UNIGRAPHICS

MULTI–AXIS MACHINING

STUDENT MANUAL

November 2002

MT11050 – Unigraphics NX

EDS Inc.

Proprietary & Restricted Rights Notices

Multi−Axis Machining Student Manual Publication History:

Original Release . . . . August 2000
Version 17.1.1 . . . . February 2001
Version 18.0 . . . . November 2001
Unigraphics NX . . . . November 2002

Table of Contents

(This Page Intentionally Left Blank)

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 Unigraphics 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 Unigraphics:

- 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 Unigraphics multi-axis programming practices

Student Responsibilities

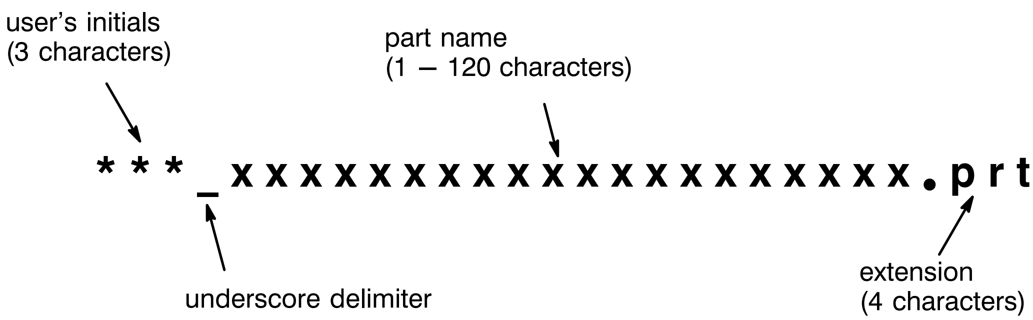
- be on time
- participate in class
- focus on the subject matter
- listen attentively and take notes
- enjoy the class

Class Standards for Unigraphics Part Files

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 File 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.

> **TIP** Currently up to 128 characters are valid for file names. A four character extension (.prt, for example) is automatically added to define the file type. This means the maximum number of user defined characters for the file name is actually 124.

The Arrow Symbol (→)

The arrow symbol (→), represents that you choose an option, then immediately choose another option. For example, **Tools→Operation Navigator→Toolpath→Replay** means:

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

Layers and Categories

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	***Valid colors***
Bodies	
Solid	*Green*
Sheet	*Yellow*
Generating Curves (non-sketch)	
Lines and Arcs	*Orange*
Conics and Splines	*Blue*
Sketches	
Sketch Curves	*Cyan*
Reference Curves	*Gray*
Datum Features	*Aquamarine*
Points and Coordinate Systems	*White*
System Display Color	*Red*

Seed Part

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

- sketch preferences
- commonly used expressions
- layer categories
- user-defined views and layouts
- part attributes

How to Use This Manual

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

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

Step 1 This is an example of a step.

❑ This is an example of an action box.

The general format for lesson content is:

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

While working through lesson activities, give attention to the CUE and Status lines for additional information.

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

Workbook Overview

The 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: **Username:** ______________________________

Password: ______________________________

Work Directory: ______________________________

Parts Directory: ______________________________

Instructor: ______________________________

Date: ______________________________

Student and Workbook Parts

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

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

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

System Privileges

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

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

Introduction to Four and Five Axis Machining

Lesson 1

PURPOSE

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

OBJECTIVES

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:

- Unigraphics 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, Unigraphics 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, make sure 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

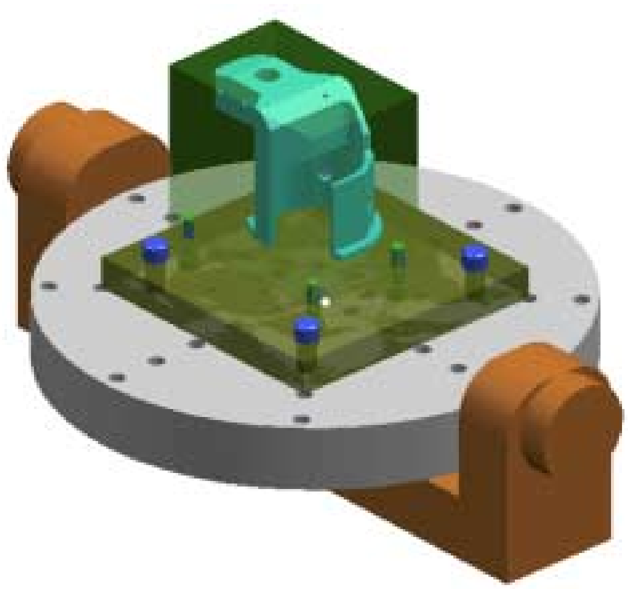
The following activity requires you to generate a tool path at other than a normal tool axis of (0,0,1).

Activity 1—1: 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.

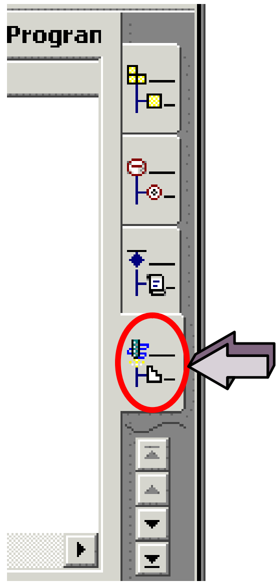
Step 1 Open an existing part file and enter the Manufacturing Application.

❑ Choose **File→Open** then choose **mam_collar_mfg.prt**.



❑ Choose **Application→Manufacturing.**

❑ Choose the Operation Navigator tab from the resource bar.

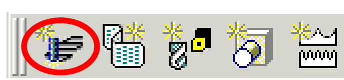


The Operation Navigator and the part are displayed.

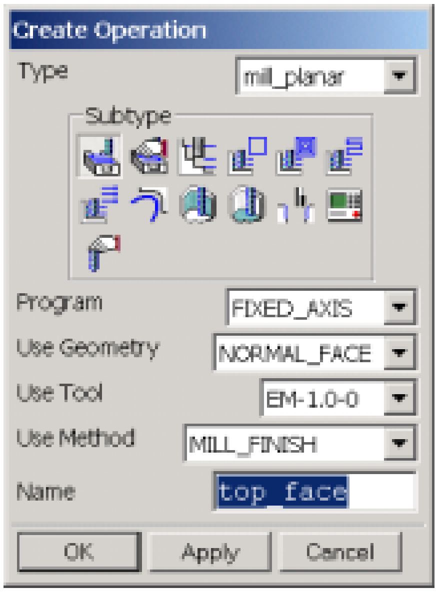
Step 2 Create the finishing operation.

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 tool bar.

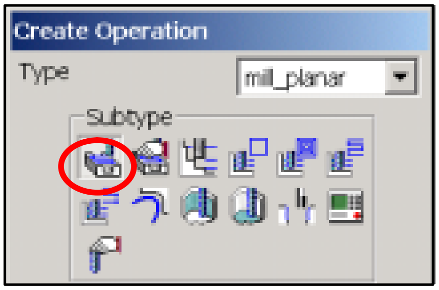


The Create Operation Dialog is displayed



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

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



❑ Choose the following Parent objects:

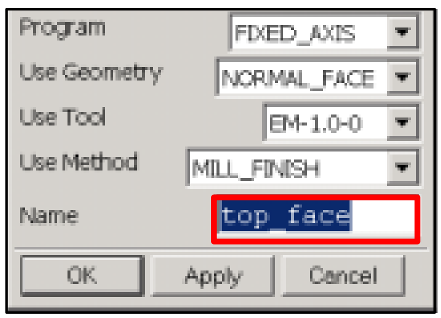
Program: FIXED_AXIS
Geometry: NORMAL_FACE
Tool: EM–1.00–0
Method: MILL_FINISH

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.

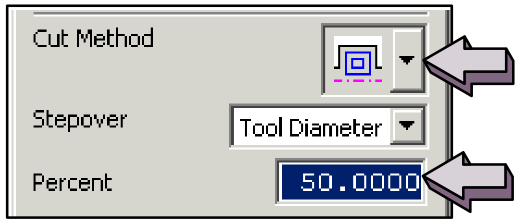
- ❑ Key in **top_face** as the name of the operation.



- ❑ 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.

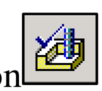
❑ Choose OK to accept the operation.

Step 3 Verify the results.

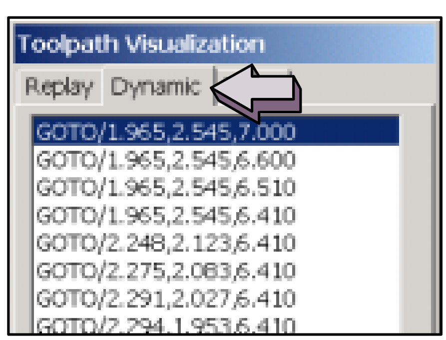
You will now verify the results by using **Toolpath Visualization**.

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

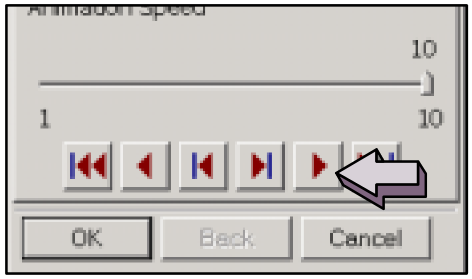
❑ Highlight the **FIXED_AXIS** program object.

❑ Choose the **Verify Toolpath** icon  from the tool bar.

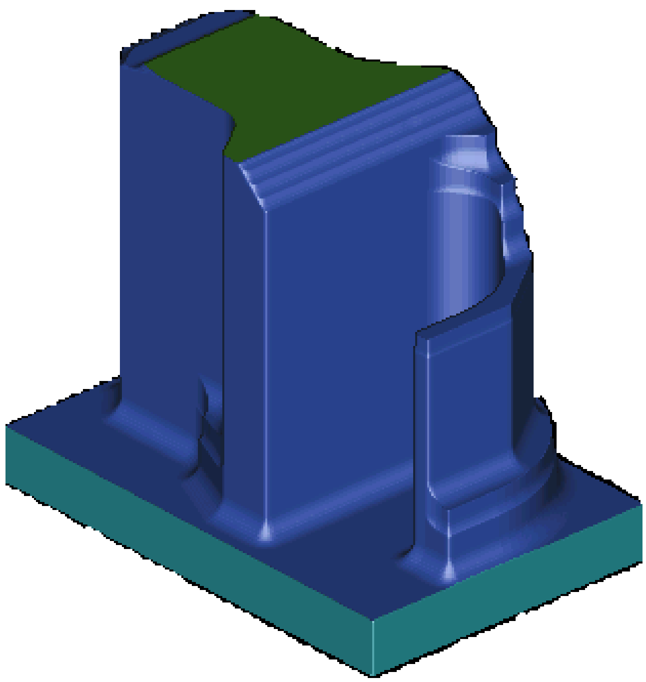
❑ Choose the **Dynamic** tab from the **Toolpath Visualization** dialog.



❑ Choose the **Play Forward** 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.



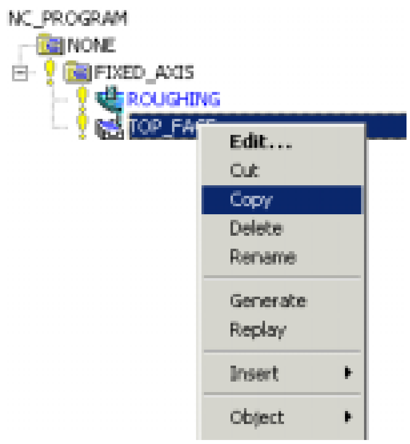
Verifying the operation indicates the tool path to be acceptable, you will now continue with the next operation.

❑ Choose **Cancel** from the **Toolpath 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.

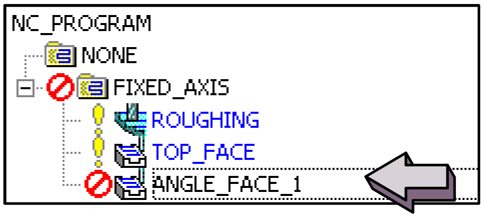
❑ Highlight the **TOP_FACE** operation and choose **MB3** → **Copy**.



❑ 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.

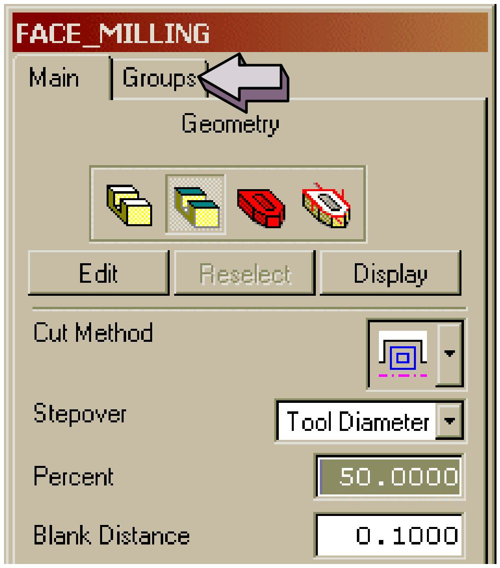
❑ Change the name of the new operation by highlighting the **TOP_FACE_COPY** operation, using **MB3,** choose **Rename**, and then type the name **ANGLE_FACE_1**.



You will now change the geometry parent object.

❑ Double click on the **ANGLE_FACE_1** operation.

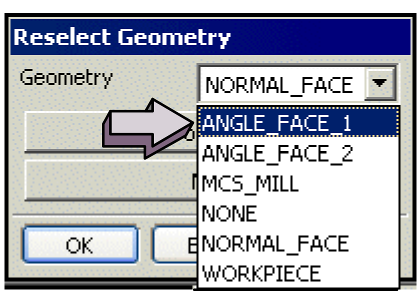
❑ Choose the Groups Property Page



❑ Choose the *Geometry* radio button at the top of the dialog, then choose **Reselect**, as shown.

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**.

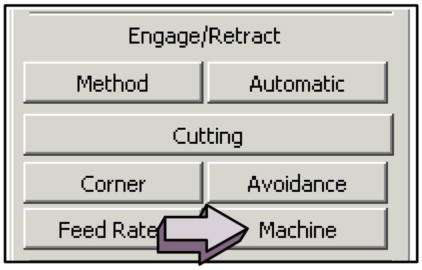
❑ Choose **OK** on the Display Parameters dialog.

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.

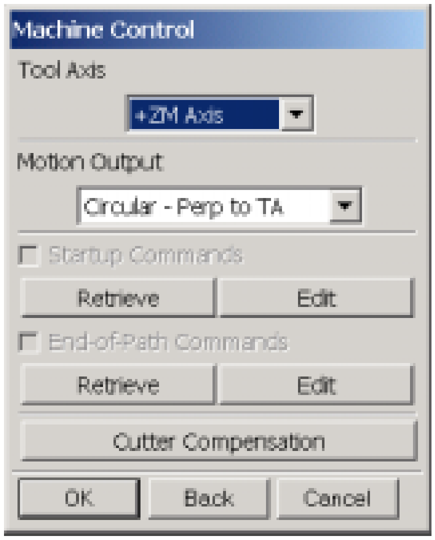
❑ Choose **OK** from the error dialog.

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



1

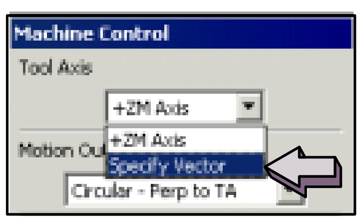
The Machine Control dialog is displayed.



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.

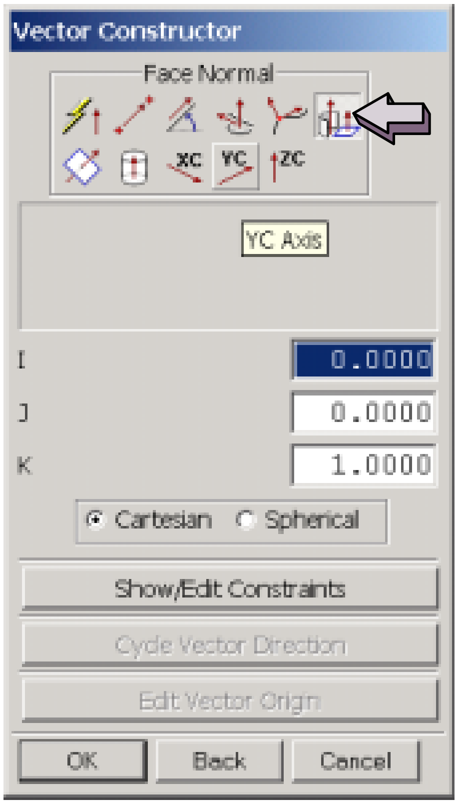
❑ Choose **Specify Vector**.



The Vector Constructor dialog is displayed.

1

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

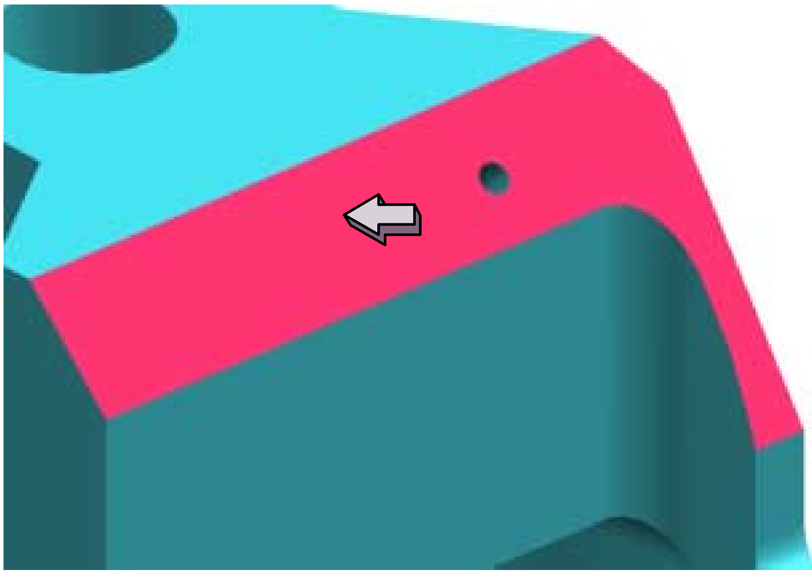


Note that **Face Normal** means to set the vector perpendicular to the face.

1

❑ 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 **Dynamic 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.

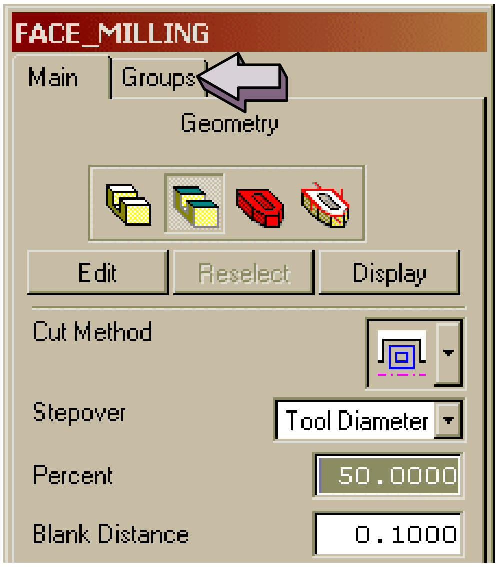
❑ Choose **MB3** → **Copy.**

❑ Choose **MB3** → **Paste**.

❑ Change the name of the new operation to **ANGLE_FACE_2**.

❑ Edit the operation by doubling clicking on **ANGLE_FACE_2**.

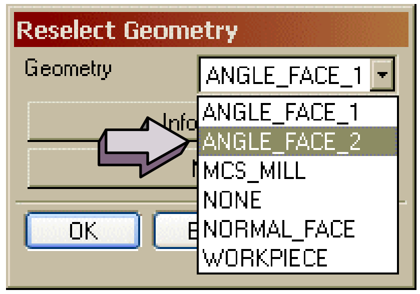
❑ Choose the Groups Property Page



❑ Choose the *Geometry* radio button at the top of the dialog, then choose **Reselect**, as shown.

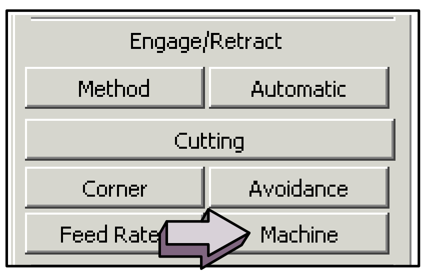
The Reselect Geometry dialog is displayed.

- ❑ Choose **ANGLE_FACE_2**, as the geometry parent, from the pull down list**.**

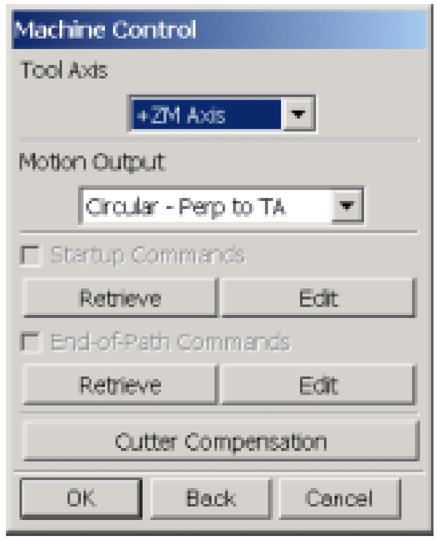


- ❑ Choose **OK.**

- ❑ Choose the **Machine** button located on the **FACE_MILLING** dialog.

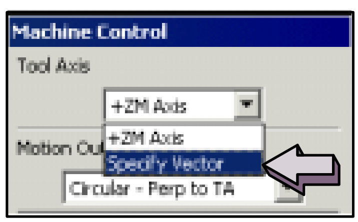


The Machine Control dialog is displayed.



❑ Select the **Tool Axis** pull down arrow.

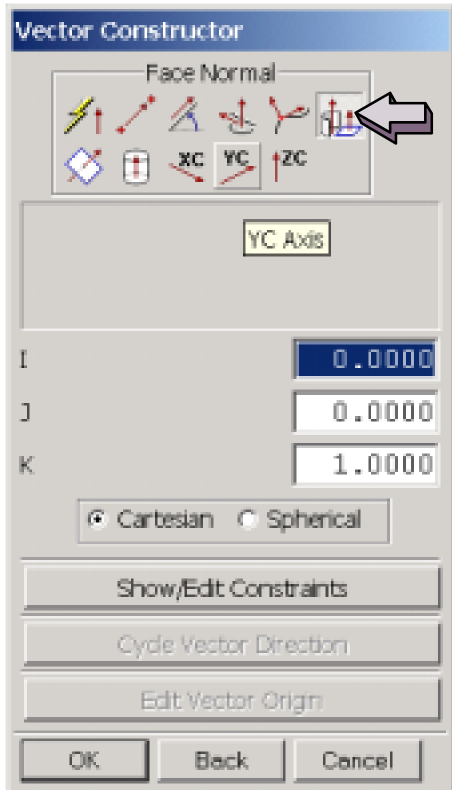
❑ Choose **Specify Vector**.



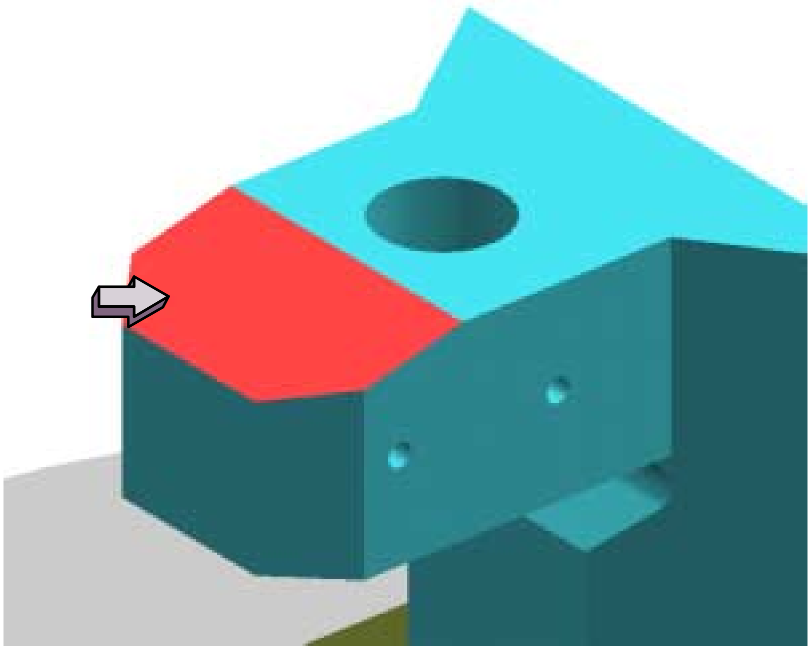
The Vector Constructor dialog is displayed.

1

❑ 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.

❑ As you have previously done, use Dynamic Verification to display the machined body.

❑ Close the part file without saving.

This concludes the activity.

Defining the Center of Rotation for a rotary axis

In order for machining to occur about a rotary axis, the position of the rotary axis must be defined. There are generally two methods to accomplish this definition:

- place the WCS/MCS at the center of axis rotation
- use a “FROM” statement to define the X,Y, Z distance from the MCS of (0,0,0) to the center of axis rotation

Placing the MCS at the Center of Axis Rotation

Utilizing this method, the programmer simply positions the part on the fixture in a normal position. Then, instead of placement of the MCS at the part (0,0,0) the MCS is placed at the center of rotation.

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
- fixture crashes may require some type of reprogramming

Placing the MCS at the Part Origin

Using this method, the programmer places the MCS at the part origin. Prior to postprocessing, the programmer provides specific information to the post with respect to the distance from the MCS (0,0,0) to the center axis of rotation. This is usually performed with a **FROM** (preferred) or **ORIGIN** statement located within the operation or CLSF.

Advantages:

- output in the program matches the part print
- fixture crashes can be solved by changing the **FROM** statement

Disadvantages:

- more work on programmers part - - - need to provide accurate **FROM** statement
- may be more confusing for machine operators

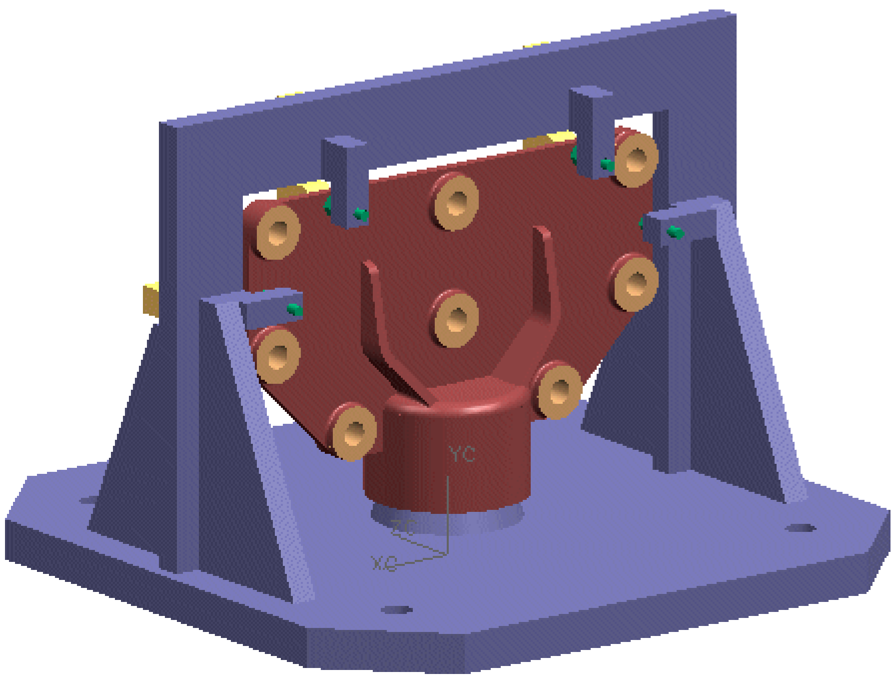
The following activity will address both methods showing results before and after a simulated fixture crash.

Activity 1—2: MCS Placement in Multi-Axis applications

In this activity, you will ensure that the MCS is at the center of axis rotation. You will then postprocess the program and examine the results.

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

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



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

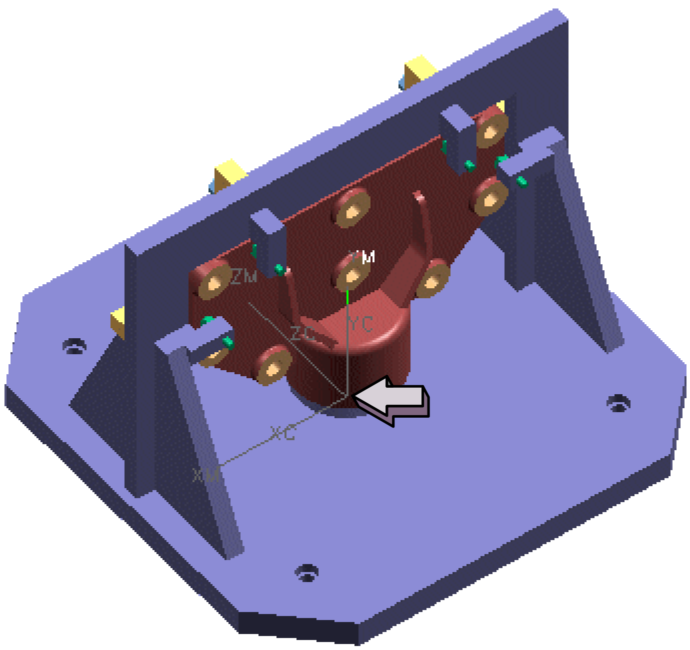
Step 2 Examine the NC Program.

