Multi-Axis Machining

Student Guide
May 2005
MT11050 — NX 3

Manual History

Proprietary and Restricted Rights Notices

The following proprietary and restricted rights notices apply.

Copyright

Restricted Rights Legend

Manual Revision	Unigraphics Version	Publication Date
	Version 16	August 2000
	Version 17.1.1	February 2001
	Version 18.0	November 2001
	Unigraphics NX	November 2002
	Unigraphics NX 2	January 2004
	NX 3	May 2005

This edition obsoletes all previous editions.

Contents

Course Overview

Course Description

The Multi-Axis Machining course teaches the use of the Manufacturing Application for creating 4 and 5-axis milling tool paths. You will learn about the Variable Contour and Sequential Mill operation types that are designed for multi-axis machining. You will also learn about the tool axes that are available within Variable Contour and Sequential Mill operations.

Intended Audience

This course is intended for Manufacturing Engineers, NC/CNC programmers and anyone with the desire to learn how to create four and five axis tool paths.

Prerequisites

The required prerequisites for the course are Practical Applications of NX or the CAST equivalent, and Mill Manufacturing Process. Any additional experience in creating multi-axis tool paths is an asset in taking this course.

Objectives

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

- choose between Variable Contour and Sequential Mill operation types
- choose the best type of tool axis for creating various multi-axis tool paths
- develop multi-axis machining practices
- develop NX multi-axis programming practices

Student Responsibilities

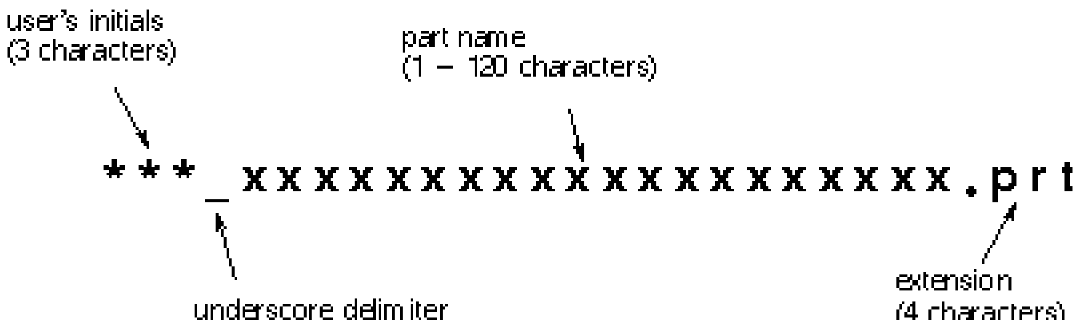
- Be on time.
- Participate in class.
- Focus on the subject matter.
- Listen attentively and take notes.
- Enjoy the class.

Class Standard for NX Parts

The following standards are 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 Naming

This class uses the following file naming standard:



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

The Arrow Symbol

The arrow symbol (→), represents that you choose an option, then immediately choose another option. 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

There are standard layer assignments and category names in each of the part files. They are 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	Color Used
Solid Bodies	Green
Sheet Bodies	Yellow
Lines and Arc (non-sketch curves)	Green
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

How to Use This Manual

It is important that you use the Student Guide in the sequence presented since 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.

☐ Choose **Edge Lengths, Corner** for the creation method.

The general format for lesson content is:

- Presentation
- Activity
- 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 exit NX and log off the terminal.

Workbook Overview

The workbook contains a project that requires you to apply the knowledge that you learned in the class and in the Student Activities. The projects do not contain detailed instructions as do the Student Activities.

The intent of the projects is to allow you to apply the skills taught in this course. At any point when you are not making progress, ask your instructor for help.

Classroom System Information

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

Student Login:

User name:

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 directories located in the Parts directory, the Students_parts and workbook.

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

The workbook 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 with these files.

Lesson

1 *Introduction to Four and Five Axis Machining*

Purpose

This lesson introduces the application of machining parts utilizing 4 and 5 axis machining principles.

Objective

At the conclusion of this lesson, you will be able to:

- Create tool paths for 4-axis positioning and contouring operations.
- Properly place the MCS for multi-axis operations.

Multi-Axis Machining Concepts

The majority of what NC/CNC programmers term as "multi-axis" can actually be considered planar or fixed axis machining. The spindle axis, on some machines, is not normal to the Z direction of the machine tool and the actual machining does not force a change in any motion of the rotary axis. This case considers using the rotary axis for *positioning* mode only.

Programming of this type of operation is relatively simple, once you understand some of the more basic concepts of multi-axis machining. Some concepts for considerations are:

- NX always requires a tool axis; if one is not specified, the default tool axis is equal to the Z of the MCS (sometimes referred to by the vector of 0,0,1)

- Fixed-Axis machining with a tool axis other than (0,0,1) involves setting the tool axis to the proper orientation

- most, if not all, NX multi-axis operations work with a tool axis other than 0,0,1

- when performing multi-axis machining, never assume the tool axis is currently correct; always make sure you specify the proper tool axis if it is not 0,0,1

- prior to rotation of the table to a new position, verify the tool has been retracted far enough to clear the part/fixture during rotational moves

- it is a recommended practice to return the tool axis back to (0,0,1) at the end of the operation

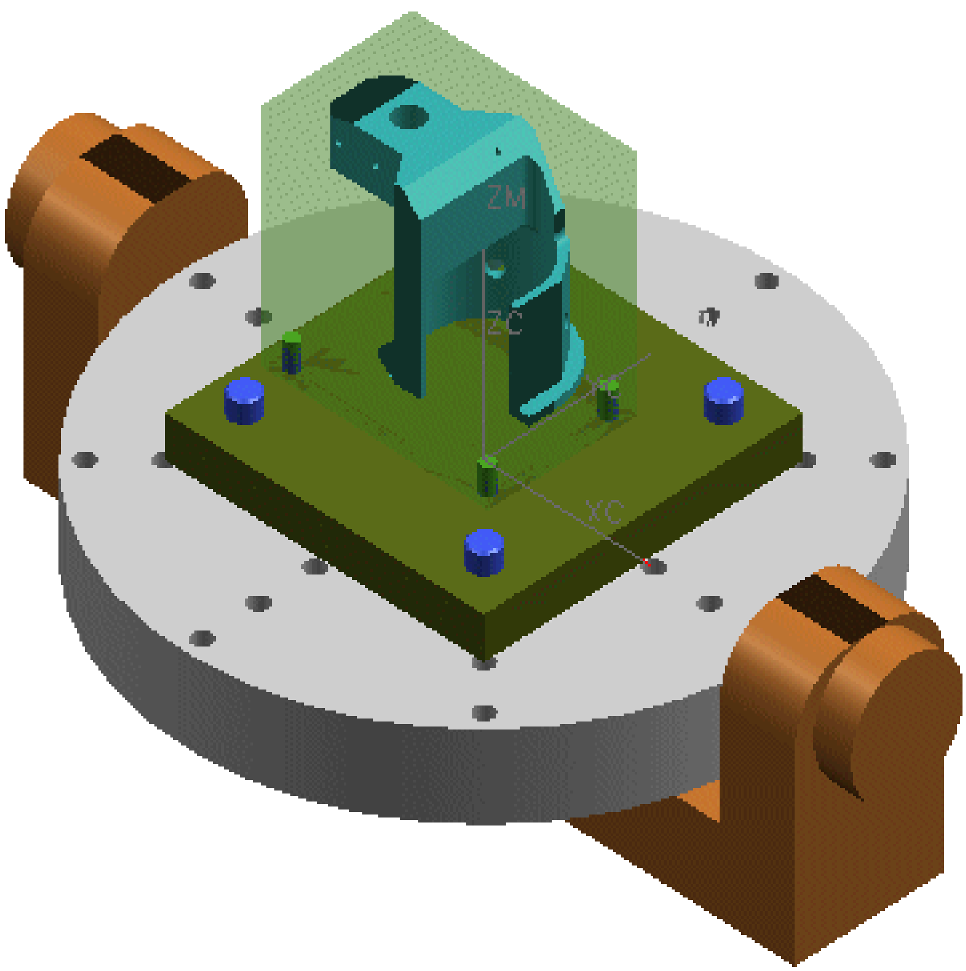
The following activity requires you to generate a tool path at other than a normal tool axis of (0,0,1).

Activity: Operations at other than 0,0,1 Tool Axis

In this activity, you will machine the top and two angled areas of a sleeve collar used in a yoke mechanism. All necessary Parent objects have been created and the part has been previously roughed. The operations which you will create will finish mill the top and two angled faces of the part.

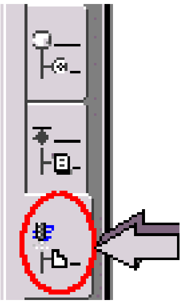
Step 1: Open an existing part file and enter the Manufacturing Application.

☐ Open the part file, **mam_collar_mfg**.



☐ Choose **Application →Manufacturing.**

☐ Choose the Operation Navigator tab from the resource bar.



The Operation Navigator and the part are displayed.

Step 2: Create the finishing operation.

The operation, **ROUGHING**, already exists to rough the pad at the top of the part. You will now create the operation to finish that particular pad.

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

☐ If necessary, set the **Type** to **mill_planar.**

☐ Choose **FACE_MILLING** as the operation type.

☐ Choose the following **Parent** objects:

Program: FIXED_AXIS

Geometry: NORMAL_FACE

Tool: EM-1.00-0

Method: MILL_FINISH

Note that the geometry parent contains a boundary that describes the top face of the part. The floor plane is set to the top face.

Also note that the tool used in this operation is a 1.00" diameter end mill with 0" corner radius.

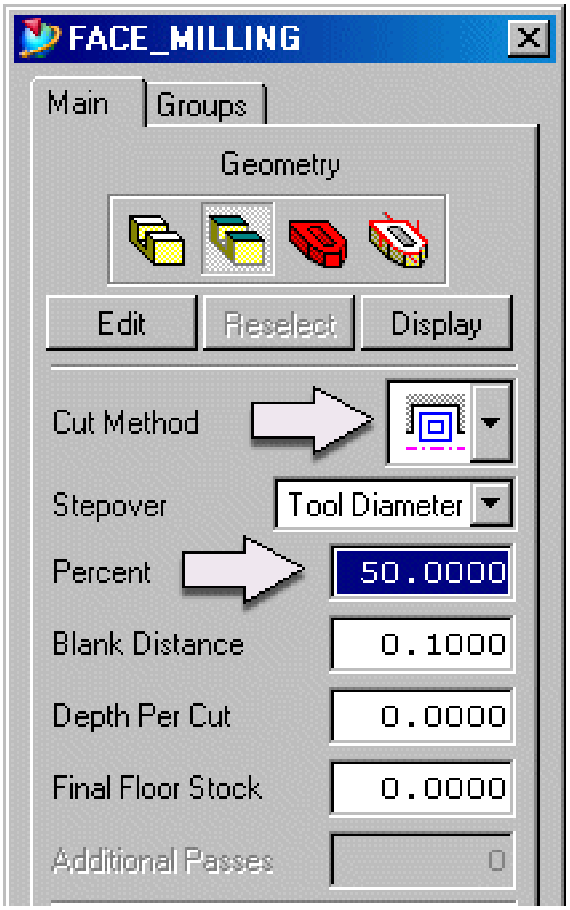
Since this operation is used for finishing, no machining stock will be left by the Method parent object.

☐ Key in **top_face** as the name of the operation.

☐ Choose **OK**.

The **FACE_MILLING** dialog is displayed.

☐ Change the **Cut Method** to **Follow Periphery** and the **Stepover Percent** to **50**.



☐ **Generate** the operation and then choose **OK** from the **Display Parameters** dialog.

The generated tool path is displayed.

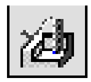
☐ Choose **OK** to accept the operation.

Step 3: Verify the results.

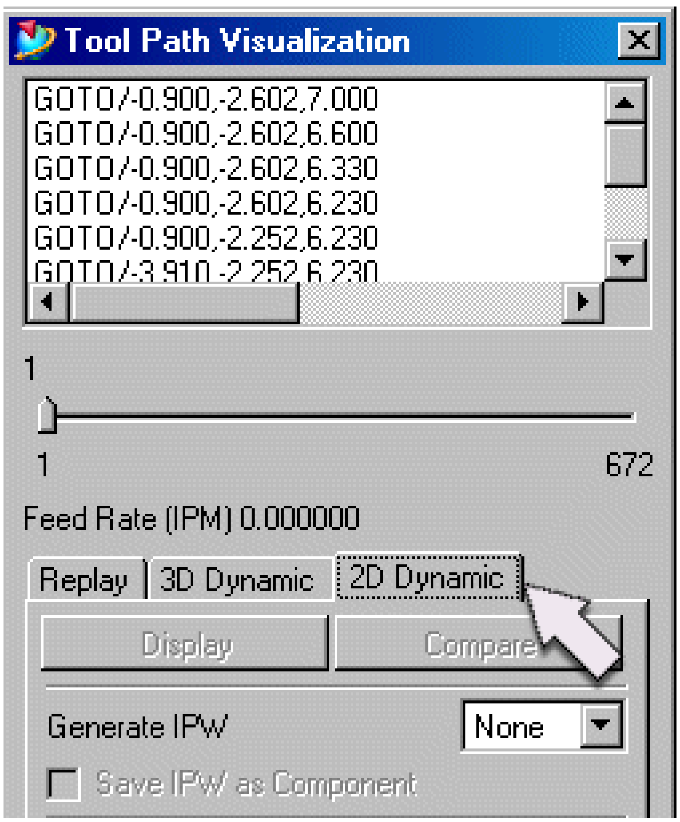
You will now verify the results by using **Tool path Visualization**.

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

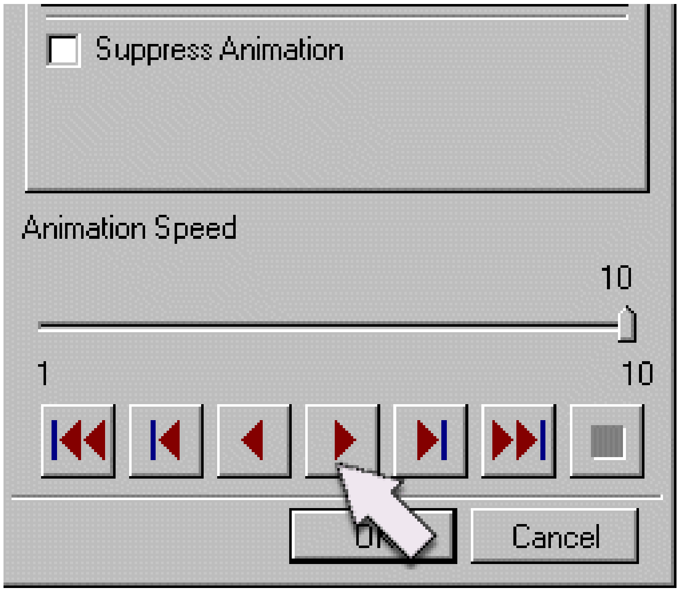
☐ Highlight the **FIXED_AXIS** program object.

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

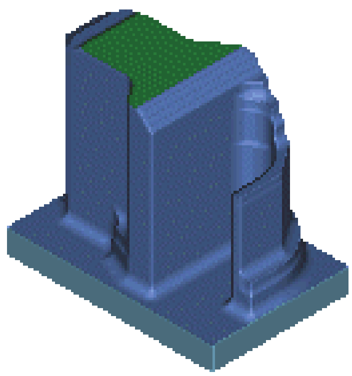
☐ Choose the **2D Dynamic** tab from the **Tool Path Visualization** dialog.



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



Two operations will be replayed. The first operation is used for roughing, the second is the finish operation that you just created.



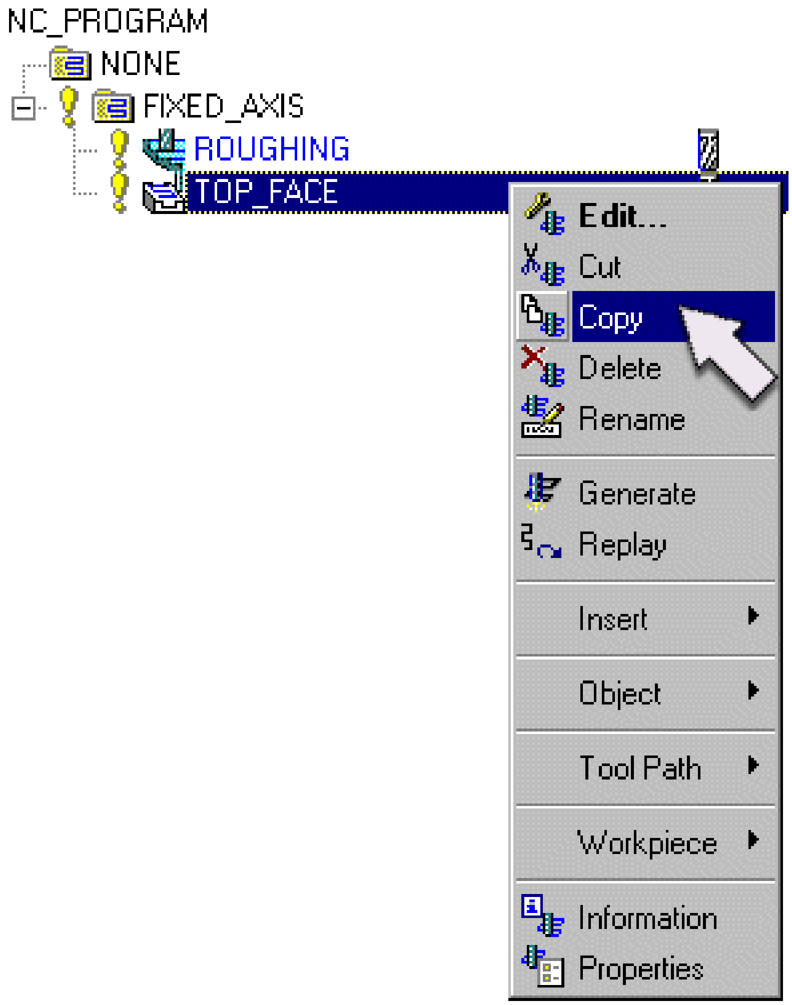
Verifying the operation indicates the tool path to be acceptable, you will now continue with the next operation.

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

Step 4: Create the first angled-face operation.

You will copy and rename the existing operation, **TOP_FACE**, to use as a template for creating the next operation.

☐ Highlight the **TOP_FACE** operation and choose **MB3** → **Copy**.



☐ Choose **MB3** → **Paste**.

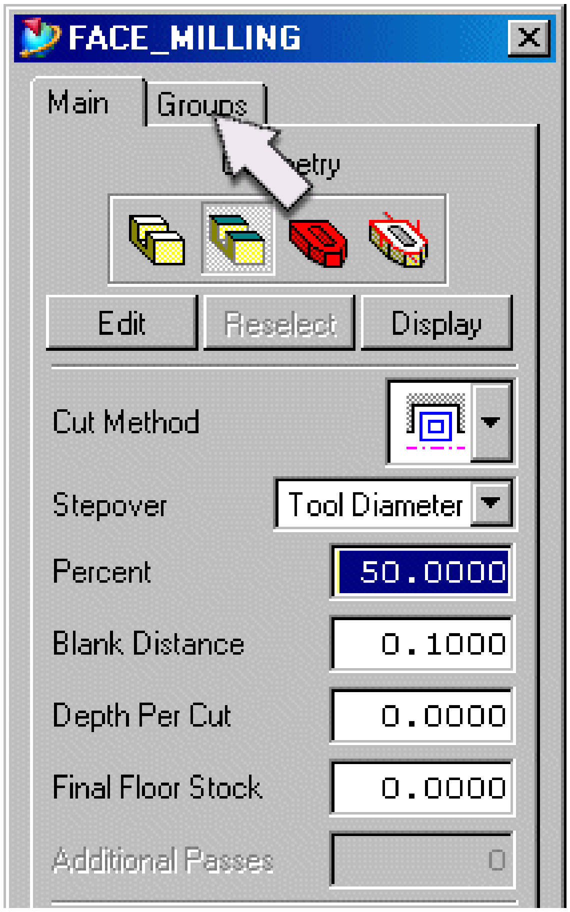
A copy of the previous operation is created, with the name **TOP_FACE_COPY**. You will now rename the operation to **ANGLE_FACE_1**.

☐ Change the name of the new operation by highlighting the **TOP_FACE_COPY** operation, using **MB3,** choose **Rename**, and then type the name **ANGLE_FACE_1**.

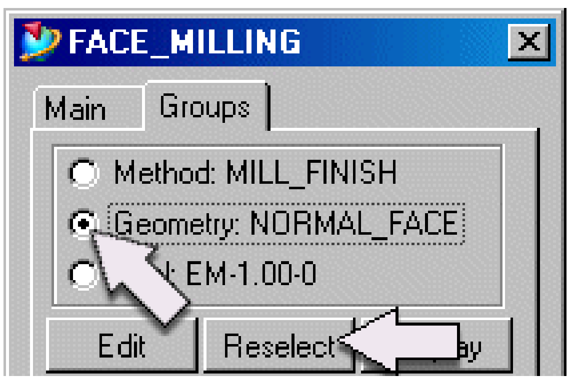
You will now change the geometry parent object.

☐ Double-click on the **ANGLE_FACE_1** operation.

☐ Choose the **Groups** property page.

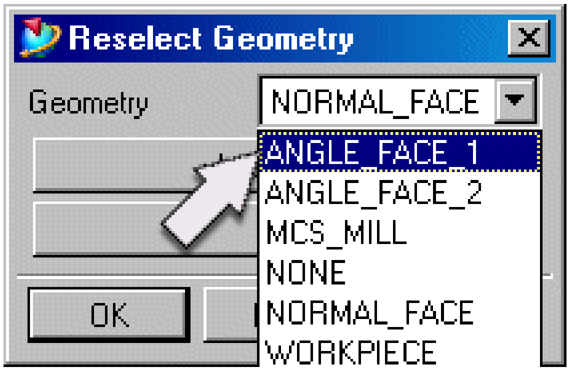


☐ Choose the **Geometry** radio button at the top of the dialog, then choose **Reselect**.



The **Reselect Geometry** dialog is displayed.

☐ Choose **ANGLE_FACE_1** from the pull-down list.



☐ Choose **OK**.

☐ Choose the **Main** property page from the **FACE_MILLING** dialog.

☐ Choose **Generate**.

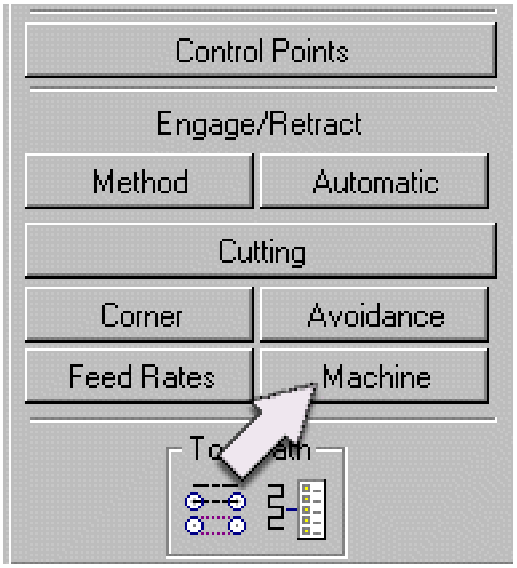
☐ Choose **OK** on the **Display Parameters** dialog.

The **Operation Parameter Error** dialog is displayed.

This dialog is informing you that the operation type, **FACE_MILLING**, will not work unless the tool axis is set normal to the floor axis. You will now redefine the tool axis normal to the floor.

☐ Choose **OK** from to the Operation Parameter Error dialog.

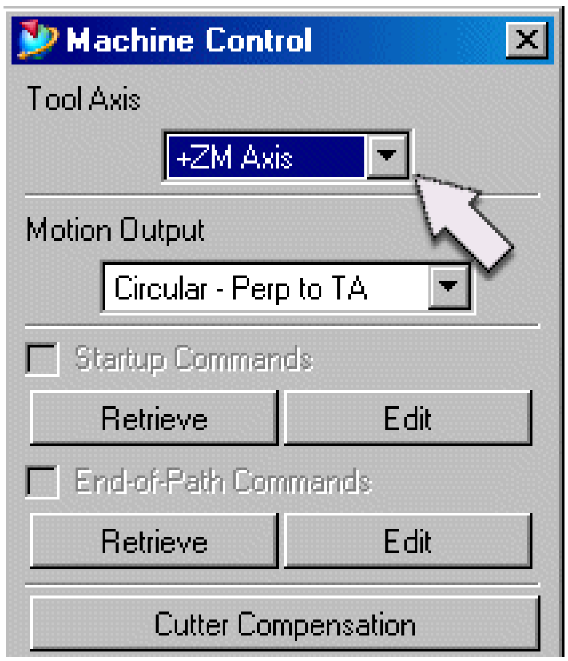
☐ Choose the **Machine** button located on the **FACE_MILLING** dialog.



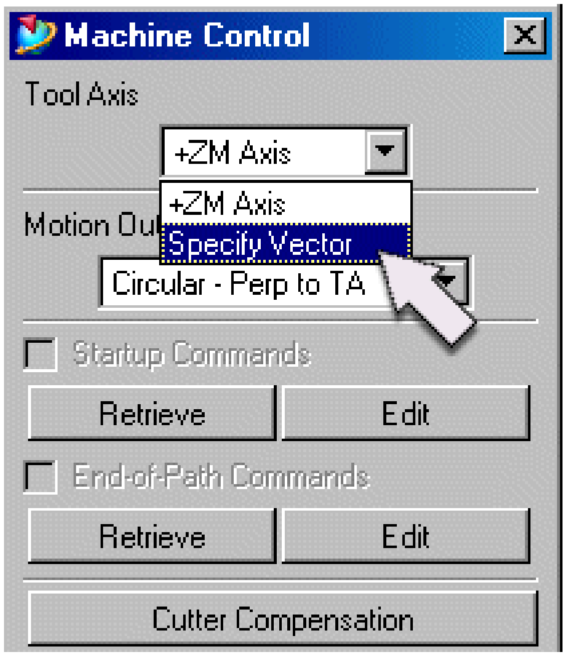
The **Machine Control** dialog is displayed.

As described earlier, there always is a defined tool axis. In this particular case, the tool axis is the same as the Z of the MCS (the definition of "+ZM Axis"). You will now change the tool axis to one that is normal to the floor plane of the **ANGLE_FACE_1** geometry parent object.

☐ Select the **Tool Axis** pull-down arrow.

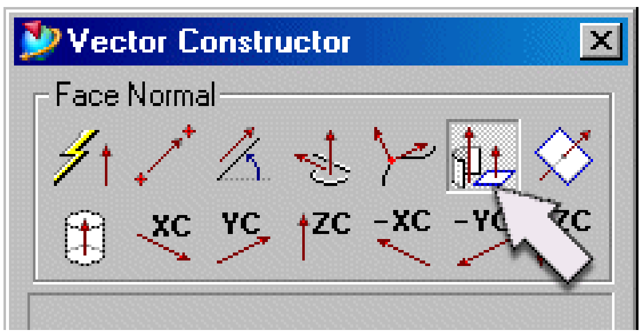


- ☐ Choose **Specify Vector** from the list.



The **Vector Constructor** dialog is displayed.

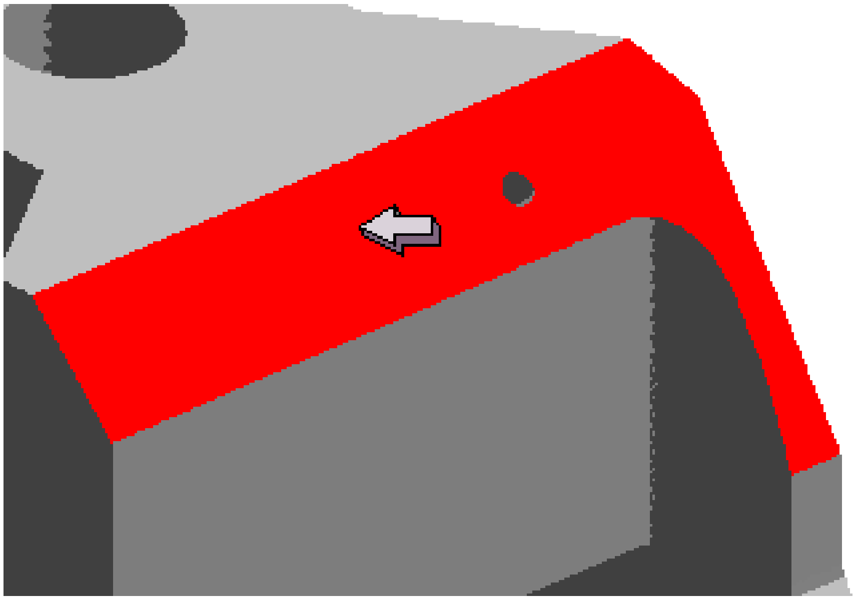
- ☐ From the **Vector Constructor** dialog, choose the **Face Normal** icon.



Note that **Face Normal** means to set the vector perpendicular to the face.

- ☐ Choose the angled face as shown in the following figure.

  Hint: You may need to blank the stock component to choose the proper face, by using the Assembly Navigator.



- ☐ Choose **OK** until the **FACE_MILLING** dialog is displayed.
- ☐ **Generate** the operation.

Step 5: Verify the results.

- ☐ Use **Verification** to verify your tool path (refer to Step 3 for detailed instructions).

Step 6: Create the second angled face operation.

You will use the copy/paste features of the Operation Navigator to create the third finish operation.

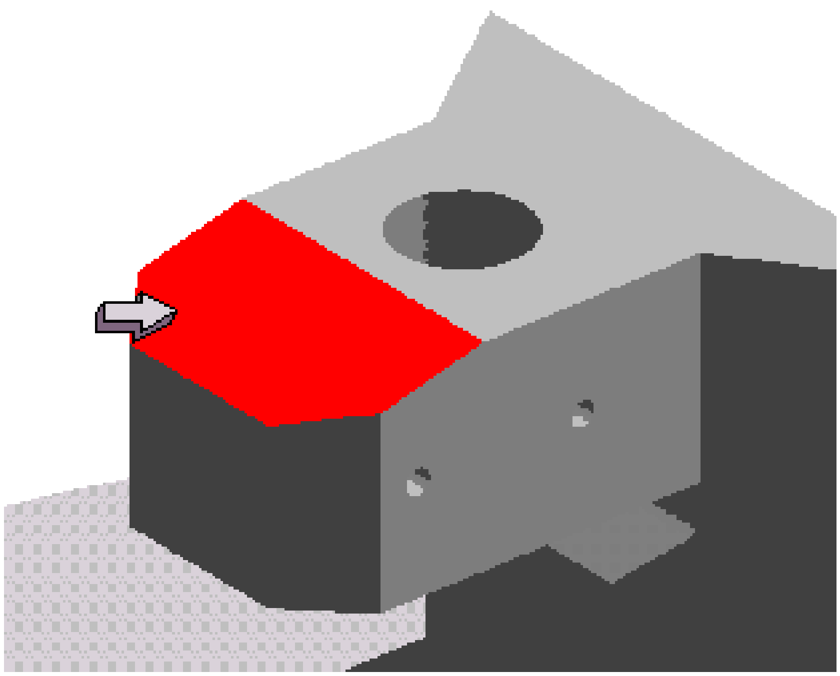
- ☐ Highlight the **ANGLE_FACE_1** operation.
- ☐ Choose **MB3** → **Copy.**
- ☐ Choose **MB3** → **Paste**.
- ☐ Change the name of the new operation to **ANGLE_FACE_2**.
- ☐ Edit the operation by doubling clicking on **ANGLE_FACE_2**.
- ☐ Choose the **Groups** property page
- ☐ Choose the **Geometry** radio button at the top of the dialog, then choose **Reselect**.

The **Reselect Geometry** dialog is displayed.

- ☐ Choose **ANGLE_FACE_2**, as the geometry parent, from the pull-down list.
- ☐ Choose **OK**.
- ☐ Choose the **Machine** button located on the **FACE_MILLING** dialog.

  The **Machine Control** dialog is displayed.
- ☐ Select the **Tool Axis** pull-down arrow.
- ☐ Choose **Specify Vector**.

  The **Vector Constructor** dialog is displayed.
- ☐ From the **Vector Constructor** dialog, choose the **Face Normal** icon.
- ☐ Choose the angled face as shown in the following figure.



- ☐ Choose **OK** until the **FACE_MILLING** dialog is displayed.
- ☐ Choose **Generate**.
- ☐ Choose **OK** on the **Display Parameters** dialog.

☐ Choose **OK** on the **FACE_MILLING** dialog to save the operation.

Step 7: Verify the results.

☐ Use **Verification** to verify the tool path.

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

Defining the Center of Rotation for a rotary axes

To machine about a rotary axes, the position of the rotary axes must be defined. There are two methods to accomplish this:

- place the WCS/MCS at the center of axis rotation. For a 4 or 5 axis machine tool, position the Main MCS at the center of rotation of the 4th or 5th axis.
- designate the MCS as a geometry group, consisting of both a Main and Local MCS. This is used by the NX/POST post processor as either fixture offsets or machine tool zero data.

Placing the MCS at the Center of Axis Rotation

Position the part on the fixture in a normal position. Place the MCS at the center of rotation of the fourth axis.

At the machine tool, the operator will then set the rotary table center as the zero point.

Advantages:

- simplest method to use and deploy
- considerably less work for the NC/CNC programmer

Disadvantages:

- output in created program does not match output or dimensions on part print
- adjustment of fixtures may require some type of reprogramming

Designate the MCS as a geometry group, consisting of both a Main and Local MCS.

The programmer designates the purpose of the coordinate system as either Main or Local in the geometry group. When post processing, using the local MCS, the data of the Main and Local coordinate system are used and the output will then match the print dimensions.

If the coordinate system is designated Local, then a special output parameter can be specified for the coordinate system. The options available are:

- None
- Use the Main MCS
- Fixture Offset
- CSYS rotation

The default setting is Fixture Offset. The designated option setting is then passed to the post processor, along with the Main and Local coordinate system to output the appropriate fixture offset values (G54...G59). The post processor needs to be modified for this action to occur.

Advantages:

- output in the program matches the part print
- fixture adjustments can be solved by changing the Main and Local designation

Disadvantages:

- programmer needs to understand the complexities associated with use of the Main and Local coordinate system and the options provided
- may be more confusing for machine operators
- machine tool post processor must be set up to obtain the correct output

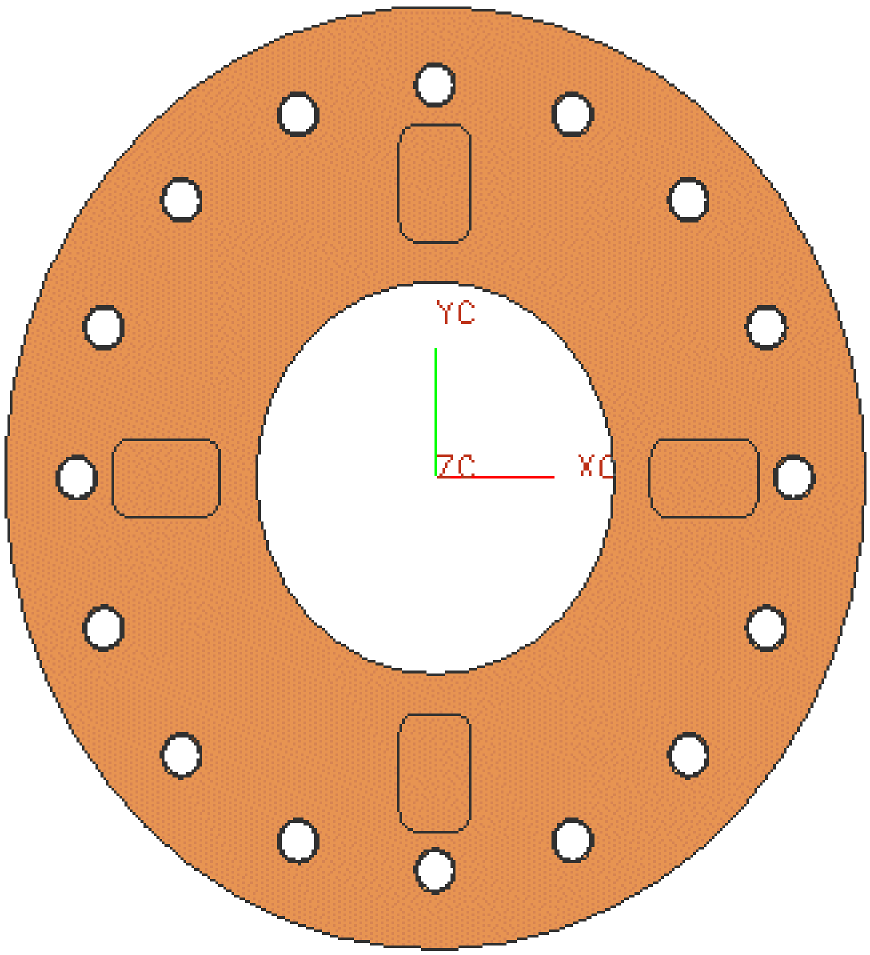
The following activity will address using a Main and Local MCS.

Activity: Main and Local MCS in Multi-Axis applications.

In this activity, you will use the Main and Local MCS, which is used by the post processor for formatting output used at the machine tool. The part file has the main and local MCS already created for you. The Main MCS is set where the machine zero would be, the same as if you were using an ORIGIN statement to govern the output. When you list the tool paths, all have the same X, Y, and Z values since they are based on the Local MCS. When you post the program, the output of the three tool paths, with their respective X, Y, and Z values, are based upon the Main MCS.

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

☐ Open the part file **mam_mcs_local_main**.



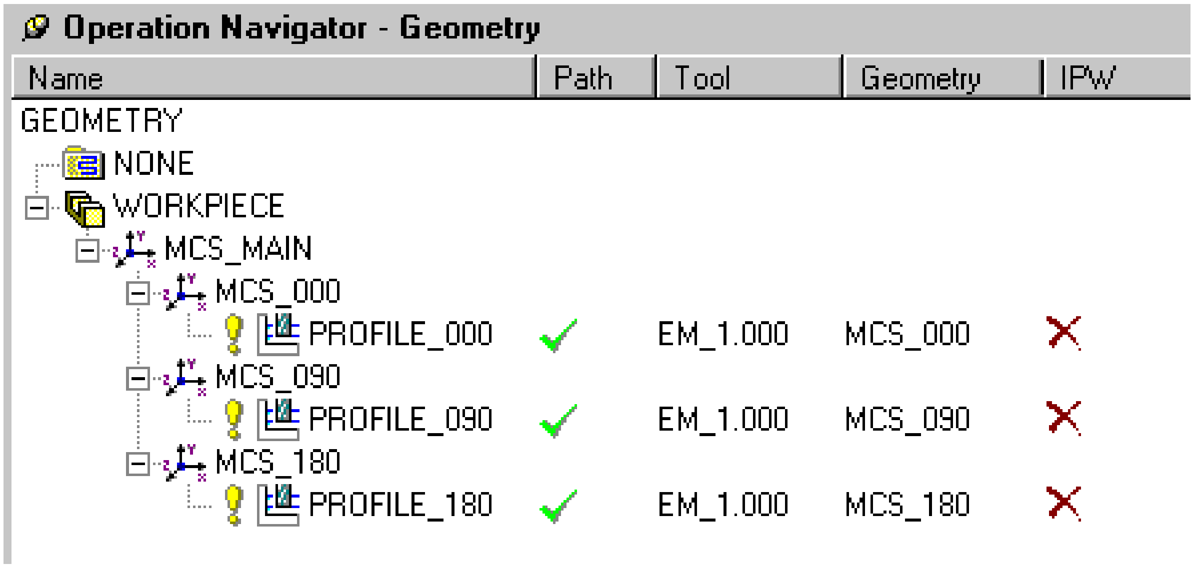
☐ Save the part as ***_mcs_local_main**.

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

Step 2: Examine the Local and Main coordinate systems.

☐ If required, change to the **Geometry** view of the Operation Navigator**.**

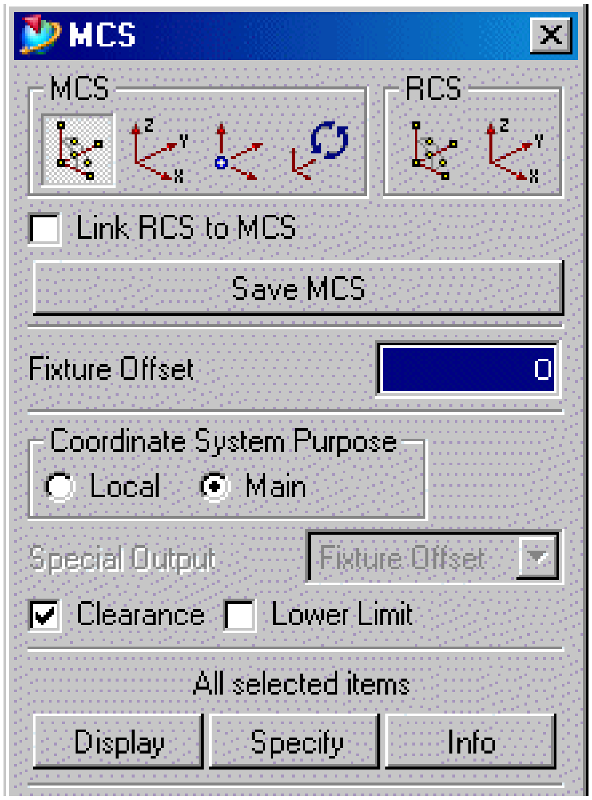
☐ Expand the **WORKPIECE** group object and all subsequent objects contained within the **WORKPIECE** parent.



You will notice that the **WORKPIECE** parent contains four different MCS coordinate systems. You will now examine each individual one.

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

The MCS dialog is displayed.

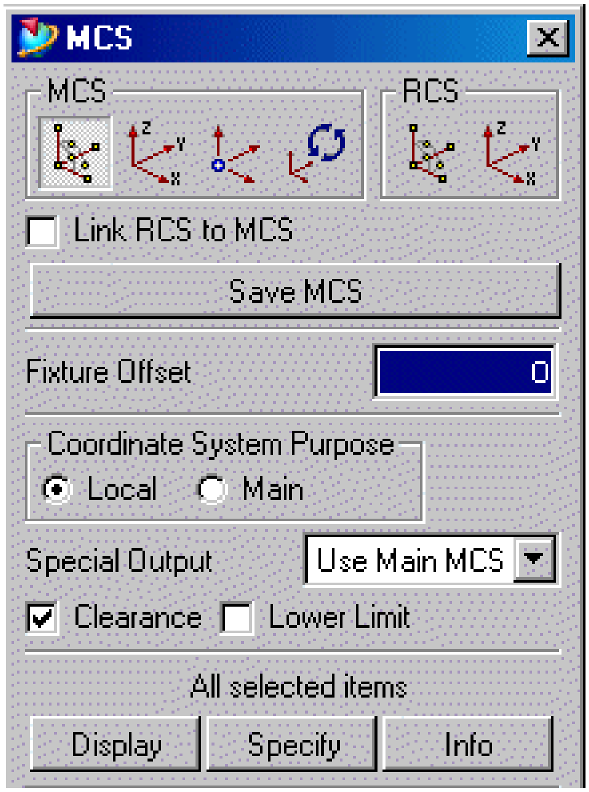


Note that the **Coordinate System Purpose** selected is **Main**.

☐ Choose **OK**.

☐ Double-click on the **MCS_000** group object.

The MCS dialog is displayed.

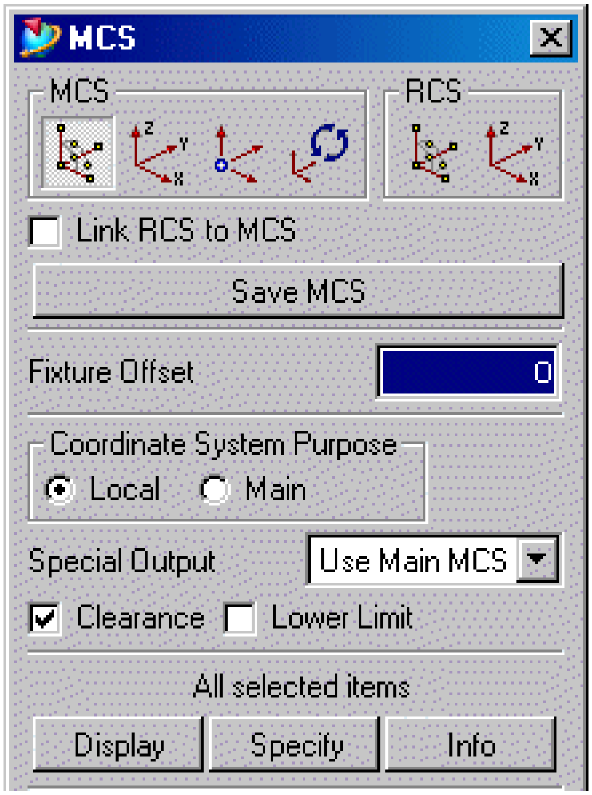


Note that the **Coordinate System Purpose** selected is **Local** and that **Special Output** is set to **Use Main MCS**.

☐ Choose **OK**.

☐ Double-click on the **MCS_090** group object.

The MCS dialog is displayed.

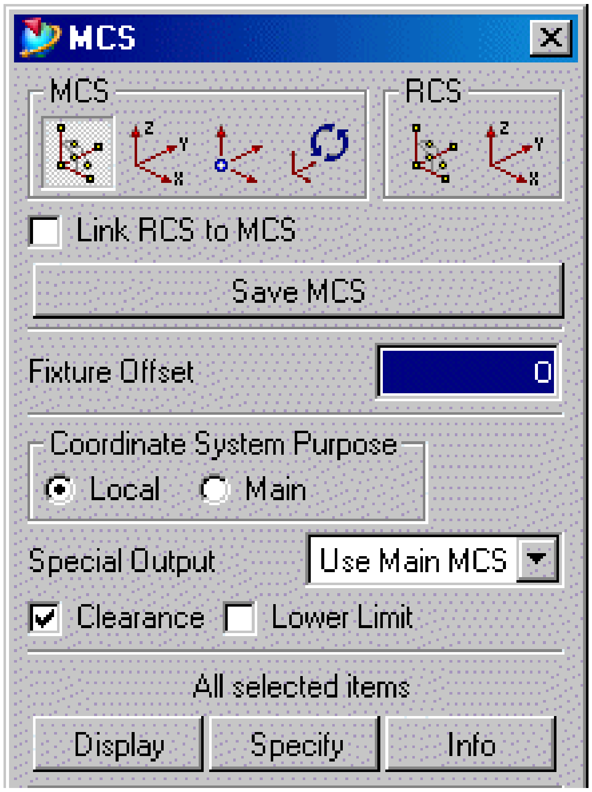


Note that the **Coordinate System Purpose** selected is **Local** and that **Special Output** is also set to **Use Main MCS**.

☐ Choose **OK**.

☐ Double-click on the **MCS_180** group object.

The MCS dialog is displayed.



Note that the **Coordinate System Purpose** selected is **Local** and that **Special Output** is set to **Use Main MCS**.

☐ Choose **OK**.

You will now list the tool paths for the existing operations that use the Local MCS and observe that the X, Y and Z values are the same for each one.

Step 3: Examine the tool path listing.

☐ Highlight the **PROFILE_000** operation, replay and list the tool path.

☐ Highlight the **PROFILE_090** operation, replay and list the tool path.

☐ Highlight the **PROFILE_180** operation, replay and list the tool path.

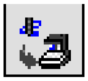
Note that all the X, Y and Z values are the same.

You will now post process the three operations and note that the X, Y and Z values are based on the MAIN MCS.

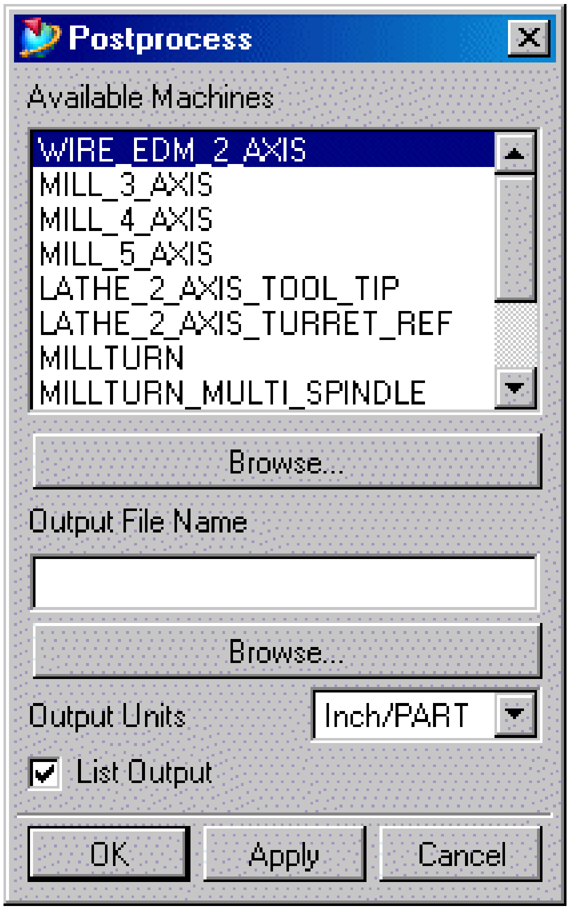
Step 4: Post process the existing operations and examine the output.

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

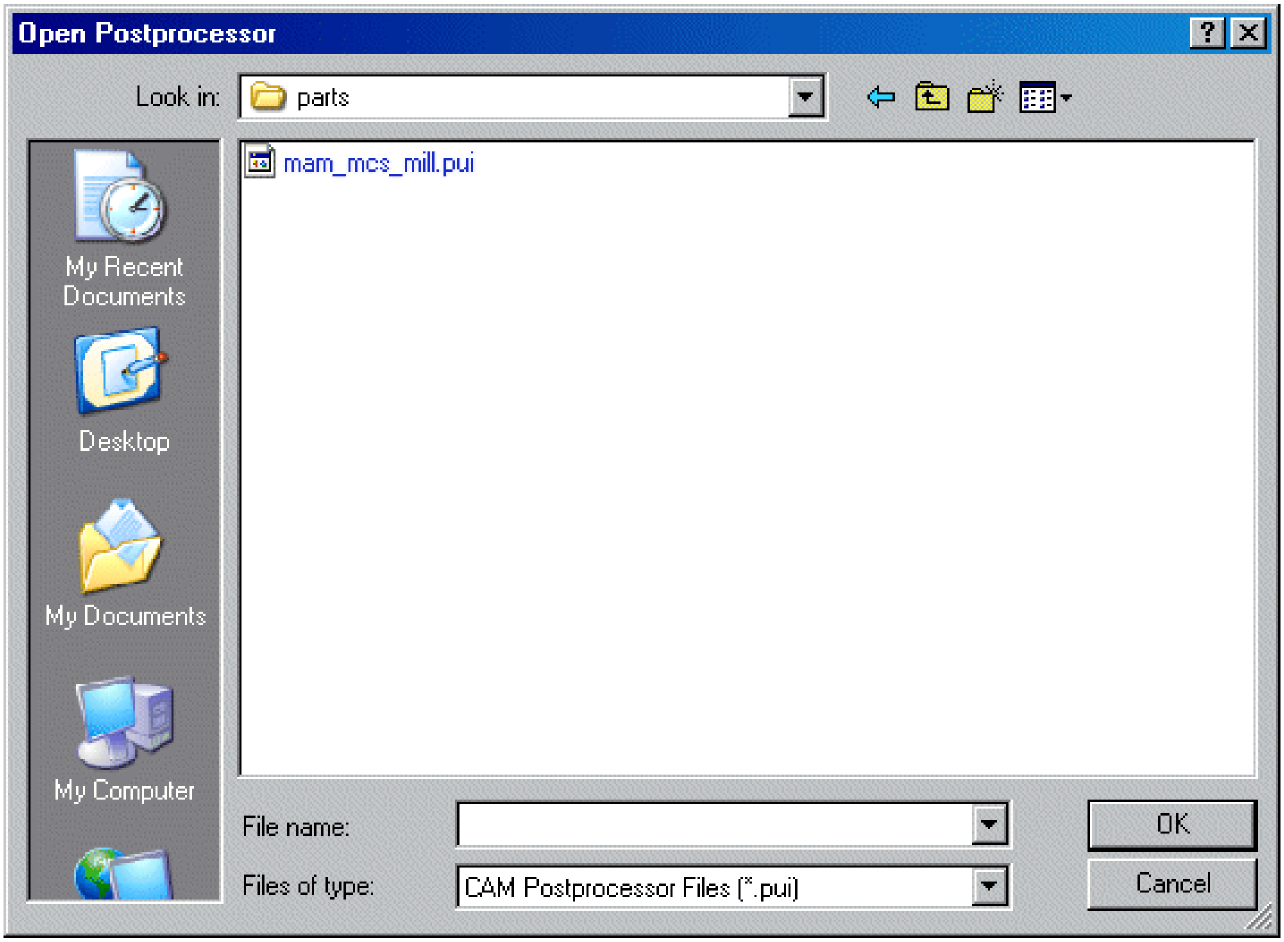
☐ Highlight the **TT1346-AA** parent group.

☐ Choose the **Postprocess** icon. 

The Postprocess dialog is displayed.



☐ Using the **Browse** button under **Available Machines**, browse to your parts directory and select the **mam_mcs_mill.pui** post processor.



☐ Choose **OK**.

☐ Choose **Apply** on the Postprocess dialog.

☐ If necessary, choose **OK** to the **Path Out of Date** dialog.

The posted output is displayed.

```
===============================================================
Information listing created by :
Date                           :
Current work part              :
Node name                      :
===============================================================
( PARTNO TT1346-AA )
( PART NAME         : )
( CREATED BY        : )
( CREATION DATE     : )
( CREATION TIME     : )
%
N0010 G40 G17 G90 G70
( PATH NAME  : PROFILE_000 )
( POCKET NO  : 0 )
( TOOL NAME  : EM_1.000    )
N0020 G91 G28 Z0.0
:0030 T00 M06
N0040 G0 G90 X37.775 Y20.4 S764 M03
N0050 G43 Z-17.5 H00
N0060 Z-17.9
N0070 Z-18.4
N0080 G1 Z-18.5 F1.4 M08
N0090 G3 X37.575 Y20.5 I-.2 J-.15
N0100 G1 X37.5
N0110 X36.5 F1.5
N0120 Y19.5
N0130 X38.5
N0140 Y20.5
N0150 X37.425
N0160 G3 X37.225 Y20.4 I0.0 J-.25
N0170 G0 Z-17.5
(============================================)
( OPERATION: PROFILE_000   TIME: 4.6 MIN.)
(============================================)
( PATH NAME  : PROFILE_090 )
( POCKET NO  : 0 )
( TOOL NAME  : EM_1.000    )
N0180 G0 X29.6 Y27.775 S764 M03
N0190 Z-17.9
```


Notice the values for the X, Y and Z axes.

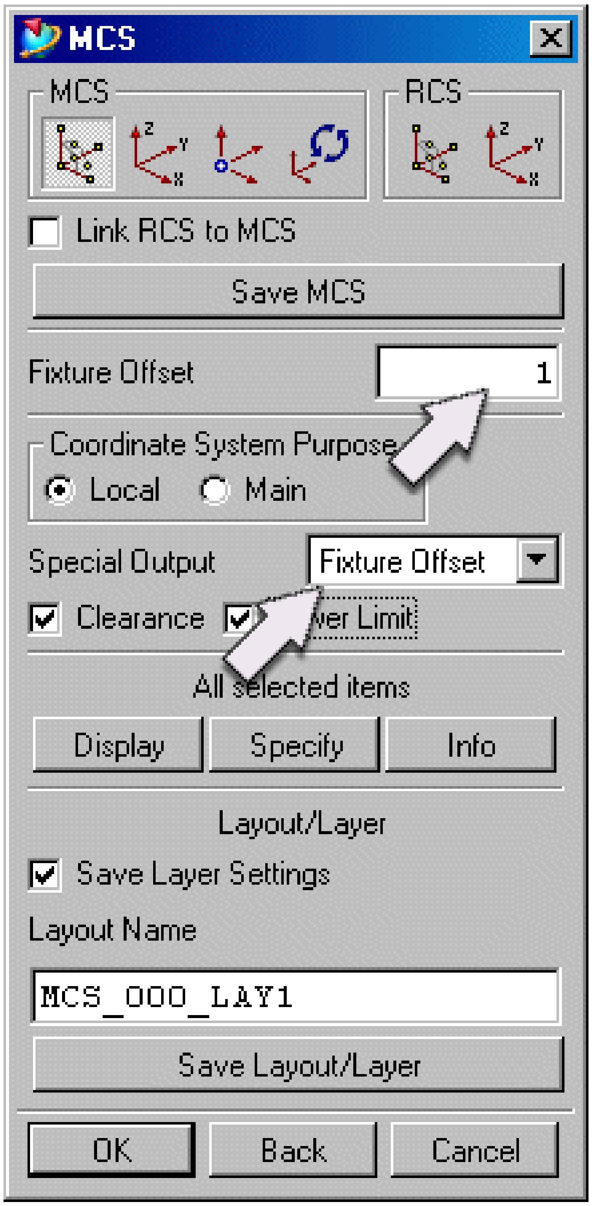
You will now modify the local MCS by adding fixture offsets and will re-post the operations.

☐ Cancel the Postprocess dialog.

Step 5: Modify the Local MCS by adding fixture offsets and re-posting the operations.

