Advanced Machining Applications

Student Guide
March 2005
MT11045 – NX 3

Manual History

Proprietary and Restricted Rights Notices

The following proprietary and restricted rights notices apply.

Copyright

Copyright ©2005 UGS Corp. All Rights Reserved.

Restricted Rights Legend

This software and related documentation are proprietary to UGS Corp. LIMITATIONS TO U.S. GOVERNMENT RIGHTS. UNPUBLISHED – RIGHTS RESERVED UNDER THE COPYRIGHT LAWS OF THE UNITED STATES. This computer software and related computer software documentation have been developed exclusively at private expense and are provided subject to the following rights. If this computer software and computer software documentation qualify as "commercial items" (as that term is defined in FAR 2.101), their use, duplication or disclosure by the U.S. Government is subject to the protections and restrictions as set forth in the UGS Corp. commercial license for the software and/or documentation as prescribed in FAR 12212 and FAR 27.405(b)(2)(i) (for civilian agencies) and in DFARS 227.7202–1(a) and DFARS 227-7202-3(a) (for the Department of Defense), or any successor or similar regulation, as applicable or as amended from time to time. If this computer software and computer documentation do not qualify as "commercial items," then they are "restricted computer software" and are provided with "restrictive rights," and their use duplication or disclosure by the U.S. Government is subject to the protections and restrictions as set forth in FAR 27.404(b) and FAR 52–227–14 (for civilian agencies), and DFARS 227.7203–5(c) and DFARS 252.227–7014 (for the Department of Defense), or any successor or similar regulation, as applicable or as amended from time to time. UGS Corp., 5800 Granite Parkway, Suite 600, Plano, TX 75024.

All trademarks belong to their perspective holders.

© UGS Corp.
All rights reserved.
Printed in the United States of America.

Manual Revision	Version	Publication Date
	Version 16.0	October 2000
	Version 17.0	January 2001
	Version 18.0	August 2001
	Unigraphics NX	September 2002
	Unigraphics NX 2	January 2004
	NX 3	March 2005

This edition obsoletes all previous editions.

Contents

©UGS Corp., All Rights Reserved

 ©UGS Corp., All Rights Reserved

©UGS Corp., All Rights Reserved

Course Overview

Course Description

The Advanced Machining Applications 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 NX 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, machine faceted bodies and use various cut patterns and operation types to perform high speed machining.

Student Responsibilities

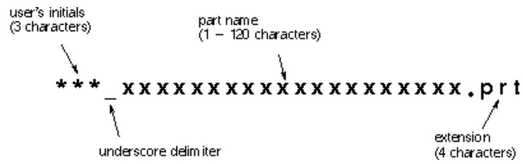
- Be on time
- Participate in class
- Stick with the subject matter
- Listen attentively and take notes
- Practice on the job what you have learned
- Have fun

Class Standards for NX Part Files

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

Class Part File Naming

This class utilizes the following file naming standard:



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

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)

 ©UGS Corp., All Rights Reserved

Colors

The following colors are preset to indicate different object types:

Object	Valid Colors
Solid Bodies	Green
Sheet Bodies	Yellow
Lines and Arc (non-sketch curves)	Green
Conics and Splines (non-sketch curves)	Green
Sketch Curves	Cyan
Reference Curves (in sketches)	Gray
Datum Features	Aquamarine
Points and Coordinate Systems	Green
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

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

©UGS Corp., All Rights Reserved

activity. Below each step are action boxes which emphasize the individual actions that must be taken to accomplish the step. As your knowledge of NX increases, the action boxes may seem redundant as the step text becomes all that is needed to accomplish a given task.

Step 1: This is an example of a step.

☐ This is an example of an action box.

The general format for lesson content is:

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

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

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

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

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

Workbook Overview

The Advanced Machining Application project allows 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:

User name:

____

Password:

____

 ©UGS Corp., All Rights Reserved

Work Directory:

Parts Directory:

Instructor:

Date:

©UGS Corp., All Rights Reserved

Lesson

1 *WAVE Geometry Linker in Manufacturing*

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.

Objective

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.

The WAVE Geometry Linker

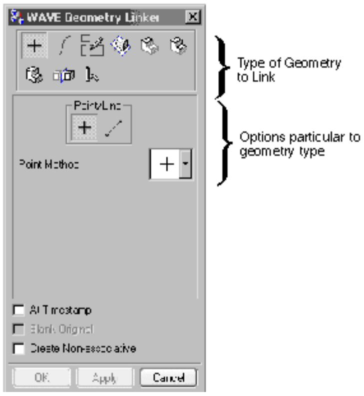
The WAVE Geometry Linker is used to associatively copy geometry from a component part in an assembly into the work part. The resulting linked geometry is associated to the parent geometry. Modifying the parent geometry will cause the linked geometry in the other parts to update.

The WAVE Geometry Linker is available with an Assemblies license. It does not require a NX 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 Insert→Associative Copy→WAVE Geometry Linker from the menu bar.

©UGS Corp., All Rights Reserved



- 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.

 ©UGS Corp., All Rights Reserved

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

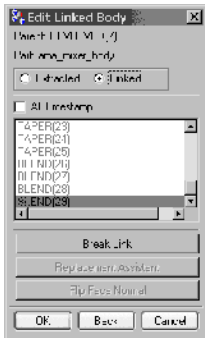
When selecting geometry to copy, you should consider how permanent the geometry will be. If you copy as little geometry as possible to do the job, performance will be improved but updates will be less robust when the parent geometry is altered.

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.

©UGS Corp., All Rights Reserved

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.

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.

 ©UGS Corp., All Rights Reserved

- *An Extracted* feature (intra-part) can be converted to a Linked feature (inter-part) 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.

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).

 ©UGS Corp., All Rights Reserved

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.

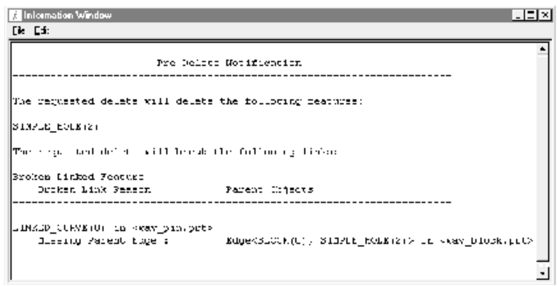
After a link is accepted, its status is changed to broken until a new parent is defined.

©UGS Corp., All Rights Reserved

Deleting Parent Geometry

To prevent unintentional deletion of the parents of linked geometry, a message will warn you if a delete operation would cause inter-part 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.



 ©UGS Corp., All Rights Reserved

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.

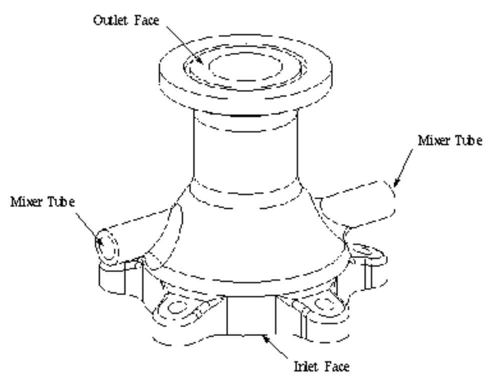
Activity: Creating an Assembly for WAVE

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 NX 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, and save it with a new name.

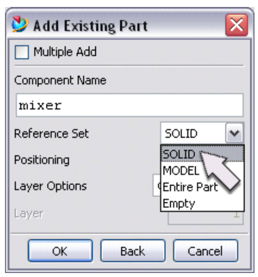
- ☐ If necessary, start NX.
- ☐ Use **File→Open**. 
- ☐ Navigate to your parts folder and open the file.
- ☐ Choose **File→Save As ***_mixer_mfg** where ***** represents your initials.

 ©UGS Corp., All Rights Reserved

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**.
- ☐ 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.

> 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.

☐ In the **File Name** field, of the Select Part Name dialog, 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**. 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 NX, all operations apply to the work part, which is currently *****_mixer_mfg**. 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**.

 ©UGS Corp., All Rights Reserved

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.

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

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.

 ©UGS Corp., All Rights Reserved

Activity: 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.

☐ Highlight the component *****_mixer_casting** in the Assembly Navigator, and make it the Work Part by using MB3, and selecting **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 **Insert→Associative Copy→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.

©UGS Corp., All Rights Reserved

☐ In the **Assembly Navigator** use **MB3** over the *****_mixer_casting** component, and choose **Make Displayed Part**.

☐ In the graphics window, use **MB3→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**(green check mark) .

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.

 ©UGS Corp., All Rights Reserved

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.

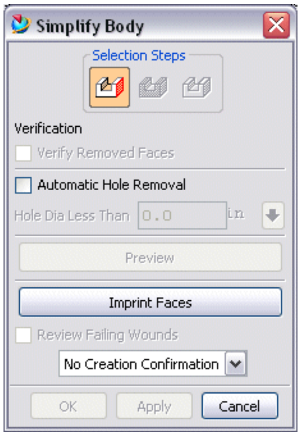
©UGS Corp., All Rights Reserved

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.



 ©UGS Corp., All Rights Reserved

Activity: 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.

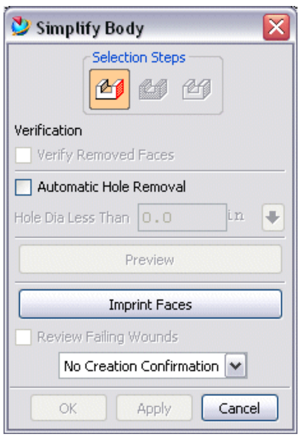
☐ In the Assembly Navigator, use **MB3** on the *****_mixer_casting** component and choose **Make Displayed Part**.

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

☐ Choose **Application→Modeling**.

☐ Choose **Insert→Direct Modeling→Simplify**.

The **Simplify Body** dialog is displayed.



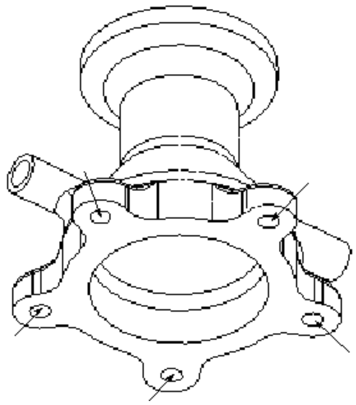
The cue line reads **Select retained faces**.

☐ 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.

©UGS Corp., All Rights Reserved



When selected as a boundary face, NX 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 the **Retained Faces** icon 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**.

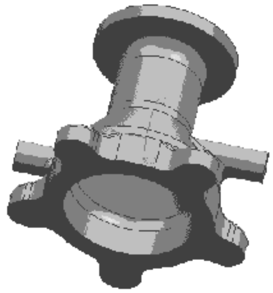
Now all faces except the five holes highlight.

☐ 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).

 ©UGS Corp., All Rights Reserved



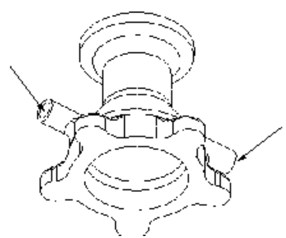
☐ Dismiss the Simplify Body information dialog by choosing **OK**.

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 the **Boundary Faces** icon.

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

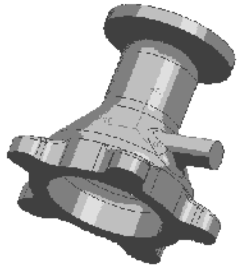


☐ Choose the **Retained Faces** icon.

☐ 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.

©UGS Corp., All Rights Reserved

Activity: 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.

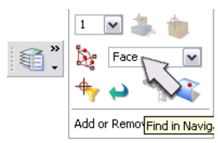
☐ Choose **Application→Modeling**.

☐ Choose **Insert →Design Feature→Extrude**.

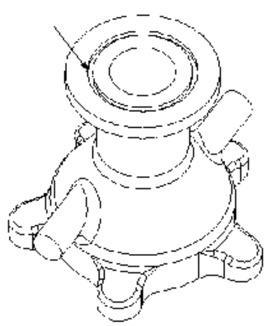
The **Extrude Widget** is displayed.

☐ Choose the **Extrude Dialog** icon.

☐ On the **Selection** toolbar change the type filter from **Any** to **Face**.

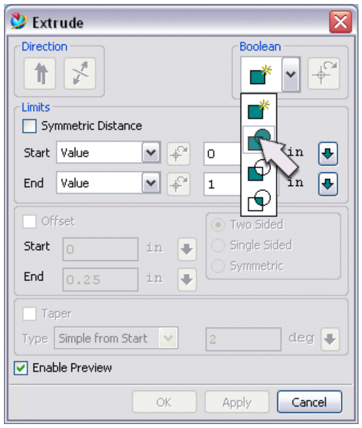


☐ Choose the bottom face of the ring groove, as shown below.



 ©UGS Corp., All Rights Reserved

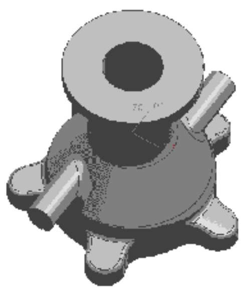
- ☐ Choose the **Unite** icon from the Boolean pull-down menu.



- ☐ Under **Limits**, End, change from **Value** to **Until Extended**.

- ☐ Select the outlet face, then choose **OK** on the Extrude dialog.

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



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.

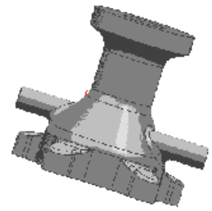
- ☐ From the menu bar choose **Insert→Offset/Scale→Offset Face**.

- ☐ In the Offset Faces dialog, key in **0.250** for the offset value.

- ☐ Select the inlet and outlet faces, and the two mixer tube faces.

- ☐ Choose **OK**.

©UGS Corp., All Rights Reserved



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**.

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.

 ©UGS Corp., All Rights Reserved

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**.

©UGS Corp., All Rights Reserved

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.

 ©UGS Corp., All Rights Reserved

Lesson

2 *In-Process Workpiece*

Purpose

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.

Objective

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.

In-Process Workpiece (IPW) Overview

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.

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 and resultant IPW for each operation.

 ©UGS Corp., All Rights Reserved

Activity: 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**.

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

☐ Choose **Application→Manufacturing**.

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

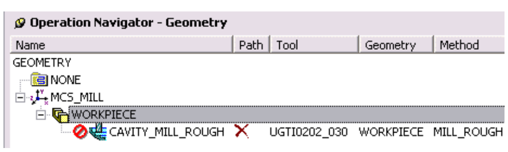
An In-Process Workpiece (IPW) column can be added to the Operation Navigator by clicking **MB3** on the Operation Navigator background, choosing **Columns**, and turning the **IPW** option **on**.

Step 2: Displaying the Part Geometry.

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

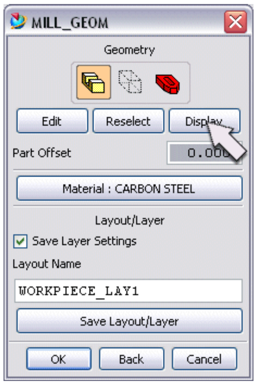
☐ In the **Geometry** view of the Operation Navigator, expand all objects.

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



The MILL_GEOM dialog is displayed.

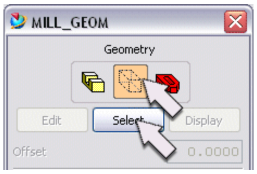
☐ With the **Part** icon selected, choose **Display**.



Step 3: Define the Blank Geometry.

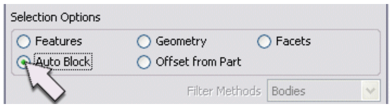
You will define the blank geometry using a method that creates a solid body automatically by enclosing the part geometry.

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

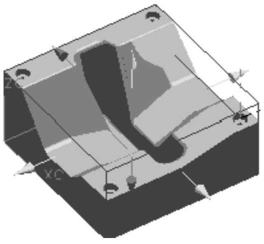


The Blank Geometry dialog is displayed.

☐ 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.



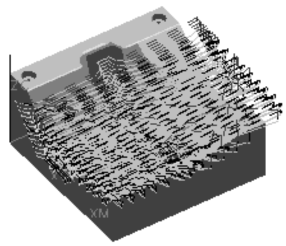
☐ Choose **OK** to accept the blank geometry with no additional offsets.

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

Step 4: Generate the tool path.

 ©UGS Corp., All Rights Reserved

- ☐ 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.

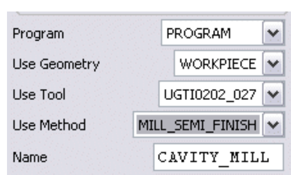
Step 5: Create a Semi-Finishing operation.

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** toolbar.

The Create Operation dialog is displayed.

- ☐ If necessary choose **mill_contour** as the **type**.
- ☐ 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.

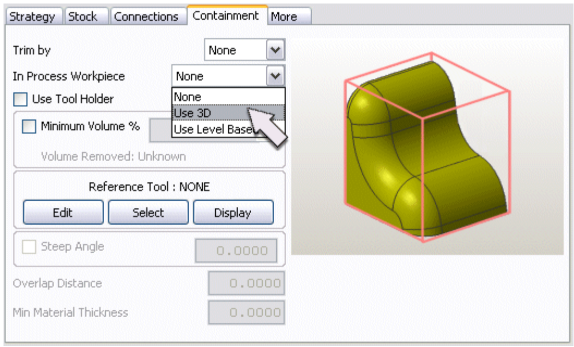
You will specify use of the IPW in the previous operation to define the blank geometry in this current operation.

- ☐ Choose the **Cutting** button.

The Cut Parameters dialog is displayed.

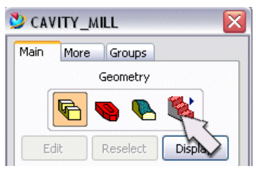
☐ Choose the **Containment** tab.

☐ Select **Use 3D** from the pull-down menu for the **In Process Workpiece**.

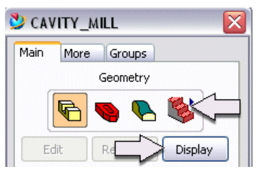


☐ Choose **OK** to accept the Cut Parameters dialog.

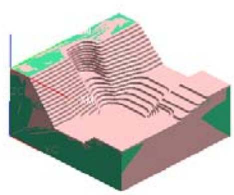
The **Blank** icon at the top of the **CAVITY_MILL** dialog Main property page has been replaced by the **Previous IPW** icon.



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



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



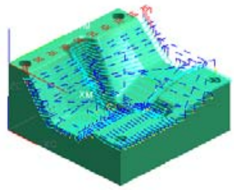
This faceted body is the IPW that this operation uses as blank geometry.

Step 7: Generate the tool path.

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

 ©UGS Corp., All Rights Reserved

☐ Turn the three **Display Parameter** options **off** and choose **OK** to continue generating the tool path.



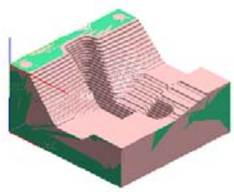
☐ Refresh the graphics display.

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.

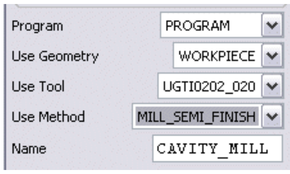
You will now create a finishing operation that uses the IPW defined by the semi-finishing operation.

☐ Select the **Create Operation** icon from the **Create** toolbar.

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

©UGS Corp., All Rights Reserved

☐ 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.

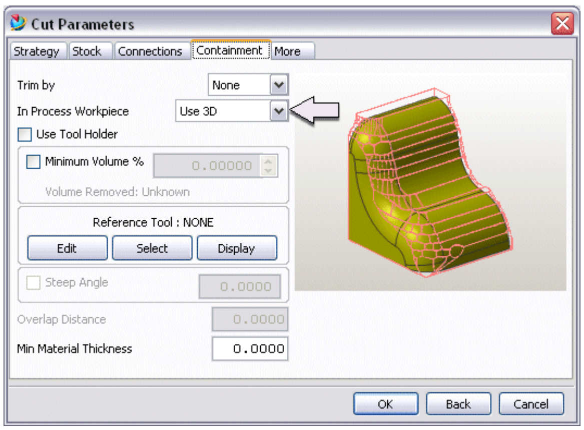
You will specify that the IPW in the previous operation will define the blank geometry in this operation.

☐ Choose the **Cutting** button.

The **Cut Parameters** dialog is displayed.

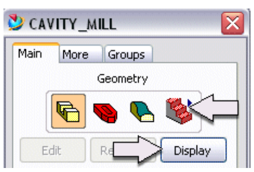
☐ Choose the **Containment** tab.

☐ Select **Use 3D** from the pull-down menu for the **In Process Workpiece**.



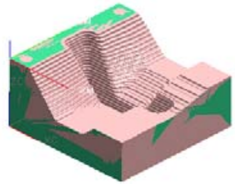
☐ Choose **OK** to accept the Cut Parameters dialog.

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



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

 ©UGS Corp., All Rights Reserved



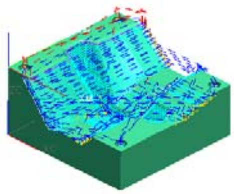
This faceted body is the IPW that this operation uses as blank geometry.

☐ Key in **0.100** in the **Global 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.



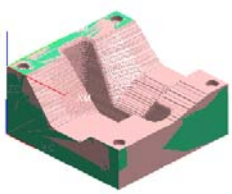
☐ Refresh the graphics display.

Step 12: Display the resulting IPW.

The IPW created by this operation can now be displayed.

☐ Choose **Display Resulting IPW**.

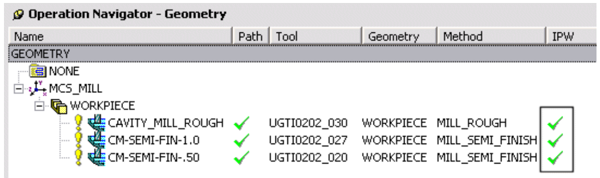
The processor may require some processing time to display the faceted body.



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

The check marks in the IPW column indicate which operations contain resulting IPW’s.

©UGS Corp., All Rights Reserved



If a 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.

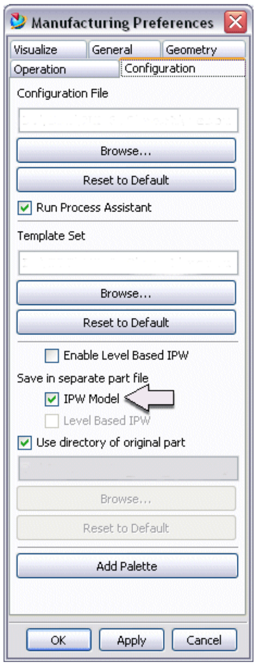
 ©UGS Corp., All Rights Reserved

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.



©UGS Corp., All Rights Reserved

Summary

Rest milling can be performed by creating an associative In-Process Workpiece \\(IPW\\) in an operation and using it as blank geometry in the next operation. The In-Process Workpiece provides an efficient and robust method of using material left by previous cutting operations as blank geometry for the next operation in the program.

In this lesson you:

- Used Auto Block to create blank geometry for the initial roughing operation.
- Turned on the Use 3D IPW option so that the IPW created by the previous operation will be used as blank geometry in the current operation.
- Used the Previous IPW option to display the IPW being used.
- Used the Display Resulting IPW option to display the IPW created in the current operation.

Lesson

3 Libraries

Purpose

This lesson 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.

Objective

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.

©UGS Corp., All Rights Reserved

Overview of CAM Libraries

Libraries are a convenient and easy tool that are used to access reference data. Currently libraries can access information related to:

- Cutting tools
- Machine tools
- Part materials
- Cutting tool materials
- Cut methods
- Speeds and feeds

Cutting tool information is located in the file tool_database.dat and can be modified through a text editor. Machining data, Cut Methods, Tool Materials, Part Materials, Tool Machining Data, Machine Tools and Importing text files can be accessed by selecting Tools → Edit Machining Data Libraries and then selection of the appropriate tab. Respective, individual data files, representing the above categories may also be modified through a text editor.

The configuration selected at the beginning of your CAM session (i.e. mill_contour) defines the location of the various external libraries. Each external library is represented by one line of information which identifies library type and points to the Event Handler (file with .tcl extension) and Definition file (file with .def extension) that are used for the Data Base Connection (DBC). The Definition file is used to establish a class hierarchy (the way tools are organized) and associated mechanism for queries, establishes dialog layout definitions, attribute mappings, option menu definitions, library reference names and delimiters.

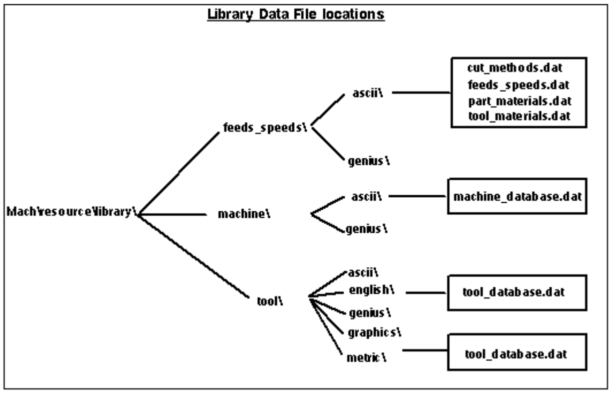
Sample Configuration File (mill_contour.dat)

```
TEMPLATE_OPERATION,${UGII_CAM_TEMPLATE_SET_DIR}mill_contour.opt
TEMPLATE_DOCUMENTATION,${UGII_CAM_SHOP_DOC_DIR}shop_doc.dat
TEMPLATE_POST,${UGII_CAM_POST_DIR}template_post.dat
USER_DEFINED_EVENTS,${UGII_CAM_USER_DEF_EVENT_DIR}ude.cdl,${UGII_CAM_USER_DEF_EVENT_DIR}ude.tcl
TEMPLATE_CLSF,${UGII_CAM_TOOL_PATH_DIR}template_clsf.dat
LISTING_FORMAT,${UGII_CAM_TOOL_PATH_DIR}clsf.def,${UGII_CAM_TOOL_PATH_DIR}clsf_listing.tcl
LIBRARY_TOOL,${UGII_CAM_LIBRARY_TOOL_ASCII_DIR}dbc_tool_ascii.def,
        ${UGII_CAM_LIBRARY_TOOL_ASCII_DIR}dbc_tool_ascii.tcl
LIBRARY_MACHINE,${UGII_CAM_LIBRARY_MACHINE_ASCII_DIR}dbc_machine_ascii.def,
        ${UGII_CAM_LIBRARY_MACHINE_ASCII_DIR}dbc_machine_ascii.tcl
LIBRARY_FEEDS_SPEEDS,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}feeds_speeds.def,
        ${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}feeds_speeds.tcl
LIBRARY_MACHINING_DATA,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}machining_data.def,
        ${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}machining_data.tcl
LIBRARY_TOOL_MACHINING_DATA,${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}tool_machining_data.def,
        ${UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR}tool_machining_data.tcl
LIBRARY_PART_MATERIAL,${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 files.

 ©UGS Corp., All Rights Reserved

Included in each release of NX are the access mechanisms to the ASCII text files. Sample ASCII based libraries are provided. GENIUS/4000 libraries are optional.



The following pertains to library files in general:

- Some library files are opened and read by the system only the first time they are read for performance reasons.
- When you edit library files, keep the *library references* (library references are names given to every entry in the library) unique.
- After editing library files, reset the configuration to force any changes to be read.
- Not all the information located in the library files are retrieved into the part file. Extra fields are used to aid in selection by NX POST and Shop Documentation.

©UGS Corp., All Rights Reserved

Activity: Preparation for modifying CAM Libraries

In order to modify the library or data files containing cutting tool information, 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 conductive for each student to have a copy of the library files in their home directory. In this activity you will make a copy of the *mach\resource* directory structure to your home directory and modify the directories for read/write access. Instructions are presented for Windows and Unix separately.

Windows Environment:

Step 1: Copy the Mach\resource directory.

- ☐ On the main menu bar, select **Help** → **NX Log File** to verify the *Mach* directory being used by looking 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\resource** directory, right-click on **Mach\resource** directory and select **Copy**.
- ☐ Highlight your home directory, right-click on your home directory and select **Paste**.

Step 2: If necessary, copy the NX environment file, ugii_env.dat to your home directory.

- ☐ From the Explorer window locate the **ugii_env.dat** file in the **\xxx\ugii** directory (where **xxx** represents the NX base directory).
- ☐ Highlight the **ugii_env.dat** file, right-click and select **Copy**.
- ☐ Highlight your home directory, right-click on your home directory and select **Paste**.

Step 3: Edit the ugii_env.dat file to redefine your Mach\resource directory location.

- ☐ Highlight the **ugii_env.dat** file from your home directory, right-click and select **Open**.
- ☐ Scroll down the file until you find the following line: **UGII_CAM_RESOURCE_DIR=${UGII_CAM_BASE_DIR} resource** and change the line to **UGII_CAM_RESOURCE_DIR=${Homedrive}resource**

 ©UGS Corp., All Rights Reserved

where Homedrive is the letter designator of the disk drive where your home directory is located.

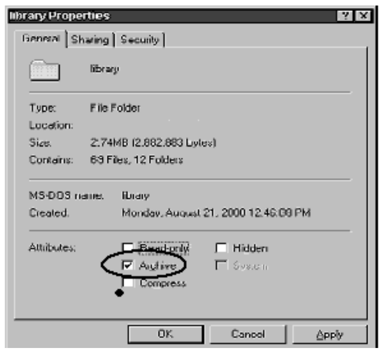
☐ Save the file and exit from Notepad.

Step 4: If necessary, change the Read-only protection on your just created local \Mach\resource\library directory to Archive.

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

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

☐ Un-check **Read-Only** and then Check **Archive**.



Step 5: Restart NX.

☐ Exit NX and then restart NX.

☐ On the menu bar, select **Help** → **NX Log File** to verify that your resource directory is being used.

Unix Environment:

Step 1: Copying the mach/resource directory.

☐ On the menu bar, select **Help** → **NX Log File** to verify the *mach* directory being used by looking for the environment variable UGII_CAM_BASE_DIR.

☐ Open a terminal window making sure that your default directory is set to your home directory.

☐ Copy the **mach/resource** directory to your home directory. The path for the file will be the value obtained for UGII_CAM_BASE_DIR. The format will be similar to the following:

cp /usr/xxx/mach/resource (where xxx represents the NX base directory).

Step 2: Copying the NX environment file, .ugii_env to your home directory.

- ☐ Copy the **.ugii_env** file from **/usr/xxx/ugii** directory to your home directory. The format will be similar to the following:

  cp /usr/ugii/.ugii_env .

Step 3: Edit the .ugii_env file to redefine your mach/resource directory location.

- ☐ Edit the **.ugii_env** file from your home directory using the vi or other Unix editor.

- ☐ Find the following line in the **.ugii_env** file **UGII_CAM_BASE_RESOURCE=${UGII_CAM_BASE_DIR}resource/** and change the line to **UGII_CAM_BASE_RESOURCE=${Home}/resource/**.

- ☐ Save the changes and exit from editor.

Step 4: If necessary, change the Read-only protection on your just created local /mach/resource directory to rwed.

- ☐ Change the directory protection by typing the following command: **chmod 777 ${HOME}/mach/resource**.

Step 5: Restart NX.

- ☐ Exit NX and then restart NX.

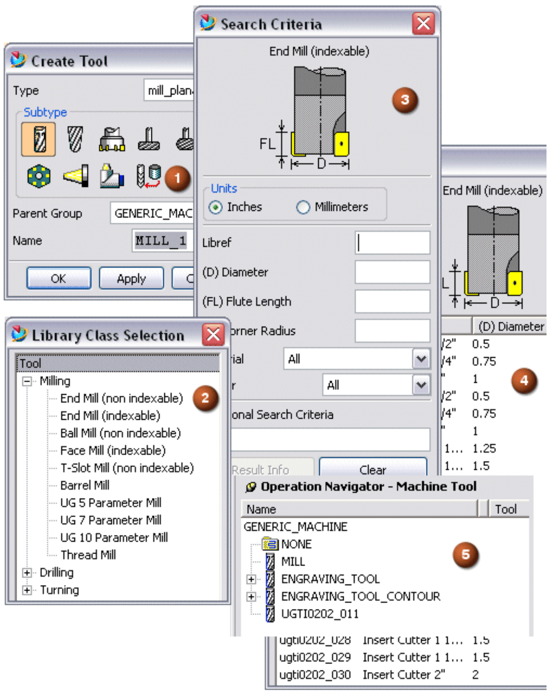
- ☐ On the menu bar, select **Help** → **NX Log File** to verify that your resource directory (based from your home directory) is being used.

 ©UGS Corp., All Rights Reserved

Cutting Tool Libraries

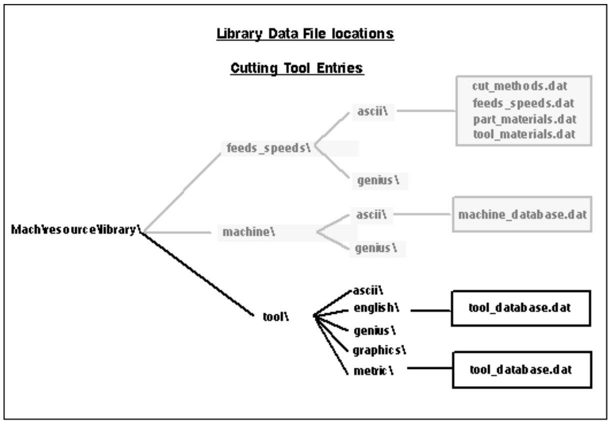
Cutting Tool Libraries contain information related to all cutting tools used in the generation of a tool path from an operation.

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) 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.

©UGS Corp., All Rights Reserved



This directory contains the following five subdirectories:

- ascii ——contains Definition and Event Handler files for ASCII text databases. These files are used for the Data Base Connection and usually are not modified by the user.

- english–contains the ASCII text database file (tool_database.dat) which contains all the data records used for English tool descriptions.

This file is edited by the user when adding or modifying tool data entries to the library.

- genius—contains the Definition and Event Handler files for Genius databases. These files are not modified by the user.

- graphics-contains part files of tool assemblies used for advanced replays with a solid tool. These assemblies are provided with the CAM release.

- metric—contains the ASCII text database file (tool_database.dat) which contains all the data records used for metric tool descriptions.

This file is edited by the user when adding or modifying tool data entries to the library.

The tool_database.dat files contains a list of tool data records that defines parameters used for tool definition. This is the only file that needs to be modified when you want to enter new tools. A data record consists of a record type and associated parameters. There are three record types represented in this file. They are:

- **#** which indicates a comment, the record is ignored.

- **FORMAT** which describes the type of parameters of subsequent DATA records.

 ©UGS Corp., All Rights Reserved

- **DATA** which consists of parameters which describe the tool.

For example:

FORMAT LIBRF T ST DESCR MATREF MATDES HLD HLDDES DIA FLEN FN HEI

describes the following data record:

DATA |ugt0201_001|02|01|End Mill|TMC0_00006|HSS-Co5-TiN | 320 | Steep Taper 20 | 10.5 | 35.3| 4| 55

Tool entries can be added to the Tool Libraries by two different methods. Data records, as described previously, can be added to the tool_database.dat file by simply editing the file or existing tools from current or legacy part files can be extracted by activating the Shop Documentation icon (Information → Documentation) and selecting Export Tool Library to ASCII datafile from the Available Templates list box of the Manufacturing Part Reports 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.

©UGS Corp., All Rights Reserved

Activity: Inserting Pre-existing Tools

Cutting tools with part files can be extracted and inserted into tool libraries. This procedure will work with the pre-V16 as well as NX part files. The following activity will take you through the process of inserting tools from a part file into a tool library.

Step 1: Open the pre-existing part file containing tool entries.

☐ Open the part file **ama_lib_tools**.

Step 2: Enter the Manufacturing Application.

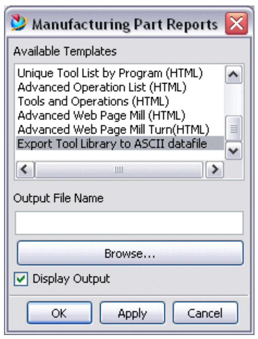
☐ Choose **Application** → **Manufacturing**.

The Operation Navigator and the Create Operation dialogs are displayed.

Step 3: Export the existing tool entries to an ASCII data file.

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

The Manufacturing Part Reports dialog is displayed.



☐ Select "**Export Tool Library to ASCII datafile**".

☐ Accept the default Output File Name.

Two files are created with a ".html" and ".dat" extension. The ".html" file is displayed in the information window when you select **OK**. The ".dat" file contains entries that you will use to insert into the **tool_database.dat** file.

☐ Choose **OK**.

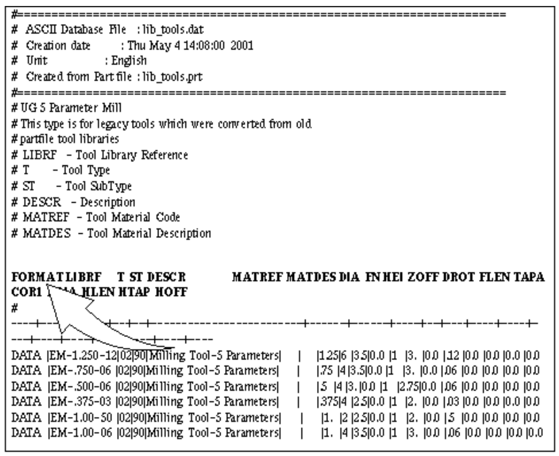
 ©UGS Corp., All Rights Reserved

Examine the listing window to see what tools have been exported to the data file. Note at the end of the listing window the location and name of the ".dat" file.

Step 4: Importing the tool data into the "tool_database.dat" file.

☐ Open the file, **tool_database.dat**, in your home **MACH\resource\library\tool\english** directory (Use the Notepad editor).

☐ Open the data file, **ama_lib_tools.dat**, created from **Step 3,** and scroll to the area that begins with **FORMAT LIBRF**.