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

❑ Highlight the **FOUR-AXIS_PROGRAM** parent object, using **MB3**, choose **Replay**.

The program machines both sides of the cast pads, spot-drills and then drills the thru-holes. If you so desire, you may also analyze the program movements by using Dynamic Verification.

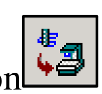
❑ Change to the **Geometry View** of the Operation Navigator.



If you verify the WCS/MCS, you will notice that it is placed at the center of axis rotation.

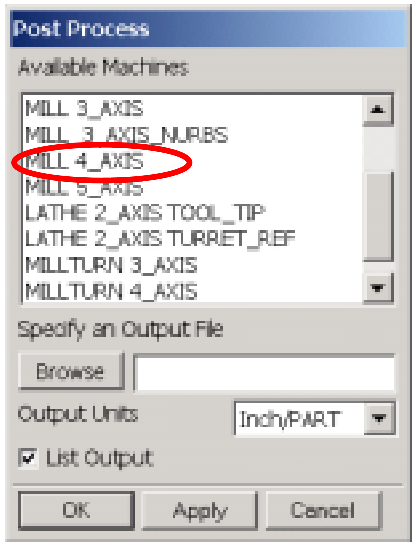
❑ Change back to the **Program Order View** of the Operation Navigator**.**

❑ Highlight the **FOUR-AXIS_PROGRAM** parent object in the Operation Navigator.

❑ Choose the UG/Postprocess icon .

The Post Process dialog is displayed.

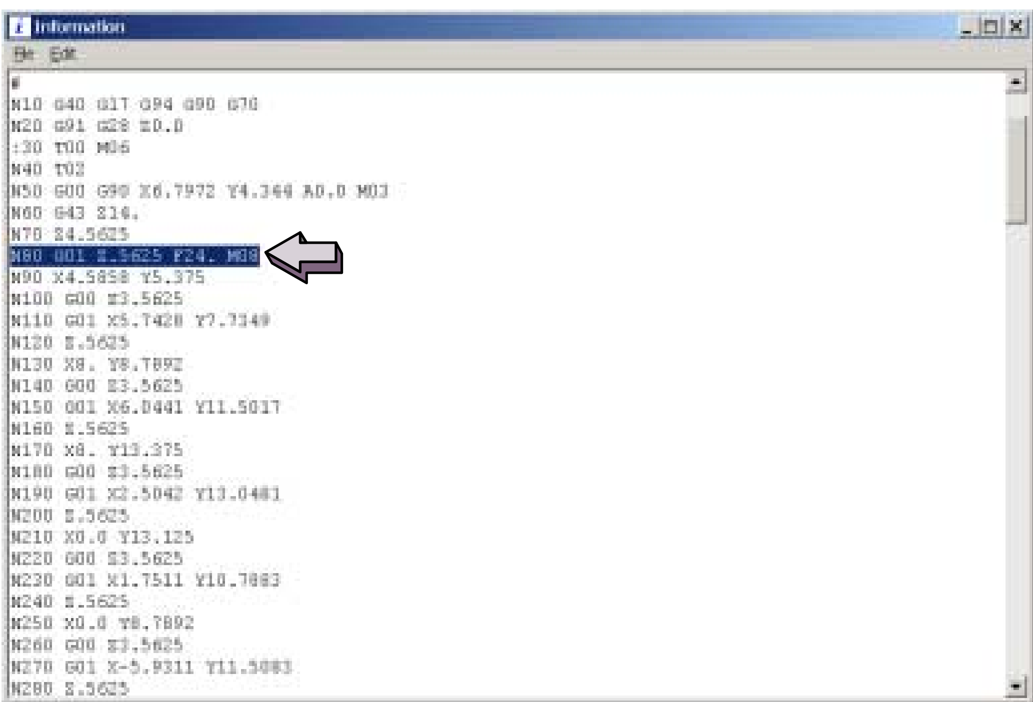
1



❑ Choose the **MILL4_AXIS** post from the list and then choose **OK**.

❑ If necessary, choose **OK** to the warning concerning overwriting an existing file**.**

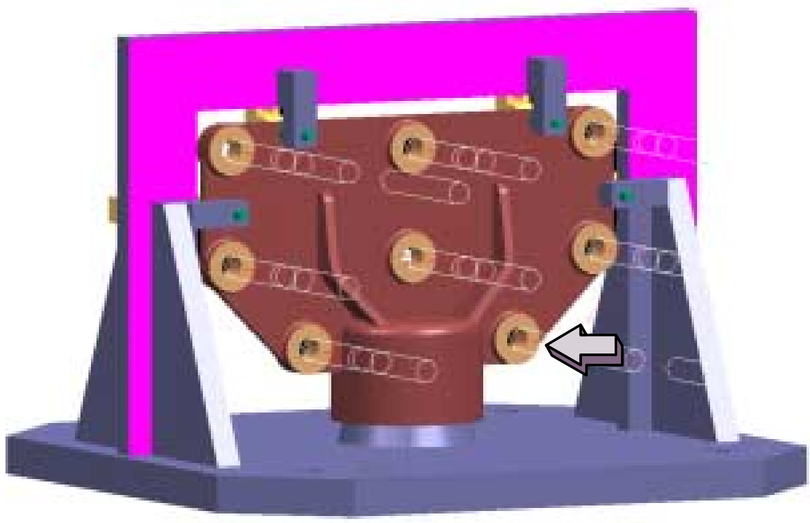
The posted output is displayed.



Notice block 80. The Z.5625 value is the last Z motion which occurs prior to the first pad being cut.

❑ Scroll down to the bottom of the list.

The last drilled hole (G81X....) is shown in the following figure:



This particular hole's dimension is measured from the center of rotation to the hole center.

No further action is required from the programmer, other than to document where the (0,0,0) location should be placed physically on the machine.

Step 3 Modify the program.

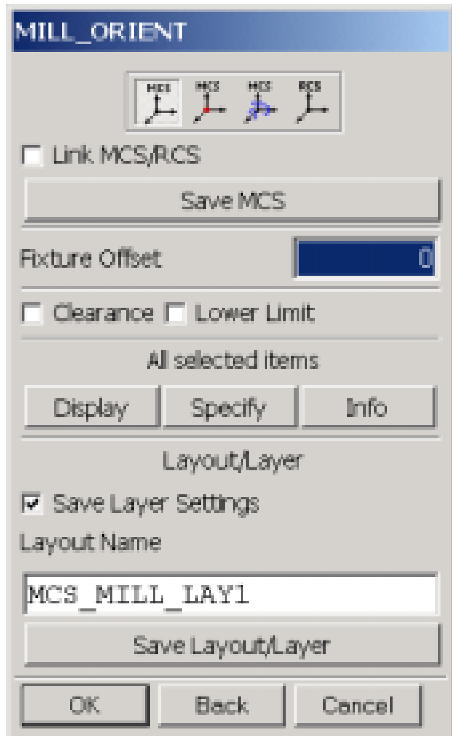
During the last production run on this particular job, the spindle head crashed into the holding fixture. Consequently, the fixture moved on the table, breaking the fixture keys. After numerous attempts to realign the fixture to its original location, it is still misaligned in the Z+ direction, when the table is at 0 degree position, by .0125".

The decision has been made to permanently modify the program to account for this discrepancy.

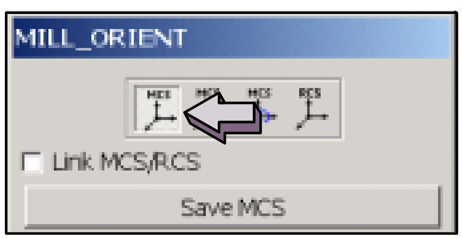
❑ Change to the *Geometry View* in the Operation Navigator.

❑ Double-click on the **MCS_MILL** parent group object.

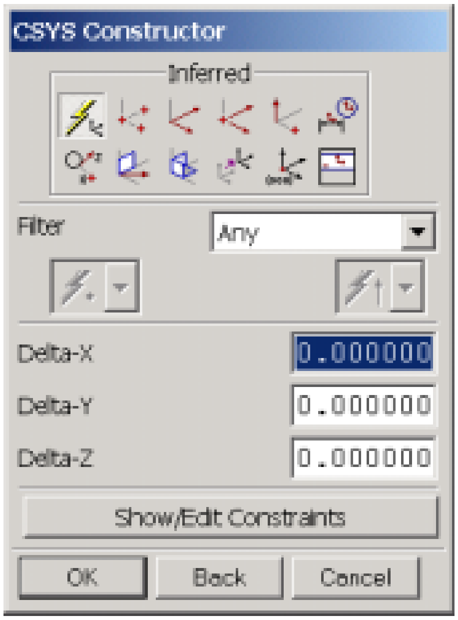
The **MILL_ORIENT** dialog is displayed.



❑ Click on the **MCS** icon at the top of the **MILL_ORIENT** dialog.

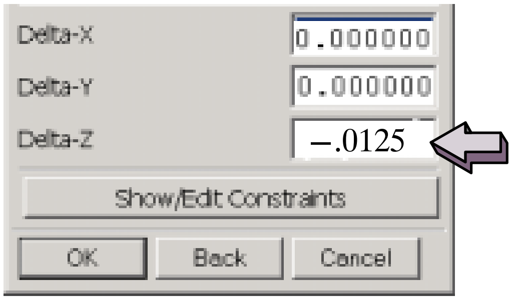


The **CSYS Constructor** dialog is displayed.



The default setting is inferred, you only need to key in the value of the offset.

❑ In the **Delta-Z** field, key in the value **-.0125**.



❑ Choose **OK**.

❑ Choose **OK** again to accept the changes in the **MILL_ORIENT** dialog.

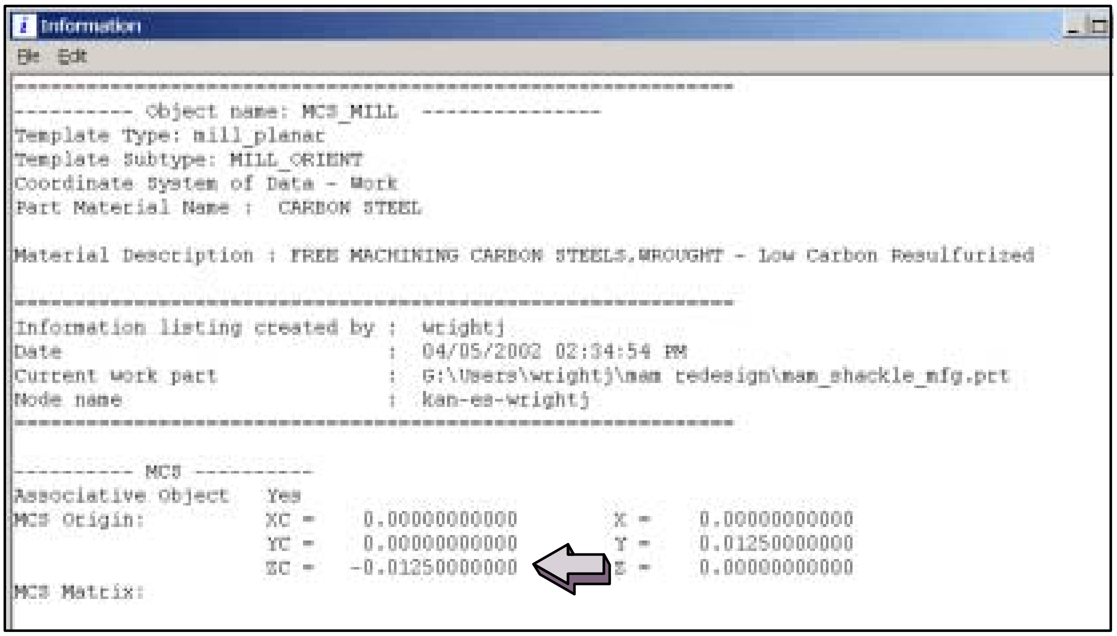
Since Unigraphics knows the new position of the MCS, it is not necessary to re-generate all of the operations that are within the **MCS_MILL** parent object. You may also notice that the subsequent operations changed from a  to a . This represents the postprocessed output as being out-of-date and needing to be re-posted.

You will now check the changes which you just made.

❑ Highlight the **MCS_MILL** parent object in the *Geometry View*.

❑ Using MB3, choose **Properties**.

The MCS is now located from the WCS by .0125”.



```
Information
File Edit
==============================================================
---------- Object name: MCS_MILL ---------------
Template Type: mill_planar
Template Subtype: MILL_ORIENT
Coordinate System of Data - Work
Part Material Name :  CARBON STEEL

Material Description : FREE MACHINING CARBON STEELS,WROUGHT - Low Carbon Resulfurized

==============================================================
Information listing created by :  wrightj
Date                           :  04/05/2002 02:34:54 PM
Current work part              :  G:\Users\wrightj\mam redesign\mam_shackle_mfg.prt
Node name                      :  kan-es-wrightj
==============================================================

---------- MCS ----------
Associative Object    Yes
MCS Origin:           XC =    0.00000000000        X =    0.00000000000
                      YC =    0.00000000000        Y =    0.01250000000
                      ZC =   -0.01250000000        Z =    0.00000000000
MCS Matrix:
```

❑ Change to the Program Order View of the Operation Navigator.

❑ Highlight the **FOUR-AXIS_PROGRAM** parent object.

❑ Postprocess using the **MILL_4_AXIS** postprocessor.

❑ Choose **OK** when you receive the warning about overwriting the existing file.

❑ Re-examine block 80.

All Z motions now have the .0125” value added to them. This completes the activity and finishes the lesson.

1

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
- moved the MCS to compensate for fixture collisions and other modifications that may take place at the machine tool

Sequential Mill Basics

Lesson 2

PURPOSE

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

OBJECTIVES

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

This lesson contains the following activities:

2

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).

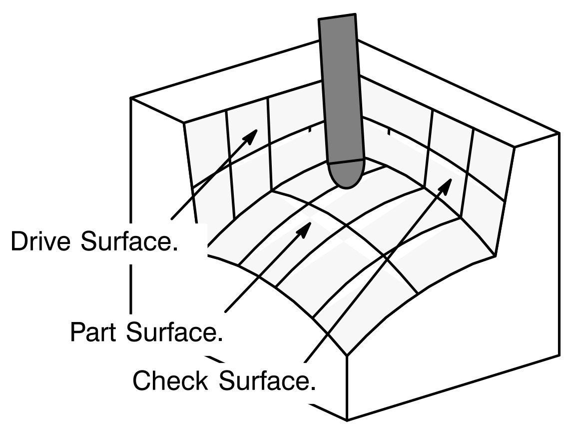
Sequential Mill also provides some unique tool axis control capabilities in maintaining a tool position relative to drive and part geometry, with the capability to recognize multiple check surfaces.

Sequential Milling Terminology

The following terms pertain to Sequential Milling:

Part, Drive, and Check surfaces -

- Part surface controls the bottom of the tool; the tool tip will follow the contour of the selected part surface
- Drive surface controls the side of the tool; the side of the tool will follow the contour of the selected drive surface
- 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.

Stopping Position -

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

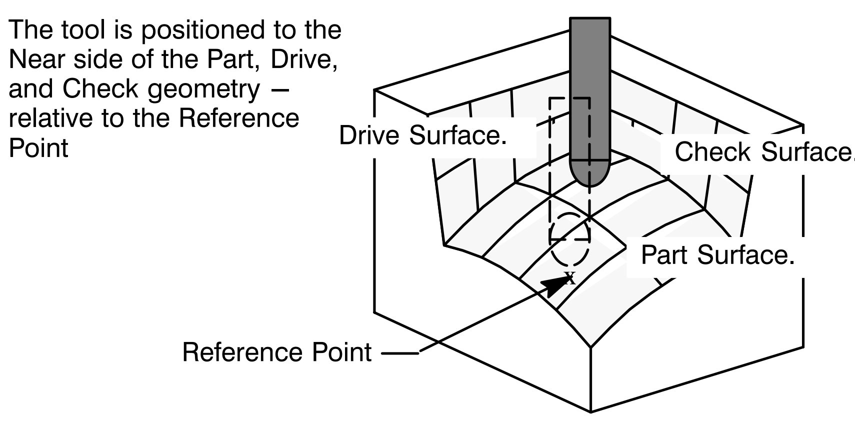
- **Near Side** indicates that the tool will stop when it reaches the closest side of the specified part, drive or check geometry, relative to the current tool position
- **Far Side** indicates that the tool will stop when it reaches the farthest side of the specified part, drive or check geometry relative to the current tool position
- **On** indicates that the tool will stop when its center axis reaches the edge of the specified part, drive or check geometry 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.

Reference Point and Near Side/Far Side -

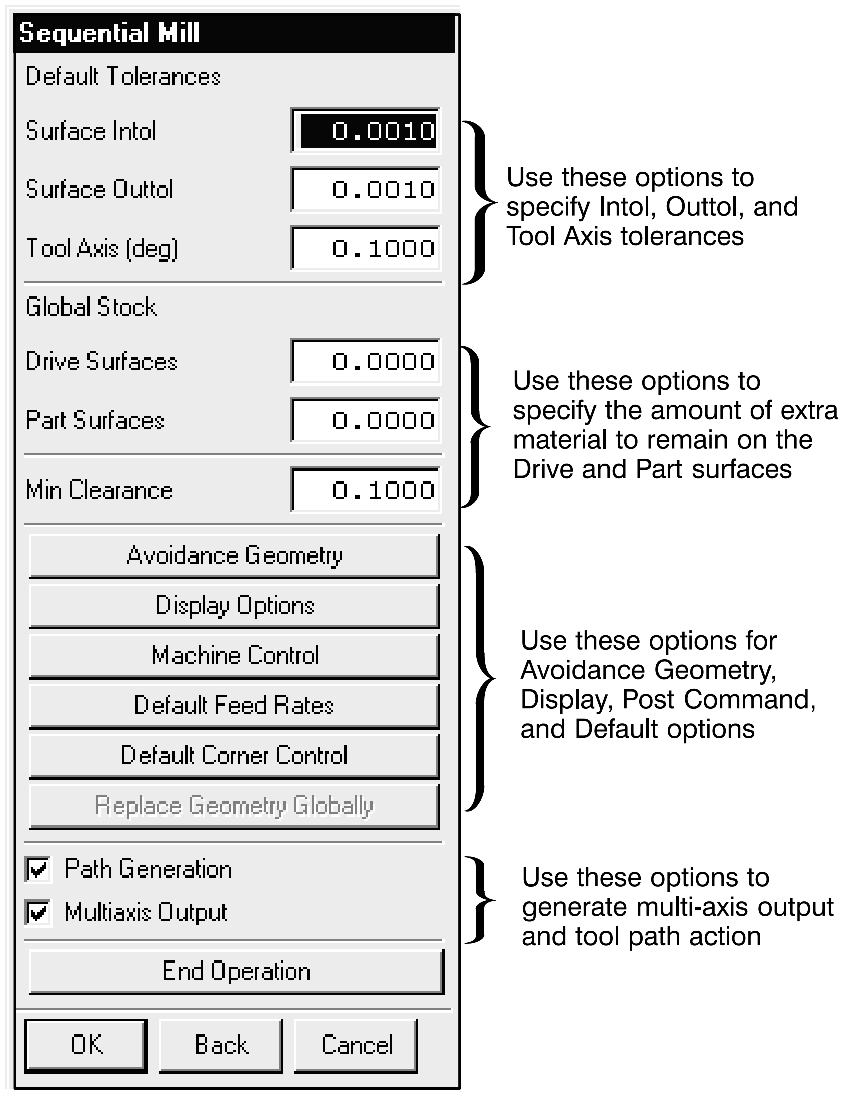
You must initially specify a tool Reference Point position to determine the correct side of the drive, part, and check geometry for tool placement. The tool does not move to the Reference Point, it only uses this point to establish the direction to part geometry.

Once you specify the Reference Point, you can specify the tool starting position as the Near Side, the Far Side, or On the Drive, Part, or Check geometry. The tool will then move to that position.



The Sequential Mill dialog -

When you create or edit a Sequential Mill Operation, the Sequential Mill dialog is displayed.

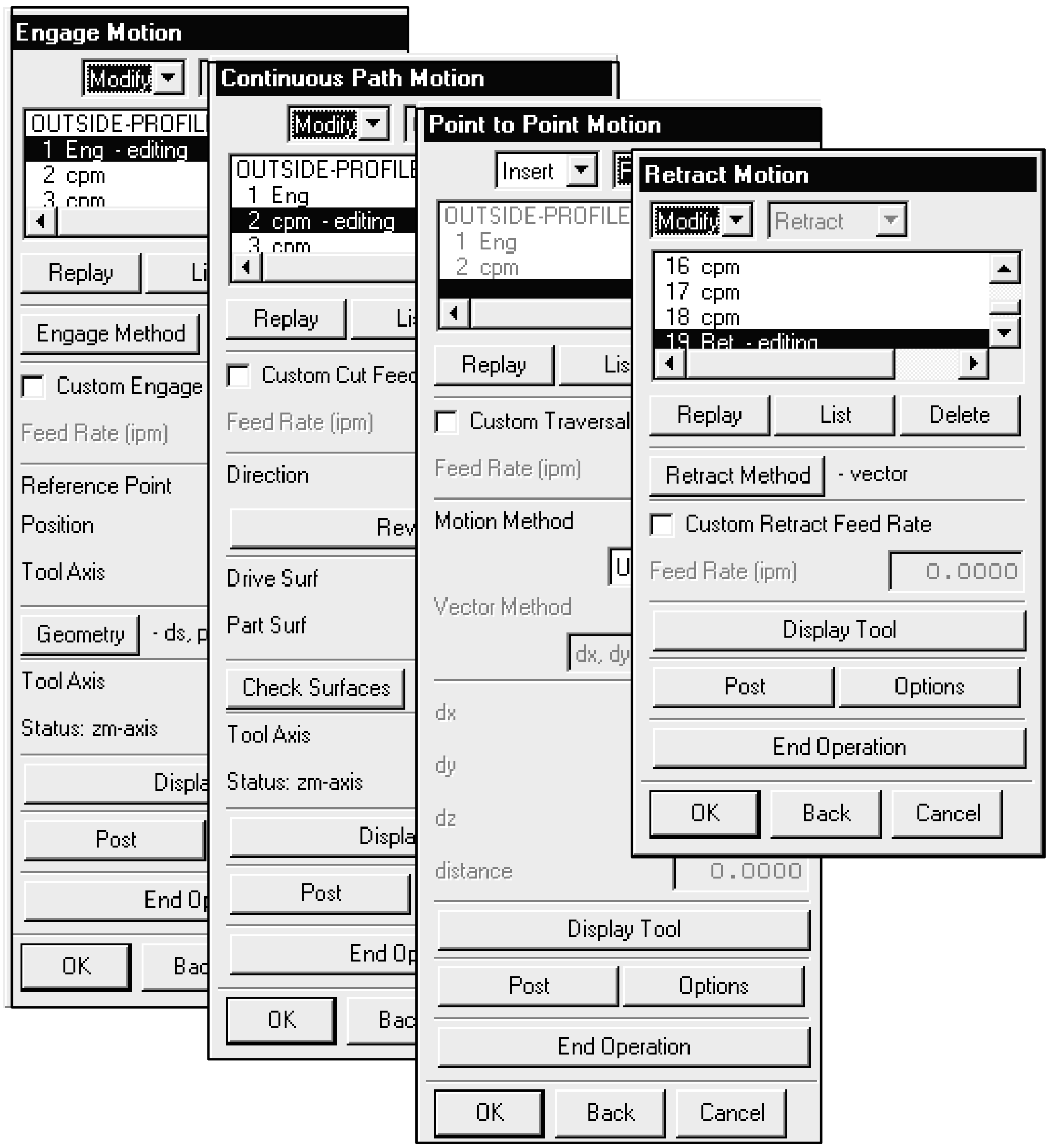


Options are available to:

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

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

Sub-operations are individual tool motions which comprise a complete tool path. The four different types of sub-operations are **Engage**, **Continuous Path**, **Point to Point** and **Retract** motion.



Normally, you will use these four dialogs in sequential order.

- initially, you specify an **Engage** move
- then, you specify **Continuous Path** motions
- finally, at the end of the tool path, you will specify a **Point to Point** and then a **Retract** move

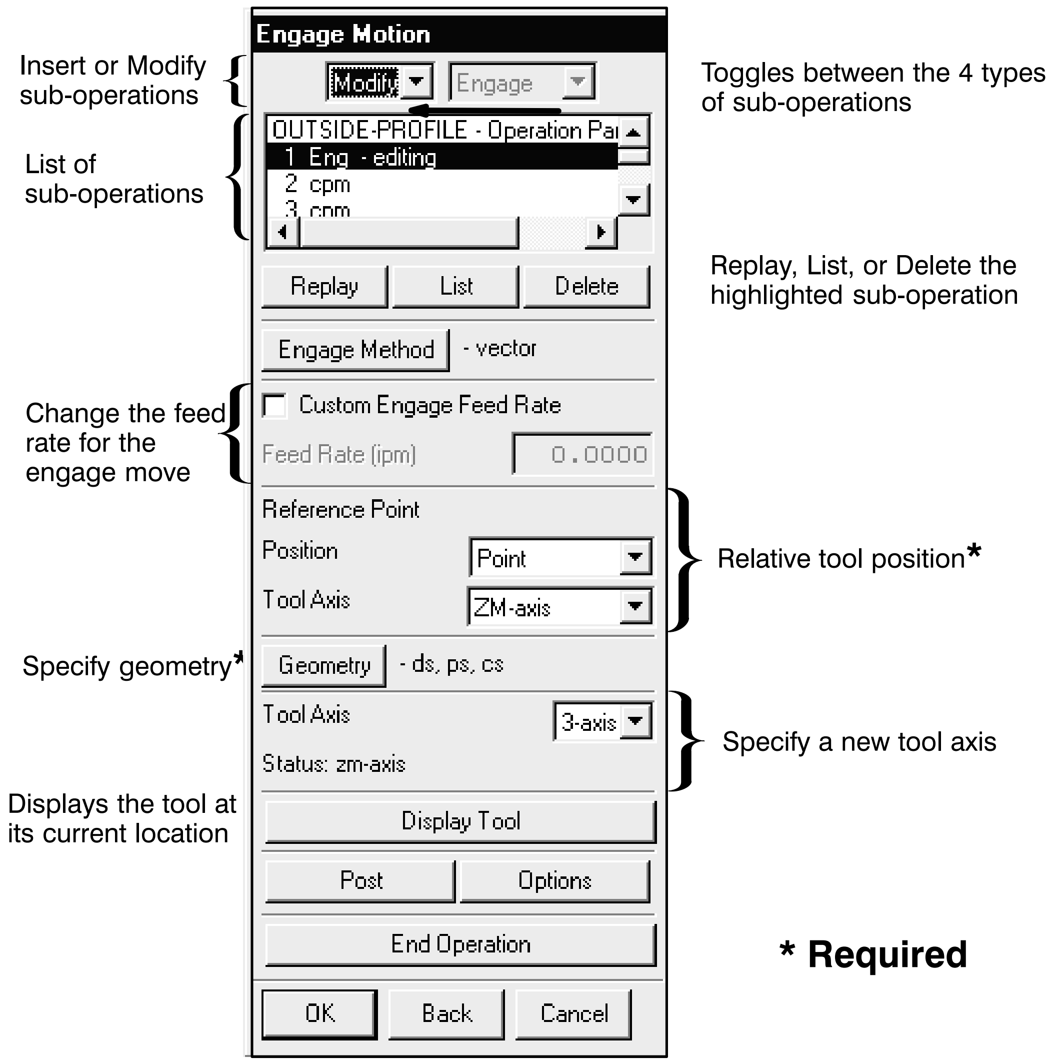
The first sub-operation uses the **Engage Motion** dialog to create a tool motion from the Start Point or the Engage Point to the initial cut position.

The **Continuous Path Motion** dialog creates a sequence of cuts, followed by the **Retract Motion** and **Point to Point Motion** dialogs used for final tool positioning.

After creating or editing an operation, you choose **End Operation** and then generate the tool path, or choose **OK** and 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.



