UNIGRAPHICS

ADVANCED MILL APPLICATIONS

STUDENT MANUAL

September 2002
MT11045 — Unigraphics NX

EDS Inc.

Proprietary & Restricted Rights Notices

Advanced Mill Applications Student Manual Publication History:

Version 16.0 . . . . October 2000
Version 17.0 . . . . January 2001
Vesion 18.0 . . . . August 2001
Unigraphics NX . . . . September 2002

Table of Contents

(This Page Intentionally Left Blank)

Course Overview

Course Description

The Advanced Mill Applications course is oriented toward the Mold/Die industry and includes topics directed at the more experienced and advanced NC/CNC user. Topics include rough and finish operations of machining cavity and core type parts, wire EDM methods and procedures, and high speed machining techniques. This course includes advanced topics that are designed to maximize productivity and efficiencies in everyday programming environments.

Intended Audience

Manufacturing Engineers, Process Planners, NC/CNC Programmers and CAD/CAM System Managers.

Prerequisites

The required prerequisites for the course are Practical Applications of Unigraphics and the Mill Manufacturing Process course or the CAST equivalent. Your experience as an NC/CNC programmer or machinist is also an asset in taking this course.

Objectives

Upon completion of this course, you will be able to create manufacturing assemblies using the Wave Geometry linker, perform roughing and finishing operations of cavity and core type parts, machine faceted bodies, create wire EDM operations and use various cut patterns and operation types to perform high speed machining.

Student Responsibilities

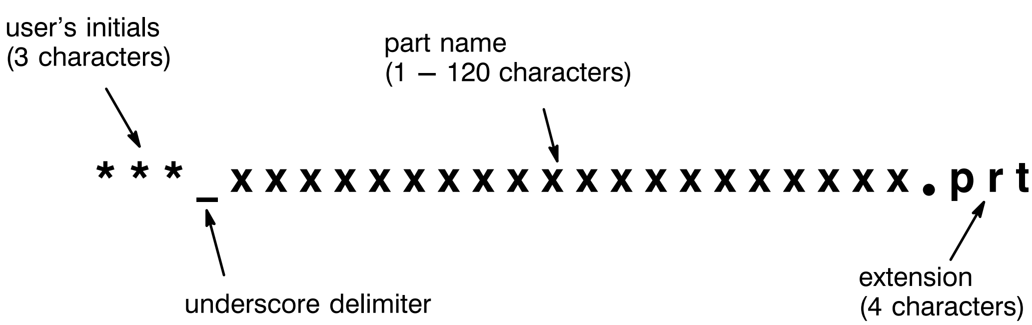
- Be on time
- Participate in class
- Stick with the subject matter
- Listen attentively and take notes
- Practice on the job what you have learned
- Have Fun

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 filenaming 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 because 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
- project
- summary

(activity and project: One or more included in most lessons)

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 Advanced Mill Application project is an ongoing activity that incorporates the skills required to perform advanced machining processes. During the course of working through this workbook exercise, you will be asked to perform tasks which will be used to machine the core block of a plastic hair dryer.

It is the intent of this project to allow the student 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: ____________________

WAVE Geometry Linker in Manufacturing
Lesson 1

PURPOSE

In this lesson, you will learn different methods available for creating machining geometry that is associated to the designer's original geometry. This geometry is still modifiable for manufacturing applications.

OBJECTIVES

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

- Use a "base part" to control the manufacturing setup
- Use the WAVE Geometry Linker to create associative, linked geometry
- Make modifications to linked geometry
- Build a simulated casting solid body

This lesson contains the following activities:

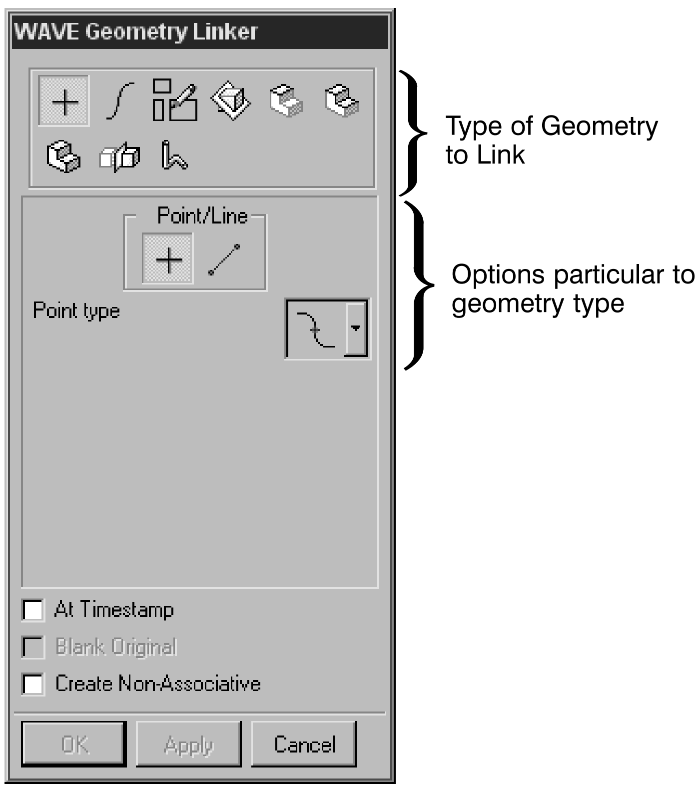
The WAVE Geometry Linker

The WAVE Geometry Linker is a tool which provides a means to associatively copy geometry from another component part in an assembly into the work part. The resulting linked geometry is associated to the parent geometry. Modifying the parent geometry will cause the linked geometry in the other parts to update.

NOTE The WAVE Geometry Linker is available with an Assemblies license. It does not require a UG/WAVE license.

Different types of objects can be selected including points, curves, sketches, datums, faces, and bodies. The linked geometry can be used for creating and positioning new features in the Work Part.

This function is accessed by choosing **Assemblies→WAVE Geometry Linker.**

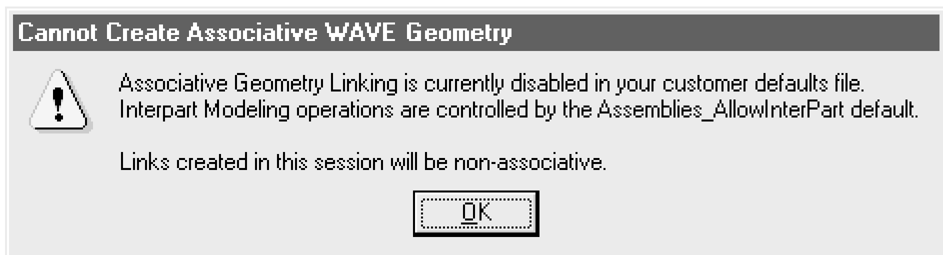


The **At Timestamp** option lets you specify where the linked object is placed in the feature list. When turned OFF, any new features added that alter the parent geometry will be reflected in the linked geometry. When turned ON, new features added after the link was created will not be reflected.

Blank Original lets you blank the original geometry so that the linked geometry in the work part will be easier to work with while the assembly is displayed.

The **Create Non-Associative** option will create a broken link. The geometry will be created in the work part but will not be associated to the parent geometry.

If interpart modeling is not enabled in the customer defaults file (*ug_english.def* or *ug_metric.def*), the following message will appear when you access the WAVE Geometry Linker. The dialog will still be available but the Create Non-Associative option will be permanently ON by default.



NOTE To enable the WAVE Geometry Linker you must change the statement **Assemblies_AllowInterPart: no** to **yes** in the *ug_english.def* or *ug_metric.def* files. It is *not on* by default.

Geometry Types

Several different types of geometry can be selected for the WAVE application.

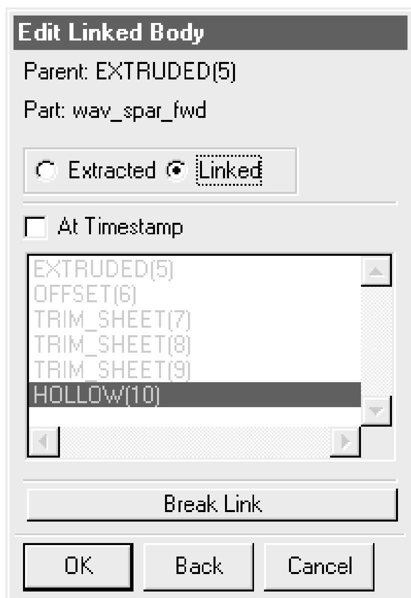
- Points
- Curves/Strings
- Sketches
- Datums
- Faces
- Regions of Faces
- Bodies
- Mirrored Bodies

When selecting the geometry to copy, you should consider how permanent the geometry will be. If you copy as little geometry as possible to do the job, performance will be improved but updates will be less robust when the parent geometry is altered.

For example, if you copy individual curves to another part, the link may not update correctly if one of the curves is deleted. On the other hand, if you copy an entire sketch, curves may be removed or added to it and the link will still update.

Editing Links

Links may be edited by choosing **Edit→Feature→Edit Parameters** in the ***Model Navigator*** and selecting a linked feature. Linked features have an **Edit** dialog similar to the one below.



While this dialog is displayed, the cursor is active in the graphics area so that new parent geometry can be selected for the link being edited. The new parent geometry must be the same type as the old geometry (curve, datum, solid body, etc.)

Parent indicates the parent geometry type. If the feature was linked, but the link has been broken, the parent is shown as a Broken Link.

Part shows the name of the part where the parent geometry is located. If the parent geometry is located in the current work part, the part name given is Work Part.

NOTE The dialog information updates when you select new parent geometry, which you can do at any time.

At Timestamp lets you specify the timestamp at which the linked feature is placed. If this option is toggled ON, the list box will display the features in the parent part. One of these features may be selected from the list to specify a new timestamp location for the linked feature being edited. If this option is toggled OFF, all features in the parent part will be reflected in the linked feature.

Break Link lets you break the association between the linked feature and its parent. This means that the linked feature will no longer update if its parent changes. You can later define a new parent by selecting geometry with the cursor.

An **Extracted** feature (intrapart) can be converted to a **Linked** feature (interpart) by selecting the appropriate option and selecting new parent geometry from another component in the assembly.

Depending on the geometry type of the feature being edited, other options may appear on the dialog.

NOTE When editing links and selecting new parent geometry, it may be easier to temporarily work in an exploded view so that you can distinguish between the existing linked geometry and the new parent geometry.

Broken Links

A link may become broken for several reasons:

- The parent geometry is deleted.
- The path from the linked geometry to its parent part is broken. This can occur if the component part containing the parent geometry is deleted or substituted.
- If the parent is removed from the start part reference set that defines the linked part.
- If you deliberately break the link (e.g., using Edit Feature or the Break option on the **WAVE Geometry Navigator** dialog).

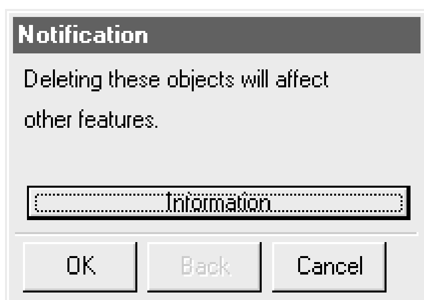
Newly Broken Links

When a link breaks for an indirect reason (i.e., any reason except the last one listed above), the link is identified as *newly broken* until you accept it. You can accept newly broken links from the WAVE Geometry Navigator dialog or the Edit during Update dialog.

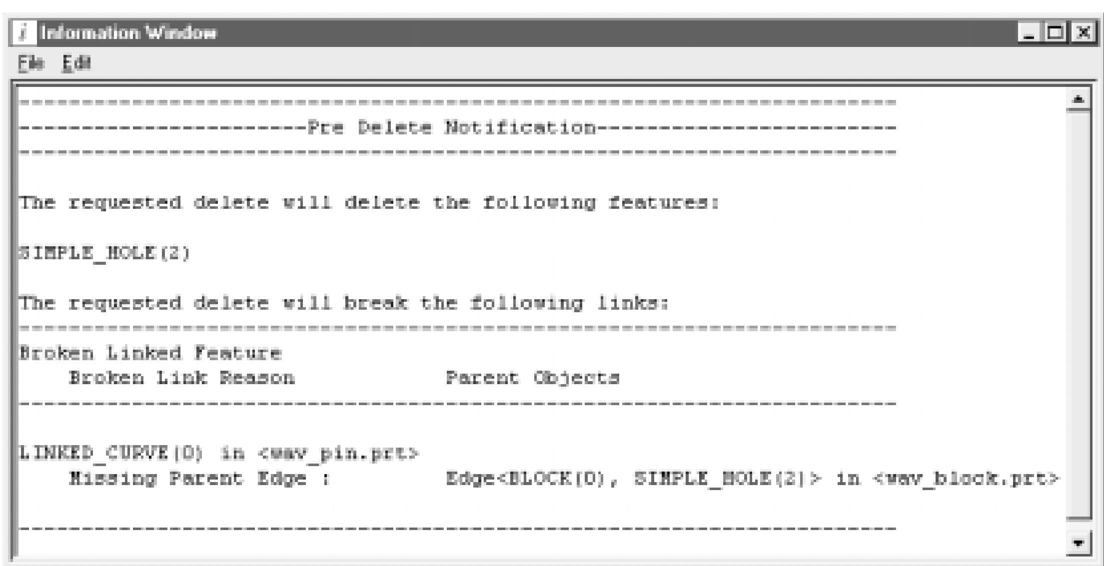
After a link is accepted, its status is changed to *broken* until a new parent is defined.

Deleting Parent Geometry

To prevent unintentional deletion of the parents of linked geometry, a message will warn you if a delete operation would cause interpart links to break. This applies to operations using **Edit→Feature→Delete**, **Edit→Delete**, and **Model Navigator→Delete** while the parts containing the linked geometry are loaded.



The **Information** option provides details about the links that will be broken in an Information window.



Deleting Linked Geometry

Linked geometry is created as a feature and can be deleted by choosing **Edit→Feature→Delete** (or choosing the Delete Feature icon).

Linked bodies may also be deleted by choosing **Edit→Delete,** but you will not have an opportunity to verify its child features before they are lost.

Assemblies and WAVE

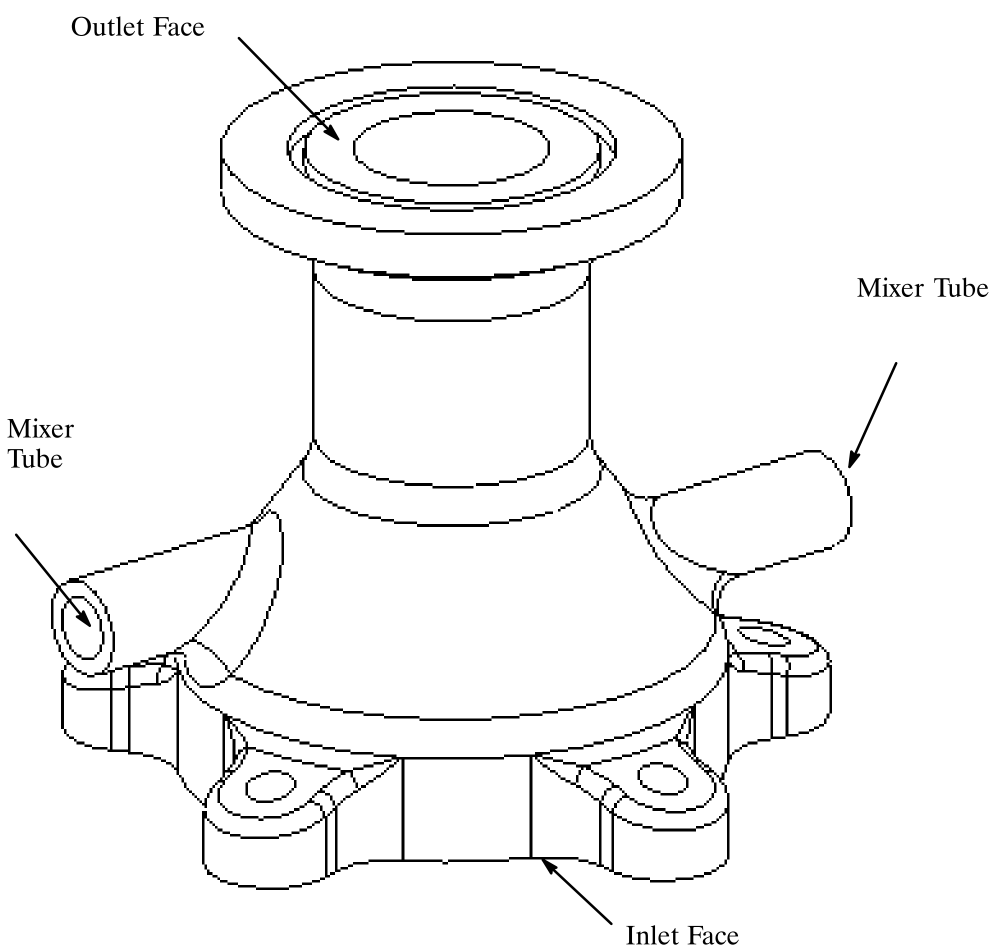
The WAVE Geometry Linker only works in the context of an assembly. An assembly link must exist between two part files before a WAVE link can be established.

Activity 1—1: Creating an Assembly for WAVE

In this activity, you will create an assembly structure for later use with the WAVE Geometry Linker. Since WAVE only works in the context of an assembly, this step is an important one.

To set this activity up, our hypothetical company has won a contract to perform the machining on a mixer housing. The customer has supplied us with a Unigraphics solid model of the designed part. Because high-production quantities are needed, the customer has decided to make the part an aluminum casting. This will greatly reduce the amount of time spent machining. Unfortunately, the customer has not supplied us with a solid model of the casting. It is our job to create the solid model of the casting. Using WAVE, we will create a simulated casting model that is associated with the original.

For the casting body, it is necessary to remove the seven drilled holes, and apply .250” machining stock on the inlet face, outlet face, and mixer tube faces. Additionally, the ring groove will not exist on the casting body.



The customer has informed us that all machined faces have 1/4” of added stock. Once the modeling changes are accomplished, we will also drill all holes, and machine the ring groove into the mixer outlet face, since the casting process was not accurate enough for the tolerances required.

Step 1 Open a seed part, then save it with a new name.

❑ Choose **File→Open→ ama_seedpart_in.prt**.

1

TIP Seed parts are often used to standardize part files for a particular company or situation. With a seed part, Layer Categories can be already defined, standard object colors can be implemented, and other preferences can be set as well.

❑ Choose **File→Save As ***_mixer_mfg.prt** where ***** represents your initials.

Step 2 Add the existing designed part as an assembly component.

Our first objective will be to add the existing mixer housing as the first component of the mixer_mfg assembly. All assembly links will be on layer 11.

❑ From the main menu, choose **Applications→Assemblies.**

❑ Change the **Work Layer** to **11**.

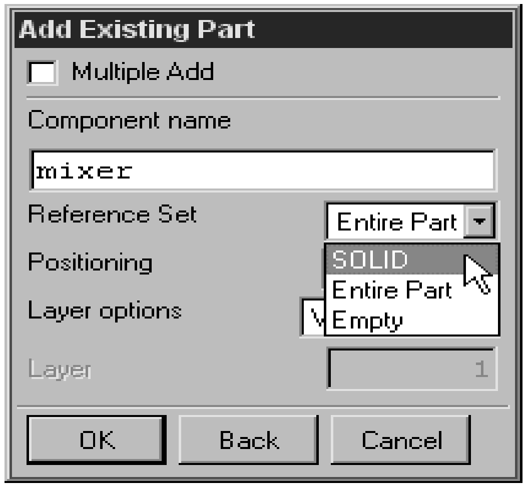
❑ From the main menu, choose **Assemblies→Components→Add Existing**

❑ In the *Select Part* dialog, click on **Choose Part File**.

❑ Find and select **ama_mixer_body.prt**, then choose **OK**.

❑ In the *Add Existing Part* dialog, change the component name to **mixer**. It can be typed in upper or lower case. Unigraphics will convert it to upper case.

❑ While still in the *Add Existing Part* dialog, choose **SOLID** from the Reference Set pull down window.



The *Add Existing Part* dialog is still displayed.

- ❑ Verify that the Positioning pull down menu is set to **Absolute**.

- ❑ Choose **OK** in the *Add Existing Part* dialog.

- ❑ Choose **Reset** in the *Point Constructor* dialog, then choose **OK**.

  The mixer body part file is now a component of *****_mixer_mfg.prt**.

- ❑ **Cancel** the *Select Part* dialog.

Step 3 Examine the current assembly structure.

- ❑ Display the **Assembly Navigator** by choosing the Assembly Navigator tab in the resource bar.

> **TIP** Clicking once on the tab temporarily displays the Assembly Navigator by sliding it to the left over the graphics display.
>
> Double–clicking on the tab displays the Assembly Navigator in a separate window which can then be moved and docked.

There are currently two parts in this assembly. The top-level control part is *****_mixer_mfg.prt**, while **ama_mixer_body.prt** is the single component. Currently, only the component contains any geometry.

The next step is to create a new component that will contain the WAVE casting body.

Step 4 Create an empty component, then apply the seed part preferences.

- ❑ Choose **Assemblies→Components→Create New.**
- ❑ Choose **OK** in the *Class Selection* dialog.
- ❑ In the *File Name* field, type in *****_mixer_casting.prt**, then choose **OK**.

  The *Create New Component* dialog is displayed.

- ❑ In the *component name* field, type **casting**, then choose **OK**.

  A new component, named **CASTING**, is displayed in the Component Name column of the Assembly Navigator. The name of the part file is *****_mixer_casting.prt.** You may need to display the Component Name column by **MB3→Columns→Component Name**.

  Next, apply the layer and color standards from the seed part file. In Unigraphics, all operations apply to the work part, which is currently *****_mixer_mfg.prt**. To apply the seed part defaults, the CASTING component should be the work part. For clarity, we will also make it the displayed part.

- ❑ In the Assembly Navigator, perform **MB3** on the CASTING component. Choose **Make Displayed Part** from the pop-up dialog.
- ❑ To illustrate the lack of user-defined defaults, choose **Format→Layer Settings**.

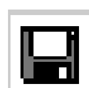
  Notice the category field is blank.

- ❑ Choose **Cancel** in the Layer Settings Dialog.
- ❑ Choose **File→Import→Part...**
- ❑ In the *Import Part* dialog, turn off **Create Named Group**, then choose **OK**.

- ❑ Browse to the **ama_seedpart_in.prt**, and double-click on it.
- ❑ Choose **OK** in the *Point Constructor* dialog. Since no geometry is being imported, position is irrelevant.
- ❑ Choose **Cancel** in the **Point Constructor** dialog.
- ❑ Choose **Format→Layer Settings**.

  Now several layer categories are defined.
- ❑ Choose **Cancel** in the *Layer Settings* dialog.

Step 5 Make the top-level part the displayed part, and save the work thus far.

- ❑ In the Assembly Navigator, **MB3** on *****_mixer_casting**, and choose **Display Parent→***_mixer_mfg.**
- ❑ In the Assembly Navigator, **MB3** on *****_mixer_mfg**, and choose **Make Work Part**.
- ❑ Choose the **Save** icon  on the **Standard** toolbar.

NOTE When you save an assembly, all modified components below the work part are also saved.

- ❑ Dismiss any warnings about the mixer body not being saved.
- ❑ This concludes this activity.

Linking Procedure

We will use the **Assemblies→WAVE Geometry Linker** dialog to create associated objects between part files. Unlike promotions, the linker allows us to copy geometry “downward” into component parts, “upward” into higher level assemblies, or “sideways” between components within an assembly. As you build your assembly you will use the “sideways” functionality.

To create linked geometry:

- Arrange your assembly display so that the part containing the geometry to be copied is visible, and the geometry of interest is selectable.
- Change work part to the part that is to receive the linked copies.
- Set the Work Layer to the layer you want to contain the linked copies.
- Choose **Assemblies→WAVE Geometry Linker**.
- Use the linker dialog to filter the type of object(s). You may select several objects of different types.
- Choose **Apply** to make copies and remain in the **Selection** dialog, or **OK** to copy objects and exit the dialog.

TIP Linked Geometry will be created in the Work Layer in the current work part, but will display in the layer in which the component object was created in higher level assemblies. To distribute linked objects according to layer planning and standards select objects for a particular layer then choose Apply. You can change Work Layer while in the **WAVE Geometry Linker** dialog, then select more objects.

TIP In the assemblies that you will be using, several bodies often occupy the same volume. Sometimes it becomes difficult to select the correct geometry. You can control the display components by turning on or off the single layer in which they reside, and blanking all parts except the one you want to link from. If it is still difficult to select the geometry you want, create a separate reference set for each object that will be a parent for linked geometry. The visibility options in the Assembly Navigator in WAVE mode are particularly useful for isolating displays of components.

Activity 1—2: Creating WAVE Geometry

In this activity, you will practice using the geometry linker. You will create a WAVE linked copy of the mixer body, then perform modifications to that copy to simulate a casting.

If necessary, open your *****_mixer_mfg** assembly part. Open the Assembly Navigator.

Step 1 Prepare the assembly.

❑ Use MB3 over the *****_mixer_casting** line in the **Assembly Navigator** to choose **Make Work Part**.

The mixer body in the graphics window fades to gray. This is a visual clue that geometry is no longer in current modeling hierarchy.

The work layer is where linked geometry will be created.

❑ Choose **Format→Layer Settings.**

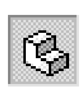
❑ Make layer 1 the work layer.

❑ Choose **OK** in the *Layer Settings* dialog.

Step 2 Create a linked body.

❑ Choose **Assemblies→Wave Geometry Linker.**

It is possible to link types of geometry other than solid bodies. Curves, Sketches, and Datum Planes are also commonly linked.

❑ Choose the **BODY** icon  in the **WAVE Geometry Linker** dialog.

❑ Select the mixer body.

❑ Choose **OK.**

Step 3 Modify the display of the linked casting.

There are now two identical bodies, lying in the same model space; the original mixer body and our linked copy. It can be difficult to determine which is which, so action is necessary to clarify the differences. First, you will remove the original body from the display. Then, you will change the display of the linked body.

❑ In the **Assembly Navigator** use MB3 over the CASTING component and choose **Make Displayed Part**.

❑ In the graphics area, use **MB3** to **Replace View→TFR–TRI**.

❑ Find and choose the **Shaded** icon.

❑ Choose **Edit→Object Display.**

❑ Select the linked body and choose **OK**.

NOTE Using the *Edit Object Display* dialog is a powerful method of differentiating between bodies that are similar in appearance.

❑ Change the *Color* to **Yellow**.

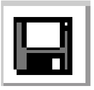
❑ Choose **OK** in the *Edit Object Display* dialog.

Step 4 Make the top-level part the displayed part, then save the work in progress.

At this point no physical difference exists between the mixer body and the mixer casting. They do have a visual difference. In the next activity, you will perform modeling changes to the mixer casting as desired.

❑ In the Assembly Navigator, **MB3** on the CASTING component, and choose **Display Parent→***_mixer_mfg.**

❑ In the Assembly Navigator, **MB3** on *****_mixer_mfg**, and choose **Make Work Part**.

❑ Choose the **Save** icon  on the **Standard** toolbar.

❑ This concludes this activity.

Simplify

Simplify is a powerful modeling tool that can be used to satisfy a wide range of needs in developing models that are associative, yet different.

Simplify provides a method of removing faces. The system must be able to extend surrounding faces to “heal the wound” where the faces have been removed.

Uses of Simplify:

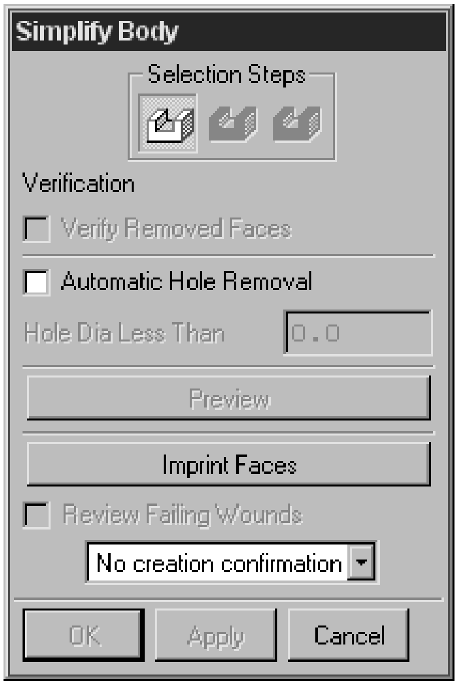
- Remove “machined” features for preparing an as cast part from a body that is not appropriately constructed for link **At Timestamp**, or from a body whose features are not accessible.
- Remove details such as holes and blends for finite element analysis.
- In casting tooling work, core and pattern preparation in parts where the regions were not modeled separately. Simplify can often be used both to remove interior faces, for patterns, and to remove exterior faces, for cores (if the system cannot heal wounds left by core removal, the pattern designer must extract regions and sew core print faces to obtain a core body).
- Preparing a body for export to a supplier who need only be concerned with the exterior envelope. Interior faces are removed using simplify, then the simplified part is linked into a new part for export to the supplier. The linked part has no “knowledge” of interior features in the original, but it can still be updated by the owning company if the parent body changes.

Simplify Body Procedure

We will use the **Simplify Body** function to remove holes from our mixer casting body.

To simplify geometry:

- Choose as a retained face one that will not be simplified away.
- Select the cylindrical faces of the holes as boundary faces.
- De-select the boundary faces as retained faces.
- Choose **Apply** to perform simplification.
- Acknowledge the simplify notice.



Activity 1—3: Using Simplify Body

In this activity, you will practice using Simplify Body as a tool to reduce the complexity of a linked solid body.

If necessary, open your *****_mixer_mfg** assembly part. Open the Assembly Navigator.

Step 1 Make the CASTING component the work and displayed part.

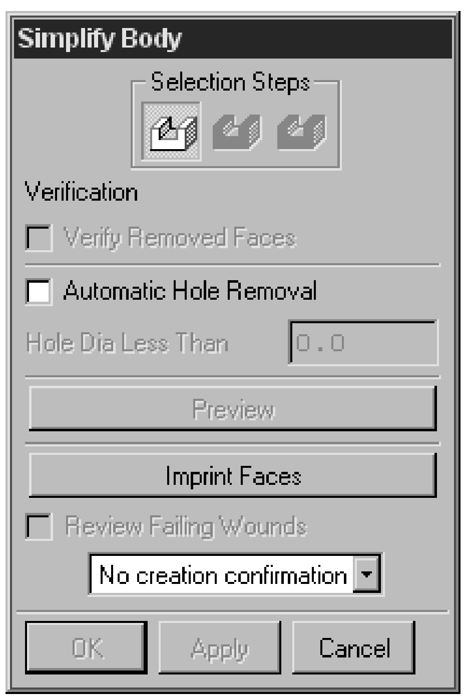
❑ In the Assembly Navigator, use MB3 on the CASTING component and choose **Make Displayed Part**.

Step 2 Perform a Simplify Body operation on the five bolt holes on the outlet face.

❑ Choose **Application→Modeling.**

❑ Choose **Insert→Feature Operation→Simplify.**

The Simplify Body dialog is displayed.

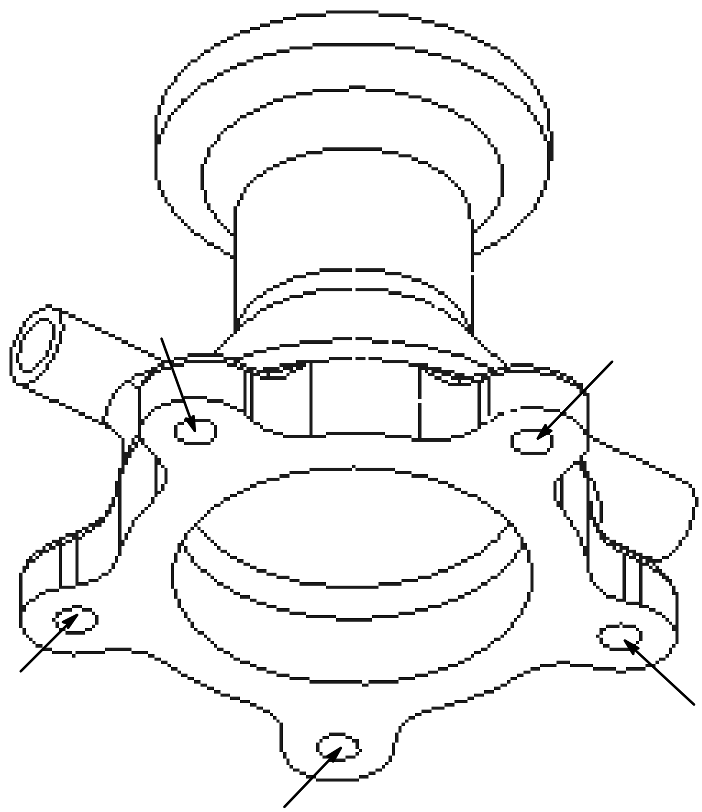


The cue line reads “Select a retained face”.

❑ Choose any face on the part that is not a hole, then choose **OK** to advance to the next step.

The cue line reads “Select a boundary face”.

❑ Choose the five cylindrical faces of the holes on the inlet face of the mixer. Do not choose the cylindrical inlet face.



When selected as a boundary face, Unigraphics also assumes that these will be retained faces, and adds them to the “retained face” selection. In this case, they will not be retained, so you will de-select them from the “retained faces” set.

❑ In the *Simplify Body* dialog, choose **Retained Faces** again.

❑ Hold down the “shift” key, and de-select the five holes again.

- ❑ Choose **Preview**.
- ❑ In the *Simplify Body Preview* dialog, choose **Preview Removed**.

  Only the hole faces highlight.
- ❑ Choose **Preview Retained.**

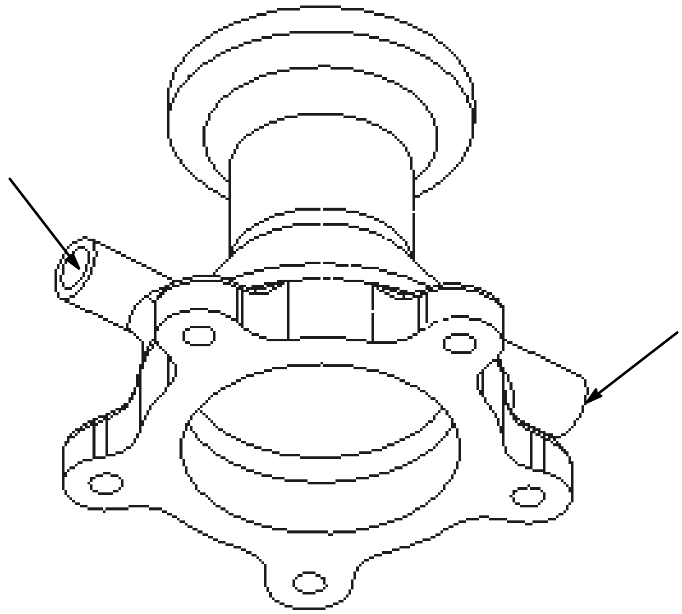
  Now all faces except the five holes highlight.
- ❑ Choose **OK** in the *Simplify Body Preview* dialog.
- ❑ Choose **Apply** in the *Simplify Body* dialog.

  The *Result of Simplification* window gives the number of faces removed and retained.
- ❑ Dismiss the *Result of Simplification* dialog by choosing **OK**.

Step 3 Simplify away the holes in the two bosses.

The retained face that was selected earlier is still active, so it is not necessary to choose another.

- ❑ In the *Simplify Body* dialog, choose **Boundary Faces**.
- ❑ This time, select the two cylindrical hole faces of the mixer tubes.



- ❑ Once again, choose **Retained Faces**.
- ❑ Using the procedure described previously, deselect the holes as retained faces.
- ❑ **Preview** the retained and removed faces.
- ❑ Choose **OK** until the body updates.
- ❑ Save the work in progress.
- ❑ This concludes this activity.

Activity 1—4: Other Modeling Techniques

Previously, **Simplify Body** was used to remove unwanted geometry from the Linked Casting body. Now, you will explore other ways to modify a linked body. The first option used is **Extrude**.

If necessary, open your *****_mixer_mfg** assembly part. Open the Assembly Navigator.

Step 1 Make the CASTING component the work and displayed part.

❑ In the Assembly Navigator, use MB3 on the CASTING component and choose **Make Displayed Part**.

Step 2 Use Extrude to fill in the ring groove.

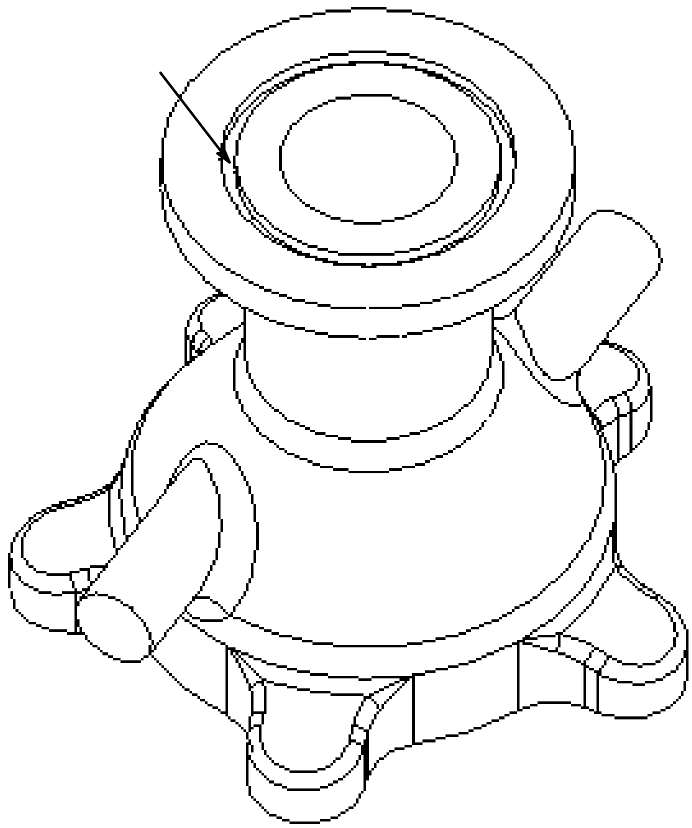
❑ Choose **Application→Modeling**.

❑ Choose **Insert→Form Feature→Extrude.**

The *Extruded Body* dialog is displayed.

❑ Choose the **Solid Face** button.

❑ Choose the bottom face of the ring groove, as shown below.



- ❑ Choose **OK** until the cue line reads “Choose Extrusion Method”.

- ❑ Choose **Trim to Face/Plane**.

  The extrusion direction arrow should point away from the linked body. If not, choose **Reverse Direction**.

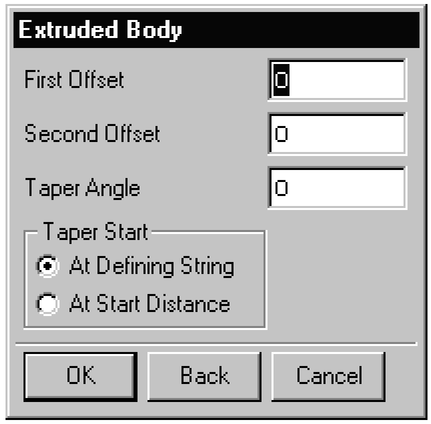
- ❑ If the extrusion direction arrow points away from the body, choose **OK**.

- ❑ In the *Trimming Face* dialog, choose Extend Trim Face.

  Extend Trim Face forces Unigraphics to extend the trimming faces as necessary to complete the extrusion operation.

- ❑ Select the outlet face, then choose **OK**.

  The following dialog allows you to apply offset to the face being extruded. In this example, offsets are not necessary.



- ❑ Choose **OK**.
- ❑ Choose **Unite**.

  The O-ring groove has been removed from the outlet face.

- ❑ **Cancel** the *Extruded Body* dialog.

Step 3 Use the Offset Face option to add machining stock.

In this step, you will add machining stock to the inlet and outlet faces, as well as the mixer tube faces.

- ❑ From the menu bar choose **Insert→Feature Operation→Offset Face.**
- ❑ In the *Offset Face* dialog, key in **0.250** for the offset value, then choose **OK**.
- ❑ Choose **Offset Faces**.
- ❑ Select the inlet and outlet faces, and the two mixer tube faces.
- ❑ Choose **OK**.
- ❑ Choose **Cancel** in the *Offset Face* dialog.

The modeling changes are complete. It will be difficult to visualize those changes in shaded mode, without a further display change to the casting.

Step 4 Change the translucency of the casting.

To make it easier to visually distinguish between the original designed part and the casting, you will make the casting model translucent.

- ❑ Use the **Shaded** icon to turn on shaded mode.
- ❑ From the menu bar choose **Edit→Object Display.**
- ❑ Select the body and choose **OK**.
- ❑ Slide the *Translucency* bar to about 50% and choose **OK**.

TIP If the solid body does not become semi-transparent, choose **Preferences→Visualization Performance**, and turn off **Disable Translucency,** located on the **General Settings** tab under Session Settings.

Step 5 Make ***_mixer_mfg the work part, and compare the two solid bodies.

To fully realize the extent of the changes made, you will display both the original and the linked body together.

- ❑ Find and depress the **Assembly Navigator** button to activate the Assembly Navigator.
- ❑ Use **MB3** on the CASTING component and choose **Display Parent→***_mixer_mfg**.
- ❑ In the Assembly Navigator, double-click on *****_mixer_mfg** to make it the work part.
- ❑ Examine the two models.

The CASTING component has stock added on the machined faces. All drilled holes have been removed, as well as the ring groove.

This is only one potential method for creating a simulated casting body. Other methods and techniques could also have been used. However, this method is fully associated to the original, so that if the customer changes the original body, the casting body will update also.

NC/CNC Programming, using the CASTING component as the BLANK, could now begin.

❑ Choose **File→Close→Save All and Close.**

This concludes this activity.

SUMMARY

The WAVE Geometry Linker provides an efficient method to associatively copy geometry used for machining from a component part in an assembly into a work part. The machining geometry is modifiable for manufacturing needs but does not change the original design intent.

In this lesson you:

- Used Assemblies to enable “Best Practices” for modeling in manufacturing
- Created a WAVE solid body that is associatively linked to the original
- Modified the WAVE geometry to simulate a casting for machining

(This Page Intentionally Left Blank)

Cavity Milling

Lesson 2

PURPOSE

This lesson will review the basics concepts of Cavity Milling and will introduce you to more advanced options and concepts that are unique to Cavity Milling. You will review the steps necessary to create Parent Groups that are used in subsequent operations.

OBJECTIVES

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

- Create Geometry Parent Groups for Cavity Milling operations
- Create advanced Cavity Milling operations
- Create Z-Level operations
- Define Blank Distance to offset Blank Geometry from Part Geometry
- Create pre-drilled Engage and Cut Region Start points

This lesson contains the following activities:

Cavity Milling Review

Cavity Milling operations, like Planar Milling, removes volumes of material in planar cut layers (perpendicular to the tool axis). The two operations types differ in the method which is used to define the material. Planar Milling uses boundaries to define Part material while Cavity Milling uses boundaries, faces, curves and solid bodies which are typical of cavities and cores.

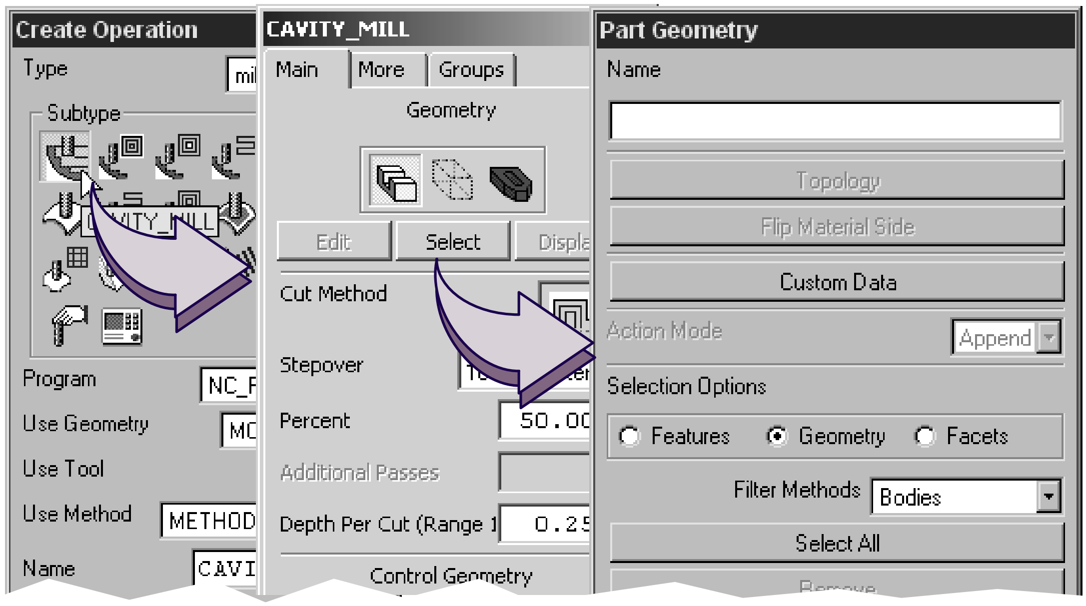
Planar Milling is designed to cut parts containing vertical walls, islands of material which are planar, and floors which are normal to the tool axis. Cavity Milling is designed to cut parts with tapered walls and contoured floors.

Cavity Milling should only be used in roughing operations since their tool paths normally will leave contoured scallops, stair steps and ridges. The scallops, stair steps, ridges and stock which remain from Cavity Milling operations should then be finished machined by using Fixed Contour operations.

Cavity Milling tool paths are easy to create. You can create tool paths by specifying only the Part and Blank (stock, forging, etc.) geometry and accepting the default option settings.

Selecting Part, Blank and Check Geometry

Cavity Milling, at a minimum, must have Part geometry defined. Blank (normally used) and Check geometry are optional. All three types of geometry may consist of faces, curves and/or bodies. Surface Regions and facets may also be used for Part and Blank geometry.



The difference between the Part and Blank geometry, defines the material to be removed. This material is often referred to as *volume* of material to be removed.

Blank geometry represents stock material (blank piece of stock) or can be a forging or casting that has a uniform offset from the selected Part geometry. This stock material may consist of faces, curves and or bodies (including facets).

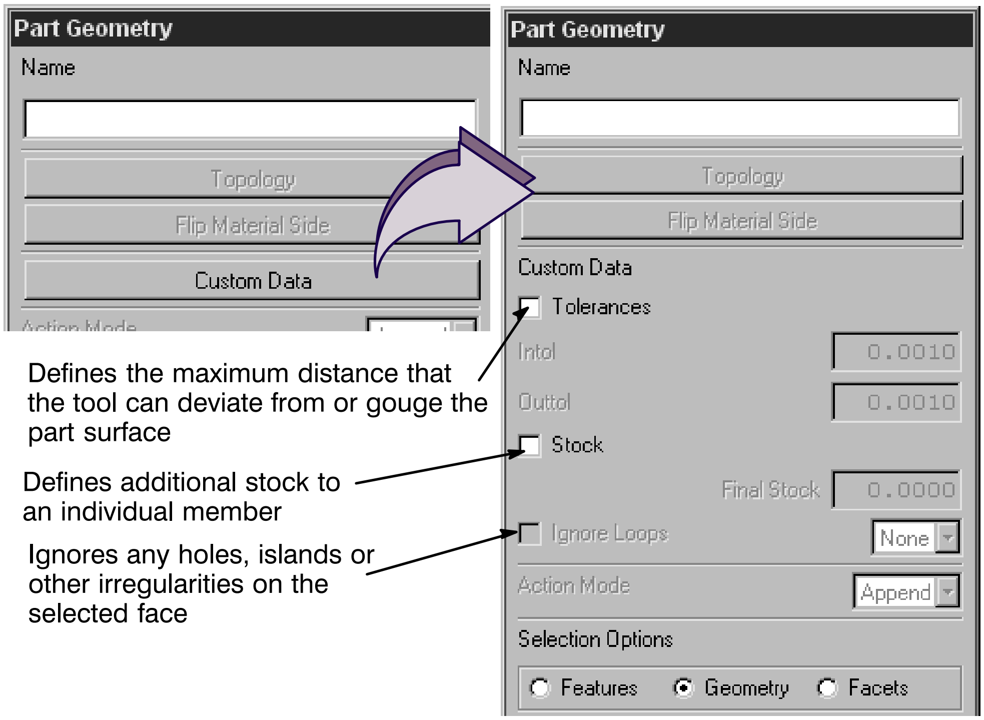
Blank geometry may also be used to cut specific areas. In this case the Blank geometry generates a boundary that surrounds only the Part and the material to be cut away. The processor would then cut from the Blank to the Part.

As mentioned previously, it is not necessary in all cases, to specify Blank geometry. You can use Part geometry only, if it encloses and defines the entire cut volume.

Custom Data

You can specify Custom Data for Part Geometry when creating a Geometry Parent Group. When you specify Custom Data for Part Geometry, when creating a Geometry Parent Group, the custom data is inherited by all operations that use that Parent Group.

You can also specify Custom Data for Part Geometry within an operation. When you select Part geometry, you can assign custom stock, tolerance, and tool conditions to the individual members for that operation only. When you select Custom Data from the Part Geometry selection dialog, the dialog expands to offer the additional options of tolerances, stock and ignore loops.



To make a custom assignment, you must:

- First select the option (Stock, Tolerance, etc.)
- Change the option value
- Select the specific geometry

The new value will be applied to any subsequent selections.

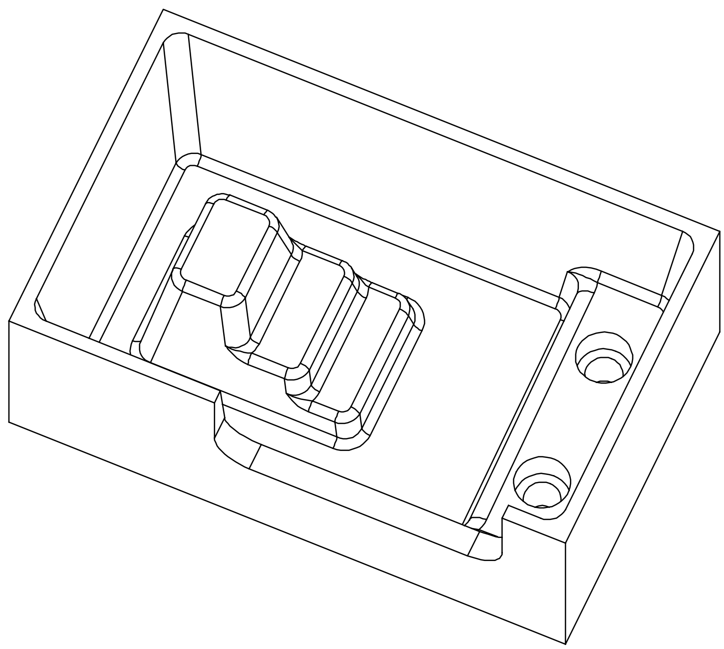
When assigning custom stock to geometry, assign equal amounts of stock to adjacent geometry. If stock is not applied in a consistent manner, the resulting tool path may not be what is expected.

Activity 2—1: Review of a Cavity Milling Operation

In this activity, you will create the Geometry, Tool, Method and Program Parent Groups that you will use to create a Cavity Mill operation. The purpose of this activity is to review the steps that are necessary to prepare a Cavity Milling operation.

Step 1 Open an existing part, save with a new name and enter the Manufacturing Application.

- ❑ Open the part **ama_form_mold_mfg.prt.**

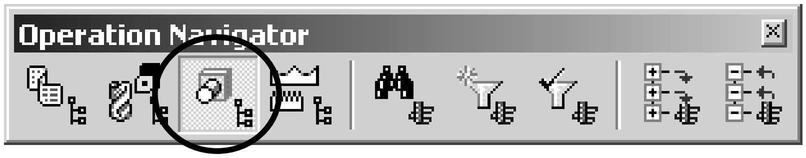


- ❑ Use the **Save→As** option under File on the menu bar and rename the part to *****_form_mold_mfg.prt** where ******* represents your initials.
- ❑ Choose **Application→Manufacturing.**

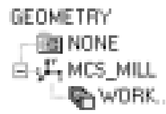
  The Operation Navigator is displayed.

Step 2 Create a Clearance Plane used for tool positioning for part clearance.

- ❑ Change to the Geometry View of the Operation Navigator (use the Geometry View icon).



The Geometry View of the Operation Navigator is displayed.



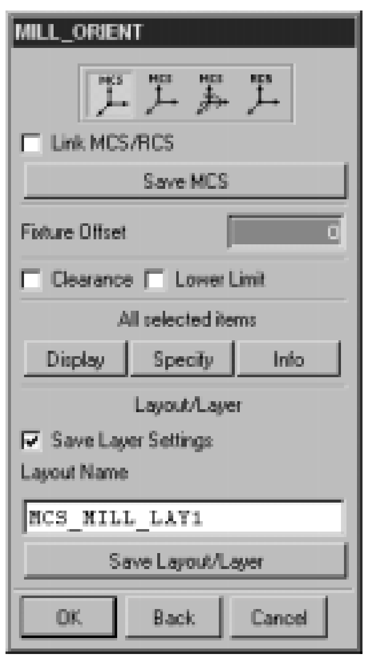
2

The Parent Group, MCS_MILL, has been provided for you. You will now verify the location of the MCS.

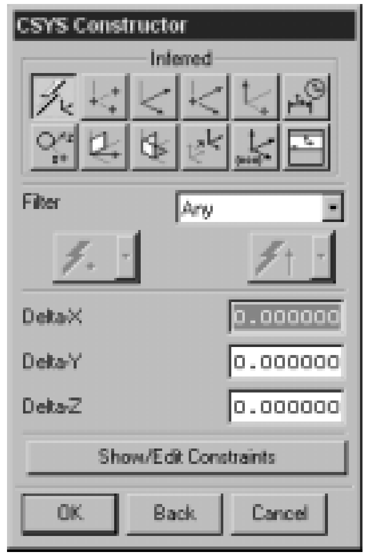
❑ Double click on the Parent Group, **MCS_MILL** in the Operation Navigator.

The Mill_Orient Dialog is displayed.

❑ As shown below, click on the **MCS** icon.



The CSYS Constructor dialog is displayed.



In the CSYS Constructor dialog, note that the values for Delta X, Y and Z are zero, indicating that the MCS currently, is not offset from the WCS, but is at the same location.

❑ Choose **Cancel.**

The Mill_Orient dialog is again displayed.

You will use the Assembly Navigator to display the Blank geometry.

❑ On your keyboard type **Control A** to activate the Assembly Navigator (or select the Assembly Navigator icon).

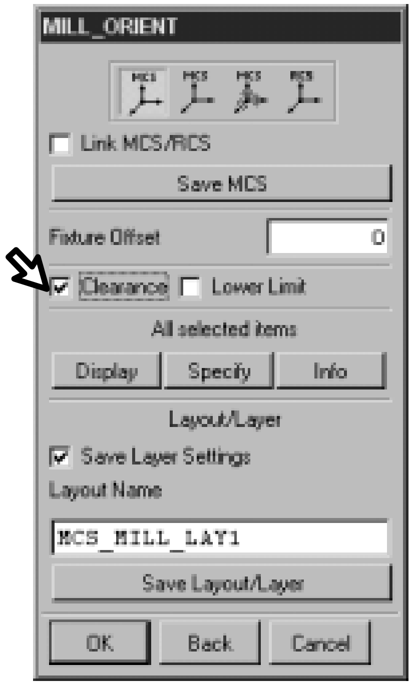
The Assembly Navigator is displayed.

❑ From the Assembly Navigator, select the **ama_form_mold_stock** component at the check box (this component has a gray checkmark) to make the Blank stock visible.

❑ On your keyboard type **Control A** to close the Assembly Navigator (or select the “**x**” on the upper right corner of the Assembly Navigator).

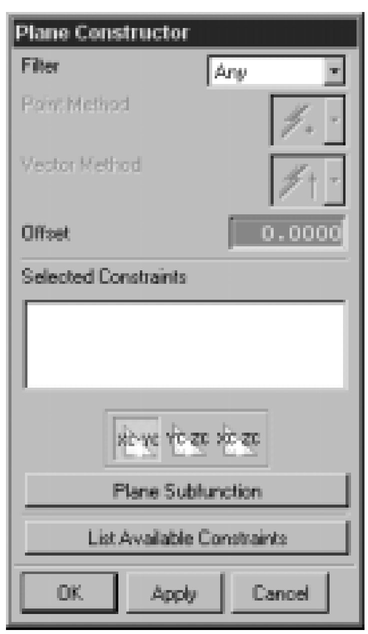
You will now create a clearance plane which will be used for tool positioning to clear the part.

❑ As shown below, select the Clearance option from the Mill_Orient dialog.

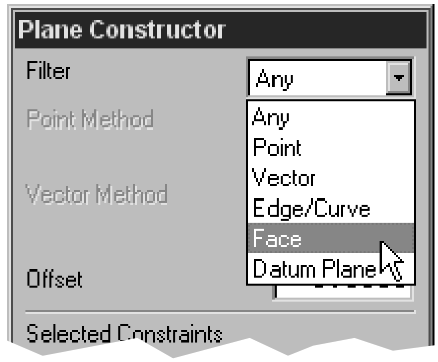


❑ Choose **Specify,** from the Mill_Orient dialog.

The Plane Constructor dialog is displayed.



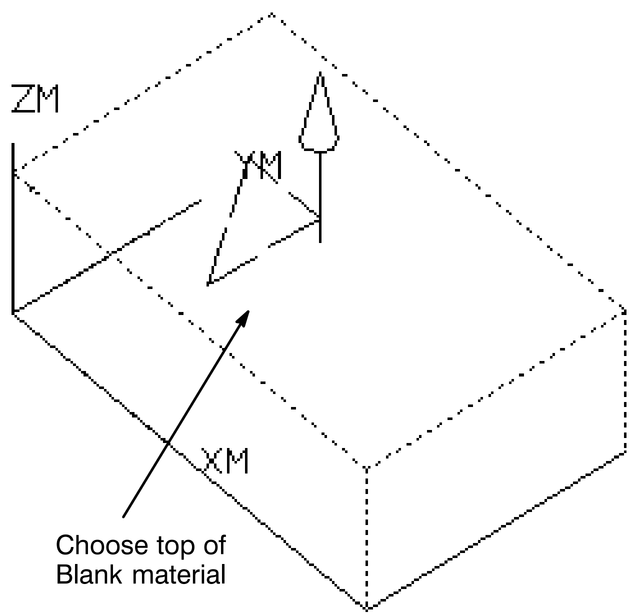
❏ From the Filter pull down menu select **Face**.



❏ Choose the top of the Blank geometry.

❏ Key in **.500** in the Offset field of the Plane Constructor dialog.

❏ Choose **OK**.



Note the Plane symbol is displayed .500 above the Blank stock geometry.

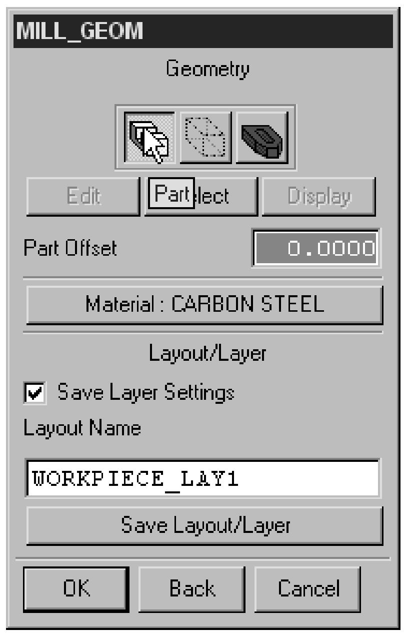
❏ On the Mill_Orient dialog, choose **OK**.

You will now select the Part Geometry under the Geometry Parent Group, Workpiece.

Step 3 Edit the Geometry Parent Group.

❑ Double click on the **WORKPIECE** in the Operation Navigator.

The Mill_Geom dialog is displayed.

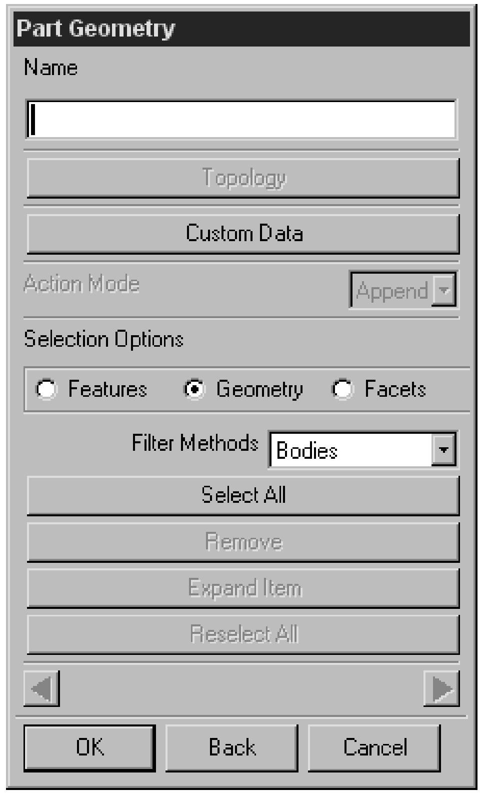


Remember, that Cavity Milling and Fixed Contour operations use Mill_Geom Parent Groups. The Parent Group named WORKPIECE is a Mill_Geom Parent Group that was previously created.

❑ Choose the **Part** icon under the Geometry label.

❑ Choose the **Select** button.

The Part Geometry dialog is displayed.



The Cue line prompts you to select Part Geometry. You have the option of selecting Features, Geometry or Facets.

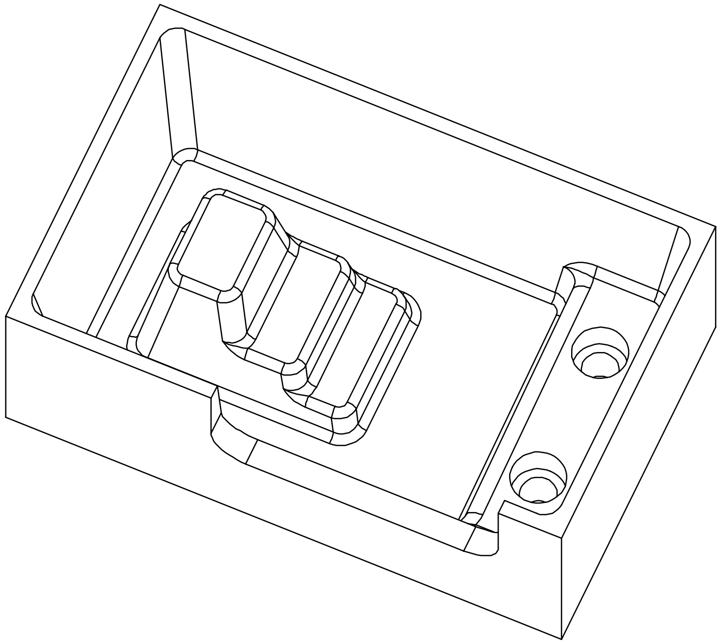
You are going to select Geometry.

❑ If necessary, choose the **Geometry** option located in the Selection Options area.

Note that the Filter Methods pull down menu default selection is for Bodies. The Part Geometry that you will be selecting is a single Body.

❑ Select the entire part as the Part Geometry (Make sure that you select the part and not the blank).

2

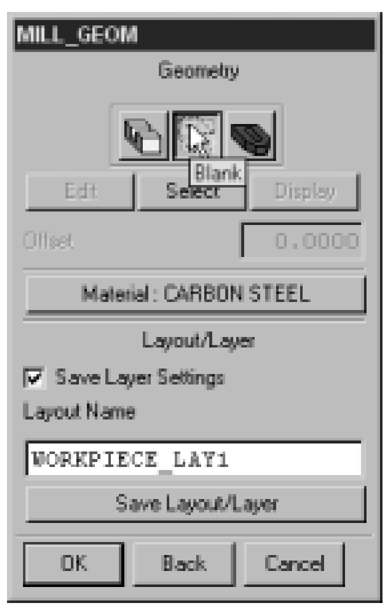


❑ Choose **OK** and return to the Mill_Geom dialog.

You will now select the Blank Geometry.

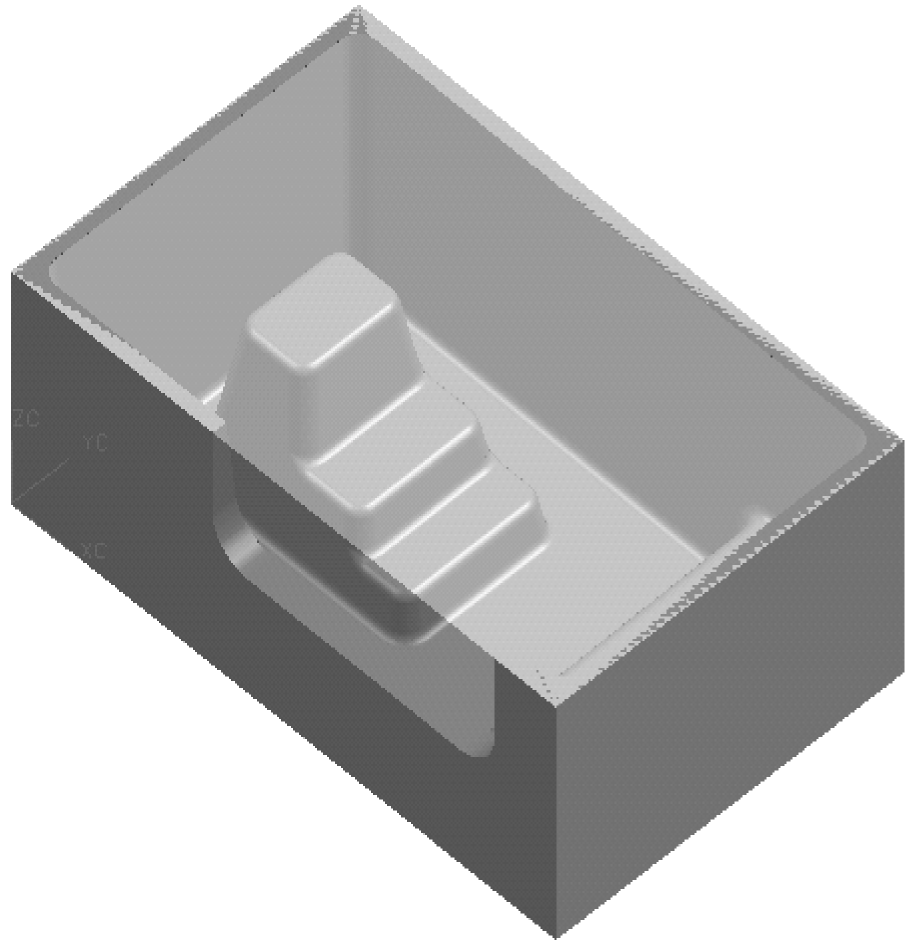
Step 4   Specify Blank Geometry.

❑ Choose the Blank icon.



❑ Choose **Select**.

Make sure that the Blank Geometry surrounding the Part Geometry is displayed.



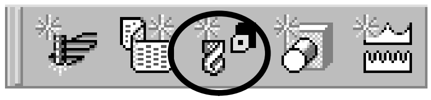
❑ As shown above, choose the Blank part as the Blank Geometry.

❑ Choose **OK**, twice.

You have selected Part and Blank Geometry which are now included in the Geometry Parent Group, Workpiece. Next you will create .750 x .060 corner radius tool for machining the cavity. This tool will become a member of the Tool Parent Group.

Step 5 Creating the Tool for machining the cavity.

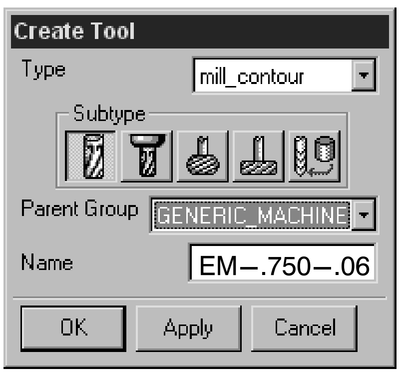
❑ As shown below, select the **Create Tool** icon from the Create toolbar.



The Create Tool dialog is displayed.

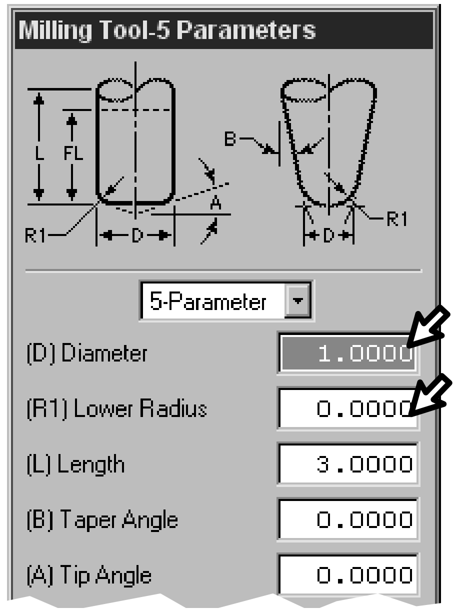
❑ On the Create Tool dialog, select the milling tool subtype.

❑ Enter EM−.750−.06 in the Name field.



❑ Choose **OK**.

The Milling Tool- 5 Parameters dialog is displayed. You will change the tool diameter from 1.000” to .750” and the lower radius from 0.0” to .060”.



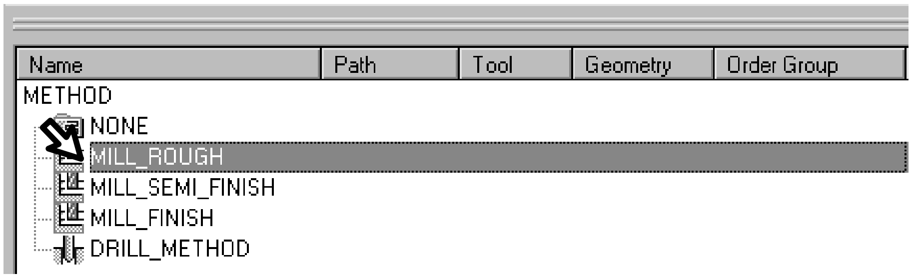
2

- ❑ Enter **.750** as the Diameter
- ❑ Enter **.06** as the Lower Radius.
- ❑ Choose **OK.**

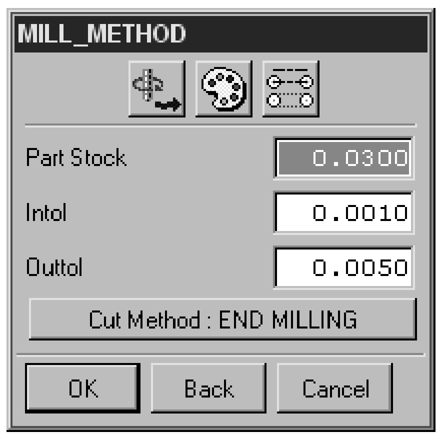
Step 6 Examine the Method Parent Group.

The Method Parent Group, Mill_Rough has already been created for you. You will examine the options available under this Parent Group.

- ❑ In the Operation Navigator Machining Method View, double click on **MILL_ROUGH**.



The Mill Method dialog is displayed. Note the tolerances and Part Stock. These values apply to all Operations that use the Parent Group, Method unless other stock values are specified within an Operation. Specifying stock values within an Operation overrides the inheritance of the Part Stock value from the Method Parent Group to the Operation.

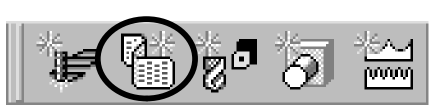


❑ Choose **OK.**

You will now create a program, named Interior, in the Program Parent Group which will be used in postprocessing the operations which you will create.

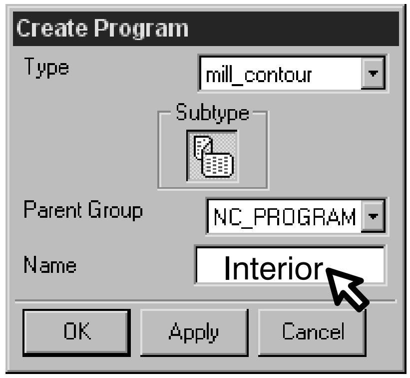
Step 7 Creating a Program.

❑ As shown below, select the **Create Program** icon from the Create toolbar.



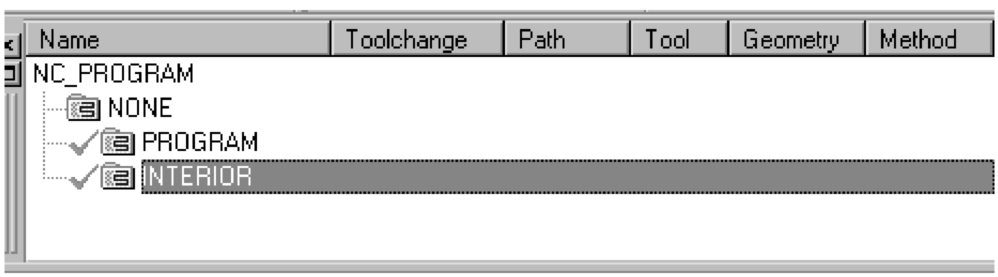
The Create Program dialog is displayed.

❑ On the Create Program dialog, select **Name** and enter the name **Interior** as shown below.



❑ Choose **OK**.

In the Operation Navigator Program Order View, the name INTERIOR is displayed



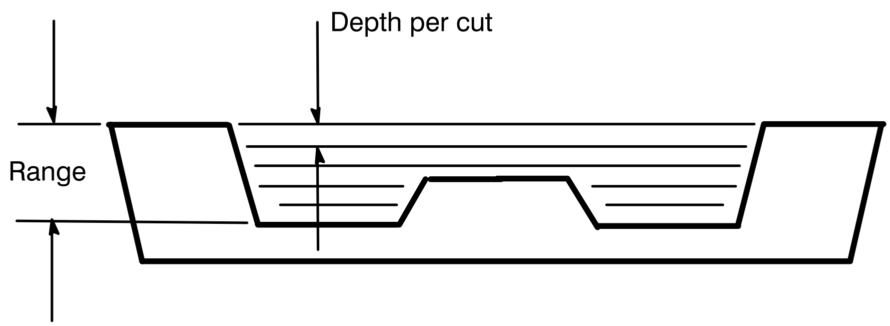
❑ **Save** the part file.

You have completed this activity. The Geometry, Tool and Program Parent Groups that you have created can now be used to create Operations and will be used in the next activity.

Cut Levels

Cut Levels establish parallel cut planes for Cavity Milling operations. Cut levels consist of ranges and depths per cut plane within the ranges.

A range consists of two planes perpendicular to the tool axis that define the cut volume. Multiple ranges may exist within an operation. Each range is evenly divided into cut depths. The processor determines the first range based on the highest and lowest points of the cut volume defined by the Part and Blank geometry.

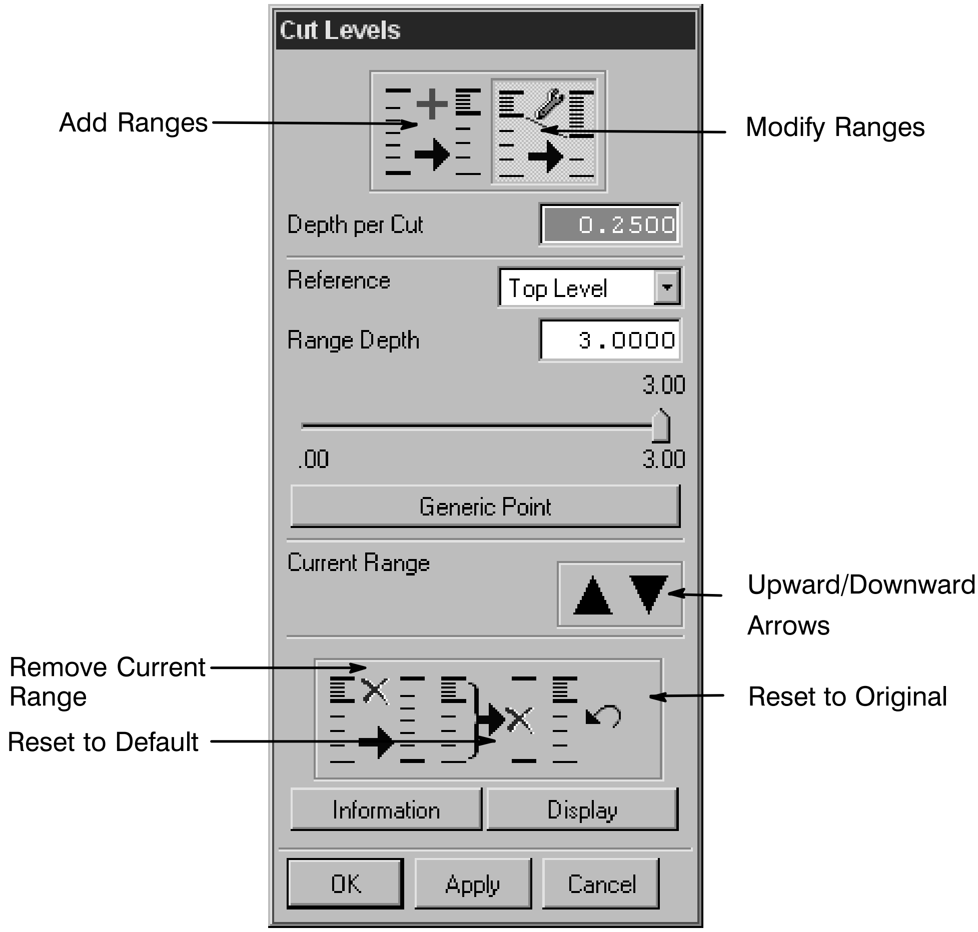


Visually, ranges are displayed by large plane symbols while cut depths within the range(s) are displayed by small plane symbols.

Multiple Ranges

Only one range may be active at a time. The currently active range is highlighted for visual reference and indicated by a number in the Status Line. Cut depths are added to a range by making that range active through the selection of up or down arrows and then specifying a Depth Per Cut value.

Ranges may be modified, added, or deleted. When a range is modified, the bottom plane moves upwards or downwards. Any ranges above or below will expand or contract to fill the void area. When a range is added, it is created above the specified plane and extends up to the bottom of the previous range, or to the top level of the cut volume if there are no ranges above. When a range is deleted, the one beneath it will fill in the gap from the top down.



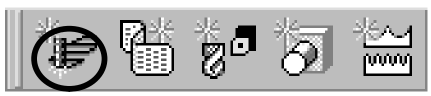
Cut Level Dialog

Activity 2–2: Cavity Milling using cut levels and ranges

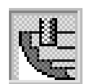
In this activity, you will create a Cavity Milling Operation to machine the interior geometry, using cut levels and cut ranges.

Step 1 Creating a new Cavity Milling Operation.

- ❑ Continue using *****_form_mold_mfg.prt.**
- ❑ As shown below, select the **Create Operation** icon from the Create toolbar.

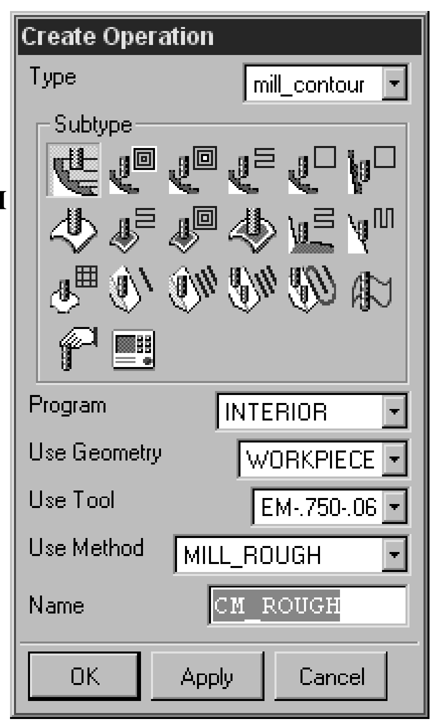


The Create Operation dialog is displayed (if necessary, select Type as mill_contour).

- ❑ Select the **Cavity Milling** icon. 
- ❑ On the Create Operation dialog, select and enter as shown below.

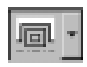
Set:

Program:	**Interior**
Use Geometry:	**Workpiece**
Use Tool:	**EM-.750-.06**
Use Method:	**MILL_ROUGH**
Name:	**CM_ROUGH**



❑ Choose **OK.**

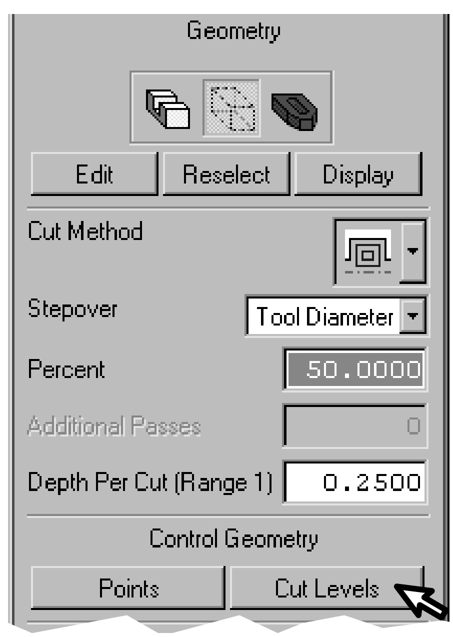
The Cavity_Mill dialog is displayed.

❑ In the Cavity Milling dialog, change the Cut Type to **Follow Periphery**. 

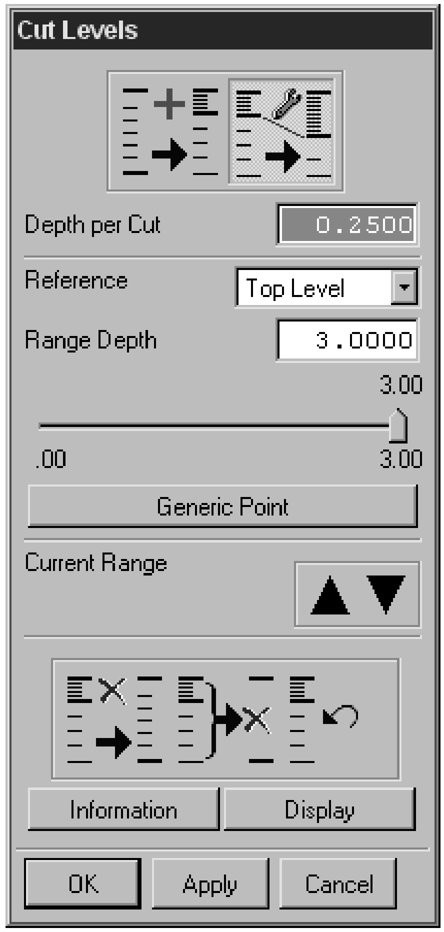
You will now select the cut levels and cut ranges for cutting the part.

Step 2 Define the Cut Ranges and Cut Level for the first range in the cavity.

❑ Choose **Cut Levels** in the Control Geometry section of the Cavity Mill dialog.

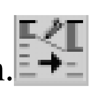


The Cut Levels dialog is displayed.

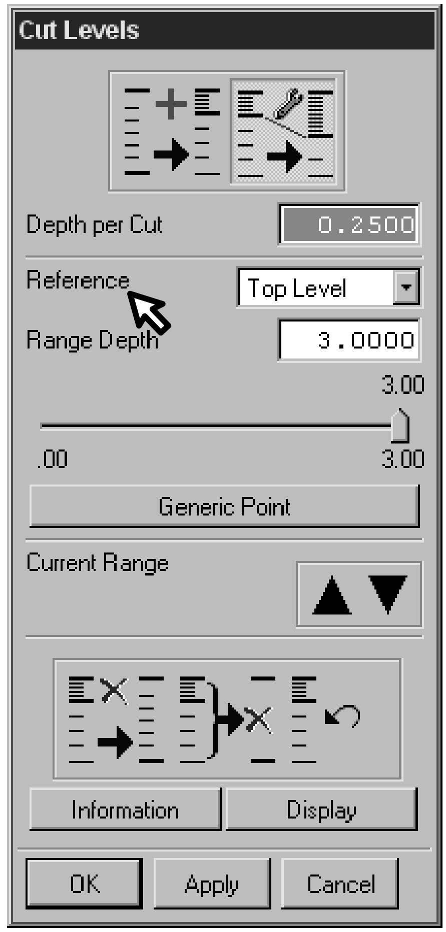


Notice in the graphics area, two plane symbols are displayed. They define the top and bottom of the first range.

Since the first range is defined for you, you must choose the Modify Range button in order to make any changes to the range.

❑ Choose the **Modify Range** icon. 

Notice that the Reference option is set to Top Level.



The Reference option determines where the Range Depth value you specify is referenced from.

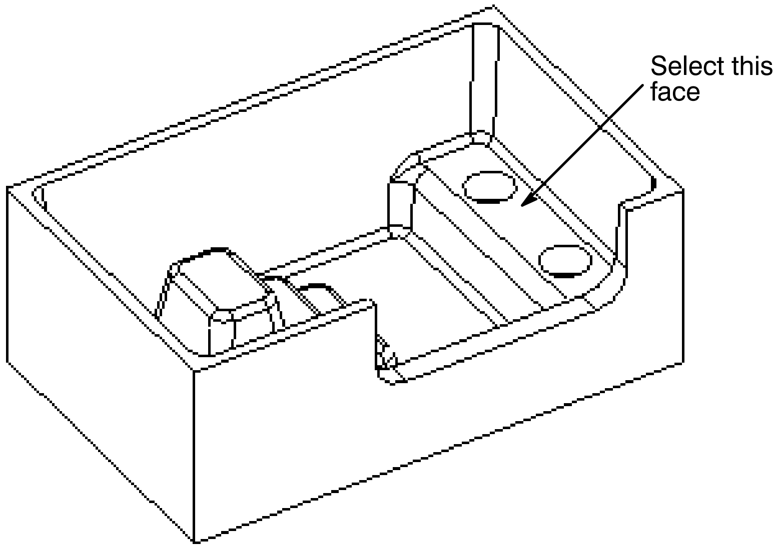
You will use the default setting of Top Level.

You are now going to modify the bottom of the first range. By default, it is defined at the lowest point of the cavity.

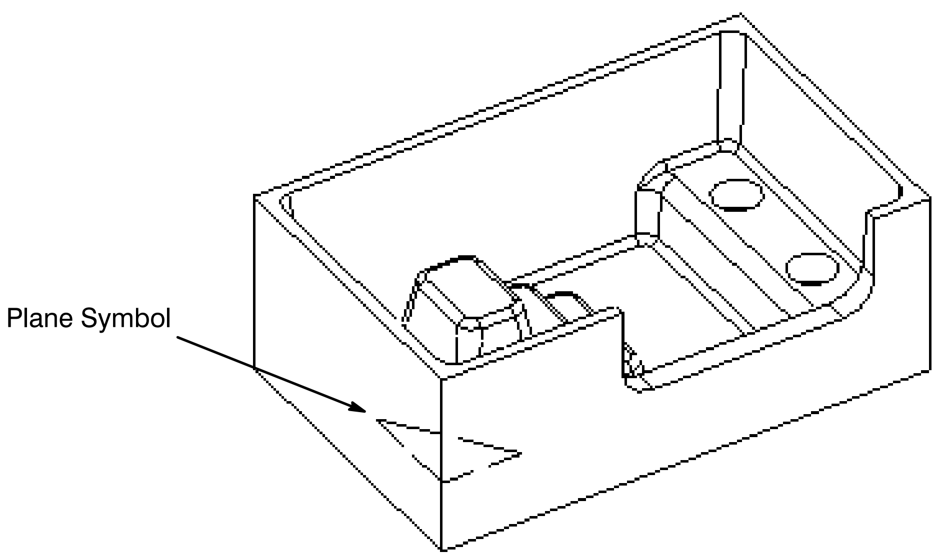
The new location will be at the top of the step.

2

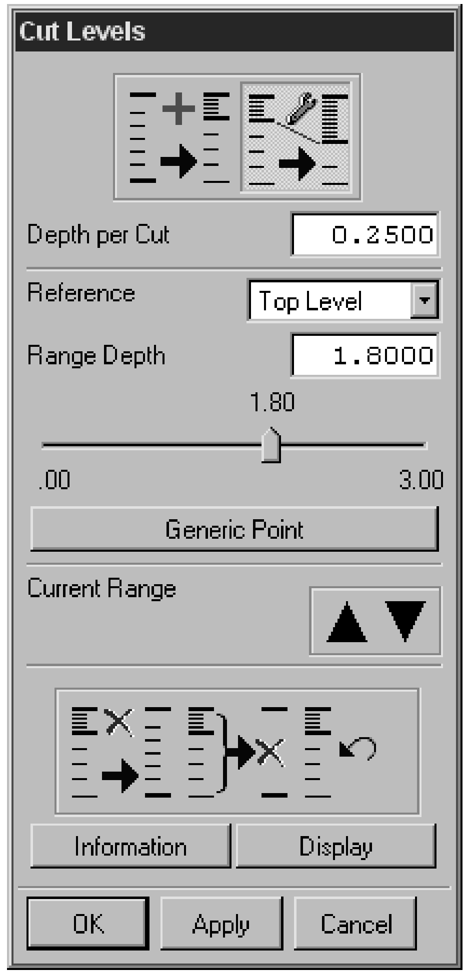
❑ Choose the top face of the step as shown.



A plane symbol is displayed and you are returned to the Cut Levels dialog.



The Range Depth is displayed on the dialog. It is the distance between the top and bottom of the range.



You will now define the maximum depth per cut within the specified range.

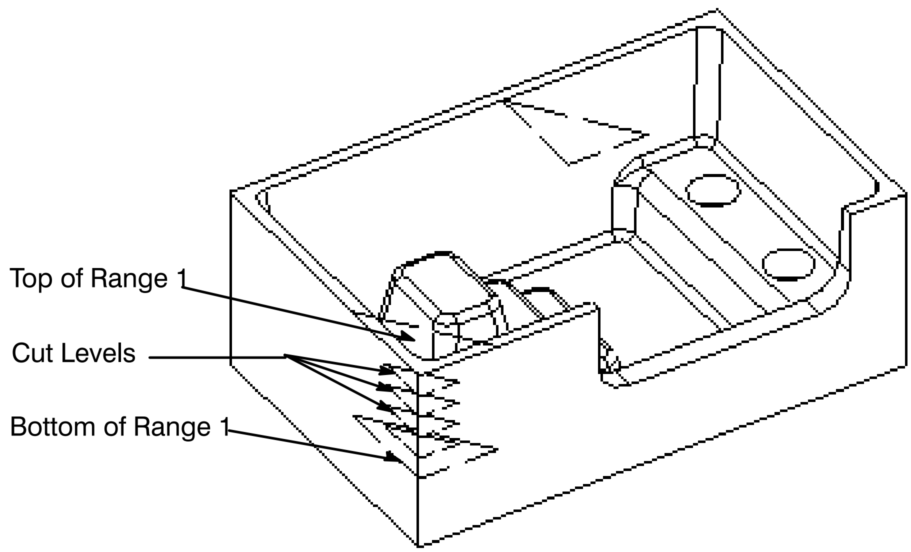
You are going to specify the cut depth in the Depth per Cut value field.

For this activity, you will divide the cut into four steps. You will divide the total depth of 1.800 by 4.

❑ Key in **1.8** / **4** in the Depth per Cut field and press the keyboard return or **Enter** key.

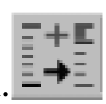
2

The Cut Levels are displayed as small plane symbols.



Step 3 Define the second Range.

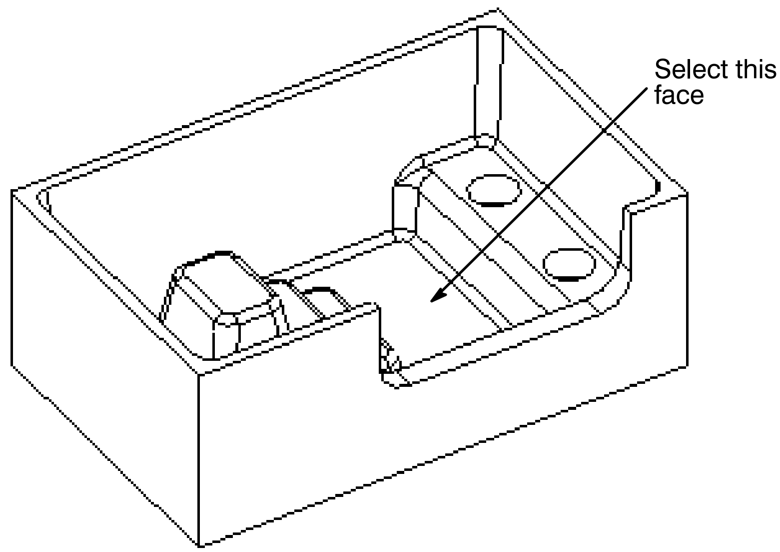
The Cut Levels dialog is displayed. You will now add another range.

❑ Choose the **Add Ranges** icon. 

You are not going to change the Reference option. You want to reference the range from the top of the Top Level which in this case is the top of the part.

The top of Range #2 is defined for you since the top of the next range always uses the bottom of the previous range.

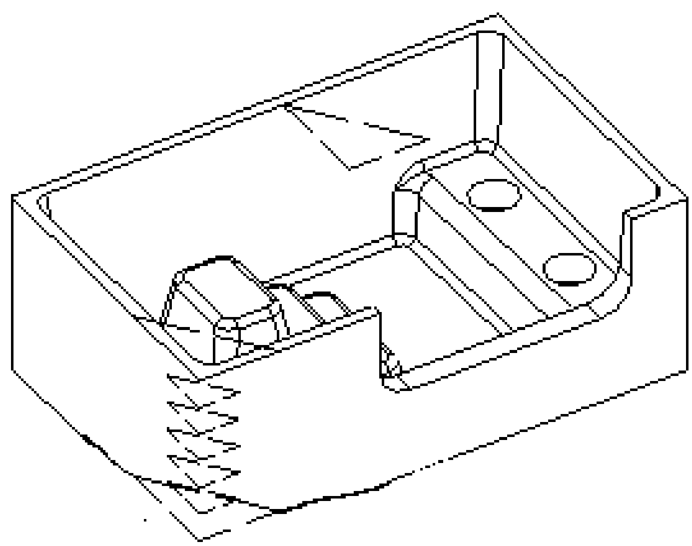
❑ Choose the bottom face of the pocket as shown.



You are going to specify the depth of cut next. You will remove material in two steps.

- ❑ Key in **(2.8−1.8)** / **2** in the Depth per Cut field and press the return or **Enter** key.

This range will be cut in two passes at .500 per cut.



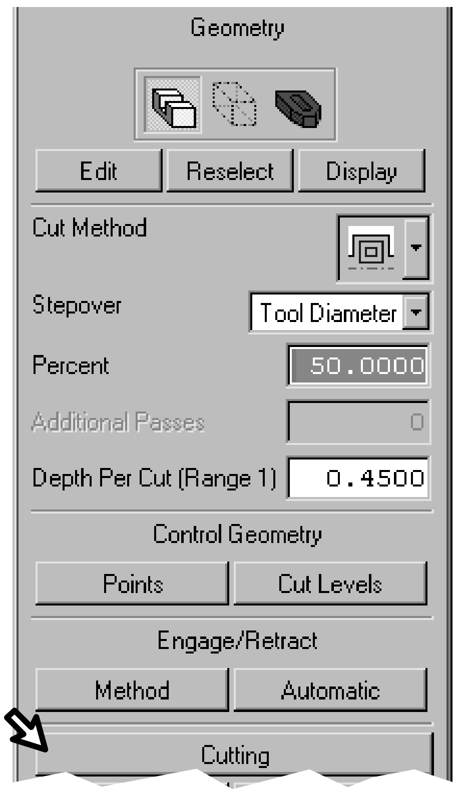
- ❑ Choose **OK**.

The Cavity Mill dialog is displayed.

You can change the cut level settings at any time by reselecting the Cut Level option.

