Turning Manufacturing Process

Student Guide
April 2006
MT11055 — NX 4

Publication Number
mt11055_g NX 4

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
	NX 3	March 2005
	NX 4	April 2006

Contents

Figures

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 NX Turning Manufacturing application.

Prerequisites

The required prerequisites for the course are:

- *Practical Applications of NX* or CAST equivalent
- *NX Manufacturing Fundamentals with Basic Design* course or the current *CAM Transition* course
- Basic understanding of the Master Model concept

A working knowledge of the following:

- The 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:

- Create cross-sectional curves for use in defining part and blank geometry.
- Define part and blank geometry for Turning operations.
- Create facing, centerline drilling, roughing, grooving, finishing, and threading operations.
- Visually verify the program by displaying 2D and 3D dynamic material removal.
- Define part geometry for parts mounted at each spindle of a multiple spindle machine.
- Manage the in-process workpiece as it is passed from one spindle to the next.
- Create a program that contains milling and turning operations.

Student Responsibilities

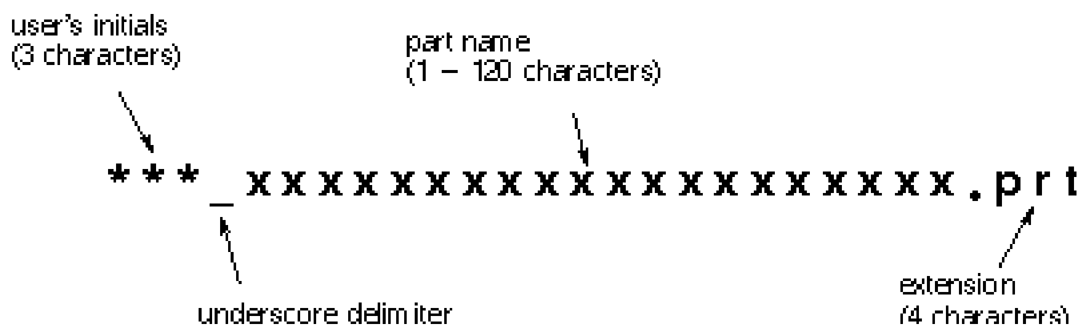
- Be on time
- Participate in class
- Focus on the subject matter
- Listen attentively and take notes
- Enjoy the class

Class Standards for NX Part Files

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

Class Part File Naming

This class utilizes the following 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 →Tool Path →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 Tool Path.
- 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 NX increases, the action boxes may seem redundant as the step text becomes all that is needed to accomplish a given task.

Step 1: This is an example of a step.

☐ This is an example of an action box.

The general format for lesson content is:

- Presentation
- Activity
- 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 NX, being ready to follow the instructor's curriculum. At the end of the day's class you should always quit NX and log off the terminal.

Workbook Overview

The 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 the 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.

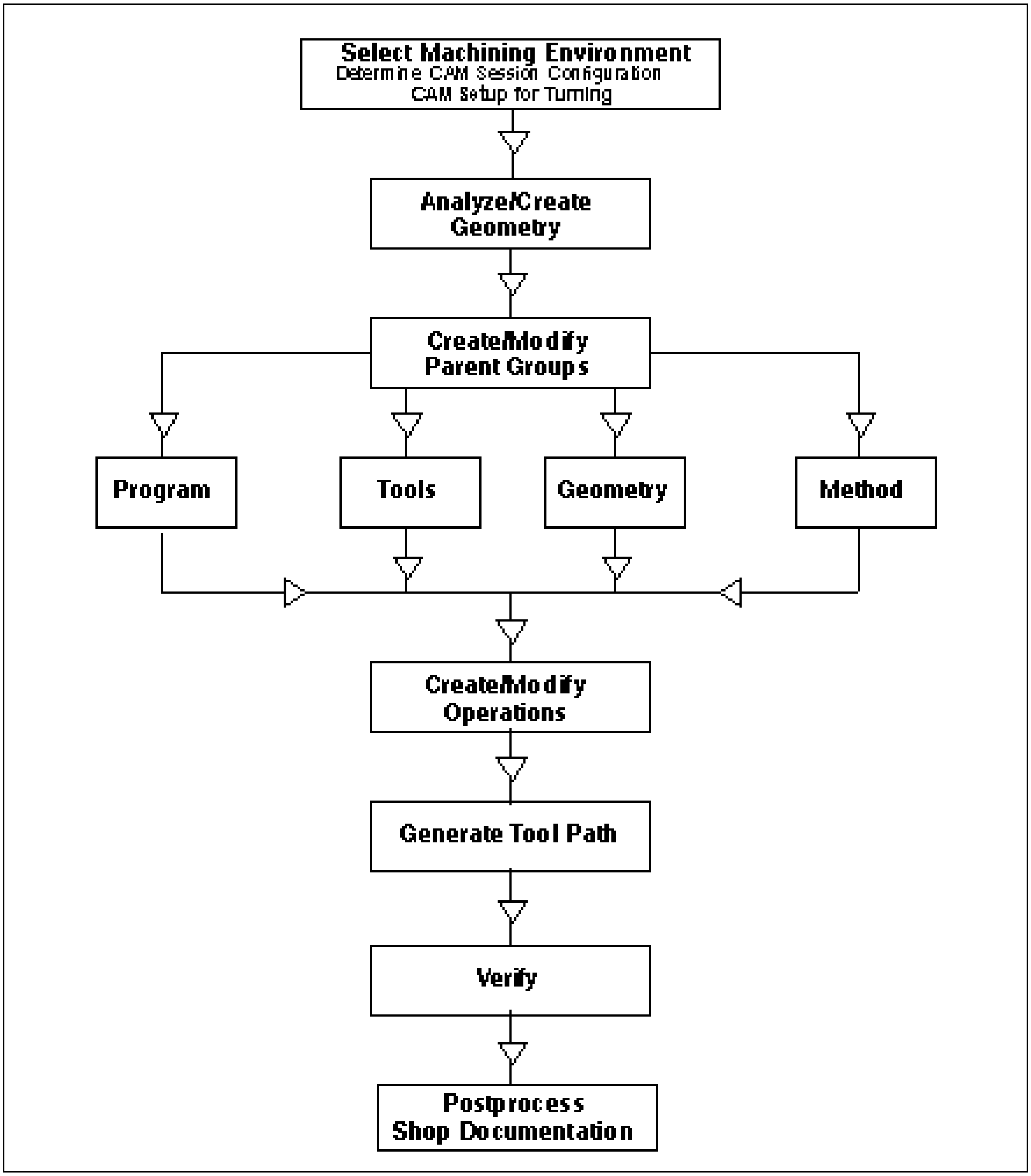


Figure 1. NX Manufacturing Process — Turning

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 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.

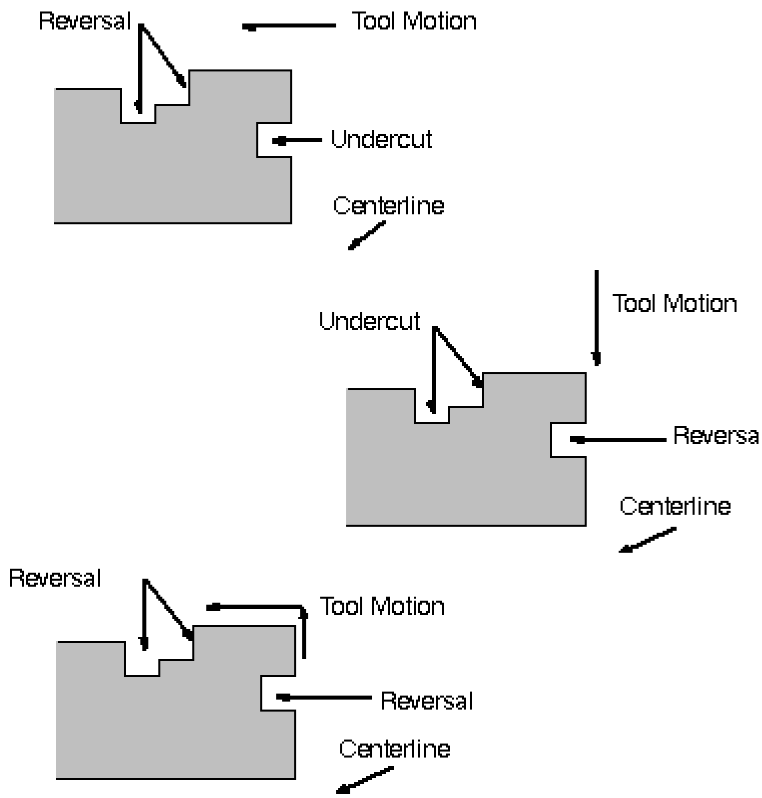
Teach Mode - is a cut type that enables you to define the motions of the operation step by step. This is comparable to manual programming.

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).



1

Lesson

1 Defining Part and Blank Geometry

Purpose

This lesson covers how to define the machine coordinate system (MCS), create blank and part boundaries, and define cut region containment.

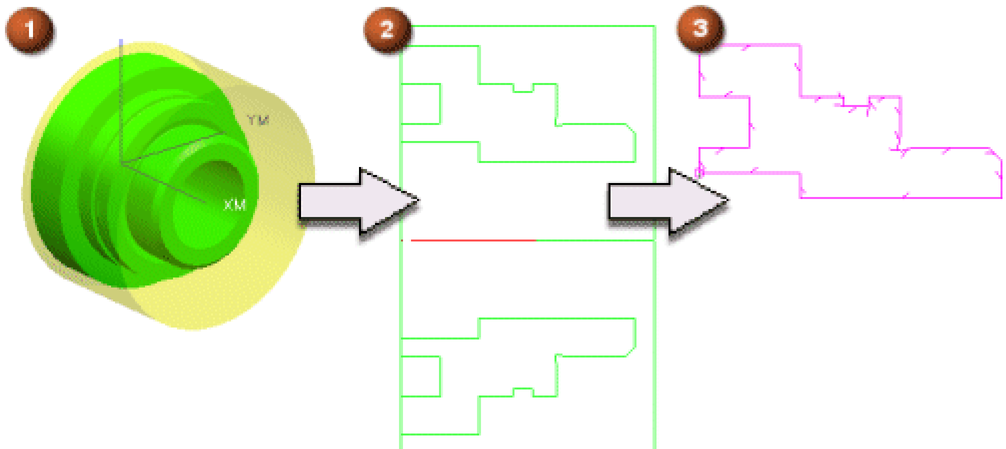
Objective

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

- Create Part and Blank boundaries by selecting their solid bodies.
- Edit the MCS_SPINDLE group.
- Create Blank boundaries as bar stock or tube stock.
- Define cut region containment within a Containment group and within an operation.

Part & Blank Boundaries

Part and Blank boundaries are required to generate a tool path. To create Part and Blank boundaries, begin by selecting the solid bodies (1) that contain the part and blank. Cross-sectional curves (2) representing a 2D cross section of a solid body are created automatically. These curves serve as a reference for tool paths and containment as you create operations. You can then display the boundaries (3) needed for generating a tool path.

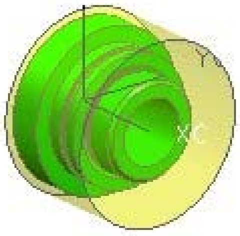


Activity: Creating and Viewing Part & Blank Boundaries

In this activity, you will select the solid bodies that define the part and blank geometry. You will then view the cross sectional curves and display the part and blank boundaries in the TURN_WORKPIECE object.

Step 1: Open and rename the part.

☐ Open the **tmp_cross_sec_mfg** part.



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

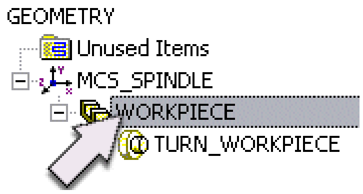
☐ Choose **Start→Manufacturing**.

Step 2: Specify Part and Blank geometry within the **WORKPIECE** group.

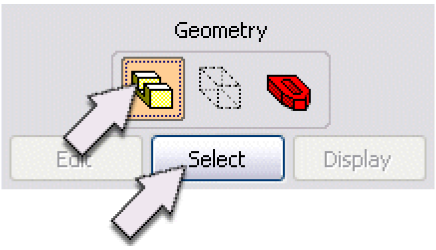
You will specify the Part and Blank geometry within WORKPIECE by selecting the solid bodies of the Part and Blank.

☐ Display the Geometry View of the Operation Navigator and expand all objects.

☐ Double-click **WORKPIECE**.

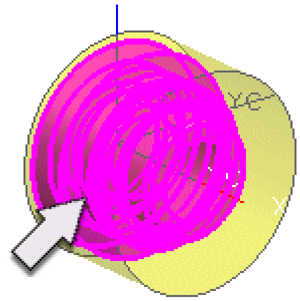


☐ In the Workpiece dialog, choose the **Part** icon and **Select**.



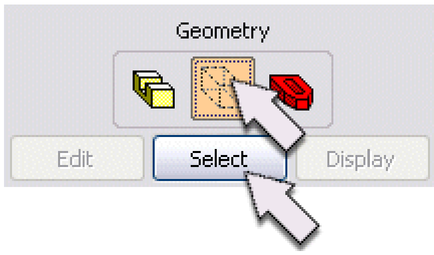
☐ In the **Part Geometry** dialog, make sure **Selection Options** are set to **Geometry** and **Filter Methods** are set to **Solid Bodies**.

☐ Select the solid body of the Part as illustrated below.

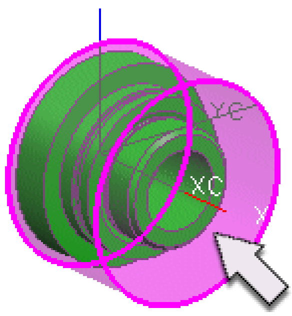


☐ **OK** to accept the Part Geometry.

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



☐ Select the stock surrounding the part as illustrated below.



☐ **OK** to accept the Blank Geometry.

You have now defined the Part and Blank Geometry and the associated cross sectional curves.

Step 3: Display cross sectional curves in the **TURN_WORKPIECE** group.

You will display the cross sectional curves in the TURN_WORKPIECE group by hiding the layers containing the solids.

☐ Choose **Format→Layer Settings**.

☐ Double-click layers 1 and 2 to hide the solid bodies they contain.

☐ **OK** the dialog.

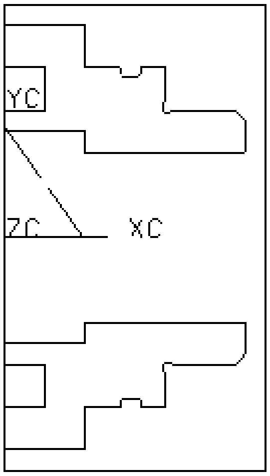
The solids will disappear from the screen, leaving only the coordinate systems.

☐ In the Operation Navigator, double-click **TURN_WORKPIECE**.

The 2D cross section appears shaded in isometric view.

☐ **MB3→Rendering Style→Static Wireframe** to change the display to remove the shading.

☐ **MB3→Orient View→Top** to change to a TOP view.

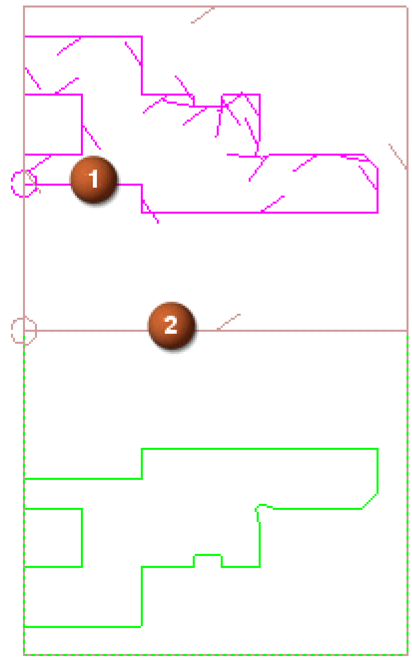


Here you see the cross sectional curves used to define the part and blank boundary. You will reference these curves when creating boundaries required by operations.

☐ In the **TURN_BND** dialog, select the **Part** icon and click **Display**.

☐ Select the **Blank** icon and click **Display**.

The Part (1) and Blank (2) boundaries are displayed on the cross sectional curves you created. These boundaries help define the tool path for operations.



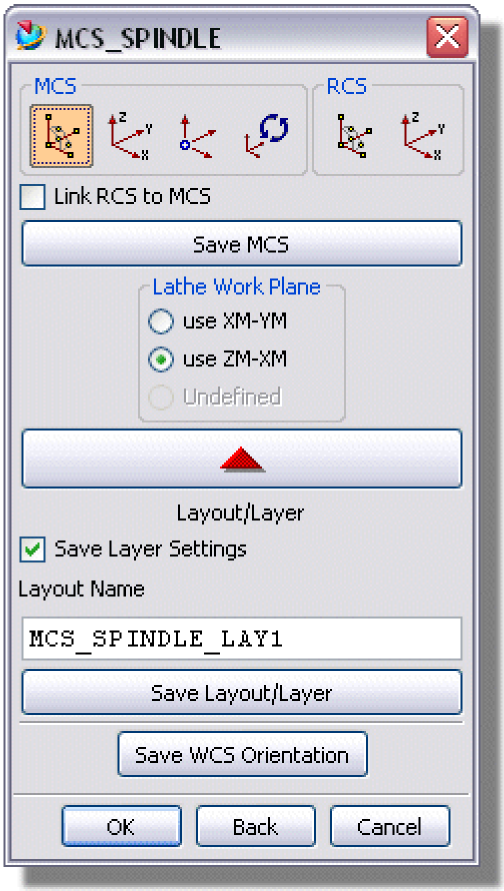
☐ Save and close the part.

The Create Geometry Dialog

The Create Geometry dialog allows you to define MCS_SPINDLE (1), WORKPIECE (2), TURN_WORKPIECE (3), TURN_PART (4), CONTAINMENT groups (5), and AVOIDANCE (6).

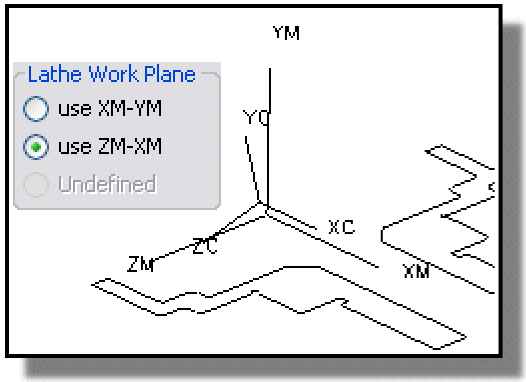
MCS Spindle

MCS SPINDLE allows you to create, edit, and store Machine Coordinate Systems, used for defining spindles.



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 respectively.

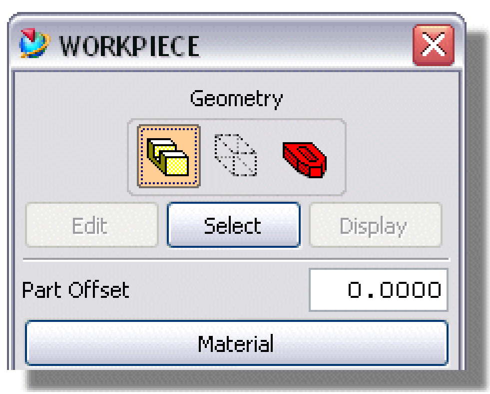
If the XC-YC plane of the WCS is not parallel to the XM-YM or ZM-XM lathe work plane as illustrated below, you can directly specify XM-YM or ZM-XM as the desired lathe work plane. In the following figure, choosing "use ZM-XM", specifies ZM-XM as the lathe work plane regardless of the orientation of the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



The Save Layout/Layer button 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

WORKPIECE allows you to define the part and blank geometry by selecting their solid bodies.

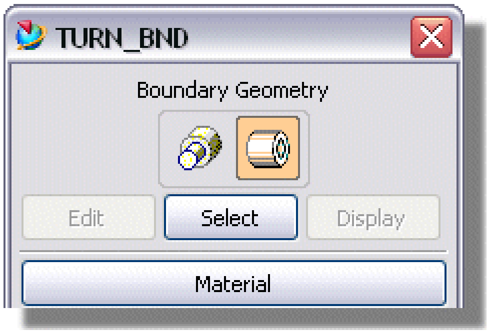


In a new CAM Setup, the MCS SPINDLE, WORKPIECE, and TURN_WORKPIECE are created for you automatically. The initial WORKPIECE contains no part or blank data, but it can be edited to include this data.

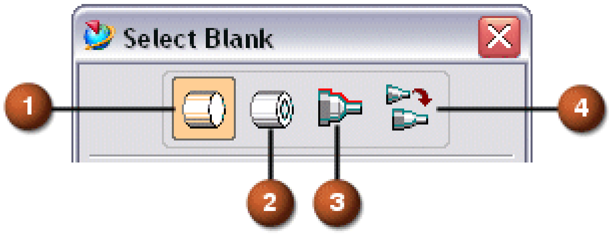
If you define the part material from this dialog, that definition will override any definition made using the menu bar (Tools→Part Material).

Turn_Workpiece

TURN_WORKPIECE contains the cross-sectional curves and boundaries derived from part and blank geometry defined in the WORKPIECE. It also allows you to define part boundaries from faces, curves, points and blank boundary geometry from bar stock, tube stock, curves or the in-process workpiece.



Blank geometry can be defined in the TURN_WORKPIECE or in the following ways.



1. Bar Stock (mount position and blank parameters)

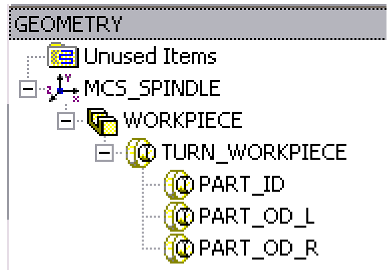
2. Tube Stock (mount position and blank parameters)

3. From Curve (existing geometry created by lathe cross section)

4. From Workspace (references in-process workpiece)

Turn_Part

When contained within TURN_WORKPIECE parent groups, PART parent groups help you to organize your geometry. The first geometry parent group defined is the MCS_SPINDLE. Within the MCS is the WORKPIECE parent group which contains the solid bodies that define the part and blank geometry. Within the WORKPIECE parent group is the TURN_WORKPIECE parent group which contain the part and blank boundaries. Within the TURN_WORKPIECE parent are the PART groups that define various Part boundaries.



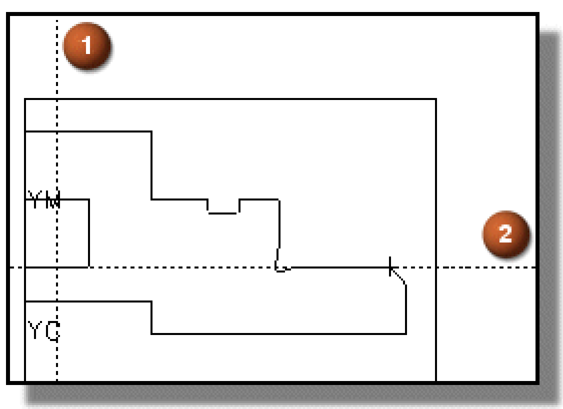
Naming or renaming the Geometry Parent Groups with meaningful names can alleviate a great deal of confusion when creating programs. Example:

- TURN_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

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.

An axial containment plane (1) can avoid collisions with the chuck. A radial containment plane (2) can avoid cutting to the center line.



Avoidance

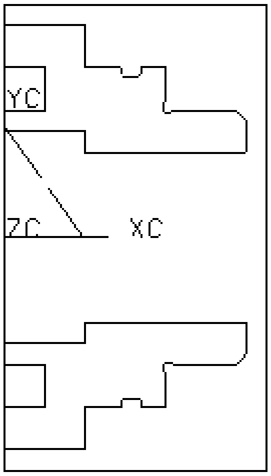
AVOIDANCE allows you to specify positions and movements for the tool as it moves toward or away from the part. You may define avoidance as a geometry group when several operations will use the same avoidance parameters. Otherwise, define avoidance within individual operations when they require unique avoidance parameters.

Activity: Defining Geometry Groups

In this activity, you will edit an existing MCS_SPINDLE group, define blank geometry within the TURN_WORKPIECE, and create a Containment group defining axial and radial trim planes.

Step 1: Open and rename the part.

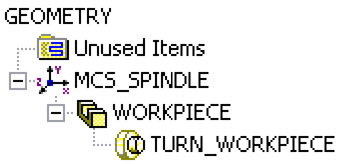
☐ Open the **tmp_geom_mfg_1** part.



☐ Save the part as *****_geom_mfg_1**, where *** represents your initials.

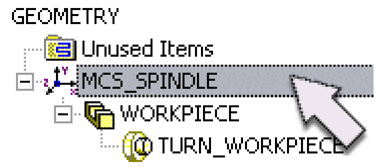
☐ Choose **Start→Manufacturing**.

☐ Display the Geometry View of the Operation Navigator and expand the MCS_SPINDLE object.

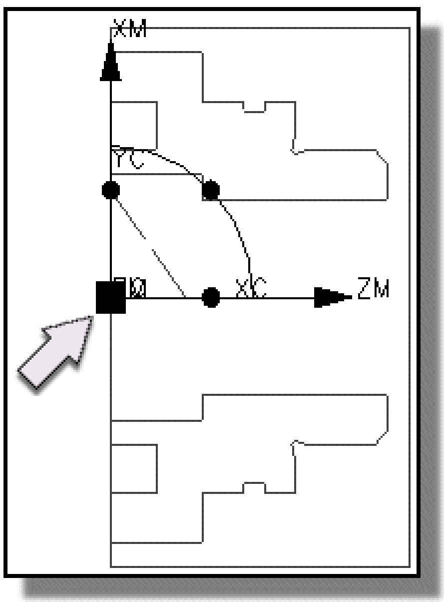


Step 2: Examine the MCS_SPINDLE parent group.

☐ Double-click **MCS_SPINDLE** to display the coordinates.



It is a good practice to check the MCS position and orientation each time you start a session or continue someone else's work.

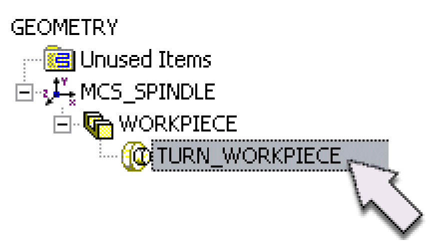


☐ **OK** to accept the MCS.

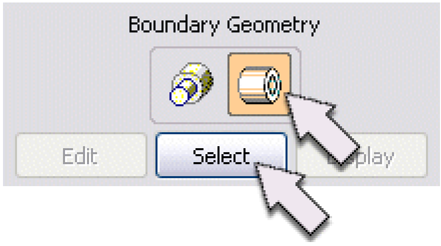
Step 3: Define blank geometry from bar stock.

You will redefine the blank geometry by keying in bar stock dimensions.

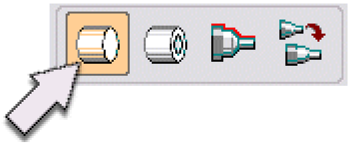
☐ Double-click **TURN_WORKPIECE** to edit the group.



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



- ☐ Choose the **Bar Stock** icon.

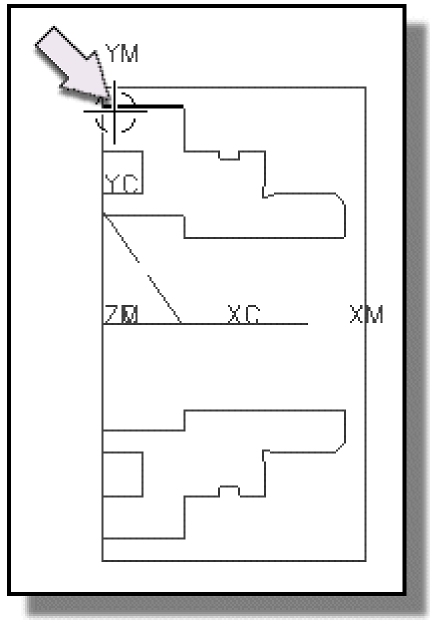


- ☐ Key in **3.25** in the Length field and **5.50** in the Diameter field.

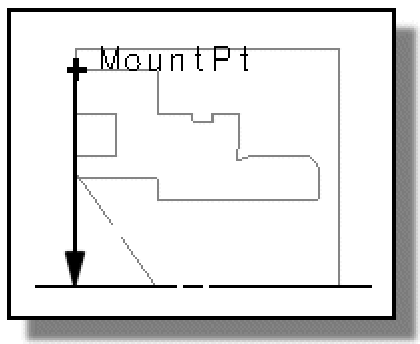
  The mounting position associates the blank geometry with the part geometry via a smart point. As a result, moving the part geometry also moves the blank geometry.

- ☐ Choose **Select** under Mounting Position.

☐ Select the end point of the horizontal line.

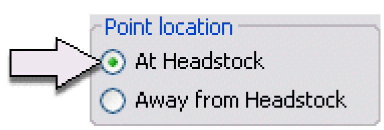


Internally, the Mount Point is projected to the centerline for blank placement.

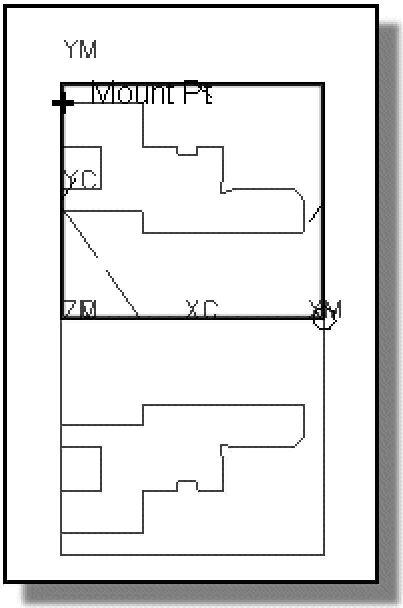


The Point Location options orient the blank geometry in relation to the mounting position you just defined.

☐ Be sure **At Headstock** is chosen for the point location.



☐ Choose **Display Blank**.

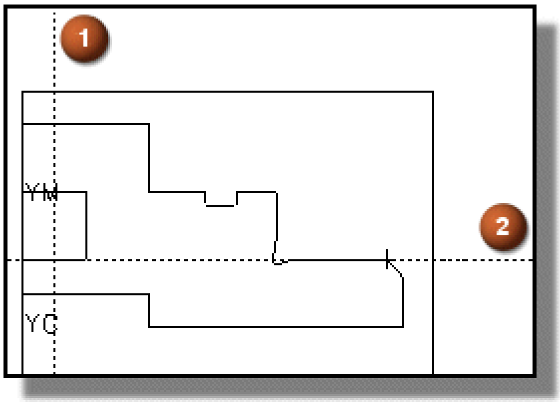


☐ **OK** to accept the blank geometry.

☐ **OK** to complete the Workpiece.

Step 4: Create a containment group.

Cut region containment restricts the cut region by allowing you to position axial (1) and radial (2) trim planes. Cut region containment is sometimes necessary to avoid colliding with the chuck or cutting too close to the center line. Containment can be defined within the Workpiece parent group or within individual operations.



☐ Choose the **Create Geometry** icon. 

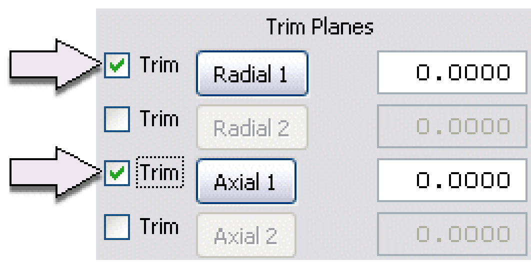
☐ Choose the **Containment** icon.

☐ Be sure the **Parent Group** option is set to **TURN_WORKPIECE**.

☐ **OK** to begin creating the Containment group.

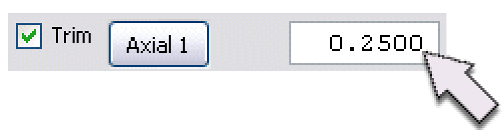
You will define containment geometry by positioning axial and radial trim planes.

☐ Turn the **Trim** options **on** next to the **Radial 1** and **Axial 1** buttons.



At 0.0000, Radial 1 initially lies on the XM axis and Axial 1 lies on the YM axis. You can reposition these trim planes by selecting points or by keying in values.

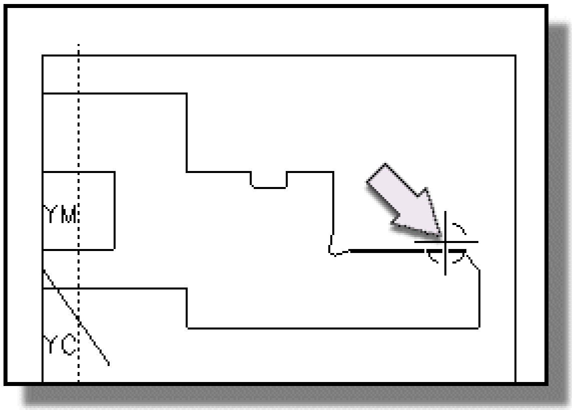
☐ Key in **0.25** in the Axial 1 field.



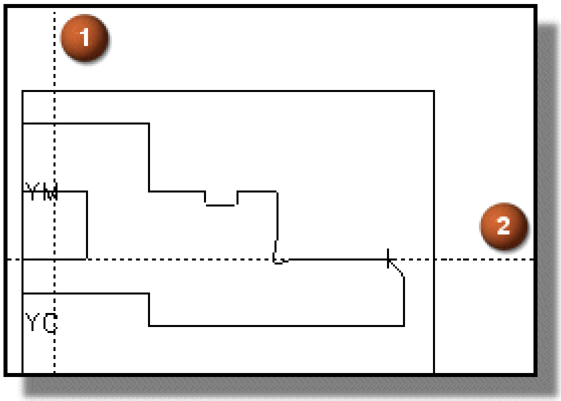
This will prevent the tool from colliding with the chuck.

☐ Choose **Radial 1**.

☐ Select the end point of the horizontal line as illustrated below.



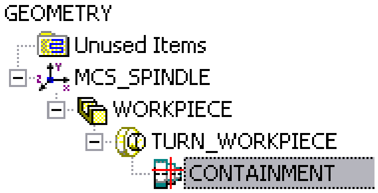
☐ Choose **Display Containment** to display both trim planes.



The cut region is limited to the area above the radial trim plane and to the right of the axial trim plane.

☐ **OK** to accept the containment.

This completes the containment group. The Geometry view of the Operation Navigator now displays four groups: MCS_SPINDLE, WORKPIECE, TURN_WORKPIECE, and CONTAINMENT. By defining the containment geometry within a group, this geometry is defined only once and can be used selectively when creating operations.



☐ Save and close the part.

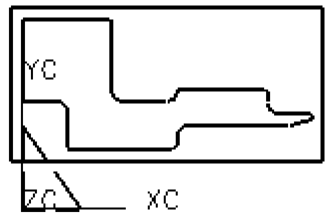
1

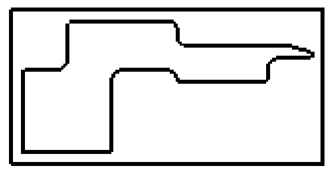
Activity: Defining Cut Region Containment within an Operation

In this activity, you will define cut region containment within an operation. This method is preferred over defining cut region containment within a group because some tool path creation requires unique containment settings.

Step 1: Open and rename the part.

☐ Open the **tmp_geom_mfg_2** part.

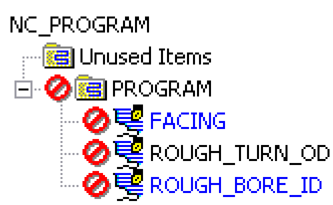




☐ Save the part as ***_geom_mfg_2**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

☐ Display the Program Order View of the Operation Navigator.



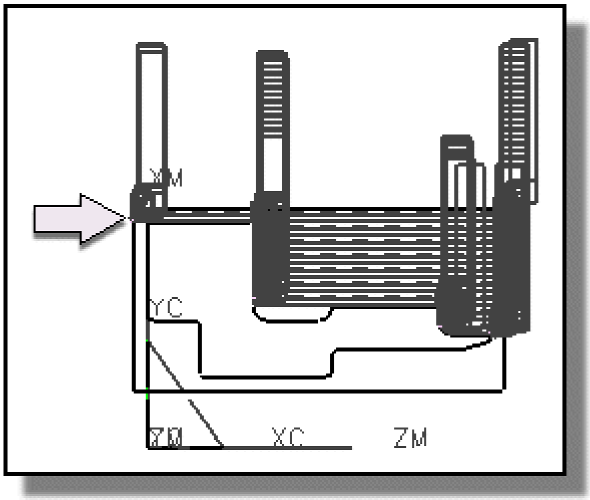
Step 2: Generate the tool paths.

☐ In the Operation Navigator, highlight the PROGRAM object and **MB3→Generate**.

☐ **OK** as needed to generated all of the tool paths.

☐ Replay the ROUGH_TURN_OD operation.

The tool cuts to the end of the blank where it will collide with the chuck.



You will define an axial trim plane for this operation to prevent the tool from colliding with the chuck.

Step 3: Define an axial trim plane.

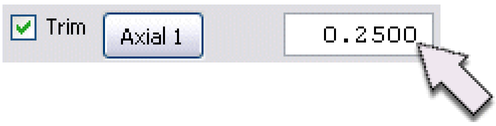
☐ In the Operation Navigator, double-click **ROUGH_TURN_OD** to edit the operation.

☐ Choose **Containment**.

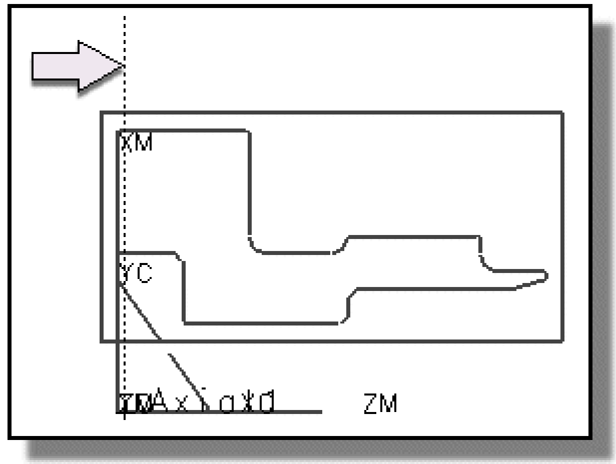
☐ Turn the **Trim** option **on** next to the **Axial 1** button.

You can reposition the Axial 1 trim plane by selecting points or by keying in values.

☐ Key in **0.25** in the Axial 1 field.



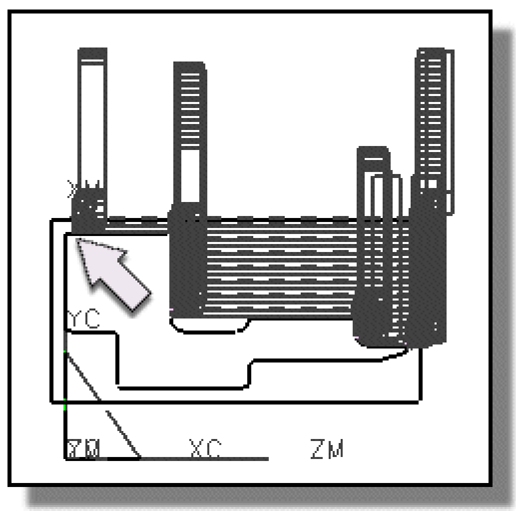
☐ **OK** the Geometry Containment dialog.



Step 4: Generate the tool path.

☐ **Generate** the tool path for the ROUGH_TURN_OD operation.

The tool path remains to the right of the axial trim plane.



☐ **OK** to complete the operation.

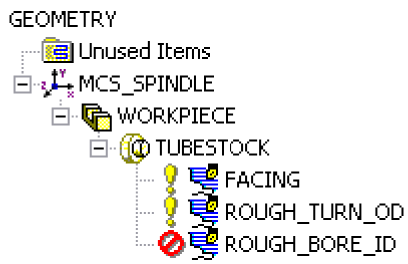
☐ Leave this part open. You will continue to use it in the next activity.

Activity: Defining Blank Geometry as Tube Stock

In this activity, you will redefine the blank geometry as tube stock. You will continue to use the part file **tmp_geom_mfg_2**.

Step 1: Display the Geometry View.

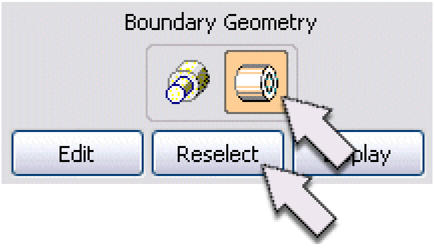
- ☐ Display the Geometry View of the Operation Navigator and expand the objects.



Step 2: Define blank geometry as tube stock.

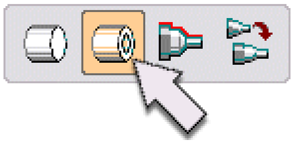
- ☐ In the Operation Navigator, double-click **TUBESTOCK** to edit the object.

- ☐ Choose **Blank** and **Reselect**.



- ☐ **OK** the warning.

- ☐ Choose **Tube Stock**.

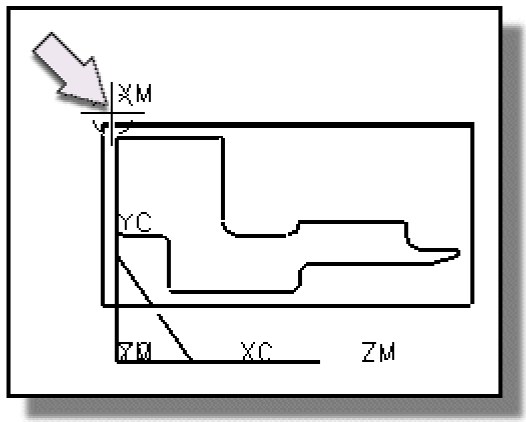


- ☐ Key in the following values in the Length, Outer Diameter, and Inner Diameter fields.
  - Length: **14.000**
  - Outer Diameter: **17.000**
  - Inner Diameter: **4.000**

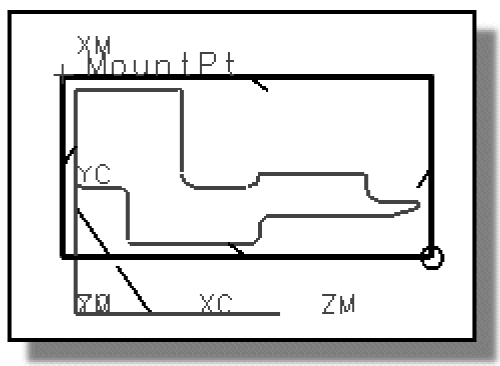
Step 3: Define the mounting position.

☐ Choose **Select** under Mounting Position.

☐ Select the end point of the horizontal line as illustrated below.



☐ Choose **Display Blank**.

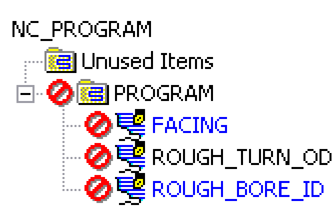


☐ **OK** to accept the blank geometry.

☐ **OK** to complete the tube stock.

Step 4: Generate the tool paths.

☐ Display the Program Order View of the Operation Navigator.



☐ In the Operation Navigator, highlight the **PROGRAM** object and **MB3→Generate**.

☐ **OK** as needed to generate all the tool paths.

☐ Save and close the part.

Summary

Turning geometry consists of spindle definition, part geometry, blank geometry, and cut region containment. Part and blank geometry are defined using boundaries. Blank boundaries can be defined as bar stock, tube stock, or by selecting the solid body that contains the blank. Part boundaries are most easily defined by selecting the solid body that contains the part. Cut region containment can be defined within a parent group or within individual operations. Defining cut region containment within individual operations is preferred because tool paths sometimes require unique containment settings.

Lesson

2 Retrieving and Creating Tools

2

Purpose

This lesson covers tool creation and retrieval.

Objective

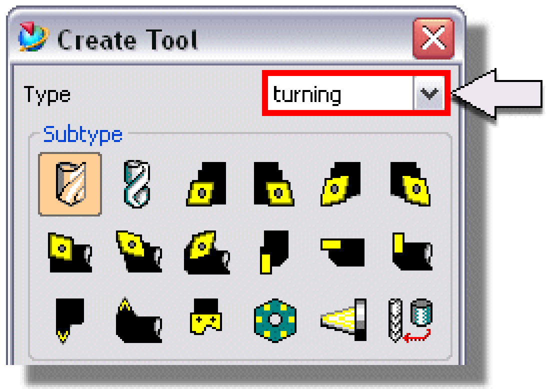
Upon completion of this lesson, you will be able to:

- Retrieve tools from the library.
- Create new tools.
- Manage tools for a two turret program.

2

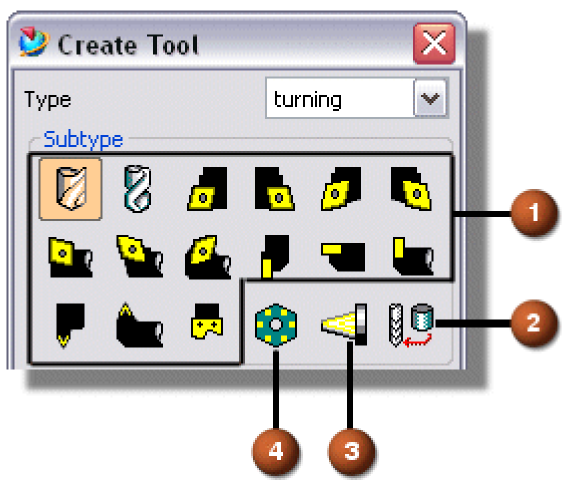
The Create Tool Dialog

The types of tools you can create are determined by the Machining Environment Setup. When the Type option is set to Turning, you can only create turning tools.



Operations must use tools of the appropriate type. Turning operations can only use turning tools. Milling operations can only use milling tools.

New tools can be created (1) or predefined tools can be retrieved from the library (2). When defining multi-turret operations tools can be assigned to pockets (3) (also called faces) within carriers (4).

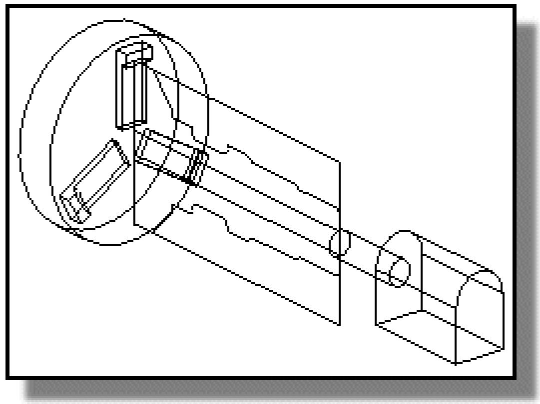


Determining the tool number by the pocket in which the tool is placed is analogous to a machine operator picking up a tool and putting it into a pocket on the machine.

Activity: Retrieving Tools from the Library

Step 1: Open and rename the part.

☐ Open the **tmp_tools_mfg_1** part.



☐ Save the part as *****_tools_mfg_1**.

☐ Choose **Start→Manufacturing**.

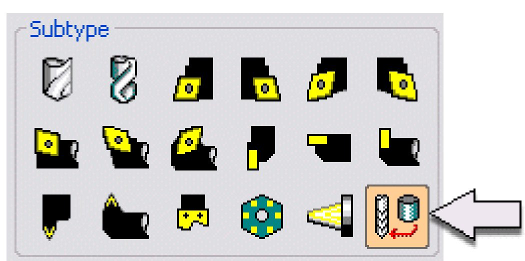
Step 2: Enter the search criteria.

☐ Display the Machine Tool View of the Operation Navigator.

☐ Choose the **Create Tool** icon. 

☐ Be sure the **Type** option is set to **turning**.

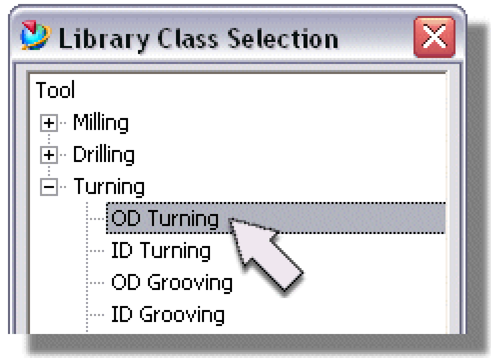
☐ Choose the **Retrieve Tool** icon.



☐ Choose **Apply**.

2

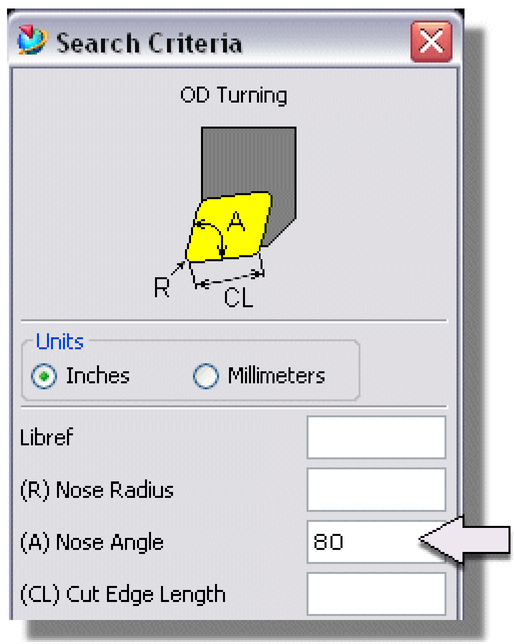
☐ Expand **Turning** and choose **OD Turning** in the Library Class Selection dialog.



☐ Choose **OK**.

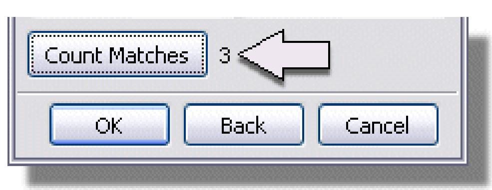
The Search Criteria dialog is displayed.

☐ Key in **80** in the Nose Angle field.



☐ Choose the **Count Matches** button.

Three matches are found.

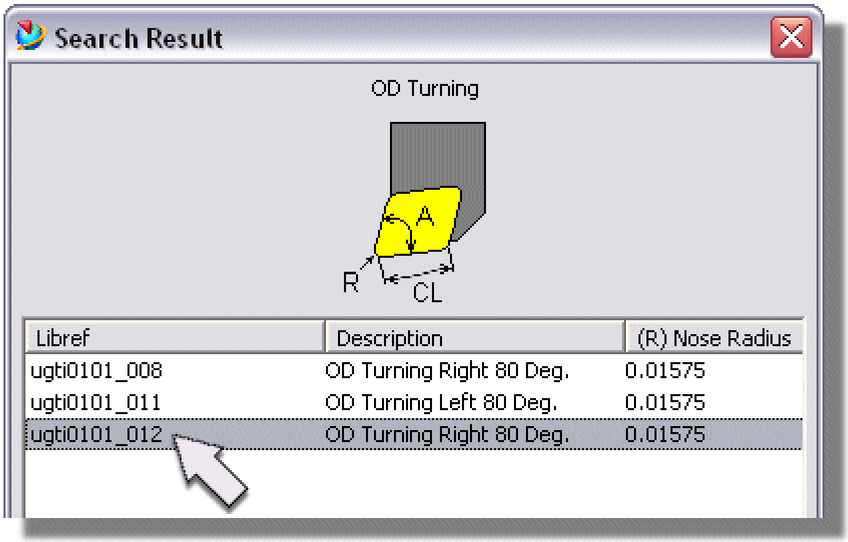


Step 3: Retrieve and display the tools.

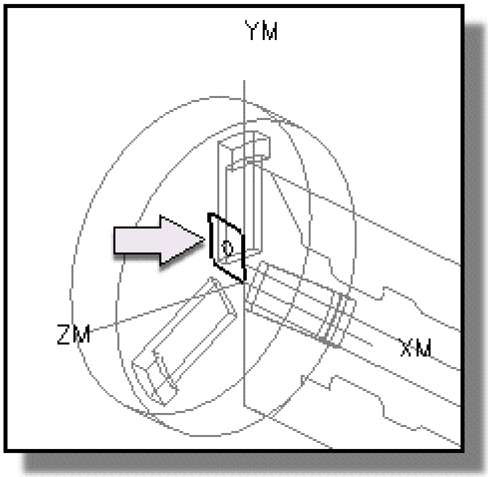
☐ Choose **OK**.

The Search Result window is displayed.

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



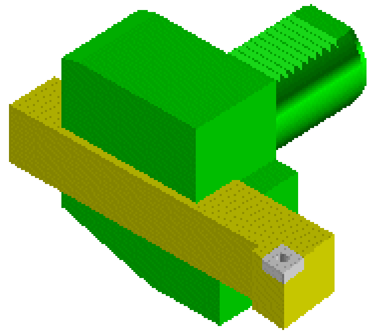
The tool displays at the WCS.



☐ Choose the tool **ugti0101_011** in the list, and **Display**.

This displays a tool assembly.

☐ Change to a shaded view to see the assembly clearly.



☐ Choose **OK** in the Search Result dialog.

☐ **OK** the warning.

The tool is retrieved into the part and displayed in the Operation Navigator.

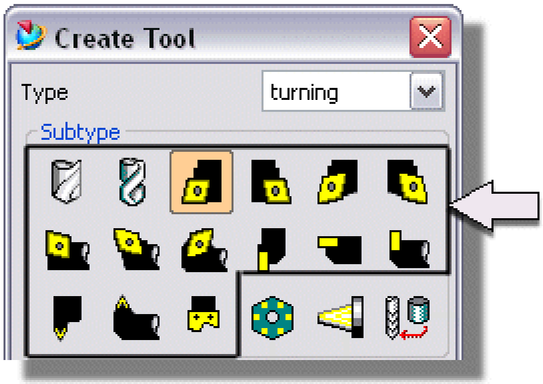
☐ Save the part.

Leave this part open. You will continue using it in the next activity.

Tools retrieved from the library have predefined tool numbers and cannot be put into faces.

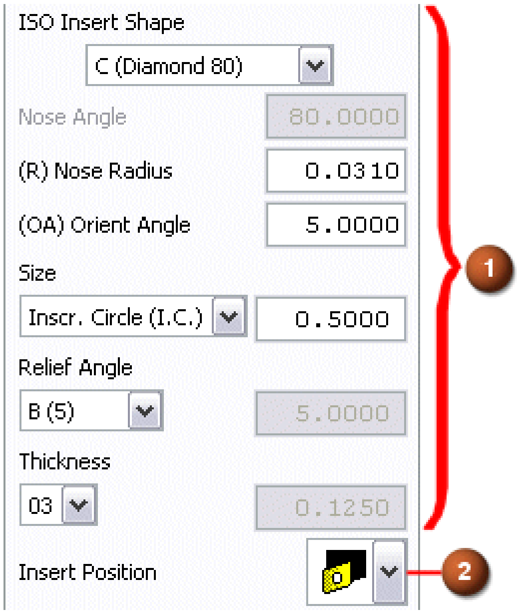
Creating New Turning Tools

You can create new tools using the tool templates or Subtypes.



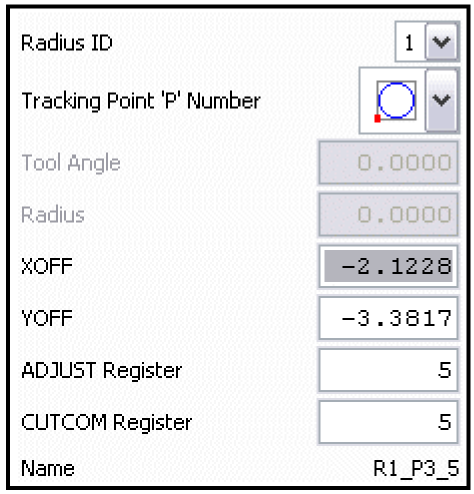
Common Parameters

The tool tab displays options that allow you to define common parameters for the specified subtype such as the size, shape, and orientation of the cutter (1). Insert Position (2) defines whether the cutter is mounted to the topside or the underside of the holder. The system determines a clockwise or counterclockwise rotation of the spindle based on the specified Insert Position and the location of the cut region above or below the centerline.



Tool Tracking

The Tracking tab displays options that define a tracking points at the tips of the cutter. Internally, the system calculates tool path output based on these points.



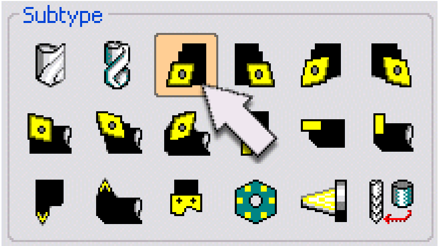
- The **Radius ID** selects which corner is the active corner.
- The **Tracking Point ’P’ Number** selects one of nine available points around the active corner as the tracking point. You may define a tracking point for each available corner of the cutting insert.
- The **Tool Angle** tracks the tool path from operation generation at the center of the tool nose radius, but makes the parameter value of the tool angle available to the post processor. This is only available when P9 is selected as the ’P’ Number.
- The **Radius** tracks the tool path from operation generation at the center of the tool nose radius This is only available when P9 is selected as the ’P’ Number.
- The **x offset (XOFF)** is the distance between the tool’s reference point and its tracking point measured along the spindle axis.
- The **y offset (YOFF)** is the distance between the tool’s reference point and its tracking point measured normal to the spindle axis.
- The **ADJUST Register** provides the location of the tool offset coordinates in the controller memory. The CUTCOM Reg adjusts the cutter path to allow for variation in the cutter size.

Activity: Creating New Tools

For this activity, you will continue using *****_tools_mfg_1**.

Step 1: Define a button tool.

- ☐ Display the Machine Tool View of the Operation Navigator.
- ☐ Choose the **Create Tool** icon. 
- ☐ Choose the **OD_80_L** subtype icon.

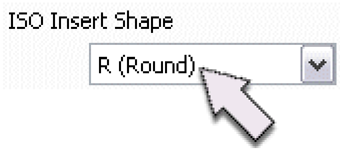


- ☐ Set the **Parent Group** option to **Face1**.
- ☐ Key in **button_.250** in the Name field.
- ☐ Choose **Apply** at the bottom of the Create Tool dialog.

Step 2: Change the insert type and enter the tool dimension.

To create a button tool, you must change the insert type.

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



- ☐ **OK** the warning.
- ☐ Key in the following tool parameters.

  Diameter: **.250**

  Orient Angle: **90**

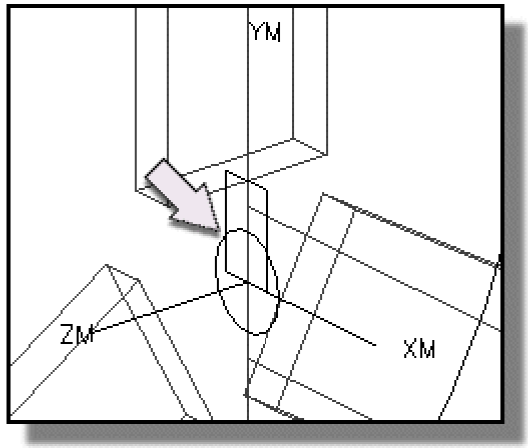
  Holder Control Width: **.1625**

  Holder Control Angle: **0**

  The tool, button_.250, has inherited the tool number from Face 1.

2

☐ Choose **Display Tool**.

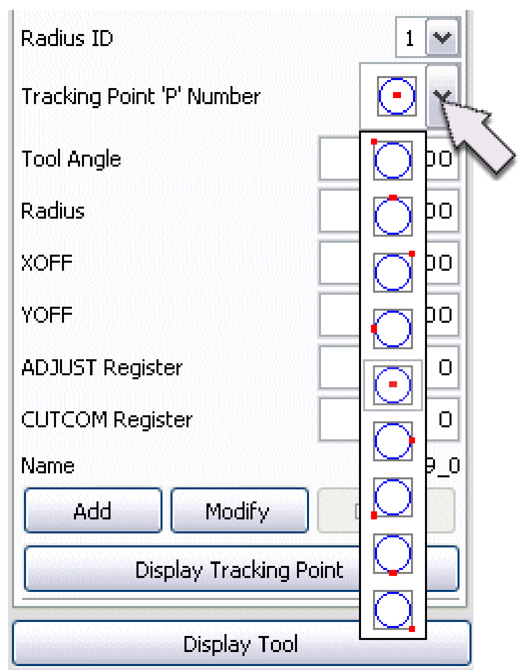


Step 3: Examine the Tool Tracking Options.

☐ Choose the **Tracking** tab.

This tool will track from the center point of the insert.

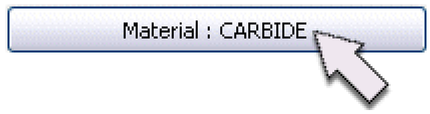
☐ Choose the **Tracking Point** arrow and examine the choices.



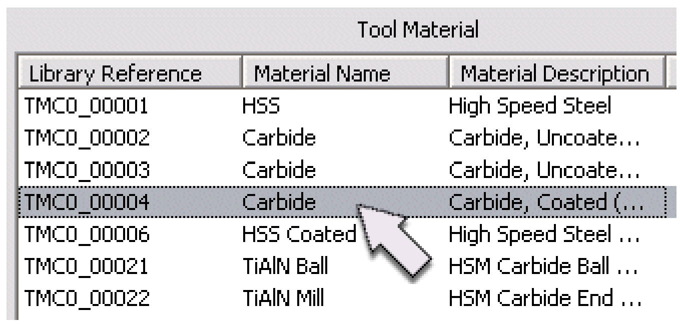
Step 4: Define the tool material.

☐ Choose the **Tool** tab.

☐ Choose the **Material** button.



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


Tool Material

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

☐ **OK** the Turning Tool-Standard dialog to complete the tool definition.

Step 5: Define a diamond tool.

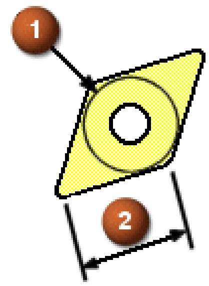
☐ Choose the **OD_55_L** subtype icon.

☐ Set the **Parent Group** option to **Face2**.

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

Step 6: Change the insert size.

The Inscr. Circle (I.C.) option allows you to key in the diameter of the inscribed circle (1). The Cut Edge Length option allows you to key in the length of the cutting edge (2).

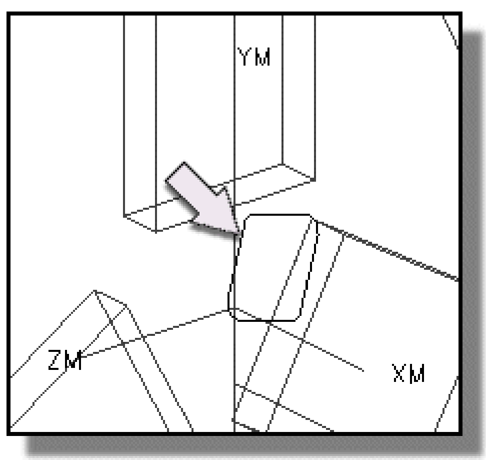


The I.C. ANSI option allows you to key in an ANSI code (a positive integer denoting eighths of an inch).

☐ Set the **Size** option to **Inscr. Circle (I.C.)**.

☐ Key in **0.2500** for the diameter of the inscribed circle.

☐ Choose **Display Tool**.



☐ **OK** the Turning Tool-Standard dialog to complete the tool definition.

Step 7: Create an ID cutting tool unassisted.

☐ 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)**

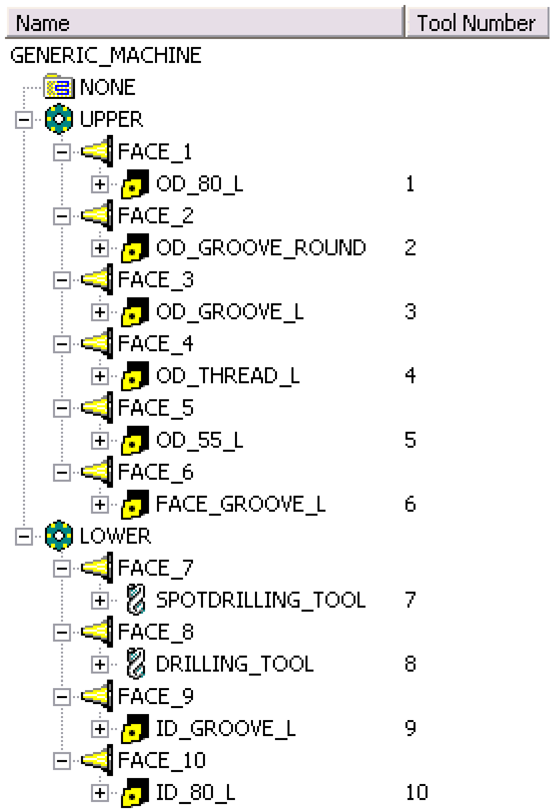
  The Cut Edge Length defines the insert size.

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

- Save and close the part.

Managing Tool Changes

Turrets and faces can be defined in the Operation Navigator, allowing you to interactively manage tool changes for multiple turret programs.



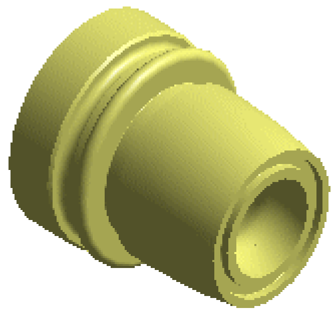
Tool numbers are assigned to faces and are inherited by tools and operations. Inheritance can be overridden by defining the tool number within the operation. Depending on your work environment, you will need to determine whether it is appropriate to define the tool number at the face level or at the tool level. Regardless of where you define the tool number, the system always displays the output tool number within the operation and in the Tool Number column of the Operation Navigator.

Activity: Defining a Single Spindle, Two Turret Program

In this activity you will use the Operation Navigator to define a two turret program. You will create turrets and faces and assign a tool to each face.

Step 1: Open and rename the part.

☐ Open the **tmp_tools_mfg_2** part.



☐ Save the part as *****_tools_mfg_2**.

☐ Choose **Start→Manufacturing**.

Step 2: Define the upper and lower turrets.

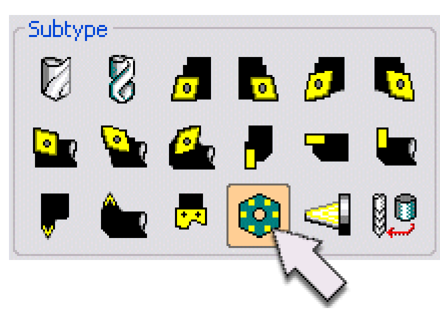
You will define upper and lower turrets and add numbered faces to each. You will see how these faces define the tool numbers inherited by the operations.

☐ Display the Machine Tool View of the Operation Navigator.

☐ Choose the **Create Tool** icon. 

☐ Be sure the **Type** option is set to **turning**.

☐ Choose the **Carrier** icon.



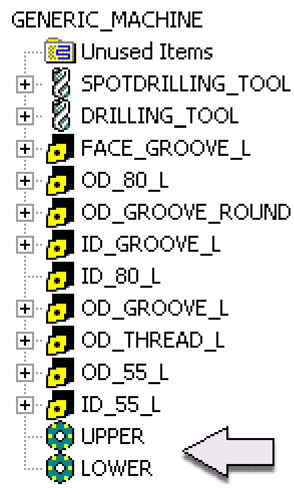
☐ Set the **Parent Group** option to **Generic Machine**.

☐ Key in **UPPER** in the Name field.

☐ **OK** to define the upper turret.

☐ **OK** to complete the turret definition.

☐ Repeat these steps to define a **LOWER** turret.

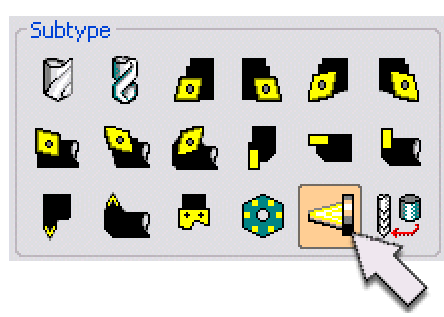


Step 3: Add faces to turrets.

You will add numbered faces to each turret. This is the most common method of determining the tool number on the machine tool. These turret numbers will be inherited by the tools and operations as tool numbers and output to the postprocessor.

☐ Choose the **Create Tool** icon.

☐ Choose the **MCT_POCKET** icon.



☐ Set the **Parent Group** option to **UPPER**.

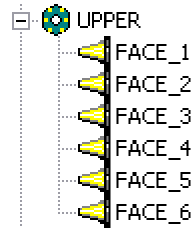
☐ Key in **FACE_1** in the Name field.

☐ **Apply** to begin defining the face.

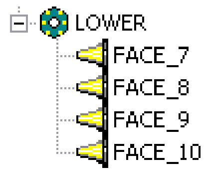
☐ Key in **1** in the Pocket ID field and **OK** the pocket dialog.

This is the number that will be assigned to the tool.

☐ Key in **FACE_2** in the Name field and repeat the process until **FACE_1** through **face_6** have been defined in the UPPER turret as illustrated below.



☐ Create faces 7–10 in the LOWER turret as illustrated below.



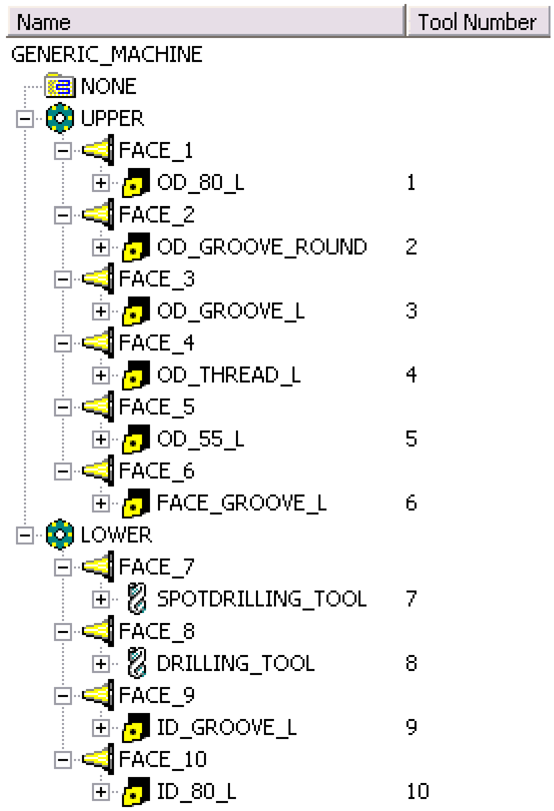
☐ **Cancel** the Create Tool dialog.

