UNIGRAPHICS

ADVANCED MILL APPLICATIONS

STUDENT GUIDE

January 2004

MT11045 — Unigraphics NX 2

EDS Inc.

Proprietary & Restricted Rights Notices

Advanced Mill Applications Student Guide Publication History:

Version 16.0 .... October 2000
Version 17.0 .... January 2001
Version 18.0 .... August 2001
Unigraphics NX .... September 2002
Unigraphics NX 2 .... January 2004

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 and use of 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 and use various cut patterns and operation types to perform high speed machining.

Student Responsibilities

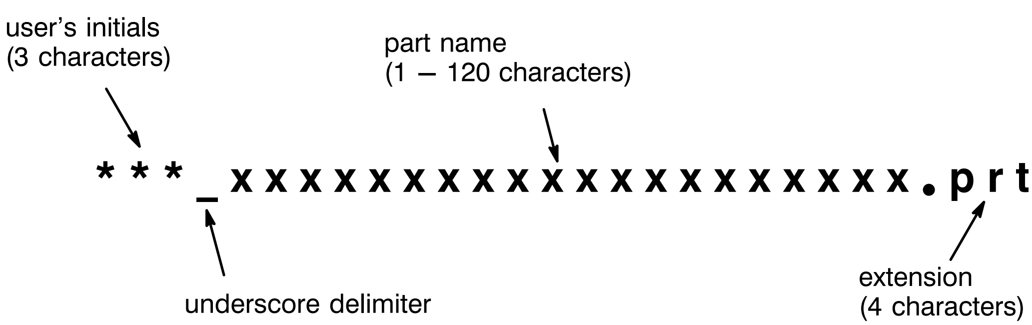
- Be on time
- Participate in class
- Stick with the subject matter
- Listen attentively and take notes
- Practice on the job what you have learned
- Have Fun

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, using the WAVE (What If Alternative Value Engineering) Geometry Linker, that is associated to the designer's original geometry.

OBJECTIVES

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

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

This lesson contains the following activities:

1

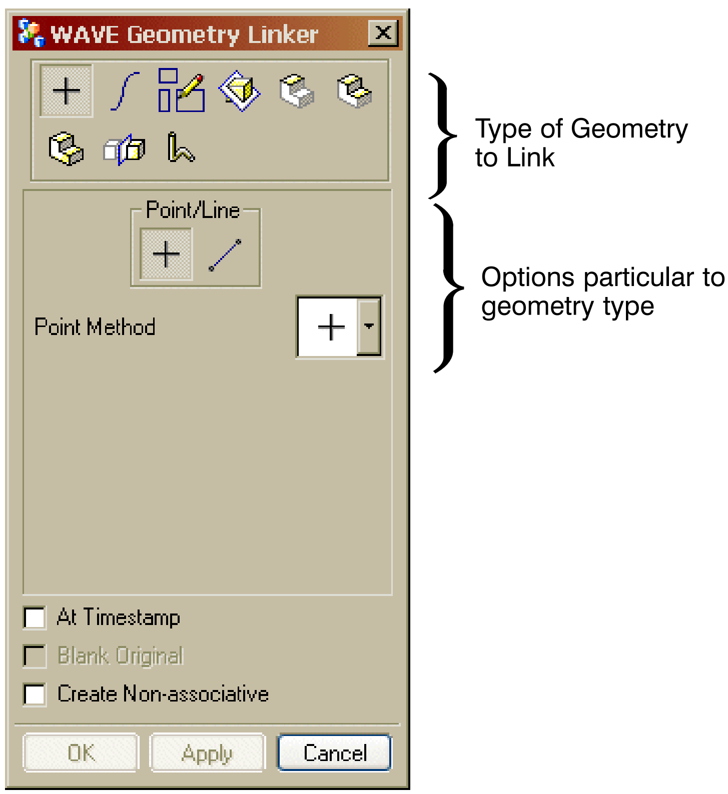
The WAVE Geometry Linker

The WAVE Geometry Linker is used to associatively copy geometry from a 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 for linking, including points, curves, sketches, datums, faces, and bodies. The linked geometry can be used for creating and positioning new features in the work part.

The Wave Geometry linker is accessed by choosing **Assemblies** → **WAVE Geometry Linker** from the menu bar.



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

- **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.
- **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.

Geometry Types used by the Geometry Linker

Several different types of geometry can be used in the WAVE application.

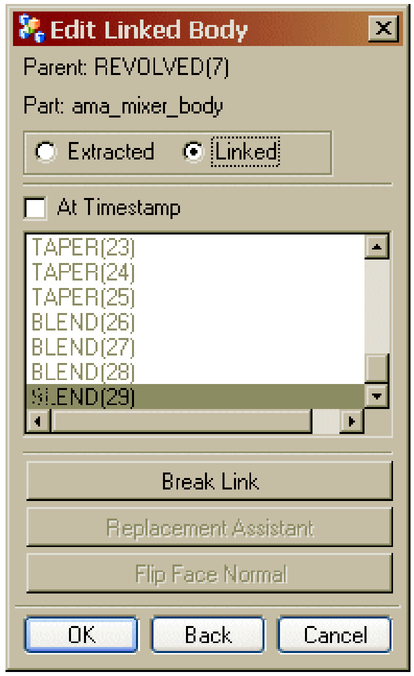
- Points
- Curves/Strings
- Sketches
- Datums
- Faces
- Regions of Faces
- Bodies and Mirrored Bodies

When selecting 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. Conversely, if you copy an entire sketch, curves may be removed or added and the link will update.

Editing Links

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



When this dialog is displayed, the cursor is active in the graphic window allowing new parent geometry selection 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** allows you to specify the timestamp at which the linked feature is placed. If 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 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.

- **Replacement Assistant** allows replacement of one linked object with another (cannot be used on linked sketches or strings).
- **Flip Face Normal** reverses the normal of the face selected.
- 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 to distinguish between the existing linked geometry and the new parent geometry.

Broken Links

A link may become broken for several of the following reasons:

- The parent geometry is deleted.
- The path from the linked geometry to the 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).

1

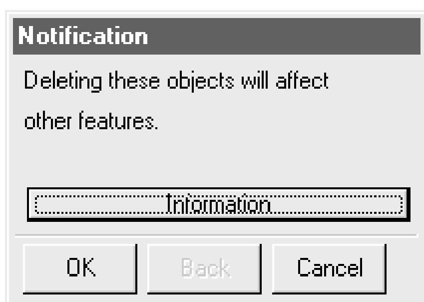
Newly Broken Links

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.

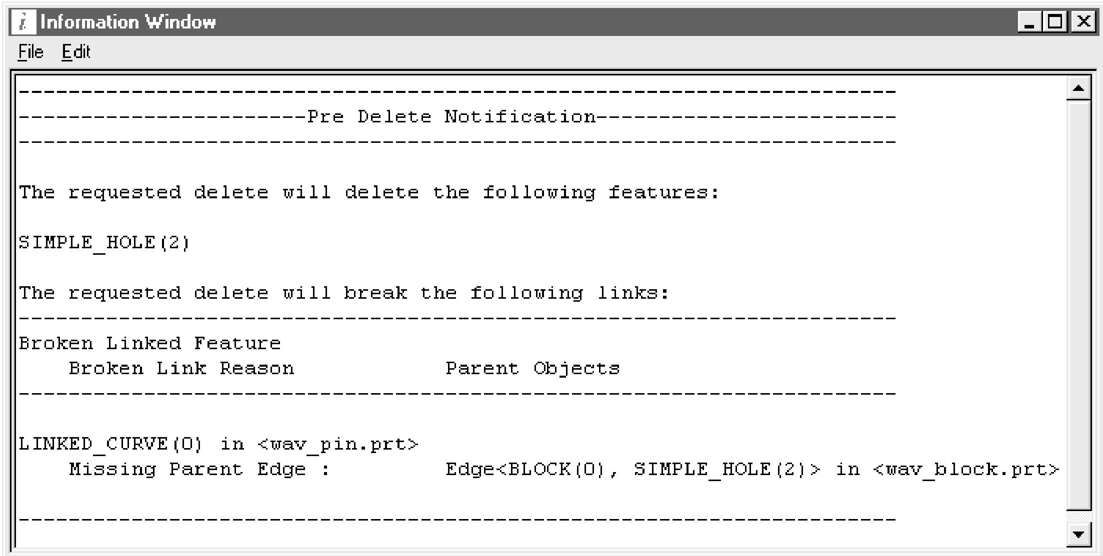
After a link is accepted, its status is changed to **broken** until a new parent is defined.

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.




```
----------------------------------------------------------------------
-----------------------Pre Delete Notification------------------------
----------------------------------------------------------------------

The requested delete will delete the following features:

SIMPLE_HOLE(2)

The requested delete will break the following links:
----------------------------------------------------------------------
Broken Linked Feature
    Broken Link Reason            Parent Objects
----------------------------------------------------------------------

LINKED_CURVE(0) in <wav_pin.prt>
    Missing Parent Edge :         Edge<BLOCK(0), SIMPLE_HOLE(2)> in <wav_block.prt>

----------------------------------------------------------------------
```

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.** If you choose this method, you will not have an opportunity to verify child features before they are removed.

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.

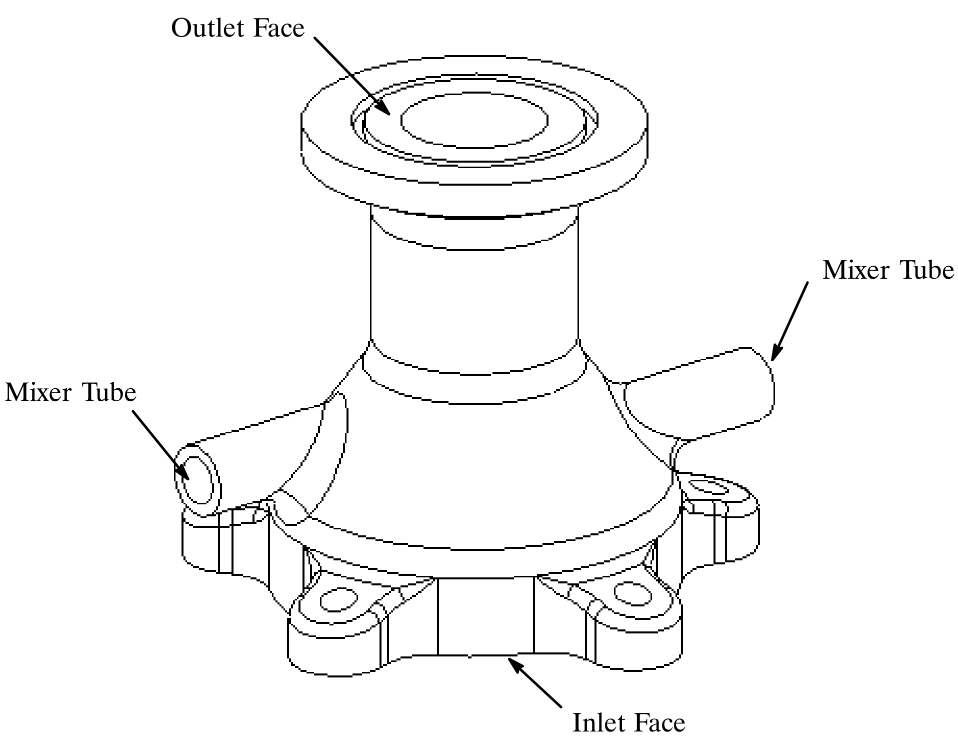
Activity 1—1: Creating an Assembly for WAVE

In this activity, you will create an assembly structure for later use with the WAVE Geometry Linker. Remember that WAVE only works in the context of an assembly.

This activity uses a hypothetical company that has been awarded a contract to machine a mixer housing.

The customer has supplied a Unigraphics solid model of the designed part. Since high-production quantities are needed, the customer has decided to make the part as an aluminum casting. This will reduce significantly, the amount of time spent machining. Unfortunately, the customer has not supplied a solid model of the casting which we will need to create. Using WAVE, you will create a simulated casting model that is associated with the original geometry.

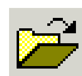
For the casting body, it will be necessary to remove the seven drilled holes, and add .250” machining stock on the inlet, outlet and mixer tube faces. Also note that the ring groove will not exist on the casting body.



All machined faces have 1/4” of added stock. Once the modeling changes are made, you will 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 the seed part, ama_seedpart_in.prt, and save it with a new name.

❑ If necessary, start Unigraphics NX.

❑ Use **File → Open.** 

❑ Navigate to your parts folder and open the file.

> **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.

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

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

Your 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.

❑ If necessary, from the main menu, choose **Applications → Assemblies**.

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

❑ Choose **OK** from **Layer Settings** dialog.

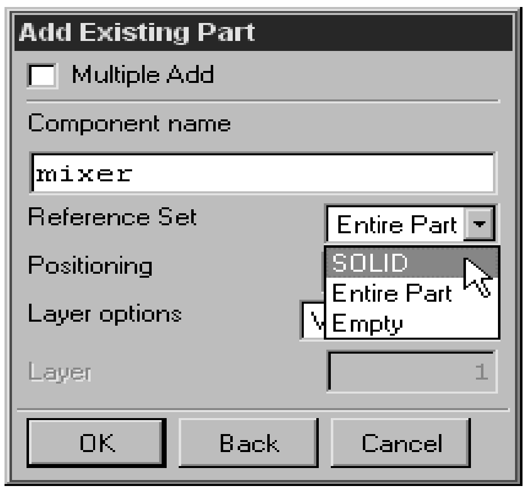
❑ From the main menu, choose **Assemblies → Components → Add Existing.**

❑ In the **Select Part** dialog, select the **Choose Part File** button.

❑ Select **ama_mixer_body**, then choose **OK**.

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

❑ If necessary, while still in the **Add Existing Part** dialog, choose **SOLID** from the **Reference Set** pull down menu.



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.

The **Point Constructor** dialog is displayed.

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

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

❑ **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**, while **ama_mixer_body** is the single component. Currently, only the component contains any geometry.

The next step will be 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** from the menu bar.

❑ Choose **OK** in the **Class Selection** dialog.

❑ In the **File Name** field, type in *****_mixer_casting**, 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 selecting **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, highlight the CASTING component, *****_mixer_casting**, and using **MB3** choose **Make Displayed Part** from the pop-up menu.

- ❑ 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...**

- ❑ If necessary, in the **Import Part** dialog, uncheck **Create Named Group**, then choose **OK**.

- ❑ Browse to the **ama_seedpart_in.prt**, and double-click on it.

  The **Point Constructor** dialog is displayed.

- ❑ Choose **OK** in the **Point Constructor** dialog. Since no geometry is being imported, position is not relevant.

  Also, there is no interaction on the screen.

- ❑ Choose **Cancel** in the **Point Constructor** dialog.

- ❑ Choose **Format → Layer Settings**.

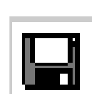
  Notice the several different layer categories defined.

- ❑ Choose **Cancel** in the **Layer Settings** dialog.

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

- ❑ In the Assembly Navigator, highlight *****_mixer_casting**, and using **MB3**, choose **Display Parent → ***_mixer_mfg.**

- ❑ In the Assembly Navigator, highlight *****_mixer_mfg**, and using **MB3**, choose **Make Work Part**.

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

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

This concludes this activity.

1

Linking Procedure

You use the **Assemblies → WAVE Geometry Linker** dialog to create associated objects between part files. The linker allows you 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 additional 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.

Step 1 Prepare the assembly.

- ❑ If necessary, open the *****_mixer_mfg** assembly part and then the Assembly Navigator.
- ❑ Using **MB3** over the *****_mixer_casting** line in the **Assembly Navigator** 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 the 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 the linked copy. It can be difficult to determine one from the other, it will be 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 *****_mixer_casting** component and choose **Make Displayed Part**.
- ❑ In the graphics area, use **MB3** and select **Replace View → TFR-TRI** from the pull down menu.
- ❑ Choose the **Shaded** icon from the main menu bar.
- ❑ Choose **Edit → Object Display.**
- ❑ Select the linked body and choose **OK**.

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

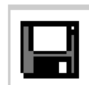
- ❑ Change the **Color** to **Yellow**.
- ❑ Choose **OK** in the **Edit Object Display** dialog.

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.

- ❑ In the Assembly Navigator, using **MB3** on the *****_mixer_casting** component, choose **Display Parent → ***_mixer_mfg.**

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

- ❑ Choose the **Save** icon  on the 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, but somewhat different.

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

Uses of Simplify:

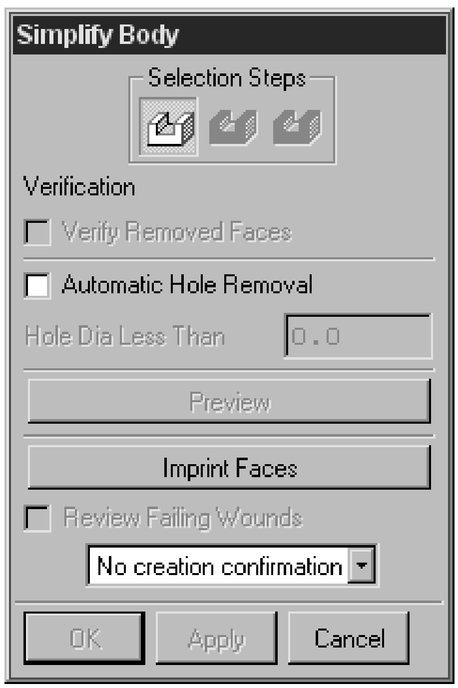
- Remove “machined” features for preparing an as cast part from a body that is not appropriately constructed for link **At Timestamp**, or from a body whose features are not accessible.
- Remove details such as holes and blends for finite element analysis.
- In casting tooling work, core and pattern 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

You will use the **Simplify Body** function to remove holes from your 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.
- Deselect 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.

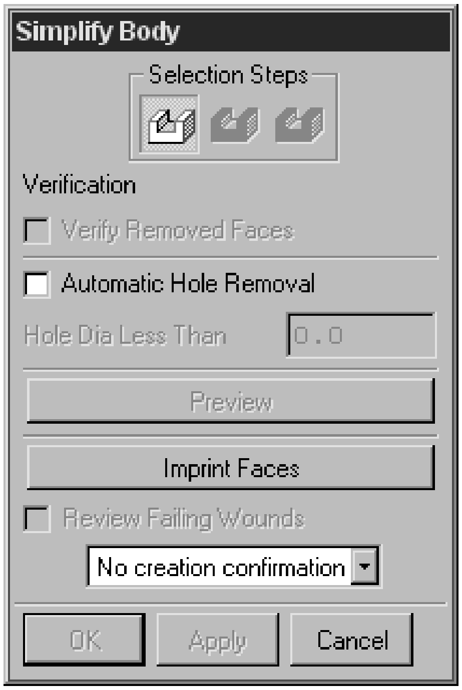
Step 1 Make the CASTING component the work and displayed part.

- ❑ If necessary, open your *****_mixer_mfg** assembly part and then open the Assembly Navigator.
- ❑ In the Assembly Navigator, use **MB3** on the *****_mixer_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.

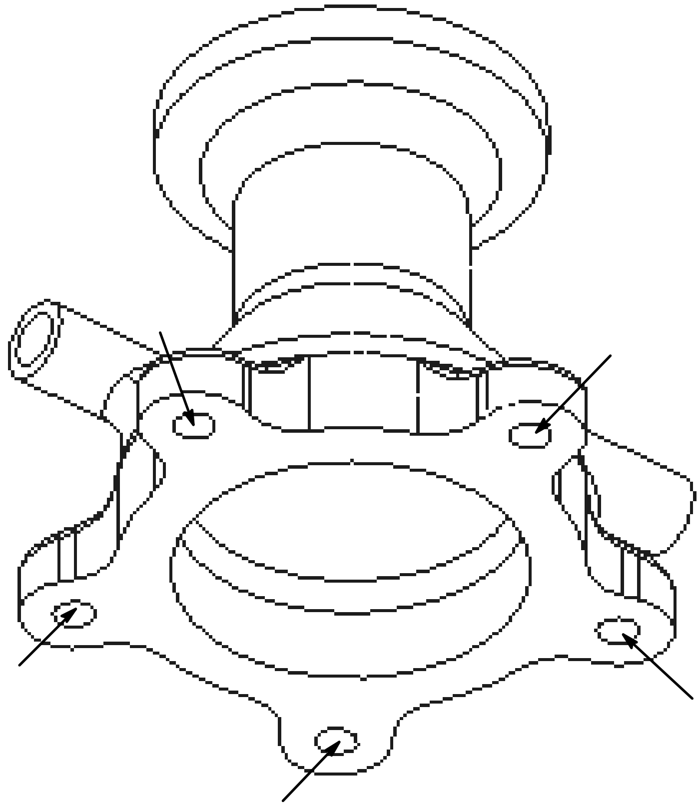


The cue line reads **Select retained faces**.

- ❑ 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 boundary faces**.

- ❑ 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 they will be retained faces, and adds them to the **retained face** selection. In this case, they will not be retained, so you will deselect them from the **retained faces** set.

- ❑ In the **Simplify Body** dialog, choose **Retained Faces** again.
- ❑ Hold down the “shift” key, and deselect the five holes that were previously chosen.
- ❑ Choose **Preview**.
- ❑ In the **Simplify Body Preview** dialog, choose **Preview Removed**.

  Only the hole faces highlight.

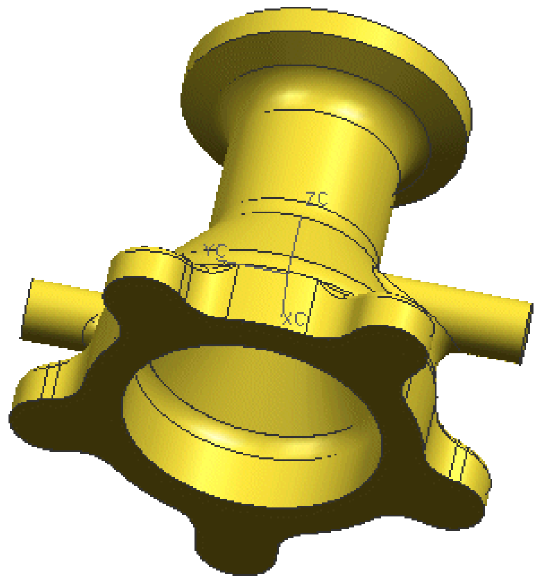
- ❑ Choose **Preview Retained.**

  Now all faces except the five holes highlight.

- ❑ Choose **OK** in the **Simplify Body Preview** dialog.

❑ Choose **Apply** in the **Simplify Body** dialog.

The Simplify Body information window gives the number of faces removed and retained (in this case 5 faces are removed, 110 faces remain).



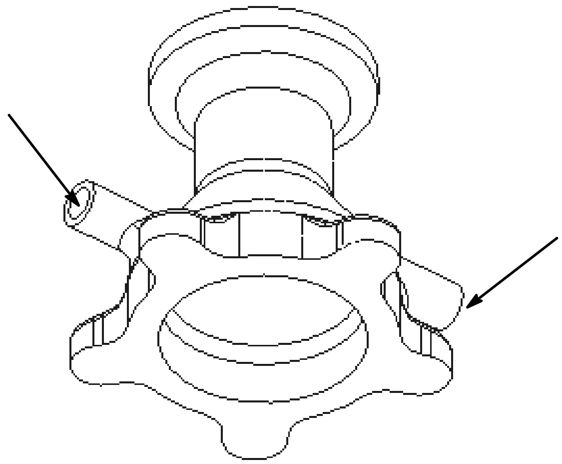
❑ Dismiss the Simplify Body information dialog by choosing **OK**.

Step 3 Simplify away the holes in the two bosses.

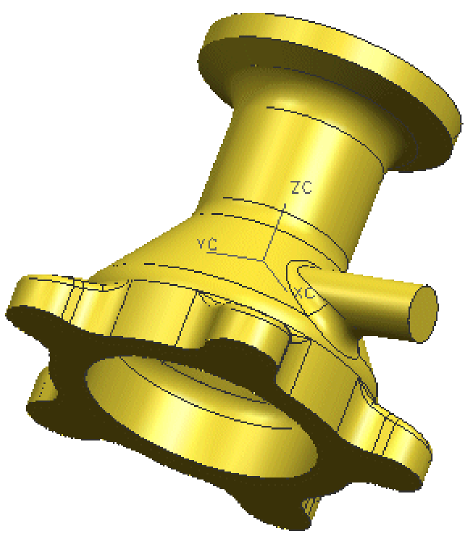
The retained face that was selected earlier is still active, so it is not necessary to choose another.

❑ In the **Simplify Body** dialog, choose **Boundary Faces**.

❑ Select the two cylindrical hole faces of the mixer tubes.



- ❑ 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 explored is **Extrude**.

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

- ❑ If necessary, open your *****_mixer_mfg** assembly part and then open the Assembly Navigator.

- ❑ If necessary, in the Assembly Navigator, using **MB3** on the *****_mixer_casting** component, choose **Make Displayed Part**.

Step 2 Use Extrude to fill in the ring groove.

- ❑ Choose **Application → Modeling**.

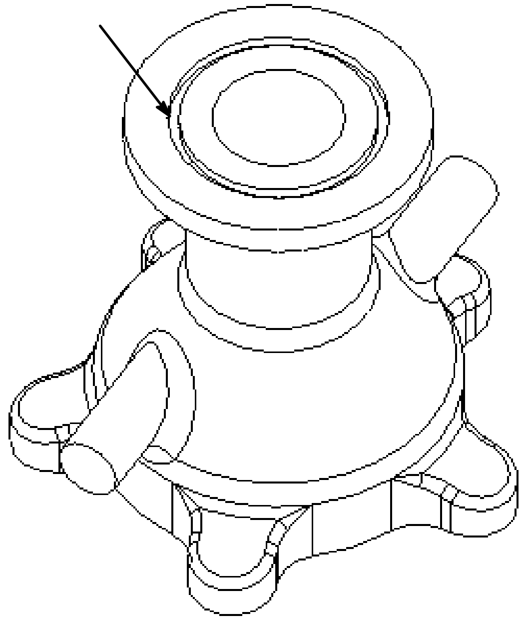
- ❑ Choose **Insert → Form Feature → Extrude.**

  The **Extruded Body** dialog is displayed.

- ❑ Choose the **Solid Face** button.

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

1



❑ Choose **OK** until the cue line reads **Choose extrusion method**.

❑ Choose **Trim to Face/Plane** from the Extruded Body dialog.

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

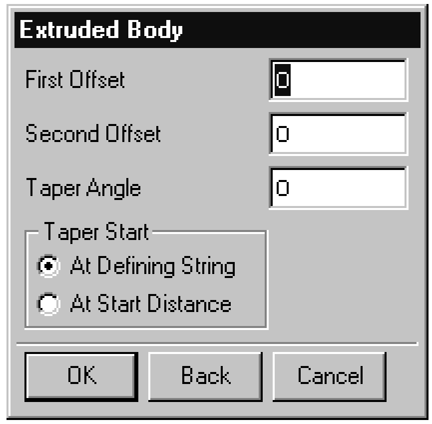
❑ If the extrusion direction arrow points away from the body, choose **OK**.

❑ In the **Trimming Face** dialog, toggle **Extend Trim Face** on.

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

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

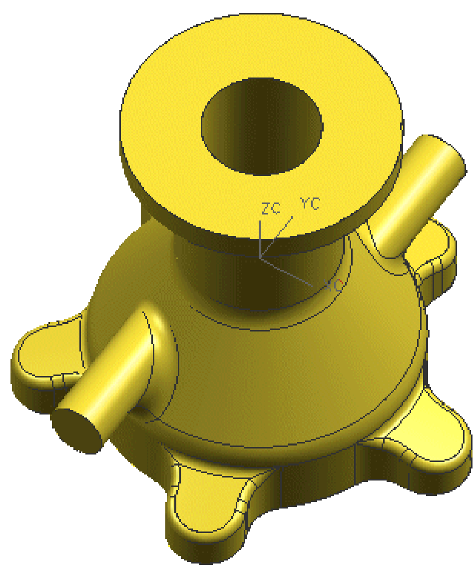
The following dialog allows you to apply offset to the face being extruded. In this example, offsets are not necessary.



❑ Choose **OK** on the **Extruded Body** dialog.

❑ Choose **Unite** on the **Boolean Operation** dialog.

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



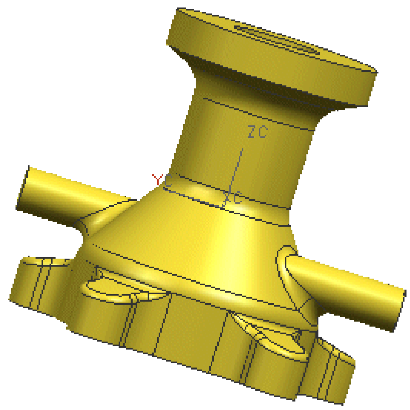
❑ **Cancel** the **Extruded Body** dialog.

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.
- ❑ Select the inlet and outlet faces, and the two mixer tube faces.
- ❑ Choose **OK**.



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.

- ❑ If necessary, 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 **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.

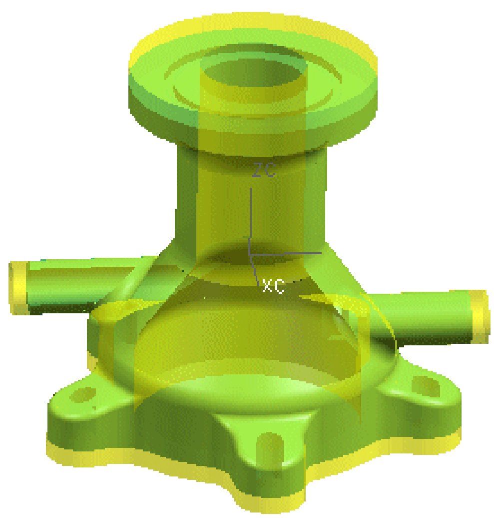
To fully realize the extent of the changes made, you will display both the original and the linked body together.

❑ 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 original body changes, the casting body will update also.

At this stage, 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)

In-Process Workpiece

Lesson 2

2

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 2—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_mfg_asmb.prt.**

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

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

❑ If necessary, display the Geometry view of the Operation Navigator.

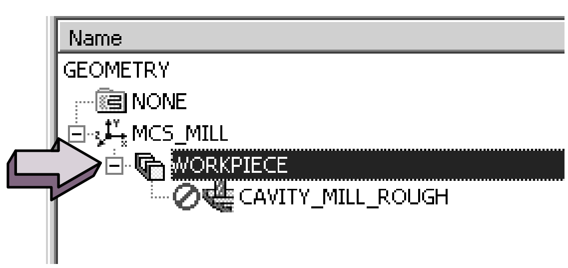
An In-Process Workpiece (IPW) column can be added to the Operation Navigator by clicking **MB3** on the Operation Navigator background, choosing **Columns** → **Configure**, and turning the **IPW** option **on**.

Step 2 Displaying the Part Geometry.

The part geometry has been defined in the **WORKPIECE** parent group.

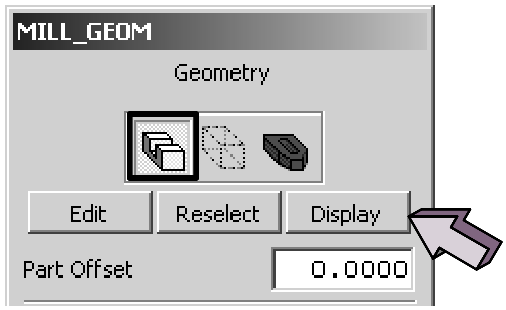
❑ In the **Geometry** view of the Operation Navigator, expand all objects.

❑ Double-click the **WORKPIECE** parent group.



The **MILL_GEOM** dialog is displayed.

❑ With the **Part** icon selected, choose **Display**.

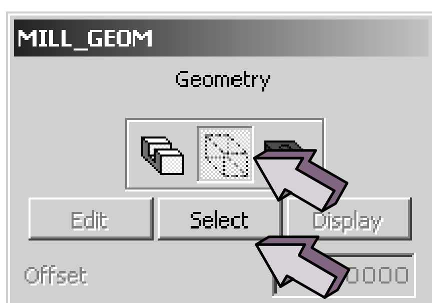


2

Step 3 Defining the Blank Geometry

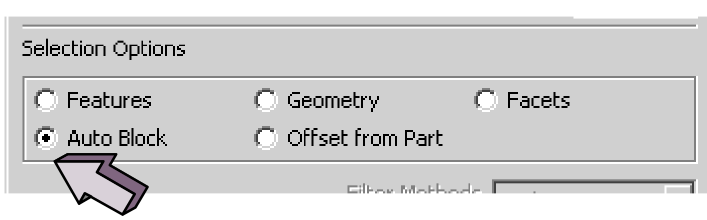
You will define the blank geometry using a method that creates a solid body automatically by enclosing the part geometry.

❑ Choose the **Blank** icon and **Select**.

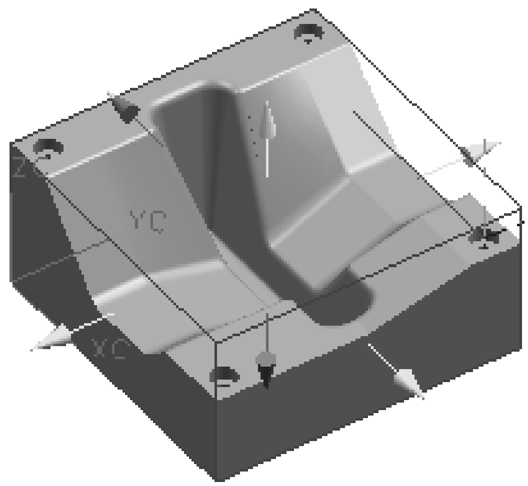


The **Blank Geometry** dialog is displayed.

❑ Turn the **Auto Block** option **on**.



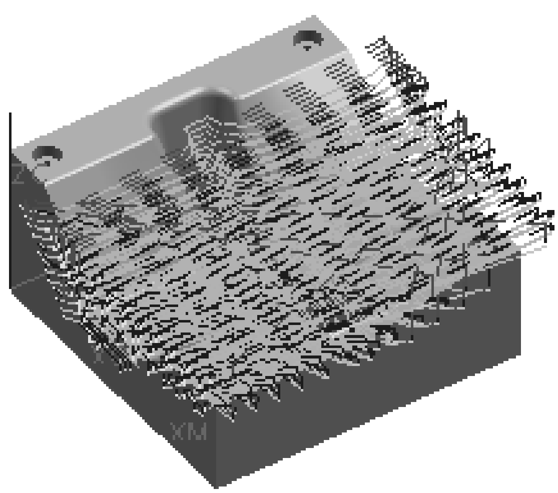
A solid body bounding the part geometry is created. The **XM**, **YM**, **ZM** fields allow you to modify the size of the body by specifying offsets from each face.



- ❑ 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 using **MB3 → Generate,** generate the roughing tool path.

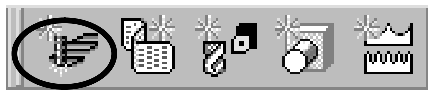


- ❑ Choose **OK** to accept the **Tool Path Generation** dialog.
- ❑ Refresh the graphics display.

Step 5 Create a Semi-Finishing operation.

You will create a semi-finishing operation that uses the IPW defined by the roughing operation.

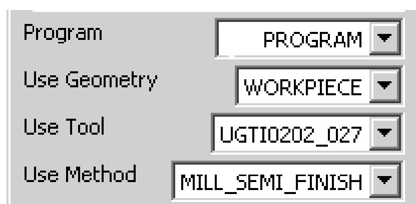
❑ As shown below, select the **Create Operation** icon from the **Create** tool bar.



The Create Operation dialog is displayed.

❑ Choose **CAVITY_MILL** as the **Subtype**.

❑ Specify the following parent groups.



❑ Key in **cm–semi–fin–1.0** in the **Name** field.

❑ Choose **OK** to begin creating the operation.

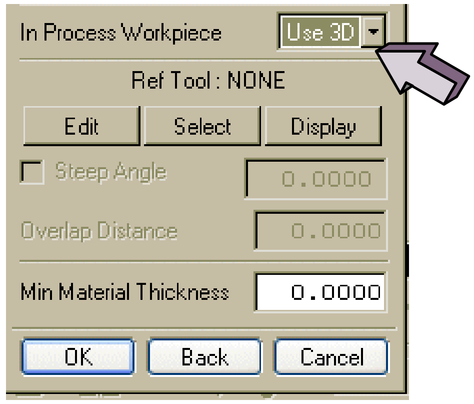
Step 6 Use the IPW as blank geometry.

You will specify use of the IPW in the previous operation to define the blank geometry in this current operation.

❑ Choose the **Cutting** button.

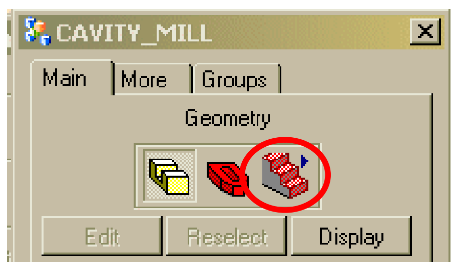
The Cut Parameters dialog is displayed.

❑ Select **Use 3D** from the pull-down menu.

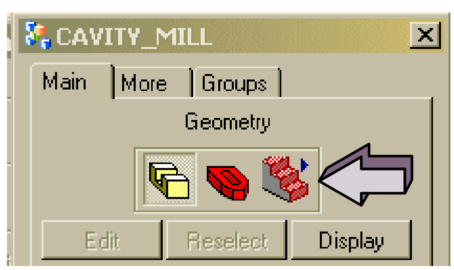


❑ Choose **OK** to accept the Cut Parameters dialog.

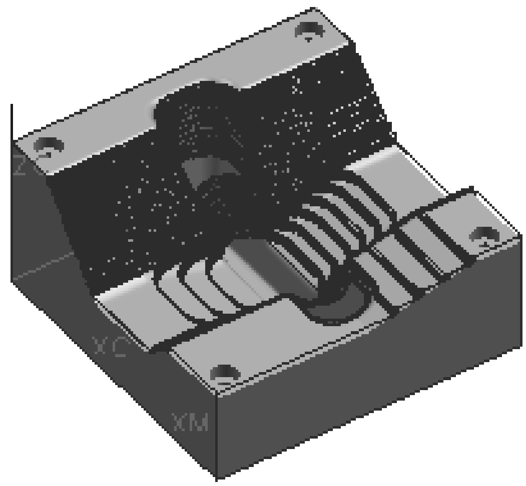
The **Blank** icon at the top of the **CAVITY_MILL** dialog Main property page has been replaced by the **Previous IPW** icon.



❑ Choose the **Previous IPW** icon and then the **Display** button.



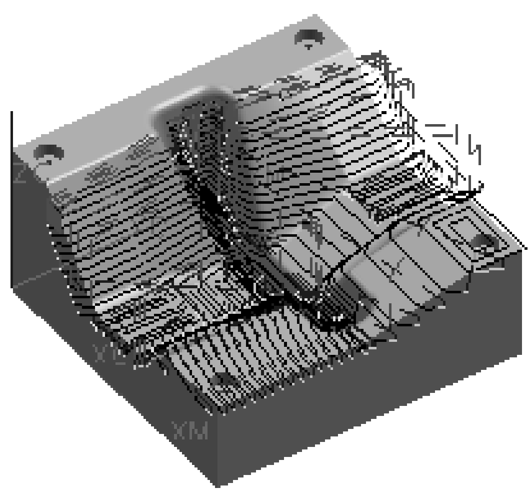
The processor may require some processing time to display the faceted body.



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.

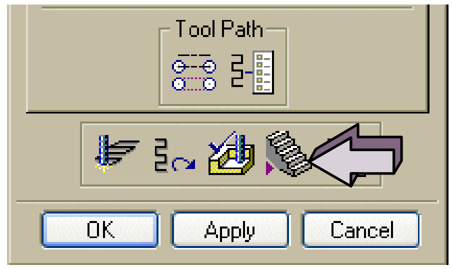


- ❑ Refresh the graphics display.

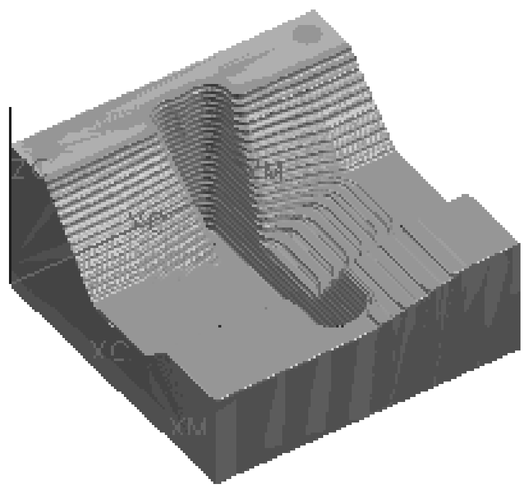
Step 8 Display the resulting IPW.

The IPW created by this operation can now be displayed.

❑ Choose the **Display Resulting IPW** icon.



The processor 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.

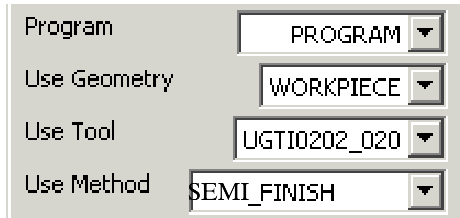
Step 9 Create a second semi-finish operation with a smaller tool.

You will now create a finishing operation that uses the IPW defined by the semi-finishing operation.

❑ Select the **Create Operation** icon from the **Create** tool bar.

❑ Be sure **CAVITY_MILL** is selected as the **Subtype**.

❑ Specify the following parent groups.



❑ Key in **cm–semi–fin–.50** in the **Name** field.

❑ Choose **OK** to begin creating the operation.

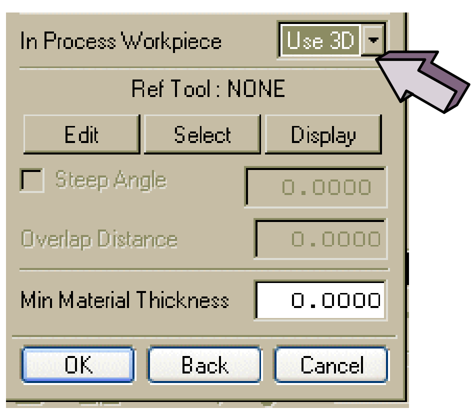
Step 10 Use the IPW as blank geometry.

You will specify that the IPW in the previous operation will define the blank geometry in this operation.

❑ Choose the **Cutting** button.

The **Cut Parameters** dialog is displayed.

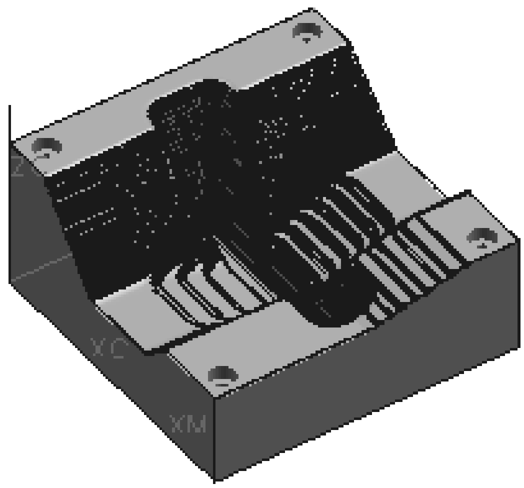
❑ Select **Use 3D** from the pull-down menu.



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

❑ Choose the **Previous IPW** icon and then the **Display** button.

The processor may require some processing time to display the faceted body.



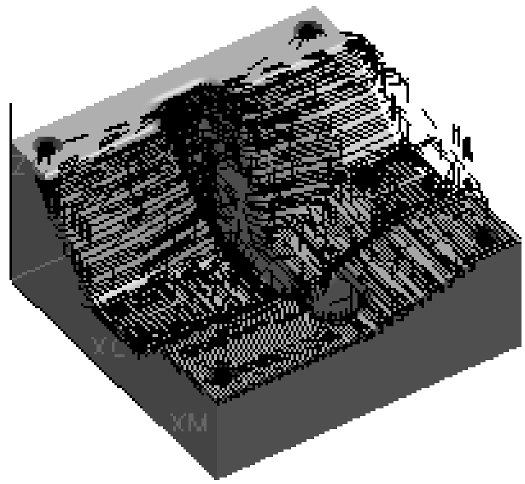
This faceted body is the IPW that this operation uses as blank geometry.

❑ 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.

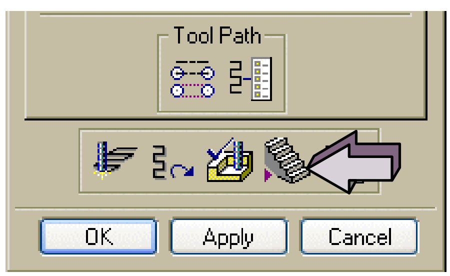


❑ Refresh the graphics display.

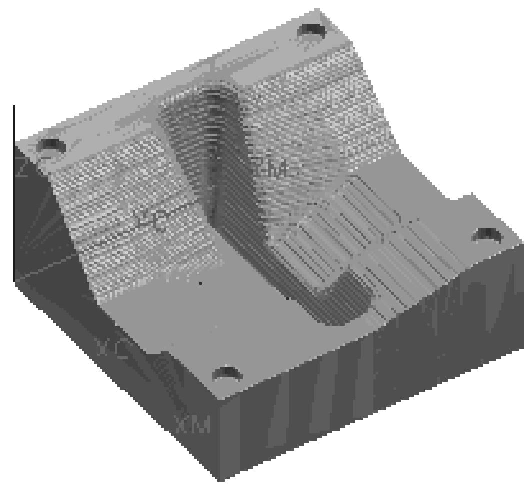
Step 12 Display the resulting IPW.

The IPW created by this operation can now be displayed.

❑ Choose **Display Resulting IPW.**

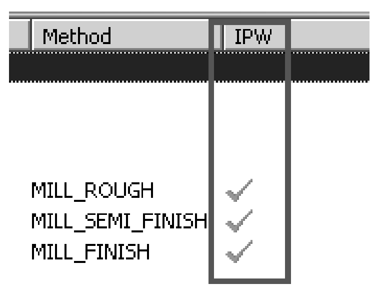


The processor may require some processing time to display the faceted body.



❑ 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.

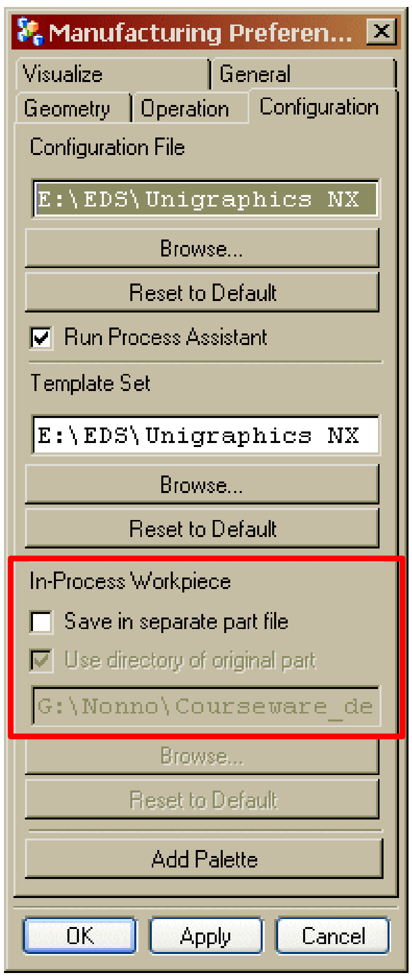
This completes the activity.

IPW and Performance

As you have noticed in the previous activity, generating the IPW takes considerable computer resources. To improve performance, an option to store the facet model representation of the IPW (FIPW) is available.

A new component part will be created that will be a combination of the work part and operation name.

A reference set will be created within this part with the name of the operation with FIPW added to this reference set. If the part already exists, the facets in the reference set will be deleted and a new faceted body will be added.



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.

2

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)

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

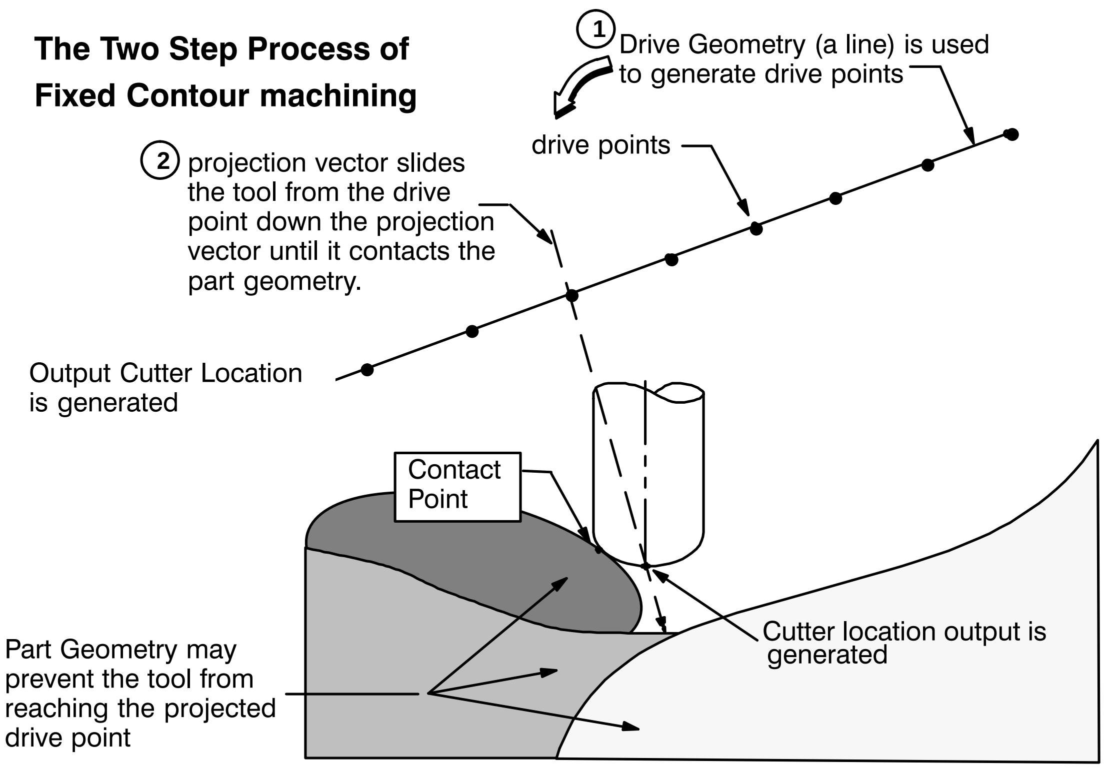
This lesson contains the following activities:

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.

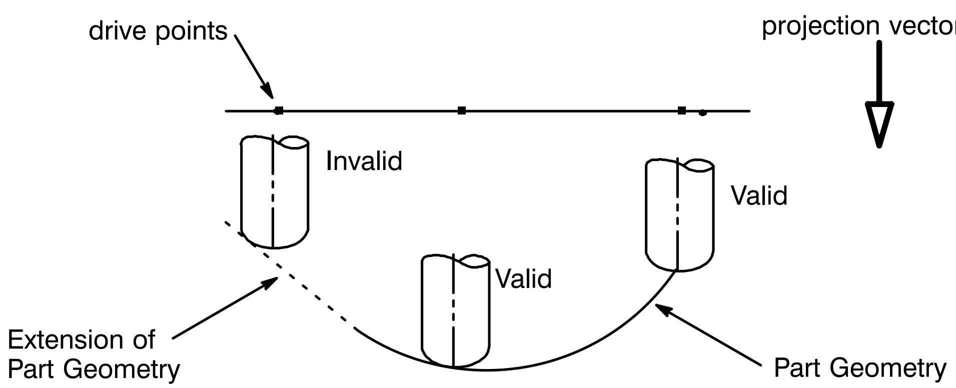


Fixed Contour Tool Path Accuracy

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.

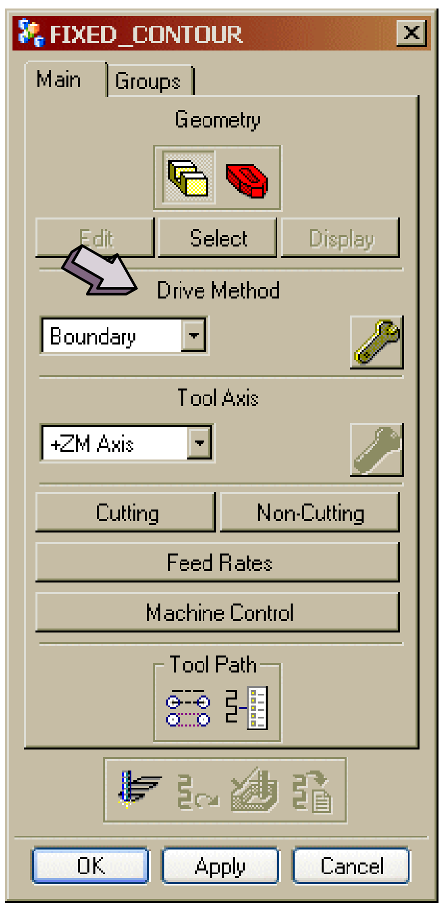
NOTE The projection vector does not need to coincide with the tool axis vector.

For more on the theory and use of the advanced concepts of surface contouring, please refer to Appendix A.

3

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.

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 surface. 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 surface in the direction of a specified projection vector. If part surfaces are not defined, the tool path can be created directly on the drive surface. 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 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 methods 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, check and 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, Wall** and **Trim** geometry.

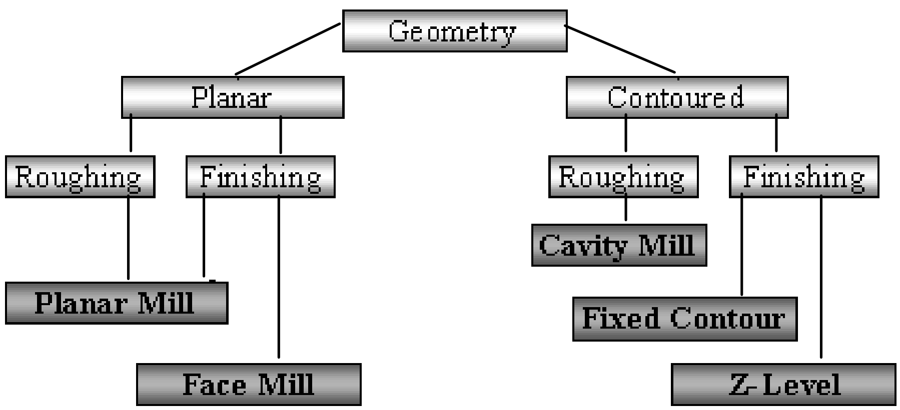
The parent group, **MILL_AREA**, which you used in Cavity Milling operations, is also used in Fixed Contour operations. It allows you to include or exclude areas to be machined in cut areas that you specify. These specific areas may have been previously roughed by **Cavity Milling** or finished by **Planar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

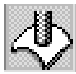
The following diagram can be used as an aid in the determination of the operation type needed for various types of geometry:

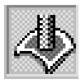
Matching the Geometry to an Operation

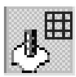


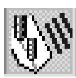
Fixed Contour Operation types

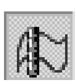
The most commonly used Fixed Contour operation types are:

- **FIXED_CONTOUR –** Generic Fixed Contour operation type. Allows 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.

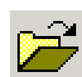
- **PROFILE_3D –** Generates a profile pass utilizing three dimensional curves, edges, faces, existing boundaries or points. Machines at a given Z-depth offset with respect to the geometry type selected. Useful in creation of addendum profile cuts for stamping dies.

Activity 3–1: Contour_Area_Non-Steep Operations

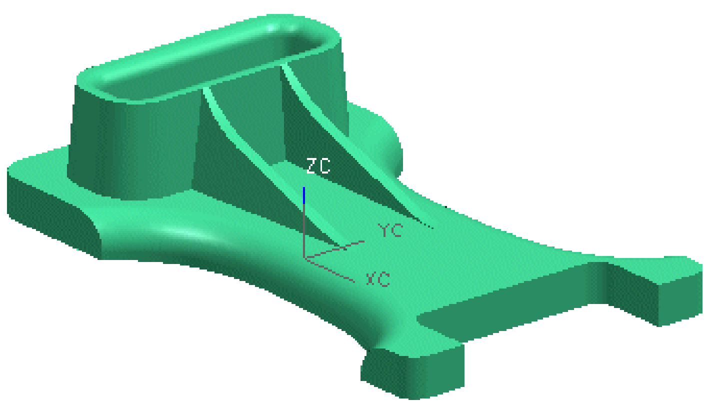
In this activity, you will finish machining the non-steep areas of the part which were previously machined, in a prior activity, with Cavity Milling operations. You will use the geometry parent group, WORKPIECE, and Fixed Contour Non-Steep operation type to remove the material left in areas not machined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

Step 1 Open the part ama_horn_mfg, and save it with a new name.

❑ If necessary, start Unigraphics NX.

❑ Use **File → Open.** 

❑ Navigate to your parts folder and open the file.

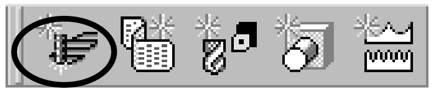


❑ Choose **File → Save As ***_horn_mfg .prt** where ***** represents your initials.

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

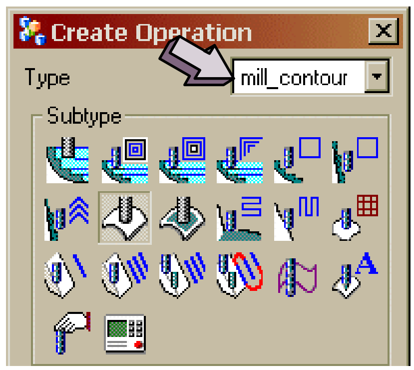
Step 2 Create the Operation.

❑ Choose the **Create Operation** icon from the **Create** tool bar.

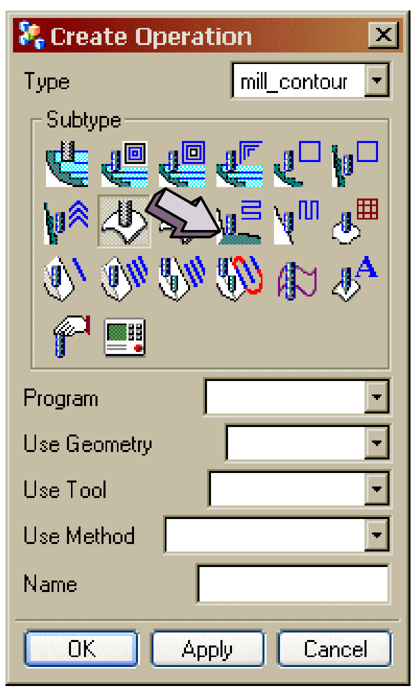


The **Create Operation** dialog is displayed.

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



❑ Select the **CONTOUR_AREA_NON_STEEP** icon.

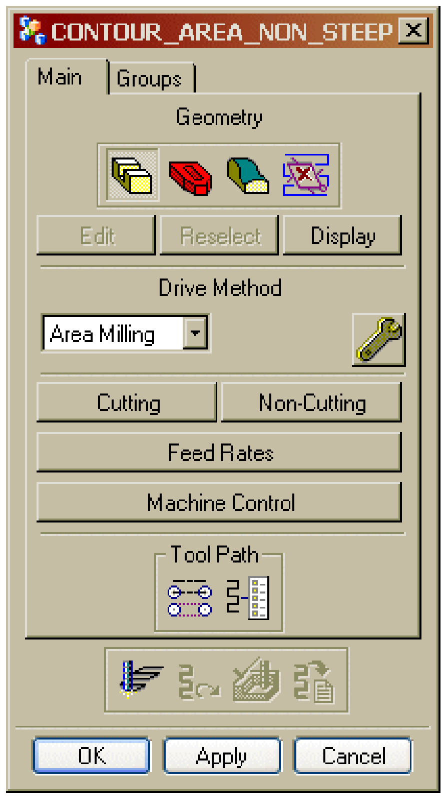


❑ Set the following:

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

❑ Choose **OK.**

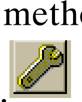
The **CONTOUR_AREA_NON_STEEP** dialog is displayed.



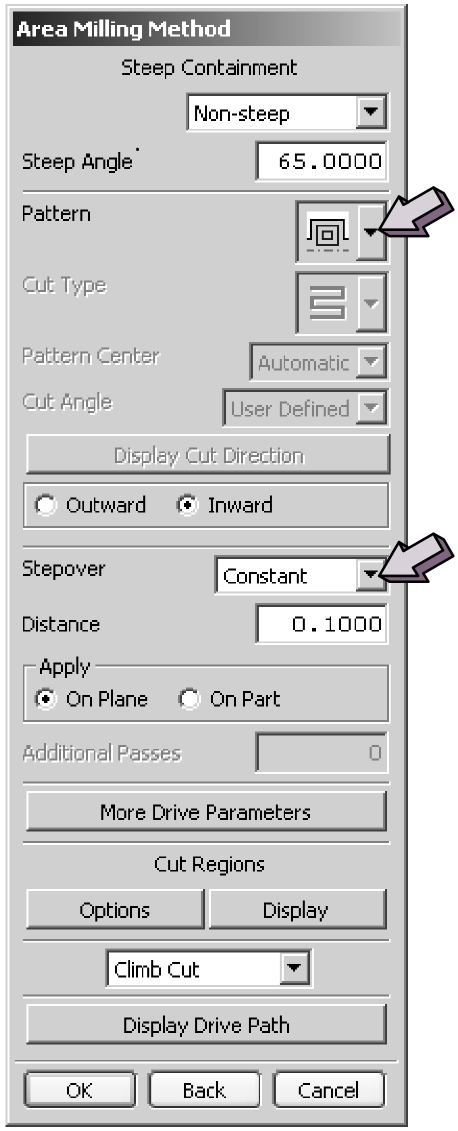
❑ Under the **Geometry** label, choose the **Part** icon  **.**

❑ Choose **Display**.

Note that the entire body is selected as part geometry.

❑ Under the drive method pull down dialog select **Edit Parameters** icon. 

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

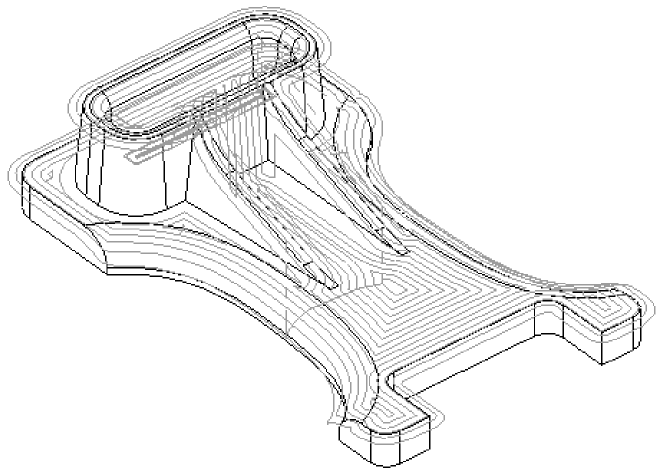


- ❑ Change the **Pattern** to **Follow Periphery** and the **Stepover** to **Constant.**

- ❑ 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 up to 65 degrees were machined.

- ❑ **Save** the part file.

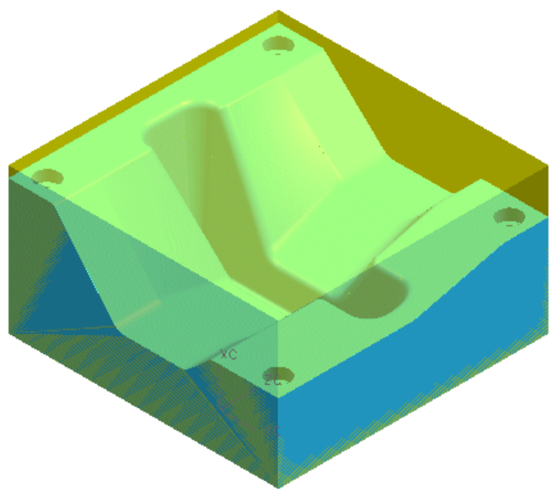
You are finished with this activity. In the next activity, you will use the **Area_Mill** geometry parent group to select a specific area of the body as part geometry.

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**.



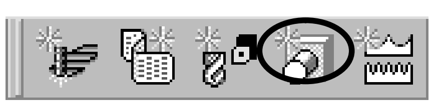
❑ **Save As** ***_ **deep_mold_mfg .prt** where *** represents your initials.

Note that blank geometry surrounds the part.

❑ Enter the **Manufacturing** application.

Step 2 Create the Geometry Parent Group, CUT_AREA_PARENT.

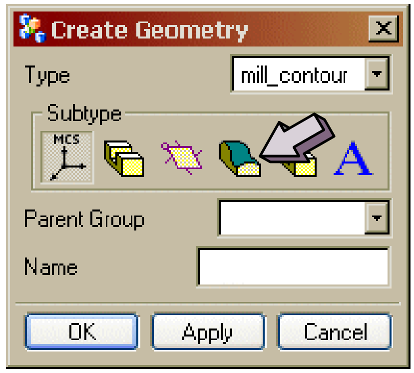
❑ As shown below, select the **Create Geometry** icon from the Create toolbar.



The **Create Geometry** dialog is displayed.

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

❑ Choose the **MILL_AREA** icon.

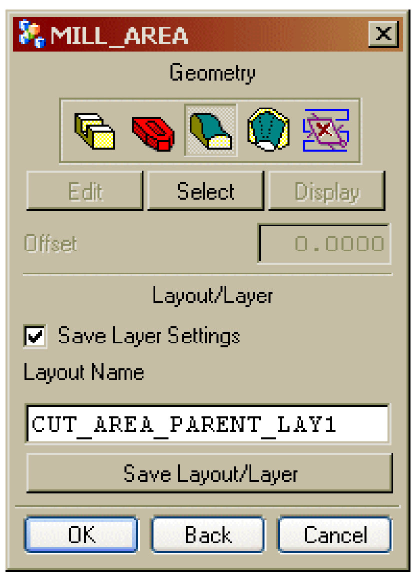


❑ Choose **WORKPIECE** as the **Parent Group**.

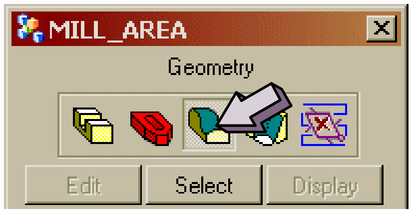
❑ Key in **CUT_AREA_PARENT** as the **Name**.

❑ Choose **OK**.

The **MILL_AREA** dialog is displayed.

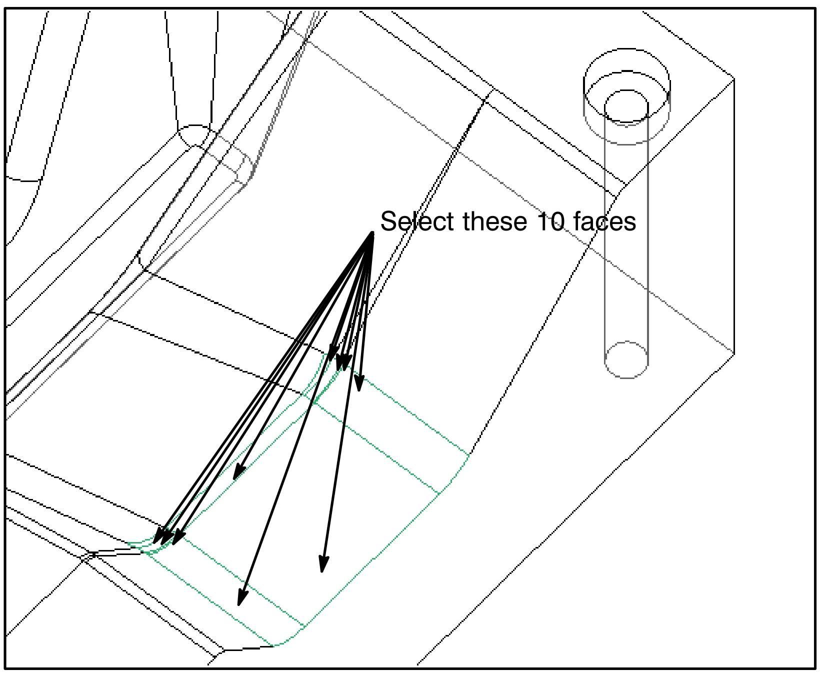


❑ Select the **CUT_AREA** icon.



❑ Choose **Select**.

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



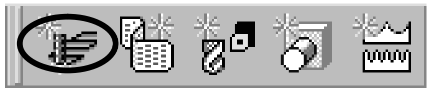
❑ In the **Cut Area** dialog, if necessary, change **Filter Methods** to **Faces**.

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

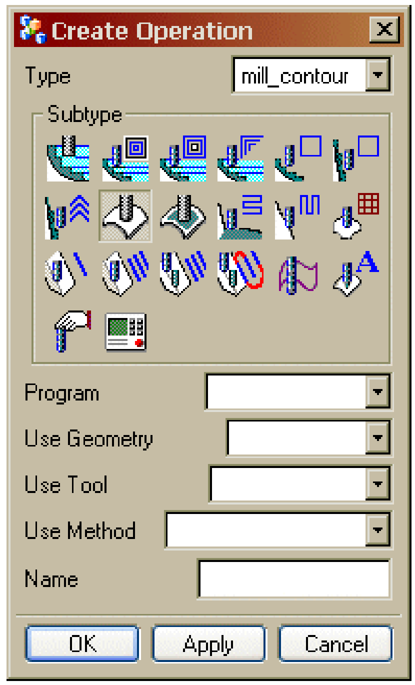
❑ Choose **OK** twice.

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

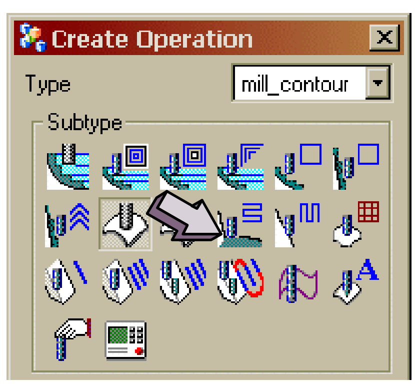
❑ Select the **Create Operation** icon from the **Create** tool bar.



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.

- ❑ Choose the **Edit Parameters** icon. 

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

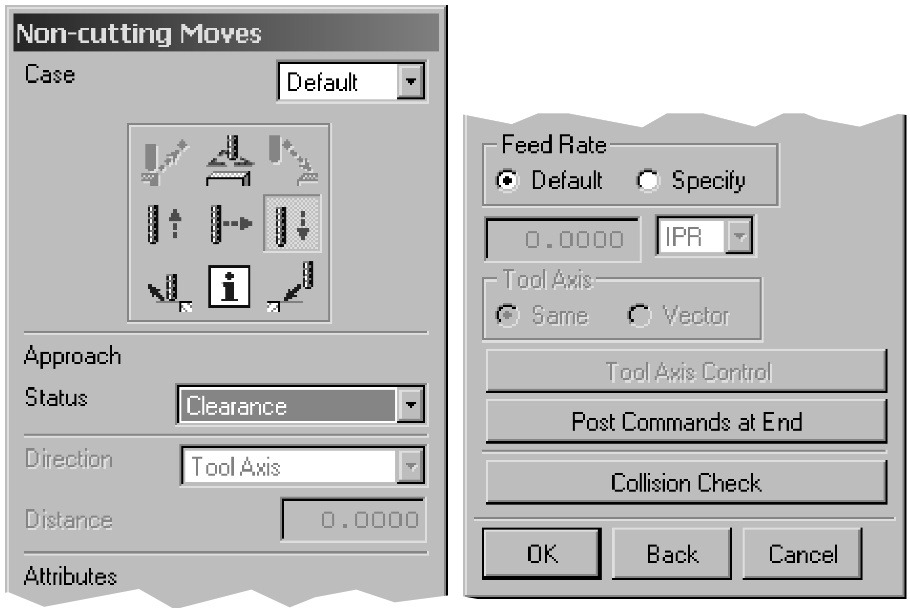
- ❑ Choose **Outward** as the pocket direction.

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

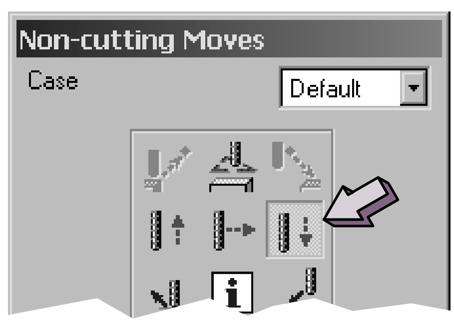
Step 5 Utilizing a clearance plane for clearing the part.

- ❑ Choose **Non-Cutting.**

The **Non-cutting Moves** dialog is displayed.



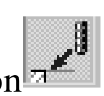
❑ Select the **Approach** icon.



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

You would apply the same procedure for setting the departure motions.

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.**

- ❑ Set the **Max Ramp Angle** to **7.000**.
- ❑ Select the **Distance** button.
- ❑ Key in **.100** as the value.
- ❑ Choose **OK** and return to the **CONTOUR_AREA** dialog.

Step 6 Set the Cutting Options.

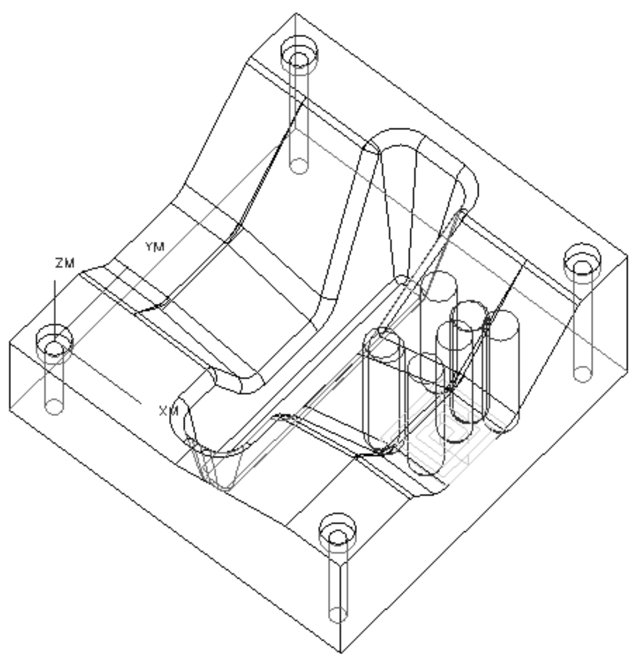
- ❑ Choose **Cutting**.
- ❑ Set **Remove Edge Traces** to **ON** (√)**.**

  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** the part file, but do not close.

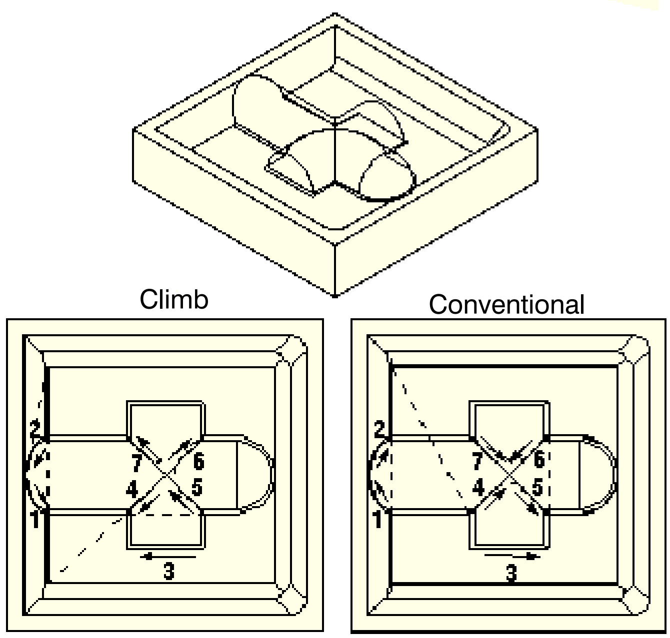
This concludes the activity.

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.

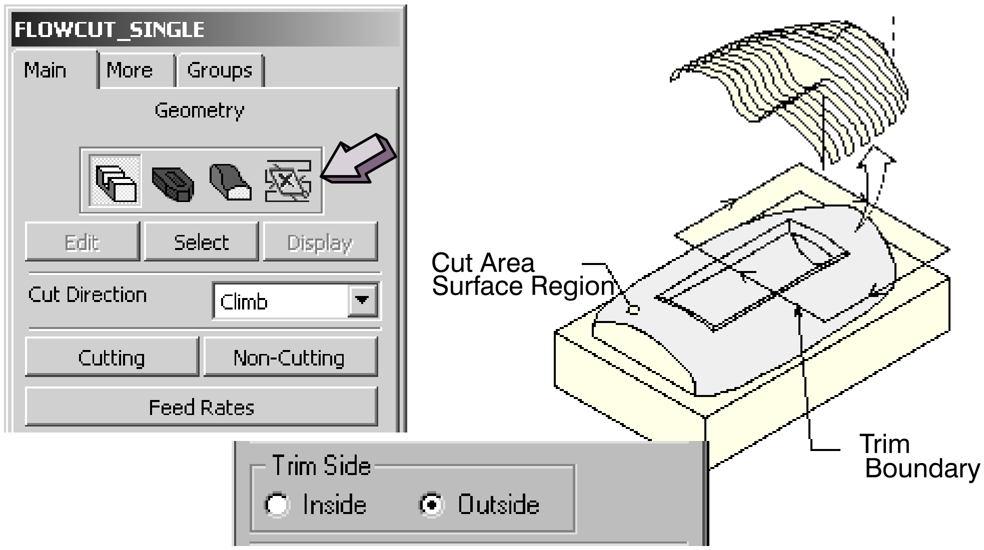
The **Mixed** option allows for the internal calculation of the cut direction.



Flow Cut drive method using Cut Area and Trim Boundary Geometry

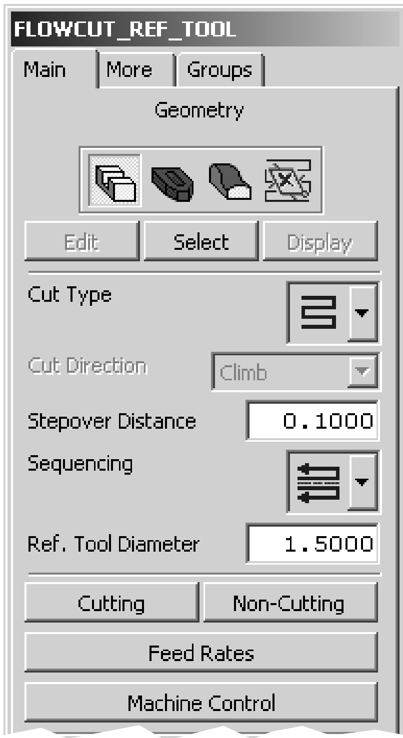
The **Flow Cut** drive method allows **Cut Area** geometry to be defined the same way as the **Area Milling** drive method. Surface regions, sheet bodies, faceted bodies and or faces can be used as the cut area. Concave valleys are analyzed within the cut area as well as concave valleys formed by the cut area and part geometry. Valleys formed by the cut area and check geometry are excluded.

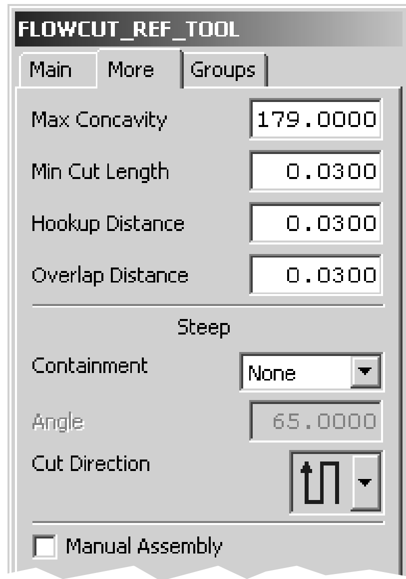
Trim boundaries can be used to further constrain cut regions. Material **Inside** or **Outside** determines the area of the cut region to be omitted**.** Trim boundaries are always **Closed,** always use an **ON** condition, and are projected to the Part geometry along the tool axis vector. More than 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.

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.

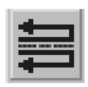
Minimum Cut Length allows you to eliminate short tool path segments that may occur in isolated areas of the part. Cutting moves shorter than this value are ignored. This option is useful in eliminating very short cutting moves that occur at the intersection of fillets.

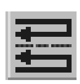
Hookup Distance allows you to eliminate unwanted gaps in the tool path by connecting disjointed cutting motions that exceed the specified Maximum Concavity angle. These unwanted motions occur where the tool retracts from the part surface and are caused by gaps between surfaces or variations in the Angle 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.

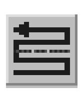
Cut Type (Zig-Zag and Zig) allows you to define how the cutter moves from one cut pass to the next.

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

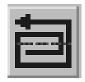
Sequencing enables you to determine the order in which the cut passes are executed.

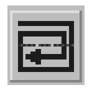
Inside-Out  results in the cut starting at the center of the Flow Cut pass and moving toward one of the outside passes. The tool then moves back to the center cut and works its way toward the opposite side. You may start the sequencing by choosing either side of the center of the Flow Cut.

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 pairs on both sides. Outside-In Alternate sequences can be generated in a Zig, Zig-Zag, or Zig-Zag with Lifts pattern.

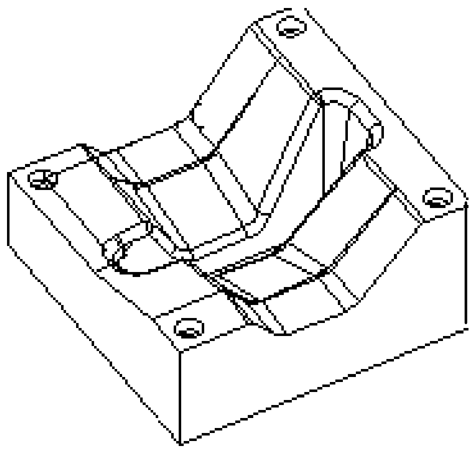
Reference Tool Diameter enables you to specify the width of the finishing cut region based on the diameter of the previous roughing (reference) tool. The tool diameter specified must be larger than the current tool.

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, **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.

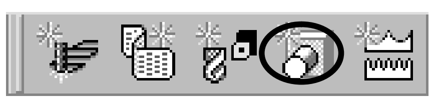
Activity 3—3: Creating a Reference Tool Operation

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 part are machined.



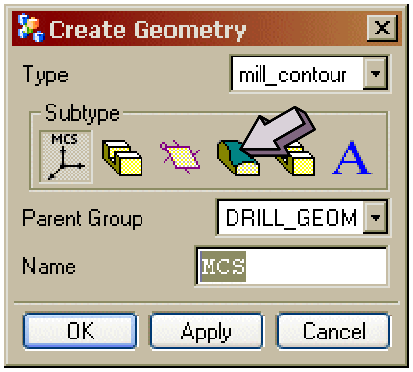
Step 1 Create the Geometry Parent Group for trim geometry.

- ❑ Continue using *****_deep_mold_mfg.prt**
- ❑ Make sure that the blank stock is displayed.
- ❑ Select the **Create Geometry** icon from the **Create** toolbar.



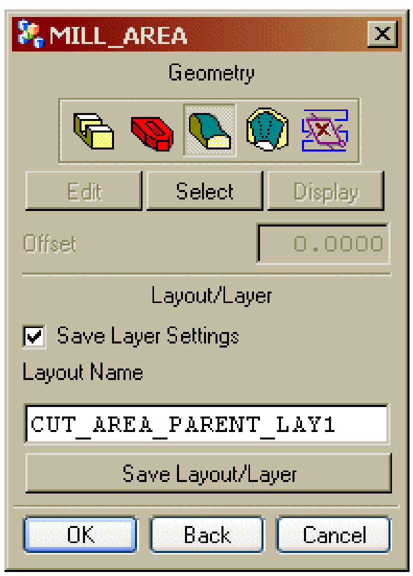
The **Create Geometry** dialog is displayed.

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

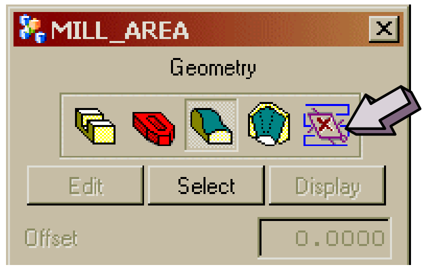


- ❑ Choose the **MILL_AREA** icon.
- ❑ Select **WORKPIECE as** the Parent **Group**.
- ❑ Key in **TRIM_BOUNDARIES** as the **Name**.
- ❑ Choose **OK**.

The **MILL_AREA** dialog is displayed.

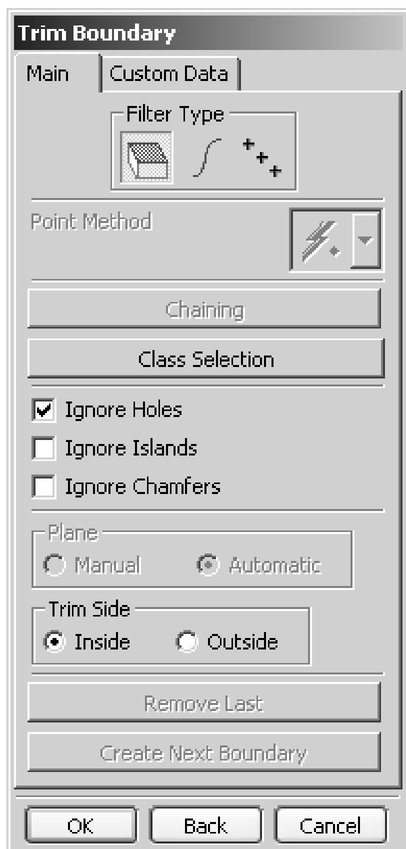


❑ Choose the **TRIM** icon.



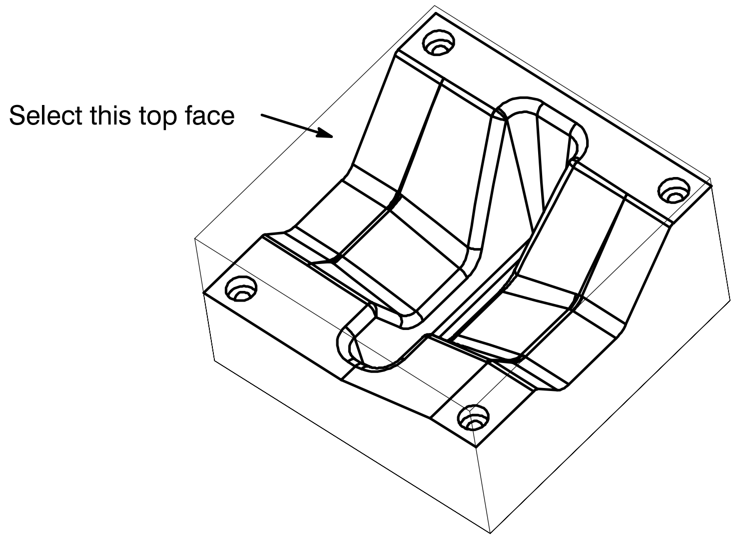
❑ Choose **Select**.

The **Trim Boundary** dialog is displayed.

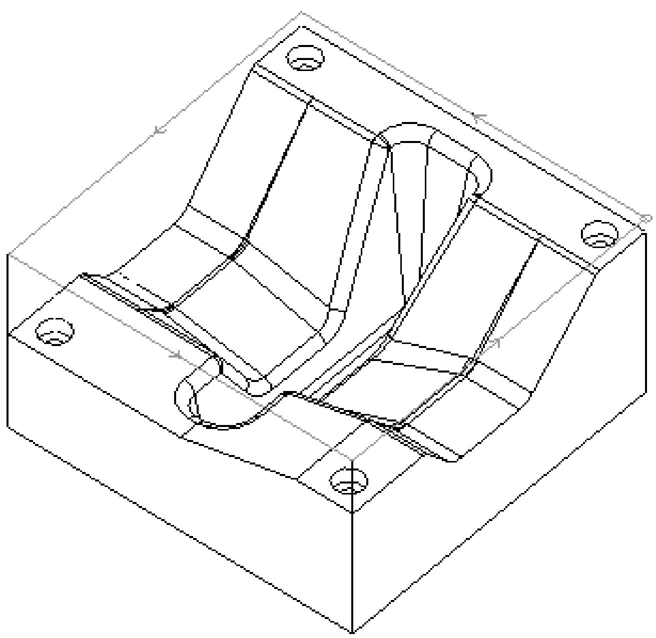


❑ In the **Trim Boundary** dialog, choose Trim Side **Outside**.

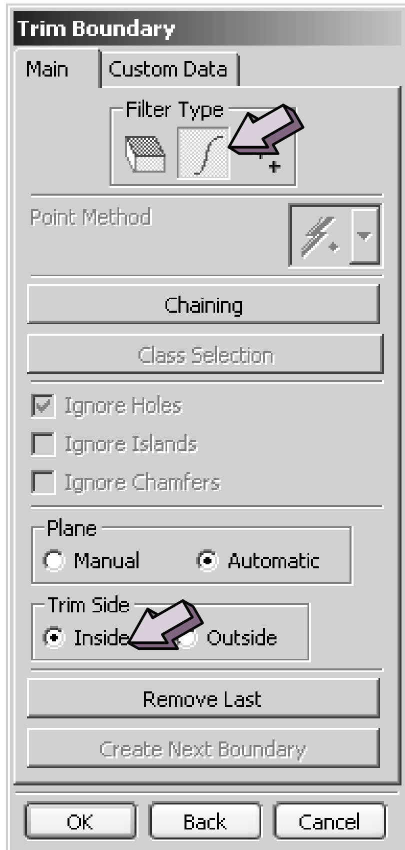
❑ Select the top face of the blank.



Note the boundary markers on the display.

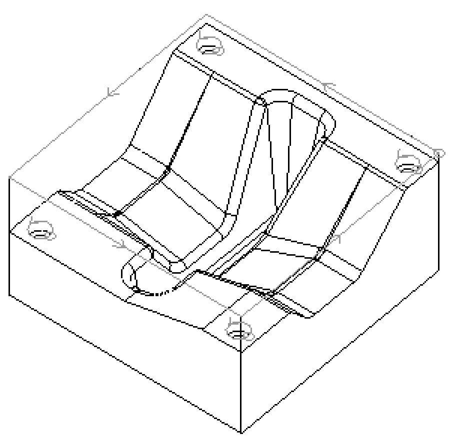


❑ On the **Trim Boundary** dialog, which is displayed, choose the **Curve Boundary** icon and set the **Trim Side** to **INSIDE.**



3

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

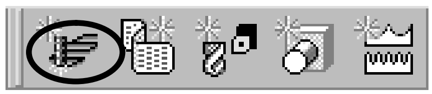


❑ Choose **OK** twice.

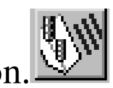
Step 2 Create the Flowcut Operation.

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.

❑ Select the **Create Operation** icon from the **Create** tool bar.



The **Create Operation** dialog is displayed.

❑ Choose the **FLOWCUT_REF_TOOL** icon. 

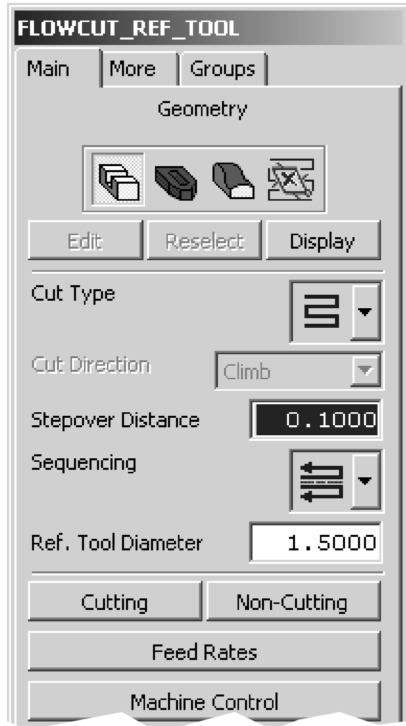
❑ Set the following:

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

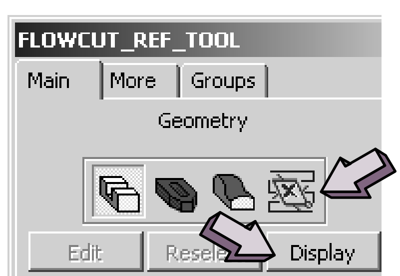
❑ Key in **FLOWCUT_1.0_BALL** for the name.