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

☐ Highlight the **MCS_000** parent group, key in **1** for the **Fixture Offset** and change the **Special Output** to **Fixture Offset.**

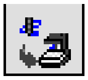


☐ Repeat the above step action item for **MCS_90** and **MCS_180** parent groups, using **2** as the fixture offset for the MCS_90 parent group and **3** as the fixture offset for the MCS_180 parent group.

☐ Choose **OK**.

☐ Change to the **Program Order** view of the operation Navigator.

☐ Highlight the **TT1346-AA** parent group.

☐ Choose the **Postprocess** icon. 

The Postprocess dialog is displayed.

☐ If necessary, browse to your home directory and select the **mam_mcs_mill.pui** postprocessor.

☐ Choose **OK**.

☐ Choose **Apply** on the Postprocess dialog.

☐ If necessary, choose **OK** to the **Path Out of Date** dialog.

☐ If necessary, choose **OK** to overwrite **Output File** dialog.

The posted output is displayed.


```
Node name                              :
================================================
( PARTNO TT1346-AA )
( PART NAME          : )
( CREATED BY         : )
( CREATION DATE      : )
( CREATION TIME      : )
%
N0010 G40 G17 G90 G70
( PATH NAME  : PROFILE_000 )
( POCKET NO  : 0 )
( TOOL NAME  : EM_1.000    )
N0020 G54
N0030 G91   8 Z0.0
:0040 TOO M
N0050 G0 G90 X1.775 Y.4 S764 M03
N0060 G43 Z.5 H00
N0070 Z.1
N0080 Z-.4
N0090 G1 Z-.5 F1.4 M08
N0100 G3 X1.575 Y.5 I-.2 J-.15
N0110 G1 X1.5
N0120 X.5 F1.5
N0130 Y-.5
N0140 X2.5
N0150 Y.5
N0160 X1.425
N0170 G3 X1.225 Y.4 I0.0 J-.25
N0180 G0 Z.5
(==============================================)
( OPERATION: PROFILE_000   TIME: 4.5 MIN.)
(==============================================)
( PATH NAME  : PROFILE_090 )
( POCKET NO  : 0 )
( TOOL NAME  : EM_1.000    )
N0190 G55
N0200 G0    775 Y.4 S764 M03
N0210 Z.1
```

Notice the values for the X, Y and Z axes and compare with the previously posted output. Also note the G54, G55 and G56 that is used for fixture offsets.

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

Summary

The majority of "multi-axis" machining can actually be considered to be planar or fixed axis in nature. The spindle axis, on some machines, is not normal to the Z direction of the machine tool and the actual machining does not force a change in rotation of the rotary axis. Designation of tool axis and MCS is crucial to perform this type of work.

In this lesson you:

- Performed planar type machining at a tool axis other than (0,0,1).
- Specified the MCS at the center of rotation for multi-axis machining.

Lesson

2 Sequential Mill Basics

Purpose

Sequential Mill operations allow you to machine contoured parts by cutting from one surface to the next in a sequence of moves referred to as sub-operations. These sub-operation types allow the flexibility to completely control cutter movements to obtain desired results.

Objective

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

- use Sequential Mill operations to create multi-axis tool paths
- create Sequential Mill rough and finish operations

Sequential Milling Overview

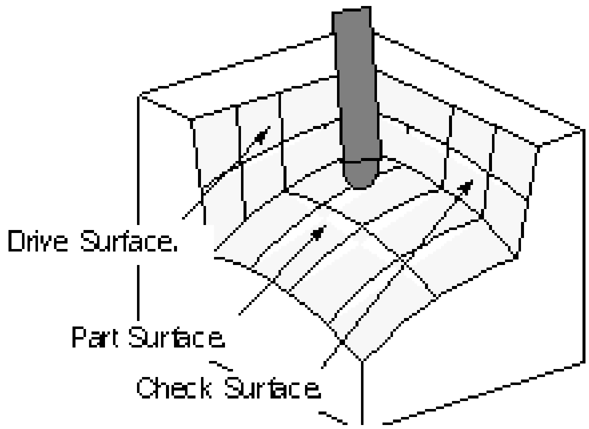
Sequential Milling operations are an alternative to Fixed or Variable Contour operations used for finishing 3, 4, and 5-axis parts. You normally use Fixed and Variable Contour operations to finish cut areas using *area* tool motion. Sequential Milling operations are used to finish cut part edges using *linear* tool motion. You can area machine using Sequential Mill, however, the area is usually limited to an offset from a single drive surface or a single part surface (or both).

Sequential Mill also provides tool axis control capabilities in maintaining a tool position relative to drive and part geometry, recognizing multiple check surfaces.

Sequential Milling Terminology

The following terms pertain to Sequential Milling:

- Part surface controls the bottom of the tool
- Drive surface controls the side of the tool
- Check surface controls the tool stopping position



In the above illustration, the tool is in contact with the Part, Drive and Check surfaces. The bottom of the tool follows the Part surface, the side of the tool follows the Drive surface until the tool contacts the Check surface.

Before you specify the part, drive, and check geometry, you must indicate where the tool will stop. You have four possible choices:

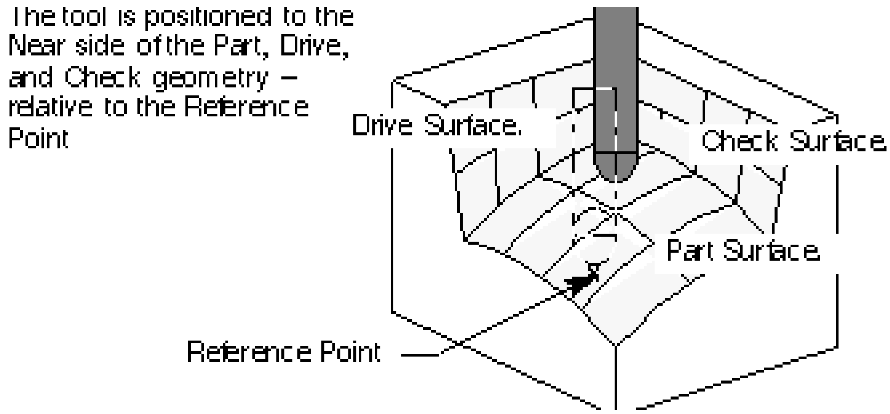
- *Near Side* indicates that the tool will stop when it reaches the closest side of the specified part relative to the current tool position
- *Far Side* indicates that the tool will stop when it reaches the farthest side of the specified part relative to the current tool position
- *On* indicates that the tool will stop when its center axis reaches the edge of the specified part relative to the current tool position
- *Ds-Cs Tangency and Ps-Cs Tangency* indicates that the tool will stop when it is at the position that the drive (or part) surface is tangent to the check surface

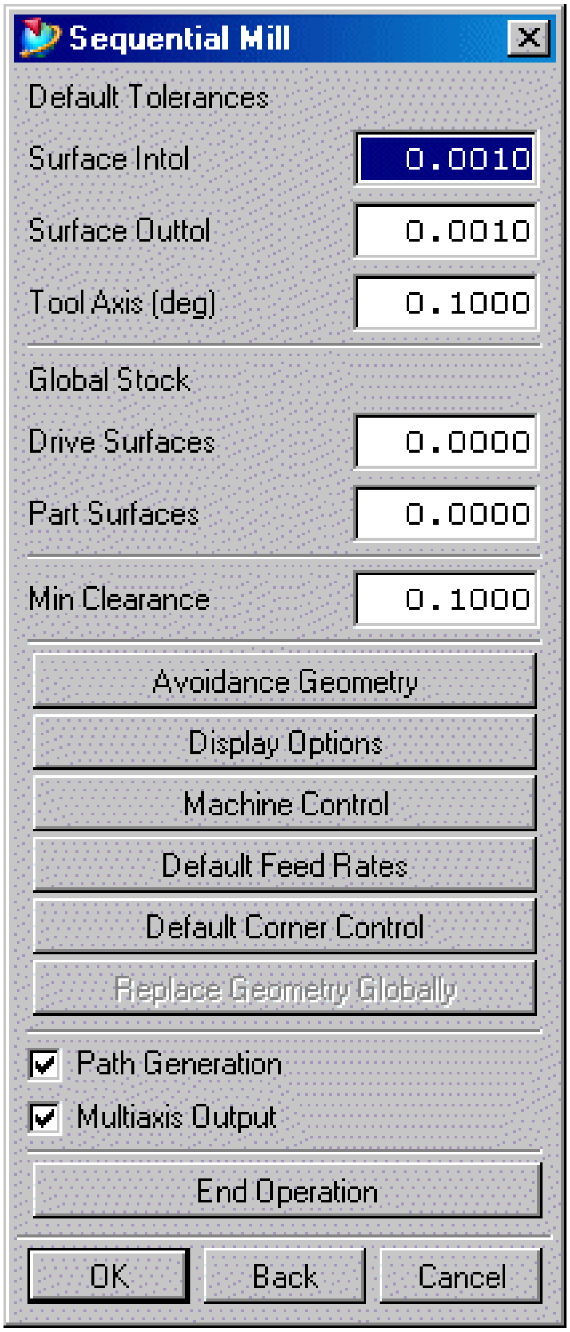
Note that when a wall is tangent to a corner radius and the tool will contact that tangency, you must choose this option. Otherwise, you must choose the Near Side, Far Side or On condition.

-

You must initially specify a tool Reference Point position to determine the side of the drive, part, and check geometry for tool placement. This establishes direction only.

Once you specify the Reference Point, you can specify the tool starting position as the Near Side, Far Side, or On the Drive, Part, or Check geometry.





The Sequential Mill dialog -

Allows you to:

- add stock to all drive and part surfaces
- specify a Minimum Clearance value to be used in Engage and Retract sub-operations
- add Corner Control
- specify Path Generation which determines whether the tool path is output for each sub-operation
- Multi-axis output

After you set the Sequential Mill operation options you can create a sub-operation to control tool motion.

Sub-operations are individual tool motions. The four different types of sub-operations are Engage, Continuous Path, Point to Point and Retract motion.

Normally, you will use these sub-operations in sequential order.

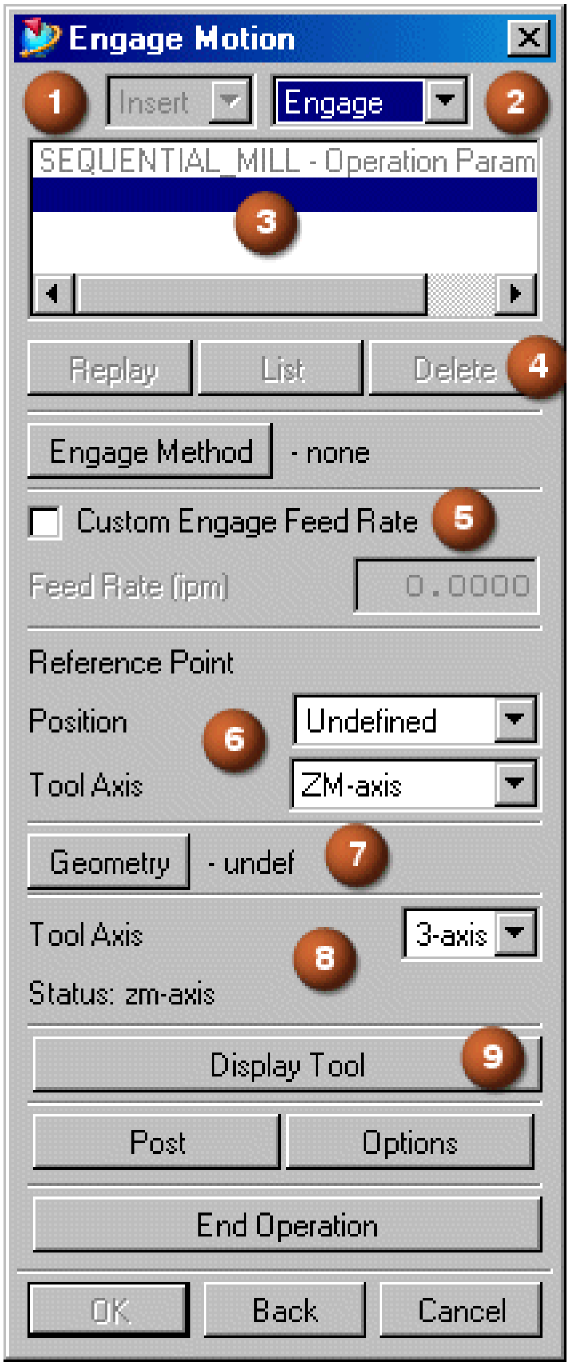
- initially, specify an Engage move

- then, specify Continuous Path motions
- at the end of the tool path, specify a Point to Point
- and then a Retract move

After creating or editing an operation, you choose End Operation either generate the tool path, or save the operation without tool path generation.

The Engage Motion

The Engage Motion sub-operation defines where the tool initially contacts the part. This is usually the first sub-operation dialog which you will encounter.



(1) Insert or modify sub-operations

(2) toggles between 4 types of sub-operations

(3) list of sub-operations

(4) replay, list or delete highlighted sub-operation

(5) change engage feed rate

(6) relative tool position (required)

(7) specify geometry (required)

(8) specify tool axis

(9) display tool at current location

The Continuous Path Motion dialog

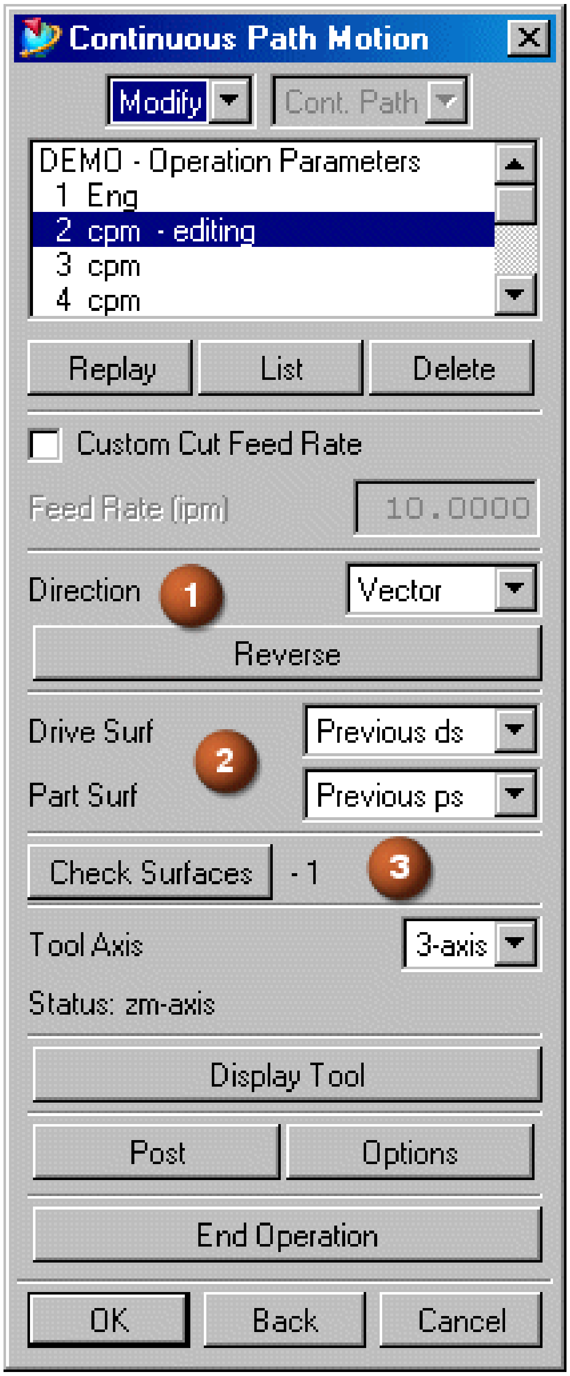
After engaging the part, the tool motion is determined by a series of Continuous Path Motion (CPM) sub-operations.

Each tool move requires specific Drive, Part and Check geometry:

- Drive geometry controls the side of the cutter

- Part geometry controls the bottom of the cutter
- Check geometry stops the cut movement

The cutter moves along the drive and part geometry until it reaches check geometry.



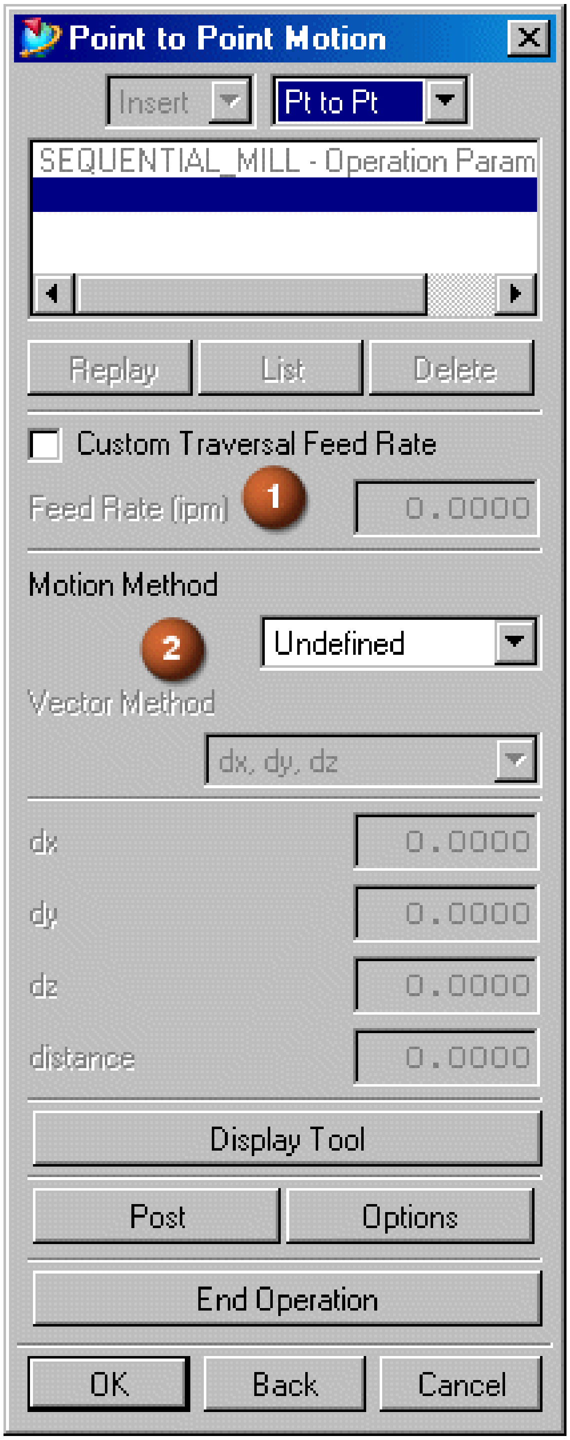
(1) specify tool direction

(2) must be specified

(3) number of check surfaces

The Point To Point Motion dialog

The Point to Point dialog enables you to create linear, non-cutting moves. It is used to move the tool to another position where continuous path motions can then continue. You may or may not need to use this dialog when creating Sequential Mill operations.

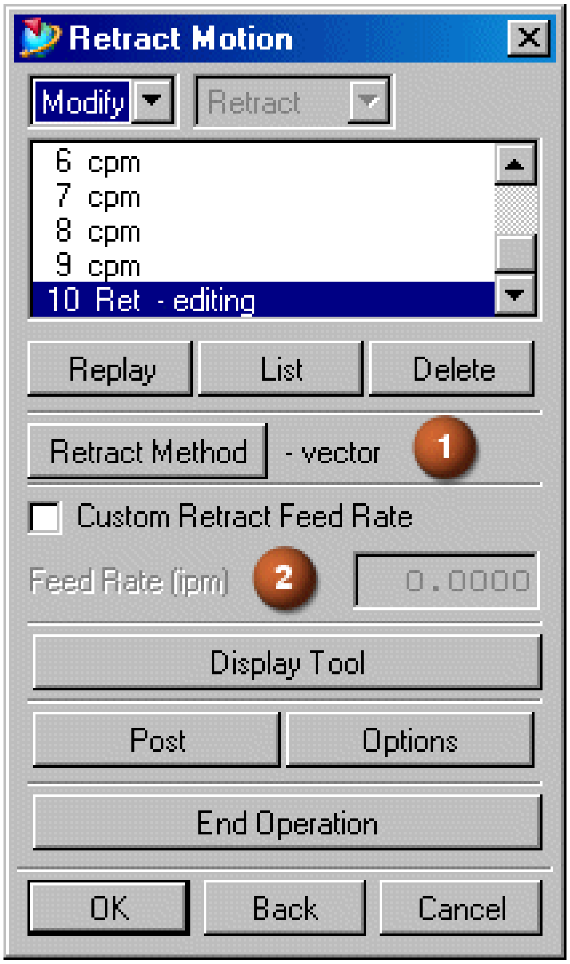


(1) specify special traverse feed rate

(2) defines the way the tool will move to the next location

The Retract Motion dialog

The Retract Motion dialog enables you to create a non-cutting move from the part to the avoidance geometry or to a defined retract point. It is similar to the Engage Motion dialog.



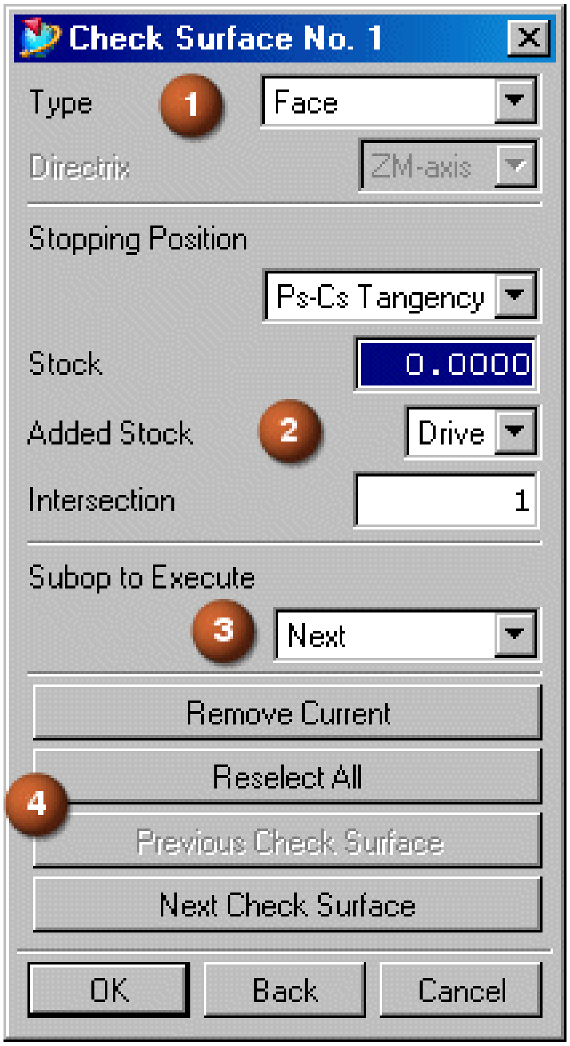
(1) type of retract move

(2) feed rate control for feed rate move

Defining the Check Surfaces

When you are creating a Continuous Path Motion sub-operation, you must define one or more Check Surfaces.

By default, the Check Surface for one sub-operation becomes the Drive Surface for the next sub-operation. This often saves you from having to specify the Drive Surface. The Part Surface, is by default, the same for each sub-operation throughout the tool path. This also saves you from having to specify the Part Surface. Normally, you only need to specify the Check Surface in each sub-operation.



(1) type of geometry used for Check surface

(2) add stock or define the tool position with respect to Check geometry

(3) action to take after sub-operation

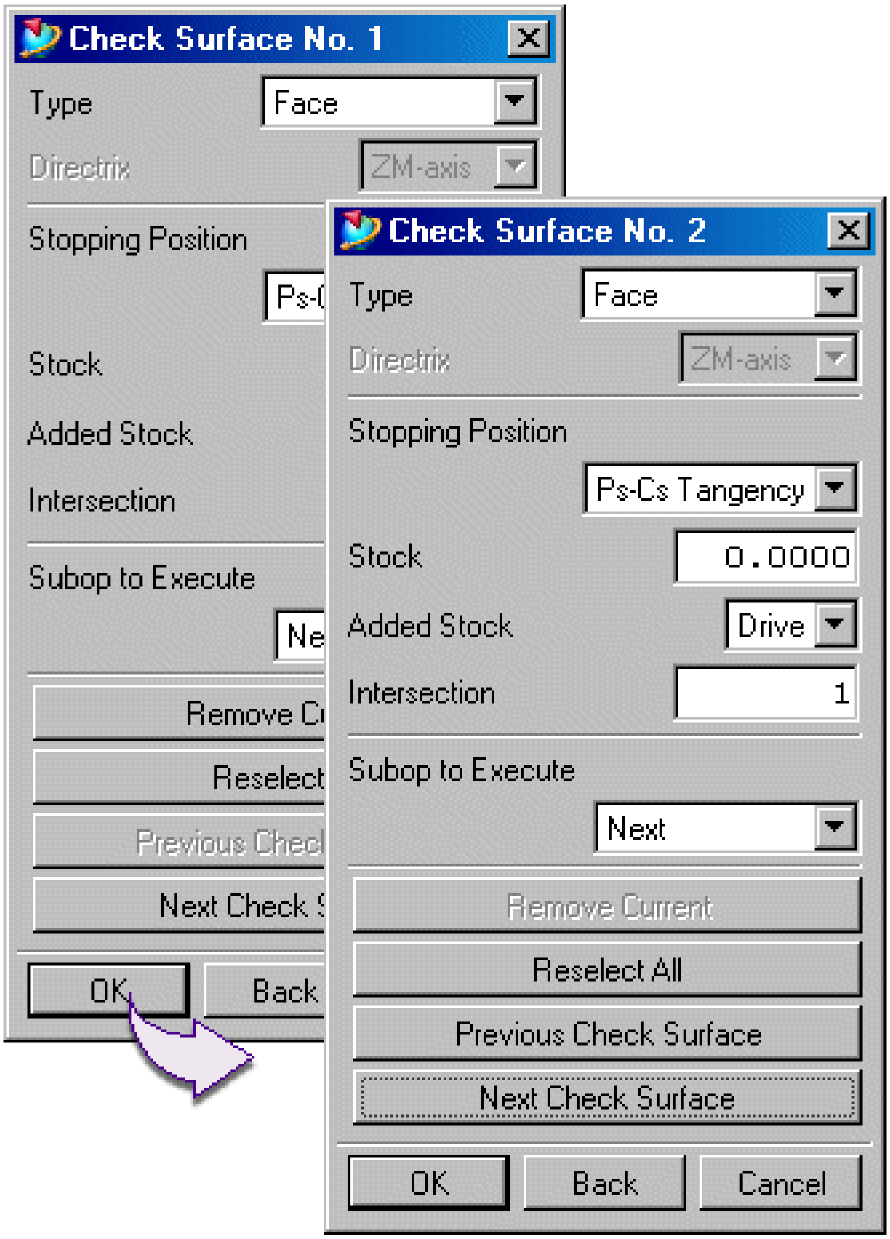
(4) navigating through multiple Check surface dialogs

Multiple Check Surfaces

In a Continuous Path Motion command the cutter moves along the Drive and Part Surface until it reaches a Check Surface.

If you specify more than one Check Surface (multiple check surfaces), motion continues until the tool reaches the first of the possible stopping positions.

You can define up to five Check Surfaces for each Continuous Path Motion sub-operation. After you have defined the first Check Surface, you are automatically prompted to define the next Check Surface.



The following activities will familiarize you with Sequential Mill operations.

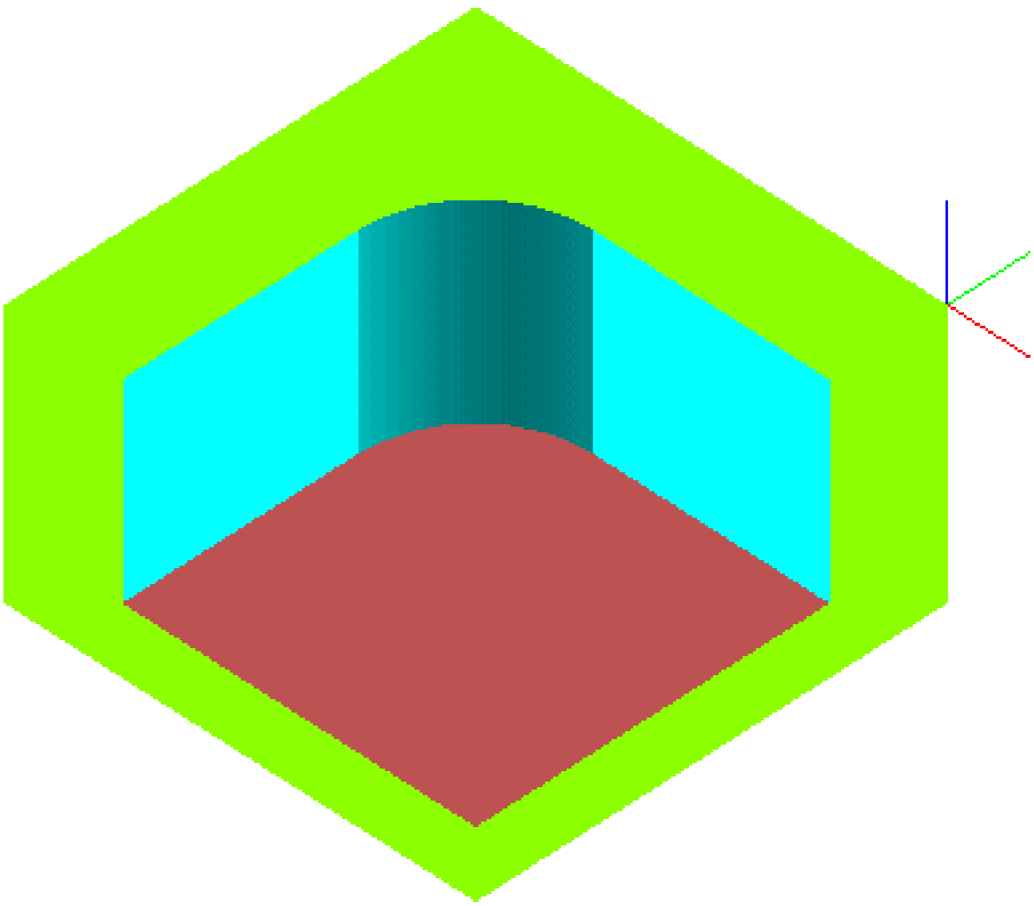
Activity: Basic Sequential Milling Techniques

In this activity, you will use basic interactions necessary to create Sequential Milling operations. You will drive a tool around a simple part, create several sub-operations, and establish Drive, Part, and Check geometry used in the various operations.

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

☐ Open the part file **mam_box_mfg**.

This part is programmed in the context of an assembly. The top-level component, **mam_box_mfg** contains all of the manufacturing data. The **mam_box_stock** file contains a WAVE-linked representation of the raw material and the **mam_box** file contains the part that is to be machined.



The raw material file, **mam_box_stock**, has been hidden from the display.

☐ Rename the part to ***_box_mfg**.

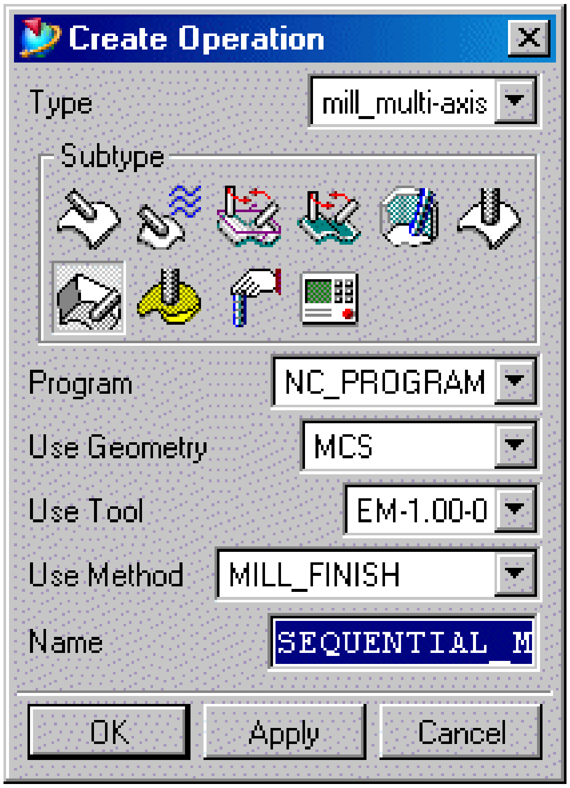
☐ Enter the **Manufacturing application**.

The necessary Parent Groups (i.e. Geometry, Machine, Program and Method) have already been created for you.

Step 2: Create a Sequential Milling operation.

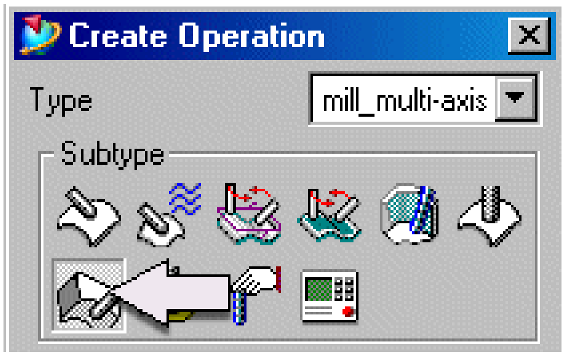
☐ Choose the **Create Operation** icon

The Create Operation dialog is displayed.

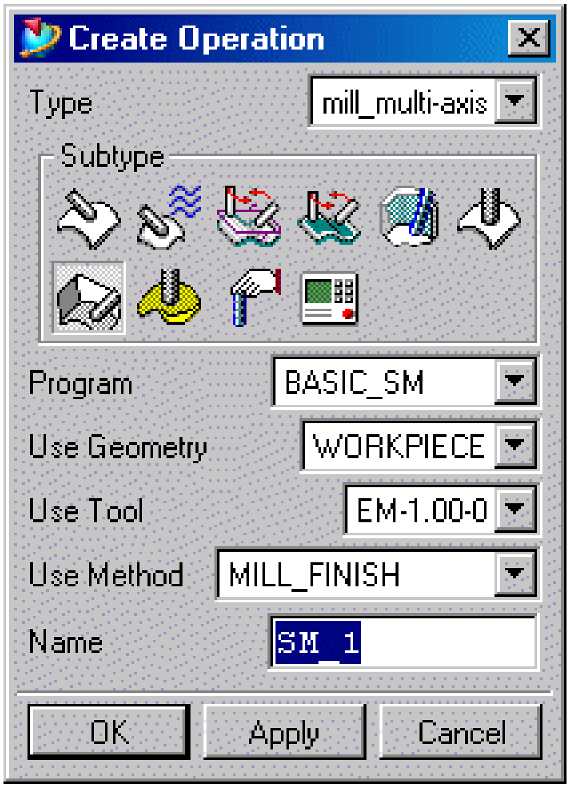


☐ If necessary, change the **Type** to **mill_multi-axis**.

☐ Choose **Sequential_Mill** as the **subtype**.



☐ Set the **Parent** objects as shown and name the operation **SM_1**:

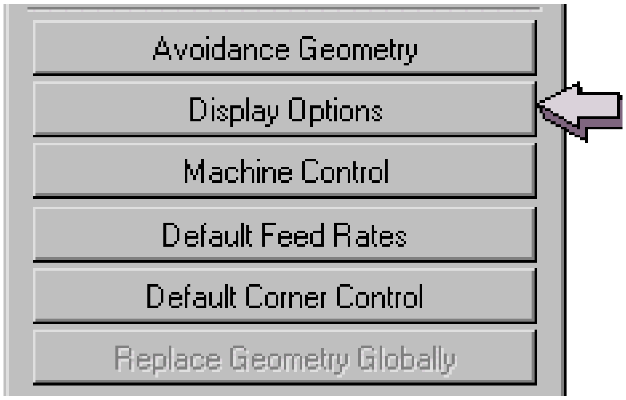


☐ Choose **OK**.

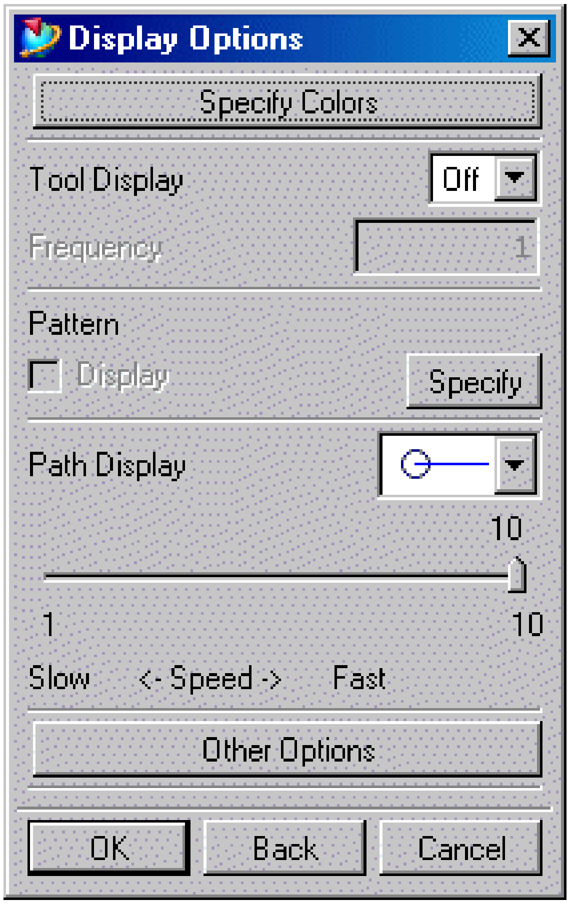
The **Sequential Mill** dialog is displayed.

This dialog allows the input of basic global parameters that are active throughout the operation (unless changed in an sub-operation).

☐ Choose the **Display Options** button.



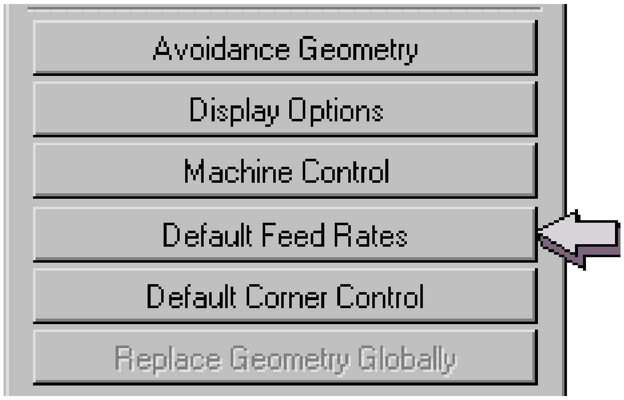
The **Display Options** dialog is displayed.



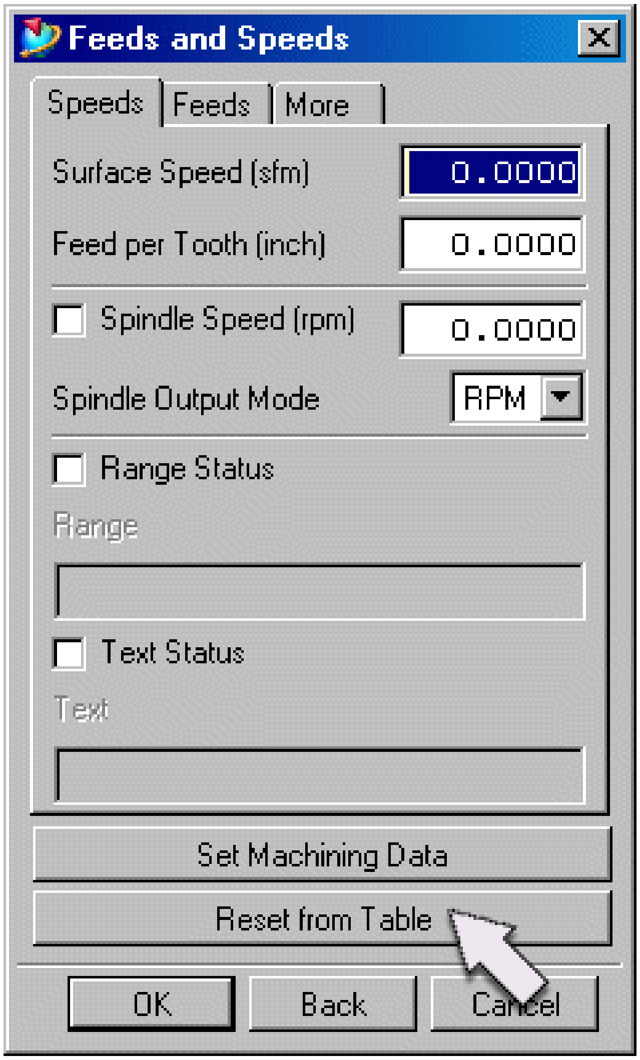
- ☐ Change the **Tool Display** to **3-D** and the **Path Display Speed** to **9**.

- ☐ Choose **OK**.

- ☐ Choose the **Default Feed Rates** button.



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



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

Based on the tool material, part material and number of cutter flutes, the feeds and speeds will be recalculated and reset.

☐ Choose **OK** until you return to the **Sequential Mill** dialog.

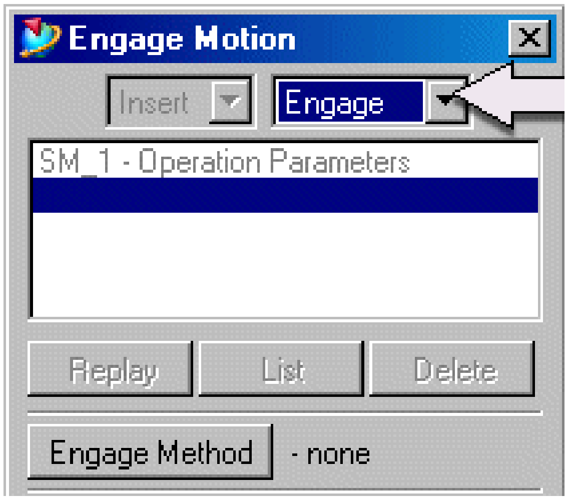
The global parameters are now set.

The **Sequential Milling** dialogs behave somewhat differently than other operation dialogs that you are normally familiar with.

Normally, for any operation, choosing **OK** to the main dialog would save the operation, which would be subsequently displayed in the Operation Navigator. In Sequential Milling, however, choosing **OK** from the main dialog results in the sub-operation dialog being displayed. This is where the actual programming process takes place.

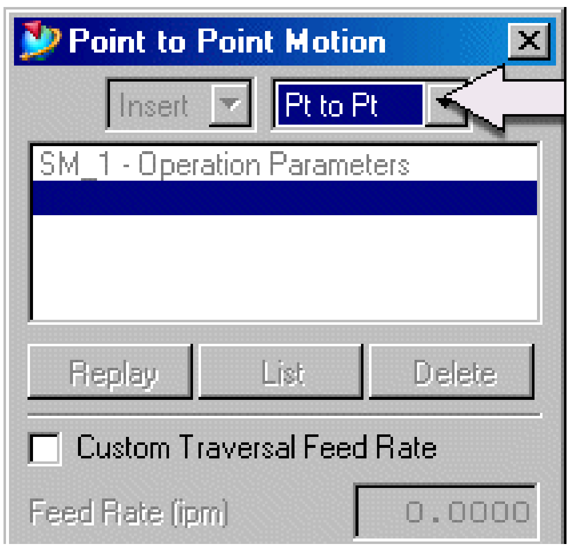
☐ Choose **OK** from the **Sequential Mill** dialog.

The **Engage Motion** sub-operation dialog is displayed. By default, the sub-operation dialog is set to **Engage**.



To properly determine the tool's current location for Near Side/Far Side, establish a **Pt to Pt** motion as the first sub-operation.

☐ Change the motion from **Engage** to **Pt to Pt**.



The dialog changes to match **Point to Point** motion.

You will now establish the tool position, specifying both the position of the tool and the tool axis.

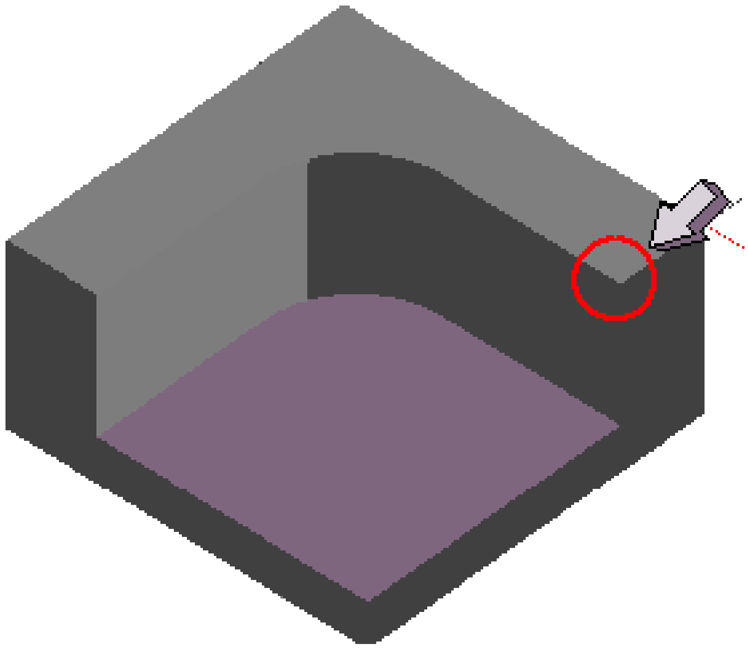
☐ Change the **Motion Method** to **Point, Tool Axis**.

The **Point Constructor** dialog is displayed.

☐ Change the **Offset** from **None** to **Rectangular**.

Note that using a **Rectangular Offset** allows an X, Y and Z delta offset from the point chosen.

☐ Choose the corner of the ledge, as shown.



The **Point Constructor** dialog has changed to allow the input of delta values from the point selected.

☐ Key in the following values:

Delta XC 1.00

Delta YC -1.00

Delta ZC 1.00

☐ Choose **OK**.

The **Vector Constructor** dialog is displayed.

You will accept the default tool axis vector of 0,0,1 which is the same as the Z coordinate of the WCS.

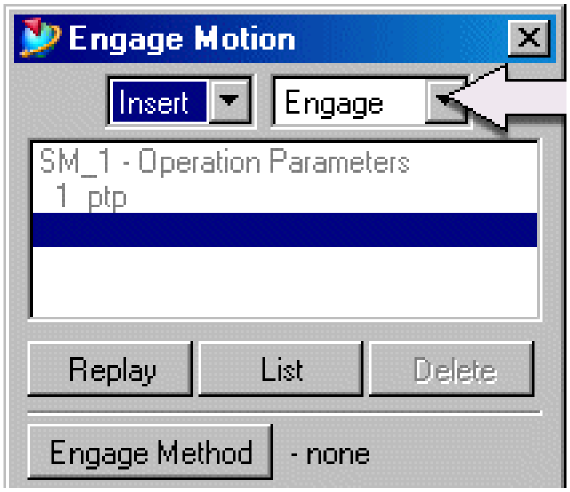
☐ Choose **OK**.

The **Point to Point** sub-operation is complete. By choosing **OK,** the sub-operation will be placed in the sub-op list and you will be ready to create the next sub-operation.

☐ Choose **OK**.

You will now define the **Engage** component.

☐ Change to an **Engage** sub-operation.



The **Engage Motion** dialog is displayed.

This dialog requires **Drive**, **Part** and **Check** geometry. Additionally, you may specify an engage method.

You will specify the geometry first and then the **Engage** method.

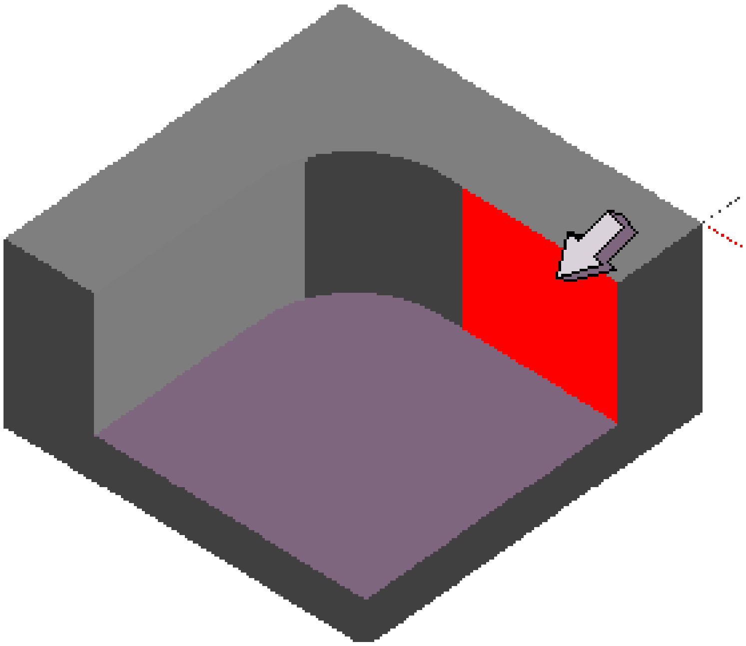
☐ Choose the **Geometry** button from the **Engage Motion** dialog.

The **Engage Geometry** dialog is displayed.

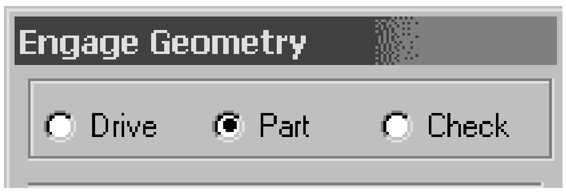
The defaults are set to **Drive** geometry, the **Type** is **Face** and the **Stopping Position** is **Near Side**.

You will now select the **Drive** geometry.

☐ Choose the face as shown.

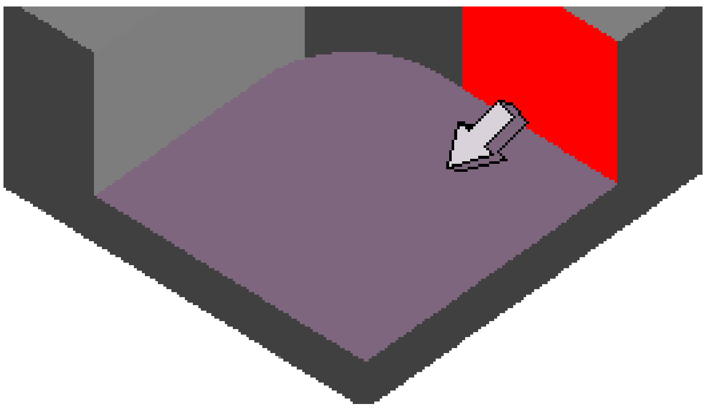


The geometry selection on the dialog advances to **Part** geometry.

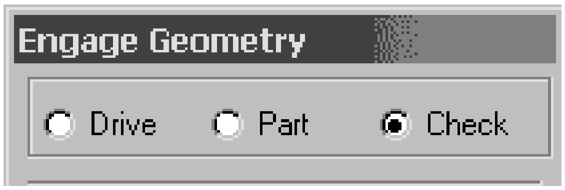


You will now select the **Part** geometry.

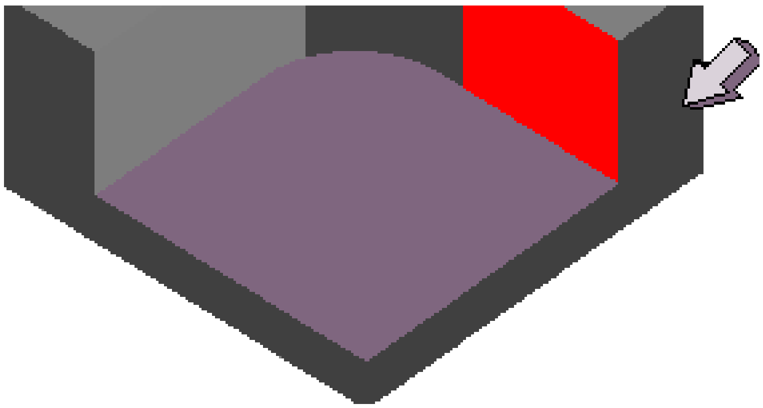
☐ Choose the bottom of the pocket as the **Part** geometry.



The geometry selection on the dialog advances to **Check** geometry.



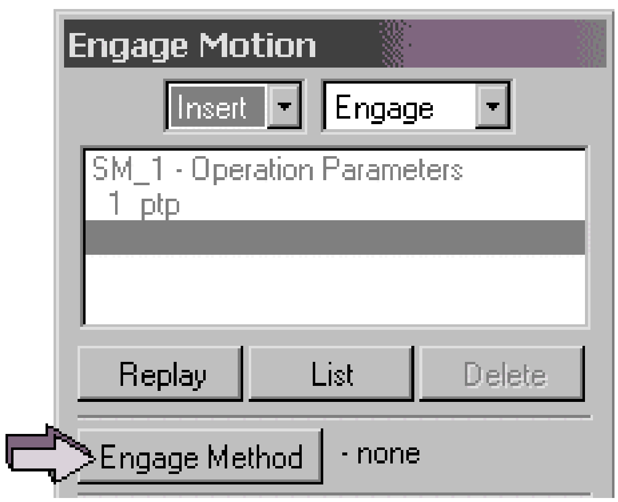
☐ Choose the face, as shown below, as the **Check** geometry.



As soon as the last geometry is selected, the dialog reverts to the **Engage Motion** sub-operation.

You will now specify the **Engage** motion.

☐ Choose the **Engage Method** button from the **Engage Motion** dialog.



The **Engage Method** dialog is displayed.

☐ Change the **Method** to **Vector Only**.

The **Vector Constructor** dialog is displayed.

- ☐ Key in the following values:

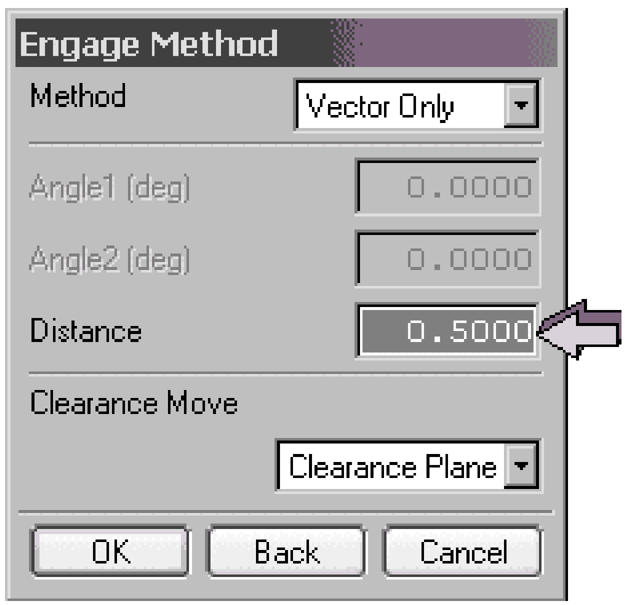
  I=-1.000

  J= 1.000

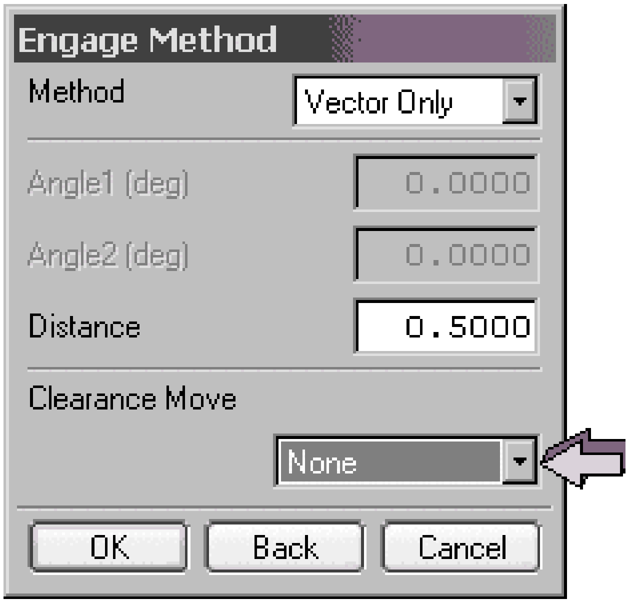
  K=-1.000

- ☐ Choose **OK**.

- ☐ Key in **0.500** in the **Distance** field of the **Engage Method** dialog.

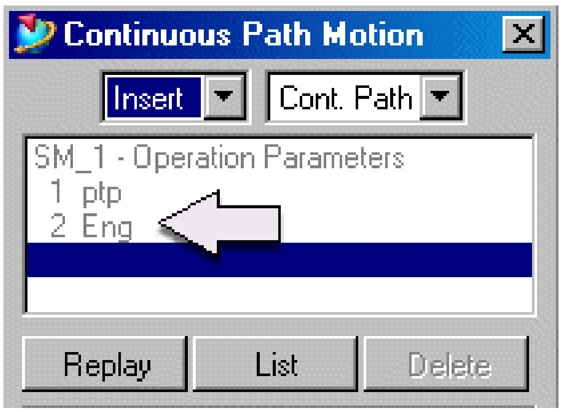


☐ Change the **Clearance Move** to **None**.