Step 4: Place tools into faces.

You will drag and drop tools into the faces you just defined. In doing so, the tools inherit the face numbers and the operations are divided between the upper and lower turrets. Determining the tool number by the face in which the tool is placed is analogous to a machine operator picking up a tool and putting it into a pocket on the machine.

☐ In the Operation Navigator, display the Tool Number column next to the Name column as illustrated below. This will allow you to see the tool numbers as the tools inherit them.

☐ Drag and drop each tool into each face as illustrated below.



☐ Save and close the part.

Summary

Tools can be created or retrieved from libraries. Turrets and faces can be defined in the Operation Navigator, allowing you to interactively manage tool changes for multiple turret programs.

Lesson

3 *Facing Operations*

3

Purpose

This lesson teaches you how to create a Facing operation. Facing the end of the part is one of the first operations you will perform in a typical turning program.

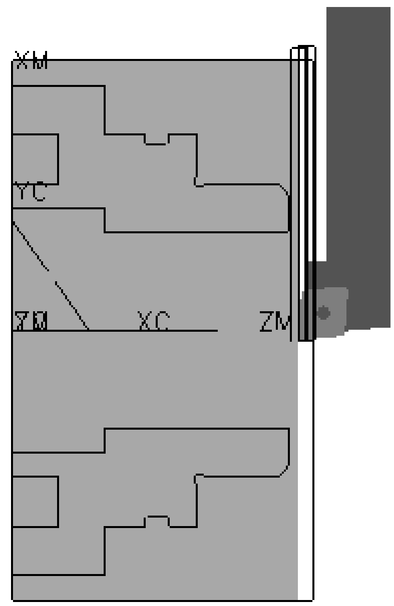
Objective

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

- Face the end of a part.
- Display the cut region.
- Limit the cut region using an axial trim plane.
- Specify face stock.
- Generate the tool path and visualize the in-process workpiece.

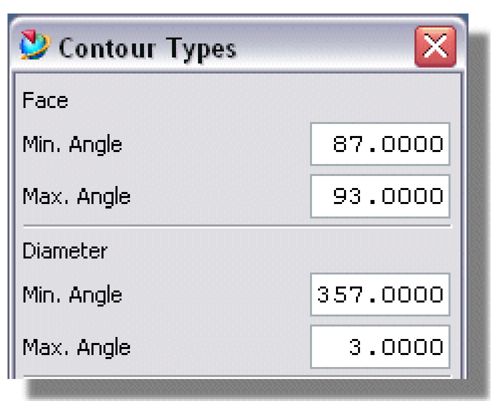
Facing Operations

Facing operations remove material from faces by roughing normal to the centerline. Many of the parameters found in Facing are common to other roughing operations.

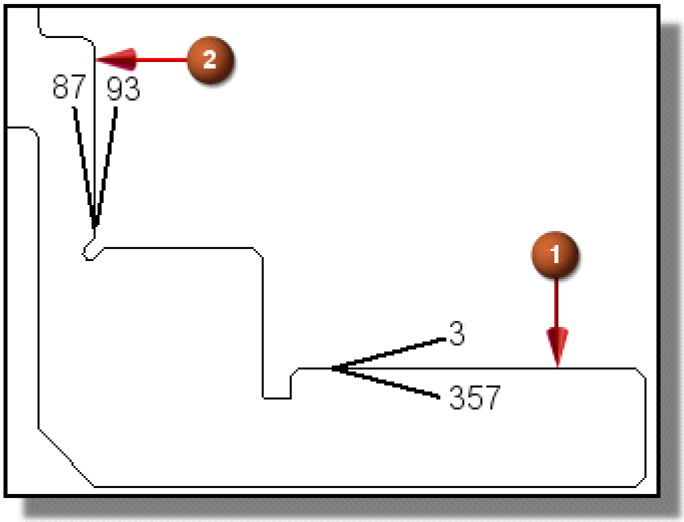


Defining Faces

The system defines faces based on parameters specified in the Contour Types dialog.

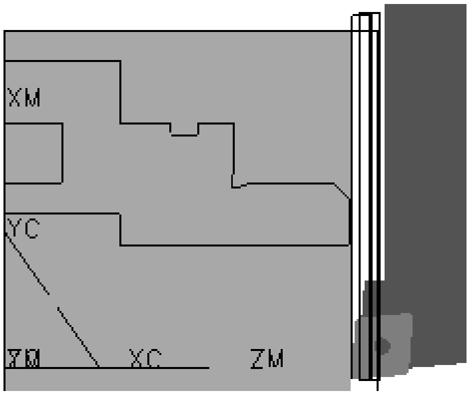


When the parameters are specified as shown above, any line or curve with a slope between 3 and 357 degrees (1) is recognized as a diameter, to which Radial stock is applied. Any line or curve with a slope between 87 and 93 degrees (2) is recognized as a face, to which Face stock is applied.



The In-Process Workpiece

The in-process workpiece is the remaining material at each stage of the cutting process. The system tracks and graphically displays the in-process workpiece as the program progresses. Each subsequent operation uses the in-process workpiece as it progresses from operation to the next.



Cut Regions and Automatic Cut Region Detection

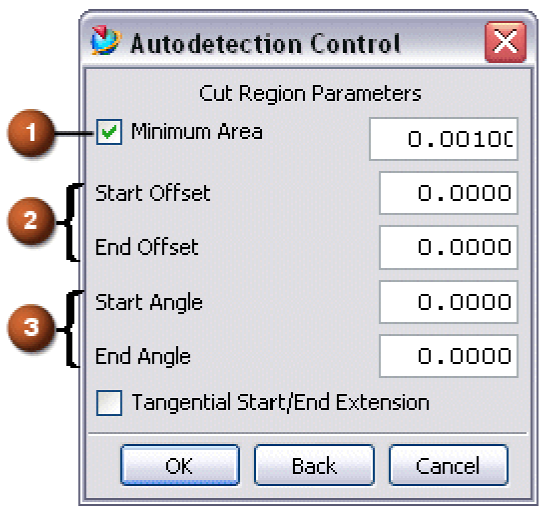
Roughing (and Finishing) operations use automatic cut region detection. The system detects the remaining material between the In-process Workpiece and the part surface and then uses the cut region for the subsequent operation.

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.

Autodetection Control

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



The Minimum Area (1) 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.

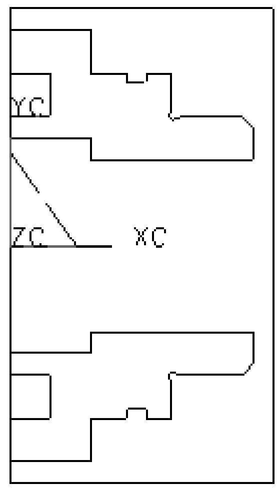
The Start/End Offset (2) and Start/End Angle (3) options apply only in the case of open boundaries, and only if a containment has not been set.

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. You will then reject the operation and set some of the options available on the Facing dialog.

Step 1: Open and rename the part.

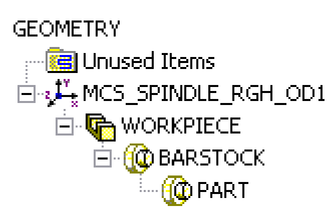
☐ Open the **tmp_face_mfg_1** part.



☐ Save the part as *****_face_mfg_1**, where *** represents you initials.

Choose **Start→Manufacturing**.

☐ Display the Geometry view of the Operation Navigator and expand the objects.

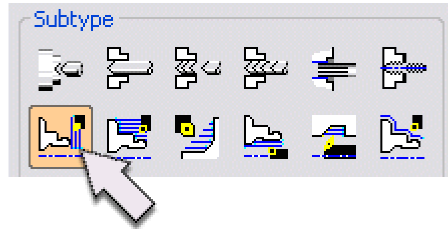


Step 2: Begin the Facing operation.

☐ Choose the **Create Operation** icon. 

☐ Set the **Type** option to **turning**.

☐ Choose the **Facing** icon.



☐ Set the following Parent Groups:

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

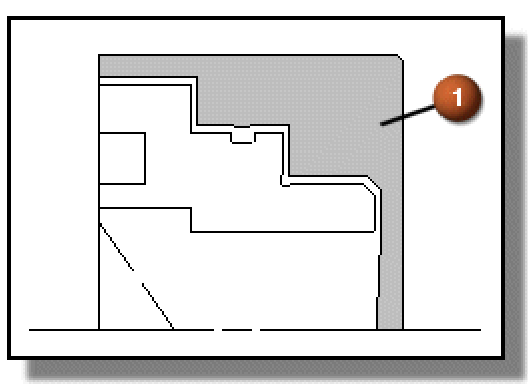
☐ Choose **OK**.

Step 3: Display the cut region.

By defining the part and blank geometry in parent groups, the system can track the material that needs to be removed. The blank geometry is defined in the BARSTOCK object and the part geometry is defined in the PART object.

☐ Choose **Display**.

The cut region (1) is displayed.



Step 4: Limit the area the tool will cut.

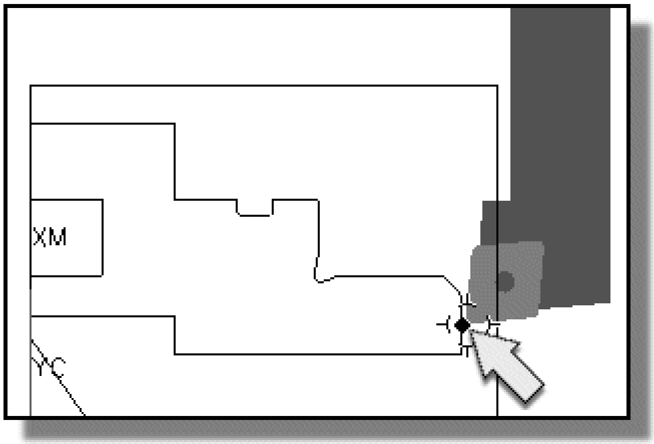
You will face the end of the part. The cut region indicates that the entire OD is going to be cut in this operation. You will use a trim plane to contain the cut region.

☐ Choose **Containment** under Cut Regions.

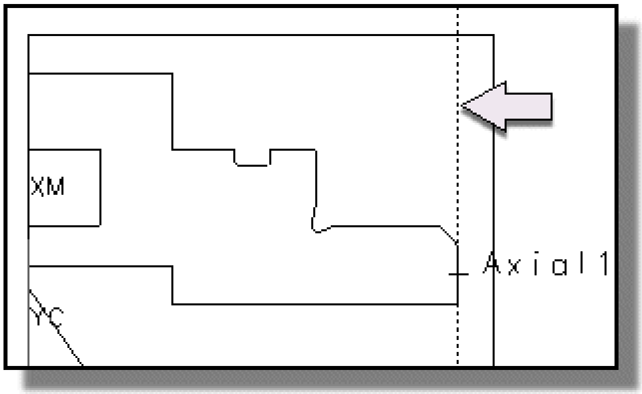
☐ Turn the **Trim** option **on** next to the **Axial 1** button.

☐ Choose the **Axial 1** button.

☐ Select the inferred point on the vertical line as illustrated below.



A point labeled Axial 1 displays on the face of the part and a dashed line indicates the trim plane.

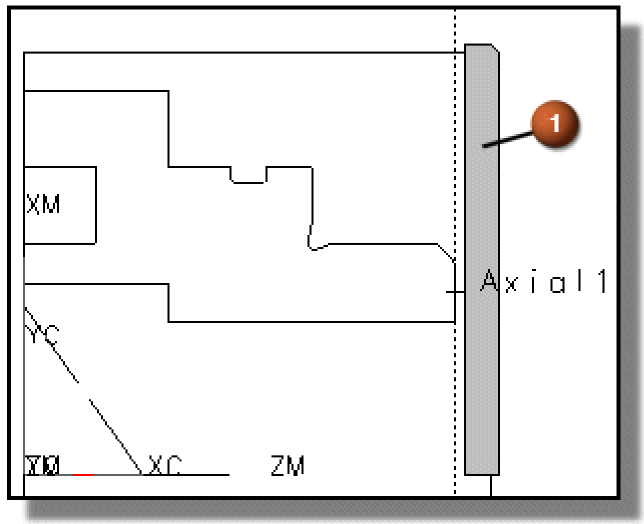


☐ **OK** to complete the geometry containment.

Step 5: Display the cut region.

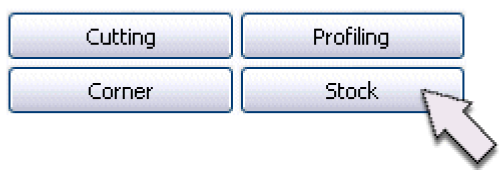
☐ In the Cut Regions area, choose **Display**.

The cut region (1) is contained to the right of the trim plane.

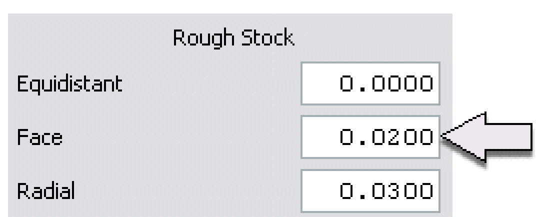


Step 6: Observe the face stock.

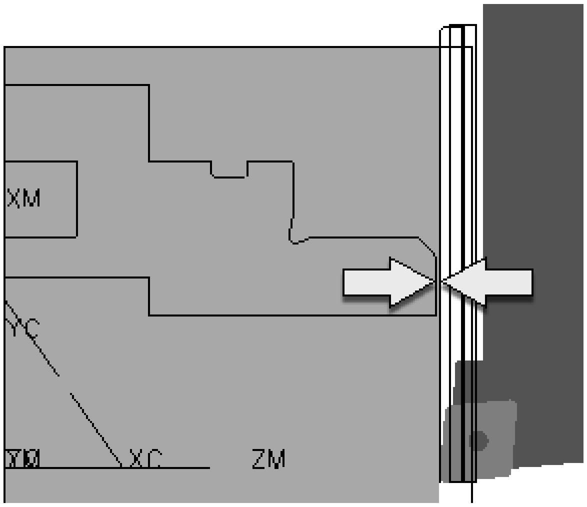
☐ Choose **Stock**.



The Face Rough Stock is currently set to 0.0200.



This is the amount of material that will be left on the face.

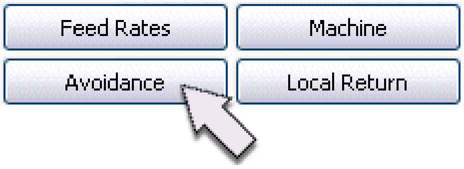


☐ **Cancel** the stock dialog.

Step 7: Define Start and Return points.

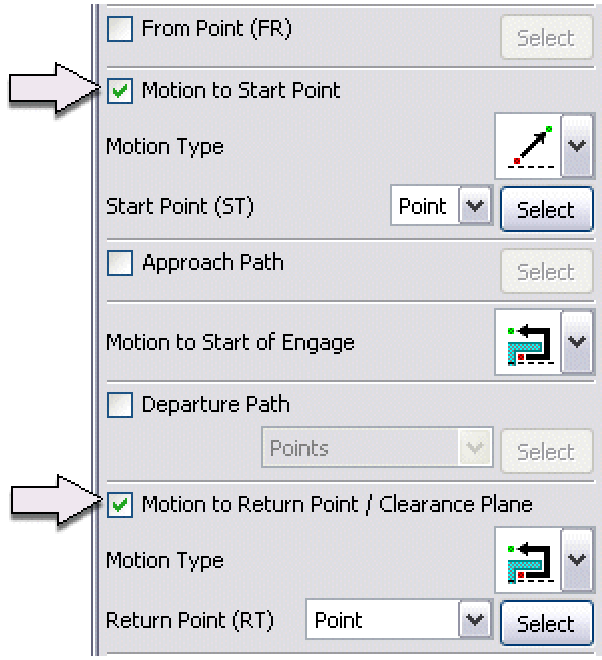
Start and Return points allow the tool to move away from or toward the part between operations. These and other avoidance parameters will be discussed in more detail in the Common Options lesson.

☐ Choose **Avoidance**.



3

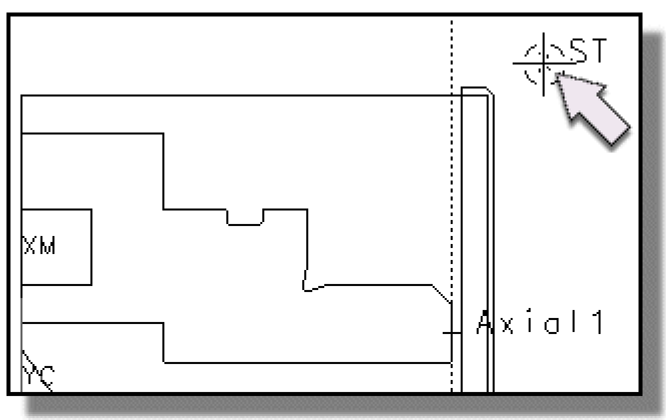
☐ Turn the **Motion to Start Point** and **Motion to Return Point** options **on**.



Motion Type determines how the tool will move between the avoidance points and the part. You will leave both of these options set to Direct. This will allow the tool to take a direct (shortest) path.

☐ Under **Motion to Start Point**, choose **Select**.

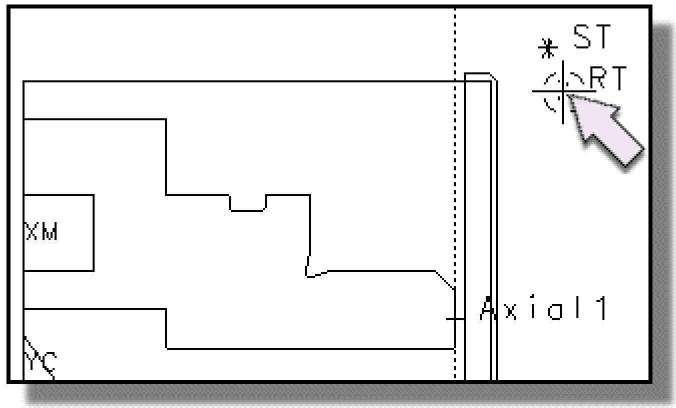
☐ Indicate a point at the approximate screen position illustrated below.



This position defines the Start Point and is labeled ST.

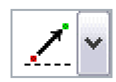
☐ Under **Motion to Return Point**, choose **Select**.

- ☐ Indicate a second point near the first.



This position is labeled RT.

- ☐ Change the **Motion Type** to **Direct**.



- ☐ **OK** to complete the avoidance parameters.

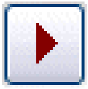
Step 8: Generate the tool path.

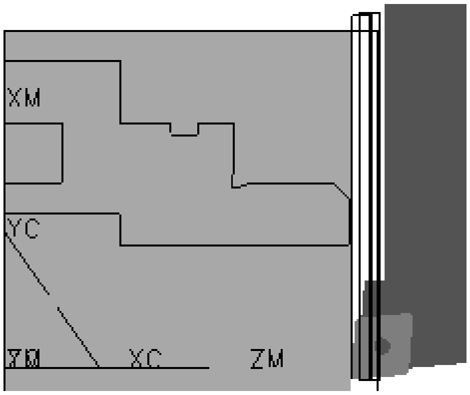
- ☐ Choose the **Generate** icon at the bottom of the dialog.

  The tool cuts the face of the part leaving behind the specified face stock.

- ☐ Choose Verify at the bottom of the dialog. 

- ☐ Turn the **2D Material Removal** option **on**.

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



☐ **OK** to complete the visualization.

☐ **OK** to complete the operation.

☐ Save and close the part.

Summary

Facing removes material by roughing normal to the centerline. The system defines faces based on parameters specified in the Contour Types dialog. Axial cut region containment allows you to restrict the region to be faced while Face Rough Stock defines the amount of material that will be left on the faces. Once generated, the tool path and the in-process workpiece can be displayed using 2D Material Removal.

3

Lesson

4 Verification

Purpose

You will use the Verify Tool Path function to visually validate single and multiple tool paths.

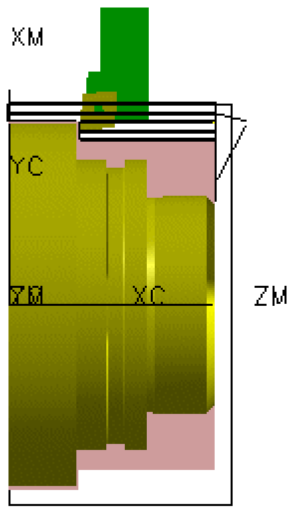
Objective

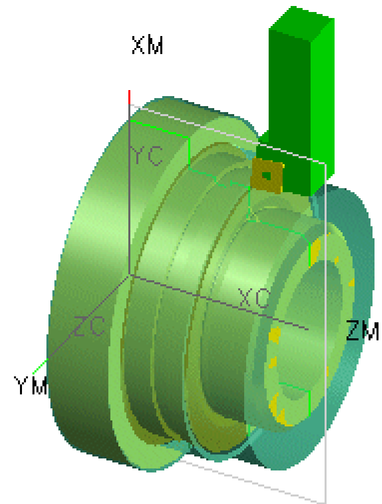
Upon completion of this lesson, you will be able to:

- Display the in-process workpiece as a 2D or 3D static image.
- Display dynamic material removal in both 2D and 3D.
- Display dynamic material removal for a single operation as well as for a sequence of operations.
- Edit display parameters for the tool and the in-process workpiece.

Displaying Cut Motion and Dynamic Material Removal

The Verify Tool Path function allows you to display dynamic material removal for a single operation or a sequence of operations and visually examine the tool movement in both 2D and 3D.

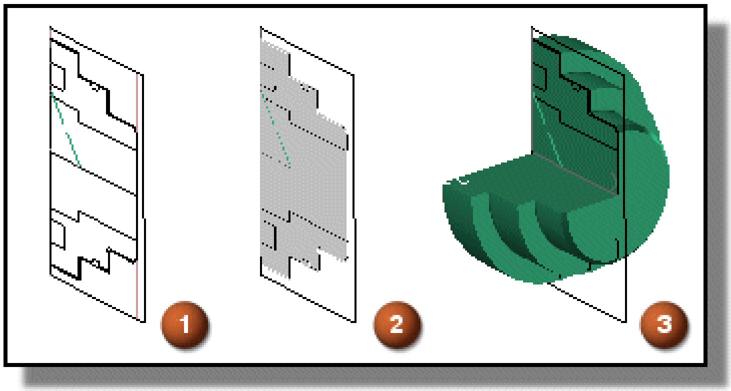




You can control the animation speed of the tool movement, the tool display, and verify that the tool is cutting the correct portions of the material. It is important, for example, to determine if the tool is gouging the part. You can display the tool, the part, and the in-process workpiece in a variety of ways in both 2D and 3D. Verify Tool Path can also report unacceptable conditions such as collisions.

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

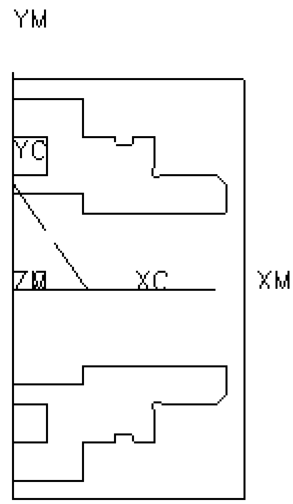
You can display the in-process workpiece as a static image by highlighting the operation in the Operation Navigator and choosing MB3→Workpiece→Show 2D (1), Show filled 2D (2), or Show 3D (3). Since the sequence of operations is important, it is best to select the operation in the Program Order View of the Operation Navigator. If operations are reordered, the system recalculates the in-process workpiece.



Activity: Verification Using Show 2D and 3D Options

Step 1: Open the part.

☐ Open the **tmp_verify_mfg_1** part.

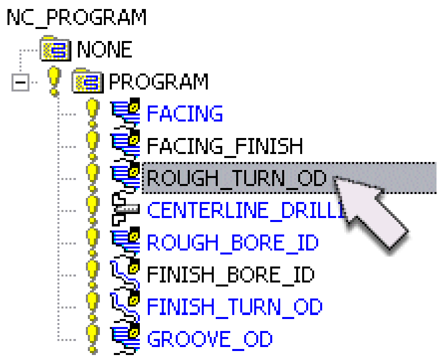


☐ Choose **Start→Manufacturing**.

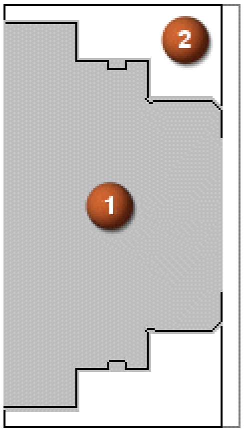
Step 2: Display the in-process workpiece in 2D.

☐ Display the Program Order View of the Operation Navigator and expand the objects.

☐ Highlight **ROUGH_TURN_OD** and **MB3→ Workpiece →Show filled 2D** .



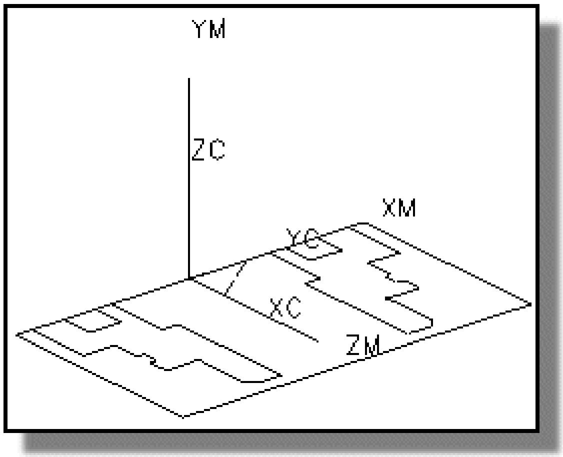
The shaded area (1) represents the in-process workpiece. The outlined area (2) represents the cut region.



☐ Refresh the graphics display.

Step 3: Display the in-process workpiece in 3D.

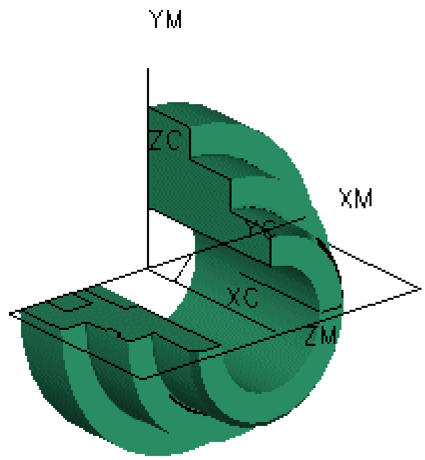
☐ Change to a Trimetric view.



☐ Highlight **ROUGH_BORE_ID** and **MB3→ Workpiece →Show Spinning 3D**.

The in-process workpiece is displayed in 3D.

☐ Choose the **Shaded** icon. 



Step 4: Compare the finished part to the in-process workpiece.

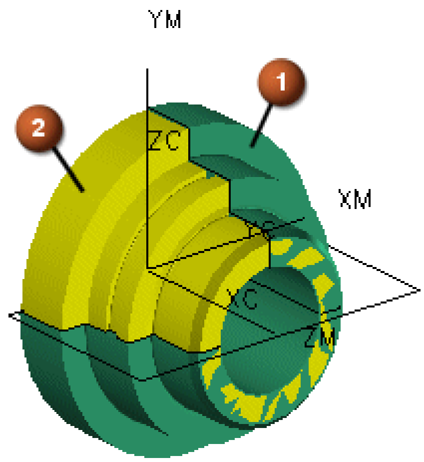
You will begin by displaying the finished part.

☐ Choose **Format→Layer Settings**.

☐ Double-click on layer **110**.

☐ Choose **OK**.

The in-process workpiece (1) is superimposed on the finished part (2).



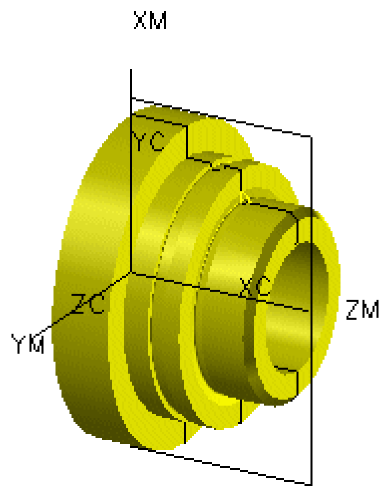
☐ Refresh the graphics window.

☐ Leave this part open. You will continue using it in the next activity.

Activity: Verification Using Tool Path Visualization

Step 1: Visualize 2D material removal for a single operation.

- ☐ Rotate  the part to the approximate viewing angle illustrated below.

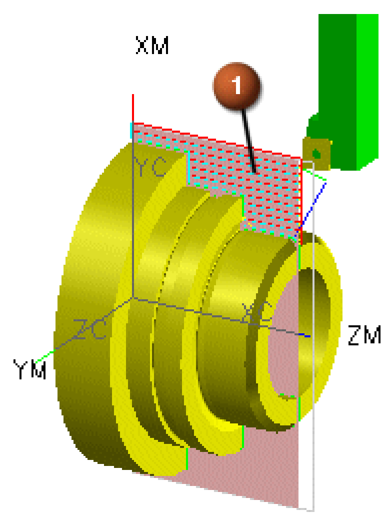


- ☐ In the Operation Navigator, highlight **ROUGH_TURN_OD**.

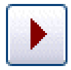
- ☐ **MB3→ Tool Path → Verify** or choose the **Verify Tool Path** icon.


- ☐ Make sure the **Replay** tab is active.

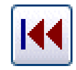
- ☐ Turn the **2D Material Removal** option **on**. This will display the blank material in a 2D plane (1).



- ☐ At the bottom of the dialog, set the **Animation Speed** to **7**.

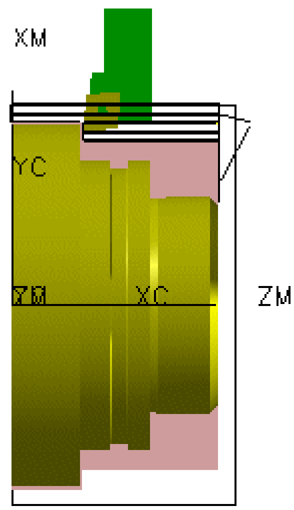
- ☐ Choose **Play**. 

- ☐ Change to a Top view.

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

- ☐ Turn the **2D Material Removal** option **off** and then **on** again. This will redisplay the blank material.

- ☐ Choose **Play**.



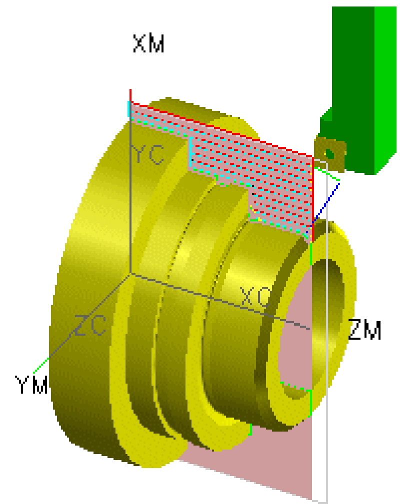
- ☐ Choose **Reverse Play**. 

- ☐ Choose **OK**.

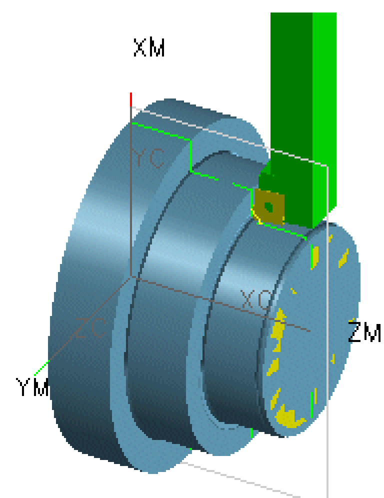
4

Step 2: Visualize 3D material removal for a single operation.

- ☐ Once again, rotate the part to the approximate viewing angle illustrated below.



- ☐ Choose the **3D Dynamic** tab.
- ☐ Choose **Play**.

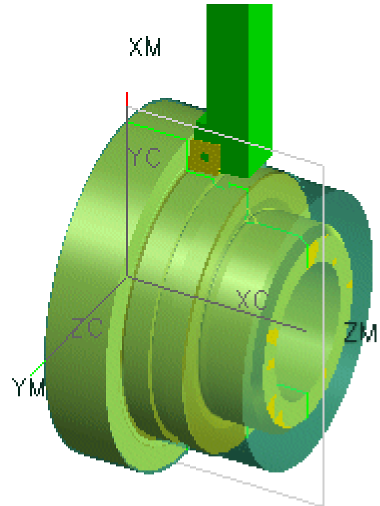


- ☐ Choose **Reset**.
- ☐ Choose **Display Options**.
- ☐ Set the **IPW Color** option to some other color that will contrast the part color.
- ☐ Set the **IPW Translucency** option to about **50**.

  This will allow you to see the part through the in-process workpiece.

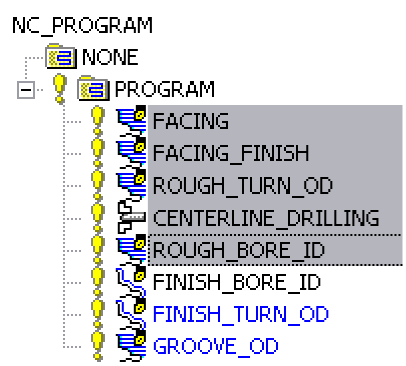
☐ **OK** the 3D Dynamic Options dialog.

☐ Choose **Play**.



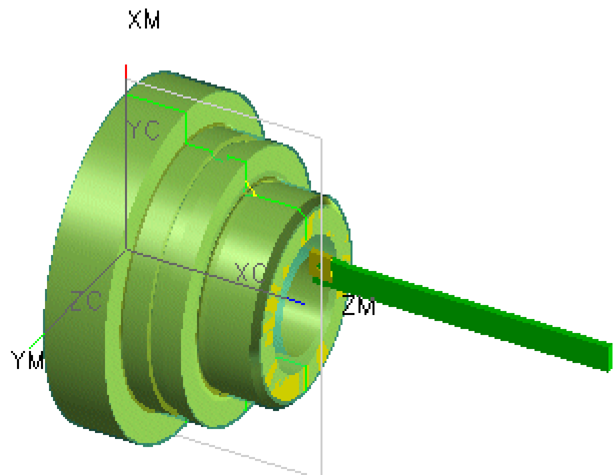
Step 3: Visualize 3D material removal for a sequence of operations.

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



☐ Set the **Animation Speed** to **2**.

☐ Choose **Play**.



The tool paths are replayed in their entirety in sequence.

Step 4: Visualize 2D material removal for a sequence of operations.

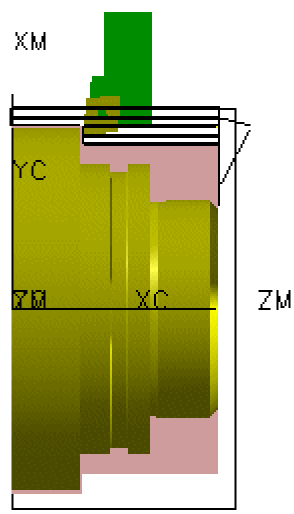
☐ Choose **Reset**.

☐ Change to a Top view.

☐ Choose the **Replay** tab.

☐ Turn the **2D Material Removal** option **on**.

☐ Choose **Play**.



☐ **OK** the Tool Path Visualization dialog.

☐ Do not save the part.

☐ Save and close the part.

Summary

The Verify Tool Path function allows you to display dynamic material removal for a single operation or a sequence of operations and visually examine the tool movement in both 2D and 3D. You can also display the in-process workpiece of any operation as a static image in both 2D and 3D.

4

Lesson

5 Common Options

5

Purpose

The purpose of this lesson is to introduce you to many of the common parameters you will use in creating tool paths. These parameters, which include stock, feeds and speeds, avoidance moves, engage and retracts, and feed rate control, are common to all or most turning operations.

Objective

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

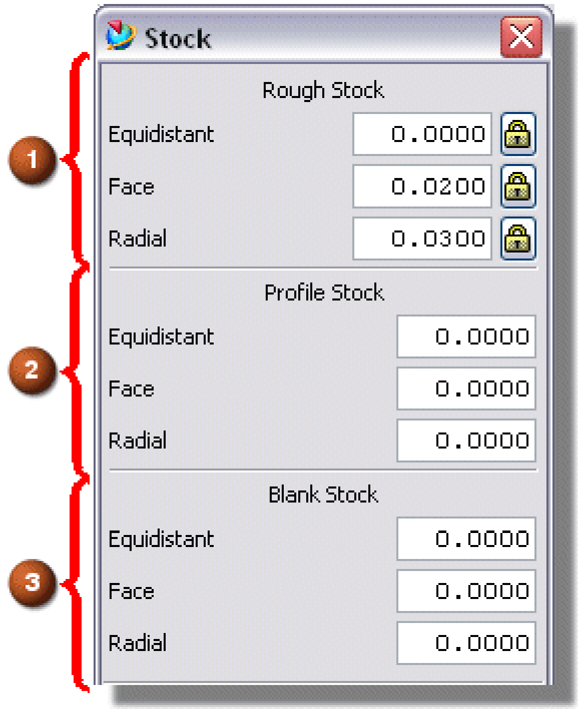
- Define From, Return, and Start points.
- Define rough stock.
- Define grind stock.
- Define Engages and Retracts.
- Define feed rates.

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			

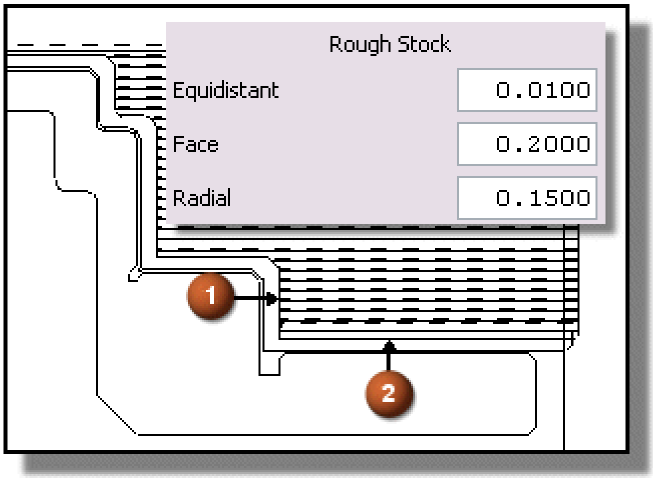
Stock

The Stock function allows you to leave material on the part. Rough Stock (1) leaves material after roughing. Profile Stock (2) leaves material after profiling. Blank Stock (3) adds material to the work piece.



For each type of stock (rough, profile, blank), separate stock values can be applied to faces and diameters. Face stock and Radial stock are applied in addition to the Equidistant stock. Equidistant stock adds the same stock to all elements. Face stock adds stock to faces only. Radial stock adds stock to diameters only.

In the following figure, the total face stock (1) is equal to the Equidistant stock plus the specified Face stock. The total diameter stock (2) is equal to the Equidistant stock plus the specified Radial stock.



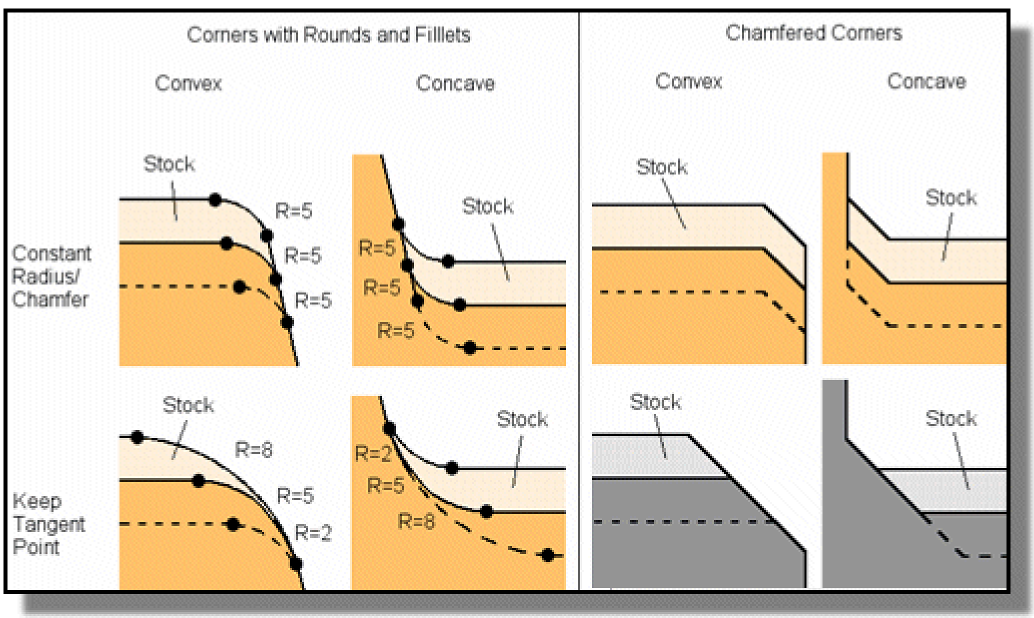
Part Boundary Parameters

The following options control part boundaries and are available by editing the part boundary of a TURN_WORKPIECE object.

Tolerance Offset

Tolerance Offset lets you control the dimensions of the design model of your part in manufacturing.

This feature is useful if you require optimal surface control for the part you want to machine. Use this feature when you need to consider tolerance values specified for the part you want to machine.

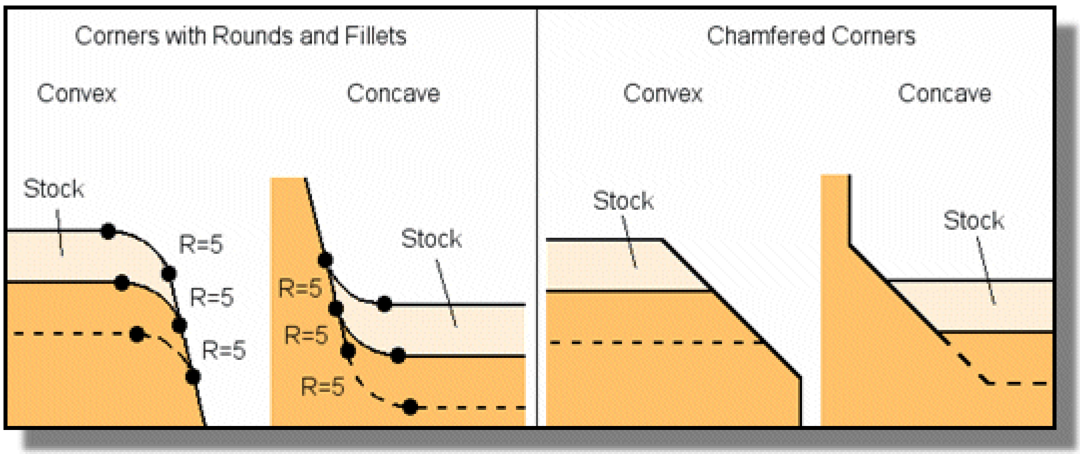


5

General Offset

General Offset lets you define stock or offset values for individual surface areas of your manufacturing part.

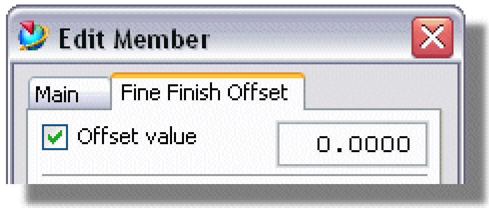
This feature is useful if you require optimal surface control for the part you want to machine. Use this feature when you need to consider individual stock/offset values for certain boundary members of your part.



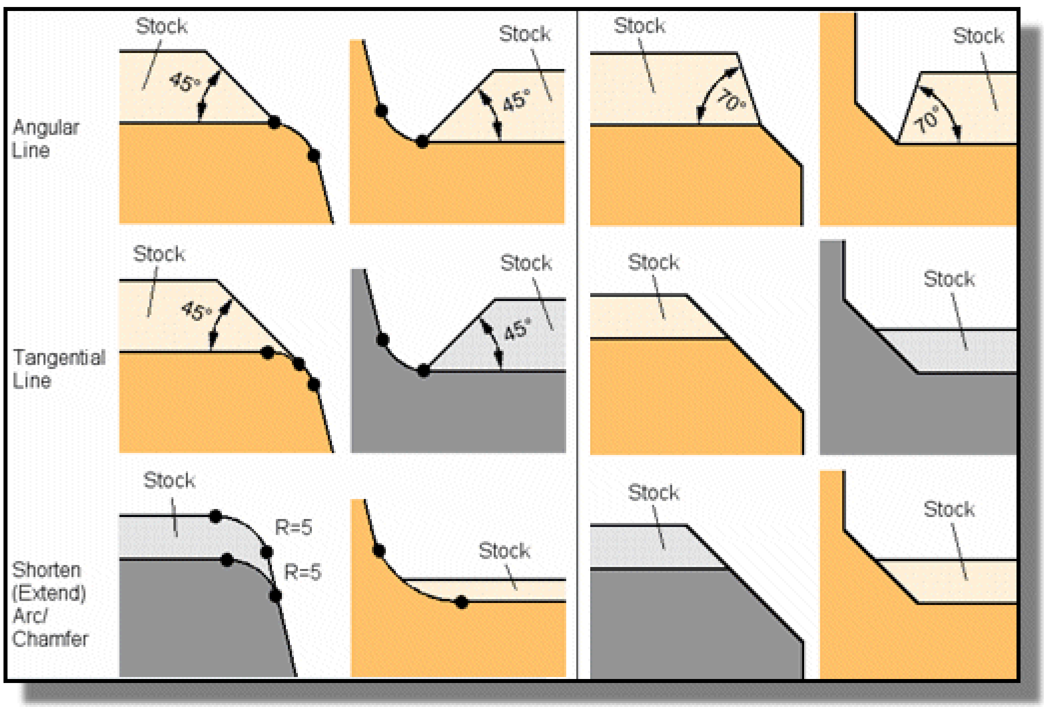
5

Fine Finish Offset

Fine Finish Offset provides corner control options specialized to leave proper finish or grind stock on selected portions of the part surface.

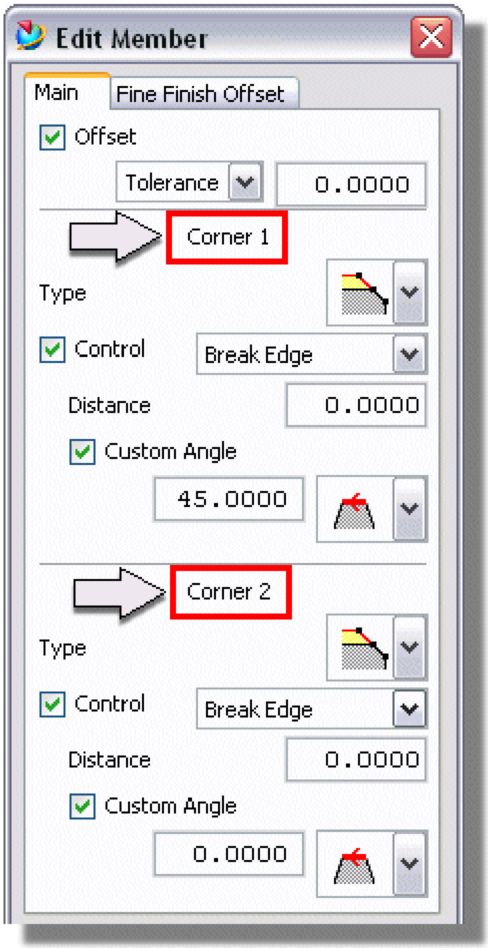


This feature is useful if you want optimal surface control for the part you want to machine. Use this feature when you need to leave finish or grind stock on your part surface.



Corner 1/Corner 2

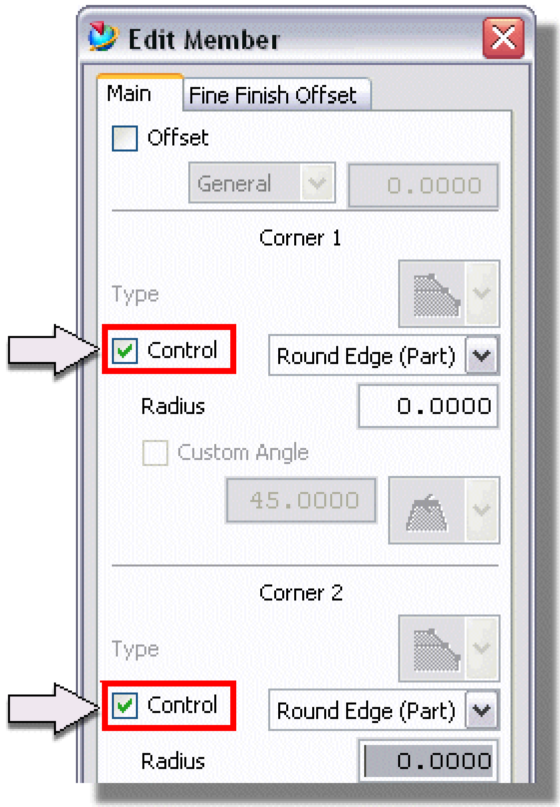
Corner 1/Corner 2 options let you control the offset/stock shape and corner treatment at either corner of the selected boundary member individually. Use this feature if you want to specify a corner control type such as Constant Radius/Chamfer for Tolerance Offset or Angular Line for Fine Finish Stock with Corner Cleanup.



5

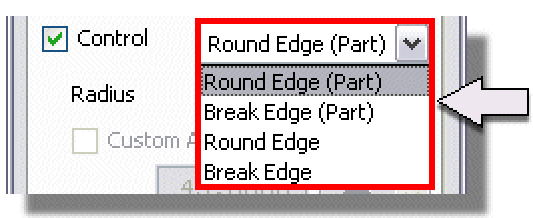
Control

The Control option menu allows you to specify additional corner control for an individual corner of the selected boundary member. It may be combined with offset/stock definition but can also be independently defined. Use this feature when you need a specific corner control option either at Corner 1 or Corner 2 of your selected boundary member.



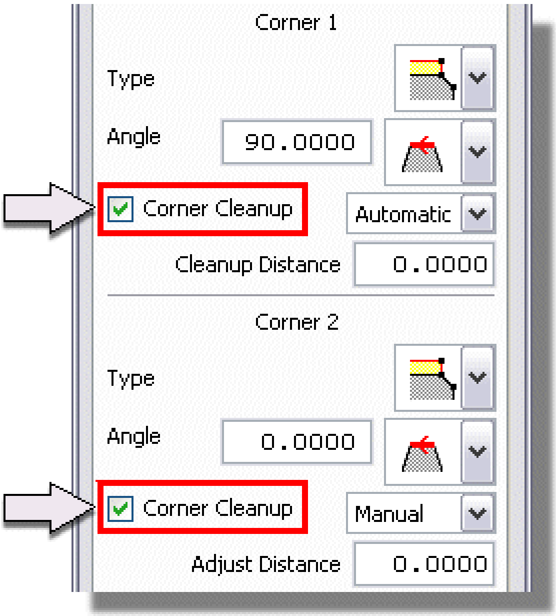
Create Chamfer/Round Control

Create Chamfer/Round Control allows you to create a chamfer or a round at a selected corner in your boundary. It may be combined with offset/stock definition but can also be independently defined. Use this feature when you need a specific corner control option either at Corner 1 or Corner 2 of your selected boundary member.

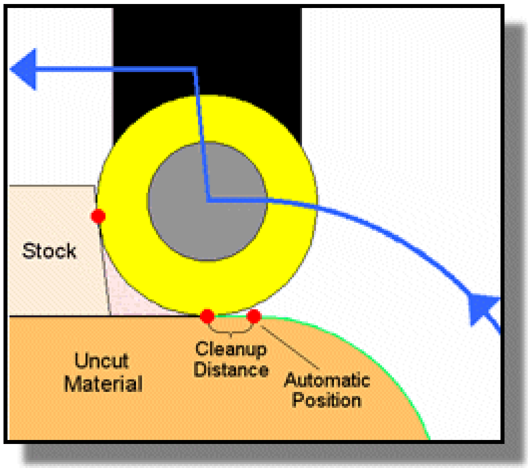


Corner Cleanup

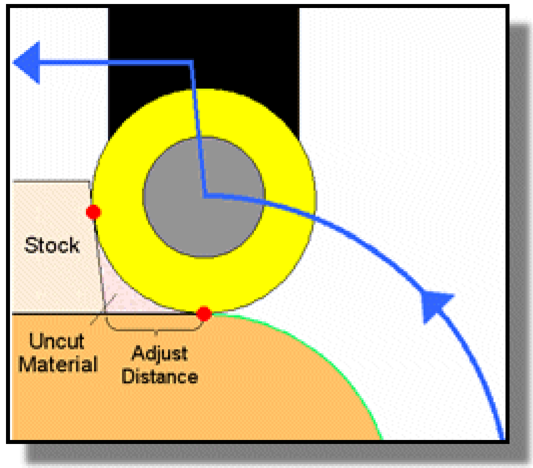
Corner Cleanup ensures that the tool's cutting motions leave a clean round or chamfer surface while respecting the angular corner of a Fine Finish Offset. You should use this feature when selecting the Angular Line option in Fine Finish Offset/Stock for diameter surfaces that have blends or chamfers next to them.



Automatic Corner Cleanup allows the contact point of the cutter to cut all the way to the tangent point of the blend before cutting along the chamfer. This assures that grind stock is never left on the blend and that the round is clean. As a result, the tool may cut into the chamfer before moving up and over the chamfer. You can specify a Cleanup Distance to cut beyond the tangent point.



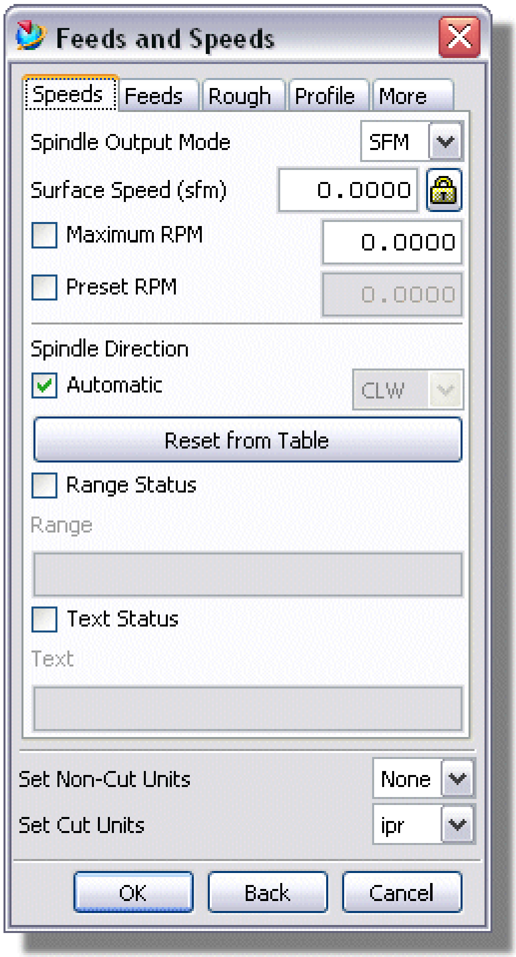
Manual Corner Cleanup respects the chamfer, but can leave unwanted grind stock on the blend near the tangent point. The Adjust Distance option can correct this by allowing the tool to cut further along the blend.



5

Feed Rates

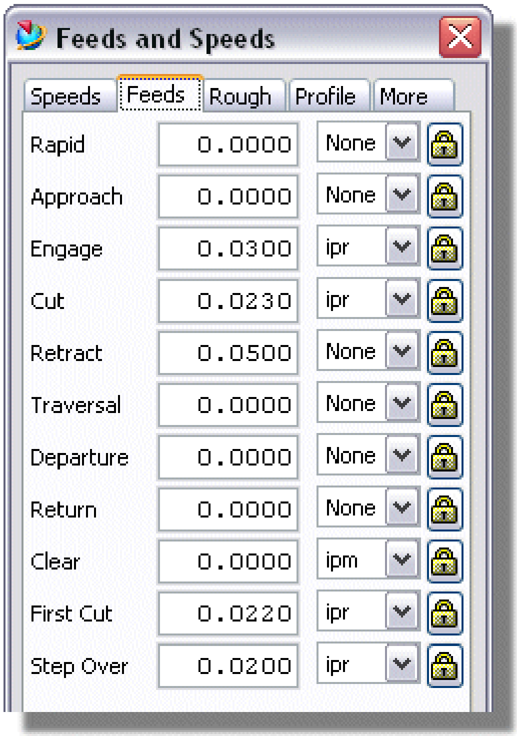
Feed Rate is the rate at which the tool advances. The Feed Rate submenu contains a variety of feed rate control parameters which allow for fine-tuning of the Turning processor's results. Speeds refer to spindle and surface speeds.



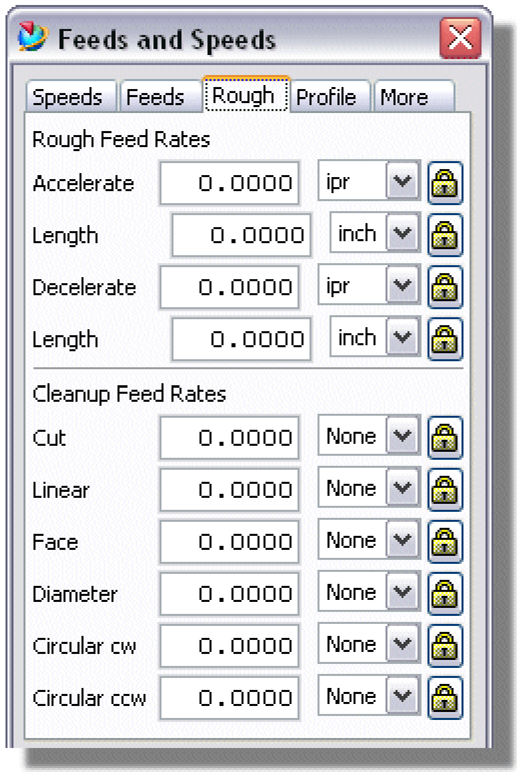
Spindle Output Mode	The method in which the Feed Rate is specified. RPM (Revolutions per Minute) refers to the number of times the spindle turns in 60 seconds. SFM/SMM (Surface Feet/Meters per Minute) refers to the distance in feet or meters that the spindle travels in 60 seconds measured at the surface of the part. Here the spindle speed adjusts its speed to the amount of material to be removed in order to ensure a constant surface speed.
Surface Speed	The speed found on the surface of the part at any moment in the machining process
Maximum RPM	Defines the maximum RPM during the SFM or SMM motion.
Preset RPM	Defines the RPM that will be output prior to entering the SFM or SMM mode.
Spindle Direction	Determines the correct spindle rotation direction based on the location of the cut region with respect to the spindle axis of rotation and whether the insert is on the top side or the underside of the tool holder. Defaults to Automatic
Set Non-Cut Units	initializes the feed rate unit for all movements which do no cut through material such as traversals and returns. This option sets the units of all non-cut moves at once so you don’t have to individually change the unit of each feed rate.
Set Cut Units	Initializes the feed rate unit for all movements which cut through material such as engage and cut moves. This option sets the units of all cutting moves at once so you don’t have to individually change the unit of each feed rate.
Reset from Table	Allows the system to recommend appropriate feeds and speeds based on the user specified part material, tool type and material, cut method, and cut depth parameters. The system extracts an appropriate Surface Speed from the library (maintained by system administration) based on the user input. The feed rate values, which appear under the Feeds tab, are then calculated.

Not all conditions will find a feed and speed solution in the library.

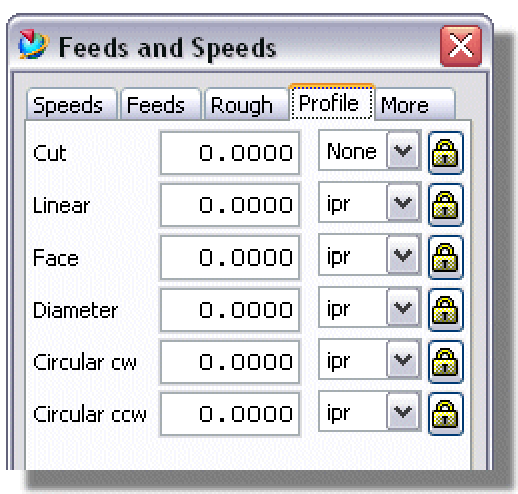
Feed rates can be specified for each type of cutting and non-cutting move. A feed rate of zero indicates that the move uses a value specified elsewhere. For example, an Engage feed rate of zero causes the tool to engage at the Cut feed rate. An Approach feed rate of zero causes the tool to move at the Rapid feed rate if the engage method is specified; otherwise, the Engage feed rate is used.



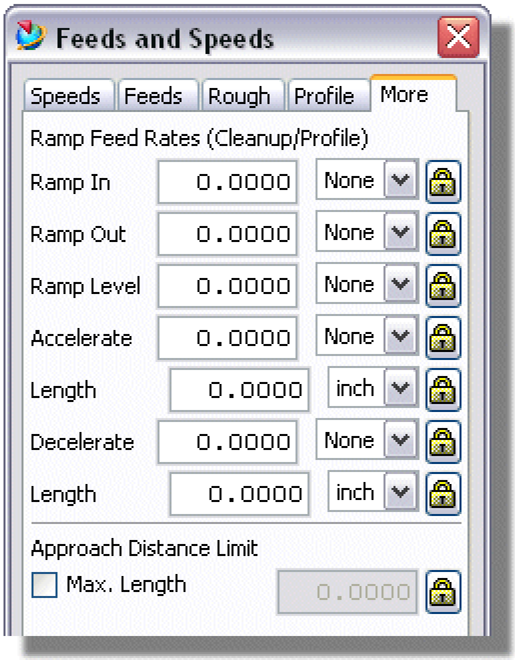
Rough Feed Rates are available to assign specific Feed Rate values to motions carried out in Roughing operations only. Acceleration and deceleration feed rates allow feed rate changes at the beginning and at the end of each singular rough cut, for example when approaching a shoulder. This feature helps to prevent insert breakage due to chips accumulating at the end of a rough cut close to a shoulder or to protect a forged part from breakage in those moments where the cutter enters or leaves the material.



Profile Feed Rates are available to assign specific feed rate values to motions carried out during finishing or profiling passes:

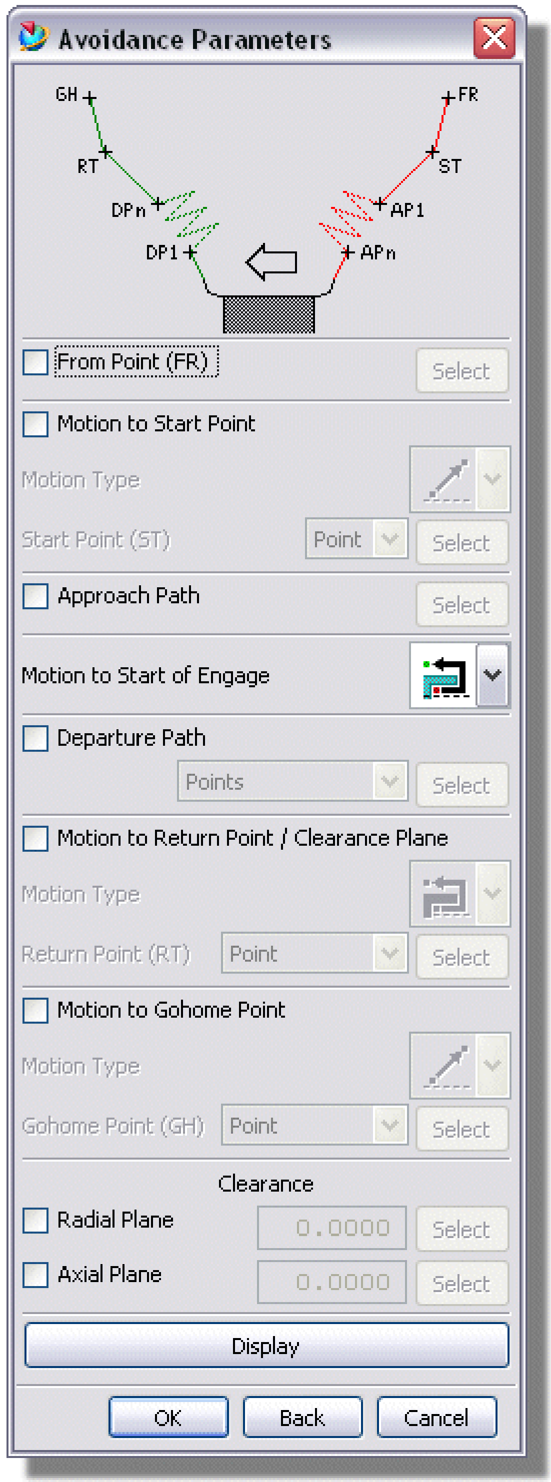


Ramp Feed Rates pertain to the cutting motions that are not made along the part profile in cleanup, finishing or additional profiling. If any of these cleanup feed rates are set to 0, those movements are carried out at the linear feed rate from the Profile/Finishing tab control feed rates for additional profiling moves.



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.



5

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
Motion to 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.
Motion to Return Point / Clearance Plane	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.
Motion to 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 work piece along the radial or axial positions, for positioning moves to the Start and Return points

Creating Associative Points

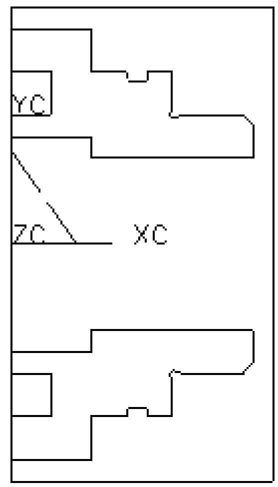
Points, Vectors, and Planes (usually used for avoidance geometry) can be created as associative geometry so that when the referenced geometry is modified, the associated geometry is also modified.

Activity: Rough 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 and rename the part.

☐ Open the **tmp_common_mfg_1** part.



☐ Save the part as *****_common_mfg_1**, where *** represents your initials.

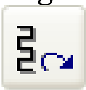
☐ Choose **Start→Manufacturing**.

Step 2: Replay the existing operation.

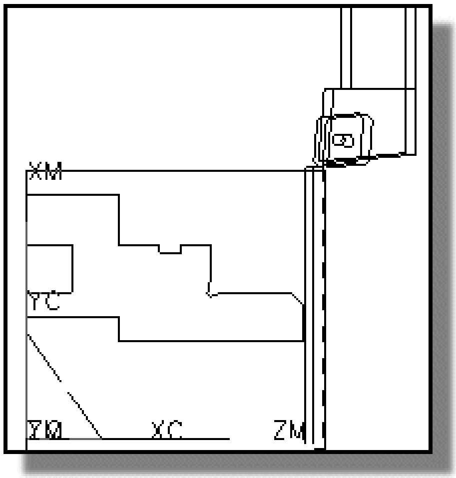
☐ In the Operation Navigator, display the Program Order View and expand the objects.

5

☐ Highlight **ROUGH_FACE** and choose the **Replay Toolpath** icon.



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



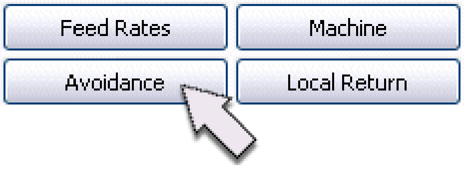
5

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

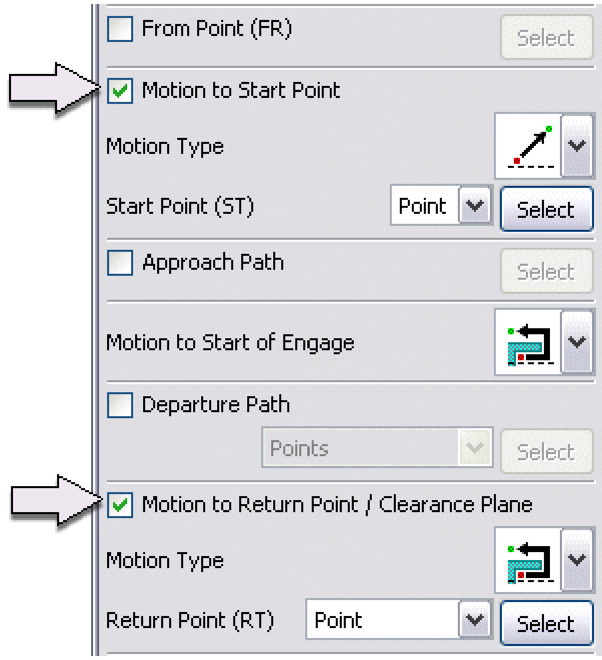
☐ In the Operation Navigator, double-click **ROUGH_FACE** to edit the operation.

You are going to specify several Avoidance settings.

☐ Choose **Avoidance**.



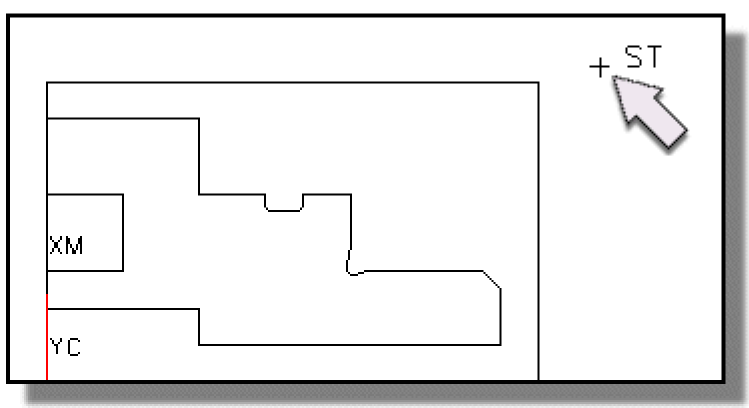
☐ Turn the **Motion to Start Point** and **Motion to Return Point** options **on**.



☐ Under each option, set the **Motion Type** to **Direct**.