❑ Choose **OK.**

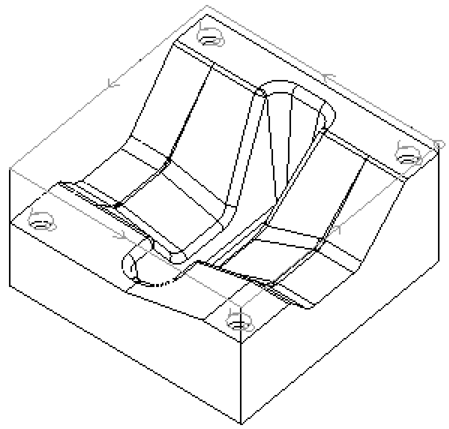
The **FLOWCUT_REF_TOOL** dialog is displayed.



- ❑ Choose the **More** tab.
- ❑ Set the **Steep Containment** from **None** to **Steep**.
- ❑ Choose the **Main** tab.
- ❑ 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.

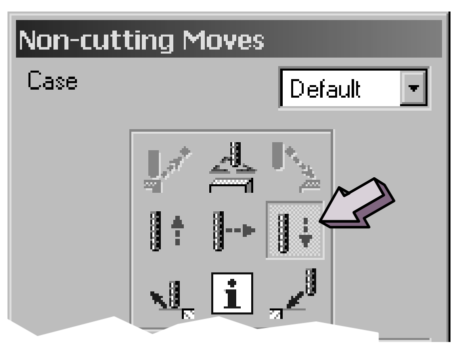


Step 3 Set the Non-Cutting Options.

You will now create a set non-cutting options when approaching the part.

❑ Choose **Non-Cutting.**

❑ Select the **Approach** icon.



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

The Clearance Plane location will be inherited from the MCS. You can use the same approach to establish the departure setting as well.

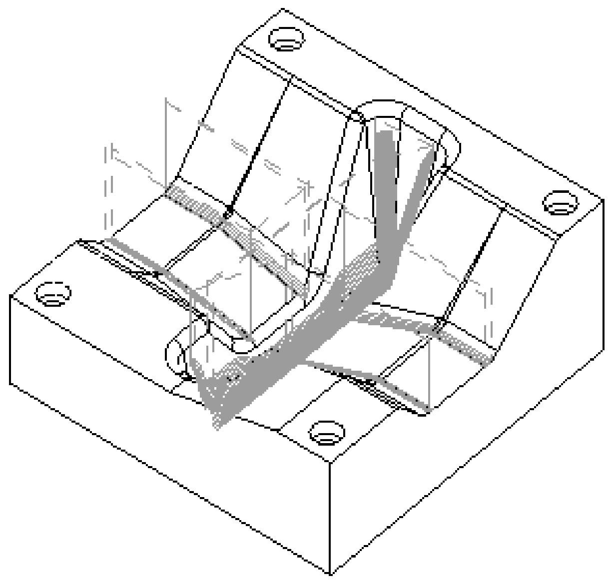
❑ Choose **OK** to return to the **FLOWCUT_REF_TOOL** dialog.

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

Step 4 Generate the tool path.

❑ **Generate** the tool path**.**

Your tool path should look similar to the following illustration.



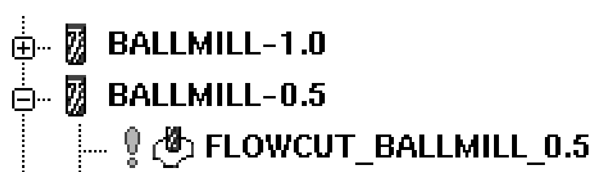
❑ After examining the tool path, choose **OK.**

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.**

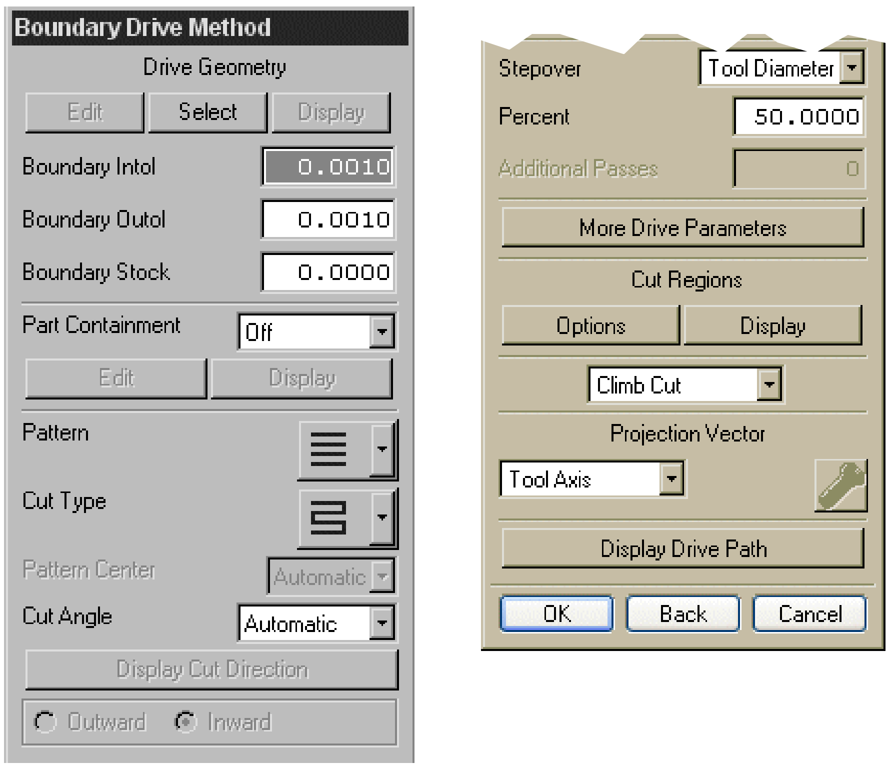
Step 6 Reusing the just created operation.

- ❑ **Copy** the operation **FLOWCUT_BALLMILL_0.5**, and **Paste** it under the **BALLMILL_0.375** tool.
- ❑ 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** and **Close** the part file.

You have completed this activity.

Boundary drive method

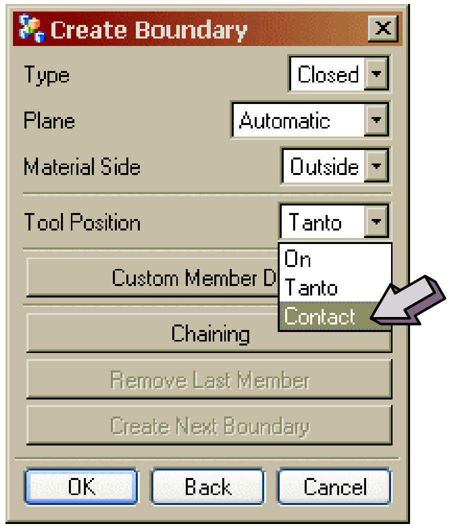
As mentioned earlier in this lesson, the Boundary drive method allows you to define cut regions by specifying Boundaries and Loops. Boundaries utilize cut regions which are defined by Boundaries, Loops, or a combination of both. The tool path is created by projecting drive points from the cut region 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.



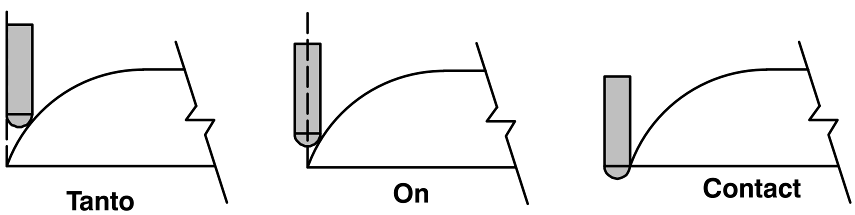
The following are some of the options available when using the Boundary drive method.

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.

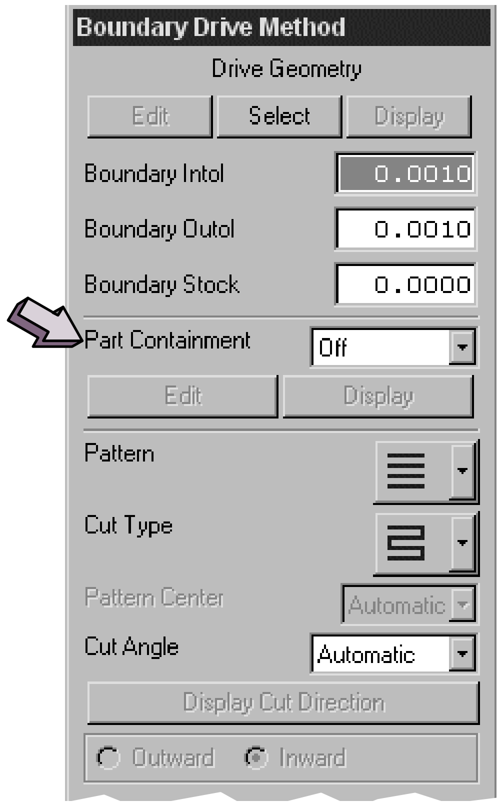


The **Contact** position cuts the entire boundary. The **Tanto** and **On** conditions leave a portion of the boundary uncut. The **Contact** condition is useful when cutting exterior surfaces, but it is not designed for cutting pockets.

NOTE You cannot mix **Contact** position with **On** or **Tanto** positions.

Boundary drive method — Part Containment

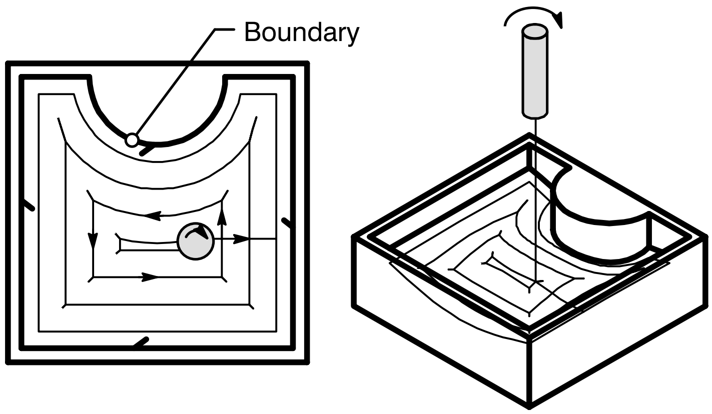
The **Part Containment** options define cut regions by using exterior edges of part surfaces. When you select this option, the system automatically identifies the exterior edges of all part surfaces and creates Loops along these edges.



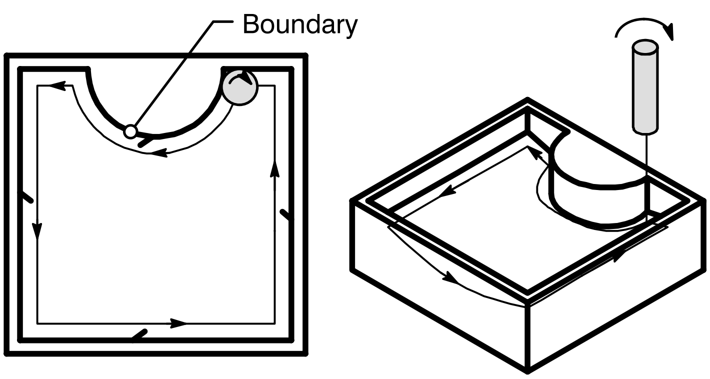
Loops define the main area to be cut as well as islands and pockets to be avoided. Loops are similar to Boundaries in that they define cut regions. They are dissimilar to Boundaries in that they are generated directly on the part surface edges and are not projected.

Boundary drive method — Pattern

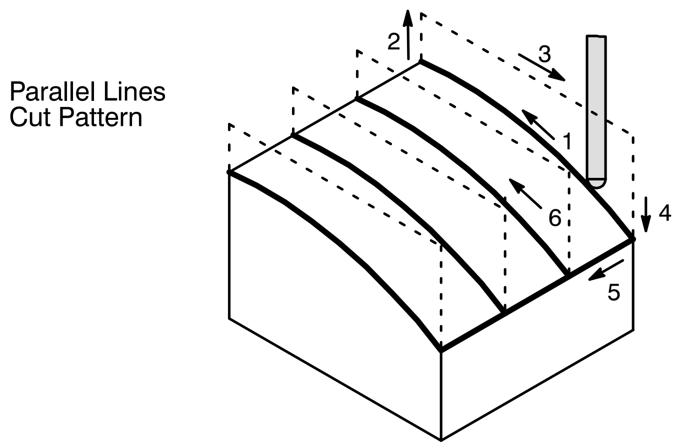
Follow Periphery creates a cut pattern that produces a sequence of concentric passes following the contours of the cut region. Like Zig-Zag, this cut type maximizes cutting moves by allowing the tool to remain continually engaged during stepovers.



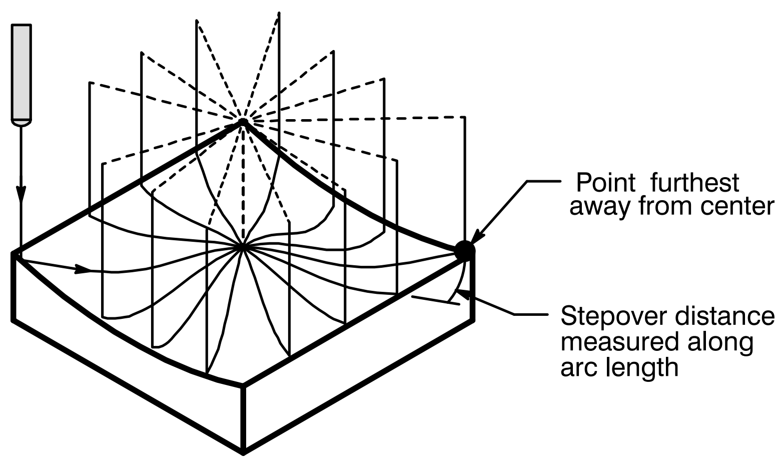
Profile creates a cut pattern that follows the perimeter of the cut region.



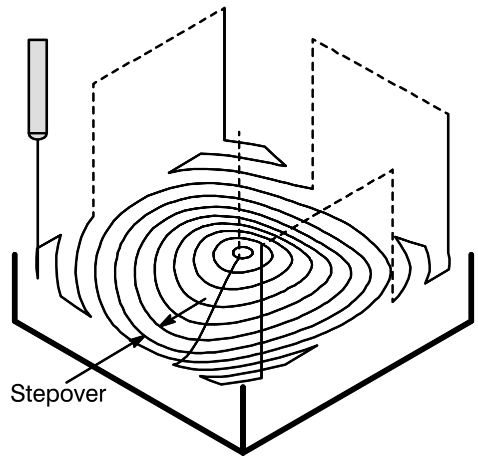
Parallel Lines creates a cut pattern defined by a series of parallel passes. This option requires you to specify a Cut Type of Zig-Zag, Zig, Zig With Contour, or Zig with Stepover and allows you to specify a Cut Angle.



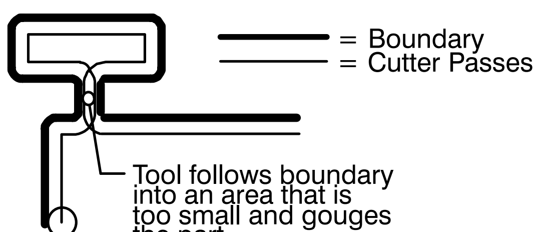
Radial Lines creates linear cut patterns extending from a user-specified or system calculated center point. This pattern allows you to specify a Cut Type, a Pattern Center and pocketing method as Inward or Outward. It also enables you to specify an angular stepover which is unique to this cut pattern. 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.



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.

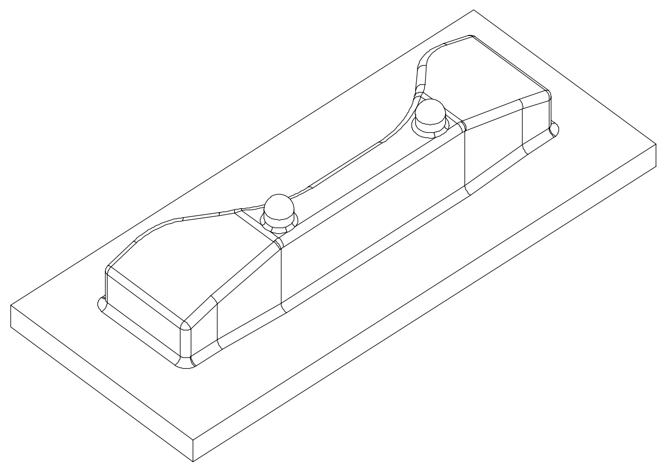


Activity 3–4: Using the Boundary drive method

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 Open and rename the part file and enter the Manufacturing Application.

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

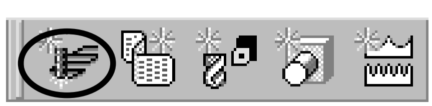


❑ **Save As** ***_ **bumper_mfg .prt** where *** represents your initials.

❑ Enter the **Manufacturing** application.

Step 2 Create a Fixed Contour Operation.

❑ Select the **Create Operation** icon from the **Create** tool bar.



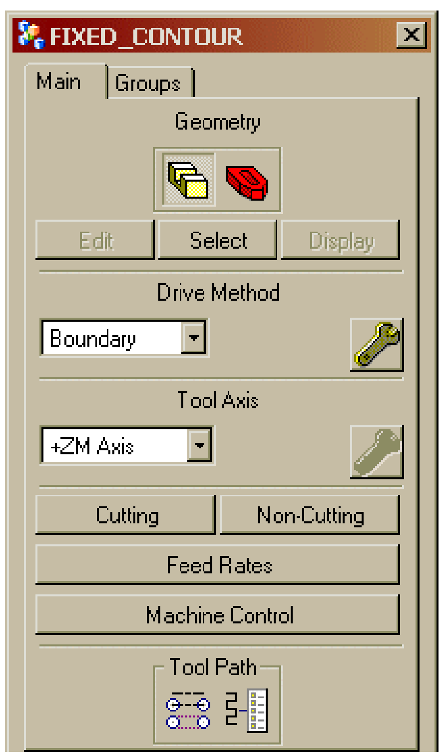
The **Create Operation** dialog is displayed.

- ❑ 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.**

  The **Fixed Contour** dialog is displayed.

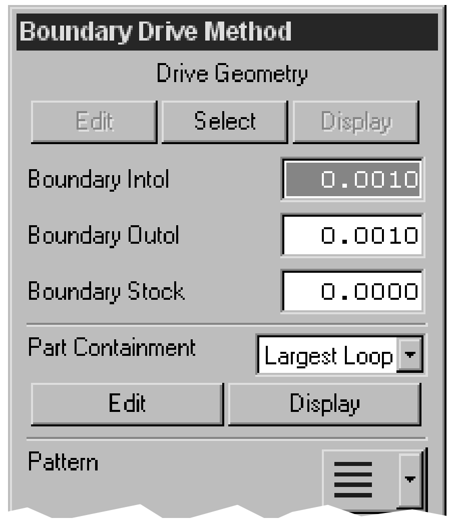


Step 3 Create the Boundary.

- ❑ Under the drive method label, choose **Boundary.**

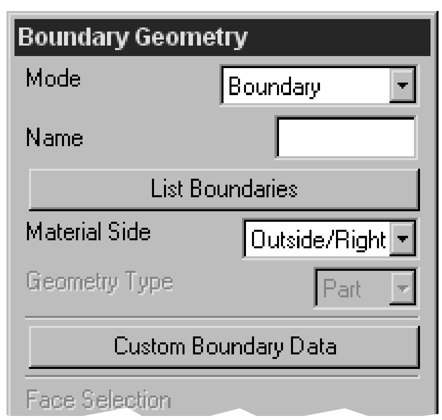
3

The **Boundary Drive Method** dialog is displayed.



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

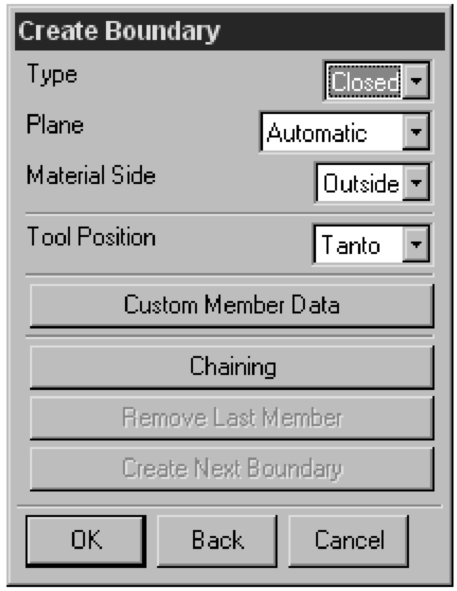
The **Boundary Geometry** dialog is displayed.



3

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

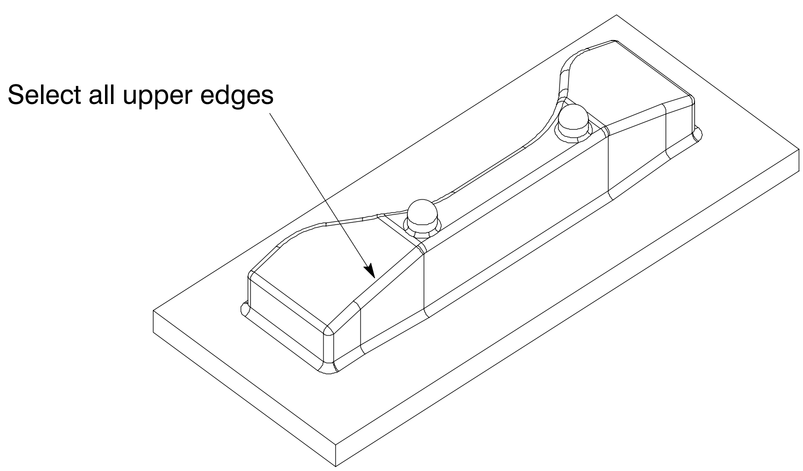
The **Create Boundary** dialog is displayed.



- ❑ 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.

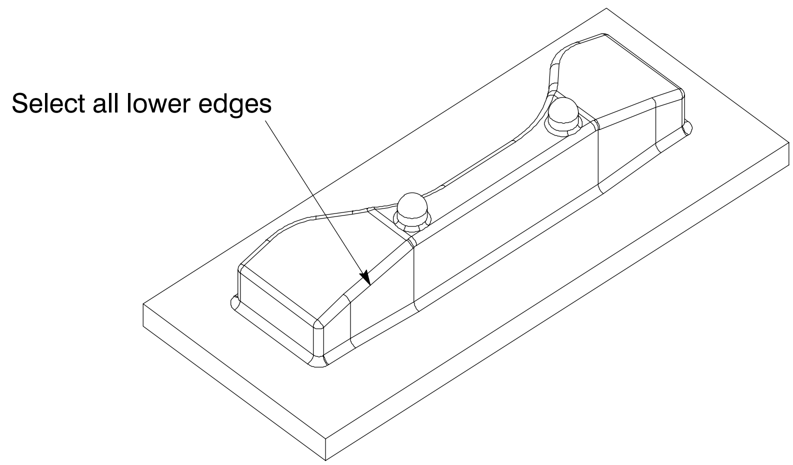


You will now create the lower boundary.

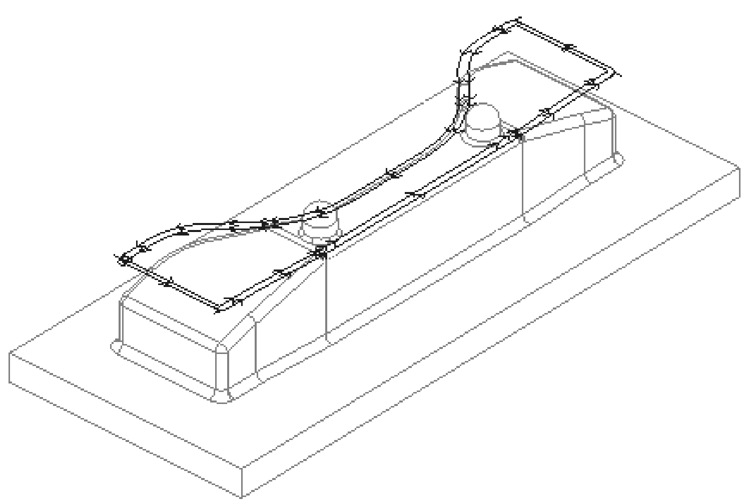
❑ After selecting all upper members, choose the **Create Next Boundary** button.

❑ Change **Material Side** to **Outside.**

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



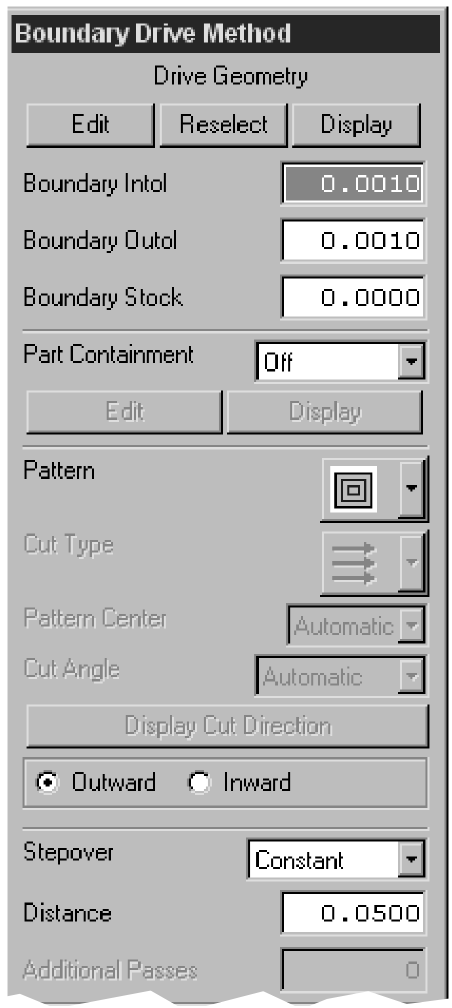
❑ Choose **OK** until you return to the **Boundary Drive Method** dialog.



Step 4 Specify Boundary drive method Option Settings.

❑ Turn **Part Containment OFF.**

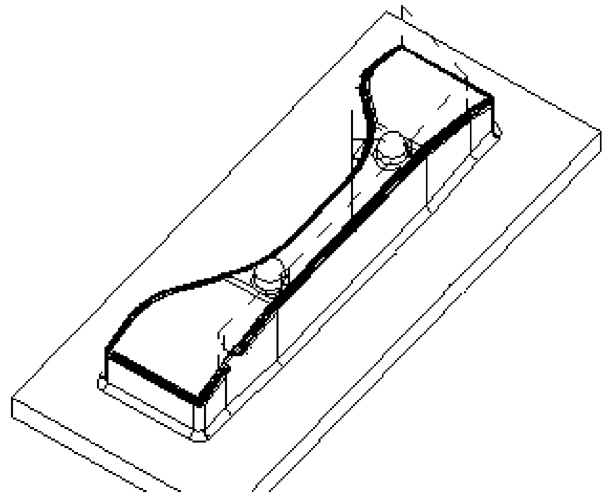
- ❑ Change the **Pattern** to **Follow Periphery.**
- ❑ Choose the **Outward** radio button.
- ❑ Change the **Stepover** to **Constant.**
- ❑ Change the **Distance** to **0.05.**



- ❑ 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.

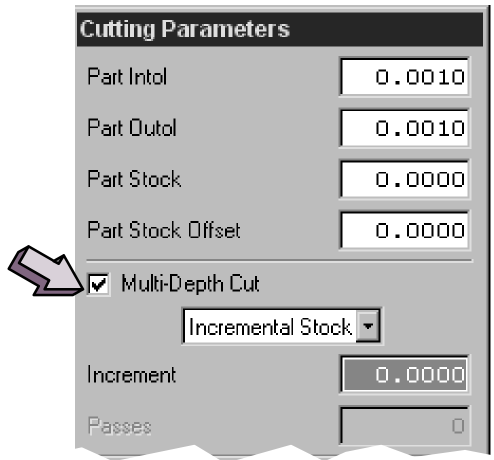


❑ Choose **OK** to accept the operation.

You are finished with this activity. Do not close the part since you will use it in the next 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.

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

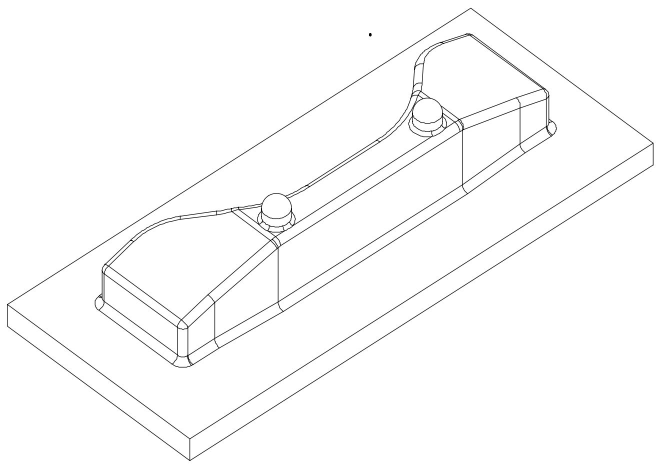
The traversal between levels can be minimized if the **Engage** option is set to **Manual** in **Non-Cutting Moves**. After finishing each level, the tool retracts to the previous level plus the **Outol** distance for that level before traversing and engaging to the next level.

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**

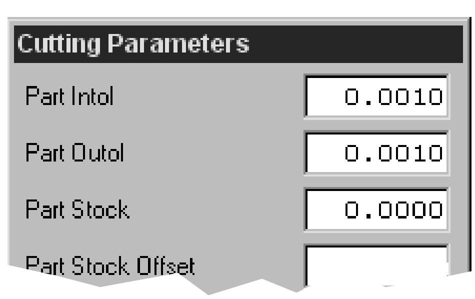


❑ In the Operation Navigator, double click on **BOUNDARY_1** .

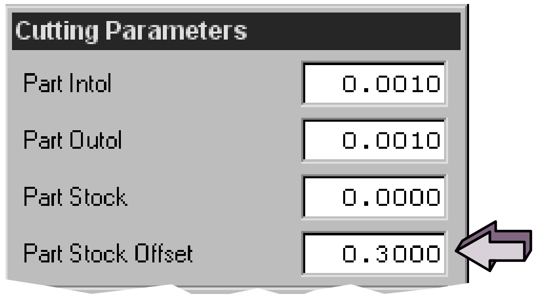
The **Fixed Contour** dialog displays.

❑ Choose **Cutting**.

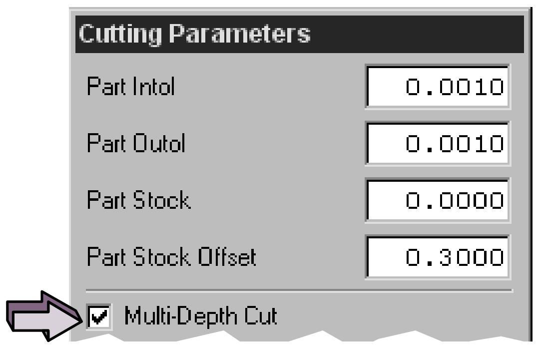
The **Cutting Parameters** dialog displays.



❑ Key in **0.300** in the **Part Stock Offset** field. This specifies the amount of stock on the surface to be machined.

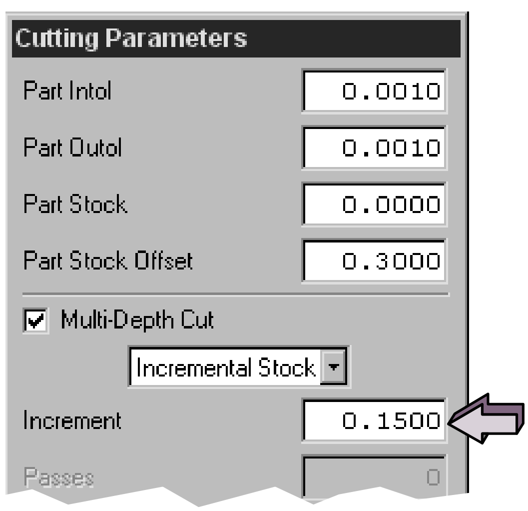


❑ Turn **Multi-Depth Cut** on.



❑ Leave the default at **Incremental Stock**.

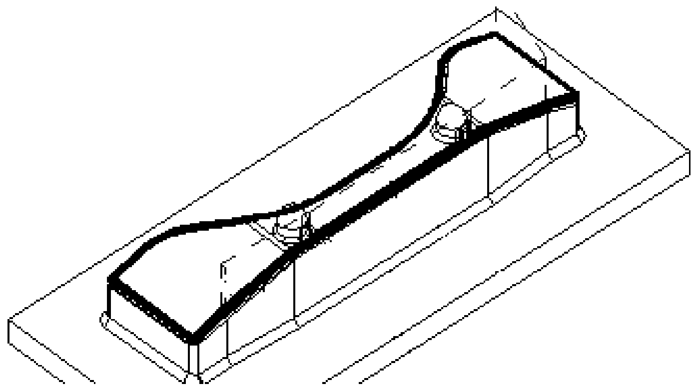
❑ Key in **0.150** as the Increment.



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

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.

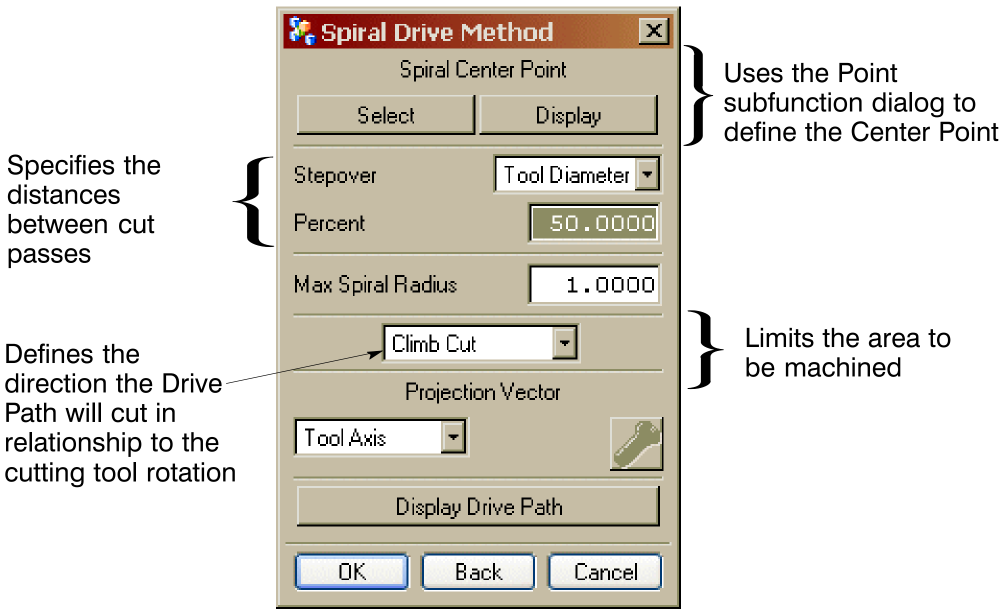
❑ **Save** the part file.

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.

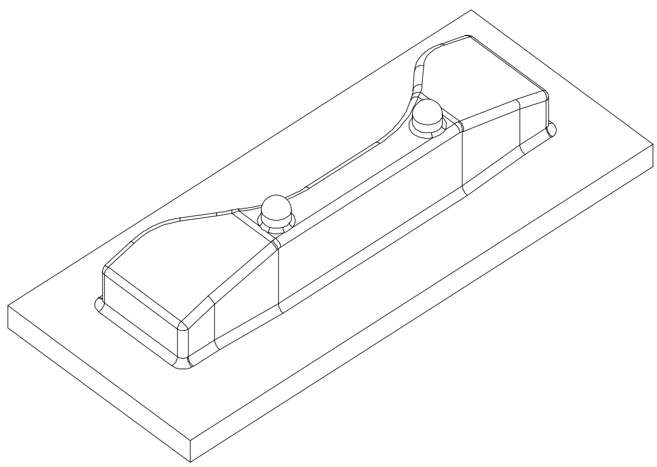
Activity 3–6: Creating a Spiral drive method Tool Path

The following activity demonstrates use of the Spiral drive method.

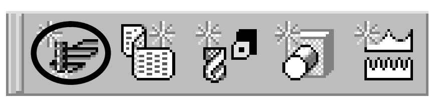
Step 1 Create a Spiral Drive operation.

❑ Continue with ***_bumper_mfg part.

.



❑ Select the **Create Operation** icon.



The **Create Operation** dialog is displayed.

Step 2 Create a Spiral Drive method, 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**

3

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

❑ Choose **OK**.

The **Fixed Contour** dialog is displayed.

Step 3 Specify the drive method.

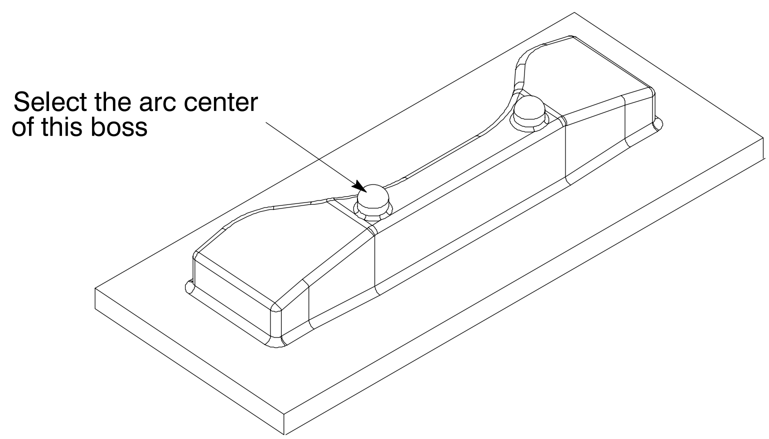
❑ Change the drive method to **Spiral.**

❑ 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.



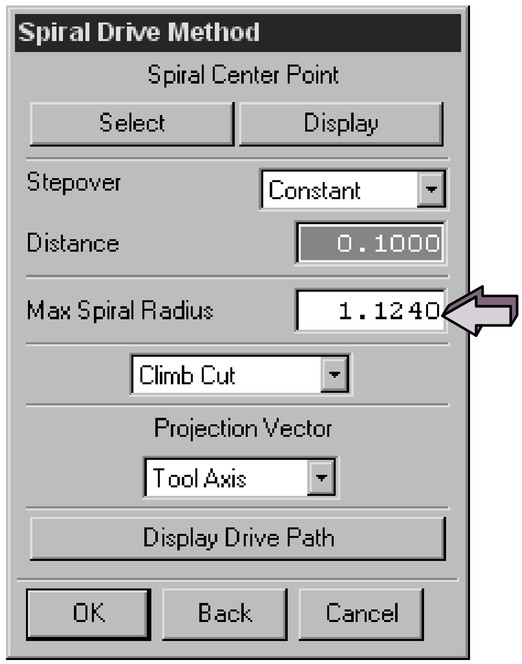
❑ Change the **Stepover** to **Constant.**

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

- 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.



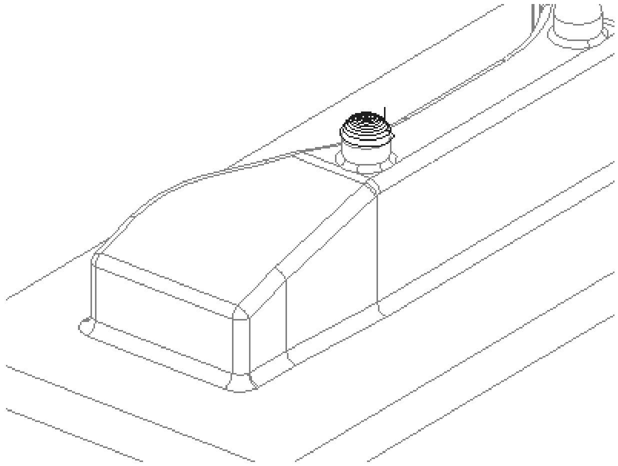
❑ Under Projection Vector, choose **Tool Axis.**

❑ 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**.**



❑ 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** and **Close** 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.

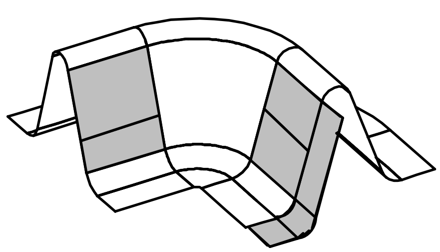
The array of drive points is first created on the selected drive surface and then projected along the specified projection vector to the part surface(s). The tool positions to the part surface at Contact Points. As the tool moves from one Contact Point to the next, the tool path is created using the 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:

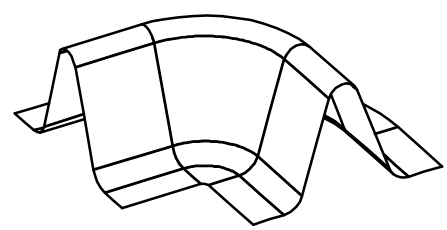
- The drive geometry must form a grid of rows and columns where each row or column has the same number of objects as every other row or column in the grid
- Any two adjacent objects must share a common edge and may not contain gaps that exceed the Chaining Tolerance defined under Preferences
- When you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

3



Incorrect:

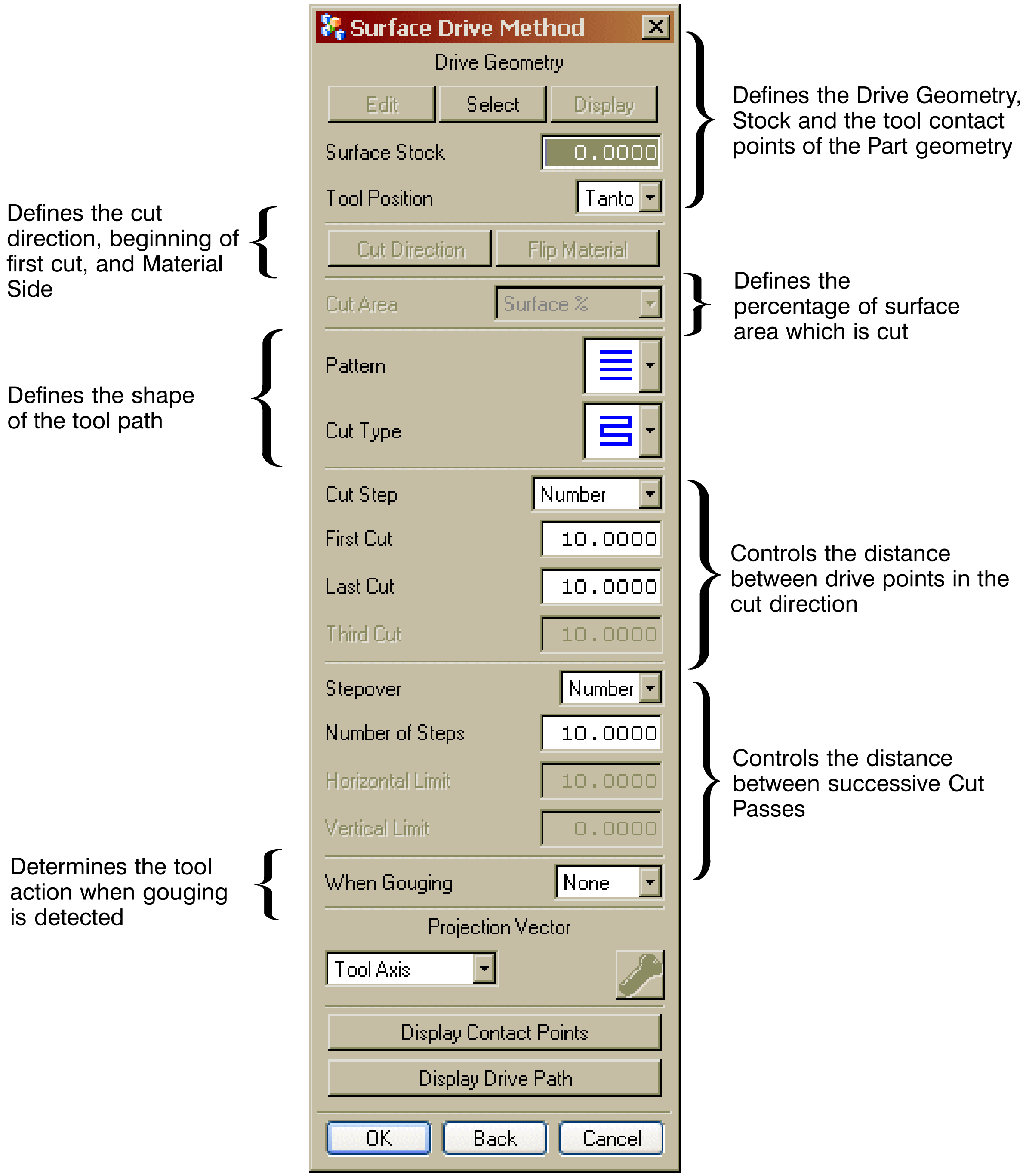
- Inconsistent number of objects in rows and in columns
- Adjacent object edges are not equal length



Correct:

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

The **Surface Drive Method** dialog follows:

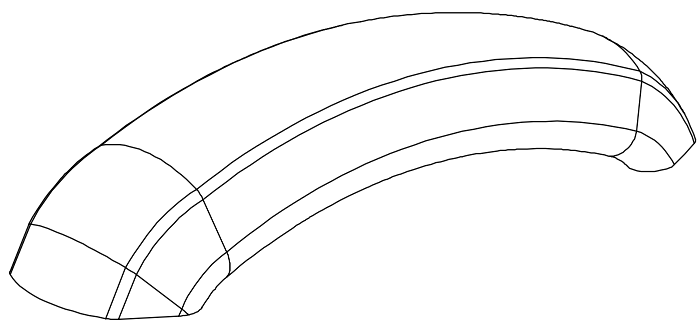


Activity 3–7: Using the Surface drive method

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_handle_mfg_asmb.prt**



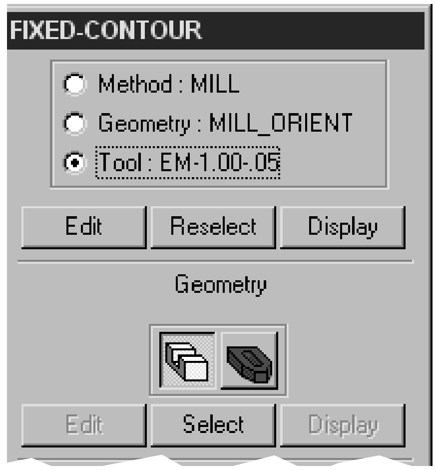
❑ Rename the part ***_**handle_mfg_asmb.prt** where *** represents your initials**.**

❑ Enter the **Manufacturing Application**.

Step 2 Edit an existing operation.

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

The **Fixed Contour** dialog is displayed.



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

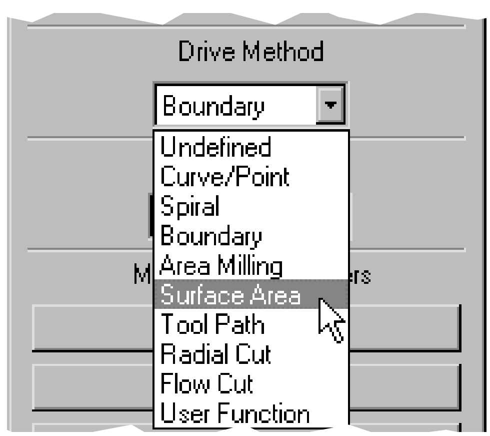
- the tool which is an end mill, designated **EM-1.00–.05**
- **Engage** and **Retract** Non-Cutting Moves

3

Step 3 Define the drive method as Surface Area.

You are going to specify the **Surface Area** drive method.

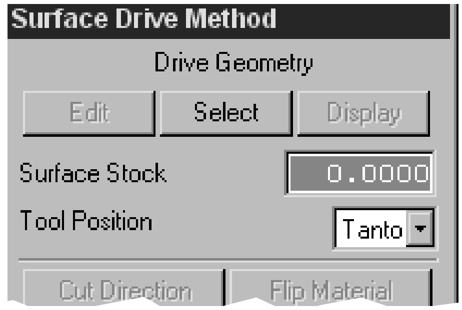
❑ In the drive method area, choose **Surface Area.**



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.

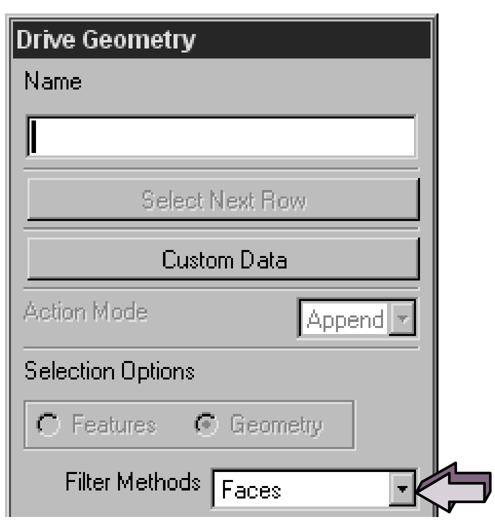
NOTE When you specified the options on the Fixed Contour dialog, you did not select any part geometry. Not selecting part geometry, in this particular operation, will make processing quicker.

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

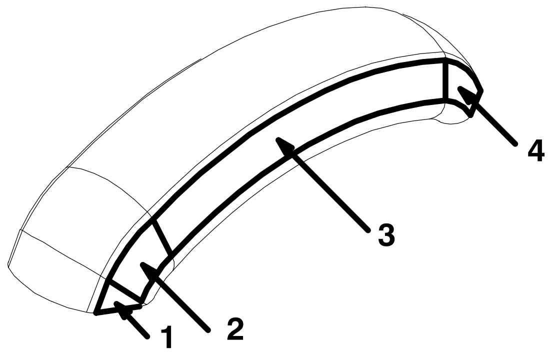
The **Drive Geometry** dialog is displayed.

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**.

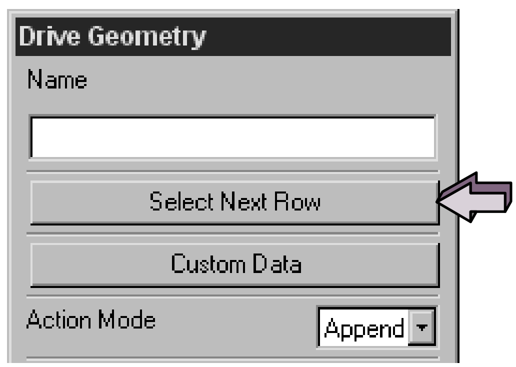


❑ In the graphic window, choose faces in the specified order as shown (four surfaces total).



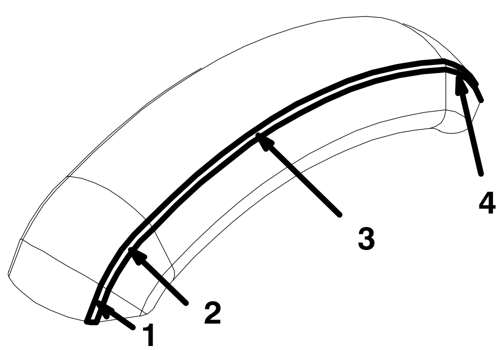
You are ready to select a new row. When you finish selecting the first row, you must specify that you want to begin selecting the next row.

❑ Choose the **Select Next Row** button.



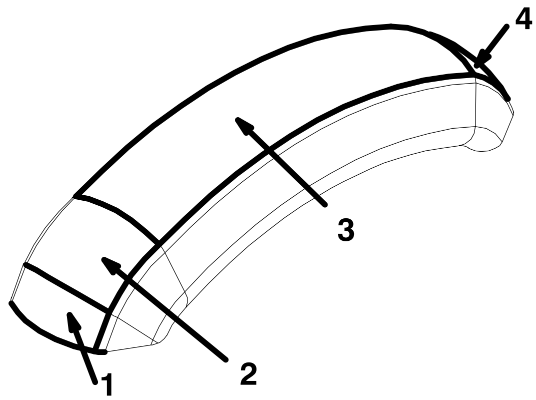
You must select the second row of surfaces and all subsequent rows in the same order as the first.

❑ Select the second row (four surfaces total).

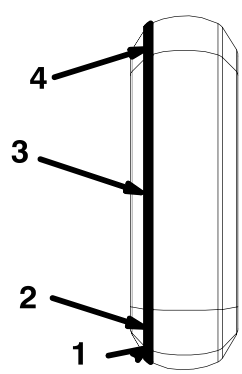


You do not need to select the **Select Next Row** button again. The number of surfaces in subsequent rows must be the same.

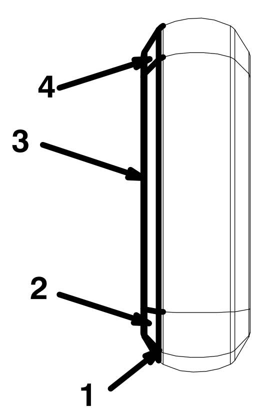
❑ Select the third row of surfaces as shown below (four surfaces total).



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

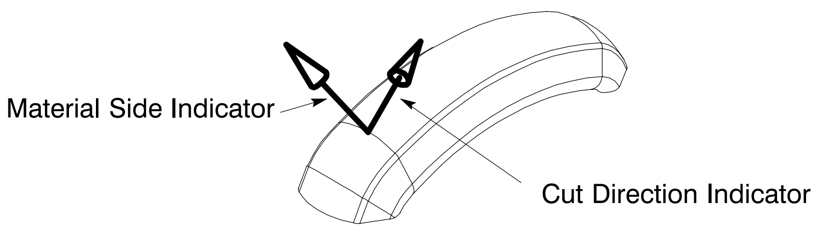


❑ Select the fifth and final row of surfaces as shown (four surfaces total).

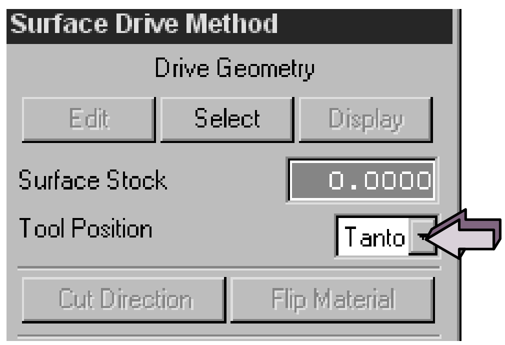


❑ Choose **OK** when finished.

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.



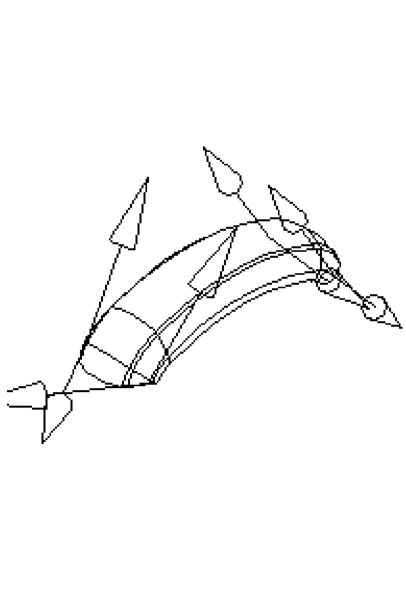
Step 5 Define the Cut Direction.

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

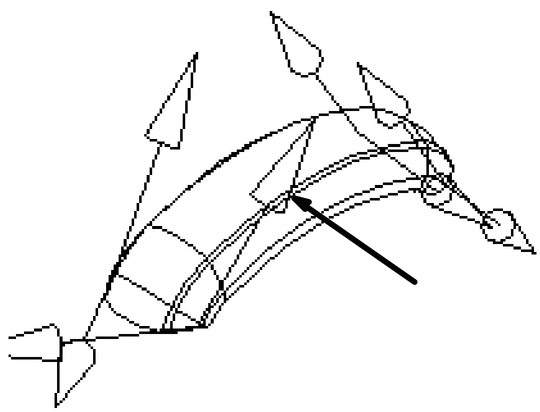
❑ Choose the **Cut Direction** button.

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.

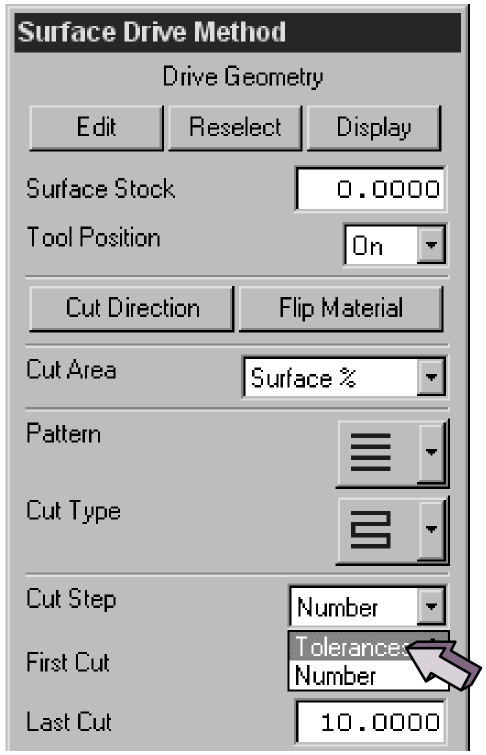


The selected cone head is marked and you are returned to the **Surface Drive Method** dialog.

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** option defines an **Intol** and **Outol** value.

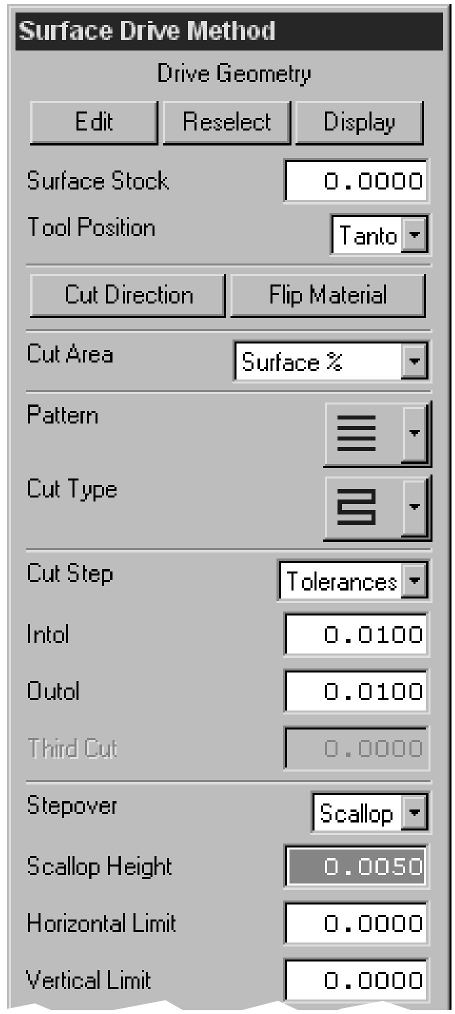
You are going to use the Tolerances option.

❑ 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**.

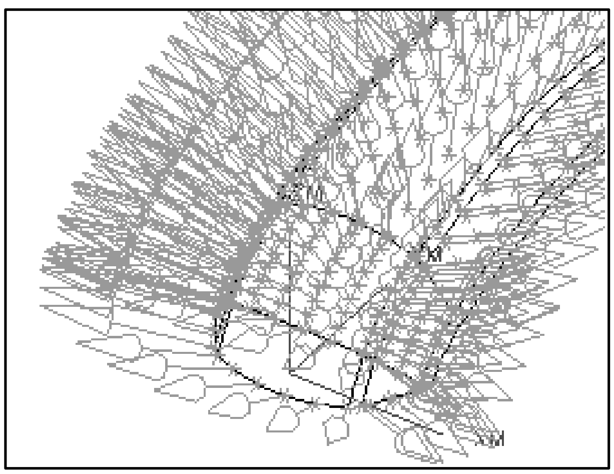
❑ Change the projection vector to **Tool Axis.**

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 surface.

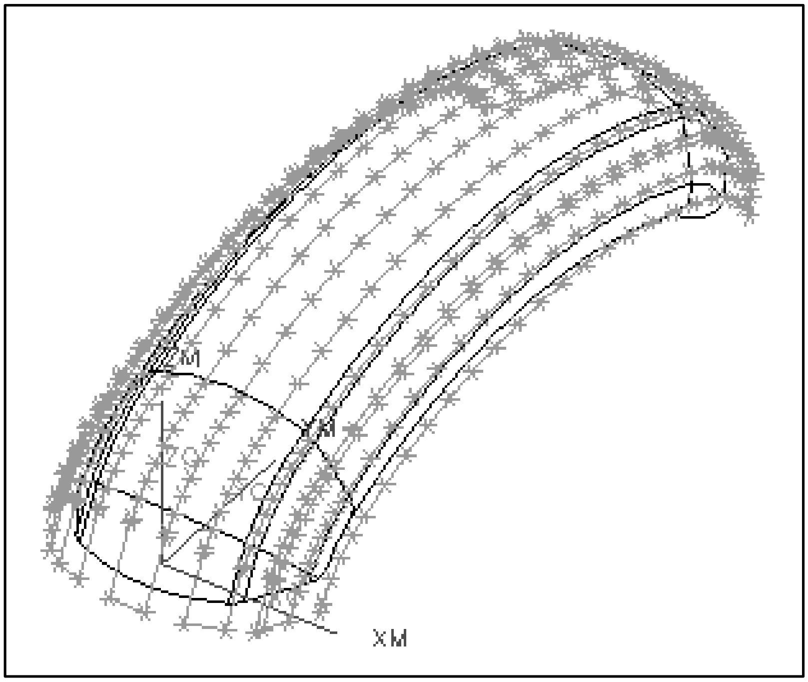


Step 10 Display the Drive Path.

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 surface. Each Drive Point is displayed on the drive surface.

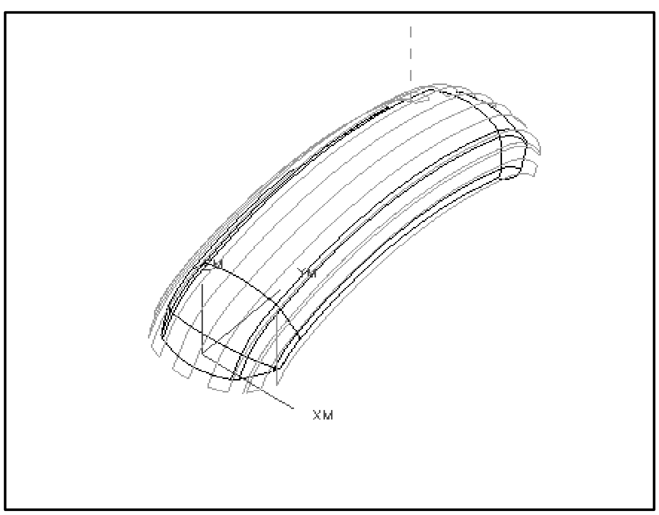


❑ Choose **OK** to return to the **Fixed Contour** dialog.

Step 11 Generate the tool path.

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

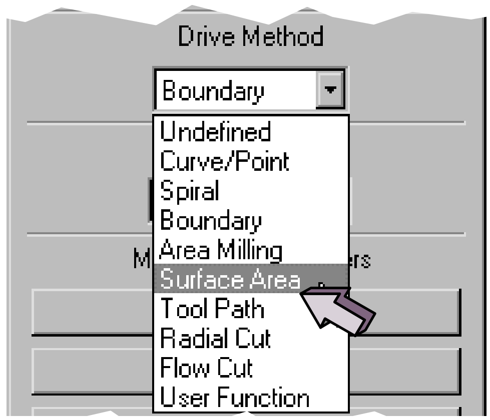
The tool path is generated. It cuts along all the drive surfaces that were specified. In the following portion of the activity, you will cut only a portion of the drive surface.



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

You are going to change the amount of the surface area 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 %**.

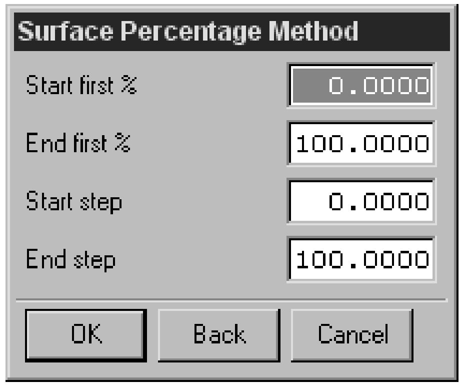
Note that you have two options:

- Surface Percentage (uses a percentage of the surface area)
- Diagonal Points (uses two diagonal points to define the cuter 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.

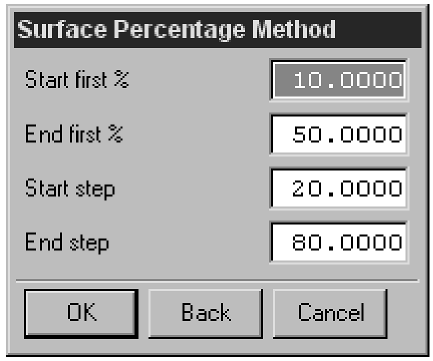
Start step – is the distance along the first Stepover direction.

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.

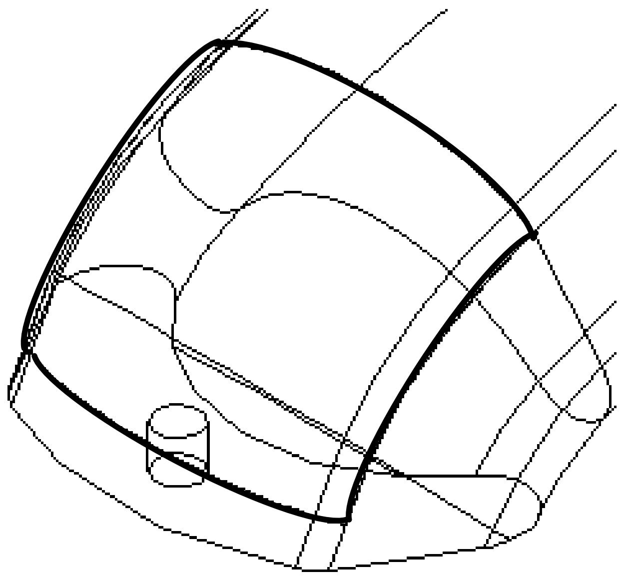
❑ Key in the following values:

- **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.

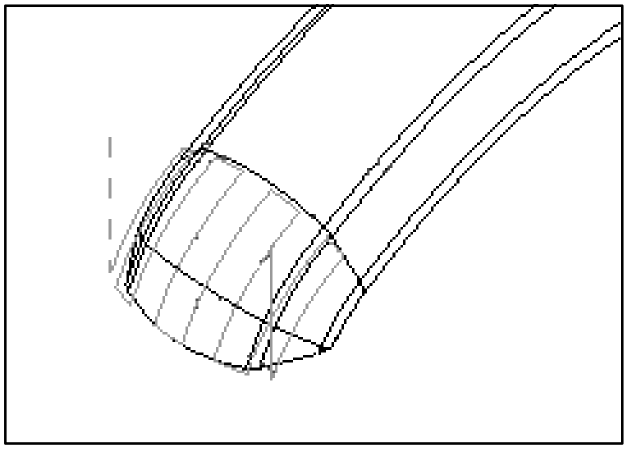
❑ Choose **Display Drive Path**.

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.

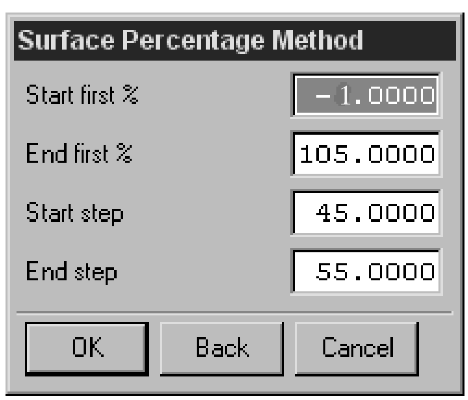
❑ In the drive method area, choose **Surface Area**.

❑ Choose **Surface %**.

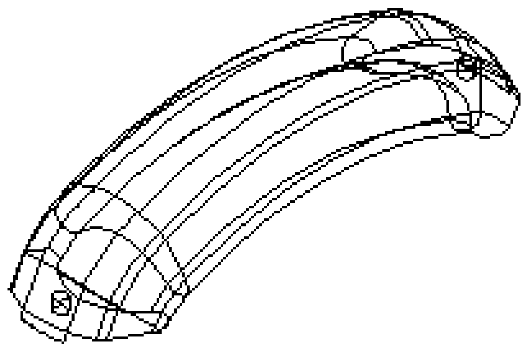
You will now change some of the Surface Percentage values.

❑ 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.

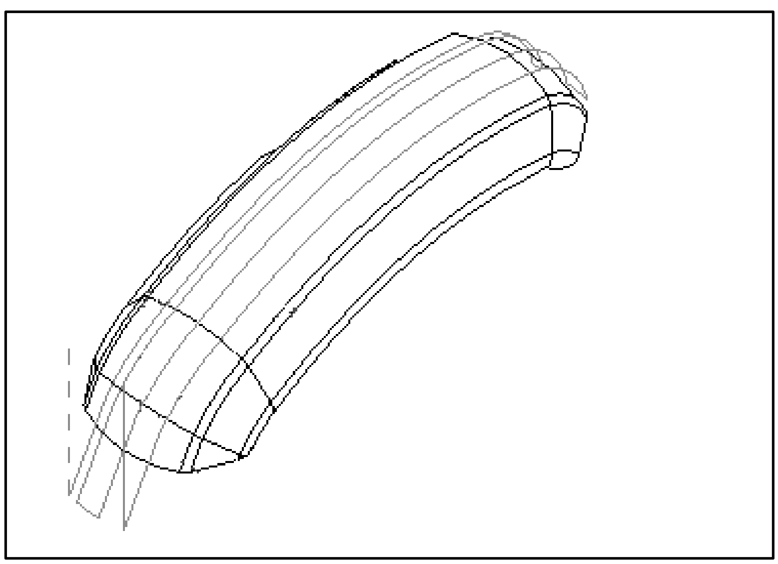


❑ Choose **OK** until you return to the **Fixed Contour** dialog.

Step 15 Generate the tool path.

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

The tool path travels beyond the drive surface.



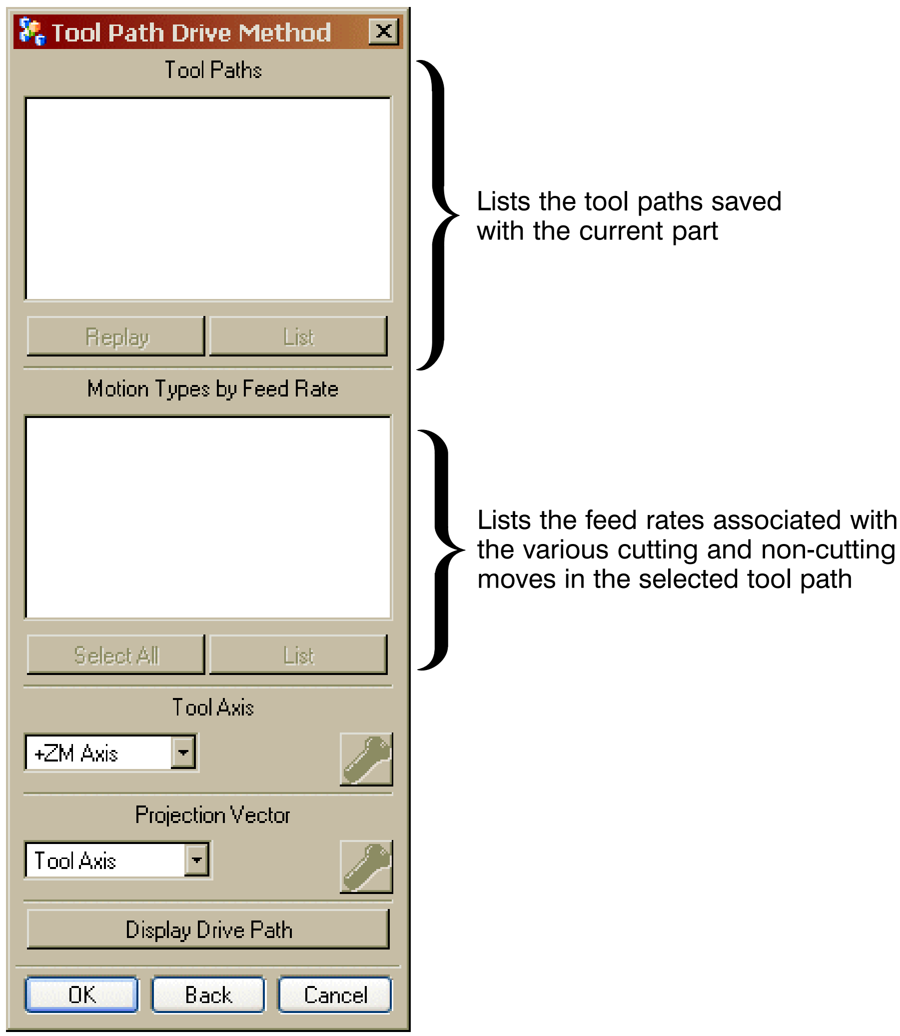
❑ Choose **OK** to save the operation.

❑ **Save** the part file.

You are finished with this activity.

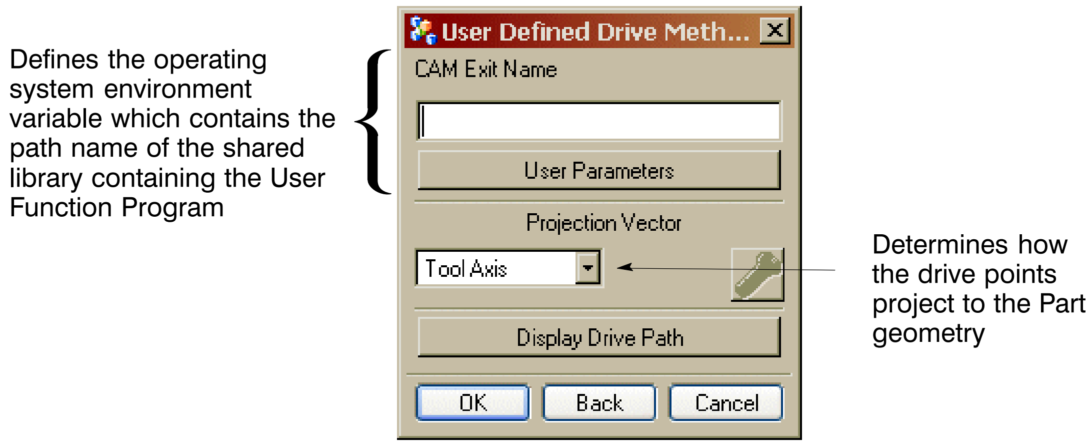
Tool Path drive method

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.



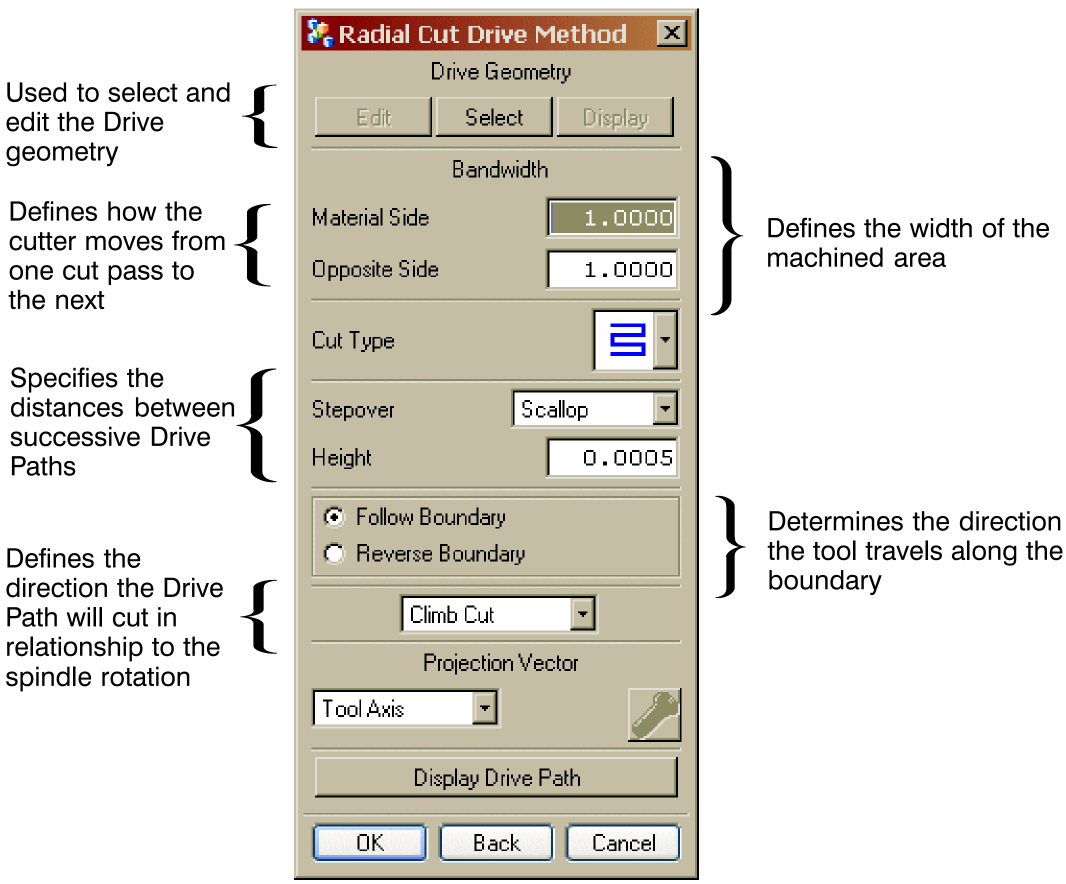
User Function

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.



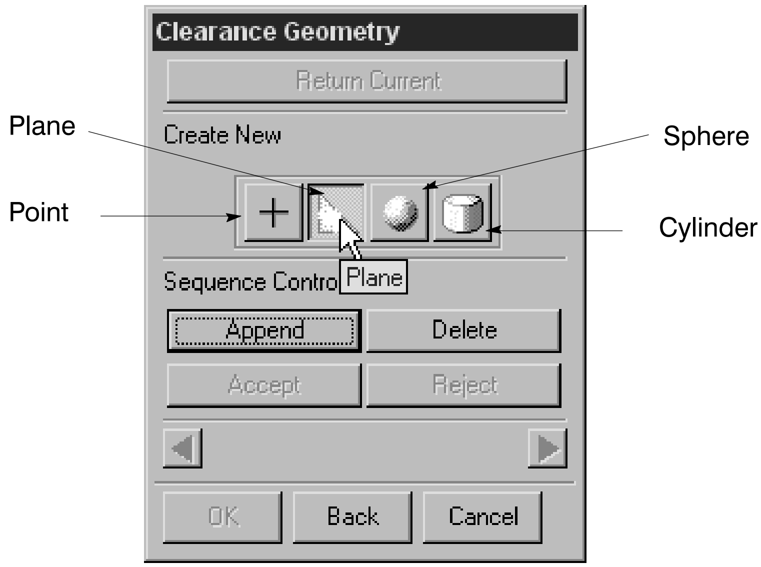
Radial Cut drive method

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.

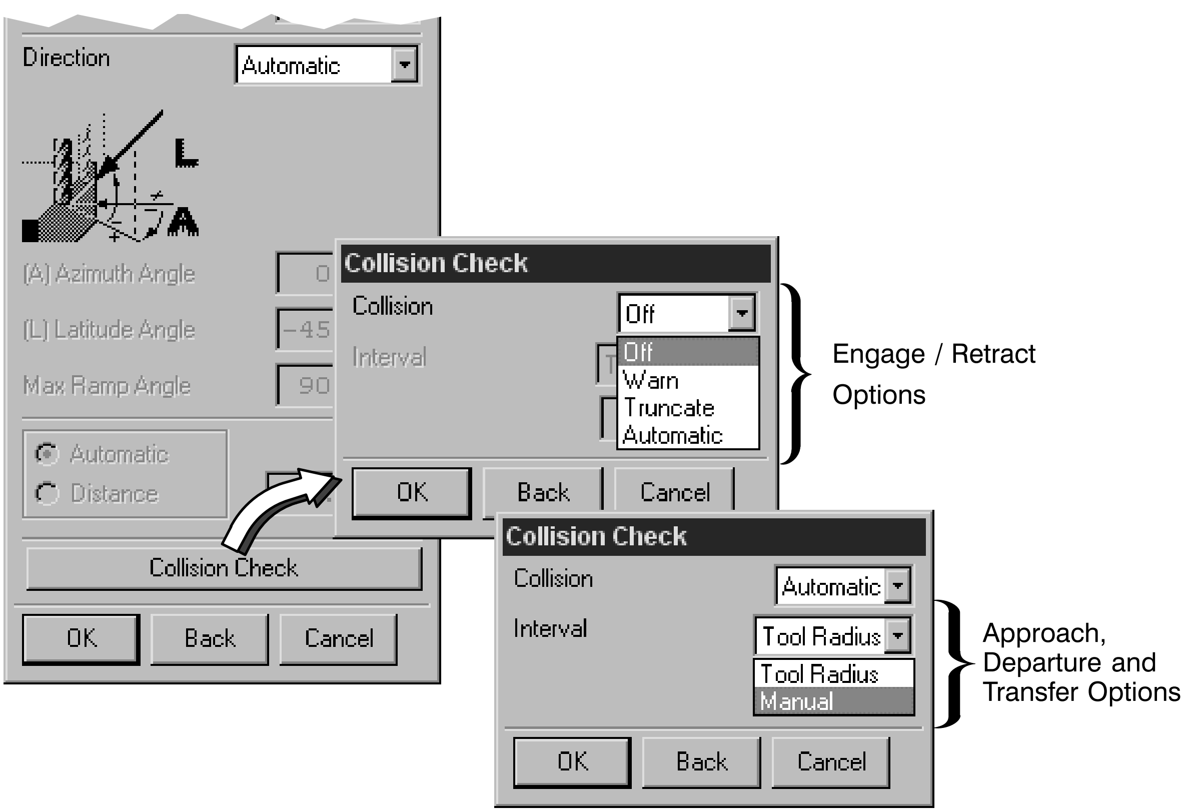
Skip – Replaces the drive points that gouge with straight line motion between the last and next non-gouging points.

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

(This Page Intentionally Left Blank)

Libraries

Lesson 4

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
- Create graphical tool assemblies
- add tooling data to database files

This lesson contains the following activities:

Overview of CAM Libraries

Libraries are convenient and easy tools 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 is the associated mechanism for queries, establishes dialog layout definitions, attribute mappings, option menu definitions, library reference names and delimiters.