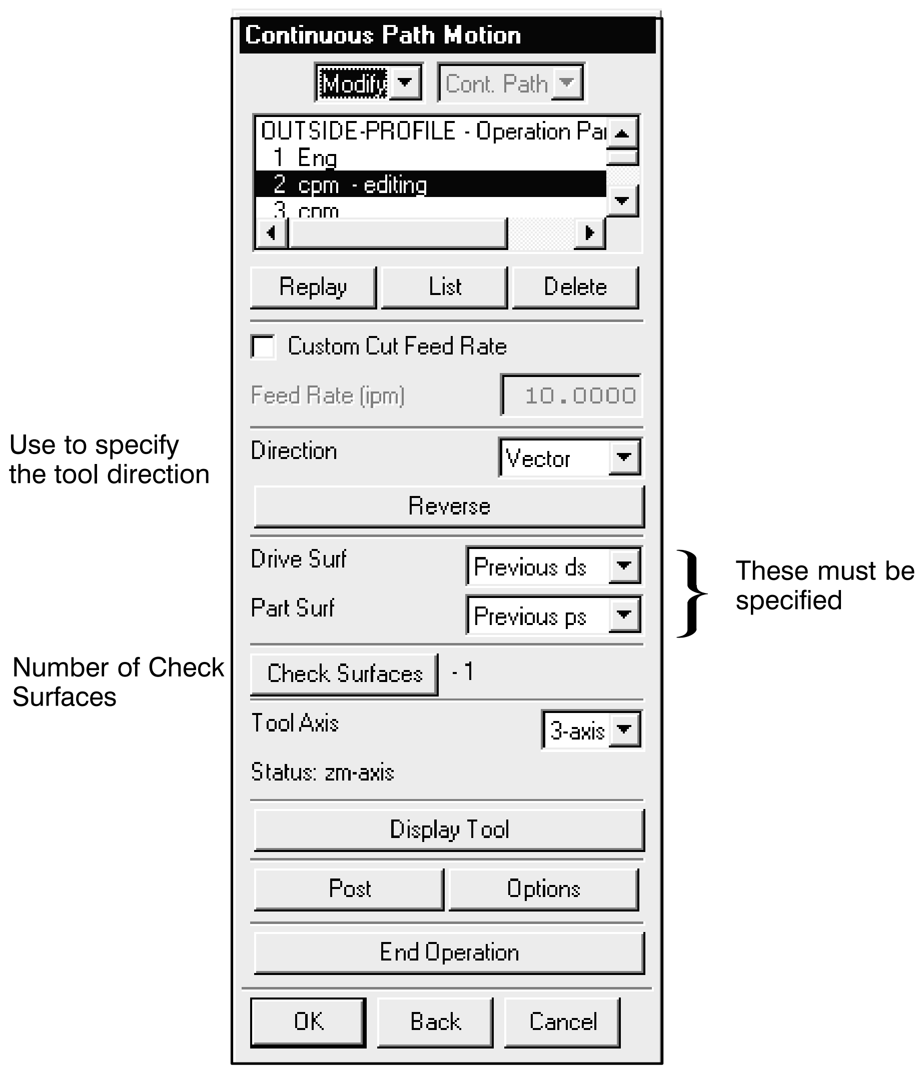
The Continuous Path Motion dialog

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.

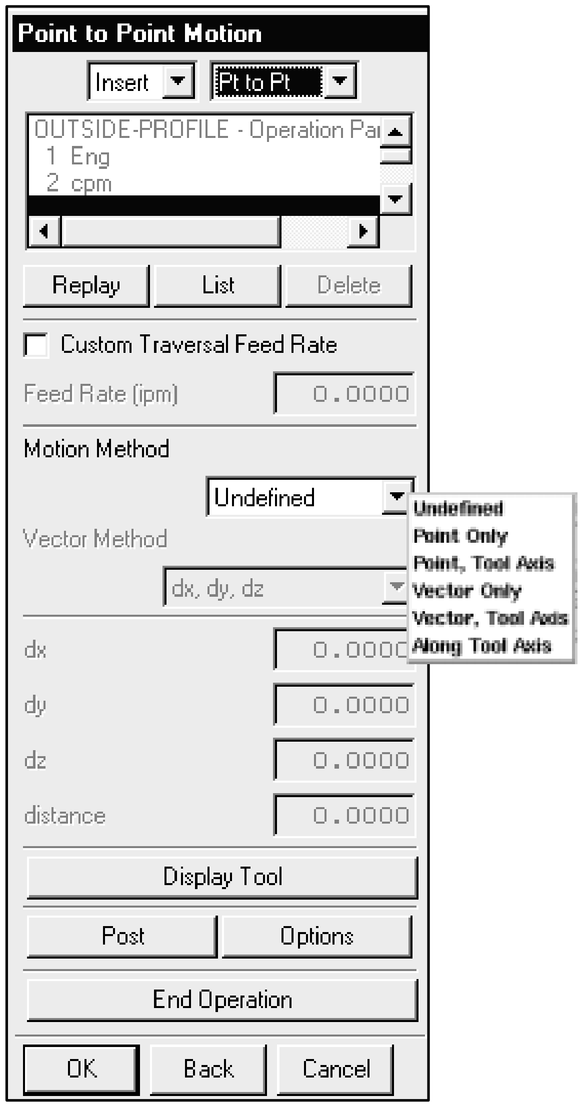


The Point To Point Motion dialog

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.

Use to specify a special traversal feed rate.

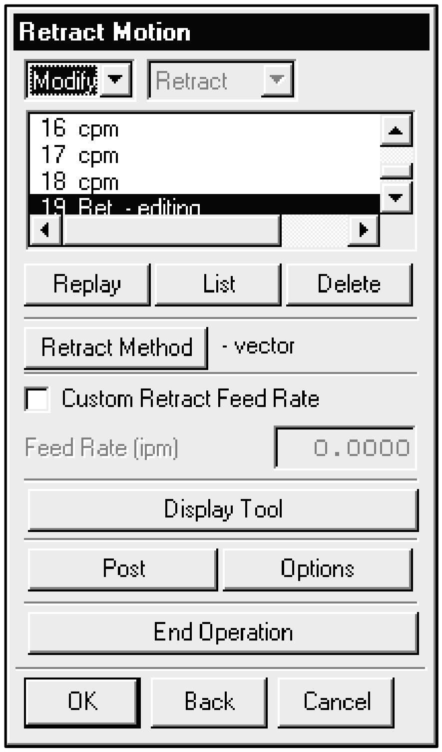
Defines the way the tool will move to the next location



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.

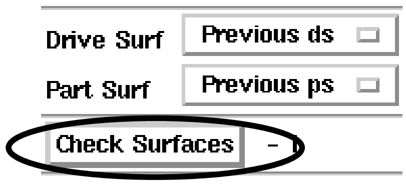
Type of retract move



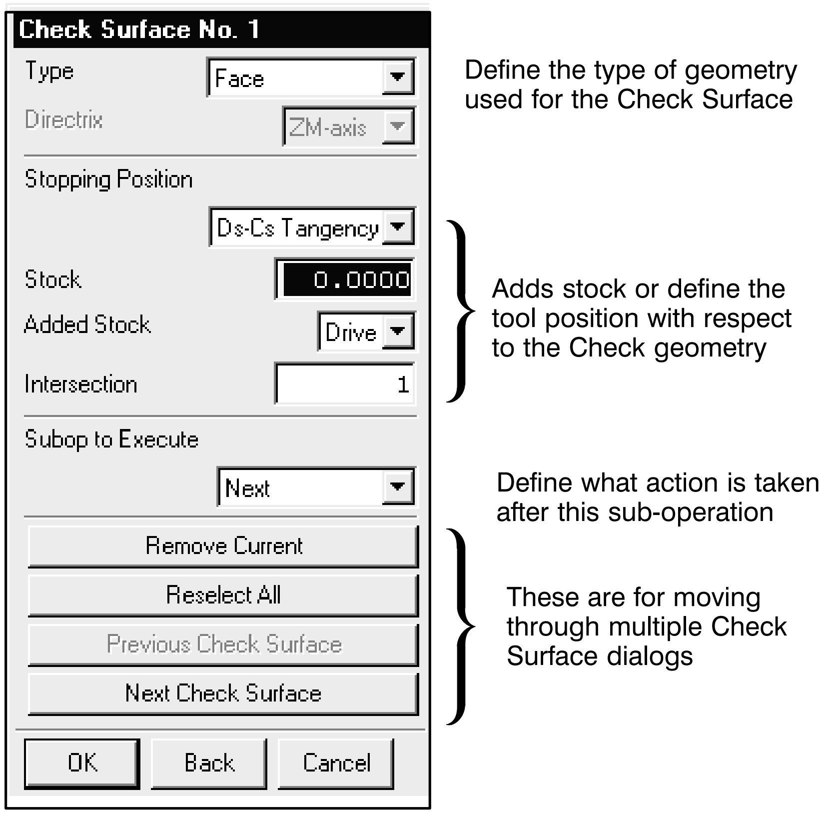
Change the feed rate for the retract move

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. Similarly, 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.

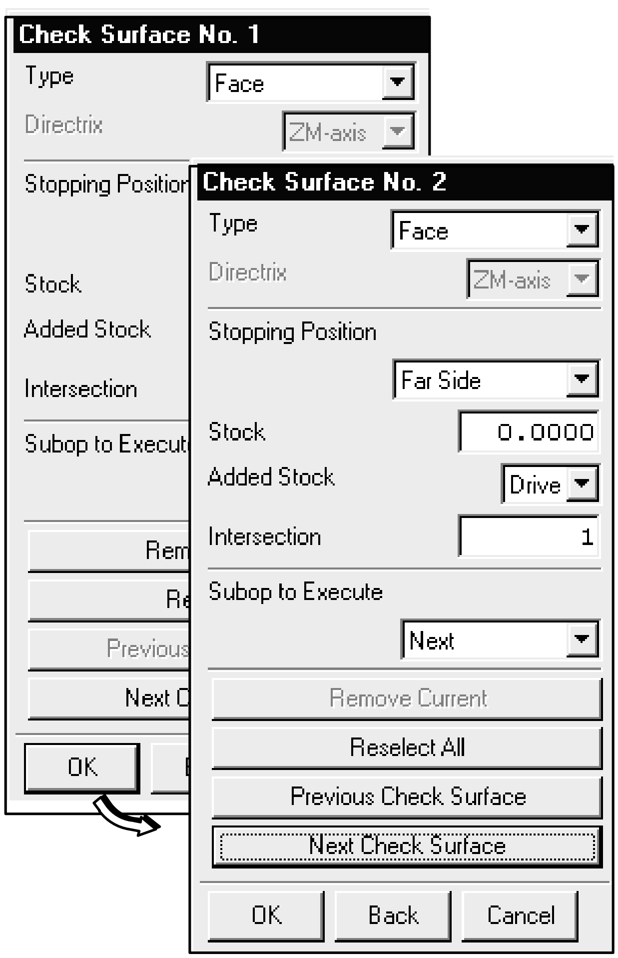


Multiple Check Surfaces

In a Continuous Path Motion command the cutter moves along the Drive Surface and the 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. You can continue to define the number of Check Surfaces which are required.



The following activities will familiarize you with Sequential Mill operations.

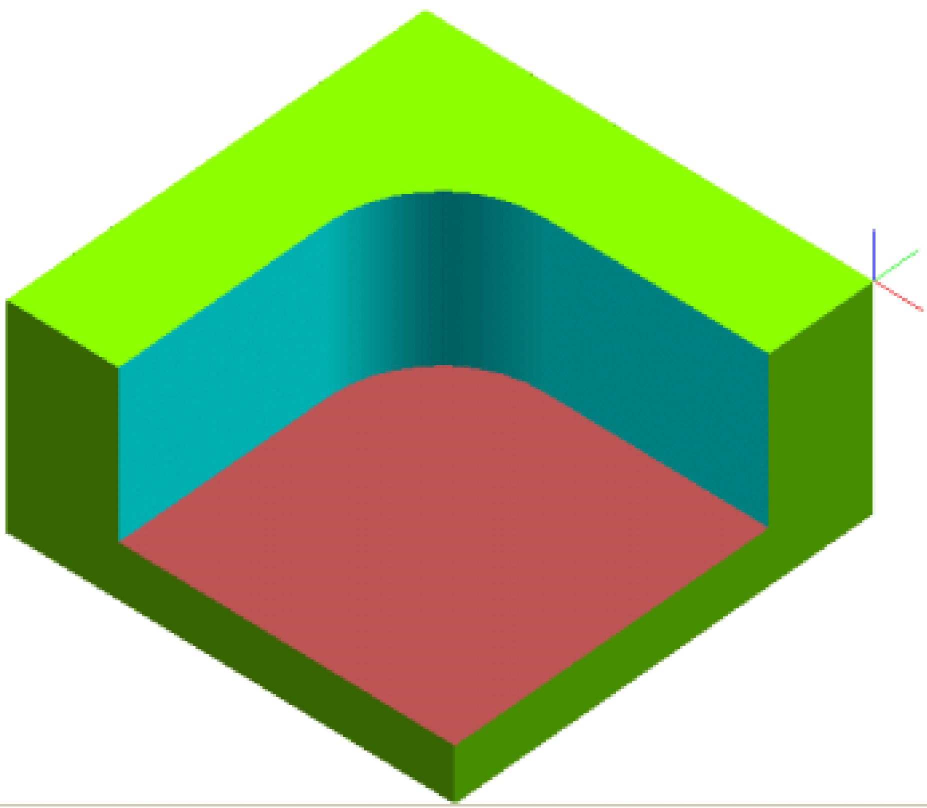
Activity 2–1: Basic Sequential Milling Techniques

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 sub-operations.

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

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

Even though this is a very simple part, it is still programmed in the context of an assembly. The top-level component, **mam_box_mfg.prt**, contains all of the manufacturing data. The **mam_box_stock.prt** file contains a WAVE-linked representation of the raw material and the **mam_box.prt** file contains the designed part that is to be machined.



The raw material file, **mam_box_stock.prt**, has been hidden from the display. To enable the display, choose the check mark next to the appropriate file in the Assembly Navigator.

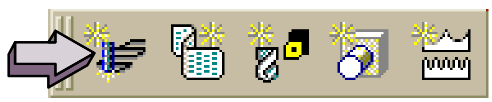
❑ Rename the part ***_box_mfg.prt** using the **File→Save As** option on the menu bar (*** represents your initials).

❑ Enter the **Manufacturing application** by choosing **Application → Manufacturing.**

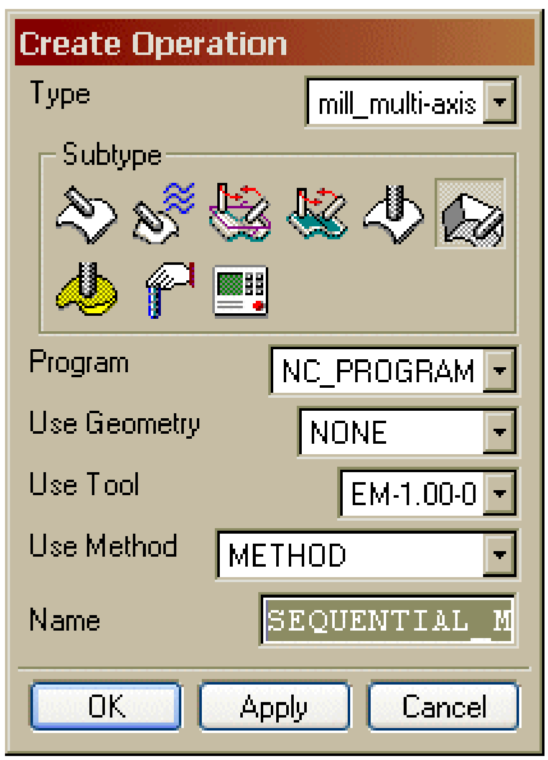
The necessary Parent Groups (i.e. Geometry, Machine, Program and Method) have already been created for you.

Step 2 Create a Sequential Milling operation.

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

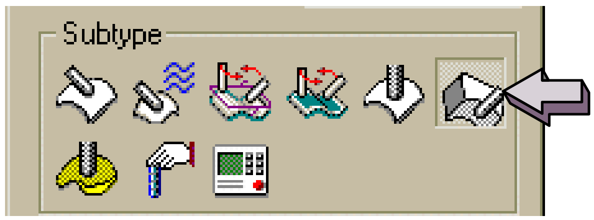


The Create Operation dialog is displayed.

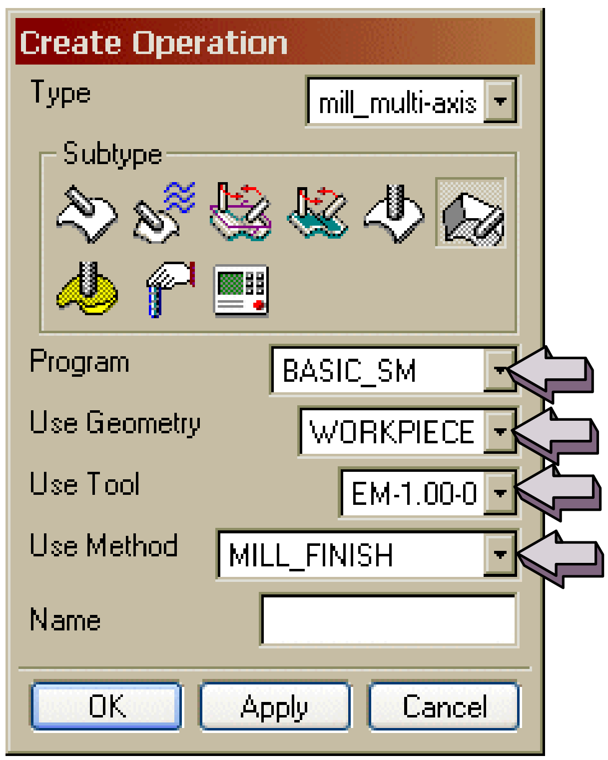


❑ If necessary, change the Type to **mill_multi-axis**.

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



❑ Set the Parent objects as follows:

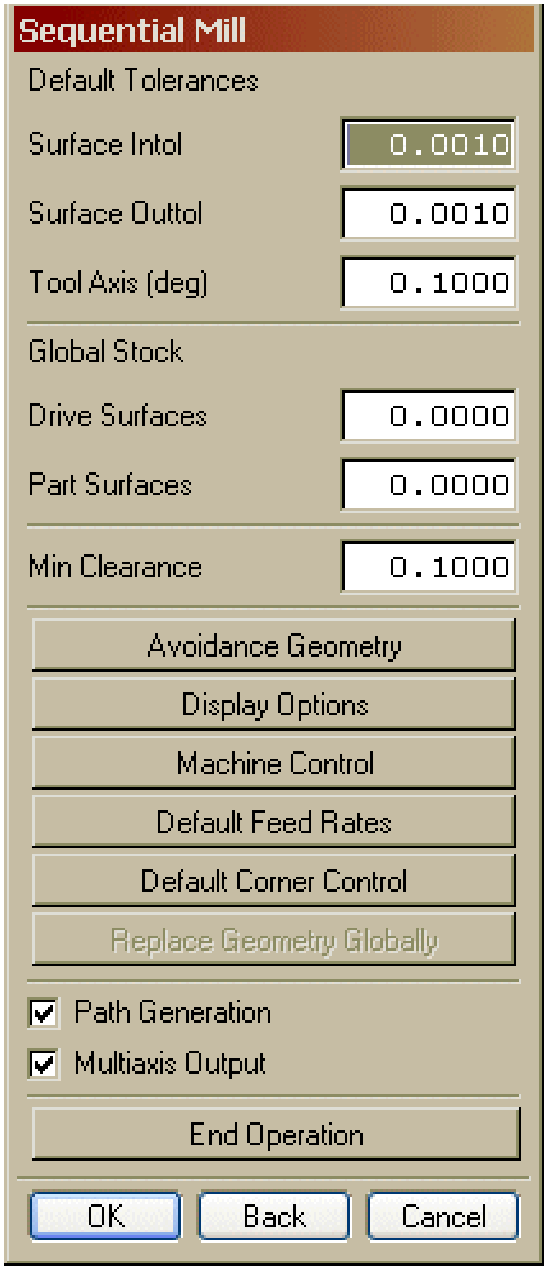


❑ Name the operation **SM_1**.

2

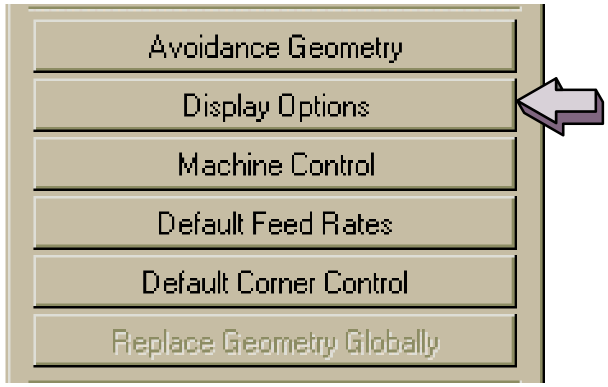
❑ Choose **OK**.

The Sequential Mill dialog is displayed.

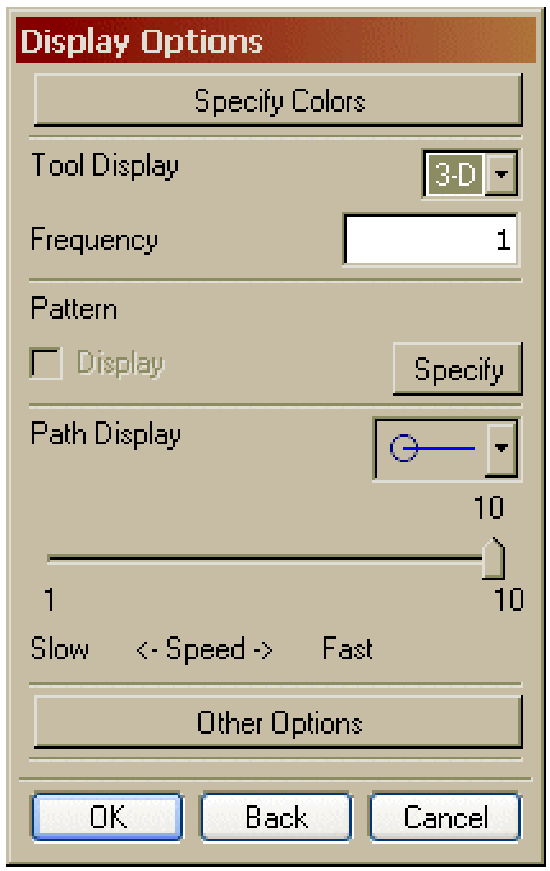


This dialog allows the input of basic global parameters. These global parameters will be active throughout the operation, unless changed in an individual sub-operation.

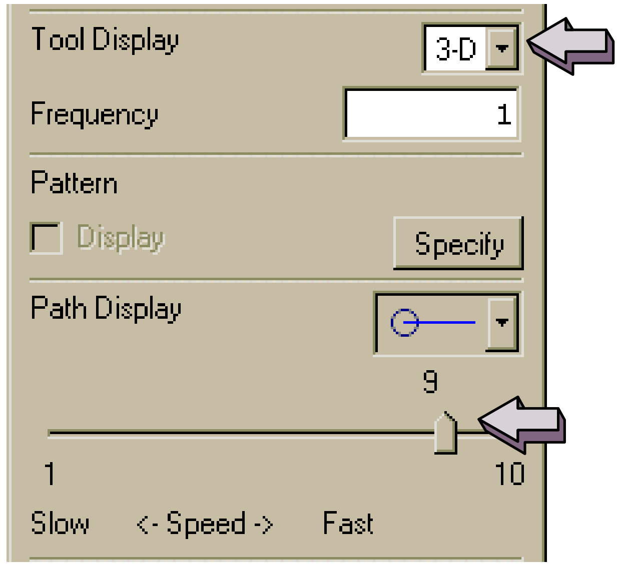
❑ Choose the **Display Options** button.



The Display Options dialog is displayed.

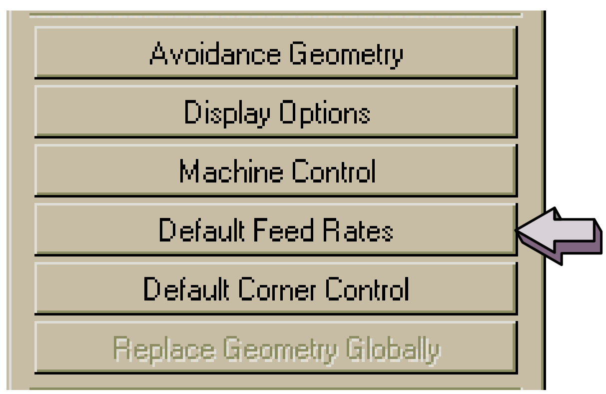


❑ Change the Tool Display to **3-D** and the Path Display Speed to **9**.

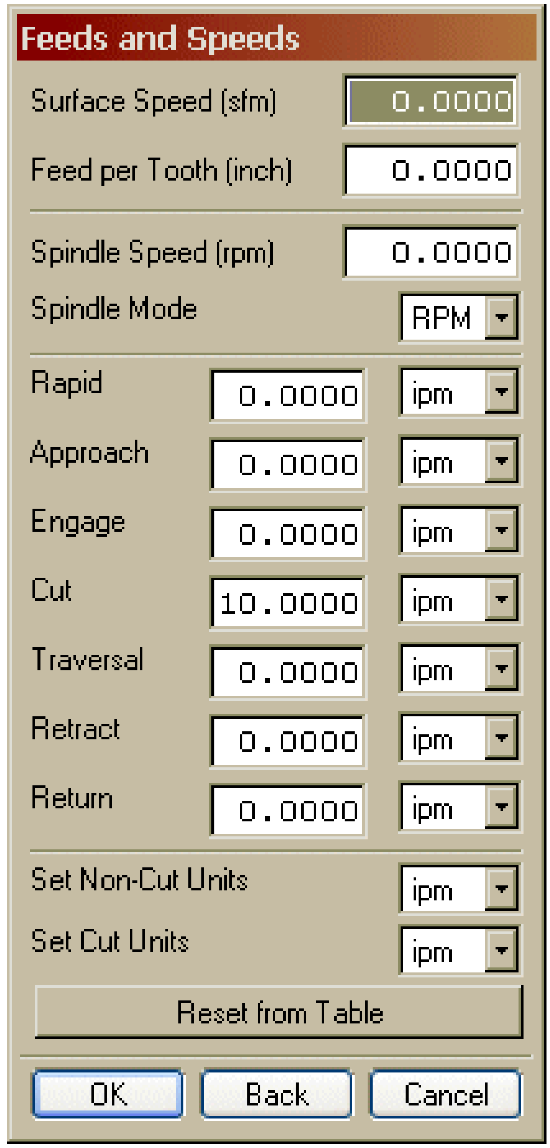


❑ Choose **OK**.

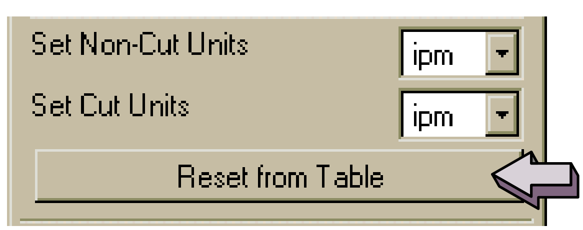
❑ Choose the **Default Feed Rates** button.



The Feeds and Speeds dialog is displayed.



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

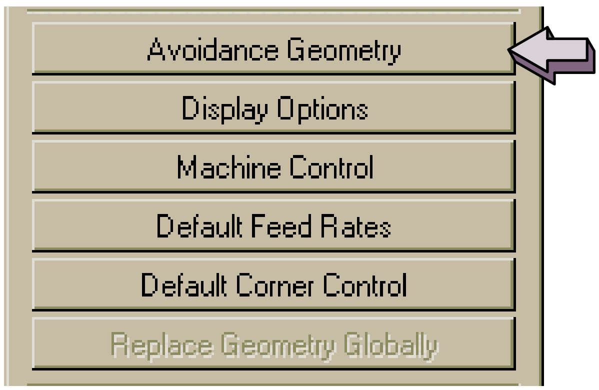


Based on the tool material, part material and number of cutter flutes, the feeds and speeds will be recalculated and reset.

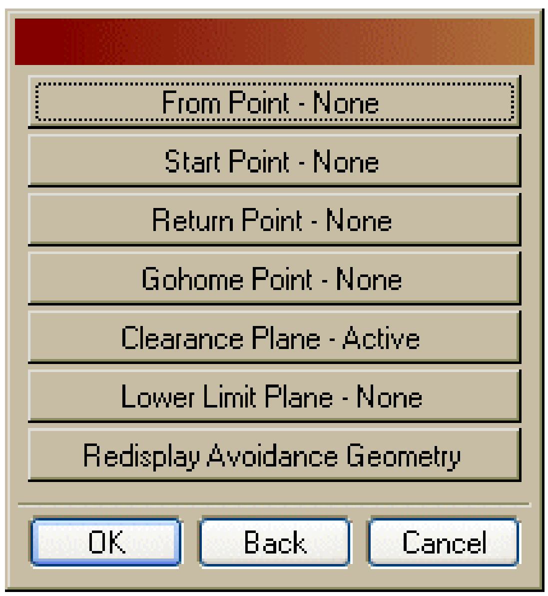
- ❑ Choose **OK.**

Finally, you will need to create a Clearance Plane for tool to part clearance. You will create a Clearance Plane .500 above the top of the part.

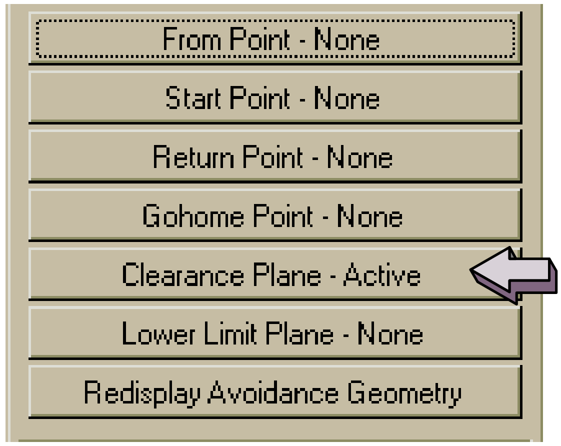
- ❑ Choose the Avoidance Geometry button.