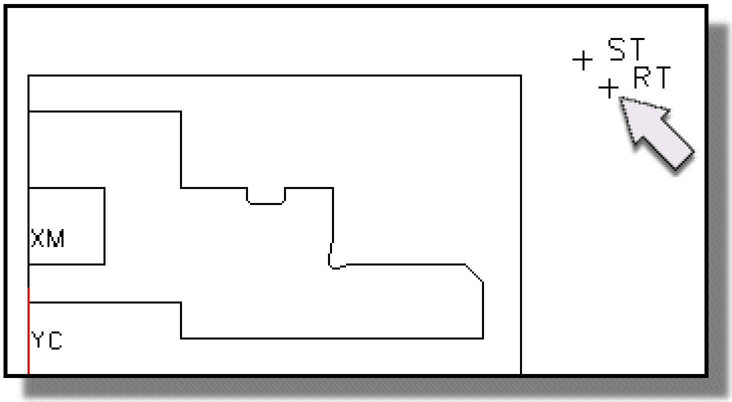
Motion Type determines how the tool will move between the avoidance points and the part. You will leave both of these options set to Direct. This will allow the tool to take a direct (shortest) path.

☐ Choose **Select** under **Motion to Start Point** and click the approximate screen position illustrated below.



5

☐ Choose **Select** under **Motion to Return Point** and click near the Start Point.



The Start and Return Point indicators are placed on top of each other.

☐ Turn the **From Point** option **on**.

☐ Choose **Select**.

In many cases, you may have a specific From point location that you use with each machine.

☐ Key in **6.00** in the XC field.

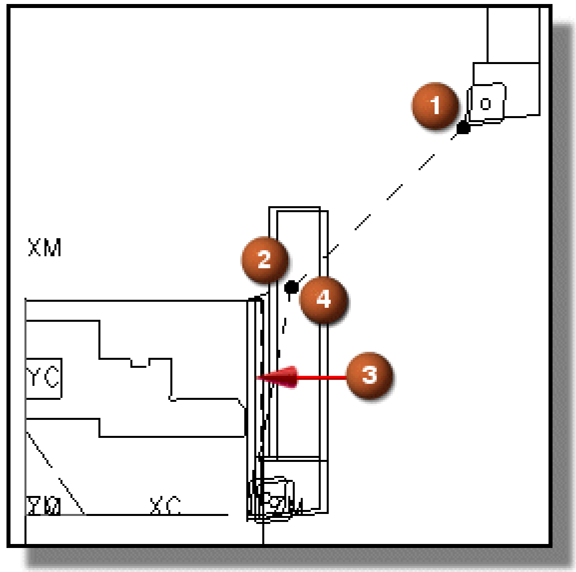
☐ Key in **5.00** in the YC field.

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

5

☐ **Generate** the tool path. 

The tool moves from the From Point (1) to the Start point (2), cuts the part (3), and then moves to the Return point (4).



Notice that creating avoidance points allowed the tool path to generate without displaying the **Last Tool Position** dialog.

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

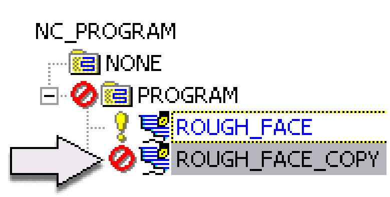
Step 4: Copy and rename the operation.

You are going to copy the operation and then edit the Method and the Stock. When you copy the operation, the Avoidance parameters you just defined will already be set.

☐ Highlight **ROUGH_FACE** 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 still highlighted in the Operation Navigator, use MB3 to **Paste** the copied operation.



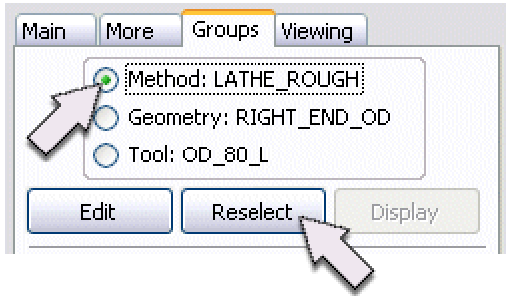
☐ Make sure that the operation that you just copied is highlighted, then use **MB3→Rename** to rename the operation to **FINISH_FACE**.

Step 5: Edit the Method.

☐ Double-click **FINISH_FACE** to edit the operation.

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

☐ Choose **Method** and **Reselect**.



☐ Set the **Method** option to **LATHE_FINISH**.

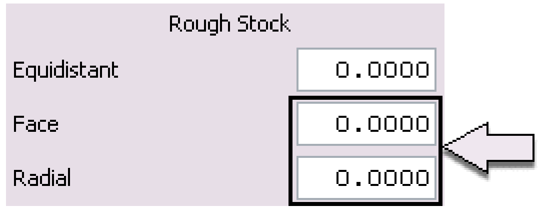
☐ **OK** to accept the Method.

Step 6: Edit the Stock values of the operation.

☐ Choose the **Main** tab.

☐ Choose **Stock**.

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

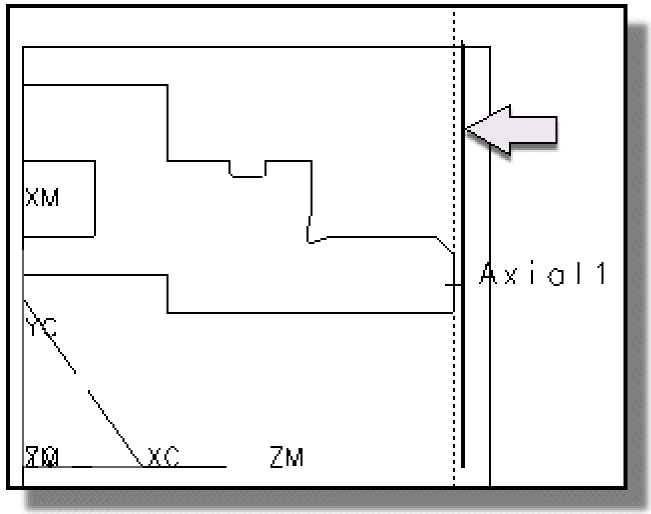


☐ **OK** to accept the stock.

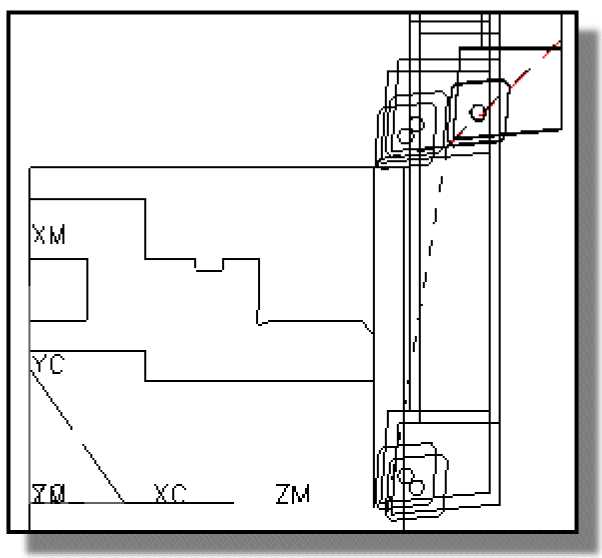
Step 7: Display the Cut Region.

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

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



☐ **Generate** the tool path. 



The tool path finishes the face of the part.

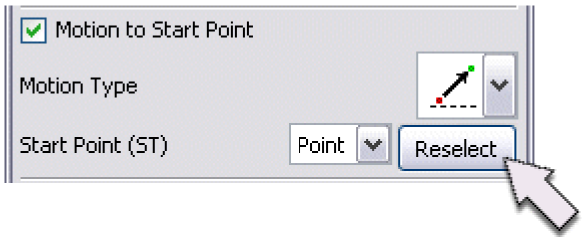
☐ Refresh the graphics display.

Step 8: Edit the Start point.

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

☐ Choose **Avoidance**.

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

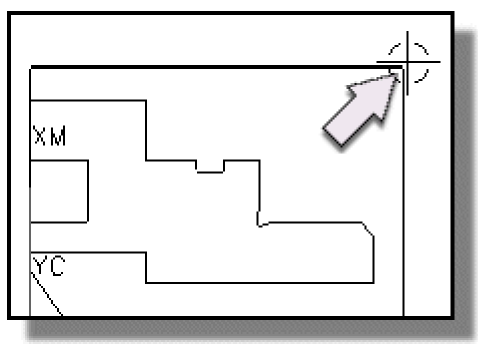


This Start point will be a smart point. You will associate the Start point to the end point of the blank. If the blank changes, the Start point will also change.

☐ Set the **Offset** option to **Rectangular**.

☐ Choose the **End Point** icon.

☐ Select the end point of the horizontal or vertical line.



☐ Key in **1.00** in the XC field.

☐ Key in **.25** in the YC field.

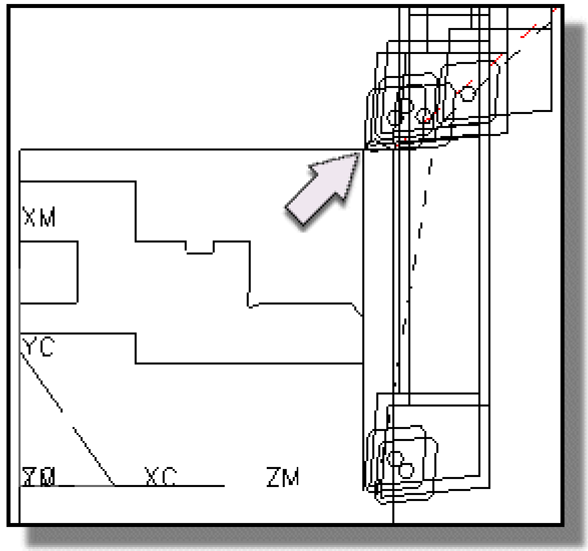
☐ **OK** to accept the Start point.

☐ **OK** to accept the avoidance parameters.

5

☐ **Generate** the tool path.

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



5

Step 9: Removing the From point.

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

☐ Choose **Avoidance**.

☐ Turn the **From Point** option **off**.

☐ **OK** to accept the avoidance parameters.

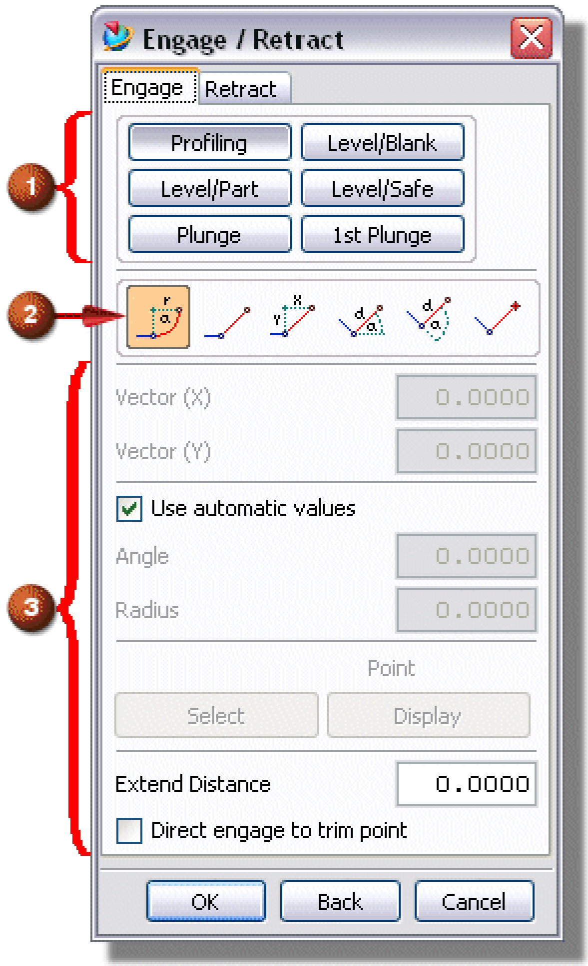
☐ **Generate** the tool path.

☐ **OK** to complete the operation.

☐ Save and close the part.

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 retract motions to safely move the tool away from the work piece at the end of each machining pass. For each engage or retract situation (1), the selected strategy (2) works together with the specified parameters (3).



5

The engage and retract conditions are defined as follows:

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. The table below 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	

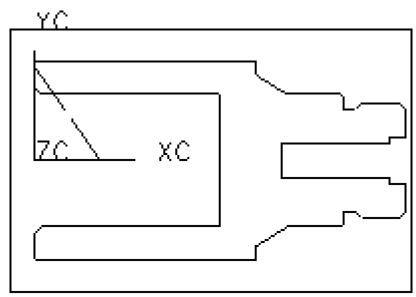
5

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 **tmp_common_mfg_2** part.



☐ Save the part as *****_common_mfg_2**, where *** represents your initials.

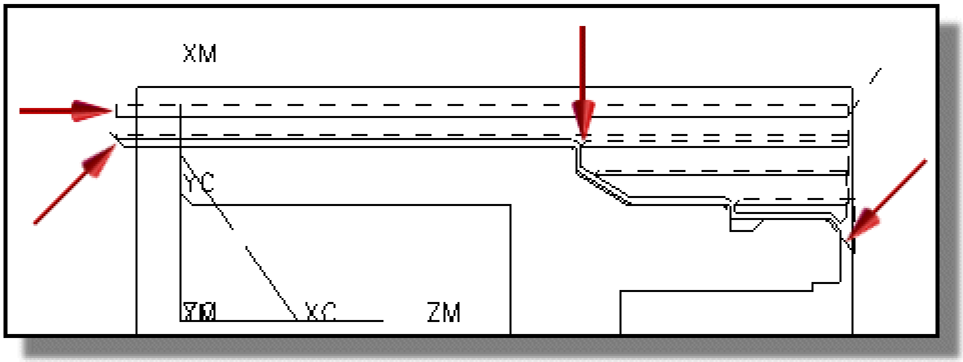
☐ Choose **Start→Manufacturing**.

Step 2: Replay the roughing OD operation.

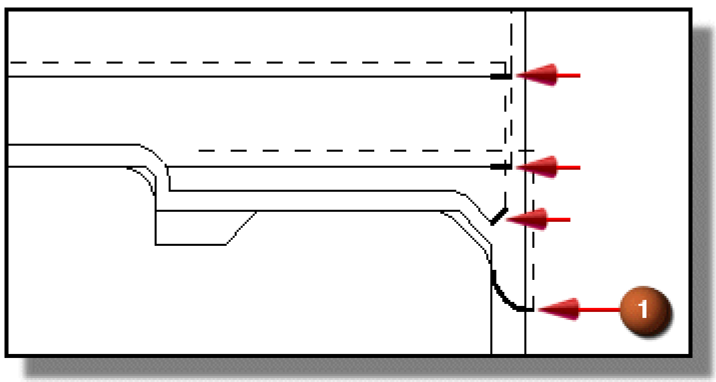
☐ Display the Program Order view of the Operation Navigator and expand the objects.

☐ Highlight **ROUGH_TURN_OD** and replay the tool path.

Notice the default engage and retract moves.



All engages and retracts are Auto Linear except those for the Profile cut which uses Auto Circular (1).



You will now change some of the engages and retracts.

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

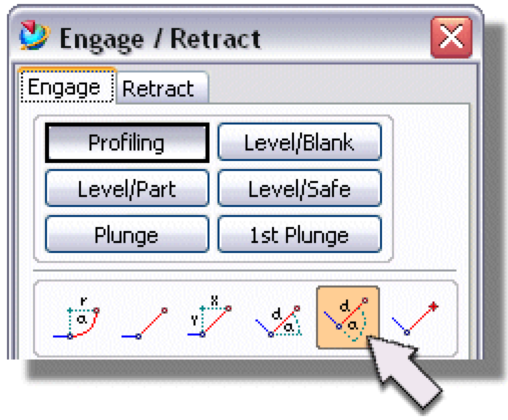
You will change the engage move to use the Relative Linear strategy. The tool will use this move when engaging the part material on a profile pass.

☐ 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.

☐ Choose **Relative Linear**.



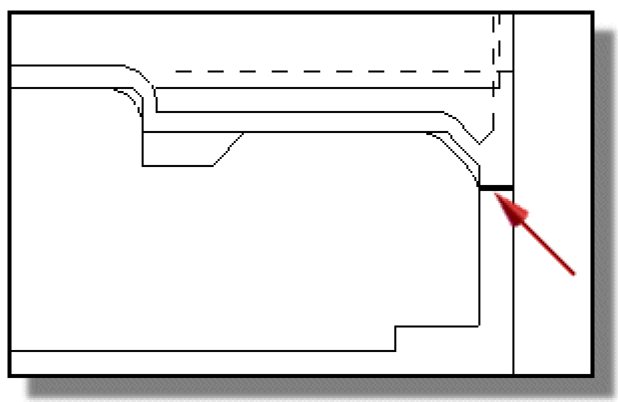
☐ Key in **90** in the Angle field.

☐ Key in **0.050** in the Distance field.

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

☐ **Generate** the tool path.

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



5

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**.

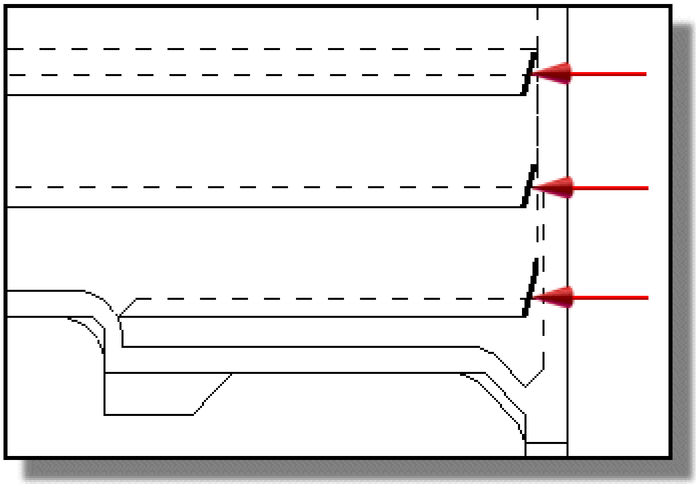
☐ Choose **Level/Blank**.

The default engage for Level/Blank is Auto Linear.

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

- ☐ Key in **255** in the Angle field.
- ☐ Key in **0.050** in the Distance field.
- ☐ Choose **OK** to return to the ROUGH_TURN_OD dialog.
- ☐ **Generate** the new tool path.

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



5

Step 5: Create a Retract move.

- ☐ Choose **Engage/Retract**.
- ☐ Choose the **Retract** tab.
- ☐ Be sure **Profiling** is selected.
- ☐ Choose the **Vector** icon.
- ☐ Key in **0.00** in the Vector (X) field.
- ☐ Key in **0.15** in the Vector (Y) field.

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

- ☐ Choose **Level/Part**.
- ☐ Choose the **Angle/Distance** icon.
- ☐ Key in **90** in the Angle field.
- ☐ Key in **0.10** in the Distance field.
- ☐ Choose **OK** to return to the ROUGH_TURN_OD dialog.

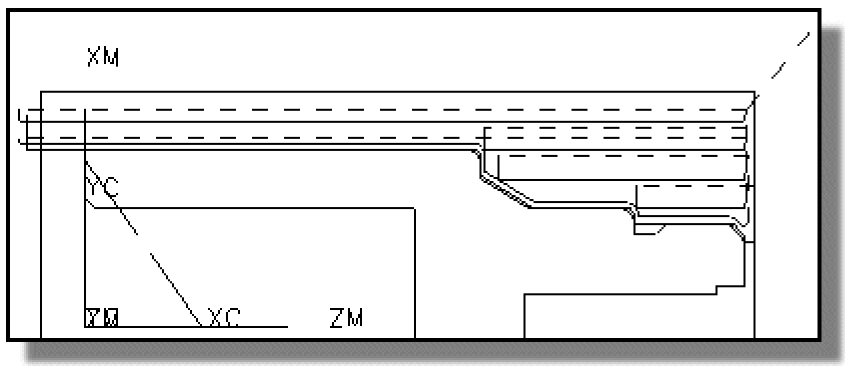
☐ **Generate** the tool path.

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.

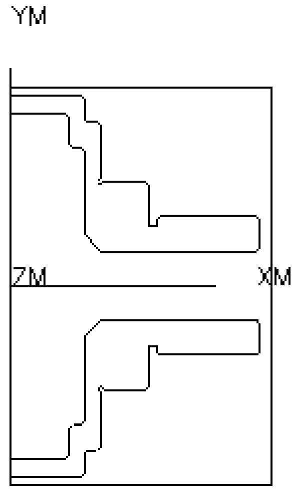
☐ Now try some of the other Engage and Retract strategies, such as To or From a Point. Examine the results.

☐ Save and close the part.

Activity: Applying Grind Stock

In this activity you will specify a Fine Finish Offset and a Tolerance Offset. Fine Finish Offset allows you to leave finish or grind stock on your part surface. Tolerance Offset allows you to achieve localized mid-tolerance part size control.

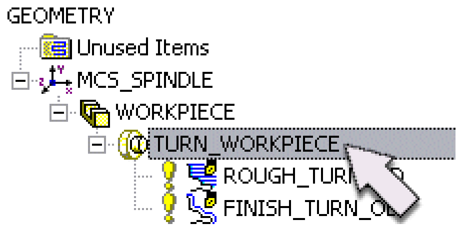
Step 1: Open the **tmp_common_mfg_3** part and start the Manufacturing application.



Step 2: Edit a boundary member.

☐ Display the Geometry view of the Operation Navigator and expand the objects.

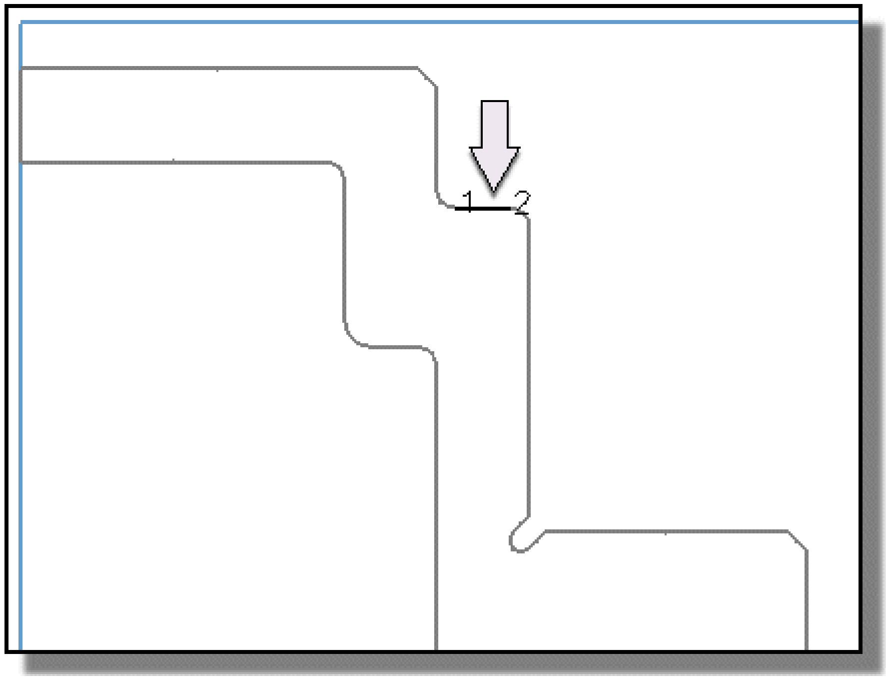
☐ Double-click TURN_WORKPIECE in the Operation Navigator to edit the object.



☐ Choose **Edit** in the TURN_BND dialog.

☐ Choose **Edit** in the Part Boundary dialog.

☐ Select the boundary member illustrated below.



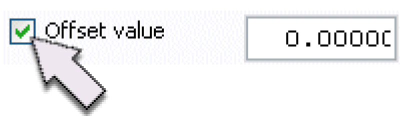
This is the member to which you will add Fine Finish and Tolerance Offsets.

Step 3: Define a fine finish offset.

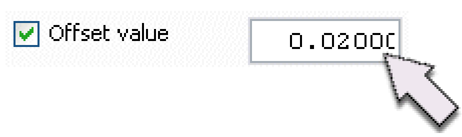
You will define a Fine Finish Offset to leave grind stock on the part diameter.

☐ Choose the **Fine Finish Offset** tab.

☐ Turn the **Offset Value** option **on**.



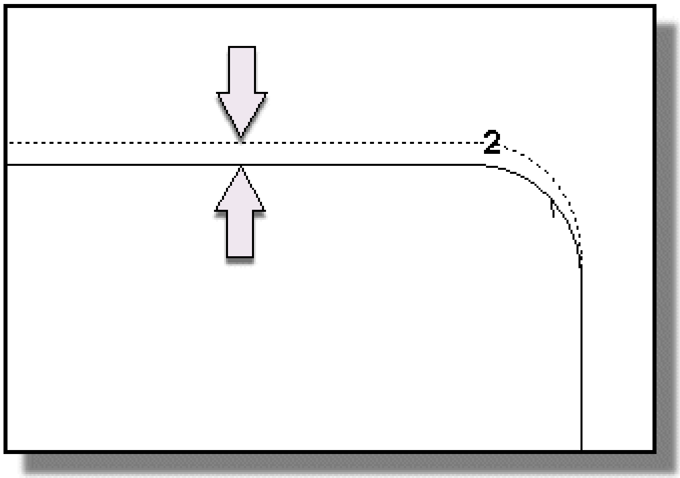
☐ Key in **0.020** in the Offset Value field.



This value is deliberately large so you can graphically see the offset. A more realistic offset may be 0.005.

5

☐ **Zoom In** on Corner 2 and choose **Preview** to see the offset.

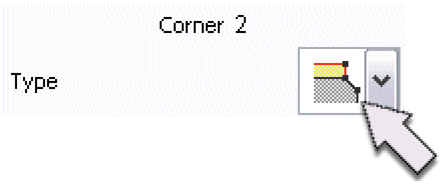


The ends of the boundary member being edited are numbered (1, 2) and are referred to as "corners". This corner control allows you to control the transition from the stocked member to the adjacent member.

Step 4: Define an angled corner.

You will define Corner 2of the Fine Finish Offset as an angled corner.

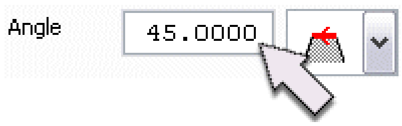
☐ Set the **Type** option for Corner 2 to **Angular Line**.



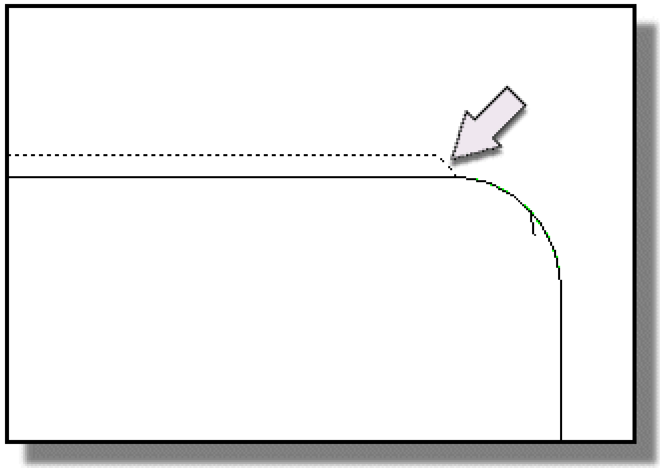
☐ Set the **Angle** option to **At current segment**.

This is the segment from which the angle will be measured.

☐ Key in **45.000** in the Angle field.



☐ Choose **Preview** to see the angled corner.



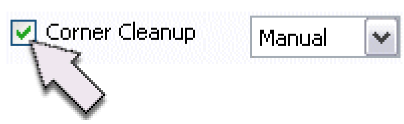
The angled corner begins at the tangency point of the blend and extends upward at 45 degrees measured from the current boundary member.

5

Step 5: Specify automatic corner cleanup.

Automatic Corner cleanup assures that the contact point of the tool reaches the tangent point of the blend despite the chamfer.

☐ Turn the **Corner Cleanup** option **on**.



☐ Set the **Corner Cleanup** option to **Automatic**.

☐ **OK** the Edit Member dialog.

☐ **OK** the Part Boundary dialog.

☐ **OK** the TURN_BND dialog.

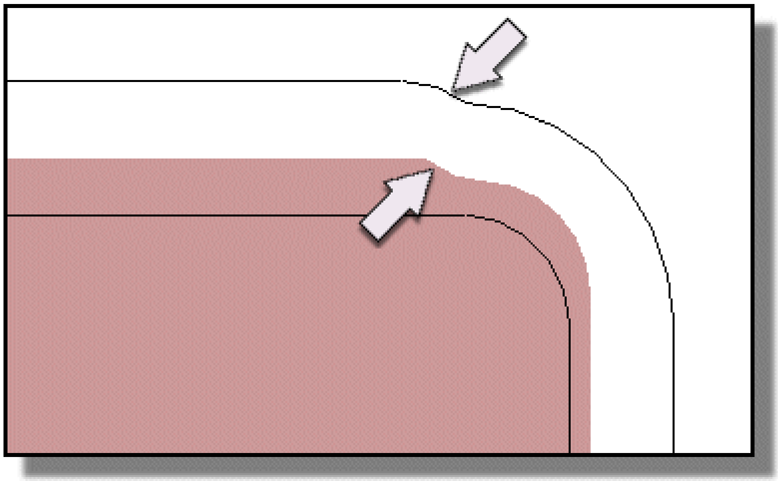
Step 6: Visualize material removal.

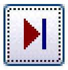
☐ Generate the tool paths for the two operations in the program.

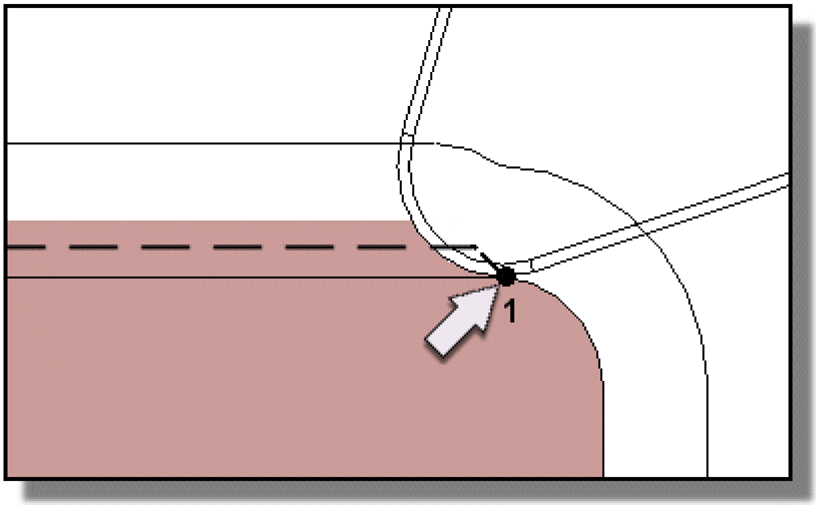
☐ Highlight FINISH_TURN_OD and choose **Verify Toolpath** from the toolbar.

☐ Turn the **2D Material Removal** option **on**.

Notice how the tool path joggles up to leave the grind stock (Fine Finish Offset) on the part.



☐ Choose the **Step** option  at the bottom of the Tool Path Visualization dialog until the tool reaches the tangent point as illustrated below.



Notice that the tool cuts into the chamfer to reach the tangent point (1). Automatic Corner cleanup assures that the contact point of the tool reaches the tangent point of the blend despite the chamfer. As a result, grind stock is not left along the blend and the round is always smooth.

☐ Continue stepping through the tool path to see the tool jog up and over the fine Finish Offset.

☐ **Cancel** the Tool Path Visualization dialog.

Step 7: Specify manual corner cleanup.

You will see how a Manual Corner Cleanup can leave unwanted material along the blend unless you specifically control the cleanup.

☐ Select the TURN_WORKPIECE object and edit the boundary again as you did in Step 2.

☐ Select the boundary member as you did in Step 2.

☐ Choose the **Fine Finish Offset** tab.

☐ Set the **Corner Cleanup** option to **Manual**.

☐ **OK** the Edit Member dialog.

☐ **OK** the Part Boundary dialog.

☐ **OK** the TURN_BND dialog.

Step 8: Visualize material removal.

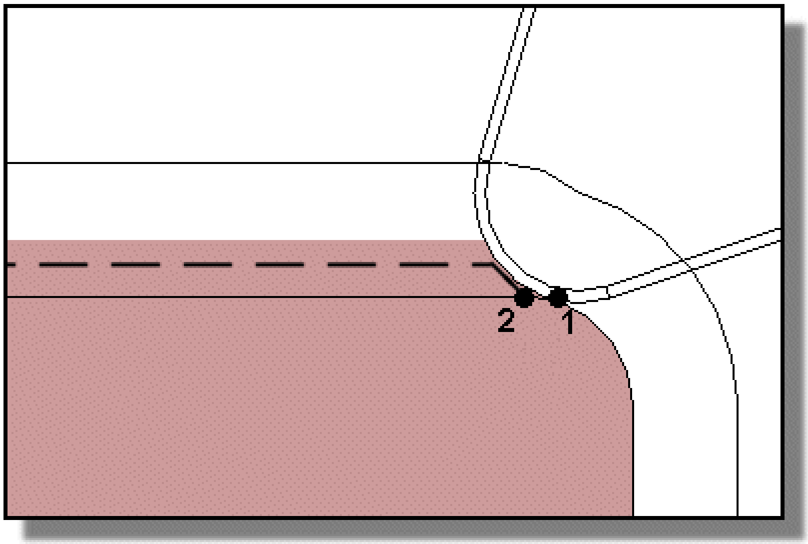
☐ Generate the tool paths for the two operations in the program.

☐ Highlight FINISH_TURN_OD and choose **Verify Toolpath** from the toolbar.

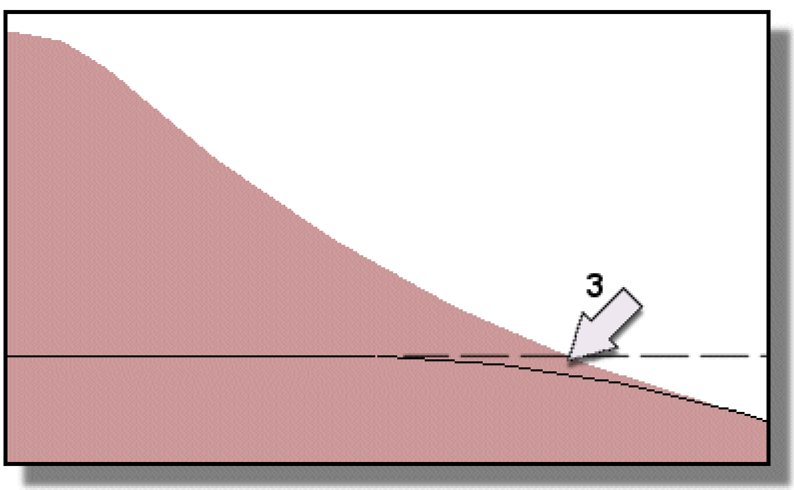
☐ Turn the **2D Material Removal** option **on**.

☐ Choose the **Step** option at the bottom of the Tool Path Visualization dialog until the tool positions near the top of the part as illustrated below.

Notice that the tool contact point (1) never reaches the blend tangent point (2). The tool does not cut into the angled corner, but instead cuts up and over the angled corner as soon as it is encountered.



As a result, unwanted grind stock is left along the blend (3) and the round is not smooth.



The Adjust Distance option can correct this by allowing the tool to cut further along the blend, cleaning up the unwanted grind stock. Automatic Corner Cleanup, however, is generally a better option to use in these cases.

☐ Continue stepping through the tool path to see the tool jog up and over the Fine Finish Offset.

5

☐ **Cancel** the Tool Path Visualization dialog.

Step 9: Define a tolerance offset.

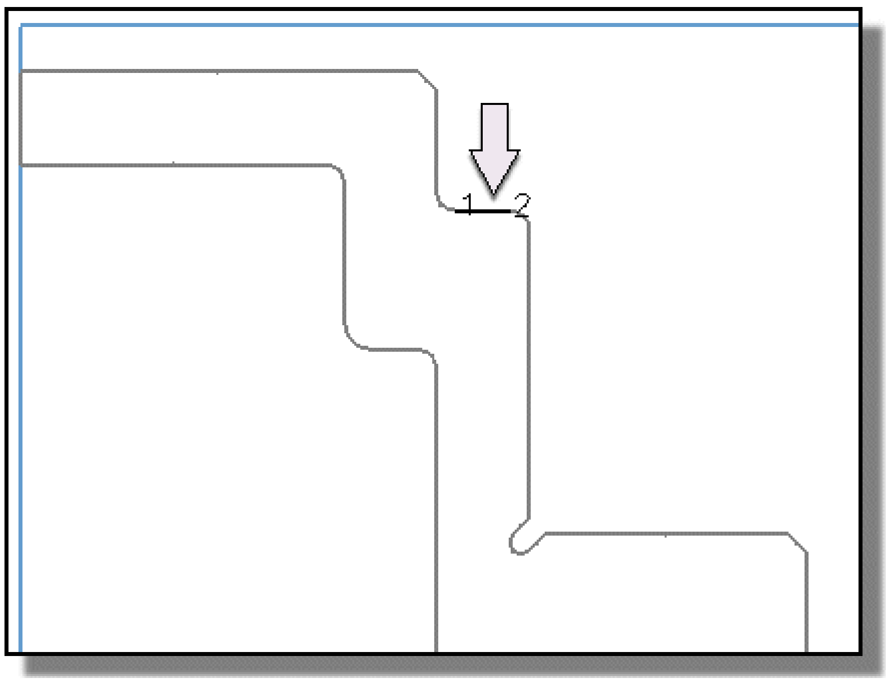
Tolerance Offset allows you to control the dimensions of the design model of your part in Manufacturing. It allows you to achieve localized mid-tolerance part size control. This offset defines a stock that will not be removed by any machining operation.

☐ Double-click TURN_WORKPIECE in the Operation Navigator to edit the object.

☐ Choose **Edit** in the TURN_BND dialog.

☐ Choose **Edit** in the Part Boundary dialog.

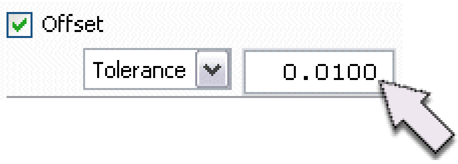
☐ Select the boundary member illustrated below to edit as you did before.



☐ Turn the **Offset** option **on**.

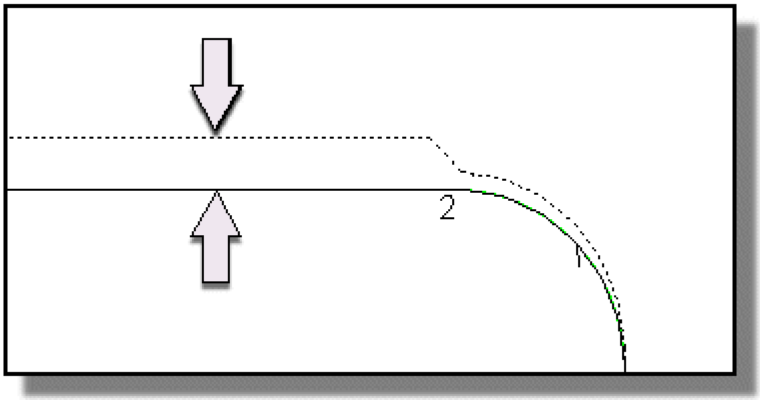
☐ Set the **Offset** option to **Tolerance**.

☐ Key in **0.010** in the Tolerance field.



This value is deliberately large so you can graphically see the offset. A more realistic offset may be 0.002.

☐ Zoom in on Corner 2 and choose **Preview** to see the combined Tolerance and Fine Finish Offsets.



You will change the Corner Cleanup for the Fine Finish Offset back to Automatic so that unwanted grind stock is not left on the blend.

☐ Choose the **Fine Finish Offset** tab.

☐ Set the **Corner Cleanup** option to **Automatic**.

5

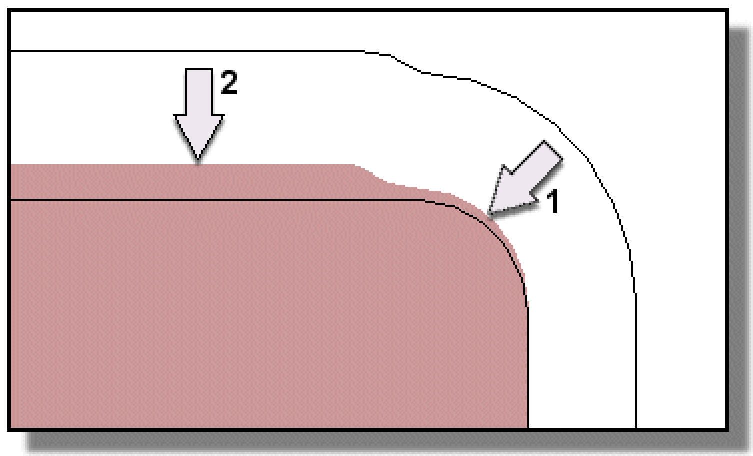
Step 10: Visualize material removal.

☐ **Generate** the tool paths for the two operations in the program.

☐ Highlight FINISH_TURN_OD and choose **Verify Toolpath** from the toolbar.

☐ Turn the **2D Material Removal** option **on**.

☐ Choose the **Play** option at the bottom of the Tool Path Visualization dialog.



Notice how the Tolerance Offset (1) gradually increases around the blend. The Fine Finish Offset (2) is then added to the Tolerance Offset once the tool contact point reaches the tangent point of the blend. As a result, there is no unwanted grind stock left on the blend and the round is smooth.

☐ **Cancel** the Tool Path Visualization dialog.

☐ Save and close the part.

5

Summary

This lesson covered many of the common parameters you will use in creating tool paths. These parameters, which include stock, feed rate, avoidance moves, engage and retracts, and feed rate control, are common to all or most turning operations. The Stock function allows you to leave material on the part. The Feed Rate is the rate at which the tool advances. Feed Rates can be specified for each type of cutting and non-cutting move. The Avoidance option enables you to define clearance motions for the tool as it moves toward or away from the part. 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.

5

Lesson

6 *Centerline Operations*

Purpose

This lesson describes the use of Centerline operations.

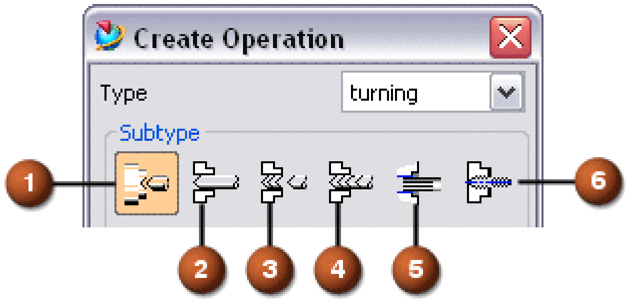
You will create several Lathe Drilling operations. You will specify the tool start point, the drill depths, and learn how to use the break chip methods.

Objective

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

- Create centerline spot drilling and drilling operations.
- Define drilling geometry both automatically and manually.
- Control chip removal by defining variable increments.

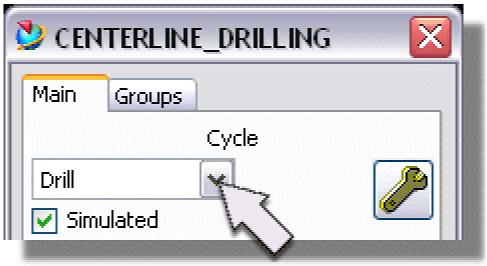
Drilling Operation Types



1. CENTERLINE_SPOTDRILLING
2. CENTERLINE_DRILLING
3. CENTERLINE_PECKDRILL
4. CENTERLINE_BREAKCHIP
5. CENTERLINE_REAMING
6. CENTERLINE_TAPPING

Drilling Cycles

Use the **Cycle** menu to choose a drilling cycle type for the operation. Cycle types determine which parameters are available and the cycle event that is output in canned machine cycles.



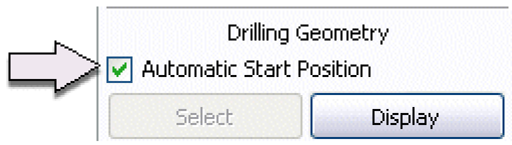
Drilling cycles may be either Canned or Simulated.

Canned	Use cycles supplied by your NC machine. Generates a CYCLE event containing all cycle parameters and a single GOTO statement representing the start position of the cycle. To use canned cycles, make sure the **Simulated** option is turned **off**.
Simulated	Generates a centerline drilling tool path and a series of GOTO statements. Does not create a CYCLE event. To use simulated cycles, make sure the **Simulated** option is turned **on**.

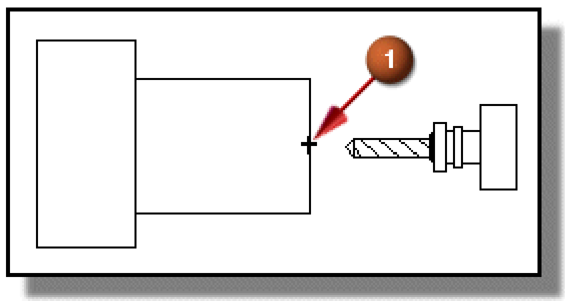
6

Drill Geometry

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



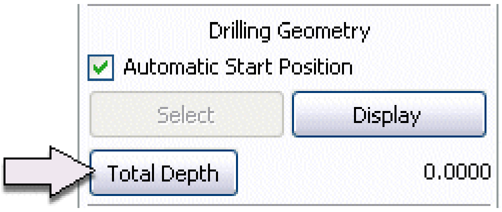
When you choose Select or Reselect under the Drilling Geometry label, the Point Constructor dialog is displayed. You are then prompted to specify a Start Point. The Start Point is the point at which the tip of the drill first contacts the part (1).



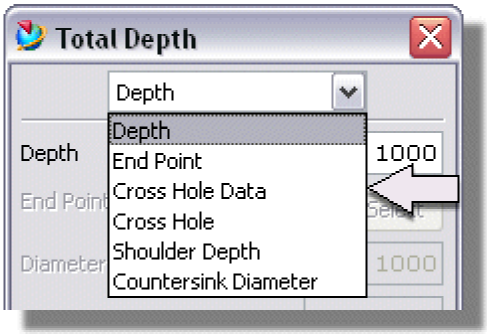
6

Total Depth

Total Depth defines the total cutting depth of the drill.



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.

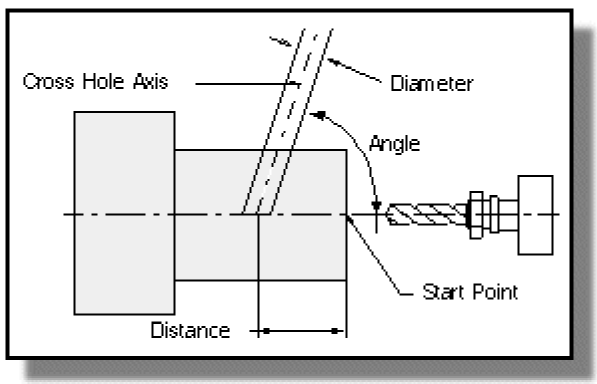
6

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

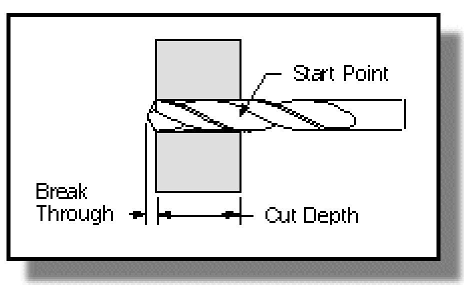
Shoulder Depth allows you to specify the drilling depth by referencing the shoulder of the tool. The tracking point is established at the tip of the tool by adding the calculated tool tip length to the specified shoulder depth.

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



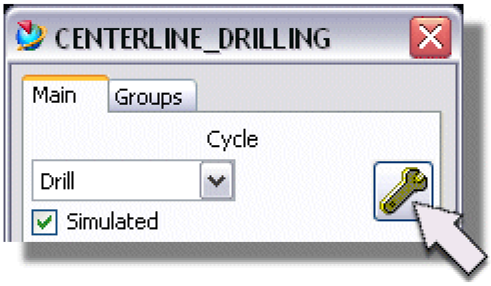
Break Through

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



Chip Removal

The **Drill, Deep** and **Drill, Break Chip** simulated cycles automatically clear or break chips during the drilling operation. You may specify chip removal parameters on the **Chip Removal** tab within the Cycle Parameters dialog by clicking the **Settings** as shown below.



Increment Type

You may specify whether the distance that the drill will move before clearing the chips (Increment Type) is Constant or Variable.

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

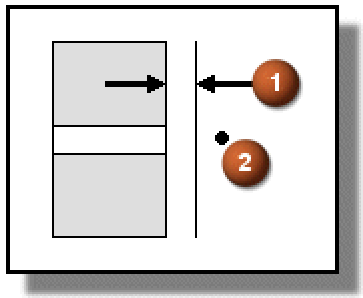
Increment refers to the distance the drill moves forward with each move.

Departure Distance refers to the distance the drill moves backward.

Clearance Distance

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

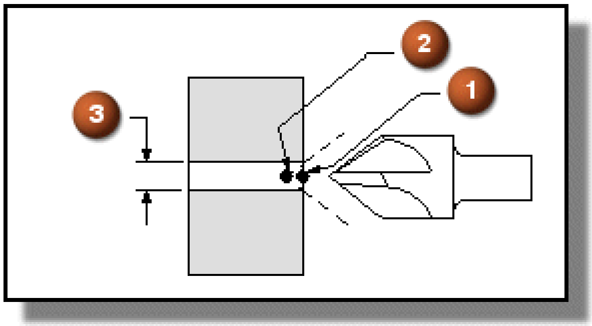
The tool rapids to the Clearance Distance (1) from the specified Start Point (2) and then proceeds at the cut feed rate.



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

Engage Distance

Engage Distance 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. The tool begins at the original start point (1), and drills to the new start point (2). The engage distance (2) is calculated based on the drilled hole (3).



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.

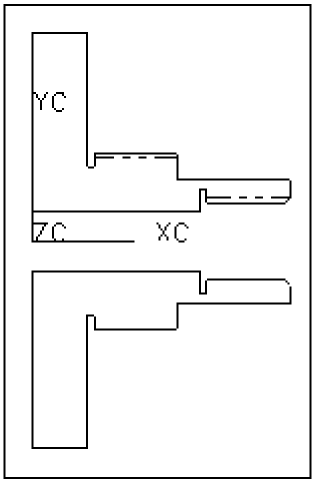
You must first specify the units (SECONDS or REVOLUTIONS) and then the number of units. A DWELL/ command is output at various times depending on the chip removal method. If you do not specify chip removal, a DWELL/ command is output when it reaches the total depth.

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 and rename the part.

☐ Open the **tmp_drill_mfg_1** part.



☐ Save the part as *****_drill_mfg_1**, where *** represents your initials.

The part and blank geometry have been predefined for this operation.

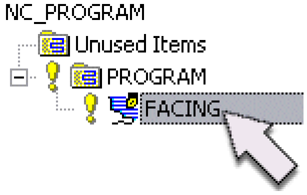
☐ Choose **Start→Manufacturing**.

6

Step 2: Verify the Facing operation tool path.

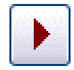
The program contains a Facing operation. The tool path generated by the subsequent spot drill operation must use the IPW created by the Facing operation.

☐ Display the Program Order View of the Operation Navigator and highlight the **FACING** operation.

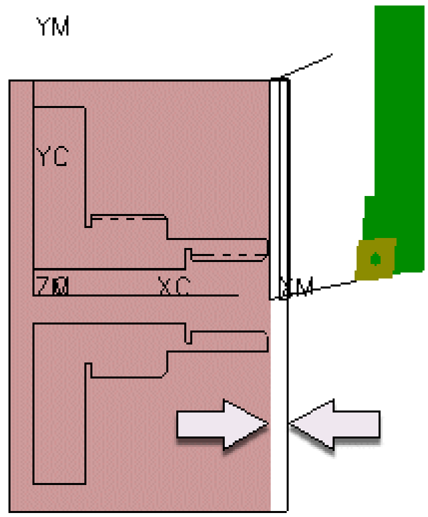


☐ Choose the **Verify Toolpath** icon in the toolbar.

☐ Turn the **2D Material Removal** option **on**.

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

Material is removed from the face of the blank.



The subsequent spot drill operation must take into consideration the material already removed by the facing operation. The system can establish the drilling start point automatically based in the in-process workpiece (IPW).

☐ **OK** to complete the tool path visualization.

Step 3: Begin a Centerline Spot Drill operation.

☐ Choose the **Create Operation** icon. 

☐ Be sure the **Type** option is set to **Turning**.

☐ Choose the **CENTERLINE_SPOTDRILL** icon. 

☐ Set the following Parent Groups:

- Program..........**PROGRAM**
- Use Geometry....**TURN_WORKPIECE**
- Use Tool..........**SPOTDRILL_.500**
- Use Method......**LATHE_CENTERLINE**

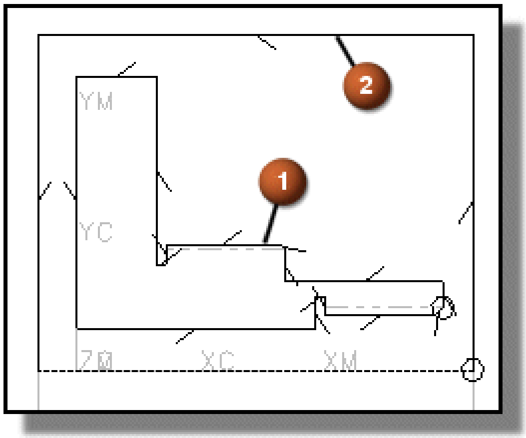
☐ Choose **OK**.

The CENTERLINE_SPOTDRILL dialog is displayed. You will display the part and blank boundaries.

☐ Choose the **Groups** tab.

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

The part (1) and blank (2) boundaries are displayed.

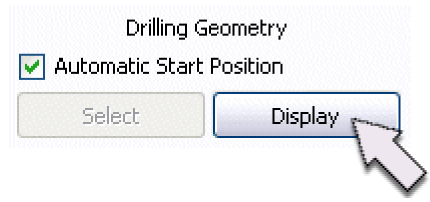


☐ Choose the **Main** tab.

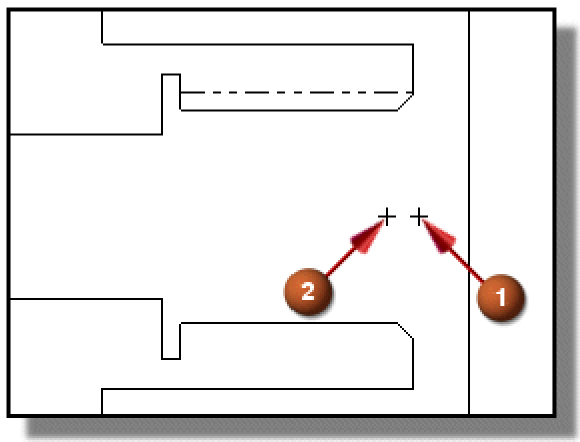
6

Step 4: Display the drilling geometry.

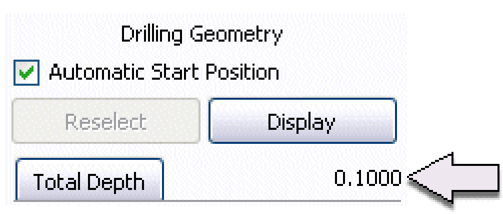
☐ With the **Automatic Start Position** option turned **on**, choose **Display**.



The start point (1) is determined automatically based on the IPW.

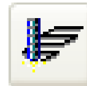


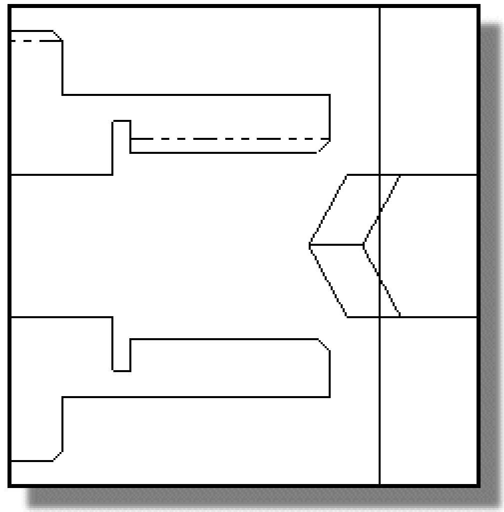
The end point (2) is determined by the Total Depth measured from the start point. Look at the default Total Depth value.



Step 5: Generate and verify the tool path.

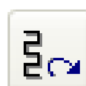
6

☐ Choose the **Generate** icon. 

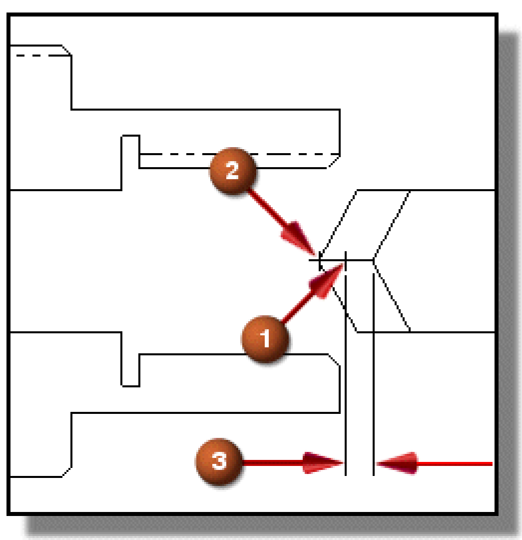


Now you can observe the relationship between the start point, the end point, and the tool path.

☐ Choose **Display** under Drilling Geometry.

☐ Choose the **Replay**  icon at the bottom of the dialog.

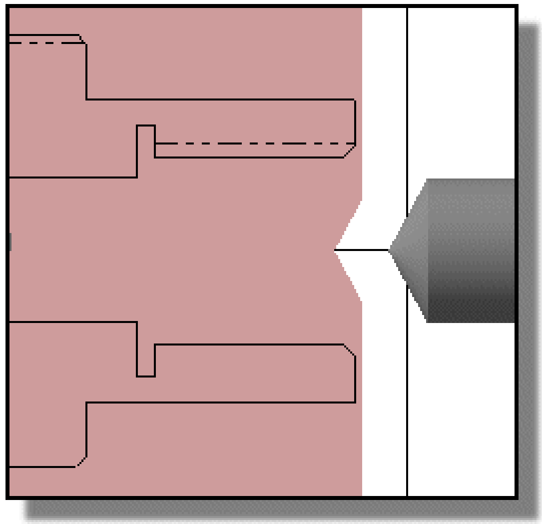
The start point (1) is the point at which the tool begins to cut. The end point (2), defined by the Total Depth value, is where the tool stops cutting. The Clearance Distance (3) allows the tool to retract out of the hole past the start point to assure that the tool clears the material.



☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

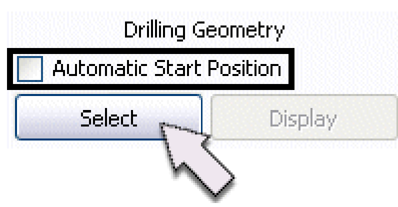
☐ Choose the **Play** icon  at the bottom of the dialog.



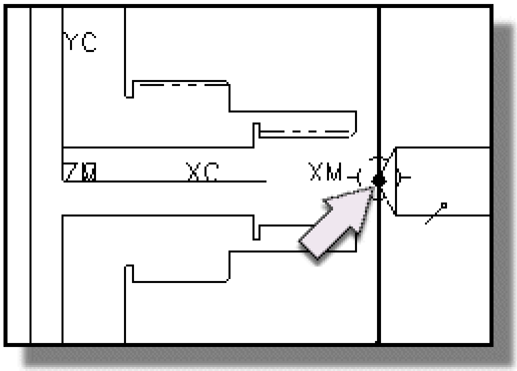
☐ **OK** to complete the tool path visualization.

Step 6: Define the start point manually.

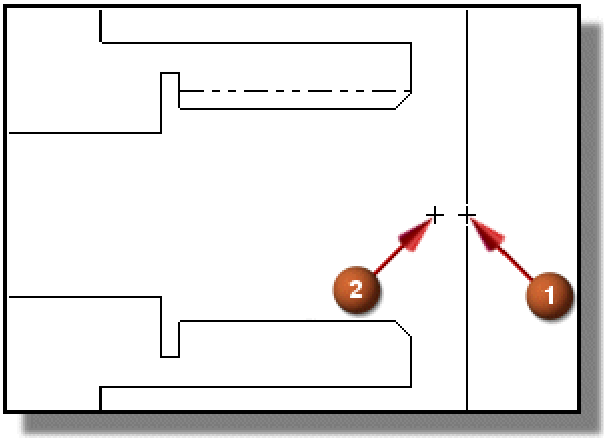
☐ Turn the **Automatic Start Position** option **off** and choose **Select**.



☐ Select the vertical line that defines the front face of the blank geometry.

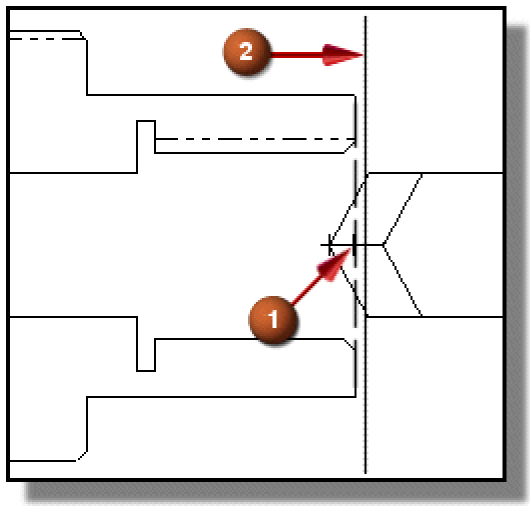


The start point (1) is positioned on the face of the blank. The end point (2) is positioned 0.100 inside this face.



Remember that the face you selected has been removed by the facing operation. Consequently, you will need to increase the Total Depth for the tool to reach the IPW. The disadvantage of this method is that you are cutting empty space as the tool moves from the start point toward the IPW.

As an alternative, you might consider positioning the start point (1) on the front face of the part. This, however, would embed the start point in the IPW (2).



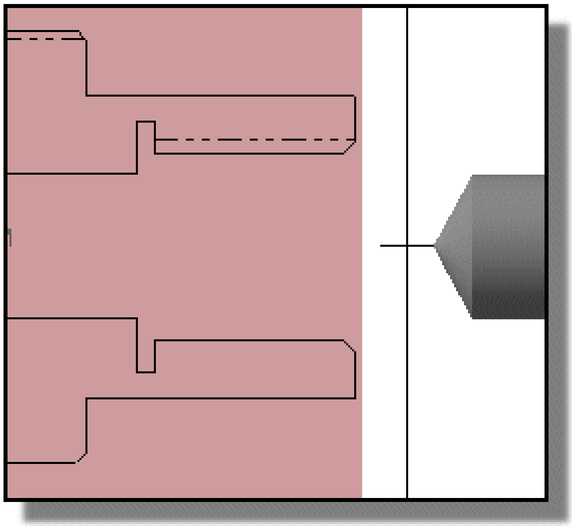
Step 7: Generate and verify the tool path.

☐ Choose the **Generate** icon. 

☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

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



The tool clearly does not reach the IPW. You will need to increase the Total Depth.

☐ **OK** to complete the tool path visualization.

Step 8: Increase the total depth.

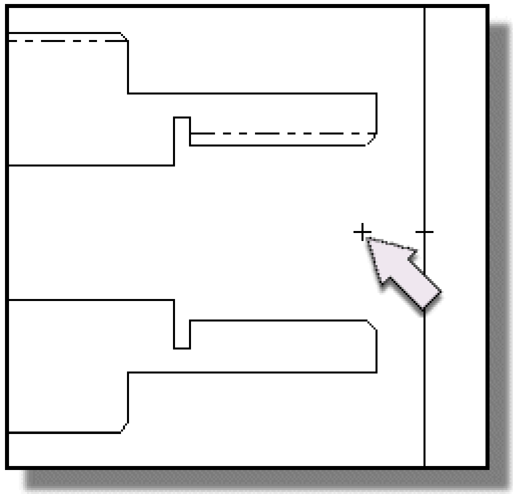
☐ Choose **Total Depth**.

☐ Key in **0.250** in the **Depth** field.

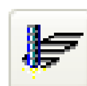
6

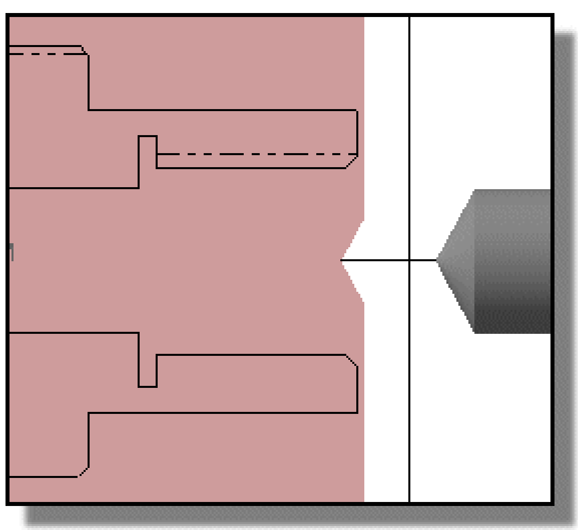
- ☐ **OK** the Total Depth dialog.

  The end point has moved to the left.



Step 9: Generate and verify the tool path.

- ☐ Choose the **Generate** icon. 
- ☐ Choose **Verify**  at the bottom of the dialog.
- ☐ Turn the **2D Material Removal** option **on**.
- ☐ Choose the **Play** icon  at the bottom of the dialog.



Now the tool reaches the IPW.

☐ **OK** to complete the tool path visualization.

You can see by these two examples that it is much easier and more effective to use Automatic Start Position than to manually define the start point.

Step 10: Specify an automatic start position.

☐ Turn the **Automatic Start Position** option **on**.

☐ Specify a Total Depth of **0.100**.

☐ Generate the tool path.

☐ **OK** to complete the operation.

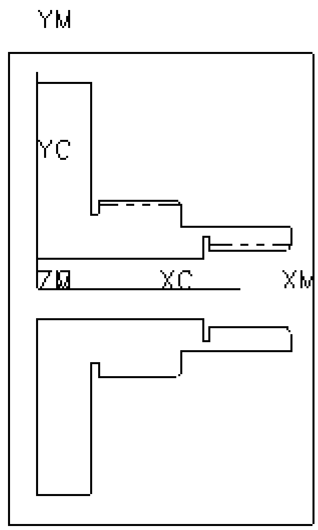
☐ Save the part.

6

Activity: Centerline Drilling – Standard Drilling

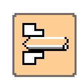
In this activity, you are going to create a standard drilling operation. You will define the Geometry, the Total Depth, and then examine the Chip Removal options.

Step 1: Continue to use the *****_drill_mfg_1** part.



Step 2: Begin a Centerline Drilling operation.

☐ Choose the **Create Operation** icon. 

☐ Choose the **CENTERLINE_DRILLING** icon. 

☐ Set the **Use Tool** option to **DRILL_.470**.

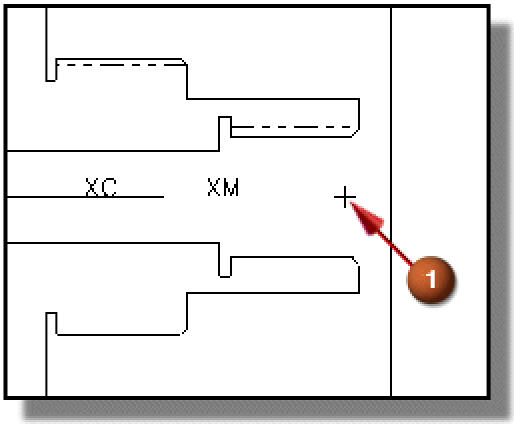
The Program, Use Geometry, and Use Method options are set correctly from the previous operation.

☐ Choose **OK**.

Step 3: Display the automatic start point.

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

The start point position (1) is based on the IPW.



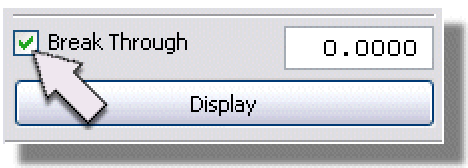
Step 4: Specify the total depth.

The end point is determined by the Total Depth measured from the start point. The default Total Depth value is 0.0000.

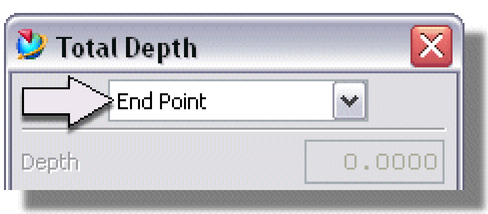
☐ Choose **Total Depth**.

You want the tool to drill completely through the part.

☐ Turn the **Break Through** option **on**.



☐ Change the Total Depth to **End Point**.

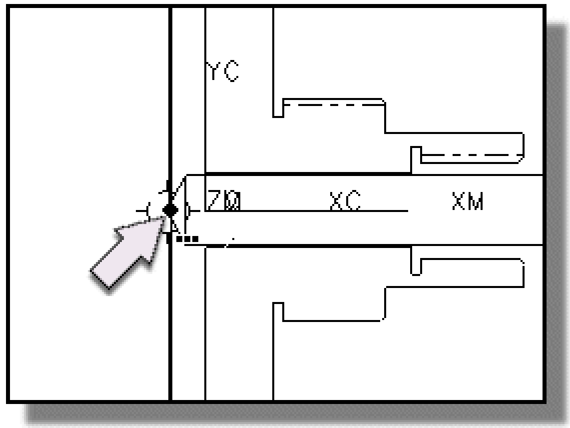


☐ Choose **Select**.

The Point Constructor dialog is displayed.

6

- ☐ Select the vertical line defining the back face of the blank.



- ☐ **OK** to accept the total depth.

