Turning Manufacturing Process

Student Guide
December 2003
MT11055 — Unigraphics NX 2

Manual History

Manual Revision	Unigraphics Version	Publication Date
	Version 16.0.2.2	October 2000
	Version 17.0.1.1	February 2001
	Version 18.0	January 2002
	Unigraphics NX	November 2002
	Unigraphics NX 2	December 2003

This edition obsoletes all previous editions.

Contents

Course Overview

Course Description

The Turning Manufacturing Process course teaches the use of the Manufacturing Application for creating Lathe tool paths.

The course is taught within the context of an NC/CNC programming session and emphasizes the programming organization and efficiency that the Manufacturing Application provides. The purpose of this class is to teach you how to use the Turning software as you would in your work environment.

Intended Audience

This course is intended for Manufacturing Engineers, Process planners and NC/CNC Programmers who use the Unigraphics NX Turning Manufacturing Application.

Prerequisites

The required prerequisites for the course are:

- Practical Applications of Unigraphics or CAST equivalent
- Mill Manufacturing Process course or the current CAM Transition course
- Basic understanding of the Master Model concept

A working knowledge of the following:

- The Unigraphics NX software interface
- Part file saving conventions
- Experience as an NC/CNC programmer

Special Considerations

When you select *lathe_mill* as the Configuration and *turning* as the CAM Setup, the system will create the mill_planer and drill templates in your setup as well as the lathe templates. This course will discuss the lathe templates and options. Mill Manufacturing is a prerequisite to this class, and therefore, you should be familiar with planar milling and drilling (as they pertain to milling) concepts.

The user interface is common to all of Manufacturing. The general use of the user interface is taught in the Mill Manufacturing Process and CAM Transition course. The specific differences in Turning are discussed in this course.

Objectives

After successfully completing this course, you will be able to perform the following activities in **Unigraphics NX 2**:

- create Lathe cross-sections
- create and assign Programs, Geometry, and Methods to organize programming information
- create Lathe tools
- create Face, Centerline, Rough, Finish, Groove, and Thread operations
- use the In-Process Workpiece to track the remaining material on your part

Student Responsibilities

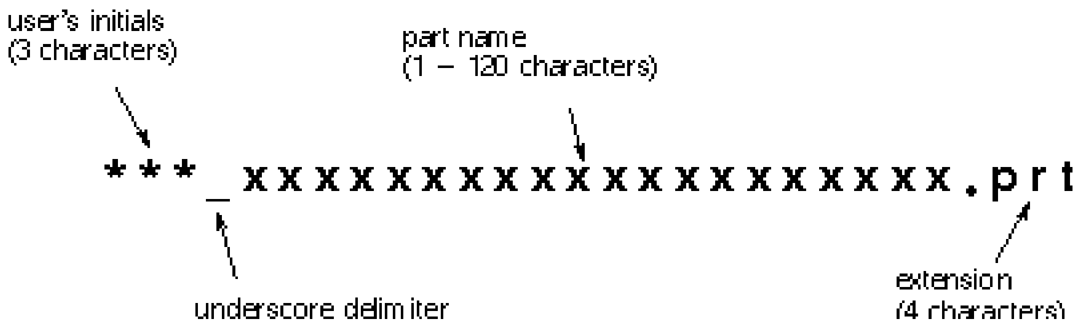
- Be on time
- Participate in class
- Focus on the subject matter
- Listen attentively and take notes
- Enjoy the class

Class Standards for Unigraphics Part Files

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

Class Part File Naming

This class utilizes the following filenaming standard:



When you save a part file for your later use. Use your initials (***) to replace the course name initials. Save these files to your 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.

Notes provide you with information to help you understand the new concepts.

The Arrow Symbol (→)

When you see the arrow symbol (→), it means that you choose an option, then immediately choose another option (stacking the option selections). For example, **Tools** →**Operation Navigator** →**Toolpath** →**Replay** means:

- put the cursor on **Tools** on the main menu bar.
- press mouse button #1 to display the pull-down menu.
- slide the cursor down to **Operation Navigator** (continuing to press mouse button # 1).
- slide the cursor down to **Toolpath**.
- slide the cursor down to **Replay**.
- release mouse button #1.

Layers and Categories

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

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

Layers 1-14, Solid Geometry (Category: SOLIDS)

Layers 15-20, Linked Objects (Category: LINKED OBJECTS)

Layers 21-40, Sketch Geometry (Category: SKETCHES)

Layers 41-60, Curve Geometry (Category: CURVES)

Layers 61-80, Reference Geometry (Category: DATUMS)

Layers 81-100, Sheet Bodies (Category: SHEETS)

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

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

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

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

Layers 151 - 180, Manufacturing (Category: MFG)

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

Colors

The following colors are preset to indicate different object types:

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

Seed Part

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

- sketch preferences
- commonly used expressions
- layer categories
- user-defined views and layouts
- part attributes, manufacturing attributes
- parent groups (Tools, Geometry, Method and Program)

How to Use This Manual

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

The format of the activities is consistent throughout this manual. Steps are labeled and specify what will be accomplished at any given point in the activity. Below each step are action boxes which emphasize the individual actions that must be taken to accomplish the step. As your knowledge of Unigraphics NX 2 increases, the action boxes may seem redundant as the step text becomes all that is needed to accomplish a given task.

Step 1: This is an example of a step.

☐ This is an example of an action box.

The general format for lesson content is:

- Presentation
- Activity
- Project
- Summary

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

At the start of each class day you will be expected to log onto your terminal and start Unigraphics NX 2, being ready to follow the instructor's curriculum.

At the end of the day's class you should always quit Unigraphics NX 2 and log off the terminal.

Workbook Overview

The workbook contains a project that requires lathe programming. You will start with creating geometry, then create tools and parent groups, create facing operations, create several OD and ID roughing operations and then finish the part.

It is the intent of this project to allow you to apply the skills taught in this course. At any point when progress is not being made, ask your instructor for help.

Classroom System Information

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

Username:

Password:

Work Directory:

Parts Directory:

Instructor:

Date:

Student and Workbook Parts

The parts for this class are stored in the class Parts directory. There are two sub-directories located in the Parts directory, the Students_parts and workbook_parts.

The Student_parts sub-directory contains the parts that you will use when working on activities in the Student Manual.

The workbook_parts sub-directory contains the parts that you will use when working on the project within the workbook.

System Privileges

You do not have the system privilege to modify any of the part files. If you attempt to do so, you will get a message saying that the file is Read Only. However, this does not restrict you from working in these files.

You can use the **File →Save As ...** option to save the current part file using your initials. For example, the part file *tmp_any.prt* rename would be (your initials)*_any.prt.*

Saving Assembly Parts

When you save an Assembly part file in the directory provided by your instructor, you may need to change the Load Option (File → Options→ Load Options) to As Saved. This will allow you to open your part and the associated assembly components.

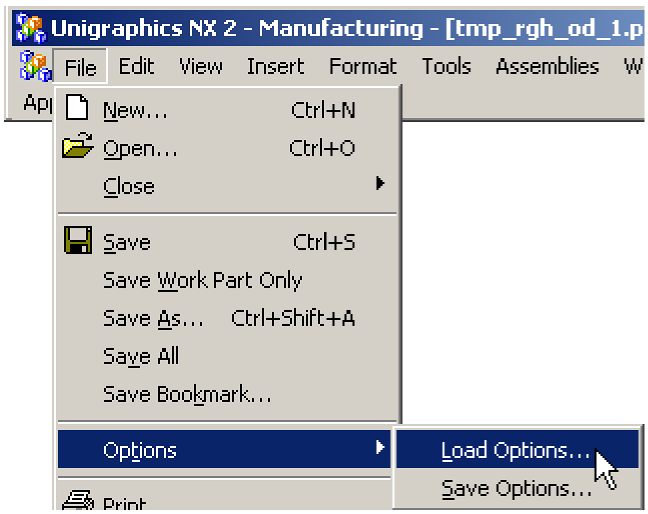
The Parts that are available in the class room have their Load Option (File → Options → Load Options) set to, From Directory. This limits the search for associated assembly files to the directory in which the associated assembly files reside.

The steps necessary are as follows:

Step 1: Change the Part File Load Options.

You are going to change the Part File Load Options from From Directory to As Saved.

- Choose **File→Options→ Load Options**.



The Load Options dialog is displayed. You change the Load Options by specifying the Load Method.

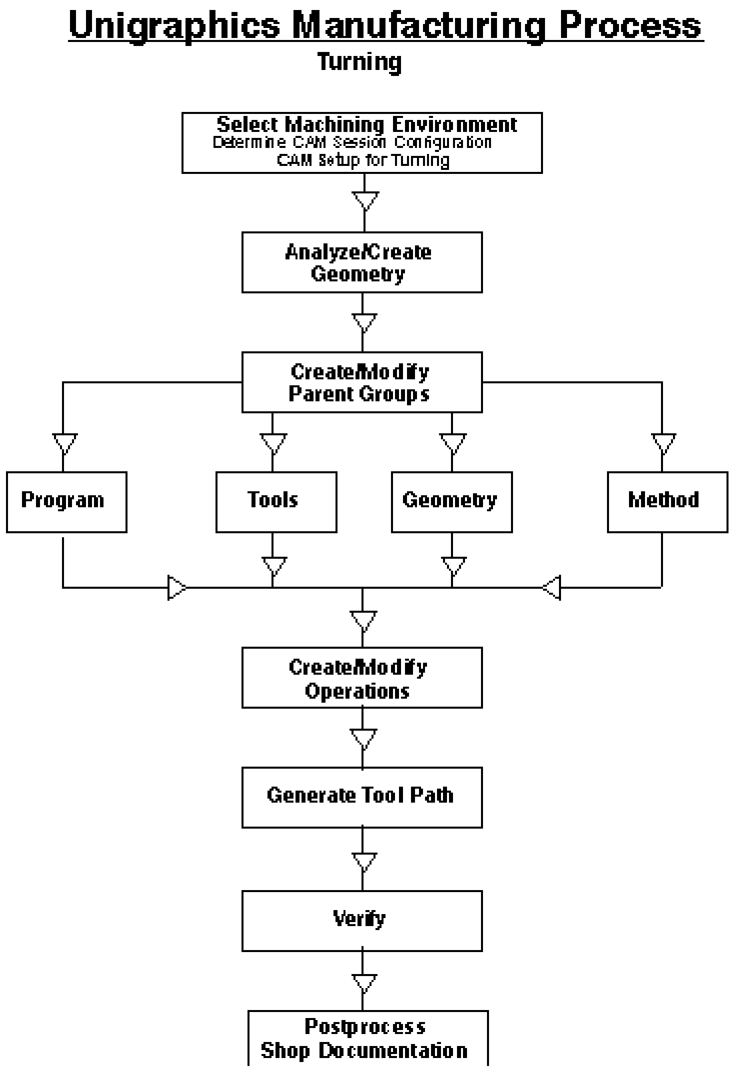
- Choose **As Saved** from the dialog.

The N/C Programming Sequence

The Manufacturing application programming sequence is to:

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

The flow chart below shows the typical Manufacturing process.



Terminology

The following terms are presented along with a brief definition so that you will be somewhat familiar with the Turning terms as used in the Unigraphics NX environment.

CAM Files

Template Part File - any part file that contains CAM information such as, tools, methods and operations that can be retrieved through the template mechanism into another part file.

Operation/Tool Path

Blank is the uncut material.

In Process Workpiece - is the geometry that represents the shape of the work piece at each stage of cutting during the machining process.

Level Angle - defines the direction of cut relative to the WCS.

Cut Type or **Cut Method -** predefined machining methods such as level zigzag, contour rough zig or plunge.

Stepover - the distance the tool travels *between* passes.

Material Side - is the side of the part geometry that is not to be cut.

Radial Move - is motion defined perpendicular to the centerline.

Radial Clearance - is a line or level defined parallel to the centerline.

Axial Move - is motion defined parallel to the center line axis.

Axial Clearance - is a line or level defined perpendicular to the centerline.

Auto Detection - identifies regions to be cut based on part geometry, In-process Workpiece, cut parameters, and tool shape.

Containment - are radial and axial positions that limit the area to be cut.

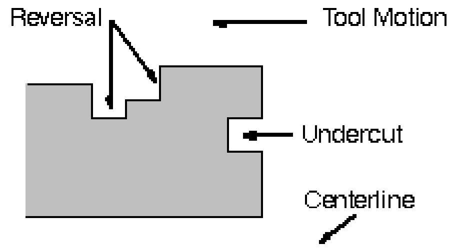
Teach Mode - is a cut type that enables you to define the motions of the operation step by step. This is comparable to manual programming.

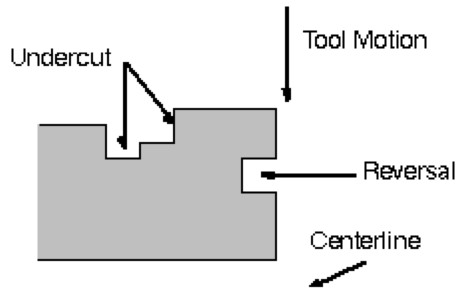
Cleanup - creates an additional pass to clear excess material left after the rough cuts.

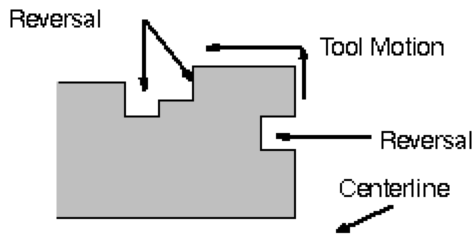
Geometry

Reversal- Any geometry that turns away from the cut motion. The tool will create a ramping motion into the material to cut the reversal.

Undercut- The tool motion will determine whether the geometry is a reversal (which may be cut) or an undercut (which is not cut).


Reversal
Tool Motion
Undercut
Centerline


Tool Motion
Undercut
Reversal
Centerline


Reversal
Tool Motion
Reversal
Centerline

Lesson

1 *Lathe Cross Sections*

Purpose

Lathe Cross Section allows you to create associative curves that represent a 2D cross section of a solid body to be used for Lathe Programming.

The body can be rotational symmetrical (e.g., cylindrical, cone, etc.) requiring a Simple cross section, or non-symmetrical (e.g., a cam shaft hex, etc.) requiring a Complex cross section.

Objective

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

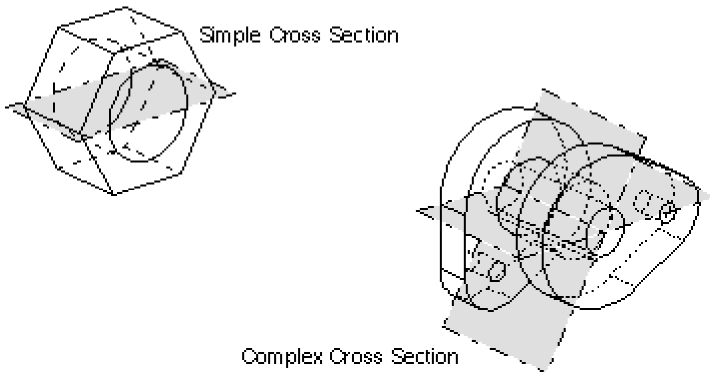
- Create Simple and Complex Cross Sections.
- Identify the Body and the Rotation, Projection and Section Planes.

The Lathe Cross Section Feature

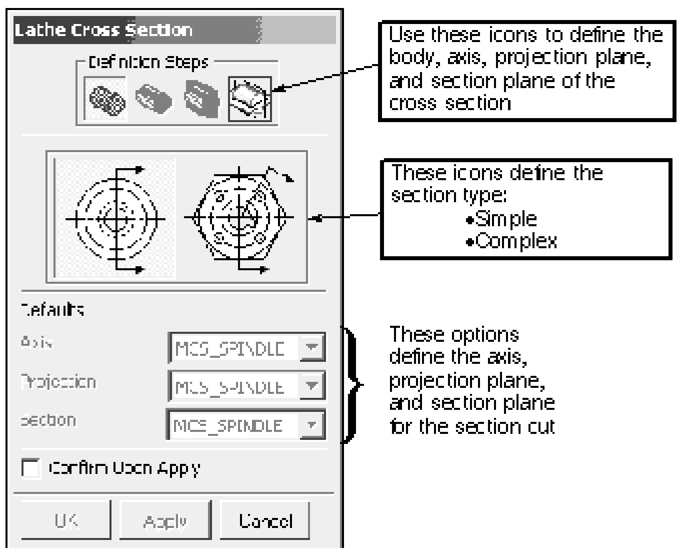
The two cross-sectioning methods are:

Simple - The Simple method uses a single section plane. The resulting cross-sectional curves lie in the plane used to create the cross section. This is intended for symmetrical parts.

Complex - The Complex method uses multiple section planes. The cross-sectional curves are rotated about a specified axis (axis of rotation) onto a projection plane. This is intended for non-symmetrical parts.



Look at the Lathe Cross Section dialog that follows.

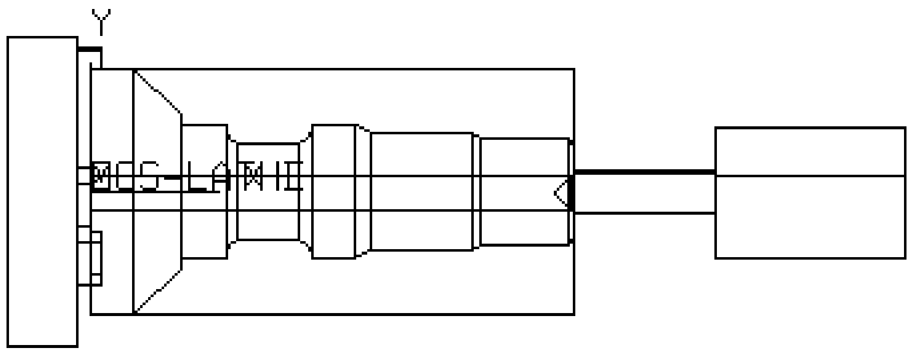


Activity: Creating a Simple Cross Section

In this activity, you are going to create two simple cross sections on the shaft shown below. You will cross section the blank and then the part geometry. The cross sections that you create will be used in a following lesson when you create lathe boundaries.

Step 1: Open and rename the part, then enter the Manufacturing application.

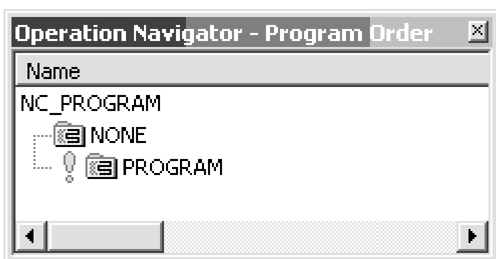
☐ Open the part **tmp_crs_sec_1.prt**.



☐ Save the part file as ***_crs_sec_1.prt**, where *** represents your initials.

☐ Choose **Application →Manufacturing**.

☐ Double-click the **Operation Navigator** tab in the resource bar and undock the Operation Navigator (using the Ctrl key) so it displays in a separate window.



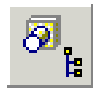
Step 2: Check the location of the MCS.

Check the location of the WCS and MCS before you start since many of the cross sections are created using the default MCS XY-Plane.

☐ Visually check the **WCS** and **MCS** locations.

Note that the WCS and MCS are both in the same location. You must also check the **MCS_SPINDLE** Parent Group.

Change the Operation Navigator view.

☐ Choose the Geometry View icon. 

☐ Double-click on the **MCS_SPINDLE** Parent Group in the Operation Navigator.

The TURN_ORIENT dialog is displayed.

☐ Visually check the **MCS** location.

The XM is along the Spindle axis and the YM is parallel to the face of the chuck.

☐ Choose **Cancel** to dismiss the dialog.

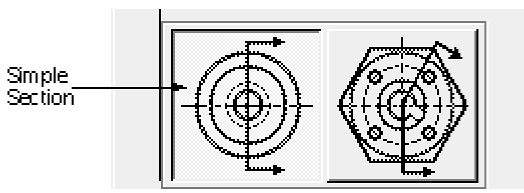
Step 3: Specifying the Body.

You first select the solid bodies and then define a section plane.

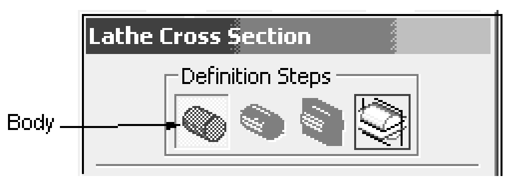
☐ Choose **Tools** →**Lathe Cross-Section**.

The Lathe Cross Section dialog is displayed.

☐ Choose the **Simple Section** icon.



Notice that the Body icon is also selected.



Before creating any cross sections, note the work layer. It is layer 50. You will create each cross section on it's own layer. This will allow you to display cross sections as required.

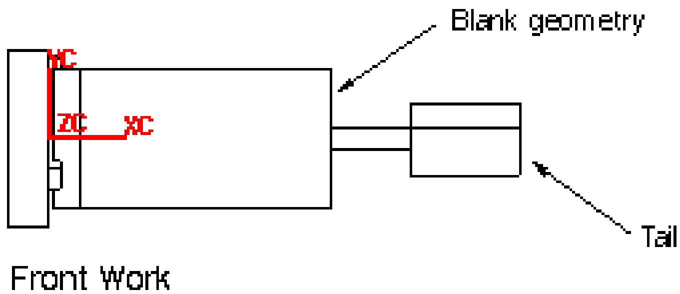
Now you will select the body or bodies to cross section. In this case, you are going to cross section the Blank geometry.

It will be helpful to eliminate some of the geometry that you are not going to need.

Use the Hidden Edge option to hide the extra geometry that is not needed at this time.

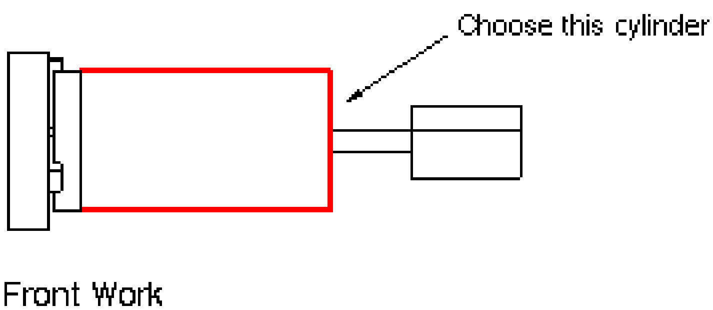
☐ Use MB3 in the work view to choose **Hidden Edges→Invisible** option on the pop up menu.

Your part should look similar to the part shown below.



You can use the Hidden Edges option in any of the views that you will be using.

☐ Select the blank geometry as shown below.



The cylinder is highlighted and displayed in red.

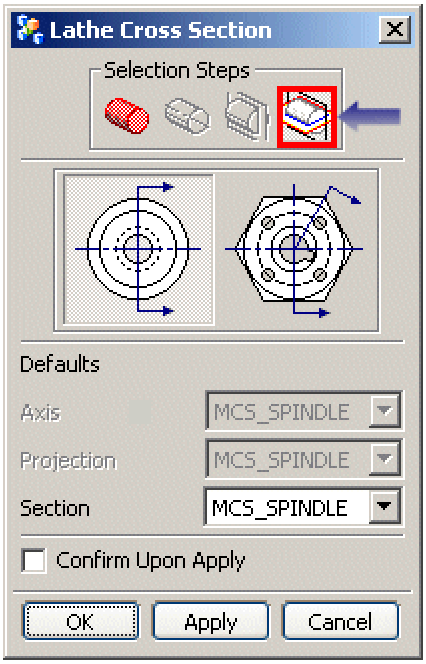
This is the only body that you are going to select.

☐ Choose **OK**.

The plane symbol is displayed at MCS XY-Plane.

Step 4: Specify the Section Plane

The Section Plane icon is selected.



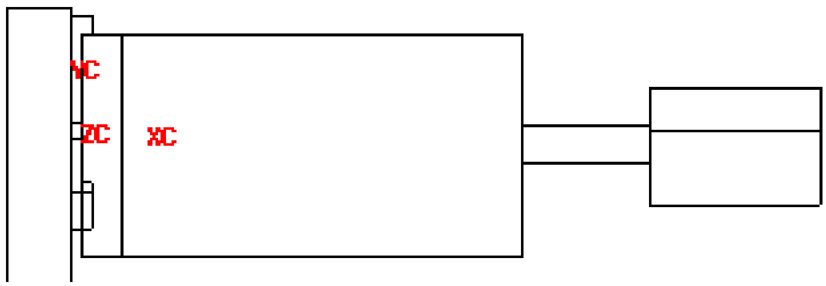
The Cue line prompts you to Specify the section plane(s).

You are going to use the default Section, MCS XY-Plane.

☐ Choose **OK**.

If you used the Hidden Edge option to hide the extra geometry, undo it now.

☐ Choose **Hidden Edges→Visible**.



The cross section is created and displayed in orange.

You will create a cross section of the part geometry. Before you begin, change the work layer.

☐ Choose **Format→Layer Settings** from the Menu Bar.

The Layer Settings dialog is displayed.

☐ Key in **55** in the Work field and select the **Make Work** button below the list box.

Now you will make layer 50 invisible. This is the layer that contains the blank cross sections.

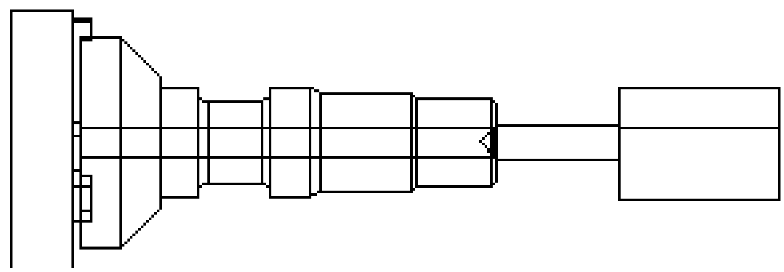
☐ Double-click on Layer **50**.

You no longer need the blank cylinder geometry. You will make it invisible as well.

☐ Double-click on layer **53**.

☐ Choose **OK**.

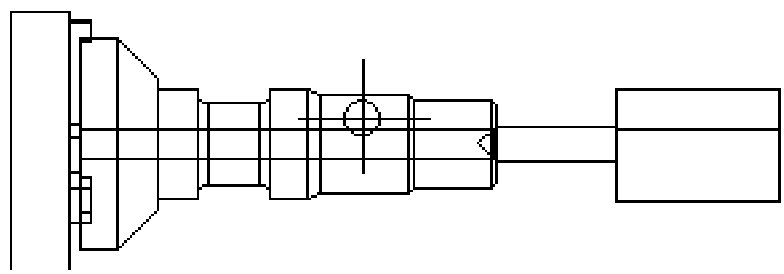
You are returned to the Lathe Cross Section dialog. Layer 55 is now the work layer and layer 50 and 53 are invisible. Your part should look similar to the part shown.



You are ready to create the part geometry cross section.

☐ Choose the **Simple Section** icon if it is not already selected.

☐ Select the part geometry as shown.



All of the part geometry is selected to cross section and is highlighted in red.

☐ Choose **OK**.

A plane symbol is displayed at the MCS XY-Plane and the Section Plane icon is selected on the Lathe Cross Section dialog.

The Cue line prompts you to specify the Section Plane.

You will use the default MCS XY-Plane again.

☐ Choose **OK**.

The cross section is created.

Step 5: Change the layer settings to ease geometry selection.

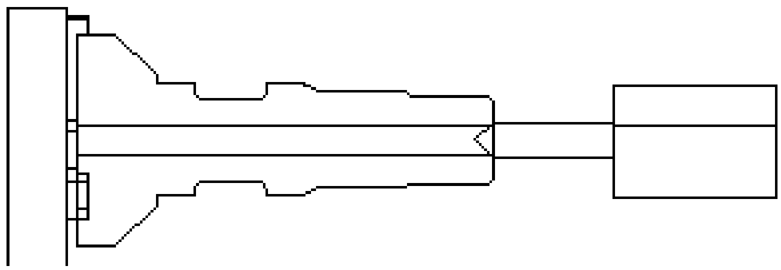
You are going to make layer 1 invisible.

☐ Choose **Format→Layer Settings** from the Menu Bar.

The Layer Setting dialog is displayed.

☐ Double-click on **Layer 1** (contains the part geometry) to make it invisible.

☐ Choose **OK**.



You now have associative curves that you can select for your lathe boundary creation.

Next you will make layer 50 selectable.

☐ Choose **Format→Layer →Settings** from the Menu Bar.

The Layer Settings dialog is displayed

☐ Double-click on Layer **50** and choose **OK**.

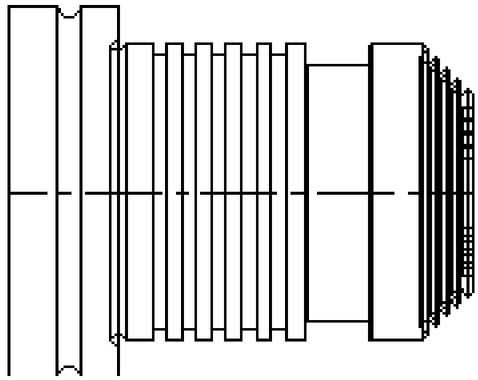
☐ Choose **File→Save** from the Menu Bar.

Activity: Create a Simple Cross Section Unassisted

Step 1: Open and rename the part.

☐ Open the part file **tmp_crs_sec_2.prt**.

☐ Save the part file as, *****_crs_sec_2.prt**, where *** represents your initials.



Step 2: Create the Cross Section geometry for this part.

☐ Put the Cross Section geometry on Layer **53**.

If you have any questions, ask your instructor.

☐ **Save** the part file

☐ **Close All** part files.

Activity: Create a Complex Cross Section

You will create a plane that intersects the outermost edges of a solid, and then rotate the resulting edge curve to the MCS XY plane.

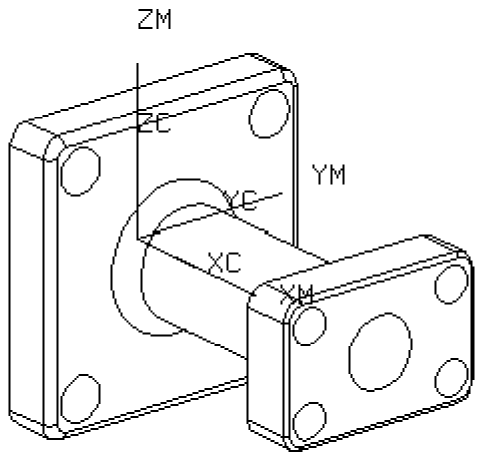
To create the edge curve, you must select a solid body and specify a Cross Section plane that intersects the outer edge of the solid.

To project the curves, you must specify the axis of curve rotation (usually around the MCS X axis) and the Projection plane to which the curves are rotated (usually the MCS XY plane).

Step 1: Open and rename the part.

☐ Open the part file **tmp_crs_sec_3.prt**.

☐ Save the part file as *****_crs_sec_3.prt**, where *** represents your initials.



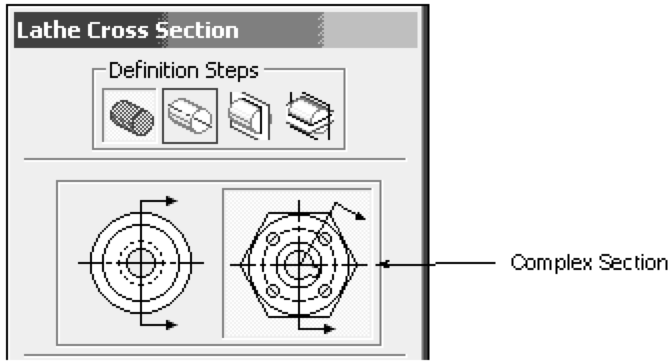
☐ Choose **Application→Manufacturing**.

Step 2: Create the Complex Cross Section geometry which represents the largest OD or swing of the part.

☐ Choose **Tools→Lathe Cross-Section**.

The Lathe Cross Section dialog is displayed.

☐ Choose the **Complex Section** icon.



Notice under the Definition Steps label, the **Body** icon is highlighted. You are prompted to choose the body.

☐ Use the cursor to select the part body

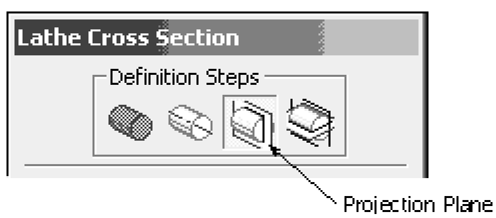
☐ Choose **OK**.

Under the Definition Steps label, notice that the Axis icon has been selected. You are prompted to specify the axis of rotation for the cross section curves.

Also note that the Axis label is highlighted. The default MCS_SPINDLE is correct for this part.

☐ Choose **OK**.

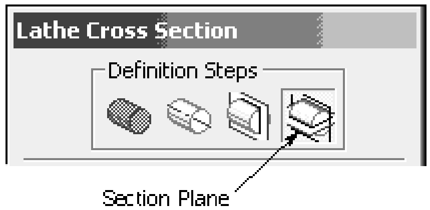
Note that under the Definition Steps label, the **Projection Plane** icon is highlighted. You are prompted to specify the plane to which the cross section curves are to be rotated.



Also note that the Projection label is highlighted. The default MCS_SPINDLE is correct for this part.

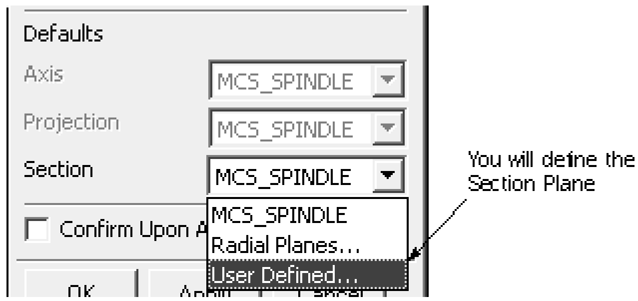
☐ Choose **OK**.

Note that under the Definition Steps label, the **Section Plane** icon is highlighted. You are prompted to specify the plane that intersects the axis and the cross section curves.



Also note that the Section label is highlighted.

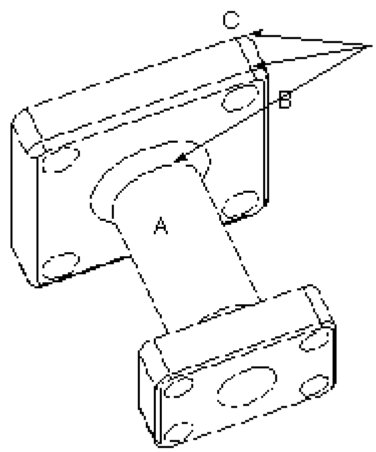
☐ Change the Section to **User Defined**.



☐ Choose the **Three Points** icon.

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

☐ As shown in the following illustration, choose the three arcs.



Because the fillets at the corners are equal, the center points of these three arcs will create a plane through:

- **A** – The MCS X axis
- **B** and **C** – The two arcs representing the outermost part of the part (on one end).

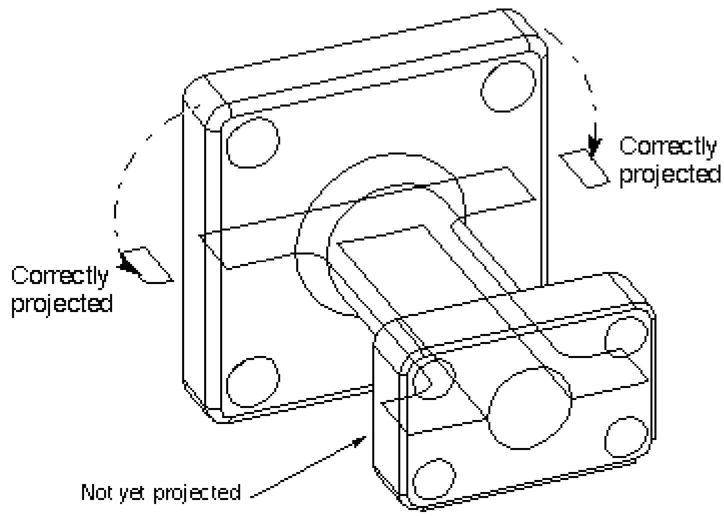
Note that you could have chosen:

- **A** – As any other arc centered on the MCS X axis.
- **B** and **C** – As any of the three other of the outer pairs of arcs.

☐ Choose **No** when you are prompted to define more planes.

Note the plane symbol on the part.

☐ On the Lathe Cross Section dialog, choose **Apply**.



The outermost edges of the curves are rotated to the MCS XY plane. Note that the smaller end shows the cross section in the MCS XY plane, but without any curve rotation. You will add these curves next.

Step 3: Remove the cross section curves.

☐ Choose **Edit→Delete**.

☐ Use the **Chain** option to select all of the curves.

☐ Choose **OK** to delete the curves and return to the Lathe Cross Section dialog.

Step 4: Respecify the cross section curves.

☐ Again, choose the solid body and then **OK** to accept it.

☐ Choose **OK** to accept the default rotation Axis.

☐ Choose **OK** to accept the default Projection Plane.

☐ Change the Section to **User Defined**.

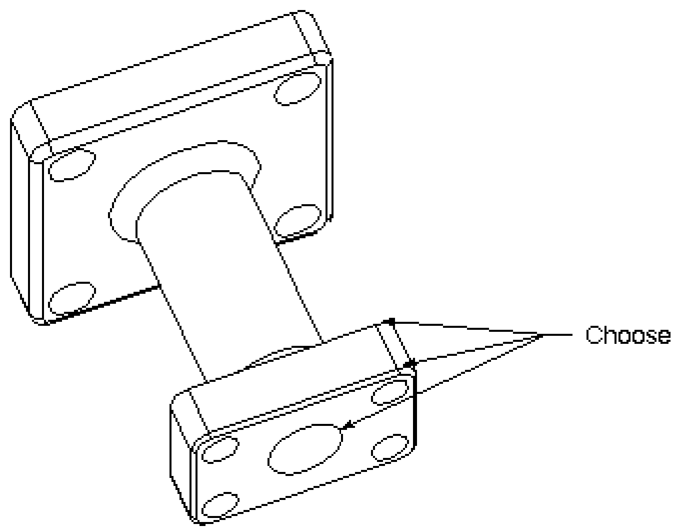
☐ Specify the arcs as before, but choose **Yes** when you are prompted to define additional cut planes.

Step 5: Create the Cross Section geometry for the smaller rectangular end of the part.

☐ Choose the **Three Points** icon.

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

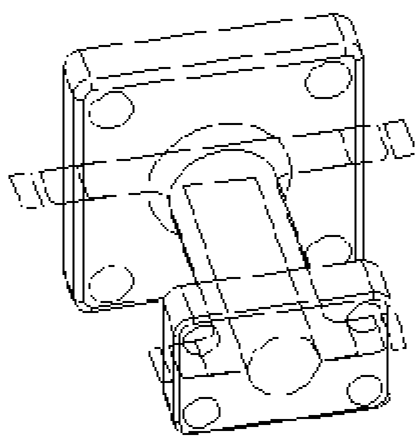
☐ As shown in the following illustration, select the three arcs.



Note the Cross Section plane symbol displayed in the plane of the three points (arc centers).

☐ Choose **No** when you are prompted to define more planes.

☐ On the Lathe Cross Section dialog, choose **Apply**.

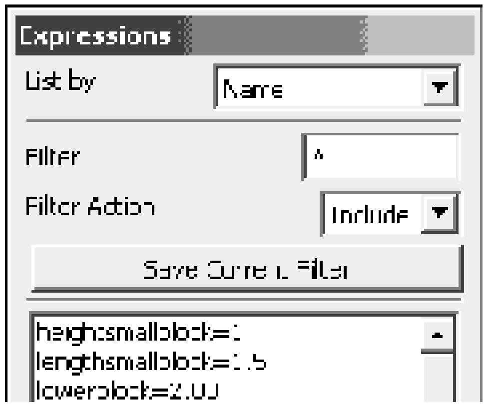


Now the curves, on both ends of the part, are projected to the MCS XY plane. The cross section curves can be used in another operation to rough turn the rectangular ends and rough or finish the cylindrical middle of the part.

Step 6: Change the width of the part geometry using an expression and view the updated cross section.

☐ Choose **Tools→Expression** from the menu bar.

The Expressions dialog is displayed. Note the list of part expressions.



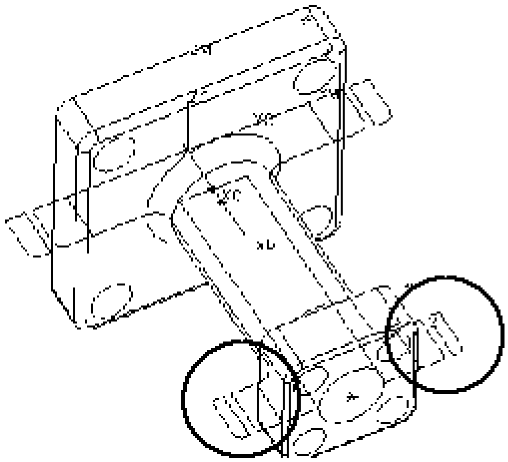
☐ Choose the **lenghtsmallblock=1.5** expression.

The expression displays in the expression list box. The slot width value displays in the value window. Use the expression list box to edit the slot width. After you have made the change, you can use the return key to enter the change and continue making edits. Use OK to accept the edits.

☐ Change the **lenghtsmallblock** to **1.00** and press the return key.

☐ Choose **OK** from the Expressions dialog to accept the slot change.

Note that the part geometry has changed, but the cross section curves have not yet updated. You need to redisplay the Lathe Cross Section dialog and then Cancel to update the curves.



☐ Choose **Tools→Lathe Cross-Section**.

☐ Choose **Cancel** from the Lathe Cross Section dialog to update the curves**.**

☐ **Save** the part file.

☐ **Close** all of the part files.

The cross section only updates while in the Manufacturing application. If the model changes outside of the Manufacturing application, and these changes affect the cross sections, they will be updated only when you re-enter the Manufacturing application.

Summary

Geometry can be created very easily by the use of Lathe Cross Sections. The symmetrical cross sections are associative and are automatically updated when geometry of the part model is modified. The following functions are used to create Lathe Cross Sections:

- Simple Cross section function
- Complex Cross section function

Lesson

2 Retrieving and Creating Tools

Purpose

This lesson is a review of Tool Creation and Retrieval. When the tool that you need to cut your part is not in the Unigraphics provided Tool Library, you will create it using the Create Tool feature of Manufacturing.

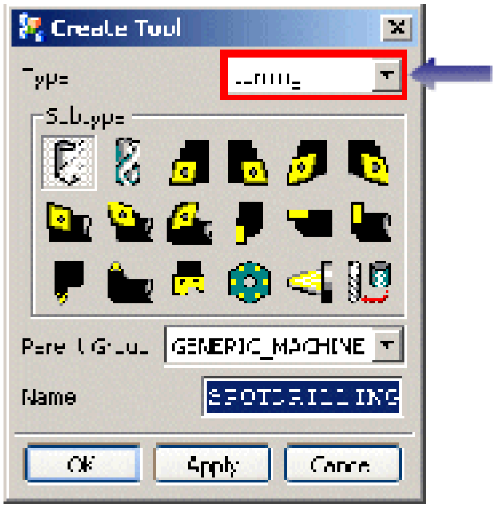
Objective

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

- Retrieve tools from theUnigraphics NX Tool Library.
- Create tools using the Create Tool dialog.

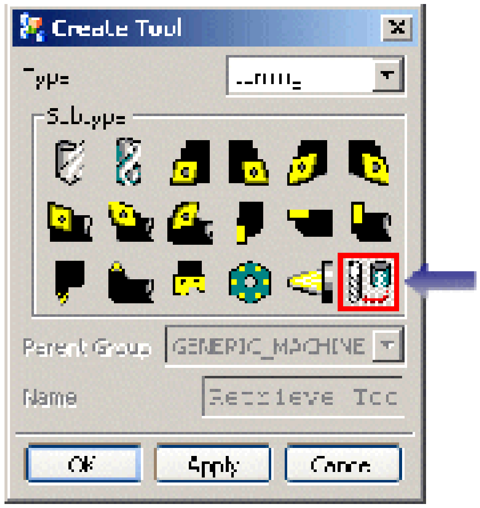
The Create Tool dialog

The types of tools that you can create are controlled by the Machining Environment Setup. In the case below, the Type selection is turning because that was the Machining Environment Setup. The tool types that you create can only be used for turning (Rough, Finish, Groove, Centerline-Drill, Thread, Teachmode).

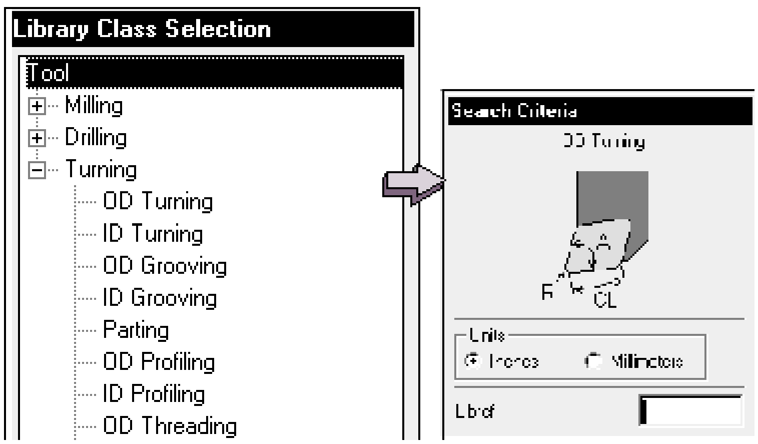


Retrieving Tools from the Unigraphics Library

The Unigraphics NX supplied Tool Library is accessed using the Retrieve Tool button. The retrieval process uses various search criteria to locate tools.



You will use the Library Class Selection and the Search Criteria dialogs to enter the search criteria information.

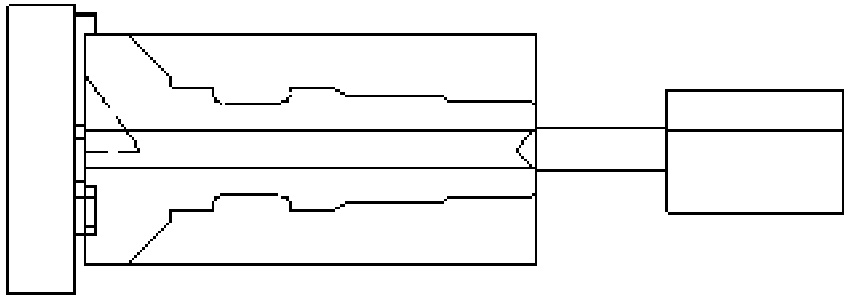


After you enter tool search criteria, the tool list is displayed. This search procedure is typical of other libraries within Unigraphics NX.

Activity: Retrieving Tools from the Library

Step 1: Open the part file and enter Manufacturing.

☐ Open the part file **tmp_tools_1.prt**.



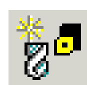
☐ Save the part file as *****_tools_1.prt**.

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

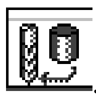
The Machining Environment is the Lathe Configuration and the Turning Setup.

Step 2: Enter the search criteria.

☐ Change to the **Machine Tool View** in the Operation Navigator.

☐ Choose the **Create Tool** icon. 

The Create Tool dialog is displayed.

☐ Choose the **Retrieve Tool** icon , and then choose **APPLY**.

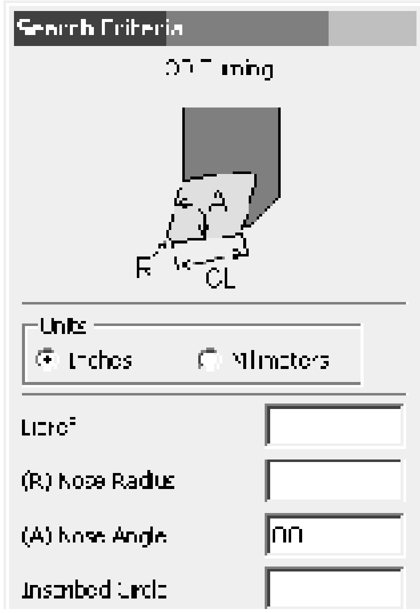
The Library Class Selection dialog is displayed.

☐ Expand **Turning**, in the list.

☐ Highlight **OD Turning**

☐ Choose **OK**.

The Search Criteria dialog is displayed.



☐ Enter **80** into the Nose Angle field.

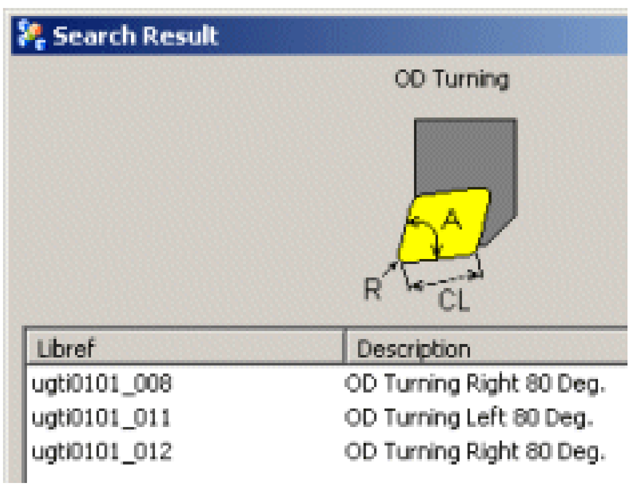
☐ Choose the **Count Matches** button.

Three matches are found.

Step 3: Retrieve and display the tools.

☐ Choose **OK**.

The Search Results window is displayed.



☐ Choose the tool **ugti0101_012** on the list and then choose **Display**.

Notice in the graphics window, the tool is displayed at the WCS. You may need to change the part from a shaded model to wire frame to see this clearly and you may have to adjust the screen position to view the tool.

☐ Choose the middle tool (ugti0101_011) on the list, and then choose **Display**. This displays a tool assembly. Change to a shaded view to see it clearly.

☐ Choose **OK**.

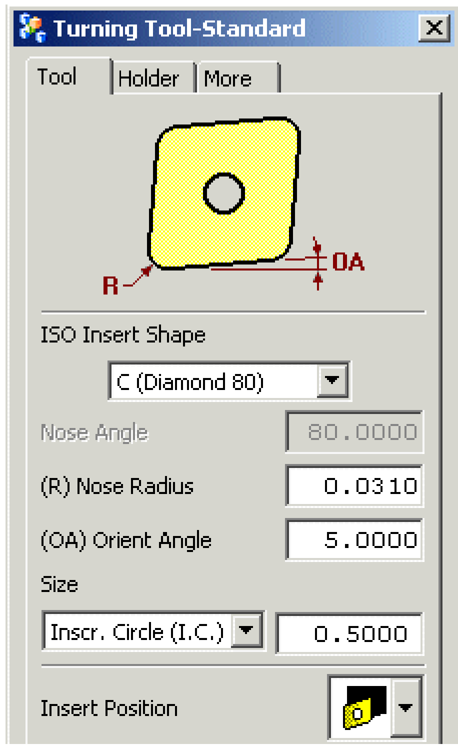
The tool is retrieved into the part. If there had been other tools in the part, this tool would have been appended to the bottom of the tool list in the Operation Navigator.

☐ **Save** the part file.

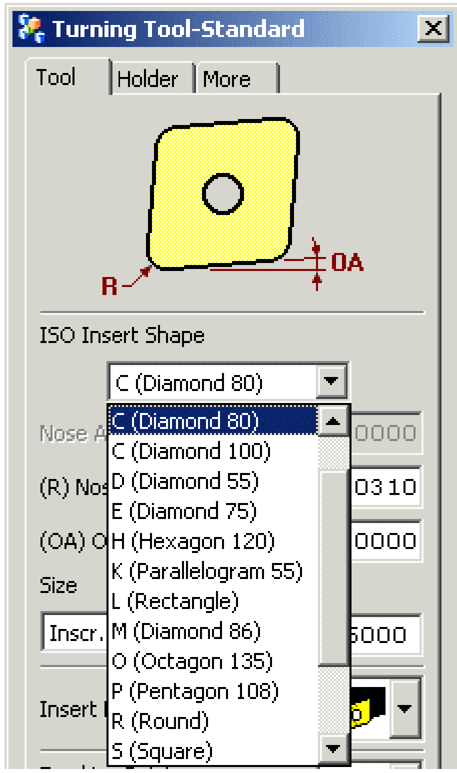
Creating New Turning Tools

You can also create new tools using the tool templates. First, choose the tool Subtype and then choose OK.

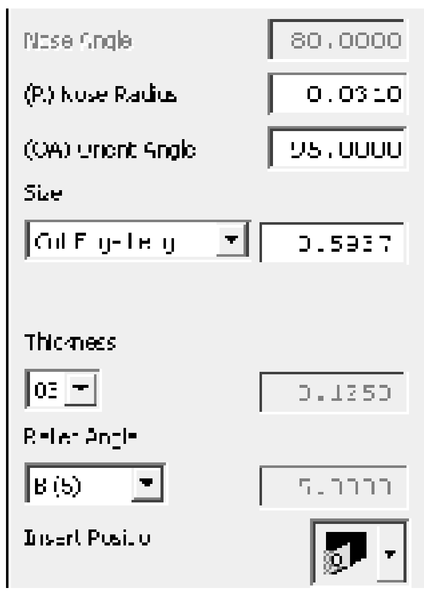
Then, adjust the tool parameters on the tool dialog.



Selection from the insert shape list defines a wide range of insert/tool standards.



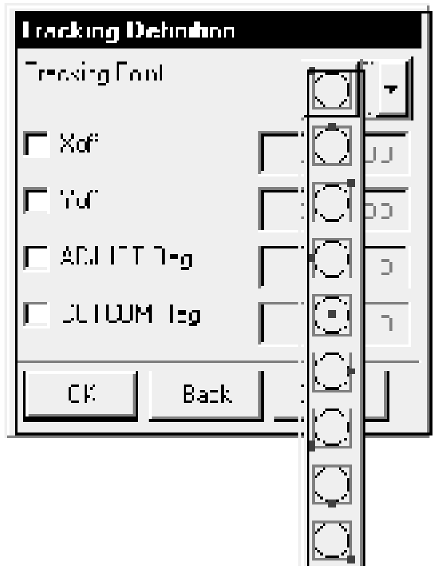
You can change the tool radius, orientation or size as shown on the 80 degree diamond dialog, shown below. Each different insert/tools have similar options that you can define to fully create the tool that you need.