The Avoidance Geometry dialog is displayed.

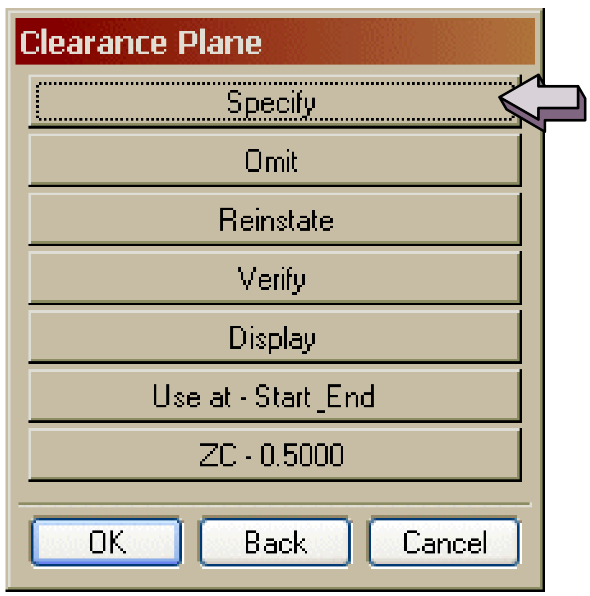


❑ Choose the **Clearance Plane** button.



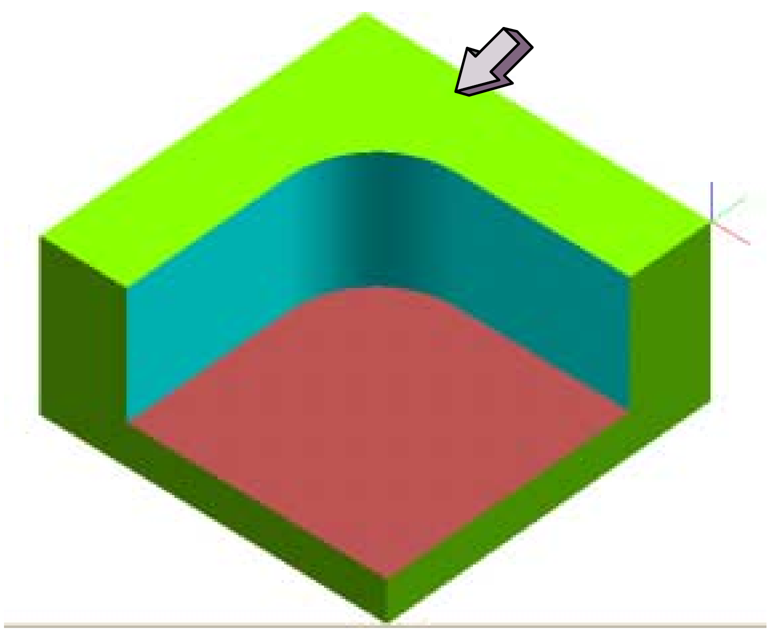
The Clearance Plane dialog is displayed.

❑ Choose **Specify**.

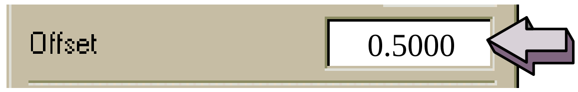


The Plane Constructor dialog is displayed.

❑ Choose the top surface of the part as shown.



❑ Key in **.500** in the Offset field.



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

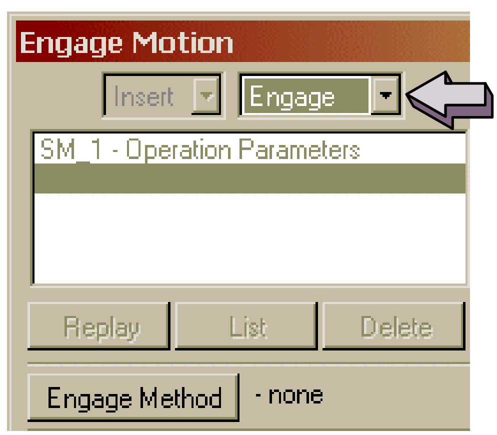
The global parameters are now set. You are now ready to begin the Sequential Milling process.

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 subsequentially 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 the dialog where the actual programming process takes place.

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

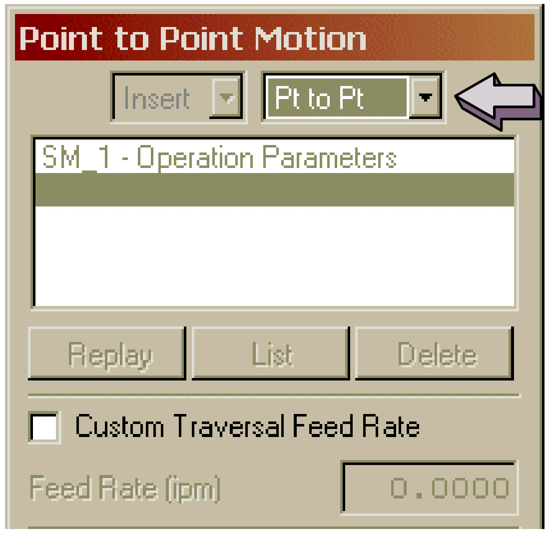
The Engage Motion sub-operation dialog is displayed. By default, the sub-operation dialog is set to **Engage**.



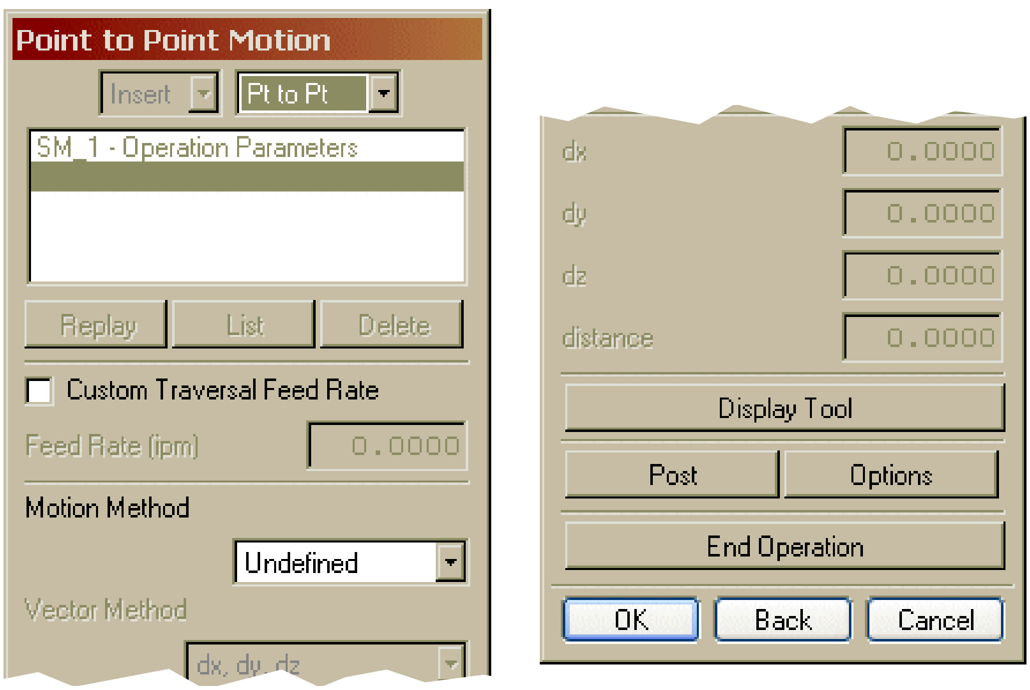
In order for the Sequential Milling processor to properly determine the tool's current location for Near Side/Far Side, it is usually beneficial to establish a **Pt to Pt** motion as the first sub-operation.

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

2

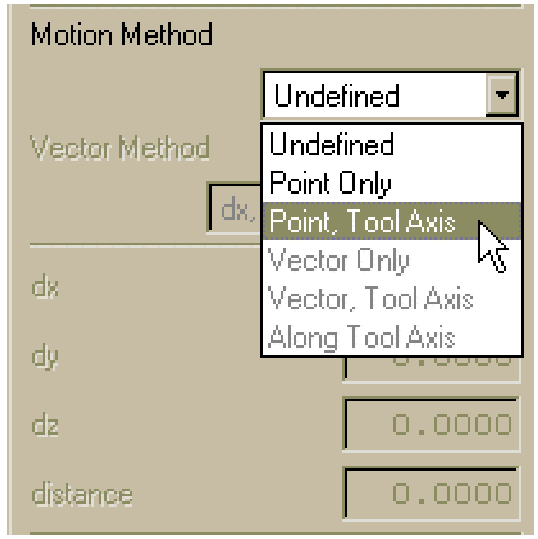


The corresponding dialog changes to match **Point to Point** motion.

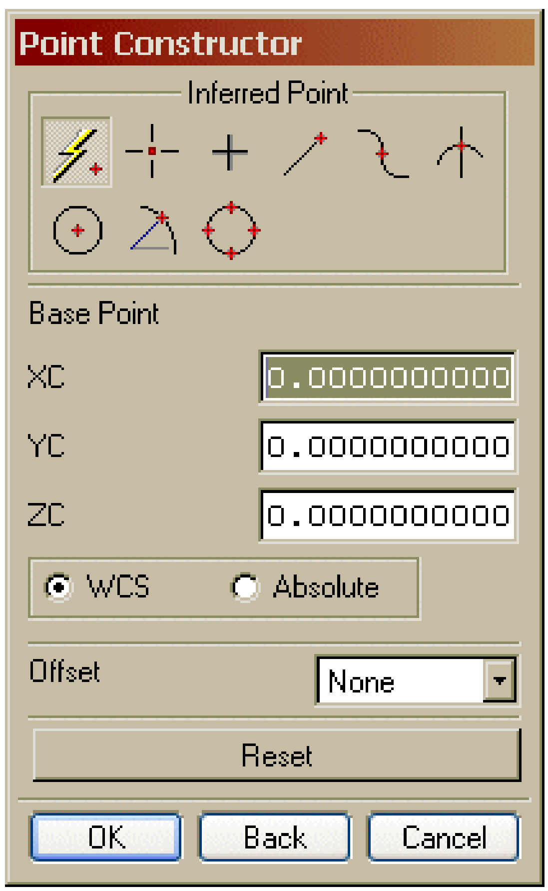


You will now establish the tool position, specifying both the position of the tool and the tool axis.

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



The Point Constructor dialog is displayed.

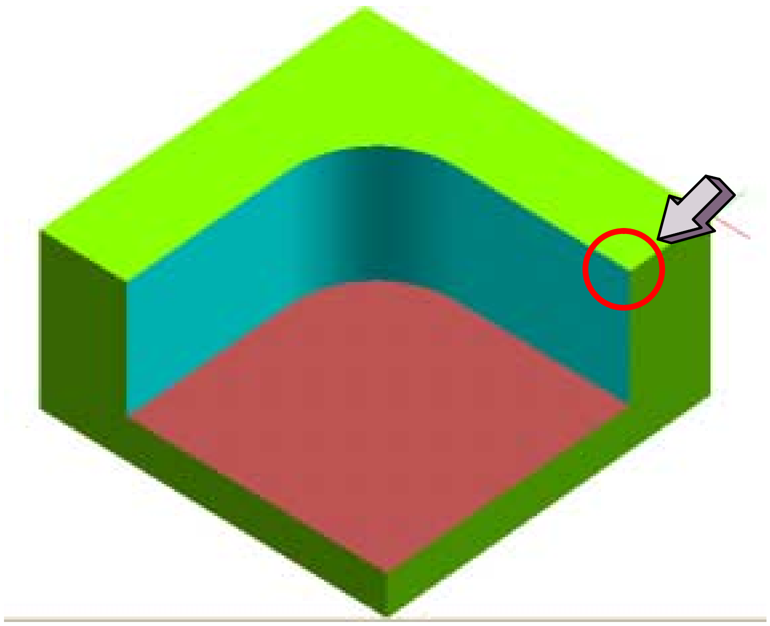


2

❑ 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.

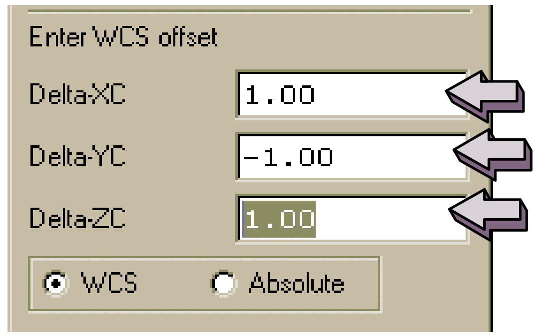
❑ Choose the corner of the ledge, as shown.



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

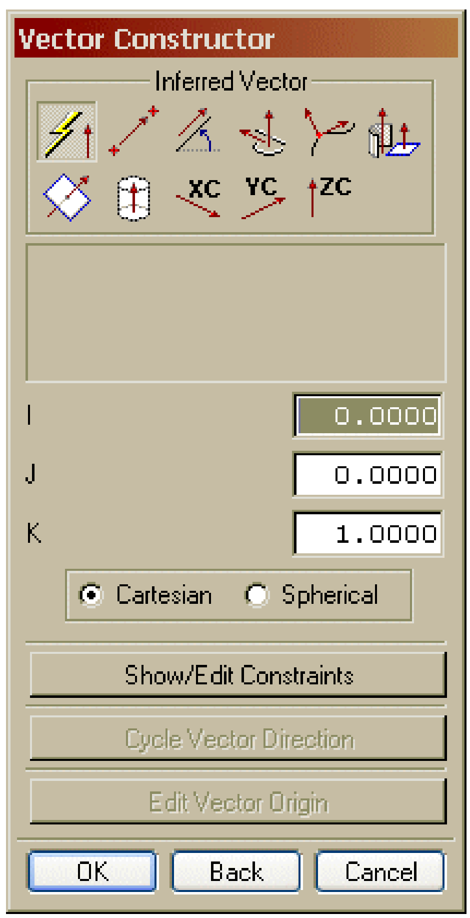
❑ Key in the following values:

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.

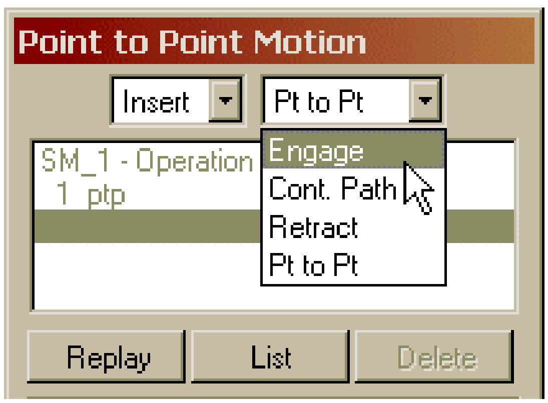
❑ Choose **OK**.

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

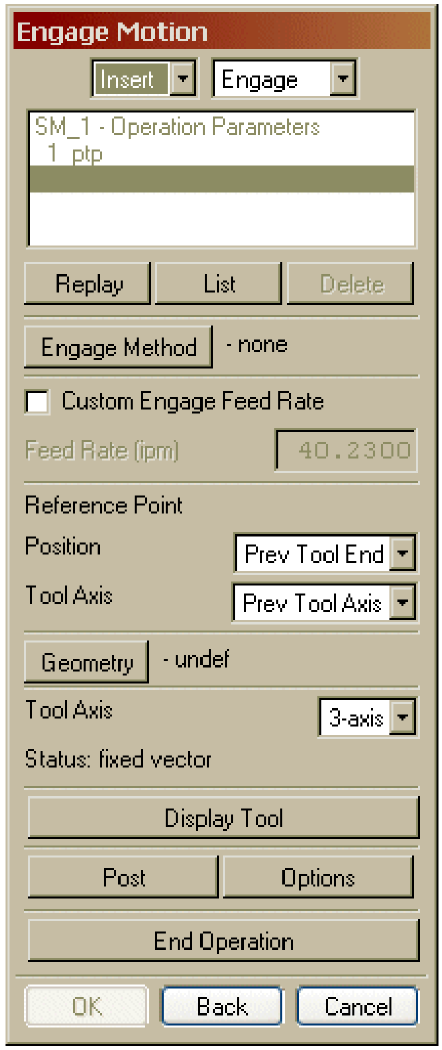
❑ Choose **OK**.

You will now define the Engage component.

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



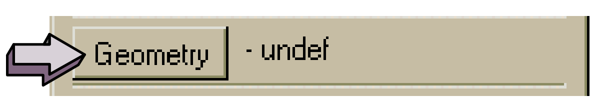
The Engage Motion dialog is displayed.



The Engage dialog requires Drive, Part and Check geometry to function properly. 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.



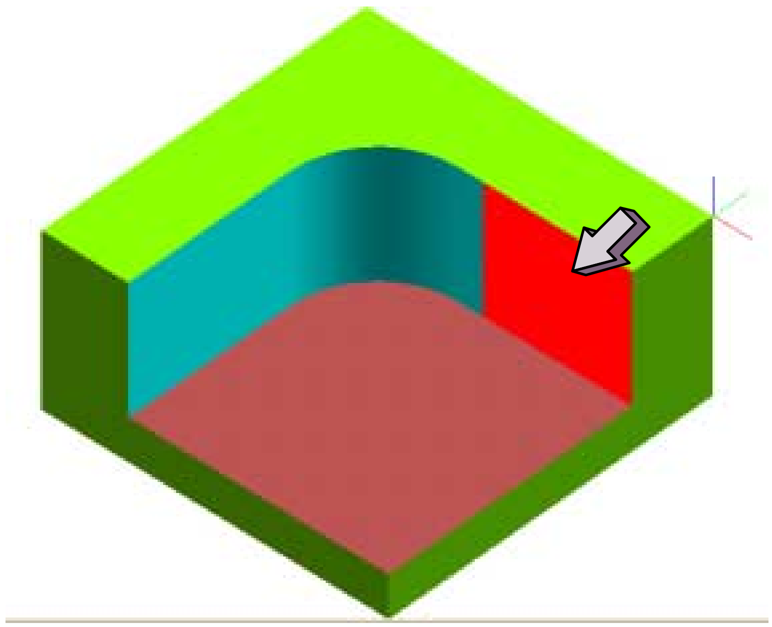
2

The Engage Geometry dialog is displayed.

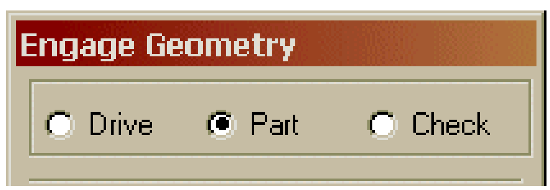
The defaults are set to **Drive** geometry, the Type is **Face** and the Stopping Position is **Near Side**.

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

❑ Choose the face as shown below.

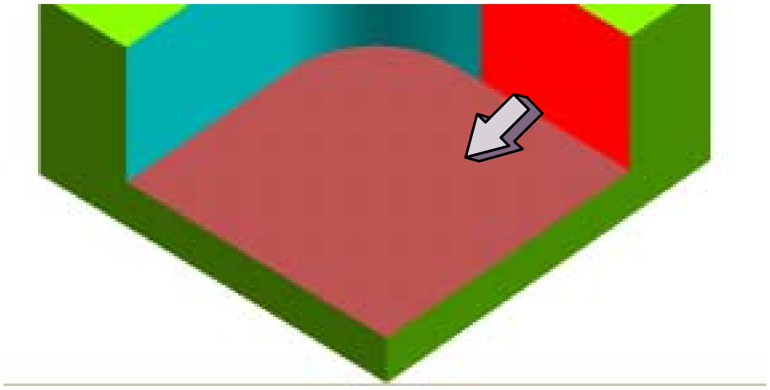


The geometry selection on the dialog advances to Part geometry.

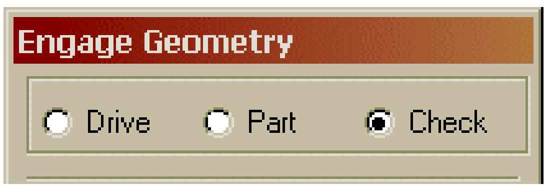


You will now select the **Part** geometry.

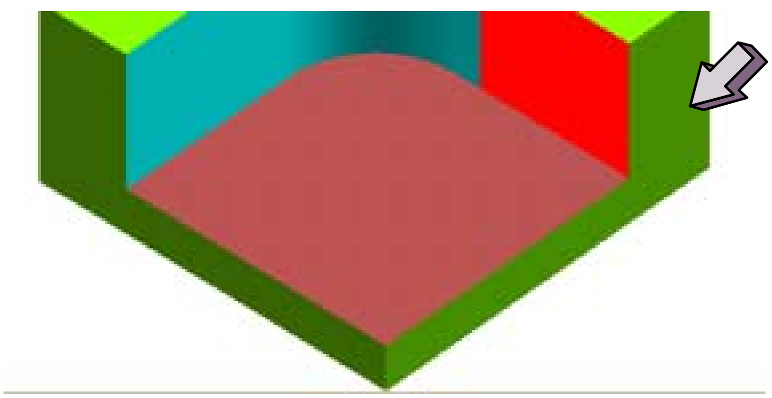
❑ Choose the bottom of the pocket as the **Part** geometry.



The geometry selection on the dialog advances to Check geometry.



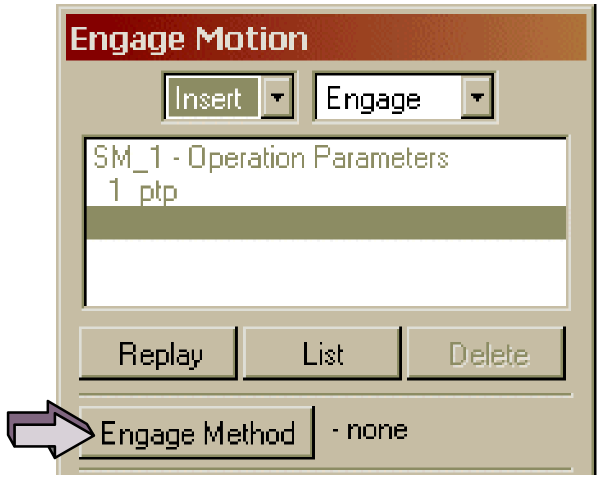
❑ Choose the face, as shown below, as the **Check** geometry.



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.

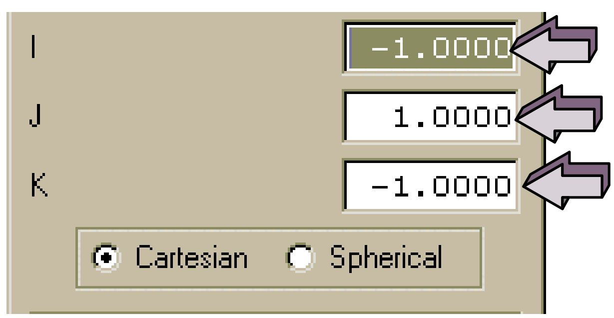
2

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

The Vector Constructor dialog is displayed.

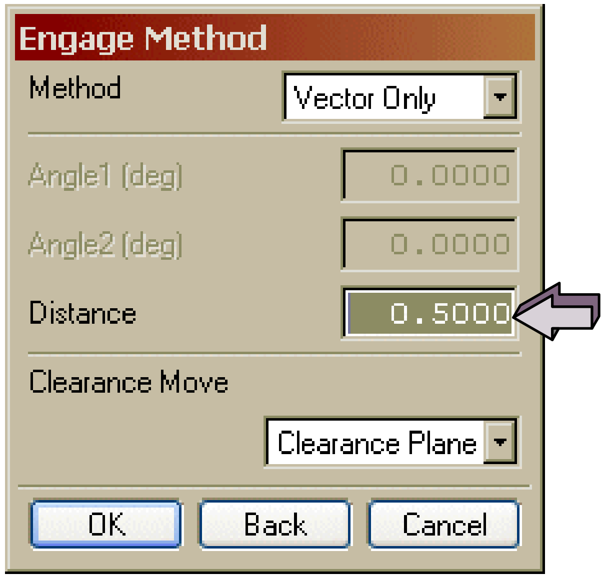
❑ Key in the following values:

I = –1.000
J = 1.000
K = –1.000

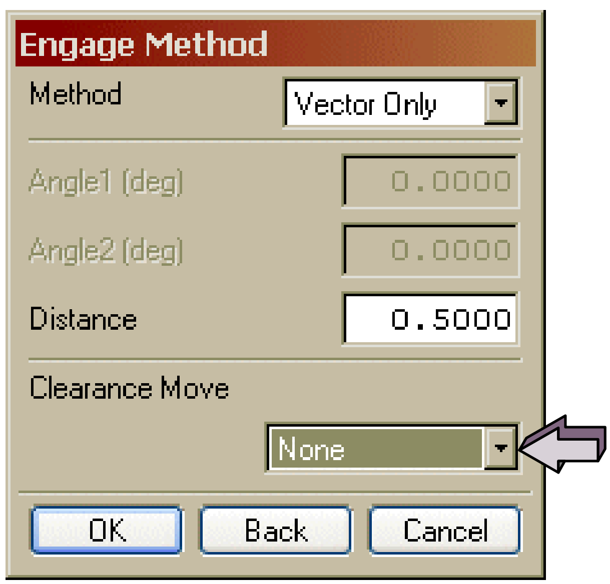


❑ Choose **OK**.

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

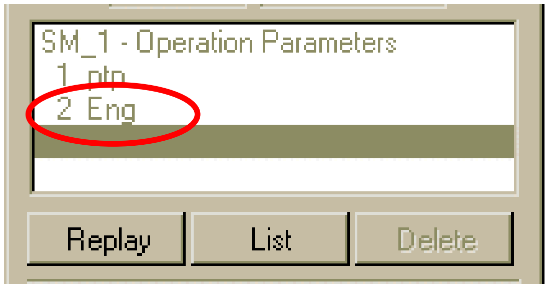


❑ Change the Clearance Move to **None**.



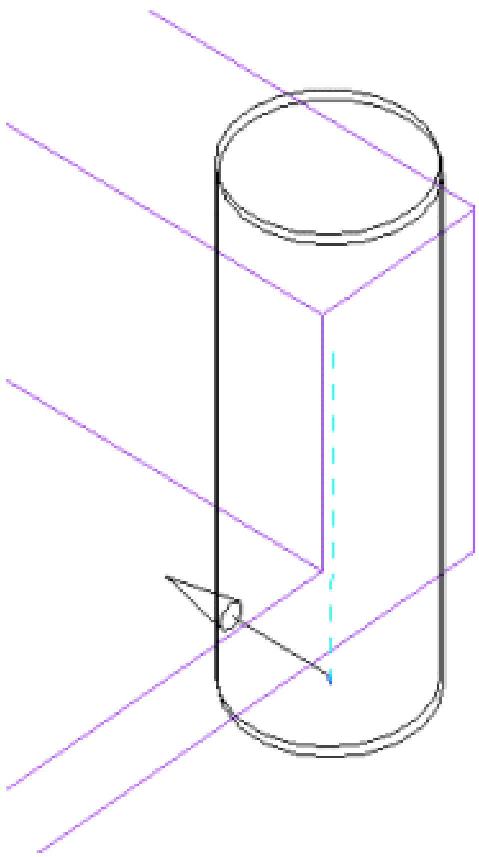
❑ Choose **OK** twice.

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.



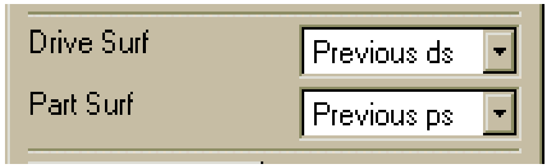
You will now create a Continuous Path Motion sub-operation.

Notice the arrow that is displayed at the bottom of the tool.



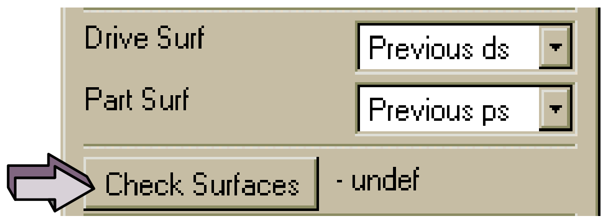
The arrow indicates the intended direction of the next cut. In this case the direction is correct. If the arrow was pointed in another direction, it would have been necessary to change direction by using the **Direction** option.

Notice the defaults for Drive Surf and Part Surf.

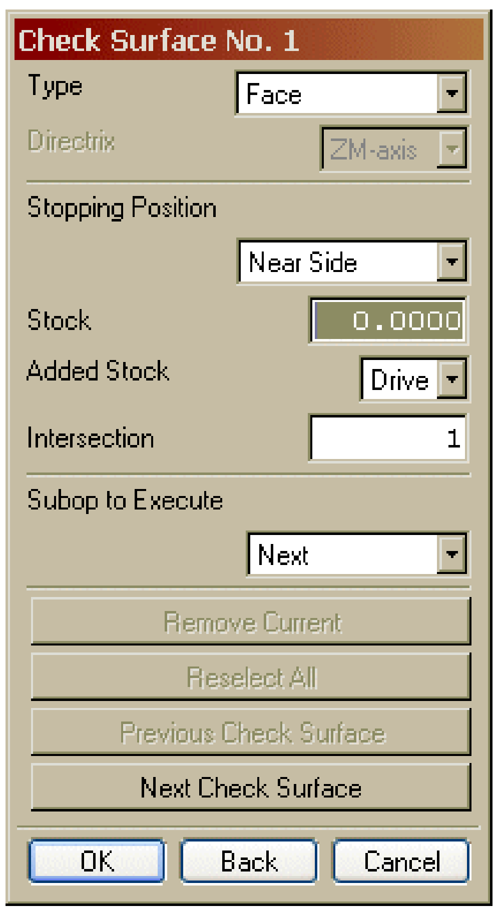


The Drive Surf(ace) is set to the Previous Drive surface (ds). The Part Surf(ace) is set to the Previous Part surface (ps). This is correct. However, it will be necessary to set the Check surface.