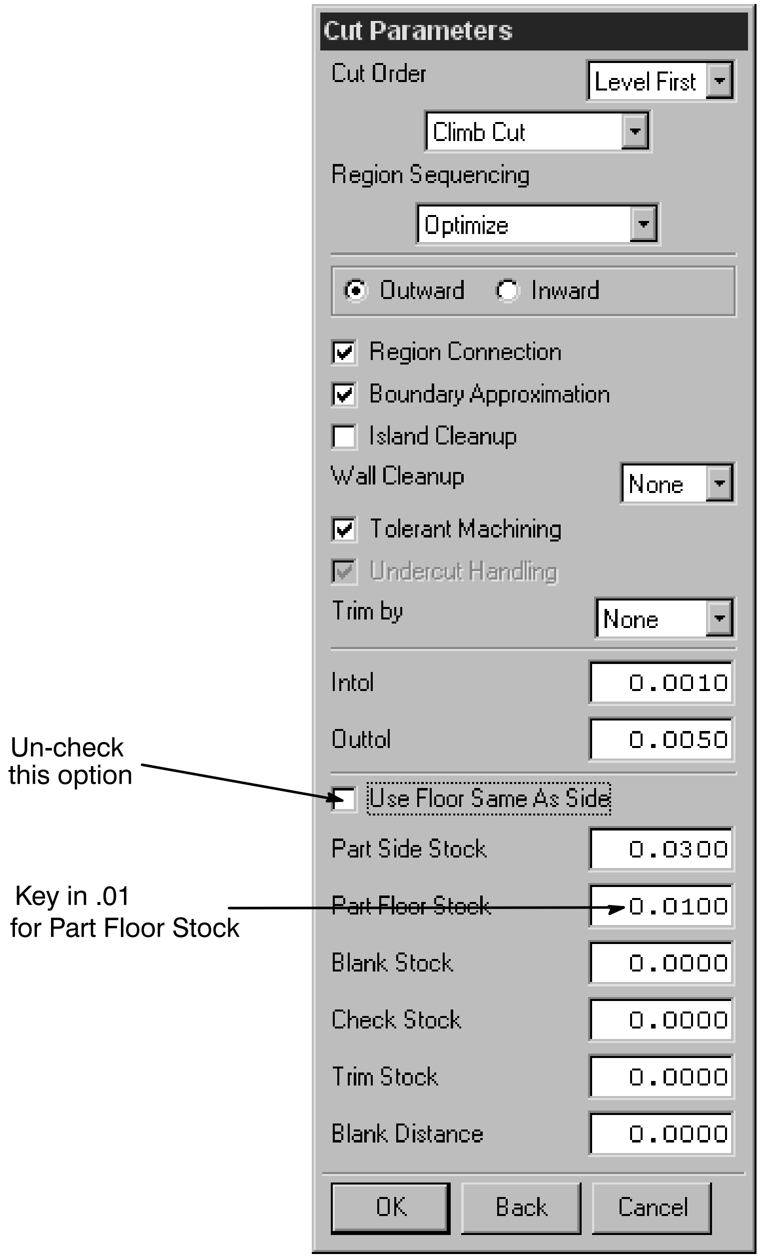
Step 4 Specify Cutting options.

You will now add specific amounts of stock to all planar floors. You will add stock to the step and bottom of the pocket that have been used as cut levels. This stock value overrides the Part Stock specified in the Parent Group, MILL_ROUGH, since it is specified within the operation.



❑ Choose **Cutting.**

The Cut Parameters dialog is displayed.

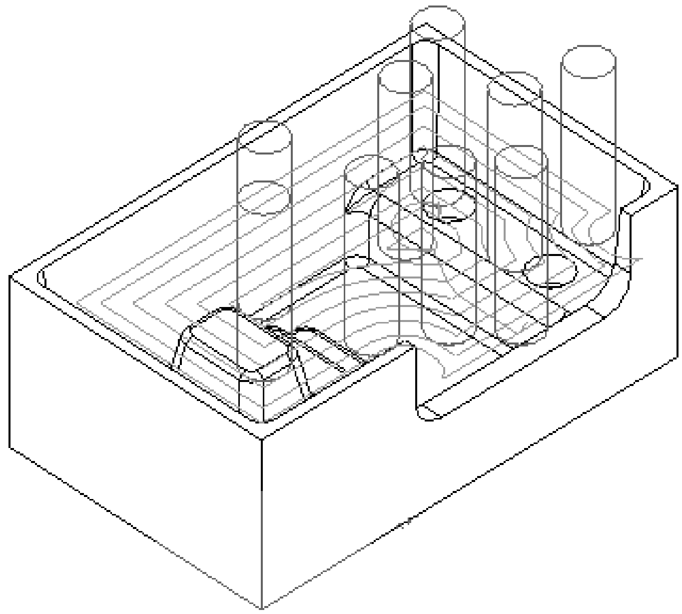


- ❑ Un-check **Use Floor Same As Side** (this prevents the floor from having the same stock value as the side).
- ❑ Key in **.01** for Part Floor Stock.
- ❑ Choose **OK.**

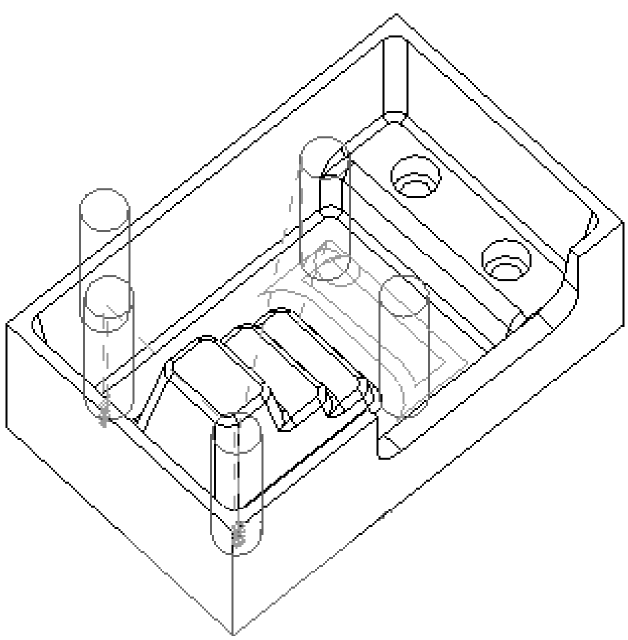
Step 5 Generate the tool path.

- ❑ Choose **Generate**.

The first level of the tool path is generated.



❑ Choose **OK** until all the levels of the tool path are generated.



Notice that the tool path is generated at all of the specified cut levels.

Step 6 Save this Operation.

You will edit this operation later in this lesson.

❑ Choose **OK** from the Cavity Milling dialog.

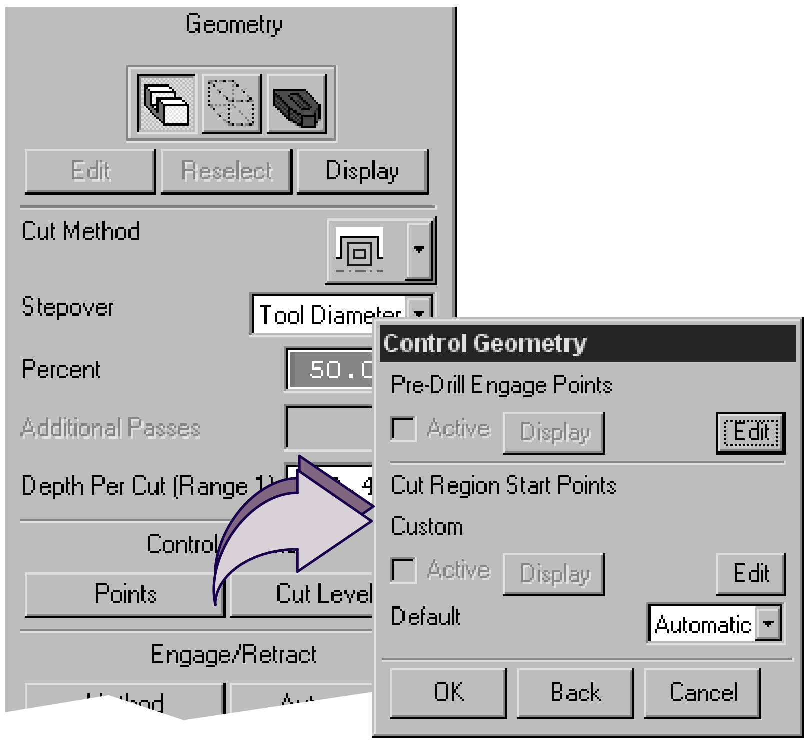
The operation is saved.

❑ **Save** the part file.

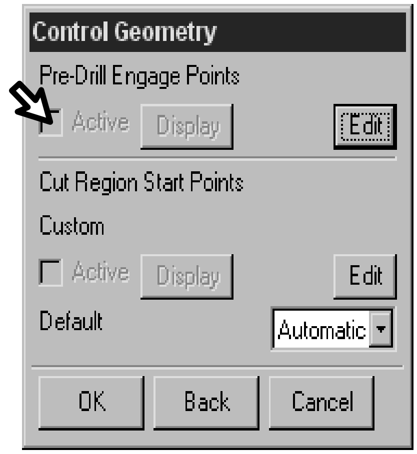
You have completed this activity.

Pre-Drill Engage and Cut Region Start Points

Pre-Drill Engage and Cut Region Start Points are found in the Control Geometry, Points section of the Cavity Milling dialog. These two options provide control over the cutting start point within single and multiple regions of Cavity Milling. They also determine the direction that the tool moves towards the cavity or core walls.



Pre-Drill Engage Points



The internal processor of Cavity Milling determines the tool path start point.

You can use the Pre-Drill Engage Points option to specify where you want the tool to start cutting. With this option, the tool moves to the pre-drill engage point you specify, then to the specified cut level. It then moves to the processor generated start point and then generates the remainder of the tool path.

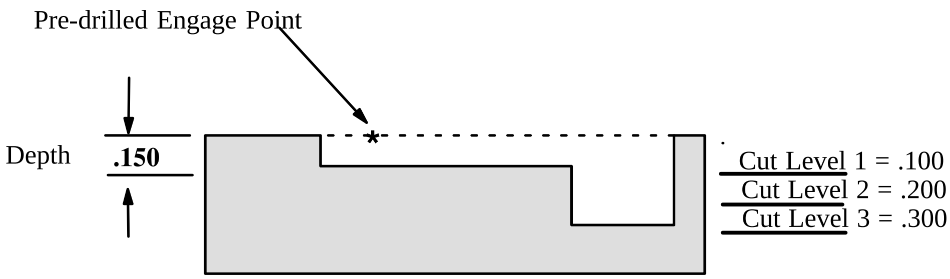
To use this option, specify a pre-drill engage point and an optional depth value. If you are going to specify a depth value, it must be done prior to specifying the start point.

There are three methods available for specifying pre-drill engage points:

- **Point/Arc -** by using existing points or arcs. The arcs are associative to the geometry. They must be explicit or sketch curves.
- **Cursor -** by using the cursor position.
- **Generic Point -** by using the option on the generic point dialog.

The depth value for a start cut point is optional. If you do not specify a value, the pre-drill engage point is used at every cut level.

As shown below, cut level 1 uses the pre-drill engage point that falls within the specified depth. Cut levels 2 and 3 do not use the specified pre-drill engage point since the cut levels are not within the specified depth. The processor will use the internally defined cut start point to cut the remaining cut levels (2 and 3).

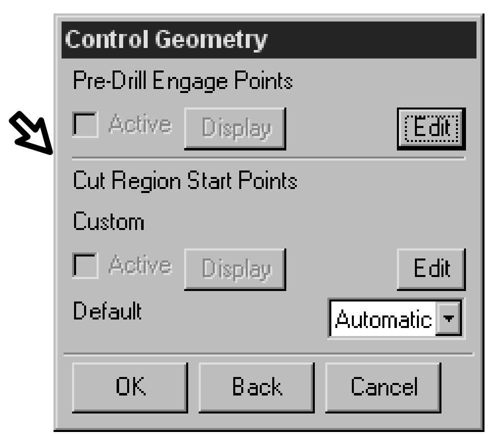


All specified depths are measured from the top plane.

NOTE You can define Pre-drilled points using either the Engage/Retract dialog or the Pre-drill Start Points option located under the Control Geometry button. The Engage/Retract Pre-drill points and settings override the points defined under the Control Geometry Option. If you specify multiple Pre-drill points you can optimize the order in which they are drilled by customizing the Engage/Retract options which are available.

Cut Region Start Points

Cut Region Start Points allows you to specify cut start points for each region in a multi-region cavity. When you use circular engages, this option can avoid engages into pocket corners by using the Automatic or User Defined method of engagement.



The Cut Region Start Points defaults are as follows:

Automatic establishes the Cut Region Start Point at the “flattest” convex corner of the cut region. If there are no convex corners, the midpoint of the longest boundary segment of the cut region is used. This option assures that the tool will stepover or engage the part at a location which is least likely to cause the tool to become buried in the material.

Standard establishes the Cut Region Start Point as close as possible to the start point of the boundary region. The shape of the boundary, cut type, and position of islands and pockets will influence how closely the processor positions the Cut Region Start Point to the Boundary Start Point. Moving the Boundary Start Point affects the location of the Cut Region Start Point.

Activity 2—3: Using a Pre-Drill Engage Point

In this activity, you will edit the current operation to use a Pre-drilled Engage Point to start your tool path. The Pre-drill Engage Point is a hole that has been previously drilled.

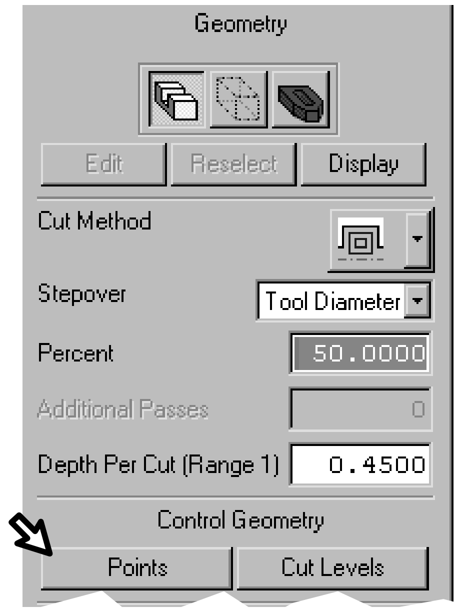
Step 1 Edit an existing operation.

- ❑ Continue using ***_form_mold_mfg.prt**
- ❑ Double-click on the **CM_ROUGH** operation in the Operation Navigator.

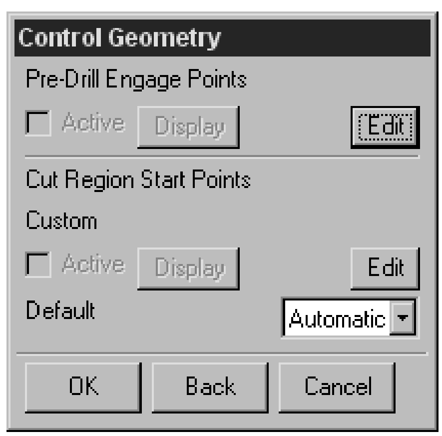
  The Cavity Milling dialog is displayed. You will now define a point that represents a hole which has been previously drilled. This will be the engage point for the tool that is used to start each cut level.

Step 2 Define a Pre-drill Engage Point for this operation.

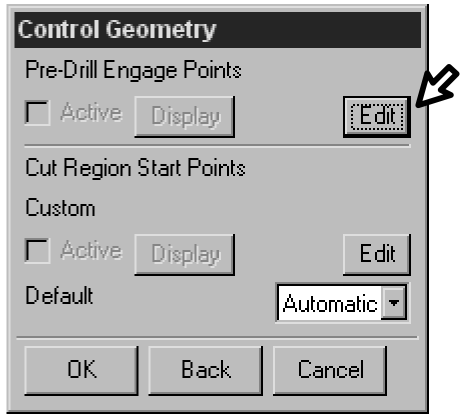
❑ Choose **Points** in the Control Geometry section.



The Control Geometry dialog is displayed. Notice that there are two sections to this dialog, Pre-Drill Engage and Cut Region Start Points.

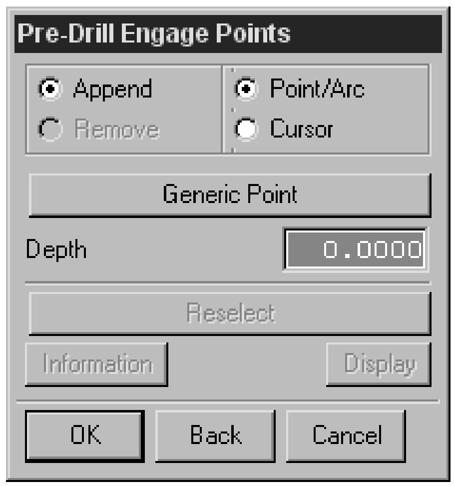


❑ Choose **Edit** in the Pre-Drill Engage Points section.



2

The Pre-Drill Engage Points dialog is displayed.



You can use the Depth parameter when you want a particular Pre-Drill Engage Point to be used only for certain cut levels. If you do *not* specify a Depth parameter, the point will be used at all cut levels. If you use the Depth parameter it must be defined *before* specifying the point.

For this activity, you will not specify a Depth parameter. This particular Pre-Drill Engage Point will be used at all cut levels.

❑ Choose **Generic Point**.

The Point Constructor dialog is displayed.

❑ Key in the following values
XC=5
YC=2.5
ZC=0

- ❑ Choose **OK**.

  The point just created is displayed (this point is at the bottom of the part, if your display setting is solid, set to wireframe to see the point).

- ❑ Choose **OK** until you return to the Cavity Milling dialog.

Step 3 Generate the tool path.

- ❑ Choose the **Generate** icon to create the tool path.

  Notice that *all* levels start at the Pre-Drill Engage Point in the center of the part, then move to the start point which is determined by the processor.

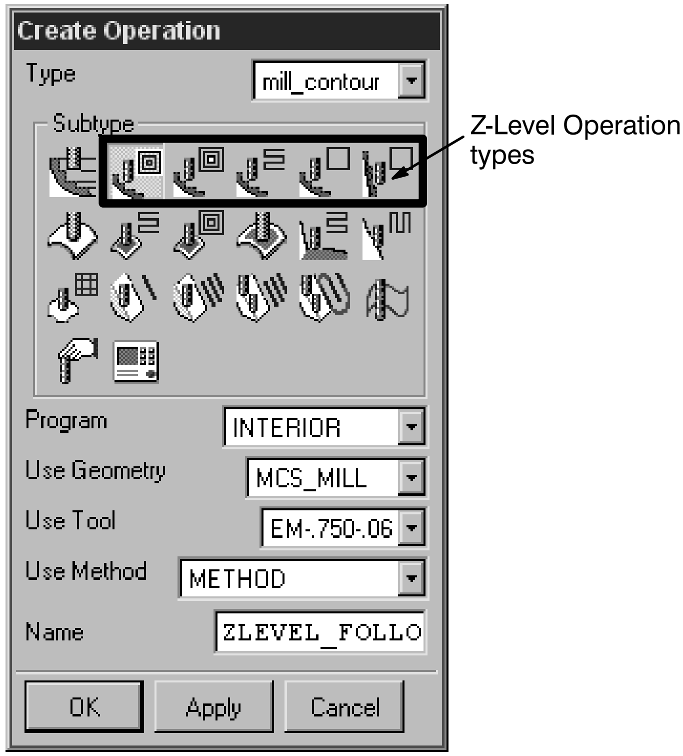
- ❑ Choose **OK** to accept the operation.

- ❑ **Save** your Part.

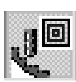
  You have completed this activity.

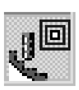
Z-Level Milling Operations

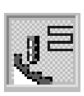
Z-Level Milling operations are used to profile solid bodies and or faces at multiple levels.

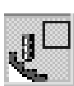


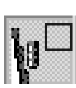
The following Z-Level Operation types are available:

-  **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.

Z-LEVEL_PROFILE and Z-LEVEL_PROFILE_STEEP operation types  allow both the cutting of steep areas only or cutting of the entire part.

As an option, you may specify Cut Area  as a subset of the part geometry in order to limit the area(s) that are to be cut. If no Cut Area geometry is defined, the entire part is considered to be the cut area.

Associated with the Cut Area option are Trim Boundaries.  Trim Boundaries further constrain cut regions. You may define the area of the cut region to be excluded from the operation by specifying Material as Inside or Outside. Trim Boundaries are always closed, always use a tool ON condition, and are projected along the tool axis vector to the part. You can have more than one Trim Boundary defined. You may also specify a Trim Stock to define the distance the tool is positioned from the Trim Boundary as well as Intol/Outol values for boundary members.

When generating a tool path, tracing of the geometry occurs, steep areas are detected, traced shapes are ordered, and subsequent cut areas to be machined are detected and then cut at all cut levels.

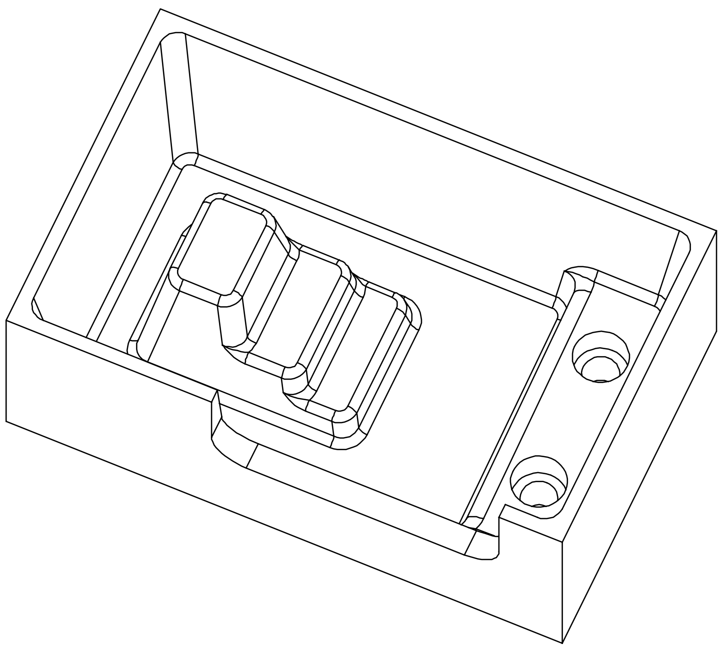
Activity 2—4: Z-Level Profile Milling

In the following activity, you will create a Z-Level Profile operation to machine the steep geometry of the island within the cavity. You will create a Geometry Parent Group (MILL_AREA) that contains the steep geometry necessary for machining. The tool path will cut only within the area specified.

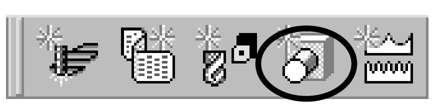
2

Step 1 Create the Geometry Parent Group.

❑ Continue using *****_form_mold_mfg.prt**

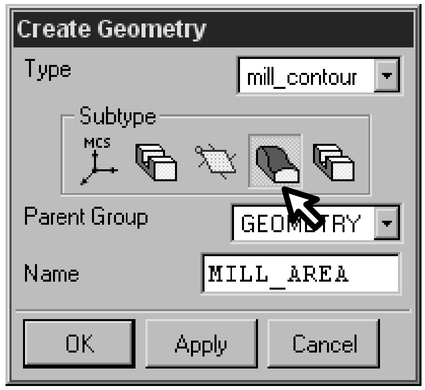


❑ As shown below, select the **Create Geometry** icon from the Create toolbar.

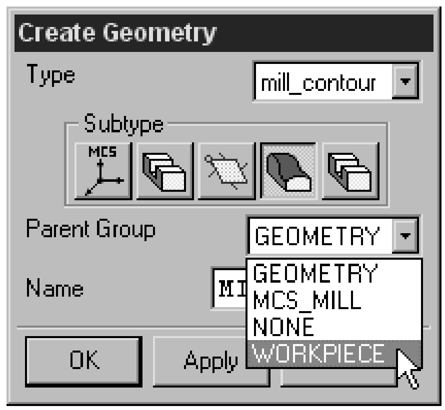


The Create Geometry dialog is displayed (make sure Type is mill_contour).

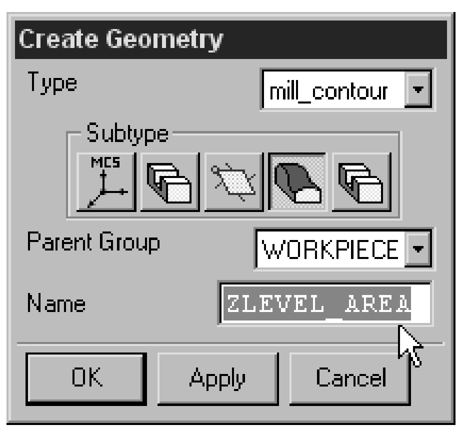
❑ In the Create Geometry dialog select the Mill_Area icon.



❑ Select **WORKPIECE** as the Parent Group.



❑ Enter **ZLEVEL_AREA** as the Name.



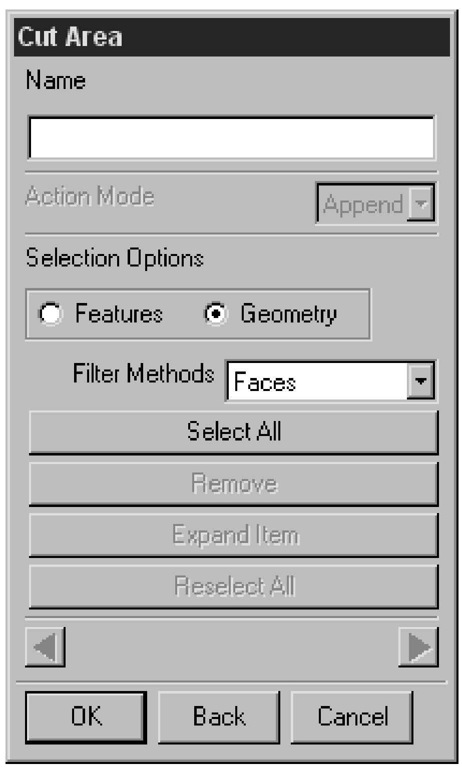
❑ Choose **OK**.

The MILL_AREA dialog is displayed.

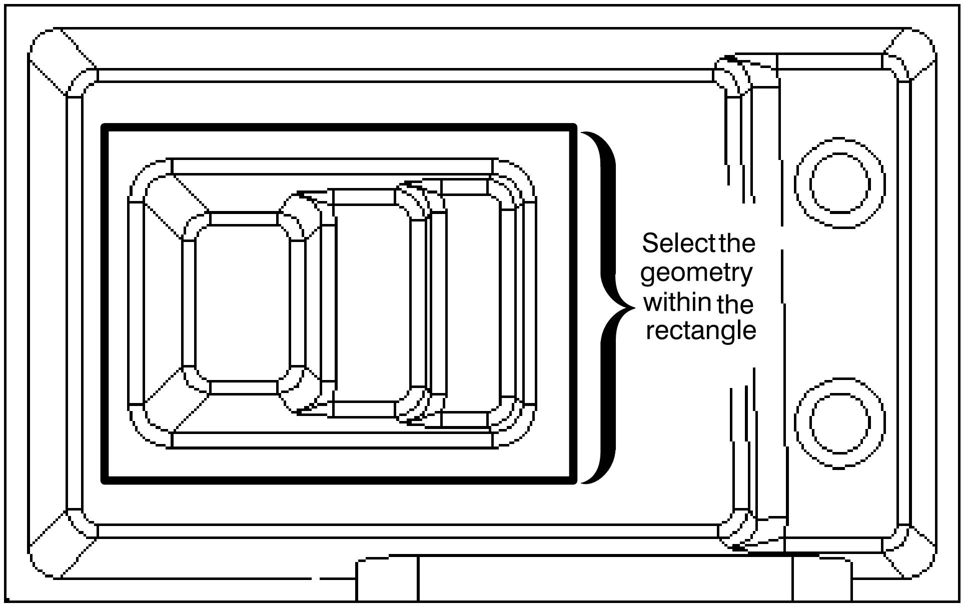
❑ Choose the Cut Area icon.

❑ Choose **Select**.

The Cut Area dialog is displayed.



❑ Select the interior island geometry as shown.



Make sure that when selecting with a rectangle, selection criteria should be inside only.

Note: If the following message appears, select OK until all objects are selected.

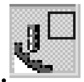
❑ Choose **OK**, twice to return to the Create Geometry dialog.

Note that you do not need to specify Blank Geometry.

To briefly review ----- you have created a geometry Parent Group, named ZLEVEL_AREA which contains the geometry of the island. This Parent Group will be used in the ZLEVEL_PROFILE operation.

You will now create the operation.

Step 2 Create the ZLEVEL_PROFILE Operation.

- ❑ Choose the Create Operation icon.
- ❑ Select the **ZLEVEL_PROFILE** icon. 
- ❑ Set the following:
  - Program: **INTERIOR**
  - Use Geometry: **ZLEVEL_AREA**
  - Use Tool **EM–.750-.06**
  - Use Method: **MILL_FINISH**
- ❑ Choose **OK.**

  The ZLEVEL_PROFILE dialog is displayed.

- ❑ Under the Geometry label, select the Part icon and choose **Display.**

  The Part Geometry is displayed. It was specified in the WORKPIECE Parent Group.

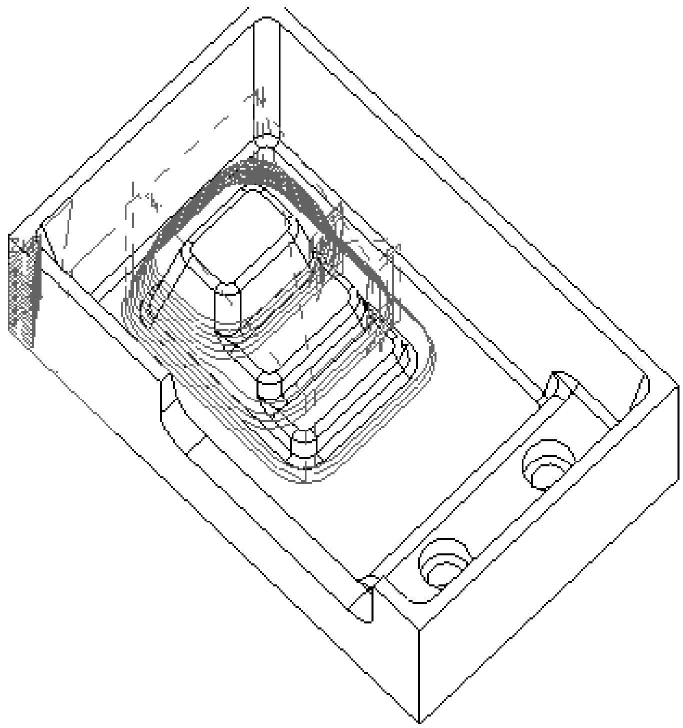
- ❑ Under the Geometry label, select the Cut Area icon and choose **Display.**

  The Cut Area Geometry is displayed. It was specified in the ZLEVEL_AREA Parent under the WORKPIECE Parent Group.

- ❑ Change the Depth Per Cut to **.15.**
- ❑ Change Cut Order to **Depth First.**

Step 3 Generate the tool path.

❑ Choose the **Generate** icon to generate the tool path.



❑ Choose **OK** to save the Operation

❑ **Save** the part file.

You have completed this activity.

The next activity will introduce you to the concept of Steep. Steep is defined as the area of a body that is within a specified angle, relative to a direction vector. When Steep Angle is toggled to ON, the areas that are greater than the specified Steep Angle are profiled. When Steep Angle is in the OFF state, the entire part is profiled. Steep is normally used to control scallop height and can also aid in avoiding plunging of the tool into material on steep surfaces.

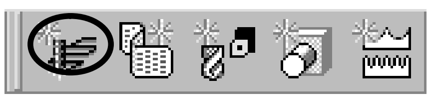
Activity 2–5: ZLEVEL_PROFILE_STEEP Operations

In the following activity, you will create a ZLEVEL_PROFILE_STEEP operation to machine *all* of the steep geometry located within the cavity. You will use the Geometry Parent Group, WORKPIECE that contains all of the Part Geometry. The tool path will cut only within the Steep areas specified.

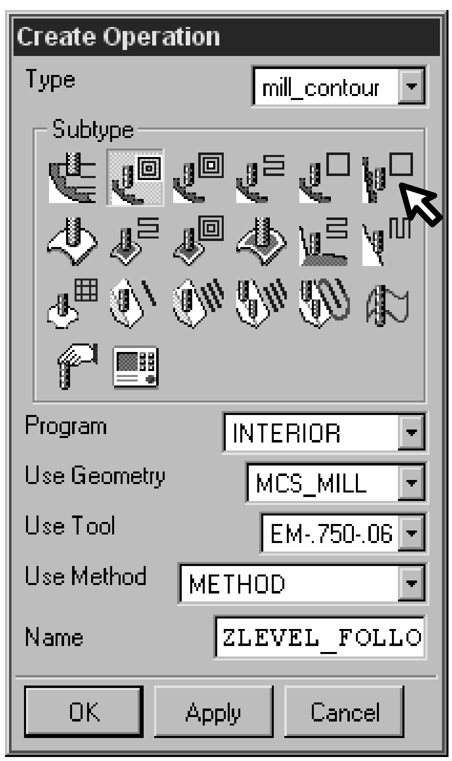
Step 1 Create the ZLEVEL_PROFILE_STEEP Operation.

❑ Continue using *****_form_mold_mfg.prt**.

❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.

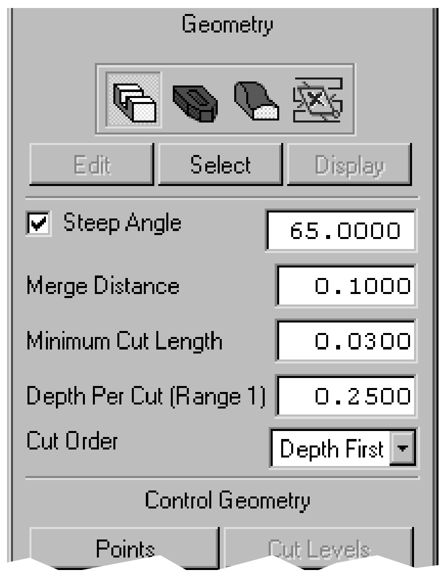


❑ Select the **ZLEVEL_PROFILE_STEEP** icon.

❑ Set the following:

- Program: **INTERIOR**
- Use Geometry: **WORKPIECE**
- Use Tool: **EM–.750-.06**
- Use Method: **MILL_FINISH**

❑ Choose **OK.**



The ZLEVEL_PROFILE_STEEP dialog is displayed.

❑ Under the Geometry label, select the Part icon and choose **Display.**

The Part Geometry is displayed. Note that the Part Geometry was specified in the Parent Group named WORKPIECE.

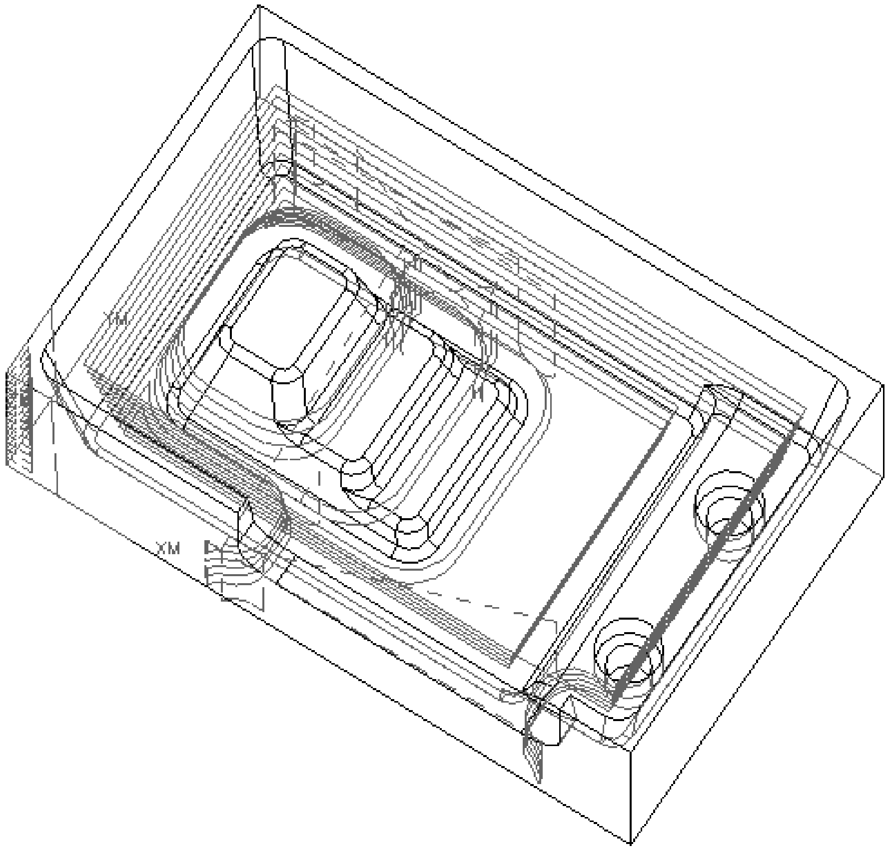
❑ Under the Geometry label, select the Cut Area icon and notice that only the Select button is available**.**

Since the Cut Area was not specified, by default, the entire part will be used for cutting.

Also note the Steep Angle and the other default option settings.

Step 2 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.



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

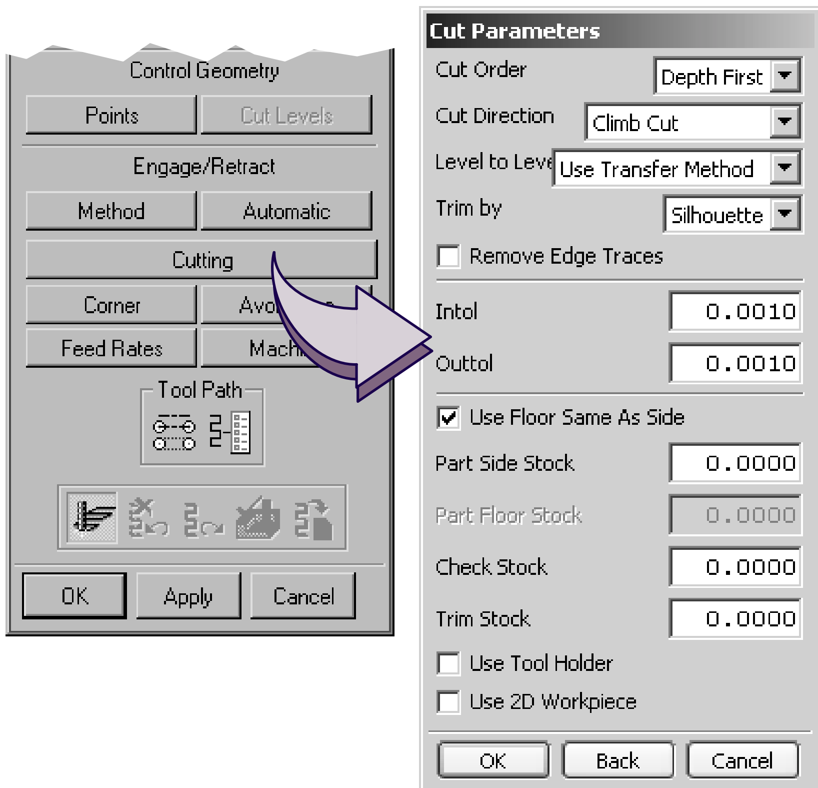
Notice the areas cut by the tool path. Remember that the Steep Angle was set to 65 degrees.

❑ **Save** the part file.

You have completed this activity.

Cavity and Z-Level Milling Stock options

Stock options for Cavity and Z-Level Milling are found on the Cut Parameters dialog. This dialog is activated by selecting the Cutting button found on the Cavity Mill or various Z-Level operation dialogs.



Some of the stock options are as follows:

– *Part Side Stock* adds stock to the individual walls of the part.

– *Part Floor Stock* adds stock to the floor.

– *Check Stock* is the distance that the tool will stay away from the check geometry.

– *Trim Stock* is the distance that the tool will stay away from the trim boundary.

– *Blank Stock* is stock applied to Blank Geometry.

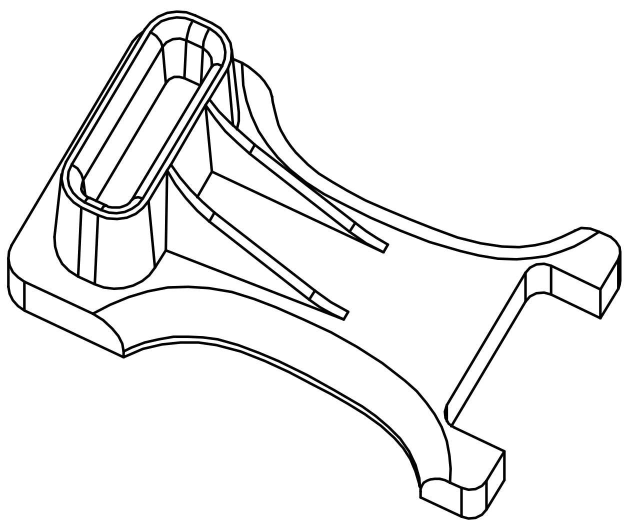
– *Blank Distance* applies to Part Geometry. This is an offset distance which can be used for a casting or forging.

Activity 2–6: Using the Blank Distance Option

In the following activity, you will learn how to set the Blank Distance for a core type part. The MCS, Part Geometry and Program Name have already been created for you.

Step 1 Open a new part file, rename and enter the Manufacturing Application.

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



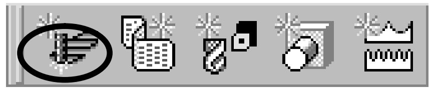
❑ Rename the part *****_horn_mfg.prt** using the **Save→As** option under File on the menu bar where *** represents your initials.

❑ Choose **Application→Manufacturing.**

The Operation Navigator is displayed.

Step 2 Create an operation utilizing Blank Distance as a part offset.

❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.

❑ Select the **Cavity Milling** icon.

❑ Set the following:

- Program: **ROUGH_WITHOUT_CASTING**
- Use Geometry: **WORKPIECE**
- Use Tool: **EM–.375–.06**
- Use Method: **MILL_FINISH**

❑ On the Create Operation dialog, Name the Operation **CM_.25_BLANKDISTANCE**

❑ Choose **OK.**

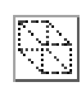
The Cavity Milling dialog is displayed.

Step 3 Verify the Part Geometry selection.

❑ Under the Geometry label, select the **Part** icon. 

❑ Choose **Display.**

Note that the Part Geometry is displayed.

❑ Under the Geometry label, select the **Blank** icon. 

Note that no Blank Geometry has been selected.

Step 4 Specify Operation settings.

❑ Set the Cut Method to **Follow Part**

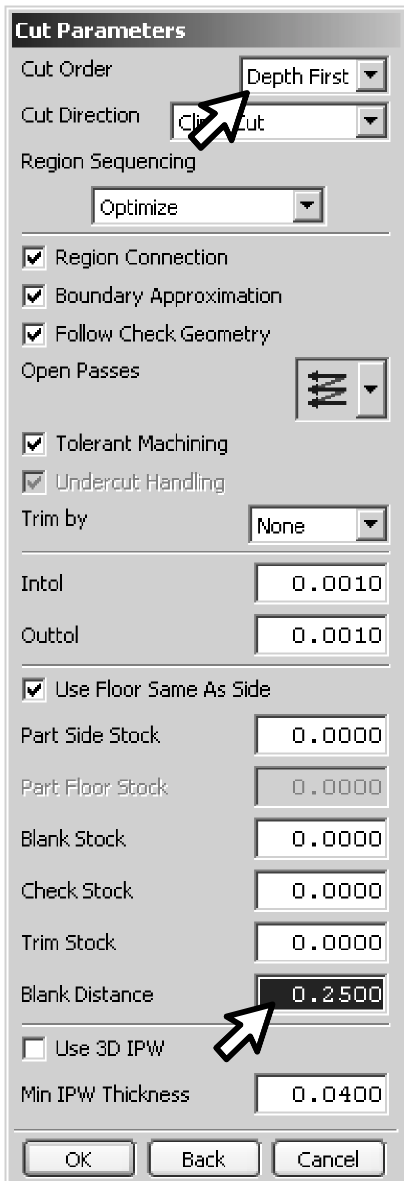
❑ Set Depth Per Cut **.5**

❑ Choose **Cutting.**

The Cut Parameters dialog is displayed.

❑ Change the Cut Order to **Depth First**.

❑ Change the Blank Distance to **.250.**



❑ Choose **OK**.

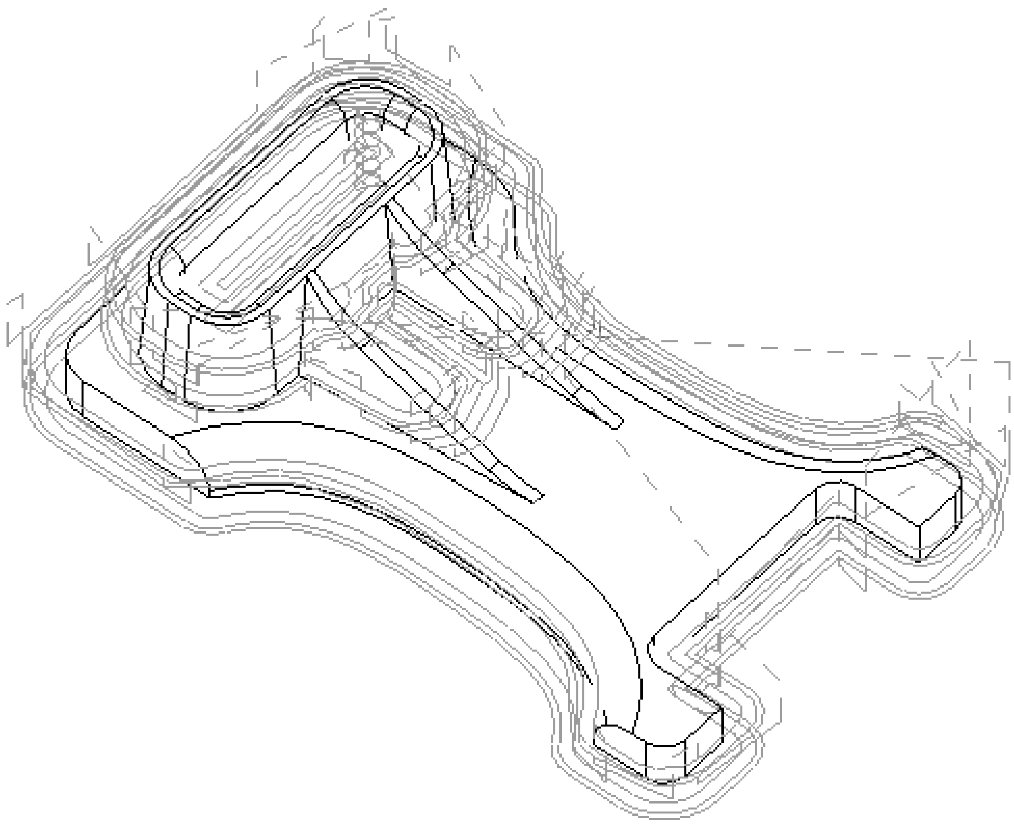
The Cavity Mill dialog is displayed.

Step 5 Generate the tool path.

❑ Choose the **Generate** icon to generate the tool path.

❑ Choose **OK** after viewing each Cut Level.

The tool path cuts all of the core geometry.



Notice that the tool path follows the part contour since you used the Blank Distance option rather than selecting other geometry (such as a solid block) to represent the Blank shape.

In this case, you specified that the Blank was near-net-shape with .250” stock overall.

❑ Choose **OK** to accept the tool path.

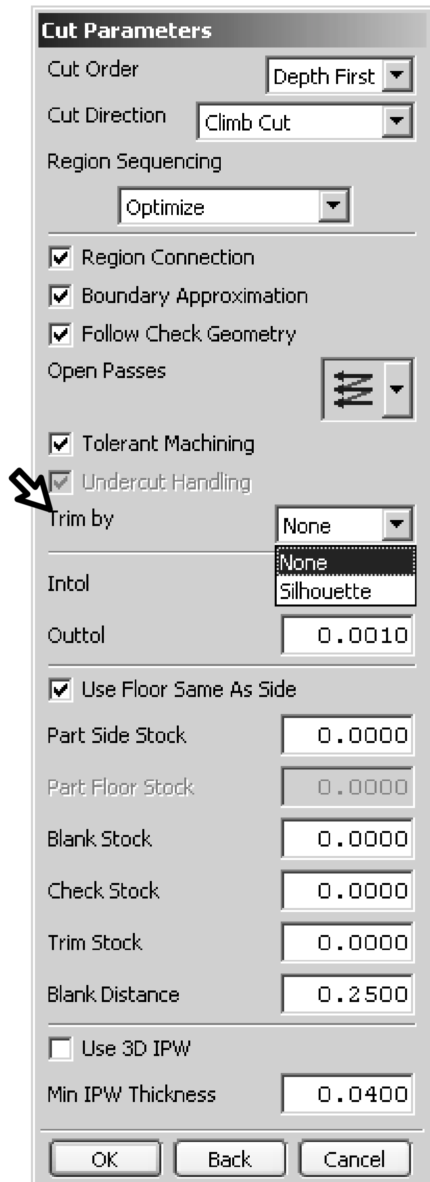
❑ **Save** the part file.

You have completed this activity.

2

Cut Parameters - Trim by

Trim by enables the Blank Geometry to be recognized on core parts when the Blank Geometry has not been explicitly defined. The Trim by method provides a Silhouette option to clean up the material which surrounds the Part geometry. It is available only when Tolerant Machining is toggled to ON.



This option positions the tool to the outer most edge periphery (silhouette) of the part geometry and then offsets it outside by the tool radius. The silhouette can be consider as a shadow of the part projected along the tool axis.

When using Trim by Silhouette, the processor uses the traces at the bottom of the defined part geometry as trim shapes. These shapes are then projected along the tool axis to each cut level and are used to generate machinable regions as trim shapes.

Cut Parameters - Tolerant Machining

The Tolerant Machining ON option is the ***preferred*** method for Cavity Milling operations. Tolerant Machining will find all machinable regions without gouging the part.

Tolerant Machining algorithms digitize a model on a rectangular grid that is determined by the defined cutting tolerance and the tool size. In most parts, the grid size range between 1-2 millimeters (.04 - .08").

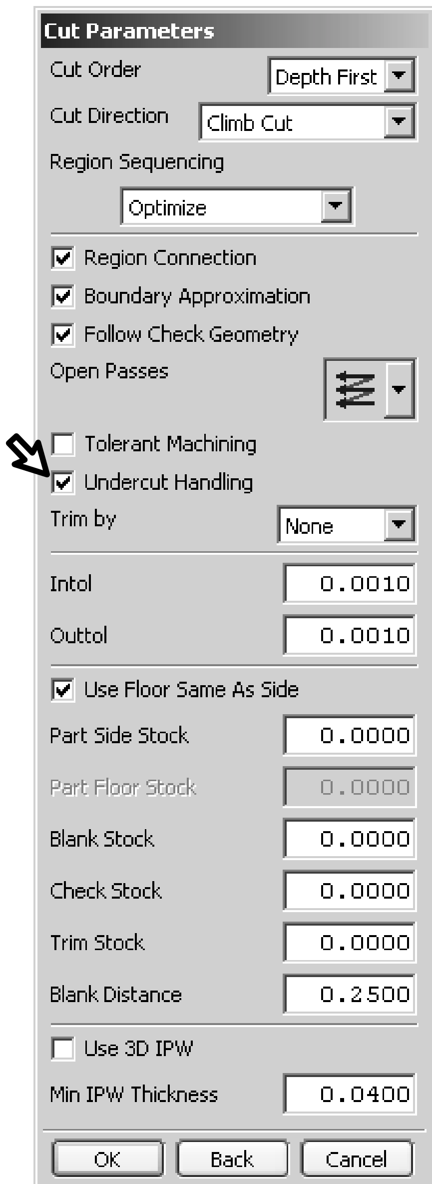
When you specify a Blank distance that is an offset from the Part, the tolerance used to trace the Blank is looser than the tolerance used to trace the Part. This is due to dimensions of Blank geometry not being as accurate as those of the Part Geometry. When you specify Blank geometry that is close to the size of Part Geometry, the Blank and Part traces will overlap and result in an undesirable cut region(s). In this case it would be better to cut a profile pass along the Part without specifying the Blank. The resultant tool path will be along the Part Geometry.

When the processor encounters geometry that contains gaps or that is not perfectly matched, it will move the tool using an approximation within the specified tolerances.

The processing time is longer when Tolerant Machining is ON. Tolerant Machining ***should always*** be turned on.

Cut Parameters - Undercut Handling

Undercut Handling is used with geometry features containing undercuts. It is applied only to non-tolerant machining.

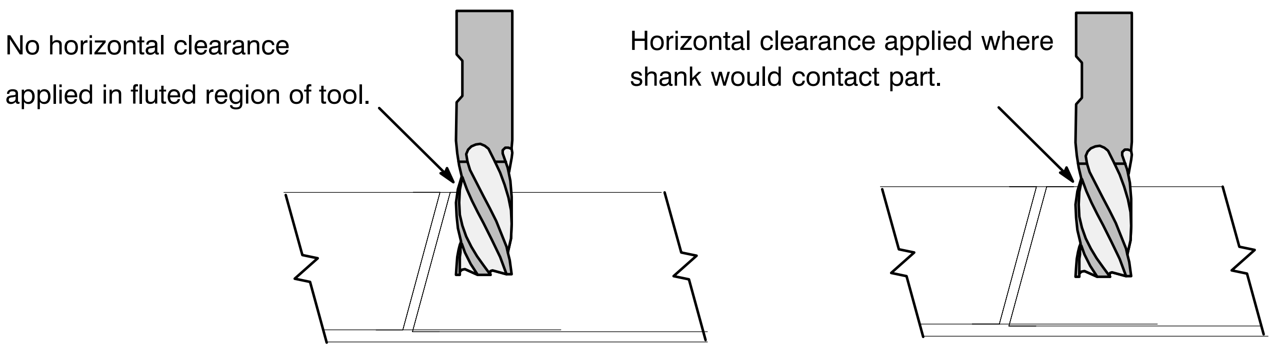


If Tolerant Machining is turned ON, undercut handling is automatically turned off.

When using the Undercut Handling option Horizontal Clearance (specified under the Engage/Retract Method) applies to the shank of the tool (the portion above the flutes) unless the Horizontal Clearance is greater than the tool radius. In this case the tool radius is used.

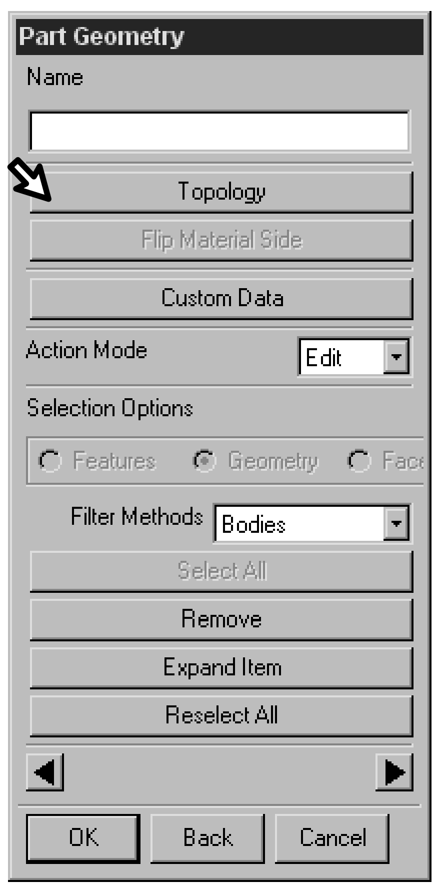
As the tool progresses deeper through the various cut levels, Horizontal Clearance will keep the shank from contacting the part geometry which forms the undercut.

In the following example the Horizontal Clearance uses the default of .100. The tool radius is .120. The tool will be offset from the undercut face by .100.



Part, Blank, Check Geometry - Topology

Topology provides options for surface analysis that allow checking for material side inconsistency, gaps and missing and duplicate surfaces.



This option is available when you are *editing* geometry and aids in the correction of model geometry errors that occur when models from other CAD systems are converted into Unigraphics models or from within a model created using Unigraphics.

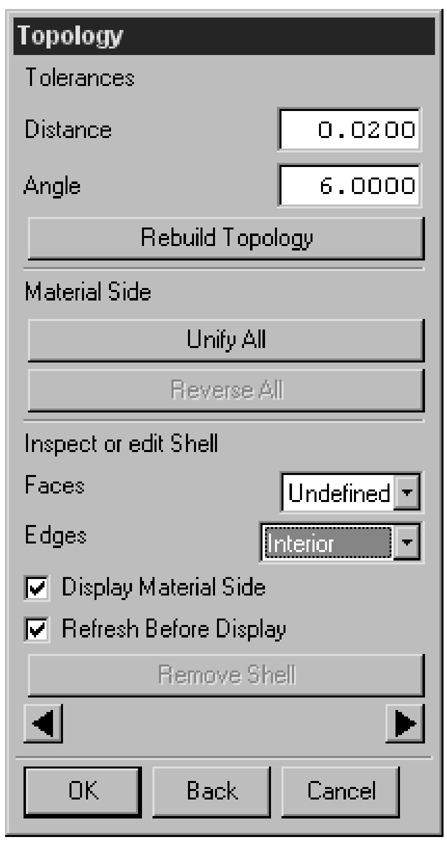
The topology processor inspects the model for missing, duplicated and non-tangent faces which can create multiple shells and an erratic tool path.

NOTE It is suggested that the Topology option be used only if tool path generation fails.

The following are common causes of tool path generation failures.

- Duplicate faces
- Missing fillets and faces
- Smaller than tolerance faces (usually fillets)

The following is a summary of the options on the Topology dialog:



Tolerances - Distance is the tolerance used for connecting faces and curves. The distance value represents the maximum value that two objects can be apart and still be considered connected. Angle is the tolerance used for determining the type of each edge (convex, concave or tangent). The Angle value represents the maximum angle that the normals of two adjacent faces or curves can vary at an edge to determine if the edge is convex, concave or tangent.

Rebuild Topology - After editing tolerances or material side, you can choose Rebuild Topology to create the shell. Surfaces are considered adjacent if the gaps are less than the tolerance specified and one or more shells are created. Model geometry is not modified.

Material Side - Material Side allows you to change the material side of any object that is used to define the cutting operation. Material side is represented by a vector arrow that points away from the material.

Unify All - allows material side to be located on the same side for all objects.

Reverse All - allows material side to be reversed for all objects.

Inspect or Edit Shell - allows the inspection of the classification of edge types and material side for individual objects.

Faces - allows the inspection of material side defined for each face. Faces can be set to undefined, same or opposite.

- **Undefined** allows you to highlight all faces where the material side is not defined
- **Same** allows you to highlight all of the faces where the material side is the same as that of the majority of faces
- **Opposite** allows you to highlight all of the faces where the material side is different than that of the majority of faces

Edges - allows the review of the classification of various edges. Edges can be set to the following:

- **Undefined** allows the highlight of any edge which is not classified by the system
- **Nonmanifold** allows the highlight of any unresolved edge where more than two faces meet along the same portion of the edge

- **Exterior** allows the highlight of all of the outside edges that define the cutting region
- **Interior** allows the highlight of all of the inside edges that define the cutting region
- **Inconsistent** allows the highlight of edges where the adjacent faces have material sides on opposite sides
- **Complex** allows the highlight of edges that are neither completely tangent, concave or convex
- **Tangent** allows the highlight of all edges that are classified as being tangent
- **Concave** allows the highlight of all edges that are classified as being concave
- **Convex** allows the highlight of all edges that are classified as being convex

Display Material Side - this option results in the display of the material side indicator whenever one of the face options is chosen.

The material side indicator is a vector that points towards the material to be removed which is away from the material side.

Refresh Before Display - the system will refresh the screen every time you choose one of the Face or Edge options.

Arrow Buttons - allows you to cycle through the different shells as you inspect and edit the topology.

SUMMARY

Cavity Milling operations are efficient and robust methods of removing volumes of material prior to finishing operations. Numerous options allows flexibility in the stock removal process.

In this lesson you:

- Created Geometry Parent Groups that were used in Cavity Milling operations
- Review Cavity Milling operations
- Used the more advanced concepts of Cavity Milling
- Used the Cut Area option for limiting the cutting area in an operation
- Applied the use of Blank Offset Distance and Blank Geometry to generate Cavity Milling operations

2

EDS

(This Page Intentionally Left Blank)

Fixed Contour

Lesson 3

PURPOSE

This lesson will show you how to create Fixed Contour operations using advanced concepts and techniques. You will also review the steps necessary to create various Parent Groups that will aid you in the selection of geometry and cutting tools. Fixed Contour operations are generally used for creation of tool paths used to finish the contoured areas of a part.

OBJECTIVES

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

- Apply the more advanced concepts of Fixed Contour operations for creating tool paths
- Create Parent Groups used for Fixed Contouring operations
- Choose the most appropriate Drive method for a Fixed Contour operation

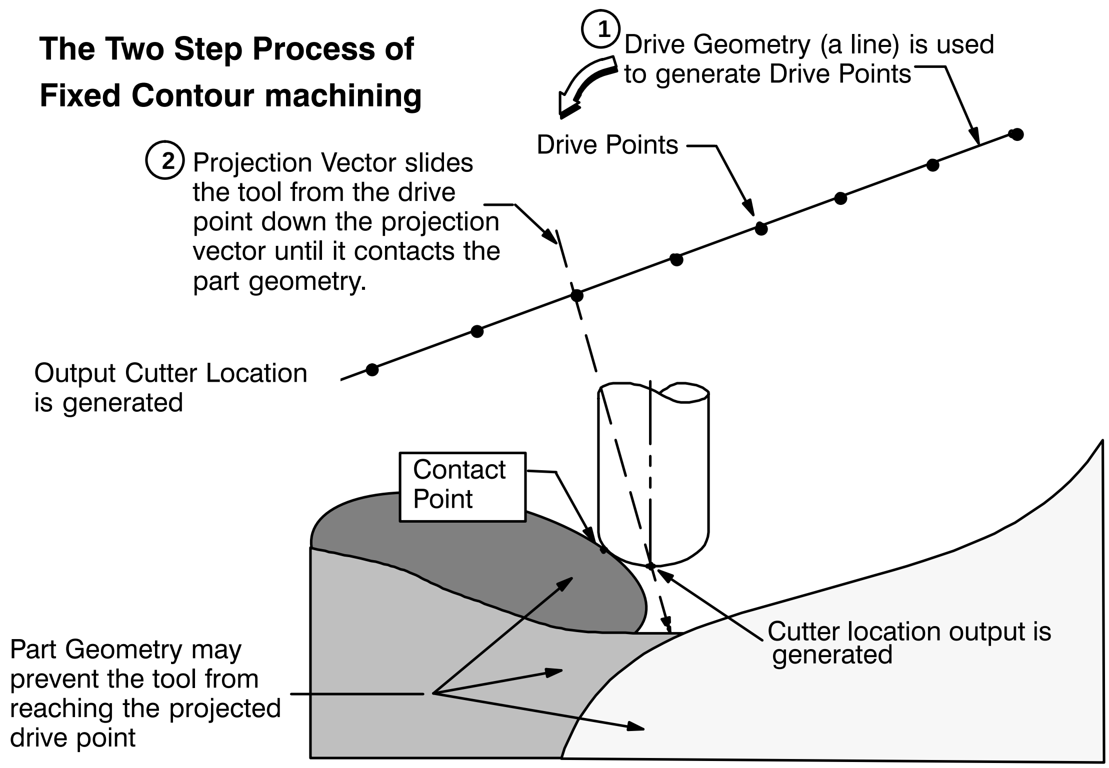
This lesson contains the following activities:

Fixed Contour Overview

Fixed Contour operations are used to finish areas formed by contoured geometry. Fixed Contour tool paths are able to follow complex contours by the control of tool axis, projection vector and drive methods. Tool paths are created in two steps. The first step generates *drive points* from the *drive geometry.* The second step projects the *drive points* along a projection vector to the part geometry.

The Drive Points are created from some or all of the part geometry, or can be created from other geometry that is not associated with the part. The points are then projected to the Part geometry.

The tool path output is created by internal processing which moves the tool from the Drive Point along the projection vector until contact is made with the Part Geometry. The position may coincide with the projected drive point or, if other part geometry prevents the tool from reaching the projected drive point, a new output point is generated and the unusable drive point is ignored.

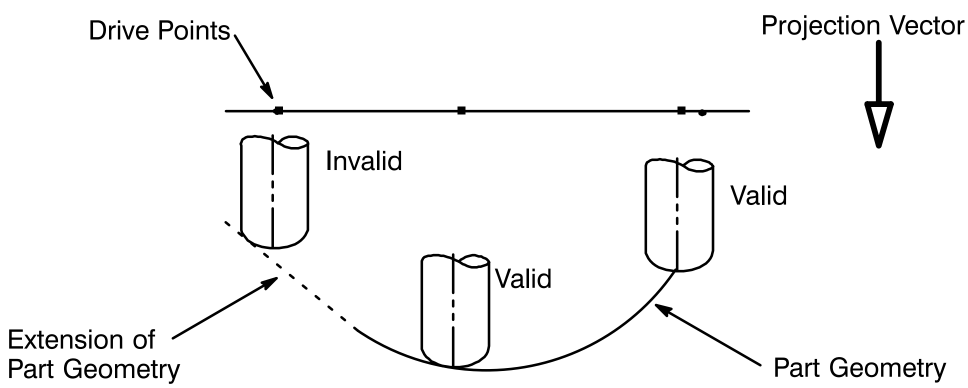


Fixed Contour Tool Path Accuracy

Fixed Contour provides several options that help insure the accuracy of the tool path. Included are:

- Check Geometry to stop tool movement
- Gouge Checking to prevent gouging of the part
- Collision Checking to prevent unintended tool contact with other geometry
- Various tolerance options

Fixed Contour operations can position to existing locations on the part geometry (which includes the edge of an object), but the tool cannot position to an extension of part geometry as shown in the following illustration.



Terminology used in Fixed Contour operations

Part Geometry – is geometry selected to cut.

Check Geometry – is geometry selected that is used to stop tool movement.

Drive Geometry – is geometry used to generate drive points.

Drive Points – are generated from the Drive Geometry and projected onto the part geometry.

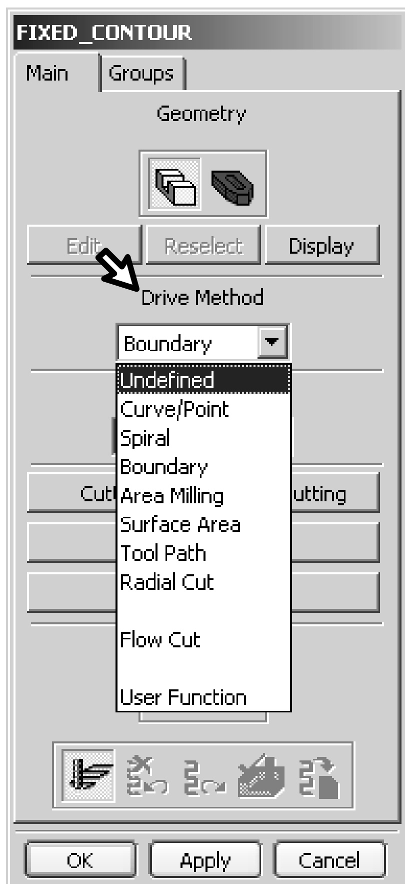
Drive Method – method of defining Drive Points required to create a tool path. Some Drive Methods allow the creation of a string of Drive Points along a curve while others allow the creation of an array of Drive Points within an area.

Projection Vector – used to describe how the Drive Points project to the Part Surface and which side of the Part Surface the tool contacts. The selected Drive Method determines which Projection Vectors are available.

NOTE The projection vector does not need to coincide with the tool axis vector.

Drive Methods for Fixed Contouring

The Drive Method defines the method of creating Drive Points.



Each drive method contains a series of dialogs that are displayed upon selection.

Curve/Point Drive Method

The Curve/Point Drive Method allows you to define Drive geometry by specifying points and curves. When specifying points, the Drive Path is created as linear segments between the points. When specifying curves, Drive Points are generated along the selected curves. In each case, the Drive geometry is projected on to the Part Surface(s) where the tool path is created. The curves may be open or closed, contiguous or noncontiguous, planar or non-planar.

When points define the Drive geometry, the cutter moves along the tool path from one point to the next in the order in which they were specified. The same point may be used more than once, provided it is not defined consecutively in the sequence. A closed Drive Path can be created by defining the same point as the first and last point in the sequence.

Spiral Drive Method

The Spiral Drive Method allows you to define Drive Points that spiral outward from a specified center point. The drive points are created within the plane normal to the projection vector and contain the center point. The Drive Points are projected on to the part surfaces along the projection vector.

Unlike other Drive Methods which require a sudden change in direction to stepover to the next cutting pass, Spiral Drive Method Stepovers are a smooth, constant transition outward. Since this drive method maintains a constant cutting motion, it is useful in high speed machining applications.

Boundary Drive Method

The Boundary Drive Method allows you to define cut regions by specifying Boundaries and Loops. Boundaries are not dependent on the shape and size of the Part Surfaces while Loops must correspond to exterior Part Surface edges. Cut regions are defined by Boundaries, Loops, or a combination of both. The tool path is created by projecting Drive Points from the defined cut region to the Part Surface(s) in the direction of a specified Projection Vector. The Boundary Drive Method is useful in machining Part Surfaces requiring minimal Tool Axis and Projection Vector control.

Area Milling Drive Method

The Area Milling Drive Method allows you to specify a cut area for tool path generation. This drive method is similar to the Boundary Drive Method, but does not require drive geometry.

Cut Area(s) may be defined by selecting Surface Regions, Sheet Bodies, or Faces. Unlike the Surface Area Drive Method, the cut area geometry does not have to be selected in an orderly grid of rows and columns.

If you do not specify a Cut Area, the processor will use the selected Part Geometry (excluding areas not accessible by the tool) as the cut area.

The Area Milling Drive method is generally the preferred Fixed Contour Drive Method for creating tool paths.

Surface Drive Method

The Surface Area Drive Method allows you to create an array of Drive Points that lie on a grid of Drive Surfaces. This Drive Method is useful in machining very complex surfaces. It provides additional control of both the Tool Axis and the Projection vector.

The tool path is created on the selected Part Surfaces by projecting points from the Drive Surfaces in the direction of a specified Projection Vector. If Part Surfaces are not defined, the tool path can be created directly on the Drive Surfaces. The Drive Surfaces do not have to be planar, but must be in an orderly grid of rows and columns. Adjacent surfaces must share a common edge and may not contain gaps that exceed the Chaining Tolerance defined under Preferences (**Preferences→ Selection→ Chaining Tolerance**). Trimmed surfaces can be used to define Drive Surfaces as long as the trimmed surface has four sides. Each side of the trimmed surface can be a single edge curve or comprised of multiple tangent edge curves that can be considered a single curve.

Tool Path Drive Method

The Tool Path Drive Method allows you to define Drive Points along the tool path of a Cutter Location Source File (CLSF) to create a similar Surface Contouring tool path. Drive Points are generated along the existing tool path and then projected on to the selected Part Surface(s) to create the new tool path that follows the surface contours. The direction in which the Drive Points are projected on to the Part Surfaces is determined by the Projection Vector.

Radial Cut Drive Method

The Radial Cut Drive Method allows you to generate Drive Paths perpendicular to and along a given boundary, using a specified Stepover distance, Bandwidth and Cut Type. This method is useful in creating cleanup operations.

Flow Cut Drive Method

Flow Cut Drive Method allows you to generate Drive Points along concave corners and valleys formed by Part Surfaces. The direction and order of the flow cuts are determined using rules based on machining best practices. The tool path is optimized for maximum part contact to minimize non-cutting moves.

User Function Drive Method

The User Function Drive method creates tool paths from special drive methods developed in User Function code. These are optional, highly specialized custom routines developed for specific complex applications.

Parent Groups associated with Fixed Contour operations

There are three different Geometry Parent Groups available for use in Fixed Contour operations. They are:

- The MILL_GEOM Parent Group which allows Part, Blank and Check Geometry.

- The MILL_BND Parent Group which also allows Part, Blank and Check as well as Trim and Floor boundary geometry.

- The MILL_AREA Parent Group allows Part and Check but not Blank Geometry. It also allows for the specification of Cut Areas and Trim geometry.

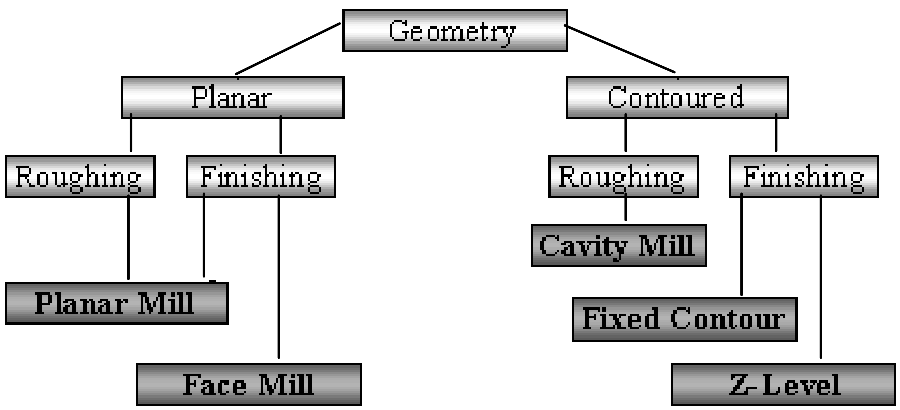
The Parent Group, MILL_AREA, which you used in Cavity Milling operations, is also used in Fixed Contour operations. It allows you to include or exclude areas to be machined in cut areas that you specify. These specific areas may have been previously roughed by Cavity Milling or already finished by Planar Mill operations.

Fixed Contour also provides several template operations that use the Parent Group, MILL_AREA. These operations also have the Area Milling Drive Method specified allowing you to quickly create finishing operations for contoured parts.

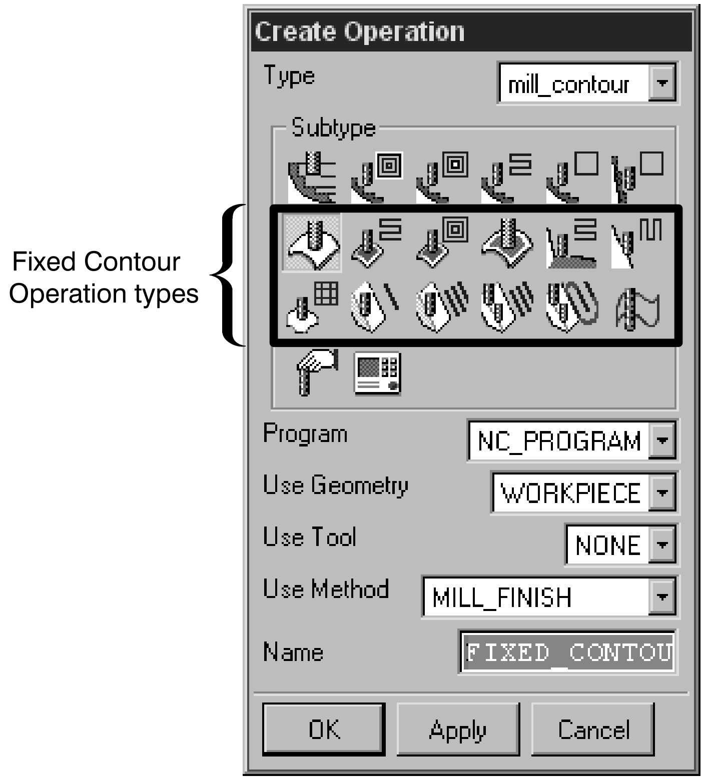
Fixed Contour operations are generally used to finish contoured types of geometry.

The following diagram can be used as an aid in the determination of the operation type needed for various types of geometry.

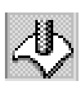
Matching the Geometry to an Operation

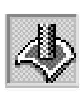


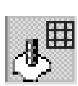
Fixed Contour Operation types

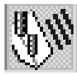


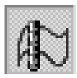
The most commonly used Fixed Contour operation types are:

- **FIXED_CONTOUR –** Generic Fixed Contour operation type. Allows for selection of various drive methods and cut types. Use when other Fixed Contour operation types are not applicable.

- **CONTOUR_AREA –** Uses Area Milling Drive Method. Ideal for cutting specific areas of part geometry.

- **CONTOUR_SURFACE_AREA–** Uses Surface Area Drive Method. Ideal for complex part surfaces where tool axis control is critical.

- **FLOWCUT_REF_TOOL –** Uses the Flow cut Drive Method. Flow Cut RTO (reference tool) will machine certain geometry types by level and provide you with the options to cut the two sides alternatively with a rounded or standard turn at each end, and side by side with the option from the steep side to non-steep side. This operation type takes into account the previous tool diameter used for roughing (you must specify this). This results in cutting parts with a more constant cutting load and a shorter distance of non-cutting moves.

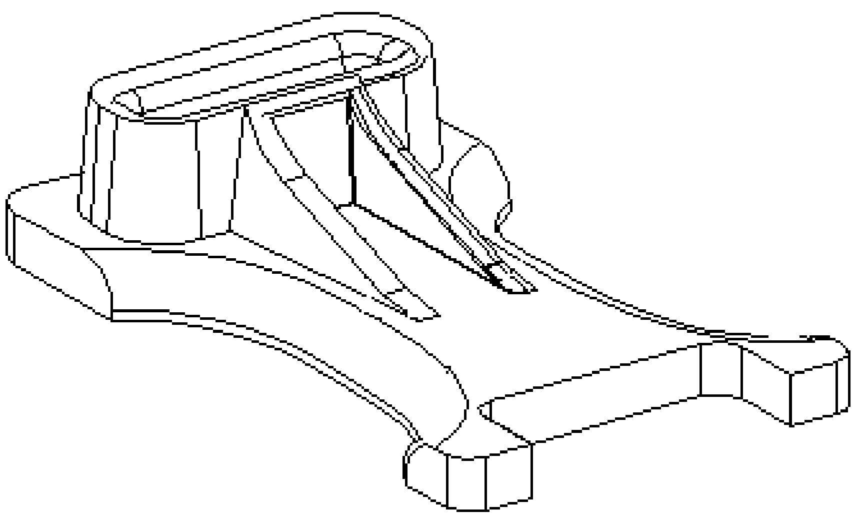
- **PROFILE_3D –** Generates a profile pass with a constant Z-depth utilizing three dimensional curves, edges, faces, existing boundaries or points. Machines at a given Z-depth offset with respect to the geometry type selected. Useful in creation of addendum profile cuts for stamping dies.

Activity 3—1: Contour_Area_Non-Steep Operations

In this activity, you will finish machining the non-steep areas of the part which were previously machined, in a prior activity, with Cavity Milling operations. You will use the Geometry Parent Group, WORKPIECE, and Fixed Contour Non-Steep operation type to remove the material left in areas not machined previously. The Geometry Parent Group WORKPIECE, which is already defined, uses the entire piece part as the Part Geometry.

Step 1 Open the part file.

❑ Open the part file *****_horn_mfg .prt** where *** represents your initials.

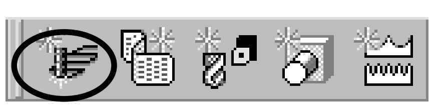


❑ If necessary, enter the Manufacturing application.

The Operation Navigator is displayed.

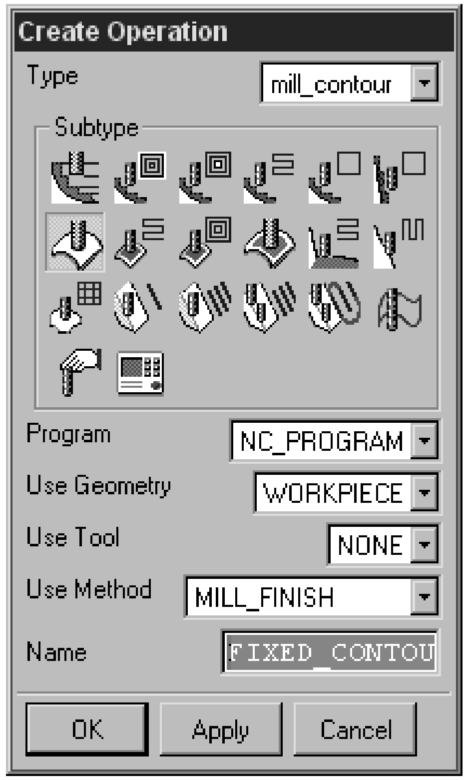
Step 2 Create the Operation.

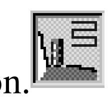
❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.

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



❑ Select the **CONTOUR_AREA_NON_STEEP** icon. 

❑ Set the following:

- Program: **ROUGH_WITHOUT_CASTING**
- Use Geometry: **WORKPIECE**
- Use Tool: **EM-.375-.06**
- Use Method: **MILL_FINISH**

❑ Choose **OK.**

The CONTOUR_AREA_NON_STEEP dialog is displayed.

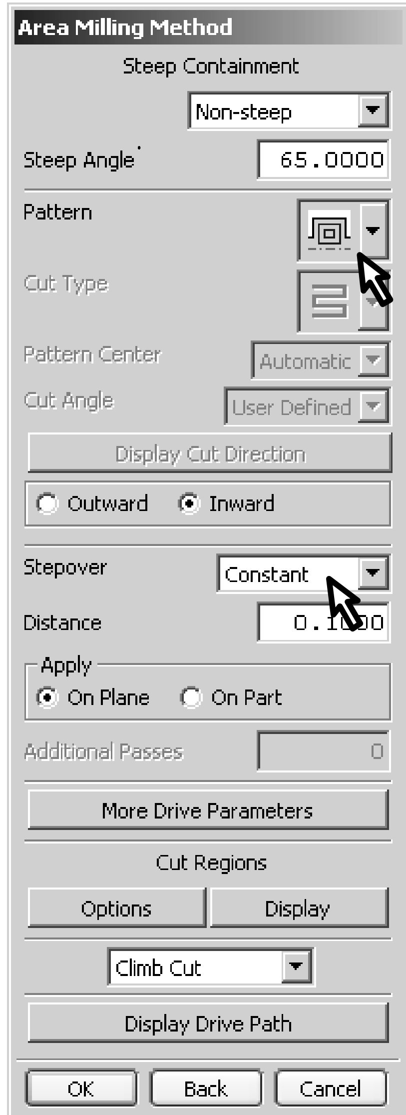
❑ Under the Geometry label, choose the **Part** icon  **.**

❑ Choose **Display**.

Note that the entire body is selected as Part Geometry.

❑ Under the Drive Method pull down dialog select **Area Milling**.

The Area Milling Method dialog is displayed. Note that Steep Containment is set to Non-Steep and the Steep Angle is **65**°.



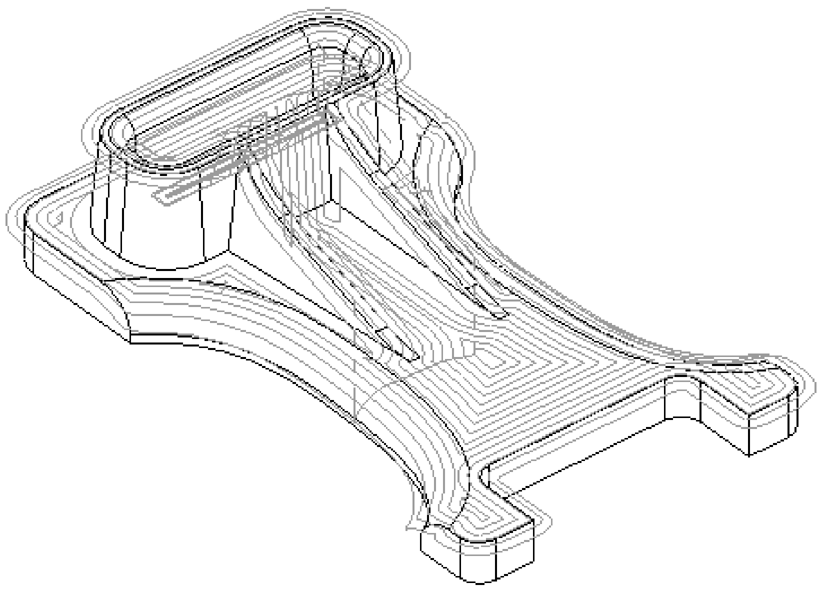
3

❑ Change the Pattern to **Follow Periphery** and the Stepover to **Constant.**

❑ Choose **OK** to return to the CONTOUR_AREA_NON_STEEP dialog.

Step 3 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path



Note that areas over 65 degrees were not machined. In the Cavity Mill ZLEVEL_PROFILE_STEEP operation, the Steep Angle was set to 65 and only areas over 65 degrees were machined.

❑ **Save** the part file.

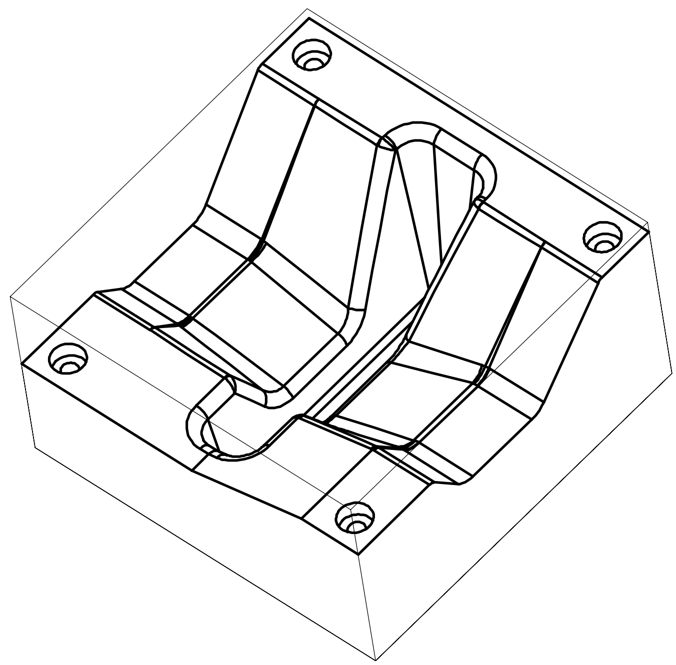
You are finished with this activity. In the next activity, you will use the Area_Mill Geometry Parent Group to select a specific area of the body as Part Geometry.

Activity 3–2: Creating and Using a Mill_Area Parent

In this activity, you will use the geometry Parent Group, MILL_AREA to isolate a non-steep portion of a mold cavity for tool path generation.

Step 1 Change to a new part file.

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



❑ **Save As** ***_ **deep_mold_mfg .prt** where *** represents your initials.

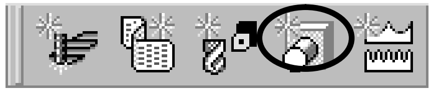
Note that Blank Geometry surrounds the part.

❑ Enter the **Manufacturing** application.

The Operation Navigator is displayed.

Step 2 Create the Geometry Parent Group, CUT_AREA_PARENT.

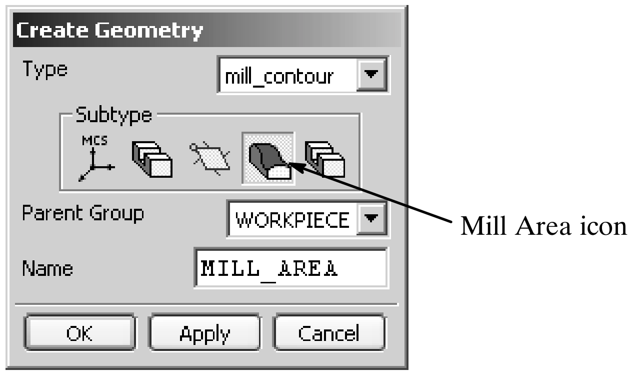
❑ As shown below, select the **Create Geometry** icon from the Create toolbar.



The Create Geometry dialog is displayed.

❑ Make sure the Type is **mill_contour** .

❑ Choose the **MILL_AREA** icon.



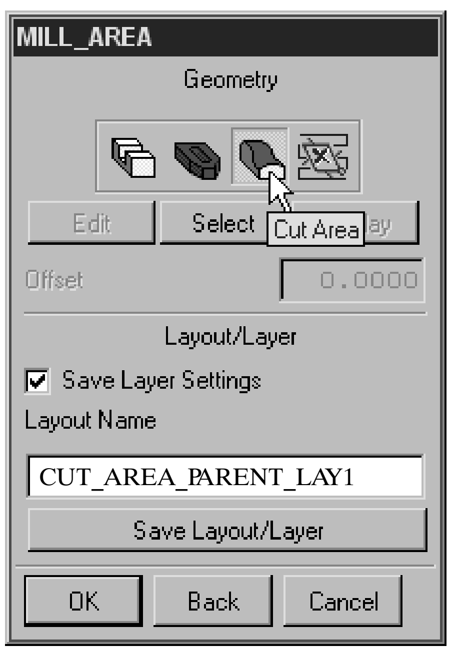
❑ Choose **WORKPIECE** as the Parent Group.

❑ Key in **CUT_AREA_PARENT** as the name.

3

❑ Choose **OK**.

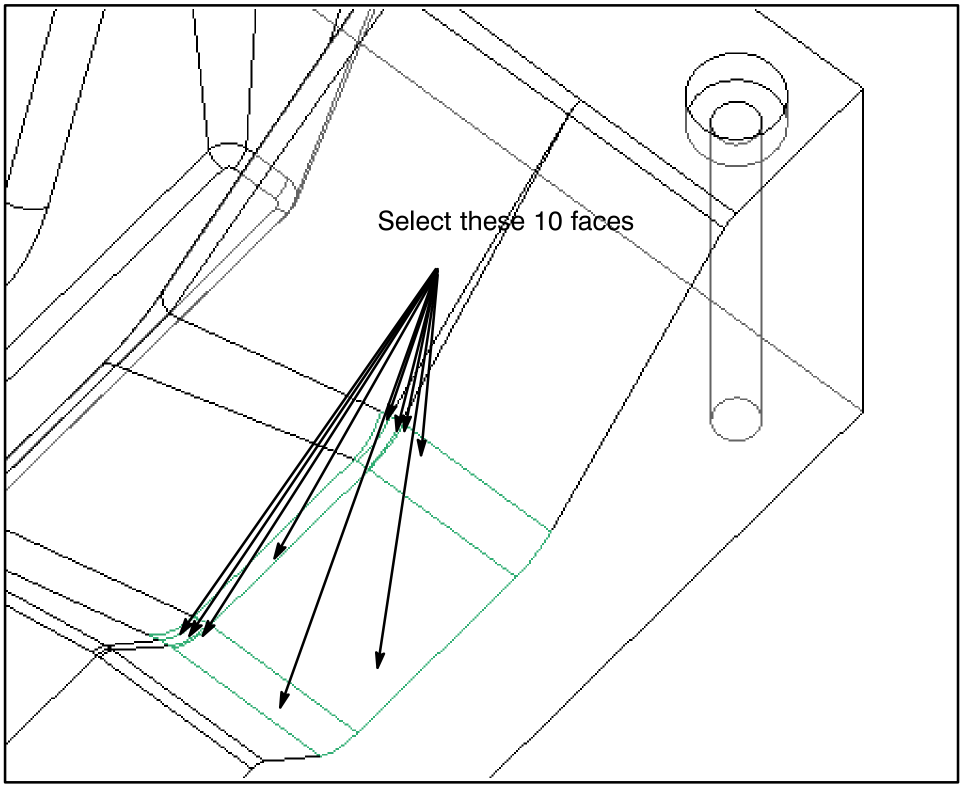
The MILL_AREA dialog is displayed.



❑ Select the **CUT_AREA** icon.

❑ Choose **Select**.

You will choose the 10 faces shown on the following illustration.



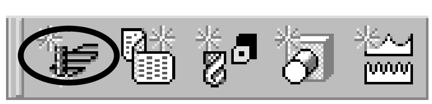
❑ In the Cut Area dialog, if necessary, change Filter Methods to **Faces**.

❑ Select the faces as shown in the previous illustration. Note the Cue line count of the faces after each selection.

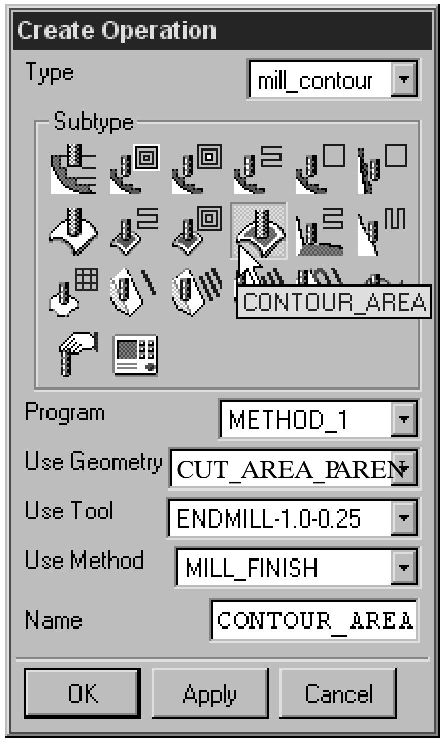
❑ Choose **OK** twice.

Step 3 Create a contour area with a non-steep containment operation.

❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.



❑ Choose the **CONTOUR_AREA** icon.

❑ If necessary, set the following:

- Program: **METHOD_1**
- Use Geometry: **CUT_AREA_PARENT**
- Use Tool: **ENDMILL-1.0-0.25**
- Use Method: **MILL_FINISH**

❑ Key in **CONTOUR_AREA_MANUAL** for the Name.

❑ Choose **OK**.

The CONTOUR_AREA dialog is displayed.

Step 4 Set the Drive Method Options.

❑ Choose **AREA_MILLING** twice.

❑ Set the Pattern to **Follow Periphery.**

❑ Choose **Outward** as the pocket direction.

❑ Choose **Tool Diameter** from the Stepover pull down menu.

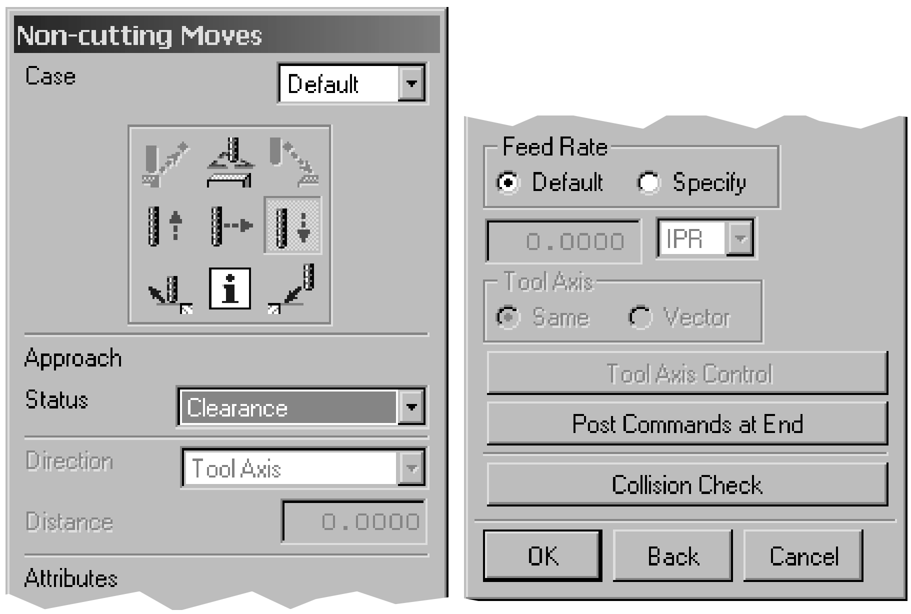
❑ Accept the default of 50 for **Percent** of tool used.

❑ Choose **OK** to return to the CONTOUR_AREA dialog.

Step 5 Create a clearance plane used for clearing the part.

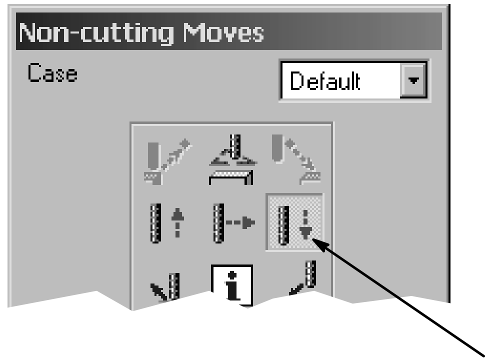
❑ Choose **Non-Cutting.**

The Non-cutting Moves dialog is displayed.