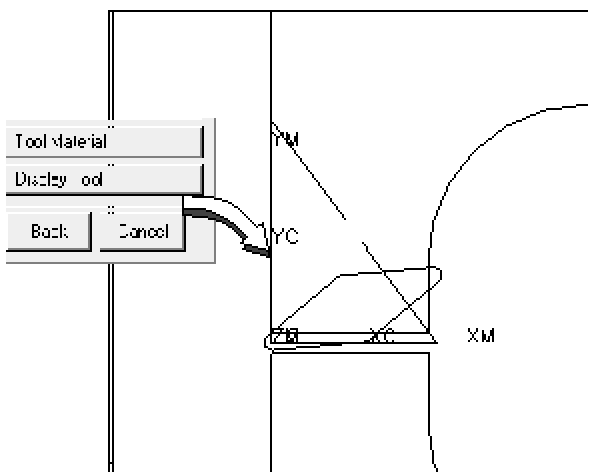
Tool Tracking

You can also change the tool tracking point.

The tool path will be output according to the tracking point. Changing the tracking point from center to bottom left for an OD tool will translate the tool path by the tool nose radius to the left and down.



You can display the tool to verify the shape. The tool is displayed at the WCS, which represents the tracking point.



At this time you should enter the tool, offset register and adjust register numbers for use when postprocessing.

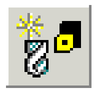
Activity: Creating New Tools

Step 1: Open the part file.

☐ Continue using *****_tools_1.prt**.

Step 2: Retrieve the tool template and name the first tool.

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

☐ Choose the **Create Tool** icon. 

☐ Choose the **OD_80_L** Tool Type from the Subtypes.

You are first going to create a button tool. This is accomplished by changing the insert type.

☐ Enter the name of the tool, **button_.250**.

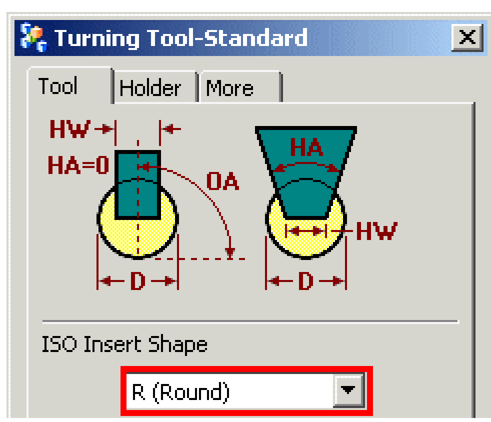
☐ Choose **APPLY** at the bottom of the Create Tool dialog.

The Turning Tool-Standard dialog is displayed.

Step 3: Change the insert type and enter the tool dimension.

☐ Set the **ISO Insert Shape** option to **R (Round)**.

The template for a button tool is displayed.



☐ Enter the following tool dimensions into the appropriate value field:

- Diameter........... **.250**
- Holder Angle........ **0**
- Holder Width........ **.1625**
- Orient Angle........**90**
- Choose **Display Tool**.

  The tool displays at the XC-YC origin.

Step 4: Examine the Tool Tracking Options.

Observe the various tracking options.

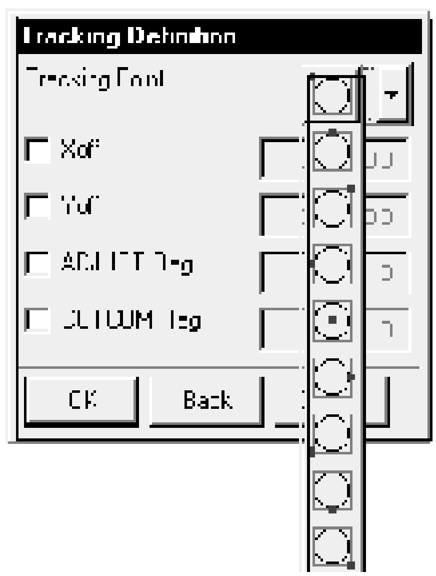
☐ Choose the **Define Tracking** button.

The Tracking Definition dialog is displayed.

☐ Choose the **Tracking Point** arrow, then examine the choices.

Notice that this tool will track from the center point of the insert. You can change the tracking position by selecting one of the icons on the list.

Notice the Offset and Cutcom settings that are available.



☐ **Cancel** the dialog, since you will not change the tracking position.

You will now define the tool material.

☐ Choose the **More** tab and then the **Material** button.

The Search Result dialog is displayed.

☐ Highlight **Carbide Coated** and then choose **OK**.

The Turning Tool-Standard dialog is displayed.

☐ Choose **OK** from the Turning Tool-Standard dialog to accept the tool definition.

Step 5: Retrieve the tool template and name the second tool.

☐ Choose the **Create Tool** icon.

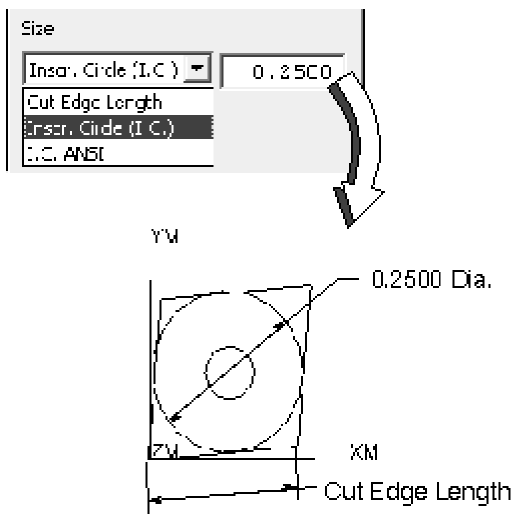
☐ Choose the **OD_55_L** Tool Type from the Subtypes.

You are going to create a diamond type tool.

☐ Choose **APPLY** at the bottom of the Create Tool dialog.

Step 6: Change the insert size.

☐ Choose **Inscr. Circle (I.C.)** for the Size.



The Inscr. Circle (I.C.) option allows you to key in the diameter of the inscribed circle as illustrated above.

The Cut Edge Length option allows you to key in the length of the cutting edge as shown above.

The I.C. ANSI option allows you to key in an ANSI code (a positive integer denoting eighths of an inch).

☐ Enter **0.2500** for the diameter of the inscribed circle.

☐ Choose **Display Tool**.

☐ Choose **OK** from the Turning Tool-Standard dialog to accept the tool definition.

Step 7: Create an ID cutting tool unassisted.

☐ This time create a Standard ID_55_L turning tool with the following dimensions.

- Nose Radius........ **.0625**
- Inscribed circle...... **.50**
- Tool Material **TMC0_00004 Carbide , Coated (Indexable).**

The Cut Edge Length defines the insert size.

The tools are displayed in the Operation Navigator in the Tool View.

Step 8: Save and Close the part file.

Summary

Lathe tools can be retrieved from existing libraries or can be created as required. The flexibility and robustness of the Create Tool dialog allows the creation of numerous turning tool types.

Lesson

3 The Create Geometry Dialog

Purpose

This lesson gives you detailed information about using the Create Geometry dialog to define blank geometry, part geometry, and part material.

Objective

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

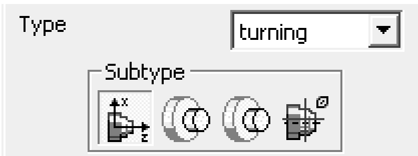
- Specify the MCS (MCS Spindle).
- Create Blank and Part boundaries.

The Create Geometry dialog

The Create Geometry dialog allows you to:

- Select geometry (Part, Blank and Containment) to be machined.
- Create one or more Machine Coordinate Systems (MCS).

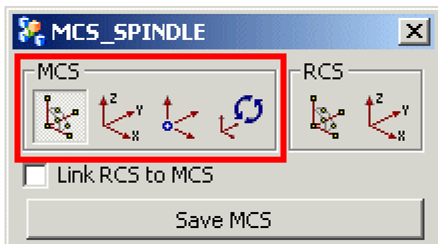
Each of these items and how they are used will be discussed in detail.



In **turning** you can create:
MCS Spindle
Workpiece
Part
Containment

MCS Spindle

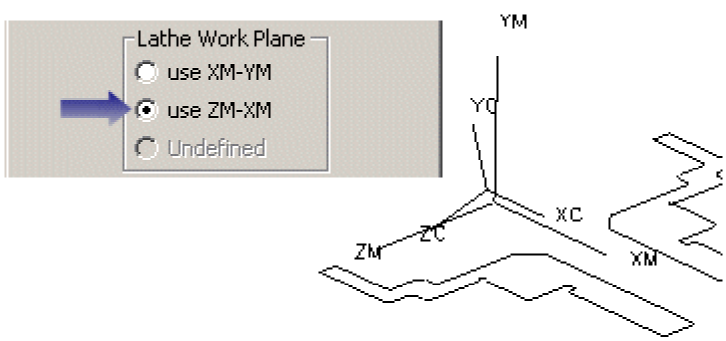
This option allows you to define and store a Machine Coordinate System (MCS). These are used for all Turning Operations.



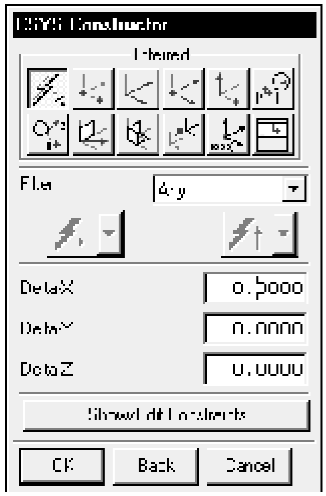
You can change the Orientation, Origin, Rotation, and display the MCS.

The Lathe Work Plane options allow you to directly specify XM-YM or ZM-XM as the desired lathe work plane. If the XC-YC plane is parallel to the XM-YM or ZM-XM work plane, the system defaults to the XM-YM or ZM-XM work plane.

If the XC-YC plane of the WCS is not parallel to the XM-YM or ZM-XM lathe work plane, you can directly specify XM-YM or ZM-XM as the desired lathe work plane. In the following figure, choosing "use ZM-XM", specifies ZM-XM as the lathe work plane regardless of the orientation of the WCS. This is the plane in which cross section curves and tool paths will be created.

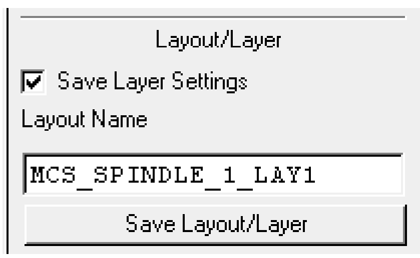


If you choose the MCS Constructor icon at the top of the dialog, you will display the CSYS Constructor dialog.



The CSYS Constructor allows you to create an MCS.

Use the TOP view.



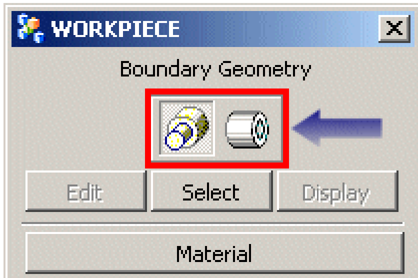
Orient the MCS and WCS, if necessary.

Note the Save Layout/Layer button located at the bottom of the MCS_SPINDLE dialog.

This option allows you to save the current Layout and associate it with the defined MCS. This is useful if you rotated or oriented your part in any other view. You can then retrieve this view using the Switch Layer/Layout option on the MB3 menu. This option is available on all of the other Geometry dialogues.

Workpiece

The WORKPIECE includes the Part and/or Blank geometry. These Parent Groups are used for passing geometry information to operations. This also provides Part and Blank geometry among all operations for use in Verification.



You will notice in the Geometry view, in the Operation Navigator, that the MCS SPINDLE and the first WORKPIECE (if expanded) are created for you. The WORKPIECE contains no information. At this point you can edit the WORKPIECE that has been created by the system or create a new one.

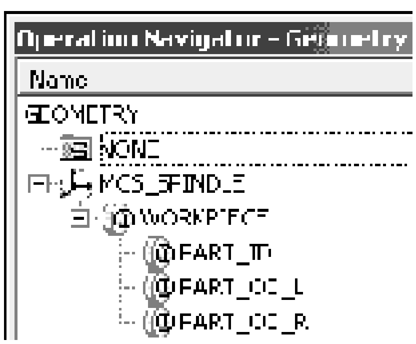
In order to define Part or Blank Geometry within the WORKPIECE dialog, you choose the Select button. This displays the Part Boundary or Select Blank dialog. If the geometry is already selected, you can also edit or reselect it from this dialog.

Note that if you define the Part Material from this dialog, that selection will override a Part Material selection made using the Menu Bar **Tools→Part Material**.

Turning Tip

When used in conjunction with WORKPIECE parent groups, PART parent groups help you to organize your geometry as recommended below.

The first geometry parent group defined is the MCS_SPINDLE. Within the MCS is the WORKPIECE parent group which defines the material and blank geometry. Within the WORKPIECE parent group are the PART groups that define various Part boundaries.

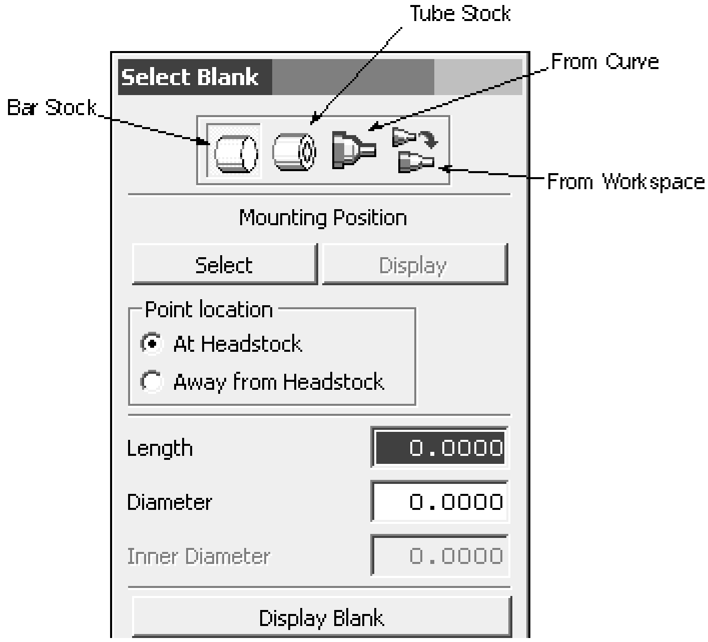


Naming or renaming the Geometry Parent Groups with meaningful names can alleviate a great deal of confusion when creating programs. Example:

- WORKPIECE Parent Group.......Contains only Blank geometry
- PART_OD Parent Group......Contains only the OD geometry
- PART_ID Parent Group......Contains only the ID geometry

Selecting Blank Geometry

- Bar Stock (mount position and blank parameters)
- Tube Stock (mount position and blank parameters)
- From Curve (existing geometry created by lathe cross section)
- From Workspace (references in-process workpiece)



Mounting Position

- Select (select a point where the material begins)
- Point Location (at or away from the head stock)

Boundaries

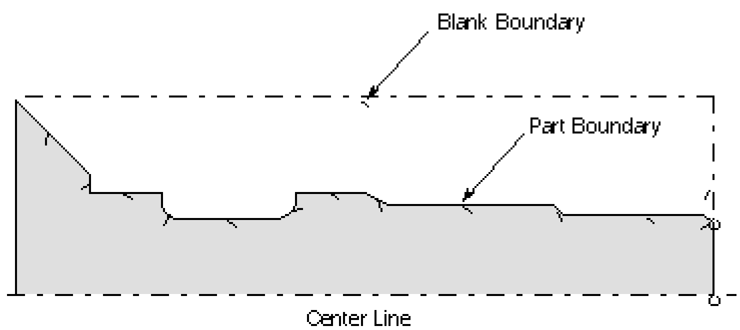
The shape of each Blank and Part is defined using boundaries. Blank boundaries are created by keying in values for tube stock, bar stock, by selecting curves, or by referencing the in-process workpiece. Part boundaries are created by selecting faces, curves, or points.

The From Workspace option  allows you to reference the In-process Workpiece (IPW) as the work piece is transferred from one spindle to another or turned around in the spindle. When the work piece is remounted, the IPW referenced from the previous workspace defines the blank geometry in the current workspace. This allows the system to progressively remove material while the work piece is passed from one workspace to the next.

Defining the Blank and Part Boundary

The Blank Boundary defines the stock that is to be removed.

The Part Boundary defines the finished part.



There are several characteristics of the boundary display:

- a small circle is displayed marking the start of the boundary
- arrows show the direction of the boundary

Boundary editing

- equidistant (adds same amount of offset to elements)
- face (adds offset only to faces)
- radial (adds offset only to radial elements)

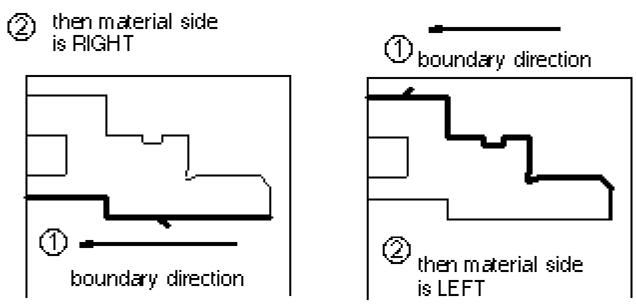
Selecting Part Geometry

This dialog is the same as the one you used in Milling to define Part boundaries. You have filter options (Faces or Curves) as well as Ignore options (Holes, Islands and Chamfers). You also must define the Material Side.

Notice that the default boundary type is set to Open. Your Part boundary may be either Open or Closed.

Material Side

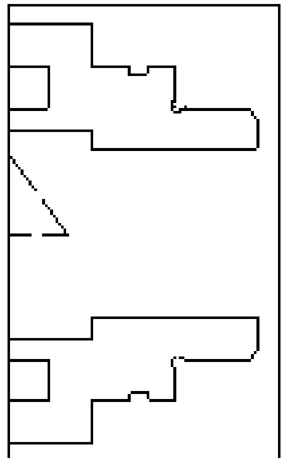
Material Side is Inside or Outside for closed boundaries and Left or Right for open boundaries. For open boundaries, when you look in the direction of the boundary, if the material to remain (the part) is to the right then the Material Side will be Right.



Activity: Selecting / Using Geometry

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

☐ Open the part file **tmp_geom_1.prt**



☐ Save the part file as ***_geom_1.prt**, where *** represents your initials.

☐ Enter the **Manufacturing** application.

This session has been initialized using the Lathe configuration and the Turning CAM setup.

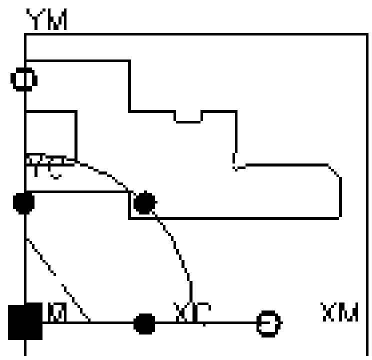
Step 2: Change the Operation Navigator view to the Geometry View.

☐ Choose the **Geometry View** icon to change the Operation Navigator to the Geometry View.

Step 3: Examine the MCS_SPINDLE Parent Group.

☐ Double-click on the **MCS_SPINDLE** Parent Group on the Operation Navigator.

The MCS is in the correct position, so you can accept the default setting. It is always good practice to check the MCS position each time you start a session or follow someone else's work.



☐ Select the MCS **Constructor** icon.

☐ Select the work plane symbol on the part to be sure the MCS is orientated exactly to this plane.

☐ Choose **OK** twice to accept the MCS.

Step 4: Select the Blank geometry.

You will select the blank geometry.

☐ Expand the MCS_SPINDLE.

☐ Double-click on the Parent Group name **WORKPIECE**.

The WORKPIECE dialog is displayed.

☐ Under Boundary Geometry, choose the **Blank** icon.

☐ Choose **Select**.

The Select Blank dialog is displayed.

☐ Choose the **From Curve** icon.

The Select Blank dialog changes to display the options available for the From Curves selection.

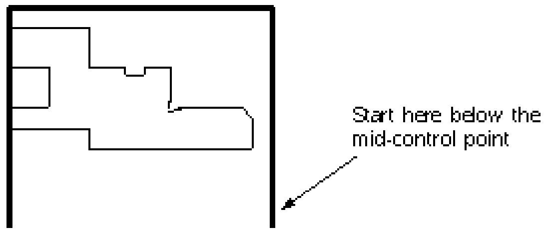
☐ Choose the **Select** button on this Select Blank dialog.

The Blank Boundary dialog is displayed.

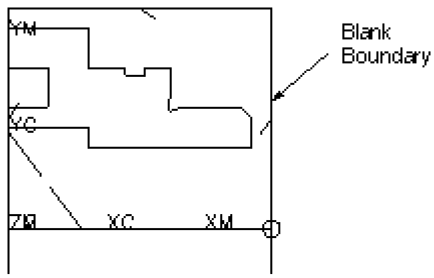
You are ready to select the geometry for the Blank Boundary.

☐ Choose the **Curve Boundary** icon.

☐ Select the curves as shown, then choose **OK**.



☐ Choose **OK**.



The Blank boundary is displayed. The system will automatically close the boundary.

☐ Choose **OK** again.

Step 5: Create a new PART Geometry Parent Group, rename and create an open part boundary.

You are going to create a new PART Geometry Parent Group, rename it and create an open part boundary that defines only the outside diameter.

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

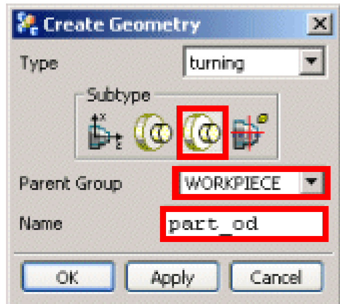
☐ If necessary, change the Type to **Turning**.

☐ Choose the **PART** icon from the Create Geometry dialog.

☐ In the Name field type **part_od**.

☐ Choose **WORKPIECE** as the Parent Group for PART_OD.

This PART_OD Parent Group must be created under the WORKPIECE Parent Group that defines the blank boundaries in order to utilize inheritance and the In-Process Workpiece.

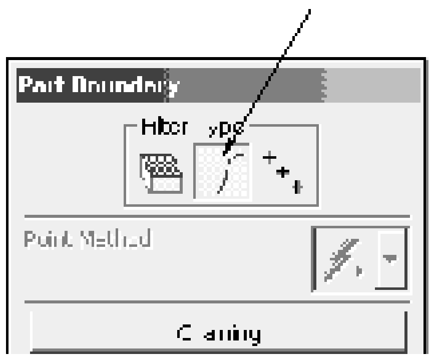


☐ Choose **OK** from the Create Geometry dialog.

The PART dialog is displayed. You are ready to select the OD part geometry for the boundary.

☐ Under the Part Boundary label, choose **Select** from the dialog.

The Part Boundary dialog is displayed.

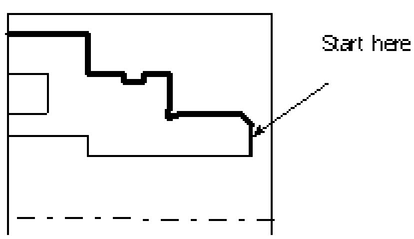


☐ Choose the **Curve Boundary** icon.

☐ Check that the boundary type is **Open**.

☐ The Material Side should be **Left**.

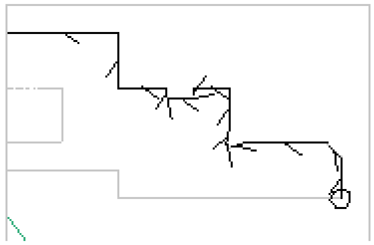
☐ Select the geometry as shown.



☐ Choose **OK** from the Part Boundary dialog.

Display the newly created boundary.

☐ Under the Part Boundary label, choose **Display**.



☐ Choose **OK**.

Step 6: Create a new PART Parent Group, rename and create an open part boundary with limited assistance.

You are going to create a new PART Parent Group, rename it and create an open part boundary that defines only the inside diameter.

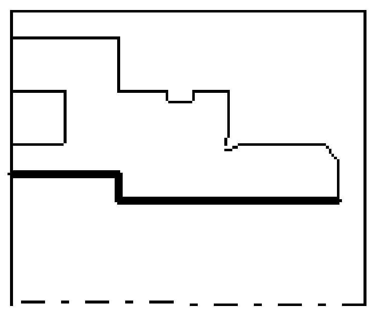
Here are a few hints:

☐ the Parent Group for the PART_ID should be WORKPIECE

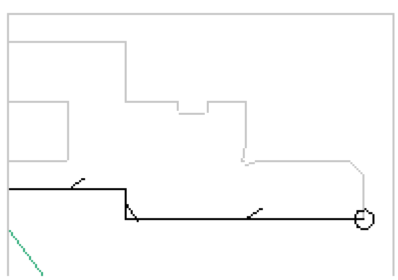
☐ the Boundary Type must be defined (open)

☐ Material Side must be defined correctly

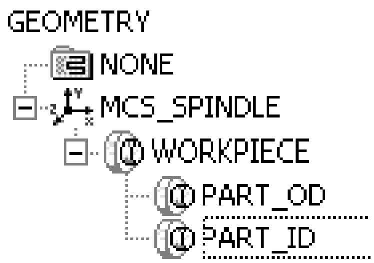
☐ Select the geometry as shown.



When displayed, the boundary should appear as shown below.



You have created the necessary geometry to program the first set up position of this part. The Operation Navigator should look similar to the one shown below.



Step 7: Edit the WORKPIECE parent group and create a closed part boundary.

You are going to delete the PART_OD and PART_ID parent groups and create a single closed part boundary within the WORKPIECE parent group that includes both the inside and outside diameter.

☐ In the Geometry view of the Operation Navigator, delete the PART_OD and PART_ID objects.

☐ Double-click on the Parent Group named **WORKPIECE**.

The WORKPIECE dialog is displayed.

☐ With the **Part** icon chosen, choose **Select**.

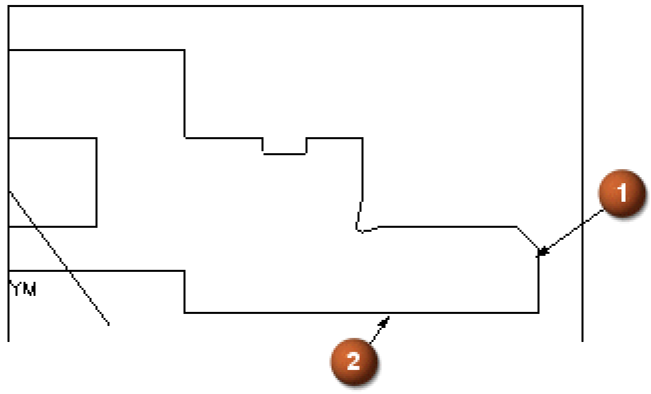
The Part Boundary dialog is displayed.

☐ Choose the **Curve Boundary** icon.

☐ Choose **Closed** for the Type and be sure **Inside** is specified as the Material Side.

☐ Choose the **Chaining** button.

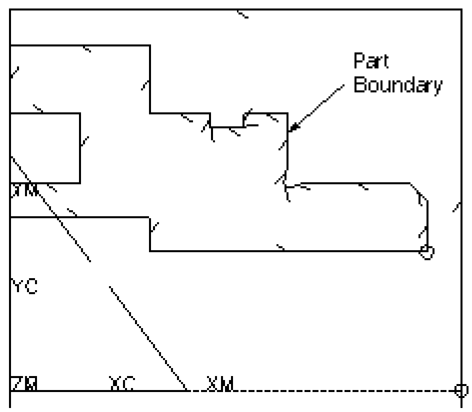
☐ Select the curves as shown, then choose **OK**.



☐ Choose **OK**.

☐ Choose **Display**.

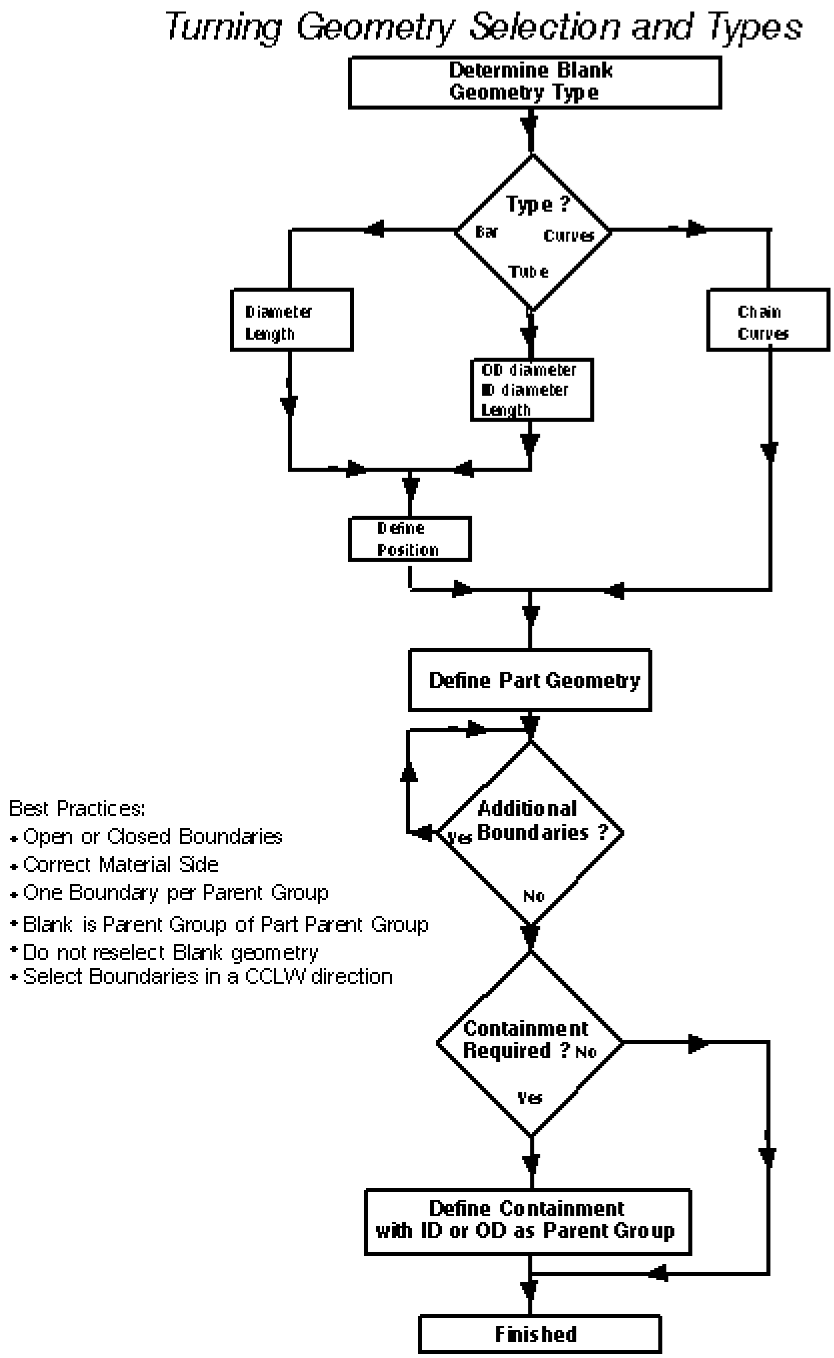
The Part boundary is displayed.



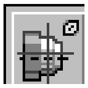
☐ Choose **OK** to complete the WORKPIECE.

☐ **Close** the part file.

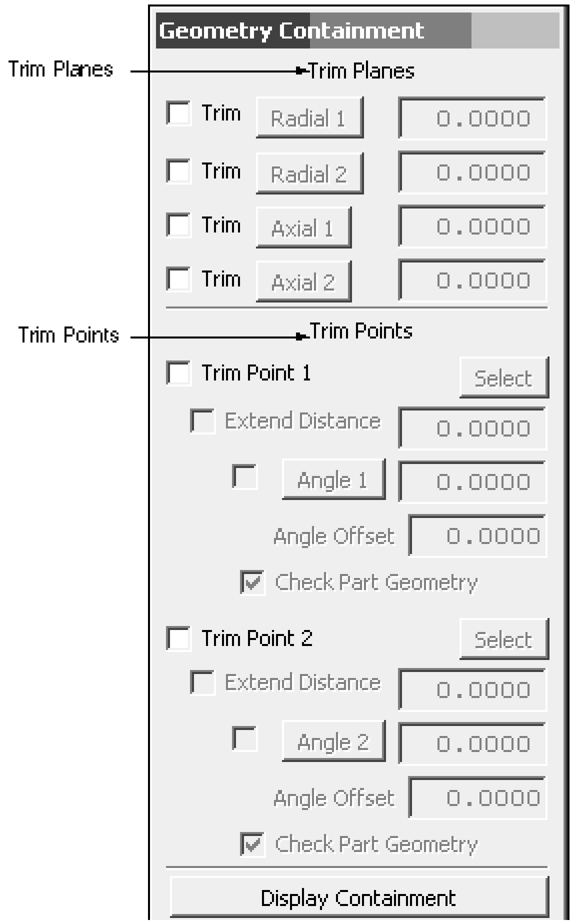
This concludes this activity.


Turning Geometry Selection and Types
Determine Blank Geometry Type
Type ?
Bar
Curves
Tube
Diameter Length
OD diameter ID diameter Length
Chain Curves
Define Position
Define Part Geometry
Additional Boundaries ?
Yes
No
Best Practices:
• Open or Closed Boundaries
• Correct Material Side
• One Boundary per Parent Group
• Blank is Parent Group of Part Parent Group
• Do not reselect Blank geometry
• Select Boundaries in a CCLW direction
Containment Required ?
No
Yes
Define Containment with ID or OD as Parent Group
Finished

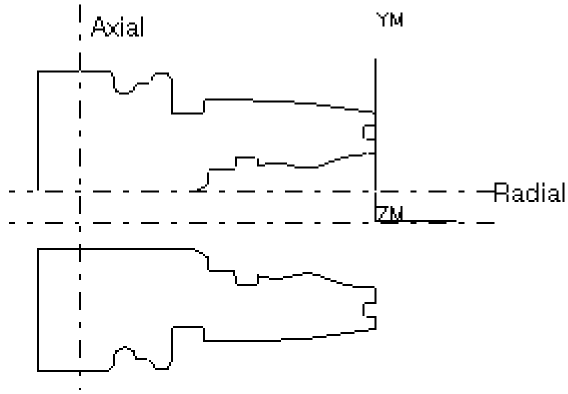
Containment

 CONTAINMENT restricts the cut region by allowing you to position axial and radial trim planes and trim points. Trim points allow you to constrain the cut region in addition to the limits imposed by axial and radial trim planes. They allow you to be very specific about the areas you wish to cut.

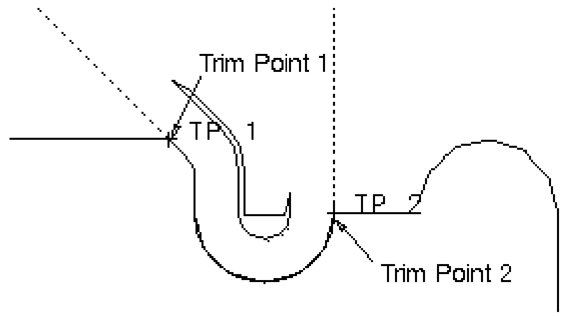
When defined in a geometry parent group, the containment constrains the cut regions for all operations contained within the parent group.



An axial containment plane can avoid collisions with the chuck. A radial containment plane can avoid cutting to the center line.



Trim points can further constrain the cut region to very specific areas.



Summary

This lesson examined the Create Geometry options and how it is use with respect to the Turning application.

In this lesson you:

- Created three different Geometry Parent Groups and assigned geometry and an MCS to the Parent Groups.

- Selected the Blank and Part geometry within the appropriate geometry Parent Group.

Lesson

4 *Facing Operations*

Purpose

This lesson teaches you how to create Facing Operations. In a typical lathe programming session, one of the first operations that you perform is facing the end of the blank.

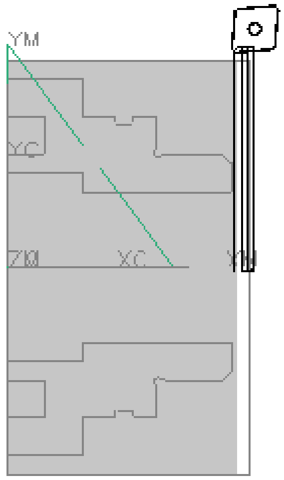
Objective

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

- Create Facing Operations.
- Define Containment in an operation.
- Use Autodetection options.

Facing Operations

The Facing Operation type is one of the Roughing Operation Types. It is used to remove material from faces (material that is perpendicular to the centerline of the part). Many of the options found on the Facing template are common to other roughing operations.



The In-Process Workpiece

The system tracks the shape of the Workpiece at the end of each cutting operation (In-Process Workpiece). The In-Process workpiece is the remaining shape of the workpiece at each stage of the cutting process. Each In-Process Workpiece is used by the next operation. This reduces the need to select geometry for each operation.

Cut Regions and Automatic Cut Region Detection

Roughing (and Finishing) operations use automatic cut region detection. The system detects the remaining material between the In-process Workpiece and the part surface and then uses the cut region for the subsequent operation.

The material that can be removed by a single operation depends on the:

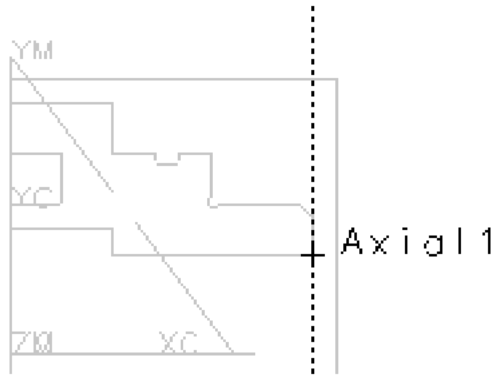
- shape of the In-Process Workpiece.
- part boundary.
- blank boundary.
- tool shape.

Containment

The Containment option restricts the cut region by allowing you to position axial and radial trim planes and trim points. Trim points allow you to constrain the cut region in addition to the limits imposed by axial and radial trim planes. They allow you to be very specific about the areas you wish to cut.

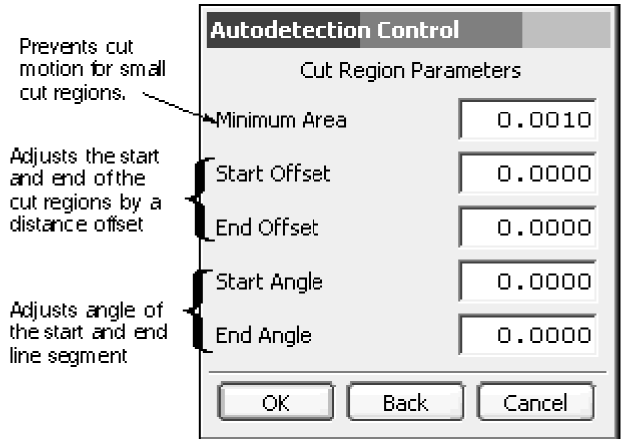
To define a trim plane, turn on the Trim option and then choose the corresponding Radial or Axial button. The Point Constructor dialog is displayed, allowing you to define the containment as a point on an axis or radius.

The example below shows an axial trim plane commonly used when facing.



Autodetection Control

Autodetection Control is used to change and adjust auto-detected cut regions.



Minimum Area

The Minimum Area can be used to eliminate small cut regions whose area is less than the minimum area value. This option prevents the generation of unwanted cut motion for small cut regions. The area is measured in square inches/millimeters. If you are having a problem detecting a small amount of material, you can reduce the minimum area.

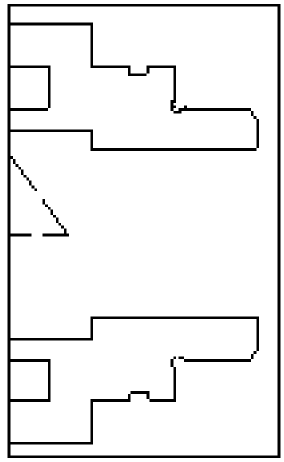
The Start/End Offset and Start/End Angle options apply only in the case of open contours, and only if a containment has not been set.

Activity: Creating a Facing Operation

In this activity, you will create an operation to cut the face of the part. You will generate the tool path using only the default settings. then you will reject the operation and set some of the options available on the Facing dialog and re-generate the tool path.

Step 1: Open the part file and enter Manufacturing.

- ☐ Open the part file **tmp_face_1.prt**.



Note that the cross sections have been created for you.

- ☐ Save the part file as *****_face_1**, where *** represents you initials**.**

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

- ☐ If necessary, undock the Operation Navigator and display it in a separate window.

  This session has been initialized using the Lathe configuration and the Lathe CAM setup.

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

  The Type may be set to mill_planar. The system will not track the Configuration or Set up after the part is saved and closed the first time. You may need to set the Type to Turning.

- ☐ Change the Type to **turning**, if necessary.

Step 2: Define the Parent Groups for the operation.

- ☐ Choose the **Facing** icon from the Create Operation dialog.

- ☐ Set the following Parent Groups:

- Program..........**PROGRAM**
- Use Geometry....**RIGHT_END_OD**
- Use Tool..........**OD_80_L**
- Use Method......**LATHE_ROUGH**

☐ Choose **OK**.

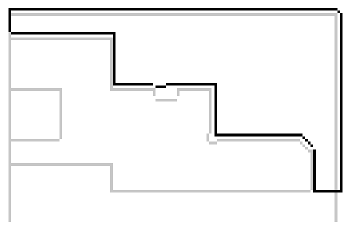
The Facing dialog is displayed.

Step 3: Examine the part Containment.

By pre-defining the geometry (Part and Blank) the system can track the material that needs to be removed. You can display the area to be removed.

☐ Choose the **Display** option located in the Cut Regions area.

The cut region is displayed.



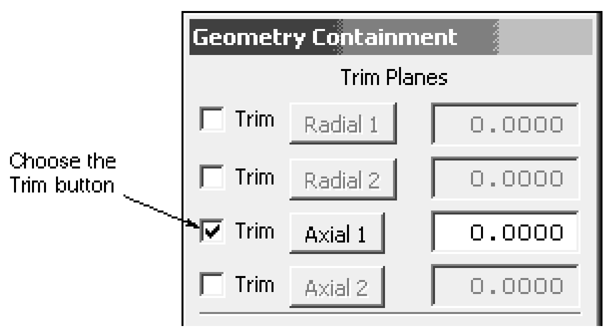
Step 4: Limit the area the tool will cut.

You want to cut only the face of the part. The cut region indicates that the entire OD is going to be cut in this operation. You will use a trim plane to contain the cut region.

☐ Choose **Containment** under Cut Regions.

The Geometry Containment dialog is displayed. You are going to specify an axial containment for this operation.

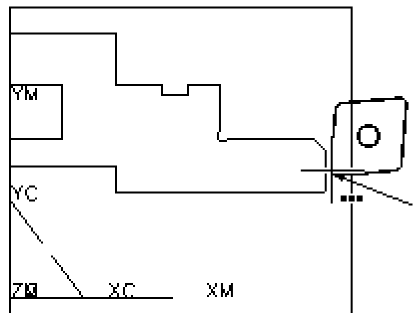
☐ Check the **Trim** option next to Axial 1.



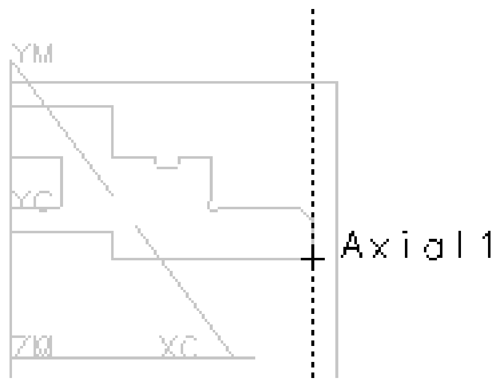
☐ Choose the **Axial 1** button.

The Point Constructor dialog is displayed. You define the containment as a point on an axis. The default is an inferred point, and you will use that setting. You will use the cursor to select the containment location.

☐ Select the face of the part as the axial containment line as shown.



A point labeled Axial 1 is displayed on the face of the part. The Geometry Containment dialog is displayed.

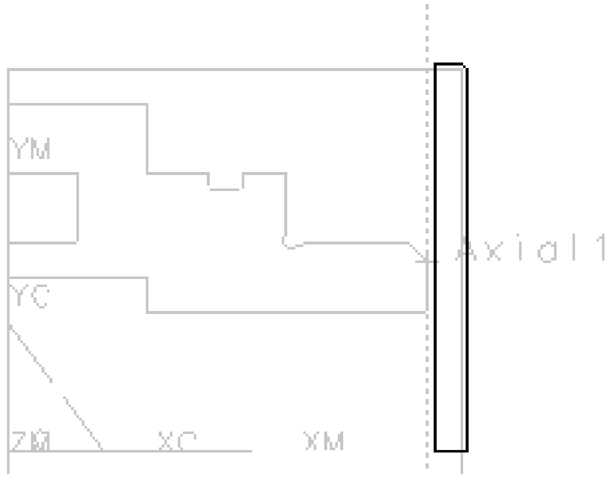


☐ Choose **OK**.

Step 5: Display the cut region.

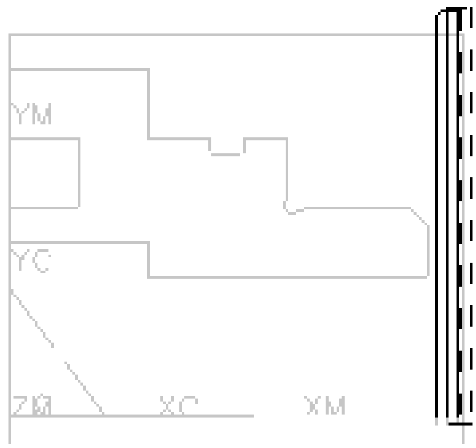
It is always a good practice to display the cut region before generating a path. You will then see what the tool will cut.

☐ In the Cut Regions area, choose **Display**.



Step 6: Generate the operation.

☐ Choose the **Generate** icon located at the bottom of the dialog.



The tool path is generated. The tool cuts the face of the part leaving the defined (default) stock on the face.

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

☐ **Save** and **Close** the part.

Summary

In this lesson you learned how to create Facing operations.

In this lesson you:

- Specified a Cut Region Containment area to be machined in a Facing operation.

- Examined default Rough Stock values.

Lesson

5 *Verification*

Purpose

You will use the Verification feature to visually validate single and multiple tool paths.

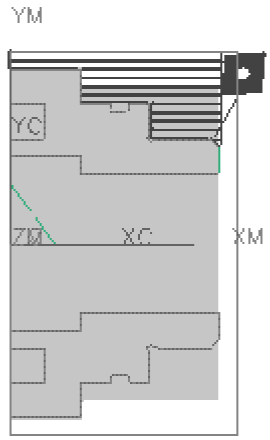
Objective

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

- Use the Show 2D and 3D options to view the In-Process Workpiece.
- Use the Show 3D option to compare the finished part to the current (In-Process Workpiece) part.
- Use Toolpath Visualization to verify cutting and non-cutting tool motions.

Verifying Cut Motion and Blank Material

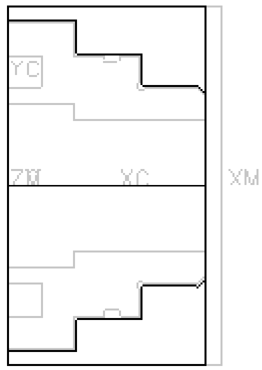
You can visually examine the cut motion and the remaining material of one or more operations in Turning using Toolpath Visualization.



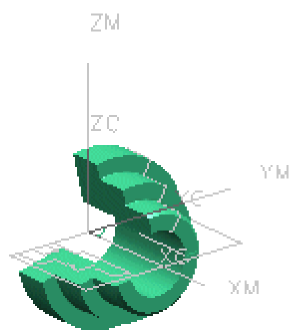
Verifying the In-Process Workpiece Using Show 2D / 3D

You can graphically display the In-Process Workpiece by highlighting the operation in the Operation Navigator and choosing **MB3→Workpiece→Show 2D** or **Show 3D**. Since the sequence of operations is important, it is best to select the operation in the Program Order view of the Operation Navigator. If operations are reordered, the system recalculates the In-Process Workpiece.

Show 2D



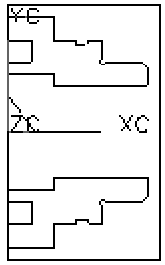
Show 3D



Activity: Verification Using Show 2D and 3D Options

Step 1: Open the part file.

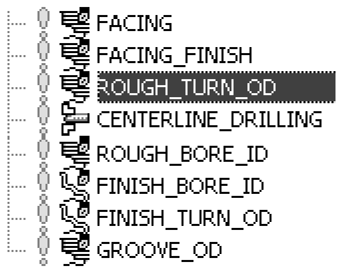
☐ Open the part file **tmp_verify_1.prt**.



☐ Enter the **Manufacturing** application.

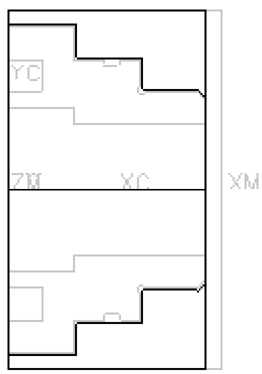
☐ If necessary, change to the **Program Order View**.

☐ In the Operation Navigator, highlight the operation **ROUGH_TURN_OD**.



☐ With MB3, choose **Workpiece** →**Show 2D**.

The results are displayed in the graphics window. The red geometry is the cut region. The white geometry is the in-process workpiece (refresh dismisses the results).

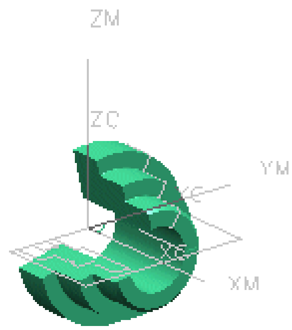


Step 2: Examine the cut motion results in 3D mode.

☐ **Refresh** the graphics window.

- ☐ In the Operation Navigator, highlight the operation **ROUGH_BORE_ID**.
- ☐ Change to the **Trimetric View**.
- ☐ With MB3, choose **Workpiece→Show 3D**.

  The in-process workpiece is displayed in 3D.
- ☐ With MB3, choose **Display Mode→Shaded**.



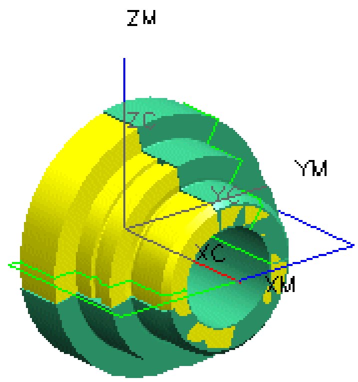
Step 3: Compare the part to the In-Process Workpiece.

You will compare the solid finished part to the In-Process Workpiece. The part has been modeled as a solid and is displayed. The solid model is located on a different layer. You will make the layer visible and then use Show 3D to compare the part to the In-Process Workpiece.

- ☐ Choose **Format→Layer Settings**.
- ☐ Double-click on layer **110**.

☐ Choose **OK**.

The solid part is displayed. Note that the yellow portion shows the finished part and the green shows the In-Process Workpiece part.

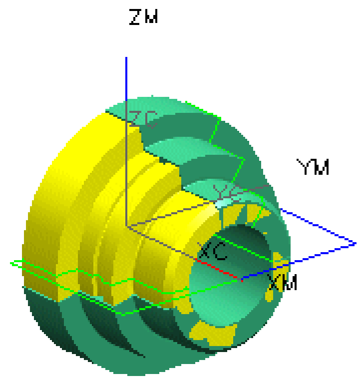


☐ Do not Save or Close the part file.

Activity: Verification Using Toolpath Visualization

Step 1: Visualizing a Single Operation

- ☐ Continue to use the same part file, **tmp_verify_1.prt**.



- ☐ In the Operation Navigator, highlight the operation **ROUGH_TURN_OD**.

- ☐ With MB3, choose **Tool Path→Verify** or choose the **Verify Toolpath** icon. 

  The Tool Path Visualization dialog is displayed.

- ☐ Turn the **2D Material Removal** option **on**. This will display the blank material.

- ☐ At the bottom of the dialog, slide the animation speed to **7**.

- ☐ Run the animation using the **Play** button. 

- ☐ Change to the **Top View**.

- ☐ Choose **Format→Layer Settings**.

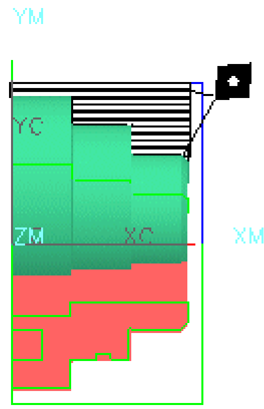
- ☐ Double click on layer **110**.

- ☐ Choose **OK**.

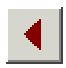
- ☐ Choose the **Rewind to Previous Operation** button.

- ☐ Run the animation using the **Play Forward** button. 

The tool path is replayed.



The Stop Animation message dialog allows you to end the animation before it has finished playing.

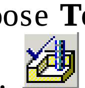
☐ Choose the **Reverse Play** button. 

This will replay the tool path backwards.

☐ Choose **OK**.

Step 2: Visualizing a Sequence of Operations.

☐ In the Operation Navigator, highlight the operations **FACING** through **ROUGH_BORE_ID**.

☐ With MB3, choose **Toolpath→Verify** or choose the **Verify Toolpath** icon. 

☐ Choose the **2D Material removal** option.

☐ Change the animation speed to **8**.

☐ Run the animation using the **Play** button. 

The tool paths are replayed in sequence in their entirety.

☐ Choose **OK**.

☐ Do not save the Part.

☐ **Close** the Part File.

Summary

In this lesson, you used three different methods to verify operations. Two of the methods displayed the In-Process Workpiece.

In this lesson you:

- Visually inspected the operations and In-Process Workpiece using Show 2D.
- Visually inspected the operations and In-Process Workpiece using Show 3D; this method displayed the In-Process Workpiece as a 3D solid.
- Replayed and visually inspected several operations in the program using Toolpath Visualization.
- Compared the solid part to the In-Process Workpiece.

Lesson

6 *Common Options*

Purpose

The purpose of this lesson is to introduce you to many of the procedures that you will use in creating tool paths. These are procedures that do not depend upon the type of operation you choose, but are general requirements for all or most operations.

Objective

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

- Use From, Return, and Start points within an operation.
- Add and remove rough stock in an operation.
- Use the tool Engage/Retract options within an operation.
- Set the feed rates within an operation.