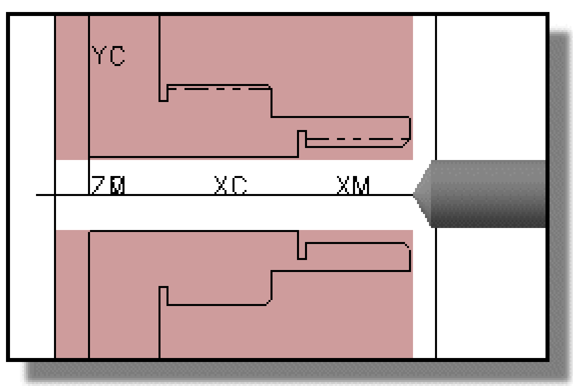
Step 5: Generate and verify the tool path.

- ☐ Choose the **Generate** icon. 

- ☐ Choose **Verify**  at the bottom of the dialog.

- ☐ Turn the **2D Material Removal** option **on**.

- ☐ Choose the **Play** icon  at the bottom of the dialog.

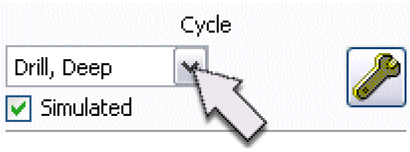


- ☐ **OK** to complete the tool path visualization.

Step 6: Change the Chip Removal setting.

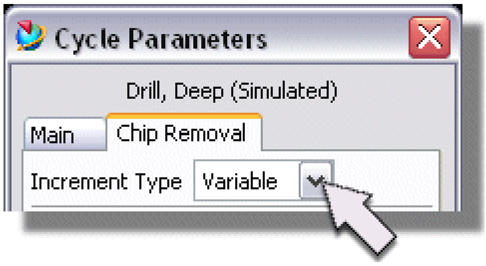
You can change the Chip Removal setting (Break Chip or Peck Drill) within an operation.

☐ Change the **Cycle** option to **Drill, Deep**.



☐ Choose **Settings** icon.  .

☐ Click the **Chip Removal** tab and change the increment type to **Variable**



☐ Enter the following values in order into the list:

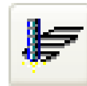
- Number of Cuts **2** Increment **.35**
- Number of cuts **4** Increment **.25**
- Number of cuts **4** Increment **.15**
- Number of cuts **2** Increment **.05**

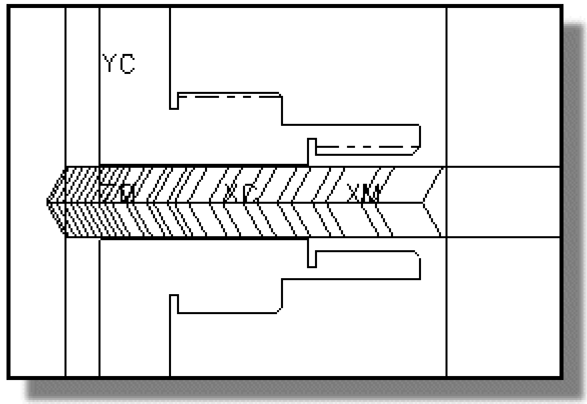
The Safe Distance determines how far the tool will retract between each cut. Use the default setting of .100.

☐ **OK** to accept the chip removal increments.

6

Step 7: Generate and verify the tool path.

☐ Choose the **Generate** icon. 



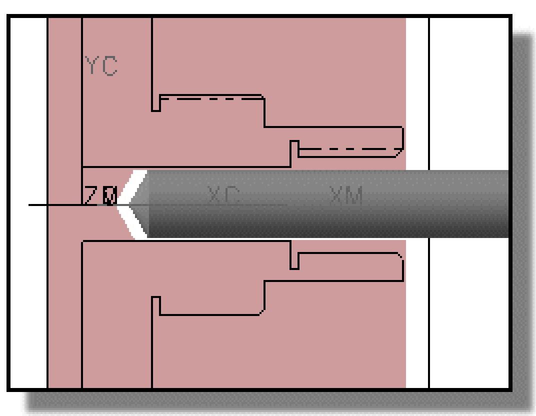
The tool moves along the centerline to the end position, drilling through the part using the increment that you entered. Note the last passes of the tool path. You specified two cuts at .05. The tool used these values, but there was material remaining. The system continued to cut to the specified depth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

☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

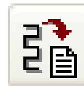
☐ Slow the **Animation Speed** down to **3**.

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

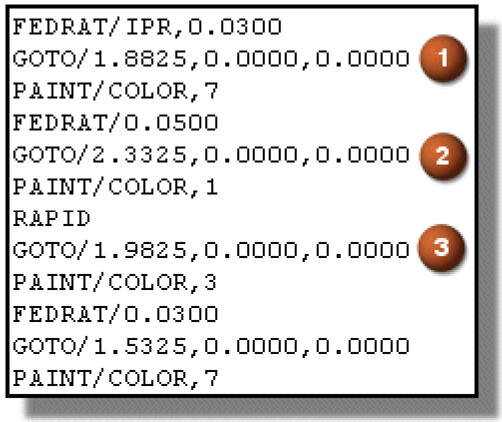


☐ **OK** to complete the tool path visualization.

Step 8: Examine the Clearance Distance 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.



```
FEDRAT/IPR,0.0300
GOTO/1.8825,0.0000,0.0000   1
PAINT/COLOR,7
FEDRAT/0.0500
GOTO/2.3325,0.0000,0.0000   2
PAINT/COLOR,1
RAPID
GOTO/1.9825,0.0000,0.0000   3
PAINT/COLOR,3
FEDRAT/0.0300
GOTO/1.5325,0.0000,0.0000
PAINT/COLOR,7
```

  The tool cuts to 1.8825 (1) then rapids to the clearance distance of 2.3325 (2). (the 2.3325 value is .100 from the IPW). The tool then rapids to a clearance distance of .100 away from the previous cut depth (3). The tool continues to cut in this manner until it clears the back of the IPW.

- ☐ Dismiss the Information window.

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

- ☐ Save the part.

6

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: Continue to use the ***_drill_mfg_1** part.

Step 2: Create an operation.

- ☐ Begin a new Centerline Drilling operation.
- ☐ 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.

Summary

In this lesson, you learned how to create centerline spot drilling and drilling operations. You learned how to define drilling geometry both automatically and manually. When defined automatically, the drilling geometry is based in the IPW making the automatic method much easier and more efficient to use than the manual method. When drilling, you learned how to control chip removal by defining variable increments.

Lesson

7 *Roughing Operations – OD Work*

7

Purpose

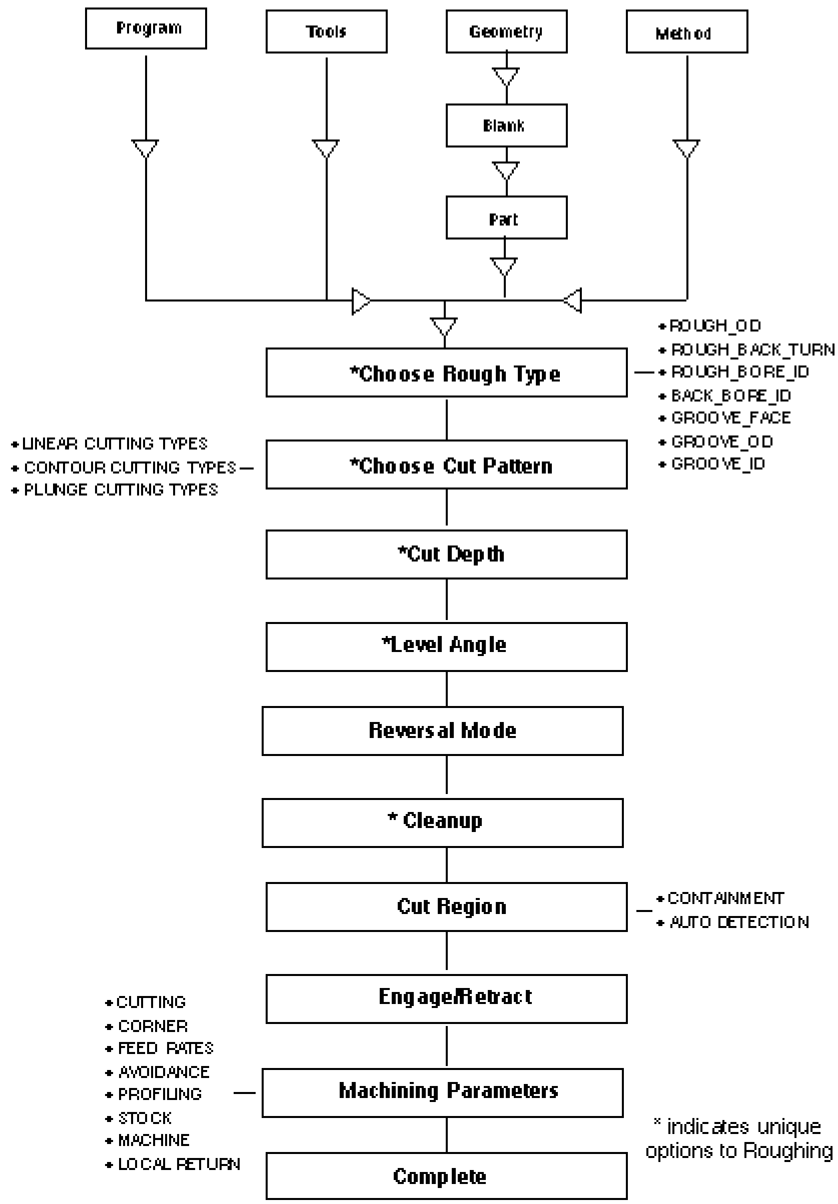
The roughing options in Turning allow you to remove large amounts of material and leave sufficient material for finishing operations.

Objective

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

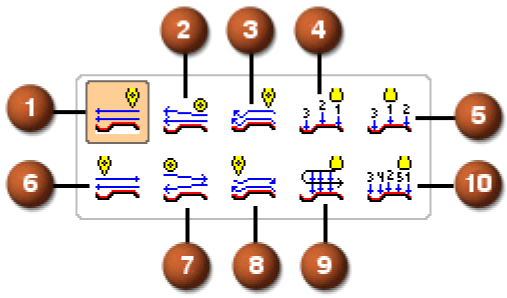
- Create a Rough Turn Outer Diameter operation.
- Change the Cut Pattern.
- Identify the appropriate Cut Depths.
- Add and remove Cleanup passes.
- Create additional Profiling passes (spring pass).
- Add a Local Return Point to the Spring pass.
- Add Post Commands to Local Return moves.
- Apply Single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

Roughing Options



7

Cut Patterns



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

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

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

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

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

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

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

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

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

10. **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.

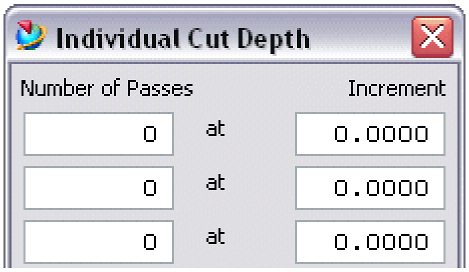
Variable Max. determines the cut regions, cuts at the maximum value as many times as possible, and then removes the remaining material in one pass if it is greater than or equal to the minimum within each separate region.

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.

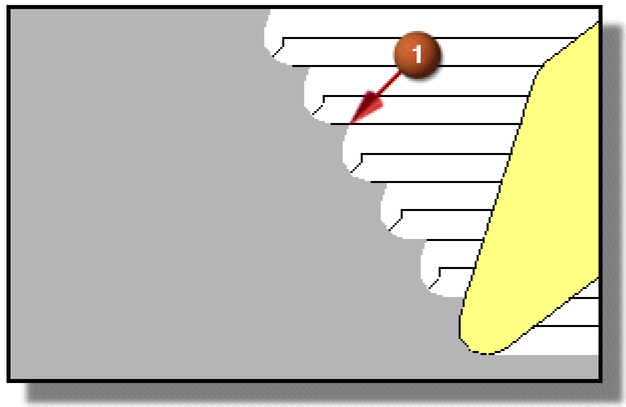
Individual produces a series of differing cut depths by allowing you to define the number of passes at each cut depth.



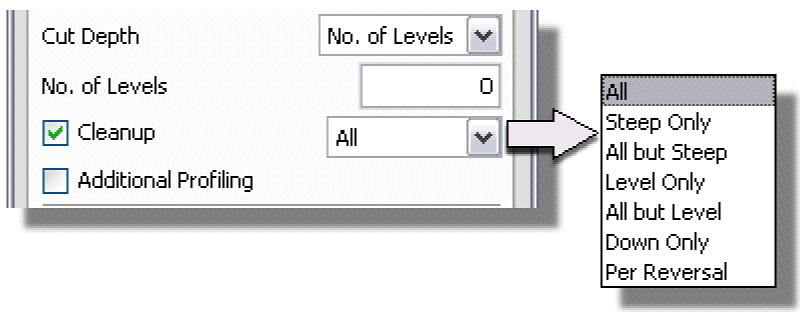
7

Cleanup

Cleanup is an additional pass applied at the end of the tool path designed to remove material such as steps (1) left by the previous rough cutting passes.



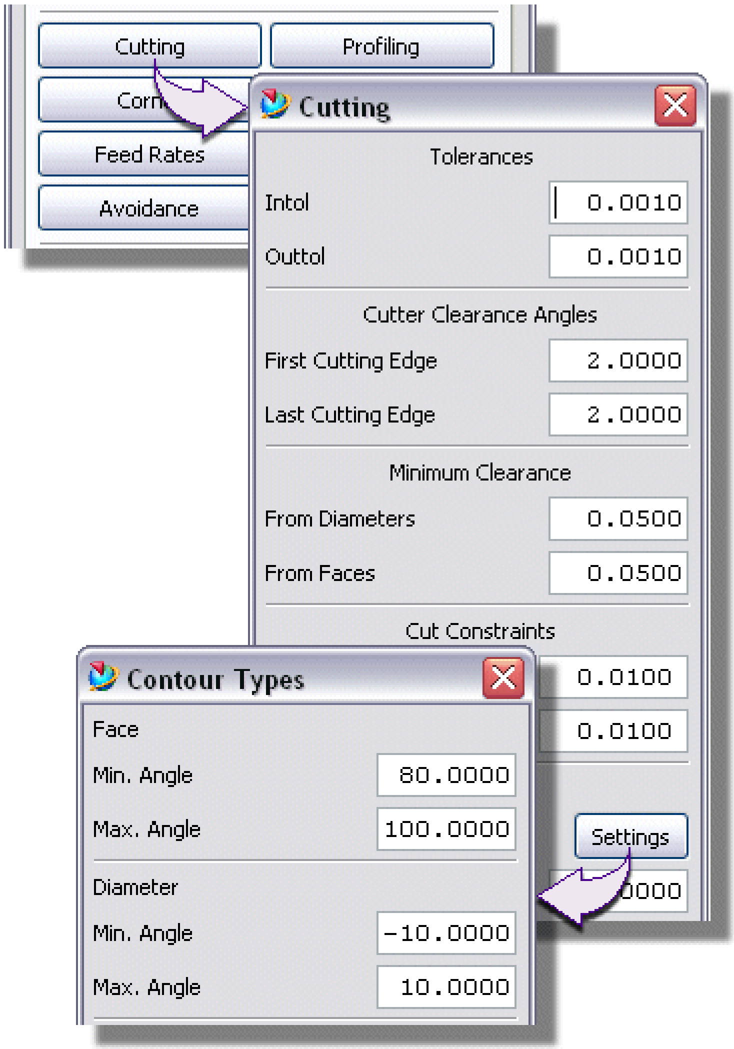
Cleanup passes can be done in several different ways.



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 Regions

This option allows you to machine all cut regions in a single operation, reducing the number of operations required to machine your part. Region Sequencing allows you to control the cut region machining sequence. Sequences include Single Direction, Inverse Direction, Alternate, and Bi-directional.

Cut Control

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

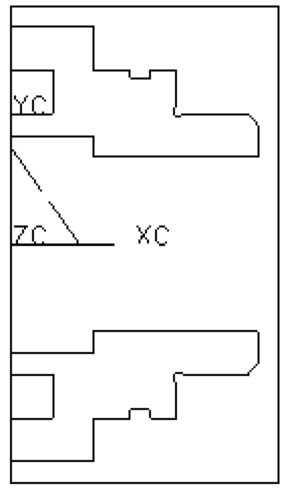
Dwell after Rough Cut

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 **tmp_rgh_od_mfg_1** part.



☐ Save the part as ***_rgh_od_mfg_1**, where *** represents your initials.

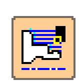
☐ Choose **Start→Manufacturing**.

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. 

☐ Be sure the **Type** option is set to **Turning**.

☐ Choose the **ROUGH_TURN_OD** icon. 

☐ Set the following Parent Groups:

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

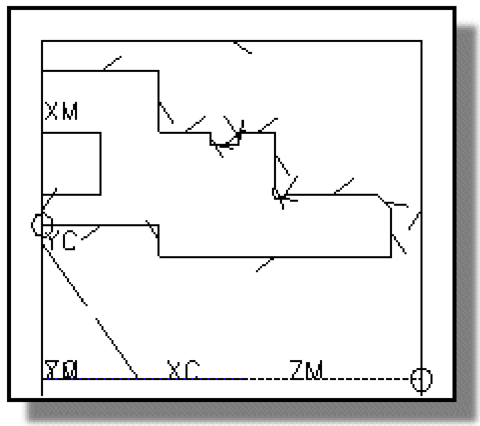
☐ Choose **OK**.

The ROUGH_TURN_OD dialog is displayed.

Next you will verify the geometry.

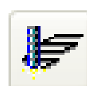
☐ Choose the **Groups** tab at the top of the dialog.

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



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

Step 3: Generate and verify the tool path.

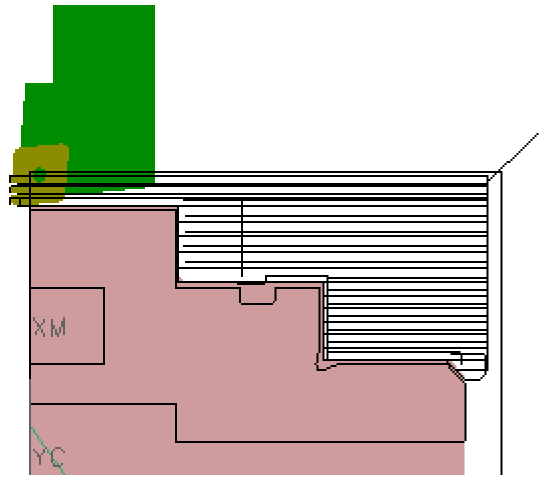
☐ Choose the **Generate** icon. 

☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

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

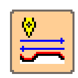
The Cut Pattern is Linear Zig.



☐ **OK** to complete the tool path visualization.

Step 4: Changing Cut Patterns.

You will change the cut pattern and then regenerate the tool path.

☐ Choose the **Linear Zig-Zag** icon. 

☐ Choose the **Generate** icon.

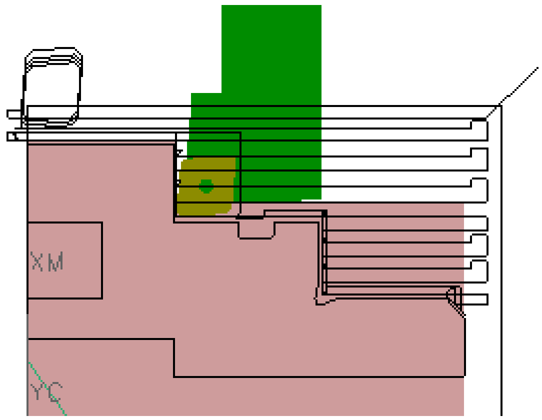
☐ Choose **Verify** at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

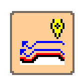
☐ Slow the **Animation Speed** down to **1**.

☐ Choose the **Play** icon at the bottom of the dialog and notice the motion of the tool.

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



☐ **OK** to complete the tool path visualization.

☐ Choose the **Contour Zig** icon. 

☐ Key in **0.100** in the Depth field.

☐ Choose the **Generate** icon.

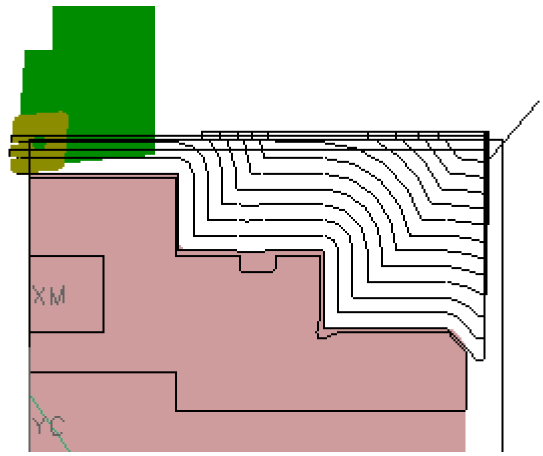
☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

7

☐ Choose the **Play**.

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.



☐ **OK** to complete the tool path visualization.

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

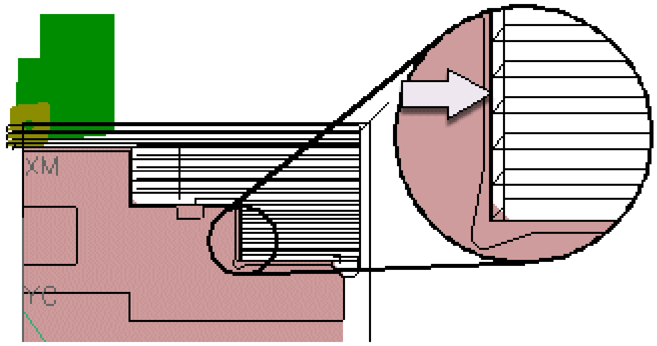
Next you will examine the Cleanup option. This option is unique to Roughing Operations.

Step 5: Examine the results of the Cleanup option.

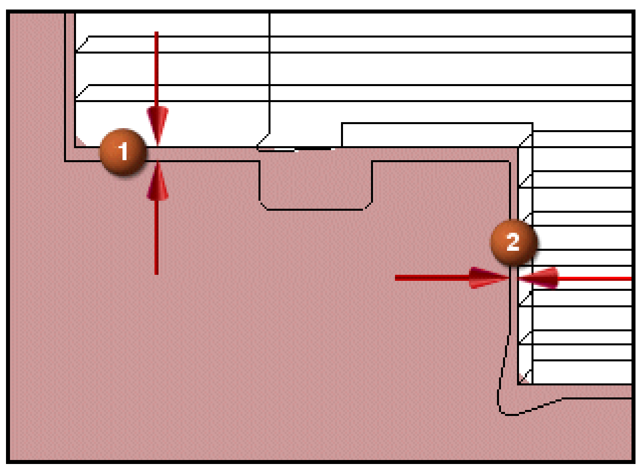
Cleanup is designed to remove material (steps in most cases) left from roughing passes. First, you are going to examine the results of the cleanup option.

☐ **Generate** and **Verify** the tool path as you did before.

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.



The cleanup pass leaves the 0.030 radial stock (1) 0.020 face stock (2) defined in the **Stock**→**Rough Stock** settings.

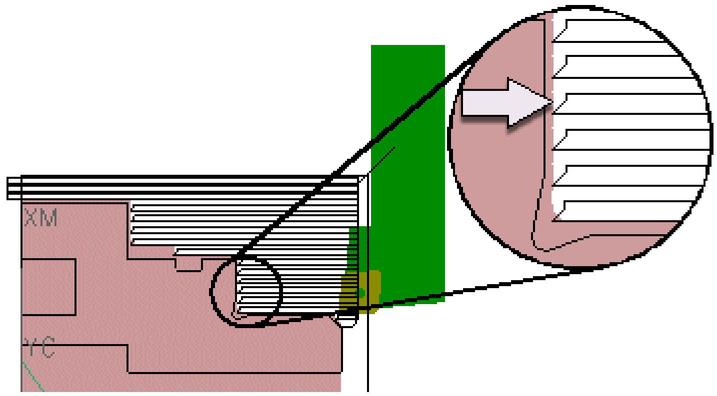


☐ Turn **off** the **Cleanup** option.

7

☐ **Generate** and **Verify** the tool path.

There is no cleanup pass.



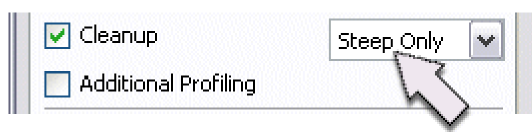
☐ **OK** to complete the operation.

Step 6: Cleanup steep areas only.

☐ Double-click **ROUGH_TURN_OD** to edit the operation.

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

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



The system delineates steep from non-steep based on the Steep parameters specified in the Contour Types dialog.

☐ Choose **Cutting**.

☐ Choose **Settings** under Cut Control.

Areas with angles falling between the specified Steep Min. Angle and Max. Angles are regarded by the processor as steep.

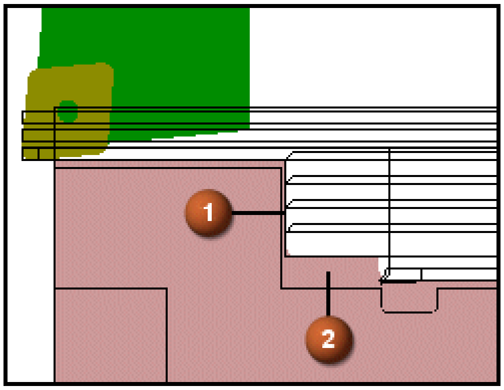
You will not make any adjustments to the steep parameters.

Steep	
Min. Angle	87.0000
Max. Angle	93.0000

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

☐ **Generate** and **Verify** the tool path.

Only steep areas (1) are cleaned up. Non-steep areas (2) are not cleaned up.



☐ Change the **Cleanup** option back to **All**.

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

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

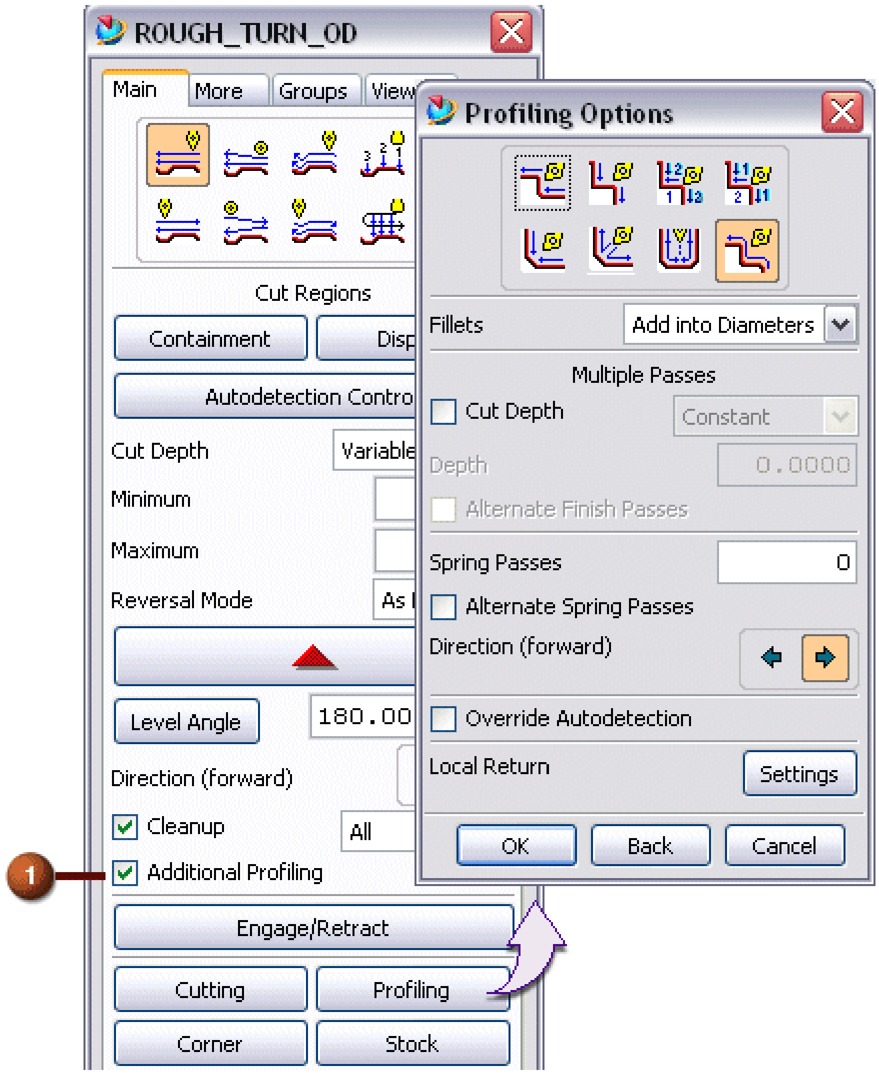
☐ Choose **OK** to complete the operation.

☐ Save the part.

7

Profiling

Profiling options provide additional Spring Passes and Multiple Finish Passes. These options differ from Cleanup in that they follow the part boundary, where as the Cleanup option only removes excess step material. The Additional Profiling option (1) will cause the operation to use the profiling parameters you define in the Profiling Options dialog.



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.

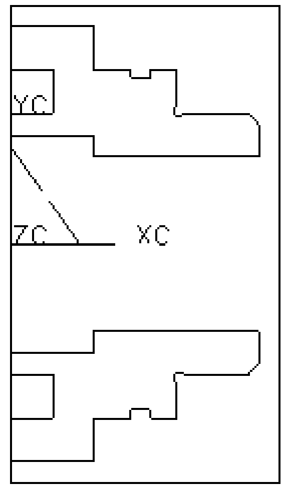
If rough and profile stock differ, the activated toggle to override the second cut region detection process would cause the processor to execute the profile cut with the wrong offset setting.

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: Continue to use the *****_rgh_od_mfg_1** part.



Step 2: Create an Additional Profiling pass (spring pass).

When creating an Additional Profiling pass (spring pass), the processor will first calculate a Cut Region for the Rough Stock and execute the rough cuts. Then it will calculate a Cut Region using Profile Stock and execute that cut.

☐ Expand **PROGRAM_PROFILE** so you can see the three operations.

☐ Double-click **ROUGH_TURN_OD_PROFILE** to edit the operation.

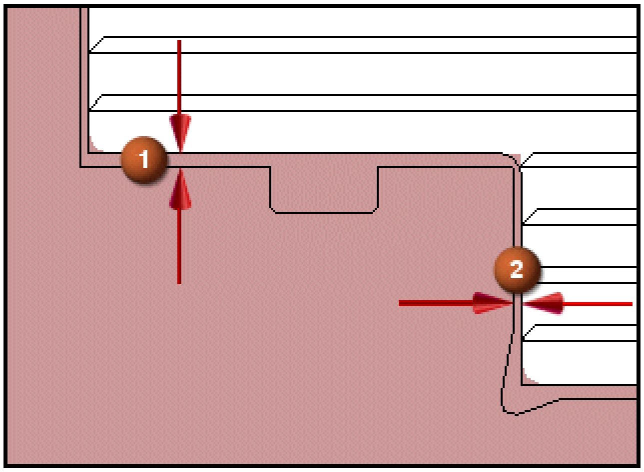
☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

7

☐ Choose the **Play**.

The cleanup pass leaves the 0.030 radial stock (1) and the 0.020 face stock (2) defined in the **Stock→Rough Stock** settings.



☐ **OK** to complete the tool path visualization.

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

☐ Choose **Profiling**.

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

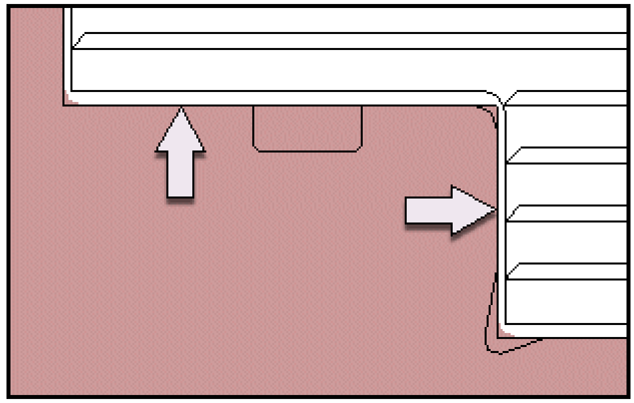
☐ Enter **1** into the Spring Passes value field.

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

7

☐ **Generate** and **Verify** the tool path as you did before.

Note that the final pass cuts to finish depth.



This is due to the Profile Stock settings of **Stock→Profile Stock** settings of 0.0000 Face and 0.0000 Radial. This is an example of a roughing operation that has a finish pass.

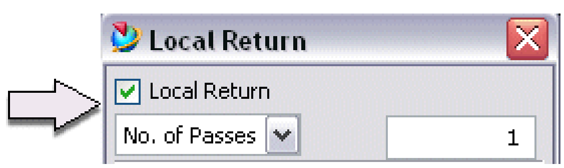
Step 3: Add a Local Return Point and an operator message.

You are going to add a local return point to the Profile pass only. This is done 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.

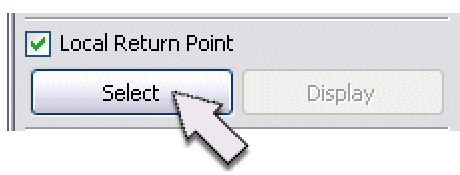
☐ Choose **Profiling**.

☐ Choose **Settings** next to Local Return.

☐ Turn **on** the **Local Return** option and set **No. of Passes** to 1.



☐ Turn **on** the **Local Return Point** option and choose the **Select** button.



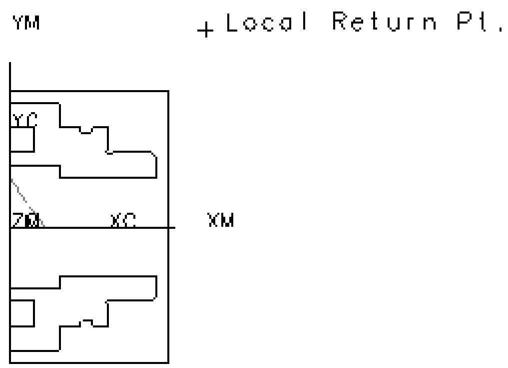
7

☐ Enter the following values:

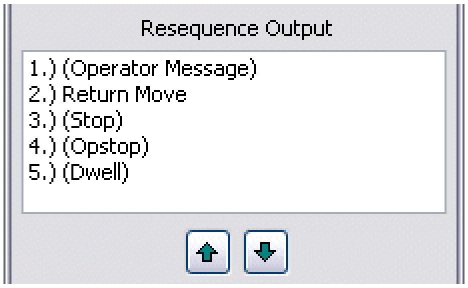
- XC = **4**
- YC = **4**

☐ **OK** to accept the Point Constructor dialog.

☐ Choose **Display** to display the local return point.



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 the **Opstop** and **Operator Message** options **on**.

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

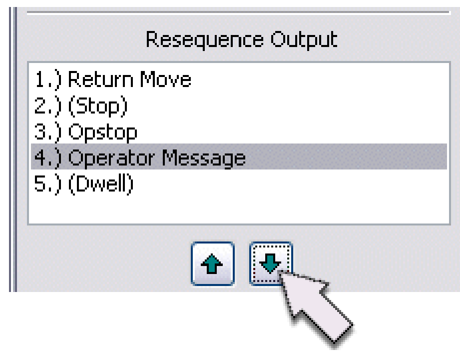
7

Step 4: 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**.



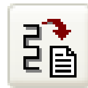
You are ready to generate the spring pass changes, the local return and additional commands.

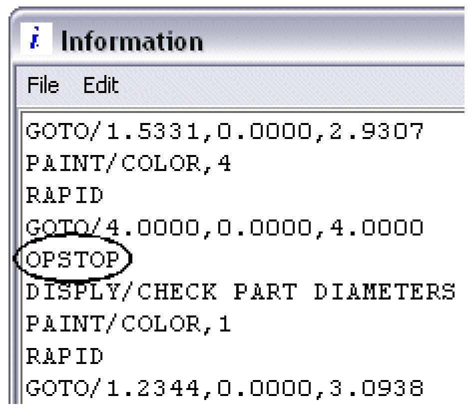
☐ Choose **OK** twice to return to the **ROUGH_TURN_OD** dialog and then **Generate** the tool path.

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.

7

Step 5: List the tool path data.

- ☐ Choose the **List** icon. 
- ☐ Scroll until you see the Opstop and Message commands in the text.



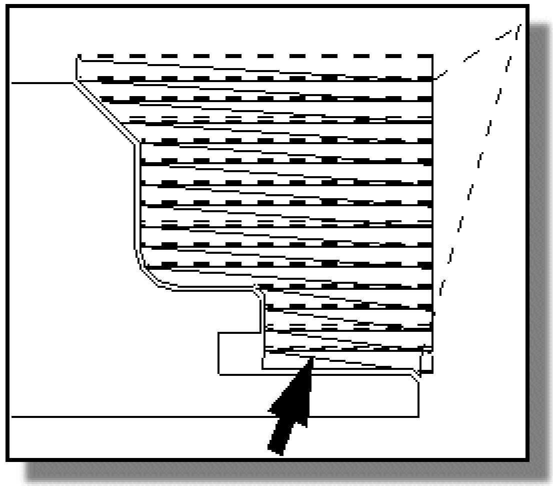
- ☐ Dismiss the Listing window.

  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.

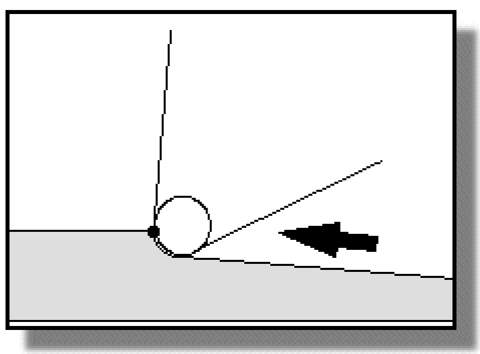
- ☐ Test this yourself by putting in an a different message and different return position using the Local Return located on the **ROUGH_TURN _OD**.
- ☐ Turn the **Additional Profiling** option **off**.
- ☐ Choose **OK** to complete the operation.
- ☐ Save and close the part.

Ramping

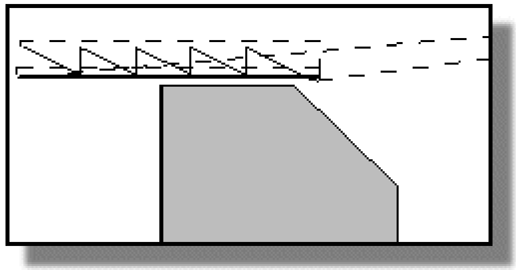
Ramping cut strategies can prolong tool life (particularly ceramic inserts) and reduce machining time when rough cutting hard materials. Notching effects on the insert can be effectively reduced or even completely avoided through ramping which leads to savings in tooling costs and machining time.



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.



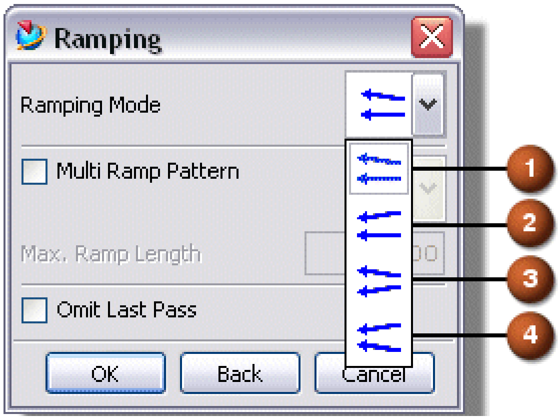
In cases where a single ramp over the length of a cut is ineffective due to a shallow cut depth, multiple ramping achieves the desired effect of a constantly changing depth by generating a "sawtooth" or "wavy" cut pattern.



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.



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

2. *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 cut level.

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

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

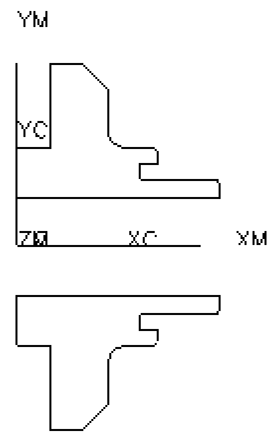
7

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 **tmp_ramp_mfg_1** part.



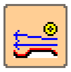
☐ Save the part as *****_ramp_1_mfg_1**, where *** represents you initials.

☐ If necessary, enter the Manufacturing application.

Step 2: Specify a Ramp Zig cut pattern.

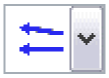
☐ Expand the **PROGRAM** Parent Group in the Operation Navigator.

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

☐ Choose the **Ramping Zig** icon. 

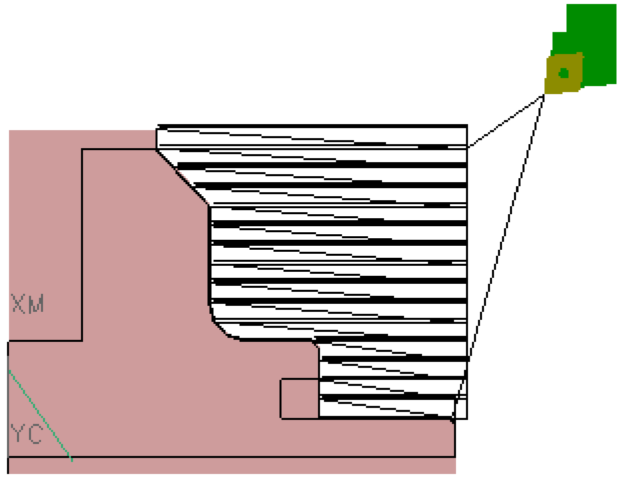
☐ Choose **Cutting**.

☐ Choose **Settings** next to Ramping.

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

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

☐ **Generate** and **Verify** the tool path.



☐ **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 **ROUGH_OD_1** in the Operation Navigator to edit the operation.

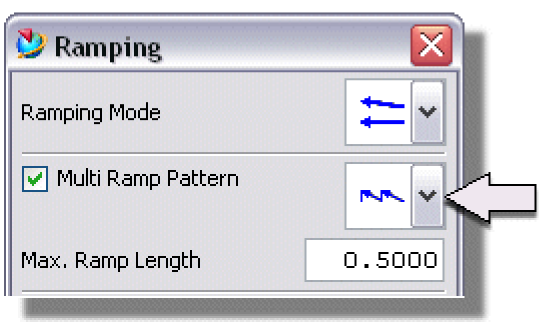
☐ Choose the **Ramping Zig** icon.

☐ 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.

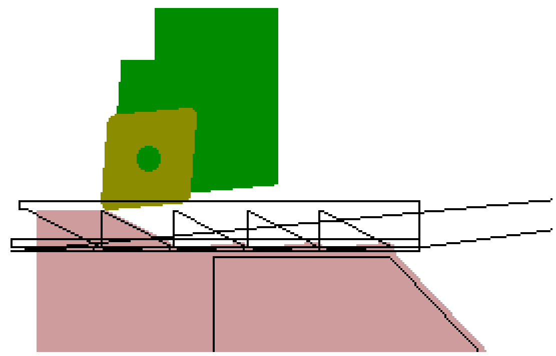
Notice that the default Multi Ramp Pattern is Ramp Out Only.



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

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

☐ **Generate** and **Verify** the tool path using an Animation Speed of **1**.



The last linear pass assures that the surface is cut flat.

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.

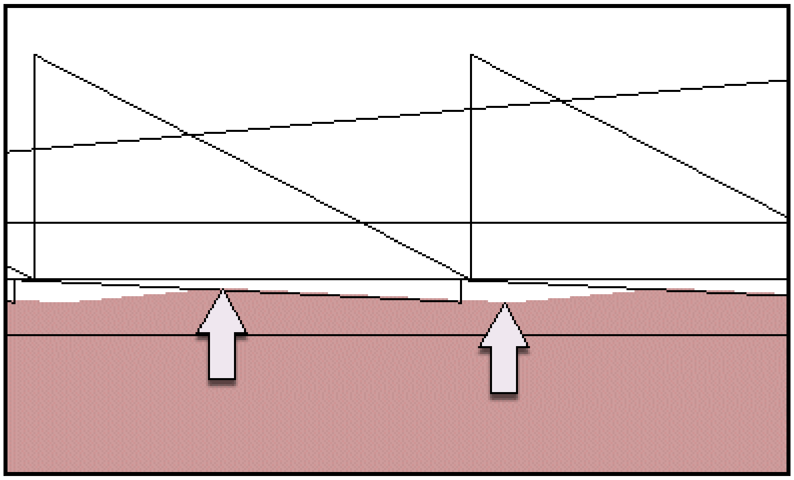
☐ Turn the **Omit Last Pass** option **on**.

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

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

☐ **Generate** and **Verify** the tool path.

Notice that the surface is not flat, but irregular.

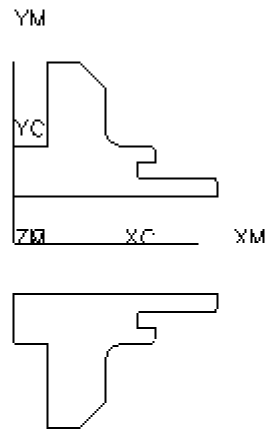


☐ **OK** to complete the operation and save the part.

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: Continue to use the *****_ramp_mfg_1** part.



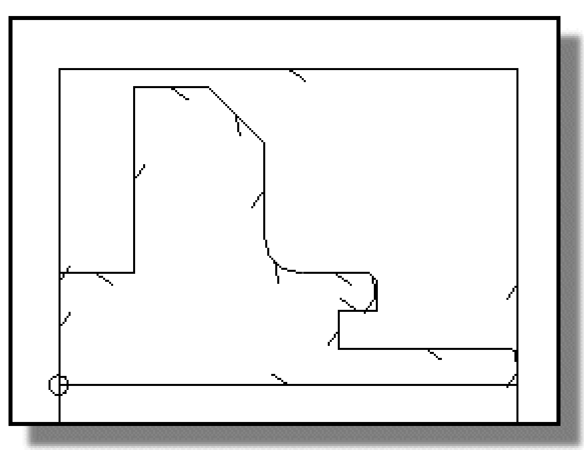
Step 2: Verify the part boundary.

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

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

☐ Highlight the **TURN_WORKPIECE** geometry group in the Operation Navigator and **MB3→Object→Display** .

The part and Blank boundaries display.



7

Step 3: Edit boundary parameters.

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

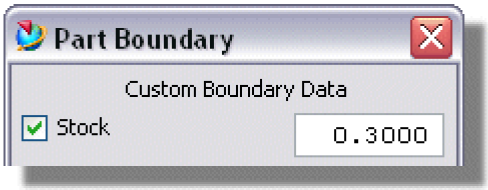
☐ Double-click **ROUGH_OD** to edit the operation.

☐ Choose the **More** tab.

☐ Choose **Edit** under Part Boundary.

The Part Boundary dialog displays allowing you to apply Stock and Cut Feed Rate 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 Feed Rate, 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.

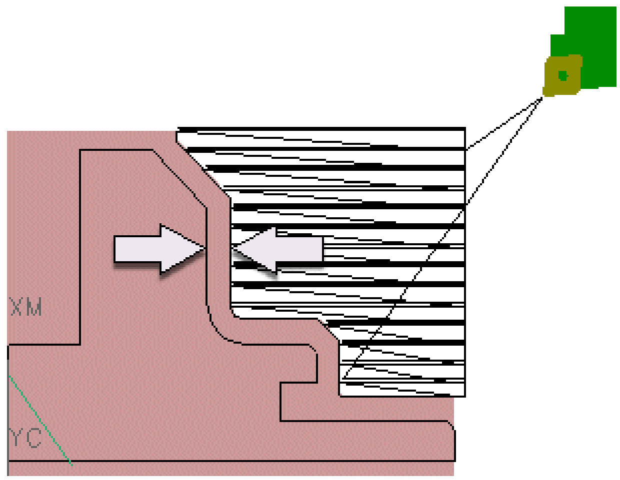
7

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.

☐ **Generate** and **Verify** the tool path as you did before.

The 0.300 stock is applied to the entire part boundary.



☐ **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.

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

☐ Set the following Parent Groups:

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

☐ **OK** to create the operation.

☐ **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.

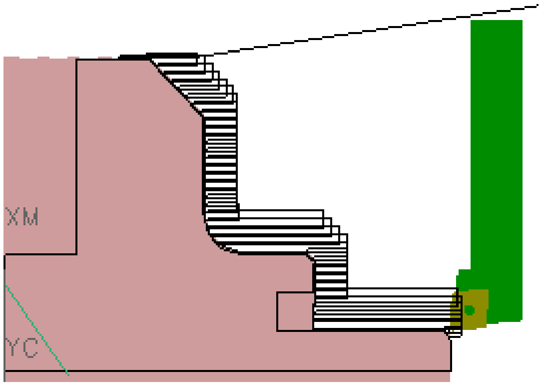
☐ In the Program Order View of the Operation Navigator, choose **PROGRAM** and **MB3→Generate**.

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

☐ With **PROGRAM** still highlighted in the Operation Navigator, choose **Verify Toolpath**  in the toolbar

☐ Turn the **2D Material Removal** option **on** and **Play**.

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



☐ **OK** to complete the operation.

☐ Save and close the part.

7

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

8 *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

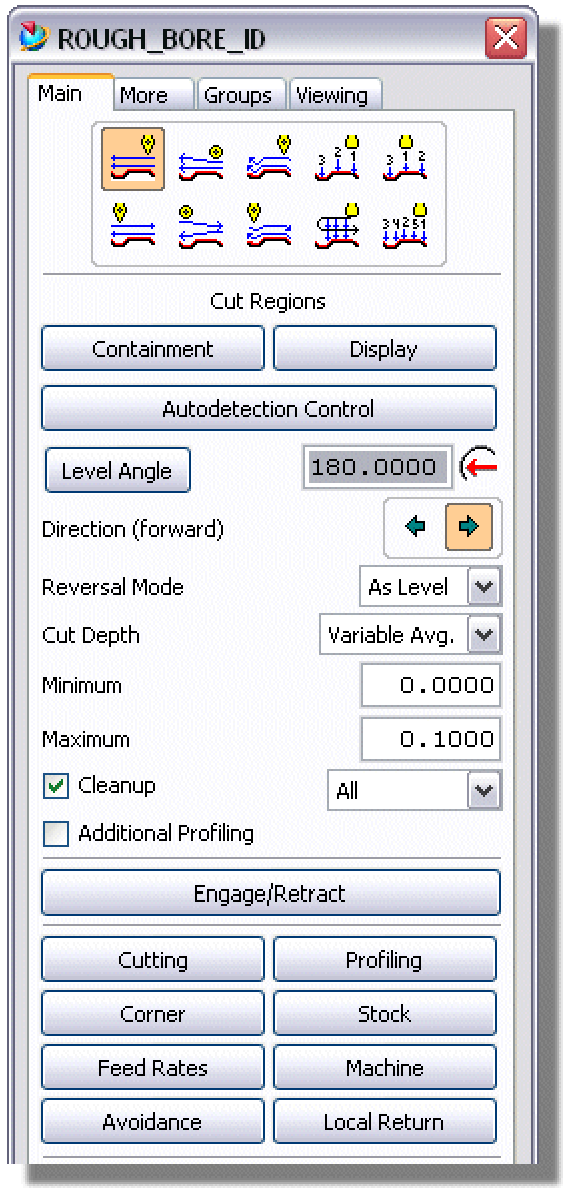
Upon completion of this lesson, you will be able to:

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

8

Roughing Operations for ID Work

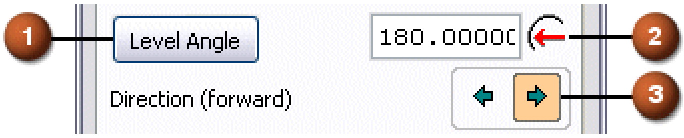
ROUGH_BORE_ID operation subtypes have the same options available as ROUGH_TURN_OD operation subtypes.



8

Level Angle/Step Angle

In Rough Turn operations, the Level Angle (Zig and Zig-Zag cuts) and Step Angle (Plunge cuts) options (1) define the angle at which the rough cuts are made. They are accompanied by a visual indicator (2) that graphically displays the cut angle. The cut angle is measured counterclockwise from the spindle centerline.



The Forward/Backward direction arrows (3) 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.

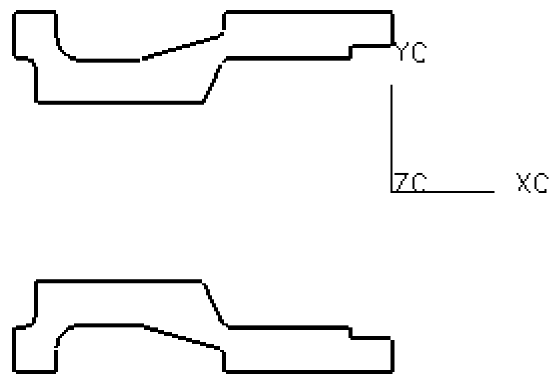
8

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 **tmp_reducer_mfg_1** part.



☐ Save the part as *****_reducer_mfg_1**, where *** represents your initials.

The cross section geometry has been predefined for this operation.

☐ Choose **Start→Manufacturing**.

Step 2: Define the parent groups for the first ID operation.

☐ Choose the **Create Operation** icon. 

☐ Be sure the **Type** option is set to **Turning**.

☐ Choose the **ROUGH_BORE_ID** icon. 

The Parent Groups for this program have been predefined for you.

☐ Set the following Parent Groups:

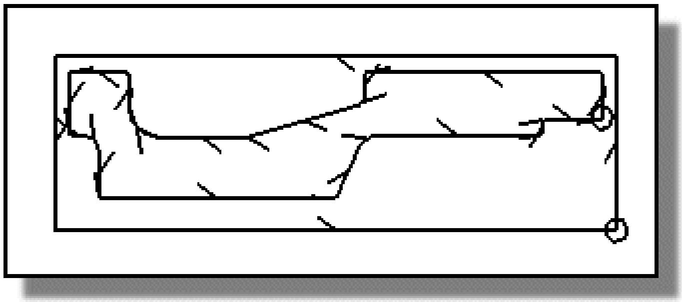
- Program..........**PROGRAM**
- Use Geometry....**PART**
- Use Tool..........**ID_80_L**
- Use Method......**LATHE_ROUGH**

☐ Choose **OK**.

The ROUGH_BORE_ID dialog is displayed.

☐ Choose the **Groups** tab.

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



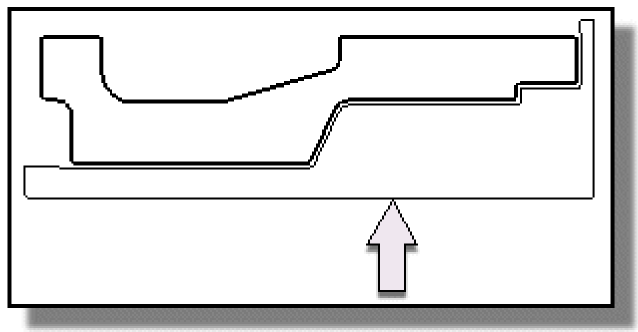
8

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.

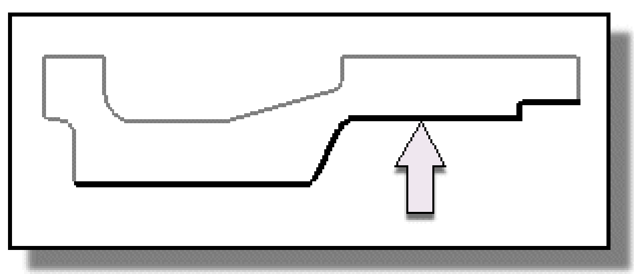
☐ Choose the **Main** tab.

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



Step 4: Limit the area the tool will cut.

The back portion of the part will be cut in a different setup. You are going to add containment to restrict the cut region to the area indicated below.



☐ 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.

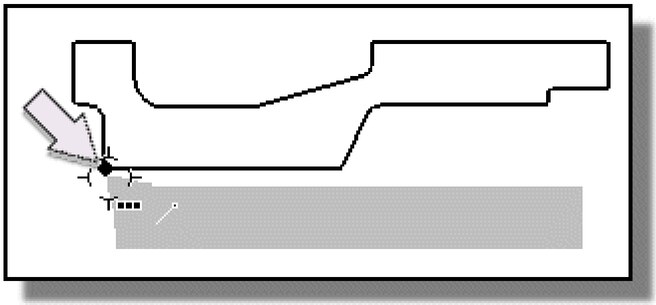
☐ Check the **Trim** option next to Axial 1.

☐ Choose the **Axial 1** button.

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

8

☐ Select the end of the horizontal line to define the containment position at the as shown below.



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

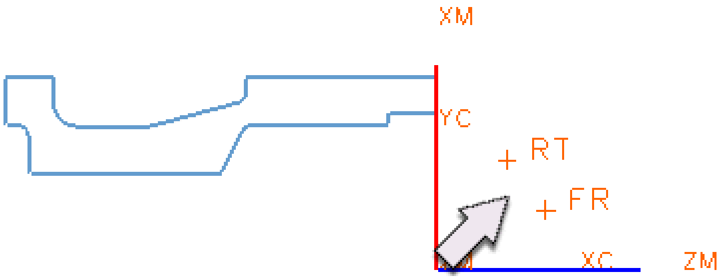
☐ Key in **–.100** in the Delta-XC field and choose **OK** until you return to the ROUGH_BORE_ID dialog.

Step 5: Avoid cutting the reversal.

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

Step 6: Define the avoidance geometry.

☐ Define a **From point** and a **Motion to Return Point** at the approximate screen position shown below.



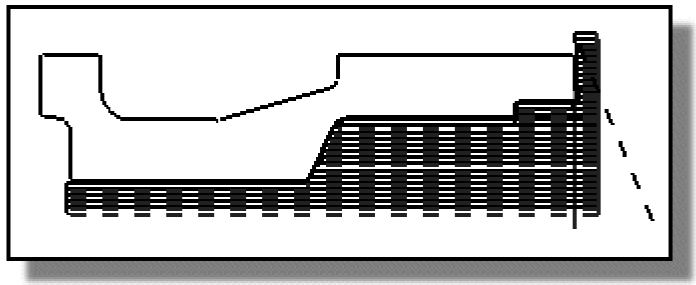
☐ Set the **Motion Type** to **Direct**.

8

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.

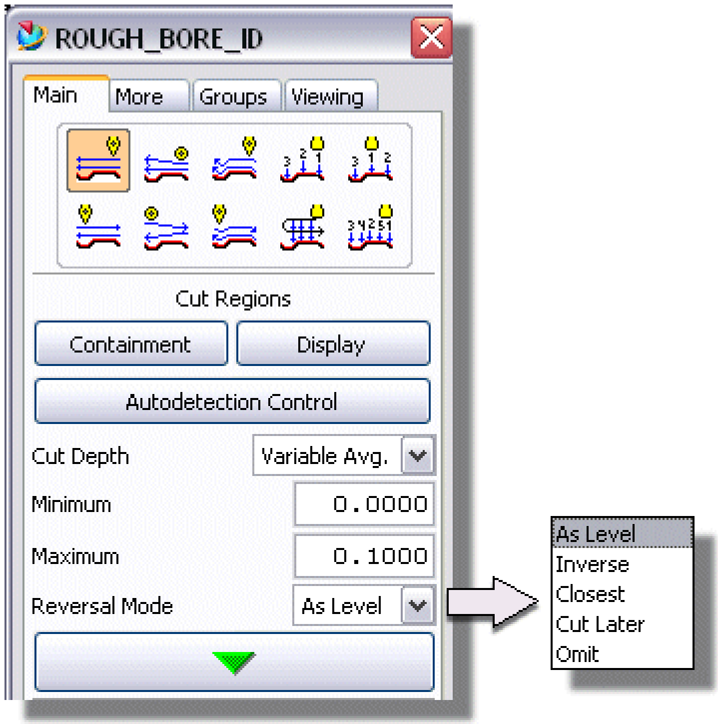


☐ Choose **OK** to complete the operation.

Step 8: Save and close the part.

Reversal Mode

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



- **As Level** completes cutting at one level within a reversal before moving to the next reversal within a cut region.
- **Inverse** reverses the cut order.
- **Closest** cuts the reversal that is the closest to the current tool position.
- **Cut Later** completes cutting the first reversal it comes to then moves to the next reversal.
- **Omit** does not cut the reversals.

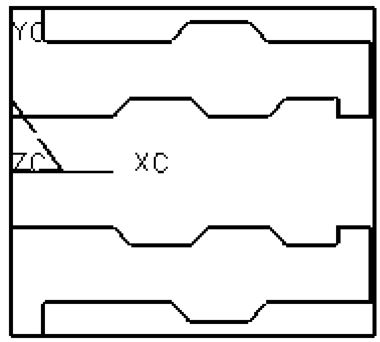
8

Activity: Roughing the ID – Reversals

You are going to examine the different reversal mode options. Reversal Mode determines the sequence in which the reversals of a cut region (i.e. the valleys of the part boundary found in this cut region) are addressed for material removal.

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

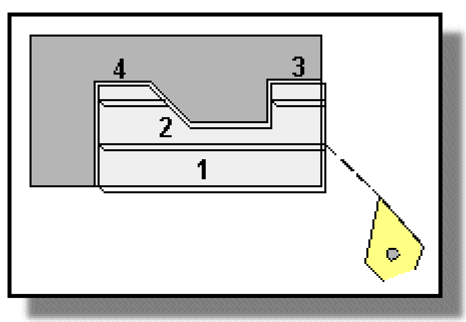
☐ Open the **tmp_bearing_sleeve_mfg_1** part.



☐ Save the part as *****_bearing_sleeve_mfg_1**, where *** represents your initials.

Step 2: Examine the As Level Reversal Mode.

The As Level Reversal Mode performs each rough cut at maximum reach into the reversal. As soon as a cut level is reached which leads into lower reversals, the system will continue to cut the reversal which comes next as seen in the direction of the level angle.

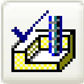


☐ Display the Program Order view and expand the T5678_B parent group.

☐ Double-click **ROUGH_BORE_ID** to edit the operation.

Notice that the Reversal Mode is set to As Level.

8

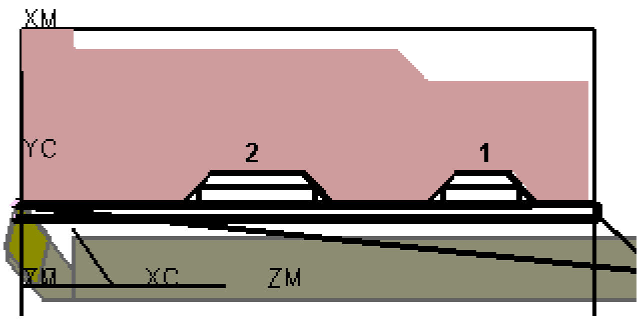
☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose the **Play**.

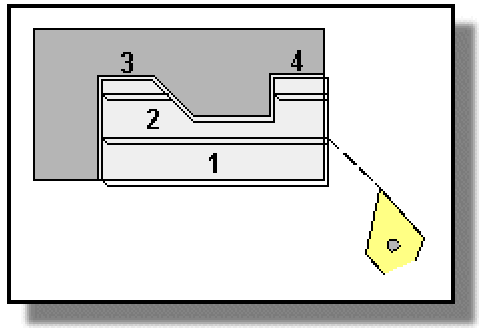
Note the order (1, 2) in which the reversals are cut.



☐ **OK** to complete the tool path visualization.

Step 3: Examine the Inverse Reversal Mode.

With an Inverse Reversal Mode, the reversals are cut in a pattern opposite to the As Level Mode. In other words, the system will cut the last reversal first and work backwards to the first reversal.

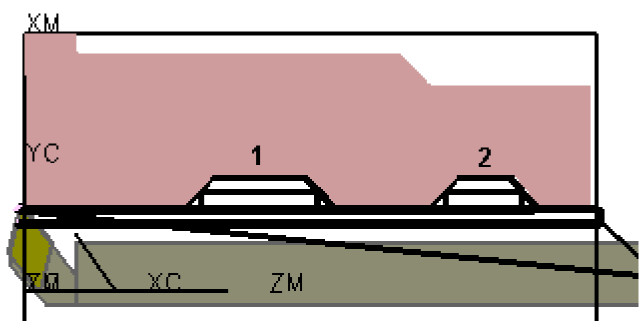


☐ Set the **Reversal Mode** option to **Inverse**.

☐ **Generate** the tool path.

☐ **Verify** the tool path as you did before.

Note the order (1, 2) in which the reversals are cut.



☐ **OK** to complete the tool path visualization.

8

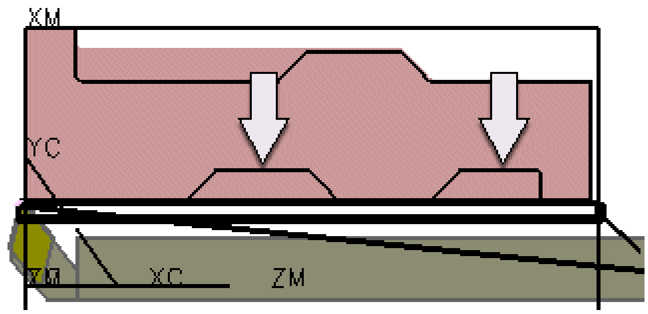
Step 4: Examine the Omit Reversal Mode.

The Omit option does not cut the reversals.

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

☐ **Generate** the tool path.

☐ **Verify** the tool path as you did before.



☐ **OK** to complete the tool path visualization.

Step 5: Examine the Closest Reversal Mode.

The Closest option is especially useful in combination with the zig-zag cut strategy, as the system will always choose to next cut that reversal which is closest to the current tool position. With a particularly complicated part boundary, this can save considerable machining time as tool travel is reduced.

☐ Change the reversal Mode to **Closest**.

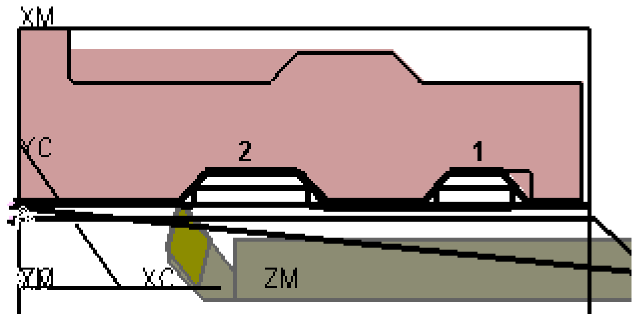
☐ Change the cut type to **Linear Zig-Zag**.

☐ **Generate** the tool path.

8

☐ **Verify** the tool path as you did before.

Note the order (1, 2) in which the reversals are cut.



The reversal on the right is cut first because it is the first reversal the tool encounters as it cuts along the OD from right to left (zig).

☐ **OK** to complete the tool path visualization.

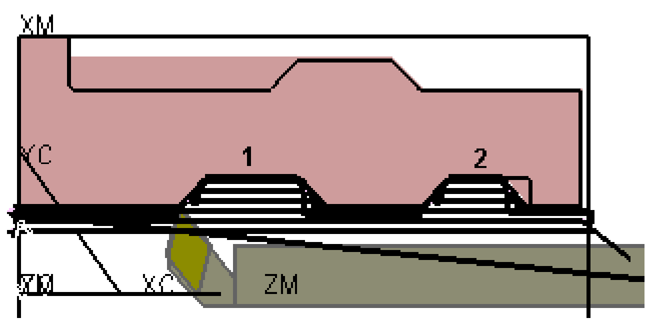
Next, you will change the maximum cut depth.

☐ Key in **0.1500** in the Maximum field. This will cause the tool to cut along the OD as it zags from left to right (zag).

☐ **Generate** the tool path.

☐ **Verify** the tool path as you did before.

Note the order (1, 2) in which the reversals are cut.



The reversal on the left is cut first because it is the first reversal the tool encounters as it cuts along the OD from right to left (zag).

☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

Step 6: Save and close the part.

Summary

In this lesson, you learned how to create roughing operations that remove ID material. You also explored the various reversal modes and observed how each mode affects the order in which reversals are cut.

Lesson

9 *Finish Operations OD and ID Work*

Purpose

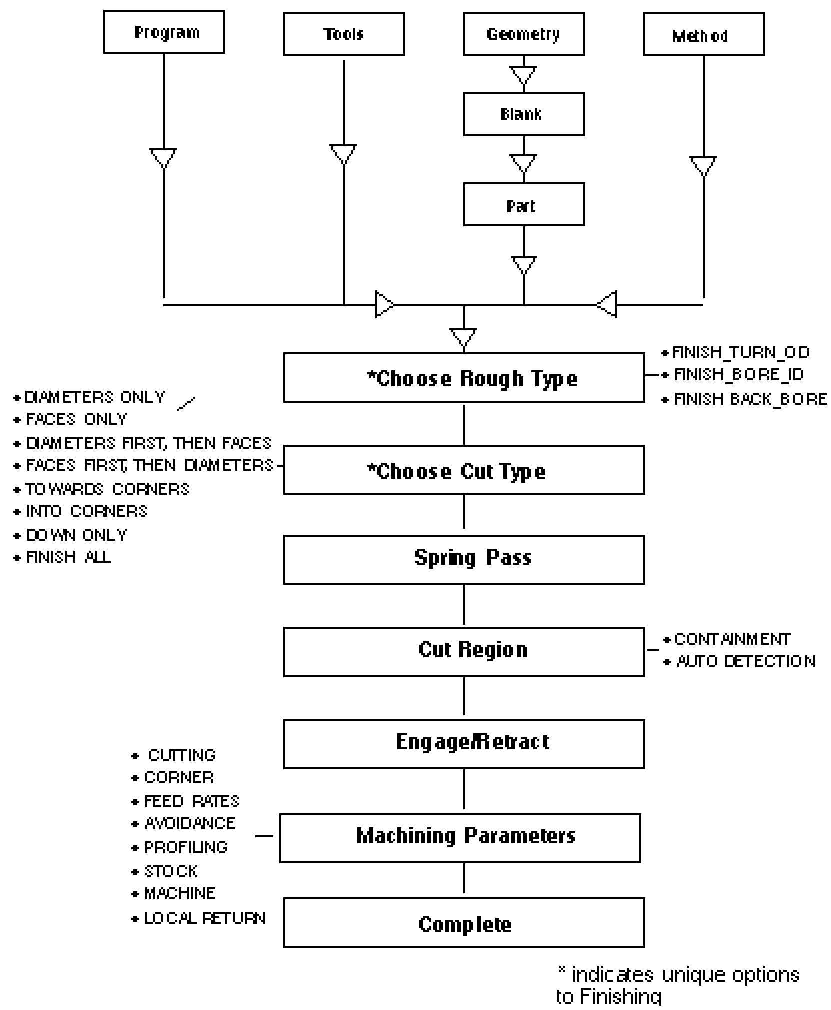
The Finish operation type is used to remove remaining material after roughing. Like roughing operations, finishing operations use the in-process workpiece to track the remaining material on the part.