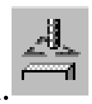


You will now create a Clearance Plane used when approaching the part.

❑ Select the **Approach** icon.



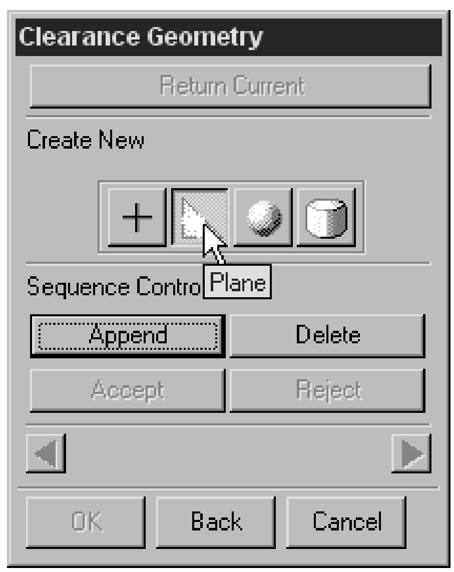
❑ Change the Approach Status to **Clearance**.

❑ Choose the **Clearance** icon. 

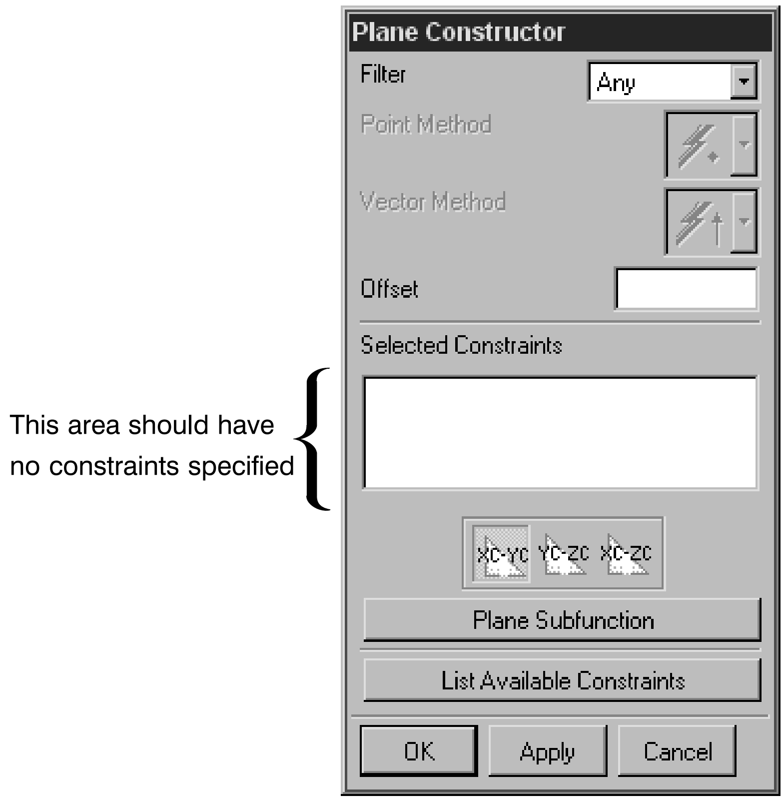
The Clearance Geometry dialog is displayed.

❑ Choose **Append** from the Clearance Geometry dialog.

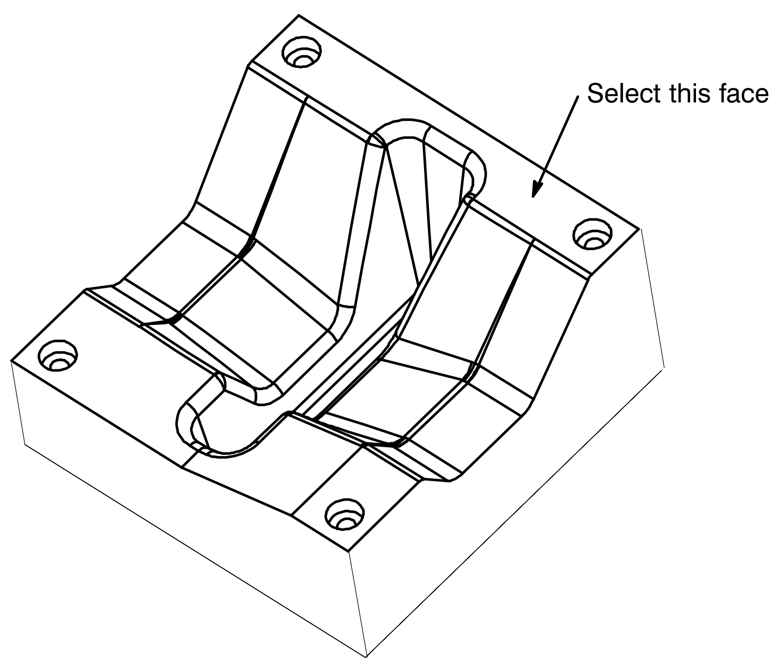
❑ Choose the **Plane** icon.



The Plane Constructor dialog is displayed. **Note:** make sure that the Selected Constraints area of the dialog is empty.



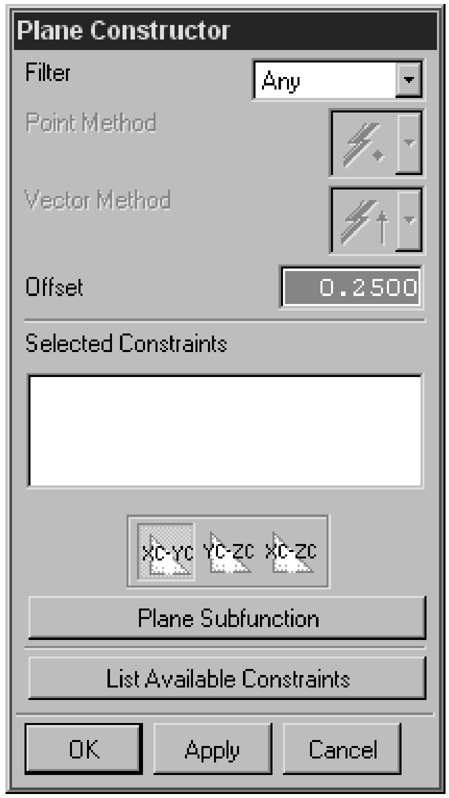
❑ As shown below, select the upper face.



Note that the Selected Constraint is **Offset to Face**.

3

- ❑ Enter **.250** as the Offset.



- ❑ Choose **OK.**

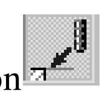
  Note that the plane symbol is displayed.

- ❑ On the Clearance Geometry dialog, choose **Accept**.

- ❑ On the Clearance Geometry dialog, choose **Return Current**.

  You have established a Clearance Plane .250 above the upper face.

  Next you will specify a helical engage motion.

- ❑ The **Engage** icon  should be selected.

- ❑ Change the Engage Status to **Manual**.

- ❑ Change the Movement to **Helical: Climb.**

- ❑ Select the **Distance** radio button.
- ❑ Set the Max Ramp Angle to **7.000**.
- ❑ Choose **OK** and return to the CONTOUR_AREA dialog.

Step 6 Set the Cutting Options.

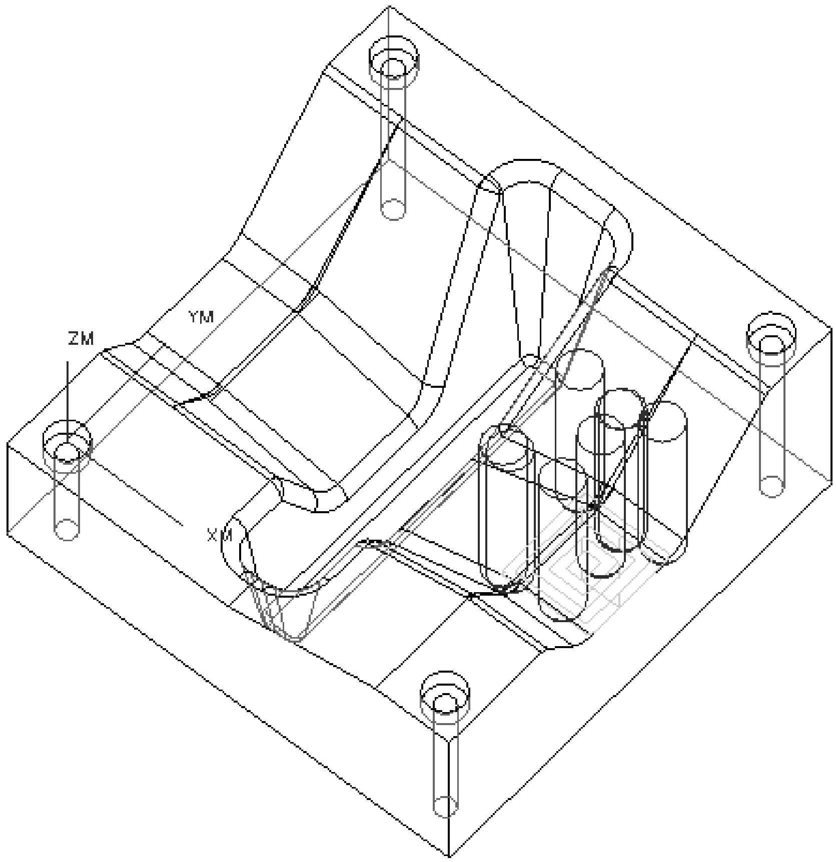
- ❑ Choose **Cutting**.
- ❑ Set Remove Edge Traces to **ON** (√)**.**

  By setting this option any portion of the tool path that lies beyond the edge of the part geometry will be removed.

- ❑ Choose **OK**.

Step 7 Generate the tool path.

- ❑ Choose the **Generate** icon and generate the tool path**.**



Note that the tool path is restricted to the faces that you selected for the Mill_Area Geometry Parent Group. In this operation, you did not need to know or specify the steep angles.

❑ **Save and Close** the part file.

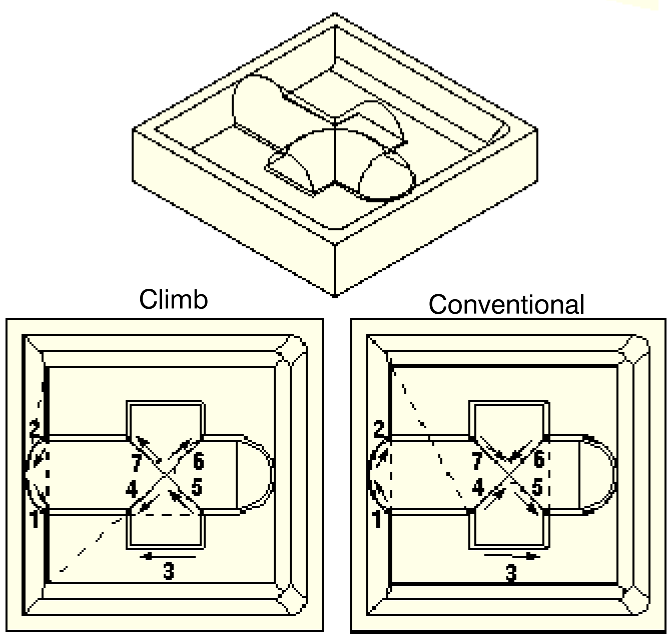
This concludes the activity.

Flow Cut Drive Methods

The Flow Cut Drive Method allows the specification of Climb, Conventional, or Mixed cut directions for single pass operations.

The Climb and Conventional options allow the climb or conventional method for all cutting passes in the operation. If a steep side can be determined, the steep side is used to calculate the Climb or Conventional cut direction. If a steep side cannot be determined, the cut direction is determined internally.

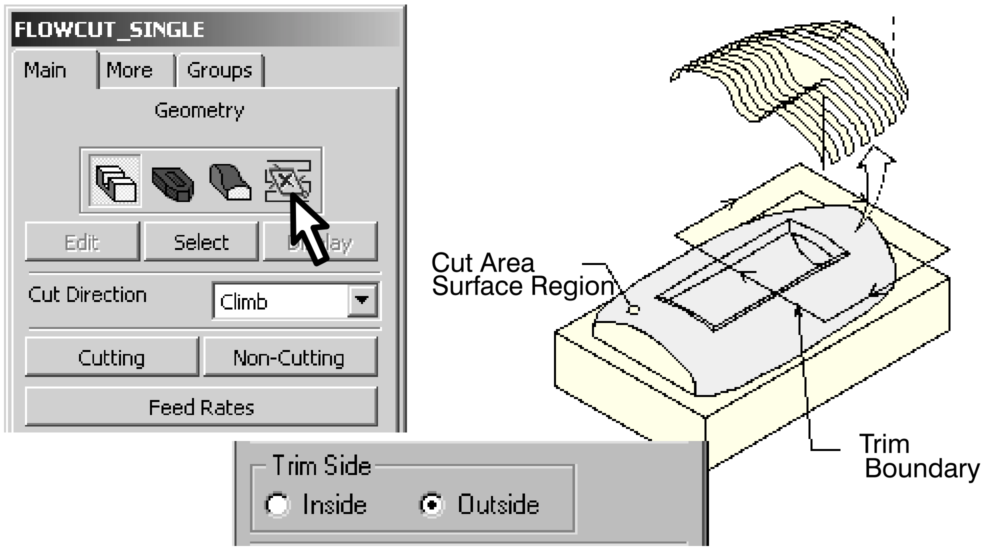
The Mixed option allows for the internal calculation of the cut direction.



Flow Cut Drive Method using Cut Area and Trim Boundary Geometry

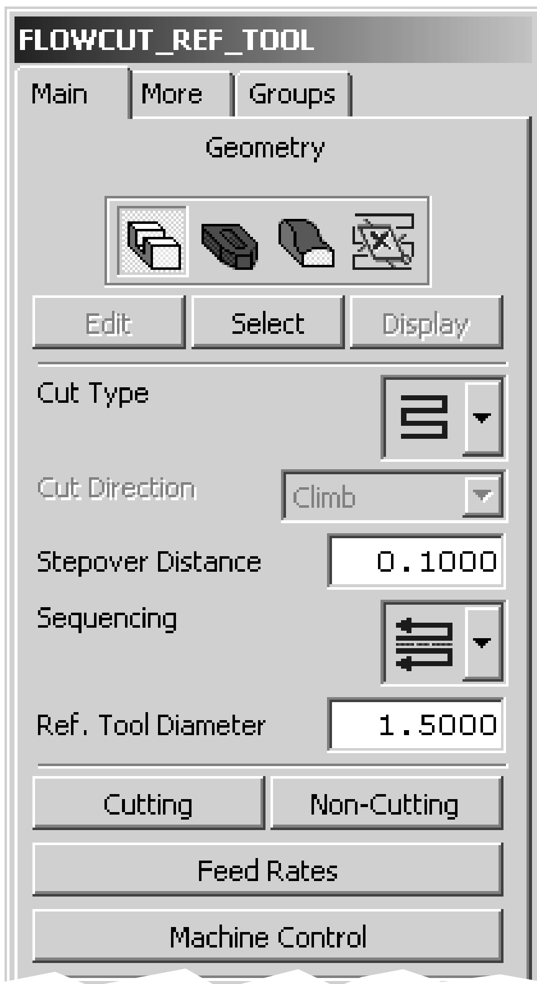
The Flow Cut Drive method allows Cut Area geometry to be defined the same way as the Area Milling Drive method. Surface Regions, Sheet Bodies, Faceted Bodies and or Faces can be used as the cut area. Concave valleys are analyzed within the cut area as well as concave valleys formed by the cut area and part geometry. Valleys formed by the cut area and check geometry are excluded.

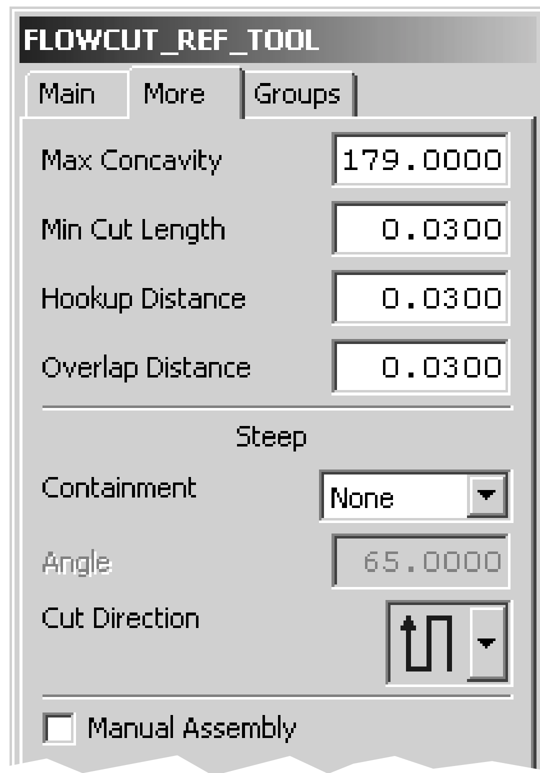
Trim boundaries can be used to further constrain cut regions. Material Inside or Outside determines the area of the cut region to be omitted. Trim boundaries are always Closed, always use an ON condition, and are projected to the Part geometry along the tool axis vector. More than one Trim Boundary may be defined. Trim Stock may be specified to define the distance the tool is positioned from the Trim Boundary.



Flow Cut Reference Tool Drive Method

Flow Cut Reference Tool Drive method produces multiple cutting passes on either side of the center flow cut by allowing you to specify a reference tool diameter to define the total width of the area to be machined and a Stepover Distance to define the interior passes.





This method is useful for cleanup machining after roughing out an area with a large tool. This method also uses the Cut Type, Stepover Distance, Sequencing, Reference Tool Diameter, Overlap Distance, and Steep Containment options.

The Flow Cut Reference Tool Options

Maximum Concavity allows you to determine where Flow Cuts are created based on the Angle of Concavity. Cutting moves are created only where the Angle of Concavity is less than or equal to the specified Maximum Concavity angle. The value you enter must be positive and less than or equal to 179.0 degrees. When the Angle of Concavity exceeds the specified Maximum Concavity angle, the tool will retract and traverse.

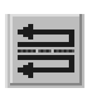
Minimum Cut Length allows you to eliminate short tool path segments that may occur in isolated areas of the part. Cutting moves shorter than this value are ignored. This option is useful in eliminating very short cutting moves that occur at the intersection of fillets.

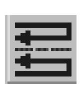
Hookup Distance allows you to eliminate unwanted gaps in the tool path by connecting disjointed cutting motions that exceed the specified Maximum Concavity angle. These unwanted motions occur where the tool retracts from the Part surface and are caused by gaps between surfaces or variations in the Angle of Concavity that exceed the specified Maximum Concavity angle. The value you enter determines the distance the tool will span to connect the end points of cutting moves. The two ends will be connected by linearly extending the two paths.

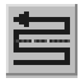
Cut Type (Zig-Zag and Zig) allows you to define how the cutter moves from one cut pass to the next.

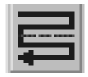
Stepover Distance allows you to specify the distance between successive passes.

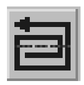
Sequencing enables you to determine the order in which the cut passes are executed.

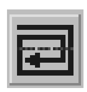
Inside-Out  results in the cut starting at the center of the Flow Cut pass and moving toward one of the outside passes. The tool then moves back to the center cut and works its way toward the opposite side. You may start the sequencing by choosing either side of the center of the Flow Cut.

Outside-In  results in the cut starting at one of the outside passes and moving to the center of the Flow Cut pass. The tool then picks up the outside cut on the opposite side and works its way to the center cut again. You may start the sequencing by choosing either side of the center of the Flow Cut.

Steep Last  results in the cut moving from non-steep side to the steep side.

Steep First  results in the cut moving from the outside pass on the steep side to the outside pass on the non-steep side. The Steep First sequence is available for Zig, Zig-Zag, and Zig-Zag with Lifts patterns.

Inside-Out Alternate  always cuts a Flow Cut valley from the middle Flow Cut pass. The cut starts at the center pass, moves to an inside pass and then to the inside pass on the opposite side. The cut then moves to the pass in the next pair on the first side and then to the pass in the same pair on the second side. If one side has more offset passes then the other side, all the extra passes on that side are machined after machining the passes which are paired on both sides. Inside-Out Alternate sequence can be generated with a Zig, Zig-Zag, or Zig-Zag with Lifts pattern.

Outside-In Alternate  always machines a Flow Cut valley from passes in an outside pair to inside pair, and then to the middle Flow Cut pass when necessary. The cut starts at one outside pass and moves to the other outside pass on the opposite side. The cut then moves to the pass in the next pair on the first side and to the pass in the same pair on the second side. After finishing the passes in the inside pair, the cut will move to the middle Flow Cut pass, if required. If one side has more offset passes then the other side, all the extra passes on that side are machined before machining the passes in pair on both sides. Outside-In Alternate sequences can be generated in a Zig, Zig-Zag, or Zig-Zag with Lifts pattern.

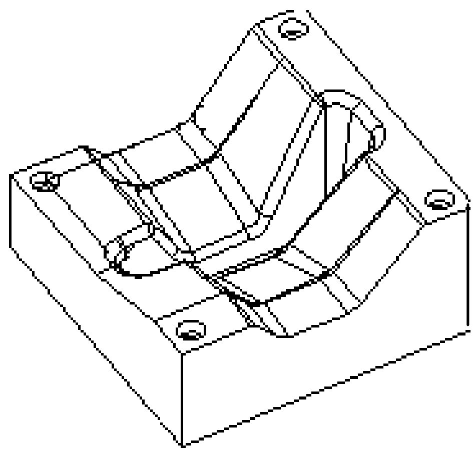
Reference Tool Diameter enables you to specify the width of the finishing cut region based on the diameter of the previous roughing (reference) tool. The tool diameter specified must be larger than the current tool.

Overlap Distance enables you to extend the width of the area defined by the Reference Tool Diameter along the tangent surfaces.

Steep enables the use of steepness to control the cut regions and their cut directions. As in Area Milling Drive Method, Flow Cut Steep Containment allows the restriction of the cut area based on the steepness of the tool path. Steepness is defined by specifying a Steep Angle and a Steep or Non-Steep option. Cut direction is defined by specifying a Steep Cut or Non-Steep Cut Direction. You can also choose to machine flow cuts on both sides alternatively with a rounded or standard turn at each end, or machine side by side from the steep side to non-steep side.

Activity 3—3: Creating a Reference Tool Operation

In this activity, you will create a Flowcut Drive Method operation and generate a subsequent tool path. You will then Copy and Paste the operation with the Operation Navigator, edit the operation, using a smaller tool, and generate a subsequent tool path. You will repeat this procedure, using smaller tools, until all areas of the are machined.

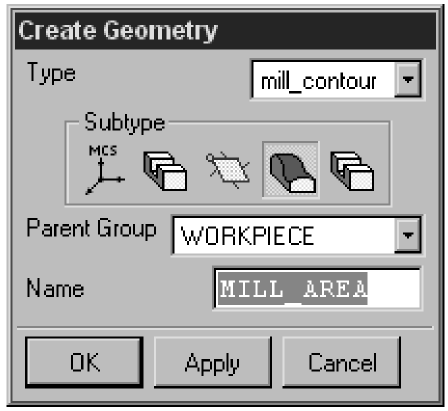


Step 1 Create the Geometry Parent Group for trim geometry.

- ❑ Continue using *****_deep_mold_mfg.prt**
- ❑ Make sure that the blank stock is displayed.
- ❑ As shown below, select the **Create Geometry** icon from the Create toolbar.

The Create Geometry dialog is displayed.

Make sure the Type is set to **mill_contour.**



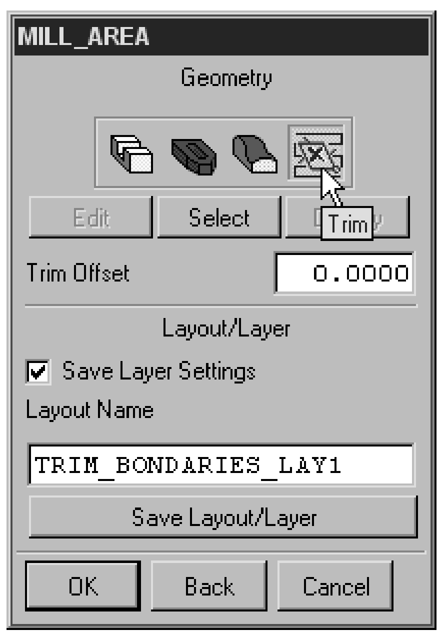
❑ Choose the **MILL_AREA** icon.

❑ Select **WORKPIECE** for the Parent Group.

❑ Key in **TRIM_BOUNDARIES** for the name.

❑ Choose **OK**.

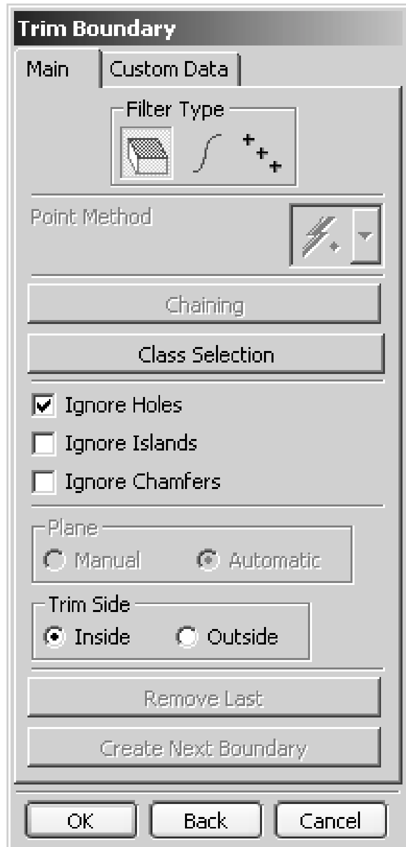
The MILL_AREA dialog is displayed.



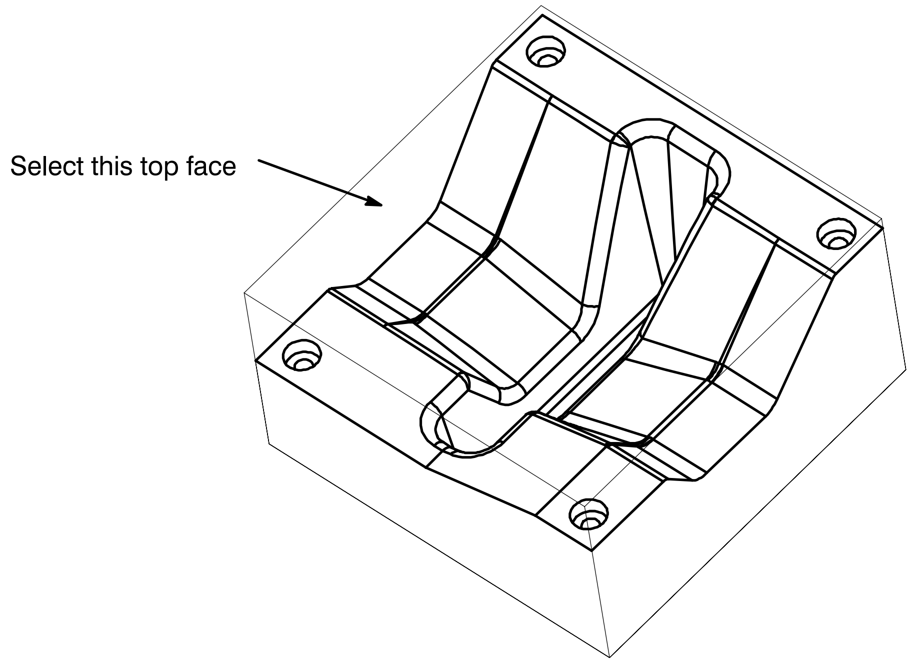
❑ Choose the **TRIM** icon.

❑ Choose **Select**.

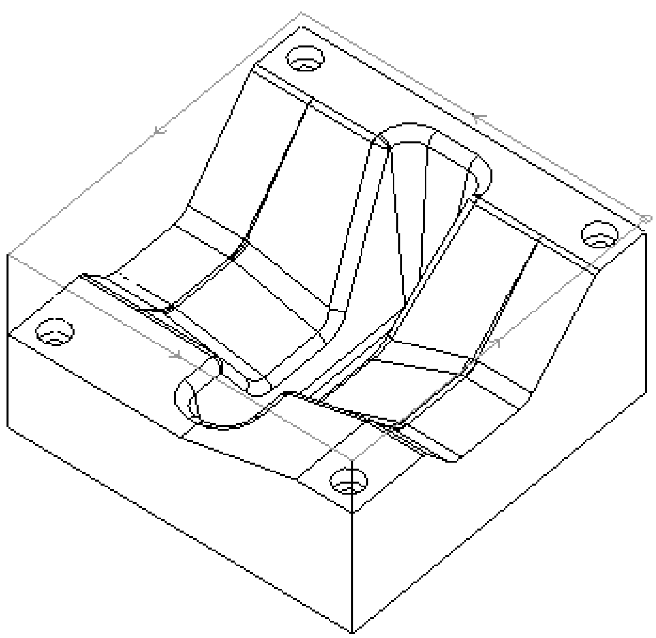
The Trim Boundary dialog is displayed.



- ❑ In the Trim Boundary dialog, choose Trim Side **Outside**.
- ❑ Select the top face of the Blank.

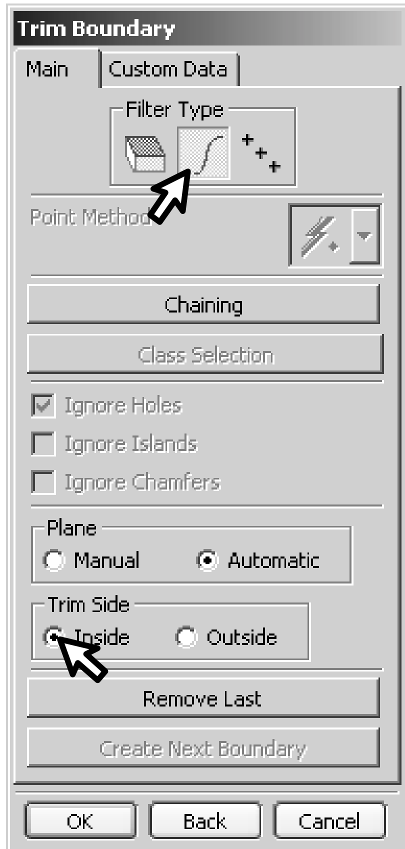


Note the boundary markers on the display.

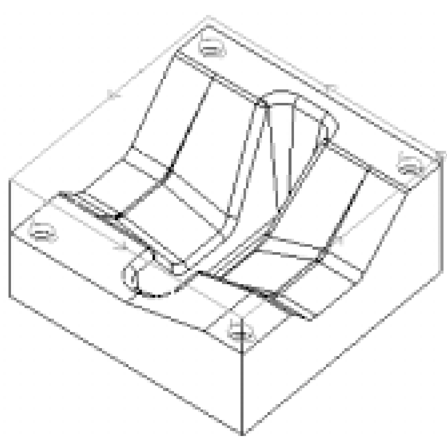


3

❑ On the Trim Boundary dialog, which is displayed, choose the **Curve Boundary** icon and set the Trim Side to **INSIDE.**



- ❑ Choose the four counterbored holes. Each time you choose a hole, you should select **Create Next Boundary**. If you do not, you will get one boundary around all four holes.

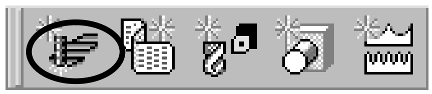


- ❑ Choose **OK** twice.

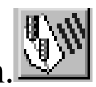
Step 2 Create the Flowcut Operation.

To create this operation, you will use the Geometry Parent group, TRIM_BOUNDARIES, that you created previously. The cutting tool will avoid the trim boundaries, which you created for the counterbored holes.

❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.

❑ Choose the **FLOWCUT_REF_TOOL** icon. 

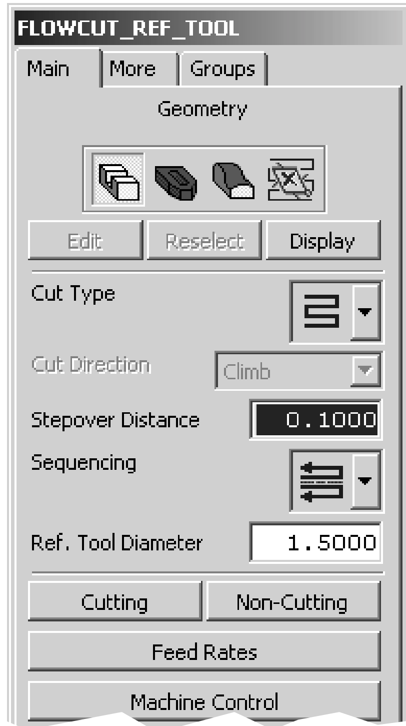
❑ Set the following:

- Program: **METHOD_1**
- Use Geometry: **TRIM_BOUNDARIES**
- Use Tool: **BALLMILL−1.0**
- Use Method: **MILL_FINISH**

❑ Key in **FLOWCUT_1.0_BALL** for the name.

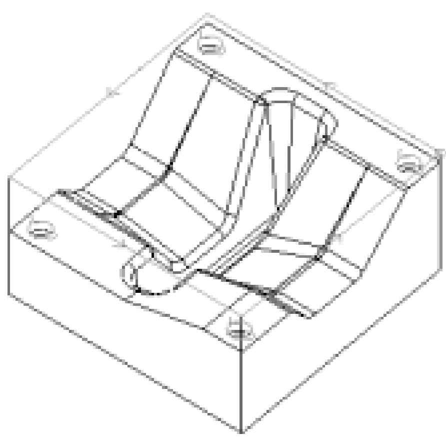
❑ Choose **OK.**

The FLOWCUT_REF_TOOL dialog is displayed.



❑ Under the Geometry label, choose the Trim icon and **Display**.

The five Trim boundaries (one at each counterbored hole and one around the Blank) are displayed.

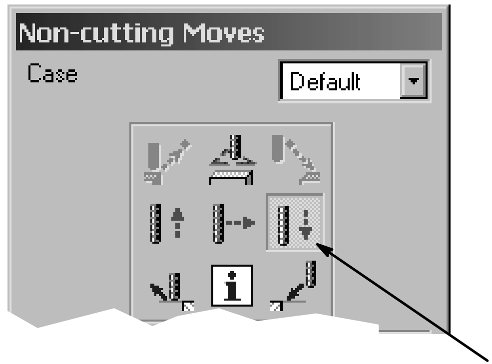


Step 3 Set the Non-Cutting Options, create a clearance plane.

❑ Choose **Non-Cutting.**

You will now create a Clearance Plane used when approaching the part.

❑ Select the Approach icon.



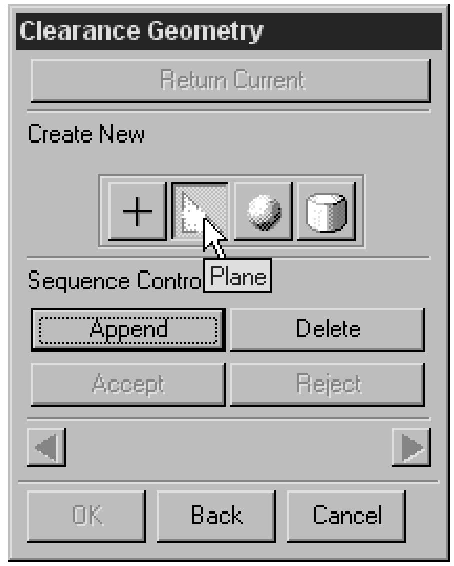
❑ Change the Approach Status to **Clearance**.

❑ Choose the **Clearance** icon.

The Clearance Geometry dialog is displayed.

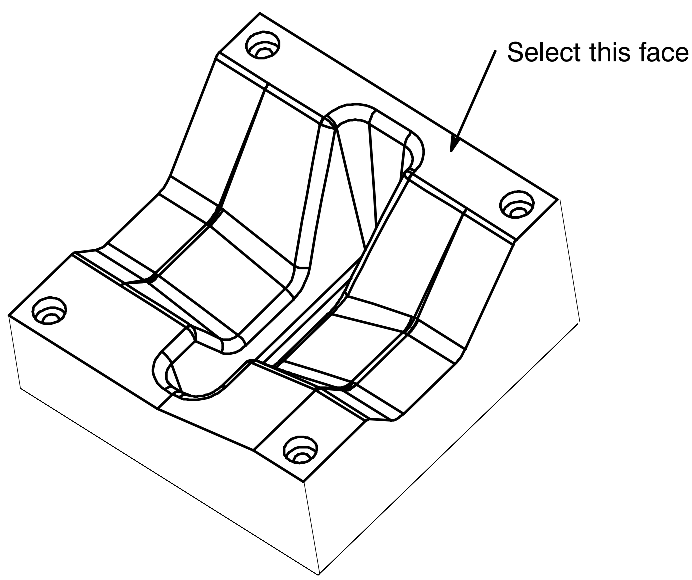
❑ Choose **Append** from the Clearance Geometry dialog.

❑ Choose the **Plane** icon.



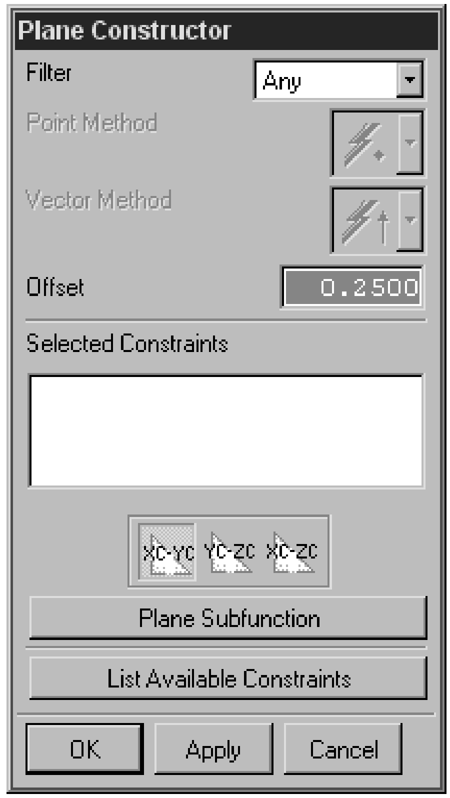
The Plane Constructor dialog is displayed

❑ As shown in the following illustration, select the upper face.



Note that the Selected Constraint is **Offset to Face**.

❑ Enter **.250** as the Offset.



❑ Choose **OK.**

Note the plane symbol is displayed.

❑ On the Clearance Geometry dialog, choose **Accept**.

❑ On the Clearance Geometry dialog, choose **Return Current**.

You have created a Clearance Plane .250" above the upper face.

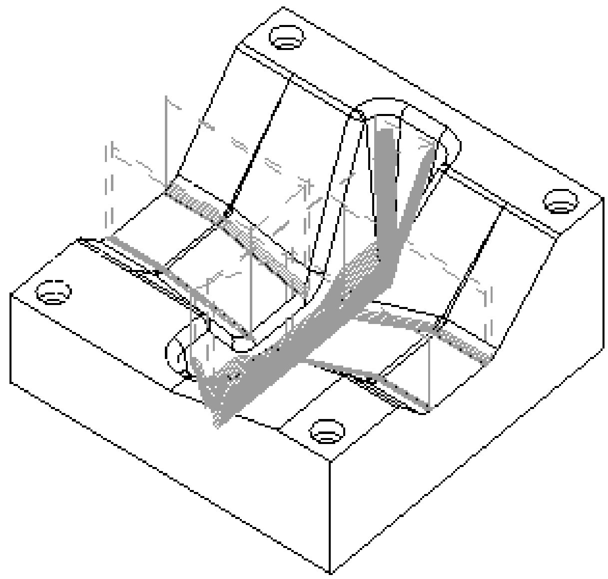
❑ Choose **OK** to return to the **FLOWCUT_REF_TOOL** dialog.

❑ Choose the **More** tab and change the **Overlap Distance** to **.250.**

Step 4 Generate the tool path.

❑ Choose the **Main** tab and **Generate** the tool path**.**

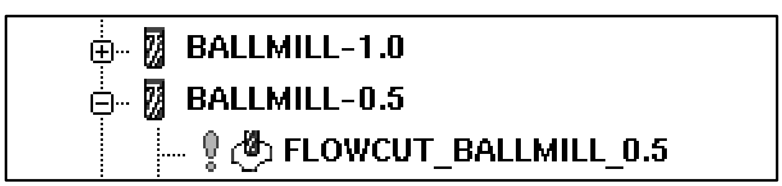
Your tool path should look similar to the following illustration.



- ❑ After examining the tool path, choose **OK.**

Step 5 Reusing the previously created Operation.

- ❑ Change the view of the Operation Navigator to the **Tool View.**
- ❑ **Copy** the operation **FLOWCUT_1.0_BALL**, and **Paste** it under the **BALLMILL_0.5** tool.
- ❑ Change the Name of the copied operation to **FLOWCUT_BALLMILL_0.5**



- ❑ Edit the new operation by double clicking on it.

- ❑ Change the Reference Tool Diameter to **1.0** (the previous operation's tool diameter).
- ❑ **Generate** the tool path.
- ❑ Choose **OK.**

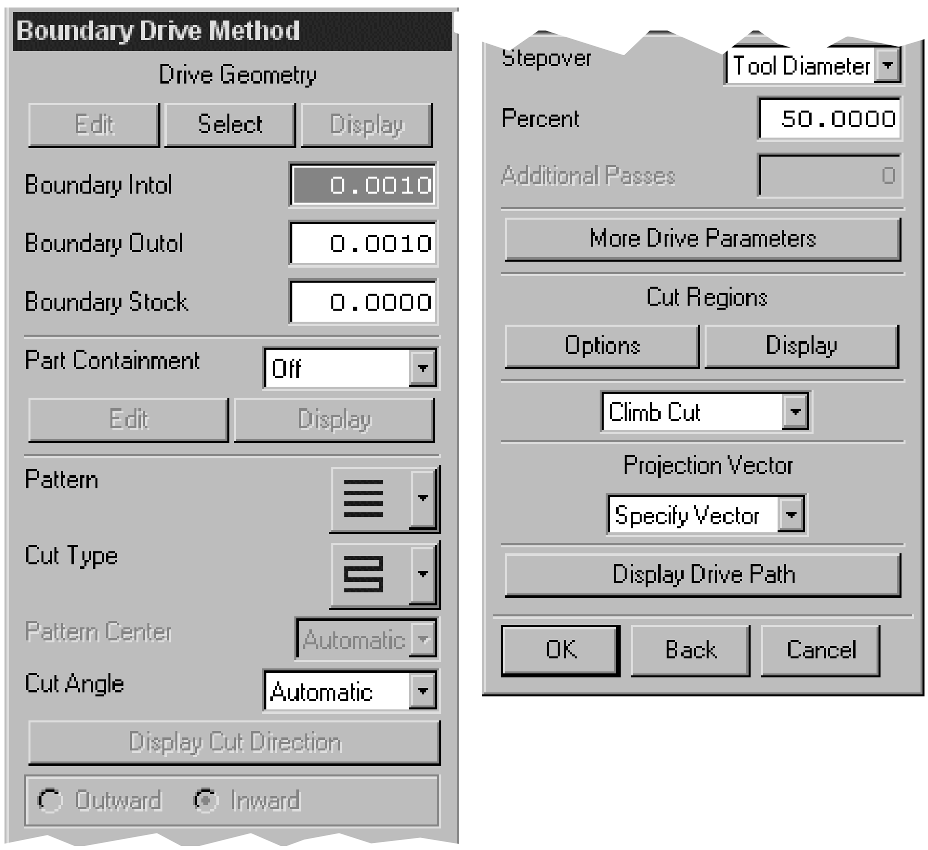
Step 6 Reusing the just created Operation.

- ❑ **Copy** the operation **FLOWCUT_BALLMILL_0.5**, and **Paste** it under the **BALLMILL_0.375** tool in the Operation Navigator Tool View.
- ❑ Change the Name of the copied operation to **FLOWCUT_BALLMILL_0.375.**
- ❑ Edit the new operation by double clicking on it.
- ❑ Change the Reference Tool Diameter to **0.5** (the previous operation's tool diameter).
- ❑ **Generate** the tool path.
- ❑ Choose **OK.**
- ❑ **Save** the part file.

You have completed and are finished with this activity.

Boundary Drive Method

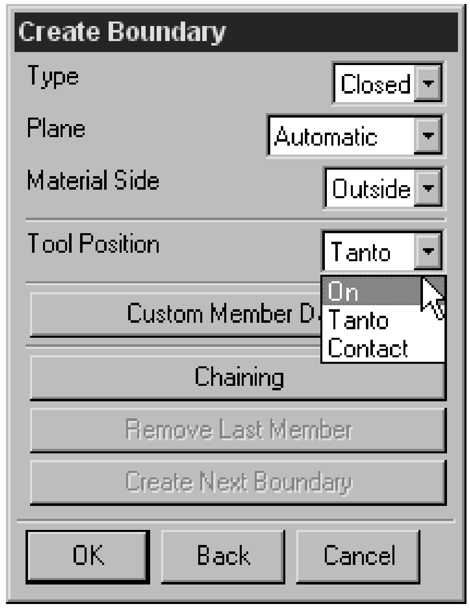
As mentioned earlier in this lesson, the Boundary Drive Method allows you to define cut regions by specifying Boundaries and Loops. Boundaries utilize cut regions which are defined by Boundaries, Loops, or a combination of both. The tool path is created by projecting drive points from the cut region to the Part Surface(s) in the direction of a projection vector. The Boundary Drive Method is useful in machining part surfaces requiring minimal tool axis and projection vector control.



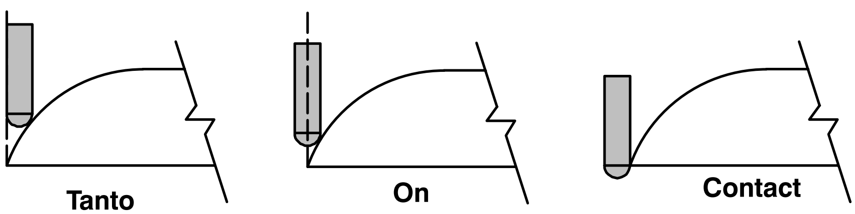
The following are some of the options available when using the Boundary Drive Method.

Boundary Drive Method - Select

When you use the **Select** Drive Geometry option, and set the Mode to **Curves/Edges**, you have an additional Tool Position option **– Contact.**



The Contact condition is dependent on the boundary and drive geometry. The tool will stay in contact with the boundary changing the contact point as necessary.

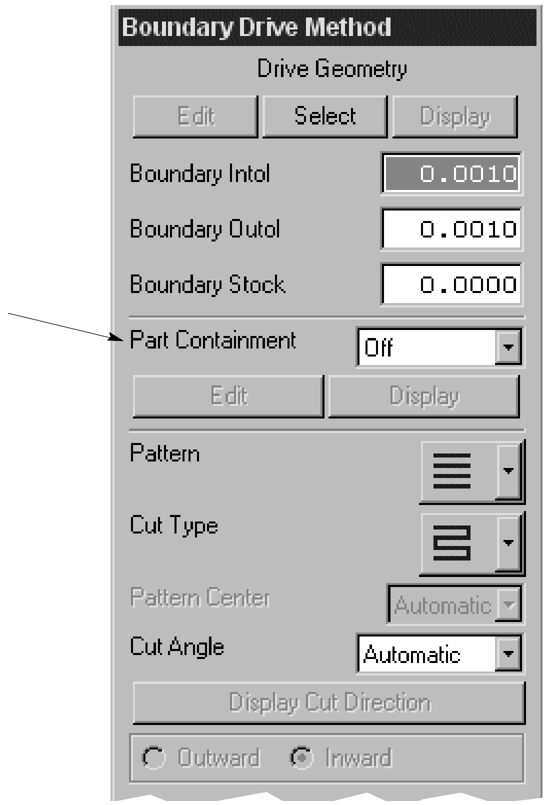


The Contact position cuts the entire boundary. The Tanto and On conditions leave a portion of the boundary uncut. The Contact condition is very useful when cutting exterior surfaces, but it is not designed for cutting pockets.

NOTE You cannot mix Contact position with On or Tanto.

Boundary Drive Method — Part Containment

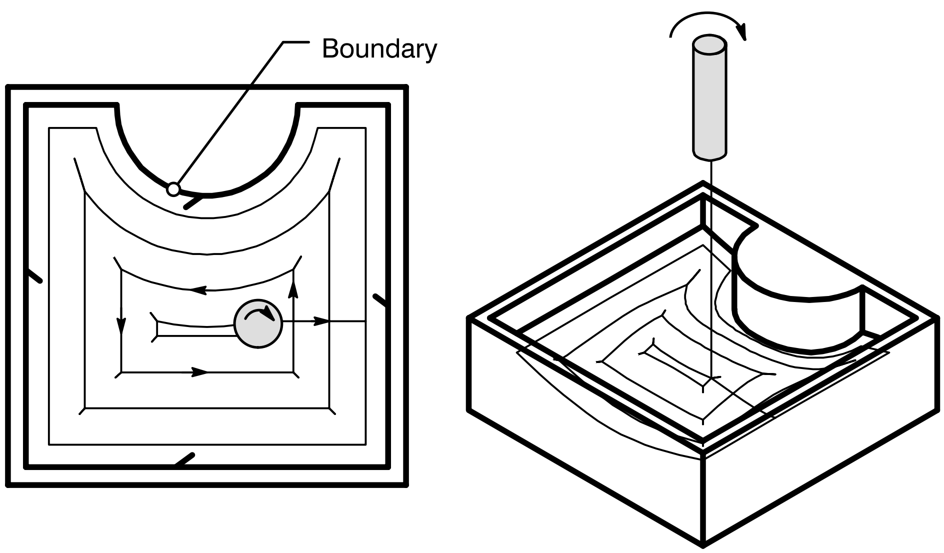
The Part Containment options define cut regions by using exterior edges of Part Surfaces. When you select this option, the system automatically identifies the exterior edges of all Part Surfaces and creates Loops along these edges.



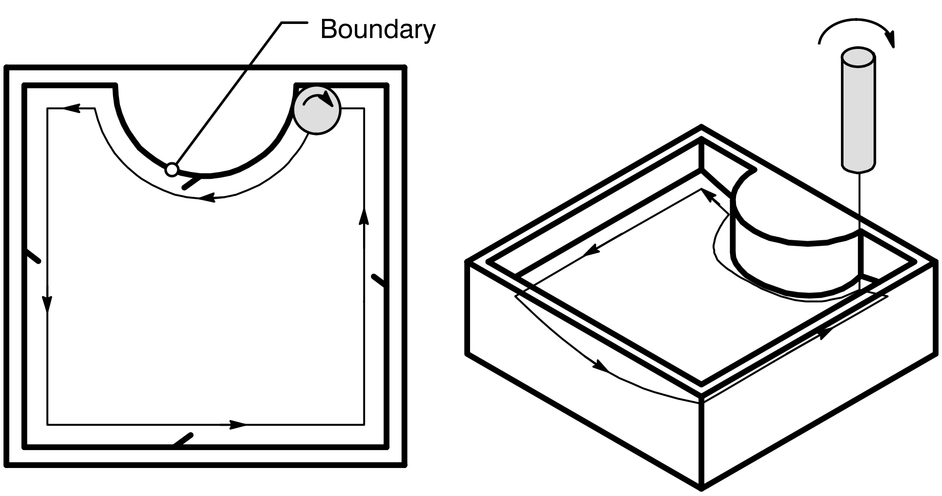
Loops define the main area to be cut as well as islands and pockets to be avoided. Loops are similar to Boundaries in that they define cut regions. They are dissimilar to Boundaries in that they are generated directly on the Part Surface edges and are not projected.

Boundary Drive Method — Pattern

Follow Periphery creates a cut pattern that produces a sequence of concentric passes following the contours of the cut region. Like Zig-Zag, this cut type maximizes cutting moves by allowing the tool to remain continually engaged during stepovers.

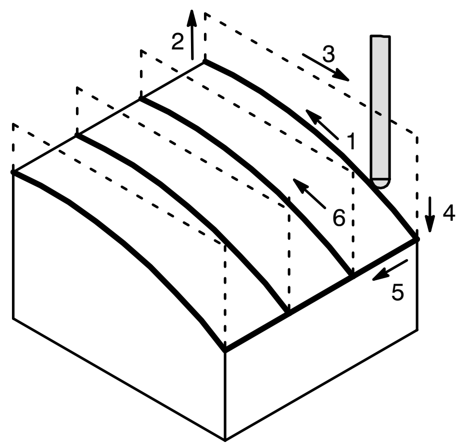


Profile creates a cut pattern that follows the perimeter of the cut region.

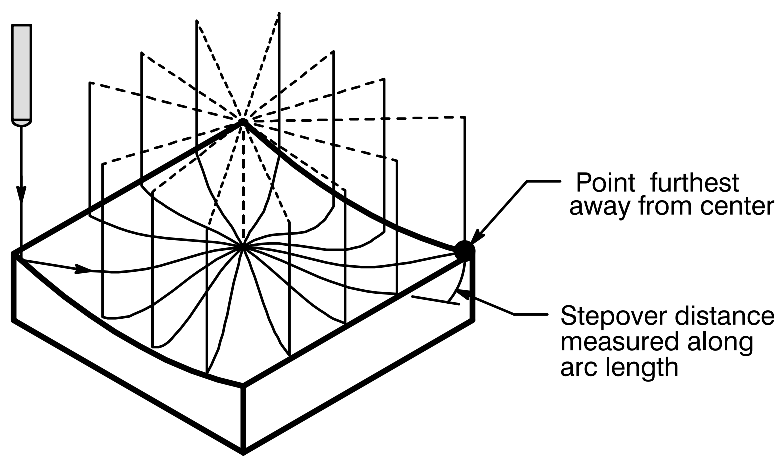


Parallel Lines creates a cut pattern defined by a series of parallel passes. This option requires you to specify a Cut Type of Zig-Zag, Zig, Zig With Contour, or Zig with Stepover and allows you to specify a Cut Angle.

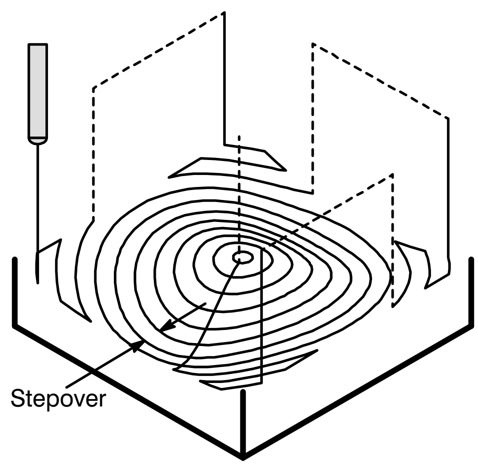
Parallel Lines
Cut Pattern



Radial Lines creates linear cut patterns extending from a user-specified or system calculated center point. This pattern allows you to specify a Cut Type, a Pattern Center and pocketing method as Inward or Outward. It also enables you to specify an angular stepover which is unique to this cut pattern. The Stepover distance for this cut pattern is measured along the arc length at the boundary point furthest away from the center.

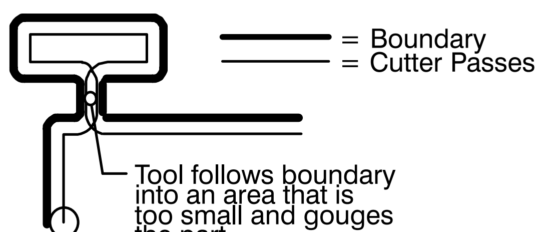


Concentric Arcs creates progressively larger or progressively smaller circular cut patterns from a user - specified or system - calculated optimum center point. This Cut Pattern allows you to specify a Cut Type, a Pattern Center, and a pocketing method as Inward or Outward. In areas such as corners that the full circular pattern cannot extend into, concentric arcs are created and connected by the specified Cut Type before the cutter moves to the next corner to continue cutting.



3

Standard Drive creates a cut pattern similar to Profile that follows the perimeter of the cut region. Unlike Profile, Standard Drive does not modify the tool path to prevent crossing over itself or to prevent gouging the part. Standard Drive causes the cutter to follow the boundary exactly as it was specified.

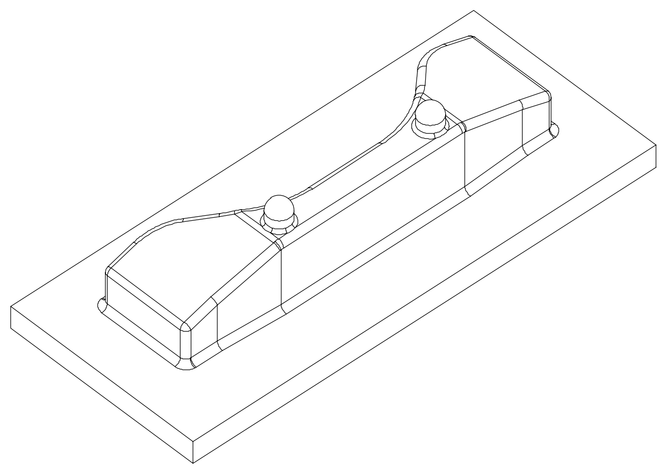


Activity 3–4: Using the Boundary Drive Method

In this activity, you will use boundaries to isolate portions of the geometry used in cutting the part. In a future activity, you will edit this particular operation and use the multi-level cutting option.

Step 1 Change part files and enter the Manufacturing Application.

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



❑ **Save As** ***_ **bumper_mfg .prt** where *** represents your initials.

❑ Enter the **Manufacturing** application.

The Operation Navigator is displayed.

Step 2 Create a Fixed Contour Operation.

❑ As shown below, select the **Create Operation** icon from the Create toolbar.

The Create Operation dialog is displayed.

- ❑ Make sure the Type is **mill_contour**.

- ❑ Choose the **Fixed Contour** icon.

  Since the Boundary Drive Method will not inherit an external boundary Parent Group (MILL_BND), you will create the boundary inside of the operation.

- ❑ Set the following:

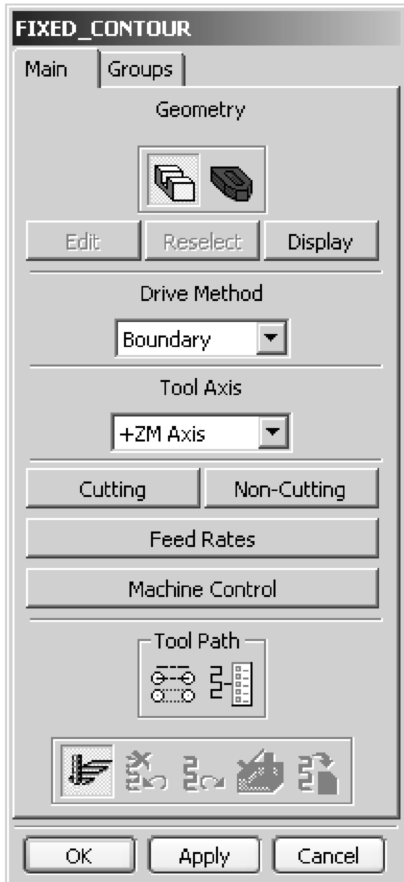
  - Program: **SPIRAL_AND_BOUNDARY**
  - Use Geometry: **WORKPIECE**
  - Use Tool: **BALLMILL-0.25**
  - Use Method: **MILL_FINISH**

- ❑ Key in **boundary_1** for the Name.

- ❑ Choose **OK.**

3

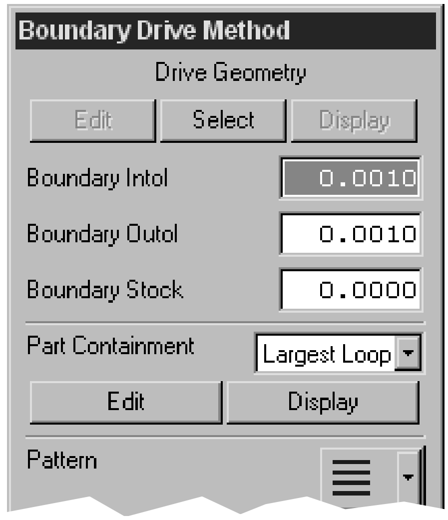
The Fixed Contour dialog is displayed.



Step 3 Create the Boundary.

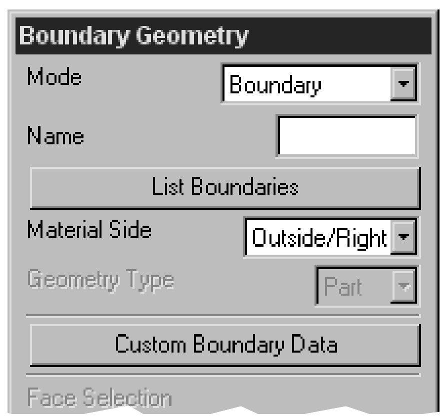
❑ Under the Drive Method label, choose **Boundary.**

The Boundary Drive Method dialog is displayed.



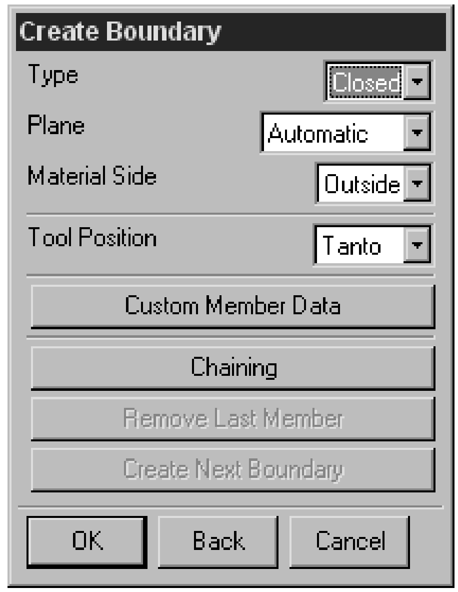
❑ Under Drive Geometry, choose **Select.**

The Boundary Geometry dialog is displayed.



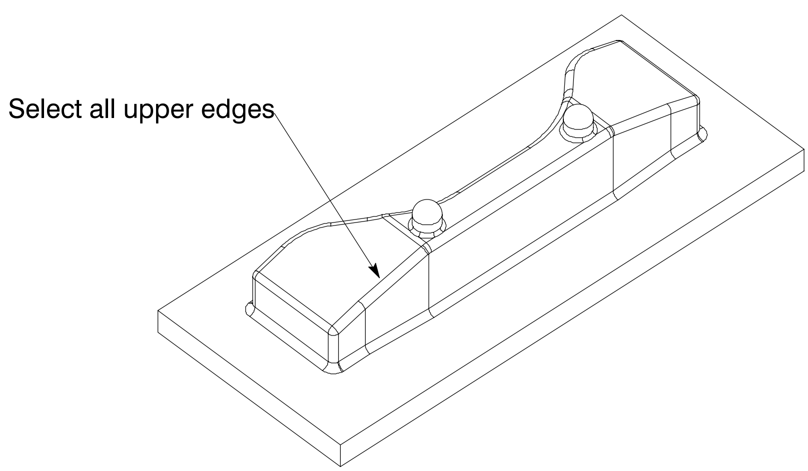
❑ Change the Mode to **Curves/Edges.**

The Create Boundary dialog is displayed.



- ❑ Change Material Side to **Inside.**
- ❑ Change Tool Position to **On.**
- ❑ Change the Plane to **User-Defined→Plane of WCS.**

  The Create Boundary dialog is again displayed.
- ❑ Select all upper edges of the radius around the bumper face.

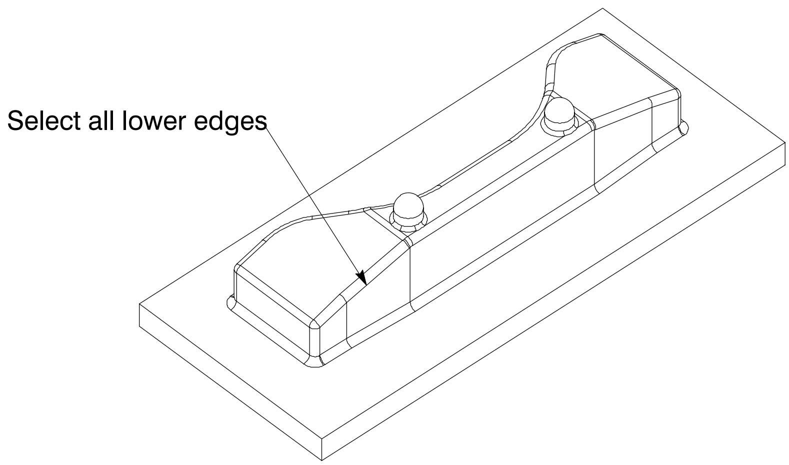


You will now create the lower boundary.

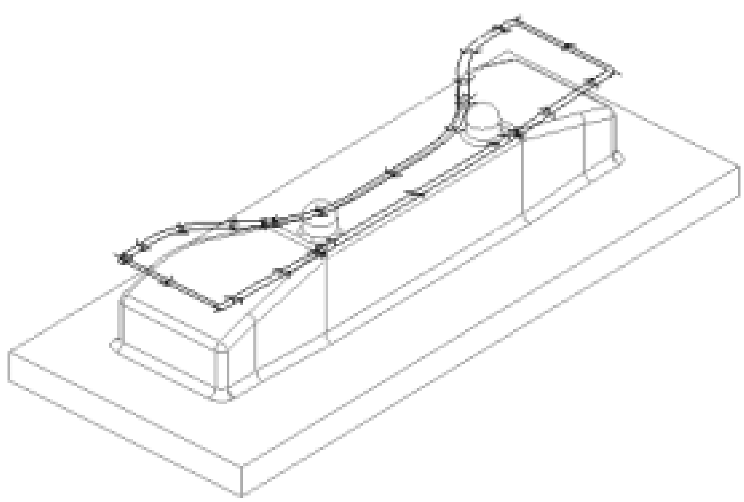
❑ After selecting all upper members, choose the **Create Next Boundary** button.

❑ Change Material Side to **Outside.**

❑ Select all the lower edges of the radius around the bumper face.

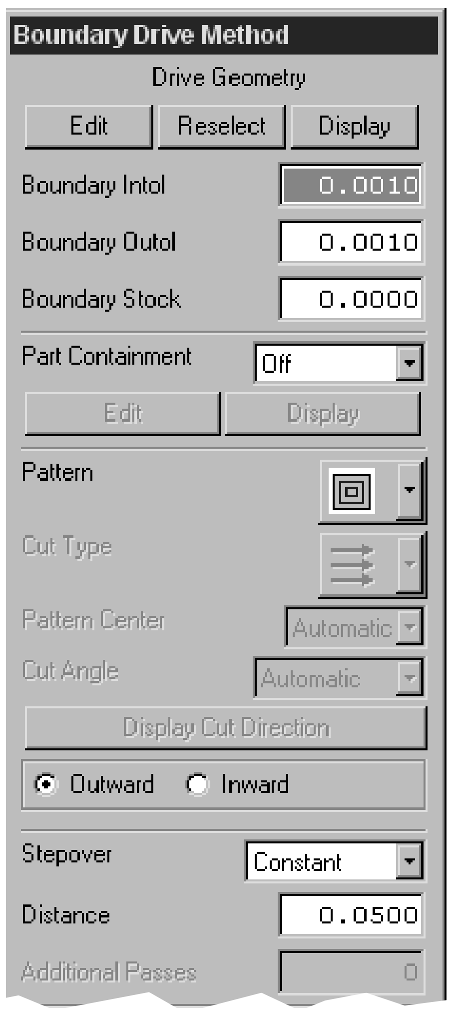


❑ Choose **OK** until you return to the Boundary Drive Method dialog.



Step 4 Specify Boundary Drive Method Option Settings.

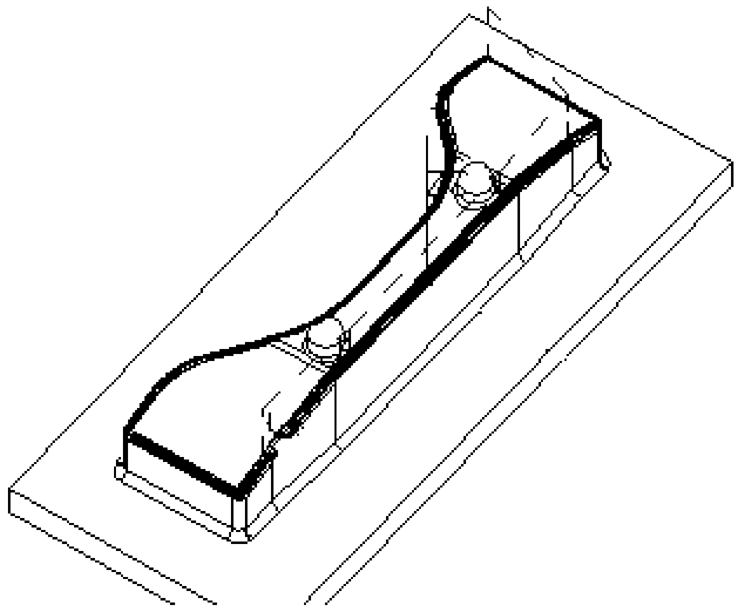
- ❑ Turn Part Containment **OFF.**
- ❑ Change the Pattern to **Follow Periphery.**
- ❑ Choose the **Outward** radio button.
- ❑ Change the Stepover to **Constant.**
- ❑ Change the Distance to **0.05.**



- ❑ Choose **Display Drive Path.**
- ❑ Choose **OK** to return to the Fixed Contour dialog.

Step 5 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.



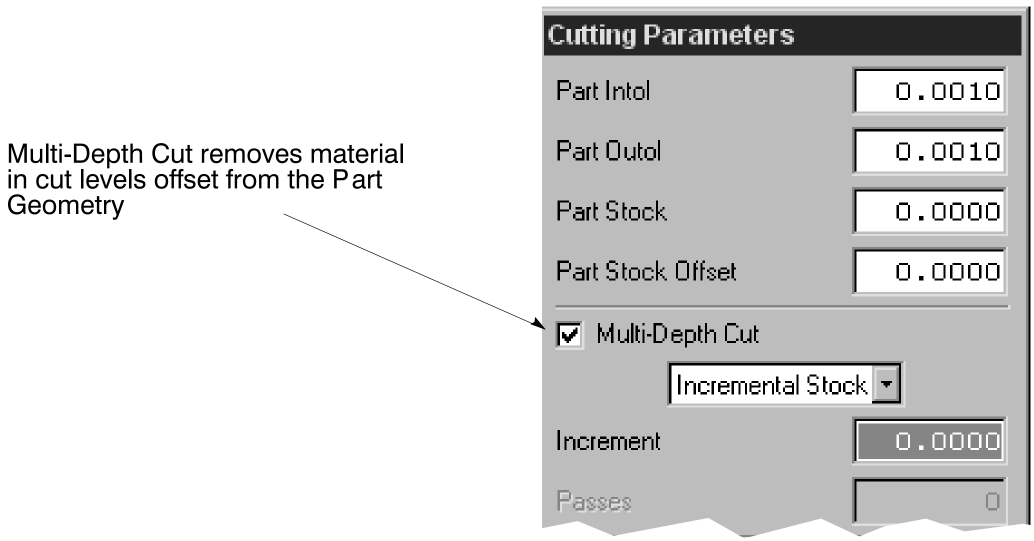
3

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

You are finished with this activity.

Multi Depth Cutting

Fixed Contour offers a Multi-Depth cut option that removes material by gradually machining toward the Part Geometry one offset cut level at a time. These settings are located under the Cutting option on the Fixed Contour dialog.



The tool path in each cut level is calculated separately as an offset of the contact points normal to the Part Geometry. Since the tool path contours can change as they diverge further away from the Part Geometry, the tool path in each cut level is calculated independently.

3

CAUTION Multi-Depth Cut ignores the custom stock values on Part Geometry.

Tolerance values

The tool path's tolerance of the final cut level applies the user-defined Intol/Outol values specified in the Operation Parameter dialog. The Intol/Outol is calculated for each of the previous levels by multiplying the total distance from the cut level to the part geometry by ten percent. This allows the Intol/Outol for rough cuts to increase in proportion to the total distance from the cut level to the Part Geometry and eliminates excessive processing time.

Traversal

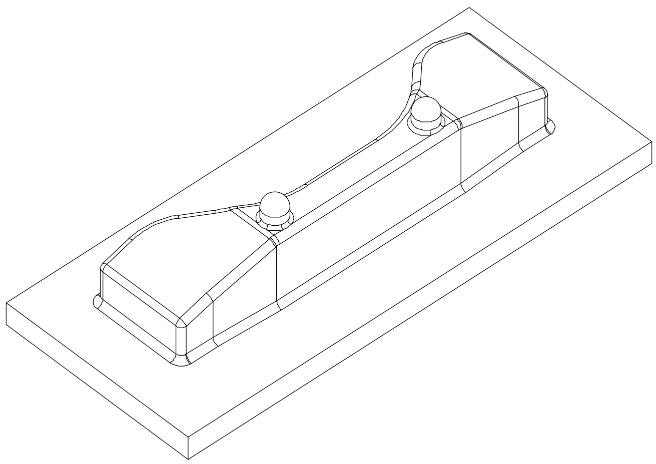
The traversal between levels can be minimized if the Engage option is set to Manual in Non-Cutting Moves. After finishing each level, the tool retracts to the previous level plus the Outol distance for that level before traversing and engaging to the next level.

Activity 3–5: Adding Multi-Depth Cutting to an Operation

In this activity, you will edit the previous operation, adding multi-depth cutting. You will be using this method to remove excessive stock.

Step 1 Edit the Boundary_1 operation.

❑ Continue using ***_bumper_mfg.prt**

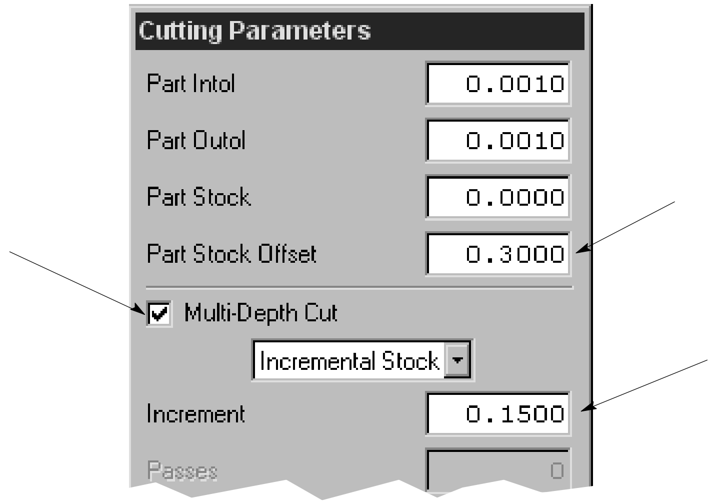


❑ In the Operation Navigator, double click on **BOUNDARY_1** .

The Fixed Contour dialog displays.

❑ Choose **Cutting**.

The Cutting Parameters dialog displays.

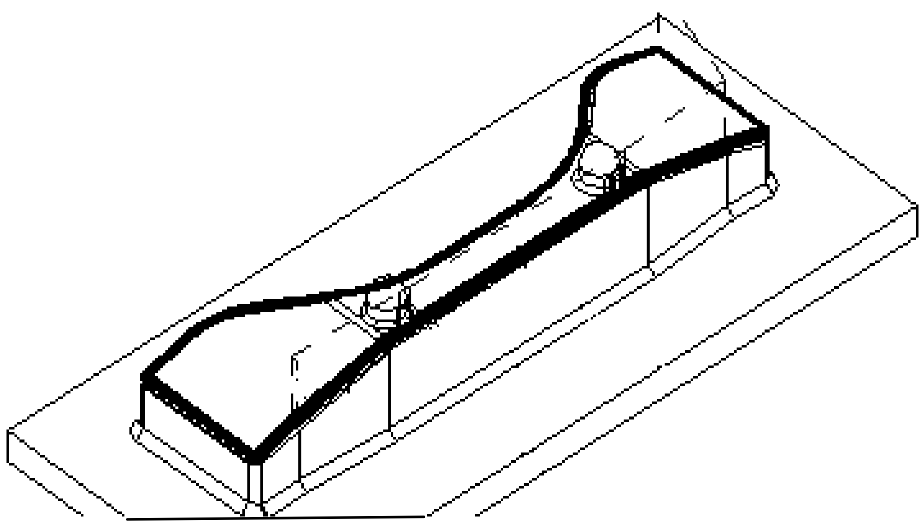


- ❑ Key in **0.300** in the Part Stock Offset field. This specifies the amount of stock on the surface to be machined.
- ❑ Choose **Multi-Depth Cut**.
- ❑ Leave the default at **Incremental Stock**.
- ❑ Key in **0.150** as the Increment.
- ❑ Choose **OK** to return to the Fixed Contour dialog.

3

Step 2 Generate the tool path.

- ❑ Choose the **Generate** icon and generate the tool path.



Two passes are generated, one at .150” stock above the part and the second with no stock.

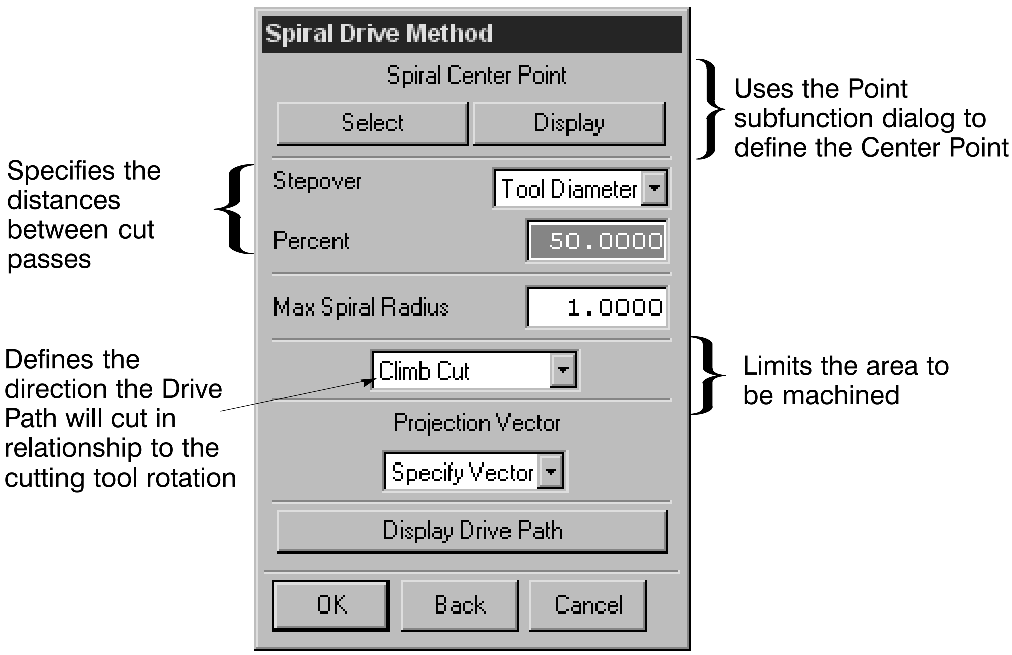
- ❑ Choose **OK** to accept the operation.
- ❑ **Save** the part file.

You are finished with this activity.

Spiral Drive Method

The Spiral Drive method defines Drive Points that spiral outward from a specified center point. The Drive Points are created within the plane normal to the projection vector and the center point. The Drive Points are then projected on to the selected part geometry along the projection vector.

Unlike other Drive Methods which require a sudden change in direction to stepover to the next cutting pass, Spiral Drive Method stepovers make a smooth, constant transition outward. Since this drive method maintains a constant cutting speed and relatively smooth motion, it is useful for high speed machining applications.



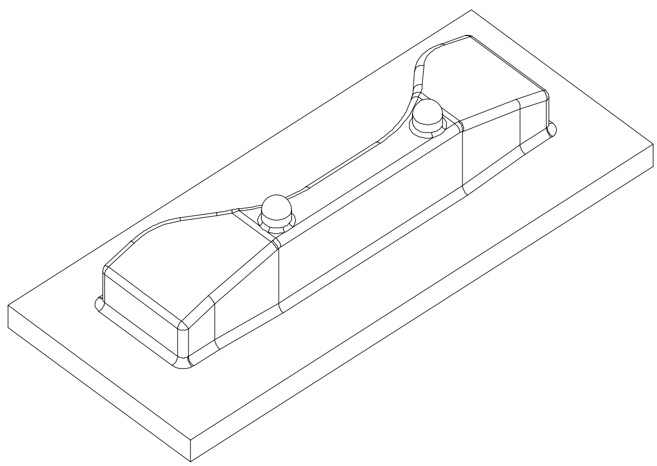
The Spiral Drive method is not constrained by the geometry to be machined. It is limited by the maximum radius value that is specified. This method works best on parts which are round in shape.

Activity 3–6: Creating a Spiral Drive Method Tool Path

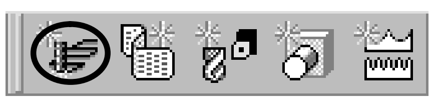
The following activity demonstrates use of the Spiral Drive Method.

Step 1 Create a Spiral Drive Fixed Contour operation.

❑ Continue with ***_bumper_mfg part.



❑ As shown below, select the **Create Operation** icon.



The Create Operation dialog is displayed.

Step 2 Create a Fixed Contour Operation.

❑ Make sure the Type is **mill_contour.**

❑ Choose the **Fixed Contour** icon.

❑ Set the following:

- Program: **SPIRAL_AND_BOUNDARY**
- Use Geometry: **WORKPIECE**
- Use Tool: **BALLMILL–0.25**
- Use Method: **MILL_FINISH**

❑ Key in **spiral_1** for the operation name.

❑ Choose **OK**.

The Fixed Contour dialog is displayed.

Step 3 Specify the Drive Method.

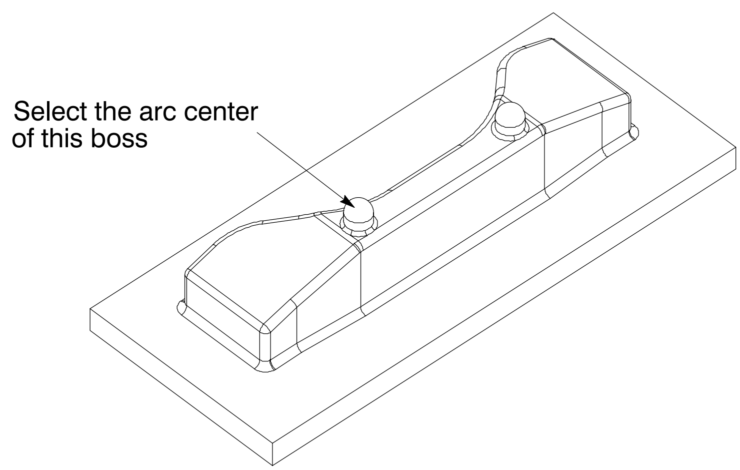
❑ Change the Drive Method to **Spiral.**

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

The Spiral Drive Method dialog displays.

❑ Under Spiral Center Point, choose **Select.**

❑ As shown below, select the arc center of the lower of the two bosses on the top of the bumper frame.



❑ Change the Stepover to **Constant.**

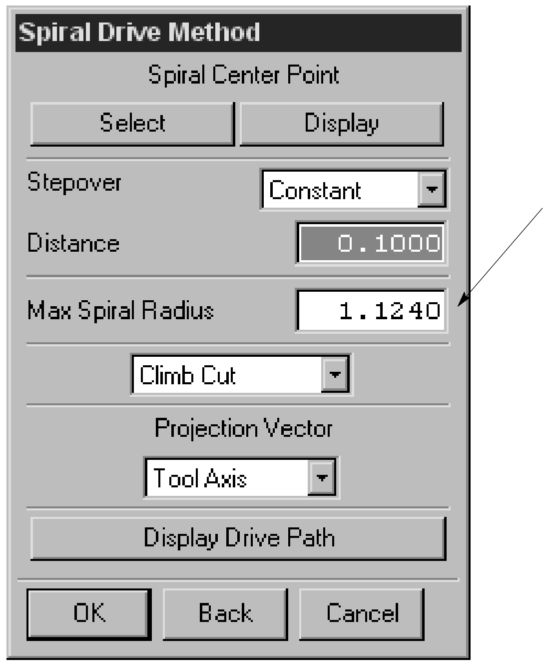
You will next specify the Max Spiral Radius value. In this case you must compute the value with the following considerations for Intol/Outtol requirements.

- Both bosses have a 1”radius
- The tool has a .125” radius

3

- The maximum radius that can be driven without falling off the boss is 1.125” (1.00” boss+.125” radius)
- Since the intol/outtol needs to be factored in, you will reduce 1.125 by 0.001”

❑ Key in **1.124** for the Max Spiral Radius.



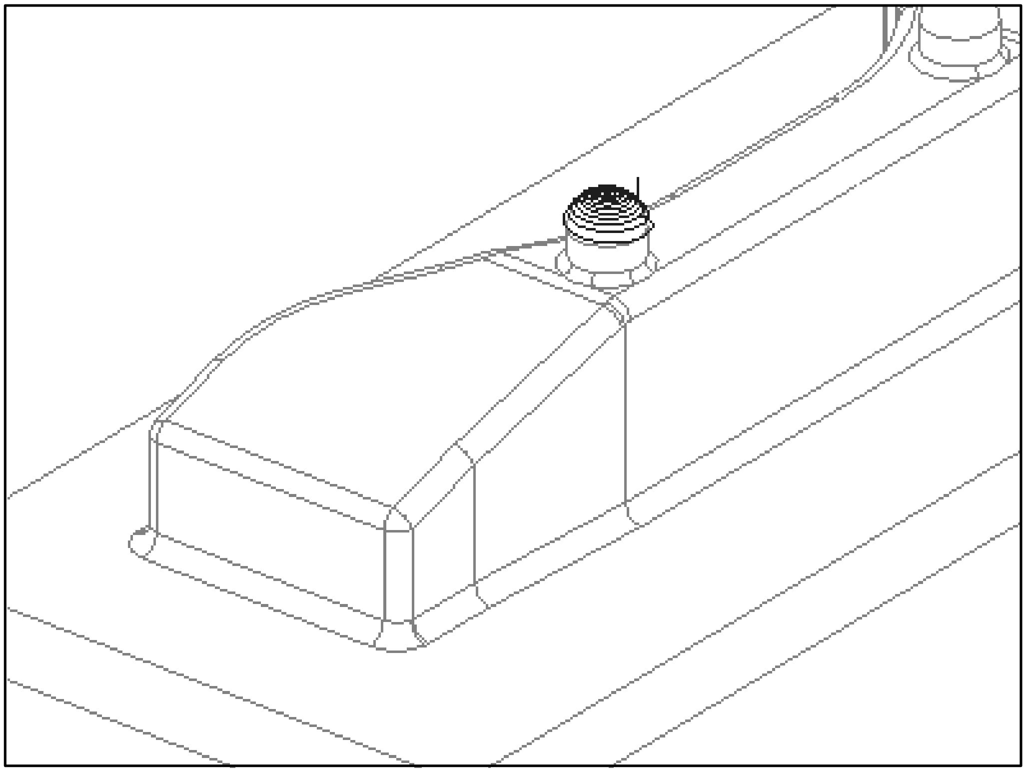
❑ Under Projection Vector, choose **Tool Axis.**

❑ Choose **Display Drive Path.**

❑ Choose **OK** to return to the Fixed Contour dialog.

Step 4 Create the first spiral Tool Path.

❑ Choose the **Generate** icon and generate the tool path**.**



3

- ❑ After examining the tool path, choose **OK** to accept the path.

Step 5 Create the second Spiral Drive Tool Path.

You can quickly create an identical tool path for the other boss on the bumper.

- ❑ Copy and paste the operation, **Spiral_1**, renaming it **Spiral_2**.
- ❑ Edit the **Spiral_2** operation.
- ❑ Change the Spiral Center to the arc center of the second boss.
- ❑ Choose the **Generate** icon and generate the **Spiral_2** tool path.
- ❑ Choose **OK** to accept the operation.
- ❑ **Save** the part file.

You are finished with this activity.

Surface Area Drive Method

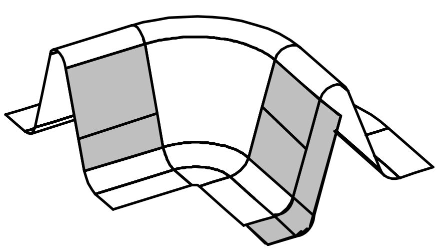
The Surface Area Drive Method provides additional control over both the Tool Axis and the Projection Vector. This method enables you to create an array of Drive Points that lie on a grid of Drive Surfaces.

The array of Drive Points is first created on the selected Drive Surface and then projected along the specified Projection Vector to the Part Surface(s). The tool positions to the Part Surface at Contact Points. As the tool moves from one Contact Point to the next, the tool path is created using the Output Cutter Location Point from the tip of the tool.

If Part Surfaces are not defined, the tool path can be created directly on the Drive Surfaces. The Drive Surfaces need not be planar, but must be arranged in an orderly grid of rows and columns.

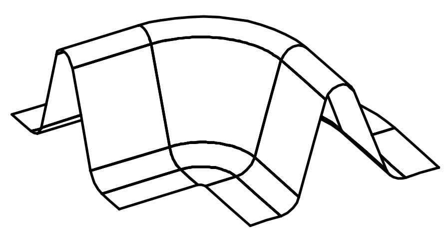
The faces and sheets that are selected as drive geometry have specific requirements. It is important to note the following:

- The drive geometry must form a grid of rows and columns where each row or column has the same number of objects as every other row or column in the grid
- Any two adjacent objects must share a common edge and may not contain gaps that exceed the Chaining Tolerance defined under Preferences
- When you use trimmed surfaces, trims will be ignored



Incorrect:

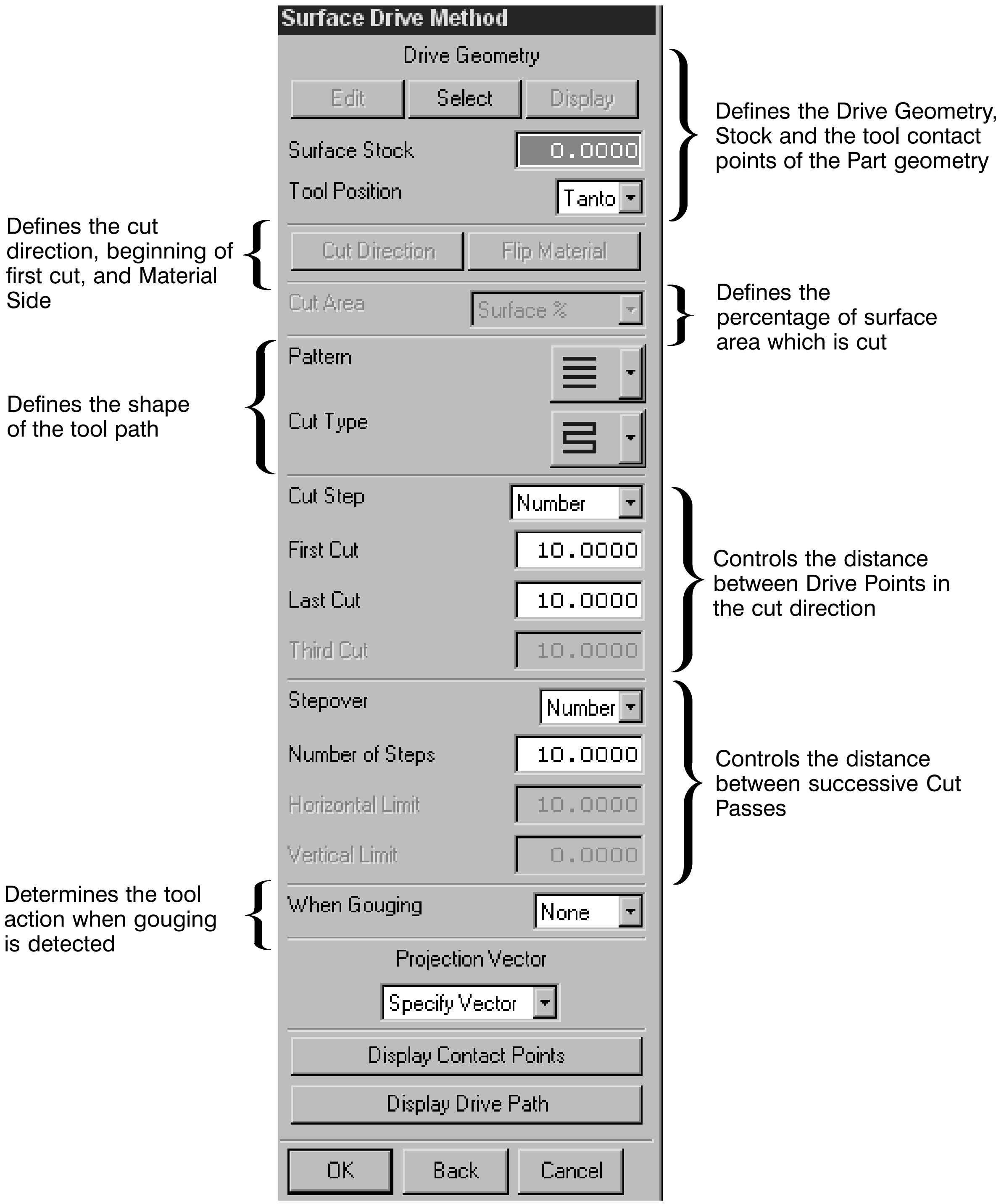
- Inconsistent number of objects in rows and in columns
- Adjacent object edges are not equal length



Correct:

- Consistent number of objects in rows and in columns
- Adjacent object edges are the same length

The Surface Drive Method dialog follows.

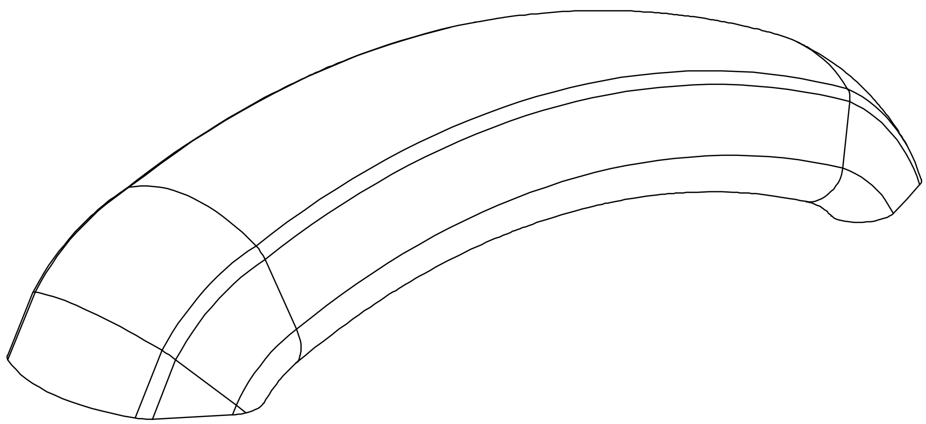


Activity 3–7: Using the Surface Drive Method

In this activity, you will edit an existing operation and change the Drive Method to Surface Drive.

Step 1 Open the part file, rename and enter the Manufacturing Application.

❑ Open the part **ama_fx_3.prt**



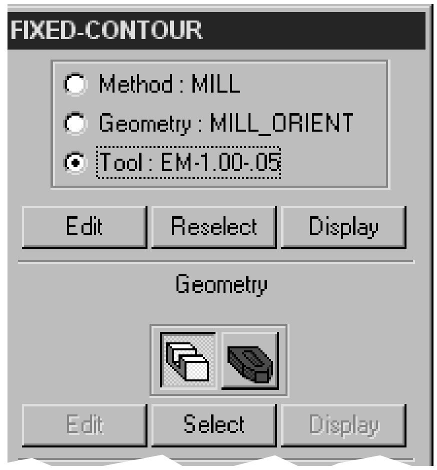
❑ Rename the part ***_**fx_3.prt** where *** represents your initials**.**

❑ Enter the **Manufacturing Application**.

Step 2 Edit an existing operation.

❑ Double-click on **SURFACE-DRIVE** in the Operation Navigator.

The Fixed Contour dialog is displayed.