❑ Choose the **Check Surfaces** button.



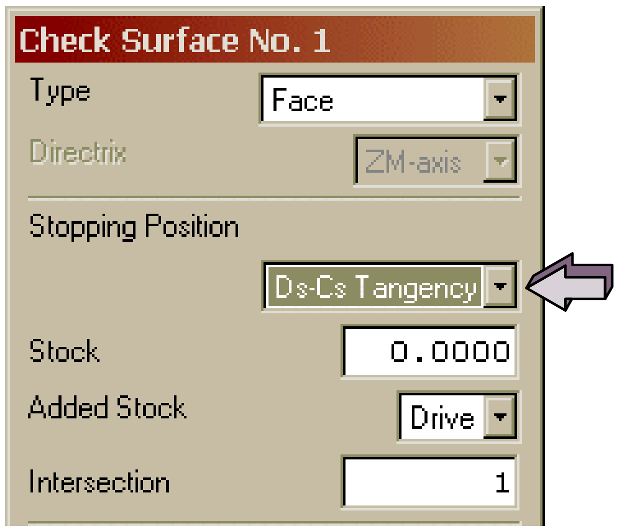
The Check Surfaces No. 1 dialog is displayed.



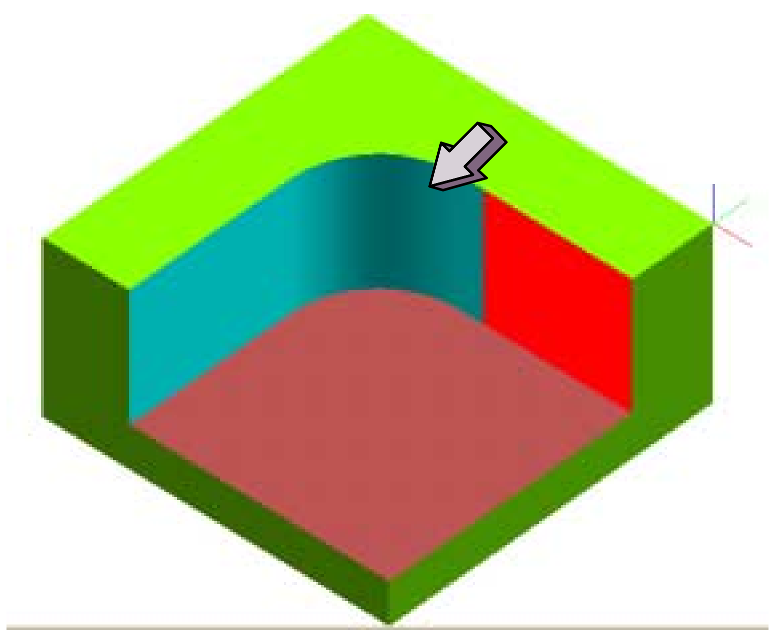
Sequential Mill is now ready for you to select the first (in this case, the only) Check surface. As soon as the surface is selected, the dialog advances to Check Surface No. 2. Therefore it is important to specify any changes to the dialog before the surface is selected.

Note that the current Drive surface is tangent to the next surface that the tool will drive to. Thus, a stopping position of Near Side is incorrect. You will want to 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, therefore skip the option to choose another.

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

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

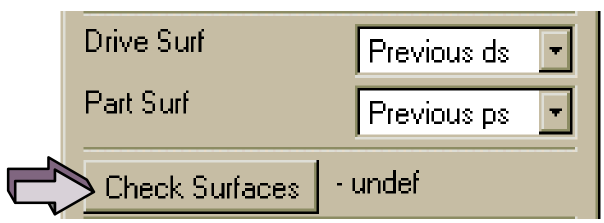
The third sub-operation, **3 cpm** has been created. You will now create another CPM (Continuous Path Motion) sub-operation.

Sequential Mill has automatically forwarded the Drive surface to the previous Check surface, which is correct in this case. It has also kept the previous Part surface as the new Part surface, which is also correct.

The Direction of Motion Vector setting is correct.

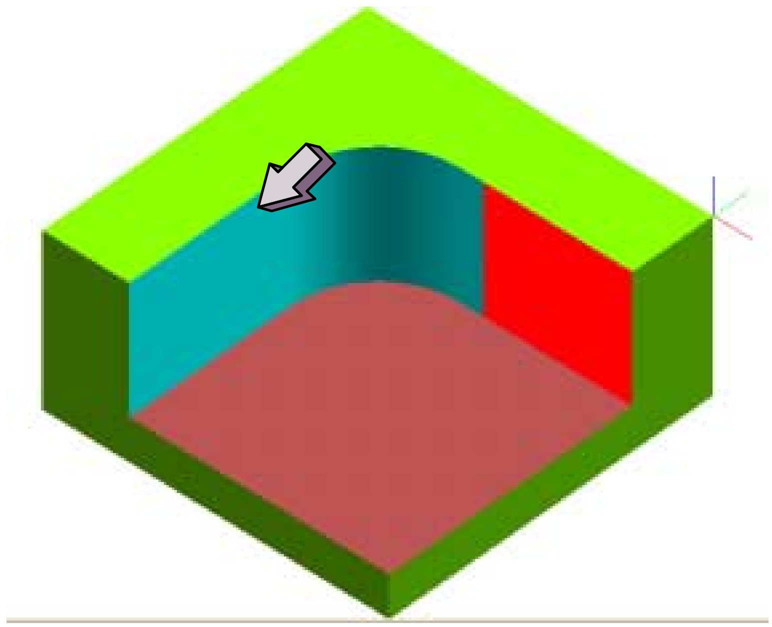
You need to choose the new Check surface.

❑ Choose the **Check Surfaces** button.



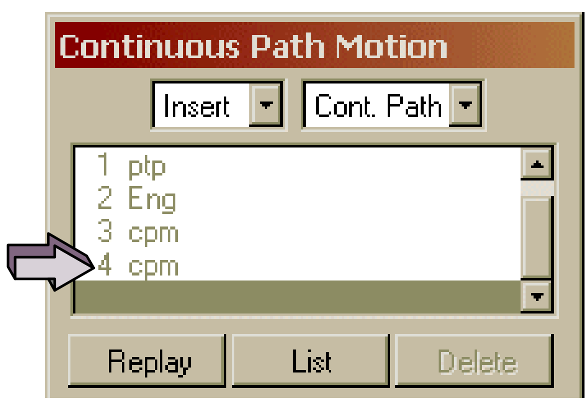
The object type of face is correct as well as the Stopping Position of Ds-Cs Tangency.

- ❑ Choose the face as shown.



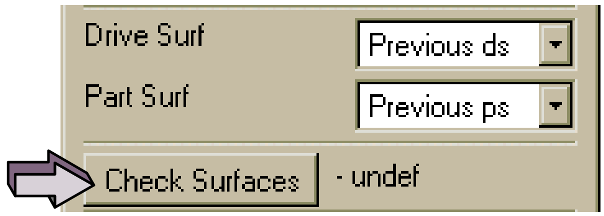
- ❑ Choose **OK** in the Check surface dialog.
- ❑ Choose **OK** in the Continuous Path Motion dialog.

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.

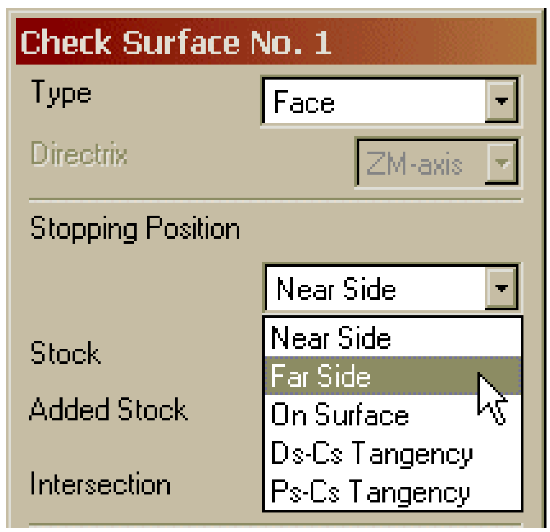
❑ Choose the Check Surfaces button.



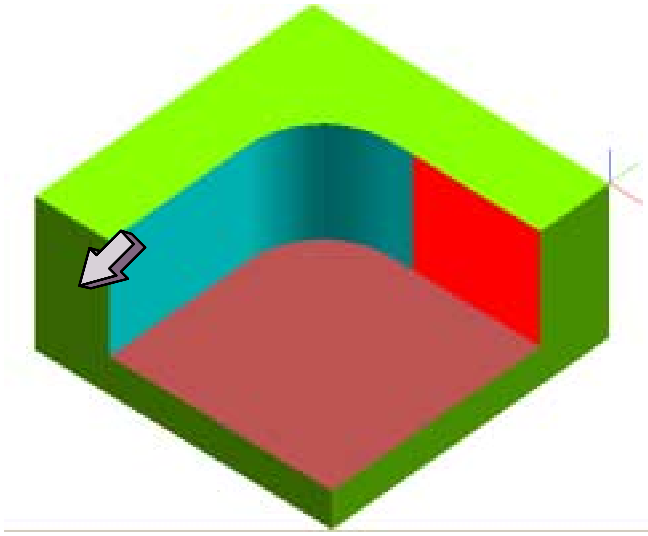
This time, the Stopping Position of Ds-Cs Tangency is incorrect. You will change it to Far Side, so that the tool is completely off the Part surface, prior to stopping.

2

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



❑ Choose the surface as shown below.



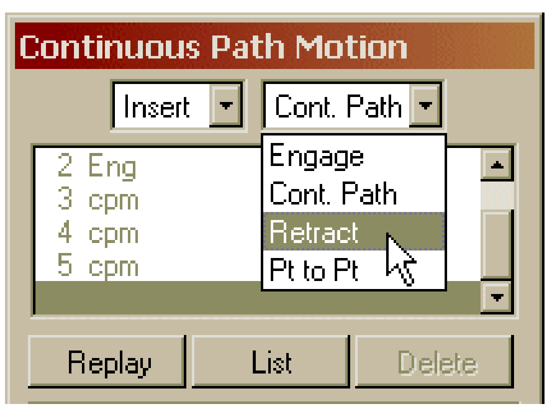
❑ Choose **OK** in the Check Surface dialog.

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

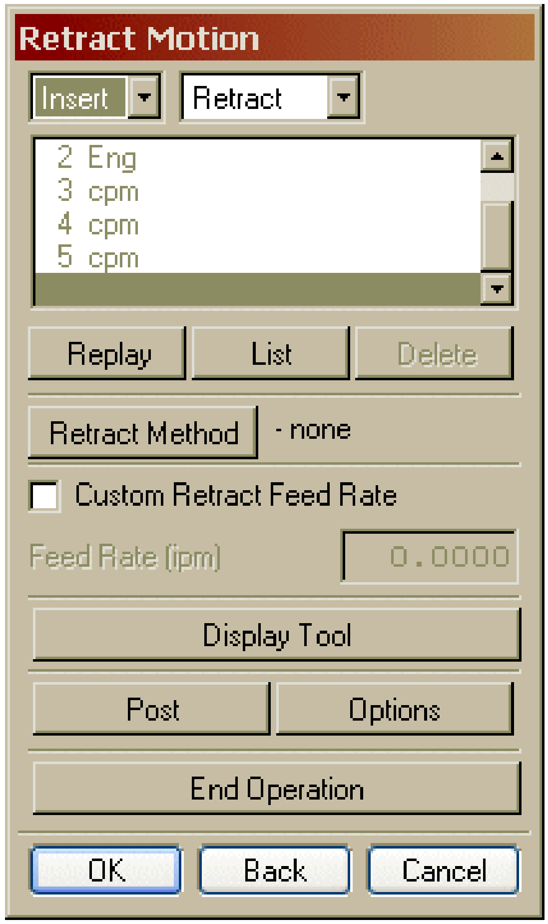
The sub-operation, **5 cpm**, is now placed in the dialog list.

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

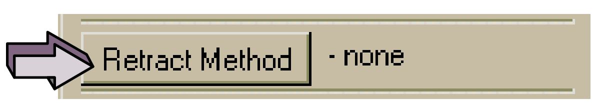
❑ Change the sub-operation to Retract.



The Retract Motion dialog is displayed.

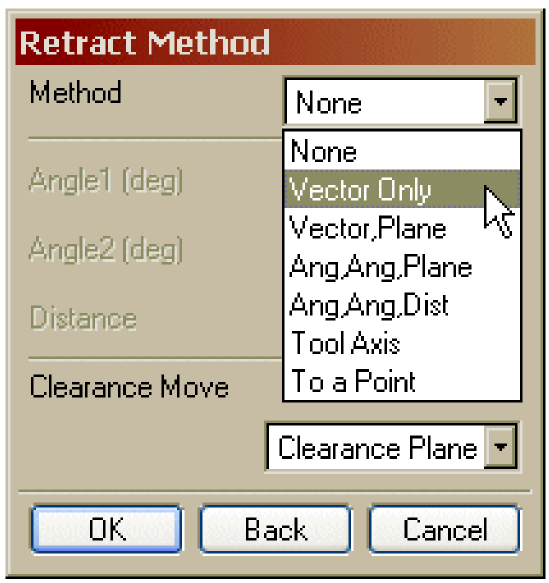


❑ Choose the **Retract Method** button.



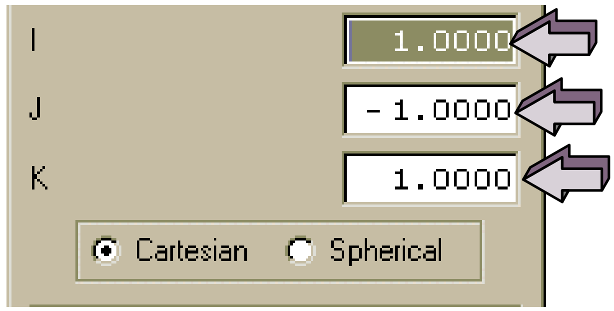
The Retract Method dialog is displayed.

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



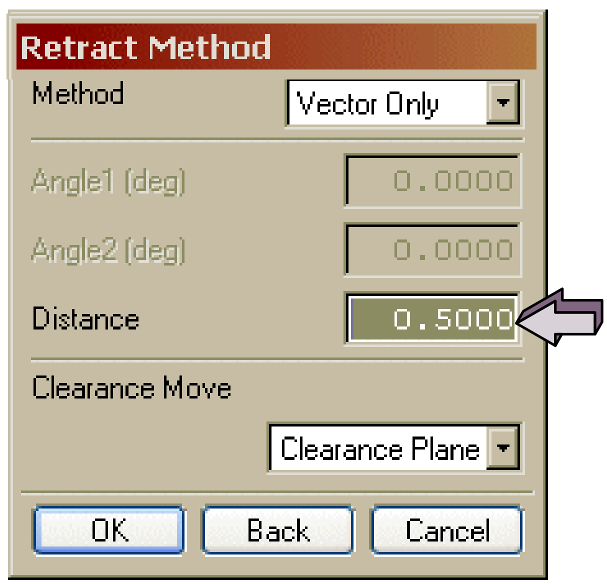
The Vector Constructor dialog is displayed.

- ❑ Key in the following values:
  I = 1.000
  J = −1.000
  K = 1.000



- ❑ Choose **OK**.

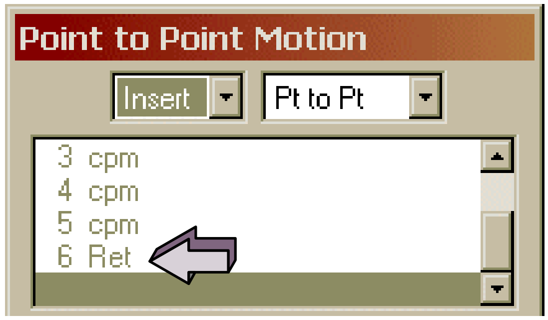
- ❑ Key in **0.500** in the **Distance** field of the Retract Method dialog.



- ❑ Choose **OK.**

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

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



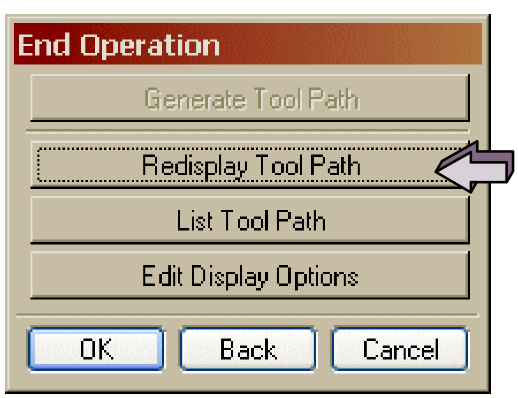
The tool retracts to the clearance plane. Programming of the wall is complete. To complete the process, use the End Operation button.

❑ Choose the **End Operation** button.

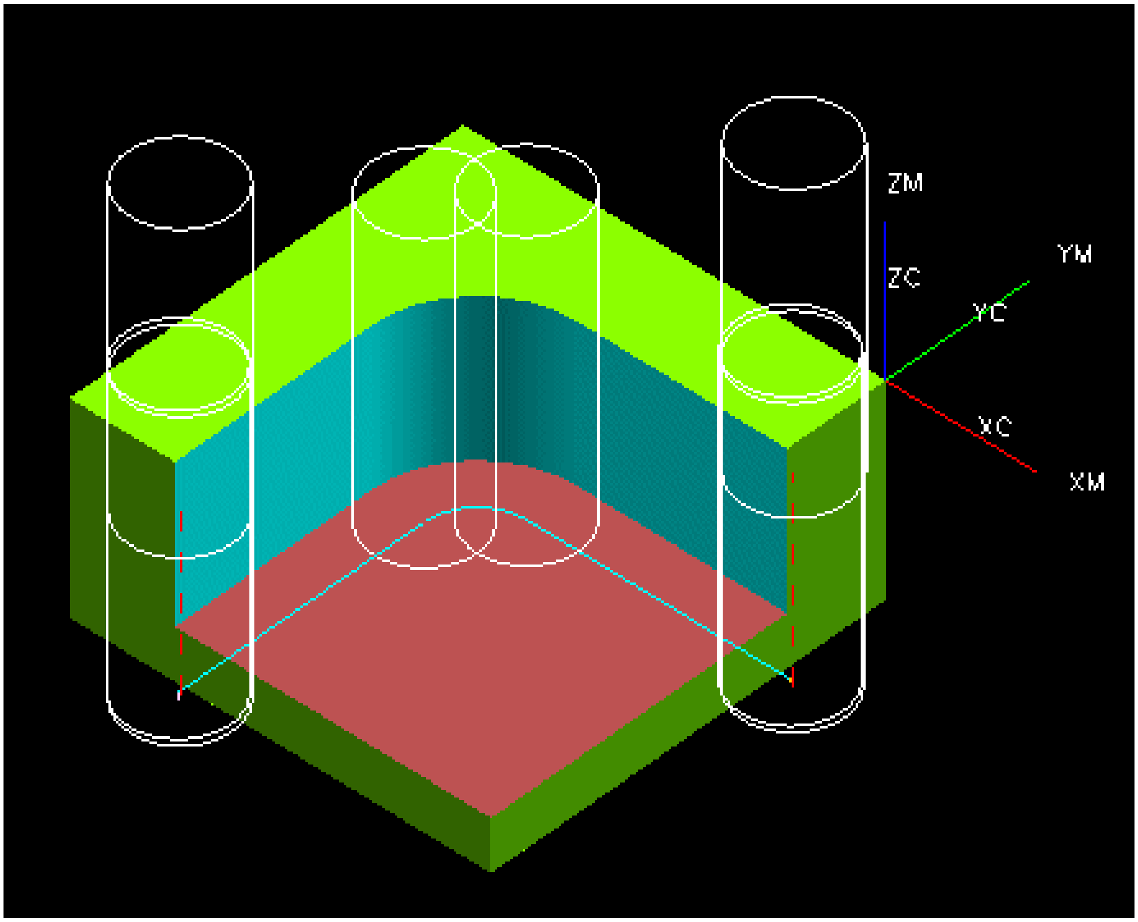
To observe the tool path again, refresh the screen and choose the Redisplay Tool Path button.

❑ In the graphics window, use **MB3, Refresh**.

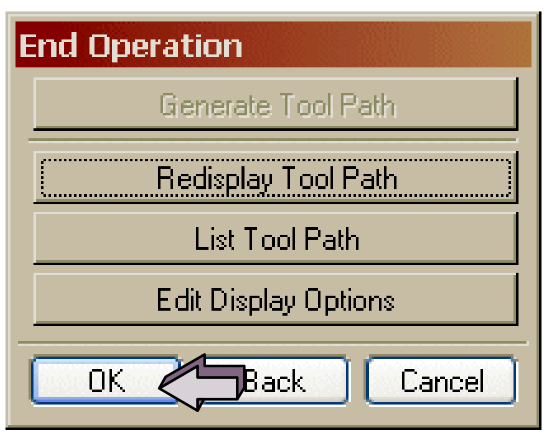
❑ Choose Redisplay Tool Path from the End Operation dialog.



The tool path is displayed.



❑ Choose **OK** from the End Operation dialog.



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

This completes the activity.

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 Sequential Mill 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).

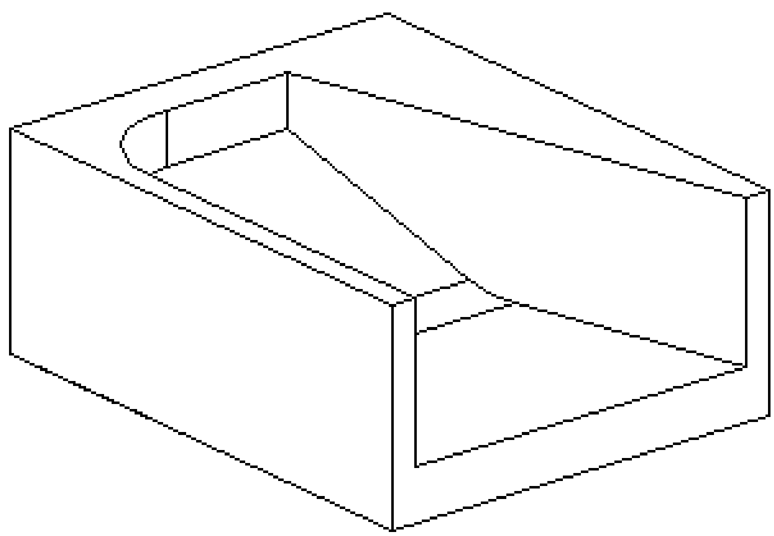
When you select either Drive surface or Part surface from the Continuous Path Motion dialog, you have the following options of **Other Surface**, **Previous ds**, **Previous ps** and **Previous cs**.

Activity 2—2: 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 respecify 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.prt** and rename it to *****_mam_sq_3.prt.**



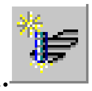
❑ Choose **Application → Manufacturing**.

❑ 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:

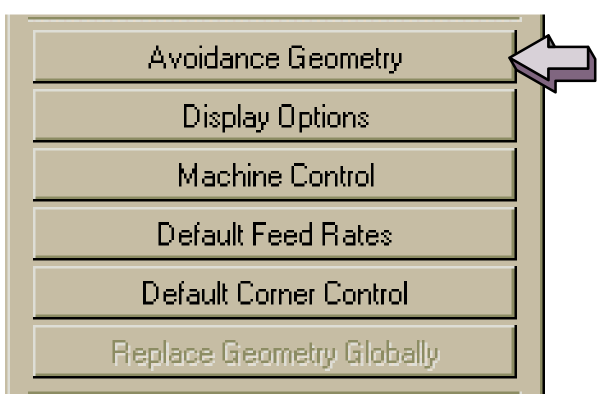
Program:	**NC_PROGRAM**
Use Geometry:	**NONE**
Use Tool:	**EM_.75_.125**
Use Method:	**MILL**

- ❑ Enter the operation name **fin-poc-walls** into the Name field.
- ❑ Choose **OK**.

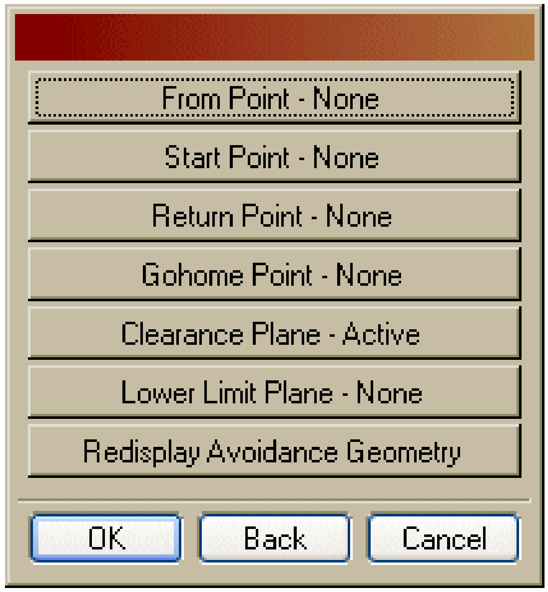
The Sequential Mill dialog is displayed.

Step 3 Define a Clearance Plane and tool necessary for machining the part.

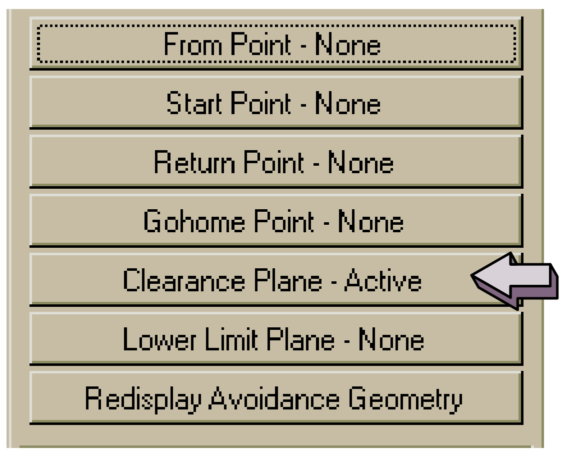
- ❑ Choose the Avoidance Geometry button.



The Mill Avoidance Control dialog is displayed.

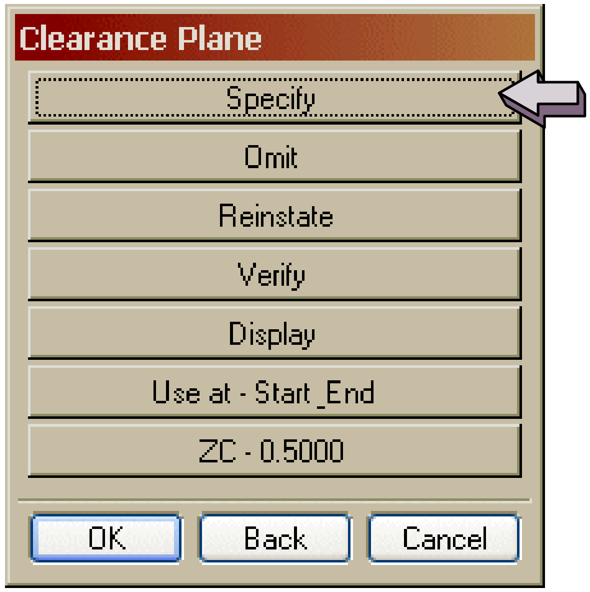


❑ Choose the **Clearance Plane** button.

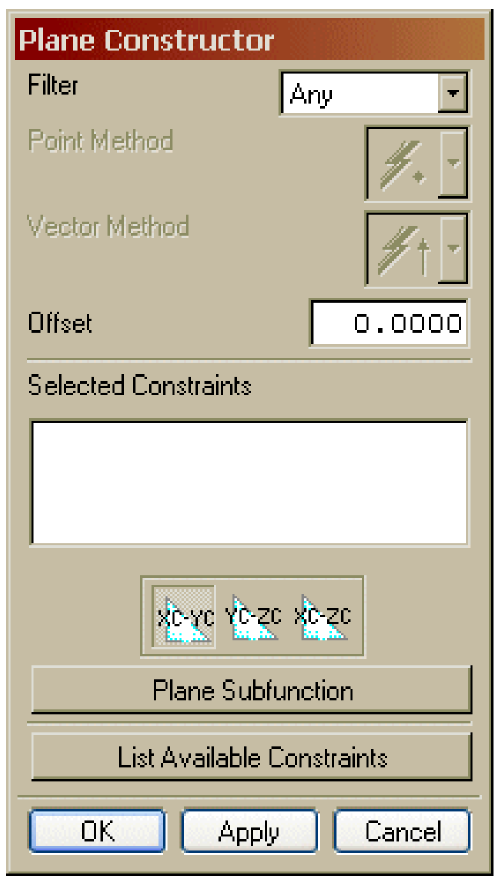


The Clearance Plane dialog is displayed.

