – A viewing orientation which provides you with an excellent view of the principal axes. In Unigraphics II, the trimetric view has the Z-axis vertical. The measure along the X-axis is 7/8 of the measure along Z, and the measure along the Y-axis is 3/4 of the measure along Z.

Unigraphics – A computer based turnkey graphics system for computer-aided design, drafting, and manufacturing, produced by UGS.

units – The unit of measure in which you may work when constructing in Unigraphics. Upon log on, you may define the unit of measure as inches or millimeters.

upgraded component – A component which was originally created pre-V10 but has been opened in V10 and upgraded to remove the duplicate geometry.

version – A term which identifies the state of a part with respect to a series of modifications that have been made to the part since its creation.

view – A particular display of the model. View parameters include view orientation matrix; center; scale; X,Y and Z clipping bounds; perspective vector; drawing reference point and scale. Eight standard views are available to the user: Top, Front, Right, Left, Bottom, Back, Tfr-ISO (top-front-right isometric), and Tfr-Tri (top-front-right trimetric).

view dependent edit – A mode in which the user can edit a part in the current work view only.

view dependent modifications – Modifications to the display of geometry in a particular view. These include erase from view and modify color, font and width.

view dependent geometry – Geometry created within a particular view. It will only be displayed in that view.

WCS – Work Coordinate System.

WCS, work plane – The WCS (Work Coordinate System) is the coordinate system singled out by the user for use in construction, verification, etc. The coordinates of the WCS are called work coordinates and are denoted by XC, YC, ZC. The XC-YC plane is called the work plane.

Work Coordinate System – See WCS.

work layer – The layer on which geometry is being constructed. You may create objects on only one layer at a time.

work part – The part in which you create and edit geometry. The work part can be your displayed part or any component part which is contained in your displayed assembly part. When displaying a piece part, the work part is always the same as the displayed part.

work view – The view in which work is being performed. When the creation mode is view dependent, any construction and view dependent editing that is performed will occur only in the current work view.

XC axis – X-axis of the work coordinate system.

YC axis – Y-axis of the work coordinate system.

ZC axis – Z-axis of the work coordinate system.

Index

Numbers

A

B

C

IN

D

E

F

G

H

I

K

L

M

N

O

IN

IN

P

R

S

(This Page Intentionally Left Blank)

Reference Chart Tear Outs

These tear out reference charts are provided for your convenience.

(This Page Intentionally Left Blank)

Student Profile
Mill Manufacturing Process

Name ______________________________ **Date** ______________
Employer ____
U.S. citizen? Yes / No
When is your planned departure time?______________am/pm

Please answer the following questions as honestly as you can. We are concerned about providing training that meets your needs. If you have any additional comments please write them on the back of this form.

1. Job title: ____
2. Current responsibilities: ____
____
3. How long have you held these responsibilities? Years _____ Months _____
4. How long have you been working with CAD/CAM/CAE systems? Years _____
5. What other CAD/CAM/CAE systems are you familiar with? ______________________
____
6. Are you currently using Unigraphics? ______ Version ______ Hours per week? __________
7. What is the function of your CAD/CAM/CAE system (documentation, modeling, analysis, translation interface, etc.)? ______________________________
____
8. What do you model in your Unigraphics part files (castings, assemblies, floor plans, etc.)?
____
____
9. Please list other completed CAD/CAM/CAE courses and the provider including *Unigraphics CBT and CAST*:

Course	**Provider**

10. Please check the box that best describes your current skill level in the various Unigraphics disciplines listed below.

	none	novice	intermediate	advanced	future use
Wireframe Modeling					
Solid Modeling					
Parametric Modeling					
Drafting					
Assemblies					
Manufacturing					

Additional Comments

NX Mill Manufacturing Process
Class Agenda

Day 1 ________Morning

- Overview
- Lesson 1 Getting Started
- *Workbook Section 1 Process Planning*
- Lesson 2 Cavity Mill – Basic