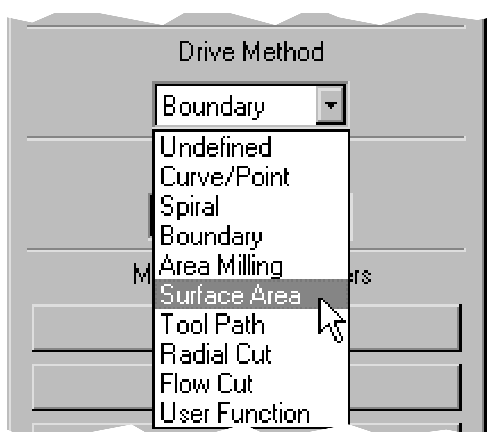
Some of the options that have been set for you are:

- the tool which is an end mill, designated EM-1.00–.05
- Engage and Retract Non-Cutting Moves

Step 3 Define the Drive Method as Surface Area.

You are going to specify the Surface Area Drive Method.

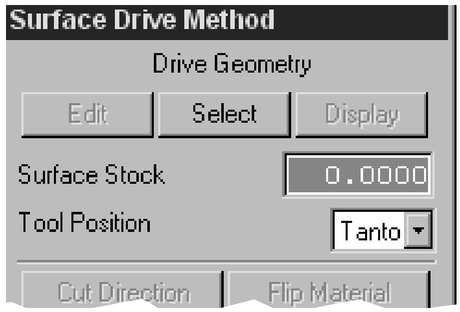
❑ In the Drive Method area, choose **Surface Area.**



The Drive Method warning dialog is displayed. This dialog displays each time you change to a different Drive Method.

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

The Surface Drive Method dialog is displayed.



Step 4 Define the Drive Geometry.

NOTE When you specified the options on the Fixed Contour dialog, you did not select any Part Geometry. Not selecting Part Geometry, in this particular operation, will make processing quicker.

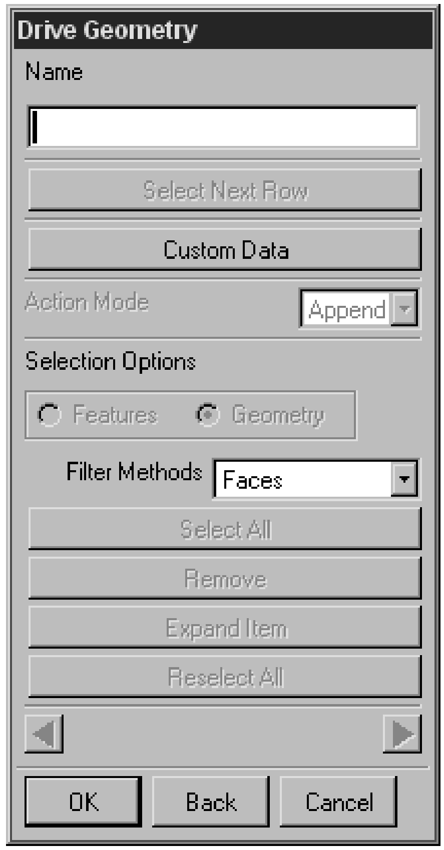
3

❑ Choose **Select** from the Drive Geometry area**.**

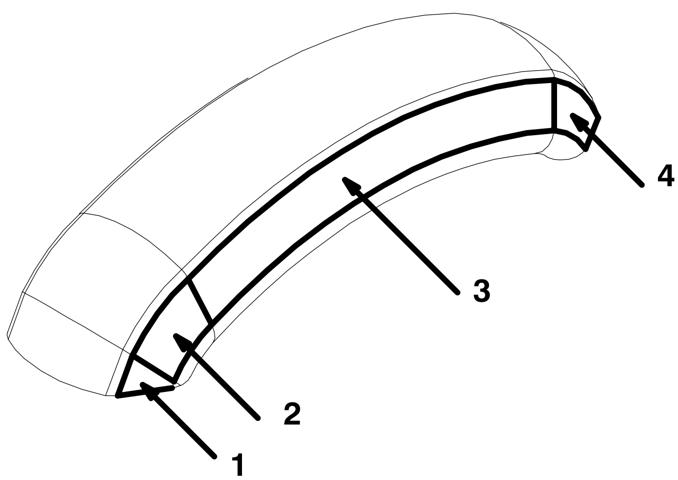
The Drive Geometry dialog is displayed.

You are now going to select the Drive Geometry for this operation. Remember that the geometry is selected in a grid pattern.

You are going to select the first row of surfaces. Notice that the Drive Geometry dialog defaults to Faces.

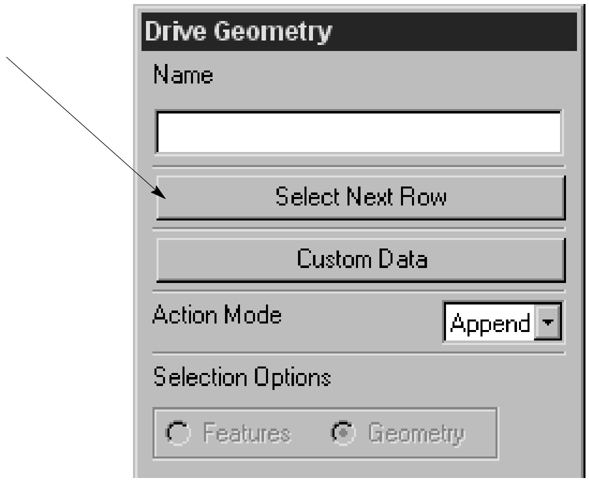


- ❑ In the graphic window, choose faces in the specified order as shown (four surfaces total).



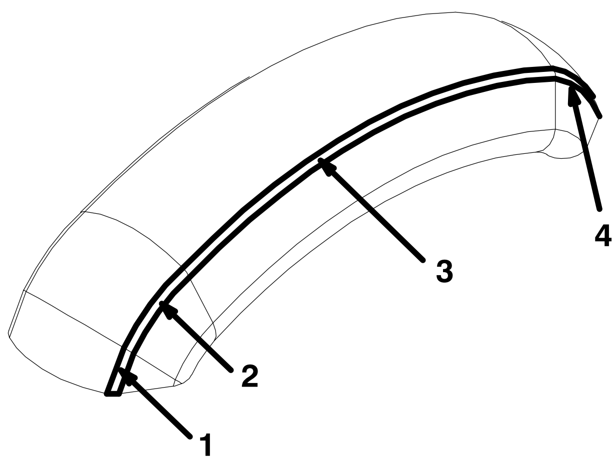
You are ready to select a new row. When you finish selecting the first row, you must specify that you want to begin selecting the next row.

- ❑ Choose the **Select Next Row** button from the Drive Geometry dialog.



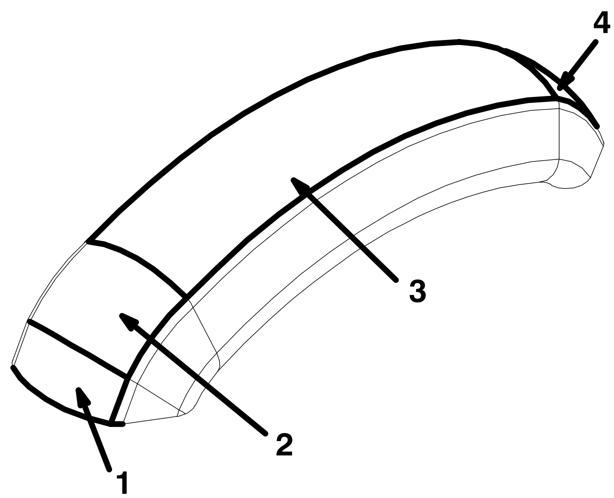
You must select the second row of surfaces and all subsequent rows in the same order as the first.

❑ Select the second row of surfaces as shown (four surfaces total).

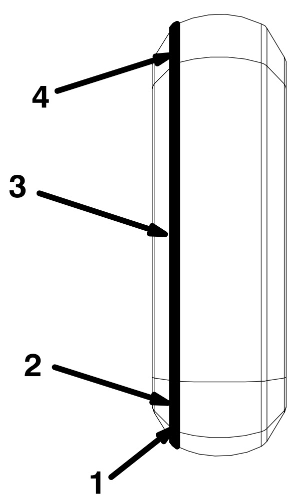


You do not need to select the Select Next Row button again. The number of surfaces in the subsequent rows must be the same.

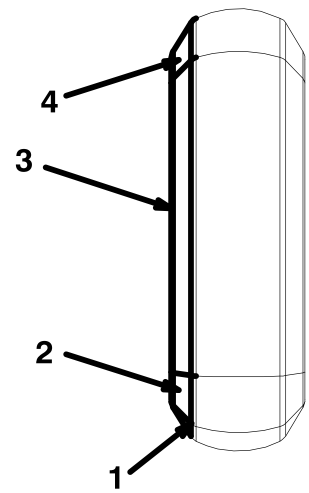
❑ Select the third row of surfaces as shown below (four surfaces total).



❑ Select the fourth row of surfaces as shown below(four surfaces total).

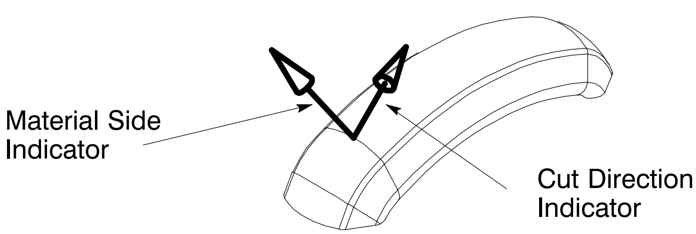


❑ Select the fifth and final row of surfaces as shown below (four surfaces total).

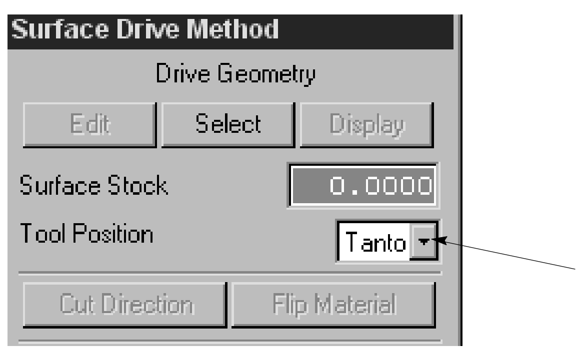


❑ Choose **OK** when finished.

Two cone heads are displayed. One indicates the Material Side and the other indicates the first Cut Direction.



NOTE When creating a Tool Path directly on the Drive Surface, Tool Position should be toggled to Tanto. Depending on the Projection Method used, ON may violate the Drive Surface.



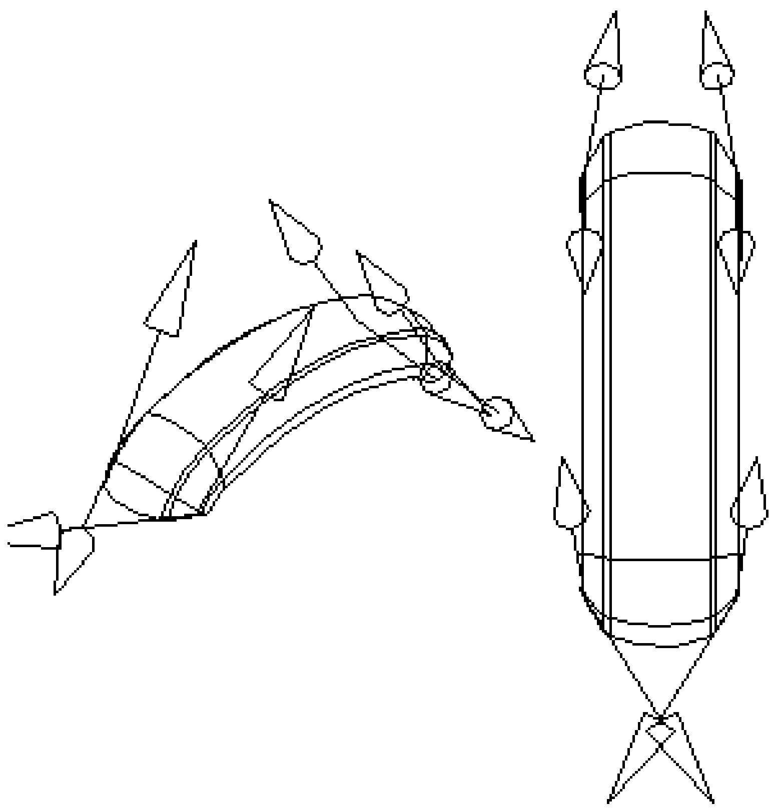
Step 5 Define the Cut Direction.

You can redefine the Cut Direction using the Cut Direction option.

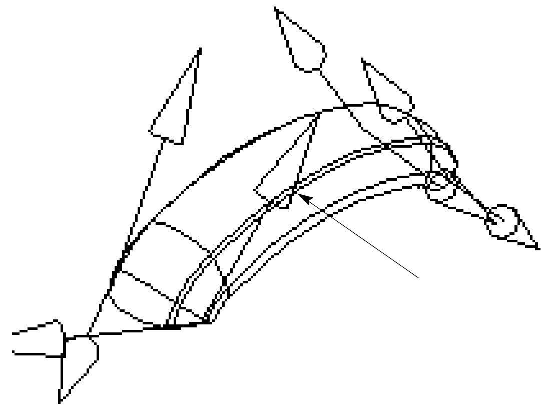
❑ Choose the **Cut Direction** button.

To redefine the Cut Direction, select one of eight displayed vectors which display in pairs at each of the surface corners.

The cone head not only defines the direction but also the quadrant in which the first cut will start.



❑ Choose the cone head as shown below.

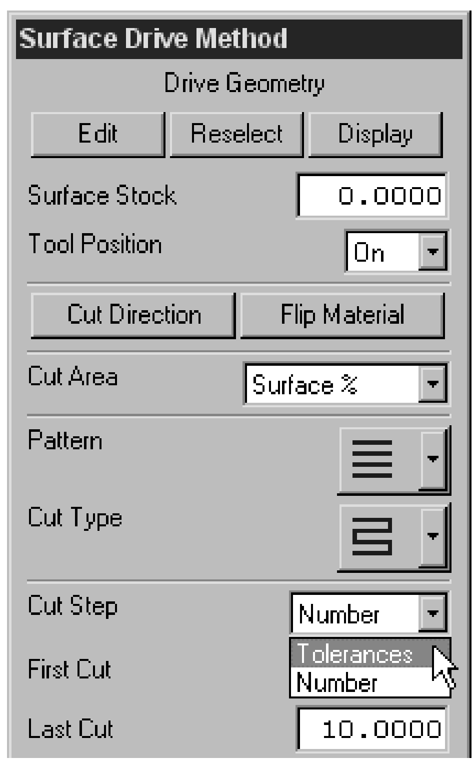


The selected cone head is marked and you are returned to the Surface Drive Method dialog.

Step 6 Define a different type Cut Step.

Cut Step determines the distance between Drive Points in the cut direction.

❑ Next to the Cut Step label, choose **Number**.



The Number option defines a minimum number of Drive Points created along cut passes.

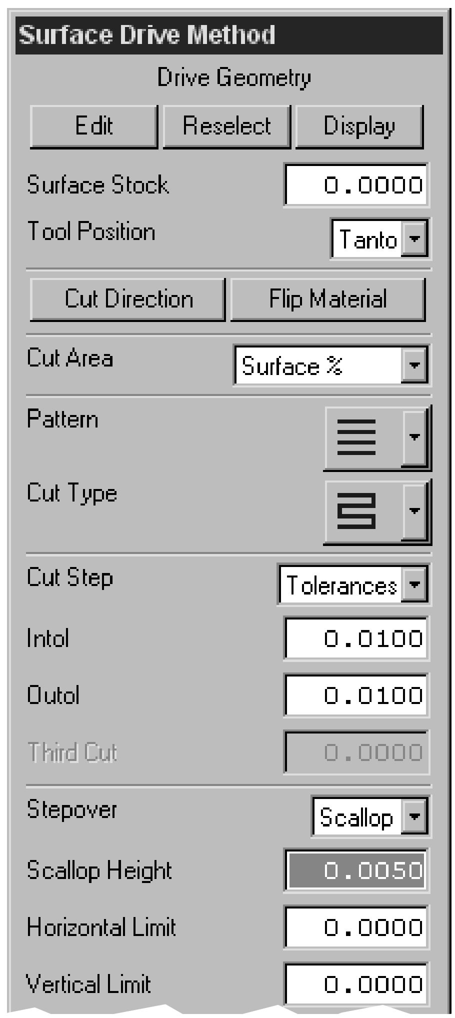
The Tolerances defines an Into and Outol value. You are going to use the Tolerances option.

❑ Choose **Tolerances** and accept the default Intol/Outol settings.

Step 7 Define a different Stepover.

Stepover determines the distance between successive Cut Passes. This distance can be controlled using a scallop height or a total number of Stepovers.

❑ Next to the Stepover label, choose **Scallop**.



The dialog has changed to allow Height, Horizontal, and Vertical Limits to be defined. The Horizontal and Vertical

Limits restrict the distance the tool may move in a direction which is normal to the Projection Vector.

You will now specify the Scallop Height.

❑ Key in **0.2** into the Scallop Height value field.

This will leave a .2" cusp between tool passes. You could use a smaller value but using this large of a value will allow for quicker processing.

❑ Set **0** into the **Horizontal** and **Vertical Limits** value fields.

Step 8 Define the Projection Vector.

The Projection Vector determines how the Drive Points are project to the Part Surface. Surface Area Drive Method provides one additional Projection Vector; Normal to Drive.

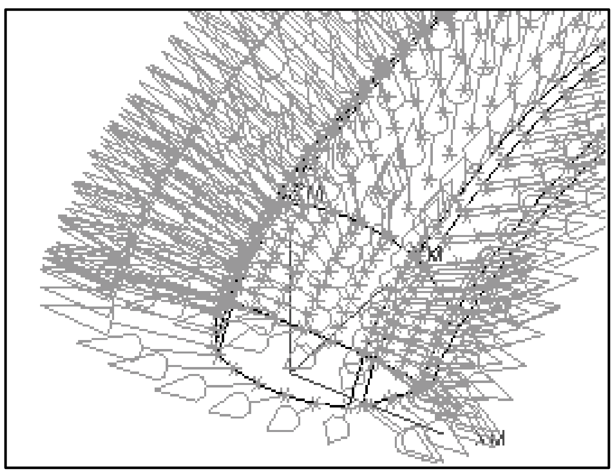
❑ Change the Projection Vector to **Tool Axis.**

Step 9 Display the Contact Points.

This option displays a surface normal vector at each of the Drive Points that has been generated.

❑ Choose **Display Contact Points**.

The Contact Points and its surface normal are displayed along the Drive Surfaces.

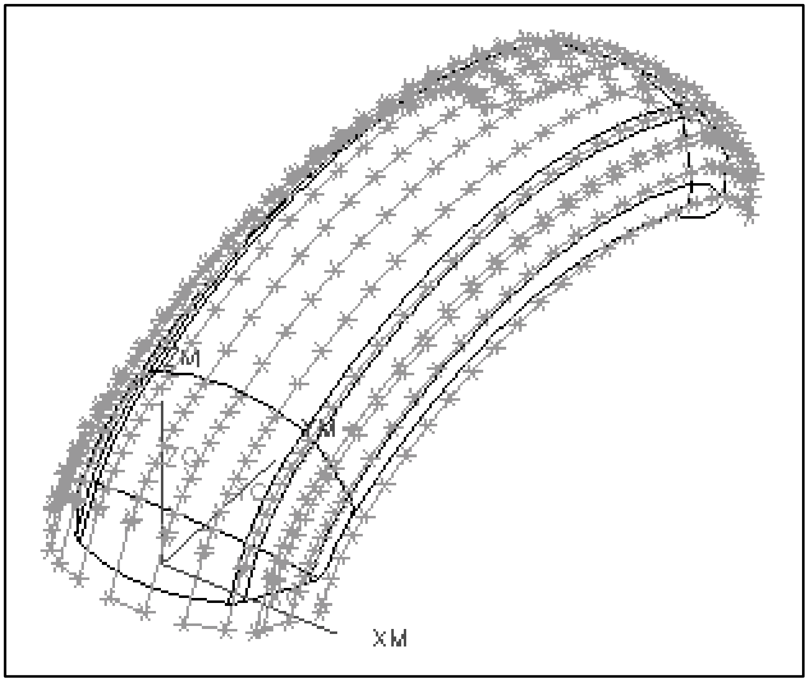


Step 10 Display the Drive Path.

This option displays the Drive Path generated according to the Drive Parameters. It is for visual reference only.

❑ Choose **Display Drive Path**.

The Drive Path is displayed along the Drive Surfaces. Each Drive Point is displayed on the Drive Surfaces.



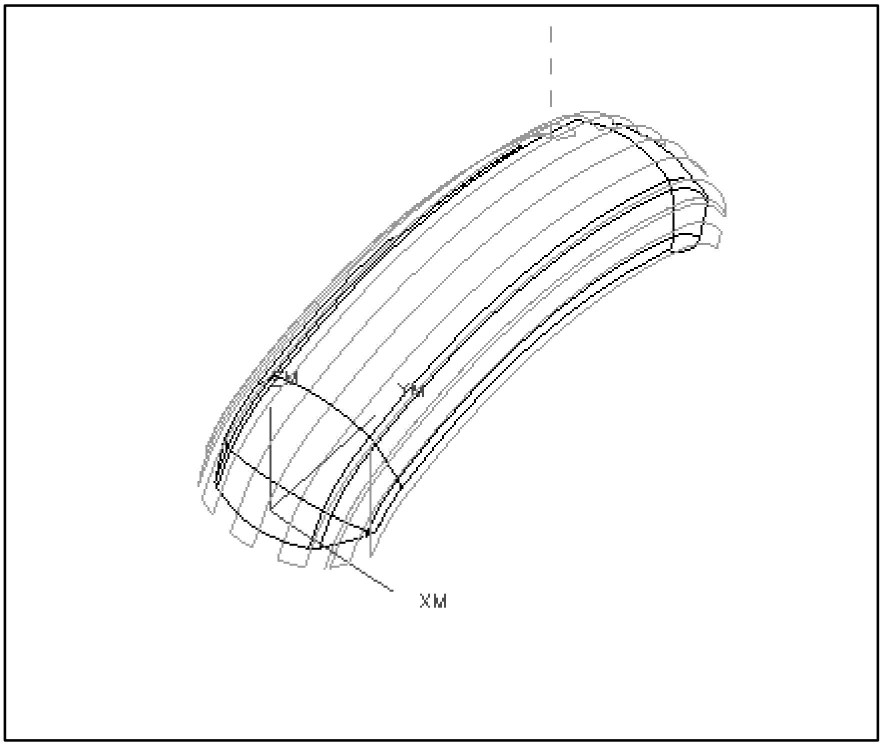
❑ Choose **OK** to return to the Fixed Contour dialog**.**

3

Step 11 Generate the tool path.

❑ Choose the **Generate** icon from the Fixed Contour dialog.

The tool path is generated. It cuts along all the Drive Surfaces that were specified. In the following portion of the activity, you will cut only a portion of the Drive Surfaces.

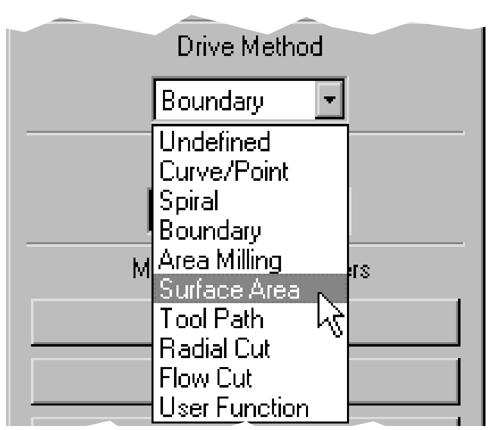


❑ **Reject** the tool path and return to the Fixed Contour dialog.

3

You are going to change the amount of the surface areas which will be cut. This is controlled using the Cut Area option, on the Surface Area Dialog.

❑ In the Drive Method area, choose **Surface Area**.



Step 12 Define the Surface Percentage.

❑ Next to the Cut Area label, choose **Surface %**.

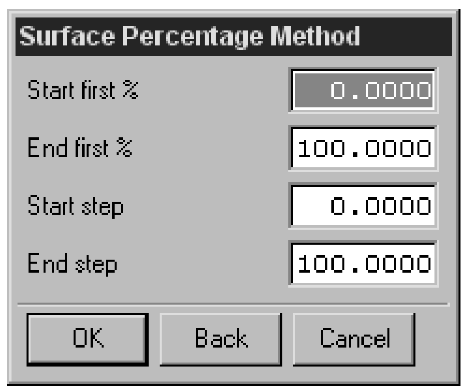
Note that you have two options:

- Surface Percentage
- Diagonal Points

Diagonal Points uses two diagonal points to define the cut area. You are going to use Surface Percentages.

❑ Choose **Surface %**.

The Surface Percentage dialog is displayed.



Start first % – is the location of the first set of Drive Points for the *First* pass.

End first %– is the location of the last set of Drive Points for the *First* pass.

Start step – is the distance along the first Stepover direction.

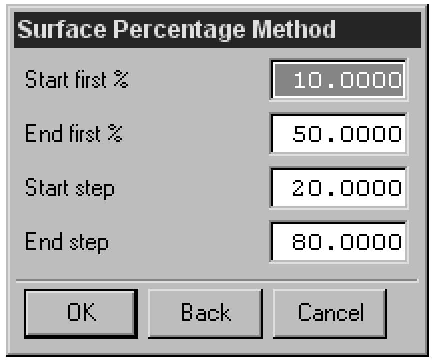
End step – is the distance along the last Stepover direction.

When using only one Drive Surface, the entire surface is 100%. For multiple surfaces, 100% is divided by the number of surfaces in that direction. Then each surface is given that percentage, regardless of size. For example, if you have 5 surfaces, each surface is allotted 20%, regardless of its relative size.

❑ Key in the following values:

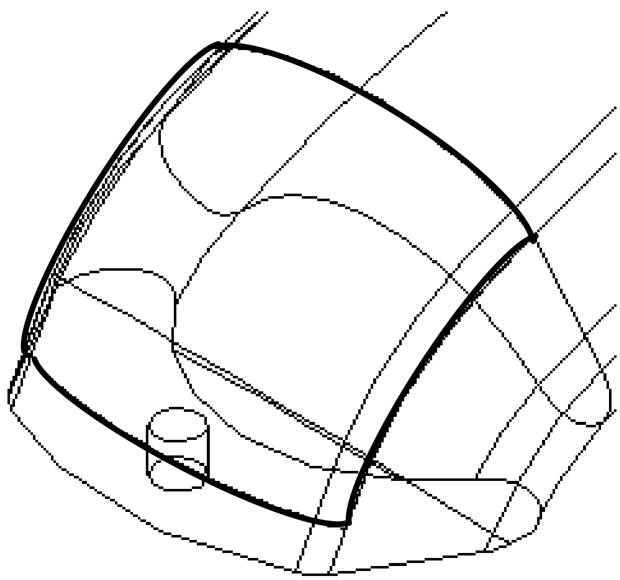
- Start first % **10**

- End first % **50**
- Start step **20**
- End step **80**



❑ Choose **OK**.

The area specified is displayed in the graphics window.



You are ready to generate the tool path. Before you do, display the Drive Path.

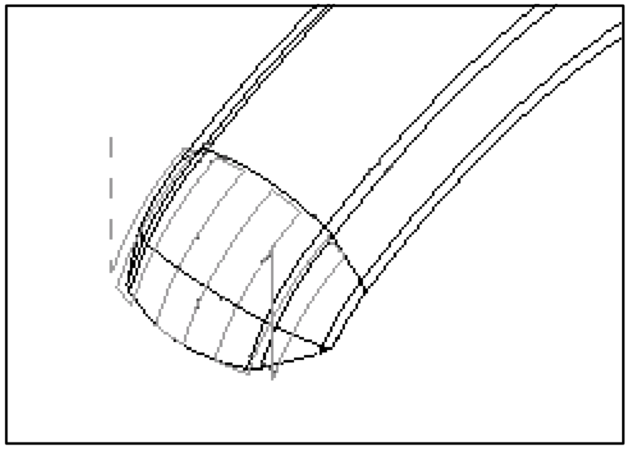
❑ Choose **Display Drive Path**.

The Drive Path is displayed.

❑ Choose **OK**.

Step 13 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.



The tool path is generated cutting only the area specified.

Step 14 Now you will change the surface percentage so that the tool actually leaves the drive surface.

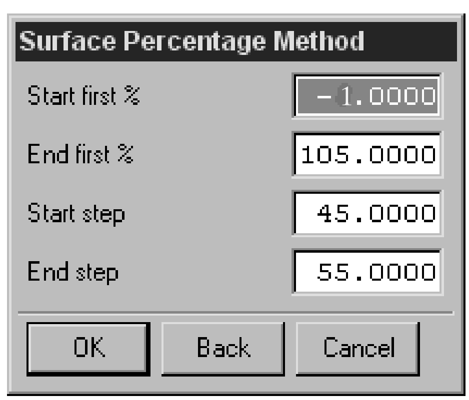
❑ In the Drive Method area, choose **Surface Area**.

❑ Choose **Surface %**.

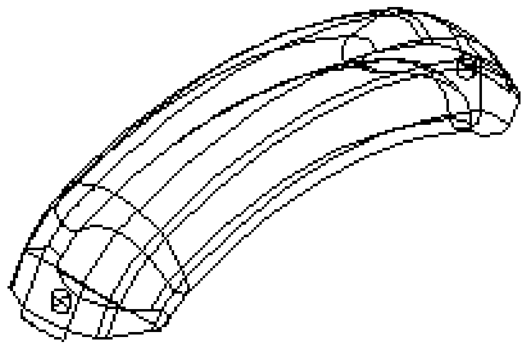
You will now change some of the Surface Percentage values.

❑ Key in the following values:

- Start first % . . . . . **−1**
- End first % **105**
- Start step **45**
- End step **55**



The area specified is displayed in the graphics window.

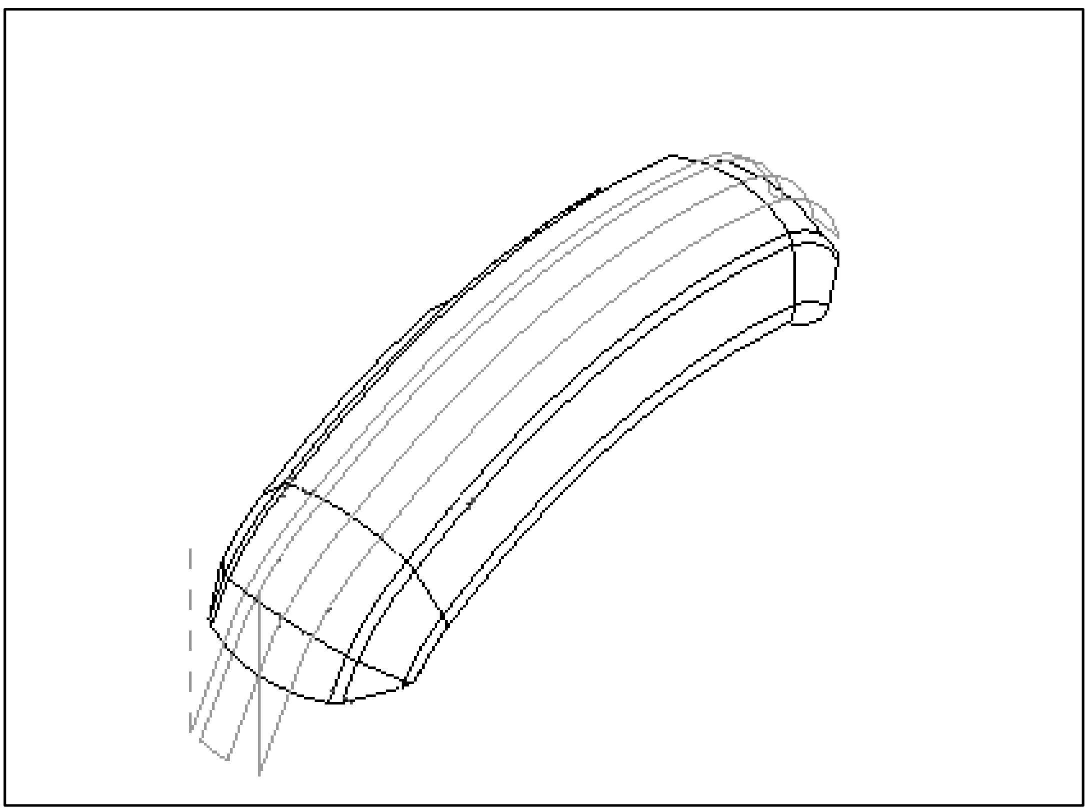


❑ Choose **OK** until you return to the Fixed Contour Dialog.

Step 15 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.

The tool path travels beyond the Drive Surfaces.



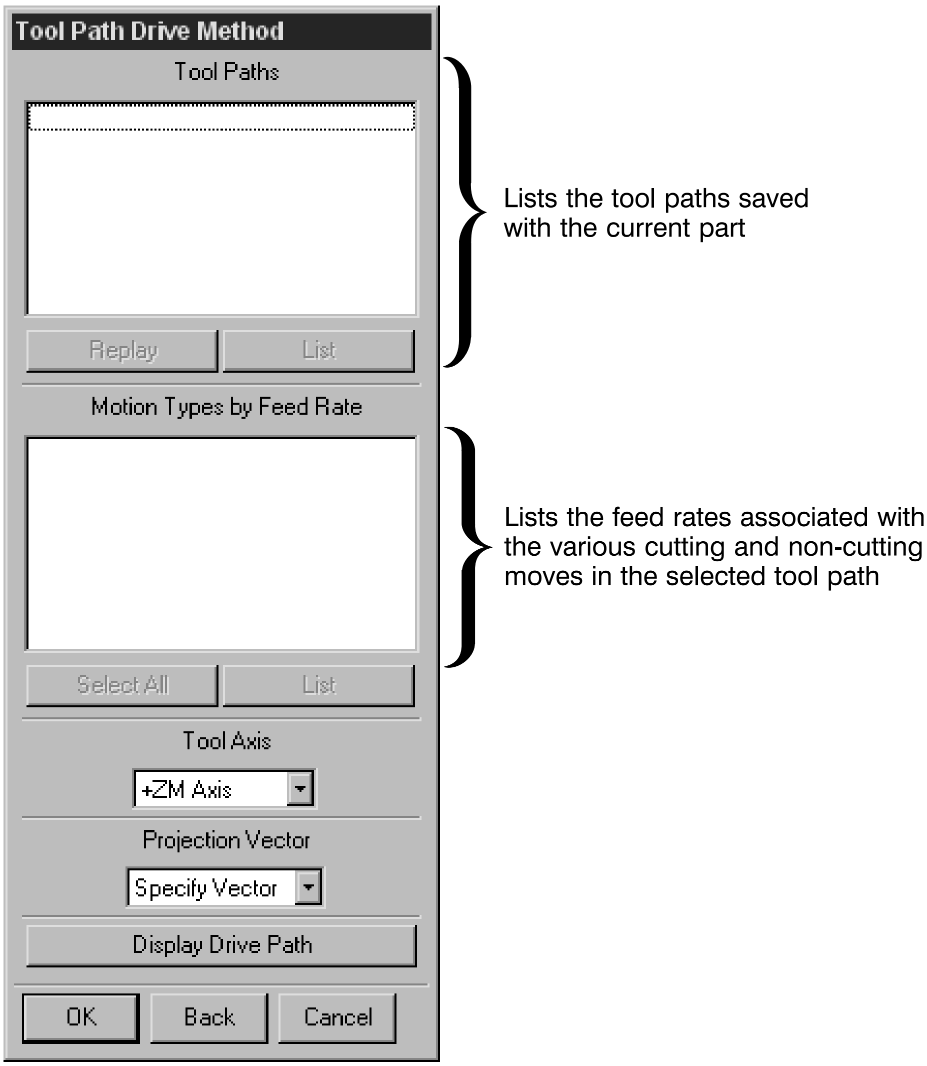
❑ Choose **OK** to save the operation.

❑ **Save** this part file.

You are finished with this activity.

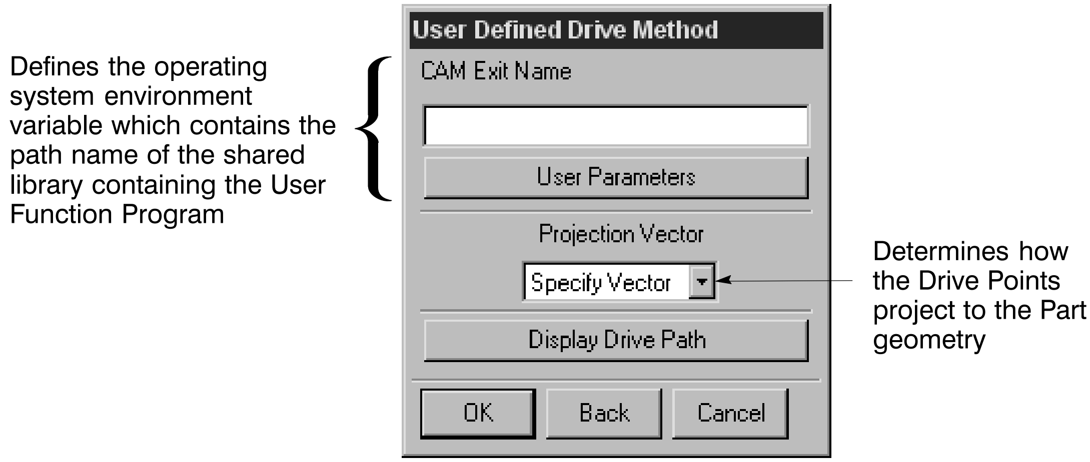
Tool Path Drive Method

This Method allows you to define Drive Points along the tool path of a Cutter Location Source File (CLSF) to create a similar Surface Contouring tool path in the current operation. Drive Points are generated along the existing tool path and projected on to the selected Part Surface(s) creating a new tool path that follows the surface contours. The direction in which the drive points are projected on to the Part Surface(s) is determined by the projection vector.



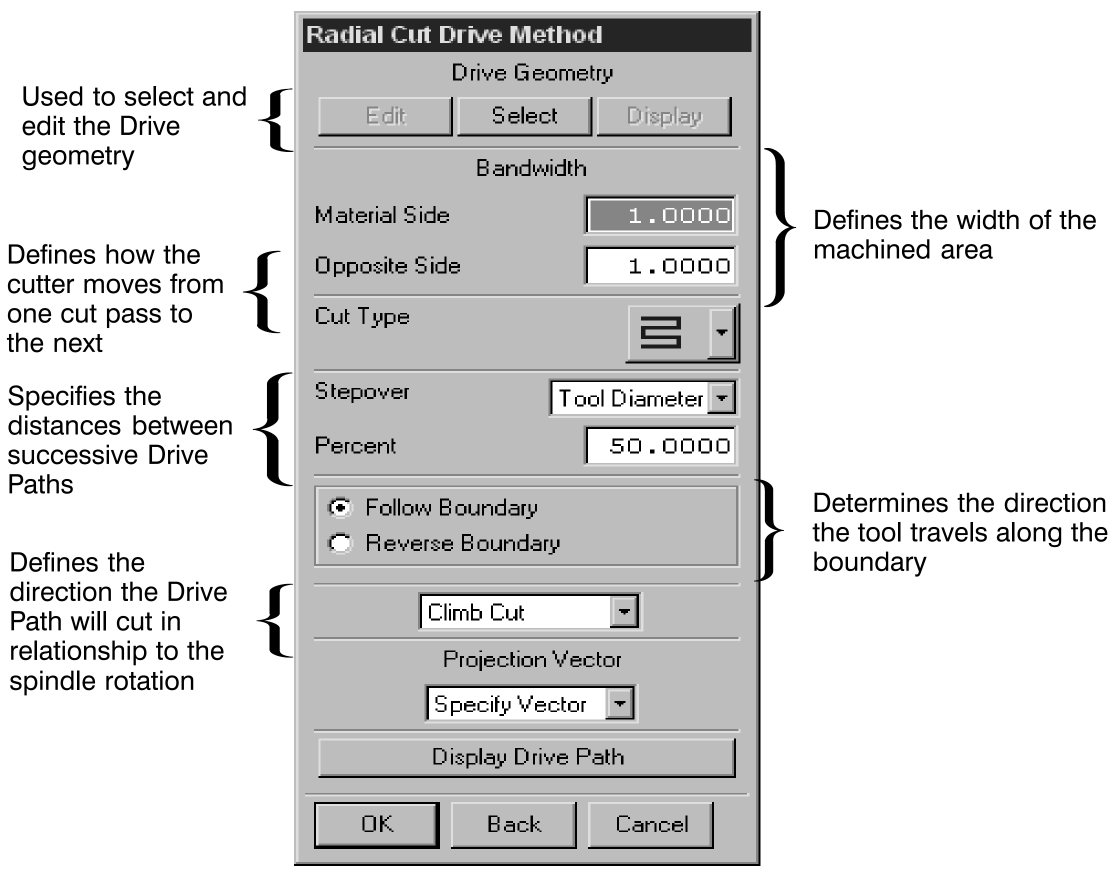
User Function

The User Function Program provides added flexibility by allowing you to use drive methods in the current operation that are created outside of Unigraphics. These are special, optional, user created User Function routines, which are then used to generate tool paths.



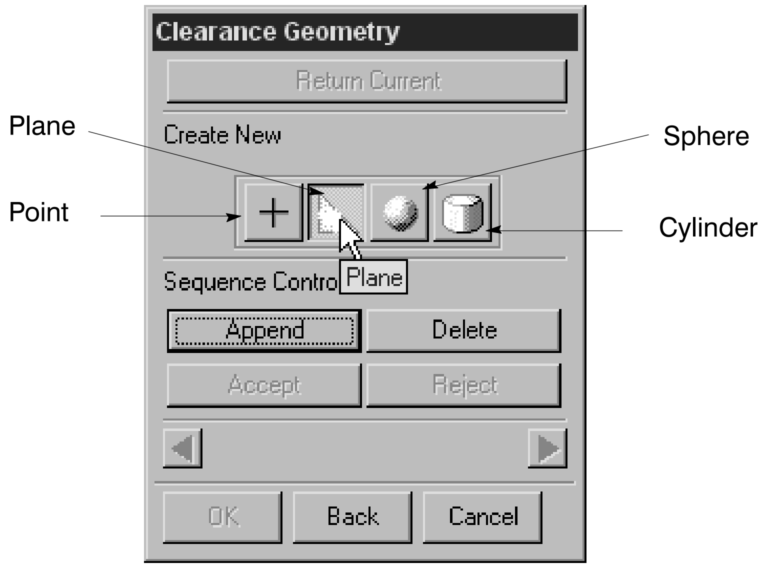
Radial Cut Drive Method

The Radial Cut Drive Method allows you to generate Drive Paths perpendicular to and along a given boundary, using a specified Stepover distance, Bandwidth and Cut Type. This method is useful in creating cleanup operations.



Non-Cutting Moves - Clearance Geometry

On the Clearance Geometry dialog there are four types of geometry that you can use as clearance geometry:



The Sphere and Cylinder options are useful for 4 and 5 axis work. You can define a sphere or cylinder to encompass the part including such items as a fixture, rotary table, or machine table to ensure that the tool will clear all objects when positioning from one location to another.

Gouge Check

Gouge Checking assures that the cutter will not violate the INTOL and OUTTOL values or unexpectedly come in contact with other areas of the part.

Gouge Checking pertains to the following:

- Check geometry (Custom Data dialog)
- Check Stock
- Non-cutting moves
- Surface Area Drive Method

The following is a summary of the options found on the Gouge dialog:

Warning – Outputs a warning message to the CLSF. It does not alter the tool path to avoid gouging the Check Geometry.

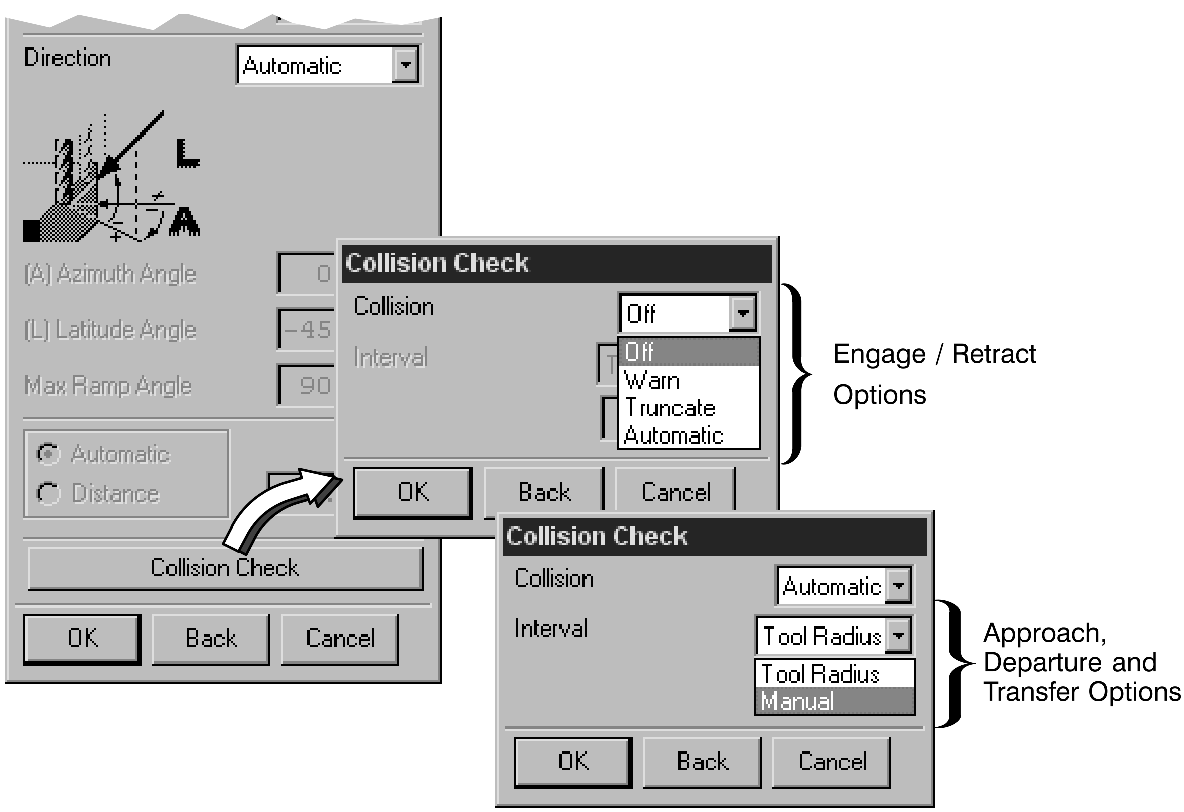
Skip – Replaces the Drive Points that gouge with straight line motion between the last and next non-gouging points.

Retract – The tool will avoid gouging the Check Geometry by using the Check Engage and Retract parameters defined in Non-Cutting moves.

Non-Cutting Moves – Collision Checking

Gouge-Checking for non-cutting moves is referred to as Collision Checking. This option is only available if you specified at least one non-cutting move type (such as an Approach or Engage move).

Non-Cutting Moves Dialog Box



SUMMARY

Fixed Contour operations allows many drive methods for controlling tool motion. The flexibility of different drive methods with the control of numerous cutting parameters allows the cutting of complex contoured geometry.

In this lesson you:

- Applied advanced Fixed Contour operations options for generating cutter paths
- Became familiar with how to select the most appropriate Drive Method and parameters within a Fixed Contour operation

Wire EDM

Lesson 4

PURPOSE

This lesson will show you how to use the fully integrated Wire EDM module. Included are internal and external trim, no core, open profile, optional cutoff and backburn operations.

OBJECTIVES

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

- Create external and internal trim operations
- Create no core and open profile operations

This lesson contains the following activities:

Overview

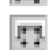
The Wire EDM module provides complete NC/CNC programming functionality for wire EDM machinery. Two and four axis internal and external trim (profile), open profile and two axis no core (pocketing) operations are currently supported.

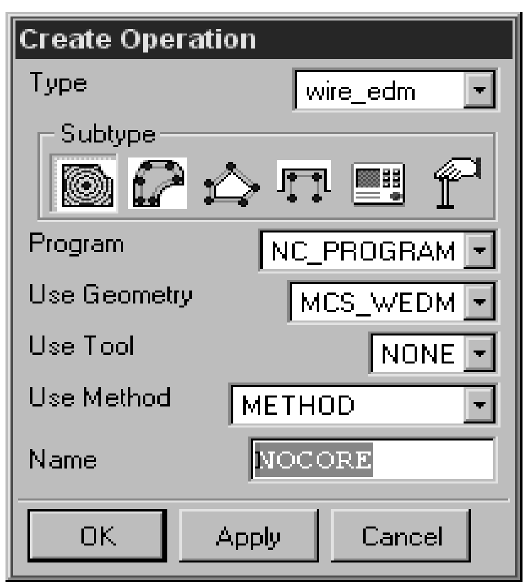
You can also generate cutoff operations applicable for external trim as well as backburn and rough passes that are applicable for internal trim.

Wire EDM also includes support of wireframe and solid models, 2 and 4-axis type machines (Agiecut 100, Agiecut 123, Charmilles, Elox, Fanuc Japax, Mitsubishi, Sodik), full model associativity, corner control and cutter compensation.

Cut Types

Most parameters used in Wire EDM are defined from the Create Operation dialog. You have the following type of cutting operations to choose from under Subtype:

- **No Core** Operations 
- **Internal Trim** operations 
- **External Trim** operations 
- **Open Profile** Operations 



External Trim, **Internal Trim** and **Open Profile** can by used with either 2 and or 4-axis machines. **No Core** can only be used with 2-axis type machines.

No Core

No Core allows the removal of internal part material by use of a concentric cut pattern. By default, the generated path starts from the center of the area to be removed and works its way to the outer most part boundary. You may also specify a start point to control the first cut move.

In cases where large amounts of material have been removed by previous machining operations, such as milling or drilling, multiple inner boundaries may be defined, representing material removed previously by such operations. This eliminates the undesirable effect of cutting air. Inner boundaries may also include the thread or start hole.

The sole purpose of the No Core option is to eliminate the creation of core (commonly referred to as slug) material which could damage the part or break the wire.

Note that the default condition for wire position is TANTO.

Activity 4—1: No Core Wire EDM Operations

In this activity, you will use the No Core operation type to remove material without leaving a material slug. After completing the first part of the activity, you will be asked to select two holes that were previously created. The smaller hole will be used as the thread hole, the larger represents material which was removed in a preceding operation. This will prevent the wire from making "air" passes in areas where material was previously removed.

Step 1 Open the part file.

- ❑ Open the part file **ama_wire_edm** and then save the part as ***wire_edm**, where *** represents your initials.
- ❑ Enter the **Manufacturing** application.

  The Machining Environment dialog is displayed.

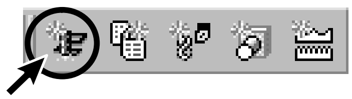
- ❑ Select **wire_edm** from the CAM Session Configuration listing window.
- ❑ Select **wire_edm** from the CAM Setup listing window.

❑ Choose the **Initialize** button.

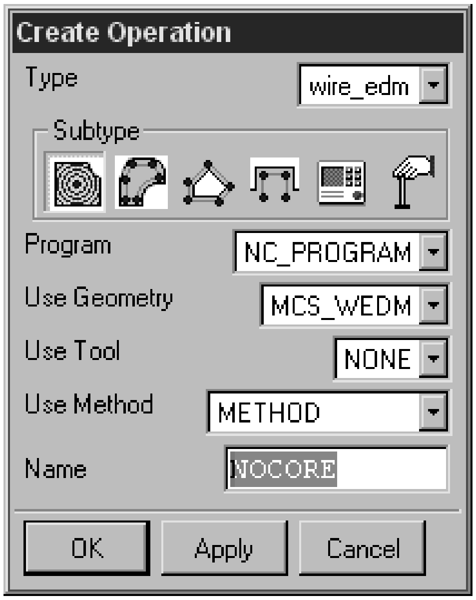
The Operation Navigator is displayed.

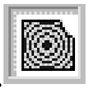
Step 2 Create the No Core Operation.

❑ Select the Create Operation icon from the Create Tool Bar.



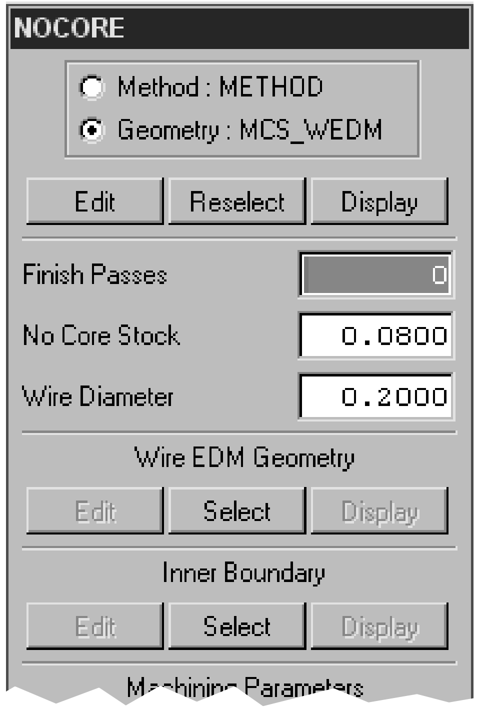
The Create Operation dialog is displayed.



❑ Select the **NOCORE** icon. 

❑ Choose **OK.**

The NOCORE dialog is displayed.

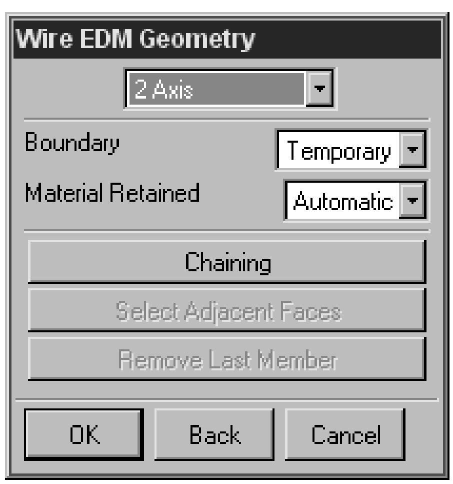


The wire diameter is set to .200 mm which will show a cluttered display when generating a tool path. For purposes of clarity you will change the wire diameter to .5 mm.

❑ Key in **.5** in the Wire Diameter field.

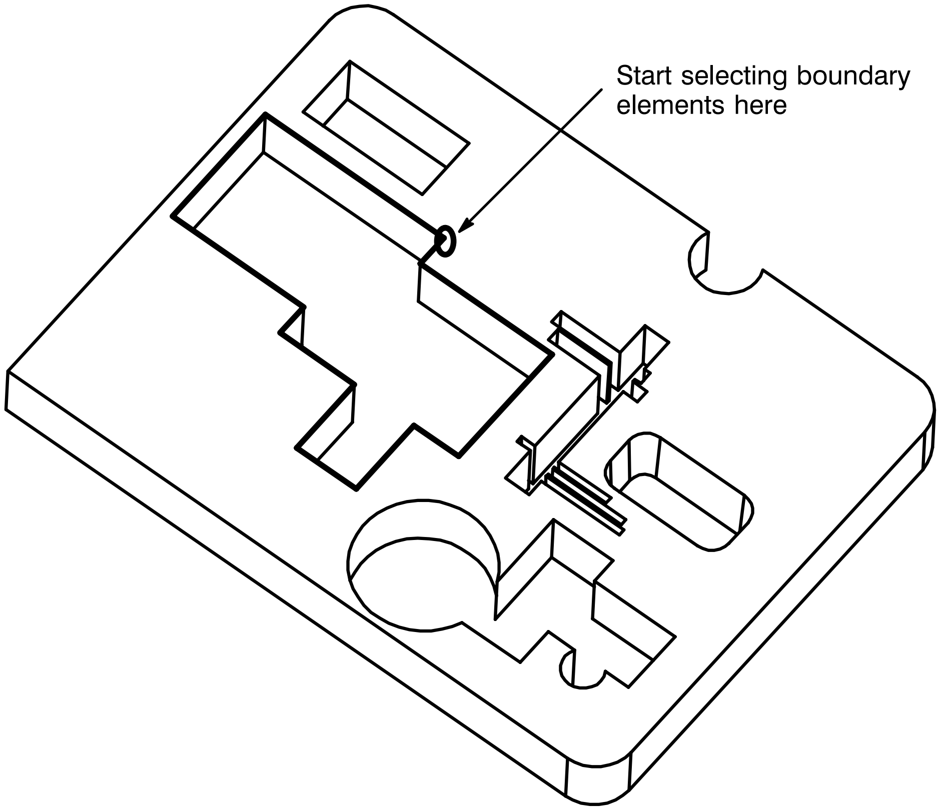
❑ Under the **Wire EDM Geometry label**, choose **Select**.

The Wire EDM Geometry dialog is displayed.



You will now select the boundary elements for the area that will be cut with the wire.

❑ Select the bold boundary elements, counterclockwise, starting as shown in the following diagram.



❑ Choose **OK** to create the boundary and return to the NOCORE dialog.

Step 3 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.

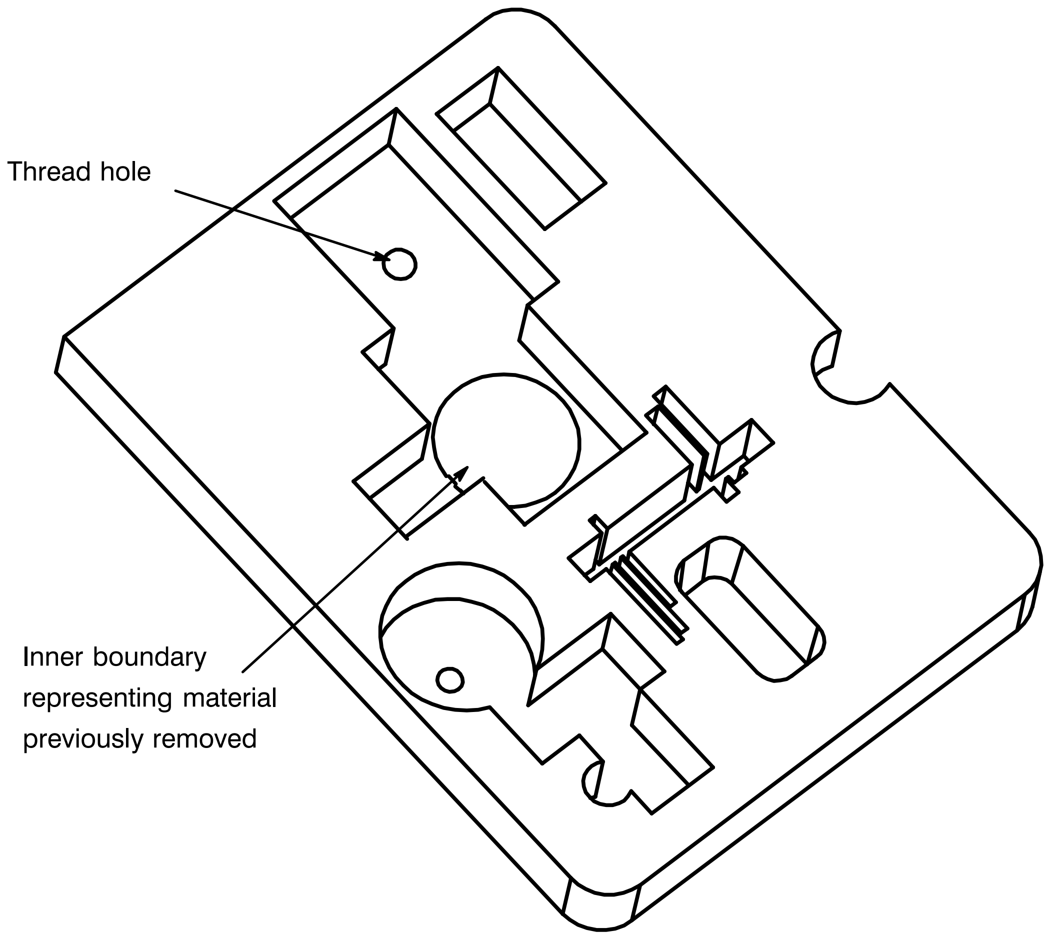
Note the starting point of the wire cut. This is calculated by the processor and may not necessarily be the most ideal condition for this particular part. You will control the start of the wire cutting operation by selecting a thread hole.

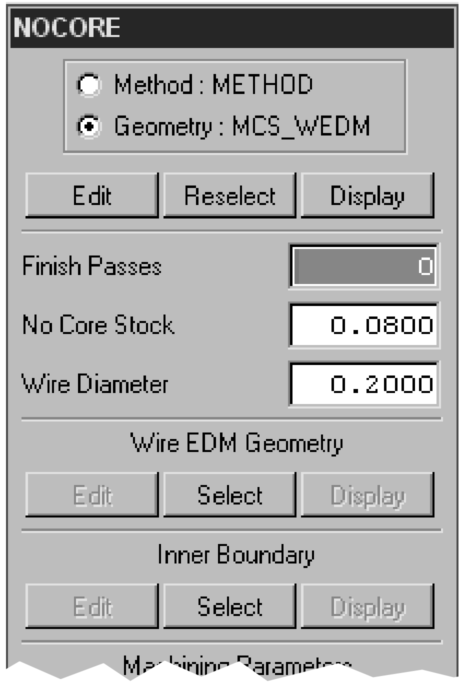
❑ **Reject** the tool path.

You will now edit the operation and select a thread hole and inner boundary which represents material removed from a prior operation.

Step 4 Edit the existing operation, adding a thread hole and inner boundary.

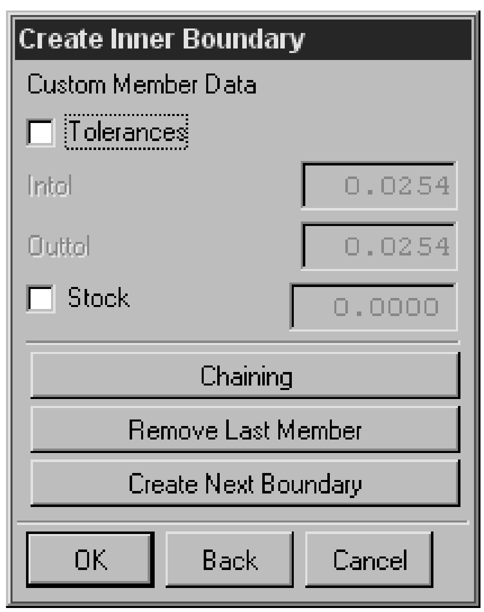
❑ Make layer 10 selectable.





❑ Under the **Inner Boundary label**, choose **Select**.

The Create Inner Boundary dialog is displayed.



❑ Select the thread hole as shown from the previous diagram.

❑ Select the **Create Next Boundary** button from the Create Inner Boundary dialog.

❑ Select the large circle as the inner boundary (as shown in the diagram previously).

❑ Choose **OK** to return to the NOCORE dialog.

Step 5 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.

Note the start position of the wire cut at the thread hole. Also note that the large circle representing material removed from a previous operation contains no tool motions.

❑ Accept the operation and **Save** the part file.

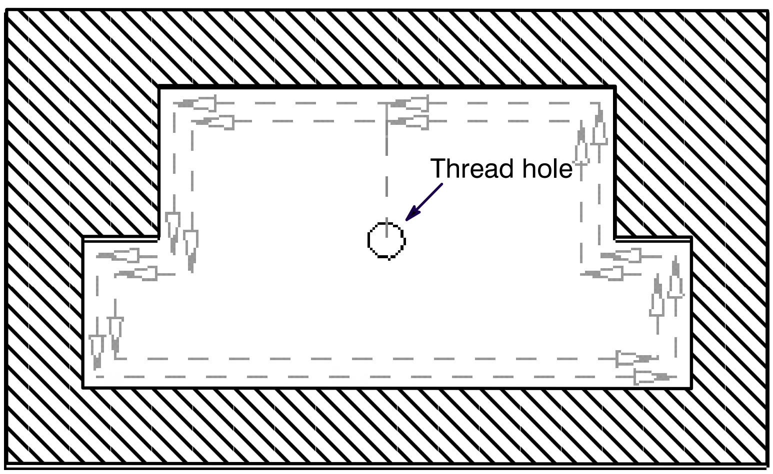
You have finished this activity.

Internal Trim

Internal trim allows you to generate a profile cut or series of cuts on the inside of a boundary. Internal trim also allows for separate sub-operations of **rough pass**, **backburn** and **finish** through the selection and or creation of geometry from the Geometry Parent Group.

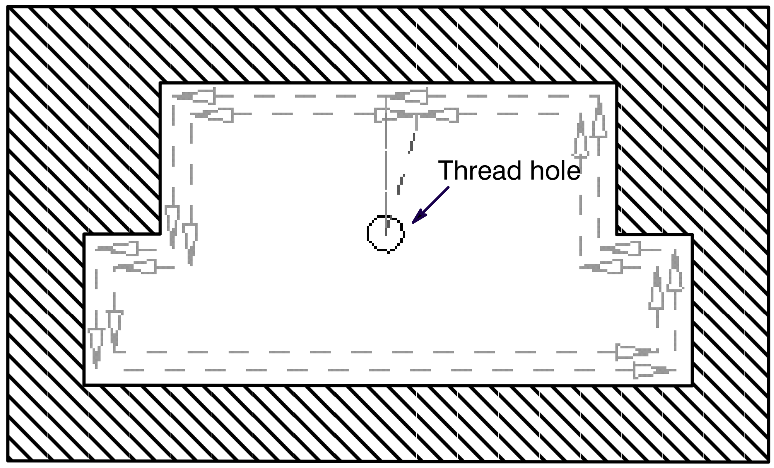
Note that by default, the wire position is always “on” the boundary. In order to create a “tanto” condition, the operation is created with the default “on”, edited and then changed to “tanto”. Also note that internal trim functions with closed boundaries only. If you do not select boundary elements to close the boundary, the closed boundary will automatically be created between the first and last boundary elements selected.

Rough pass will cut the majority of material encompassed by the boundary and leaves a tab that prevents the material (or slug) from dropping through the part.



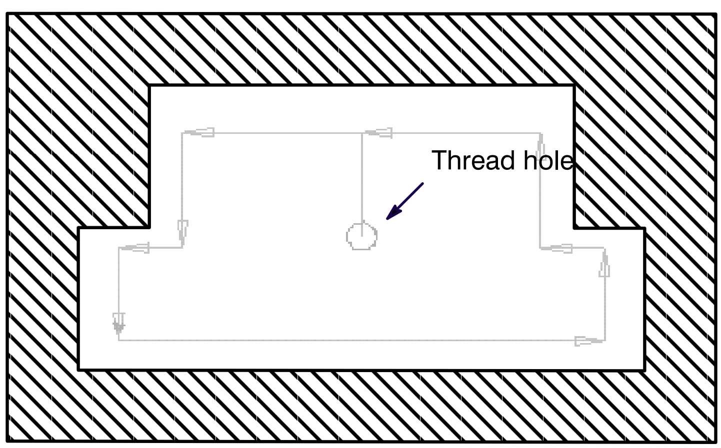
Internal trim (1 rough pass, 1 finish pass, backburn distance .100, wire diameter .100 for clarity only)

Backburn cuts through the tab, created by the rough pass, to allow the remaining material to fall through the part.



Internal trim multiple backburn passes (backburn distance .100, wire diameter .100 for clarity only)

Trim is the main profiling pass or passes.



Internal trim single pass, wire diameter .100 for clarity only)

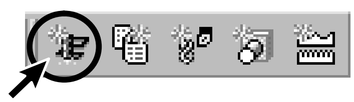
Activity 4—2: Creating Internal Trim Operations

In this activity, you will create various internal trim operations which allows for profile cuts on the inside of boundaries.

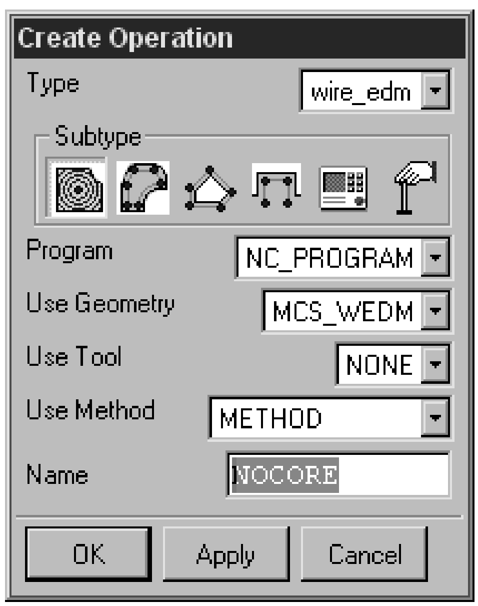
Step 1 Create an Internal Trim Operation.

- ❑ Continue with ***wire_edm** from the previous activity.
- ❑ If necessary enter the **Manufacturing** application.

  The Operation Navigator is displayed.
- ❑ Select the Create Operation icon from the Create Tool Bar.



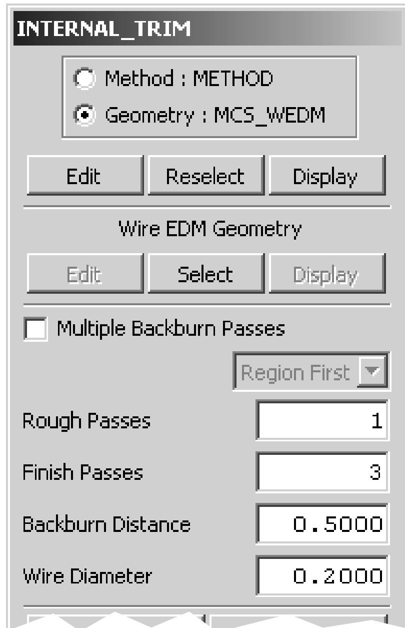
The Create Operation dialog is displayed.



- ❑ Select the **Internal Trim** icon. 

❑ Choose **OK.**

The Internal Trim dialog is displayed.

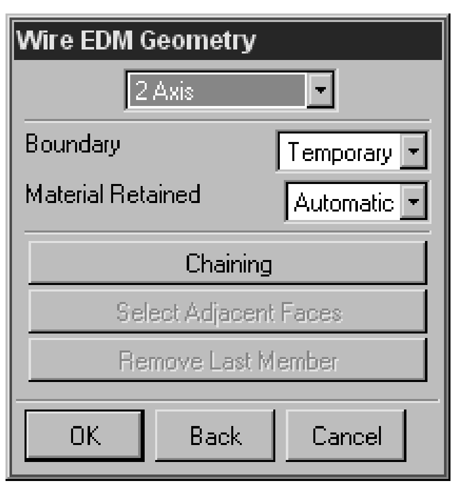


The wire diameter is set to .200 mm which will show a cluttered display when generating a tool path. For purposes of clarity you will change the wire diameter to .5 mm. Also note that the default for number of rough passes is 1 and for finish passes 3. You will accept these defaults.

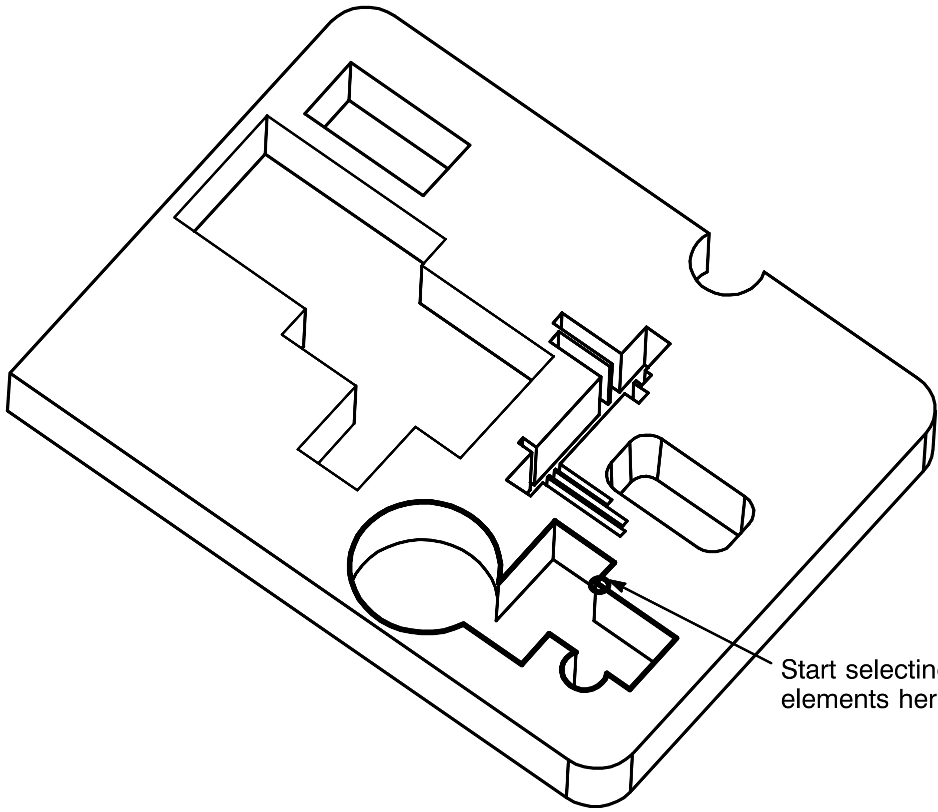
❑ Key in **.5** in the Wire Diameter field.

❑ Under the **Wire EDM Geometry label**, choose **Select**.

The Wire EDM Geometry dialog is displayed.



You will now select the boundary elements for the area that will be cut with the wire.



❑ Choose **OK** to create the boundary and return to the Internal Trim dialog.

Step 2 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.

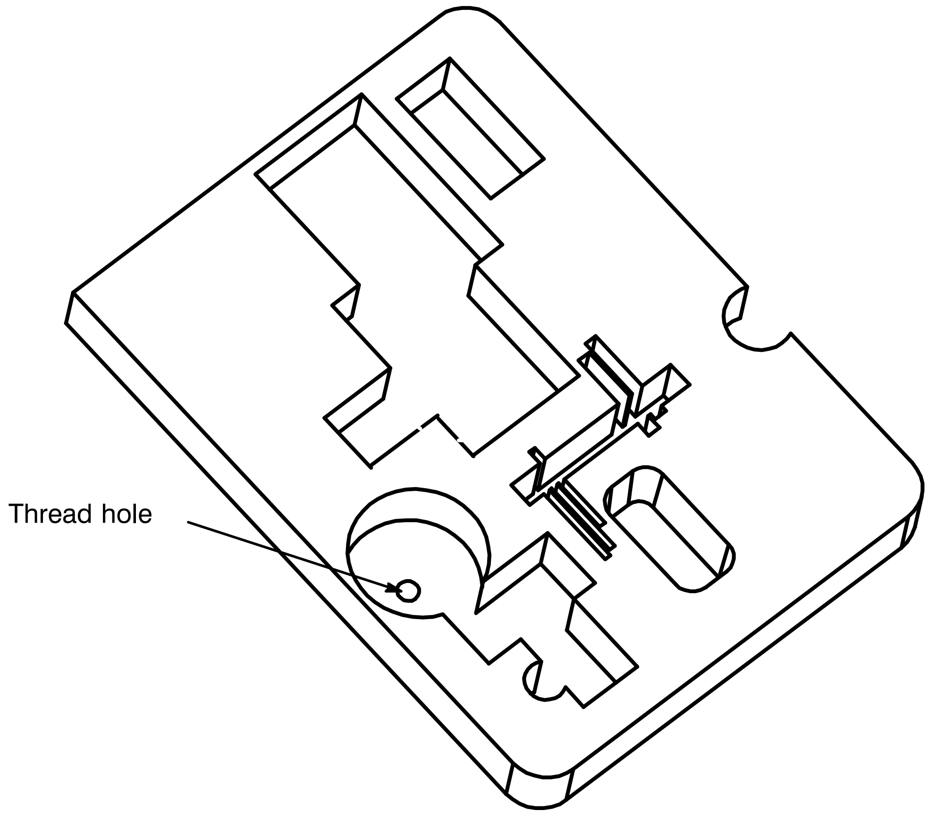
Note the starting point of the wire cut. The starting point is calculated by the processor and may not necessarily be the most optimal point to start. You will now select a thread hole to control the starting point of the cutting operation. You will use a different method of selection than you did in the previous activity involving the **No Core** operation type.

❑ **Reject** the tool path

You will now edit the operation and select a thread hole to control the starting point of the cutting operation.

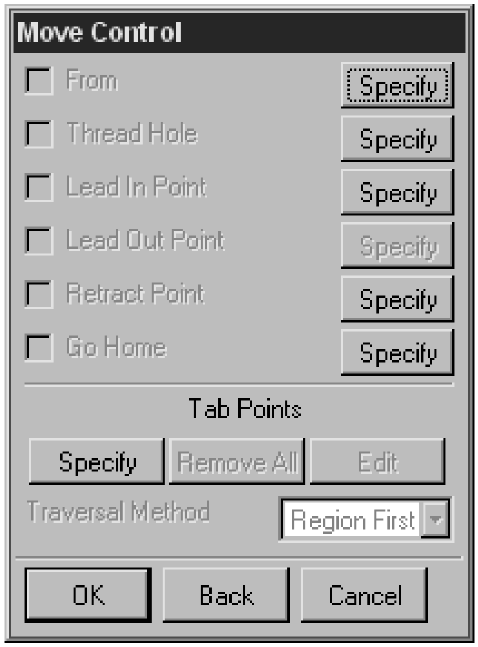
Step 3 Edit the existing operation, adding a thread hole.

❑ If necessary, make layer 10 selectable.



❑ Choose **Move**.

The Move Control dialog is displayed.

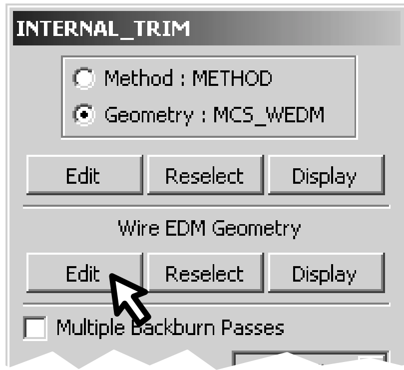


- ❑ Select **Specify** (next to **Thread Hole**).
- ❑ Select the thread hole (as shown in the diagram previously).
- ❑ Choose **OK** until you return to the Internal Trim dialog.

Step 4 Create the tool path.

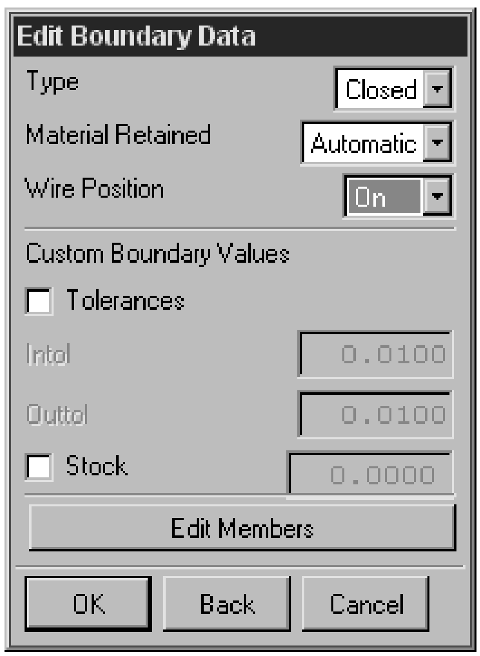
- ❑ Choose the **Generate** icon and generate the tool path.

Do not accept the tool path. Note the start position of the wire cut at the thread hole. Also note the tool path. Remember that the wire position by default is “on”. The rough and finish passes are visually on top of one another. For purposes of clarity you will change the wire position from “on” to “tanto”. You will now edit the operation and change the wire position to “tanto”.



❑ Under the Wire EDM Geometry label, select **Edit**.

The Edit Boundary Data dialog is displayed.



❑ Change the Wire Position from **On** to **Tanto**.

❑ Choose **OK**.

❑ Choose the **Generate** icon and generate the tool path.

Note the rough and finish passes and the start of the wire cut.

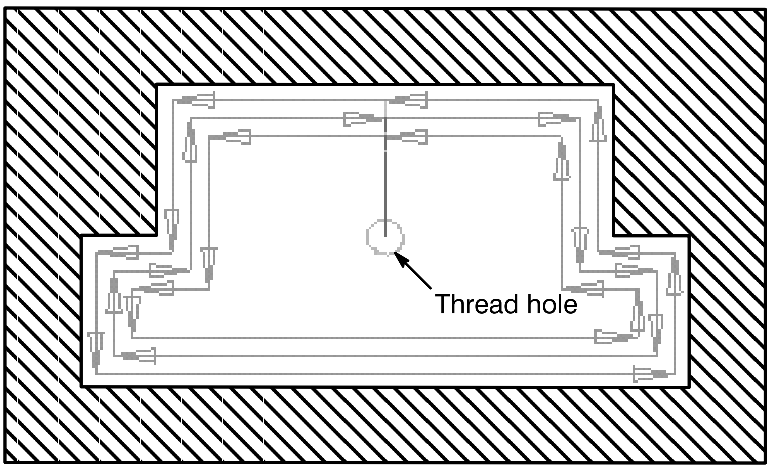
❑ Accept the operation and save the part file.

You have completed this activity.

Internal Trim Sub-operations

You have just completed an activity which familiarizes you with the various options available for creating internal trim operations. Internal trim also allows for the creation of separate sub-operations of **rough**, **backburn** and **finish** by the creation of a sequence of internal trim operations through the Geometry Parent Group.

Selection of the Geometry Parent Group allows for the creation of a sequence of operations that will perform a rough pass, backburn operation and finish pass.



Internal trim sequence of operations including rough pass, backburn operation and finish pass (wire diameter .100 for clarity only)

The following activity will familiarize you with the procedure of creating a sequence of internal trim sub-operations.

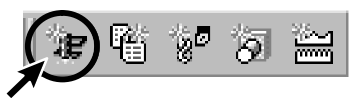
Activity 4–3: Creating a Sequence of Sub-Operations

In this activity, you will create a Sequence of internal trim sub-operations and then generate subsequent tool paths.

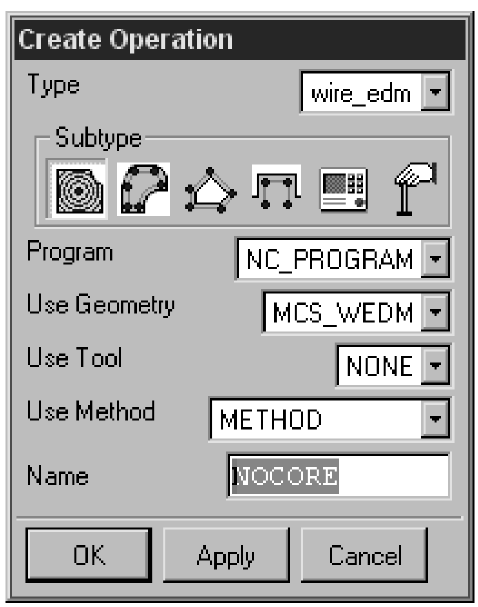
Step 1 Create an Internal Trim sequence of sub-operations.

- Continue with *****wire_edm** from the previous activity.
- If necessary enter the **Manufacturing** application.

  The Operation Navigator is displayed.
- Select the Create Operation icon from the Create Tool Bar.



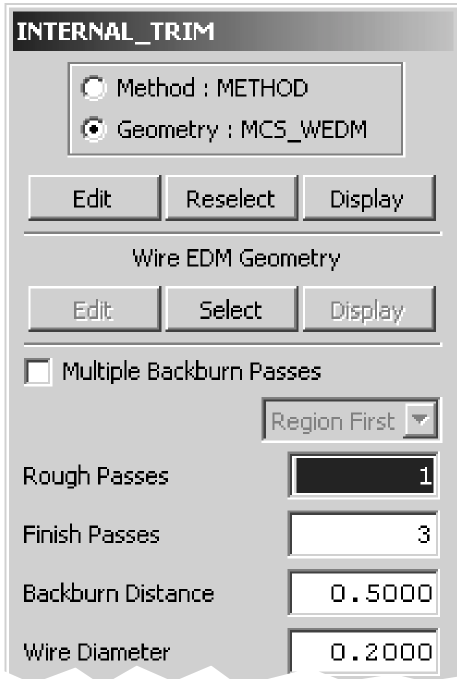
The Create Operation dialog is displayed.



- Select the **Internal Trim** icon. 

❑ Choose **OK.**

The Internal Trim dialog is displayed.

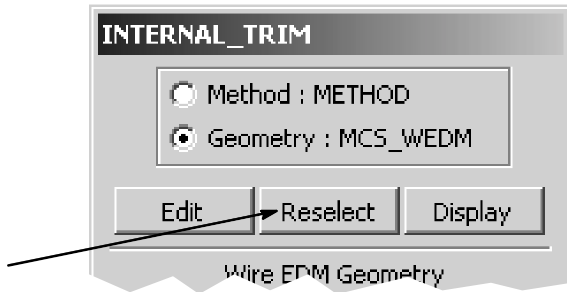


The wire diameter is set to .200 mm which will show a cluttered display when generating a tool path. For purposes of clarity you will change the wire diameter to .5 mm. Also note that the default for number of rough passes is 1 and for finish passes 3. You will accept those defaults.

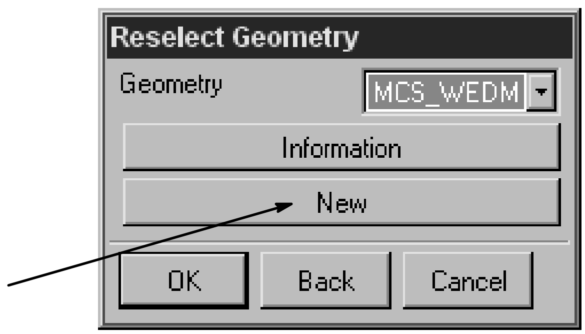
❑ Key in **.5** in the Wire Diameter field.

You will now create the Geometry Parent Group necessary for the internal trim sequence.

❑ Under the **Geometry: MCS_WEDM** label, choose **Reselect**.

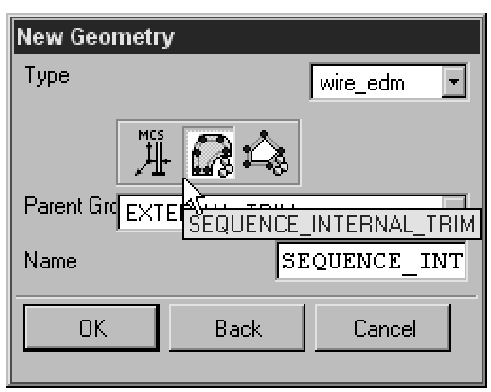


The Reselect Geometry dialog is displayed.



❑ Choose, **New**.

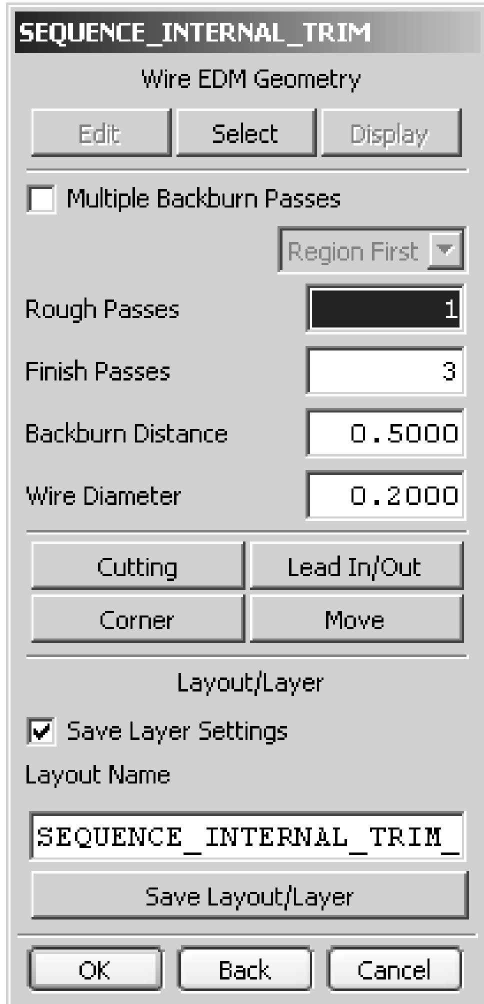
The New Geometry dialog is displayed.



❑ Select the **SEQUENCE_INTERNAL_TRIM** icon.

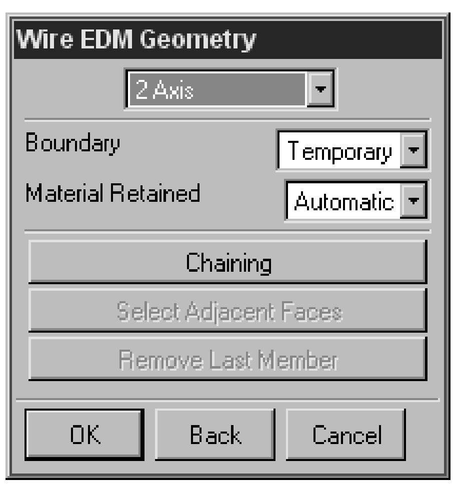
❑ Choose, **OK**.

The SEQUENCE_INTERNAL_TRIM dialog is displayed.

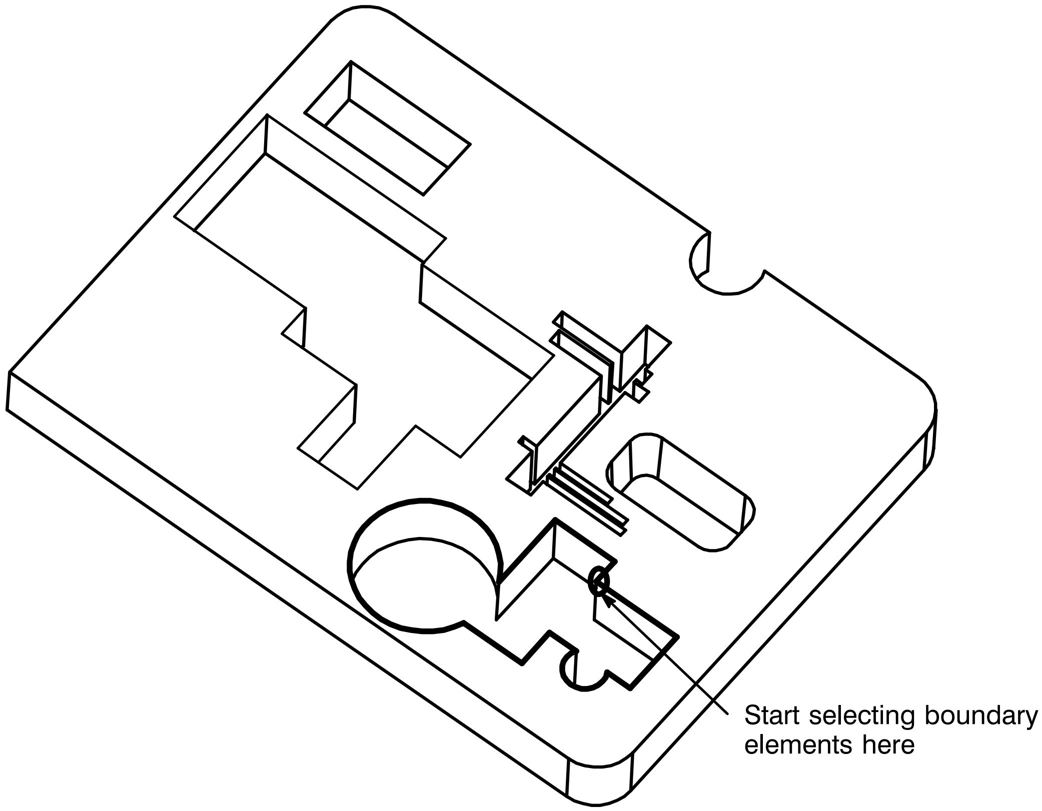


❑ Under the **Wire EDM Geometry** label, choose **Select**.

The Wire EDM Geometry dialog is displayed.



You will now select the boundary elements for the area that will be cut with the wire.



❑ Choose **OK** to create the boundary and return to the **SEQUENCE_INTERNAL_TRIM** dialog.

The wire diameter is set to .200 mm which will show a cluttered display when generating a tool path. You will change the wire diameter to .5 mm for purposes of clarity.

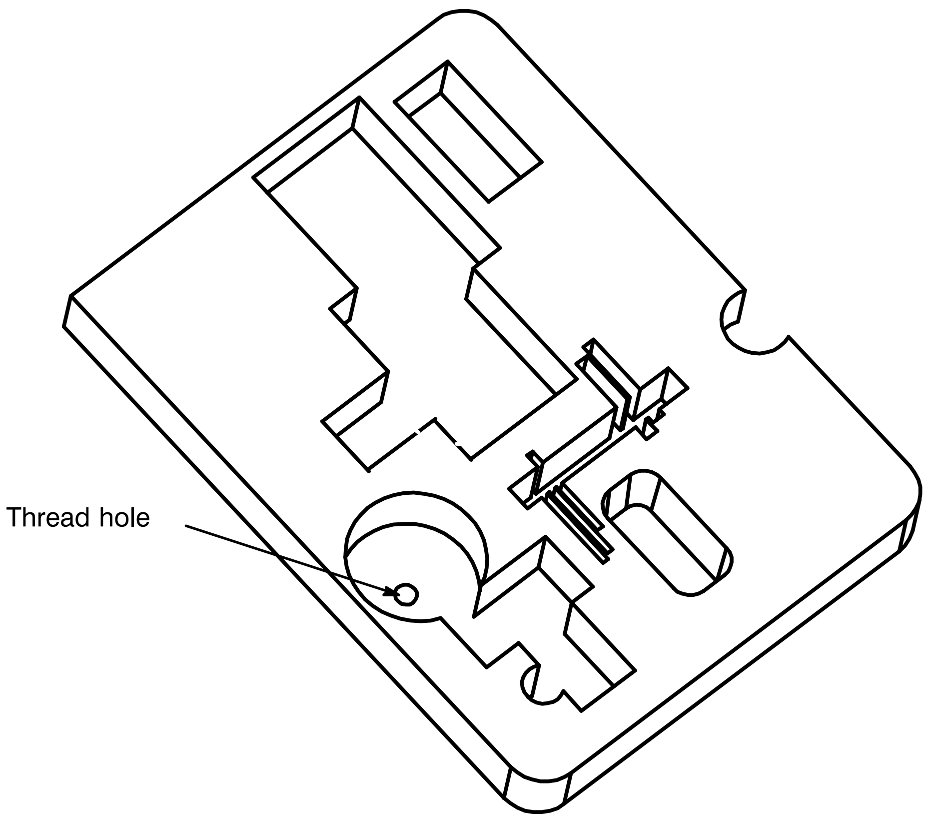
4

❑ Key in **.5** in the Wire Diameter field.

❑ Choose, **OK**.

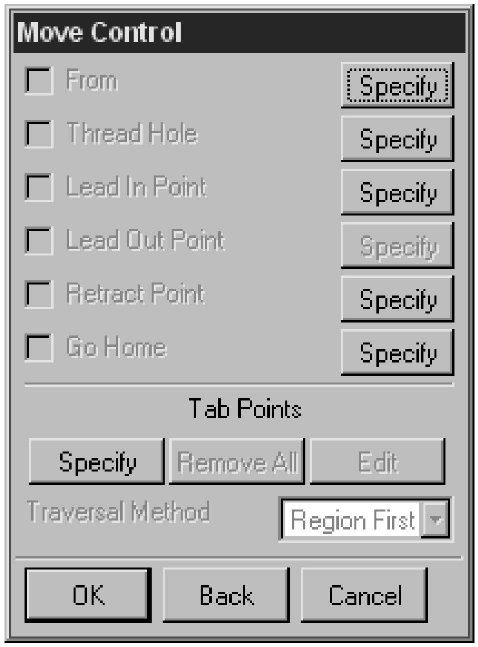
You will now select a thread hole to control the starting point of the cutting operation.

❑ If necessary, make layer 10 selectable.



❑ Choose **Move**.

The Move Control dialog is displayed.



❑ Select **Specify** (next to **Thread Hole**).