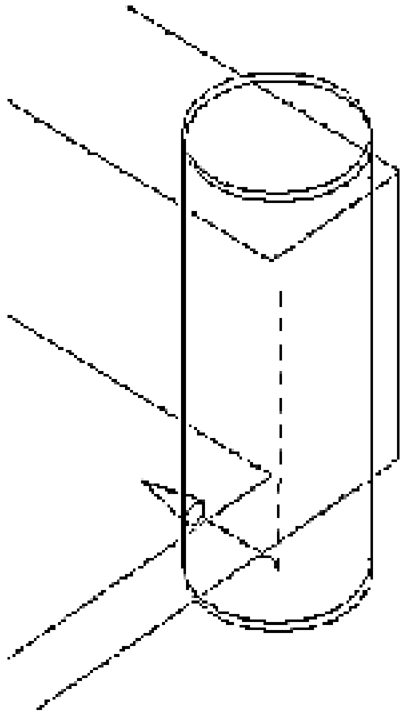
☐ Choose **OK** twice.

The second sub-operation, **2 Eng** is created. The tool side is now positioned tangent to **Drive** and **Check** geometry and tangent to the **Part** geometry with the bottom of the tool.

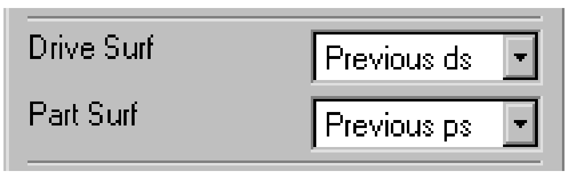


You will now create a **Continuous Path Motion** sub-operation.

The arrow displayed at the bottom of the tool indicates the direction of the next cut. In this case the direction is correct. If the arrow was pointed in another direction, it would have been necessary to change direction by using the **Direction** option.

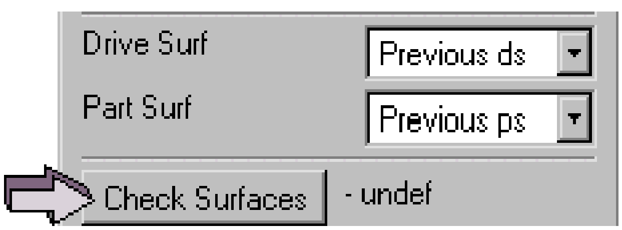


Notice the defaults for **Drive Surf** and **Part Surf**.



The **Drive Surf** is set to the **Previous ds** (drive surface). The **Part Surf** is set to the **Previous ps** (part surface). It will be necessary to set the **Check** surface.

☐ Choose the **Check Surfaces** button.

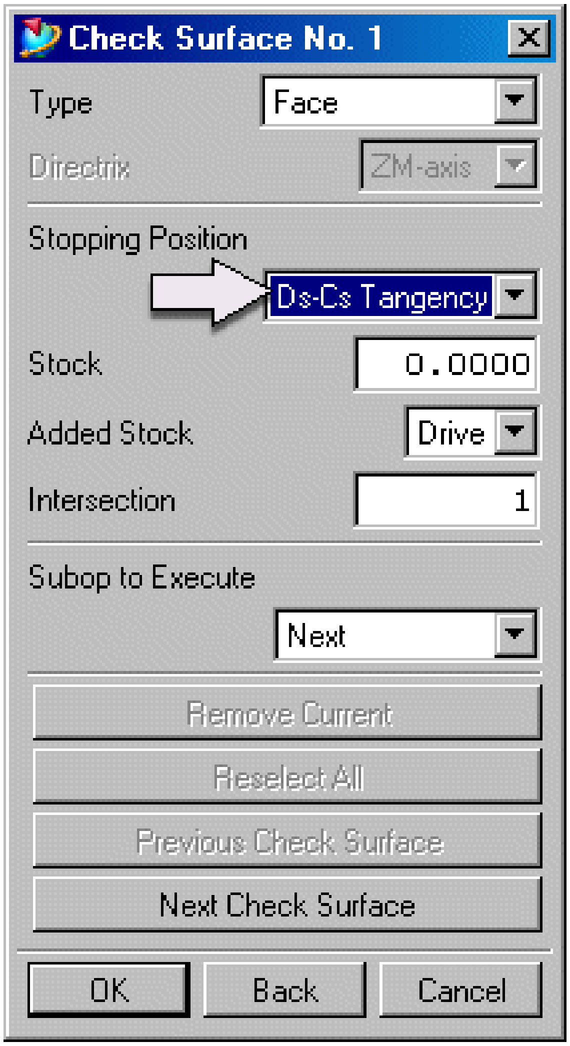


The **Check Surfaces No. 1** dialog is displayed.

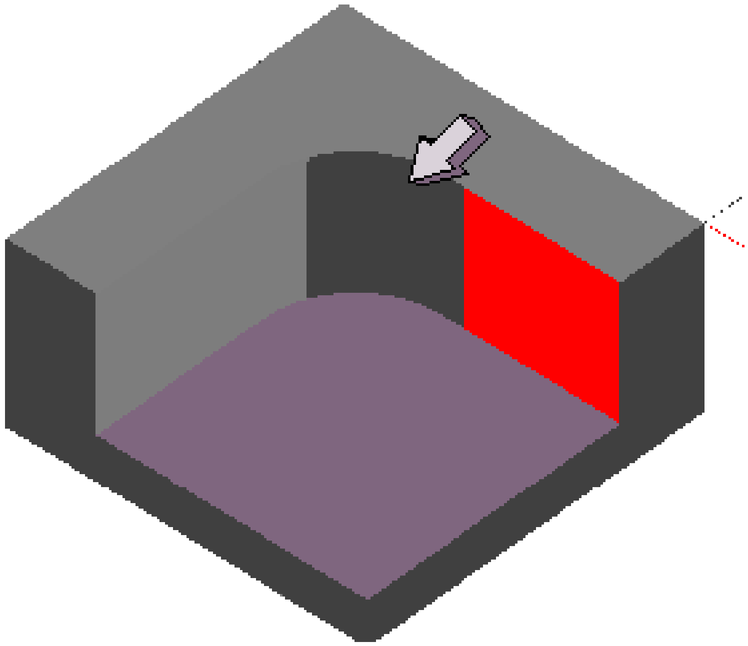
You are now ready to select the first (in this case, the only) **Check** surface. As soon as the surface is selected, the dialog advances to **Check Surface No. 2**. It is important to specify any changes to the dialog before the surface is selected.

Note that the current **Drive** surface is tangent to the next surface that the tool will drive to. A stopping position of **Near Side** is incorrect. You will change the stopping position to **Drive Surface/Check Surface Tangency**.

☐ Change the **Stopping Position** to **Ds-Cs Tangency**.



☐ Choose the Blend face as shown.



There will not be a second **Check** surface to select.

☐ Choose **OK** in the **Check Surface** dialog.

☐ Choose **OK** in the **Continuous Path Motion** dialog.

The third sub-operation, **3 cpm** has been created. You will now create another **CPM** sub-operation.

The processor has automatically forwarded the **Drive** surface to the previous **Check** surface. It has also kept the previous **Part** surface as the new **Part** surface.

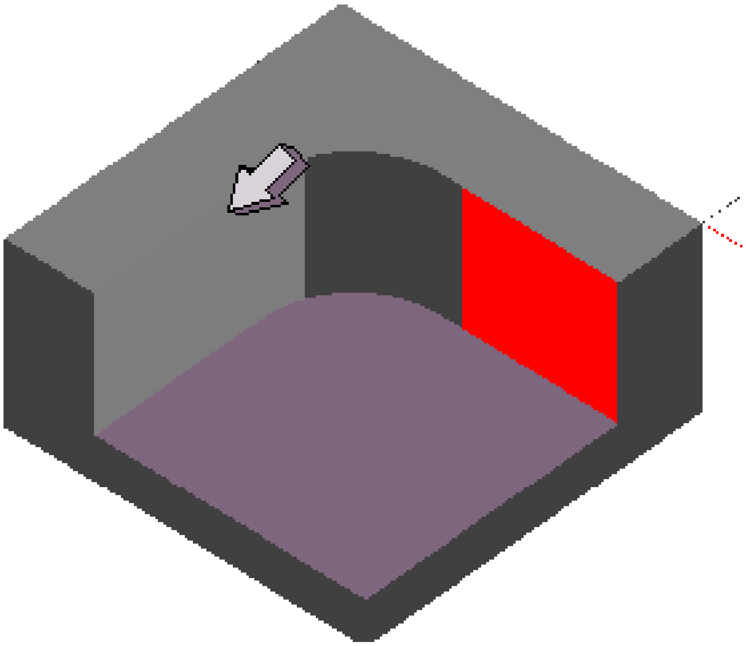
The **Direction of Motion Vector** setting is correct.

You need to choose a new **Check** surface.

☐ Choose the **Check Surfaces** button.

The object type of face is correct as well as the **Stopping Position** of **Ds-Cs Tangency**.

☐ Choose the face as shown.



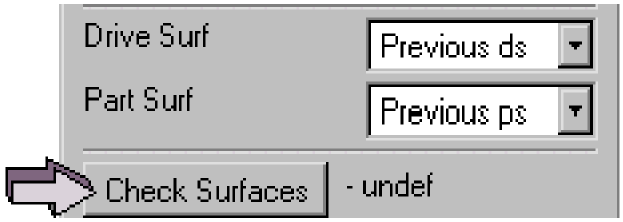
☐ Choose **OK** in the **Check** surface dialog.

☐ Choose **OK** in the **Continuous Path Motion** dialog.

The sub-operation, **4 cpm**, is now placed in the dialog list.

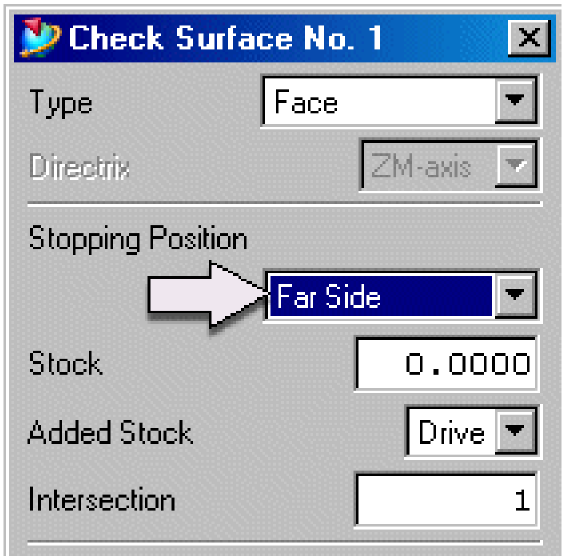
Sequential Mill is now ready for the next sub-operation. Once again, the defaults are correct. You only need to choose a new **Check** surface.

☐ Choose the **Check Surfaces** button.

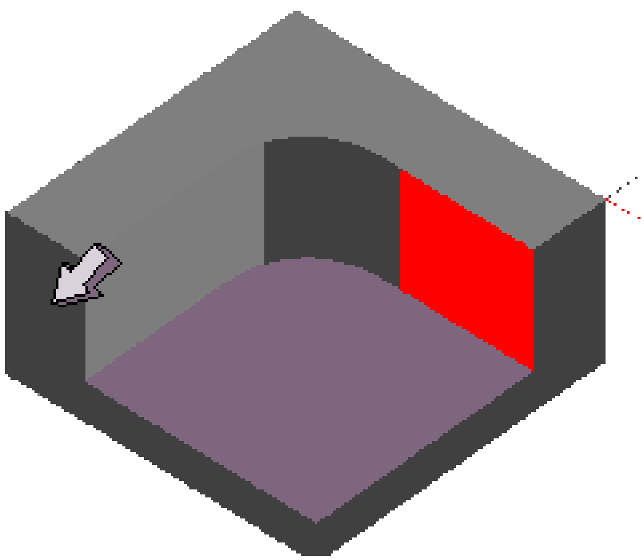


This time, the **Stopping Position** of **Ds-Cs Tangency** is incorrect. You will change it to **Far Side**, so that the tool is completely off the **Part** surface, prior to stopping.

☐ Change the **Stopping Position** to **Far Side**.



☐ Choose the surface as shown below.



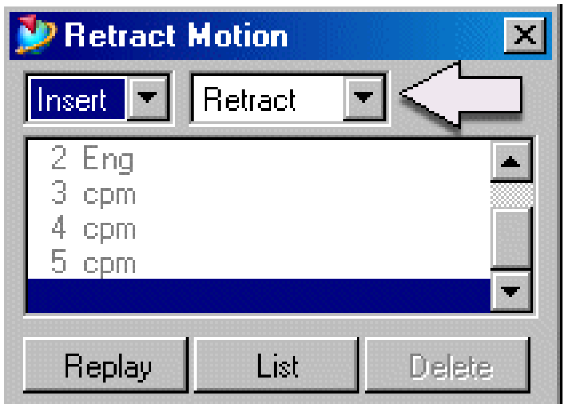
☐ Choose **OK** in the **Check Surface** dialog.

☐ Choose **OK** in the **Continuous Path Motion** dialog.

The sub-operation, **5 cpm**, is now placed in the dialog list.

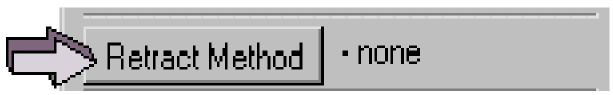
The machining operation is complete. You will now retract the tool a safe distance from the work piece.

☐ Change the sub-operation to **Retract**.



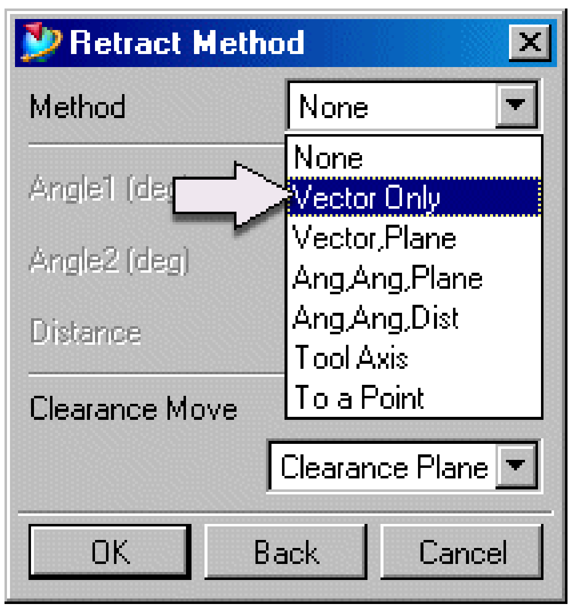
The **Retract Motion** dialog is displayed.

☐ Choose the **Retract Method** button.



The **Retract Method** dialog is displayed.

☐ Change the **Method** from **None** to **Vector Only**.



The **Vector Constructor** dialog is displayed.

☐ Key in the following values:

I= 1.000

J= -1.000

K= 1.000

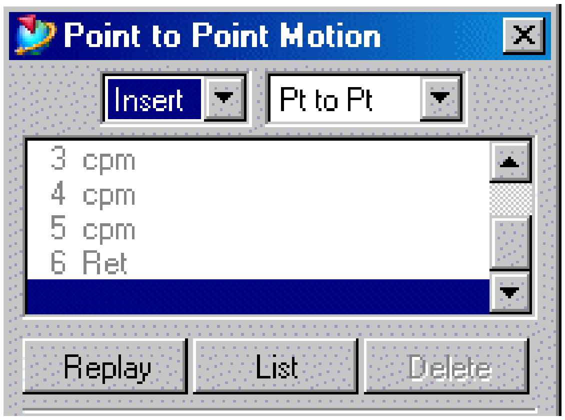
☐ Choose **OK**.

☐ Key in **0.500** in the **Distance** field of the **Retract Method** dialog.

☐ Choose **OK**.

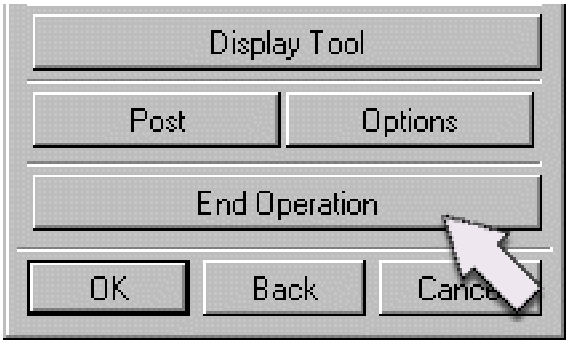
☐ Choose **OK** in the sub-operation dialog.

The sub-operation, **6 Ret,** is now placed in the list.



The tool retracts to the clearance plane. Programming of the wall is complete. To complete the process, select the **End Operation** button.

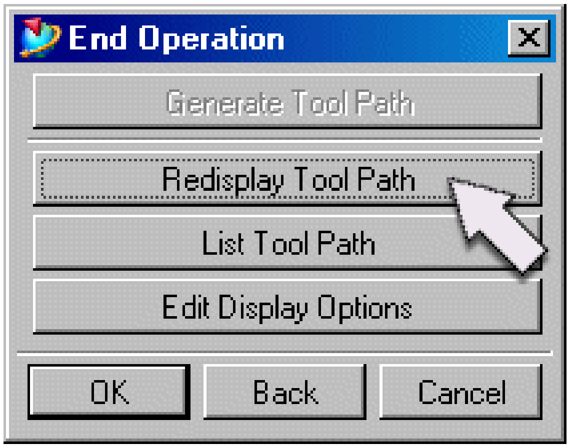
☐ Choose the **End Operation** button.



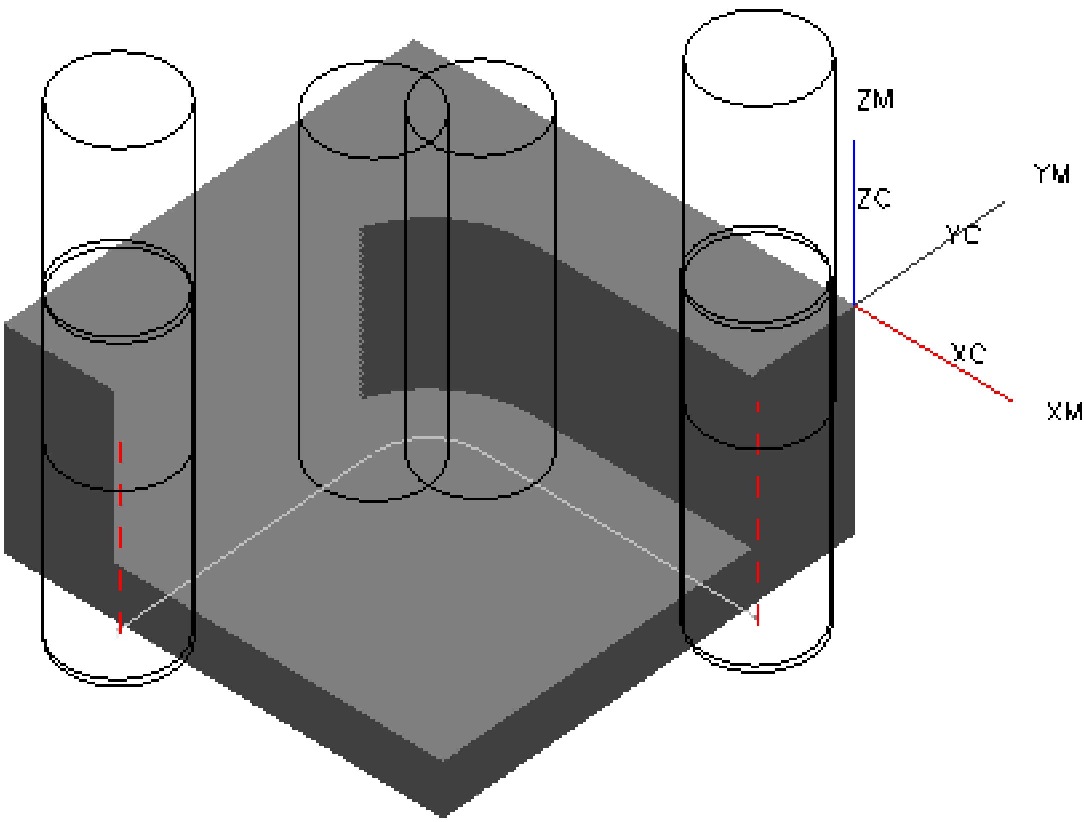
To observe the tool path , refresh the screen and display the tool path.

☐ In the graphics window, use **MB3→Refresh**.

☐ Choose **Redisplay Tool Path** from the **End Operation** dialog.



The tool path is displayed.



- ☐ Choose **OK** from the **End Operation** dialog.
- ☐ **Save** and **Close** the part file.

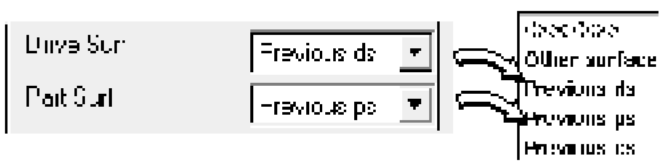
More on Check Surfaces

In the previous activity, you used the same Part surface for each Continuous Path Motion sub-operation. The sub-operation ended after the tool moved along the Drive surface to the Check surface. The Check surface then became the Drive surface for the next sub-operation and the Continuous Path Motion dialog anticipated this choice by selecting Previous Check Surface as the Drive surface at the beginning of each Continuous Path Motion sub-operation.

It is also possible to exchange the Part surface for the next Check surface. One consideration that should be made when exchanging the Check surface as the new Part surface is the Stopping Position. If the Check surface is tangent to the present Part surface and PS-CS Tangency is chosen, the front edge of the cutting tool will be positioned to prevent gouging of the tool into the Check surface. This may cause the tool to be Out of Position to the new Part surface at the beginning of the next move. To compensate for this action, it may become necessary to drive the tool on to the Check surface, even though a tangency condition exists.

In the following activity, the Drive and Part surfaces, as well as the Check geometry will change throughout the operation as you generate the tool path. You will see that the Check surface in a current sub-operation can become the Part surface, as well as the Drive surface, in the next sub-operation. You will also see that the processor is able to anticipate your choice for Drive and Part surfaces in Continuous Path Motion sub-operations, so that you only need to specify the Check surface(s).

When selecting either Drive or Part surface from the Continuous Path Motion dialog, you have the options of Other Surface, Previous ds, Previous ps and Previous cs.

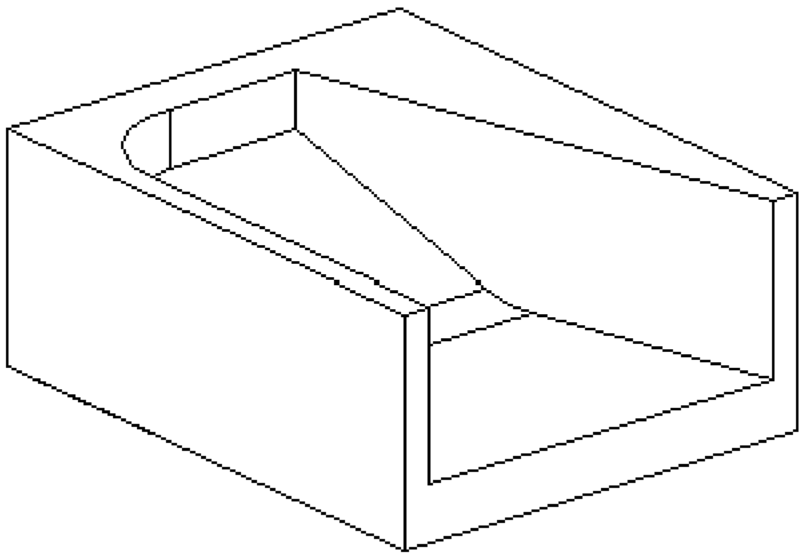


Activity: Sequential Milling of a Multi-Surfaced Floor

In this activity, you will machine a floor that is flat, sloped, and curved. The part requires that you re-specify the part surface when the floor surface changes.

Step 1: Open a new part, rename and begin a Sequential Mill operation.

☐ Open the part file **mam_sq_3** and rename it to **_mam_sq_3**.



☐ Choose **Application →Manufacturing**.

☐ Choose the Operation Navigator tab from the resource bar.

☐ In the Operation Navigator, **Replay** the operation named **DEMO**.

You will now create an operation identical to the operation which you just replayed.

Step 2: Create the Sequential Mill operation.

☐ Choose the **Create Operation** icon.

☐ If necessary change the **Type** to **mill_multi_axis**.

☐ Choose the **SEQUENTIAL_MILL** icon. 

☐ On the Create Operation dialog, set:

Program:**MULTI-FLOOR-PROG**

Use Geometry: **WORKPIECE**

Use Tool:**EM_.75_.125**

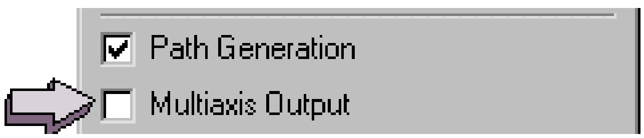
Use Method:**MILL_FINISH**

☐ Enter the operation name **fin-poc-walls** into the **Name** field.

☐ Choose **OK**.

The **Sequential Mill** dialog is displayed.

☐ On the **Sequential Mill** dialog, verify **Multiaxis Output** check box is **OFF**.



The global parameters are now set and you are ready to begin the Sequential Milling process.

☐ Choose **OK** and continue to the **Engage Motion** dialog.

Step 3: Specify an Engage motion.

You will now create a vector that will be used for engaging the part.

☐ Choose the **Engage Method** button.

The **Engage Method** dialog is displayed.

☐ Change the **Method** to **Vector Only**.

The **Vector Constructor** dialog is displayed.

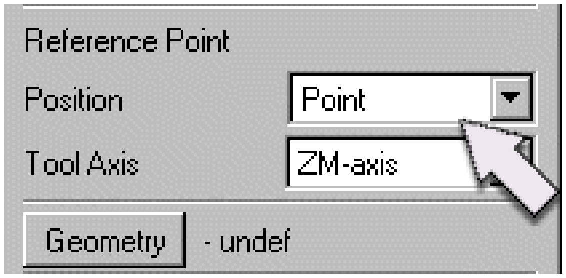
☐ Key in the **-1.000** value for **I**.

☐ Choose **OK**.

☐ Key in **0.500** in the **Distance** field of the **Engage Method** dialog.

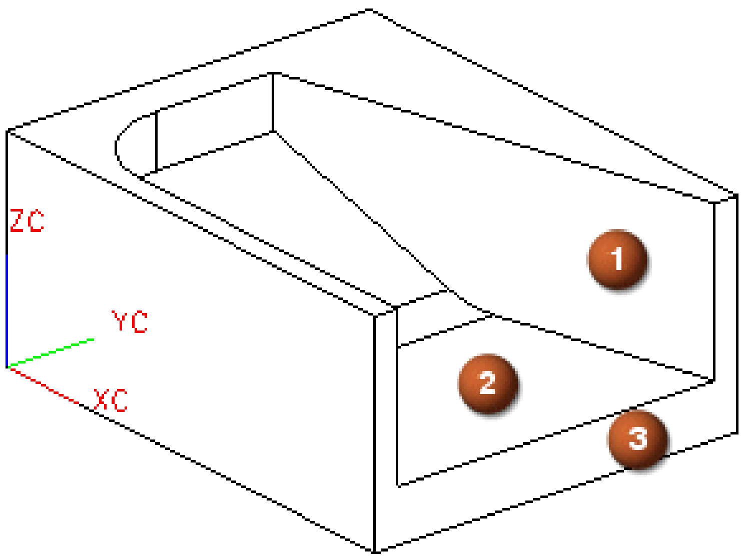
☐ Choose **OK** to return to the **Engage Motion** dialog.

- ☐ Under **Reference Point** label, specify a **Position Point** at: **X=11, Y=6.5**, **Z=2**



- ☐ Choose **OK** in the **Point Constructor** dialog.

- ☐ Choose the **Geometry** button and specify the **Drive** and **Part** surfaces as shown.



(1) Drive Surface

(2) Part Surface

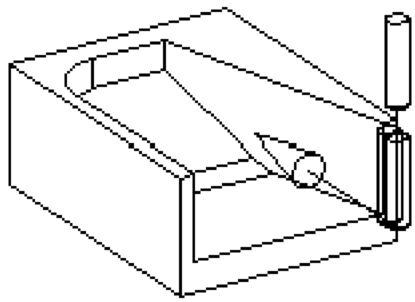
(3) Check Surface (add .250 stock)

- ☐ Specify **.250 Stock** for the **Check** surface, prior to selecting the surface.

  You must enter any **Stock** value and change the **Stopping Position** status before you select the **Check Surface**.

- ☐ Choose **OK**.

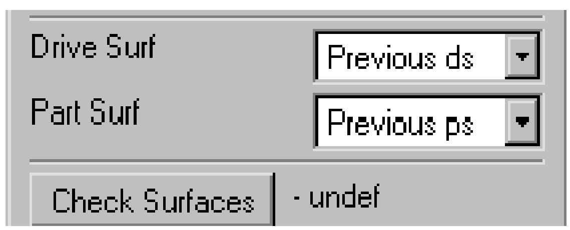
The tool moves from the Clearance plane to the position just specified.



The tool direction arrow shows the current direction of motion. Throughout this activity, change the direction arrow whenever necessary so that it points in the intended cut direction.

Step 4: Specify Continuous Path motion.

Sequential Mill expects the next **Drive** surface to be the previous **Drive** surface, and the next **Part** surface to be the previous **Part** surface.



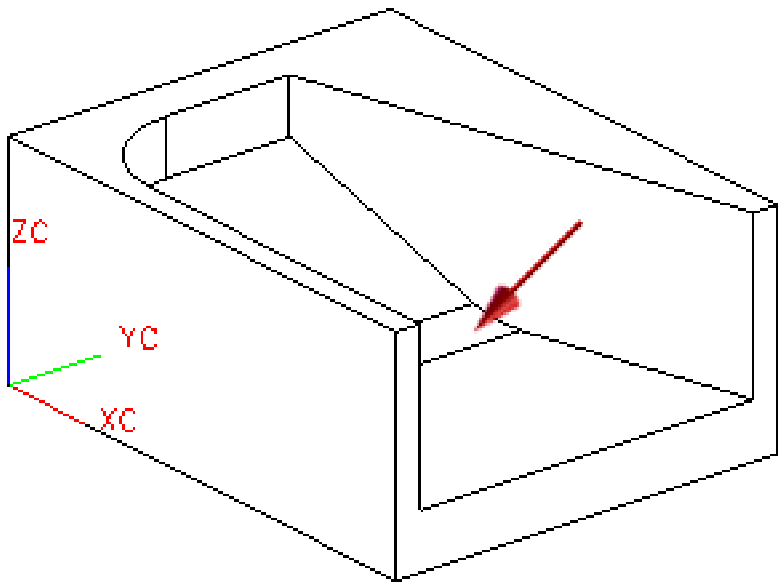
For the remainder of this activity, you will be prompted to change the **Drive** and **Part** surfaces only if the processor does not correctly select the proper surface. Each sub-operation will require you to select a new **Check** surface.

☐ Choose the **Check Surfaces** button.

☐ Change the **Check Stock** to **0**.

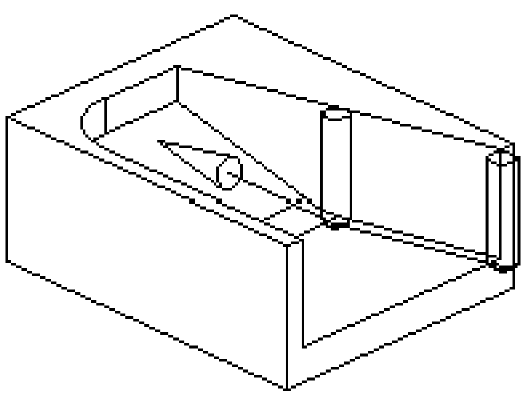
☐ Change the **Check** surface **Stopping Position** to **Ps-Cs Tangency**.

☐ Specify a new **Check** surface as shown.



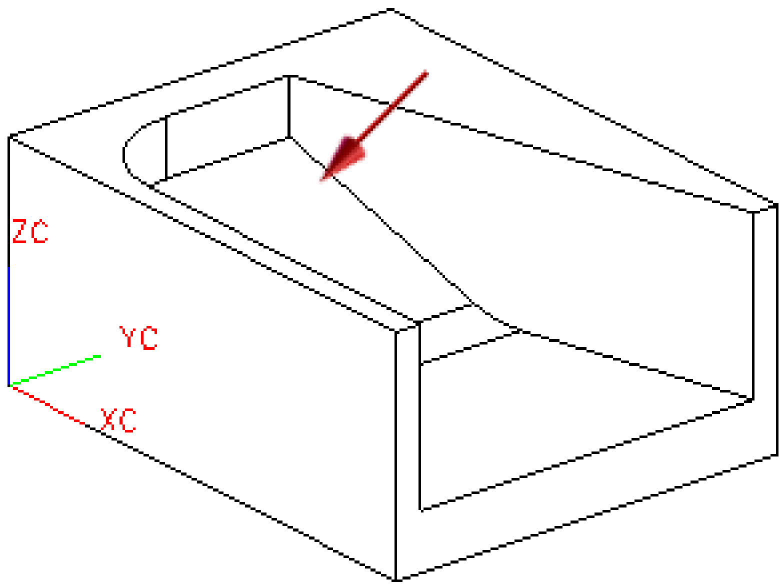
☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

The tool moves to the new position.



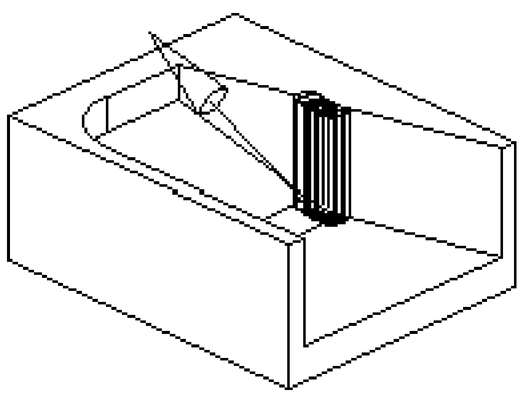
Note the status of the **Part Surface** to previous **Check** surface.

☐ Specify a new **Check** surface as shown.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

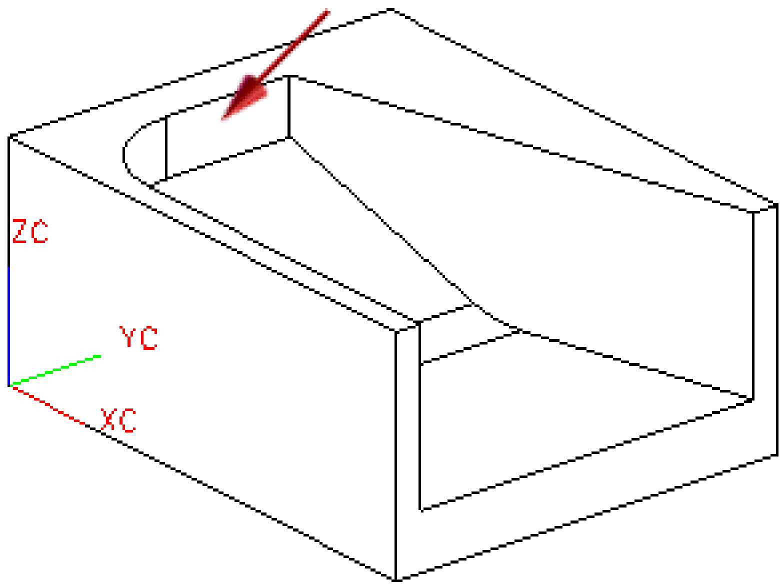
The tool moves to the new position.



Note that the Sequential Mill processor did not change the status of the **Drive** or **Part** surfaces.

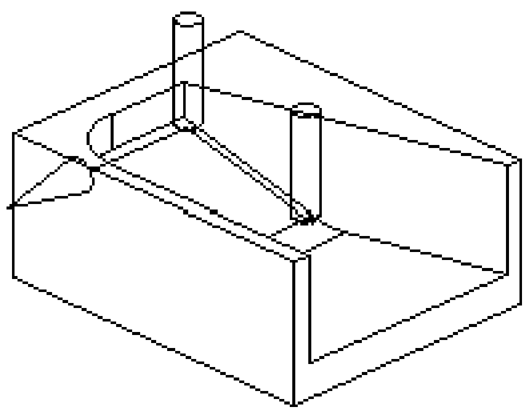
☐ Specify **Check** surface **Stopping Position** as **Near Side**.

☐ Specify a new **Check** surface as shown.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

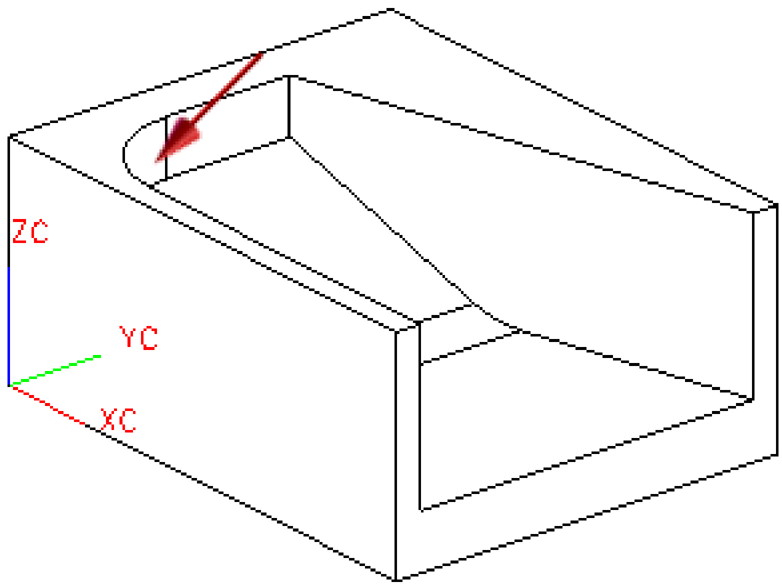
The tool moves to the new position.



Note that the Sequential Mill processor expects that the next **Part** surface will be the previous **Part** surface.

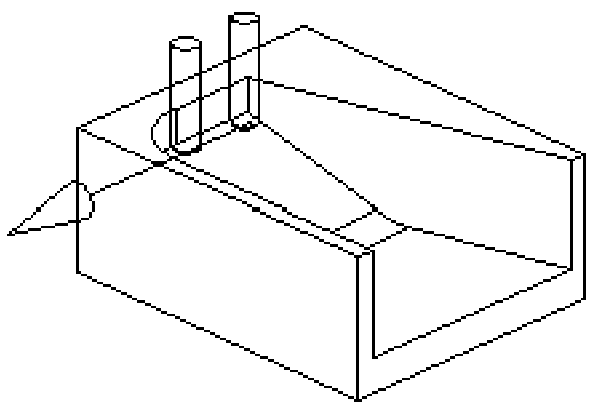
☐ Specify the **Check** surface **Stopping Position** as **Ds-Cs Tangency**.

☐ Specify a new **Check** surface as shown.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

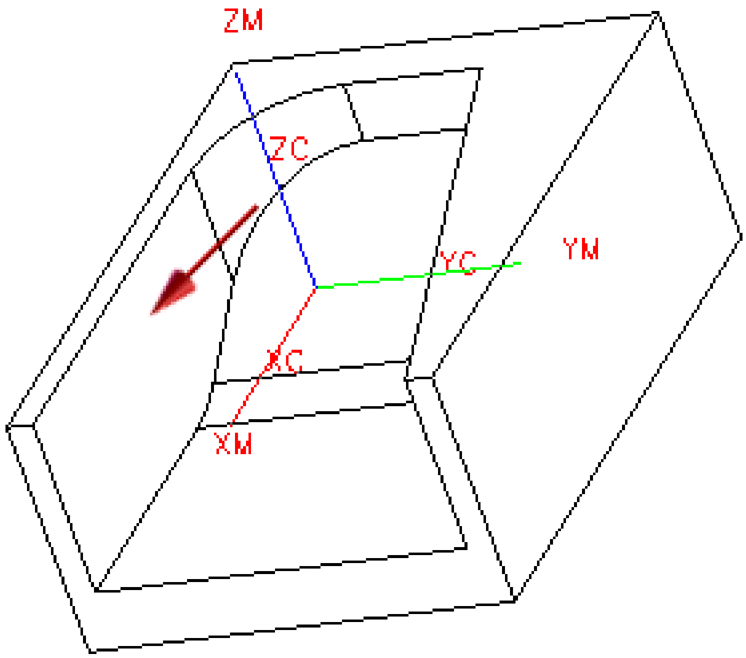
The tool moves to the new position.



Note that the status of the **Drive** or **Part** surfaces did not change.

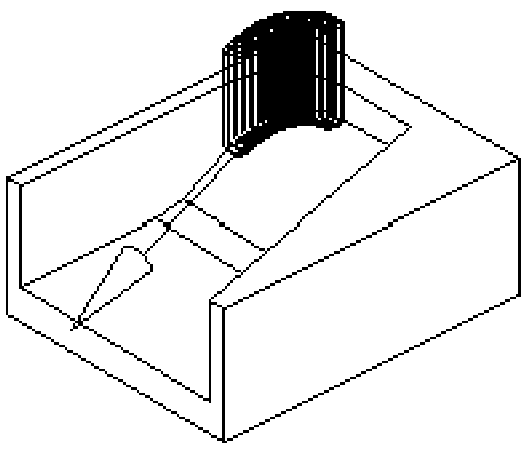
☐ Specify the **Check** surface **Stopping Position** as **Near Side**.

☐ Specify a new **Check** surface as shown.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

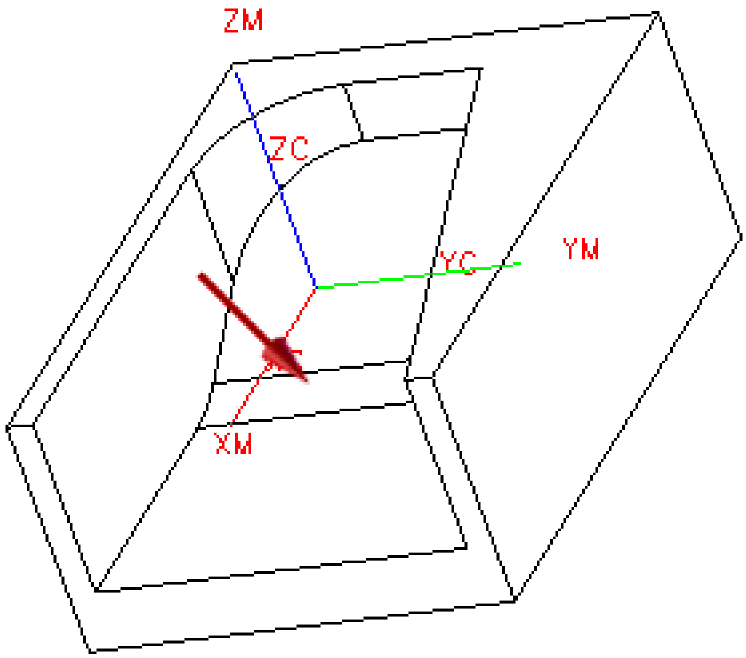
The tool moves to the new position.



Note the status of the **Drive** or **Part** surfaces did not change.

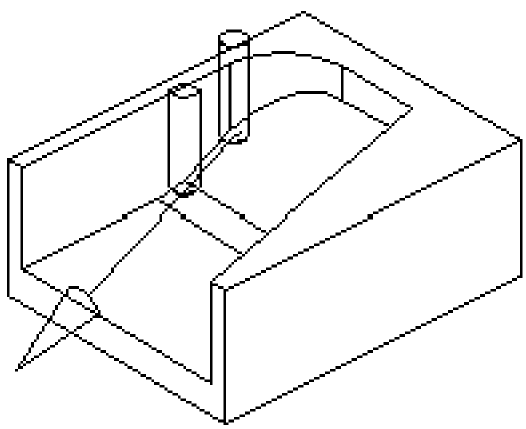
☐ Specify the **Check** surface **Stopping Position** as **Ps-Cs Tangency** .

☐ Specify a new **Check** surface.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

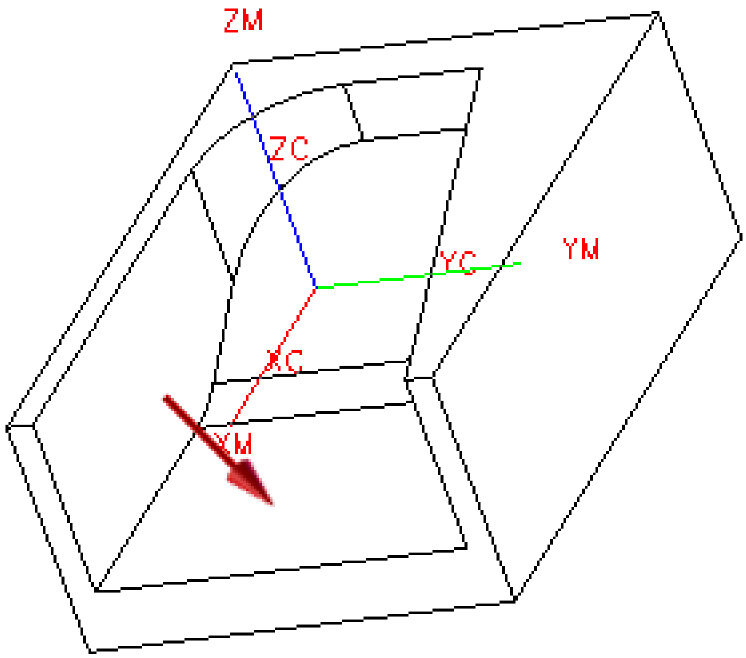
The tool moves to the new position.



Note the status of the **Drive** and **Part** surfaces changed.

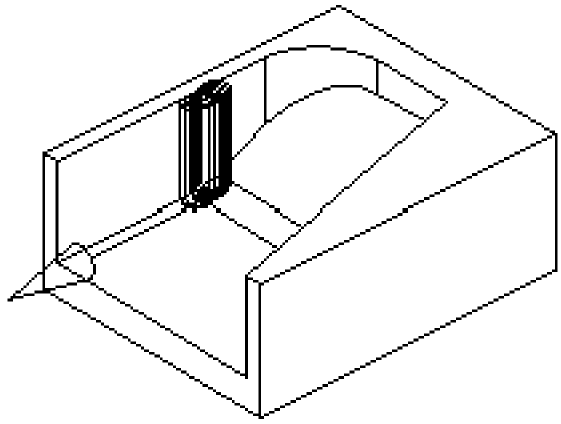
☐ Specify the **Check** surface **Stopping Position** as **Ps-Cs Tangency**.

☐ Specify a new **Check** surface as shown below.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

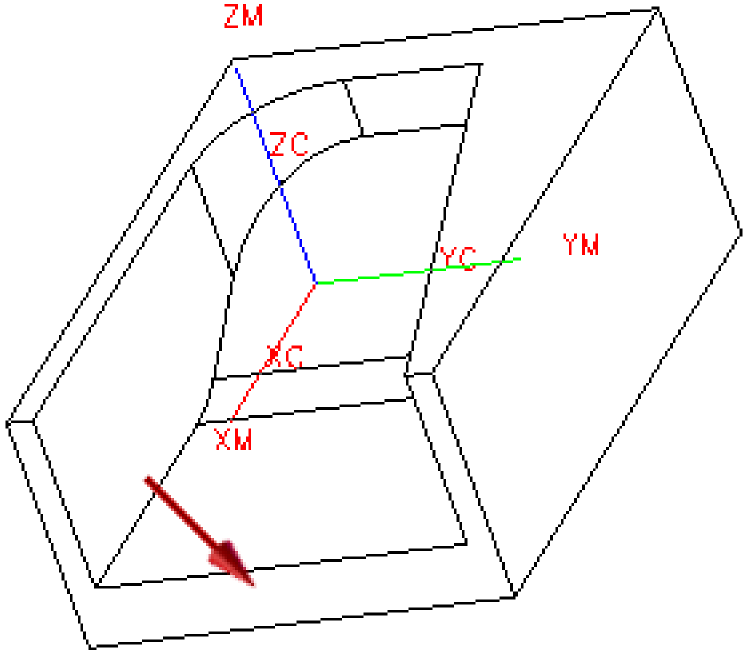
The tool moves to the new position.



The status of the **Drive** or **Part** surfaces did not change.

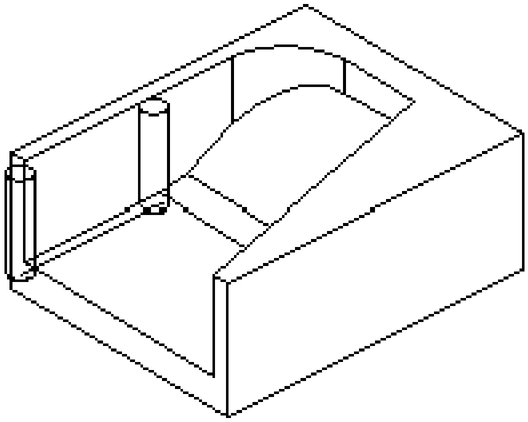
☐ Specify the **Check** surface **Stopping Position** as **Far Side**.

☐ Specify a new **Check** surface as shown.



☐ Return to the **Continuous Path Motion** dialog and choose **OK**.

The tool moves to the new position.



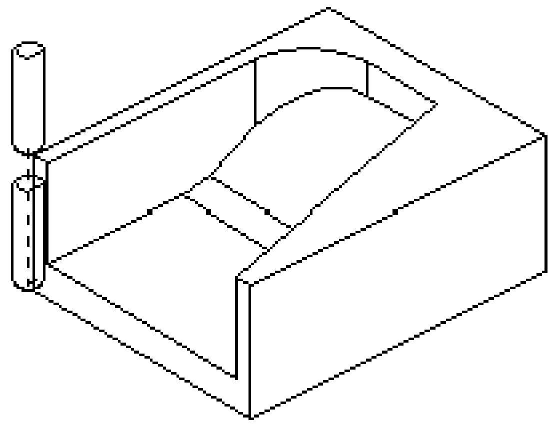
☐ Change **Cont. Path** to **Retract**.

The **Retract Motion** dialog is displayed.

☐ Change the **Retract Method** to **Vector Only** and then **+XC Axis**.

- ☐ Change the **Distance** to **.200**.
- ☐ Return to the **Retract Motion** dialog and choose **OK**.

  The tool retracts to the **Clearance Plane**.



- ☐ Choose **End Operation** and then **OK** to save the operation.

  The entire tool path is displayed.

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

Summary

Sequential Milling operations allow complete control of cutter movement and are useful in the finish machining of complex, multi-axis geometry. The more experienced programmer will use Sequential Milling techniques to simplify the creation of complex tool paths.

The following functions are used in Sequential Milling applications:

- Selecting of specific tool axis.
- Specifying tool starting and stopping positions based on contact with Part, Drive, and Check surfaces.

Lesson

3 *Sequential Mill Intermediate*

Purpose

Sequential Mill operations are flexible in the types of geometry that it recognizes. The use of solid as well as non-solid geometry enhances the flexibility of the types of parts that can be machined. This allows for the easy use of geometry imported from other systems without having to create solid geometry.

Objective

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

- Generate Sequential Mill tool paths using wire frame geometry.

Sequential Milling using non-solid geometry

Occasionally, geometry being machined was not originally created in NX. The geometry may have come from another CAD/CAM system, either through IGES, STEP or a direct translator. Sometimes the translation results in non-solid geometry such as sheets and faces.

It is not always necessary to reconstruct the geometry into a solid body to perform Sequential Mill operations.

In order to program non-solid geometry, Sequential Mill provides two types of geometry for selection, in addition to Face. They are:

- Curve Directrix allowing the selection of curves or face edges to drive the cutting tool; the Directrix parameter forces the calculation of a tabulated cylinder (which is normally parallel to the tool axis) to aid in positioning the tool.

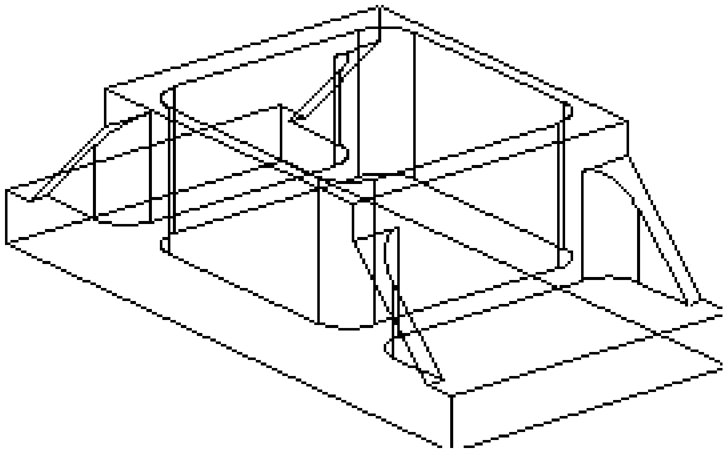
- Temporary Plane allowing the creation of a temporary plane to aid in positioning the tool.

Activity: Sequential Mill Machining of Non-solid Geometry

In this activity, you will machine wire frame geometry using Sequential Milling operation types.

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

☐ Open the part file **mam_wireframe_mfg**.



☐ Enter the **Manufacturing application**.

Step 2: Begin the Sequential Mill manufacturing process.

You will now create a Sequential Mill operation that will finish the front edge of the part. The tool has already been created for you.

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

The Create Operation dialog is displayed.

☐ If necessary, change the **Type** to **mill_multi-axis**.

☐ Choose **Sequential _Mill** as the **subtype**.

☐ Set the **Parent** objects as follows:

Program: **MACHINE_WIREFRAME**

Use Geometry: **WORKPIECE**

Use Tool: **EM-0.875–0–CARBIDE**

Use Method: **MILL_FINISH**

☐ Name the operation **SM_1**.

☐ Choose **OK**.

The Sequential Mill dialog is displayed.

Step 3: Create the Sub-operations.

You have specified the global settings. You will now create the individual sub-operations that will drive the tool around the edge of the flange.

- ☐ Choose **OK** in the Sequential Mill dialog.

  The **Engage Motion** sub-operation dialog is displayed. By default, the sub-operation dialog is set to **Engage**.

  You will establish the position and tool axis using a **Pt-to-Pt** sub-operation.

- ☐ Change the motion from **Engage** to **Pt to Pt**.

  The corresponding dialog changes to match **Point to Point** motion.

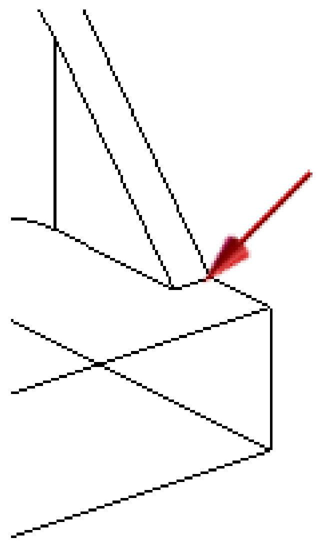
- ☐ Change the **Motion Method** from **Undefined** to **Point, Tool Axis**.

  The **Point Constructor** dialog is displayed. You will establish the tool position by using an offset from the part geometry.

- ☐ Change the **Offset** to **Rectangular**.

  Note that using a Rectangular Offset allows an X, Y and Z delta offset from the point chosen.

- ☐ Choose the end point, as shown.



The **Point Constructor** dialog has changed to allow the input of delta values from the point selected.

☐ Key in the following values:

Delta XC 1.00

Delta YC 1.00

Delta ZC 1.00

☐ Choose **OK**.

The **Vector Constructor** dialog is displayed.

You will accept the default tool axis vector of 0,0,1 which is the same as the Z coordinate of the WCS.

☐ Choose **OK** in the **Vector Constructor** dialog.

The **Point to Point** sub-operation is complete. By choosing **OK,** the sub-operation will be placed in the sub-op list and Sequential Mill will be ready to create the next sub-operation.

☐ Choose **OK** in the sub-operation dialog.

You will now define the **Engage** component.

☐ Change to an **Engage** sub-operation.

The **Engage Motion** dialog is displayed.

The **Engage** dialog requires **Drive, Part** and **Check** geometry.

You will specify the geometry and then the **Engage** method.

☐ Choose the **Geometry** button from the **Engage Motion** dialog.

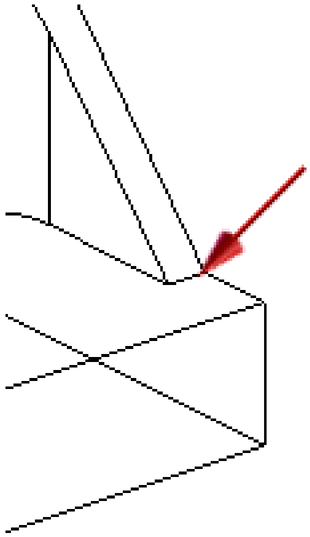
The **Engage Geometry** dialog is displayed.

The defaults are set to **Drive** geometry, the **Type** is **Face** and the **Stopping Position** is **Near Side**.

You will now select the **Drive** geometry, prior to selecting the geometry you will change the **Type** of geometry selected from **Face** to **Curve, Directrix**.

☐ Change the **Type** from **Face** to **Curve, Directrix**.

☐ Choose the curve as shown.



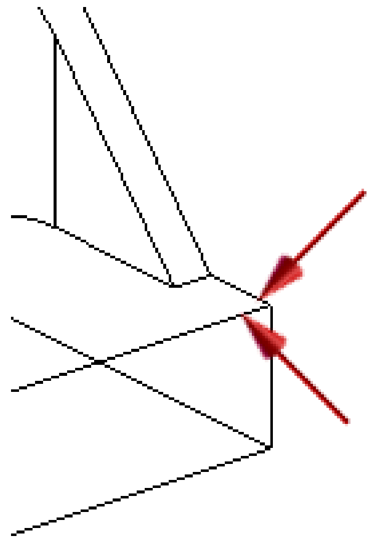
As soon as the **Drive** geometry is selected, the dialog advances to enable **Part** geometry selection. You will now select the **Part** geometry.