```
#=====================================================================
#  ASCII Database File   : lib_tools.dat
#  Creation date         : Thu May 4 14:08:00 2001
#  Unit                  : English
#  Created from Part file : lib_tools.prt
#=====================================================================
# UG 5 Parameter Mill
# This type is for legacy tools which were converted from old
# partfile tool libraries
# LIBRF   - Tool Library Reference
# T       - Tool Type
# ST      - Tool SubType
# DESCR   - Description
# MATREF  - Tool Material Code
# MATDES  - Tool Material Description


FORMAT LIBRF    T ST DESCR           MATREF MATDES DIA  FN HEI ZOFF DROT FLEN TAPA
COR1 [illegible] HLEN HTAP HOFF
#
---+-----+-+-+-----------------------+-----+-----+----+--+---+----+---+
--+----+----+-----+----+---
DATA |EM-1.250-12|02|90|Milling Tool-5 Parameters|    |   |1.25|6 |3.5|0.0 |1  |3.  |0.0 |.12 |0.0 |0.0 |0.0 |0.0
DATA |EM-.750-06 |02|90|Milling Tool-5 Parameters|    |   |.75 |4 |3.5|0.0 |1  |3.  |0.0 |.06 |0.0 |0.0 |0.0 |0.0
DATA |EM-.500-06 |02|90|Milling Tool-5 Parameters|    |   |.5  |4 |3. |0.0 |1  |2.75|0.0 |.06 |0.0 |0.0 |0.0 |0.0
DATA |EM-.375-03 |02|90|Milling Tool-5 Parameters|    |   |.375|4 |2.5|0.0 |1  |2.  |0.0 |.03 |0.0 |0.0 |0.0 |0.0
DATA |EM-1.00-50 |02|90|Milling Tool-5 Parameters|    |   |1.  |2 |2.5|0.0 |1  |2.  |0.0 |.5  |0.0 |0.0 |0.0 |0.0
DATA |EM-1.00-06 |02|90|Milling Tool-5 Parameters|    |   |1.  |4 |3.5|0.0 |1  |3.  |0.0 |.06 |0.0 |0.0 |0.0 |0.0
```

☐ Select the lines beginning with **FORMAT LIBRF** and ending with **DATA |EM-1.00-06** (hold down **MB1** and drag the mouse through the lines).

☐ Select **MB3,** then **Copy**.

☐ Select the Notepad Window containing the contents of the file **tool_database.dat** (this file was previously opened).

☐ Select **MB3**, then **Paste** the contents into the file **tool_database.dat** (You can paste anywhere in the file, but it is suggested that you paste prior to the first **FORMAT** line).

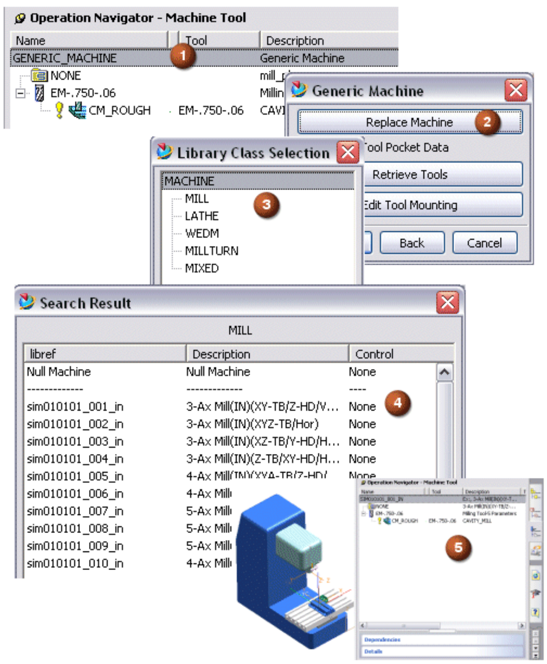
☐ Save the contents of the **tool_database.dat** file.

©UGS Corp., All Rights Reserved

Machine Tool Libraries

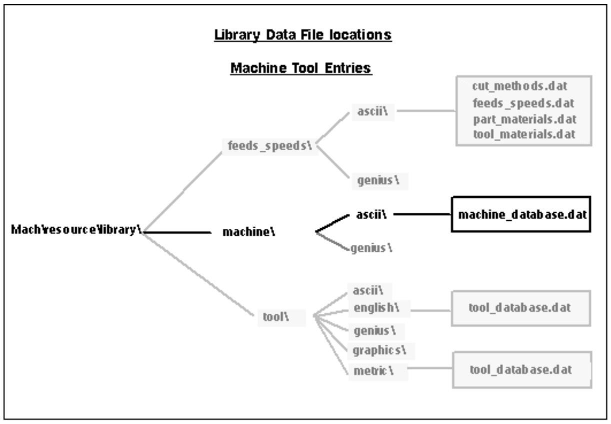
Machine Tool libraries contain information related to various machine tools configured for CAM. This information is used by the Integrated Simulation and Verification (IS&V) module used to simulate the execution of tool paths on a particular machine tool and is also used to add machine tool entries to the Postprocess dialog box (used to select a particular machine for post processing).

To access information from the Machine Tool libraries, from the Machine Tool view of the Operation Navigator, editing the Generic_Machine object (1), displays the Generic Machine Selection dialog. 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.

 ©UGS Corp., All Rights Reserved



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 post processor which is displayed on the Postprocess 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 post processor data file. Specifying the post processor 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
```

©UGS Corp., All Rights Reserved

Activity: Machine Tool Libraries

In this activity, you will become familiar with the procedures to access Machine Tool data from CAM Libraries. You will see how this library is used in conjunction with the Postprocess dialog by replacing the Available Machines with a 5–axis post processor.

Step 1: Open the part file.

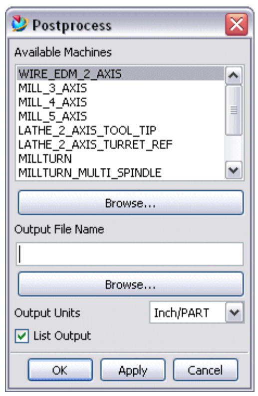
☐ Open the part file, **ama_lib_function** and if necessary, enter the Manufacturing application.

☐ Highlight the Program group object.

Step 2: Review available machines in the Post Process Dialog.

☐ Select the Post Process icon.

The Postprocess dialog is displayed.



☐ Choose **Cancel** or **OK**.

Step 3: Change the Postprocess Dialog to show a 5-axis machine tool only.

☐ If necessary change the Operation Navigator to **Machine Tool** view.

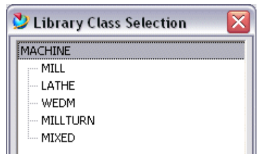
☐ Highlight the **GENERIC_MACHINE** object, using **MB3**, select **Edit**.

The Generic Machine dialog is displayed.

 ©UGS Corp., All Rights Reserved

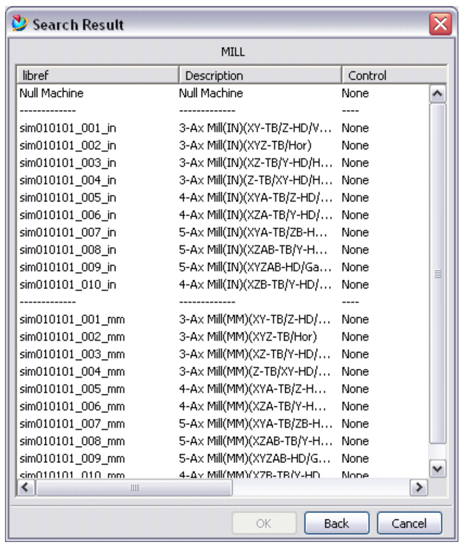
☐ Select the **Replace Machine** button.

The Library Class Selection Menu is displayed.



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

The Search Result listing is displayed.

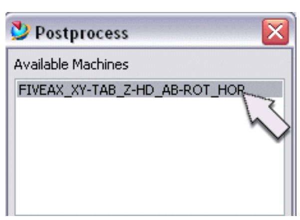


☐ Highlight **sim010101_008_in**, then select **OK** until you are back to the Operation Navigator.

☐ Select the Post Process icon.

The Postprocess dialog is displayed.

Notice that only the **FIVEAX_XY-TAB_Z_HD_AB-ROT_HOR** machine is displayed in the Postprocess dialog.

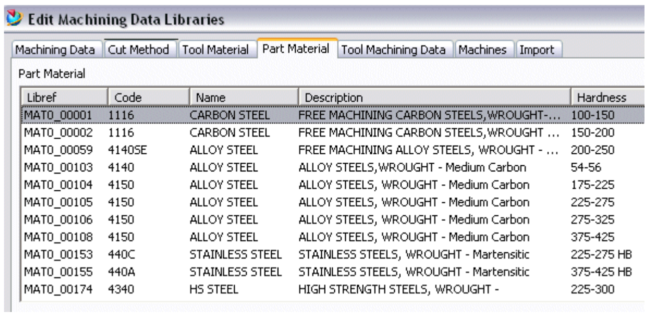


☐ Choose the **Cancel** button.

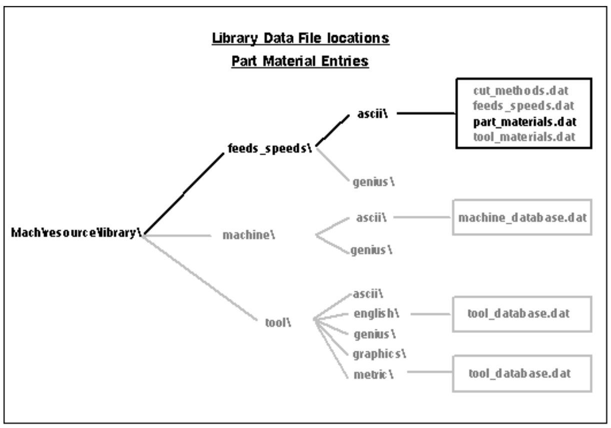
Part Material Libraries

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 modify, insert or remove Part Material entries, select from the menu bar Tools → Edit Machining Libraries and select the Part Material tab. Select the desired Part Material from the Part Material list and make any modifications necessary. New entries can be created by selecting the Insert button and typing data in the appropriate fields.



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.

 ©UGS Corp., All Rights Reserved

You can also add or modify Part Material information by editing the data records in the *part_materials.dat* file, using a text editor.

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 Resulfu-
rized|150-200|MAT0_00002
DATA|1116|CARBON STEEL|FREE MACHINING CARBON STEELS,WROUGHT- Low Carbon Resulfu-
rized|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 Resulfu-
rized|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
```

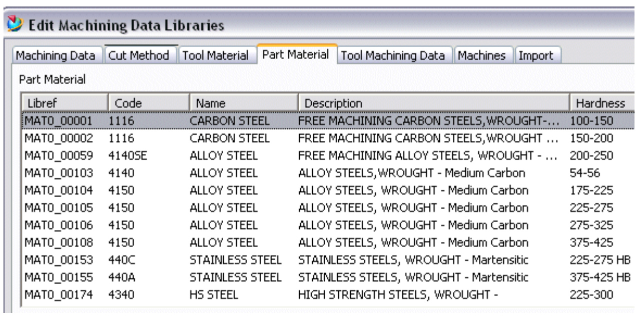
©UGS Corp., All Rights Reserved

Activity: Part Materials Library

In this activity, you will become familiar with accessing Part Material from the Library.

Step 1: Add a new part materials which will become available for selection.

- ☐ Continue with the opened part, **ama_lib_function**.
- ☐ If necessary, enter the **Manufacturing Application.**
- ☐ If necessary, Select **Tools** → **Edit Machining Data Libraries** from the menu bar.
- ☐ Select the **Part Materials** tab.



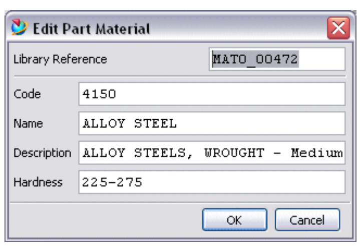
Libref	Code	Name	Description	Hardness
MAT0_00001	1116	CARBON STEEL	FREE MACHINING CARBON STEELS,WROUGHT-...	100-150
MAT0_00002	1116	CARBON STEEL	FREE MACHINING CARBON STEELS,WROUGHT ...	150-200
MAT0_00059	4140SE	ALLOY STEEL	FREE MACHINING ALLOY STEELS, WROUGHT - ...	200-250
MAT0_00103	4140	ALLOY STEEL	ALLOY STEELS,WROUGHT - Medium Carbon	54-56
MAT0_00104	4150	ALLOY STEEL	ALLOY STEELS, WROUGHT - Medium Carbon	175-225
MAT0_00105	4150	ALLOY STEEL	ALLOY STEELS, WROUGHT - Medium Carbon	225-275
MAT0_00106	4150	ALLOY STEEL	ALLOY STEELS, WROUGHT - Medium Carbon	275-325
MAT0_00108	4150	ALLOY STEEL	ALLOY STEELS, WROUGHT - Medium Carbon	375-425
MAT0_00153	440C	STAINLESS STEEL	STAINLESS STEELS, WROUGHT - Martensitic	225-275 HB
MAT0_00155	440A	STAINLESS STEEL	STAINLESS STEELS, WROUGHT - Martensitic	375-425 HB
MAT0_00174	4340	HS STEEL	HIGH STRENGTH STEELS, WROUGHT -	225-300

Step 2: Insert a new part material.

You will insert a new part material that is similar to the existing MAT0_00105, but has a slightly different hardness.

- ☐ Select **MAT0_00105** from the Edit Machining Data Libraries dialog.
- ☐ Choose the **Insert** button.

  The Edit Part Material dialog is displayed.



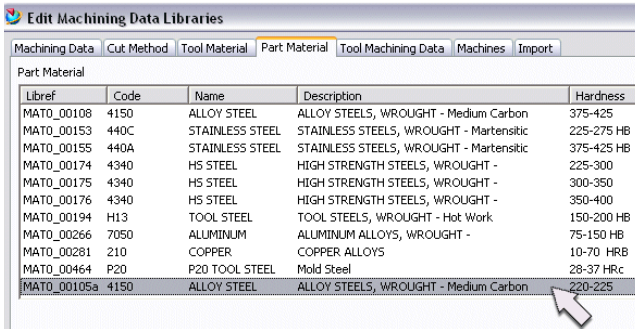
  You will change the Library Reference and the Hardness.

- ☐ Key **MAT0_00105A** in the Library Reference field.
- ☐ Key **220–225** in the Hardness field.

 ©UGS Corp., All Rights Reserved

☐ Choose **OK**.

☐ Scroll to the bottom of the list and note your new entry.



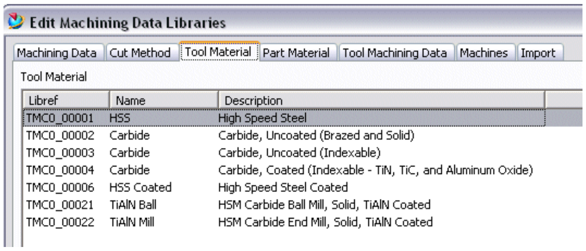
☐ Do not save or close the part.

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

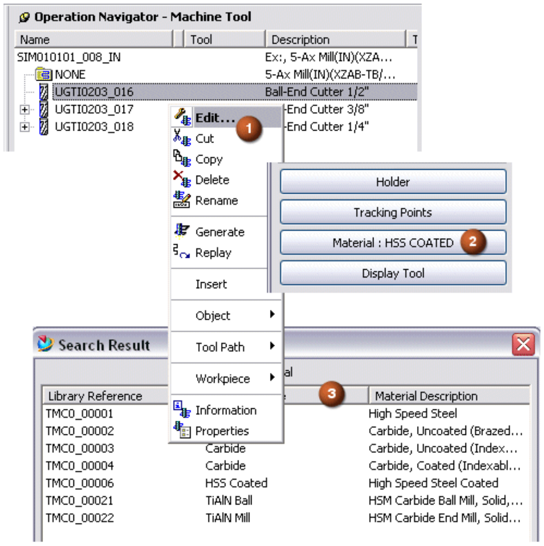
Cutting Tool Material Libraries

Cutting Tool Material libraries contains information which pertains to the cutting tool material type which is used in the calculation of feeds and speeds.

To modify, insert or remove Cutting Tool Material entries, select from the menu bar Tools → Edit Machining Libraries and select the Tool Material tab. Select the desired Tool Material from the Tool Material list and make any modifications necessary. New entries can be created by selecting the Insert button and typing data in the appropriate fields.

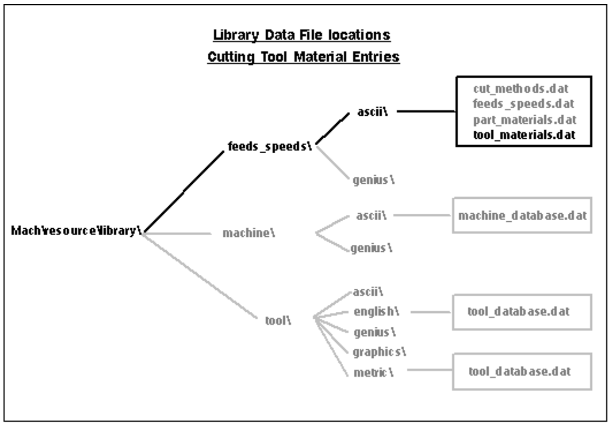


To access information from the Cutting Tool libraries, in the Machine Tool View of the Operation Navigator, editing any tool object (1) displays the Tool Parameter dialog. Selection of the Material: button (2) from this dialog, creates a query of the tool_materials.dat file with a listing displayed in the Search Results dialog. 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.

 ©UGS Corp., All Rights Reserved

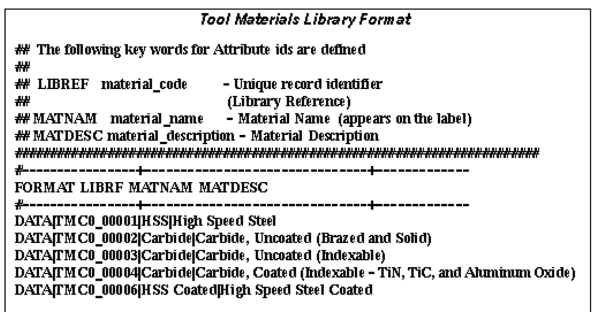


This directory contains the following two subdirectories:

- 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 also add or modify Cutting Tool Material information by editing the data records in the tool_materials.dat file by the use of a text editor. 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
```

©UGS Corp., All Rights Reserved

Activity: Cutting Tool Materials Libraries

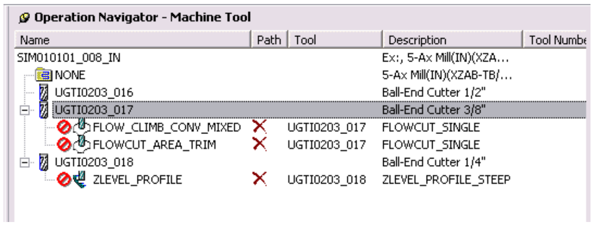
In this activity, you will become familiar with the Cutting Tool Materials data. Cutting Tool Materials are used in the calculation of feeds and speeds.

Step 1: Accessing the Cutting Tool Material library.

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

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

☐ If necessary, change the view of the Operation Navigator to the **Machine Tool** view.

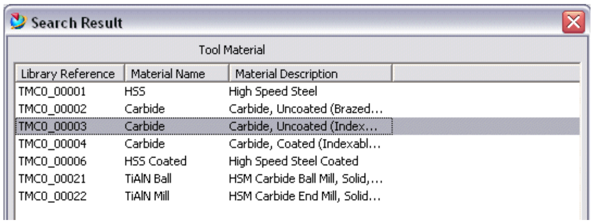


☐ Highlight **UGTI0203_017**, select **MB3**, then **Edit**.

The Cutting Tool Parameter List is displayed.

☐ Select the **Material** button.

The Search Result list is displayed.



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

☐ Choose **OK** until your are returned to the Operation Navigator.

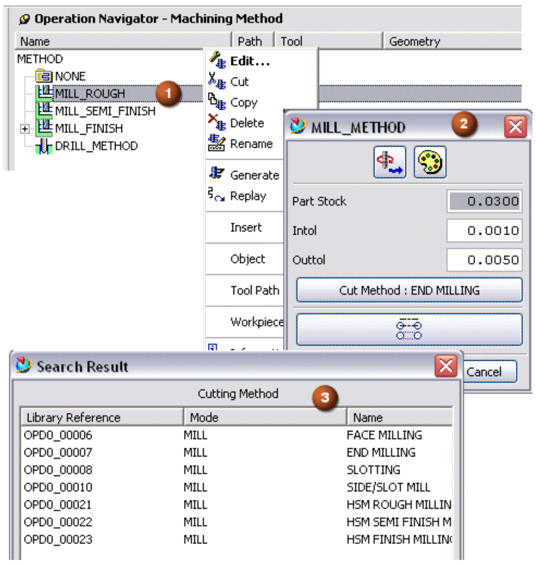
☐ Do not save or close the part file.

 ©UGS Corp., All Rights Reserved

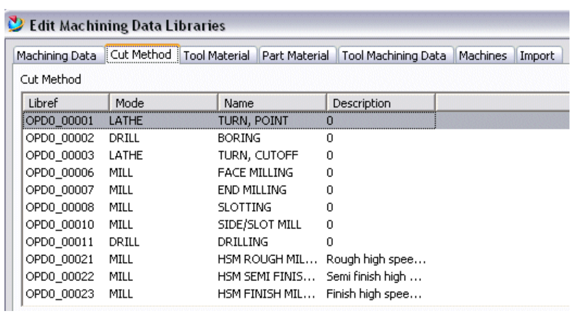
Cut Method Libraries

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).

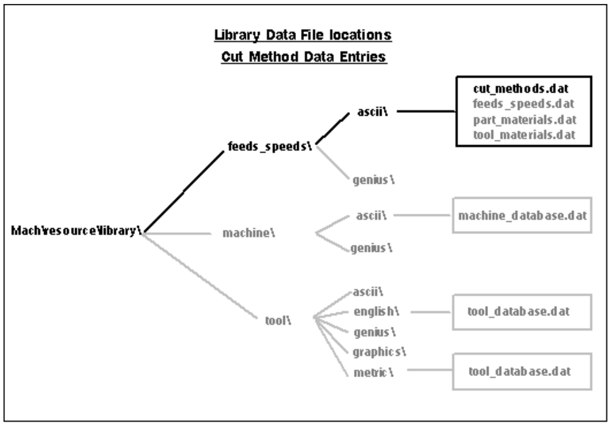


To modify, insert or remove Cut Method entries, select from the menu bar Tools → Edit Machining Libraries and select the Cut Method tab. Select the desired Cut Method from the list and make any modifications necessary. New entries can be created by selecting the Insert button and typing data in the appropriate fields.



Cut Method data is located in the **Mach\resource\library \feeds_speeds** directory.

©UGS Corp., All Rights Reserved



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 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
```

 ©UGS Corp., All Rights Reserved

Activity: Cut Method Libraries

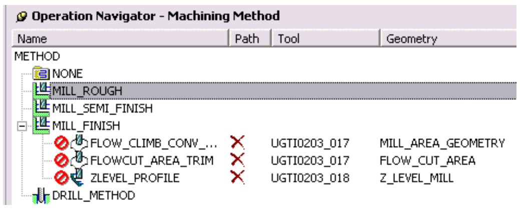
In this activity, you will become familiar with the Cut Methods Library.

Step 1: Accessing the Cut Method library.

☐ Continue with the opened part, **ama_lib_function**.

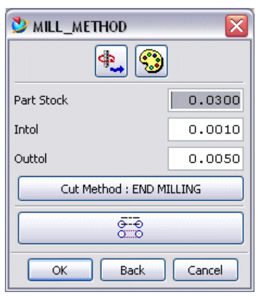
☐ If necessary, enter the **Manufacturing** application.

☐ If necessary, change the view of the Operation Navigator to the **Machining Method** view.



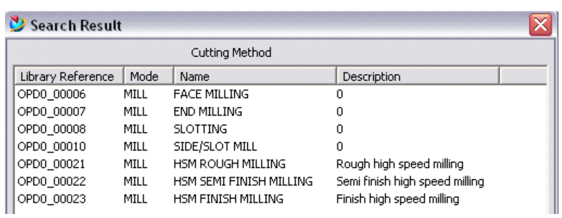
☐ Highlight **MILL_ROUGH**, select **MB3**, then **Edit**.

The Mill_Method dialog is displayed.



☐ Select the **Cut Method** button.

The Cutting Method Search List is displayed.



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

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

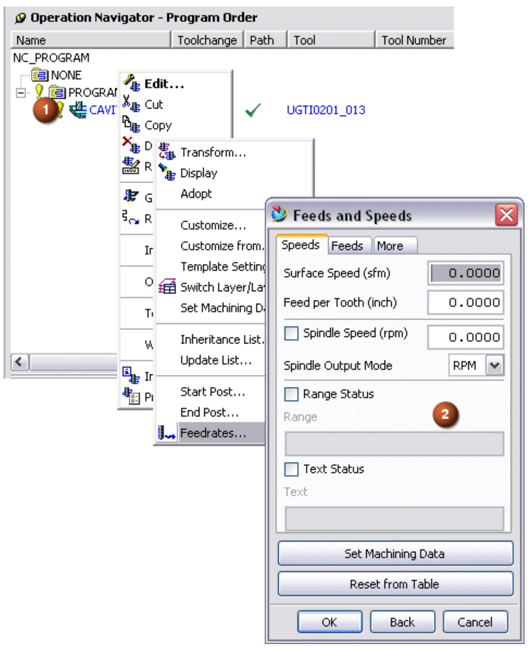
☐ Close the part.

©UGS Corp., All Rights Reserved

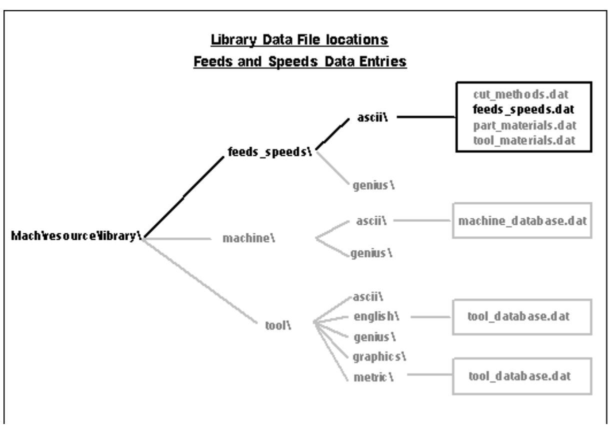
Feeds and Speeds Libraries

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 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:

 ©UGS Corp., All Rights Reserved

- ascii——contains Definition and Event Handler files for ASCII text databases. These files are associated with the Data Base Connection (DBC) and usually are not modified by the user. It also contains the feeds_speeds.dat file. This file is used for defining feeds and speeds in an operation.

- genius—contains the Definition and Event Handler files for Genius databases. These files are not modified by the user.

You can add or modify Feeds and Speeds information by editing the data records in the **feeds_speeds.dat** file. Data records consist of (LIBREF), Cut Method Library reference (OPERTYPE), Part Material Library reference (PARTMAT), Tool Material Library reference (TOOLMAT), Depth of Cut (DPT_CUT_IN or DPT_CUT_MM), Surface Speed (SURF_SPEED_FPM or SURF_SPEED_MPM) and Feed per Tooth (FEED_IPT or FEED_MMPT).

When adding entries for Feeds and Speeds, be sure that the Library Reference for the Part Material (located in part_materials.dat), Tool Material (located in tool_materials.dat) and Cut Method (located in cut_methods.dat) exist, are unique and of the correct type.

Be sure to reset the configuration to force the update of the files that have been modified.

Feeds and Speeds Library Format

```
# LIBRF        - Unique record identifier
#                (Library Reference)
# OPERTYPE     - cutmthd_libref            Cut Method Library Reference
# PARTMAT      - part_material_libref Part  Part Material Library Reference
# TOOLMAT      - tool_material_libref      Tool Material Library Reference
# DPT_CUT_IN   - dpth_of_cut               Depth_of_cut(inch)
# DPT_CUT_MM   - dpth_of_cut               Depth_of_cut(mm)
# SURF_SPEED_FPM  - surface_speed          Suface Speed(FPM)
# SURF_SPEED_MPM  - surface_speed          Suface Speed(MPM)
# FEED_IPT     - feed_per_tooth            Feed per Tooth(IPT)
# FEED_MMPT    - feed_per_tooth            Feed per Tooth(MMPT)
#-------------------------------------------------------------------------

FORMAT LIBRF OPERTYPE PARTMAT TOOLMAT DPT_CUT_IN DPT_CUT_MM IN-
DEX1 INDEX2 SURF_SPEED_FPM SURF_SPEED_MPM FEED_IPT FEED_MMPT