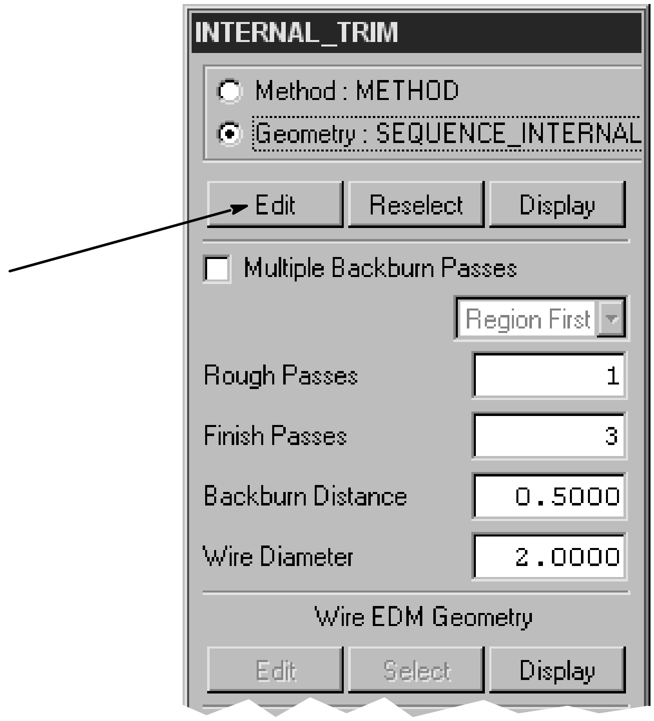
❑ Select the thread hole (as shown in the diagram previously).

❑ Choose **OK** until you return to the Internal Trim dialog.

Step 2 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.

Do not accept the tool path. Note the start position of the wire cut at the thread hole and the tool path. Remember that the wire position by default is “on”. The rough and finish passes are visually on top of one another. For purposes of clarity you will change the wire position from “on” to “tanto”. You will now edit the operation and change the wire position to “tanto”.

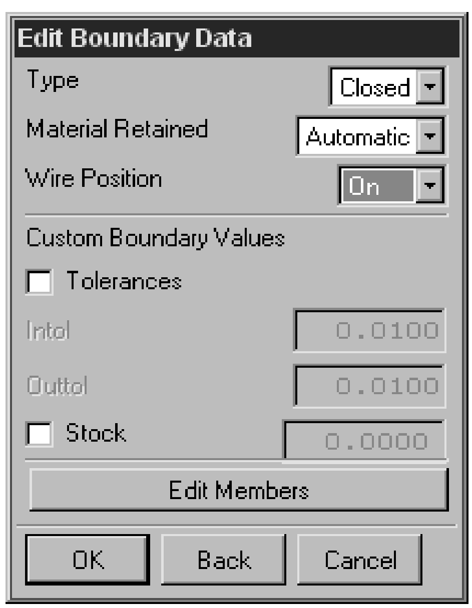


❑ Under the Geometry: **SEQUENCE_INTERNAL_TRIM** label, select **Edi**t.

The SEQUENCE_INTERNAL_TRIM dialog is displayed (ignore any warning messages that may appear).

❑ Under the Wire EDM Geometry label, select **Edit**.

The Edit Boundary Data dialog is displayed.



- ❑ Change the Wire Position from **On** to **Tanto**.

- ❑ Choose **OK** until you return to the INTERNAL_TRIM dialog.

- ❑ Choose the **Generate** icon and generate the tool path.

  Note the rough and finish passes and the start of the wire cut.

- ❑ Accept the operation and **Save** the part file.

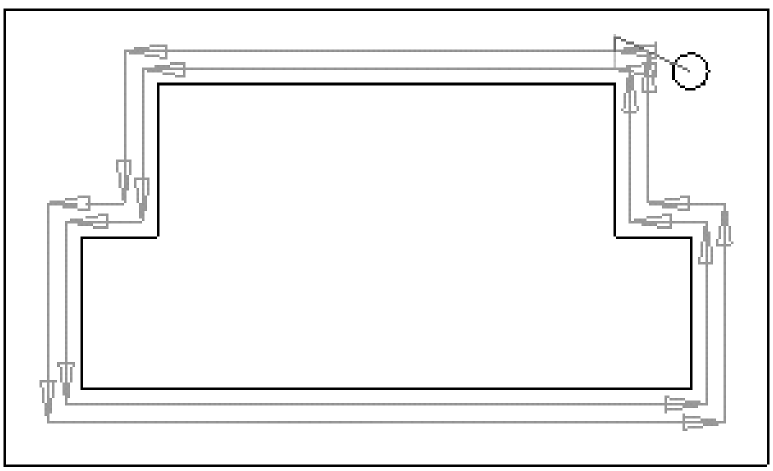
  You have completed the activity.

External Trim

External trim allows you to generate a profile cut or series of cuts around the outside of a boundary. External trim allows for separate sub-operations of **trim** and **cutoff** through the selection and or creation of geometry from the Geometry Parent Group.

Note that by default, the wire position is always “on” the boundary. In order to create a “tanto” condition, the operation is created with the default “on”, edited and then changed to “tanto”. Also note that external trim functions with closed boundaries only. If you do not select boundary elements which create a closed condition, the closed boundary will automatically be created between the first and last boundary elements selected.

Rough pass will cut the majority of material encompassed by the boundary and leaves a tab that prevents the material (or slug) from dropping through the part.



External trim (1 rough pass, 1 finish pass, cutoff distance .100, wire diameter .100 for clarity only)

Activity 4—4: Creating External Trim Operations

In this activity, you will create external trim operations which creates external profile cuts around the outside of the boundary.

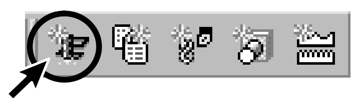
Step 1 Create an External Trim Operation.

❑ Continue with *****wire_edm** from the previous activity.

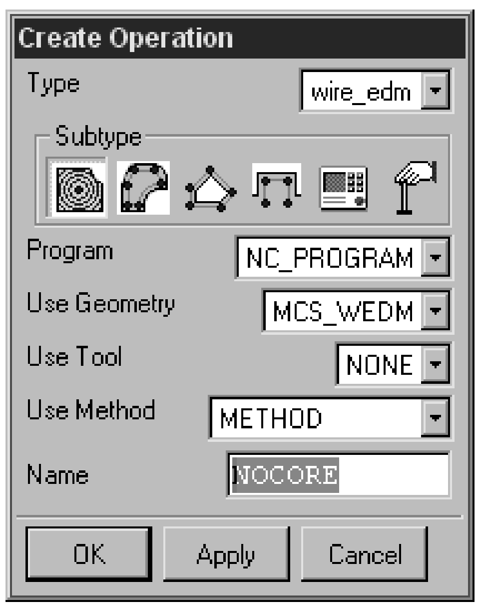
❑ If necessary enter the **Manufacturing** application.

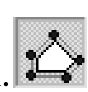
The Operation Navigator is displayed.

❑ Select the Create Operation icon from the Create Tool Bar.



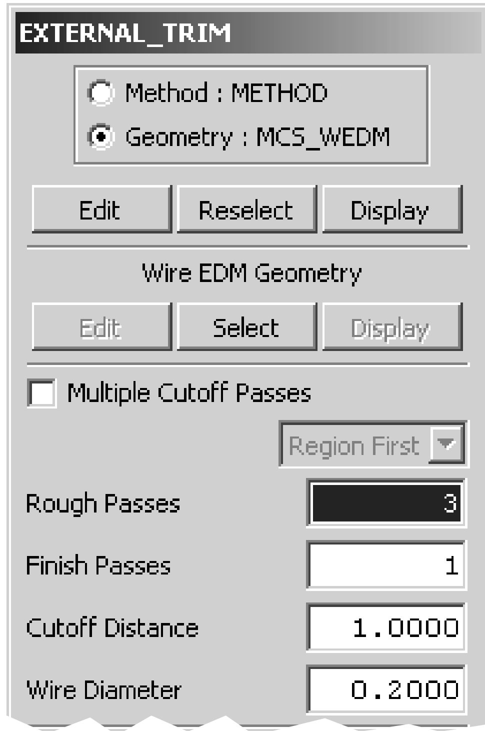
The Create Operation dialog is displayed.



❑ Select the **External Trim** icon. 

❑ Choose **OK.**

The External Trim dialog is displayed.



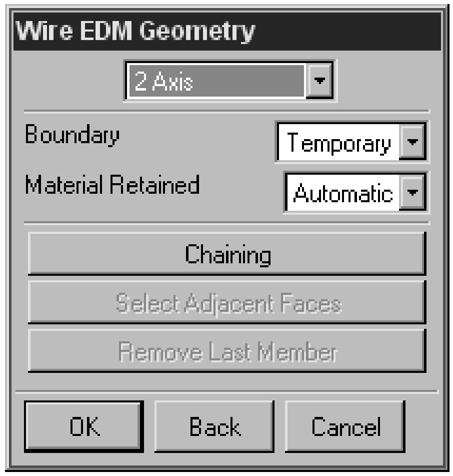
Note that the default for number of rough passes is 3 and for finish passes 1. You will change the number of rough passes from 3 to 1.

❑ Key in **1** in the Rough passes field.

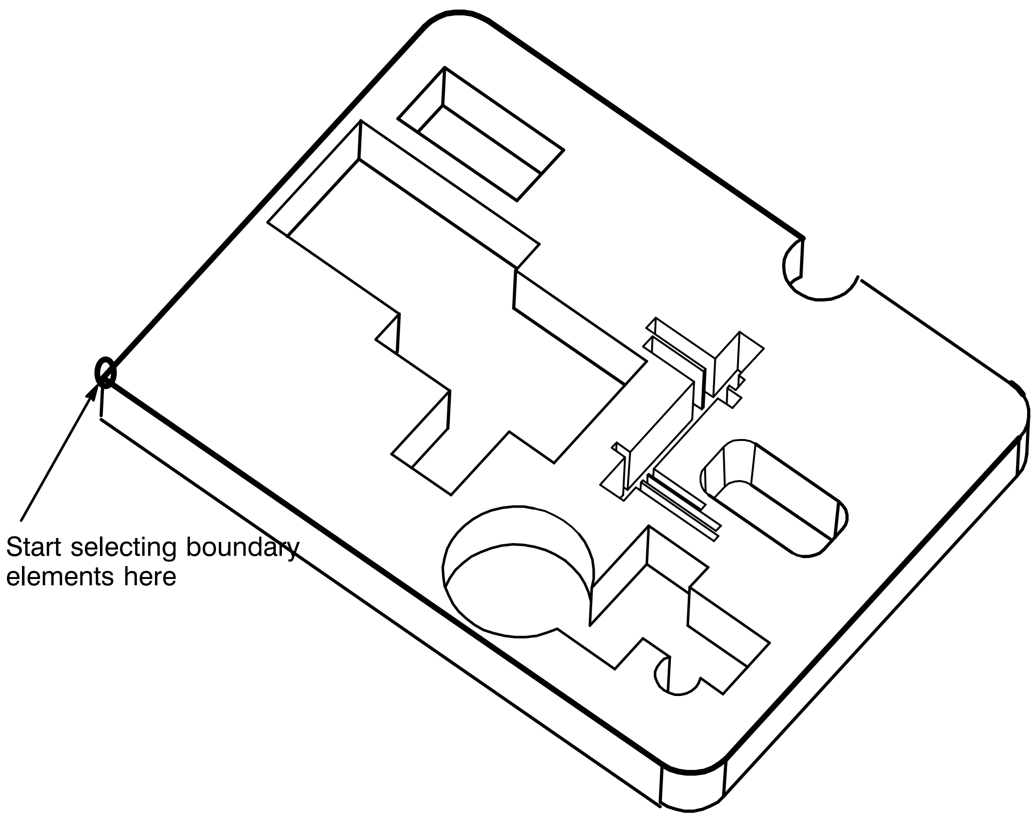
❑ Under the **Wire EDM Geometry label**, choose **Select**.

4

The Wire EDM Geometry dialog is displayed.



You will now select the boundary elements for the area that will be cut with the wire. The type of part that you are cutting would be synonymous to a punch die. The punch would be the material which would “fall” out of the cut.



❑ Choose **OK** to create the boundary and return to the External Trim dialog.

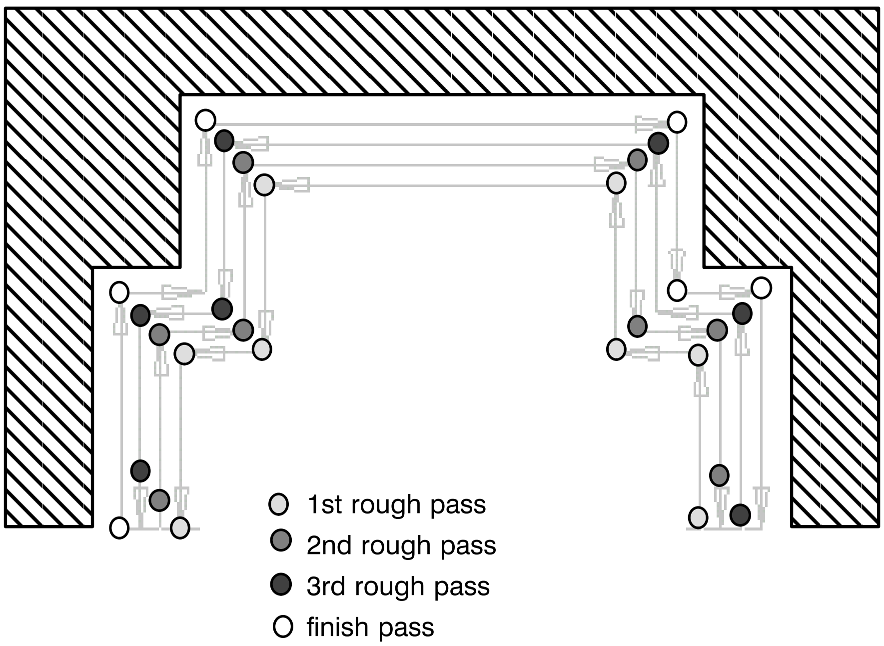
Step 2 Create the tool path.

- ❑ Choose the **Generate** icon and generate the tool path.
- ❑ Accept the operation and **Save** the part file.

You have completed the activity.

Open Profile

Open profile allows you to generate a profile cut or series of cuts which follows the shape of an open or closed boundary. Unlike external or internal trim, when creating the boundary, you can specify material side as to right or left of boundary elements.

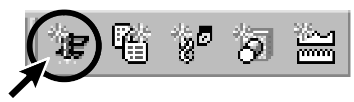


Activity 4—5: Creating Open Profile Operations

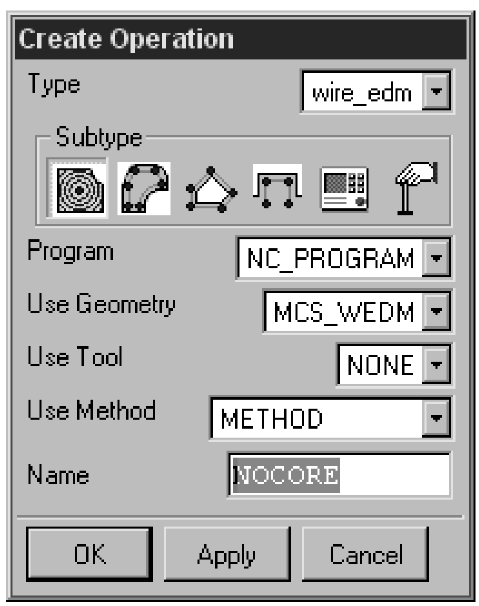
In this activity you will create an open profile operation.

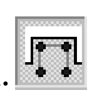
- ❑ Continue with *****wire_edm** from the previous activity.
- ❑ If necessary enter the **Manufacturing** application.

  The Operation Navigator is displayed.
- ❑ Select the Create Operation icon from the Create Tool Bar.



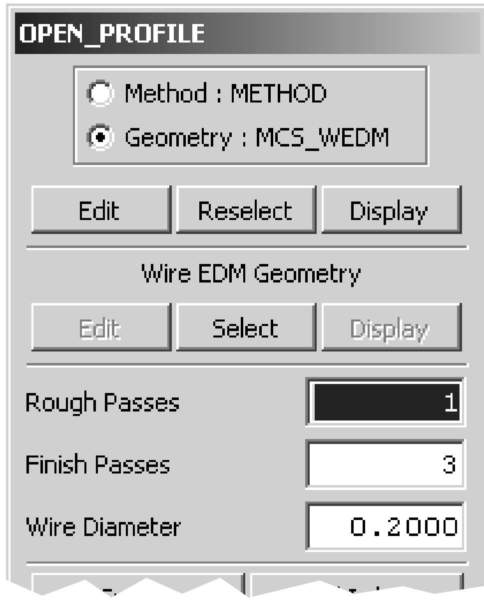
The Create Operation dialog is displayed.



- ❑ Select the **Open Profile** icon. 
- ❑ Choose **OK.**

  The Open Profile dialog is displayed.

4

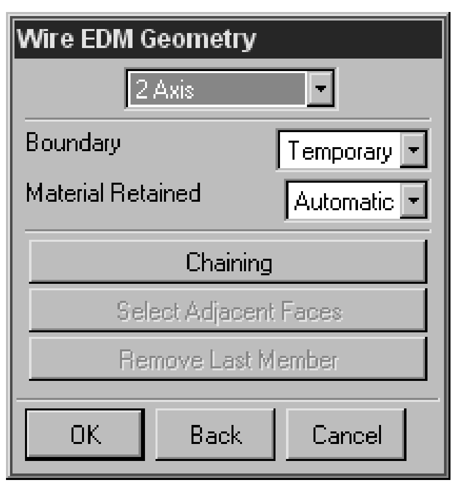


Note that the default for number of rough passes is 1 and for finish passes 3. You will change the number of finish passes from 3 to 1.

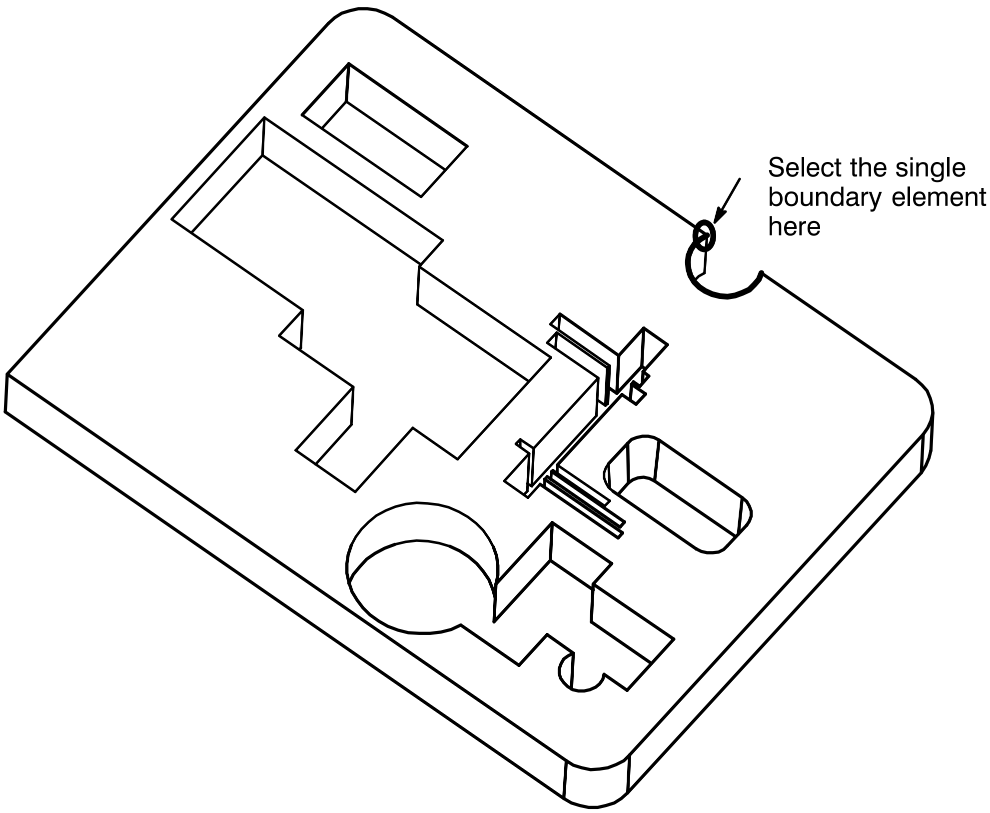
❑ Key in **1** in the Finish Passes field.

❑ Under the **Wire EDM Geometry** label, choose **Select**.

The Wire EDM Geometry dialog is displayed.



You will now select the boundary for the area that will be cut with the wire.



❑ Choose **OK** to create the boundary and return to the Open Profile dialog.

Step 1 Create the tool path.

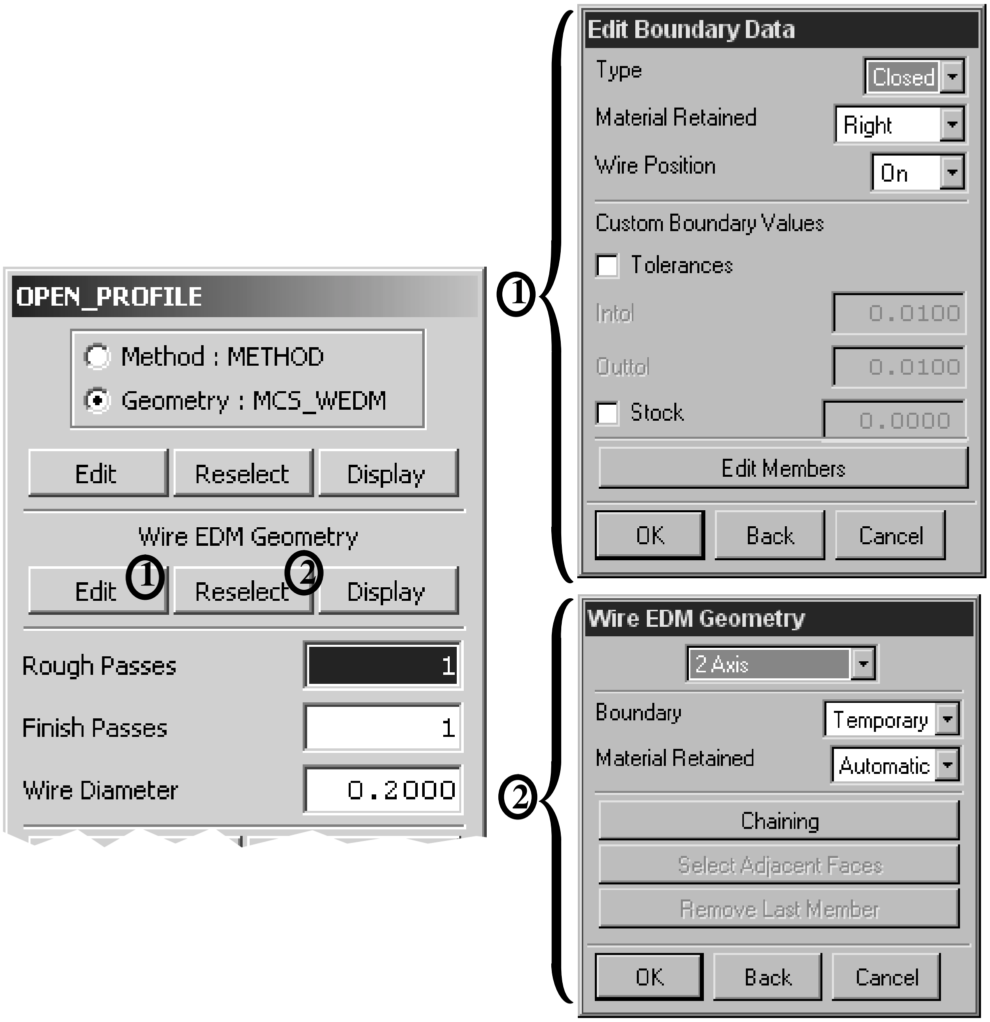
❑ Choose the **Generate** icon and generate the tool path.

❑ Accept the operation and **Save** the part file.

You have completed this activity.

Wire EDM Geometry

The Wire EDM Geometry selections allows you to Edit, Select/Reselect or Display the boundary(s) which were used to define the operation.



Edit allows modifications to various parameters in relationship to a boundary.

The **Type** parameter allows the change of existing boundaries from Open to Closed or from Closed to Open.

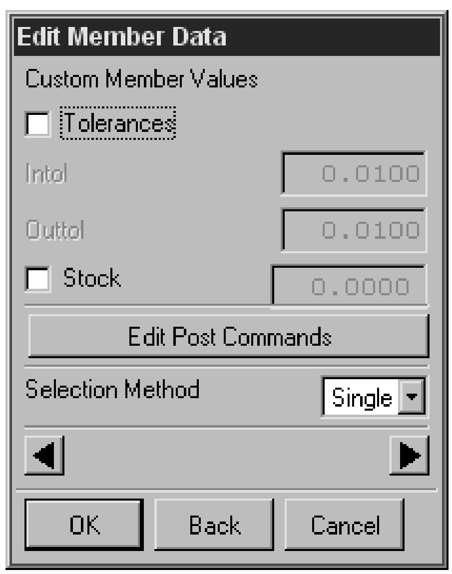
The **Material Retained** parameter allows the material to be defined with respect to the boundary. Choice for No Core, Internal and External trim is **Automatic** only. Open Profile choices are **Left** or **Right**.

Wire Position parameter allows you to specify where the wire will be positioned relative to the boundary. Choices are ON and TANTO. ON allows the center of the wire to be located on the boundary elements. TANTO positions the edge of the wire to the boundary elements. No Core operations, by default, have wire position TANTO. Internal Trim, External Trim and Open profile, by default, have wire position ON.

The **Tolerances** parameters allows you to set custom Intol and Outtol values. Custom Tolerance can be toggled on or off. Intol is the maximum allowable distance that the wire may deviate from the tool path into the part. Outtol is the maximum allowable distance that the wire may deviate from the tool path away from the part. Setting these values will override the global tolerance parameters that are applied to the boundary.

The **Stock** parameter allows you to set the final amount of stock to be left by the operation. This parameter may be toggled ON or OFF.

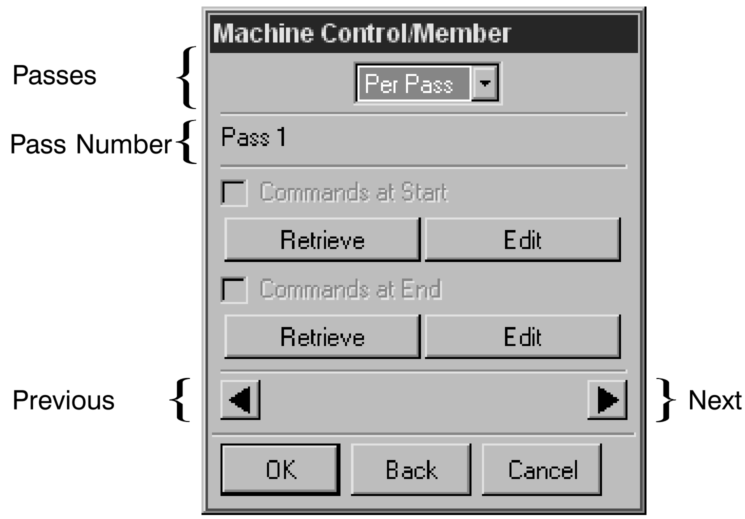
Edit Members displays the Edit Member Data dialog which allows the specification of custom tolerance and stock values and also allows the association of post commands with boundary members.



Tolerances allows the specification of Intol and Outtol for a single or number of boundary elements.

Stock allows the specification of stock for a single or number of boundary elements.

Edit Post Commands displays the Machine Control/Member dialog which allows the association of post commands (User Defined Events) with boundary members. For each boundary member you may specify the pass numbers to which the Events will apply and application of those Events to the beginning or end of each pass.



The **Passes** option toggles between **All Passes** and **Per Pass**. **All Passes** allows the specification of User Defined Events for all passes of the current boundary segment. The **Pass Number** and **Previous/Next** arrow buttons are deactivated when this option is selected.

Per Pass allows the specification of User Defined Events for single passes of the current boundary segment. The pass to which the Events apply is determined by selecting the **Next** and **Previous** arrow buttons.

Commands at Start allows the specification of User Defined Events for the beginning of the pass.

Retrieve allows the choice of a previously defined set of User Defined Events.

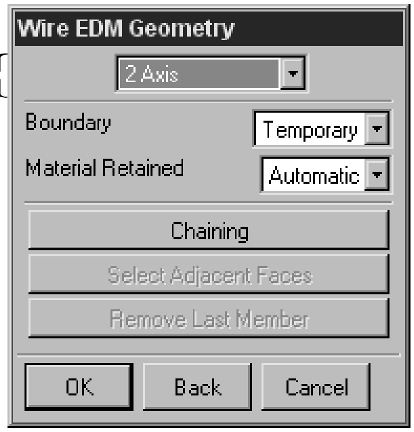
Edit displays the User Defined Events dialog which allows the definition of a set of User Defined Events.

Commands at End allows the specification of User Defined Events for the end of pass by using a previously defined set or a new set of User Defined Events.

4

Select/Reselect displays the Wire EDM Geometry menu allowing the selection or reselection of boundary geometry.

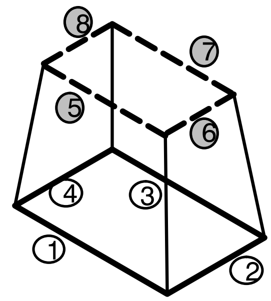
Axis type



When defining the boundaries you specify the machine type, by axis designation. Choices available are 2 Axis, 4 Axis Wireframe and 4 Axis Solid.

2 Axis allows the single selection or chain selection of boundary elements for a 2 Axis designated machine.

4 Axis Wireframe allows the single or chain selection of boundary elements for a 4 Axis machine. 4 Axis Wireframe requires an upper and lower boundary. The lower boundary will always be the boundary with the smallest ZM value. The system will automatically create the corresponding upper boundary, providing the upper and lower boundaries are connected in some fashion.



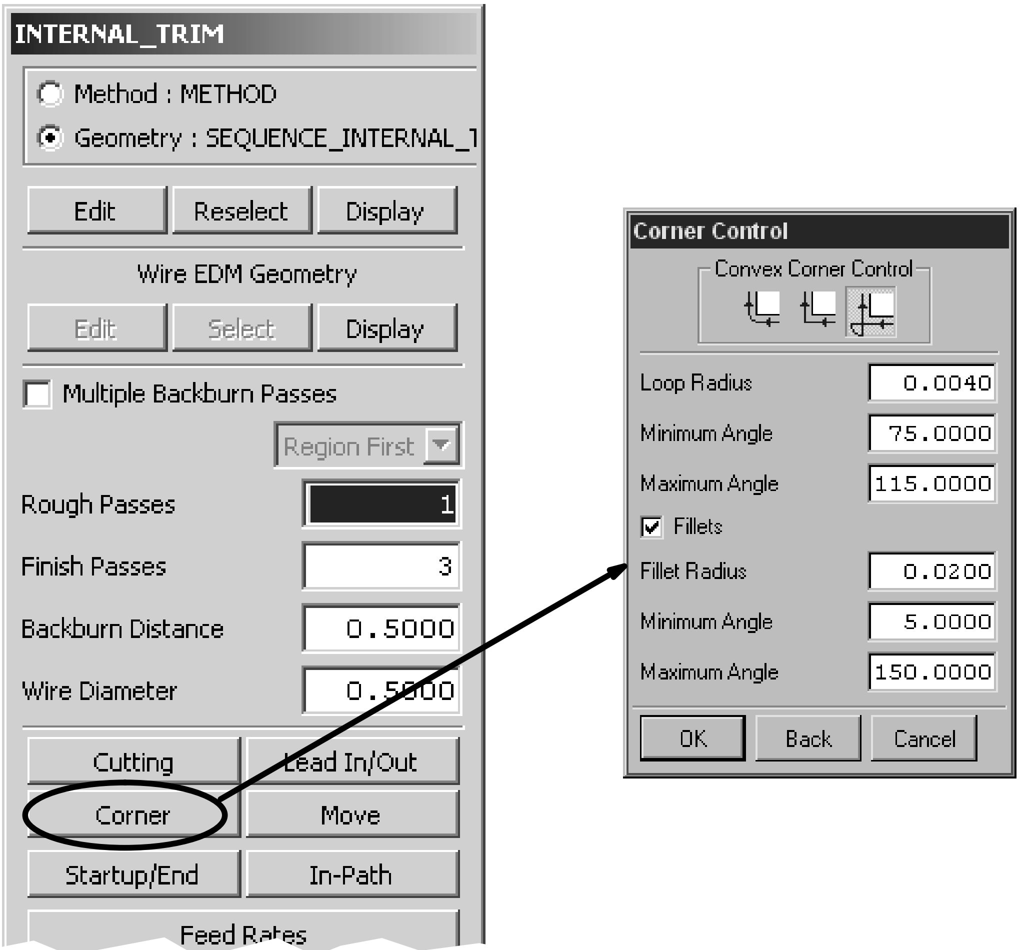
In the above diagram, if you select the sequence of boundary elements 1, 2, 3, 4 as the lower boundary, the system automatically creates the upper boundary (elements 5, 6, 7 ,8). Notice that the upper and lower boundary elements are connected by lines.

4 Axis Solid allows the specification of a boundary by selecting a sequence of closed side faces which will create the lower and corresponding upper boundaries. You may also use the Select Adjacent Faces option. When using this option, selecting the top face, will define all adjacent faces as the boundary.

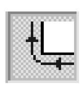
Material Retained is used to specify part material in relationship to the boundary being created. This option is only available for Open Profile operations and is specified as **Left** which defines the part material to be on the left side of the boundary or **Right** which defines the part material to be on the right side of the boundary. Either **Left** or **Right** is with respect to the direction of the boundary.

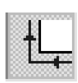
Corner Control

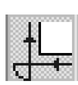
Corner control options allow you to determine the type of cut that you want to make on convex corners. If the Wire Position is set to ON (default condition), all corner control parameters are ignored. The Corner Control dialog is displayed when you select Corner under the Machining Parameters option.



There are three different methods of convex corner control:

 **Add Arcs** which leaves a rounded edge.

 **Extend Tangent** which leaves a sharp edge.

 **English D Loop** which extends past a corner and then arcs back to the next cutting direction. With this option the following parameters are available:

Loop Radius which is the actual arc generated.

Minimum Angle is the minimum size angle of a convex corner in the loop.

Maximum Angle is the maximum size angle of a convex corner.

Parameters associated with corner control are as follows:

Fillets, toggling this option on will create fillets on all concave corners that are specified by the following parameters:

Fillet Radius is the radius of the fillet which the wire cut will follow.

Minimum Angle is the minimum angle of the concave filleted corner.

Maximum Angle is the maximum angle of the concave filleted corner.

Machining Parameters

Various Machining Parameters allows you to specify parameters pertaining to wire cutting of a part. The options described pertain to the Cut Types of No Core, Internal Trim, External Trim and Open Profile.

No Core Stock allows the setting of the amount of stock which is left during the wire cutting passes. This applies to the No Core operation only.

Rough Passes specifies how many rough passes will be made for Internal Trim, External Trim and Open Profile operations.

Finish Passes specifies how many finish passes will be made for No Core, Internal Trim, External Trim and Open Profile operations.

Backburn Distance defines the length of material left as the tab in an Internal Trim operation. The value entered is the length of the tab.

Number of Passes defines how may passes are made in one operation. Number of passes is available for Internal Trim, External Trim and Open Profile. Number of Passes refers to the main trim operation and does not apply to Cutoff, Backburn or Rough Pass.

Cutoff Distance is the length of material left as the cutoff tab in an External Trim operation. Value entered is the length of the cutoff tab.

Cutoff Stock is the amount of stock which is left after the cutoff function in an External Trim operation. The stock value is only applied when the cutoff is specified as a separate sub-operation (when created within a Geometry Parent Group).

Wire Diameter defines the diameter of the cutting wire. This option is available for all operations. Value entered is the diameter of the wire.

Upper Plane/Lower Plane defines the parallel planes in which the wire guide holders move. Values define the upper and lower cutting limits of the wire and determine the length of wire displayed on the graphics window when using 3-D tool display.

The position and orientation of the planes are with respect to the Machine Co-ordinate System. The planes are normal to the ZM axis and the values are taken from the MCS origin.

Stop Point allows the definition of a Stop Point for Internal Trim, External Trim and Open Profile operations. The Stop Point suspends cutting from a specified distance before completing the final cut that will separate the part from the slug. This allows for manual intervention by the generation of an optional stop or stop code.

For External Trim and Open Profile operations, the Stop Point is created in the last pass at the specified distance from the end of the cut.

For Internal Trim operations, the Stop Point is created at the specified distance fro the end of the cut.

Associated with the Stop Point are the **Type** and **Distance** Options. **Type** may be set to Opstop (Optional Stop) which creates an optional stop code or Stop which creates a stop code. For Internal Trim operations, when Rough Pass and Backburn are created as sub-operations, Opstop is applied to the Backburn operation. For External Trim Operations, if Cut Off is created as a separate sub-operation, Opstop is then applied to the Cut Off operation.

Distance is the length that will be left in an uncut state when the Stop occurs. For External Trim operations, the stop point distance must be less than the specified Cutoff Distance. For Internal Trim operations, the stop point distance must be less than the specified Backburn distance.

Stepover parameters are used when making multiple cutting passes at a specified offset. When using ON as the Wire Position, Stepover parameters are ignored. Parameters associated with Stepover are:

Stepover Type which may be set to one of the following:

Distance allows a constant stepover as a specified distance. The stepover cannot be greater than the wire diameter. The value entered is the length of the Step Size.

Variable allows a series of stepover distances for a specified number of passes. The stepover cannot be greater than the wire diameter. Values are entered for the distance of the Step Size and a value for the Number of Passes.

% of Wire defines a constant stepover as a percentage of the wire diameter. The stepover cannot be greater than the wire diameter. The value entered is a % of the Wire diameter.

Absolute Stock allows each stepover as the distance measured from the part. The stepover cannot be greater than the wire diameter. The number of stock entries available is based on the number of specified rough and finish passes. Stock values specified must be progressively larger from the top to the bottom of the Absolute Stock dialog.

Pass Cut Direction allows you to specify the direction of the wire cut. This option is available for Internal and External Trim operation types. Direction may be set to **Alternate**, which results in alternate clockwise, counter-clockwise direction when cutting more than one pass, **CW** which results in all passes made in a clockwise direction, and **CCW** which results in all passes made in a counter-clockwise direction.

SUMMARY

The Wire EDM functions in Unigraphics CAM provides an efficient and robust capability for generating Wire EDM tool paths. Numerous operation types provide the flexibility needed for the simplest to the most complex type of parts.

The follow operation types are used in the Wire EDM module:

- No_Core
- Internal Trim with sub operations of rough pass, backburn and finish
- External Trim with sub operations of trim and cutoff
- Open Profile

(This Page Intentionally Left Blank)

In-Process Workpiece

Lesson 5

PURPOSE

Cavity Milling allows you to perform rest milling by creating an associative In-Process Workpiece (IPW) in an operation and using it as blank geometry in the next operation. It also allows you to display the previous IPW and the resultant IPW for each operation.

OBJECTIVES

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

- Use Auto Block to create blank geometry for the initial roughing operation
- Turn on the Use 3D IPW option so that the IPW created by the previous operation will be used as blank geometry in the current operation
- Use the Previous IPW option to display the IPW being used
- Use the Display Resulting IPW option to display the IPW created in the current operation

This lesson contains the following activity:

In-Process Workpiece (IPW) Overview

In order for you to make operations as efficient as possible, you must be able to ascertain what has and has not been machined in each operation. Conditions such as cutting tool lengths and diameters, draft angles and undercuts, fixture and tool clearances, will affect the amount of material or stock that each operation may leave. The representation of the material that remains after each operation is referred to as the In-Process Workpiece or IPW.

In a process commonly known as Rest Milling, the IPW is used for input into the subsequent operation which may be used for additional roughing, semi-finishing or finishing operations. The end result is a finished part that has all excess material or stock completely removed.

Activity 5—1: Creating and Using the IPW

In the following activity, you will create and use multiple In-Process Workpiece (IPW) objects to rough, semi-finish, and finish a die cavity block. The first Cavity Milling operation is provided. You will generate an IPW in this operation and use it in a subsequent semi-finishing operation. You will then generate an IPW in the semi-finishing operation and use it in a subsequent finishing operation.

Step 1 Open an existing part, save with a new name and enter the Manufacturing Application.

❑ Open the part **ama_ipw.prt.**

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

❑ Choose **Application→Manufacturing.**

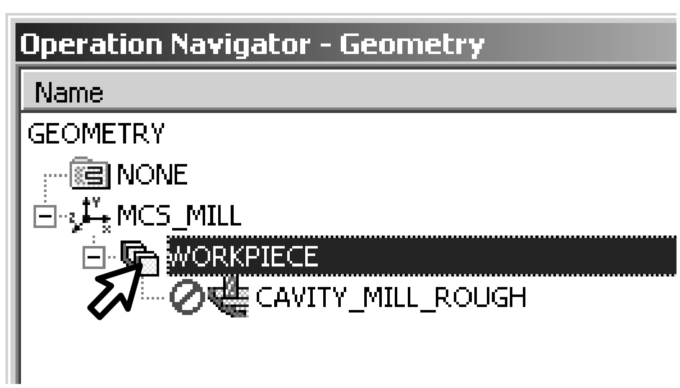
The Operation Navigator is displayed.

An In-Process Workpiece (IPW) column can be added to the Operation Navigator by clicking **MB3** on the Operation Navigator background, choosing **Columns→Configure**, and turning the **IPW** option **on**.

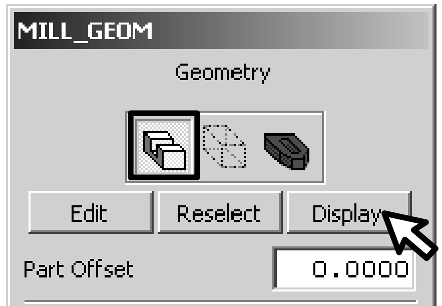
Step 2 Displaying the Part Geometry.

The part geometry has been defined in the WORKPIECE parent group.

❑ In the Geometry view of the Operation Navigator, double-click the WORKPIECE parent group.



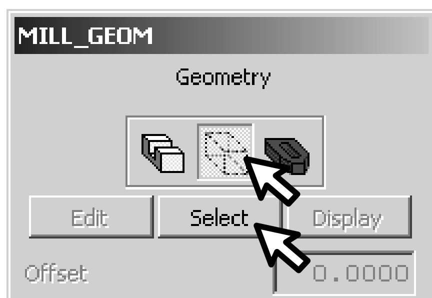
❑ With the **Part** icon highlighted, choose **Display**.



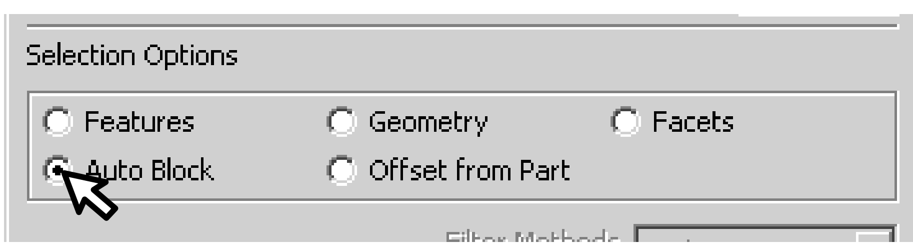
Step 3 Defining the Blank Geometry

You will define the blank geometry using a method that creates a solid body automatically by enclosing the part geometry.

❑ Choose the **Blank** icon and **Select**.

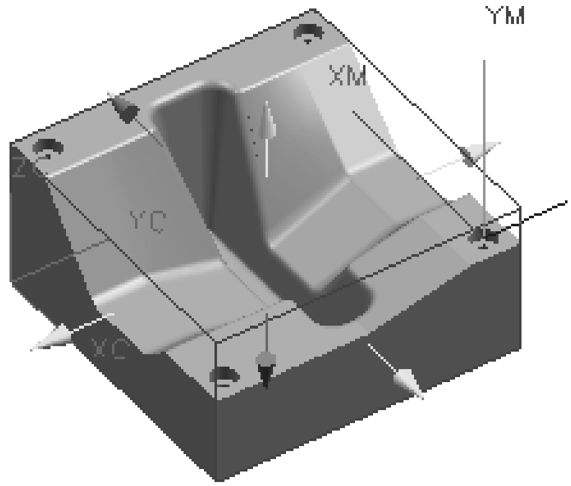


❑ Turn the **Auto Block** option **on**.



5

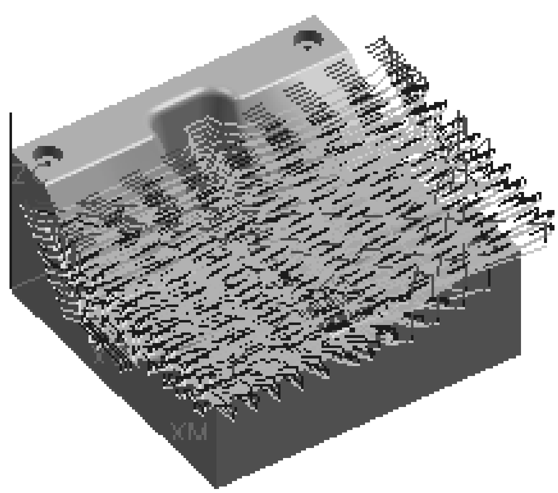
A solid body bounding the part geometry is created. The XM, YM, ZM fields allow you to modify the size of the body by specifying offsets from each face.



- ❑ Choose **OK** to accept the blank geometry with no additional offsets.
- ❑ Choose **OK** to accept the MILL_GEOM dialog.

Step 4 Generate the tool path.

- ❑ Highlight the CAVITY_MILL_ROUGH operation in the Operation Navigator and **MB3→Generate** to generate the roughing tool path.



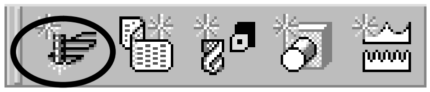
- ❑ Choose **OK** to accept the Tool Path Generation dialog.
- ❑ Refresh the graphics display.

5

Step 5 Create a Semi-Finishing operation.

You will create a semi-finishing operation that uses the IPW defined by the roughing operation.

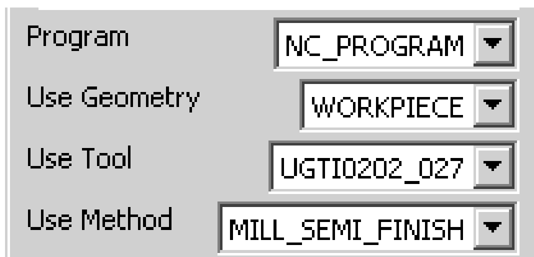
❑ As shown below, select the **Create Operation** icon from the Create toolbar.



The Create Operation dialog is displayed.

❑ Choose **CAVITY_MILL** as the Subtype.

❑ Specify the following parent groups.



❑ Key in **cavity_mill_semi_finish** in the Name field.

❑ Choose **OK** to begin creating the operation.

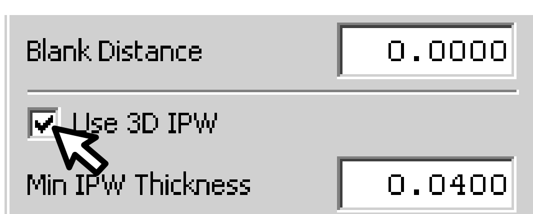
Step 6 Use the IPW as blank geometry.

You will specify that you wish to use the IPW in the previous operation to define the blank geometry in this operation.

❑ Choose **Cutting**.

The Cut Parameters dialog is displayed.

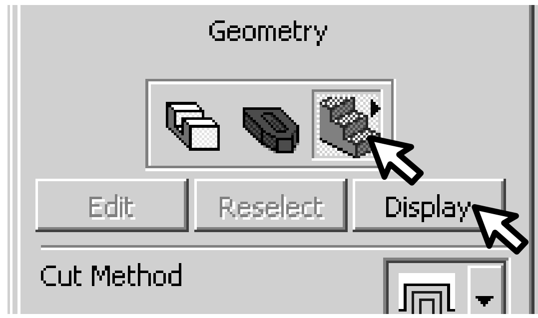
❑ Turn the **Use 3D IPW** option **on**.



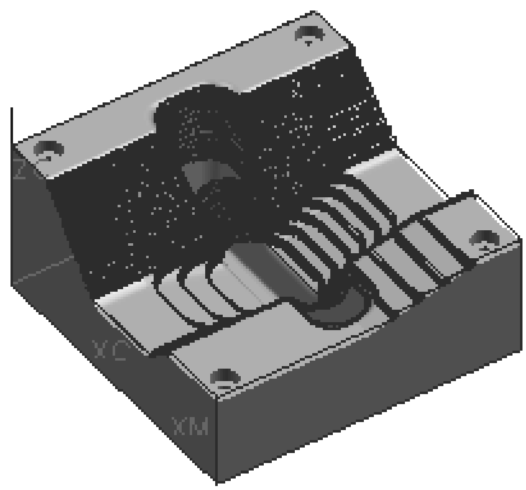
❑ Choose **OK** to accept the Cut Parameters dialog.

The Blank icon at the top of the Cavity Mill dialog has been replaced by the Previous IPW icon.

❑ Choose **Previous IPW** and **Display**.



The system may require some processing time to display the faceted body.

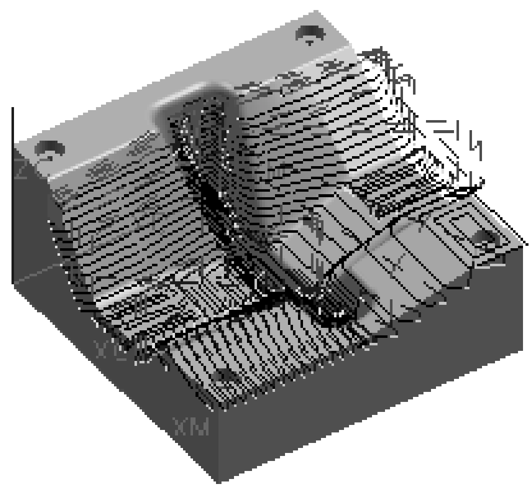


This faceted body is the IPW that this operation uses as blank geometry.

Step 7 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.

❑ Turn the three **Display Parameter** options **off** and choose **OK** to continue generating the tool path.

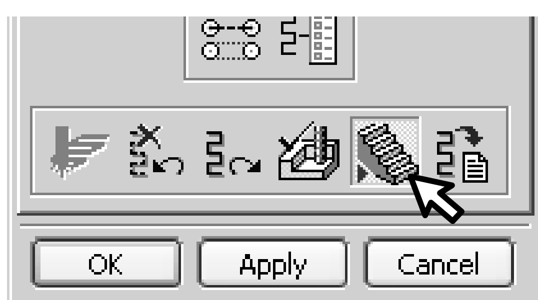


❑ Refresh the graphics display.

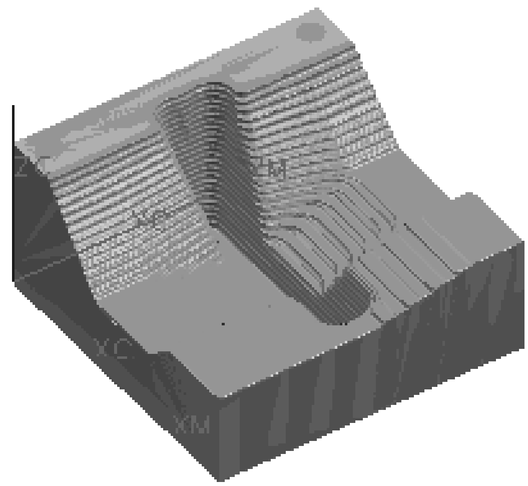
Step 8 Display the resulting IPW.

The IPW created by this operation can now be displayed.

❑ Choose **Display Resulting IPW**.



The system may require some processing time to display the faceted body.



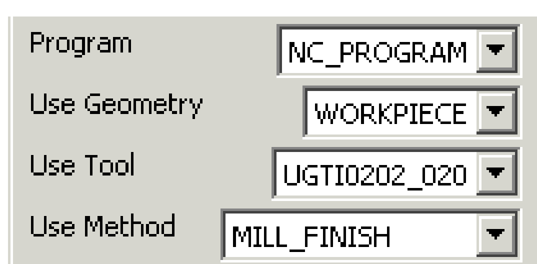
This faceted body is the IPW the next operation will use as blank geometry.

- ❑ Refresh the graphics display.
- ❑ Choose **OK** to complete the operation.

Step 9 Create a Finishing operation.

You will create a finishing operation that uses the IPW defined by the semi-finishing operation.

- ❑ Select the **Create Operation** icon from the Create toolbar.
- ❑ Be sure **CAVITY_MILL** is selected as the Subtype.
- ❑ Specify the following parent groups.



- ❑ Key in **cavity_mill_finish** in the Name field.
- ❑ Choose **OK** to begin creating the operation.

Step 10 Use the IPW as blank geometry.

You will specify that the IPW in the previous operation will define the blank geometry in this operation.

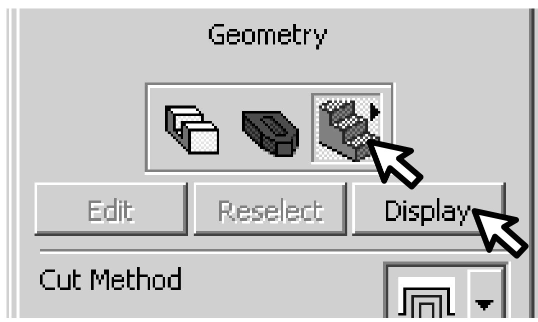
❑ Choose **Cutting**.

The Cut Parameters dialog is displayed.

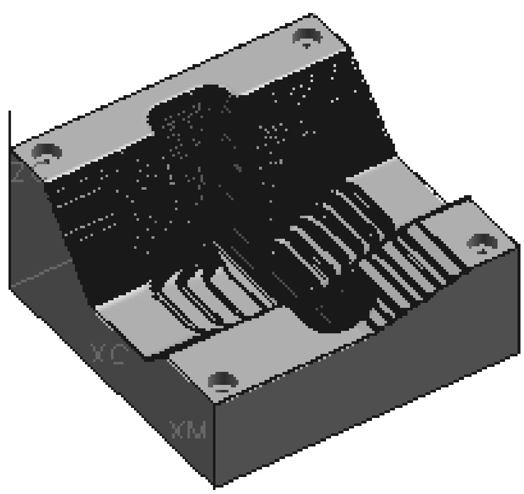
❑ Turn the **Use 3D IPW** option **on**.

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

❑ Choose **Previous IPW** and **Display**.



The system may require some processing time to display the faceted body.



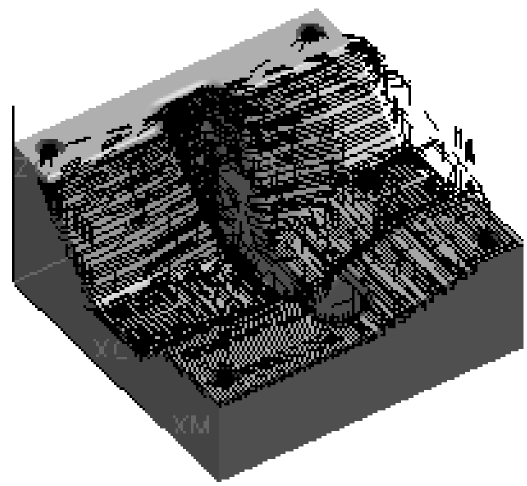
This faceted body is the IPW that this operation uses as blank geometry.

❑ Key in **0.100** in the Depth Per Cut field.

Step 11 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.

❑ Turn the three **Display Parameter** options **off** and choose **OK** to continue generating the tool path.

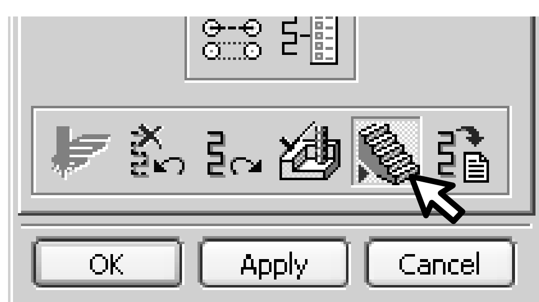


❑ Refresh the graphics display.

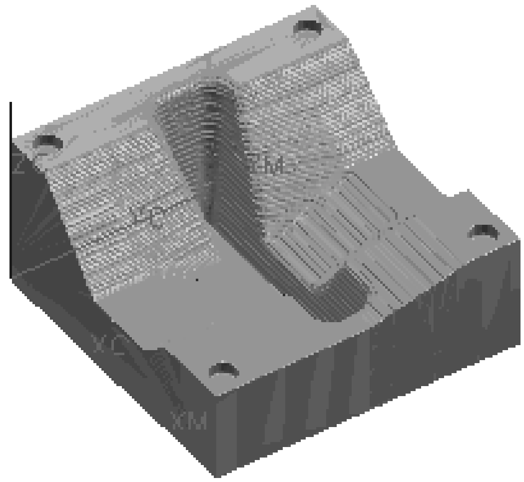
Step 12 Display the resulting IPW.

The IPW created by this operation can now be displayed.

❑ Choose **Display Resulting IPW**.

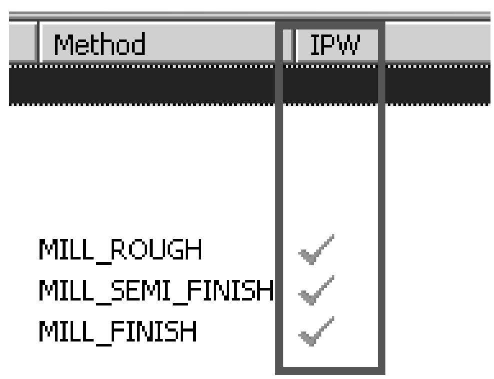


The system may require some processing time to display the faceted body.



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

The check marks in the IPW column indicate which operations contain resulting IPW's.



If an new operation is inserted in the program sequence, if an operation is deleted, or if the operations are reordered, clock icons indicate that the resulting IPW's are out of date. This simply means that when generating the tool paths, the IPW's will need to be updated internally, requiring additional processing time.

❑ **Save** the part file.

SUMMARY

Rest milling can be performed by creating an associative In-Process Workpiece (IPW) in an operation and using it as blank geometry in the next operation.

The In-Process Workpiece provides an efficient and robust method of using material left by previous cutting operations as blank geometry for the next operation in the program.

In this lesson you:

- Used Auto Block to create blank geometry for the initial roughing operation
- Turned on the Use 3D IPW option so that the IPW created by the previous operation will be used as blank geometry in the current operation
- Used the Previous IPW option to display the IPW being used
- Used the Display Resulting IPW option to display the IPW created in the current operation

(This Page Intentionally Left Blank)

Machining Faceted Geometry

Lesson 6

PURPOSE

In numerous applications, faceted geometry is used to create prototype design and manufacturing models. Direct Machining of Facets allows you to directly machine faceted geometric without having to go through the tedious process of converting the facets to a wireframe or solid geometric model.

OBJECTIVES

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

- Import an STL file into Unigraphics
- Generate tool paths on faceted geometry

This lesson contains the following activity:

Direct Machining of Facets

6

It is extremely important for designers and manufacturing departments of a company to be able to reverse engineer a product, when exact math data may not be available, to be competitive in the marketplace. These examples are ideal situations for Direct Machining of Facets (DMF). The process of scanning or digitizing a part creates a cloud of points, which can be converted into a faceted model. This faceted model can then be imported into Unigraphics for modeling and or machining applications.

Real life examples such as the machining of dies and discrete part manufacturing lend themselves to DMF.

The DMF allows you to generate tool paths on faceted part geometry without the need to create surface geometry. DMF can be used with Fixed Axis Surface Contouring, Cavity Milling and Z-Level Milling operations by allowing the selection of Faceted Bodies as valid part geometry for tool path generation.

Activity 6–1: Machining of Faceted Geometry

In this activity you will generate Cavity Milling tool paths on faceted geometry imported into Unigraphics. All geometry will be created with a metric database.

Step 1 Create a metric Unigraphics base file used for importing the faceted model.

❑ Choose **File→New** and name the part file *****_DMF**, where ******* represents your initials.

❑ If necessary, change the units to millimeters.

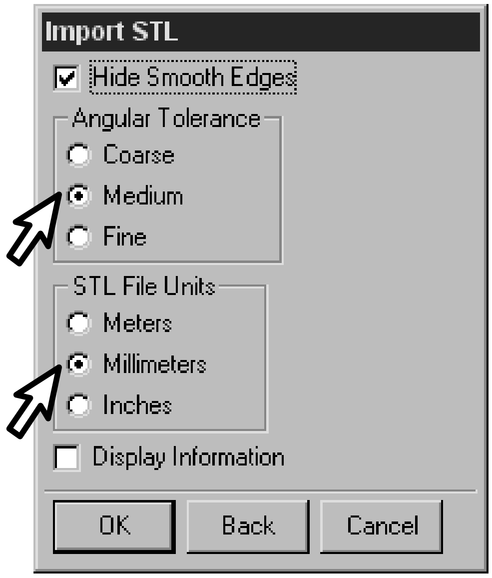
❑ Select **OK**.

A new part file has been created which will be used to import the faceted model of the cavity of a plastic hair dryer. The faceted model is in STL format.

Step 2 Import the faceted model.

❑ Choose **File→ Import→ STL**

The Import STL dialog is displayed.

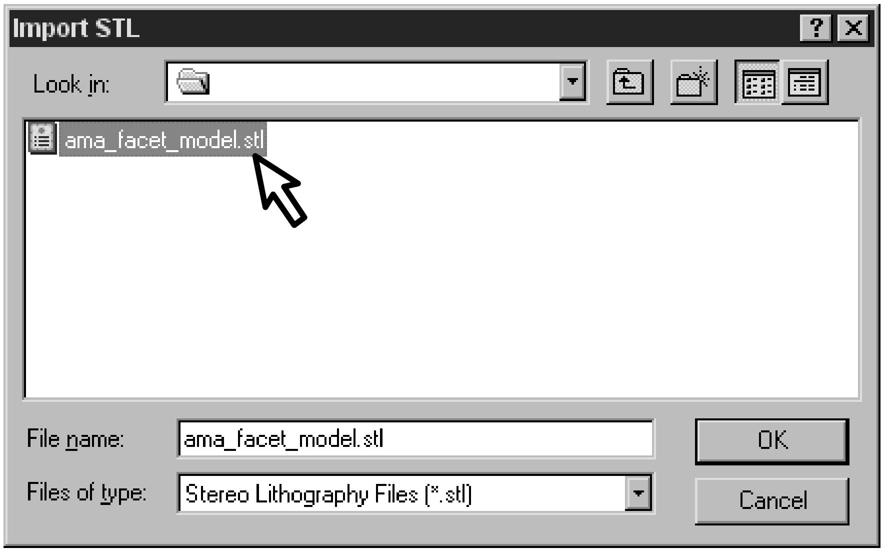


6

❑ If necessary select Angular Tolerance **Medium** and STL File Units **Millimeter.**

❑ Choose **OK**.

The Import STL File Selection dialog is displayed.



❑ Select **ama_facet_model.stl**.

The faceted model data file usually has an ".stl" file extension.

❑ Select **OK**.

The file is imported into the Unigraphics "base" file which you previously created.

❑ Change the display to solid and fit the view to the screen.

Step 3 Create the Cavity Milling operation necessary to machine the imported faceted model.

- ❑ Enter the Manufacturing Application.

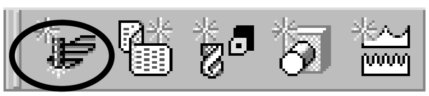
  The Machining Environment dialog is displayed.

- ❑ Select **mill_contour** as the CAM Session Configuration, select **mill_contour** for the CAM Setup and then select **Initialize**.

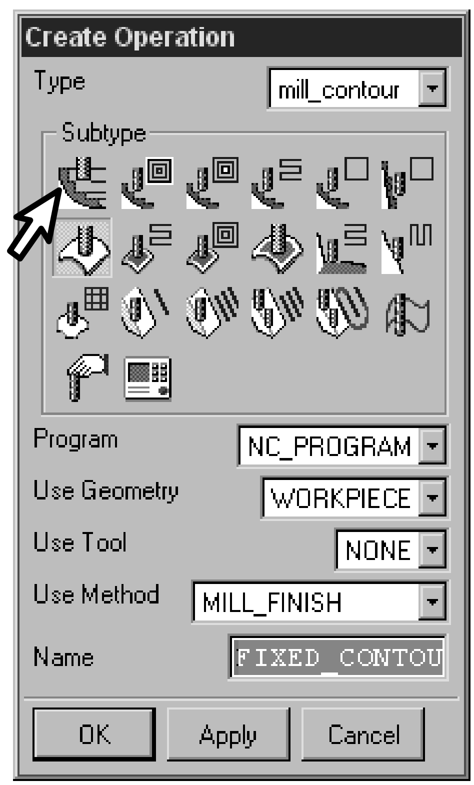
  The Operation Navigator is displayed.

- ❑ If necessary, change the view of the Operation Navigator to the **Program View.**

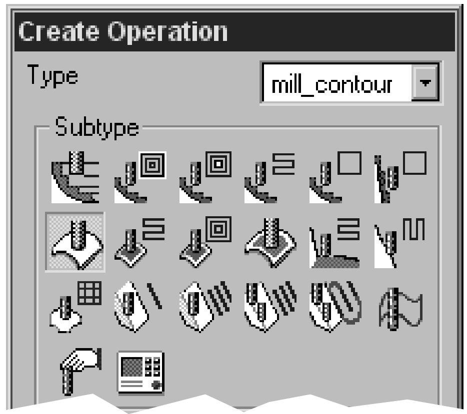
- ❑ Select **Create Operation** from the CAM Create toolbar.



The Create Operation dialog is displayed.



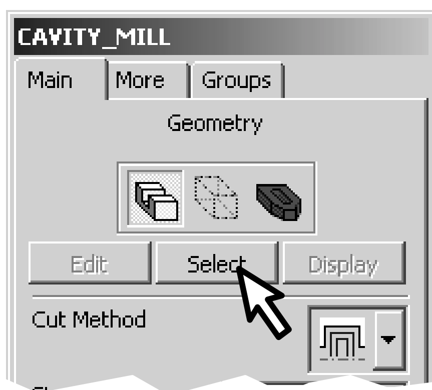
- ❑ Select the **Cavity_Mill** subtype.



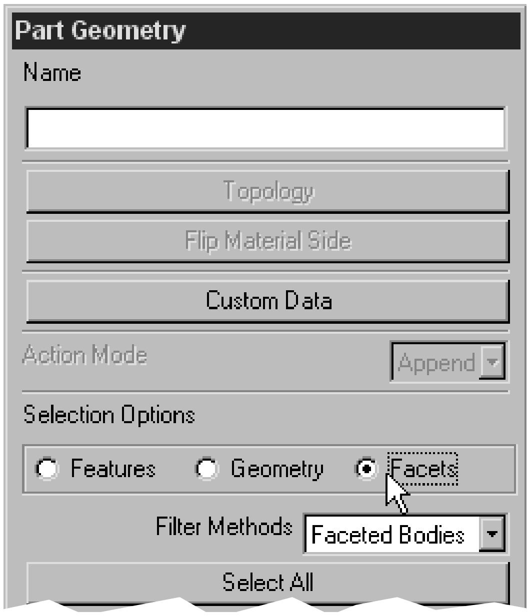
❑ Select **OK**.

The Cavity_Mill dialog is displayed. You will use an 8mm ball tool, **ugt0203_003**, to rough out the cavity. Other parameters used will be default parameters. You will now select the geometry to machine the part.

❑ Choose the **Select** icon from the Geometry section of the Cavity_Mill dialog.



❑ Select Facets from the Part Geometry dialog.



You will now select all of the faceted geometry of the part.

❑ Choose **Select All** from the Part Geometry dialog.

❑ Select **OK.**

Step 4 Generate the tool path.

❑ Choose the **Generate** icon and generate the tool path.

❑ Close the part.

SUMMARY

Direct Machining of Facets (DMP) in Unigraphics provides an easy and efficient method of machining parts that have been reversed engineered and are converted to STL data files.

In this lesson you:

- Imported an STL file into Unigraphics, creating a faceted model
- Directly machined a faceted model using the Cavity Milling operation type

High Speed Machining
Lesson 7

PURPOSE

This lesson will introduce you to the concepts of High Speed Machining (HSM), which increases productivity and improves the quality of the final part being machined. HSM achieves these results through the use of consistent volume removal concepts and smooth cutter path generation.

OBJECTIVES

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

- Generate High Speed Machining operations
- Generate Nurbs output

This lesson contains the following activities:

High Speed Machining- An Overview

The concept of High Speed Machining (HSM) is not new. In fact, it was demonstrated over 75 years ago. The original concept was conceived by Dr. Carl Salomon in 1924 and was patented (German patent) in April of 1931. In recent years, the concept was further developed through United States Air Force research funding and has slowly been brought out of the classified world into the everyday commercial applications.

HSM technology has shown increases in productivity and improved part quality. Characteristics of HSM are high spindle speeds, fast feed rates, light cuts, smooth tool movement and constant volume removal. Due to the rapid changes in dynamics of chip removal at these very high speeds, cut methods and characteristics of the tool path are critical factors in the success of the cutting process. Factors such as sudden stops, sharp corners, reversal of cut direction and erratic tool movements will directly affect the speed at which cuts are made.

There are two basic goals for HSM. They are:

- Maintain constant material volume removal
- Generate smooth tool moves throughout the entire cutting path

Applications abound in the aerospace and mold and die industry for HSM technology. Cutting thin wall parts in the aerospace industry is a typical application. In the mold and die industry, contoured surface cutting can be accomplished at high spindle speed and feed rates. Incorporating very small stepovers results in very fine finishes that generally require no hand finishing work. Since tool deflection is at a minimum, greater accuracies can be achieved.

Basic requirements

HSM is currently being used in a variety of applications. Chip removal characteristics translates into very high spindle speeds, typically 25000-100000 rpm. Corresponding feed rates from 250 -1200 ipm are typical. Due to these high spindle speeds and feed rates, heat is dissipated through the chip, allowing the cutting tool to run cooler and being able to keep its cutting ability for longer periods of time. The higher spindle speed results in faster feed rates that will in turn maintain the proper chip load per tooth. At these rates, the depth of cut can be reduced and high volume removal rates can be maintained. The reduced depth of cut minimizes tool and part deflection which results in better control of surface finishes and part dimensions. HSM allows the roughing of parts to net sizes and shapes. The reduced cutter stress afforded by this process also minimizes the risk of cutter breakage and allows for longer production runs and unattended machining.

HSM also has the potential of reducing the time to manufacture. Normally, much of the cost and time delay in manufacturing is due to two primary factors. First is the time that it takes to design the casting or forging, and then the time to do the manufacturing. Due to the higher material removal rate associated with HSM, consideration can be given to manufacturing parts from plate and bar stock rather than castings and forgings, saving the wait time to normally procure the material.

HSM technology is actually placing a burden on programmers and the methods which they use to produce tool paths. This is primarily a result of the NC/CNC programs being so much longer because of shallow depths of cut and fine stepovers. Also, HSM machinery can cut parts faster and therefore reduce cycle times.

Methods for most High Speed Machining applications

The following methods should always be considered when doing HSM:

- Constant volume removal - an ideal goal is to maintain consistent volume removal. With this goal, care must be taken when making stepover moves.
- Smooth tool movement - due to high feed rates, it is imperative to avoid abrupt starts and stops as well as sharp corners. Acceleration at starts and deceleration into stops are crucial. All corners should be rounded or rolled, that is the cutter should make changes in its direction by going through arc moves.
- Rounded or rolled exterior as well as interior corners.
- Multiple roughing tools and depth ranges - for deep cavities and pockets, the preferred method of cutting is to use multiple tools with progressively longer lengths. Shorter tools are rigid and can cut well when taking deeper cuts. The depth of the feature(s) that are being cut is divided into ranges that match the longer tool lengths.
- For roughing passes, use helical engage with a ramp angle of between 5-10 degrees. For semi-finish and finish cuts, use circular engages.
- When finishing, scallops must be kept to a minimum. Keep these heights as low as 0.00005” and set Intol/Outol between 0.0005 and 0.00005”. This will substantially increase the size of the program but will greatly decrease the amount of hand finishing (if any) that must be done.

- Optimization of tool paths - for tool paths cutting multiple regions, engaging and retracting within those regions, optimization of the order of those regions is necessary.
- Multiple pocket cutting order - must be able to specify the order of the pocket or feature being cut to reduce part stress and warpage.
- When cutting thin wall multi-pocket or cavity parts, always cut the level first in all multiple features before progressing to the next level. This should be done in both roughing and finishing so as to maintain rigidity in the thin walls.

High Speed Machining vs Conventional Machining

Conventional machining normally uses high speed steel or carbide cutters, cutting up to 700 surface feet per minute (SFM), with feed rates up to 50 inches per minute (IPM). Tremendous heat is generated in the process. This excess heat is transmitted to the cutter and workpiece and is dissipated through the use of large amounts of coolant.

High speed machining usually uses small, solid carbide, or large carbide inserted tools. Cutting begins at 1000 SFM, with feed rates of 90 to 100 inches per minute without the use of coolant. With the use of coolant, feed rates in excess of 1000 IPM can be achieved.

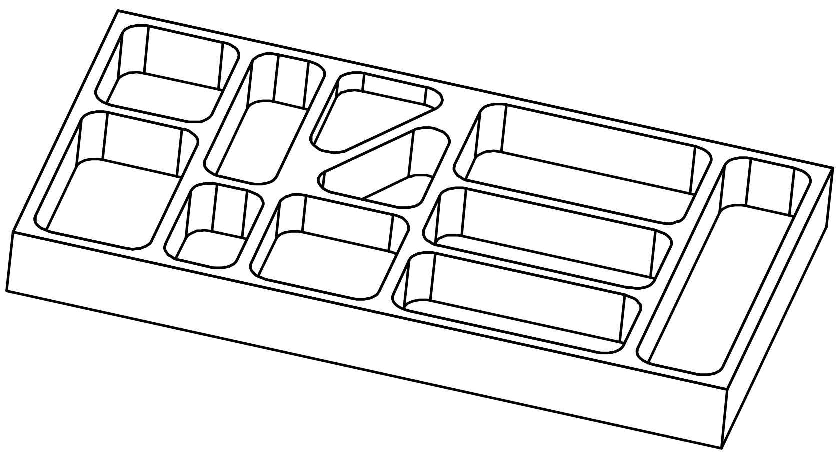
The key to successfully perform high speed machining are various methods used in the cutting operations. Techniques such as roughing at a shallow depth with maximum width of cut promotes longer tool life and higher cutting accuracy. In some cases, semi-finish passes can be completely eliminated. Tool paths, which are generated for these types of applications, need to use very fine Intol/Outtol values (.0001”) for optimum finishes. Scallop height needs to be kept at a minimum with stepover values set to .00005” or less.

Activity 7—1: Creating a High Speed Machining Operation

In this activity, you will edit a conventional pocketing operation, changing parameters that will make the operation ideal for High Speed Machining.

Step 1 Open, then rename a part file and review an existing operation.

❑ Open the part file **ama_hsm_1.prt.**



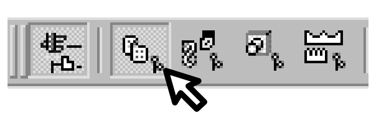
❑ **Save As ***_hsm_1.prt** where *** represents your initials.

❑ Enter the **Manufacturing** Application.

This part file contains an operation that roughs a pocket with default template settings and a clearance plane .100 above the part. You will edit the operation and change numerous parameters that will make the operation applicable to High Speed Machining.

You will review the existing operation prior to making any modifications.

❑ If necessary, change the view of the Operation Navigator to the Program View.



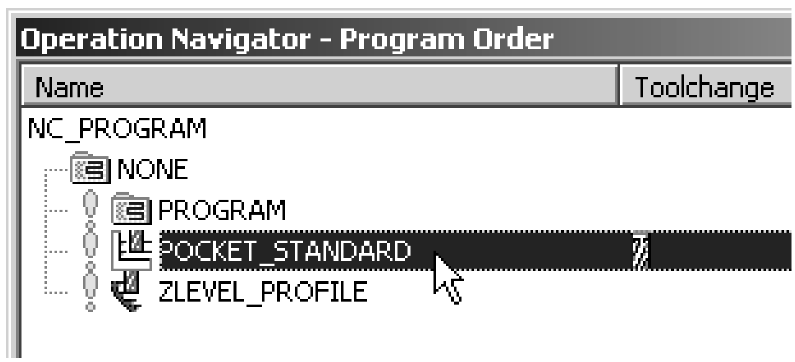
❑ Select the operation, **pocket_standard,** use MB3 and select **Toolpath → Verify.**

The tool path for the **pocket_standard** operation is displayed. Pay particular attention to the method of engagement, the sharp corners generated within the tool path, the depth of cut, etc. The tool path display has been intentionally slowed so that you may observe the various movements.

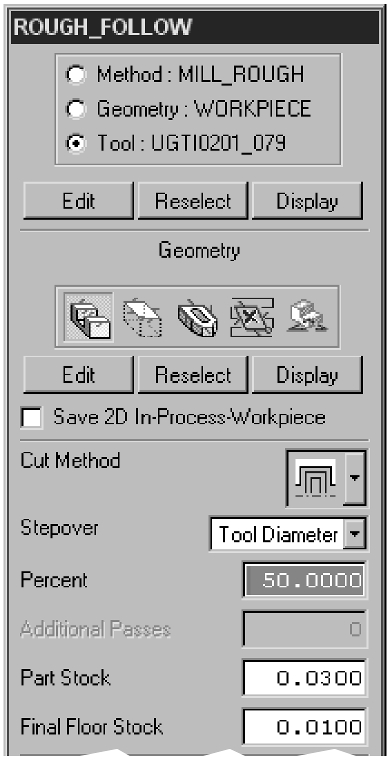
You will now edit the **pocket_standard** operation and modify parameters that are applicable for High Speed Machining.

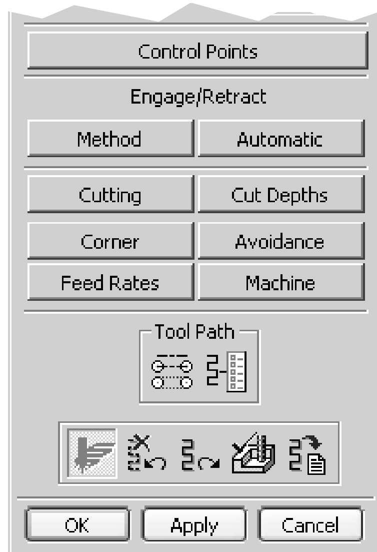
Step 2 Edit the existing operation and modify parameters suitable for High Speed Machining.

❑ Double click on the operation, **pocket_standard.**



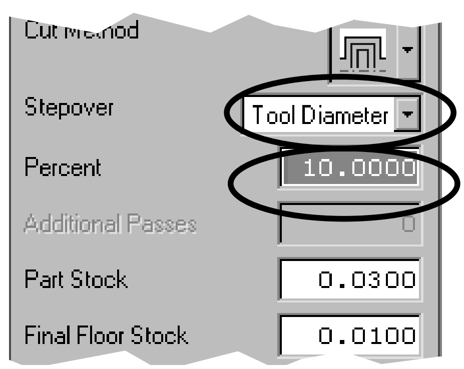
The ROUGH_FOLLOW dialog is displayed.





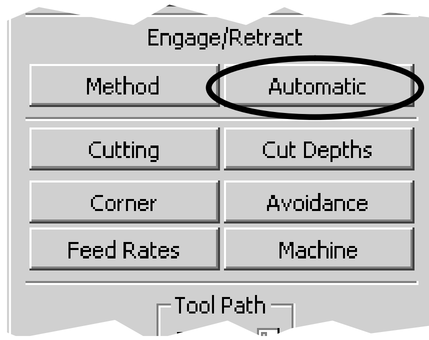
Currently, all parameters set are default parameters. You will now change parameters for High Speed Machining.

❑ From the **Stepover** pull down menu, select **Tool Diameter** and then change **Percent** to **10.** This will minimize the stepover value to 10% of the effective cutter diameter and will help maintain constant tool loading conditions.



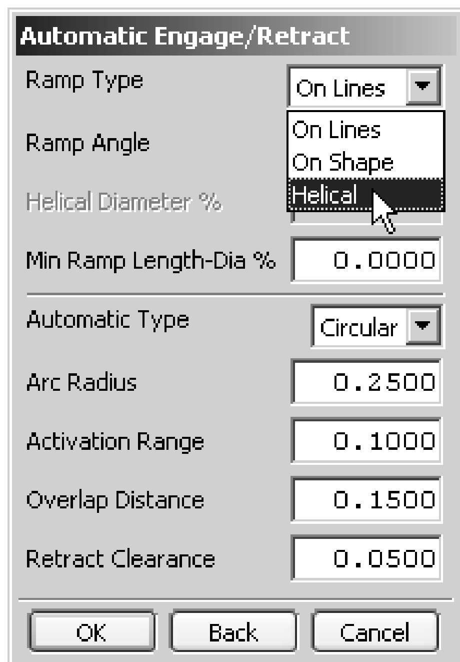
You will now change the parameters used for engagement from one cut level to the next to Helical.

❑ Select **Automatic** under **Engage/Retract.**



The Automatic Engage/Retract dialog is displayed.

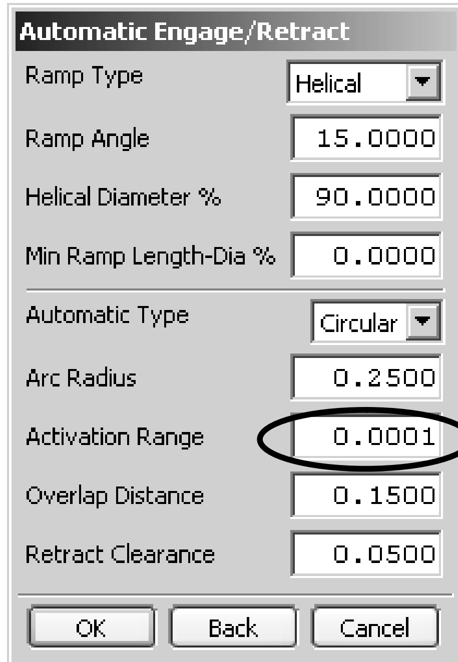
❑ Select the **Ramp Type** pull down menu and then select **Helical.**



Changing the Ramp Type to Helical insures that tool engagement at the next cut level, will be with a helical move. This will keep forces on the cutter consistent, more so than if you were to plunge to the next level.

Note: Cutter geometry determines specific ramp angles and overlap distance requirements.

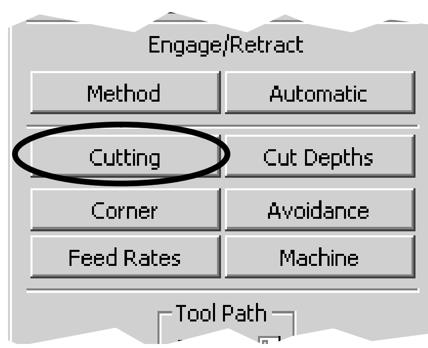
❑ Change the **Activation Range** to **.0001.**



Changing the Activation Range to this small value insures that the cutter will not engage the part as it approaches the walls.

❑ Choose **OK**.

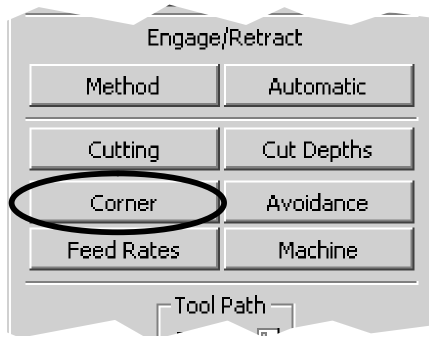
❑ Select **Cutting** and change the **Intol** parameter to **.00005** and the **Outtol** parameter to **.0005**.



As the machine tool travels faster and very accurately, a much finer surface finish can be achieved. The movement per block can be shorter to obtain the best dimensional accuracy.

❑ Choose **OK**.

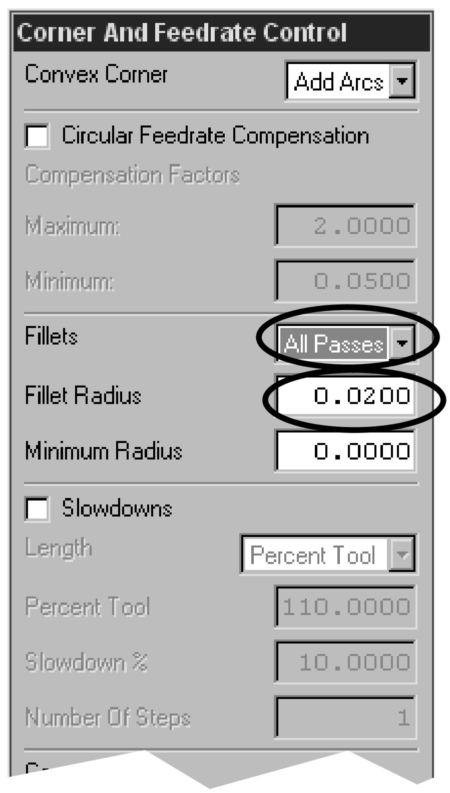
❑ Select **Corner.**



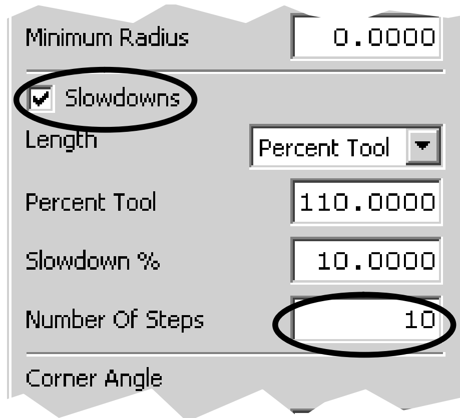
This option will add a fillet at all corners (corner roll) which eliminates sharp and sudden moves.

7

❑ Select **All Passes** from the **Fillets** pull down menu and accept the default of **.020** for **Fillet Radius**.



❑ Select **Slowdowns** (checked). Change Number of **Steps** to **10**.

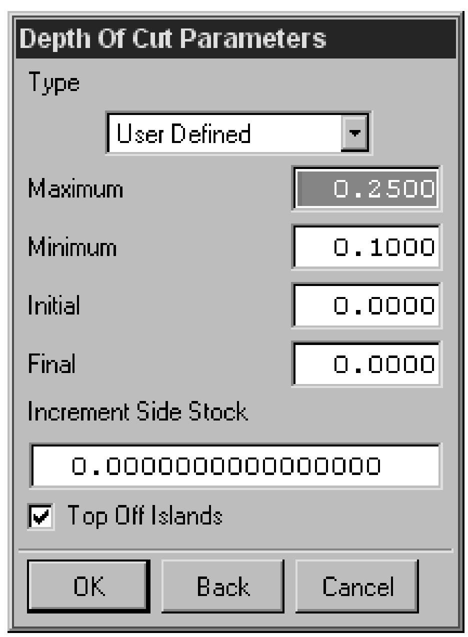


Slowdown is used to slow the feed rates as you approach corners or obstructions in the tool path. Slowdown can be controlled by length, starting location and the rate of slowdown. Number Of Steps allows you to set the abruptness of slowdown. The greater the number, the more even the slowdown.

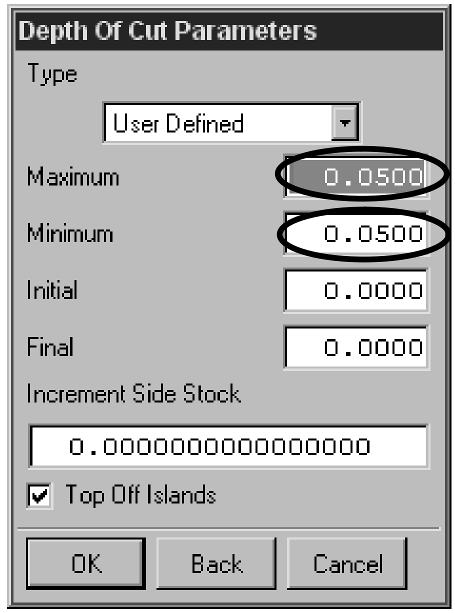
❑ Choose **OK**.

❑ Select **Cut Depths**.

The Depth of Cut Parameters dialog is displayed.

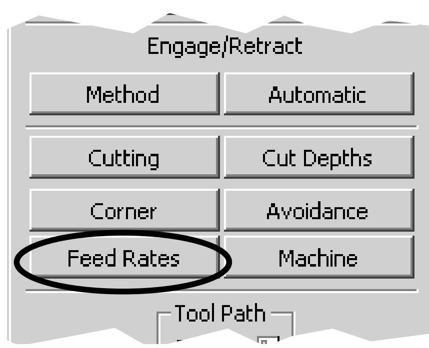


❑ Change the **Maximum** and **Minimum** values to **.050.**

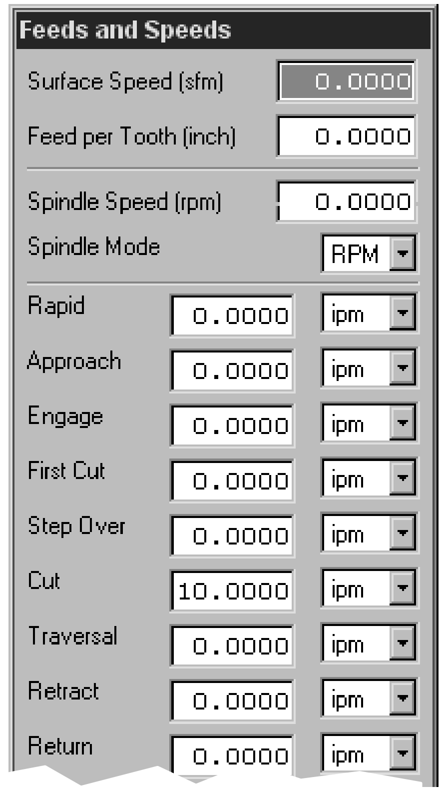


Reducing the depth of cut while increasing feed rates is a core process change of High Speed Machining. More material can be removed in less time. An advantage of a shallower depth of cut is the cutter will take less power to cut through the material.

- ❑ Choose **OK**.
- ❑ Select **Feed Rates**.

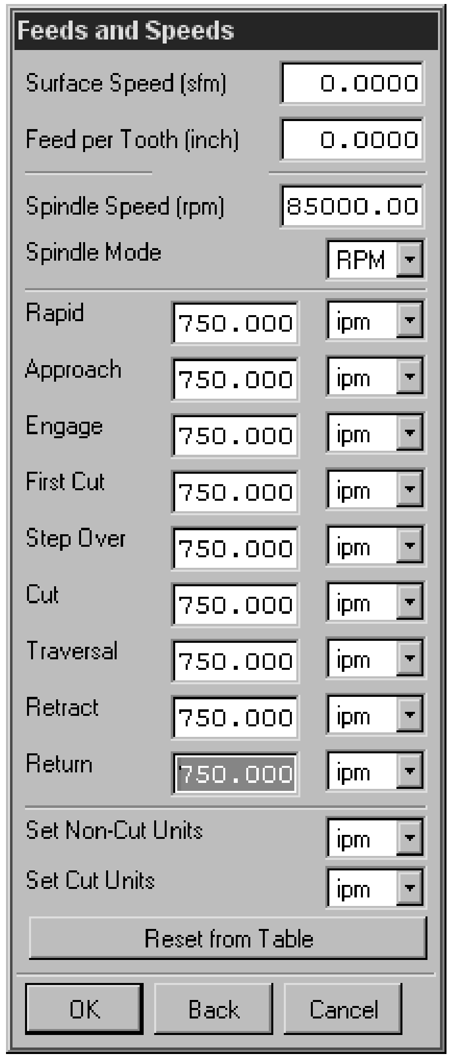


The Feeds and Speeds dialog is displayed.



You now want to change your feed rate and spindle speed to the maximum values that your HSM process can address; in this example you will set the feed rate to 750 inches per minute and the spindle RPM to 85000.

❑ Change the **Spindle Speed** to **85000** and all Feed Rate values to **750**.



Changing all the feed rate values to the same feed rate insures a constant load on the cutting tool.

❑ Choose **OK**.

Step 3 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.

Examine the various motions. Once you generate the 1st cut level, you may want to stop the display motion and change the Path Display speed to 10. After generating the tool path at all cut levels, list the tool path and examine the feed rates.

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

❑ **Save** the part file.

You have completed this activity and will be using this part file in the activity on mixed cut directions.

Mixed Cut Directions

Cut patterns for high speed machining must allow constant volume removal and eliminate burying the cutter into material. They must also provide a smooth transition from level to level, eliminating constant retracting, traversing and engaging.

Mixed cut directions are useful when large open areas are cut and you want the cutter to cut back and forth instead of beginning each cut at the same end of the part. This will minimize the time that is spent traversing between the various cut levels and from the end of one cut to the beginning of the next.

The next activity will familiarize you with using mixed cut directions and making direct moves when cutting between levels in a Zlevel cutting operation.

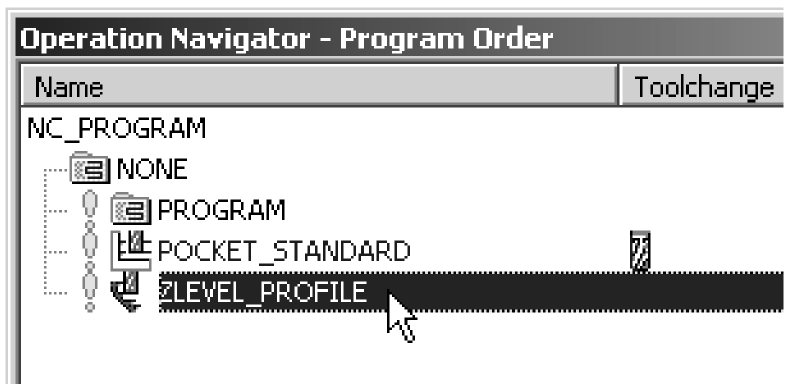
Activity 7—2: Mixed Cut Directions

In this activity, you will use the part file from the previous activity and explore the use of the Mixed Cut Direction option.

Step 1 Edit the existing operation and modify parameters so that the tool is in constant contact with the part.

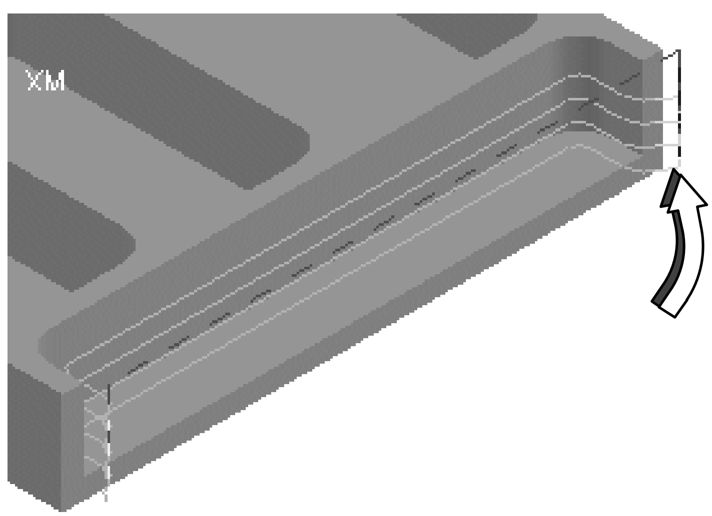
❑ Continue with the part file *****_hsm_1.prt.**

❑ Double click on the operation, **zlevel_profile.**



The ZLEVEL_PROFILE dialog is displayed.

❑ **Replay** the tool path.



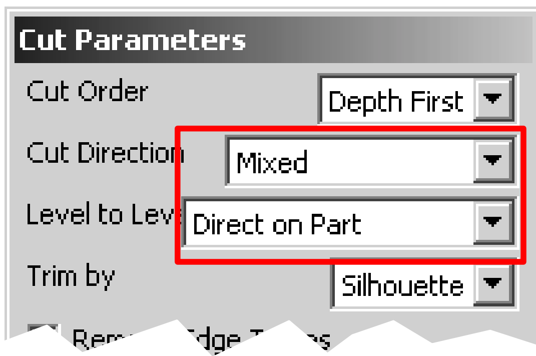
Notice that the tool retracts and engages for each cut level.