The Common Options Discussed in this Lesson

- stock
- avoid - Return and Start Point, From Point
- engage/retract
- feed rates

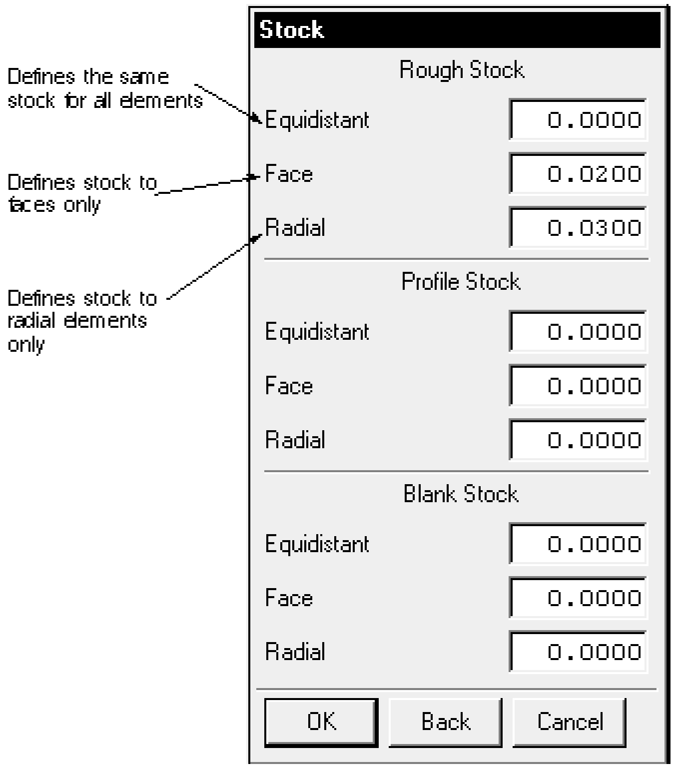
Common Options in Turning

Option	Rough	Finish	Thread	Drill	Teach Mode
Level Angle	X				
Reversal Mode	X	X			
Containment	X	X			
Autodetection	X	X			
Engage/Retract	X	X	X		X
Cutting	X	X			X
Corner	X	X			X
Feed Rates	X	X	X	X	X
Profile	X	X			
Stock	X	X			X
Machine	X	X	X	X	
Avoidance	X	X	X	X	
Local Return	X	X			

Stock

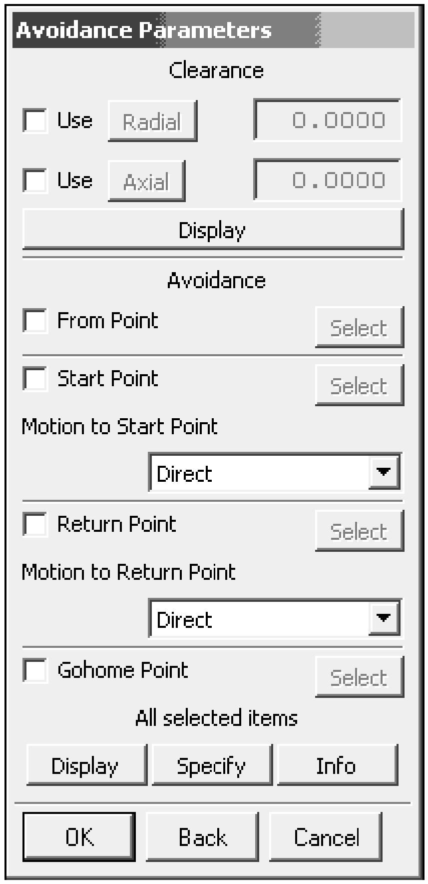
The Stock function allows you to leave material on the part.

- Rough Stock - leaves material after roughing
- Profile Stock - leaves material after profiling
- Blank Stock - adds material to the work piece



Avoidance Geometry

The Avoidance option enables you to define clearance motions for the tool as it moves *toward* or *away* from the part. You can specify points or planes to help you define these clearance motions.



FROM Point defines the initial cutter location at the start of a tool path. The FROM/ command is output as the first entry in the tool path, but there is no tool movement. It is advisable to specify a FROM point when using the GPM for postprocessing the tool path since the first GOTO point is removed from the output.

Start Point defines the tool position location in the cutter start up sequence. If a From Point has been defined, the cutter will rapid from it to the Start Point**.** The Start Point outputs a GOTO/ command at the rapid feed rate after the FROM/ and post commands and before the first Engage move.

Return Point defines the tool location at the end of a cutting sequence. The Return Point outputs a GOTO/ command at the Rapid feed rate following the Final Retract motion.

GOHOME Point outputs a GOHOME/ command as the final entry in the tool path. The From Point is often used as the GOHOME point.

Clearance defines a safe distance from the workpiece along the radial or axial positions, for positioning moves to the Start and Return points.

Creating Associative Points

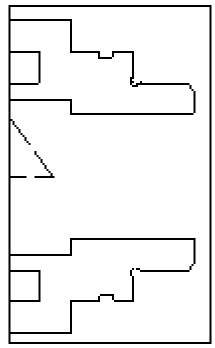
Points, Vectors, and Planes (usually used for avoidance geometry) can be created as associative geometry so that when the referenced geometry is modified, the associated geometry is also modified.

Activity: Stock and Avoidance Options

In this activity, you are going to create an operation to finish the face of the part by removing the Rough Face stock. You will edit the Rough Face operation, add a From point. a Start point, and a Return point in order to gain more tool control between operations.

Step 1: Open the part, rename it and enter the Manufacturing application.

☐ Open the part **tmp_common_1.prt**.



☐ **Save** the part file as *****_common_1.prt**, where *** represents your initials.

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

Step 2: Replay the existing operation.

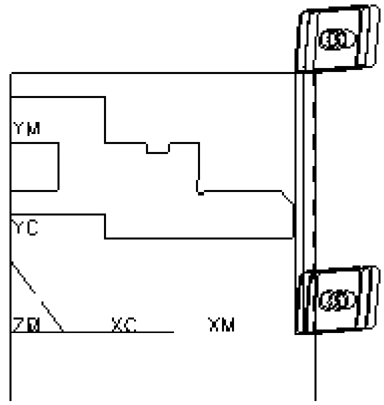
☐ In the Operation Navigator, display the **Program Order View**, if necessary.

☐ Expand the Parent Group object.

The operation ROUGH_FACE is listed.

☐ Highlight the **ROUGH_FACE** operation name in the Operation Navigator, then choose the **Replay Toolpath** icon.

The tool removes material in two passes leaving .02 (the roughing default) stock on the face.



Step 3: Add a From, Start, and Return Point to this operation.

☐ Double-click on the **ROUGH_FACE** operation.

You are going to add several Avoidance settings.

☐ Choose **Avoidance**.

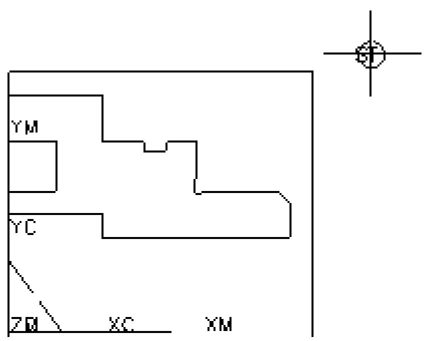
☐ Check the **Start Point** and **Return Point** options.

You can specify the Start and Return points in one step.

☐ Choose **Specify** from the dialog.

☐

Using the Inferred Point method, place your cursor in the graphics window and click twice.



The point indicators are placed on top of each other.

You should always create a From point in the first operation of your program in order to orient the tool.

☐ Check the **From Point** option.

- ☐ Choose **Specify** from the dialog.

  In many cases, you may have a specific From point location that you use with each machine. In this case a position is provided for you.

- ☐ Enter the following values: XC=6; YC=5.

- ☐ Choose **OK**.

  The cue line indicates that you can now specify a new Start point. The point location for the previously defined Start Point is displayed in the Base Point area. You can choose Cancel or OK. If you choose Cancel you can reselect a new Return point. If you choose OK the displayed location will be used.

- ☐ Choose **OK** until you return to the FACING dialog.

- ☐ Choose the **Generate** icon.

- ☐ Choose **OK** to accept the tool path.

  The ROUGH_FACE tool path is generated. Note the tool moves from the From Point then to the Start point, returning to the Return point.

  You will copy this tool path, edit the method and stock, and then generate the path.

Step 4: Copy and rename the operation.

You are going to copy the operation. When you copy the operation, the Avoidance parameters you just defined will already be set.

- ☐ Highlight the **ROUGH_FACE** operation name in the Operation Navigator, then (using MB3) choose **Copy**.

  The operation is copied. Now you will paste it into the list on the Operation Navigator.

- ☐ With the **ROUGH_FACE** operation name highlighted in the Operation Navigator, use MB3 to **Paste** the copied operation.

  Change the operation name. This step will help distinguish it from other operations.

- ☐ Make sure that the operation that you just copied is highlighted, using **MB3→Rename**, rename the operation to **finish_face**.

Step 5: Edit the operation.

☐ Double-click the **FINISH_FACE** operation name in the Operation Navigator.

The Facing dialog is displayed. The first option you will change is the Method.

☐ Choose the **Groups** tab at the top of the Facing dialog.

☐ Choose **Method**.

☐ Choose **Reselect**.

The Method dialog is displayed.

☐ Choose **LATHE_FINISH** from the list, and then **OK**.

The FACING dialog is displayed.

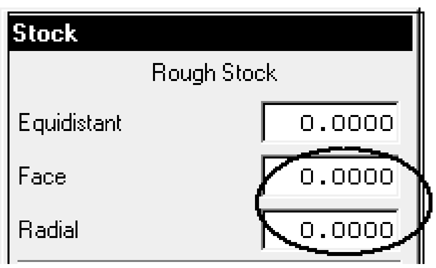
Step 6: Edit the Stock values of the operation.

☐ Choose the **Main** tab.

☐ Choose **Stock**.

The Stock dialog is displayed. You are going to check that the stock values are set to zero.

☐ Verify the stock values are zero **0**.



☐ Choose **OK**.

Step 7: Display the Cut Region.

It is always a good practice to display the cut region before generating a path.

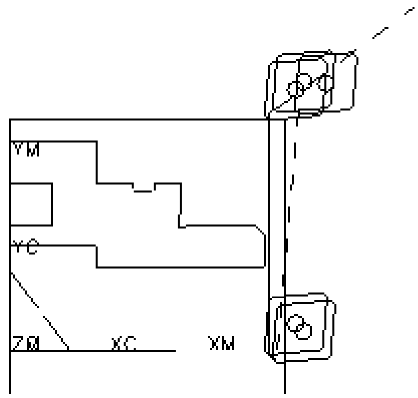
☐ In the Cut Region area, choose **Display**.

The cut region is displayed.

Step 8: Generate the tool path.

☐ Choose the **Generate** icon.

The tool path is generated.



You are going to edit the Start point in the FINISH_FACE operation.

Step 9: Edit the Start Point.

☐ Choose **Avoidance**.

The Avoidance Parameters dialog is displayed.

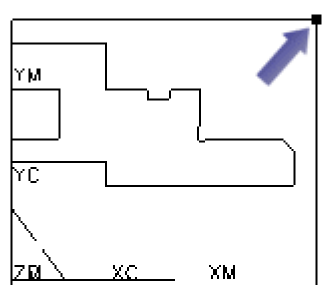
☐ Choose **Reselect** next to the Start Point option.

The Point Constructor dialog is displayed. This Start point is going to be a smart Start point. You will associate the Start point to the End point of the Blank. If the Blank size changes, the Start point will also change, preventing the tool from crashing into the Blank.

☐ Change the Offset to **Rectangular**.

☐ Choose the **End Point** icon.

☐ Choose the end point of the blank geometry as shown.

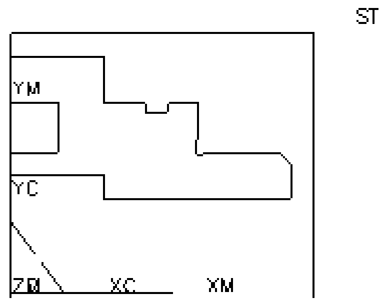


☐ Enter **1.00** into the XC value field.

☐ Enter **.25** into the YC value field.

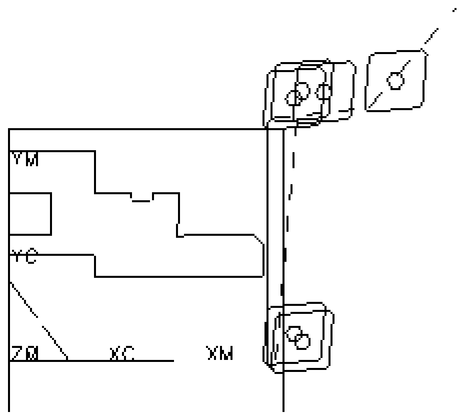
☐ Choose **OK**.

The Start point is displayed by **ST** on the graphics window.



☐ Choose **OK** to return to the FACING dialog.

☐ **Generate** the tool path.



The tool moves to the Start point above centerline and then moves to the first cut position.

You are going to remove the From point since this is not the first operation in the program.

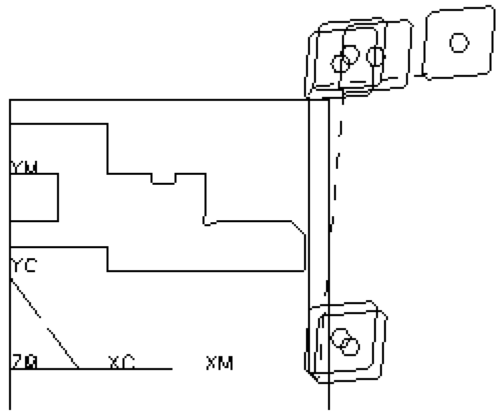
Step 10: Removing the From Point.

☐ Choose **Avoidance**.

☐ Choose the **From Point** option so that there is no longer a check mark in front of it.

☐ Choose **OK** to return to the FACING dialog.

☐ **Generate** the tool path.

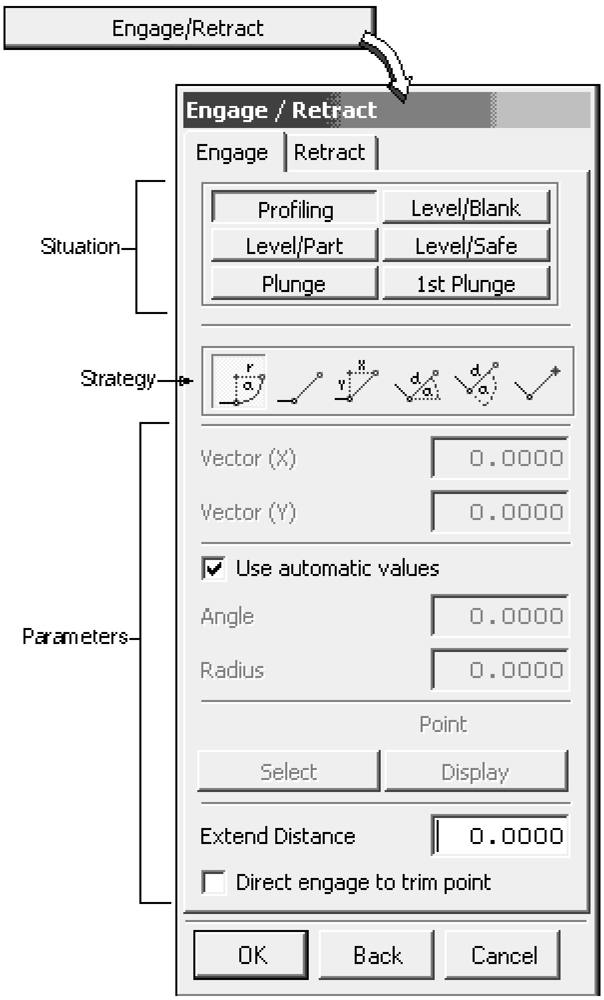


☐ Choose **OK** to accept the tool path.

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

Engage and Retract Moves

Engage and Retract moves control the tool motion to bring the tool safely into contact with the Blank and Part to begin each machining pass. They also control toolretract motion safely from the work piece at the end of each machining pass. *F*or each engage or retract situation, you see the selected strategy together with the specified parameters.



The engage and retract conditions are defined as follows:

Option	**Description**
Profiling	The engage/retract that is applied to the start or end of a profile pass.
Level/Blank	the **engage** is applied when a roughing pass cuts only blank material (does not contact any part material). The **retract** is applied when the rough pass cuts to the end of the blank, without cutting any part material.
Level/Part	The **engage** is applied whenever a rough pass in a reversal engages to part material. The **retract** is applied whenever a rough cut stops at part geometry. The retract motion is applied after a rough cut or — if applicable — after the cleanup cut.
Level/Safe	The engage that is applied to the last pass of roughing when the pass contours the part shape.
Plunge	The engage/retract that is used in the plunging passes in grooving.
1st Plunge	Gives separate control for engage and retract for the first plunge pass.

Engage and Retract Strategies

Each of the previous listed Engage and Retract options allows you to define specific Engage/Retract Strategies. Below is a table that indicates which Strategy options are available for each Engage/Retract Situation.

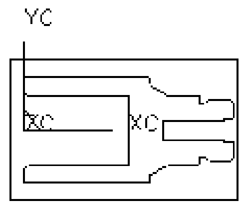
Option	**Profile**	**Level/ Blank**	**Level/Part**	**Level/Safe (no Retract)**	**Plunge and 1st Plunge**
Auto Circular	X				
Auto Linear	X	X	X	X	X
Vector	X	X	X	X	X
Angle/ Distance	X	X	X	X	X
Relative Linear	X				
From a Point (eng) To a Point (ret)	X	X	X		X (no ret)
Two Circles		X			
2 Pt. Tangent			X	X	

Activity: Engage and Retract Moves

In this activity, you are going to edit an existing operation and examine the automatic engage and retracts. You will then change and add different engage and retract moves, then examine the results. The purpose of this activity is to introduce some of the many methods of engagement and retracting.

Step 1: Open the part, rename it, and enter Manufacturing.

☐ Open the part **tmp_common_2.prt**.



☐ Save the part file as *****_common_2.prt**, where *** represents your initials.

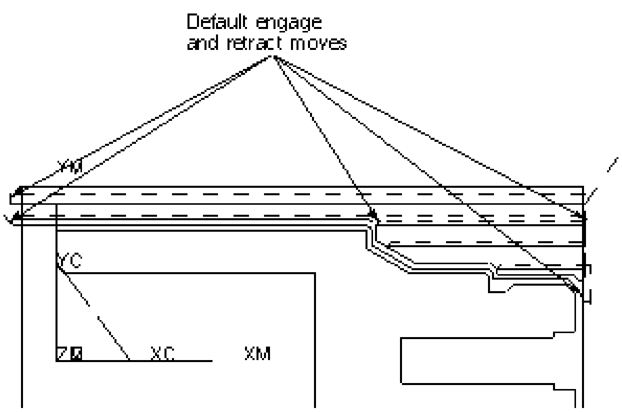
☐ Choose **Application→Manufacturing**, if necessary.

This session has been initialized using the Lathe Configuration and the Lathe CAM setup.

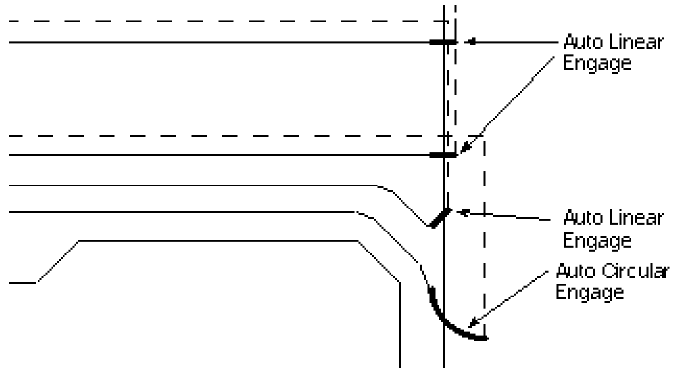
Step 2: Replay the existing operation.

☐ In the Operation Navigator Program Order view, expand the Parent Groups.

☐ Highlight the **ROUGH_TURN_OD** operation name in the Operation Navigator, then choose the **Replay Toolpath** icon.



The tool path is replayed. Notice the default engage and retract moves. All engages and retracts are Auto Linear except those for the Profile cut which uses Auto Circular.



You will now change some of the engages and retracts.

Step 3: Create an Engage move on a Profile pass.

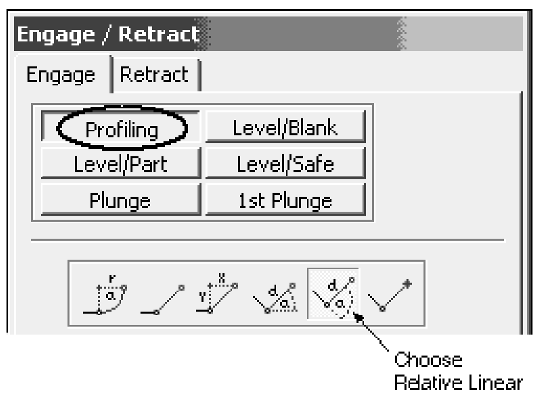
You will change the engage move to use the Relative Linear strategy. The tool will use this move when engaging the part material on a profile pass.

☐ Double-click on the **ROUGH_TURN_OD** operation.

☐ Choose **Engage/Retract**.

The Engage/Retract dialog is displayed. The default engage for Profiling is Auto Circular.

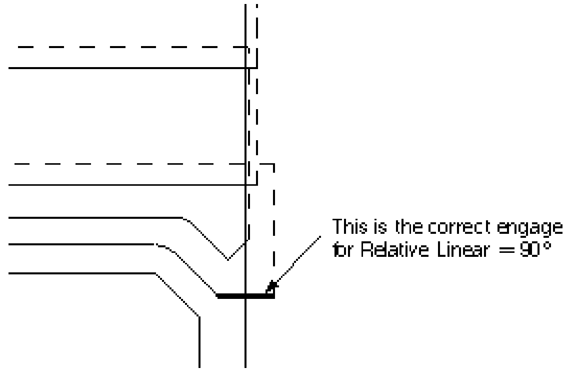
☐ Click the **Relative Linear** icon.



☐ Enter the following values: Angle = 90; Distance = .05.

☐ Choose **OK** to return to the ROUGH_TURN_OD dialog.

☐ **Generate** the tool path.



The tool path is generated. The tool now engages at 90 degrees into the material on the Profile pass.

Step 4: Create an Engage move when engaging the Blank material.

This time you will disable the automatic setting on the linear engage type and input manual values.

☐ Choose **Engage/Retract**.

☐ Choose **Level/Blank**.

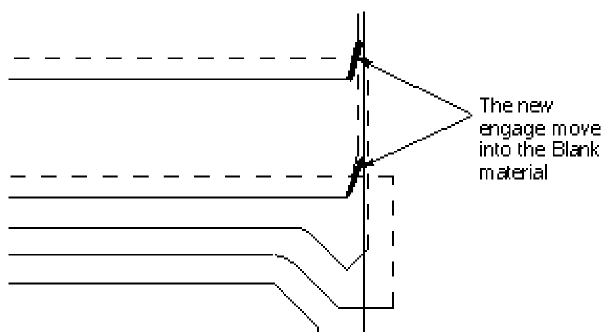
The default engage for Level/Blank is Auto Linear.

☐ Deselect the **Use automatic values** option.

☐ Enter the following values: Angle = 255; Distance = .05

☐ Choose **OK** to return to the ROUGH_TURN_OD dialog.

☐ **Generate** the new tool path.



The tool path is generated. The tool now engages at 255 degrees into the Blank material.

Step 5: Create a Retract move.

☐ Choose **Engage/Retract**.

☐ Choose the **Retract** tab.

☐ Be sure **Profiling** is selected.

☐ Choose the **Vector** icon.

☐ Enter the following values: Vector X = 0; Vector Y = .15.

Step 6: Create a Retract move when exiting the Part material.

☐ Choose **Level/Part**.

☐ Choose the **Angle/Distance** icon.

☐ Enter the following values: Angle = 90; Distance = .1.

☐ Choose **OK** to return to the ROUGH_TURN_OD dialog.

Now you are ready to generate the tool path with the new settings.

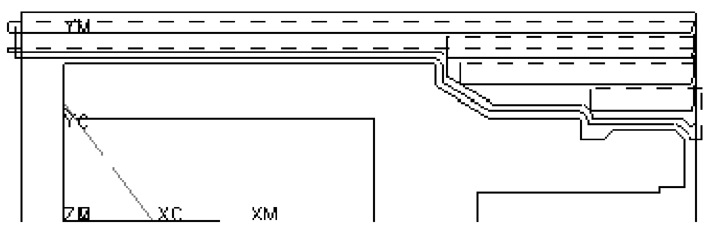
☐ **Generate** the tool path.

The first retract is the Level/Blank retract. It is using the default Auto Linear.

The second pass retract uses the Level/Part retract, which uses Angle/Distance to define the move.

The last pass retract is the Profiling pass which uses a Vector to define the move.

Note that the retracts look the same. When you specified a Vector of X=0 and Y=.15 for the Profile retract and Angle/Distance of 90 and .1 for the Part retract move, they produce the same result. The result is a direct move along the Y axis. The only difference is that the profiling move is longer than the Part Retract move.



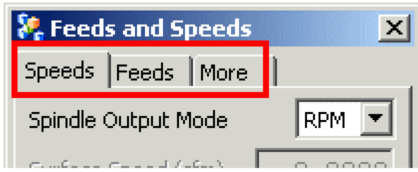
Step 7: Change the Engage and Retract types.

☐ Now try some of the other Engage and Retract strategies, such as To or From a Point. Examine the results.

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

Feed Rates

The Feed rates dialog contains a variety of feed rate control parameters that allow you to "fine-tune" the Turning tool path results. You access the options using one of the tabs at the top of the dialog.



The first tab contains surface spindle speed adjustments.

The second tab contains feed rate adjustments for each type of tool move and cut.

The third tab contains feed rate adjustments for roughing and profiling passes.

Summary

In this lesson, you learned to use some of the options that are commonly used within Turning.

In this lesson you:

- Discussed and used some of the Stock options.
- Defined the From, Start, and Return points in an operation.
- Used several different Engage and Return moves within an operation.
- Discussed the different types of feed rate options available within Turning.

Lesson

7 *Centerline Operations*

Purpose

This lesson will describe the use of Centerline operations.

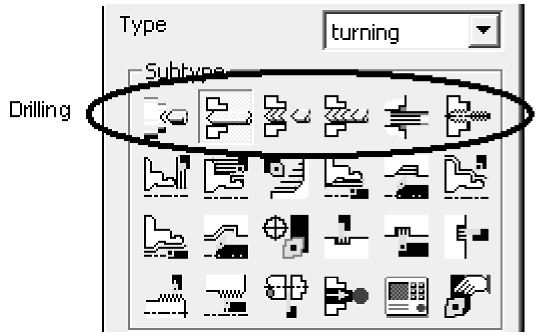
You will create several Lathe Drilling operations. You will specify the tool start point, the drill depths, and learn how to use the break chip methods.

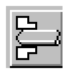
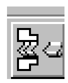
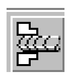
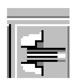
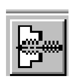
Objective

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

- Create Centerline operations.
- Use the Break Chip method.

Drilling Operation Templates



- CENTERLINE_SPOTDRILLING 
- CENTERLINE_DRILLING 
- CENTERLINE_PECKDRILL 
- CENTERLINE_BREAKCHIP 
- CENTERLINE_REAMING 
- CENTERLINE_TAPPING 

The Create Drill dialog

The following topics are discussed in this lesson:

- Drill Geometry

  Start Point

- Total Depth

  Depth

  Break Through

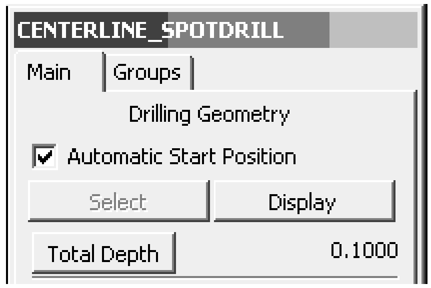
- Chip Removal

  Method

  Increment Type

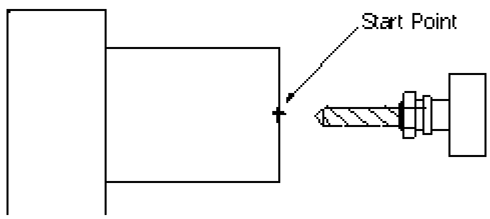
- Minimum Clearance

Drill Geometry



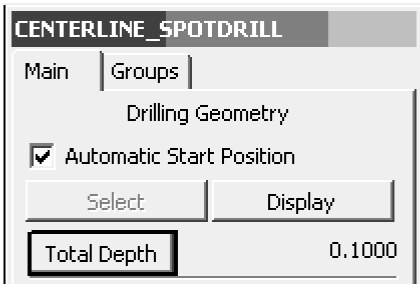
Automatic Start Position determines the start position based on the current IPW and tool shape.

When you choose Select or Reselect, under the Drilling Geometry label, the Point Constructor dialog is displayed. You are then prompted to specify a Start Point. The Start Point is the point at which the tip of the drill first contacts the part.

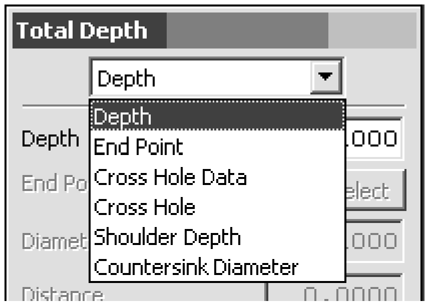


Total Depth

The total depth option is used to specify the depth that the drill will cut.



There are 6 options available:



Depth requires that you key in a positive value next to the Depth label. This value is applied along the drill axis (parallel to the centerline through the Start Point).

End Point uses the Point Constructor dialog to specify the drilling end point.

Cross Hole Data requires that you enter values for:

Diameter – the diameter of the cross hole.

Distance – the distance between the drill Start Point and the intersection of the cross hole axis with the drill axis (parallel to centerline through the start point).

Angle – the angle that the cross hole axis makes with the drill axis (parallel to the XM or YM axis).

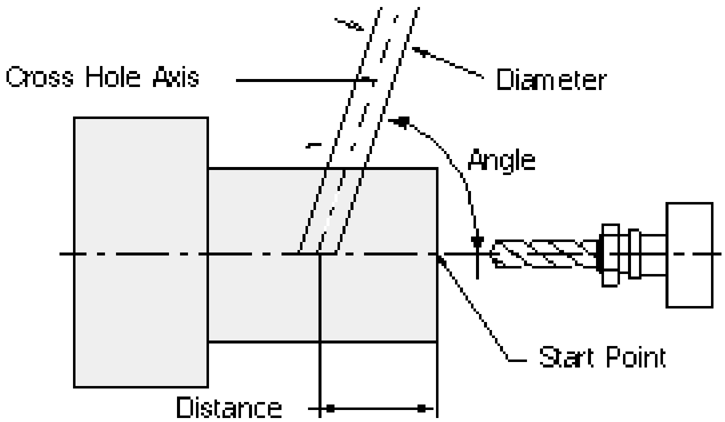
When you enter the data to define the cross hole, the depth value is calculated.

Cross Hole allows you to select an existing circle as a cross hole. The drilling depth is based on the distance from the start point along the drilling axis to the selected cross hole. The drill depth is calculated such that the drill fully pierces the side of the cross hole and then retracts. It allows you to specify the Cross Hole:

- using the cursor
- entering the depth
- specifying an end point

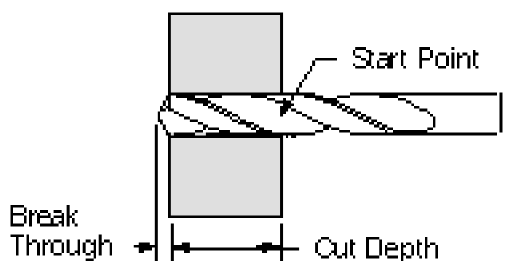
Shoulder Depth allows you to specify the drilling depth by referencing the shoulder of the tool. The tracking point is established at the tip of the tool by adding the calculated tool tip length to the specified shoulder depth.

Countersink Diameter determines the cut depth by allowing you to specify the diameter to be cut by the spot drilling tool.



Break Through

The **Break Through** option controls the distance that the tip of the tool can exceed the specified drilling depth.



Chip Removal

You can use the templates provided or change the Chip Removal option within the Standard Drill dialog.

Methods

The Break Chip or Peck Drill method is used for clearing chips.

Break Chip periodically retracts the tool a short distance at the feed rate you specify to break the chip.

Peck Drill periodically withdraws the tool from the hole at the departure feed rate to clear off the chips.

Increment Type

After you specify a method, you must specify whether the distance that the drill will move before clearing the chips (Increment Type) is Constant or Variable.

Increment refers to the distance the drill moves forward with each move.

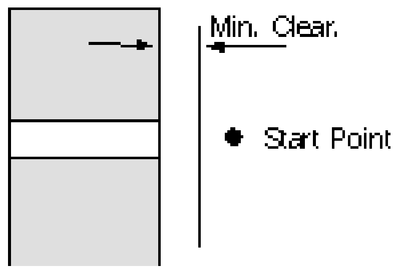
Departure Distance refers to the distance the drill moves backward.

Number of Cuts refers to the number of times you wish to repeat the specified Variable Increment.

Minimum Clearance

The Minimum Clearance is a safety zone around the workpiece for non-cutting tool moves.

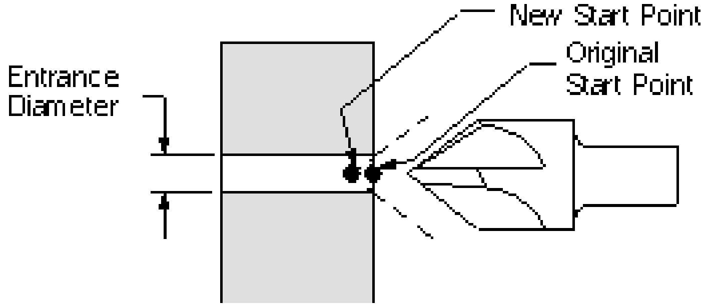
The tool will Rapid to the Minimum Clearance distance from the specified Start Point, and then proceeds at the cut feed rate.



At the end of the drilling operation, the tool returns to the Minimum Clearance distance before going to the Return and/or GOHOME points.

Entrance Diameter

Entrance Diameter uses the diameter of an existing drilled hole to adjust the contact point of the drill with the material. The calculations are made using the defined entrance diameter and the drill point angle.



Dwell

This option allows you to specify a dwell interval (Delay) in tool motion at the end of a machining pass to relieve tool pressure.

You must first specify the units (SECONDS or REVOLUTIONS) and then the number of units when:

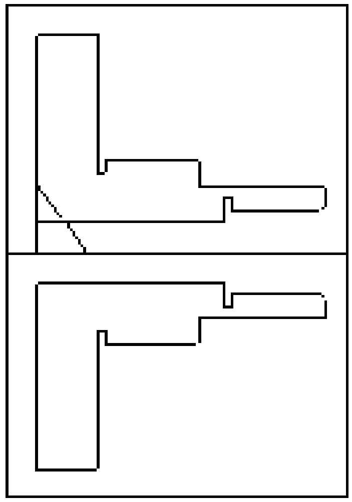
- Chip Removal is set to None or Break Chip, the DELAY/ command is output after the drilling to depth is complete and before the final move out of the drilled hole.
- Chip Removal is set to Peck Drill, the DELAY/ command is output after each move out of the drill hole.

Activity: Centerline Drilling - Spot Drilling

In this activity, you are going to create an operation to spot drill the face in preparation for a standard drilling operation. The tools for this operation are contained within the part file. You will define the Parent Groups, select the geometry, and examine the Total Depth options.

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

☐ Open the part file **tmp_drill_1.prt**.



☐ Save the part file as *****_drill_1.prt**, where *** represents your initials.

The cross section geometry has been pre-defined for this operation.

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

Step 2: Define the Parent Groups for this operation.

☐ Choose the **Create Operation** icon.

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

☐ Choose the **CENTERLINE_SPOTDRILL** icon from the Create Operation dialog.

The tool for this operation is a .500 center drill.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **ID**
- Use Tool = **SPOTDRILL_.500**

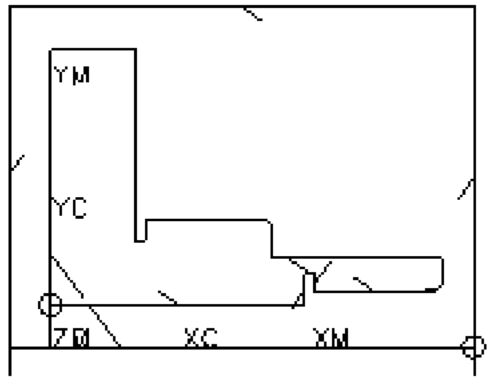
- Use Method = **LATHE_CENTERLINE**

☐ Choose **OK**.

The CENTERLINE_SPOTDRILL dialog is displayed.

☐ Choose the Groups tab.

☐ Choose the **Geometry** button, then choose **Display**.



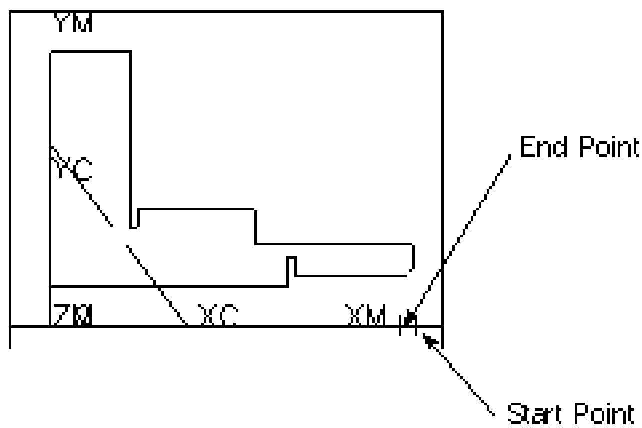
☐ Choose the Main tab.

Step 3: Define the drilling geometry.

You may define the geometry that will determine the drilling start point or allow the system to do so automatically.

☐ With the Automatic Start Position option turned **on**, choose **Display**.

The start position is determined automatically based on the current IPW and tool shape.

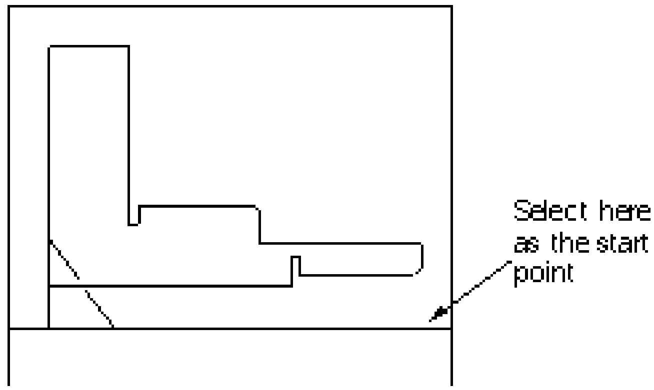


☐ Turn the Automatic Start Position option to **OFF**.

☐ Choose **Select**.

The Point Constructor dialog is displayed. You will select the horizontal line (centerline).

☐ Select the line towards the right end as shown.



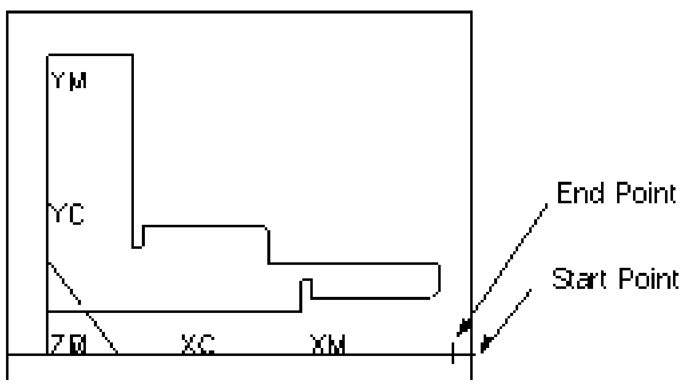
The CENTERLINE_SPOTDRILLING dialog is displayed.

Step 4: Examine the Total Depth for the Spotdrilling operation.

Examine the default depth value that is defined for a spotdrilling operation.

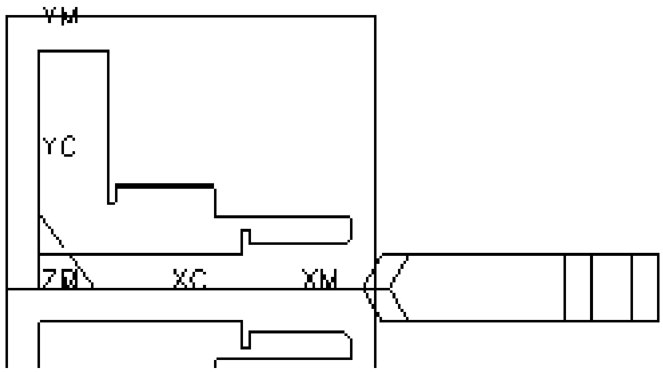
The default depth is .100. You will use this setting.

The horizontal line that you selected is displayed along with the start and end point indicator.



Step 5: Generate the tool path.

☐ Choose the **Generate** icon.



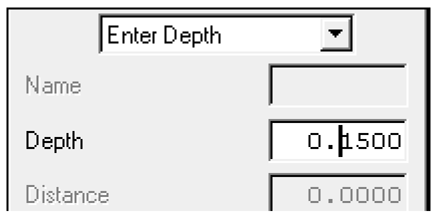
The tool path is generated. The tool moves along the centerline to the end position (.100 from the blank material).

Step 6: Change the depth value for the end point.

You are going to change the drill depth for this spotdrill operation.

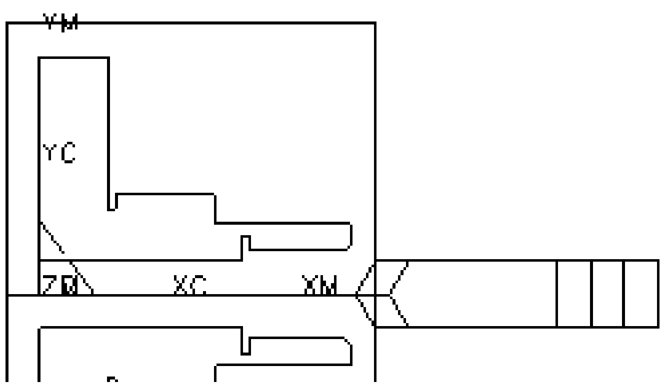
☐ Choose the **Total Depth** button in the Drilling Geometry area.

☐ Enter **.150** into the value field and then choose **OK**.



Step 7: Generate the tool path.

☐ Choose the **Generate** icon.



The tool path is generated. The tool moves along the centerline to the end position (.150 from the blank material).

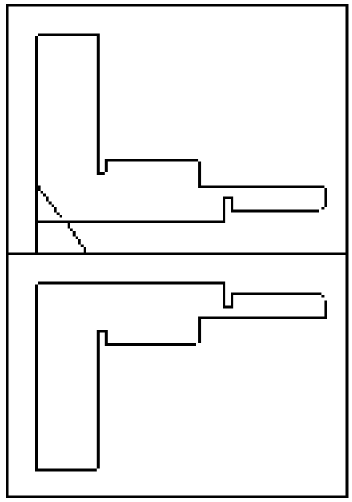
Step 8: **OK** to complete the operation and save the part file.

Activity: Centerline Drilling - Standard Drilling

In this activity, you are going to create a standard drilling operation. You will define the Geometry, the Total Depth, and then examine the Chip Removal options.

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

☐ Continue to use the part file *****_drill_1.prt**.



Step 2: Define the Parent Groups for this operation.

☐ Choose the **Create Operation** icon.

☐ Choose the **CENTERLINE_DRILLING** icon from the Create Operation dialog.

The tool for this operation is a .470 standard drill.

You only need to change the drill for this operation.

☐ Change the following Parent Group:

- Tool = DRILL_.470

☐ Choose **OK**.

Step 3: Define the drilling geometry.

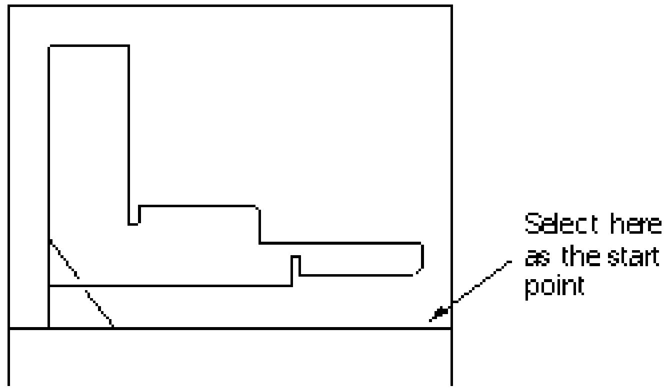
You must define the geometry that will determine the drilling start point.

☐ Turn the Automatic Start Position option to **OFF**.

- Choose **Select**.

  The Point Constructor dialog is displayed. You will select the horizontal line (centerline) to define the Start point.

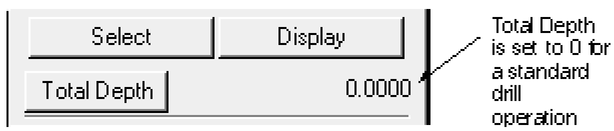
- Select the centerline towards the right end as shown.



The CENTERLINE_DRILLING dialog is displayed.

Step 4: Specify the Total Depth for the Drilling operation.

Examine the depth value that is defined for a drilling operation. It is set to 0.

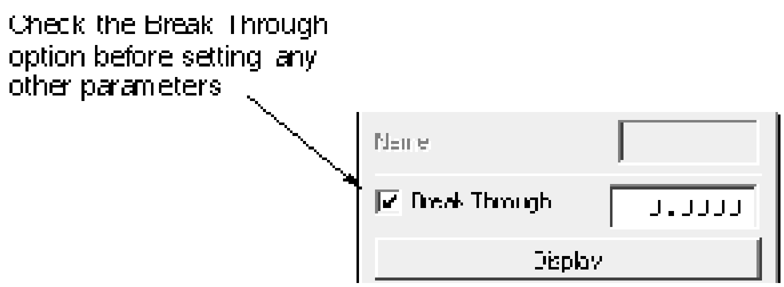


☐ Choose the **Total Depth** button in the Drilling Geometry area.

The Total Depth dialog is displayed.

You want the tool to drill through the part.

☐ Check the **Break Through** option located at the bottom of the dialog.



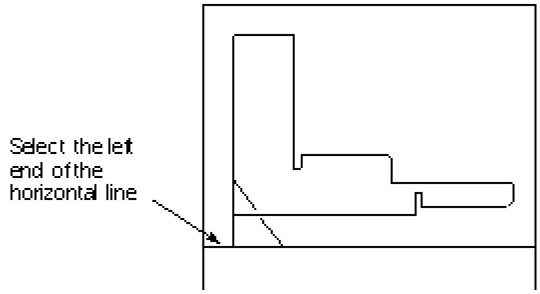
You will specify a Point to determine the drill depth.

☐ Change the Total Depth to **End Point**. This option is located at the top of the dialog.

☐ Choose **Select**.

The Point Constructor dialog is displayed.

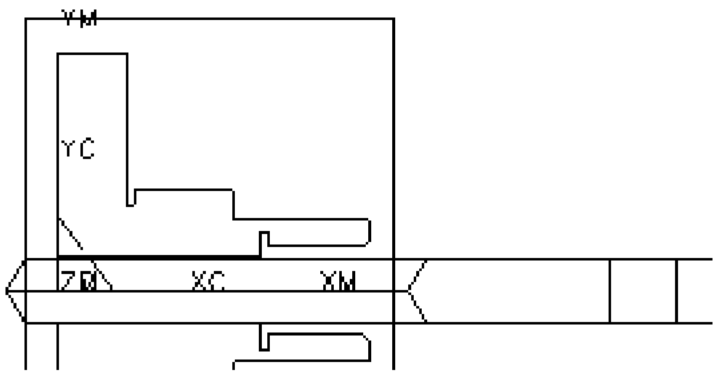
☐ Select the centerline towards the left end as shown and choose **OK**.



The CENTERLINE_DRILLING dialog is displayed.

Step 5: Generate the tool path.

☐ Choose the **Generate** icon.



The tool path is generated. The tool moves along the centerline to the end position, drilling through the part.

Step 6: Change the Chip Removal setting.

You can change the Chip Removal setting (Break Chip or Peck Drill) within an operation.

☐ Change the Chip Removal to **Peck Drill**.

☐ Choose **Settings** and change the Increment Type to **Variable**.

☐ Enter the following values in order into the list:

- Number of Cuts 2 Increment .35
- Number of cuts 4 Increment .25
- Number of cuts 4 Increment .15
- Number of cuts 2 Increment .05

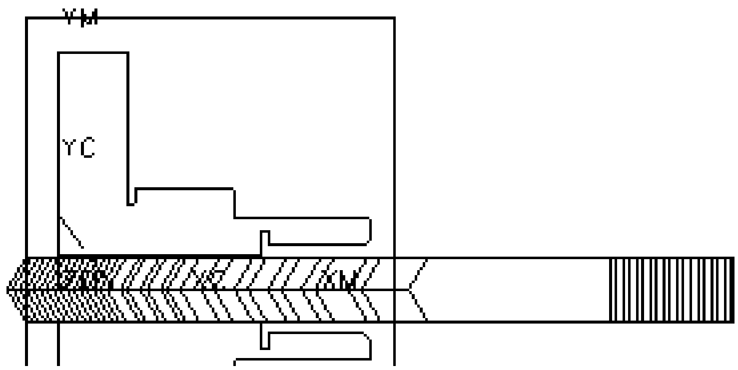
Note the Safe distance option. This determines how far the tool will retract between each cut. Use the default setting of .100.

☐ Choose **OK**.

Step 7: Generate the tool path.

☐ **Reject** the tool path.

☐ Choose the **Generate** icon.



The tool path is generated. The tool moves along the centerline to the end position, drilling through the part using the values that you entered. Note the last passes of the tool path. You specified two cuts at .05. The tool used these values, but there was material remaining. The system continued to cut to the specified depth using the last specified value (.05).

Step 8: Examine the Minimum Clearance setting in the tool path listing.

☐ Choose the **List** icon at the bottom of the CENTERLINE_DRILLING dialog.

The data is listed in the window.


```
GOTO/2.1500,0.0000,0.0000
PAINT/COLOR,7
FEDRAT/0.0500
GOTO/2.6000,0.0000,0.0000
PAINT/COLOR,1
RAPID
GOTO/2.2500,0.0000,0.0000
PAINT/COLOR,6
FEDRAT/0.0300
GOTO/2.1500,0.0000,0.0000
PAINT/COLOR,3
GOTO/1.8000,0.0000,0.0000
PAINT/COLOR,7
FEDRAT/0.0500
GOTO/2.6000,0.0000,0.0000
PAINT/COLOR,1
RAPID
GOTO/1.9000,0.0000,0.0000
PAINT/COLOR,6
FEDRAT/0.0300
GOTO/1.8000,0.0000,0.0000
PAINT/COLOR,3
GOTO/1.5500,0.0000,0.0000
PAINT/COLOR,7
FEDRAT/0.0500
GOTO/2.6000,0.0000,0.0000
```

Move to a Minimum Clearance from the blank material

Move to the Minimum Clearance

Move to a Minimum Clearance distance above the last cut depth

Move to a Minimum Clearance from the blank material

The tool cuts to 2.15 then rapids to the minimum clearance distance of 2.6000. (The 2.6000 value is .100 from the face of the blank material). The tool then rapids to a minimum clearance distance of .100 away from the previous cut depth. The tool continues to cut in this manner until the tool clears the back of the part material.

☐ Dismiss the Information window.

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

Step 9: Verify the tool path

☐ Highlight the Program name in the Operation Navigator.

☐ Choose the **Verify Tool Path** icon on the Manufacturing Operations tool bar.

☐ Choose 2D Material Removal.

☐ Choose the **Play** button.

☐ Choose **OK** to complete the Tool Path verification.

Step 10: Save the part file.

Activity: Centerline Drilling - Unassisted

In this activity, you are going to create an operation to remove the material within the larger ID. You are going to create this activity using fewer instructions.

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

- ☐ Continue to use the part file *****_drill_1.prt**.

Step 2: Choose the CENTERLINE_DRILL icon and define the Parent Groups for this operation. Use the .612 diameter drill.

Step 3: Define the following options for this operation:

- ☐ Drill Geometry
- ☐ Total Depth
- ☐ Chip Removal (Optional)
- ☐ Dwell (Optional)

Step 4: Generate and accept the tool path.

Step 5: Save and Close the part file.

Summary

In this lesson, you created spot drilling and standard drilling operations to remove material from the ID of the part.

In this lesson you:

- Defined the Drill Start Point.
- Specified the drill Cut Depth using the Point method and the Depth method.
- Set the Break Through option.
- Used the Chip Removal option.
- Set the Chip Removal Increments option.
- Examined Minimum Clearance moves.