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 convex corners.
- Use the Fillet options to finish the concave corners.

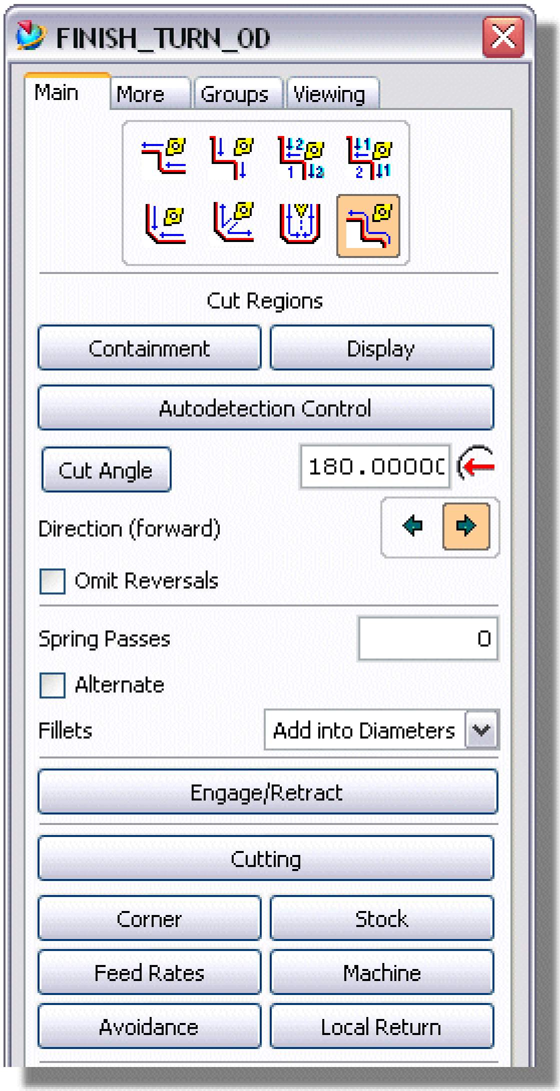
Creating Finishing Operations



9

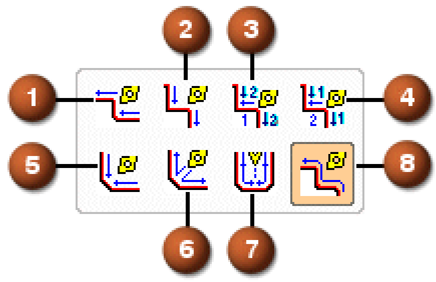
The FINISH_TURN options

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.



1. **Finish Diameters Only** creates finish cuts on diameters only.

2. **Finish Faces Only** creates finish cuts on faces only.

3. **Finish Diameter First, then Faces** creates finish cuts on all of the diameters, then creates finish cuts on the faces.

4. **Finish Faces First, then Diameters** creates finish cuts on all of the faces, then creates finish cuts on the diameters.

5. **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.

6. **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.

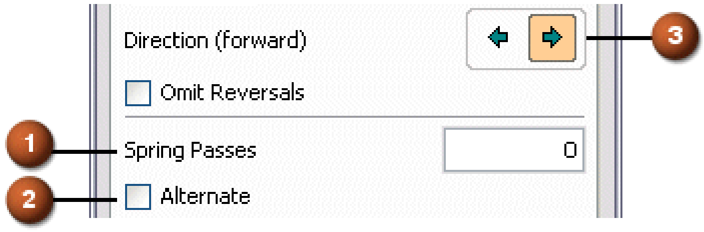
7. **Down Only** create finish cuts that always cut towards the bottom of the groove.

8. **Finish All** creates finish cuts that contour the boundary in a contiguous pass.

9

Spring Pass

The Spring Passes option (1) allows more than one finish pass.



Alternate

The Alternate option (2) alternates finish spring passes in a Zig Zag cut pattern.

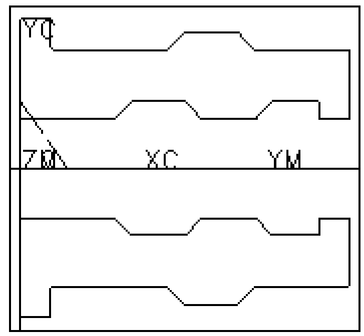
Direction

The Direction arrows (3) 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 **tmp_bearing_sleeve_mfg_2** part.



- ☐ Save the part as ***_**bearing_sleeve_mfg_2**, where *** represents your initials.

- ☐ Choose **Start→Manufacturing**.

Step 2: Examine the rough tool path.

- ☐ In the Operation Navigator, double-click **ROUGH_TURN_OD_1** to edit the operation.

- ☐ Choose **Display** in the Cut Regions area.

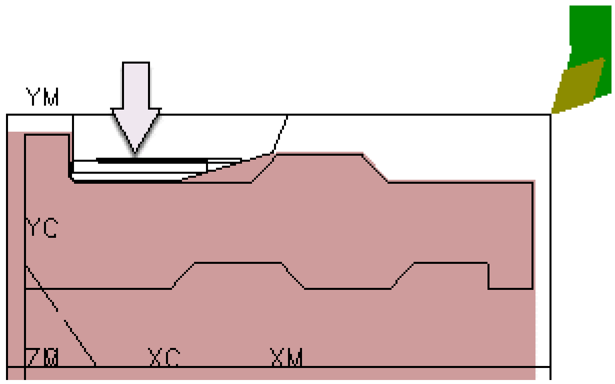
- ☐ Choose **Verify**  at the bottom of the dialog.

- ☐ Turn the **2D Material Removal** option **on**.

- ☐ Slow the **Animation Speed** down to **1**.

9

☐ Choose **Play**.



The tool removes most of the material in the reversal. You are going to create an OD Finish operation to remove the remaining material.

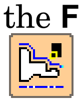
☐ **OK** to complete the tool path visualization.

☐ **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. 

☐ 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**.

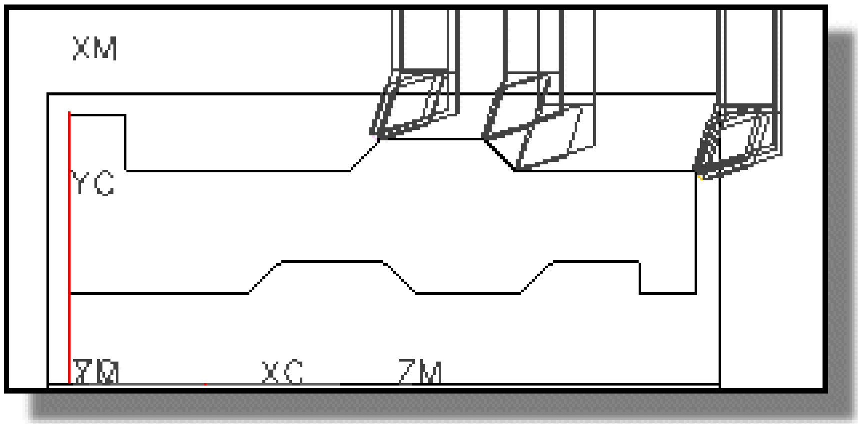
☐ Choose the **Main** tab.

9

Step 4: Generate the tool path and examine the results.

☐ Set the Tool Display to **2-D**, and **Generate** the tool path.

The tool does not cut the entire OD. Although the cut type is set to **Finish All**, the tool stops before the left reversal.



Step 5: Turn on Machine all Regions

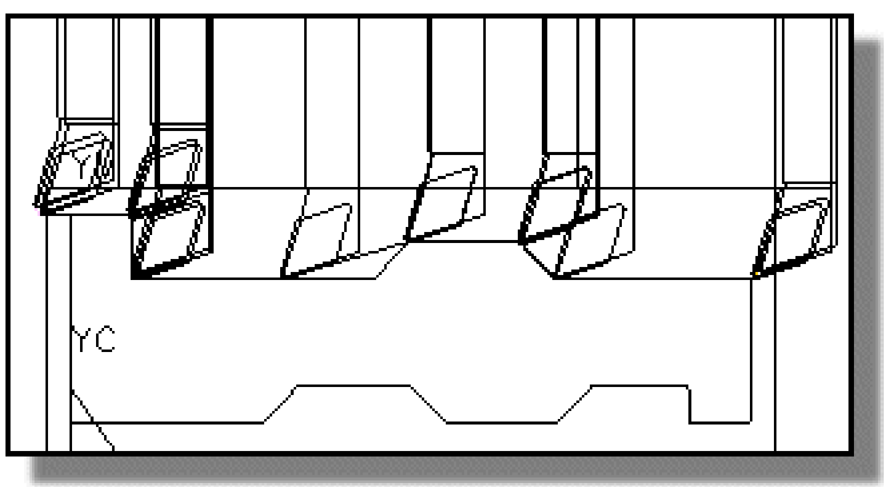
You will activate the Machine all Regions option to finish the outside diameter in a single operation.

☐ Choose **Cutting**.

☐ Turn the **Machine all Regions** option **on**.

☐ **OK** the Cutting dialog.

☐ Generate the tool path.



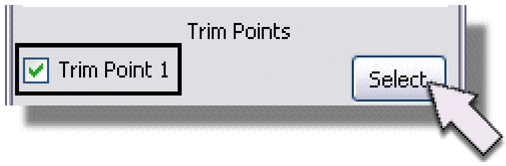
The tool now cuts the entire outside diameter, including the material to the left of the reversal area. However, the tool proceeds too far into the blank, where it could collide with the chuck.

Step 6: 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.

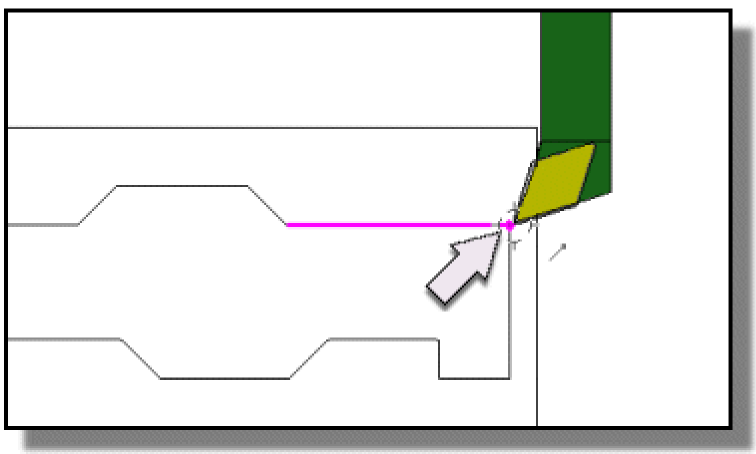
☐ Choose **Containment** under Cut Regions.

☐ 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.

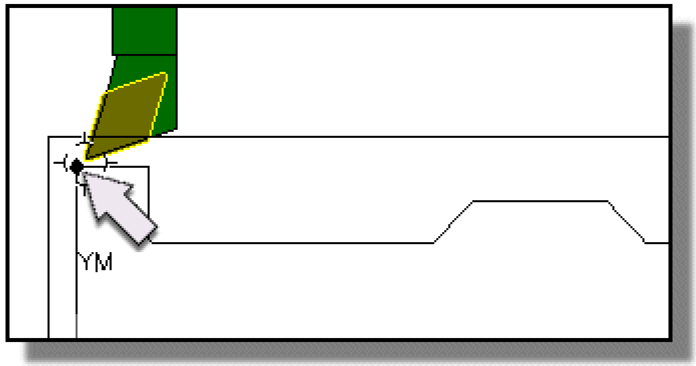


The point is labeled TP 1.

☐ Turn on **Trim Point 2** and choose **Select**.

9

☐ Select the end of the horizontal line near the back of the part.

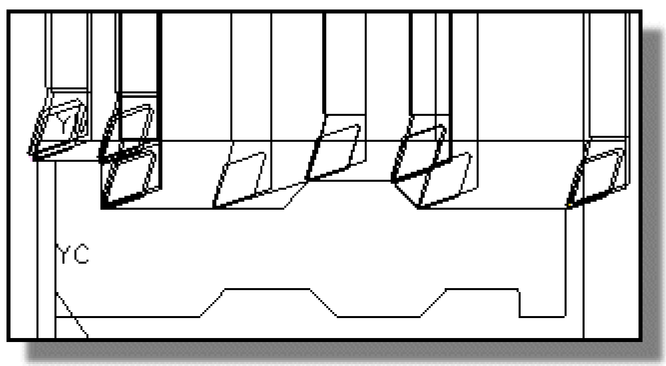


The point is labeled TP 2.

☐ Choose **OK** to accept the trim points.

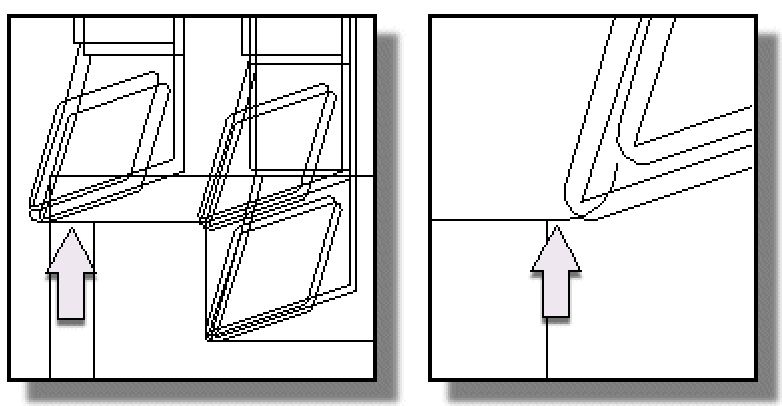
Step 7: Generate the tool path and examine the results.

☐ **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.



9

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 8: 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** the **Direct retract from trim point** option.

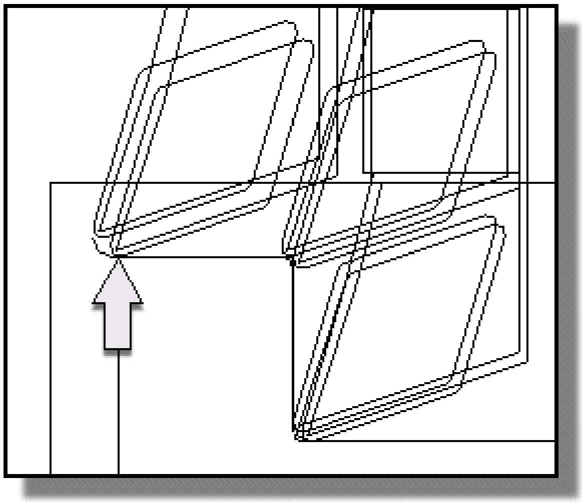
When Extend Distance is set to 0.0000, the cutting move to the second trim point is completely eliminated. Entering a value for Extend Distance determines the length of the cutting move following the trim point.

☐ **OK** the Engage/Retract dialog.

Step 9: Generate the tool path and examine the results.

☐ **Generate** the tool path.

The tool now retracts directly from the second trim point.

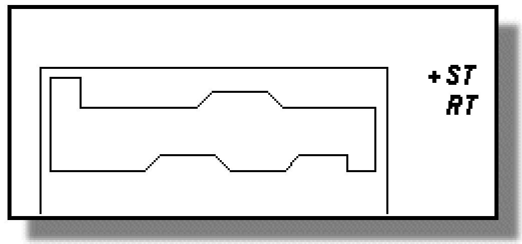


Next, you will define Avoidance geometry.

Step 10: Define the avoidance geometry.

There are no Start and Return points defined. You will define both for this operation.

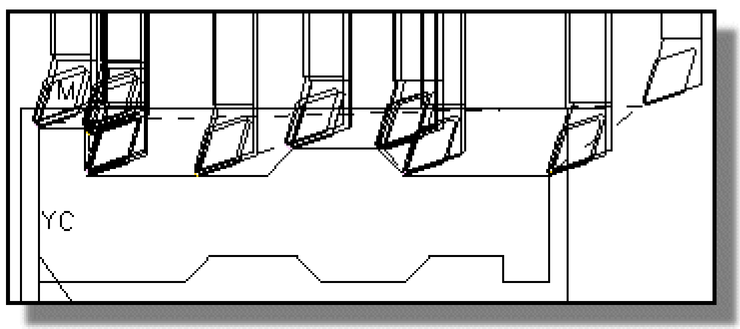
☐ Define a **Motion to Start Point** and **Motion to Return Point** as shown.



☐ Set the **Motion Type** to **Axial→Radial** for both points.

Step 11: Generate the tool path.

☐ Choose the **Generate** icon.



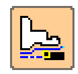
The tool cuts to the finish dimension of the part.

Step 12: Save the part.

Activity: Creating a Finish ID Operation

In this activity, you are going to create an ID Turning Operation with limited assistance.

Step 1: Create a new operation using the finish bore subtype.

- ☐ Continue to use the *****_bearing_sleeve_2_mfg** part.
- ☐ Choose the **Create Operation** icon.
- ☐ Choose the **FINISH_BORE_ID** icon. 

Step 2: Define the Parent Groups.

- ☐ Program..........**PROGRAM**
- ☐ Use Geometry....**PART**
- ☐ Use Tool..........**ID_55_L**
- ☐ Use Method......**LATHE_FINISH**

Step 3: Define the following options for this operation:

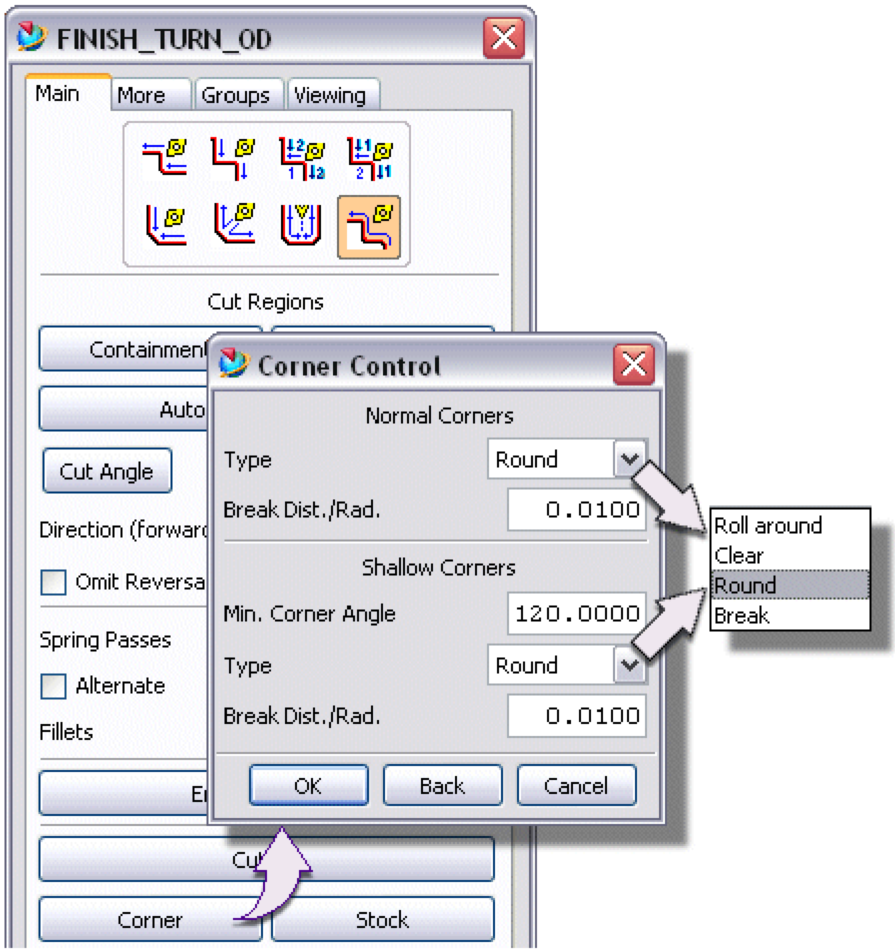
- ☐ Start and Return Point
- ☐ Omit Reversals **on**

Step 4: Generate the tool path.

Step 5: Save and close the part.

Corner Control

The Corner Control option allows you to control tool path when the tool cuts boundary corners. You can remove a specified amount of material or none at all.



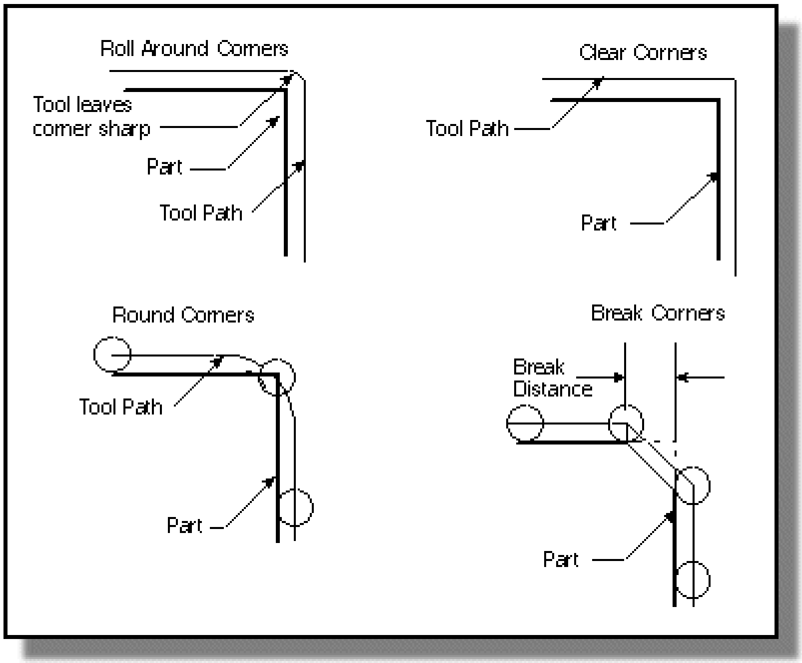
Type Options

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.

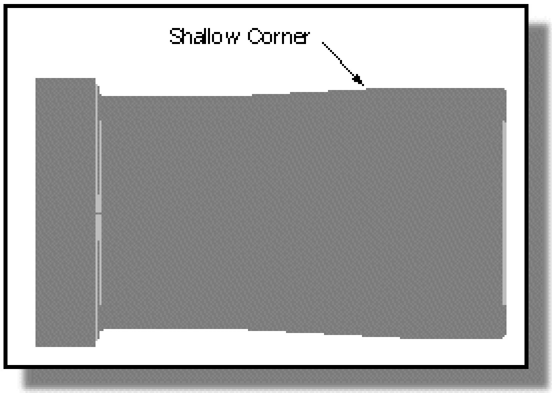
Round Corners– adds a radius to the corner. You enter the desired radius value.

Break Corners– allows chamfering of corners. The tool moves the specified distance in a straight line across the corner.



Shallow Corners

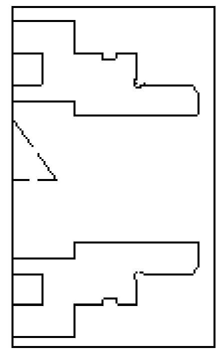
This option allows you to control tool motion when convex corners has an included angle larger than 180 degrees.



Activity: Using Corner Control

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

☐ Open the **tmp_fin_od_mfg_1** part.



☐ Save the part as *****_fin_od_mfg_1**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

Step 2: Edit an existing operation.

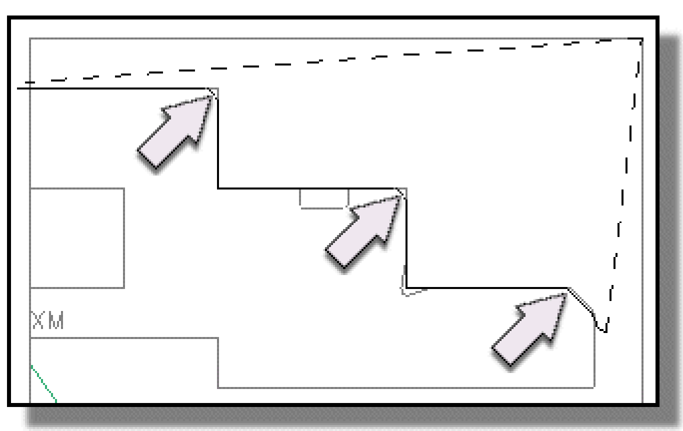
☐ Double-click **FINISH_TURN_OD** in the Operation Navigator to edit the operation.

☐ Choose **Replay** from the FINISH_TURN_OD dialog.

Step 3: Examine the Clear Normal Corners option.

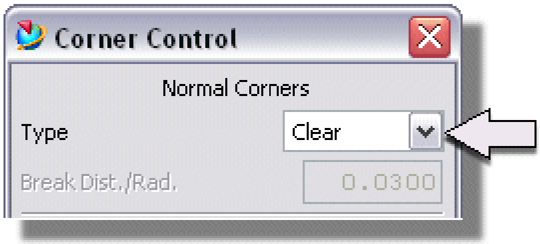
☐ Turn the **Tool Display** option **off** and **Replay** the tool path.

The tool rounds the corners. This is the default setting for the Corner Control option.



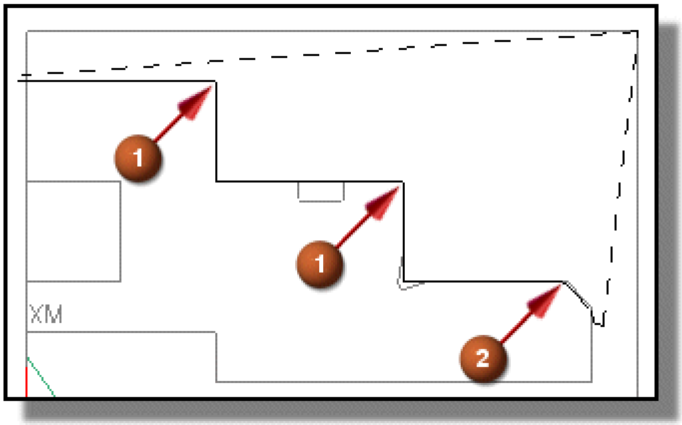
☐ Choose the **Corner** button.

☐ 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 (1). The tool still rolls around the Shallow Corner (2).



Step 4: Examine the Round Normal Corners option.

Generally, you have to round the sharp corners and push the burr in front of the tool.

☐ Again, choose the **Corner** button.

☐ In the Normal Corners area, change the Type to **Round**.

☐ Key in **0.0500** in the Break Dist/Rad field.

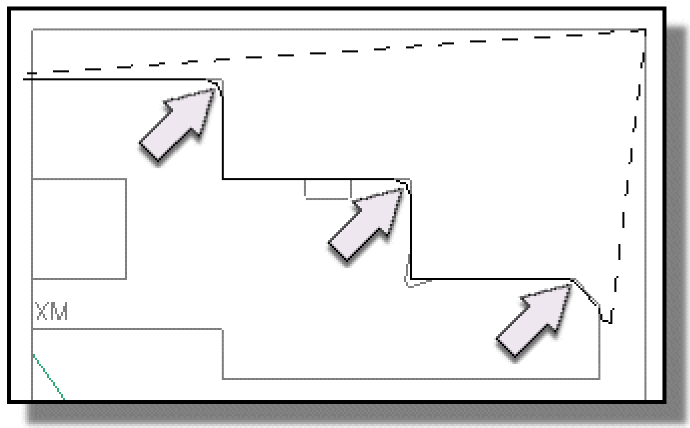
☐ Choose **OK**.

The FINISH_TURN_OD dialog is displayed.

9

☐ **Generate** the tool path.

The tool path is generated. The tool rounds the corners using a 0.0500 radius.



Step 5: Verify the tool path.

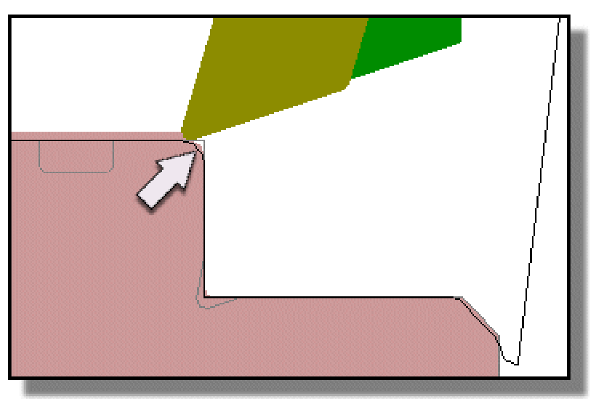
☐ Choose **Verify**  at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose **Play**.

You can clearly see that the tool rounds the corner with a 0.0500 radius.



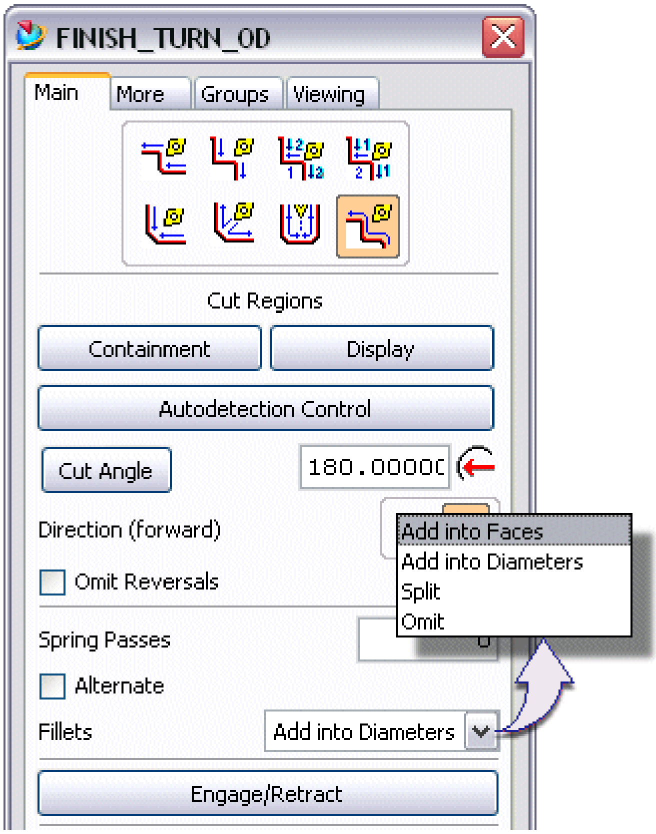
☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

Step 6: Save and close the part.

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.

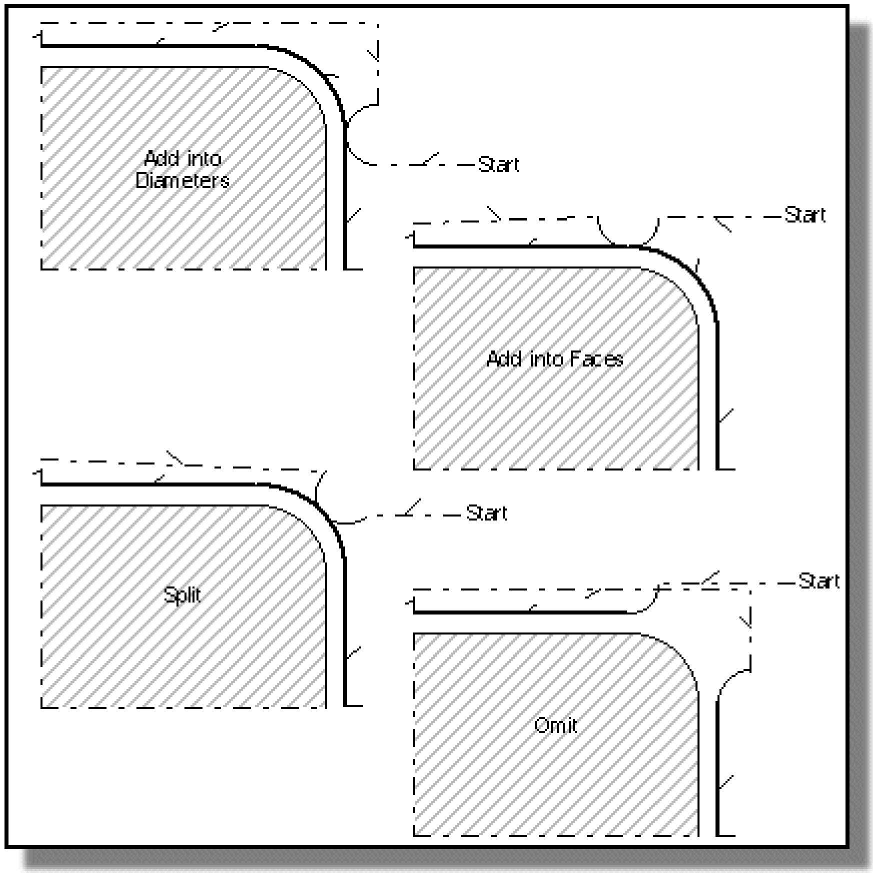
Add into Diameters - cuts the fillets when cutting the adjacent diameters. For example, if you selected Diameters First, Then Faces as the cut type, the fillets are cut when the adjacent diameters are cut rather than the face.

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.

9

The following illustrations show the fillet options for "Diameters then Faces" with auto circular engage and auto linear retract.

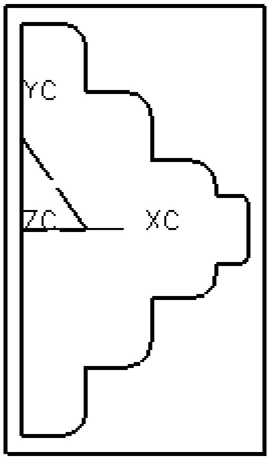


9

Activity: Cutting Fillets

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

☐ Open the **tmp_fillets_mfg_1** part.

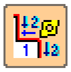


☐ Save the part as *****_fillets_mfg_1**, where *** represents your initials.

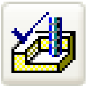
☐ Choose **Start→Manufacturing**.

Step 2: Observe the Add into Faces option.

☐ Double-click **FINISH_OD** in the Operation Navigator to edit the operation.

Notice the cut type is set to **Diameters First, Then Faces**  and the Fillets option is set to **Add into Faces**.

☐ **Generate** the tool path.

☐ Choose **Verify**  at the bottom of the dialog.

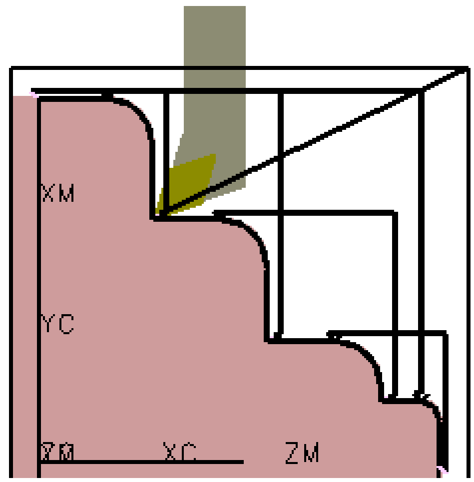
☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

9

- ☐ Choose **Play**.

  The fillets are cut when the adjacent faces are cut because the cut type is set to **Diameters First, Then Faces** and the Fillets option is set to **Add into Faces**.

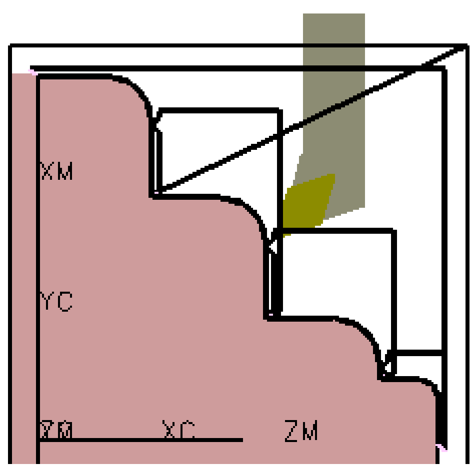


- ☐ **OK** to complete the tool path visualization.

Step 3: Observe the Add into Diameters option.

- ☐ Set the **Fillets** option to **Add into Diameters**.
- ☐ **Generate** the tool path.
- ☐ **Verify** the tool path as you did before.

  The fillets are cut when the adjacent diameters are cut because the cut type is set to **Diameters First, Then Faces** and the Fillets option is set to **Add into Diameters**.



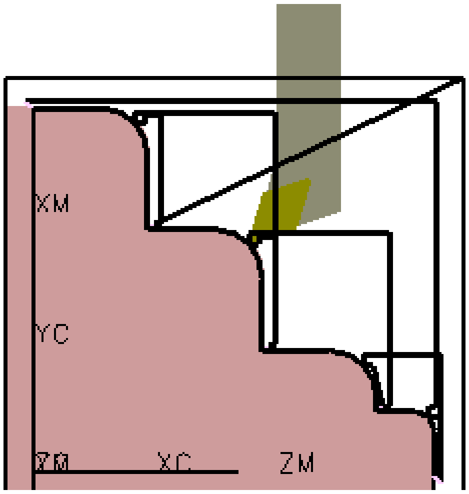
- ☐ **OK** to complete the tool path visualization.

9

Step 4: Observe the Split option.

- ☐ Set the **Fillets** option to **Split**.
- ☐ **Generate** the tool path.
- ☐ **Verify** the tool path as you did before.

  Half of the fillet is cut when cutting the adjacent face and the other half of the fillet is cut when cutting the adjacent diameter.



- ☐ **OK** to complete the tool path visualization.
- ☐ **OK** to complete the operation.

Step 5: Save and close the part.

Summary

In this lesson, you created Finish OD and ID operations, specified trim points and explored various corner control and fillet options.

Lesson

10 *Grooving*

Purpose

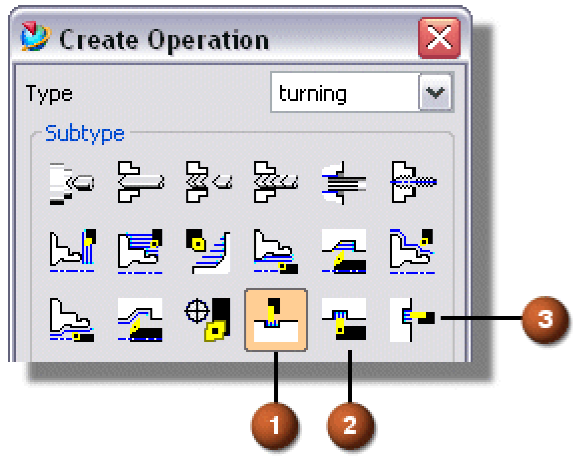
Grooving is a roughing operation that is used to remove material in grooves or undercut areas.

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.

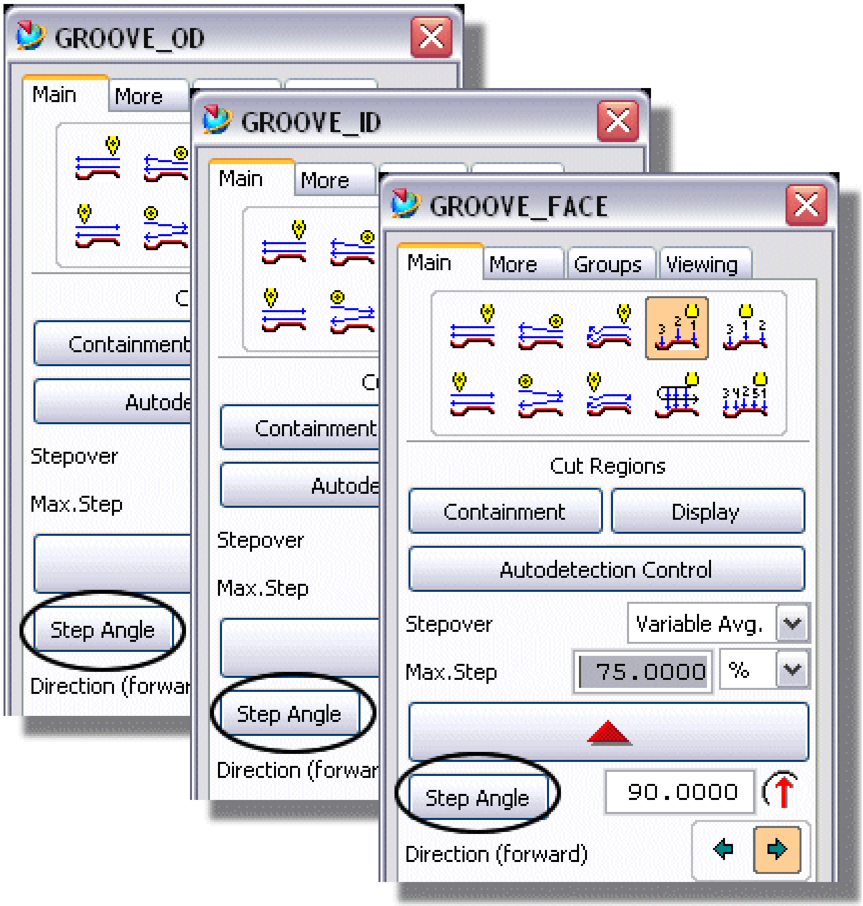
Grooving Operation Types



1. GROOVE_OD
2. GROOVE_ID
3. GROOVE_FACE

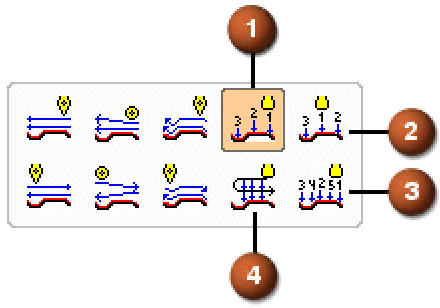
The Groove Options

The Groove options located on the Groove OD, Groove ID, and Groove Face dialogs are identical, with the exception of the Step Angle. The Step Angle determines the angle at which the tool will cut.


GROOVE_OD
Main
More
Containment
Autode
Stepover
Max.Step
Step Angle
Direction (forwar
GROOVE_ID
Main
More
Containment
Autode
Stepover
Max.Step
Step Angle
Direction (forwar
GROOVE_FACE
Main
More
Groups
Viewing
Cut Regions
Containment
Display
Autodetection Control
Stepover
Variable Avg.
Max.Step
75.0000
%
Step Angle
90.0000
Direction (forward)

Cut Patterns

There are four plunging cut patterns.



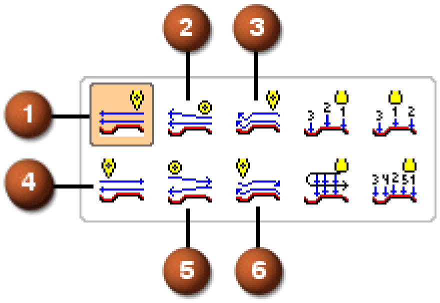
1. **Plunge Zig** are typical plunge cuts in one direction within the cut region.

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

3. **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.

4. **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.



1. **Linear Zig** creates linear cuts by level in one direction within the cut region.

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

3. **Contour Zig** creates cuts by level that follow the contour of the groove in one direction within the cut region.

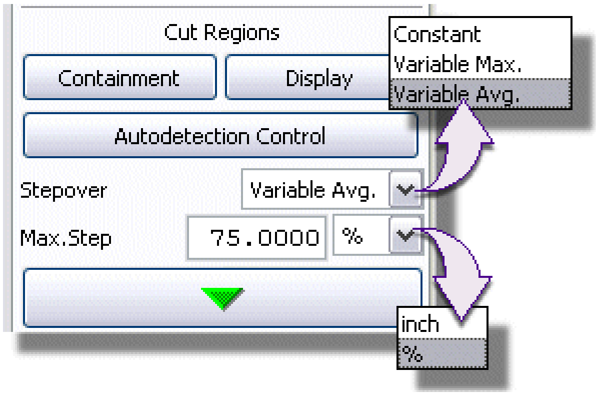
4. **Linear Zig-Zag** creates linear cuts by level in both directions within the cut region.

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

6. **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.



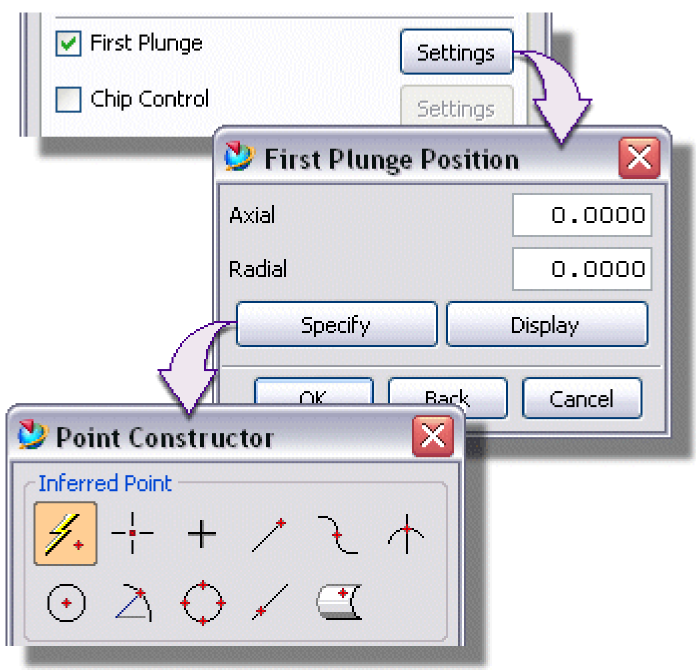
The Stepover options are defined in the table below.

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.

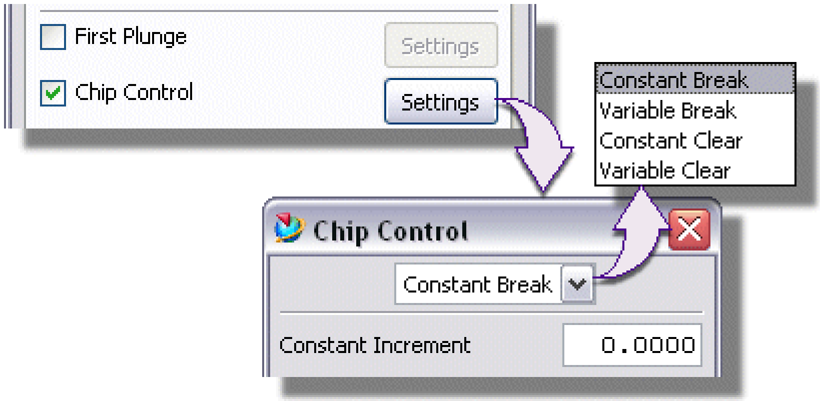
First Plunge

The first plunge of the grooving tool path is automatically determined. The option, First Plunge, allows you to reposition the first plunge by defining an axial or radial position or by defining a point.



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.

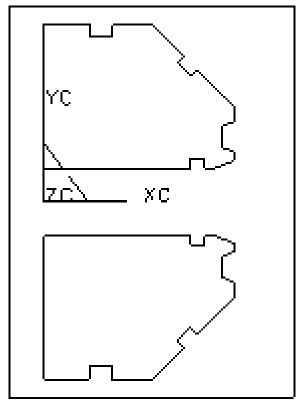


Activity: Creating an OD Groove Operation

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 **tmp_grv_all_mfg_1** part.



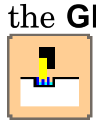
☐ Save the part as *****_grv_all_mfg_1**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

Step 2: Create an outside diameter grooving operation.

☐ Choose the **Create Operation** icon.

☐ Set the **Type** option 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**

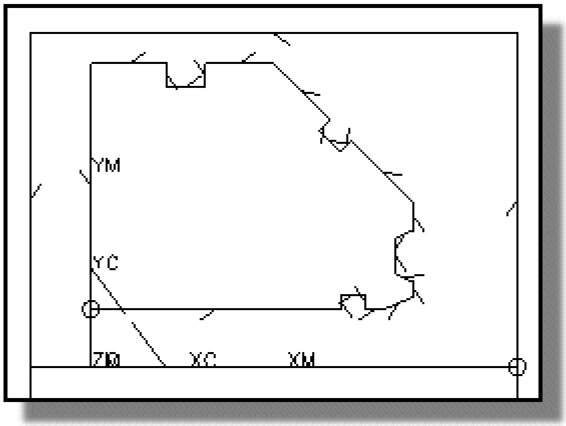
☐ Choose **OK**.

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.

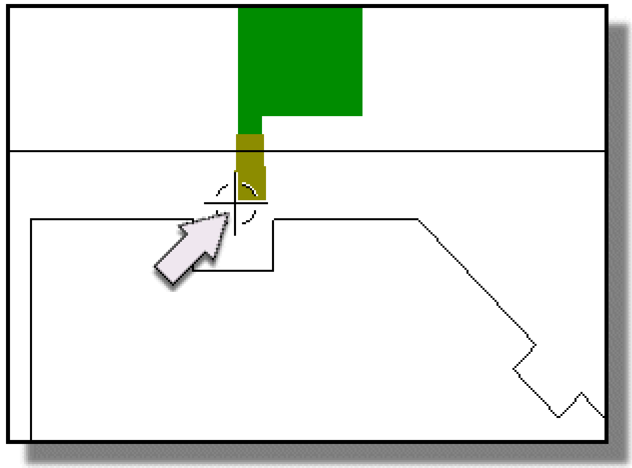
Step 4: Specify the cut region.

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.

☐ Choose **Containment** under Cut Regions.

☐ Turn the **Select Manually** option **on** and then choose **Select** under Cut Regions.

☐ Use **Cursor Location** to select near the desired cut region as illustrated below.

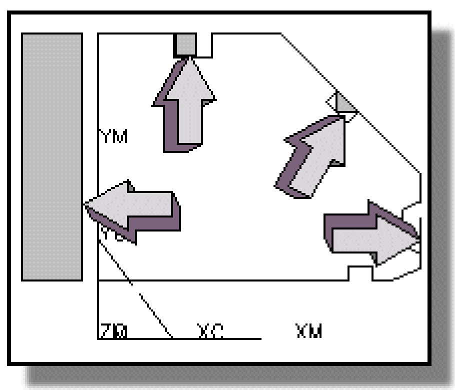


☐ Choose **OK** to accept the cut region.

Step 5: Display the cut regions.

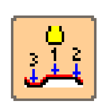
☐ Choose **Display** under Cut Regions.

Notice that there are multiple cut regions. The operation will be performed on the region closest to the RSP point.



Step 6: Change the cut type.

You will plunge using alternating stepover directions.

☐ Choose the **Plunge Alternate** icon. 

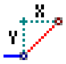
☐ Turn the **Cleanup** option **off**.

☐ Turn the **Additional Profiling** option **on**.

Step 7: Set Engage Parameters

☐ Choose **Engage / Retract**.

☐ On the **Engage** tab, make sure **Profiling** is selected.

☐ Choose **Vector**. 

☐ In the **Vector (Y)** field, enter **-.06**.

☐ **OK** to accept the Engage / Retract parameters

Step 8: Generate and verify the tool path.

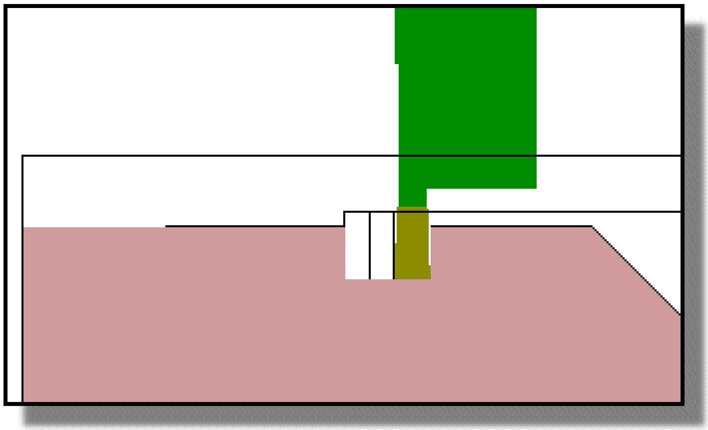
☐ Choose the **Generate** icon.

☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose **Play**.



☐ **OK** to complete the tool path visualization.

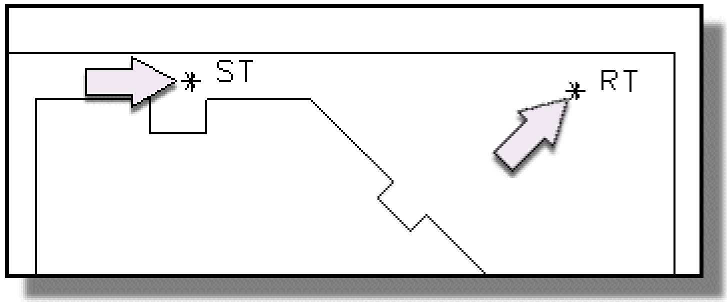
Step 9: Define avoidance parameters.

You will define a Start Point, a Return Point.

☐ Choose **Avoidance**.

10

☐ Define a **Motion to Start Point** and **Motion to Return Point** as shown.

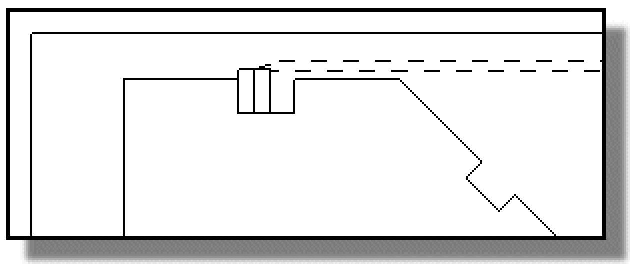


☐ Choose **Radial→Axial** for the **Motion Type** for both points.

☐ Choose **OK** to accept the Avoidance Parameters.

Step 10: Generate the tool path.

☐ Choose the **Generate** icon.

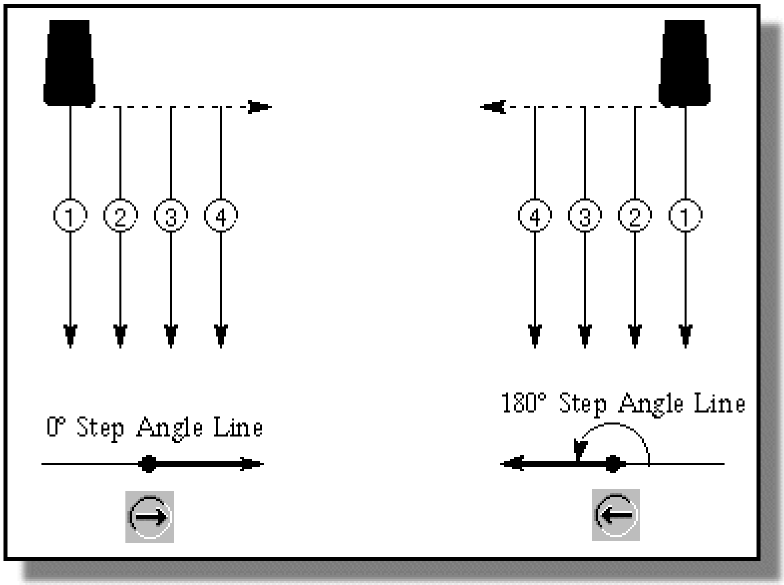


☐ Choose **OK** to complete the operation.

☐ Save the part.

Step Angle

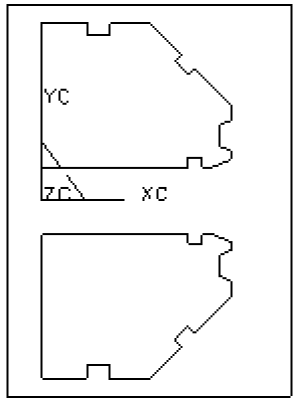
The Step Angle allows you to create plunge cuts at any angle. Plunge cuts are perpendicular to the step angle line and are generated in sequence in the direction of the Step Angle line, indicated in the dialog by a red arrow.



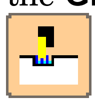
Activity: Creating an Angled OD Groove Operation

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 *****tmp_grv_all_mfg_1** part.



Step 2: Create an outside diameter grooving operation.

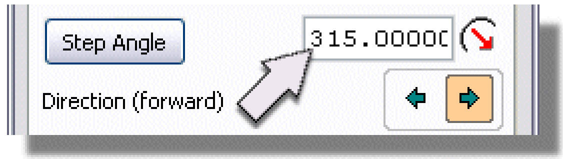
- ☐ Choose the **Create Operation** icon.
- ☐ Choose the **GROOVE_OD** icon from the Create Operation dialog. 
- ☐ Set the following Parent Groups:
  - Program..........**PROGRAM**
  - Use Geometry....**PART**
  - Use Tool..........**GRV_45_DEG**
  - Use Method......**LATHE_GROOVE**
- ☐ Key in GRV_ANGLE for the name.
- ☐ Choose **OK**.

10

Step 3: Change the step angle.

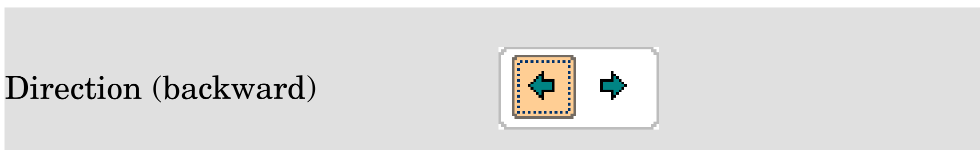
You will change the Step Angle so that the plunges cut directly into the groove.

☐ 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.

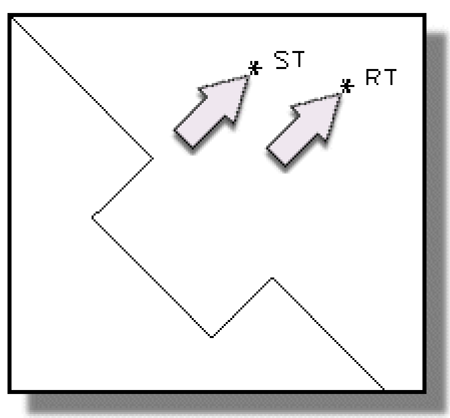
☐ Change the **Direction** to **Backward**.



Step 4: Define avoidance parameters.

☐ Choose **Avoidance**.

☐ Define a **Motion to Start Point** and **Motion to Return Point** as shown.

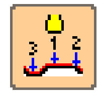


☐ Choose **OK** to accept the Avoidance Parameters.

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.

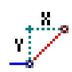
☐ Turn the **Cleanup** option **off**.

☐ Turn the **Additional Profiling** option **on**.

Step 7: Set the Engage and Retract parameters

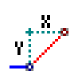
☐ Choose **Engage / Retract**.

☐ On the **Engage** tab, make sure**Profiling** is selected.

☐ Choose **Vector**. 

☐ Key in **-.06** in the **Vector (X)** and **Vector (Y)** fields.

☐ Choose the **Retract** tab and make sure **Profiling** is selected.

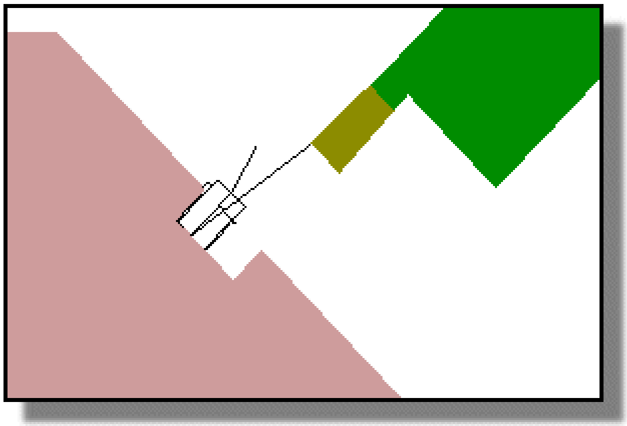
☐ Choose **Vector**. 

☐ Key in **.1** in the **Vector (X)** and **Vector (Y)** fields.

☐ **OK** to accept the Engage / Retract parameters

Step 8: Generate and verify the tool path.

- ☐ Choose the **Generate** icon.
- ☐ Choose **Verify**.
- ☐ Turn the **2D Material Removal** option **on**.
- ☐ Slow the **Animation Speed** down to **1**.
- ☐ Choose **Play**.

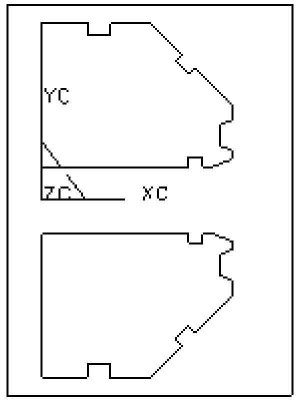


- ☐ **OK** to complete the tool path visualization.
- ☐ Choose **OK** to complete the operation.
- ☐ Save the part.

Activity: Creating a Face Groove Operation

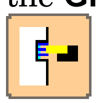
In this activity, you are going to create an operation that cuts a groove into the front face of the part using a linear zig-zag cut motion.

Step 1: Continue to use the part file *****_grv_all_mfg_1.prt**.



Step 2: Create a face groove operation.

☐ Choose the **Create Operation** icon.

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

☐ Set the following Parent Groups:

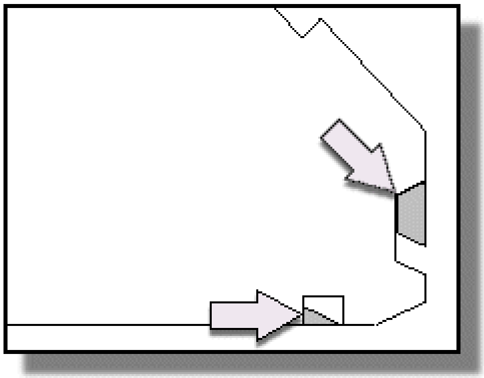
- Program..........**PROGRAM**
- Use Geometry....**PART**
- Use Tool..........**RTJ_GROOVE_TOOL**
- Use Method......**LATHE_GROOVE**

☐ Choose **OK**.

Step 3: Display the cut regions.

☐ Choose **Display** under Cut Regions.

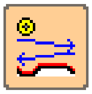
Notice the two cut regions at the lower right corner of the part.



You will allow the system to automatically determine which cut region to use based on the Start Point position.

Step 4: 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 5: 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.

☐ Key in **0.100** in the Maximum field.

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.

☐ Turn the **Cleanup** option **off**.

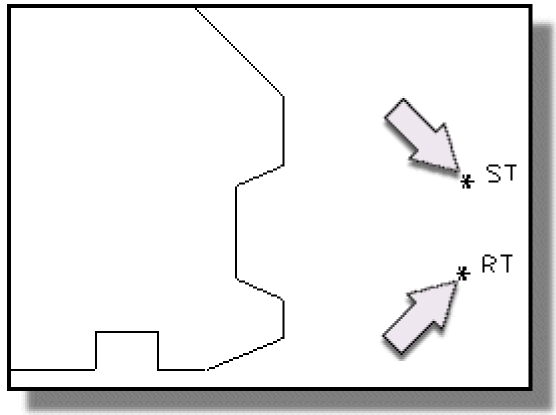
☐ Turn the **Additional Profiling** option **on**.

Step 7: 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 motion type.



☐ Choose **OK** to accept the Avoidance Parameters.

Step 8: Define the ramping mode.

☐ Choose **Cutting** and select the **Settings** button next to **Ramping**.

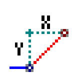
☐ Set the **Ramping Mode** to **Ramp In First**.

☐ **OK** to accept the ramping.

Step 9: Set Engage parameters.

☐ Choose **Engage / Retract**.

☐ On the **Engage** tab, make sure **Profiling** is selected.

☐ Choose **Vector**. 

☐ Key in **-.06** in the **Vector (X)** field.

☐ **OK** to accept the Engage / Retract parameters

Step 10: Generate and verify the tool path.

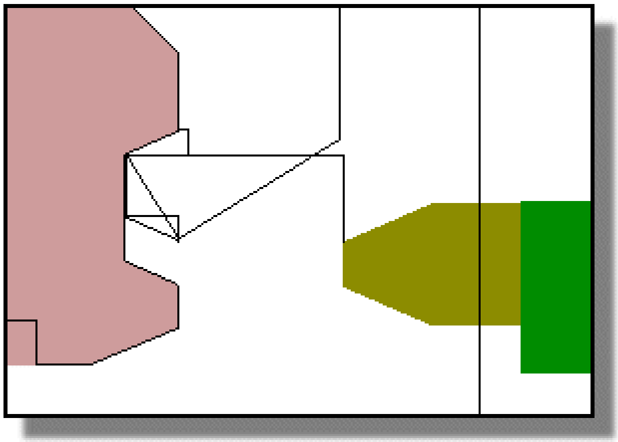
☐ Choose the **Generate** icon.

☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose **Play**.



☐ **OK** to complete the tool path visualization.

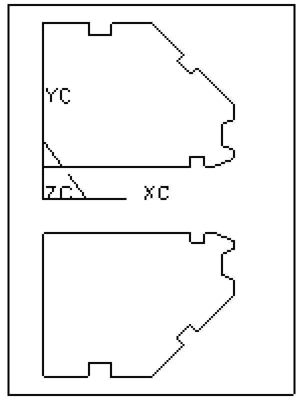
☐ Choose **OK** to complete the operation.

☐ Save the part.

Activity: Creating an ID Groove Operation

In this activity, you are going to create a simple outside diameter grooving operation.

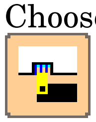
Step 1: Continue to use the *****_grv_all_mfg_1** part.



Step 2: Create an inside diameter grooving operation.

☐ 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**

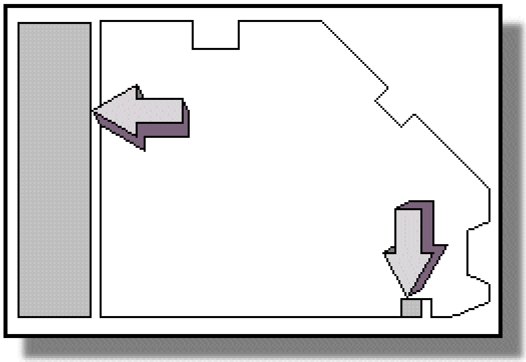
☐ Choose **OK**.

10

Step 3: Display the cut regions.

☐ Choose **Display** under Cut Regions.

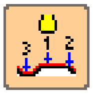
Notice that there are two cut regions.



You will allow the system to automatically determine which cut region to use based on the Start Point position.

Step 4: Change the cut type.

You will plunge using alternating stepover directions.

☐ Choose the **Plunge Alternate** icon. 

Step 5: 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 %.

☐ Turn the **Cleanup** option **off**.

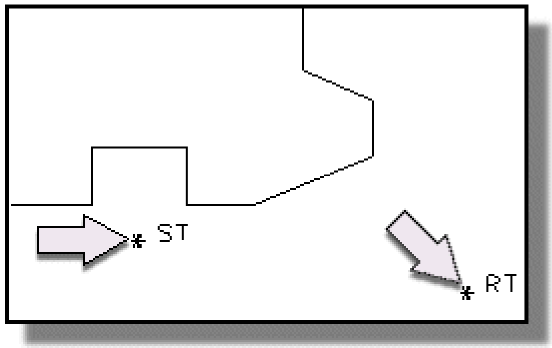
☐ Turn the **Additional Profiling** option **on**.

Step 6: 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 with a **Radial→Axial** motion type.

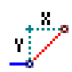


☐ Choose **OK** to accept the Avoidance Parameters.

Step 7: Set the Engage and Retract parameters

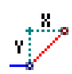
☐ Choose **Engage / Retract**.

☐ On the **Engage** tab, make sure**Profiling** is selected.

☐ Choose **Vector**. 

☐ Key in **.1** in the **Vector (Y)** field.

☐ Choose the **Retract** tab and make sure **Profiling** is selected.

☐ Choose **Vector**. 

☐ Key in **.06** in the **Vector (X)** field.

☐ **OK** to accept the Engage / Retract parameters

Step 8: Generate the tool path.

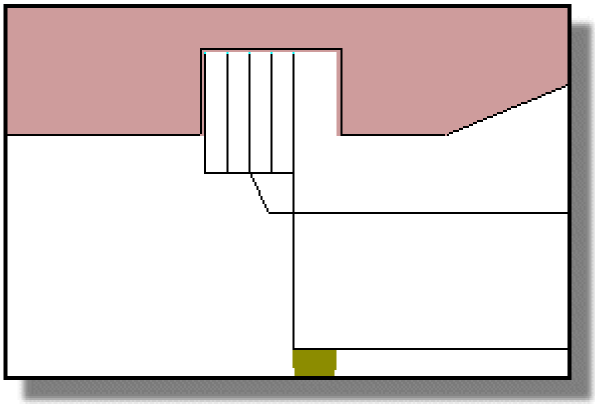
☐ Choose the **Generate** icon.

☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose **Play**.



☐ **OK** to complete the tool path visualization.

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

☐ Save and close the part.

Summary

Grooving removes material in grooves or undercut areas.

In this lesson you:

- Learned how to create outside diameter, inside diameter, and face groove operations.
- 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

11 *Teach Mode*

Purpose

Teach Mode allows you to define cutting and non-cutting moves individually and in sequence. It is useful for finishing in areas that require a high degree of cutter control.

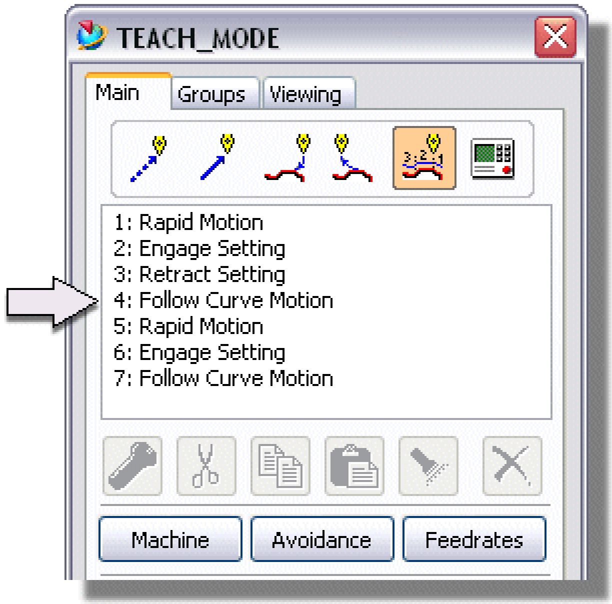
Objective

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

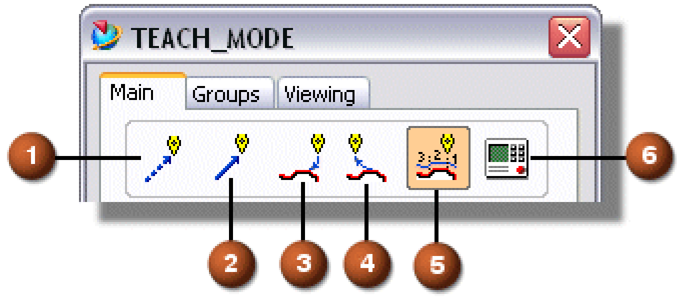
- Identify the different types of suboperations.
- Determine the sequence of the suboperations.
- Identify the Drive geometry.
- Create a Teach Mode operation that cut faces and diameters.