You will now change the cut parameters to allow the cutter to move directly from one cut level to the next without engaging, traversing and retracting.

❑ Refresh the graphics display.

❑ Choose **Cutting**.

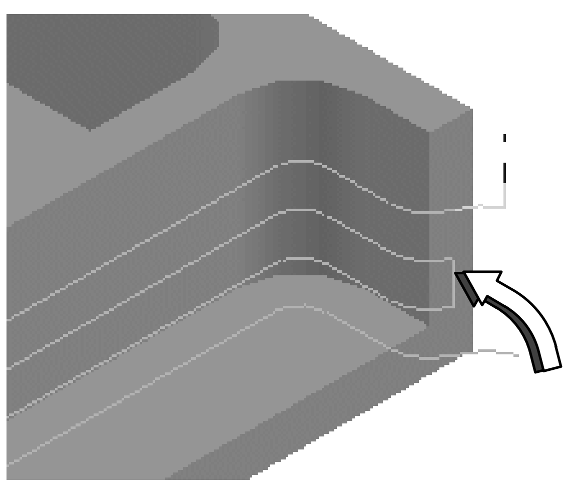
❑ Set the **Cut Direction** option to **Mixed** and the **Level to Level** option to **Direct on Part**.



❑ Choose **OK** to accept the Cut Parameters.

Step 2 Create the tool path.

❑ Choose the **Generate** icon and generate the tool path.



Notice how the cutting tool engages the part, feeds down the wall of the part to get to the next level, and alternates the direction of cut from one level to the next.

❑ Refresh the graphics display.

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

❑ **Save** the part file.

You have completed this activity and will be using this part file in the activity on NURBS.

Nurbs

Many machine tool controllers have the option of creating non uniform rational B-splines, commonly referred to as NURBS. NURBS output will cause the tool to drive along these spline curves (degree 3, cubic splines) instead of line/arc segments. The result is a very smooth and accurate surface cut (particularly on contoured surfaces) that *may* result in reduced output for the machine tool controller.

Not all controllers can handle NURBS, and those that do normally use different formats. Currently, Fanuc, Seimens, Heidenhain, or Fidia controllers are support for Nurbs input.

NURBS are available only for fixed axis machining methods of Fixed Axis Surface Contouring, Planar Milling, and Cavity Milling.

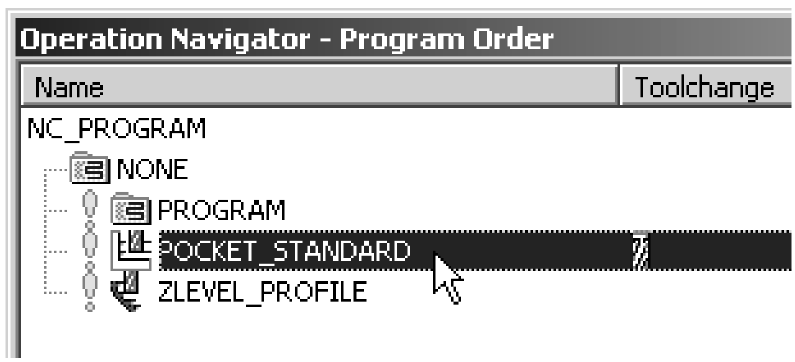
Activity 7–3: NURBS

In this activity, you will use the part file from the previous activity and explore the use of the NURBS option.

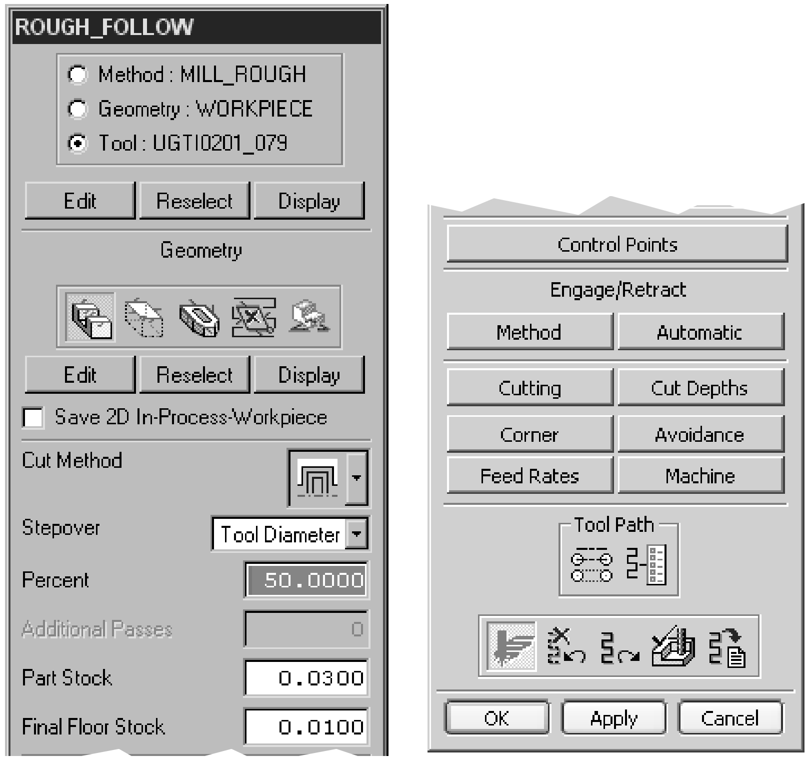
Step 1 Edit the existing operation and modify parameters suitable for the output of NURBS data.

❑ Continue with the part file *****_hsm_1.prt.**

❑ Double click on the operation, **pocket_standard.**

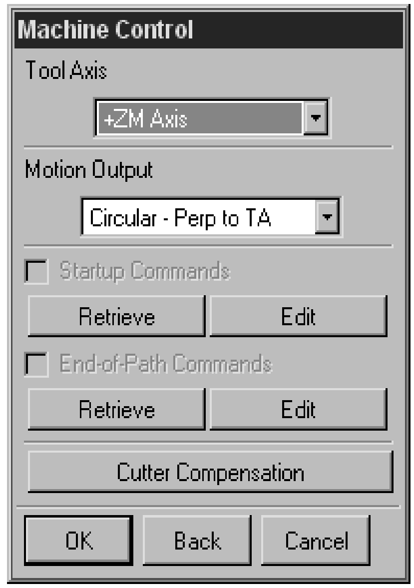


The ROUGH_FOLLOW dialog is displayed.

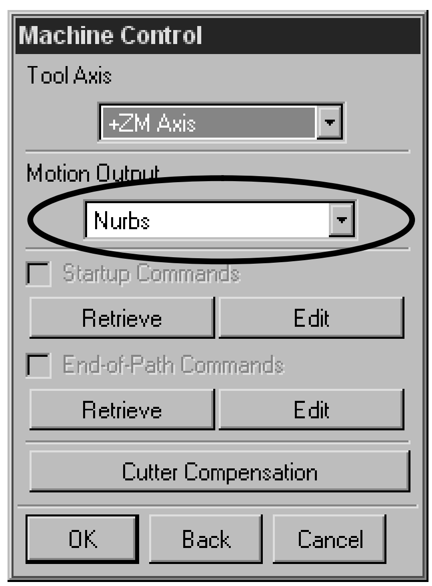


❑ Select **Machine**.

The Machine Control dialog is displayed.

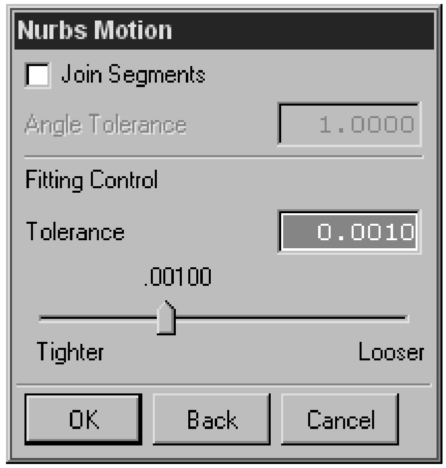


❑ Choose **Nurbs** from the **Motion Output** pull down menu.



7

The Nurbs Motion dialog is displayed.



Join Segments determines whether or not the separate curve segments join together into one curve to form a single Nurbs for each cut.

The finished part should not deviate from the design geometry more than the specified Intol/Outtol if the defined **Fitting Control Tolerance** is within the Intol/Outtol band. If a smoother path is desired, specify a tighter Fitting Control Tolerance. When this option is used, the tool displays only at the beginning and end of each cutting pass regardless of the specified Display Frequency.

In addition to determining the degree of smoothness applied to angles (as described above), Fitting Control Tolerance also determines how accurately the tool path follows the NURBS. A tighter tolerance (defined by a smaller value) causes the tool path to follow the NURBS more accurately.

The Fitting Control Tolerance may be specified by either keying in a value or by moving the slider bar.

Fitting Control Tolerance and Join Segments can be used together to achieve the desired "polished" finished surface by not only smoothing the surface, but also avoiding sharp turns and irregular movements. By using a tight Fitting Control Tolerance and an Angle Tolerance of less than five degrees, you can fit to a smooth single NURBS tool path.

Angle Tolerance is available when Join Segments is toggled ON. This option allows you to determine which angles formed between the joined curves forming the NURBS will be smoothed. Angles smaller than or equal to the specified Angle Tolerance will be smoothed. Angles greater than the specified Angle Tolerance will not be smoothed. To obtain reliable results, you should use an Angle Tolerance of 5 or less degrees.

Angles Smaller than the Specified Angle Tolerance are Smoothed.

Larger Fitting Control Tolerance Creates a Smoother Blend.

- ❑ Choose **Join Segments** (checked) from the **Nurbs Motion** dialog.

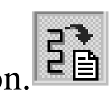
You will accept the other options as defaults.

- ❑ Choose **OK,** twice**.**

Step 2 Create the tool path.

- ❑ Choose the **Generate** icon and generate the tool path.

Examine the output.

- ❑ List the tool path output by selecting the **Listing** icon. 

Notice the tool path listing for NURBS, it will be similar to the following:

```
GOTO/−1.9522,0.3426,−0.3950
GOTO/−1.9522,0.6165,−0.3950
GOTO/−1.9589,0.6739,−0.3950
NURBS/
KNOT/1.0000000
CNTRL/−1.9721,0.7307,−0.3950
CNTRL/−2.0055,0.7816,−0.3950
CNTRL/−2.0522,0.8165,−0.3950
PAINT/COLOR,37
RAPID
```

- ❑ **Close** the part file.

You have completed this activity and the lesson.

7

SUMMARY

High Speed Machining technology has shown dramatic increases in productivity and improved part quality. The characteristics of HSM such as high spindle speeds, fast feed rates, light cuts, smooth tool movement and constant volume removal are obtainable through various parameter settings.

In this lesson you:

- Explored various parameters within operations that lend themselves to High Speed Machining concepts
- Generated operations, using parameters that were conducive to HSM
- Explored parameters and techniques for generating NURBS output

NC Assistant
Lesson 8

PURPOSE

This lesson will familiarize you with the functionality of the NC Assistant. The NC Assistant is a very useful tool used to analyze corner and fillet radii, draft angles and cutting depths. Analyzing these features will aid you in the determination of the tool configuration needed to cut the part.

OBJECTIVES

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

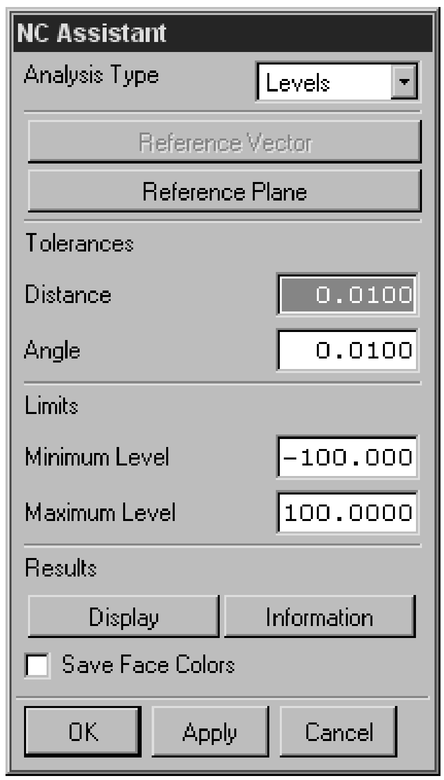
- Use the NC Assistant
- Determine cutter geometry based on information feedback from the NC Assistant

This lesson contains the following activities:

Overview of the NC Assistant

The NC Assistant is an analysis tool that assists you in the selection of the proper tool needed for machining various geometric configurations. The Assistant provides you with information on planar levels (depths), corner radii, fillet radii, and draft angles. The information provided is color coded for easy detection on the model (model must be shaded) along with an information listing giving pertinent data concerning the geometry being analyzed. This information is useful in deciding the cutting tool parameters which are necessary to machine the selected part geometry.

The NC Assistant is activated once you are in the Manufacturing Application from the Main Menu bar by choosing **Analysis→NC Assistant**.



You select the geometry to be analyzed and then set the various parameters. Currently four Analysis Types are available: Levels, Corner Radii, Fillet Radii, and Draft Angles.

When analyzing Levels, information is provided on the distance of planar levels from a reference plane. If a reference plane is not specified, the MCS is used as a basis. This information can be used as an aid in determining the length of the tool(s) that is needed.

Analysis of Corner Radius provides information on the minimum corner radii of the faces selected. This information will aid you in determining the proper tool diameter.

Analysis of Fillet Radius will show the minimum fillet radius of the selected faces with reference to a vector. This information will help you to determine the tool nose radius, required if any.

Analysis of Draft Angle will determine the slopes of the faces selected with reference to a specified vector. This information will help you to determine the taper of the tool (also can be a quick aid in determining various areas of draft on a casting or injection mold).

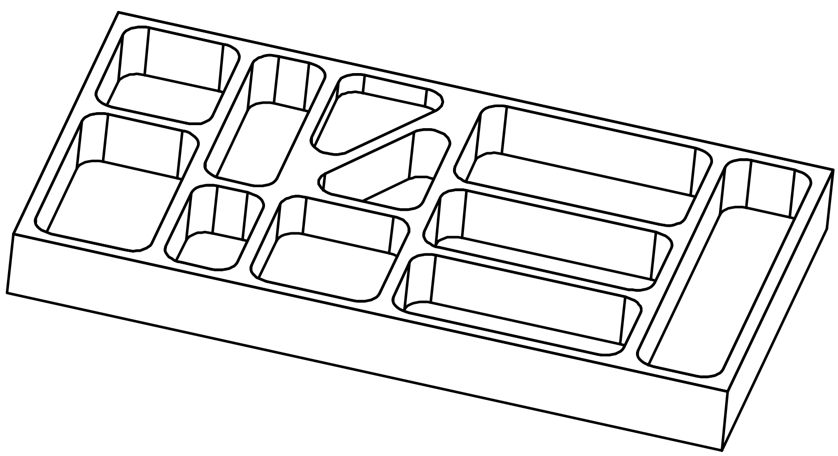
When analyzing the various types, limits can be set. For example, if you wanted to check for all corner radii that were greater than .500 inches and less than .750 inches, values can be set for the minimum of .500 and maximum of .750.

Activity 8—1: Using the NC Assistant

In this activity, you will become familiar with the various features of the NC Assistant. You will use the NC Assistant to determine the length, cutting diameter and corner radius of the tool(s) necessary to finish all pockets of the part. Since you will only be analyzing this part, there is no need to rename or save it.

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

❑ Open the part **ama_nc_assistant.prt.**



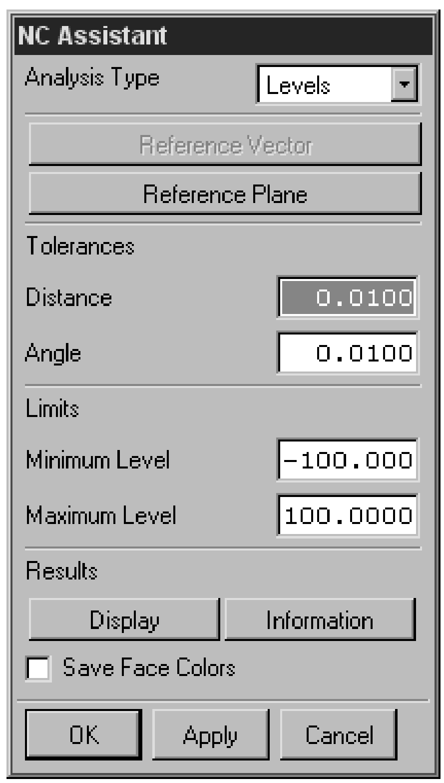
❑ Enter the **Manufacturing** Application.

Step 2 Activate the NC Assistant.

You will need to determine the tool configuration(s) necessary to finish machine all pockets in this part. Visually, it is difficult to detect draft, if any on the pocket walls. It would be cumbersome to verify all corner radii and floor depths. To make your job easier, you will use the NC Assistant, to analyze the geometry configuration. You will now activate the NC Assistant.

❑ From the Main Menu Bar select, **Analysis → NC Assistant**.

The NC Assistant Dialog is displayed.



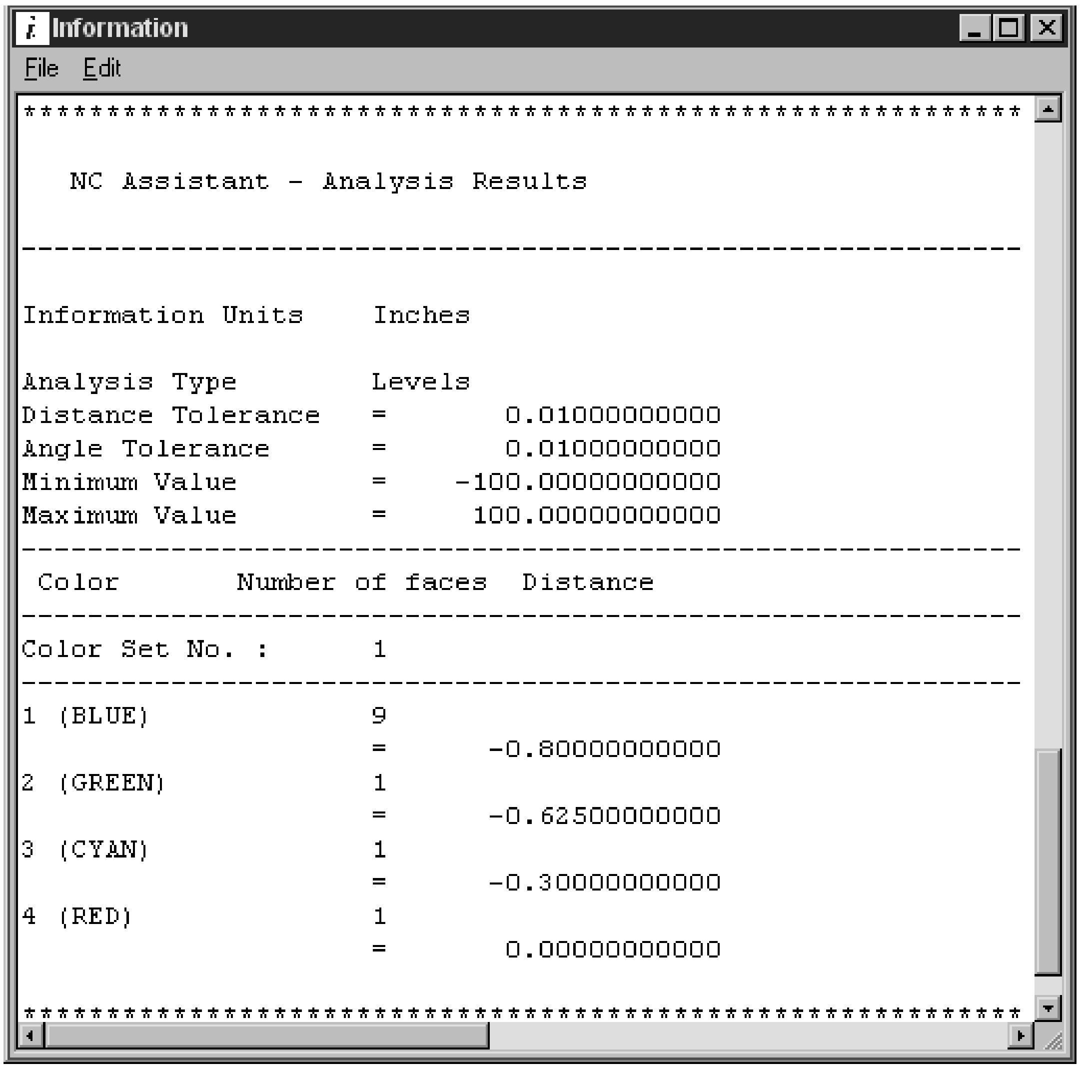
8

Step 3 Use the NC Assistant to determine cutter length.

The first item for consideration is to determine the length of the cutter that will be needed. For this determination you will use the NC Assistant to determine the various Levels. If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will accept the defaults for all parameters and select the entire part for faces.

- ❑ Use **MB1** and drag a rectangle around the entire part.
- ❑ Choose **Apply**.

An Information Window is displayed containing the results of the Level analysis.



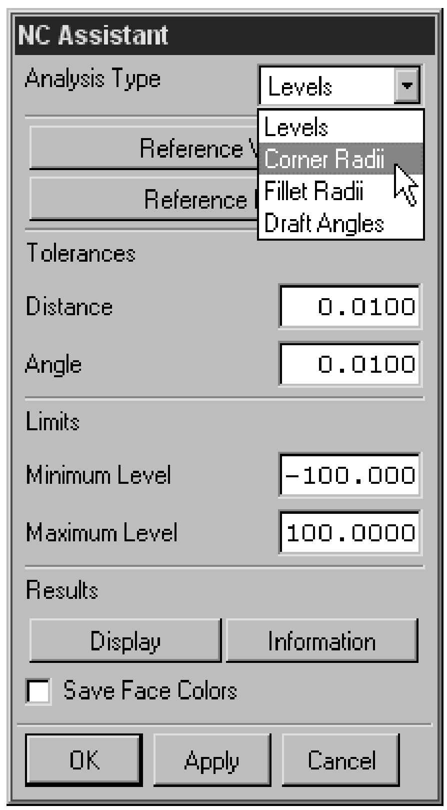

```
*************************************************************

   NC Assistant - Analysis Results

-------------------------------------------------------------

Information Units    Inches

Analysis Type        Levels
Distance Tolerance   =       0.01000000000
Angle Tolerance      =       0.01000000000
Minimum Value        =    -100.00000000000
Maximum Value        =     100.00000000000
-------------------------------------------------------------
 Color       Number of faces  Distance
-------------------------------------------------------------
Color Set No. :      1
-------------------------------------------------------------
1 (BLUE)             9
                     =      -0.80000000000
2 (GREEN)            1
                     =      -0.62500000000
3 (CYAN)             1
                     =      -0.30000000000
4 (RED)              1
                     =       0.00000000000

*************************************************************
```

Notice the color set number, the color associated with the color set number, number of faces and distance. Compare the colors with those now displayed in the graphics window. By examining the distance values, the deepest level or floor is located .800 below the top of the part (blue). Therefore the length of the tool is .800 plus whatever clearance value that you would want to use.

Step 4 Use the NC Assistant to determine cutter diameter.

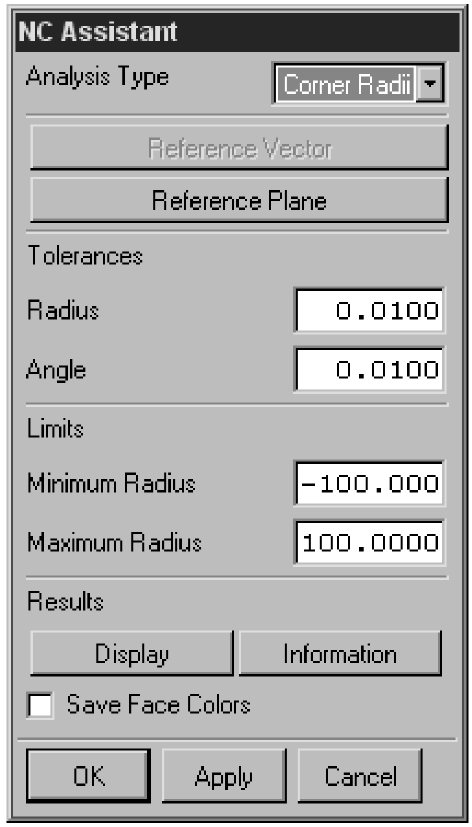
The second item for consideration is to determine the diameter of the cutter that will be needed. For this determination you will use the NC Assistant to determine the various corner radii of the part.

❑ Choose **Corner Radii** from the **Analysis Type** pull down menu.



8

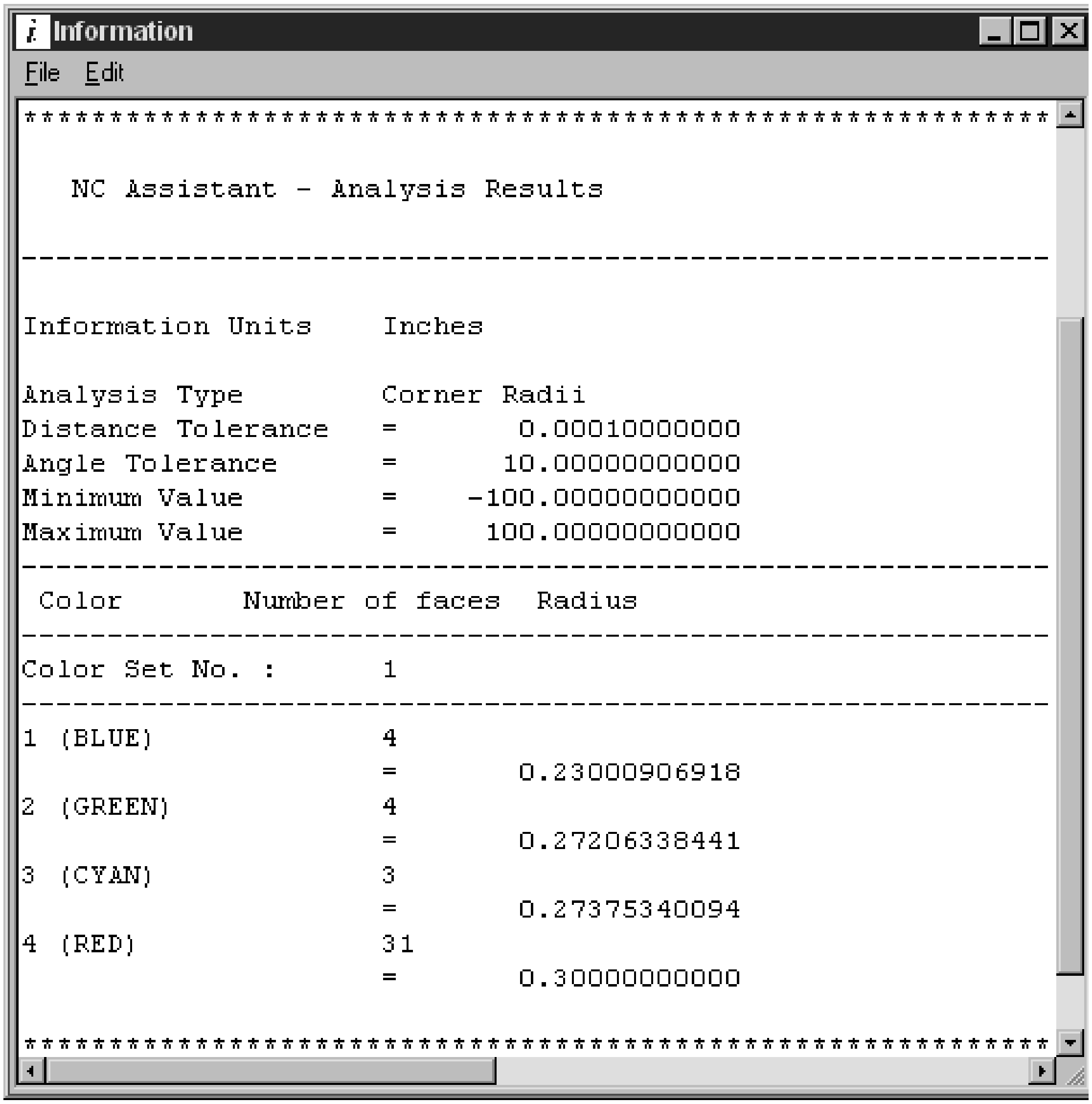
The dialog for Corner Radii analysis is displayed.



Notice the limits for **Minimum** and **Maximum Radius**. If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will change the Radius and Angle tolerances and select the entire part for faces.

- ❑ Change the **Radius** value to **.0001.**
- ❑ Change the **Angle** value to **10.0**.
- ❑ Use MB1 and drag a rectangle around the entire part.
- ❑ Choose **Apply**.

An Information Window is displayed containing the results of the Corner Radii analysis.




```
****************************************************************

   NC Assistant - Analysis Results

----------------------------------------------------------------

Information Units     Inches

Analysis Type         Corner Radii
Distance Tolerance    =        0.00010000000
Angle Tolerance       =       10.00000000000
Minimum Value         =     -100.00000000000
Maximum Value         =      100.00000000000
----------------------------------------------------------------
 Color        Number of faces   Radius
----------------------------------------------------------------
Color Set No. :       1
----------------------------------------------------------------
1 (BLUE)              4
                      =        0.23000906918
2 (GREEN)             4
                      =        0.27206338441
3 (CYAN)              3
                      =        0.27375340094
4 (RED)               31
                      =        0.30000000000

****************************************************************
```

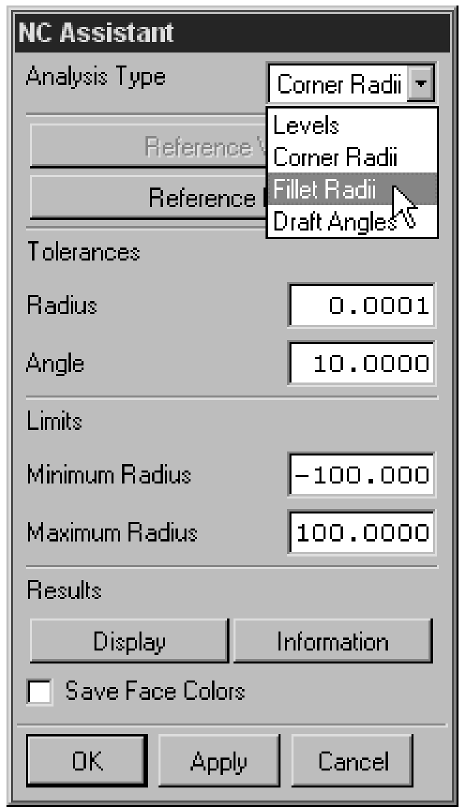
Notice the color set number, the color associated with the color set number, number of faces and radius. Compare the colors with those now displayed in the graphics window. By examining the corner radii values, the largest is .300 (red), the smallest .230, the closest standard size end mill required would be .4375 inches.

8

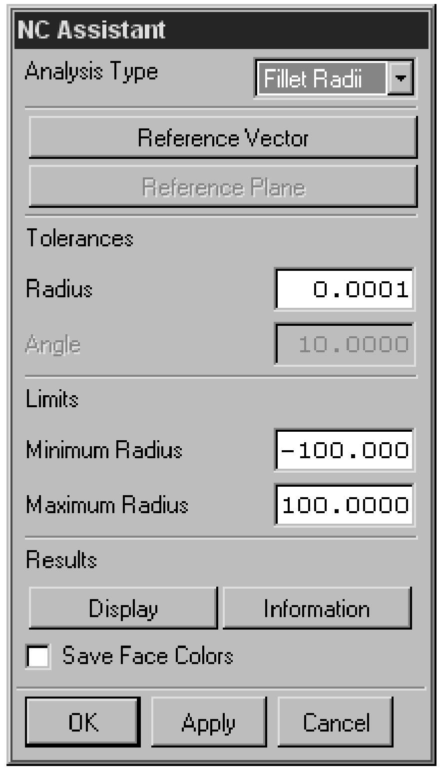
Step 5 Use the NC Assistant to determine cutter corner radius.

The third item for consideration is to determine the corner radius of the cutter that will be needed. For this determination you will use the NC Assistant to determine the various fillet radii of the part.

❑ Choose **Fillet Radii** from the **Analysis Type** pull down menu.



The dialog for Fillet Radii analysis is displayed.



If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will accept all defaults and select the entire part for faces.

❑ Use MB1 and drag a rectangle around the entire part.

❑ Choose **Apply**.

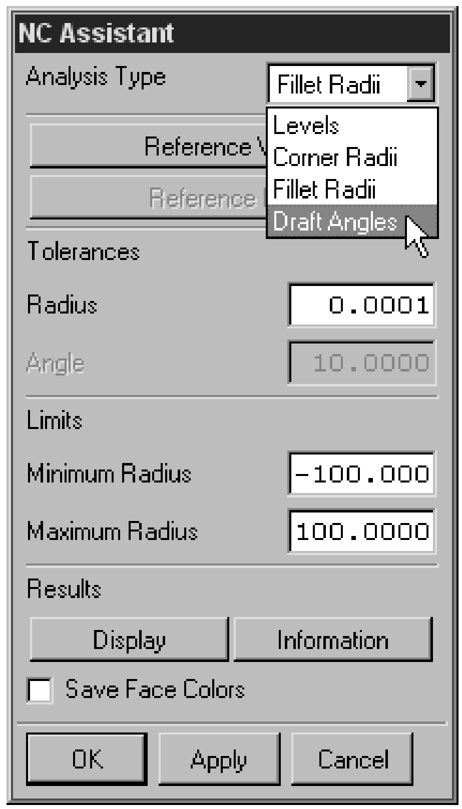
An Information Window is displayed containing the results of the Fillet Radii analysis.

Notice that there were no Fillet Radii displayed.

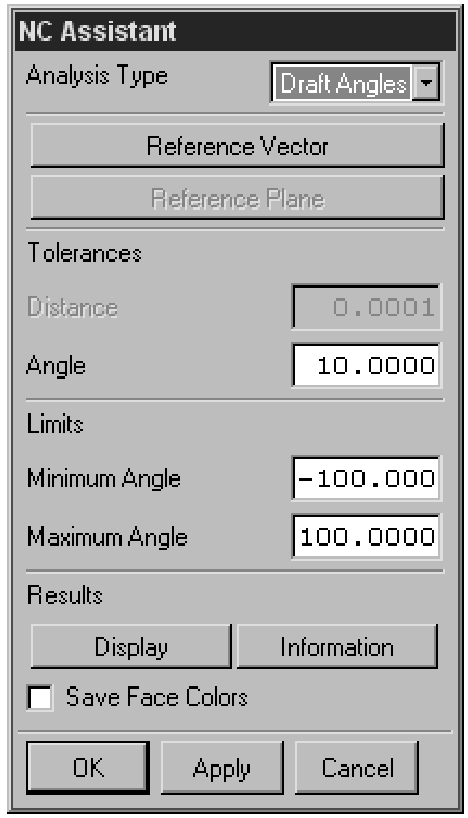
Step 6 Use the NC Assistant to determine draft angle on walls.

The fourth item for consideration is to determine any draft angles on the part that are machinable through the use of an angle cutter. For this determination you will use the NC Assistant to determine the draft angles.

❑ Choose **Draft Angles** from the **Analysis Type** pull down menu.



The dialog for Draft Angles analysis type is displayed.



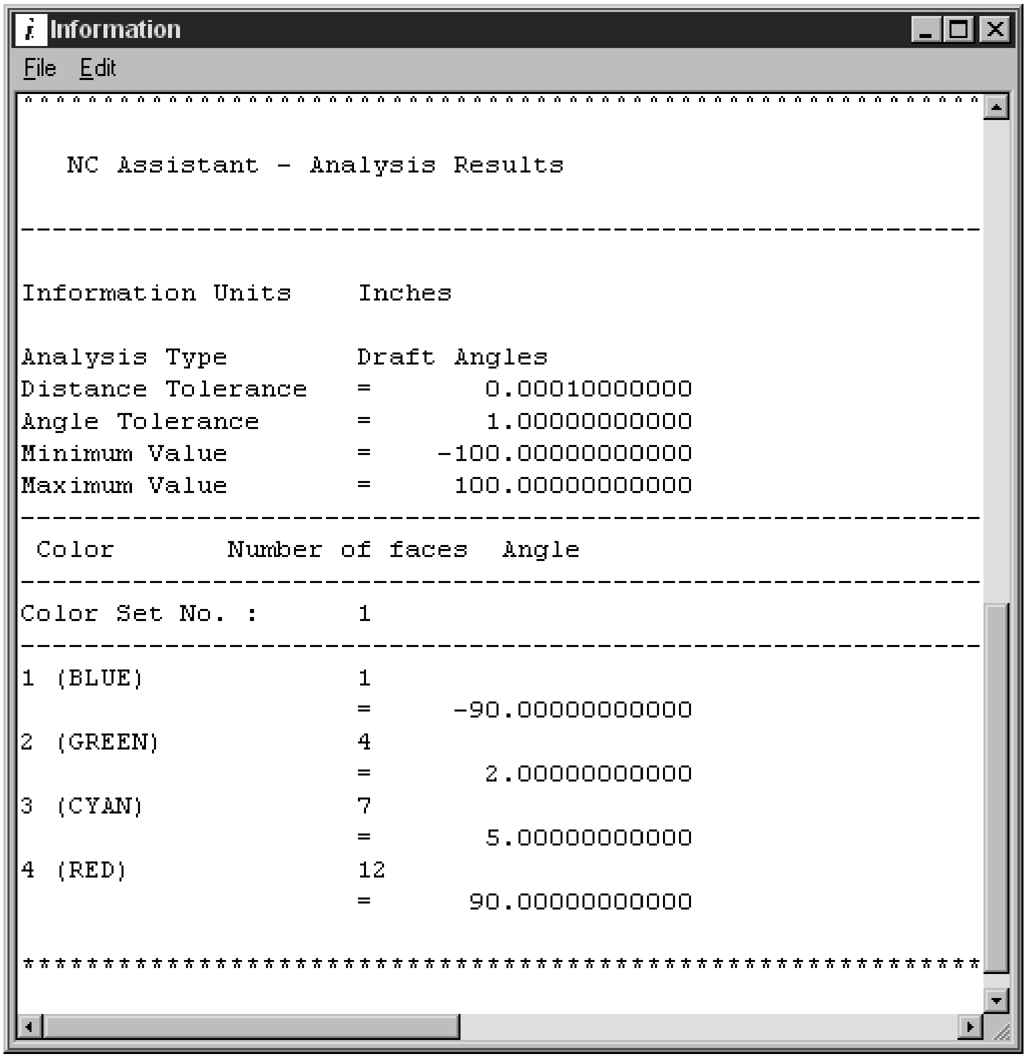
If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will change the **Angle** tolerance to 1.0 degree, accept all other defaults and select the entire part for faces.

❑ Under Tolerances, change **Angle** to **1.0**

❑ Use MB1 and drag a rectangle around the entire part.

❑ Choose **Apply**.

An Information Window is displayed containing the results of the Draft Angles analysis.



```
Information
File  Edit
***************************************************************

   NC Assistant - Analysis Results

---------------------------------------------------------------

Information Units     Inches

Analysis Type         Draft Angles
Distance Tolerance    =        0.00010000000
Angle Tolerance       =        1.00000000000
Minimum Value         =     -100.00000000000
Maximum Value         =      100.00000000000
---------------------------------------------------------------
 Color        Number of faces   Angle
---------------------------------------------------------------
Color Set No. :       1
---------------------------------------------------------------
1 (BLUE)              1
                      =      -90.00000000000
2 (GREEN)             4
                      =        2.00000000000
3 (CYAN)              7
                      =        5.00000000000
4 (RED)               12
                      =       90.00000000000

***************************************************************
```


Notice the color set number, the color associated with the color set number, number of faces and angle. Compare the colors with those now displayed in the graphics window. The green faces represent 2 degrees draft, the cyan 5 degrees. All other walls have no draft. Two different angle cutters would be necessary or a ball tool could be used to profile the draft angle onto the wall.

- ❑ **Cancel** the NC Assistant dialog and dismiss the Information window.

- ❑ **Close** the part.

You are finished with this activity and also with the lesson.

SUMMARY

The NC Assistant is an efficient tool to use for analyzing part geometry for various corner radii, fillet radii, floor depths and draft angles. This information is beneficial in the determination of cutter parameters used for cutting your part.

In this lesson you:

- Became familiar with the functionality of NC Analysis
- Performed various analysis functions which were used to determine cutter length, diameter, corner radius and draft angle

8

Templates

Lesson 9

PURPOSE

Templates contain predefined parameters that enable you to quickly and easily define new operations and group objects tailored to your specific needs. Templates eliminate the tedious task of redefining parameters from a set of standard defaults each time you define a new operation or group object.

OBJECTIVES

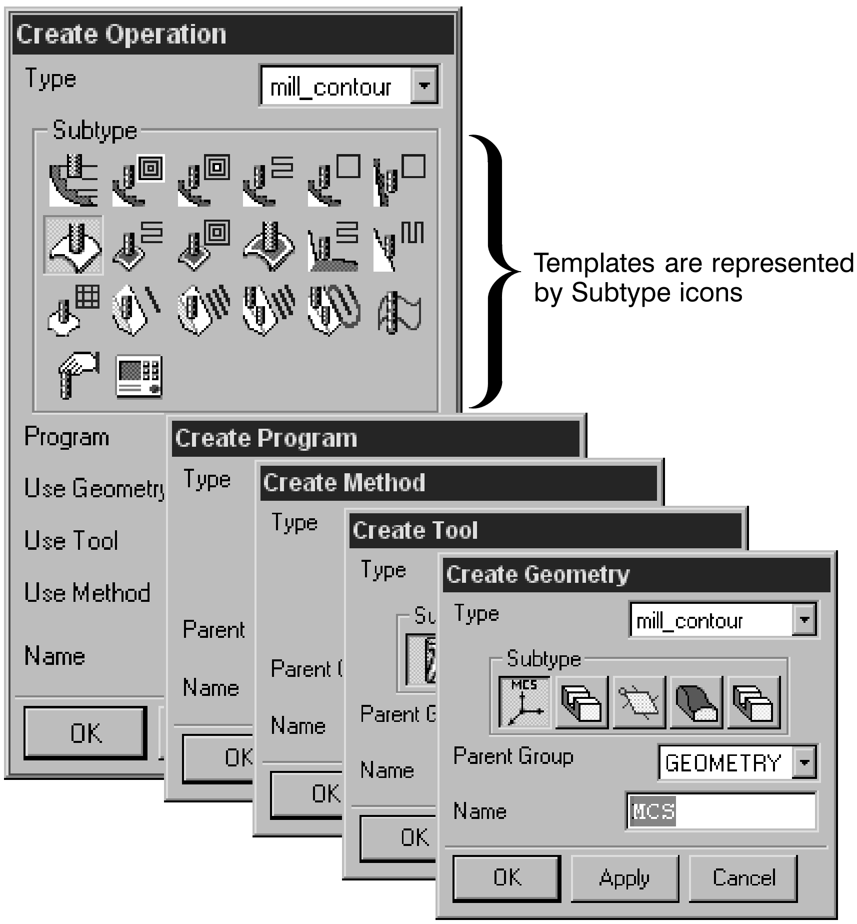
Upon completion of this lesson, you will be able to:

- Create and use templates
- Use a sequence of operations template to create numerous operations automatically for similar parts

This lesson contains the following activities:

Templates Overview

Templates are operations and group objects within part files that contain predefined parameters that allow you to define new operations and groups quickly and easily for specific tasks. They eliminate the laborious procedure of redefining parameters from a standard set of system defaults each time a new operation or group is defined.

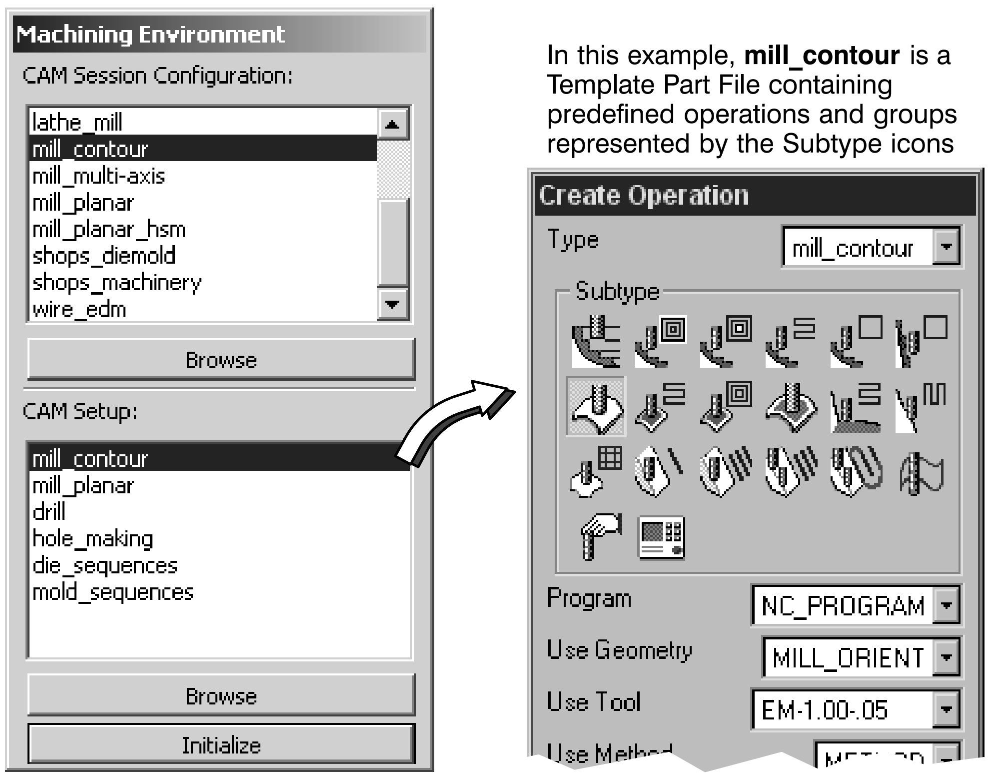


Templates are represented as subtype icons which are available in the various Create dialogs (also referred to as object dialogs). They define operation parameters including numerical values such as part Intol/Outol, feed rates, etc. and can display customized object editing dialogs.

In addition, templates determine the operations and groups that initially appear in the Operation Navigation Tool when you first enter the Manufacturing Application.

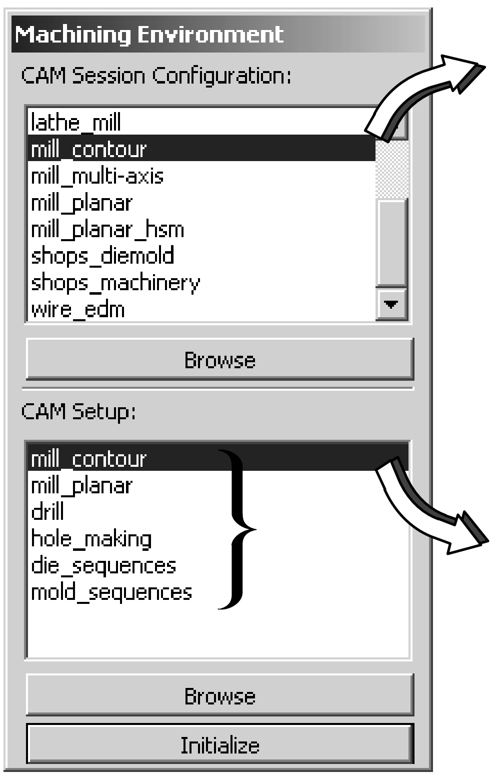
Template Part Files

A template part file contains a collection of predefined operations and groups (referred to as templates) from which you may select. Template part files appear in the CAM Setup portion of the Machining Environment dialog and are listed as Type options in the operation and group creation dialogs.



Template Sets

A Template Set is a collection of template part files. Template Sets are specified from within the Configuration files. Basic Template Sets are provided and maintained in every software release. Template Sets may also be created, customized and maintained by users. Template Sets created by users may use any valid naming convention and may be located in any directory. They must have a .opt file extension. Supplied Template Sets reside in the \mach\resource\template_set directory.



These are files with ".dat" extensions that contain the location of template files for: operations, documentation, posts, user defined events and library locations. These files are located in the \mach\resource\configuration directory. Example: mill_contour.dat.

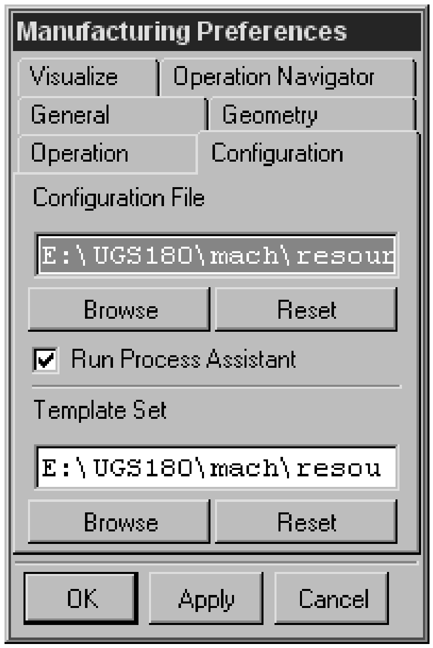
```
TEMPLATE_OPERATION,${UGII_CAM_TEMPLATE_
TEMPLATE_DOCUMENTATION,${UGII_CAM_SHOP_
TEMPLATE_POST,${UGII_CAM_POST_DIR}templ
USER_DEFINED_EVENTS,${UGII_CAM_USER_DEF_
TEMPLATE_CLSF,${UGII_CAM_TOOL_PATH_DIR}
LISTING_FORMAT,${UGII_CAM_TOOL_PATH_DIR
LIBRARY_TOOL,${UGII_CAM_LIBRARY_TOOL_AS
LIBRARY_MACHINE,${UGII_CAM_LIBRARY_MACH
LIBRARY_FEEDS_SPEEDS,${UGII_CAM_LIBRARY_
LIBRARY_PART_MATERIAL,${UGII_CAM_LIBRAR
LIBRARY_TOOL_MATERIAL,${UGII_CAM_LIBRAR
LIBRARY_CUT_METHOD,${UGII_CAM_LIBRARY_F
WIZARD,${UGII_CAM_WIZARD_DIR}wizard_mil
```

This is the contents of a template set. It is a list of part files that contain parameters used to generate the option. The template set is a file with a ".opt" extension. These files are located in the \mach\resource\template_set directory. Example: mill_contour.opt

```
${UGII_CAM_TEMPLATE_PART_ENGLISH_
${UGII_CAM_TEMPLATE_PART_ENGLISH_
${UGII_CAM_TEMPLATE_PART_ENGLISH_
${UGII_CAM_TEMPLATE_PART_ENGLISH_
${UGII_CAM_TEMPLATE_PART_ENGLISH_
##
${UGII_CAM_TEMPLATE_PART_METRIC_D
${UGII_CAM_TEMPLATE_PART_METRIC_D
${UGII_CAM_TEMPLATE_PART_METRIC_D
${UGII_CAM_TEMPLATE_PART_METRIC_D
${UGII_CAM_TEMPLATE_PART_METRIC_D
```

9

You can specify the Template Set by selecting **Preferences→Manufacturing** and then selecting the Configuration tab. You can browse and or specify the directory and name of where the template set is located. When you specify a template set, the available Type options in the object dialogs change to reflect the new template set files. You may also select template files with the browse option in the Type option list.



When creating a new operation, you specify the template part by choosing a Type and then the template by choosing an operation type (subtype icon) in the Create Operation dialog. The template parameters are then copied into the new operation.

When creating a new Group, you specify the template part by choosing a Type and then specify the Group Template by choosing the appropriate subtype icon in the Create Program, Create Tool, Create Geometry or Create Method Group dialog.

9

Creating and Using Template Sets

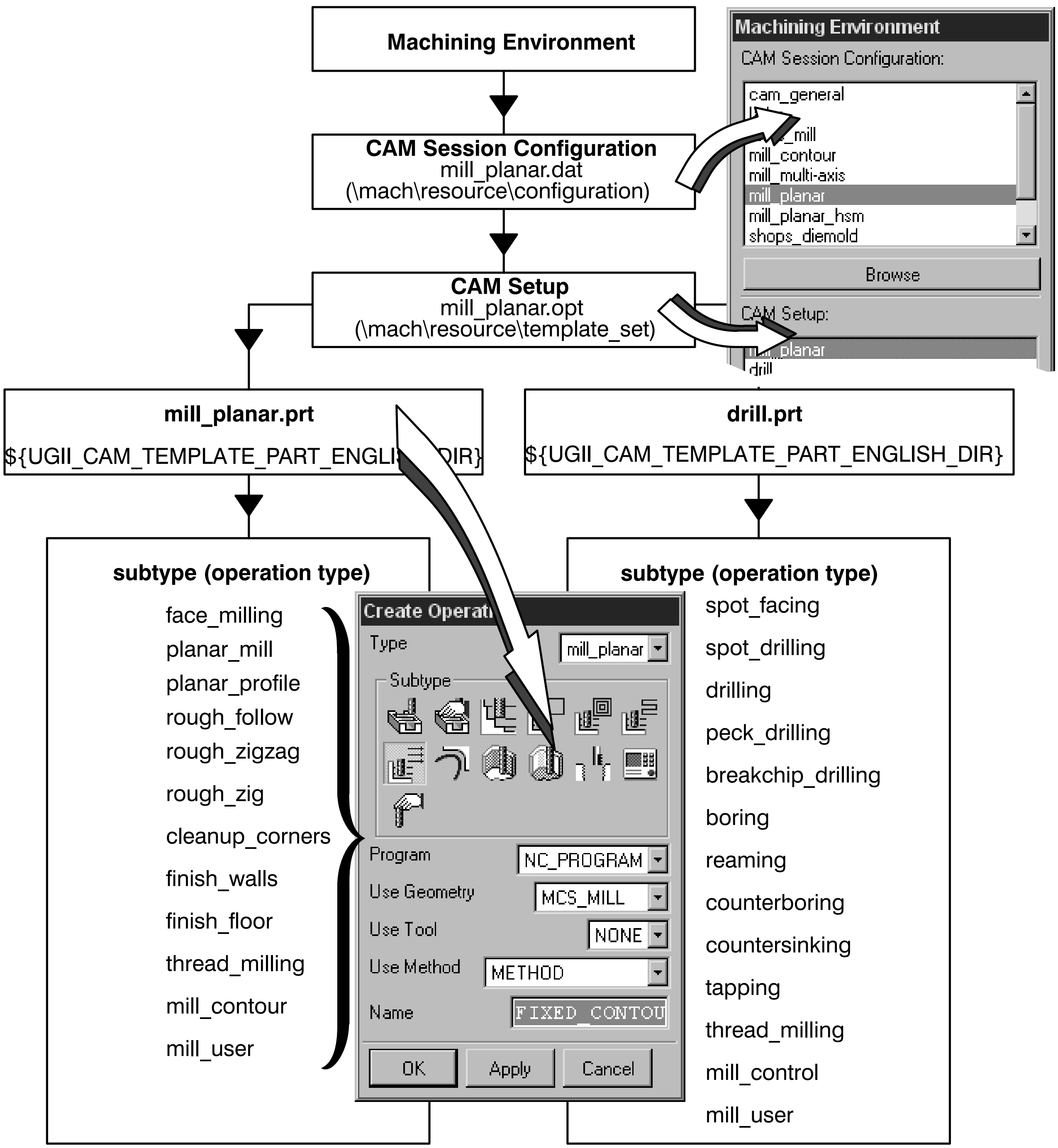
Template Sets make template part files available for selection from within dialogs. This prevents you from having to browse for individual part files. A Template Set is an text file (.opt extension) which list the part files which contain the templates that are to be utilized. The default Template Set is determined by the Configuration file. There is a limit of 30 part files in a Template Set.

When creating operations or groups in part files that are members of a Template Set you may also want to customize the operation dialogs so that only specific options are displayed creating operations using the template (be sure that **MB3→Object→Template Setting→Template** has been toggled to ON).

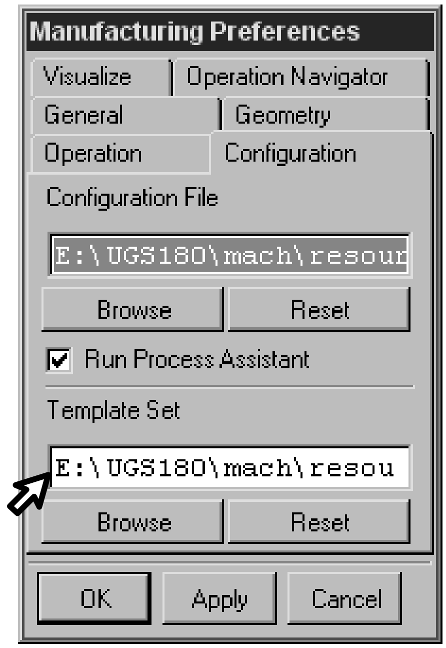
Once the template files have been created and saved, they are then grouped together into a Template Set. A Template Set is simply an ASCII text file that contains the directory path and file name of each template file in the set. This file will always have a .opt extension associated with it.

The following flowchart, for the **mill_planar** default Template Set, shows the relationship between various files used in Template Sets and their dialogs.

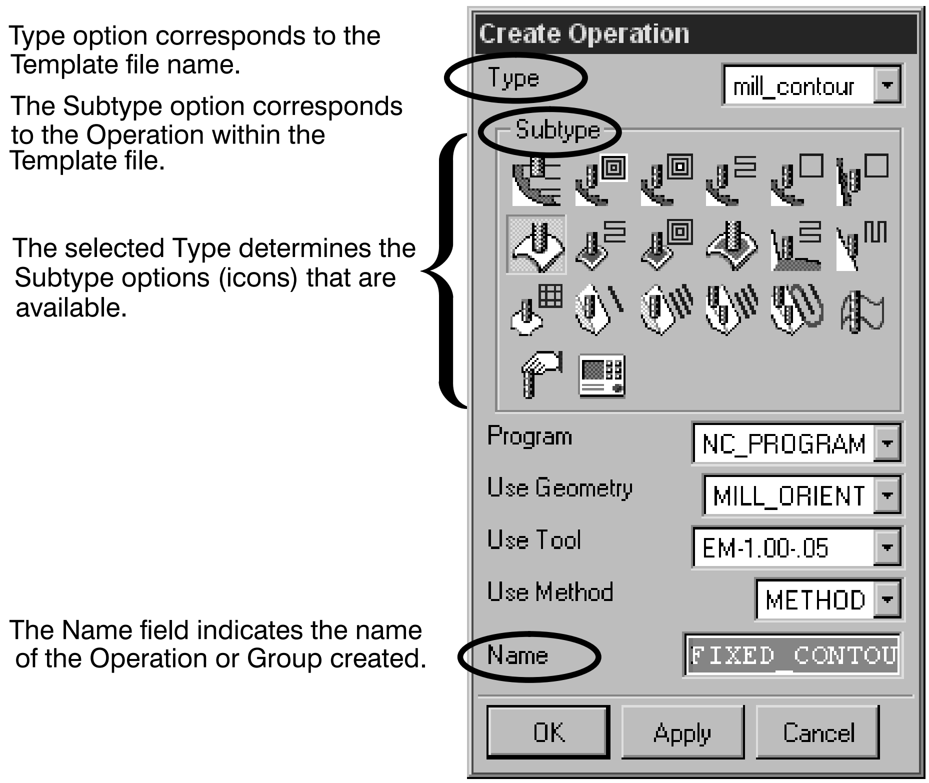
Template Set / Dialog Relationships (mill_planar)



Once a template set has been created, you may select which template set to use when creating new operations by selecting **Preferences→ Manufacturing,** select the **Configuration** tab, and then selecting the **Browse** button under **Template Set**. You can then browse to the location of your template set.

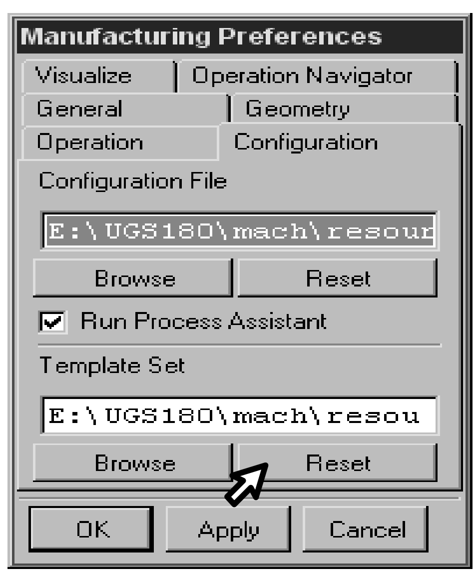


The Template Set that has been selected determines the Type and Subtype options which appear in the Create Operation and Group dialogs.



The creation of tools within an operation will occur only if the template has Load with Parent set to ON. If the template loads a tool with the same name but different parameters as a tool in the current part file, a new tool with different name is created with the new operation.

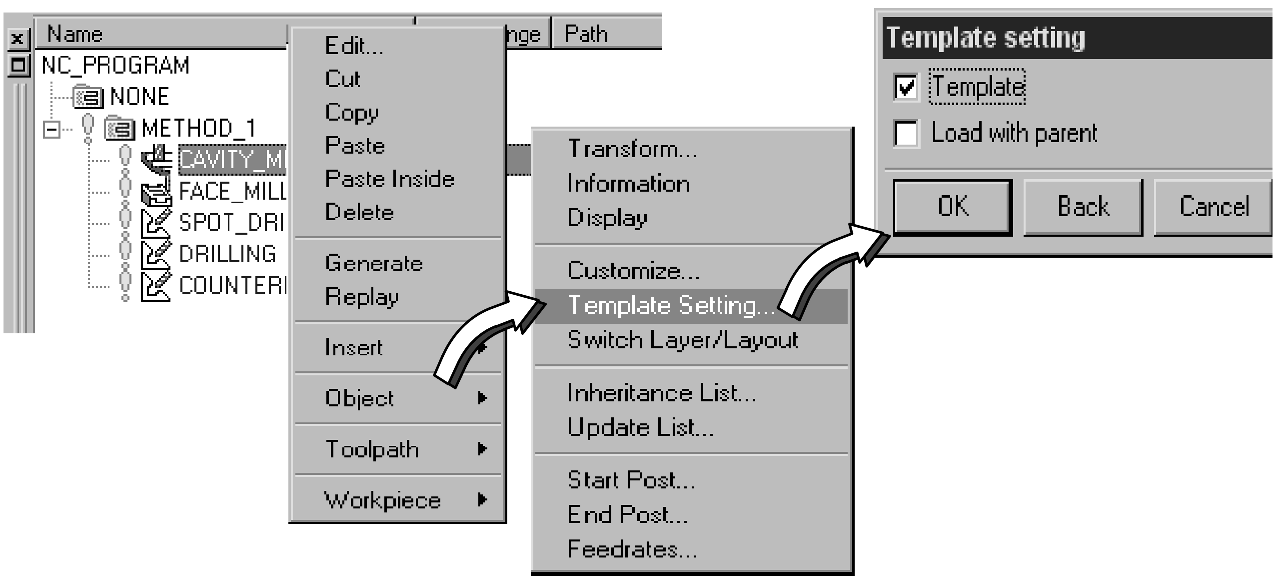
You can reset the current Template Set back to the default operation Template Set by using the **Reset** button in the Manufacturing Preferences Configuration tab dialog. The default Template Set is determined by the Configuration file.



Creating a Template

As discussed earlier, a template is a predefined operation or group within a part file. Template files can be new or can be copied, customized and/or renamed from existing template files.

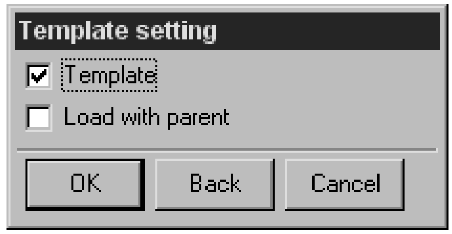
Template files can contain many operations and groups. You can specify which operations and groups are used as templates by choosing the Template Setting option in the Operation Navigator.



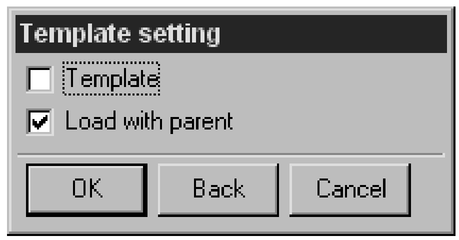
Highlight the desired operations and/or groups in the Operation Navigator and with MB3 choose **OBJECT→Template Setting.** This will display the template setting dialog which contains the Template and Load with Parent toggle settings.

9

If the template option is toggled ON (checked) the highlighted operations or group will be available as a Template whenever the particular part is used as a Template Part file. All operations and groups for which the template option is toggled ON (checked) will appear as subtype icons in the appropriate Create dialogs.



The Load with Parent option allows the determination of which operations and groups that will appear in the Operation Navigator when Setup is initially selected from the Machining Environment dialog. It allows the specification of certain operations and/or groups in addition to the current Parent Group being created. For example, any time that an MCS Geometry group is created, a WORKPIECE Geometry group is also created within the MCS Geometry group when Load with Parent is specified for the WORKPIECE.



Load with Parent allows the automatic loading of operations when creating groups. This is useful when loading a predefined sequence of operations. The operations which are loaded may have four parents (Program, Tool, Method and Geometry) but only are loaded when creating Geometry groups. For example, if you specify Load with Parent in the Machine Tool view, the Parent group, Tool will not load the operations. The Parent group, Geometry will.

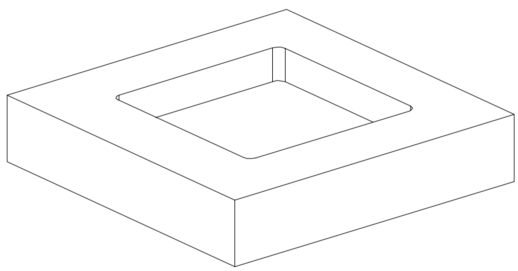
The Load with Parent option also allows the determination of which operations and groups in a template part file will appear in the Operation Navigator when the template part file is specified as a Setup in the Machining Environment dialog. The Setup is the initial template part file which is selected when first entering the Manufacturing Application. When Load with Parent is specified for a continuous sequence of operations and groups beginning at the parent root level, all of the operations and groups are created when the part is selected as a Setup and will appear in the Operation Navigator. An easy way to remember the Load with Parent option function is use to the following analogy: “Whenever you create my Parent group, I will be created also.”

Activity 9–1: Creating a Template

In this activity, you will create rough and finish pocketing operations in an existing part file. You will use this part file as a template for machining pockets with different geometry in the next activity. This activity establishes the template file settings.

Step 1 Opening a part file.

❑ Open **ama_single_pocket.prt.**

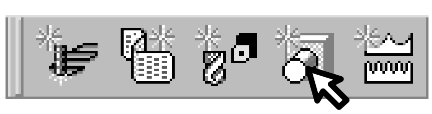


❑ **Save As ***_single_pocket .prt** where *** represents your initials.

❑ Enter the **Manufacturing** Application.

This part file contains several tools and Blank geometry which surrounds the part. The Configuration used is **cam_general** and the Setup is **mill_planar**.

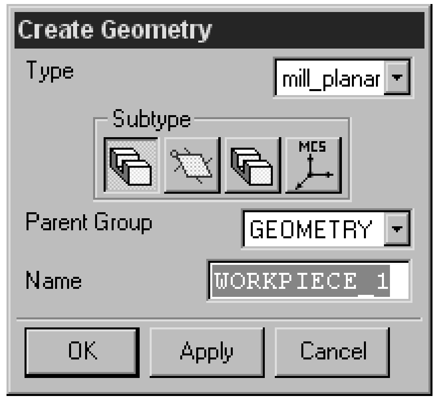
❑ As shown below, select the **Create Geometry** icon from the Create toolbar.



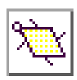
9

The Create Geometry dialog is displayed.

- ❑ Make sure the Type is set to **mill_planar.**

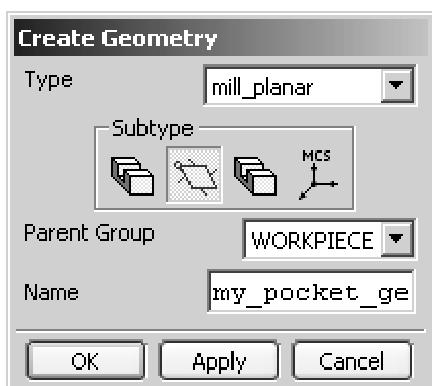


Step 2 Create a Geometry Parent Group to contain a sequence of Operations.

- ❑ From the Create Geometry dialog, choose the **Mill_Bnd** icon. 

- ❑ Choose **WORKPIECE** as the Parent Group.

- ❑ Change the **Name** to **my_pocket_geom**.



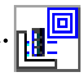
- ❑ Choose **OK** twice.

Step 3 Create three template pocketing operations.

- ❑ As shown below, select the **Create Operation** icon from the Create toolbar.

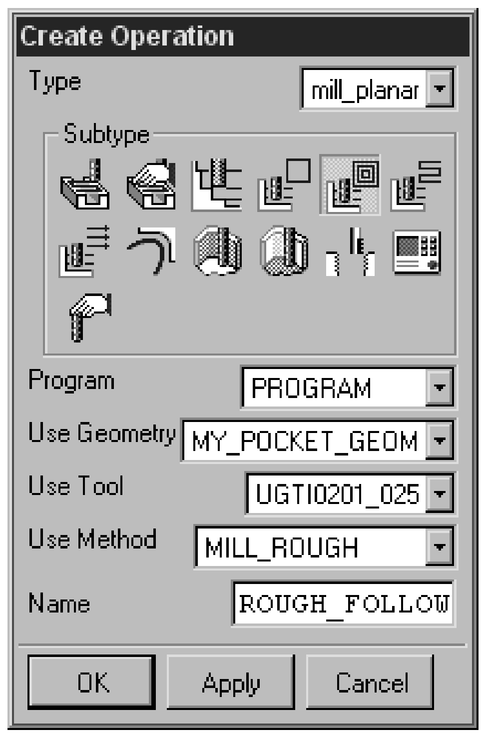
9

The Create Operation dialog is displayed.

❑ Choose the **Rough_Follow** icon. 

❑ Set the following:

- Program: **PROGRAM**
- Use Geometry: **MY_POCKET_GEOM**
- Use Tool: **UGTI0201_025**
- Use Method: **MILL_ROUGH**



❑ Choose **OK**.

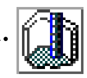
You do not want or need to create a tool path for this Template part file. You will create a tool path in a later part having different geometry and will use this template operation to generate the tool path.

- ❑ Choose **OK**.

  You will now create a template operation for finishing the pocket floors.

- ❑ Select the **Create Operation** icon from the Create toolbar.

  The Create Operation dialog is displayed.

- ❑ Choose the **Finish_Floor** icon. 

- ❑ Change the Tool to **UGTI0201_012.**

- ❑ Change the Method to **MILL_FINISH**.

- ❑ Choose **OK**.

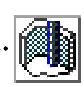
  Again, you do not want or need to create a tool path in this, or any, template part file.

- ❑ Choose **OK**.

  Next, you will create an operation for finishing the pocket walls.

- ❑ Select the **Create Operation** icon from the Create toolbar.

  The Create Operation dialog is displayed.

- ❑ Choose the **Finish_Walls** icon. 

- ❑ Change the Tool to **UGTI0201_011.**

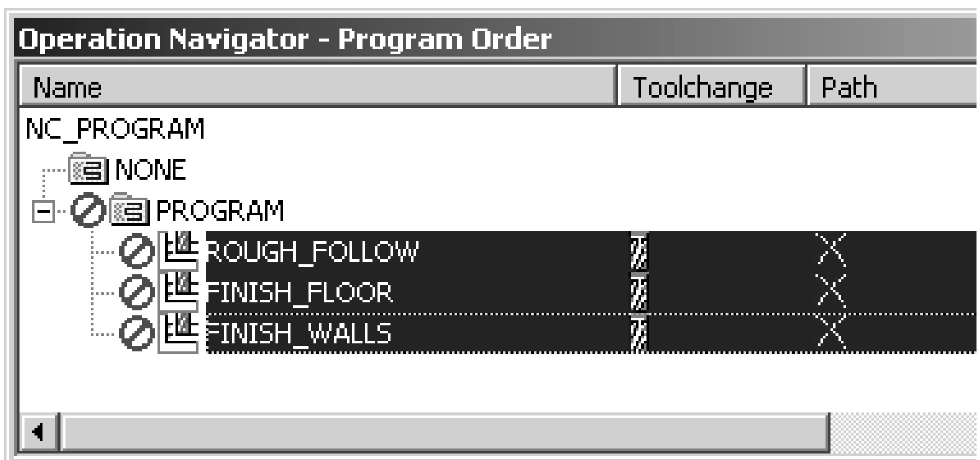
❑ Choose **OK** twice.

You have created a simple sequence to machine pockets. The sequence can be automatically created in a later part file if the Load with Parent and Template setting, for that particular operation, is active. You will do this next.

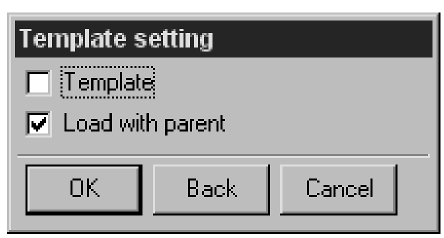
Step 4 Making the sequence available in other part files.

You will want the operations that you have just created to be used in another part. These operations will be created only if the option, Load with Parent, is toggled ON.

❑ Highlight the three operations **ROUGH_FOLLOW, FINISH_FLOOR,** and **FINISH_WALLS.**



❑ Use MB3→**Object**→**Template Setting** to set Template, **Off**; and Load with Parent, **On.**

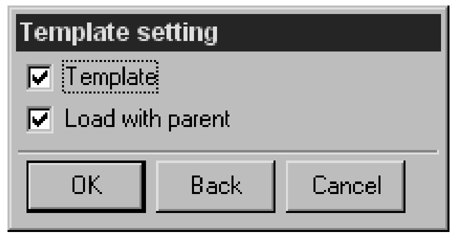


❑ Choose **OK.**

The Load with Parent setting assures that the three operations will be created in a subsequent part file if the Geometry Parent Group (MY_POCKET_GEOM) is created. Next you will make the Parent Group, **MY_POCKET_GEOM**, Load with Parent.

❑ Change the display of the Operation Navigator to the Geometry View.

❑ Choose the **MY_POCKET_GEOM** Parent Group and toggle the Template and Load with Parent, **ON** (check mark the boxes).



❑ Choose **OK.**

The template setting will create an icon in the Create Geometry dialog in part files using this particular template file. The Load with Parent setting will automatically create the Parent Group.

❑ **Save** and **Close** the part file.

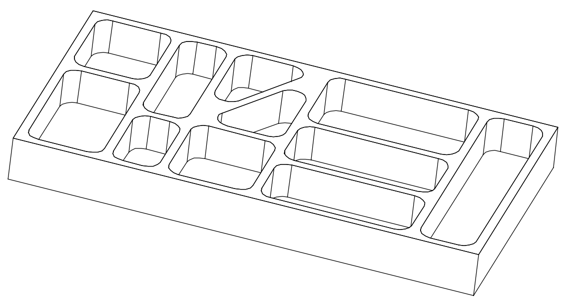
Be sure to save and close the part file since it will be used as a template in the next activity.

Activity 9—2: Using a Template

In this activity, you will use the part file that you just created as a pocketing template file to machine new pocket geometry. Since you specified most options in the template file, including the operations, you will not need to respecify them in the new part file.

Step 1 Open a new part file.

❑ Open **ama_multi_pocket.prt.**



❑ **Save As ***_multi_pocket.prt** where *** represents your initials.

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

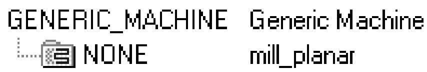
❑ The CAM Session Configuration should be **cam_general.**

❑ The CAM Setup should be **mill_planar**.

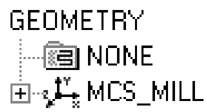
❑ Choose **Initialize**.

❑ In the Program Order View of the Operation Navigator, note that no operations are listed.

❑ Change to the Tool View of the Operation Navigator and note that no tools are present.



❑ Change to the Geometry View of the Operation Navigator and note that the MCS_MILL and Workpiece Parent Groups were created by the selection of mill_planar as the Setup.

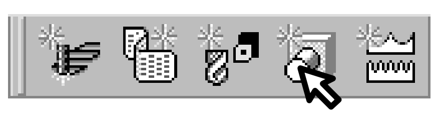


Since you chose mill_planar as the Setup file, it is the current template being used for this part.

Step 2 Change template files.

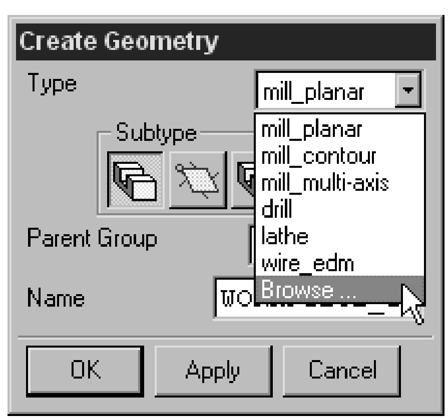
Another method of choosing a template is to change the **Type** in the create dialogs. When using this method you are limited to the options that are available in the template file.

❑ Select the **Create Geometry** icon from the Create toolbar.



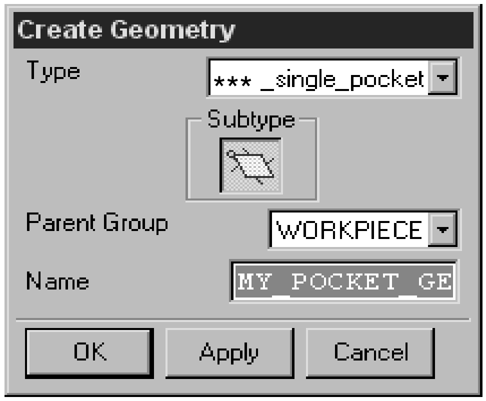
The Create Geometry dialog is displayed.

❑ In the Create Geometry dialog next to the Type label, change mill_planar to **Browse** and choose *****_single_pocket.prt**.



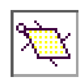
9

Notice that the only available Subtype is MY_POCKET_GEOM. This is a direct result of the options chosen when the original template was created.



Step 3 Specifying Part, Blank, and Floor geometry.

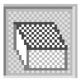
You will create a Parent Group named MY_POCKET_GEOM and specify the Part, Blank, and Floor geometry.

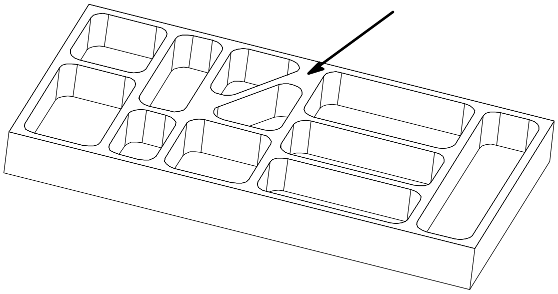
❑ Choose the **MY_POCKET_GEOM** icon. 

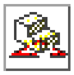
❑ Choose **WORKPIECE** as the Parent Group.

❑ Choose **OK**.

The MY_POCKET_GEOM dialog is displayed.

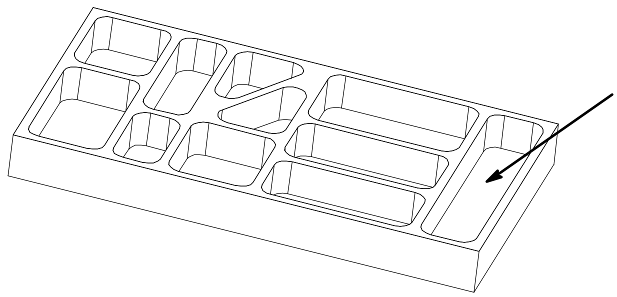
- ❑ In the MY_POCKET_GEOM dialog, choose the **Part** icon and then **Select.**
- ❑ Choose the Material side as **Outside.**
- ❑ Using the **Face Boundary**  method, select the top face of the part.



- ❑ Choose **OK**.
- ❑ Choose the **Floor** plane icon  and then **Select.**

The Plane Constructor dialog is displayed.

❑ Select the floor of any pocket.



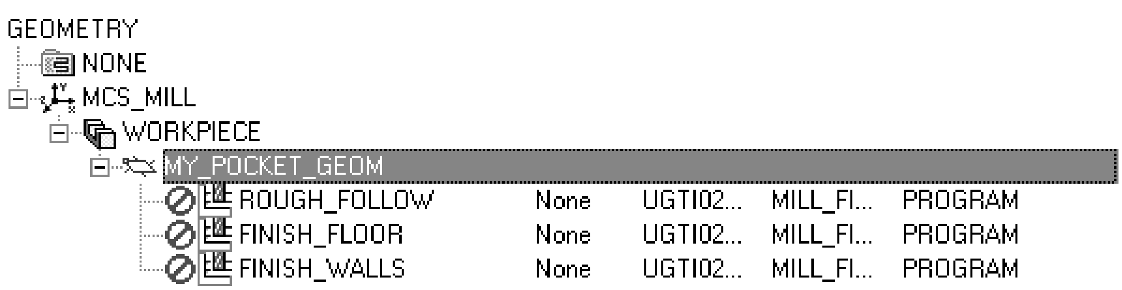
❑ Choose **OK** twice.

You have specified all of the information needed to generate the tool paths.

Step 4 Generating the tool paths

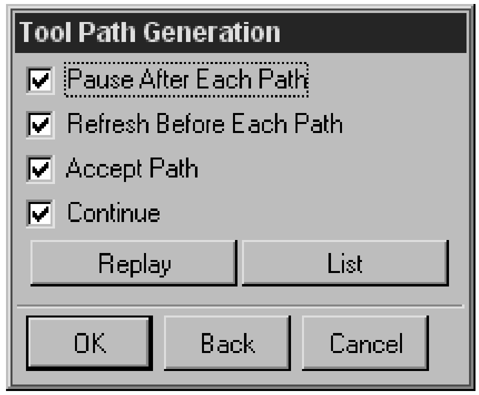
❑ In the Geometry View of the Operation Navigator, expand all Parent Groups.

Note that the operations were created to rough and finish the part.



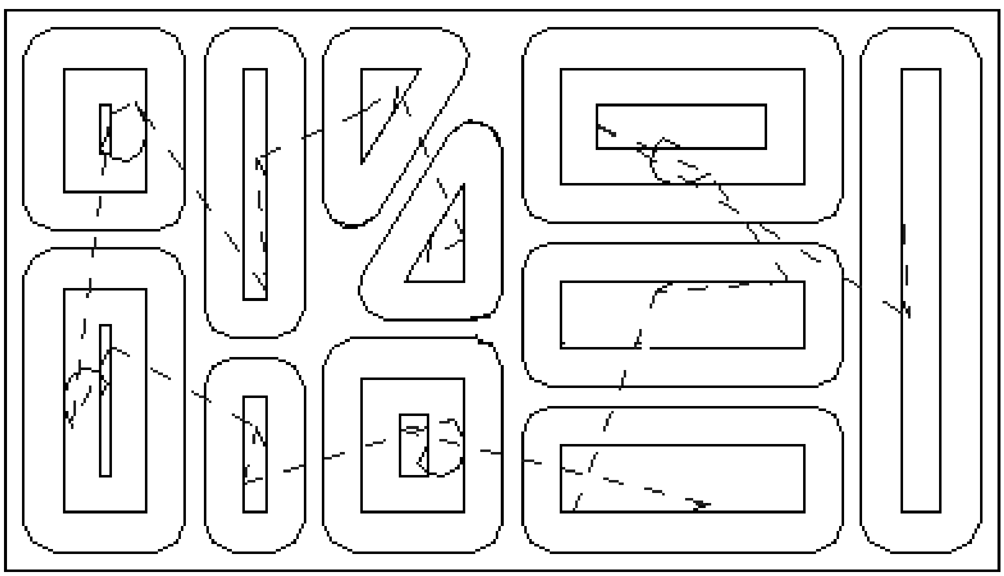
❑ Choose **MY_POCKET_GEOM** and using MB3 **Generate** the tool paths.

The Tool Path Generation dialog is displayed.

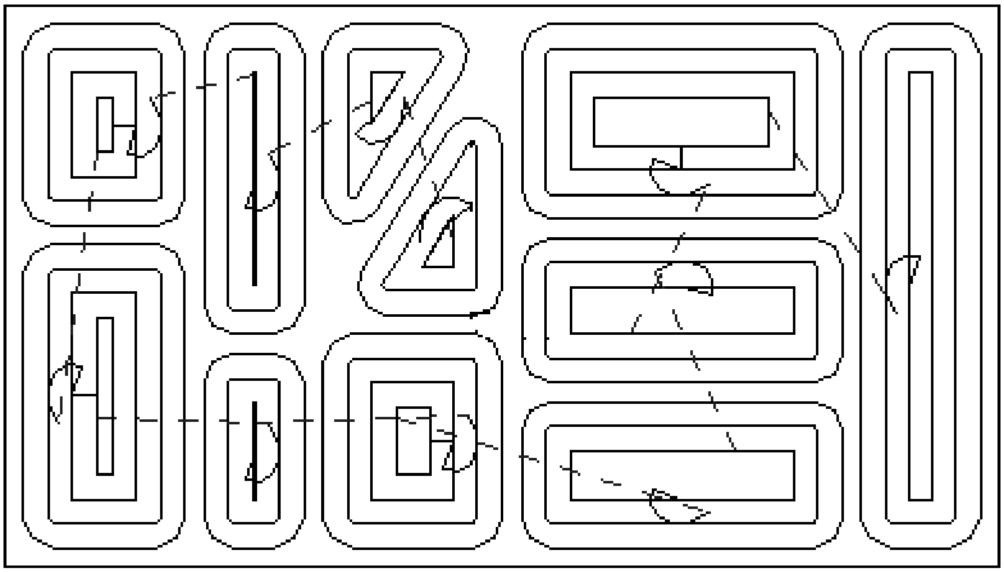


❑ Choose **OK** as necessary to continue generating the tool paths.

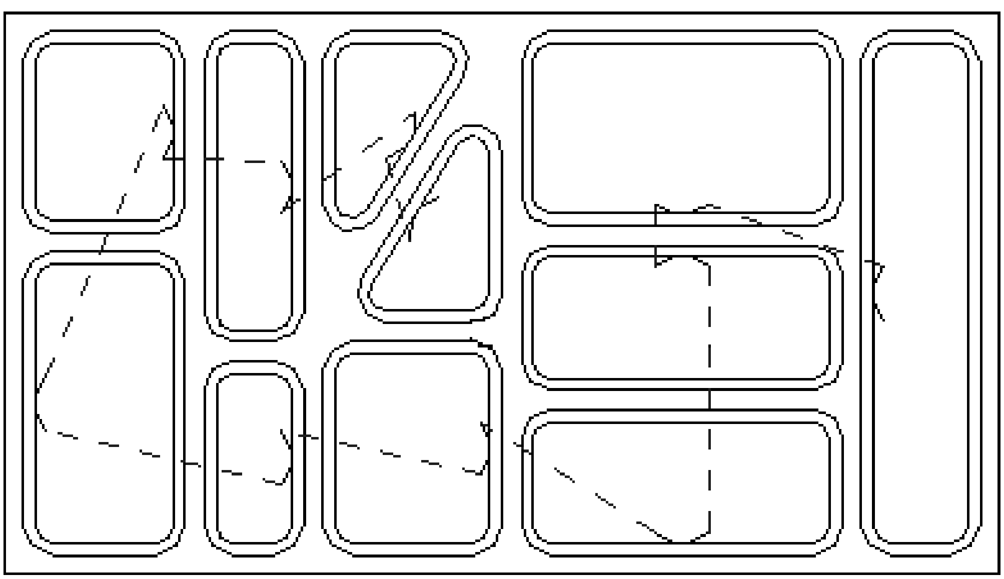
The ROUGH_FOLLOW operation creates a tool path similar to the following illustration.



The FINISH_FLOOR operation creates a tool path similar to the following illustration.



The FINISH_WALLS operation creates a tool path similar to the following illustration.



- ❑ Choose **OK** to accept the tool paths.
- ❑ **Save** and **Close** the part file.

You are finished with this activity.

9

Review of the Procedure

This is a good time to review the several steps you took in creating the pocketing sequence used for machining the previous activity. After this review, you will see that you can also create a sequence of operations for machining complex geometry just as easily. This complete sequence is built upon the same principles you used in creating the previous pocketing sequence.

These are the steps that you took in building the pocketing sequence:

In the Template part file, you:

- Created the geometry Parent Group, MY_POCKET_GEOM which contains the sequence of operations used to machine the part.
- You changed the MY_POCKET_GEOM Template setting to ON. This creates the Subtype MY_POCKET_GEOM in the Create Geometry dialog when using this template in another part.
- You created Template Operations under the Parent Group MY_POCKET_GEOM which roughed and then finished the floor and walls of the pockets. The Load with Parent setting was toggled to ON and the Template setting was toggled to OFF.

In the part file that used your template you:

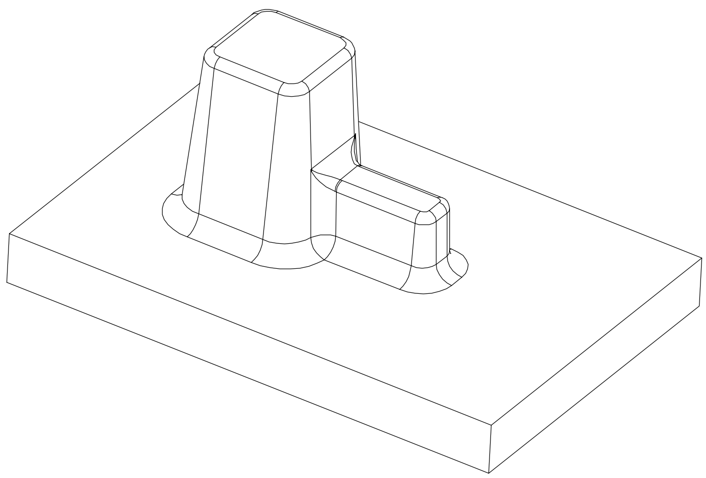
- Used mill_planar as the Setup to Initialize the CAM session.
- Selected *****_single_pocket.prt** as the Type (Template) using the Browse feature.
- Created the MY_POCKET_GEOM in the Create Geometry dialog (created by the template) which was used to choose the part and floor geometry.
- Created three pocketing operations by just selecting the part and floor geometry. Most of the effort in creating these operations was through the use of templates.

Activity 9—3: Using the Die_Sequence Template

This activity shows the use of a die machining sequence of operations, included in the Manufacturing Application, to machine more complicated part geometry. This sequence, is used to rough, semi-finish and finish machine a part based on die machining practices.

Step 1 Open the part file.

❑ Open **ama_deep_core.prt.**



❑ Save the part as, ******_deep_core.prt*** where ******* represents your initials**.**

9

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

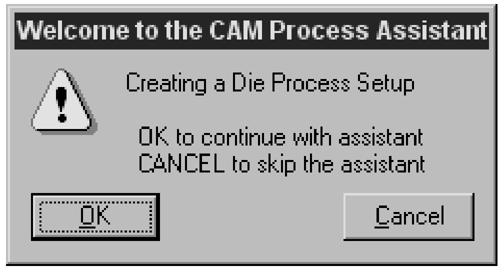
❑ The CAM Session Configuration should be **mill_contour**.

❑ The CAM Setup Template should be **die_sequences**.

This is the template that contains the die machining sequence.

❑ Choose **Initialize**.

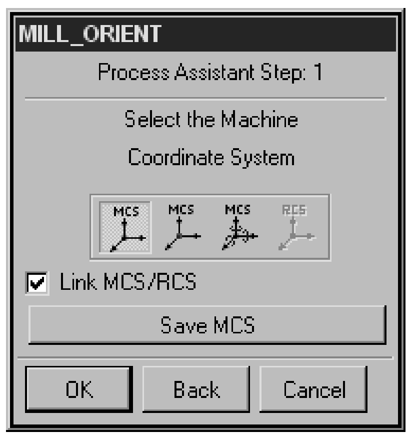
The CAM Process Assistant for Die Machining is displayed.



The Process Assistant guides you in selecting the geometry for the machining sequence.

❑ Choose **OK.**

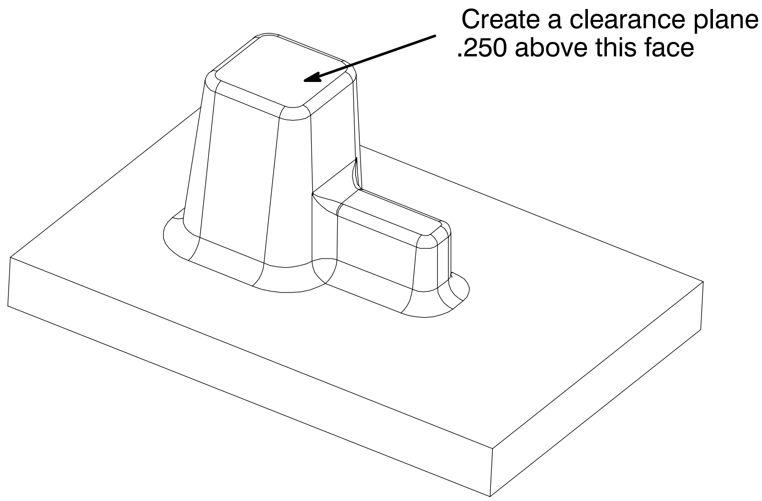
The Process Assistant Step 1 dialog is displayed, asking for the selection of the MCS. By selecting OK, the MCS is set to the WCS.



❑ Choose **OK.**

The Process Assistant Step 2 dialog is displayed. The part and blank geometry were selected automatically. The Process Assistant was designed to select the part and blank geometry by searching for geometry with assigned attributes of those names. The attribute names were assigned by the designer. You do not need to select part or blank geometry for any operation in the machining sequence.

❑ In the Process Assistant Step 2: dialog, choose **Specify** and then create a clearance plane .250 above the face as shown.



❑ Choose **OK** until the Process Assistant Step 3 dialog is displayed.

❑ In the Process Assistant Step 3: dialog, choose **Display**.

The part geometry is displayed.

❑ Choose **OK**.

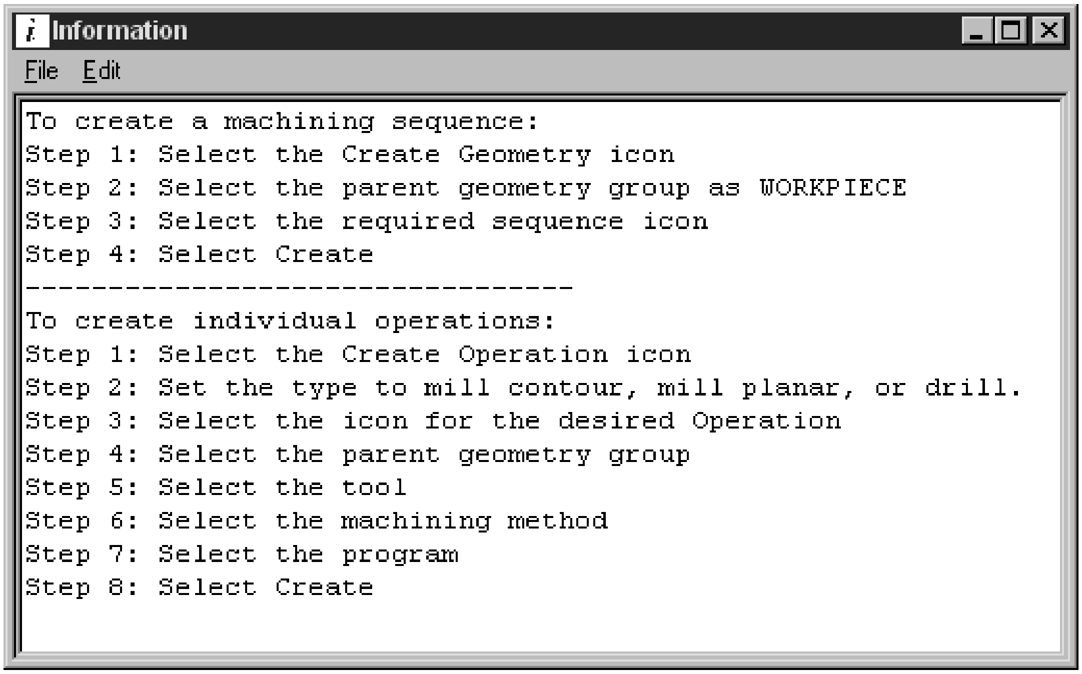
The Process Assistant Step 4: is displayed.