☐ Change the **Type** to **Temporary Plane**.

The **Plane Selection** method dialog is displayed. There are numerous options in which a temporary plane can be created. You will use the **Two Lines** method.

☐ Choose the **Two Lines** button.

☐ Select the two lines as shown.

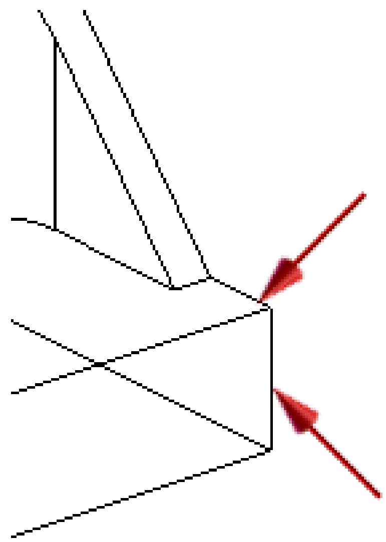


As soon as the **Part** geometry is selected, the dialog advances to enable **Check** geometry selection. You will now select the **Check** geometry.

☐ Choose the **Temporary Plane** option.

☐ Choose the **Two Lines** button from the **Plane** selection dialog.

☐ Select the two lines as shown.



☐ Choose **OK**.

The **Engage** sub-operation is created and placed in the sub-operation list. You are now ready to create the **Continuous Path Motion** sub-operations.

The defaults for the new **Drive** and **Part** geometry are correct.

☐ Choose the **Check Surfaces** button.

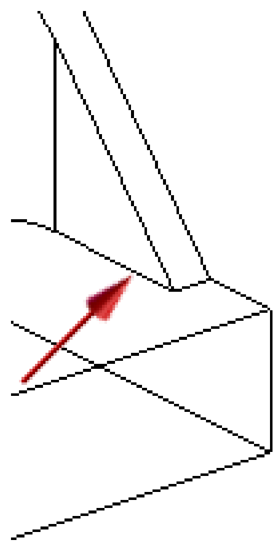
The **Check Surfaces** dialog is displayed.

Before selecting the new **Check** geometry you need to determine the proper **Stopping Position** of the cutting tool.

☐ Change the **Stopping Position** from **Near Side** to **Far Side**.

☐ Change the **Type** from **Temporary Plane** to **Curve, Directrix**.

☐ Choose the curve as shown.



As you select the first **Check** geometry element, the dialog advances so that you can select the next item for **Check** geometry.

☐ Choose **OK** in the **Check Surface** dialog.

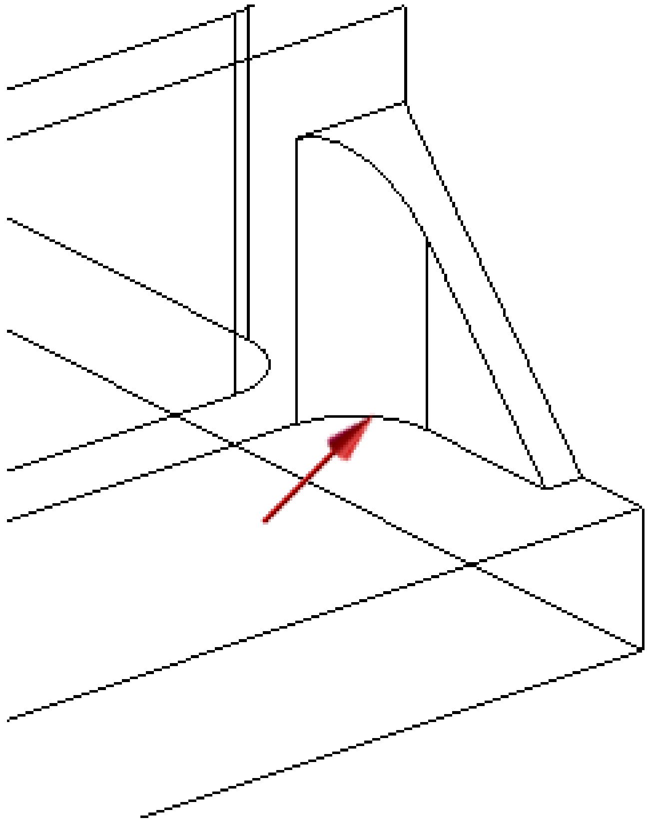
☐ Choose **OK** to create the sub-operation.

You will now create the **Continuous Path Motions** to complete the area to be machined.

☐ Choose the **Check Surfaces** button.

☐ Change the Stopping Position to **Ds-Cs Tangency**.

☐ Choose the curve as shown.



☐ Choose **OK** in the **Check Surface** dialog.

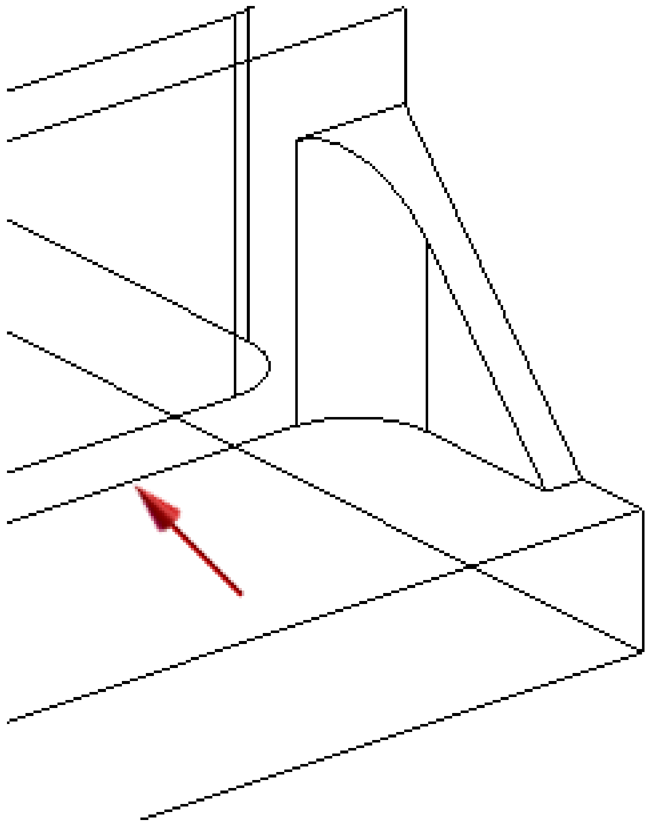
☐ Choose **OK** to create the new sub-operation.

There are now four sub-operations in the sub-operation list.

Continue the tool path motion by selection of the next element of **Check** geometry.

☐ Choose the **Check Surfaces** button.

☐ Choose the curve as shown.



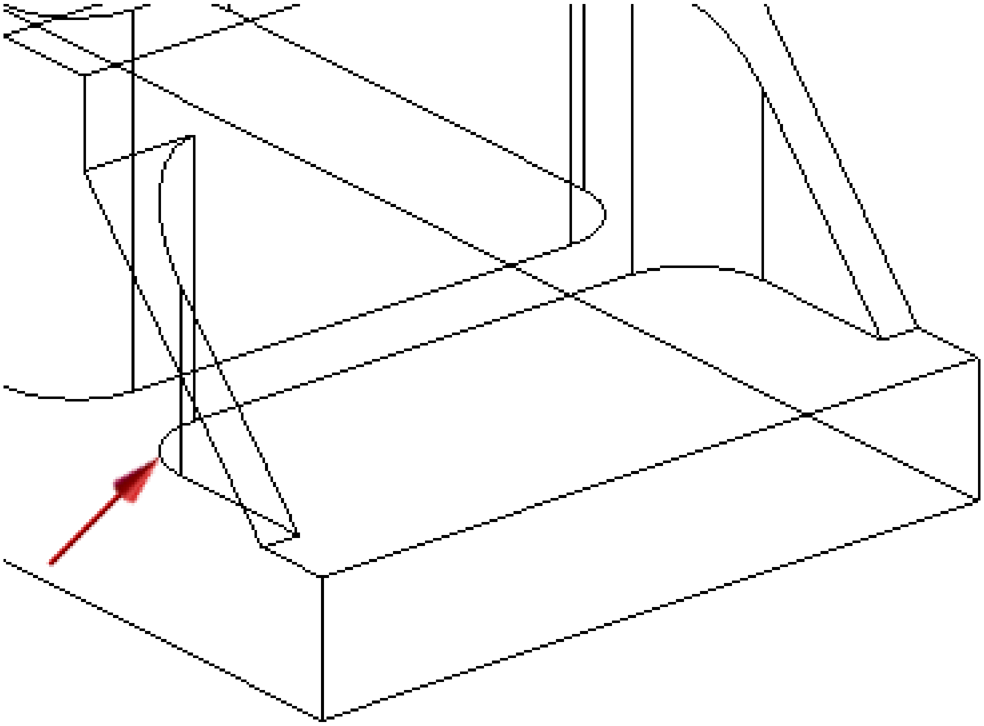
☐ Choose **OK** in the **Check Surfaces** dialog.

☐ Choose **OK** to create the new sub-operation.

The fifth sub-operation, **5 cpm**, has been created. Continue by selecting the next curve.

☐ Choose the **Check Surfaces** button.

☐ Choose the curve as shown.



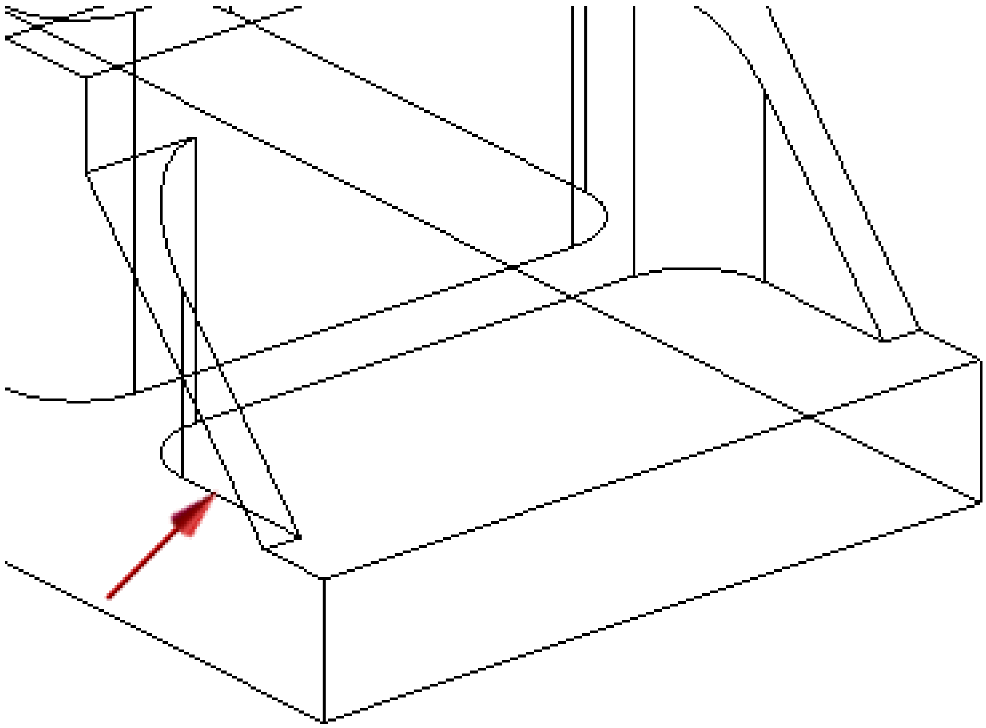
☐ Choose **OK** in the **Check Surfaces** dialog.

☐ Choose **OK** to create the new sub-operation.

The sixth sub-operation, **6 cpm**, has been created. Continue by selecting the next curve.

☐ Choose the **Check Surfaces** button.

☐ Choose the curve as shown.



☐ Choose **OK** in the **Check Surfaces** dialog.

☐ Choose **OK** to create the new sub-operation.

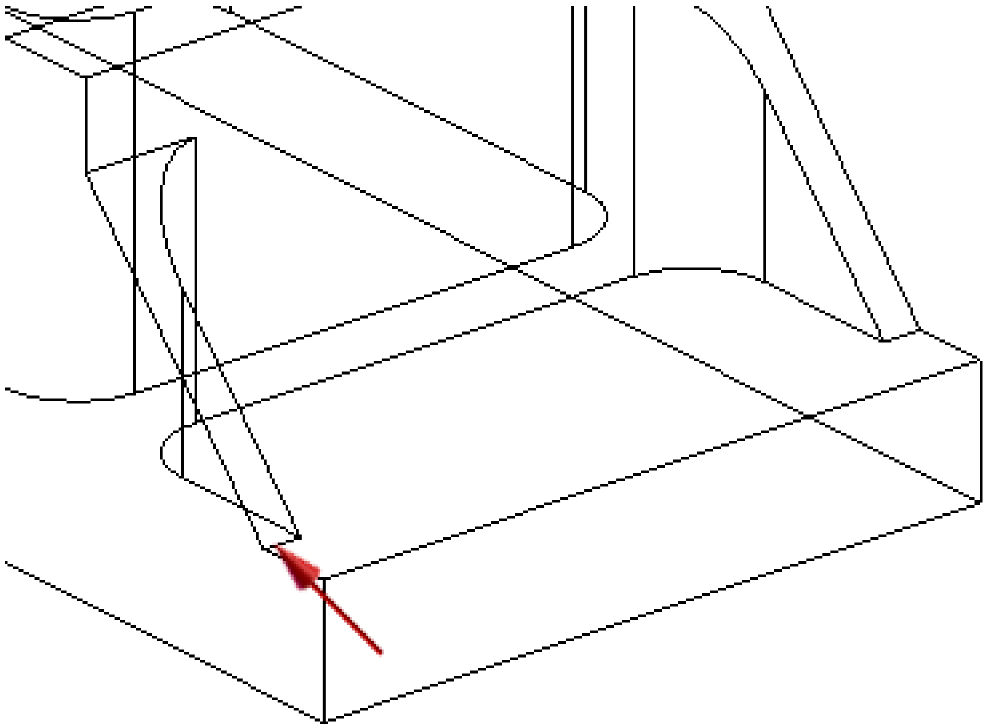
The seventh sub-operation, **7 cpm**, has been created. Continue by selecting the next curve.

For the next curve, you will want the cutter to position to the far side of the curve.

☐ Choose the **Check Surfaces** button.

☐ Change the **Stopping Position** to **Far Side**.

☐ Choose the curve as shown.



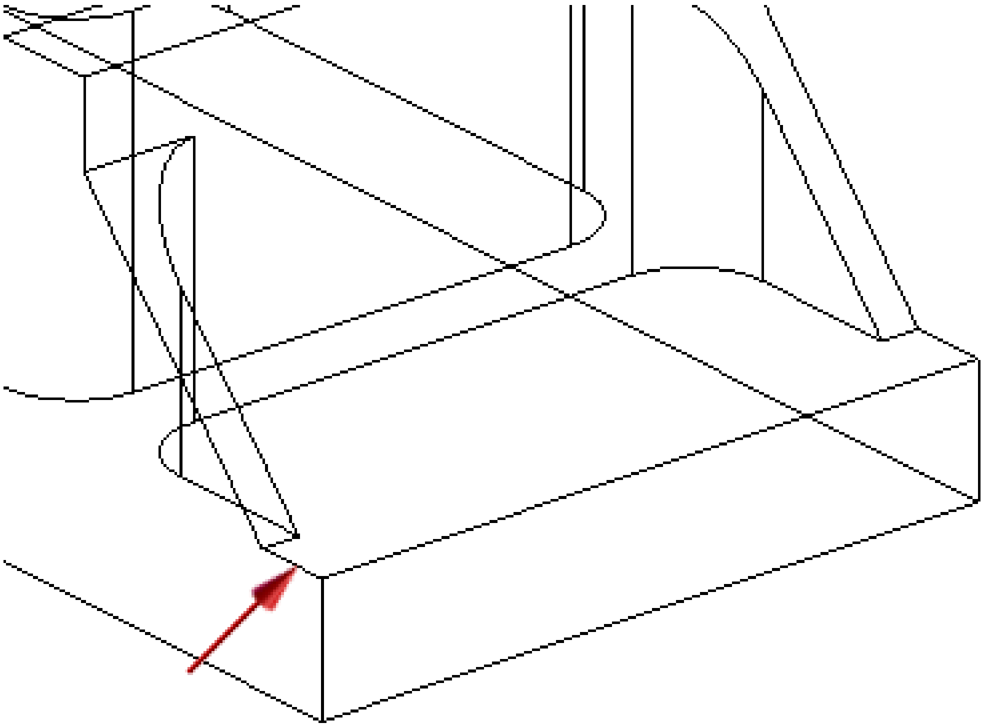
☐ Choose **OK** on the **Check Surfaces** dialog.

☐ Choose **OK** to create the new sub-operation.

The eighth sub-operation, **8 cpm**, has been created. Continue by selecting the final curve for **Continuous Path Motion**.

☐ Choose the **Check Surfaces** button.

☐ Choose the curve as shown.



☐ Choose **OK** in the **Check Surfaces** dialog.

☐ Choose **OK** to create the new sub-operation.

You have completed the selection of geometry for continuous path motion and need to retract the cutter.

☐ Change from **Cont. Path** to **Retract** motion.

☐ Choose the **Retract Method** button.

☐ Change the **Retract Method** from **None** to **Vector Only**.

☐ Key in the following values in the **Vector Constructor** dialog:

I = 1.000

J = –1.000

K = 1.000

☐ Choose **OK** in the **Vector Constructor** dialog.

☐ Key in **0.500** in the **Distance** field of the **Retract Method** dialog.

☐ Choose **OK** in the **Retract Method** dialog.

- ☐ Choose **OK** to create the retract sub-operation.

  The operation is now complete.

- ☐ Choose the **End Operation** button to complete the operation.
- ☐ Choose **OK** in the **End Operation** dialog.
- ☐ Close the file without saving.

Summary

The ability of utilizing wire frame type geometry in Sequential Milling operations, affords the flexibility of the types of parts that can be machined. The use of wire frame geometry allows the import of data from other systems without having to create solid geometry.

In this lesson you:

- Created Engage tool motion to engage the part.
- Created Continuous Path Motion to drive the cutter along various geometric entities.
- Created Retract tool motion to disengage the tool from the part.

Lesson

4 Sequential Mill Advanced

Purpose

Some of the more advanced features of Sequential Milling allow for multiple passes and control of the tool axis. These options allow for increased flexibility for roughing and finishing operations.

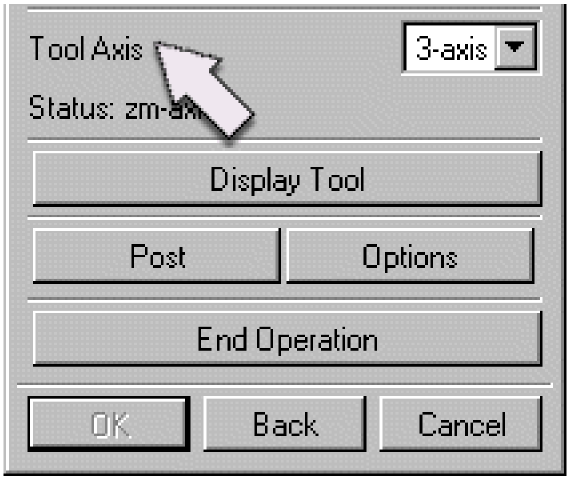
Objective

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

- Use standard and nested loops for creating roughing and finishing passes.
- Completely control the tool axis in 3, 4 and 5-axis applications.

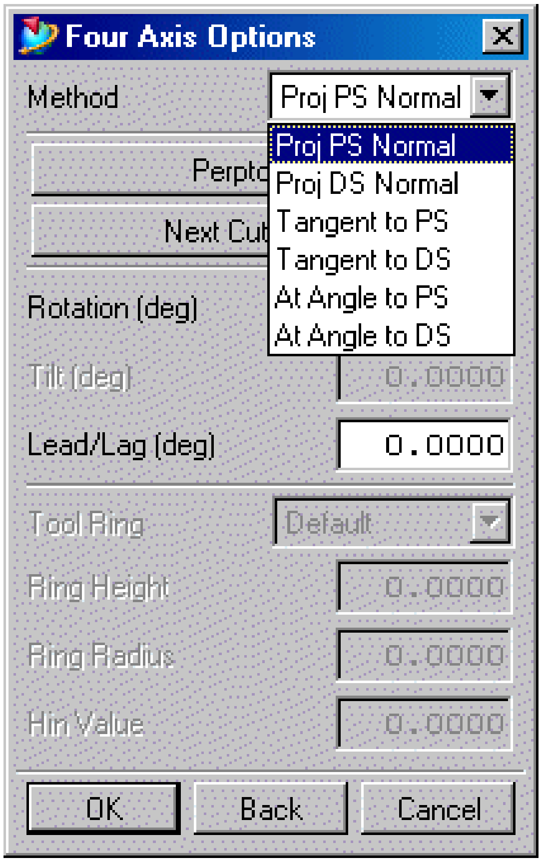
Tool Axis Control

In Sequential Mill, you define the tool axis by first specifying 3, 4 or 5-axis tool positioning which is found on the Engage and Continuous Path Motion dialogs.



3-axis allows you to specify the ZM axis or a fixed vector.

4-axis allows the tool to remain perpendicular to a specified vector and can be further adjusted by:



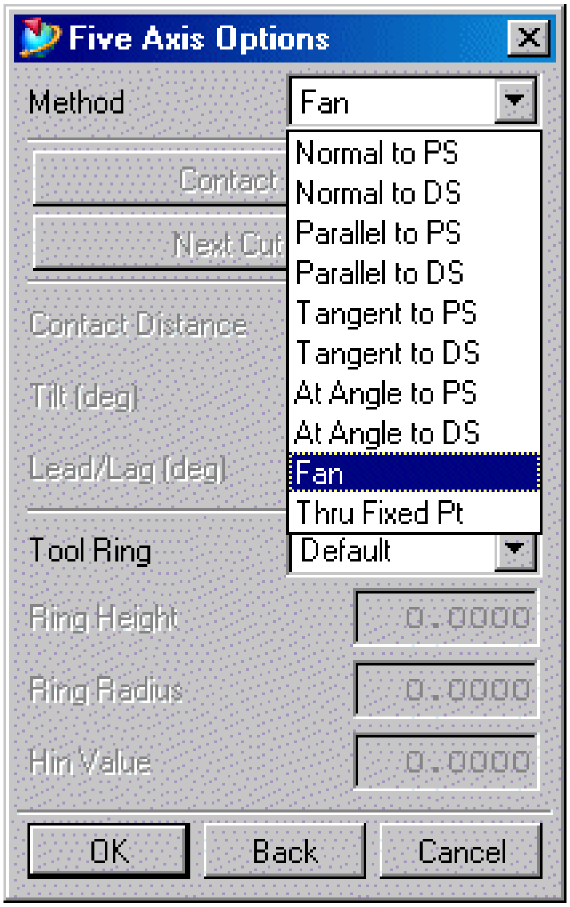
- another vector - projected PS (or DS) Normal
- A "ring" height on the tool - tangent to PS (or DS)
- An angle - at angle to PS (or DS)

Project Part Surface (or Drive Surface) Normal indicates that the tool axis is calculated by rotating the surface normal by a lead or lag angle, projecting the resulting vector onto a plane perpendicular to the specified Perpto vector, and then rotating it in that plane by a specified angle. This option causes the Perpto vector and the Next Cut Direction buttons to appear.

Tangent To PS (or DS) indicates that the side of the tool is tangent to the designated surface while the tool axis remains perpendicular to the specified Perpto vector.

At Angle To Ps (or Ds) indicates the tool axis maintains a fixed angle with the designated surface normal while remaining perpendicular to the specified Perpto vector.

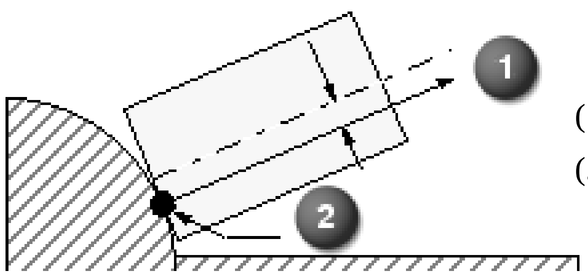
5-axis allows the tool axis to :



- remain normal, parallel or angled to the Part or Drive surfaces
- fan between surfaces
- pivot from a point

5–Axis Tool Axis Control	
Variable Contour	**Sequential Mill**
Toward or Away From Point	**Thru Fixed Point**
Normal to Part	**Normal to PS**
Normal to Drive	**Normal to DS**
Swarf Drive	**Parallel to PS** **Parallel to DS**
Relative to Drive	**At Angle to DS** **At Angle to PS**
–	**Tangent to PS**
–	**Fan**
–	**Tangent to DS**

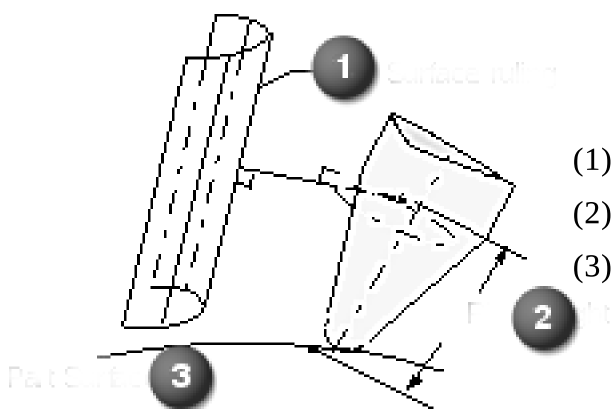
Normal To Ps (or Ds) causes the tool axis to remain perpendicular to the specified surface. This generally involves keeping the center of the bottom of the tool in contact with the surface. Optionally, you can offset the contact point from the bottom center of the tool.



(1) Surface normal at contact point

(2) “new” contact point

Parallel to Ps (or Ds) causes the side of the tool to be kept parallel to the surface rulings at the contact point. A ring on the tool must be specified to indicate where the side of the tool must touch the surface.

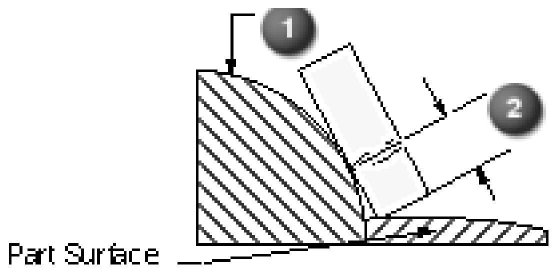


(1) Drive Surface ruling

(2) Ring height

(3) Part Surface

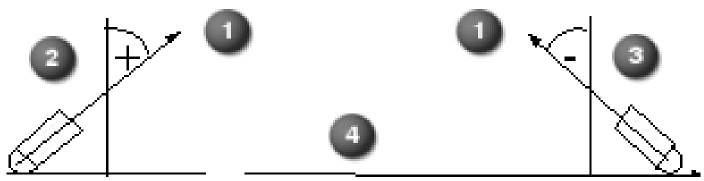
Tangent to Ps (or Ds) causes the side of the tool to be tangent to the specified surface while the tool axis stays perpendicular to the current direction of motion. You must specify a ring height.



(1) Drive Surface

(2) Ring height

At Angle to Ps (or Ds) causes the tool axis to maintain a fixed angle (Tilt) with the surface normal and a fixed angle with the current direction of motion (a Lead or Lag angle).



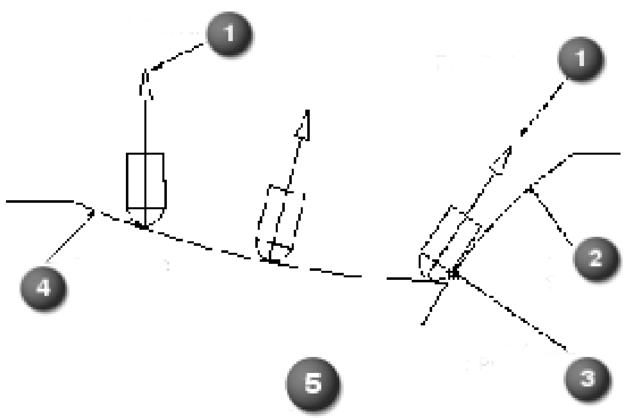
(1) Tool Axis

(2) Lead

(3) Lag

(4) Direction of motion

Fanning is an even distribution of tool axis change from the start to the stop position. This can be useful, for example, when the tool is canted at either or both positions.



(1) Final Tool Axis

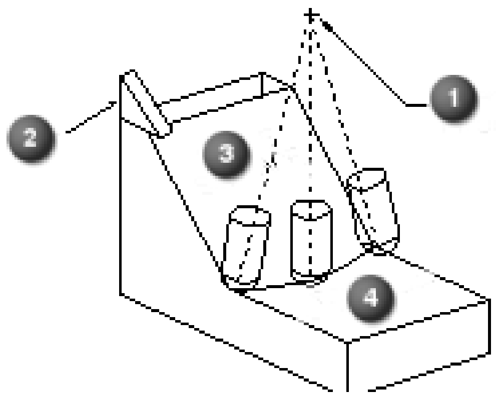
(2) Check Surface

(3) Check Surface contact point

(4) Part Surface

(5) 5–Axis Fanning

Thru Fixed Point indicates that the tool axis always lies along the line joining the tool end tip and a user-defined point. Use the *Point Constructor* dialog to define the point.



(1) User defined pivot point

(2) Check Surface

(3) Drive Surface

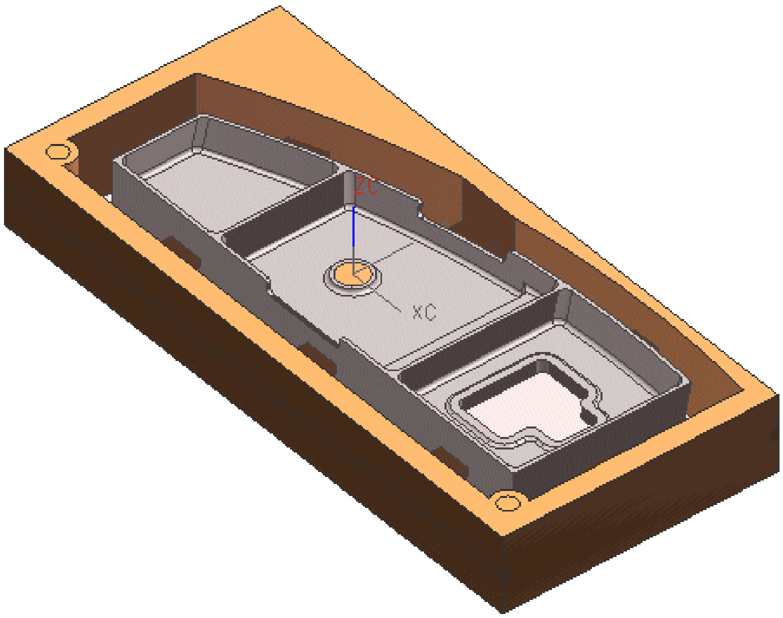
(4) Part Surface

Activity: Sequential Mill Five-Axis Fan Motion

In this activity, you will create a Sequential Milling operation to finish the walls of a pocket on an aircraft structural component.

Step 1: Open, rename and examine the part file.

☐ **Open** the part file **mam_spar_mfg**.



The spar is cut from a forged block of aluminum and is held in place by clamps along the slits that run the length of the block on either side. Dowel pins are used to locate the block.

The orange material represents the "window frame" portion of the block. Small tabs run from it to the part to secure it during machining.

This part has been partially machined. You will first examine the machining progress made to this point.

☐ Rename the part *****_spar_mfg**.

☐ Enter the **Manufacturing application**.

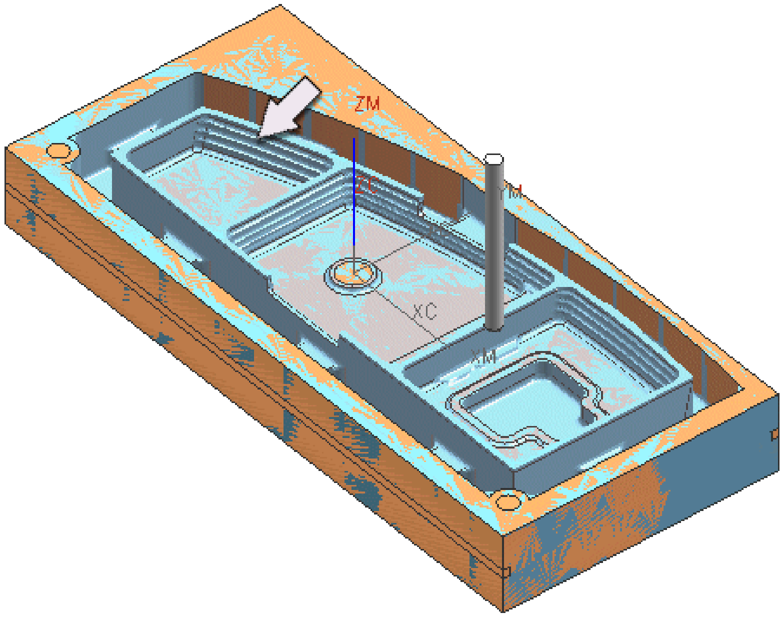
☐ Choose the Operation Navigator tab from the toolbar.

☐ Highlight the **SIDE_1** program object and then use **MB3,** select **Tool Path** and then **Verify.**

☐ Select the **3D Dynamic** tab from the Tool Path Visualization dialog.

☐ Choose the Play Forward button from the bottom of the dialog.

The In-Process work piece of the part is represented. You will begin machining the left most pocket in the part.



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

Step 2: Create the Sequential Mill Operation.

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

The Create Operation dialog is displayed.

☐ If necessary, change the Type to **mill_multi-axis**.

☐ Choose **Sequential_Mill** as the subtype.

☐ Set the Parent objects as follows:

Program: **FINISH_1**

Use Geometry: **PART_AND_BLANK**

Use Tool: **EM-.5–.130–CARBIDE**

Use Method: **MILL_FINISH**

Name: **SM_FINISH_WALLS_POCKET_1**

☐ Choose **OK**.

The Sequential Mill dialog is displayed.

Step 3: Set Tool Display options and create a Point to Point Motion.

You will now set the tool display options, which will make the tool easier to visualize.

☐ Choose the **Display Options** button.

The Display Options dialog is displayed.

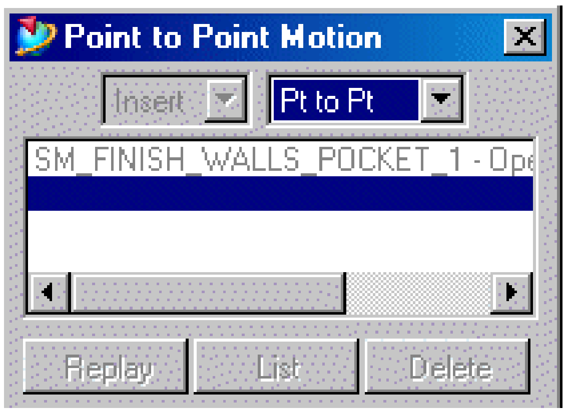
☐ Change the Tool Display to **3-D** and the Path Display Speed to **9**.

☐ Choose **OK** twice.

The Engage Motion dialog is displayed.

You will now establish the tool location and axis by using a **Point to Point** sub-operation.

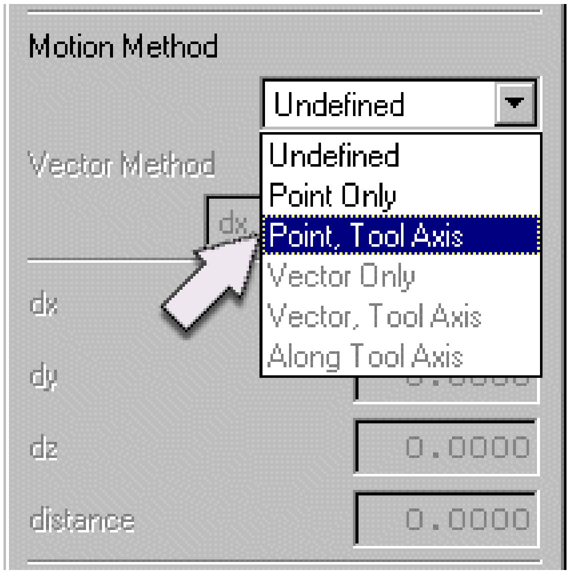
☐ Change the motion from **Engage** to **Pt to Pt**.



The corresponding dialog changes to match **Point to Point** motion.

You will now establish the tool position, specifying both the position of the tool and the tool axis.

☐ Change the Motion Method from **Undefined** to **Point, Tool Axis**.



The Point Constructor dialog is displayed.

☐ Key in the following values for the Base Point:

XC -5.00

YC 0.00

ZC 2.00

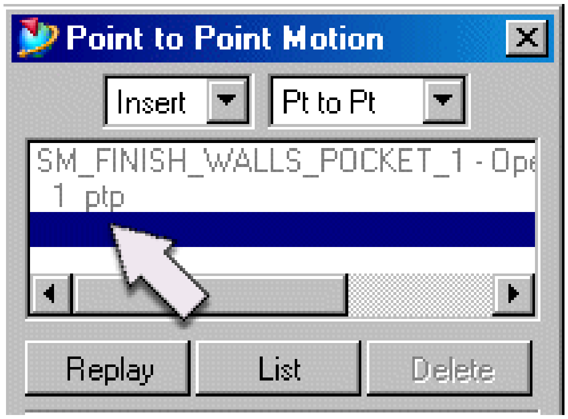
☐ Choose **OK**.

The Vector Constructor dialog is displayed.

You will accept the default tool axis vector of 0,0,1 which is the same as the Z coordinate of the WCS.

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

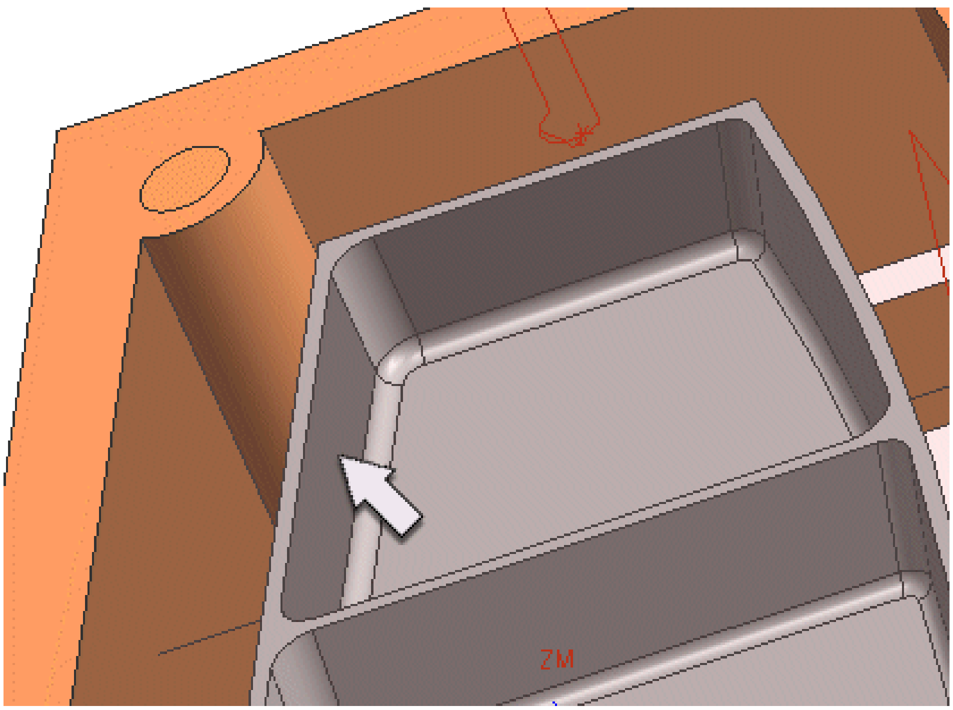
☐ Choose **OK** to accept the first sub-operation.



The first sub-operation, **1 ptp** is created and inserted into the sub-operation list.

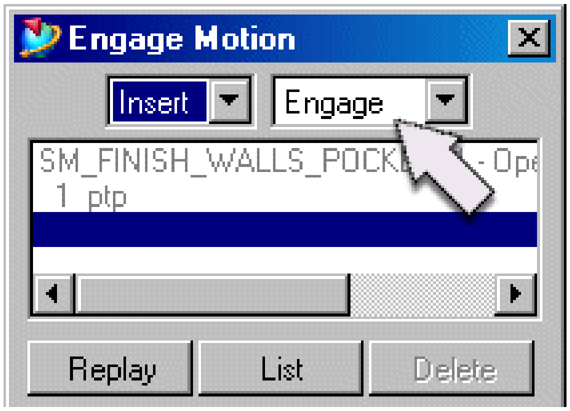
Step 4: Create the Engage Motion.

A best practice is to establish a cutting tool along a straight wall as well as to feed into the wall away from a corner to eliminate tool chatter. You will engage the wall as shown.



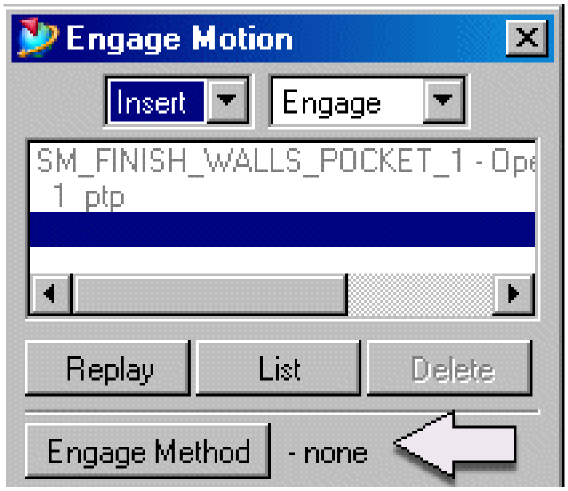
You will now define the Engage component.

☐ Change to an **Engage** sub-operation.



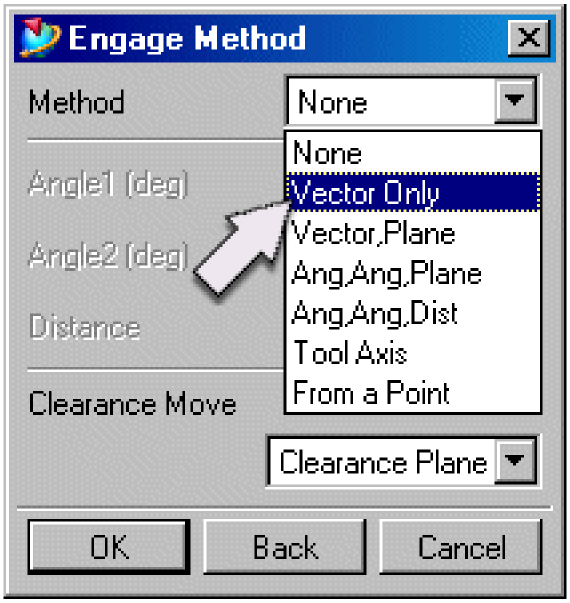
The Engage Motion dialog is displayed.

☐ Choose the Engage Method button.



The Engage Method dialog is displayed.

☐ Change the Method from **None** to **Vector Only**.



The Vector Constructor dialog is displayed.

☐ Key in the following values:

I= 0.000

J= -1.000

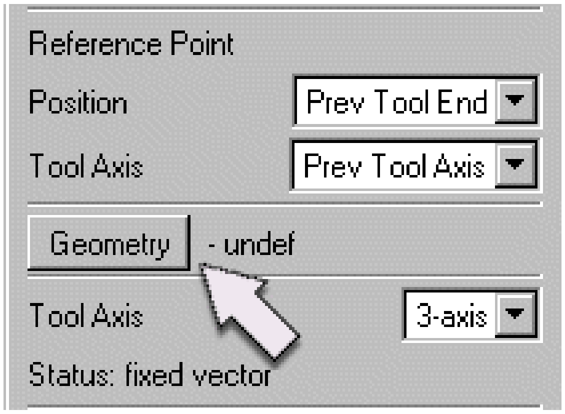
K=-.500

☐ Choose **OK**.

☐ Key in **2.00** in the **Distance** field of the Engage Method dialog.

☐ Choose **OK**.

☐ Choose the Geometry button from the Engage Motion dialog.

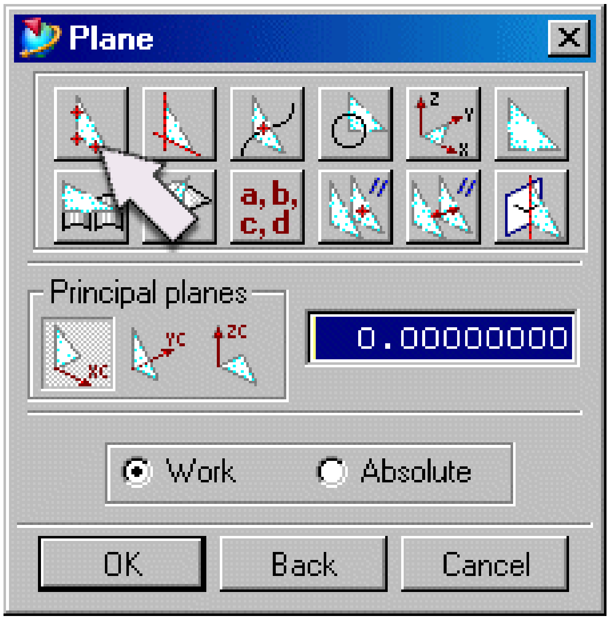


The Engage Geometry dialog is displayed.

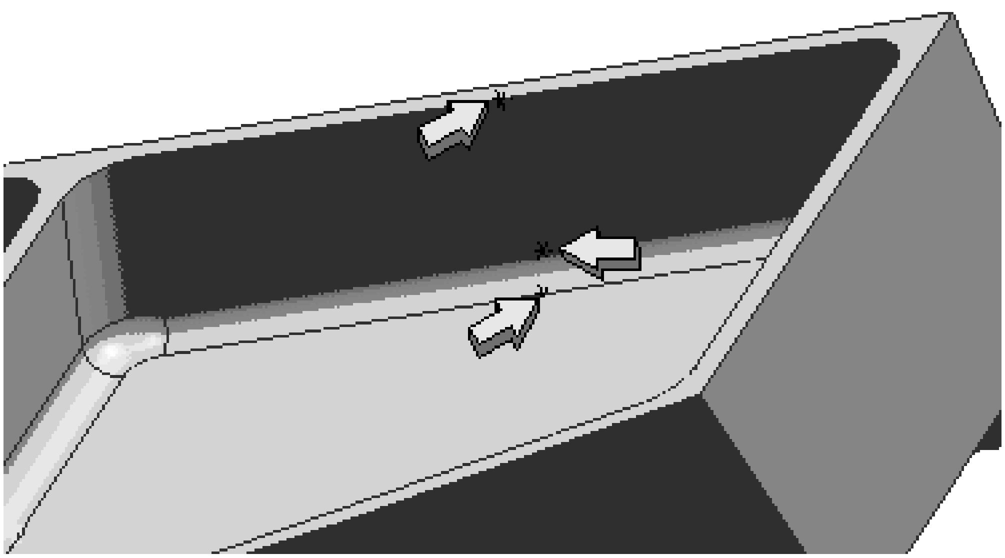
You will first create a temporary check plane as the Drive geometry using the Three Points option for plane creation.

☐ In the Engage Geometry dialog, change the Type from **Face** to **Temporary Plane**.

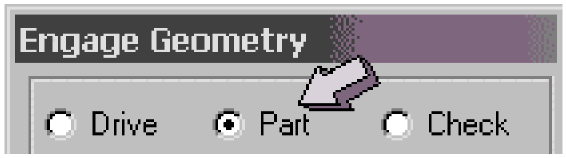
☐ Choose the Three Points option from the Plane dialog.



☐ Choose the three control points as shown. Note that in the following views, the part has been rotated 180 degrees to facilitate looking at the wall being created. The stock and check geometry have been removed from the view for purposes of clarity.

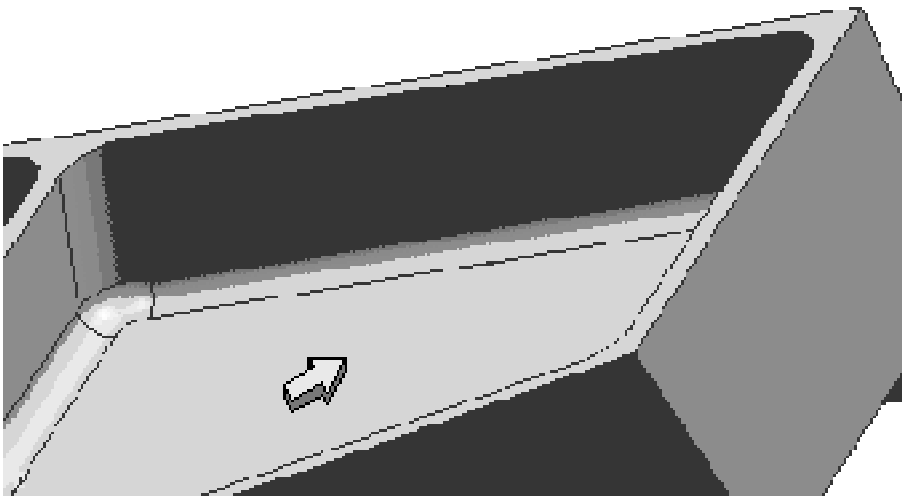


As the last point is selected, the geometry selection advances to Part geometry.

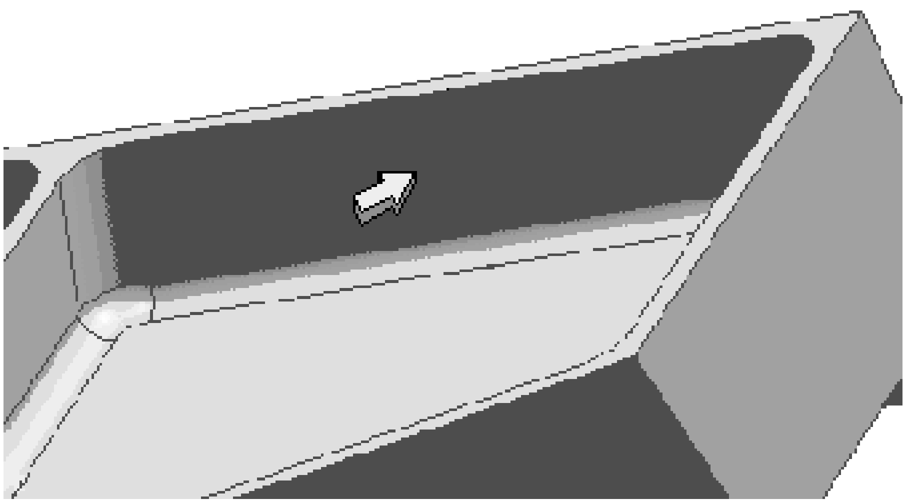


☐ Change the Type back to **Face**.

☐ Choose the bottom face of the pocket as the Part geometry.

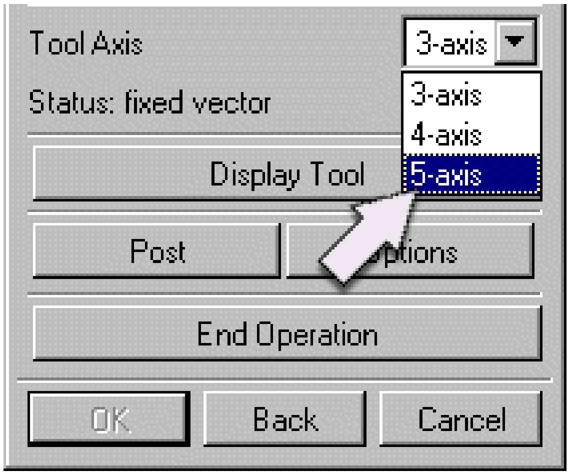


☐ Choose the wall face as the Check geometry.



After selecting the Check geometry, the Engage Motion dialog is displayed. Before proceeding any further you will want to change the Tool Axis to 5-axis fan motion.

- ☐ Change the Tool Axis from 3-axis to 5-axis.



The Five Axis Option dialog is displayed. Notice that the Method defaults to **Fan**, which is acceptable in this instance.

- ☐ Choose **OK** in the Five Axis Options dialog.

- ☐ Choose **OK** to create the Engage sub-operation.

  You are now ready to create the first Continuous Path Motion.

Step 5: Create the first Continuous Path Motion.

The radii in the pocket corners are slightly larger than the tool radius and allows the opportunity to drive the corner fillet with less tool chatter.

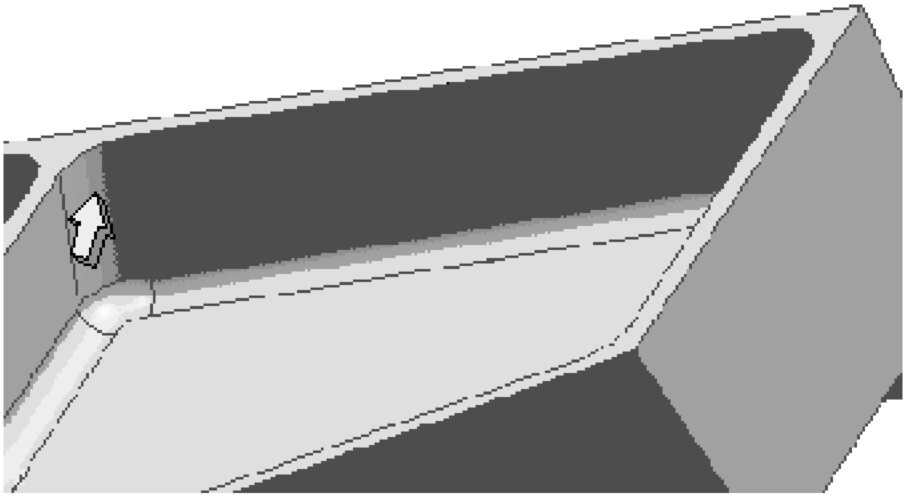
Continuous Path Motion is the default as the next sub-operation type. You will need to choose the fillet as the next Check surface.

- ☐ Change the Drive Surface to **Previous Cs**.

- ☐ In the Continuous Path Motion dialog, choose the **Check Surfaces** button.

- ☐ In the Check Surfaces dialog, change the Stopping Position to **Ds-Cs Tangency**.

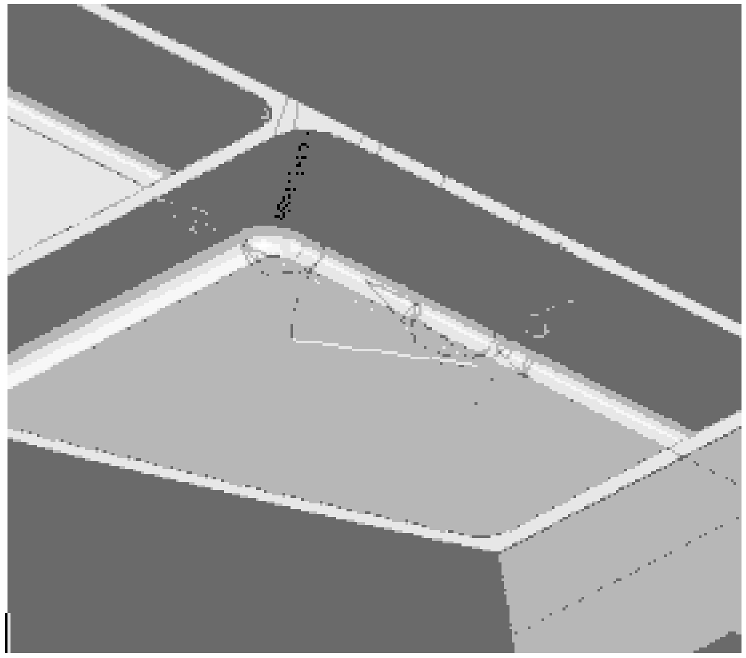
☐ Choose the corner fillet surface as shown.



☐ Choose **OK** in the Check Surfaces dialog.

☐ Choose **OK** in the Continuous Path Motion dialog.

The tool drives into the corner and sub-operation 3 is created.



☐ Choose the Check Surfaces button.

☐ Choose the next surface in line.

☐ Choose **OK** until the next sub-operation is created.

Step 6: Finish the operation.

- ☐ Continue to drive around the inner wall of the pocket, using the next surface in line as the new Check surface.

- ☐ When you reach the original surface that you used for engaging the part, drive past the temporary plane made up of control points on the edges of the surfaces. This should prevent any scallops from being left on the wall.

- ☐ Retract the tool from the pocket and end the operation.

- ☐ Save the part file.

  You finish machined the wall of the pocket. One of the walls of the pocket is at an extreme closed angle. Extra stock was left on that wall.