Sample Configuration file (mill_contour.dat)

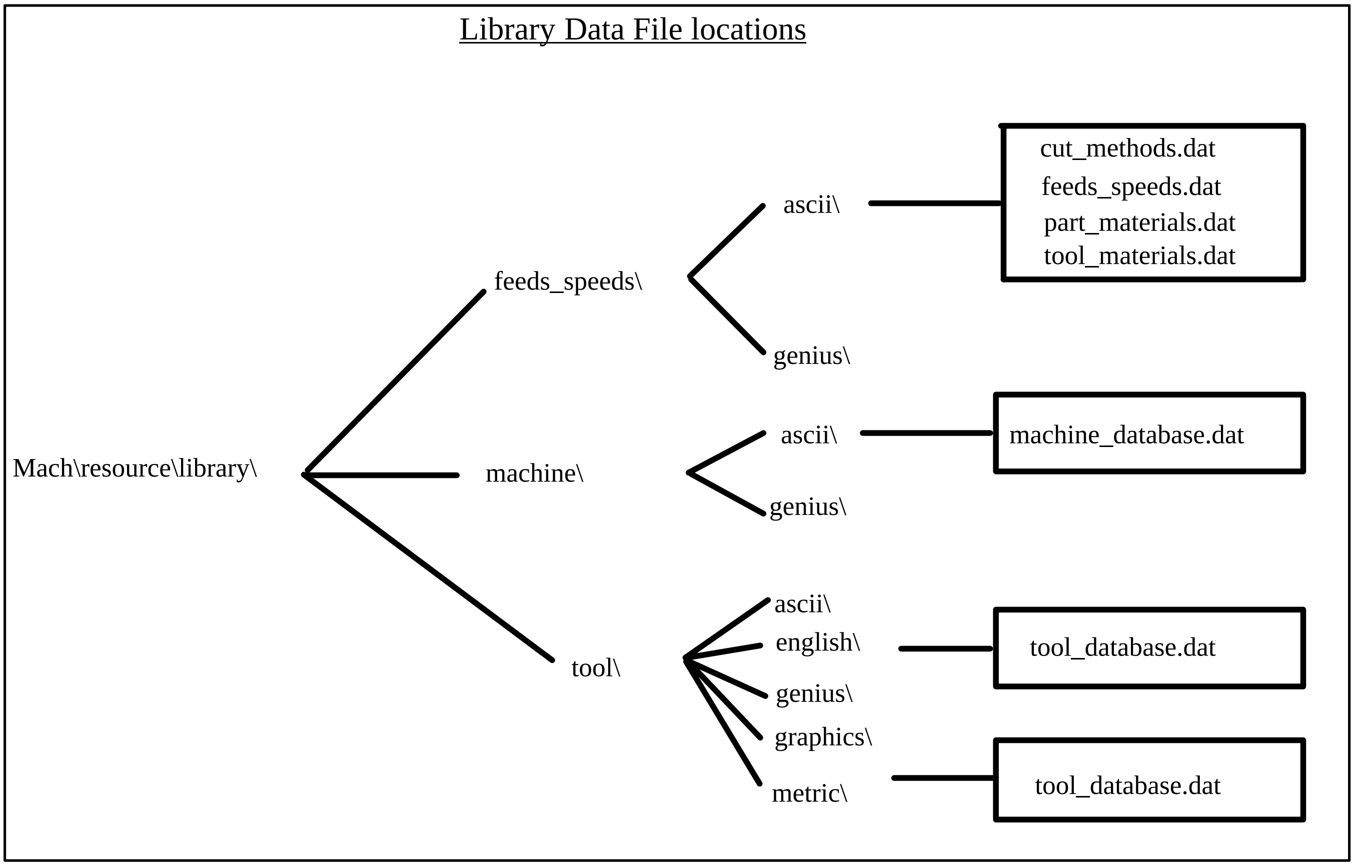
```
TEMPLATE_OPERATION,${UGII_CAM_TEMPLATE_SET_DIR}mill_contour.opt
TEMPLATE_DOCUMENTATION,${UGII_CAM_SHOP_DOC_DIR}shop_doc.dat
TEMPLATE_POST,${UGII_CAM_POST_DIR}template_post.dat
USER_DEFINED_EVENTS,${UGII_CAM_USER_DEF_EVENT_DIR}ude.cdl,$
        {UGII_CAM_USER_DEF_EVENT_DIR}ude.tcl
TEMPLATE_CLSF,${UGII_CAM_TOOL_PATH_DIR}template_clsf.dat
LISTING_FORMAT,${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 Unigraphics Manufacturing Application.

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 initially only 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 4—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** directory structure to your home directory and modify the directories for read/write access. Instructions are presented for Windows and Unix separately.

Windows:

Step 1 Copying the Mach directory.

- ❑ On the Unigraphics Main Menu Bar, select **Help → Unigraphics NX Log File** to verify the **Mach** directory being used by searching for the environment variable **UGII_CAM_BASE_DIR**.

- ❑ Open a Windows Explorer window and locate the directory from the previous action item.

- ❑ Highlight the **Mach** directory, right click and select **Copy**.

- ❑ Highlight your **home directory**, right click and select **Paste.**

Step 2 Edit the *ugii_env.dat* file to redefine your Mach 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_BASE_DIR=${UGII_BASE_DIR}\mach
  and change the line to
  UGII_CAM_BASE_DIR=${HOME}\mach
  where **HOME** is the system environment variable designating where your home directory is located.

4

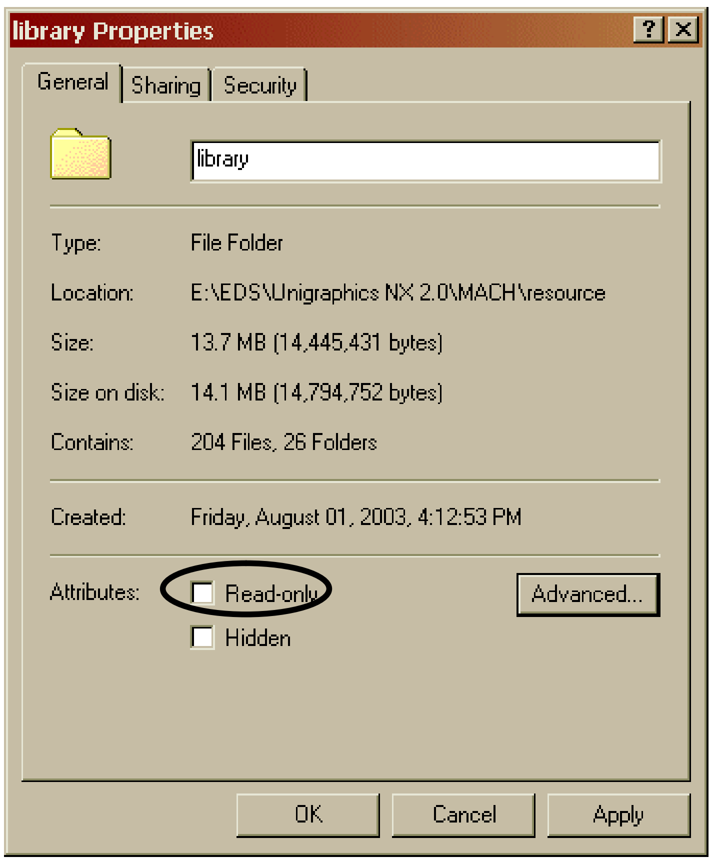
❑ Save the file and exit from Notepad.

Step 3 If necessary, change the Read-only protection on your just created local \Mach\resource\library subdirectory.

❑ With the Windows Explorer, locate your home **\mach\resource\library** directory.

❑ Highlight the directory and with **MB3** select **Properties**.

❑ Uncheck **Read-Only**.



Step 4 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 directory.

- ❑ On the Unigraphics Main Menu Bar, select **Help → Unigraphics NX Log File** to verify the **mach** directory being used by searching 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** 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 .

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* 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_DIR=${UGII_BASE_DIR}\mach
  and change the line to
  UGII_CAM_BASE_DIR=${HOME}\mach\.

- ❑ Save the changes and exit from the editor.

4

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**

Step 5 Restart Unigraphics.

- ❑ Exit Unigraphics and then restart Unigraphics.

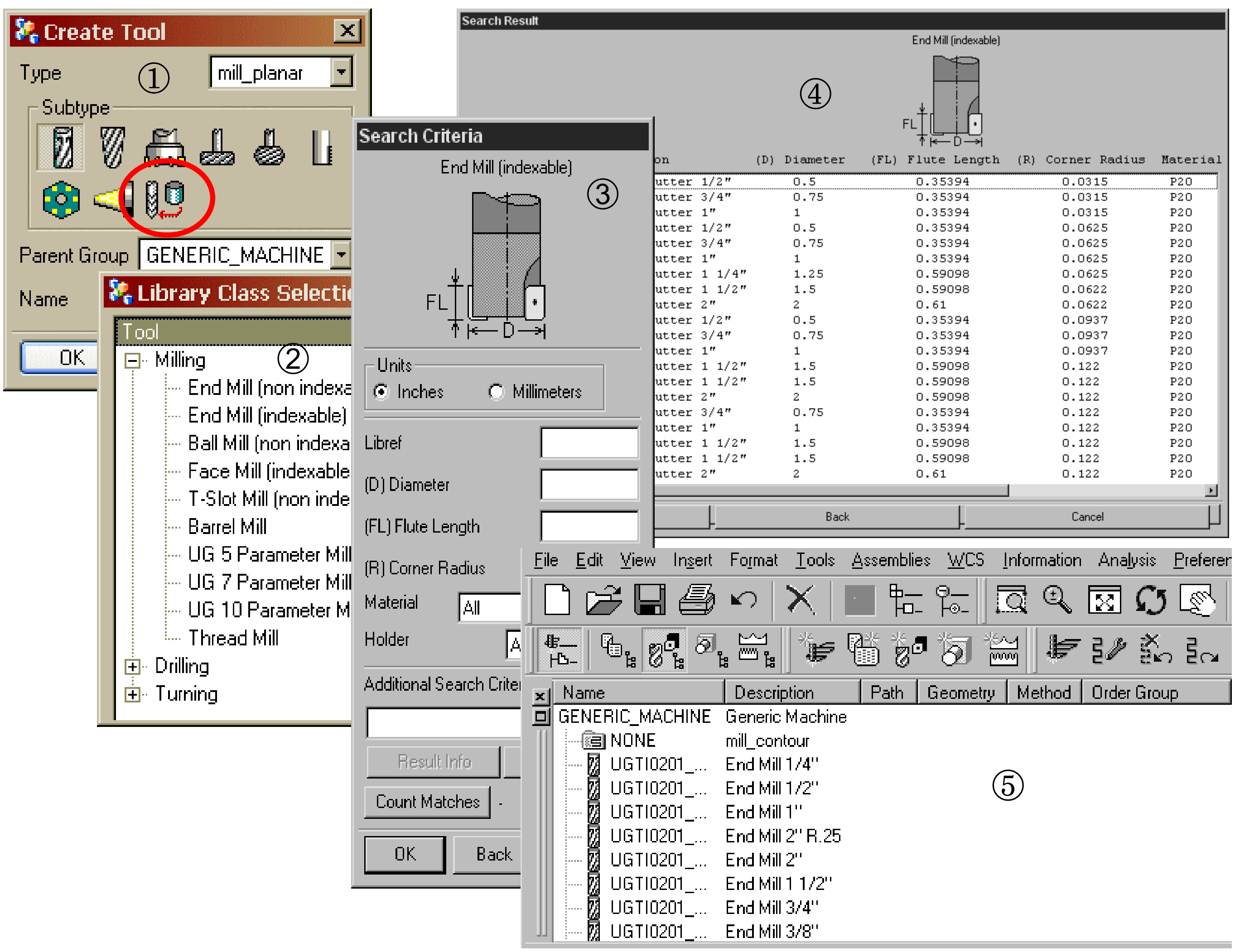
- ❑ On the Unigraphics Main Menu Bar, select **Help → UG Log File** to verify that **mach** directory (based from your home directory) is being used.

  This completes this activity.

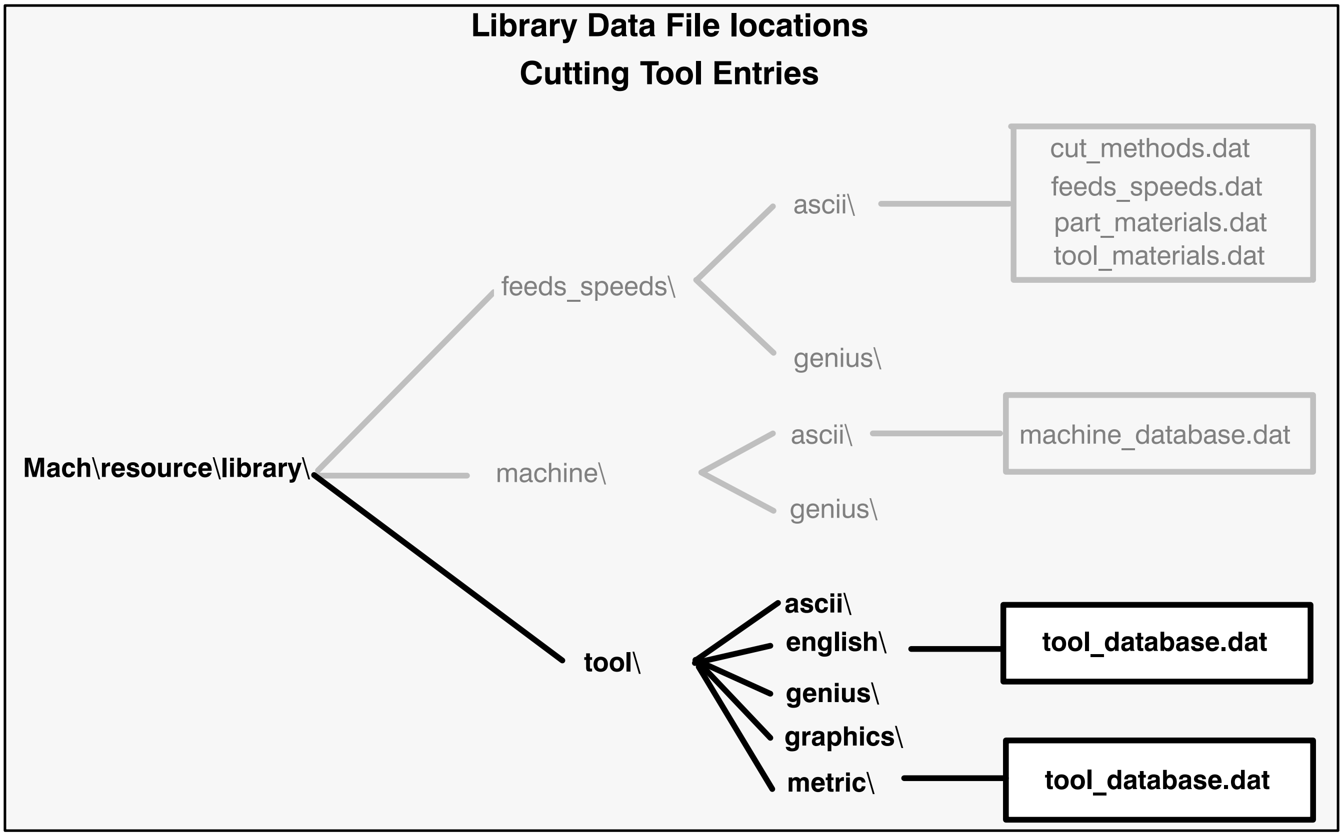
Cutting Tool Libraries

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 which contains all of 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.
- metric---contains the ASCII text database file which contains all of 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.

4

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 data file** from the dialog. This will export all tooling data to the file that you designate. You then will ***cut*** the tool data record(s) from the designated file and ***paste*** it into the **tool_database.dat** file.

The following is an example of the various attributes used with the **Tool Library** formats. All fields are documented within the file. Note that they are not the same for each tool.

Tool Library Format

```
# LIBRF   – Library Reference
# T       – Tool Type
# ST      – Tool SubType
# DESCR – Description
# MATREF – Reference to cutter material table
# MATDES – Cutter material description
# HLD     – Holding system (Type of Machine Adapter)
# HLDDES – Holding system description
# DIA     – Diameter
# FLEN   – Flute Length
# FN      – Number of Flutes
# HEI     – Height
# HLD    –Tool Holding System
# HLDDES  – Tool Holding System Description
# DIA     – Tool Diameter
# FN      – Tool Flutes Number
# HEI     – Tool Length (Height)
# ZOFF    – Tool Z Offset
# DROT  – Tool Direction (3=clockwise, 4=counterclockwise)
# FLEN    – Tool Flute Length (Cutting Depth)
# TAPA    – Tool Taper Angle
# COR1    – Tool Corner1 Radius
# HDIA    – Tool Holder Diameter
# HLEN    – Tool Holder Length
# HTAP    – Tool Holder Taper
# HOFF    – Tool Holder Offset

FORMAT LIBRF       T  ST DESCR          MATREF     MATDES      HLD HLDDES
DIA  FN HEI   ZOFF    DROT FLEN   TAPA COR1  HDIA    HLEN   HTAP HOFF
DATA   |ugti0201_011|02|01|End  Mill  1/4”     |TMC0_00006|HSS–Co5–TiN|300|Steep  Taper
SKG30|.25  |2 |.90551 |–2.87402|3   |.51181 |    |0.0  |.5315  |1.21654|31. |0.0
```

Note that the **MATREF** attribute is a reference to the cutter material located in the file **\MACH\resource\library\feeds_speeds\ascii\tool_materials.dat**.

Activity 4–2: Creating a tool with holder

In this activity you will create a tool that is mounted in a holder. This tool, as well as all other tools in the part file, will be inserted into the tool library in the activity which follows. These tools will become available whenever you perform a search for tools. Defining a tool with the holder, allows collision checking as well as better visualization when verifying a tool path.

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

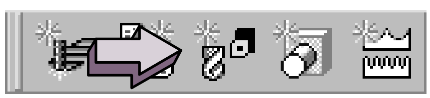
❑ Open **ama_lib_tools.prt.**

❑ **Save As ***_lib_tools .prt** where *** represents your initials.

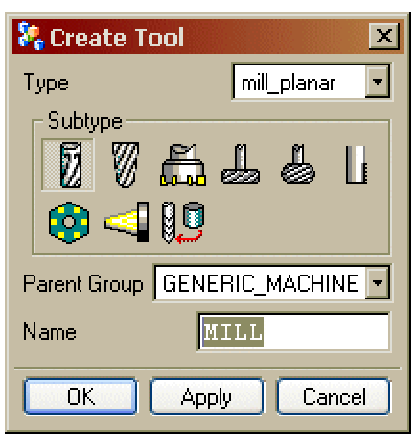
❑ Enter the **Manufacturing** Application.

You will create a 3/4 inch in diameter end mill with three flutes and no corner radius.

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



The **Create Tool** dialog is displayed.

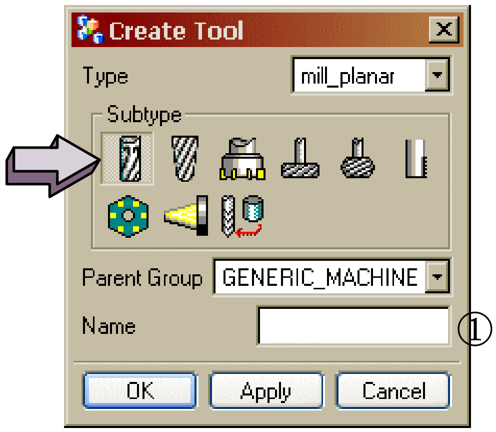


❑ If necessary, change the **Type** to **mill_planar**.

You will create a 3/4” end mill with 3 flutes and no corner radius.

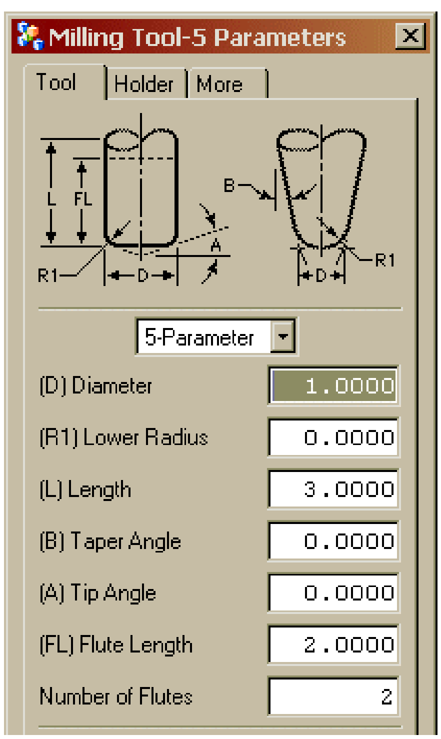
❑ Choose the **Mill** icon on the **Create Tool** dialog.

❑ Key in **em–.75–0_holder** ① for the **Name** of the tool.

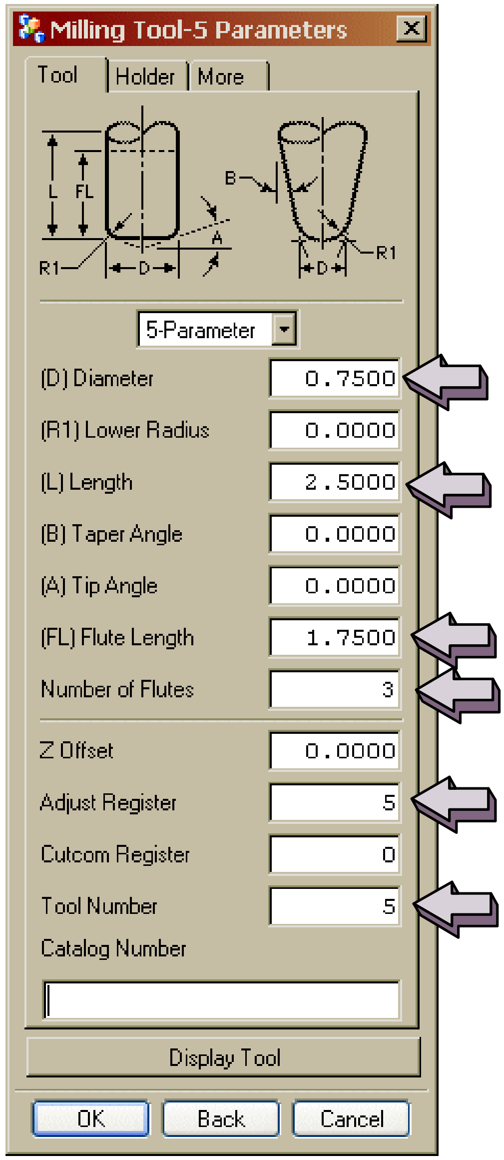


❑ Choose **OK**.

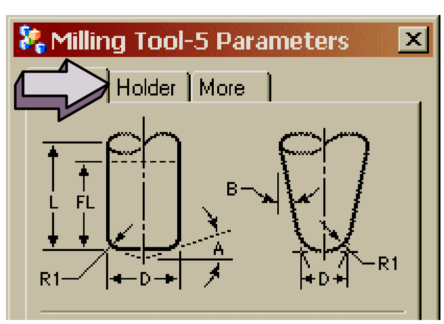
The **Milling Tool 5 Parameters** dialog is displayed.



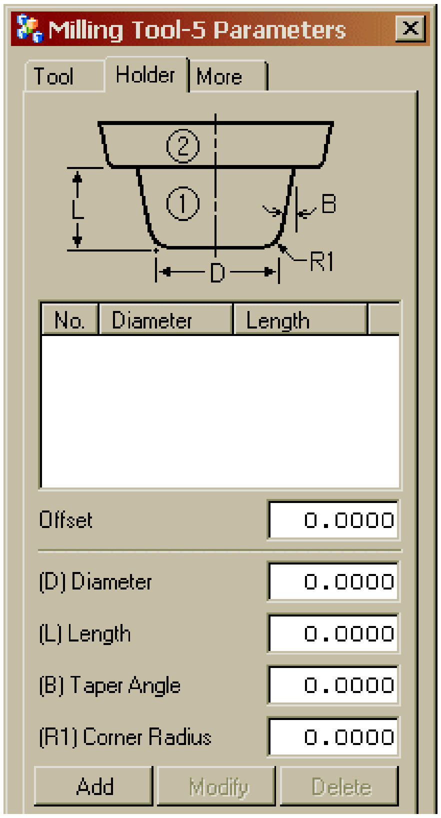
❑ Change the parameters as shown:



❑ Choose the **Holder** tab.

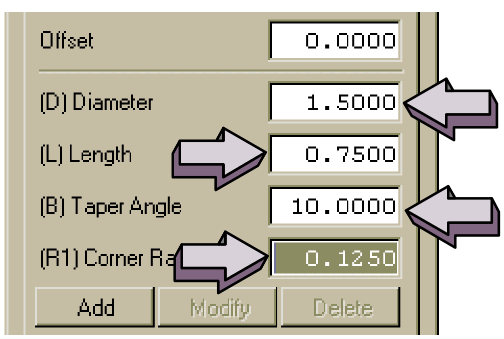


The **Tool Holder** dialog is displayed.

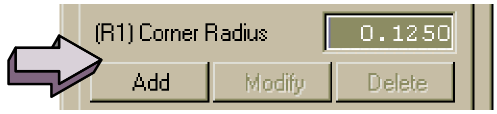


Tool holders are created from the end of the tool upwards. The first diameter created will be closest to the top of the tool.

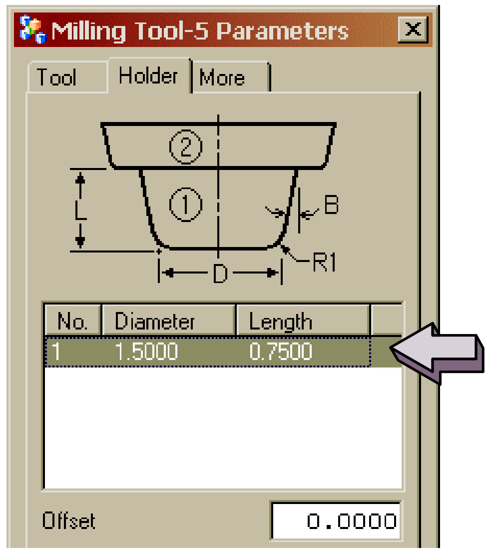
❑ Key in the values shown.



❑ Choose the **Add** button.

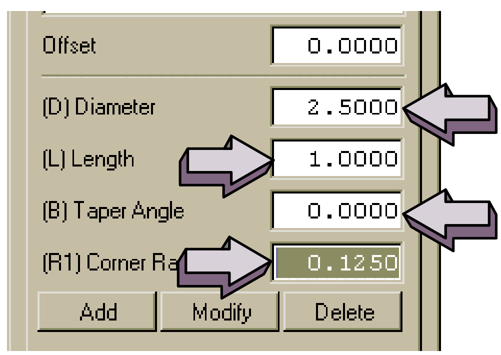


The bottom component of the tool holder, designated as (1) appears on the **Holder** list window.

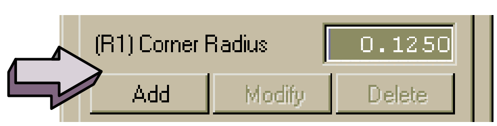


You will now create the 2nd component of the tool holder.

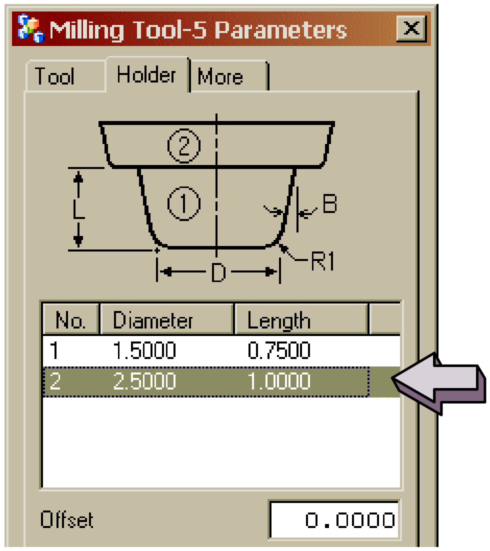
❑ Key in the values as shown.



❑ Choose the **Add** button.

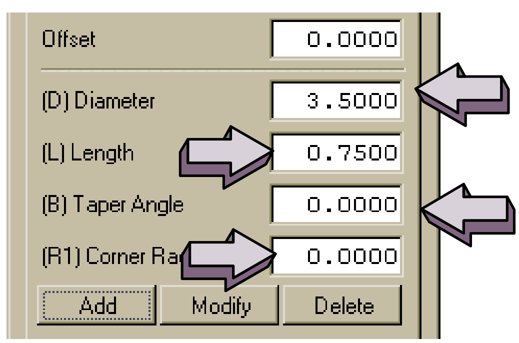


The 2nd component of the tool holder, designated as (2) appears on the **Holder** list window.

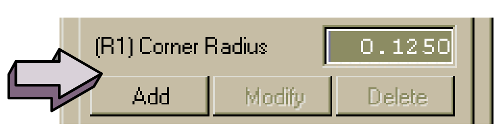


You will now create the third component of the tool holder. Note that this component is not represented on the tool holder dialog. However, remember that we define the tool holder components from the end of the tool upwards. In this case the third component is the upper flange of the tool holder which registers the holder with the spindle of the machine tool.

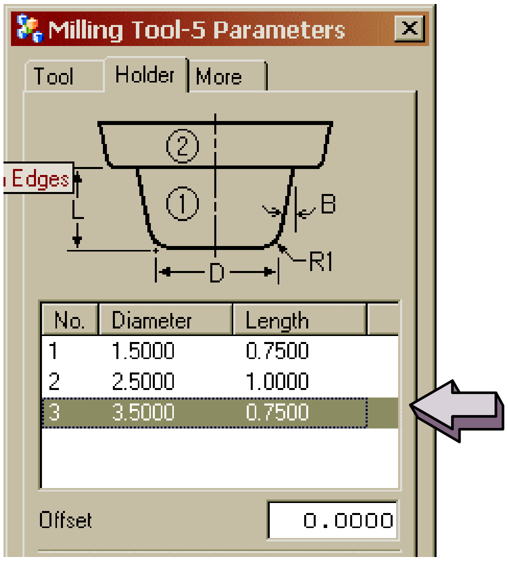
❑ Key in the values as shown.



❑ Choose the **Add** button.

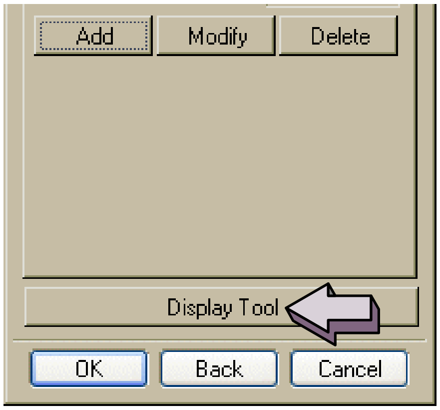


The 3rd component of the tool holder, designated as (3) appears on the **Holder** list window.

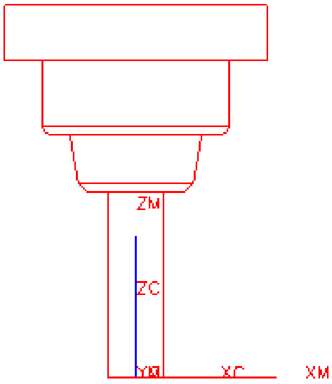


You have completed defining the tool with the holder. You will now display the tool.

❑ Choose the **Display Tool** button on the **Holder** dialog.



The tool with holder is displayed.



- ❑ On your own, create a face mill and drill with holders.
- ❑ **Save** and then close the part file.

  This completes the activity.

Existing tools

Cutting tools within part files can be extracted and inserted into tool libraries. This procedure is applicable with pre-V16 as well as NX part files. The next activity will take you through the process of extract existing tools from an existing part file and inserting them into the tool library.

Activity 4—3: Inserting Pre-existing Tools

The following activity will take you through the process of inserting tools from the part file from the previous activity and placing them into the tool library.

Step 1 Open the pre-existing part file containing tool entries and enter the Manufacturing Application.

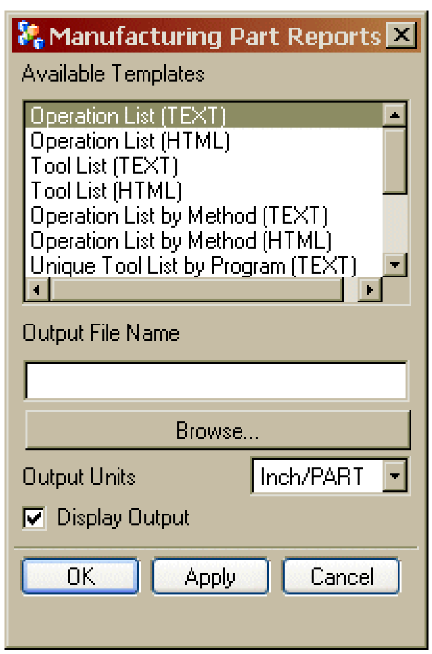
❑ If necessary, open the part file *****_lib_tools.prt** where *** represents your initials.

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

Step 2 Export the existing tool entries to an ASCII data file.

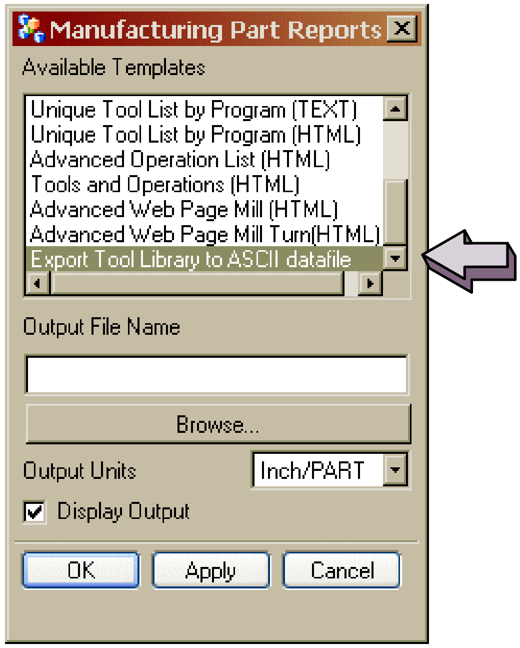
❑ Choose **Information** → **Shop Documentation** (or select the Shop Documentation icon).

The **Manufacturing Part Reports** dialog is displayed.

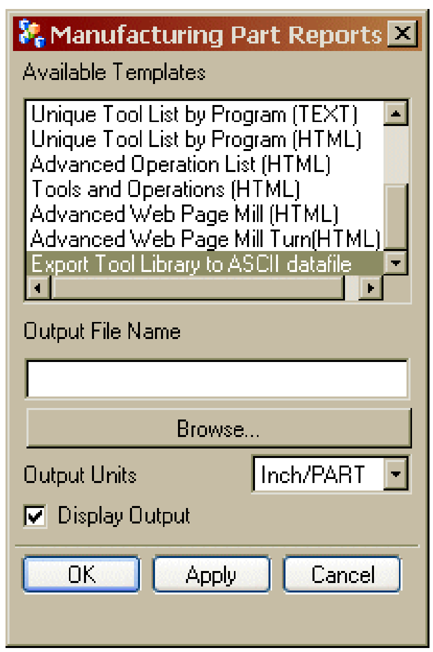


4

❑ Select **Export Tool Library to ASCII data file**.



❑ Choose **OK** to accept the default name.



Two files are created, one with a ".html" and the other with a ".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.

```
==========================================================================
Information listing created by :
Date                           :
Current work part              :  ***_lib_tools.prt
Node name                      :
==========================================================================

#=========================================================================
#
#   ASCII Database File   : ***_lib_tools.dat
#   Creation date         :
#   Unit                  : English
#   Created from Partfile : ***_lib_tools.prt
#
#=========================================================================

Type=02/90 #1 Name='EM-1.250-12'          Type='Milling Tool-5 Parameters'
Type=02/90 #2 Name='EM-.750-06'           Type='Milling Tool-5 Parameters'
Type=02/90 #3 Name='EM-.500-06'           Type='Milling Tool-5 Parameters'
Type=02/90 #4 Name='EM-.375-03'           Type='Milling Tool-5 Parameters'
Type=02/90 #5 Name='EM-1.00-50'           Type='Milling Tool-5 Parameters'
Type=02/90 #6 Name='EM-1.00-06'           Type='Milling Tool-5 Parameters'
Type=02/90 #7 Name='EM-.75-0_HOLDER'      Type='Milling Tool-5 Parameters'
Type=02/92 #1 Name='SP1'                  Type='Milling Tool-10 Parameters'
Type=03/90 #1 Name='DR-.484'              Type='Drilling Tool'
Type=03/90 #2 Name='SPOT-.500-90'         Type='Drilling Tool'
Type=03/90 #3 Name='RM-.500'              Type='Drilling Tool'

------------------------------------------------------------------------------
------------------------------------------------------------------------------


Output directory    =
Ascii database file = ***_lib_tools.dat
Unigraphics tools   = 11
```

4

Step 3 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, *****_lib_tools.dat**, created from **Step 3,** and scroll to the area that begins with **FORMAT LIBRF**.

```
#   HEI      - Tool Length (Height)
#   ZOFF     - Tool Z Offset
#   DROT     - Tool Direction (3=clockwise, 4=counter
#   FLEN     - Tool Flute Length (Cutting Depth)
#   TIPA     - Tool Tip Angle
#   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 UGT UGST DESCR
# ----+---------------+--+--+---+----+------------
DATA  |EM-1.250-1     |02|90|1  |1   |Milling Tool-
DATA  |EM-.750-06     |02|90|1  |1   |Milling Tool-
DATA  |EM-.500-06     |02|90|1  |1   |Milling Tool-
DATA  |EM-.375-03     |02|90|1  |1   |Milling Tool-
DATA  |EM-1.00-50     |02|90|1  |1   |Milling Tool-
DATA  |EM-1.00-06     |02|90|1  |1   |Milling Tool-
DATA  |EM-.75-0_HOLDER|02|90|1  |1   |Milling Tool-

#
# 10 Parameter Mill
#
#   LIBRF    - Tool Library Reference
#   T        - Tool Type
#   ST       - Tool SubType
#   UGT      - UG Tool Type
#   UGST     - UG Tool SubType
#   DESCR    - Description
#   MATREF   - Tool Material Code
#   MATDES   - Tool Material Description
#   TLNUM    - Tool Number
#   ADJREG   - Tool Length Adjust Register
#   CUTCOMREG - Tool CUTCOM Register
#   HLD      - Tool Holding System
#   HLDDES   - Tool Holding System Description
#   DIA      - Tool Diameter
```

❑ Select the lines beginning with **FORMAT LIBRF** and ending with **DATA |EM–.75–0_Holder** (hold down **MB1** and drag the mouse through the lines). Include any extra tools that you created in addition to **EM–.75–0_Holder** in the previous activity.

❑ 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.

This completes the activity.

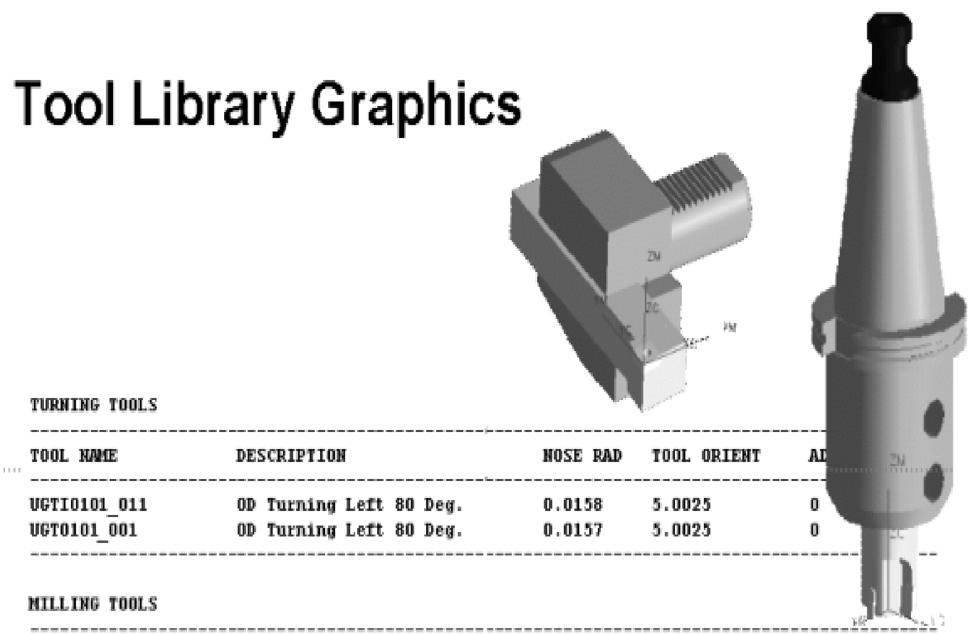
4

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 used in the **Toolpath Visualization** dialog, under Display Options, Tool option and also in the Integration, Simulation and Verification module. 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. The key to creating a graphical tool assembly is to have an assembly of the desired tool and a library entry in the tool library with **exactly the same name** as the tool assembly.



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.

Activity 4—4: Creating a Graphical Tool Assembly

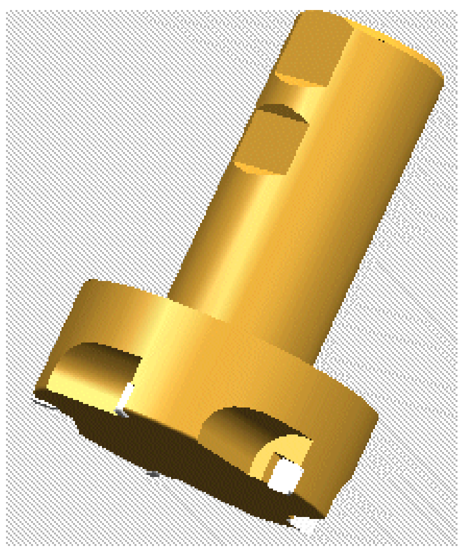
This lesson will guide you through the creation of a graphical tool assembly. You will open an existing assembly, modify two components and then save the new assembly.

Since the associated modeling of the various components of the graphical tool assembly can be very time consuming, the components necessary to complete this activity have already been modeled for you.

In a future lesson you will use the graphical tools in the simulation of a tool path.

Step 1 Open and examine the part file representing the new tool to be added to the tool assembly.

❑ Open the part file **ama_fm_4.0_5_flutes.prt**.



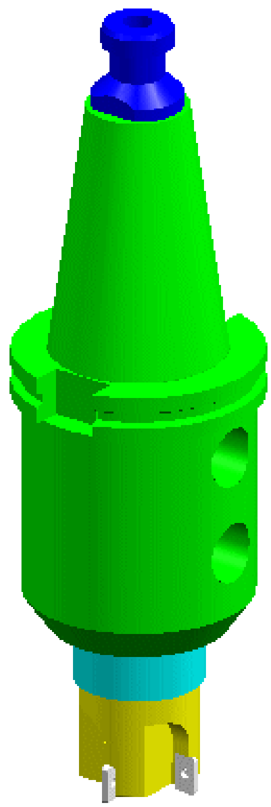
Examine the part carefully. This part represents a four inch face mill with 5 carbide inserts. You will substitute this tool in an existing assembly.

4

Step 2 Open an existing tool assembly, rename and save in the graphics library.

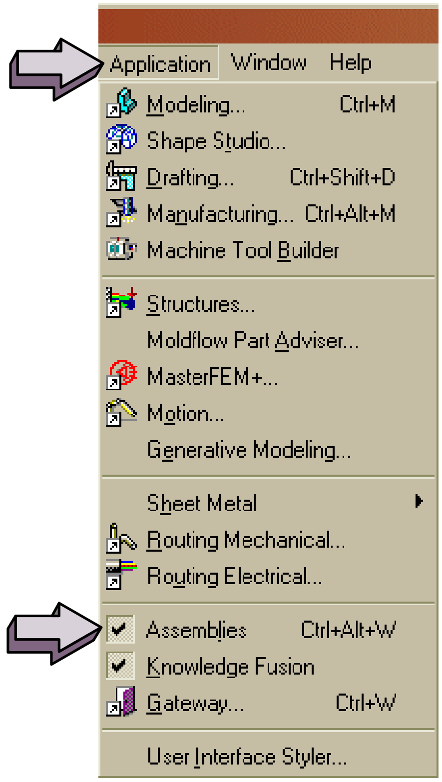
You will open an existing tool assembly that will be used to hold the face mill from the previous step. You will save this assembly with a new part name.

❑ Open the existing tool assembly,**ugti0202_019.prt** located in your home **MACHN\resource\library\tool\graphics** directory.

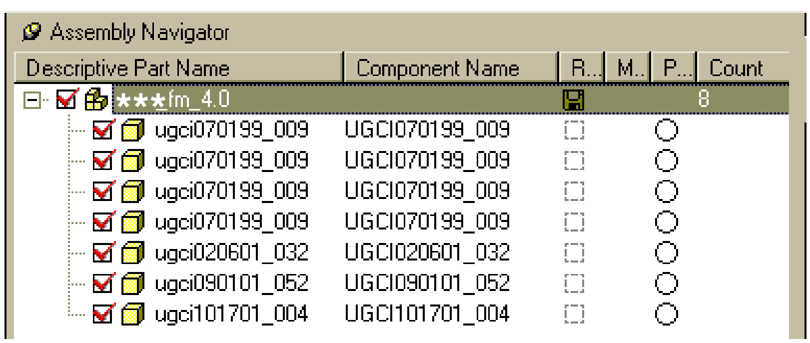


❑ Use the **Save** → **As** option under **File** on the menu bar and rename the part to *****_fm_4.0.prt** where ******* represents your initials.

❑ If necessary, turn the **Assemblies** application on.

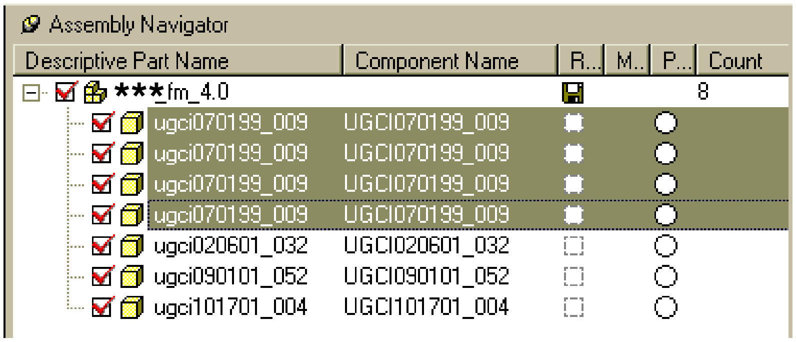


❑ Select the **Assembly Navigator** from the **Resource** bar.



You will now delete the inserts from the assembly.

❑ Highlight the four **ugci070199_009** components, which represent the inserts, using **MB3**, **Delete** them from the assembly.

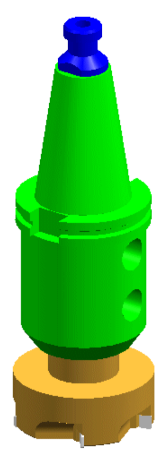


You will now replace the existing end mill with the 4.00” face mill.

❑ Highlight the **ugci020601_032** component, which represents the existing inserted end mill, using **MB3** → **Close** → **Part**.

❑ Highlight the **ugci020601_032** component, using **MB3** → **Open** → **Component As...** and choose the **ama_fm_4.0_5_flutes** part file from the list.

The assembly now contains the four inch face mill.



❑ **Save** and **Close** the assembly.

❑ Using a text editor, open the file **tool_database.dat** from your home directory **MACH\resource\library\tool\english**.

❏ Scroll down to the **Face Mill** section of the file.

```
#
# Face Mill (indexable)
#
#  LIBRF     - Tool Library Reference
#  T         - Tool Type
#  ST        - Tool SubType
#  UGT       - UG Tool Type
#  UGST      - UG Tool SubType
#  DESCR     - Description
#  MATREF    - Tool Material Code
#  MATDES    - Tool Material Description
#  TLNUM     - Tool Number
#  ADJREG    - Tool Length Adjust Register
#  CUTCOMREG - Tool CUTCOM Register
#  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)
#  D2        - Tool Diameter 2
#  COR1      - Tool Corner1 Radius
#  HDIA      - Tool Holder Diameter
#  HLEN      - Tool Holder Length
#  HTAP      - Tool Holder Taper
#  HOFF      - Tool Holder Offset

FORMAT LIBRF         T  ST UGT UGST DESCR         MATREF     MATDES TLNUM
# ----+-------------+--+--+---+----+-------------+----------+------+-----+
DATA  |ugti0212_005|02|12|01 |01  |Face Mill 2"|TMC0_00003|P20   |     |
DATA  |ugti0212_006|02|12|01 |01  |Face Mill 3"|TMC0_00003|P20   |     |
```

❏ Add the following line to the **DATA** area of the **Face Mill** section of the file:

DATA |*_fm_4.0 |02|12|01 |01 |Face Mill 4”|TMC0_00003|P20 | | |**
|320|Steep Taper SKG45|4.|5|1.75
|–3.34646|3|.375|3.5|.0315|2.3622|.62598|29.|0.0

Verify that the name matches **exactly** the name you gave to the assembly.

This tool will now be available for graphical display when simulating tool paths.

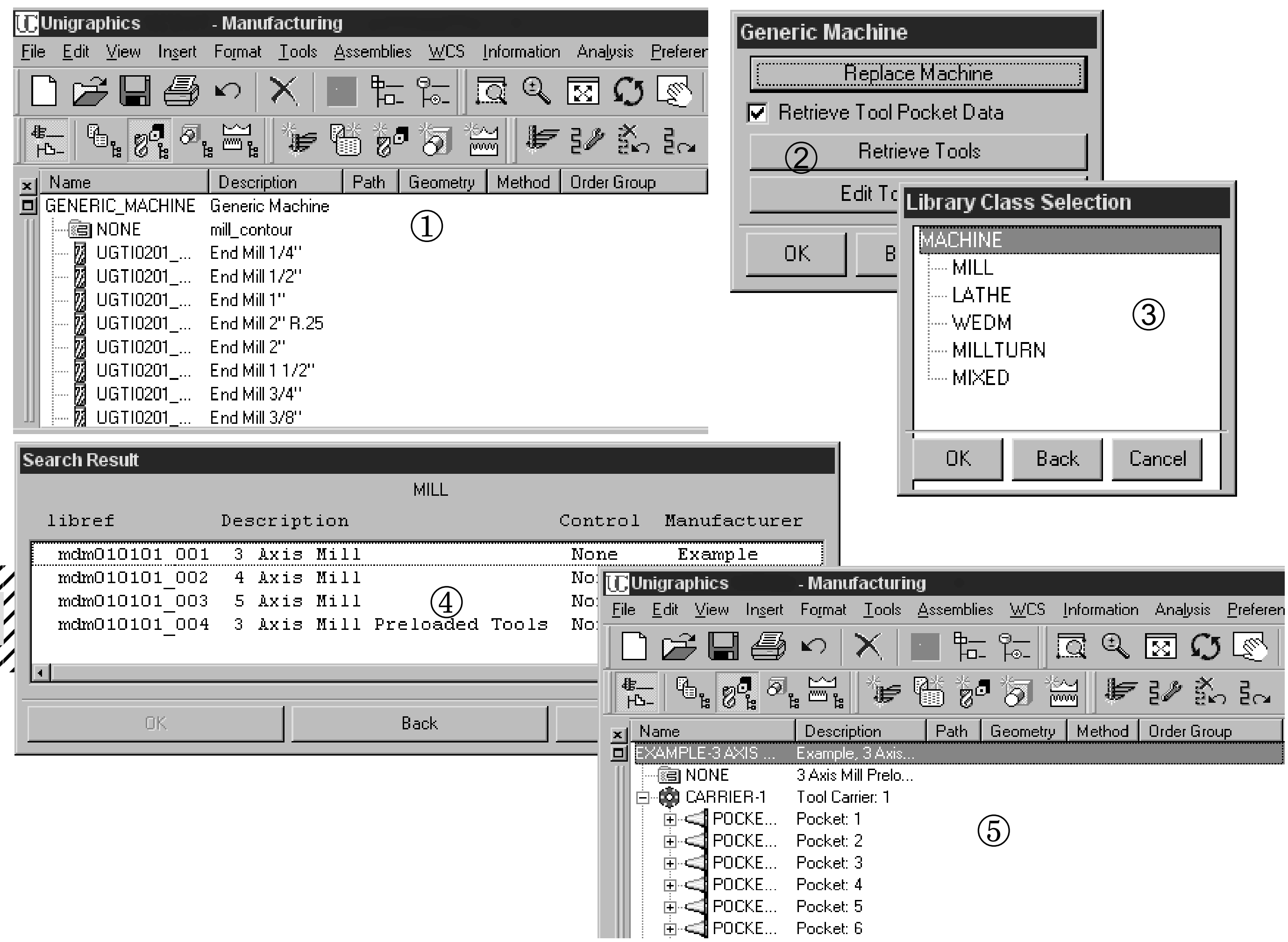
This completes the activity.

4

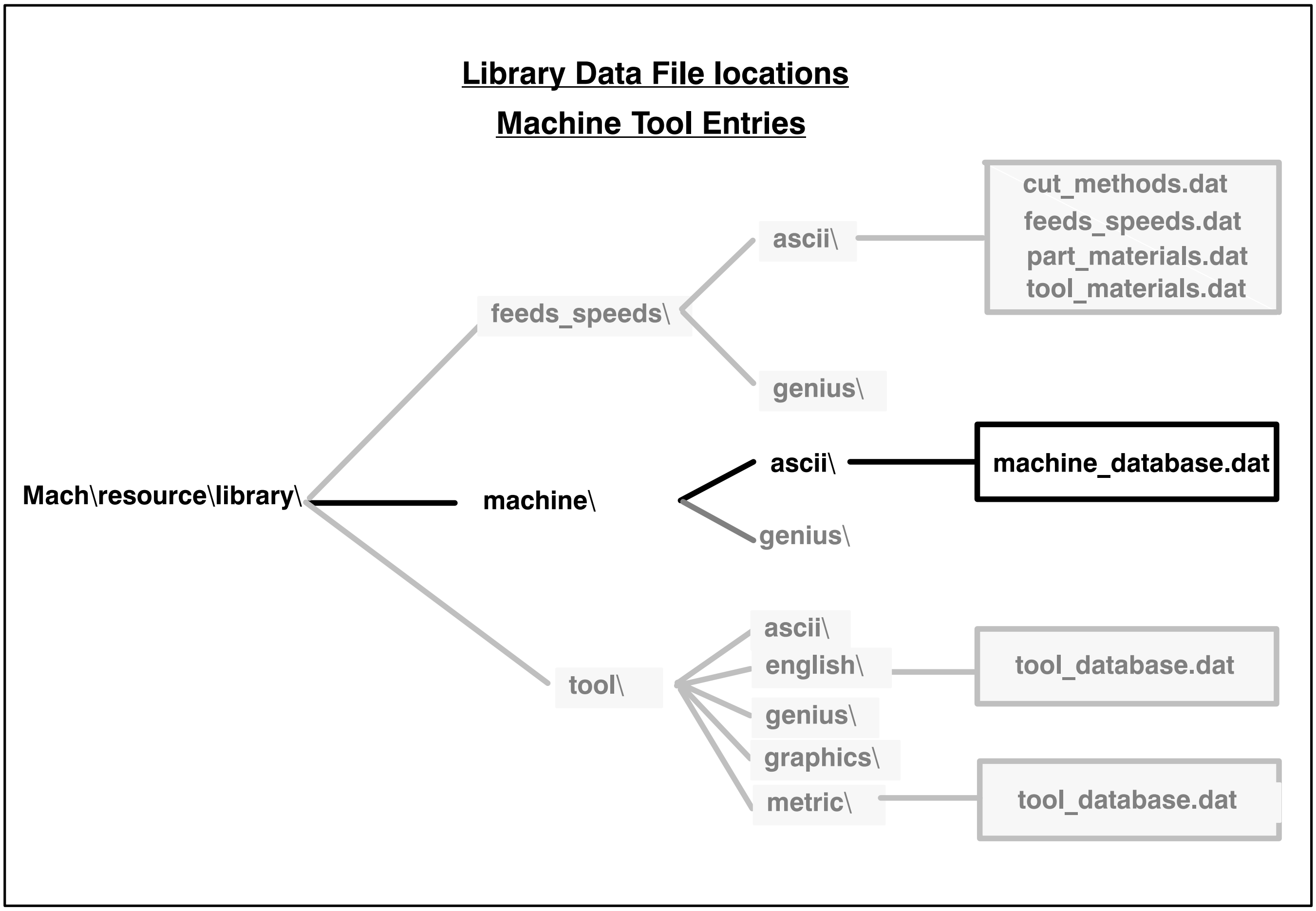
Machine Tool Libraries

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** dialog. Selection of the **Replace Machine** button (②) from this dialog, 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
```

4

Activity 4—5: Machine Tool Libraries

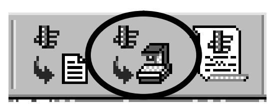
In this activity you will become familiar with the procedure to access Machine Tool data from CAM Libraries. You will see how this library is used in conjunction with the Post Process dialog by replacing the Available Machines with a 5-axis postprocessor.

Step 1 Open the part file.

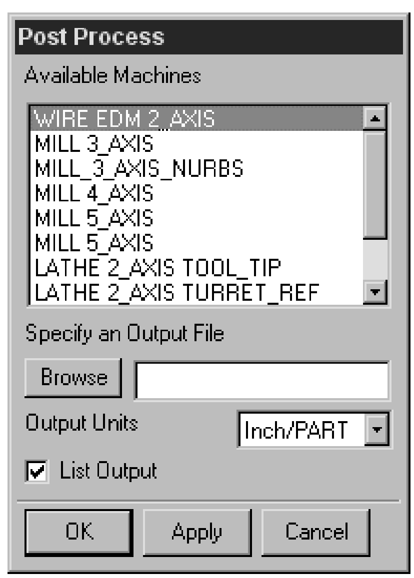
- ❑ Open the part file, **ama_lib_function.prt.**
- ❑ If necessary, enter the **Manufacturing Application.**

Step 2 Review available machines in the Post Process dialog.

- ❑ Select the Post Process Icon**.**



The Post Process dialog is displayed.



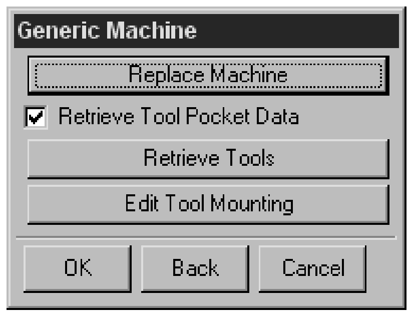
- ❑ Choose **Cancel** or **OK.**

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.

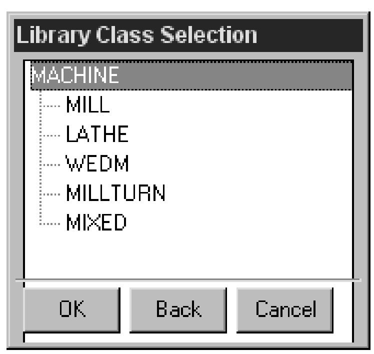
❑ Highlight the **GENERIC_MACHINE**, then using **MB3,** select **Edit**.

The **Generic Machine** dialog is displayed.



❑ Select the **Replace Machine** button.

The **Library Class Selection Menu** is displayed.

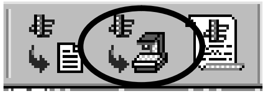


❑ Highlight **MILL**, then select **OK**.

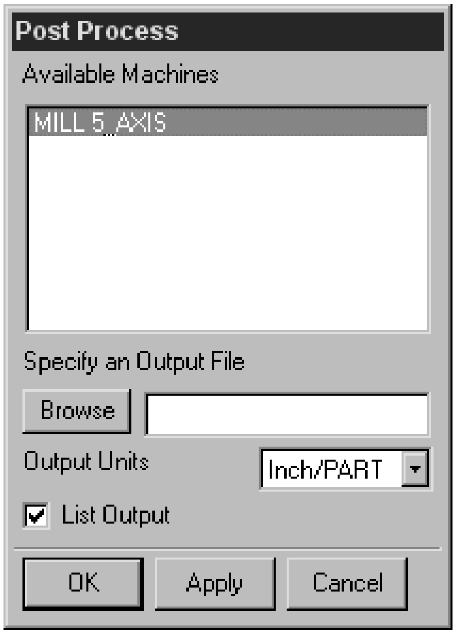
The **Search Result** listing is displayed.

❑ Highlight **mdm010101_003 5-Axis Mill**, then select **OK** until all dialogs are dismissed.

❑ 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.

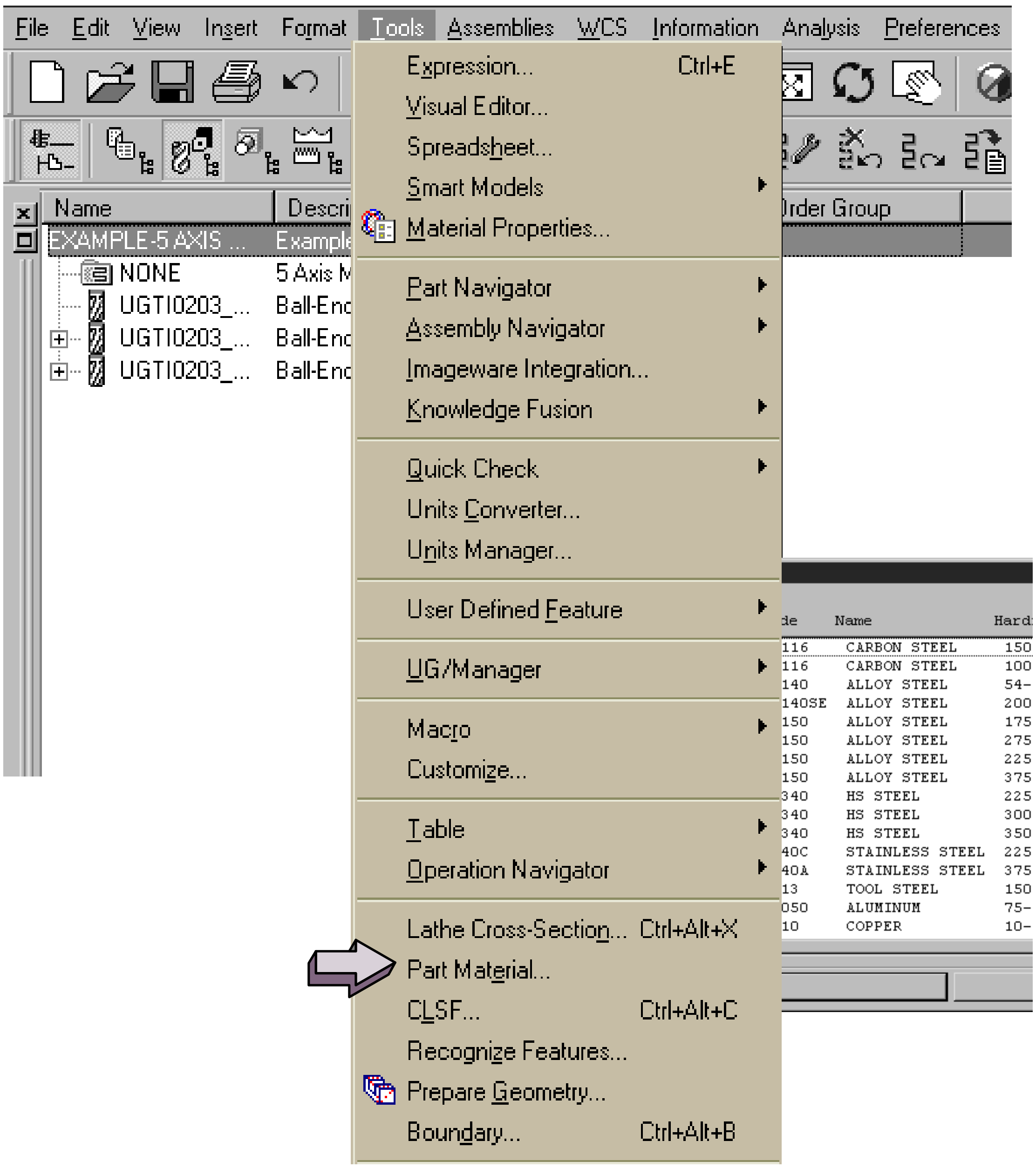
❑ Choose the **Cancel** button.

You are finished with this activity and will be using this part file in the next activity.

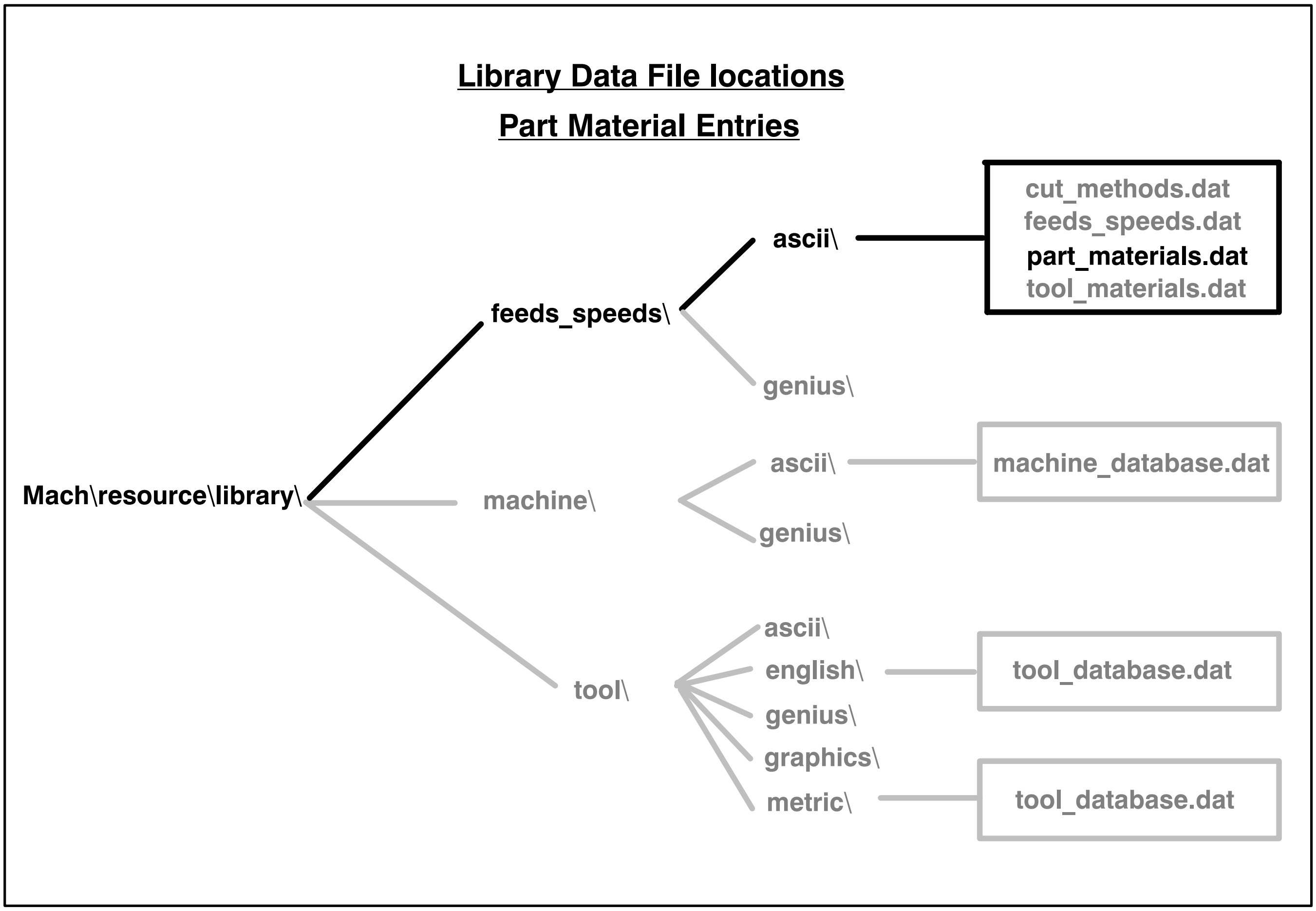
Part Material Libraries

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.



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
```

4

Activity 4—6: Part Materials Libraries

In this activity you will become familiar with accessing Part Material from the Library. You will modify the part material data file to add a new material type.

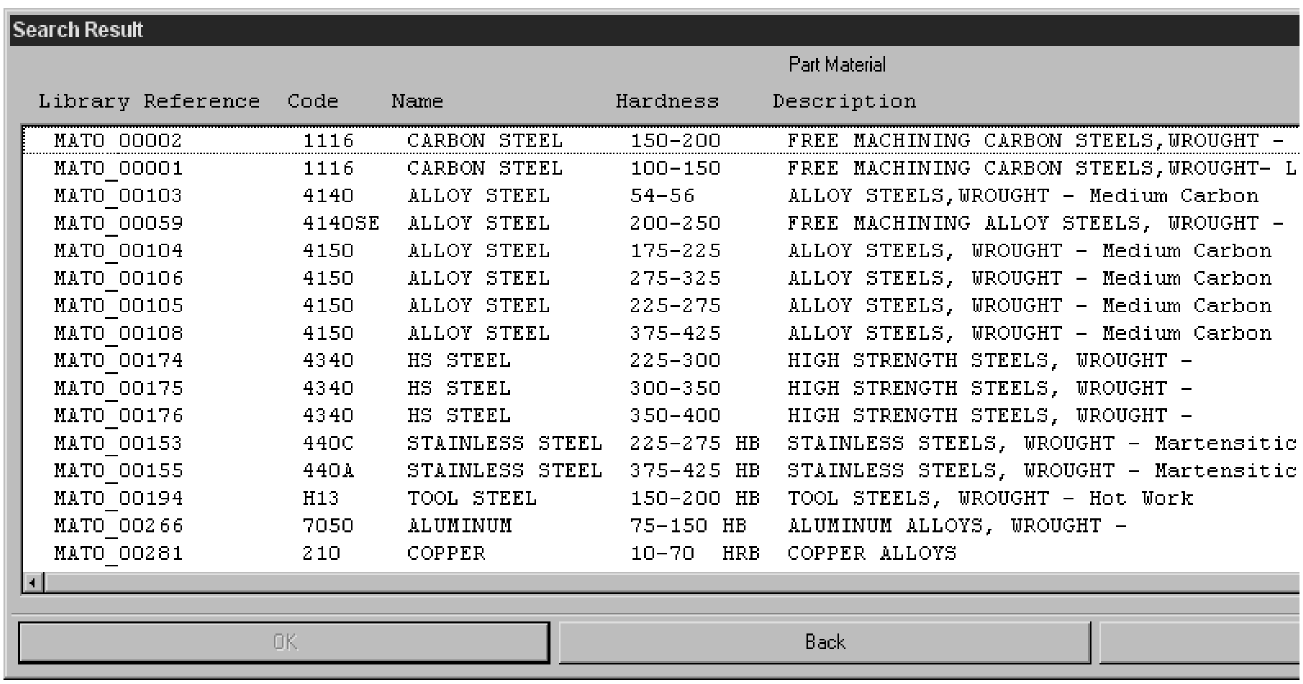
Step 1 List all part materials which are available for selection.

❑ Continue with the opened part, **ama_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

❑ Choose **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.**

Step 3 Adding a new part material to the part material database.

You will now add the *faux pax* material, **UNOBTAINIUM**, to the part material database.

❑ Using your text editor, open the **part_materials.da**t file located in your **MACH\resource\library\feeds_speeds\ascii** home directory.

part_materials.dat - Notepad

File Edit Format Help

```
# PART_MATERIALS.DAT
###########################################################################
#
#   PURPOSE:
#
#      This is the database file used for defining Part Material.
#
#   REVISIONS:
#     Sl.No.  Date          Name          Reason
#     01      06/15/99     Subhash       Initial
#     02      07/31/99     Subhash       Include Format statement
#     03      10/29/99     Murthy        Changes to data and file description
###########################################################################
###########################################################################
##
##  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 Res
DATA|1116|CARBON STEEL|FREE MACHINING CARBON STEELS,WROUGHT- Low Carbon Resu
DATA|4140|ALLOY STEEL|ALLOY STEELS,WROUGHT - Medium Carbon|54-56|MAT0_00103
DATA|4140SE|ALLOY STEEL|FREE MACHINING ALLOY STEELS, WROUGHT - Medium Carbor
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT - Medium Carbon|175-225|MAT0_001
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT - Medium Carbon|275-325|MAT0_001
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT - Medium Carbon|225-275|MAT0_001
DATA|4150|ALLOY STEEL|ALLOY STEELS, WROUGHT - Medium Carbon|375-425|MAT0_001
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 HE
DATA|440A|STAINLESS STEEL|STAINLESS STEELS, WROUGHT - Martensitic|375-425 HE
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
```

❑ After the last DATA statement, add the following line (it is

somewhat easier to copy and paste an existing line, making any modifications necessary to that line):
DATA|7777|UNOBTAINIUM|HARD TO GET|100–170 HRB|MAT0_07777

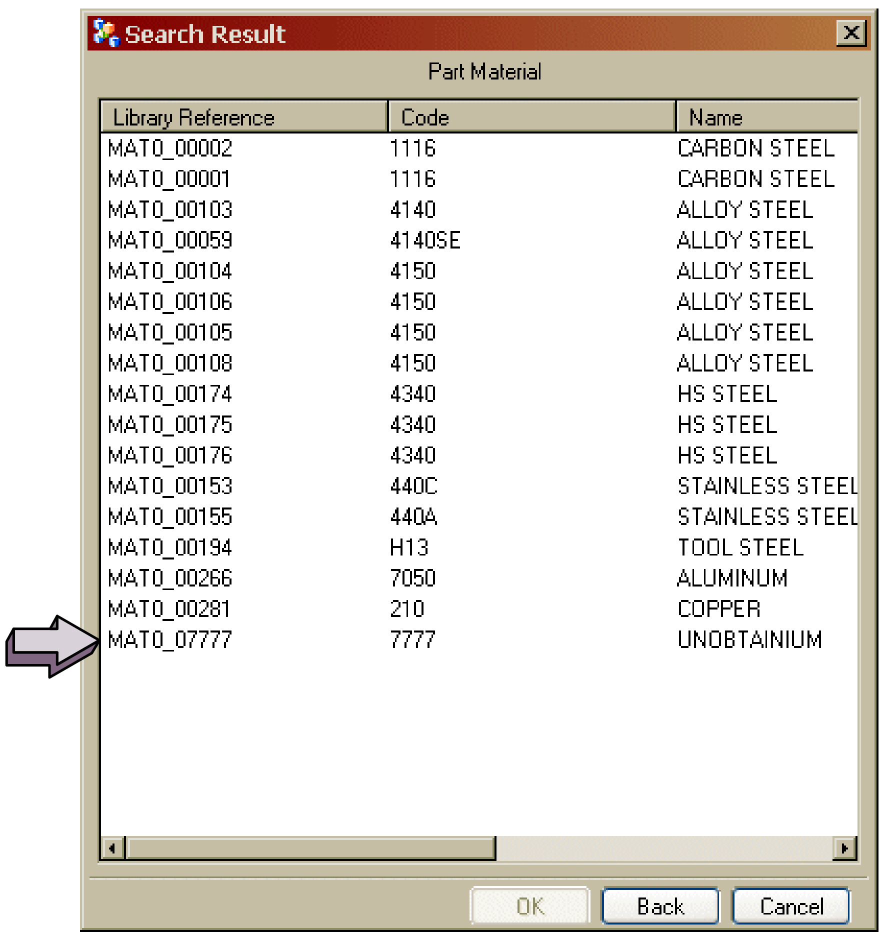
❑ On a separate sheet of paper, record the material code (MAT0_07777). You will need to reference this later.

❑ Save the file and return to Unigraphics.

❑ If necessary, reset the Configuration file.

❑ Choose **Tools** → **Part Material** from the Main menu bar.

The new part material, MAT0_07777 is now listed in the dialog.

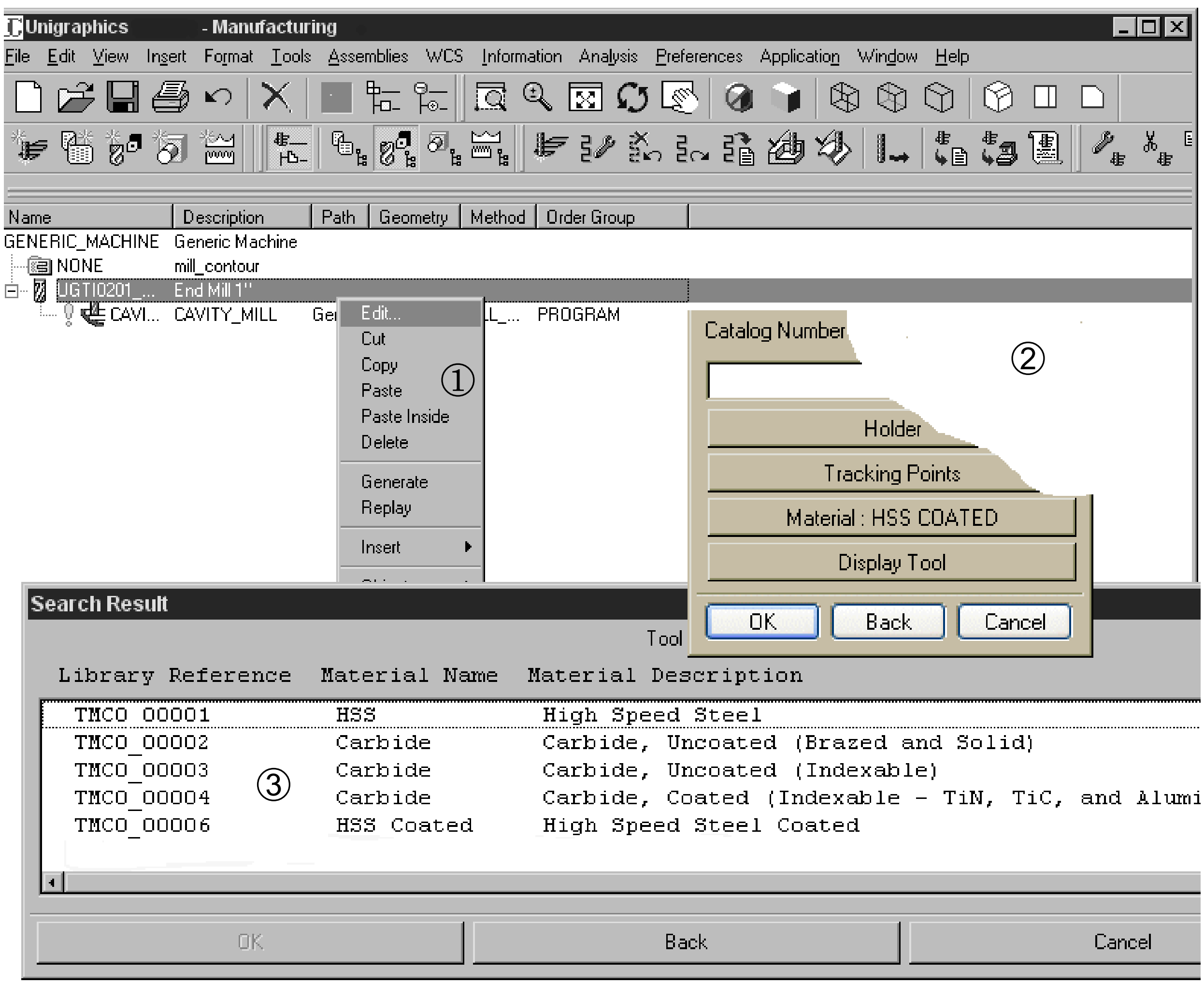


❑ **Cancel** the Search Result list window.

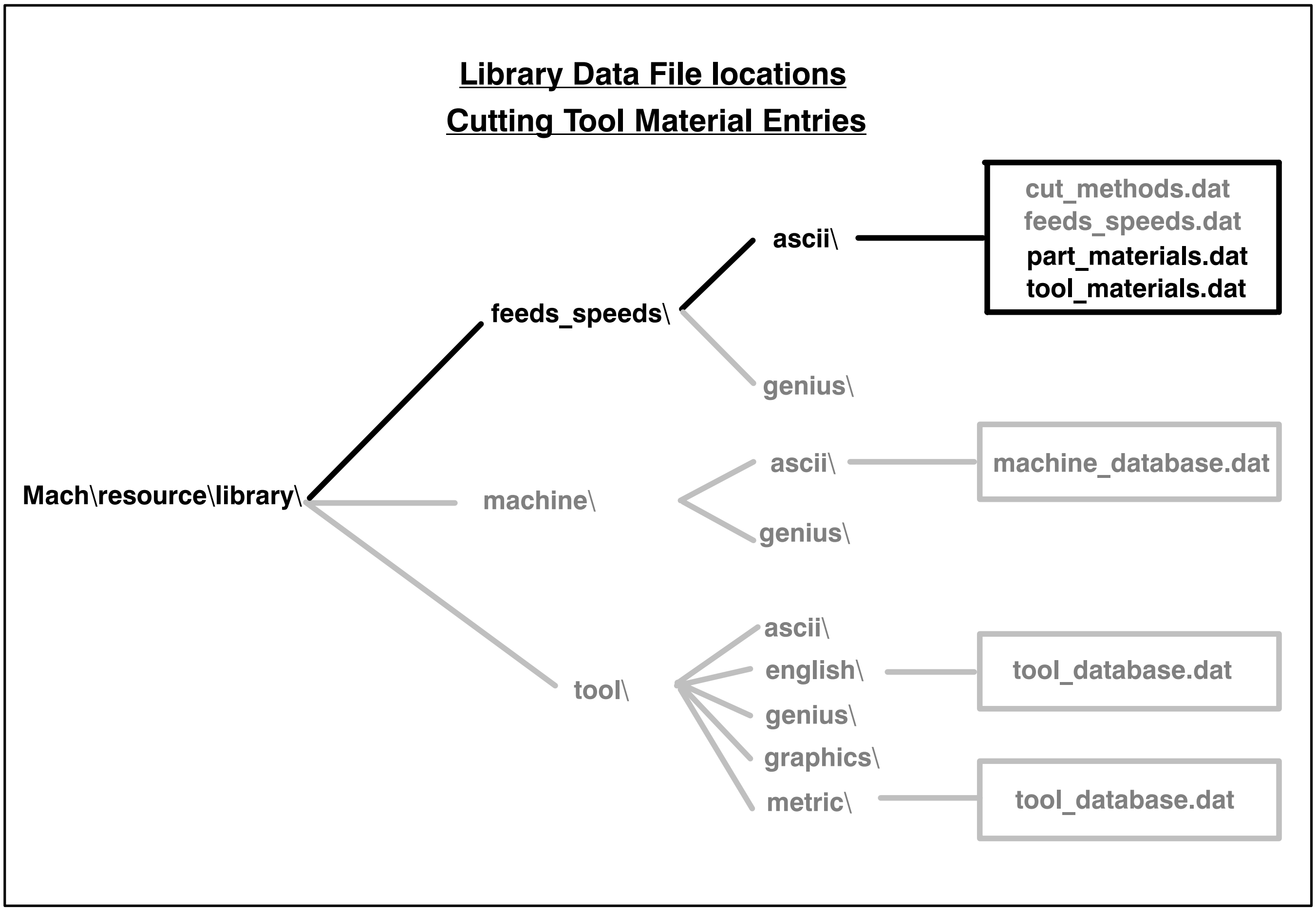
Cutting Tool Material Libraries

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 Material** 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
```

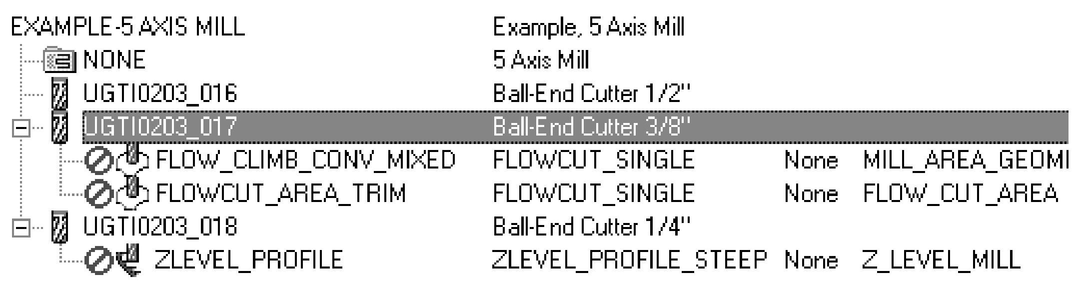
4

Activity 4—7: Cutting Tool Materials Libraries

In this activity you will become familiar with the Cutting Tool Materials data. You will then add a new Cutting Tool Material to the library. Cutting Tool Materials are used in the calculation of feeds and speeds.

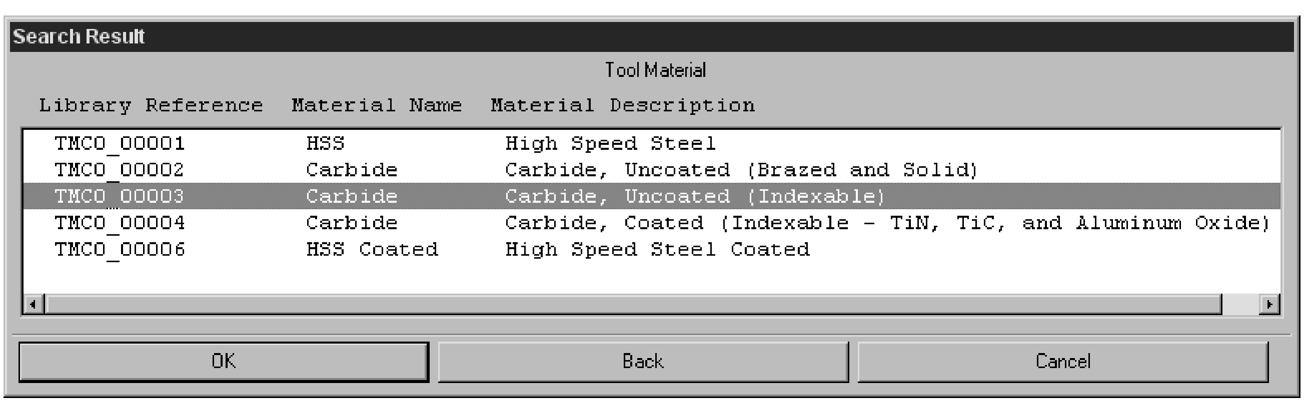
Step 1 Accessing the Cutting Tool Material library.

- ❑ 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 Search Result list is displayed.

- ❑ Select the **TMC0_00003 Carbide, Uncoated (Indexable)** as the material type.

- ❑ Choose **OK** until all dialogs are dismissed.

Step 2 Adding a new cutter material to the cutting tool material database.

You will now add the cutter material, **PCD Diamond Coated**, to the cutter material database.

- ❑ Using your text editor, open the **tool_materials.dat** file located in your **MACH\resource\library\feeds_speeds\ascii** home directory.

tool_materials.dat - Notepad

File Edit Format Help

```
# TOOL_MATERIALS.DAT
##########################################################################
#
#   PURPOSE:
#
#      This is the database file used for defining Tool Material.
#
#   REVISIONS:
#     Sl.No.   Date          Name          Reason
#     01      06/15/99     Subhash      Initial
#     02      07/31/99     Subhash      Include Format statement
#     03      10/29/99     Murthy       Changes to data and file description
#
##########################################################################
##########################################################################
##
##  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
```

- ❑ After the last DATA statement, add the following line (it is somewhat easier to copy and paste an existing line, making any modifications necessary to that line):
DATA|TMC0_0009|Diamond|PCD Diamond Coated

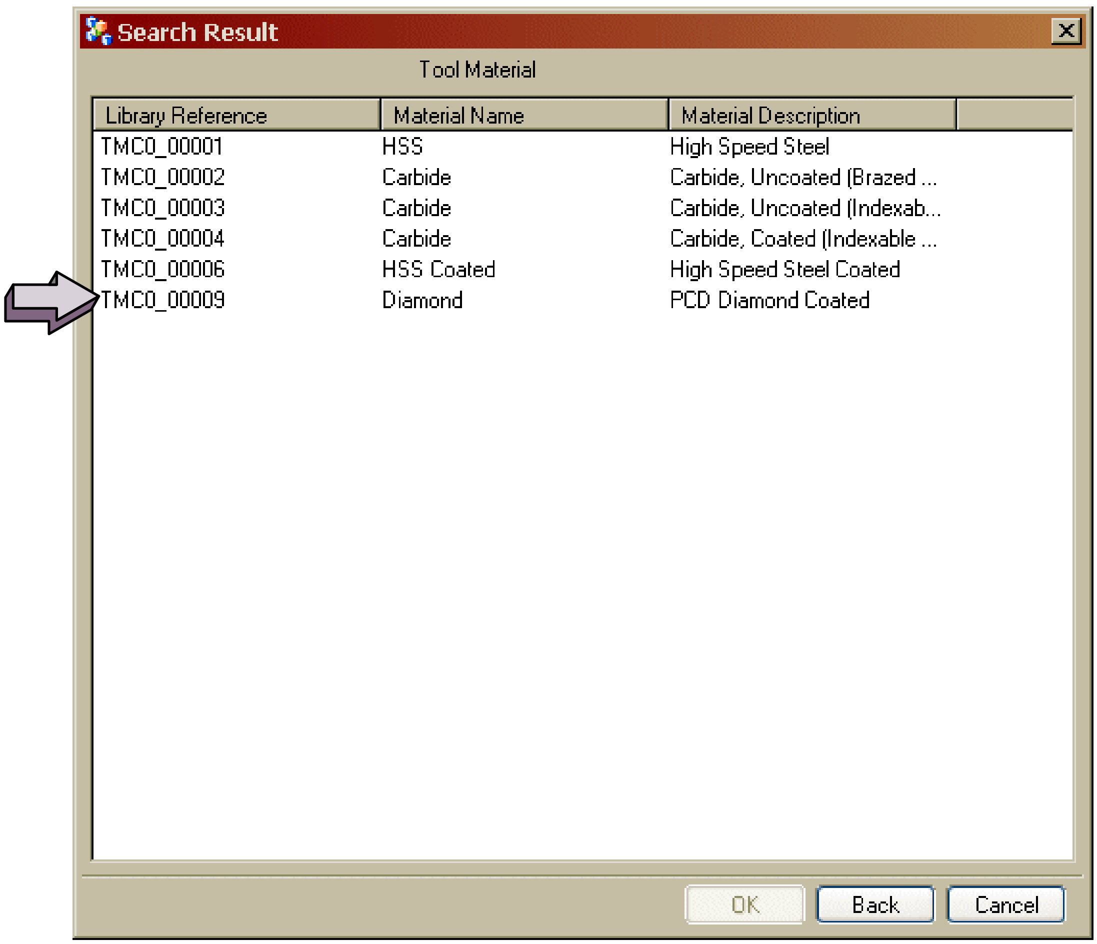
- ❑ On a separate sheet of paper, record the tool material code (TMC0_00009). You will need to reference this later.

❑ Save the file and return to Unigraphics.

❑ If necessary, reset the Configuration file.

❑ Open an existing tool and select the tool material button.

The new cutting tool material, **TMC0_00009** is now listed in the dialog.



❑ **Cancel** the **Search Result** list window.

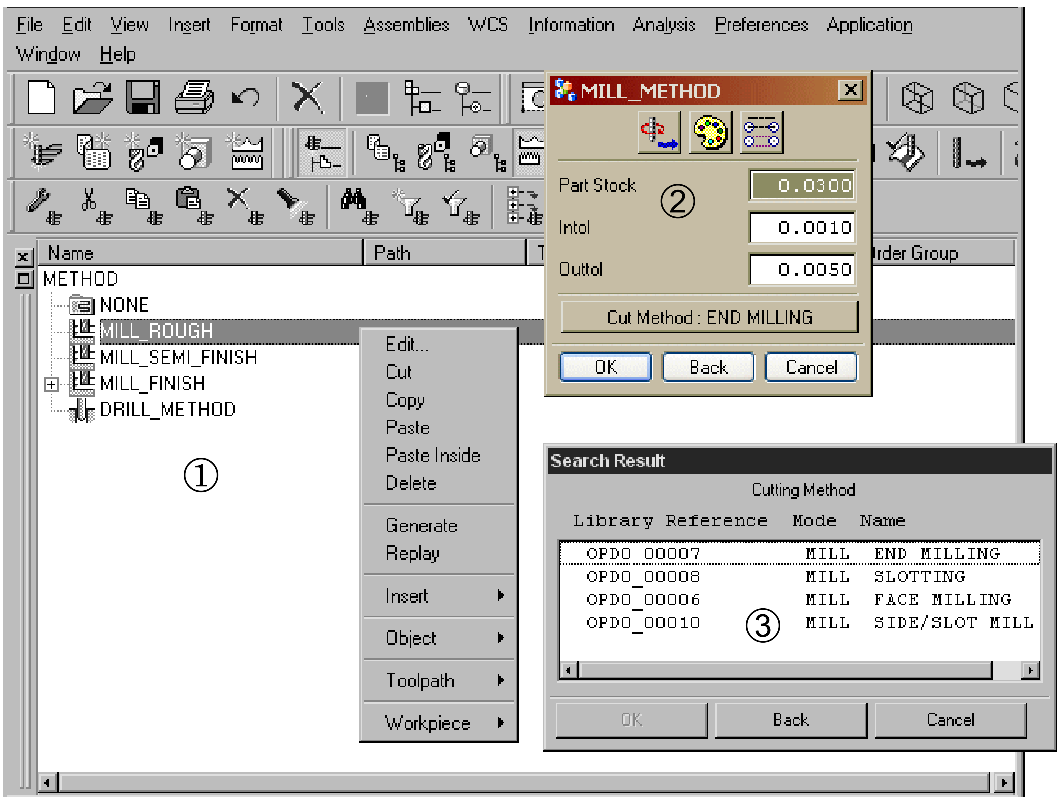
❑ **Cancel** the Tool dialog.

You are finished with this activity and will be using this part file in the next activity.

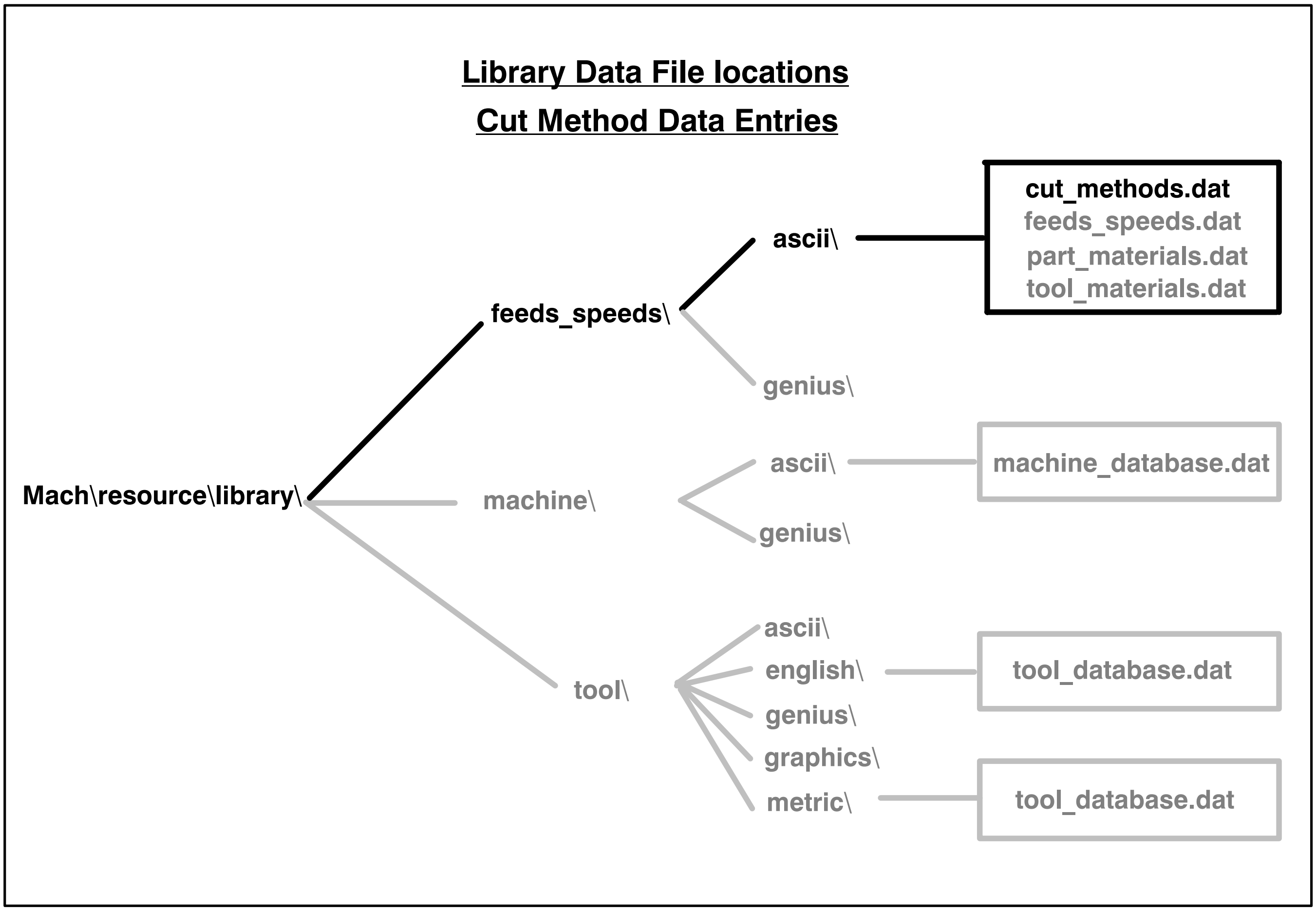
Cut Method Libraries

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 4—8: Cut Method Libraries

In this activity you will become familiar with the Cut Methods library. You will then enter a new cut method in the Cut Methods database file.

Step 1 Accessing the Cut Method library.

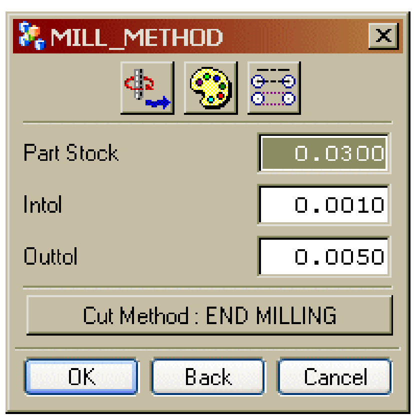
❑ Continue with the opened part, **ama_lib_function.prt.**

❑ If necessary, enter the **Manufacturing Application.**

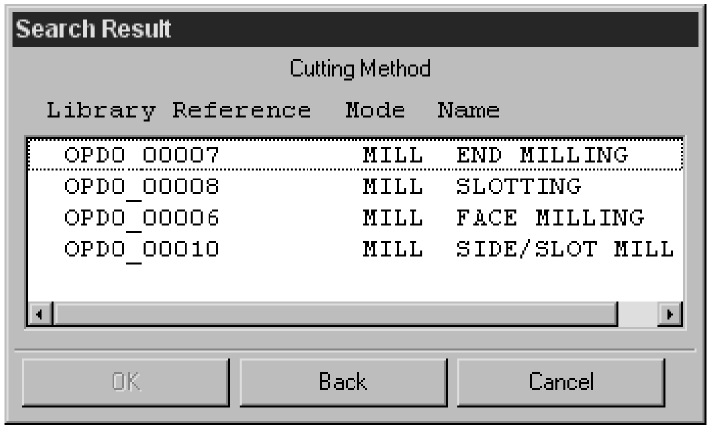
❑ If necessary, change the view of the Operation Navigator to the **Machine Method** view.

❑ Highlight **MILL_ROUGH**, select **MB3**, then Edit.

The Mill_Method dialog is displayed.



❑ Select the **Cut Method** button.



The Search Result list is displayed.