#-------------------------------------------------------------------------
DATA|FSDO_00001|OPD0_00001|MAT0_00001|TMC0_00001|.040|1.|||200.|60.8|.007|0.1778
DATA|FSDO_00002|OPD0_00001|MAT0_00001|TMC0_00001|.150|4.|||150.|45.6|.015|0.381
```

©UGS Corp., All Rights Reserved

Activity: Feeds and Speeds

In this activity, you will set the options necessary for system generated feeds and speeds. You will then change some of these settings to see how they affect feeds and speeds which are calculated by the system.

Step 1: Open the part file.

- ☐ Open the part file **ama_lib_act_feeds_speeds** and then rename it to *****_lib_act_feeds_speeds**.

- ☐ Enter the **Manufacturing** application.

  The Operation Navigator is displayed.

Step 2: Define the Part Material.

You only need to define the Part material once.

- ☐ Change the Operation Navigator to the **Geometry** view.

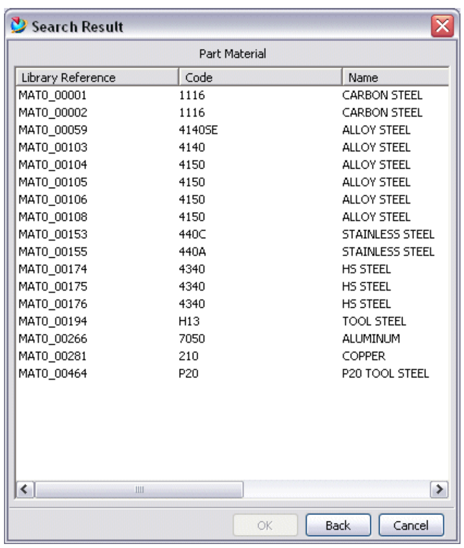
- ☐ Expand all of the Group objects.

- ☐ 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 **Aluminum** from the list and then choose **OK** twice to return you back to the Operation Navigator.

Step 3: Define the Cut Method.

 ©UGS Corp., All Rights Reserved

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

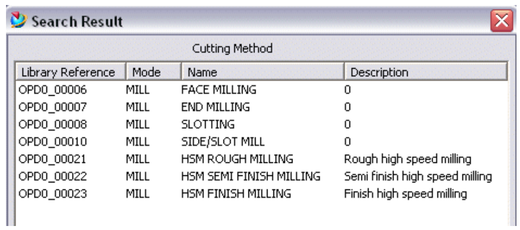
☐ Change the Operation Navigator to the **Method** view.

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

The Mill Method dialog is displayed.

☐ Choose the **Cut Method** button.

The Search Result dialog is displayed.



Search Result

Cutting Method

Library Reference	Mode	Name	Description
OPD0_00006	MILL	FACE MILLING	0
OPD0_00007	MILL	END MILLING	0
OPD0_00008	MILL	SLOTTING	0
OPD0_00010	MILL	SIDE/SLOT MILL	0
OPD0_00021	MILL	HSM ROUGH MILLING	Rough high speed milling
OPD0_00022	MILL	HSM SEMI FINISH MILLING	Semi finish high speed milling
OPD0_00023	MILL	HSM FINISH MILLING	Finish high speed milling

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

☐ Select **End Milling** and then choose **OK** until you return to the Operation Navigator.

Step 4: Define the Tool Material.

You can define the tool material when you first create a tool or when editing an existing tool.

In this case, you are going to edit an existing tool.

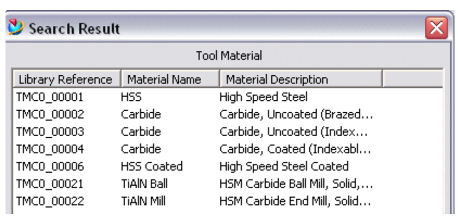
☐ Change the Operation Navigator to the **Tool** view.

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

The Milling Tool Parameters dialog is displayed. Note the lower portion of the dialog. This is where you define the Tool Material. Right now the material type is HSS COATED.

☐ Choose the **Material** button.

The available tool material types are displayed.



Search Result

Tool Material

Library Reference	Material Name	Material Description
TMC0_00001	HSS	High Speed Steel
TMC0_00002	Carbide	Carbide, Uncoated (Brazed...
TMC0_00003	Carbide	Carbide, Uncoated (Index...
TMC0_00004	Carbide	Carbide, Coated (Indexabl...
TMC0_00006	HSS Coated	High Speed Steel Coated
TMC0_00021	TiAlN Ball	HSM Carbide Ball Mill, Solid,...
TMC0_00022	TiAlN Mill	HSM Carbide End Mill, Solid...

☐ Select **TMCO_00002** Carbide on the list, then choose **OK** until you return to the Operation Navigator.

©UGS Corp., All Rights Reserved

Remember, you could have also changed the Material type from within the operation by editing the tool description.

Step 5: Define the Cut Depth.

This option is set from within the operation and is used in the calculation of feeds and speeds.

You are now going to edit an existing operation.

☐ Change the Operation Navigator to the **Program** view.

☐ Expand all objects.

☐ Highlight the operation named **CAVITY_MILL**. then **MB3, Edit**.

The Cavity_Mill dialog is displayed.

☐ Change the **Global Depth Per Cut** to **.375**.

☐ Select the **Cut Levels** button under Control Geometry.

The Cut Levels dialog is displayed.

☐ Change the **Local Depth Per Cut** to **.375**.

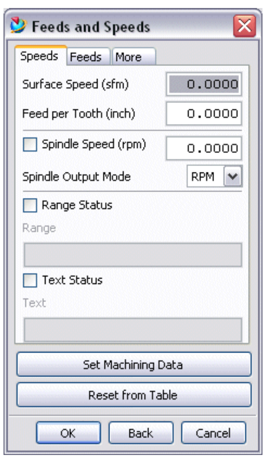
☐ Choose **OK** until the Operation Navigator is displayed.

Step 6: Set the Feeds and Speeds.

☐ Highlight the **CAVITY_MILL** operation.

☐ Use MB3 and choose **Object** → **Feedrates**.

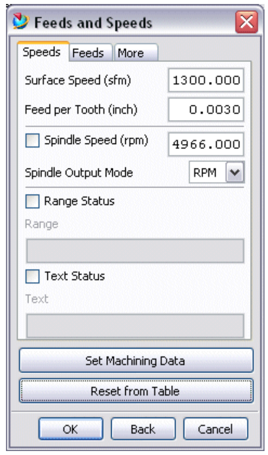
The Feeds and Speeds dialog is displayed.



☐ Choose the **Reset from Table** button.

 ©UGS Corp., All Rights Reserved

The feeds and speeds parameters are calculated and displayed in the value fields.



☐ Choose **OK** to return to the Operation Navigator.

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

Step 7: Changing the tool material and adjusting the Feeds and Speeds.

You are going to change the tool material then recalculate the Feeds and Speeds for the operation.

☐ Change the Operation Navigator to the **Machine Tool** view.

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

☐ Choose the **Material** button.

The Search Result dialog is displayed.

☐ Select **TMCO_00001** HSS on the list, then choose **OK** until you return to the Operation Navigator.

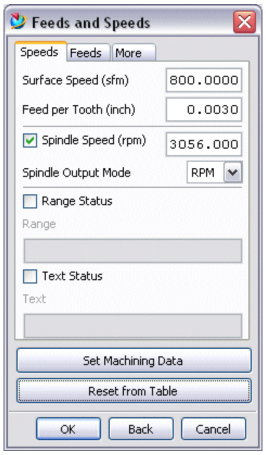
☐ Change the Operation Navigator to the **Program View**.

☐ Highlight the **Cavity_Mill** operation.

☐ Choose **Object** → **Feedrates**.

©UGS Corp., All Rights Reserved

☐ Choose **Reset from Table**.



Notice the change in the speeds and feeds

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

 ©UGS Corp., All Rights Reserved

Summary

Libraries are used for numerous applications in the Manufacturing application. Libraries are convenient and easy tools that can be used to access reference data with respect to cutting tool, machine tool, part material, cutting tool material, cut method and feeds and speeds.

In this lesson you:

- Reviewed Cutting Tool Libraries.
- Inserted pre-existing cutting tools into libraries.
- Reviewed the Machine Tool, Part Material, Cutting Tool Material and Cut Method Libraries.
- Changed various option settings to show their effect on feeds and speeds.

©UGS Corp., All Rights Reserved

Lesson

4 *Machining Faceted Geometry*

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 wire frame or solid geometric model.

Objective

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

- Import an STL file into NX.

- Generate tool paths on faceted geometry.

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. This example is an 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 NX 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.

©UGS Corp., All Rights Reserved

Activity: Machining of Faceted Geometry

Step 1: Create a metric NX 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.

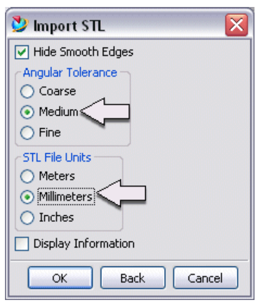
☐ Choose **OK**.

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**.

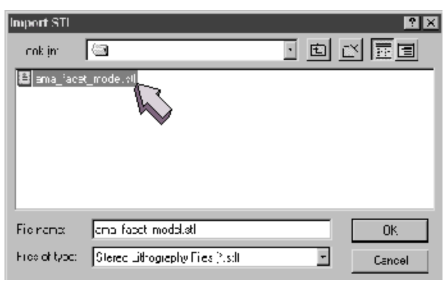
The **Import STL** dialog is displayed.



☐ 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**.

 ©UGS Corp., All Rights Reserved

The file is imported into the NX "base" file which you previously created.

☐ Change the display to solid and fit the view to the screen.

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

☐ Enter the **Manufacturing** application.

The **Machining Environment** dialog is displayed.

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

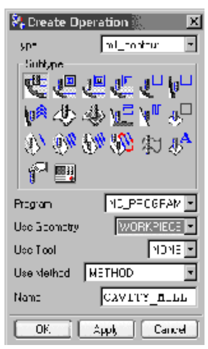
☐ Choose the **Operation Navigator** tab from the toolbar.

The Operation Navigator is displayed.

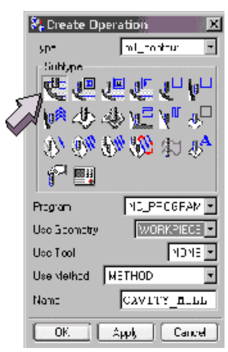
☐ If necessary, change the view of the Operation Navigator to the **Program Order View**.

☐ Select **Create Operation** from the **CAM Create** toolbar.

The **Create Operation** dialog is displayed.



☐ Select the **Cavity_Mill** subtype.

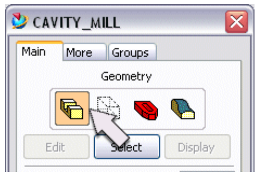


©UGS Corp., All Rights Reserved

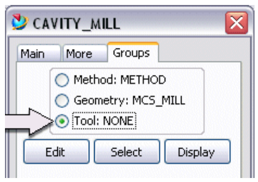
☐ Select **OK**.

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.

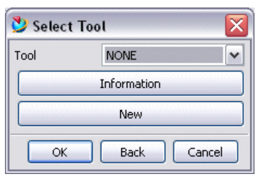
☐ Choose the Groups tab from the CAVITY_MILL dialog.



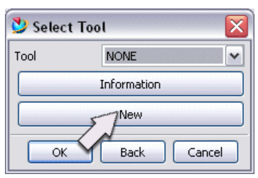
☐ If necessary, choose the Groups tab then choose**Tool: NONE** and then the **Select** button.



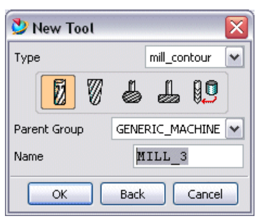
The **Select Tool** dialog is displayed.



☐ Choose the **New** button.

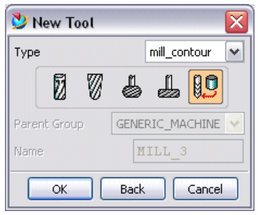


The **New Tool** dialog is displayed.

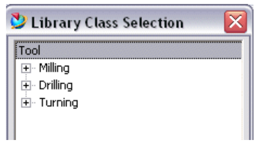


 ©UGS Corp., All Rights Reserved

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

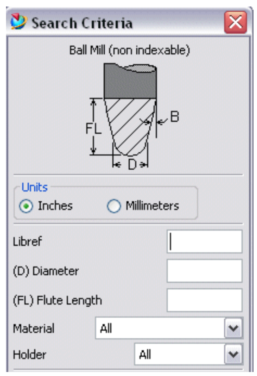


The **Library Class Selection** dialog is displayed.

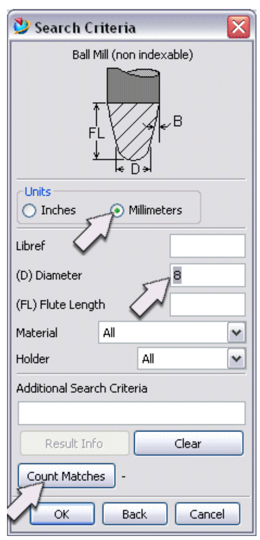


☐ Expand the **Milling** group object and then double-click on the **Ball Mill** (non indexable) object.

The Search Criteria dialog is displayed.



☐ If necessary, choose the Millimeters radio button, key **8** in the **Diameter** field and then choose the Count Matches button.



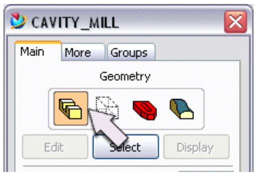
©UGS Corp., All Rights Reserved

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

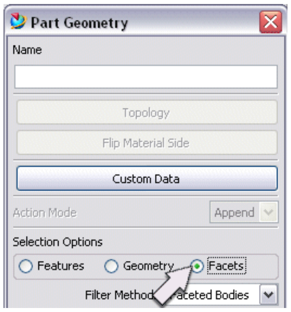
- ☐ Choose **OK** from the Search Criteria dialog.

  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 the **Facets** radio button 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.

 ©UGS Corp., All Rights Reserved

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 NX, creating a faceted model.

- Directly machined a faceted model using the Cavity Milling operation type.

Lesson

5 High Speed Machining

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.

Objective

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

- Generate High Speed Machining operations.
- Generate Nurbs output.

©UGS Corp., All Rights Reserved

High Speed Machining- An Overview

The concept of High Speed Machining (HSM) is not new. The original concept was conceived by Dr. Carl Salomon in 1924 and was patented 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 step overs 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 (or higher) 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. Reduced cutter stress 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

 ©UGS Corp., All Rights Reserved

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 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 longer because of shallow depths of cut and fine step overs.

©UGS Corp., All Rights Reserved

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 step over 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 work to finish the part (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 part deflection.
- 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.

 ©UGS Corp., All Rights Reserved

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 work piece 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 step over values set to .00005" or less.

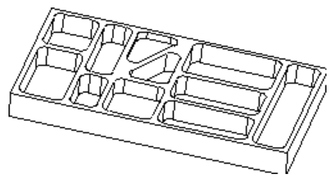
©UGS Corp., All Rights Reserved

Activity: 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 the part file and review an existing operation.

☐ Open the part file **ama_hsm_1**.



☐ Save as ******_hsm_1***, 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.

☐ Highlight 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 operation and modify parameters that are applicable for High Speed Machining.

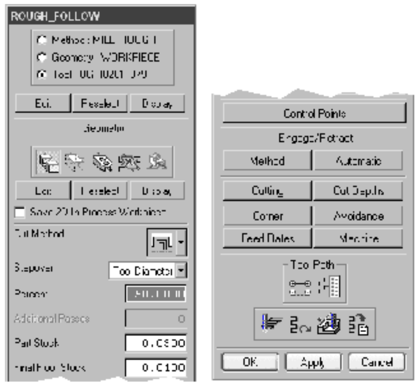
☐ Cancel the Tool Path visualization dialog.

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

☐ Double-click on the operation, **pocket_standard**.

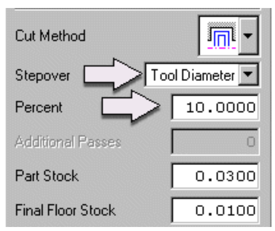
 ©UGS Corp., All Rights Reserved

The ROUGH_FOLLOW dialog is displayed.



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

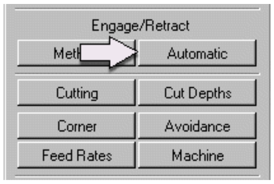
☐ From the **Stepover** pull-down menu, select **Tool Diameter** and then change **Percent** to **10**. This will minimize the step over value to 10% of the effective cutter diameter and will help maintain constant tool loading conditions.



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

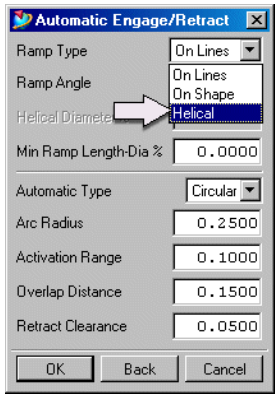
©UGS Corp., All Rights Reserved

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



The Automatic Engage/Retract dialog is displayed.

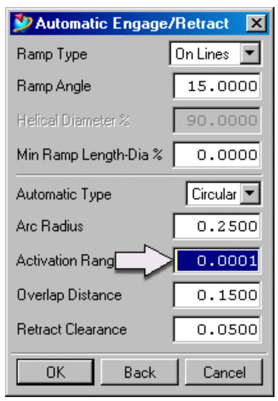
☐ Select the **Ramp Type** pull-down menu and then select **Helical**.



Changing the **Ramp Type** to **Helical** insures 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** and choose **OK**.



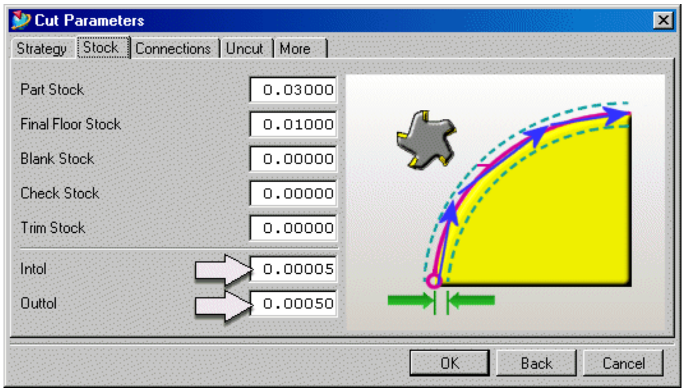
Changing the **Activation Range** to this small value insures that the cutter will not engage the part as it approaches the walls.

☐ Select the **Cutting** button.

 ©UGS Corp., All Rights Reserved

☐ Select the **Stock** tab.

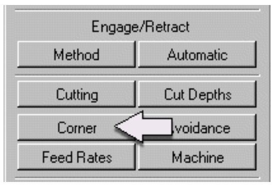
☐ Change the **Intol** parameter to **.00005** and the **Outtol** parameter to **.0005**.



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.

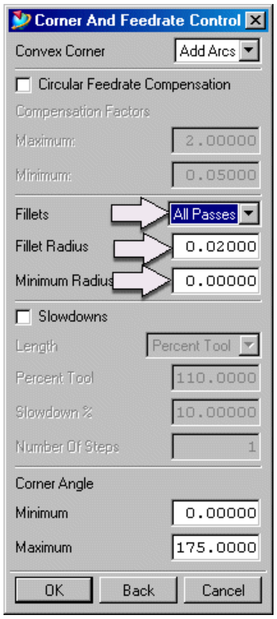
☐ Choose **OK**.

☐ Select **Corner**.

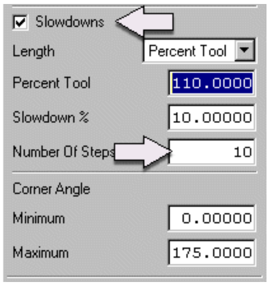


This option will add a fillet at all corners (corner roll) which eliminates sharp and sudden moves.

☐ 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.

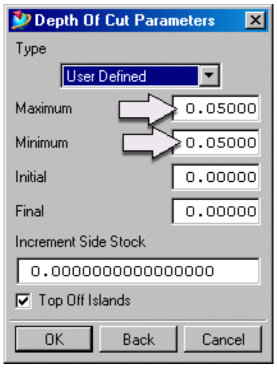
☐ Choose **OK**.

☐ Select **Cut Depths**.

The Depth of Cut Parameters dialog is displayed.

 ©UGS Corp., All Rights Reserved

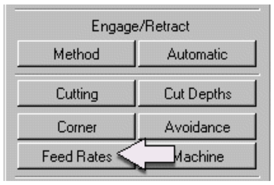
☐ Change the **Maximum** and **Minimum** values to **.050**.



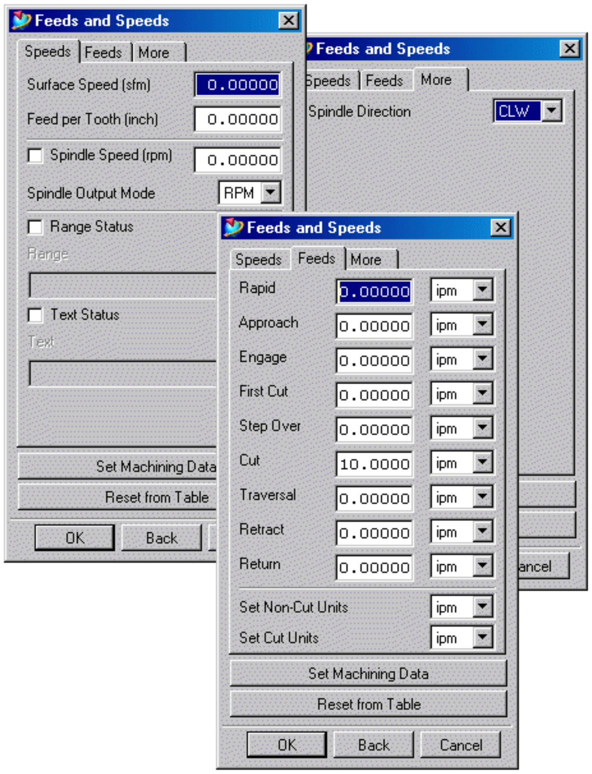
Reducing the depth of cut while increasing feed rates is a core process change of High Speed Machining. More material can be removed in less time. An advantage of a shallower depth of cut is less horsepower required to cut through the material.

☐ Choose **OK**.

☐ Select **Feed Rates**.

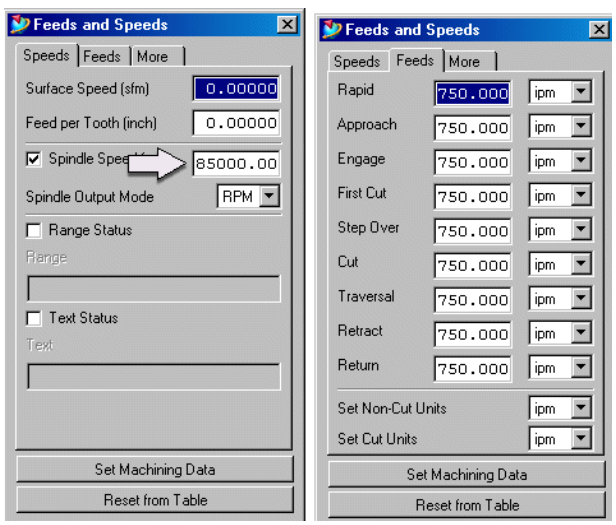


The Feeds and Speeds dialog is displayed.



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.

 ©UGS Corp., All Rights Reserved

- ☐ 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.

©UGS Corp., All Rights Reserved

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.

 ©UGS Corp., All Rights Reserved

Activity: 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.

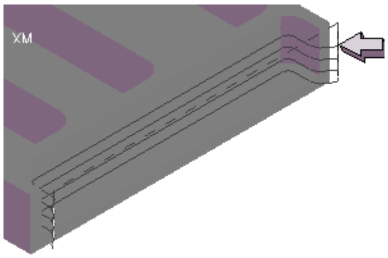
Step 1: Edit the existing operation and modify parameters so that the tool is in constant contact with the part.

☐ Continue with the part file *****_hsm_1**.

☐ Double-click on the operation, **zlevel_profile**.

The ZLEVEL_PROFILE dialog is displayed.

☐ **Replay** the tool path (you may want to slow down the display speed).



Notice that the tool retracts and engages for each cut level.

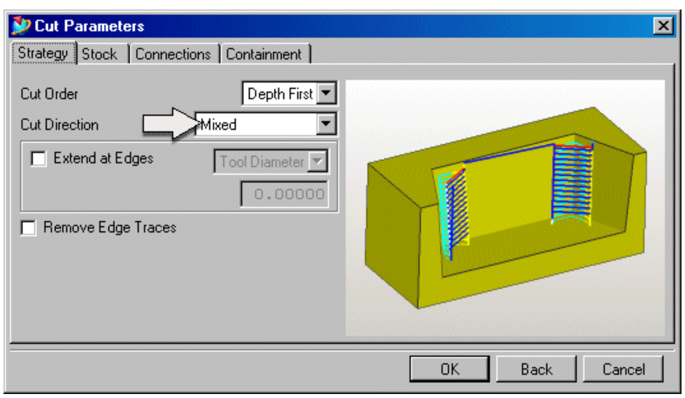
You will now change the cut parameters to allow the cutter to move directly from one cut level to the next without engaging, traversing and retracting.

☐ **Refresh** the graphics display.

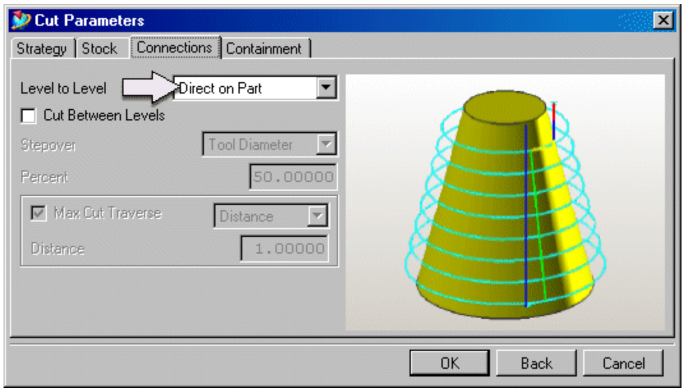
☐ Choose the **Cutting** button.

☐ If necessary choose the **Strategy** tab.

☐ Set the **Cut Direction** option to **Mixed**.



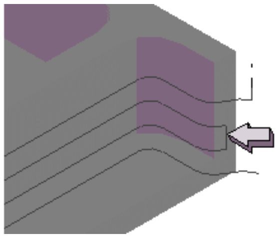
☐ Choose the **Connections** tab and change the **Level to Level** option to **Direct on Part**.



☐ 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.

 ©UGS Corp., All Rights Reserved

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.

©UGS Corp., All Rights Reserved

Activity: NURBS

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

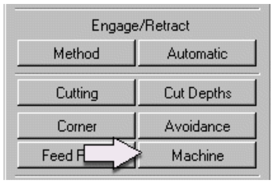
Step 1: Edit the existing operation and modify parameters suitable for the output of NURBS data.

☐ Continue with the part file *****_hsm_1**.

☐ Double-click on the operation, **pocket_standard**.

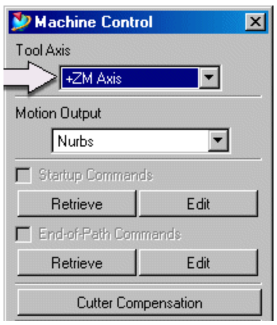
The ROUGH_FOLLOW dialog is displayed.

☐ Select the **Machine** button.

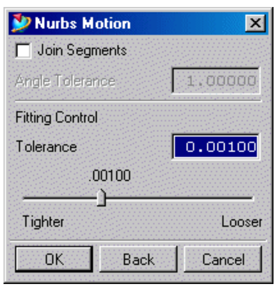


The Machine Control dialog is displayed.

☐ 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 Nurb 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

 ©UGS Corp., All Rights Reserved

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 NURB. A tighter tolerance (defined by a smaller value) causes the tool path to follow the NURB 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 NURB 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 NURB 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 degrees or less.

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.

©UGS Corp., All Rights Reserved

☐ List the tool path output.

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

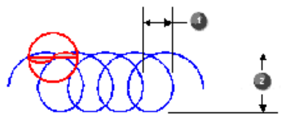
☐ **Close** the part file.

 ©UGS Corp., All Rights Reserved

Trochoidal Cut Pattern

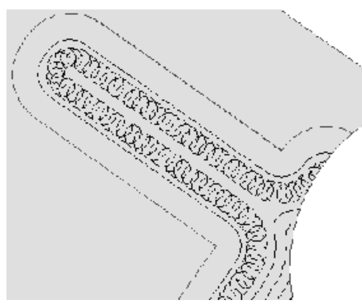
The Trochoidal Cut pattern is used when there is a need to limit excessive step over to prevent tool breakage when the tool is fully embedded into a cut and you want to avoid cutting excessive material. Most cut patterns generate embedded regions between islands and parts during the engage process as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

©UGS Corp., All Rights Reserved

Activity: 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.

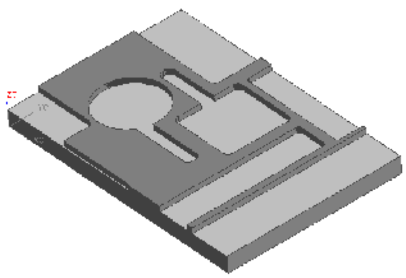
Step 1: Opening the part file ama_trochoidal_cut_method.

☐ If necessary, start NX.

☐ Use **File→Open**. 

☐ Navigate to your parts folder and open the file.

☐ Briefly examine the part.

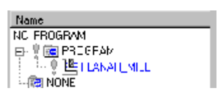


Step 2: If necessary, enter the Manufacturing application and display the Operation Navigator.

☐ 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.

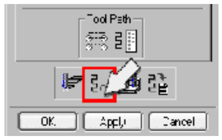
Step 3: Replay and then modify the PLANAR_MILL operation to utilize the Trochoidal Cut method.

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

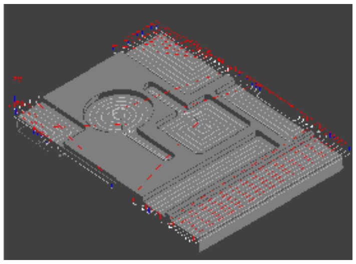
The PLANAR_MILL dialog is displayed.

 ©UGS Corp., All Rights Reserved

☐ Choose the Replay icon.

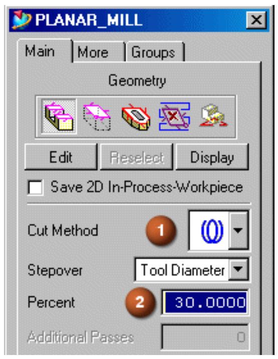


The tool path is displayed.



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 **Percent (2)** to **30**.



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

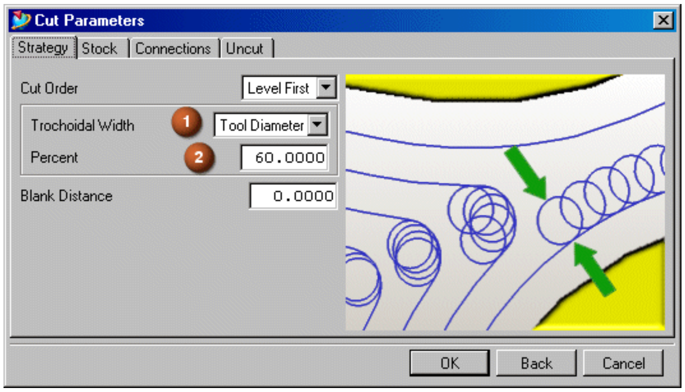
☐ Choose **OK** on the **Reselect Tool** dialog.

☐ Under the **Main** tab, select the **Cutting** button.

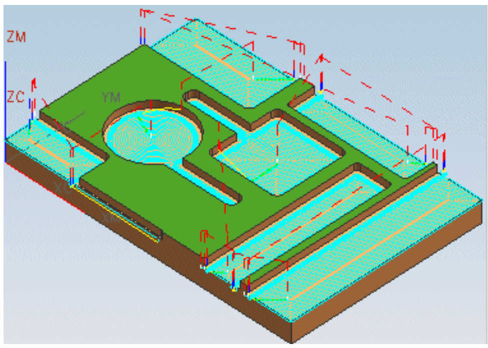
☐ From the **Cut Parameters** dialog, if necessary, choose the **Strategy** tab.

©UGS Corp., All Rights Reserved

☐ If necessary change **Trochoidal Width (1)** to **Tool Diameter**, **Percent (2)** to **60**.



☐ **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.

 ©UGS Corp., All Rights Reserved

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.

©UGS Corp., All Rights Reserved

Lesson

6 *NC Assistant*

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.

Objective

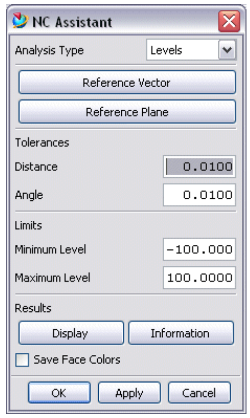
Upon completion of this lesson, you will be able to:

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

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. 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.

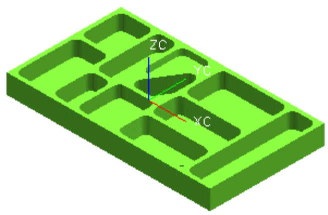
 ©UGS Corp., All Rights Reserved

Activity: 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**.

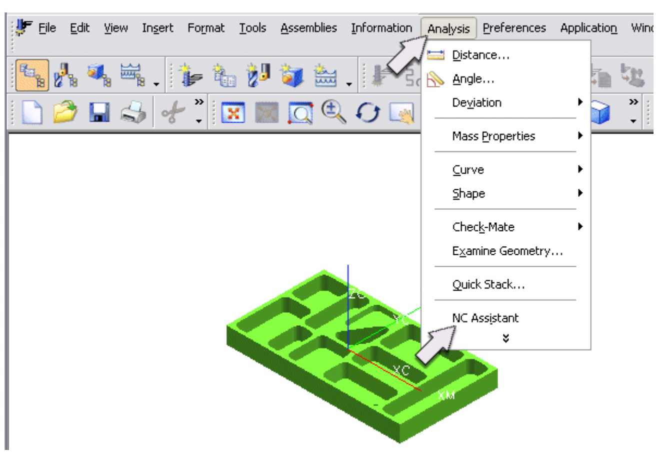


☐ Enter the **Manufacturing** application.

Step 2: Activate the NC Assistant.

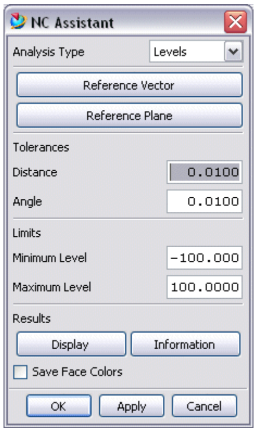
You will need to determine the tool configuration(s) necessary to finish machine all pockets in this part. Visually, it is difficult to detect draft, if any, on the pocket walls. It would be cumbersome to verify all corner radii and floor depths. To make your job easier, you will use the NC Assistant, to analyze the geometry configuration. You will now activate the NC Assistant.

☐ From the Main Menu Bar select, **Analysis** → **NC Assistant**.



©UGS Corp., All Rights Reserved

The **NC Assistant** dialog is displayed.



Step 3: Use the NC Assistant to determine cutter length.

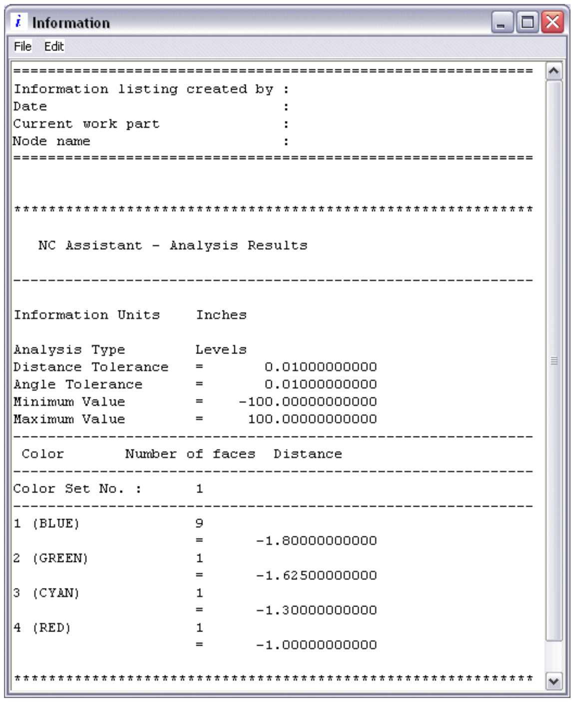
The first item for consideration is to determine the length of the cutter that will be needed. For this determination you will use the NC Assistant to determine the various **Levels**. If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will accept 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.

 ©UGS Corp., All Rights Reserved

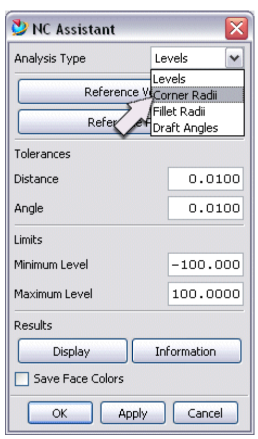


Notice the color set number, the colors associated with the color set number, number of faces and distance. Compare the colors with those now displayed in the graphics window. By examining the distance values, the deepest level or floor is located .800 below the top of the part (blue). Therefore the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

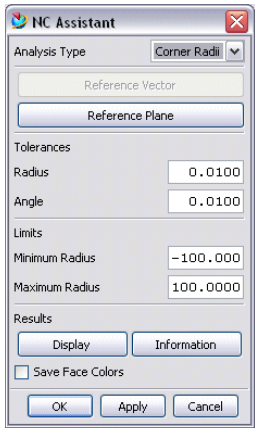
The second item for consideration is to determine the diameter of the cutter that will be required. For this determination you will use the NC Assistant to determine the various corner radii of the part.

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



©UGS Corp., All Rights Reserved

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.

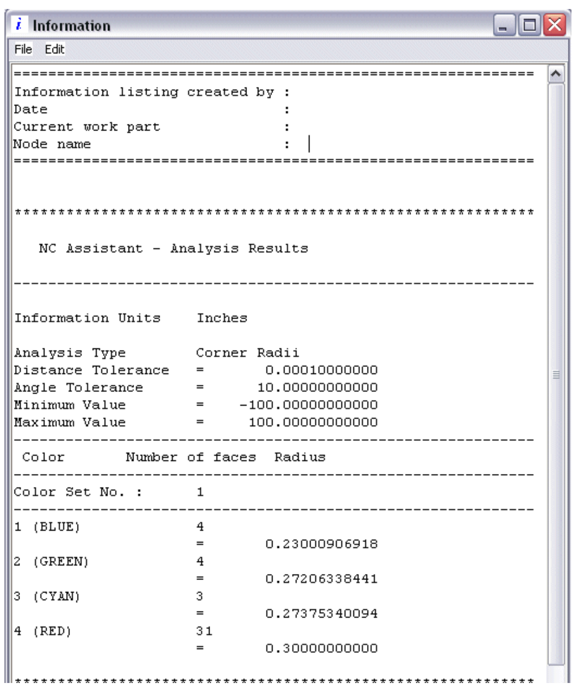
☐ Change the **Radius** value to **.0001**.

☐ 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.



 ©UGS Corp., All Rights Reserved

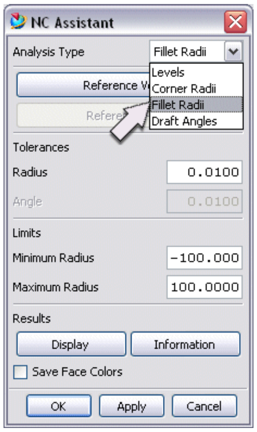
Notice the color set number, the colors associated with the color set number, number of faces and radius. Compare the colors with those now displayed in the graphics window. By examining the corner radii values, the largest is .300 (red), the smallest .230, the closest standard size end mill required would be .4375 inches.

©UGS Corp., All Rights Reserved

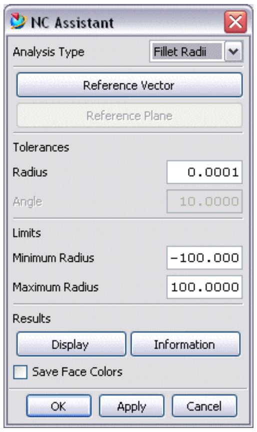
Step 5: Use the NC Assistant to determine cutter corner radius.

The third item for consideration is to determine the corner radius of the cutter that will be needed. For this determination you will use the **NC Assistant** to determine the various fillet radii of the part.

☐ Choose **Fillet Radii** from the **Analysis Type** pull-down menu.



The dialog for **Fillet Radii** analysis is displayed.



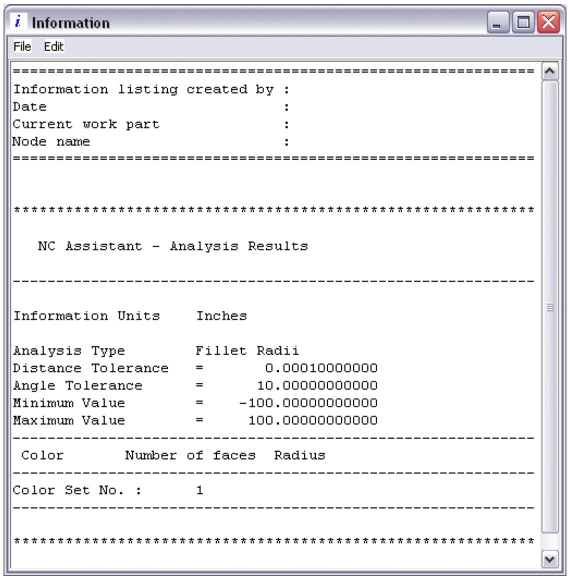
If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will accept all defaults and select the entire part for faces.

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

☐ Choose **Apply**.

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

 ©UGS Corp., All Rights Reserved

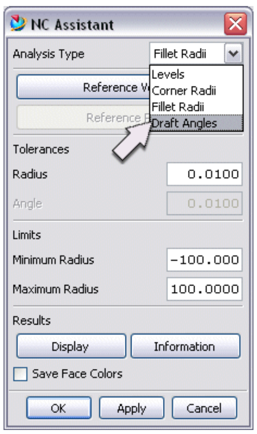


Notice that there were no Fillet Radii displayed.

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

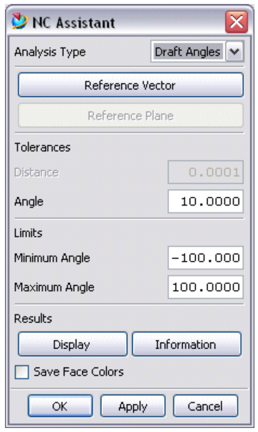
The fourth item for consideration is to determine any draft angles on the part that are machinable through the use of an angle cutter. For this determination you will use the **NC Assistant** to determine the draft angles.

☐ Choose **Draft Angles** from the **Analysis Type** pull-down menu.



The dialog for **Draft Angles** analysis type is displayed.

©UGS Corp., All Rights Reserved



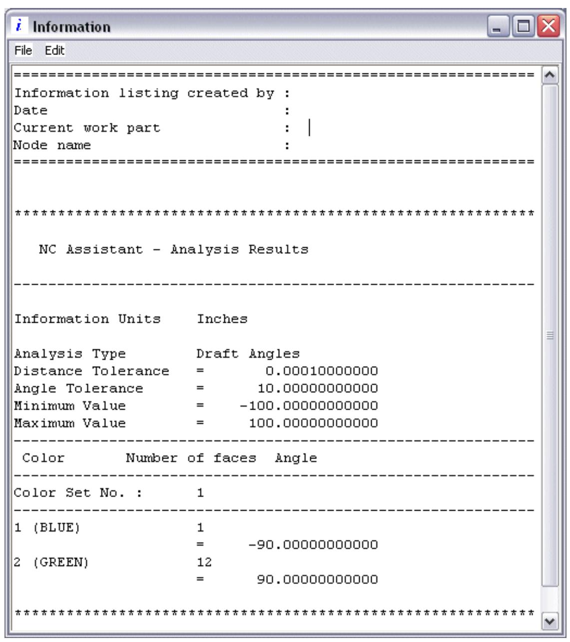
If you now look at the cue line, you will see that you are being asked to set parameters or select faces. You will change the **Angle** tolerance to 1.0 degree, accept all other defaults and select the entire part for faces.

☐ 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.



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

 ©UGS Corp., All Rights Reserved

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.

©UGS Corp., All Rights Reserved

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.

 ©UGS Corp., All Rights Reserved

Lesson

7 *Templates*

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.

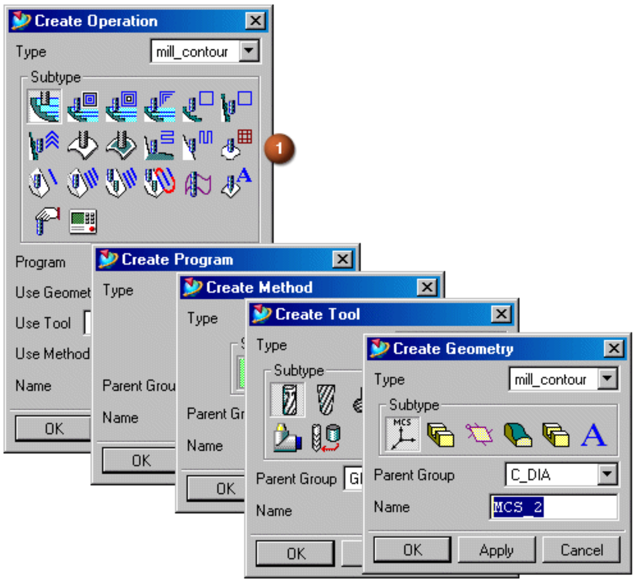
Objective

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.

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.



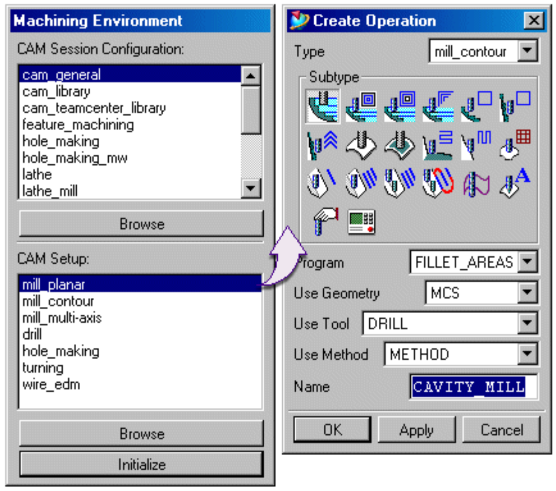
Templates are represented as Subtype (1) icons which are available in the various Create dialogs (also referred to as object dialogs). They define operation parameters including numerical values such as part Intol/Outol, feed rates, etc. and can display customized dialogs.

In addition, templates determine the operations and groups that initially appear in the Operation Navigation Tool when you first enter the Manufacturing application.

 ©UGS Corp., All Rights Reserved

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.

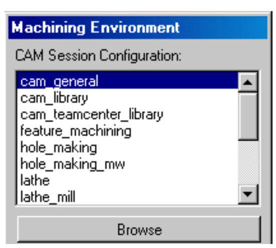


In the above example, mill_contour is a template part file containing predefined operations and groups which are represented by Subtype icons.

Template Sets

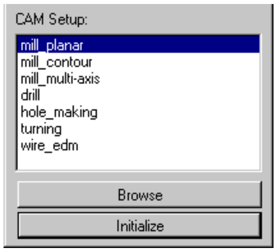
A Template Set is a collection of template part files. They are specified from within the Configuration files. Basic Template Sets are provided and maintained in each software release and/or may be created, customized and maintained by users. Template Sets created by users may use any valid naming convention, 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 window lists files with ".dat" extensions containing the location of template files for operations, documentation, post processors and libraries.



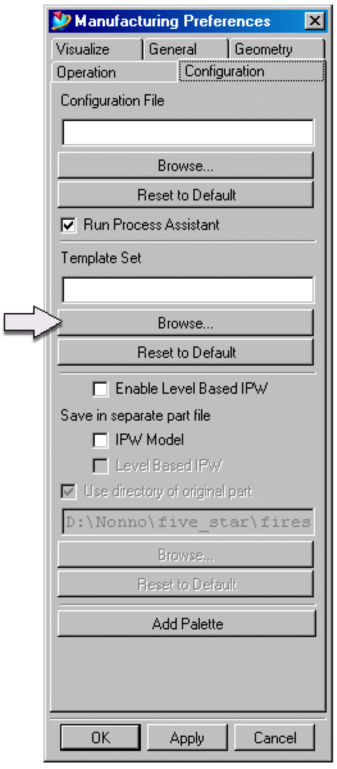
These files are located in the *\mach\resource\configuration* directory.

The CAM Setup 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.

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

 ©UGS Corp., All Rights Reserved

icon in the Create Program, Create Tool, Create Geometry or Create Method group dialog.

Creating and Using Template Sets