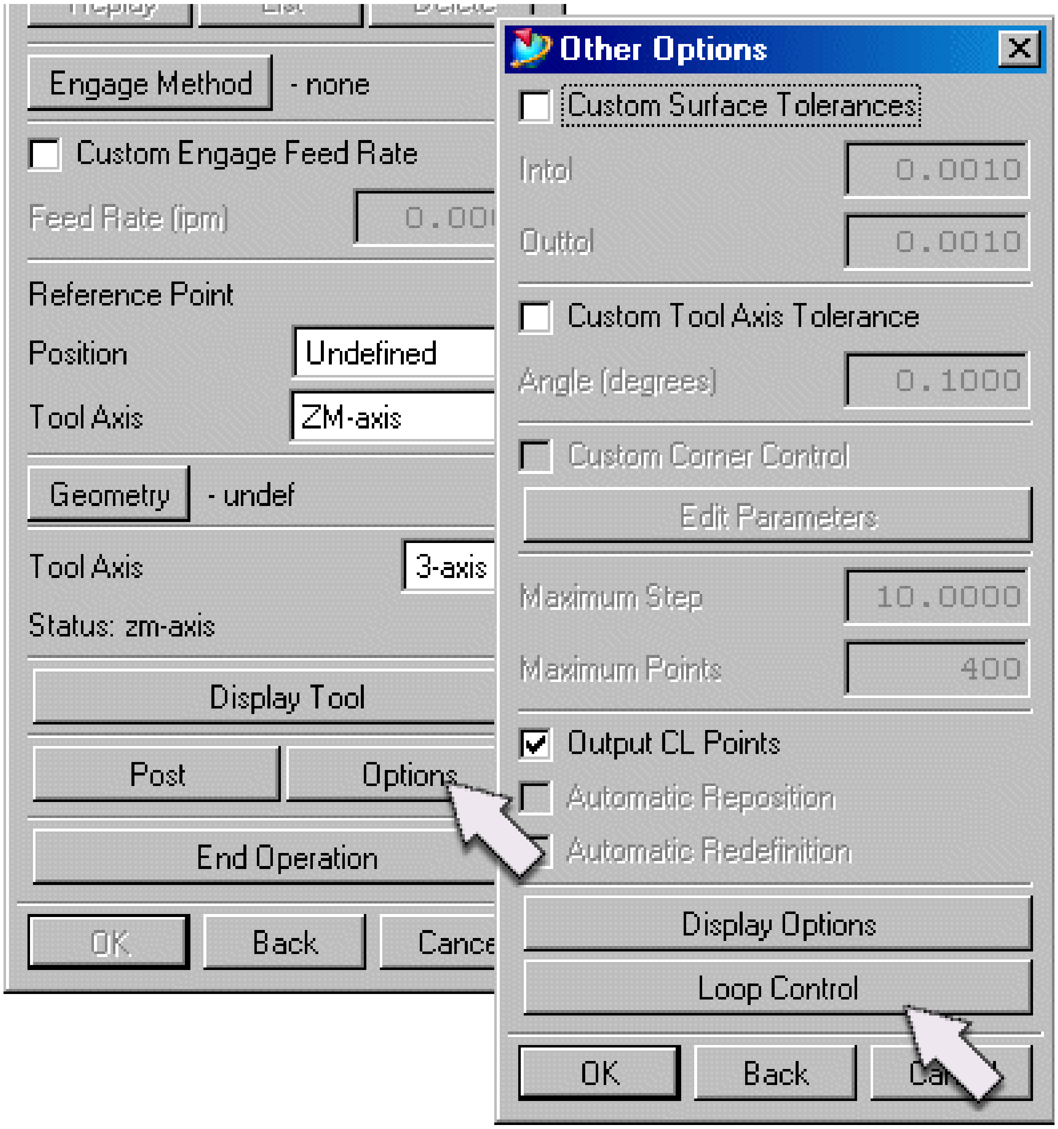
  In a future activity, you will use Sequential Mill looping functionality, with five-axis motion, to remove the excessive stock.

Standard and Nested Loops

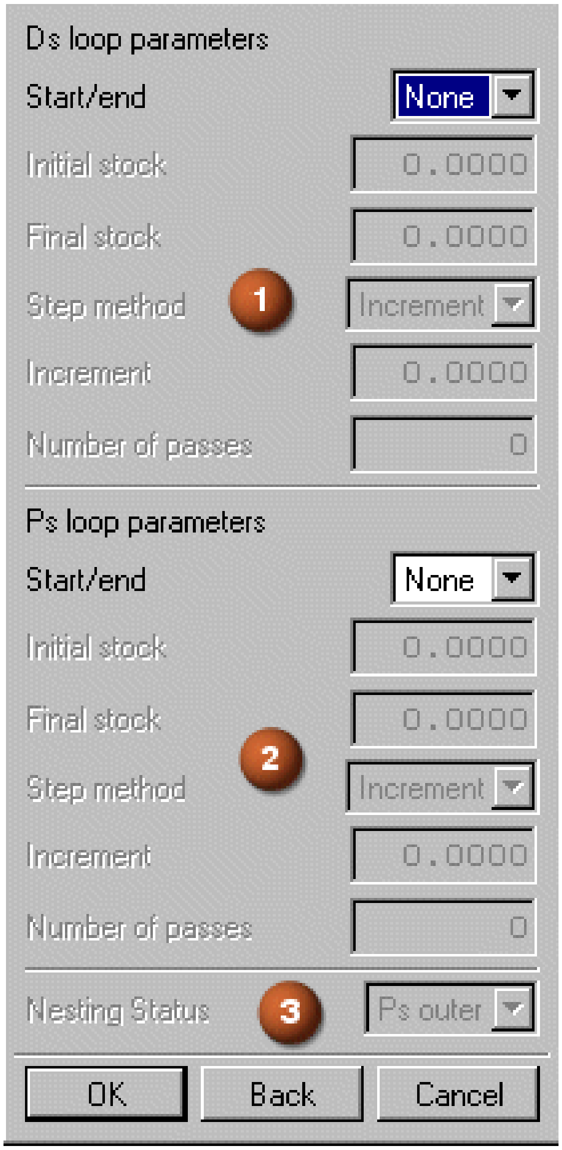
Standard Loops

Loops are modified copies of an original tool path. They are copies of a portion of a tool path that are repeated to remove extra stock.

The Loop option is located in any of the Motion dialogs (Engage, Retract, Continuous Path, or Point to Point) under the Options → Loop Control.



The Loop Control dialog follows:



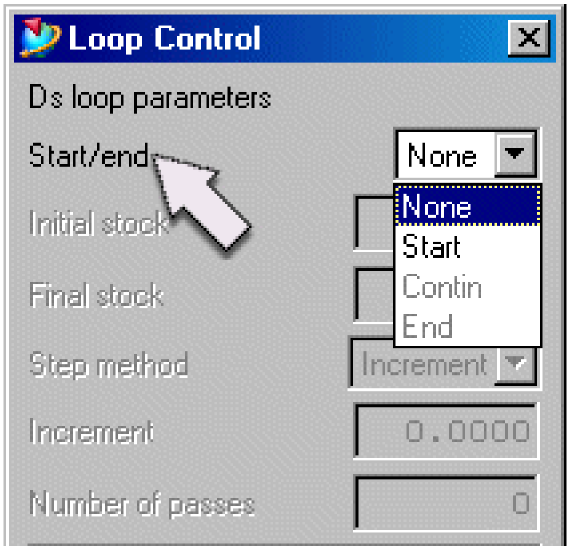
(1) starts and ends the looping routines that cut toward the drive surface

(2) starts and ends the looping routines that cut toward the part surface

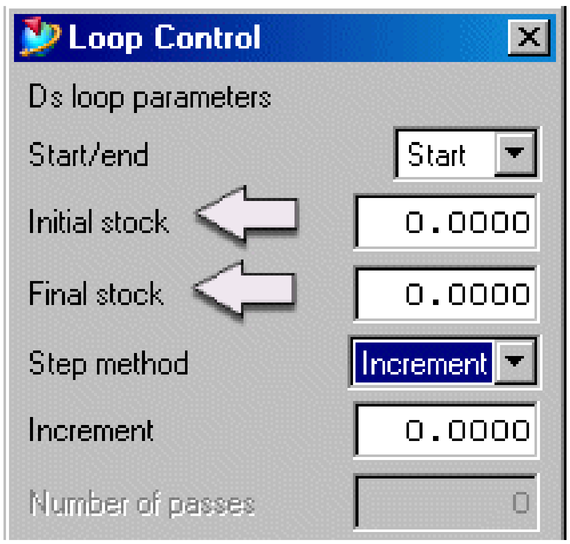
(3) specifies the inner and outer loops when both start on the same sub-operation

Before you begin the creation of a loop, the tool should be in the proper position within the operation (where you want the tool to start repeating from).

Choose Options →Loop Control →Start/End and change to Start. This establishes the beginning of the Loop.



You can also specify Loop Stock. This is the stock that is applied to the geometry within the loop. It is removed as the looping routine progresses.



To end the loop, you should be in the desired position within the operation and then stop the loop. Choose Options →Loop Control→Start/End and change to End.

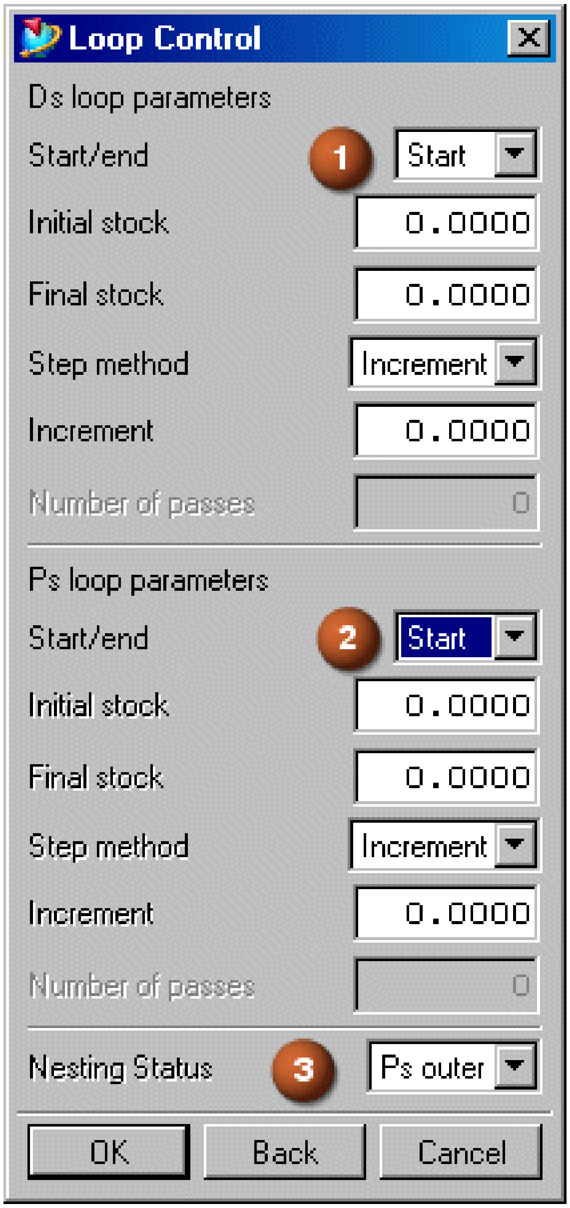
The tool path is then recomputed by adding the loop Stock and moving toward the part in a specified number of steps. The path will display in the graphics window.

You can also create an operation without a loop. You can later edit the operation and then add the loop.

Nested Loops

A Drive surface and a Part surface loop within the same sub-operation or a later sub-operation is considered a nested loop (one inside of another).

If the *Ds* loop and the *Ps* loop are started within the same sub-operation, you must determine whether you want the *Ds* loop or the *Ps* loop to be cut first. The Nesting Status option defines this for you. This option is only available after both the *Ds* and *Ps* Start/End Parameters are defined.



(1) Drive Surface Start/End parameter

(2) Part Surface Start/End parameter

(3) Nesting Status option

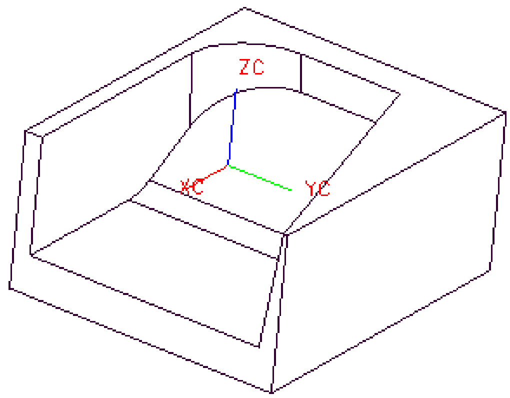
The next activity will familiarize you with some of the basic concepts of looping within Sequential Mill.

Activity: Sequential Mill – Using Loops

In this activity, you will replay and examine Sequential Mill looping operations.

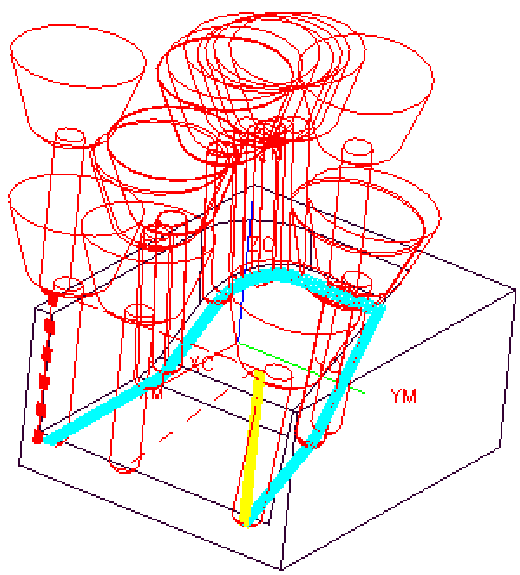
Step 1: Open a new part file and replay an existing operation.

☐ Open the file **mam_sq_3_loop**.



☐ Enter the **Manufacturing application**.

☐ From the Operation Navigator, **Replay** the **FINWALLS** tool path.



The tool path makes several passes toward the part walls and floors. You will now examine the loop settings.

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

Note that the **Multiaxis Output** option is selected.

☐ Choose **OK**.

The Point to Point Motion dialog is displayed.

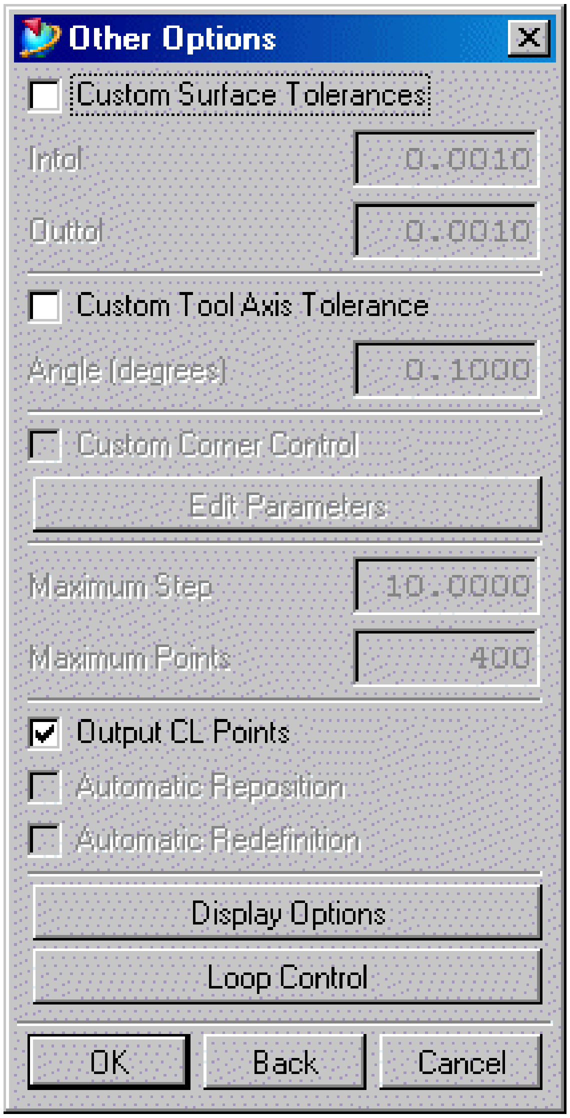
☐ Choose **OK**.

The Engage Motion dialog is displayed.

Normally, you start the looping process from within this dialog.

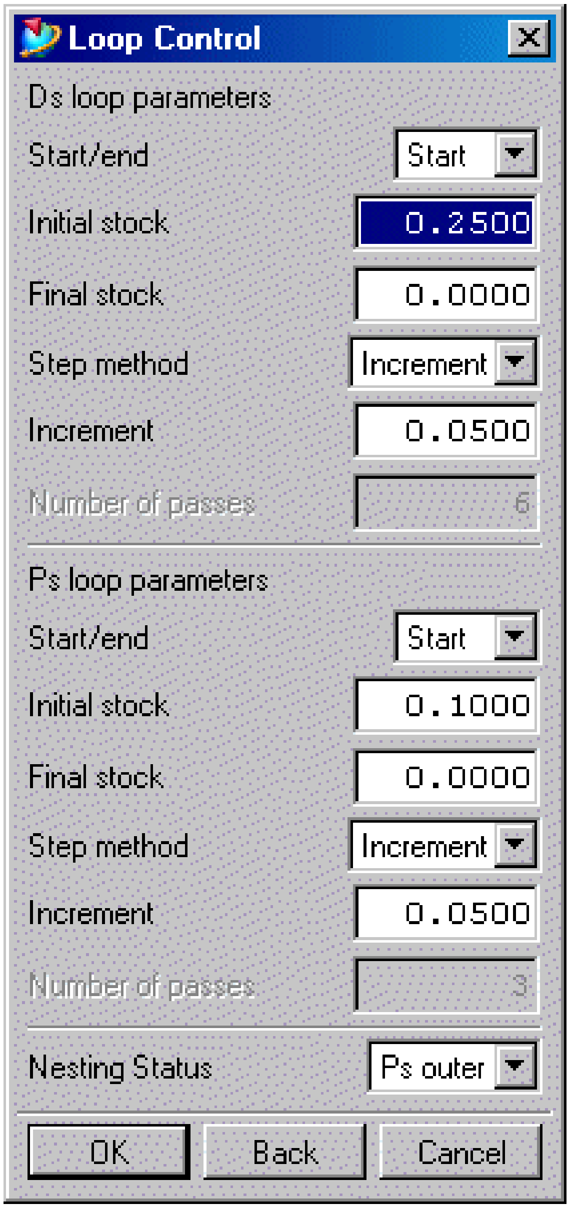
☐ Choose **Options**.

The Other Options dialog is displayed.



☐ Choose **Loop Control**.

The Loop Control dialog is displayed.



Note the Ds and PS loop settings.

These settings will create five passes, each pass will remove .050 stock.

☐ Choose **OK** three time to return to the Continuous Path Motion dialog.

☐ On the Continuous Path dialog, choose **Options** and then **Loop Control** to check the Loop Control status. They are set to **Contin**.

☐ Choose **OK** twice to return to the Continuous Path Motion dialog.

Step 2: End the loop.

☐ On the Continuous Path Motion dialog, double-click on the sub-operation **11 Ret**.

The tool path updates to the current location.

☐ On the Retract Motion dialog, choose the Options button, then the Loop Control button to check the loop status. They are set to **End.**

Step 3: Start the looping process.

☐ Choose **OK** three times until the Loop Debug Options dialog is displayed.

☐ On the Loop Debug Options dialog, choose **OK**.

The tool begins to cut as specified.

☐ Choose **End Operation** and then choose **OK** from the End Operation dialog to save the operation and return to the Operation Navigator.

The entire tool path is now displayed.

☐ Close the part.

Activity: Removing Excess Stock from a Closed Wall

In this activity, you will use the looping functionality of Sequential Mill to remove the excess stock on a undercut wall. You will make a copy of the previous operation that you created and modify that operation for doing looping activities.

Step 1: Copy a previous Sequential Mill operation.

☐ Open the part file*****_spar_mfg** (or choose from Window on the toolbar)

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

☐ Expand the **SIDE_1** and **FINISH_1** Program object.

☐ Highlight the **SM_FINISH_WALLS_POCKET_1** operation that you previously created, use **MB3, Copy**.

☐ Highlight the **PM_FINISH_BOSSES** operation, use **MB3, Paste.**

☐ Use **MB3**, **Rename** the operation to **SM_SEMI-FINISH_WALLS_POCKET_1**.

Step 2: Edit the operation.

You will want to edit the operation which you just copied and renamed. You will be using most of the same defaults as in the previous operation. However, some parameters will change.

☐ Double– click on the **SM_SEMI_FINISH_WALLS_POCKET_1** operation.

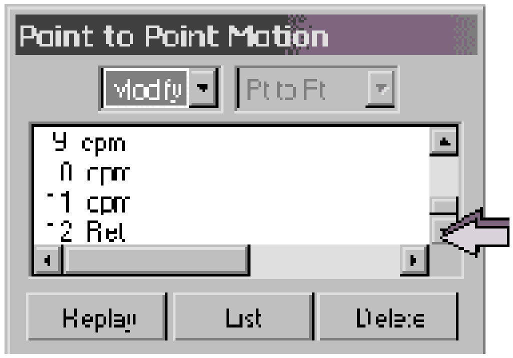
The Sequential Mill dialog is displayed.

☐ Change the **Global Stock** on **Drive Surfaces** to **.030**.

☐ Change the **Global Stock** on **Part Surfaces** to **.030**.

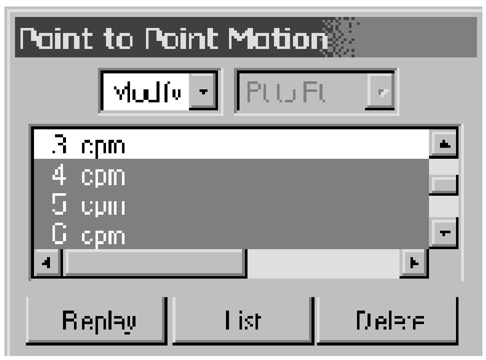
☐ Choose **OK** in the Sequential Milling dialog.

☐ Scroll down to the bottom in the sub-operation list.

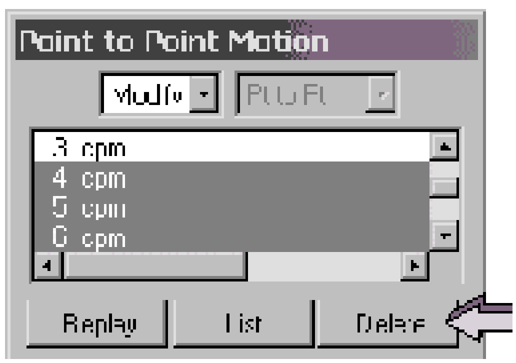


☐ Highlight the **12 Ret** sub-operation.

☐ Hold down the shift key, scroll back up in the dialog and choose the **4 cpm** sub-operation.

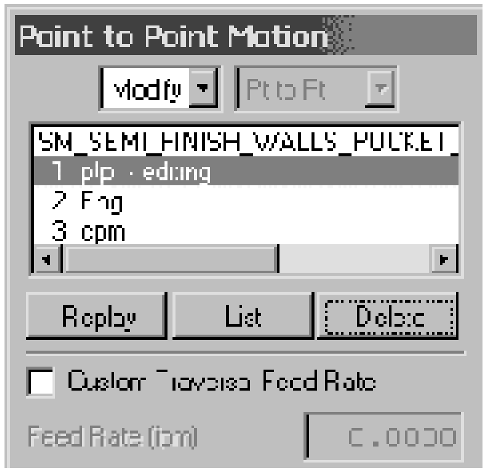


☐ Choose the Delete button and confirm the choice in the pop-up dialog.



There should now be three sub-operations remaining in the sub-operation list —a Point to Point; an Engage and a CPM.

The dialog should look as follows:

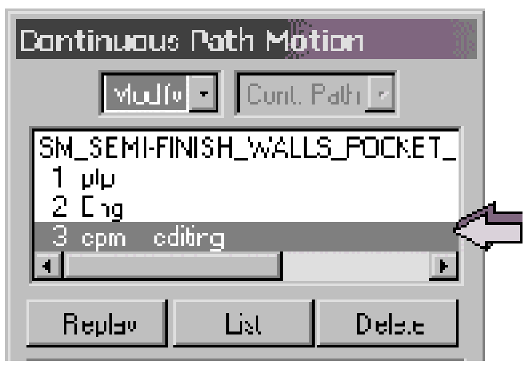


Since this operation will leave stock on the wall and the tool radius is nearly the size of the corner fillet, the corner fillet radii will not be selected. When stock is added to the fillet, it becomes impossible for the tool to reach its designated tangency point.

Step 3: Edit the sub-operation 3 cpm.

☐ Double–click on sub-operation 3 in the sub-operation list box.

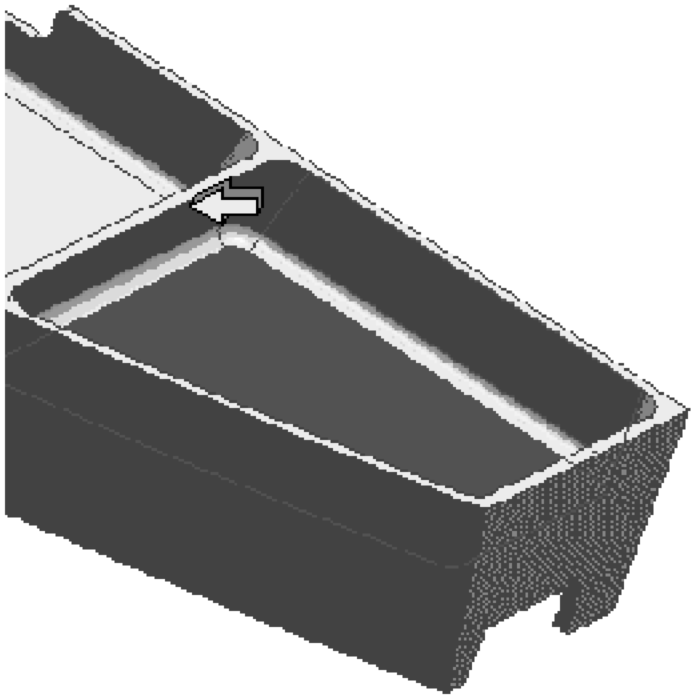
Note that in order to edit a sub-operation, simply highlighting the operation will not place it in edit mode. A double-click on the sub-operation is necessary. When successful, the word "editing" will appear following the sub-operation name.



☐ Choose the Check Surfaces button.

☐ Change the Stopping Position to Near Side.

☐ Choose the undercut wall as shown.



☐ Choose **OK** in the Check Surfaces dialog.

☐ Choose **OK** to accept the modified CPM sub-operation.

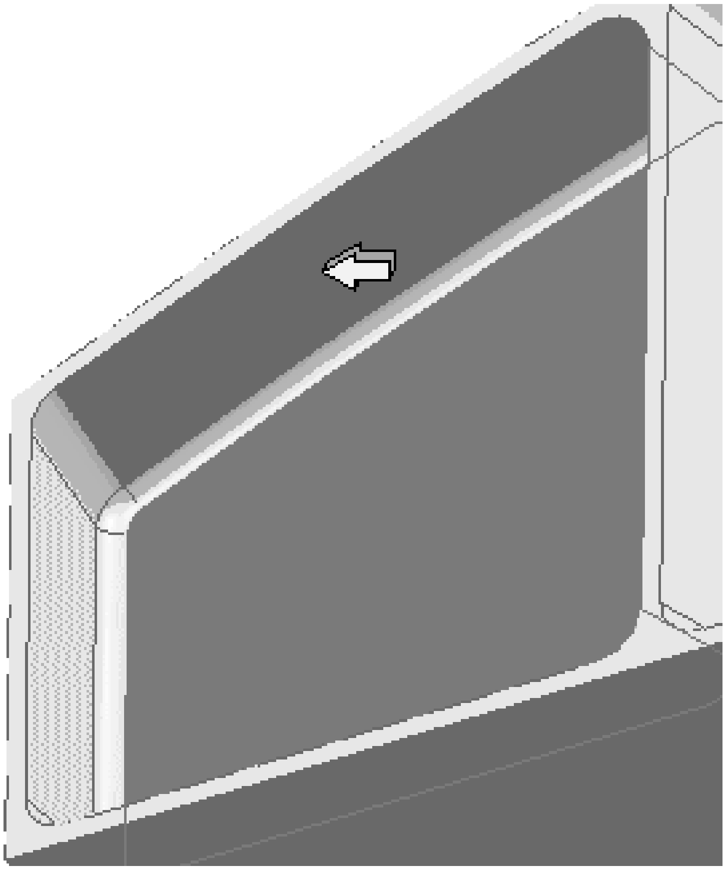
Since there are not any more sub-operations to edit, Sequential Mill automatically switches to Insert mode.

Step 4: Create additional sub-operations.

You will now create the additional sub-operations, necessary to finish the undercut area of the pocket.

☐ Choose the **Check Surfaces** button.

☐ Choose the wall as shown below.



☐ Choose **OK** in the Check Surfaces dialog.

☐ Choose **OK** to create the sub-operation.

The sub-operation is created. You will now position the cutter to the middle of the Check surface which you previously selected and then will retract the tool.

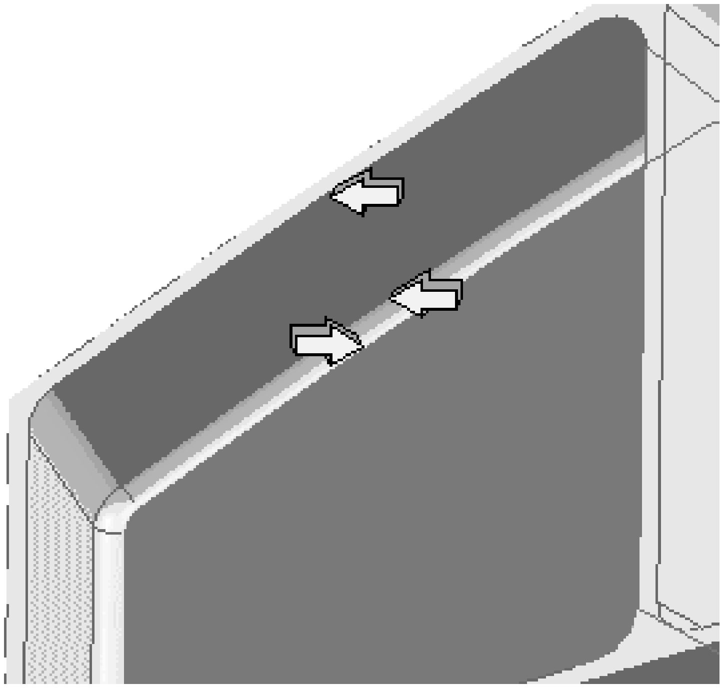
☐ Choose the **Check Surfaces** button.

☐ Change the Type to **Temporary Plane**.

☐ Choose the Three Points method.

☐ In the Point Constructor dialog, choose the Control Point icon.

☐ Choose the three edges in the area as shown below.



☐ Choose **OK** in the Check Surface dialog.

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

☐ Change the motion type to **Retract**.

☐ Choose the **Retract Method** button.

☐ Change the Method from **None** to **Vector Only**.

☐ Key in the following values to create the vector:

I = 0.0

J = -1.0

K = 1.0

☐ Choose **OK** in the Vector Constructor dialog.

☐ Key in **1.0** in the **Distance** field.

☐ Choose **OK** in the Retract Method dialog.

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

The sub-operation, **6 Ret** is created.

- ☐ Choose **End Operation**.
- ☐ Choose **OK** in the End Operation dialog.
- ☐ Save the Part file.

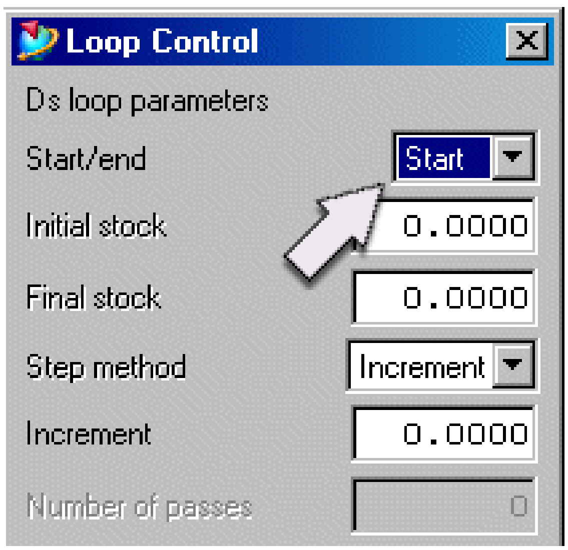
Activity: Using Looping to Remove Excess Stock

In this activity, you will edit the previous operation, modify the operation by using the looping option, which will create a series of passes for stock removal.

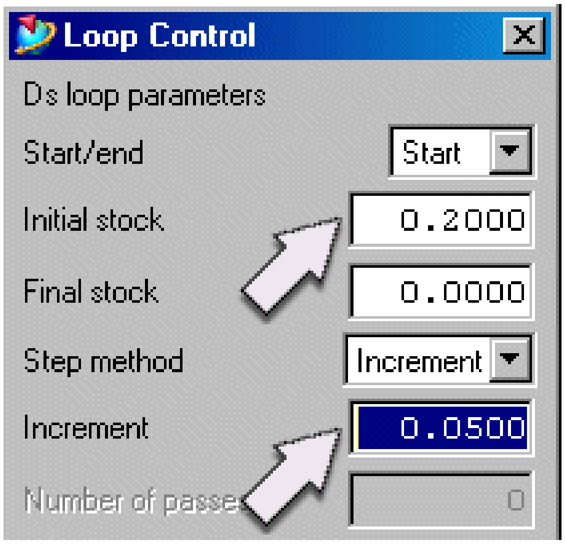
Step 1: Edit an existing operation.

- ☐ Continue using *****_spar_mfg**.
- ☐ In the Operation Navigator, double-click on the **SM_SEMI_FINISH_WALLS_POCKET_1** operation.
- ☐ Choose **OK** in the Sequential Mill dialog.
- ☐ Choose **OK** in the Point to Point Motion dialog to advance to sub-operation 2.
- ☐ In the Engage Motion dialog, choose the **Options** button.
- ☐ Choose **Loop Control**.

  The Loop Control dialog is displayed.
- ☐ Change the Ds loop parameters Start/End from **None to Start**.



- ☐ Key in **0.2** in the **Initial stock** field and **.05** in the **Increment** field.



- ☐ Choose **OK**.
- ☐ Choose **OK** in the **Other Options** dialog.
- ☐ Choose **OK** in the **Engage Motion** dialog.
- ☐ Continue to choose **OK** until sub-operation **6 Ret** is highlighted (Retract Motion dialog is displayed).
- ☐ Choose **Options**.
- ☐ Choose **Loop Control**.
- ☐ Change the Ds loop parameters from **Contin** to **End**.
- ☐ Choose OK in the **Loop Control** dialog.
- ☐ Choose **OK** in the **Other Options** dialog.

  Sequential Milling is now ready to create the additional loop passes.
- ☐ Choose **OK** in the **Loop Debug Options** dialog.
- ☐ When satisfied with the additional passes, choose **End Operation** in the **Point to Point Motion** dialog.
- ☐ Choose **OK** in the **End Operation** dialog.
- ☐ Visually examine the output using **Visualization**.

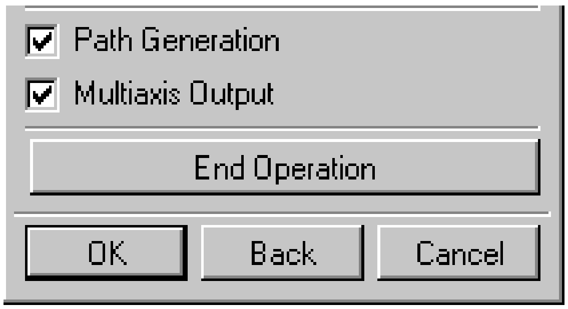
☐ Save and close the part.

Additional Sequential Mill Options

The following are Sequential Mill options that you have not used in the activities. You can review these options with your instructor or on your own.

Path Generation and Multiaxis Output

You can turn Path Generation on or off from the Sequential Mill dialog. When this option is toggled on, the tool path segment is graphically displayed as each sub-operation is accepted. When it is off, the tool path segment is not calculated or displayed.

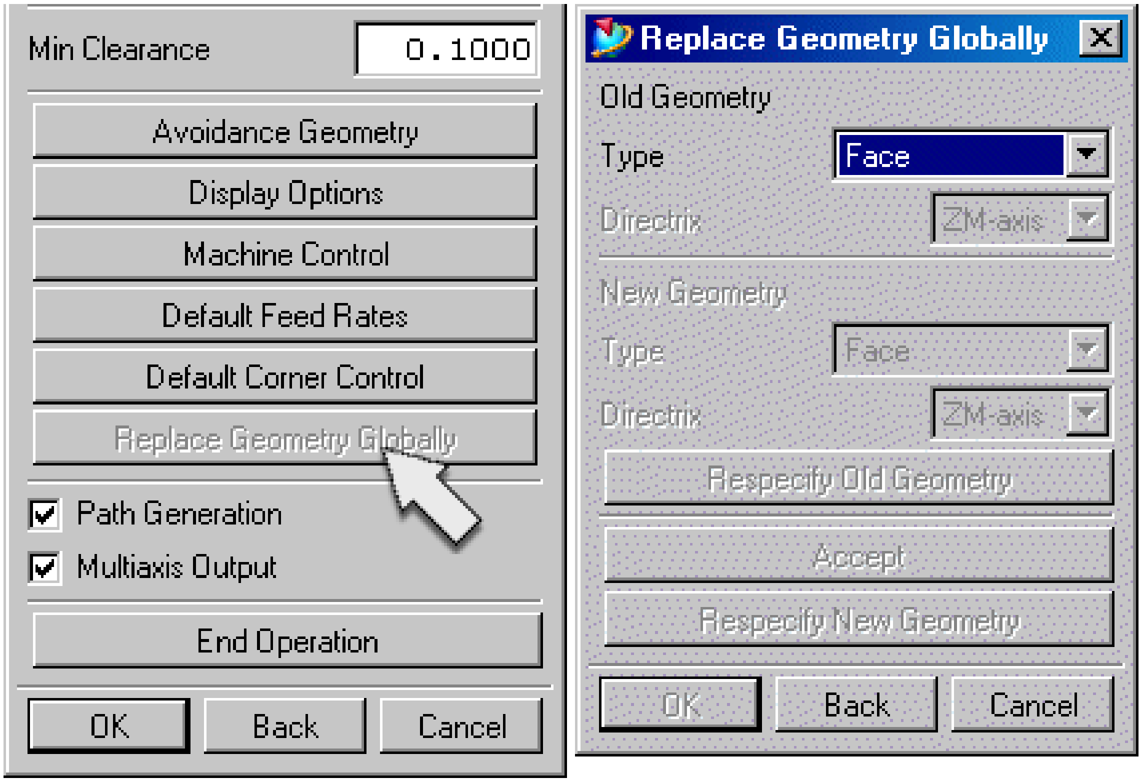


Multiaxis Output is an option. This outputs the I, J, and K components of the tool axis vector with each output point. This option must be active if 4 or 5-axis tool positioning is used. The default is **ON** (box is checked).

Replace Geometry Globally

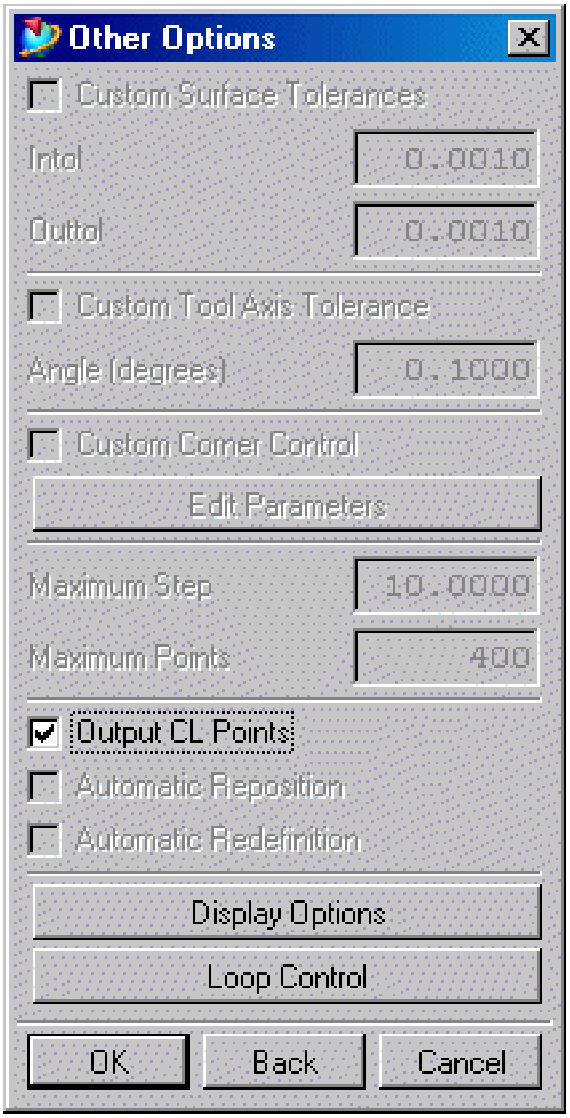
Replace Geometry Globally, replaces faces, curves and temporary planes by other faces, curves and temporary planes throughout the operation.

This option is located on the Sequential Mill dialog.



Other Options

Use the Other Options dialog to set custom tolerances, limit the step distance and number of output points, set tool path and tool display options, use of Corner Control, and to specify looping routines for a specific sub-operation.



A summary of the options on the Other Options dialog follows:

Custom Surface Tolerances specifies special *Intol* and *Outtol*values for the current sub-operation. This option is only active in the Engage and Continuous Path Motion dialog.

Custom Tool Axis Tolerance specifies a special tool axis tolerance for the current sub-operation. This option is only active in the Engage and Continuous Path Motion dialog.

Custom Corner Control specifies the cutter feed rate, slowdowns, and fillet radius at corners. By toggling the Custom Corner Control button and selecting the Edit Parameters action button, you bring up the Corner and Feed rate Control dialog. This option is available for Continuous Path sub-operations only.

Maximum Step specifies the maximum length of individual tool moves in the sub-operation. If you change the default value of ten inches, the new value becomes the default value for the current and subsequent sub-operations. If you are editing a sub-operation, changing the Maximum Step does not affect subsequent sub-operations. You must specify a positive value. This option is available for Continuous Path sub-operations only.

Maximum Points specifies the maximum number of points generated in a sub-operation. If you change the default value of 400, the new value becomes the default value for the current and subsequent sub-operations. If you are editing a sub-operation, changing the Maximum Points does not affect subsequent sub-operations.

Output CL Points temporarily suspends the output of points to the CL source file. By suspending the output of CL points, you can move the tool in several sub-operations that are not included in the tool path. When you finally position the tool to the correct geometry, activate this option and the tool path continues (this is similar to APT's CUT/DNTCUT).

Automatic Redefinition establishes a check plane at the last valid tool location when the processor is unable to complete the tool path for a sub-operation. You can continue programming from the new check plane.

Automatic Reposition is useful if the tool is not within tolerance to the Drive or Part surface at the *start* of a sub-operation.

Display Option sets tool, pattern, and tool path display options for the current sub-operation. This is the same Display Options dialog used in Operation Parameters.

Loop Control specifies a looping routine for area clean-up of Drive or Part geometry, or both.

Most Sequential Milling processor errors are caused by the tool being out of tolerance to the geometry.

Sequential Milling Best Practices

Engaging:

- use a reference point that is *near* the startup geometry
- when using the Fan tool axis, use *Tangent to Drive*
- use the Direction Move option on the Engage Geometry dialog when the tool can move to more than one location or if the tool is not close to the surface
- remember that the Direction Move is applied first to the Drive, second to the Part, and last to the one or more Check surfaces
- use *Side Indication* on the *Engage Geometry* dialog when the tool is on or overlaps a surface

You should imagine the tool moving initially after you specify the Drive surface. Then, if you need to specify a direction for the Part surface, do so from the imagined position. Then imagine the tool moving to the new position if you need to specify a Direction Move for the Check surface.

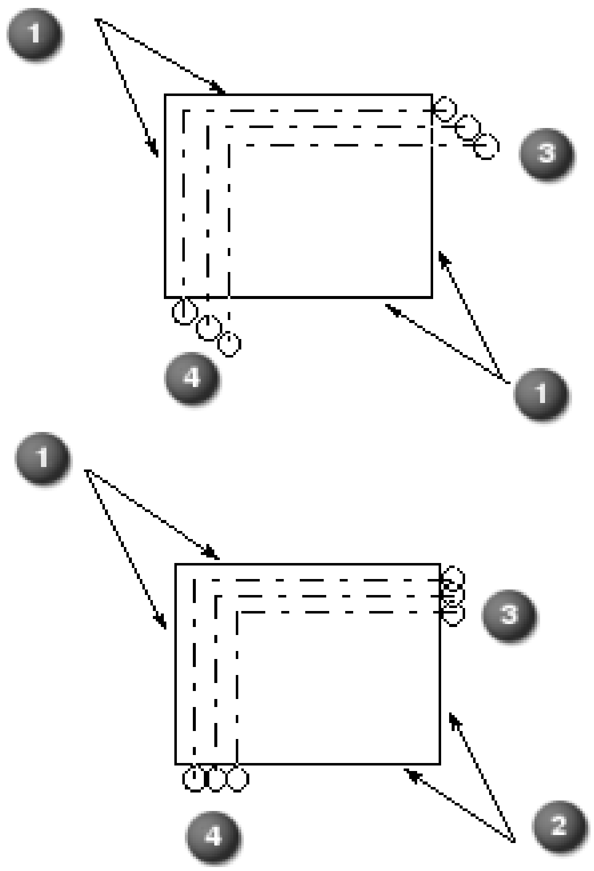
Continuous Path:

- if the Drive and Part surfaces are flat and long, reduce the *Maximum Step* (on the Other Options dialog)
- when using a Fan tool axis, reduce the Maximum Step (on the Other Options dialog)
- when using a Fan tool axis around curved geometry, limit the motion to 60 degrees

Looping:

- start a loop on an *Engage* or *Point to Point Motion* sub-operation; starting a loop on a *Continuous Path Motion* sub-operation can cause the tool to be out of tolerance
- the last loop sub-operation should be a *Retract* or *Point to Point Motion* move
- if you do not want the tool retracting during the loop, be careful in ending the loop on a Continuous Path Motion sub-operation so that the loop ends with the tool in the same position and orientation as at the start of the loop

- use caution when specifying *Added Stock* to *Check Geometry* . In a loop, you may want to choose None when you do not use a Check Surface as a Drive or Part surface in the next sub-operation. See the following example.



(1) Added stock = Drive

(2) Added stock = None

(3) Start

(4) End

Summary

The more advanced features of Sequential Milling allow for multiple passes and complete control of the tool axis. These options allow for increased flexibility for roughing and finishing operations. Some of the more advanced features are:

- Looping control allowing for removal of excess stock.
- Fanning tool axis control.
- Complete control of tool positioning.

Lesson

5 *Variable Contour - Basics*

Purpose

Variable Contour operations are used to finish areas formed by contoured geometry. Variable Contour tool paths are able to follow complex contours by the control of tool axis, projection vector and drive methods.

Objective

At the conclusion of this lesson, you will be able to:

- create multi-axis tool paths by choosing a tool axis that is most appropriate for the part geometry
- incorporate complementary programming practices that are necessary for multi-axis machining

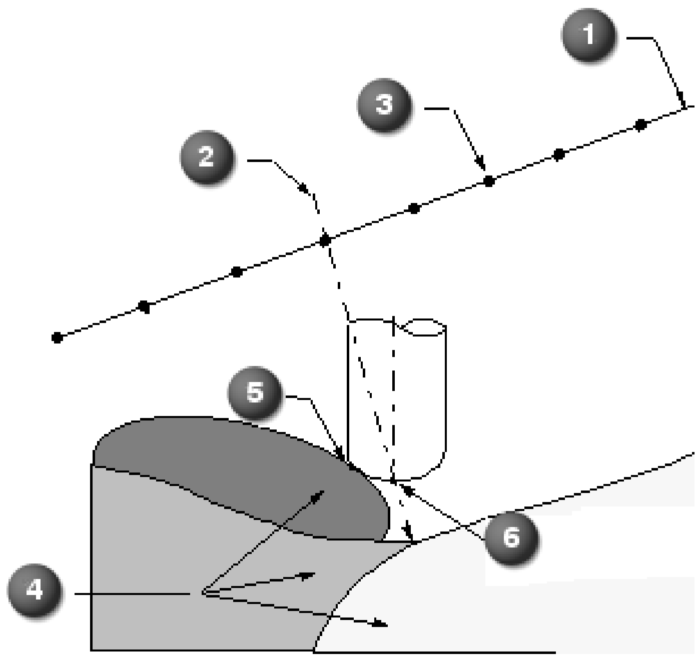
Variable Contour Operations

Variable Contour operations are used to finish areas formed by contoured geometry by the control of tool axis, projection vector and drive methods.

Tool paths are created through the generation of drive points from the drive geometry and then projecting those points along a projection vector to the part geometry.

The drive points are created from part geometry or can be created from other geometry that is not associated with the part. The points are then projected to the part geometry.

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



(1) Drive geometry is used to generate points

(2) Projection vector moves the tool from the drive point, down the projection vector until it contacts part geometry

(3) Drive points

(4) Part geometry may keep the tool from reaching the projected drive point

(5) Contact point

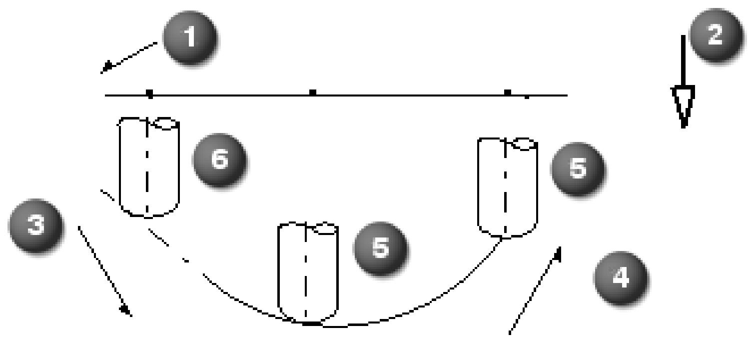
(6) Cutter location output is generated

Tool Path Accuracy

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

- Check geometry to stop tool movement
- gouge checking to prevent gouging of the part
- Various tolerance options

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



(1) Drive points

(2) Projection vector

(3) extension of part geometry

(4) Part geometry

(5) Valid

(6) Invalid

Terminology used in Variable Contour

- Part Geometry - is geometry selected to cut
- Check Geometry - is used to stop tool movement
- Drive Geometry - is used to generate drive points
- Drive Points - are generated from the Drive geometry and projected onto part geometry
- Drive Method - method of defining Drive Points required to create a tool path; some drive methods allow creation of a string of drive points along a curve while others allow the creation of an array of drive points within an area
- Projection Vector - used to describe how the Drive Points project to the Part Surface and which side of the Part Surface the tool contacts; the selected drive method determines which Projection Vectors are available

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

Variable Contour vs Fixed Contour

The primary difference between Fixed Contour and Variable Contour lies with the various methods of tool axis control and the drive methods available.

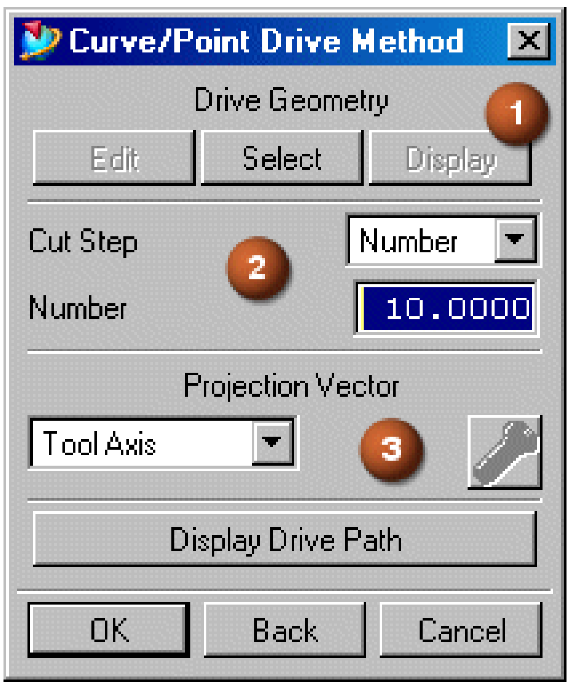
Drive Methods for Variable Contouring

Curve/Point Drive Method

Allows you to define drive geometry by specifying points and curves. Using points, the drive path is created as linear segments between the points. Using curves, drive points are generated along the curves. The drive geometry is projected on to the part surface(s) where the tool path is created. The curves may be open, closed, contiguous, non-contiguous, planar or non-planar.

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

The Curve/Point Drive Method dialog allows you to specify the distance between drive points and the projected location of drive points. You can also use the Display Drive Point option to view the location of the drive points before generating the tool path.



(1) Used to select and edit drive geometry

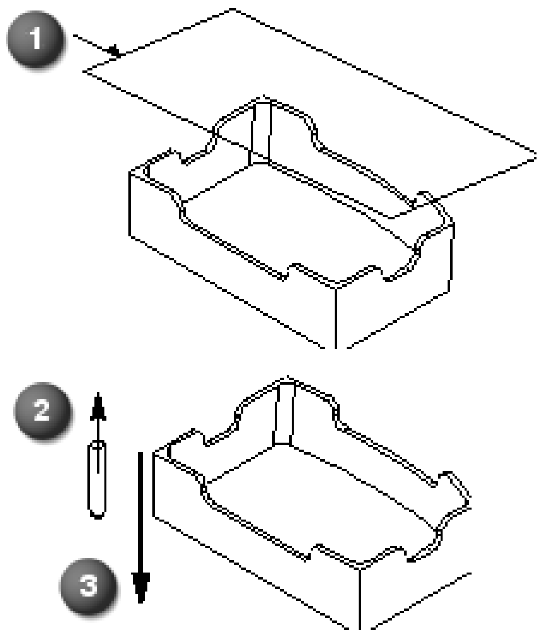
(2) Controls the distance between drive points

(3) determines the method of projection onto the part

Boundary Drive Method

The Boundary Drive Method allows you to define cut regions by specifying Boundaries and Loops. Boundaries are not dependent on the shape and size of the part surfaces while Loops must correspond to exterior part surface edges. Cut regions are defined by Boundaries, Loops, or a combination of both.

The Boundary Drive Method allows you to use a permanent boundary, part curves or faces to generate drive points.



(1) Boundary

(2) Tool axis

(3) Projection vector

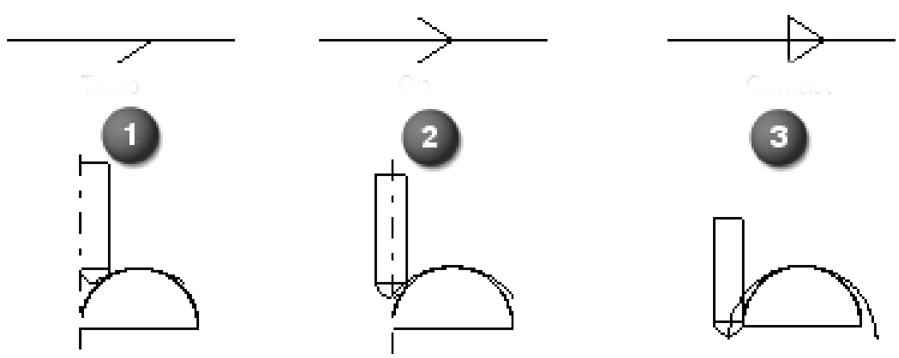
Drive points are generated within the boundary and are then projected linearly onto the part geometry according to the specified projection vector.

The Boundary Drive Method is preferred to the Surface Area Drive Method. You can quickly create a boundary and tool path without the surface design requirements of the Surface Area Drive Method.

This method does not allow as many choices of tool axis options that are available in the Surface Area Drive Method and is better suited for roughing operations. The Surface Area Method is better suited for finishing operations.

Each boundary member is assigned an On, Tanto, or Contact tool position (unique to Variable Contour Boundary Drive Method). The Contact tool position can be used when specifying boundaries using curves and edges.

The boundary members graphically represent the associated tool positions as illustrated below:



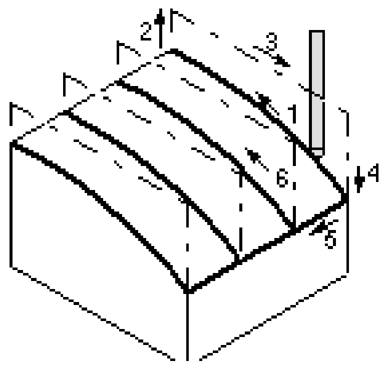
(1) tanto condition (2) on condition (3) contact condition

Options associated with the boundary drive method follows:

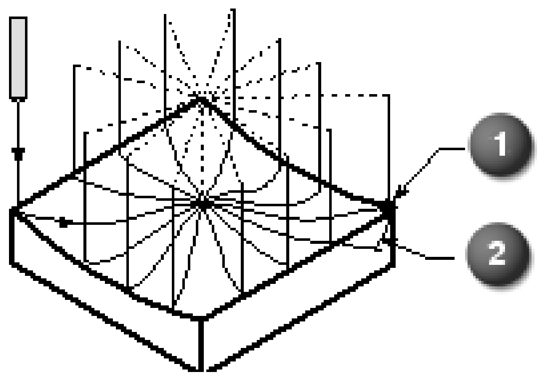
Pattern option enables you to define the shape of the tool path. Some patterns cut the entire region, while others cut just around the perimeter of the region. Some patterns follow the shape of the cut region, while others are independent of it.

The selected Pattern determines which other options are available. If you select Parallel Lines as the cut pattern, the Cut Type, Cut Angle, and Degrees options become available. If you select Follow Pocket, only the Inward and Outward options are available.

Parallel Lines creates a cut pattern defined by a series of parallel passes. You are required to specify a Cut Type of Zig-Zag, Zig, Zig With Contour, or Zig with Stepover and a specific Cut Angle.