❑ Select the **OPD0_00010 MILL SIDE/SLOT MILL** method**.**

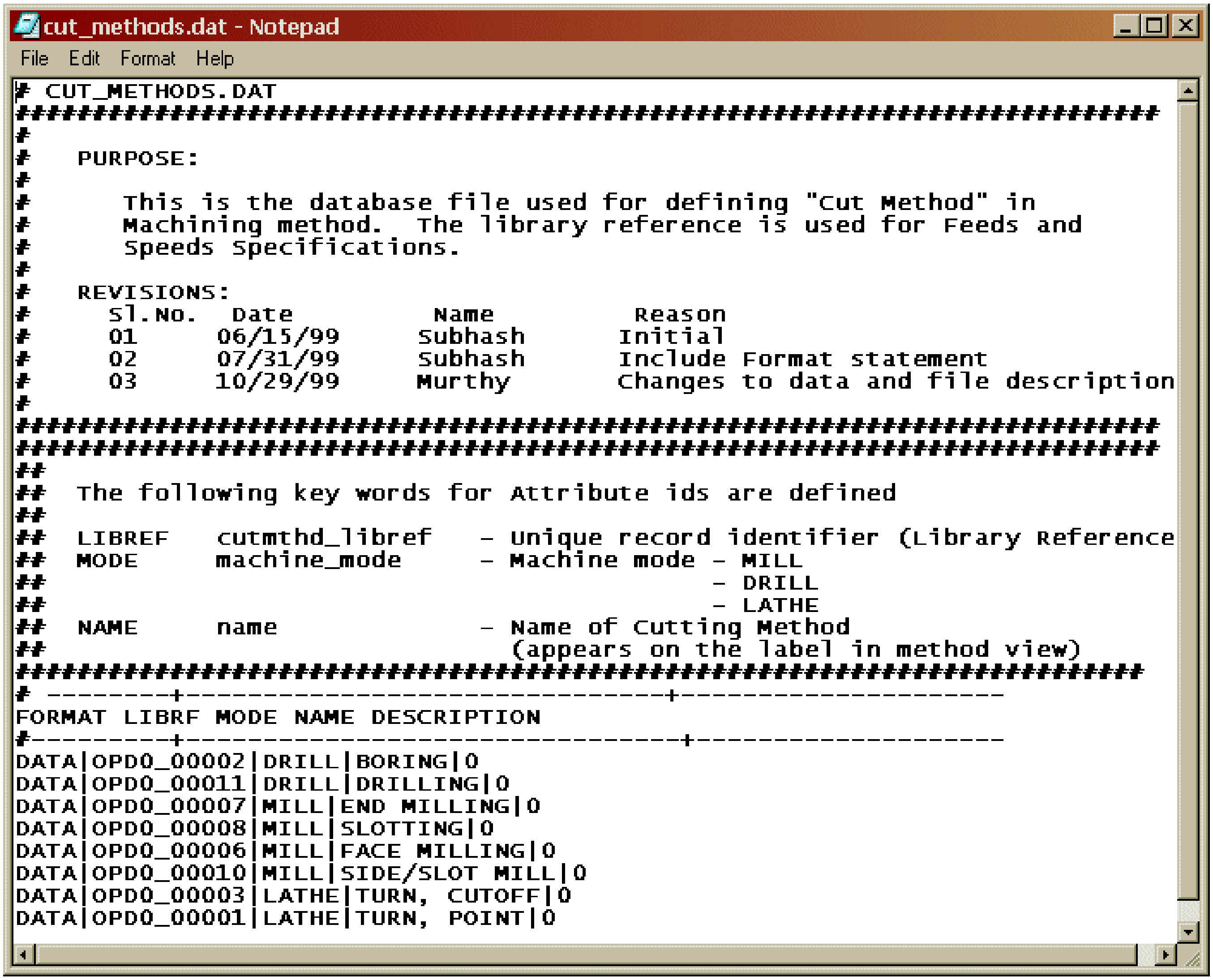
❑ Choose **OK** and notice the label on the **Cut Method** button.

❑ Dismiss all dialogs.

Step 2 Adding a new cut method to the cut method database.

You will now add the **Cut Method**, **Plunging**, to the Cut Method database.

❑ Using your text editor, open the **cut_methods.da**t file located in your **MACH\resource\library\feeds_speeds\ascii**\ home directory.



cut_methods.dat - Notepad

File Edit Format Help

```
# CUT_METHODS.DAT
#########################################################################
#
#   PURPOSE:
#
#      This is the database file used for defining "Cut Method" in
#      Machining method.  The library reference is used for Feeds and
#      Speeds Specifications.
#
#   REVISIONS:
#     Sl.No.   Date         Name         Reason
#     01      06/15/99     Subhash      Initial
#     02      07/31/99     Subhash      Include Format statement
#     03      10/29/99     Murthy       Changes to data and file description
#
#########################################################################
#########################################################################
##
##  The following key words for Attribute ids are defined
##
##  LIBREF    cutmthd_libref    - Unique record identifier (Library Reference
##  MODE      machine_mode      - Machine mode - MILL
##                                             - DRILL
##                                             - LATHE
##  NAME      name              - Name of Cutting Method
##                                (appears on the label in method view)
########################################################################
# --------+-------------------------------+--------------------
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
```

❑ After the last DATA statement, add the following line (it is somewhat easier to copy and paste an existing line, making any modifications necessary to that line):
DATA|OPD0_00072|MILL|PLUNGING|0

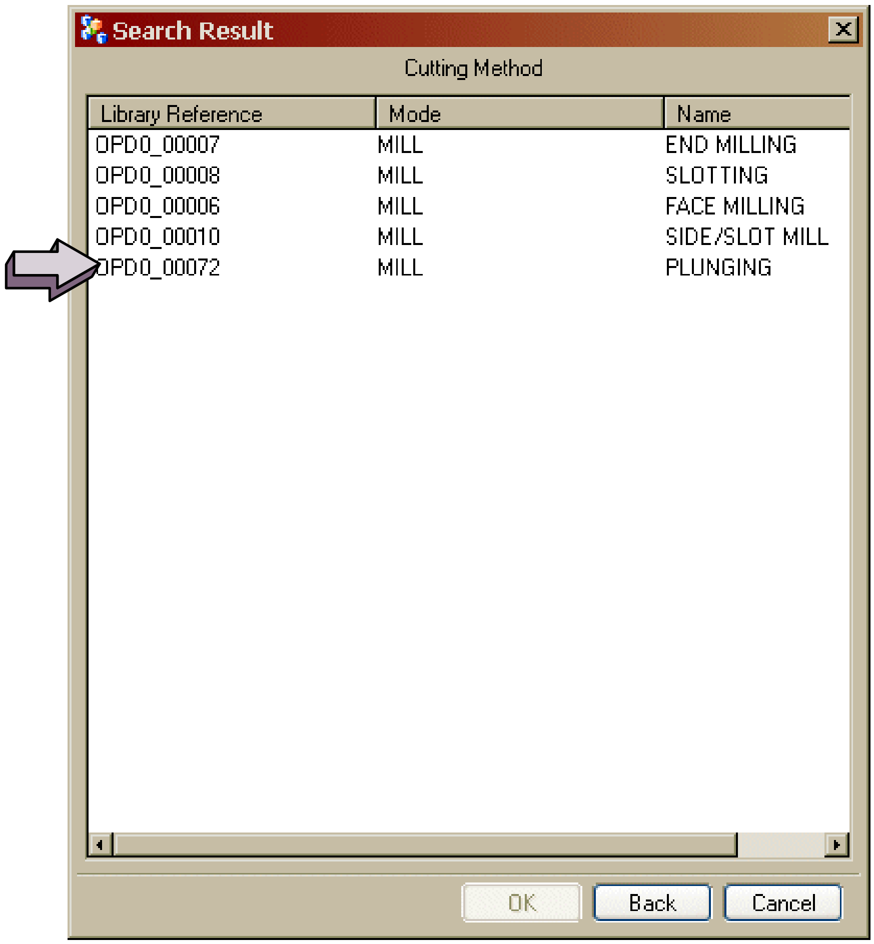
❑ On a separate sheet of paper, record the tool material code (OPD0_00072). You will need to reference this later.

❑ Save the file and return to Unigraphics.

❑ In the **Method** view, open the **MILL_FINISH** object.

❑ Choose the **Cut Method** button.

The new **Cut Method**, **OPD0_00072** is now listed in the **Search Result** and is available for selection.



❑ **Cancel** the **Search Result** list window.

❑ **Cancel** the **Mill_Method** dialog.

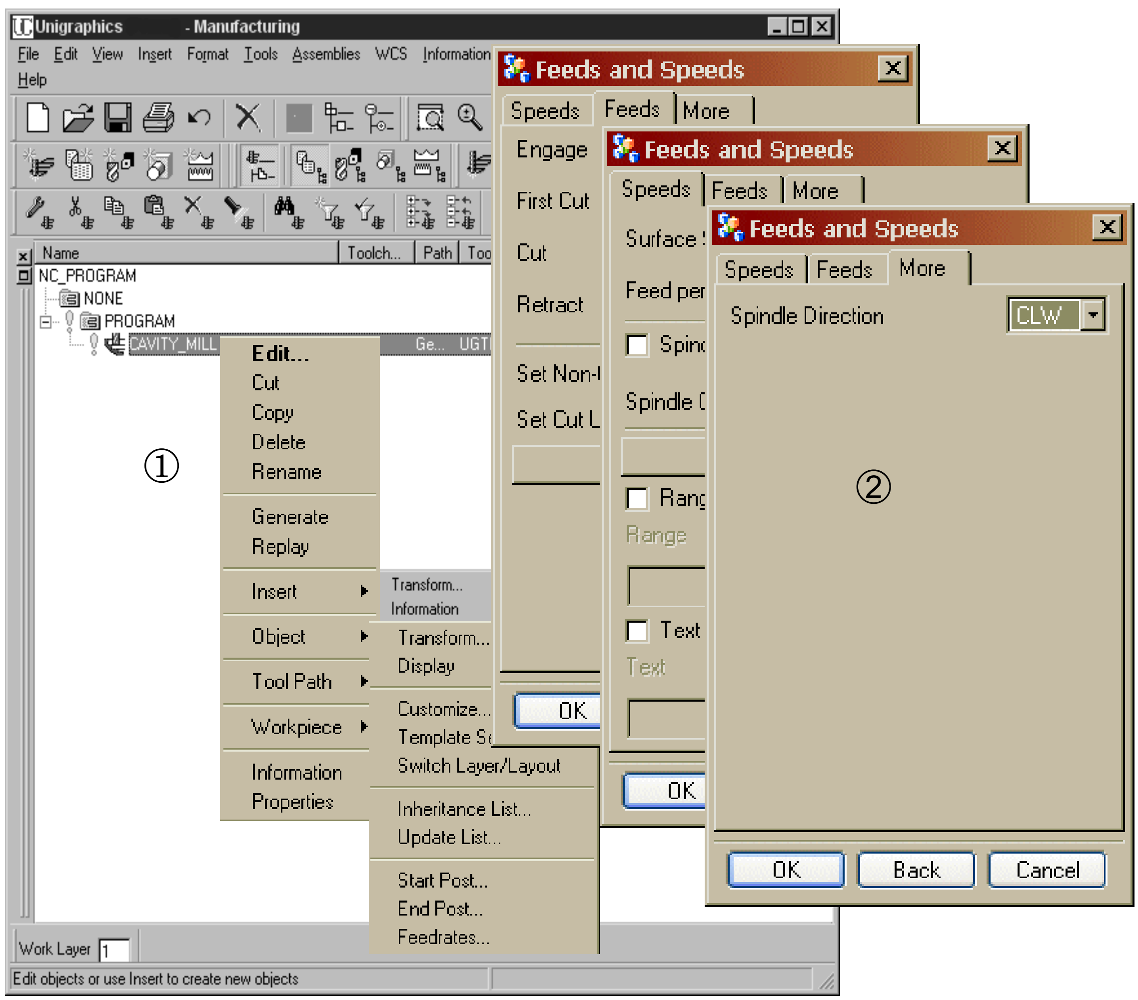
❑ Close the part.

You are finished with this activity.

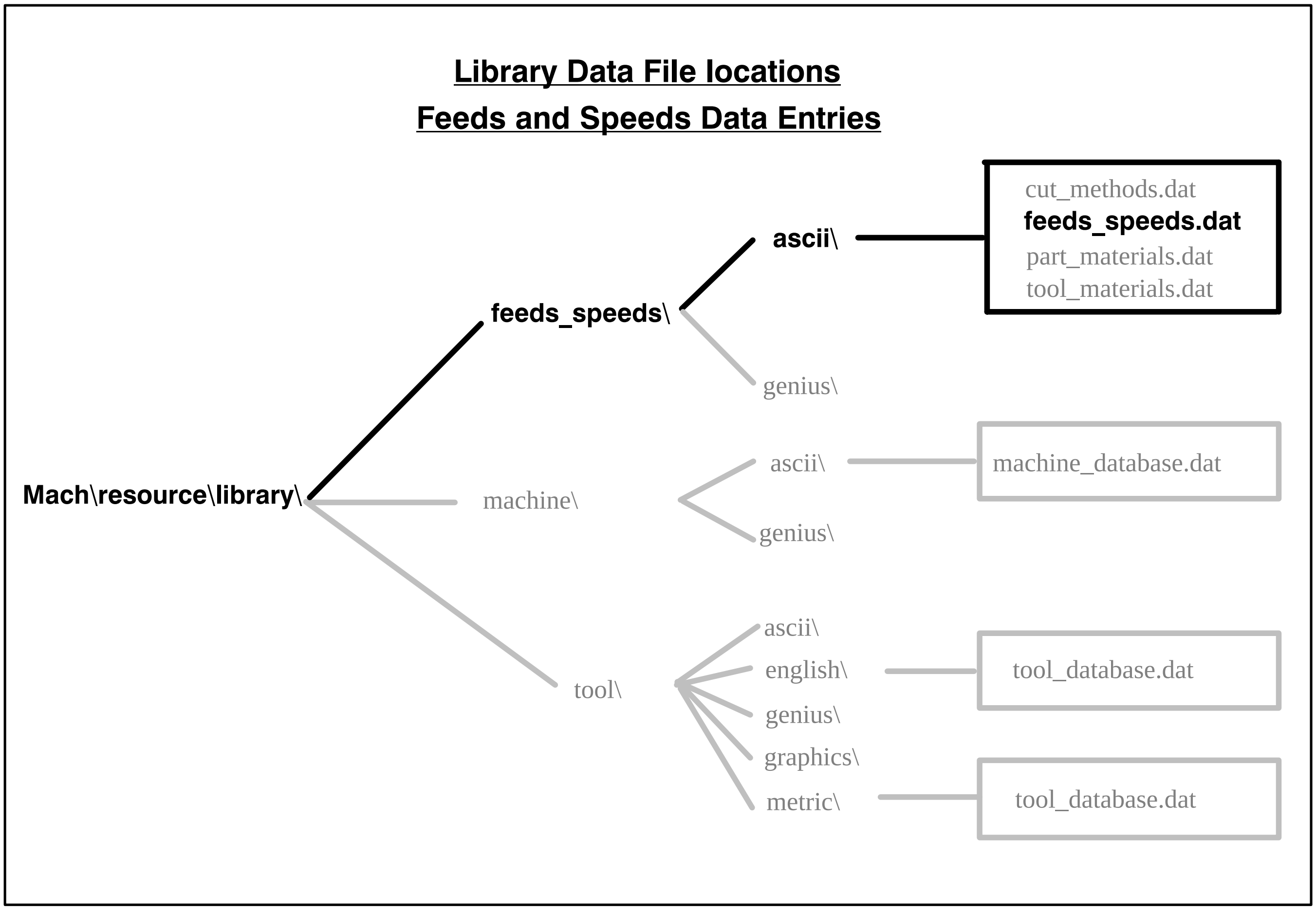
Feeds and Speeds Libraries

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**.



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.

4

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**, verify 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               Surface Speed(FPM)
# SURF_SPEED_MPM – surface_speed               Surface 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
```

4

Activity 4—9: 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. The options that will be selected are options added in previous activities.

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.

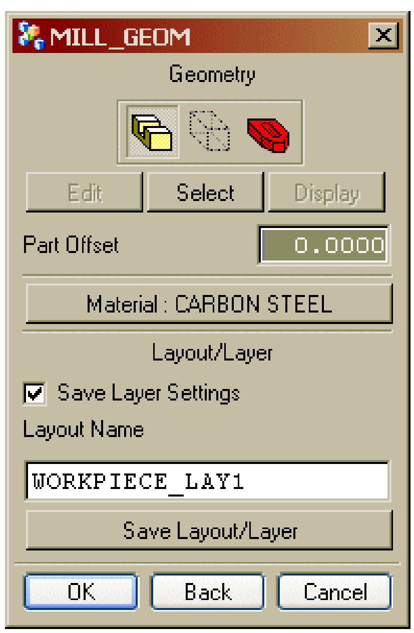
Step 2 Defining the Part Material.

You only need to define the **Part Material** once.

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

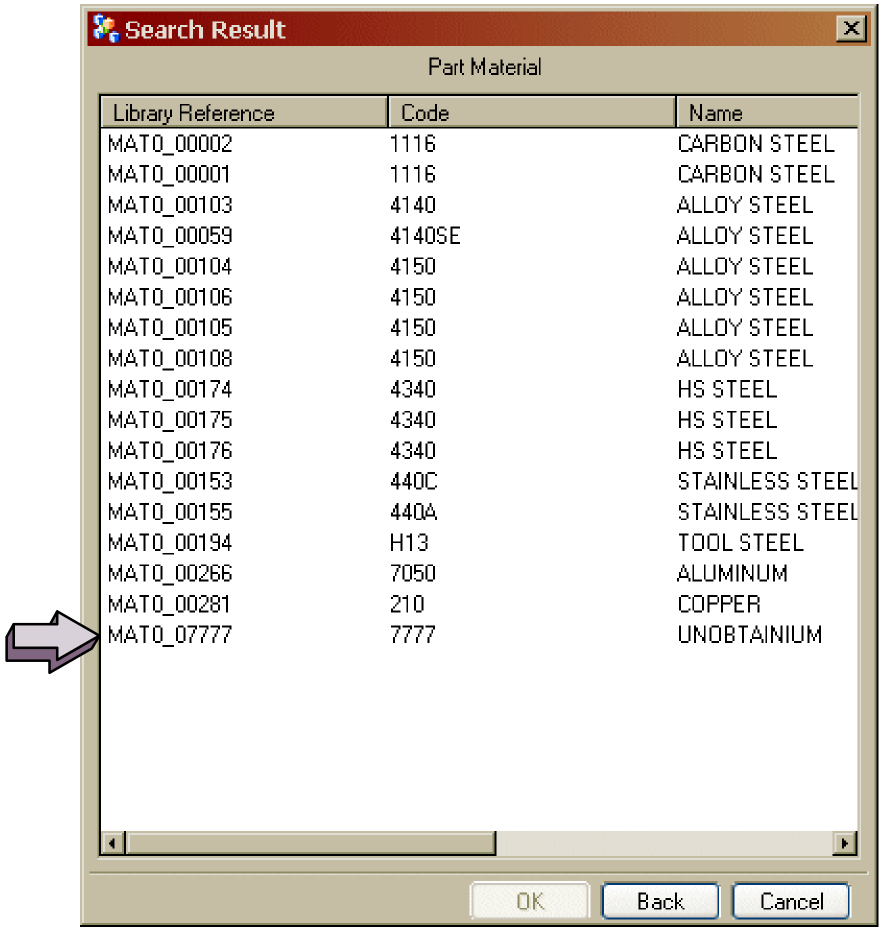
❑ Highlight **Workpiece**, then **MB3, Edit**.

The **Mill_Geom** dialog is displayed.



❑ Choose the **Material** button.

The **Search Result** window is displayed. You will select the material type from here.



❑ Select **UNOBTAINIUM** from the list and then choose **OK** twice.

Step 3 Defining the Cut Method.

You will now define the **Cut Method** by editing the **MILL_ROUGH** method.

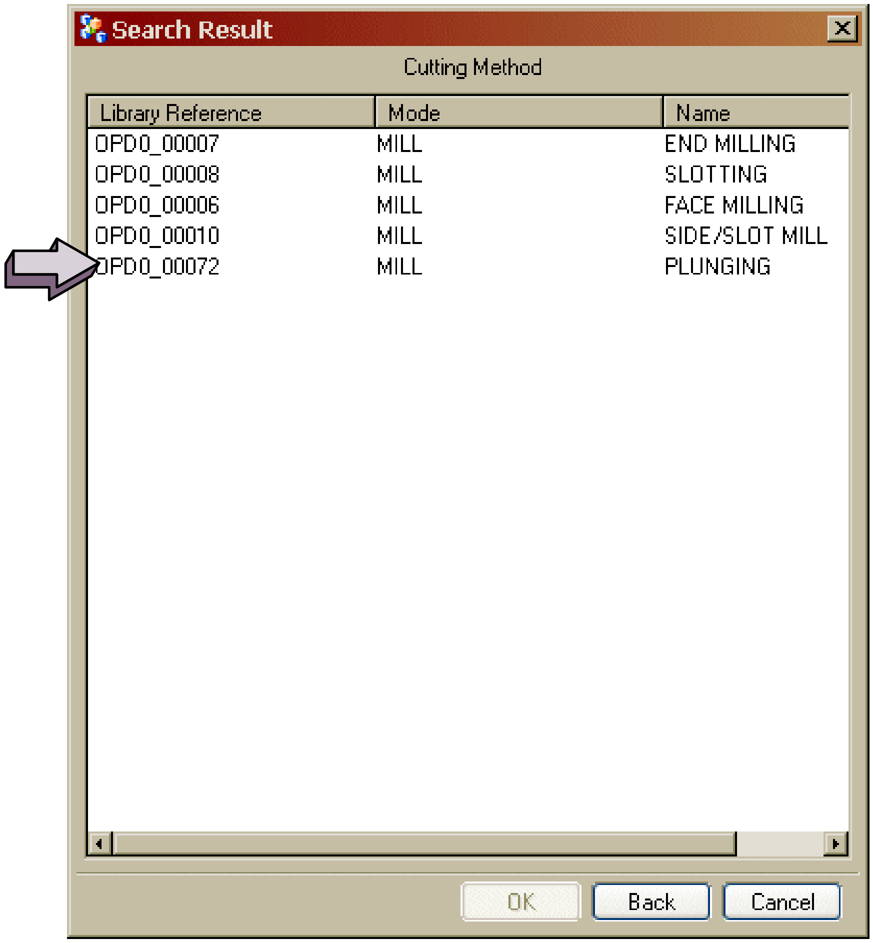
❑ Change the Operation Navigator to the **Method** view and expand all objects.

❑ Highlight **MILL_ROUGH**, then **MB3, Edit**.

The **MILL_METHOD** dialog is displayed.

❑ Choose the **Cut Method** button.

The **Search Result** dialog is displayed.



From this dialog, you can choose the type of cutting.

Select **PLUNGING** and then choose **OK** twice.

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.

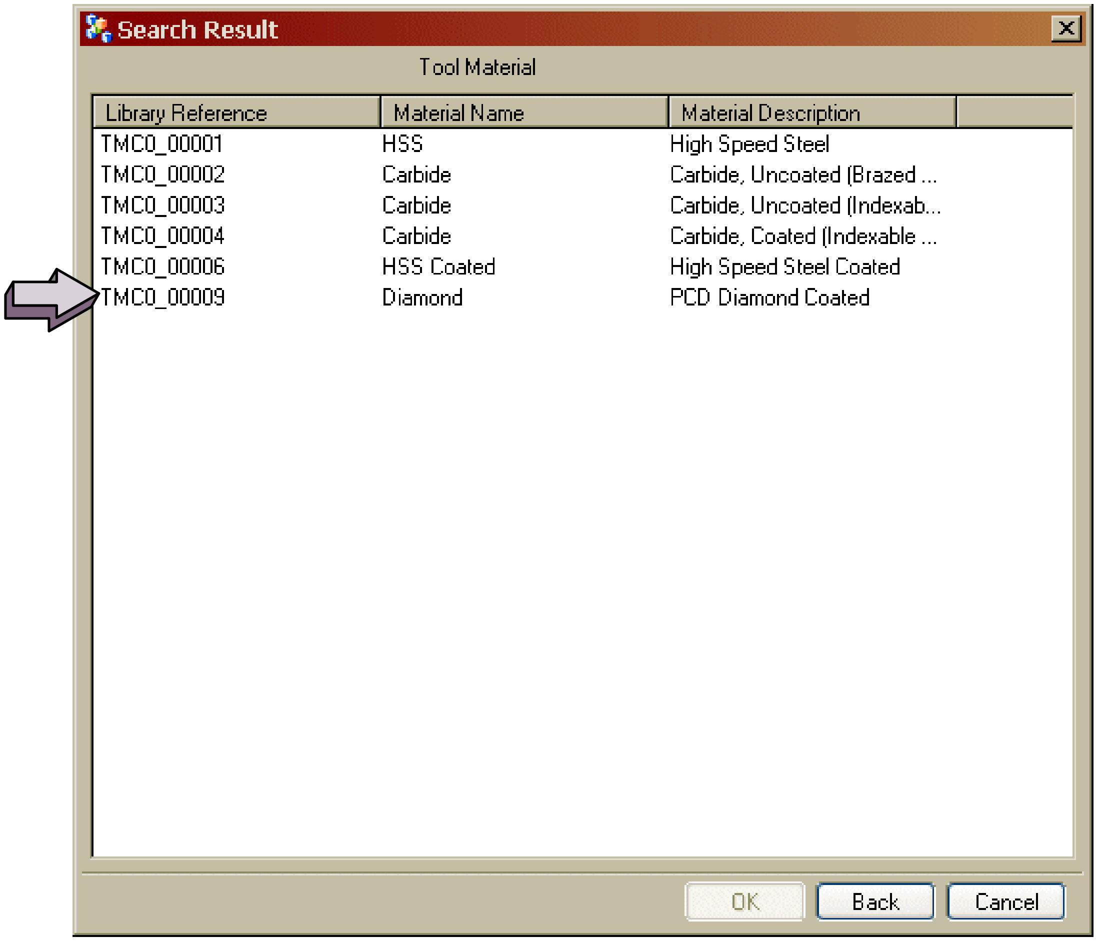
❑ Change the Operation Navigator to the **Tool** view.

❑ Highlight the **UGTI0201_013** tool name, then **MB3, Edit**.

The Milling End Mill 1” dialog is displayed. Note the lower portion of the dialog. This is where you define the **Tool Material**. Currently the material type is **HSS COATED**.

❑ Choose the **Material** button.

The available tool material types are displayed.



❑ Select **TMCO_00009** Diamond on the list, then choose **OK** twice.

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.

❑ Change the Operation Navigator to the **Program Order** view and expand all group objects.

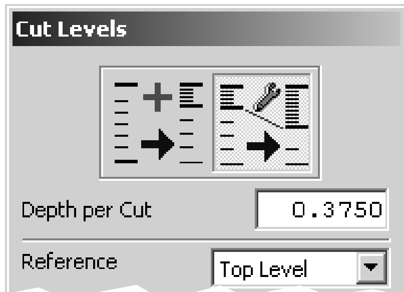
❑ Highlight the operation named **Cavity Mill,**then **MB3, Edit**.

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**.



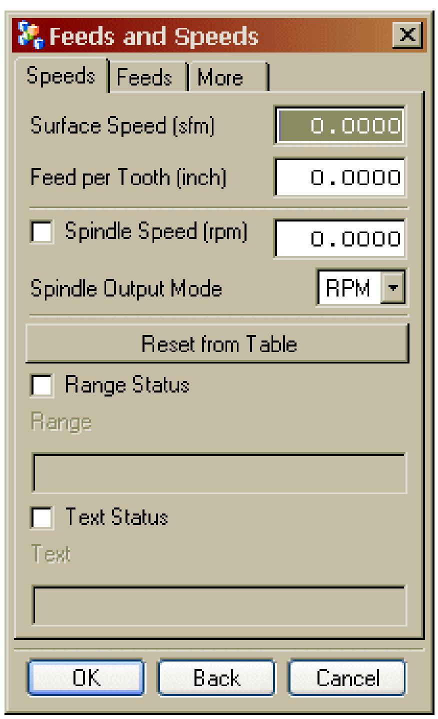
❑ Choose **OK** twice.

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.

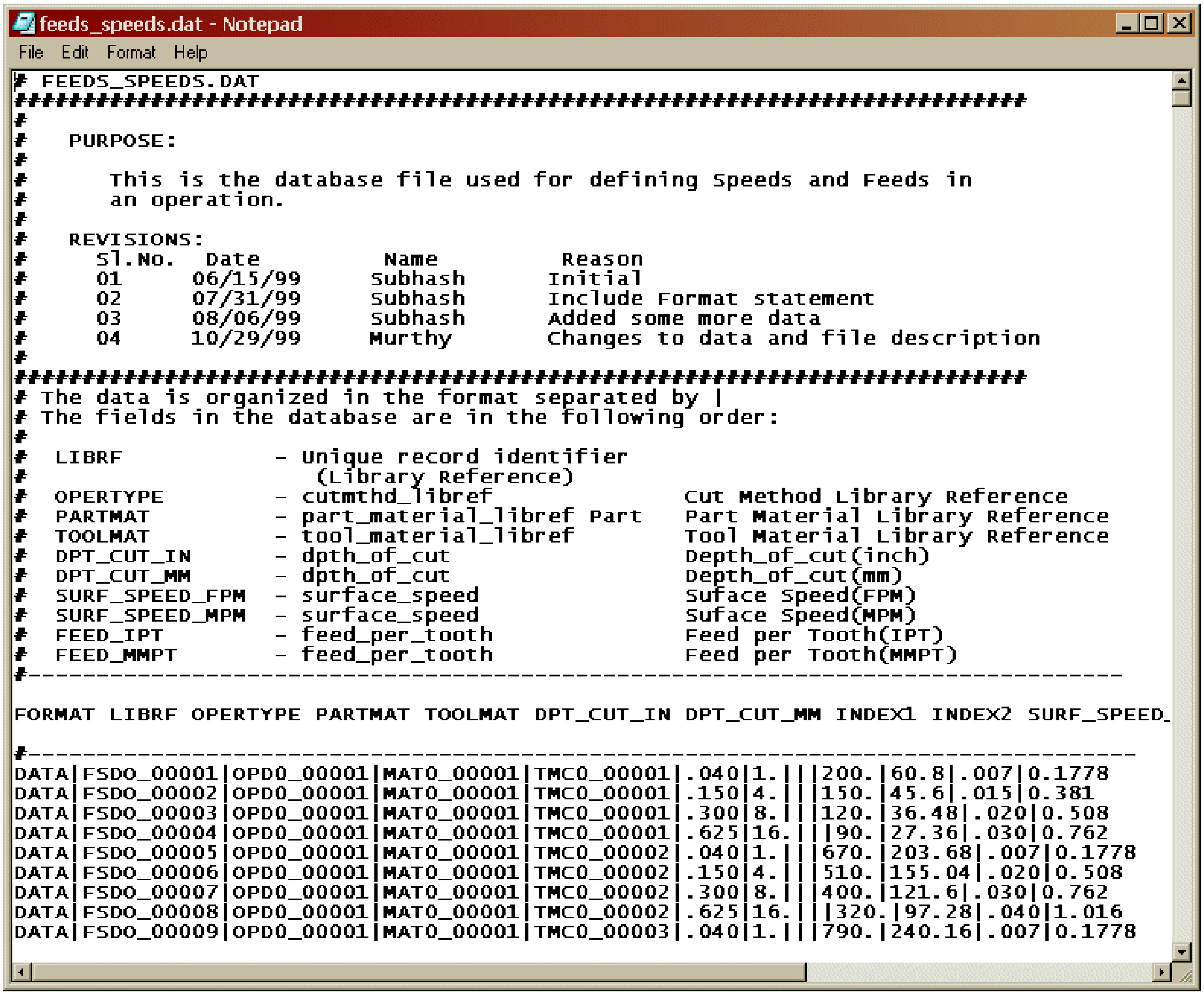
An error message is returned stating that no appropriate feed speed data was found.

This error message was the result of not having the proper data in the feeds_speeds.dat file which is used to define feeds and speeds in an operation.

❑ Choose **OK** to return to the **Feeds and Speeds** dialog.

Step 7 Creating the data entry necessary for feed and speed calculations.

❑ Using your text editor, open the **feeds_speeds.da**t file located in your **MACH\resource\library\feeds_speeds\ascii**\ home directory.



feeds_speeds.dat - Notepad

File Edit Format Help

```
# FEEDS_SPEEDS.DAT
######################################################################
#
#   PURPOSE:
#
#      This is the database file used for defining Speeds and Feeds in
#      an operation.
#
#   REVISIONS:
#     Sl.No.   Date          Name          Reason
#     01      06/15/99      Subhash      Initial
#     02      07/31/99      Subhash      Include Format statement
#     03      08/06/99      Subhash      Added some more data
#     04      10/29/99      Murthy       Changes to data and file description
#
######################################################################
# The data is organized in the format separated by |
# The fields in the database are in the following order:
#
#  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 INDEX1 INDEX2 SURF_SPEED_

#-----------------------------------------------------------------------------
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
DATA|FSDO_00003|OPD0_00001|MAT0_00001|TMC0_00001|.300|8.|||120.|36.48|.020|0.508
DATA|FSDO_00004|OPD0_00001|MAT0_00001|TMC0_00001|.625|16.|||90.|27.36|.030|0.762
DATA|FSDO_00005|OPD0_00001|MAT0_00001|TMC0_00002|.040|1.|||670.|203.68|.007|0.1778
DATA|FSDO_00006|OPD0_00001|MAT0_00001|TMC0_00002|.150|4.|||510.|155.04|.020|0.508
DATA|FSDO_00007|OPD0_00001|MAT0_00001|TMC0_00002|.300|8.|||400.|121.6|.030|0.762
DATA|FSDO_00008|OPD0_00001|MAT0_00001|TMC0_00002|.625|16.|||320.|97.28|.040|1.016
DATA|FSDO_00009|OPD0_00001|MAT0_00001|TMC0_00003|.040|1.|||790.|240.16|.007|0.1778
```

❑ After the last DATA statement, add the following line (it is somewhat easier to copy and paste an existing line, making any modifications necessary to that line):
DATA|FSDO_09999|OPD0_00072|MAT0_07777|TMC0_00009|.040|1.|||673.|60.8|0.00567|0.1778

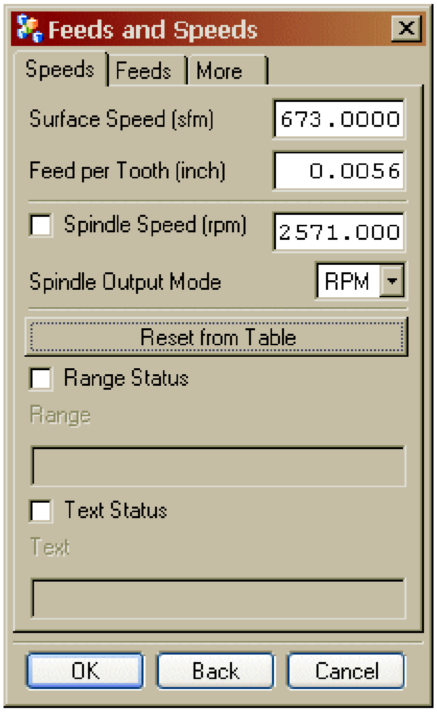
This line represents the following: when plunge milling in Unobtainium using a PCD coated tool, up to a depth of .040, use a cutting speed of 673 SFM and a chipload of .00567” per tooth.

❑ Save the file and return to Unigraphics.

❑ Choose the **Reset from Table** button on the **Feeds and Speeds** dialog.

4

Proper feeds and speeds are calculated based on cutting tool material, part material and method used.



NOTE Resetting the speeds and feeds turns off the inheritance of feed rates from the method parent.

❑ **Save** and **close** the part file.

You have completed the activity and the lesson.

4

SUMMARY

Libraries are used for numerous applications in the Manufacturing Application. Libraries are convenient and easy tools that can be modified and 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
- Created graphic tools for the Tool Graphics library
- Reviewed the Machine Tool, Part Material, Cutting Tool Material and Cut Method Libraries and added entries to their respective data files
- Modified various option settings to show their effect on feeds and speeds

Machining Faceted Geometry
Lesson 5

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

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 5—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.

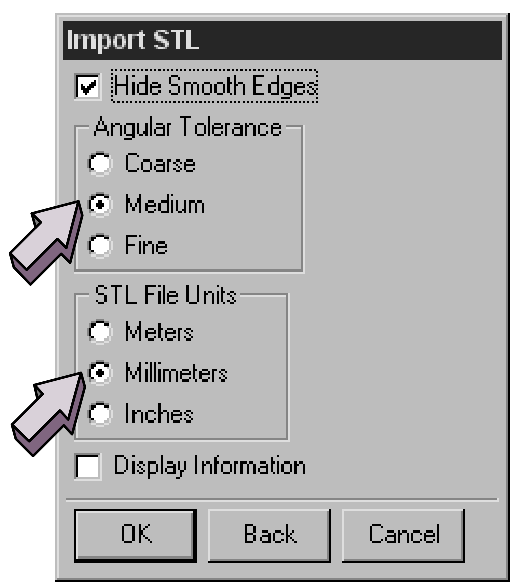
❑ Select **OK**.

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**

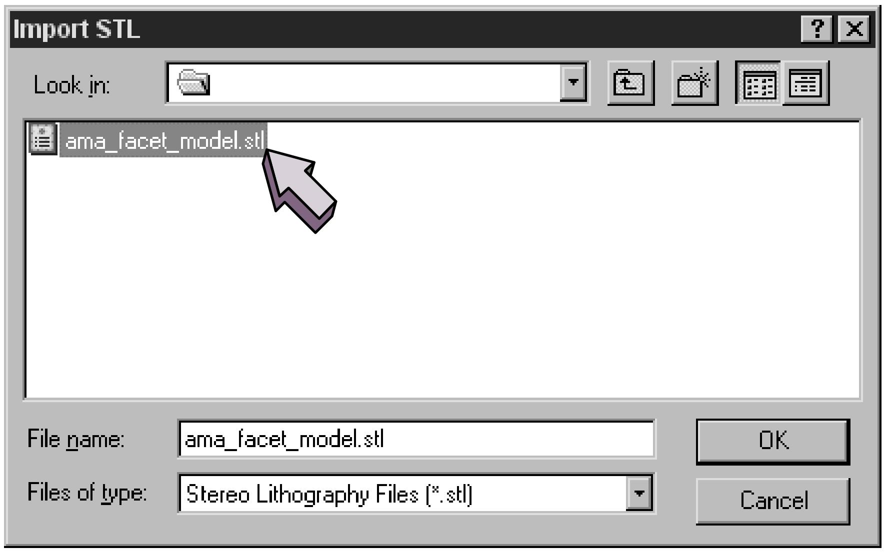
The **Import STL** dialog is displayed.



❑ If necessary select **Angular Tolerance** as **Medium** and STL **File Units** as **Millimeters.**

❑ 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.

5

Step 3 Create the Cavity Milling operation necessary to machine the imported faceted model.

❑ Enter the **Manufacturing Application**.

The **Machining Environment** dialog is displayed.

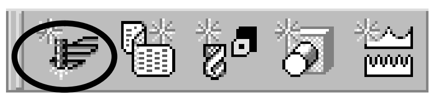
❑ Select **mill_contour** as the **CAM Session Configuration**, select **mill_contour** for the **CAM Setup** and then select **Initialize**.

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

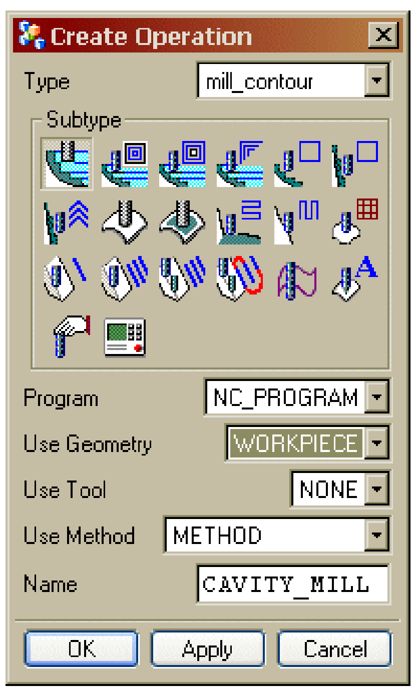
The Operation Navigator is displayed.

❑ If necessary, change the view of the Operation Navigator to the **Program Order View.**

❑ Select **Create Operation** from the **CAM Create** tool bar.

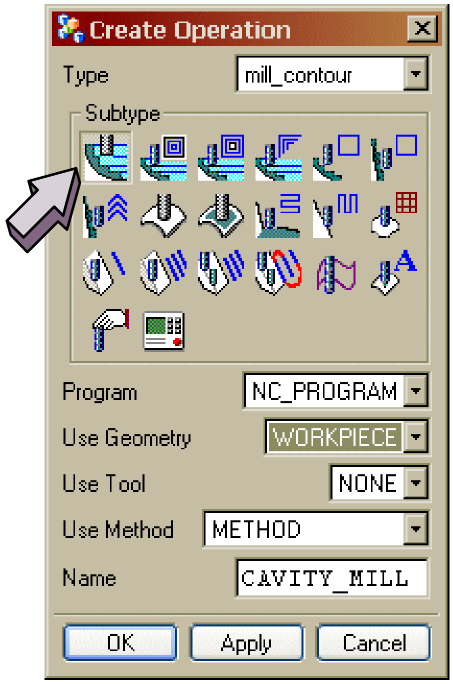


The **Create Operation** dialog is displayed.



5

❑ Select the **Cavity_Mill** subtype.



❑ Select **OK**.

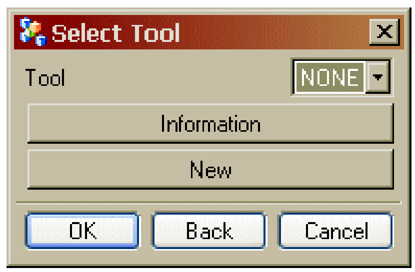
The Cavity_Mill dialog is displayed. You will use an 8mm ball tool, **ugt0203_003**, to rough out the cavity. Other parameters used will be default parameters. You will first select the ball tool and then the geometry to machine the part.

❑ Choose the Groups tab from the CAVITY_MILL dialog.

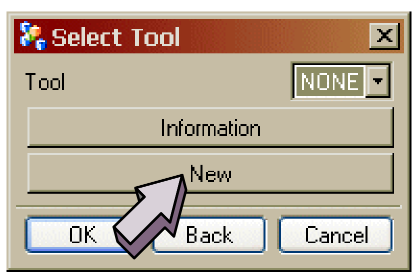
5

❑ If necessary, choose **Tool: NONE** and then the **Select** button.

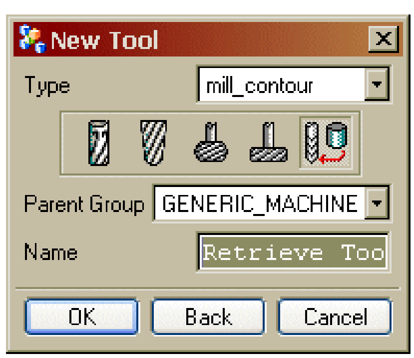
The **Select Tool** dialog is displayed.



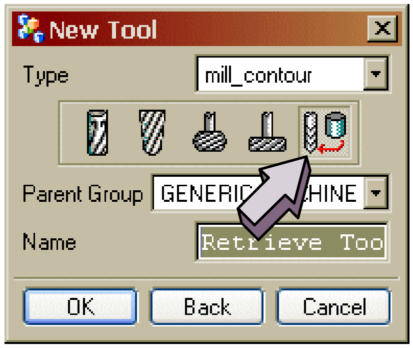
❑ Choose the **New** button.



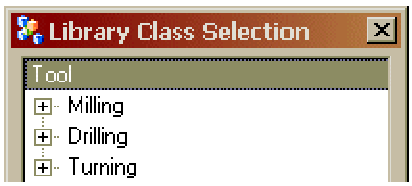
The **New Tool** dialog is displayed.



❑ Choose the **Retrieve Tool** icon and then choose **OK**.

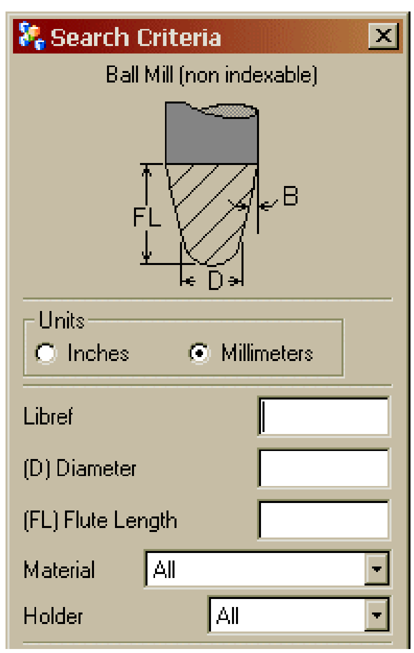


The **Library Class Selection** dialog is displayed.

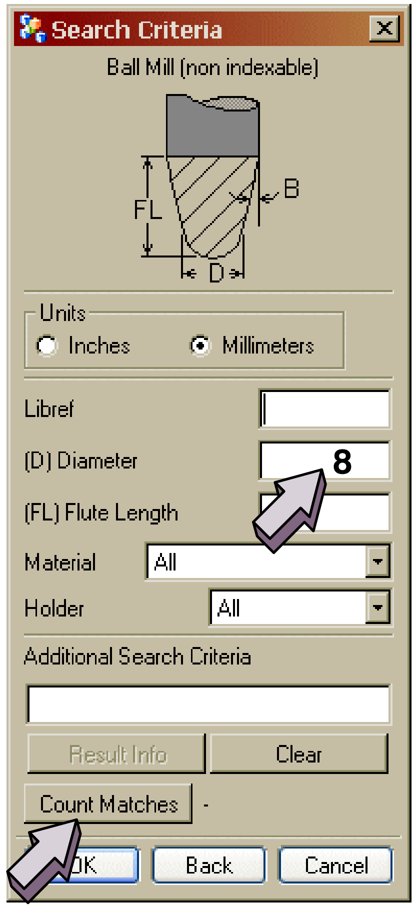


❑ Expand the **Milling** group object and then double-click on the **Ball Mill (non indexable)** object.

The **Search Criteria** dialog is displayed.



Key **8** in the **Diameter** field and then choose the Count Matches button.



Two matches, meeting the criteria of 8mm diameter, are found.

❑ Choose **OK** from the **Search Criteria** dialog.

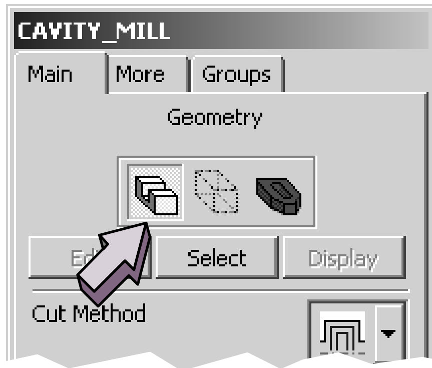
The **Search Result** dialog is displayed.

❑ Highlight **ugt0203_003** from the **Search Result** dialog.

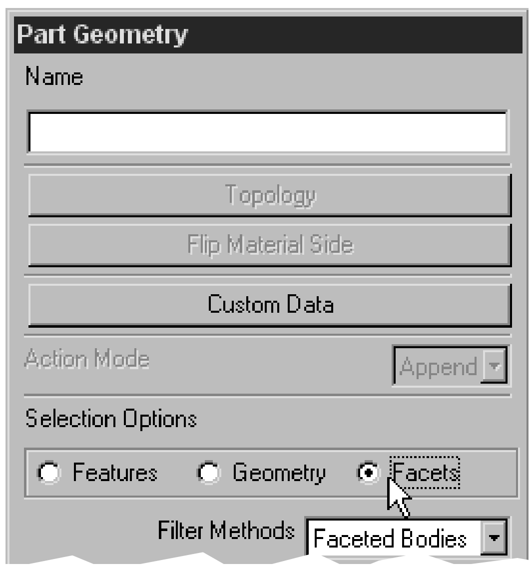
❑ Choose **OK**.

❑ Select the **Main** tab from the **CAVITY_MILL** dialog.

❑ 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.

❑ Examine the tool paths just created.

❑ Close the part.

This concludes the activity and the lesson.

SUMMARY

Direct Machining of Facets (DMP) provides an easy and efficient method of machining parts that have been reversed engineered and imported from 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

(This Page Intentionally Left Blank)

High Speed Machining

Lesson 6

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 places 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 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.

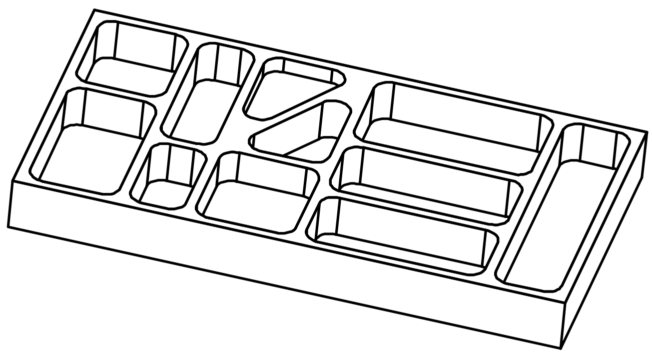
The key to successfully perform high speed machining are various methods used in the cutting operations. Techniques such as roughing at a shallow depth with maximum width of cut promotes longer tool life and higher cutting accuracy. In some cases, semi-finish passes can be completely eliminated. Tool 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 6–1: 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.**



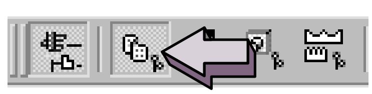
❑ **Save As ***_hsm_1.prt** where *** represents your initials.

❑ 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 Order** view.



❑ Select the operation, **pocket_standard,** use **MB3** and select **Toolpath → Verify.**

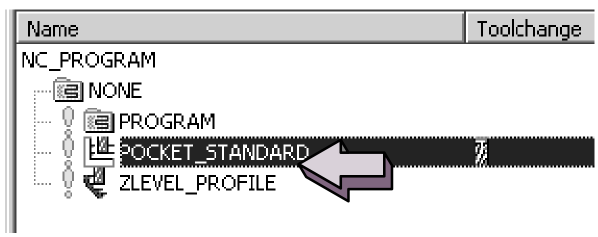
The tool path for the **pocket_standard** operation is displayed. Pay particular attention to the method of engagement, the sharp corners generated within the tool path, the depth of cut, etc. The tool path display has been intentionally slowed so that you may observe the various movements.

You will now edit the **pocket_standard** operation and modify parameters that are applicable for High Speed Machining.

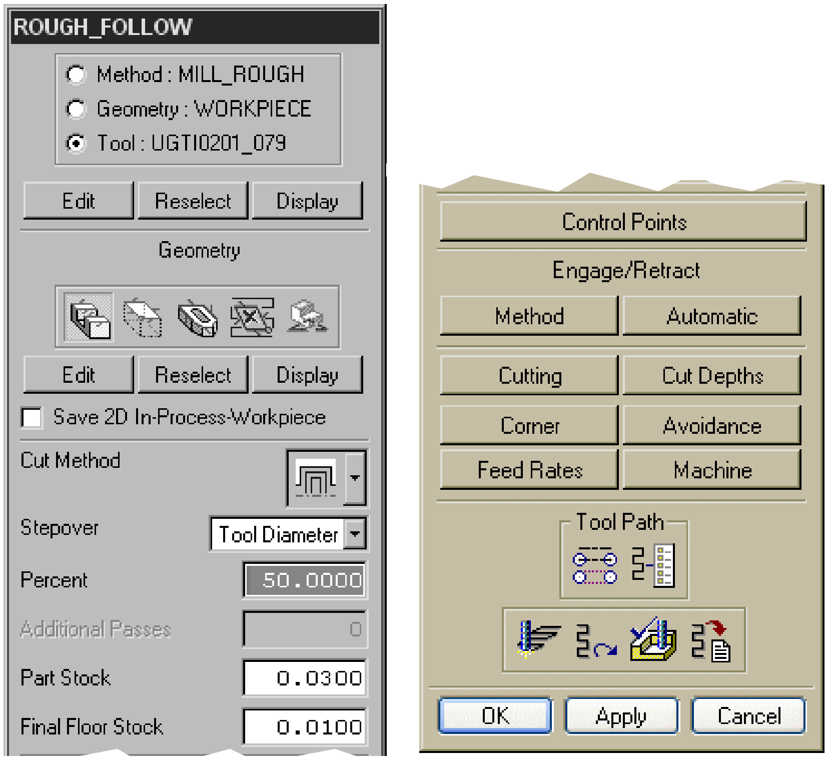
❑ **Cancel** the **Tool Path** visualization dialog.

Step 2 Edit the existing operation and modify parameters suitable for High Speed Machining.

❑ Double click on the operation, **pocket_standard.**

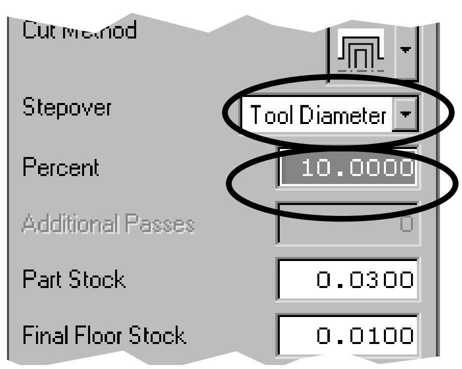


The **ROUGH_FOLLOW** dialog is displayed.



Currently, all parameters set are default parameters. You will now change parameters for High Speed Machining.

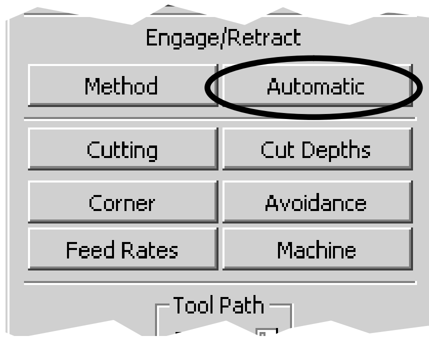
❑ From the **Stepover** pull down menu, select **Tool Diameter** and then change **Percent** to **10.** This will minimize the stepover value to 10% of the effective cutter diameter and will help maintain constant tool loading conditions.



You will now change the parameters used for engagement from one cut level to the next to **Helical**.

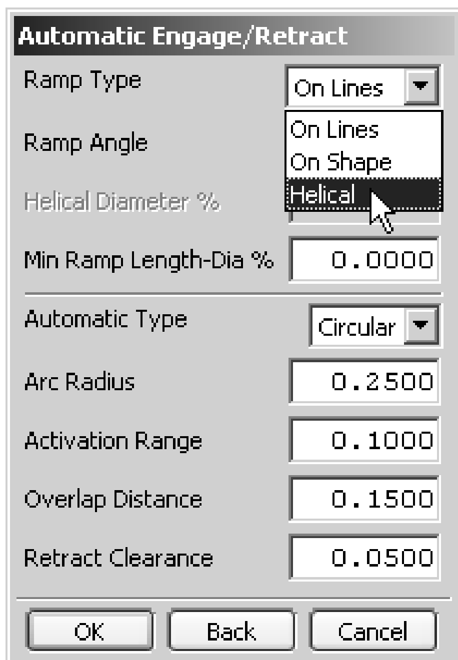
❑ Select **Automatic** under **Engage/Retract.**

6



The **Automatic Engage/Retract** dialog is displayed.

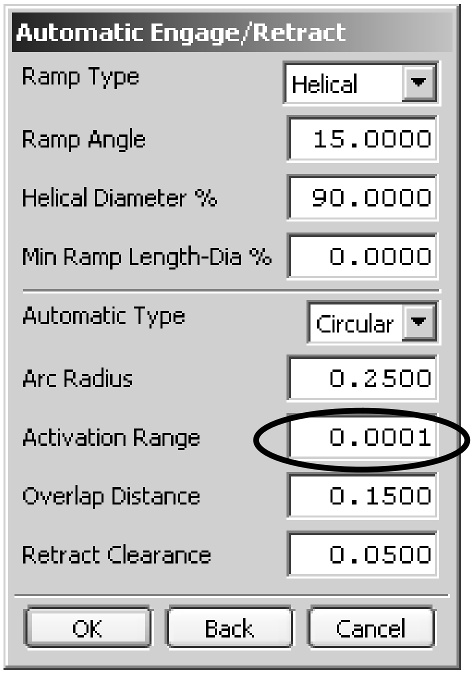
❑ Select the **Ramp Type** pull down menu and then select **Helical.**



Changing the **Ramp Type** to **Helical** insures the 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.

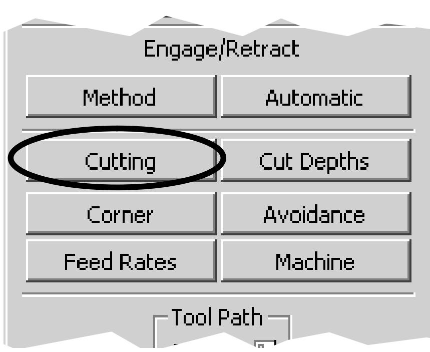
❑ Change the **Activation Range** to **.0001.**



Changing the **Activation Range** to this small value insures that the cutter will not engage the part as it approaches the walls.

❑ Choose **OK**.

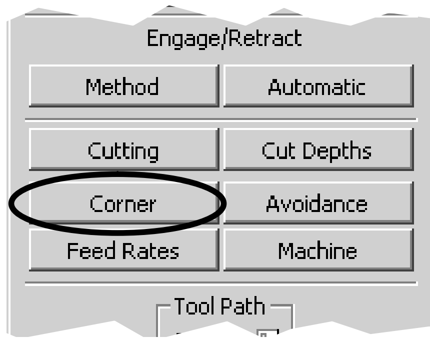
❑ Select **Cutting** and change the **Intol** parameter to **.00005** and the **Outtol** parameter to **.0005**.



As the machine tool travels quicker and very accurately, a much finer surface finish can be achieved. The movement per block can be shorter to obtain the best dimensional accuracy.

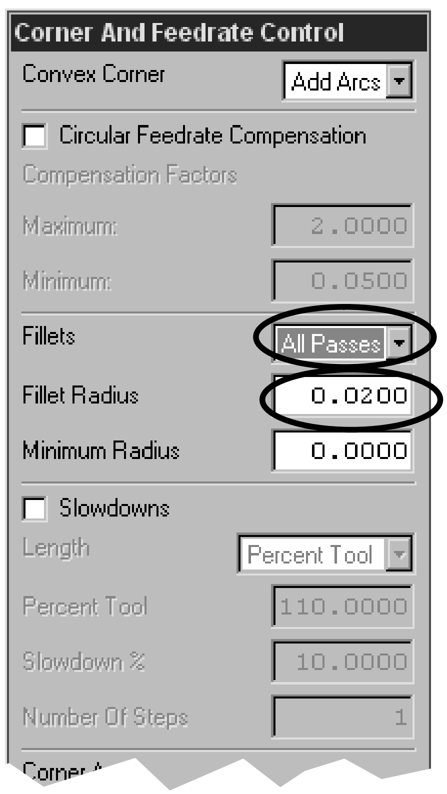
❑ Choose **OK**.

❑ Select **Corner.**

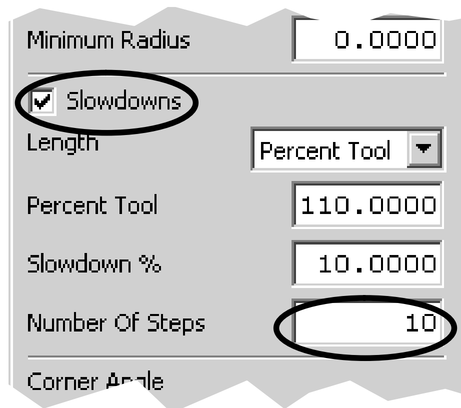


This option will add a fillet at all corners (corner roll) which eliminates sharp and sudden moves.

❑ 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**.

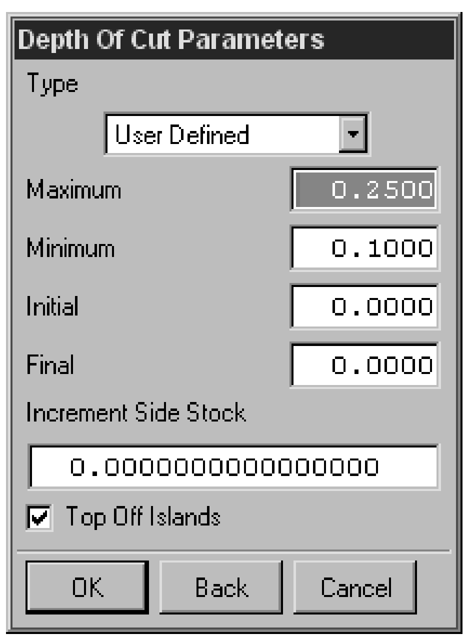


Slowdown is used to slow the feed rates as you approach corners or obstructions in the tool path. Slowdown can be controlled by length, starting location and the rate of slowdown. **Number Of Steps** allows you to set the abruptness of slowdown. The greater the number, the more even the slowdown.

❑ Choose **OK**.

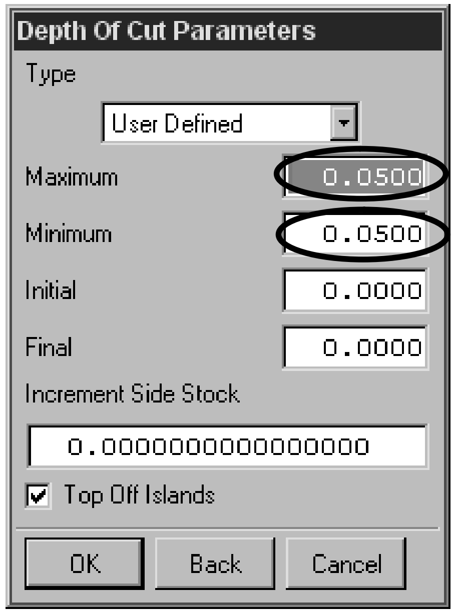
❑ Select **Cut Depths**.

The **Depth of Cut** Parameters dialog is displayed.



❑ Change the **Maximum** and **Minimum** values to **.050.**

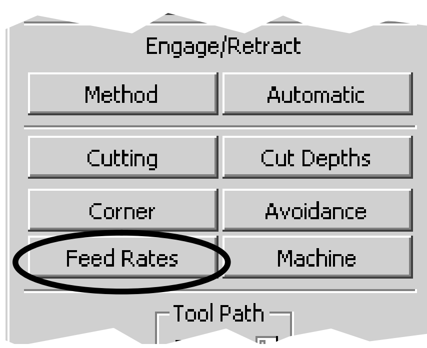
6



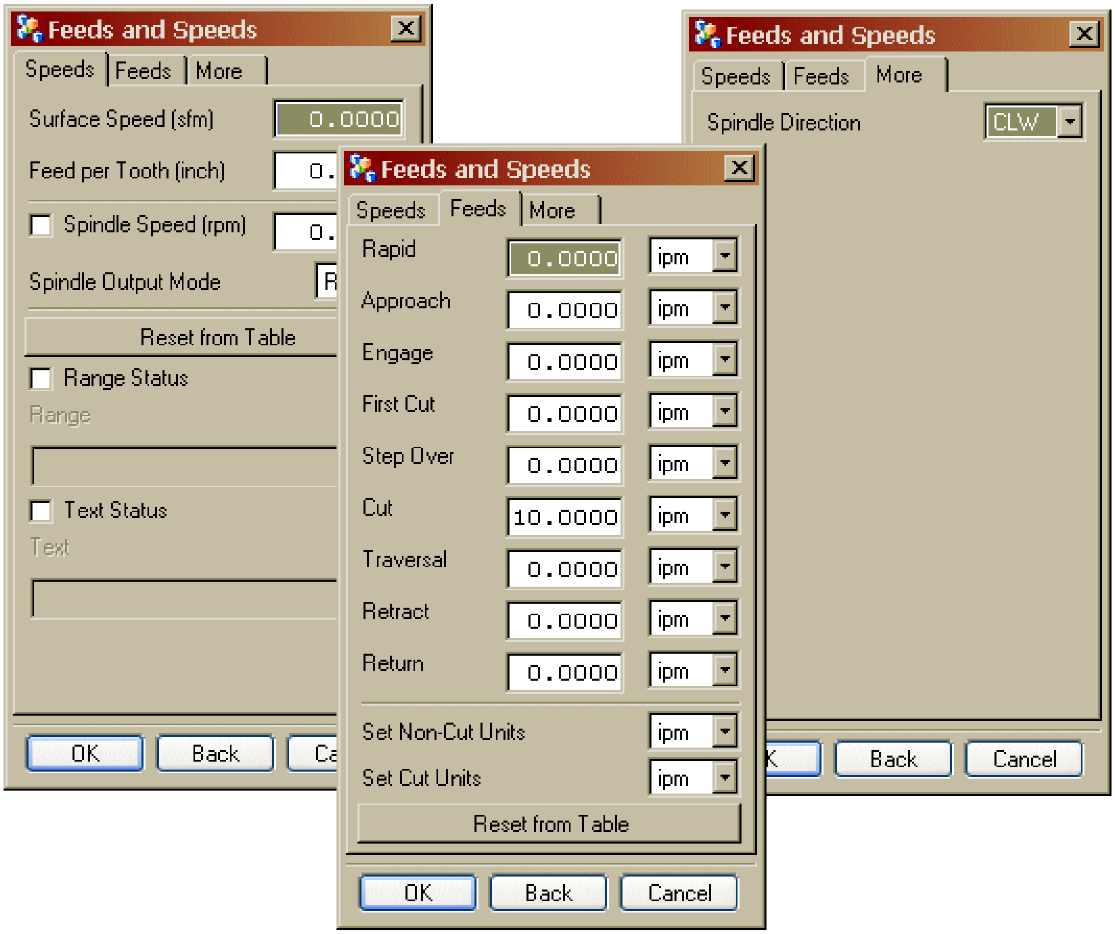
Reducing the depth of cut while increasing feed rates is a core process change of High Speed Machining. More material can be removed in less time. An advantage of a shallower depth of cut is less horsepower required to cut through the material.

❑ Choose **OK**.

❑ Select **Feed Rates**.

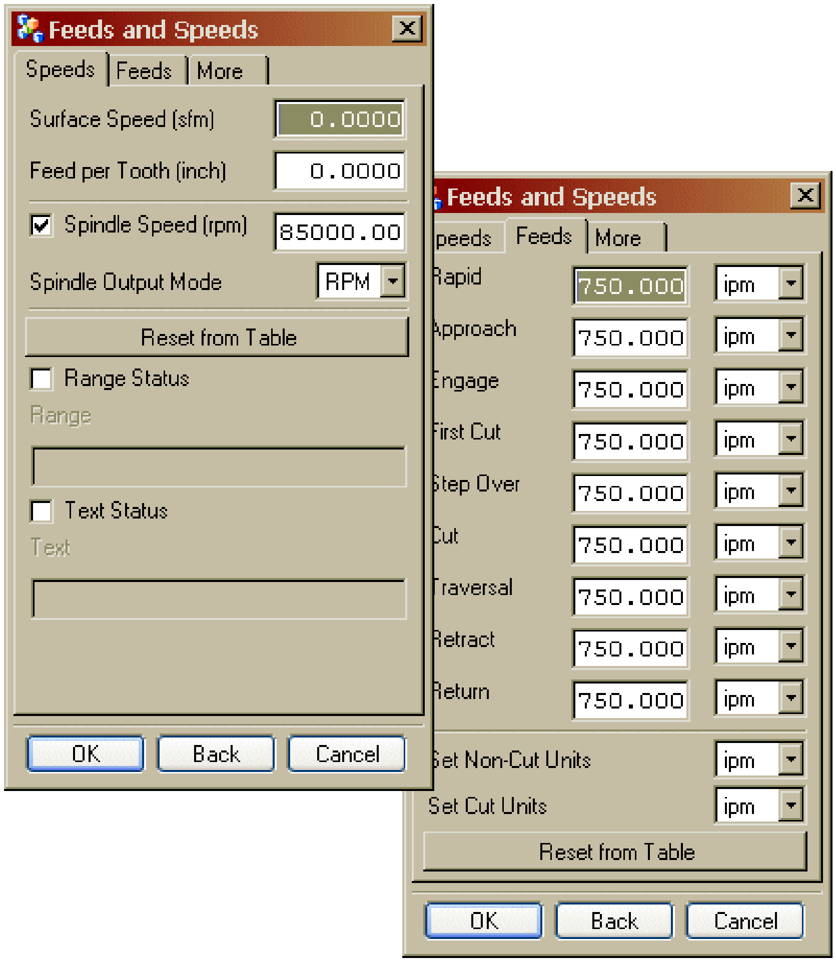


The **Feeds and Speeds** dialog is displayed.



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.

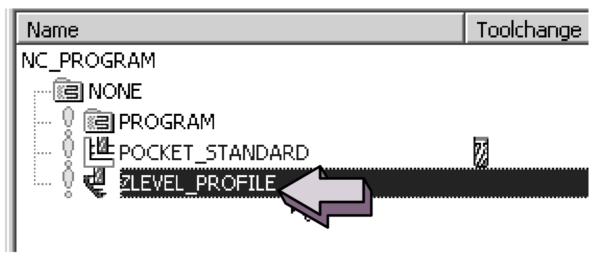
Activity 6—2: Mixed Cut Directions

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.

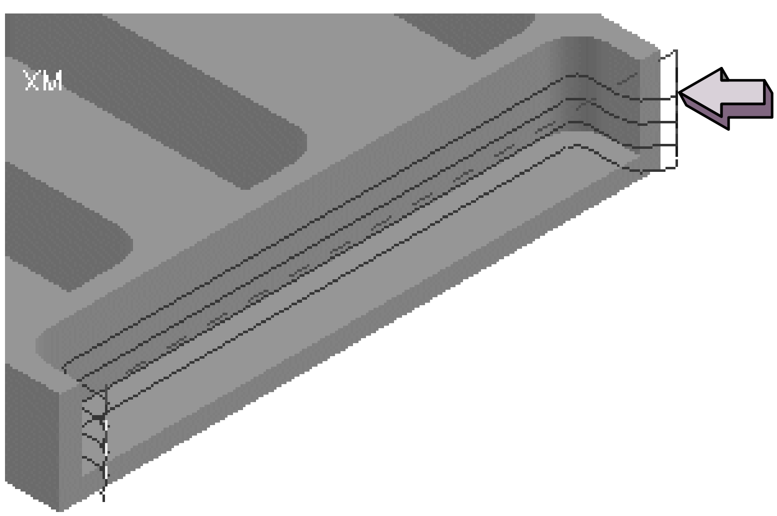
❑ Continue with the part file *****_hsm_1.prt.**

❑ Double click on the operation, **zlevel_profile.**



The **ZLEVEL_PROFILE** dialog is displayed.

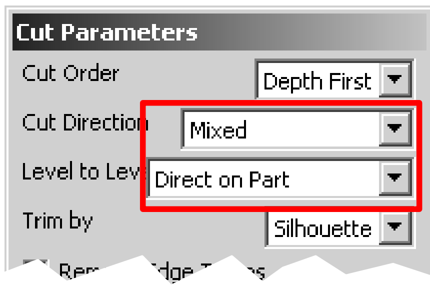
❑ **Replay** the tool path.



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.

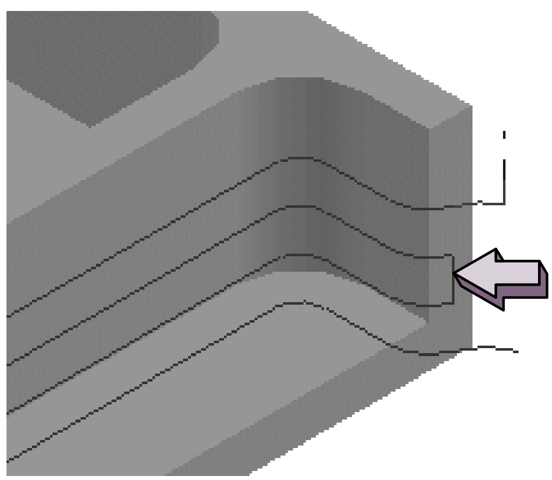
- ❑ Refresh the graphics display.
- ❑ Choose **Cutting**.
- ❑ Set the **Cut Direction** option to **Mixed** and the **Level to Level** option to **Direct on Part**.



- ❑ 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 and Fidia controllers are supported for Nurbs input.

NURBS are available only for fixed axis machining methods of Fixed Axis Surface Contouring, Planar Milling, and Cavity Milling.

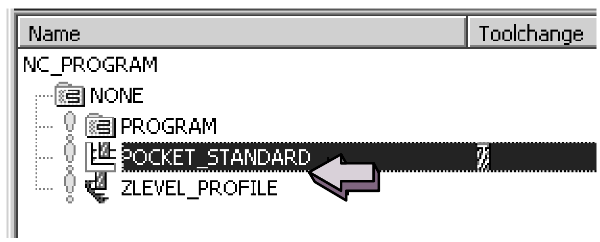
Activity 6–3: NURBS

In this activity, you will use the part file from the previous activity and explore the use of the NURBS option.

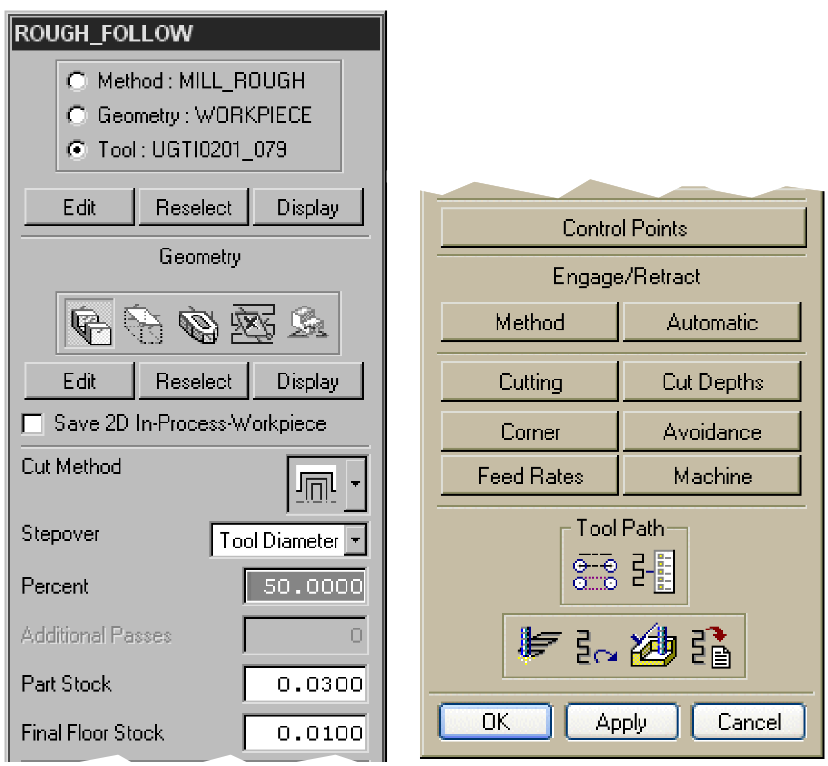
Step 1 Edit the existing operation and modify parameters suitable for the output of NURBS data.

❑ Continue with the part file *****_hsm_1.prt.**

❑ Double click on the operation, **pocket_standard.**

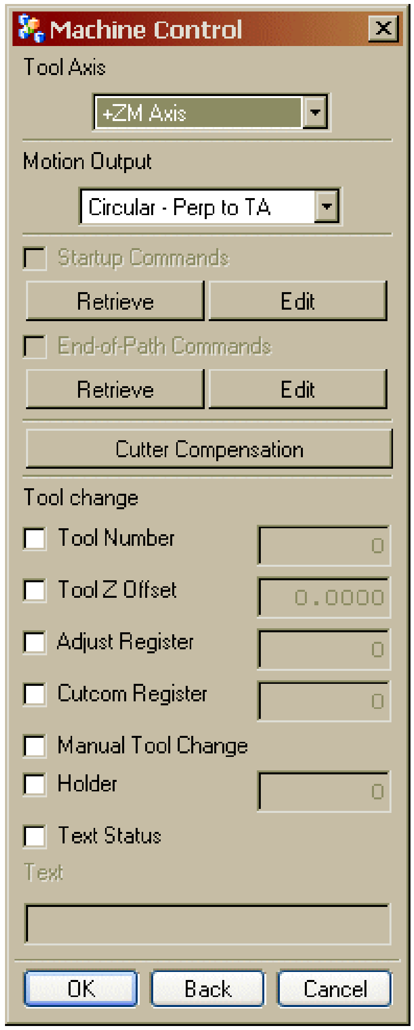


The **ROUGH_FOLLOW** dialog is displayed.

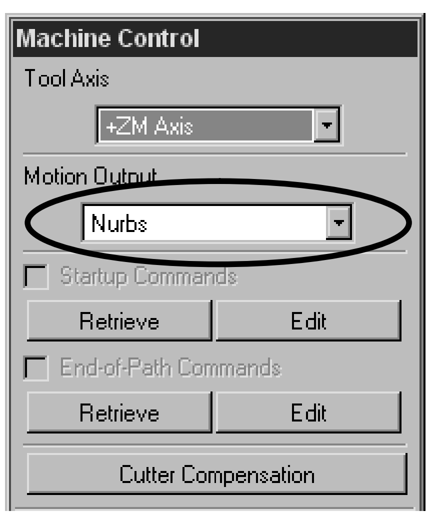


❑ Select the **Machine** button.

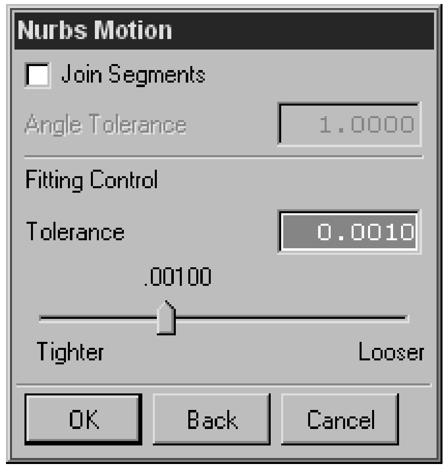
The **Machine Control** dialog is displayed.



❑ Choose **Nurbs** from the **Motion Output** pull down menu.



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.

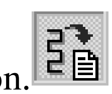
You will accept the other options as defaults.

❑ 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

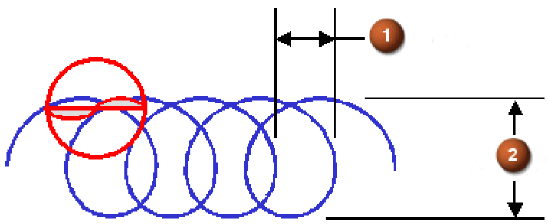
❑ **Close** the part file.

You have completed this activity.

Trochoidal Cut Pattern

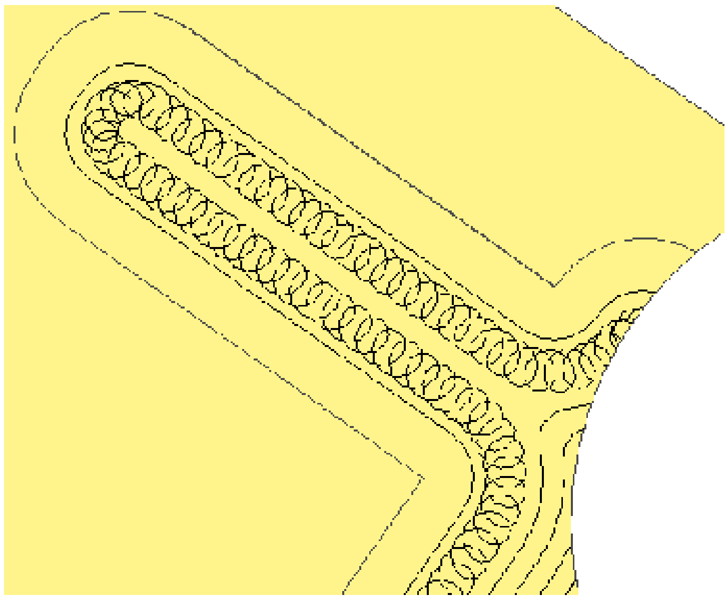
The **Trochoidal Cut** pattern is used when there is a need to limit excessive step over to prevent tool breakage when the tool is fully embedded into a cut and you want to avoid cutting excessive material. Most cut patterns generate embedded regions between islands and parts during the engage process as well as in narrow areas of a part.

The use of **Trochoidal Cut** pattern eliminates this problem by creating a trochoidal cut offset from the part. The tool path cuts along the part and then uses a smooth follow pattern to cut the regions inward. This cutting method can be described as a method of milling where the cutter moves in a circular looping pattern while the center of the circle moves along a path. This is similar in appearance to a stretched out spring.



- (1) Stepover
- (2) Path Width

The following figure illustrates the **Trochoidal Cut** pattern. Note the looping pattern. The cutter machines the material in small looping motions, spinning as it moves in a looping cut pattern. Compare this with the conventional method of cutting where the cutter moves forward in a straight path and is surrounded by material on all sides.



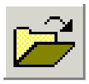
This type of cut pattern is useful in high speed machining applications since this method avoids embedding the tool in material and limits the amount of step over which can occur. This cut pattern is available in Planar, Cavity and Face Milling.

Activity 6–4: Trochoidal Cut Pattern

In this activity you will first examine an existing planar milling operation that uses the Follow Part cut method. You will then change the cut method to Trochoidal, select a different cutting tool and regenerate the operation to observe the changes in the corresponding tool paths.

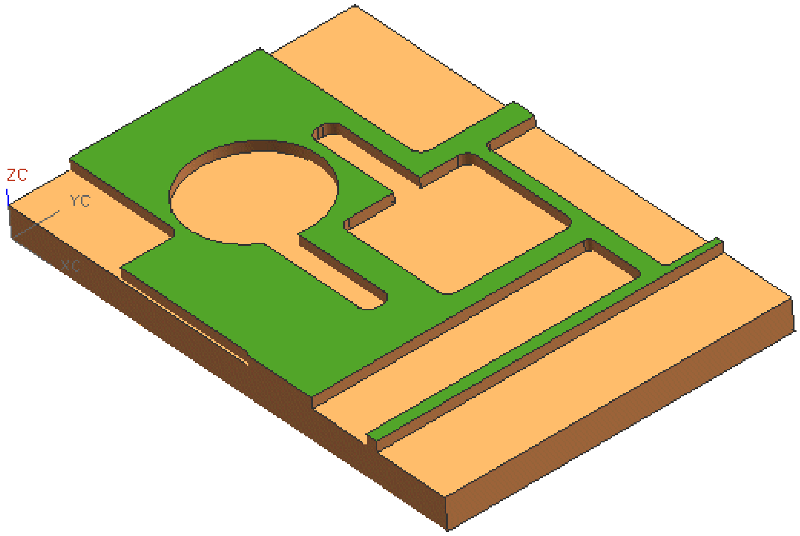
Step 1 Opening the part file ama_trochoidal_cut_method.

❑ If necessary, start Unigraphics NX 2.

❑ Use **File → Open**. 

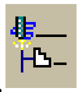
❑ Navigate to your parts folder and open the file.

❑ Briefly examine the part.

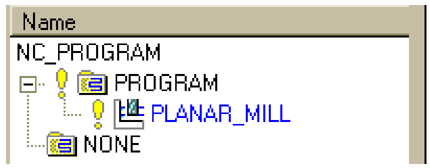


Step 2 If necessary, enter the Manufacturing Application and display the Operation Navigator.

❑ Choose **Application → Manufacturing** from the Menu bar.

❑ Choose the **Operation Navigator** icon from the **Resource** bar. 

❑ If necessary, change to the **Program Order** view of the Operation Navigator and expand the group objects.

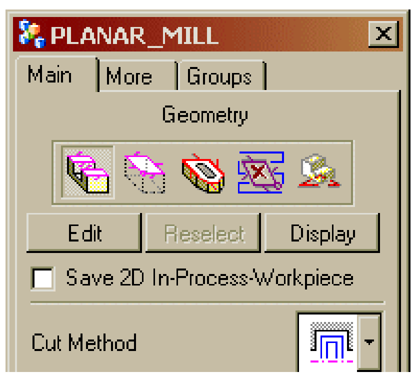


You will notice one operation, **PLANAR_MILL** displayed. This operation will be replayed and then modified by applying the **Trochoidal Cut** method.

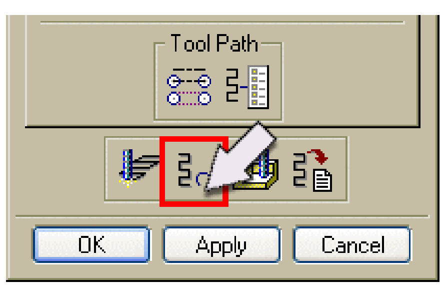
Step 3 Replay and then modify the PLANAR_MILL operation to utilize the Trochoidal Cut method.

❑ Double-click the **PLANAR_MILL** operation.

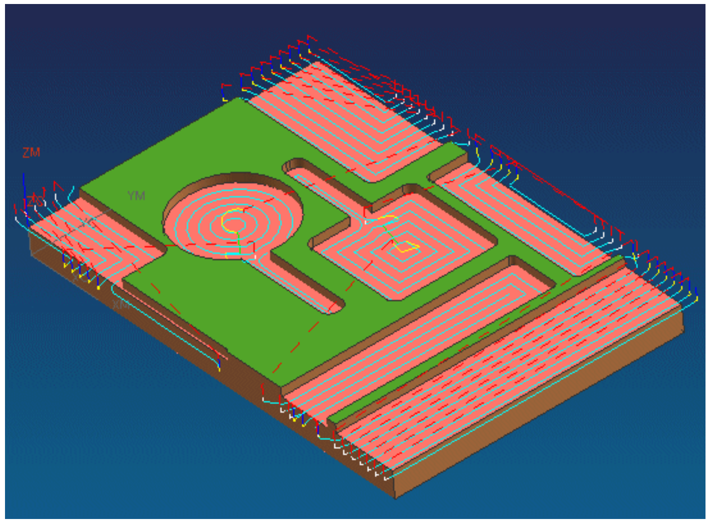
The **PLANAR_MILL** dialog is displayed.



❑ Choose the **Replay** icon.

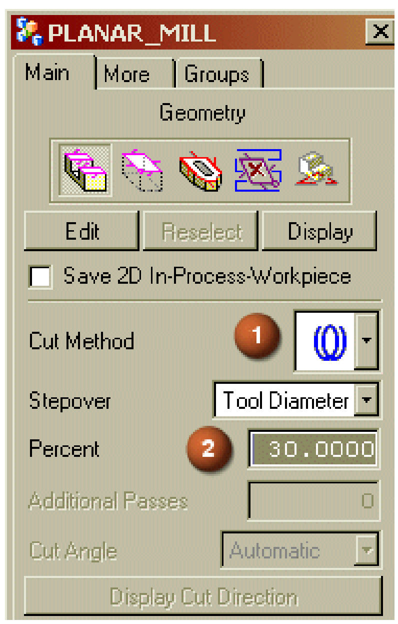


The tool path is displayed.



You will now modify the operation by changing the Cut Method to Trochoidal as well as using a different diameter tool.

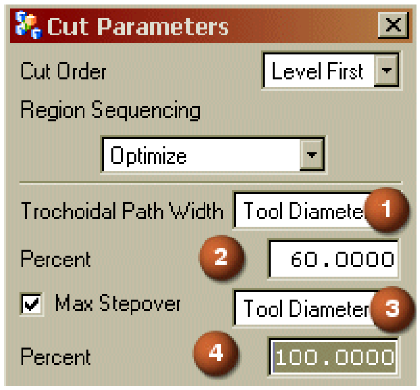
❑ Change the **Cut Method** to **Trochoidal** (1) and the **Stepover Percent** (2) to 30.



❑ Under the **Groups** tab, change the tool form **UGTI0201_012** to **UGTI0201_011**.

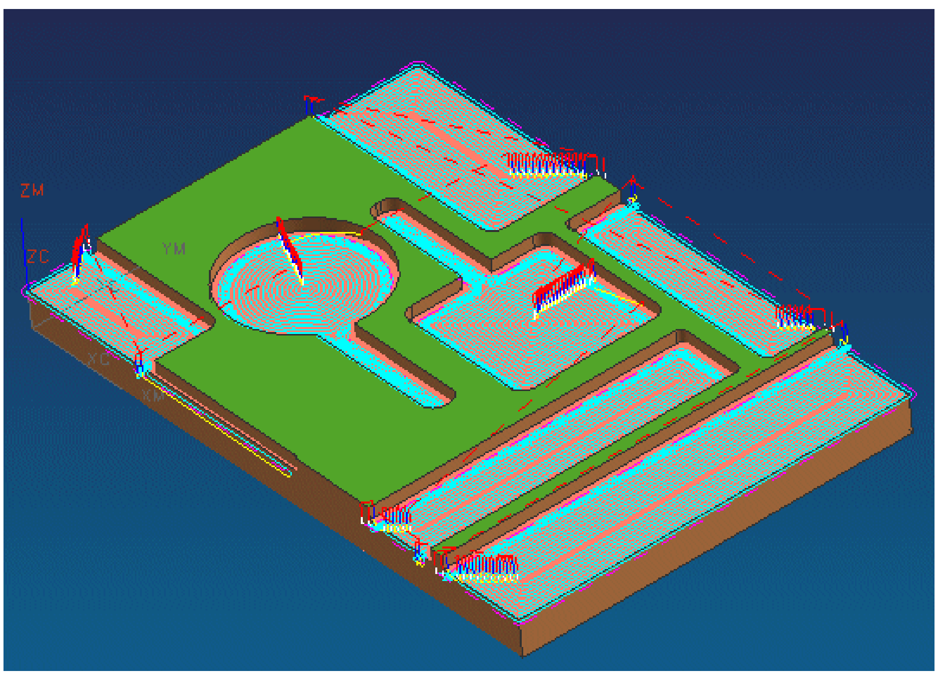
❑ Choose **OK** on the **Reselect Tool** dialog.

- ❑ Under the **Main** tab, select the **Cutting** button and from the **Cut Parameters** dialog, if necessary, change the **Trochoidal Path Width** (1) to **Tool Diameter, Percent** (2) to **60**, **Max Stepover** (3) to **Tool Diameter and Percent** (4) to **100**.



- ❑ Choose **OK** on the **Cut Parameters** dialog.

- ❑ Generate the tool path.



Step 4 Examine in detail, the tool path created using the Trochoidal Cut method.

- ❑ Zoom in on the various cut areas and examine the tool path in detail.

- ❏ **Close** the part file without saving.

  This concludes the activity and the lesson.

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
- Used the Trochoidal Cut method to generate tool paths that avoids embedding the tool in material and limits the amount of step over
- Explored parameters and techniques for generating NURBS output

EDS

NC Assistant

Lesson 7

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:

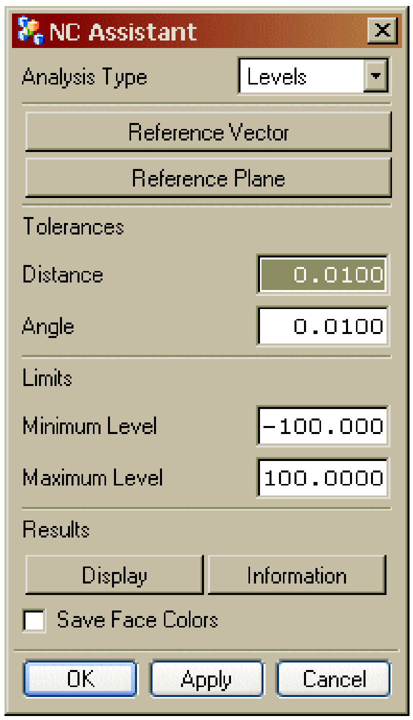
- Use the NC Assistant
- Determine cutter geometry based on information feedback from the NC Assistant

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 reference. 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 proper tool diameter(s).

Analysis of **Fillet Radius** displays 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).

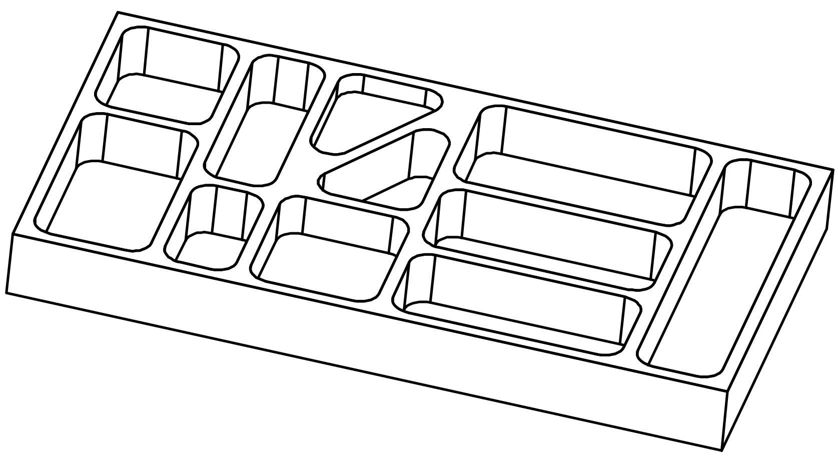
When analyzing the various types, limits can be set. For example, if you wanted to check for all corner radii that were greater than .500 inches and less than .750 inches, values can be set for the minimum of .500 and maximum of .750.

Activity 7—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.**



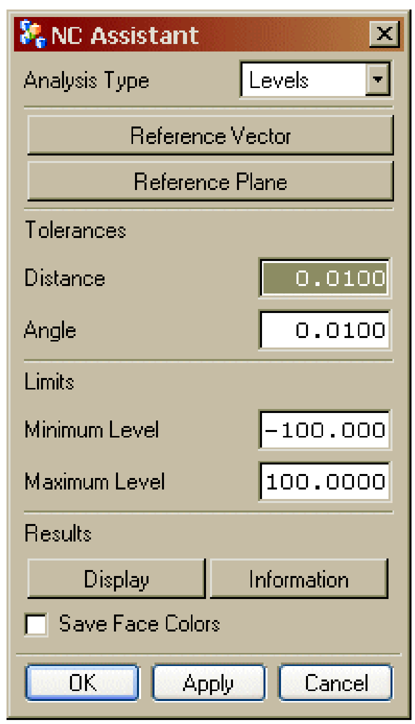
❑ Enter the **Manufacturing** Application.

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.



7

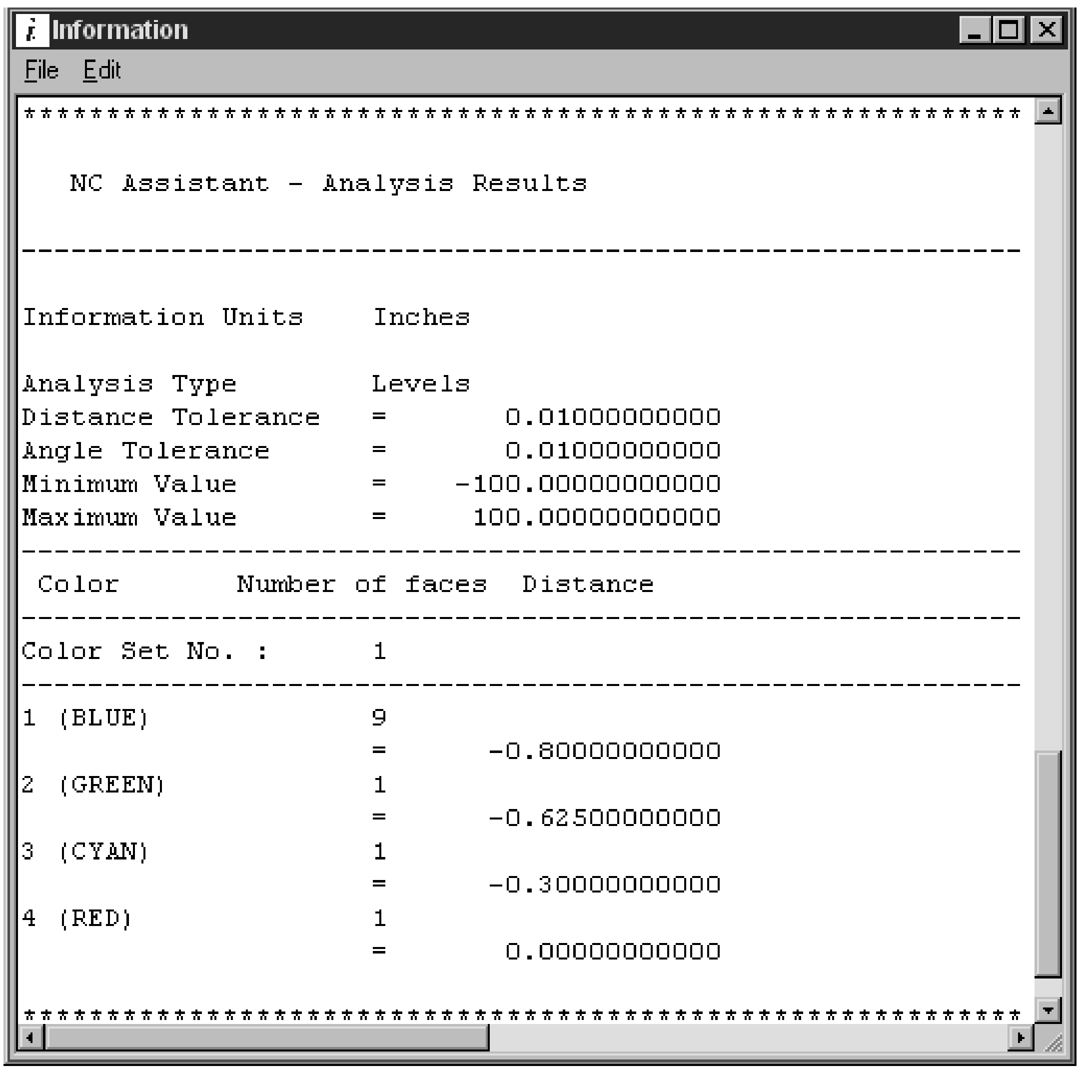
Step 3 Use the NC Assistant to determine cutter length.

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 face selection.

❑ Use **MB1** and drag a rectangle around the entire part.

❑ Choose **Apply**.

An Information Window is displayed containing the results of the Level analysis.