Afternoon

- *Workbook Section 2 Preparing the Cover Housing for Machining*
- Lesson 3 Visualization
- *Workbook Section 3 Regeneration and Tool Path Verification*
- Lesson 4 Assembly Modeling for Manufacturing

Day 2 ________Morning

- Lesson 5 Coordinate Systems
- Lesson 6 Smart Objects
- *Workbook Section 4 MCS and Smart Objects*

Afternoon

- Lesson 7 The Machining Environment
- Lesson 8 The Operation Navigator
- Lesson 9 Face Milling
- *Workbook Section 5 Face Milling*

Day 3 ________Morning

- Lesson 10 Creating Drilling Tools and Operations
- Lesson 11 Drill Geometry Parent Groups
- *Workbook Section 6 Drill Parent Groups and Operations*

Afternoon

- Lesson 12 Advanced Cavity Milling Topics
- *Workbook Section 7 Advanced Cavity Milling topics*
- Lesson 13 Planar Mill – Basics

Day 4 ________Morning

- Lesson 14 Planar Mill – Intermediate
- Lesson 15 Planar Mill – Advanced

Afternoon

- *Workbook Section 8 Planar Mill, Profile operations*
- Lesson 16 Z—Level Milling
- *Workbook Section 9 Z-Level Operations*

Day 5 ________Morning

- Lesson 17 MILL_AREA Geometry Parent Groups
- Lesson 18 Fixed Contour Operation Types

- *Workbook Section 10 Fixed Contour Mill Area Operations*

Afternoon

- Lesson 19 Tool Path Information Output
- *Workbook Section 11 Post Processing and Shop Documentation*

Mill Manufacturing Process Training Course Evaluation

Name *(Optional)*____ **Date** _________________

Instructor _________________________ **Location** ____________________ **NX Version 1**

*Please give your **honest** opinion about the training you have received during this class. Provide additional comments on the reverse side of this evaluation form.*

☐ *Please check the box if you would like your comments, regarding the training you just received, featured in our training publications. We will contact you if more information is needed.*

Hotel Accommodations (if applicable) Hotel name ____________________________________

What was your overall impression of this hotel?	Poor 2	3	4	5	6	7
Facilities – How would you rate the training facilities?	Poor 2	3	4	5	6	7
Instruction – How would you rate the instruction?	Poor 2	3	4	5	6	7
Was the instructor knowledgeable of the subject?	Poor 2	3	4	5	6	7

Comments ____

1. Were the course objectives clearly defined and were they met? Yes No
   Please explain:____
2. Were concepts effectively communicated so that you understand how to apply the software? Yes No Please explain:____
3. How well prepared do you now feel to use the functions covered in this course in your day to day activities? Please explain:____
4. Were the student activities effective in learning the Manufacturing Process? Yes No Please explain:____

5. What additional topics would you like to see covered in this course? Please explain:__________

6. Do you have any other suggestions on how the course could be improved? Yes No
   Please explain:__________

7. In order to continually improve our courseware, a post class survey is conducted; would you be willing to participate in this survey. ☐ (If you checked this box, make sure that your name is on this sheet.)

Course – What was your overall impression? Poor 2 3 4 5 6 7

Additional Comments __________

The NC Programming Sequence — 7 Easy Steps

The Manufacturing application programming sequence is to:

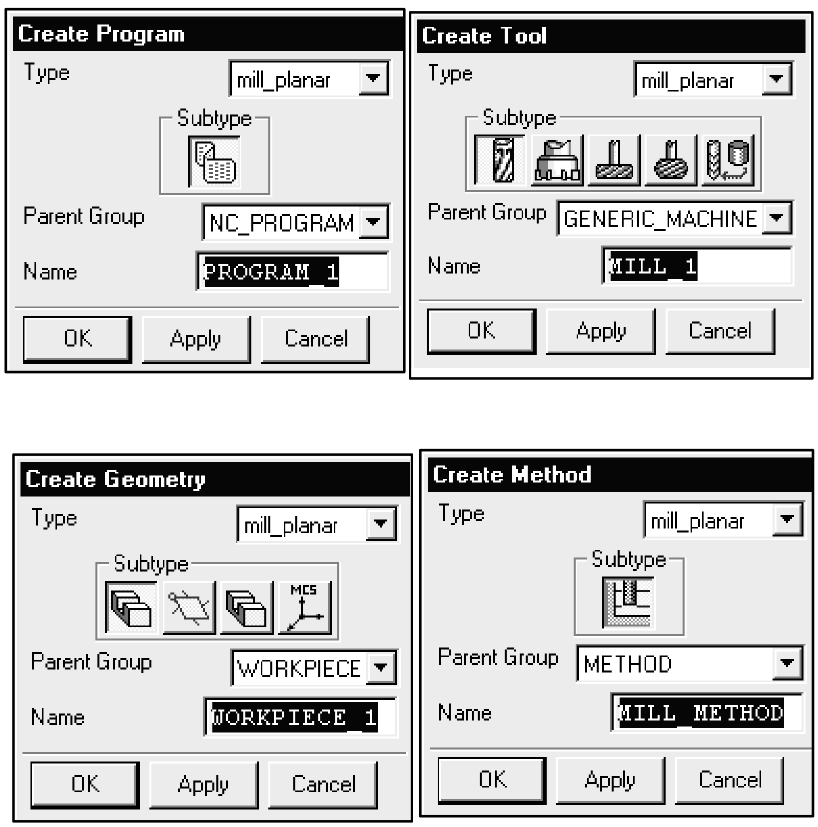
- Create the Manufacturing Assembly
- Choose the Manufacturing Environment
- Establish the Group Parents
- Create the Operation(s)
- Verify Your Work
- Postprocess
- Shop Docs

(This Page Intentionally Left Blank)

Create Parent Groups

Parent Group	Data Contained
Tool	Tool Data
Method	Machining data such as Feeds, Speeds and Tolerances.
Geometry	Geometry data such as, Part, Blank, ID, OD, MCS, and Clearance Plane, etc.
Program	The output order of Operations

These dialogs are shown in the following below:

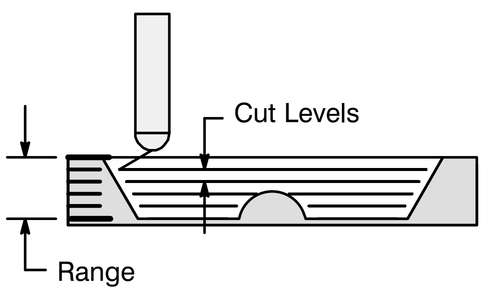


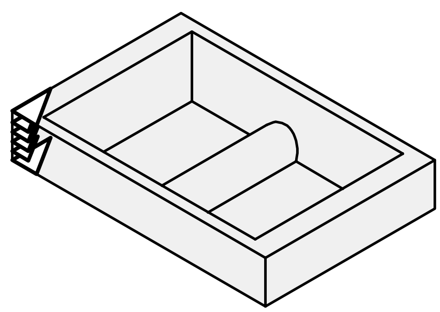
(This Page Intentionally Left Blank)

How Cavity Milling Creates Tool Paths

In order to efficiently use Cavity Milling, it is important to understand how Cavity Mill creates tool paths. The process is as follows:

- Select the Blank material.
- Select the Part geometry.
- Cavity Milling then automatically sets the top and bottom of the Blank geometry as the highest and lowest level of cutting.
- Based on the user-defined Cut Levels, the system creates a plane that is perpendicular to the tool axis
- Intersection curves between the Cut Level planes and the geometry are created. These intersection curves are called Traces.
- At each Cut Level, Cavity Milling creates a cut pattern to efficiently remove material from that level.
- Lastly, the system combines all the different Cut Levels with Engage and Retract moves.