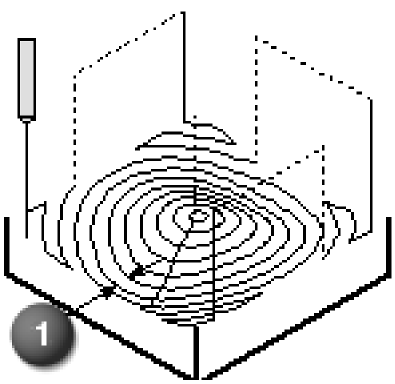
Radial Lines creates linear cut patterns extending from a user-specified or system calculated optimum center point. You are required to specify a Cut Type, a Pattern Center, and pocketing method as Inward or Outward. You may also specify an angular stepover, which is unique to this type. The Stepover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



(1) point furthest away from center

(2) stepover distance measured along arc length

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



(1) stepover

Cut Type defines how the cutter moves from one cut pass to the next. The options are used in combination with Parallel Lines, Radial Lines, or Concentric Arcs cut patterns. When used in combination with the Parallel Lines pattern, Zig-Zag, Zig and Zig with Contour work in the same way as their counterparts in Planar and Cavity Milling.

Pattern Center allows interactive or automatic definition of the center point of Concentric Arcs and Radial Lines cut patterns.

Cut Angle determines the angle of rotation for the Parallel Lines Cut Patterns. This option is available if the Cut Angle is set to Specify. Enter the degrees of rotation for the Parallel Lines Cut Pattern.

Outward and *Inward* allow you to specify a pocketing method that determines whether to cut from the inside out or the outside in of a Follow Pocket, Concentric Arcs, or Radial Lines cut type.

Stepover specifies the distances between successive cut passes.

Constant specifies a fixed stepover distance between successive cut passes. When used with the Radial Lines cut type, the constant distance is measured along the arc length at the boundary point farthest away from the center.

Scallop determines the stepover distance based on the scallop height you enter.

Tool Diameter defines the stepover in terms of a percentage of the effective tool diameter.

Variable allows you to vary the stepover distance within a specified minimum and maximum value. The required input values differ depending on the selected cut type.

Angular defines a constant stepover by keying in an angle. This option is used only in combination with the Radial Cut pattern.

Additional Passes specifies an additional number of passes that allows the tool to step toward the boundary in successive concentric cuts for Profile and Standard cutting patterns.

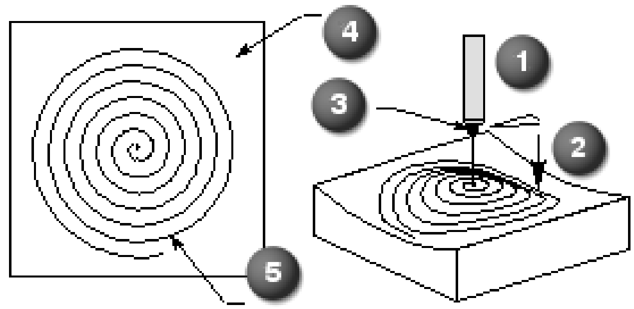
More Drive Parameters displays a dialog containing the following options:

> *Options* displays a dialog that enables you to create start points automatically or interactively and to specify how cut regions will be displayed when the Display button is selected.
>
> *Display* generates a temporary screen display of the cut regions for visual reference. The display is generated using the parameters specified under Cut Region Display options.
>
> *Display Drive Path* displays the Drive Path used to generate the tool path. The path is created as a temporary element projected onto the WCS along the tool axis and is for visual reference only.

Spiral Drive Method

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

Spiral Drive Method stepovers are a smooth, constant transition outward. This drive method maintains a constant cutting motion and is applicable to high speed machining applications.



(1) Drive points projected from plane

(2) Projection vector

(3) Center point defines the center of the spiral, cut starts here

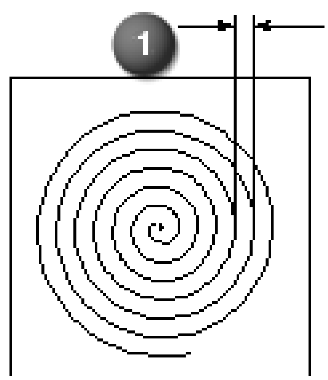
(4) Part surface

(5) Spiral drive

If you do not specify a center point, the system uses the (0,0,0) of the Absolute Coordinate System. If the Center Point is not on the part geometry, it follows the defined projection vector to the part geometry. The direction of the spiral (clockwise vs. counterclockwise) is controlled by the Climb or Conventional cut direction.

The following parameters pertain to Spiral Drive method:

Stepover allows you to specify the distances between successive cut passes and are a smooth constant transition outward; does not require an abrupt change of direction.

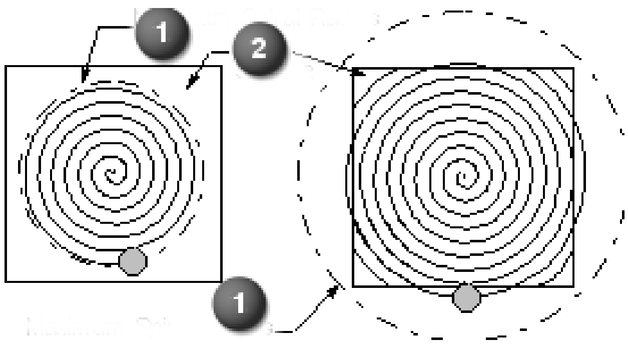


(1) stepover

Constant allows you to specify a fixed distance between successive cut passes. Key in the desired distance between subsequent cut passes.

Tool Diameter allows you to define the Stepover in terms of a percentage of the effective tool diameter.

Maximum Spiral Radius allows you to limit the area to be machined by specifying a Maximum Radius. This constraint reduces processing time by limiting the number of drive points created. The radius is measured in the plane normal to the Projection Vector.



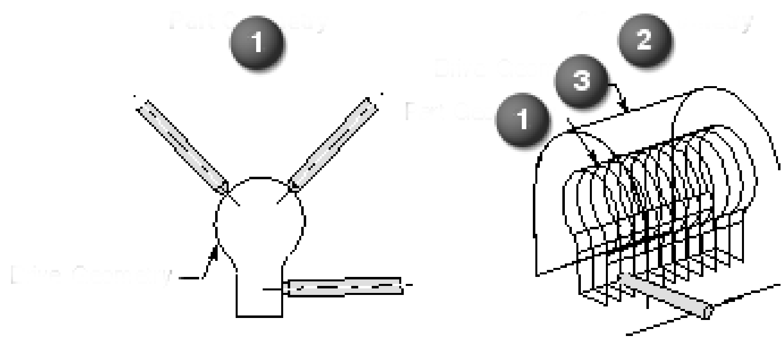
(1) maximum spiral radius

(2) part surface

If the specified radius is contained within the part geometry, the center of the tool positions to the radius before retracting. If the specified radius exceeds the part geometry, the tool continues to cut until it can no longer position to the part geometry. The tool then retracts and engages.

Surface Area Drive Method

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



(1) Part geometry

(2) other geometry

(3) drive geometry

To generate Drive Points from part geometry, select the surfaces as drive geometry and do not select any part geometry. The drive points are then generated on the drive geometry.

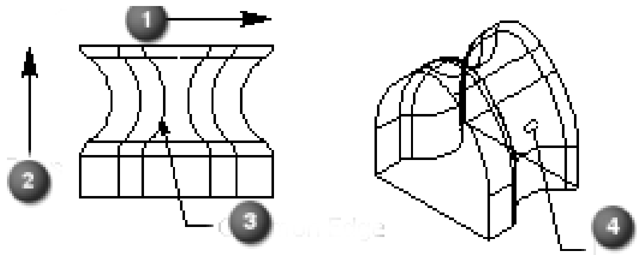
To generate Drive Points from other geometry, select the drive and part geometry. The Drive Points are then generated on the drive geometry and are projected onto the part geometry according to the Projection vector.

In either case, the tool axis can follow the drive geometry contour.

The Surface Area Drive method also provides an additional Projection Vector option, Normal to Drive, which enables you to evenly distribute drive points onto convex part geometries.

The Surface Area Drive method is much more accurate than the Boundary Drive method for complex parts and is useful for finishing types of operations.

The limiting factor of the Surface Area Drive method is that surfaces must be arranged in an orderly grid of rows and columns and adjacent surfaces must share a common edge.

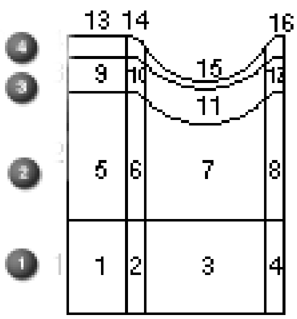


(1) columns

(2) rows

(3) common edge

(4) drive surface



Drive geometries must be selected in an orderly sequence defining the rows

(1) Row 1

(2) Row 2

(3) Row 3

(4) Row 4

The following parameters pertain to Surface Area Drive method:

Select allows you to select the Drive Geometry. Note that you can use part geometry as drive geometry.

Surface Stock offsets drive points along surface normals a specified distance.

Tool Position determines the tool contact points on the drive geometry as either On or Tanto.

Cut Direction is the tool path direction and the quadrant where the first cut will begin. It is specified by selecting one of the vector arrows which appear in pairs at each of the surface corners.

Flip Material reverses the direction of the Material Side Vector which determines the side of the surface the tool contacts when machining directly the drive geometry. When machining part geometry, the Projection vector determines the Material Side.

Cut Area defines how much of the total drive geometry area to utilize by specifying surface percentages or diagonal points and to display the boundary of the cut area.

Surface % specifies the drive geometry area to cut by using positive or negative percentage values for the beginning and end of first and last pass, and the first and last Stepover.

Diagonal Points uses the cursor to indicate two diagonal points defining the area.

Pattern defines the shape of the tool path as Follow Pocket or Parallel Lines.

Cut Type in combination with the Parallel Lines pattern defines cutter movement from one cut pass to the next. The types are: Zig-Zag, Zig-Zag with Lifts and Zig.

Cut Step controls the distance between drive points created along the drive curve. For complex parts, the closer the drive points, the more accurate the tool path. You can control the cut step by specifying a Tolerance or by specifying a Number of points.

Stepover controls the distance between successive cut passes. Stepover choices are:

> Scallop, (for Parallel Lines pattern) which requires you to enter the Height of the scallop and the Horizontal and Vertical Limit to restrict the distance the tool moves in a direction normal to the Projection Vector. This option avoids leaving wide ridges on near vertical surfaces by limiting the horizontal distance of the Stepover.
>
> Number, which requires you to enter the First and Second Directions of cutting (for Follow Pocket) or the Number of Steps (for Parallel Lines). These are used to generate drive points.

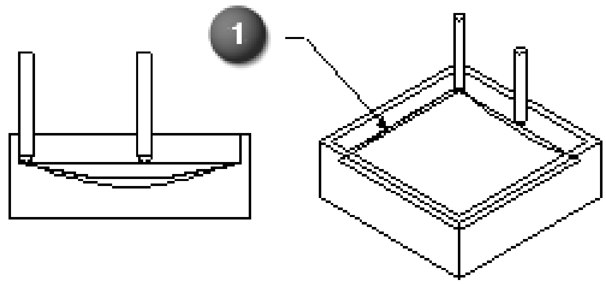
When Gouging indicates the processor action when gouging is detected. The actions are: None; issue a Warning in the tool path output; Skip the output point; or Retract and avoid the gouge.

Cutting Parameters affect each of the Drive Methods. The corresponding Cutting Parameters dialog follows:

> The Part Stock option defines an envelope of material surrounding the part geometry which remains on the part after machining. The stock specified applies to those part entities which do not have Custom Stock specified (under Custom Data in the Part geometry dialog).
>
> The Check Stock option is an envelope of material surrounding the Check geometry, which the tool will not gouge.

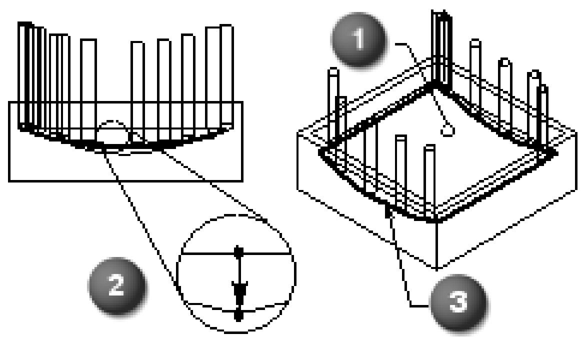
Tool Path Drive Method

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



Tool path created using Planar Mill, profile cut type

(1) planar mill tool path



Results of using Planar Mill tool path, projected on to the contoured part geometry

(1) part surface

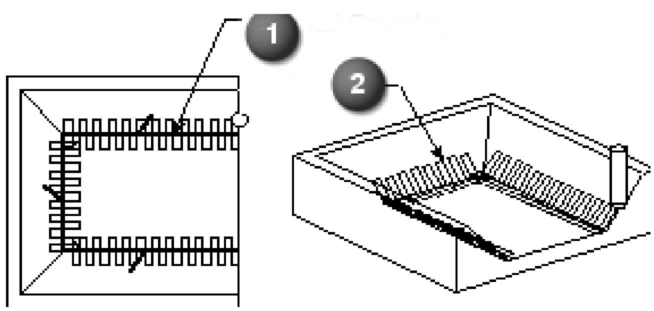
(2) drive point projection

(3) surface contour tool path

When you select Tool Path as the drive method, you must specify an existing CLSF to be used to generate drive points.

Radial Cut Drive Method

The Radial Cut Drive Method allows you to generate drive paths perpendicular and along a given boundary, using a specified Stepover distance, Bandwidth and Cut Type. This method is useful in clean-up type applications.



(1) selected boundary

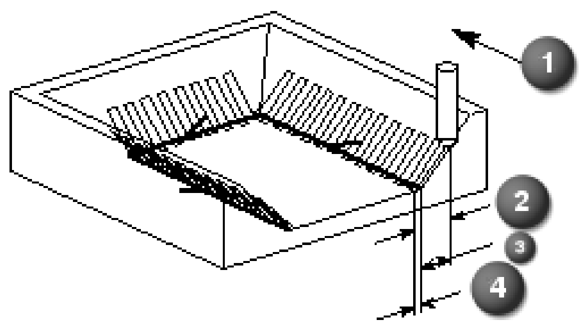
(2) tool path

The tool will Zig or Zig-Zag along the boundary in the direction of the boundary indicators. This can be changed by selecting Reverse Boundary. The following Radial Cut Drive Method options are available:

Select displays the **Permanent Boundary** or **Temporary Boundary** dialog allowing you to define the area to be cut. The Permanent Boundary dialog is only displayed if permanent boundaries currently exist. If multiple boundaries are defined, a lift is applied, allowing the tool to traverse from one boundary to the next.

Bandwidth defines the total width of the machined area measured in the plane of the boundary. The bandwidth is the sum of the Material Side and Opposite Side offset values.

The **Material Side** is the right side of the boundary as you look in the direction of the boundary indicators. The **Opposite Side** is the left side. The sum of the Material Side and Opposite Side cannot equal zero.



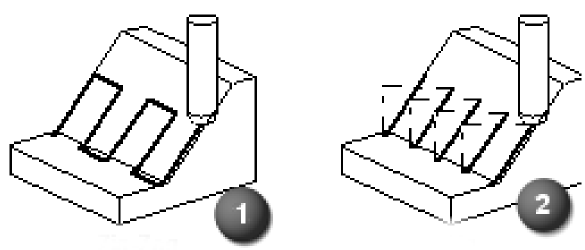
(1) looking in the direction of boundary indicators

(2) bandwidth

(3) material side

(4) opposite side

Cut Type enables you to define how the cutter moves from one cut pass to the next. The following options are available:



(1) Zig-Zag

(2) Zig

Stepover specifies the distances between successive drive paths.

Each Stepover methods require you to enter a corresponding *Distance* value.

Follow Boundary and Reverse Boundary allow you to determine the direction the tool travels along the boundary.

Contour Profile Drive Method

This method is a simple to use drive method to cut the undercut or overcut walls of a part and is especially effective in machining multi-pocket type parts. Selection of the bottom of the pocket, setting of various cut parameters, and generation of the operation are the only steps required for use.

User Function Drive Method

User Function Drive method creates tool paths from special drive methods developed using User Function programming. These are optional, highly specialized custom routines developed for specific applications.. Options available are:

CAM Exit Name is the name of an operating system environment variable which contains the path name of the shared library containing the User Function Program.

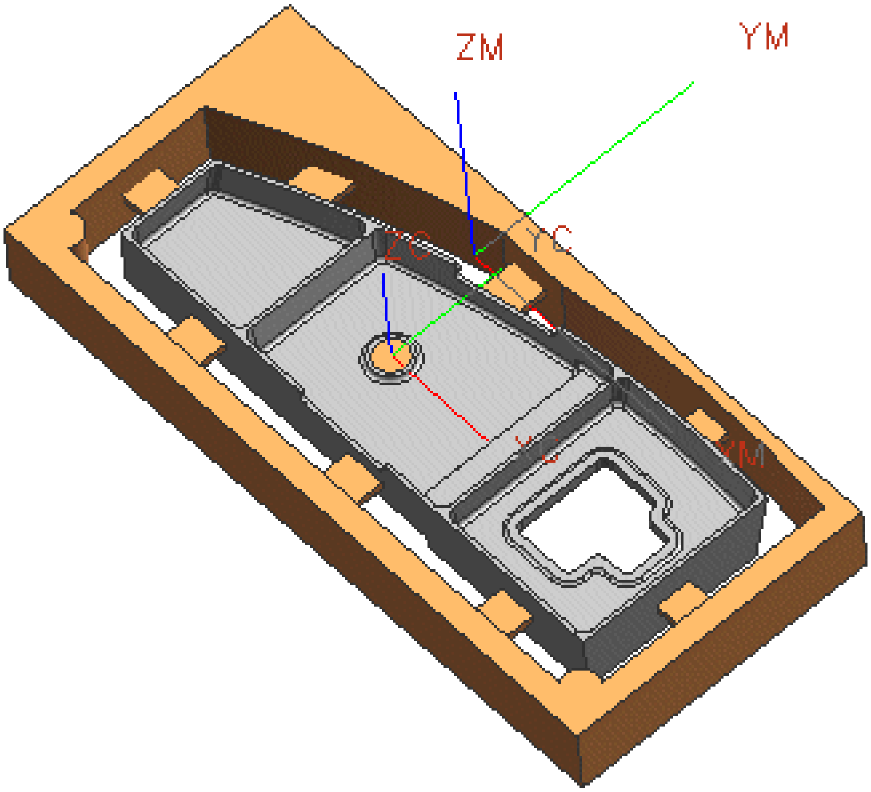
Users Parameters access a user exit specifying parameters for the drive path. The User Function program associates these parameters with the calling operation, using the name of the operation as the link.

Activity: Contour Profile Drive Method

In this activity you will use the Contour Profile drive method to machine the canted walls of the part.

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

☐ Open the part file **mam_spar_mfg**.



This part has already been roughed machined as well as the floor have been finished. All that remains to finish is the interior walls of the three rectangular pockets.

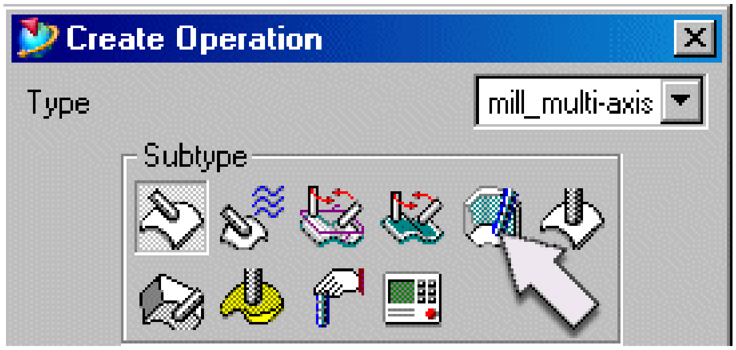
☐ Enter the **Manufacturing** application.

Step 2: Create a Variable Axis Profiling operation.

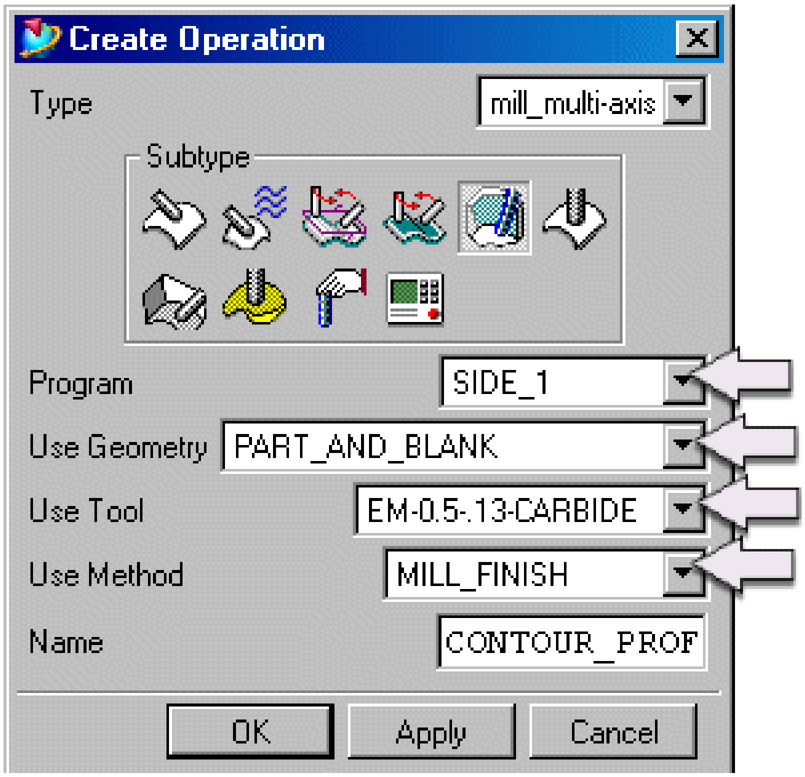
☐ Select the **Create Operation** icon.

☐ If necessary, set the **Type** to **mill_multi_axis**.

☐ Choose the **CONTOUR_PROFILE** icon.



☐ Set the group objects as shown:



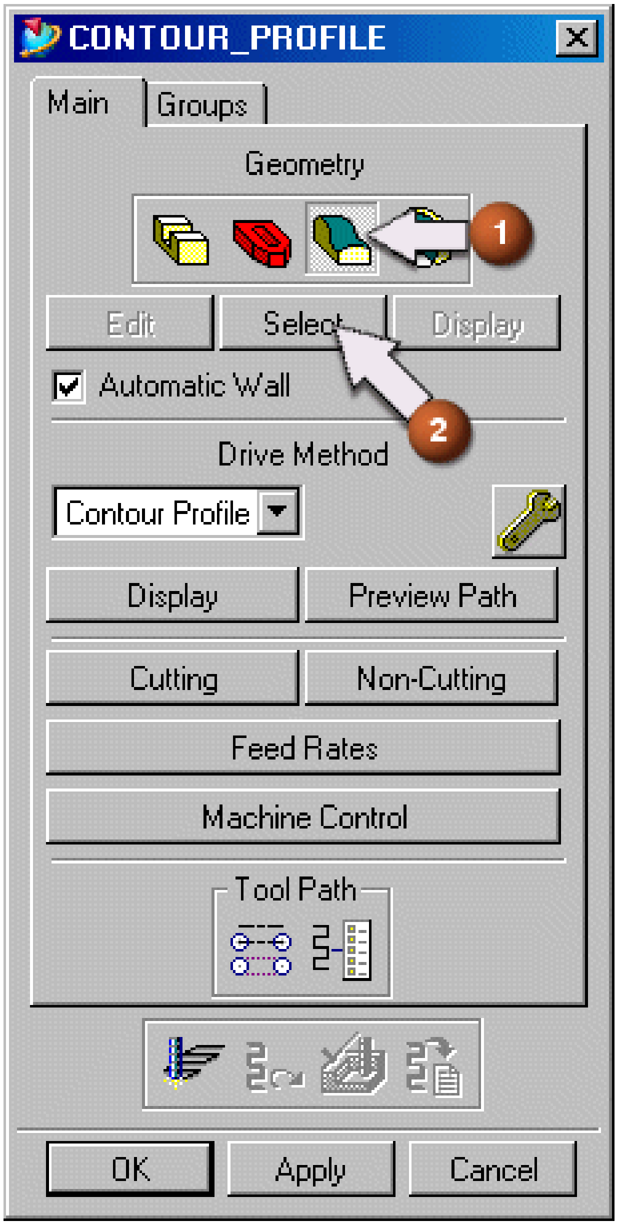
The CONTOUR_PROFILE dialog is displayed.

☐ Choose **OK**.

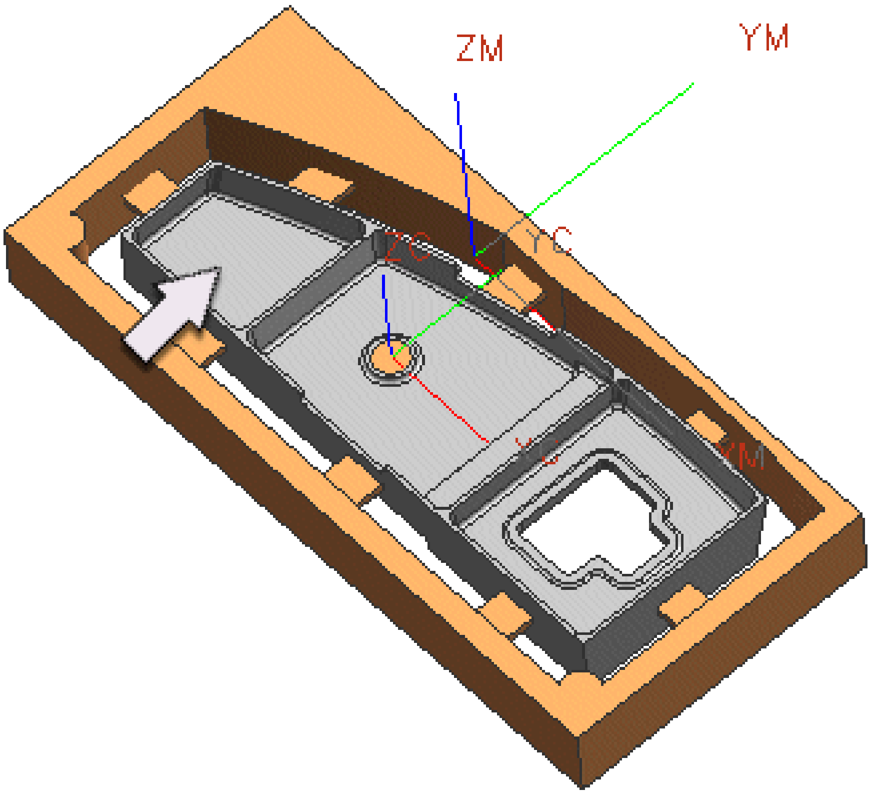
Step 3: Selection of Parameters.

As stated earlier, the only requirements necessary to use this drive method is the selection of the floor of the pocket, setting various cutting parameters and generating the operation. You will first select the floor of the pocket.

☐ Under the Geometry area of the dialog, choose the **Cut Area** (1) icon and then **Select** (2).



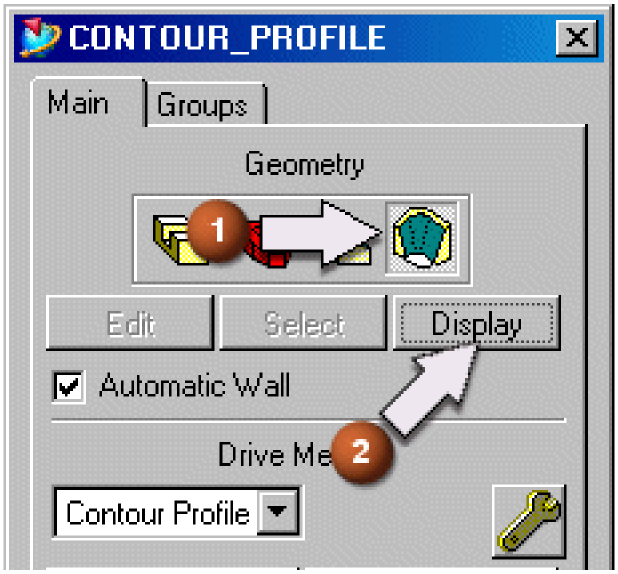
☐ Choose the bottom of the pocket as shown.



☐ Choose **OK**.

☐ Select the **Wall** icon from the Geometry area of the CONTOUR_PROFILE dialog.

☐ Choose **Display**.

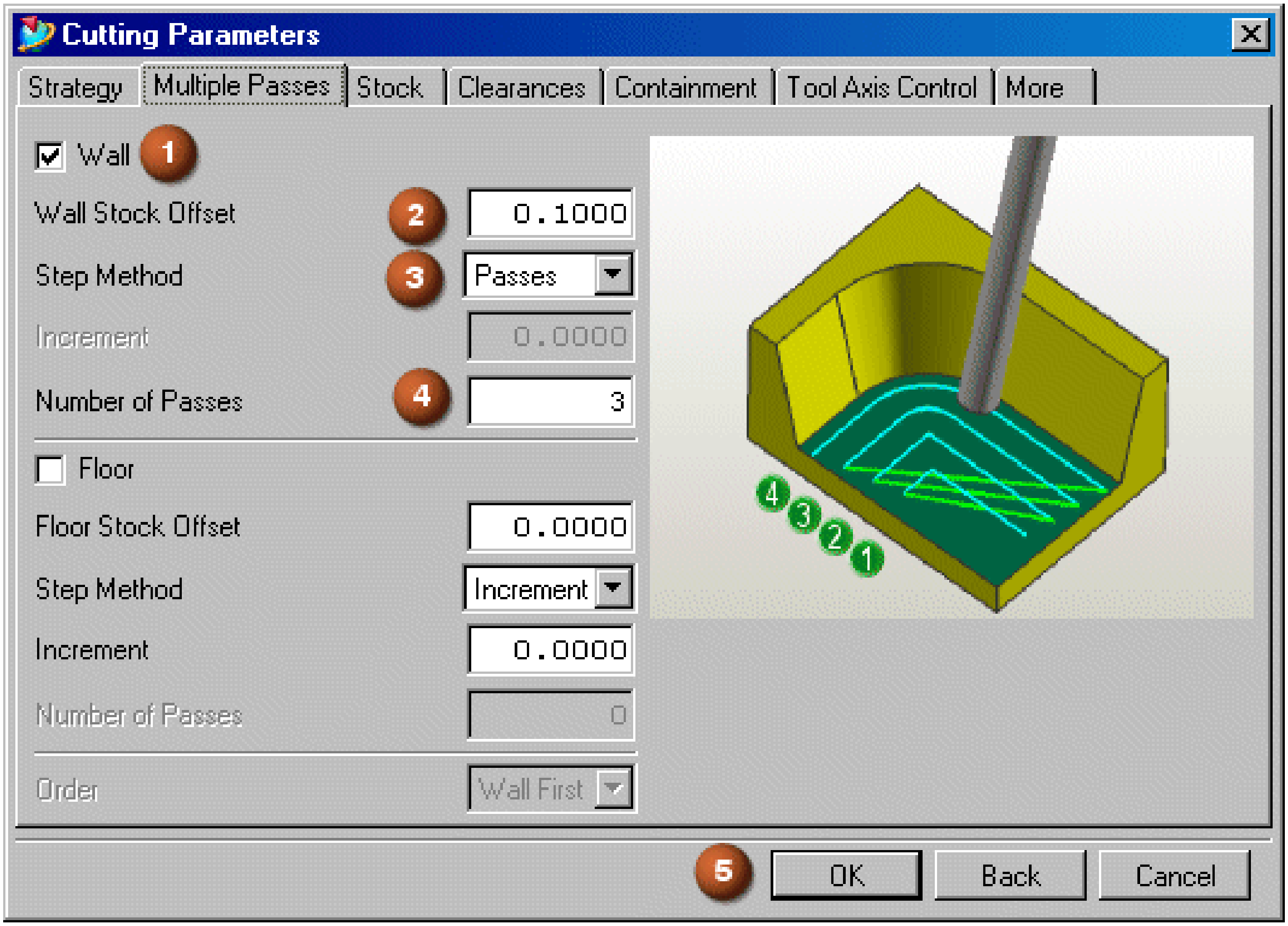


Note that the Automatic Wall parameter is On. The walls, forming the sides of the pockets are automatically detected (even though the floor is a radius). The operation is now ready to be generated, however we need to make multiple passes to keep the cutter from deflecting. You will now select those parameters.

☐ Choose the **Cutting** button from the CONTOUR_PROFILE dialog.

The Cutting Parameters dialog is displayed.

- ☐ Select the **Multiple Passes** tab.
- ☐ Turn on the **Wall** (1) option.
- ☐ Key in **0.1** for the **Wall Stock Offset** (2).
- ☐ Set the **Step Method** to **Passes** (3).
- ☐ Set the **Number of Passes** to **3** (4).



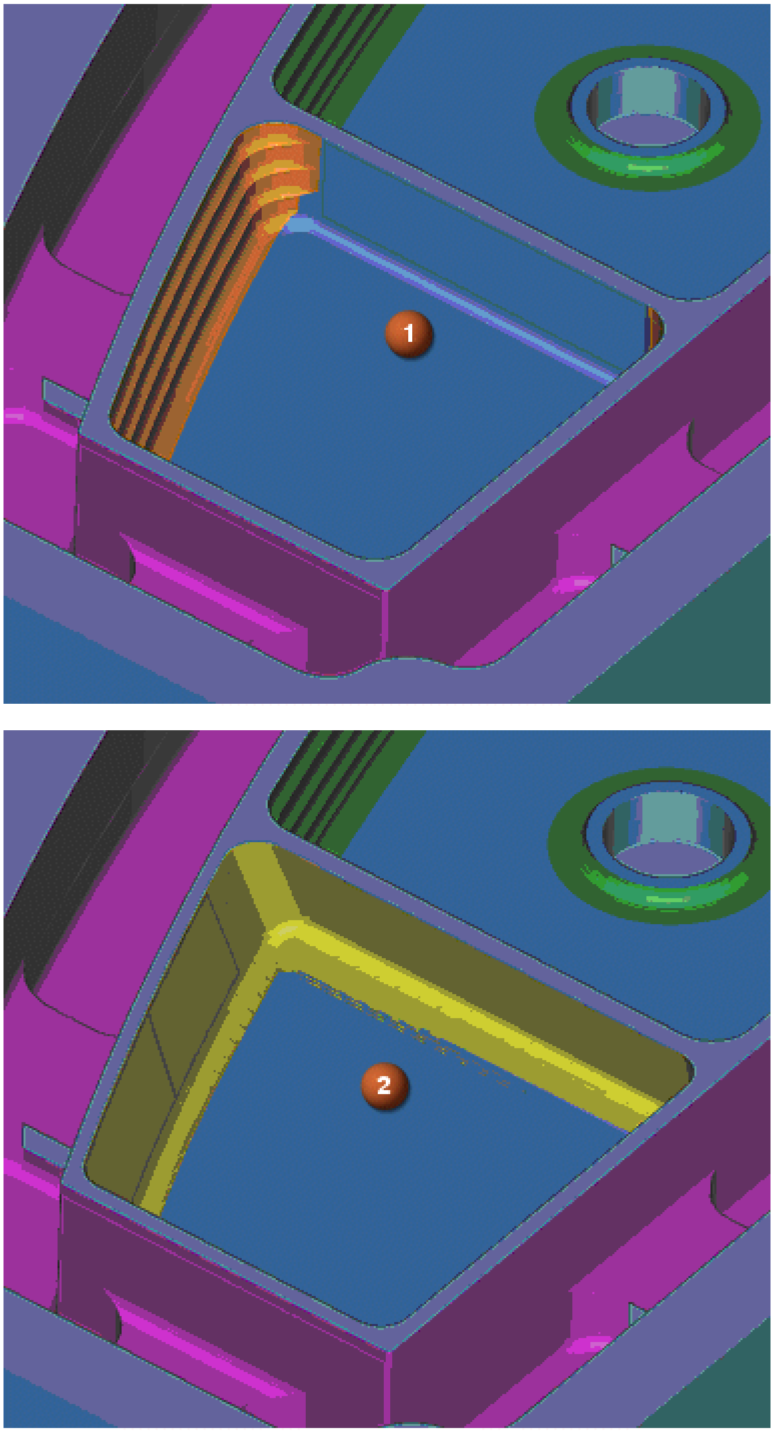
- ☐ Choose **OK** (5).

  You have set the cutting parameters to remove .100 stock in three equally spaced passes.

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

- ☐ Choose the **Generate** button from the CONTOUR_PROFILE dialog.

☐ Examine the tool path.



(1) Tool path prior to stock removal; (2) tool path after stock removal

☐ If time permits, create a second Contour Profile operation to machine the walls of the next pocket.

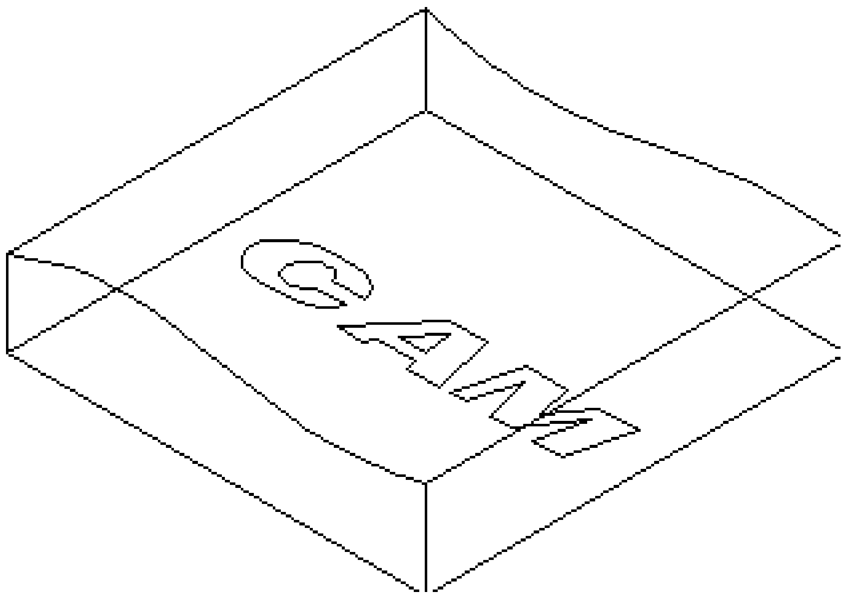
☐ Close the part file.

Activity: Overview of Variable Contour Options

In this activity, you will review the basic methods that Variable Contour uses to create tool paths. You will observe that some of the Fixed Contour options are not available in Variable Contour, as well as some options are only available in Variable Contour.

Step 1: Open an existing part file.

☐ Open the part file **mam_vx_0**.



☐ Enter the **Manufacturing** application.

☐ Select the Operation Navigator tab from the toolbar.

Step 2: Review an existing operation.

You will review the options by examining their settings.

☐ In the Operation Navigator, expand the Program named **OVERVIEW** and double-click on the operation named **REVIEW**.

The Variable Contour dialog is displayed.

Step 3: View the Variable Contour dialog options.

You will review the option settings on the Variable Contour dialog, then you will note the option settings on the Surface Area Drive Method dialog. These options are required to create the tool path.

☐ Choose the **Groups** tab and if necessary, choose the **Geometry** radio button, then select **Display**.

The geometry Parent Group, WORKPIECE, is displayed. This is the Part geometry.

☐ **Refresh** the graphics window and select the **Main** tab.

☐ Under the Geometry label, note that the Part icon is already selected. Choose **Display**.

The geometry Parent Group named WORKPIECE is displayed since it was selected as the part geometry.

☐ Choose the **Check** icon.

Note that no previous Check geometry was selected.

☐ Under the Drive Method label, view the Drive Methods that are available.

Note that the Area Milling, Flow Cut and Text Drive Methods are not available and are grayed out.

☐ On the Variable Contour dialog, under the Tool Axis label, note the various tool axes which are available.

Step 4: View the Surface Area Drive Method settings.

The Surface Area Drive Method is the most commonly used method of creating variable axis tool paths.

☐ Under the Drive Method label, choose **Surface Area**.

The Surface Drive Method dialog is displayed.

☐ Under the Drive Geometry label, choose **Display**.

Note that the top face was selected as the Drive Geometry. The Drive Points will be generated on this surface and projected to the part geometry based on the Projection Vector.

☐ Under the Projection Vector label, choose **Specify Vector**.

The Projection Vector is **I=0, J=0, K=-1** and is displayed pointing downward.

The Drive Points will be projected to the top of the part geometry, which are also their current location.

☐ Choose **Cancel**.

☐ Choose **Display Drive Path**.

The temporary Drive Points are displayed (which are used to create contact points.

☐ Choose **Display Contact Points.**

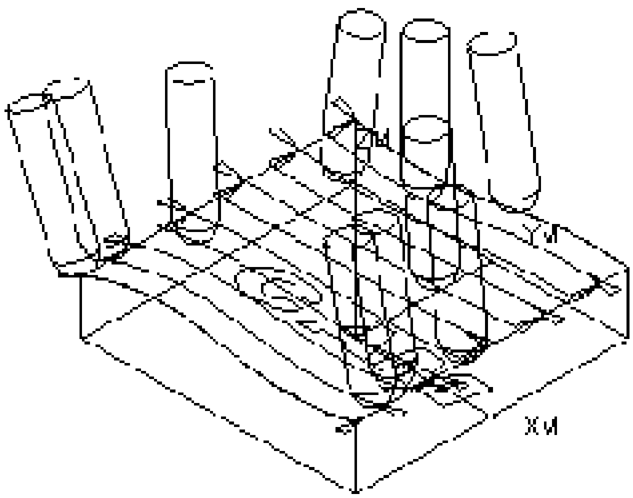
The surface normals are displayed at each tool contact point. The Surface Area Drive Method is the only Drive Method that allows you to display contact points.

☐ Choose **Cancel**.

Step 5: Generate and view the tool path.

You will now create a tool path using the settings which you just reviewed.

☐ **Generate** the tool path.



☐ **Close** the part file.

Tool Axis Control

The Variable Contour Tool Axes can be grouped based on the geometry that determines the tool axis.

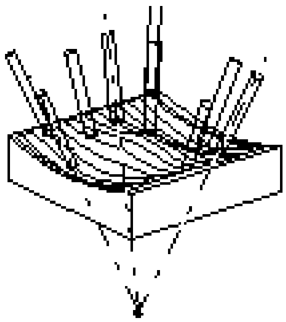
The choice of tool axis depends upon the Drive Method you choose. For instance, the Surface Area Drive Method allows you to specify many 4 and 5 axis tool positions that are not available by using any other Drive Method. The table which follows shows the various drive methods with permissible tool axis:

Tool Axis	Drive method					
	Curve/ Point	Spiral	Bndry	Surface Area	Tool Path	Radial
Away From point	X	X	X	X	X	X
Toward Point	X	X	X	X	X	X
Away From Line	X	X	X	X	X	X
Toward Line	X	X	X	X	X	X
Relative to Vector	X	X	X	X	X	X
4–axis Norm. To Part	X	X	X	X	X	X
4–axis Rel. To Part	X	X	X	X	X	X
Dual 4–Axis on Part	X	X	X	X	X	X
Inter- polate	X			X		
Swarf Drive				X		
Normal To Drive				X		
Relative To Drive				X		
4–axis Norm. To Drive				X		
4–axis Rel. To Drive				X		

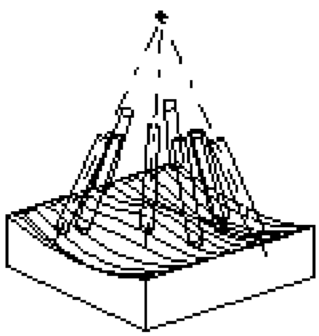
Dual 4–Axis on Drive				X		
Same as Drive Path					X	

Point and Line Tool Axes

The following tool axis types use focal points and can produce 5-axis movements:

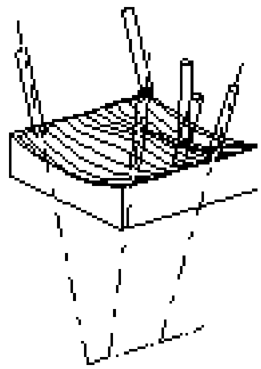


Away From Point

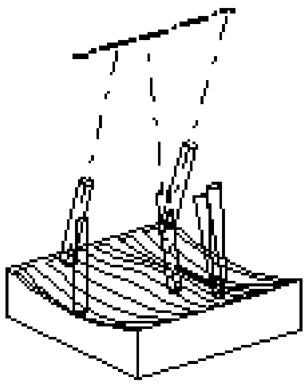


Towards Point

The following tool axis types use focal lines and can produce 4-axis movements:



Away From Line



Towards Line

Away and *Towards* refers to a vector direction.

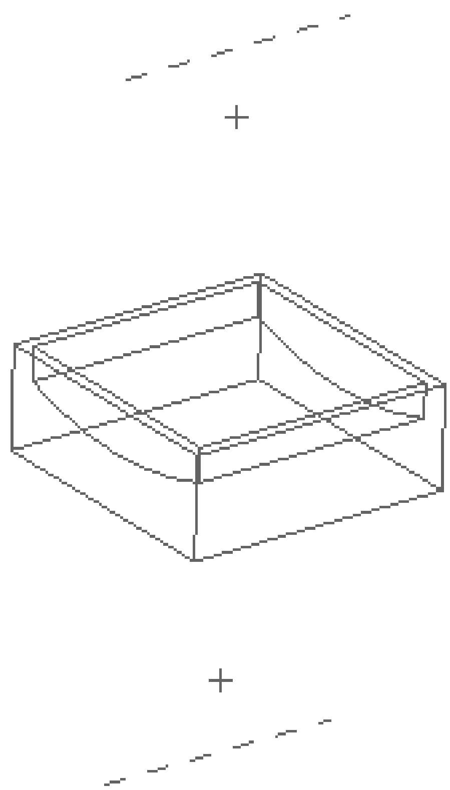
Consideration must be given to machine configuration, part fixturing and amount of swing or tilt of the table and or head when selecting tool axis types. It is advisable to select the method which minimizes the amount of table and or head tilt.

Activity: Point and Line Tool Axis Types

In this activity, you will replay a series of Variable Contour operations that use point and line geometry to control the tool axis.

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

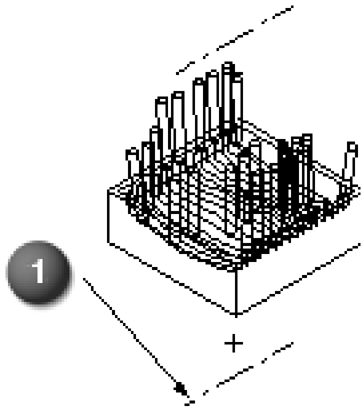
☐ Open the part file **mam_vx_4**.



☐ If necessary, enter the Manufacturing application and the display the Program Order view in the Operation Navigator.

Step 2: Replay the operations.

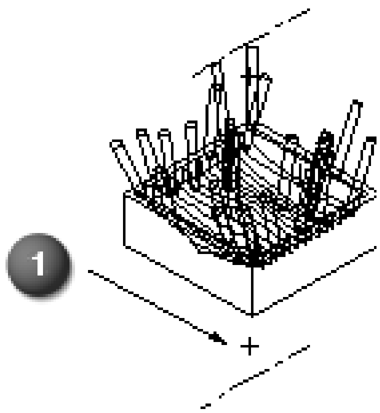
☐ **Replay** the **AWAYLINE** operation.



(1) Focal line used with tool axis

☐ The tool path is replayed using the tool axis option **Away from Line**.

☐ **Replay** the **AWAYPT** operation.



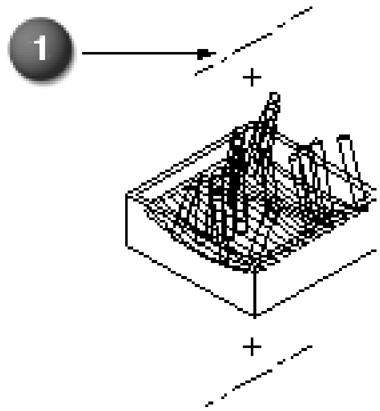
(1) Focal point used with tool axis

The tool path is replayed using the tool axis option **Away from Point.**

Notice the amount of difference in tool tilt between the two different methods. Proper placement of the focal point and line can greatly reduce the amount of tool tilt resulting in reduced risk of head or tool interference with clamps and or fixturing.

☐ **Replay** the **TOWARDLINE** operation.

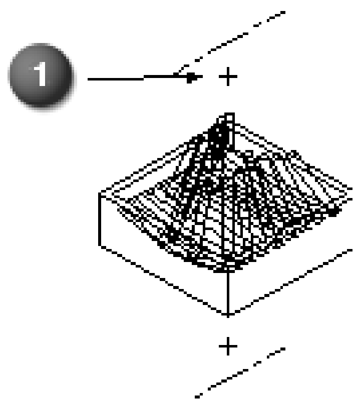
☐ **Replay** the tool path.



(1) Focal line used with tool axis

The tool path is replayed using the tool axis option **Towards Line.**

☐ **Replay** the **TOWARDPT** operation.



(1) Focal point used with tool axis

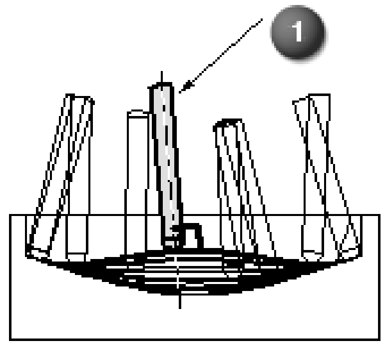
The tool path is replayed using the tool axis option **Towards Point**.

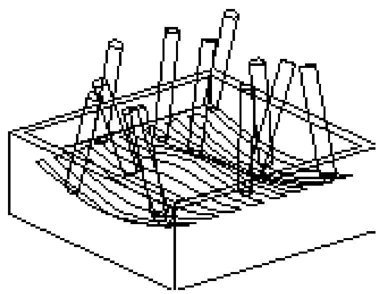
Notice the difference in the amount of tool tilt. The method chosen, **towards** or **away** from a **point** or **line**, along with their respective placement of the geometry being cut, gives you precise control of the tilt of the tool.

☐ **Close** the part file.

Normal Tool Axis

Normal Tool Axis maintains a tool axis that is perpendicular to the part geometry, drive geometry, or rotational axis (4-axis) at each contact point. This is a preferred method of tool axis control when the contoured geometry that is being machined does not change radically in shape and or direction.





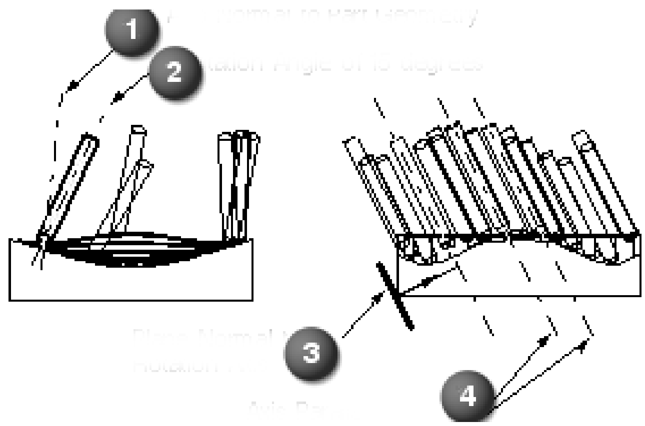
(1) Normal to part geometry at each drive point

The following tool axis types use the Normal tool axis:

- Normal To Part
- 4-axis Norm To Part
- Normal To Drive Surf (Surface Area Drive)
- 4-axis Norm To Drive (Surface Area Drive)

The 4-axis type options allow you to apply a rotational angle to the tool axis. This rotational angle effectively rotates the part about an axis as it would on a machine tool with a single rotary table. The 4-axis orientation causes the tool to move within planes which are normal to the defined rotational axis.

In the following example, the rotational angle causes the tool axis to lean forward in relation to an otherwise normal tool axis.



(1) axis normal to part geometry

(2) rotation angle of 15 degrees

(3) plane normal to rotation axis

(4) axis parallel to plane

Relative Tool Axis

Relative tool axis maintains a tool axis that is perpendicular to the part geometry, drive geometry, or rotational axis (4-axis) at each contact point and allows the application of Lead or Tilt angle to the tool axis.

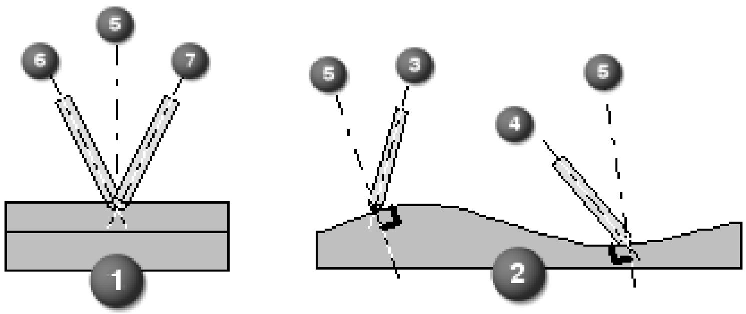
You can apply Lead or Tilt to the following tool axis types:

- Relative To Part
- 4-axis Rel To Part
- Relative to Vector
- Dual 4-axis
- Relative To Drive (Surface Area Drive)
- 4-axis Rel To Drive (Surface Area Drive)

Lead and Tilt Angle

Lead Angle defines the angle of the tool forward or backward along the tool path. A positive Lead Angle leans the tool forward based on the direction of the tool path. A negative Lead Angle (lag) leans the tool backwards based on the direction of the tool path.

Tilt Angle defines the angle of the tool, side to side. A positive value tilts the tool to the right as you look in the direction of cut. A negative value tilts the tool to the left.



(1) Tool direction (front view)

(2) Tool direction (right view)

(3) Lead

(4) Lag

(5) Normal axis

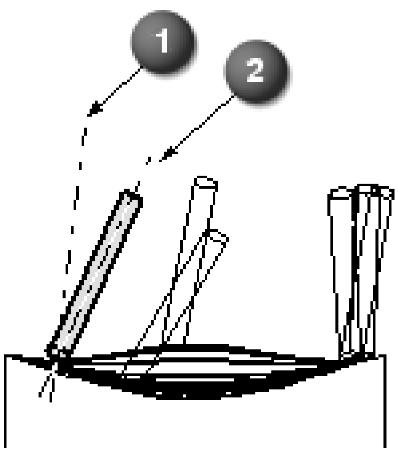
(6) Negative tilt

(7) Positive tilt

You can specify a Minimum and Maximum angle of movement for the Lead and Tilt of the tool axis.

Unlike a Lead angle, a 4-axis rotational angle always leans to the same side of the normal axis and is independent of the direction of the tool movement.

The rotational angle causes the tool axis to lean to the right of the part geometry normal axis in both zig and zag moves. The tool moves within planes normal to the defined rotational axis.



(1) axis normal to part geometry

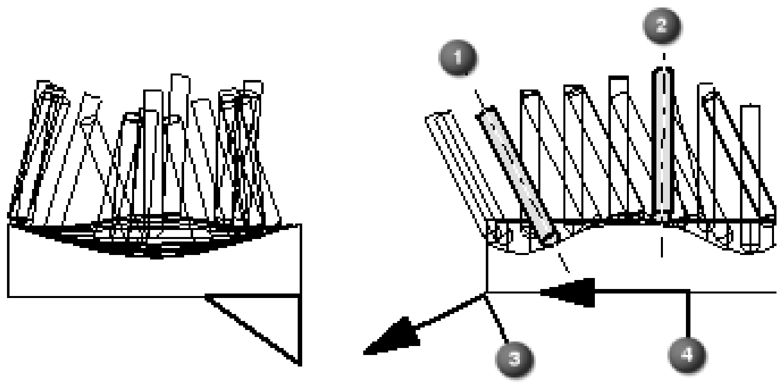
(2) rotational angle of 15 degrees

Dual 4-Axis

Dual 4-Axis applies rotational, Lead and Tilt angle to the Zig and the Zag moves independently.

You can specify a 4-axis rotation angle, a lead angle, and a tilt angle. The 4-axis rotation angle rotates the part about an axis as it would on a machine tool with a single rotary table.

In Dual 4-Axis mode, these parameters may be defined separately for Zig and Zag moves.



(1) zig cut

(2) zag cut

(3) zig cut, tool axis

(4) zag cut, tool axis

Activity: Normal to Part and Relative to Part

In this activity, you will compare two similar and frequently used tool axes; normal to Part and Relative to Part.

Step 1: Open a new part file.

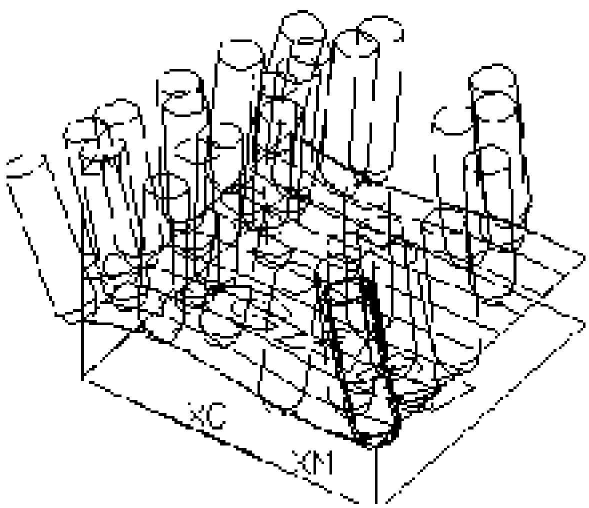
☐ Open the part file **mam_vx_0** and enter the **Manufacturing** application.