```
Information
File  Edit
************************************************************

   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

************************************************************
```

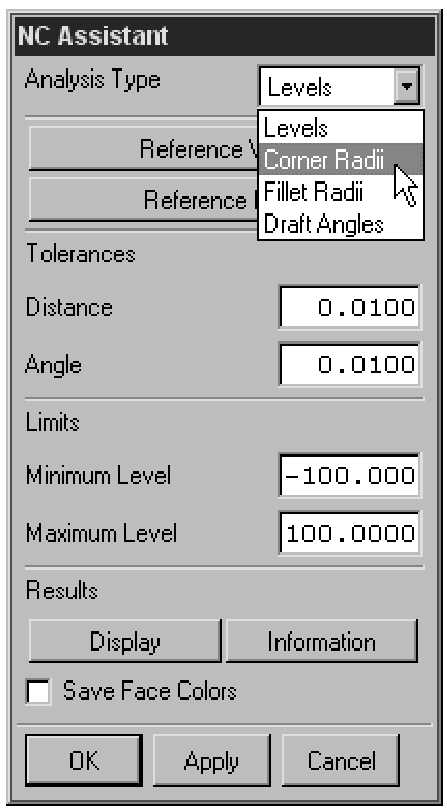
Notice the color set number, the colors associated with the color set number, number of faces and distance. Compare the colors with those now displayed in the graphics window. By examining the distance values, the deepest level or floor is located .800 below the top of the part (blue). Therefore the 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.

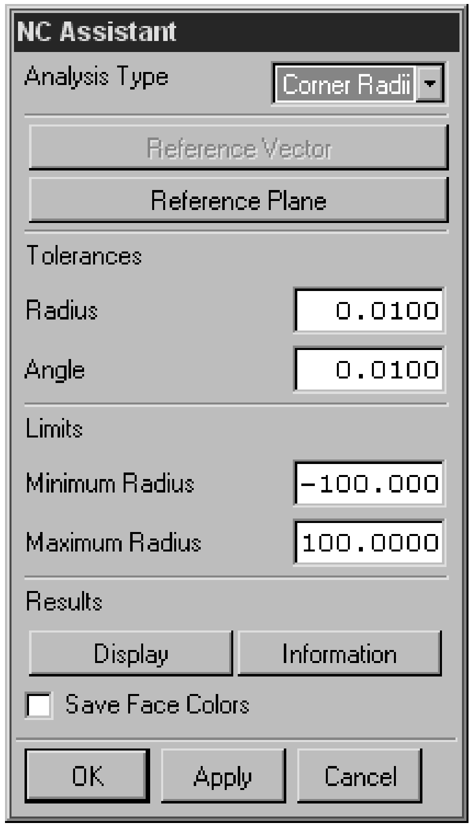
The second item for consideration is to determine the diameter of the cutter that will be required. For this determination you will use the NC Assistant to determine the various corner radii of the part.

7

❑ Choose **Corner Radii** from the **Analysis Type** pull down menu.



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.

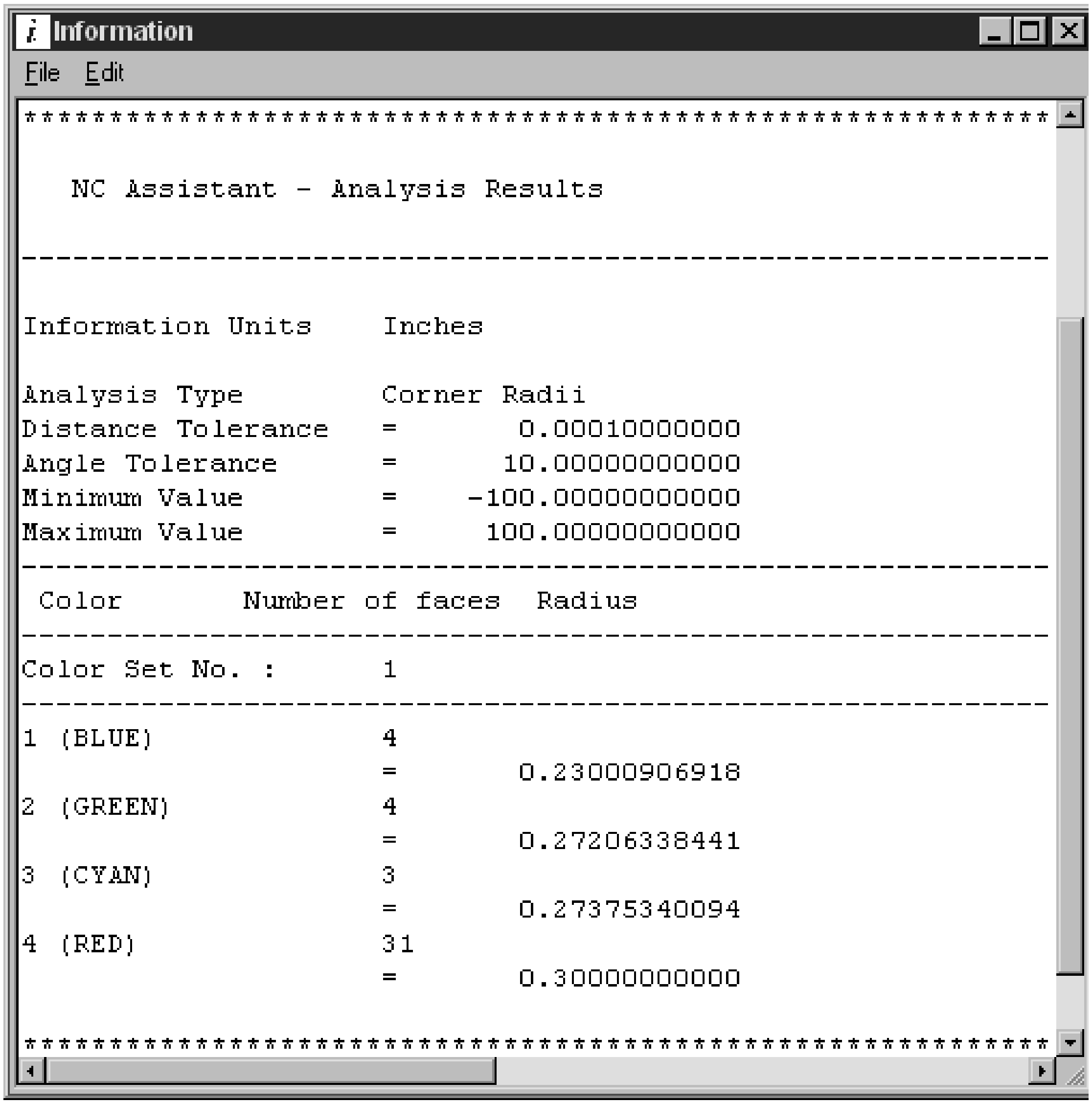
❑ Change the **Radius** value to **.0001.**

❑ 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.



Information

File Edit

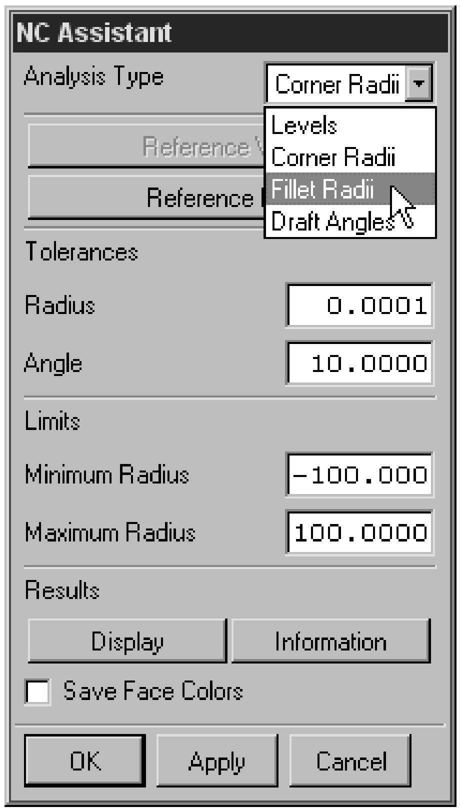
```
***************************************************************


   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 colors 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.

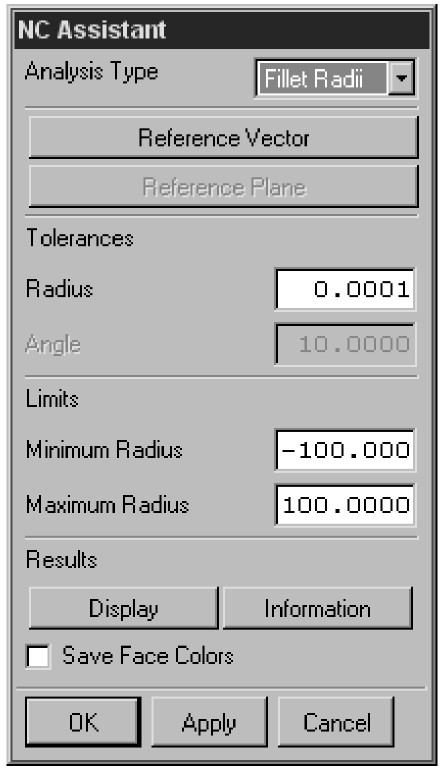
Step 5 Use the NC Assistant to determine cutter corner radius.

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**.

7

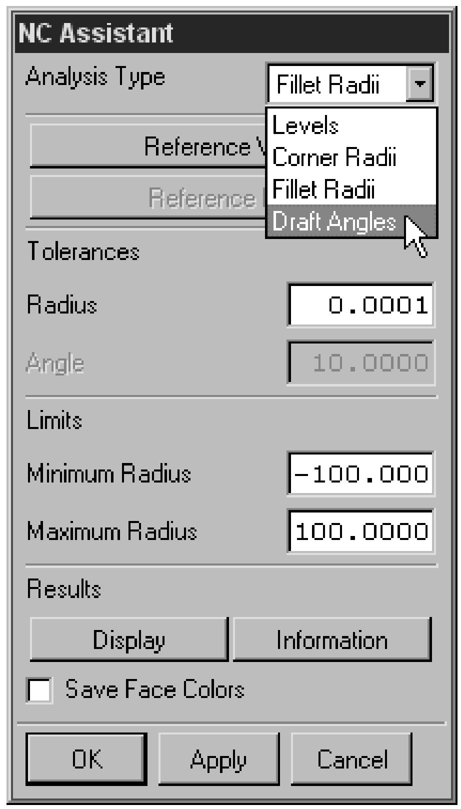
An **Information Window** is displayed containing the results of the Fillet Radii analysis.

Notice that there were no Fillet Radii displayed.

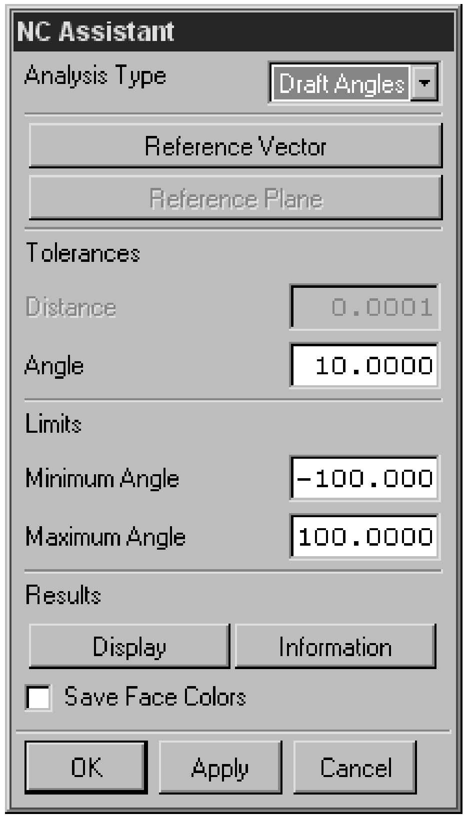
Step 6 Use the NC Assistant to determine draft angle on walls.

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.



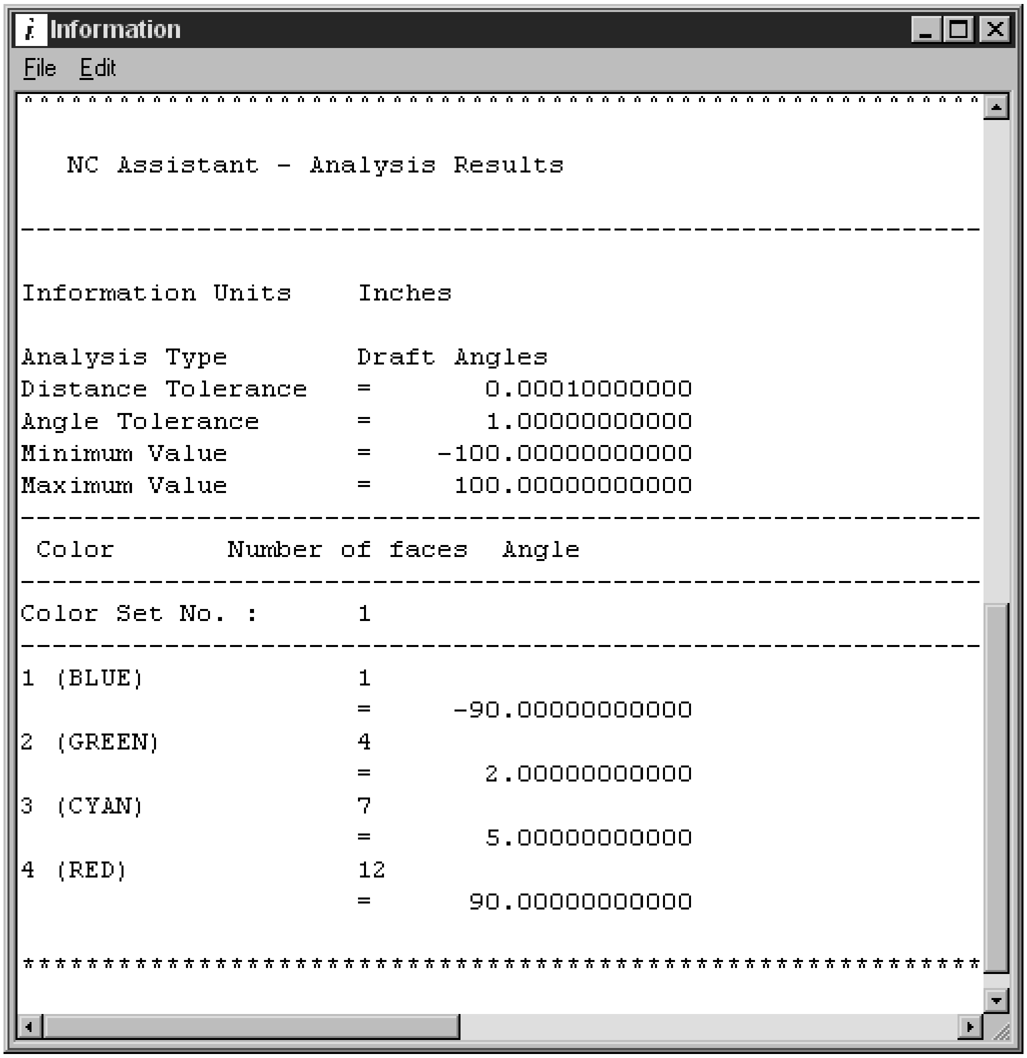
If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will change the **Angle** tolerance to 1.0 degree, accept all other defaults and select the entire part for faces.

❑ 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.




```
************************************************************

   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 colors 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.

7

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 Assistant.
- Performed various analysis functions which were used to determine cutter length, diameter, corner radius and draft angle

Templates

Lesson 8

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 and allows you to customize your programming environment to your shop standards and procedures.

OBJECTIVES

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

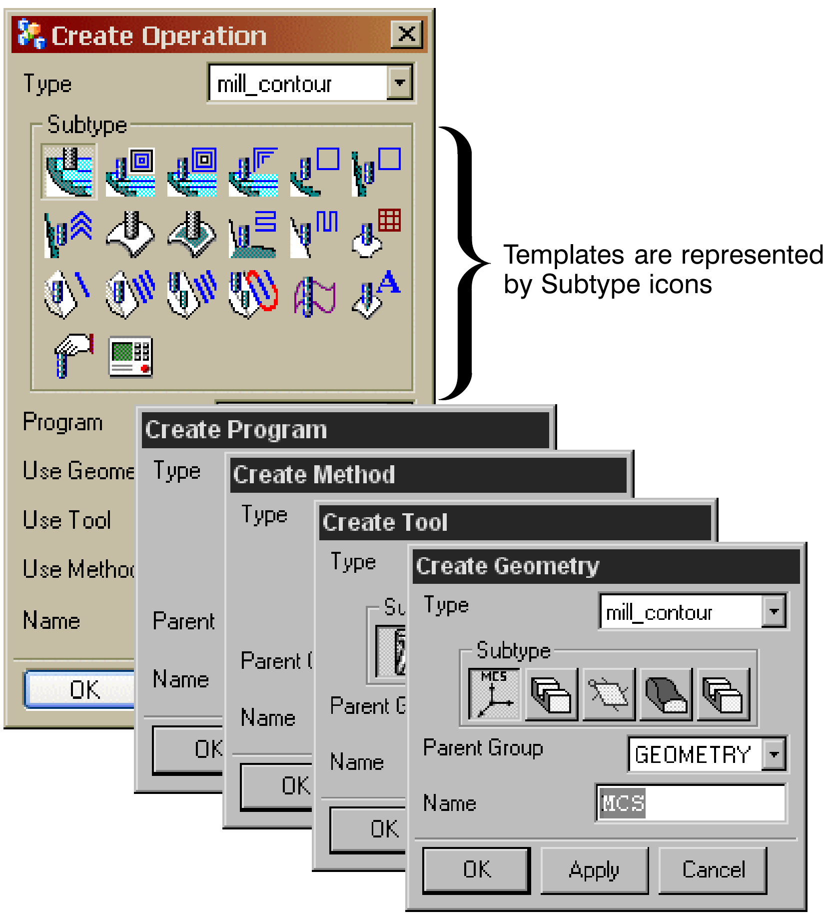
- Understand the function and mechanism of templates
- Create and use templates
- Use a sequence of operations template to create numerous operations automatically for similar parts

This lesson contains the following activities:

Activity **Page**

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 determine the initial setup and control the creation of operations, tools, programs, geometry and methods group objects. They also reduce the laborious procedure of redefining parameters from a standard set of defaults each time a new operation or group is defined.

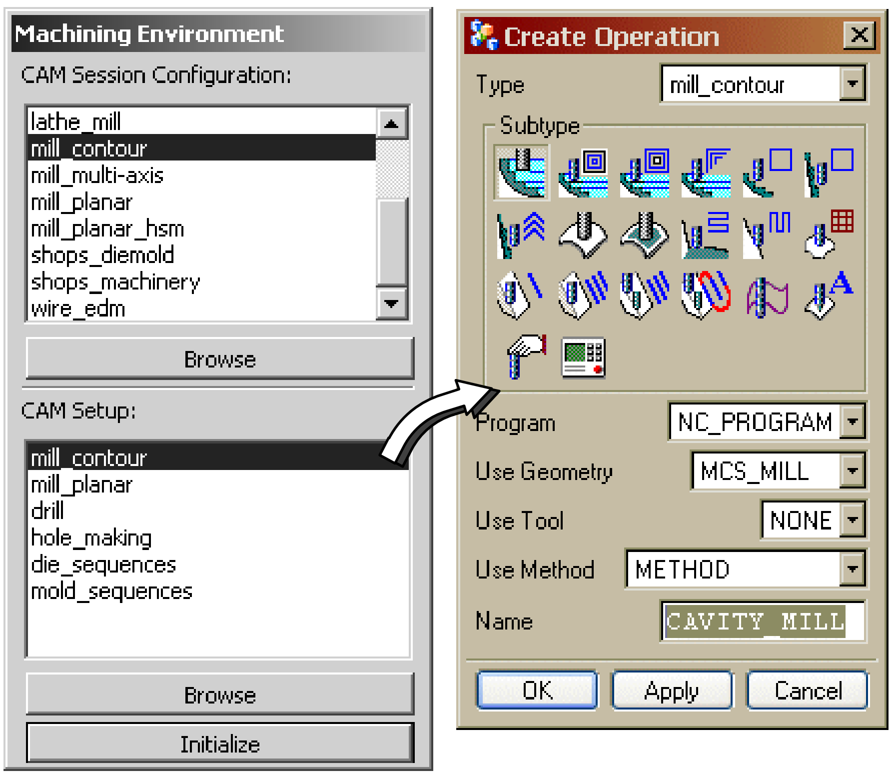


Templates are represented as **Subtype** icons which are available in the various **Create** dialogs (also referred to as object dialogs). They define operation parameters including numerical values such as part Intol/Outol, feed rates, etc. and can display customized dialogs.

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/or groups (templates) for selection. **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.

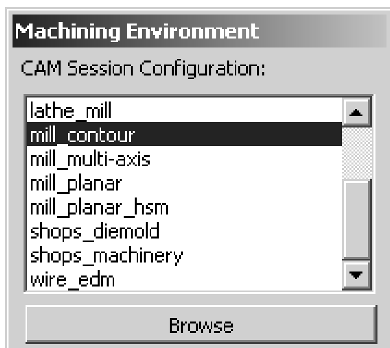


In the above example, **mill_contour** is a **template part file** containing predefined operations and groups which are represented by **Subtype** icons.

Template Sets

A **Template Set** is a collection of **template part files**. They are specified from within the **Configuration** files. Basic **Template Sets** are provided and maintained in each software release and/or may be created, customized and maintained by users. **Template Sets** created by users may use any valid naming convention, may be located in any directory and must have a **.opt** file extension. Supplied **Template Sets** reside in the **\mach\resource\template_set** directory.

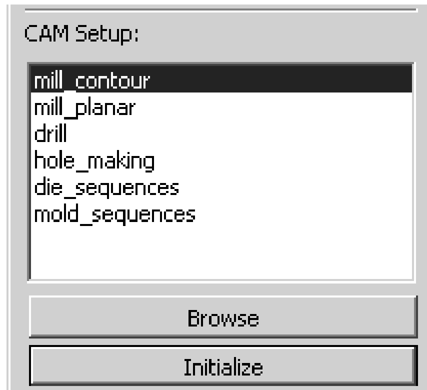
The **CAM Session Configuration** scrollable window lists files with ".dat" extensions containing the location of template files for operations, documentation, post processors and libraries.



These files are located in the **\mach\resource\configuration** directory. For example, part of **mill_contour.dat** contains the following:

```
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
```

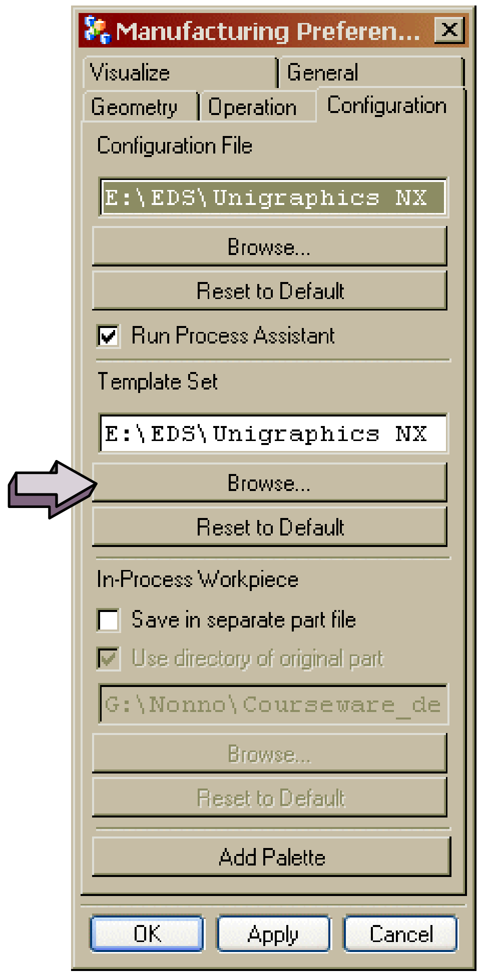
The **CAM Setup** scrollable window lists **Template Set** files with ".opt" extensions containing a list of part files with parameters used to generate the option.



These files are located in the **\mach\resource\template_set** directory. For example, **mill_contour.opt** contains the following:

```
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}mill_contour.prt
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}mill_planar.prt
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}drill.prt
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}hole_making.prt
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}die_sequences.prt
${UGII_CAM_TEMPLATE_PART_ENGLISH_DIR}mold_sequences.prt
##
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}mill_contour.prt
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}mill_planar.prt
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}drill.prt
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}hole_making.prt
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}die_sequences.prt
${UGII_CAM_TEMPLATE_PART_METRIC_DIR}mold_sequences.prt
```

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.

Creating and Using Template Sets

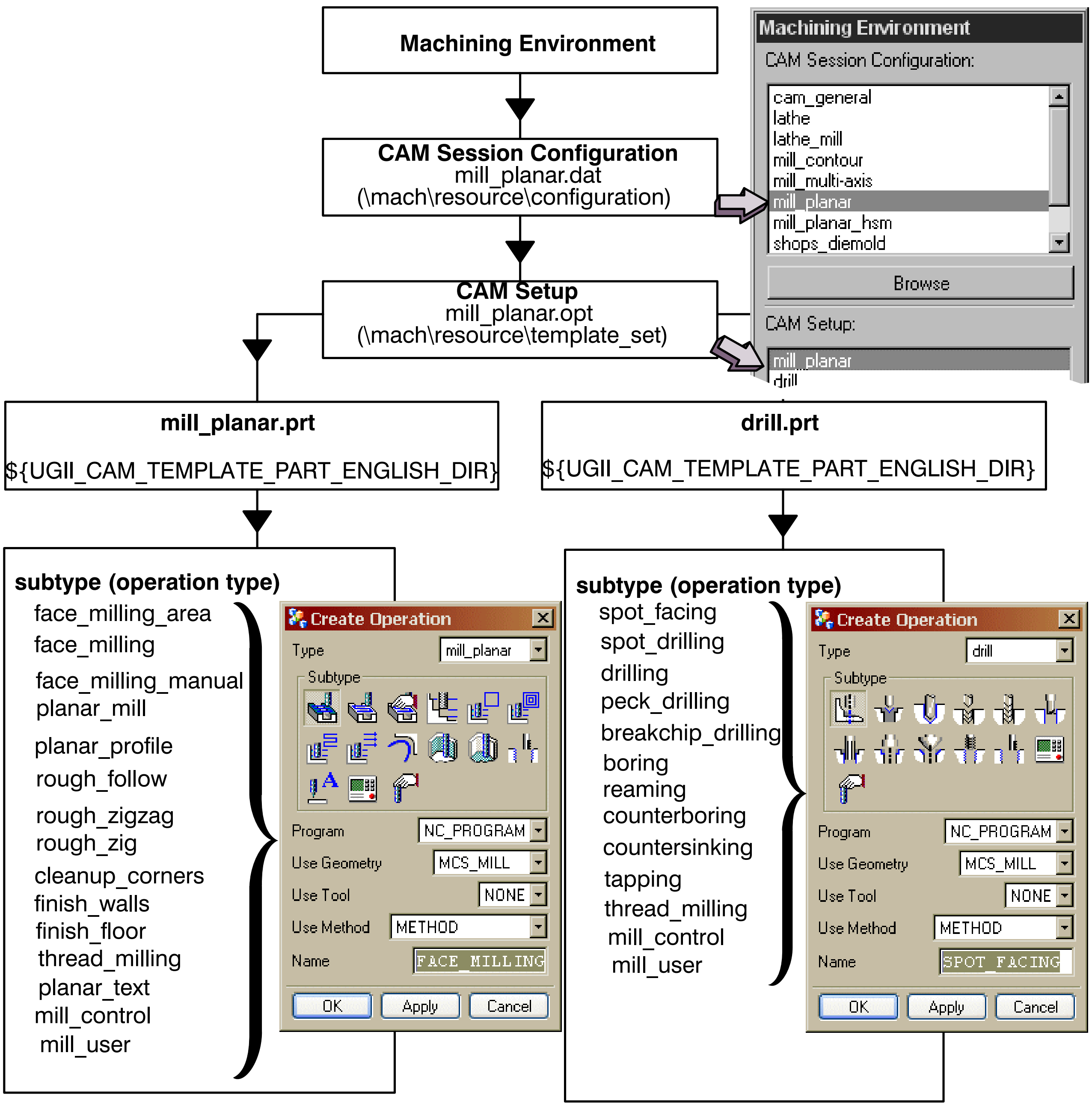
Template Sets make template part files available for selection from within dialogs. This prevents you from having to browse for individual part files. A Template Set is an text file (.opt extension) which list the part files which contain the templates that are to be utilized. The default **Template Set** is determined by the **Configuration** file.

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 when creating operations using the template (verify 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)



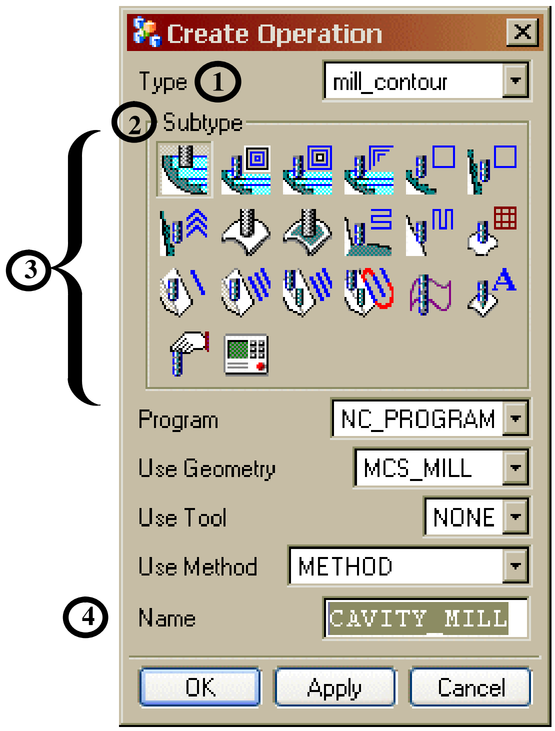
8

The **Type** option (1) corresponds to the template file name.

The **Subtype** (2) option corresponds to the operation name within the template file.

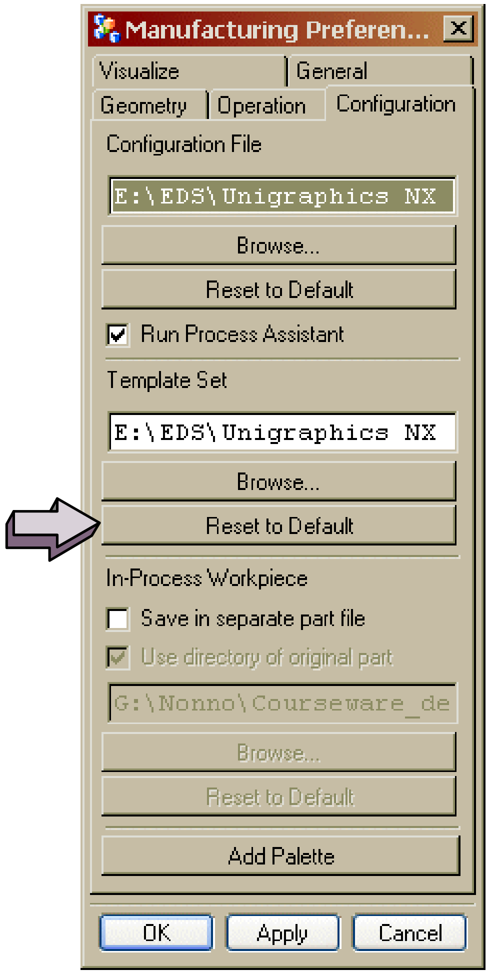
The selected **Type** (3) determines the **subtype** options (icons) that are available (note that the subtype icon file names must match the subtype name).

The **Name** field indicates the name of the operation or group object created.



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 a 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 to Default** button in the **Manufacturing Preferences Configuration** tab dialog. The default **Template Set** is determined by the **Configuration** file.

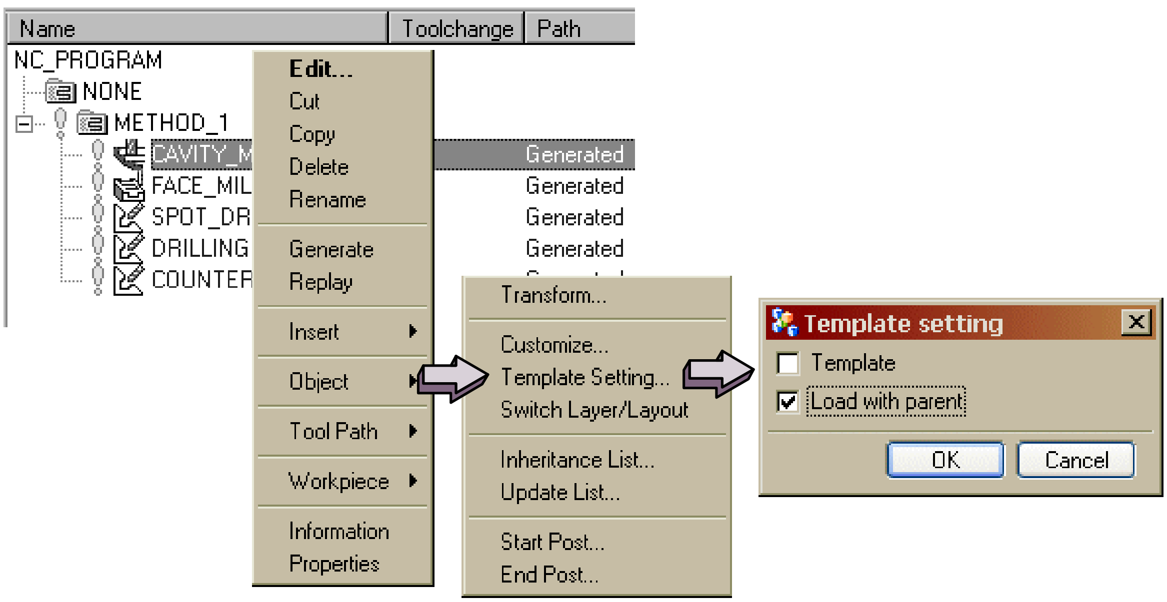


8

Creating a Template

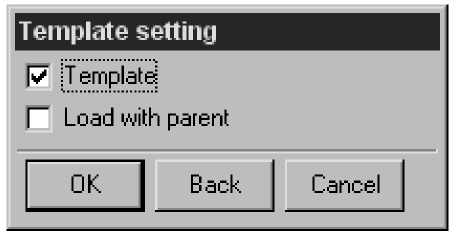
As discussed earlier, a template is a predefined operation or group within a part file. Template files can be new or can be copied, customized and/or renamed from existing template files.

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.

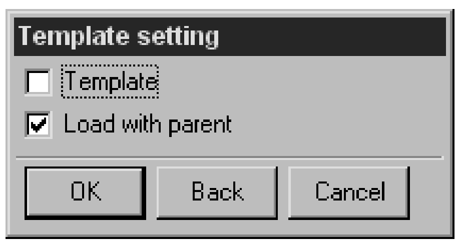


Highlight the desired operations and/or groups in the Operation Navigator and with **MB3** choose **OBJECT** → **Template Setting.** This will display the **Template setting** dialog which contains the **Template** and **Load with parent** toggle settings.

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 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.

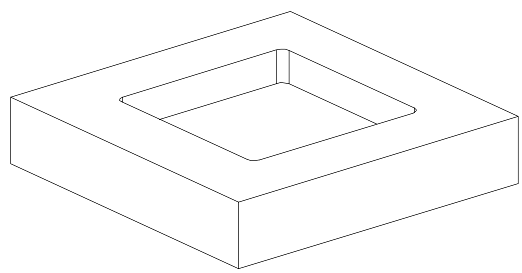
The **Load with parent** option also allows the determination of which operations and groups in a **template part** file will appear in the Operation Navigator when the template part file is specified as a **Setup** in the **Machining Environment** dialog. The **Setup** is the initial **template part** 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 8–1: 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 activity which follows. This activity establishes the template file settings.

Step 1 Open and rename a part file.

❑ Open **ama_single_pocket.prt.**

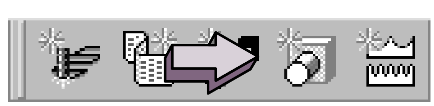


❑ **Save As ***_single_pocket .prt** where *** represents your initials.

❑ Enter the **Manufacturing** Application.

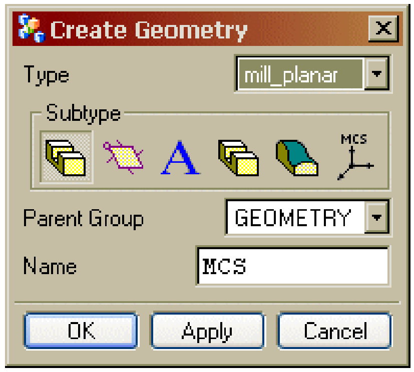
This part file contains several tools and blank geometry which surrounds the part. The **Configuration** used is **cam_general** and the **Setup** is **mill_planar**.

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



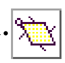
8

The **Create Geometry** dialog is displayed.



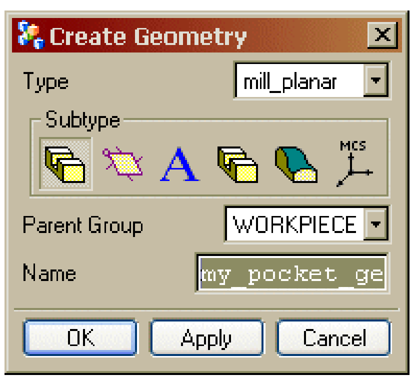
❑ Make sure the **Type** is set to **mill_planar.**

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.

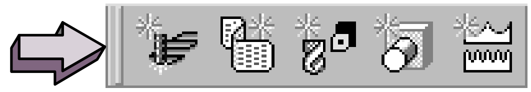
❑ Change the **Name** to **my_pocket_geom**.



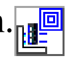
❑ Choose **OK** twice to exit all dialogs.

Step 3 Create three template pocketing operations.

❑ Choose the **Create Operation** icon from the **Create** tool bar.

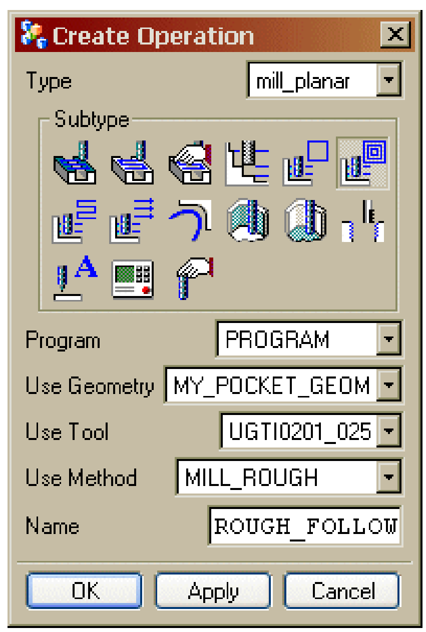


The **Create Operation** dialog is displayed.

❑ Choose the **Rough_Follow** icon. 

❑ Set the following:

- Program: **PROGRAM**
- Use Geometry: **MY_POCKET_GEOM**
- Use Tool: **UGTI0201_025**
- Use Method: **MILL_ROUGH**



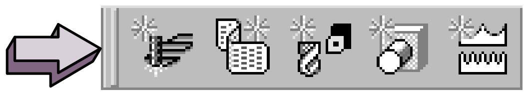
❑ Choose **OK** in the **Create Operation** dialog.

You do not need to create a tool path for this **Template** part file. You will create a tool path in a future activity, using different geometry with this template operation.

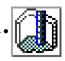
❑ Choose **OK** in the **Rough_follow** dialog.

You will now create a template operation for finishing the pocket floors.

❑ Choose the **Create Operation** icon from the **Create** tool bar.



The **Create Operation** dialog is displayed.

❑ Choose the **Finish_Floor** icon. 

❑ Change the **Tool** to **UGTI0201_012.**

❑ Change the **Method** to **MILL_FINISH**.

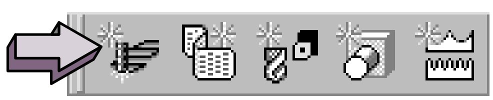
❑ Choose **OK** in the **Create Operation** dialog.

Remember that you do not need to create a tool path in a template part file.

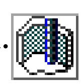
❑ Choose **OK** in the **Finish_Floor** dialog.

Next, you will create an operation for finishing the pocket walls.

❑ Choose the **Create Operation** icon from the **Create** tool bar.



The **Create Operation** dialog is displayed.

❑ Choose the **Finish_Walls** icon. 

❑ Change the **Tool** to **UGTI0201_011.**

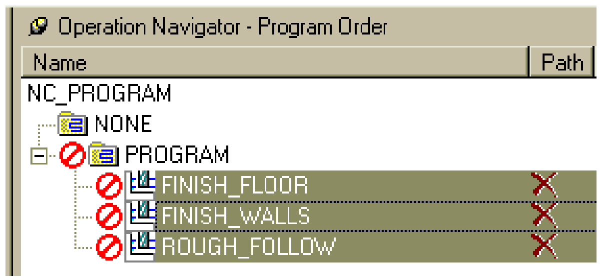
❑ Choose **OK** twice.

You have just created a simple sequence for machining 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.

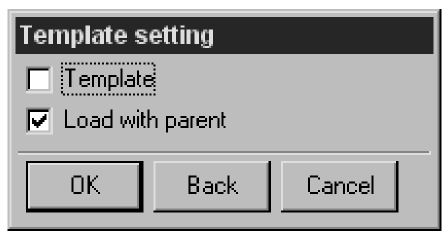
Step 4 Making the pocket sequence available in other part files.

You will want the template operations that you have just created to be available for use in another part. These operations will be available 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.**

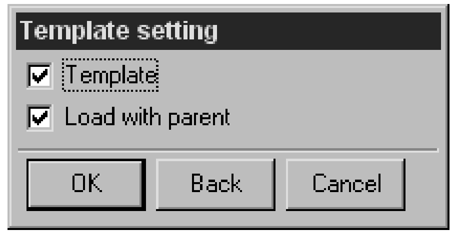


❑ Choose **OK.**

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. For this to occur, the parent group, **MY_POCKET_GEOM**, must have the **Template setting**, **Load with parent** toggled **On**.

❑ 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.

8

Template Review

- **Template Parts**
  - determine the initial **Setup**
  - control the creation of operations, tools, programs, methods and geometry groups
  - the **Setup** is only selected once in a part file
  - the **Setup** template part determines what is created in the work part when the **Setup** is initialized
  - the **Setup** template part becomes the first **Type** in the create dialog
  - the **Type** is the template part for creating objects
  - the **Type** template part controls what can be created in the work part
  - the **Type** can be changed during a programming session
- **Template Settings**
  - If **Load with parent** is active, then:
    - if parent is created, so is the object
    - any group object only has one parent
    - any operation, has four parents and:

    (1) the **Geometry** parent is the "**Load with**" parent

    (2) if **Tool**, **Method** or **Program** are specified they are created if not found in the work part

    (3) template used determines if the tool is created with the operation
  - If **Template** is active, then:
    - this object can be used as a template
    - controls the icons created on the **Create** dialogs
    - customized dialog and inheritance list are included

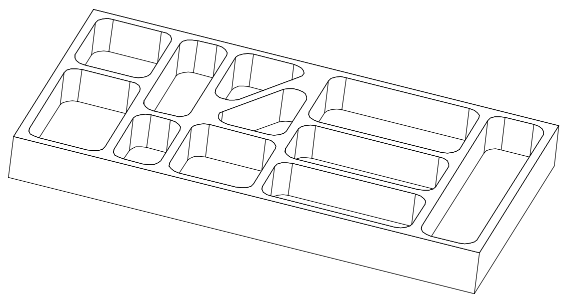
- **Template Generalities**
  - for performance purposes, templates are opened and read only the first time that they are required
  - once a template part has been browsed, it remains on the list of **Setups** or **Types** for the current session and will not be opened again, even if it has changed
  - when editing template parts, reset the **Configuration** to force the updated template files to be read when you select or browse for them

Activity 8—2: Using a Template

In this activity, you will use the part file that you 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 and rename it.

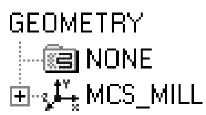
- ❑ Open **ama_multi_pocket.prt.**



- ❑ **Save As ***_multi_pocket.prt** where *** represents your initials.
- ❑ Choose **Application** → **Manufacturing**.
- ❑ Choose the **CAM Session Configuration** as **cam_general.**
- ❑ Choose the **CAM Setup** as **mill_planar**.
- ❑ Choose the **Initialize** button.
- ❑ In the **Program Order** view of the Operation Navigator, note that no operations are listed.

❑ Change to the **Machine 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**.

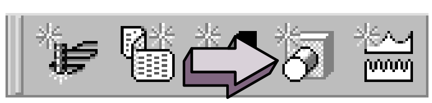


Since you chose **mill_planar** as the **Setup** file, it is the current template being used for this part.

Step 2 Change template files.

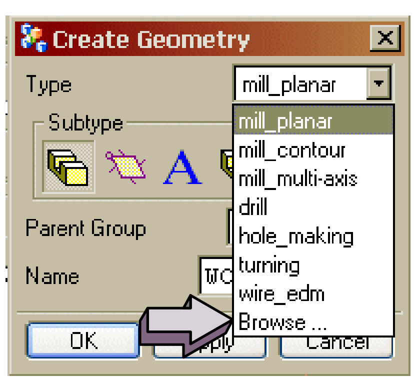
Another method of choosing a template is to change the **Type** in the create dialogs. When using this method you are limited to the options that are available in the template file.

❑ Choose the **Create Geometry** icon from the **Create** tool bar.



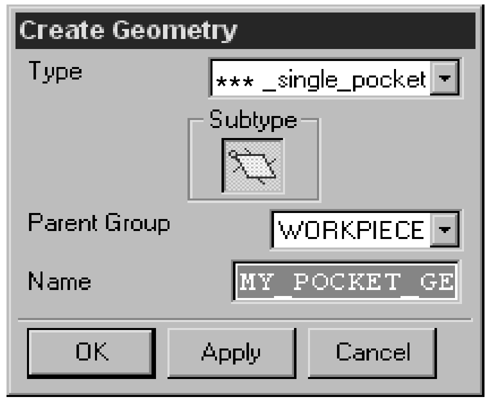
The **Create Geometry** dialog is displayed.

❑ In the **Create Geometry** dialog, next to the **Type** label, change **mill_planar** to **Browse** and choose *****_single_pocket.prt**.



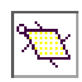
❑ Choose **OK** in the **Template part** dialog.

Notice that the only available **Subtype** is **MY_POCKET_GEOM**. This is a 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. 

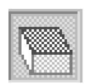
❑ Choose **WORKPIECE** as the parent group.

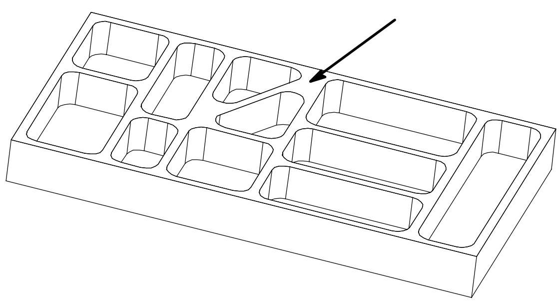
❑ Choose **OK**.

The **MY_POCKET_GEOM** dialog is displayed.

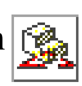
❑ In the **MY_POCKET_GEOM** dialog, choose the **Part** icon and then **Select**.

❑ Choose the Material side as **Outside.**

❑ Using the **Face Boundary**  method, select the top face of the part.



❑ Choose **OK**.

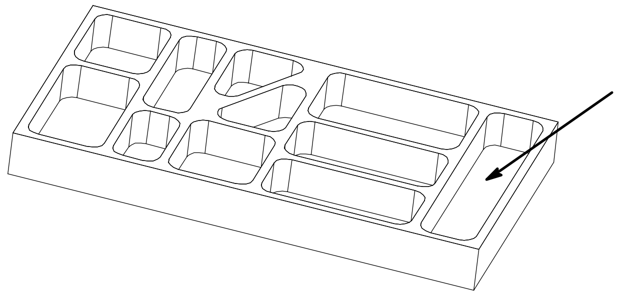
❑ Choose the **Floor** plane icon  and then **Select.**

The **Plane Constructor** dialog is displayed.

❑ Set the **Filter** to **Face**.

❑ Select the floor of any of the pockets.

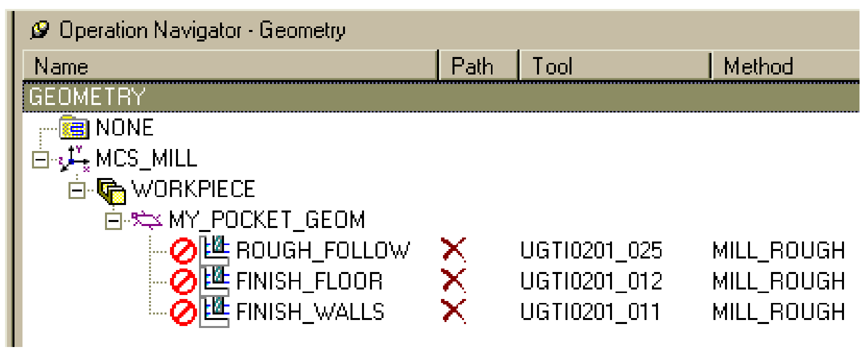
8



❑ Choose **OK** twice.

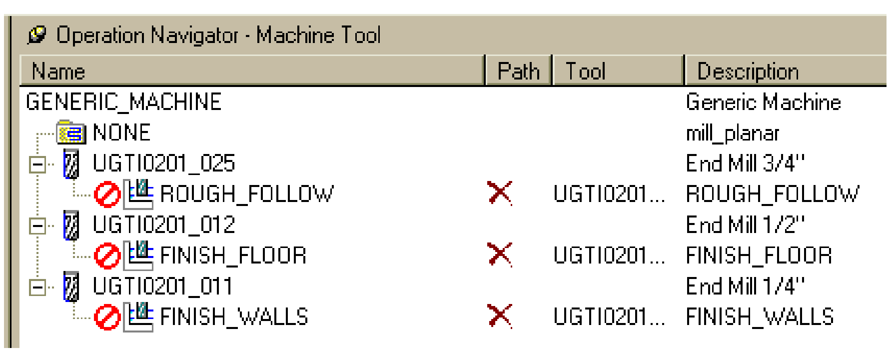
You have specified all of the information needed to generate the tool paths.

❑ If necessary, change to the **Geometry** view of the Operation Navigator and expand all parent groups.



Note the three operations created from the template to rough and finish the part.

❑ If necessary, change to the **Machine Tool** view of the Operation Navigator and expand all parent groups.



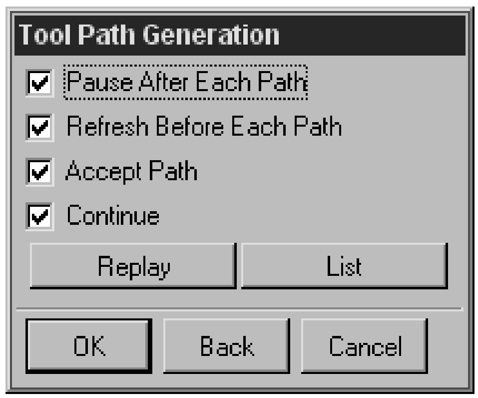
Note the three tools created from the template used in roughing and finishing the part.

Step 4 Generating the tool paths.

❑ Change to the **Geometry** view of the Operation Navigator.

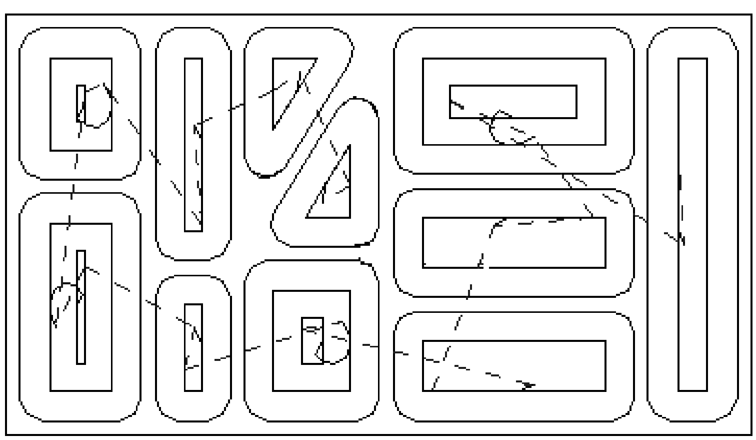
❑ Choose the **MY_POCKET_GEOM** group object and using **MB3, Generate** the tool paths.

The **Tool Path Generation** dialog is displayed.

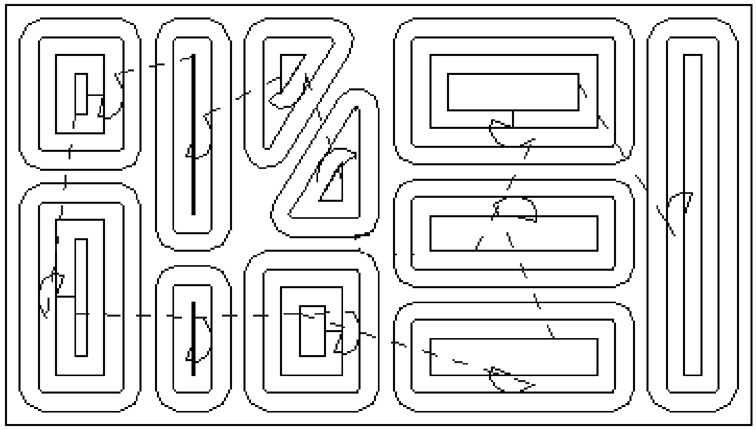


❑ Choose **OK** as necessary to continue generating the tool paths.

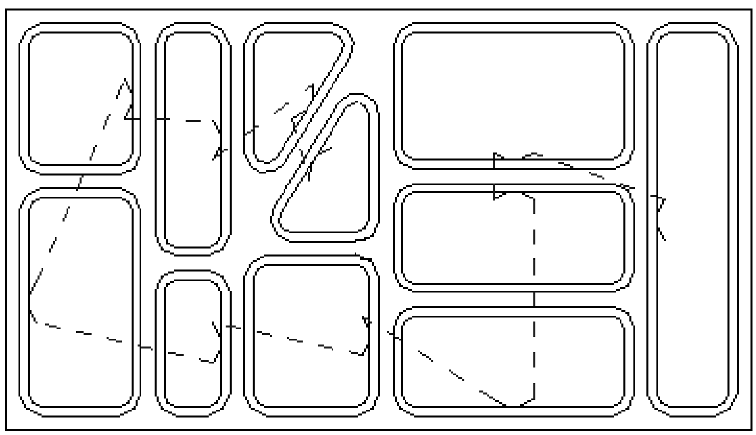
The **ROUGH_FOLLOW** operation creates a tool path similar to the following illustration:



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.

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 toggled 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** group object in the **Create Geometry** dialog (created by the template) which was used to choose the part and floor geometry.
- Created three pocketing operations by selecting the part and floor geometry. Most of the effort in creating these operations was through the use of templates.

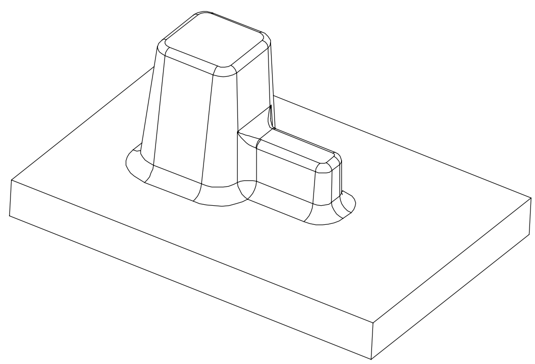
In the next activity you will use a die machining sequence incorporating, roughing and finishing operations to machine a die set.

Activity 8—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**.**

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

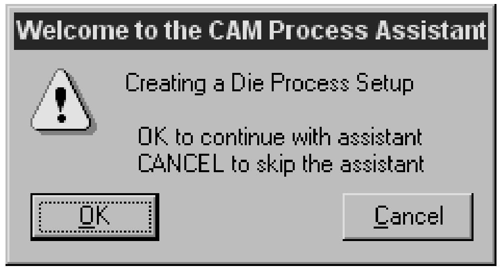
❑ Choose the **CAM Session** Configuration **mill_contour**.

❑ Choose the **CAM Setup** template **die_sequences**.

This is the template that contains the die machining sequence.

❑ Choose **Initialize**.

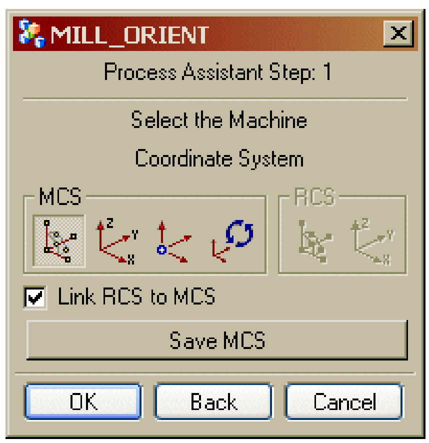
The **CAM Process Assistant for Die Machining is displayed**.



The Process Assistant guides you in selecting the geometry for the machining sequence.

❑ Choose **OK.**

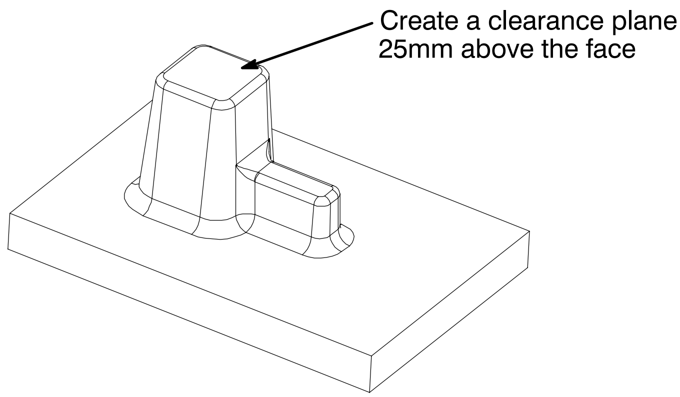
The **Process Assistant Step: 1** dialog is displayed, asking for the selection of the **MCS**. By selecting **OK**, the **MCS** is set to the **WCS**.



❑ 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 25mm 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**.

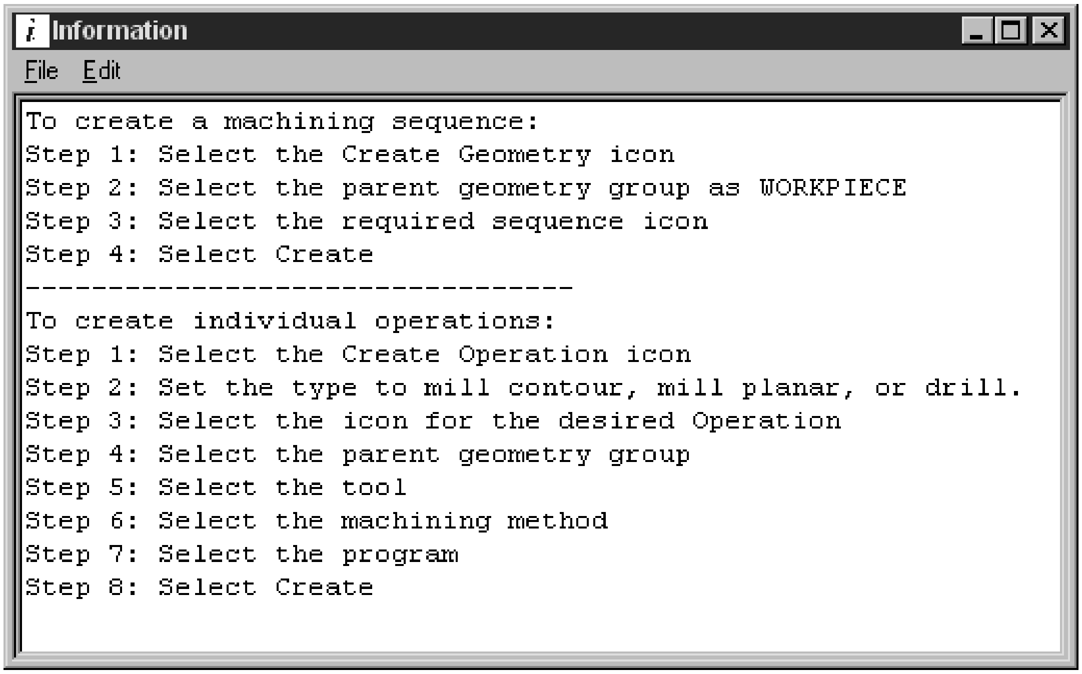
  The **Process Assistant Step: 4** is displayed.

- ❑ In the **Process Assistant Step: 4** dialog, choose **Display.**

  The Blank geometry is displayed.

- ❑ Choose **OK**.

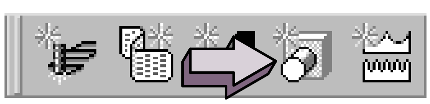
A dialog window appears, referring you to an information window for further instructions. The information window which is displayed, gives the steps necessary for creating a machining sequence and/or individual operations.



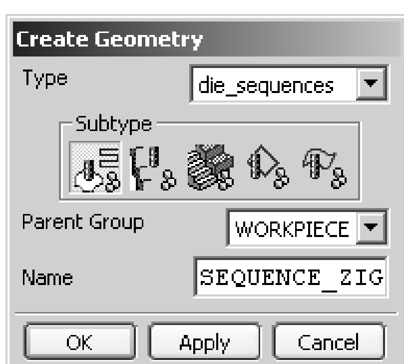
❑ Choose **OK**.

The Operation Navigator is displayed.

❑ Choose the **Create Geometry** icon from the **Create** tool bar.

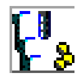


The **Create Geometry** dialog is displayed.



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 select 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 select a trim boundary around the outside of the part to prevent the tool from "water falling" down the side of the part.

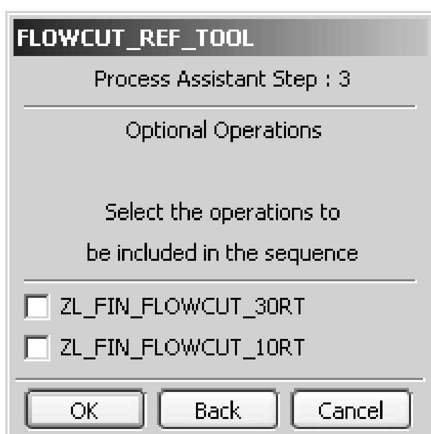
❑ At the prompt to select the trim boundary, choose **Select**.

❑ Set the **Trim Side** to **Outside**.

8

❑ 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.

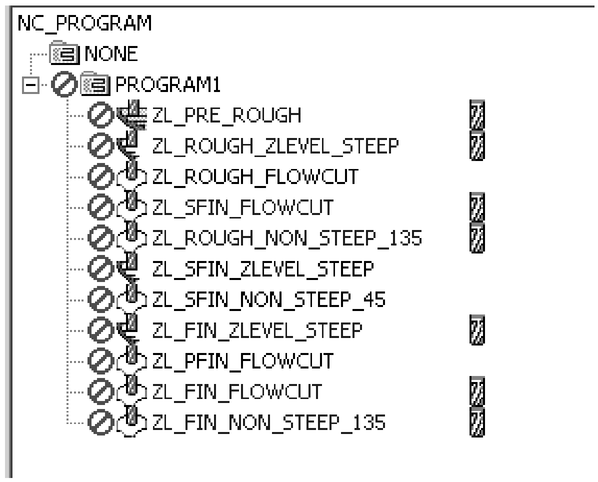


❑ Choose **OK**.

❑ Choose **OK** for any other Process Assistant Steps**.**

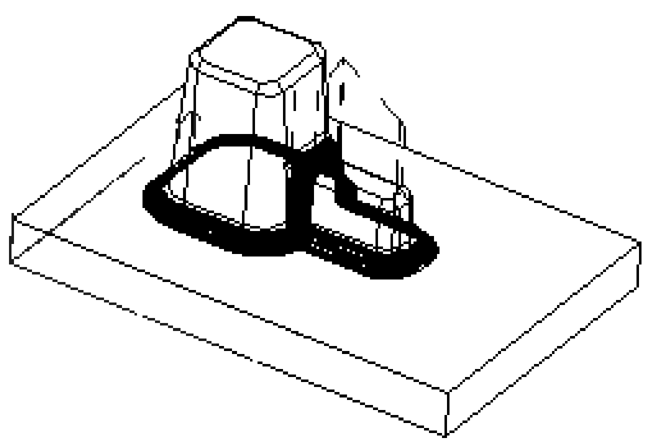
The machining sequence of operations is created.

❑ In the Operation Navigator **Program Order** view, expand **PROGRAM1** and note the various operations.



Step 3 Generating the tool paths.

❑ **Highlight the PROGRAM1 group object, using MB3, select Generate.**



❑ Examine the various tool paths.

❑ **Close** the part file.

You are finished with this activity.

More on Templates

Changing the Machining Environment

The option **Tools** → **Operation Navigator** → **Delete Setup** returns you to the **Machining Environment** dialog. This option will delete all CAM information in the part file and will allow the selection of another **Machining Environment**. Note that use of this option will permanently remove all of your operations, tools, etc.

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).

Subtype Icons

Standard icons are provided for all **Subtypes** found in the standard **Create** dialogs. When you are creating customized dialogs and using custom icons, the icon file name must match the **Subtype** name in order for the correct icon to appear on the dialog. Otherwise a default icon will be substituted

Icons are 24 x 24 x 16 color bitmaps (.bmp file extensions). When creating custom icons, place all icons in the user bitmap directory which is defined by the environment variable **UGII_BITMAP_PATH** located in the ugii_env.dat file.

Activity 8—4: Using Icons in a Customized Template

In this activity, you will examine an existing template that is used for tapping. The template contains center drilling, drilling and tapping operations, based on tap size required. You will examine the template in detail and see how custom icons are used for subtype representation. The use of custom icons will further enhance your abilities to customize your machining environment.

Step 1 Open the existing template file.

❑ If necessary, start Unigraphics NX.

❑ Use **File → Open.** 

❑ Navigate to your parts folder and open the file **ama_tap_template**.

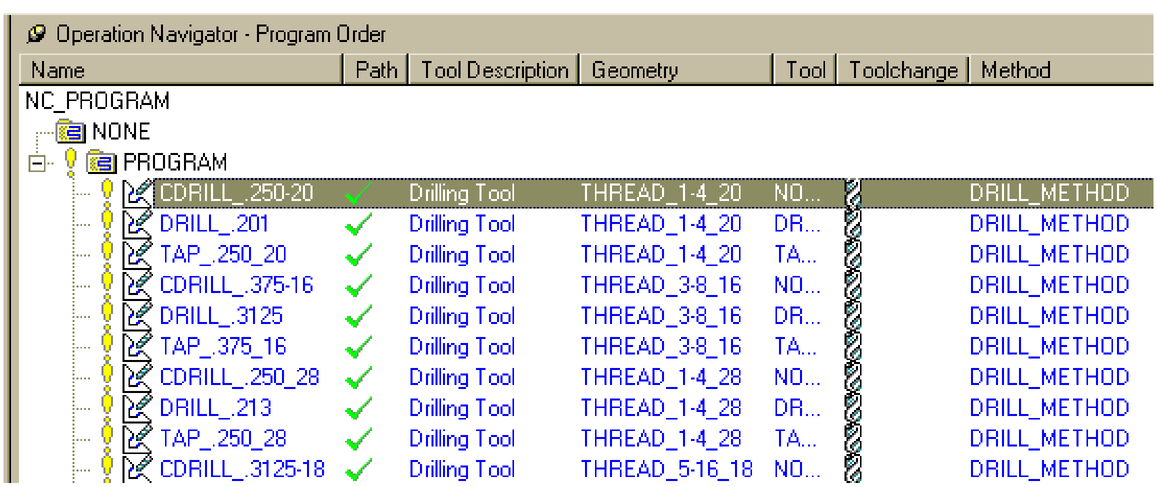
❑ Save the part as, *****_tap_template.prt** where ******* represents your initials**.**

❑ Enter the **Manufacturing** application.

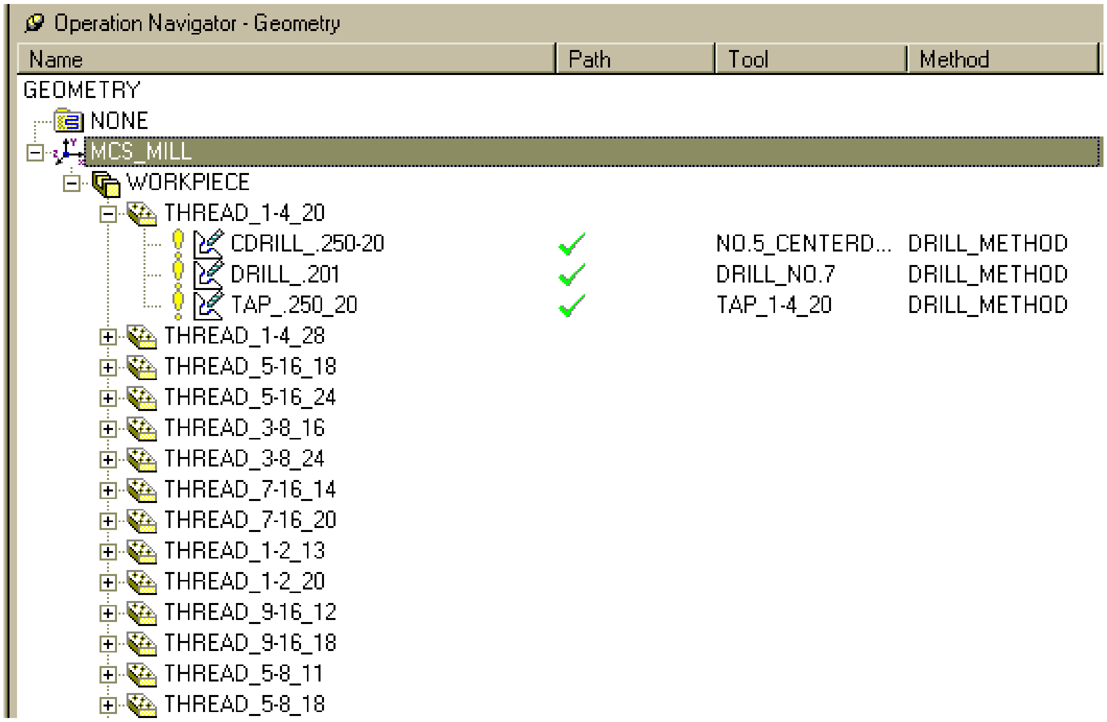
Step 2 Examine the various operations.

❑ Display the **Program Order** view in the Operation Navigator.

Note that operations were previously created to center drill, drill and tap the various tap sizes that are available.

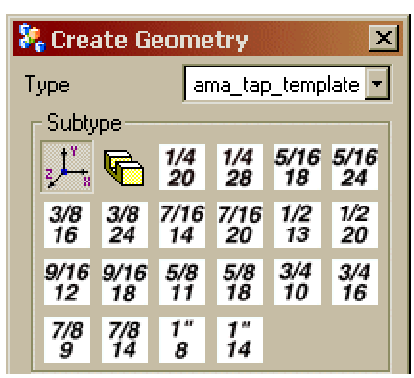


❑ Display the **Geometry** view in the Operation Navigator and expand the **THREAD_1-4_20** geometry group object.

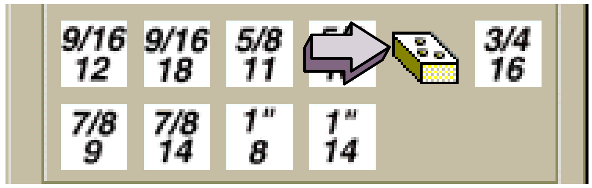


Pay close attention to the name of the individual geometry objects. Individual bitmaps have been previously created and exist for each tap size. The names of these bitmap files match the name of the geometry object. For instance, the bitmap for the **THREAD_9-16_12** object is **thread_9-16_12.bmp**.

When this template is used by your part file, the **Subtype** options will be represented by icons that have the same name as the operation that you are performing.



If these names do not match, the default bitmap icon will appear.



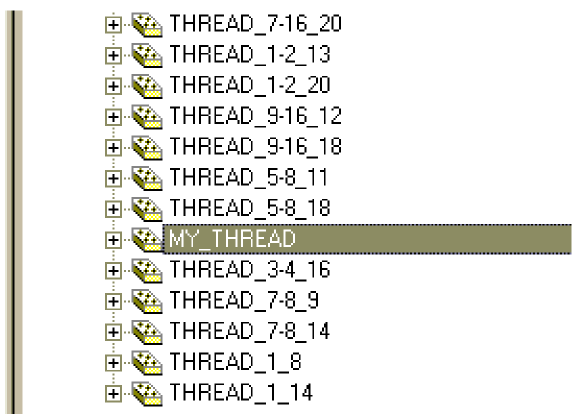
You must specify where the bitmap files reside by changing the environment variable **UGII_BITMAP_PATH** (the bitmap files for this class, reside in a subdirectory of your part files). The environment variable, has already been set for you.

To further show this relationship, you will rename the group object, **THREAD_3-4_10 to MY_THREAD**. When the template is retrieved in your part file, the default subtype icon will appear for the **THREAD_3-4_10** object.

Step 3 Renaming the geometry object.

- ❑ Highlight the **THREAD_3-4_10** group object and using **MB3**, rename the object to **MY_THREAD**.

8



- ❑ **Save** the template part file.

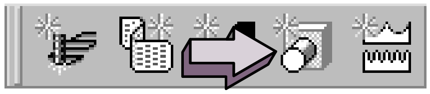
Step 4 Using the template file that you just modified.

You will retrieve an existing part file and change the template to the template file that you modified in the previous step. Remember that in order for modifications to the template file to take effect, you must **Browse** or **select** the **template file** from the **type** pull-down menu.

❑ Open the part file **ama_ipw.prt**.

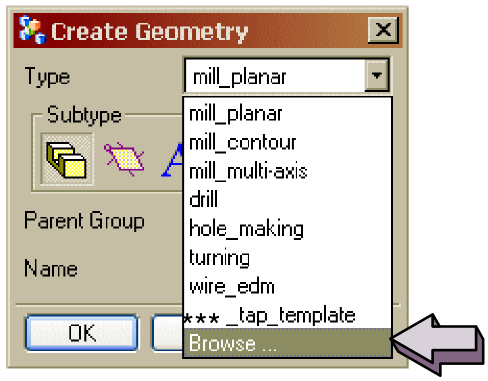
❑ If necessary, choose **Application** → **Manufacturing**.

❑ Choose the **Create Geometry** icon from the **Create** tool bar.



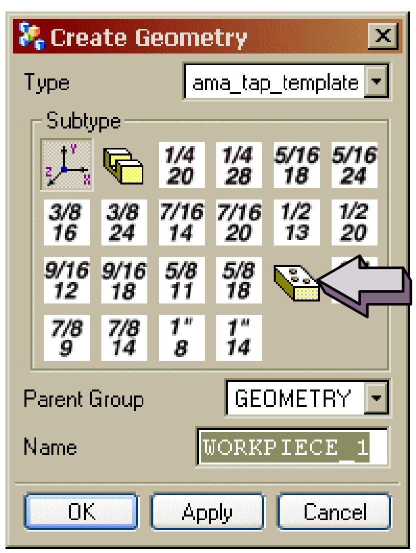
The **Create Geometry** dialog is displayed.

❑ Choose (or Browse) for ***_tap_template** for the **Type**.



❑ Choose **OK**.

The **Create Geometry** dialog is displayed. Note the default icon for the **3/4 10 tap**.



You will now rename the operation to match the existing icon name and will reload the template for the change to be implemented.

❑ Cancel the **Create Geometry** dialog.

Step 5 Rename the operation and reload the template.

❑ Change the displayed part to your template part file (***.tap_template.prt).

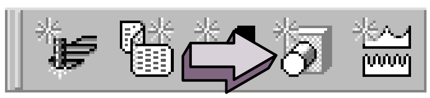
❑ If necessary, change to the **Geometry** view in the Operation Navigator.

❑ Rename the **MY_THREAD** operation to **THREAD_3-4_10**.

❑ **Save** your template part file.

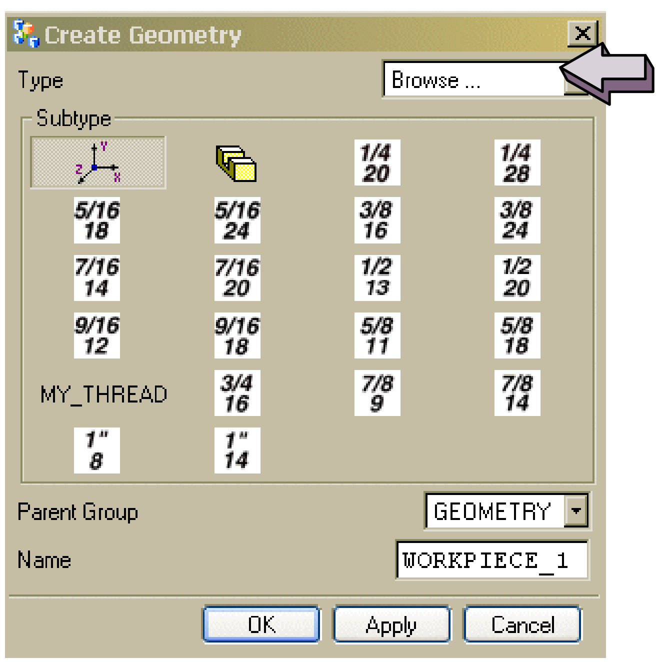
❑ Change the displayed part to **ama_ipw.prt**.

❑ Choose the **Create Geometry** icon from the **Create** tool bar.



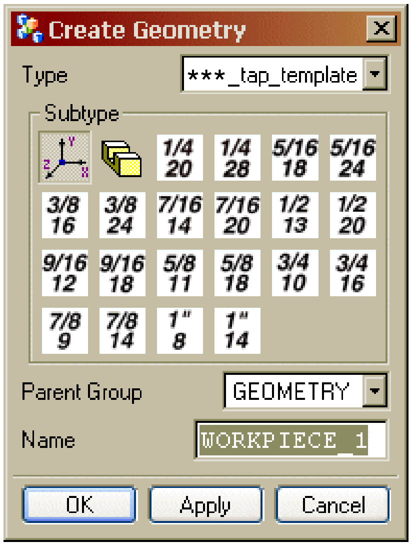
The **Create Geometry** dialog is displayed. You will now reload the template.

❑ Choose **Browse** for *****_tap_template** for the **Type**.



❑ Choose **OK** to the **Template part error** dialog.

The **Create Geometry** dialog is displayed, with the proper icons.



❑ Choose **Cancel**.

❑ **Close** all parts without saving.

This completes the activity and the lesson.

SUMMARY

The Template function provides an efficient means to customize your CAM environment. Numerous parameters used by various operations, custom operation sequences, tool and post processor 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
- Became familiar with template icons; their naming conventions and interactions

(This Page Intentionally Left Blank)

Hole Making

Lesson 9

PURPOSE

In this lesson, you will learn how to create and optimize a Hole Making program.

OBJECTIVES

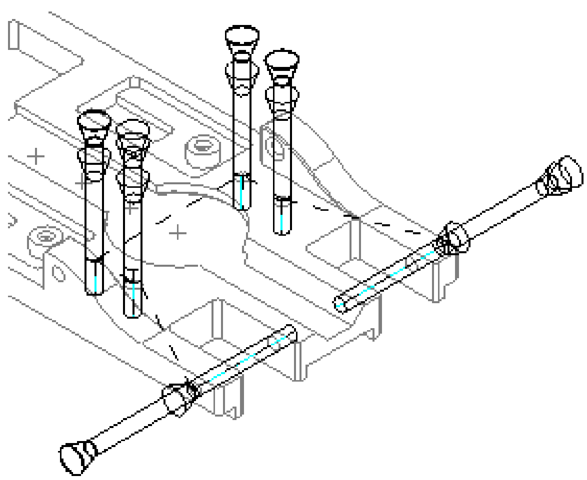
Upon completion of this lesson, you will be able to:

- Create a hole making program that machines simple, countersunk, and counterbore holes
- Identify where tools must be edited or created for Hole Making and apply the necessary changes
- Tag simple geometry so to be recognized as machinable features for the Hole Making processor
- Optimize the Hole Making program
- Use the features and functionality pertaining to Alternate Groups of Operations, Cut Area, Feature Recognition, Feature Status, Holder Types, 3D In Process Work Piece, Maximum Cut Depth and Extended Length and Multiple Selection

This lesson contains the following activities:

Hole Making Overview

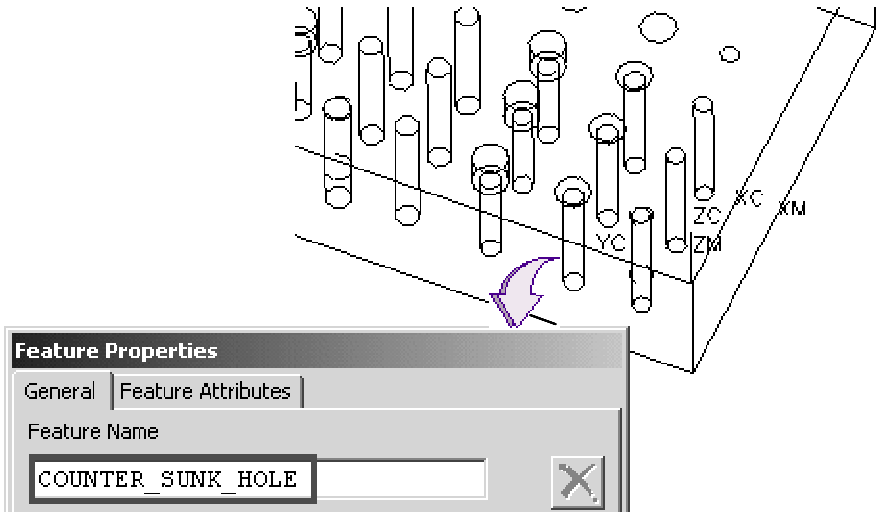
Hole Making is an advanced Manufacturing application that automates the creation of operations such as spot drilling, drilling, countersinking, counterboring, reaming, tapping, and deburring through the use of intelligent models containing manufacturing features (User Defined Features, User Defined Attributes, and Unigraphics based Features) and embedded machining rules.



Manufacturing features contain information that allows the Hole Making processor 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. Hole Making 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.

Hole Making Templates

A Hole Making 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 Hole Making processor makes intelligent decisions about how to machine these feature groups based on **Knowledge Fusion** rules.

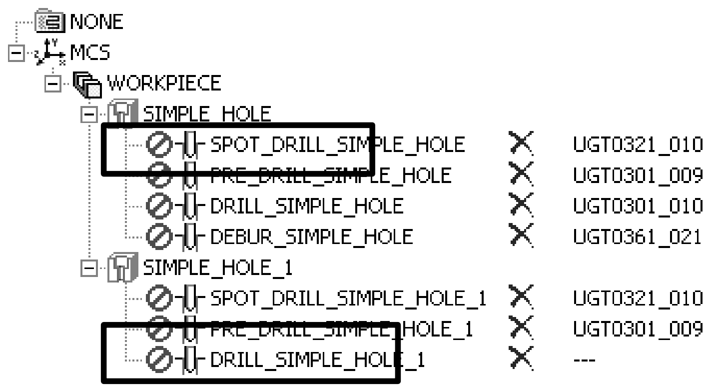


Knowledge Fusion rules are associated with feature groups through **Adopted Operations** displayed in the Knowledge Fusion Navigator. This association allows the Hole Making processor to apply specific machining rules to feature groups.

9

Feature Groups

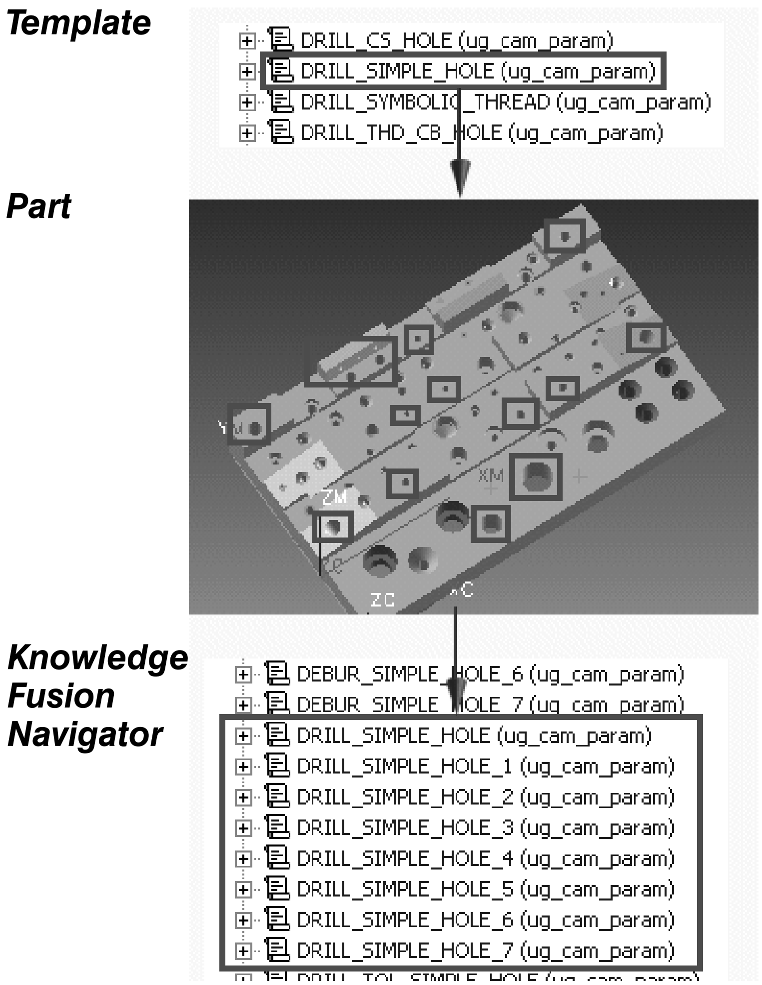
The **Geometry** view of the Operation Navigator associates manufacturing features with feature groups. Feature groups organize operations according to feature type. The feature group named SIMPLE_HOLE for example, contains all operations (spot drill, drill, ream) that need to be performed on simple 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.

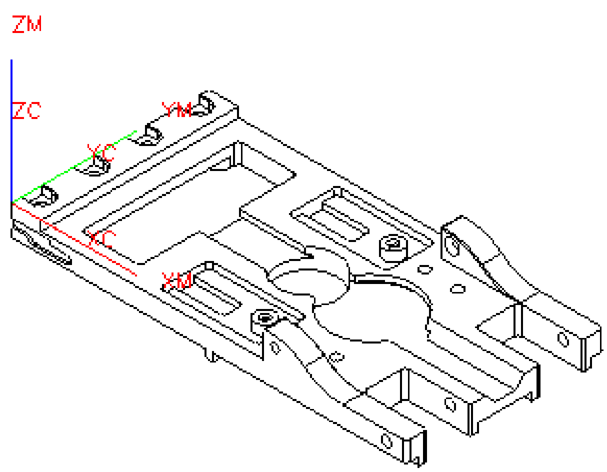
Editing Knowledge Fusion rules requires a working knowledge of the language in which the Knowledge Fusion rules are written and is typically not done by the end-user. Knowledge Fusion will not be emphasized in this course.

Activity 9–1: Machining Holes

In this activity, you will create hole making operations that machine simple, countersunk, and counterbore holes.

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.

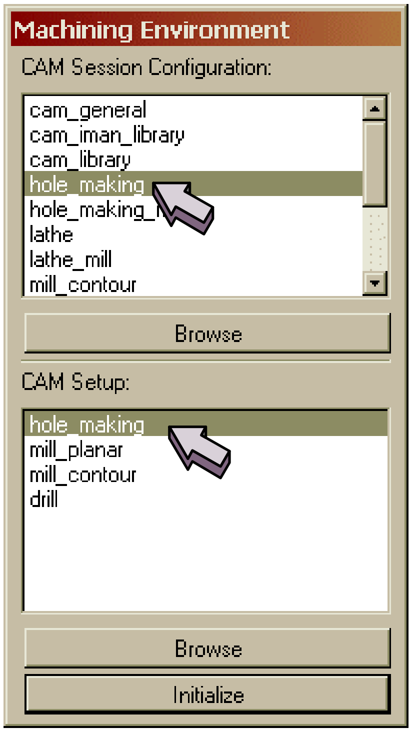
❑ Choose **Application → Manufacturing.**

Step 2 Define the Machining Environment.

The **Machining Environment** dialog displays since 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 **hole_making** as the **CAM Session Configuration**.

❑ Choose **hole_making** as the **CAM Setup**.

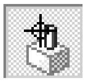


Hole_making is the standard CAM Setup used for hole making applications. Other setups may also be available depending on your working environment.

❑ Choose **Initialize**.

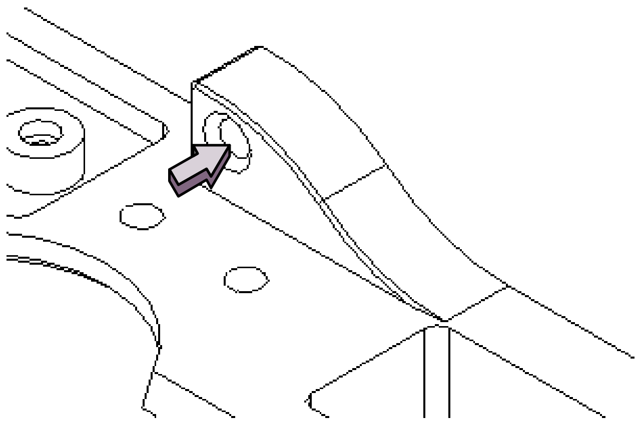
Step 3 Display Feature Properties.

You will display the feature name of a countersunk hole.

❑ Choose the **Select Features** icon. 

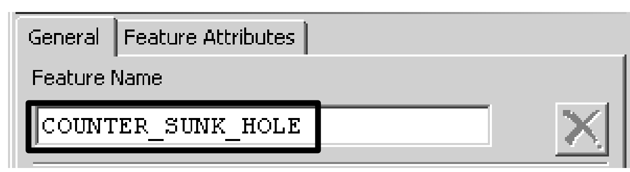
9

❑ Select the countersunk hole illustrated below.



❑ Choose **MB3,** then select **Properties**.

In the **COUNTER_SUNK_HOLE** dialog, the feature name is **COUNTER_SUNK_HOLE**.



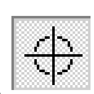
Since the hole making template also contains a feature called **COUNTER_SUNK_HOLE**, the Hole Making processor will recognize this feature and apply the appropriate machining rules based on the shape, size, and surface finish of the feature.

❑ **Cancel** the **Feature Properties** dialog.

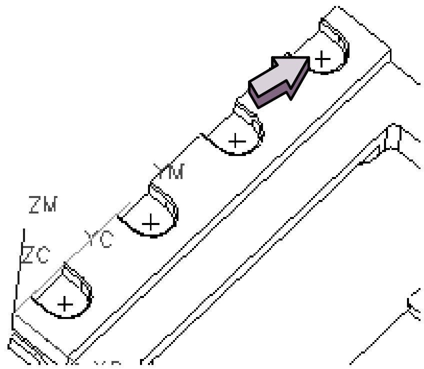
9

Step 4 Display Object Properties.

You will display the properties of an attributed point.

❑ Choose the **Select General Objects** icon. 

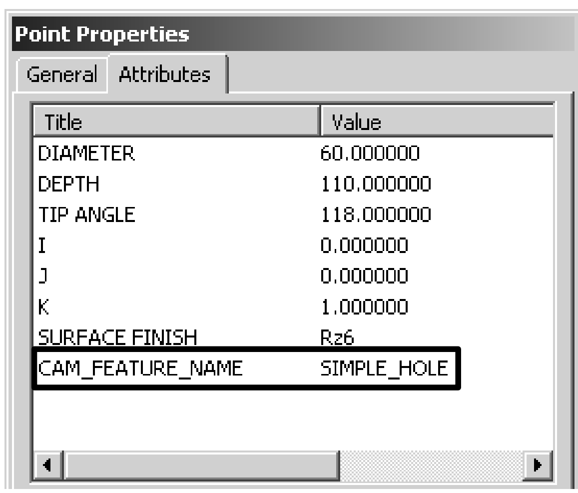
❑ Select the point illustrated below.



❑ 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 Hole Making processor will recognize this point as a feature and apply the appropriate machining rules based on the attributes.

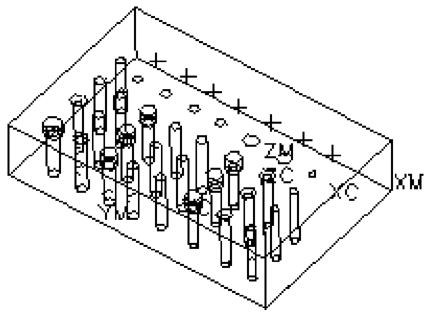
❑ **Cancel** the **Point Properties** dialog.

Step 5 Examine the Hole Making Template.

You will review 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 observed in your part.

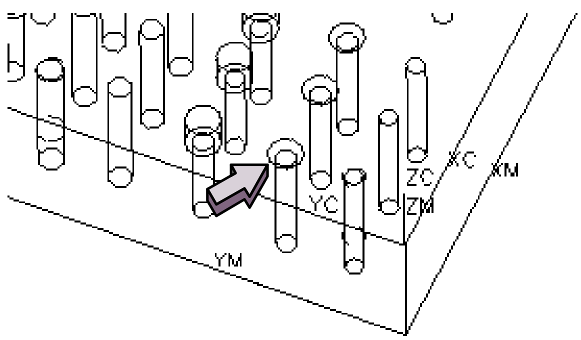
❑ In the menu bar, choose **Window** → **More Parts**.

❑ Choose **hole_makin**g in the **Change Displayed Part** dialog and then choose **OK**.



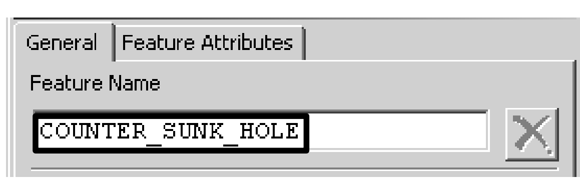
❑ Choose the **Select Features** icon.

❑ Select the countersunk hole illustrated below.



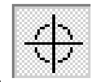
❑ Using **MB3**, choose **Properties**.

❑ In the **COUNTER_SUNK_HOLE Properties** dialog, notice that the feature name, **COUNTER_SUNK_HOLE**, is the same as the one in your part.

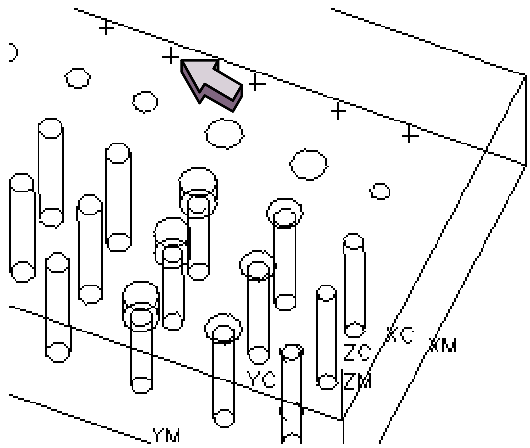


Since your part contains machinable features called **COUNTER_SUNK_HOLE**, the Hole Making processor will recognize all occurrences of this feature and apply the appropriate rules for machining.

❑ **Cancel** the **Feature Properties** dialog.

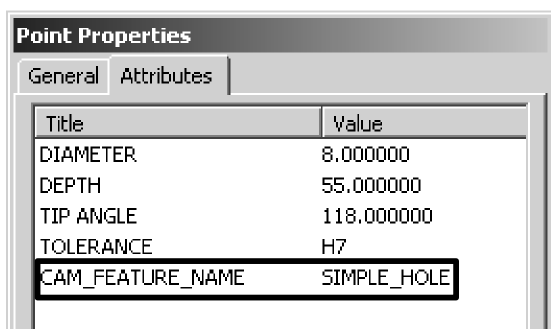
❑ Choose the **Select General Objects** icon. 

❑ Select the point illustrated below.



❑ Using **MB3**, choose **Properties**.

❑ In the **Point Properties** dialog, choose the **Attributes** tab. Notice the feature name is **SIMPLE_HOLE**.



Since your part contains attributed points called **SIMPLE_HOLE**, the Hole Making processor 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.

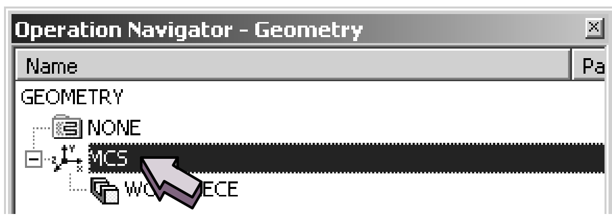
Step 6 Specify an Appropriate Tool Axis.

You will specify a tool axis allowing the Hole Making processor 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**) to display in a separate window.

❑ Choose the **Geometry** view icon in the tool bar to display the Geometry view of the Operation Navigator.

❑ Double-click on the **MCS** icon in the Operation Navigator.



❑ Verify the **Tool Axis** option is set to **All Axes**.

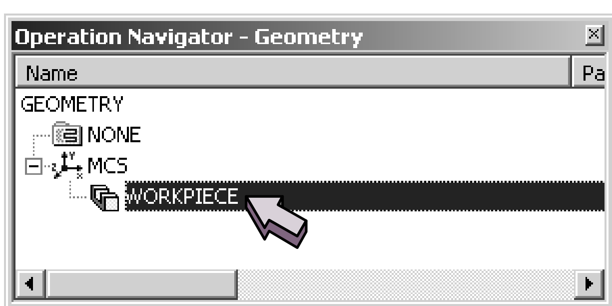
❑ Choose **OK** to accept the tool axis.

Step 7 Specify the Part Geometry.

You will specify the solid body as the part geometry.

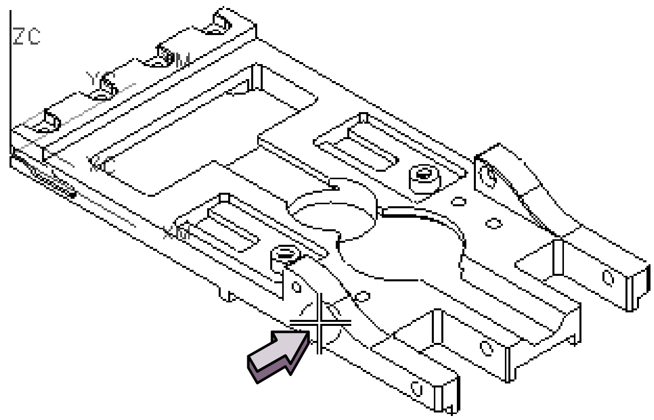
9

❑ Double-click on the **Workpiece** icon in the Operation Navigator.



❑ Verify the **Part** icon is chosen and choose **Select**.

❑ Choose the solid body.



❑ Choose **OK** to accept the part geometry.

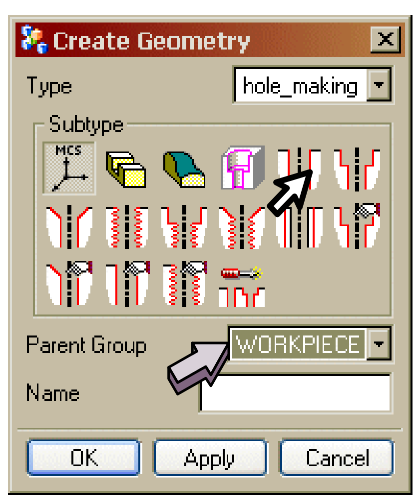
❑ Choose **OK** to accept the **MILL_GEOM** dialog.

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 on the tool bar.

❑ Choose the **SIMPLE_HOLE** icon and verify the **Parent Group** option is set to **Workpiece**.



❑ Choose **OK** to create the operations.

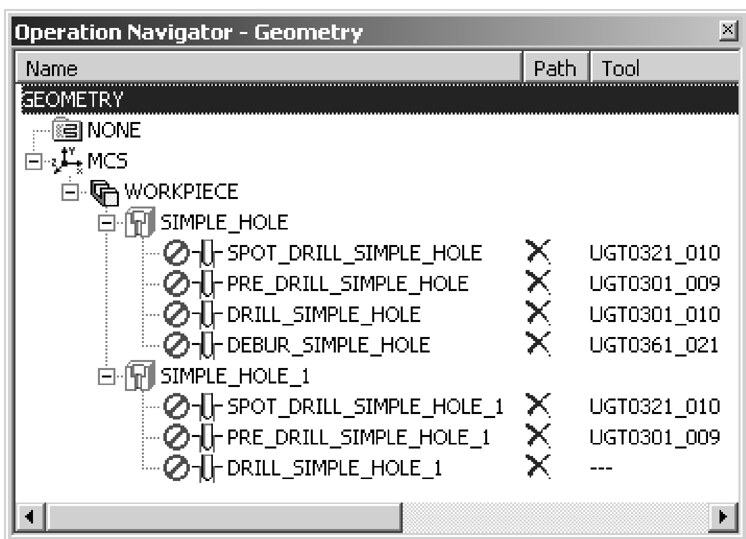
The processor will take a few moments to process the information.

- ❑ 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 Hole Making processor assigns feature group numbers randomly, the operations listed in one feature group (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.

9

❑ Enlarge the Information window and examine the contents.

The Hole Making 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.

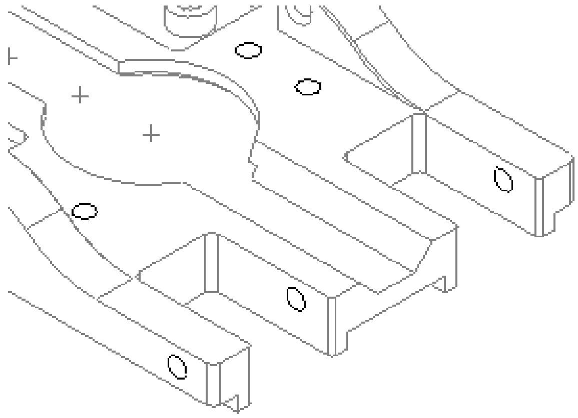
```
Instantiating Feature Geometry Template : SIMPLE_HOLE