(This Page Intentionally Left Blank)

Coordinate System Summary

Absolute Coordinate System

- Sometimes abbreviated ABS
- Fixed in model space
- Cannot be seen
- User may save Csys to mark location
- Useful as a reference in large assemblies

Work Coordinate System

- Sometimes abbreviated WCS
- Displayed in graphics area with C after each leg
- Moveable and changeable
- Used heavily in modeling
- I,J,K Vectors are based on orientation of WCS

Machine Coordinate System

- Sometimes abbreviated MCS
- Displayed in graphics area with M after each leg
- Moveable and changeable
- Zero point for tool path
- Z-axis is the default tool axis

Reference Coordinate System

- Sometimes abbreviated RCS
- Limited functionality
- Eliminates re-specification of parameters by allowing the retrieval and mapping of stored parameters

(This Page Intentionally Left Blank)

Smart Objects

The following table defines the type of Smart Objects that you can create and the Operation Types in which they are available:

All Operations	Drill	Planar Mill	Cavity Mill	Fixed Contour
Fixed Tool Axis Vector	Projection Vector	Initial Engage	Initial Engage	Projection Vectors
From Point	Material Side Vector	Final Retract	Final Retract	Non-Cutting: Direction
Start Point	Part Surface	Predrilled Engage Points	Predrilled Engage Points	
Return Point	Bottom Surface	Cut Region Start Points	Cut Region Start Points	
GOHOME Point		Floor Plane	Cut Levels	
Clearance Plane		Boundary Plane		
Lower Limit				
GOHOME point				
MCS				
RCS				

To create Smart Objects, you use the Point, Plane, Vector and CSYS *Constructor* dialog boxes.

(This Page Intentionally Left Blank)

Planar Mill vs. Cavity Milling

Planar Mill is intended to cut planar geometry.

Use Planar Mill to create tool paths for:

- Planar geometry
- Roughing or finishing
- A Single Profile pass

Cavity Milling is intended to rough geometry.

The tool can follow contoured geometry at each Cut Level. The Z-axis is fixed while cutting, but follows irregular shapes in the X and Y-axis.

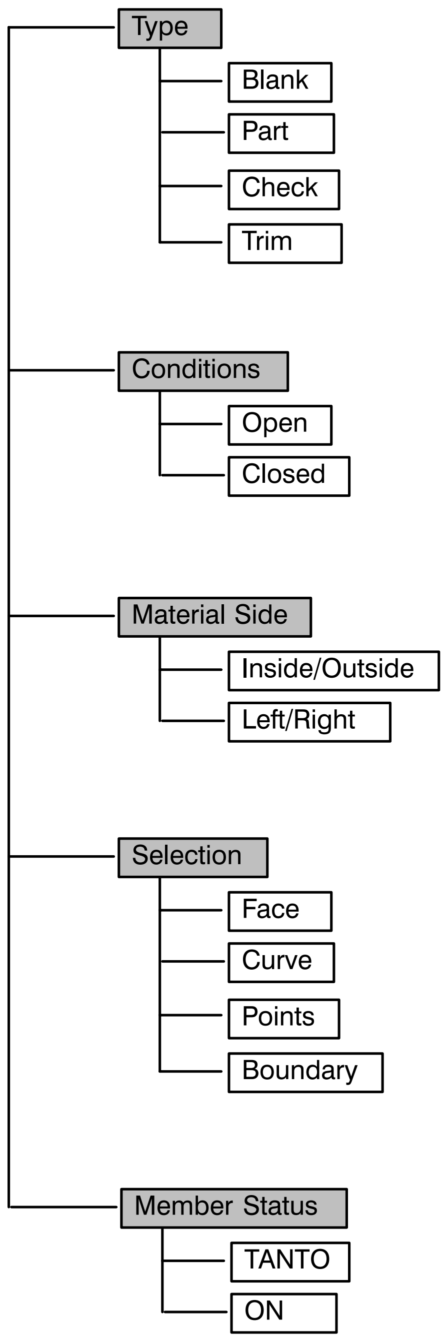
You use Cavity Milling to create tool paths for:

- Contour geometry
- Roughing

(This Page Intentionally Left Blank)

Boundary Flow Chart

The following is a flow chart outlining the characteristics related to boundaries.