The Teach Mode options

Teach Mode allows you to create a sequence of suboperations that define individual cutting and non-cutting motions.



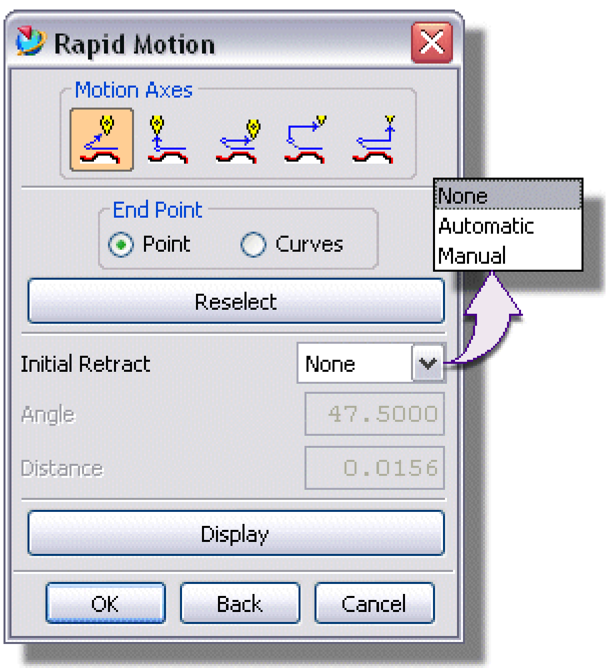
Suboperation Types



1. Linear Rapid
2. Linear Feed
3. Engage Setting
4. Retract Setting
5. Follow Curve Motion
6. Machine Control Events

Linear Rapid

Linear Rapid allows you to define a point from which the tool will move to the engage position and the motion it will use to get there.



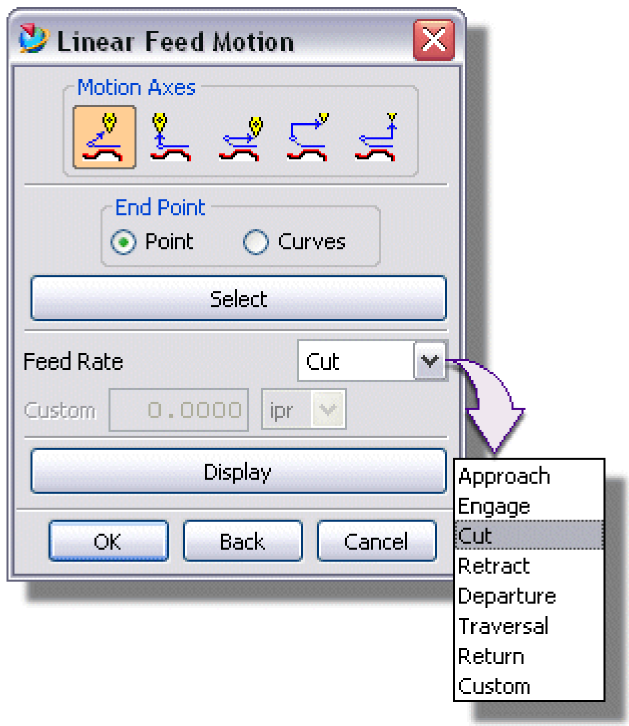
The Motion Axis options allow you to define various radial and axial movements for the rapid motion.

The End Point options allow you to define a point to which the tool will rapid.

The Initial Retract options allow 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.

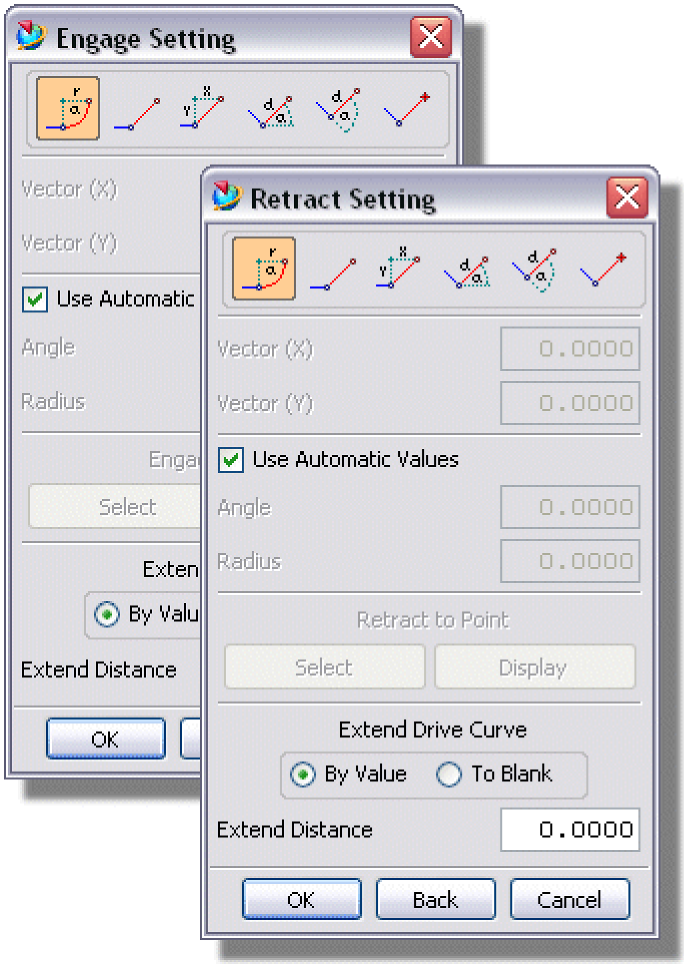
Linear Feed

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 different 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 suboperations.



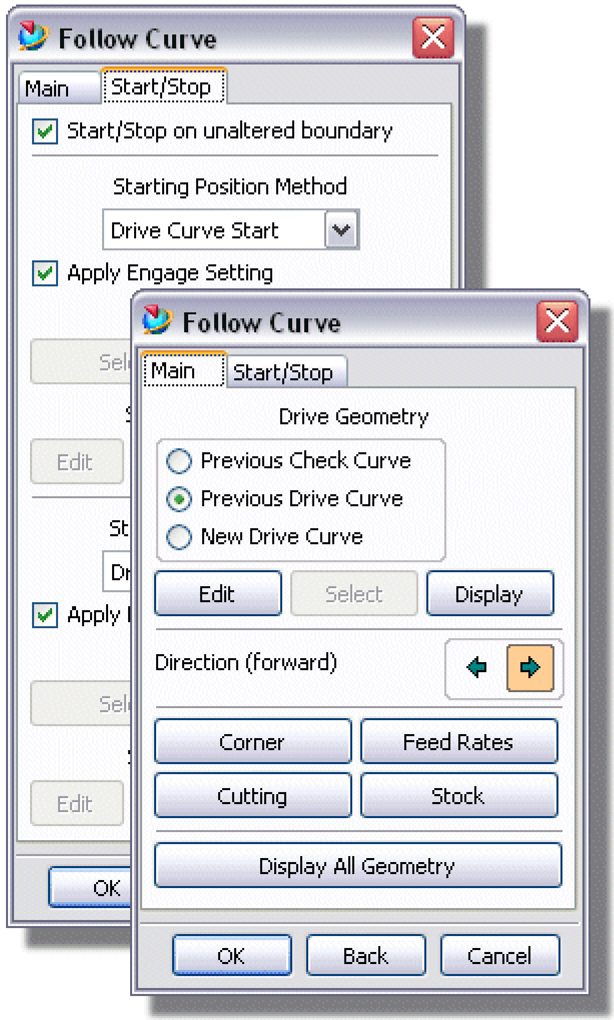
Extend Drive Curve is used to create overlapping tool paths on the drive curve starts/ends within the In-process Workpiece. This will allow the tool to engage TO rather than INTO the blank.

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 suboperation(s).

Follow Curve Motion

Follow Curve allows you to define the drive geometry along selected curve segments and to specify starting and stopping points along those segments.



The Point start/stop positioning method gives you precise control of the starting/stopping positions. You can also specify geometry that defines engage and retract positions.

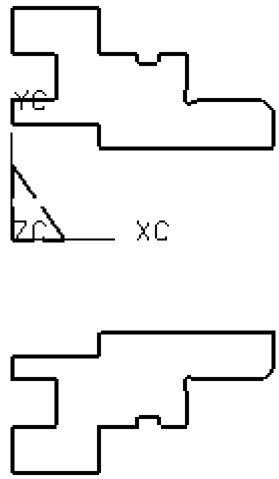
Option	No Previous Follow Curve	Previous Follow Curve
Previous Check Curve		Uses check geometry as stop position method
Previous Drive Curve		Uses same geometry as Previous Follow Curve
New Drive Curve	Selects new curve to machine	

Activity: Creating a Teach Mode Operation

In this activity, you are going to create a Teach Mode operation that finishes a portion of the outside diameter of the part.

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

☐ Open the **tmp_teach_mfg_1** part.



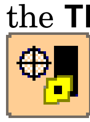
☐ Save the part as *****_teach_mfg_1**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

Step 2: Begin Creating the Operation.

☐ Choose the **Create Operation** icon.

☐ Set the **Type** option to **turning**.

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

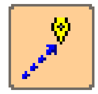
☐ Set the following Parent Groups:

- Program..........**PROGRAM**
- Use Geometry....**PART**
- Use Tool..........**OD_35_L**
- Use Method......**LATHE_FINISH**

☐ Choose **OK**.

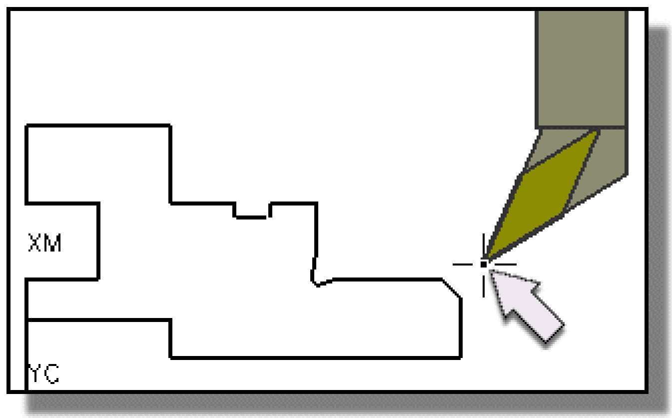
Step 3: Define the Initial Rapid Motion

You will begin the sequence of cutting and non-cutting moves with a rapid motion to the engage position.

☐ Choose **Linear Rapid**. 

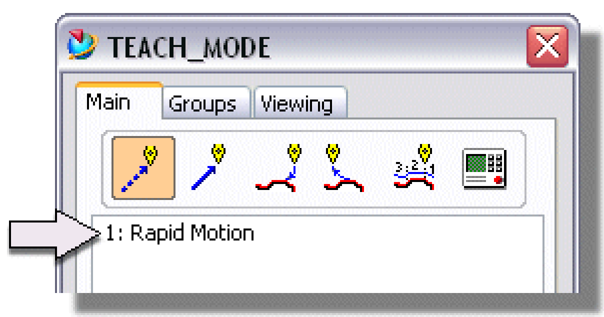
☐ In the Rapid Motion dialog, make sure the **Point** option is turned **on** and choose **Select**.

☐ Indicate the point at the approximate screen position illustrated below.



☐ **OK** to accept the rapid motion.

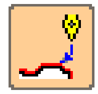
The Rapid Motion suboperation is now listed in the Teach Mode dialog.

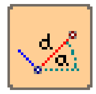


The end point position is displayed in the graphics window and is labeled EP. This is the position from which the tool will move to the engage.

Step 4: Define the Engage Suboperation.

You will now define an engage suboperation. Once defined, the engage suboperation will apply to every subsequent cutting move in the operation until a different engage suboperation is specified.

☐ Choose **Engage Setting**. 

☐ Choose **Angle/Distance**. 

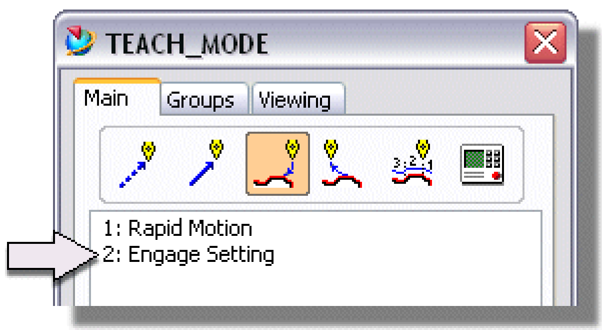
☐ Key in **270** in the Angle field.

☐ Key in **0.100** in the Distance field.

These parameters will engage the tool in a downward direction.

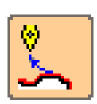
☐ **OK** to complete the Engage Setting suboperation.

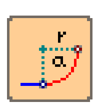
The Engage Setting suboperation is now listed in the Teach Mode dialog.



Step 5: Define the Retract Suboperation.

You will now define a retract suboperation. Once defined, the retract suboperation will apply to every subsequent cutting move in the operation until a different retract suboperation is specified.

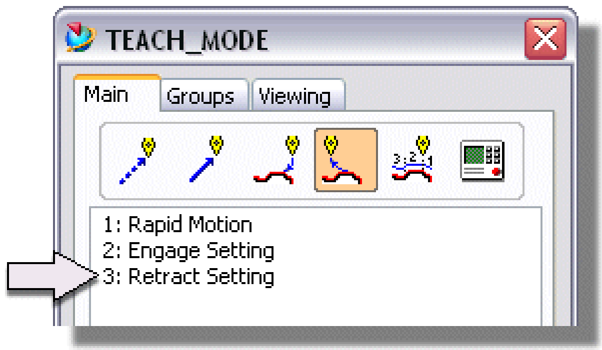
☐ Choose **Retract Setting**. 

☐ Choose **Auto Circular**. 

This option specifies that the tool will retract in a circular motion at the end of each cutting move.

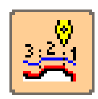
☐ **OK** to complete the Retract Setting suboperation.

The Retract Setting suboperation is now listed in the Teach Mode dialog.



Step 6: Define the Follow Curve Motion.

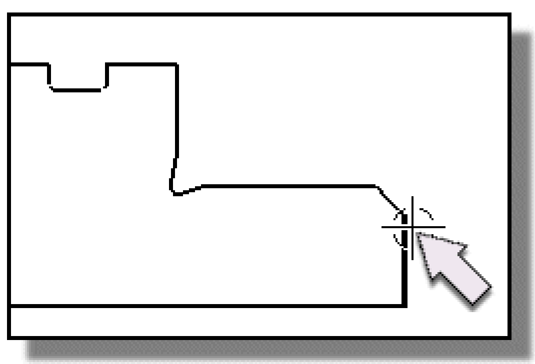
Next, you will define the Follow Curve Motion. You will define the drive geometry and the parameters that determine how the tool will cut along the drive geometry.

☐ Choose **Follow Curve Motion**. 

You will specify a boundary along which the tool will cut.

☐ Choose **New Drive Curve** and **Select**.

☐ Select the curve toward the top end as illustrated below.

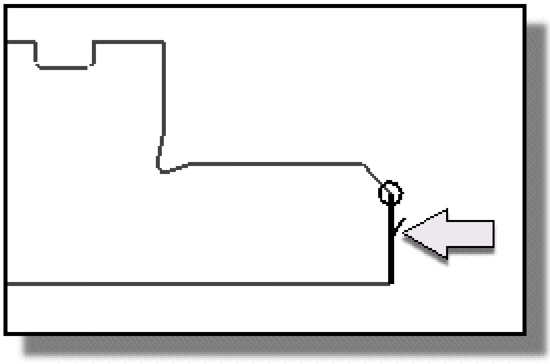


The end you select defines the direction of the boundary. The direction of the boundary is used to establish the forward cut direction. If you had chosen the bottom of the curve, you could reverse the cut direction by choosing the **Backward** arrow.

☐ Set the **Material Side** option to **Right**.

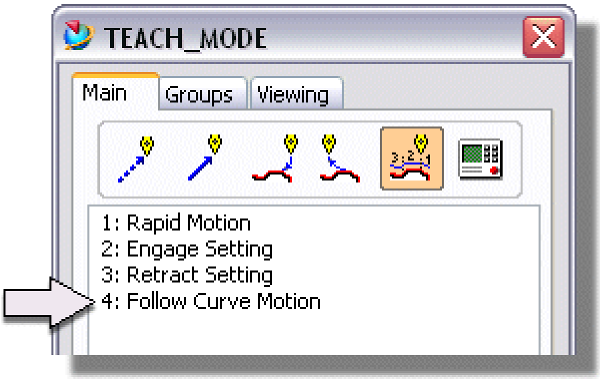
☐ **OK** to accept the drive geometry.

☐ Choose **Display** to display the boundary.



☐ **OK** to complete the Follow Curve Motion suboperation.

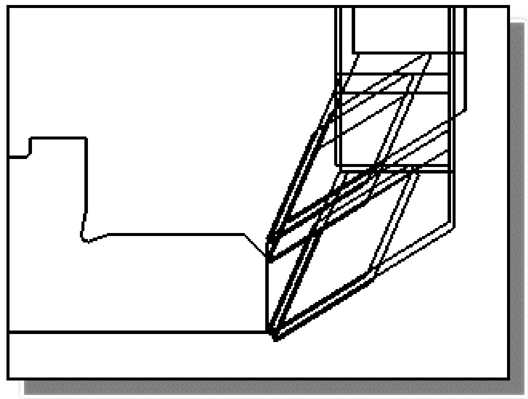
The Follow Curve Motion suboperation is now listed in the Teach Mode dialog.



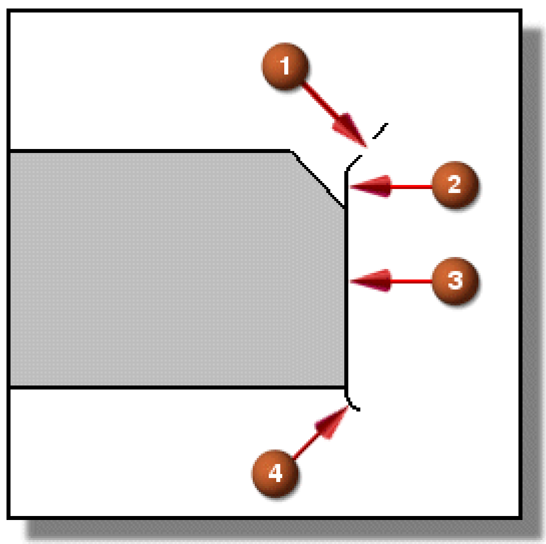
Step 7: Generate the tool path.

You will generate the tool path defined so far by the Rapid, Engage, Retract, and Follow Curve Motion suboperations.

☐ Choose the **Generate** icon.



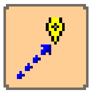
☐ Turn the **Tool Display** option **off** and generate the tool path again so you can see the motions clearly.



1. Rapid
2. Engage
3. Follow Curve Motion
4. Retract

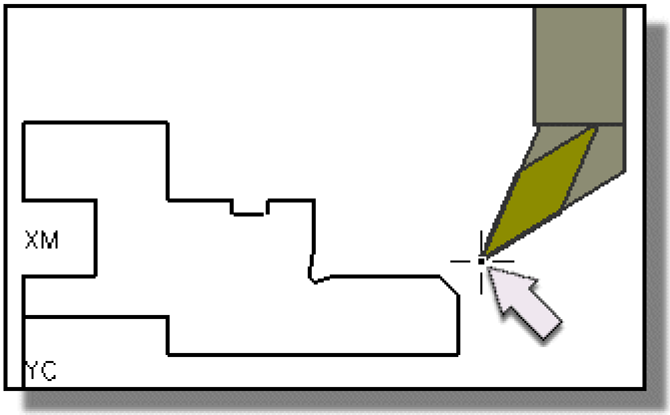
Step 8: Defining a Rapid Motion

Next, you will define the end point of another linear rapid motion. This will position the tool for the next engage, cut, and retract sequence.

☐ Choose **Linear Rapid**. 

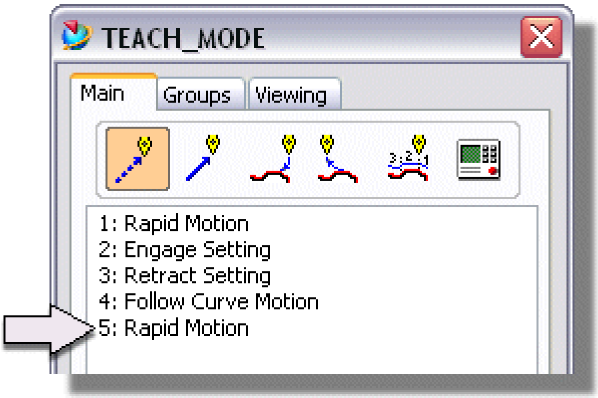
☐ In the Rapid Motion dialog, make sure the **Point** option is turned **on** and choose **Select**.

☐ Indicate the point at the approximate screen position illustrated below.



☐ **OK** to accept the rapid motion.

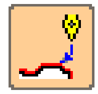
The Rapid Motion suboperation is now listed in the Teach Mode dialog.

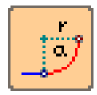


The end point position is displayed in the graphics window and is labeled EP. This is the position from which the tool will move to the engage.

Step 9: Define the Engage suboperation.

The next sequence of cutting moves will require a different engage setting.

☐ Choose **Engage Setting**. 

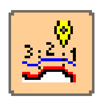
☐ Choose **Auto Circular**. 

☐ **OK** to complete the Engage Setting suboperation.

The Engage Setting suboperation is now listed in the Teach Mode dialog. This engage setting will apply to every subsequent cutting move in the operation until a different engage setting is specified.

Step 10: Define the Follow Curve Motion.

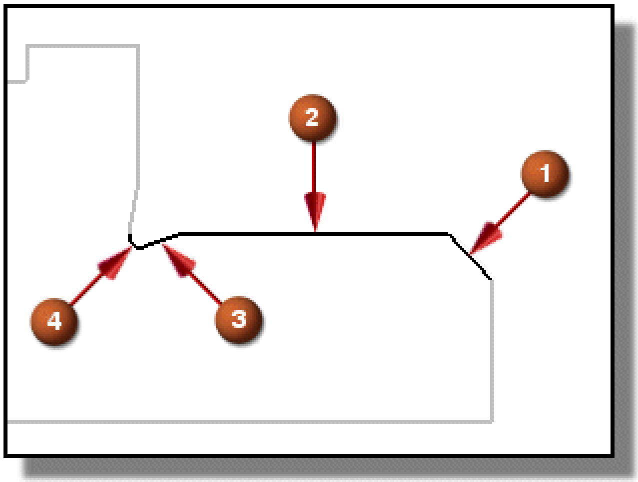
Next, you will define the next Follow Curve Motion. You will define the drive geometry and the parameters that determine how the tool will cut along the drive geometry.

☐ Choose **Follow Curve Motion**. 

You will specify a boundary along which the tool will cut.

☐ Choose **New Drive Curve** and **Select**.

☐ Select the four curves in the order illustrated below.

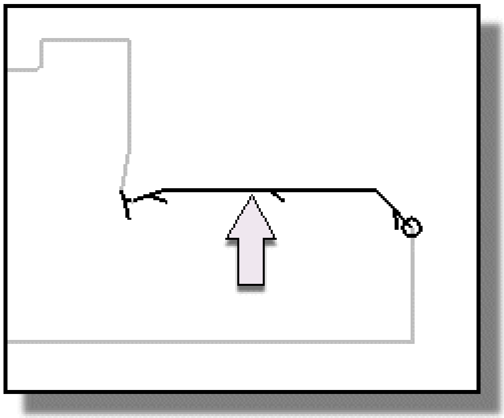


The order in which you select the curves defines the direction of the boundary. The direction of the boundary is used to establish the forward cut direction.

☐ Leave the **Material Side** option set to **Left**.

☐ **OK** to accept the drive geometry.

☐ Choose **Display** to display the boundary.



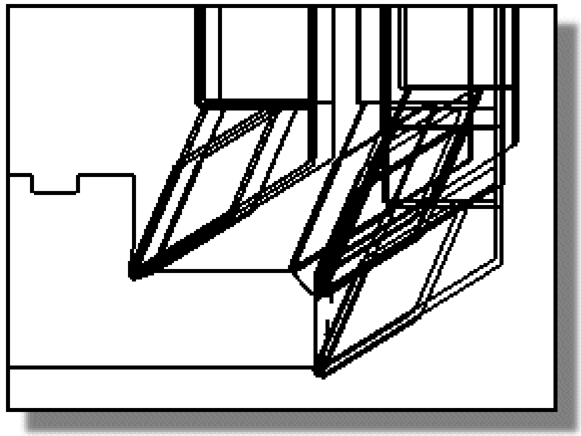
☐ **OK** to complete the Follow Curve Motion suboperation.

The Follow Curve Motion suboperation is now listed in the Teach Mode dialog.

Step 11: Generate the tool path.

☐ Set the **Tool Display** option to **2–D**.

☐ Choose the **Generate** icon.

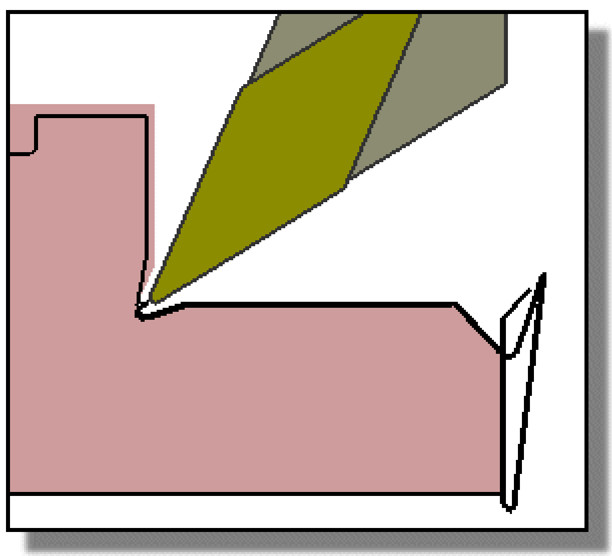


☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

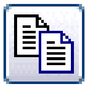
☐ Choose **Play**.



☐ **OK** to complete the tool path visualization.

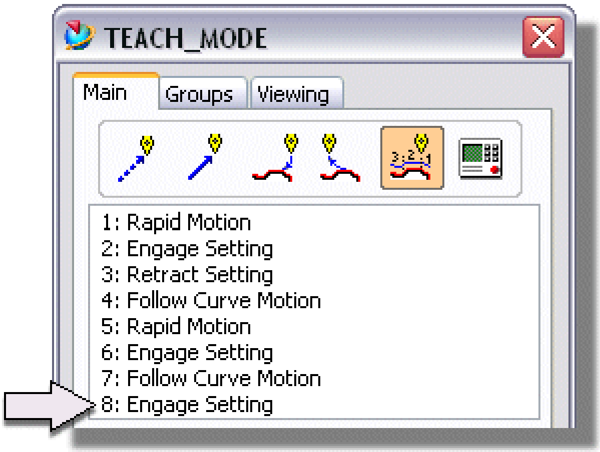
Step 12: Copying an Engage Setting.

The next Follow Curve Motion will require an Engage Setting similar to the one defined for the first Follow Curve Motion. You will copy and paste the first Engage Setting and then make a minor edit.

☐ Choose **2: Engage Setting** in the list box and choose the **Copy** icon. 

☐ Choose **7: Follow Curve Motion** in the list box and choose the **Paste** icon. 

The Engage Setting suboperation (line 8) now appears after the selected Follow Curve Motion suboperation. This Engage Setting will apply to the next Follow Curve Motion in the sequence.



You will edit this Engage Setting by adding an extend distance to prevent the engage from hitting the stock.

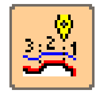
☐ Double-click **8: Engage Setting** in the list box to edit the suboperation.

☐ Key in **0.050** in the Extend Distance field.

☐ **OK** to accept the Engage Setting.

This extension will prevent the tool from hitting the stock on the next diameter.

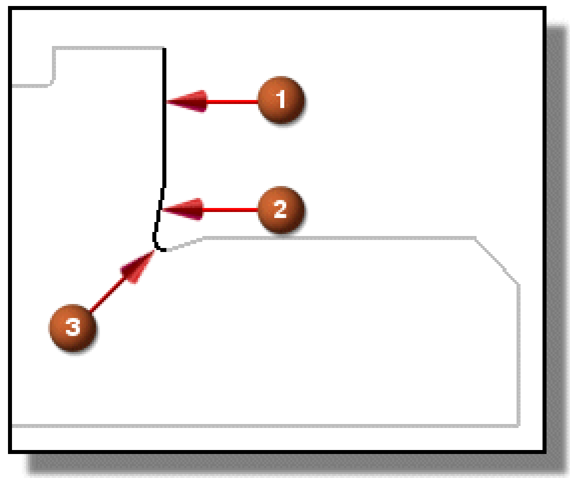
Step 13: Define the Follow Curve Motion.

- ☐ Choose **Follow Curve Motion**. 

  You will specify a boundary along which the tool will cut.

- ☐ Choose **New Drive Curve** and **Select**.

- ☐ Select the three curves in the order illustrated below.

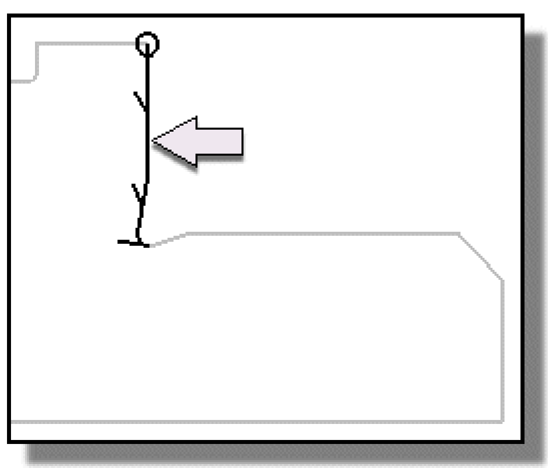


The order in which you select the curves defines the direction of the boundary. The direction of the boundary is used to establish the forward cut direction.

- ☐ Set the **Material Side** option to **Right**.

- ☐ **OK** to accept the drive geometry.

- ☐ Choose **Display** to display the boundary.

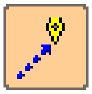


- ☐ **OK** to complete the Follow Curve Motion suboperation.

  The Follow Curve Motion suboperation is now listed in the Teach Mode dialog.

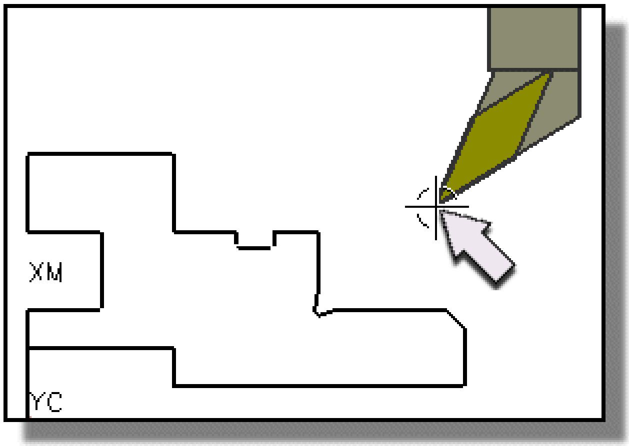
Step 14: Define a Rapid Motion

You will define the end point of a linear rapid motion.

- ☐ Choose **Linear Rapid**. 

- ☐ In the Rapid Motion dialog, make sure the **Point** option is turned **on** and choose **Select**.

- ☐ Indicate the end point at the approximate screen position illustrated below.



- ☐ **OK** to accept the rapid motion.

  The Rapid Motion suboperation is now listed in the Teach Mode dialog.

Step 15: Generate the tool path.

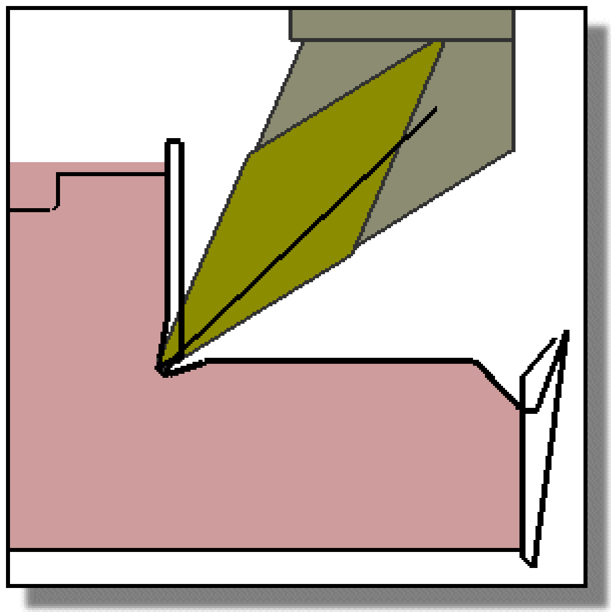
☐ Choose the **Generate** icon.

☐ Choose **Verify**.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the **Animation Speed** down to **1**.

☐ Choose **Play**.



☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

☐ Save and close the part.

Summary

Teach Mode allows you to create a sequence of suboperations that define individual cutting and non-cutting motions. It is useful for finishing areas that require a high degree of cutter control. In this lesson, you created a Teach Mode operation that cut faces and diameters. You defined drive geometry, avoidance moves, and applied appropriate engages and retracts.

11

Lesson

12 *Threading Operations*

12

Purpose

This lesson teaches you how to create OD and ID threading operations.

Objective

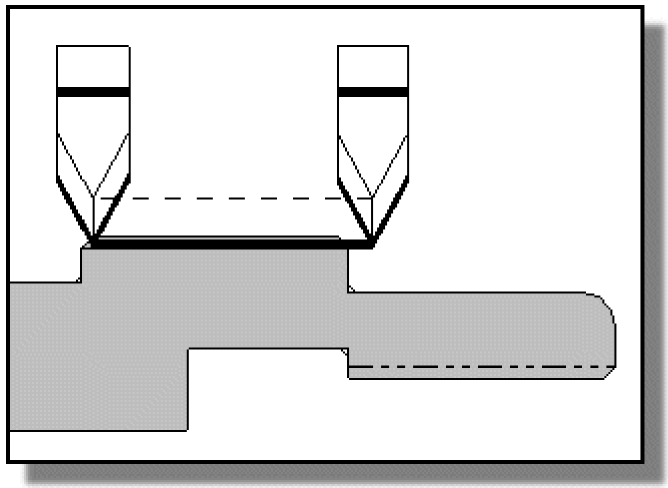
Upon completion of this lesson, you will be able to:

- 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.

12

Threading Operations

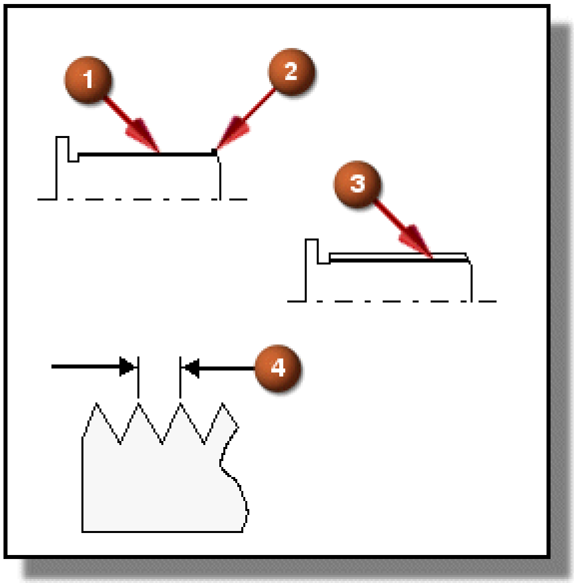
Threading operations can be developed to cut both straight and tapered threads. They can be single start or multiple start threads which are either internal, external, or face threads (such as the thread advance used in a three-jaw chuck).



12

Developing a Threading Operation

Threading operations require you to specify a crest line, a root line, and a pitch. The crest line (1) represents the outer tip of the thread. The end you select (2) determines the start of the thread. The root line (3) determines the depth of the thread. The pitch (4) is the distance between corresponding points measured from one thread to the next.



Crest Line

The Crest Line represents the outer tip of the thread.

The Start Point is the point at which the threading motion begins.

The system 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 total depth of the thread. The total depth can be reached using constant, variable, or percentage increments.

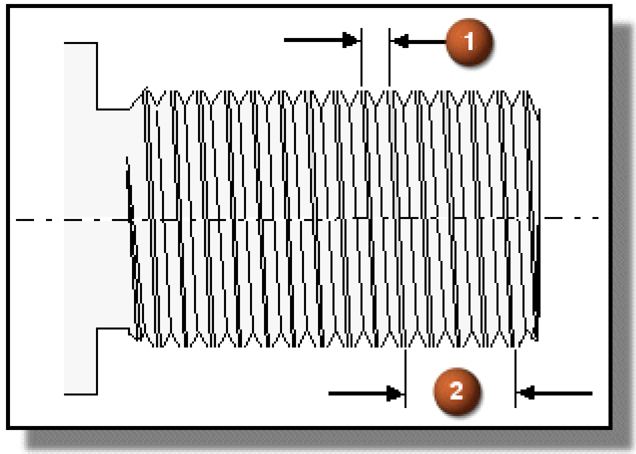
Pitch

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.

12

Pitch/Lead/Tpi

You must specify the Pitch and Lead (1) or Threads Per Inch (2).



The Number of Starts determines the Pitch and Lead as shown in the following example of a 1/4-20 thread:

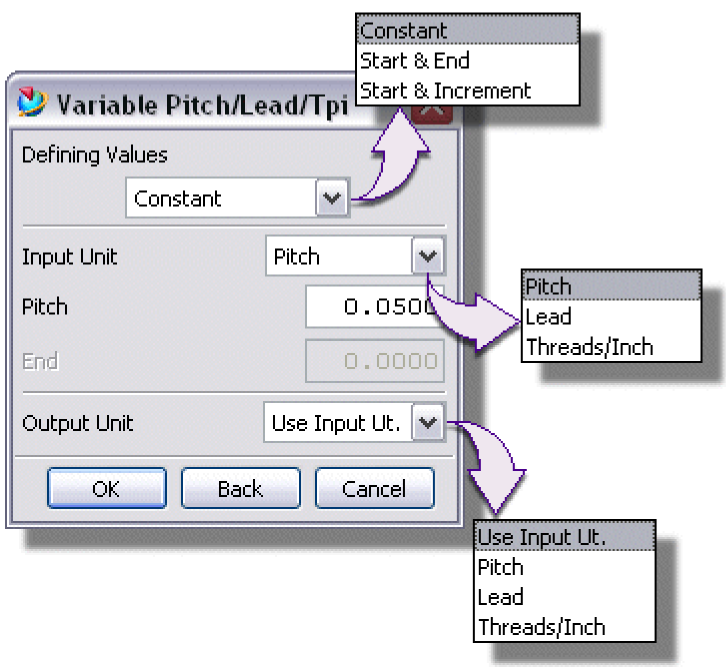
Threads Per Inch = 20

Pitch = (1 (inch) / 20) = .05

Lead = .05 x 1 (No. of Starts)

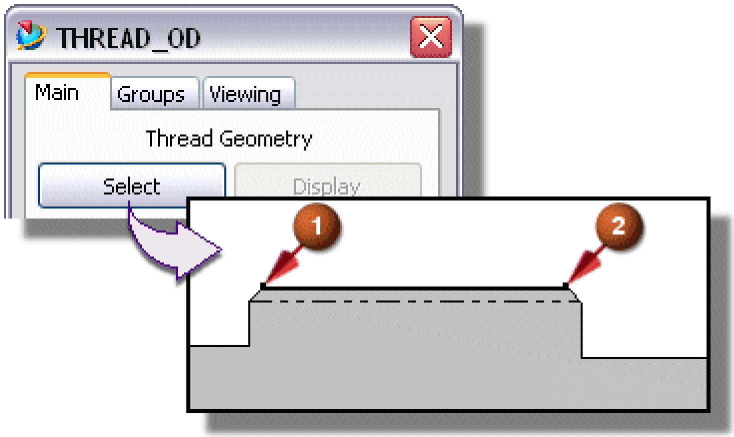
In this case, the Pitch and Lead are .050 because there is only one Start. If there were two Starts, the Lead would be .100, or twice the Pitch.

When you choose the Specify button, the following dialog is displayed. You can define the following options from this dialog.



Thread Geometry

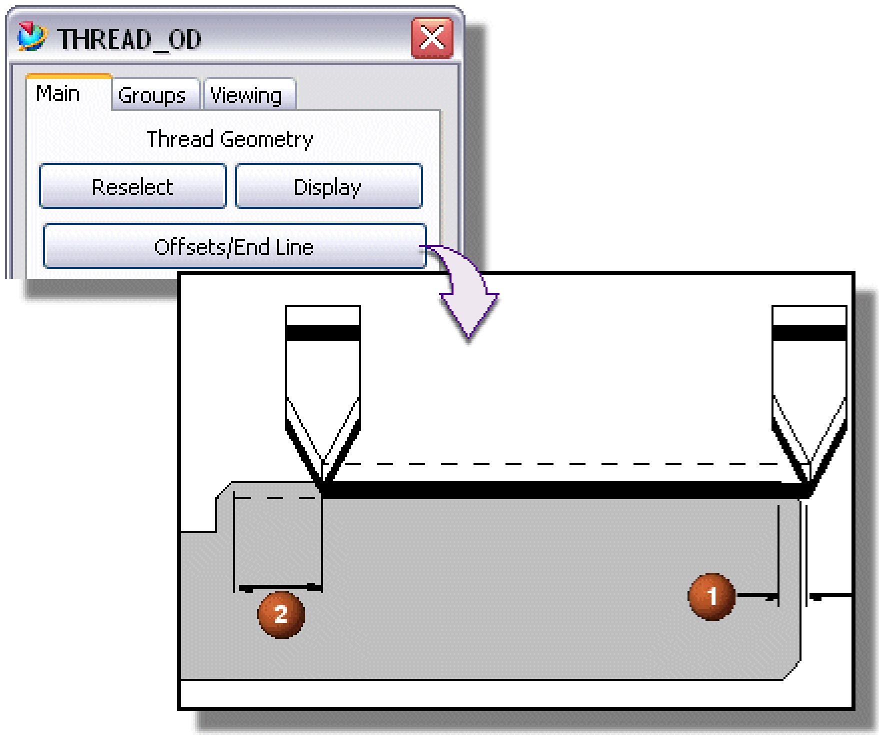
Thread Geometry defines the Start (1) and End (2) points of the thread.



12

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.



1. 0.1500 Start Offset

2. –.05000 End Offset

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.

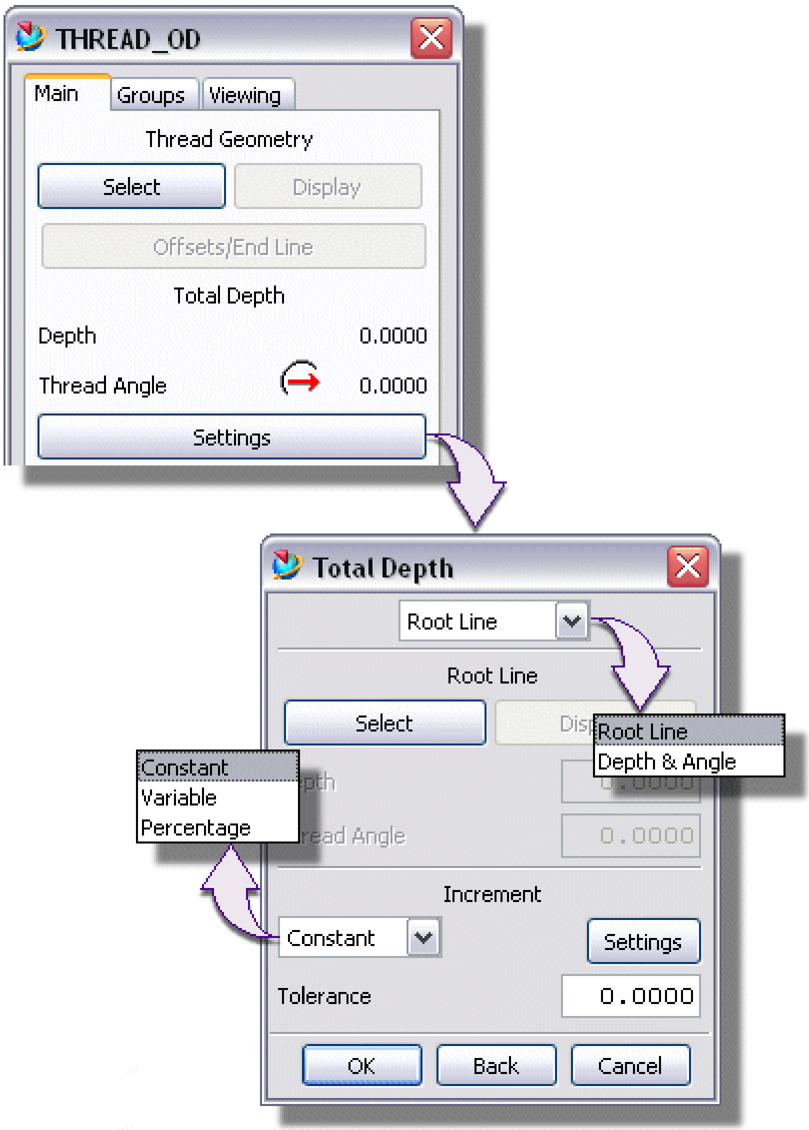
Init From Tool causes the system to calculate the End Offset so that the tool clears the thread.

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.

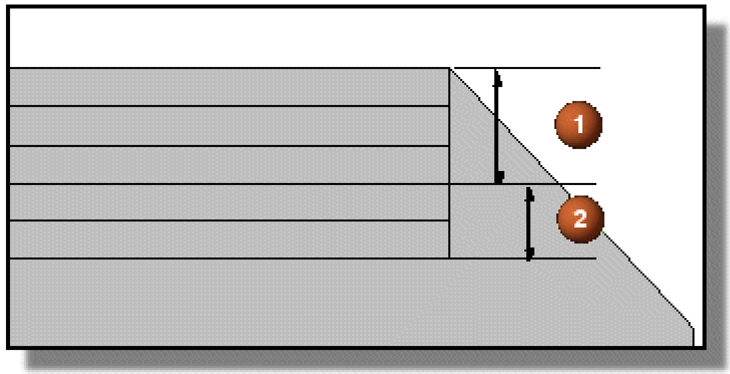
The Total Depth gives you options to establish a Root Line by using the cursor to select a line, or by entering Depth and Angle values.



Each Increment method provides a Tolerance option which allows you to specify the increment for the last roughing pass.

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.

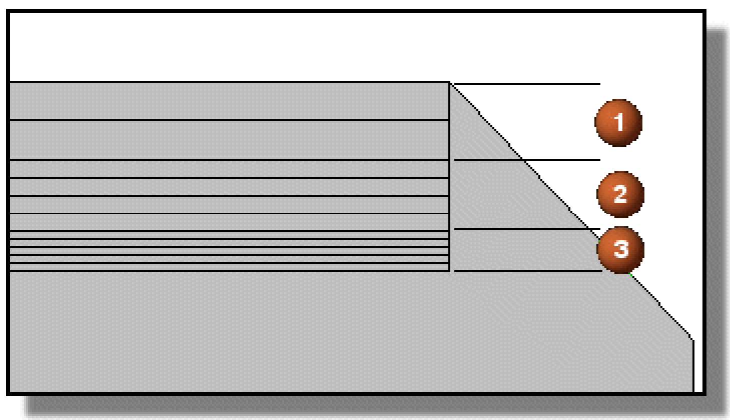
In the following illustration, a constant increment of 0.0150 has been specified. The system takes the average of the last two passes to keep them as close as possible to the specified constant increment value.



1. Three passes at 0.0150 depth each

2. Two passes at 0.0140 depth each

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.



1. Two passes at 0.0150 depth each

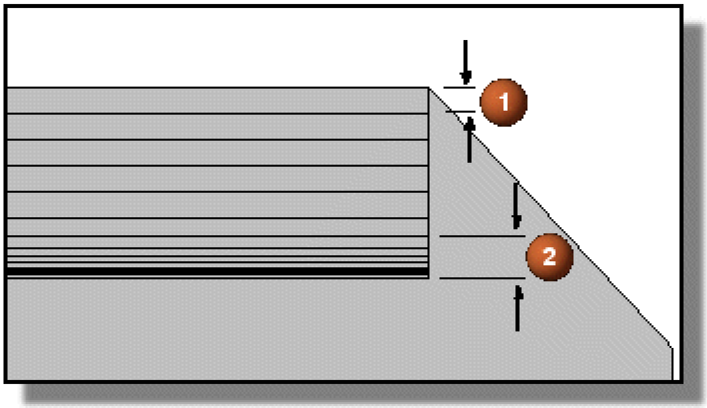
2. Four passes at 0.0070 depth each

3. Remaining depth at 0.0030 per pass

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 the depth of each cutting pass to decrease as the tool moves deeper into the thread. The Maximum value prevents the initial passes from cutting too deeply. As the tool approaches the Total Depth, the increment depth can become very small. Therefore, to control the number of passes, you must enter a Minimum increment value so that the tool can reach the total depth.

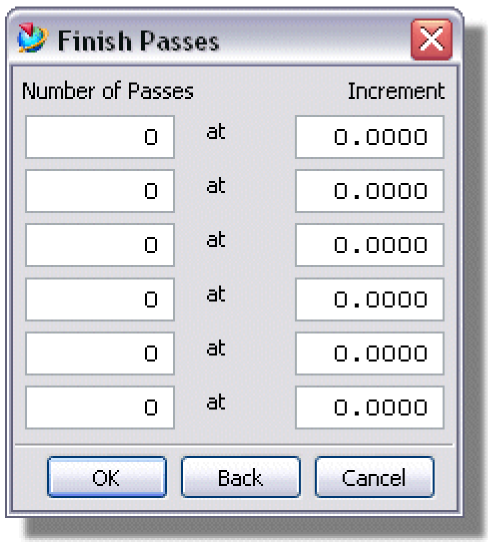


1. 0.0100 Maximum
2. Each pass is 30% of remaining depth

Finish Passes

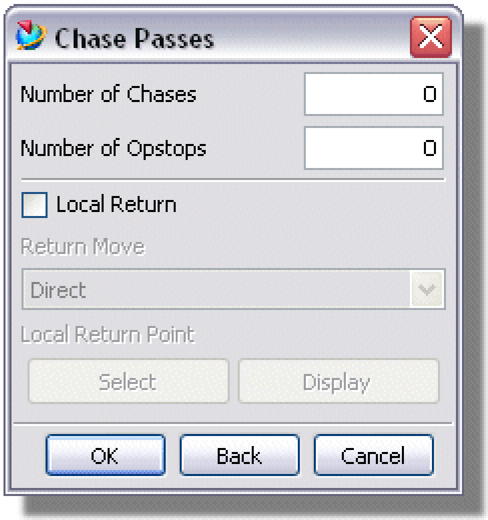
You can specify a number of finish passes and increments when cutting threads.

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.

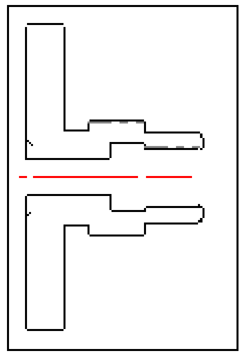
Chase passes are always bracketed by OPSTOP, OPSKIP/ON and OPSKIP/OFF commands.

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 **tmp_thread_mfg_1** part.



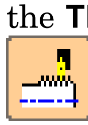
☐ Save the part as *****_thread_mfg_1**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

Step 2: Define the parent groups for this operation.

☐ Choose the **Create Operation** icon.

☐ Set the **Type** option to **turning**.

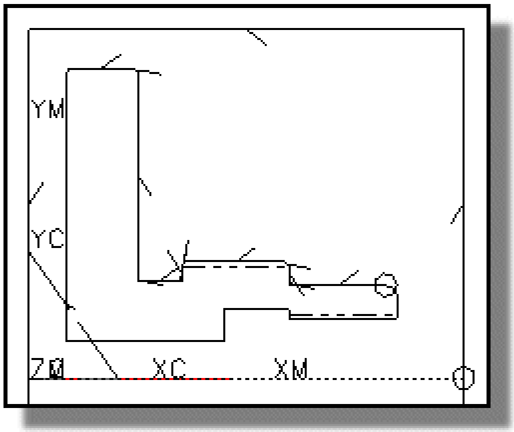
☐ Choose the **THREAD_OD** icon from the Create Operation dialog. 

☐ Set the following Parent Groups:

- Program..........**PROGRAM**
- Use Geometry....**PART**
- Use Tool..........**THREAD_EXT**
- Use Method......**THREAD_METHOD**

☐ Choose **OK**.

☐ Choose the **Groups** tab and display the geometry.



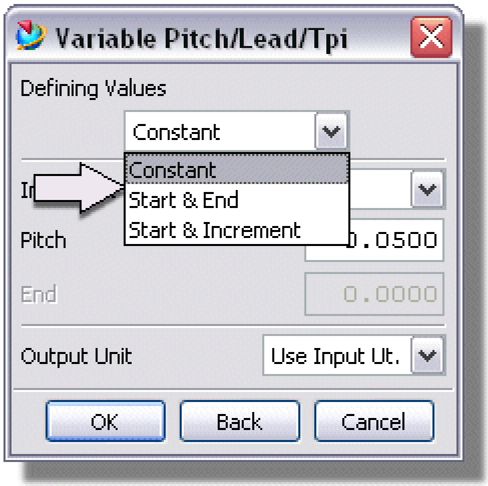
☐ Refresh the graphics display.

Step 3: Define the pitch.

☐ Choose the **Main** tab.

☐ Choose **Specify** next to Pitch/Lead/Tpi .

☐ Display the **Defining Values** options.



☐ Choose **Constant**.

☐ Key in **1/18** in the Pitch field.

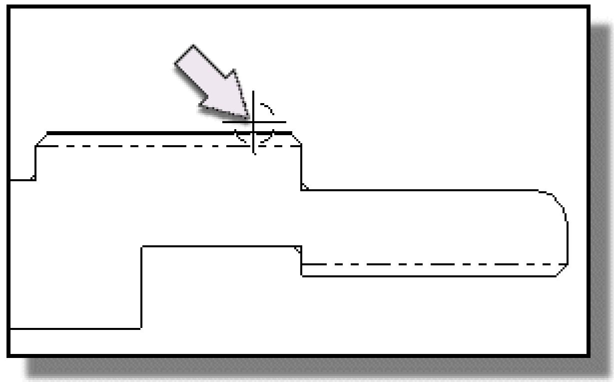
☐ **OK** to accept the pitch.

12

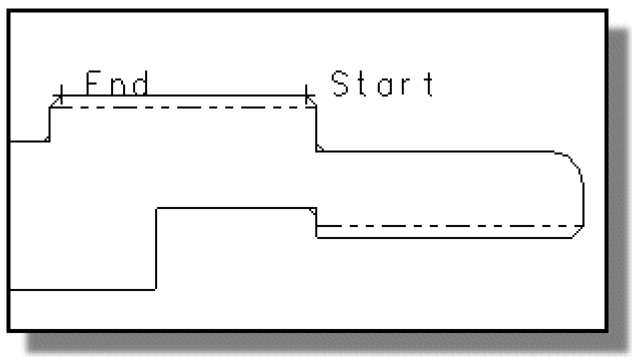
Step 4: Define the crest line.

☐ Choose the **Select** option under Thread Geometry.

☐ Select the crest line near the right end as illustrated below.



The start point should appear at the right end of the selected line.



Step 5: Define the total depth.

☐ Choose **Settings**.

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**.

☐ Key in **0.0334** in the Depth field.

☐ Key in **180** in the Thread Angle field.

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**.

You must specify the Percentage and the Maximum and Minimum values.

☐ Key in the following values in the Percentage Increment dialog:

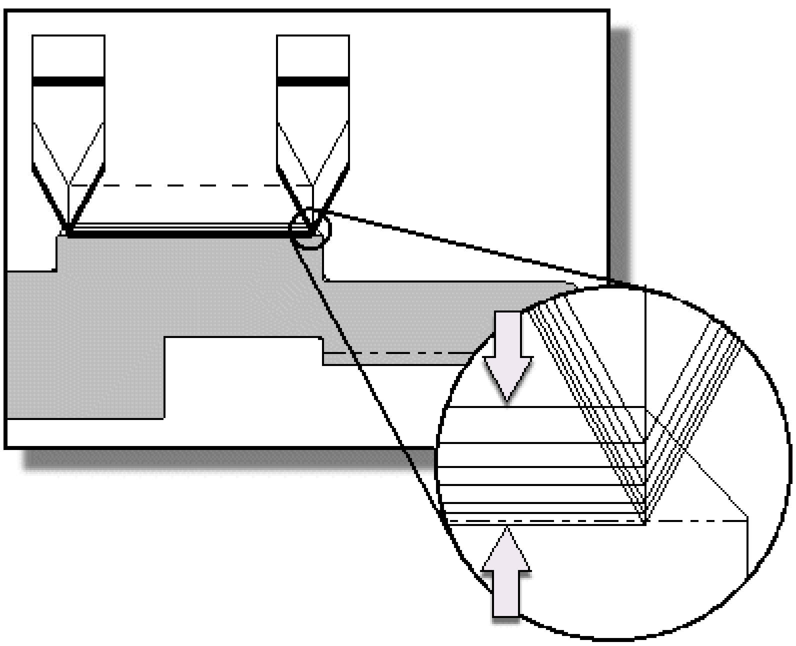
- Percentage = **30**
- Maximum = **.04**
- Minimum = **.005**

☐ Choose **OK**.

☐ **OK** to accept the total depth.

☐ **Generate** the tool path.

Zoom in on the start point to see the incremental cut depth.



Now you will define the Total Depth by selecting a root line.

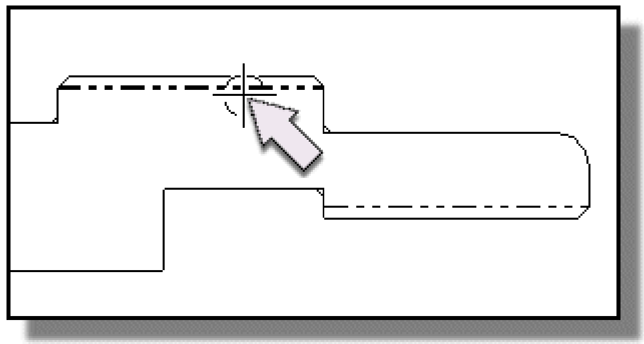
☐ Refresh the graphics display.

☐ Choose **Settings**.

- ☐ Choose **Root Line**.

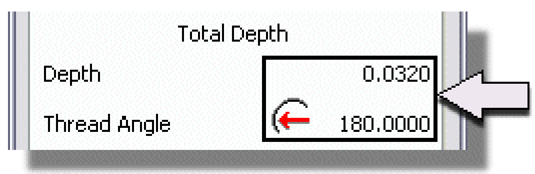
- ☐ Choose **Select**.

- ☐ Choose the Thread Root Line as illustrated below.



- ☐ **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.

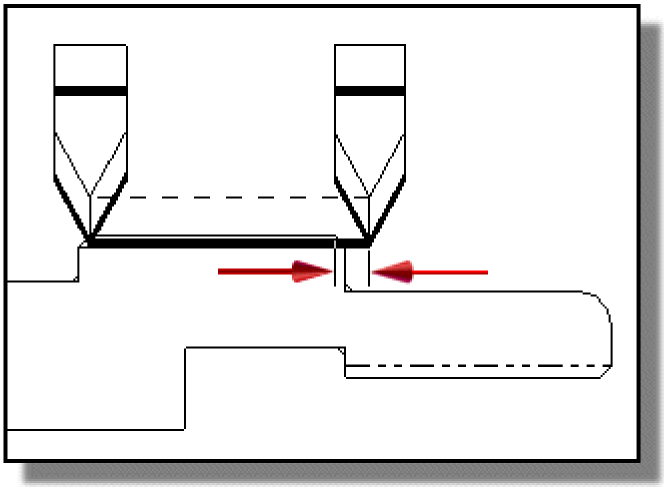
- ☐ Choose **Offsets/End Line**.

- ☐ Key in **0.100** in the Start field.

- ☐ Choose **OK**.

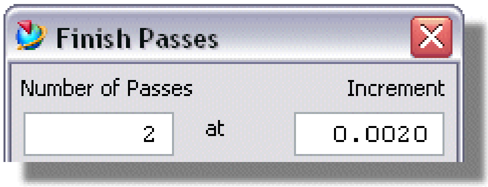
- ☐ **Generate** the tool path.

  The start point has been offset 0.100 to the right.



Step 7: Specify a finish pass.

- ☐ Choose **Finish Passes**.
- ☐ Enter **2** passes of **.002**.



- ☐ Choose **OK** to return to the THREAD_OD dialog.

Step 8: Specify a chase pass.

- ☐ Choose **Chase Passes**.
- ☐ Key in **2** in the Number of Chases field.
- ☐ Choose **OK** to return to the THREAD_OD dialog.
- ☐ **Generate** the tool path.

  The tool path is generated with the Finish and Chase Passes.

- ☐ **OK** to complete the operation
- ☐ Save the part.

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: Continue to use the *****_thread_mfg_1** part.

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....**PART**
- ☐ 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.

Summary

Threading operations can cut both straight and tapered threads. Threading operations require you to specify a crest line, a root line, and a pitch. The crest line represents the outer tip of the thread. The end you select determines the start of the thread. The root line determines the depth of the thread. The pitch is the distance between corresponding points measured from one thread to the next.

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

13 Using Multiple Spindles

Purpose

This lesson will teach you how to use cross sectional curves to define part geometry for parts mounted at each spindle of a multiple spindle machine. You will also learn how to manage the in-process workpiece as it is passed from one spindle to the next.

Objective

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

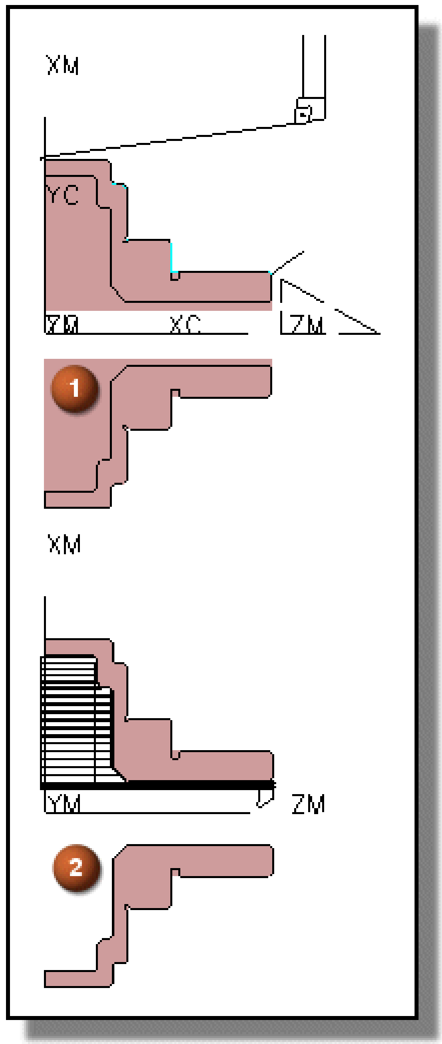
- Define part geometry for parts mounted at each spindle of a multiple spindle machine.

- Define blank geometry at Spindle 2 by referencing the IPW generated at Spindle 1.

- Generate the tool paths for the program and observe material removal at both spindles.

MCS Objects Available for Lathe Cross Sections

An in-process workpiece (IPW) can be created and referenced as it is transferred from one spindle to another or turned around in the spindle. When the workpiece is remounted, the IPW 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 another.



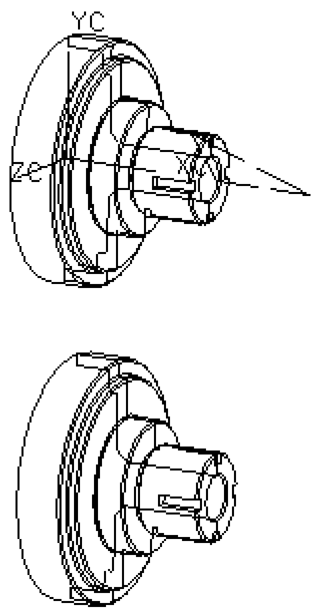
1. Spindle 1
2. Spindle 2

Activity: Using a Multiple Spindle Setup

You are going to define part geometry at two spindles. You will then define the blank geometry at Spindle 2 by referencing the IPW generated at Spindle 1. You will generate the tool paths for the program and see how the system progressively removes material while passing the workpiece from one spindle to the next.

Step 1: Examine the setup for a two spindle machine.

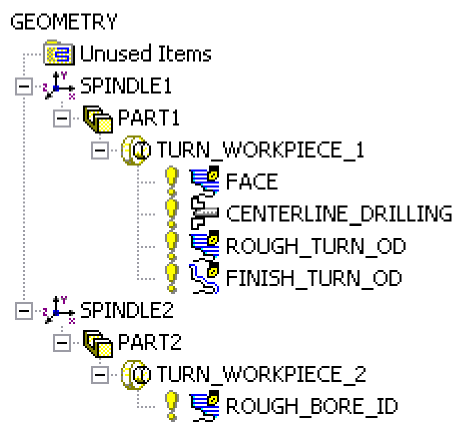
☐ Open the **tmp_turn_assy_1** part.



☐ Save the part as *****_turn_assy_1**, where *** represents your initials.

☐ Choose **Start→Manufacturing**.

☐ Display the Geometry view of the Operation Navigator and expand the objects.

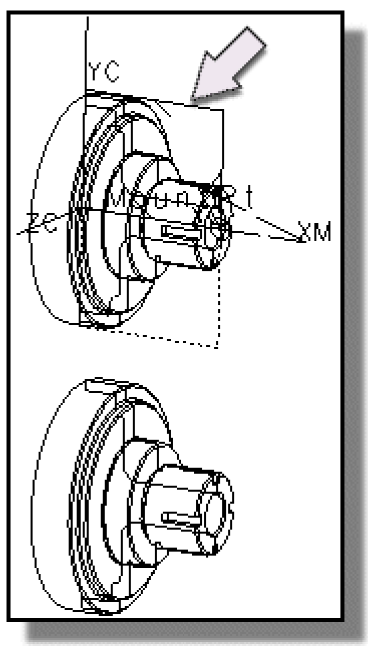


Step 2: Display the blank boundary at Spindle 1.

☐ Double-click **TURN_WORKPIECE_1** in the Operation Navigator.

☐ Choose the **Blank** icon in the TURN_BND dialog.

☐ Choose **Display**.



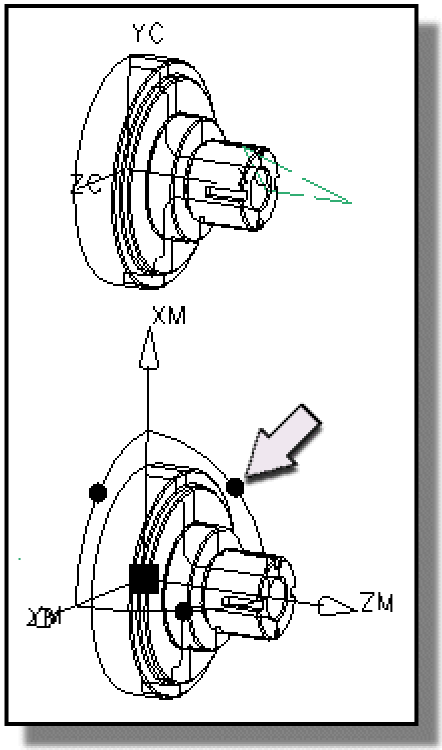
☐ **Cancel** the TURN_BND dialog.

Step 3: Display the spindles.

Each spindle is defined in the assembly model by an MCS.

☐ Notice the location of the MCS at the first spindle.

☐ Double-click **SPINDLE2** in the Operation Navigator to display the second MCS.



☐ **Cancel** the MCS_SPINDLE dialog.

13

Step 4: View the cross section curves.

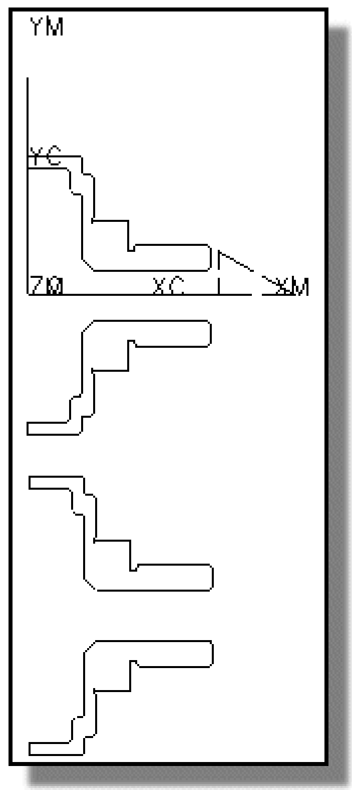
You will now change to a top view and display only the cross section curves.

☐ Choose **Format→Layer Settings** from the menu bar.

☐ Make layers **1** and **2 Invisible**.

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

☐ In the graphic window, choose **MB3→Replace View→Top**.



Step 5: Define part geometry at Spindle 1.