*******************************************************************
Identifying all features of name: SIMPLE_HOLE

10 features were found with feature name: SIMPLE_HOLE

Classifying features according to the following criteria

       DIAMETER

 2 group(s) were found after classification criteria were applied
```

❑ 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. Your results may be different for this particular feature group because the Hole Making processor 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.

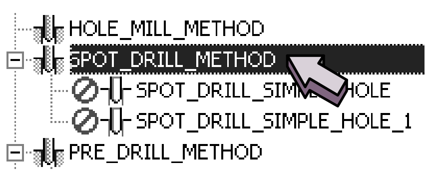
Step 11 Perform a Tool Query.

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.

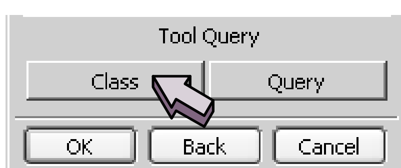
❑ Choose the **Machining Method** view icon in the tool bar to display the Machining Method view of the Operation Navigator.

❑ Expand the **Machining Method** groups.

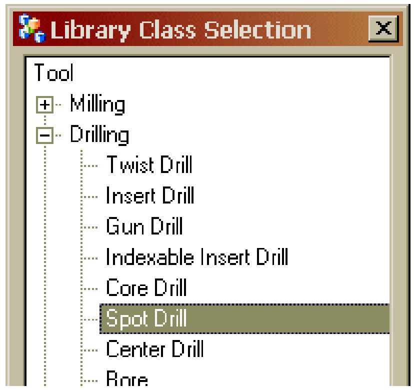
❑ Double-click on the **SPOT_DRILL_METHOD** group icon.



❑ 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, and 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.

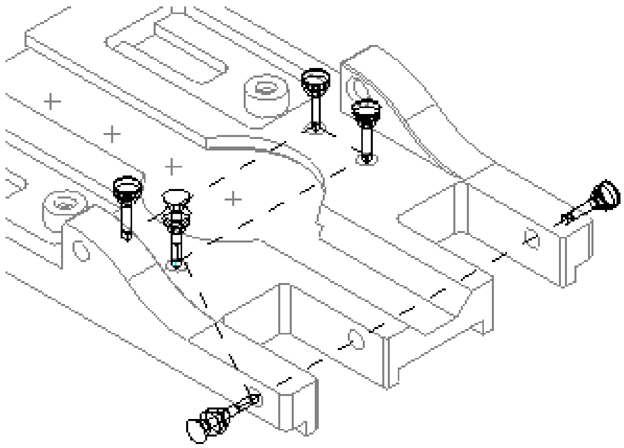
9

❑ Double-click on the **SPOT_DRILL_SIMPLE_HOLE** operation to display the operation dialog.

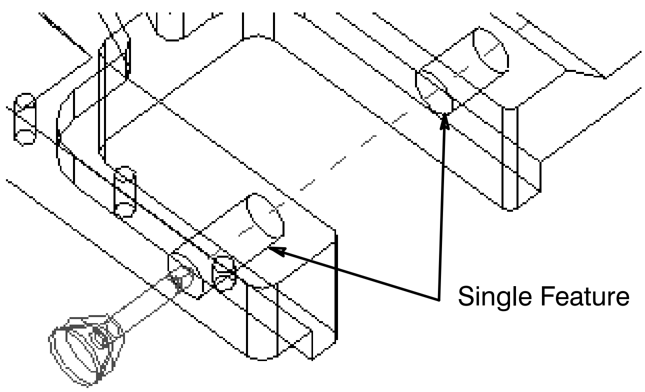
❑ Choose the **Edit Display** icon under **Tool Path** and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the **Display Options**.

❑ **Generate** the tool path. Your tool path might differ slightly.



Notice at the end of the part, only the outermost horizontal holes are spot drilled and the operation does not attempt to spot drill the inner holes. This is a result of each pair of holes (the two holes drilled from the left and the two holes drilled from the right) being modeled as a single feature.

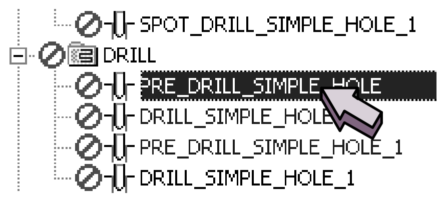


❑ Choose **OK** to accept the operation.

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.

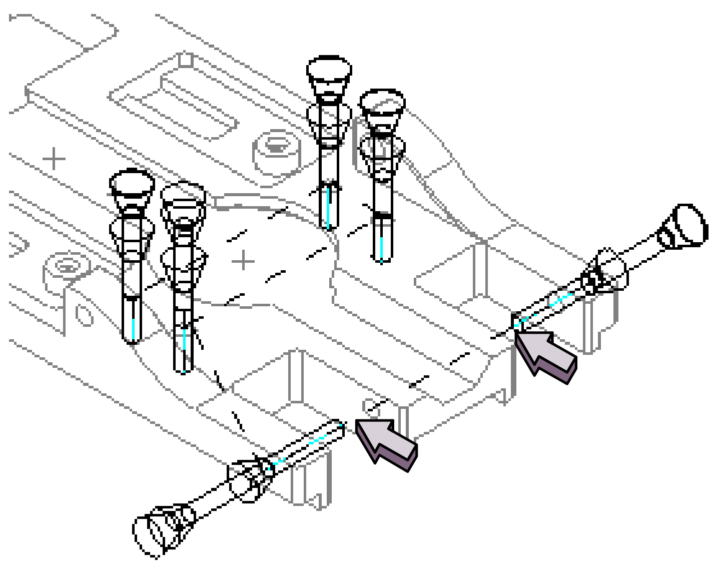


❑ Choose the **Edit Display** icon under **Tool Path** and set the **Tool Display** option to **3-D**.

❑ Choose **OK** to accept the **Display Options**.

❑ **Generate** the tool path.

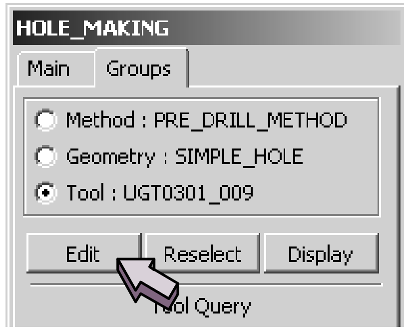
Notice that the tool is not long enough to drill through the fork area at the end of the part. This is a case where you will need to edit the length of the tool.



Step 14 Edit the Tool Length.

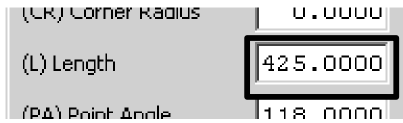
❑ Choose the **Groups** tab.

❑ With **Tool:UGT0301_009** chosen, choose **Edit**.



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

❑ Change the **Length** to **425.**

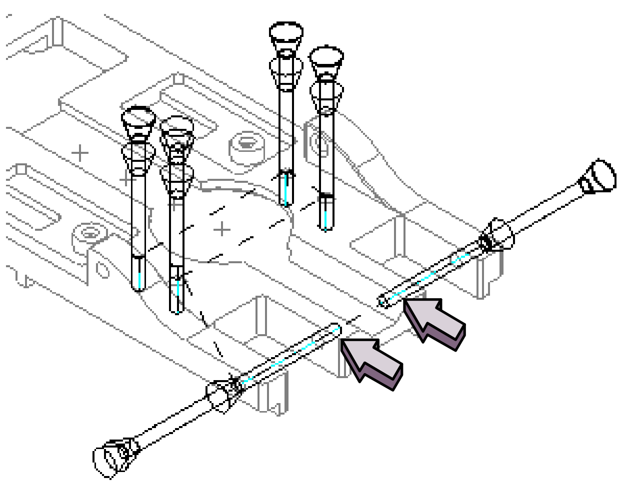


❑ Choose **OK.**

❑ Choose **YES** to the **Edit Tool** warning message.

❑ Choose the **Main** tab.

❑ **Generate** the tool path.



The tools are now long enough to drill the full depth of the holes.

❑ Choose **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.

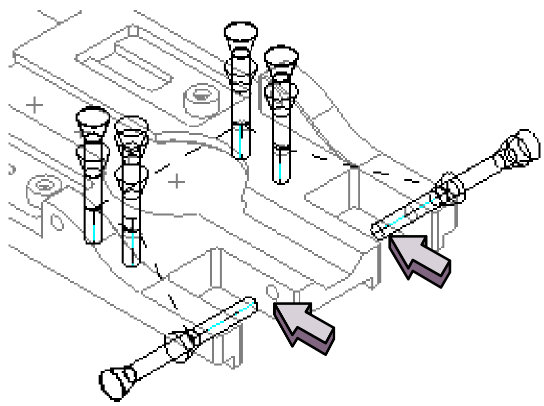
- ❑ Double-click on the **DRILL_SIMPLE_HOLE** operation to display the operation dialog.

- ❑ Choose the **Edit Display** icon under **Tool Path** and set the **Tool Display** option to **3-D**.

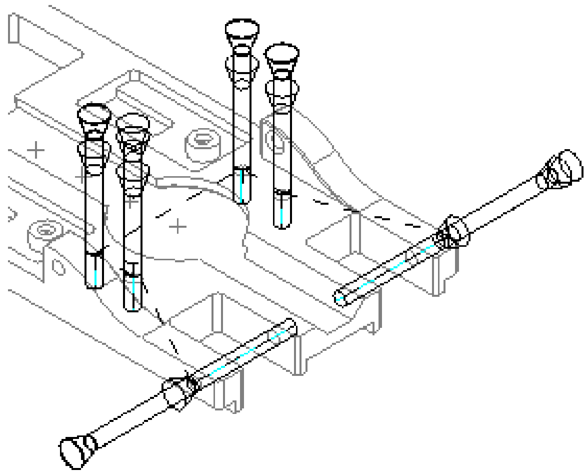
- ❑ Choose **OK** to accept the **Display Options**.

- ❑ **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 previously.



- ❑ Change the length of the tool to 425 and regenerate the tool path as you previously did.



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

Step 16 Generate the Spot Drill Tool Path.

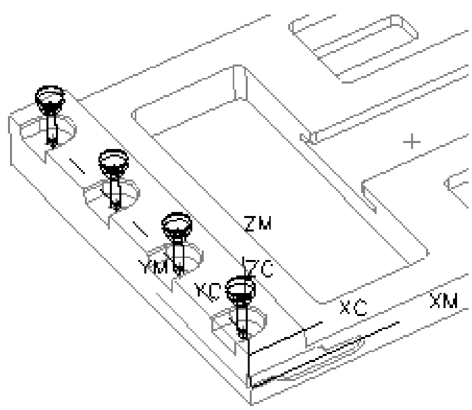
You will display the tools and generate the tool path for the **SPOT_DRILL_SIMPLE_HOLE_1** operation.

❑ 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.

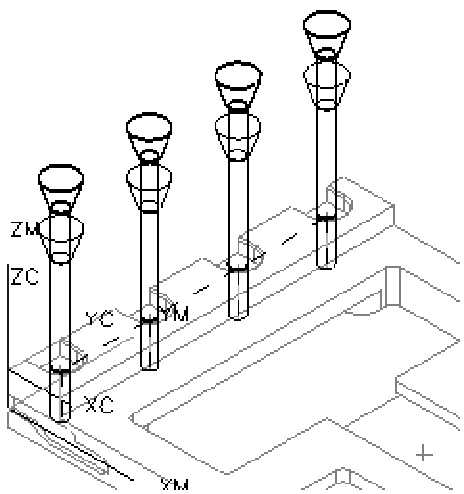
You will display the tools and generate the tool path for the **PRE_DRILL_SIMPLE_HOLE_1** operation.

❑ 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 received an error message stating that no tool has been specified.

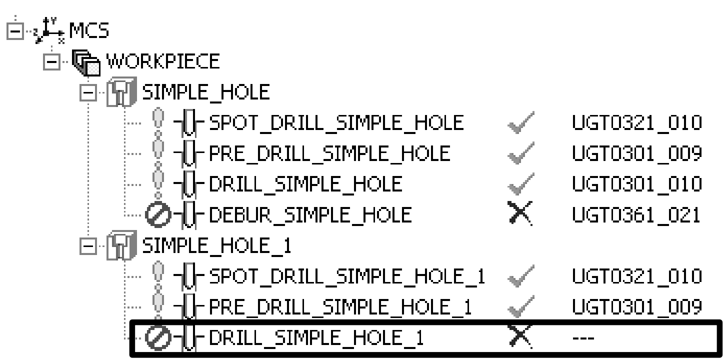
The hole making template does not contain the tool required to drill the 60 mm diameter holes.

- ❑ Choose **OK** to the error message.
- ❑ Choose **OK** to the operation.

Step 19 Adding a Tool.

You should verify that every operation contains a tool. The best approach to accomplish 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 contain 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 previously.

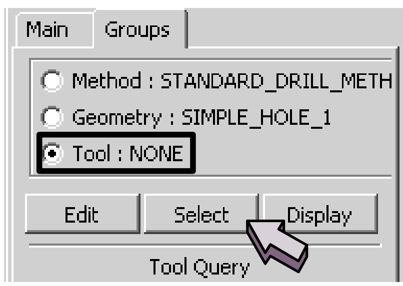
You will find that the holes have a diameter of 60 and a depth of 110. You can now create the required tool.

- ❑ Double-click the **DRILL_SIMPLE_HOLE_1** operation icon to edit the operation.

- ❑ Choose the **Groups** tab.

  Notice that the **Tool:** option object is **NONE**.

- ❑ Choose the **Select** button.



- ❑ Choose **New** on the **Select Tool** dialog.

- ❑ Choose the **STD_DRILL** icon and then **OK** to accept it.

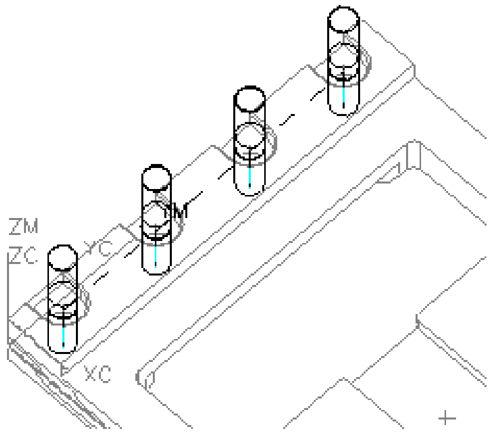
- ❑ Key in **60** in the **Diameter** field.

- ❑ Key in **200** in the **Length** field.

- ❑ Choose **OK** to create the tool.

- ❑ Choose the **Main** tab.

❑ Generate the tool path.



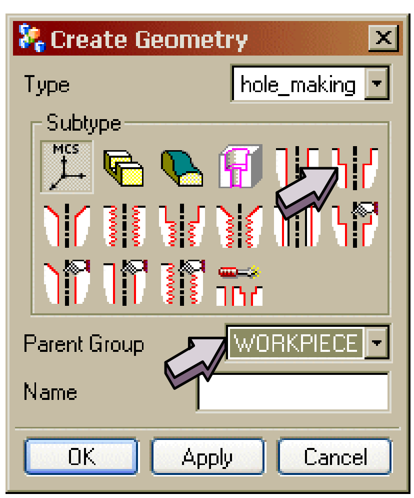
❑ Choose **OK** to complete the operation.

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 tool bar.

❑ Choose the **CB_HOLE** icon and verify the **Parent Group** option is set to **Workpiece**.



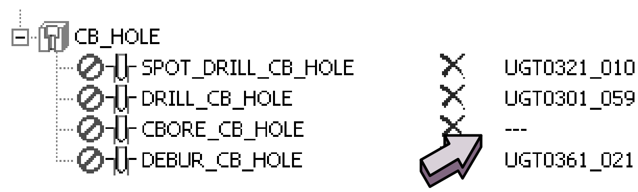
❑ Choose **OK** to create the operations.

The processor 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.

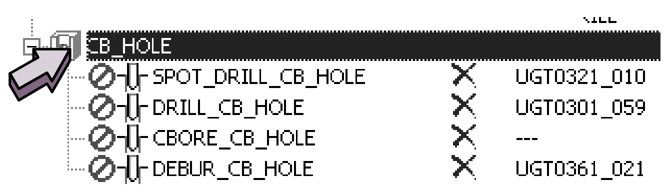
Notice that one of the operations does not have a tool.



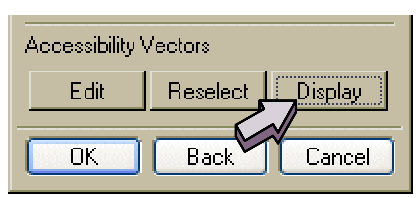
Step 21 Adding a Tool.

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.

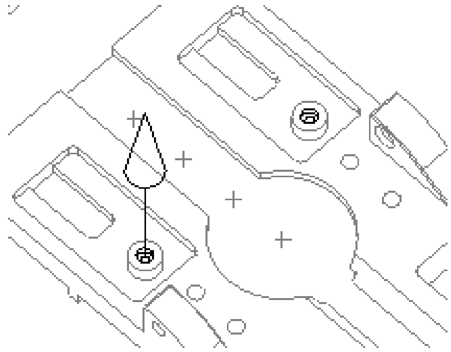
❑ Double-click on the **CB_HOLE** feature group icon.



❑ Under **Accessibility Vectors,** choose **Display**.

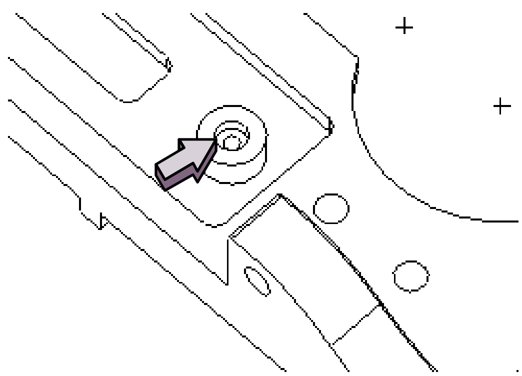


The **Vector** is displayed in the Graphics window.



The vector identifies one of the features associated with the **CB_HOLE** feature group.

- ❑ **Cancel** the **CB_HOLE** dialog.
- ❑ Choose **Information** → **Feature** from the tool bar.
- ❑ Select the counterbore hole illustrated below.



- ❑ Choose **OK** in 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.

You can now 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**.

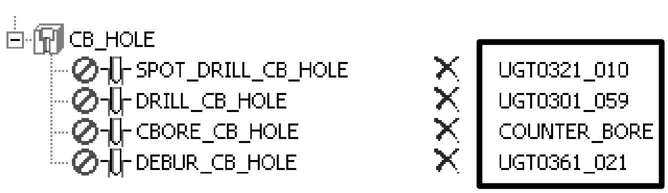
❑ Choose the **COUNTER_BORE** icon and **OK** to accept it.

❑ Key in **50** in the **Diameter** field.

❑ Choose **OK** to create the tool.

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

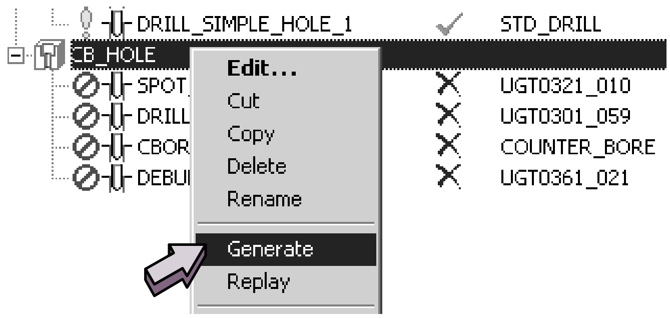
All of the operations within the **CB_HOLE** feature group now contain tools.



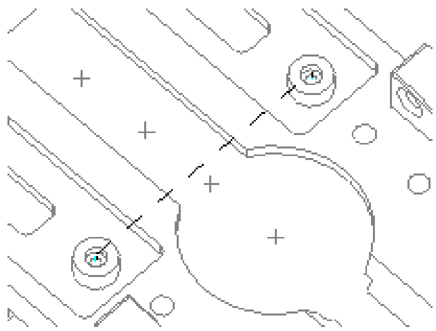
Step 22 Generate the Tool Paths for the Counterbore Holes.

❑ Highlight the **CB_HOLE** feature group and choose **MB3 →** **Generate** to generate the tool paths.

9



❑ Choose **OK** to accept each one of the tool paths.



❑ **Save** the part file.

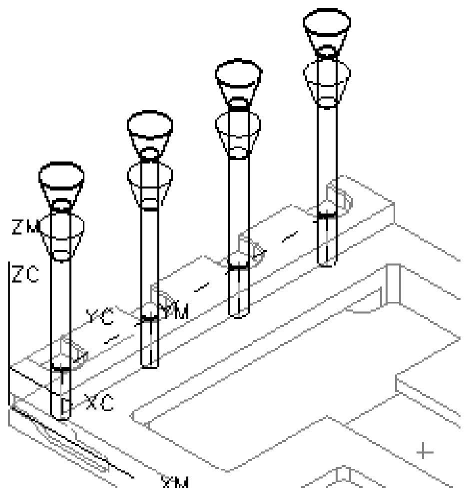
You have completed this activity.

Next, you will learn how to tag simple geometry such as points and arcs that are recognized as machinable features.

9

Tagging

The Hole Making processor recognizes NX based features such as simple holes, countersunk holes, counterbored holes, and symbolic threads as machinable features and applies machining rules based on their shape, size, and other attributes such as surface finish. Simple geometry such as points and arcs, 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.



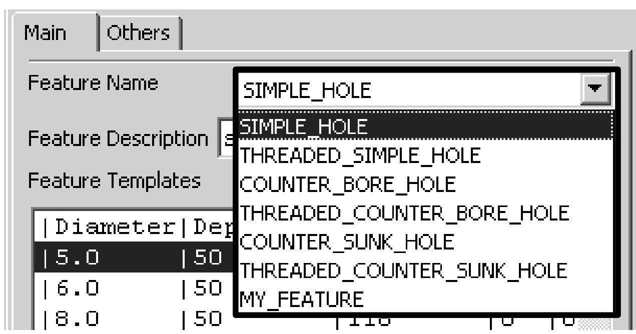
You were able to machine these points since they had been previously tagged as simple holes with a specific diameter and depth. Tagging allows you to apply attributes such as feature name, diameter, depth, tip angle, and surface finish so that the Hole Making processor can recognize simple geometry as machinable features and apply machining rules.

Activity 9–2: Tagging Points

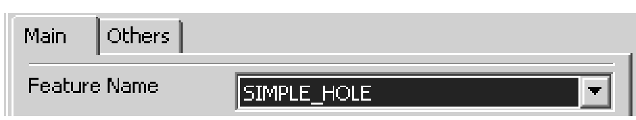
In this activity, you will tag points so that the Hole Making processor recognizes them as simple holes.

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 assigned attributes.

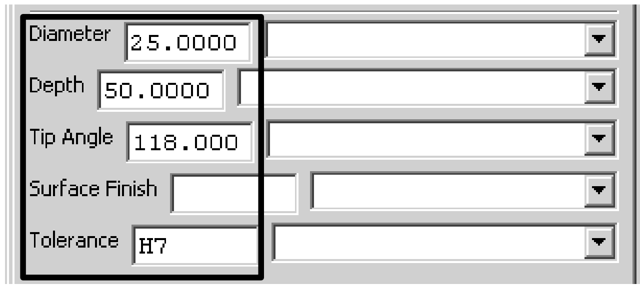


- ❑ Verify that **SIMPLE_HOLE** is specified as the **Feature Name**.



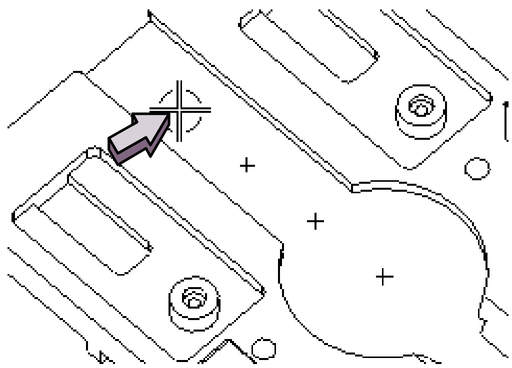
9

❑ Fill in the following values in the **Tagging Attributes to Points** dialog. Verify the **Surface Finish** field has been blanked out and that the **Tolerance** field says **H7**.



By specifying a tolerance of H7 the Hole Making processor will 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, for the attributes to be applied individually to each point.

❑ 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.

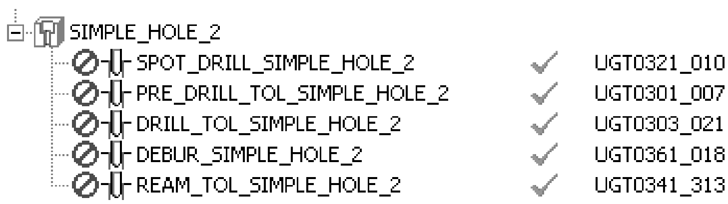
- ❑ Choose the **Create Geometry** icon in the tool bar.

- ❑ Choose the **SIMPLE_HOLE** icon and verify 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.

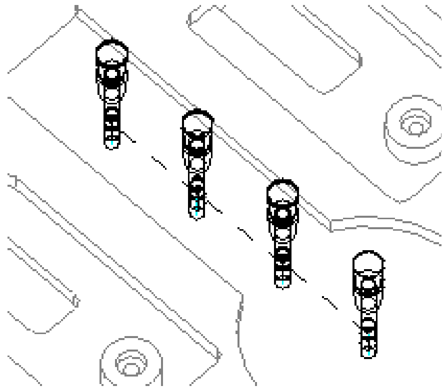


Step 3 Generate the Tool Paths for the Simple Holes.

- ❑ Highlight the **SIMPLE_HOLE_2** feature group and **MB3** → **Generate** to generate the tool paths.

- ❑ Choose **OK** to accept each of the tool paths.

  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.

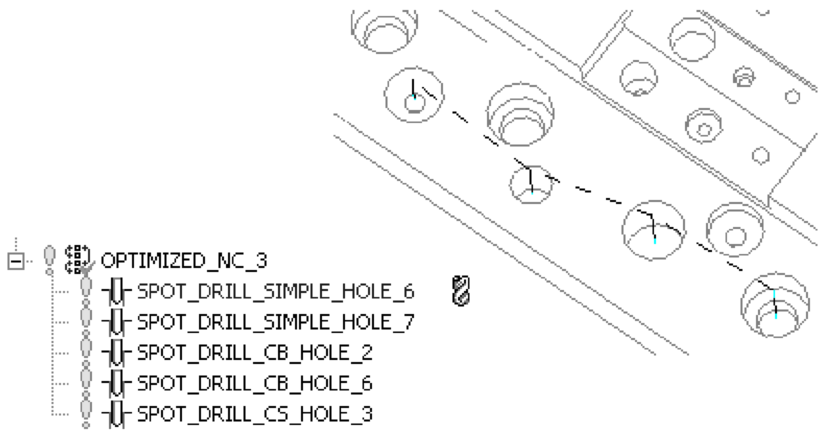


❑ **Save** the part file.

You have completed this activity.

Optimization

Optimization improves machining effectiveness 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:

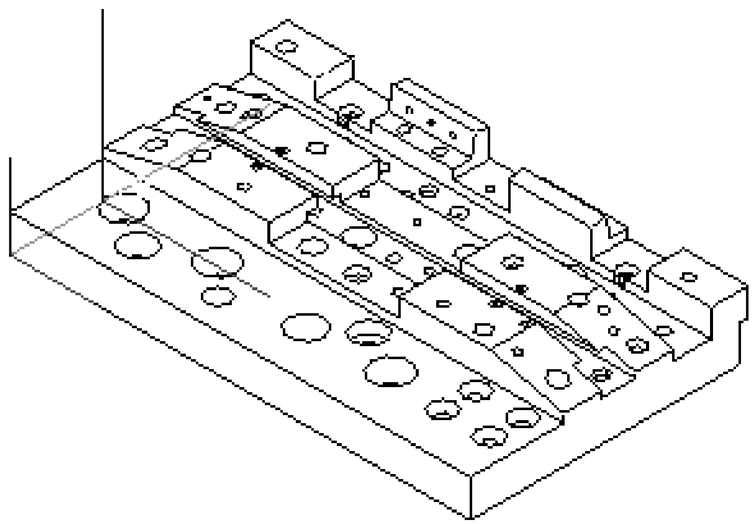
- **Consolidate Tools** causes the program to use as few tools as possible without compromising the effectiveness of machining.
- **Minimize Tool Changes** reorders operations to minimize the number of tool changes that occur within the program. The Hole Making processor does this without violating operation 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.

Activity 9—3: Optimizing a Spot Drill Subprogram

In this activity, you will examine an 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.**



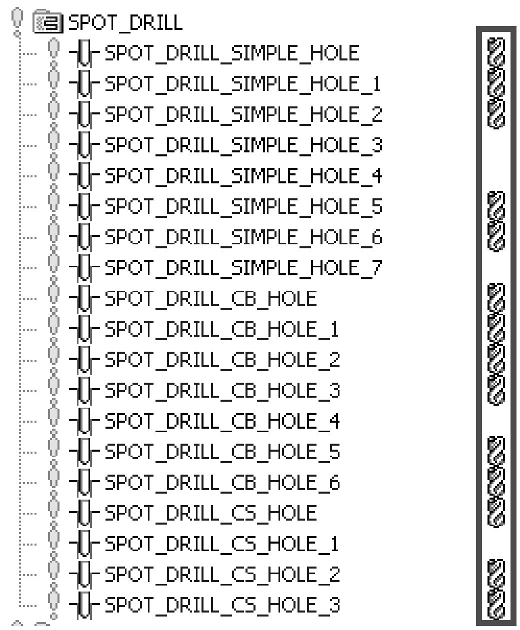
❑ Use the **Save** → **As** option under **File** on the menu bar and rename the part to *****_optimization.prt** where ******* represents your initials.

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

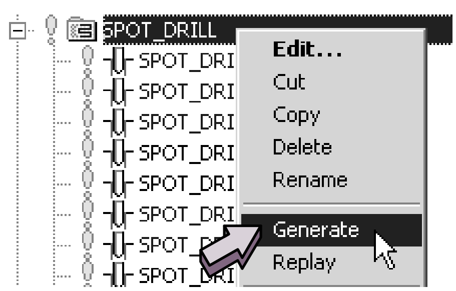
Step 2 Generate the generic Tool Paths.

You will generate the tool paths for the existing **SPOT_DRILL** sub operation to illustrate the inefficiency of the tool movements.

In the **Program Order** view, notice the excessive number of tool changes in the **SPOT_DRILL** group object.



❑ Highlight the **SPOT DRILL** sub operation icon and use **MB3** → **Generate**.



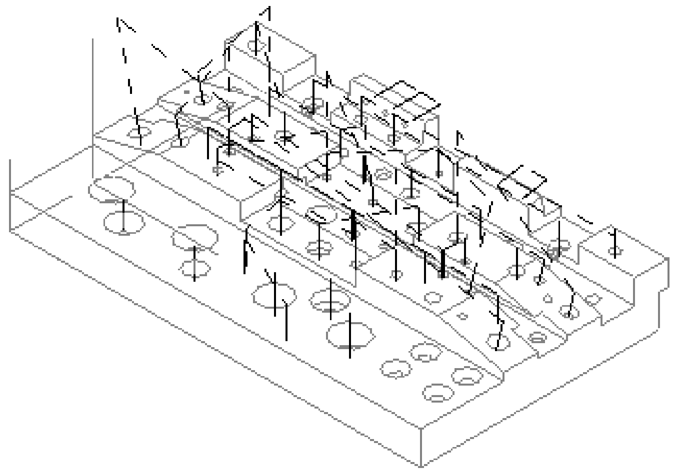
❑ Turn the **Pause After Each Path** and **Refresh Before Each Path** options **off**.

9

❑ Choose **OK** to generate and display all of the tool paths for the **DRILL** sub operation.

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

Notice the excessive and disorganized non-cutting tool movements.

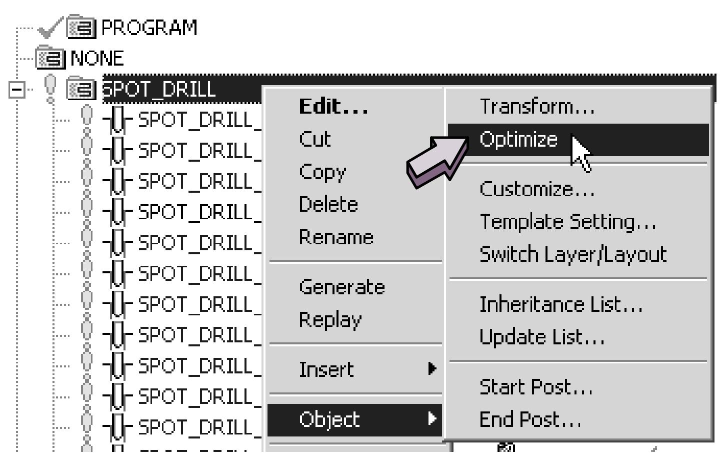


It is more efficient to use one tool where possible regardless of feature type and to minimize tool changes and traversals. You will see how optimization can accomplish this.

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**.

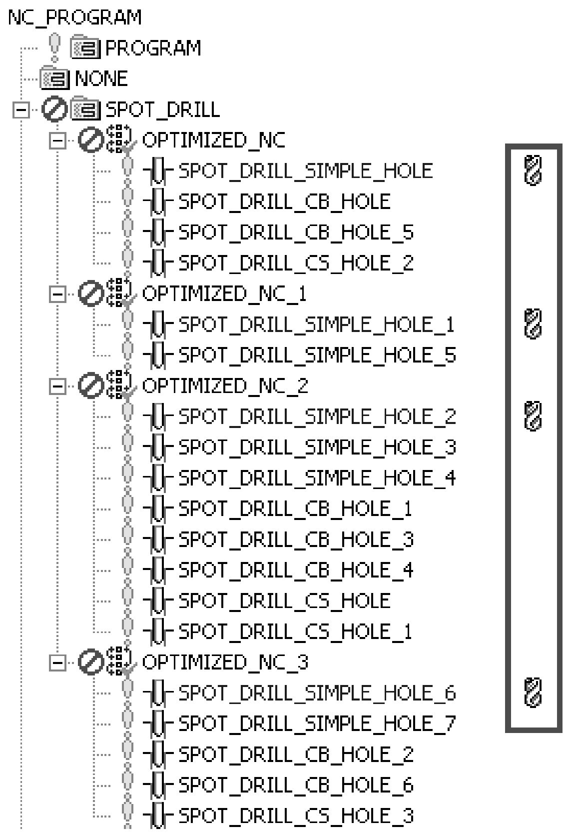


- ❑ Verify that all three options are turned **on** and then choose **OK** to begin the optimization process.

The Hole Making processor will take a few moments to process the holes.

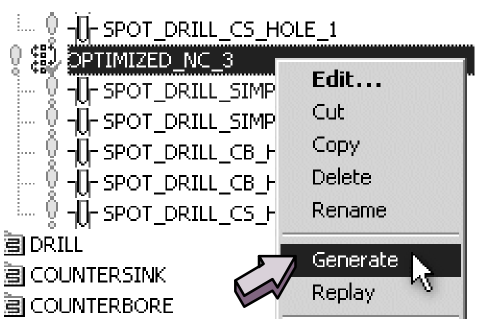
❑ When processing is complete, expand the **Optimization** groups.

The number of tool changes has been reduced to four.



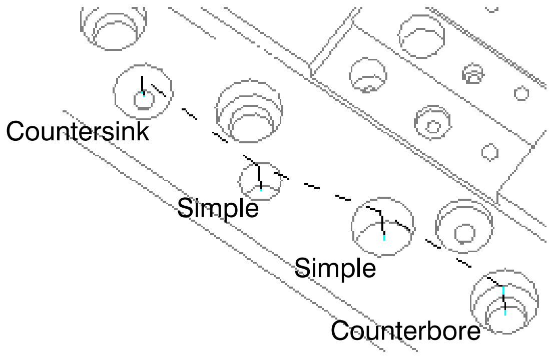
Step 4 Generate the Optimized Tool Paths.

❑ Choose the **OPTIMIZE_NC_3** icon and then use **MB3** → **Generate** to generate the tool path for this optimization group.



❑ Choose **OK** to complete the tool path generation.

The Hole Making processor no longer machines manufacturing features in the order according to feature type. 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 use **MB3** → **Generate** to generate tool paths for all of the optimization subprograms.

❑ **Save** the part file.

You have completed this activity.

9

Additional Hole Making Topics

Alternate Groups of Operations

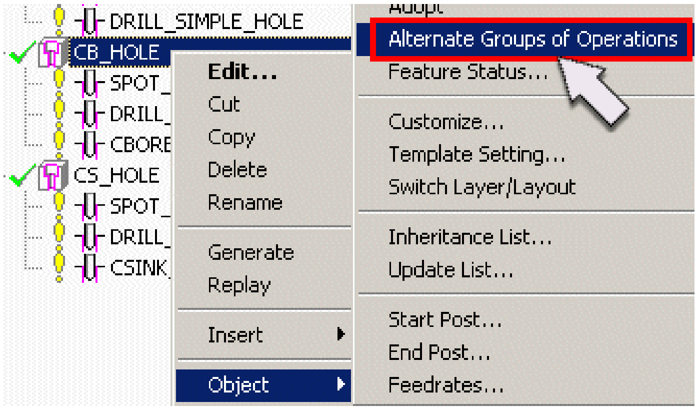
Alternate Groups of Operations allows you to create groups of operations that perform machining tasks that are different than the feature templates that are provided as a standard with Hole Making.

As an example, you can define one group of operations that drills before countersinking and another group of operations that countersinks before drilling. When defining the hole making process, you can then choose the appropriate group of operations for machining that type of particular feature. You would then use this option when you want to minimize the number of feature templates being used.

Alternate Groups of Operations are created in the **Geometry** view of the Operation Navigator by highlighting a feature group, and then by selecting **MB3 → Object → Alternate Groups of Operations**. In the **Alternate Groups of Operations** dialog, the upper list which is displayed contains the operations allowed for the selected feature group. The lower list displayed contains the alternate groups and their respective operations in their current order.

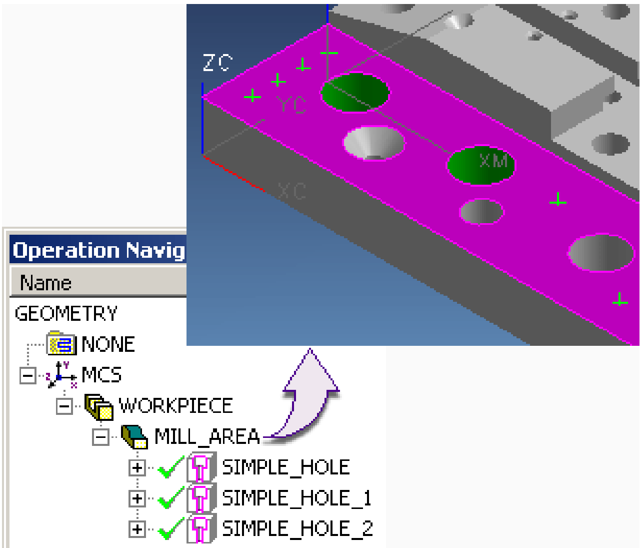
You can create the alternate groups in the lower list by choosing the **New Group** icon. You may add operations to the alternate group by highlighting the operation in the **Available Operations** list and then choose the **Move Operation to Group** icon.

To delete operations or alternate groups, use the **Delete Group/Operation** icon. For specific Knowledge Fusion rules, use the **Alternate Group Rule** field to write the KF rule that will be used to decide which group of operations will machine that particular feature group. For example, a rule might be as follows: **If the material of the part is steel, Alternate Group one is TRUE else False**.



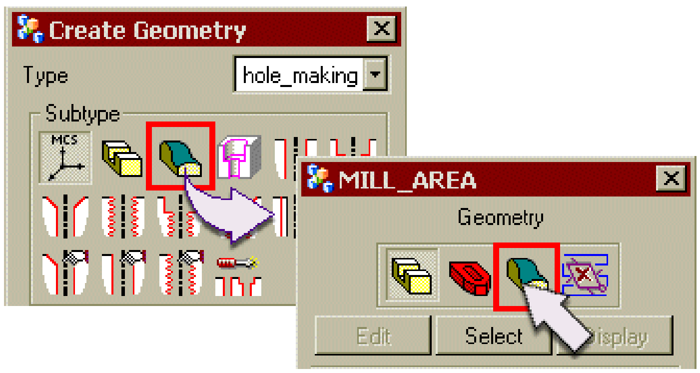
Cut Area

Cut Area allows you to reduce the scope of machineable feature identifications by creating a **MILL_AREA** group that identifies specific faces to be machined. When there is no **Cut Area** geometry identified, the feature identification will be the entire part. The **WORKPIECE** group should be specified as the parent when creating this **MILL_AREA** group.



9

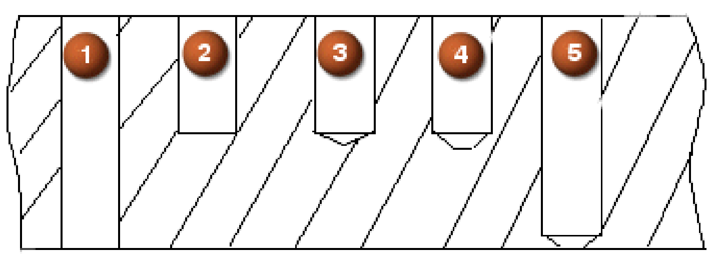
The face geometry defining the **Cut Area** is specified within the **MILL_AREA** group. Selected points and arcs will be identified during the feature grouping regardless of part or **Cut Area** geometry since **Cut Area** geometry is based on faces and part geometry is based on bodies. Points and arcs are considered as neither faces or bodies.



Feature Recognition

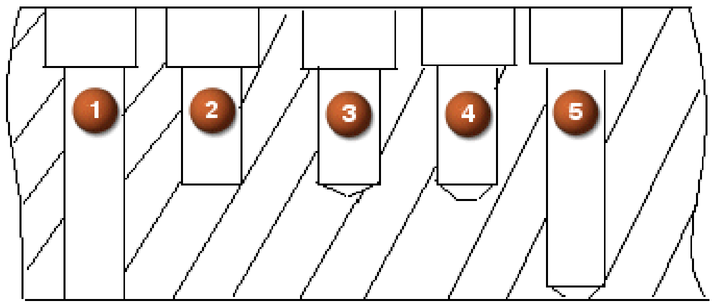
Hole Making will recognize simple, counterbored and countersunk hole features that have not been explicitly created as a Unigraphics feature. These features are based on cylinders with diameter and depth parameters, cones with maximum and minimum cone diameters as well as cone angle and planes.

A simple hole is created from a cylinder with planes designating the top and bottom of the hole.



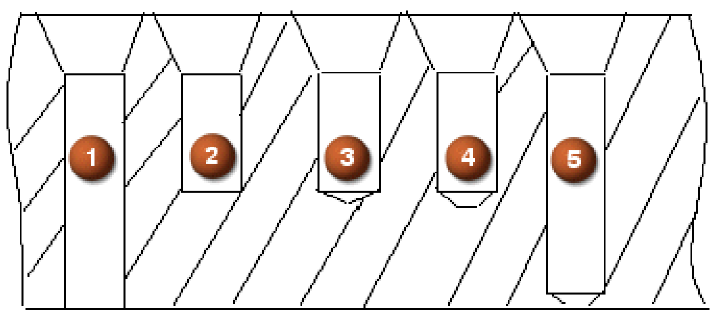
- (1) Cylinder
- (2) Cylinder and plane
- (3) Cylinder and cone
- (4) Cylinder, Cone, Plane
- (5) Cylinder, Cone, Plane

A counterbored hole consists of a cylinder and plane, which designate the counterbore with the addition of a simple hole feature.



- (1) Cylinder, plane, simple hole
- (2) Cylinder, plane, simple hole
- (3) Cylinder, plane, cone, simple hole
- (4) Cylinder, plane, cone, simple hole
- (5) Cylinder, plane, cone, simple hole

A countersink hole consists of a cone with the addition of a simple hole.



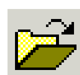
- (1) Cone, plane, simple hole
- (2) Cone, plane, simple hole
- (3) Cone, plane, cone, simple hole
- (4) Cone, plane, cone, simple hole
- (5) Cone, plane, cone, simple hole

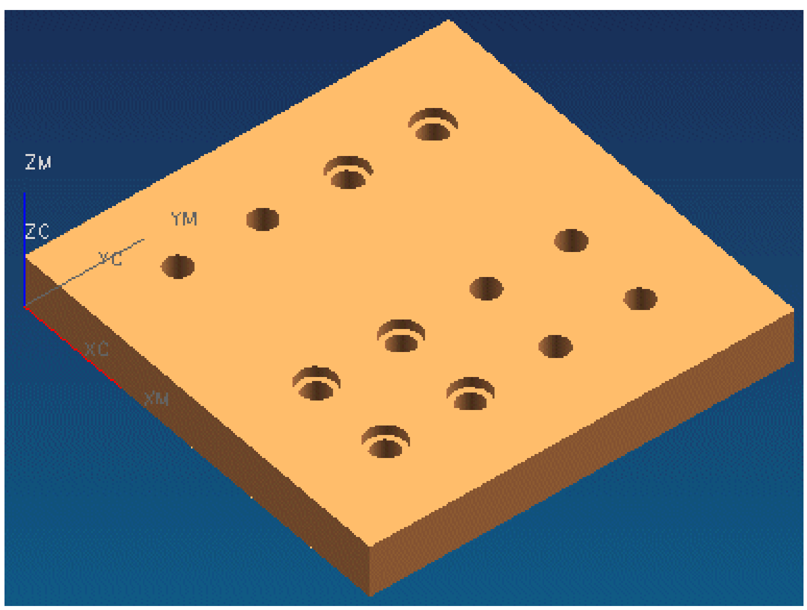
To use this functionality, declare all simple hole, counterbored hole and countersunk hole shapes as machinable features, create the operation and generate the tool path.

Activity 9—4: Feature Recognition

In this activity you will utilize the Hole Making processor to machine simple, counterbored and countersunk hole shapes that have not been created explicitly as features. You will first replay existing operations that will machine the holes that have been modelled as features. You will then use the Feature Recognition option to recognize holes that were created by subtracting cylinders from the solid body. Finally, you will create operations that will machine all of the holes in the part.

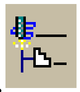
Step 1 Open the part ama_hole_making_feature.

- ❑ If necessary, start Unigraphics NX 2.
- ❑ Use **File → Open**. 
- ❑ Navigate to your parts folder and open the file.
- ❑ Briefly examine the part.

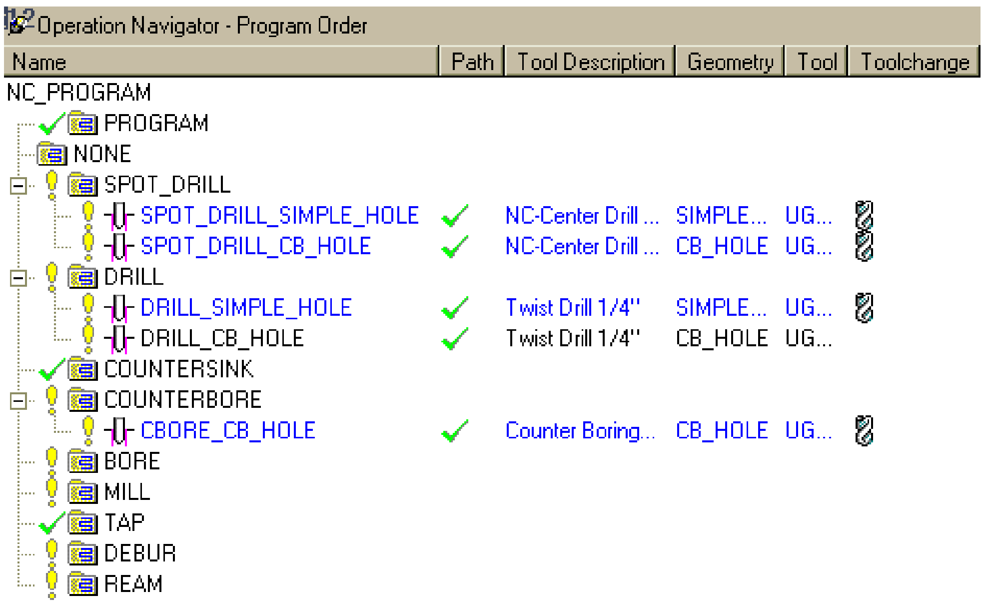


Step 2 If necessary, enter the Manufacturing Application and display the Operation Navigator.

- ❑ Choose **Application → Manufacturing** from the Menu bar.

❑ Choose the **Operation Navigator** icon from the **Resource** bar. 

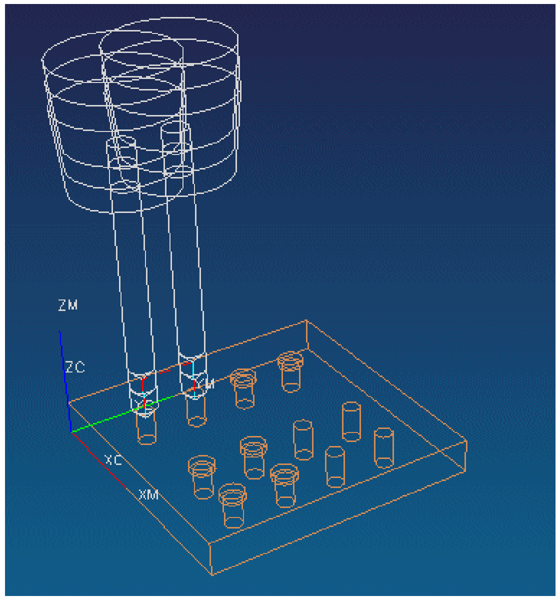
❑ If necessary, change to the **Program Order** view of the Operation Navigator and expand the **SPOT _DRILL**, **DRILL** and **COUNTERBORE** objects.



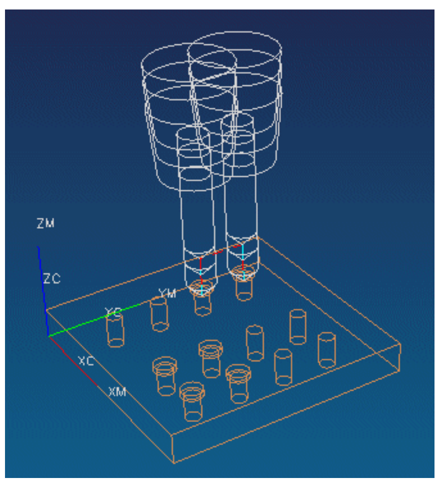
You will replay existing operations and observe that only features which have been modeled are machined.

Step 3 Replaying the SPOT_DRILL_SIMPLE_HOLE and SPOT_DRILL_CB_HOLE operations.

❑ Highlight the **SPOT_DRILL_SIMPLE_HOLE** operation, using **MB3**, **Replay** the operation.



❑ Highlight the **SPOT_DRILL_CB_HOLE** operation, using **MB3**, **Replay** the operation.



Note that only four holes, that were previously modelled as features, are recognized as machineable features and have tool paths associated with them.

❑ **Replay** the tool paths for the **DRILL** and **COUNTERBORE** objects as well.

You will now create operations that will machine all of the holes that are contained in the part.

Step 4 Creating operations that will machine non modelled as well as modelled features.

In order for the Hole Making processor to select the holes that were not modelled as simple hole features, you will need to use the **Feature Recognition** option.

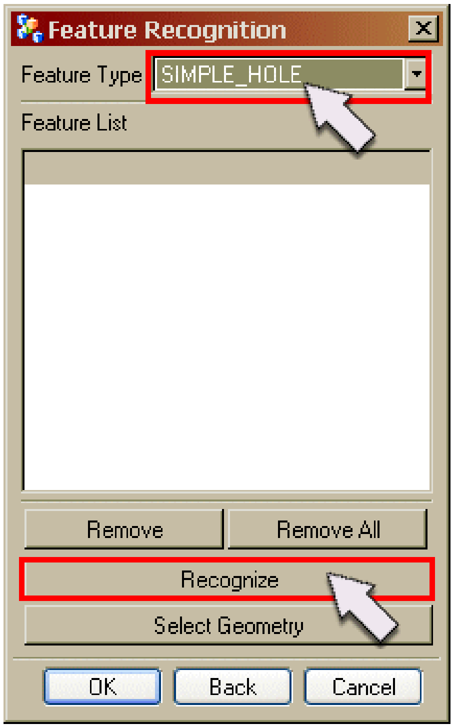
❑ Choose **Tools** → **Recognize Features** from the tool bar.

The **Feature Recognition** dialog is displayed.

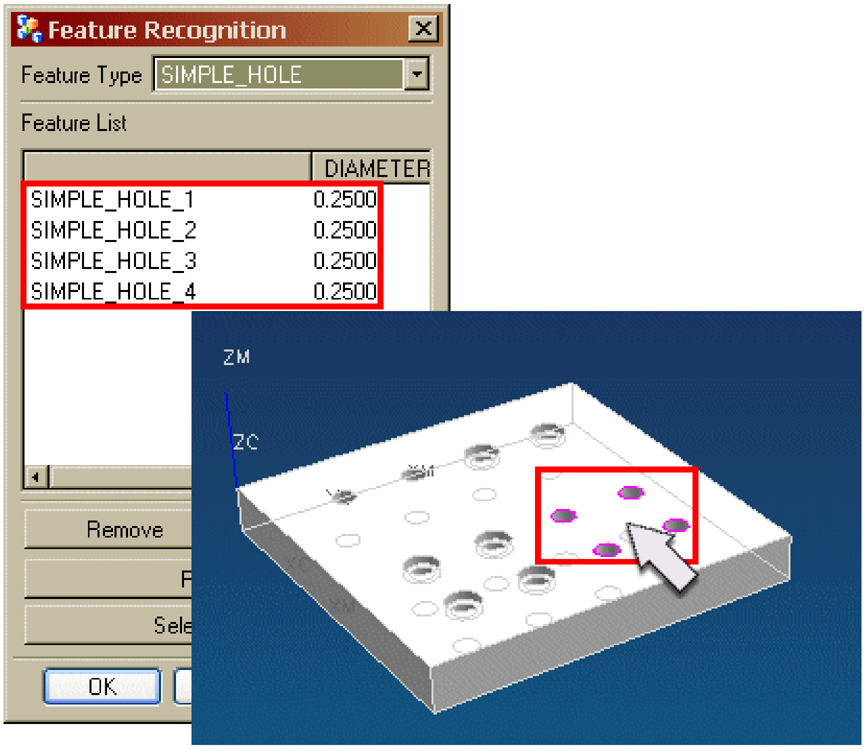
❑ Set the **Feature Type** to **SIMPLE_HOLE**.

❑ Choose **Recognize**.

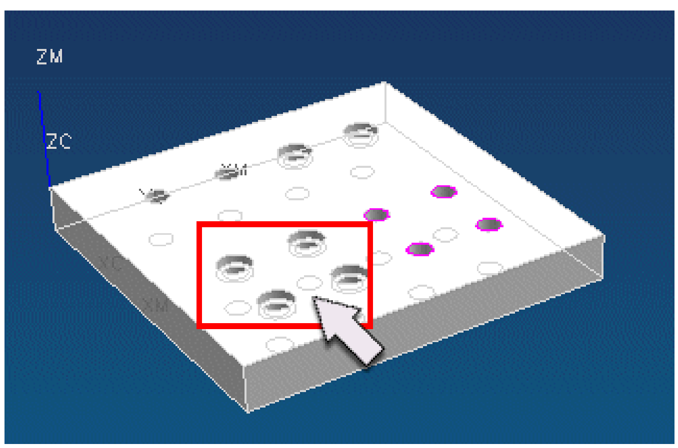
The **Feature Recognition** dialog is displayed.



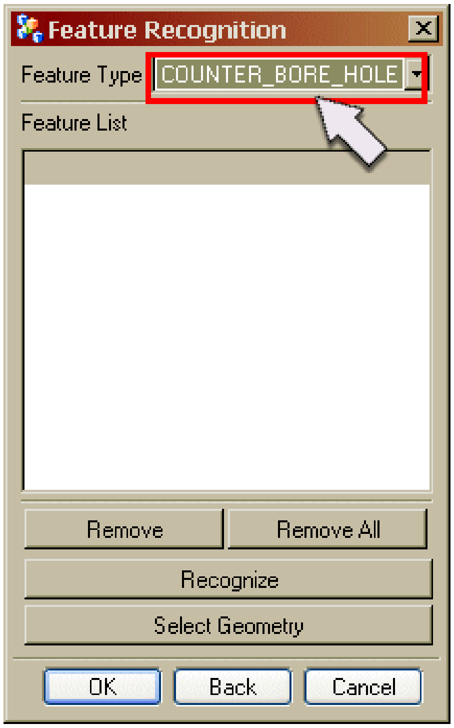
The four holes are now recognized as being simple holes, are displayed in the **Feature List** box of the **Feature Recognition** dialog and are highlighted on the part.



You will now use the **Feature Recognition** function to recognize the counterbored hole feature types as machineable features.

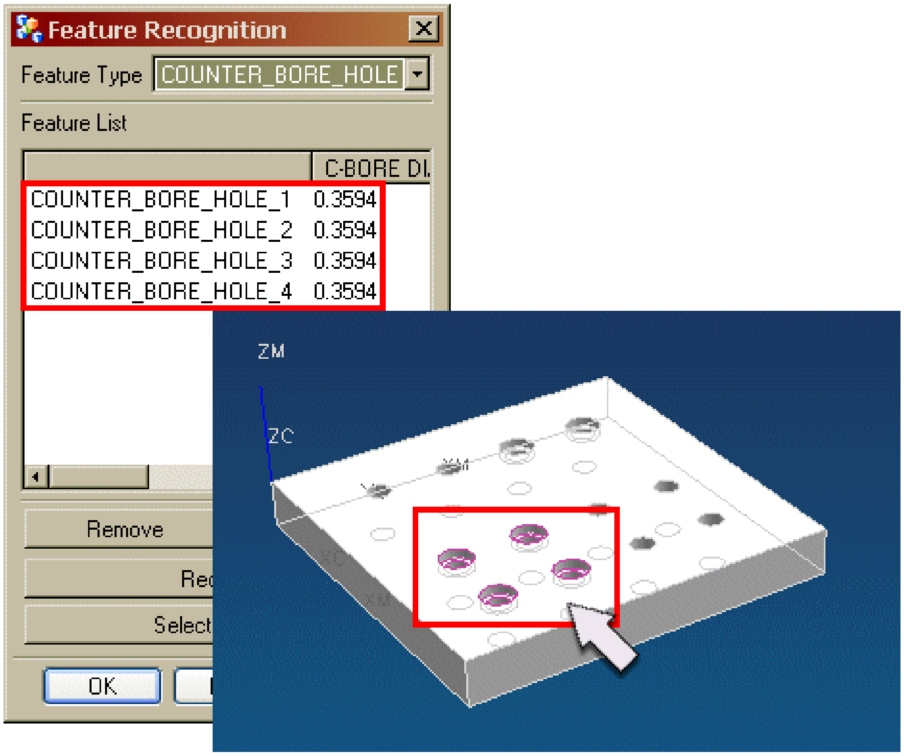


❑ Set the **Feature Type** option to C**OUNTER_BORE_HOLE**.



❑ Choose **Recognize**.

The four holes are now recognized as being counter bored holes, are displayed in the **Feature List** box of the **Feature Recognition** dialog and are highlighted on the part.



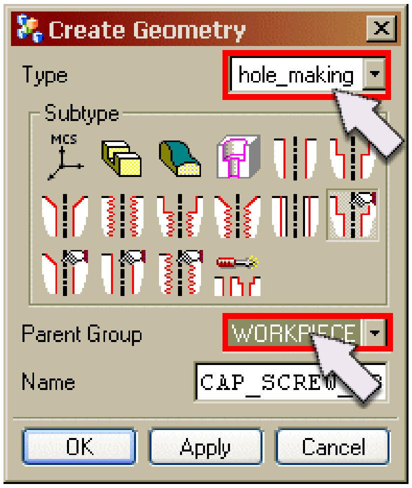
❑ Choose **OK** on the **Feature Recognition** dialog.

You will now create the operations that will machine all holes that are contained in the part.

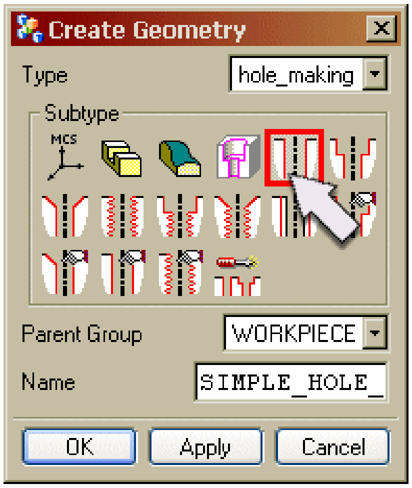
❑ Choose the **Create Geometry** icon on the tool bar.

❑ If necessary, set the **Type** to **hole_making** and the **Parent Group** to **WORKPIECE**.

9

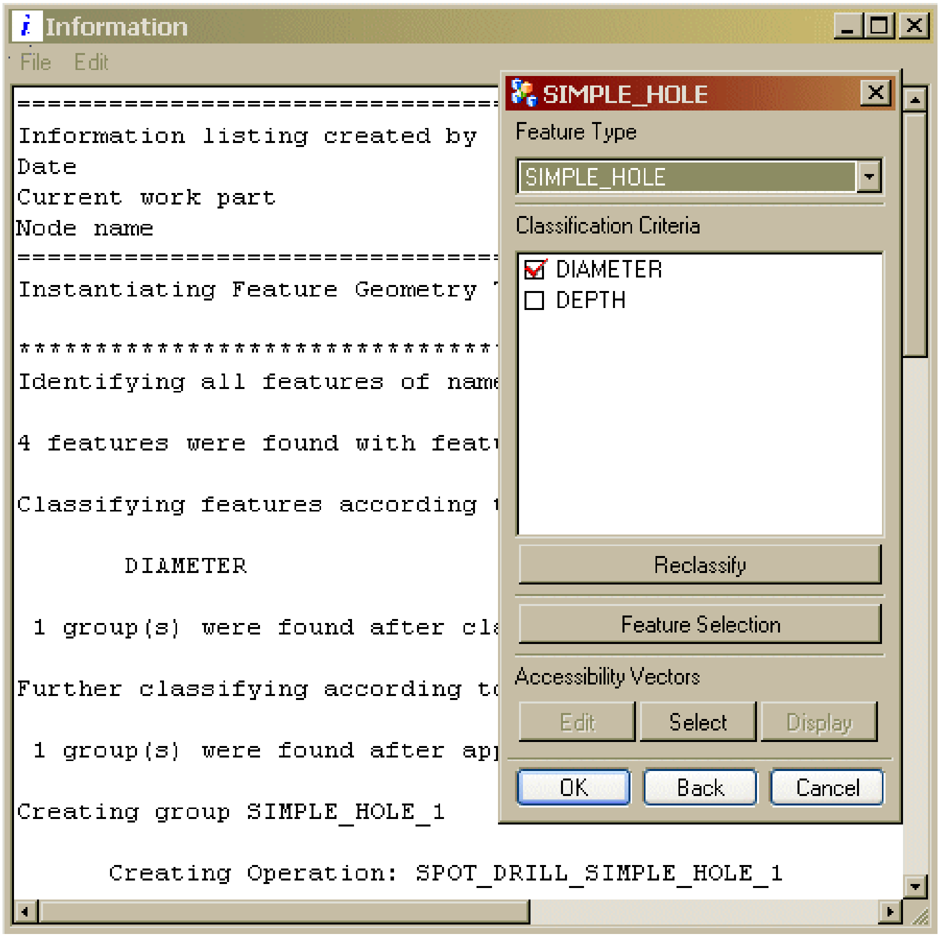


❑ Choose the **SIMPLE_HOLE** icon from the **Create Geometry** dialog.



❑ Choose **OK** from the **Create Geometry** dialog to begin generating the operations.

The **Information Window** and **SIMPLE_HOLE** dialog is displayed.



❑ Dismiss the **Information** window and choose **OK** on the **SIMPLE_HOLE** dialog.

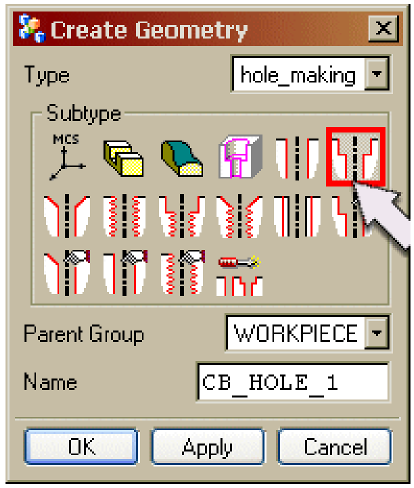
The diameter of the hole will be used as the classification criteria. You will now create the operations for the counter bored holes.

❑ Choose the **Create Geometry** icon from the tool bar.

The **Create Geometry** dialog is displayed.

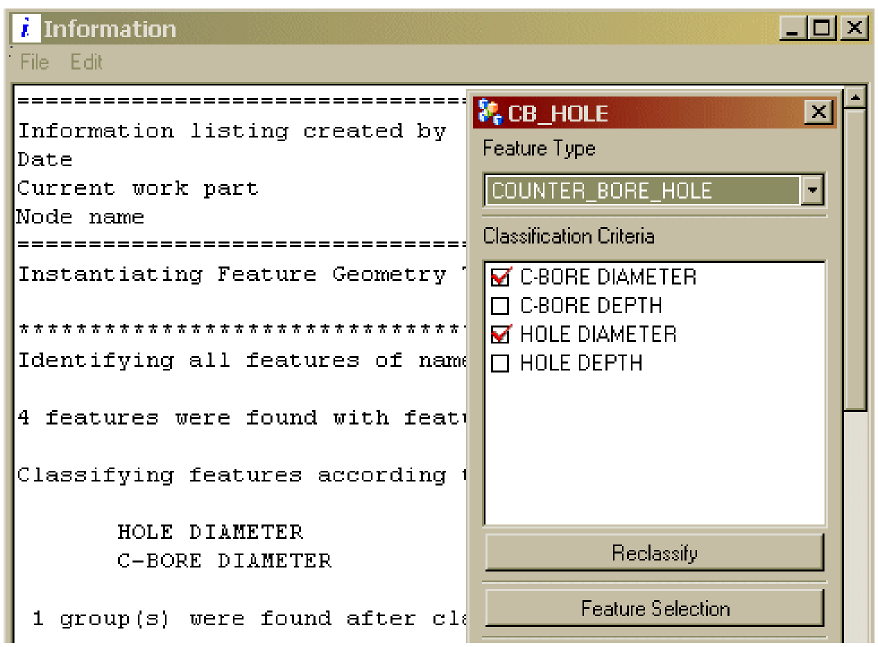
9

❑ Choose the **CB_HOLE** icon.



❑ Choose **OK** from the **Create Geometry** dialog to begin generating the operations.

The **Information Window** and **CB_HOLE** dialog is displayed.

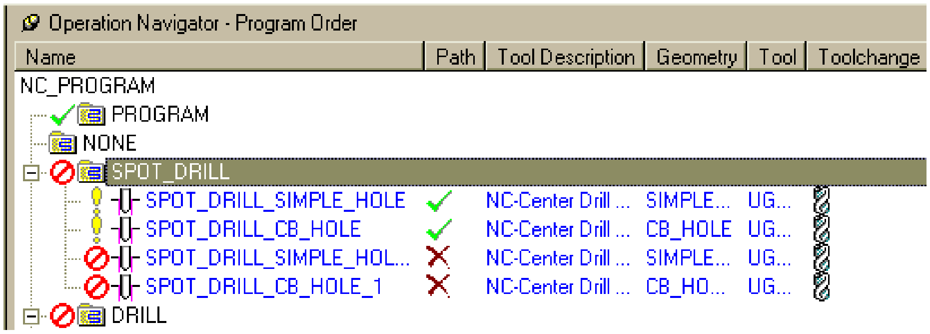


❑ Dismiss the **Information** window and choose **OK** on the **CB_HOLE** dialog. The diameter of the hole and counter bore will be used as classification criteria.

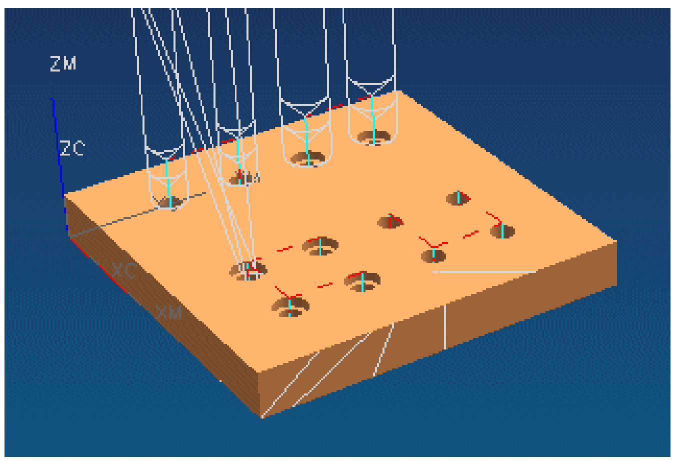
9

You will now generate the tool paths.

❑ Highlight the **SPOT_DRILL** object, using **MB3 → Generate** the operation.



The tool path is generated.



❑ Repeat the above process for the **DRILL** and **COUNTERBORE** operations.

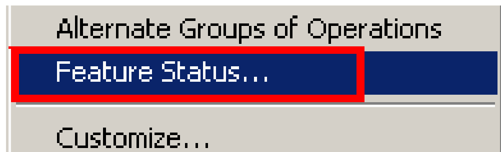
❑ **Close**, without saving, the part file.

This concludes the activity.

Feature Status

The **Feature Status** option allows you to identify features that have not been machined. This can occur due to collisions with clamps or the tool holder.

To check **Feature Status**, choose the feature group icon  in the Operation Navigator and use **MB3 → Object Feature Status**.



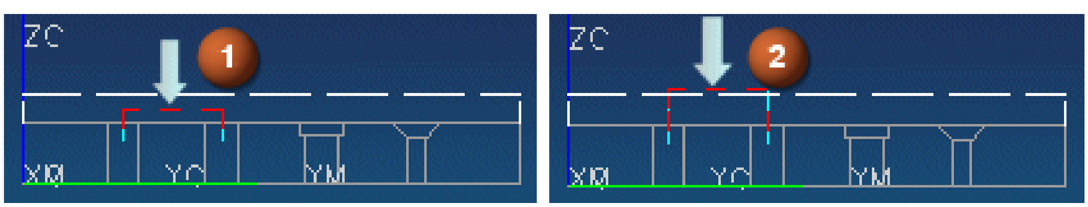
Holder Types

Holder Type is found in the **Tap** tool dialog and allows you to specify either a rigid or float type tapping holder. For legacy operations, **Holder Type** needs to be customized in the **Tap** dialog.

3D in Process Work Piece

Hole Making allows you to create and use an associative In Process Work piece (IPW). Using an IPW assures that blank geometry is recognized and each subsequent cut region is based on remaining material. This will prevent the tool from colliding with any material that remains from previous operations.

The following illustration represents a tool path (1) not using and (2) using an IPW:

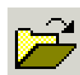


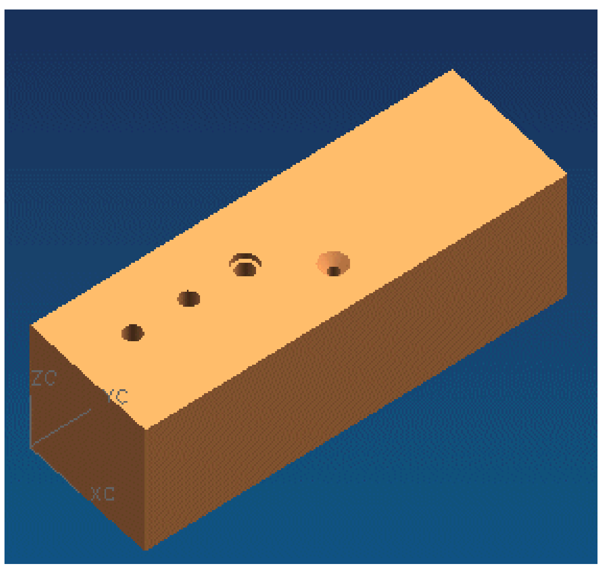
When the IPW is not used, the tool will traverse through the material. When using the IPW the tool traverses above and clears any material which remains. In sequential operations, an IPW is created and then used as blank geometry in the operation which follows. The resultant IPW can be displayed for each operation.

Activity 9–5: Using the IPW in Hole Making

In this activity, you will first examine a Hole Making operation and observe that the tool collides with the work piece. You will then create and use an IPW that will prevent the tool form colliding with any material that may remain from previous operations.

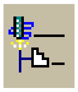
Step 1 Opening the part file ama_hole_making_ipw.

- ❑ If necessary, start Unigraphics NX 2.
- ❑ Use **File → Open**. 
- ❑ Navigate to your parts folder and open the file.
- ❑ Briefly examine the part.

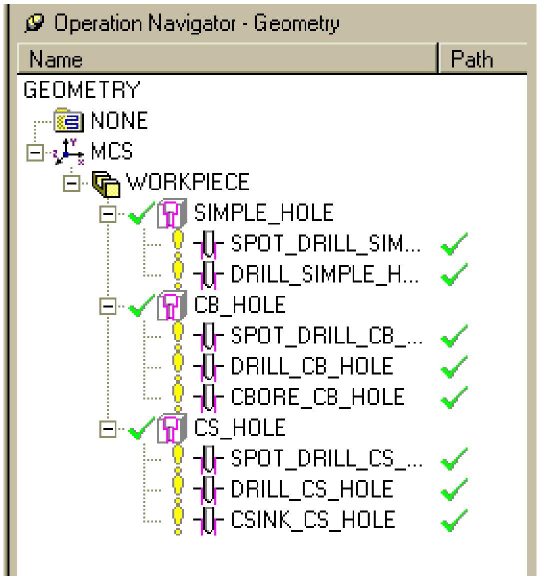


Step 2 If necessary, enter the Manufacturing Application and display the Operation Navigator.

- ❑ Choose **Application → Manufacturing** from the Menu bar.

❑ Choose the **Operation Navigator** icon from the **Resource** bar. 

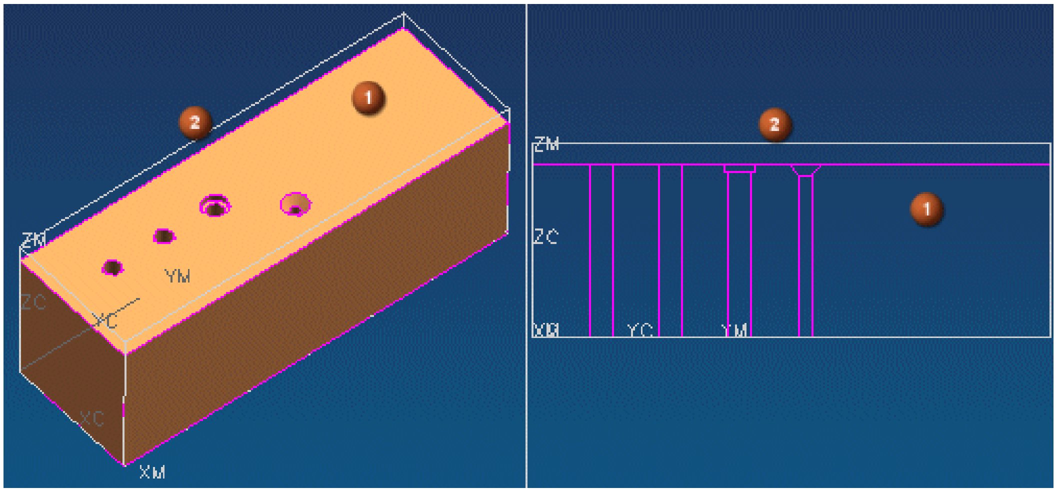
❑ If necessary, change to the **Geometry** view of the Operation Navigator and expand the group objects.



You will notice that operations exist to spot drill, drill, counter bore and counter sink the holes on the part. Next you will verify the part and blank material and will replay the operation **SPOT_DRILL_SIMPLE_HOLE**.

Step 3 Verify the part and blank material and then replay the SPOT_DRILL_SIMPLE_HOLE operation.

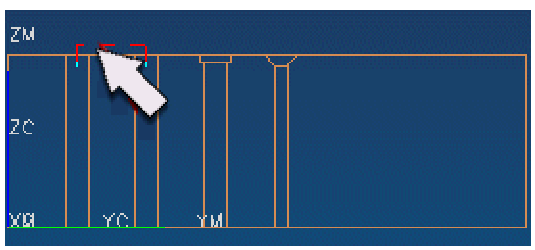
❑ Double-click the **WORKPIECE** object and display the **Part** (1) and **Blank** (2) material.



The operation recognizes **Blank** geometry only if the **IPW** is utilized. In this example, the **Blank** geometry, which extends above the actual **Part** surface, in not recognized and the tool collides with the **Blank** material.

You will now replay the tool path, and observe how the tool collides with the **Blank** material.

- ❑ Choose **OK** on the **MILL_GEOM** dialog.
- ❑ Highlight the **SPOT_DRILL_SIMPLE_HOLE** operation, using **MB3**, **Replay** the operation.



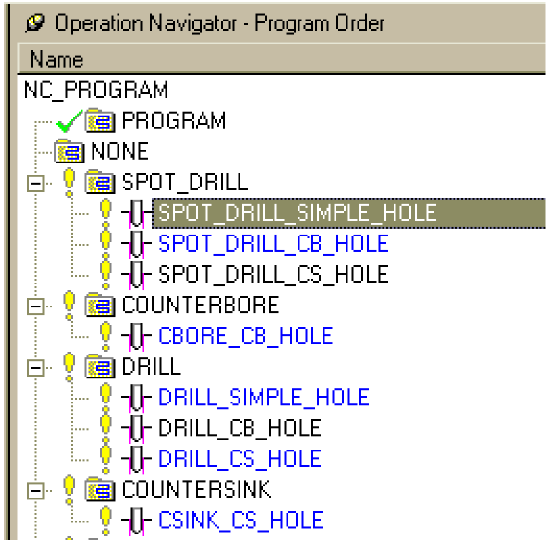
Note how the tool, when positioning from one hole to the next, does not retract high enough, to avoid colliding with the **Blank** material.

You will now activate the IPW, which will keep the tool from colliding with the **Blank** material.

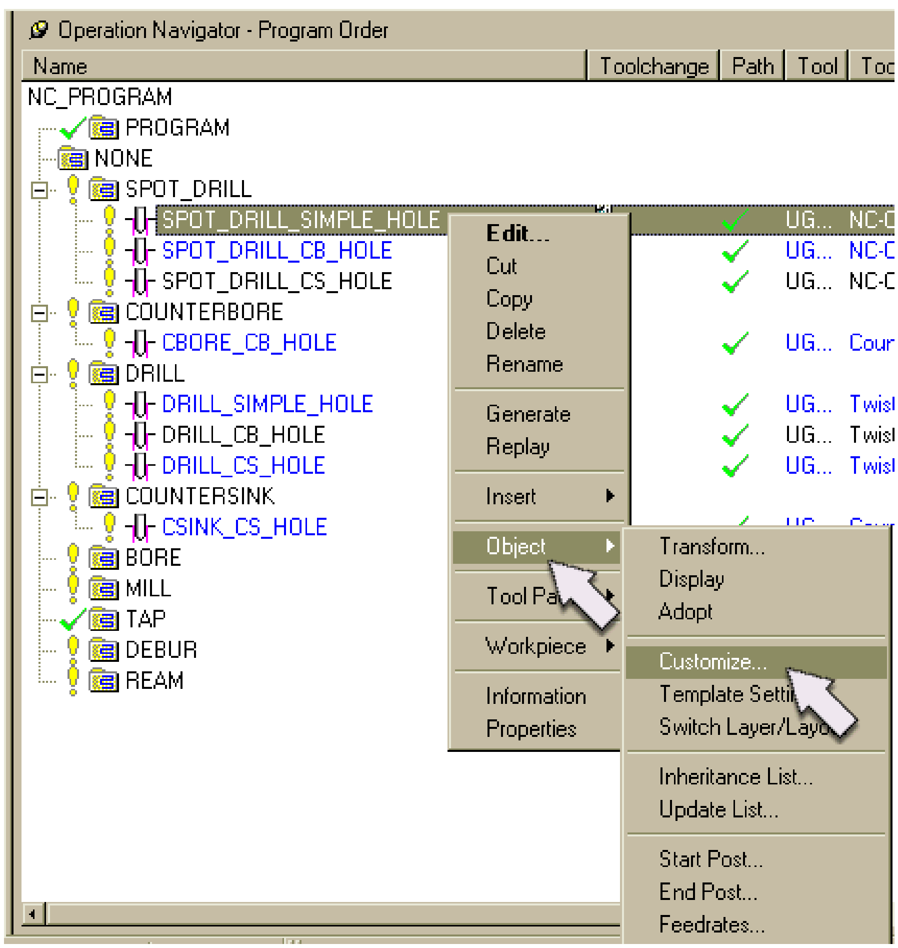
NOTE **Use 3D IPW** option ***must*** *be* customized into each **HOLE_MAKING** operation dialog.

Step 4 Incorporating and using the IPW.

- ❑ Change to the **Program Order** view of the Operation Navigator and if necessary expand all objects.

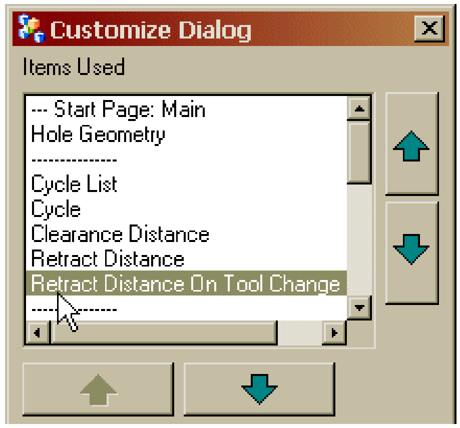


❑ Highlight the **SPOT_DRILL_SIMPLE_HOLE** operation and using **MB3**, select **Object → Customize.**



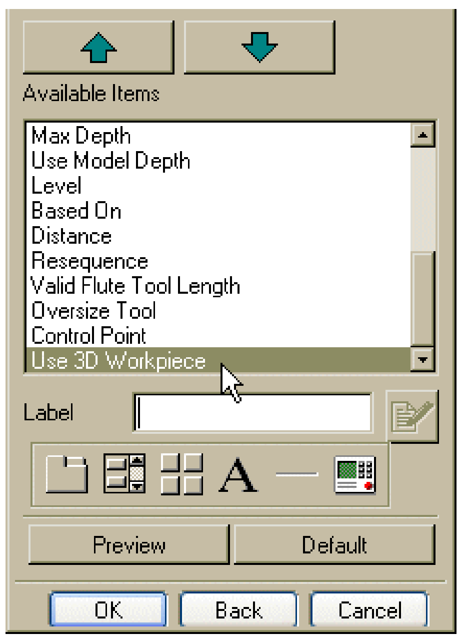
9

❑ Choose **Retract Distance On Tool Change** from the **Items Used List** of the **Customize Dialog**.



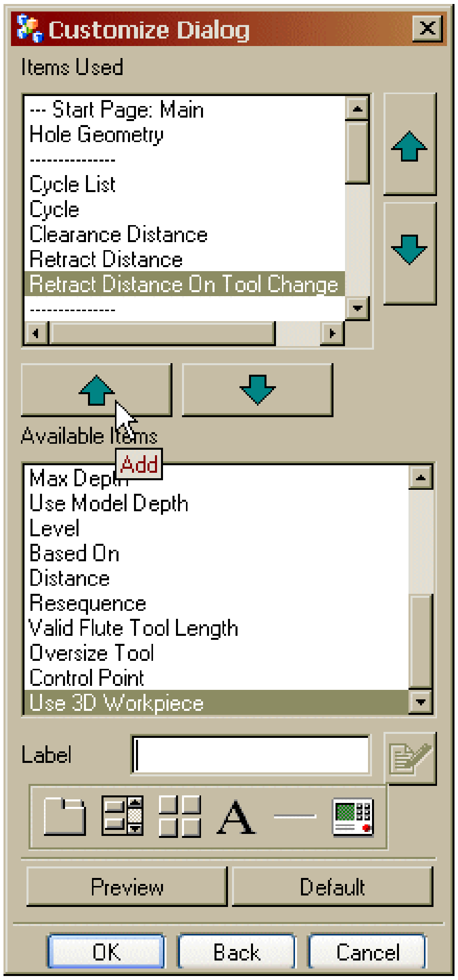
This will be the area of the **Hole_Making** dialog in which the **Use 3D IPW** option will appear.

❑ Choose **Use 3D Workpiece** from the **Available Items** list.



❑ Choose the **ADD** arrow to move the **Use 3D Workpiece** option into the **Items Used** list.

9



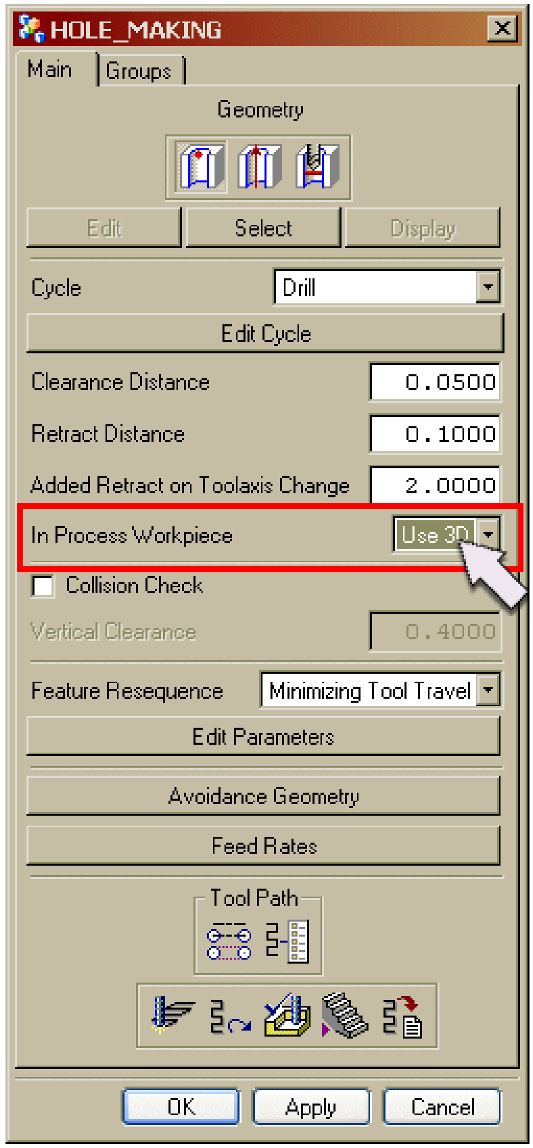
❑ Choose **OK** to accept the **Customize Dialog**.

You will now activate the **Use 3D IPW** that will enable this operation to use the IPW.

❑ From the Operation Navigator, double-click on the **SPOT_DRILL_SIMPLE_HOLE** operation.

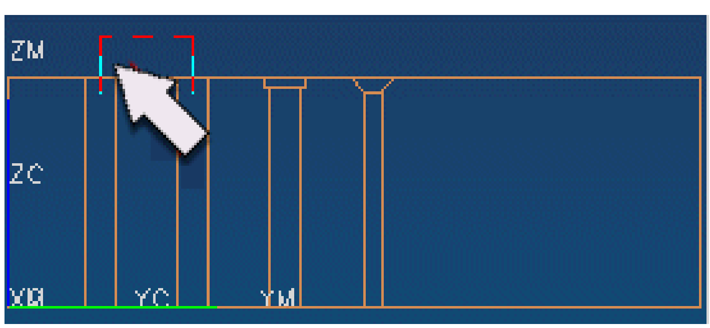
The **HOLE_MAKING** dialog is displayed.

❑ Set the **In Process Workpiece** option to **Use 3D**.



You will now generate the operation.

❑ Choose the **Generate** icon.



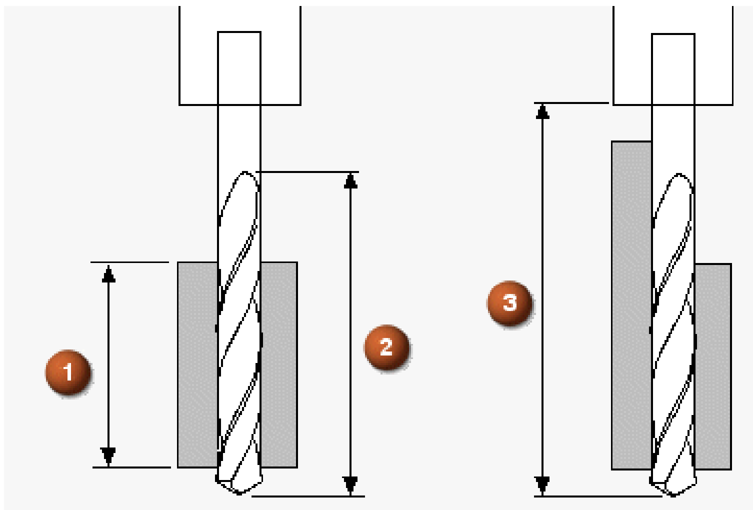
Notice how the tool retracts to a higher position when moving from one hole to the next.

- ❑ Choose **OK** on the **HOLE_MAKING** dialog.
- ❑ **Close**, without saving the part file.

  This concludes this activity.

Maximum Cut Depth and Extended Length

Check Flute Length and **Check Tool Length** are parameters that are used to verify that the appropriate tools are retrieved into the part when performing tool queries from the tool library. **Check Flute Length** validates that the tools copied from the tool library have a **flute length** that is long enough for the required cut depth. **Check Tool Length** validates that the tools copied from the tool library have a **tool length** that is long enough to avoid collisions between the tool holder and adjacent walls of the part.



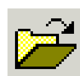
- (1) Cut Depth
- (2) Flute Length
- (3) Tool Length

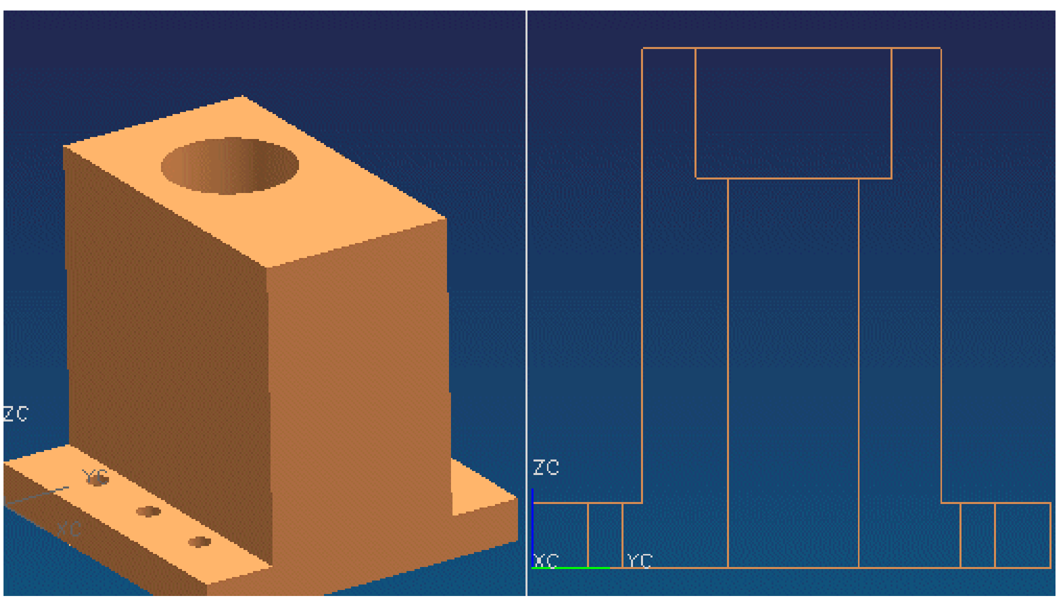
NOTE **Check Flute Length** and **Check Tool Length** must be customized into the **HOLE_MAKING** dialog.

Activity 9–6: Maximum Cut Depth and Extended Length

In this activity you will first examine a Hole Making operation and observe that the tool holder will collide with the work piece since the tool does not have enough length to drill through the part. You will then search the tool library for tools with appropriate tool lengths that can perform the required operation without colliding with the part.

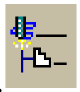
Step 1 Open the part ama_hole_making_mx_ct_dp.

- ❑ If necessary, start Unigraphics NX 2.
- ❑ Use **File → Open**. 
- ❑ Navigate to your parts folder and open the file.
- ❑ Briefly examine the part.

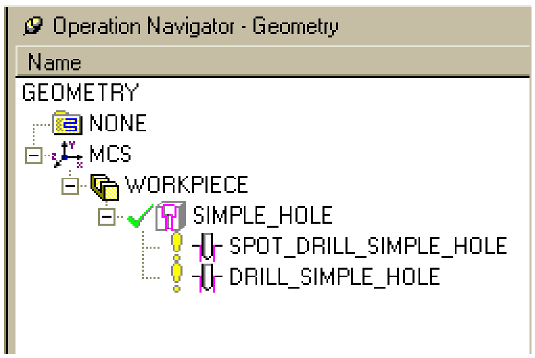


Step 2 If necessary, enter the Manufacturing Application and display the Operation Navigator.

- ❑ Choose **Application → Manufacturing** from the Menu bar.

- ❑ Choose the **Operation Navigator** icon from the **Resource** bar. 

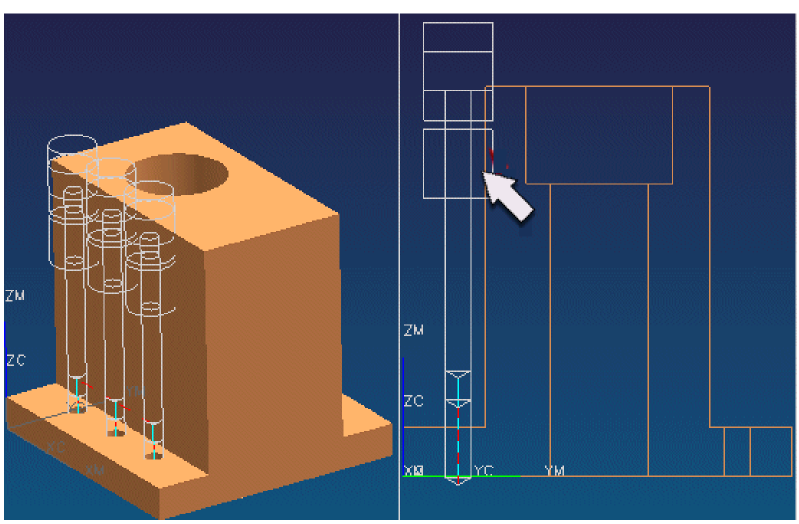
- ❑ If necessary, change to the **Geometry** view of the Operation Navigator and expand the group objects.



You will replay the **DRILL_SIMPLE_HOLE** operation and observe how the tool holder collides with the part. When the tool was selected originally from the library search, consideration was not given to check the flute or tool length.

Step 3 Replaying the DRILL_SIMPLE_HOLE object.

- ❑ Highlight the **DRILL_SIMPLE_HOLE** operation, using **MB3**, **Replay** the operation.



9

Note how the tool, when positioning from one hole to the next, does not retract high enough to avoid colliding with the part. By using the **Check Tool Length** option when searching the library for tools, collisions like those previously displayed can be avoided.

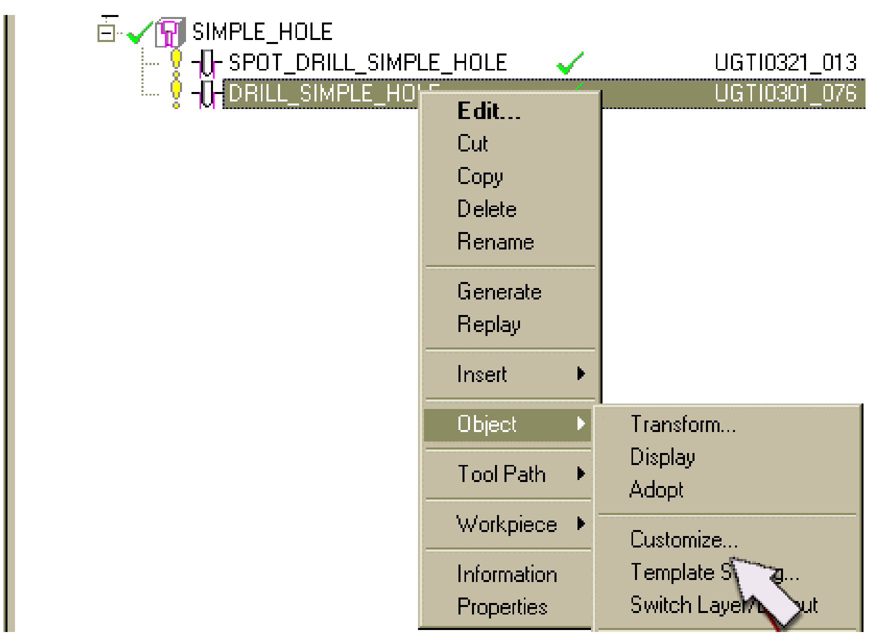
If appropriate tools are not found in the search, you will be prompted to create a new tool.

You will now use the **Check Flute Length** and **Check Tool Length** options to search the tool library for proper length tools.

NOTE **Check Flute Length** and **Check Tool Length** option must be customized into each **HOLE_MAKING** operation dialog.

Step 4 Incorporating and using the Check Flute Length and Check Tool Length options.

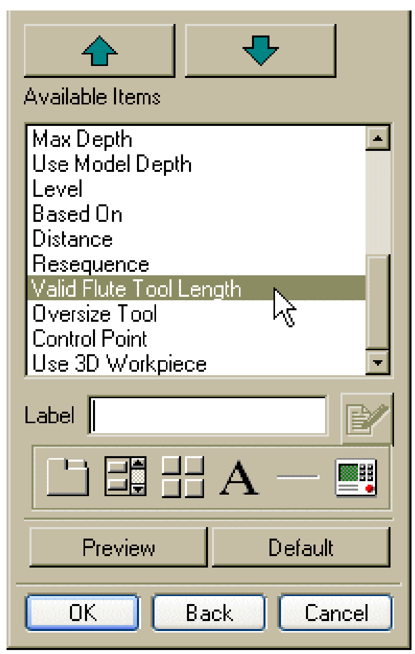
❑ Highlight the **DRILL_SIMPLE_HOLE** operation, and using **MB3**, select **Object → Customize**.



❑ **Choose Tool Class Query** from the **Items Used List** of the **Customize Dialog**.

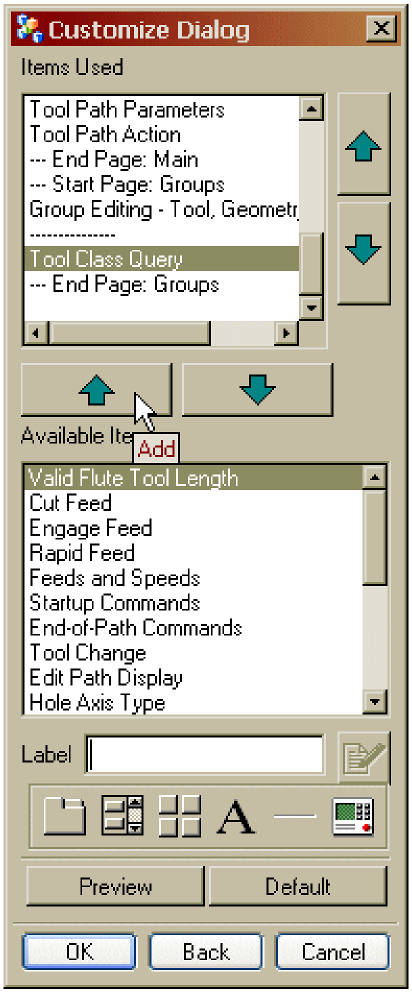
This will be the area of the **Hole_Making** dialog in which the **Check Flute Length** and **Check Tool Length** option will appear.

❑ Choose **Valid Flute Tool Length** from the **Available Items** list.



9

❑ Choose the **ADD** arrow to move the **Valid Flute Tool Length** option into the **Items Used** list.



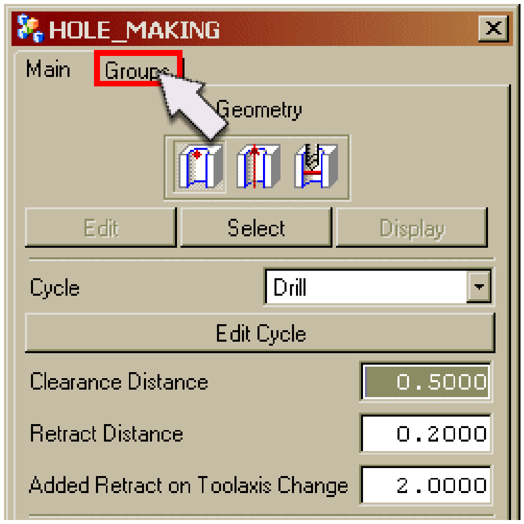
❑ Choose **OK** to accept the **Customize** dialog.

You will now retrieve tools using the **Check Flute Length** and **Check Tool Length** options when querying the library.

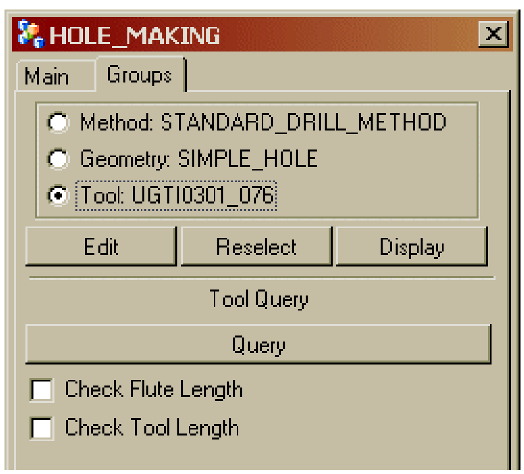
❑ From the Operation Navigator, double-click on the **DRILL_SIMPLE_HOLE** operation.

The **HOLE_MAKING** dialog is displayed.

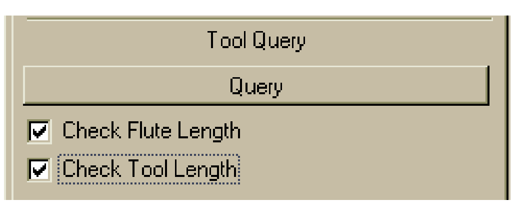
❑ Choose the **Groups** tab at the top of the **HOLE_MAKING** dialog.



The **Check Flute Length** and **Check Tool Length** options are turned off. You will turn these options on and then query the library for the proper tooling.

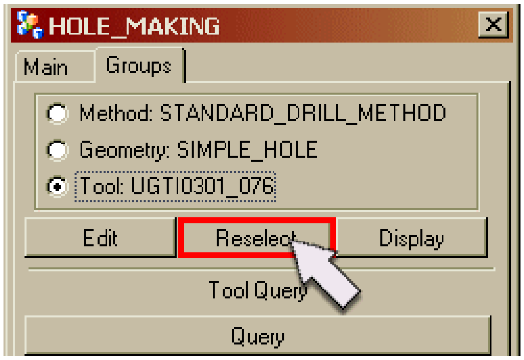


❑ Turn the **Check Flute Length** and **Check Tool Length** options **ON**.

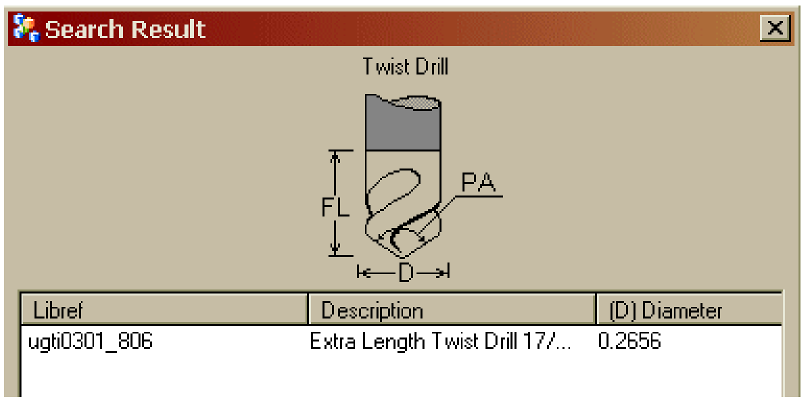


9

❑ Choose **Reselect**.



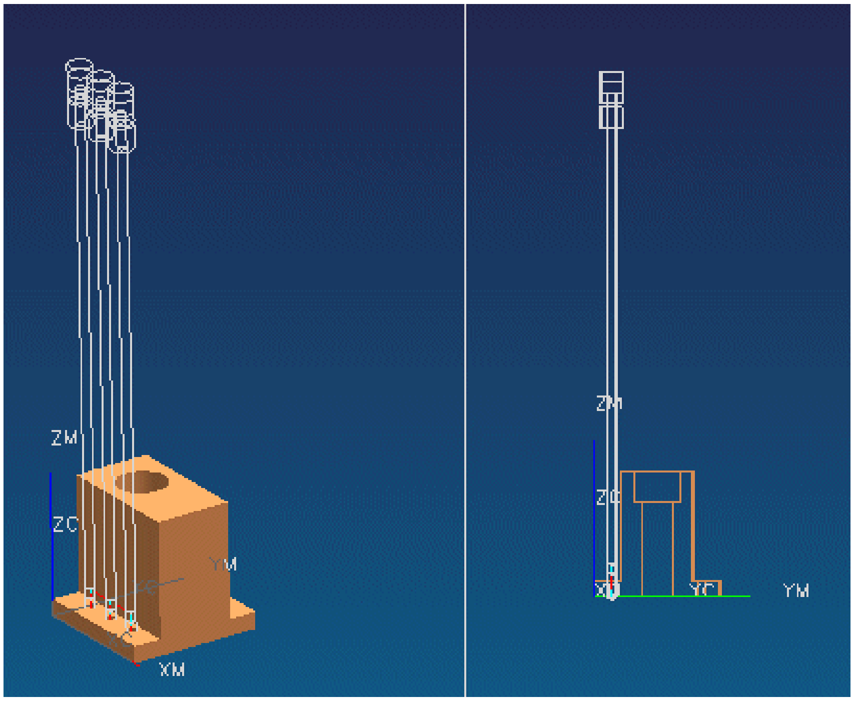
The **Search Result** dialog is displayed with the proper tool.



❑ Choose the tool from the list and then choose **OK** on the **Search Result** dialog.

❑ Choose the **Main** tab from the top of the **HOLE_MAKING** dialog.

❑ Choose the **Generate** icon.



Notice how the extended tool length now clears the part.

❑ Choose **OK** on the **HOLE_MAKING** dialog.

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

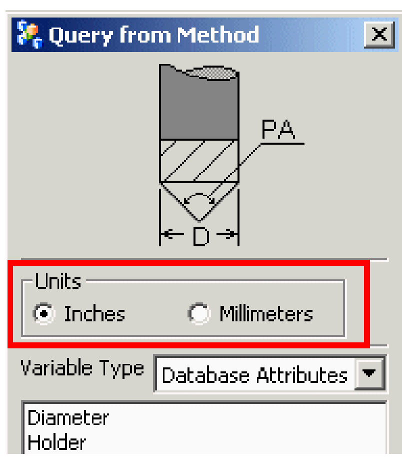
This concludes the activity.

Multiple Selection

Features can be selected using the Class Selection dialog when appending or removing them from a feature group. This allows the selection of several features at once.

Inch and Metric Availability within Tool Query

Units options are selectable from the **Query From Method** dialog. The toggle buttons available allow you to specify Inches or Millimeters as the units of the tools to be retrieved into the part when performing queries from the tool library. This allows you to search for metric tools in an inch part and inch tools in a metric part. If the retrieved tool has a different unit from that of the part, the tool parameter values are converted into the units of the part. This setting is applicable only to queries that are being used in the ASCII tool database. To specify the **Units** query parameter, double-click on a **Machining Method** object in the Operation Navigator. Select **Query** and then select either the **Inches** or **Millimeters** button.



Then accept the **Query** from the **Method** dialog. This setting will be saved with the Machining Method object selected and is then applied to that object when the query is executed and when subsequent tools are retrieved.

To execute a tool query, double-click an operation inside the **Machining Method** object that has been edited. Select the **Groups** tab, turn the **Tool** button **ON** and choose **Reselect**. Then select the tool you desire from the **Search Result** dialog. If the query cannot find any tools in the specified Inches or Millimeters library, then the **Reselect Tool** dialog will be displayed.

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. Using Hole Making greatly simplifies the process of making holes, regardless of the type of application.

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
- Utilized Alternate Groups of Operations and Cut Area features of Hole Making to further limit the types of holes to be machined
- Used Feature Recognition and Feature Status to identify and utilize specific features in the hole making process
- Used the IPW to avoid collisions

9

Integrated Simulation and Verification

Lesson 10

PURPOSE

In this lesson, you will learn how to use the Integrated Simulation and Verification module to verify tool paths by means of machine tool simulation.

OBJECTIVES

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

- Interact with the Integrated Simulation and Verification module
- Configure and mount parts and fixtures on existing machine tools configured for simulation
- Simulate tool paths

This lesson contains the following activities:

10

Integrated Simulation and Verification Overview

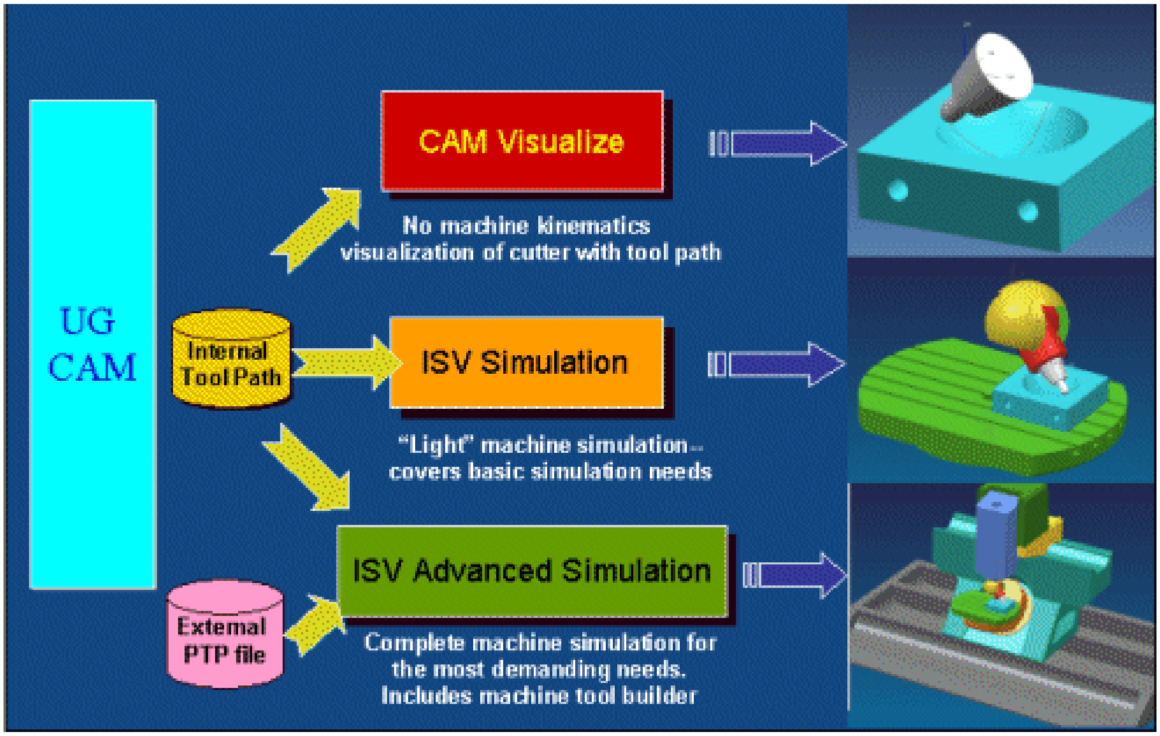
The Integrated Simulation and Verification module (IS&V) allows you to simulate a machine tool with an actual piece part, giving you an overview of the entire machining process. The simulation process animates the exact machine tool motions, taking into account controller functions and cutting tool configurations.