Lesson

8 Roughing Operations - OD Work

Purpose

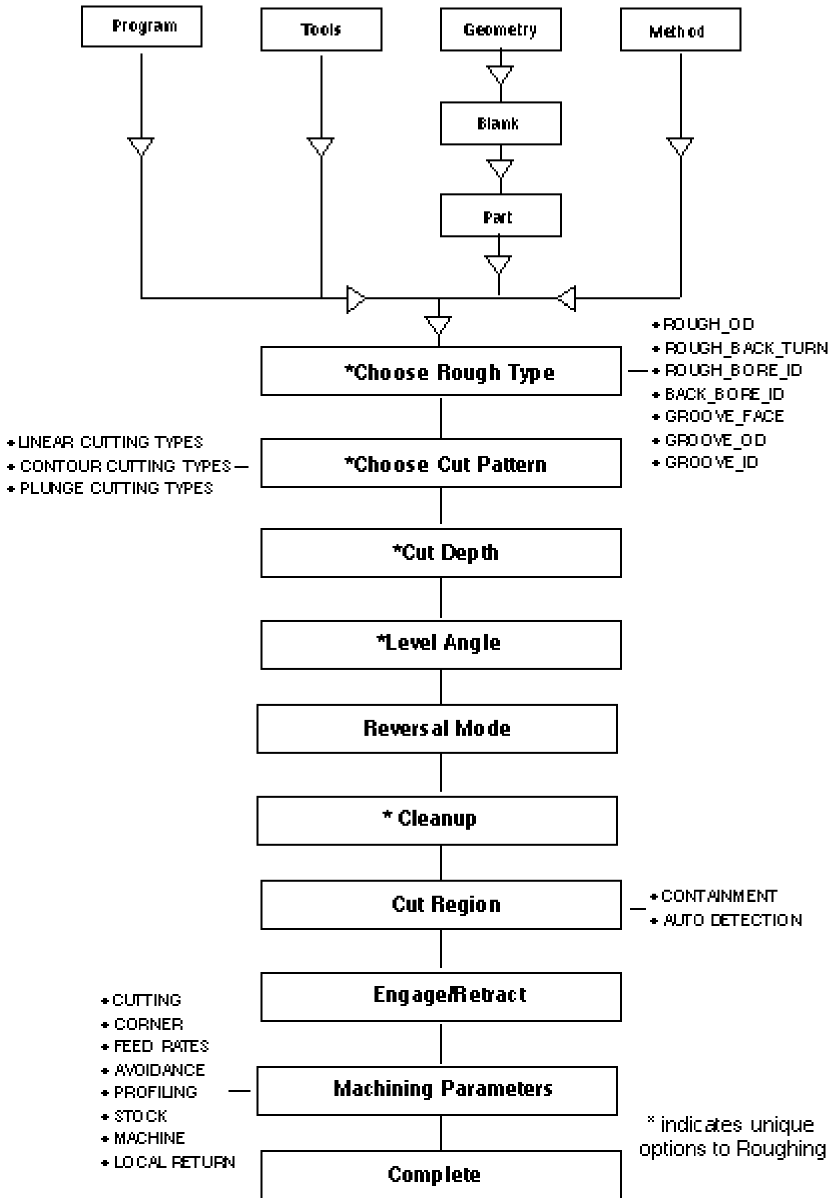
The roughing options in Turning allow you to remove large amounts of material and leave sufficient material for finishing operations.

Objective

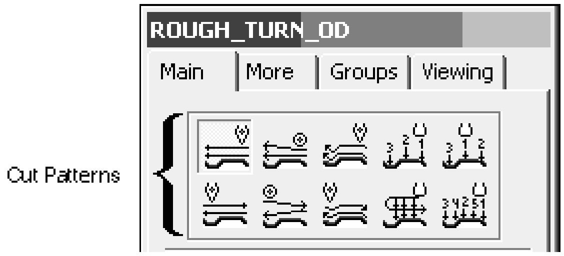
Upon completion of this lesson, you will be able to:

- Create a Rough Turn Outer Diameter operation.
- Change the Cut Pattern.
- Identify the appropriate Cut Depths.
- Add and remove Cleanup passes.
- Create additional Profiling passes (spring pass).
- Add a Local Return Point to the Spring pass.
- Add Post Commands to Local Return moves.
- Apply Single and Multiple Ramping Cut Strategies.
- Edit Part Boundary parameters from within an operation.

Roughing Options



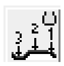
Cut Patterns

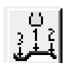


 **Linear Zig** is a straight cut in one direction. Each pass is parallel to the previous pass.

 **Ramping Zig** is for inclined/declined cuts in one direction.

 **Contour Zig** are parallel contour cuts in one direction. Each pass follows the parts profile.

 **Plunge Zig** are typical plunge cuts in one direction within the cut region.

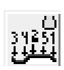
 **Plunge Alternate** cuts with alternating stepover direction. Each subsequent plunge is applied to the opposite side of the first plunge.

 **Linear Zig-Zag** is a straight cut with each subsequent pass cutting in an alternating direction.

 **Ramping Zig-Zag** is for inclined/declined cuts in alternating directions.

 **Contour Zig-Zag** are parallel contour cuts with alternating direction. Each pass follows the part profile.

Plunge Zig-Zag is for repeated plunge cuts to a specified level and in alternating directions.

 **Plunge Castling** is a unique plunge pattern which leaves pillars between plunges, then returns to cut the pillars. The result is even cutting pressure on the tool.

Stepover (Cut Depth)

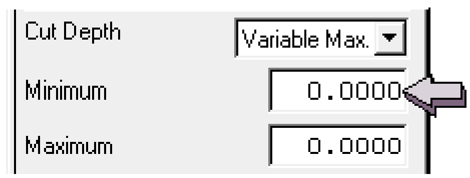
Stepover (for Plunge cuts) and Cut Depth (for Zig and Zig_Zag cuts) determine how the tool will move from one cut level to the next. The cut pattern determines which types are available.

Stepover/Cut Depth Types and Cut Patterns

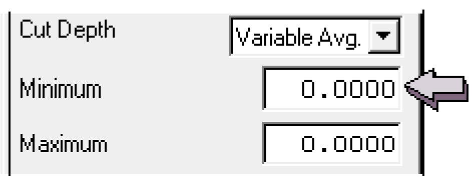
Types	Linear Zig	Linear Zig-Zag	Cont. Zig	Cont. Zig-Zag	Plunge Zig	Plunge Alt.	Plunge Castle
Const.	X	X	X	X	X	X	X
Variable Max.	X	X			X	X	X
Variable Avg.	X	X			X	X	X
# of Levels	X	X					
# of Passes			X	X			
Indiv.	X	X	X	X			

Constant specifies the maximum depth of cut to be made on each roughing pass. This value is used as many times as possible and then any remaining material is removed in one pass.

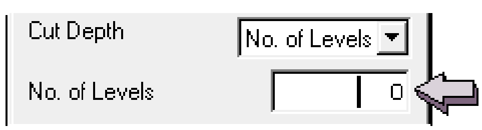
Variable Max. determines the cut regions, cuts at the maximum value as many times as possible, and then removes the remaining material in one pass if it is greater than or equal to the minimum within each separate region.



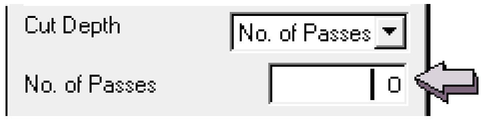
Variable Avg. calculates the minimum number of passes required based on never cutting more than the maximum or less than the minimum for each region.



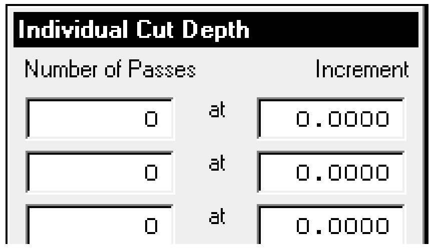
# of Levels produces equal depths of cut by allowing you to define the number of levels for linear roughing.



# of Passes allows you to define the number of contour passes to be taken to remove the material.

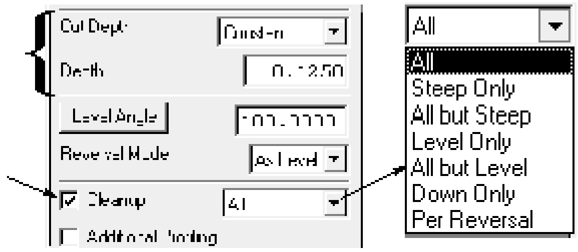


Individual produces a series of differing cut depths by allowing you to define the number of passes at each cut depth.



Cleanup

Cleanup is designed to remove material (steps in most cases) left from roughing passes. The cleanup pass is applied at the end of a cut to remove the material (such as a step) left by the previous roughing cut.



All creates a cleanup on all contours.

Steep Only creates a cleanup on only the steep elements. Steep parameters are defined using the Contour Types located under the Cutting button.

All but Steep creates a cleanup for all geometry that does not qualify as steep.

Level Only creates a cleanup on geometry that is defined as level only. Level parameters are defined using the Contour Types located under the Cutting button.

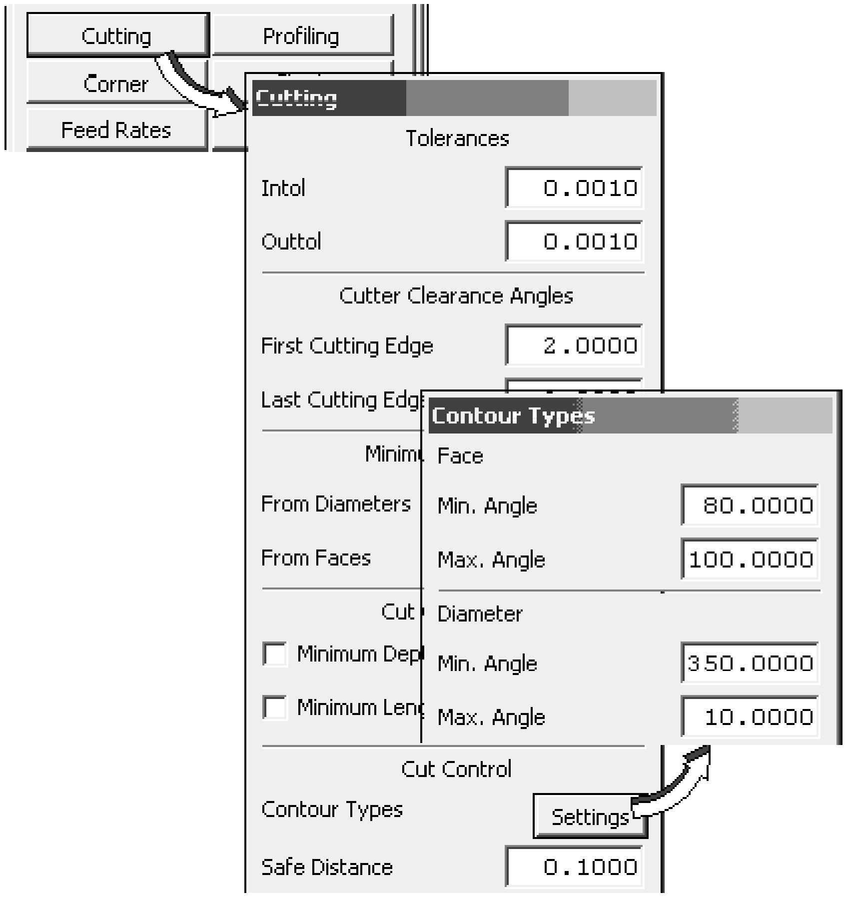
All but Level creates a cleanup on all geometry except the geometry that does not qualify as Level.

Down Only creates a cleanup in a down direction. This option is commonly used in grooving.

Per Reversal creates a cleanup in the reversals only.

Contour Types

The parameters that define Faces and Diameters are located on the Contour Types dialog. This dialog is accessed when you choose **Cutting →Settings.**



Cutting

The options that are located under the Cutting button allow you to add additional tool control.

Tolerance

Tolerances allows you to set values for **Intol** and **Outtol**. The tolerance is applied to the Part boundary and determines the acceptable amount of deviation from the boundary.

Cutter Clearance Angles

This option is used as a clearance for the Lead and Heel Angle. This option is used to prevent the tool from gouging.

Minimum Clearance

Minimum Clearance defines a safe clearance distance away from the blank boundaries. You can define individual distances for both:

- diameters
- faces

Minimum Clearance ensures that positioning moves between cutting regions or during a traversal move, clears the In-Process Workpiece.

Cut Constraints

These options allow you to define a Minimum depth of cut and Minimum length of cut for the operation. These options only apply to the Linear cutting options.

Cut Control

The Cut Control option allows you to control the extent of the tool's entry into undercut areas on your workpiece.

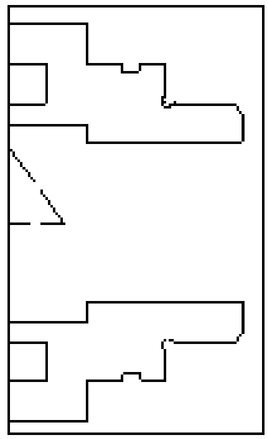
Dwell after Rough Cut

This option outputs a dwell command after each and every rough plunge motion. The dwell is not triggered for subsequent motions along the part profile which may also be generated by the rough plunge cut operation. It is, however, initiated during the successive incremental plunge motions when chip control is activated. Dwell may be input as seconds or revolutions.

Activity: Creating an OD Rough Turn Operation

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

☐ Open the part file **tmp_rgh_od_1.prt**.



☐ Save the part file as *****_rgh_od_1.prt**, where *** represents your initials.

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

Step 2: Define the Parent Groups for this operation.

☐ Expand the **PROGRAM_RGH_OD** Parent Group in the Operation Navigator.

☐ Choose the **Create Operation** icon in the Manufacturing Create toolbar.

Note that Type might be set to mill_planar. The Configuration or Set up will not be tracked after the part is saved and closed for the first time. You may need to set the Type back to Turning.

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

☐ Choose the **ROUGH_TURN_OD** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

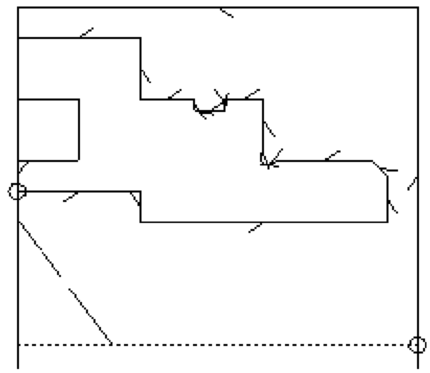
- Program = **PROGRAM_RGH_OD**
- Use Geometry = **RIGHT_END_OD**
- Use Tool = **OD_80_L**
- Use Method = **LATHE_ROUGH**

☐ Choose **OK**.

The ROUGH_TURN_OD dialog is displayed.

Next you will verify the geometry.

☐ Choose the **Groups** tab at the top of the dialog, choose the **Geometry** button, then **Display**.

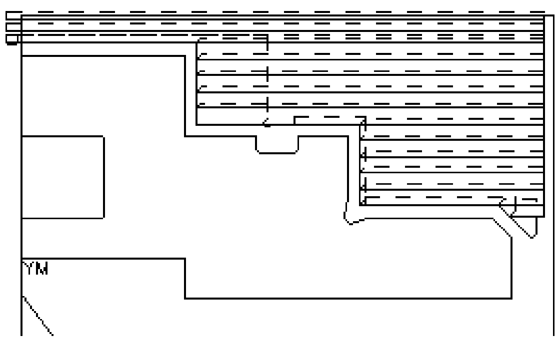


☐ Choose the **Main** tab at the top of the dialog.

Step 3: Generate the tool path using the default settings.

☐ Choose the **Generate** icon located in the lower portion of the dialog.

The Cut Pattern is Linear Zig.



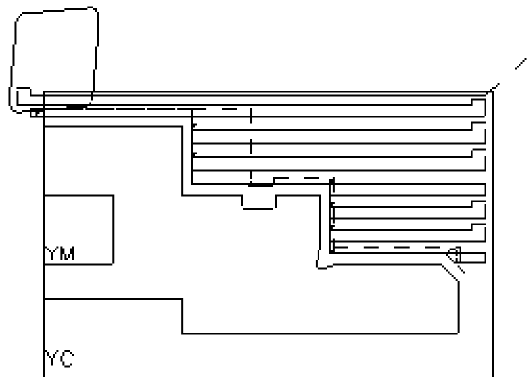
Step 4: Changing Cut Patterns.

You will regenerate the tool path using a different Cut Pattern.

☐ Choose the **Linear Zig-Zag** icon.

☐ Choose the **Generate** icon.

The tool lifts from the material then feeds back in at the same point. Note this pattern is not acceptable for this part/tool combination. You will try a different cut pattern.

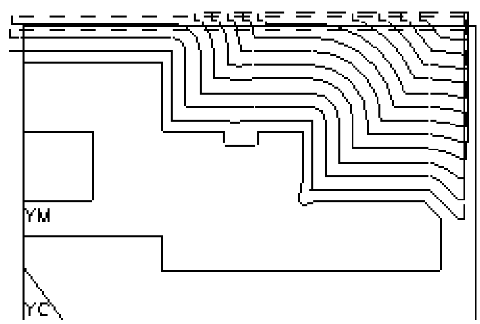


☐ Choose the **Contour Zig** icon.

☐ Key in **0.100** for the Depth.

☐ Choose the **Generate** icon.

The tool path is generated and looks very different. This time the tool cuts in a contour pattern following the shape of the finished part. This pattern could work very well for a non-stepped profile, but for this deeply stepped part, the tool will bury into the faces.



☐ Change the Cut Pattern back to **Linear Zig**.

Now examine the Cleanup option. This option is unique to Roughing Operations.

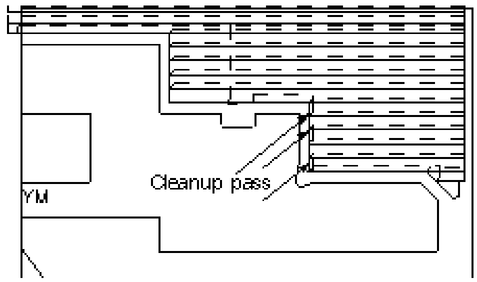
Step 5: Examine the results of the Cleanup option.

Cleanup is designed to remove material (steps in most cases) left from roughing passes. First, you are going to examine the results of the cleanup option.

☐ **Generate** the tool path.

☐ Examine the tool path.

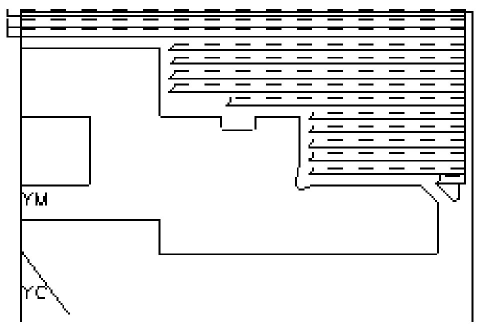
The tool path continues the pass along the contour of the part in order to remove the material left from the previous pass. If you were to turn the Cleanup option **OFF**, no cleanup would be created.



☐ Turn **OFF** the Cleanup option.

☐ **Generate** the tool path.

Cleanup is not generated. The tool path leaves material on the faces of the part.

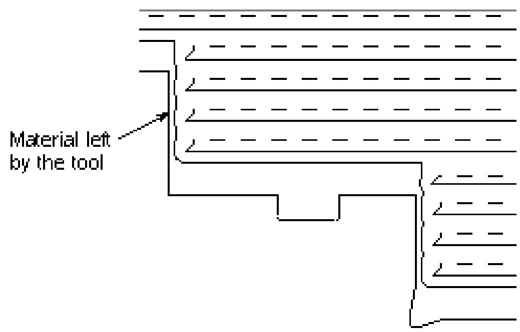


☐ Choose **OK** to accept the operation.

Step 6: Visually Examine the tool path.

You are going to examine the results of the tool path using the Show 2D option.

☐ Highlight the **ROUGH_TURN_OD** operation name (if necessary) and choose **Workpiece →Show 2D**.



The In-Process Workpiece shows the material remaining. The cleanup in the previously generated path removes this material.

Try viewing the Workpiece in the 3D mode.

☐ Highlight the **ROUGH_TURN_OD** operation name (if necessary) and choose **Workpiece →Show 3D**. Change to the trimetric view, for easier viewing.

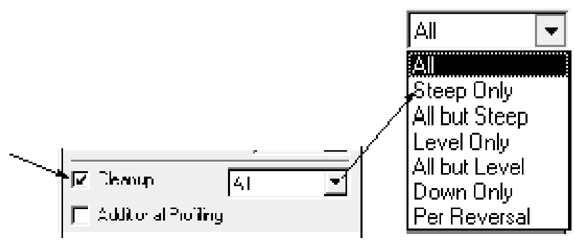
Step 7: Edit the ROUGH_TURN_OD tool path.

☐ Double-click the **ROUGH_TURN_OD** operation name in the Operation Navigator.

The ROUGH_TURN_OD dialog is displayed. You are ready to continue examining the Cleanup options.

☐ Click on the **Cleanup** option to turn it **on**.

☐ Next to the Cleanup option, choose **Steep Only**.



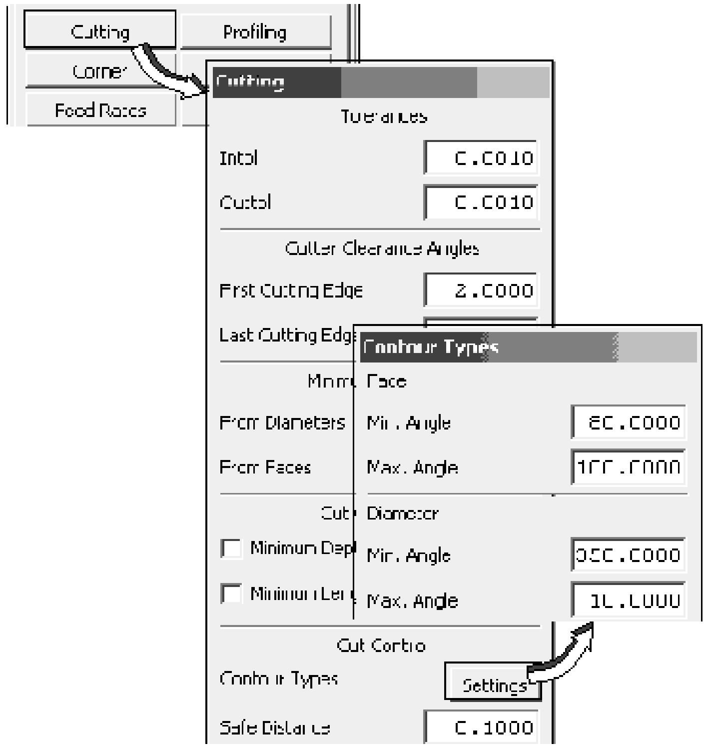
Steepness is a parameter that you can define.

The steepness options are located in the Contour Types area under the Settings button.

☐ Choose **Cutting**.

The Cutting dialog is displayed.

☐ Choose the **Settings** button next to Contour Types.

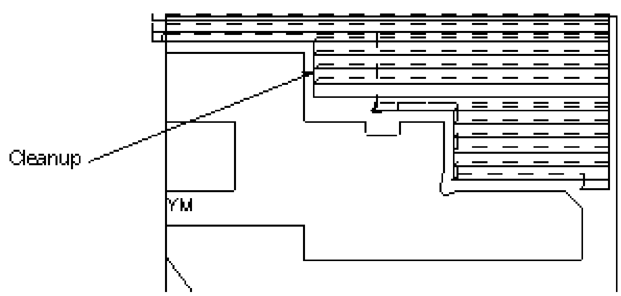


Notice the options available. You will not make any adjustments to the steep parameters.

☐ Choose **Back** until you return to the **ROUGH_TURN_OD** dialog.

☐ **Generate** the tool path.

Only the steep areas (faces) area cleaned up.



☐ Change the Cleanup back to **All**.

☐ **Generate** the tool path once again.

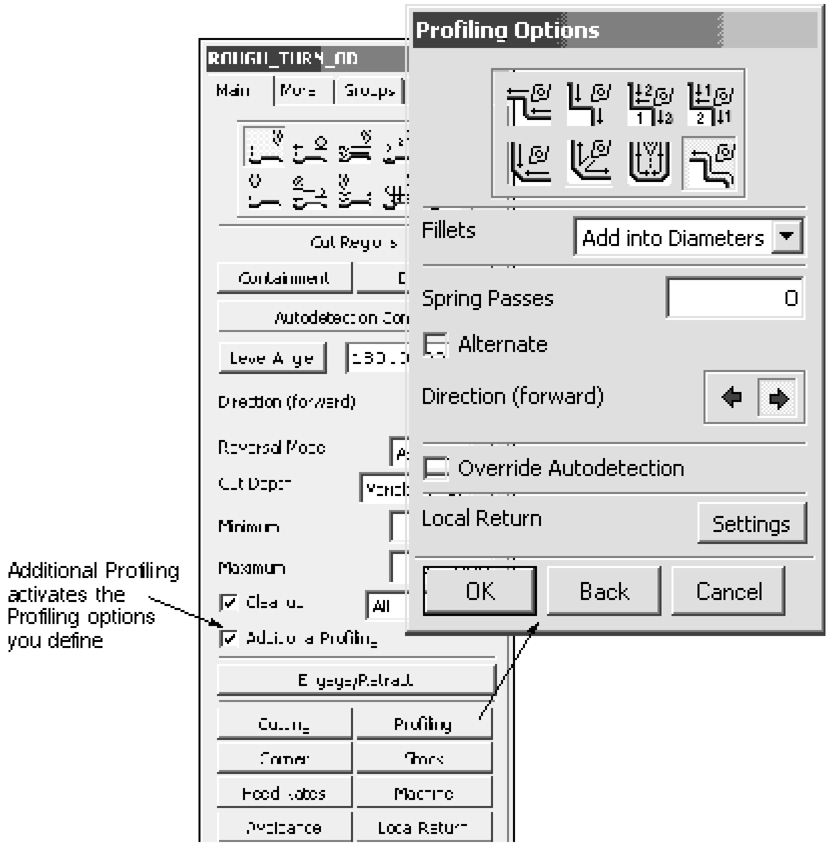
Note that the tool cuts slightly into the thread relief and groove.

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

☐ **Save** the part file.

Profiling

Profiling options provide additional Spring Passes. This option is different from the Cleanup option in that the Spring Pass follows the part boundary, where as the Cleanup option only removes excess step material.



Note that you must turn the Additional Profiling option **ON** in order to use the Profile option that you defined.

The Profiling options allow you to make finish passes within a Roughing operation.

Override Autodetection

After the roughing passes, the system re-evaluates the material and the In-Process Workpiece (using a second cut region detection process) and then performs the profiling cuts.

Override Autodetection eliminates the second cut region detection and allows the tool to cut exactly the same region that the roughing pass cuts.

Override Autodetection requires Rough and Profile Stock to be identical.

If rough and profile stock differ, the activated toggle to override the second cut region detection process would cause the processor to execute the profile cut with the wrong offset setting.

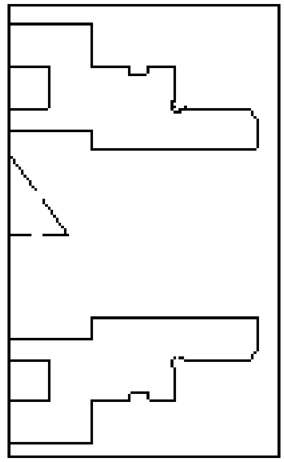
Local Returns

Local Return moves are available in Roughing, Finishing and Threading operations. Local Return in Roughing can be defined separately for Rough and Profile passes.

Activity: Creating Additional Profile Passes

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

☐ Continue to use the part file *****_rgh_od_1.prt**.



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

Step 2: Edit an existing operation.

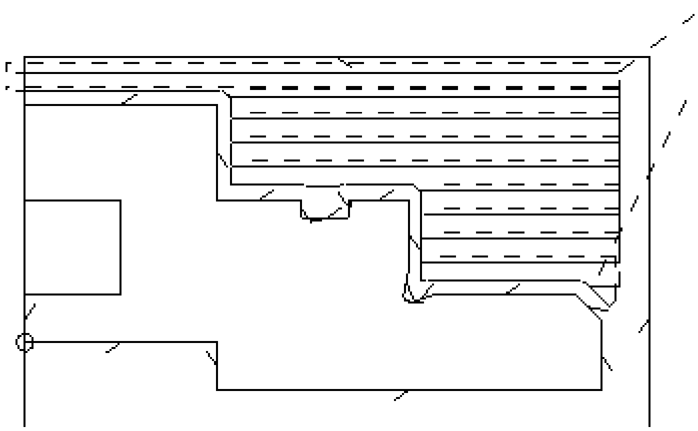
☐ Expand the Program name **PROGRAM_PROFILE**.

☐ Double-click the **ROUGH_TURN_OD_PROFILE** operation name in the Operation Navigator.

The ROUGH_TURN_OD dialog is displayed.

☐ **Display** the geometry.

☐ **Replay** the tool path.



The tool path is generated. Note the roughing cuts. You are going to add a Profiling pass to the operation and examine the results.

Step 3: Create an Additional Profiling pass (spring pass).

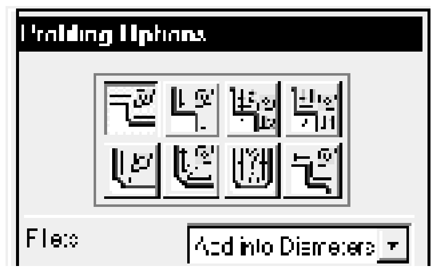
You can create additional spring passes for your operation. There are several steps necessary to add spring passes.

The processor will calculate a Cut Region for the Rough Stock and execute the rough cuts, then calculate a Cut Region using Profile Stock and then execute that cut.

☐ On the **ROUGH_TURN_OD** dialog, turn **ON** the **Additional Profiling** option.

☐ Choose **Profiling**.

The Profiling Options dialog is displayed.



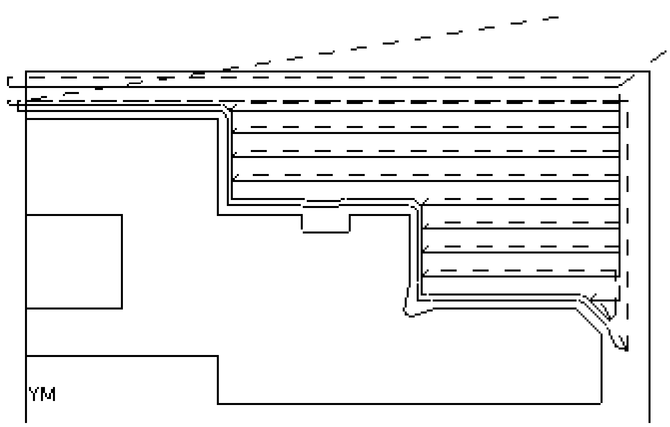
You are going to use the default Cut Pattern (Profile All) the first time and add one spring pass to this operation.

☐ Enter **1** into the Spring Passes value field.

☐ Choose **OK** from the Profiling Options dialog.

The ROUGH_TURN_OD dialog is displayed.

☐ **Generate** the tool path.



The Spring Pass is created. Note that the rough pass left the stock defined in the **Stock→Rough Stock** settings (.02 face and .03 radial). Also note that the final pass cuts to finish depth. This is due to the **Stock→Profile** settings being set to 0 Face and 0 Radial. This is an example where a Rough turn operation can have a finish pass.

Step 4: Add a Local Return Point and an operator message.

You are going to add a local return point to the Profile pass only. This is accomplished by defining the Local Return option inside of the Profiling Options dialog. You can also add a Local Return to the rough pass. To apply the Local Return to the rough pass you select the Local Return button on the ROUGH_TURN_OD dialog.

☐ Choose **Profiling**.

The Profiling Options dialog is displayed.

☐ Choose the **Settings** button next to the Local Return label, at the bottom of the dialog.

☐ Turn **ON** the **Local Return**.

☐ Turn **ON** the **Local Return Point** option, then choose the **Select** button.

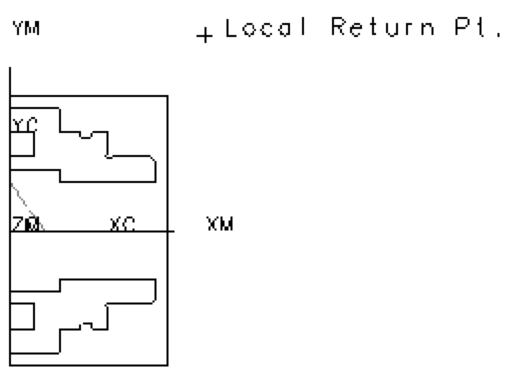
☐ Enter the following values:

- XC = **4**
- YC = **4**

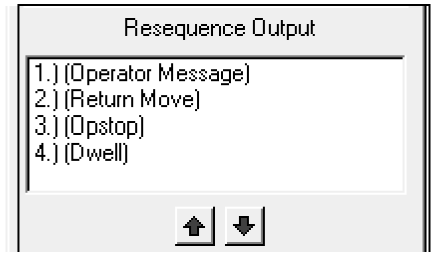
☐ Choose **OK**.

☐ Choose **Display** from the Local Return Point area.

The point is displayed.



Now you are going to add an Opstop and an Operator message. Note in the Resequence output list that these commands are in brackets. As you pick the options to activate them, the brackets will be removed.



☐ Turn **ON** the **Opstop** and **Operator Message** options.

☐ In the Operator Message text field type **Check Part Diameters**.

Step 5: Resequence the commands.

You can resequence the commands that you have activated using the Resequence Output window and arrows. Notice that the commands that you selected are no longer in brackets.

☐ Highlight **Operator Message** in the Resequence Output area.

☐ Use the down arrow to move the **Operator Message** below **Opstop**.

You are ready to generate the spring pass changes, the local return and additional commands.

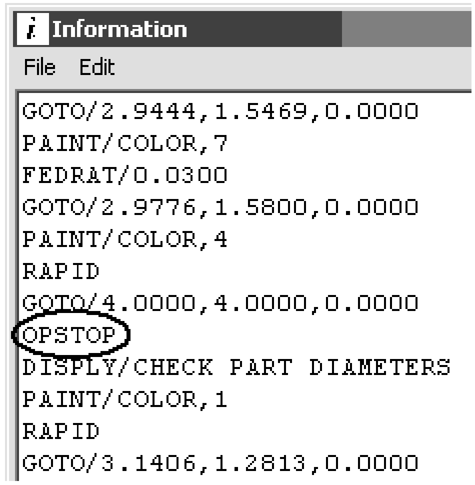
☐ Choose **OK** until you return to the **ROUGH_TURN_OD** dialog and then **Generate** the tool path.

The tool path is generated. The tool cuts the part including a cleanup cut. Before cutting the spring pass the tool moves to the Local Return Point, then cuts the spring pass.

List the tool path data.

☐ Choose the **List** icon.

☐ Scroll until you see the Opstop and Message commands in the text.



☐ Dismiss the Listing window.

This Local Return Point that you have just created is only applied to the Profile pass. Remember, on the main ROUGH_TURN _OD dialog you can define a Local Return. This Local Return is applied at the end of each Rough pass.

☐ Test this yourself by putting in an a different message and different return position using the Local Return located on the **ROUGH_TURN _OD**.

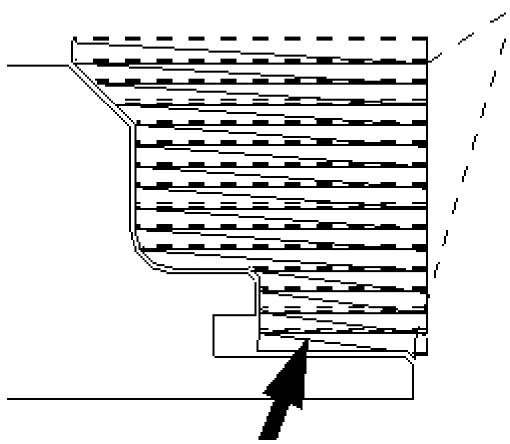
☐ Turn **Off** the Additional Profiling options.

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

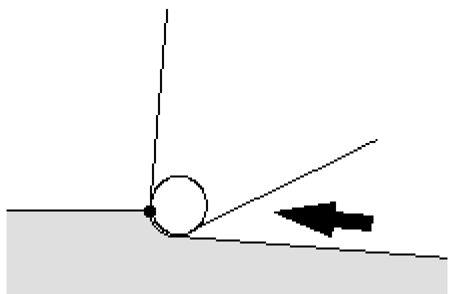
☐ **Save** the part file.

Ramping

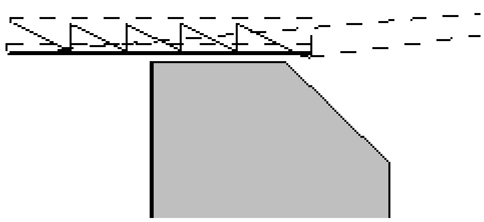
Ramping cut strategies can prolong tool life (particularly ceramic inserts) and reduce machining time when rough cutting hard materials. Notching effects on the insert can be effectively reduced or even completely avoided through ramping which leads to savings in tooling costs and machining time.



By ramping every pass or every other pass in the roughing cut pattern, the hot point on the cutting edge of the insert moves continually, distributing stress and heat and extending the life of the insert.



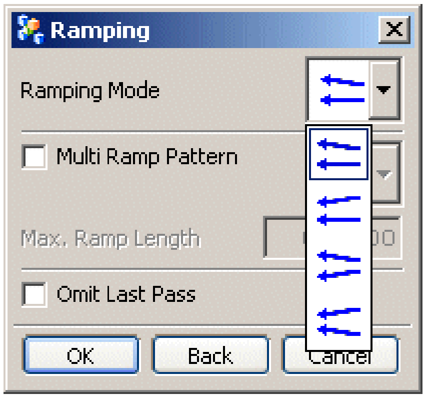
In cases where a single ramp over the length of a cut is ineffective due to a shallow cut depth, multiple ramping achieves the desired effect of a constantly changing depth by generating a "sawtooth" or "wavy" cut pattern.



Ramping Zig  ramps into or out of the part in one cut direction.

Ramping Zig-Zag  ramps into or out of the part in both cut directions.

The following ramping modes are used in combination with Ramping Zig and Ramping Zig-Zag.



Ramp Out on Every Other Pass starts with a deep cut and diminishes the depth of cut. This is followed by a level cut.

Ramp In on Every Other Pass starts at the surface of the work piece and ramps downward into the material. this is followed by a level cut.

Ramp Out First alternately ramps every cutting pass with the first pass ramping out.

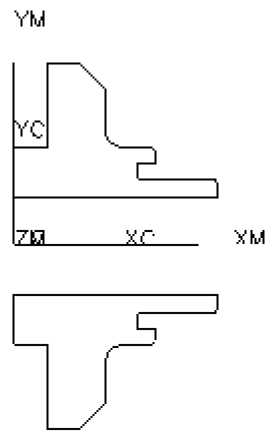
Ramp In First alternately ramps every cutting pass with the first pass ramping in.

Activity: Creating Ramping Cut Patterns

In this activity, you are going to create single and multiple ramping cut patterns.

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

☐ Open the part file **tmp_ramp_1.prt**.



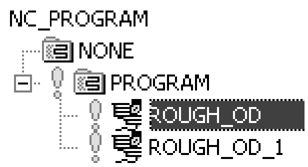
☐ Save the part file as *****_ramp_1.prt**, where *** represents you initials.

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

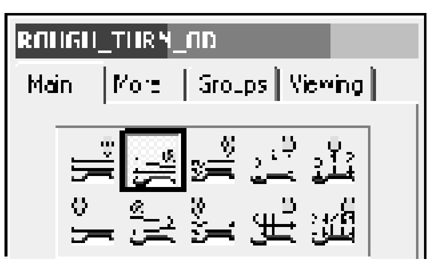
Step 2: Specify a Ramp Zig cut pattern.

☐ Expand the **PROGRAM** Parent Group in the Operation Navigator.

☐ Double-click on the **ROUGH_OD** icon in the Operation Navigator to edit the operation.

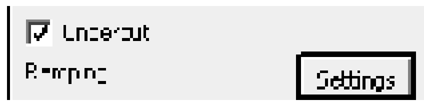


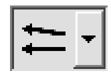
☐ Choose the **Ramping Zig** icon.



☐ Choose **Cutting**.

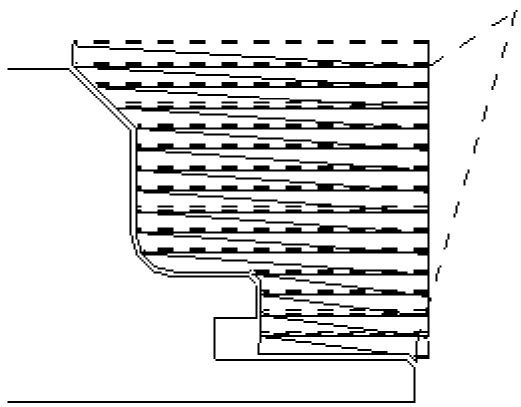
☐ Choose **Settings** next to Ramping.



☐ Choose **OK** to accept Ramp Out on Every Other Pass  as the Ramping Mode.

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

☐ **Generate** the tool path.



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

Step 3: Specify a Multi Ramp pattern

Multi Ramp patterns are useful in cut regions that are long and shallow where ramping over the length of the cut is ineffective because the inclination is nearly horizontal.

☐ Double-click on the **ROUGH_OD_1** icon in the Operation Navigator to edit the operation.

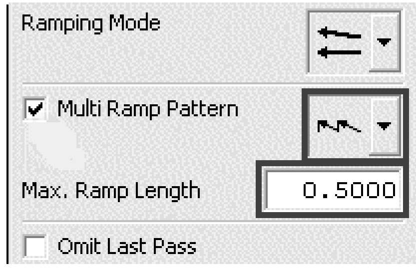
☐ Choose the **Ramping Zig** icon.

☐ Choose **Cutting**.

☐ Choose **Settings** next to Ramping.

☐ Turn the Multi Ramp Pattern option **ON** and key in **0.500** in the Max. Ramp Length field.

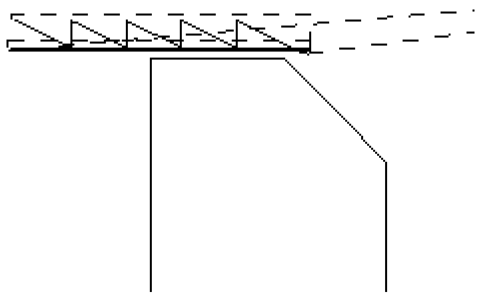
Notice that the default Multi Ramp Pattern is Ramp Out Only.



☐ Choose **OK** to accept the ramping parameters.

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

☐ **Generate** the tool path.



Step 4: Omit the last pass

The Omit Last Pass option suppresses the final linear pass along the part surface.

☐ Choose **Cutting**.

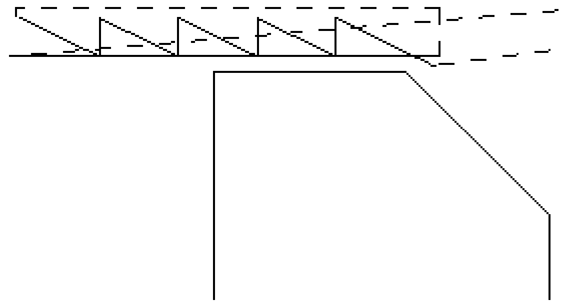
☐ Choose **Settings** next to Ramping.

☐ Turn the Omit Last Pass option **ON**.

☐ Choose **OK** to accept the ramping parameters.

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

☐ **Generate** the tool path.



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

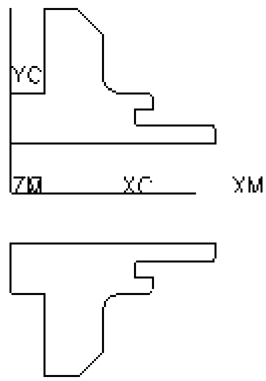
☐ **Save** the part.

Activity: Custom Boundary Data

You can edit part boundary and member parameters for individual operations in cases where the parameters must differ from those defined at the geometry group level. These parameters include custom stock, cut feed rate, and machine control events.

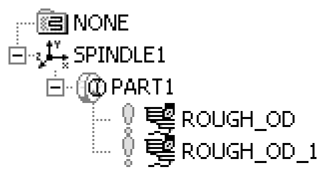
Step 1: Open the part file and enter the Manufacturing application.

☐ Continue to use the part file ***_ramp_1.prt**.



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

☐ Display the Geometry View in the Operation Navigator and expand the objects.



Step 2: Verify the Part Boundary.

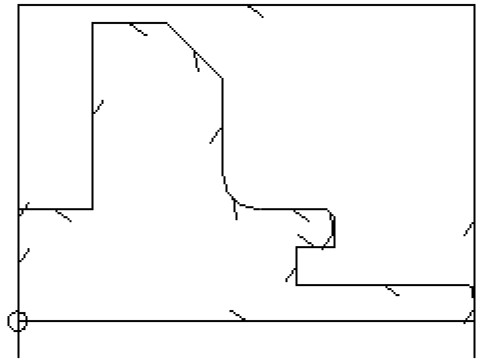
You will verify that the part boundary has been defined in the PART1 geometry parent group.

☐ Display the Geometry View of the Operation Navigator.

☐ Highlight the PART1 geometry group in the Operation Navigator.

☐ **MB3** select **Object Display**.

The part and Blank boundaries display.



☐ Choose **Cancel** to dismiss the TURN_BND dialog.

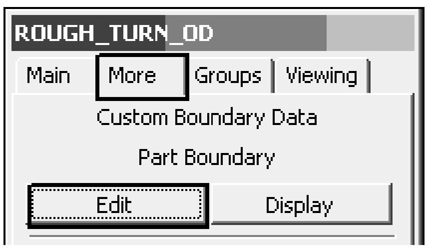
Step 3: Edit Boundary Parameters

You will see how the part boundary parameters can be edited from within the operation.

☐ Double-click on the **ROUGH_OD** icon to edit the operation.

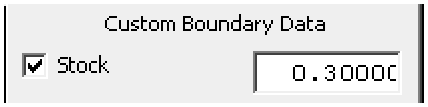
☐ Choose the **More** tab.

☐ Choose **Edit** under part boundary.



The Part Boundary dialog displays allowing you to apply Stock and Cut Feedrate to the entire boundary, but only for the current operation. The Stock you apply here overrides the Custom Boundary Data Offset specified in the geometry group.

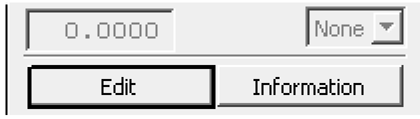
☐ Turn the Stock option **ON** and key in **0.300** in the Stock field.



Step 4: Edit Member Parameters

You will see how the member parameters can be edited from within the operation.

☐ Choose Edit in the Part Boundary dialog.



The Edit Member dialog displays allowing you to apply Stock, Cut Feedrate, and machine control events to individual boundary members. The Stock you specify here overrides the Custom Member Data Offset specified in the geometry group.

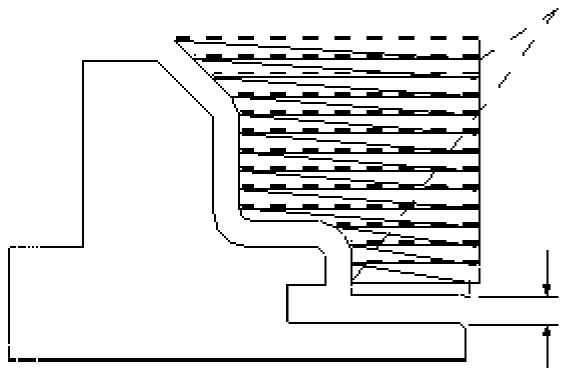
☐ Choose **OK** twice to return to the ROUGH_TURN_OD dialog.

Step 5: Generate the tool path

You will generate the tool path and see how the 0.300 stock has been applied to the part boundary.

☐ Choose the Main tab and Generate the tool path.

The 0.300 stock is applied to the entire part boundary.



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

Step 6: Create a new operation.

You will create a new roughing operation to see how the part boundary stock you specified earlier does not apply to this operation.

☐ Choose the **Create Operation** icon in the Manufacturing Create toolbar.

☐ Choose the **ROUGH_TURN_OD** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**

- Use Geometry = **PART1**
- Use Tool = **OD_80_L**
- Use Method = **LATHE_ROUGH**

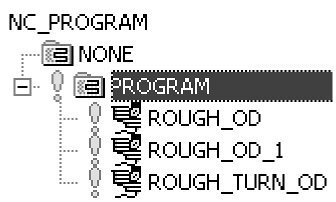
☐ Choose **OK** to begin creating the operation.

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

Step 7: **Generate** the tool paths.

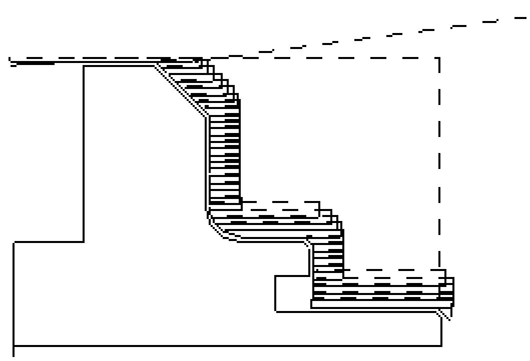
You will generate the tool paths for the entire program and see how the final tool path removes the stock you specified in the first operation.

☐ In the Program Order View of the Operation Navigator, choose the PROGRAM icon and **MB3→Generate**.



☐ Choose **OK** to generate each of the three tool paths.

Notice that the last tool path removes the part boundary stock you specified in the ROUGH_OD operation.



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

☐ **Save** the part.

Summary

In this lesson, you learned how to create a roughing operation to cut the OD of the part.

In this lesson you:

- Created roughing operations.
- Used several different roughing Cut Patterns.
- Removed and added a cleanup pass to an operation.
- Created an additional pass (spring pass) and adjust the profiling options for that pass.
- Added a Local Return and post commands that were applied to a spring pass.
- Applied Single and Multiple Ramping Cut Strategies.
- Edited Part Boundary parameters from within an operation.

Lesson

9 Roughing Operations - ID Work

Purpose

This lesson is designed to teach you how to create roughing operations to remove ID material. These operations are similar to OD roughing operations. This lesson will also allow you to use some of the options that you have used previously in a little different context.

Objective

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

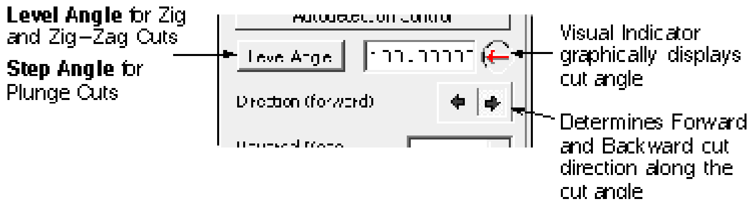
- Create a roughing operation to remove material from the part ID.
- Use Containment to limit the cut area.
- Manually specify a Cut Region.
- Use the different Reversal Mode options to remove material from part reversals.

Roughing Operations for ID Work

ROUGH_BORE_ID operation subtypes have the same options available as ROUGH_TURN_OD operation subtypes.

Level Angle/Step Angle

In Rough Turn operations, the Level Angle (Zig and Zig-Zag cuts) and Step Angle (Plunge cuts) options define the angle at which the roughcuts are made. They are accompanied by a visual indicator that graphically displays the cut angle. The cut angle is measured counterclockwise from the spindle centerline.



The Forward/Backward direction arrows determine the cut direction along the cut angle. The Forward Direction arrow cuts in the direction indicated by the Level Angle visual indicator. The Backward Direction arrow cuts in the opposite direction indicated by the Level Angle visual indicator.

There are two methods of determining the cut angle:

- Specifying an existing line using the selection ball
- Entering a value

When you select a line to determine the cut angle, the angle is based on the location of the cursor relative to the line’s midpoint.

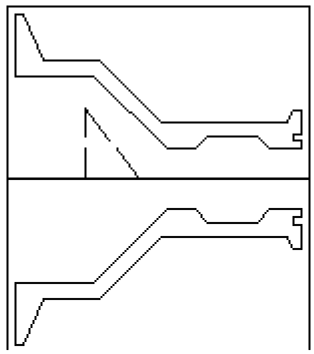
In the following activities you will create a ROUGH_BORE_ID operation to remove most of the ID material. You will use several Reversal Mode options and examine the tool path results.

Activity: Roughing the ID

In this activity, you are going to create two operations to remove material on the ID of the part. You will limit the tool path using Containment, observe how the system automatically determines which of several cut regions to use, and learn how to manually specify the desired cut region.

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

☐ Open the part file **tmp_reducer_1.prt**.



☐ Save the part file as *****_reducer_1.prt**, where *** represents your initials.

The cross section geometry has been pre-defined for this operation.

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

Step 2: Define the Parent Groups for the first ID operation.

☐ Choose the **Create Operation** icon.

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

☐ Choose the **ROUGH_BORE_ID** icon from the Create operation dialog.

The Parent Groups for this program have been pre-defined for you.

☐ Set the following Parent Groups:

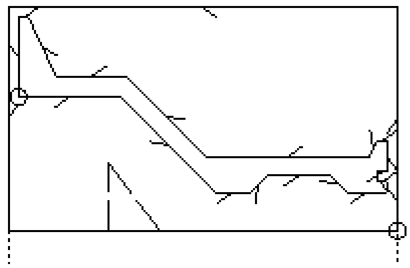
- Program = **PROGRAM**
- Use Geometry = **WORKPIECE**
- Use Tool = **ID_80_L**
- Use Methods = **LATHE_ROUGH**

☐ Choose **OK**.

The **ROUGH_BORE_ID** dialog is displayed.

☐ Choose the **Groups** tab.

☐ Choose the **Geometry** button and then choose **Display**.



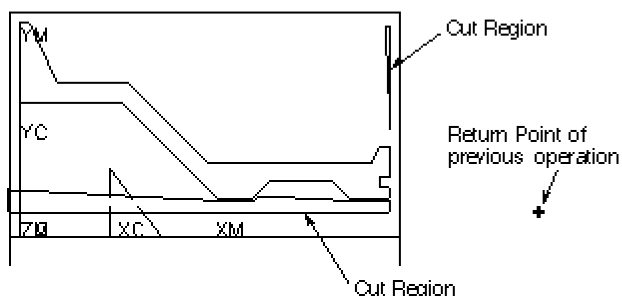
Step 3: Display the Cut Region.

It is always a good practice to display the cut region before generating a path. You will then see what the tool will cut.

☐ Choose the **Main** tab.

☐ In the Cut Regions area, choose **Display**.

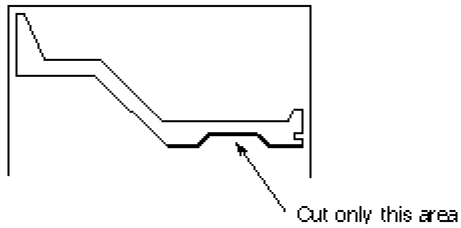
Notice that two cut regions are displayed.



An operation can use only one cut region. If the operation does not contain a Start Point, the cut region closest to the Return Point of the previous operation is used. In this case, the cut region you wish to use is the one closest to the Return Point of the previous operation.

Step 4: Limit the area the tool will cut.

You are going to add containment to this tool path. The back portion of the part (taper and bore) will be cut in a different setup. You only need to machine the three ID diameters and the two tapers.



☐ Choose **Containment** from the Cut Regions section of the dialog.

The Geometry Containment dialog is displayed. You are going to specify an axial containment for this operation.

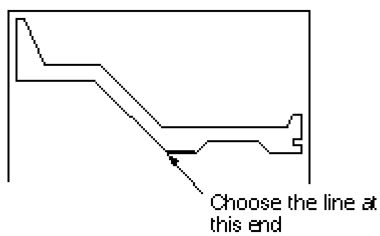
☐ Check the **Trim** option next to Axial 1.