❑ Choose **Specify**.

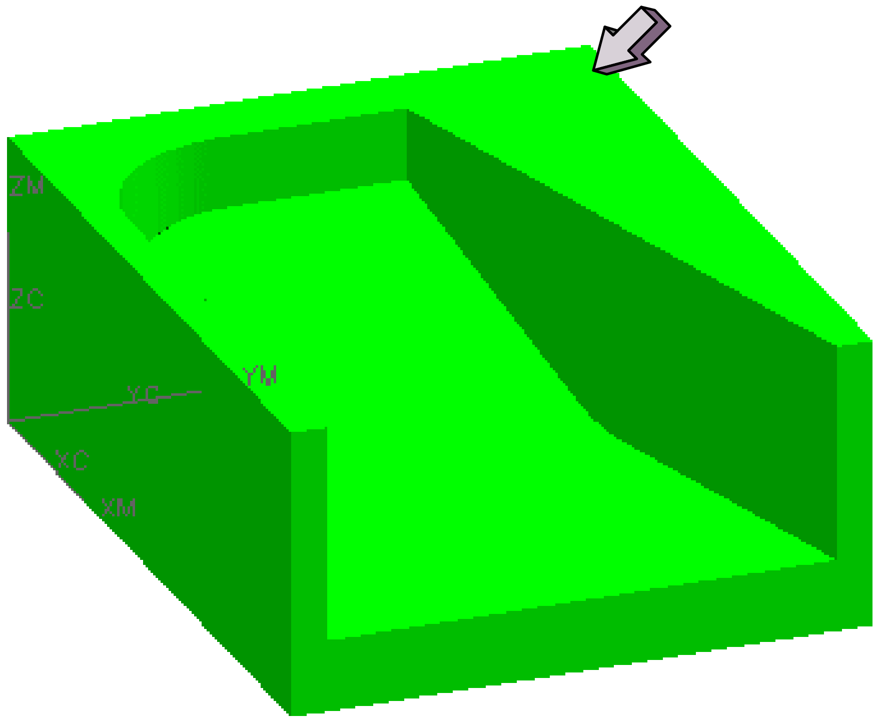


The Plane Constructor dialog is displayed.

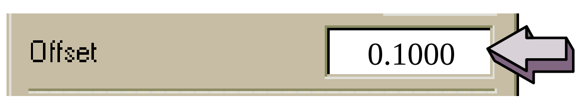


2

❑ Choose the top surface of the part as shown.



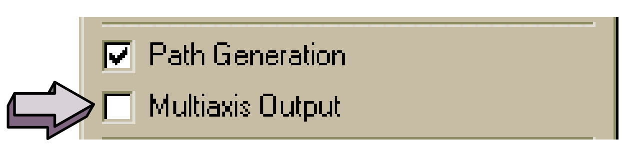
❑ Key in **.100** in the Offset field.



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

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

❑ On the Sequential Mill dialog, make sure that the Multiaxis option check box is OFF (***not*** checked).



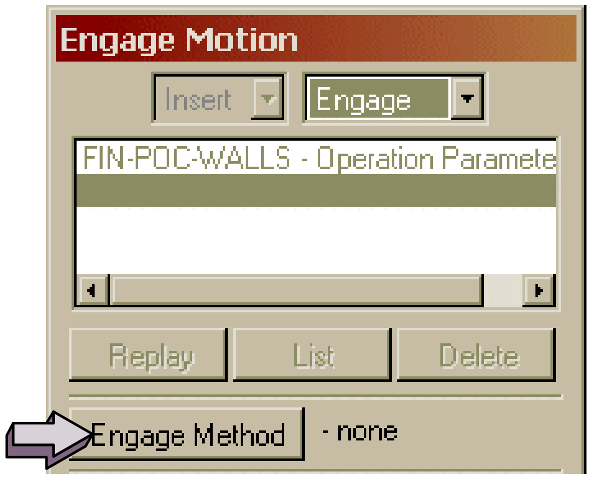
❑ Choose **OK** and continue to the Engage Motion dialog.

❑ Choose the **Display Tool** button to display the tool at its reference position.

Step 4 Specify an Engage motion.

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

❑ Choose the Engage Method button.

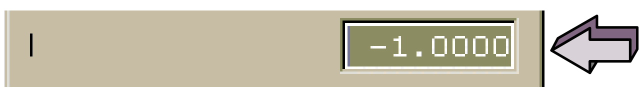


The Engage Method dialog is displayed.

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

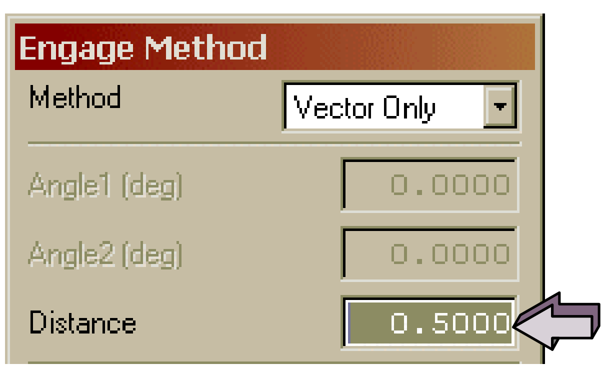
The Vector Constructor dialog is displayed.

❑ Key in the -1.000 value for I.



❑ Choose **OK.**

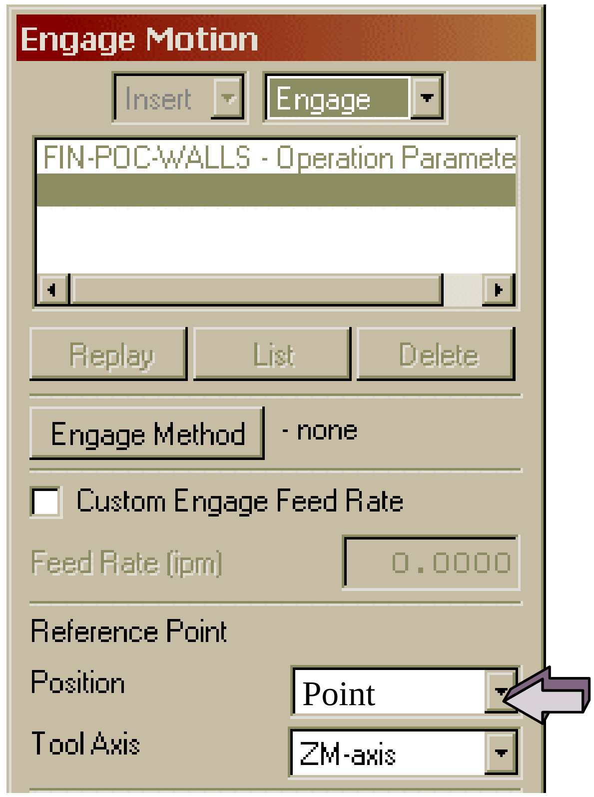
❑ Key in **0.500** in the **Distance** field of the Engage Method dialog.



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

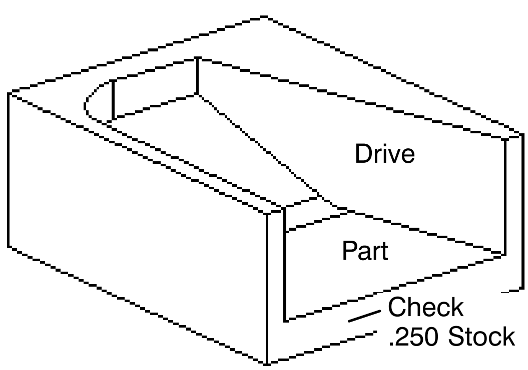
2

❑ 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 below.

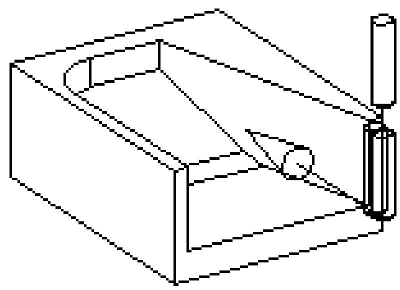


- ❑ Specify **.250 Stock** for the Check surface, prior to selecting the surface.

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

- ❑ Choose **OK**.

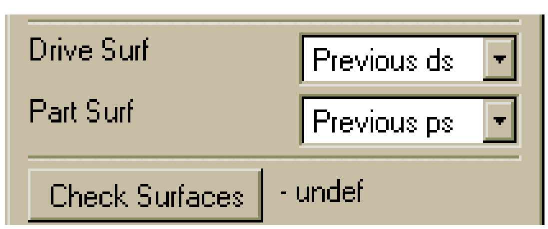
The tool moves from the Clearance Plane to the position just specified.



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 5 Specify Continuous Path motion.

Note that the Sequential Mill processor, by settings, expects the next Drive surface to be the previous Drive surface, and that the next Part surface will be the previous Part surface.



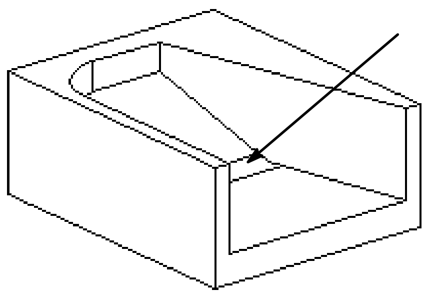
For the remainder of this activity, you will be prompted to change the Drive and Part surfaces only if the processor does not correctly select the proper surface. Each sub-operation will require that you select a new Check surface.

❑ Choose the Check Surfaces button.

❑ Change the Check Stock to **0**.

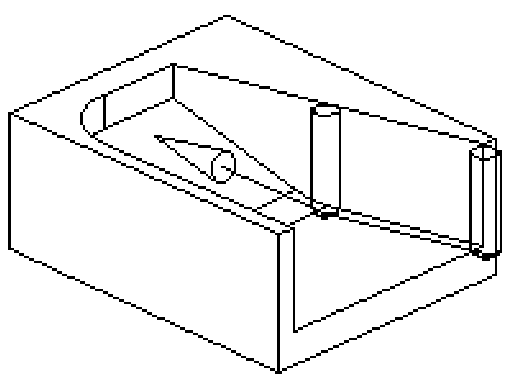
❑ Change the Check surface Stopping Position to **Ps-Cs Tangency**.

❑ Specify a new Check surface as shown below.



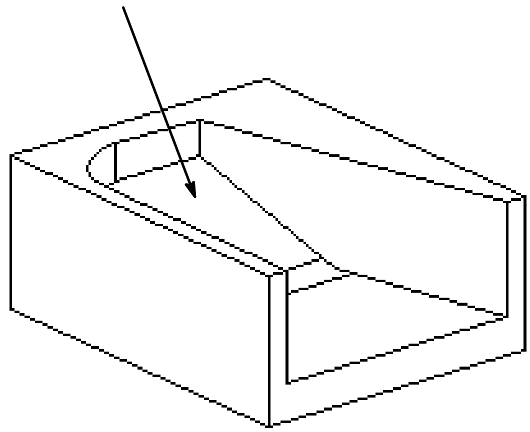
❑ Return to the Continuous Path Motion dialog and choose **OK**.

The tool moves to the new position.



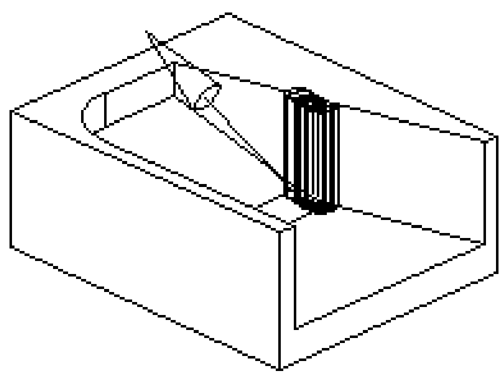
Note that the Sequential Mill processor changed the status of the Part Surface to previous Check surface.

❑ Specify a new Check surface as shown below.



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

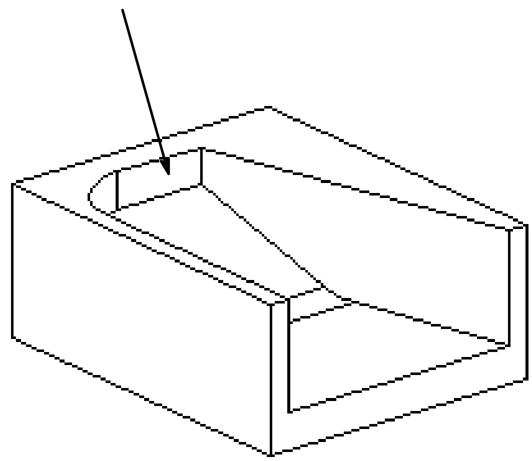
The tool moves to the new position.



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

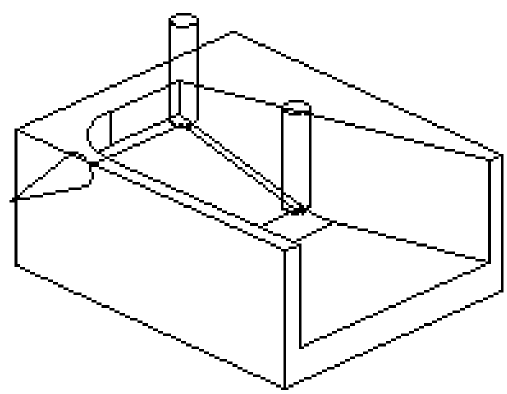
❑ Specify Check surface Stopping Position as **Near Side**.

❑ Specify a new Check surface as shown below.



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

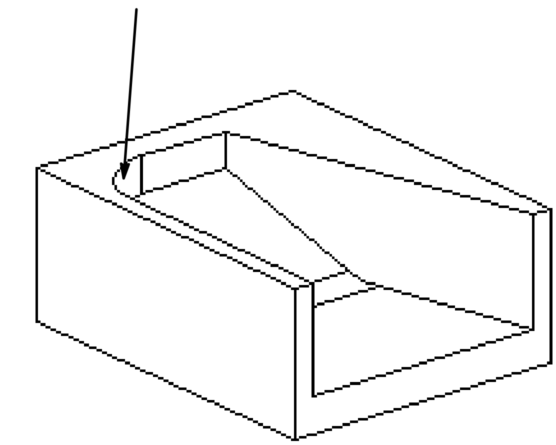
The tool moves to the new position.



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

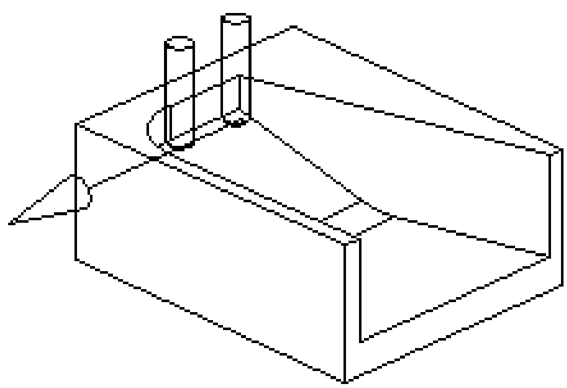
❑ Specify the Check surface Stopping Position as **Ds-Cs Tangency**.

❑ Specify a new Check surface as shown below.



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

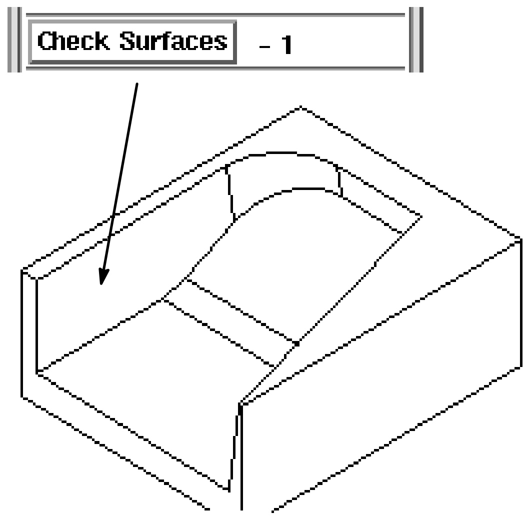
  The tool moves to the new position.



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

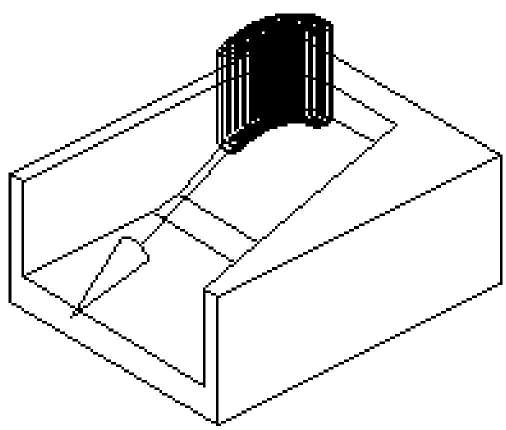
- ❑ Specify the Check surface Stopping Position as **Near Side**.

- ❑ Specify a new Check surface as shown below.

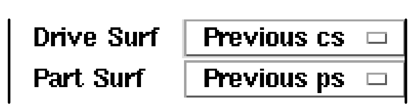


❑ Return to the Continuous Path Motion dialog and choose **OK**.

The tool moves to the new position.

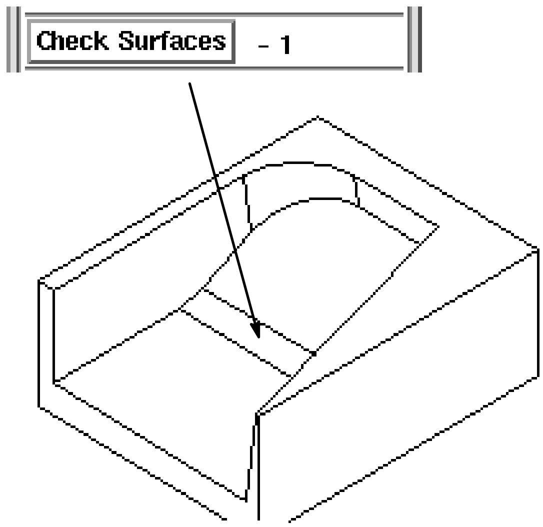


Note that the Sequential Mill processor did not change the status of the Drive or Part surfaces.



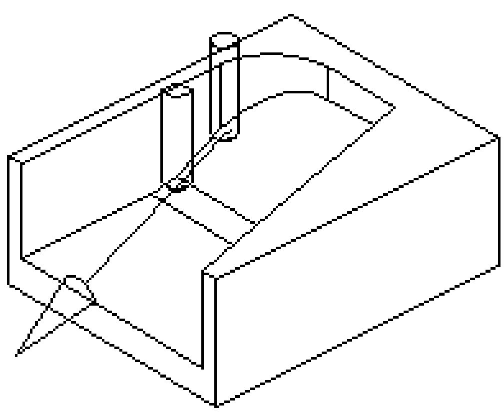
❑ Specify the Check surface Stopping Position as **Ps-Cs Tangency** .

❑ Specify a new Check surface as shown below.

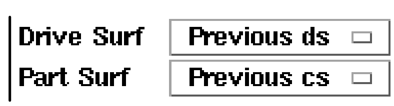


❑ Return to the Continuous Path Motion dialog and choose **OK**.

The tool moves to the new position.

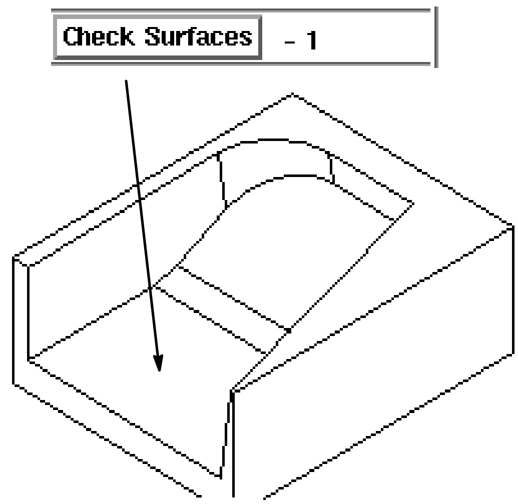


Note that the Sequential Mill processor changed the status of the Drive and Part surfaces.



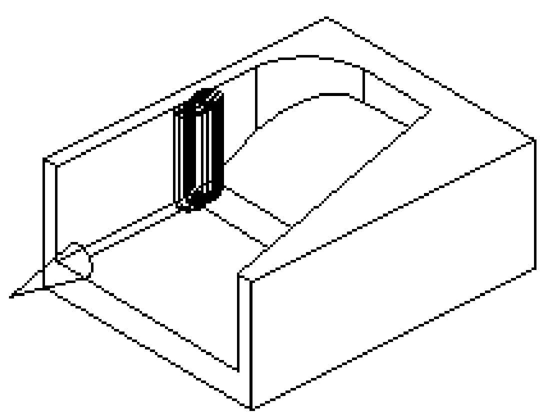
❑ Specify the Check surface Stopping Position as **Ps-Cs Tangency**.

❑ 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.

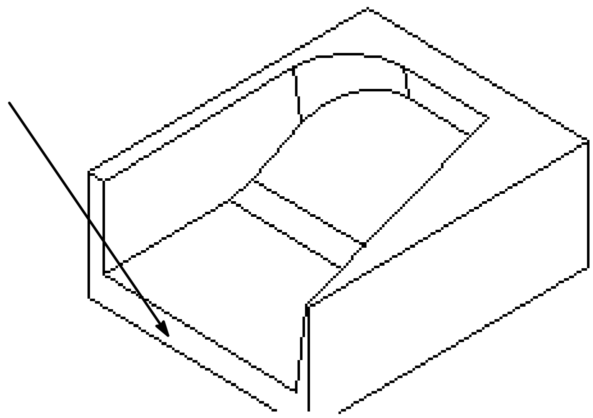


Note that the Sequential Mill processor did not change the status of the Drive or Part surfaces.

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

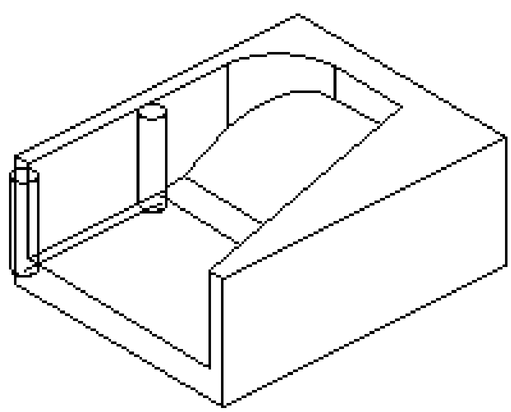
❑ Specify the Check surface Stock as **.25.**

❑ Specify a new Check surface as shown in the following illustration.



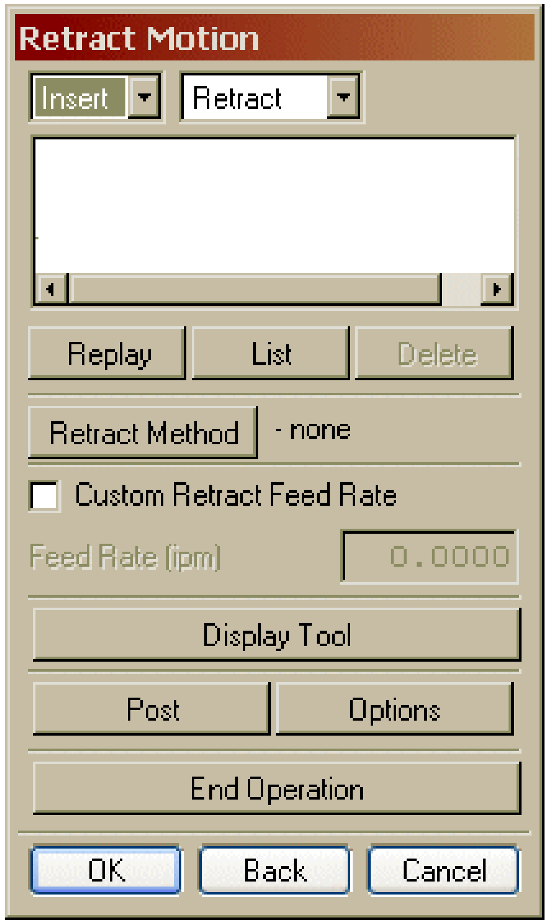
❑ Return to the Continuous Path Motion dialog and choose **OK**.

The tool moves to the new position.



❑ Change **Cont. Path** to **Retract**.

The Retract Motion dialog is displayed.



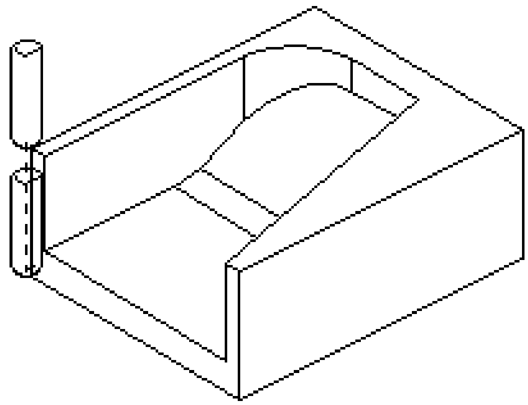
❑ Change the Retract Method to **Vector Only** → **+XC Axis**.

❑ Change the Distance to **.200**.

❑ Change the Clearance Move to **Clearance Plane**.

❑ Return to the Retract Motion dialog and choose **OK**.

The tool retracts to the Clearance Plane.



❑ Choose **End Operation** → **OK** to save the operation.

The entire tool path is displayed.

❑ Save and close the part file.

You have completed this activity as well as the lesson.

SUMMARY

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

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

Sequential Mill Intermediate
Lesson 3

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.

OBJECTIVES

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

- generate Sequential Mill tool paths using wireframe geometry

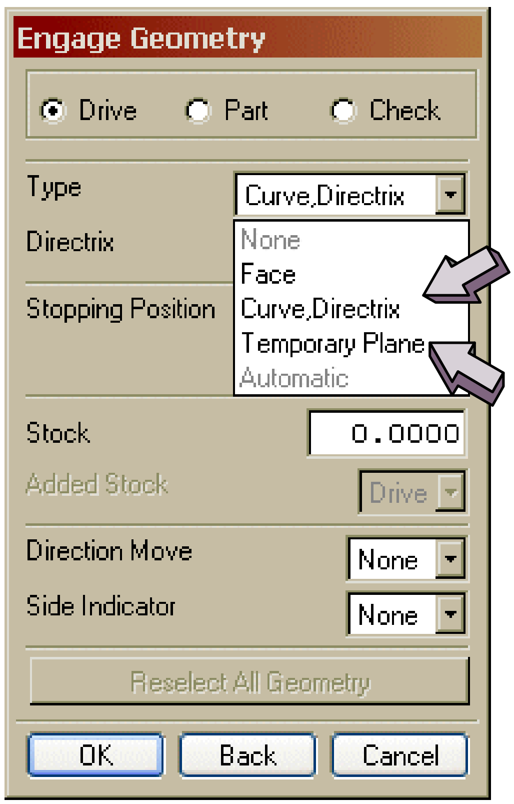
This lesson contains the following activities:

Sequential Milling using non-solid geometry

Occasionally, geometry being machined was not originally created in Unigraphics. 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:



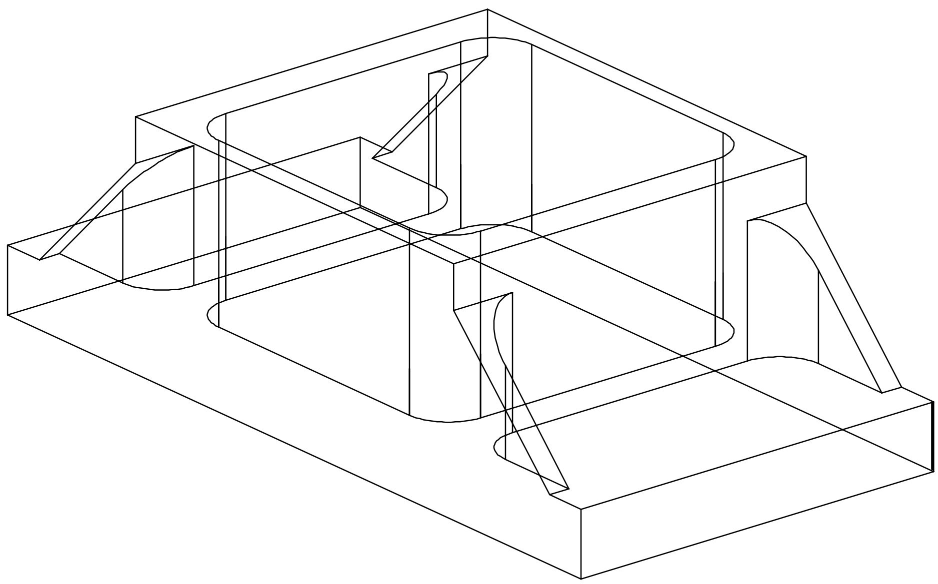
- Curve Directrix allows the selection of curves or face edges in order to drive the cutting tool; the Directrix parameter forces the calculation of a tabulated cylinder (which is normally parallel to the tool axis) to aid in positioning the tool
- Temporary Plane allows the creation of a non-permanent plane to aid in positioning the tool; the parameters of the plane are stored within Sequential Mill, but do not create geometry that is visible

Activity 3–1: Seq. Mill machining of non-solid geometry

In this activity, you will machine wireframe 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.prt**.