IS&V features collision checking which allows collision detection between machine components, fixtures, tools, parts and the in-process work piece. You may also view machine controller functions including macros, subroutine calls, cycles and function M, G and H commands.

IS&V can improve the quality of machining processes by allowing the comparison of the designed part to the part which is being manufactured. Reduction in cost can be obtained by the elimination of expensive and time consuming dry runs; reduction in manual operator intervention; and the reduced risk of expensive damage to machine tools, fixtures and parts by elimination of collisions.

IS&V consists of the following components:

- Visualize
- Simulation
- Advanced Simulation
- Machine Tool Builder
- Machine Tool Driver
- Setup Configurator



Visualize

- basic level of visualization of tool paths
- represents tool only moves, no kinematic model of machine tool
- performs gouge and collision checking with part and IPW
- display of material removal is 2D only
- optionally produces IPW used in roughing

Simulation

- basic level of tool path simulation
- uses kinematic model of machine tool - simulation shows configuration of machine including head and table movement
- common machine tool library
- common controller library
- shows collision detection with associated components

Advanced Simulation

- uses kinematic model of machine tool - simulations shows configuration of machine including head and table movement
- includes Machine Tool Builder capabilities
- provides gouge checking and collision detection
- provides methods to configure the manner of simulation (controller configuration)
- includes interface to Post Builder - enhanced, to generate machine controller drivers automatically (V3.1+ of PostBuilder)
- provides the ability to simulate existing machine G & M codes (reverse postprocessing)

Machine Tool Builder

- used to build a Machine Tool or device
- uses geometric model, created as an assembly
- builds a kinematic model of machine tool members
- defines a mounting model
- animates motion along machine axis for testing purposes
- may edit model through associated kinematics tree manipulation

Machine Tool Builder

Name	Classification	Axis Name	Junctions
FOURAX_XY-TAB_Z-HD_A-ROT_VERT			
MACHINE_BASE	_MACHINE_BASE, _BASIC_LICENSE_MACHINE		MACHINE_ZERO*
Z_BASE			
SPINDLE		Z	TOOL_MOUNT_JCT
Y_BASE			
Y_SLIDE		Y	
X_SLIDE		X	
A_BASE			
A_SLIDE		A	ROT_JCT
SETUP	_SETUP_ELEMENT		PART_MOUNT_JCT
PART	_PART, _SETUP_ELEMENT		
BLANK	_WORKPIECE, _SETUP_ELEMENT		
FIXTURE	_SETUP_ELEMENT		

Activity 10–1: Using Simulation

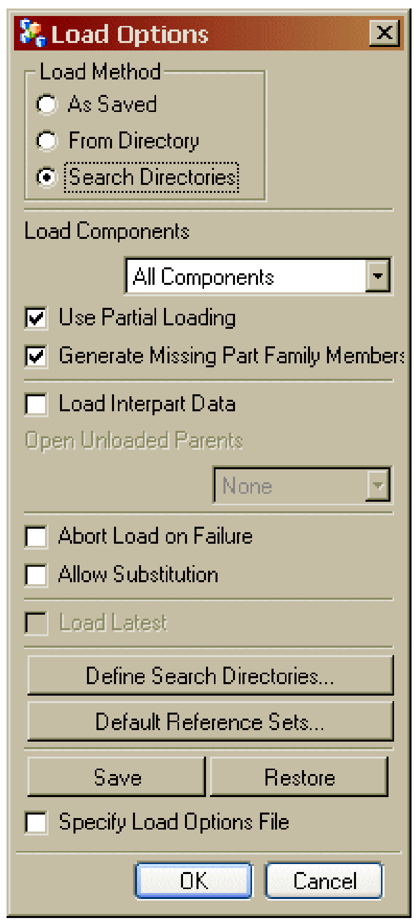
In this activity you will become familiar with some of the components used in the simulation process. You will execute a simulation of sample components supplied with this release of Unigraphics. For the simulation of the sample part to work correctly, you must first define the search directory of where the files are located.

Step 1 Define a new Search Directory for part retrieval.

❑ If necessary, start NX 2.

❑ Choose **File** → **Options** → **Load Options**.

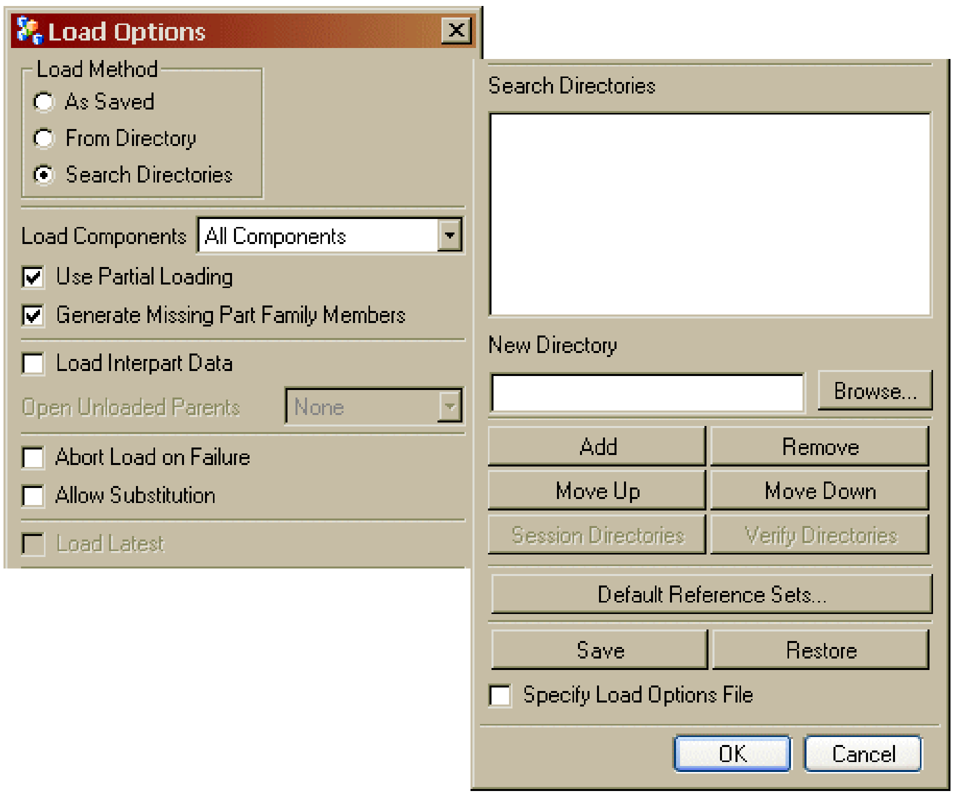
The **Load Options** dialog is displayed.



❑ If necessary, choose the **Search Directories** option.

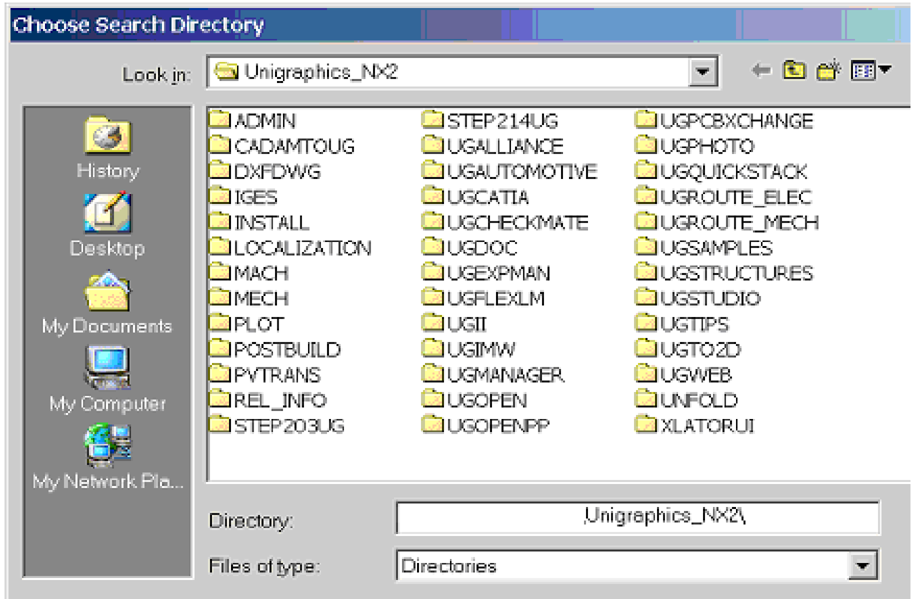
❑ Select the **Define Search Directories** button.

An updated **Load Options** dialog is displayed.



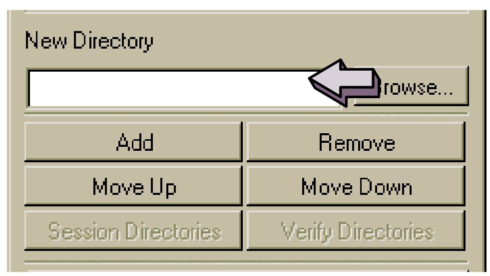
❑ Select the **Browse** button.

❑ Filter to your home Unigraphics directory and then the MACH subdirectory.



❑ Choose **OK** to the **Choose Search Directory**.

The directory name will appear in the “New Directory” field.

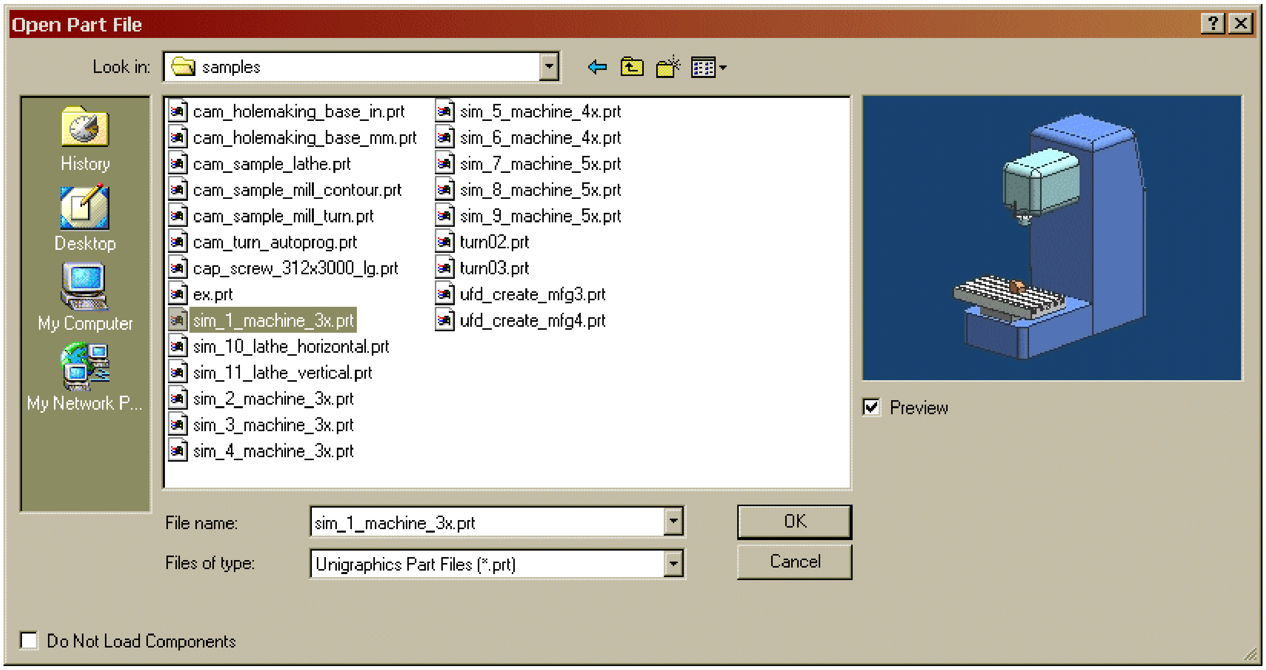


❑ Focus on the “New Directory” field, add three period characters to the end of the field and then select the Enter key from the keyboard.

❑ Choose **OK** on the **Load Options** dialog.

❑ Choose **File → Open**.

- ❑ Browse to your home **MACH\samples** directory.

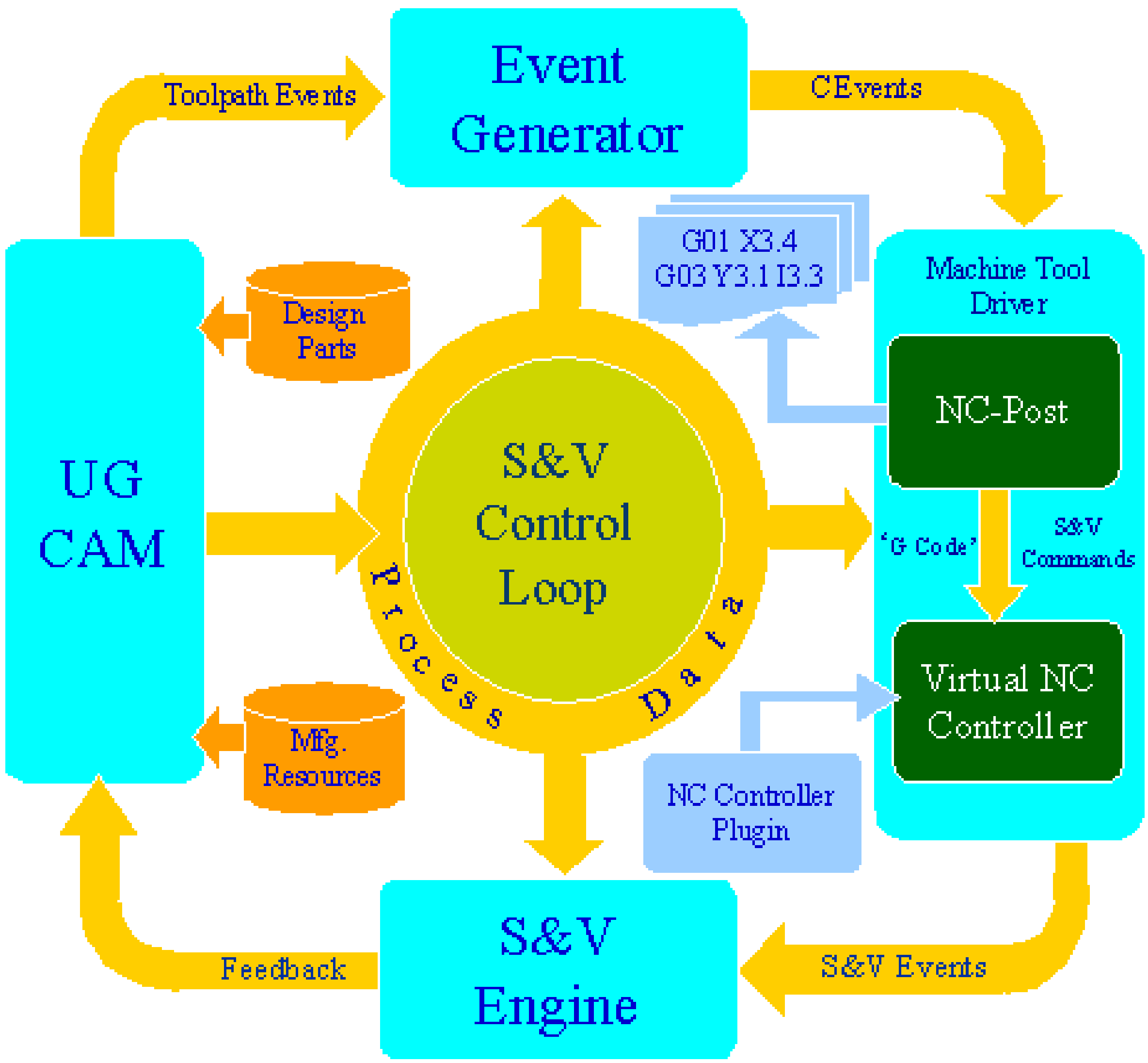


- ❑ Select the **sim_1_machine_3x.prt** from the **Open Part File** dialog.
- ❑ Choose **OK**.
- ❑ If necessary, start the **Manufacturing Application**.
- ❑ If necessary, switch to the **Program Order** view of the Operation Navigator.
- ❑ Using **MB3** on the **NC Program** parent, select **Toolpath** → **Simulate**.
- ❑ In the **Simulate Control Panel**, choose the **Play** button.
- ❑ Open the part **sim_10_lathe_horizontal** and repeat the simulation process.
- ❑ Open the part **sim_lathe_vertical** part and repeat the simulation process.
- ❑ **Close** all parts without saving.

This concludes the activity.

Machine Tool Driver

- generates motion control program and emulates CNC controller
- accurate path based on machine tool configuration
- handles specific machine tool features including macros, cycles and subroutines
- can be customized using Tcl scripting language
- text and graphics feedback initiated by Events

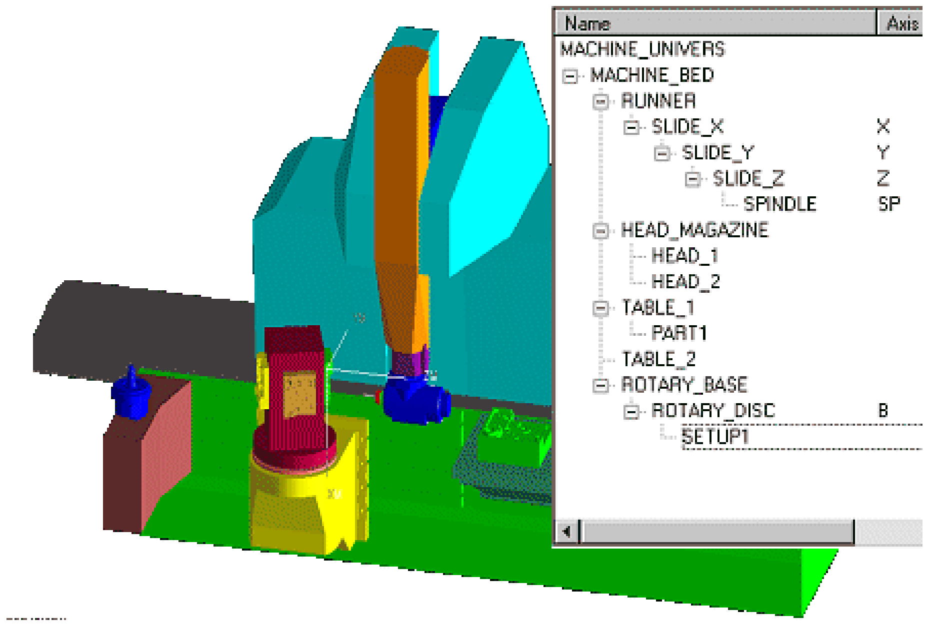


The **Machine Tool Driver** (**MTD**) creates the CNC program that emulates the CNC controller. The CNC controller emulator (or Virtual NC controller) is a programmable interface that instructs the machine tool model on actual movements and how those movements are displayed. Any motion and feedback displayed during machine tool simulation is controlled by the dedicated **MTD**.

For comparison purposes, the **MTD** is analogous to the machine tool simulator as the CNC controller is analogous to the machine tool that it controls. For each machine tool in the machine tool library, there is a **MTD** driver available (eleven generic **MTD's** come standard with NX 2). For creating an **MTD** for a new machine tool, you can modify a generic driver to work with that machine. **MTD's** are written int the Tcl scripting language but may also be developed in higher level languages such as C++. **MTD's** can emulate special cycles, User Defined Events (UDE's), macros and other CNC controller dependent functions that the Manufacturing application does not support.

Setup Configurator

- similar functionality to Machine Tool Builder
- used for mounting work piece and fixtures to machine tool
- used for defining machine state

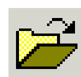


Activity 10–2: Use of Simulate on a Four Axis Part

In this activity, you will select a machine tool from the existing library of machine tools, add a fixture and part component using the Machine Tool Builder and Mating Conditions, and simulate a machining operation.

Step 1 Opening the part file ama_simulate.prt.

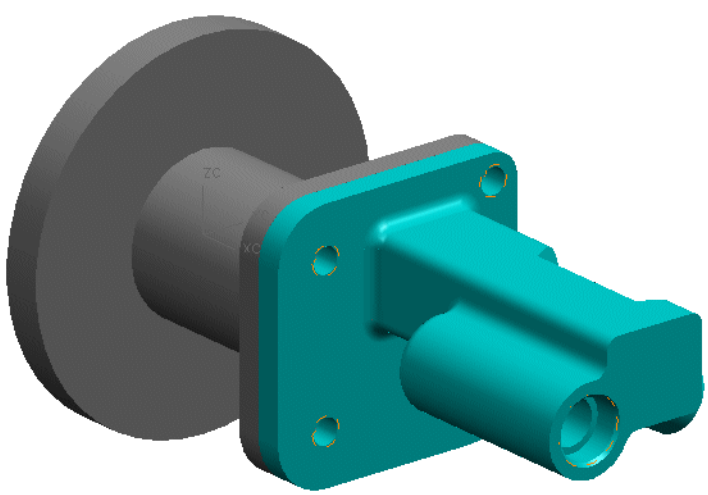
❑ If necessary, start Unigraphics NX 2.

❑ Use **File → Open**. 

❑ Navigate to your parts folder and open the file.

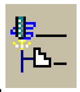
❑ Choose **File → Save as *** _simulate.prt** where *** represents your initials.

❑ Briefly examine the part.

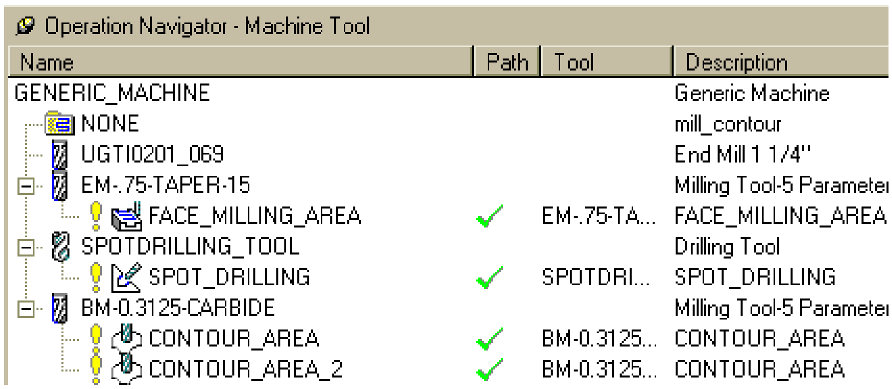


Step 2 If necessary, enter the Manufacturing Application and display the Operation Navigator.

❑ Choose **Application → Manufacturing** from the Menu bar.

❑ Choose the **Operation Navigator** icon from the **Resource** bar. 

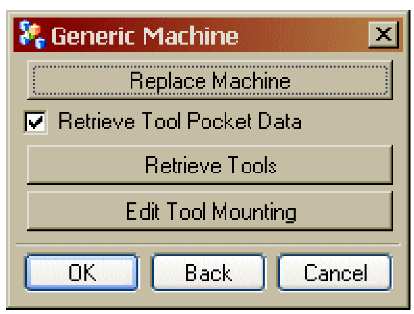
❑ If necessary, change to the **Machine Tool view** of the Operation Navigator and expand the group objects.



Step 3 Selecting the machine tool.

❑ Highlight the **GENERIC_MACHINE** group object, use **MB3**, and choose **Edit**.

The **Generic Machine** dialog is displayed.

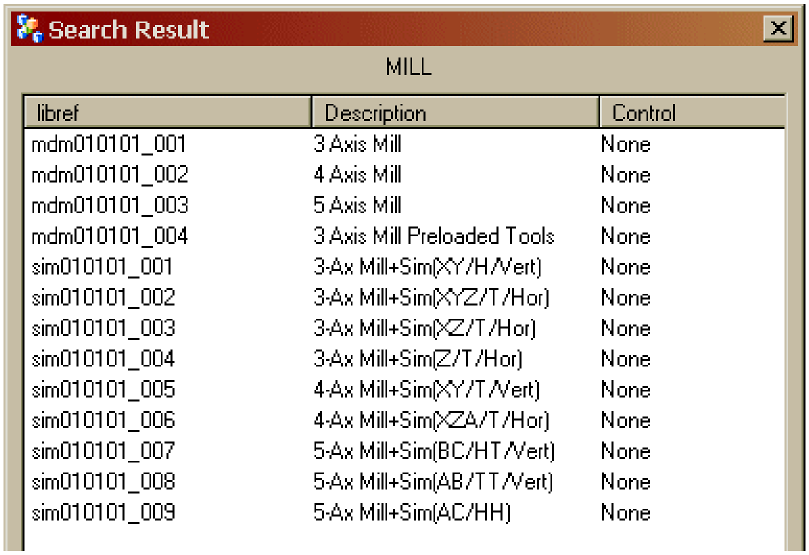


❑ Select the **Replace Machine** button.

The **Library Class Selection** dialog displays.

❑ Double-click on the **MILL** item.

The **Search Result** list is displayed, showing the various machines available for selection.

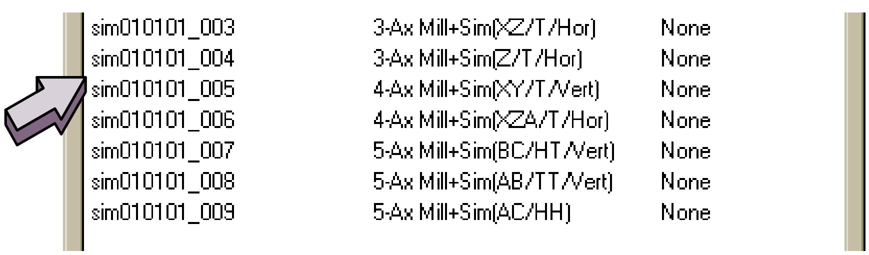

Search Result

MILL

libref	Description	Control
mdm010101_001	3 Axis Mill	None
mdm010101_002	4 Axis Mill	None
mdm010101_003	5 Axis Mill	None
mdm010101_004	3 Axis Mill Preloaded Tools	None
sim010101_001	3-Ax Mill+Sim(XY/H/Vert)	None
sim010101_002	3-Ax Mill+Sim(XYZ/T/Hor)	None
sim010101_003	3-Ax Mill+Sim(XZ/T/Hor)	None
sim010101_004	3-Ax Mill+Sim(Z/T/Hor)	None
sim010101_005	4-Ax Mill+Sim(XY/T/Vert)	None
sim010101_006	4-Ax Mill+Sim(XZA/T/Hor)	None
sim010101_007	5-Ax Mill+Sim(BC/HT/Vert)	None
sim010101_008	5-Ax Mill+Sim(AB/TT/Vert)	None
sim010101_009	5-Ax Mill+Sim(AC/HH)	None

The machines beginning with “sim” are machines ready for use in simulations. These machines have been previously modelled with the kinematics (motion of machine tool members) already defined and have Machine Tool Drivers already developed.

❑ Choose the 4-axis vertical milling machine, **sim010101_005** from the **Search Result** list.



sim010101_003	3-Ax Mill+Sim(XZ/T/Hor)	None
sim010101_004	3-Ax Mill+Sim(Z/T/Hor)	None
sim010101_005	4-Ax Mill+Sim(XY/T/Vert)	None
sim010101_006	4-Ax Mill+Sim(XZA/T/Hor)	None
sim010101_007	5-Ax Mill+Sim(BC/HT/Vert)	None
sim010101_008	5-Ax Mill+Sim(AB/TT/Vert)	None
sim010101_009	5-Ax Mill+Sim(AC/HH)	None

❑ Choose **OK** on the **Search Result** list.

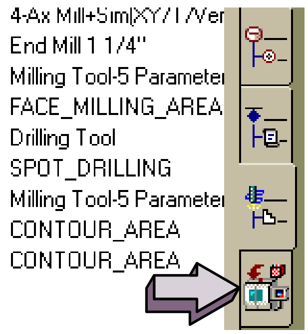
❑ Choose **OK** on the **Generic Machine** dialog.

❑ With the cursor in the graphic window, use **MB3**, **Fit** to size the view to the graphics window.

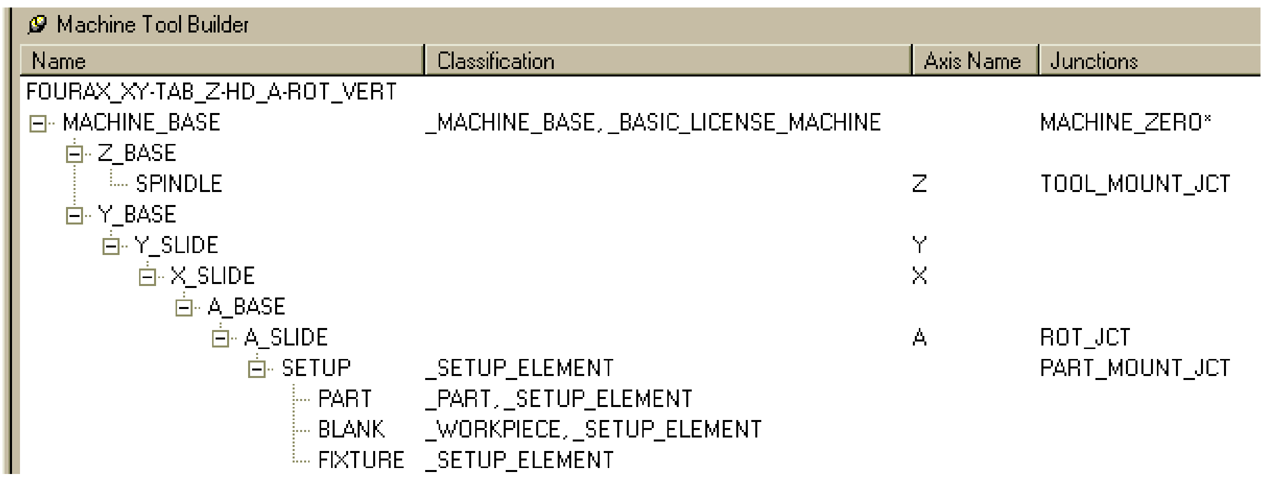
The machine tool, **sim010101_005** has been loaded. You must now mount the work piece to the machine. This is accomplished by using the **Machine Tool Navigator**.

Step 4 Mounting the work piece to the machine tool.

- ❑ Open the **Machine Tool Navigator** by selecting the **Machine Tool Navigator** icon from the **Resource** bar.



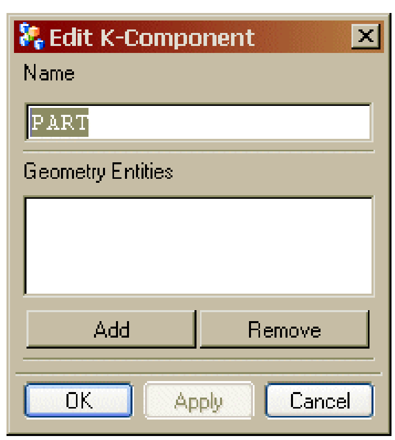
- ❑ Expand all objects.



Two objects, **Part** and **Fixture** must be assigned.

- ❑ Double-click on the **Part** component.

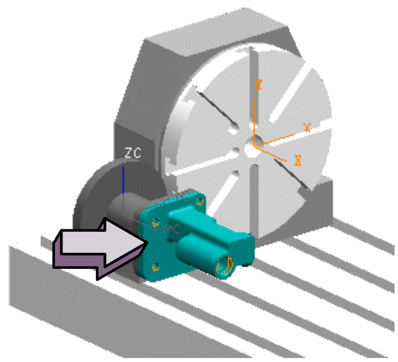
The **Edit-K Component** dialog is displayed.



- ❑ Choose the **Add** button.

  The **Class Selection** dialog is displayed.

- ❑ Select the part geometry.



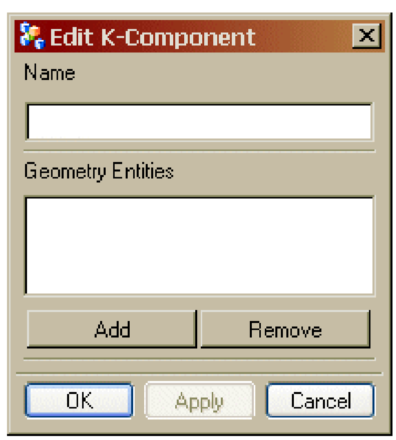
- ❑ Choose **OK** in the **Class Selection** dialog.

- ❑ Choose **OK** in the **Edit K-Component** dialog.

  You will now select the **Fixture** component.

- ❑ Double-click on the **Fixture** component in the **Machine Tool Builder** dialog.

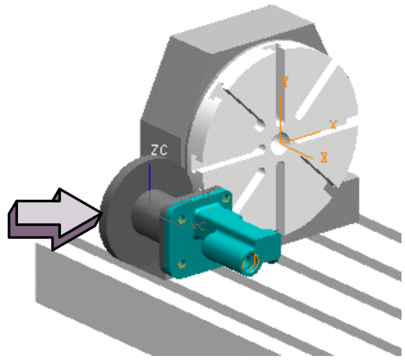
  The **Edit-K Component** dialog is displayed.



- ❑ Choose the **Add** button.

  The **Class Selection** dialog is displayed.

❑ Select the fixture geometry.



❑ Choose **OK** in the **Class Selection** dialog.

❑ Choose **OK** in the **Edit K-Component** dialog.

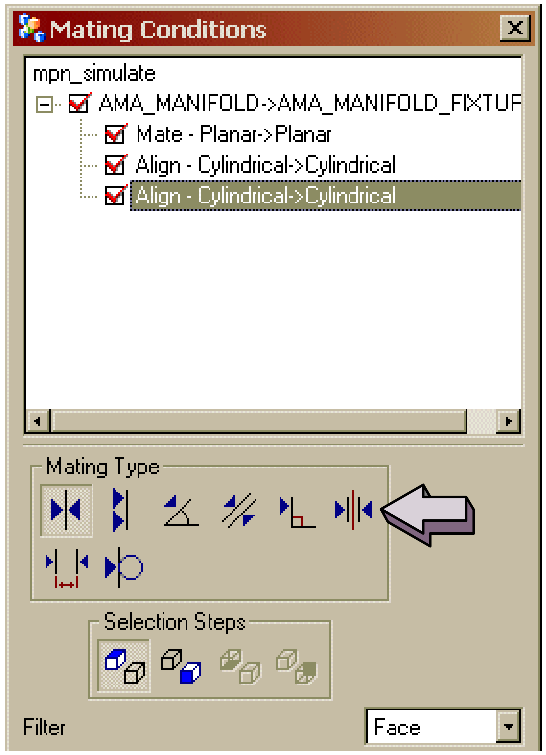
The components have been assigned to the machine tool and have to be positioned to the center of the rotary table. You will accomplish this by mating the two components using **Mating Conditions**.

❑ Verify that the **Assemblies Application** is turned on.

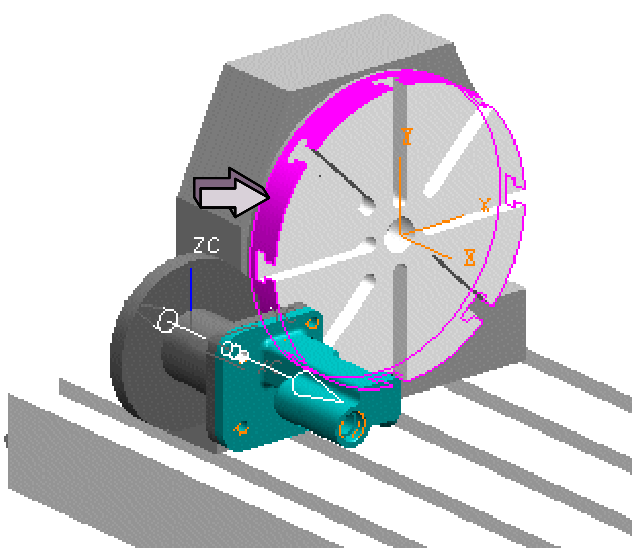
❑ Choose **Assemblies** → **Components** → **Mate Components**.

The **Mating Conditions** dialog is displayed.

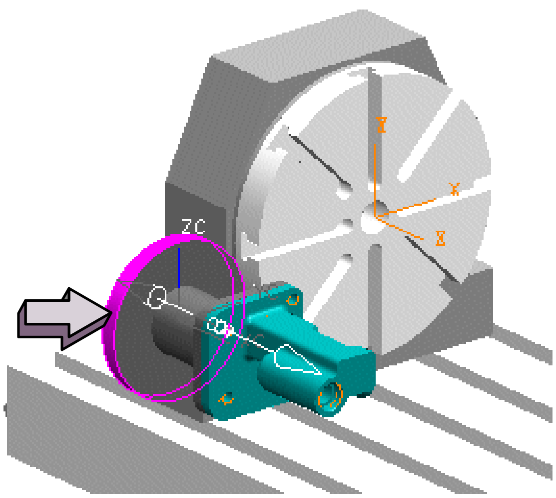
❑ Choose the icon represent the center **Mating Type**.



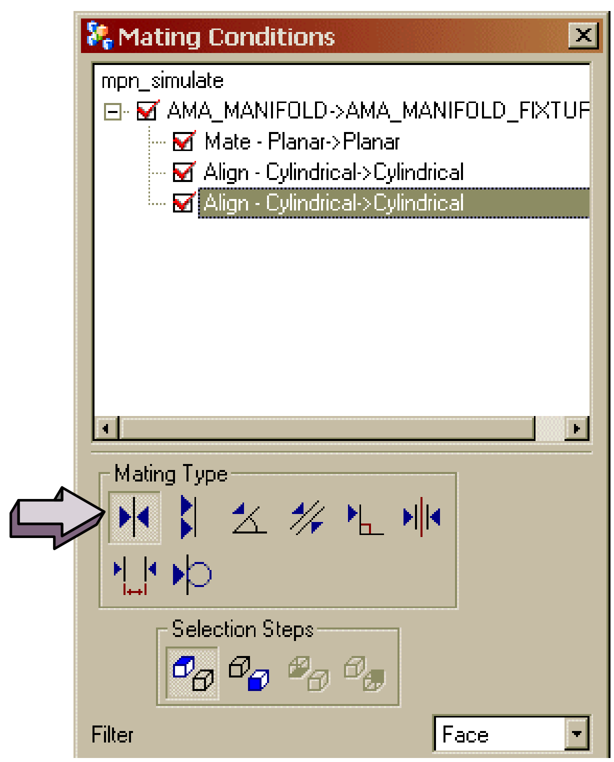
❑ Choose the cylindrical face of the headstock as the **FROM** face.



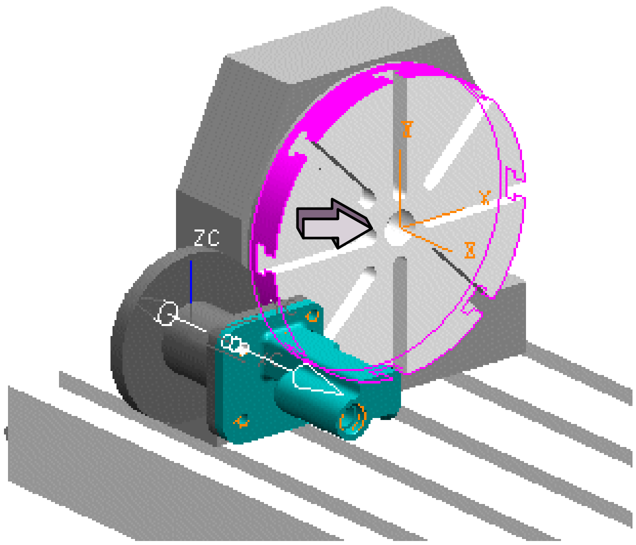
❑ Choose the cylindrical face of the fixture as the **TO** face.



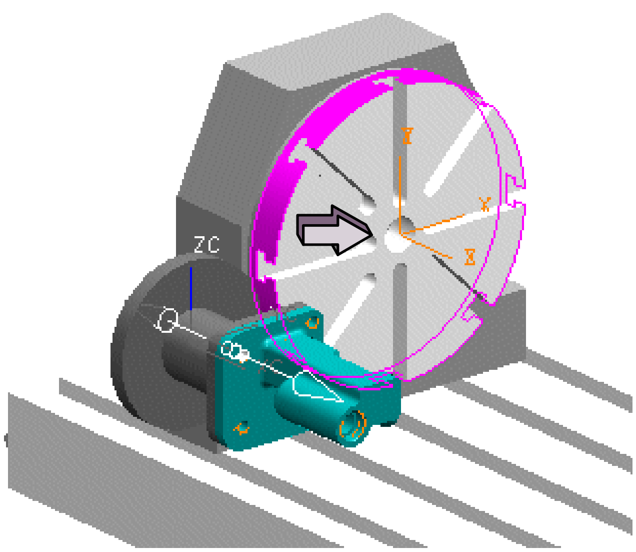
❑ Choose the **Mating Type**, **Mate** icon.



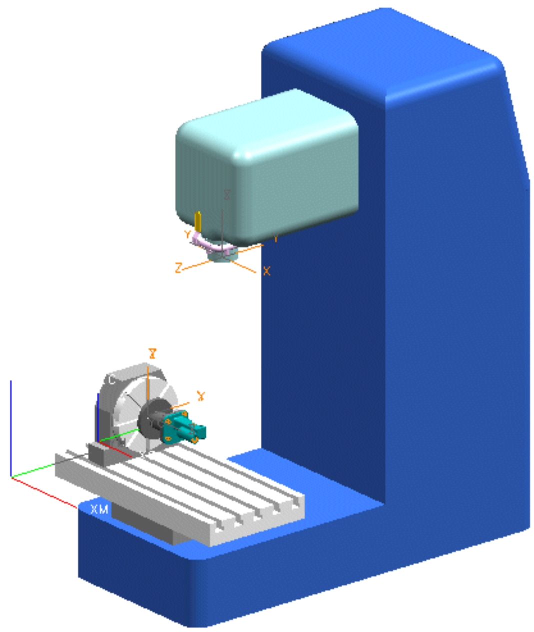
❑ Choose the planar face of the headstock as the **FROM** face.



❑ Choose the backside planar face of the fixture as the **TO** face.



❑ Choose **OK** until the fixture is oriented to the machine.



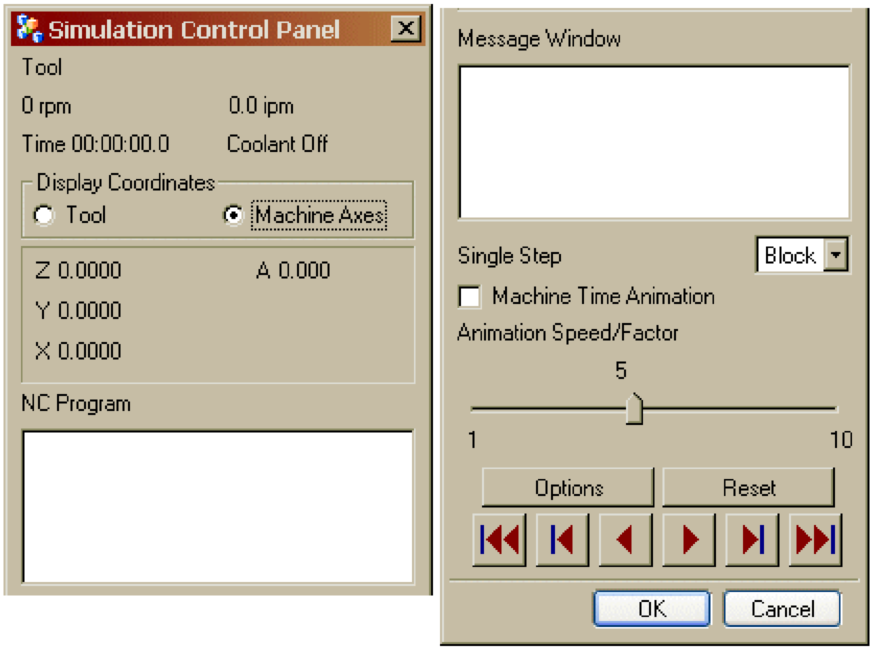
Step 5 Simulation of the program.

❑ Switch to the **Program Order** view in the Operation Navigator.

❑ Highlight the **Program** group object.

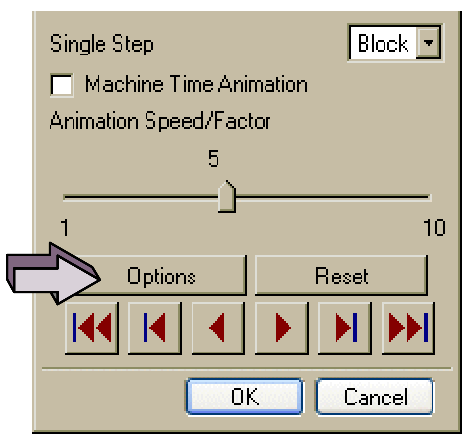
❑ Using **MB3**, choose **Tool Path** → **Simulate**.

The **Simulate Control Panel** dialog is displayed.



You will set the options, necessary for collision checking between the spindle nose and the part.

❑ Select the **Options** button at the bottom of the dialog.



The Simulation Options dialog is displayed.

❑ Highlight **Collision Checker**.

❑ Choose the **Options** button.

The **Collision Configuration** dialog is displayed.

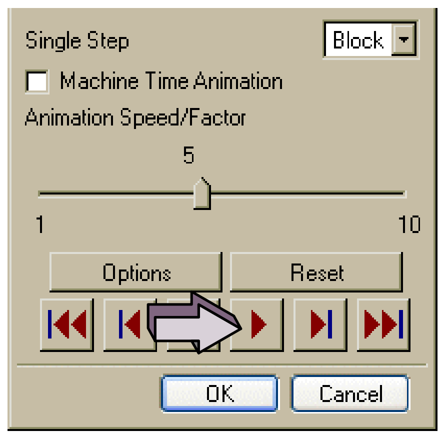
❑ In the **Name** field, choose **SPINDLE** from the drop down list.

❑ From the top of the dialog, choose the **Second Object or Class** icon.

❑ In the **Name** field, choose **PART** from the drop down list.

❑ Choose **OK** twice.

❑ Select the **Play** button.



The operations are simulated. Try various settings and options that are available to you on the **Simulate Control Panel**.

❑ **Save** and **Close** the part file.

You have completed the activity and the lesson.

SUMMARY

The Integrated Simulation and Verification module (IS&V) allows you to simulate a machine tool with an actual piece part, giving you an overview of the entire machining process. The simulation process animates the exact machine tool motions, taking into account controller functions and cutting tool configurations.

In this lesson you:

- reviewed the components that comprise the Integrated Simulation and Verification module
- mounted a part and fixture to an existing machine tool for simulation purposes
- simulated numerous tool paths

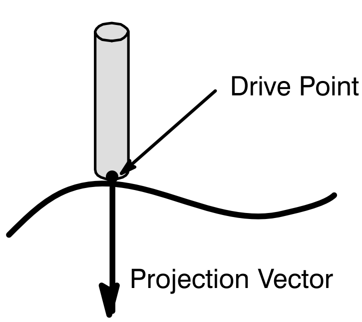
Advanced Surface Contouring

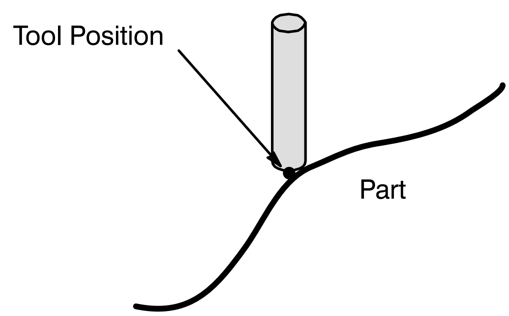
Appendix A

Projection

Mathematics of Projection:

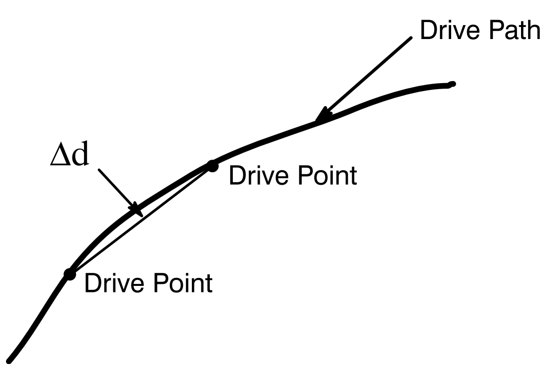
- Place tool end at drive point
- Project tool along projection vector
- Tool stops when making contact with part
- If necessary, adjust the tool axis and repeat the above steps until the tool axis is satisfied
- Add more intermediate drive points to satisfy the Intol/Outol with the part

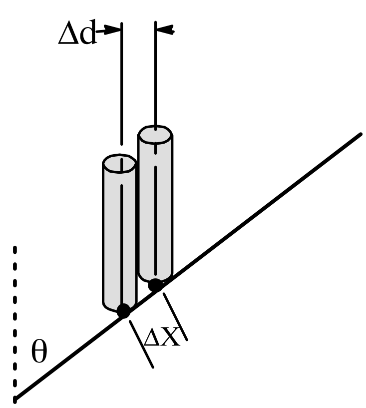




Projection and Steep Surface:

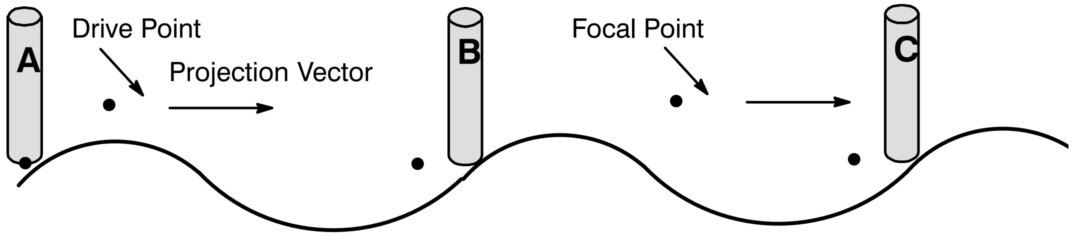
- $\Delta X = \Delta d/\sin\theta \cong \Delta d/\theta$
  ΔX becomes large if θ is very small (steep surface)
- The source of Δd is the chordal deviation of the drive path





Projection and Material Side:

- Surface contouring does not have explicit definition of material side for part geometry, only the drive surface has explicit material side
- Material side of the part is determined implicitly by the projection vector



If "Away from Point" ⇒ **C**

otherwise ⇒ **B**

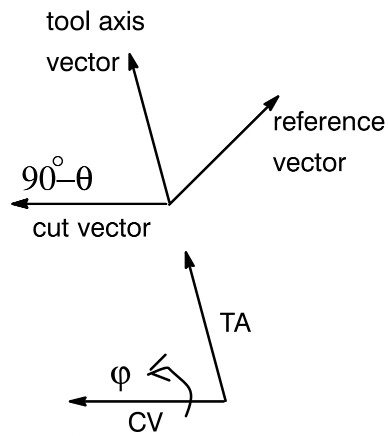
- In the case of Area Milling Drive (no projection vector), the tool axis vector is used to decide Material Side

Tool Axis

Definition of Lead/Tilt angles:

Lead $= \theta$

Tilt $= \varphi$

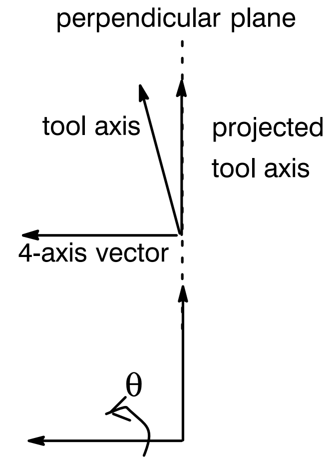


- Begin with cut vector, rotate it toward the Reference vector $90° - \theta$ degrees
- Then rotate around the cut vector φ degrees (counter clockwise)
- Reference vector is the surface normal relative to the part/drive or a vector which is relative to a vector

Definition of 4-axis rotation angle:

Rotation angle= θ

- Compute tool axis vector without 4-axis constraint first
- Project this tool axis vector onto the perpendicular plane of the 4-axis vector
- Rotate the projected tool axis vector along 4-axis vector θ degrees (counterclockwise)

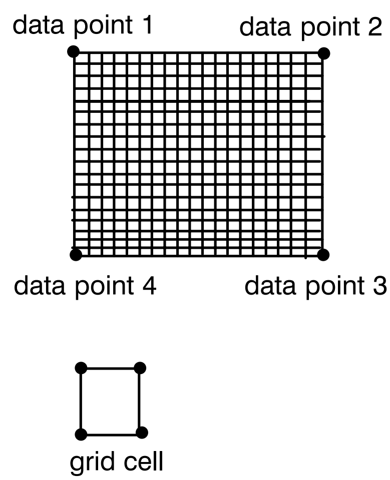


The unconstrained tool axis vector could be:

- Normal to Part / Drive
- Relative to Part / Drive

Interpolated tool axis algorithm:

- Divide the whole parameter (u,v) space for the drive surfaces by a 19x19 grid
- Compute the tool axis at each grid point using the data points weighted by the inverse of the distance square
- Inside each grid cell, compute the tool axis vector as the linear / spline interpolation of the tool axis vector at the four corners.

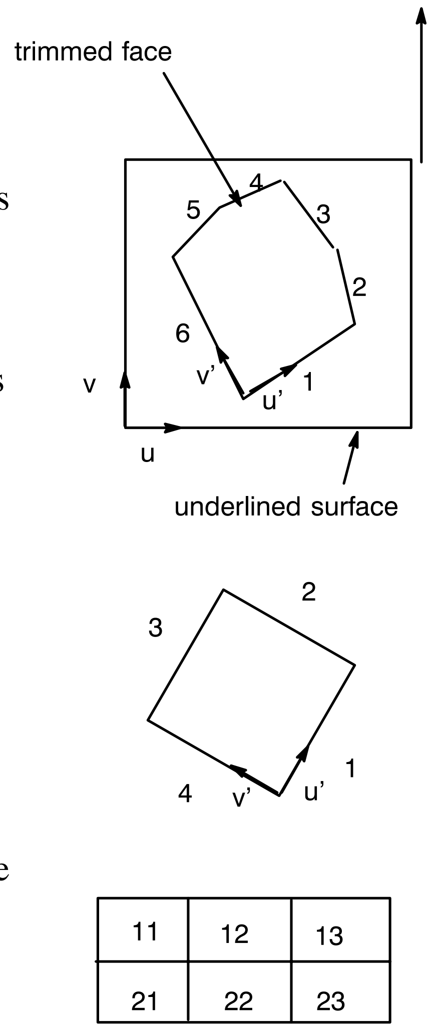


Drive Surface

A

Remap of drive surface:

Remap algorithm:



- Merge the exterior edges of the trimmed face to 4 sides
- Re-proportion the parameters of the exterior edges according to the arc length
- Use the arc length proportional edge parameters to construct the new (u',v') space for the trimmed face (Coon's Mapping)
- Finally, align the multiple drive surfaces into a rectangular grid formation

Limitations of remap:

- Fails on 3-sided faces
- Fails on faces that do not have rectangular shape
- May fail on faces with too many edges
- Multiple drive surfaces must be in grid formation

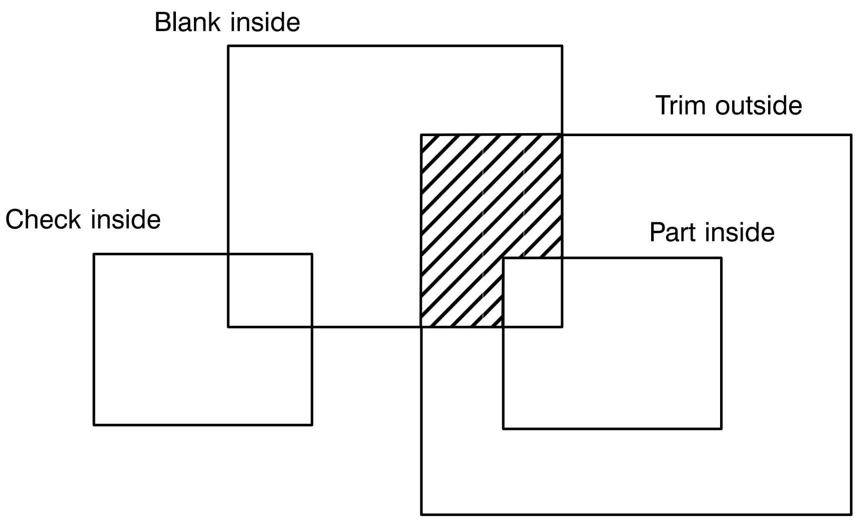
Swarf developable surface:

- Developable surfaces are special kinds of ruled surfaces when the surface normal vectors on any given rule line are the same (ruled surface without twisting)
- Only developable surfaces can be milled by swarfing without undercut or overcut

Planar Milling

A

- Blank – the region to be included
- Part – the region that can not be violated
- Check – the additional region that can not be violated
- Trim – as a final step, the region to be trimmed away



Boolean Logic

Boundary Drive

- Drive boundary – similar to “blank” if no part containment, otherwise it is like “part”
- Part containment – similar to “blank”

Area Milling Drive

- Cut area – similar to “blank”
- Trim – behaves slightly different from planar milling

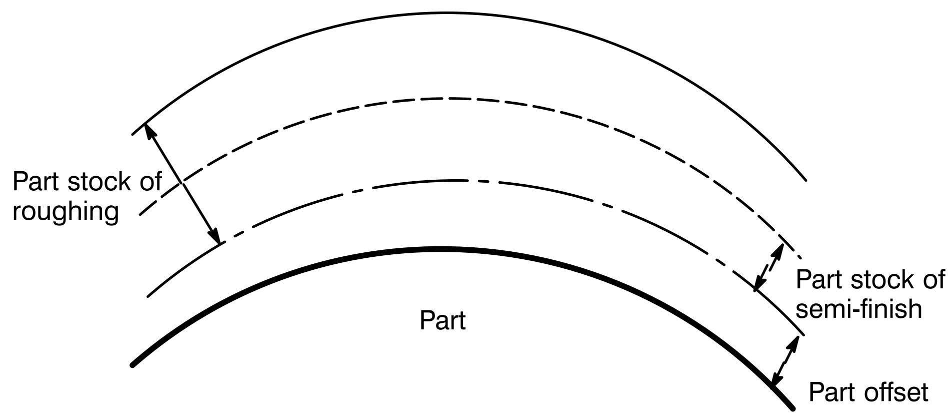
Stock

A

Part offset and part stock

	What	Where
Part Offset	Offset of part as the permanent definition of the final shape of the product	Geometry Group
Part Stock	Leftover material on part by a given operation	Operation

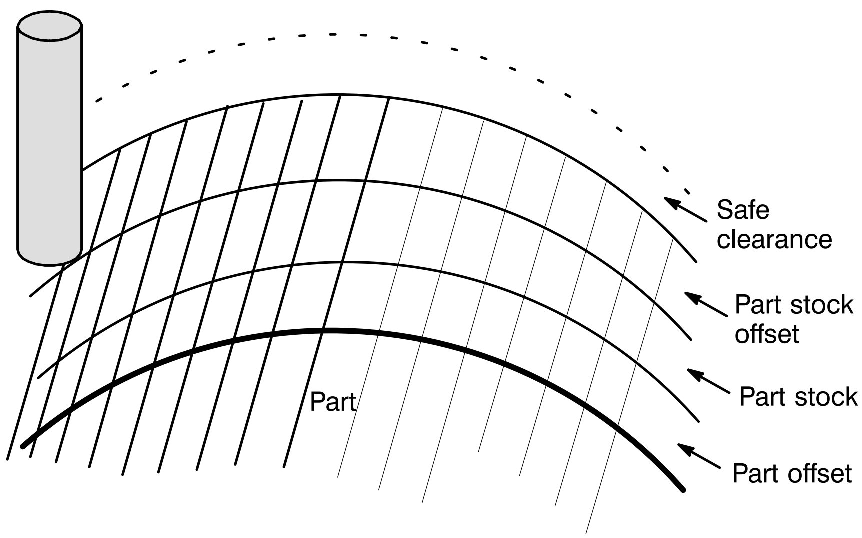
- Part stock is defined on “top” of part offset



Safe clearance and part stock offset

	What	Where
Part Stock Offset	Difference between the part stock from the previous operation and the part stock of the current operation	Operation
Safe Clearance	The additional safety zone for collision checking	Operation

- Safe clearance is defined on "top" of part stock offset



- Part stock offset is used in multiple pass, engage / retract and collision checking
- Safe clearance is used in engage / retract and collision checking

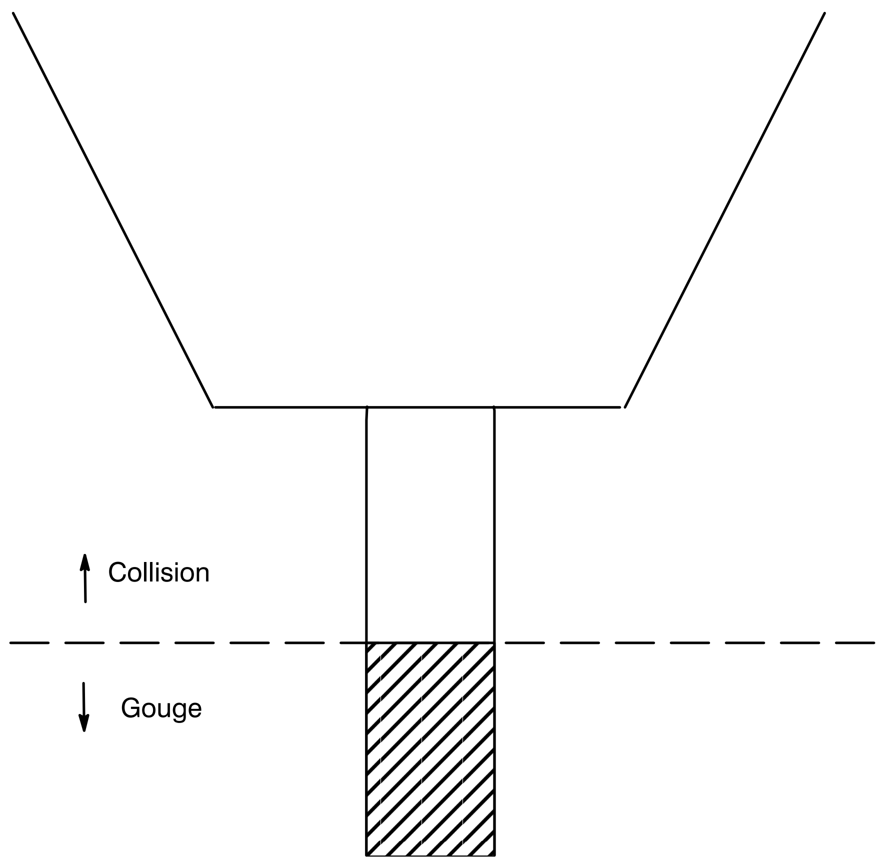
Gouge / Collision

A

Definitions:

	Rapid moves	Feed moves
Cutting part of tool assembly	Collision	Gouge
Non-cutting part of tool assembly	Collision	Collision

- Usually gouge check against part offset + part stock
- Usually collision check against part offset + part stock + part stock offset + safe clearance



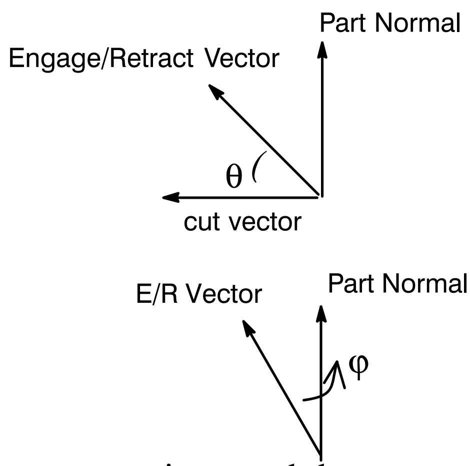
Usage:

	Collision check	**Gouge check**
Tool path generation	No	Yes on Part
Drive Path Generation	No	Optional on Drive
Engage / Retract	No	Optional on Part
Transfer moves	Optional on Part	Optional on Part
Cut Region computation (Cut Area)	Optional (holder) on Part / Check	Yes on Part
Check Geometry	No	Optional on Check
Gouge Check (Operation Navigator)	No	(No part stock)

Noncut Moves

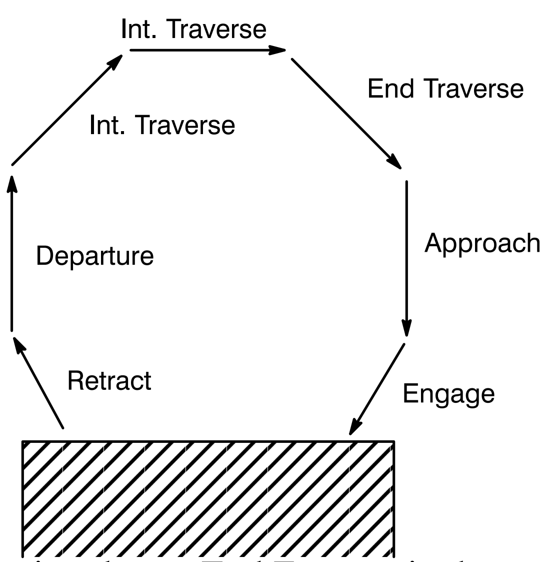
Azimuth / Latitude:

Latitude= θ
Azimuth= φ



- Begin with cut vector, rotate it toward the part normal θ degrees
- Then rotate around the part normal φ degrees (counter clockwise)

End / Intermediate traverse:



- There is only one End Traverse in the sequence, but there may be zero or multiple Int Traverse
- The Start and End positions of the End Traverse move are determined by other moves in the sequence

Project using Advanced Features
Appendix B

Since it is sometimes difficult to comprehend and use new procedures and functionality together in a process, the following series of activities will guide you through using several of the concepts that you have learned in this class.

In the following activity, our fictitious company has received an order to produce a prototype hydraulic manifold. The final product will be a casting, however our customer is assembling a prototype machine and the casting tool required will not be completed in time. Your task is to rough machine a simulated casting from a billet. A four-axis machining center is available for you use.

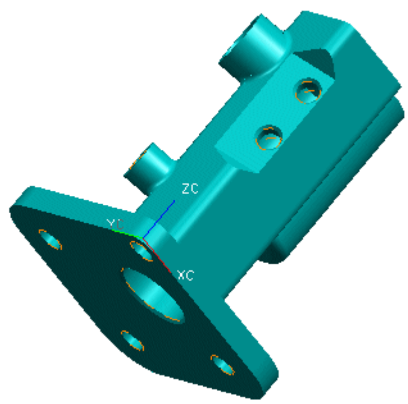
You will use the WAVE Geometry Linker and the In Process Work Piece (IPW) together to rough out a simulated casting. You will then use Z-Level Profile Steep and Mill Area Non-Steep to finish the casting simulation.

In the first part of this activity, you will open and examine the customer's part file and then create a suitable assembly, using WAVE, to simulate the casting of this part.

Activity: Part 1 Creating the Assembly for WAVE

Step 1 Open the part file ama_manifold.prt.

❑ Examine the part by rotating the object and use of the Information function on the main menu bar.



The part will be mounted with the base (large flat area with four holes) on a rotary table. All four sides of the part will be machined in one setup.

Step 2 Create the WAVE assembly.

❑ Open the file, **ama_seedpart_in.prt**, from the **student_home\parts** directory.

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

Standards require that you use a seed part whenever possible. Seed parts enforce standards such as layer control and color settings. Standards also dictate part file naming conventions. In this case, a manufacturing assembly is represented by "**mfg**" appended to the end of the file name.

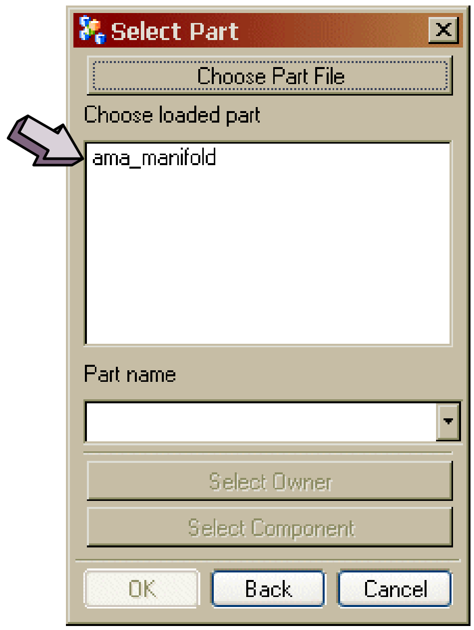
Step 3 Add the customer part file as a component.

Our intentions is to maintain full associativity to the customer's part file since we anticipate modeling changes as the prototype part is tested.

❑ Choose **Applications** → **Assemblies**.

❑ Choose **Assemblies** → **Components** → **Add Existing**.

❑ Highlight the **ama_manifold** file name and choose **OK**.



The **Add Existing Part** dialog is displayed.

❑ Set the **Reference Set** to **BODY**.

❑ Choose **OK**.

The **Point Constructor** dialog is displayed.

Since this is the first component to be added to the assembly, the position does not concern us.

❑ Choose **OK** on the **Point Constructor** dialog.

❑ Choose **Cancel** in the **Select Part** dialog.

Step 4 Create a new component and apply the seed part standards.

The customer's manifold part has been added as a component to the assembly. You will now add a component that will serve as a repository for the WAVE geometry.

❑ Choose **Assemblies** → **Components** → **Create New**.

The **Class Selection** dialog is displayed.

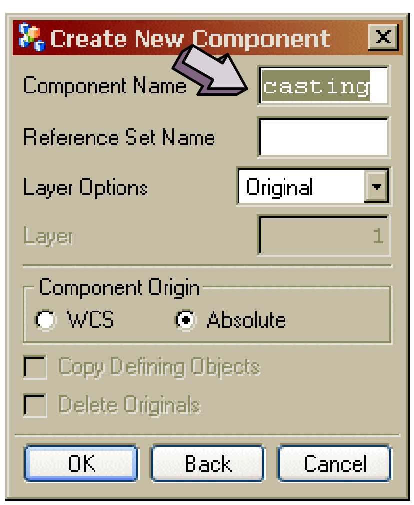
❑ Choose **OK** on the **Class Selection** dialog.

The **Part** selection dialog is displayed.

❑ Key in *****_manifold_casting** as the new component part file name.

The **Create New Component** dialog is displayed.

❑ Key in"**casting**" as the component name.



❑ Choose **OK** on the Create **New Component** dialog.

You will now apply the standards from the seed part file.

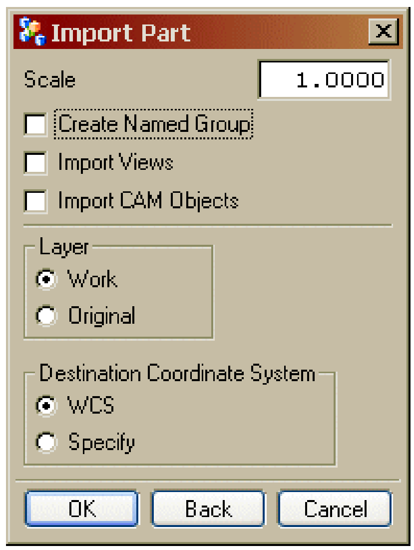
❑ Click on the **Assembly Navigator** tab on the resource bar.

❑ In the **Assembly Navigator**, double-click on the “**casting**” component (***_manifold_casting)

The “Why” Double-clicking on a component in the Assembly Navigator results in the component becoming the Work part.

❑ From the main menu bar, choose **File → Import → Part**.

The **Import Part** dialog is displayed.



❑ If necessary, turn off the **Create Named Group** option.

❑ Choose **OK** on the **Import Part** dialog.

The **Import Part** file selection dialog is displayed.

❑ Highlight the file named **ama_seedpart_in.prt** and choose **OK**.

The **Point Constructor** dialog is displayed.

❑ Accept the defaults by choosing **OK** on the **Point Constructor** dialog.

No geometry is added to the part, only layer categories and object color standards are derived from the seed part.

❑ Cancel the **Point Constructor** dialog.

Step 5 Save the modified assembly.

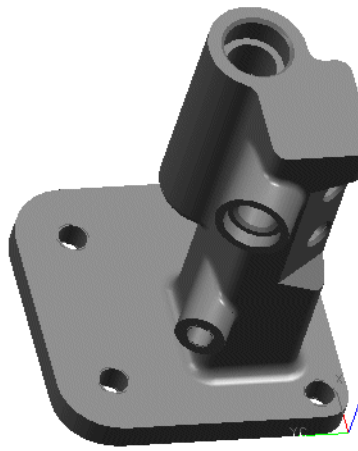
The *****_manifold_casting.prt** file has been created, however, the top-level component has not been saved. If you were to close the assembly at this time and then re-open it, the results would be incorrect.

❑ In the Assembly Navigator, make *****_manifold_mfg** component the work part.

❑ **Save**, but do not close the part.

This concludes this part of the activity.

As stated earlier, the ultimate goal of this project is to create a prototype, realistic casting that can be machined and tested. You will use the WAVE geometry linker and modeling tools to cover the openings in the casting for subsequent machining operations.



Activity: Part 2 Using the WAVE Geometry Linker

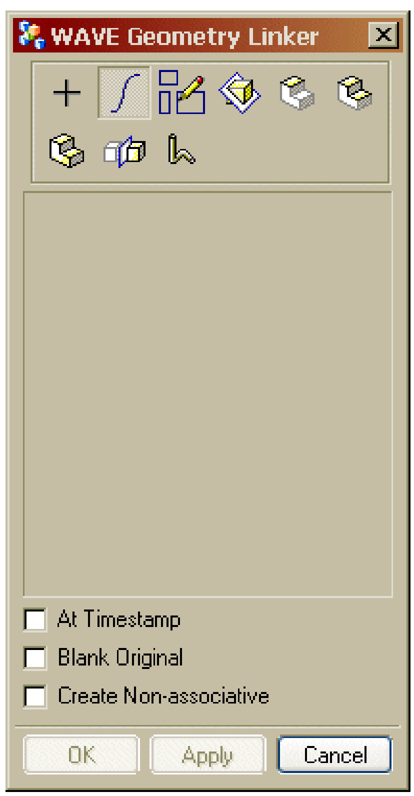
The WAVE geometry linker can be used to make associative copies of various types of geometry. In this part of the activity, you will use the linker to link face edges. In essence you will be "plugging" the holes for later machining operations.

B

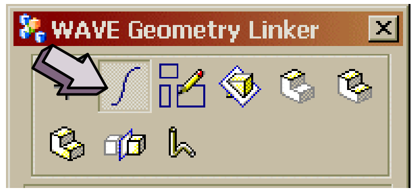
Step 1 Using the WAVE geometry linker.

- ❑ Continue using *****_manifold_mfg.prt** from Part 1 of the activity.
- ❑ In the Assembly Navigator, double-click on *****_manifold_casting** to make it the work part.
- ❑ Choose **Application** → **Modeling** from the main menu bar.
- ❑ Choose **Assemblies** → **WAVE Geometry Linker** from the main menu bar.

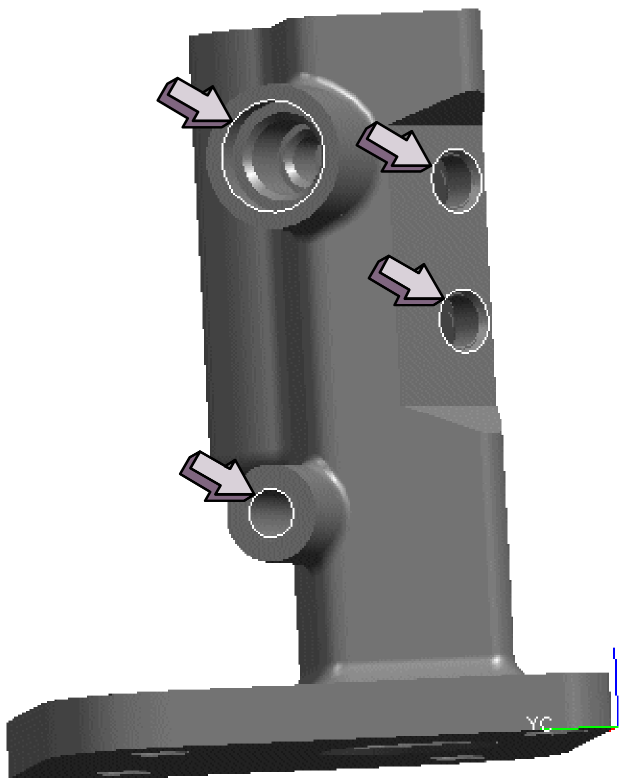
The **WAVE Geometry Linker** dialog is displayed.



❑ In the WAVE Geometry Linker dialog, select the **Curves** icon.



❑ Select the four curves, representing holes, as shown.



❑ Choose **OK** on the **WAVE Geometry Linker** dialog.

Step 2 Extruding the curves.

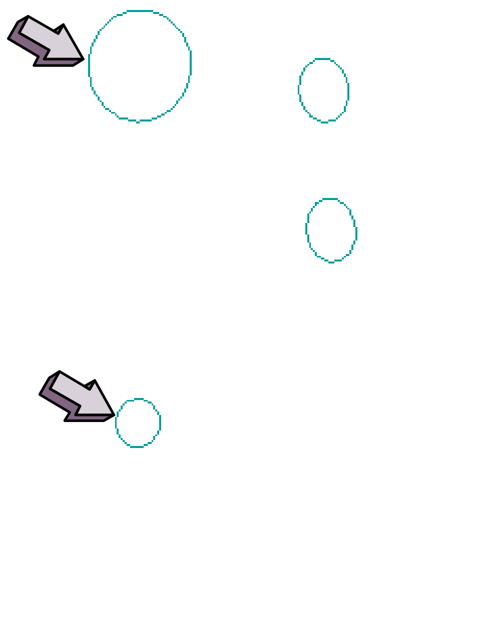
You will now extrude the linked curves into solids. When creating the machining operation, the solids created in this step will be chosen as part geometry. This will prevent the cutting tool(s) from entering those areas.

❑ In the **Assembly Navigator**, select the check mark next to **ama_manifold** to hide this component from the display

❑ From the Main menu bar, choose **Insert → Form Feature → Extrude**.

The **Extruded Body** dialog is displayed.

❑ Select the two holes that are on the same plane, but dissimilar in size as shown.

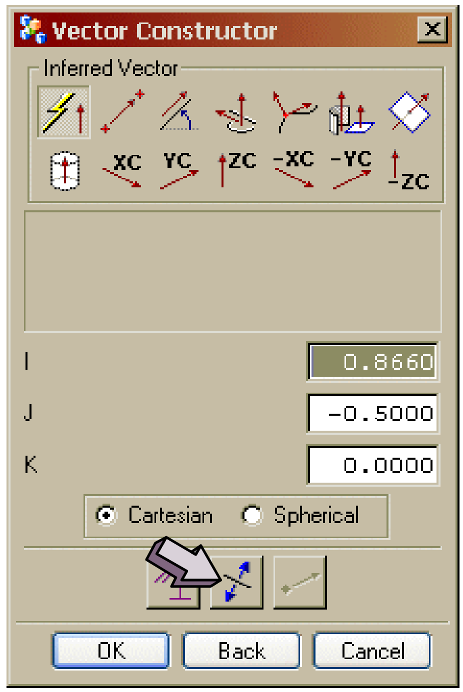


❑ Choose **OK** on the **Extruded Body** dialog.

❑ Choose **Direction and Distance** from the redisplayed **Extruded Body** dialog.

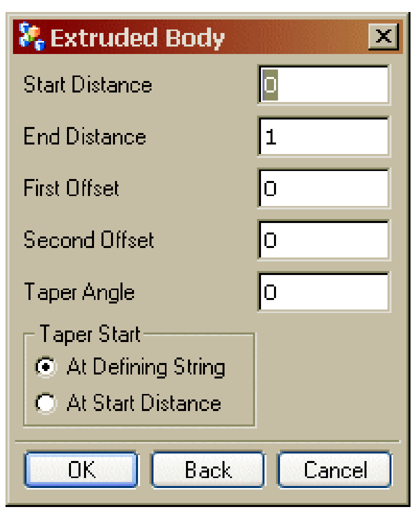
The **Vector Constructor** dialog is displayed.

❑ A direction vector is displayed from the lower hole that was previously selected. This vector should point in the –Y direction (pointing in towards the part). Use the **Cycle Vector Direction** icon, from the **Vector Constructor** dialog, if it does not.



❑ Once the correct vector is selected, choose **OK**.

The **Extruded Body** dialog is redisplayed.



❑ If necessary, set the values as shown above.

❑ Choose **OK** on the **Extruded Body** dialog.

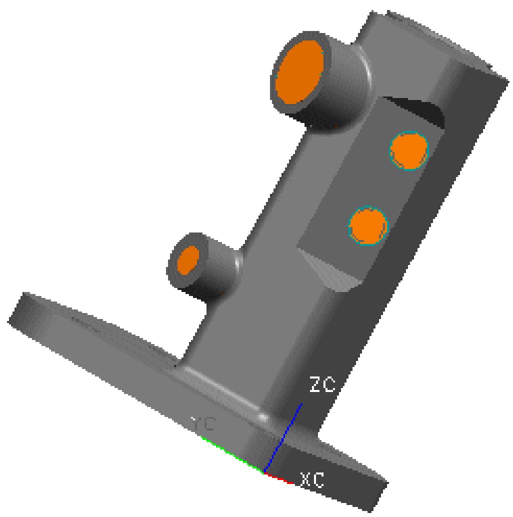
❑ Repeat the procedure for the other two curves. This time the vector should be pointed in the −X direction (in towards the part). Use the **Create** option when creating the second set of solids.

❑ Cancel the **Extruded Body** dialog.

B

Step 3 Save the assembly .

❑ In the Assembly Navigator, click on the checkmark next to ama_manifold to display the part again.



Notice how the four holes have been essentially “plugged”.

❑ In the Assembly Navigator, make *****_manifold_mfg** component the work part by double-clicking on the object.

❑ **Save** and **Close** the part.

This concludes this part of the activity.

Activity: Part 3 Using the IPW

In the previous two parts of the activity, you closed the machined holes in the manifold to simulate a casting. The following steps have been performed for you to save you some time in completing the activity:

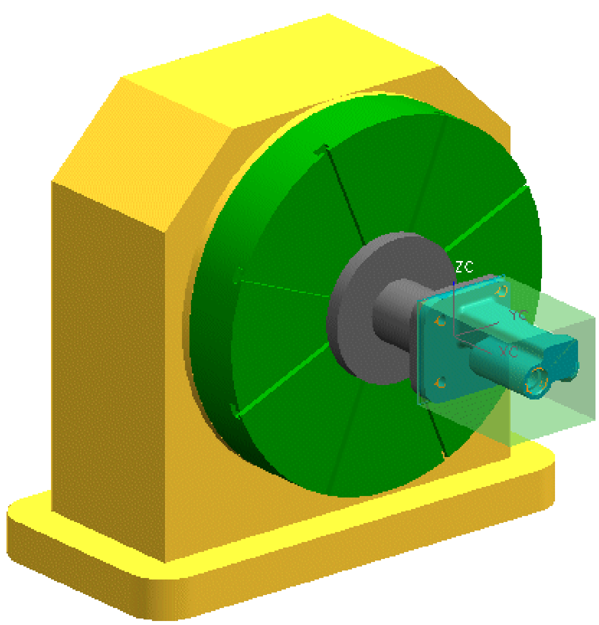
- the billet representing the raw stock has been created and added to the assembly
- a mounting plate for the part has been designed and added to the assembly
- the rotary axis of the machining center has been added to the assembly
- all components were mated and or positioned in the correct relationship with the manifold

You will pick-up the process at the stage where the NC programming will begin.

Step 1 Open and rename the existing assembly.

❑ Open **ama_manifold_mfg.prt** from your parts directory.

❑ Choose **File** → **Save As** → *****_manifold_mfg2.prt**, where ***** represents your initials.

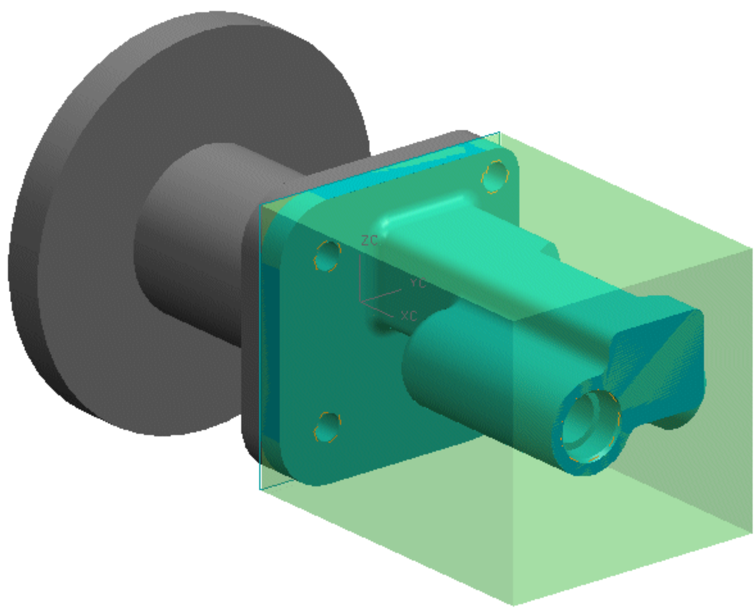


❑ Examine the assembly by opening the **Assembly Navigator**.

Since it is not necessary to see the rotary head or table, you will hide those objects by using the Assembly Navigator.

❑ Hide the display of the rotary table and head by clicking on the check marks next to them in the **Assembly Navigator**.

❑ Fit the view to the graphics window.



Step 2 Set up the manufacturing process.

❑ Choose **Application** → **Manufacturing** from the main menu bar.

The **Machining Environment** dialog is displayed.

❑ Choose **cam_general** as the **Configuration** and **mill_contour** as the **Setup**.

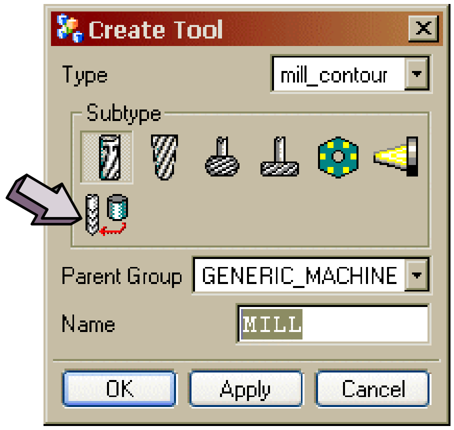
❑ Choose **Initialize**.

You will now retrieve a tool, used for roughing, from the tool library.

❑ Choose **Insert** → **Tool** from the main menu bar.

The **Create Tool** dialog is displayed.

❑ Choose the **Retrieve Tool** icon.



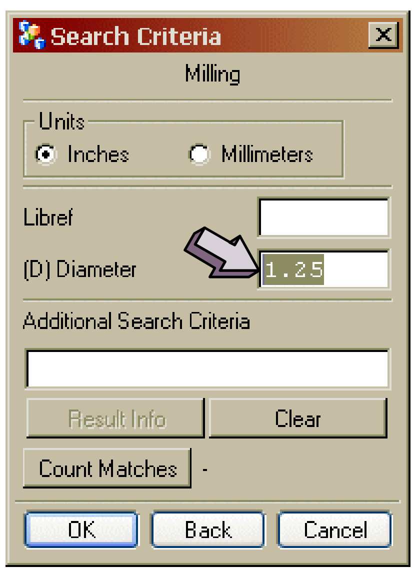
❑ Choose **OK** on the **Create Tool** dialog.

The **Library Class Selection** dialog is displayed.

❑ Highlight the **Milling** group object on the **Library Class Selection** dialog and then choose **OK**.

The **Search Criteria** dialog is displayed.

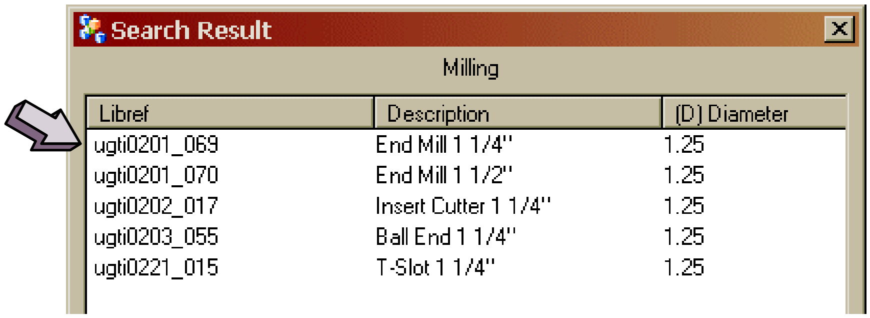
❑ Key in **1.25** in the **Diameter** field



❑ Choose **OK** on the **Search Criteria** dialog.

The **Search Result** listing is displayed.

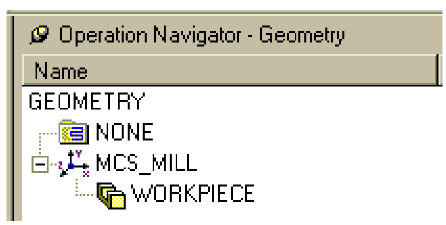
❑ Highlight the **UGTI0201_069** tool and choose **OK**.



You have selected your tool and will now select the appropriate geometry objects.

❑ Click on the **Operation Navigator** tab from the resource bar.

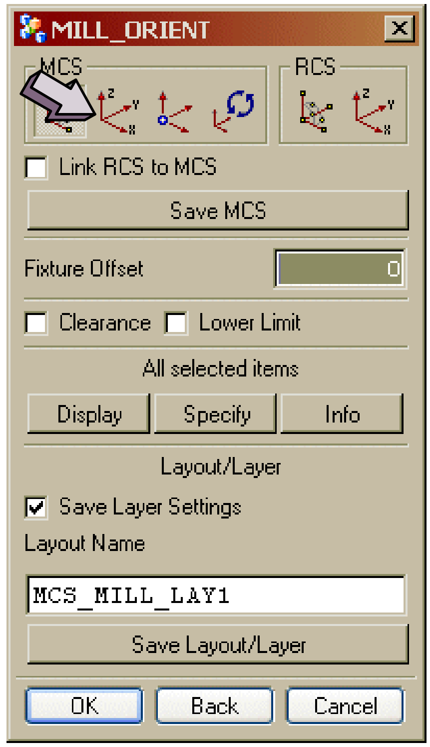
❑ Using **MB3**, switch to the **Geometry** view and expand all group objects.



❑ Double-click on the **MCS_MILL** geometry parent.

The **MILL_ORIENT** dialog is displayed.

❑ Select the **CSYS Constructor** icon.



The **CSYS Constructor** dialog is displayed.

❑ Choose **OK** on the **CSYS Constructor** dialog.

The **MCS** is now positioned and oriented to the **WCS**. It is generally a common practice to place the NC/CNC program zero at the center of rotation for rotary axis work.

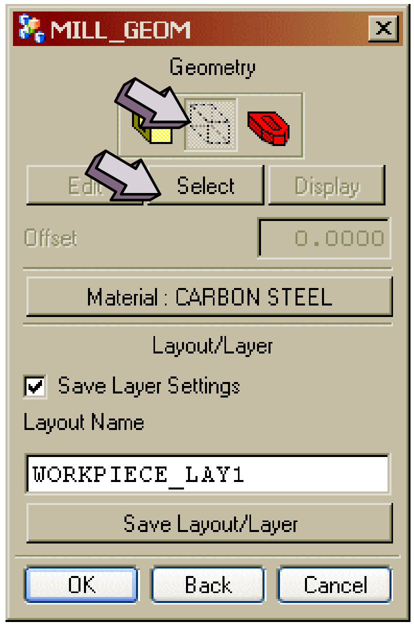
❑ Choose **OK** on the **MILL_ORIENT** dialog.

You will now select the blank geometry.

❑ In the **Geometry** view of the Operation Navigator, double-click on the **Workpiece** group object.

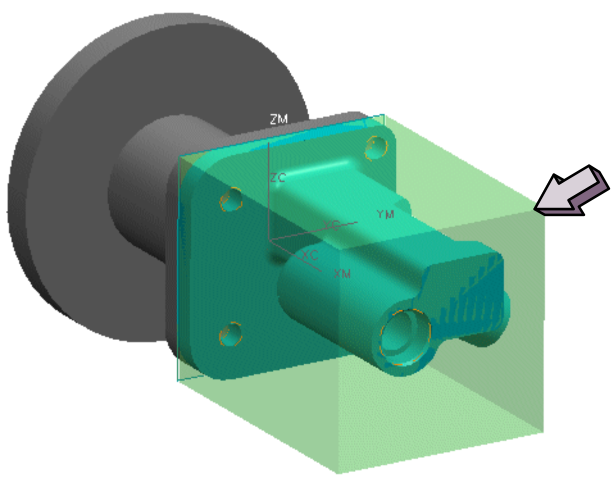
The **MILL_GEOM** dialog is displayed.

❑ Choose the **Blank** icon and then choose **Select**.



The **Blank Geometry** dialog is displayed.

❑ Choose the blank geometry as shown.

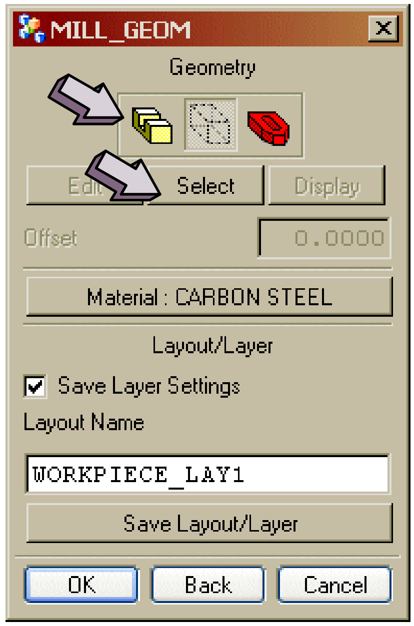


❑ Choose **OK** on the **Blank Geometry** dialog.

You will now select the part geometry.

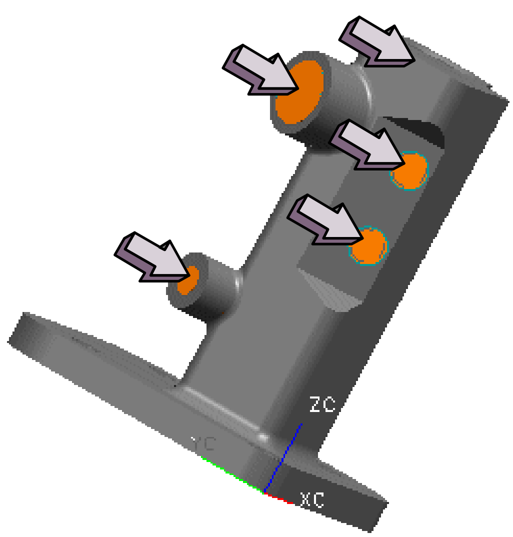
❑ In the **Assembly Navigator**, choose the checkmark next to **ama_manifold_stock**.

❑ From the **MILL_GEOM** dialog, choose the **Part** icon and then choose **Select**.



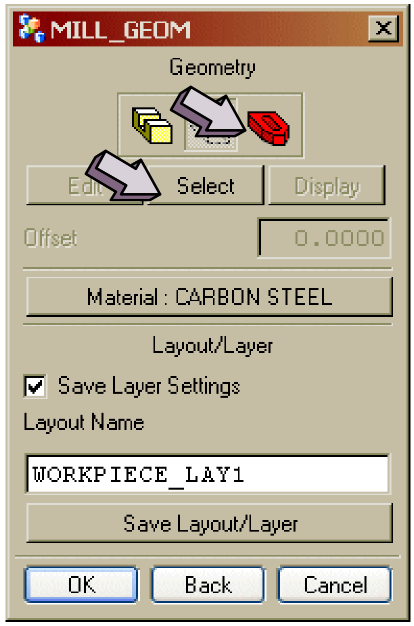
The **Part Geometry** dialog is displayed.

❑ Select the five components as shown in the figure (part and four plugs which you created to fill the holes).



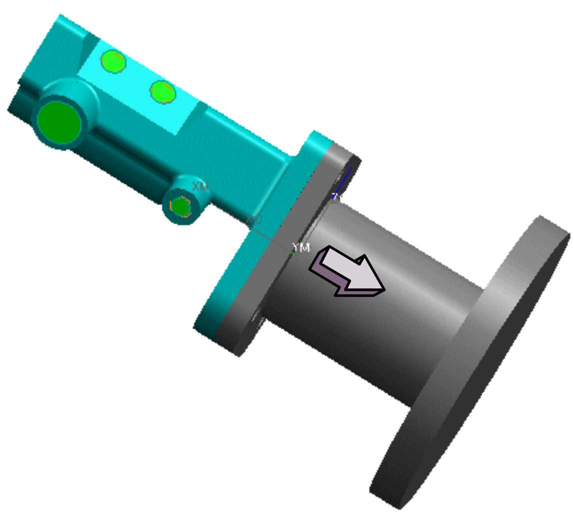
❑ Choose **OK** on the **Part Geometry** dialog.

❑ From the **MILL_GEOM** dialog, choose the **Check** icon and then choose **Select**.



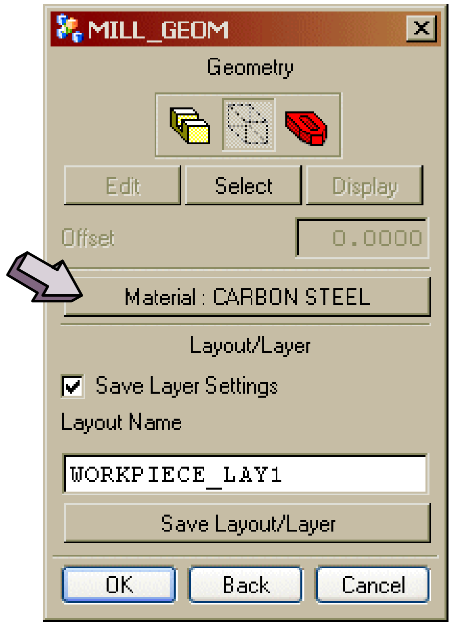
The **Check Geometry** dialog is displayed.

❑ Choose the fixture base as shown.



❑ Choose **OK**.

❑ From the **MILL_GEOM** dialog, change the **Material** to **Aluminum**.



❑ Choose **OK** in the **MILL_GEOM** dialog.

Step 3 Create the first Cavity Milling operation.

The next set of steps will lead you through the creation of a Cavity Milling operation, used in roughing the part.

❑ From the main menu bar, choose **Insert → Operation**.

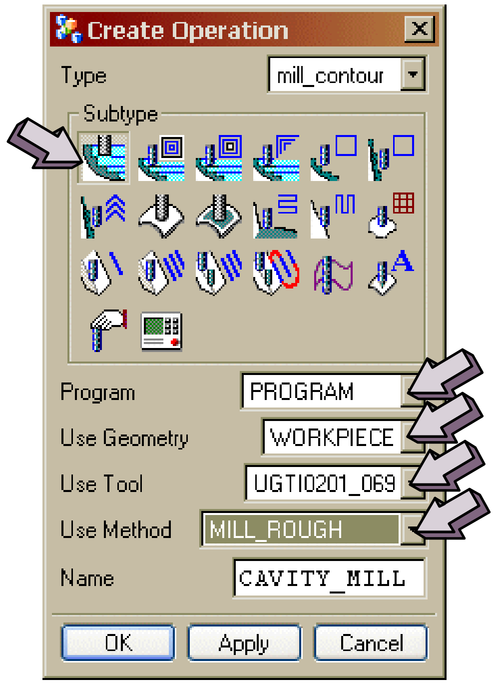
The **Create Operation** dialog is displayed.

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

❑ Choose the **CAVITY_MILLING** icon.

B

❑ Set the parent group objects as shown.



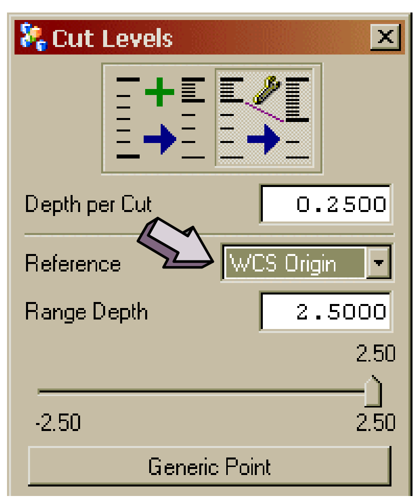
❑ Choose **OK**.

The **CAVITY_MILL** dialog is displayed.

This part will be rough machined from all four sides. Z zero has been determined as an appropriate location to stop the machining from any of the sides.

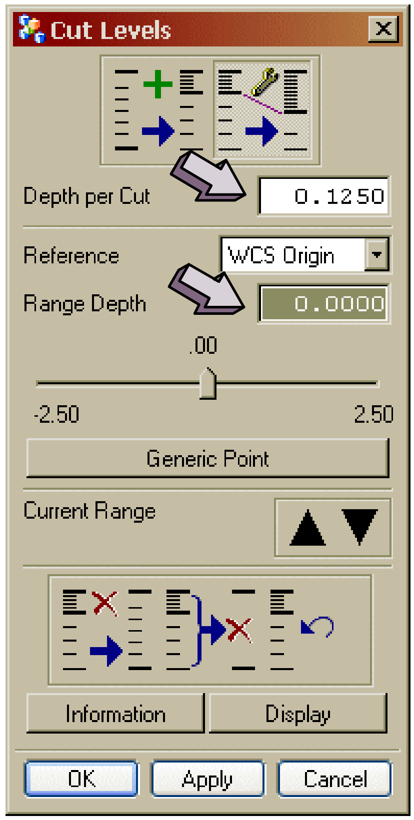
❑ Choose the **Cut Levels** button.

❑ Change **Reference** to **WCS Origin**.



❑ Set the **Depth per Cut** to **.125**.

❑ Change **Range Depth** to **0.0**.



❑ Choose **OK**.

Recalculate the feed rate based on part material and cutter type.

❑ Choose the **Feed Rates** button.

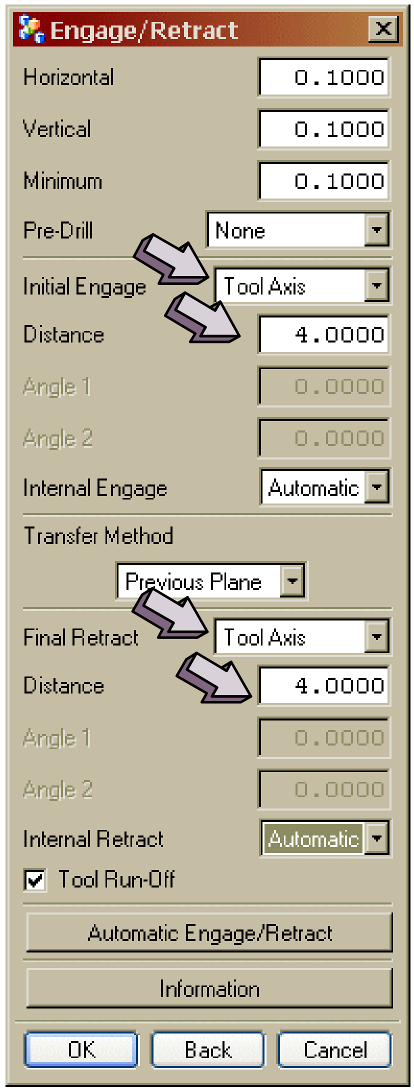
❑ Choose **Reset from Table**.

❑ Choose **OK**.

Specify the Engage/Retract direction.

❑ Under the **Engage/Retract** area of the **CAVITY_MILL** dialog, choose **Method**.

❑ Change **Initial Engage** and **Final Retract** to **Tool Axis** and then change the **Distance** parameter to **4.0**.



❑ Choose **OK** in the **Engage/Retract** dialog.

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

❑ Turn off **Pause After Display**.

❑ Choose **OK**.

❑ Once the tool path has finished generating, choose **OK**.

❑ Choose **OK** in the **CAVITY_MILL** dialog.

Step 4   Create the next Cavity Milling operation.

Now that the first operation, for roughing, has been created, the remaining three roughing operations will be a simple copy and paste. The only changes that you need to incorporate will be to turn on the IPW processing and modify the tool axis.

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

❑ Highlight the **CAVITY_MILL** operation.

❑ Use **MB3** → **Copy**.

❑ Use **MB3** → **Paste**.

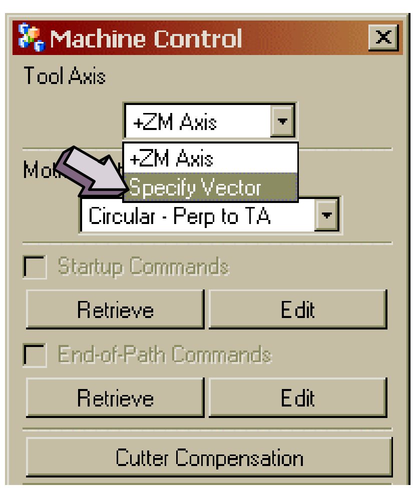
❑ **Rename** the copied operation to **CAVITY_MILL_2**.

❑ Edit the operation **CAVITY_MILL_2** by double-clicking it.

❑ Choose the **Machine** button.

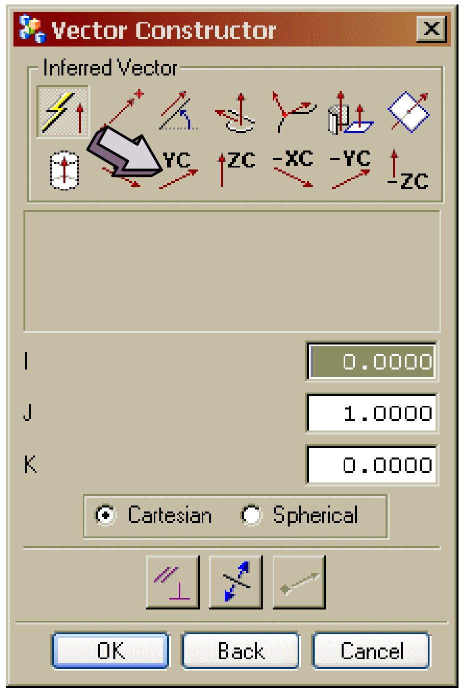
The **Machine Control** dialog is displayed.

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



The **Vector Constructor** dialog is displayed.

❑ From the **Vector Constructor** dialog, choose the **YC Axis** icon.



❑ Choose **OK**.

The warning indicates the current cut levels are not compatible with the tool axis. The processor will modify the cut levels to make them perpendicular to the new tool axis.

❑ Choose **OK** on the **Warning** dialog.

❑ Choose **OK** on the **Machine Control** dialog.

❑ Choose the **Cut Levels** button.

❑ Change the **Reference** to **WCS Origin**.