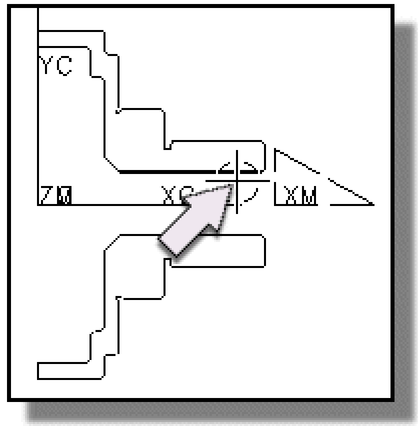
☐ In the Geometry View of the Operation Navigator, double-click **TURN_WORKPIECE_1** to edit the object.

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

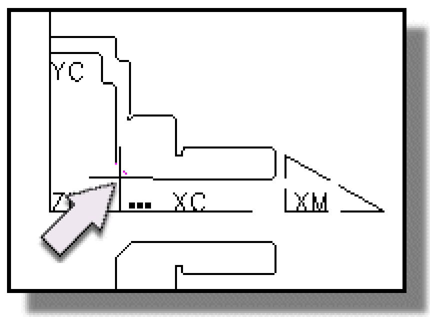
☐ With the **Curve Boundary** icon active, choose **Closed** under Type. Be sure Material Side is **Inside**.

☐ Choose **Chaining**.

☐ At Spindle 1, select the horizontal line at the approximate position illustrated below (right end).

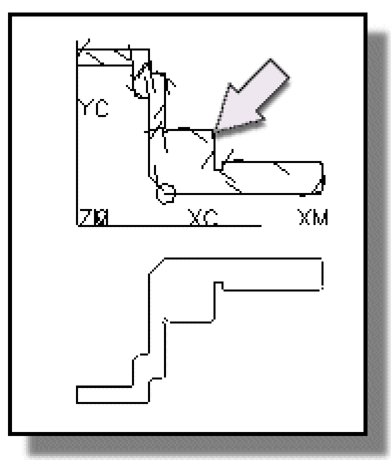


☐ Select the angled line.



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

☐ Choose **Display**.

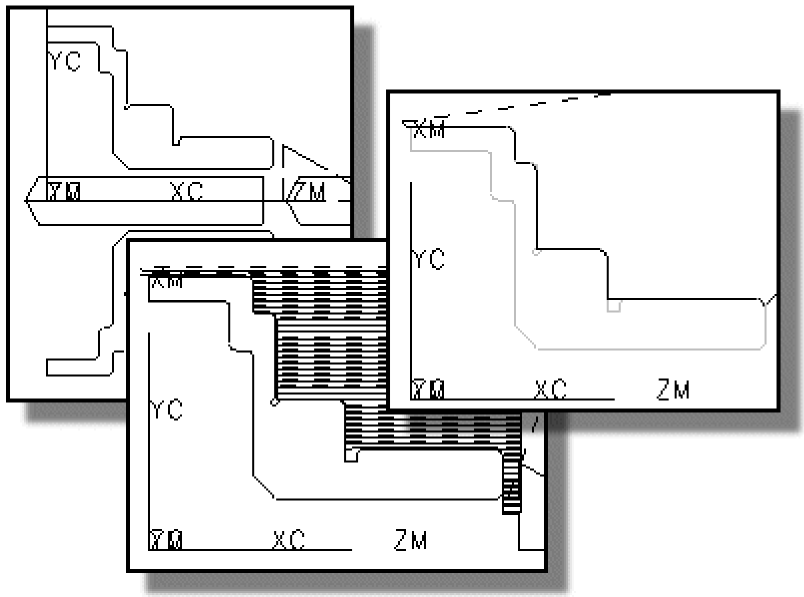


☐ **OK** the TURN_BND dialog to complete the boundary.

Step 6: Generate the tool paths at Spindle 1.

☐ Choose **SPINDLE1** in the Operation Navigator and **MB3→Generate**.

☐ Choose **OK** three times in the Tool Path Generation dialog to generate all three tool paths at Spindle 1.



Step 7: Define part geometry at Spindle 2.

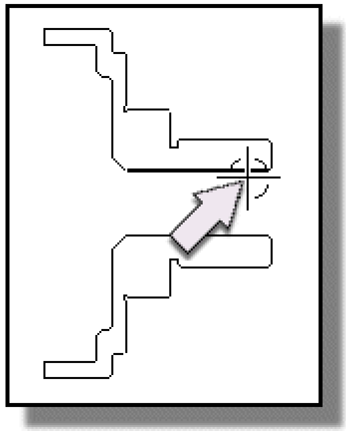
☐ In the Geometry View of the Operation Navigator, double-click the **TURN_WORKPIECE_2** object.

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

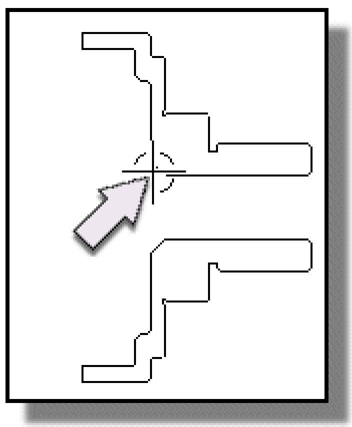
☐ With the **Curve Boundary** icon chosen, choose **Closed** under Type. Be sure Material Side is **Inside**.

☐ Choose **Chaining**.

☐ At Spindle 2, select the horizontal line at the approximate position illustrated below (right end).



☐ Select the angled line.



☐ Choose **OK** twice to complete the part boundary.

Step 8: Define blank geometry for the rough bore operation.

Blank geometry for the Rough Bore ID operation at Spindle 2 is defined by referencing the IPW generated by the previous operations at Spindle 1.

☐ Double-click **TURN_WORKPIECE_2** in the Operation Navigator.

☐ Choose the **Blank** icon.

☐ Choose **Select**.

☐ Choose the **From Workspace** icon.

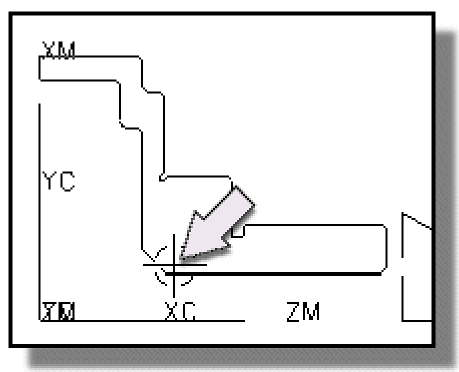
To define the blank geometry for this operation (ROUGH_BORE_ID), you will first define a Reference Position in the source, or "from" workspace and then define a Target Position in the current, or "to" workspace. Any two points will work as long as they correspond in each workspace.

Step 9: Define the reference position.

You will first define a reference position on the part at Spindle 1.

☐ Choose **Select** under Reference Position.

☐ Choose the end point of the horizontal line on the part at Spindle 1 as illustrated below.



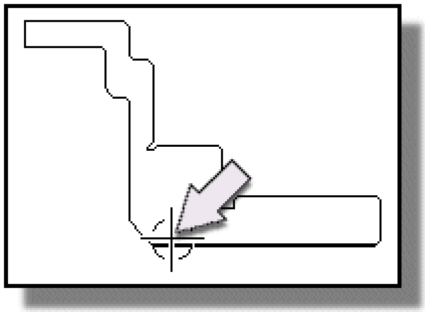
The point is labeled "Ref Pt".

Step 10: Define the target position.

You will now define a corresponding target position on the part at Spindle 2.

☐ Choose **Select** under Target Position.

☐ Choose the end point of the horizontal line on the part at Spindle 2 as illustrated below.

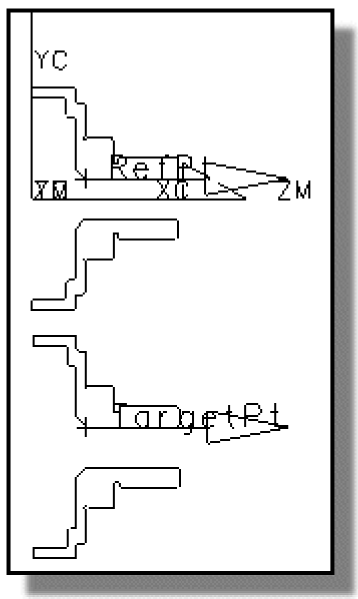


The point is labeled "Target Pt".

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. The orientation of the blank geometry at Spindle 2 is correct because the workpiece is transferred from one spindle to the next and is not flipped. If the workpiece were flipped at Spindle 2, you could flip the blank by choosing Flip Direction.

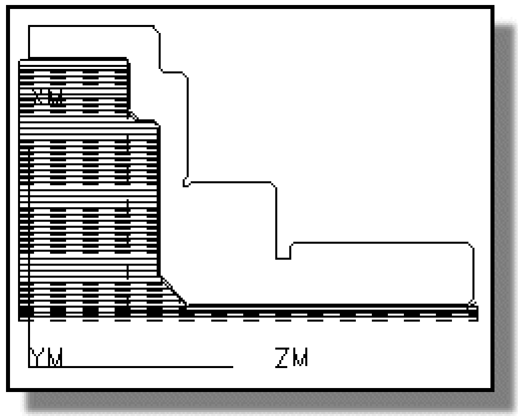
☐ **OK** to accept the blank definition.

☐ **OK** to accept the TURN_BND dialog.

Step 12: Generate the tool path for the rough bore operation.

Now that the blank material for the Rough Bore ID operation has been defined by referencing the IPW from Spindle 1, you can generate the tool path.

☐ Highlight the ROUGH_BORE_ID operation and **MB3→Generate**.



☐ **OK** to accept the tool path.

Step 13: Verify material removal for the program.

You will see how the system progressively removes material while passing the workpiece from one workspace to another.

☐ Display the Program Order View of the Operation Navigator.

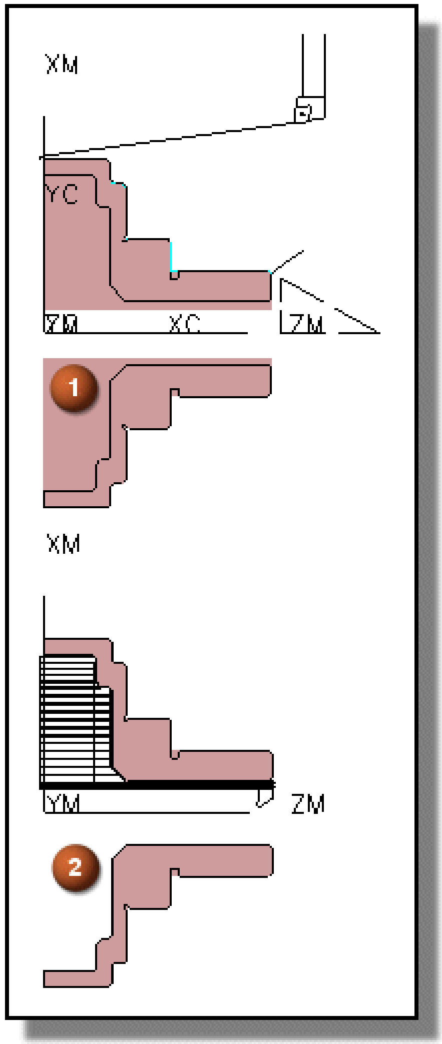
☐ Highlight **PROGRAM**.

☐ Choose the **Verify Toolpath** icon in the toolbar.

☐ Turn the **2D Material Removal** option **on**.

☐ Slow the Animation Speed down to **3**.

☐ Choose **Play**.



1. Spindle 1
2. Spindle 2

☐ Choose **OK** to complete the Toolpath Visualization.

Step 14: Save and close the part.

Summary

In this lesson, you learned how to use cross sectional curves to define part geometry for parts mounted at each spindle of a multiple spindle machine. You also learned how to manage the in-process workpiece as it is passed from one spindle to the next.

Lesson

14 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.

14

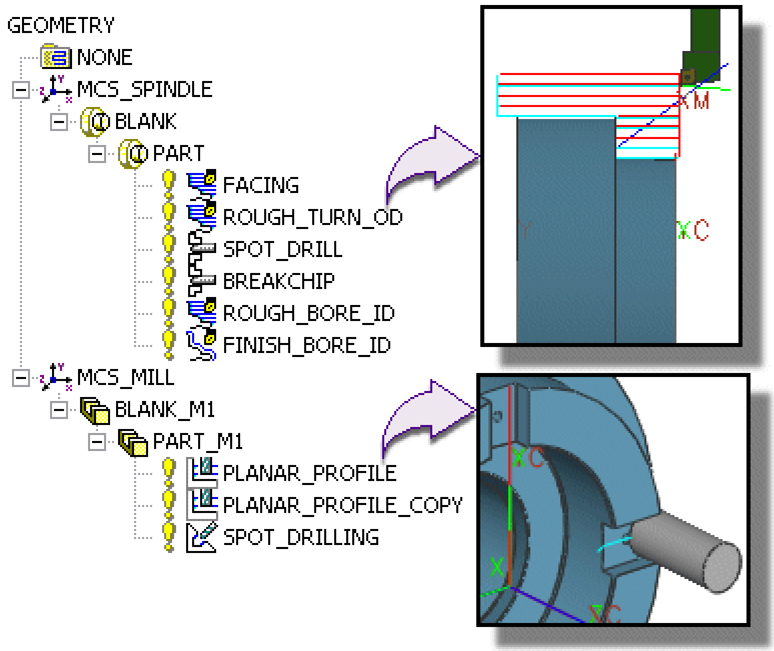
Objective

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.

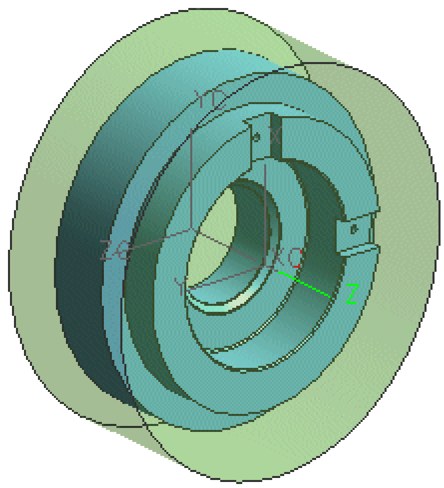
The process defined in this lesson describes how to set up the machine coordinate systems and how to utilize the turning IPW in the subsequent milling operations to create a simple mill-turn program. It also describes how to postprocess the mill-turn program.

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 **tmp_mill_turn_assy_mfg_1** part.



☐ Save the part as *****_mill_turn_assy_mfg_1**, where *** represents your initials.

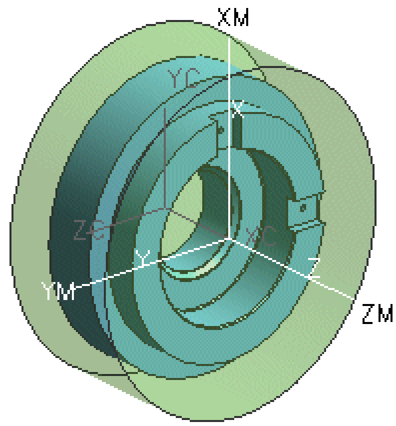
☐ Choose **Start→Manufacturing**.

☐ Choose **Start→Assemblies** if necessary to turn the **Assemblies** option **on**.

14

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 displayed above.

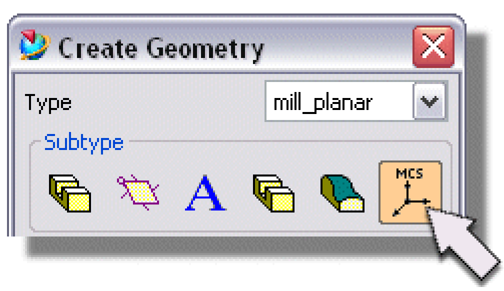
Step 3: Create a MCS_MILL object.

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.

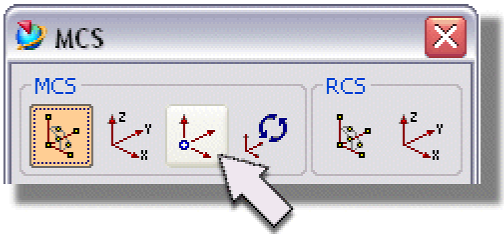


☐ Set the **Parent Group** option to **Geometry**.

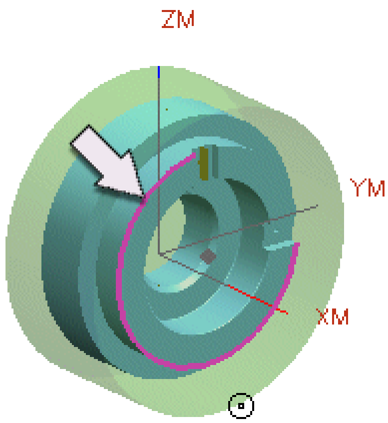
☐ Key in **MCS_MILL** in the Name field.

☐ Choose **OK** to begin creating the MCS_MILL object.

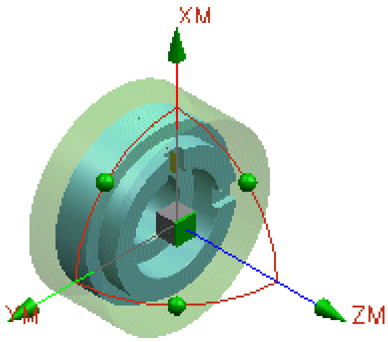
☐ Choose the **Origin** icon.



☐ Choose the arc center of the outer arc for the location of the MCS_MILL object as shown below.



☐ Drag the handles of the MCS_MILL coordinate system to orient it as shown below.



The MCS_SPINDLE and MCS_MILL coordinate systems are at the same location and have the same orientation.

☐ Choose **OK** to create the MCS_MILL coordinate system.

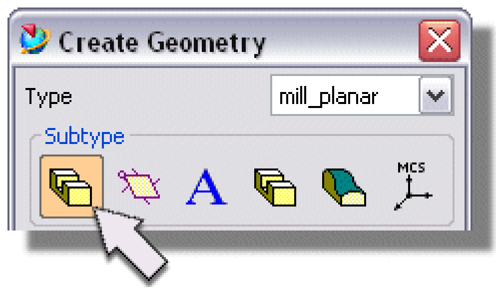
14

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.

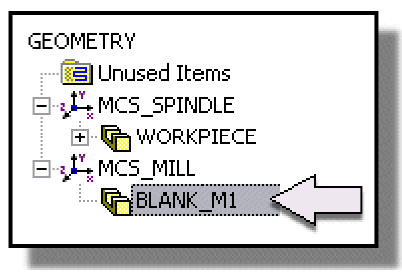


☐ Set the **Parent Group** option to **MCS_MILL**.

☐ 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.

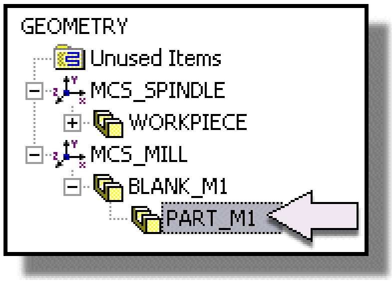
☐ Choose the **WORKPIECE** icon.

☐ 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: Exporting the turning IPW.

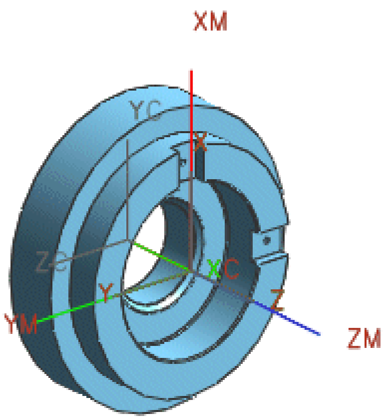
You will highlight the last turning operation and save the IPW for the turning subprogram as a separate part. Later, you will import this part back into the assembly and use it as blank geometry for the milling operations.

☐ In the Operation Navigator, highlight the last turning operation (FINISH_BORE_ID) and **MB3→Workpiece→Save as**.

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

☐ In the Assembly Navigator, choose the check mark next to **tmp_swug_mill_turn_blk** to remove the blank material from the display.

Only the part should be displayed in the assembly.



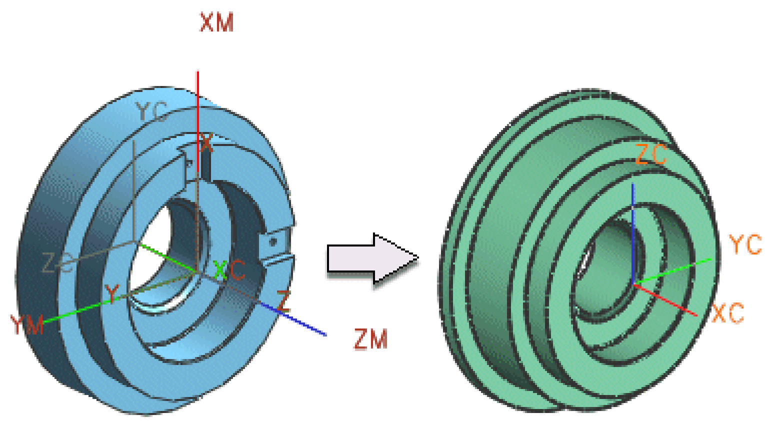
Step 6: Look at the IPW part file.

You will open the ipw part file and observe the orientation of the WCS.

☐ Open the part file ***__ipw.prt**.

☐ Choose **Initialize** at the bottom of the Machining Environment dialog.

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.



14

Step 7: 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.

☐ Choose **Window→***_mill_turn_assy_mfg_1.prt** in the menu bar to display the assembly.

☐ Choose the **Add Existing Component** icon. 

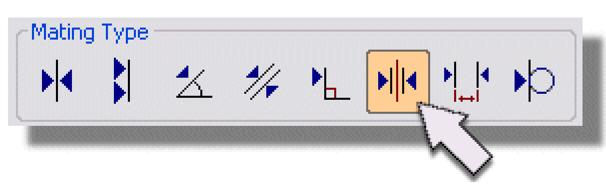
☐ Choose ***_ipw** in the listing window.

☐ Choose **OK** to accept the selected part.

☐ Set the **Positioning** option to **Mate**.

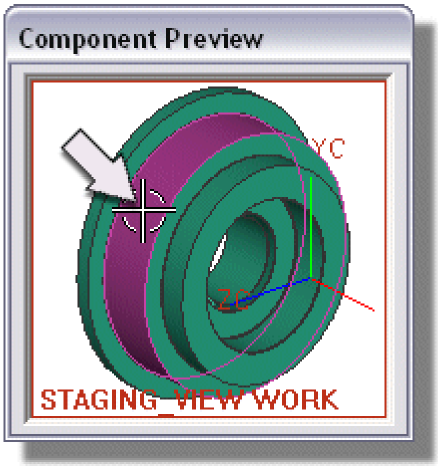
☐ Choose **OK** to accept it.

☐ Choose the **Center** icon under Mating Type.

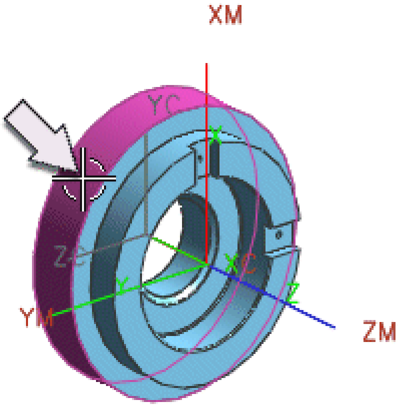


☐ In the Component Preview window, **MB3→Orient View→Trimetric**.

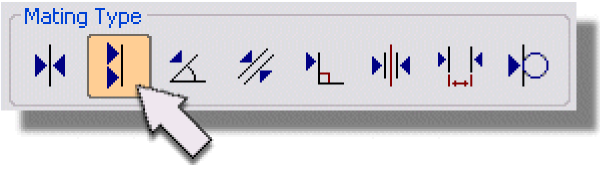
☐ Choose the cylindrical face of the IPW as the From face.



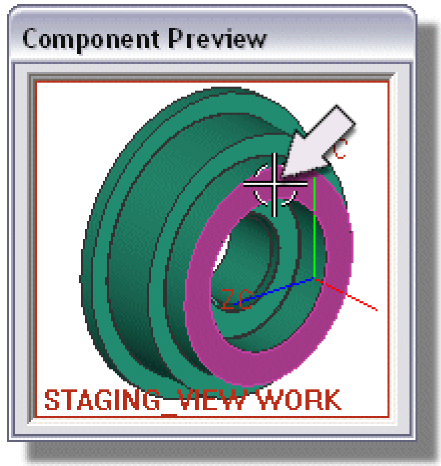
☐ Choose the cylindrical face of the part as the To face.



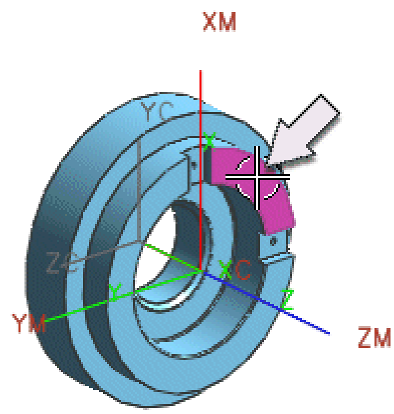
☐ Choose the Align icon under Mating Type.



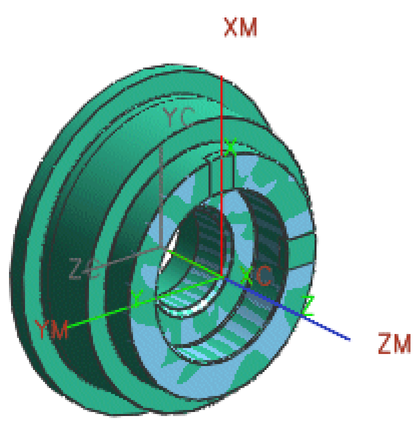
☐ Choose the front face of the IPW as the From face.



☐ Choose the front face of the part as the To face.



☐ Choose **OK** twice to see the IPW superimposed on the part.



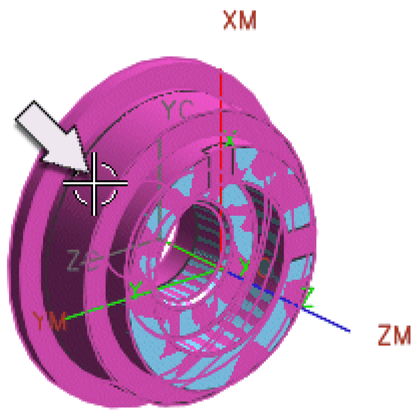
☐ **Cancel** the Select Part dialog to complete the assembly.

Step 8: Select the IPW as the blank geometry.

☐ Double-click the **BLANK_M1** object in the Operation Navigator.

☐ In the Workpiece dialog, choose the **Blank** icon and **Select**.

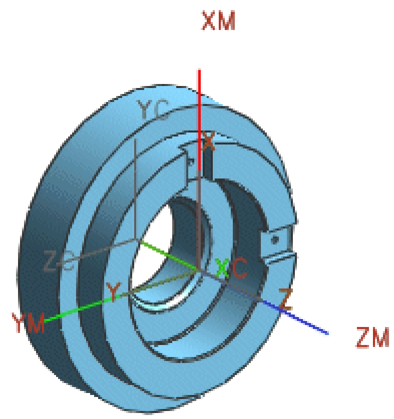
☐ Select the turning IPW as the blank geometry and then choose **OK**.



☐ **OK** to accept the Workpiece dialog.

☐ Choose the check mark next to *****ipw** in the Assembly Navigator to remove the blank component from the display.

Only the part should be displayed in the assembly.



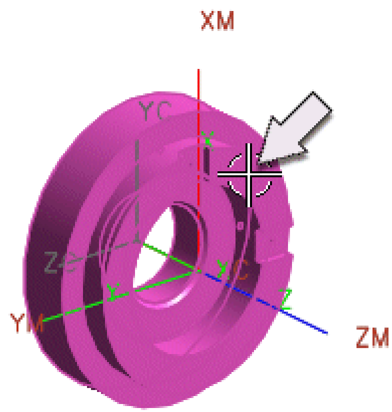
Step 9: Select the part geometry.

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 the **PART_M1** object in the Operation Navigator.

☐ In the Workpiece dialog, choose the **Part** icon and **Select**.

☐ Select the part and choose **OK**.



☐ **OK** to accept the Workpiece dialog.

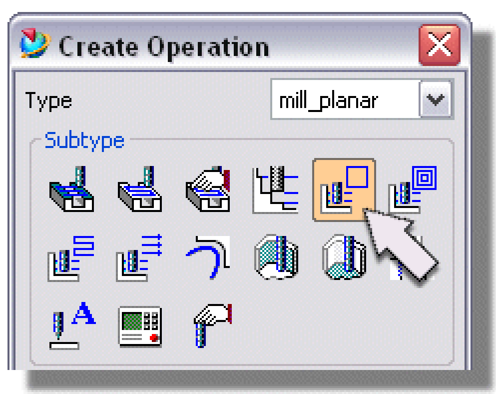
Step 10: Begin creating the first milling operation.

You will create a Planar Mill operation that machines one of the slots on the front face of the part.

☐ Choose **Create Operation**. 

☐ Set the **Type** option to **mill_planar**.

☐ Choose the **PLANAR_PROFILE** icon.



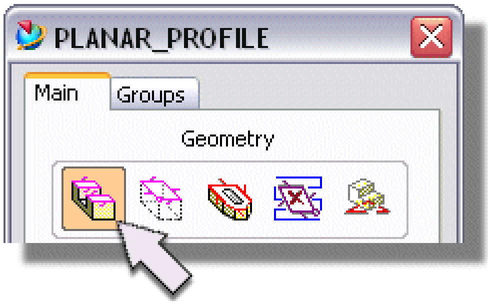
☐ Set the following Parent Groups:

- 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 11: Define the part boundary.

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



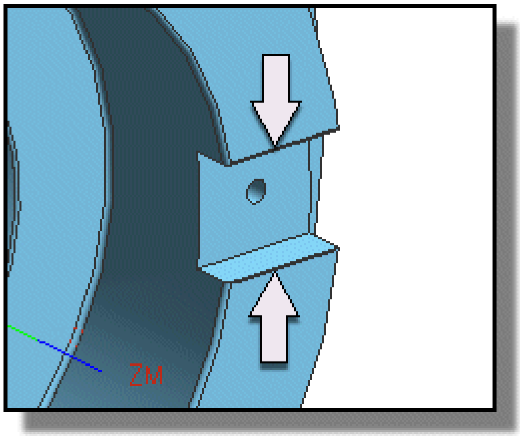
☐ Set the **Mode** option to **Curves/Edges**.

☐ Set the **Type** option to **Open**.

☐ Set the **Plane** option to **User-Defined**.

☐ Choose the **Two Lines** icon.

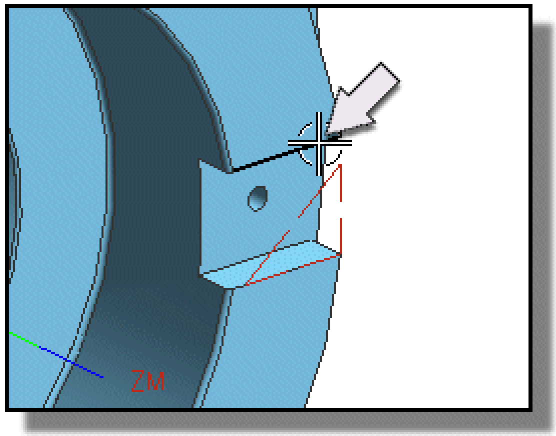
☐ Select the two lines illustrated below.



☐ Set the **Material Side** option to **Right** and be sure the **Tool Position** option is set to **Tanto**.

14

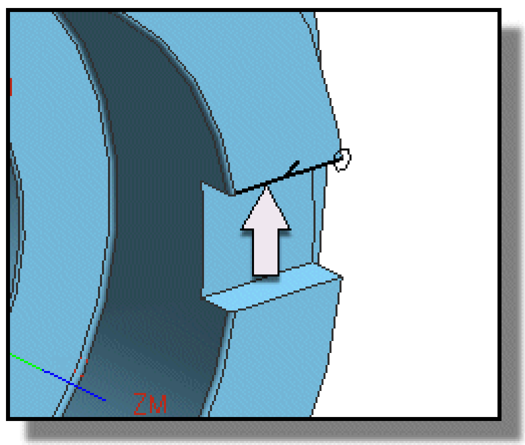
☐ Choose the upper line at the end toward the outside diameter.



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

☐ Choose **OK** to complete the boundary geometry.

☐ Choose **Display**.

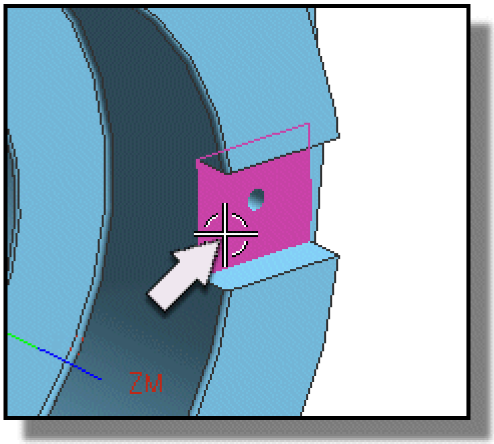


Step 12: Define the remaining operation parameters.

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

☐ Set the Filter option to **Face**.

☐ Select the face as illustrated below and **OK** to define it as the floor.

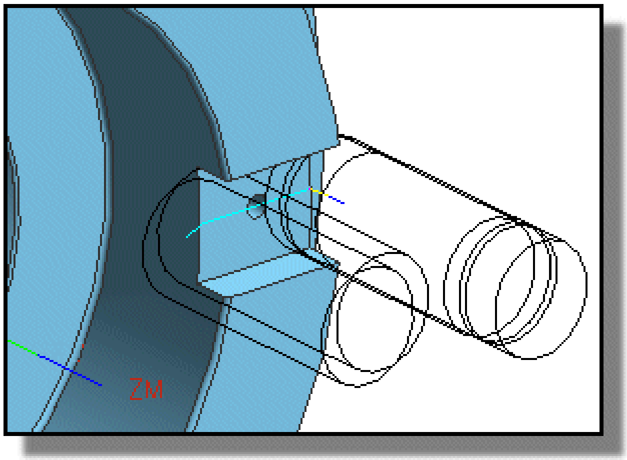


☐ Be sure the **Cut Depth** option is set to **Floor Only**.

☐ 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 13: 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 14: Redefine the part boundary.

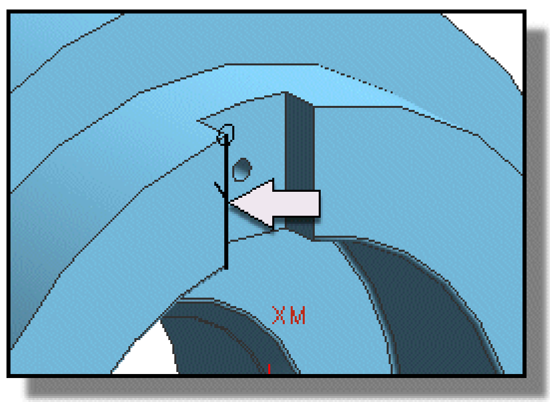
You will redefine the part boundary so that the operation machines the other slot.

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

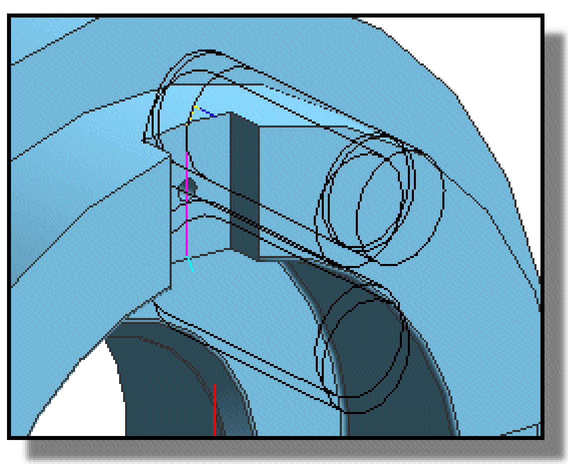
☐ Choose the **Part** icon and **Reselect**.

☐ **OK** the warning.

☐ Create the boundary illustrated below as you did before.



☐ **Generate** the tool path.



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

Step 15: Create the spot drilling operation.

You will create an operation that spot drills the holes in the slots.

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

☐ Set the Type option to **drill**.

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

☐ Set the following Parent Groups:

- Program..........**TAPE_12345_A**
- Use Geometry....**PART_M1**
- Use Tool..........**SPOTDRILL_.250**
- Use Method......**SPOT_DRILLING**

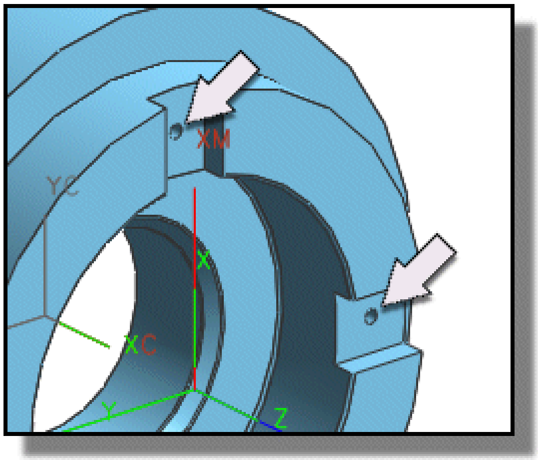
☐ Choose **OK** to begin creating the operation.

Step 16: Define the drilling geometry.

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

☐ Choose **Select** in the Point dialog.

☐ Select the two holes.

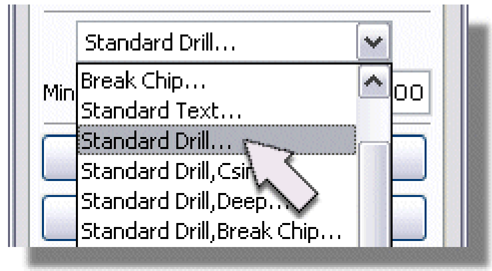


☐ Choose **OK** to accept them.

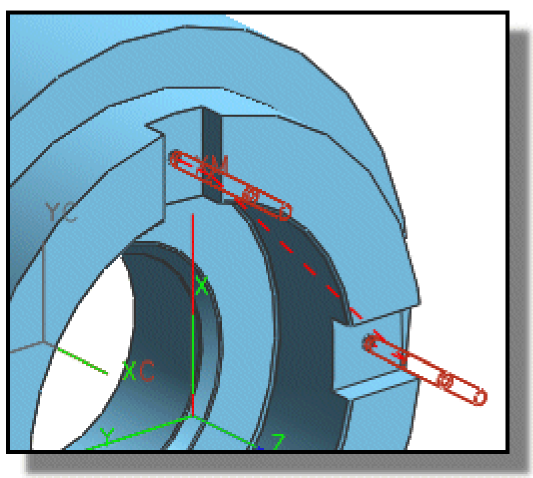
☐ **OK** the Point dialog.

Step 17: 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.

14

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.
- ☐ In the Program Order View of the Operation Navigator, highlight**TAPE_12345_A**.
- ☐ Choose **MB3→Generate**.
- ☐ Choose the **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_mfg_1.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.

```
:2340 T02 M06
N2350 T01
N2360 G0 G90 X4.25 Y0.0 C270. S0 M03
N2370 G43 Z-.4 H07
N2380 G82 Z-.5 R-.4 F.1
N2390 C0.0
N2400 G80
N2410 M02
%
```

- ☐ Dismiss the Information window.
- ☐ Save and close the part.

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.

Appendix

A Merging Lathes

Purpose

This lesson will teach you how to use the Synchronization Editor to control the synchronization and merging of machining operations and events for Merging Lathes and Multiple Spindle Production Centers.

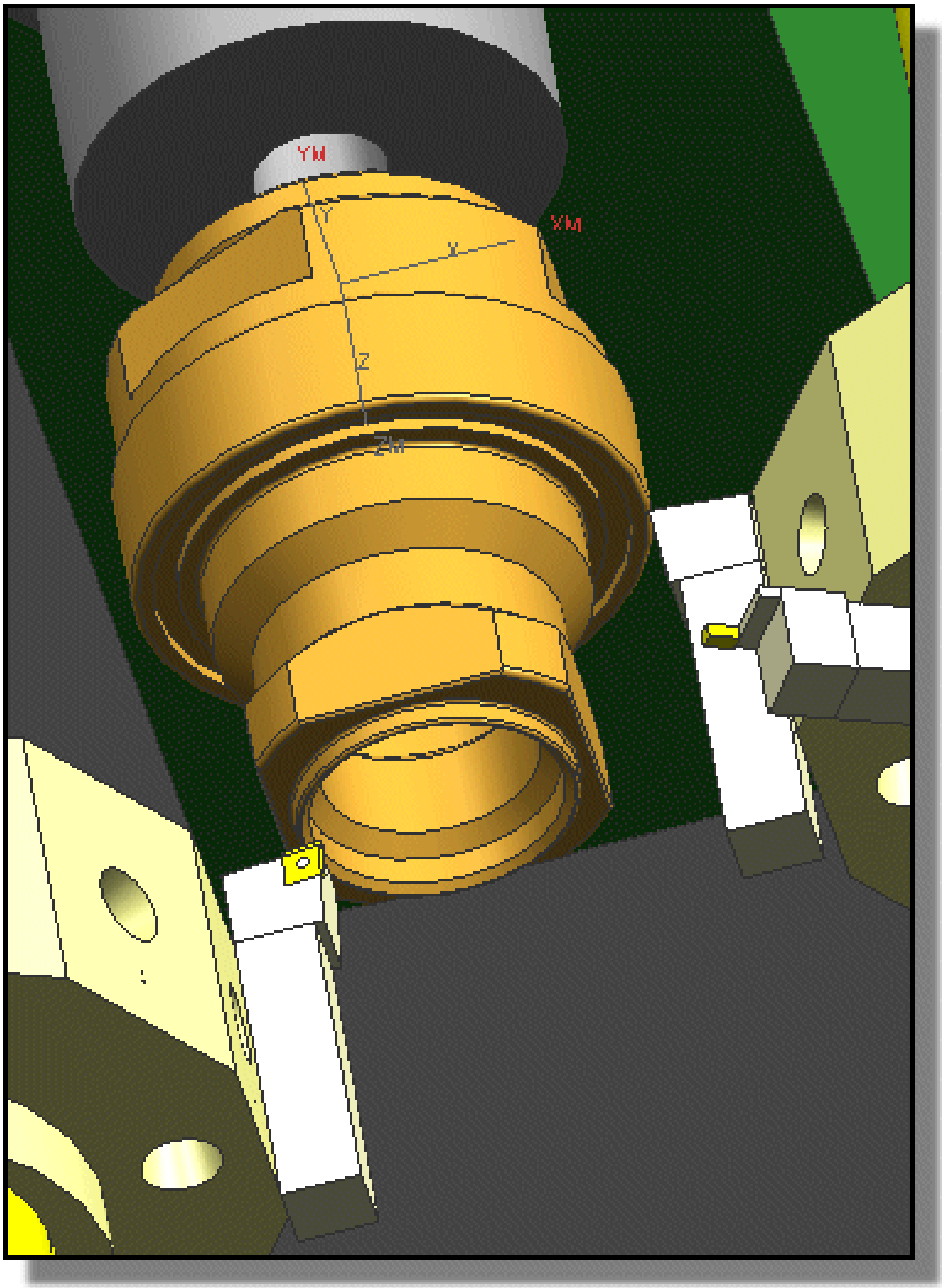
Objectives

After completing this lesson, you will:

- Define a Single Spindle, Two Turret Program.
- Use the Synchronization Editor to synchronize turret movements within the program.

Merging Lathes

Merging Lathes allows you to program turning machines that can simultaneously cut with two or more tools. You can apply synchronization states to tool paths when programming any type of merging lathe from Vertical Turret Lathes to complex production centers. It includes a method for synchronizing the motion of turrets, capturing the synchronization positions, and displaying and simulating the synchronized results for verification and post processing.



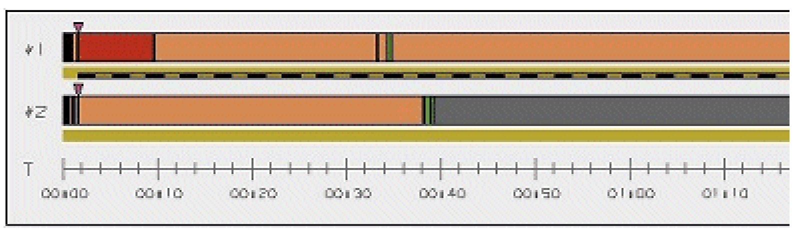
Synchronization Editor

The Synchronization Editor lets you control the synchronization and merging of machining operations and events for Merging Lathes and Multiple Spindle Production Centers. It gives you an overview of all operations assigned to your turrets, slides or other machine tool components that are working in parallel. These are presented in a multi-column list window that lets you conveniently reorder the operations and events using cut, copy, paste, or drag and drop.

The Synchronization Editor supports the insertion of wait and synchronization marks, rearranging the operations and events into the proper (parallel) time sequence for you. Integrated visualization of machining times (time chart) and simulation control (ISV) provide immediate verification feedback. This feature is useful if you need to optimize your NC-program for simultaneous machining of multiple slides or turrets. Use this feature if you want to achieve shorter machining times with better balanced NC-programs for Merging Lathes and Multiple Spindle Production Centers.

Time Chart Diagram

The Time Chart Diagram window gives you an indication of the efficiency of your program in graphical form. This feature is useful if you want to inspect how far your program has been optimized through the usage of simultaneous machining with multiple slides or turrets. Use this feature if you want to see a graphical representation of your NC-program's machining times versus idle times, the synchronization marks and wait events. It helps you to judge whether your NC-program for Merging Lathes is sufficiently balanced.



Simulation Control for Multiple Slide / Multiple Spindle Machining

The integrated Simulation Control provides functionality that lets you control the simulation of your Multiple-Slide / Multiple-Spindle NC-program from within the Synchronization Editor. It allows you to produce a program for Merging Lathes is efficient and free from collisions before it is sent out to production.

Select the Play button to start the simulation from within the Synchronization Editor, or choose Step Forward/Backward in order to single step through your program. At any time while the simulation is running, you may click the Stop icon to terminate the simulation. Pressing the Reset button makes the simulation return to the event or operation where your simulation started. Also, you may adjust your simulation speed using an integrated UI control. If you select the Simulation Options icon, you can activate collision checking or configure collision pairs.

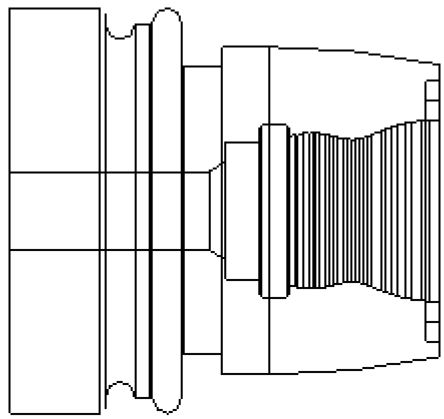
ISV for Merging Lathes

The CAM Integrated Simulation and Verification module (ISV) provides simulation and time calculation for Merging Lathes and Multiple Spindle Production Centers. You use it in the same way as the current milling ISV module, but while creating and synchronizing manufacturing operations to be machined on merging lathes and multiple spindle production centers.

Activity: Defining a Single Spindle, Two Turret Program

In this activity you will use the Operation Navigator to change a single turret turning program into a two turret program. You will then use the Synchronization Editor to synchronize turret motions and insert dwells.

Step 1: Open the **tmp_merging_lathe_mfg** part and start the Manufacturing application.



Step 2: Add pockets to carriers.

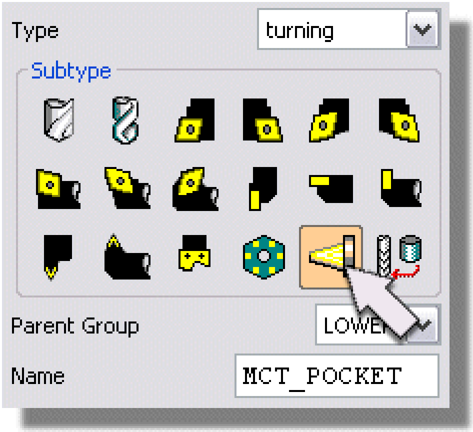
You will add numbered pockets to each carrier. In doing so, you will define tool numbers at the pocket level. This is the most common method of determining the tool number on the machine tool and the method you will most likely use. These pocket numbers will be inherited by the tools and operations as tool numbers and output to the postprocessor.

☐ Display the Machine Tool view of the Operation Navigator.

☐ Choose the **Create Tool** icon.

☐ Be sure the **Type** option is set to **turning**.

☐ Choose the **MCT_POCKET** icon.



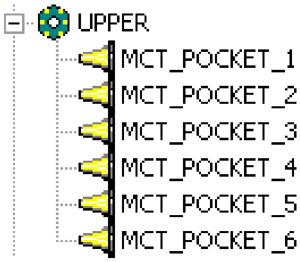
☐ Set the **Parent Group** option to **UPPER**.

☐ Key in MCT_POCKET_1 in the **Name** field.

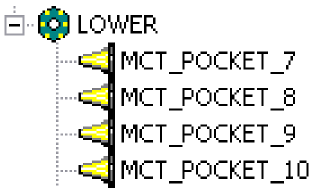
☐ **Apply** to begin defining the pocket.

☐ Key in **1** in the Pocket ID field and **OK** the pocket dialog.

☐ Key in MCT_POCKET_2 in the **Name** field and repeat the process until MCT_POCKET_1 through 6 have been defined in the UPPER carrier as illustrated below.



☐ Create pockets 7–10 inside the LOWER carrier as illustrated below.



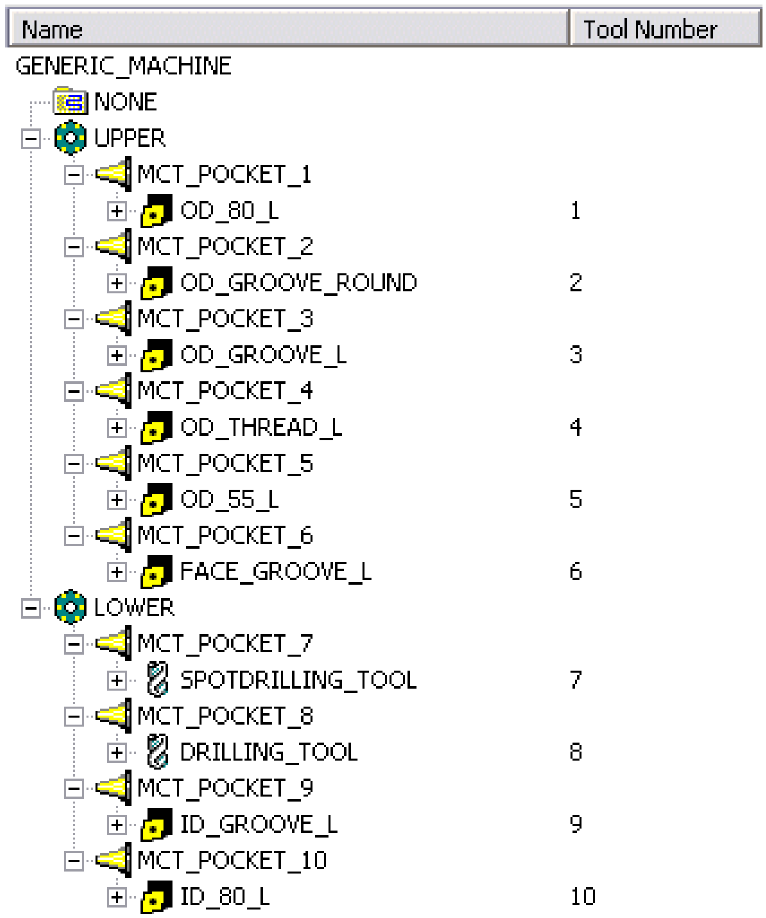
☐ **Cancel** the Create Tool dialog.

A

Step 3: Place tools into pockets.

You will drag and drop tools into the pockets you just defined. In doing so, the tools inherit the pocket numbers and the operations are divided between the upper and lower turrets. Determining the tool number by the pocket in which the tool is placed is analogous to a machine operator picking up a tool and putting it into a pocket on the machine.

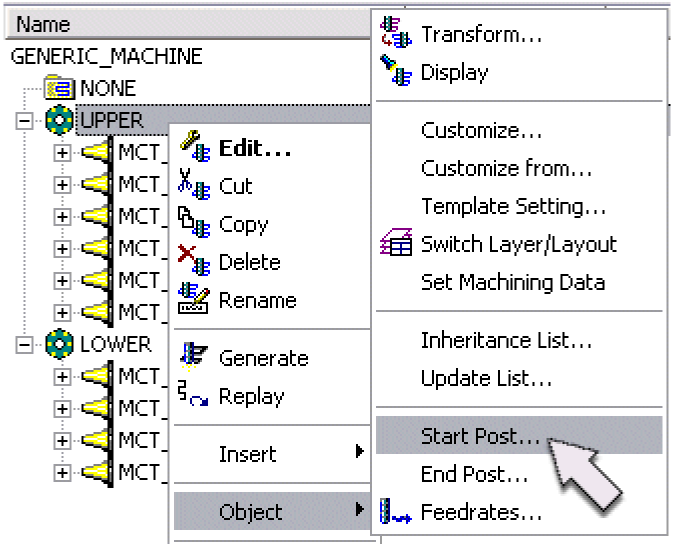
☐ Drag and drop each tool into each pocket as illustrated below.



Step 4: Specify the postprocessors to use for each turret.

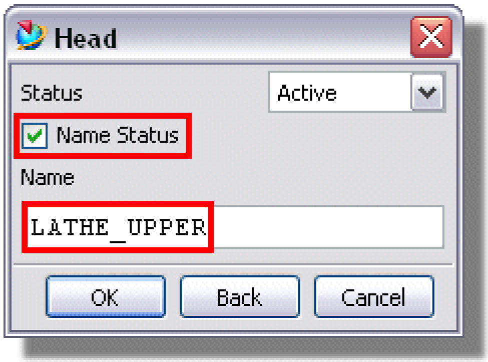
You must associate the UPPER and LOWER turrets with the appropriate postprocessors.

☐ In the Machine Tool view of the Operation Navigator, choose **UPPER** and **MB3→Object→Start Post**.



☐ In the User Defined Events dialog, choose **Head** in the Available list (you may need to scroll) and choose **Add**.

☐ In the Head dialog, turn the **Name Status** option **on** and key in **LATHE_UPPER** in the Name field.



LATHE_UPPER is the "trigger", defined within the postprocessor, that specifies the correct post for the upper turret.

☐ **OK** the Head dialog.

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

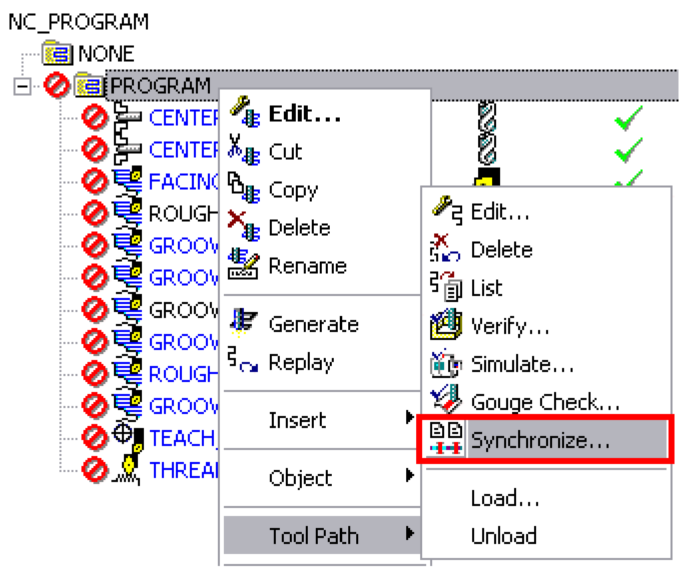
A

☐ Repeat these steps for the LOWER turret, keying in **LATHE_LOWER** in the Name field.

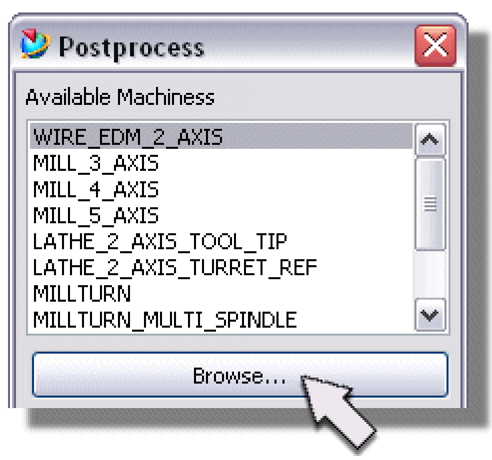
Step 5: Specify the postprocessor for the program.

The program must be assigned a single postprocessor. Yet, the program must use two postprocessors, one for the upper turret and one for the lower turret. To accommodate this situation, you must specify a single postprocessor that is linked to the postprocessors for the upper and lower turrets.

☐ In the Program Order view of the Operation Navigator, highlight PROGRAM and **MB3→Tool Path→Synchronize**.

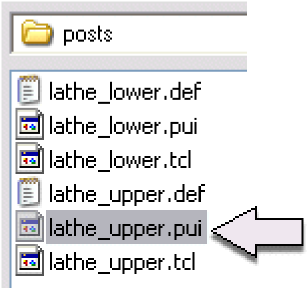


☐ Choose **Browse**.



A

☐ Browse to where your part files are saved, open the **posts** folder, and choose **lathe_upper.pui**.



This is linked to the postprocessors for the upper and lower turrets.

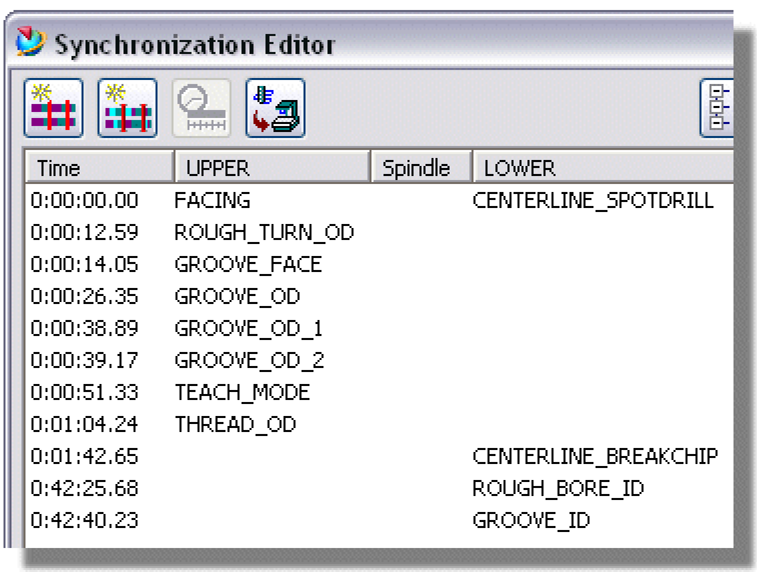
☐ **OK** the Open Postprocessor dialog.

☐ **OK** the Postprocess dialog.

Step 6: Synchronize the operations.

The Synchronization Editor reflects the program order as defined by the Program Order view. While the program order cannot be changed in the Synchronization Editor, the Synchronization Editor does allows you to determine when each turret becomes active. This allows you to determine when they run independently and simultaneously.

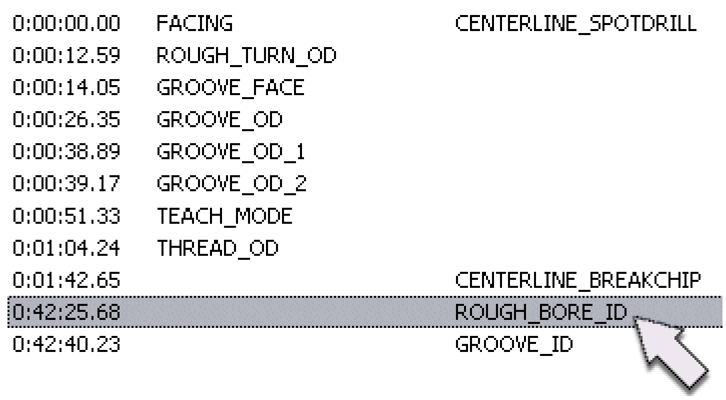
In this example, the facing and spotdrilling operations execute simultaneously. The remaining operations execute sequentially.



Facing should not occur while spotdrilling because this will cause a collision between upper and lower turrets. Rough od and rough id operations, however, should run simultaneously for program efficiency. Synchronization marks must be inserted to coordinate the turret movements.

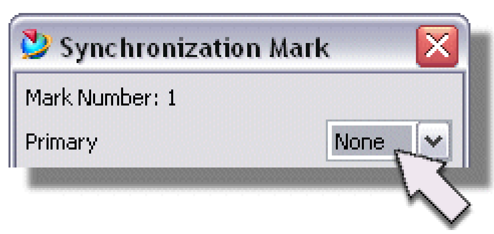
You will begin by specifying that the spotdrilling operation will run first followed by the drilling operation. The upper turret will remain idle while these operations are executing.

- ☐ Highlight **ROUGH_BORE_ID**.

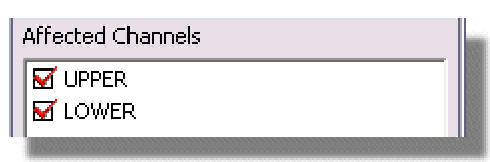


- ☐ Choose the **Insert Sync Event** icon. 

- ☐ Set the **Primary** option to **None**.



None must be specified when a single turret is active and no dependency between turrets (primary turret) is needed. Because you must always synchronize at least two turrets, UPPER and LOWER under must both remain turned on.

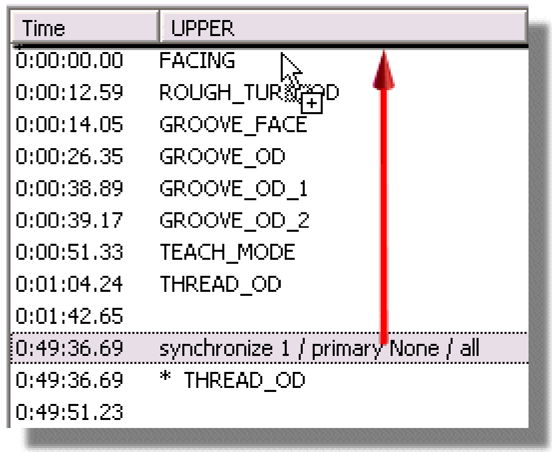


In this case, the synchronization involves allowing one turret to remain idle while the other performs drilling operations.

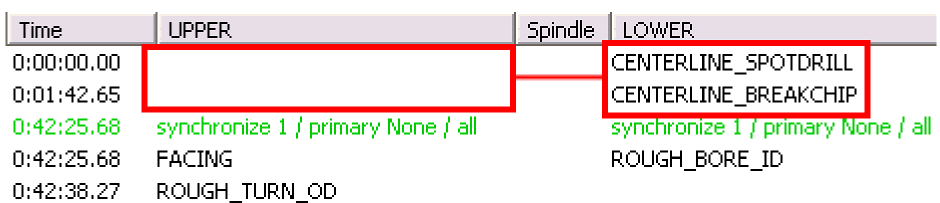
☐ **OK** the Synchronization Mark dialog.

The Synchronization Mark always appears above the selected operation.

☐ In the UPPER column, drag and drop the synchronization mark so it is positioned above FACING.

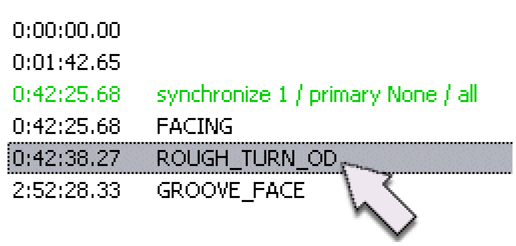


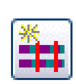
Now the spotdrilling and drilling operations run in sequence using the lower turret while the upper turret sits idle.



Next, you want facing to occur after drilling. You do not want the facing operation to run in conjunction with any other operation. This will require that the upper turret is active while the lower turret sits idle. To do this, you must position a synchronization mark after FACING.

☐ Highlight **ROUGH_TURN_OD**.

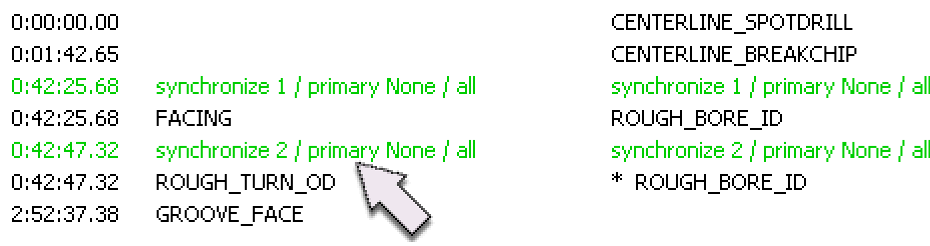


☐ Choose the **Insert Sync Event** icon. 

☐ Set the **Primary** option to **None**.

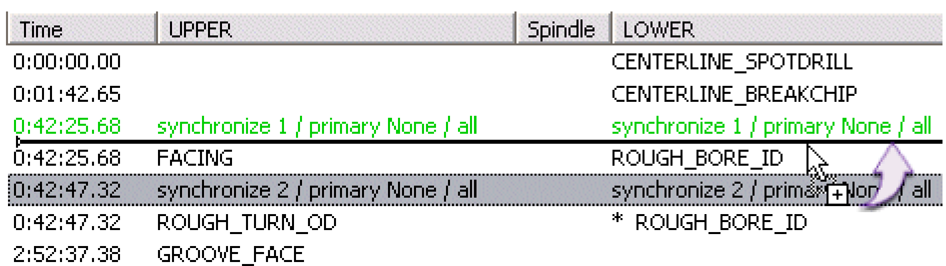
☐ **OK** the Synchronization Mark dialog.

The Synchronization Mark always appears above the selected operation.

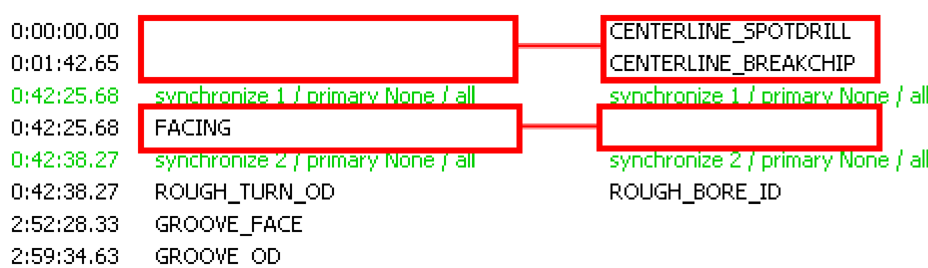


The asterisk next to ROUGH_BORE_ID indicates that the rough bore id operation has been interrupted by the facing operation.

☐ Drag the synchronization2 mark in the LOWER column above ROUGH_BORE_ID.

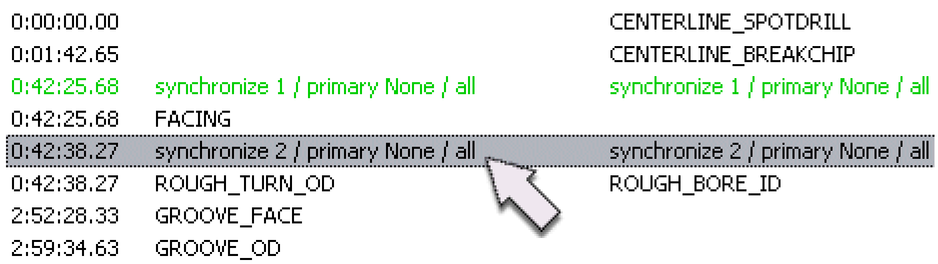


Spotdrilling and drilling operations run in sequence using the lower turret while the upper turret sits idle. The facing operation then runs using the upper turret while the lower turret sits idle.



Notice that the ROUGH_TURN_OD and the ROUGH_BORE_ID operations both run at the same time. This is what you want. However, when two operations run at the same time, you must designate one of the two operations as Primary. By doing this, the system is able to resolve conflicting operation parameters such as spindle speeds.

☐ Double-click the second synchronization mark.



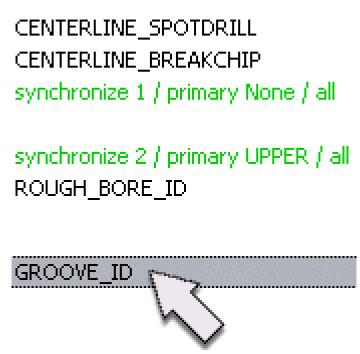
☐ In the Synchronization Mark dialog, set the **Primary** option to **UPPER**.

The parameters defined by the upper turret operation, ROUGH-TURN_OD, are now used to resolve conflicting operation parameters.

☐ **OK** the Synchronization Mark dialog.

Next, you want the GROOVE_OD and GROOVE_ID operations to run at the same time.

☐ Highlight **GROOVE_ID**.



☐ Choose the **Insert Sync Event** icon.

☐ Set the **Primary** option to **LOWER**.

☐ **OK** the Synchronization Mark dialog.

The Synchronization Mark always appears above the selected operation.

The asterisk next to TEACH_MODE indicates that the teach mode operation has been interrupted by the groove id operation.

☐ Drag the synchronization3 mark in the UPPER column above GROOVE_OD.

Now the GROOVE_OD and GROOVE_ID operations run at the same time. The parameters defined by the lower turret operation, GROOVE_ID, are used to resolve conflicting operation parameters.