☐ Choose the **Axial 1** button.

The Point Constructor dialog is displayed. You define the containment as a point on an axis. The default is an inferred point; you will use that setting. You will also add an offset to this axial position.

☐ Next to the Offset label, change the option to **Rectangular**.

☐ Select the containment position as shown below.



You are going to add an .100 offset to the axial containment.

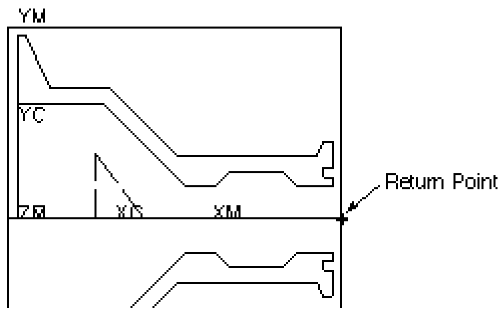
☐ Enter the negative number **-.100** into the Delta-XC value field and choose **OK** until you return to the ROUGH_BORE_ID dialog.

Step 5: Avoid cutting the reversal.

☐ Change the Reversal Mode option to **Omit**.

Step 6: Define the Avoidance geometry.

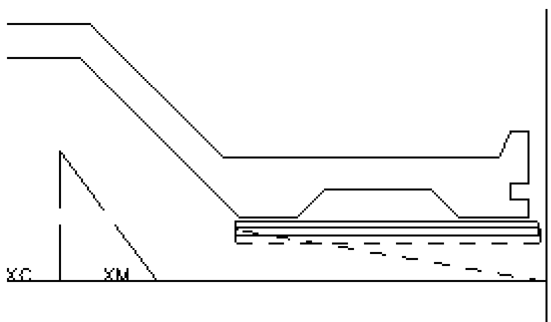
☐ Define a **Return Point** at the end of the horizontal line as shown.



Step 7: Generate the tool path.

☐ Choose the **Generate** icon.

The tool path is generated. The tool cuts the ID and the axial trim you defined. The tool also omits the reversal.



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

Step 8: Define the Parent Groups for the second ID operation.

☐ Choose the **Create Operation** icon.

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

☐ Choose the **ROUGH_BORE_ID** icon from the Create Operation dialog.

The Parent Groups for this program have been pre-defined for you.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **WORKPIECE**
- Use Tool = **ID_55_L**

- Use Method = **LATHE_ROUGH**

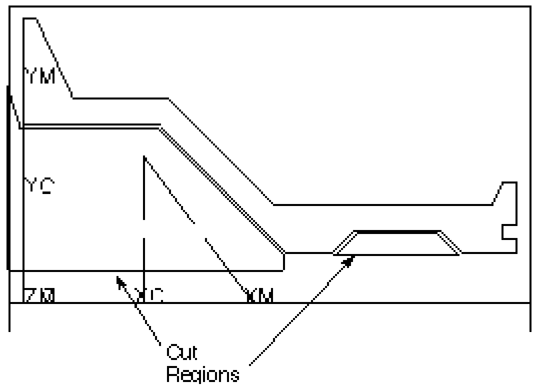
☐ Choose **OK**.

The **ROUGH_BORE_ID** dialog is displayed.

Step 9: Display the Cut Region.

It is always a good practice to display the cut regions before generating a path. You will then see what the tool will cut.

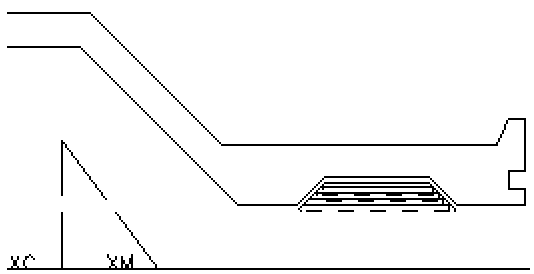
☐ In the Cut Regions area, choose **Display**.



An operation can use only one cut region. If the operation does not contain a Start Point, the system uses the cut region closest to the Return Point of the previous operation. In this case, the cut region you wish to use is not the one closest to the Return Point of the previous operation.

Step 10: Generate the tool path.

☐ Choose the **Generate** icon.



This is not the cut region you wish to machine. You must manually specify the other cut region.

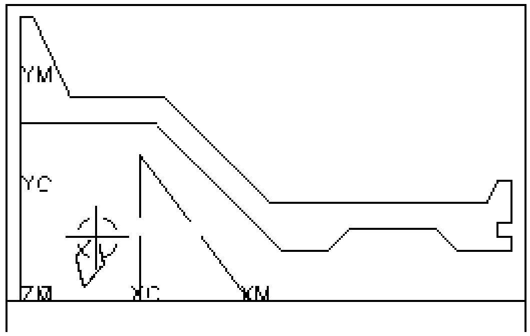
Step 11: Specify the other cut region.

☐ Choose **Containment** under Cut Regions.

The Geometry Containment dialog displays.

☐ Choose **Select Manually** and then **Select** under Cut Regions.

☐ Select near the desired cut region as illustrated below.

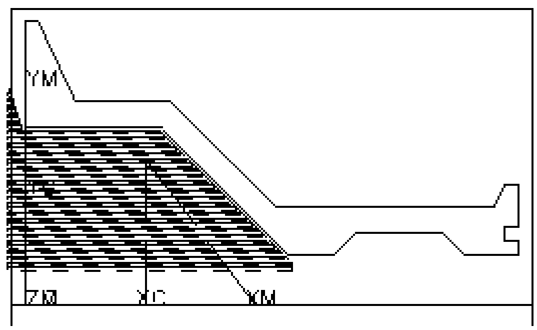


☐ Choose **OK** to accept the cut region.

Step 12: Generate the tool path.

☐ Choose the **Generate** icon.

The tool path is generated. The tool now cuts the correct region.

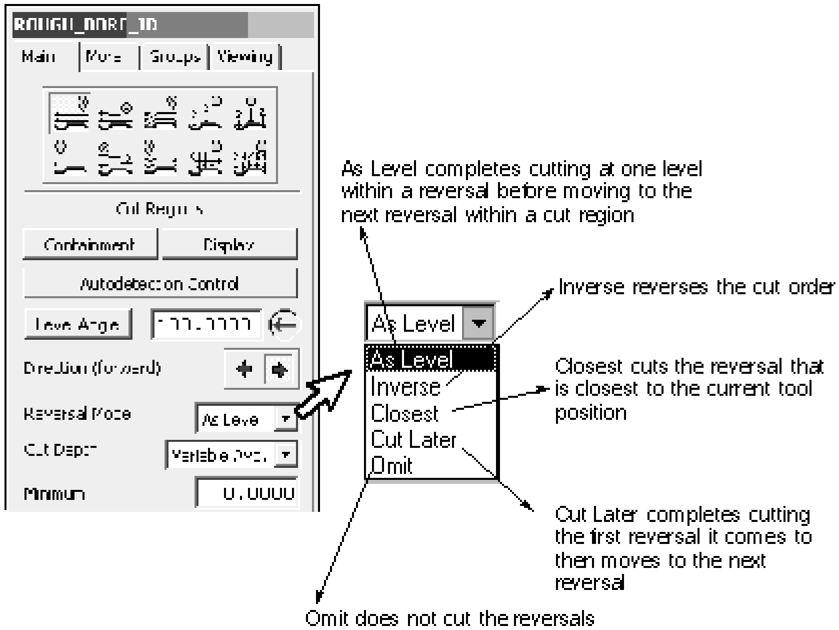


☐ Choose **OK** to complete the operation.

Step 13: **Save** the part file.

Reversal Mode

The Reversal Mode determines which reversal of a cut region will be cut first.

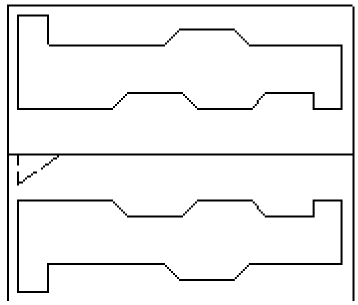


Activity: Roughing the ID - Reversals

In this activity, you are going to edit an existing tool path. You are going to replay the tool path and note that the tool stops cutting before it reaches the end of the part. You are going to adjust this to cut past the end of the part. You are then going to examine the different reversal mode options.

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

- ☐ Open the part file **tmp_bearing_sleeve_1.prt**.



- ☐ Save the part file as *****_bearing_sleeve_1.prt**, where *** represents your initials.

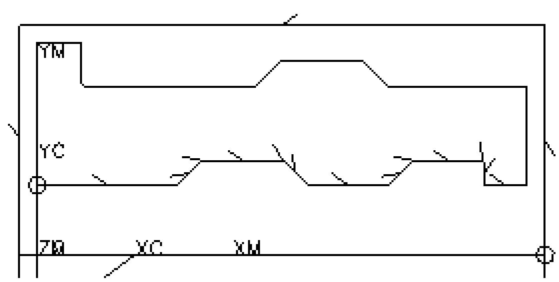
  The cross section geometry has been pre-defined for this operation.

Step 2: Edit an existing operation.

- ☐ Expand the Program Parent Group.
- ☐ Double-click the **ROUGH_ID_REVERSAL** operation name in the Operation Navigator.
- ☐ Choose the **Groups** tab and examine the settings in this operation.

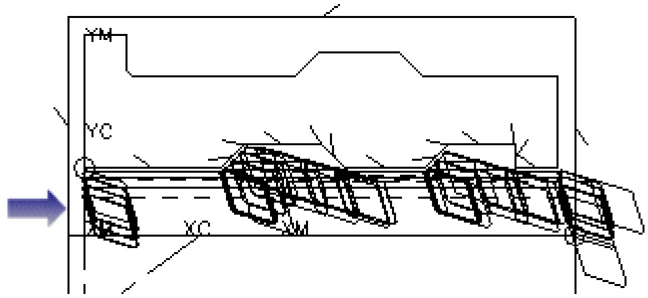
  The tool is a 55 degree diamond and the Return Point has been set.

- ☐ **Display** the geometry.



Replay the tool path and note that the tool stops cutting before it reaches the end of the bore.

☐ Choose the **Main** tab and **Replay** the tool path.

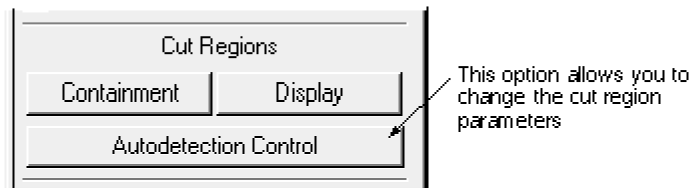


Step 3: Adjust the Autodetection settings.

☐ Choose **Display** in the Cut Regions area.

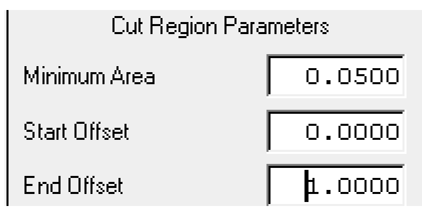
Notice the material to be cut.

☐ Choose **Autodetection Control** in the Cut Regions area.



The Autodetection dialog is displayed. You are going to enter an End Offset to force the cutter to cut past the end of the part.

☐ Enter **1.0** into the End Offset field and a then choose **OK**.



☐ Choose **Display** in the Cut Regions area.

Notice the material to be cut.

Step 4: Generate the operation and observe the change in the cutting procedure.

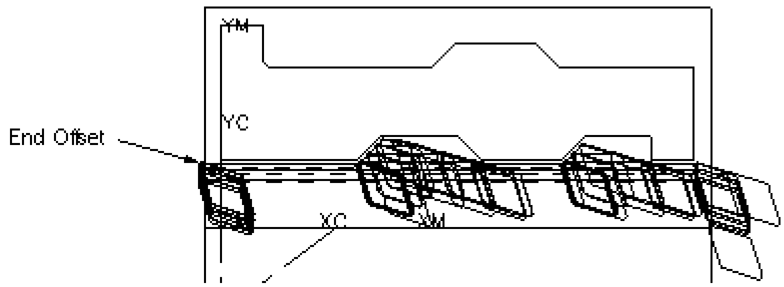
☐ Choose the **Generate** icon.

The tool makes several cuts to remove the material from the ID and cuts past the end of the bore. The offset changes you made are reflected in the cut region display. Look at the left end of the cut region. The tool moves further to the left with positive values until the cut region ends with the blank shape. Values beyond that limit have no further influence on the cut region shape.

The end offset only works with open boundaries.

☐ Choose **Replay** from the dialog.

The tool moves into the first reversal area and cuts to the maximum depth for that reversal. The tool then moves to the second reversal and removes the maximum material.



Step 5: Change the Reversal Mode option.

☐ Next to the Reversal Mode label, change to **Inverse**.

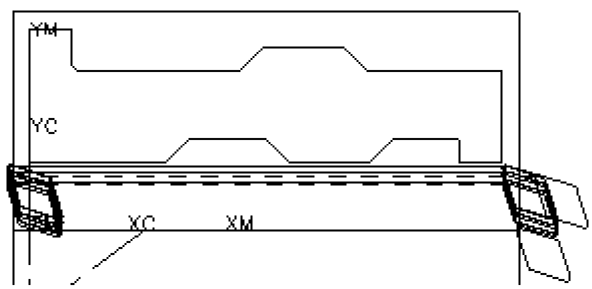
☐ **Generate** the tool path.

The tool cuts the reversals in the opposite order.

☐ Change the reversal Mode to **Omit**.

The most commonly used Reversal option is Omit.

☐ **Generate** the tool path.



The tool path is generated. The tool does not cut into the reversals.

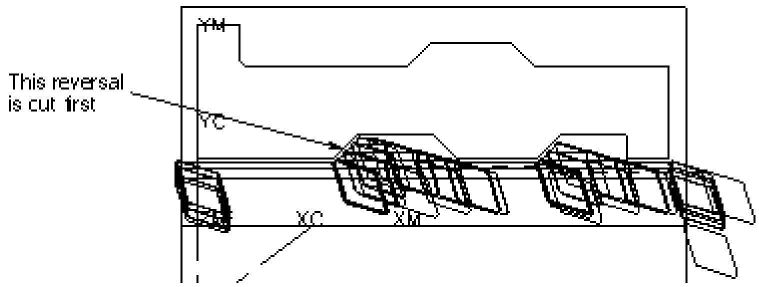
☐ Change the reversal Mode to **Closest**.

To show the use of this option, you will change the cut type to Linear Zig-Zag. This option is designed to cut the reversal that is closest to the current tool position. This tool may not be the best selection. This is for illustration purposes only.

☐ Change the cut type to **Linear Zig-Zag**. 

☐ **Generate** the tool path.

The tool cuts the ID in a Zig-Zag cut type. The first reversal is the reversal closest to the current tool position. In this case it is the left hand reversal.



Now change the cut depth and examine the tool path results.

☐ Change the Depth to **.250**.

☐ **Generate** the tool path.

This time the right hand reversal is cut first since the closest tool position was nearest to the right hand reversal.

Step 6: **Save** and **close** the part file.

Summary

In this lesson, you saw that OD and ID roughing operations are similar. The tool (type and orientation) and the part geometry determine if it is OD or ID cutting.

In this lesson you:

- Created a roughing operation to remove material from the ID of the part.
- Used the Containment option to limit the cut area of the operation.
- Identified reversals and used the reversal mode options to cut reversals.

Lesson

10 Finish Operations OD and ID Work

Purpose

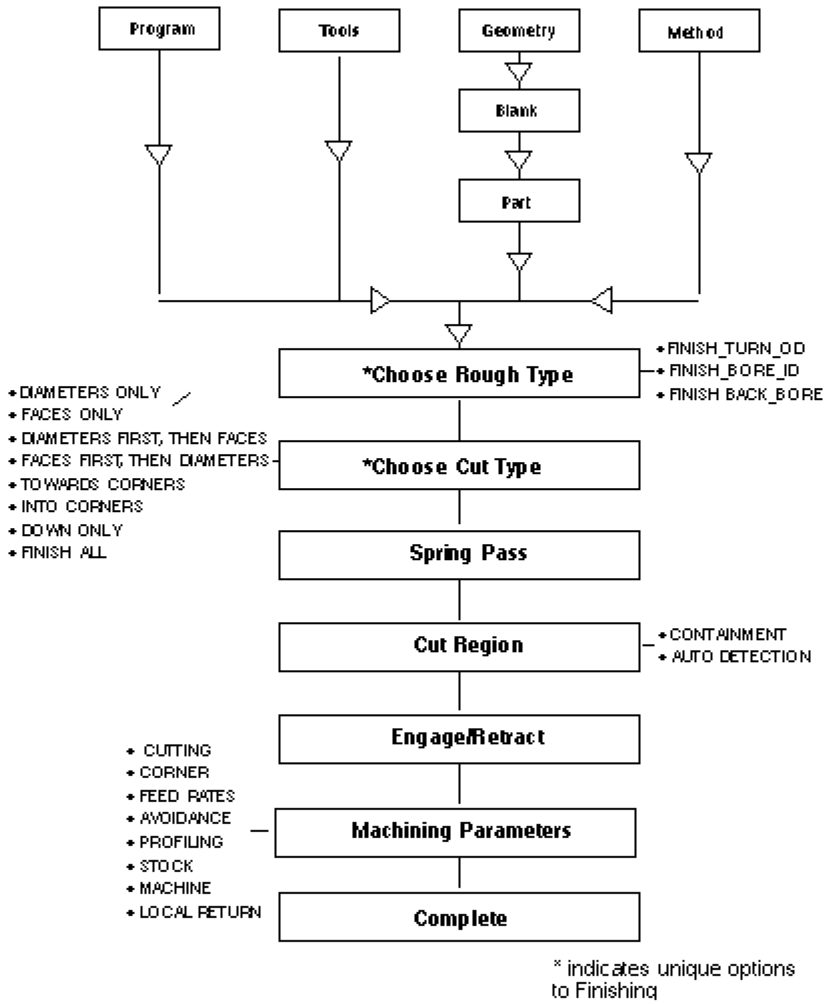
The Finish operation type is used to remove the remaining material after the rough operations. It also uses the In-Process Workpiece (automatic detection) to track the remaining material on the part.

Objective

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

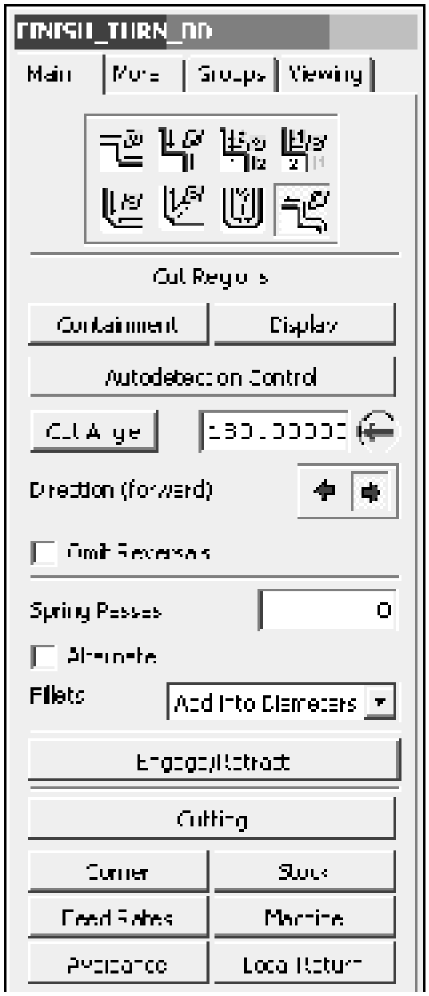
- Create a Finish operation to cut the part OD.
- Create a Finish operation to cut the part ID.
- Use the Corner Control options to finish the part convex corners.

Creating Finishing Operations



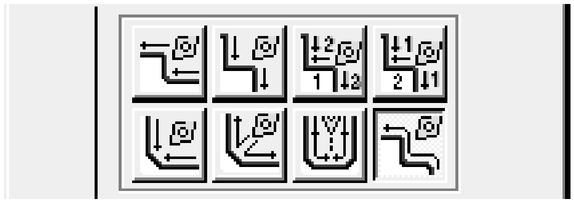
The FINISH_TURN options

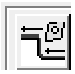
These options allow you to create Finish passes, including spring passes.

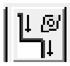


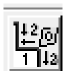
Finish Cut Patterns

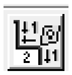
The cut patterns found on the Finish dialog are the same cut patterns available for Profile passes on the Rough dialog.

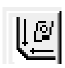


 **Finish Diameters Only** creates finish cuts on diameters only.

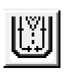
 **Finish Faces Only** creates finish cuts on faces only.

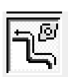
 **Finish Diameter First, then Faces** creates finish cuts on all of the diameters, then creates finish cuts on the faces.

 **Finish Faces First, then Diameters** creates finish cuts on all of the faces, then creates finish cuts on the diameters.

 **Towards Corners** creates finish cuts towards the corners. This is accomplished with two cut moves. The retract move applied in this cut pattern is an automatic retract.

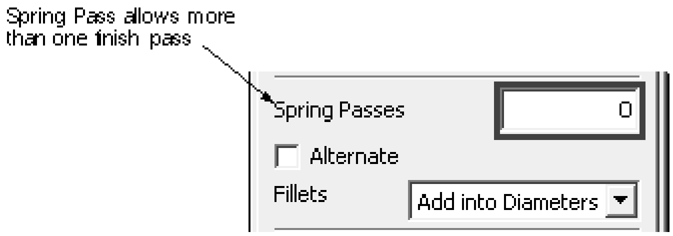
 **Into Corners** creates finish cuts into the corners. This is accomplished with two cut moves. The engage and retract moves share the same angle value which is computed automatically.

 **Down Only** create finish cuts that always cut towards the bottom of the groove.

 **Finish All** creates finish cuts that contour the boundary in a contiguous pass.

Spring Pass

The Spring Pass option allows more than one finish pass.



Alternate

The Alternate option alternates Finish spring passes in a Zig Zag cut pattern.

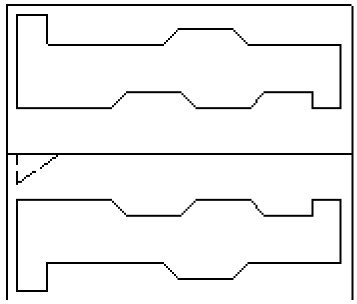
Direction

The Direction arrows invert the cutting direction for Finishing and Spring passes with respect to the boundary direction.

Activity: Creating a Finish OD Operation

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

☐ Open the part file **tmp_bearing_sleeve_2.prt**.



The cross section geometry has been pre-defined for this operation.

☐ Save the part file as ***_**bearing_sleeve_2.prt**, where *** represents your initials.

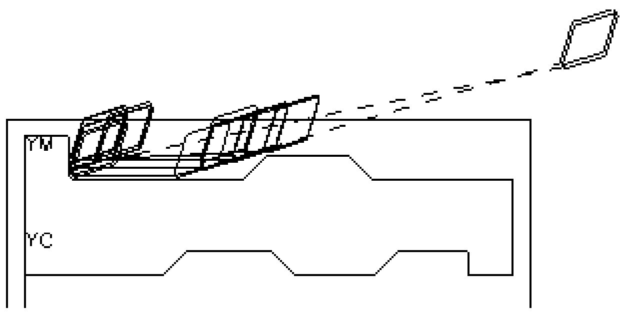
☐ If necessary, enter the **Manufacturing** application.

Step 2: Examine the Rough Tool Path.

☐ Double-click on the **ROUGH_TURN_OD_1** tool path name in the Operation Navigator.

☐ Choose **Display** in the Cut Regions area to display the material.

☐ **Replay** the tool path.



The tool removes the material in the reversal. You are going to create an OD Finish operation.

☐ **Cancel** the operation.

Step 3: Create a Finish operation to cut the OD.

Define the Parent Groups for this operation.

☐ Choose the **Create Operation** icon.

☐ Choose the **FINISH_TURN_OD** icon from the Create Operation Dialog.

The Parent Groups for this program have been pre-defined for you.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **PART**
- Use Tool = **OD_55_L**
- Use Method = **LATHE_FINISH**

☐ Choose **OK**.

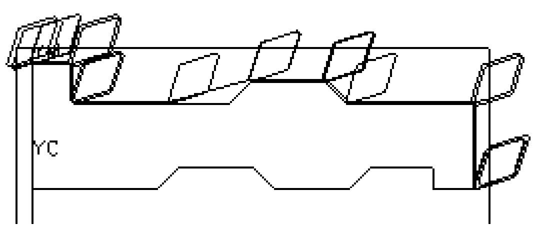
The **FINISH_TURN_OD** dialog is displayed.

☐ Choose the **Groups** tab, the **Geometry** button, and then **Display**.

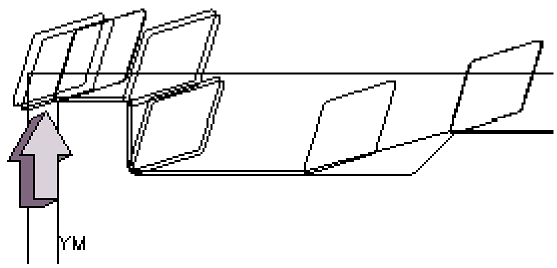
Step 4: Generate the tool path and examine the results.

☐ Set the Tool Display to **2-D**, and **Generate** the tool path.

The tool removes the remaining material and cuts into the reversals. The tool does not cut the back side of the left reversal. You will cut that later. The cut type is set to **Finish All**, so the tool cuts material on all faces and diameters.



Notice that the tool cuts the front face of the part. Also, at the end of the tool path, the tool cuts deeper into the blank material after it passes the part geometry.



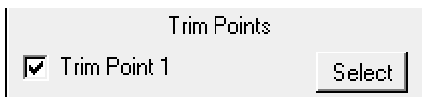
Trim Points will allow you to constrain the cut region to avoid cutting these areas.

Step 5: Create Trim Points.

Trim points allow you to constrain the cut region so that you can be very specific about the areas you wish to cut.

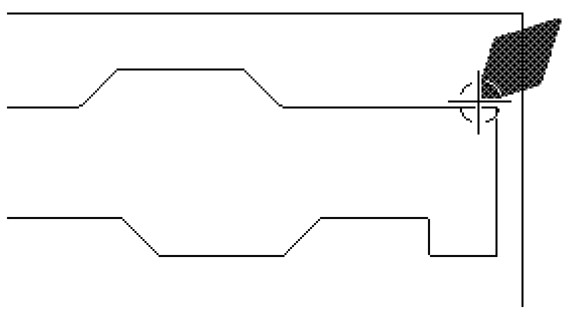
☐ Choose **Containment** under Cut Regions.

☐ Turn on **Trim Point 1** and choose **Select**.



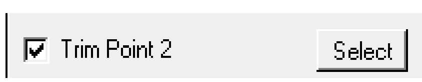
Trim points should be defined on the part boundary. If you define a point slightly off the boundary, the point projects to the part boundary at a minimum distance.

☐ Select the end of the horizontal line near the front of the part.

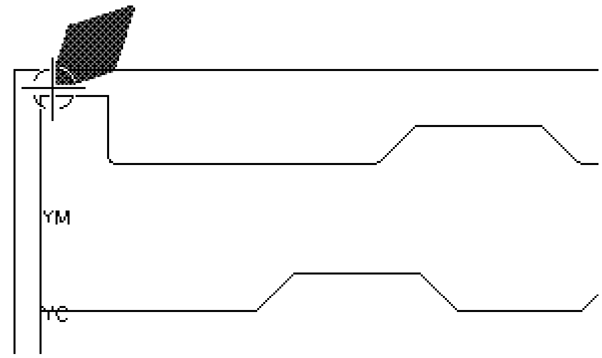


The point is labeled TP 1.

☐ Turn on **Trim Point 2** and choose **Select**.



☐ Select the end of the horizontal line near the back of the part.

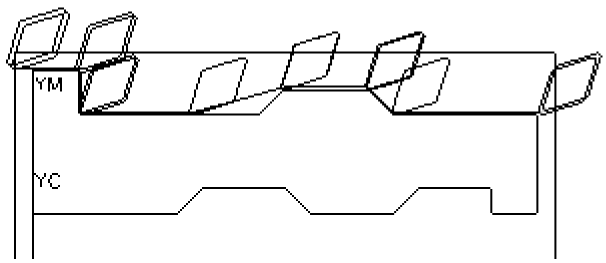


The point is labeled TP 2.

☐ Choose **OK** to accept the trim points.

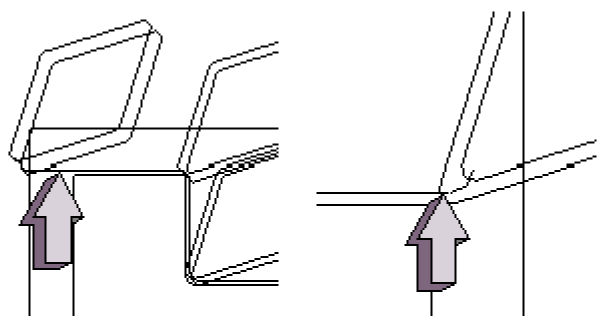
Step 6: Generate the tool path and examine the results.

☐ **Generate** the tool path.



☐ Zoom in on the trim points.

Notice that the tool cuts a short distance before the first trim point and a short distance after the second trim point.



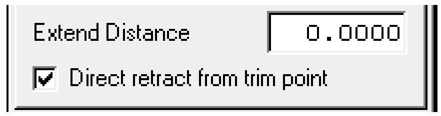
Because trim points are associative to the geometry from which they were created, the tool cuts along the associated geometry as it approaches the first trim point and continues cutting along the associated geometry after it passes the second trim point until it cuts through the in-process workpiece.

You can control the distance the tool cuts before the first trim point and after the second trim point.

Step 7: Retracting directly from the Trim Point.

You will eliminate the unwanted cutting move that occurs after the second trim point.

- ☐ Choose **Engage/Retract**.
- ☐ Choose the **Retract** tab.
- ☐ Turn on **Direct retract from trim point**.



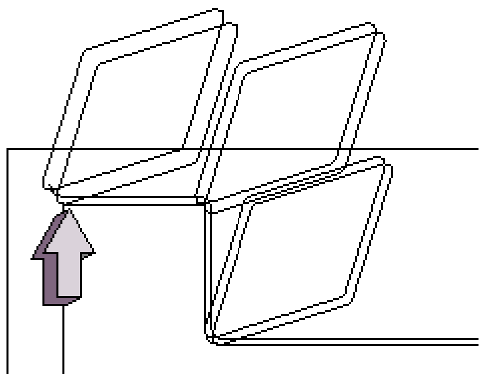
When Extend Distance is set to 0.0, the cutting move to the second trim point is completely eliminated. Entering a value for Extend Distance determines the length of the cutting move following the trim point.

- ☐ **OK** the Engage/Retract dialog.

Step 8: Generate the tool path and examine the results.

- ☐ **Generate** the tool path.

The tool now retracts directly from the second trim point.

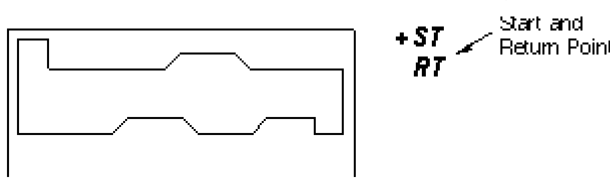


Next, you will define Avoidance geometry.

Step 9: Define the Avoidance geometry.

There are no Start and Return points defined. Define both for this operation.

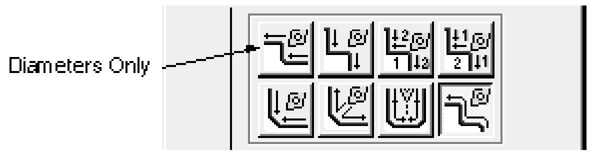
- ☐ Define a **Start** and **Return Point** as shown.



Step 10: Change the Cut Type for this operation.

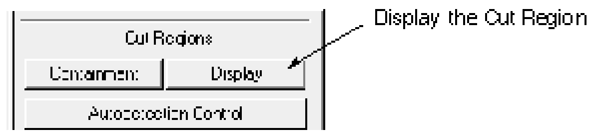
The Cut Types are the same options available when defining Profiling passes in a Roughing operation. The default is Finish All.

☐ Change the Cut Type to **Diameters Only**.



Examine the remaining material.

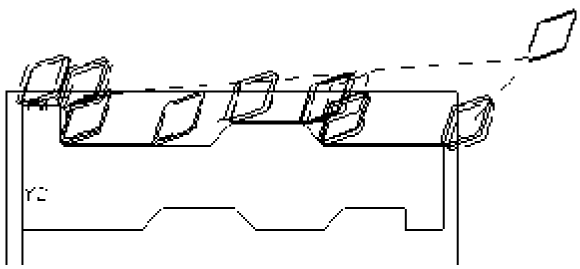
☐ Choose **Display** in the Cut Regions area to display the material.



The cut region is displayed.

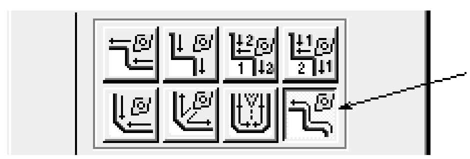
Step 11: Generate the tool path.

☐ Choose the **Generate** icon.



The tool path is generated. The tool cuts to the finish dimension of the part omitting the faces.

☐ Change the Cut Type to **Finish All**.



Step 12: Using Omit Reversal.

You can only "Omit" Reversals in finish operations. In this operation most of the material has been removed from the reversal and you are now ready to finish that area. Note the effect of using the Omit Reversal option in this case.

☐ Turn the **Omit Reversals** option **ON**.

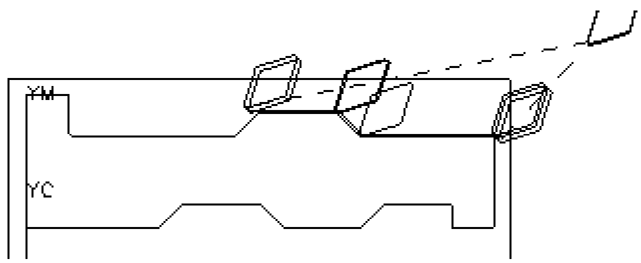
☐ Once again, choose **Display** in the Cut Regions area to display the material to be cut.

Notice the different Cut Regions. Only the front Cut Region is to be cut even though you might expect that the tool would continue cutting through the reversal area at the current level. However, the tool would be "cutting air" because the material in the reversal area has already been removed.

This finish operation will cut the region closest to the last tool location. You can use a Start Point to position the tool close to the region you want to cut.

Step 13: Generate the tool path.

☐ Choose the **Generate** icon.



☐ Choose **OK** to accept the path.

Step 14: Create a second operation to cut the second region on the part.

☐ Highlight the FINISH_TURN_OD operation that was just developed and choose **Copy** and then **Paste**.

☐ Double-click on the copy.

Step 15: On your own, Edit and Generate the operation to cut the second region.

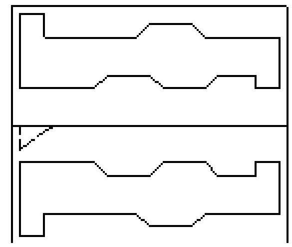
Step 16: **Save** the part file.

Activity: Creating a Finish ID Operation

In this activity, you are going to create an ID Turning Operation with limited assistance.

Step 1: Continue to use same the part file.

- ☐ Continue to use the part file*****_bearing_sleeve_2.prt**.



Step 2: Create a new operation using the finish bore subtype.

- ☐ Choose the **Create Operation** icon.
- ☐ Choose the **FINISH_BORE_ID** icon.

Step 3: Define the Parent Groups.

- ☐ Program = **PROGRAM**
- ☐ Use Geometry = **PART**
- ☐ Use Tool = **ID_55_L**
- ☐ Use Method = **LATHE_FINISH**

Step 4: Define the following options for this operation:

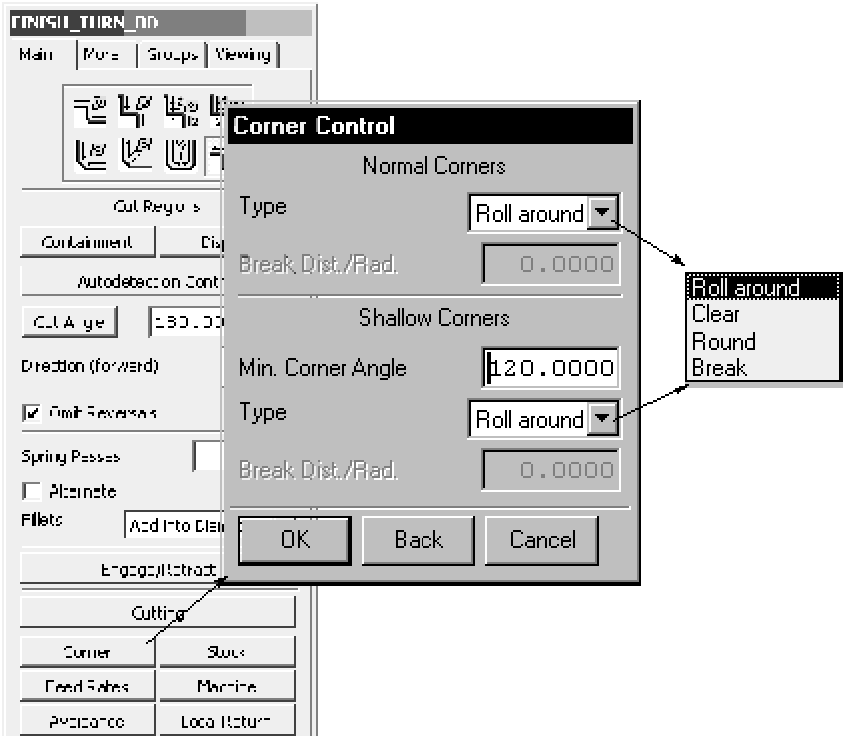
- ☐ Start and Return Point
- ☐ Omit Reversals **ON**

Step 5: Generate the tool path.

Step 6: Save and close the part file.

Corner Control

The Corner Control option allows you to control tool motion when the tool cuts into the workpiece at the boundary corners. You can remove a specified amount of material or none at all.



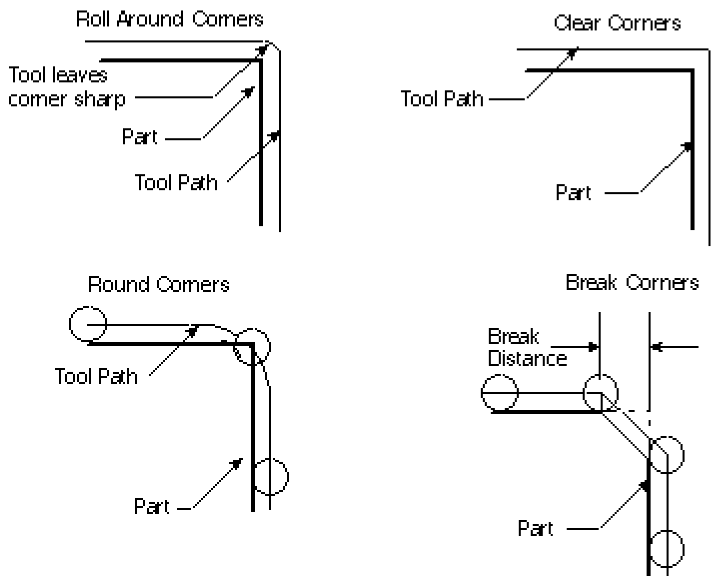
Type Options

Roll Around Corners– keeps the tool in constant contact with the material by inserting an arc into the tool path to keep the tool tangent to the boundary as it moves around the corner.

Clear Corners – causes the tool to leave the Part Boundary at the corners.

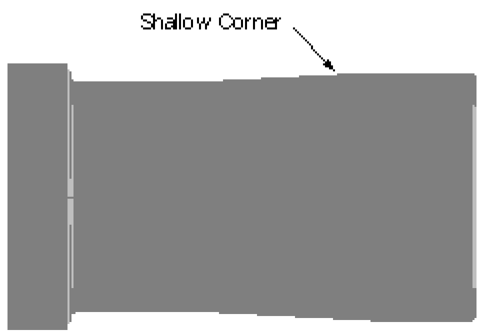
Round Corners– adds a radius to the corner. You enter the desired radius value.

Break Corners– allows chamfering of corners. The tool moves the specified distance in a straight line across the corner.



Shallow Corners

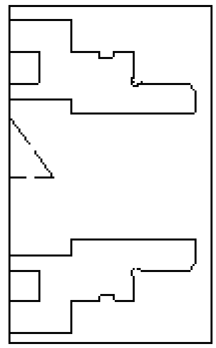
This option allows you to control tool motion when convex corners has an included angle larger than 180 degrees.



Activity: Using Corner Control

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

☐ Open the part file **tmp_fin_od_1.prt**.



☐ Save the part file as ******_fin_od_1.prt***, where *** represents your initials.

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

Step 2: Edit an existing operation.

☐ Double-click the **FINISH_TURN_OD** operation name in the Operation Navigator.

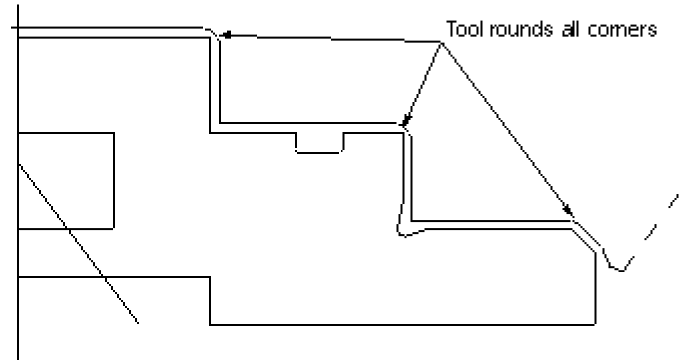
☐ Choose **Replay** from the FINISH_TURN_OD dialog.

Step 3: Examine the corner option results.

The Corner option allows you to control the edge break on convex corners. You can roll around corners, which is the default or you can break, clear, or round the corners.

☐ Look at the resulting tool path displayed in the graphics window.

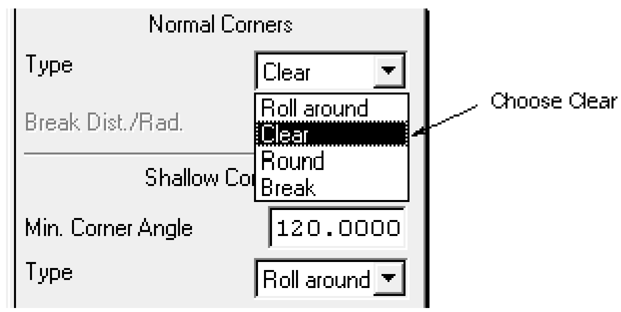
The tool rounds the corners. This is the default setting for the Corner Control option. Change the setting for the Normal Corner option and compare the results.



☐ Choose the **Corner** button.

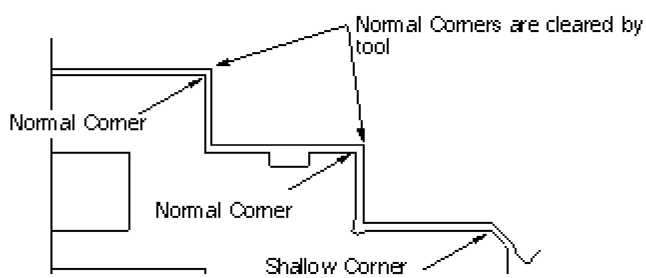
The Corner Control dialog is displayed.

☐ In the Normal Corners area, change the Type to **Clear**, and then choose **OK**.



☐ **Generate** the tool path.

The tool clears the edge of the Normal convex corners on the part. The tool still rolls around the Shallow Corner (the edge of the chamfer).



Generally, you have the round the sharp corners and push the burr in front of the tool. Change the option to Round, and observe the results. You will specify a radius when using this option.

☐ Again, choose the **Corner** button.

☐ In the Normal Corners area, change the Type to **Round**.

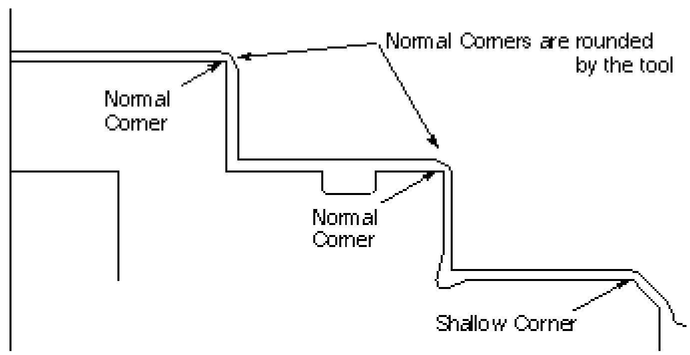
☐ Enter **.05** into the Break Dist/Rad value field.

☐ Choose **OK**.

The FINISH_TURN_OD dialog is displayed.

☐ **Generate** the tool path.

The tool path is generated. The tool rounds the Normal corner using a .05 radius and rounds the corner of the shallow corner.



Step 4: Accept and verify the tool path.

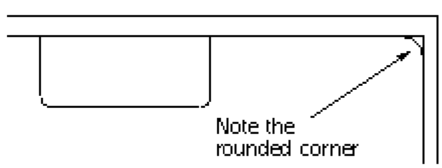
Accept the tool path and verify that the tool path cut the .05 corner.

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

Verify the rounded corners.

☐ If necessary, highlight the operation name **FINISH_TURN_OD** in the Operation Navigator.

☐ Using MB3, choose **Workpiece→Show 2D**.

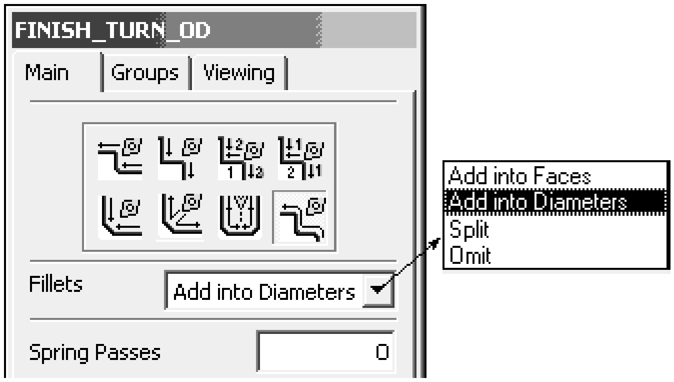


You can see that the tool rounds the corner removing .05 material.

Step 5: **Save** the part file.

Fillets

The Fillets option determines when fillets will be cut. This works in conjunction with the cut patterns Diameters First, then Faces and Faces First, and then Diameters options.



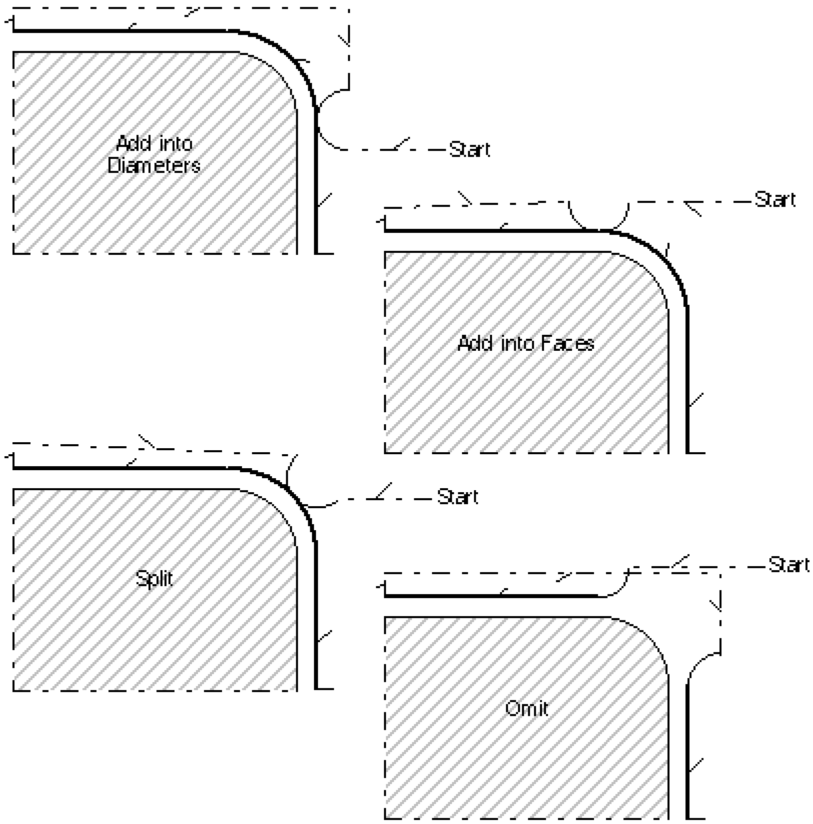
Add into Faces - cuts the fillets when cutting the adjacent face. For example, if you selected Diameters First, Then Faces as the cut type, the fillets are cut when the adjacent faces are cut.

Add into Diameters - cuts the fillets when cutting the adjacent diameters. For example, if you selected Diameters First, Then Faces as the cut type, the fillets are cut when the adjacent diameters are cut rather than the face.

Split - cuts half of the fillet when cutting the adjacent face and the other half of the fillet when cutting the adjacent diameter.

Omit - does not cut the fillets at all.

The following illustrations show the fillet options for "Diameters then Faces" with auto circular engage and auto linear retract.



Summary

Finish operations remove material from previous turning operations.

In this lesson, you:

- Created a Finish OD and ID operation.
- Used the Corner Control option.

Lesson

11 Grooving

Purpose

Grooving is a roughing operation that is used to remove material in grooves or undercut areas.

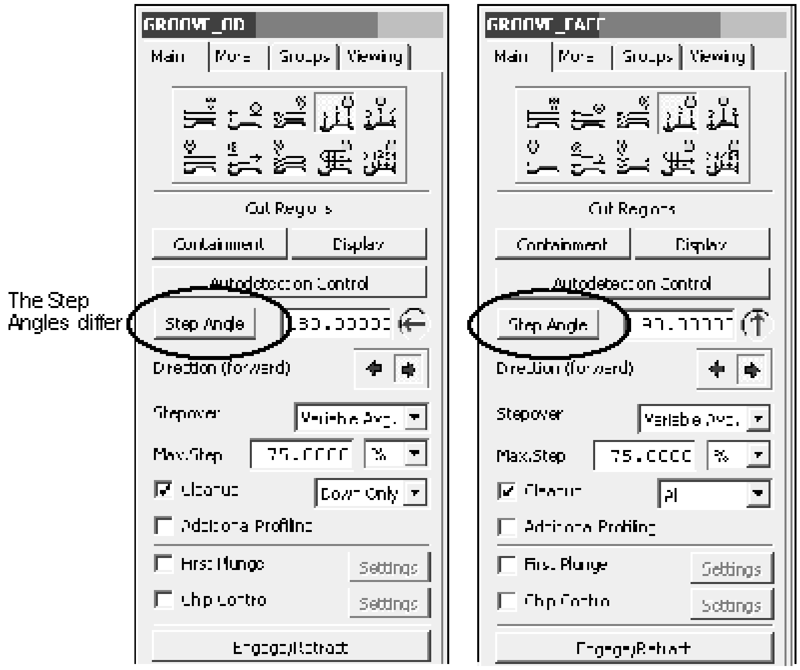
Objective

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

- Identify the different groove methods.
- Identify the options unique to grooving.
- Specify Stepover and Cut Depth.
- Manually specify the cut region.
- Create GROOVE_OD, GROOVE_ID, and GROOVE_FACE operations.

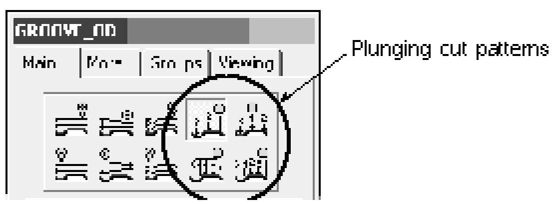
The Groove options

The Groove options located on the three dialogs are the same with the exception of the Step angle. This option determines the angle at which the tool will cut. You can adjust the Step Angle within each operation.



Cut Patterns

There are four plunging cut patterns.



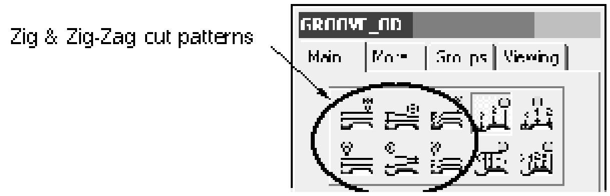
Plunge Zig are typical plunge cuts in one direction within the cut region.

Plunge Alternate cuts with alternating stepover direction. Each subsequent plunge is applied to the opposite side of the first plunge.

Plunge Zig-Zag causes the tool to plunge to a specified depth of cut, then perform a series of plunge cuts to remove all the material at that depth. It then plunges again to the cut depth, and removes all material at that level. These series of cuts are performed back and forth in a zig zag pattern until the bottom of the groove is reached.

Plunge Castling is a unique plunge pattern which leaves pillars between plunges and then returns to cut the pillars. The result is an even cutting pressure on the tool.

There are six cut patterns that cut by level.



Linear Zig creates linear cuts by level in one direction within the cut region.

Ramping Zig is for inclined/declined cuts in alternating directions.

Contour Zig creates cuts by level that follow the contour of the groove in one direction within the cut region.

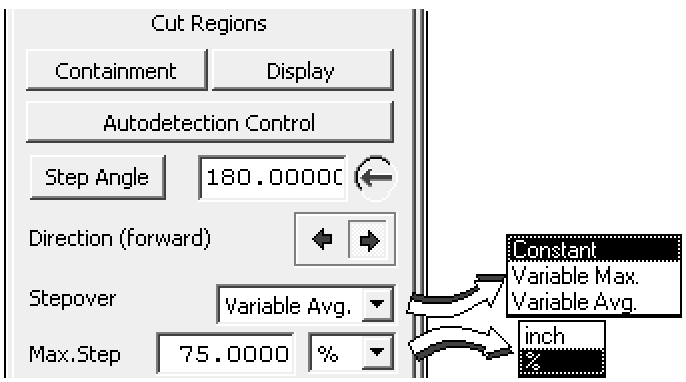
Linear Zig-Zag creates linear cuts by level in both directions within the cut region.

Ramping Zig Zag is for inclined/declined cuts in alternating directions.

Contour Zig-Zag creates cuts by level that follow the contour of the groove in both directions within the cut region.

Stepover

The Stepover determines the amount that the tool moves over between plunges. You have three Stepover methods.