Step 2: View the tool path.

Note the tool axis in the first pass. The tool axis is **Normal to Part**, always perpendicular to the part geometry.

☐ Expand the **TOOL_AXIS** Program Parent Group.

☐ **Replay** the operation **NORM_PART**.



You will change the Tool Axis to **Relative to Part** and compare the tool paths.

Step 3: Create a tool path using Relative to Part Tool Axis.

☐ **Edit** the operation **NORM_PART**.

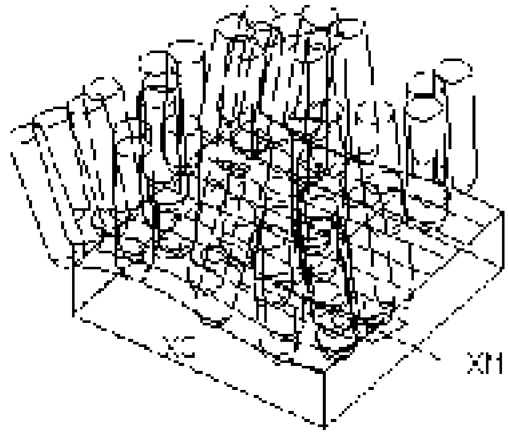
The Variable Contour dialog is displayed.

☐ Under the Tool Axis area of the dialog, select **Relative to Part** as the tool axis.

You are prompted to change the Lead and Tilt angles. Use the defaults of 0°

☐ Choose **OK**.

☐ **Generate** the tool path.



Compare this tool path to the previous one. Note that the tool paths are nearly identical. Both tool paths are created using the surface normal at each contact point.

☐ Choose **Cancel**.

Step 4: Use Lead with Relative to Part Tool Axis.

You will now see the effect of adding a **Lead angle** to the **Relative to Part** tool axis.

☐ **Edit** the operation **REL_PART_LEAD**.

The Variable Contour dialog is displayed.

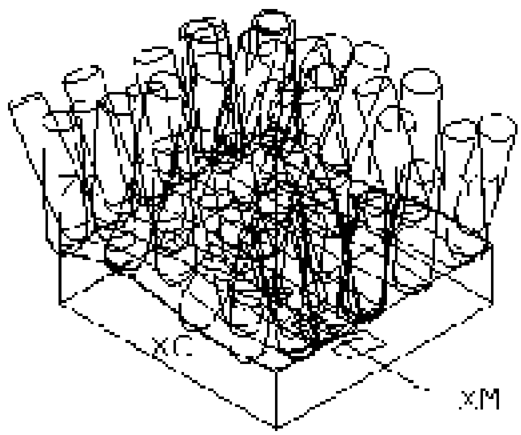
☐ Under the Tool Axis label, choose **Relative to Part**.

You are prompted for **Lead** and **Tilt** angle settings.

You will use the specified settings, which are exaggerated so that you can easily see the angle of **Lead**.

☐ Choose **OK**.

☐ **Generate** the tool path.



Note that the tool leans forward as it cuts.

☐ Choose **Cancel**.

Step 5: Use Tilt with a Relative to Part Tool Axis.

This time you will see the effect of adding a **Tilt** angle to the **Relative to Part** tool axis.

☐ **Edit** the operation **REL_PART_TILT**.

The Variable Contour dialog is displayed.

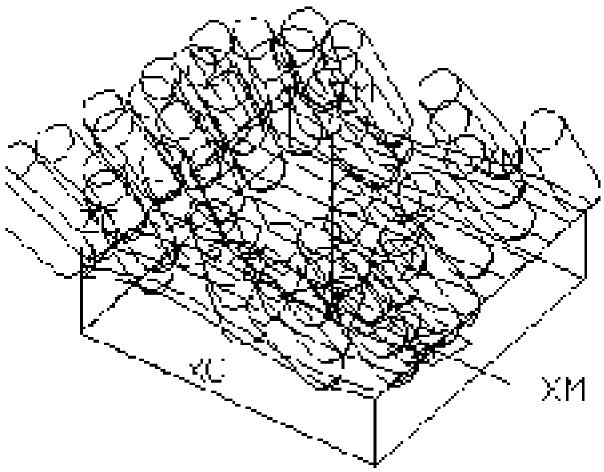
☐ Under the Tool Axis label, choose, **Relative to Part**.

You are prompted for **Lead** and **Tilt** angle settings.

Note the specified settings.

☐ Choose **OK**.

☐ **Generate** the tool path.



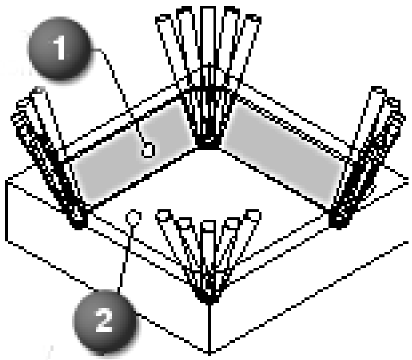
Note that the tool tilts to the right as it cuts.

☐ Choose **Cancel**.

☐ Close the part file.

Swarf Drive Tool Axis

Swarf Drive tool axis maintains a tool axis that is parallel to the drive geometry. The drive geometry guides the side of the tool while the part geometry guides the end of the tool.

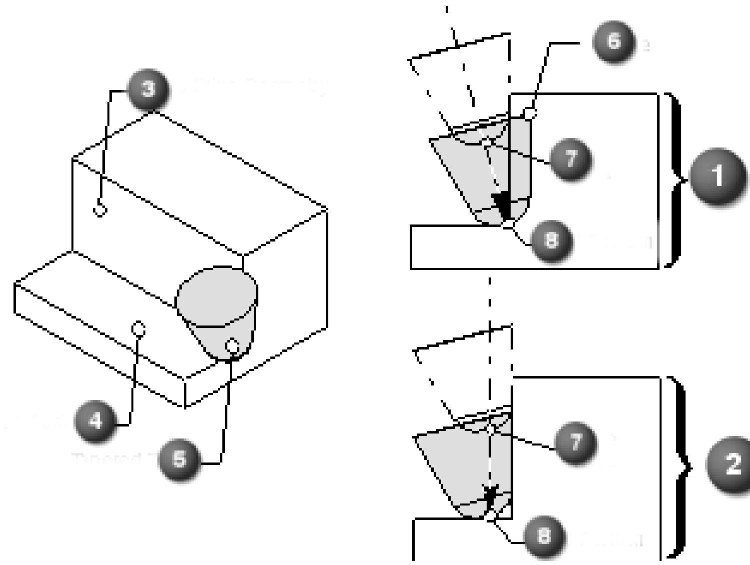


(1) drive geometry

(2) part geometry

The Swarf Drive tool axis should be used only when the drive geometry consists of ruled surfaces, since the drive geometry rulings define the swarf ruling projection vector.

This projection vector can prevent the gouging of the drive geometry when using a tapered tool as shown by the following:



(1) tool axis projection vector

(2) swarf ruling projection vector

(3) ruled drive geometry

(4) part surface

(5) tapered tool

(6) gouge

(7) drive point

(8) tool position

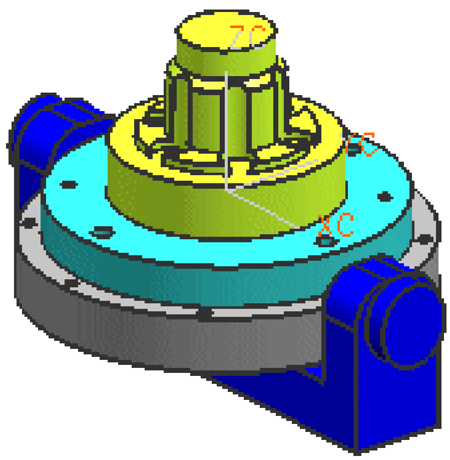
In this example, a comparison is made between the Swarf Drive Projection Vector and the Tool Axis Projection Vector. The drive points are projected along the specified vector to determine the tool position, showing the Tool Axis Projection Vector method gouging the drive geometry, while the Swarf Ruling Projection Vector method results in the tool positioning tangent to the drive geometry.

Activity: Using Special Tool Axis and non Part Geometry

The part in this activity has been partially machined. You are going to continue to machine the core for a hub cover used on a four wheel drive vehicle. To maximize the part finish, you will be using a short tool to prevent cutter deflection.

Step 1: Open an existing part file.

☐ Open the part file **mam_hub_core_mfg_asmb**.



There are two existing sample operations that you will examine and then create like operations. First you will examine the various parts which comprise the assembly.

☐ Save the part as *****_hub_core_mfg_asmb**.

Step 2: Examine the assembly.

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

☐ Choose the Assembly Navigator tab from the toolbar.

The Assembly Navigator and the part model are displayed.

Note that the assembly consists of a mounting plate, compound rotary table, numerous bolts and the hub cover core part.

Step 3: Examine various operations.

☐ Choose the Operation Navigator tab from the toolbar.

The Operation Navigator is displayed.

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

☐ Examine the various operations.

Note that the HUB-PROJECT-PROGRAM group object contains a rough and finish operation.

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

Note the various tools that are defined.

☐ Choose the Assembly Navigator tab from the toolbar.

For creating additional operations, it would be somewhat easier for selection and visualization purposes, to remove from the display, various parts of the manufacturing setup.

☐ Select the red check marks for the screws (mam_soc_hd_screw.5x8), table assembly (mam_compound_table_asmb) and mounting plate (mam_mounting_plate). This will turn off the display of these components.

Step 4: Create the operations to finish the fluted area of the part.

☐ Choose the **Create Operation** icon.

☐ If necessary, set the Type to **mill_multi_axis**.

☐ Choose the **VARIABLE_CONTOUR** icon.

☐ Set the Parent Groups as follows:

Program: **HUB-FINISH**

Use Geometry: **WORKPIECE**

Use Tool: **BALL_MILL-.75**

Use Method: **MILL_FINISH**

Name: **vc_flute_fin**

☐ Choose **OK**.

The Variable_Contour dialog is displayed.

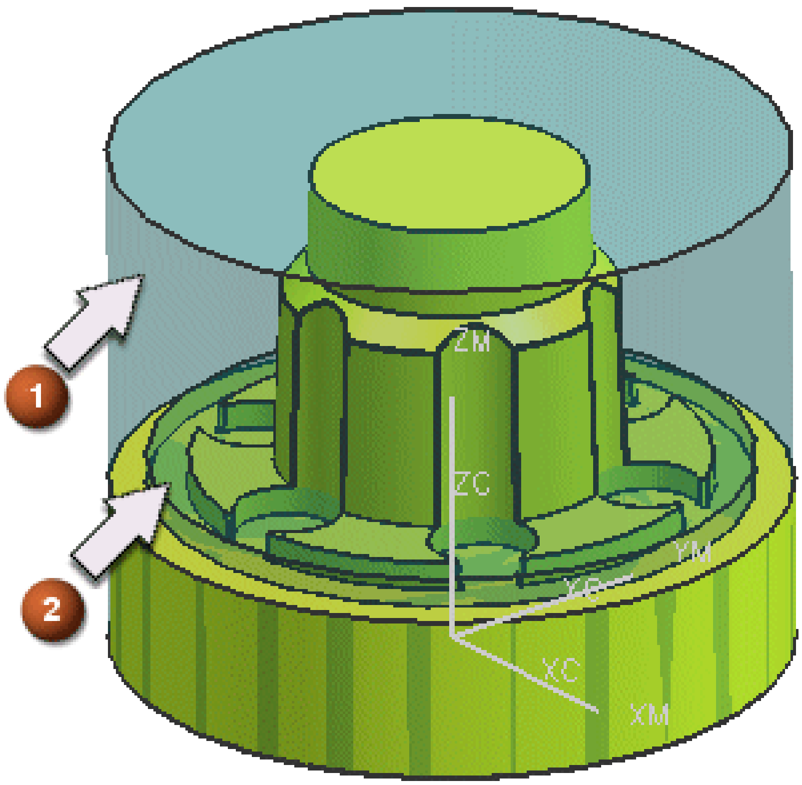
☐ Change the Drive Method from Boundary to Surface Area.

☐ Choose **OK** on the Drive Method Information dialog.

You will now select the drive geometry to control the tool motion. The part consists of many faces which are irregular in shape and uneven in contour. You will begin the selection process by selection of the outer face of the cylinder that defines the raw stock.

☐ Make Layer 2 and 5 selectable.

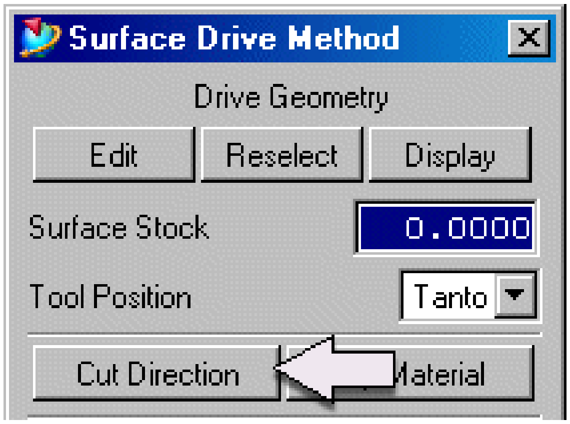
☐ Choose the **Select** button and select the outside face of the cylinder that represents the stock (1).



☐ Choose **OK**.

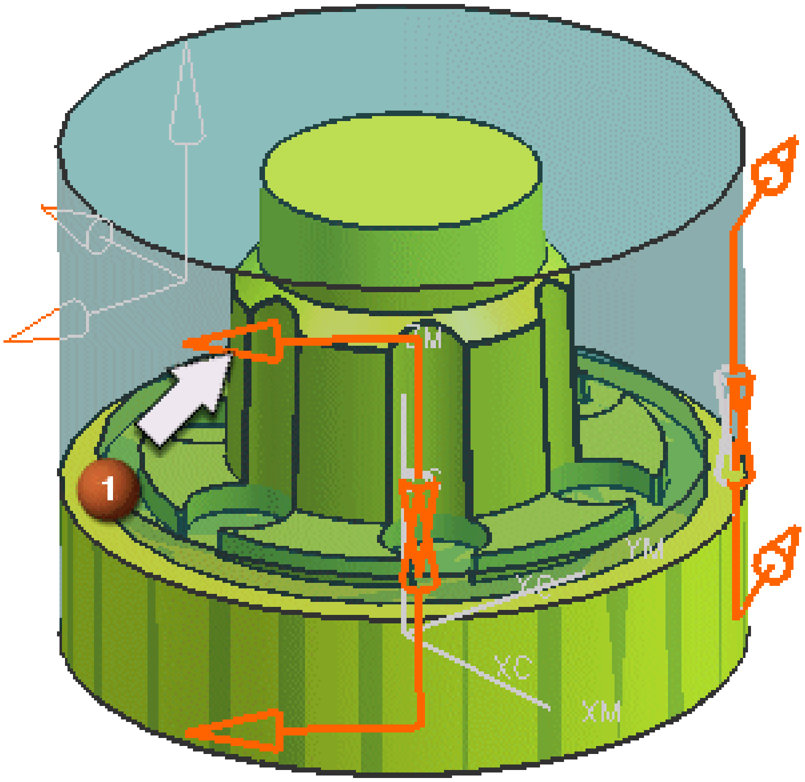
You will now set the direction of cut and its cut area in relation to the overall size of the outside face of the stock geometry. You will also set the Cut Type.

☐ Choose the **Cut Direction** button.

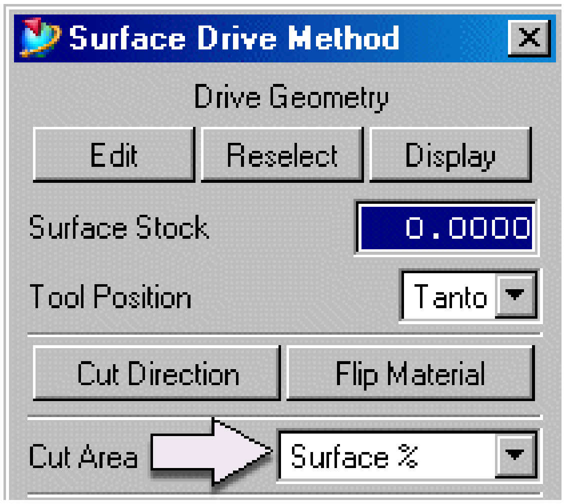


Cut direction vectors are displayed.

☐ Choose the vector as shown (1).



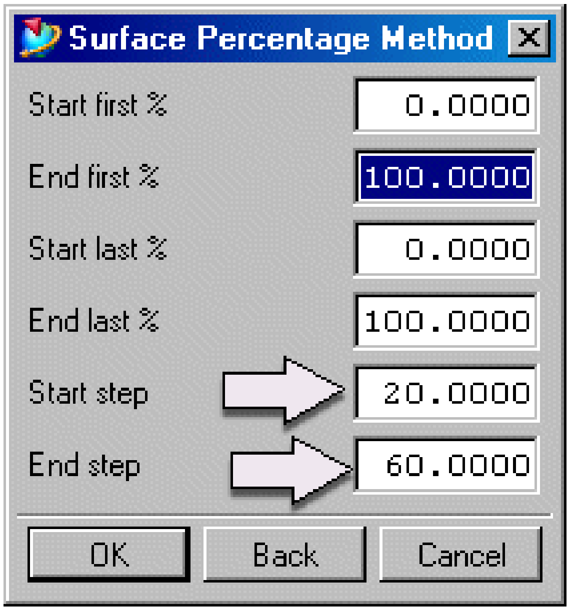
☐ Choose **Surface %** from the Cut Area pull-down menu.



Note the system highlight at the top and bottom of the cylinder.

☐ Refresh the screen.

☐ Set the start and end values as shown:



☐ Choose **OK**.

Note the area that is now highlighted. The cutter will now be limited to this area which encompasses the flutes.

☐ Change the **Cut Type** to **Zig**.

You will now set the tool axis and projection vector.

- ☐ Change the Tool Axis from Normal to Part to **Relative to Drive**.
- ☐ Set the **Tilt** angle to **45**.
- ☐ Choose **OK**.
- ☐ Set the Projection Vector to **Toward Line**.

  The Line Definition dialog is displayed.
- ☐ Choose the **Point and Vector** button.
- ☐ Choose **OK** on the Point Constructor dialog (accept the defaults).

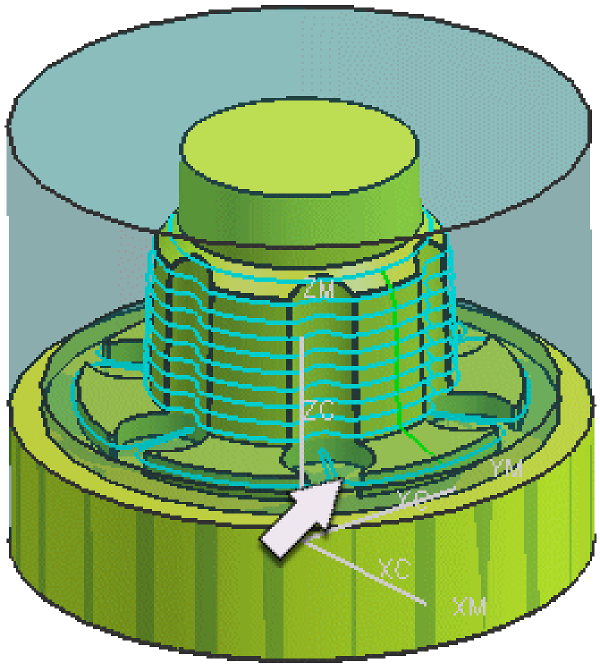
  The Vector Constructor dialog is displayed.
- ☐ Choose the **ZC Axis** icon
- ☐ Choose **OK**, twice.

  The Variable_contour dialog is displayed.

Step 5: Generate the tool path.

- ☐ Choose the **Generate** icon.

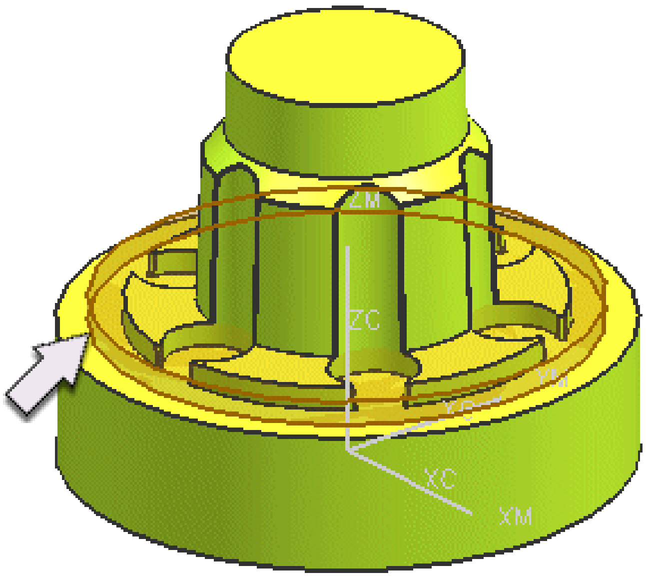
  Notice the last pass make an erratic move next to the clearance hole near the bottom of the part.



To avoid this move, you will need to select Check (2) geometry.

☐ Choose the Check geometry icon from the Variable_contour dialog.

☐ Choose the Select button and then choose the small disc area located near the bottom of the part.



You will now set the parameters used in collision detection.

☐ Choose **OK**.

☐ Choose the **Cutting** button.

☐ Select the **Clearances** tab.

☐ Select **Skip** from the **When Gouging** pull-down menu.

☐ Set the Check Safe Clearance parameter to **.01**

☐ Choose **OK**.

☐ Choose Generate and review the tool path.

☐ Close the part file.

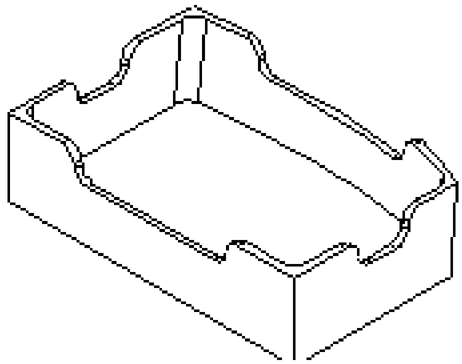
Activity: Swarf Drive Tool Axis

In this activity, you will create an operation to finish the walls of a tapered walled part using the Swarf Drive Tool Axis. The part will be modified in order to start the tool path at the center of one of the walls, which prevents the cutter from engaging the interior corner of the part.

Step 1: Open a part file and rename it for the current activity.

☐ Open the part file **mam_tub_ftg_mfg_asmb**.

☐ Rename the part to ******_tub_ftg_mfg_asmb***.



To prevent engaging and then gouging an interior corner of the part, you will engage the part from the middle of one of the long sides of the part. You must first subdivide one of the tapered side walls by creating a curve (in this case a line) before you can subdivide the face.

Step 2: Enter the Assemblies application and create a WAVE Linked body used for dividing the faces.

You will first change the work layer to the layer used for manufacturing data and will then create the WAVE linked body.

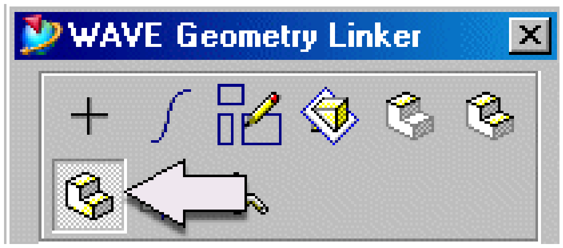
☐ Verify that Assemblies is turned on.

☐ Set the Work layer to **151**.

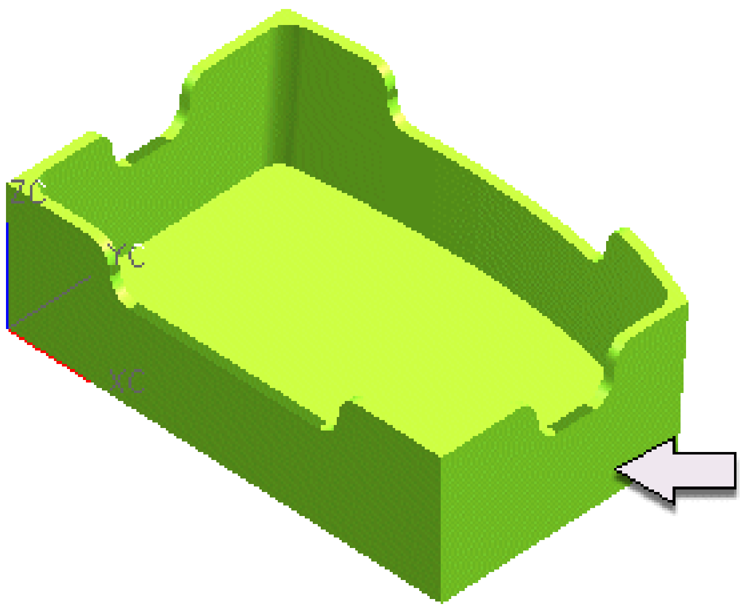
☐ Choose **Insert→Associative Copy→Wave Geometry Linker** from the main menu bar.

The WAVE Geometry Linker dialog is displayed.

☐ Select the **Body** icon.



☐ Select the solid body that is the part.



☐ Choose **OK**.

A linked body has been added to layer 151 which is associative to the engineering model (mam_tub_fitting) that appears in the Assembly Navigator. This linked body can now be modified, whereas the engineering model (mam_tub_fitting) can not.

Step 3: Turn off the display of the component and change the color of the linked body.

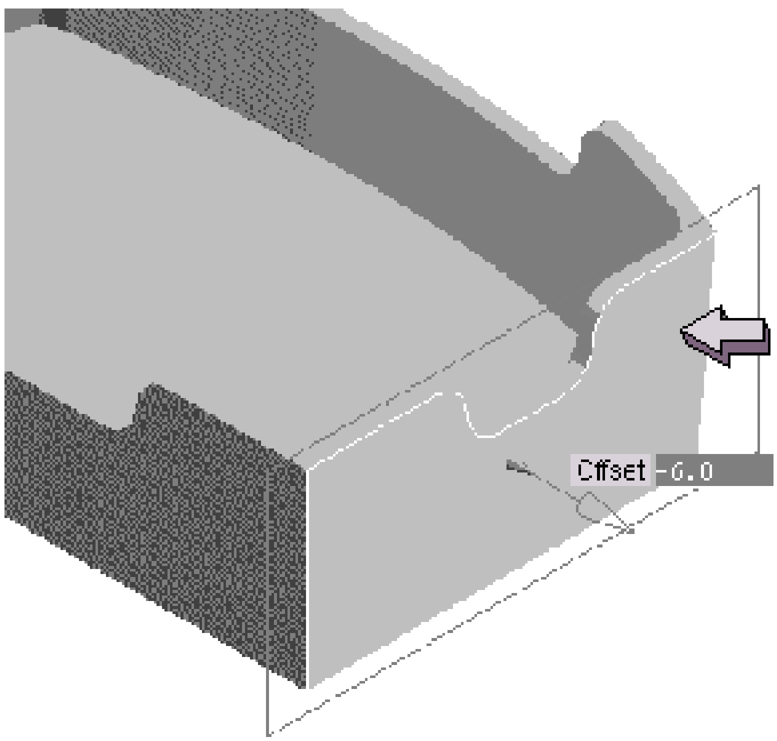
☐ Using the Assembly Navigator, turn off the display of the component by clicking the check mark in front of the **mam_tub_fitting** component (the check mark will turn from red to gray).

- ☐ Change the color of the linked body by choosing **Edit → Object Display** from the main menu bar.
- ☐ Choose the linked body and **OK** the selection.

  The **Edit Object Display** dialog is displayed.
- ☐ Change the color to one that is not the same as the engineering part.
- ☐ Choose **OK**.

Step 4: Enter the Modeling application, create a datum plane and curve for subdividing and subdivide the tapered sidewall face.

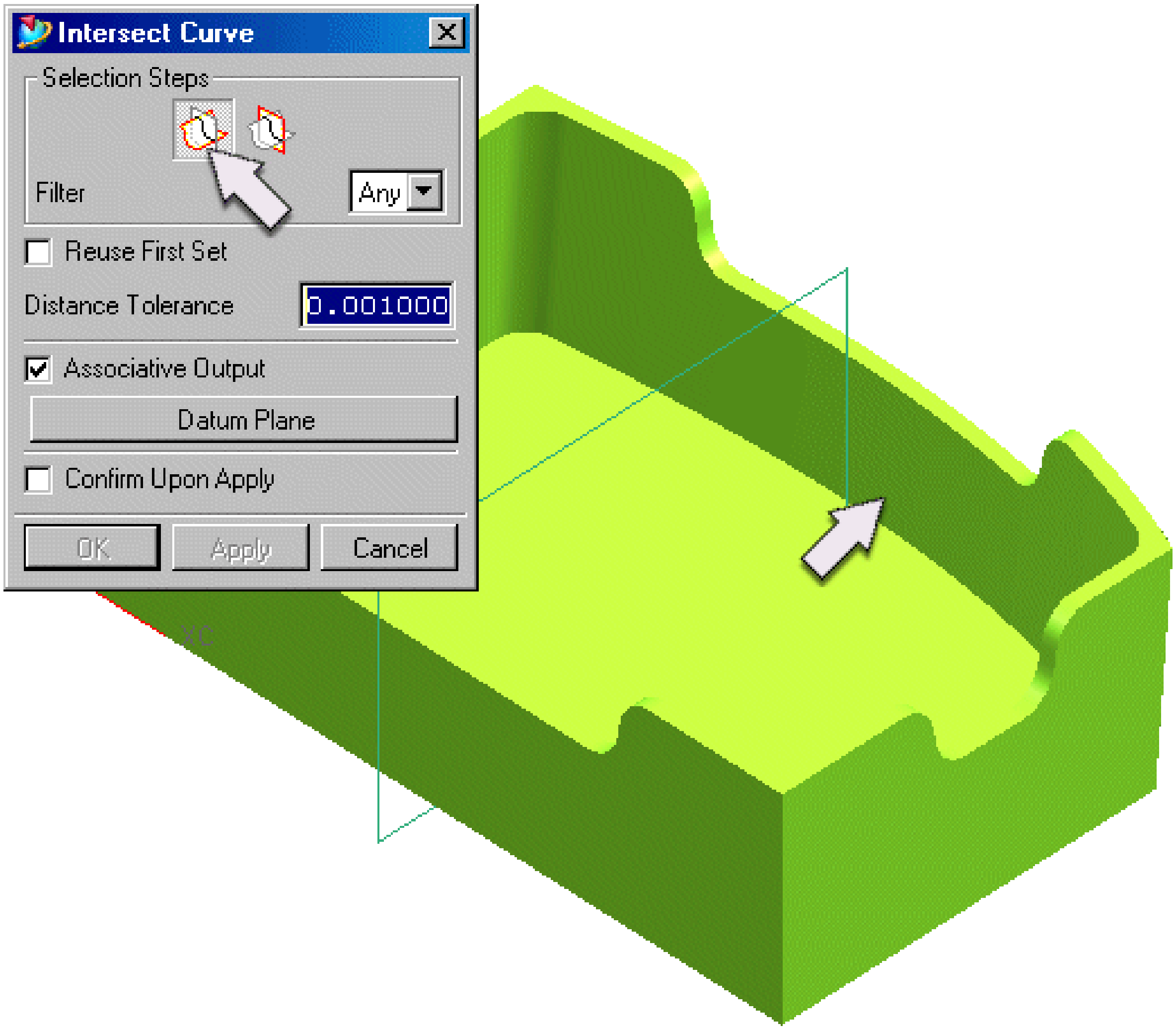
- ☐ Enter the **Modeling** application.
- ☐ Choose **Insert→Datum/Point→Datum Plane** from the menu bar.
- ☐ Select the end face and key in the value **-6.0** (Hint: use Offset as a constraint).



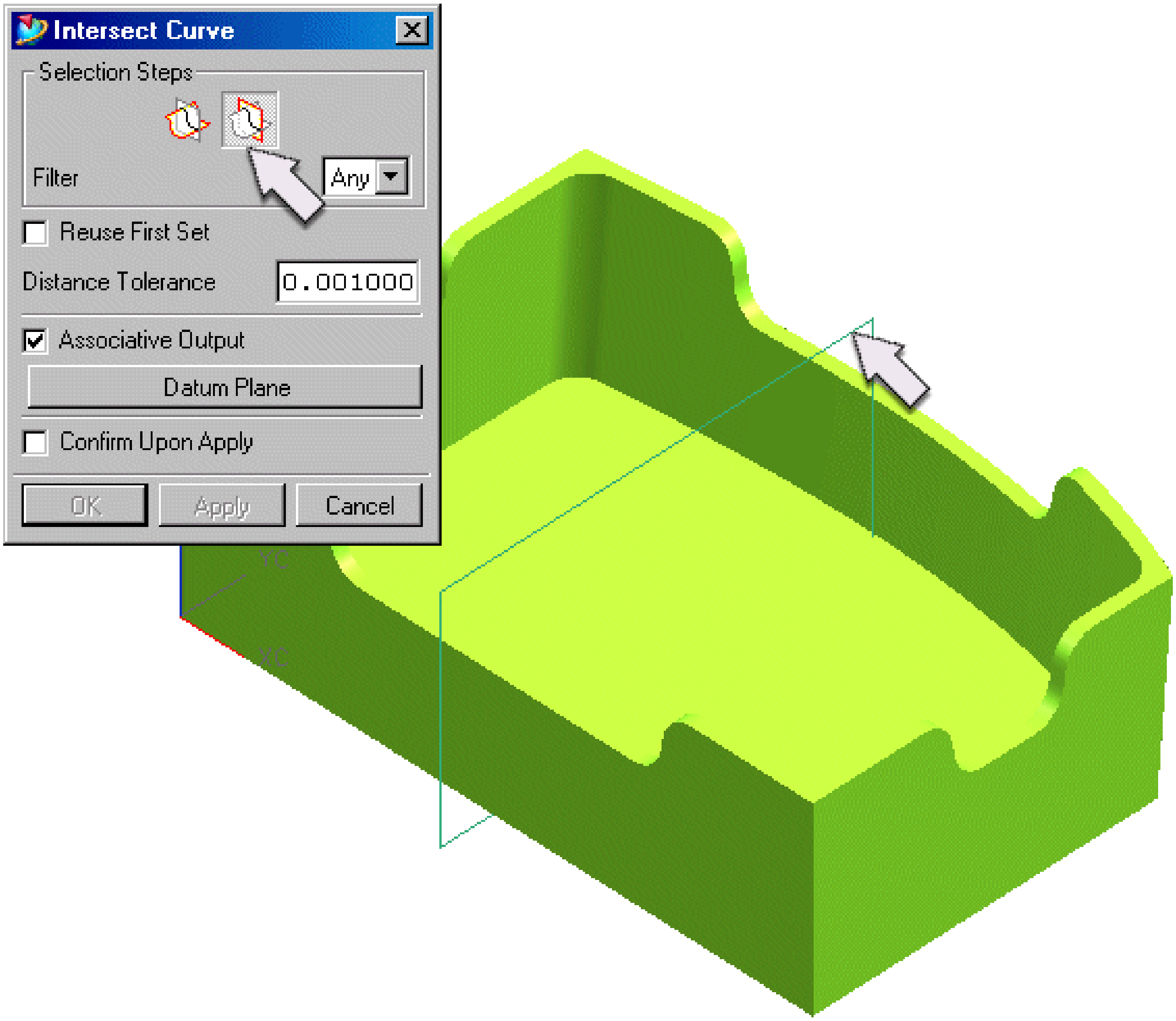
- ☐ Choose **OK**.
- ☐ Choose **Insert→Curve from Bodies→Intersect** from the menu bar.

The Intersect Curve dialog is displayed.

☐ Choose the **First Set** icon from the dialog and then select the contoured face.



☐ Choose the **Second Set** icon from the dialog and then select the datum plane.



☐ Choose **OK**.

You created a line that will be used to subdivide the face. A Datum Plane was used to create the curve to allow movement of the line easily. You will now move the Datum Plane to its proper layer and will then subdivide the face.

☐ Choose **Format→Move to Layer** and select the datum plane.

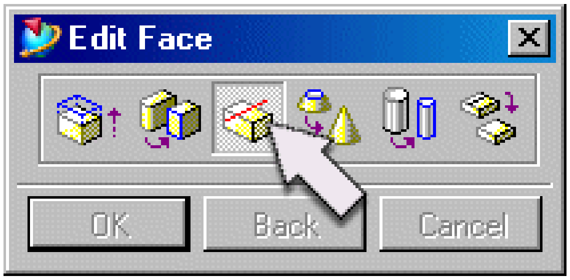
☐ Choose **OK**.

☐ In the **Layer Move** dialog, set the **Destination Layer** to **61**.

☐ Choose **OK**.

☐ Choose **Edit** →**Face** from the menu bar.

☐ Choose the **Subdivide Face** icon from the Edit Face dialog.



☐ Select the face to subdivide.

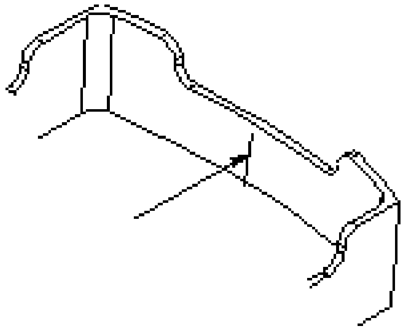
☐ Choose **OK** from the Selection Confirmation dialog.

The curve Subdivide Face dialog is displayed.

☐ Choose **Blank**.

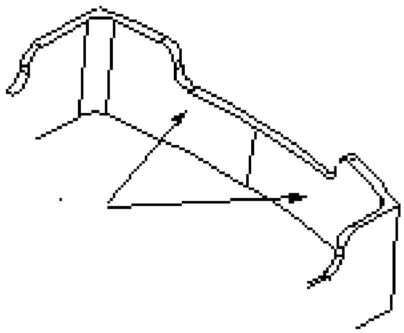
You must select the curve you are going to use to subdivide the face.

☐ Select the newly created line.



☐ Choose **OK**.

The face is divided into two faces.



☐ Choose **Cancel**.

☐ From the menu bar, choose **Preferences**→ **Selection**.

☐ Set the Chaining tolerance to **.001**.

☐ Choose **OK**.

You will now create the tool path to cut the part.

Step 5: Change to the Manufacturing application.

☐ Enter the **Manufacturing** application.

☐ Choose **mill_multi-axis** as the CAM Session Configuration.

☐ Choose **mill_multi-axis** as the CAM Setup.

This will initialize the part with multi-axis parameters.

☐ Choose **Initialize**.

Step 6: Create an end mill needed to machine the part.

☐ Choose the **Create Tool** icon.

☐ Choose the **Mill** tool icon.

☐ Name the tool **EM-.75-.06**.

☐ Choose **OK**.

☐ In the Milling Tool-5 Parameters dialog, enter a diameter of **.75** and a lower radius of **.06**.

☐ Choose **OK**.

Step 7: Edit the MCS in the Operation Navigator and set the Clearance Plane 1.000 above the top face of the part.

☐ Double-click the **MCS** group object in the **Geometry** view of the Operation Navigator.

The **MILL_ORIENT** dialog is displayed.

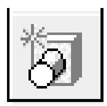
☐ Check the **Clearance** box and select the **Specify** button.

The **Plane Constructor** dialog is displayed.

☐ Set the clearance plane as being 1.000 above the uppermost top face of the part.

Step 8: Specify the Part Geometry.

You will now select the floor of the part as the Part Geometry.

☐ Choose the **Create Geometry** icon. 

☐ Set the Operation Navigator to the **Geometry View**.

☐ Choose the **Mill_Geom** icon.

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

☐ **Name** the geometry Parent Group **PART_FLOOR**.

Note that the **Type** is **mill_multi_axis**.

☐ Choose **OK**.

☐ In the **MILL_GEOM** dialog, under the Geometry label, choose the **Part** icon.

☐ Choose **Select**.

The **Part Geometry** dialog is displayed.

☐ The **Selection Options** should be set to **Geometry**.

☐ Change the **Filter Methods** to **Faces**.

☐ Select the floor of the part.

☐ Choose **Accept** from the **Selection Confirmation** dialog.

☐ Choose **OK** until you return to the **Create Geometry** dialog.

☐ Verify that the Parent Group, **PART_FLOOR** was created in the Operation Navigator.

Step 9: Create the Variable Contour operation using the Surface Area Drive Method.

☐ Choose the **Create Operation** icon.

☐ Choose the **Variable Contour** icon.

☐ Set the following:

Program: **Program**

Use Geometry: **PART_FLOOR**

Use Tool: **EM-.75-06**

Use Method: **MILL_FINISH**

Name: **fin-poc-walls**

☐ Choose **OK**.

The Variable Contour dialog is displayed.

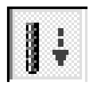
Step 10: Define an Engage and Retract move using the Non-cutting Moves option.

☐ Choose **Non-cutting** from the Machining Parameters area.

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

You will now define an **Approach** move for the **Default Case**.

You can also define different **Approach** moves for the **Initial**, **Final Check**, **Local**, and **Reposition** moves.

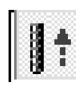
☐ Choose the **Approach** icon. 

The options available have changed to reflect the **Approach** move options.

☐ Next to the **Status** label, change **None** to **Clearance**.

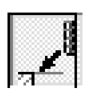
The **Clearance** geometry is assigned to the default **Approach** move. The **Non-cutting Moves** dialog is displayed again.

You will now specify a **Departure** move for the **Default** case.

☐ Choose the **Departure** icon. 

☐ Next to the Status label, change **None** to **Clearance**.

You will now define an **Engage** move for the **Initial** case.

☐ Choose the **Engage** icon. 

☐ Next to the **Status** label, change **Use Default** to **Manual**.

☐ Under the **Movement** label, change **Linear** to **Arc: Normal Tool axis**.

☐ Change the Radius Type to **Radius**.

☐ Key in **.500** into the Radius value field.

Step 11: Set the Display options.

☐ Choose the **Edit Display** icon in the Tool Path section of the dialog.

The Display Options dialog is displayed.

☐ Set the following options:

Tool Display = 3D

Frequency = 5

Path Display Speed = 8

☐ Choose **OK**.

The Variable Contour dialog is displayed.

You have specified how to drive the bottom of the tool. You must specify how to drive the side of the tool. This is done by choosing a Drive Method from the available types.

Step 12: Define the Drive Method.

☐ Under the Drive Method label, select **Surface Area.**

☐ If necessary, accept the warning message.

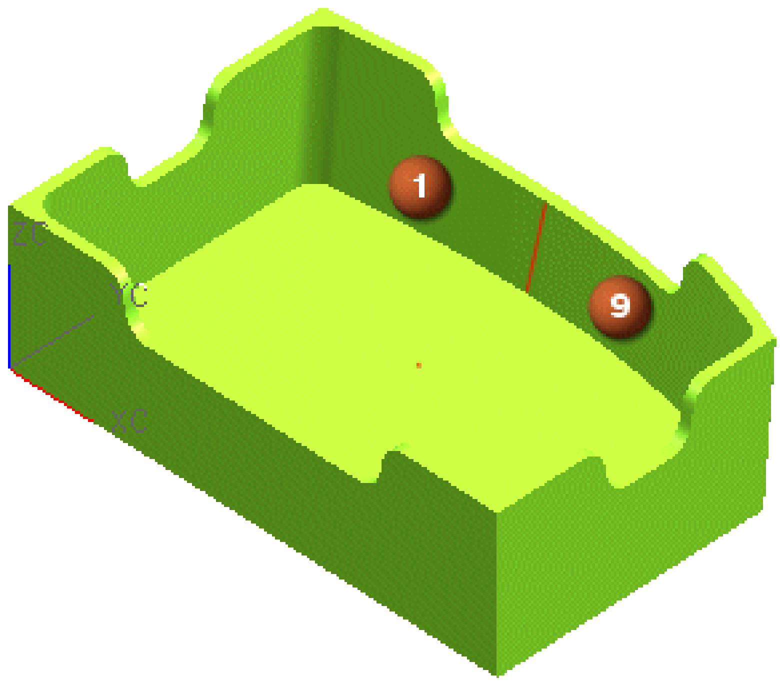
The Surface Drive Method dialog is displayed.

Step 13: Select the Drive Geometry.

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

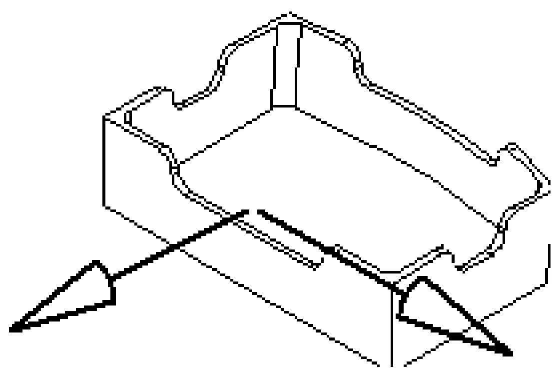
The Drive Geometry dialog is displayed.

☐ Select all the **interior** faces, beginning at (1) and ending at (9), in a counterclockwise direction.



☐ Choose **OK** from the Drive Geometry dialog when the face selection is complete.

If the material side and direction indicator appears as follows:



☐ Choose **Flip Material** from the Surface Drive Method dialog (perform this action only if the indicators appears as above).

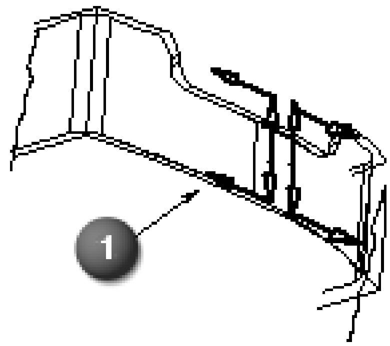
Step 14: Define the drive direction.

You must now define the direction of the cut.

☐ Choose **Cut Direction** from the Surface Drive Method dialog.

Cut direction arrows are displayed. You will select the direction arrow that points in the direction that you will want to cut.

☐ Choose the lower left arrow (1).



Step 15: Set the Number of Passes.

☐ Next to Stepover, set the option to **Number**.

☐ In the Number of Steps field, enter **0**.

Zero indicates that only one pass will be made.

Step 16: Define the Gouge Action.

☐ Next to the When Gouging label, change **None** to **Skip**.

The Skip option will move the tool to the next non-gouging point if gouging is detected.

Step 17: Define the Tool Axis.

☐ Under the Tool Axis label, change **Normal to Part** to **Swarf Drive**.

Remember, Swarf Drive enables you to define a tool axis that follows the swarf rulings of the drive geometry with the side of the tool.

Notice that several vector indicators appear. They are relative to the first drive geometry you selected. The vector you select defines the swarf ruling direction that the tool axis will follow. The vector should point towards the tool holder.

☐ Choose the arrow pointing up.

Step 18: Define the Projection Vector.

The Projection Vector determines the direction that the drive points are projected upon the part geometry.

☐ Under Projection Vector, change **Specify Vector,** to **Tool Axis**.

The Surface Area Drive Method parameters are now complete.

- ☐ Choose **OK**.

  The Variable Contour dialog is displayed.

Step 19: Create the tool path.

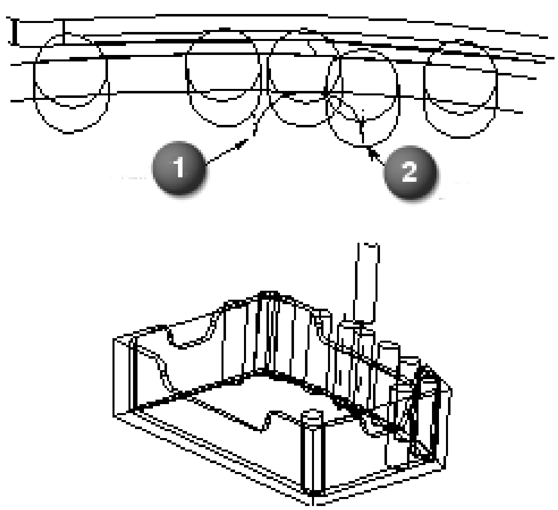
- ☐ Choose the **Generate** button.

  The tool path is generated and the option menu is displayed.

  The tool engages and retracts along the defined radius of the non-cutting move.

  The side of the tool maintains wall contact throughout the cut.

  Notice that the tool appears as to be gouging the part. It is not. The tool is longer than the surface it is cutting which makes it appear to be violating the geometry.



(1) retract

(2) engage

- ☐ Choose **OK** from the option menu.

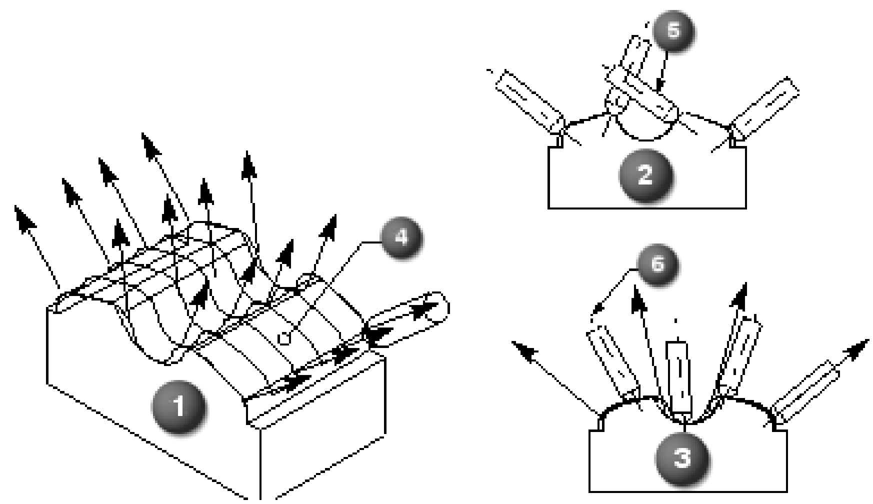
- ☐ **Save** and **Close** the part file.

Interpolated Tool Axis

Interpolate tool axis enables the control of the tool axis at specific points by defining vectors. It allows for control of excessive change of the tool axis as a result of very complex drive or part geometry, without the construction of additional tool axis control geometry (e.g., points, lines, vectors, smoother drive geometry). Interpolate can also be used to adjust the tool axis to avoid overhangs or other obstructions.

You can define as many vectors extending from specified positions on the drive geometry as required to create smooth tool axis movements. The tool axis, at any arbitrary point on the drive geometry, will be interpolated by the user-specified vector. The more vectors specified, the more control you have of the tool axis.

This option is available only when using the Curve/Point or Surface Area drive method.



(1) user-defined controlling vectors

(2) excessive tool axis change

(3) smoother tool axis movement

(4) drive surfaces

(5) tool axis normal to drive surface

(6) interpolated tool axis

Interpolated tool axis dialog options are:

Specify as defines the vectors used to interpolate the tool axis. You can define as many vectors as necessary to control the tool axis.

Vector defines vectors by first specifying a data point on the drive geometry and then specifying a vector.

Angle/PS (or DS) defines vectors by specifying a data point on the drive geometry and then specifying Lead and Tilt angles relative to the part (or drive geometry) surface normal at the tool contact point with the part geometry. Lead and Tilt angles must be within -90 to 90 degree range.

After you choose OK to accept the desired vector or angle, you can continue defining data points and vectors until you choose *Back* in the Point Constructor dialog. Selecting *Back* accepts all of the defined vectors and returns you to the Interpolated Tool Axis dialog.

Data Point allows you to create, delete and modify vectors used to interpolate the tool axis.

Add enables you to create new data points. First specify a data point on the drive geometry and then a vector direction. After specifying the data point, a vector normal to the drive geometry is displayed.

Remove enables you to delete data points. Use the Arrow Buttons to highlight the desired data point or select the desired data point directly from the screen and then choose Remove.

Edit enables you to modify the tool axis at an existing data point. It does not allow you to move data points.

Display displays all currently defined data points for visual reference.

Interpolation method determines which algorithm is used to calculate the tool axis from one drive point to the next.

- *Linear* interpolates the tool axis using a constant rate of change between drive points
- *Cubic Spline* interpolates the tool axis using a variable rate of change between drive points; this method allows a smoother transition between points

Interpolate displays drive tool axis vectors at each drive point (when Specify as Vector is used) or drive points and interpolated lead and tilt angle values (when Specify as Angle/PS or Angle/DS is used).

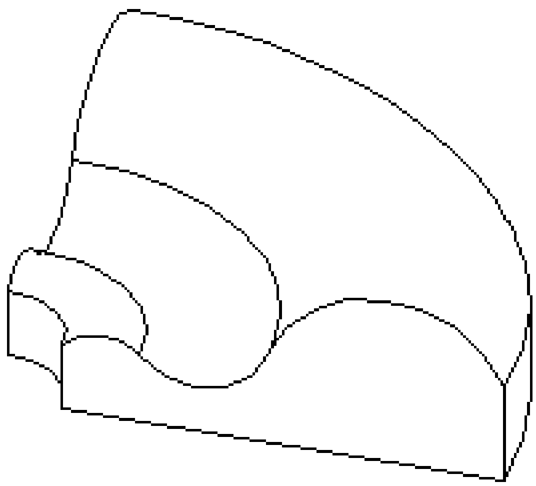
Reselect removes all defined data points.

Activity: Using the Interpolated Tool Axis

In this activity, you will create an operation using an Interpolated Tool Axis. The tool will start at the rear of the part with a tool axis that is normal and will then cut to the front of the part, ending with a tool axis that is aligned with the ZC axis. As the tool moves from the rear to the front, its orientation changes incrementally along the tool path.

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

☐ Open the part file **mam_interpolate_mfg_asmb** and rename it to *****_interpolate_mfg_asmb**.



☐ Enter the Manufacturing application.

☐ Choose the Operation Navigator icon from the toolbar.

Step 2: Create a Variable Contour Operation.

☐ Choose the **Create Operation** icon.

☐ If necessary, change the **Type** to **mill_multi-axis**.

☐ Choose the **Variable Contour** icon.

☐ In the Create Operation dialog, set the following:

Program: **PROGRAM-AXIS-LIMITS**

Use Geometry: **WORKPIECE**

Use Tool: **BALL_MILL-1.0**

Use Method: **MILL_FINISH**

Name: **interpolate**

- ☐ Choose **OK**.

  The Variable Contour dialog is displayed.

- ☐ Under the Machining Parameters label, select **Non-Cutting**.

- ☐ Specify the **Approach** and **Departure** motions to Clearance Plane.

Step 3: Define the Drive Geometry.

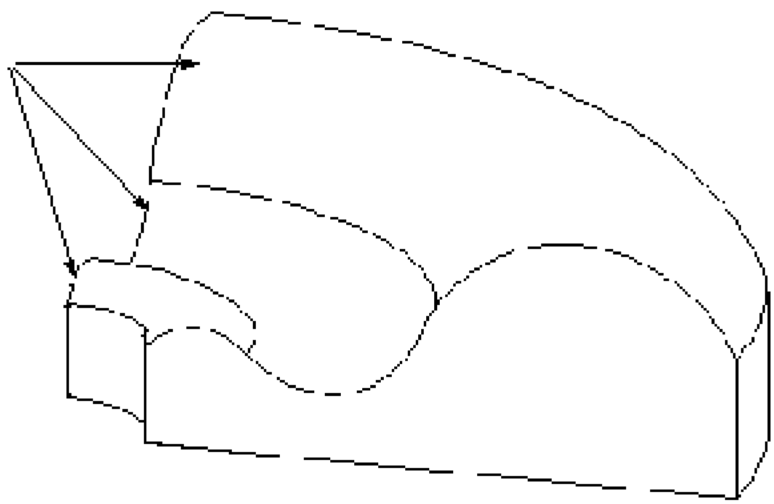
- ☐ Under the Drive Method label, choose **Surface Area**.

  The Surface Drive Method dialog is displayed.

Step 4: Specify a Drive Method.

- ☐ Under the Drive Geometry label, choose **Select**.

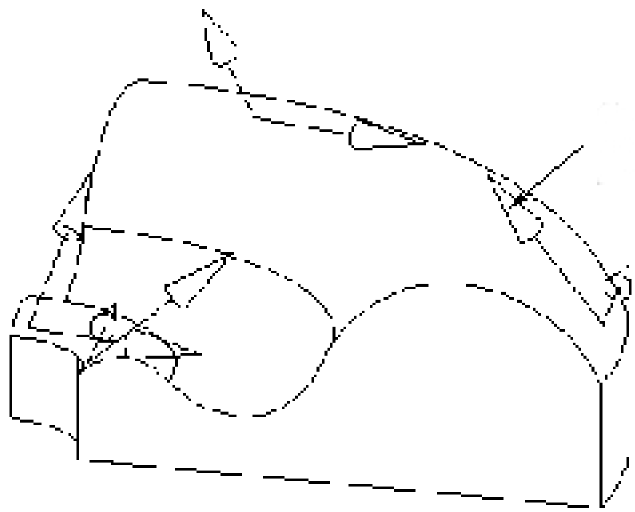
- ☐ Choose the surfaces as shown.



- ☐ Choose **OK**.

- ☐ Choose the **Cut Direction** button**.**

☐ Choose the Cut Direction arrow as shown.



☐ Change the Cut Type to **Zig**.

☐ Change the Cut Step to **Tolerances**.