❑ Enter the **Manufacturing application** by choosing **Application → Manufacturing.**

Step 2 Set up the view.

When working with wireframe geometry types, it sometimes becomes difficult to determine the true depth of a particular object. When viewing the object, the object will sometimes appear to flip or reverse position suddenly. With a solid object, this can be eliminated by simply shading the part, however with wireframe geometry, this does not apply.

There are various techniques that can be applied to keep the proper view perspective.

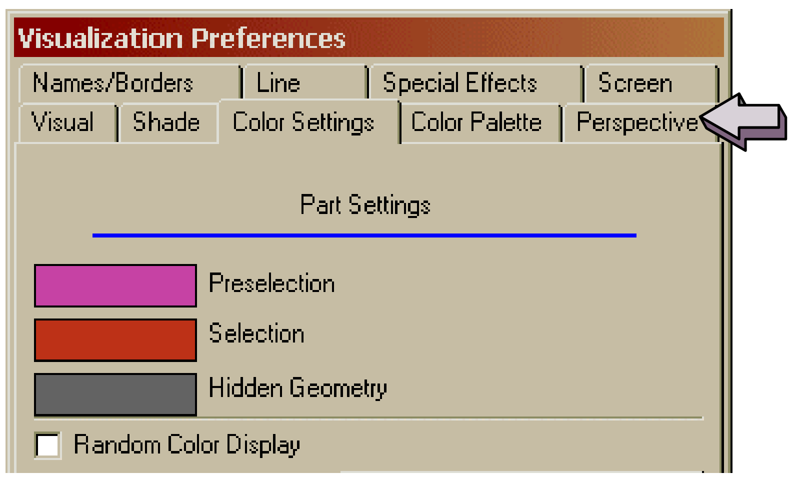
- Using **MB3 Rotate**, move the object several times to that you understand the part configuration (note that you can activate rotation also by moving the mouse to the graphics window and depressing **MB2**; as long as MB2 is depressed, the mouse is in rotation mode; holding the **Shift** key while depressing MB2 results in panning; holding the **Ctrl** key and MB2 enables zooming).

- When you are finished rotating the part, choose **MB3** → **Orient View** → **Trimetric**.

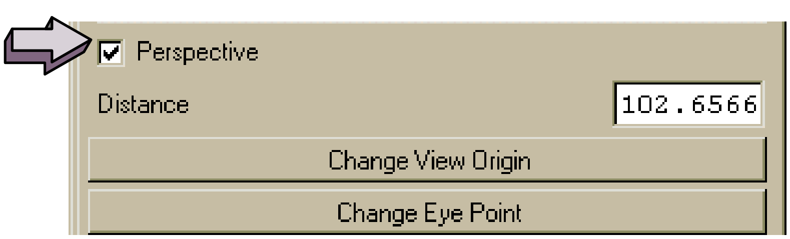
  One method to distinguish the front of the part from the back is to turn on Perspective. Perspective forces a fore-shortened view of the part. Objects closer to you appear larger; object further away appear smaller.

- From the menu bar, choose **Preferences** → **Visualization**.

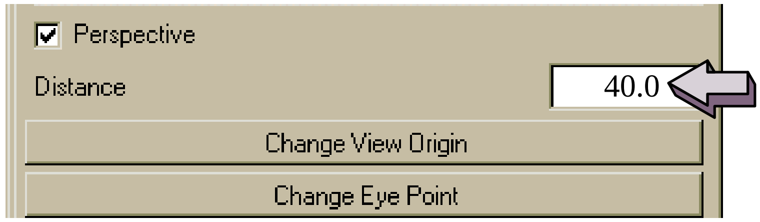
- Choose the **Perspective** property page (tab).



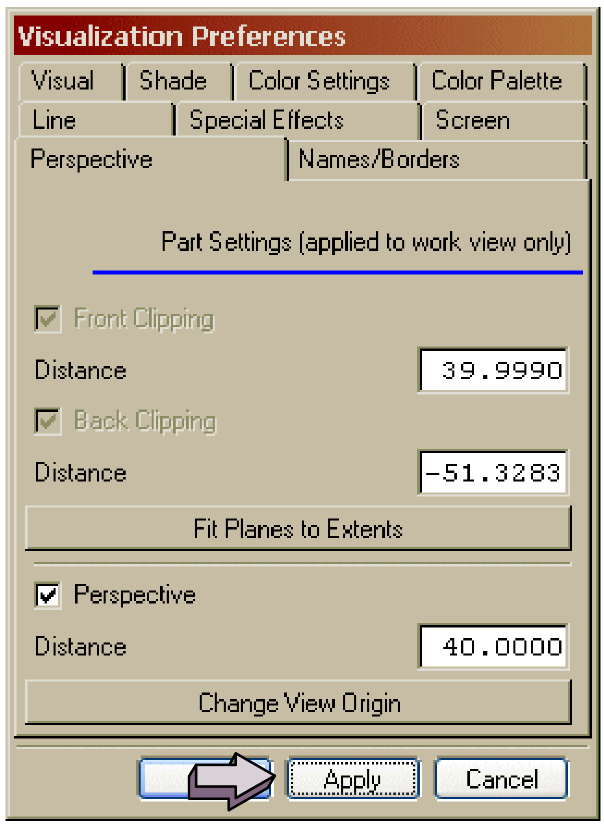
- Turn on Perspective.



❑ Key in **40.0** as the **Distance**.



❑ Choose **Apply**.



❑ Rotate the view again.

While perspective can help to view the objects there are other options as well that may be better suited. You will now work with the clipping plane option, to better enhance the visualization.

3

❑ Turn off the Perspective button.

❑ Choose **OK** in the Visualization Preferences dialog.

❑ In the graphics window, use **MB3** → **Orient View** → **Trimetric**.

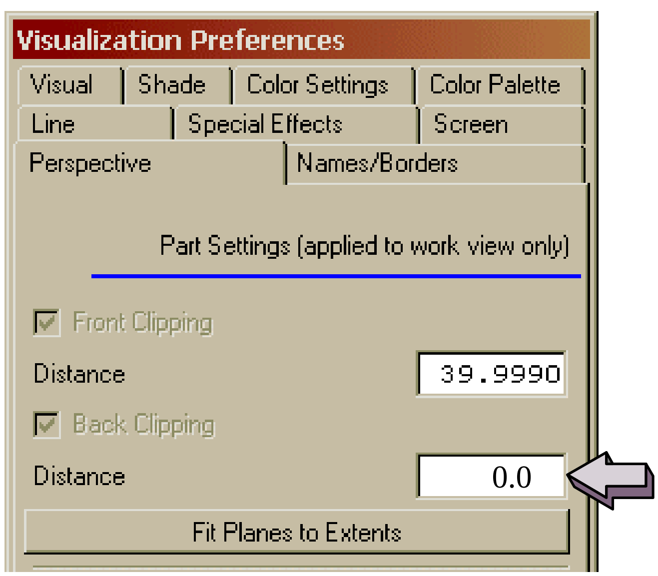
❑ From the menu bar, choose **Preferences** → **Visualization**.

❑ If necessary choose the **Perspective** property page.

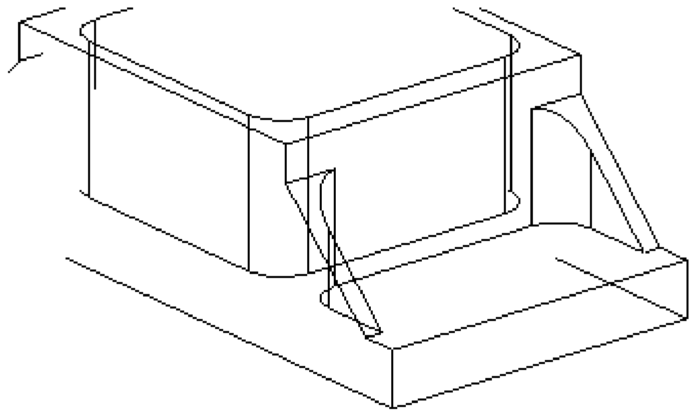
3

This time you will limit the geometry shown by setting a clipping plane.

❑ In the Back Clipping Distance field, key in 0.0.



❑ Choose **OK** on the Visualization Preferences dialog.

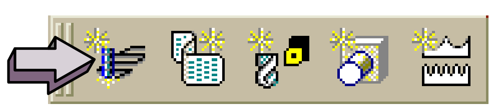


Notice the view of the geometry. Since you will be working with the geometry in the front of the part, the clipping plane setting will serve you well. Also note that any change to the graphics view that causes a regeneration, such as **Fit,** will result in the clipping plane being reset.

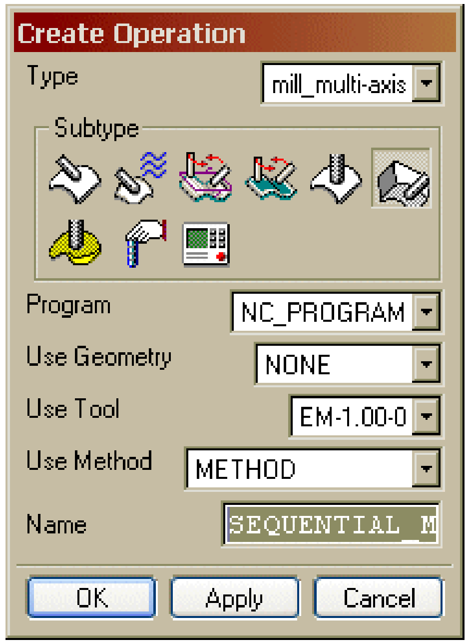
Step 3 Begin the Manufacturing Process.

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.



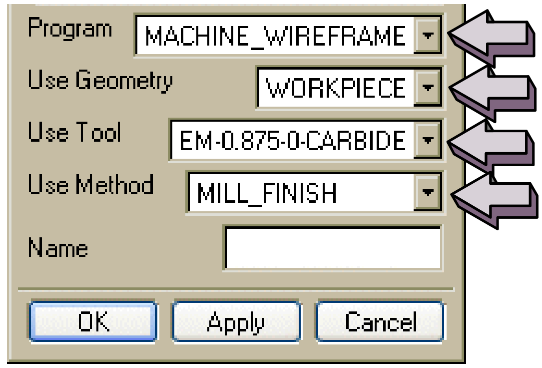
The Create Operation dialog is displayed.



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

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

❑ Set the Parent objects as follows:



❑ Name the operation **SM_1**.

❑ Choose **OK.**

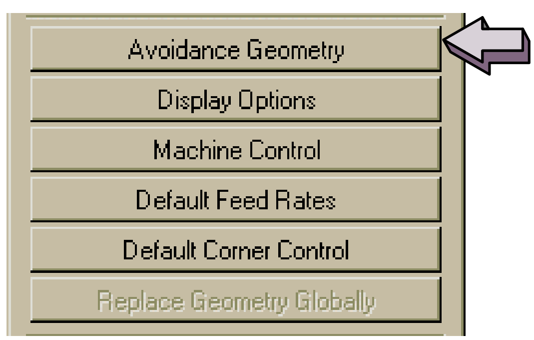
The Sequential Mill dialog is displayed.

3

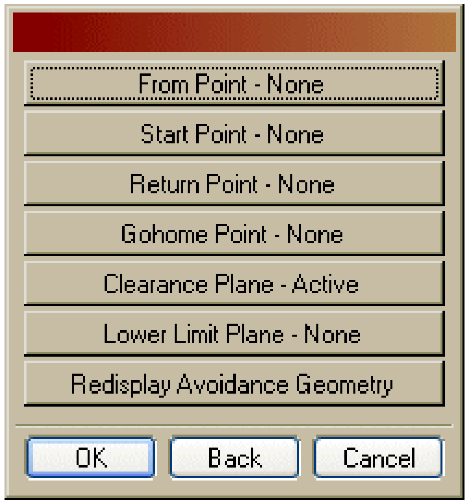
Step 4 Set the global parameters.

You will now create a Clearance Plane, used for tool clearance, and will specify the Tool Display parameters.

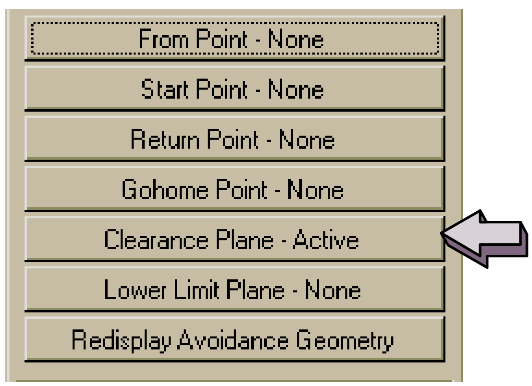
❑ Choose the Avoidance Geometry button.



The Avoidance Geometry dialog is displayed.



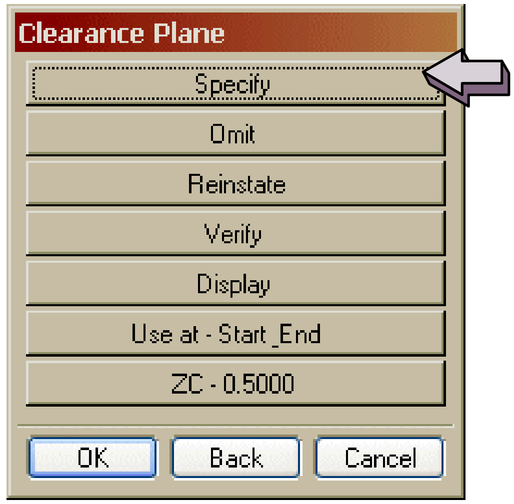
❑ Choose the **Clearance Plane** button.



3

The Clearance Plane dialog is displayed.

❑ Choose **Specify**.

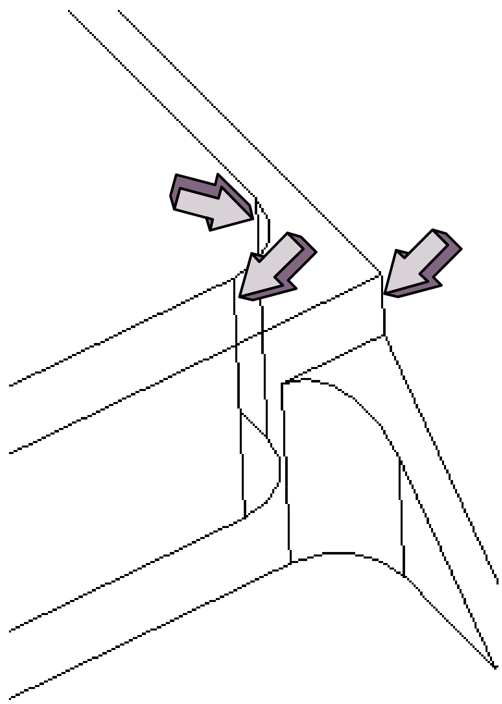


The Plane Constructor dialog is displayed.

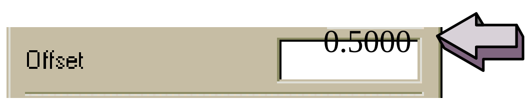
You will construct the plane through three points, in this case three end points.

❑ Change the Filter from **Any** to **Point**.

❑ Choose three end points (of the vertical lines) at the top of the part as shown, to create the clearance plane (make sure the Point Method is changed to End Point and the cone head vector points upward).

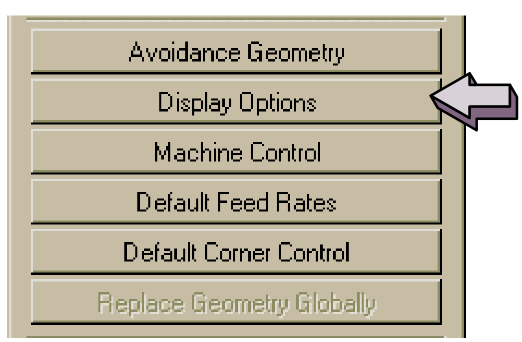


❑ Key in **.500** in the Offset field.

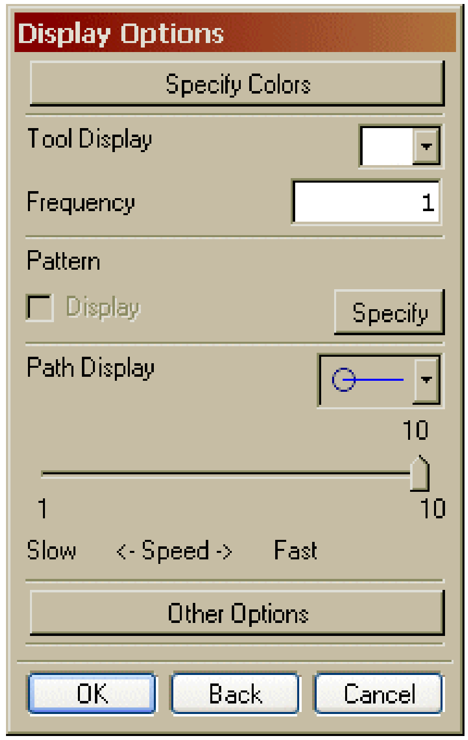


❑ Choose **OK** until you return to the Sequential Mill dialog.

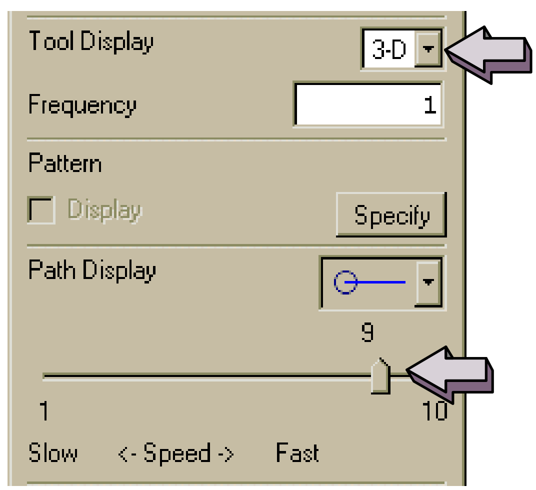
❑ Choose the **Display Options** button.



The Display Options dialog is displayed.



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



❑ Choose **OK**.

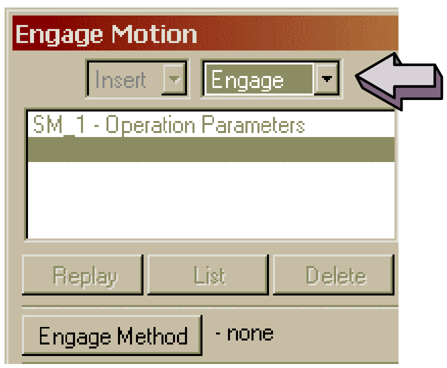
Step 5 Create the Sub-operations.

You have specified the global settings---those settings that are in effect throughout the operation. You will now create the individual sub-operations that will drive the tool around the edge of the flange.

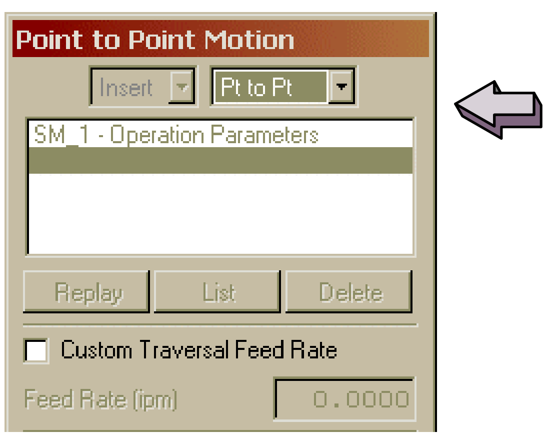
❑ Choose **OK** in the Sequential Mill dialog.

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

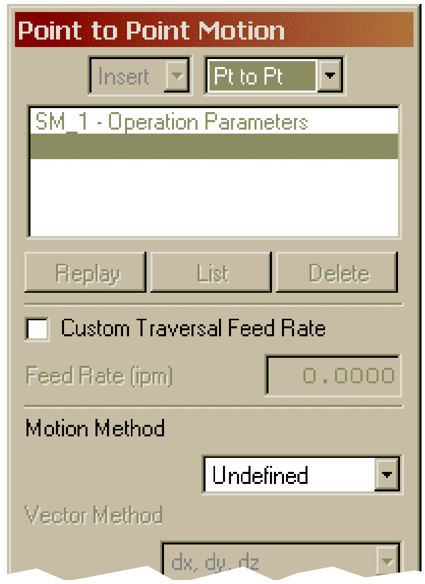
As in the previous activity, you will establish the position and tool axis using a Pt-to-Pt sub-operation.



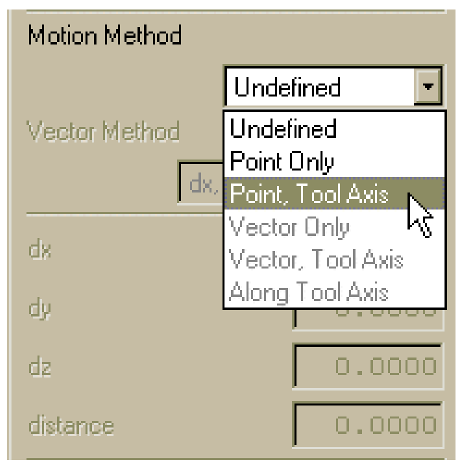
❑ Change the motion from **Engage** to **Pt to Pt**.



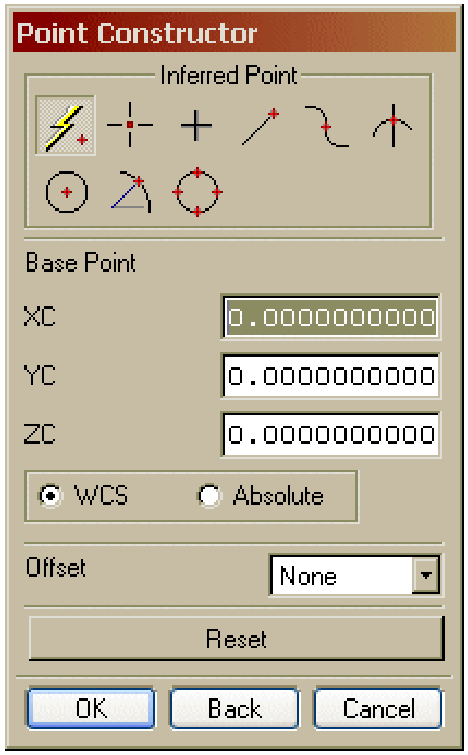
The corresponding dialog changes to match **Point to Point** motion.



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



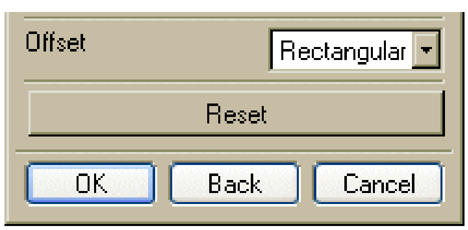
The Point Constructor dialog is displayed. You will establish the tool position by using an offset from the part geometry.



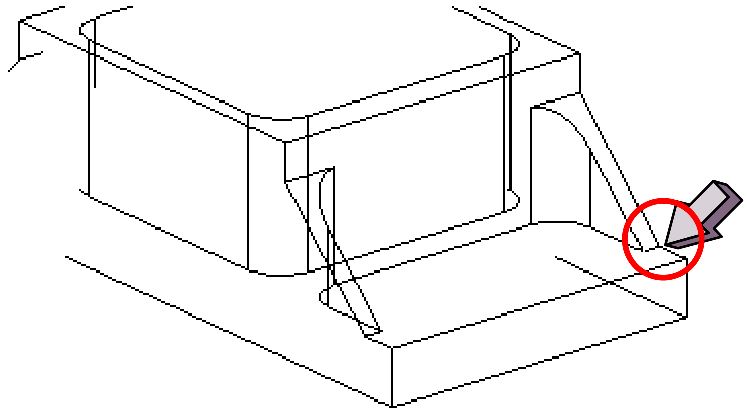
3

❑ Change the Offset to Rectangular.

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



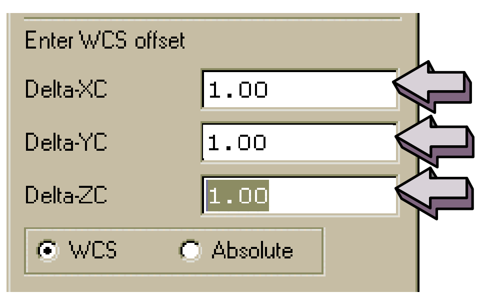
❑ Choose the point, as shown.



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

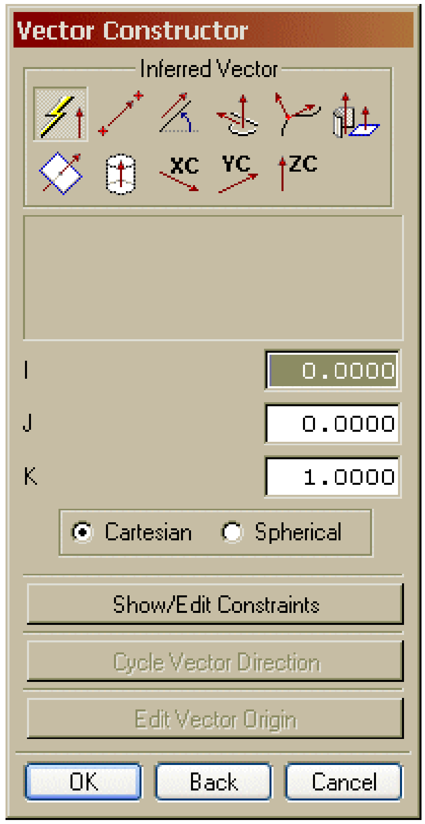
❑ Key in the following values:

Delta XC 1.00
Delta YC 1.00
Delta ZC 1.00



❑ Choose **OK**.

The Vector Constructor dialog is displayed.



3

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

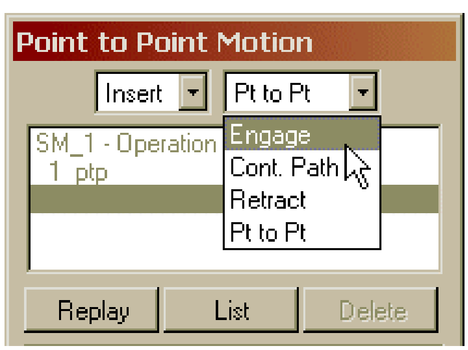
❑ Choose **OK** in the Vector Constructor dialog.

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.

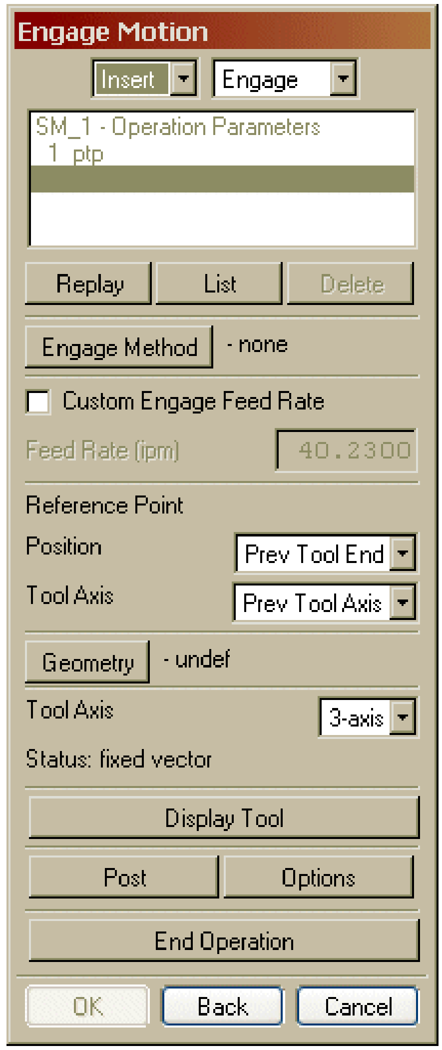
❑ Choose **OK** in the sub-operation dialog.

You will now define the Engage component.

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



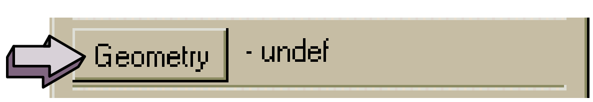
The Engage Motion dialog is displayed.



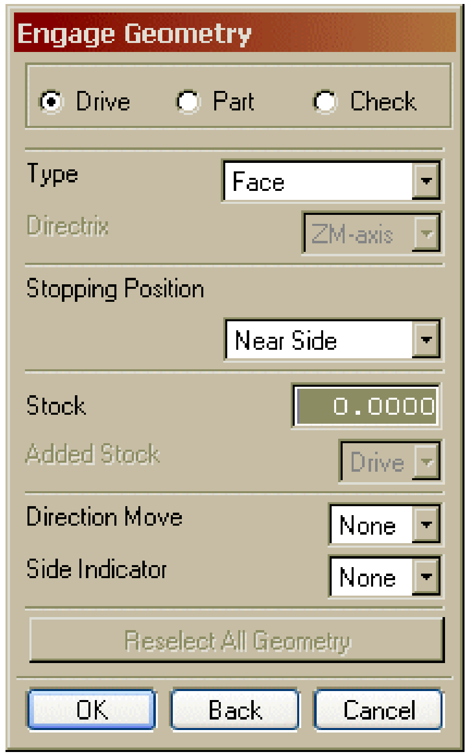
The Engage dialog requires Drive, Part and Check geometry. Additionally, you may specify an engage method.