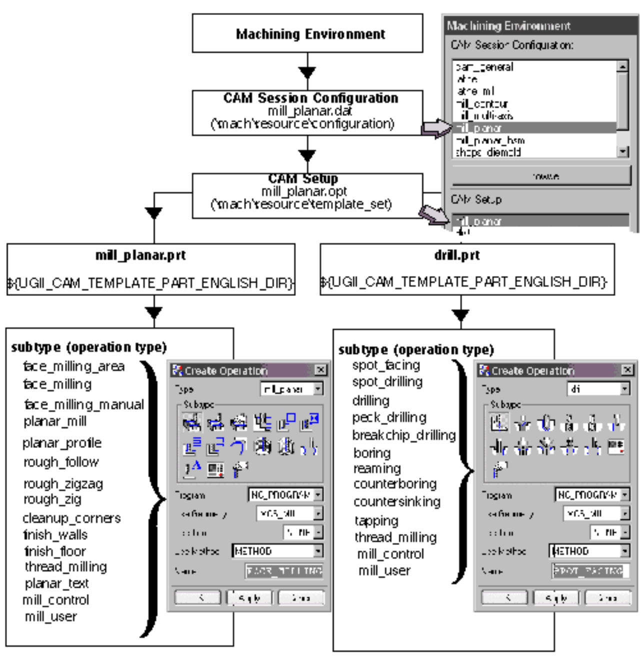
Template Sets make template part files available for selection from within dialogs. This prevents you from having to browse for individual part files. A Template Set is a text file (.opt extension) listing the part files which contain the templates that are to be utilized. The default Template Set is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

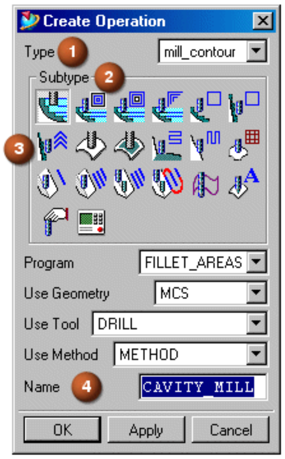


The Type option (1) corresponds to the template file name.

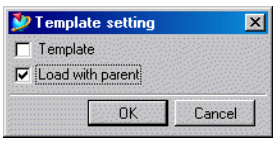
The Subtype (2) option corresponds to the operation name within the template file.

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

The Name field (4) indicates the name of the operation or group object created.



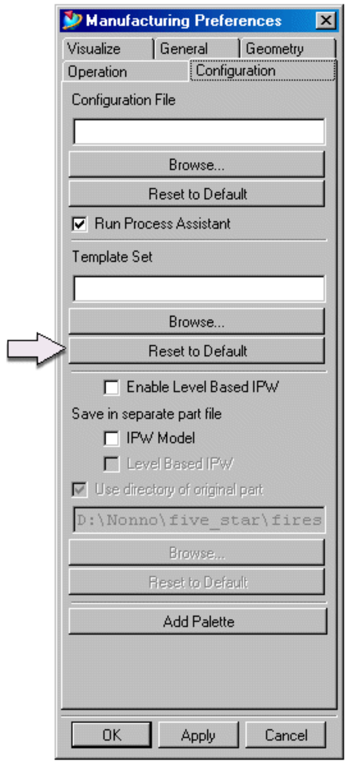
The creation of tools within an operation will occur only if the template has Load with Parent set to on. If the template loads a tool with the same name but different parameters as a tool in the current part file, a new tool with a different name is created with the new operation.



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

 ©UGS Corp., All Rights Reserved

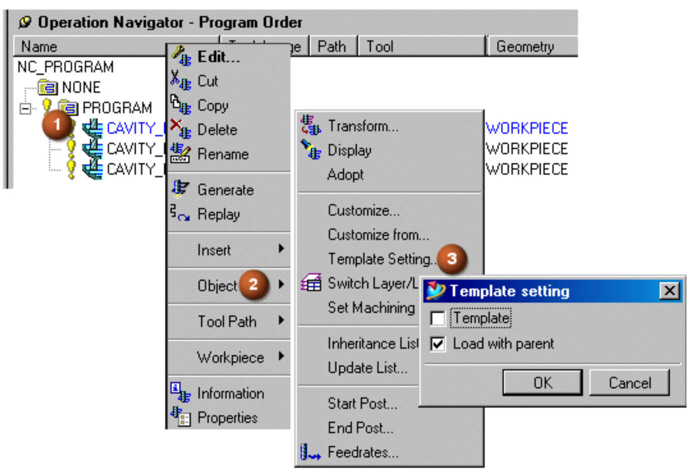
Configuration file.



Creating a Template

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

Template files can contain many operations and groups. You can specify which operations and groups are used as templates by choosing the Template Setting option in the Operation Navigator.



©UGS Corp., All Rights Reserved

Highlight the desired operations and/or groups in the Operation Navigator (1) and with MB3 choose OBJECT (2) →Template Setting (3). 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.

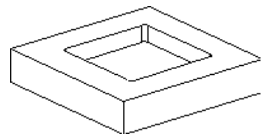
The Load with parent option also allows the determination of which operations and groups in a template part file will appear in the Operation Navigator when the template part file is specified as a Setup in the Machining Environment dialog. The Setup is the initial template part file which is selected 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."

 ©UGS Corp., All Rights Reserved

Activity: 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**.

☐ Save As *****_single_pocket**, 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**.

☐ Select the **Create Geometry** icon from the Create toolbar.

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 toolbar.

The Create Operation dialog is displayed.

☐ Choose the **Rough_Follow** icon as the subtype.

©UGS Corp., All Rights Reserved

☐ 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 toolbar.

The **Create Operation** dialog is displayed.

☐ Choose the **Finish_Floor** icon.

☐ Change the **Tool** to **UGTI0201_012**.

☐ Change the **Method** to **MILL_FINISH**.

☐ Choose **OK** 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** toolbar.

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

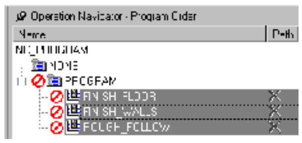
 ©UGS Corp., All Rights Reserved

if the **Load with parent** and **Template** setting, for that particular operation, is active. You will do this next.

Step 4: Make 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.

©UGS Corp., All Rights Reserved

☐ **Save** and **Close** the part file.

 ©UGS Corp., All Rights Reserved

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

©UGS Corp., All Rights Reserved

- **In general templates:**
  - for performance purposes, are opened and read only the first time that they are required
  - once it 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, reset the Configuration to force the updated template files to be read when you select or browse for them

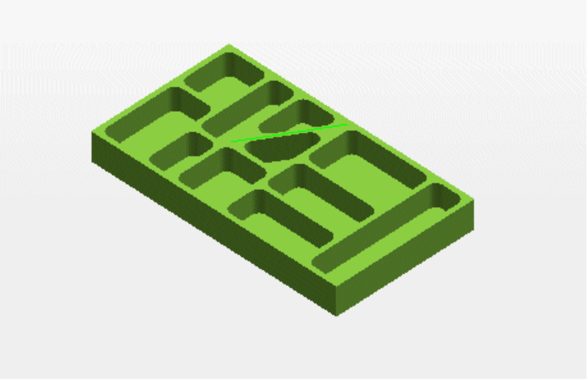
 ©UGS Corp., All Rights Reserved

Activity: 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 specify them in the new part file.

Step 1: Open a new part file and rename it.

☐ Open **ama_multi_pocket**.



☐ Save As *****_multi_pocket**.

☐ 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, it is the current template being used for this part.

Step 2: Change template files.

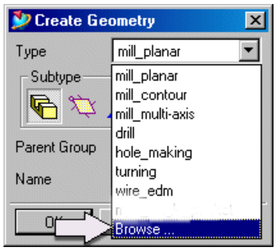
Another method of choosing a template is to change the **Type** in the create dialogs. When using this method you are limited to the options that are available in the template file.

©UGS Corp., All Rights Reserved

☐ Choose the **Create Geometry** icon from the **Create** toolbar.

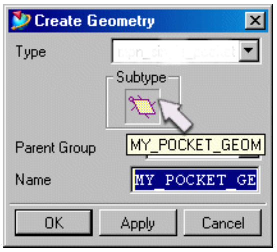
The Create Geometry dialog is displayed.

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



☐ 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: Specify 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**.

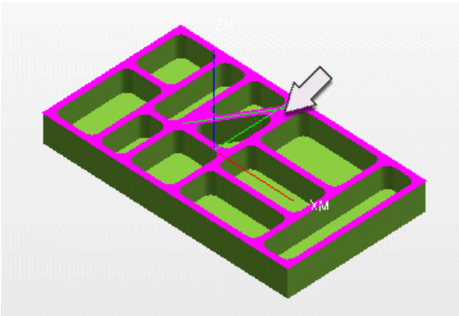
 ©UGS Corp., All Rights Reserved

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**.

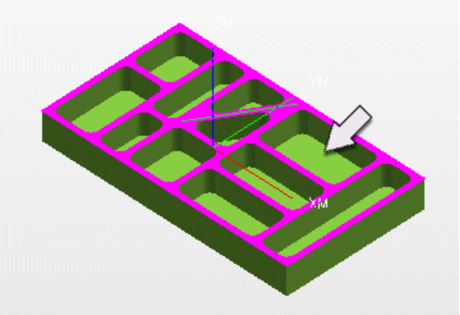
☐ Choose the **Floor** plane icon and then **Select**.

The **Plane Constructor** dialog is displayed.

☐ Set the **Filter** to **Face**.

©UGS Corp., All Rights Reserved

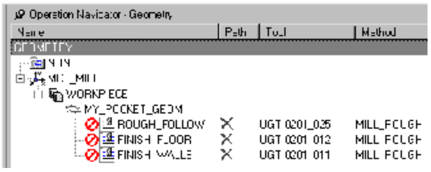
☐ Select the floor of any of the pockets.



☐ Choose **OK** twice.

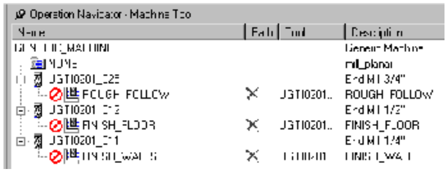
You have specified all of the information needed to generate the tool paths.

☐ 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: Generate the tool paths.

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

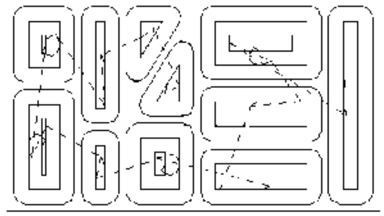
☐ Choose the **MY_POCKET_GEOM** group object and using MB3, **Generate** the tool paths.

The Tool Path Generation dialog is displayed.

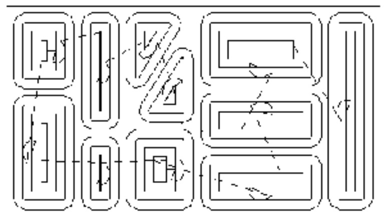
☐ Choose **OK** as necessary to continue generating the tool paths.

 ©UGS Corp., All Rights Reserved

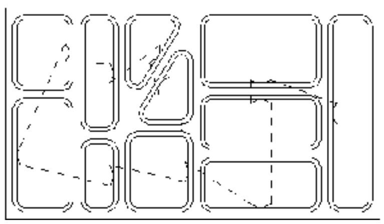
The **ROUGH_FOLLOW** operation creates a tool path similar to the following illustration:



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.

©UGS Corp., All Rights Reserved

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. The 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.

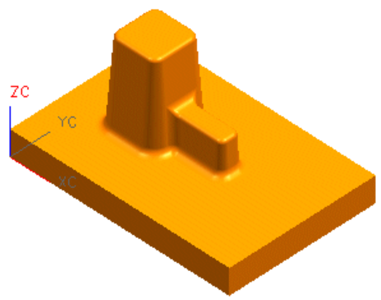
 ©UGS Corp., All Rights Reserved

Activity: Using the Die_Sequence Template

This activity shows the use of a die machining sequence of operations, included in the Manufacturing application, to machine somewhat complicated part geometry. This sequence is used to rough, semi-finish and finish a part based on die machining practices.

Step 1: Open the part file.

☐ Open **ama_deep_core**.



☐ Save the part as *****_deep_core**.

☐ Choose **Application→Manufacturing**.

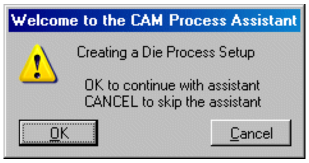
☐ Choose the CAM Session Configuration **mill_contour**.

☐ 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.

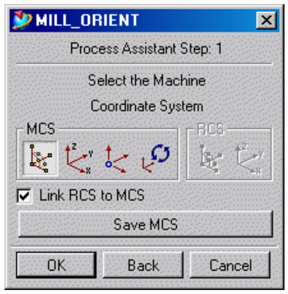


The Process Assistant guides you in selecting the geometry for the machining sequence.

☐ Choose **OK**.

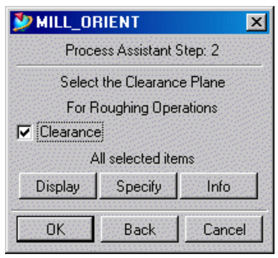
©UGS Corp., All Rights Reserved

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.

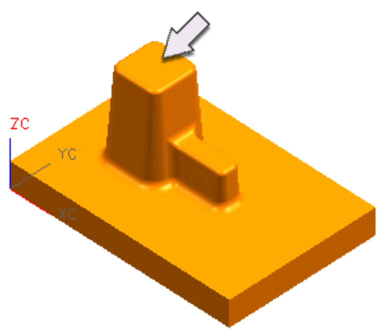


☐ Choose **OK**.

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 this 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.

 ©UGS Corp., All Rights Reserved

☐ In the **Process Assistant Step: 3** dialog, choose **Display**.

The part geometry is displayed.

☐ Choose **OK**.

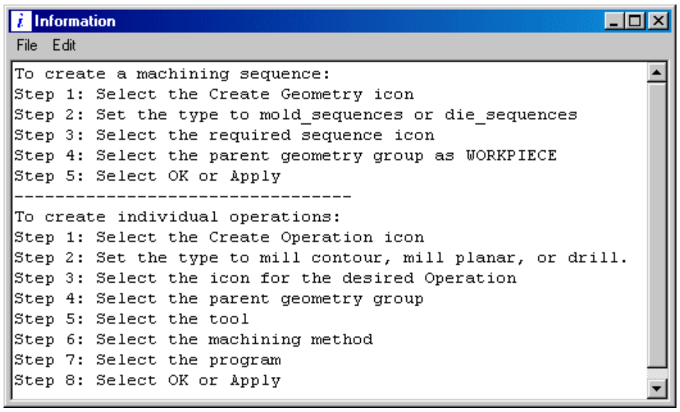
The **Process Assistant Step: 4** dialog is displayed.

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

The Blank geometry is displayed.

☐ Choose **OK**.

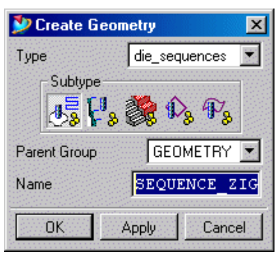
A dialog window appears, referring you to an information window for further instructions. The information window which is displayed, gives the steps necessary for creating a machining sequence and/or individual operations.



☐ Choose **OK**.

☐ Choose the **Create Geometry** icon from the Create toolbar.

The Create Geometry dialog is displayed.



Notice the various **Subtype** icons on the Create Geometry dialog. Each represents a cutting sequence (Zigzag, Zlevel, IPW, Profile 2D, Profile 3D).

Step 2: Create the Sequence.

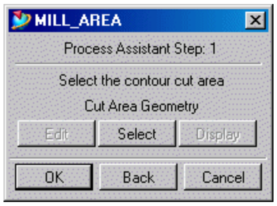
☐ In the Create Geometry dialog, choose **SEQUENCE_ZLEVEL**.

©UGS Corp., All Rights Reserved

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

☐ Choose **OK**.

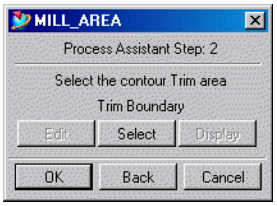
A new **Process Assistant** starts.



The Process Assistance asks for the selection of the cut area geometry. If you do not specify a cut area, all part geometry will be machined by default.

☐ At the prompt, choose **OK** to select all of the part geometry.

The **Process Assistant Step:2** dialog is displayed asking you to specify the trim area.



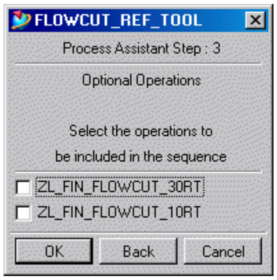
You can specify the trim boundary which limits the tool path to the area inside or outside of the boundary. You will select a trim boundary around the outside of the part to prevent the tool from moving down the side of the part.

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

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

☐ Select the bottom of the part and choose **OK** twice.

The **Process Assistant Step:3** dialog is displayed asking you to specify optional operations for cutting the part.



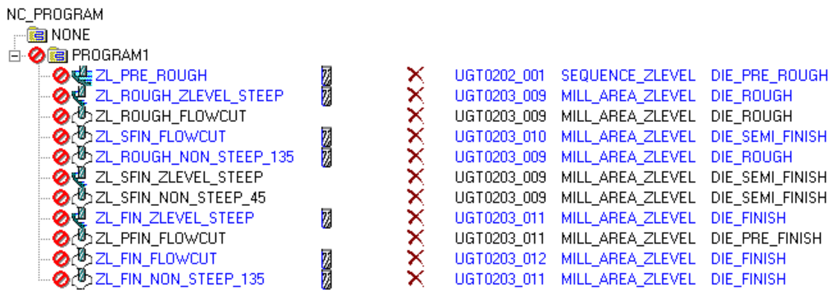
☐ Choose **OK**.

 ©UGS Corp., All Rights Reserved

☐ 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: Generate the tool paths.

☐ Highlight the individual operations, group using MB3, select **Generate** and examine each tool path individually.

☐ **Close** the part file.

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 bit maps (.bmp file extensions). NX automatically searches *UGII_USER_DIR\application* for bit map files. When creating custom icons, place all icons in this directory and define the environment variable UGII_USER_DIR located in the ugii_env.dat file.

 ©UGS Corp., All Rights Reserved

Activity: 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 NX.

☐ Use **File→Open**.

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

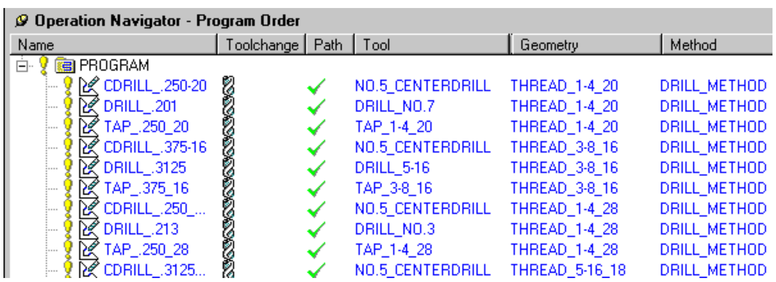
☐ Save the part as, *****_tap_template**.

☐ Enter the Manufacturing application.

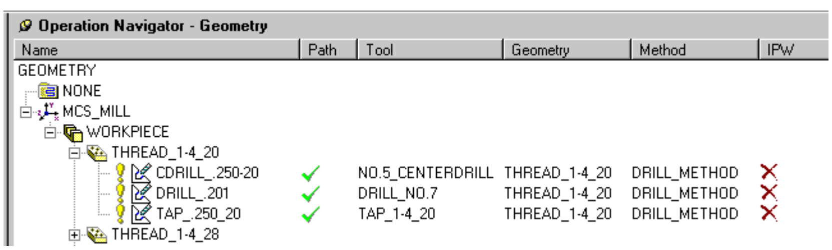
Step 2: Examine the various operations.

☐ 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.

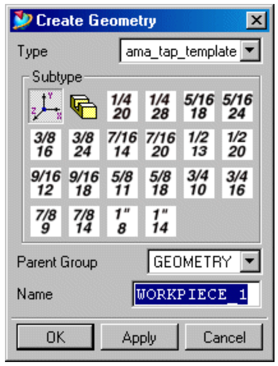


☐ Display the Geometry view in the Operation Navigator and expand the **THREAD_1-4_20** geometry group object.

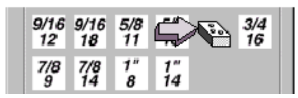


Pay close attention to the name of the individual geometry objects. Individual bit maps have been previously created and exist for each tap size. The names of these bit map files match the name of the geometry object. For instance, the bit map for the **THREAD_9-16_12** object is **thread_9-16_12.bmp**.

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 bit map icon will appear.

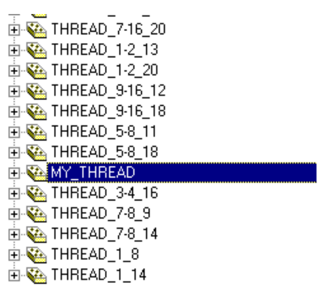


NX, by default, uses the environment variable **UGII_USER_DIR\application** to search for bit map files. This variable must be set and is located in the UGII_ENV.dat file. The environment variable, has already been set for you for this activity.

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: Rename the geometry object.

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



☐ **Save** the template part file.

 ©UGS Corp., All Rights Reserved

Step 4: Use 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**.

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

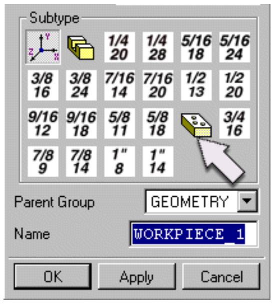
☐ Choose the **Create Geometry** icon from the Create toolbar.

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).

☐ 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**.

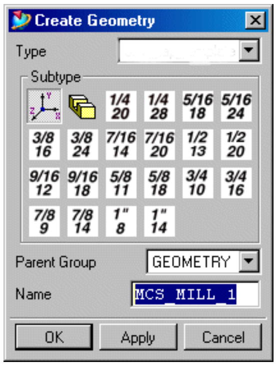
©UGS Corp., All Rights Reserved

☐ Choose the **Create Geometry** icon from the Create toolbar.

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

☐ Browse or select *****_tap_template** for the **Type**.

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



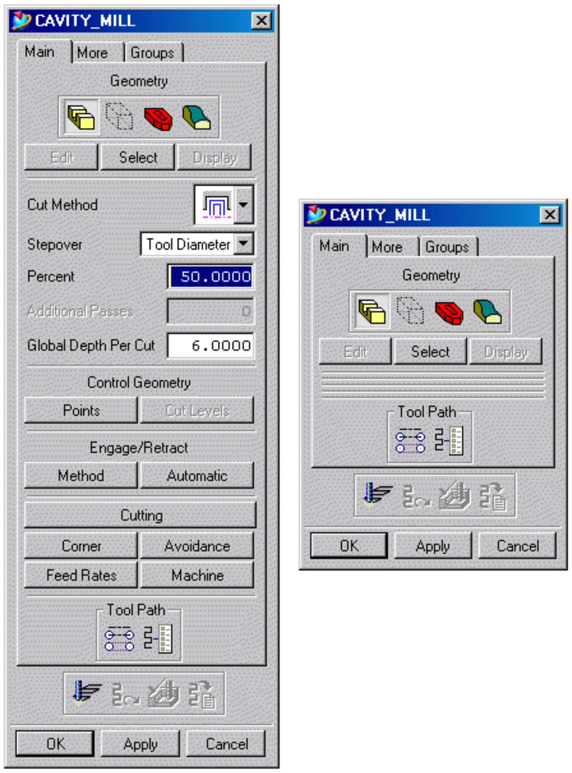
☐ Choose **Cancel**.

☐ **Close** all parts without saving.

 ©UGS Corp., All Rights Reserved

Customizing Dialogs

A unique and powerful feature of NX CAM is the ability to customize dialogs by the addition or removal of items from operations, geometry or method objects. You can also create you own unique dialogs. The following example shows a standard NX dialog and the same dialog after customizing.



The Template status for operations is on (by default).

©UGS Corp., All Rights Reserved

Activity: Customizing Dialogs

In this activity, you will customize several dialogs.

Step 1: Create a template part file.

Customizing is practical for options and parameters that are used numerous times with little or no changes being made. In this activity you will create a new template part file to store the custom dialogs that you create.

☐ Navigate to your parts folder and open the file **ama_seedpart_in**.

☐ Save the part as, *****_customized_in**.

☐ Enter the Manufacturing application.

☐ Use the **CAM_GENERAL** Session Configuration and the **mill_contour** Setup.

Step 2: Customize a Cavity Milling Operation.

The first operation you will customize will be a Cavity Milling operation.

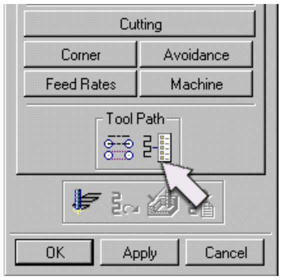
☐ Choose the **Create Operation icon**.

☐ Choose the Cavity Milling icon from the Create Operation dialog, in the Name field, key in **CUSTOM_CM**.

☐ Choose **OK**.

The CAVITY_MILL dialog is displayed.

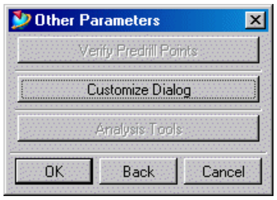
☐ Towards the bottom of the dialog, choose the Options (1) icon.



The Other Parameters dialog is displayed.

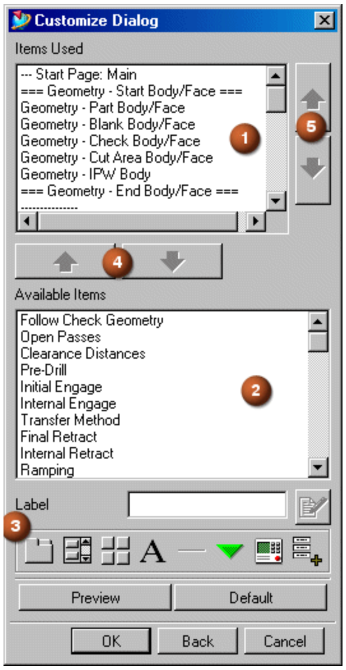
 ©UGS Corp., All Rights Reserved

☐ Choose the Customize Dialog button.



The Customize Dialog is displayed. This dialog consists of four areas of importance:

- (1) scroll box consisting of items that are currently in the operation dialog
- (2) scroll box consisting of items that can be added to the operation dialog
- (3) dialog options allowing for property pages, scrolled windows, group areas, labels, separator delimiters, More / Less button, User Defined Events, and All other Parameters
- (4) in between the two scroll boxes (1) (2) are Add and Remove arrows that push selected items from one scroll box to the other
- (5) Up and Down arrows that allow the positioning of various items within the dialog.



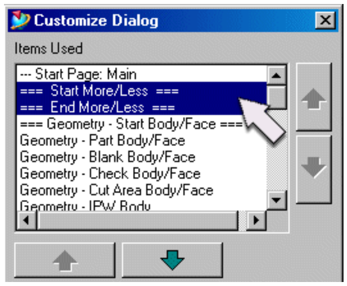
Start the customizing process by adding a **More / Less** item to the dialog.

©UGS Corp., All Rights Reserved

☐ In the Items Used scroll box (1), highlight the **Start Page: Main item**.

☐ At the bottom of the dialog (3), select the **More / Less** button.

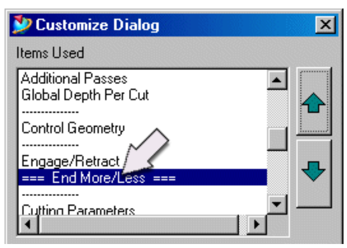
This places the More / Less button on the dialog.



Currently, the More / Less button does not encompass any dialog options. You will now position the end of the button down, so that a portion of the dialog is within the confines of the More / Less item.

☐ In the Items Used scroll box (1) , highlight the **End More/Less** item

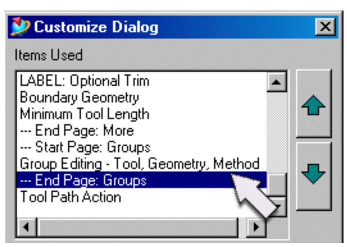
☐ Using the Move down arrow (5), move the item below the **Engage / Retract** item (each click of the arrow will move the item one position).



☐ Choose **OK**.

☐ Choose **OK** again.

The dialog is displayed in a much shorter form.



 ©UGS Corp., All Rights Reserved

- ☐ Practice using the More/Less item to become more familiar with it.

- ☐ Choose OK to the Cavity Mill operation.

Step 3: Add a property page.

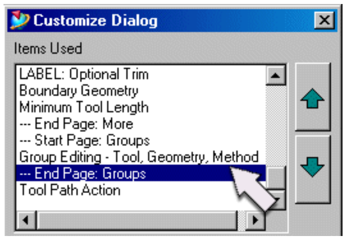
You will now add a property page and place items on the new page.

- ☐ Highlight the **CUSTOM_CM** operation in the Operation Navigator.

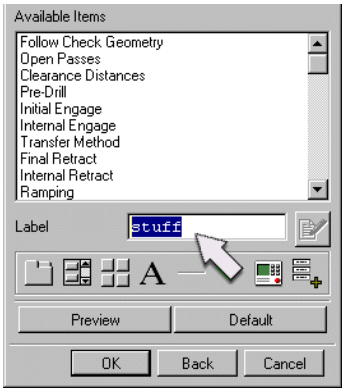
- ☐ Use MB3, **Object→Customize**.

  The Customize Dialog is displayed.

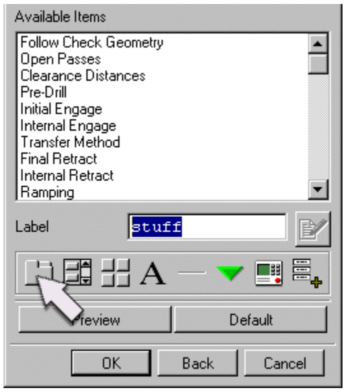
- ☐ In the Items Used Scroll box (1), highlight the **End Page: Groups** item.



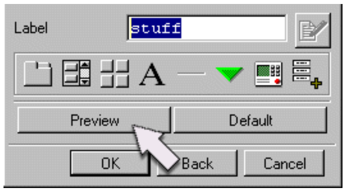
- ☐ Key in **STUFF** in the Label field.



☐ Select the **Property page** icon.



☐ Choose the Preview button.



The Cavity Milling dialog is displayed with a new property page.

☐ Click on the **STUFF** property page.

The STUFF property page is blank. You will now add items to this page.

☐ Choose the **Back** button.

☐ Highlight the **Start Page: STUFF** item in the Items Used scroll box (1).

☐ Highlight the **Slowdowns** item in the Available Items scroll box (2).

☐ Choose the **Add button** (4).

☐ Choose the **Preview** button to review the additional item.

☐ Click on the **STUFF** property page.

The STUFF property page has Slowdowns parameters listed but lacks a clear label. You will now add the label.

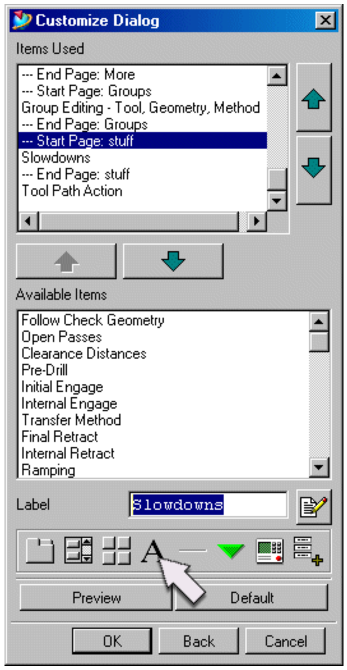
☐ Choose the **Back** button.

☐ Highlight the **Start Page: STUFF** item in the Items Used scroll box (1).

 ©UGS Corp., All Rights Reserved

☐ In the Label field, key in **SLOWDOWNS**.

☐ Click on the **Label** button.



☐ Use the **Preview** button to view the dialog.

☐ Choose the **Back** button.

☐ Choose **OK**.

You have customized the Cavity Milling dialog. Take the time to create a dialog for other operations that you may use on a regular basis. Remember that you can use customizing to change default values as well.

©UGS Corp., All Rights Reserved

Activity: Customizing a Program Header

In this activity, you will customize the Program object in the Program Order view of the Operation Navigator.

Step 1: Examine the current Program object.

☐ Continue using the part file from the previous activity.

☐ If necessary, change to the Program Order view in the Operation Navigator.

☐ Double click on the Program object.

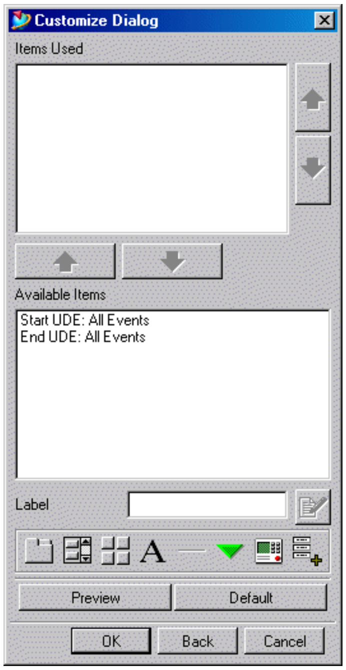
Note that the object will not open since the object contains no data.

Step 2: Customize the program object.

☐ Highlight the **Program** object.

☐ Use MB3, **Object→Customize**.

The Customize Dialog is displayed.



There are no items in the Items Used scroll box. Specifically, that is why double-clicking the Program object did not work.

Step 3: Create a User Defined Event.

You will add a User Defined Event (UDE) to the Items Used list.

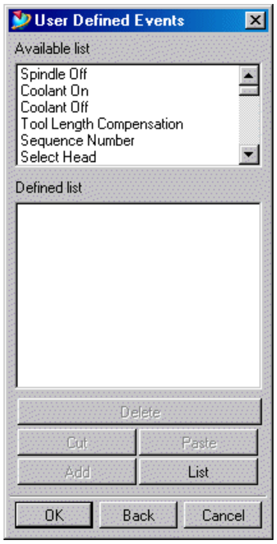
☐ **Cancel** the dialog.

 ©UGS Corp., All Rights Reserved

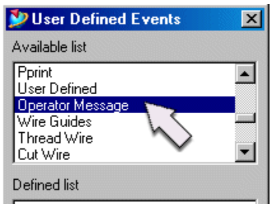
☐ Highlight the **Program** object.

☐ Use MB3, **Object→Start Post...**.

The User Defined Events dialog is displayed.



☐ Scroll down the list until the **PPrint** item is shown, then double-click on the **Operator Message** item.



The Operator Message Item is displayed.

☐ Key in **This is a Comment** in the text field.

☐ Choose **OK**.

☐ Choose **OK** on the User Defined Events dialog.

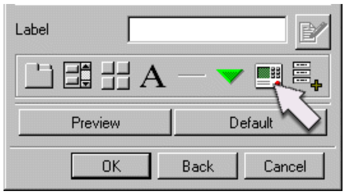
Step 4: Add the User Defined Event to the Program Object dialog.

☐ Highlight the **Program** object in the Program Order view of the Operation Navigator.

☐ Use MB3, **Object→Customize**.

©UGS Corp., All Rights Reserved

☐ Select the User Defined Events button.



The User Defined Events Customize dialog is displayed.

☐ Highlight the **Operator Message** on the list.

☐ Choose **OK**.

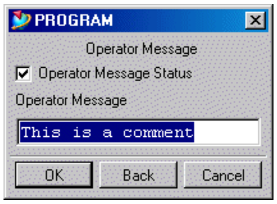
☐ Choose **OK** in the Customize Dialog.

Step 5: Test the Results.

You have successfully added a User Defined Event to an object that does not have a dialog. Since the item you added has a dialog, it will be displayed when you double-click on the object.

☐ Double-click on the **Program** object in the Program Order view of the Operation Navigator.

The dialog exhibits the characteristics of the Operator Message dialog.



☐ On your own, add another User Defined Event to the Program object (Hint: try the Origin UDE).

 ©UGS Corp., All Rights Reserved

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.
- Customized dialogs.

©UGS Corp., All Rights Reserved

Lesson

8 *Hole Making*

Purpose

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

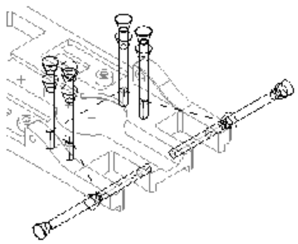
Objective

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

- Create a hole making program that machines simple, countersunk, and counter bored 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.

Hole Making Overview

Hole Making is an advanced manufacturing application that automates the creation of operations such as spot drilling, drilling, countersinking, counter boring, 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.

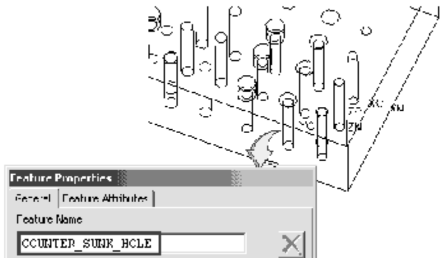


Manufacturing features contain information that allows the Hole Making processor to make machining decisions based on rules defined in templates and applied through Knowledge Fusion. Manufacturing features most commonly consist of NX modeled features such as simple, countersunk, and counter bore holes 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.

 ©UGS Corp., All Rights Reserved

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.

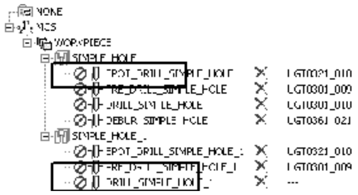


Knowledge Fusion rules are associated with feature groups through Adopted Operations displayed in the Knowledge Fusion Navigator. This association allows the Hole Making processor to apply specific machining rules to feature groups.

©UGS Corp., All Rights Reserved

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.

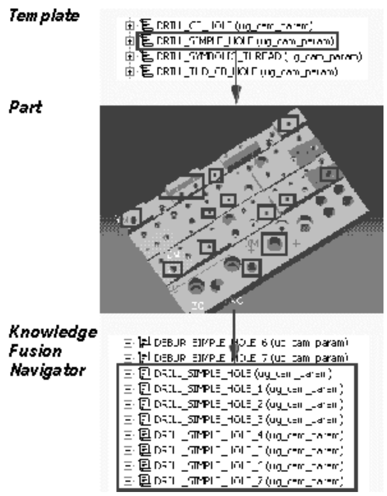


 ©UGS Corp., All Rights Reserved

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.

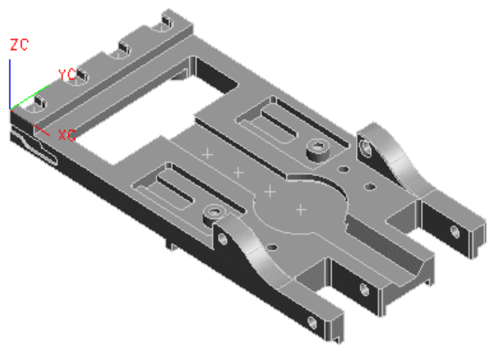
©UGS Corp., All Rights Reserved

Activity: Machining Holes

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

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

☐ Open the part **ama_holemaking**.



☐ Save the file as ******_holemaking***, 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**.

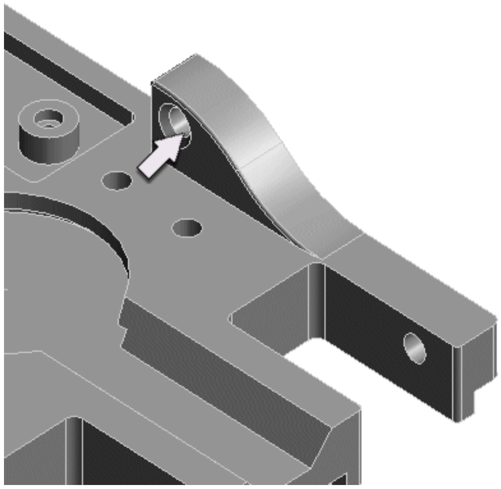
Step 3: Display Feature Properties.

You will display the feature name of a countersunk hole.

☐ Choose the **Select Features** icon. 

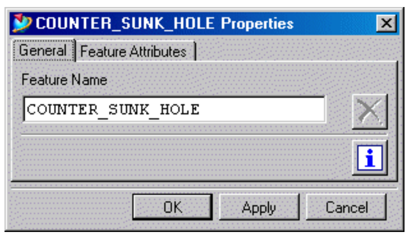
 ©UGS Corp., All Rights Reserved

☐ Select the countersunk hole illustrated below.



☐ Choose MB3, **Properties**.

The COUNTER_SUNK_HOLE Properties dialog is displayed showing the feature name as **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.

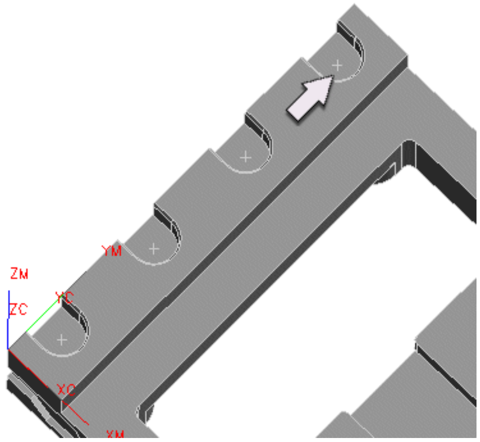
Step 4: Display Object Properties.

You will display the properties of an attributed point.

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

©UGS Corp., All Rights Reserved

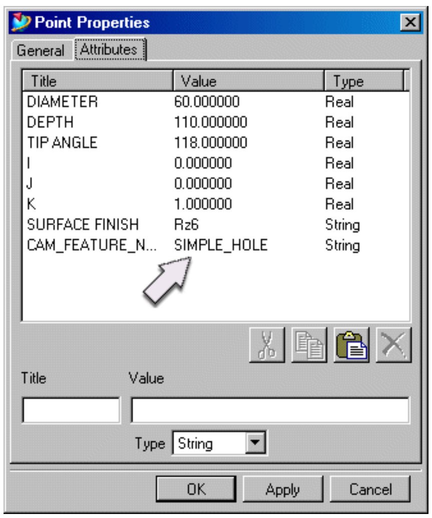
☐ Select the point as illustrated.



☐ Choose MB3, **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.

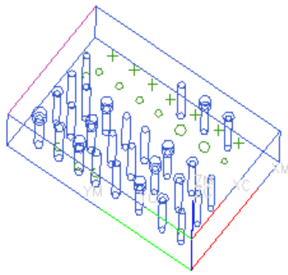
Step 5: Examine the Hole Making Template.

You will review the Hole Making template. 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.

 ©UGS Corp., All Rights Reserved

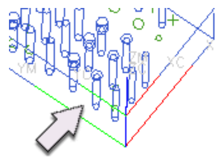
☐ In the menu bar, choose **Window→More**.

☐ Choose **hole_making** in the **Change Window** dialog and then choose **OK**.



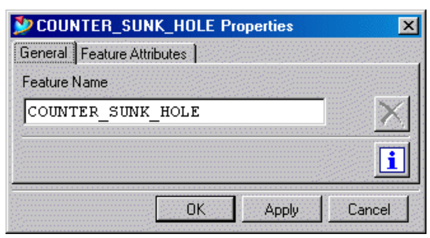
☐ Choose the **Select Features** icon.

☐ Select the countersunk hole illustrated below.



☐ Choose MB3. **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.



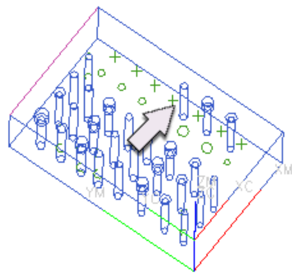
Since your part contains machinable features called **COUNTER_SUNK_HOLE**, the Hole Making processor will recognize all occurrences of this feature and apply the appropriate rules for machining.

☐ **Cancel** the Feature Properties dialog.

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

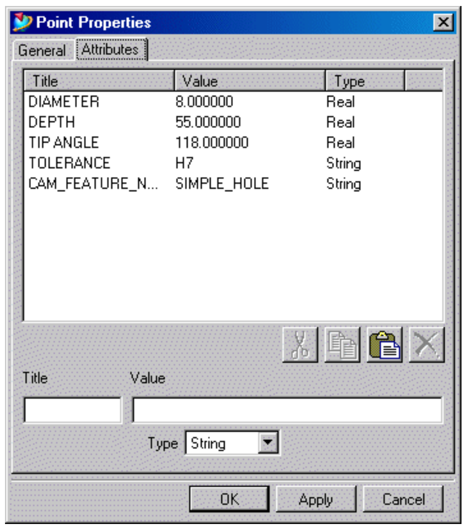
©UGS Corp., All Rights Reserved

☐ Select the point (4th hole from the right corner) as illustrated.



☐ Choose MB3, **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).

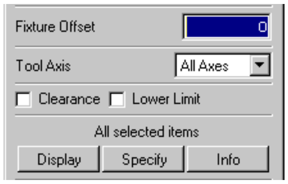
 ©UGS Corp., All Rights Reserved

☐ 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.

☐ Display the **Geometry** view of the Operation Navigator.

☐ Double-click on the **MCS** object.

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



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

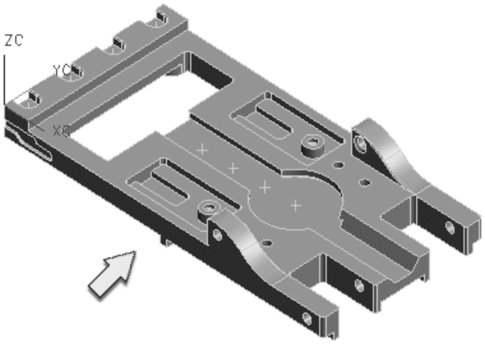
Step 7: Specify the Part Geometry.

You will specify the solid body as the part geometry.

☐ 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.

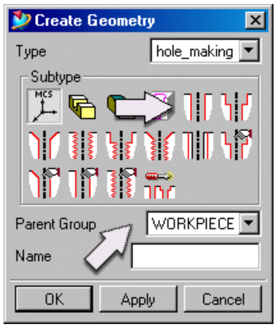
Step 8: Create Operations to Machine Simple Holes.

You are now ready to create the operations that machine the simple holes.

©UGS Corp., All Rights Reserved

☐ Choose the **Create Geometry** icon on the toolbar.

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



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

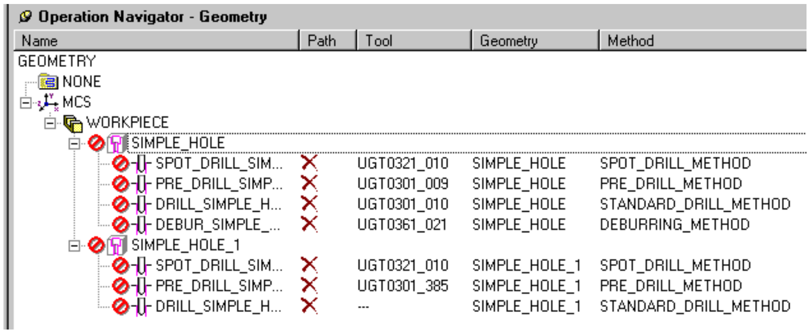
The processor will take a few moments to process the information.

☐ 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.



 ©UGS Corp., All Rights Reserved

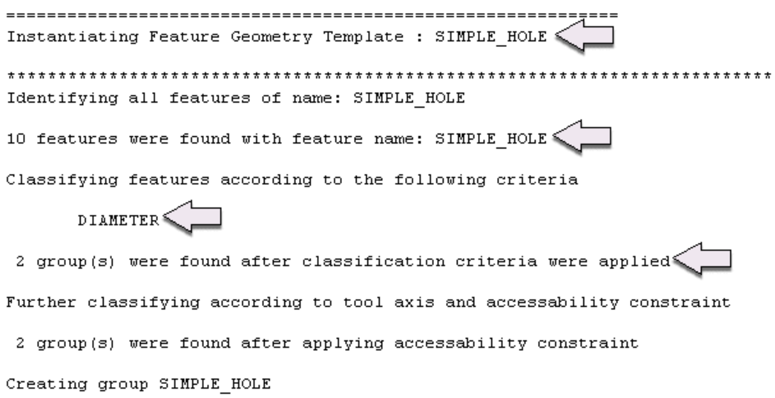
Your results may differ slightly from those illustrated above. Since the Hole Making processor assigns feature group numbers randomly, the operations listed in one feature group (SIMPLE_HOLE) might be listed in the other feature group in your part.

Step 9: View the Information Window.

The Information window displays a record of the data that was processed and the output that was generated.

☐ 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.

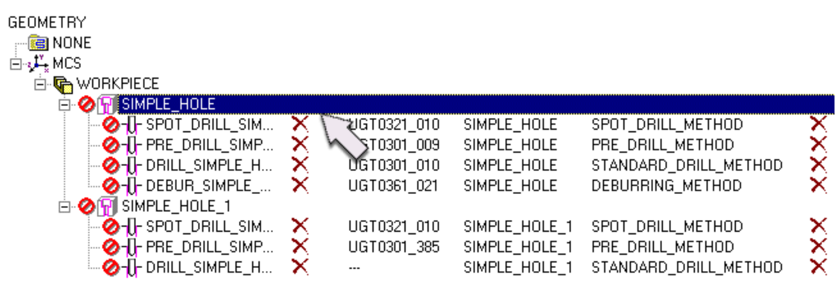