❑ In the Process Assistant Step 4: dialog, choose **Display.**

The Blank Geometry is displayed.

❑ Choose **OK**.

A dialog window appears, referring you to an information window for further instructions. The information window which is displayed, gives the steps necessary for creating a machining sequence and/or individual operations.

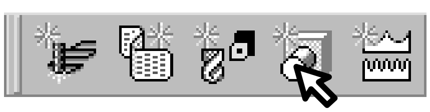



```
To create a machining sequence:
Step 1: Select the Create Geometry icon
Step 2: Select the parent geometry group as WORKPIECE
Step 3: Select the required sequence icon
Step 4: Select Create
---------------------------------
To create individual operations:
Step 1: Select the Create Operation icon
Step 2: Set the type to mill contour, mill planar, or drill.
Step 3: Select the icon for the desired Operation
Step 4: Select the parent geometry group
Step 5: Select the tool
Step 6: Select the machining method
Step 7: Select the program
Step 8: Select Create
```

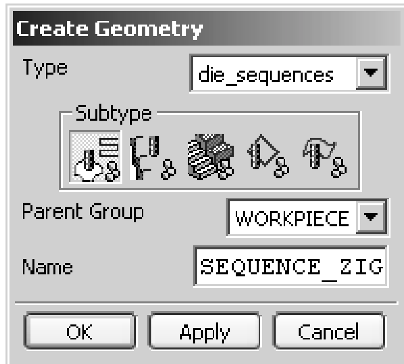
❑ Choose **OK**.

The Operation Navigator is displayed.

❑ Select the **Create Geometry** icon from the Create Toolbar.

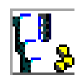


The Create Geometry dialog is displayed.



Notice the various Subtype icons on the Create Geometry dialog. Each represents a cutting sequence (Zigzag, Zlevel, Profile 2D, Profile 3D).

Step 2 Creating the Sequence.

❑ In the Create Geometry dialog, choose **SEQUENCE_ZLEVEL.** 

❑ Choose the **WORKPIECE** as the Parent Group.

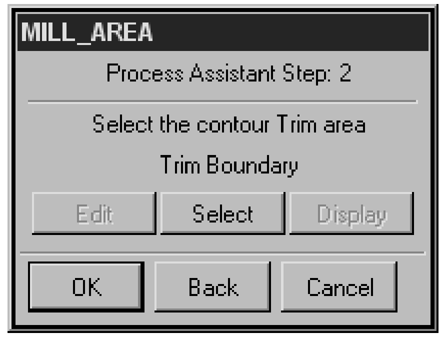
❑ Choose **OK**.

A new Process Assistant starts.

The Process Assistance asks for the selection of the cut area. If you do not, specify a cut area, all part geometry will be machined by default.

❑ At the prompt, to select the cut area geometry, choose **OK** to cut all of the part geometry.

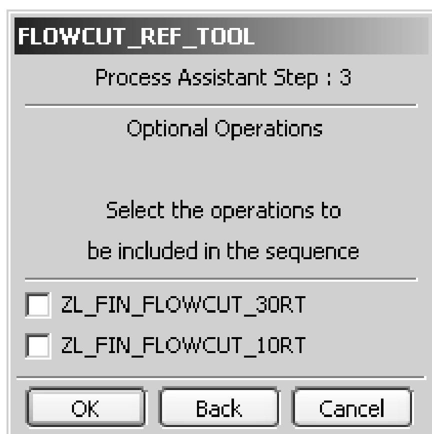
The Process Assistant Step:2 dialog is displayed asking you to specify the trim area.



You can specify the trim boundary which limits the tool path to the area inside or outside of the boundary. You will put a trim boundary around the outside of the part to prevent the tool from “water falling” down the side of the part.

- ❑ At the prompt to select the trim boundary, choose **Select**.
- ❑ Set the Trim Side to **Outside**.
- ❑ Select the bottom of the part and choose **OK** twice.

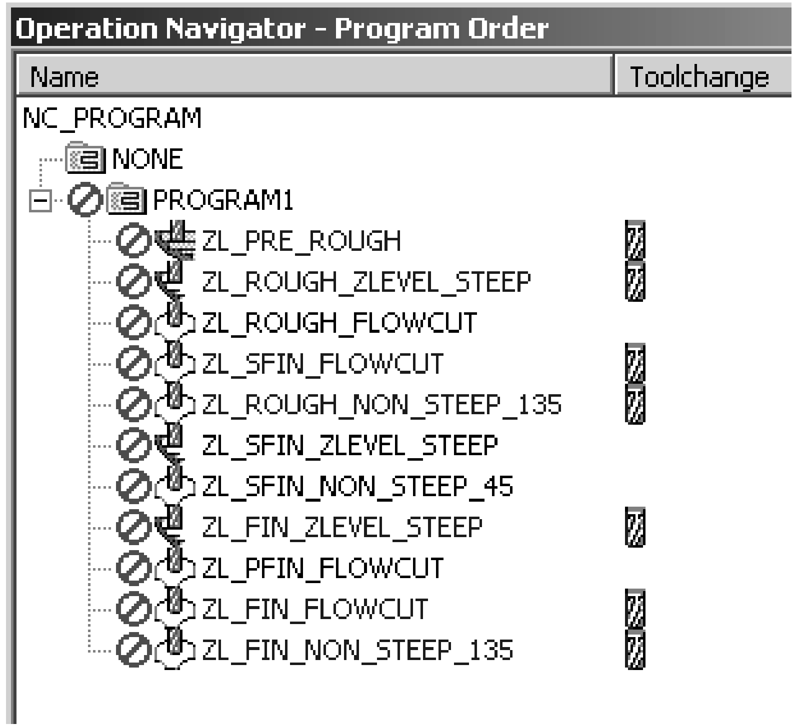
The Process Assistant Step:3 dialog is displayed asking you to specify optional operations for cutting the part.



- ❑ Choose **OK**.
- ❑ Choose **OK** for any other Process Assistant Steps**.**

The machining sequence of operations is created.

- ❑ In the Operation Navigator Program Order View, expand PROGRAM1 and note the operations.



❑ **Save** and **Close** the part file.

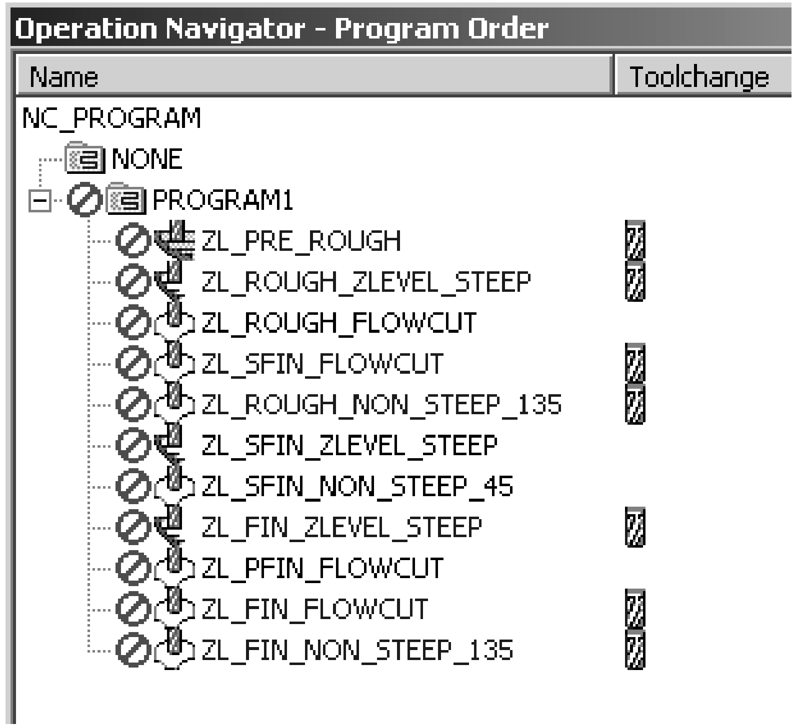
Step 3 Viewing the tool paths.

At this point, you could generate the tool paths to see the sequence of operations. However, it is faster to switch to another part file that is identical to this one except that the tool paths have been generated and are ready for Replay.

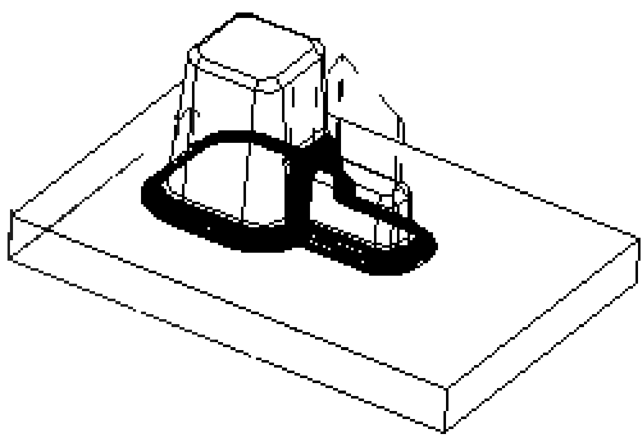
❑ Open **ama_deep_core_with_paths.prt.**

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

❑ Expand PROGRAM1 and note the operations.



❑ Highlight PROGRAM1 and **Replay** the machining sequence.



❑ Close the part file.

You are finished with this activity.

9

More on Templates

Changing the Machining Environment

The option **Tools→Operation Navigator→Delete Setup** can return you to the Machining Environment dialog. You will delete all CAM information in the part file and then choose another Machining Environment. Note that if you use this option, all of your operations, tools, etc. will be permanently removed.

The option **Preferences→Manufacturing→ Configuration** tab, selecting a configuration file, changes the CAM Session Configuration *without* changing the Setup. The current data is preserved while the Configuration data (e.g., templates available, etc.) changes.

The option **Preferences→Manufacturing→ Configuration** tab, **Reset** button under Configuration File does not change Setup but it does restore the original CAM Session Configuration.

Template Operations

The Template status for all Tools, Methods, Geometry and Programs is Off (by default).

The Template status for Operations is On (by default).

SUMMARY

The Template function provides an efficient means to customize your CAM environment. Numerous parameters used by various operations, custom operation sequences, tool and postprocessor availability, and numerous other items that are used repeatedly, can be included into custom templates. The possibilities are only limited to your imagination.

In this lesson you:

- Created a sequence of template operations
- Interacted with a supplied template containing a sequence of operations to machine part geometry
- Became familiar with the advanced concepts of using templates

(This Page Intentionally Left Blank)

Libraries

Lesson 10

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 in NX.

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 NX 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)

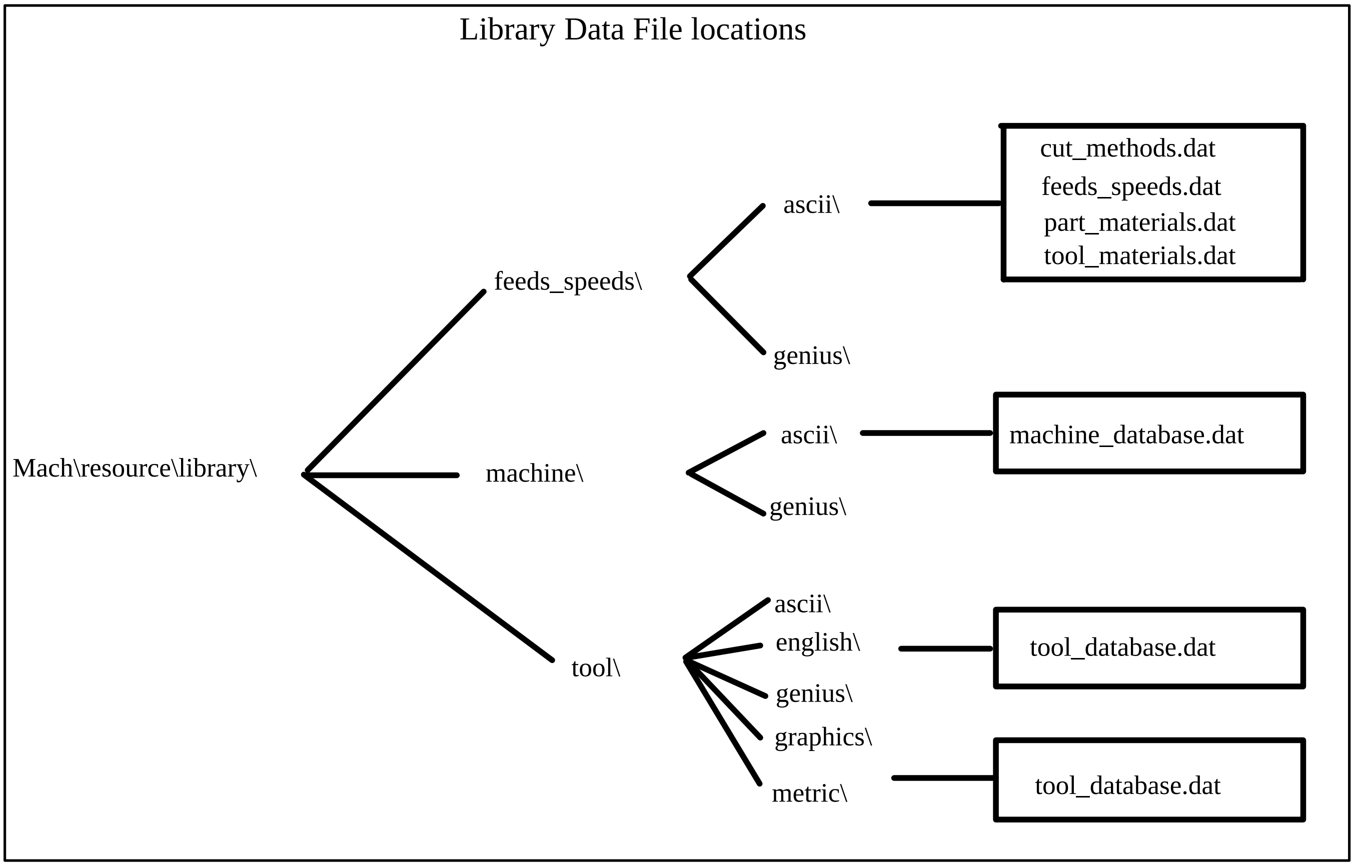
```
TEMPLATE_OPERATION,${UGII_CAM_TEMPLATE_SET_DIR}mill_contour.opt
TEMPLATE_DOCUMENTATION,${UGII_CAM_SHOP_DOC_DIR}shop_doc.dat
TEMPLATE_POST,${UGII_CAM_POST_DIR}template_post.dat
USER_DEFINED_EVENTS,${UGII_CAM_USER_DEF_EVENT_DIR}ude.cdl,$
        {UGII_CAM_USER_DEF_EVENT_DIR}ude.tcl
TEMPLATE_CLSF,${UGII_CAM_TOOL_PATH_DIR}template_clsf.dat
LISTING_FORMAT,${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 Unigraphics are the access mechanisms to the ASCII text 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.



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 UGPOST and Shop Documentation

Activity 10–1: Preparation for modifying CAM Libraries

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 Main Menu Bar, select **Help→Unigraphics NX 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 located in the *instal* 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.**

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 With→Notepad.**

- ❑ 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.

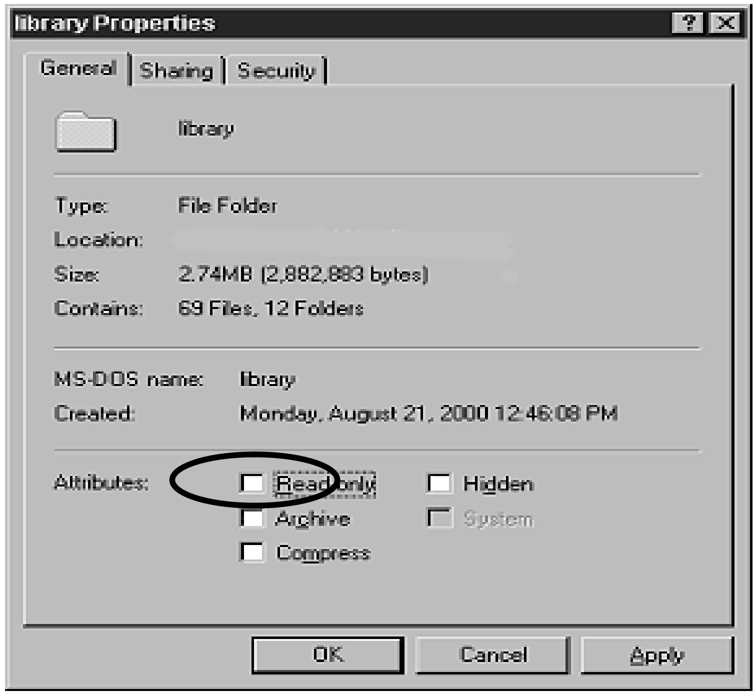
- ❑ Save the file and exit from Notepad.

Step 4 If necessary, change the Read-only protection on your just created local \Mach\resource\library.

- ❑ With the Windows Explorer, locate your home \mach\resource\library directory.

- ❑ Highlight the directory and with **MB3** select **Properties**.

- ❑ Uncheck **Read-Only**.



Step 5 Restart Unigraphics

- ❑ Exit Unigraphics and then restart Unigraphics.
- ❑ On the Unigraphics Main Menu Bar, select **Help→Unigraphics NX Log File** to verify that ***your*** resource directory is being used.

Unix:

Step 1 Copying the *mach/resource* directory.

- ❑ On the UnigraphicsMain Menu Bar, select **Help→Unigraphics NX 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/ugsnx/mach/resource .

Step 2 Copying the UG environment file, *.ugii_env* to your home directory.

- ❑ Copy the *.ugii_env* file located in the *instal* 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/

- ❑ Save the changes and exit from the 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 Unigraphics.

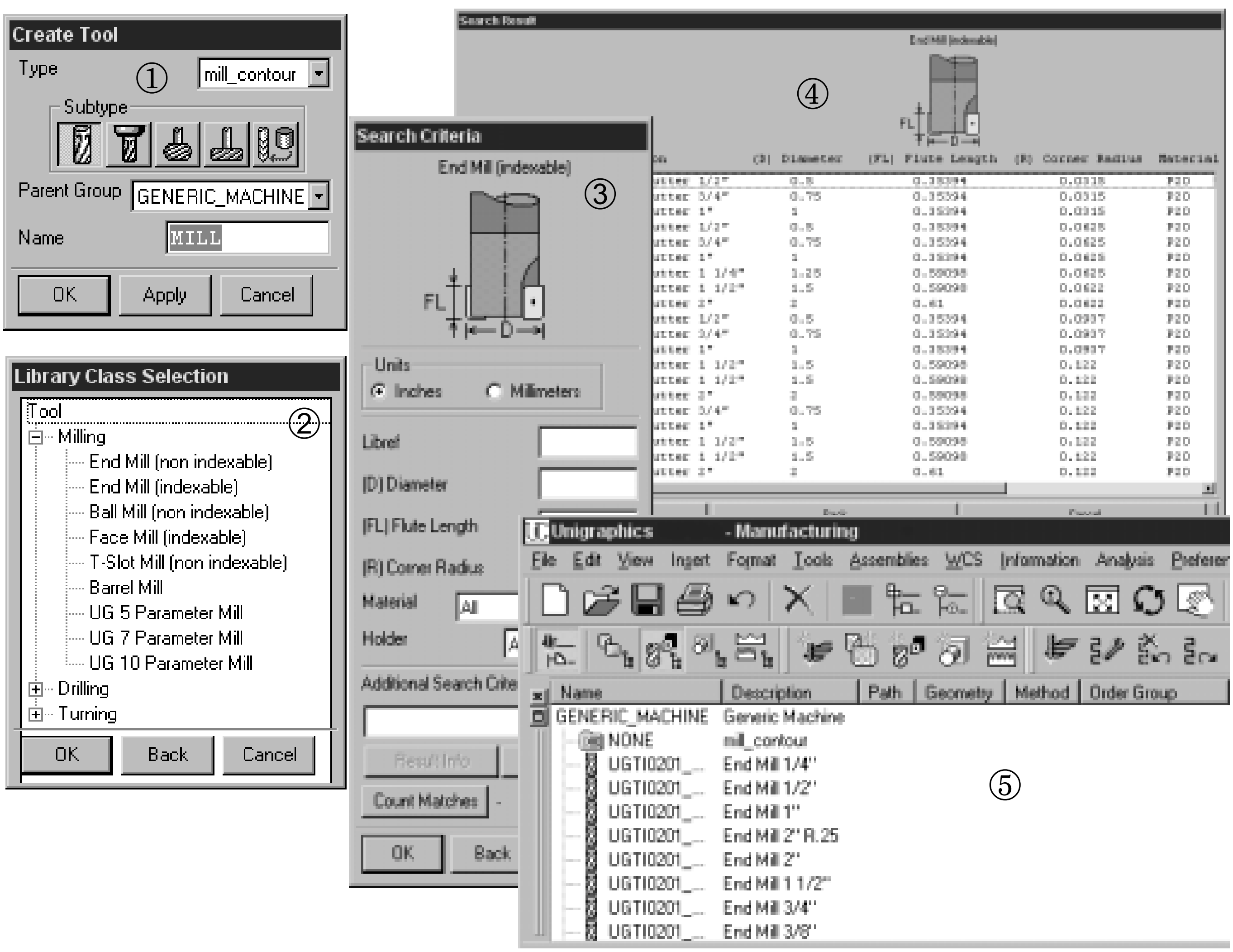
- ❑ On the Unigraphics Main Menu Bar, select **Help→UG Log File** to verify that ***your*** resource directory (based from your home directory) is being used.

  This completes this activity.

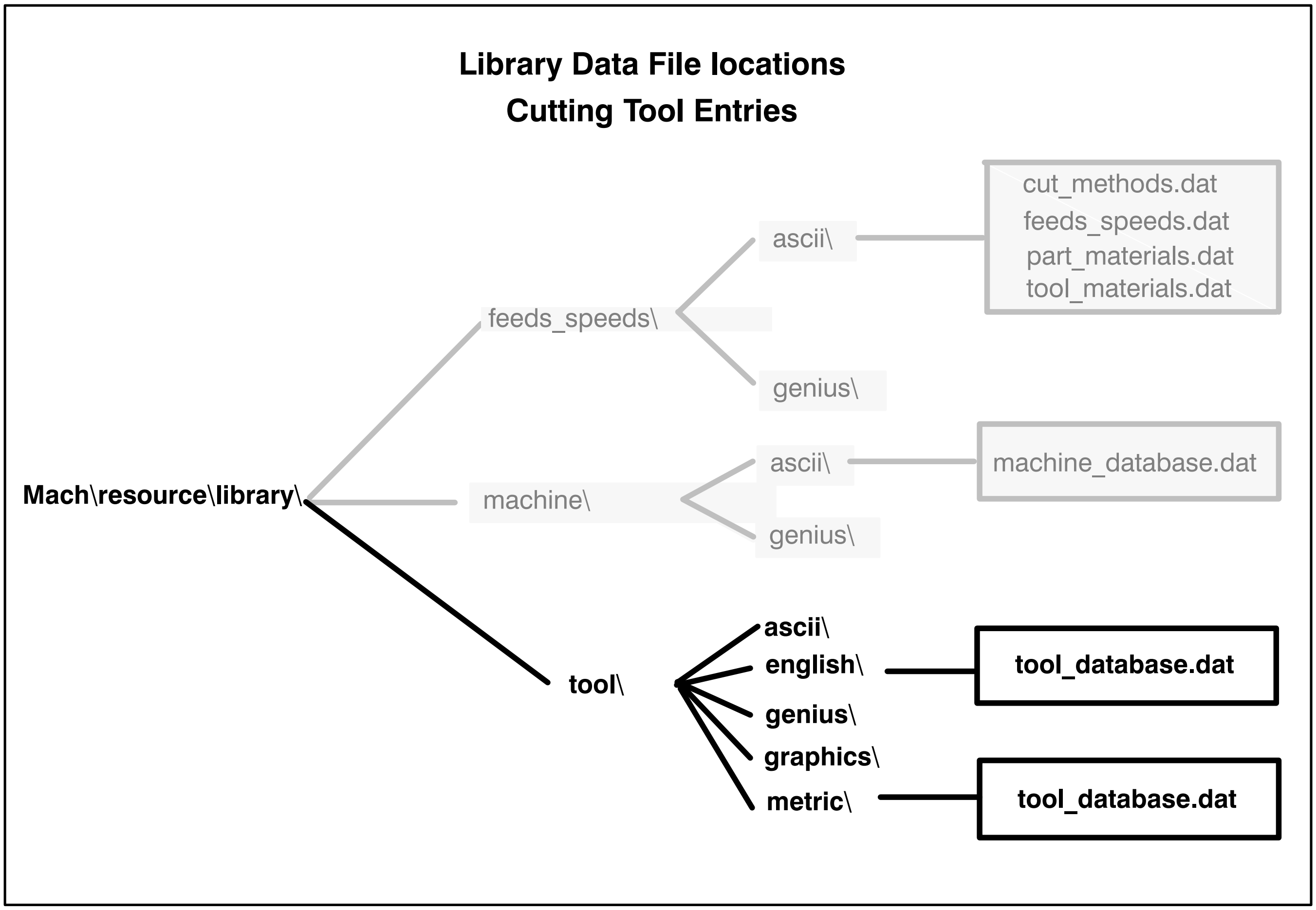
Cutting Tool Libraries

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, 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 used for advanced replays with a solid tool. These assemblies are provided with each

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**.

10

Activity 10—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 **ama_lib_tools.prt**

Step 2 Enter the Manufacturing Application.

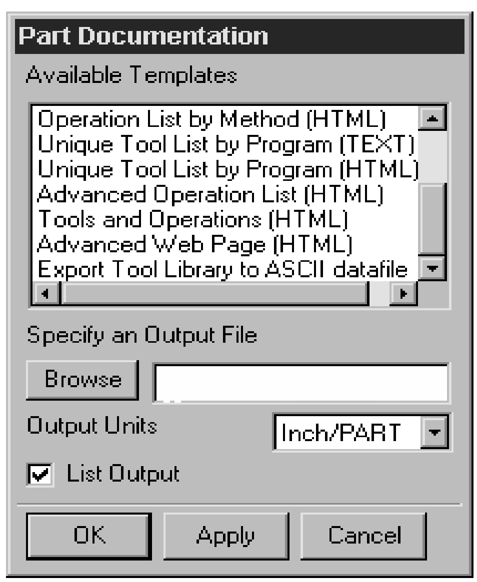
❑ Choose **Application** → **Manufacturing**

The Operation Navigator and the Create Operation dialogs are displayed.

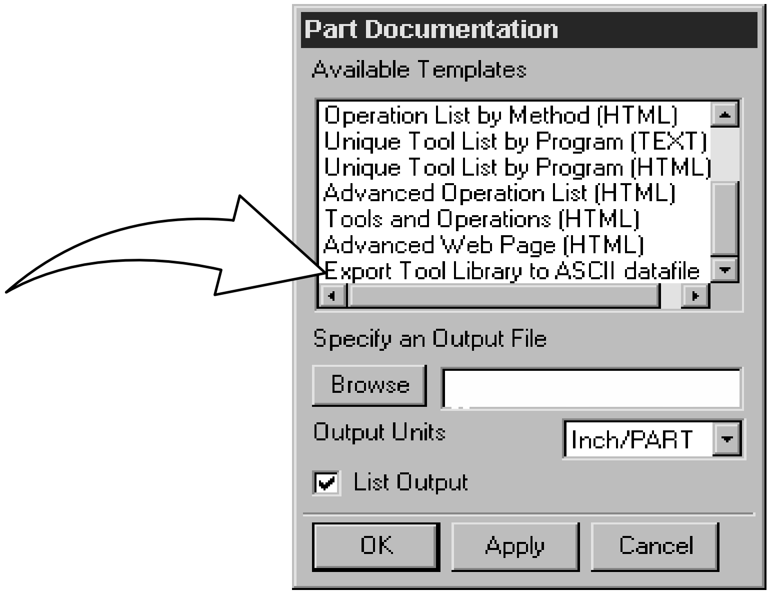
Step 3 Export the existing tool entries to an ASCII data file.

❑ Choose **Information** → **Shop Documentation** (or select the Shop Documentation Icon).

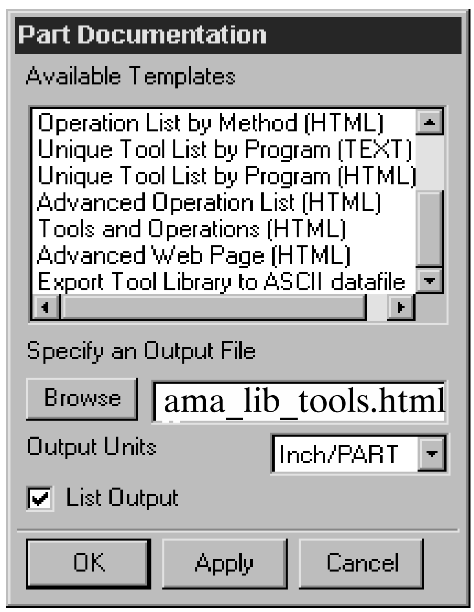
The Part Documentation dialog is displayed.



❑ 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, **ama_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 HDIA HLEN HTAP HOFF
#
——+————————+—+—+————————————————+————+————+———+—+—
–+———+———+———+———+———+———+———+———+———
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).

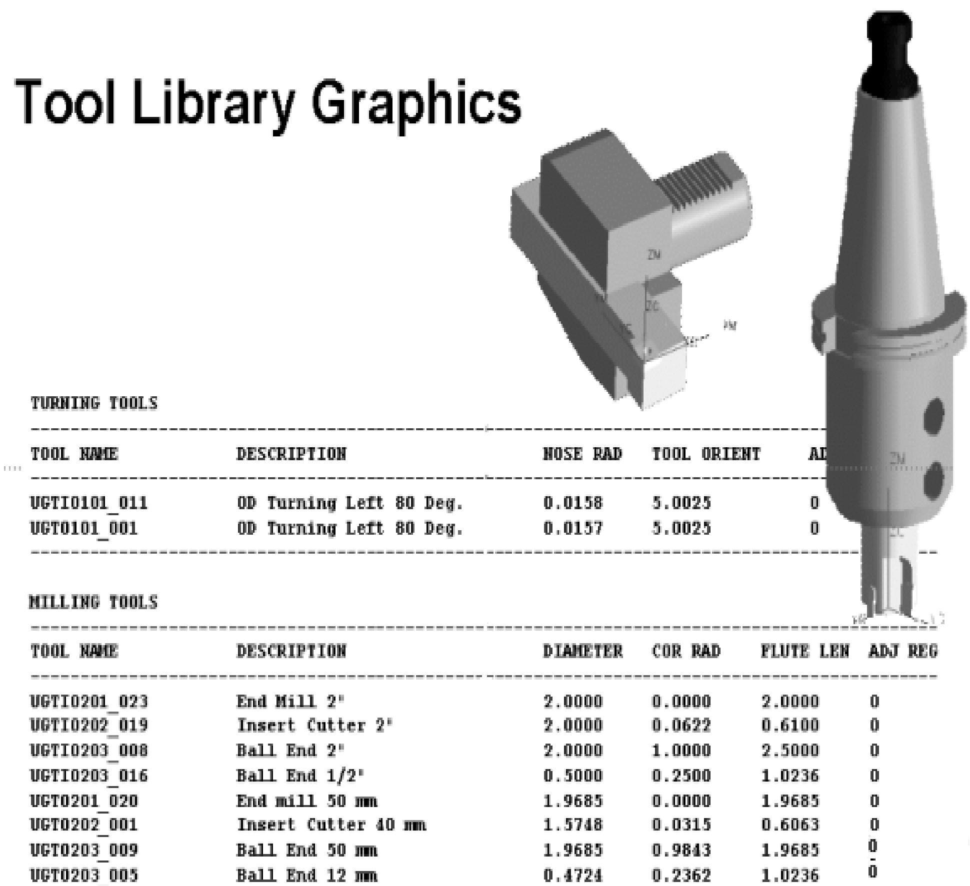
- ❑ Save the contents of the **tool_database.dat** file.

Tool Graphics Library

The Tool Graphics Library contains information related to the display of tool assemblies configured for CAM.

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 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*.

Assemblies are provided for turning and milling. These files are stored in the *graphics* subdirectory.



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

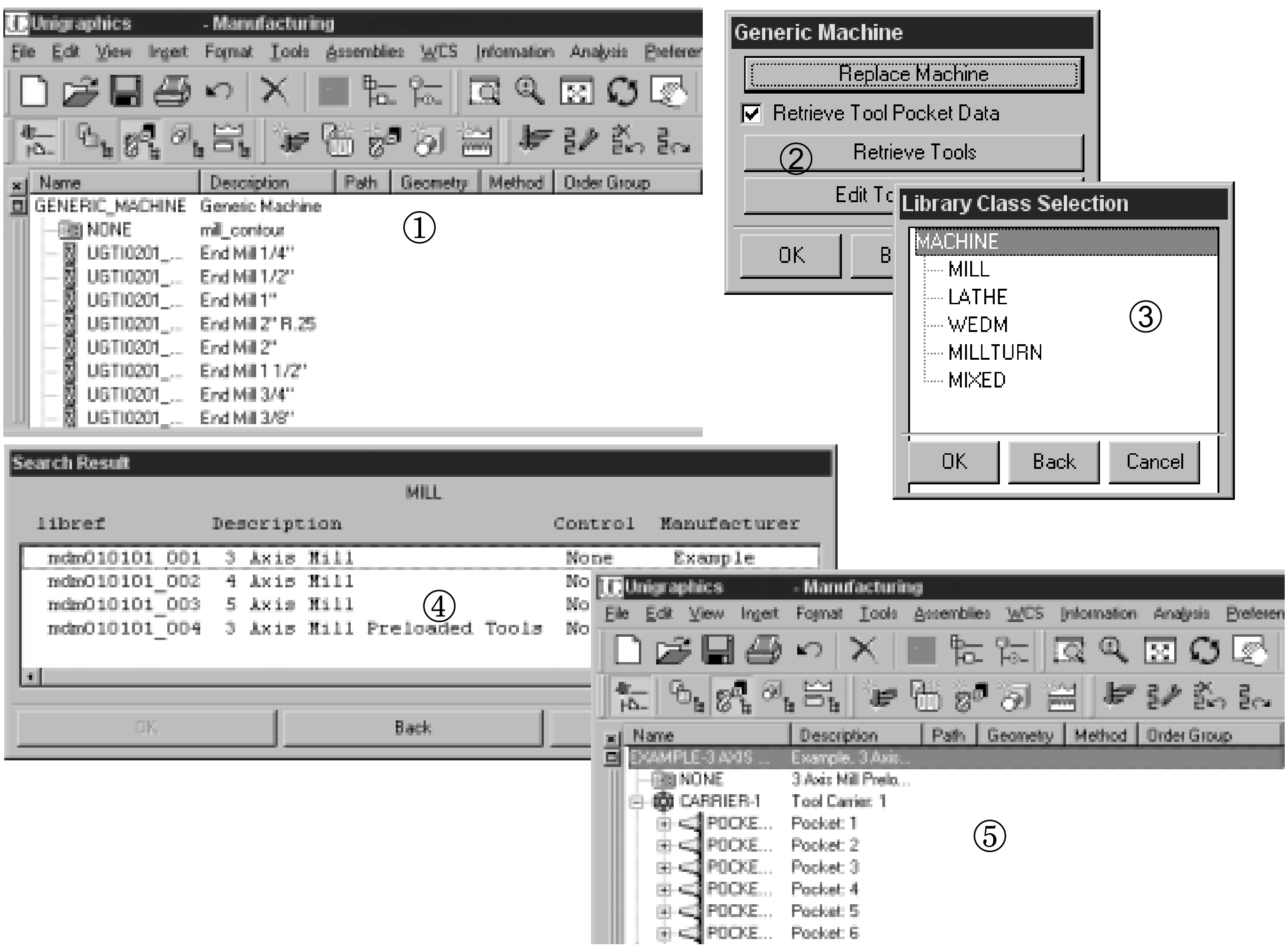
TOOL NAME	DESCRIPTION	DIAMETER	COR RAD	FLUTE LEN	ADJ REG
UGTI0201_023	End Mill 2"	2.0000	0.0000	2.0000	0
UGTI0202_019	Insert Cutter 2"	2.0000	0.0622	0.6100	0
UGTI0203_008	Ball End 2"	2.0000	1.0000	2.5000	0
UGTI0203_016	Ball End 1/2"	0.5000	0.2500	1.0236	0
UGT0201_020	End mill 50 mm	1.9685	0.0000	1.9685	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.

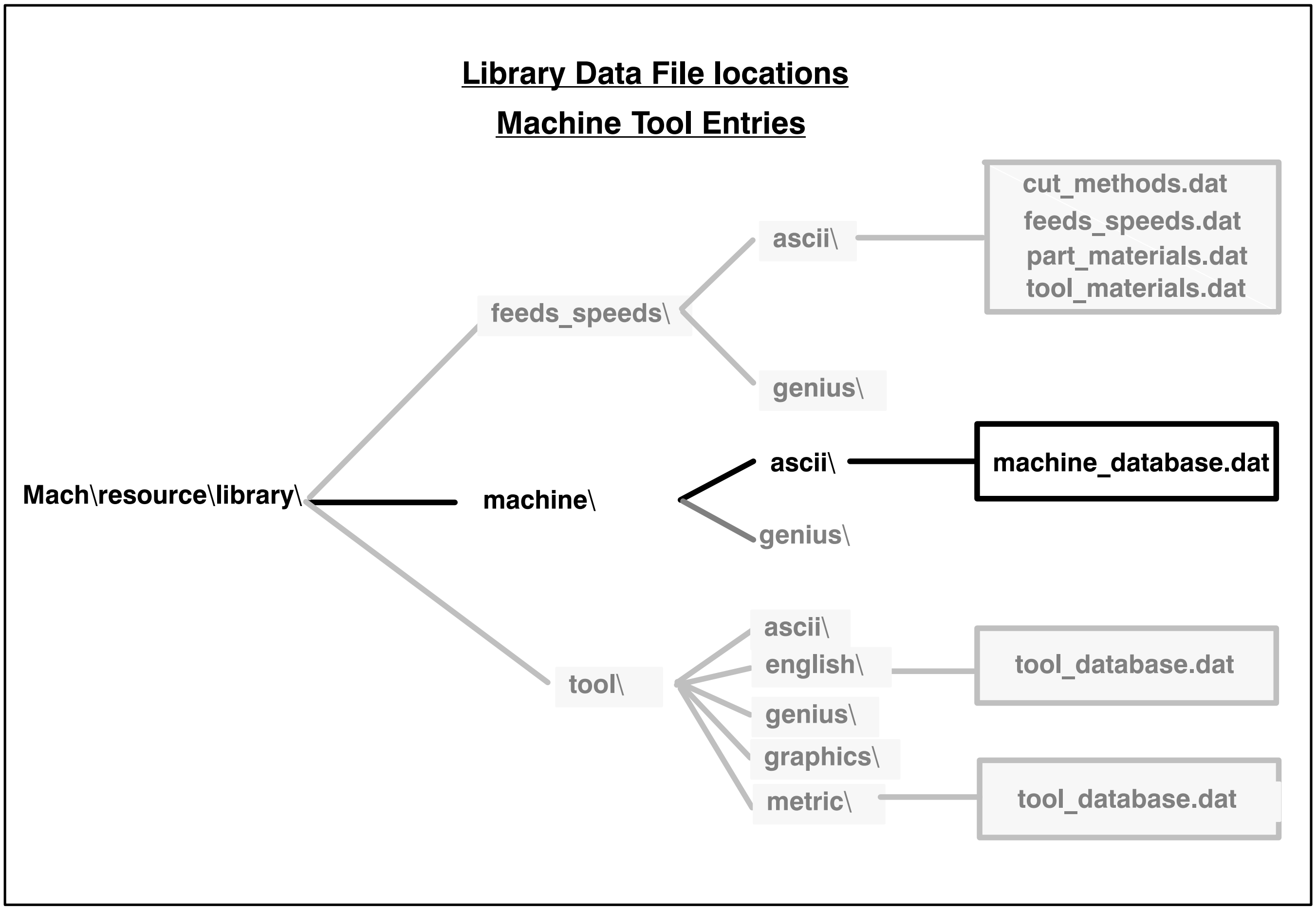
Machine Tool Libraries

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 10–3: Machine Tool Libraries

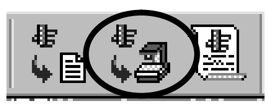
In this activity you will become familiar with the procedure to access Machine Tool data from CAM Libraries. You will see how this library is used in conjunction with the Post Process dialog by replacing the Available Machines with a 5-axis postprocessor.

Step 1 Open the part file.

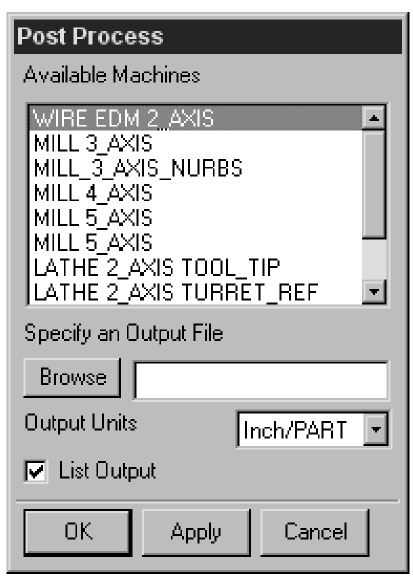
- ❑ Open the part file, **ama_lib_function.prt.**
- ❑ If necessary, enter the **Manufacturing Application.**

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.**

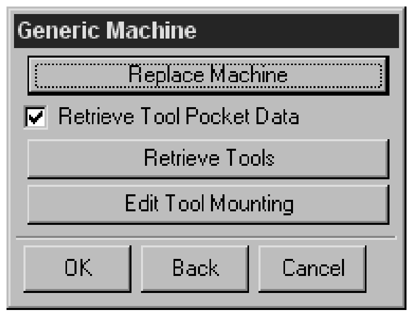
10

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**.**

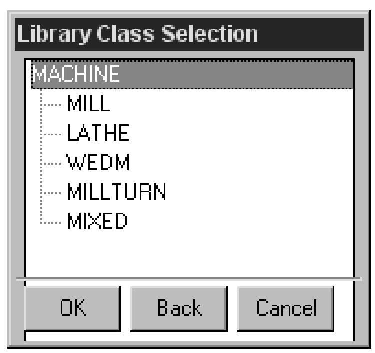
❑ Highlight the GENERIC_MACHINE, then **MB3,** and select edit.

The Generic Machine dialog is displayed.



❑ Select the Replace Machine button.

The Library Class Selection Menu is displayed.

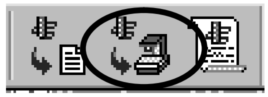


❑ Highlight **MILL**, then select **OK**.

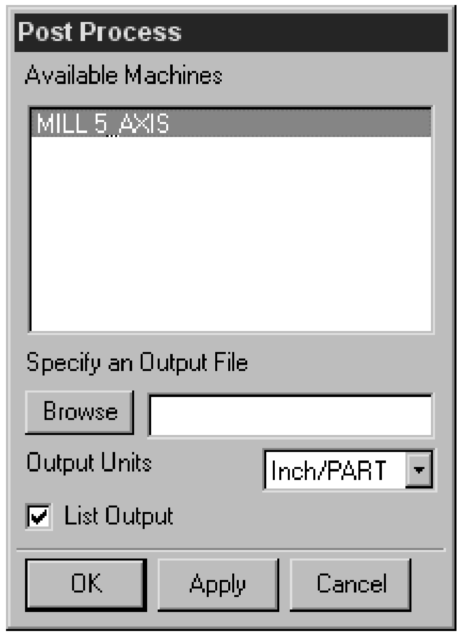
The Search Result listing is displayed.

❑ 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.

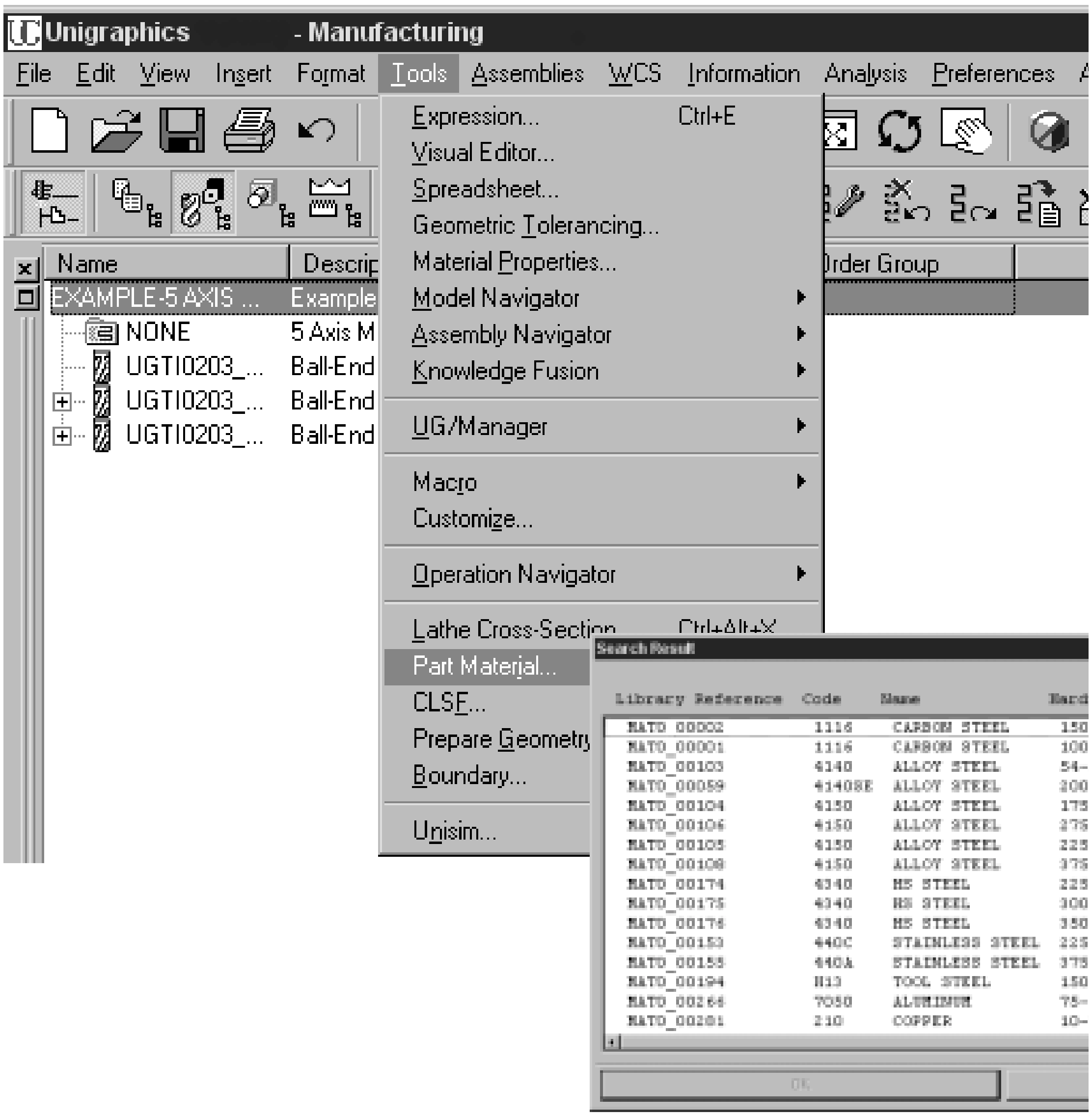
❑ Choose the **Cancel** button.

You are finished with this activity and will be using this part file in the next activity.

Part Material Libraries

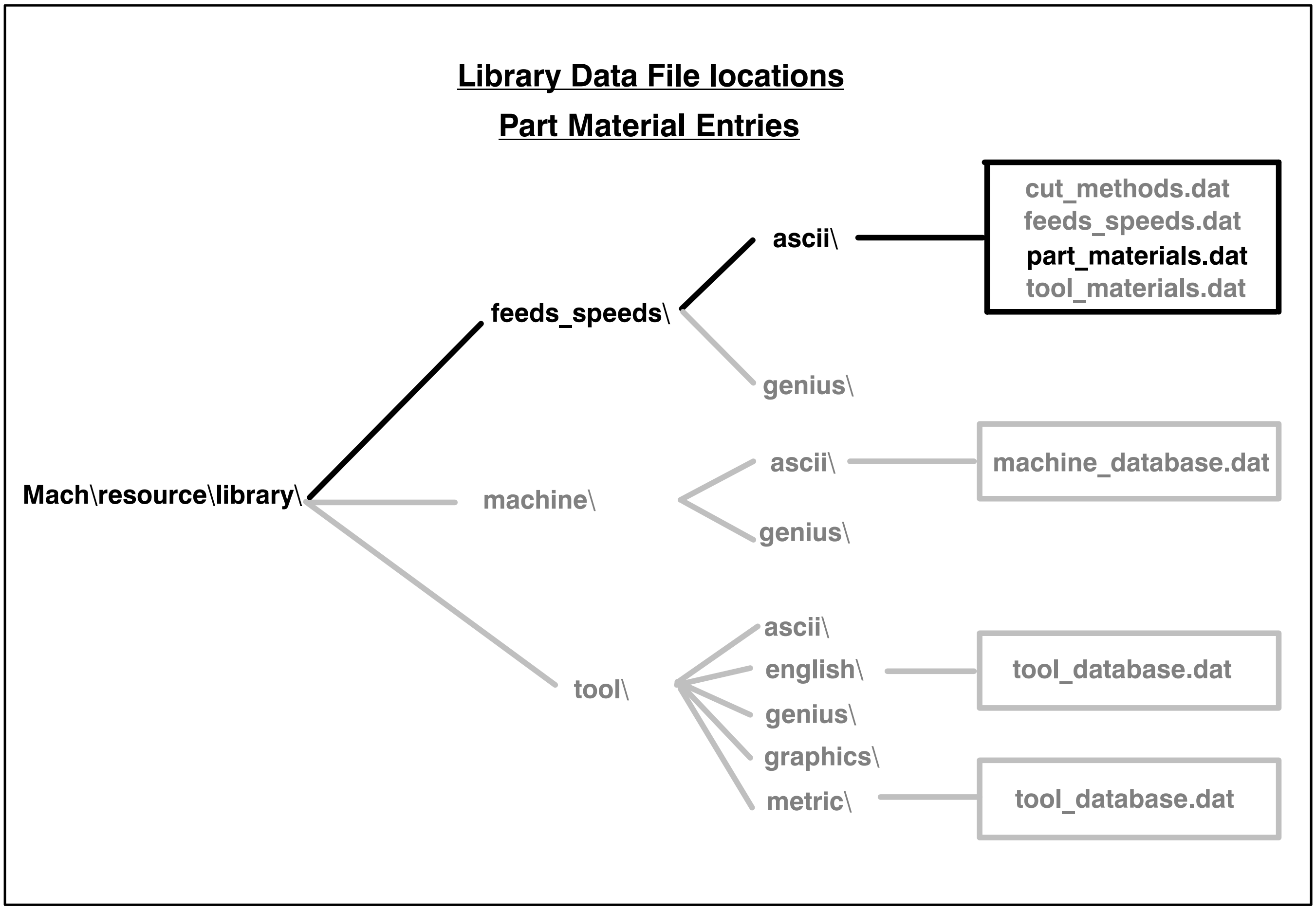
Part Material libraries contains information which is used in the calculation of feeds and speeds. This is *not* the same as Part Material used in Modeling.

To select the Part Material for a Set-up, select from the main 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.

10



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).

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 10–4: Part Materials Libraries

In this activity you will become familiar with accessing Part Material from the Library.

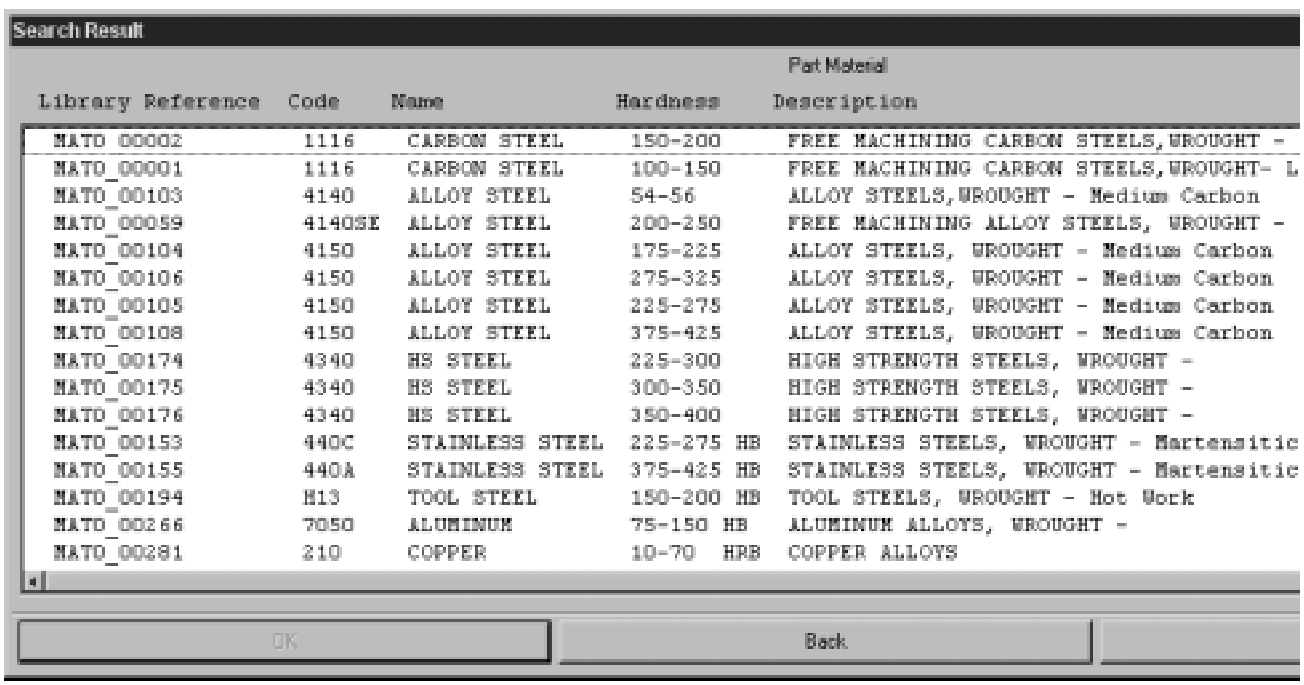
Step 1 List all part materials which are available for selection.

❑ Continue with the opened part, **ama_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

❑ Select **Tools** → **Part Material** from the Main 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

OK Back

Step 2 Select the desired part material.

❑ Select **MAT0_00105** from the Search Results list

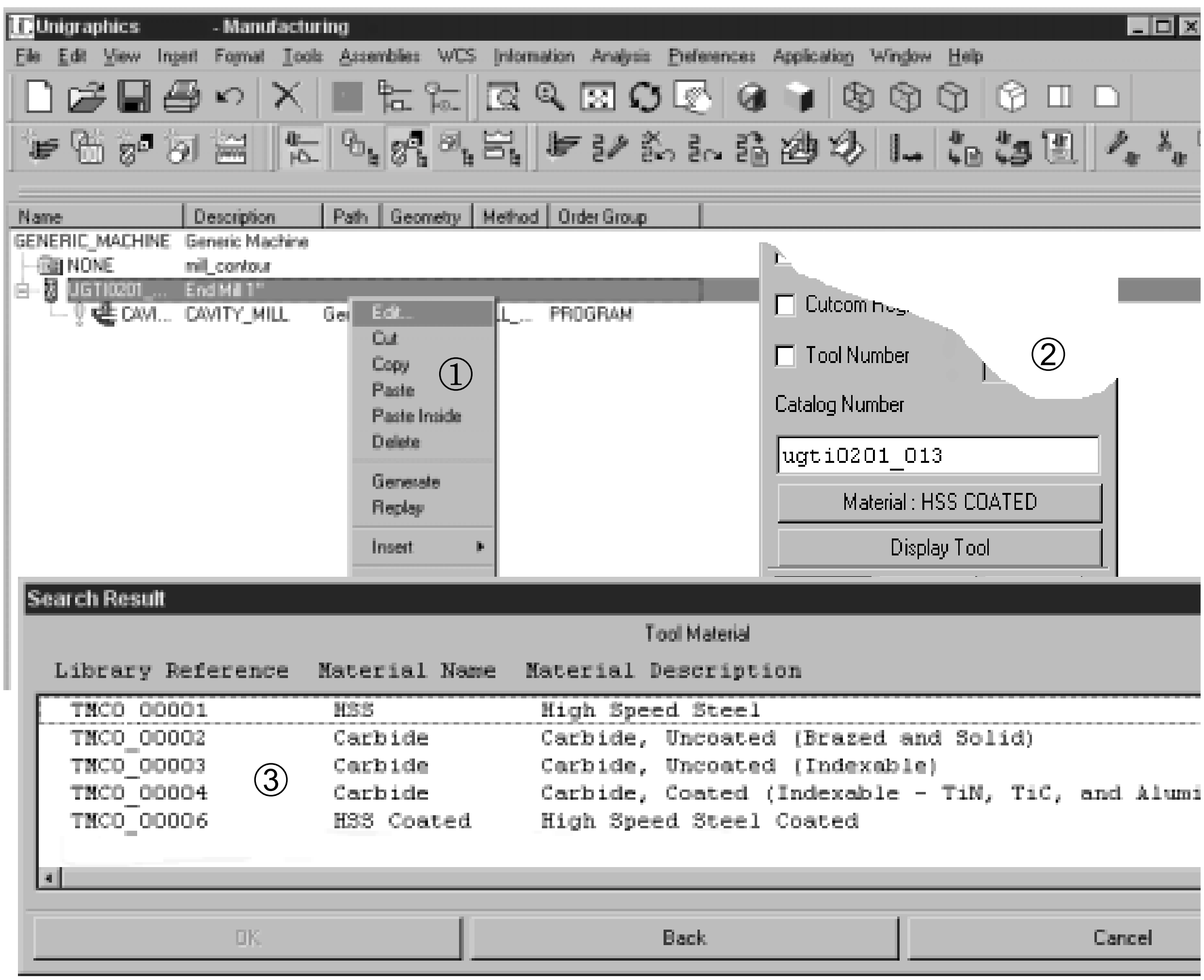
❑ Choose **OK.**

You are finished with this activity and will be using this part file in the next activity.

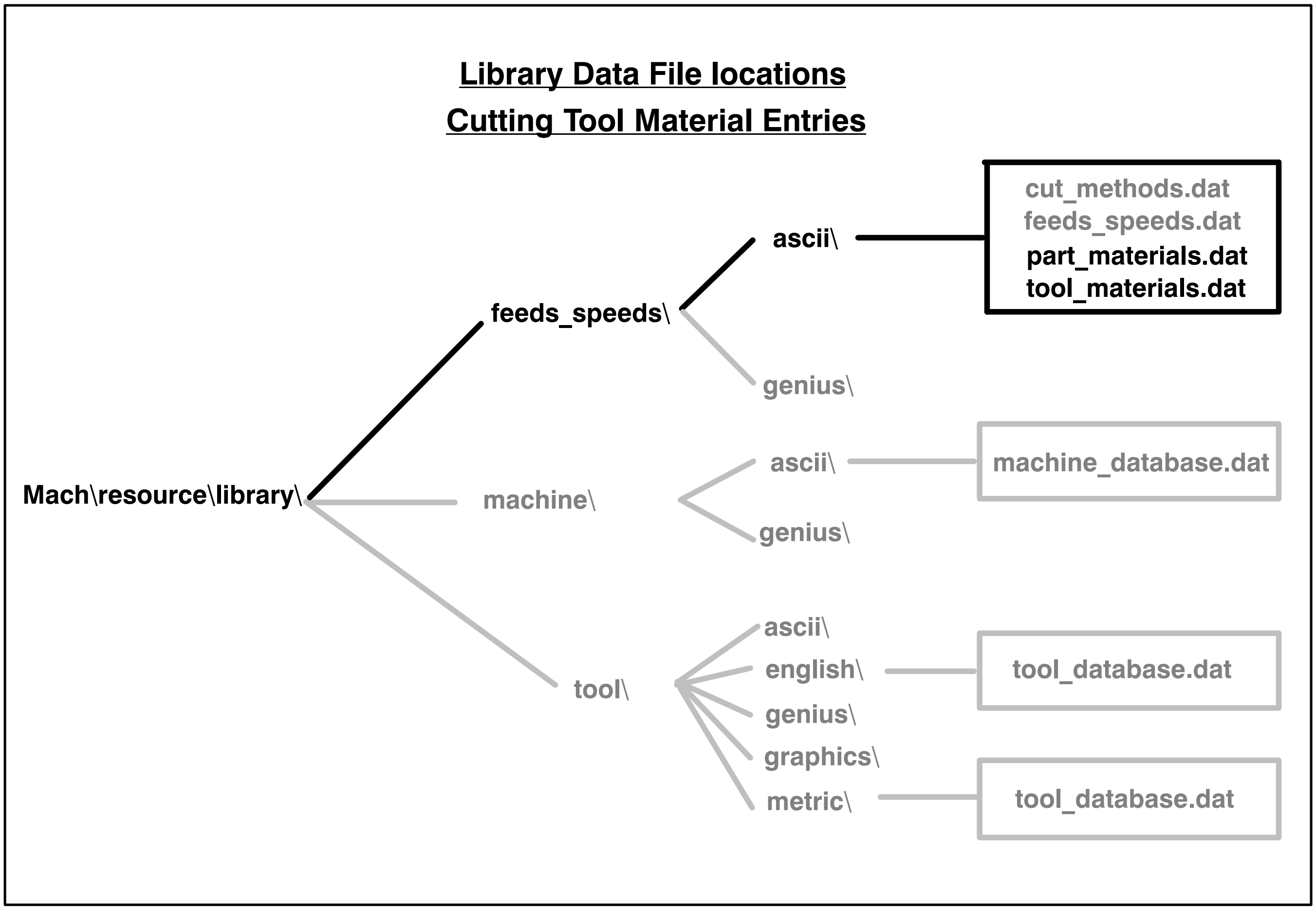
Cutting Tool Material Libraries

Cutting Tool Material libraries contains information which pertains to the cutting tool material type 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. Selection of the *Material:* button (②) from this dialog, 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).

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 10—5: Cutting Tool Materials Libraries

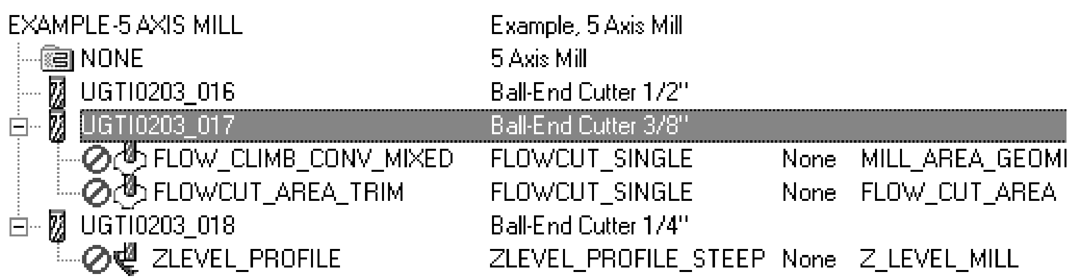
In this activity you will become familiar with the Cutting Tool Materials data. Cutting Tool Materials are used in the calculation of feeds and speeds.

Step 1 Accessing the Cutting Tool Material library.

❑ Continue with the opened part, **ama_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

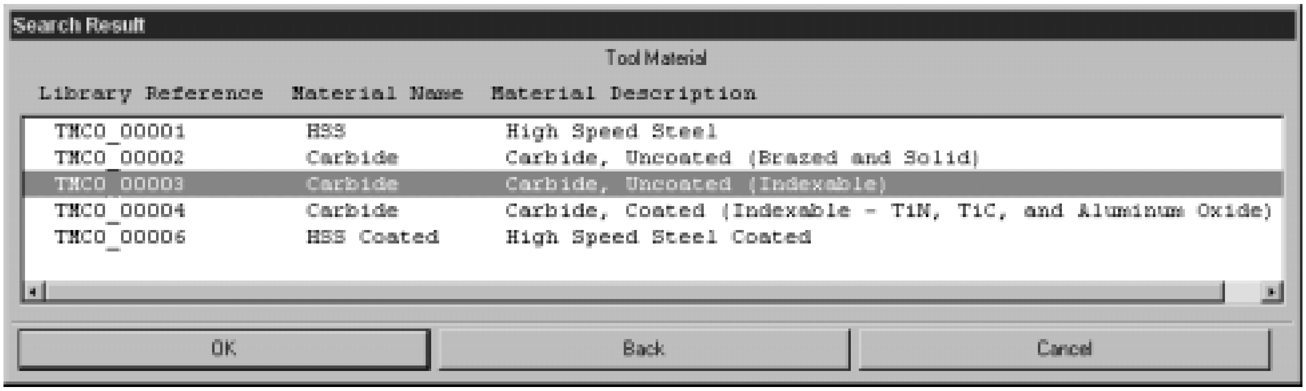
❑ If necessary, change the view of the Operation Navigator to the **Machine Tool** view.



❑ 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.

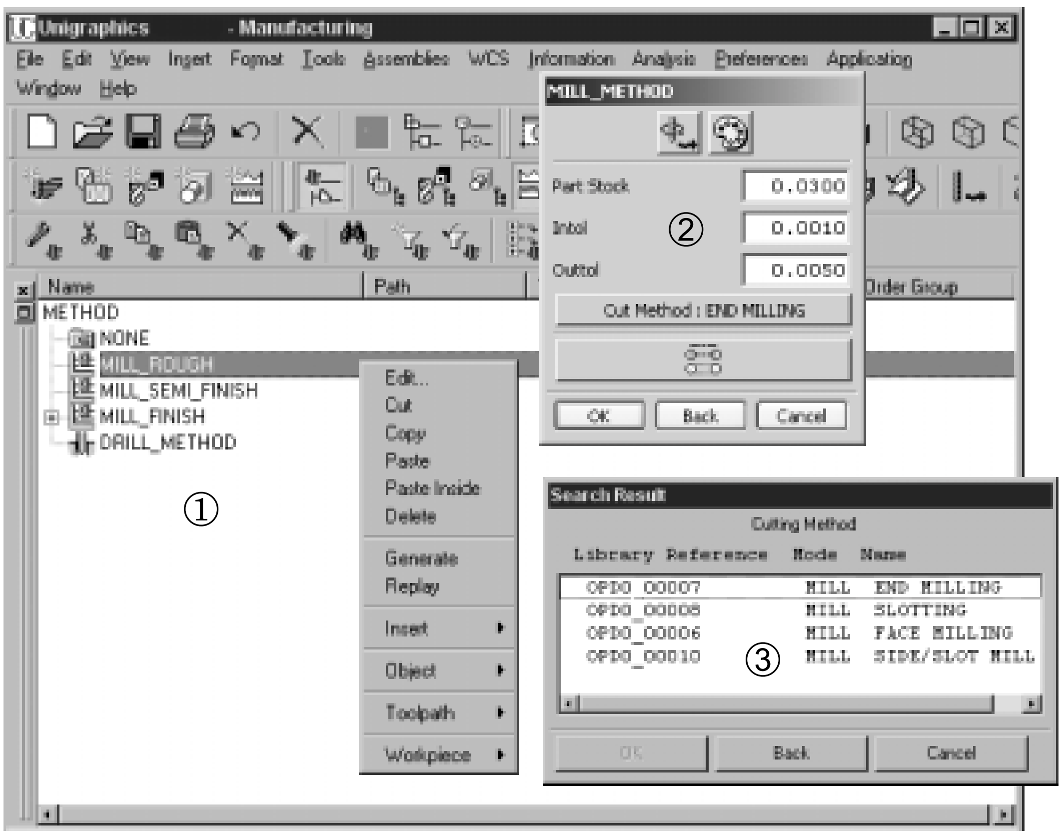
❑ Choose **OK** until your are returned to the Create Operation Menu.

You are finished with this activity and will be using this part file in the next activity.

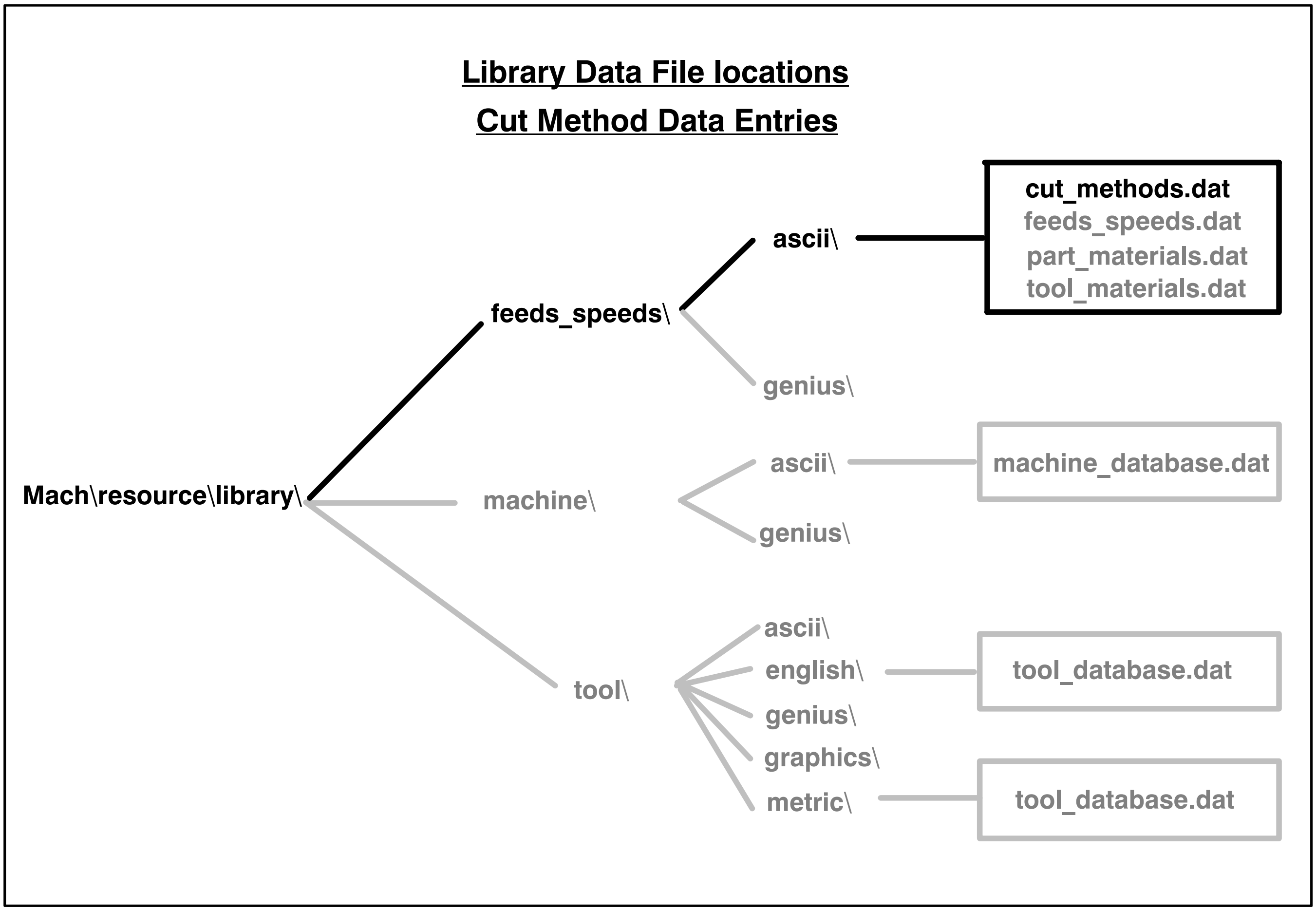
Cut Method Libraries

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. Selection of the *Cut Method:* button (②) from this dialog, 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
```

Activity 10–6: Cut Method Libraries

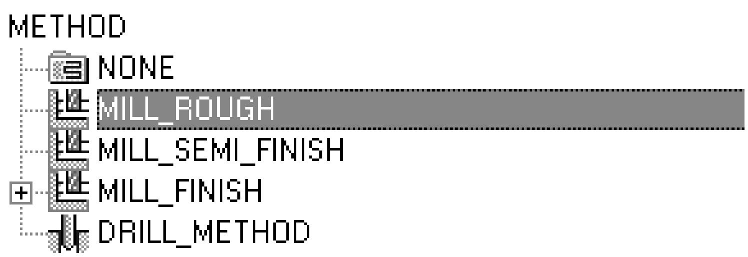
In this activity you will become familiar with the Cut Methods library.

Step 1 Accessing the Cut Method library.

❑ Continue with the opened part, **ama_lib_function.prt.**

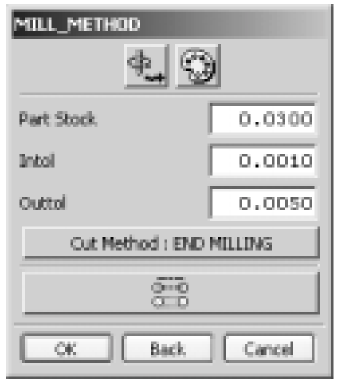
❑ If necessary, enter the **Manufacturing Application.**

❑ If necessary, change the view of the Operation Navigator to the **Machine Method** view.

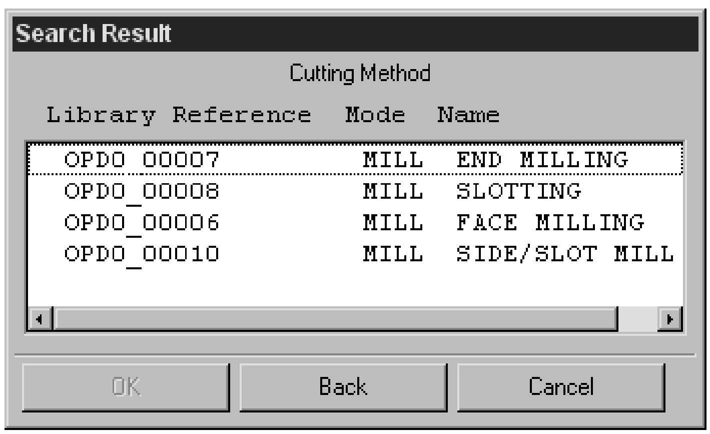


❑ Highlight **MILL_ROUGH**, select **MB3**, then Edit.

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.

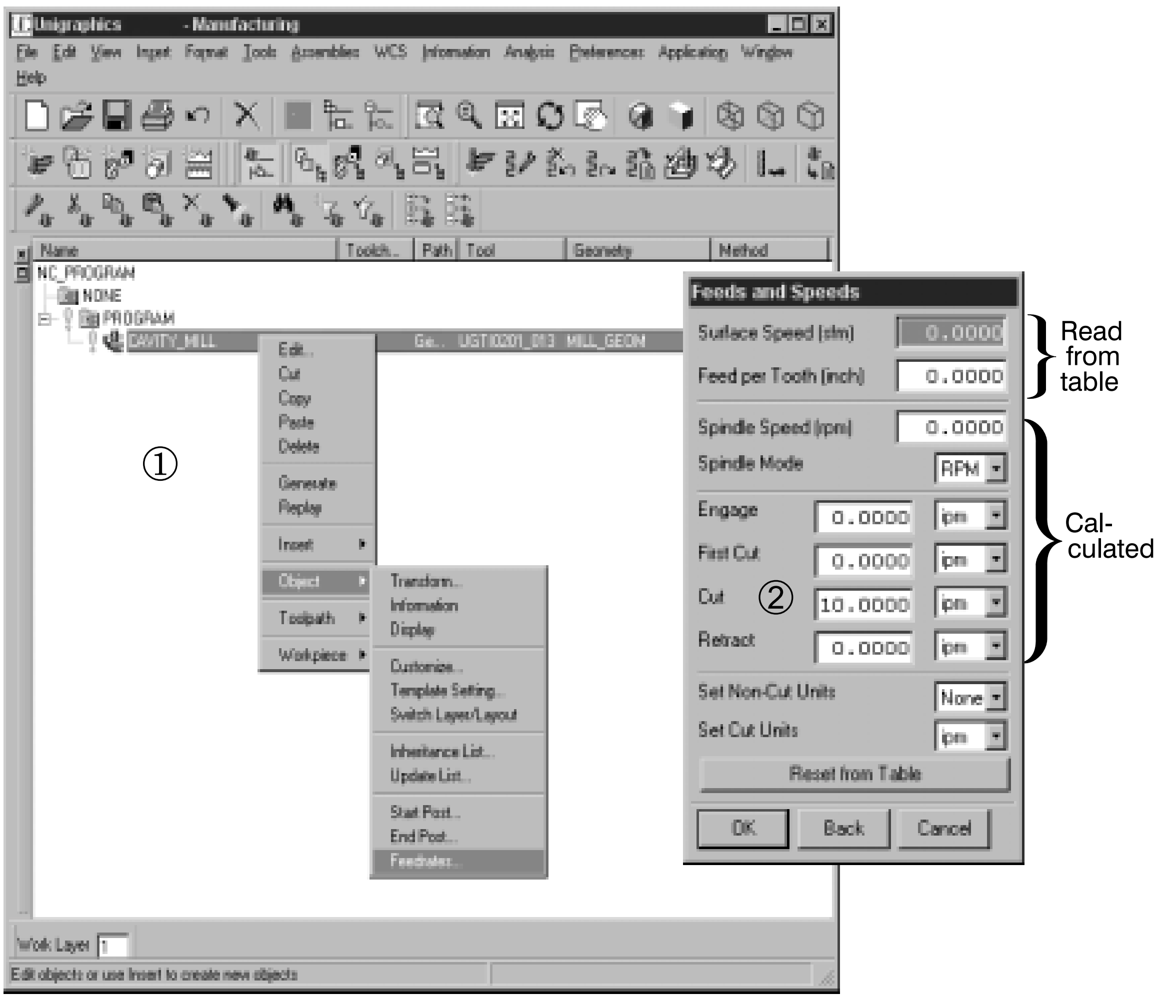
❑ Close the part.

You are finished with this activity.

Feeds and Speeds Libraries

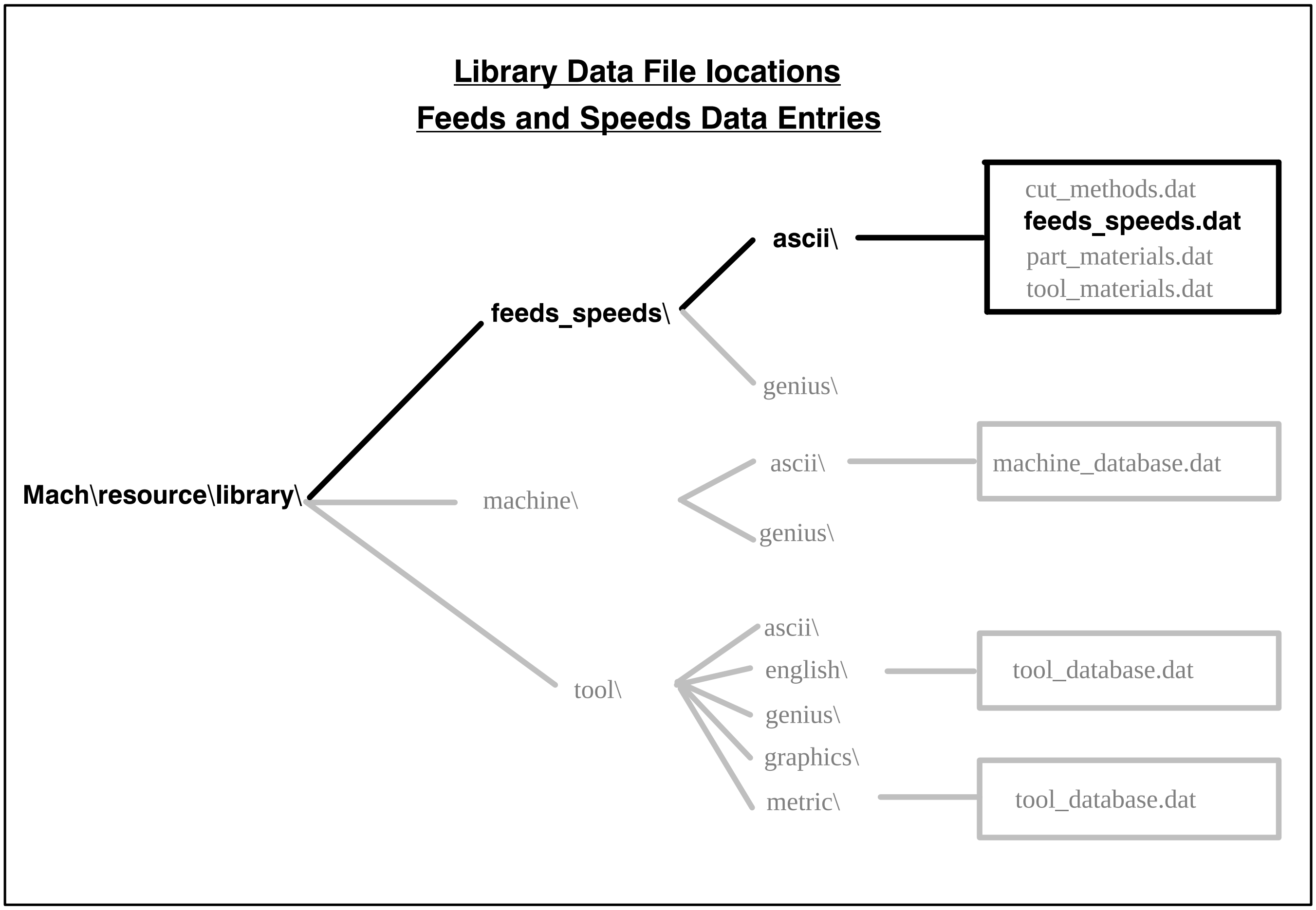
Feeds and Speeds libraries contains information which pertains to feeds and speeds used in the generation of an Operation.

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 is displayed. Selection of the *Reset from Table* button (②) from this dialog, 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.



10

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.