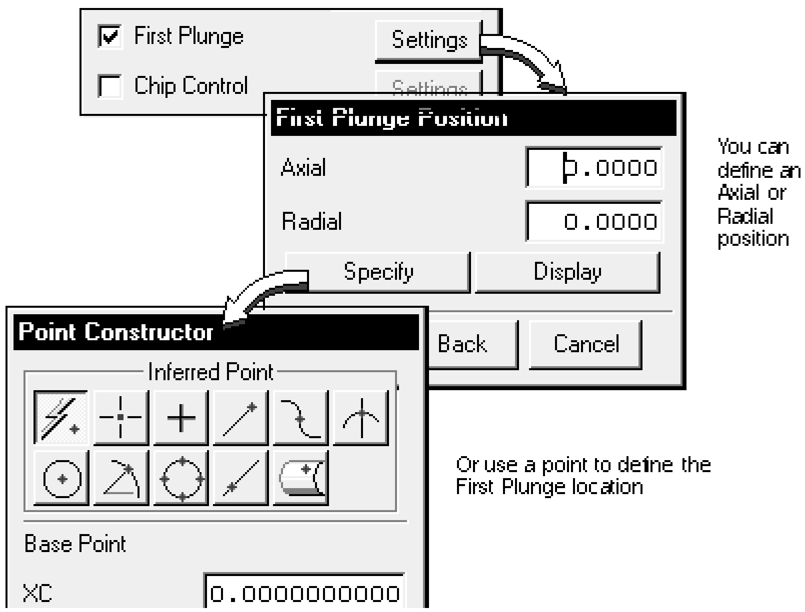
The Stepover options are defined in the table below.

Option	**Description**
Constant	Specifies the maximum depth of cut to be made on each roughing pass. This value is used as many times as possible and then remaining material is cut in one pass.
Variable Max.	Determines the cut regions, cuts at the maximum value as many times as possible and then cuts the remaining material in one pass if it is greater than or equal to the minimum within each separate region.
Variable Avg.	Calculates the minimum number of passes required based on never cutting more than the maximum or less than the minimum for each region.

The Step value is based on a percentage of the tool width or a specified value in inches.

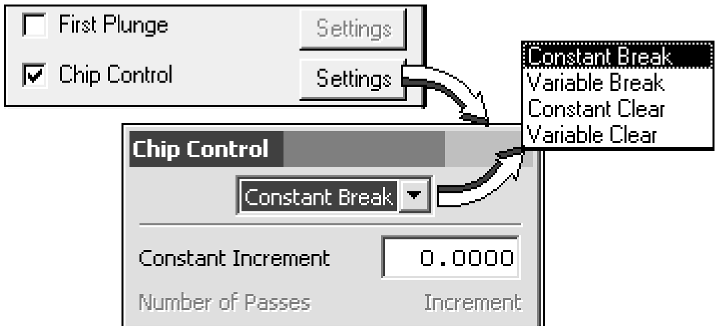
First Plunge

The first plunge of the grooving tool path is automatically determined. The option, First Plunge, allows you to reposition the first plunge.



Chip Control

The typical plunge cut is one non-interrupted cutting motion. Chip Control allows you to interrupt each plunge cut to clear the chips from the groove.

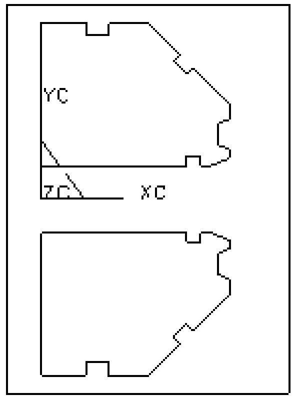


Activity: Creating an OD Groove Operation

In this activity, you are going to create a simple outside diameter grooving operation.

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

☐ Open the part file **tmp_grv_all_1.prt**.



☐ Save the part file with as *****_grv_all_1.prt**, where *** represents your initials.

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

Step 2: Create an Outside Diameter Grooving operation.

You will create a simple GROOVE_OD operation. Define the Parent Groups for this operation.

☐ Choose the **Create Operation** icon.

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

☐ Choose the **GROOVE_OD** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **PART**
- Use Tool = **GROOVE_OD**
- Use Method = **LATHE_GROOVE**

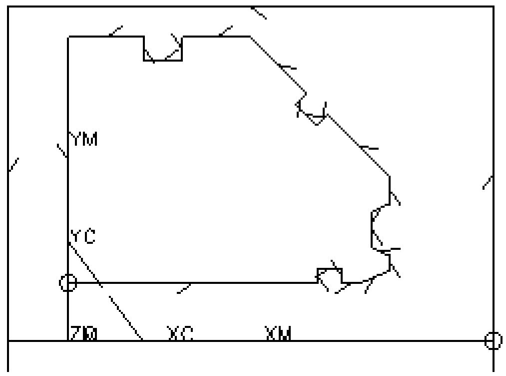
☐ Choose **OK**.

The GROOVE_OD dialog is displayed.

Step 3: Verify the Geometry.

Next you will verify the part and blank geometry.

☐ Choose the **Groups** tab at the top of the dialog, choose the **Geometry** button, then choose **Display**.



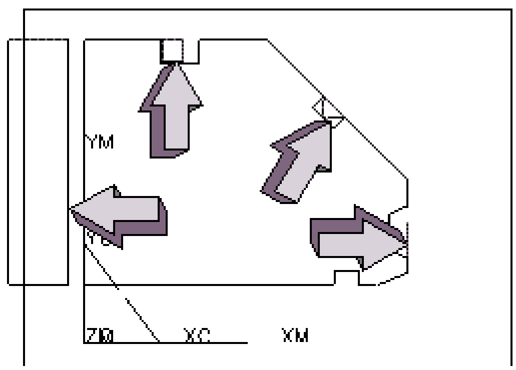
Both the Part and the Blank geometry have been defined in parent groups.

☐ Choose the **Main** tab at the top of the dialog.

Step 4: Display the Cut Regions.

☐ Choose **Display** under Cut Regions.

Notice that there are multiple cut regions.



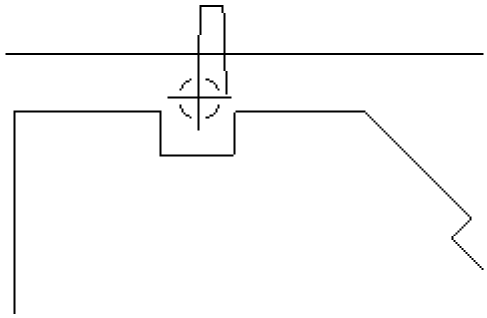
A turning operation can only cut a single region. In cases like this one where there is more than one cut region, you can manually select which region to cut.

Step 5: Specify the Cut Region

☐ Choose **Containment** under Cut Regions.

☐ Choose **Select Manually** and then **Select** under Cut Regions.

- ☐ Use **Cursor Location** to select near the desired cut region as illustrated below.



- ☐ Choose **OK** to accept the cut region.

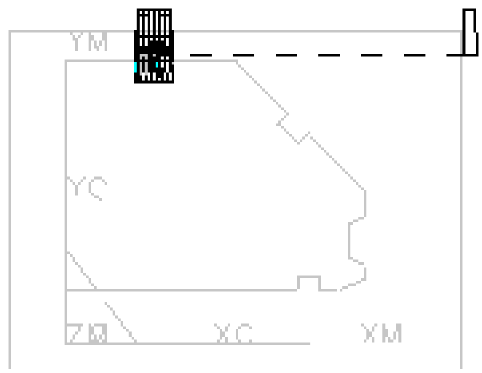
Step 6: Change the Cut Type.

You will plunge using alternating stepover directions.

- ☐ Choose the **Plunge Alternate** icon.

Step 7: Generate the tool path.

- ☐ Choose the **Generate** icon.

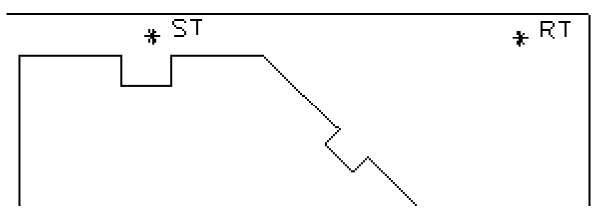


Step 8: Define Avoidance Parameters.

You will define a Start Point, a Return Point.

- ☐ Choose **Avoidance**.

- ☐ Define a **Start** and **Return Point** as shown.

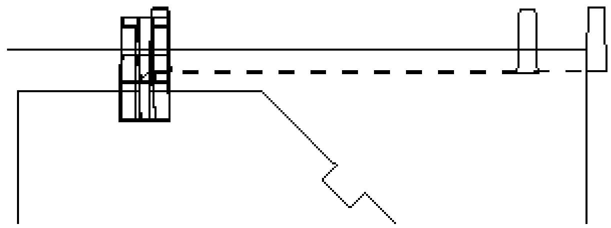


☐ Choose **Radial→Axial** for Motion to Start Point and Motion to the Return Point.

☐ Choose **OK** to accept the Avoidance Parameters.

Step 9: Generate the tool path.

☐ Choose the **Generate** icon.

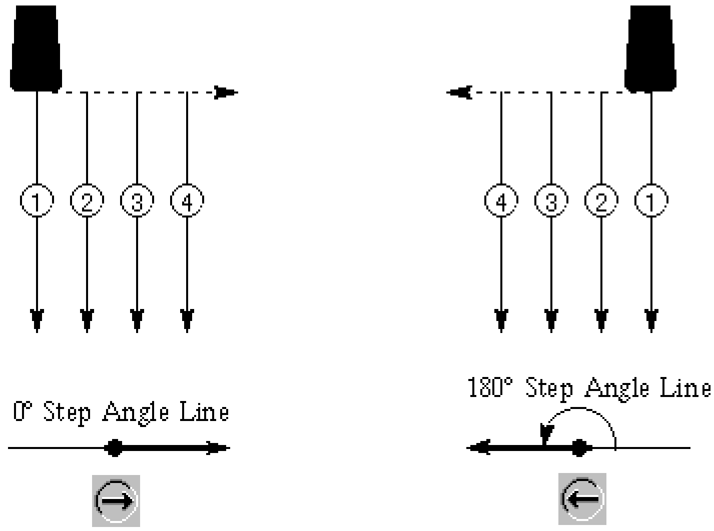


☐ **Save** the part file.

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

Step Angle

The Step Angle allows you to create plunge cuts at any angle. Plunge cuts are perpendicular to the step angle line and are generated in sequence in the direction of the Step Angle line, indicated in the dialog by a red arrow.

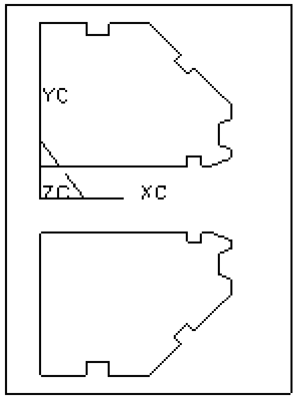


Activity: Creating an Angled OD Groove Operation

In this activity, you are going to create an outside diameter operation that cuts an angled groove using plunge cuts.

Step 1: Continue to use the same part file.

☐ Continue to use the part file *****tmp_grv_all_1.prt**.



☐ Choose the **Create Operation** icon.

☐ Choose the **GROOVE_OD** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **PART**
- Use Tool = **GRV_45_DEG**
- Use Method = **LATHE_GROOVE**

☐ Key in GRV_ANGLE for the name.

☐ Choose **OK**.

The GROOVE_OD dialog is displayed.

Step 2: Change the Step Angle.

You will change the Step Angle so that the plunges cut directly into the groove.

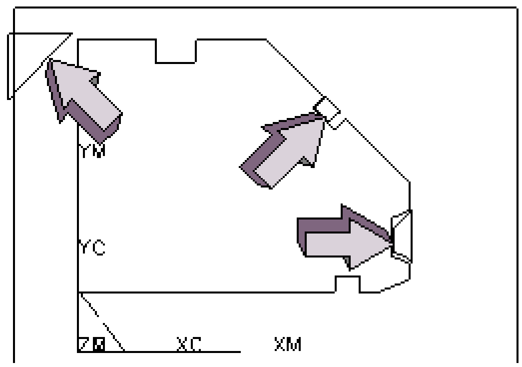
☐ Key in **315** next to Step Angle.

The angle is measured counter clockwise from the centerline and defines the orientation and the direction of progress for the plunge cuts. Notice that the arrow indicates the cut angle for visual reference.

Step 3: Display the Cut Regions.

☐ Choose **Display** under Cut Regions.

Notice that there are multiple cut regions.



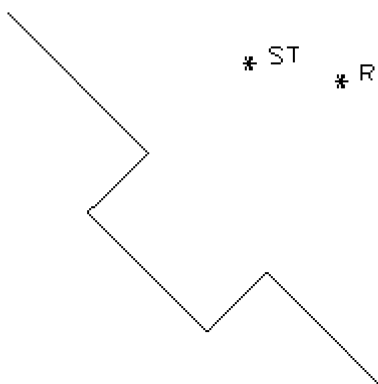
A turning operation can only cut a single region. In the previous operation, you manually selected the cut region. In this operation, you will allow the system to automatically determine the cut region to use based on the Start Point position.

Step 4: Define Avoidance Parameters.

You will define a Start Point and Return Point.

☐ Choose **Avoidance**.

☐ Define a **Start** and **Return Point** as shown.



☐ Choose **OK** to accept the Avoidance Parameters.

The system will use the cut region closest to the Start Point.

Step 5: Change the Cut Type.

You will plunge cut using alternating stepover directions.

☐ Choose the **Plunge Alternate** icon.

Step 6: Create an Additional Profiling Pass.

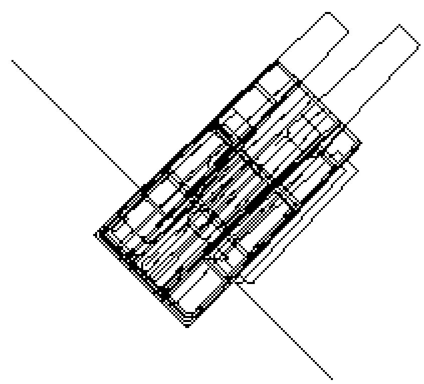
An additional profile pass allows you to perform a cleanup of the part surface after rough cuts have been made.

☐ Choose **Additional Profiling**.

The specific profiling movement that is determined by the Profiling option.

Step 7: Generate the tool path.

☐ Choose the **Generate** icon.



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

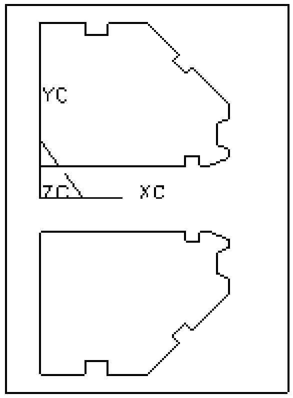
☐ **Save** the part file.

Activity: Creating a Face Groove Operation

In this activity, you are going to create an operation that cuts a groove into the front face of the part using a linear zig-zag cut motion.

Step 1: Continue to use the same part file.

☐ Continue to use the part file *****_grv_all_1.prt**.



☐ Choose the **Create Operation** icon.

☐ Choose the **GROOVE_FACE** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **PART**
- Use Tool = **RTJ_GROOVE_TOOL**
- Use Method = **LATHE_GROOVE**

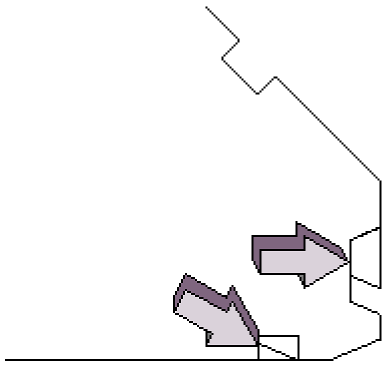
☐ Choose **OK**.

The GROOVE_FACE dialog is displayed.

Step 2: Display the Cut Regions.

☐ Choose **Display** under Cut Regions.

Notice the two cut regions at the lower right corner of the part.



You will allow the system to automatically determine which cut region to use based on the Start Point position.

Step 3: Change the Cut Type.

In this operation, you will rough out the groove one level at a time using a zig-zag motion.

☐ Choose the **Ramping Zig-Zag** icon.

Step 4: Specify the Cut Depth.

The Cut Depth determines the depth of each zig-zag cutting pass.

☐ Be sure **Constant** is specified as the Cut Depth.

Step 5: Create an Additional Profiling Pass.

An additional profile pass allows you to perform a cleanup of the part surface after rough cuts have been made.

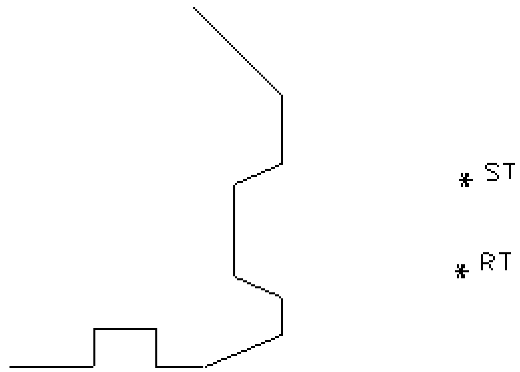
☐ Choose **Additional Profiling**.

Step 6: Define Avoidance Parameters.

You will define a Start Point and Return Point.

☐ Choose **Avoidance**.

☐ Define a **Start** and **Return Point** as shown using **Axial→Radial** for the motions.



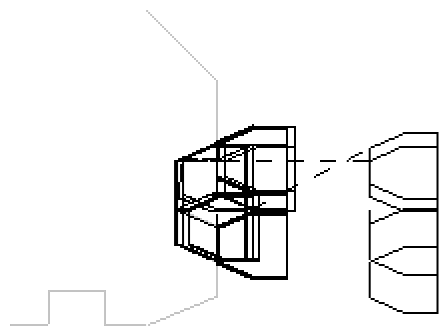
☐ Choose **OK** to accept the Avoidance Parameters.

Step 7: Choose **Cutting** and select the **Settings** button next to **Ramping**.

Step 8: Set the **Ramping Mode** to **Ramp In First**.

Step 9: Generate the tool path.

☐ Choose the **Generate** icon.



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

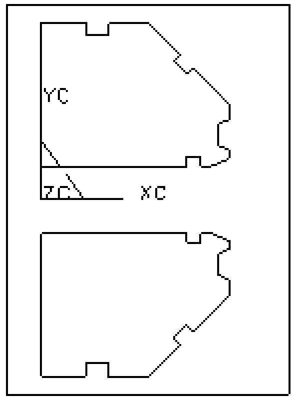
☐ **Save** the part file.

Activity: Creating an ID Groove Operation

In this activity, you are going to create a simple outside diameter grooving operation.

Step 1: Continue to use the same part file.

☐ Continue to use the part file *****_grv_all_1.prt**.



☐ Choose the **Create Operation** icon.

☐ Choose the **GROOVE_ID** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **PART**
- Use Tool = **ID_GRV_TOOL**
- Use Method = **LATHE_GROOVE**

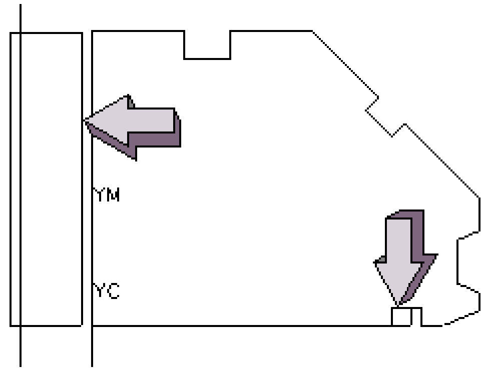
☐ Choose **OK**.

The GROOVE_ID dialog is displayed.

Step 2: Display the Cut Regions.

☐ Choose **Display** under Cut Regions.

Notice that there are two cut regions.



You will allow the system to automatically determine which cut region to use based on the Start Point position.

Step 3: Change the Cut Type

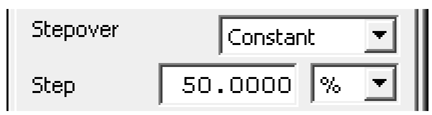
You will plunge using alternating stepover directions.

☐ Choose the **Plunge Alternate** icon.

Step 4: Specify the Stepover.

When plunging, the stepover determines the maximum amount of material that can be removed on each cutting pass. This value is used as many times as possible and then any remaining material is cut in one pass.

☐ Choose **Constant** for the Stepover and key in **50** for the Step %.

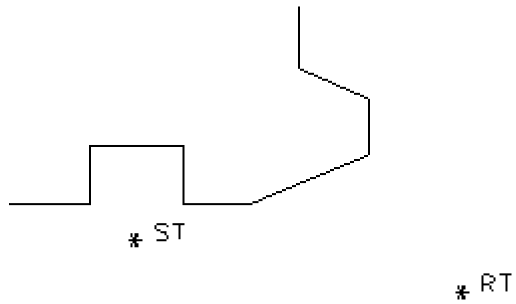


Step 5: Define Avoidance Parameters.

You will define a Start Point, a Return Point and motions that prevent the tool from colliding with the part.

☐ Choose **Avoidance**.

☐ Define a **Start** and **Return Point** as shown.

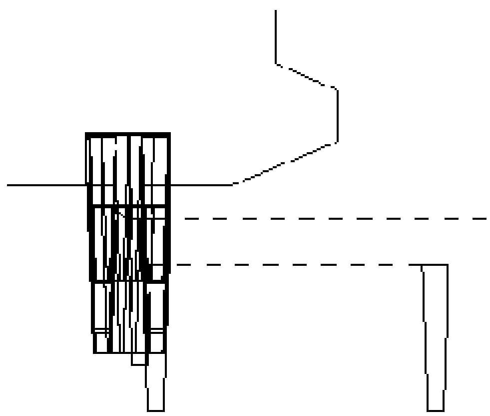


☐ Choose **Radial→Axial** for Motion to Start Point and Motion to Return Point.

☐ Choose **OK** to accept the Avoidance Parameters.

Step 6: Generate the tool path.

☐ Choose the **Generate** icon.



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

Step 7: **Save** and **Close** the part file.

Summary

Grooving removes material in grooves or undercut areas.

In this lesson you:

- Learned the different cut patterns.
- Learned about the unique options in Groove.
- Learned to specify Stepover and Cut Depth.
- Learned to manually specify the cut region.

Lesson

12 *Teach mode*

Purpose

Teach Mode allows you to manually define the cut sequence by selecting the Drive geometry. This method of selecting the geometry and other non-cutting moves allows for very precise finish cutting.

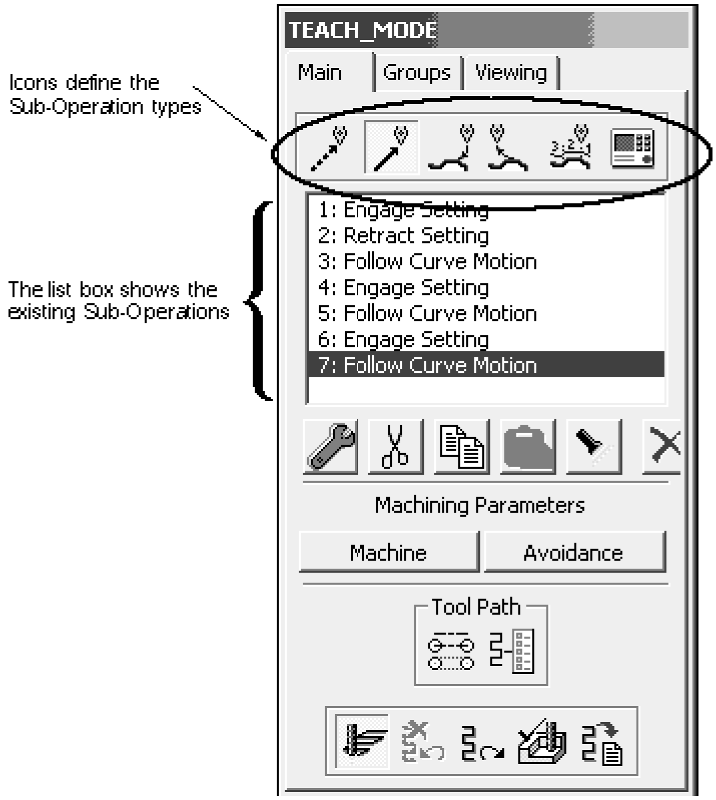
Objective

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

- Identify the different types of sub-operations.
- Determine the sequence of the sub-operations.
- Identify the Drive geometry.
- Identify the Check geometry.

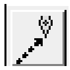
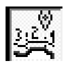
The Teach Mode options

Teach Mode allows you to create a sequence of sub-operations used to drive the cutter. The Main dialog is where you define the type of sub-operations.



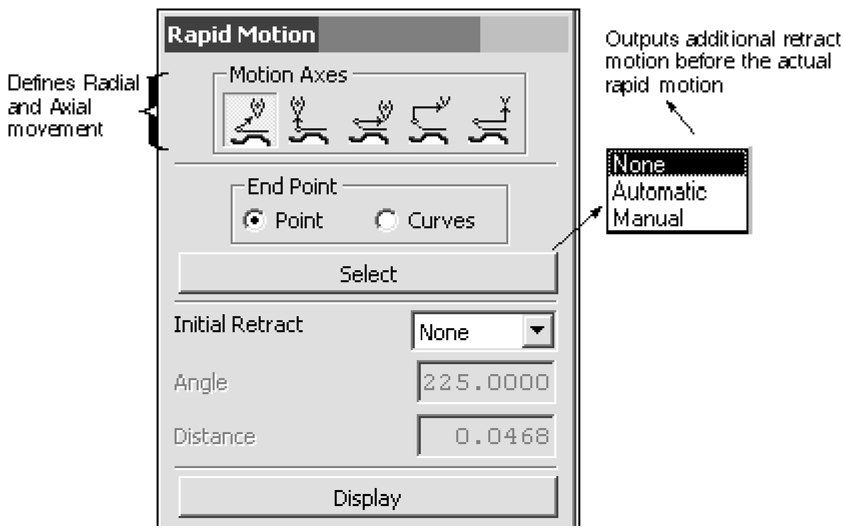
Sub-Operation Types

There are 6 Sub-Operation types:

- Linear Rapid 
- Linear Feed 
- Engage Setting 
- Retract Setting 
- Follow Curve Motion 
- Machine Control Events 

Linear Rapid

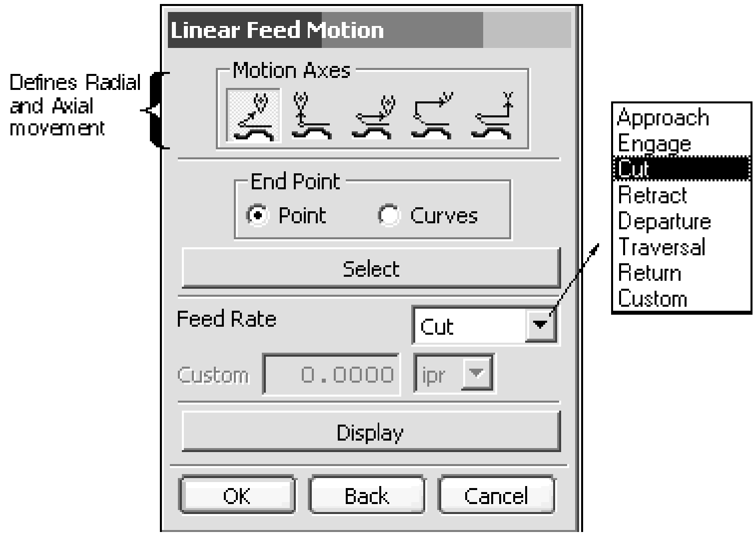
Linear Rapid allows you to define a point to which the tool will position in Rapid mode.



The option, Pop-away Move, allows you to specify an additional retract motion from last tool position which is output before the Rapid motion. The retract motion can be specified by its angle to the positive rotation axis and the motion length. These values can be inferred from the selected tool or can be specified by the selection of Automatic or Manual. It is disabled by selecting the option menu entry None.

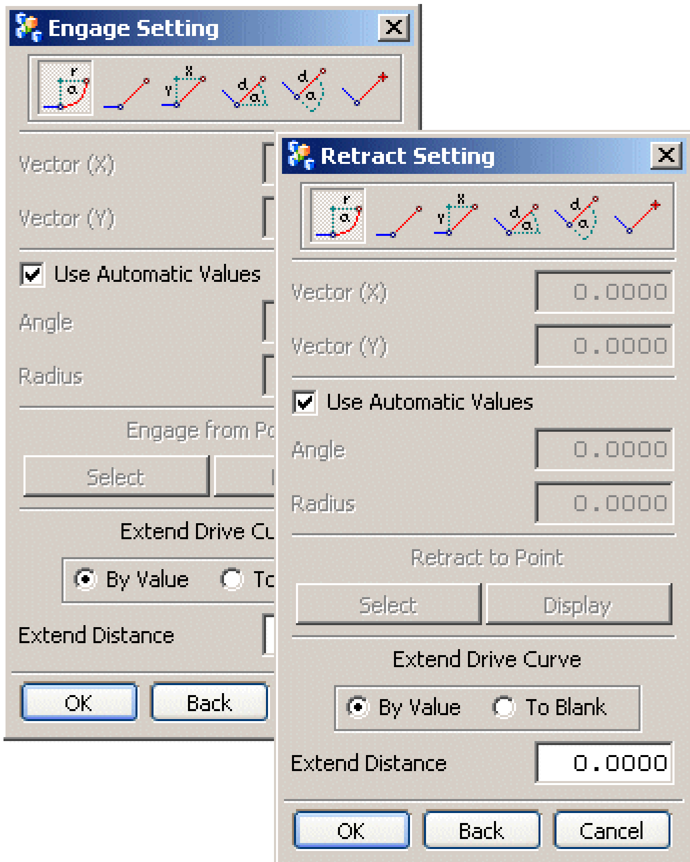
Linear Feed

Linear Feed defines a motion from the current tool position to a new position in Cut Mode at a specified feed rate. You can apply different feed rates to the cut motions.



Engage and Retract Setting

The Engage Setting dialog is used to specify a modal engage move that is used by the subsequent Follow Curve Sub-Operations.



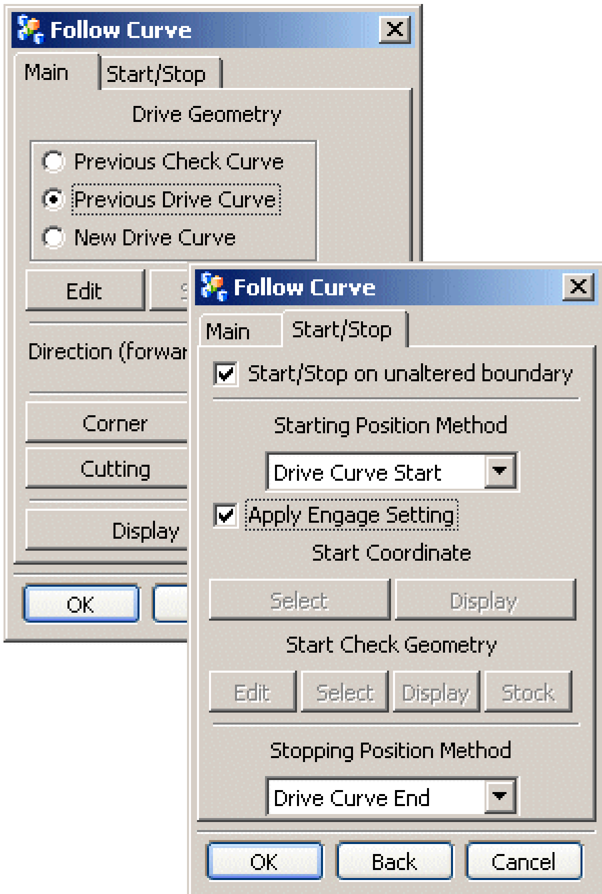
Extend Drive Curve is used to create overlapping tool paths on the drive curve starts/ends within the In-process Workpiece. This will allow the tool to engage TO rather than INTO the blank.

You can manually or automatically extend the drive curve at the start and end by distance (manual) or to the blank geometry (automatic).

Both the Engage/Retract settings need to be set prior to the motion Sub-Operation(s).

Follow Curve Motion

Follow Curve allows you to interactively define the drive geometry. This allows you to machine selected segments.



Drive Geometry

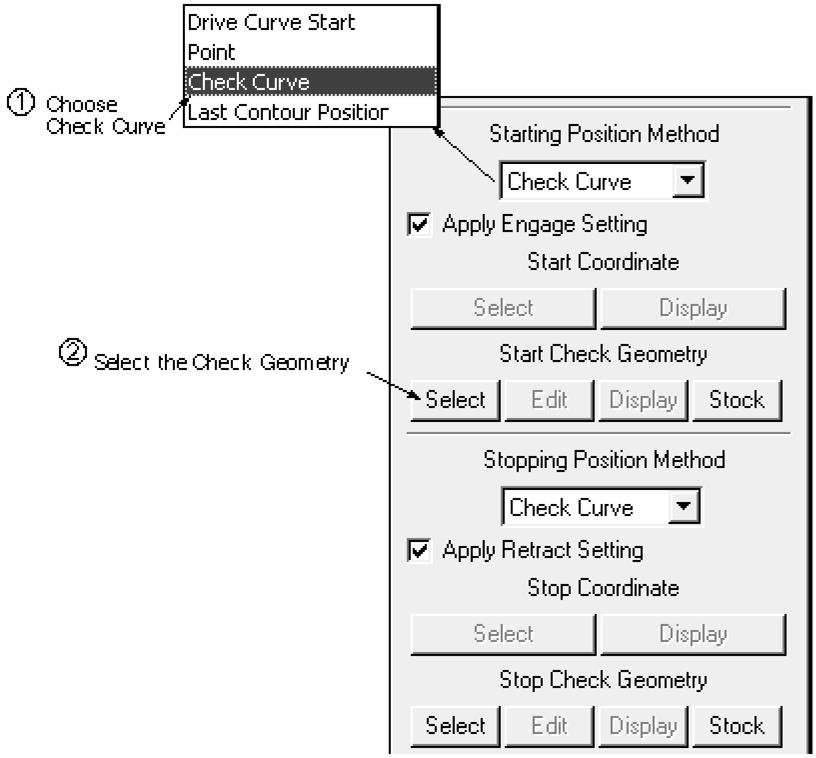
The start/stop positioning method "Point" gives you better control of the starting/stopping positions. This is useful in controlling when a tool should roll around a corner that is selected as a start/stop position.

When selecting the point where to start/stop, you can also specify corresponding geometry that defines a position from where engage or retract.

Option	**No Previous Follow Curve**	**Previous Follow Curve**
Previous Check Curve		Uses check geometry as stop position method
Previous Drive Curve		Uses same geometry as Previous Follow Curve
New Drive Curve	Selects new curve to machine	

Check Geometry

Check Geometry is used to prevent the tool from gouging the part or fixture. You can specify a Check Curve as a Starting Position and a Stopping Position as shown below.



Summary

This lesson was an introduction to Teach Mode.

In this lesson you:

- Learned the different types of sub-operations and how they are used.
- Learned the sequence that you use to typically define the sub-operations.

Lesson

13 Threading Operations

Purpose

This lesson teaches you how to create OD and ID threading operations.

Objective

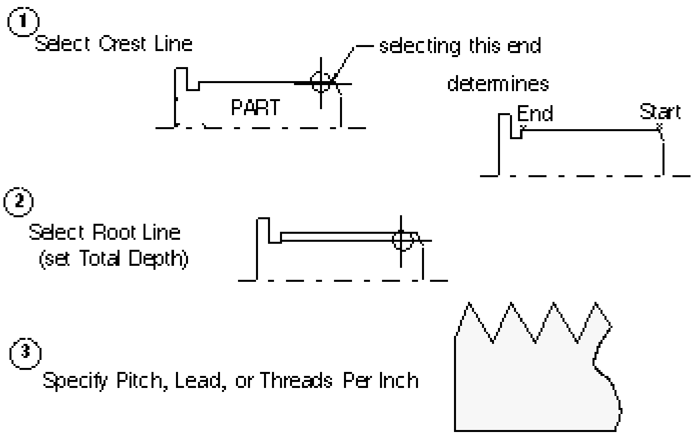
Upon completion of this lesson, you will be able to:

- Define the thread geometry.
- Identify the thread root.
- Identify the crest line.
- Specify the pitch.
- Define the cut increments.
- Create finish passes.
- Create chase passes.

Threading Operations

Threading operations can be developed to cut both straight and tapered threads. They can be single start or multiple start threads which are either internal, external, or face threads (such as the thread advance used in a three-jaw chuck).

Developing a Threading Operation



Crest Line

The Crest Line represents the outer tip of the thread.

The Start Point is the point at which the threading motion begins.

Unigraphics determines the length and angle of the thread based on the selected line, and displays START and END at each end of the line.

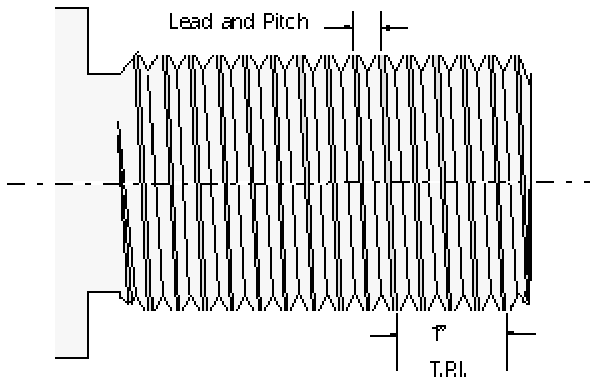
Root

The Root Line determines the depth of the thread.

Pitch

The Pitch is the distance between corresponding points measured from one thread to the next. You specify the Pitch by defining the Pitch and Lead or by defining the Threads Per Inch.

Pitch/Lead/Tpi



You must specify the Pitch, Lead or Threads Per Inch.

The Number of Starts determines the Pitch and Lead as shown in the following example of a 1/4-20 thread:

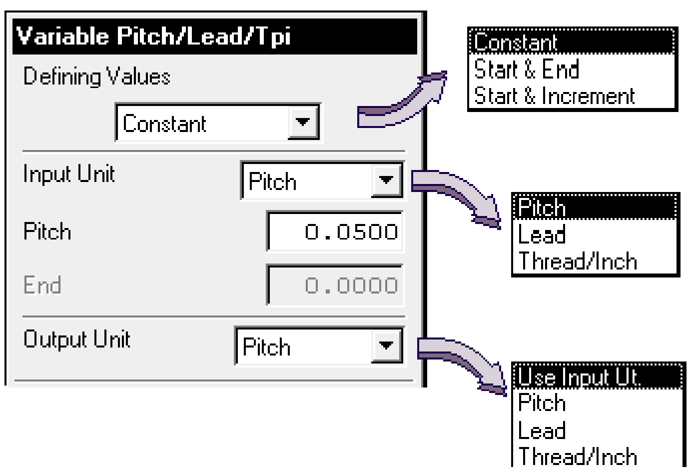
Threads Per Inch = 20

Pitch = (1 (inch) / 20) = .05

Lead = .05 x 1 (No. of Starts)

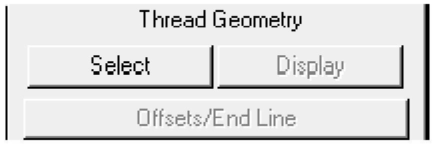
In this case, the Pitch and Lead are .050 because there is only one Start. If there were two Starts, the Lead would be .100, or twice the Pitch.

When you choose the Specify button, the following dialog is displayed. You can define the following options from this dialog.



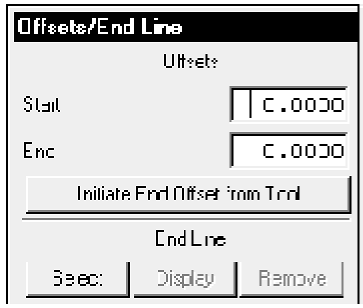
Thread Geometry

Thread Geometry defines the Start and End Point of the thread.



Offset/End Line

Offset/End line adjusts the start point and end point for the thread motion. Positive Offset values increase the thread length; negative Offset values shorten the thread length.



Offsets are measured along the thread angle, which is dependent upon the method of depth specification.

Start adjusts the Start Point of the thread.

End adjusts the End Point of the thread.

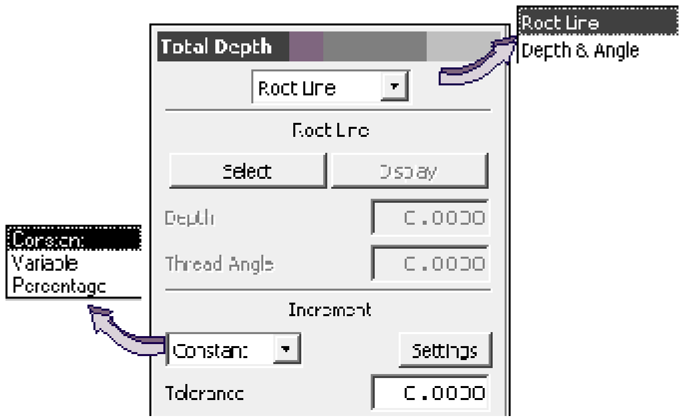
Init From Tool causes the system to calculate the End Offset so that the tool clears the thread.

End Line defines the end of the thread by selecting a line which intersects the Crest Line.

Total Depth

Total Depth determines the thread root line.

The Total Depth gives you options to establish a Root Line by using the cursor to select a line, or by entering Depth and Angle values.



Each Increment method provides a Tolerance option which allows you to specify the increment for the last roughingpass.

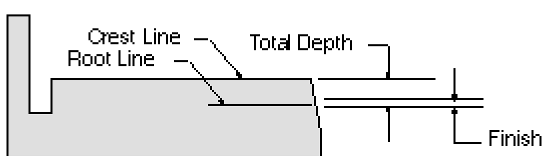
Constant - Specify a single increment value that is used throughout the tool path. You may wish to use this method when specifying relatively few roughing passes since tool pressure rapidly increases with each pass. The tool moves the input distance as it cuts along the thread angle until it reaches the start of the Finish Passes.

Variable - Specify a set of variable increments and the number of times each is to be repeated. This allows the greatest control over individual passes.

If the sum of the increments does not equal the start of the Finish Passes depth, the system repeats the last non-zero increment value until the appropriate depth is reached. If the sum of the increments exceeds the start of the Finish Passes depth, the system ignores the excess increments.

Percentage - This method is particularly useful in roughing threads because it resembles the finishing technique (even allowing you to bypass that option and using the Chase option instead) and is easily modified to accommodate different materials, thread pitch, RPM changes, etc.

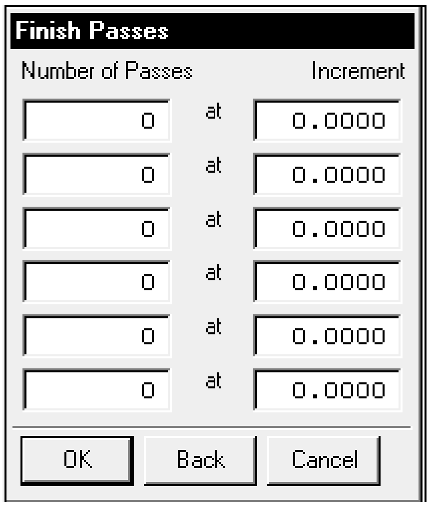
This method allows you to specify the increment depth of each pass as a percentage of the total roughing depth that remains at the time of the pass. This causes step distances to decrease as the tool moves deeper into the thread.



Finish Passes

You can specify a number of finish passes and increments when cutting your threads.

When you generate a thread tool path, the tool cuts to the Roughing Thread Depth using the number of passes determined by the Increment Method and data specified under the Total Depth option. The tool then proceeds to the Finish Thread Depth using the number of passes determined by the Finish Passes data you specified.

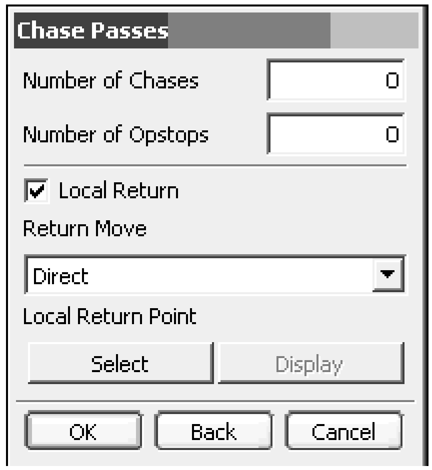


Chase Passes

You can specify a number of spring passes to control size of thread and minimize tool deflection using the Chase Passes option. The passes are made at the Total Thread Depth.

You can also specify the number of OPSTOPS to be output and a local return move.

Chase passes are always bracketed by OPSTOP, OPSKIP/ON and OPSKIP/OFF commands.

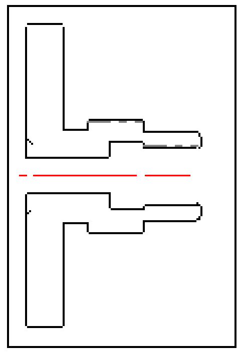


Activity: OD Threading

In this activity, you will create an operation to cut an external thread. This will include defining the necessary thread parameters, such as pitch, Offset/End lines, crest, and root geometry.

Step 1: Open the part file, save as and enter the Manufacturing application.

☐ Open the part file **tmp_thread_1.prt**.



☐ Save the part file as *****_thread_1.prt** , where *** represents your initials.

This session has been initialized using the Lathe configuration and the Lathe CAM setup.

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

Step 2: Define the Parent Groups for this operation.

☐ Choose the **Create Operation** icon.

☐ Change the Type to **Turning**, if necessary.

☐ Choose the **THREAD_OD** icon from the Create Operation dialog.

☐ Set the following Parent Groups:

- Program = **PROGRAM**
- Use Geometry = **OD**
- Use Tool = **THREAD_EXT**
- Use Method = **THREAD_METHOD**

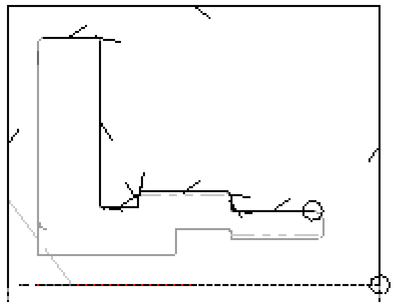
☐ Choose **OK**.

The THREAD_OD dialog is displayed.

Verify the geometry.

☐ Choose the **Groups** tab, choose the Geometry button, and then choose **Display**.

The geometry boundaries are displayed.



Step 3: Define the Pitch. This determines the thread size.

☐ Choose the **Main** Tab.

☐ Choose **Specify** next to the Pitch/Lead/Tpi label.

The Variable Pitch/Lead/Tpi dialog is displayed. The default is Constant value and Pitch. You will define the Pitch value.

☐ Choose **Constant** from the dialog.

The methods of specifying the pitch are displayed.

☐ Again, choose **Constant**.

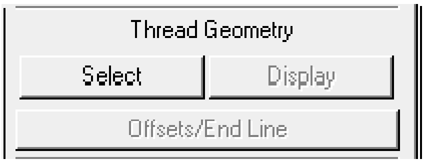
☐ Next to the Pitch label, key in **18** in the value field.

☐ Choose **OK**.

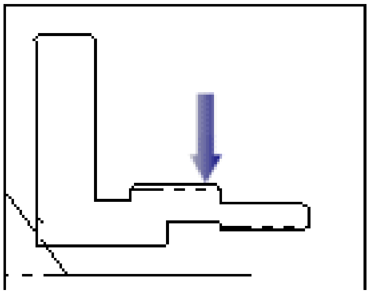
Step 4: Define the crest line.

The crest line is the top of the thread. This is defined using the Thread Geometry selection.

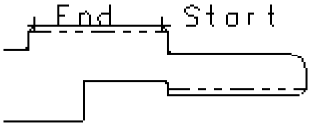
☐ Choose the **Select** option in the Thread Geometry area.



☐ Select the crest line as shown.



The start and end of the thread are defined.



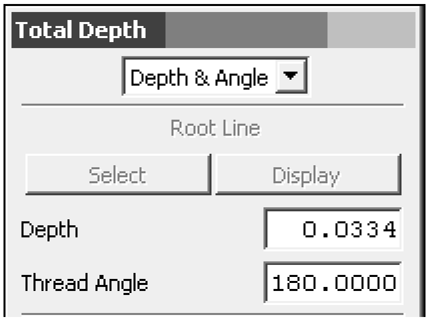
The THREAD_OD dialog is displayed.

Step 5: Define the Total Depth the tool will cut

☐ Choose the **Settings** button under Total Depth.

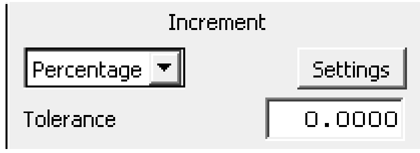
The Total Depth dialog is displayed. You will first define the Total Depth and Thread angle by keying in values. You will then define the Total Depth by selecting the root line.

☐ Choose **Depth & Angle** and key in **0.0334** for the Depth, and **180** for the Thread Angle.



The Increment defines how the tool will cut from the crest line to the final depth. Percentage is one of the most common methods used to define the Increment.

☐ Change the Increment to **Percentage** and choose **Settings**.



The Percentage Increment dialog is displayed. You must specify the Percentage and the Maximum and Minimum values.

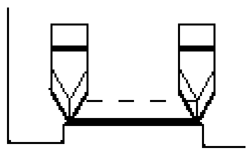
☐ Set the following values:

- Percentage = 30
- Maximum = .04
- Minimum = .005

☐ Choose **OK**.

☐ Choose **OK** to accept the total depth.

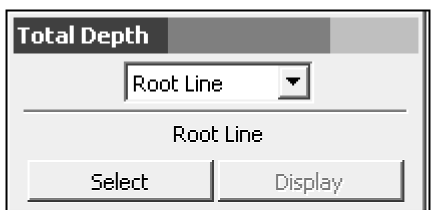
☐ **Generate** the tool path.



Now, you will see how to define the Total Depth by selecting a root line.

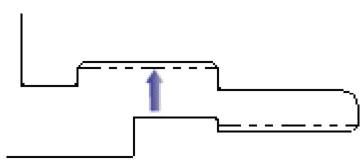
☐ Choose the **Settings** button under Total Depth.

☐ Choose **Root Line** for the Total Depth and choose **Select**.



You can now select the appropriate geometry.

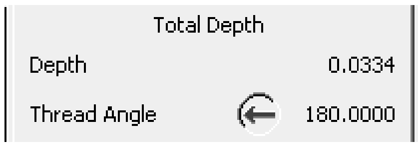
☐ Choose the Thread Root Line as shown below.



The root line is highlighted in the graphics screen and the Total Depth dialog is displayed.

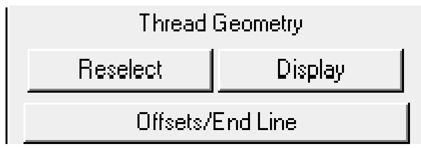
☐ Choose **OK** to accept the total depth.

The Total Depth and Thread Angle have been calculated from the selected root line.



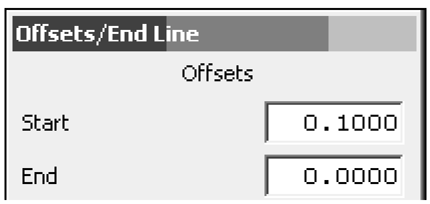
Step 6: Define an Offset for the Start of the Thread.

☐ Choose **Offsets/End Line** in the Thread Geometry area.



The Offsets/Endline dialog is displayed.

☐ Key in **0.100** in the Start field.

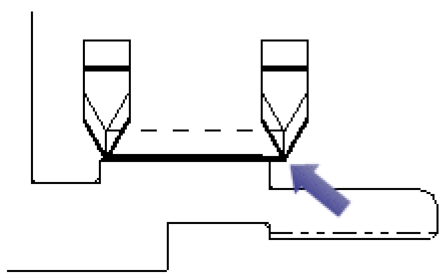


☐ Choose **OK**.

The THREAD_OD dialog is displayed.

You are ready to generate the tool path.

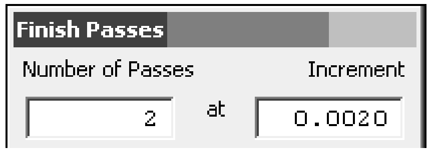
☐ **Generate** the tool path.



Step 7: Specify a Finish Pass.

☐ Choose the **Finish Passes** button.

☐ Enter **2** passes of **.002**.



☐ Choose **OK** to return to the THREAD_OD dialog.

Step 8: Specify a Chase Pass.

☐ Choose the **Chase Passes** button.

☐ Enter **2** Chase passes.

☐ Choose **OK** to return to the THREAD_OD dialog.

☐ Reduce the **Path Display** speed.

☐ **Generate** the tool path.

The tool path is generated with the Finish and Chase Passes.

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

Activity: ID Threading

In this activity, you will create an operation to cut an internal thread. This time you will have fewer instructions.

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

- ☐ Continue to use the part file **_thread_1.prt**.

Step 2: Choose the **THREAD_ID** icon and define the Parent Groups for this operation.

Step 3: Define the Parent Groups.

- ☐ Program = **PROGRAM**
- ☐ Use Geometry = **ID**
- ☐ Use Tool = **THREAD_INT**
- ☐ Use Method = **THREAD_METHOD**

Step 4: Define the following options for this operation.

- ☐ Pitch and Constant value
- ☐ Crest Line
- ☐ Root Line
- ☐ Total Depth and Increment

Step 5: **Generate** the tool path.

Step 6: Edit the operation and define the following options:

- ☐ Final Pass
- ☐ Chase Pass
- ☐ Engage and Retract
- ☐ Avoidance Geometry

Step 7: **Generate** the tool path.

Step 8: **Save** and **close** the part file.

Summary

In this lesson, you created an operation to cut threads on the OD of the part. You also created threads on the ID of the part.

In this lesson you:

- Selected the thread geometry.
- Defined the root and crest line of the threads.
- Specified the thread pitch.
- Defined the cut increments.
- Specified finish and chase passes.

Lesson

14 Using Multiple Spindles

Purpose

This lesson will teach you how to create Complex Cross Section curves relative to multiple MCS objects and how to use those curves to define part geometry for parts mounted at each spindle of a multiple spindle machine.