```
============================================================
Instantiating Feature Geometry Template : SIMPLE_HOLE

****************************************************************************
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

Further classifying according to tool axis and accessability constraint

 2 group(s) were found after applying accessability constraint

Creating group SIMPLE_HOLE
```

☐ Dismiss the Information window.

Step 10: Examine the Feature Groups.

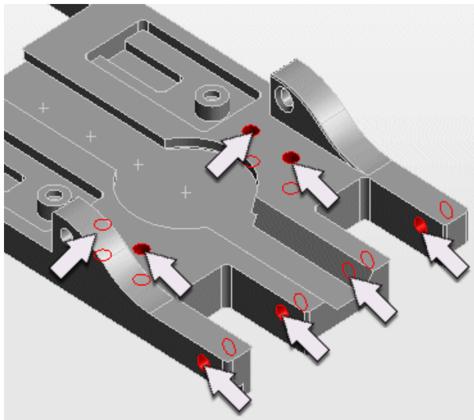
☐ Double-click on the **SIMPLE_HOLE** feature group icon in the Operation Navigator.



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

©UGS Corp., All Rights Reserved

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.

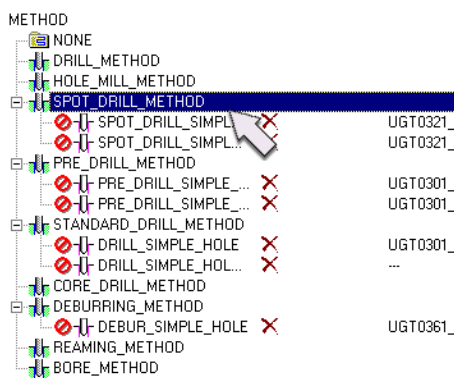
Step 11: Perform a Tool Query.

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**.

☐ Expand the **Machining Method** groups.

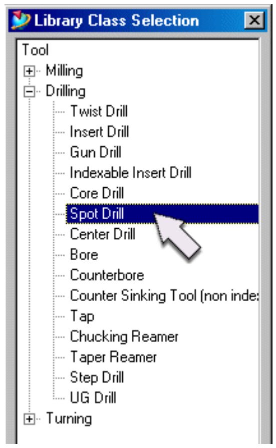
☐ Double-click on the **SPOT_DRILL_METHOD** group icon.



☐ Choose **Class** under **Tool Query**.

 ©UGS Corp., All Rights Reserved

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(s) 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.

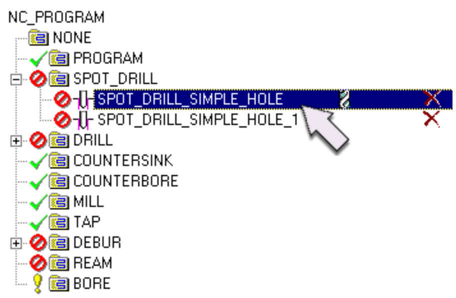
☐ **Cancel** the hole_making dialog.

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.

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

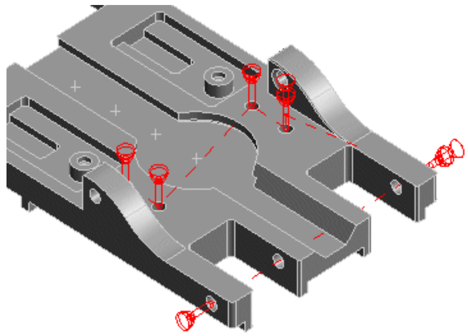


©UGS Corp., All Rights Reserved

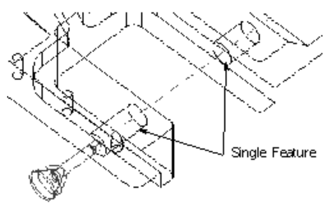
☐ Choose the **Edit Display** icon under Tool Path and set the **Tool Display** option to **3-D**.

☐ 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.

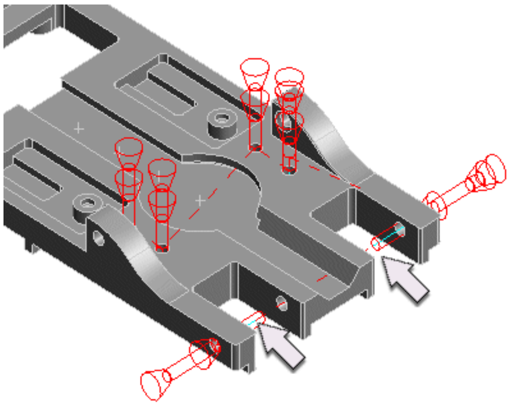
☐ Choose the **Edit Display** icon under **Tool Path** and set the **Tool Display** option to **3-D**.

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

☐ **Generate** the tool path.

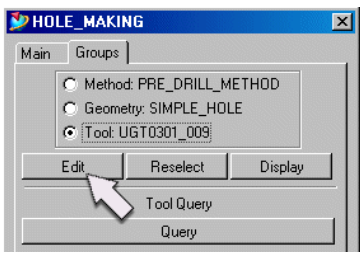
 ©UGS Corp., All Rights Reserved

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.

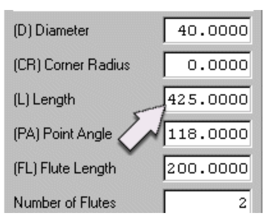


Step 14: Edit the Tool Length.

- ☐ Choose the **Groups** tab from the Hole_Making dialog.
- ☐ With **Tool:UGT0301_009** chosen, choose **Edit**.



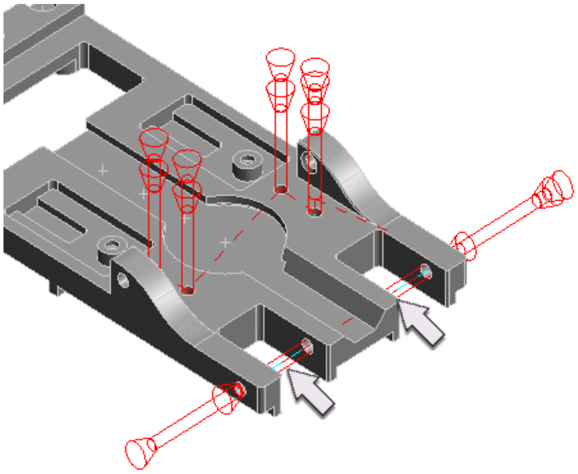
- ☐ Choose **OK** to the **Group Editing** warning message.
- ☐ Change the **Length** to **425**.



- ☐ Choose **OK**.
- ☐ Choose **YES** to the **Edit Tool** warning message.
- ☐ Choose the **Main** tab.

©UGS Corp., All Rights Reserved

☐ **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.

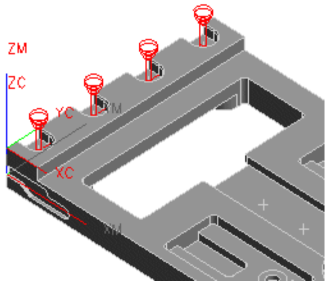
☐ Double-click on the **SPOT_DRILL_SIMPLE_HOLE_1**.

 ©UGS Corp., All Rights Reserved

☐ Choose **Edit Display** and set the **Tool Display** 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.

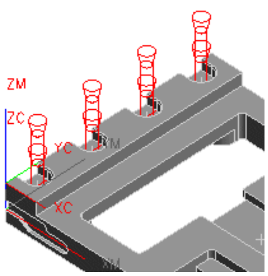
You will display the tools and generate the tool path for the **PRE_DRILL_SIMPLE_HOLE_1** operation.

☐ Double-click on the **PRE_DRILL_SIMPLE_HOLE_1**.

☐ Choose the **Edit Display** icon 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.

©UGS Corp., All Rights Reserved

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**.

☐ Choose the **Edit Display** 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.

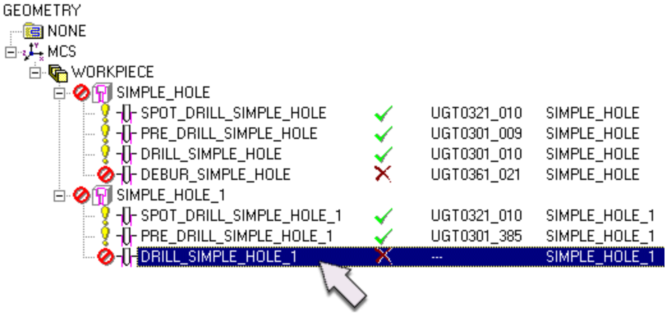
☐ Choose **OK** to the operation.

Step 19: Add a Tool.

You should verify that every operation contains a tool. To accomplish this 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**.

 ©UGS Corp., All Rights Reserved

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.

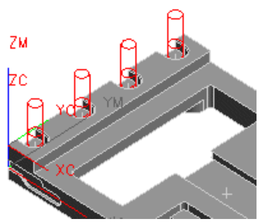
☐ Key in **60** in the **Diameter** field.

☐ Key in **200** in the **Length** field.

☐ Choose **OK** to create the tool.

☐ Choose the **Main** tab.

☐ **Generate** the tool path.



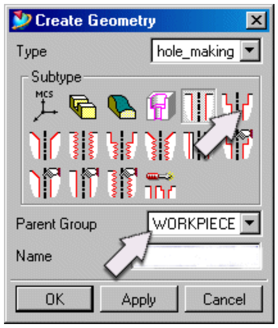
☐ Choose **OK** to complete the operation.

Step 20: Create Operations to Machine the Counter bore holes.

☐ Choose the **Create Geometry** icon in the toolbar.

©UGS Corp., All Rights Reserved

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



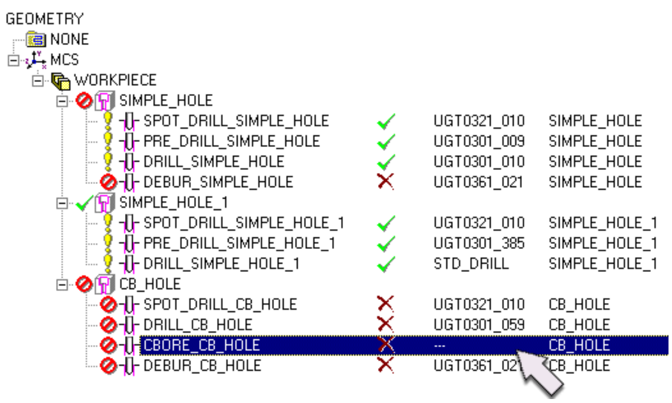
☐ Choose **OK** to create the operations.

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.

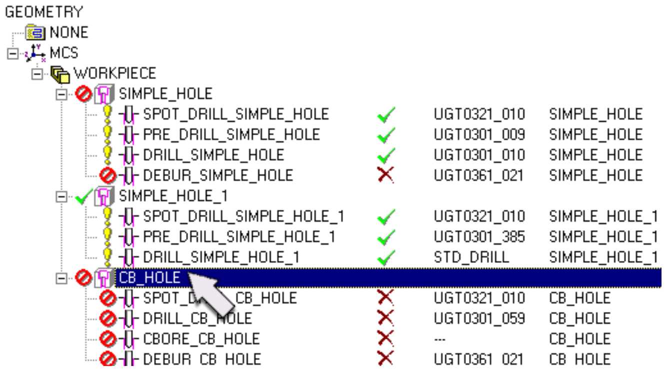


Step 21: Add a Tool.

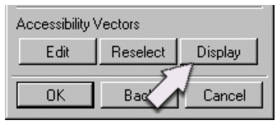
You must first identify the features that are associated with the **CB_HOLE** feature group. You can then analyze the feature to identify the diameter and depth of the holes.

 ©UGS Corp., All Rights Reserved

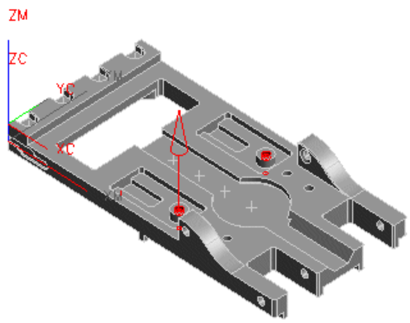
☐ Double-click on the **CB_HOLE** feature group icon.



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



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

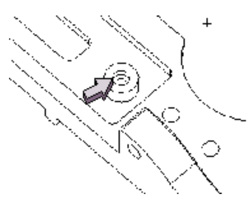


The vector identifies one of the features associated with the **CB_HOLE** feature group.

☐ **Cancel** the CB_HOLE dialog.

☐ Choose **Information→Feature** from the toolbar.

☐ Select the counter bore hole as illustrated.



☐ Choose **OK** in the **Feature Browser** dialog.

The **Information** window indicates that the counter bore diameter is 50 and the counter bore depth is 20.

©UGS Corp., All Rights Reserved