❑ Key in **0.0** in the **Range Depth** field.

❑ Choose **OK**.

❑ Choose the **Cutting** button.

❑ For the **In Process Workpiece** select **Use 3D**.

- ❑ Choose **OK**.
- ❑ Choose **Generate** to generate the tool path.

  The processing will be somewhat longer than the first Cavity Mill operation due to the considerable amount of calculations that are made for the In Process work piece.
- ❑ Turn off the **Pause After Display** dialog when it appears.
- ❑ Choose **OK**.
- ❑ After generation of the tool path, choose **OK** on the **CAVITY_MILL** dialog.

Step 5 Create the final Cavity Milling operation.

In this step, you will create another Cavity Mill operation, modifying the cut levels and tool axis.

- ❑ On the Operation Navigator, highlight the **CAVITY_MILL_2** operation.
- ❑ Use **MB3 → Copy**.
- ❑ Use **MB3 → Paste**.
- ❑ **Rename** the copied operation to **CAVITY_MILL_3**.
- ❑ Edit the operation **CAVITY_MILL_3** by double-clicking it.
- ❑ Choose the **Cutting** button.
- ❑ For the **In Process Workpiece**, select **NONE**.
- ❑ Choose **OK**.
- ❑ Choose the **Machine** button.
- ❑ Change the **Tool Axis** to **Specify Vector**.
- ❑ Choose the **ZC Axis** icon.

- ❑ Choose **OK** to the warning dialog.
- ❑ Choose **OK** on the **Machine Control** dialog.
- ❑ Choose the **Cut Levels** button.
- ❑ Change the **Reference** to **WCS Origin**.
- ❑ Change the **Range Depth** to **0.0**.
- ❑ Choose **OK**.
- ❑ Choose the **Cutting** button.
- ❑ For the **In Process Workpiece**, select **Use 3D**.
- ❑ Choose **OK**.
- ❑ **Generate** the operation.
- ❑ When the operation has been generated, choose **OK**.
- ❑ Verify the results of the entire program (Hint: go to the **Program Order** view, highlight the **NC Program** group object, using **MB3 → Toolpath → Verify**).

Each operation used only the blank material that was available. The first Cavity Milling operation used the entire blank, while each subsequent Cavity Milling operation used the **In Process Workpiece** from the previous operation. The result is a very efficient compliment of roughing operations with little, if any, non-cutting motion.

This concludes this part of the activity.

Finishing the Simulated Casting

In the previous parts of the activity, you used Cavity Mill and the In-Process Workpiece to rough a blank shape, removing most of the excess or stock material. Your goal is to simulate the casting body, therefore you need to finish the exterior portion of the casting body before you can begin the actual machining of the interior portion.

You will use a combination of Z-Level Profile Steep and Contour Area Non-Steep operations to finish the exterior of the simulated casting body.

Activity: Part 4 Combining Steep and Non-Steep operations

In this part of the activity, you will combine operations for steep and non-steep machining to finish the exterior of the simulated casting body. Additionally, you will execute tool axis changes for both types of operations to machine completely around the body.

Step 1 Open the part file.

- ❑ Choose **File** → **Open** → **ama_manifold_mfg_3.prt**.
- ❑ Choose **File** → **Save As** → *****_manifold_mfg_3.prt**.
- ❑ Choose **Application** → **Manufacturing**.

Step 2 Examine the current part.

You will now examine the existing rouging operations which were created earlier.

- ❑ If necessary, change to the **Program Order** view in the Operation Navigator and highlight the **Program** group object.
- ❑ Choose the **Verify** icon.

  The **Tool Path Visualization** dialog is displayed.
- ❑ Choose the **Dynamic** tab.
- ❑ Choose the **Play** button at the bottom of the **Tool Path Visualization** dialog.
- ❑ After examining the geometry, choose **OK**.

Step 3 Determine the appropriate tool size.

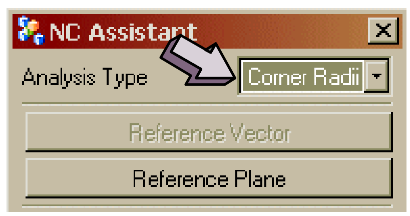
To determine the appropriate tool size for finishing purposes, you need to know the smallest radius that is on the outside of the part. You will use the **NC Assistant** to determine this radius.

❑ In the Assembly Navigator, turn off the display of the **ama_manifold_fixture** component by clicking on the red check mark next to the name.

❑ Choose **Analysis** → **NC Assistant** from the main menu bar.

The **NC Assistant** dialog is displayed.

❑ Change the **Analysis Type** to **Corner Radii**.



❑ Click and drag a rectangle around the part in the graphics window.

❑ Choose **Apply.**

There are two different corner radii listed. The smaller size, 0.125, does not affect the cutting operations since it is on the inside of the part only. The .250 corner radius listed will, however, affect the tool size.

You will now check the fillet radii.

❑ Change the **Analysis Type** to **Fillet Radii**.

❑ Click and drag a rectangle around the part in the graphics window.

❑ Choose **Apply**.

The smallest fillet radii that applies to machining the part is 0.1398; therefore a tool with a radius of .130 will suffice. The desired diameter of the tool should be no more than twice the corner radius.

❑ Choose **Cancel** in the **NC Assistant** dialog.

❑ From the main menu bar, choose **Insert** → **Tool**.

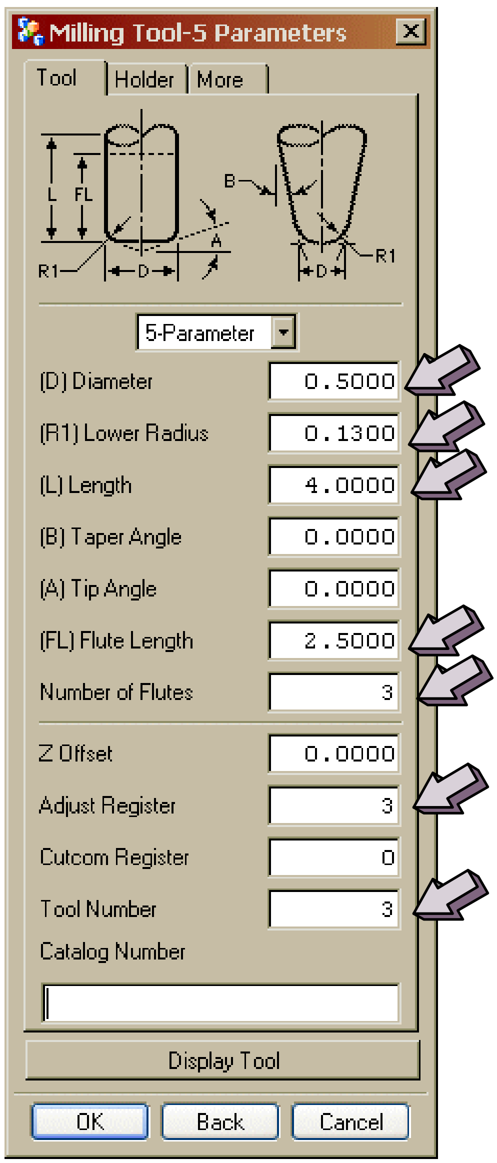
❑ If necessary, set the **Type** to **mill_contour**.

❑ Choose the **MILL** icon.

❑ Key in **em-.5-.13** for the **Name**.

❑ Choose **OK**.

❑ Key in the parameters as shown.



❑ Choose **OK**.

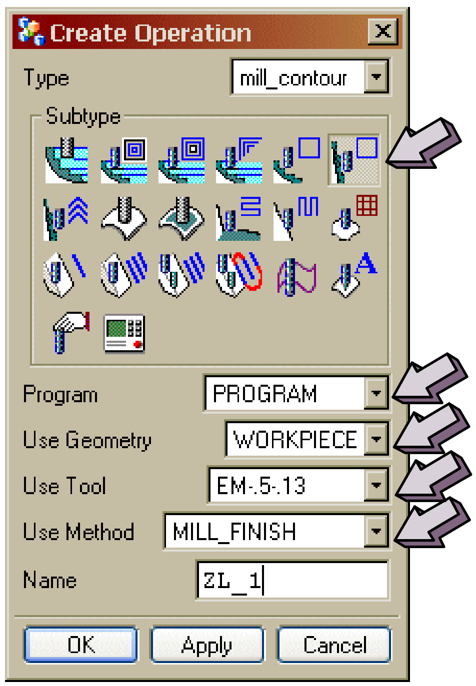
Step 4 Create a Z-Level Steep operation.

You will now create a **Z-Level Profile Steep** operation to machine those areas of the part that are close to parallel with the current tool axis.

❑ From the main menu bar, choose **Insert → Operation**.

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

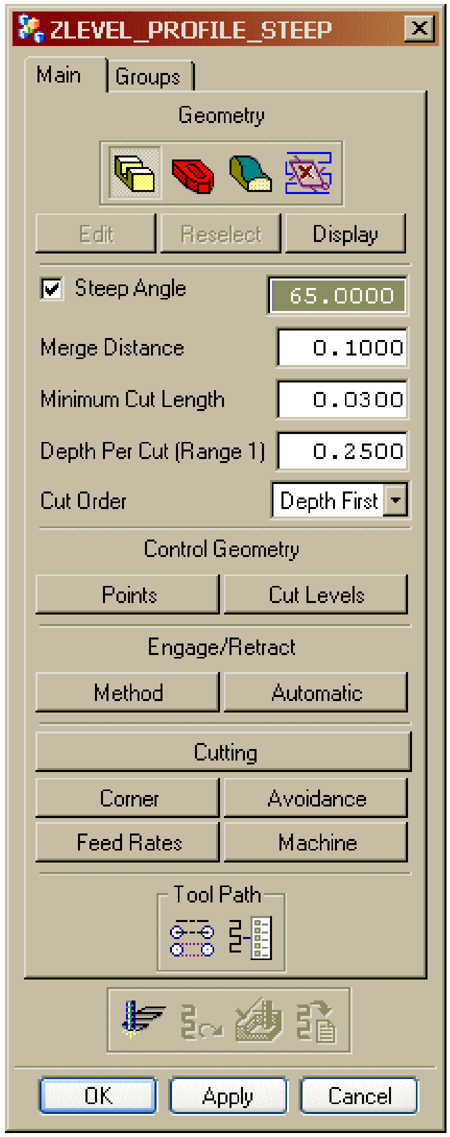
❑ Choose the **Z_LEVEL_PROFILE_STEEP** icon and set the parent objects as shown.



❑ Key in **ZL_1** as the **Name**.

❑ Choose **OK**.

The ZLEVEL_PROFILE_STEEP dialog is displayed.



- ❑ Change the **Steep Angle** to **50**.
- ❑ Choose the **Cut Levels** button.
- ❑ In the **Cut Levels** dialog, change the **Reference** to **WCS Origin**.
- ❑ Set the **Range Depth** to **0.0**.
- ❑ Set the **Depth Per Cut** to **0.03**.
- ❑ Choose **OK** in the **Cut Levels** dialog.

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

❑ When the operation has generated, choose **OK**.

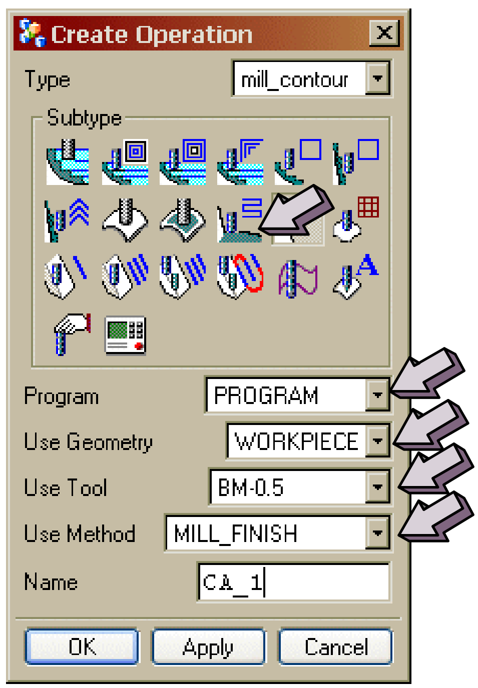
Step 5 Create a Contour Area Non-Steep operation.

You have completed the machining of the steep portions of the part and need to create a Fixed Contour operation, using the Mill Area drive method to machine those areas that are not steep.

❑ From the main menu bar, choose **Insert** → **Operation**.

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

❑ Choose the **CONTOUR_AREA_NON_STEEP** icon and set the parent objects as shown.



❑ Key in **CA_1** as the **Name**.

❑ Choose **OK**.

The **CONTOUR_AREA_NON_STEEP** dialog is displayed.

❑ Select **Area Milling** as the **Drive Method**.

The **Area Milling Method** dialog is displayed.

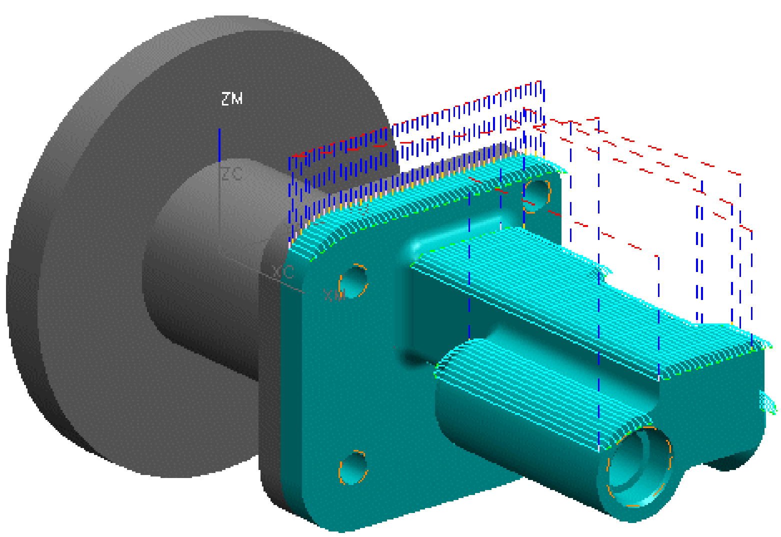
- ❑ Set the **Steep Angle** to **50**.

  This Steep Angle value matches the Steep Angle from the previous Z-Level operation.

- ❑ Change the **Pattern** to **Parallel Lines**.
- ❑ Change the **Stepover** to **Scallop**.
- ❑ Change the **Height** to **0.005**.
- ❑ Under **Apply**, choose the **On Part** radio button.
- ❑ Choose **OK**.
- ❑ In the **CONTOUR_AREA_NON_STEEP** dialog, choose the **Cutting** button.
- ❑ In the **Check Stock** field, key in **-.09**.

  The negative check stock will allow the ball-nose tool to follow the edge of the check stock.

- ❑ Change the **When Gouging** option to **Retract**.
- ❑ Choose **OK**.
- ❑ In the **CONTOUR_AREA_NON_STEEP** dialog, choose **Generate** to generate the tool path.



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

Step 6 Repeat the finishing process using new tool axis options.

You will now repeat the **ZLEVEL_STEEP** and **CONTOUR_AREA_NON_STEEP** machining operations, changing the tool axis each time.

- ❑ In the **Program Order** view, highlight the **ZL_1** operation, using **MB3 → Copy**.
- ❑ **Paste** the operation immediately below the **ZL_1** operation.
- ❑ **Rename** the operation to **ZL_2**.
- ❑ Double-click on the new operation, **ZL_2**, to edit.
- ❑ Choose the **Machine** button.
- ❑ Change the **Tool Axis** to **Specify Vector**.
- ❑ In the **Vector Constructor** dialog, choose the **Spherical Coordinates** radio button.
- ❑ Key **120.0** for the **Phi** angle; **90.0** for the **Theta** angle.
- ❑ Choose **OK**.
- ❑ Choose **OK** to the warning dialog.
- ❑ Choose **OK** in the **Machine Control** dialog.
- ❑ Choose the **Cut Levels** button.
- ❑ Change the **Reference** to **WCS Origin**.
- ❑ Set the **Range Depth** to **0.0**.
- ❑ Choose **OK**.

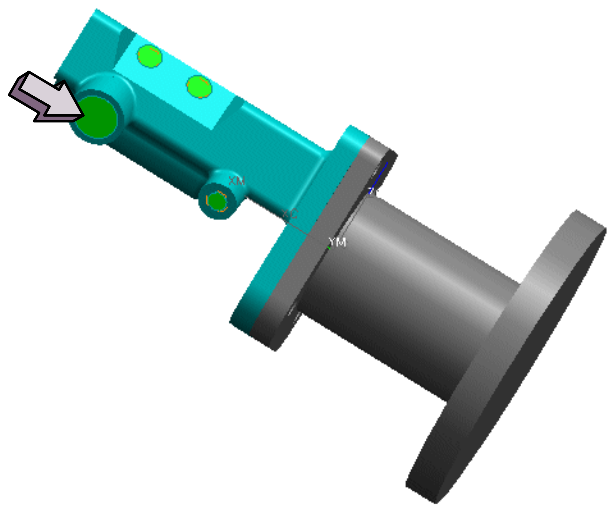
You will also need to reset the **Clearance Plane** for this operation.

- ❑ Choose the **Avoidance** button.

❑ Choose the **Clearance Plane** button.

❑ Choose **Specify**.

❑ Select the face as shown.



❑ Key in **2.00** for the **Offset**.

❑ Choose **OK** until the **ZLEVEL_PROFILE_STEEP** dialog is displayed.

Note that once that the Clearance Plane has been specified in the operation dialog, it will no longer inherit the Clearance Plane established in the MCS_MILL dialog.

❑ **Generate** the operation.

❑ When you have finished examining the operation, choose **OK**.

❑ In the Operation Navigator, highlight the **CA_1** operation, use **MB3** and choose **Copy**.

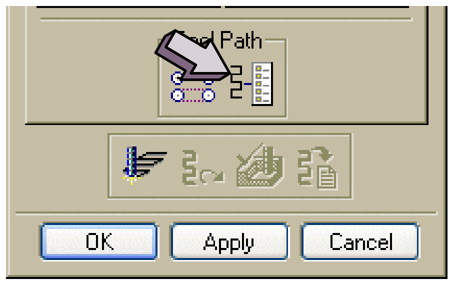
❑ Use **MB3** to paste the copied operation following the **CA_1** operation.

❑ **Rename** the copied operation to **CA_2**.

- ❑ Double-click on the operation **CA_2** to edit.

  This operation will require a tool axis change, just like in the previous operation. The option to change the tool axis is not on the dialog. You will add the option through Customized dialogs.

- ❑ On the **CONTOUR_AREA_NON_STEEP** dialog, choose the **Options** icon.

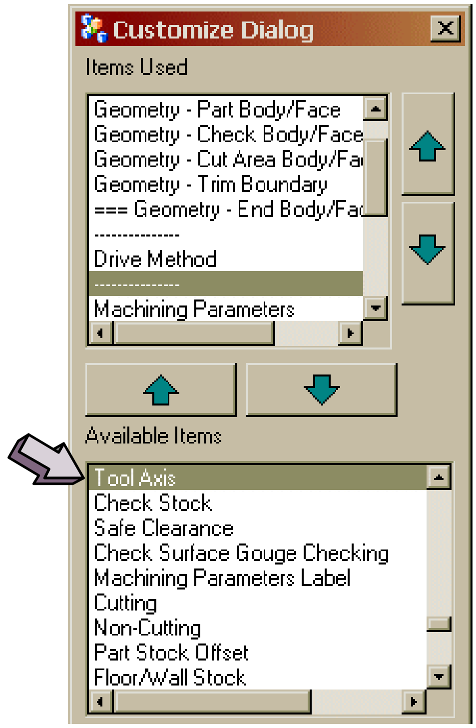


  The **Other Parameters** dialog is displayed.

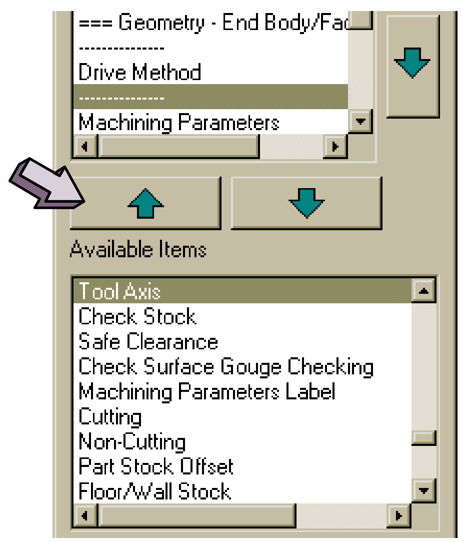
- ❑ Select the **Customize Dialog** button from the **Other Parameters** dialog.

  The **Customize Dialog** is displayed.

- ❑ From the bottom scroll listing window, highlight the **Tool Axis** option.



❑ Choose the **Add** button.



❑ Choose **OK** until the **CONTOUR_AREA_NON_STEEP** dialog appears.

❑ Choose the newly created **More** tab.

❑ Change the **Tool Axis** from **ZM Axis** to **Specify Vector**.

❑ In the **Vector Constructor** dialog, choose the **Spherical Coordinates** radio button.

- ❑ Key in **120.0** for the **Phi** angle; **90.0** for the **Theta** angle.
- ❑ Choose **OK**.
- ❑ Choose the **Main** tab.
- ❑ **Generate** the operation.
- ❑ On your own, create the final two finishing operations. Remember to place the tool axis at Phi=240.0 and Theta at 90.0, and then place the operations in the proper location in the Program Order view.

This completes the activity.

(This Page Intentionally Left Blank)

Glossary

ABS – Absolute coordinate system.

Absolute Coordinate System – Coordinate system in which all geometry is located from a fixed or absolute zero point.

active view – One of up to 49 views per layout in which you can directly work.

angle – In Unigraphics, an angle measured on the X-Y plane of a coordinate system is positive if the direction that it is swept is counterclockwise as viewed from the positive Z axis side of the X-Y plane. An angle swept in the opposite direction is said to be negative.

arc – An incomplete circle; sometimes used interchangeably with the term "circle."

ASCII – American Standard Code for Information Interchange. It is a set of 8-bit binary numbers representing the alphabet, punctuation, numerals, and other special symbols used in text representation and communications protocol.

aspect ratio – The ratio of length to height which represents the change in size of a symbol from its original.

assembly – A collection of piece parts and sub-assemblies representing a product. In Unigraphics, an assembly is a part file which contains components.

assembly part – A Unigraphics part file which is a user-defined, structured combination of sub-assemblies, components and/or objects.

associativity – The ability to tie together (link) separate pieces of information to aid in automating the design, drafting, and manufacture of parts in Unigraphics.

attributes – Pieces of information that can be associated with Unigraphics geometry and parts such as assigning a name to an object.

block font – A Unigraphics character font which is the default font used for creating text in drafting objects and dimensions.

body – Class of objects containing sheets and solids (see solid body and sheet body).

bottom-up modeling – Modeling technique where component parts are designed and edited in isolation of their usage within some higher level assembly. All assemblies using the component are automatically updated when opened to reflect the geometric edits made at the piece part level.

boundary – A set of geometric objects that describes the containment of a part from a vantage point.

CAD/CAM – Computer Aided Design/Computer Aided Manufacturing.

category, layer – A name assigned to a layer, or group of layers. A category, if descriptive of the type of data found on the layers to which it is assigned, will assist the user in identifying and managing data in a part file.

chaining – A method of selecting a sequence of curves which are joined end-to-end.

circle – A complete and closed arc, sometimes used interchangeably with the term "arc."

component – A collection of objects, similar to a group, in an assembly part. A component may be a sub-assembly consisting of other, lower level components.

component part – The part file or "master" pointed to by a component within an assembly. The actual geometry is stored in the component part and referenced, not copied, by the assembly. A separate Unigraphics part file that the system associates with a component object in the assembly part.

cone direction – Defines the cone direction using the Vector Subfunction.

cone origin – Defines the base origin using the Point Subfunction.

half angle – The half vertex angle defines the angle formed by the axis of the cone and its side.

constraints – Refer to the methods you can use to refine and limit your sketch. The methods of constraining a sketch are geometric and dimensional.

construction points – Points used to create a spline. Construction points may be used as poles (control vertices), defining points, or data points. See POLES, DEFINING POINTS, and DATA POINTS.

control point – Represents a specific location on an existing object. A line has three control points: both end points and the midpoint of the line. The control point for a closed circle is its center, while the control points for an open arc are its end and midpoints. A spline has a control point at each knot point. A control point is a position on existing geometry. Any of the following points: 1. Existing Points 2. Endpoints of conics 3. Endpoints and midpoints of open arcs 4. Center points of circles 5. Midpoints and endpoints of lines 6. Endpoints of splines.

convert curve – A method of creating a b-curve in which curves (lines, arcs, conics or splines) may be selected for conversion into a b-curve.

Coordinate System — A system of axes used in specifying positions (CSYS).

counterclockwise — The right-hand rule determines the counter- clockwise direction. If the thumb is aligned with the ZC axis and pointing in the positive direction, counterclockwise is defined as the direction the fingers would move from the positive XC axis to the positive YC axis.

current layout — The layout currently displayed on the screen. Layout data is kept in an intermediate storage area until it is saved.

curve — A curve in Unigraphics is any line, arc, conic, spline or b-curve. A geometric object; this may refer to a line, an arc, a conic, or a spline.

defaults — Assumed values when they are not specifically defined.

defining points — Spline construction points. Splines created using defining points are forced to pass through the points. These points are guaranteed to be on the spline.

degree-of-freedom arrows — Arrow-like indicators that show areas that require more information to fully constrain a sketch.

design in context — The ability to directly edit component geometry as it is displayed in the assembly. Geometry from other components can be selected to aid in the modeling. Also referred to as edit in place.

dimensional constraint — This is a scalar value or expression which limits the measure of some geometric object such as the length of a line, the radius of an arc, or the distance between two points.

directory — A hierarchical file organization structure which contains a list of filenames together with information for locating those files.

displayed part — The part currently displayed in the graphics window.

edit in place — See design in context.

emphasize work part — A color coding option which helps distinguish geometry in the work part from geometry in other parts within the same assembly.

endpoint — An endpoint of a curve or an existing point.

expression — An arithmetic or conditional statement that has a value. Expressions are used to control dimensions and the relationships between dimensions of a model.

face — A region on the outside of a body enclosed by edges.

feature – An all-encompassing term which refers to all solids, bodies, and primitives.

file – A group or unit of logically related data which is labeled or “named” and associated with a specified space. In Unigraphics, parts, and patterns are a few types of files.

filtering – See object filtering.

font box – A rectangle or “box” composed of dashed line objects. The font box defines the size, width and spacing of characters belonging to a particular font.

font, character – A set of characters designed at a certain size, width and spacing.

font, line – Various styles of lines and curves, such as solid, dashed, etc.

free form feature – A body of zero thickness. (see body and sheet body)

generator curve – A contiguous set of curves, either open or closed, that can be swept or revolved to create a body.

geometric constraint – A relationship between one or more geometric objects that forces a limitation. For example, two lines that are perpendicular or parallel specifies a geometric constraint.

grid – A rectangular array of implied points used to accurately align locations which are entered by using the “screen position” option.

guide curve – A set of contiguous curves that define a path for a sweep operation.

virtual intersection – Intersection formed by extending two line segments that do not touch to the position that they cross. The line segments must be non-parallel and coplanar.

inflection – A point on a spline where the curve changes from concave to convex, or vice versa.

interactive step – An individual menu in a sequence of menus used in performing a Unigraphics function.

isometric view (Tfr-ISO) – Isometric view orientation – one where equal distances along the coordinate axes are also equal to the view plane. One of the axes is vertical.

knot points – The defining points of a spline. Points along a B-spline, representing the endpoints of each spline segment.

layer – A layer is a partition of a part. Layers are analogous to the transparent material used by conventional designers. For example, the user may create all geometry on one layer, all text and dimensions on a second, and tool paths on a third.

layout – A collection of viewports or window areas, in which views are displayed. The standard layouts in Unigraphics include one, two, four or six viewports.

layouts – Standard layouts are available to the user. These include:
L1 – Single View,
L2 – Two Views,
L3 – Two Views,
L4 – Four Views,
L6 – Six Views.

Information window – The window used in listing operations, such as **Info**.

loaded part – Any part currently opened and in memory. Parts are loaded explicitly using the *File→Open* option and implicitly when they are used in an assembly being opened.

menu – A list of options from which the user makes a selection.

model space – The coordinate system of a newly created part. This is also referred to as the "absolute coordinate system." Any other coordinate system may be thought of as a rotation and/or translation of the absolute coordinate system.

name, expression – – The name of an expression is the single variable on the left hand side of the expression. All expression names must be unique in a part file. Each expression can have only one name. See expression.

objects – All geometry within the Unigraphics environment.

offset face – A Unigraphics surface type created by projecting (offsetting) points along all the normals of a selected surface at a specified distance to create a "true" offset.

options – A number of various alternatives (functions, modes, parameters, etc.) from among which the user can choose.

origin – The point X = 0, Y = 0, Z = 0 for any particular coordinate system.

parametric design – Concept used to define and control the relationships between the features of a model. Concept where the features of the model are defined by parameters.

part – A Unigraphics file containing a .prt extension. It may be a piece part containing model geometry, a sub-assembly, or a top-level assembly.

part or model – A collection of Unigraphics objects which together may represent some object or structure.

partially loaded part – A component part which, for performance reasons, has not been fully loaded. Only those portions of the component part necessary to render the higher level assembly are initially loaded (the reference set).

point set – A distribution of points on a curve between two bounding points on that curve.

Point Subfunction Menu – A list of options (methods) by which positions can be specified in Unigraphics.

read-only part – A part for which the user does not have write access privilege.

real time dynamics – Produces smooth pan, zoom, and rotation of a part, though placing great demand on the CPU.

Refresh – A function which causes the system to refresh the display list on the viewing screen. This removes temporary display items and fills in holes left by ***Blank*** or ***Delete.***

right-hand rule, conventional – The right-hand rule is used to determine the orientation of a coordinate system. If the origin of the coordinate system is in the palm of the right fist, with the back of the hand lying on a table, the outward extension of the index finger corresponds to the positive Y axis, the upward extension of the middle finger corresponds to the positive Z axis, and the outward extension of the thumb corresponds to the positive X axis.

right-hand rule for rotation – The right-hand rule for rotation is used to associate vectors with directions of rotation. When the thumb is extended and aligned with a given vector, the curled fingers determine the associated direction of rotation. Conversely, when the curled fingers are held so as to indicate a given direction of rotation, the extended thumb determines the associated vector.

screen cursor (cursor) – A marker on the screen which the user moves around using some position indicator device. Used for indicating positions, selecting objects, etc. Takes the form of a full-screen cross.

sheet – A object consisting of one or more faces not enclosing a volume. A body of zero-thickness. Also called sheet body.)

sketch – A collection of geometric objects that closely approximates the outline of a particular design. You refine your sketch with dimensional and geometric constraints until you achieve a precise representation of your design. The sketch can then be extruded or revolved to obtain a 3D object or feature.

Sketch Coordinate System (SCS) – The SCS is a coordinate system which corresponds to the plane of the sketch. When a sketch is created the WCS is changed to the SCS of the new sketch.

solid body – An enclosed volume. A type of body (see Body).

spline – A smooth free-form curve.

stored layout – The last saved version of a layout.

stored view – The last saved version of a view.

string – A contiguous series of lines and/or arcs connected at their end points.

sub-assembly – A part which both contains components and is itself used as a component in higher-level assemblies.

surface – The underlying geometry used to define a face on a sheet body. A surface is always a sheet but a sheet is not necessarily a surface (see sheet body). The underlying geometry used to define the shape of a face on a sheet.

system – The Unigraphics System.

temporary part – An empty part which is optionally created for any component parts which cannot be found in the process of opening an assembly.

top-down modeling – Modeling technique where component parts can be created and edited while working at the assembly level. Geometric changes made at the assembly level are automatically reflected in the individual component part when saved.

trim – To shorten or extend a curve.

trimetric view (Tfr-Tri) – A viewing orientation which provides you with an excellent view of the principal axes. In Unigraphics II, the trimetric view has the Z-axis vertical. The measure along the X-axis is 7/8 of the measure along Z, and the measure along the Y-axis is 3/4 of the measure along Z.

Unigraphics – A computer based turnkey graphics system for computer-aided design, drafting, and manufacturing, produced by UGS.

units – The unit of measure in which you may work when constructing in Unigraphics. Upon log on, you may define the unit of measure as inches or millimeters.

upgraded component – A component which was originally created pre-V10 but has been opened in V10 and upgraded to remove the duplicate geometry.

version – A term which identifies the state of a part with respect to a series of modifications that have been made to the part since its creation.

view – A particular display of the model. View parameters include view orientation matrix; center; scale; X,Y and Z clipping bounds; perspective vector; drawing reference point and scale. Eight standard views are available to the user: Top, Front, Right, Left, Bottom, Back, Tfr-ISO (top-front-right isometric), and Tfr-Tri (top-front-right trimetric).

view dependent edit – A mode in which the user can edit a part in the current work view only.

view dependent modifications – Modifications to the display of geometry in a particular view. These include erase from view and modify color, font and width.

view dependent geometry – Geometry created within a particular view. It will only be displayed in that view.

WCS – Work Coordinate System.

WCS, work plane – The WCS (Work Coordinate System) is the coordinate system singled out by the user for use in construction, verification, etc. The coordinates of the WCS are called work coordinates and are denoted by XC, YC, ZC. The XC-YC plane is called the work plane.

Work Coordinate System – See WCS.

work layer – The layer on which geometry is being constructed. You may create objects on only one layer at a time.

work part – The part in which you create and edit geometry. The work part can be your displayed part or any component part which is contained in your displayed assembly part. When displaying a piece part, the work part is always the same as the displayed part.

work view – The view in which work is being performed. When the creation mode is view dependent, any construction and view dependent editing that is performed will occur only in the current work view.

XC axis – X-axis of the work coordinate system.

YC axis – Y-axis of the work coordinate system.

ZC axis – Z-axis of the work coordinate system.

Index

Numbers

A

B

C

D

E

F

G

H

I

K

L

M

N

O

P

R

S

Reference Chart Tear Outs

These tear out reference charts are provided for your convenience.

(This Page Intentionally Left Blank)

STUDENT PROFILE

In order to stay in tune with our customers we are requesting that you take a little time to answer these questions. This information will be kept confidential, and will not be shared with anyone outside of Education Services. PLM Solutions Education Services thanks you for your participation and hopes your training experience will be an outstanding one. Please print the following:

Your Name____**Date**___________________________

Course Number__________________**Course Title**_______________________________________

Employer____

When is your planned departure time?_______________ **U.S. citizen?** ❑ **YES** ❑ **NO**

1. Please select the job that best describes your present one.

❑ Designer MFG	❑ Designer Product	❑ Designer Tooling	❑ Quality Engineer
❑ Engineer MFG	❑ Engineer Product	❑ Engineer Tooling	❑ Process Planner
❑ NC Dies	❑ NC Other	❑ NC Tooling	❑ Supplier Quality
❑ Drafter MFG	❑ Drafter Product	❑ Drafter Tooling	❑ Workflow Admin.
❑ Manager Design	❑ Manager Engineering	❑ Manager Shop Floor	❑ Collaboration Admin.
❑ Sys. Admin Local Network	❑ Sys. Admin Enterprise	❑ Systems Manager	❑ Configuration Analyst

2. Your job responsibilities are:____

3. Reason for training____

4. What other CAD/CAM/CAE /PDM software have you used?______________________

5. What types of parts/data do you model/analyze or store?______________________

   ____

6. Have you ever had any other instructor–lead/ on–line or self–paced classes for the following:

Subject	From Whom	When	Course Name
❑ **Unigraphics**			
❑ **I–deas**			
❑ **Imageware**			
❑ **Teamcenter Manufacturing**			
❑ **Teamcenter Engineering (I–Man)**			
❑ **Team Center Enterprise (Metaphase)**			
❑ **Dimensional Management / Visualization**			

Please list your CAD/CAM/CAE/PDM experience and indicate the level of understanding by using the terms novice, intermediate or advanced.

Advanced Mill Applications
Class Agenda

Day 1 ________Morning

- Introduction and Overview
- Lesson 1 Wave Geometry Linker in Manufacturing
- *Workbook* *Project Description*
- Lesson 2 In–Process Workpiece

Afternoon

- Lesson 3 Fixed Contour
- *Workbook Section 1* *Process Planning*
- *Workbook Section 2* *Manufacturing Operation Preparation*
- *Workbook Section 3* *Cavity Mill*

Day 2 ________Morning

- Lesson 4 Libraries
- *Workbook Section 4* *Fixed Contour*
- Lesson 5 Machining Faceted Geometry

Afternoon

- Lesson \6 High Speed Machining
- *Workbook Section 5* *Flowcut*
- Lesson 7 NC Assistant
- Lesson 8 Templates
- *Workbook Section 6* *Fixed Contour Finishing*

Day 3 ________Morning

- Lesson 8 Templates (continued)
- Lesson 9 Hole Making
- *Workbook Section 7* *Holemaking*

Afternoon

- Lesson 10 Integrated Simulation and Verification

(This Page Intentionally Left Blank)

EDS

Level 1 Evaluation

Course Code_______ Course__________________ Date________ Instructor_____________

1. STRONGLY DISAGREE	2. DISAGREE	3. SOMEWHAT AGREE	4. AGREE	5. STRONGLY AGREE

Content:

1. The course objectives were clear______________________________ 1 2 3 4 5
3. The goals and objectives for this course were met_______________ 1 2 3 4 5
4. The course flowed in a logical and meaningful manner___________ 1 2 3 4 5
5. The course was the appropriate length for the info taught________ 1 2 3 4 5

Materials:

1. The training materials supported the course objectives__________ 1 2 3 4 5
2. The training materials were logically sequenced________________ 1 2 3 4 5
3. Enough information was provided to complete workshops________ 1 2 3 4 5
4. The course materials provided clear and descriptive directions____ 1 2 3 4 5
5. The exercises and workshops supported the learning experience___ 1 2 3 4 5
6. The materials were easy to read and understand________________ 1 2 3 4 5

Instructor:

1. The instructor was knowledgeable about the subject
2. He/She answered my questions appropriately__________________ 1 2 3 4 5
3. He/She was well prepared to deliver the course________________ 1 2 3 4 5
4. He/She was organized and well spoken_________________________ 1 2 3 4 5
5. The instructor encouraged questions in class___________________ 1 2 3 4 5
6. The instructor made good use of the training time_______________ 1 2 3 4 5
7. The instructor completed the course___________________________ 1 2 3 4 5
8. The instructor used examples relevant to the course_____________ 1 2 3 4 5
9. The instructor provided enough time to complete the exercises____ 1 2 3 4 5
10. The instructor used review tests and gave feedback______________ 1 2 3 4 5

Student Self–Evaluation:

1. I met the prerequisites for the class (I had the skills I needed)_____ 1 2 3 4 5
2. I will be able to use the skills I have learned on my job___________ 1 2 3 4 5
3. My expectations for this course were met_______________________ 1 2 3 4 5
4. I am confident that with practice I will become proficient________ 1 2 3 4 5

Facilities:

1. The computer equipment was reliable__________________________ 1 2 3 4 5
2. The software performed properly______________________________ 1 2 3 4 5
3. The overhead projection unit was clear and working properly_____ 1 2 3 4 5
4. The registration and confirmation process was efficient_________ 1 2 3 4 5
5. The training facilities were comfortable and clean and provided a good learning environment______________________________________ 1 2 3 4 5

(Over)

Hotels: (Optional)

1. **Was this hotel recommended during your registration process?___________** ☐ **YES** ☐ **NO**

   If not how was it chosen____

2. **Was the hotel clean?____** ☐ **YES** ☐ **NO**

3. **Did it have a full–serve restaurant?___________________________________** ☐ **YES** ☐ **NO**

4. **Overall impression of the Hotel_______________________________**

1	2	3	4	5

5. **What was the name of the hotel?_______________________________**

☐ **Please check this box if you would like your comments below featured in our training publications. (Your name is required at the bottom of this form)**

☐ **Please check this box if you would like to receive more information on our other courses and services. (Your name is required at the bottom of this form)**

We would appreciate it if you would take a minute to add some personal comments on this form. These comments help us to better understand the ever–changing student needs.

Thank you for choosing EDS PLM Education Services, and we hope we can be of help to you in the future.

Comments:____

Name Optional:___________________________________ Location_____________________

Software Version__________________

The NC Programming Sequence — 7 Easy Steps

The Manufacturing application programming sequence is to:

- Create the Manufacturing Assembly
- Choose the Manufacturing Environment
- Establish the Group Parents
- Create the Operation(s)
- Verify Your Work
- Postprocess
- Shop Docs

(This Page Intentionally Left Blank)