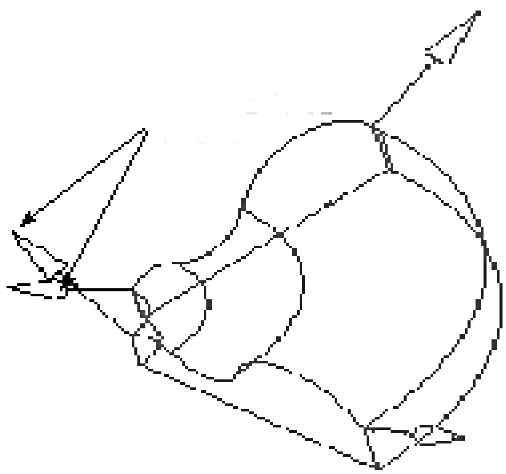
☐ Under the Stepover label, change the Number of Steps to **4**.

☐ Change the Tool Axis to **Interpolate**.

The Interpolated Tool Axis dialog is displayed.

Four default vector arrows show the current tool axis vector direction.

☐ As shown, choose the front arrows (using the cursor or the Selection Arrows, choose one at a time) and under the **Data Point** label, specify **Edit** → **ZC Axis** for each vector direction arrow selected.



Each vector now points along the +ZC axis.

☐ Choose **OK**.

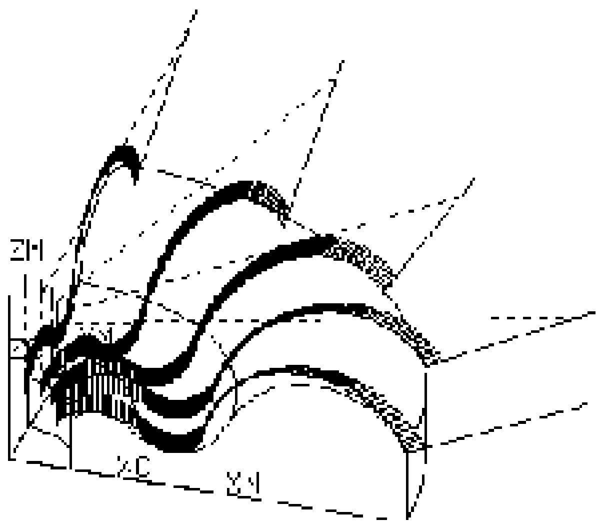
The Surface Drive Method dialog is displayed.

☐ Choose **OK** to return to the Variable Contour dialog.

☐ Under the Tool Path label, choose the **Edit Display** icon and change the Tool Display to **Axis**.

☐ Choose **OK** to return to the Variable Contour dialog.

☐ Choose the **Generate** icon**.**



Notice that the tool starts cutting along the surface normal vector at the rear of the part, gradually changing its axis to the vectors specified at the front of the part, which is parallel to the +ZC axis.

☐ Verify the Interpolate Tool Axis positions.

☐ **List** the tool path and verify the start and finish tool axis.

By listing the tool path, you can see the tool axis position at the first GOTO, is not parallel to the ZC axis. As the tool moves, the tool axis position interpolates and becomes parallel to the ZC axis at the last GOTO.

☐ **Close** the Information window.

☐ Choose **OK**.

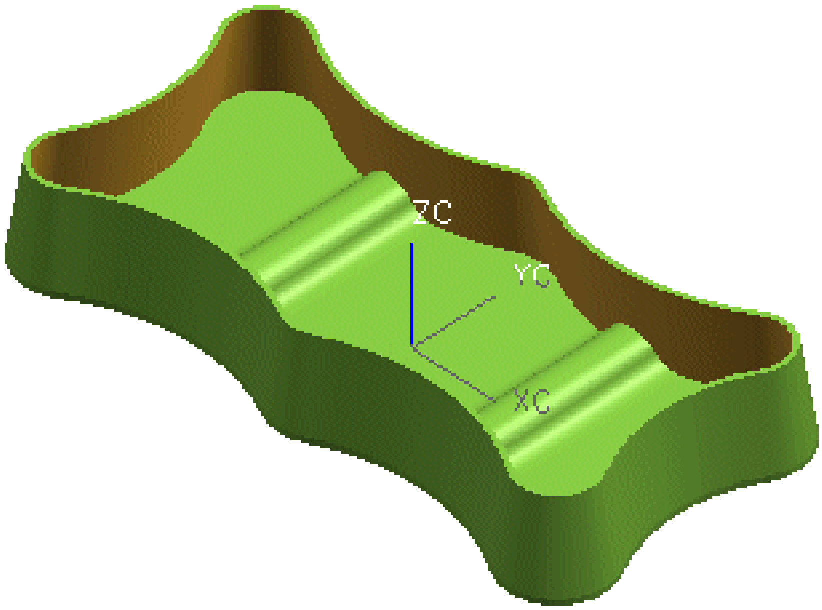
☐ **Save** the part file.

Activity: Selection and use of Various Tool Axes

In this activity, you will select several tool axes and view the effect of each in machining the part. You will see that one tool axis type is sometimes more suitable for machining the geometry than another.

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

☐ Open the part file **mam_canted_wall_mfg_asmb** and rename it to *****_canted_wall_mfg_asmb**.



☐ Enter the Manufacturing application.

☐ Choose the Create Operation icon.

☐ Choose the **Variable Contour** icon.

☐ In the Create Operation dialog, set the following:

- Program: **FINISH_PROGRAM**
- Use Geometry: **MCS**
- Use Tool: **EM-.75-12**
- Use Method: **MILL_FINISH**
- Name: **proficiency**

☐ Choose **OK**.

The Variable Contour dialog is displayed.

☐ Under the Drive Method label, Choose **Surface Area**.

☐ Create tool paths to machine the interior walls using several of the available tool axes.

You can verify your selections by using the operation, **SOLUTION**, to compare the results.

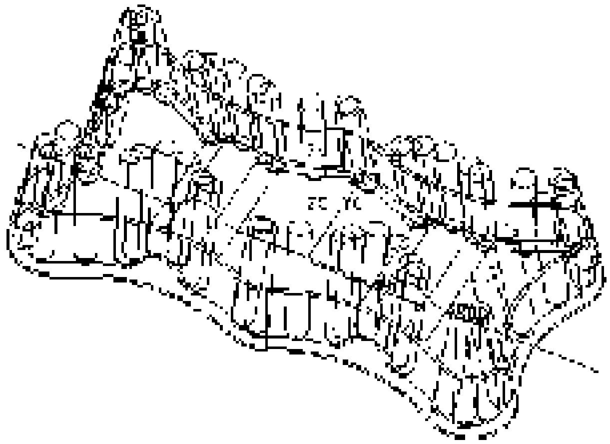
Note your success or difficulties in using these tool axes. You can discuss these notes with your instructor.

Tool Axis Review Questions

Try to answer these questions on your own. Your Instructor will provide the correct answers and any additional information that you may require.

1. Which Tool Axis is most suitable for machining the part?
____

Is it similar to the following? ______________________________________

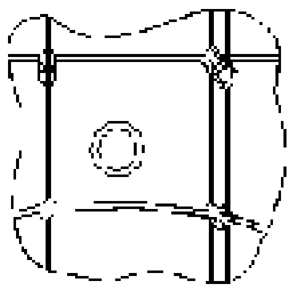


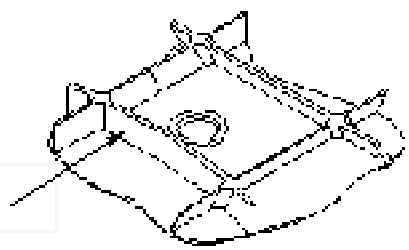
2. What is another name for a Relative Tool Axis having a Lead and Tilt Angle of 0? ____

3. How is the Swarf tool axis calculated?________________________________
____
____

4. How is the Interpolate tool axis calculated?___________________________
____

5. For the following geometry, what tool axis would you use to machine the interior walls as shown?____





6. What Drive Method would you use? _________________________________

7. What Projection Vector would you use? ____________________________

Variable Contour Best Practices

Surface Area vs Boundary Drive Methods

Either the Surface Area or Boundary Drive Method can be used for creating tool paths. There are some considerations to keep in mind in deciding which method is more conducive to use in multi-axis finish machining.

- The Boundary drive method provides several options that are not available in the Surface Area drive method.

- **More Drive Parameters**, options allow greater control of island and wall machining as well as leaving stock for a finish pass. Additional corner control and application of parameters is also controlled from this dialog.

Notes:

Surface Area Drive Method

The **Surface %** method, under **Cut Area**, allows a percentage of the surface that is selected to be machined. Negative values as well as values over 100% are allowed.

Notes:

A comparison of Variable Contour vs. Sequential Milling

Variable Contour and Sequential Mill operations allow you to specify Drive, Part and Check surfaces. Generally, the Drive geometry guides the side of the tool and the Part geometry guides the bottom of the tool. The Check geometry stops tool movement. Specifying Part and Check geometry is very similar in Variable Contour and Sequential Mill operations.

Part Geometry

Variable Contour does not always require that you specify Part geometry. When you do not, Drive geometry is used as Part geometry.

Sequential Mill requires selection of Part geometry. The default selection is the previous Part geometry.

Drive Geometry

Drive geometry is used to create drive points that are projected to the Part geometry. You may use geometry other than that contained within the model. This "external" drive geometry can be points, curves, a boundary, etc. that you select after you choose an appropriate Drive Method.

Drive geometry in Sequential Mill is used to control the side of the tool without developing and projecting drive points. Typically, you select a part wall that you want the side of the tool to contact as it follows the Part surface.

Check Geometry

Variable Contour does not require Check geometry. Check geometry is typically used to prevent collision and gouging.

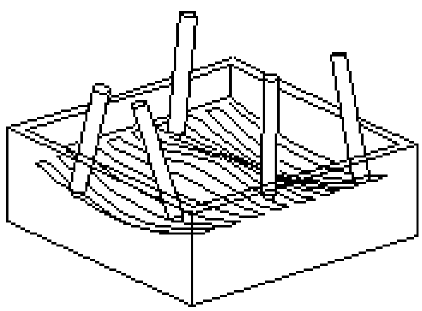
Sequential Mill requires selection of Check geometry. The Check geometry is used for tool positioning at the beginning of the next sub-operation and for preventing collision and gouging.

General Considerations

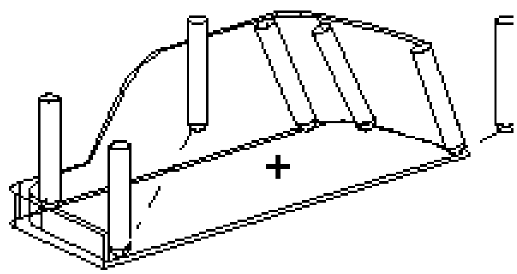
The overriding consideration in choosing between Variable Contour and Sequential Mill is: "Which method creates the best tool path and is easiest to use."

The answer depends upon whether the part model has features that only Variable Contour or Sequential Mill can resolve. If both processors are capable, you should consider the following relative strength of each processor:

Variable Contour	Sequential Mill
preferred method for area milling	preferred method for linear milling
primary cutting with bottom of tool	primary cutting with side of tool
numerous drive methods for tool path containment	single drive method
numerous cut patterns for specific applications	no cut patterns other than looping or nested loops
sheet body and surface region geometry allowed	temporary plane geometry allowed
constant tool axis	can change tool axis during operation
edits apply to entire tool path	edits apply to part of tool path
best at convex wall cuts	best at overcut and undercut type walls
easy to create operation	numerous steps in operation creation
easy to create multiple depth paths	N/A



Variable Contour



Sequential Mill

Tool Axis Usage

The following table compares tool axes usage in both Variable Contour and Sequential Mill operations:

Tool Axis Usage	
Variable Contour	**Sequential Mill**
3 Axis	
Normal to Part (default)	ZM Axis (default)
Relative to Vector	Vector
4 Axis	
Away from line (4) / Toward line (4)	-
4–axis normal to part / 4–axis normal to drive	-
4–axis relative to part	-
4–axis relative to drive	at angle to Drive Surface/at angle to Part Surface
dual 4–axis on part / dual 4–axis on drive	-
-	tangent to Part Surface
-	tangent to Drive Surface
-	project Drive Surface Normal
-	project Part Surface Normal
5 Axis	
Away from point	thru fixed point
toward point	thru fixed point
normal to part	normal to Part Surface
normal to drive	normal to Drive Surface
swarf drive	parallel to PS /Parallel to DS
relative to drive	at angle to DS / at angle to PS
interpolate	-
same as drive path	-
user function	-
-	tangent to PS
-	tangent to DS
-	fan

Summary

Variable Contour operations provide an efficient and robust capability to machine complex geometry for multiple axes machining processes (4 plus axis). Numerous types of tool axis control and drive methods, give the NC/CNC programmer the ability to machine the simplest to the most complex of parts. The following features are common to variable contour operations.

- Complete tool axis control that allows for minimal tool and table rotations.
- Numerous drive methods to achieve the simplest to the more complex of surface machining techniques.

Lesson

6 *Variable Contour - Advanced*

Purpose

This lesson will introduce advanced concepts in conjunction with Variable Contour operations.

Objective

At the conclusion of this lesson, you will be able to:

- create associative drive surfaces used to control the tool axis
- use Associative Datum planes to create surfaces and geometric objects used for creation of start points and initial tool axis

Advanced Variable Contour machining

The activity which follows will take you through some of the steps that allow greater control of the tool axis and avoids chaining tolerance errors that occur when drive surfaces edges do not match precisely.

Surfaces will be extracted from the solid body by use of the WAVE Geometry Linker with timestamp applied. The use of the timestamp will prevent new geometry, which is created after the timestamp, from being used by the WAVE Linked surfaces.

Associative Datum planes are created for use in creating the initial start point and tool axis as well as for the creation of various geometric elements that will be required for tool control.

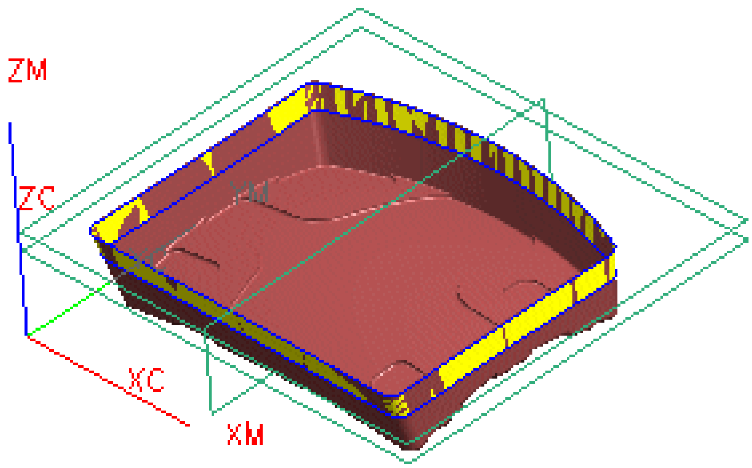
All surfaces which have not been intersected by parallel datum planes will be selected as part surfaces. The tool axis will be set to swarf drive and the projection vector will be the tool axis.

Activity: Examining the Part and Part Objects

The part in this activity can represent any type of five-axis work that could be performed on an aircraft structural assembly, the inside of a mold or some other type of part that requires five-axis machining. You will be required to semi-finish and finish the walls with two different diameter cutters. Assume that the part has already been roughed.

Step 1: Open an existing part file.

- ☐ Open the part file **mam_vc_nc_assy**.



There are two existing sample operations that you will examine and then create like operations. First you will examine the various parts which comprise the assembly.

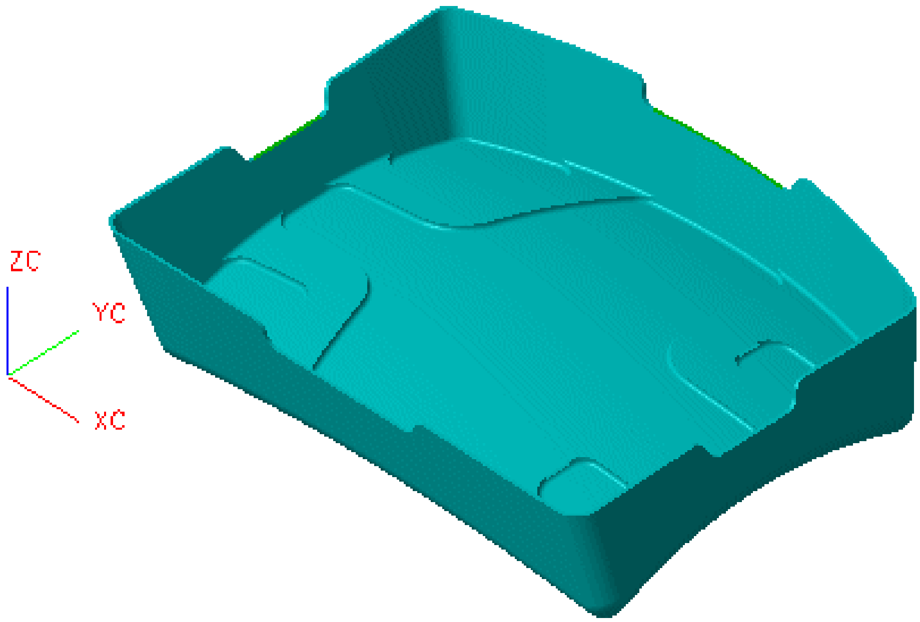
Step 2: Examine the assembly.

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

- ☐ Choose the Assembly Navigator tab from the toolbar.

  The Assembly Navigator and the part model are displayed.

☐ Make the **mam_vc_solidbody** component the **Displayed Part**.

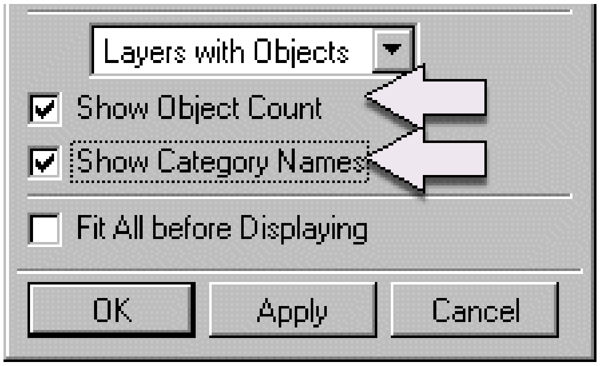


Note the cut out areas on top of the walls.

Step 3: Examine layers in the assembly.

☐ Choose **Format→Layer Settings** from the menu bar.

☐ Check the **Show Object Count** and **Show Category Names** boxes.



Note the change in the Layer/Status/Count/Category area of the dialog.

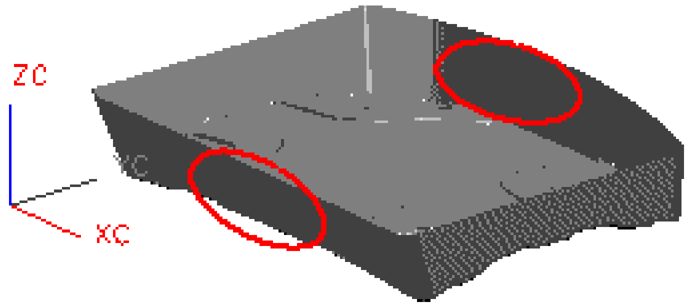
You will now examine the layers.

☐ Make all layers **Invisible**.

☐ Make **layer 15** Selectable.

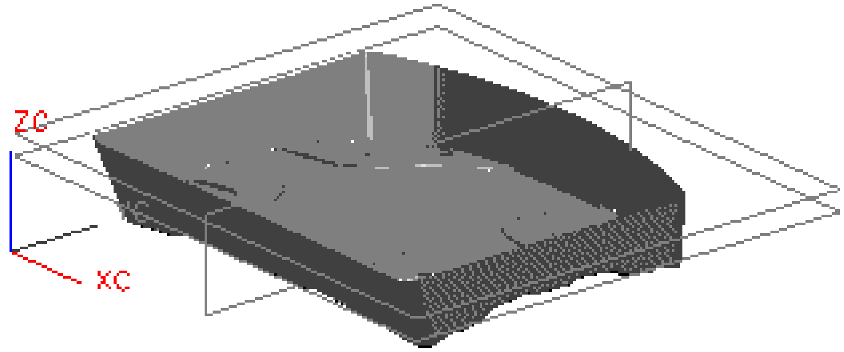
☐ Choose **OK**.

Now examine the WAVE Linked surfaces.



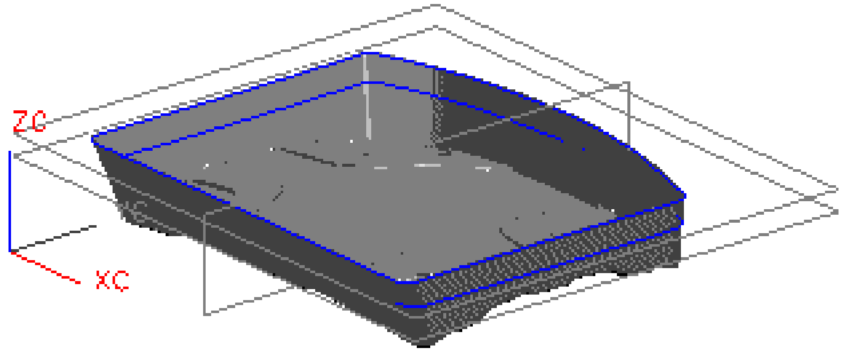
Note that the cut outs were not passed to the WAVE Linked geometry due to the use and placement of the timestamp.

☐ Make **layer 61** Selectable.



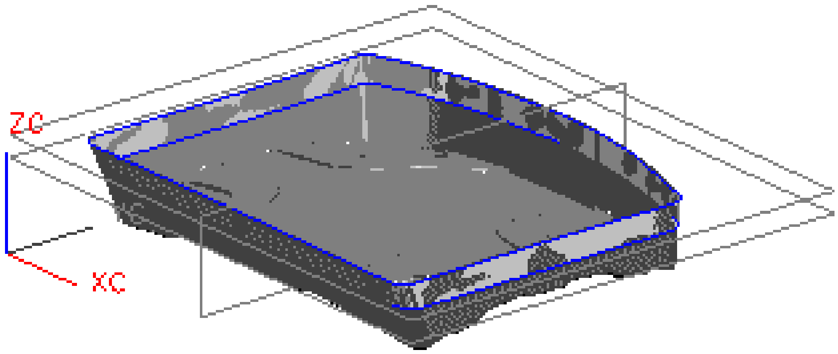
The view now shows the part with the Associative Datum planes that are used to create the necessary intersection curves.

☐ Make **layer 41** Selectable.



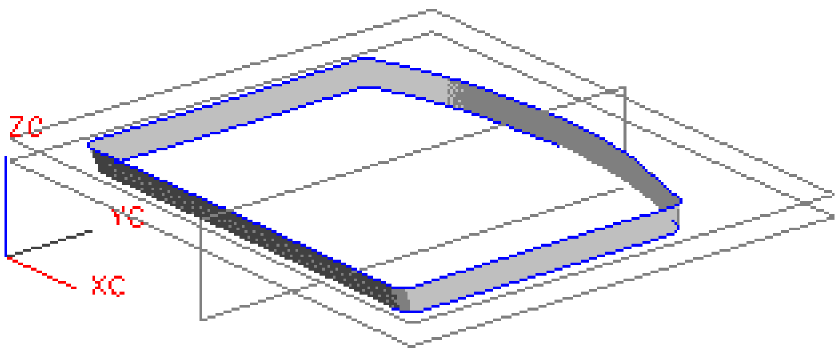
The view now shows the part with the Associative Curves that are used to create the necessary ruled surface.

☐ Make **layer 81** Selectable.



The view now shows the part with the Associative Ruled surface that is used to create the Drive surfaces.

☐ Make **layer 15** Invisible.



The single vertical plane was used to split the WAVE Linked surface prior to the creation of the Intersection Curves. This can be at any angle to establish the initial tool axis.

☐ Dismiss the Layer Settings dialog by choosing **Cancel**.

Step 4: Enter the Manufacturing application and review the existing operations.

You will review the operations by examining their settings.

☐ Enter the **Manufacturing** application.

☐ Choose the Operation Navigator tab from the menu bar.

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

☐ Expand the MCS and WORKPIECE Parent group objects.

☐ Double-click on the operation **VC_RGH_POC_1.50_WO_CS**.

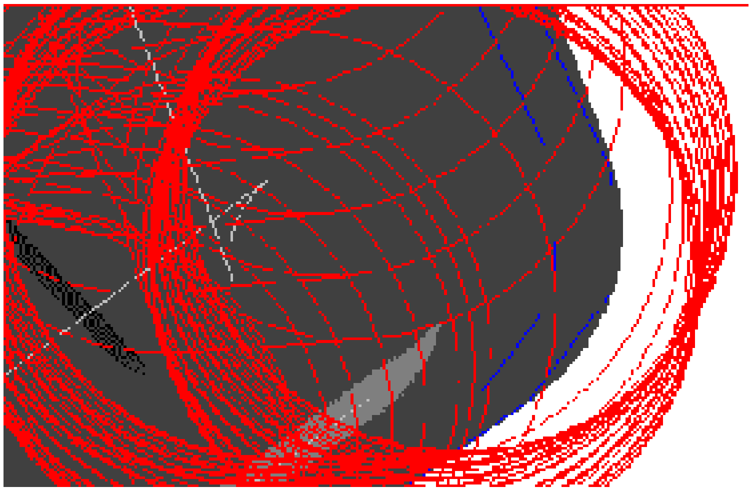
The Variable_Contour dialog is displayed.

☐ Display the Part geometry and then the Check geometry.

Notice that the Select button appears when you choose the Check geometry icon. There is no Check geometry available for display.

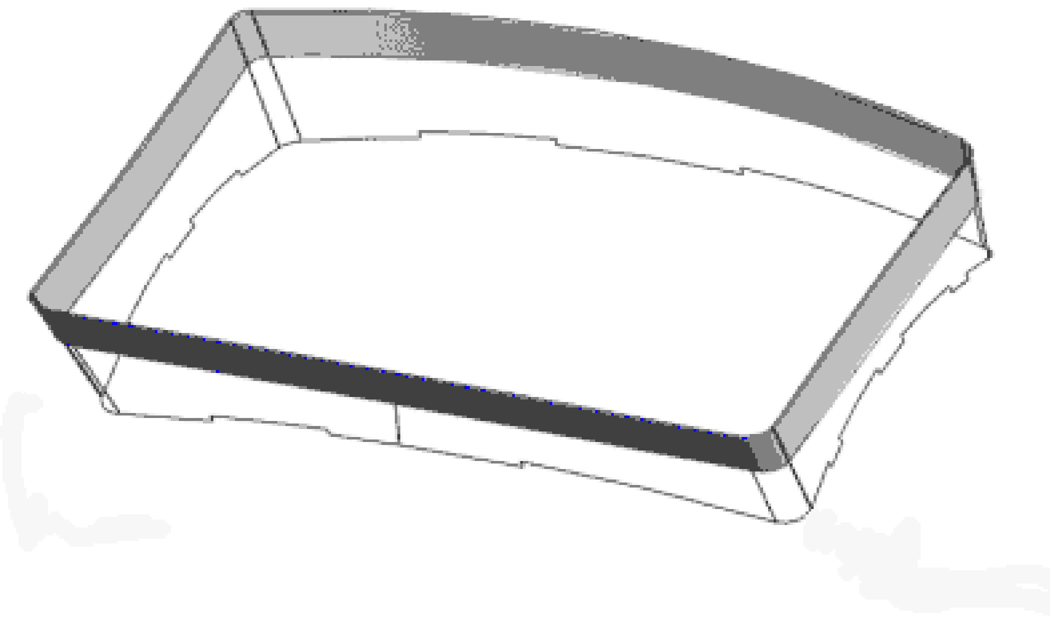
☐ Replay the operation and zoom in at the corner of the part.

Notice the cutter violates the drive surfaces.



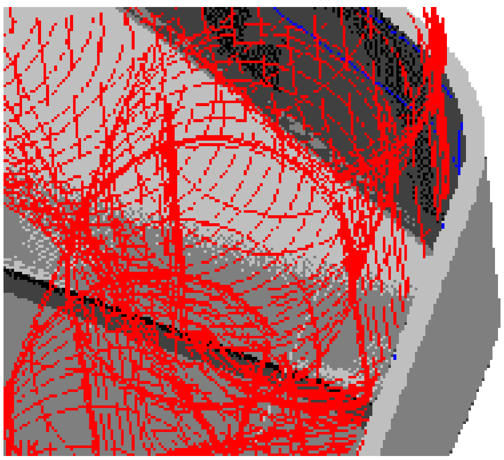
☐ Double-click on the operation **VC_RGH_POC_1.50_W_CS**.

☐ Display the Part geometry and then the Check geometry.



Notice that the side walls of the part have been selected as Check surfaces.

☐ Replay the operation and zoom in at the corner of the part.



Notice the cutter does not violate the walls.

☐ Replay the operation **VC_FIN_POC_1.00**.

☐ Examine the operation parameters and the surfaces used.

Notice that this operation does not need Check surfaces.

Step 5: Create new operations.

☐ Create operations to semi-finish and finish the pocket, using the previous operations as a guide only. Do not copy them.

☐ Close the part file.

Summary

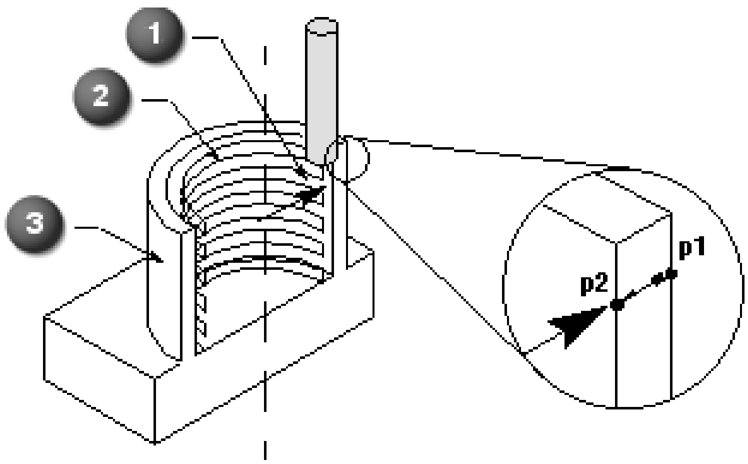
Variable Contour operations provide an efficient and robust capability to machine complex geometry for 4 and 5-axis machining centers. This lesson familiarizes you with some the requirements that are necessary to make the programming task simpler.

Appendix

A Projection Vectors

The Projection Vector indicates the side of the part surface to be cut. It is also used to project drive points from the drive to the part surface.

The following illustration shows a Projection Vector (defined as Away From Line, i.e. the center line) indicating the side of the part surface to be cut. It also shows a drive point projected, along the projection vector, from the drive surface (P1) to the part surface (P2).



(1) projection vector

(2) part surface

(3) drive surface

Note that, in this example, the drive point is projected in the *opposite* direction of the Projection Vector arrowhead. The drive point is always projected toward the part surface along the projection vector but without regard to the Projection Vector arrowhead.

A Projection Vector is required for all Variable Contour Drive Methods.

The following options allow you to define the Projection Vector:

- Specify Vector — — —fixed projection vectors
- Tool Axis — — — variable projection vector
- Away from Point — — — variable projection vector
- Toward Point — — — variable projection vector
- Away from line — — — variable projection vector
- Toward line — — — variable projection vector
- Normal to Drive — — — surface area drive method only
- Swarf Ruling — — — surface area drive method only
- User Function

Specify Vector - Fixed Projection Vectors

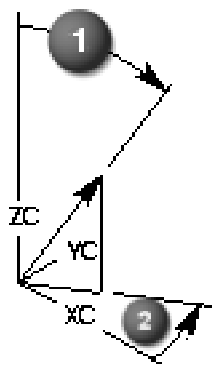
I, J, K define the vector by keying in values relative to the origin of the Work Coordinate System.

Line End Points by defining two points, selecting an existing line, or defining a point and a vector.

2 Points by using the point Constructor to specify two points. The first point defines the tail of the vector; the second point defines the arrowhead of the vector.

Tangent to Curve defines a vector tangent to a selected curve. Specify a point on the curve, select an existing curve, and select one of two displayed tangent vectors.

Spherical Coordinates defines a fixed vector by keying in two angular values, designated as *Phi* and *Theta*. Phi is the angle measured from +ZC and rotated in the ZC-XC plane from ZC to XC. Theta is the rotation angle about the ZC axis from XC to YC.



(1) Phi

(2) Theta

Variable Contour Projection Vectors

Tool Axis define a projection vector relative to the existing tool axis. When using tool axis, the vector always points in the *opposite* direction of the tool axis vector.

Away From Point creates a projection vector extending away from a specified focal point toward the part surface. Useful in machining the inside spherical (or sphere like) surfaces where the focal point is the center of the sphere.

Towards Point creates a projection vector extending from the part surface to a specified focal point. Useful in machining the outside spherical (or sphere like) surfaces where the focal point is the center of the sphere.

Away From Line creates a projection vector extending from a specified line to the part surface.

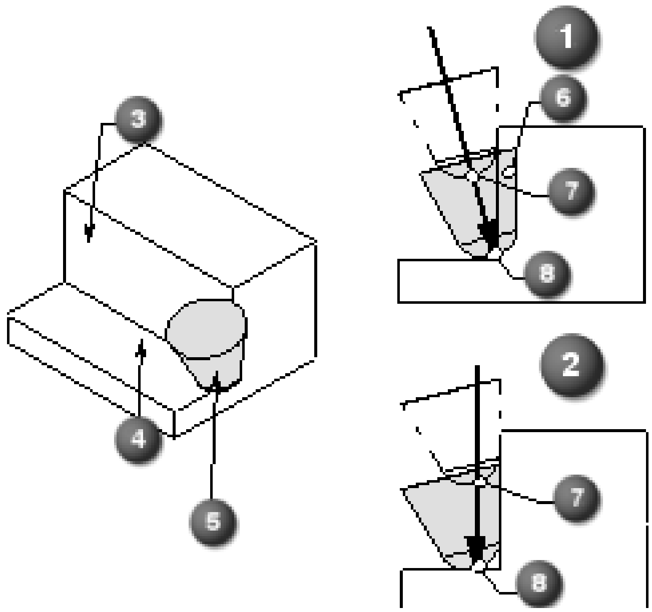
Towards Line creates a projection vector extending from the part surface to a specified line.

Surface Area Drive Method Projection Vectors

Normal to Drive define projection vectors relative to the drive surface normals.

Swarf Ruling allows you to define the projection vector parallel to the swarf rulings of the drive surfaces when you use a swarf drive tool axis. It should be used only when the drive surfaces are equivalent to ruled surfaces, since the drive surface rulings define the swarf projection vector.

The Swarf Ruling projection vector can prevent gouging the drive surface when using a tapered tool as illustrated below:



(1) Tool Axis Projection Vector

(2) Swarf Ruling Projection Vector

(3) Ruled Drive Surface

(4) Part Surface

(5) Tapered Tool

(6) gouge

(7) drive point

(8) tool position

The above figure compares the Swarf Ruling projection vector to the Tool Axis projection vector (the Tool Axis projection vector is the reverse of the Tool Axis Vector). Drive points are projected along the specified vector to determine the tool position. When using the Tool Axis projection vector, drive points are projected along the tool axis (at an angle to the drive surface), causing the tool to gouge the drive surface. When using the Swarf Ruling projection vector, drive points are projected along the drive surface swarf rulings causing the tool to position tangent to the drive surface.

The following is a summary table showing the types of projection methods available for each tool axis. The **x** indicates that the Projection Method is not available.

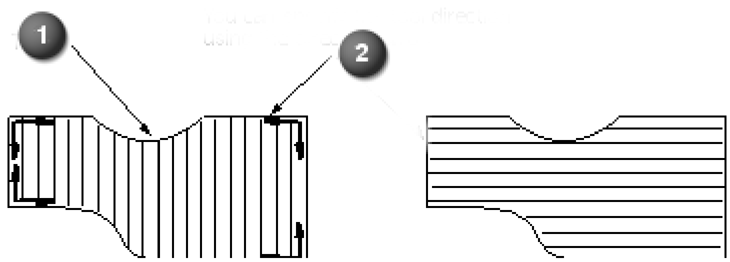
Tool Axis	Projection Methods					
	Fixed Vector	Tool Axis	Toward / Away Point	Toward / Away Line	Norm Drive	Swarf Rule
Away From Point						X
Toward Point						X
Away From Line						X
Toward Line						X
Relative To Vector						X
Normal to Part						X
Relative to Part						X
4–axis Normal to Part						X
4–axis Relative to Part						X
Dual 4–Axis on Part						X
Interpolate						X
Normal to Drive						X
Swarf Drive						
Relative to Drive						
4–axis Norm to Drive						
4–axis Rel to Drive						
Dual 4–Axis on Drive						
Same as Drive Path					X	X

Appendix

B Zig-Zag Surface Machining

Zig-Zag Surface machining is designed for machining a single trimmed surface. Zig-Zag Surface also provides the capability to offset the tool from holes trimmed in the surface (by the radius of the tool plus any specified stock).

You can specify a tool path direction or accept a system generated tool path direction. Either *Zig* or *Zig-Zag* cut types are available.



(1) trim

(2) specify cut direction by selecting direction arrows

Zig-Zag Surface tool paths are generated in parallel passes. The drive points are generated on the surface to be machined. You can control the number of input points by a chordal deviation (adjusting the step tolerance) in the direction of cut. This is the allowable deviation from the surface. Scallop height controls the distance between parallel passes according to the maximum height of material (scallop) you specify to be left between passes. This is affected by the cutter definition and the curvature of the surface.

Zig-Zag Surface also provides gouge check so that the system can check for violation of the surface.

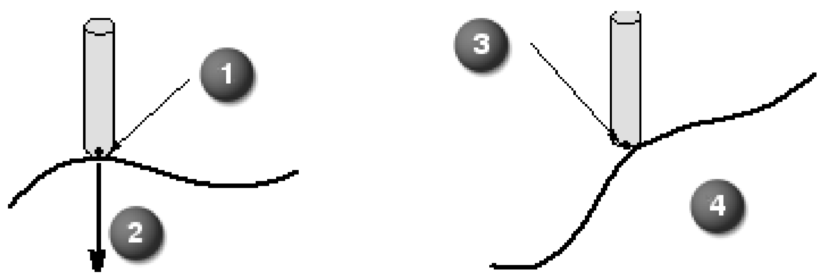
Appendix

C Advanced Surface Contouring

Projection

Mathematics of Projection:

- Place tool end at drive point
- Project tool along projection vector
- Tool stops when making contact with part
- If necessary, adjust the tool axis and repeat the above steps until the tool axis is satisfied
- Add more intermediate drive points to satisfy the Intol/Outol with the part



(1) drive point

(2) projection vector

(3) tool position

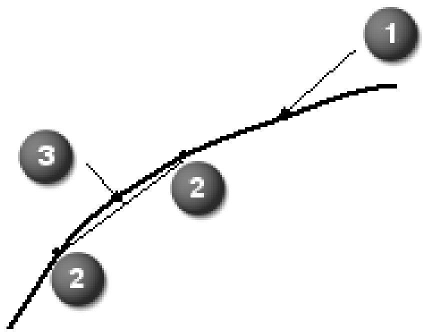
(4) part

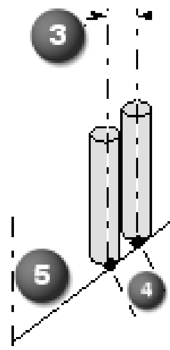
Projection and Steep Surface:

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

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

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





(1) drive path

(2) drive point

(3) Δd

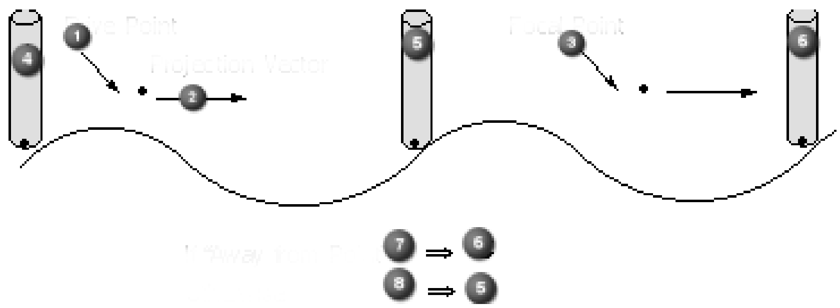
(4) Δx

(5) Θ

Projection and Material Side:

- Surface contouring does not have explicit definition of material side for part geometry, only the drive surface has explicit material side

- Material side of the part is determined implicitly by the projection vector



(1) drive point

(2) projection vector

(3) focal point

(4) A

(5) B

(6) C

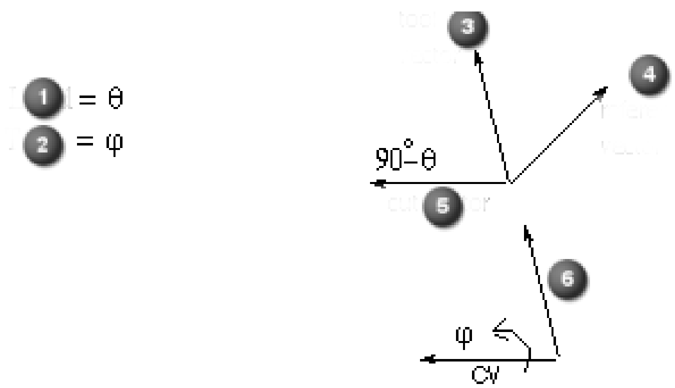
(7) away from point

(8) all other cases

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

Tool Axis

Definition of Lead/Tilt angles:



(1) lead

(2) tilt

(3) tool axis vector

(4) reference vector

(5) cut vector

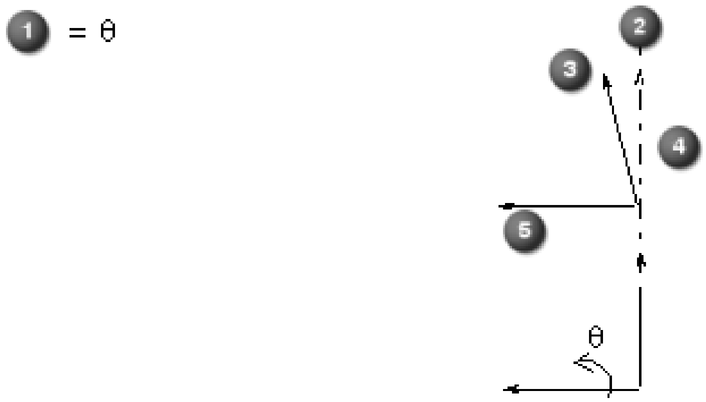
(6) tool axis

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

- Then rotate around the cut vector Φ degrees (counter clockwise)

- Reference vector is the surface normal relative to the part/drive or a vector which is relative to a vector

Definition of 4-axis rotation angle:



(1) rotation angle

(2) perpendicular plane

(3) tool axis

(4) projected tool axis

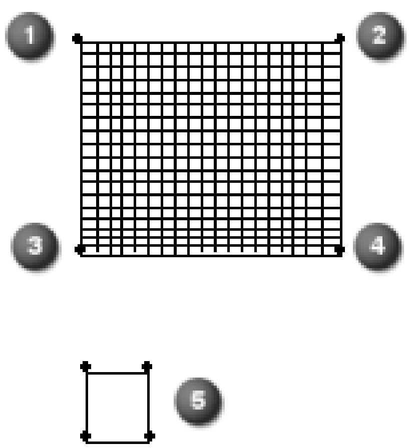
(5) 4–axis vector

- Compute tool axis vector without 4–axis constraint first
- Project this tool axis vector onto the perpendicular plane of the 4–axis vector
- Rotate the projected tool axis vector along 4–axis vector Θ (counterclockwise)

The unconstrained tool axis vector could be:

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

Interpolated tool axis algorithm:



(1) data point 1; (2) data point 2

(3) data point 3; (4) data point 4

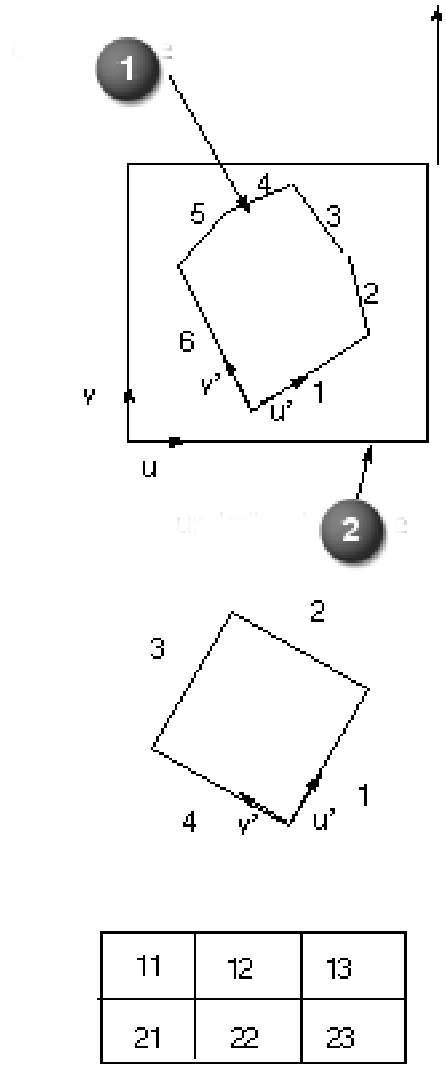
(5) grid cell

- divide the whole parameter (u,v) space for the drive surfaces by a 19x19 grid
- compute the tool axis at each grid pt using the data pts weighted by the inverse of the distance square
- inside each grid cell, calculate the tool axis vector as the linear/spline interpolation of the tool axis vector at the four corners.

Drive Surface

Remap of drive surface:

Remap algorithm:



(1) trimmed face; (2) underlined surface

- merge the exterior edges of the trimmed face to 4 sides
- re-proportion the parameters of the exterior edges according to arc length
- use the arc length proportional edge parameters to construct the new (u',v') space for the trimmed face (Coon's mapping).
- align the multiple drive surfaces into a rectangular grid pattern

Limitations of remap

- fails on 3–sided faces
- fails on faces that do not have rectangular shapes
- may fail on faces with too many edges
- multiple drive surfaces must be in grid formation

Swarf developable surface:

- Developable surfaces are special kinds of ruled surfaces when the surface normal vectors on any given rule line are the same (ruled surface without twisting)

- Only developable surfaces can be milled by swarfing without undercut or overcut

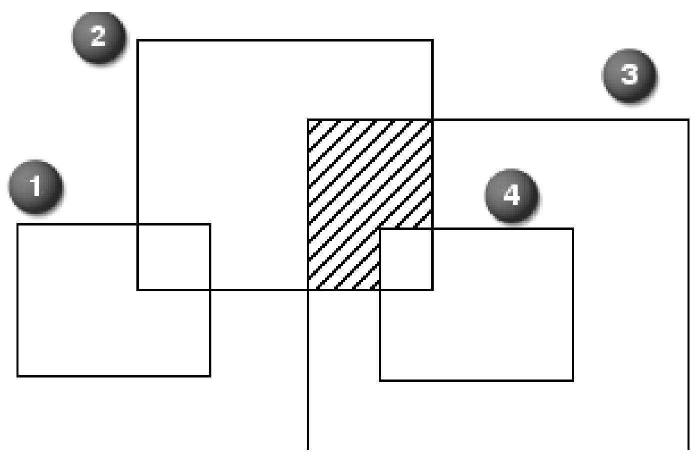
Planar Milling

- Blank - the region to be included

- Part - the region that can not be violated

- Check - the additional region that can not be violated

- Trim - as a final step, the region to be trimmed away



(1) check inside

(2) blank inside

(3) trim outside

(4) part inside

Boolean Logic

Boundary Drive

- Drive boundary - similar to "blank" if no part containment, otherwise it is like "part"
- Part containment - similar to "blank"

Area Milling Drive

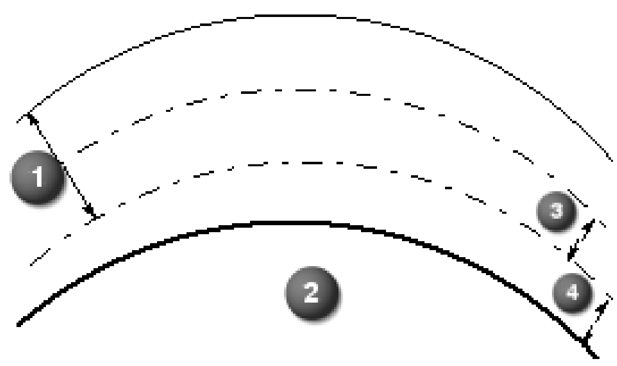
- Cut area - similar to "blank"
- Trim - behaves slightly different from planar milling

Stock

Part offset and part stock

	What	**Where**
Part Offset	Offset of part as the permanent definition of the final shape of the product	Geometry Group
Part Stock	Leftover material on part by a given operation	Operation

- Part stock is defined on "top" of part offset



(1) part stock of roughing

(2) part

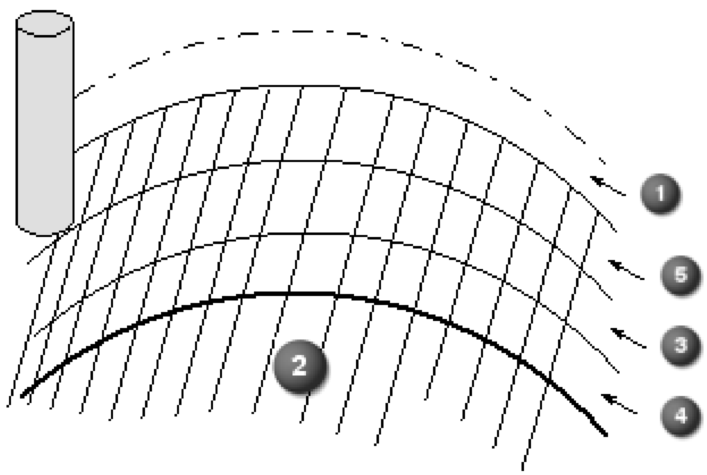
(3) part stock of semi-finish

(4) part offset

Safe clearance and part stock offset

	What	**Where**
Part Stock Offset	Difference between the part stock from the previous operation and the part stock of the current operation	Operation
Safe Clearance	The additional safety zone for collision checking	Operation

- Safe clearance is defined on "top" of part stock offset



(1) safe clearance

(2) part

(3) part stock

(4) part offset

(5) part stock offset

- Part stock offset is used in multiple pass, engage/retract and collision checking

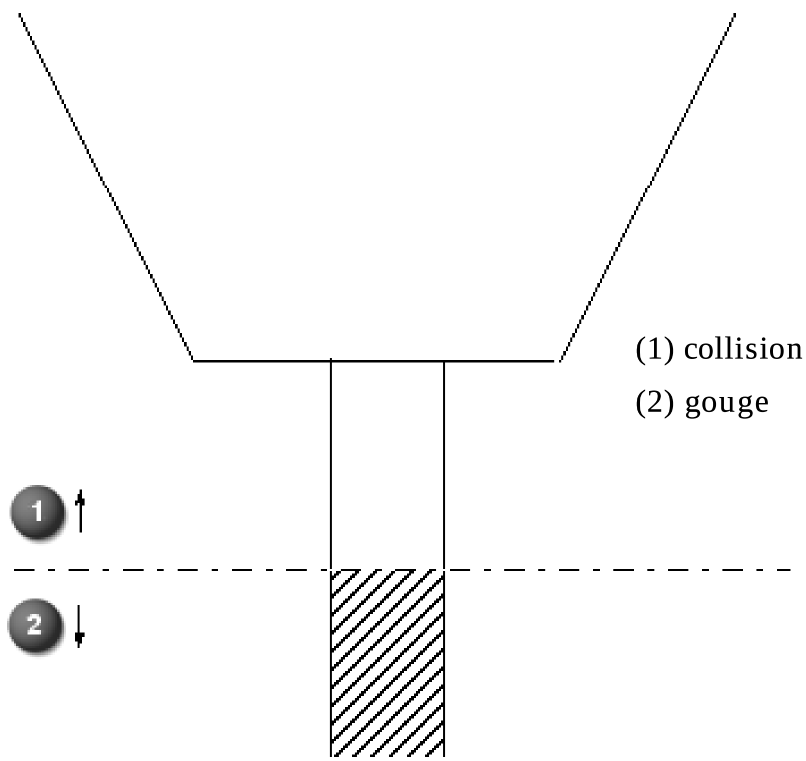
- Safe clearance is used in engage/retract and collision checking

Gouge / Collision

Definitions:

	Rapid moves	Feed moves
Cutting part of tool assembly	Collision	Gouge
Non-cutting part of tool assembly	Collision	Collision

- Usually gouge check against part offset + part stock
- Usually collision check against part offset + part stock + part stock offset + safe clearance



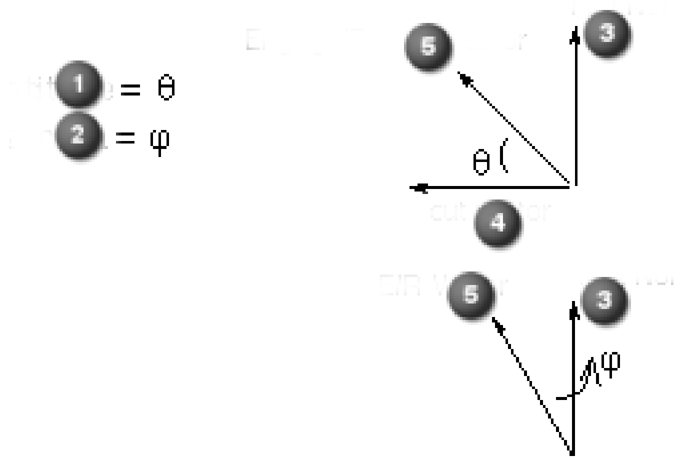
Usage:

	Collision check	Gouge check
Tool Path Generation	No	Yes on Part
Drive Path Generation	No	Optional on Drive
Engage/Retract	No	Optional on Part
Transfer Moves	Optional on Part	Optional on Part

Cut Region Computation **(Cut Area)**	Optional (holder) on Part/Check	Yes on Part
Check Geometry	No	Optional on Check
Gouge Check **(Operation Navigator)**	No	(No Part Stock)

Noncut Moves

Azimuth / Latitude:



(1) latitude

(2) azimuth

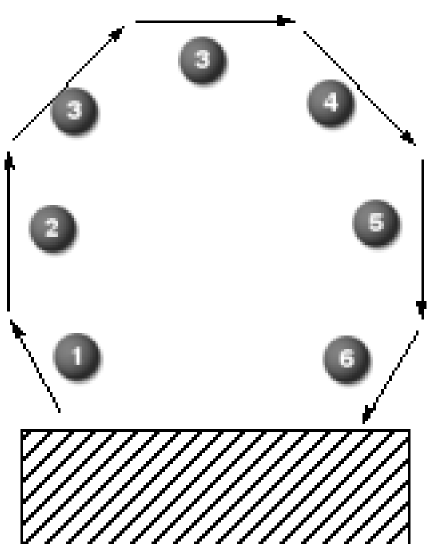
(3) part normal

(4) cut vector

(5) engage/retract vector

- Begin with cut vector, rotate it toward the part normal Θ degrees
- Then rotate around the part normal Φ degrees (counter clockwise)

  End / Intermediate traverse:



(1) retract

(2) departure

(3) int traverse

(4) end traverse

(5) approach

(6) engage

- There is only one End Traverse in the sequence, but there may be zero or multiple Int Traverse
- The Start and End positions of the End Traverse move are determined by other moves in the sequence

Index

V

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.

This page left blank intentionally.

NX Multi-Axis Machining Course Agenda

Day 1 Morning

- Overview of Variable Contour
- Lesson 1. Introduction to four and five axis machining
- *Workbook Introduction: The Manufacturing Process*
- *Workbook Section1: Drilling the Top Flange*
- Lesson 2. Sequential Mill Basics

Day 1 Afternoon

- Lesson 3. Sequential Mill Intermediate
- Lesson 4. Sequential Mill Advanced
- *Workbook Section2: Sequential Mill – Cutting the Manifold Flange*

Day 2 Morning

- Lesson 5. Variable Contour - Basics
- *Workbook Section 3: Variable Contour – Cutting the Manifold Flange*

Day 2 Afternoon

- Lesson 6. Variable Contour - Advanced
- Time Permitting: Appendices – Projection Vectors; Zig-Zag Surface Machining; Advanced Surface Contouring

This page left blank intentionally.

Class Layers and Categories

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

Model Geometry

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

Drafting Objects

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

Engineering Disciplines

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

This page left blank intentionally.

Hot Key Chart

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

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

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

UGS

Evaluation – Delivery
NX 3 Multi-Axis Machining, Course #MT11050

Dates _____________ thru______________

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

Instructor: ______________________________ ☑

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

Instructor: ______________________________ ☒

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

Comments on overall impression of instructor(s):

Overall impression of instructor(s).......... **Poor** ☐ ☐ ☐ ☐ ☐ ☐ **Excellent**

Suggestions for improvement of course delivery:____

____

What you liked best about the course delivery:____

____

Class Logistics:

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

UGS

Evaluation - Courseware
NX 3 Multi-Axis Machining, Course #MT11050

:

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

Material:

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

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

Comments on Course and Material: ________________________________

____

____

____

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

____

Student:

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

Name (optional): ____________________ **Location/room** ____________________

☐ **Please "check" this box if you would like your comments featured in our training publications.**
(Your name is required at the bottom of this form)

☐ **Please "check" this box if you would like to receive more information on our other courses and services.**
(Your name is required at the bottom of this form)

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