```
COUNTER_BORE_HOLE(51)
------------------------------------------------------------
p198=25                                                      25
Hole Diameter

p196=50                                                      50
C-Bore Diameter

p197=20                                                      20
C-Bore Depth

p199=0.0                                                      0
Positioning Dimension Parallel Distance Set To Zero


Feature Parameters for: COUNTER_BORE_HOLE(51)
------------------------------------------------------------
Feature Type - COUNTER_BORE_HOLE(51) THRU
Cbore Diameter       =      50.000000000
Cbore Depth          =      20.000000000
Hole Diameter        =      25.000000000
Ref Point   X        =    1110.000000000
            Y        =     175.000000000
            Z        =     115.000000000
Direction   X        =       0.000000000
            Y        =       0.000000000
            Z        =      -1.000000000

Feature Associativity for: COUNTER_BORE_HOLE(51)
------------------------------------------------------------

Parent(s):

    BLOCK(0)
    BOSS(49)
```

☐ 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.

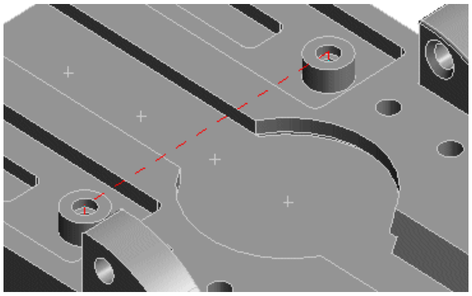
All of the operations within the **CB_HOLE** feature group now contain tools.

Step 22: Generate the Tool Paths for the Counter bore Holes.

 ©UGS Corp., All Rights Reserved

☐ Highlight the **CB_HOLE** feature group and choose MB3, **Generate** to generate the tool paths.

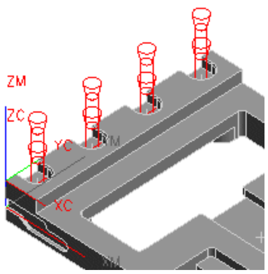
☐ Choose **OK** to accept each one of the tool paths.



☐ **Save** the part file.

Tagging

The Hole Making processor recognizes NX based features such as simple holes, countersunk holes, counter bored 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.

 ©UGS Corp., All Rights Reserved

Activity: Tagging Points

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

Step 1: Tag Points with Attributes.

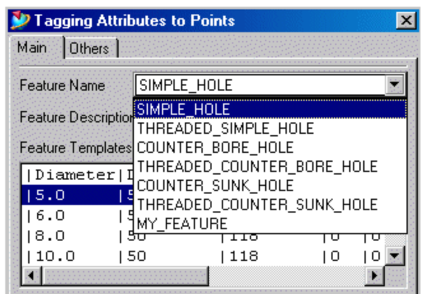
- ☐ Continue using *****_holemaking**.

- ☐ Choose **Tools→Machining Feature Manager** from the menu bar.

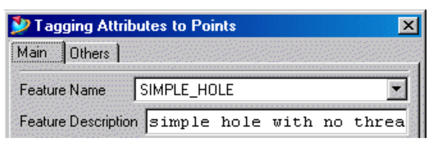
  The Machining Feature Manager is displayed.

- ☐ Select the **Tag Points** icon. 

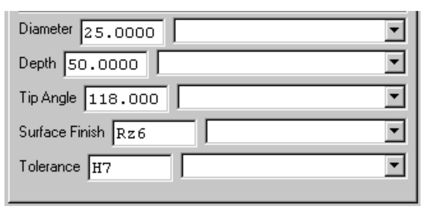
- ☐ Display the menu options next to **Feature Name** to see the different feature names that can be assigned attributes.



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



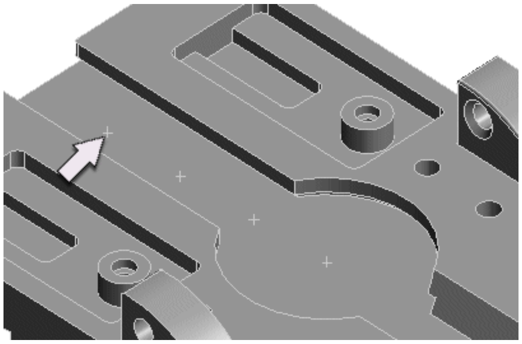
- ☐ Fill in the following values in the **Tagging Attributes to Points** dialog. Verify the **Surface Finish** field has been blanked out and that the **Tolerance** field says **H7**.



By specifying a tolerance of H7 the Hole Making processor will include a reaming operation.

©UGS Corp., All Rights Reserved

☐ Select the point as illustrated.



☐ 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.

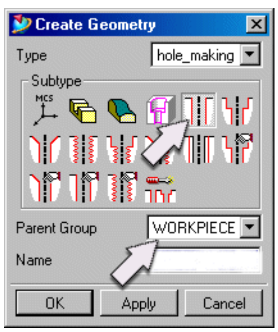
☐ Close the **Tagging Attributes to Points** dialog.

☐ **Close** the Machining Feature Manager dialog.

Step 2: Create Operations to Machine the Simple Holes.

☐ Choose the **Create Geometry** icon in the toolbar.

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

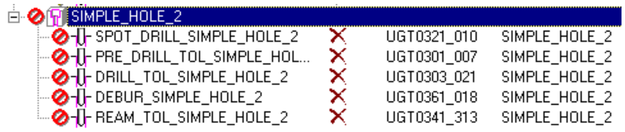


☐ Choose **OK** to create the operations.

☐ Once the processing is complete, choose **OK** in the **SIMPLE_HOLE** dialog to accept **DIAMETER** as the classification criteria.

☐ Dismiss the Information window.

 ©UGS Corp., All Rights Reserved

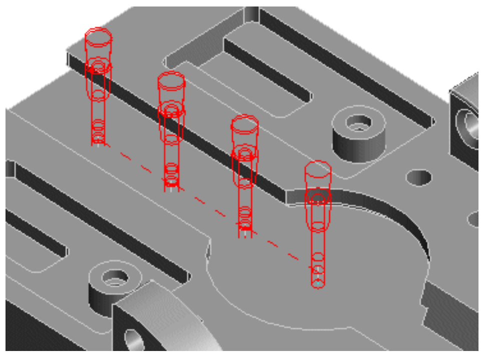


Step 3: Generate the Tool Paths for the Simple Holes.

☐ Highlight the **SIMPLE_HOLE_2** feature group and using MB3, **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.

©UGS Corp., All Rights Reserved

Optimization

Optimization improves machining effectiveness by consolidating tools to minimize the number of tools used, reordering operations to eliminate redundant tool changes and resequences 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.

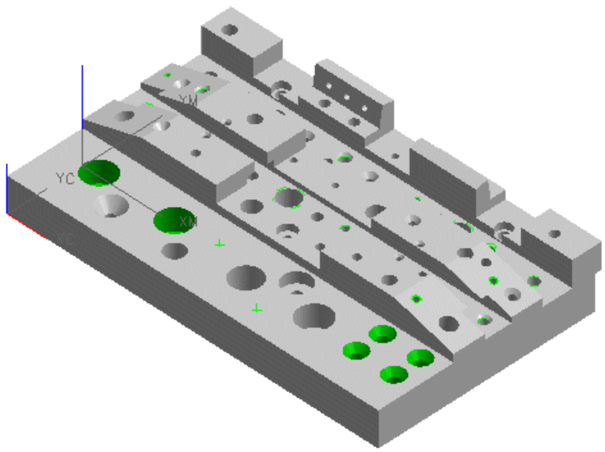
- **Create Optimization Group** creates groups containing operations that define an optimal tool path. Features cut with the same tool are resequenced to minimize tool travel distance within and between operations.

 ©UGS Corp., All Rights Reserved

Activity: Optimizing a Spot Drill Subprogram

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

☐ Open the part **ama_optimization**.



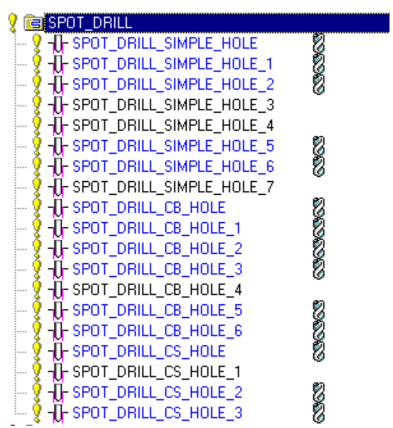
☐ **Save** the part as*****_optimization**, where ******* represents your initials.

☐ 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.



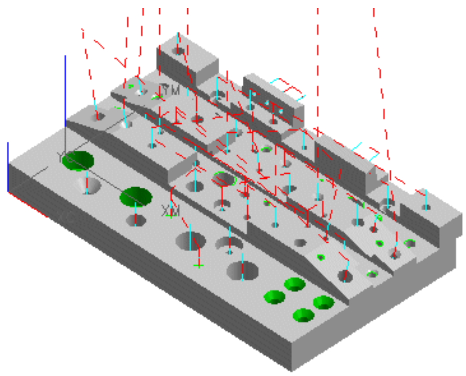
☐ Highlight the **SPOT DRILL** sub operation icon and use MB3, **Generate**.

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

☐ 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 use MB3, **Object→Optimize** .

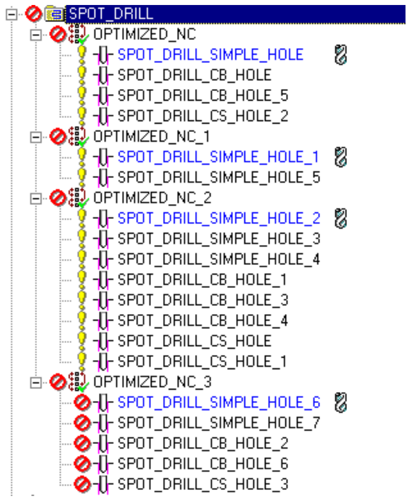
☐ Verify that all three options are turned **on** and then choose **OK** to begin the optimization process.

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

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

 ©UGS Corp., All Rights Reserved

The number of tool changes has been reduced to four.

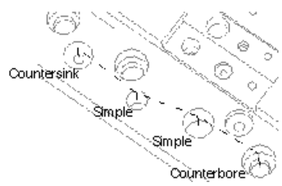


Step 4: Generate the Optimized Tool Paths.

☐ Choose the **OPTIMIZE_NC_3** object and then **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** object and then **Generate** the tool paths for all of the optimization subprograms.

☐ **Save** the part file.

©UGS Corp., All Rights Reserved

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.

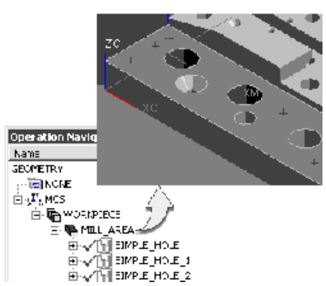
Alternate Groups of Operations are created in the Geometry view of the Operation Navigator by highlighting a feature group, and then by selecting MB3 →Object →Alternate Groups of Operations. In the Alternate Groups of Operations dialog, the upper list which is displayed contains the operations allowed 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.

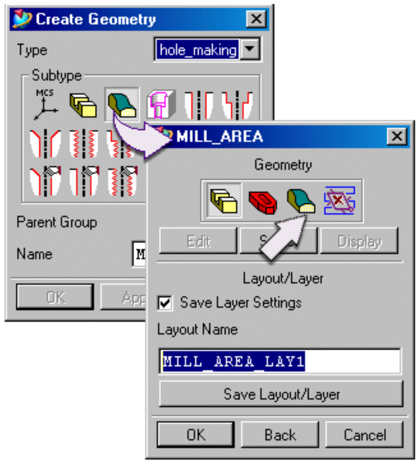
Cut Area

Cut Area allows you to reduce the scope of machinable 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.



 ©UGS Corp., All Rights Reserved

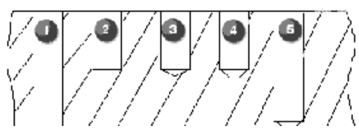
The face geometry defining the Cut Area is specified within the MILL_AREA group. Selected points and arcs will be identified during the feature grouping regardless of part or Cut Area geometry since Cut Area geometry is based on faces and part geometry is based on bodies. Points and arcs are considered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



Feature Recognition

Hole Making will recognize simple, counter bored and countersunk hole features that have not been explicitly created as a NX 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.

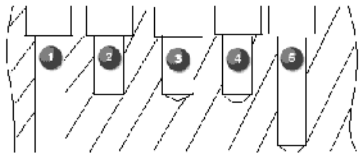
A simple hole is created from a cylinder with planes designating the top and bottom of the hole.



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

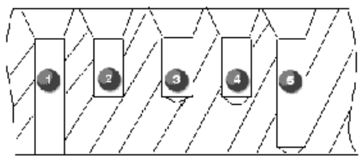
©UGS Corp., All Rights Reserved

A counter bored hole consists of a cylinder and plane, which designate the counter bore 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, counter bored hole and countersunk hole shapes as machinable features, create the operation and generate the tool path.

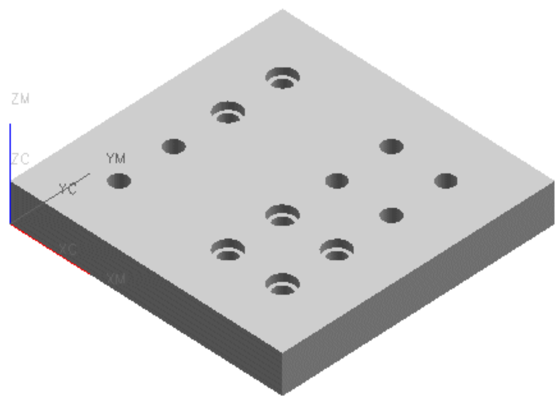
 ©UGS Corp., All Rights Reserved

Activity: Feature Recognition

In this activity you will utilize the Hole Making processor to machine simple, counter bored 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.

- ☐ Briefly examine the part.

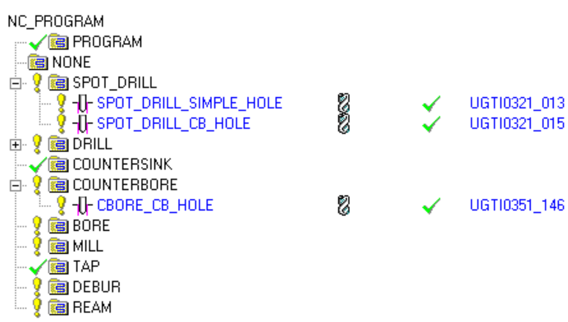


Step 2: Enter the Manufacturing Application and display the Operation Navigator.

- ☐ 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.

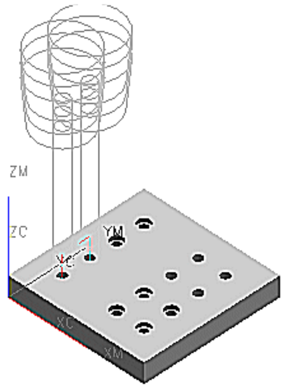


You will replay existing operations and observe that only features which have been modeled are machined.

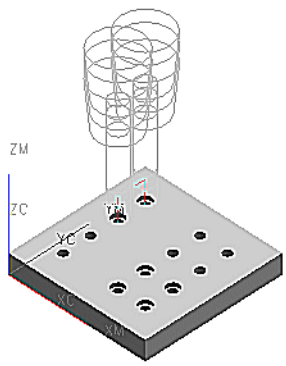
©UGS Corp., All Rights Reserved

Step 3: Replay 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 machinable 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: Create 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.

 ©UGS Corp., All Rights Reserved

☐ Choose **Tools→Machining Feature Manager** from the toolbar.

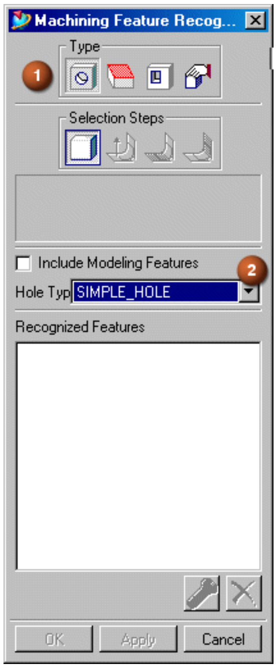
The Machining Feature Manager dialog is displayed.

☐ Select the **Recognize Feature** icon.

The Machining Feature Recognition dialog is displayed.

☐ Set the **Type** to **HOLE**(1).

☐ Set the **Hole Type** to **SIMPLE_HOLE**(2).



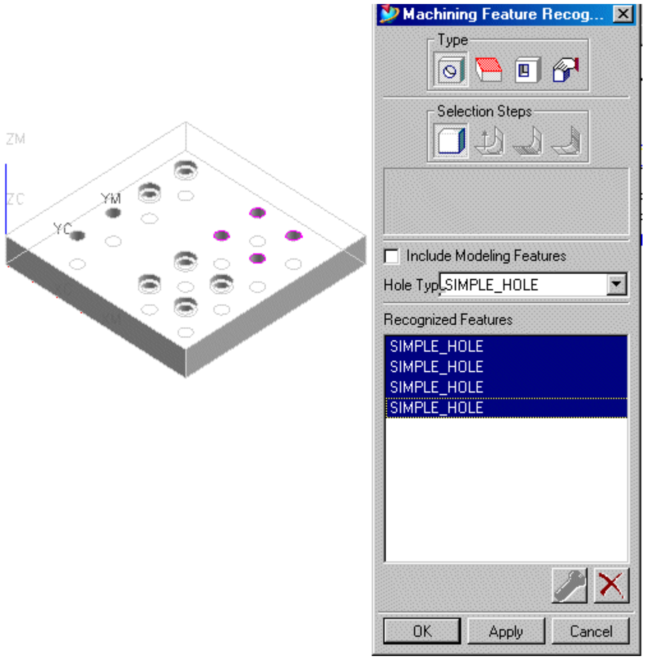
☐ Select the body containing the features (part).

☐ Choose the **Apply** button.

☐ Choose the **Part**.

©UGS Corp., All Rights Reserved

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.



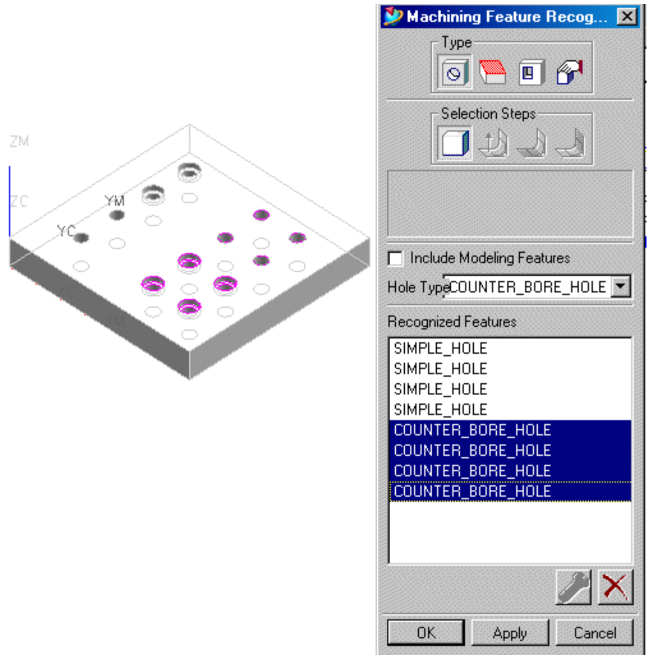
You will now use the Feature Recognition function to recognize the counter bored hole feature types as machinable features.

☐ Set the **Hole Type** to **COUNTER_BORE_HOLE**.

☐ Choose the **Apply** button.

 ©UGS Corp., All Rights Reserved

The four holes are now recognized as being counter bored holes, are displayed in the Recognized Features box of the Machining Feature Recognition dialog and are highlighted on the part.



☐ Choose **OK** on the **Machining Feature Recognition** dialog.

☐ Close the **Machining Feature Manager**.

You will now create the operations that will machine all holes that are contained in the part.

☐ Choose the **Create Geometry**icon.

☐ If necessary, set the **Type** to **hole_making** and the **Parent Group** to **WORKPIECE**.

☐ 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.

©UGS Corp., All Rights Reserved

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 toolbar.

The **Create Geometry** dialog is displayed.

☐ Choose the **CB_HOLE** icon.

☐ Choose **OK** from the **Create Geometry** dialog to begin generating the operations.

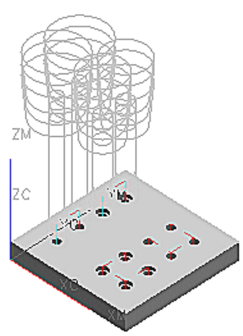
The Information Window and CB_HOLE dialog is displayed.

☐ 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.

You will now generate the tool paths.

☐ Highlight the **SPOT_DRILL** object, using MB3, **Generate** the tool path.

The tool path is generated.



☐ Repeat the above process for the **DRILL** and **COUNTERBORE** operations.

☐ **Close**, without saving, the part file.

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, in the Geometry view of the Operation Navigator, highlight the feature, use MB3→ Object→ Feature Status.

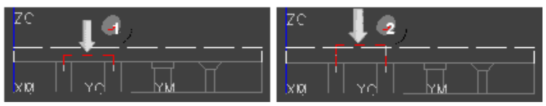
Holder Types

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.

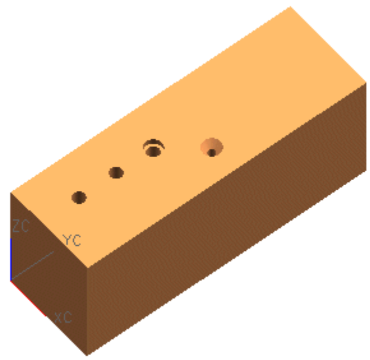
©UGS Corp., All Rights Reserved

Activity: Using the IPW in Hole Making

Step 1: Opening the part file, enter the Manufacturing application and display the Operation Navigator.

☐ Open the file **ama_hole_making_ipw**.

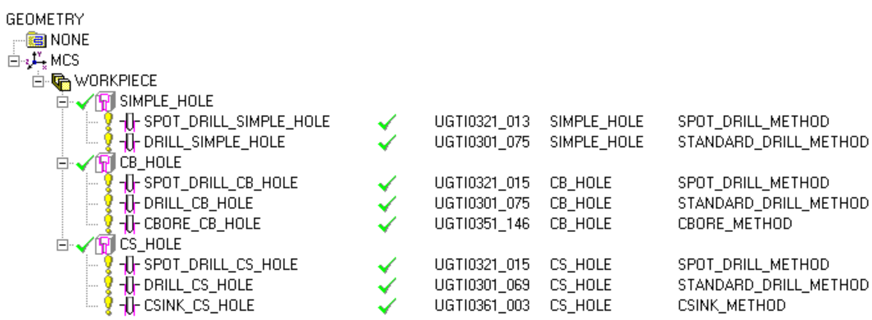
☐ Briefly examine the part.



☐ Choose **Application→Manufacturing** from the menu bar.

☐ Choose the **Operation Navigator** from the resource bar.

☐ If necessary, change to the Geometry view of the Operation Navigator and expand the group objects.

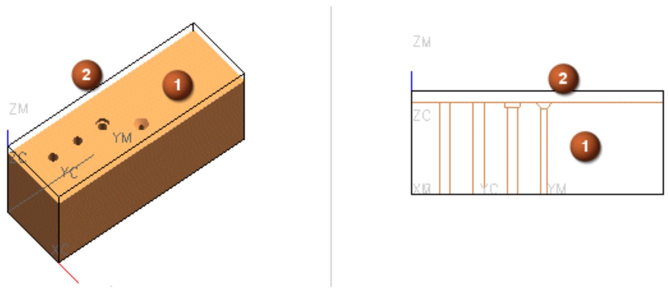


You will notice that operations exist to spot drill, drill, counter bore and counter sink the holes in the part. Next you will verify the part and blank material and will replay the operation SPOT_DRILL_SIMPLE_HOLE.

 ©UGS Corp., All Rights Reserved

Step 2: 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.

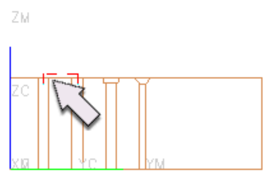


The operation recognizes Blank geometry only if the IPW is utilized. In this example, the Blank geometry, which extends above the actual Part surface, in not recognized and the tool collides with the Blank material.

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.**Use 3D IPW** option must be customized into each HOLE_MAKING operation dialog.

Step 3: Incorporate and use the IPW.

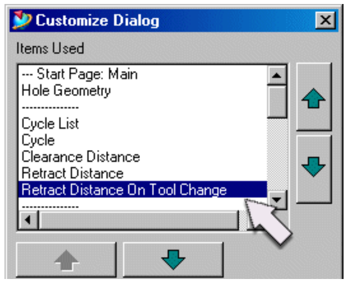
☐ Change to the Program Order view of the Operation Navigator and if necessary expand all objects.

☐ Highlight the **SPOT_DRILL_SIMPLE_HOLE** operation and using MB3, select **Object→Customize**.

The Customize Dialog is displayed.

©UGS Corp., All Rights Reserved

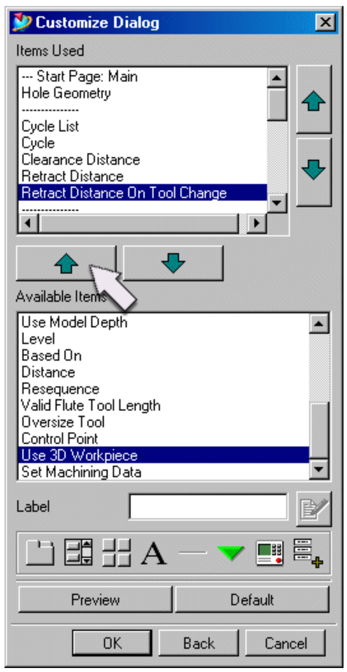
☐ Highlight **Retract Distance On Tool Change** from the **Items Used List**.



This will be the area of the Hole_Making dialog in which the Use 3D IPW option will appear.

☐ Highlight **Use 3D Workpiece** from the **Available Items** list.

☐ Select the **ADD** arrow to move the **Use 3D Workpiece** option into the **Items Used** list.



☐ Choose **OK** to accept the Customize Dialog.

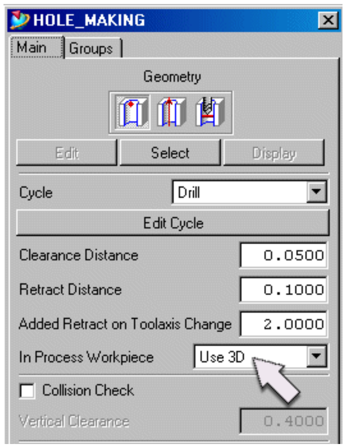
You will now activate the Use 3D IPW that will enable this operation to use the IPW.

☐ In the Operation Navigator, double-click on the **SPOT_DRILL_SIMPLE_HOLE** operation.

The HOLE_MAKING dialog is displayed.

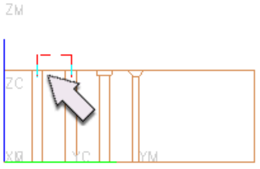
 ©UGS Corp., All Rights Reserved

☐ Set the **In Process Workpiece** option to **Use 3D**.



You will now generate the operation.

☐ Choose the **Generate** icon.



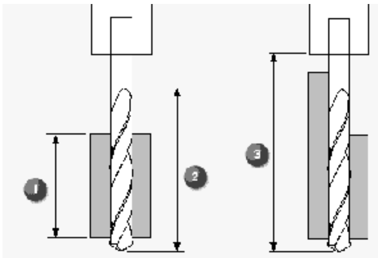
Notice how the tool retracts to a higher position when moving from one hole to the next.

☐ Choose **OK** on the HOLE_MAKING dialog.

☐ **Close**, without saving the part file.

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

Check Flute Length and Check Tool Length must be customized into the HOLE_MAKING dialog.

 ©UGS Corp., All Rights Reserved

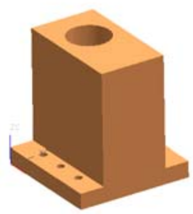
Activity: Maximum Cut Depth and Extended Length

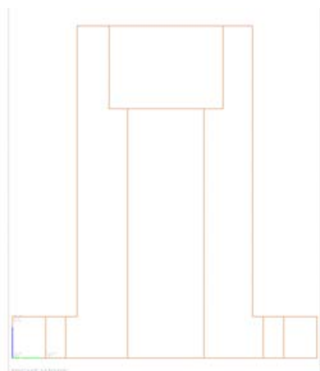
In this activity you will first examine a Hole Making operation and observe that the tool holder will collide with the work piece since the tool does not have enough length to drill through the part. You will then search the tool library for tools with appropriate tool lengths that can perform the required operation without colliding with the part.

Step 1: Open the part, enter the Manufacturing Application and display the Operation Navigator.

☐ Open the part **ama_hole_making_mx_ct_dp**.

☐ Briefly examine the part.





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

☐ Choose the **Operation Navigator** icon from the resource bar.

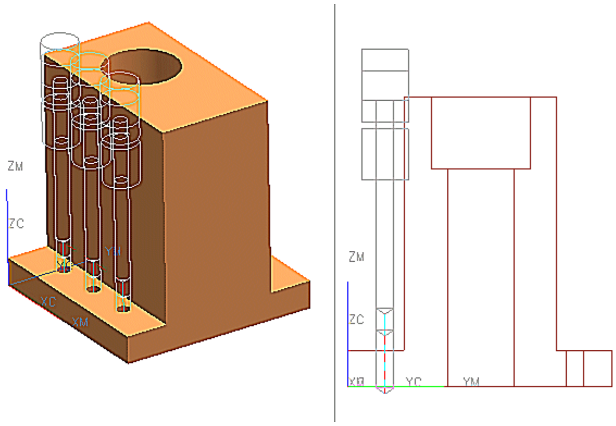
☐ If necessary, change to the Geometry view of the Operation Navigator and expand the group objects.

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 2: Replay the DRILL_SIMPLE_HOLE object.

☐ Highlight the **DRILL_SIMPLE_HOLE** operation, using MB3, Replay the operation.

©UGS Corp., All Rights Reserved



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.

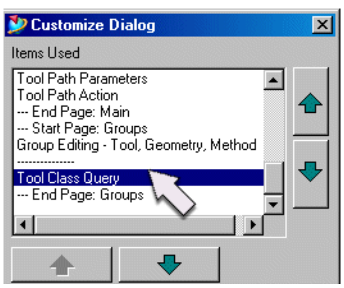
You will now use the Check Flute Length and Check Tool Length options to search the tool library for proper length tools.

Check Flute Length and Check Tool Length option must be customized into each HOLE_MAKING operation dialog.

Step 3: Incorporate and use the Check Flute Length and Check Tool Length options.

☐ Highlight the **DRILL_SIMPLE_HOLE** operation, and use **MB3→Object→Customize**.

☐ Highlight **Tool Class Query** from the **Items Used** List of the Customize Dialog.

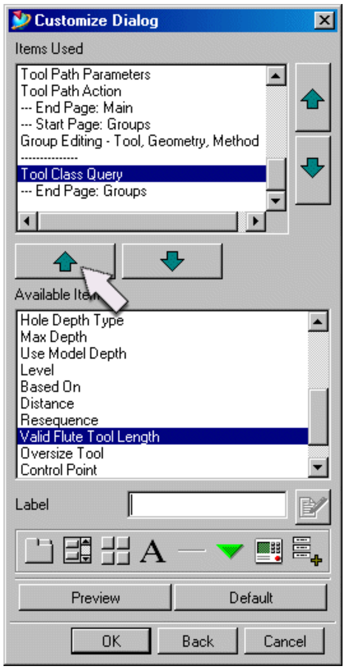


This will be the area of the Hole_Making dialog in which the **Check Flute Length** and **Check Tool Length** option will appear.

☐ Highlight **Valid Flute Tool Length** from the **Available Items** list.

 ©UGS Corp., All Rights Reserved

☐ Choose the **ADD** arrow.



☐ 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.

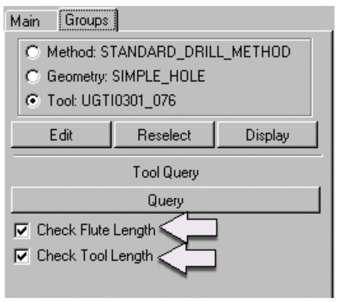
☐ From the Operation Navigator, double-click on the **DRILL_SIMPLE_HOLE** operation.

The HOLE_MAKING dialog is displayed.

☐ Choose the Groups tab at the top of the HOLE_MAKING dialog.

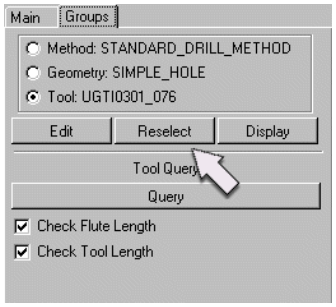
The Check Flute Length and Check Tool Length options are turned off. You will turn these options on and then query the library for the proper tooling.

☐ Turn the Check Flute Length and Check Tool Length options on.

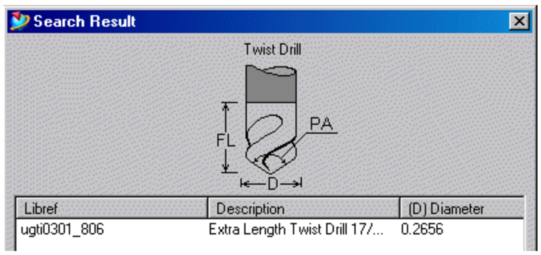


©UGS Corp., All Rights Reserved

☐ Choose **Reselect**.



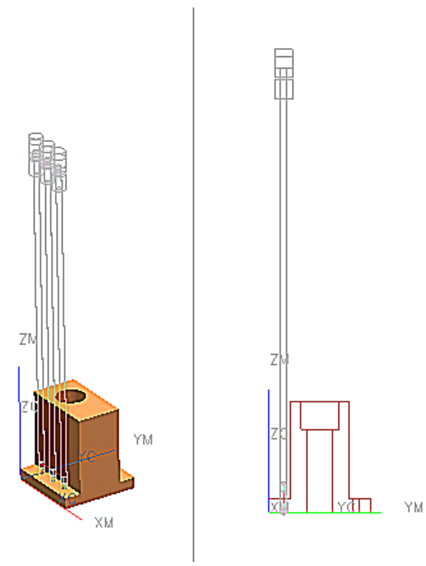
The Search Result dialog is displayed with the proper tool.



☐ Highlight 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.

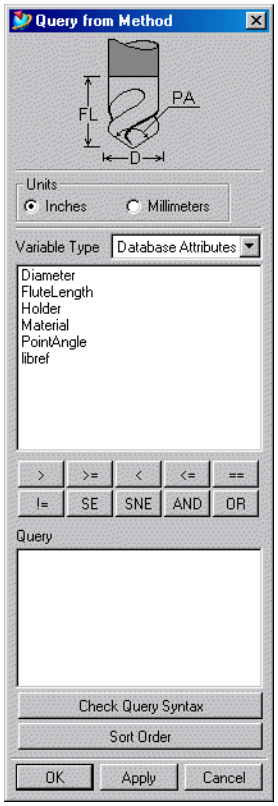
 ©UGS Corp., All Rights Reserved

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.

 ©UGS Corp., All Rights Reserved

Summary

Hole Making is an advanced application that automates the creation of operations such as spot drilling, drilling, countersinking, counter boring, 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 counter bore 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.

Lesson

9 Integrated Simulation and Verification

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.

Objective

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.

©UGS Corp., All Rights Reserved

Integrated Simulation and Verification Overview

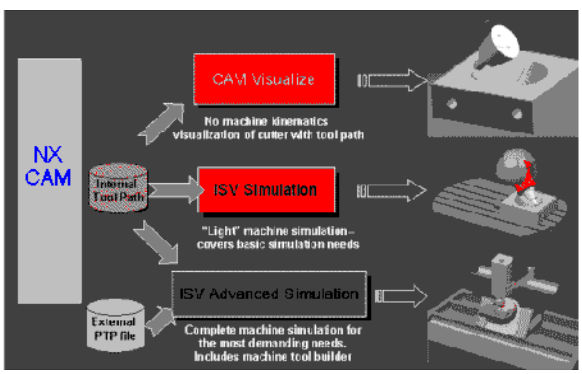
The Integrated Simulation and Verification module (IS&V) allows you to simulate a machine tool with an actual piece part, giving you an overview of the entire machining process. The simulation process animates the exact machine tool motions, taking into account controller functions and cutting tool configurations.

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



 ©UGS Corp., All Rights Reserved

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

©UGS Corp., All Rights Reserved

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

 ©UGS Corp., All Rights Reserved

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 Post Builder)
- provides the ability to simulate existing machine G & M codes (reverse postprocessing)

©UGS Corp., All Rights Reserved

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

 ©UGS Corp., All Rights Reserved

Activity: Using Simulation

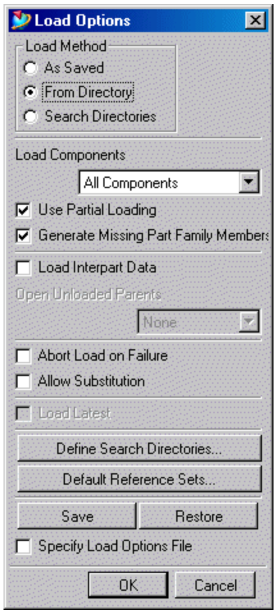
In this activity you will become familiar with some of the components used in the simulation process. You will execute a simulation of sample components supplied with this release of NX. For the simulation of the sample part to work correctly, you must first define the search directory of where the files are located.

Step 1: Define a new Search Directory for part retrieval.

☐ If necessary, start NX.

☐ Choose **File→Options→Load Options**.

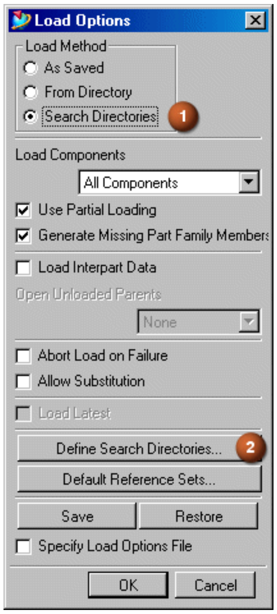