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
```

10

Activity 10–7: 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 **ama_lib_act_feeds_speeds.prt** and then rename it to ***_**lib_act_feeds_speeds.prt**.
- ❑ Enter the **Manufacturing Application.**

  The Operation Navigator is displayed.

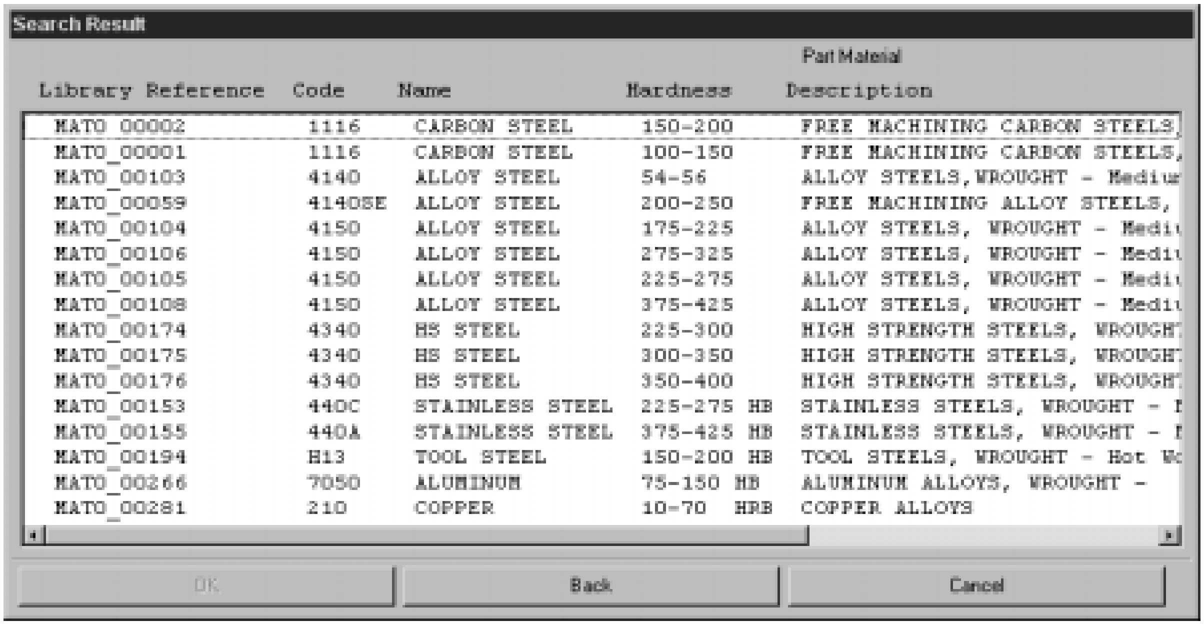
Step 2 Defining the Part Material.

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.



Library Reference	Code	Name	Hardness	Part Material 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 - Mediu
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**.

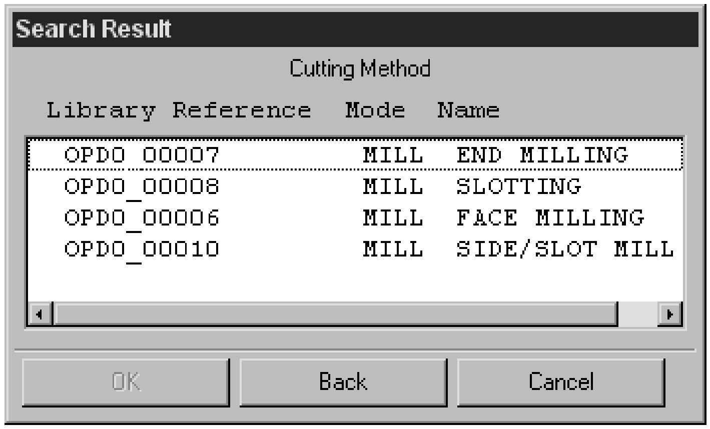
❑ Highlight **Workpiece**, then **MB3,** Edit.

The Mill Method dialog is displayed.

❑ Choose the **Cut Method** button.

10

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.

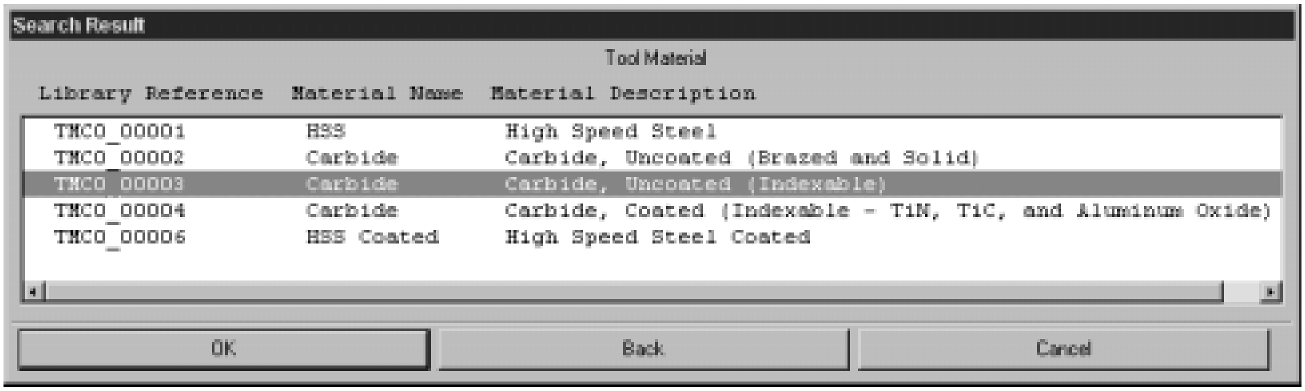
❑ Change the Operation Navigator to the **Tool View**.

❑ 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**.

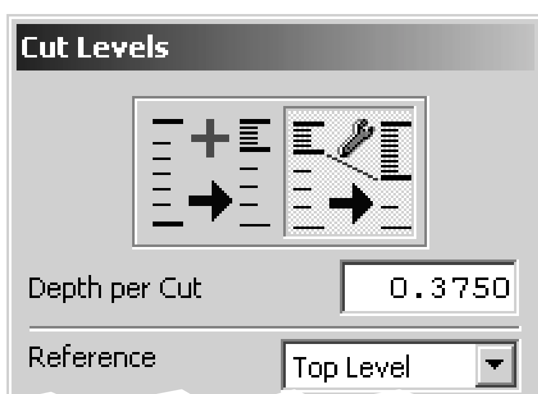
❑ Highlight the operation named **Cavity Mill,**then **MB3,** edit.

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**.



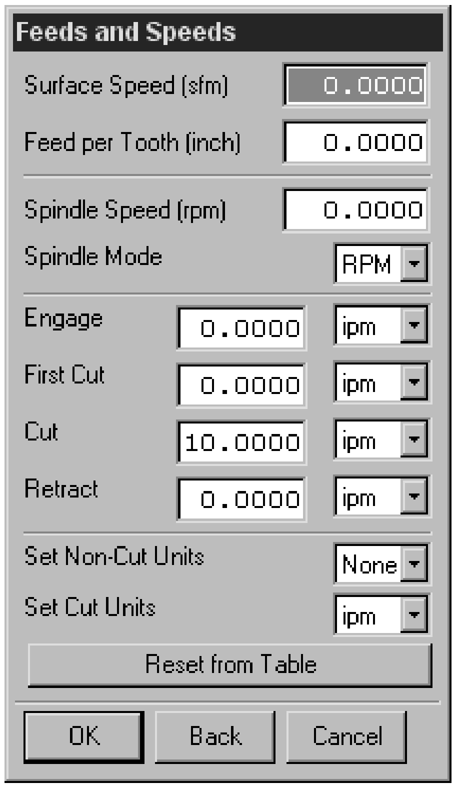
❑ Choose **OK** until the Create Operation dialog is displayed.

Step 6 Setting the Feeds and Speeds.

❑ Highlight the **Cavity_Mill** operation.

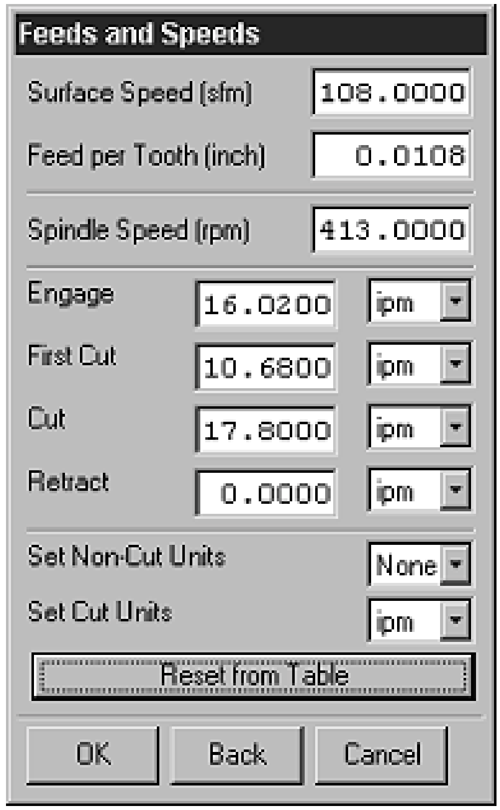
❑ Use MB3 and choose **Object→Feedrates**.

The Feeds and Speeds dialog is displayed.



❑ Choose the **Reset from Table** button.

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.

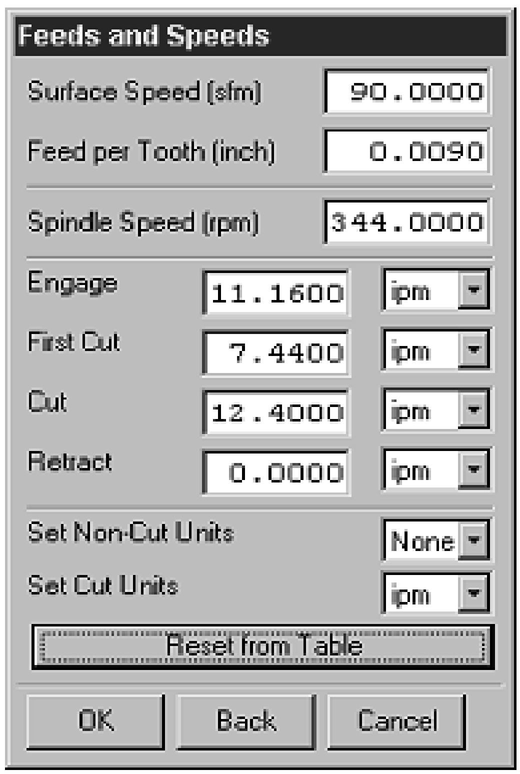
❑ Select **TMCO_00001** HSS on the list, then choose **OK** until you return to the Create menu.

❑ 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.

You have completed the activity and the lesson.

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

(This Page Intentionally Left Blank)

Hole Making
Lesson 11

PURPOSE

In this lesson, you will learn how to create and optimize a Hole Making program.

OBJECTIVES

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

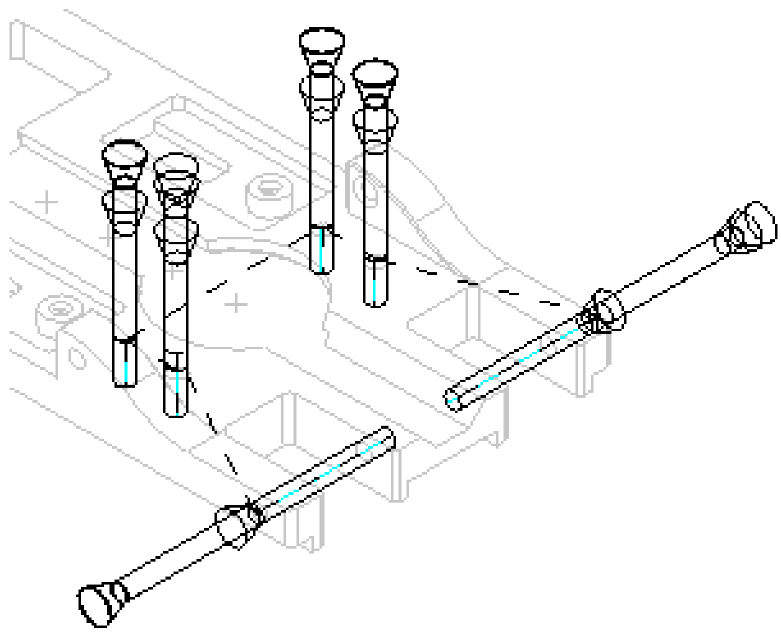
- Create a hole making program that machines simple, countersunk, and counterbore holes
- Identify where tools must be edited or created and apply the necessary changes
- Tag simple geometry so it will be recognized as machinable features
- Optimize the program

This lesson contains the following activities:

Activity **Page**

Hole Making Overview

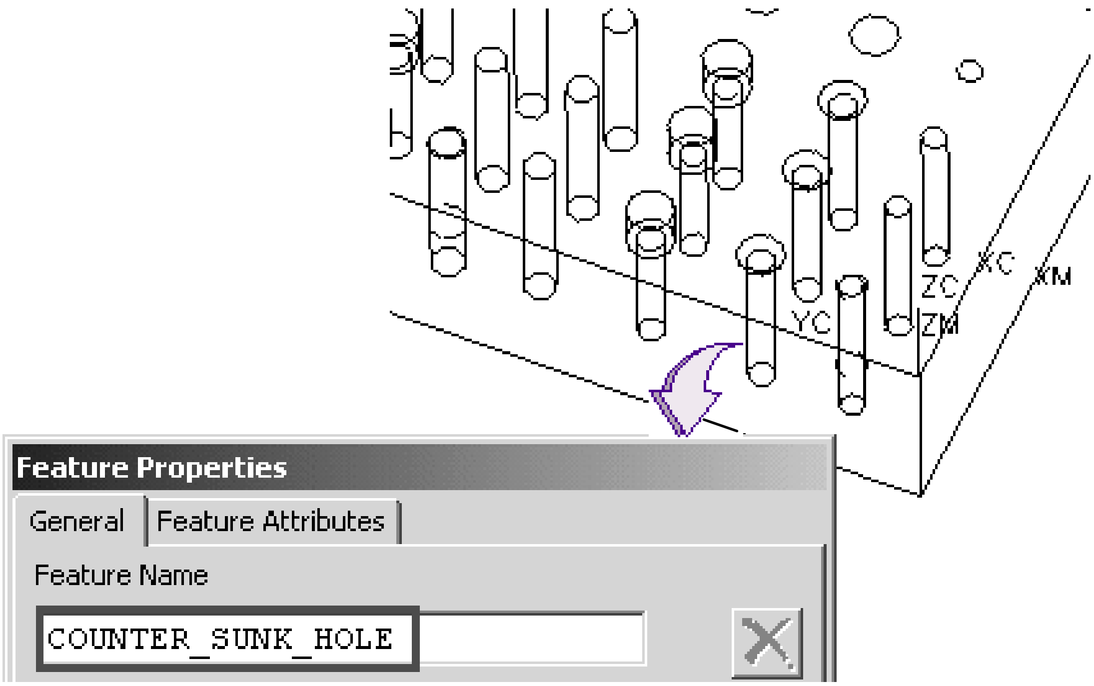
Hole Making is an advanced Manufacturing application that automates the creation of operations such as spot drilling, drilling, countersinking, counterboring, reaming, tapping, and deburring through the use of intelligent models containing manufacturing features (User Defined Features, User Defined Attributes, and UG Based Features) and embedded machining rules.



Manufacturing features contain information that allows the CAM system to make machining decisions based on rules defined in templates and applied through Knowledge Fusion. Manufacturing features most commonly consist of Unigraphics modeled features such as simple, countersunk, and counterbore holes and simple geometry such as points and arcs that contain attributed information describing the shape and size of holes. The system groups similar features together (simple holes of a particular diameter, for example), creates appropriate operations, chooses appropriate tools, specifies cut methods and machining parameters, and outputs optimized tool paths.

Templates

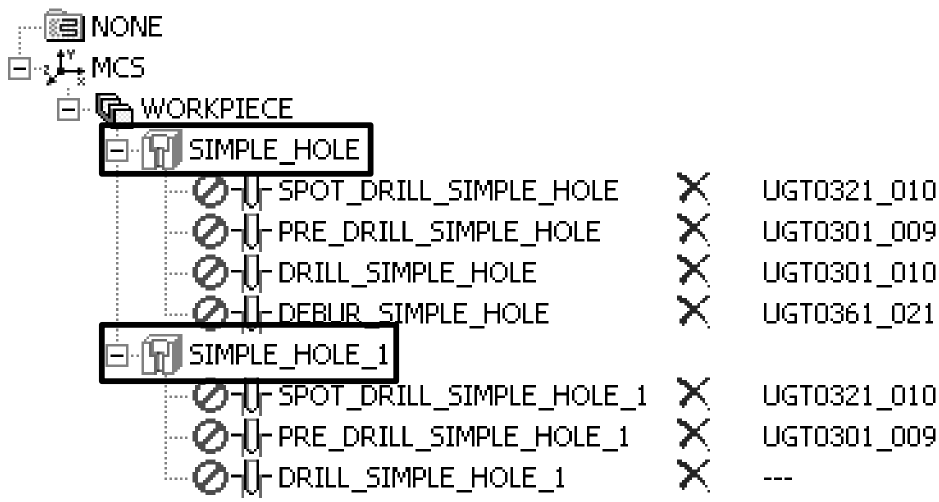
A template is a part file that contains predefined manufacturing features, feature groups, tools, operations, machining rules, and adopted operations. The feature groups defined in the template contain features modeled in various ways that the Hole Making processor will recognize in your part. A countersunk hole, for example, can be a fully modeled feature or simply a point with attributed information that describes its shape and size. The system makes intelligent decisions about how to machine these feature groups based on Knowledge Fusion Rules.



Knowledge Fusion Rules are associated with feature groups through Adopted Operations displayed in the Knowledge Fusion Navigator. This association allows the system to apply specific machining rules to feature groups so that the features can be machined correctly.

Feature Groups

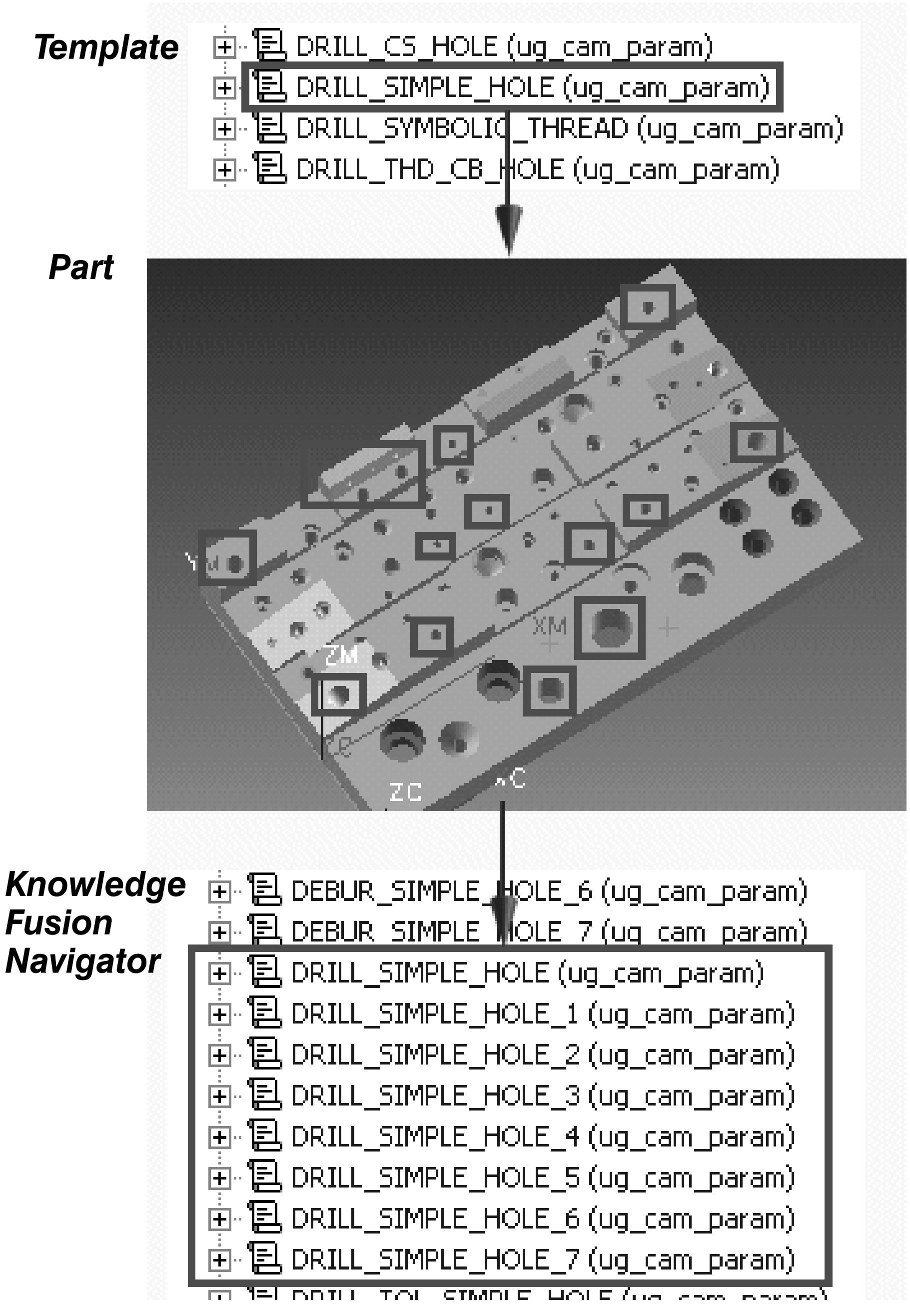
The Geometry view of the Operation Navigator associates manufacturing features with feature groups. Feature groups organize operations according to feature type. The feature group named SIMPLE_HOLE for example, contains all operations (spot drill, drill, ream) that need to be performed on simple holes of a particular diameter.



The Knowledge Fusion Navigator

The Knowledge Fusion Navigator displays adopted operations. Adopted operations are operations to which machining rules have been applied using Knowledge Fusion.

When you create a program to machine simple hole features for example, the adopted operations that apply to the machining of simple holes are copied into the part file. The machining rules embedded in these adopted operations determine which operations to use in the NC/CNC program based on the properties and attributes of the features in the part. The operations that are not needed are suppressed and are not displayed in the Operation Navigator.



To display the Knowledge Fusion Navigator you would choose **Application→Knowledge Fusion** from the menu bar. You would then choose the Knowledge Fusion Navigator tab in the resource bar.

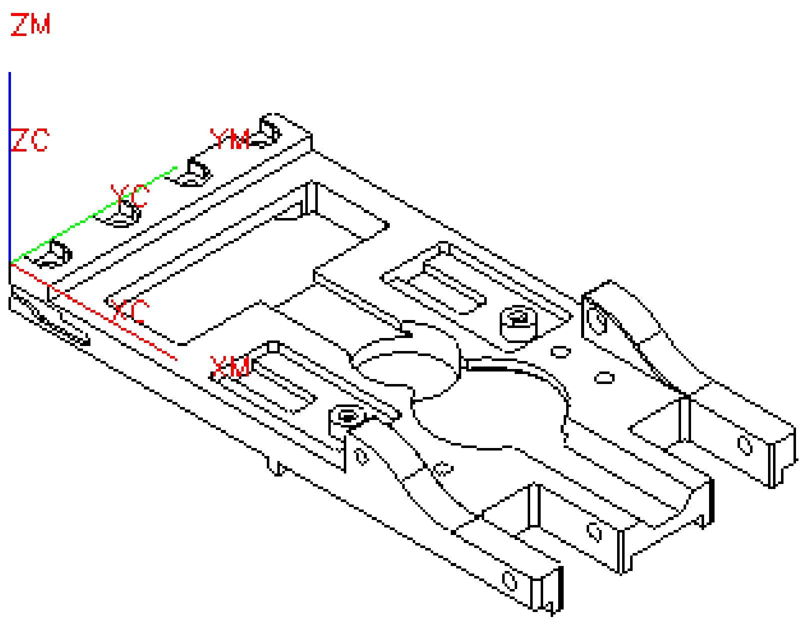
Editing Knowledge Fusion Rules requires a working knowledge of the language in which the Knowledge Fusion Rules are written and is typically not done by the end-user, so the Knowledge Fusion Navigator will not be emphasized in this course.

Activity 11–1: Machining Holes

In this activity, you will create hole making operations that machine simple, countersunk, and counterbore.

Step 1 Open an existing part, save with a new name and enter the Manufacturing Application.

❑ Open the part **ama_holemaking.prt.**



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

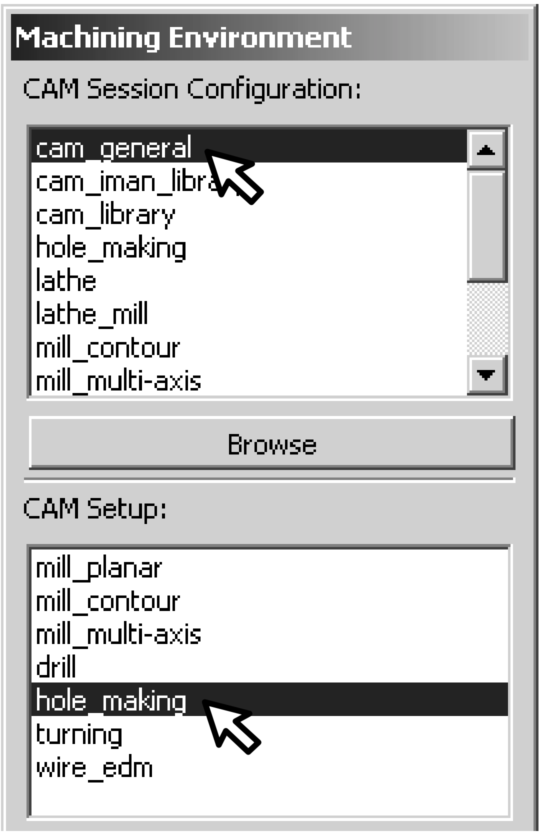
❑ Choose **Application→Manufacturing.**

Step 2 Define the Machining Environment.

The Machining Environment dialog displays because the part has not been saved in the Manufacturing application. The CAM Setup you choose determines the template that will be used to define the machining environment.

❑ Choose **cam_general** as the CAM Session Configuration.

❑ Choose **hole_making** as the CAM Setup.

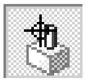


Hole_making is the standard CAM Setup supplied with the system. Other setups may also be available depending on your working environment.

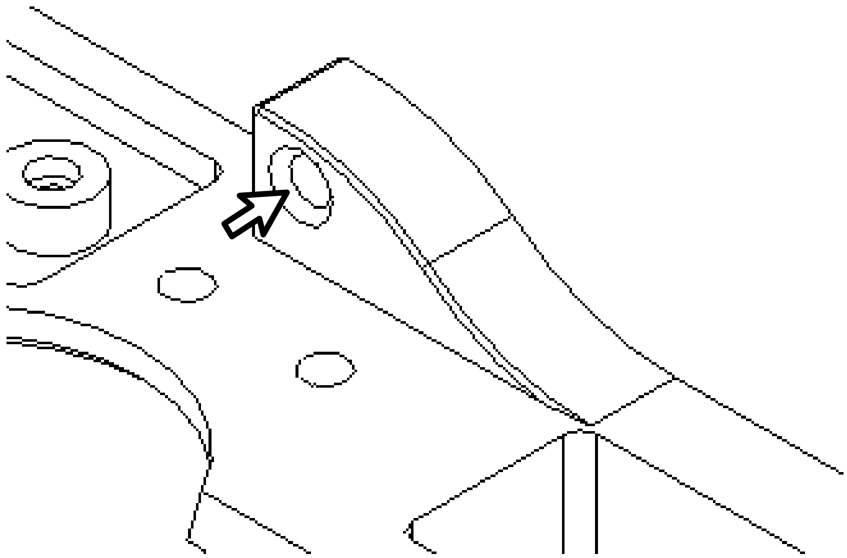
❑ Choose **Initialize**.

Step 3 Display Feature Properties.

You will display the feature name of a countersunk hole.

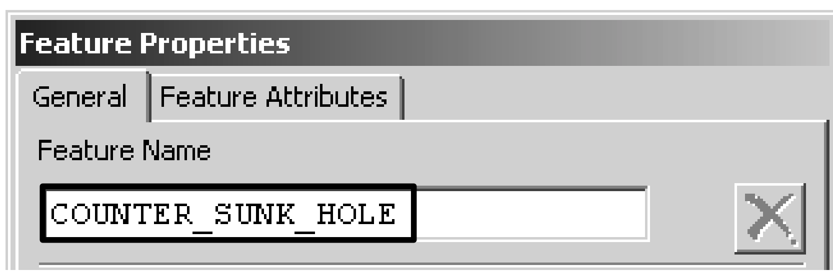
❑ Choose the **Select Features** icon. 

❑ Select the countersunk hole illustrated below.



❑ Choose **MB3,** then select **Properties**.

In the Feature Properties dialog, the feature name is COUNTER_SUNK_HOLE.

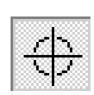


Because the hole making template also contains a feature called COUNTER_SUNK_HOLE, the system will recognize this feature and will apply the appropriate machining rules based on the shape, size, and surface finish of the feature.

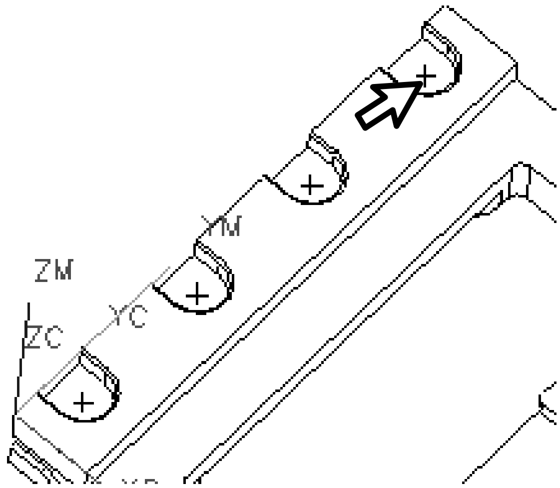
❑ **Cancel** the Feature Properties dialog.

Step 4 Display Object Properties.

You will display the properties of an attributed point.

❑ Choose the **Select General Objects** icon. 

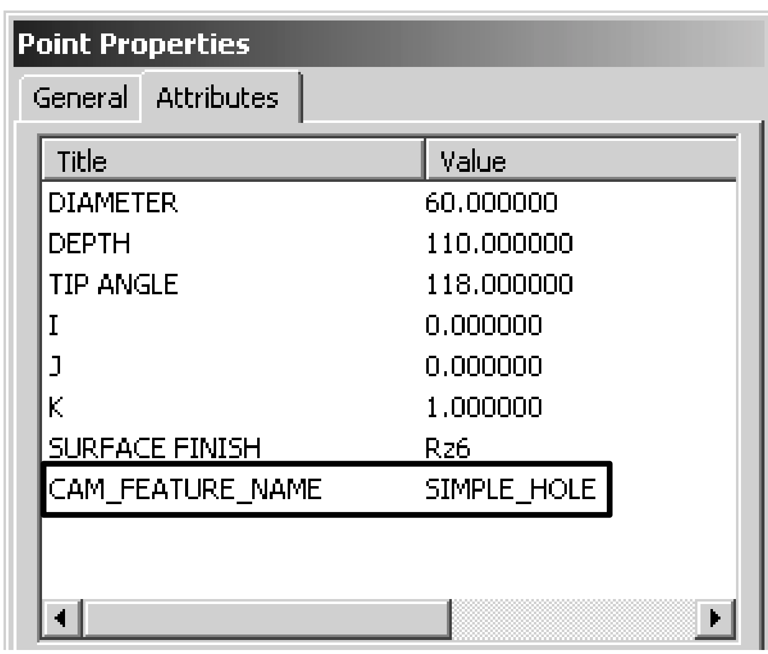
❑ Select the point illustrated below.



❑ Choose **MB3,** then select **Properties**.

❑ In the Point Properties dialog, choose the **Attributes** tab.

Notice the feature name is SIMPLE_HOLE.



Since the hole making template also contains an attributed point called SIMPLE_HOLE, the system will recognize this point as a feature and will apply the appropriate machining rules based on the attributes.

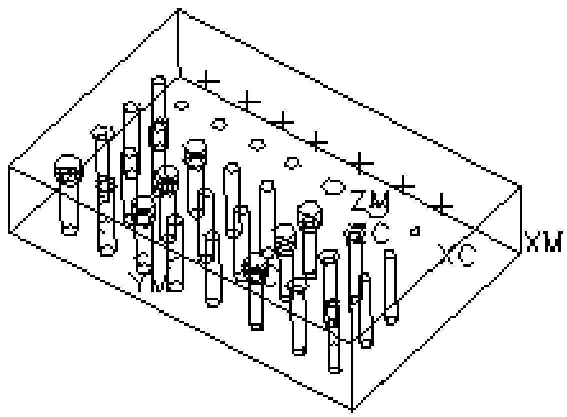
❑ **Cancel** the Point Properties dialog.

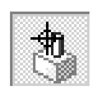
11

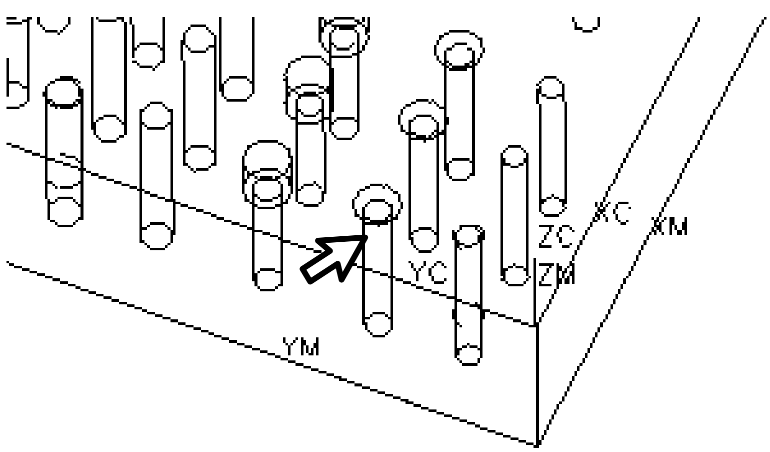
Step 5 Examine the Hole Making Template.

You will take a look at the Hole Making template. Remember, you specified this template when you chose hole_making as the CAM Setup. You will see that this template contains a countersunk hole feature and an attributed point similar to the ones you just looked at in your part.

- ❑ In the menu bar, choose **Window** → **More Parts**.
- ❑ Choose **hole_makin**g in the Change Displayed Part dialog and then choose **OK**.

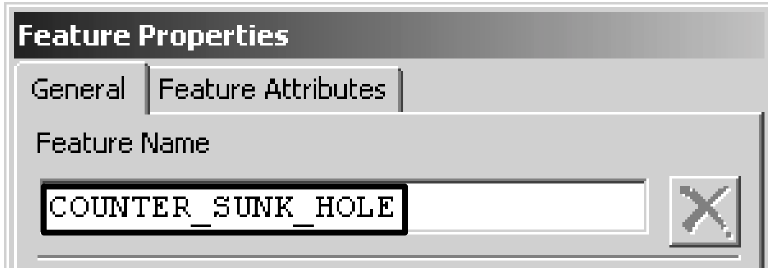


- ❑ Choose the **Select Features** icon. 
- ❑ Select the countersunk hole illustrated below.



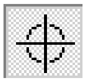
- ❑ Use **MB3**, choose **Properties**.

❑ In the Feature Properties dialog, notice that the feature name, COUNTER_SUNK_HOLE, is the same as the one in your part.

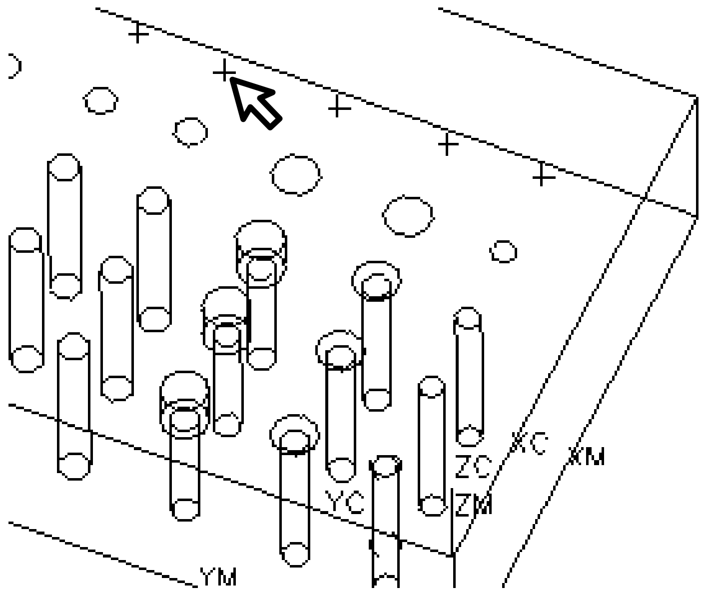


Because your part contains machinable features called COUNTER_SUNK_HOLE, the system will recognize all occurrences of this feature and will apply the appropriate rules for machining.

❑ **Cancel** the Feature Properties dialog.

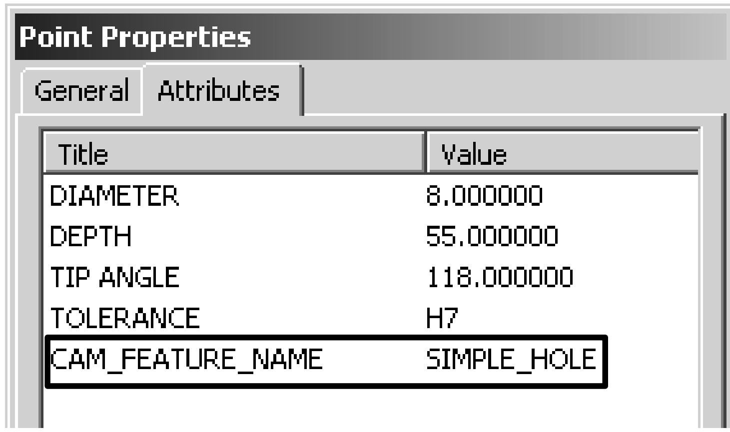
❑ Choose the **Select General Objects** icon. 

❑ Select the point illustrated below.



❑ Use **MB3**, choose **Properties**.

❑ In the Point Properties dialog, choose the **Attributes** tab. Notice the feature name is SIMPLE_HOLE.



Because your part contains attributed points called SIMPLE_HOLE, the system will recognize all occurrences of this point as a machinable feature and will apply the appropriate rules for machining.

- ❑ **Cancel** the Point Properties dialog.

- ❑ In the menu bar, choose **Window** → ***_holemaking** to display the part.

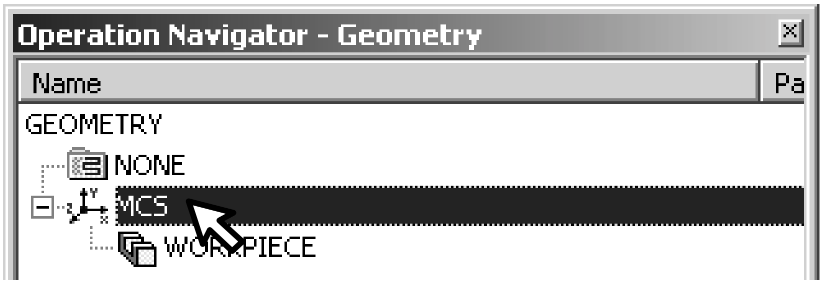
Step 6 Specify an Appropriate Tool Axis.

You will specify a tool axis allowing the system to create tool paths for holes at any angle (this would be applicable for a 5-axis machine only).

- ❑ Double-click the **Operation Navigator** tab in the resource bar and undock the Operation Navigator (using the Ctrl key) so it displays in a separate window.

- ❑ Choose the **Geometry View** icon in the toolbar to display the Geometry View of the Operation Navigator.

- ❑ Double-click on the **MCS** icon in the Operation Navigator.



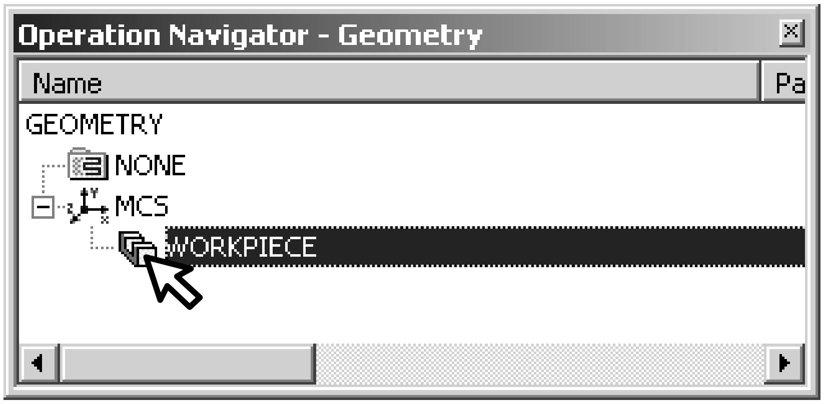
❑ Be sure the **Tool Axis** option is set to **All Axes**.

❑ Choose **OK** to accept the tool axis.

Step 7 Specify the Part Geometry.

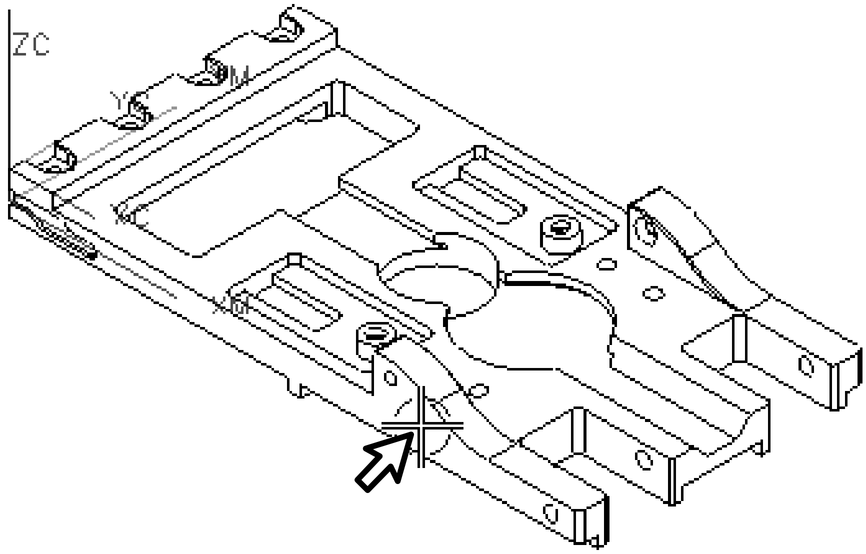
You will specify the solid body as the part geometry.

❑ Double-click on the **Workpiece** icon in the Operation Navigator.



❑ Be sure the **Part** icon is chosen and choose **Select**.

❑ Select the solid body.



❑ Choose **OK** to accept the Part Geometry.

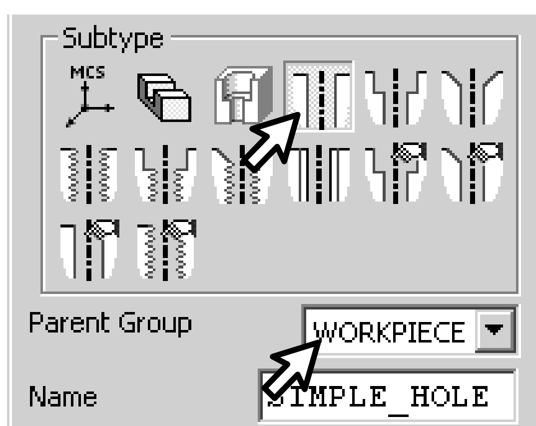
❑ Choose **OK** to accept the MILL_GEOM dialog.

Step 8 Create Operations to Machine Simple Holes.

You are now ready to create the operations that machine the simple holes.

❑ Choose the **Create Geometry** icon in the toolbar. 

❑ Choose the **SIMPLE_HOLE** icon and be sure the **Parent Group** option is set to **Workpiece**.



❑ Choose **OK** to create the operations.

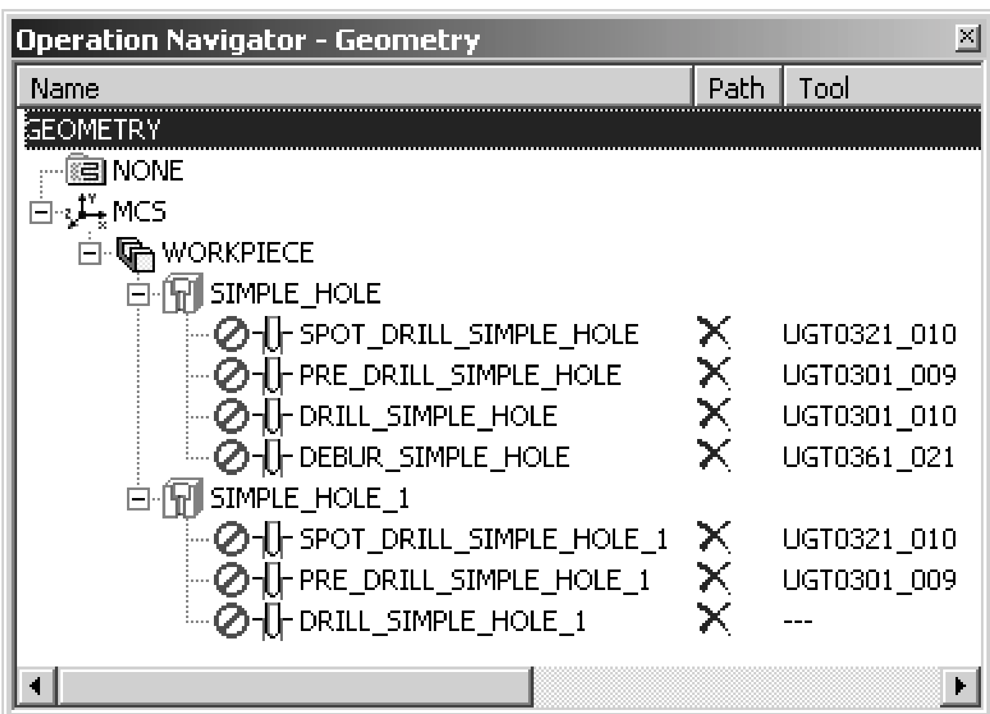
The system will take a few moments to process.

❑ Once the processing is complete, choose **OK** in the SIMPLE_HOLE dialog to accept **DIAMETER** as the classification criteria.

By choosing Diameter as the classification criteria, you have specified that each feature group will contain operations associated with simple holes of a particular diameter.

❑ Display the Geometry view of the Operation Navigator and expand the feature groups.

Notice in the Operation Navigator that two simple hole feature groups, SIMPLE_HOLE and SIMPLE_HOLE_1 were created. The simple holes in this part had only two diameters. The simple holes of one diameter require spot drilling, pre-drilling, drilling, and deburring. The simple holes of the other diameter require spot drilling, pre-drilling, and drilling.



NOTE Your results may differ slightly from those illustrated above. Since the system assigns feature group numbers randomly, the operations listed in one feature group here (SIMPLE_HOLE) might be listed in the other feature group in your part.

Step 9 View the Information Window.

The Information window displays a record of the data that was processed and the output that was generated.

❑ Enlarge the Information window and examine the contents.

The processor used the SIMPLE_HOLE template, found ten simple hole features in the part and classified these features into two feature groups according to diameter.

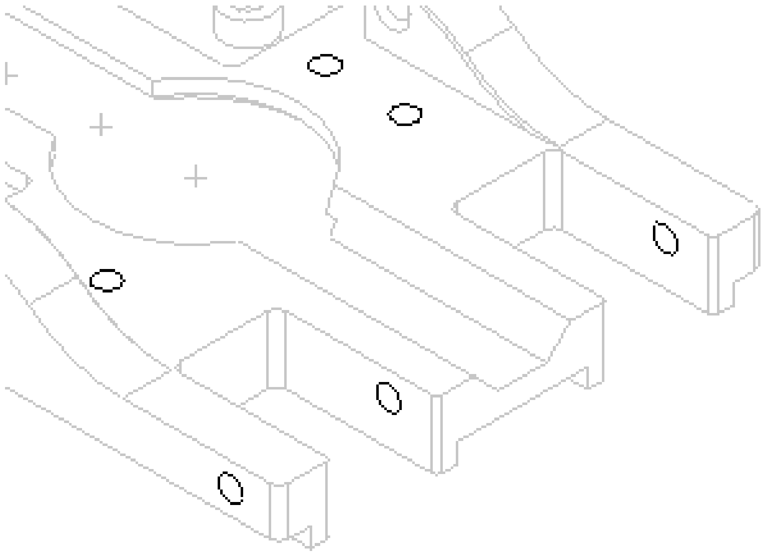
```
Instantiating Feature Geometry Template : SIMPLE_HOLE

*****************************************************
Identifying all features of name: SIMPLE_HOLE

10 features were found with feature name: SIMPLE_HOLE

Classifying features according to the following crite

      DIAMETER

 2 group(s) were found after classification criteria
```

❑ Dismiss the Information window.

Step 10 Examine the Feature Groups.

❑ Double-click on the SIMPLE_HOLE feature group icon in the Operation Navigator.

The associated features, all of which have the same diameter, are highlighted on the part. Again, your results may be different for this particular feature group because the system numbers the feature groups randomly. If double-clicking on SIMPLE_HOLE does not highlight the holes illustrated below, double-click on SIMPLE_HOLE_1.



❑ Choose **Cancel** to dismiss the Simple Hole dialog.

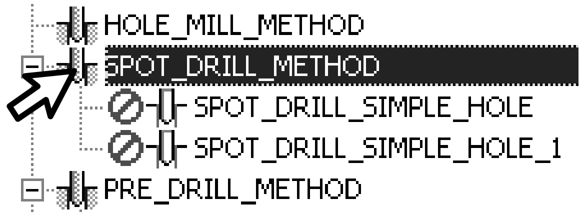
Step 11 Perform a Tool Query.

The Machining Method view allows you to perform a tool query which displays the attributes the machining rules used when selecting tools for each machining method.

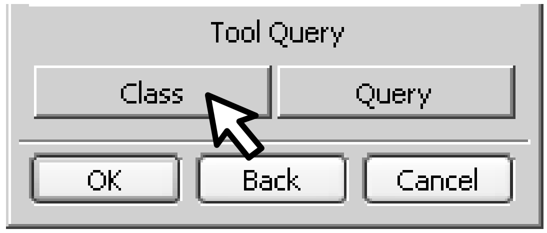
❑ Choose the **Machining Method View** icon in the toolbar to display the Machining Method view of the Operation Navigator.

❑ Expand the Machining Method Groups.

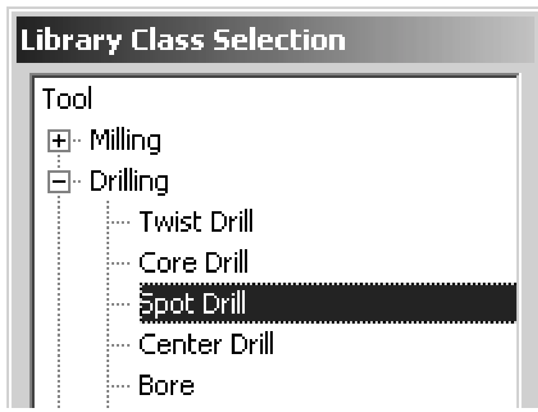
❑ Double-click on the SPOT_DRILL_METHOD group icon.



❑ Choose **Class** under Tool Query.



The Library Class Selection dialog indicates that Spot Drill is the class of tool used for spot drilling operations.



❑ **Cancel** the Library Class Selection dialog.

❑ Choose **Query** under Tool Query.

This dialog displays the parameters for this tool class and the specific Knowledge Fusion Rule that was applied. Editing these items requires a working knowledge of the language in which the Knowledge Fusion Rules are written and is typically not done by the end-user, so it will not be covered here.

❑ **Cancel** the Query from Method dialog.

❑ **Cancel** the hole_making dialog.

Step 12 Generate the Spot Drill Tool Paths.

You will display the tools and generate the tool paths for the Spot Drill operations.

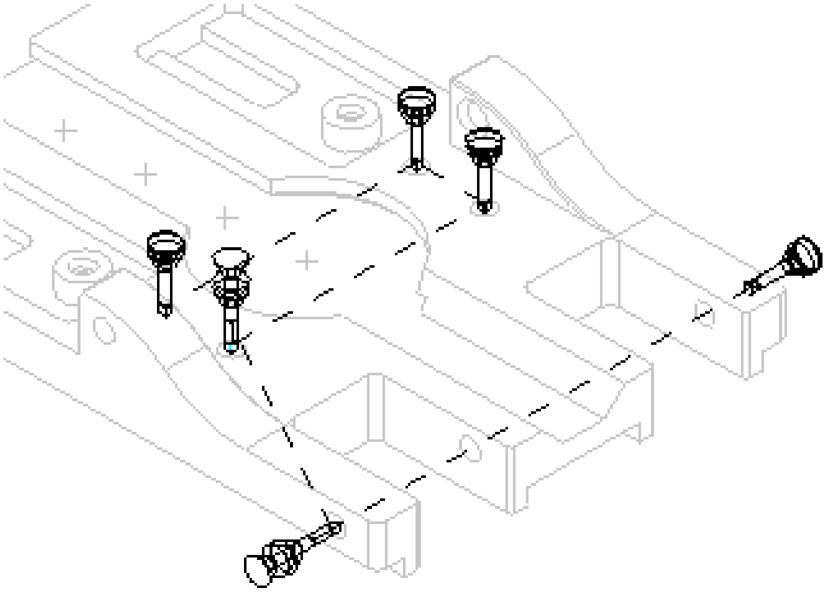
❑ Display the Program Order View of the Operation Navigator.

❑ Double-click on the SPOT_DRILL_SIMPLE_HOLE operation to display the operation dialog.

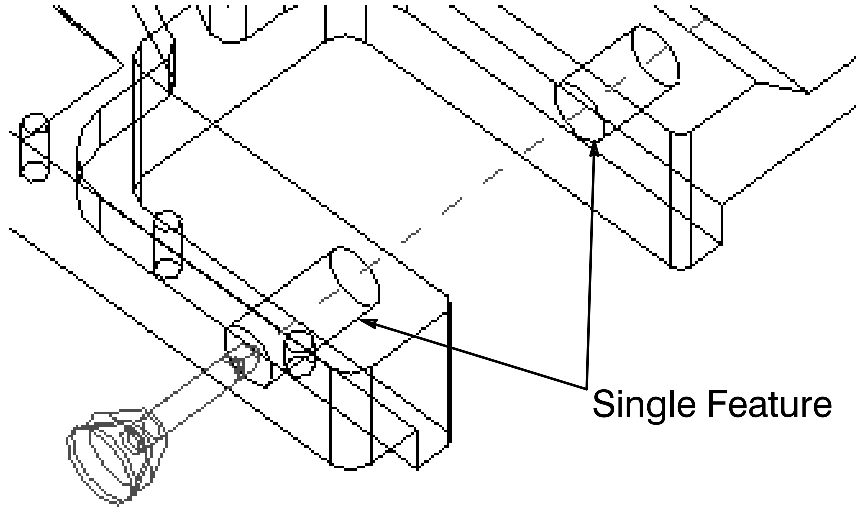
❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the Display Options.

❑ Generate the tool path. Your tool path might differ slightly.



Notice at the end of the part that only the outermost horizontal holes are spot drilled and that the operation does not attempt to incorrectly spot drill the inner holes. This occurs because each pair of holes (the two holes drilled from the left and the two holes drilled from the right) was correctly modeled as a single feature.

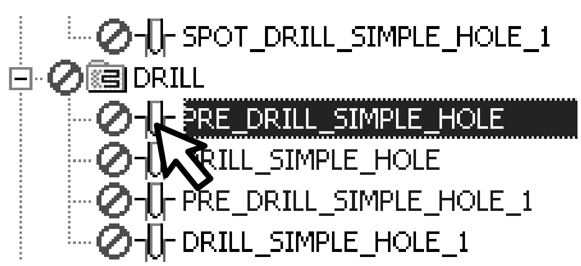


❑ **OK** to accept the operation.

Step 13 Generate the Pre-Drill Tool Path.

You will display the tools and generate the tool path for the PRE_DRILL_SIMPLE_HOLE operation.

❑ Double-click on the PRE_DRILL_SIMPLE_HOLE operation to display the operation dialog.

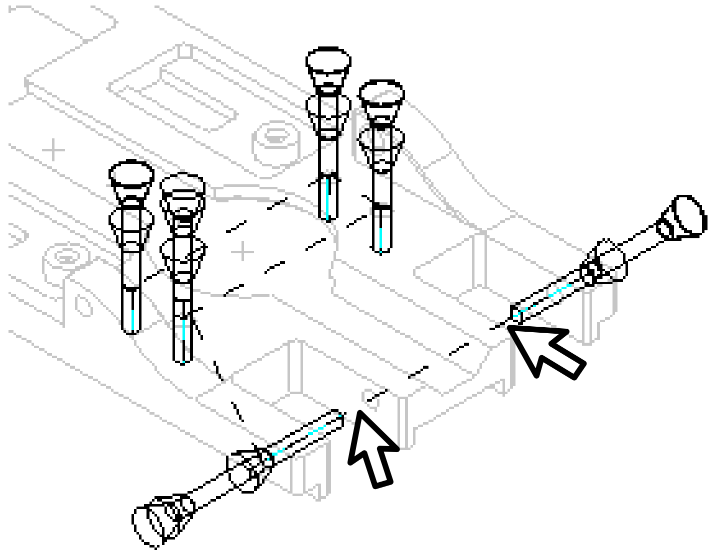


❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the Display Options.

❑ Generate the tool path.

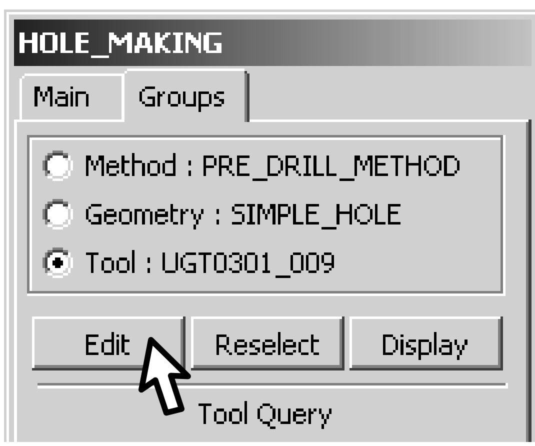
Notice that the tool is not long enough to drill through the fork at the end of the part. This is a case where you will need to edit the length of the tool.



Step 14 Edit the Tool Length.

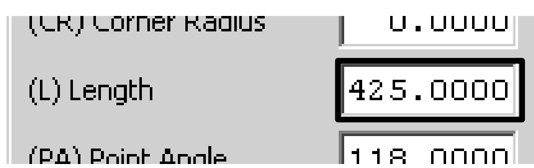
❑ Choose the **Groups** tab.

❑ With **Tool:UGT0301_009** chosen, choose **Edit**.



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

❑ Change the Length to **425.**



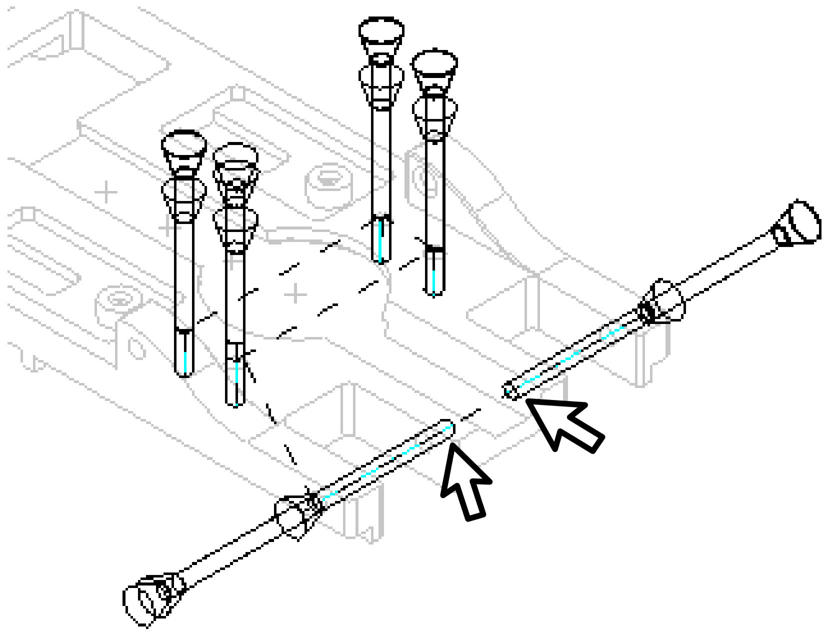
❑ Choose **OK.**

11

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

❑ Choose the **Main** tab.

❑ Generate the tool path.



The tools are now long enough to drill the full depth of the holes.

❑ **OK** to complete the operation.

Step 15 Generate the Drill Tool Path

You will display the tools and generate the tool path for the DRILL_SIMPLE_HOLE operation.

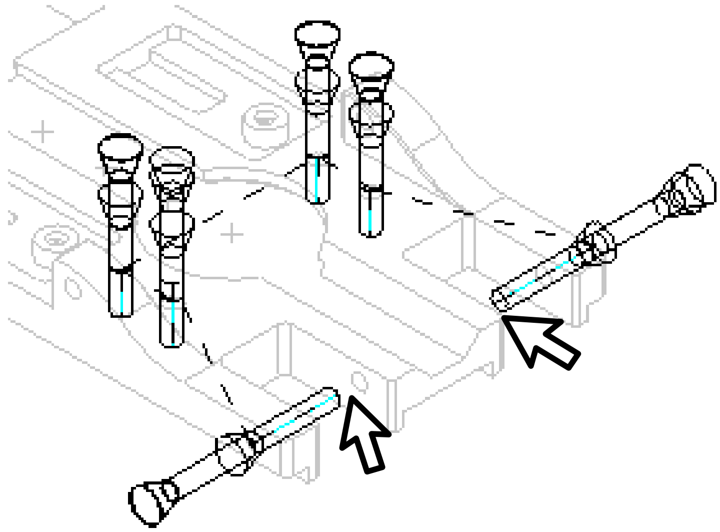
❑ Double-click on the DRILL_SIMPLE_HOLE operation to display the operation dialog.

❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

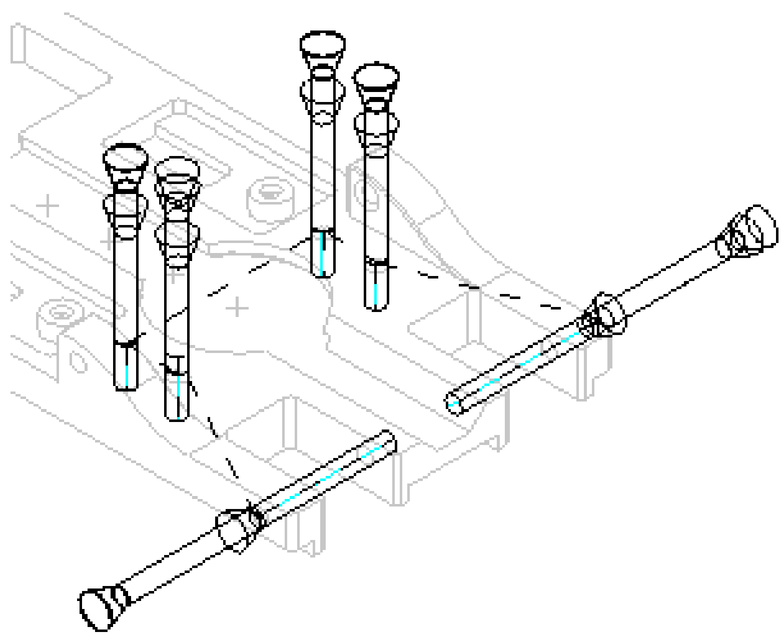
❑ **OK** to accept the Display Options.

❑ Generate the tool path.

Again, the tool is not long enough to drill through the fork at the end of the part. You will need to edit the length of the tool as you did before.



❑ Change the length of the tool to 425 and regenerate the tool path as you did before.



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

Step 16 Generate the Spot Drill Tool Path.

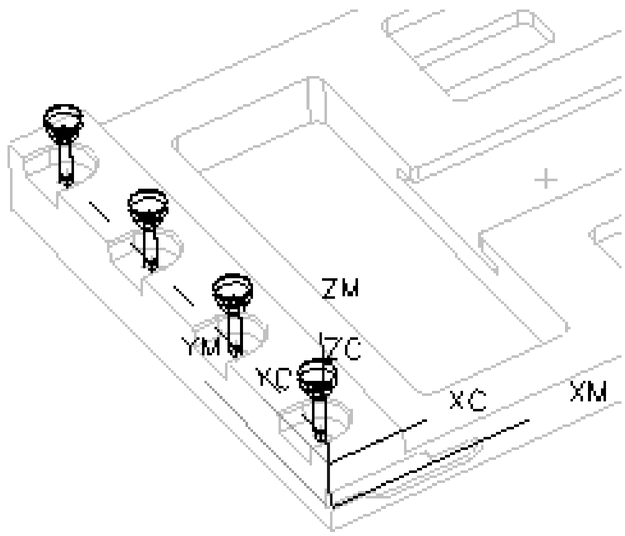
You will display the tools and generate the tool path for the SPOT_DRILL_SIMPLE_HOLE_1 operation.

❑ Double-click on the SPOT_DRILL_SIMPLE_HOLE_1 operation to display the operation dialog.

❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the Display Options.

❑ Generate the tool path.



This operation spot drills the 60 mm diameter holes defined by attributes. If you look closely, you will notice that there are no hole features modeled.

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

Step 17 Generate the Pre-Drill Tool Path.

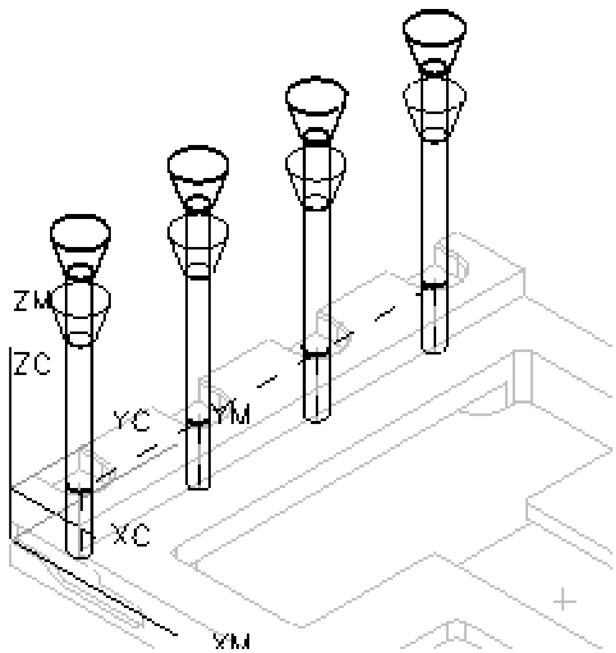
You will display the tools and generate the tool path for the PRE_DRILL_SIMPLE_HOLE_1 operation.

❑ Double-click on the PRE_DRILL_SIMPLE_HOLE_1 operation to display the operation dialog.

❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the Display Options.

❑ Generate the tool path.



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

Step 18 Generate the Drill Tool Path.

You will display the tools and generate the tool path for the DRILL_SIMPLE_HOLE_1 operation.

❑ Double-click on the DRILL_SIMPLE_HOLE_1 operation to display the operation dialog.

❑ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the Display Options.

❑ Generate the tool path.

This time you receive an error message stating that no tool has been specified.

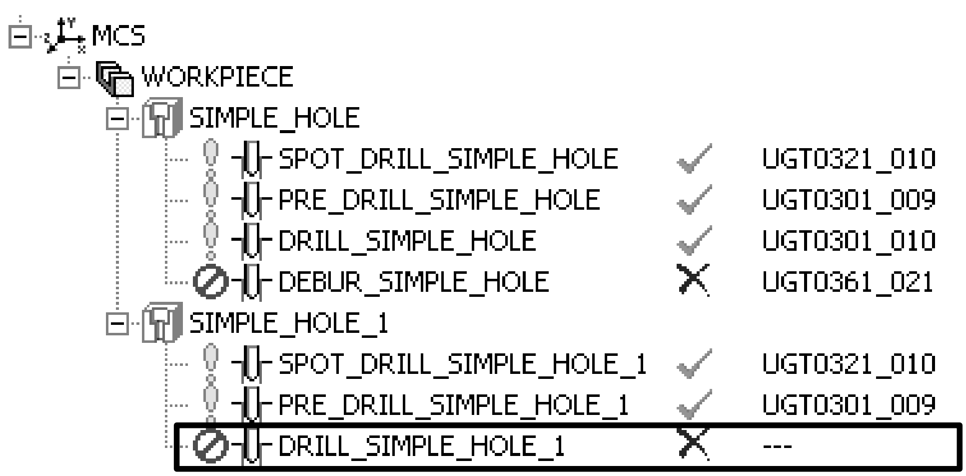
The hole making template does not contain the tool required to drill the 60 mm diameter holes.

- ❑ Choose **OK** to the error message.
- ❑ Choose **OK** to the operation.

Step 19 Adding a Tool.

You should check to be sure that every operation contains a tool. The best way to do this is to look at the Geometry View of the Operation Navigator. You can then add the required tools.

- ❑ Display the Geometry View of the Operation Navigator.
- ❑ Expand the SIMPLE_HOLE feature groups until you find the operation that does not have a tool.



- ❑ You can display the diameter and depth of the holes defined by the attributed points by selecting one of the points and choosing **MB3→Properties** as you did earlier.

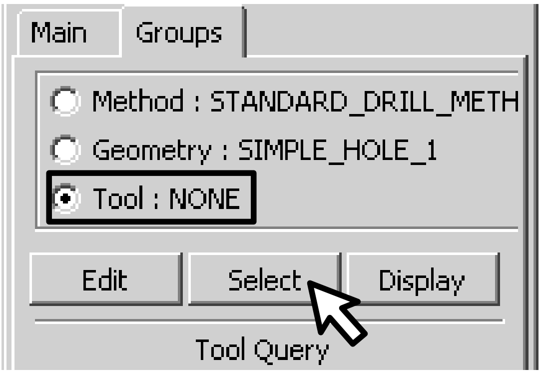
You will find that the holes have a diameter of 60 and a depth of 110. Now you can create the required tool.

- ❑ Double-click the **DRILL_SIMPLE_HOLE_1** operation icon to edit the operation.

- ❑ Choose the Groups tab.

  Notice that the Tool option says **NONE**.

- ❑ Choose **Select**.



- ❑ Choose **New**.

- ❑ Choose the **STD_DRILL** icon and then **OK** to accept it.

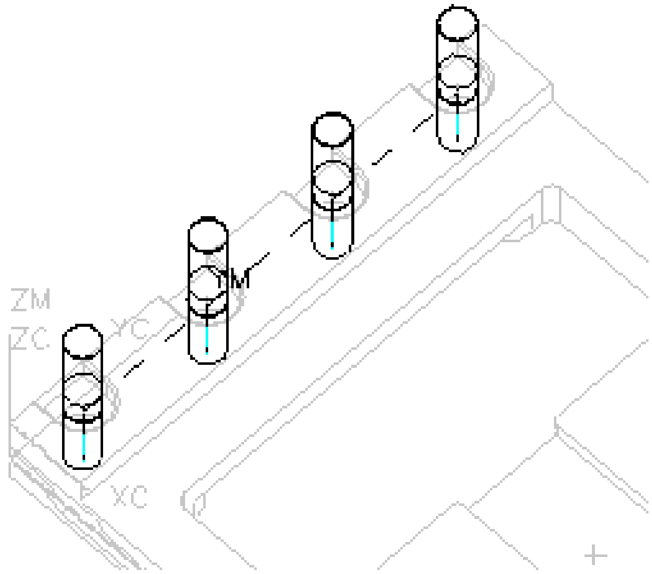
- ❑ Key in **60** in the Diameter field.

- ❑ Key in **200** in the Length field.

- ❑ Choose **OK** to create the tool.

- ❑ Choose the **Main** tab.

❑ Generate the tool path.



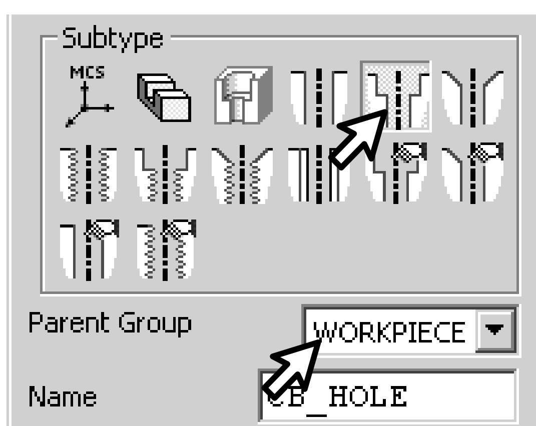
❑ Choose **OK** to complete the operation.

Step 20 Creating Operations to Machine the Counterbore Holes.

Next, you will create the operations that machine the counterbore holes.

❑ Choose the **Create Geometry** icon in the toolbar.

❑ Choose the **CB_HOLE** icon and be sure the **Parent Group** option is set to **Workpiece**.



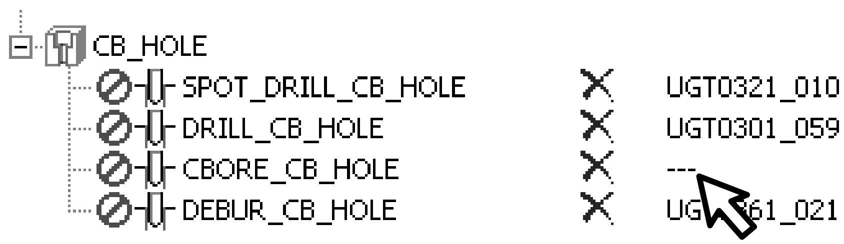
❑ Choose **OK** to create the operations.

The system will take a few moments to process.

❑ Once the processing is complete, choose **OK** in the CB_HOLE dialog to accept **HOLE DIAMETER** and **C-BORE DIAMETER** as the classification criteria.

❑ Dismiss the Information window.

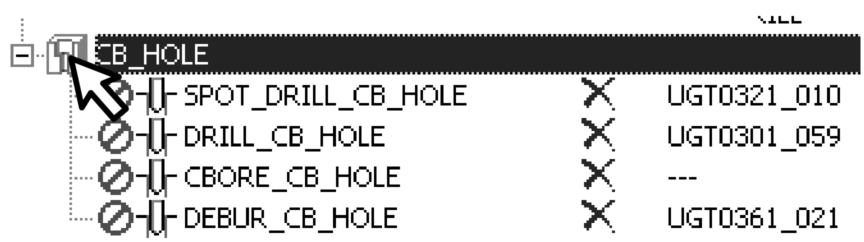
Notice that one of the operations does not have a tool.



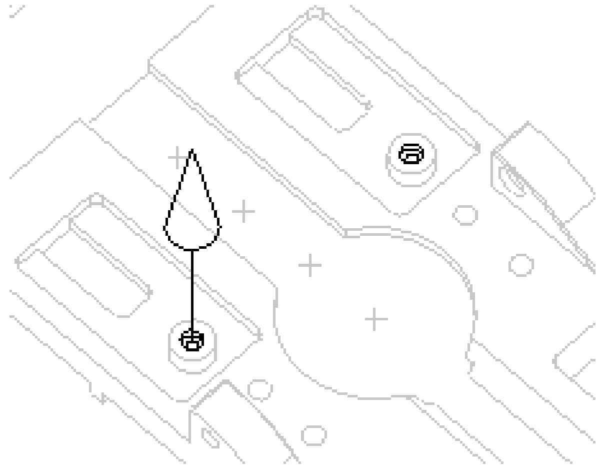
Step 21 Adding a Tool.

You must first identify the features that are associated with the CB_HOLE feature group. You can then analyze the feature to identify the diameter and depth of the holes.

❑ Double-click on the CB_HOLE feature group icon.

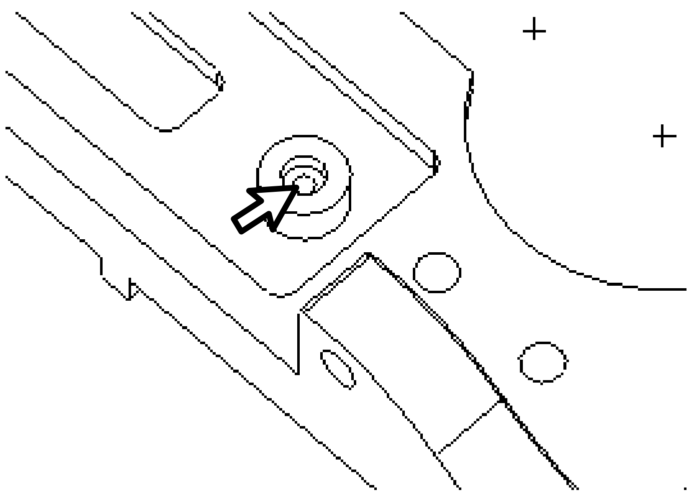


❑ Choose **Display**.



The vector clearly identifies one of the features associated with the CB_HOLE feature group.

- ❑ **Cancel** the CB_HOLE dialog.
- ❑ Choose **Information→Feature** from the toolbar.
- ❑ Select the counterbore hole illustrated below.



- ❑ **OK** the Feature Browser dialog.

The Information window indicates that the counterbore diameter is 50 and the counterbore depth is 20.

```
Feature Parameters for: COUNTER_BORE_HOLE(51)
---------------------------------------------
Feature Type - COUNTER_BORE_HOLE(51)  THRU
Cbore Diameter        =      50.00000000000
Cbore Depth           =      20.00000000000
```

❑ Dismiss the Information window.

Now you can create the required tool.

❑ Double-click the **CBORE_CB_HOLE** operation icon to edit the operation.

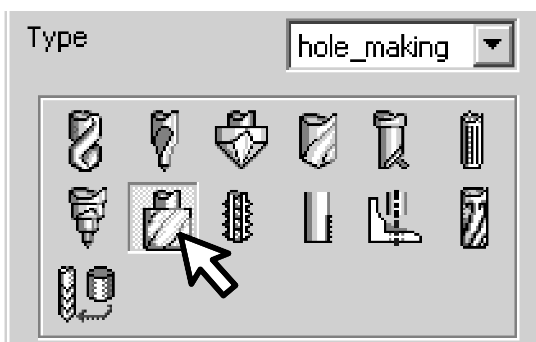
❑ Choose the Groups tab.

Notice that the Tool option says **NONE**.

❑ Choose **Select**.

❑ Choose **New**.

❑ Choose the **COUNTER_BORE** icon and **OK** to accept it.

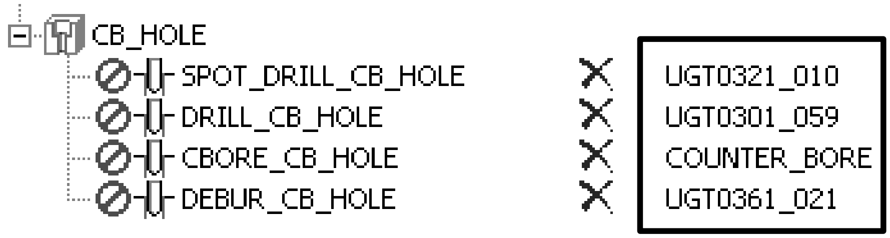


❑ Key in **50** in the Diameter field.

❑ Choose **OK** to create the tool.

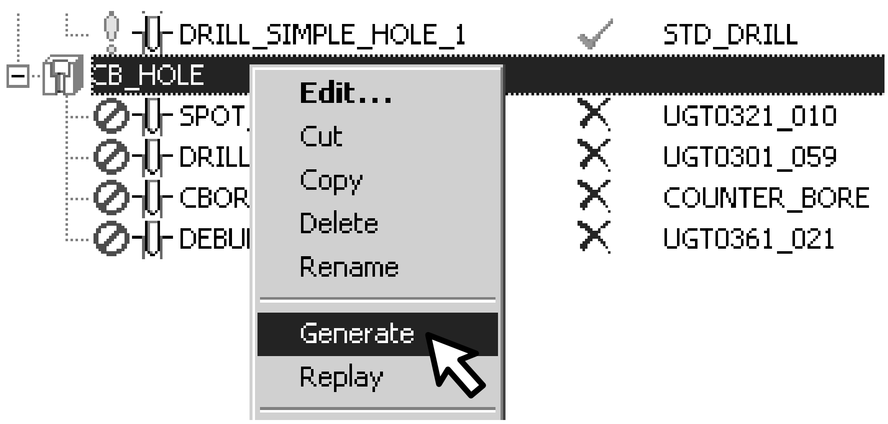
❑ Choose **OK** to complete the operation.

All of the operations within the CB_HOLE feature group now contain tools.

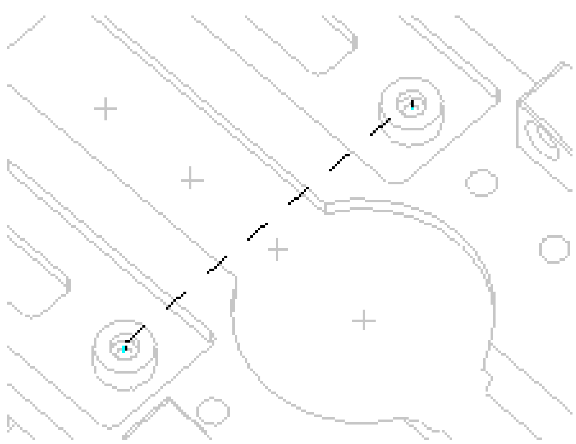


Step 22 Generate the Tool Paths for the Counterbore Holes.

❑ Highlight the CB_HOLE feature group and choose **MB3→Generate** to generate the tool paths.



❑ Choose **OK** to accept each one of the tool paths.



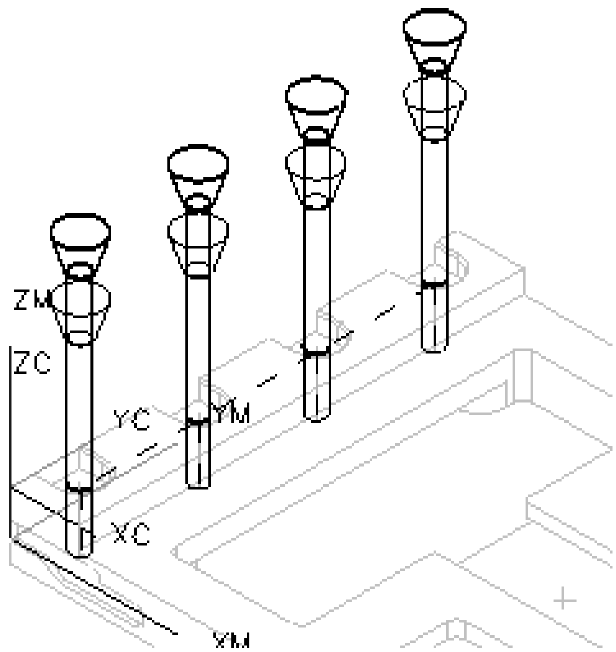
❑ **Save** the part file.

You have completed this activity.

Next, you will learn how to tag simple geometry such as points and arcs so that the hole making processor recognizes them as machinable features.

Tagging

The hole making processor recognizes NX based features such as simple holes, countersunk holes, counterbore holes, and symbolic threads as machinable features and applies appropriate machining rules based on their shape, size, and other attributes such as surface finish. Simple geometry such as points and arcs, however, can only be recognized by the hole making processor if they are tagged. Recall in the previous activity that you were able to machine holes where only tagged points represented the holes.



You were able to machine these points because they had been previously tagged as simple holes with a specific diameter and depth. Tagging allows you to apply attributes such as feature name, diameter, depth, tip angle, and surface finish so that the system can recognize simple geometry as machinable features and apply appropriate machining rules.

Activity 11–2: Tagging Points

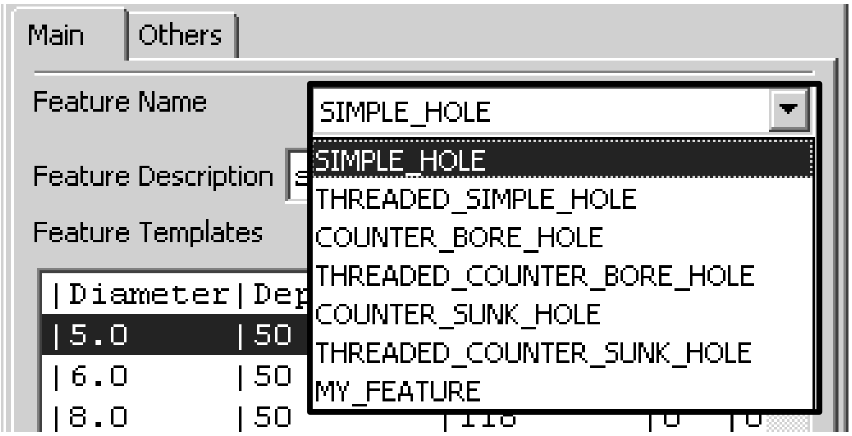
In this activity, you will learn to tag points so that the hole making processor recognizes them as simple holes.

Step 1 Tag Points with Attributes.

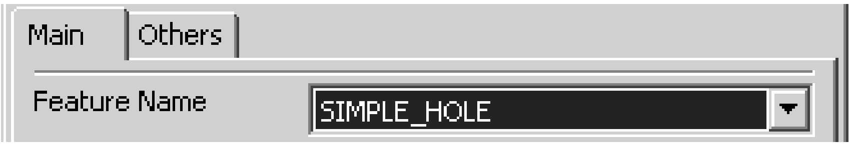
❑ Continue using *****_holemaking.prt.**

❑ Choose **Tools→Tagging→Point Tagging** from the menu bar.

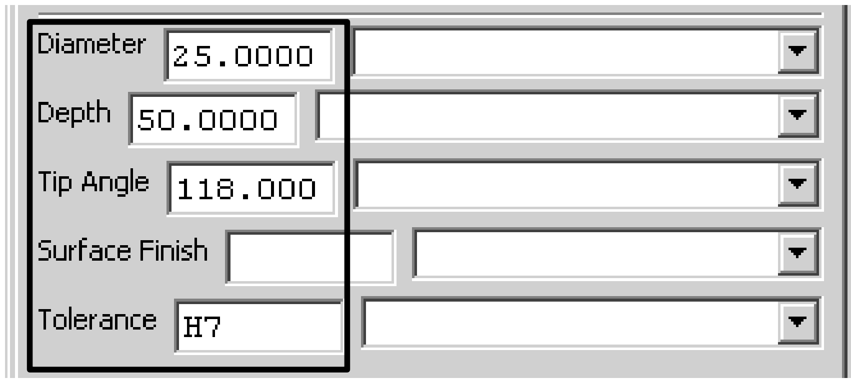
❑ Display the menu options next to Feature Name to see the different feature names that can be attributed.



❑ Be sure SIMPLE_HOLE is specified as the Feature Name.

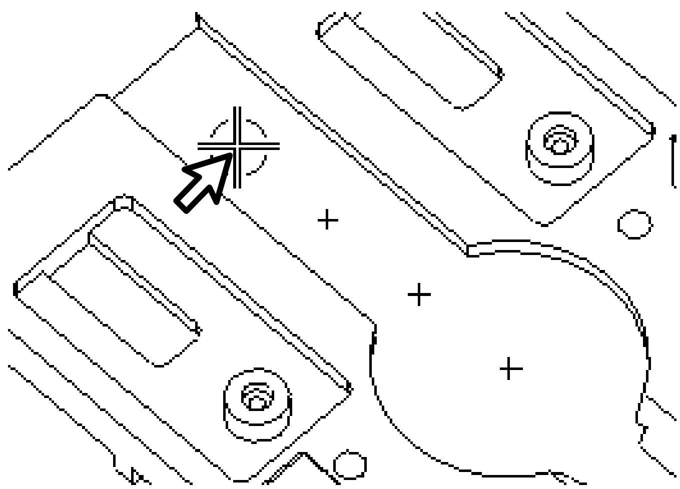


❑ Fill in the following values in the Tagging Attributes to Points dialog. Be sure the Surface Finish field has been blanked out and that the Tolerance field says **H7**.



Specifying a tolerance of H7 will cause the hole making processor to include a reaming operation.

❑ Select the point as illustrated below.



❑ Choose **Apply**.

❑ Select each of the remaining three points, choosing **Apply** after selecting each point so that the attributes are applied to each one.

❑ Choose **OK** to accept the Tagging Attributes to Points dialog.

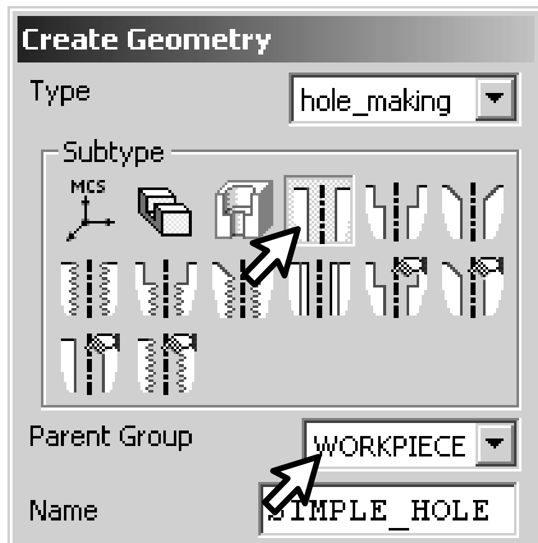
❑ Choose **OK** to the message dialog to complete the tagging.

Step 2 Create Operations to Machine the Simple Holes.

You can now create the operations that machine the simple holes.

❑ Choose the **Create Geometry** icon in the toolbar.

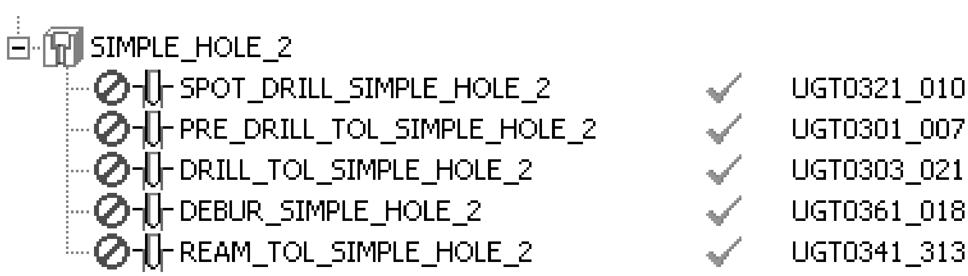
❑ Choose the **SIMPLE_HOLE** icon and be sure the **Parent Group** option is set to **Workpiece**.



❑ Choose **OK** to create the operations.

❑ Once the processing is complete, choose **OK** in the SIMPLE_HOLE dialog to accept **DIAMETER** as the classification criteria.

❑ Dismiss the Information window.



Step 3 Generate the Tool Paths for the Simple Holes.

❑ Highlight the SIMPLE_HOLE_2 feature group and **MB3→Generate** to generate the tool paths.

❑ Choose **OK** to accept each one of the tool paths.

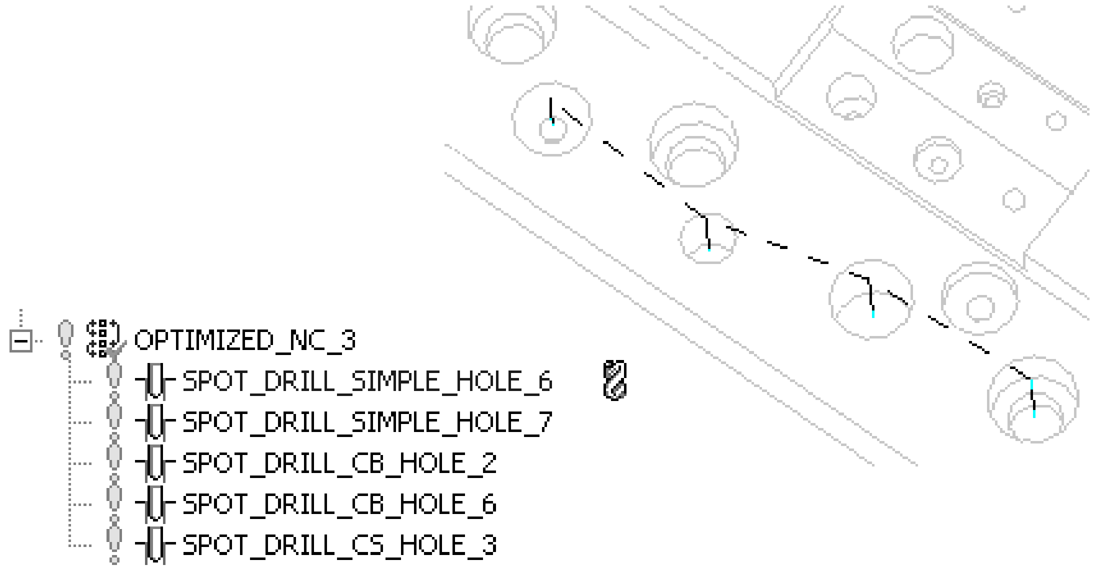
If you like, you can double-click on each operation, choose the **Edit Display** icon under Tool Path, set the **Tool Display** option to **3-D** and Replay the tool path to see the tools.

❑ **Save** the part file.

You have completed this activity.

Optimization

Optimization improves machining efficiency by consolidating tools to minimize the number of tools used, reordering operations to eliminate redundant tool changes, and resequencing features to minimize the tool travel distance.



The Optimization dialog displays three options:

- **Consolidate Tools** causes the program to use as few tools as possible without compromising the effectiveness of the machining.
- **Minimize Tool Changes** reorders operations to minimize the number of tool changes that occur within the program. The system does this without violating operation order constraints. For instance a drilling operation will never be placed before a spot drilling operation.
- **Create Optimization Group** creates groups containing operations that define an optimal tool path. Features cut with the same tool are resequenced to minimize tool travel distance within and between operations.

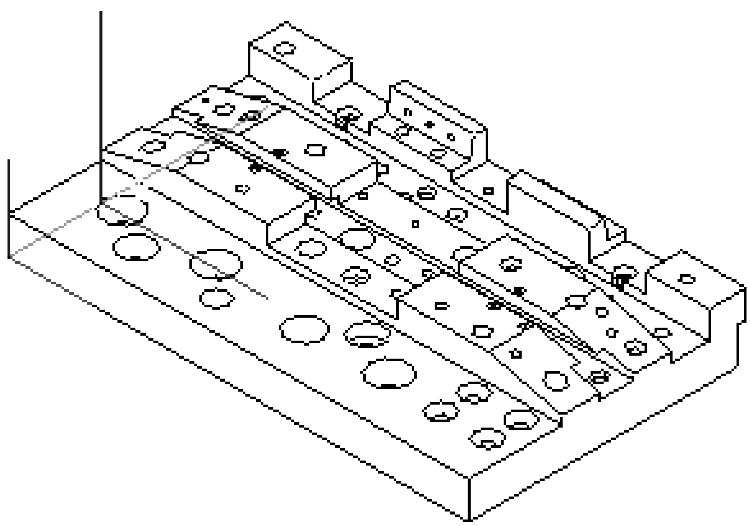
11

Activity 11–3: Optimizing a Spot Drill Subprogram

In this activity, you will examine an unoptimized spot drill subprogram and then observe how optimization improves efficiency.

Step 1 Open an existing part, save with a new name and enter the Manufacturing Application.

- ❑ Open the part **ama_optimization.prt.**



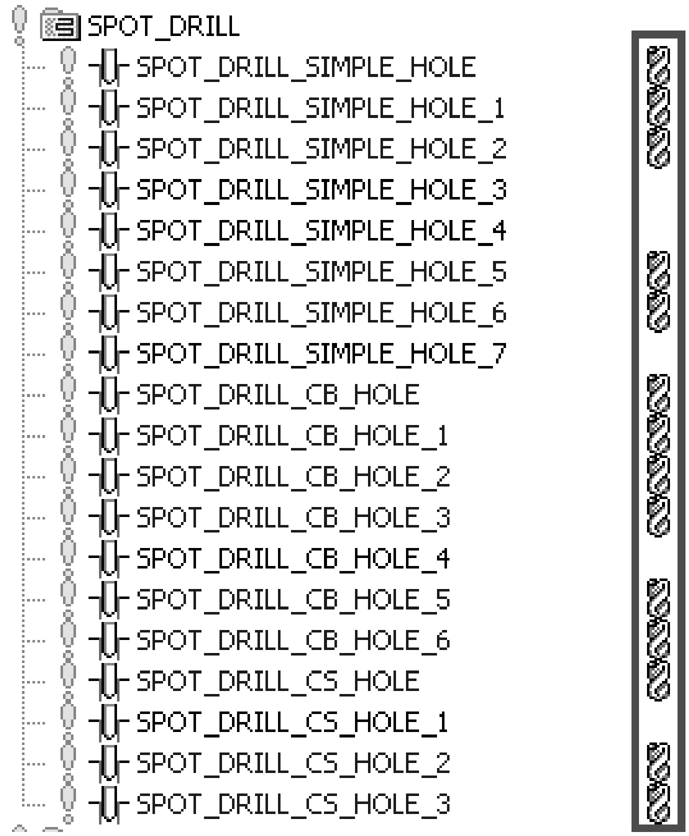
- ❑ Use the **Save→As** option under File on the menu bar and rename the part to *****_optimization.prt** where ******* represents your initials.

- ❑ Choose **Application→Manufacturing.**

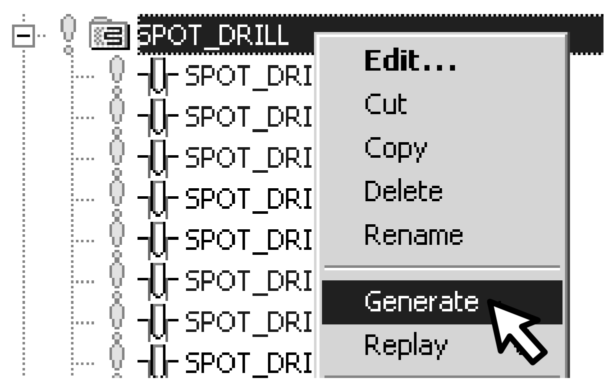
Step 2 Generate the Unoptimized Tool Paths.

You will generate the tool paths for the existing SPOT_DRILL suboperation to illustrate the inefficiency of the tool movements.

In the Program Order View, notice the excessive number of tool changes.



❑ Choose the SPOT DRILL suboperation icon and **MB3→Generate**.

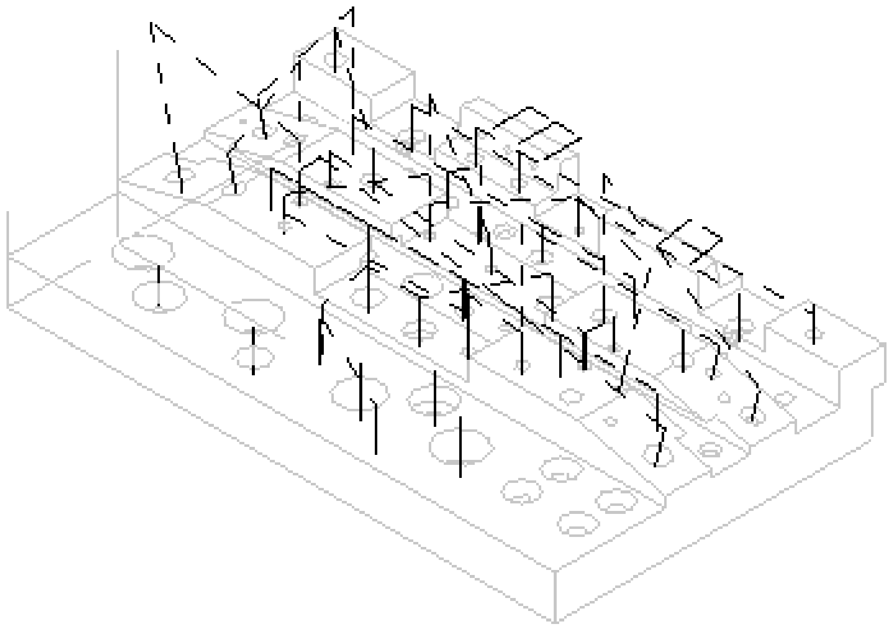


❑ Turn the **Pause After Each Path** and **Refresh Before Each Path** options **off**.

❑ Choose **OK** to generate and display all of the tool paths for the DRILL suboperation.

❑ Choose **OK** to accept the tool paths.

Notice the excessive and somewhat disorganized non-cutting tool movements.

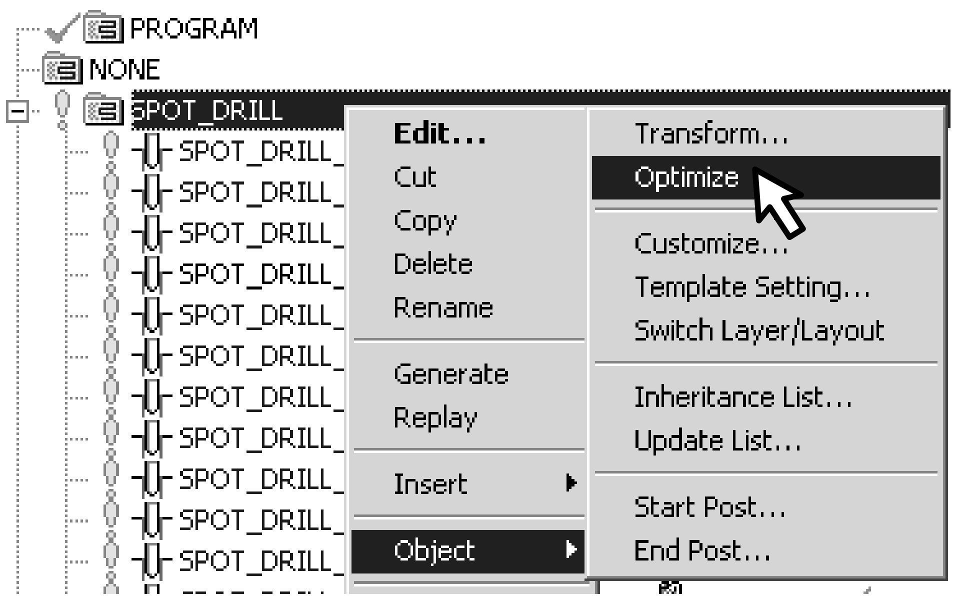


It is much more efficient to use one tool where possible regardless of feature type and to minimize tool changes and traversals. You will see how optimization can accomplish this.

Step 3 Optimize the Program.

Optimization should be done in the Program Order view. This will allow you to observe the reordering of operations and the creation of Optimization groups.

❑ Choose the SPOT_DRILL subprogram icon in the Operation Navigator and choose **MB3→Object→Optimize**.

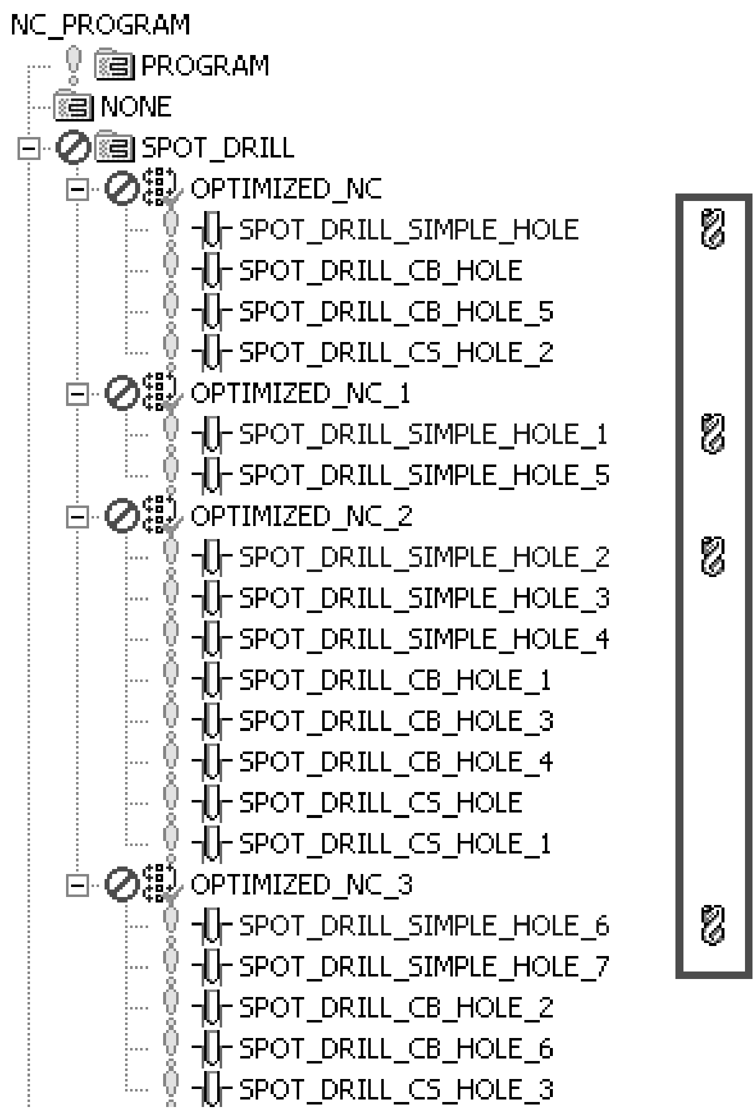


❑ Be sure all three options are turned **on** and then choose **OK** to begin the optimization.

The system will take a few moments to process.

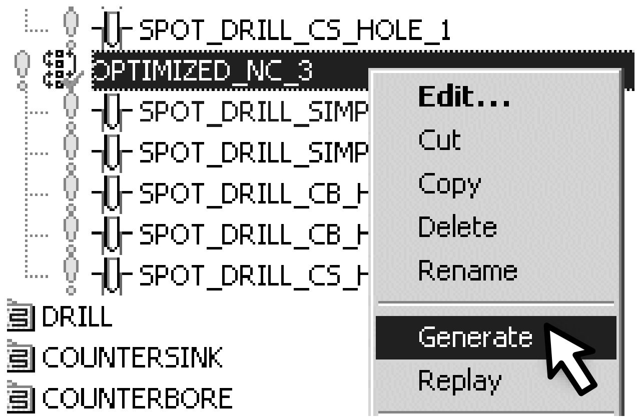
❑ When processing is complete, expand the Optimization groups.

The number of tool changes has been reduced to four.



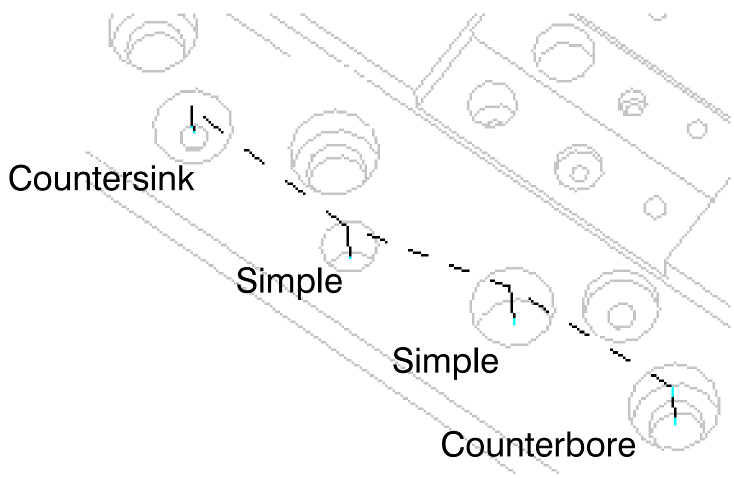
Step 4 Generate the Optimized Tool Paths.

❑ Choose the OPTIMIZE_NC_3 icon and then choose **MB3→Generate** to generate the tool path for this optimization group.



❑ Choose **OK** to complete the tool path generation.

The system no longer machines manufacturing features in order according to feature type. Now, as illustrated below, all manufacturing features that can be cut by the same tool regardless of feature type are grouped and an optimal tool path is generated to minimize tool travel distance.



❑ Choose the SPOT_DRILL icon and then choose **MB3→Generate** to generate tool paths for all of the optimization subprograms.

❑ **Save** the part file.

You have completed this activity and the lesson.

SUMMARY

Hole Making is an advanced application that automates the creation of operations such as spot drilling, drilling, countersinking, counterboring, reaming, tapping, and deburring through the use of intelligent models containing manufacturing features (User Defined Features, User Defined Attributes, and NX Based Features) and embedded machining rules.

In this lesson you:

- Created a hole making program that machines simple, countersunk, and counterbore holes
- Identified where tools must be edited or created and applied the necessary changes
- Tagged simple geometry so it would be recognized as machinable features
- Optimized a program

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

IN

I

K

L

M

N

O

P

R

S

X

Y

Z

(This Page Intentionally Left Blank)

IN

Reference Chart Tear Outs

These tear out reference charts are provided for your convenience.

(This Page Intentionally Left Blank)

Student Profile
Advanced Mill Applications

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

Advanced Mill Applications
Class Agenda

Day 1 ________Morning

- Introduction and Overview
- Lesson 1 Wave Geometry Linker in Manufacturing
- *Workbook Section 1 Project Description*

Afternoon

- *Workbook Section 2 Process Planning*
- *Workbook Section 3 Manufacturing Operation Preparation*
- Lesson 2 Cavity Milling
- *Workbook Section 4 Cavity Mill*

Day 2 ________Morning

- Lesson 3 Fixed Contour
- *Workbook Section 5 Fixed Contour*

Afternoon

- Lesson 3 Fixed Contour
- *Workbook Section 6 Flowcut*
- *Workbook Section 7 Fixed Contour Finishing*
- Lesson 4 Wire EDM

Day 3 ________Morning

- Lesson 5 In–Process Workpiece
- Lesson 6 Machining Faceted Geometry
- Lesson 7 High Speed Machining
- Lesson 8 NC Assistant

Afternoon

- Lesson 9 Templates
- Lesson 10 Libraries
- Lesson 11 Hole Making

(This Page Intentionally Left Blank)

Advanced Mill Applications
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