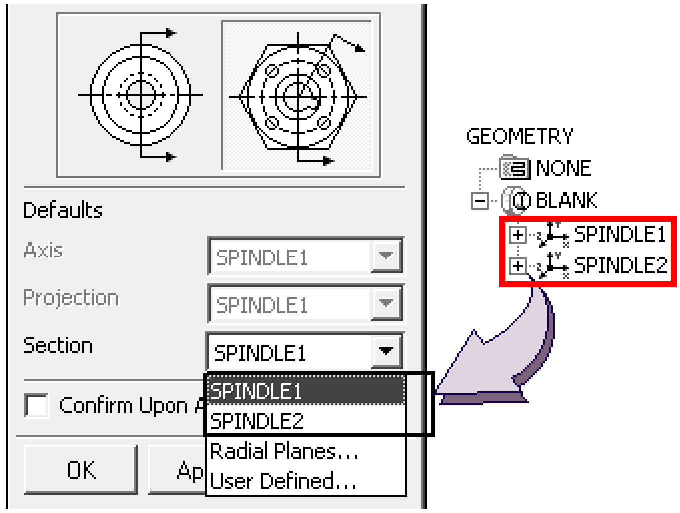
Objective

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

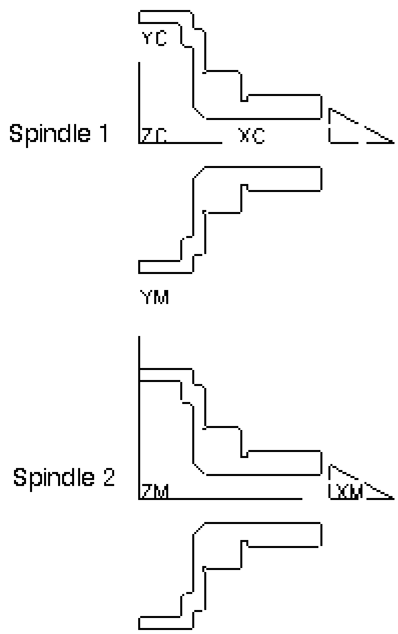
- Create Complex Cross Section curves relative to multiple MCS objects.
- Define part geometry for parts mounted at each spindle of a multiple spindle machine.
- Generate a roughing tool path at one spindle.
- Generate a finishing tool path at the other spindle.
- Create in-process workpieces.
- Use the current in-process workpiece to define the blank geometry for the next operation.
- Flip and reorient tool to machine back side of part.

MCS Objects Available for Lathe Cross Sections

The spindles defined in the Operation Navigator by MCS objects are available for selection in the Axis, Projection, and Section option menus of the Lathe Cross Section dialog.



This allows you to create complex cross section curves for parts mounted at each spindle of a multiple spindle machine.

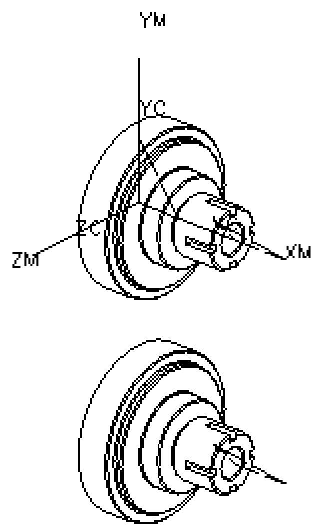


Activity: Using a Multiple Spindle Setup

You are going to create Complex Cross Section curves relative to two MCS objects, each defining a spindle. You will use these cross section curves to define part geometry at each spindle. You will then rough the part on one spindle and finish it on the other spindle.

Step 1: Examine the Setup for a Two Spindle Machine.

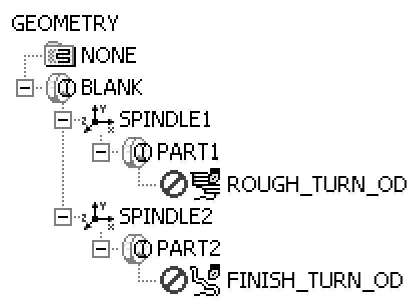
☐ Open the part **tmp_turn_assy.prt**.



☐ Save the part file as *****_turn_assy.prt**, where *** represents your initials.

☐ Choose **Application→Manufacturing**.

☐ Display the Geometry view of the Operation Navigator and expand the objects.



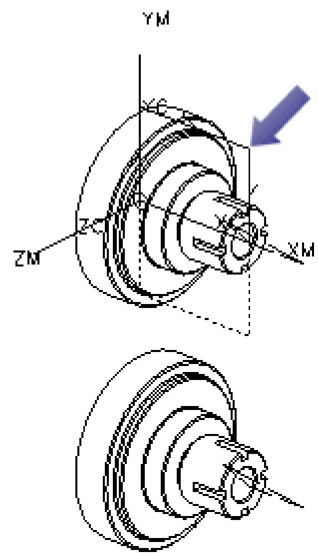
Step 2: Display the Blank Boundary.

The blank boundary has been defined in the BLANK object.

☐ Double-click the BLANK object in the Operation Navigator.

☐ Choose the **Blank** icon in the WORKPIECE dialog.

☐ Choose **Display**.



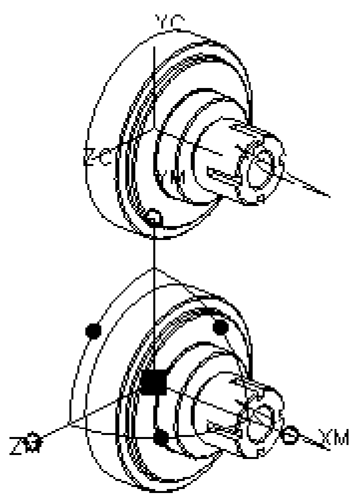
The blank boundary will be used at Spindle 1 for the roughing operation. The In Process Workpiece will then be used at Spindle 2 for the finishing operation.

☐ Cancel the WORKPIECE dialog.

Step 3: Display the spindles.

Each spindle is defined in the assembly model by an MCS.

☐ Double-click on the SPINDLE2 object in the Operation Navigator to display the second MCS.



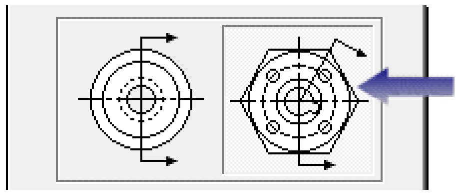
☐ **Cancel** the MCS_SPINDLE dialog.

Step 4: Create Cross Section curves at the First Spindle.

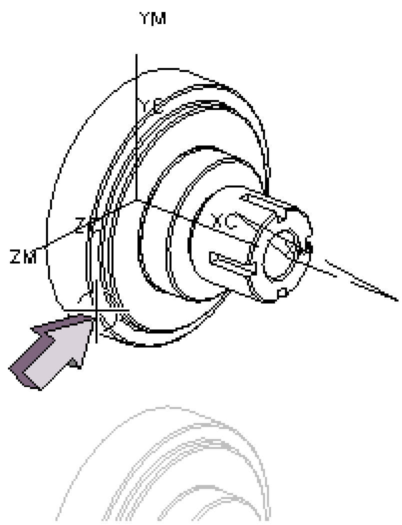
You will create cross section curves for the part mounted on SPINDLE1. The section plane has been rotated to avoid intersecting the keyways.

☐ Choose **Tools**→ **Lathe Cross-Section** in the menu bar.

☐ Choose the **Complex Section** icon.



☐ Select the body mounted at the first spindle.



☐ Choose **OK** to accept the body.

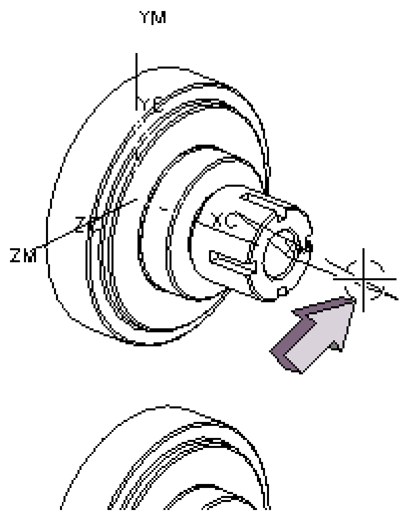
☐ Choose **OK** to accept SPINDLE1 as the rotation axis.

☐ Choose **OK** to accept SPINDLE1 as the projection plane

☐ Choose **User Defined** to define the section plane.

☐ Choose **Existing Plane** in the Plane dialog

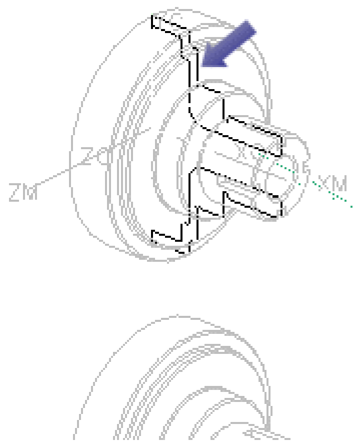
☐ Select the plane symbol at SPINDLE1.



☐ Choose **No** to specify that you do not wish to select additional planes.

☐ Choose **OK** in the Lathe Cross Section dialog.

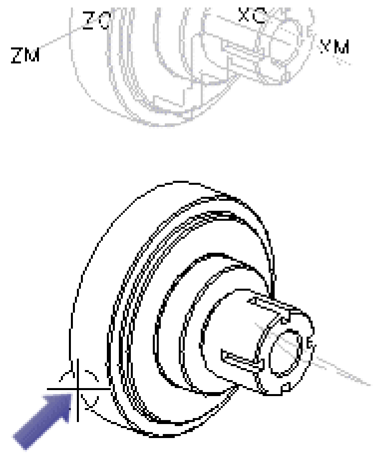
The cross section curves are created for the part mounted on the first spindle.



Step 5: Create Cross Section Curves at the Second Spindle

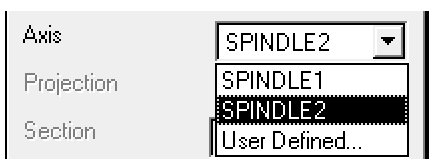
You will now create cross section curves for the part mounted on SPINDLE2.

☐ Select the body mounted at the second spindle.



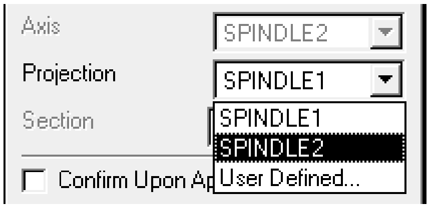
☐ Choose **OK** to accept the body.

☐ Choose SPINDLE2 as the rotation axis.



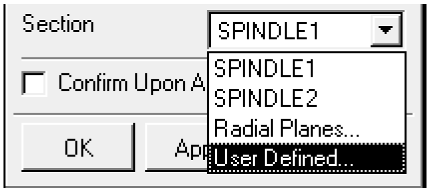
☐ Choose **OK** to accept the rotation axis

☐ Choose **SPINDLE2** as the projection plane.



☐ Choose **OK** to accept the projection plane.

☐ Choose User Defined to define the section plane.



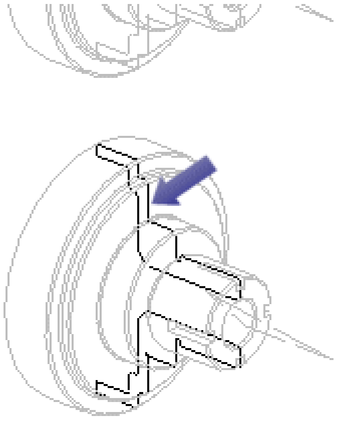
☐ Choose **Existing Plane**.

☐ Select the plane symbol at SPINDLE2.

☐ Choose **No** to specify that you do not wish to select additional planes.

☐ Choose **OK** in the Lathe Cross Section dialog.

The cross section curves are created for the part mounted on the second spindle.



☐ **Cancel** the Lathe Cross Section dialog.

Step 6: View the Cross Section Curves

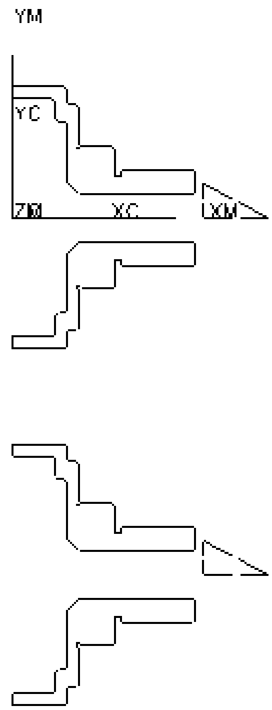
You will now change to a top view and display only the cross section curves.

☐ Choose **Format→Layer Settings** from the menu bar.

☐ Choose layer 2 and then **Invisible**.

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

☐ Choose **MB3→Replace View→Top**.



Step 7: Define Part Geometry for the Roughing Operation.

You will now define the part geometry at Spindle1 for the ROUGH_TURN_OD operation.

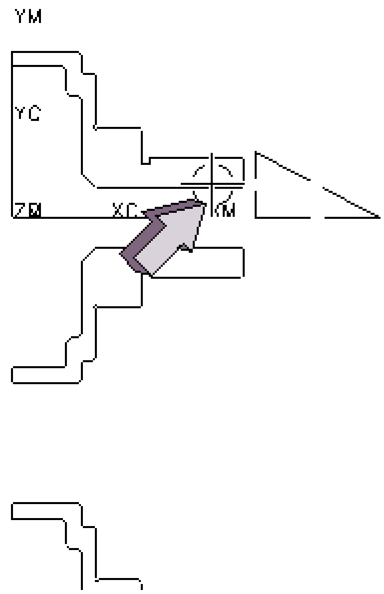
☐ In the Geometry view of the Operation Navigator, double-click on the PART1 object.

☐ With the **Part** icon chosen, choose **Select**.

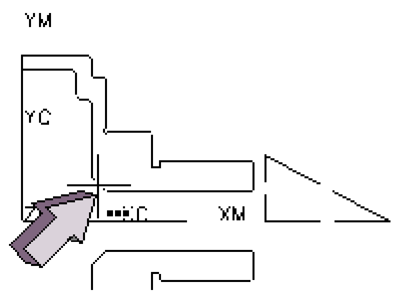
☐ With the **Curve Boundary** icon chosen, choose **Closed** under Type. Be sure Material Side is **Inside**.

☐ Choose **Chaining**.

☐ Select the horizontal line at the approximate position illustrated below (right end).

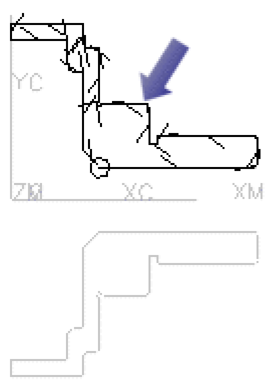


☐ Select the angled line.



☐ Choose **OK** in the Part Boundary dialog.

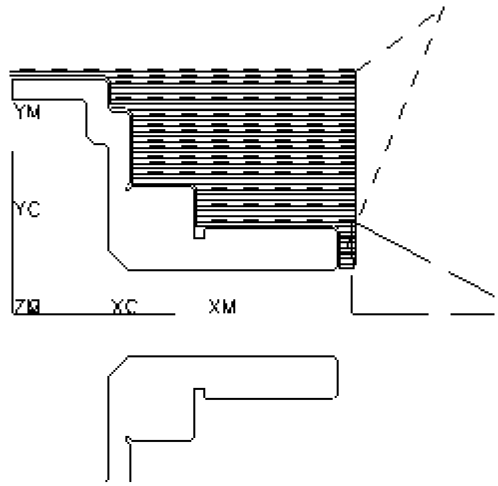
☐ Choose **Display**.



☐ **OK** the TURN_BND dialog to complete the boundary.

Step 8: Generate the Finishing Tool Path

☐ Choose ROUGH_TURN_OD in the Operation Navigator and **MB3→Generate**.



☐ Choose **OK** to complete the tool path.

Step 9: Define Part Geometry for the Roughing operation.

You will now define the part geometry at Spindle2 for the FINISH_TURN_OD operation.

☐ In the Geometry view of the Operation Navigator, double-click on the PART2 object.

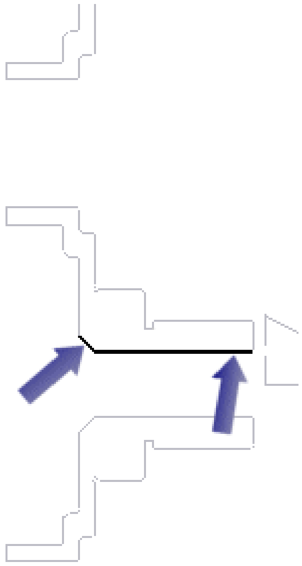
☐ With the **Part** icon chosen, choose **Select**.

☐ With the **Curve Boundary** icon chosen, choose **Closed** under Type. Be sure Material Side is **Inside**.

☐ Choose **Chaining**.

☐ Select the horizontal line at the approximate position illustrated below (right end).

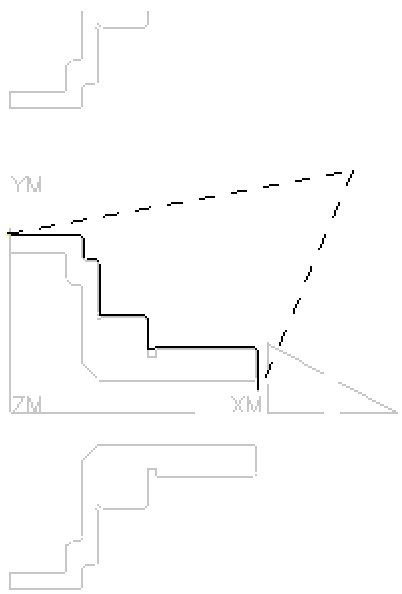
☐ Select the angled line.



☐ Choose **OK** twice to create the part boundary.

Step 10: Generate the Roughing Tool Path

☐ Choose FINISH_TURN_OD in the Operation Navigator and **MB3→Generate**.



☐ Choose **OK** to complete the tool path.

Step 11: **Save** and **close** the part file.

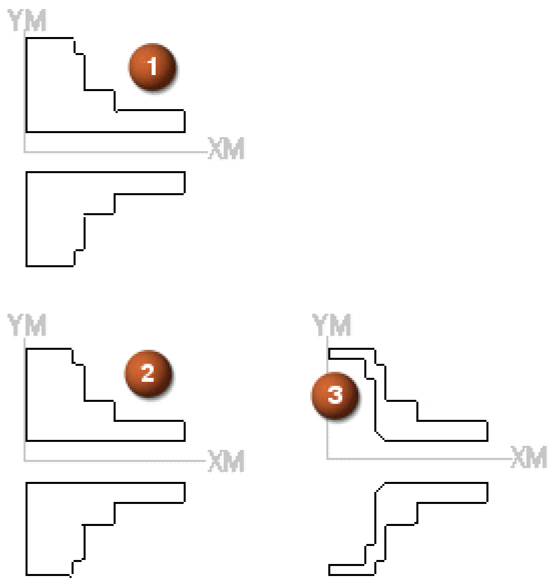
IPW Management for Multiple Spindles

An in-process workpiece (IPW) can be created and referenced as it is transferred from one spindle to another or is turned around in the spindle. When the workpiece is remounted, the IPW referenced from the previous workspace defines the blank geometry in the current workspace. This allows the system to progressively remove material while passing the workpiece from one workspace to the next.

1. Spindle 1, Roughing OD

2. Spindle 2, Finishing OD

3. Spindle 3, Facing left side



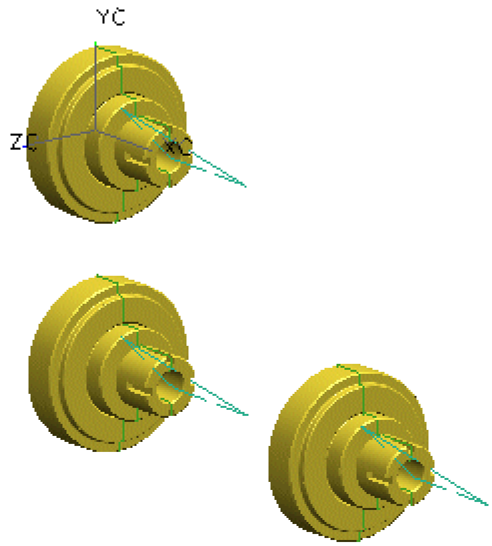
Activity: Creating and Using an IPW

You are going to open an assembly model containing three parts, each mounted on a separate spindle. You will define the initial blank geometry at the first spindle, generate the tool path, and use the resulting IPW as blank geometry at the second spindle for the next operation. You will then repeat this process to create tool paths at the second and third spindles.

Step 1: Examine the Setup for a Three Spindle Machine.

☐ Open the part **tmp_turn_assy1.prt**.

Because this is an assembly, make sure the Load Method is set to From Directory. If your part does not appear as illustrated below, choose File→Options→Load Options→From Directory.



☐ Save the part file as *****_turn_assy1.prt**, where *** represents your initials.

☐ Choose **Application→Manufacturing**.

☐ Display the Geometry view of the Operation Navigator and expand the objects.

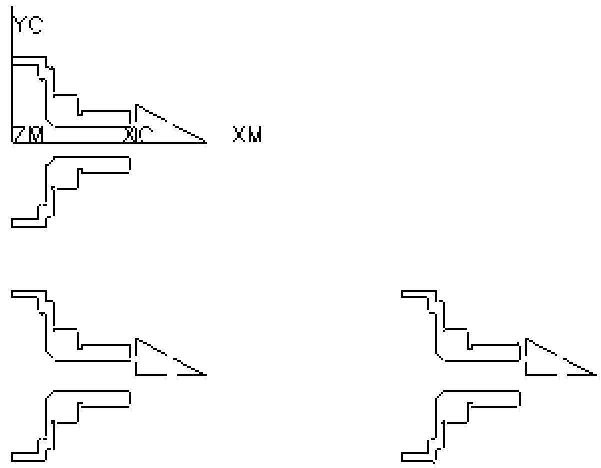
☐ Choose **Format→Layer Settings**.

☐ Choose layer **2** and **Invisible**.

☐ **OK** the Layer Settings dialog.

☐ Choose **MB3→Replace View→Top**.

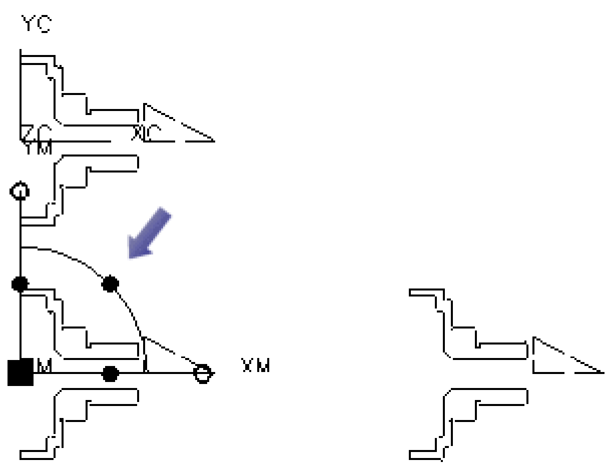
☐ Choose **MB3→Display Mode→Wireframe** so you can see the cross section curves at each spindle clearly.



Step 2: Display the spindles.

Each of the three machine spindles is defined on the system as an MCS object. This setup represents a multiple spindle lathe machine that transfers the part from one spindle to another as the NC program is executed.

☐ Double-click the SPINDLE2 object in the Operation Navigator to display the second MCS.



☐ **Cancel** the MCS_SPINDLE dialog.

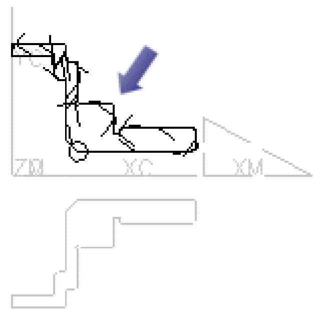
☐ Double-click the SPINDLE3 object to display the third MCS and then **Cancel** the dialog.

Step 3: Display the part boundary.

You will observe that the part boundary has been defined at each spindle.

☐ Double-click the PART1 object in the Operation Navigator.

☐ With the **Part** icon chosen, choose **Display** to see the part boundary at SPINDLE1.



☐ **Cancel** the TURN_BND dialog.

☐ Display the part boundaries at SPINDLE2 and SPINDLE3.

Step 4: Define the initial blank boundary.

You will define the initial blank boundary in the first workspace (SPINDLE1).

☐ Double-click the PART1 object in the Operation Navigator.

☐ Choose the **Blank** icon.

☐ Choose **Select**.

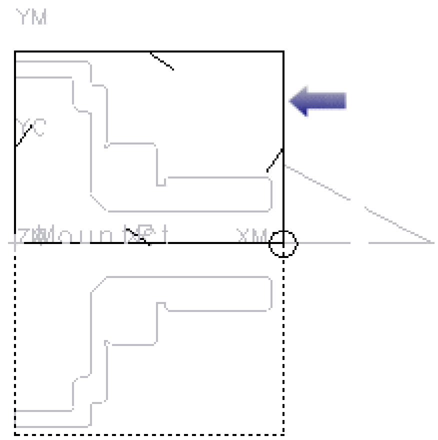
☐ Be sure the **Bar Stock** icon is chosen and choose **Select** under Mounting Position.

☐ Be sure the XC, YC, and ZC values are all set to zero and **OK** to accept the Base Point.

☐ Key in **8.000** in the Length field and **11.500** in the Diameter field.

☐ **OK** to accept the blank boundary.

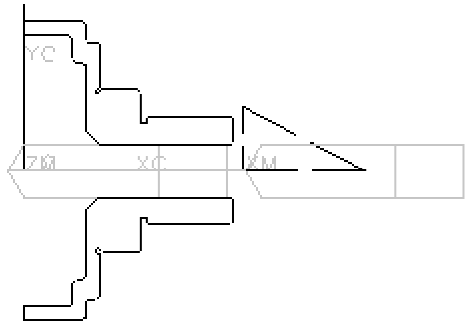
☐ **OK** to accept the TURN_BND dialog.



Step 5: Generate the tool path for the first operation.

Now that the blank boundary has been defined, you can generate the tool path at SPINDLE1 for the first operation.

☐ In the Operation Navigator, highlight the CENTERLINE_DRILLING operation and **MB3→Generate**.



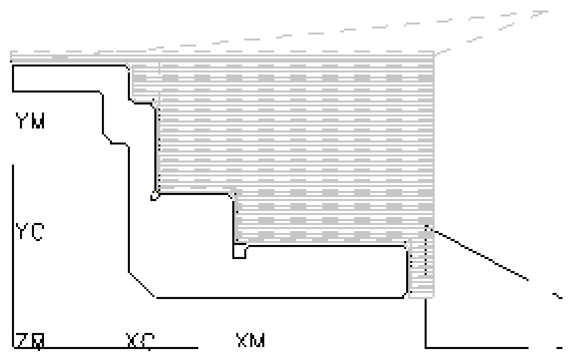
☐ **OK** to accept the tool path.

☐ Refresh the graphics display.

Step 6: Generate the tool path for the second operation.

You can generate the tool path at SPINDLE1 for the second operation.

☐ In the Operation Navigator, highlight the ROUGH_TURN_OD operation and **MB3→Generate**.



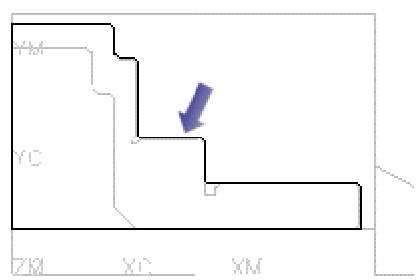
☐ **OK** to accept the tool path.

☐ Refresh the graphics display.

Step 7: Display the IPW for the second operation.

The IPW generated by the second operation will become the blank geometry for the third operation at spindle three.

☐ Highlight the second operation, **ROUGH_TURN_OD**, and **MB3→Workpiece→Show 2D**.



This is the IPW the third operation will use.

Step 8: Defining blank material for the third operation.

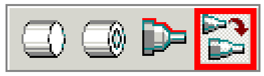
Blank geometry for the third operation (at SPINDLE2) is defined by referencing the IPW generated by the second operation at SPINDLE1.

☐ Double-click the PART2 object in the Operation Navigator.

☐ Choose the **Blank** icon

☐ Choose **Select**.

☐ Choose the **From Workspace** icon.



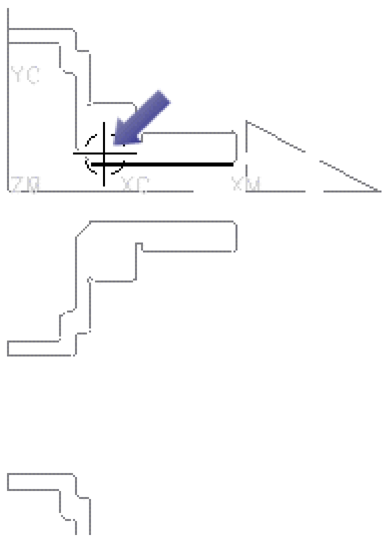
To define the blank geometry for this operation (FINISH_TURN_OD), you will first define a Reference Position in the source, or "from" workspace and then define a Target Position in the current, or "to" workspace. Any two points will work as long as they correspond in each workspace. This is the IPW the third operation will use.

Step 9: Define the Reference Position.

You will first define a reference position on the part at SPINDLE1.

☐ Choose **Select** under Reference Position.

☐ Choose the end point of the horizontal line on the part at SPINDLE1 as illustrated below.



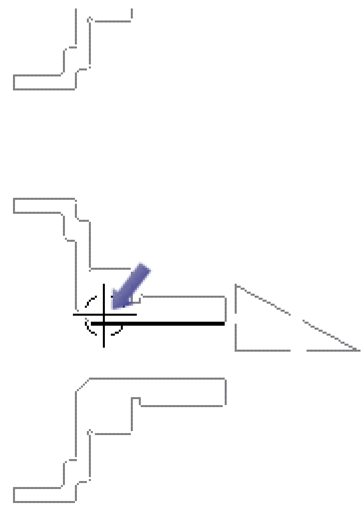
The point is labeled "Ref Pt".

Step 10: Define the Target Position.

You will now define a corresponding target position on the part at SPINDLE2.

☐ Choose **Select** under Target Position.

☐ Choose the end point of the horizontal line on the part at SPINDLE2 as illustrated below.

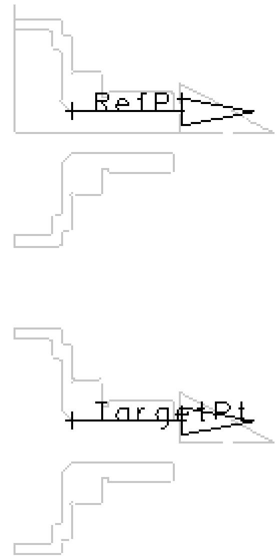


The point is labeled "Target Pt".

Step 11: Display the blank orientation.

The blank geometry must maintain the same relative orientation to the workpiece as the IPW.

☐ Choose **Display Blank**.



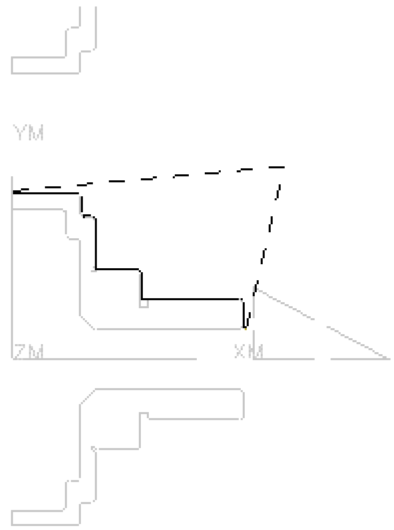
The vectors point in the same direction. In this case, the orientation of the blank geometry at SPINDLE2 is correct because the workpiece is transferred from one spindle to another and is not flipped. If the workpiece were flipped at SPINDLE2, you could flip the blank by choosing Flip Direction.

- ☐ **OK** to accept the blank definition.
- ☐ **OK** to accept the TURN_BND dialog.

Step 12: Generate the tool path for the third operation.

Now that the blank material for the third operation has been defined by referencing the IPW, you can generate the third operation.

- ☐ Highlight the FINISH_TURN_OD operation and **MB3→Generate**.

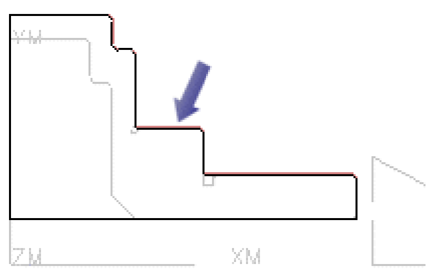


- ☐ **OK** to accept the tool path.
- ☐ Refresh the graphics display.

Step 13: Display the IPW for the third operation.

The IPW generated by the third operation will become the blank material for the fourth operation.

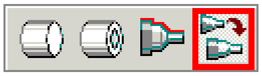
- ☐ In the Operation Navigator, highlight the third operation, FINISH_TURN_OD, and **MB3→Workpiece→Show 2D**.



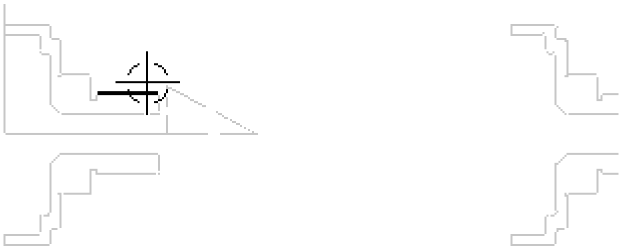
Step 14: Define blank material for the fourth operation.

Blank material for the fourth operation is defined by referencing the IPW generated by the third operation.

- ☐ Double-click the PART3 object in the Operation Navigator.
- ☐ Choose the **Blank** icon
- ☐ Choose **Select**.
- ☐ Choose the **From Workspace** icon.



- ☐ Choose **Select** under Reference Position.
- ☐ Choose the end point of the horizontal line on the part at SPINDLE2 as illustrated below.



The point is labeled "RefPt".

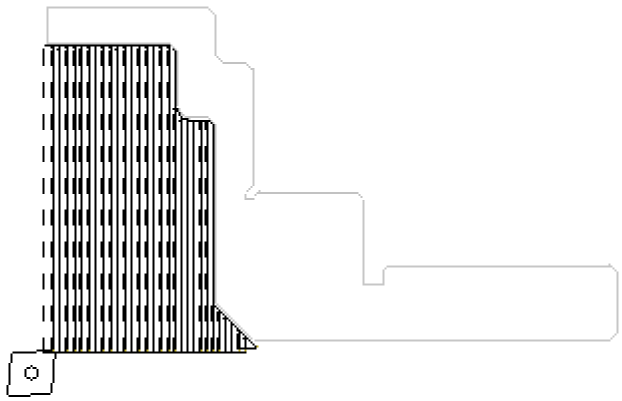
- ☐ Choose **Select** under Target Position.
- ☐ Choose the end point of the corresponding horizontal line on the part at SPINDLE3 as illustrated below.

The point is labeled "TargetPt".

- ☐ **OK** to accept the blank definition.
- ☐ **OK** to accept the TURN_BND dialog.

Step 15: Create the fourth operation.

In the fourth operation, the part is transferred to SPINDLE3 where it is held at the opposite end. You will create a facing operation that uses the same tool as the previous operation, but flipped and rotated so it can cut the back face of the part.



- ☐ Choose the **Create Operation** icon in the toolbar.
- ☐ Be sure the **Type** option is set to **turning**.
- ☐ Choose **FACING** as the subtype.
- ☐ Specify the following parent groups.

  Program: **PROGRAM**
  Use Geometry: **PART3**
  Use Tool: **OD_80_L**
  Use Method: **LATHE_ROUGH**

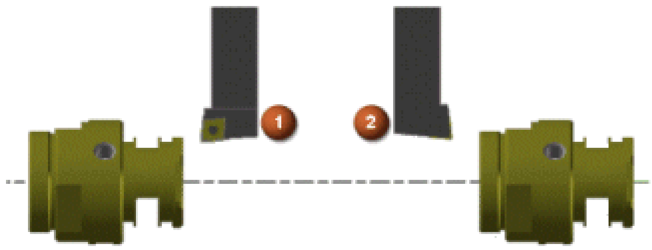
- ☐ **OK** to begin creating the operation.

Step 16: Flip the tool.

This operation uses the same tool as the previous roughing and finishing operations. To machine the back side of the part, the tool must be flipped.

1. As Mounted

2. Flipped



Flipping the tool allows you to machine the opposite end of the part using the same tool when the part is transferred from the main spindle to a sub-spindle. It allows you to access undercuts on the backside of the part and can save you from defining more tools than are mounted on the machine.

When the tool is flipped, the Insert Position is effectively switched between Topside and Underside, allowing the system to maintain the correct spindle rotation direction.

☐ Choose **Machine**.

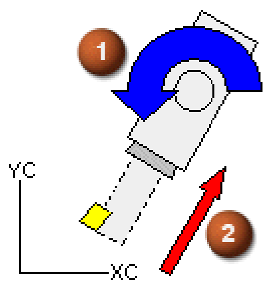
☐ Turn the **Flip tool around holder** option **on**.

Step 17: Reorient the tool.

The Reorient Tool Holder option specifies a fixed forward or backward rotation angle for the tool holder. This allows you to access undercuts and other areas with a single tool that would otherwise be difficult or impossible to access without this control. A vector parallel to the tool holder and pointing away from the insert defines the tool axis. The angle you specify references the tool axis vector and rotates the tool holder relative to the WCS. If the tool is first flipped, you should use a negative angle.

1. Tool Holder Rotation

2. Tool Axis Vector



☐ Turn the **Reorient Tool Holder** option **on**.

☐ Key in **-90** in the Tool Holder Angle field.

Because the tool has been flipped, a negative value must be used to rotate the tool counterclockwise.

☐ **OK** to accept the Machine Control dialog.

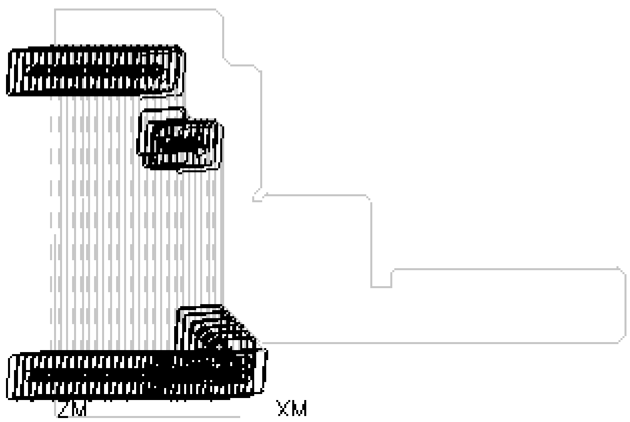
The Level Angle is currently set to 270, causing the tool to cut backward, in a downward direction. This needs to be changed so the tool cuts forward, in an upward direction.

☐ Key in **90** in the Level Angle field.

Step 18: Generating the tool path for the fourth operation.

☐ Set the **Tool Display** option to **2-D**.

☐ **Generate** the tool path.

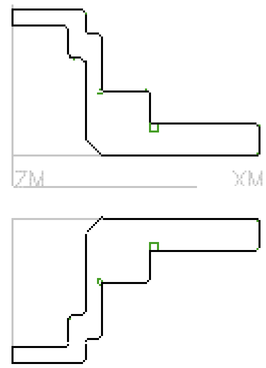


☐ **OK** to complete the operation.

☐ Refresh the graphics display.

Step 19: Displaying the IPW for the Fourth Operation.

☐ In the Operation Navigator, highlight the FACING operation and **MB3→Workpiece→Show 2D**.



This is the IPW for the fourth operation at spindle three.

☐ Close the part file.

Summary

In this lesson, you learned how to create Complex Cross Section curves relative to multiple MCS objects and how to use those curves to define part geometry for parts mounted at each spindle of a multiple spindle machine. You then learned how to create and reference the in-process workpiece as it is transferred from one spindle to another.

In this lesson, you:

- Created Complex Cross Section curves relative to two MCS objects.
- Defined part geometry for parts mounted at each spindle of a two spindle machine.
- Generated a roughing tool path at one spindle.
- Generated a finishing tool path at the other spindle.
- Created in-process workpieces
- Used the current in-process workpiece to defined the blank geometry for the next operation
- Flip and reorient tool to machine back side of part

Lesson

15 Mill-Turn

Purpose

This lesson will teach you how to create a program that contains milling and turning operations. You will also learn how to apply the turning IPW to subsequent milling operations.

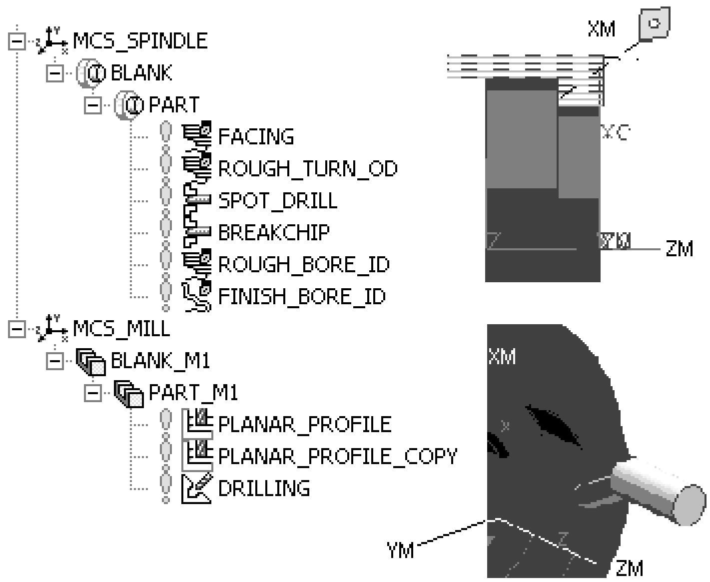
Objective

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

- Define the MCS for milling operations.
- Save the turning IPW as a separate part file.
- Add the turning IPW to the assembly.
- Define the milling blank geometry using the turning IPW.
- Add milling and drilling operations to the program.
- Postprocess the program.

Overview of Creating a Mill-Turn Program

A mill-turn program requires the definition of at least two machine coordinate systems; one for the milling program and one for the turning program. This is required since turning programs and milling programs utilize machine coordinate systems differently. Turning operations typically generate the tool path in the ZM-XM plane with the ZM axis along the spindle centerline, while milling operations typically generate the tool path in the XM-YM plane.



The turning IPW (in-process workpiece) can be used as blank geometry for the subsequent milling operations by saving it as a separate part file and then importing it back into the assembly.

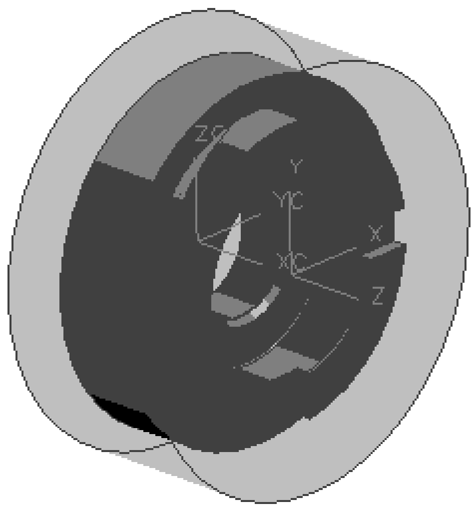
The process defined in this lesson describes how to set up the machine coordinate systems and how to utilize the turning IPW in the subsequent milling operations to create a simple mill-turn program. It also describes how to postprocess the mill-turn program.

Activity: Creating a Mill-Turn Program

In this activity, you are going to create a program that first turns and then mills a part. The turning operations have been provided. You will begin by defining the MCS and geometry groups for the milling operations. You will then save the turning IPW as a separate part file. You will add the turning IPW to the assembly and use it to define the blank geometry for the milling operations. You will then create the milling and drilling operations and generate the tool paths.

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

☐ Open the part file **tmp_mill_turn_assy.prt**.



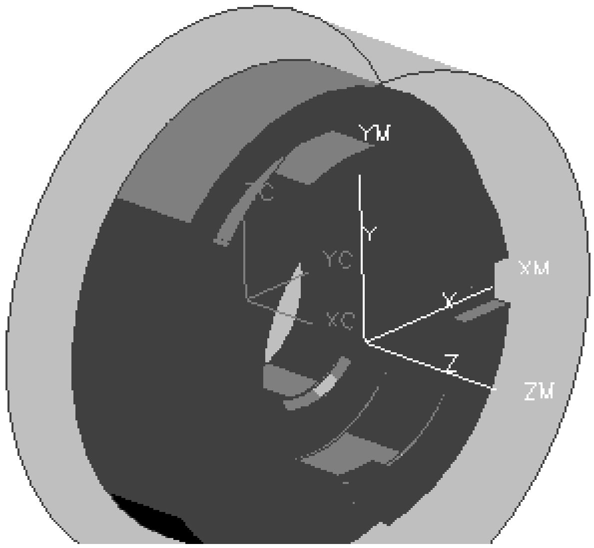
☐ Save the part file as *****_mill_turn_assy.prt**, where *** represents your initials.

☐ Enter the **Manufacturing** application.

☐ Enter the **Assemblies** application.

Step 2: Examine the MCS_SPINDLE orientation.

☐ In the Geometry view of the Operation Navigator, choose MCS_SPINDLE and **MB3→Object→Display**.



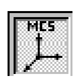
You will define the milling MCS at the same location and orientation as the turning MCS.

Step 3: Create a MCS_MILL object.

Mill operations must use a MCS separate from the one defined for the turning operations.

☐ Choose the **Create Geometry** icon. 

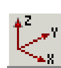
☐ Set the Type option to **mill_planar**.

☐ Choose the **MCS** icon. 

☐ Set the Parent Group option to **Geometry**.

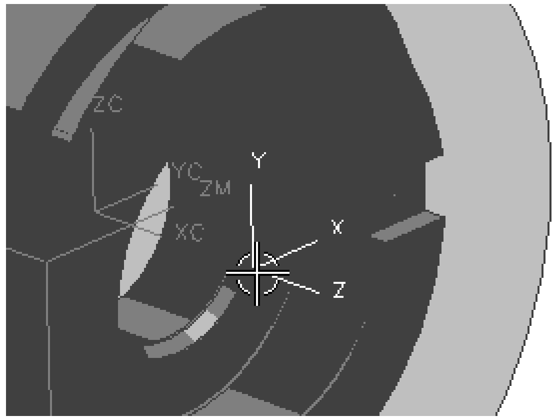
☐ Key in **MCS_MILL** in the Name field.

☐ Choose **OK** to begin creating the MCS_MILL object.

☐ Choose the **Constructor** icon. 

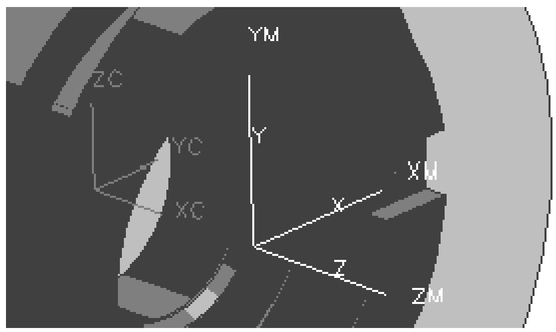
☐ Choose the **Offset from CSYS** icon. 

☐ Select the existing coordinate system as illustrated below.



This coordinate system defines both the location and orientation of the MCS_SPINDLE.

☐ Choose **OK** twice to create the MCS_MILL coordinate system.



The MCS_SPINDLE and MCS_MILL coordinate systems are at the same location and have the same orientation.

Step 4: Create Blank and Part geometry groups.

You will create geometry groups for milling of the part and blank geometry. Later, you will define the geometry within these groups.

☐ Choose the **Create Geometry** icon. 

☐ Choose the **WORKPIECE** icon. 

☐ Set the Parent Group option to **MCS_MILL**.

☐ Key in **BLANK_M1** in the Name field.

☐ Choose **OK** to begin creating the BLANK_M1 object.

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

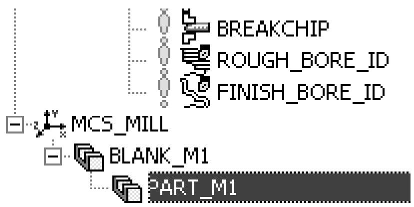
☐ Choose the **Create Geometry** icon.

☐ Set the Parent Group option to **BLANK_M1**.

☐ Key in **PART_M1** in the Name field.

☐ Choose **OK** to begin creating the PART_M1 object.

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



Step 5: Move the WCS to the MCS_SPINDLE.

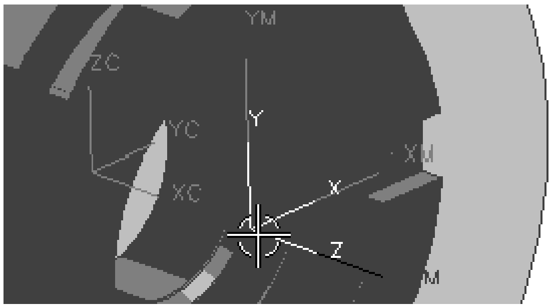
You will move the WCS to the MCS_SPINDLE location. The location of the WCS is important since the IPW references the WCS when imported into the assembly.

The orientation of the WCS is such that XC is aligned with the spindle centerline and YC is the cross-axis.

☐ Choose **WCS→Orient** from the toolbar.

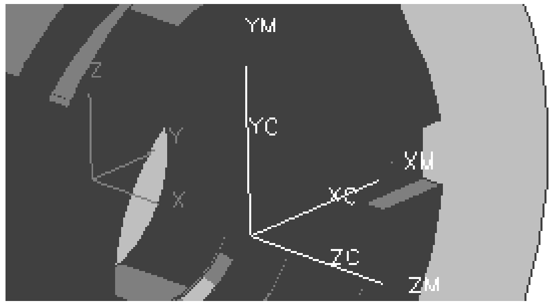
☐ Choose the **Offset from CSYS** icon.

☐ Choose the coordinate system as illustrated below.



☐ Choose **OK** to complete the move.

Notice that the WCS has changed orientation. ZC is aligned with the spindle and XC is the cross-axis.



Step 6: Rotate the WCS.

The WCS must be orientated so that XC is aligned with the spindle and YC is the cross-axis as it was originally.

☐ Choose **WCS→Rotate** from the toolbar.

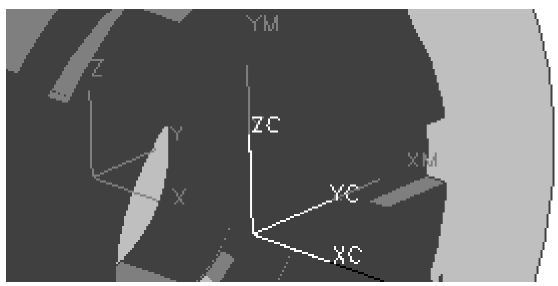
☐ Turn the **-YC Axis: XC→ZC** option **on**.

☐ Be sure **90.000** appears in the Angle field.

☐ Choose **Apply**.

☐ Turn the **-XC Axis: ZC→YC** option **on**.

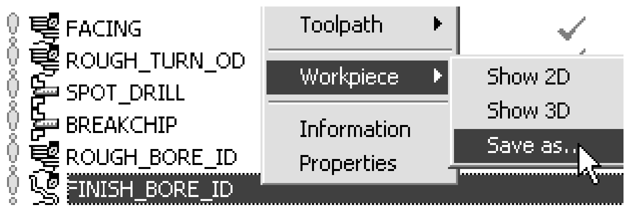
☐ Choose **OK**.



Step 7: Exporting the turning IPW.

You will highlight the last turning operation and save the IPW for the turning subprogram as a separate part. Later, you will import this part back into the assembly and use it as blank geometry for the milling operations.

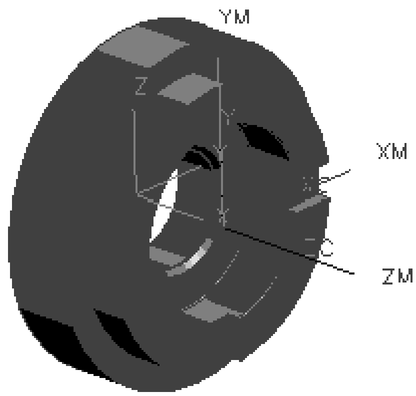
☐ Highlight the last turning operation (FINISH_BORE_ID) and **MB3→Workpiece→Save as**.



☐ Save the part file *****_ipw.prt**, where *** represents your initials.

☐ Choose the check mark next to **tmp_swug_mill_turn_blk** in the Assembly Navigator to remove the blank material from the display.

Only the part should be displayed in the assembly.

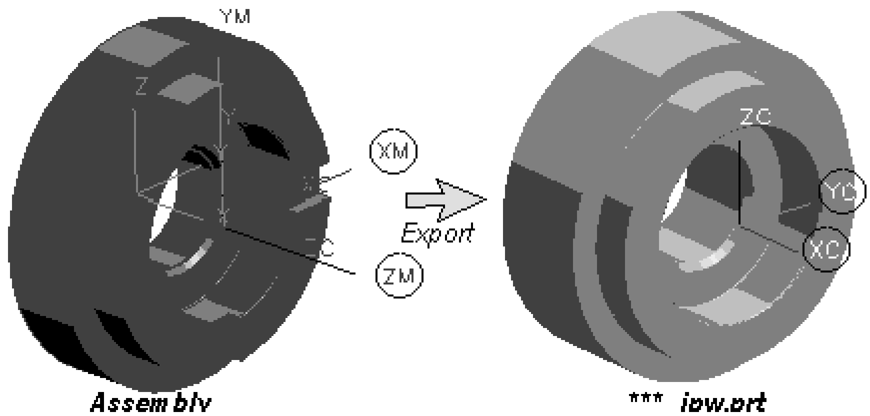


Step 8: Look at the IPW part file.

You will open the ipw part file and observe the orientation of the WCS.

☐ Open the part file *****_ipw.prt** and view it in TFR_ISO.

Notice that XC in the exported IPW is aligned with the centerline of the spindle and YC is the cross-axis. When you export the IPW, the system assumes your turning work plane (ZM-XM) is an X-Y work plane.

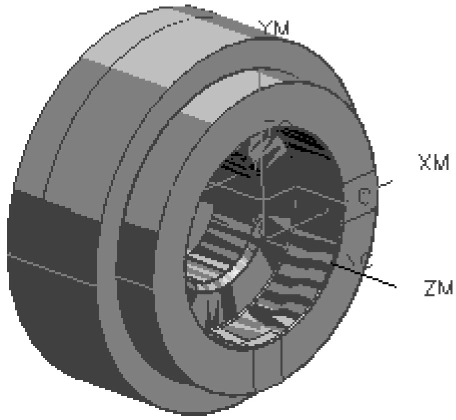


Step 9: Add the turning IPW to the assembly.