The **Load Options** dialog is displayed.



☐ If necessary, choose the **Search Directories (1)** option.

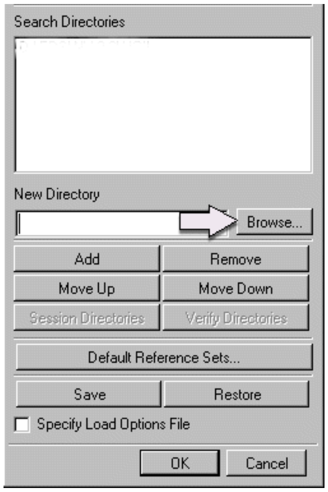
©UGS Corp., All Rights Reserved

☐ Select the **Define Search Directories (2)** button.



An updated Load Options dialog is displayed.

☐ Select the **Browse** button.



☐ Filter to your home **NX** directory and then the **MACH** subdirectory.

 ©UGS Corp., All Rights Reserved

☐ Choose **OK** to the **Choose Search Directory**.

The directory name will appear in the "New Directory" field.

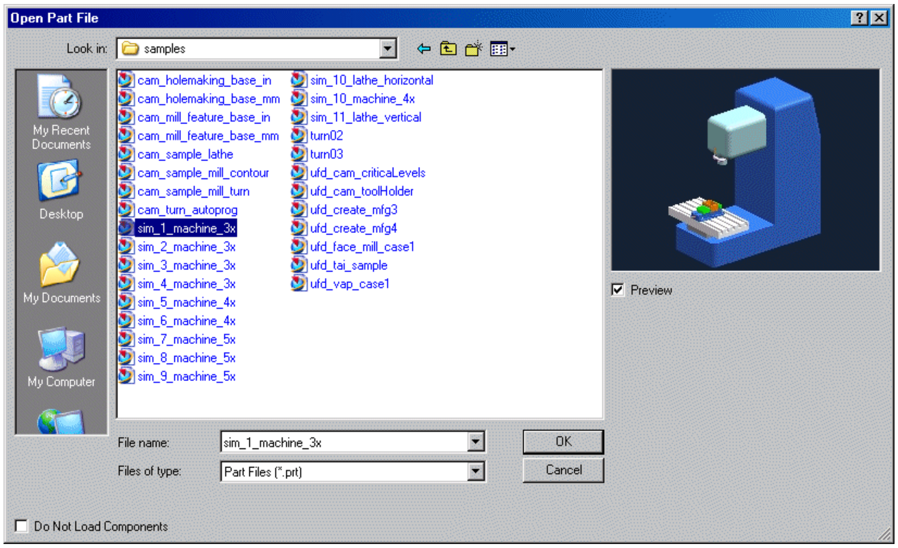
☐ Focus on the "New Directory" field, add three period characters to the end of the field and then select the Enter key from the keyboard.

☐ 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.

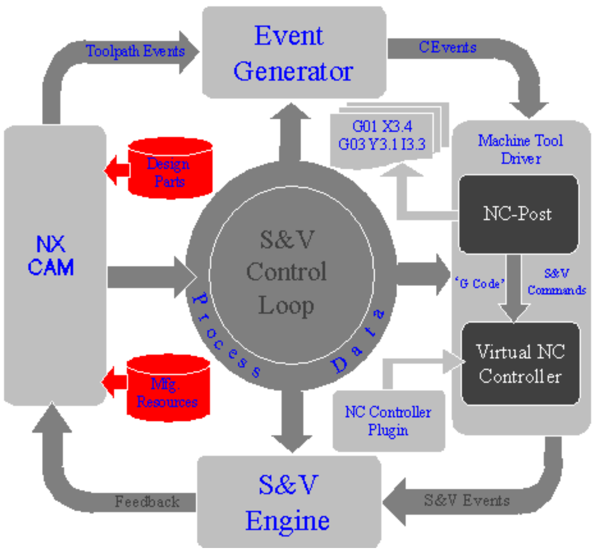
☐ If necessary, switch to the **Program Order** view of the Operation Navigator.

- ☐ Using **MB3** on the **NC Program** parent, select **Tool path→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_11_lathe_vertical** part and repeat the simulation process.
- ☐ **Close** all parts without saving.

 ©UGS Corp., All Rights Reserved

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



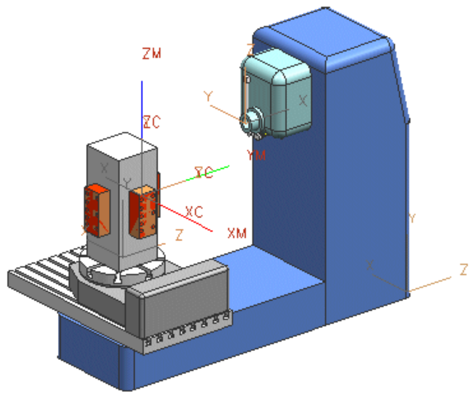
The **Machine Tool Driver (MTD)** creates the CNC program that emulates the CNC controller. The CNC controller emulator (or Virtual NC controller) is a programmable interface that instructs the machine tool model on actual movements and how those movements are displayed. Any motion and feedback displayed during machine tool simulation is controlled by the dedicated MTD.

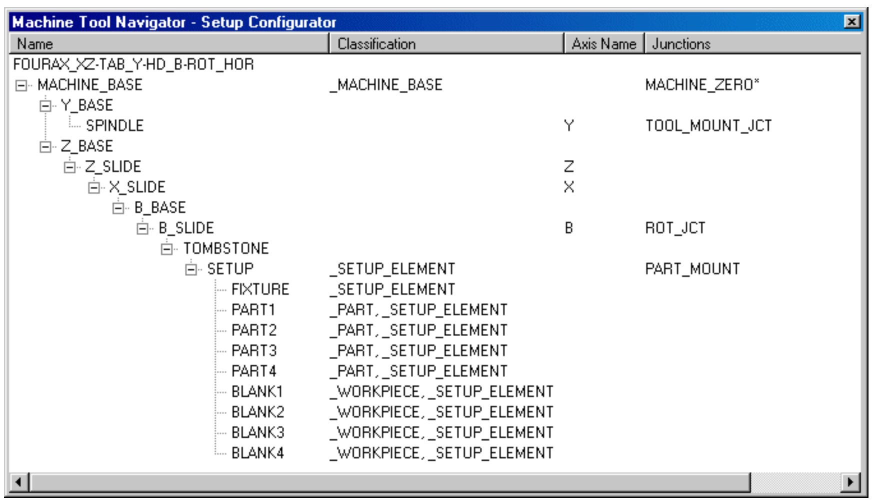
For comparison purposes, the MTD is analogous to the machine tool simulator as the CNC controller is analogous to the machine tool that it controls. For each machine tool in the machine tool library, there is a MTD driver available (twelve generic MTD's come standard with NX). For creating an MTD for a new machine tool, you can modify a generic driver to work with that machine. MTD's are written in 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.

©UGS Corp., All Rights Reserved

Setup Configurator

- similar functionality to Machine Tool Builder
- used for mounting work piece and fixtures to machine tool
- used for defining machine state





Machine Tool Navigator - Setup Configurator

Name	Classification	Axis Name	Junctions
FOURAX_XZ-TAB_Y-HD_B-ROT_HOR			
MACHINE_BASE	_MACHINE_BASE		MACHINE_ZERO*
Y_BASE			
SPINDLE		Y	TOOL_MOUNT_JCT
Z_BASE			
Z_SLIDE		Z	
X_SLIDE		X	
B_BASE			
B_SLIDE		B	ROT_JCT
TOMBSTONE			
SETUP	_SETUP_ELEMENT		PART_MOUNT
FIXTURE	_SETUP_ELEMENT		
PART1	_PART, _SETUP_ELEMENT		
PART2	_PART, _SETUP_ELEMENT		
PART3	_PART, _SETUP_ELEMENT		
PART4	_PART, _SETUP_ELEMENT		
BLANK1	_WORKPIECE, _SETUP_ELEMENT		
BLANK2	_WORKPIECE, _SETUP_ELEMENT		
BLANK3	_WORKPIECE, _SETUP_ELEMENT		
BLANK4	_WORKPIECE, _SETUP_ELEMENT		

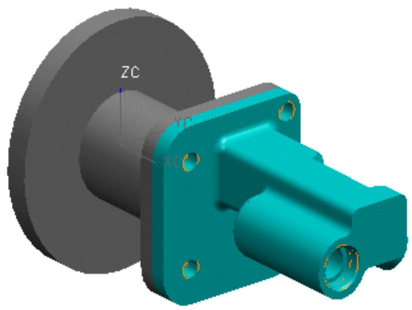
 ©UGS Corp., All Rights Reserved

Activity: 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 then simulate a machining operation.

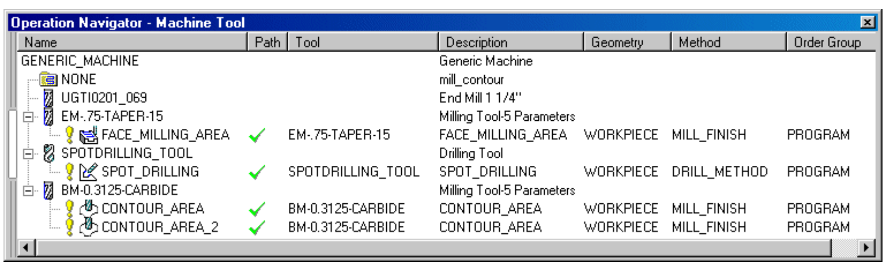
Step 1: Opening the part file ama_simulate.

- ☐ If necessary, start NX.
- ☐ Use **File→Open**.
- ☐ Navigate to your parts folder and open the file.
- ☐ Choose File →Save as *****_simulate** 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.



Operation Navigator - Machine Tool

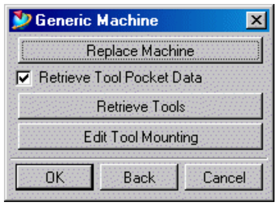
Name	Path	Tool	Description	Geometry	Method	Order Group
GENERIC_MACHINE			Generic Machine			
NONE			mill_contour			
UGTI0201_069			End Mill 1 1/4"			
EM-.75-TAPER-15			Milling Tool-5 Parameters			
FACE_MILLING_AREA	✓	EM-.75-TAPER-15	FACE_MILLING_AREA	WORKPIECE	MILL_FINISH	PROGRAM
SPOTDRILLING_TOOL			Drilling Tool			
SPOT_DRILLING	✓	SPOTDRILLING_TOOL	SPOT_DRILLING	WORKPIECE	DRILL_METHOD	PROGRAM
BM-0.3125-CARBIDE			Milling Tool-5 Parameters			
CONTOUR_AREA	✓	BM-0.3125-CARBIDE	CONTOUR_AREA	WORKPIECE	MILL_FINISH	PROGRAM
CONTOUR_AREA_2	✓	BM-0.3125-CARBIDE	CONTOUR_AREA	WORKPIECE	MILL_FINISH	PROGRAM

©UGS Corp., All Rights Reserved

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** group object.

The Search Result list is displayed, showing the various machines available for selection.

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.

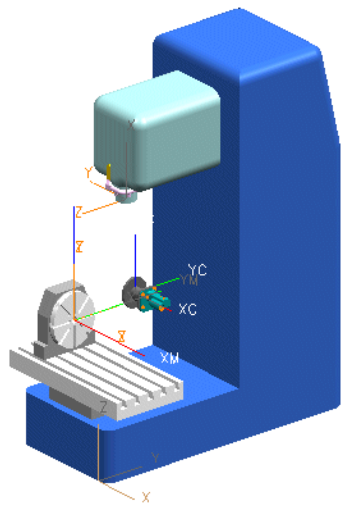
☐ Choose the 4-axis vertical milling machine, **sim010101_005_in** from the Search Result list.

☐ Choose **OK** on the Search Result list.

☐ Choose **OK** on the **Generic Machine** dialog.

 ©UGS Corp., All Rights Reserved

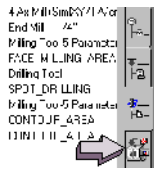
☐ With the cursor in the graphic window, use MB3, **Fit** to size the view to the graphics window.



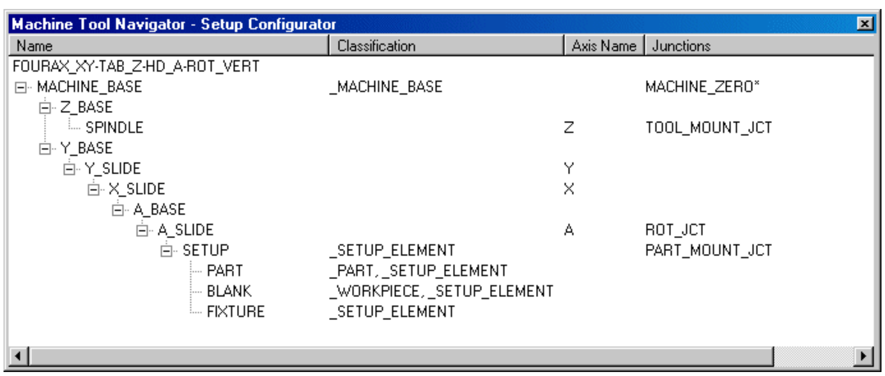
The machine tool, **sim010101_005_in** has been loaded. You must now mount the work piece to the machine. This is accomplished by using the **Machine Tool Navigator**.

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.

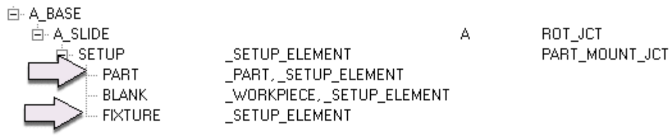


Machine Tool Navigator - Setup Configurator

Name	Classification	Axis Name	Junctions
FOURAX_XY-TAB_Z-HD_A-ROT_VERT			
MACHINE_BASE	_MACHINE_BASE		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		

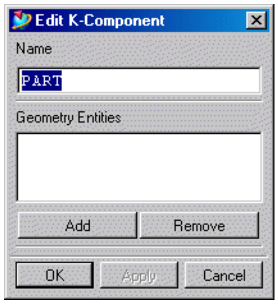
©UGS Corp., All Rights Reserved

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.

☐ Select the part geometry.

☐ Choose **OK** in the Class Selection widget.

☐ 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.

☐ Select the fixture geometry.

☐ Choose **OK** in the Class Selection widget.

☐ 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**.

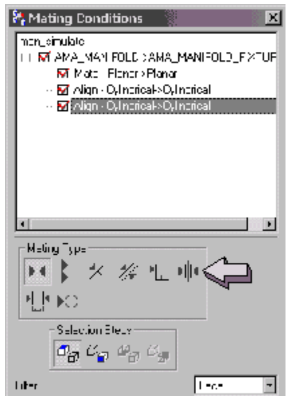
☐ Verify that the **Assemblies** application is turned on.

☐ Choose **Assemblies** → **Components** → **Mate Components**.

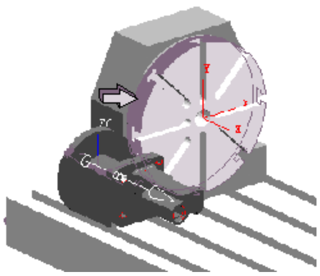
 ©UGS Corp., All Rights Reserved

The Mating Conditions dialog is displayed.

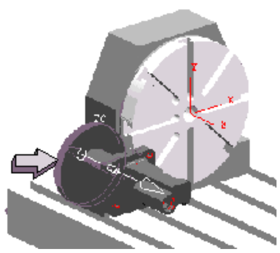
☐ Choose the icon representing the center **Mating Type**.



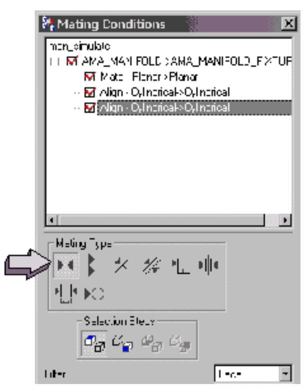
☐ Choose the cylindrical face of the headstock as the **FROM** face.



☐ Choose the cylindrical face of the fixture as the **TO** face.

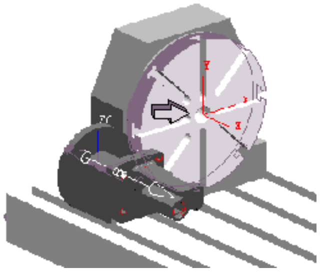


☐ Choose the **Mating Type**, **Mate** icon.

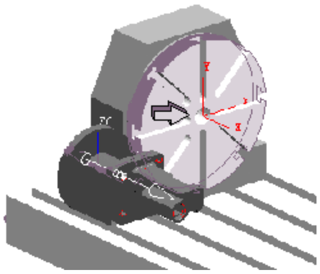


©UGS Corp., All Rights Reserved

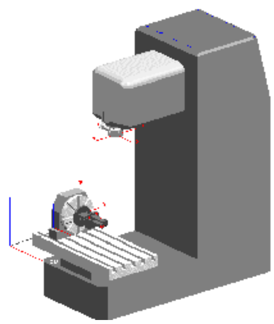
☐ Choose the planar face of the headstock as the **FROM** face.



☐ 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.

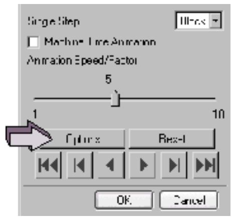
☐ Using MB3, choose **Tool Path** → **Simulate**.

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.

 ©UGS Corp., All Rights Reserved



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.

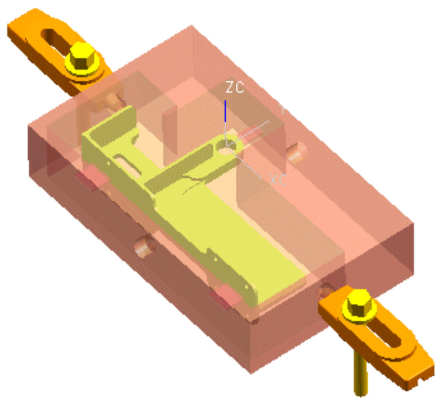
©UGS Corp., All Rights Reserved

Activity: Using a pre-built Simulation Machine

In this activity, you will use an existing Simulation Machine to simulate an NC program. NX has numerous styles of pre-built machines ready for use.

Step 1: Opening the part file and examine the NC program.

☐ Open the part file **ama_isv_support_mfg**.



This part is a manufacturing assembly of an aerospace support strut. Due to the low quantity required, a solid block of aluminum will be used for stock. The ama_isv_support_frame component serves as a window frame around the part to hold it in place.

☐ Enter the **Manufacturing** application.

☐ Display the **Program Order** view of the Operation Navigator.

☐ Highlight the **SIDE_1** operation, use MB3, **Tool path→Verify**.

☐ Choose the **3D Dynamic** tab on the Tool Path Visualization dialog.

☐ Choose the **Play** button at the bottom of the dialog.

The tool path is rapidly displayed. It is difficult to determine if any collisions have occurred. You will use the IS&V simulation tool to examine the object in greater detail.

☐ Cancel the Tool Path Visualization dialog.

Step 2: Choose the Machine Tool for Simulation.

NX contains several pre-built IS&V machines in a library. You will choose one of these machines to run your simulation.

 ©UGS Corp., All Rights Reserved

☐ Change to the **Machine Tool** view of the Operation Navigator.

☐ Highlight the **GENERIC_MACHINE** group object, use MB3, choose **Edit**.

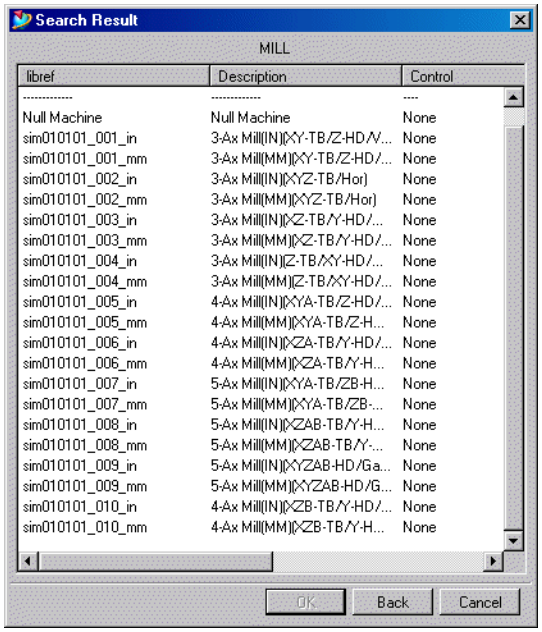
The Generic Machine dialog is displayed.

☐ Choose the **Replace Machine** button.

The Library Class Selection dialog displays.

☐ Double-click on the **MILL** group object.

The Search Result list is displayed, showing the various machines available for selection.



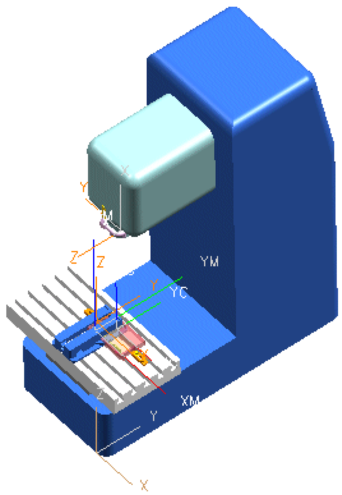
Any item that begins with “sim” in the name is a ready made simulation machine.

☐ Choose **sim010101_001_in** from the Search Result list.

☐ 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.



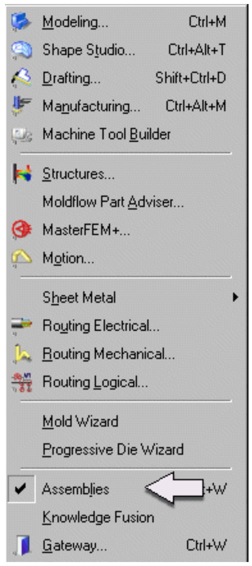
The machine tool, **sim010101_001_in** has been loaded and is ready for use. Before testing can begin, you must specify the relationship between the setup and the machine tool.

Step 3: Mount the manufacturing assembly to the machine tool.

Mounting a fixture or manufacturing assembly to a machine tool can be accomplished in one of two ways. Option 1 is to place the setup on the machine in a fixed location using Assemblies; option 2 is to use a Part Mount Junction. In this activity you will use the Assemblies option.

 ©UGS Corp., All Rights Reserved

☐ Verify the Assemblies application is on by choosing Application from the main menu bar. There should be a check mark next to Assemblies, if not turn the option on by clicking it.



This machine tool has a vice that will not be used and will be removed from the display.

☐ Choose the Assembly Navigator tab.

☐ In the Assembly Navigator, use MB3, **Expand All**.

☐ Choose the red check mark next to **vise_body_3** to remove from the display.

☐ Choose the red check mark next to **vise_jaw_3** to remove from the display.

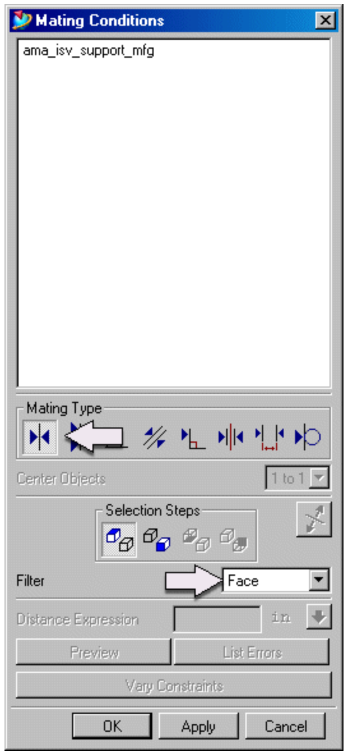
You will now mate the setup to the machine.

☐ Choose **Assemblies→Components→Mate Component** (or select the Mate Components icon on the Assemblies toolbar).

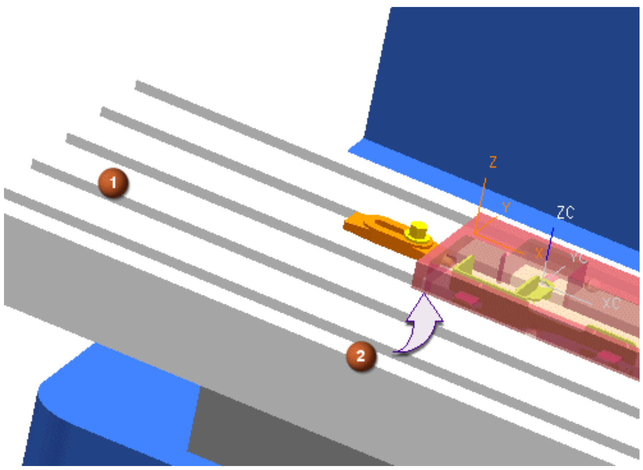
To perform the mating properly, you will mate the machine to the setup, rather than mate the setup to the machine. This method will maintain the correct MCS/WCS relationship.

©UGS Corp., All Rights Reserved

☐ Verify the **Mating_Type** is set to **Mate** and the **Filter** is set to **Face**.



☐ Choose the top face of the machine table as the **From (1)** object, and the bottom face of the blank as the **To (2)** object.

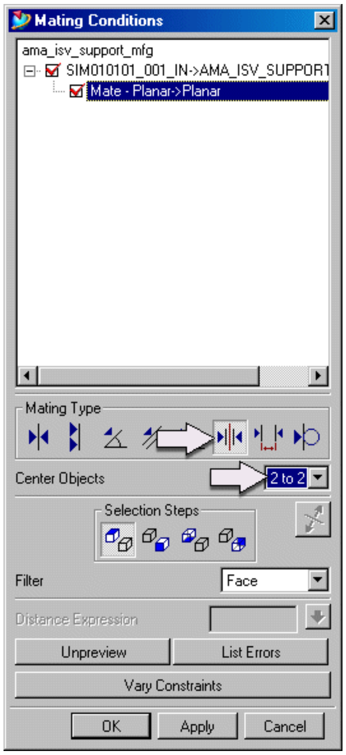


☐ Choose **Preview**.

The machining center should move until the mating condition is applied. You will now center the setup in the X-axis.

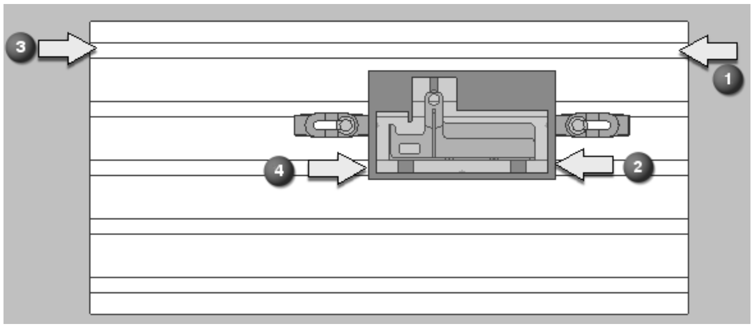
 ©UGS Corp., All Rights Reserved

☐ Choose the **Center** icon and set the **Center Objects** at **2 to 2**.



☐ Choose the front face of the table as the **First From (1)** object and then the front face of the blank as the **First To (2)** object.

☐ Choose the back face of the table as the **Second From (3)** object and the back face of the blank as the **Second To (4)** object.



☐ Use the **Preview** button to view the results.

The Status line should indicate that the mating condition is fully constrained.

Step 4: Assign the Setup, Blank and Fixture Elements.

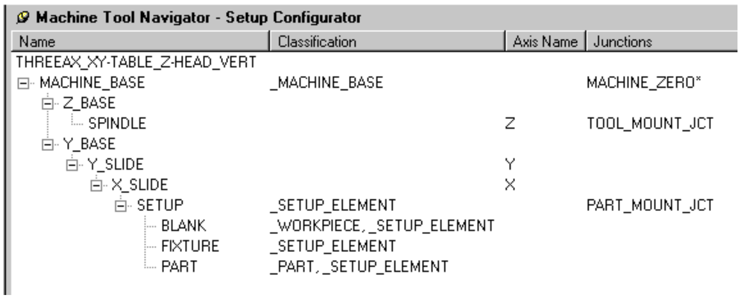
©UGS Corp., All Rights Reserved

Before testing can begin, the components of the assembly representing the designed part, the fixture and the raw stock must be defined.

☐ If necessary, enter the Manufacturing application.

☐ Select the **Machine Tool Navigator** tab.

☐ Use MB3, in the **Machine Tool Navigator** to **Expand All**.



Machine Tool Navigator - Setup Configurator

Name	Classification	Axis Name	Junctions
THREEAX_XY-TABLE_Z-HEAD_VERT			
MACHINE_BASE	_MACHINE_BASE		MACHINE_ZERO*
Z_BASE			
SPINDLE		Z	TOOL_MOUNT_JCT
Y_BASE			
Y_SLIDE		Y	
X_SLIDE		X	
SETUP	_SETUP_ELEMENT		PART_MOUNT_JCT
BLANK	_WORKPIECE, _SETUP_ELEMENT		
FIXTURE	_SETUP_ELEMENT		
PART	_PART, _SETUP_ELEMENT		

Examine the tree structure of the machine tool. Notice that it describes the various elements that comprise the machine itself. The SETUP sits on the X_SLIDE, which resides on the Y_SLIDE. Notice the relationship of the SETUP and the components that reside within it.

☐ Double click on the **Part** element.

The Edit K-Component dialog is displayed.

☐ Choose the **Add** button.

☐ Choose the part **isv_support**, then choose **OK** until the Edit K-Component dialog is no longer available.

☐ Double click on the **Fixture** element in the Machine Tool Navigator.

☐ Remove the two existing components by highlighting them and then choosing **Remove**.

☐ Choose the **Add** button.

☐ Select the two clamps and two bolts, then choose **OK** until the Edit K-Component dialog is no longer available.

☐ Double click on the **Blank** element in the Machine Tool Navigator.

☐ Choose the **Add** button.

 ©UGS Corp., All Rights Reserved

☐ Select the translucent blank body, **isv_support_stock** and the window frame, **window_frame**, then choose OK until the Edit K-Component dialog is no longer available.

You are now ready to begin the simulation process.

Step 5: Simulate the Machining Process.

☐ Close the Machine Navigator dialog.

☐ Change to the **Program Order** view in the Operation Navigator.

☐ Highlight the **SIDE_1** object, use MB3, **Tool Path**→ **Simulate**.

☐ Choose the **Play** button on the Simulation Control Panel dialog.

☐ Choose **OK** when finished.

☐ **File**→**Close** all parts.

This completes the activity.

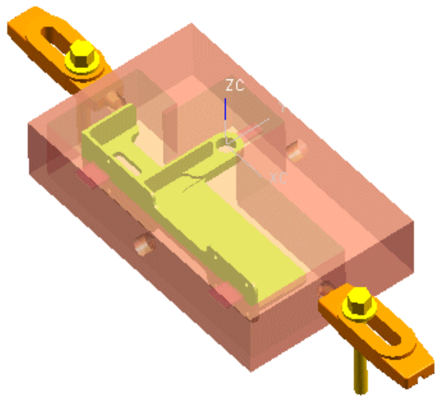
©UGS Corp., All Rights Reserved

Activity: Using the Part Mount Junction

In the previous activity you used the concept of Assemblies to mount the setup onto the machine tool. In this activity you will create a Part Mount Junction to locate the manufacturing setup.

Step 1: Opening the part file.

☐ Open the part file **ama_isv_support_mfg**.



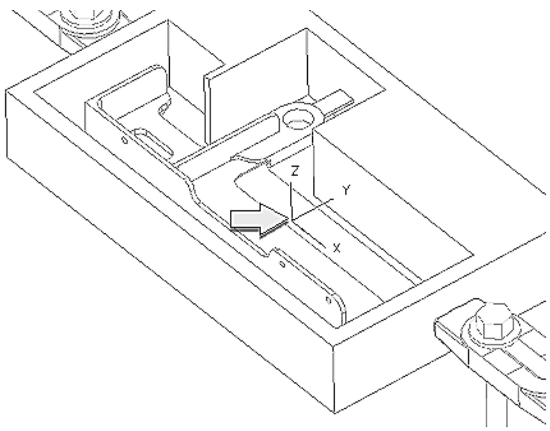
☐ Enter the **Manufacturing** application.

Step 2: Create a Part Mount Junction.

You must specify the Part Mount Junction on the setup. You must first move the WCS to the position where the Part Mount Junction will reside.

☐ Make Layer 51 the Work Layer.

☐ Move the WCS to the saved CSYS as shown.

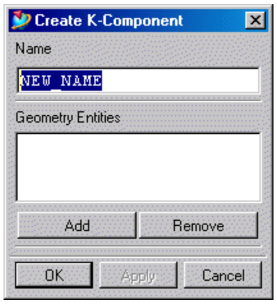


☐ Open the **Machine Tool** navigator.

 ©UGS Corp., All Rights Reserved

☐ Highlight the **NO_NAME** object, use MB3, **Insert→ K-Component**.

The Create K-Component dialog is displayed.



☐ In the Name field key in **setup**.

☐ Choose **OK** in the Create K-Component dialog.

☐ Use MB3 on the **setup** object and choose **Junctions**.

The Junctions dialog is displayed.

☐ Choose **Add**.

☐ Change the name to **Part_Mount_jct**.

☐ Choose **Define Coordinate System**.

Since the WCS is already in the proper location, no movement or manipulation of the CSYS is required.

☐ Choose **OK** in the CSYS dialog.

☐ Choose **OK** in the Edit Junction dialog.

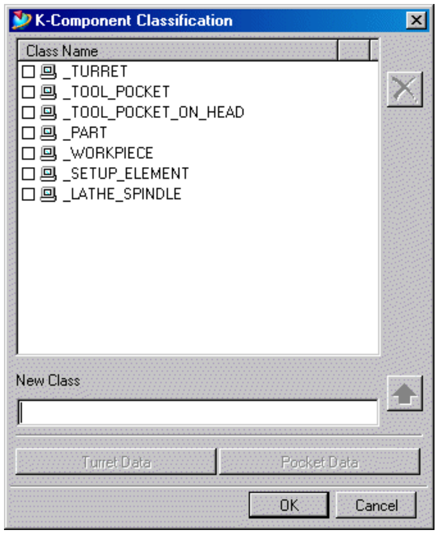
☐ Choose **OK** in the Junctions dialog.

Step 3: Define the Setup Geometry.

☐ In the Machine Tool Navigator, use MB3 on the **SETUP** object and choose **Classify**.

©UGS Corp., All Rights Reserved

The K-Component Classification dialog is displayed.



- ☐ Select the **_SETUP_ELEMENT** component.
- ☐ Choose **OK**.
- ☐ In the Machine Tool Navigator dialog, use MB3 on the **SETUP** object and choose **Insert→K-Component**.
- ☐ Key in **part** as the **Name**.
- ☐ Choose the **ADD** button.
- ☐ Choose all of the **setup** geometry (part, blank, clamps).
- ☐ Choose **OK**.
- ☐ In the Machine Tool Navigator dialog, use MB3 on the **Part** object and choose **Classify**.
- ☐ Turn on the check mark adjacent to the **_PART** element.
- ☐ Choose **OK**.

Step 4: Load the Machine Tool.

The final step is to load the machine tool from the library.

- ☐ Change to the **Machine Tool** view in the Operation Navigator.
- ☐ Highlight the **GENERIC_MACHINE** object, use MB3, choose **EDIT**.

  The Generic Machine dialog is displayed.

 ©UGS Corp., All Rights Reserved

☐ Choose the **Replace Machine** button.

The Library Class Selection dialog is displayed.

☐ Choose the **MILL** object, then choose **OK**.

☐ From the Search Result list, choose **sim010101_001_in**, then choose **OK**.

The Setup Replacement dialog is displayed. This dialog indicates the presence of a Part Mount Junction and allows the option to use if so desired.

☐ If necessary, select the **Position Machine on to Setup** radio button.

☐ Choose **OK**.

The Setup is placed on top of the vise, on the machine table.

☐ Choose **OK** in the Generic Machine dialog.

Step 5: Simulate the Program.

☐ Close the Machine Navigator dialog.

☐ Change to the **Program Order** view in the Operation Navigator.

☐ Highlight the **SIDE_1** object, use MB3, **Tool Path**→ **Simulate**.

☐ Choose the **Play** button on the Simulation Control Panel dialog.

☐ Choose **OK** when finished.

☐ **File**→**Close** all parts.

This completes the activity and the lesson.

©UGS Corp., All Rights Reserved

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.
- Used Mating Conditions to mount the setup to the machine.
- Used a Part Junction to mount the setup to the machine.
- Ran simulation on a sample part.

 ©UGS Corp., All Rights Reserved

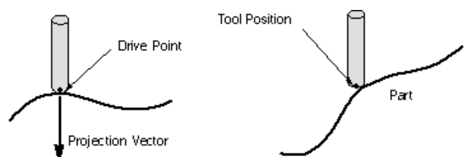
Appendix

A Advanced Surface Contouring

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

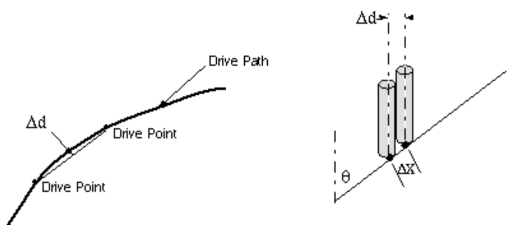


Projection and Steep Surface:

- $\Delta X = \Delta d / \sin\Theta \cong \Delta d / \Theta$

  ΔX becomes large if Θ is very small (steep surface)

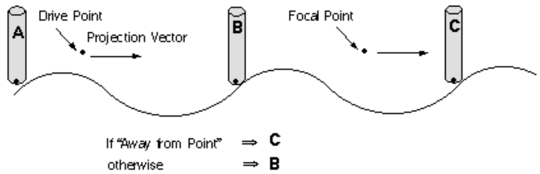
- The source of Δd is the chordal deviation of the drive path



©UGS Corp., All Rights Reserved

Projection and Material Side:

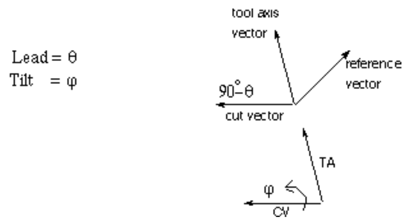
- Surface contouring does not have explicit definition of material side for part geometry, only the drive surface has explicit material side
- Material side of the part is determined implicitly by the projection vector



- In the case of Area Milling Drive (no projection vector), the tool axis vector is used to decide Material Side

Tool Axis

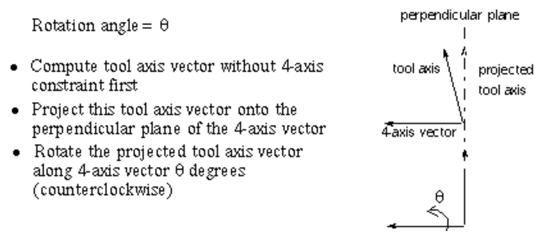
Definition of Lead/Tilt angles:



- Begin with cut vector, rotate it toward the Reference vector 90°- Θ 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