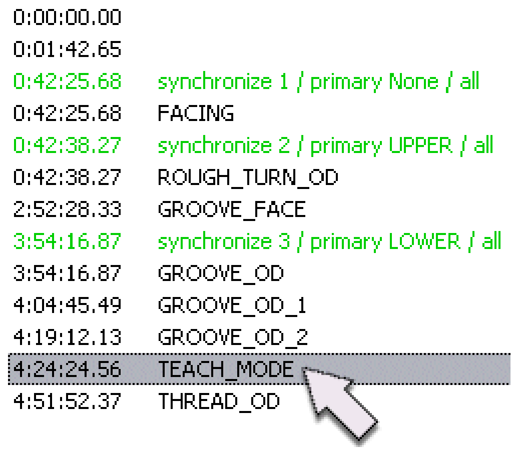
Time	UPPER	Spindle	LOWER
0:00:00.00			CENTERLINE_SPOTDRILL
0:01:42.65			CENTERLINE_BREAKCHIP
0:42:25.68	synchronize 1 / primary None / all		synchronize 1 / primary None / all
0:42:25.68	FACING		
0:42:38.27	synchronize 2 / primary UPPER / all		synchronize 2 / primary UPPER / all
0:42:38.27	ROUGH_TURN_OD		ROUGH_BORE_ID
2:52:28.33	GROOVE_FACE		
3:54:16.87	synchronize 3 / primary LOWER / all		synchronize 3 / primary LOWER / all
3:54:16.87	GROOVE_OD		GROOVE_ID
4:04:45.49	GROOVE_OD_1		
4:19:12.13	GROOVE_OD_2		
4:24:24.56	TEACH_MODE		
4:51:52.37	THREAD_OD		

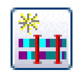
Centerline spotdrilling is followed in sequence by centerline drilling. Both are performed by the lower turret while the lower turret remains idle. The upper turret then faces the part while the lower turret remains idle. Rough turn od and rough turn id operations are then performed simultaneously by the upper and lower turrets. The upper turret is designated as primary to resolve conflicting operation parameters. The face of the part is then grooved by the upper turret while the lower turret remains idle. Next, the groove od and groove id operations are run simultaneously with the lower turret designated as primary. Finally, the remaining operations are executed in sequence by the upper turret while the lower turret remains idle.

Step 7: Insert a dwell.

Dwells can be added to pause operations. This sometimes becomes necessary when a manual process must be performed by the operator. Like synchronization marks, dwells are inserted above the selected operation.

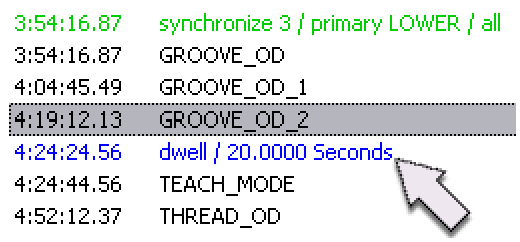
☐ Highlight **TEACH_MODE**.



☐ Choose the **Insert Dwell Event** icon. 

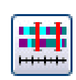
☐ Key in **20.000** in the Dwell Value field.

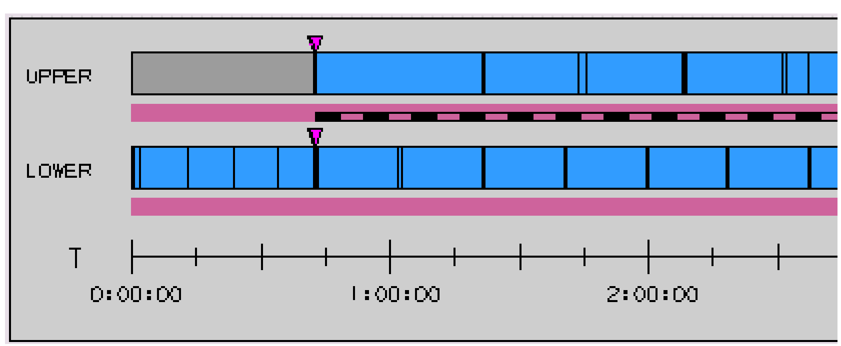
☐ **OK** the Dwell dialog.



Step 8: View the time chart.

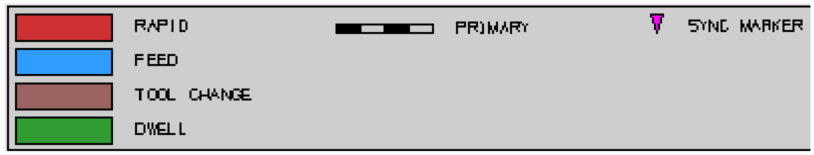
The time chart is a graphical representation of turret events along a time line.

☐ Choose the **Open time chart** icon. 



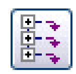
☐ Click on the operations and the synchronization marks in the Synchronization Editor to graphically locate their positions on the time chart.

☐ Choose the **Toggle Legend** icon to display a legend of the colors and symbols used in the time chart. 



Step 9: View operation details.

Expand All allows you to view every move and command in each of the operations.

☐ Choose the **Expand All** icon. 

Time	UPPER	Spindle	LOWER
0:00:00.00	start of program		start of program
0:00:00.00	start of group		start of group
0:00:00.00			machine mode
0:00:12.00			first turret
0:00:12.00			start of path / CENTERLINE_SPOTDRILL
0:00:12.00			set csys
0:00:12.00			first tool
0:00:12.00			msys
0:00:12.00			from/1.8000,0.0000,0.2500
0:00:12.18			initial/0.0000,0.0000,0.2500
0:00:12.45			cut/0.0000,0.0000,0.1000
0:01:42.45			dwell / %.4f %s
0:01:42.45			retract/0.0000,0.0000,0.2500
0:01:42.47			rapid/1.8000,0.0000,0.2500

Step 10: Show postprocessor output.

The Show Machine Code Lines icon shows the postprocessor output.

A

☐ Choose the **Show Machine Code Lines** icon. 

Time	UPPER	Spindle	LOWER
0:00:00.00			%
			N0010 G94 G90 G70
0:00:12.00			N0020 G92 X0.0 Z0.0
			:0030 T07 H00 M06
0:00:12.18			N0040 G94 G00 G90 Z.25
0:00:12.45			N0050 G97 S0 M04
			N0060 G95 G01 Z.1 F.03
0:01:42.45			N0070 G04 X1
0:01:42.45			N0080 Z.25 F.05
0:01:42.47			N0090 G94 G00 X1.8
0:01:54.65			N0100 G92 X1.8 Z.25
			:0110 T08 H00 M06
0:01:54.65			N0120 G94 X0.0 Z.22
0:01:54.92			N0130 G97 S0 M04
			N0140 G95 G01 Z-.88 F.03

Postprocessing is the preparation of machine code used to drive a specific machine tool. Postprocessing converts the internal tool path containing events and motions into a format compatible to the machine tool's controller. By default, the postprocessor will create a file with the root name of the part file and a .ptp extension. To postprocess, a tool path and postprocessor are required.

Choosing the Postprocess icon  will create the .ptp files for the upper and lower turrets in your current directory

☐ Close the part.

A

Summary

Merging Lathes allows you to program turning machines that can simultaneously cut with two or more tools. It includes a method for synchronizing the motion of turrets, capturing the synchronization positions, and displaying and simulating the synchronized results for verification and post processing.

In this lesson you:

- Define a Single Spindle, Two Turret Program.
- Use the Synchronization Editor to synchronize turret movements within the program.

A

Appendix

B Vertical Turret Lathe

Purpose

This lesson will teach you how to create a Vertical Turret Lathe (VTL) program.

Objective

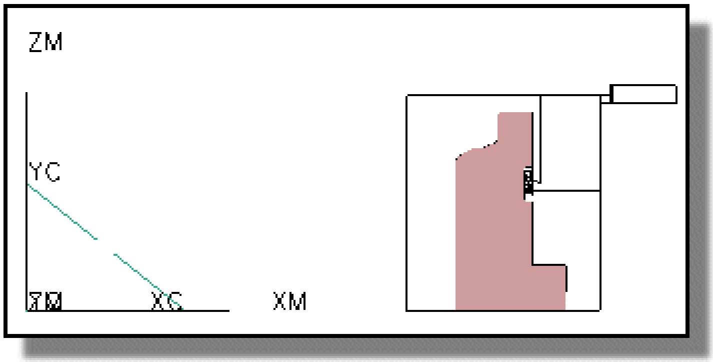
Upon completion of this lesson, you will:

- Examine the setup of a VTL.
- Create facing, roughing, grooving, and finishing operations.
- Postprocess the program.

B

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.

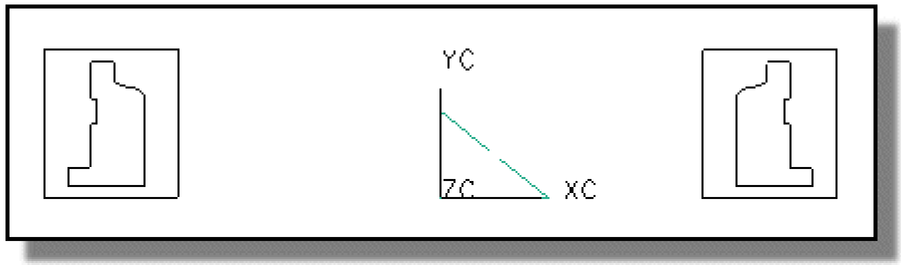


Activity: Examining a VTL Setup and Program

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_mfg_1.prt**.

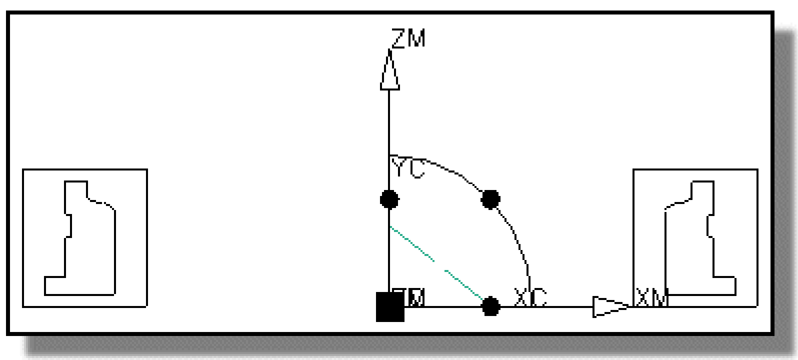


☐ Enter the **Manufacturing** application.

Step 2: View the MCS.

☐ In the Geometry View of the Operation Navigator, double-click **MCS_SPINDLE**.

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.



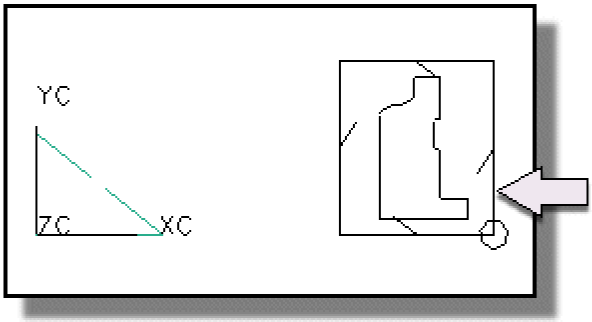
☐ **Cancel** the TURN_ORIENT dialog.

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 **BLANK** in the Operation Navigator.

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



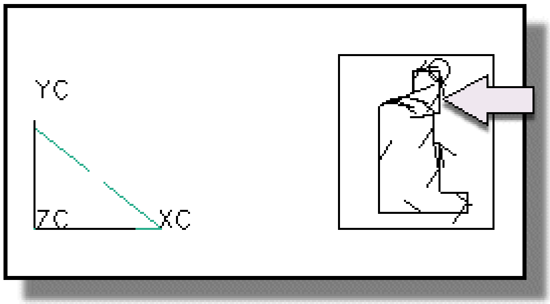
☐ **Cancel** the TURN_BND dialog.

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 **PART** in the Operation Navigator.

☐ Choose **Display**.



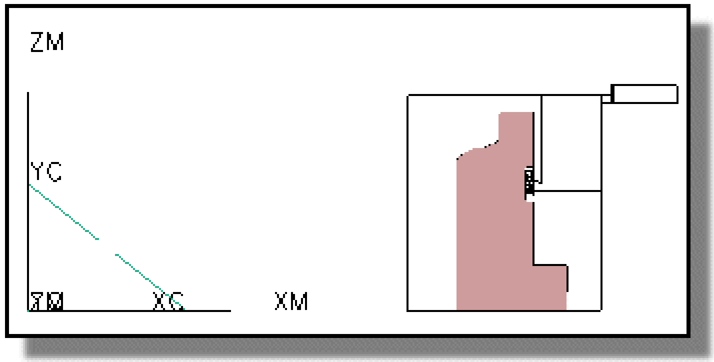
☐ **Cancel** the PART dialog.

Step 5: Verify the tool path.

☐ In the Program Order View of the Operation Navigator, highlight **T563489A** and choose the **Verify Toolpath** icon in the toolbar.

☐ Turn the **2D Material Removal** option **on**.

☐ Choose **Play**.



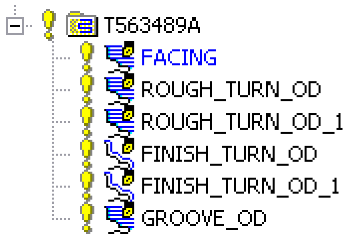
This is the program you will create in the next activity.

Choose **OK** to complete the Toolpath Visualization.

☐ Save and close the part file.

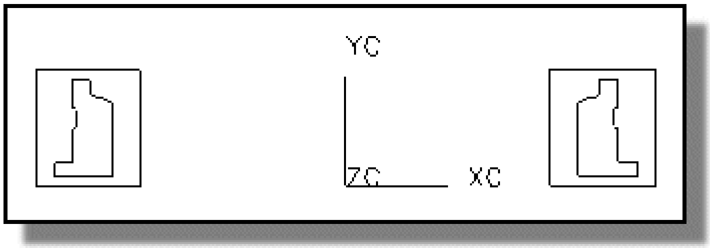
Activity: Creating the Operations

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.

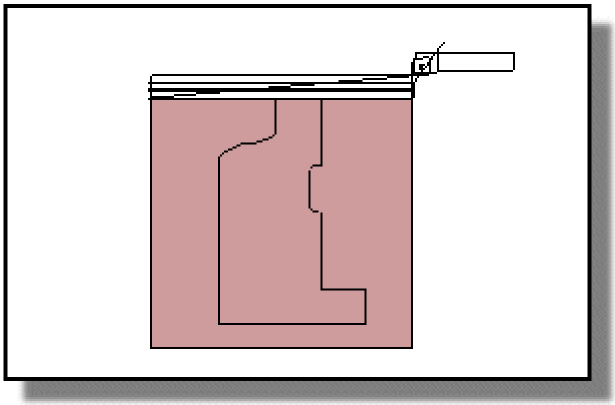
☐ Open the part file **tmp_vtl_mfg_2.prt**.



☐ Enter the **Manufacturing** application.

☐ Save the part file as *****_vtl_mfg_2.prt**, where *** represents your initials.

Step 2: Rough the face.



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

☐ Be sure the **Type** option is set to **turning**.

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

☐ Specify the following parent groups.

- Program..........**T563489A**
- Use Geometry....**PART**
- Use Tool..........**OD_80_5R_RH_RGH**
- Use Method......**LATHE_ROUGH**

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

Step 3: Specify the operation parameters.

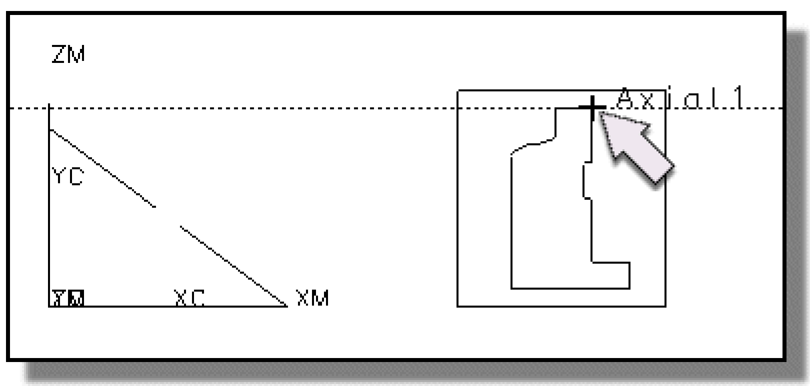
☐ Be sure the **Cut Depth** option is set to **Variable Avg**.

☐ Key in **0.040** in the **Minimum** field and **0.750** in the **Maximum** field.

☐ Choose **Containment**.

☐ Turn the Axial 1 **Trim** option **on**.

☐ Choose **Axial 1** and specify a Trim Plane on face of the part as illustrated below.

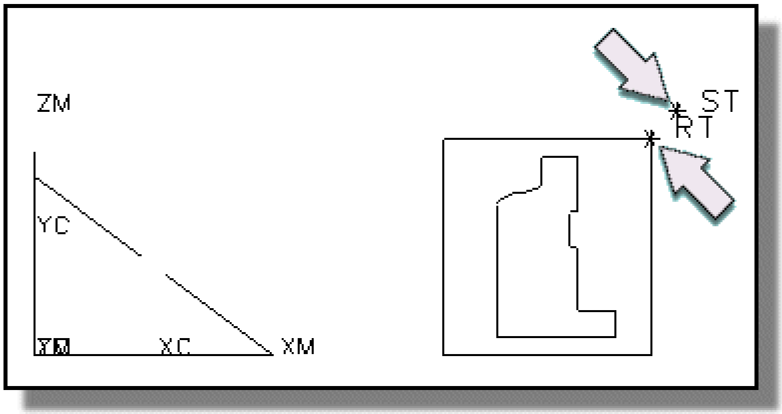


☐ **OK** to accept the Trim Plane.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

B

☐ Choose **Specify** and create a Start Point and a Return Point at the approximate screen positions illustrated below.



☐ **OK** to accept the points.

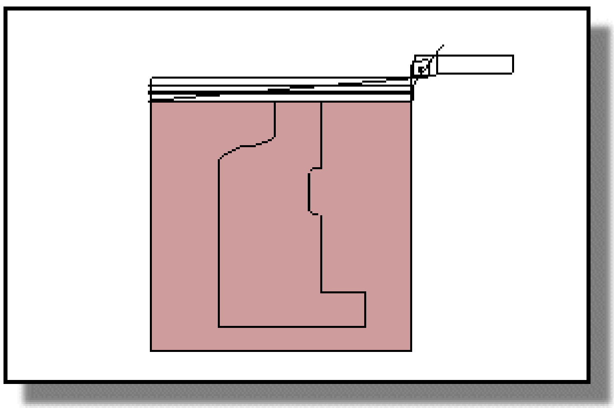
Step 4: Generate the tool path and visualize material removal.

☐ Generate the tool path.

☐ Choose the **Verify** icon at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

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

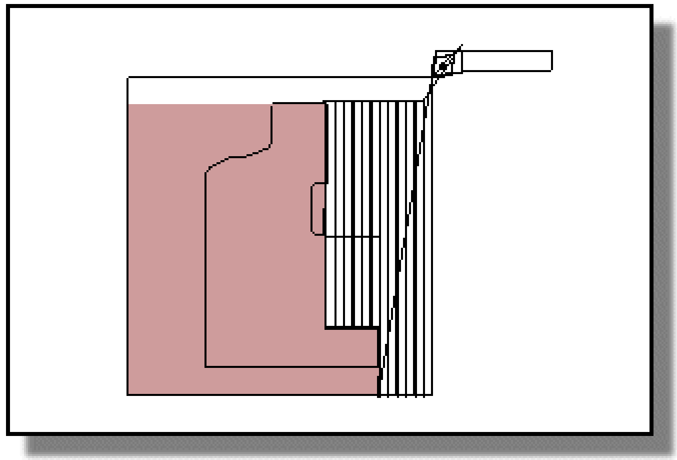


☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

B

Step 5: Rough the OD.



☐ 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_5R_RH_RGH**
- Use Method......**LATHE_ROUGH**

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

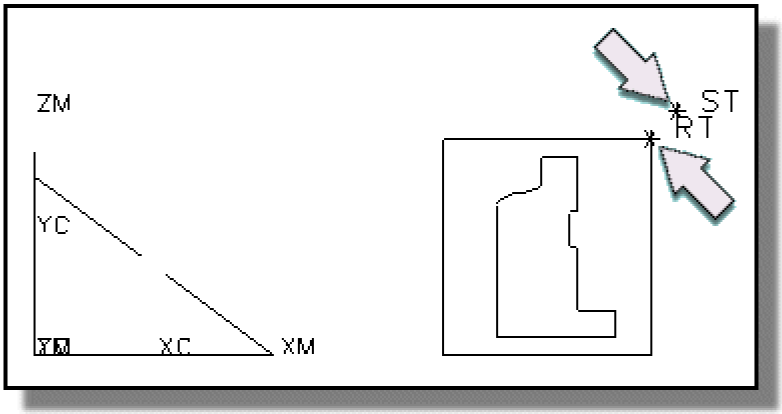
Step 6: Specify the operation parameters.

☐ Be sure the **Cut Depth** option is set to **Variable Avg**.

☐ Key in **0.040** in the **Minimum** field and **0.750** in the **Maximum** 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.

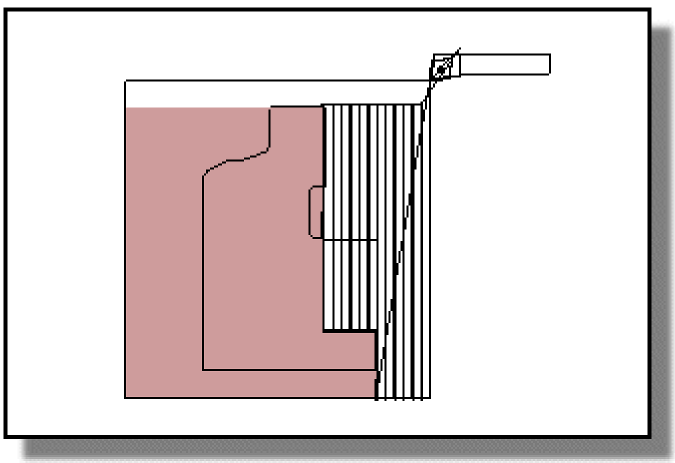
Step 7: Generate the tool path and visualize material removal.

☐ Generate the tool path.

☐ Choose the **Verify** icon at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

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



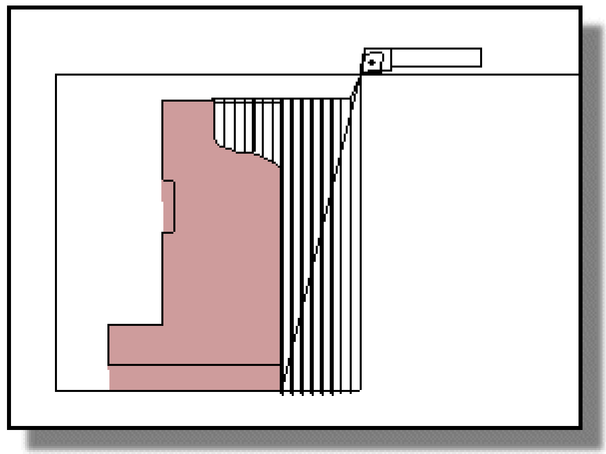
☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

B

Step 8: Rough the ID.

You will use the same turning tool to rough bore as you 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_5R_RH_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.

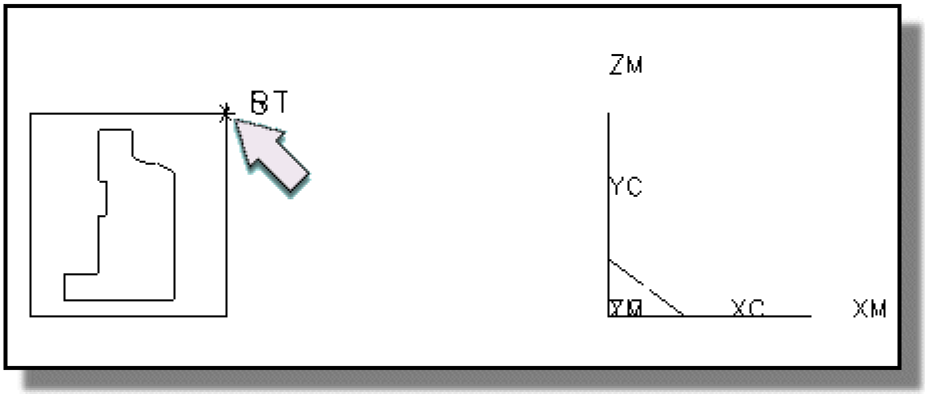
☐ Be sure the **Cut Depth** option is set to **Variable Avg**.

☐ Key in **0.040** in the **Minimum** field and **0.750** in the **Maximum** field.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

B

☐ Choose **Specify** and create both the Start Point and Return Point at the end of the horizontal line as illustrated below. Be sure you define them to the left of the centerline.



☐ **OK** to accept the points.

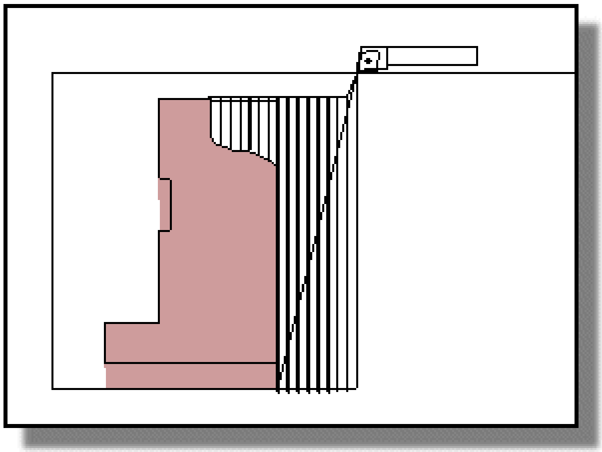
Step 10: Generate the tool path and visualize material removal.

☐ Generate the tool path.

☐ Choose the **Verify** icon at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

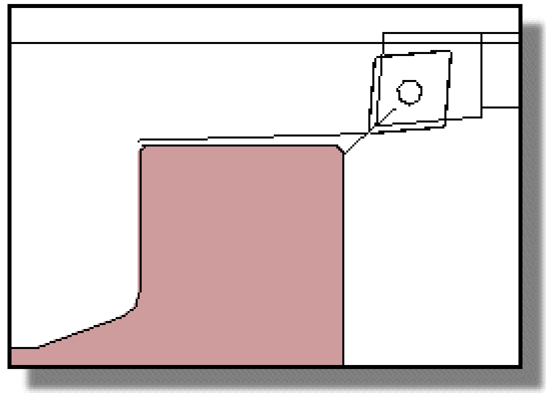
☐ Choose the **Play** icon at the bottom of the dialog.



☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

Step 11: Finish the face.



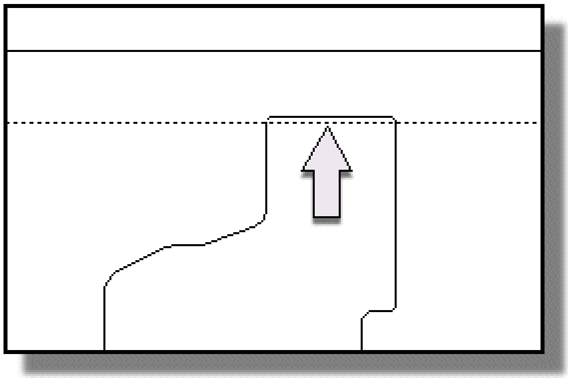
- ☐ Choose the **Create Operation** icon in the toolbar.
- ☐ Be sure the **Type** option is set to **turning**.
- ☐ Choose the **FINISH_TURN_OD** icon as the subtype.
- ☐ Specify the following parent groups.
  - Program..........**T563489A**
  - Use Geometry....**PART**
  - Use Tool..........**OD_80_3R_RH_FIN**
  - Use Method......**LATHE_FINISH**
- ☐ **OK** to begin creating the operation.

Step 12: Specify the operation parameters.

- ☐ Key in **180** in the **Cut Angle**.
- ☐ Choose **Containment**.

☐ Turn the Axial 1 **Trim** option **on** and key in **21.825**.

The axial trim plane should appear just below the part face.

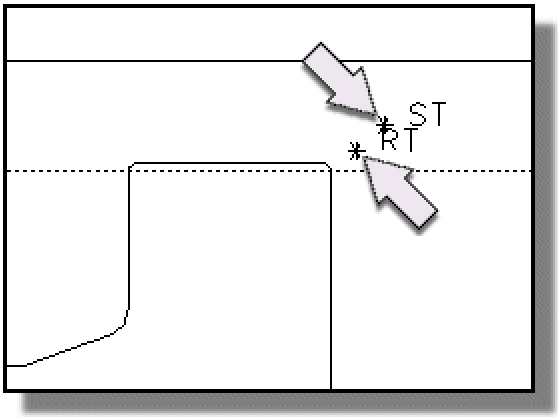


☐ **OK** to accept the Trim Plane.

☐ 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 you define them to the right of the centerline.



☐ **OK** to accept the points.

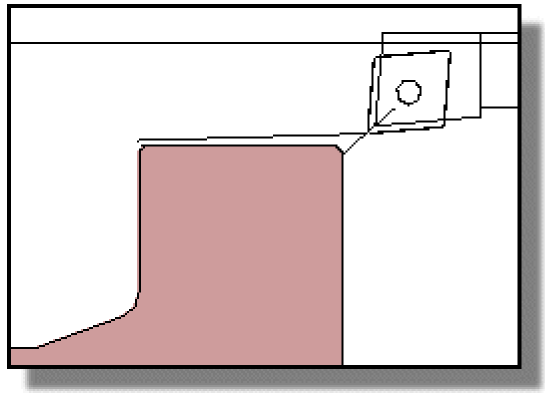
Step 13: Generate the tool path and visualize material removal.

☐ Generate the tool path.

☐ Choose the **Verify** icon at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

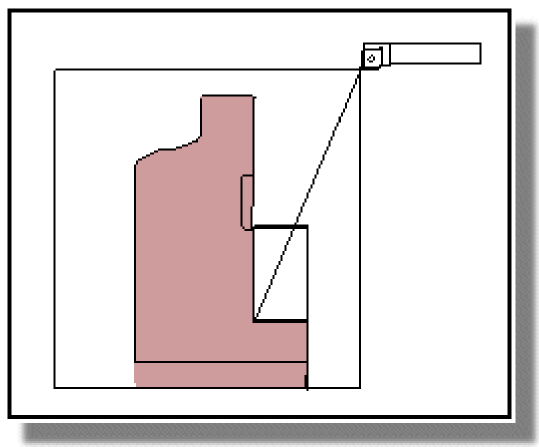
☐ Choose the **Play** icon at the bottom of the dialog.



☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

Step 14: Finish the OD.



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

☐ Be sure the **Type** option is set to **turning**.

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

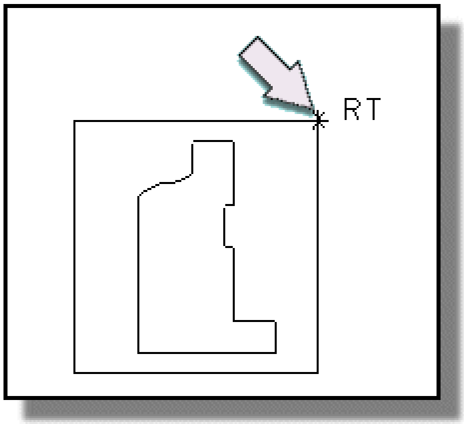
☐ Specify the following parent groups.

- Program..........**T563489A**
- Use Geometry....**PART**
- Use Tool..........**OD_80_3R_RH_FIN**
- Use Method......**LATHE_FINISH**

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

Step 15: Specify the operation parameters.

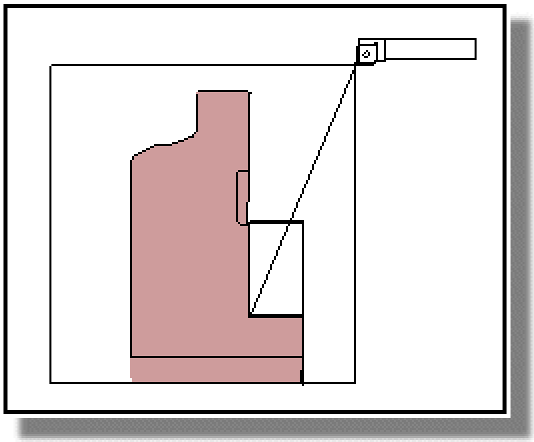
- ☐ Choose the **Diameters First, Then Faces** icon at the top of the dialog.
- ☐ Choose **Avoidance** and turn the **Return Point** option **on**.
- ☐ Choose **Select** and create a Return Point at the end of the horizontal line as illustrated below.



- ☐ **OK** to accept the point.

Step 16: Generate the tool path and visualize material removal.

- ☐ Generate the tool path.
- ☐ Choose the **Verify** icon at the bottom of the dialog.
- ☐ Turn the **2D Material Removal** option **on**.
- ☐ Choose the **Play** icon at the bottom of the dialog.

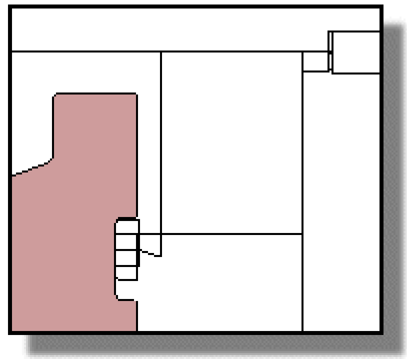


- ☐ **OK** to complete the tool path visualization.

B

☐ **OK** to complete the operation.

Step 17: Cut the groove.



☐ 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 18: Specify the operation parameters.

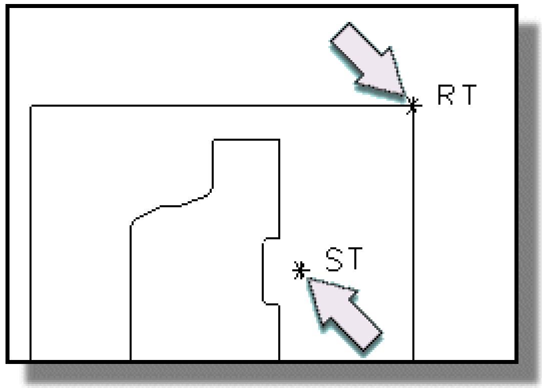
☐ Choose the **Plunge Alternate** icon.

☐ Turn the **Additional Profiling** option **on**.

☐ Choose **Avoidance** and turn the **Start Point** and **Return Point** options **on**.

B

☐ Choose **Specify** and create a Start Point and a Return Point at the screen positions illustrated below.



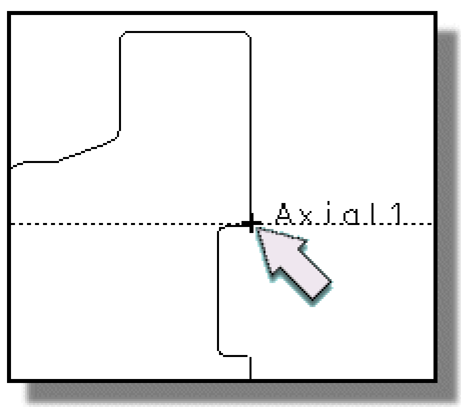
☐ Set the **Motion to Start Point** option for both the Start Point and Return Point to **Radial→Axial**.

☐ **OK** to accept the points.

☐ Choose **Containment**.

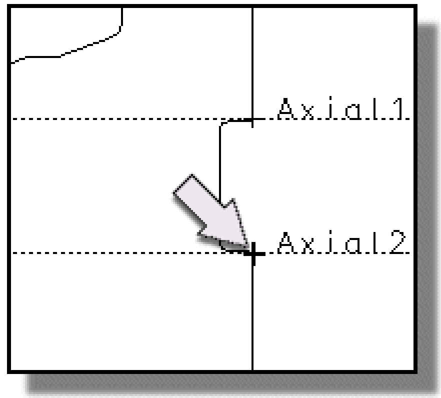
☐ Turn the Axial 1 and Axial 2 **Trim** options **on**.

☐ Choose **Axial 1** and select the side of the groove.



B

☐ Choose **Axial 2** and select the other side of the groove.



☐ **OK** to accept the Trim Planes.

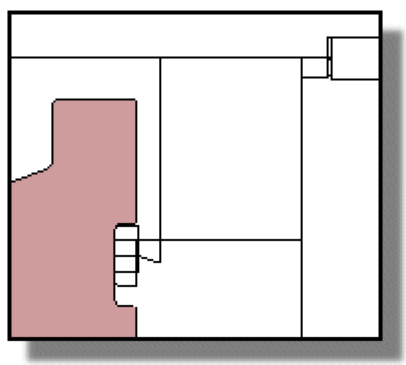
Step 19: Generate the tool path and visualize material removal.

☐ Generate the tool path.

☐ Choose the **Verify** icon at the bottom of the dialog.

☐ Turn the **2D Material Removal** option **on**.

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



☐ **OK** to complete the tool path visualization.

☐ **OK** to complete the operation.

B

Step 20: Postprocess the program.

☐ In the Program Order View of the Operation Navigator, highlight **T563489A**.

☐ Choose the **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.

☐ Scroll through the Information window.

☐ Dismiss the Information window.

☐ Save and close the part file.

Summary

In this lesson you created a Vertical Turret Lathe (VTL) program.

You learned how to:

- Create facing, roughing, grooving, and finishing operations for a VTL.
- Postprocess the program.

Appendix

C Lathe Cross Sections

Purpose

Lathe Cross Section allows you to create associative curves that represent a 2D cross section of a solid body. These cross sectional curves are needed for boundary creation in NX3 and before.

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 cross sections.
- Create complex cross sections.
- Observe the associativity between cross sectional curves and parent geometry.

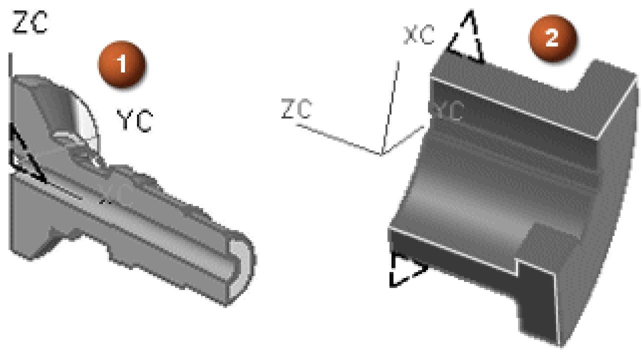
C

The Lathe Cross Section Feature

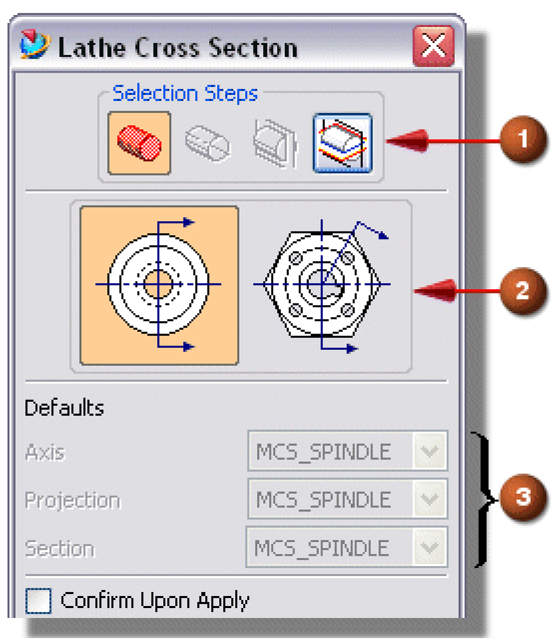
The two cross sectioning methods are:

Simple - The Simple method (1) uses a single section plane. The resulting cross-sectional curves lie in the plane used to create the cross section. This method is intended for parts with consistent cross sections.

Complex - The Complex method (2) uses multiple section planes. The cross-sectional curves are rotated about a specified axis onto a projection plane. This method is intended parts with variable cross sections.



The Selection Steps icons (1) define the body, axis, projection plane, and section plane of the cross section. The next two icons (2) define the cross sectioning method. The Defaults options (3) define the axis, projection plane, and section plane for the cross sectional curves. The MCS_SPINDLE option automatically designates the plane defined by the lathe work plane. The Radial Planes option defines the section planes using the rotation axis and specifying a point. This option is available only when defining complex sections. The User Defined option defines the section planes using the Plane Subfunction.

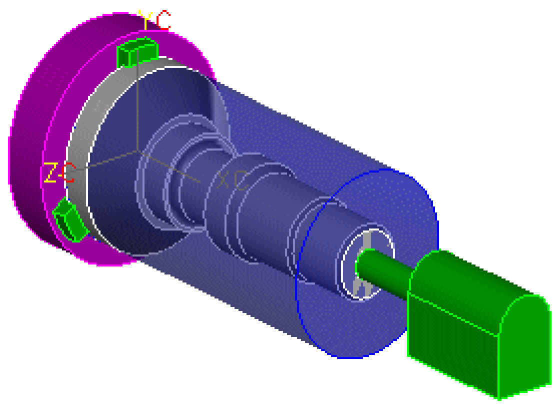


Activity: Creating a Simple Cross Section

In this activity, you are going to create simple cross sections for the blank and the part geometry.

Step 1: Open and rename the part.

- ☐ Open the part **tmp_crs_sec_mfg_1.prt**.
- ☐ Be sure the blank geometry is displayed as translucent. If it is not, choose **Preferences→Visualization→Visual** tab and turn the **Translucency** option **on**.



- ☐ Save the part file as *****_crs_sec_mfg_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.

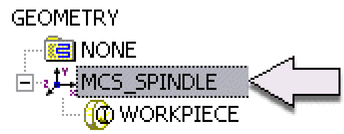
- ☐ Visually check the **WCS** and **MCS** locations.

  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.

- ☐ Display the Geometry View of the Operation Navigator.

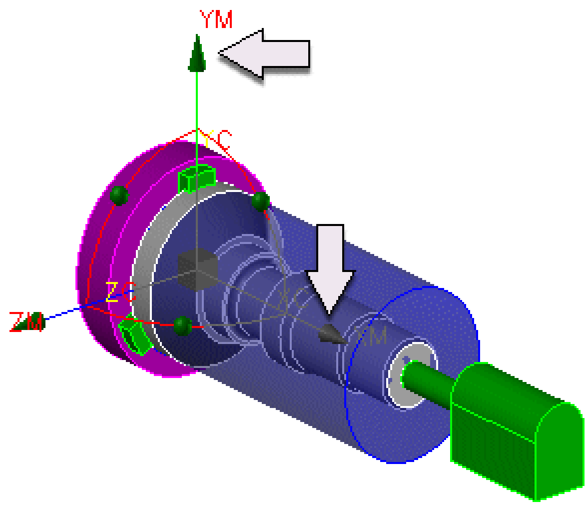
☐ Double-click on the **MCS_SPINDLE** Parent Group in the Operation Navigator.



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.



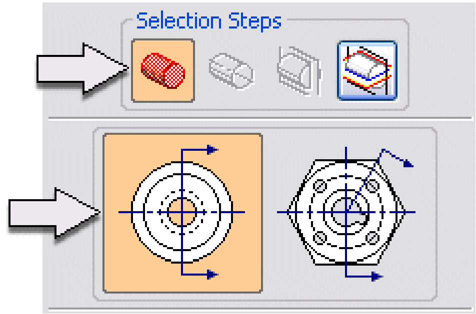
☐ Choose **Cancel** to dismiss the dialog.

Step 3: Cross section the blank.

To create cross sectional curves, you must first select a solid body and then define a section plane.

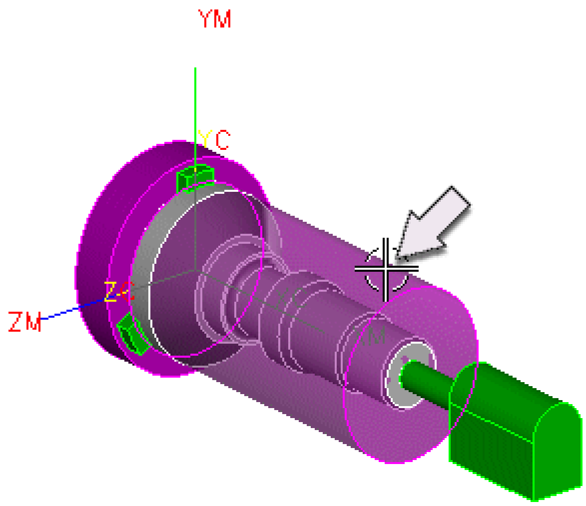
☐ Choose **Tools→Lathe Cross-Section**.

The Lathe Cross Section dialog is displayed. The Body and Simple Section icons are chosen.



Before creating cross sections, note that the work layer is layer 50. This layer currently contains no geometry. Creating each cross section on a separate layer will allow you to easily display the curves.

☐ Select the blank geometry.



☐ Choose **OK**.

The plane symbol is displayed at MCS XY-Plane.

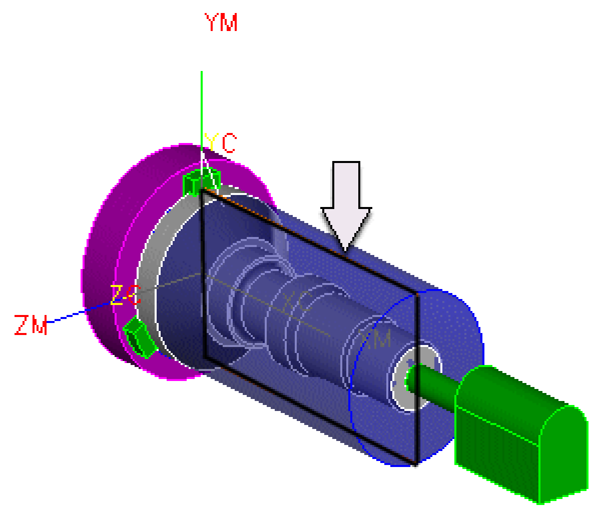
Step 4: Specify the section plane.

The Section Plane icon is automatically selected.

You are going to use the default Section, MCS_SPINDLE.

☐ Choose **OK**.

You now have associative curves that can be used for blank boundary creation.



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

Step 5: Change the work layer.

Creating each cross section on a separate layer will allow you to easily display the curves.

☐ Choose **Format→Layer Settings** from the menu bar.

☐ Key in **55** in the Work field and choose 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.

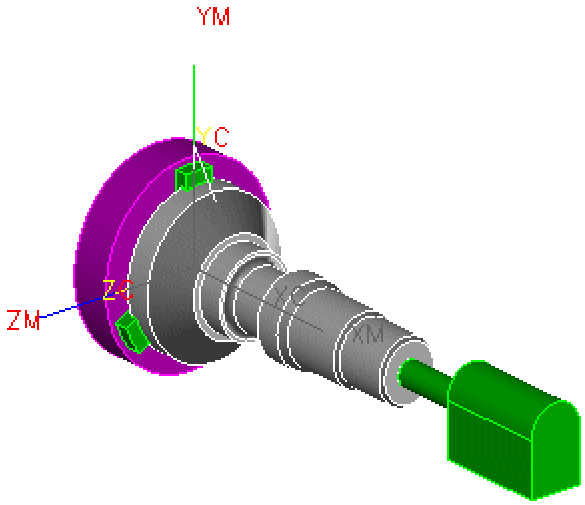
☐ Double-click on Layer **50**.

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

☐ Double-click on layer **53**.

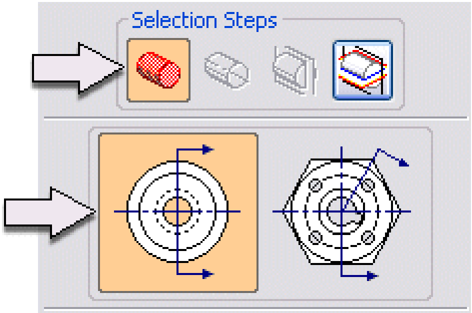
☐ Choose **OK**.

Layer 55 is now the work layer and layers 50 and 53 are invisible. You should be able to see the part geometry clearly.



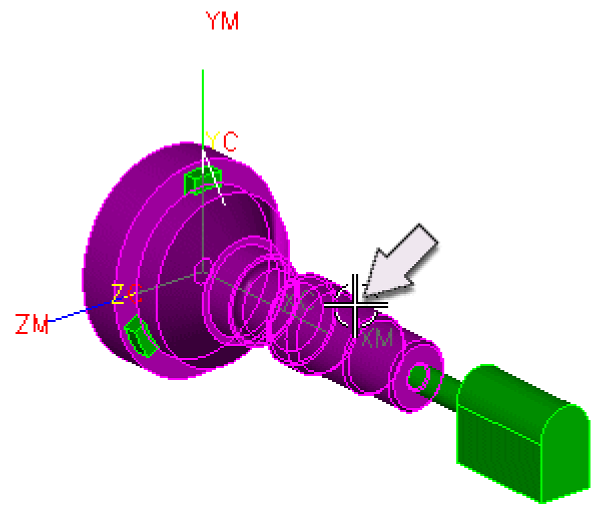
Step 6: Cross Section the part.

You will now cross section the part geometry. The Body and Simple Section icons are chosen.



C

☐ Select the part geometry.

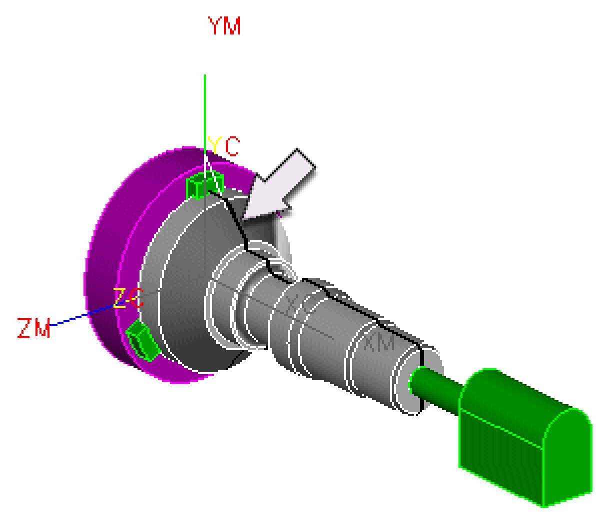


☐ Choose **OK**.

Again, you are going to use the default Section, MCS_SPINDLE.

☐ Choose **OK**.

You now have associative curves that can be used for part boundary creation.



Step 7: Change the layer settings.

You are going to change the layer settings to make the curves clearly visible. This will allow you to easily select the cross sectional curves when you create the boundaries.

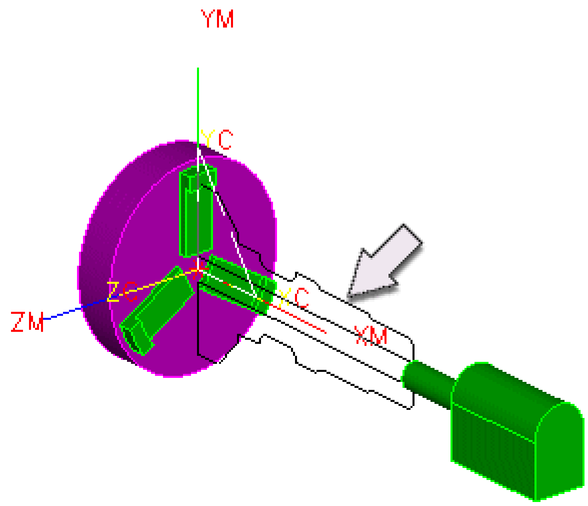
☐ Choose **Format→Layer Settings** from the menu bar.

The Layer Settings dialog is displayed.

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

☐ Choose **OK**.

You should be able to clearly see the part cross sectional curves.

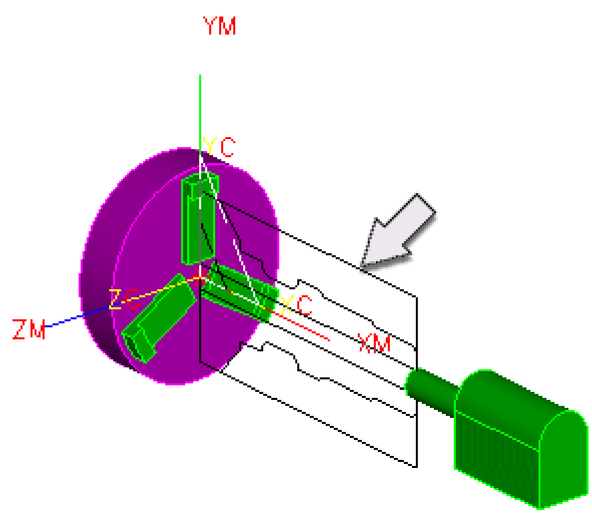


Next you will make layer 50 selectable.

☐ Choose **Format→Layer →Settings** from the menu bar.

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

You should be able to clearly see the blank cross sectional curves.



☐ Choose **File→Save** from the menu bar.

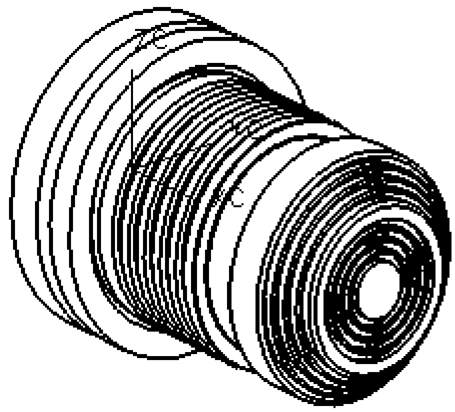
☐ Close the part.

C

Activity: Create a Simple Cross Section Unassisted

Step 1: Open and rename the part.

- ☐ Open the part file **tmp_crs_sec_mfg_2.prt**.
- ☐ Save the part file as *****_crs_sec_mfg_2.prt**, where *** represents your initials.



Step 2: Create the cross section geometry.

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

  If you have any questions, ask your instructor.

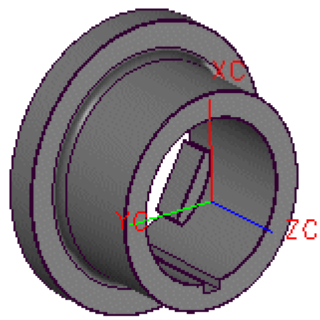
- ☐ **Save** the part file.
- ☐ Save and close the part.

Activity: Create a Complex Cross Section

In this activity, you are going to create complex cross sections for the part geometry. These cross sections will define the hole diameter and the minimum cross section of the part.

Step 1: Open and rename the part.

- ☐ Open the part file **tmp_crs_sec_mfg_3.prt**.



- ☐ Save the part file as *****_crs_sec_mfg_3.prt**, where *** represents your initials.

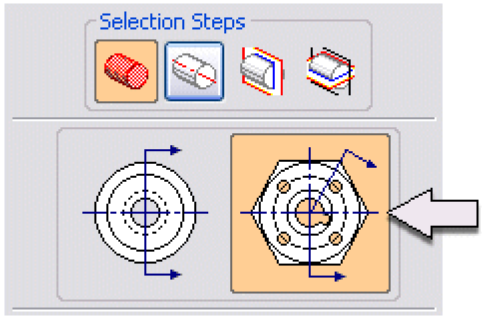
- ☐ Choose **Application→Manufacturing**.

Step 2: Create the cross section geometry that represents the smallest ID of the part.

- ☐ Make layer **50** the work layer.

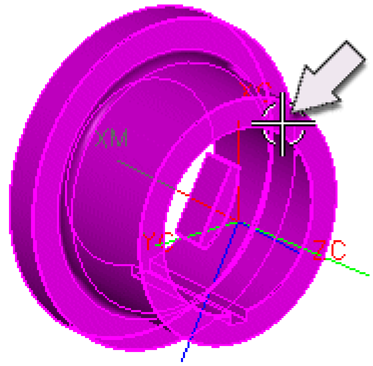
- ☐ Choose **Tools→Lathe Cross-Section**.

- ☐ Choose the **Complex Section** icon.



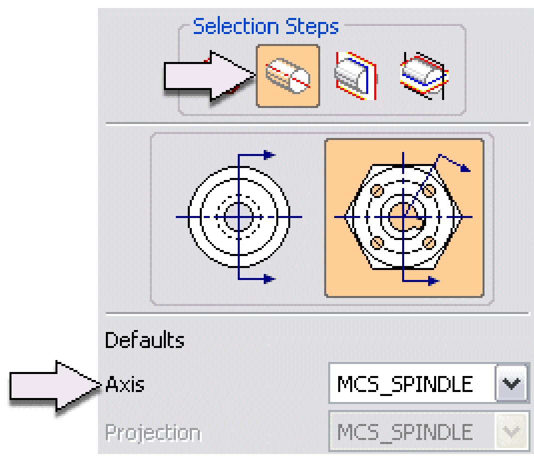
Notice under the Selection Steps label, the Body icon is highlighted.

☐ Select the part body.



☐ Choose **OK**.

Under the Definition Steps label, notice that the Axis icon has been selected. Also notice that the Axis label is highlighted.



You are prompted to specify the axis of rotation for the cross section curves. The default MCS_SPINDLE is correct for this part.

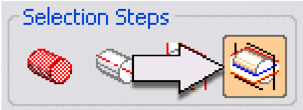
☐ Choose **OK**.

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.

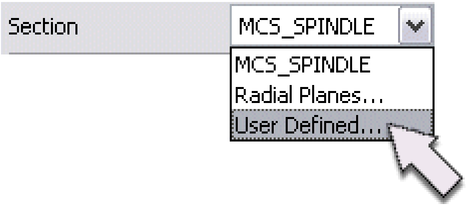
☐ Choose **OK**.

Note that under the Definition Steps label, the Section Plane icon is highlighted. You are prompted to specify the plane that will be used to create the cross section curves.



Also note that the Section label is highlighted.

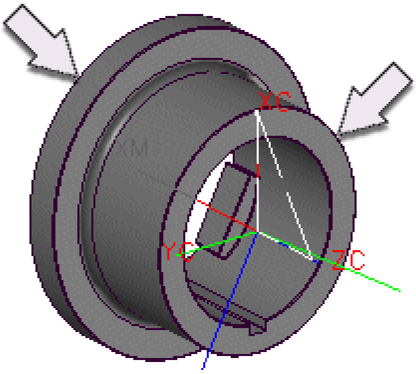
☐ Change the Section to **User Defined**.



☐ Choose the **Three Points** icon.

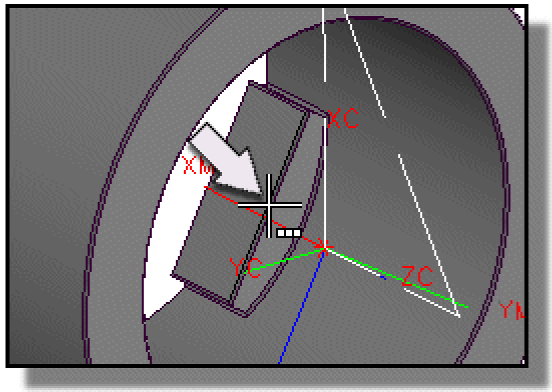
☐ Choose the **Arc/Ellipse/Sphere Center** icon.

☐ Select the two arcs as shown in the following illustration.



☐ Choose the **Control Point** icon.

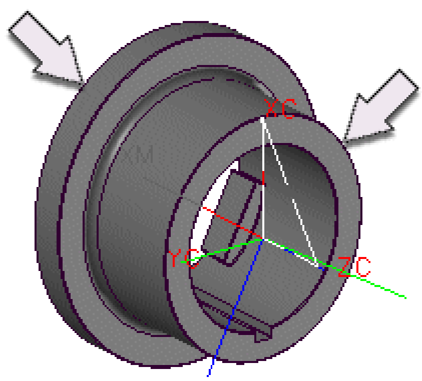
- ☐ Choose the midpoint of the edge of the pad. This point defines the smallest ID of the part.



- ☐ Choose **Yes** when you are prompted to define more additional cut planes.

Step 3: Create the cross section geometry that represents the hole diameter.

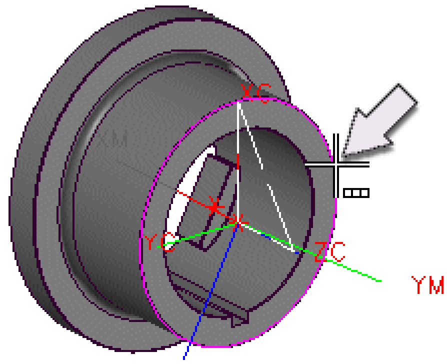
- ☐ Choose the **Three Points** icon.
- ☐ Choose the **Arc/Ellipse/Sphere Center** icon.
- ☐ Select the two arcs as shown in the following illustration.



- ☐ Choose the **Quadrant Point** icon.

C

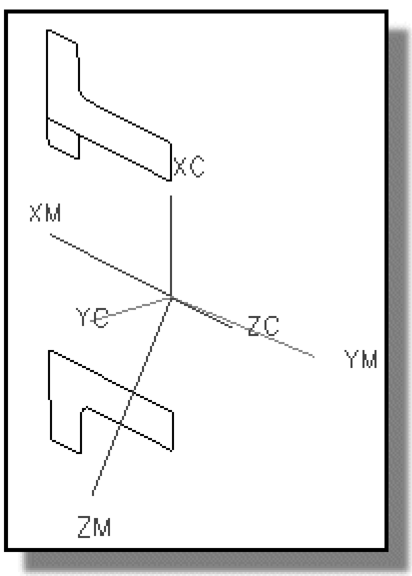
☐ Select the quadrant point of the arc at the approximate location illustrated below.



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

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

☐ Make the solid body (layer 1) invisible so you can see the cross section curves clearly.



☐ Redisplay the part.

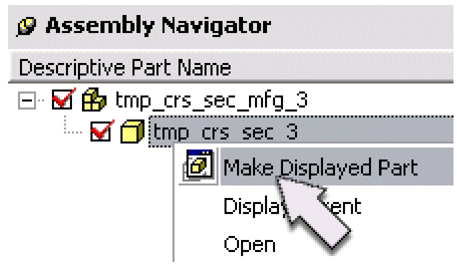
☐ **Cancel** the Lathe Cross Section dialog.

Next, you will edit the part geometry and observe how the associated cross sectional curves update automatically.

C

Step 4: Using expressions, change the height of the pad and view the updated cross sectional curves.

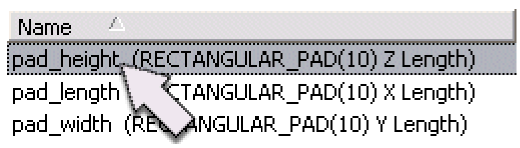
☐ In the Assembly Navigator, choose **tmp_crs_sect_3** and **MB3→Make Displayed Part**.



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

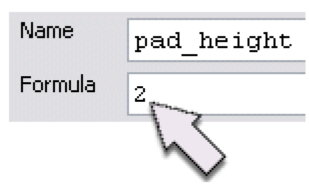
The Expressions dialog is displayed. Note the list of part expressions.

☐ Choose the **pad_height** expression.



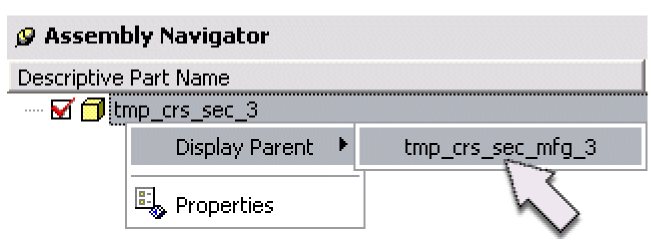
You will change the pad height from 0.500 to 2.000.

☐ Key in **2** in the Formula field.



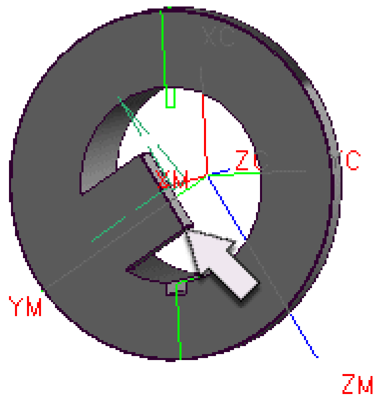
☐ **OK** the Expressions dialog to accept the edit.

☐ In the Assembly Navigator, choose **tmp_crs_sect_3** and **MB3→Display Parent→tmp_crs_sec_mfg_3**.

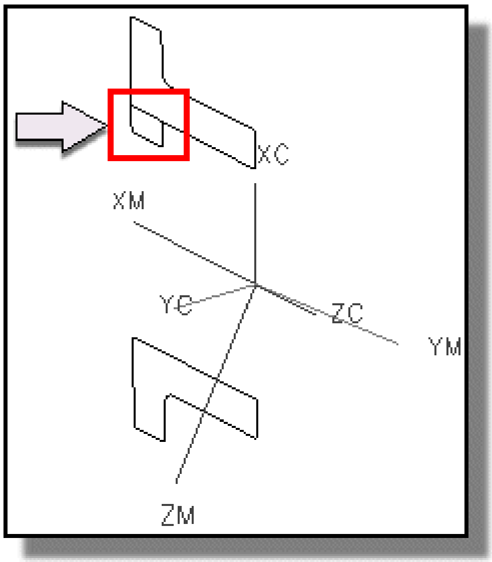


C

☐ Rotate the part to see how the pad height has changed.



☐ Make the solid body (layer 1) invisible so you can see to cross section curves clearly. Although the part geometry has changed, the cross section curves have not yet updated

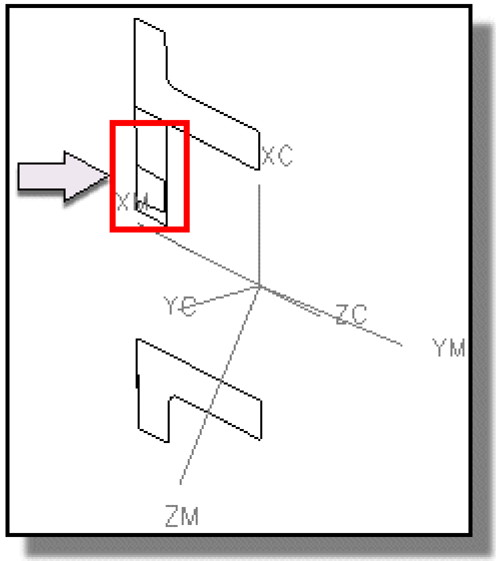


You need to redisplay the Lathe Cross Section dialog and then Cancel to update the curves.

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

C

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



☐ Redisplay the part so you can see both the part and the curves.

☐ **Save** the part.

☐ **Close** all parts.

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.

C

Summary

Associative cross sectional curves required for boundary creation can be created very easily using the Lathe Cross Sections function. The following functions are used to create cross section curves:

- Tools→Lathe Cross-Section→Simple Section
- Tools→Lathe Cross-Section→Complex Section

C

Index

This page left blank intentionally.

LEARNING ADVANTAGE

UGS Education Services offers a blend of training solutions for all of our product lifecycle management products.
Our Online Store “Learning Advantage” was developed to provide our customers with “just in time’ training for the latest in application developments.

Here are some of the Learning Advantages:

- Customers have direct access
- Self-paced course layout
- Online Assessments
- Just in time training for the latest release

To learn more about the “Learning Advantage” visit our website http://training.ugs.com or email us at training @ugs.com

Training solutions for all of our PLM software: NX TEAMCENTER TECNOMATIX

training@ugs.com http://training.ugs.com

STUDENT PROFILE

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

Please **"Print"…**

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

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

Hotel/motel you are staying at during your training ____

Planned departure time on last day of class ____

Employer _______________________________________ **Location** ________________________

Your title and job responsibilities ____________________/__________________________________

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

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

Reason for training ____

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

Software	From Whom	When	Course Name	Medium

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

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

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

Platform (operating system) ____

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

TMP Course Agenda

Day 1 **Morning**

- Introduction & Overview
- Lesson 1. Defining Part & Blank Geometry
- Lesson 2. Retrieving & Creating Tools
- Workbook Sections 1 -- 7

Afternoon

- Lesson 3. Facing Operations
- Workbook Section 8
- Lesson 4. Verification

Day 2 **Morning**

- Lesson 5. Common Options
- Lesson 6. Centerline Operations
- Workbook Sections 9 & 10
- Lesson 7. Roughing Operations – OD Work
- Workbook Section 11

Afternoon

- Lesson 8. Roughing Operations – ID Work
- Lesson 9. Finish Operations OD and ID Work

Day 3 **Morning**

- Lesson 10. Grooving
- Workbook Sections 12 - 17
- Lesson 11. Teach Mode
- Lesson 12. Threading Operations
- Workbook Sections 18 - 20

Afternoon

- Lesson 13. Using Multiple Spindles
- Lesson 14 Mill Turn

Accelerators

The following Accelerators can be listed from within an NX session by choosing Information→Custom Menubar→Accelerators.

Function	***Accelerator***
File→New...	Ctrl+N
File→Open...	Ctrl+O
File→Save	Ctrl+S
File→Save As...	Ctrl+Shift+A
File→Plot...	Ctrl+P
File→Execute→Grip...	Ctrl+G
File→Execute→Debug Grip...	Ctrl+Shift+G
File→Execute→NX Open...	Ctrl+U
Edit→Undo	Ctrl+Z
Edit→Cut	Ctrl+X
Edit→Copy	Ctrl+C
Edit-Paste	Ctrl+V
Edit→Delete...	Ctrl+D or Delete
Edit→Selection→Top Selection Priority - Feature	F
Edit→Selection→Top Selection Priority - Face	G
Edit→Selection→Top Selection Priority - Body	B
Edit→Selection→Top Selection Priority - Edge	E
Edit→Selection→Top Selection Priority - Component	C
Edit→Selection-Select All	Ctrl+A
Edit→Blank→Blank...	Ctrl+B
Edit→Blank→Reverse Blank All	Ctrl+Shift+B
Edit→Blank→Unblank Selected...	Ctrl+Shift+K
Edit→Blank→Unblank All of Part	Ctrl+Shift+U
Edit→Transform...	Ctrl+T
Edit→Object Display...	Ctrl+J
View→Operation→Zoom...	Ctrl+Shift+Z
View→Operation→Rotate...	Ctrl+R
View→Operation→Section...	Ctrl+H
View→Layout→New...	Ctrl+Shift+N
View→Layout→Open...	Ctrl+Shift+O
View→Layout→Fit All Views	Ctrl+Shift+F
View→Visualization→High Quality Image...	Ctrl+Shift+H
View→Information Window	F4
View→Current Dialog	F3
View→Reset Orientation	Ctrl+F8
Insert→Sketch...	S
Insert→Design Feature→Extrude...	X
Insert→Design Feature→Revolve...	R
Insert→Trim→Trimmed Sheet...	T

Evaluation – Delivery
NX 4 TMP, Course # MT11055
Dates _____________ thru______________

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

	STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
Instructor: ______________________ ☑						
If there were 2 instructors, please evaluate the 2nd instructor with "X's"						
Instructor: ______________________ ☒						
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

This page left blank intentionally.