You will add the turning IPW part to the assembly and then use it as the blank material for the milling operations. The part will be imported into the assembly relative to the WCS.

- ☐ Choose **Window→***_mill_turn_assy.prt** in the menu bar to display the assembly.
- ☐ Choose the **Add Existing Component** icon. 
- ☐ Choose ***_ipw.prt** in the listing window.
- ☐ Choose **OK** to accept the selected part.
- ☐ Set the Positioning option to **Reposition**.
- ☐ Choose **OK** to accept it.
- ☐ Choose **Reset** to zero out the Base Point values.
- ☐ Be sure the **WCS** option is turned **on**.
- ☐ Choose **OK** to add the component to the assembly.
- ☐ **Cancel** the Reposition Component dialog to complete the assembly.

The IPW (displayed in green) is superimposed on the part.



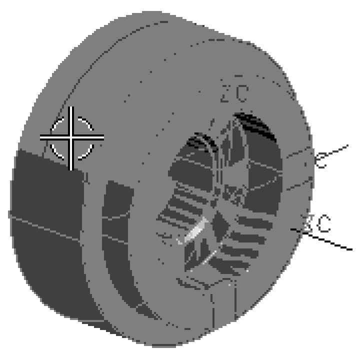
Step 10: Select the IPW as the blank geometry.

You will select the IPW as the blank geometry for the milling operations.

☐ Double-click on the BLANK_M1 icon in the Operation Navigator.

☐ In the Workpiece dialog, choose the **Blank** icon and **Select**.

☐ Select the turning IPW (displayed in green) as the blank geometry and then choose **OK**.



☐ Choose **OK** to complete the editing of the BLANK_M1 object.

☐ Choose the check mark next to *****ipw_.prt** in the Assembly Navigator to remove the blank component from the display.

Only the part should be displayed in the assembly.

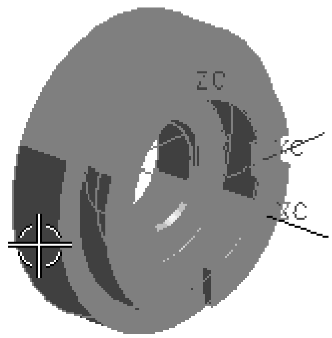
Step 11: Select the part geometry.

You will define the part geometry by selecting the solid body. Although the body is not used by the planar mill operations, it is necessary for gouge checking.

☐ Double-click on the PART_M1 icon.

☐ In the Workpiece dialog, choose the **Part** icon and **Select**.

☐ Select the part and choose **OK**.



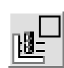
☐ Choose **OK** to complete the editing of the PART_M1 object.

Step 12: Begin creating the first milling operation.

You will create a Planar Mill operation that machines one of the slots on the front face of the part.

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

☐ Set the **Type** option to **mill_planar**.

☐ Choose the **PLANAR_PROFILE** icon. 

☐ Set the following Parent Groups:

- Program = **TAPE_12345_A**
- Use Geometry = **PART_M1**
- Use Tool = **EM_1.25_.03**
- Use Method = **MILL_FINISH**

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

Step 13: Define the part boundary.

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

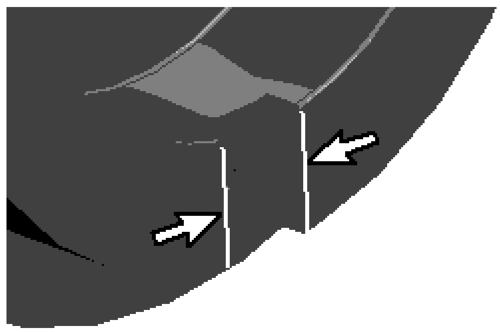
☐ Set the Mode option to **Curves/Edges**.

☐ Set the Type option to **Open**.

☐ Set the Plane option to **User-Defined**.

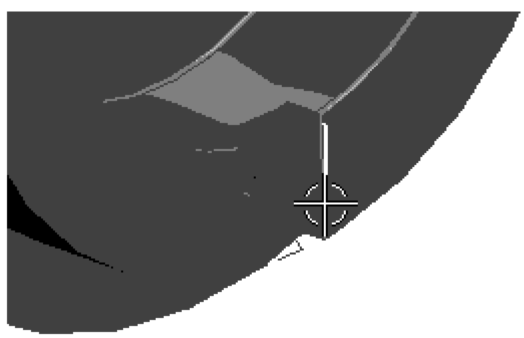
☐ Choose **Two Lines**.

☐ Select the two lines illustrated below.



☐ Set the Material Side option to **Right** and be sure the Tool Position option is set to **Tanto**.

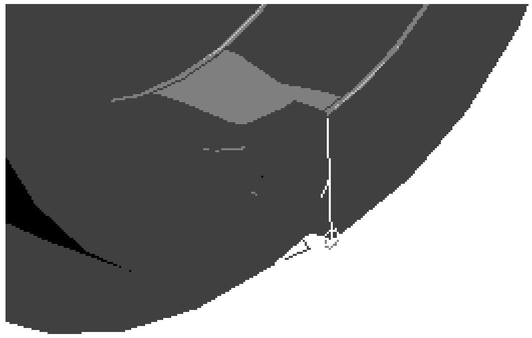
☐ Choose the line for the boundary toward the bottom as illustrated below.



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

☐ Choose **OK** to complete the boundary geometry.

☐ Choose **Display**.

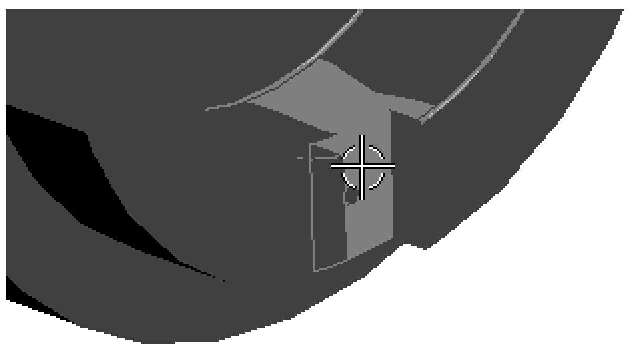


Step 14: Define the remaining operation parameters.

☐ Choose the **Floor** icon  in the PLANAR_PROFILE dialog and **Select**.

☐ Set the Filter option to **Face**.

☐ Select the face as illustrated below and **OK** to define it as the floor.

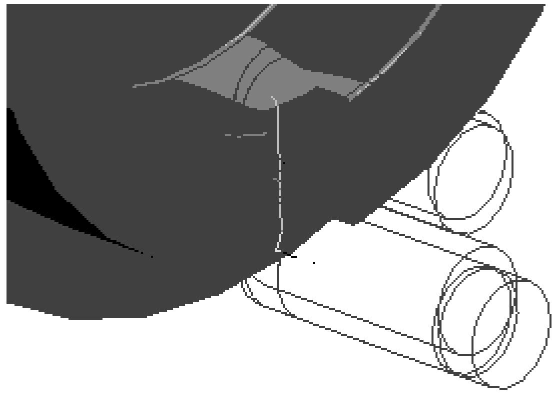


☐ Be sure the Cut Depth option is set to **Floor Only**.

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

☐ **Generate** the tool path.

☐ Choose **OK** to complete the tool path.



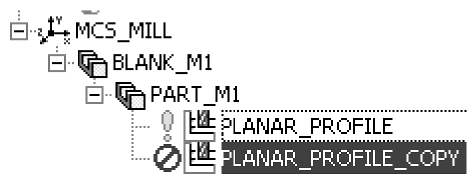
☐ Choose **OK** to complete the operation.

Step 15: Create the second milling operation.

You will create a Planar Mill operation that machines the second slot. You will do this by copying the first operation and defining a new part boundary.

☐ Highlight the PLANAR_PROFILE operation in the Operation Navigator and **MB3→Copy**.

☐ With the PLANAR_PROFILE operation still highlighted, use **MB3→Paste**.



Step 16: Redefine the part boundary.

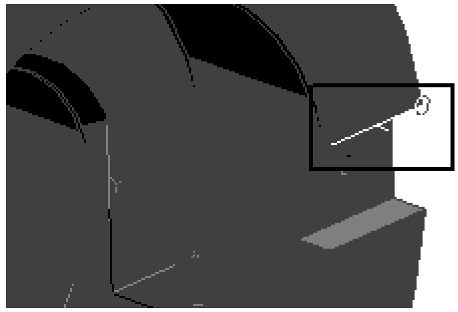
You will redefine the part boundary so that the operation machines the other slot.

☐ Double-click on the **PLANAR_PROFILE_COPY** icon in the Operation Navigator to edit the operation.

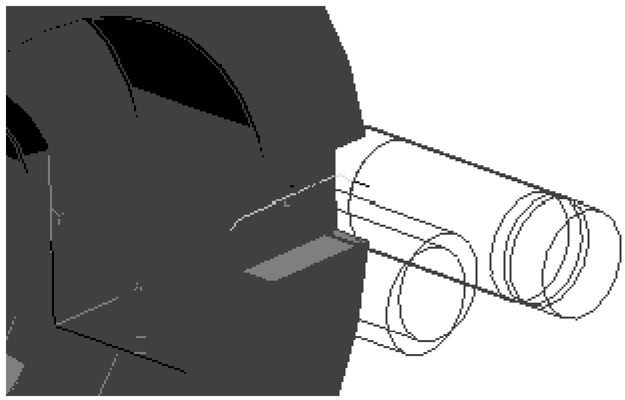
☐ Choose the **Part** icon and **Reselect**.

☐ Choose **OK** to the warning.

☐ Create the boundary illustrated below as you did before.



☐ **Generate** the tool path.



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

Step 17: Create the drilling operation.

You will create an operation that drills the holes in the slots.

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

☐ Set the Type option to **drill**.

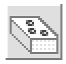
☐ Choose the **DRILLING** icon  from the Create Operation dialog.

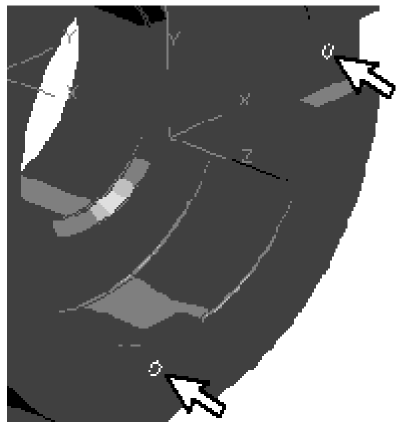
☐ Set the following Parent Groups:

- Program = **TAPE_12345_A**
- Use Geometry = **PART_M1**
- Use Tool = **DRL_.250**
- Use Method = **DRILLING**

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

Step 18: Define the drilling geometry.

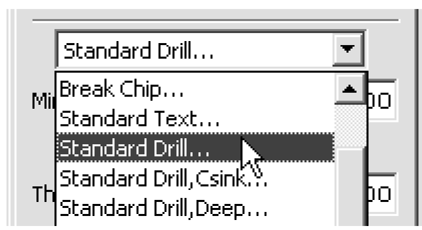
- ☐ Choose the **Holes** icon  and **Select**.
- ☐ Choose **Select** in the Point dialog.
- ☐ Select the two holes.



- ☐ Choose **OK** to accept them.
- ☐ Choose **OK** to the Point dialog.

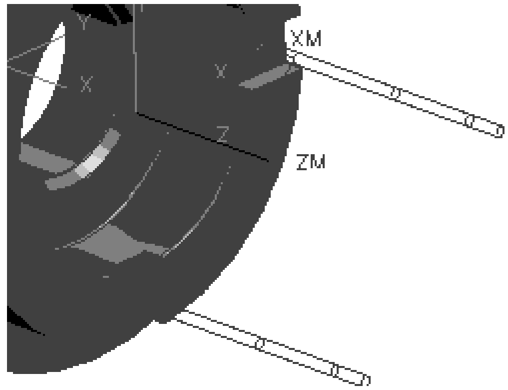
Step 19: Define the remaining operation parameters.

- ☐ Choose **Standard Drill**.



- ☐ Choose **OK** to accept 1 as the Number of Sets.
- ☐ Choose **Rtrcto - None**.
- ☐ Choose **Distance**.
- ☐ Key in **1.000** in the Rtrcto field and **OK** to accept it.
- ☐ Choose **OK** in the Cycle Parameters dialog.

☐ **Generate** the tool path.



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

☐ Save the part file.

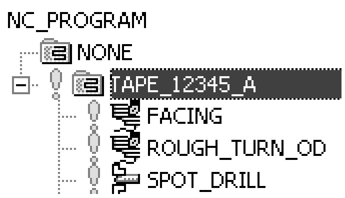
Activity: Postprocessing

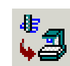
In this activity, you are going to postprocess the mill-turn program and view the output.

Step 1: Postprocess the mill-turn program.

☐ Display the Program Order View of the Operation Navigator.

☐ Highlight the **TAPE_12345_A** program.



☐ Choose the **UG/Post Postprocess** icon in the toolbar. 

☐ Choose **MILLTURN** in the Available Machines list.

☐ Specify the directory and file name. The system will save the file as *****_mill_turn_assy.prt** in the current directory by default.

☐ Choose **OK** to create the file.

Step 2: View the output.

☐ Scroll through the Information window to where the milling operations begin.

At this point, a milling tool is used and the spindle functions as a rotating table top to accommodate the milling operations.

```
:2100 T27 M06
N2110 T08
N2120 G12.1
N2130 G94 G0 G90 X-5.1618 Y0.0 C88.89 S0 M03
N2140 G43 Z.1 H27
N2150 Z-.4
N2160 G1 Z-.5 F10. M08
N2170 X-5.1219 C89.359
N2180 X-5.0731 C89.712
```

☐ Dismiss the Information window.

☐ Save the part file.

Summary

A mill-turn program allows you to perform both milling and turning operations in a single program.

In this lesson you:

- Defined the MCS for milling operations.
- Saved the turning IPW as a separate part file.
- Added the turning IPW to the assembly.
- Defined the milling blank geometry using the turning IPW.
- Added milling and drilling operations to the program.
- Postprocessed the program.

Lesson

16 Vertical Turret Lathe

Purpose

This lesson will teach you how to create a Vertical Turret Lathe (VTL) program.

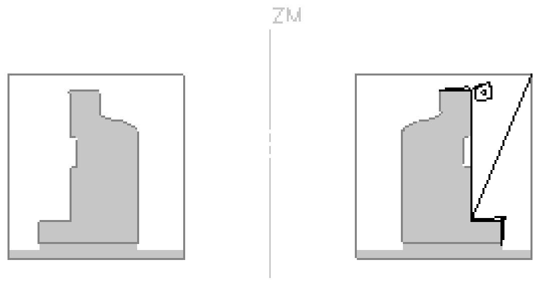
Objective

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

- Orient the view of the part for a VTL.
- Orient the WCS to facilitate the input of angle values.
- Orient the MCS according to the requirements of your postprocessor.
- Specify the Lathe Work Plane.
- Create facing, roughing, grooving, and finishing operations.
- Postprocess the program.

Overview of Creating VTL Program

A Vertical Turret Lathe (VTL) program requires the MCS to be oriented according to the requirements of the postprocessor. The vertical centerline of the spindle must be defined by either the ZM axis or the XM axis. The Lathe Work Plane must then be specified accordingly. For convenience, the part should be viewed in a vertical orientation and the WCS should be rotated so that YC is pointing upward and XC is pointing to the right. This will allow angles to be referenced from the conventional three o'clock position.



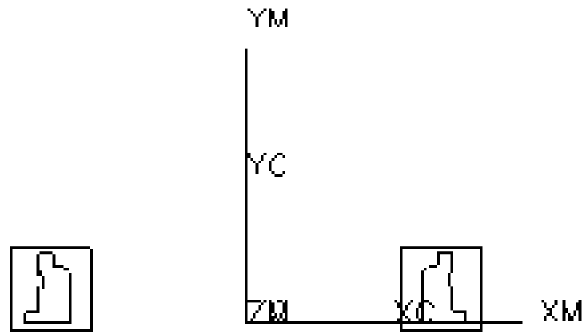
Activity: Examining a VTL Setup and Program

In this activity, you are going to examine the finished setup and program.

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

☐ Open the part file **tmp_vtl_2.prt**.

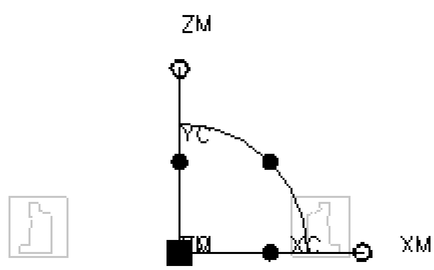
☐ Enter the **Manufacturing** application.



Step 2: View the MCS.

☐ In the Geometry View of the Operation Navigator, double-click the MCS_SPINDLE object.

The vertical centerline of the spindle must be defined by either the ZM or XM axis, depending on your postprocessor. In this case, it is defined by the ZM axis.

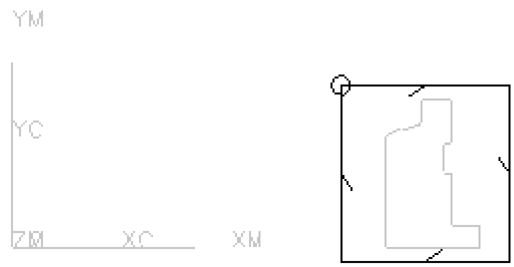


☐ **Cancel** the TURN_ORIENT dialog.

Step 3: View the Blank Geometry. The blank geometry has been defined just once in a parent group so that all of the operations can reference it.

☐ Double-click the BLANK object in the Operation Navigator.

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

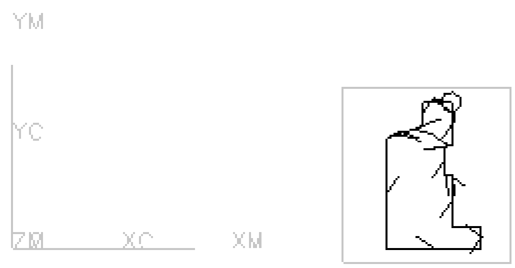


☐ **Cancel** the TURN_BND dialog.

Step 4: View the Part Geometry. The Part geometry has been defined just once in a parent group so that all of the operations can reference it.

☐ Double-click the PART object in the Operation Navigator.

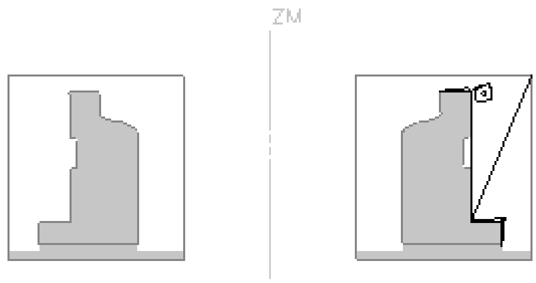
☐ Choose **Display**.



☐ **Cancel** the PART dialog.

Step 5: Replay the program.

☐ In the Program Order View of the Operation Navigator, highlight the T563489A object and **MB3→Replay**.



This is the program you will create.

☐ **Cancel** the TURN_ORIENT dialog.

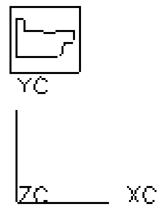
☐ Close the part file.

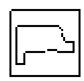
Activity: Defining a VTL Setup

In this activity, you are going to define the spindle setup for a VTL program. You will begin by rotating the view so that the centerline displays vertically. You will then rotate the WCS so that YC is pointing upward and XC is pointing to the right. This will allow angles to be referenced from the conventional three o'clock position. Lastly, you will orient the MCS according to the requirements of your postprocessor and then specify the Lathe Work Plane that corresponds to that orientation.

Step 1: Open the part file.

- ☐ Open the part file **tmp_vtl.prt**.

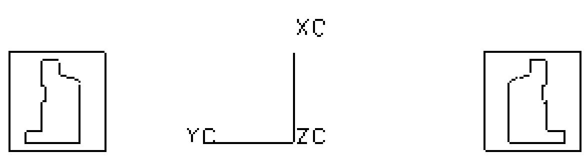




- ☐ Save the part file as *****_vtl.prt**, where *** represents your initials.

Step 2: Rotate the part on the screen so that the centerline displays vertically.

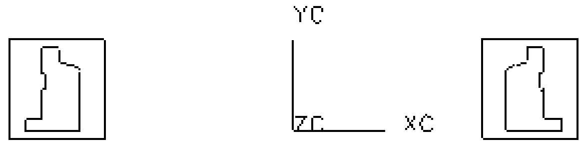
- ☐ Choose **View→Operation→Rotate** in the menu bar.
- ☐ Choose the **Z-Axis** icon.
- ☐ Be sure **90** has been entered in the Angle Increment field and **OK** the Rotate View dialog.



Step 3: Rotate the WCS so that the positive XC axis is pointing to the right.

- ☐ Choose **WCS→Rotate** in the menu bar.
- ☐ Turn the **–ZC Axis: YC→XC** option **on**.

☐ Be sure **90** has been entered in the Angle field and **OK**.



This WCS orientation establishes the zero degree angle reference at the conventional three o'clock position. This will make entering angle values much less confusing later on.

☐ Choose **Format→Layout→Save As** to save a new layout.

Step 4: Orient the MCS according to what your postprocessor requires. The vertical centerline of the spindle must be defined by either the ZM axis or the XM axis.

☐ Enter the **Manufacturing** application.

☐ In the Geometry View of the Operation Navigator, double-click the MCS_SPINDLE object.

☐ Choose the **Rotate** icon in the MCS section of the TURN_ORIENT dialog.

☐ Turn the **–ZM Axis: YM→XM** option **on**.

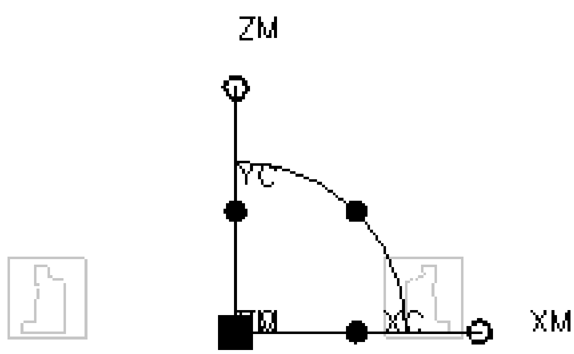
☐ Be sure **90** has been entered in the Angle field and **Apply**.

☐ Turn the **–XM Axis: ZM→YM** option **on**.

☐ Be sure **90** has been entered in the Angle field and **Apply**.

☐ **Cancel** once.

The MCS should appear as illustrated below, with ZM representing the centerline of the spindle.



Step 5: Specify the Lathe Work Plane that corresponds to the orientation of the MCS.

☐ In the TURN_ORIENT dialog under Lathe Work Plane, turn the **use ZM–XM** option **on**.

This is the plane in which your tool paths will be created

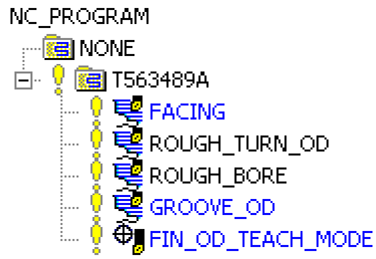
☐ **OK** the TURN_ORIENT dialog.

☐ Save and close the part file.

You have completed the VTL spindle setup. Next, you will create the operations.

Activity: Creating the Operations

In this activity, you are going to create a program consisting of facing, od roughing, boring, grooving, and finishing operations.

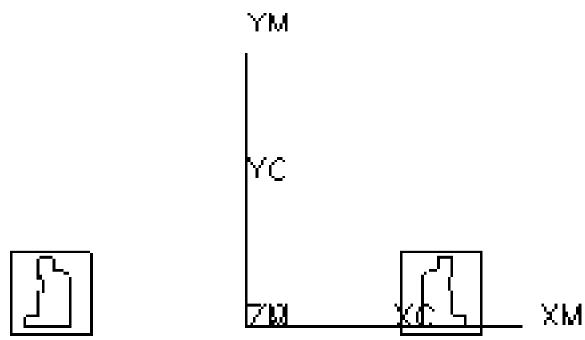


Step 1: Open the part file and enter the Manufacturing application.

☐ Open the part file **tmp_vtl_1.prt**.

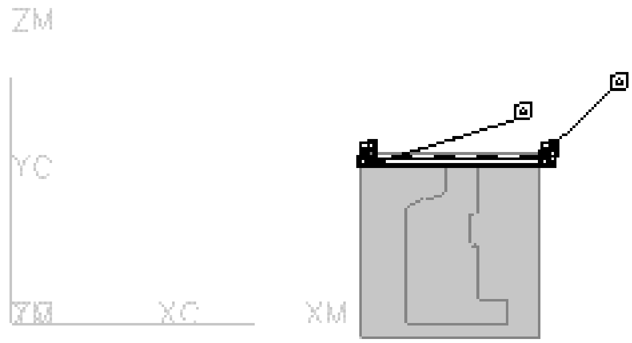
This part file contains the VTL spindle setup you just defined and all the tools and geometry necessary to create the operations.

☐ Enter the **Manufacturing** application.



☐ Save the part file as *****_vtl_1.prt**, where *** represents your initials.

Step 2: Create the FACING operation.



- ☐ Choose the **Create Operation** icon in the toolbar.
- ☐ Be sure the **Type** option is set to **turning**.
- ☐ Choose **FACING** as the subtype.
- ☐ Specify the following parent groups.

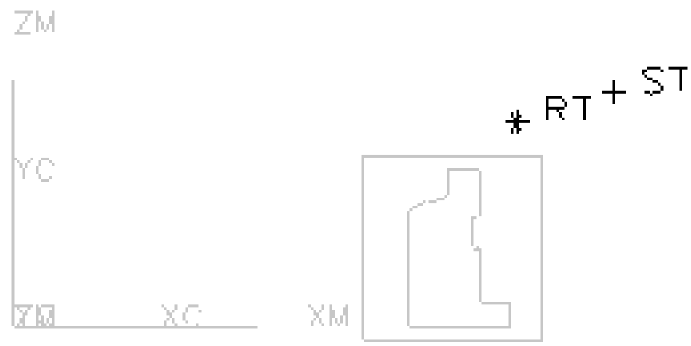
  Program: **T563489A**
  Use Geometry: **PART**
  Use Tool: **OD_80_DIA_L_RGH**
  Use Method: **LATHE_ROUGH**

- ☐ **OK** to begin creating the operation.

Step 3: Specify the Operation Parameters.

- ☐ Be sure **180.000** appears in the Level Angle field.
- ☐ Set the **Cut Depth** option to **Constant**.
- ☐ Key in **0.750** in the Depth field.
- ☐ Turn the **Cleanup** option **off**.
- ☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below.

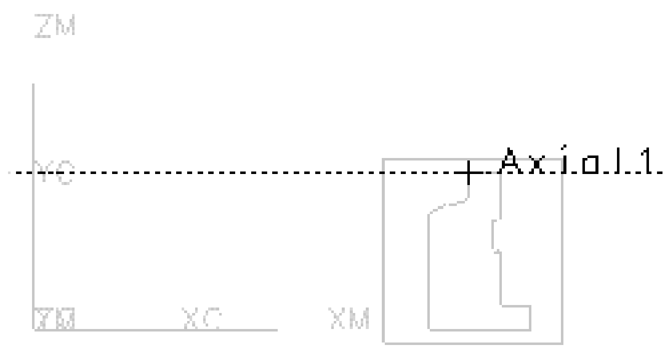


☐ **OK** to accept the points.

☐ Choose **Containment**.

☐ Turn the Axial 1 **Trim** option **on**.

☐ Choose **Axial 1** and specify a Trim Plane on the top face of the part as illustrated below.

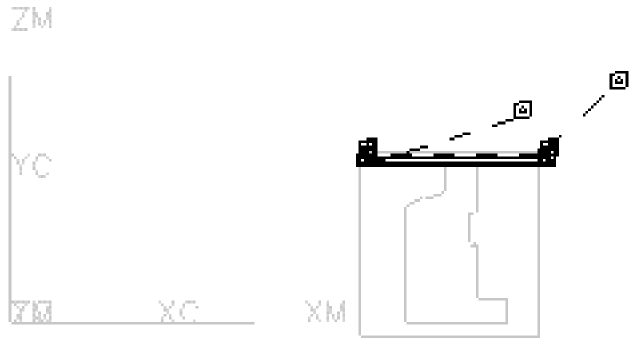


☐ **OK** to accept the Trim Plane.

Step 4: Generate the Tool Path

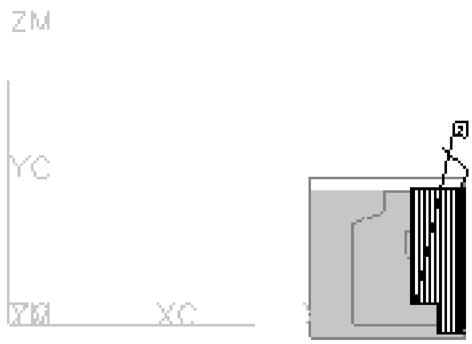
☐ Choose **Edit Display** and set the **Tool Display** option to **2–D**.

☐ Generate the tool path.



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

Step 5: Create the ROUGH_TURN_OD operation.



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

☐ Choose **ROUGH_TURN_OD** as the subtype.

☐ Specify the following parent groups.

Program: **T563489A**
Use Geometry: **PART**
Use Tool: **OD_80_DIA_L_RGH**
Use Method: **LATHE_ROUGH**

☐ **OK** to begin creating the operation.

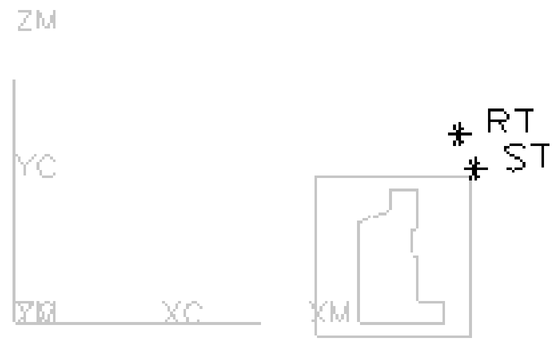
Step 6: Specify the Operation Parameters.

☐ Set the **Cut Depth** option to **Constant**.

☐ Key in **0.750** in the Depth field.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below.

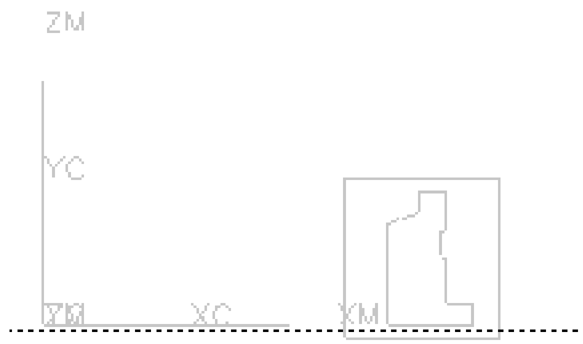


☐ **OK** to accept the points.

☐ Choose **Containment**.

☐ Turn the Axial 1 **Trim** option **on**.

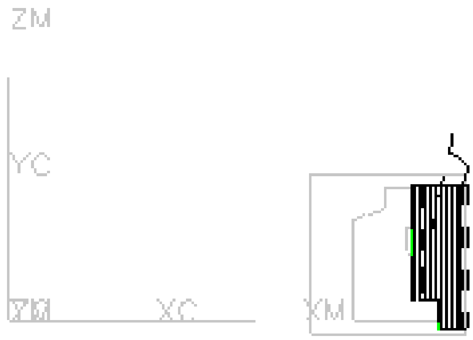
☐ Key in **–1.000** in the Axial 1 field to specify the trim plane illustrated below.



☐ **OK** to accept the Trim Plane.

Step 7: Generate the Tool Path

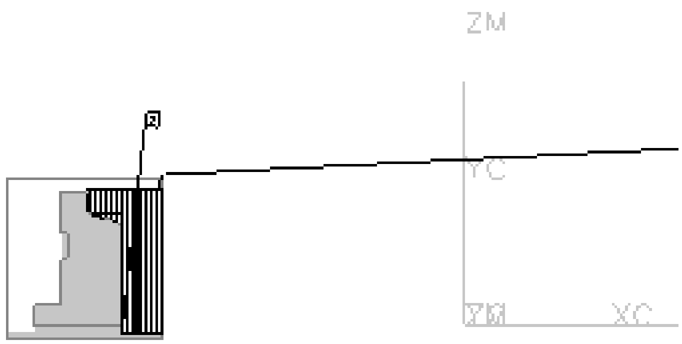
☐ Choose **Generate**.



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

Step 8: Create the ROUGH_BORE operation.

We will use the same turning tool to rough bore as we used to rough turn. The processor will automatically reverse the spindle when the tool crosses the centerline.



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

☐ Choose **ROUGH_TURN_OD** as the subtype.

☐ Specify the following parent groups.

Program: **T563489A**
Use Geometry: **PART**
Use Tool: **OD_80_DIA_L_RGH**
Use Method: **LATHE_ROUGH**

☐ Key in **ROUGH_BORE** in the Name field.

☐ **OK** to begin creating the operation.

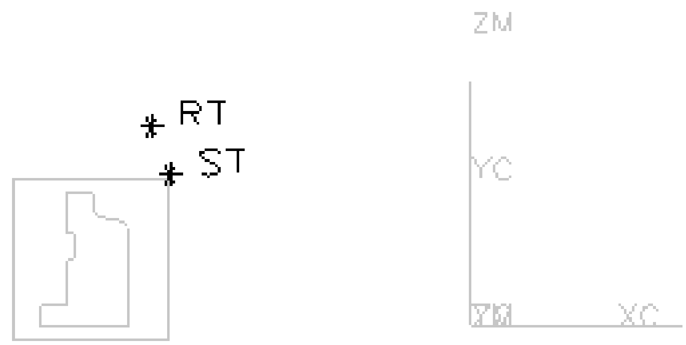
Step 9: Specify the Operation Parameters.

☐ Set the **Cut Depth** option to **Constant**.

☐ Key in **0.750** in the Depth field.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below.

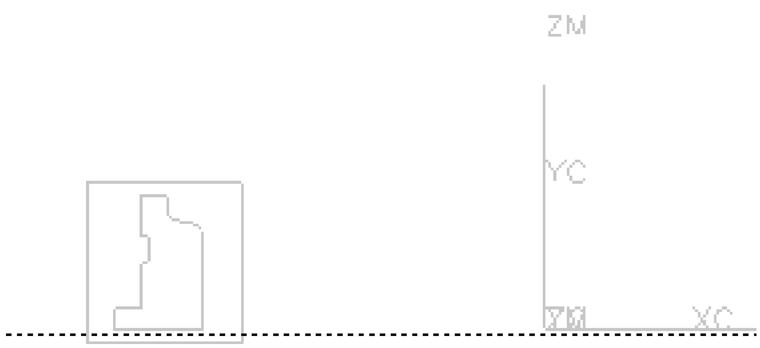


☐ **OK** to accept the points.

☐ Choose **Containment**.

☐ Turn the Axial 1 **Trim** option **on**.

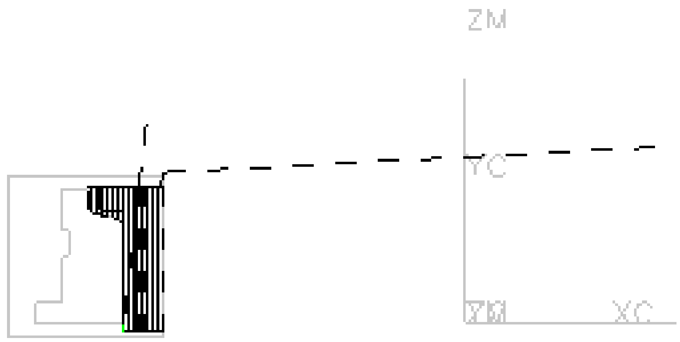
☐ Key in **–1.000** in the Axial 1 field to specify the trim plane illustrated below.



☐ **OK** to accept the Trim Plane.

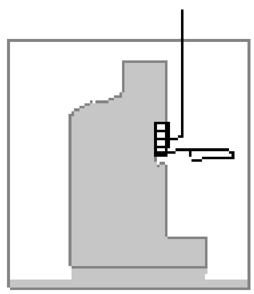
Step 10: Generate the Tool Path

☐ Choose **Generate**.



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

Step 11: Create the GROOVE_OD operation.



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

☐ Choose **GROOVE_OD** as the subtype.

☐ Specify the following parent groups.

Program: **T563489A**
Use Geometry: **PART**
Use Tool: **OD_GROOVE_L**
Use Method: **LATHE_ROUGH**

☐ **OK** to begin creating the operation.

Step 12: Specify the Operation Parameters.

☐ Choose the **Plunge Alternate** icon.

☐ Key in **90.000** in the Step Angle field.

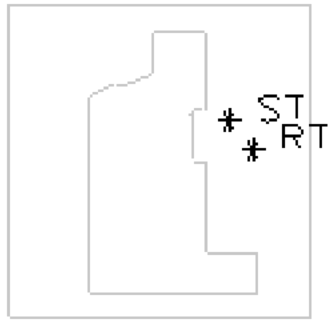
☐ Set the **Stepover** option to **Constant**.

☐ Be sure the Step field says **75.000 %**.

☐ Turn the **Additional Profiling** option **on**.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below. Be sure these points lie within the width of the groove.



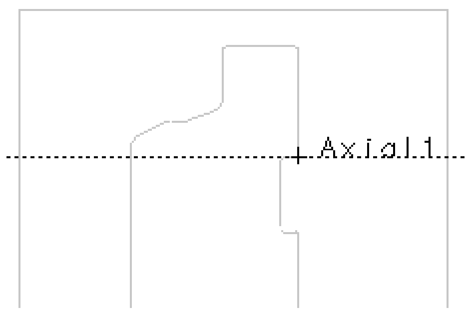
☐ Set the **Motion to Start Point** option to **Radial→Axial**.

☐ **OK** to accept the points.

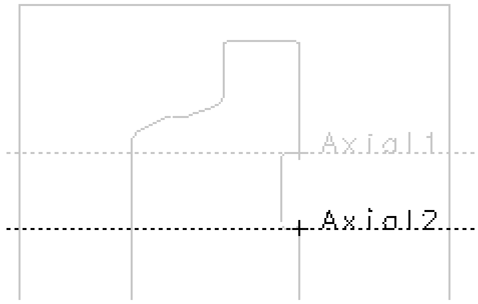
☐ Choose **Containment**.

☐ Turn the Axial 1 and Axial 2 **Trim** options **on**.

☐ Choose **Axial 1** and select an inferred point on the side of the groove.



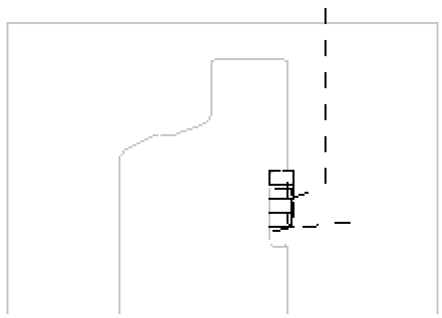
☐ Choose **Axial 2** and select an inferred point on the other side of the groove.



☐ **OK** to accept the Trim Planes.

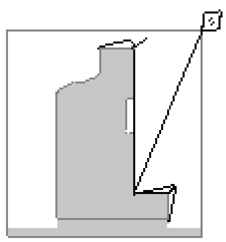
Step 13: Generate the Tool Path

☐ Choose **Generate**.



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

Step 14: Create the FIN_OD_TEACH_MODE operation.



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

☐ Choose **TEACH_MODE** as the subtype.

☐ Specify the following parent groups.

Program: **T563489A**
Use Geometry: **PART**
Use Tool: **FIN_OD_80DEG_L**
Use Method: **LATHE_FINISH**

☐ Key in **FIN_OD_TEACH_MODE** in the Name field.

☐ **OK** to begin creating the operation.

Step 15: Create an Engage Setting suboperation.

☐ Choose the **Engage Setting** icon.

☐ Be sure the **Auto Circular** icon is chosen.

☐ Key in **0.200** in the Extend Distance field.

☐ **OK** to accept the Engage Setting.

Step 16: Create a Retract Setting suboperation.

☐ Choose the **Retract Setting** icon.

☐ Be sure the **Auto Circular** icon is chosen.

☐ Key in **0.200** in the Extend Distance field.

☐ **OK** to accept the Retract Setting.

Step 17: Create a Follow Curve Motion suboperation.

☐ Choose the **Follow Curve Motion** icon.

☐ Be sure **Previous Drive Curve** is turned **on**.

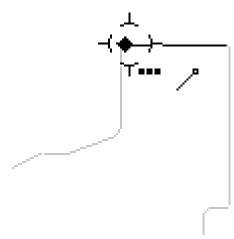
This will cause this suboperation to use the part geometry.

☐ Choose the **Start/Stop** tab.

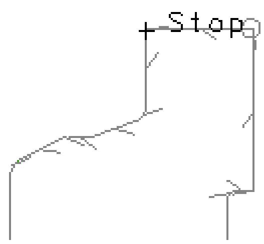
☐ Be sure the **Starting Position Method** is set to **Drive Curve Start**.

☐ Set the **Stopping Position Method** to **Point** and choose **Select**.

☐ Select the end of the horizontal line as illustrated below.



☐ Choose the **Main** tab and **Display All Geometry** to verify that the drive geometry is correct.

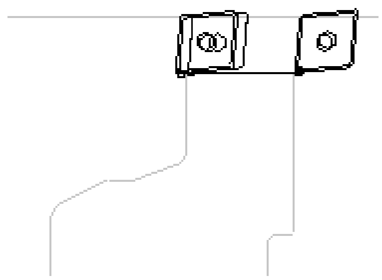


☐ **OK** to accept the Follow Curve Motion.

Step 18: Generate the Tool Path

☐ Choose **Edit Display** and set the **Tool Display** option to **2–D**.

☐ Choose **Generate**.



Step 19: Create a Rapid Motion suboperation.

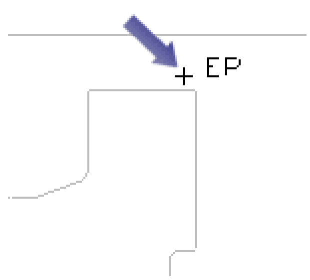
☐ Choose the **Linear Rapid** icon.

☐ Be sure the **Move Radial and Axial** icon is chosen.

☐ With the **Point** option turned **on**, choose **Select**.

☐ Choose the **Cursor Location** icon.

☐ Indicate an end point at the approximate screen position illustrated below.



☐ **OK** to accept the Rapid Motion.

Step 20: Create a Retract Setting suboperation.

☐ Choose the **Retract Setting** icon.

☐ Choose the **Vector** icon.

☐ Key in **0.060** in both the Vector (X) and Vector (Y) fields.

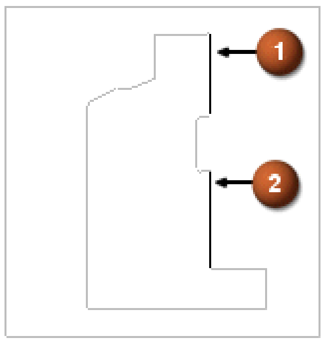
☐ **OK** to accept the Retract Setting.

Step 21: Create a Follow Curve Motion suboperation.

☐ Choose the **Follow Curve Motion** icon.

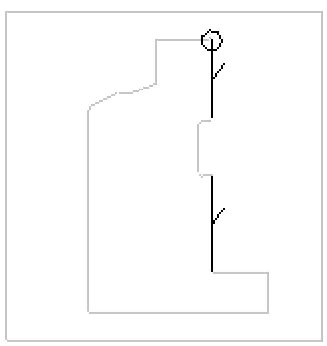
☐ Turn the **New Drive Curve** option **on** and choose **Select**.

☐ Choose the two vertical curves in the order illustrated below. Be sure you choose each curve at the top end.



☐ **OK** to accept the drive geometry.

☐ Choose **Display** to verify the drive geometry.

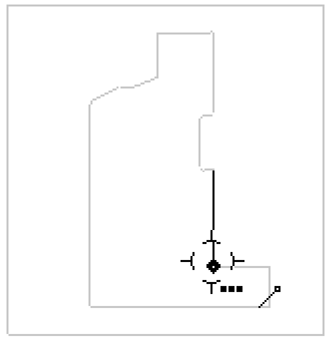


☐ Choose the **Start/Stop** tab.

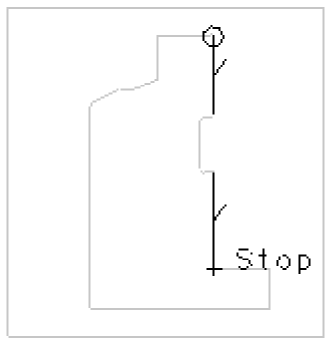
☐ Be sure the **Starting Position Method** is set to **Drive Curve Start**.

☐ Set the **Stopping Position Method** to **Point** and choose **Select**.

☐ Select the lower end of the vertical line as illustrated below.



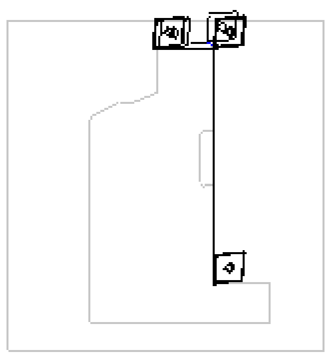
☐ Choose the **Main** tab and **Display All Geometry** to verify that the drive geometry is correct.



☐ **OK** to accept the Follow Curve Motion.

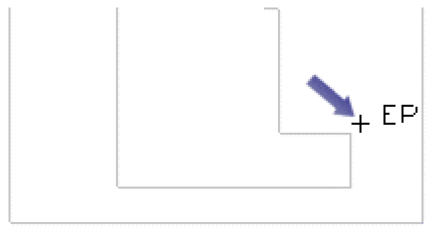
Step 22: Generate the Tool Path

☐ Choose **Generate**.



Step 23: Create a Rapid Motion suboperation.

- ☐ Choose the **Linear Rapid** icon.
- ☐ Be sure the **Move Radial and Axial** icon is chosen.
- ☐ With the **Point** option turned **on**, choose **Select**.
- ☐ Choose the **Cursor Location** icon.
- ☐ Indicate an end point at the approximate screen position illustrated below.



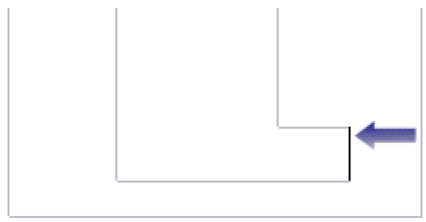
- ☐ **OK** to accept the Rapid Motion.

Step 24: Create a Retract Setting suboperation.

- ☐ Choose the **Retract Setting** icon.
- ☐ Be sure the **Auto Circular** icon is chosen.
- ☐ Key in **0.200** in the Extend Distance field.
- ☐ **OK** to accept the Retract Setting.

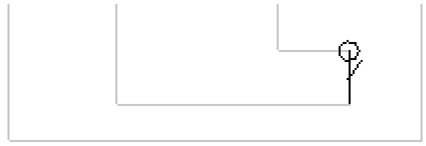
Step 25: Create a Follow Curve Motion suboperation.

- ☐ Choose the **Follow Curve Motion** icon.
- ☐ Turn the **New Drive Curve** option **on** and choose **Select**.
- ☐ Choose the vertical curve at the top end as illustrated below.



- ☐ **OK** to accept the drive geometry.

☐ Choose **Display** to verify the drive geometry.



☐ Choose the **Start/Stop** tab.

☐ Be sure the **Starting Position Method** is set to **Drive Curve Start**.

☐ Be sure the **Stopping Position Method** is set to **Drive Curve End**.

☐ **OK** to accept the Follow Curve Motion.

Step 26: Create a Rapid Motion suboperation.

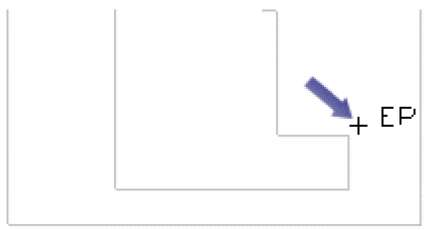
☐ Choose the **Linear Rapid** icon.

☐ Be sure the **Move Radial and Axial** icon is chosen.

☐ With the **Point** option turned **on**, choose **Select**.

☐ Choose the **Cursor Location** icon.

☐ Indicate an end point at the approximate screen position illustrated below.



☐ **OK** to accept the Rapid Motion.

Step 27: Create a Retract Setting suboperation.

☐ Choose the **Retract Setting** icon.

☐ Choose the **Vector** icon.

☐ Key in **0.100** in both the Vector (X) and Vector (Y) fields.

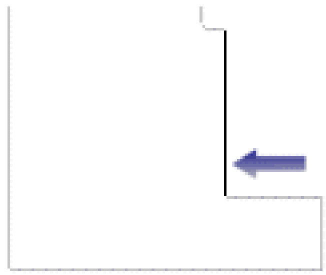
☐ **OK** to accept the Retract Setting.

Step 28: Create a Follow Curve Motion suboperation.

- ☐ Choose the **Follow Curve Motion** icon.
- ☐ Turn the **New Drive Curve** option **on** and choose **Select**.
- ☐ Under Material Side, turn the **Right** option **on**.
- ☐ Choose the horizontal curve at the end illustrated below.

- ☐ **OK** to accept the drive geometry.
- ☐ Choose **Display** to verify the drive geometry.

- ☐ Choose the **Start/Stop** tab.
- ☐ Be sure the **Starting Position Method** is set to **Drive Curve Start**.
- ☐ Set the **Stopping Position Method** to **Check Curve**.
- ☐ Choose **Select** under Stop Check Geometry.
- ☐ Select the vertical curve.

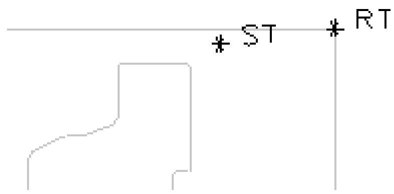


- ☐ **OK** to accept the check geometry.
- ☐ **OK** to accept the Follow Curve Motion.

Step 29: Specify Avoidance Geometry.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

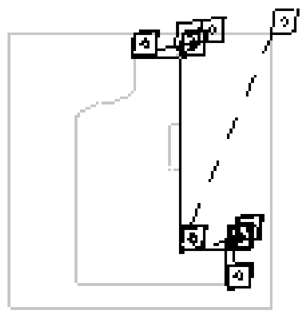
☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below.



☐ **OK** to accept the points.

Step 30: Generate the Tool Path

☐ Choose **Generate**.



☐ **OK** to complete the operation.

☐ Save and close the part file.

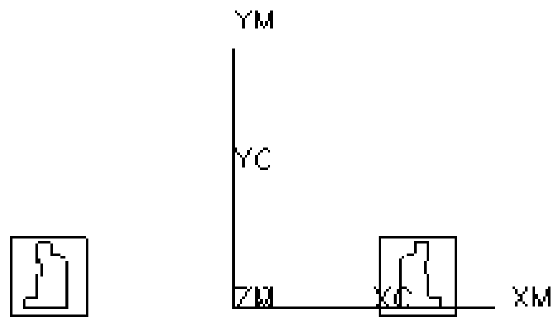
Activity: Postprocessing

In this activity, you are going to postprocess the mill-turn program and view the output.

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

☐ Open the part file **tmp_vtl_2.prt**.

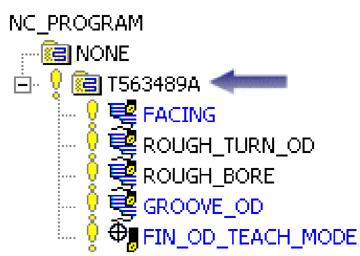
☐ Enter the **Manufacturing** application.



Step 2: Postprocess the mill-turn program.

☐ Display the Program Order View of the Operation Navigator.

☐ Highlight the **T563489A** program.



☐ Choose the **UG/Post Postprocess** icon in the toolbar.

☐ Choose **LATHE_2_AXIS_TOOL_TIP** in the Available Machines list.

☐ Specify the directory and file name. The system will save the file as ***_**tmp_vtl_2.prt** in the current directory by default.

☐ Choose **OK** to create the file.

Step 3: View the output.

- ☐ Scroll through the Information window.
- ☐ Dismiss the Information window.
- ☐ Save the part file.

Summary

In this lesson you created a Vertical Turret Lathe (VTL) program.

You learned how to:

- orient the view of the part for a Vertical Turret Lathe (VTL)
- orient the WCS to facilitate the input of angle values
- orient the MCS according to the requirements of your postprocessor
- specify the Lathe Work Plane
- create facing, roughing, grooving, and finishing operations
- postprocess the program

Index

M

O

R

T

V

PLM SOLUTIONS

STUDENT PROFILE

EDS

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	o	o	o	o
CAD assemblies	o	o	o	o
CAD drafting	o	o	o	o
CAM	o	o	o	o
CAE	o	o	o	o
PDM – data management	o	o	o	o
PDM – system management	o	o	o	o

Platform (operating system) ____

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

This page left blank intentionally.

Turning Manufacturing Process Course Agenda

Day 1

- Introduction & Overview – *Workbook (Section 1, 2, 3)*
- Lesson 1. Lathe Cross Section – *Workbook (Sections 4, 5)*
- Lesson 2. Retrieving and Creating Tools – *Workbook (Section 6)*

Afternoon

- Lesson 3. The Create Geometry Dialog – *Workbook (Sections 7, 8, 9)*
- Lesson 4. Facing Operations
- Lesson 5. Verification

Day 2

- Lesson 6. Common Options
- Lesson 7. Centerline Operations – *Workbook (Sections 10, 11, 12)*
- Lesson 8. Roughing Operations – OD Work

Afternoon

- Lesson 9. Roughing Operations – ID Work
- Lesson 10. Finishing Operations – OD and ID Work
- Lesson 11. Grooving– *Workbook (Sections 13, 14, 15, 16)*

Day 3

- Lesson 12. Teach Mode– *Workbook (Sections 17, 18, 19)*
- Lesson 13. Threading Operations – *Workbook (Section 20)*
- Lesson 14. Using Multiple Spindles

Afternoon

- Lesson 15. Mill–Turn
- Lesson 16. Vertical Turret Lathe
  Workbook (Sections 21, 22) - Postprocess, Shop Documentation

---

This page left blank intentionally.

Class 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		

PLM Solutions

Evaluation – Delivery

Turning Manufacturing Process, Course #MT11055

Dates _____________ thru______________

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

Instructor: ________________________________ ☑

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

Instructor: ________________________________ ☒

		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:

		STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
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

PLM Solutions

Evaluation - Courseware

Turning Manufacturing Process, Course #MT11055

Dates _____________ thru _____________

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:

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.