 ©UGS Corp., All Rights Reserved

Definition of 4-axis rotation angle:

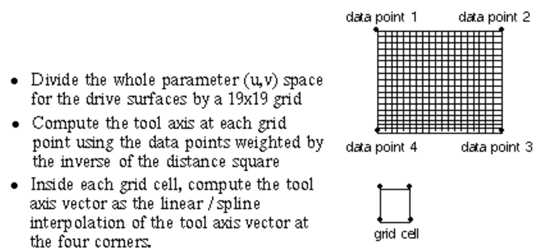


•

The unconstrained tool axis vector could be:

- Normal to Part / Drive
- Relative to Part / Drive

Interpolated tool axis algorithm:

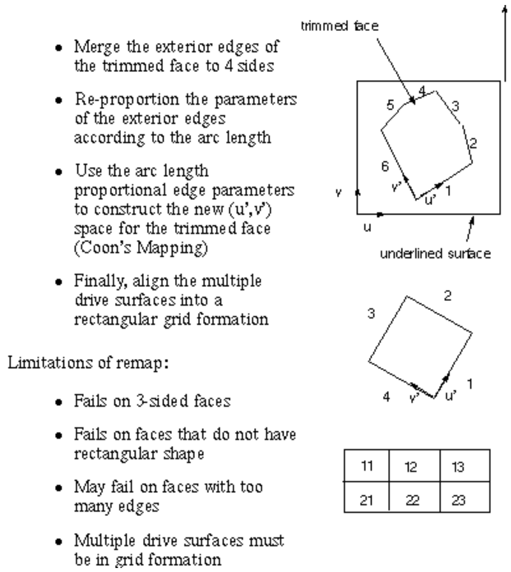


©UGS Corp., All Rights Reserved

Drive Surface

Remap of drive surface:

Remap algorithm:



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

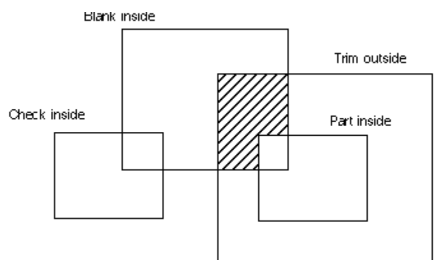
Planar Milling

- 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



 ©UGS Corp., All Rights Reserved

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

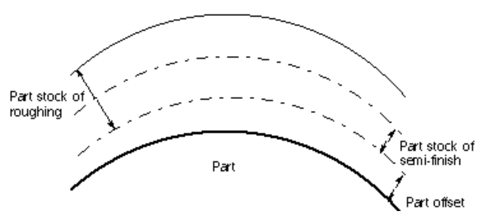
- Cut area - similar to "blank"
- Trim - behaves slightly different from planar milling

Stock

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

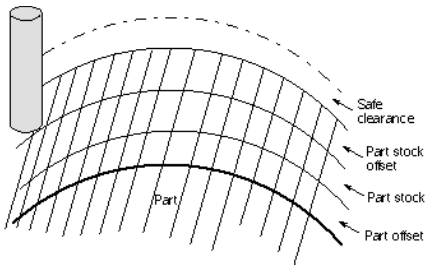


©UGS Corp., All Rights Reserved

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

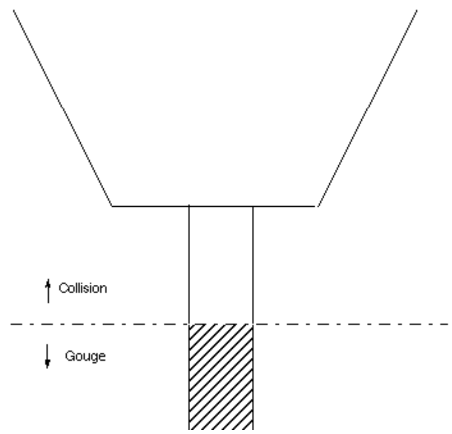
- Safe clearance is used in engage/retract and collision checking

Gouge / Collision

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



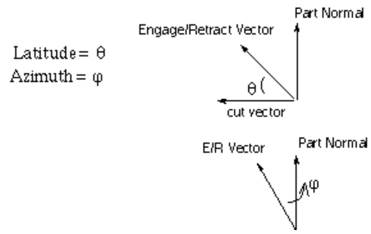
Usage:

	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)

©UGS Corp., All Rights Reserved

Noncut Moves

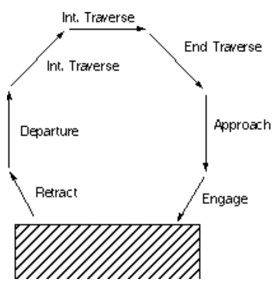
Azimuth / Latitude:



- 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

 ©UGS Corp., All Rights Reserved

Appendix

B Project using Advanced Features

Purpose

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 your 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.

Advanced Features

Step 1: Open the part file ama_manifold.

☐ 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**, from the **student_home\parts** directory.

☐ Choose **File** → **Save As** → *****_manifold_mfg** 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 are 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.

 ©UGS Corp., All Rights Reserved

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**.

☐ Choose **OK**.

The Select Part Name dialog is displayed.

☐ Key in *****_manifold_casting** as the new component part file name.

☐ Choose **OK**.

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.

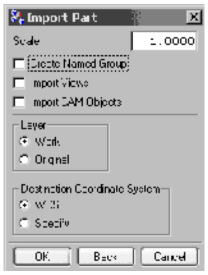
☐ In the Assembly Navigator, double-click on the "casting" component (***_manifold_casting).

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**.

©UGS Corp., All Rights Reserved

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** 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** 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.

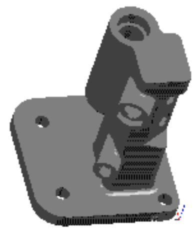
☐ In the Assembly Navigator, make *****_manifold_mfg** component the work part.

☐ **Save**, but do not close the part.

This concludes this part of the activity.

 ©UGS Corp., All Rights Reserved

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.



©UGS Corp., All Rights Reserved

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.

Step 1: Using the WAVE geometry linker.

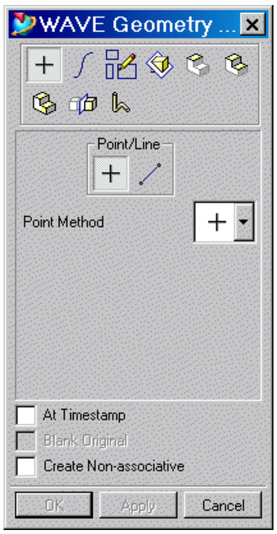
☐ Continue using *****_manifold_mfg** from Part 1 of the activity.

☐ In the Assembly Navigator, double-click on *****_manifold_casting** to make it the work part.

☐ Choose **Application** → **Modeling** from the main menu bar.

☐ Choose **Insert** → **Associative Copy** → **WAVE Geometry Linker** from the main menu bar.

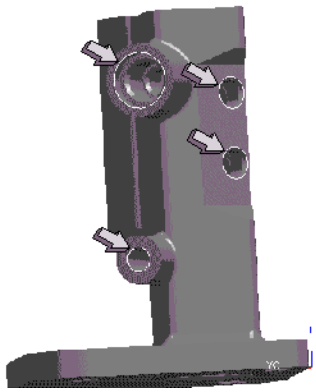
The WAVE Geometry Linker dialog is displayed.



☐ In the **WAVE Geometry Linker** dialog, select the **Curves** icon.

 ©UGS Corp., All Rights Reserved

☐ 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.

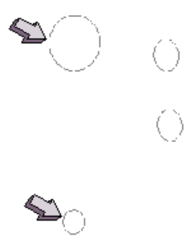
☐ From the Main menu bar, choose **Insert** → **Design Feature** → **Extrude**.

The Extruded Widget is displayed.

☐ Choose the **Extrude Dialog** icon.

☐ On the **Selection** toolbar change the type filter from **Any** to **Curve**.

☐ Select the two holes that are on the same plane, but dissimilar in size as shown.

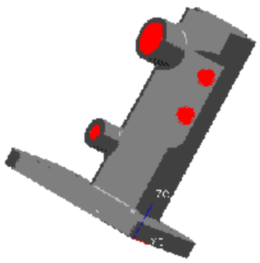


☐ Choose **OK** on the **Extrude** dialog.

☐ Repeat the procedure for the other two curves.

Step 3: Save the Assembly.

☐ In the Assembly Navigator, click on the check mark 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.

 ©UGS Corp., All Rights Reserved

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:

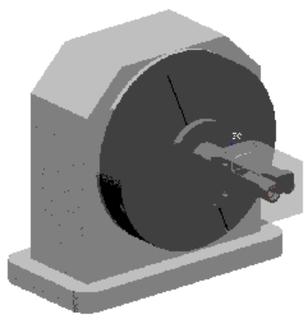
- the billet representing the raw stock has been created and added to the assembly
- a mounting plate for the part has been designed and added to the assembly
- the rotary axis of the machining center has been added to the assembly
- all components were mated and or positioned in the correct relationship with the manifold

You will start the process at the stage where the NC programming will begin.

Step 1: Open and rename the existing assembly.

☐ Open **ama_manifold_mfg** from your parts directory.

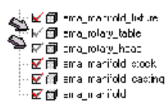
☐ Choose **File** → **Save As** → *****_manifold_mfg2**, where ***** represents your initials.



☐ Examine the assembly by opening the **Assembly Navigator**.

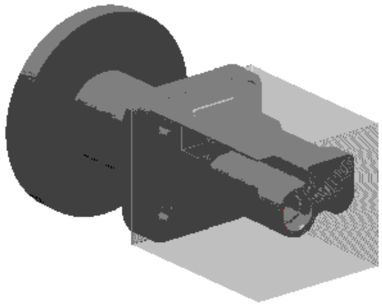
Since it is not necessary to see the rotary head or table, you will hide those objects by using the Assembly Navigator.

☐ Hide the display of the rotary table and head by clicking on the check marks next to them in the Assembly Navigator.



©UGS Corp., All Rights Reserved

☐ 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.

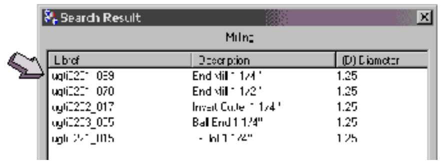
☐ Key in **1.25** in the **Diameter** field

☐ Choose **OK** on the **Search Criteria** dialog.

The Search Result listing is displayed.

 ©UGS Corp., All Rights Reserved

☐ 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 **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.

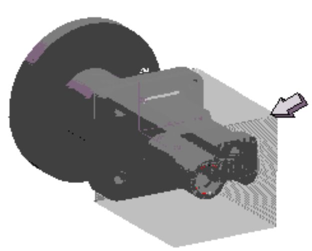
☐ In the **Geometry** view of the Operation Navigator, double-click on the **Workpiece** group object.

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.



©UGS Corp., All Rights Reserved

☐ Choose **OK** on the Blank Geometry dialog.

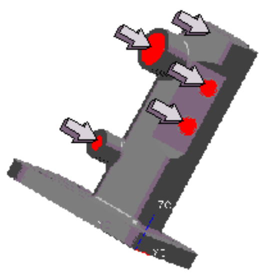
You will now select the part geometry.

☐ In the Assembly Navigator, choose the check mark 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 **Search Result** dialog.

☐ Choose **OK** in the **MILL_GEOM** dialog.

 ©UGS Corp., All Rights Reserved

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.

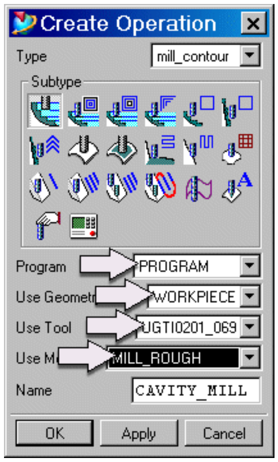
☐ From the main menu bar, choose **Insert** → **Operation**.

The Create Operation dialog is displayed.

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

☐ Choose the **CAVITY_MILLING** icon.

☐ 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 **Measured from** to **WCS Origin**.

☐ Change the **Global Depth Per Cut** to **.125**.

Change the **Range Depth** to **0.0**.

Recalculate the feed rate based on part material and cutter type.

☐ Choose the **Feed Rates** button.

☐ Choose **Reset from Table**.

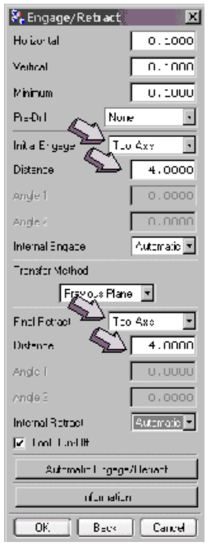
©UGS Corp., All Rights Reserved

☐ 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.

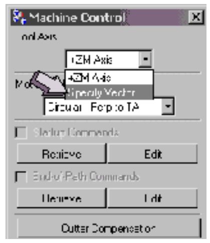
☐ Highlight the **CAVITY_MILL** operation.

 ©UGS Corp., All Rights Reserved

- ☐ 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 **Measured from** to **WCS Origin**.
- ☐ Key in **0.0** in the **Range Depth** field.
- ☐ Choose **OK**.
- ☐ Choose the **Cutting** button.
- ☐ Choose the **Containment** tab.
- ☐ 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 **Measured from** to **WCS** Origin.

 ©UGS Corp., All Rights Reserved

- ☐ Change the **Range Depth** to **0.0**.
- ☐ Choose **OK**.
- ☐ Choose the **Cutting** button.
- ☐ Choose the **Containment** tab.
- ☐ 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.

 ©UGS Corp., All Rights Reserved

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**.
- ☐ Choose **File** → **Save As** → *****_manifold_mfg_3**.
- ☐ 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 **3D 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**.

©UGS Corp., All Rights Reserved

- ☐ 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.

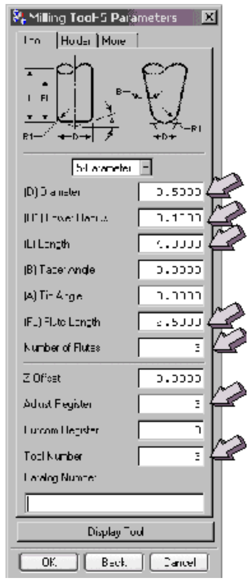
- ☐ Change the **Analysis Type** to **Fillet Radii**.
- ☐ Click and drag a rectangle around the part in the graphics window.
- ☐ Choose **Apply**.

  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**.

 ©UGS Corp., All Rights Reserved

☐ Key in the parameters as shown:



☐ Choose **OK**.

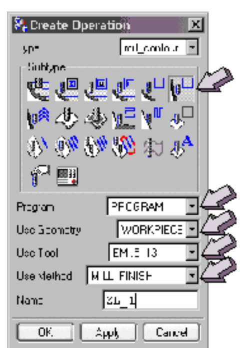
Step 4: Create a Z-Level Steep operation.

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.

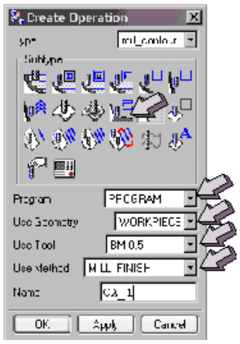
©UGS Corp., All Rights Reserved

- ☐ Change the **Steep Angle** to **50.**
- ☐ Choose the **Cut Levels** button.
- ☐ In the Cut Levels dialog, change the **Measured from** to **WCS Origin**.
- ☐ Set the **Range Depth** to **0.0**.
- ☐ Set the **Local 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.

 ©UGS Corp., All Rights Reserved

☐ 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.

☐ Choose the **Stock** tab.

☐ In the **Check Stock** field, key in **-.09**.

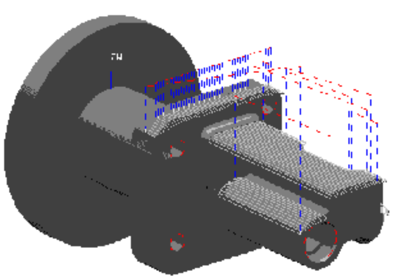
The negative check stock will allow the ball-nose tool to follow the edge of the check stock.

☐ Choose the **Clearances** tab.

☐ 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.

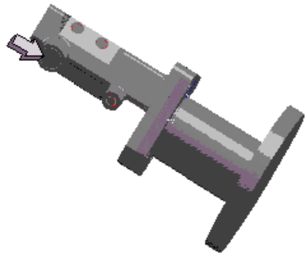
You will now repeat the ZLEVEL_STEEP and CONTOUR_AREA_NON_STEEP machining operations, changing the tool axis each time.

- ☐ 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**.
- ☐ 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 **Measured from** 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**.

 ©UGS Corp., All Rights Reserved

☐ 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**.

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.

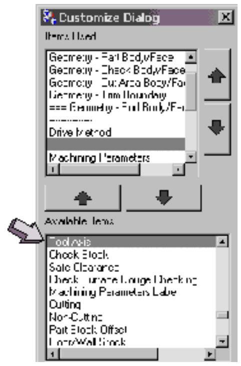
The Other Parameters dialog is displayed.

☐ Select the **Customize Dialog** button from the **Other Parameters** dialog.

The Customize Dialog is displayed.

©UGS Corp., All Rights Reserved

☐ 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.

 ©UGS Corp., All Rights Reserved

Index

©UGS Corp., All Rights Reserved

 ©UGS Corp., All Rights Reserved

STUDENT PROFILE

In order to stay in tune with our customers we ask for some background information. This information will be kept confidential and will not be shared with anyone outside of Education Services.

Please **"Print"…**

Your Name ____ **U.S. citizen** ☐**Yes** ☐**No**

Course Title/Dates ____/_____**thru**_____

Hotel/motel you are staying at during your training ____

Planned departure time on last day of class ____

Employer _______________________________________ **Location** _________________________

Your title and job responsibilities ____________________/_________________________________

Industry: ☐**Auto** ☐**Aero** ☐**Consumer products** ☐**Machining** ☐**Tooling** ☐**Medical** ☐**Other**

Types of products/parts/data that you work with ____

Reason for training ____

Please verify/add to this list of training for *Unigraphics, I-deas, Imageware, Teamcenter Mfg., Teamcenter Eng. (I-Man), Teamcenter Enterprise (Metaphase), or Dimensional Mgmt./Visualization.* **Medium** means Instructor-lead (**IL**), On-line (**OL**), or Self-paced (**SP**)

Software	From Whom	When	Course Name	Medium

Other CAD/CAM/CAE /PDM software you have used ____

Please "check" ! your ability/knowledge in the following…

Subject	None	Novice	Intermediate	Advanced
CAD modeling	q	q	q	q
CAD assemblies	q	q	q	q
CAD drafting	q	q	q	q
CAM	q	q	q	q
CAE	q	q	q	q
PDM – data management	q	q	q	q
PDM – system management	q	q	q	q

Platform (operating system) ____

Thank you for your participation and we hope your training experience will be an outstanding one.

This page left blank intentionally.

Advanced Machining Applications Course Agenda

Day 1

- Introduction &Overview
- Lesson 1. Wave Geometry Linker in Manufacturing
- Workbook: Wave Geometry Linker
- Lesson 2. In-Process Workpiece

Afternoon

- Lesson 3. Libraries
- Workbook: Libraries
- Lesson 4. Machining Faceted Geometry

Day 2

- Lesson 5. High Speed Machining
- Lesson 6. NC Assistan

Afternoon

- Lesson 7. Templates
- Workbook: Templates
- Lesson 8 Holemaking

Day 3

- Lesson 8. Holemaking
- Lesson 9. Integrated Simulation and Verification
- Workbook: Integrated Simulation and Verification

Afternoon

- Project

This page left blank intentionally.

Layers and Categories

The following layer and category standards will be followed in this class.

Model Geometry

Object Type	Layer Assignment	Category Name
Solid Geometry	1-20	SOLIDS
Inter-part Modeling	15-20	LINKED_OBJECTS
Sketch Geometry	21-40	SKETCHES
Curve Geometry	41-60	CURVES
Reference Geometry	61-80	DATUMS
Sheet Bodies	81-100	SHEETS

Drafting Objects

Object Type	Layer Assignment	Category Name
Drawing Borders	101-110	FORMATS

Engineering Disciplines

Object Type	Layer Assignment	Category Name
Mechanism Tools	121-130	MECH
Finite Element Meshes and Engr. Tools	131-150	CAE
Manufacturing	151-180	MFG
Quality Tools	181-190	QA

This page left blank intentionally.

Hot Key Chart

Hot Key	Function	Hot Key	Function
Ctrl-A		Ctrl-N	File, New
Ctrl-B	Edit, Blank	Ctrl-O	File, Open
Ctrl-C	Copy	Ctrl-P	File, Plot
Ctrl-D	Delete	Ctrl-Q	
Ctrl-E	Tools, Expression	Ctrl-R	View, Operation, Rotate (full menu)
Ctrl-F	Fit View	Ctrl-S	File, Save
Ctrl-G	Grip Execute	Ctrl-T	Edit, Transform
Ctrl-H		Ctrl-U	Execute User Function
Ctrl-I	Information, Object	Ctrl-V	Paste
Ctrl-J	Edit, Object Display	Ctrl-W	Application, Gateway
Ctrl-K		Ctrl-X	Cut
Ctrl-L	Format, Layer Settings	Ctrl-Y	
Ctrl-M	Application, Modeling	Ctrl-Z	Edit, Undo

Ctrl-Shift-A	File, Save As	Ctrl-Shift-N	Format, Layout, New
Ctrl-Shift-B	Edit, Blank, Reverse Blank All	Ctrl-Shift-O	Format, Layout, Open
Ctrl-Shift-C	View, Curvature Graph	Ctrl-Shift-P	Tools, Macro, Playback
Ctrl-Shift-D	Drafting	Ctrl-Shift-Q	Quick Shaded Image
Ctrl-Shift-E		Ctrl-Shift-R	Tools, Macro, Record
Ctrl-Shift-F	Format, Layout, Fit All Views	Ctrl-Shift-S	Toolsm Macro, Step
Ctrl-Shift-G	Debug Grip	Ctrl-Shift-T	Preferences, Selection
Ctrl-Shift-H	High Quality Image	Ctrl-Shift-U	Edit, Blank, Unblank All Of Part
Ctrl-Shift-I		Ctrl-Shift-V	Format, Visible In View
Ctrl-Shift-J	Preferences, Object	Ctrl-Shift-W	
Ctrl-Shift-K	Edit, Blank, Unblank Selected	Ctrl-Shift-X	
Ctrl-Shift-L		Ctrl-Shift-Y	
Ctrl-Shift-M		Ctrl-Shift-Z	View, Operation, Zoom (full menu)

Alt-Tab	Toggles Application	Ctrl-Alt-B	Tools, Boundary
Alt-F4	Closes Active Window	Ctrl-Alt-C	Tools, CLSF
F1	Help on Context	Ctrl-Alt-M	Application Manufacturing
F3	View Current Dialog	Ctrl-Alt-N	Tools, Unisim
F4	Information Window	Ctrl-Alt-W	Application Assemblies
F5	Refresh	Ctrl-Alt-X	Tools, Lathe Cross-Section
F6	Quick Zoom		
F7	Quick Rotate		

Evaluation – Delivery

NX 3 Advanced Machining Applications, Course #MT11045

Dates _____________ thru______________

Please share your opinion in all of the following sections with a "check" in the appropriate box:

Instructor: ________________________________ R

If there were 2 instructors, please evaluate the 2nd instructor with "X's"

Instructor: ________________________________ T

	STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
1. …clearly explained the course objectives	☐	☐	☐	☐	☐	☐
2. …was knowledgeable about the subject	☐	☐	☐	☐	☐	☐
3. …answered my questions appropriately	☐	☐	☐	☐	☐	☐
4. … encouraged questions in class	☐	☐	☐	☐	☐	☐
5. …was well spoken and a good communicator	☐	☐	☐	☐	☐	☐
6. …was well prepared to deliver the course	☐	☐	☐	☐	☐	☐
7. …made good use of the training time	☐	☐	☐	☐	☐	☐
8. …conducted themselves professionally	☐	☐	☐	☐	☐	☐
9. …used examples relevant to the course and audience	☐	☐	☐	☐	☐	☐
10. …provided enough time to complete the exercises	☐	☐	☐	☐	☐	☐
11. …used review and summary to emphasize important information	☐	☐	☐	☐	☐	☐
12. …did all they could to help the class meet the course objectives	☐	☐	☐	☐	☐	☐

Comments on overall impression of instructor(s):

Overall impression of instructor(s)…………………………… Poor ☐ ☐ ☐ ☐ ☐ ☐ Excellent

Suggestions for improvement of course delivery: ____

____

What you liked best about the course delivery: ____

____

Class Logistics:

1. The training facilities were comfortable, clean, and provided a good learning environment	☐	☐	☐	☐	☐	☐
2. The computer equipment was reliable	☐	☐	☐	☐	☐	☐
3. The software performed properly	☐	☐	☐	☐	☐	☐
4. The overhead projection unit was clear and working properly	☐	☐	☐	☐	☐	☐
5. The registration and confirmation process was efficient	☐	☐	☐	☐	☐	☐

Hotels: **(We try to leverage this information to better accommodate our customers)**

1. **Name of the hotel** _________________________ **Best hotel I've stayed at** ☐ ☐ ☐ ☐ ☐ ☐
2. **Was this hotel recommended during your registration process?……………** ☐ **YES** ☐ **NO**
3. **Problem? (brief description)** ____

SEE BACK

Evaluation - Courseware
NX 3 Advanced Machining Applications, Course #MT11045

:

Please share your opinion for all of the following sections with a "check" in the appropriate box

Material:

		STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
1.	The training material supported the course and lesson objectives	☐	☐	☐	☐	☐	☐
2.	The training material contained all topics needed to complete the projects	☐	☐	☐	☐	☐	☐
3.	The training material provided clear and descriptive directions	☐	☐	☐	☐	☐	☐
4.	The training material was easy to read and understand	☐	☐	☐	☐	☐	☐
5.	The course flowed in a logical and meaningful manner	☐	☐	☐	☐	☐	☐

6. How appropriate was the length of the course relative to the material? ☐ Too short ☐ Too long ☐ Just right

Comments on Course and Material:____

____

____

____

Overall impression of course..**Poor** ☐ ☐ ☐ ☐ ☐ ☐ **Excellent**

____

Student:

		STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
1.	I met the prerequisites for the class (I had the skills I needed)	☐	☐	☐	☐	☐	☐
2.	My objectives were consistent with the course objectives	☐	☐	☐	☐	☐	☐
3.	I will be able to use the skills I have learned on my job	☐	☐	☐	☐	☐	☐
4.	My expectations for this course were met	☐	☐	☐	☐	☐	☐
5.	I am confident that with practice I will become proficient	☐	☐	☐	☐	☐	☐

Name (optional):________________________ **Location/room**__________________________

☐ **Please "check" this box if you would like your comments 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)

Thank you for your business. We hope to continue to provide your training and personal development for the future.