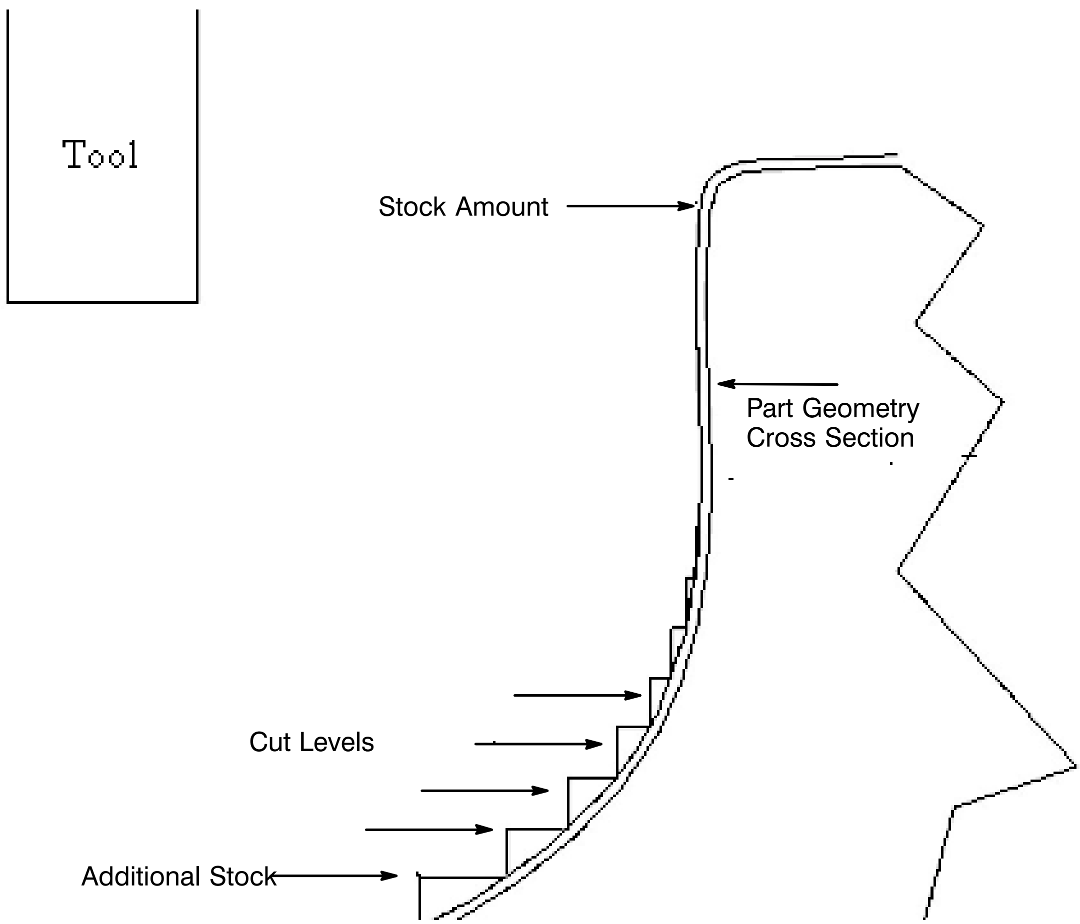
(This Page Intentionally Left Blank)

Cut Levels

Cavity Milling cuts geometry in planes or levels.

The advantage to this approach is that tool paths remain relatively short, because Z-axis moves are only specified occasionally.

The disadvantage is that when machining geometry that is close to horizontal the system may leave more stock than desired. See the diagram below.



As shown in the diagram above, the closer the geometry approaches horizontal, the more stock that is left. Through the use of the Cut Level dialog box, you can reduce that additional stock amount.

(This Page Intentionally Left Blank)