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

❑ Choose the Geometry button from the Engage Motion dialog.



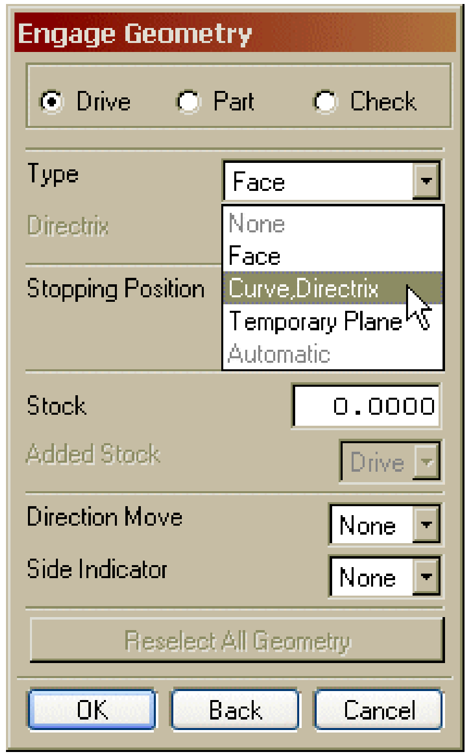
The Engage Geometry dialog is displayed.



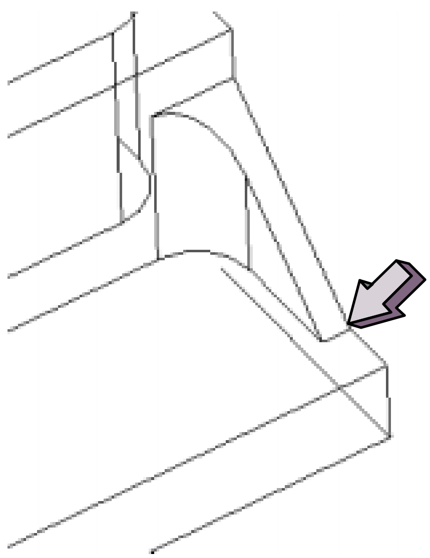
The defaults are set to **Drive** geometry, the Type is **Face** and the Stopping Position is **Near Side**.

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

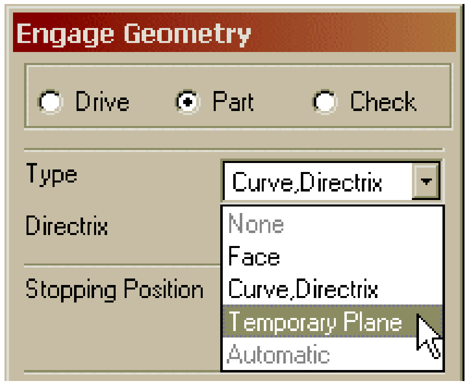


❑ Choose the curve as shown below.

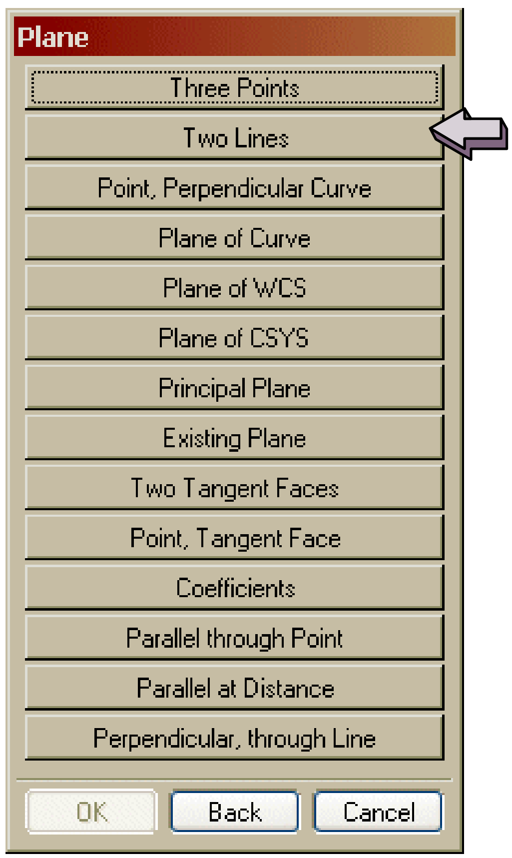


As soon as the Drive geometry is selected, the dialog advances to enable Part geometry selection. You will now select the Part geometry.

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



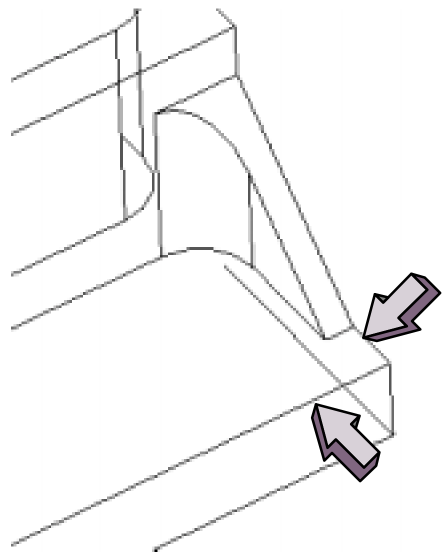
The Plane Selection method dialog is displayed. There are numerous options in which a temporary plane can be created. You will use the **Two Lines** method.



❑ Choose the **Two Lines** button.

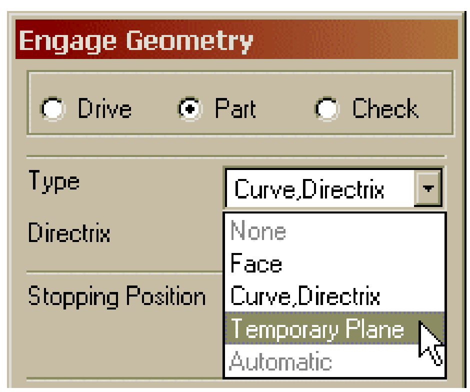
3

❑ Select the two lines as shown below.



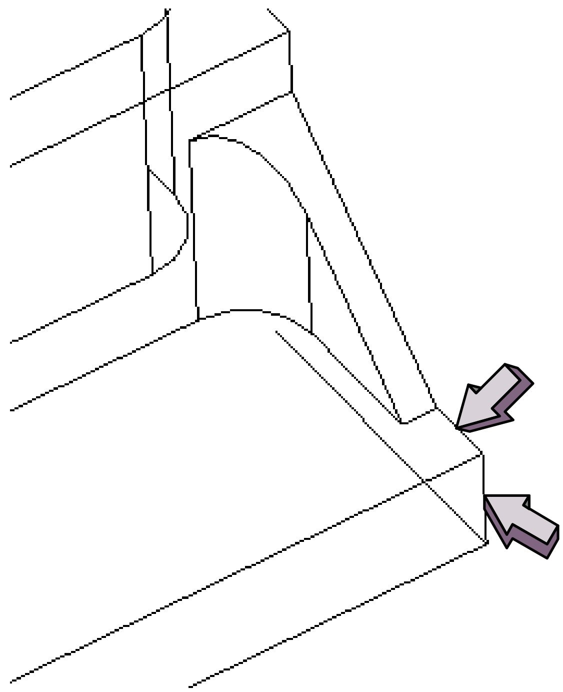
As soon as the Part geometry is selected, the dialog advances to enable Check geometry selection. You will now select the Check geometry.

❑ Choose the **Temporary Plane** option once again.



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

❑ Select the two lines as shown below.



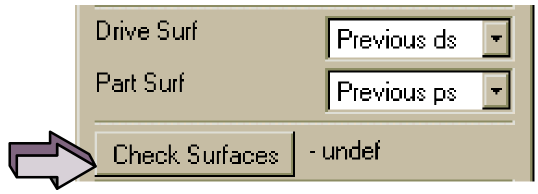
❑ Choose **OK.**

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

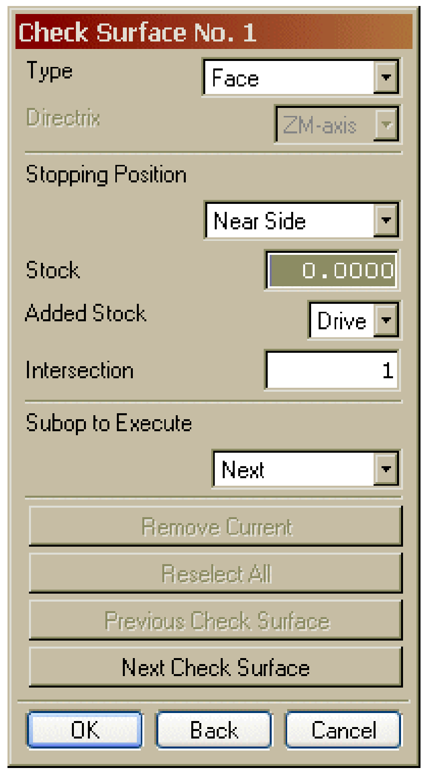
Due to the method of selection of the Part and Drive geometry, the defaults for the new Drive and Part geometry are correct.

❑ Choose the **Check Surfaces** button.

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

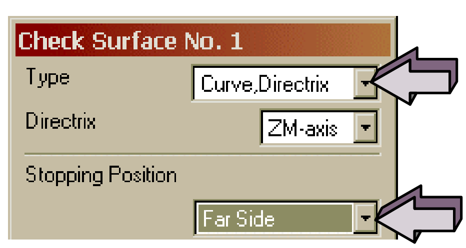


The Check Surfaces dialog is displayed.



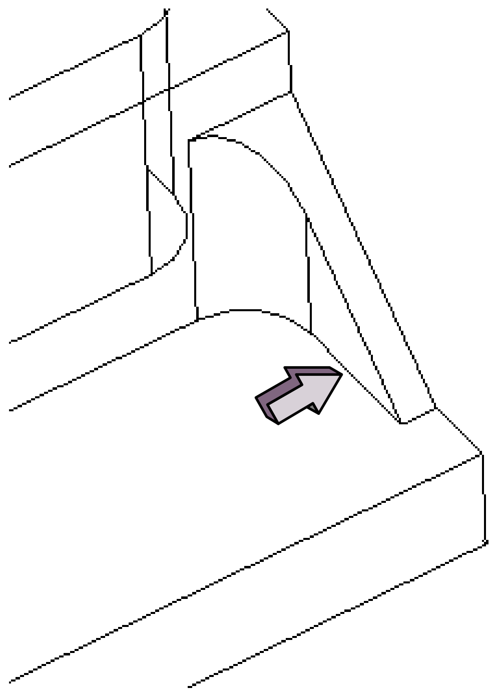
Before selecting the new Check geometry you need to determine the proper **Stopping Position** of the cutting tool.

- ❑ Change the Stopping Position from **Near Side** to **Far Side**.
- ❑ Change the Type from **Temporary Plane** to **Curve, Directrix**.



3

❑ Choose the curve as shown below.



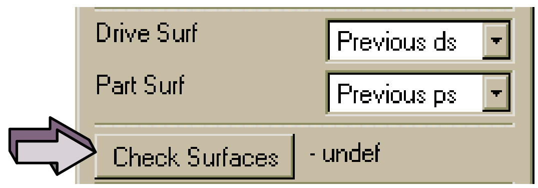
Remember that as soon as you select the first Check geometry, in this case a curve, the dialog advances so that you can select the next item for Check geometry.

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

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

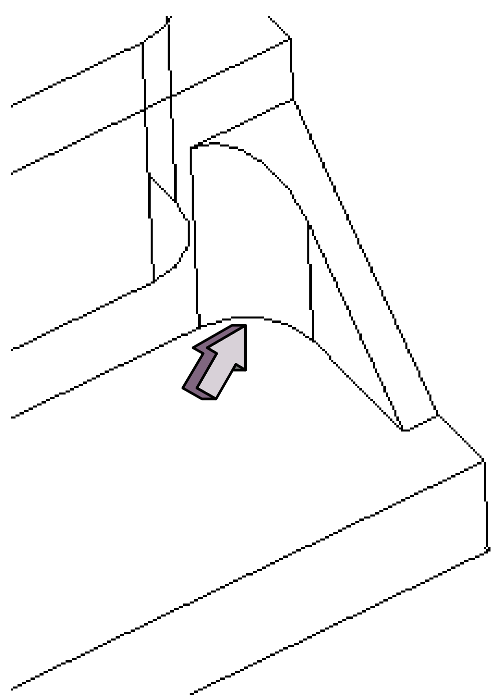
You will now create the Continuous Path Motions to complete the area to be machined.

❑ Choose the **Check Surfaces** button.



❑ Change the Stopping Position to **Ds-CS Tangency**.

❑ Choose the curve as shown below.



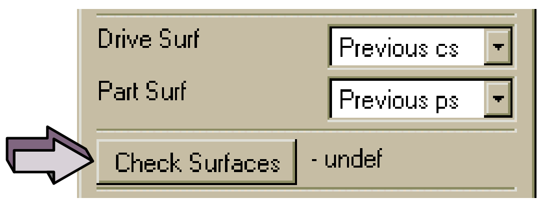
❑ Choose **OK** in the Check Surface dialog.

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

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

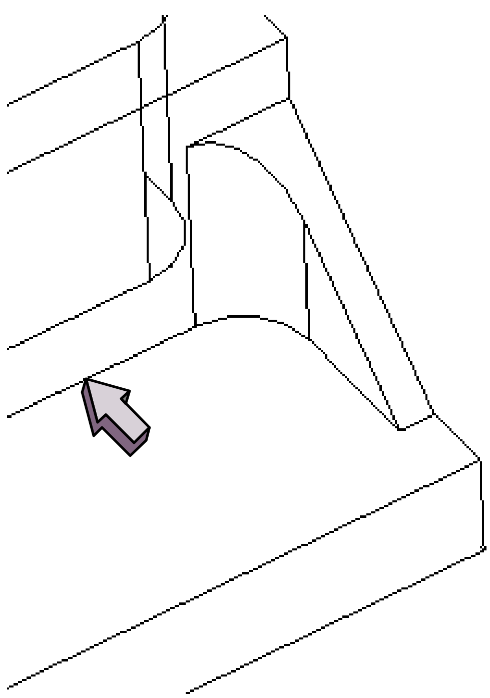
Continue the tool path motion by selection of the next element of Check geometry.

❑ Choose the **Check Surfaces** button.



3

❑ Choose the curve as shown below.

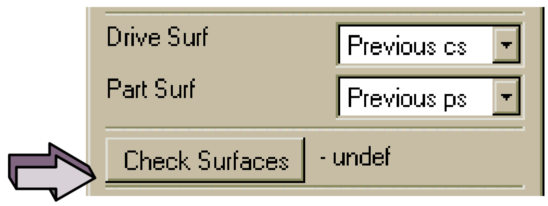


❑ Choose **OK** in the Check Surfaces dialog.

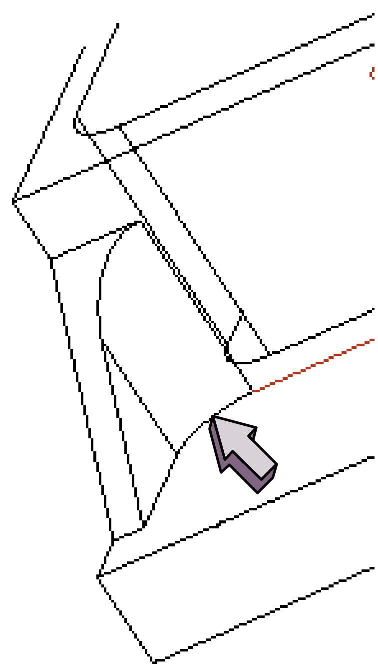
❑ Choose **OK** to create the new sub-operation.

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 below.

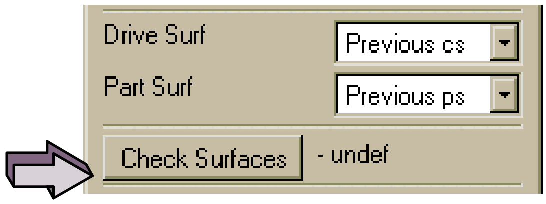


❑ Choose **OK** in the Check Surfaces dialog.

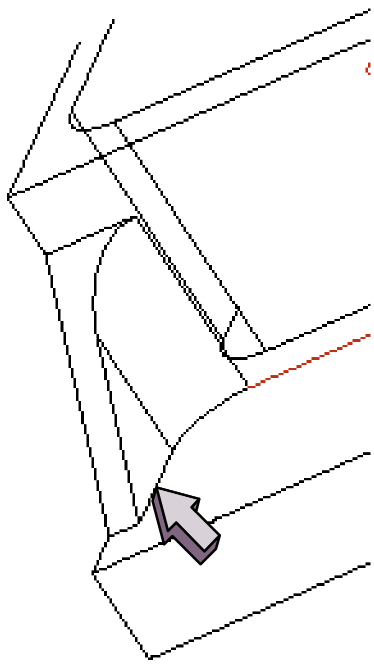
❑ Choose **OK** to create the new sub-operation.

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 below.



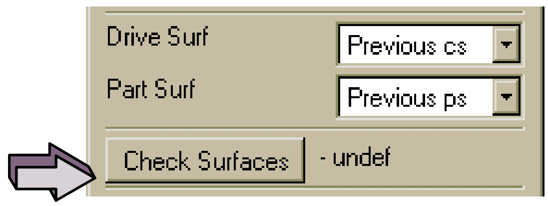
❑ Choose **OK** in the Check Surfaces dialog.

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

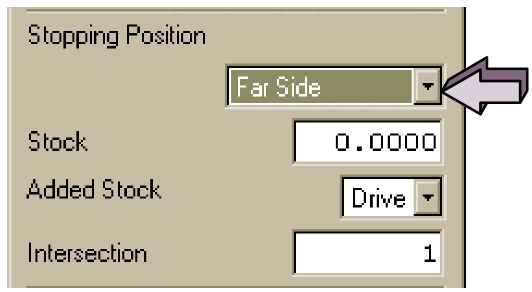
The seventh sub-operation, **7 cpm**, has been created. Continue by selecting the next curve.

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

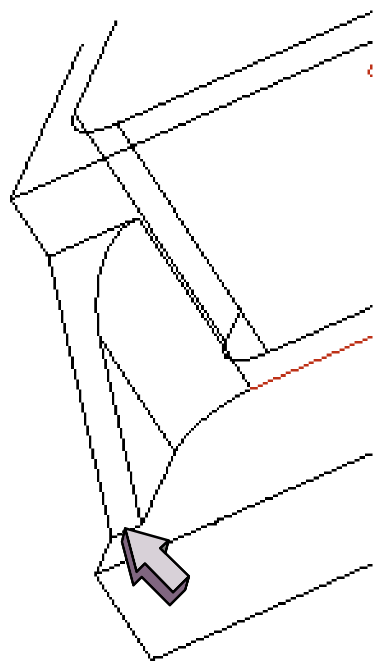
❑ Choose the **Check Surfaces** button.



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



❑ Choose the curve as shown below.

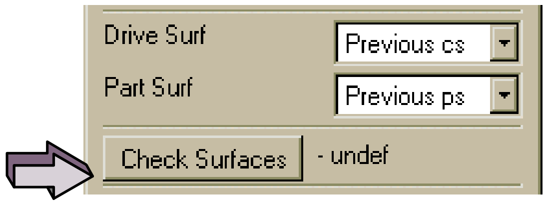


❑ Choose **OK** on the Check Surfaces dialog.

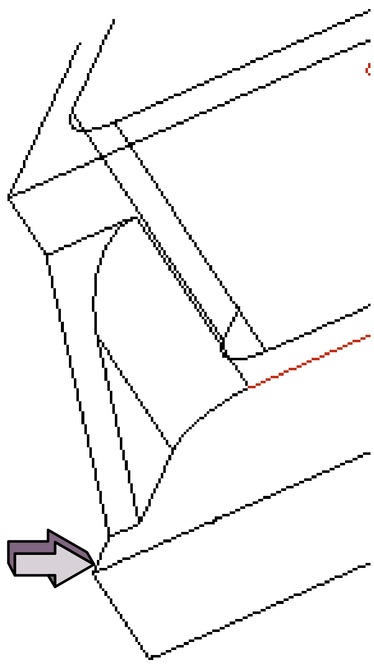
❑ Choose **OK** to create the new sub-operation.

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 below.

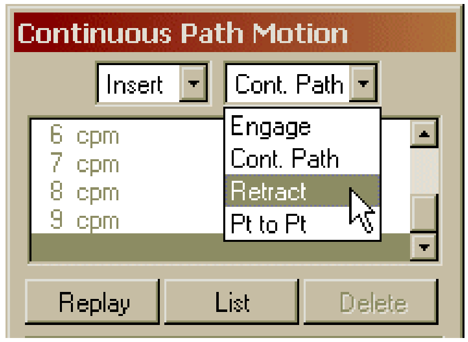


❑ Choose **OK** in the Check Surfaces dialog.

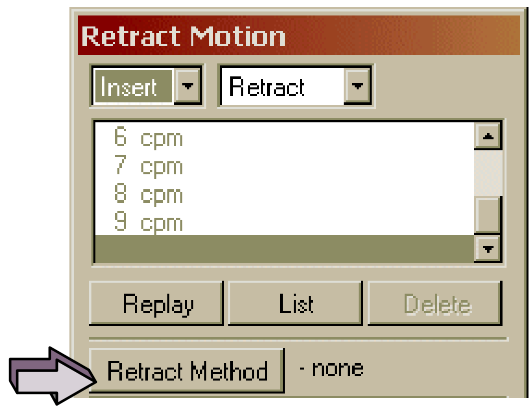
❑ Choose **OK** to create the new sub-operation.

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

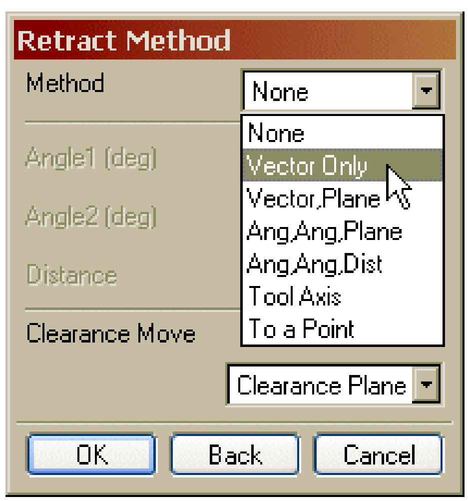
❑ Change from **Cont. Path** to **Retract** motion.



❑ Choose the Retract Method button.



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

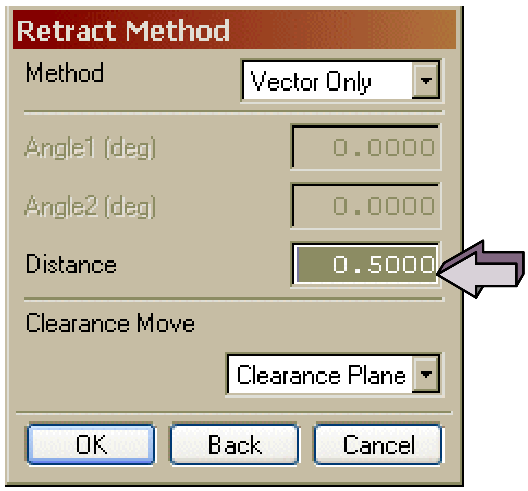


❑ Key in the following values in the Vector Constructor dialog:

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.

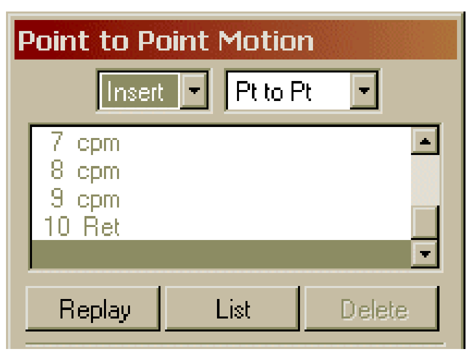


❑ Choose **OK** in the Retract Method dialog.

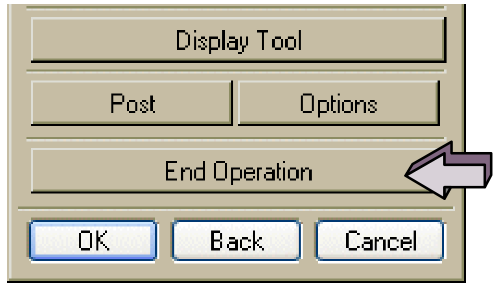
3

❑ 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.

This completes the activity and the lesson.

SUMMARY

The ability of utilizing wireframe type geometry in Sequential Milling operations, affords the flexibility of the types of parts that can be machined. The capability of using wireframe 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

Sequential Mill Advanced

Lesson 4

PURPOSE

Some of 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.

OBJECTIVES

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

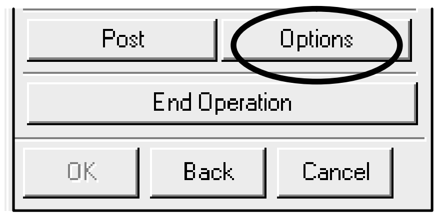
This lesson contains the following activities:

Standard and Nested Loops

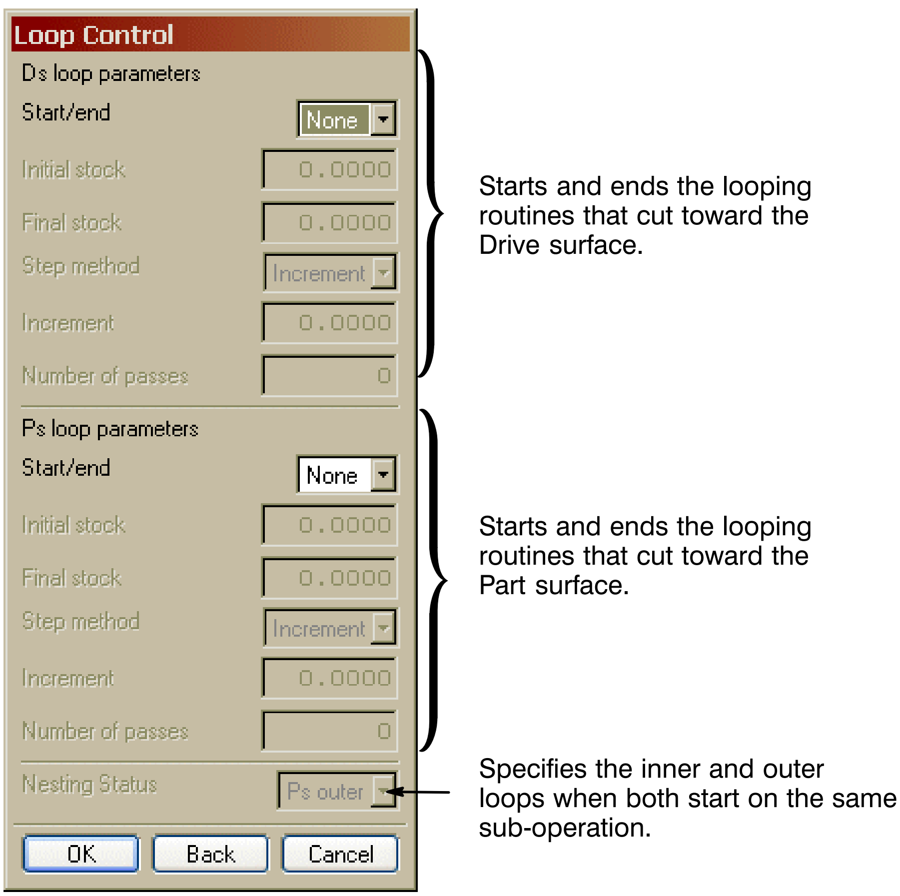
Standard Loops

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 button.

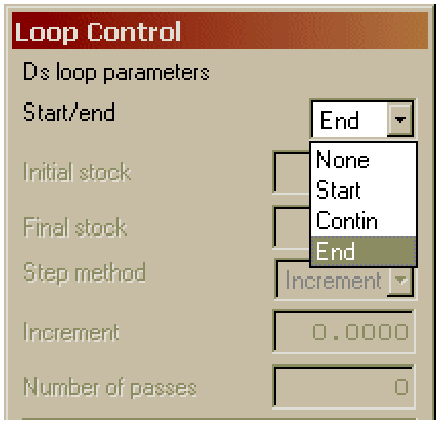


The Loop Control dialog follows:

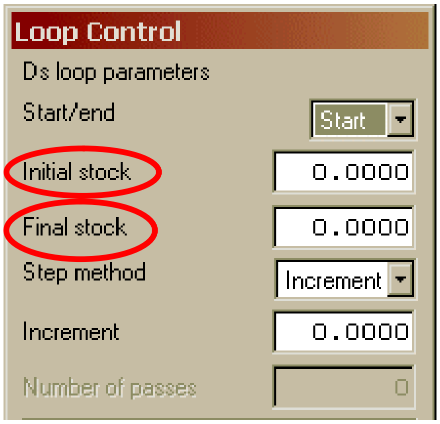


Before you begin the creation of a loop, the tool should be in the proper position within the operation (where you want the tool to start repeating from).

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**.

4

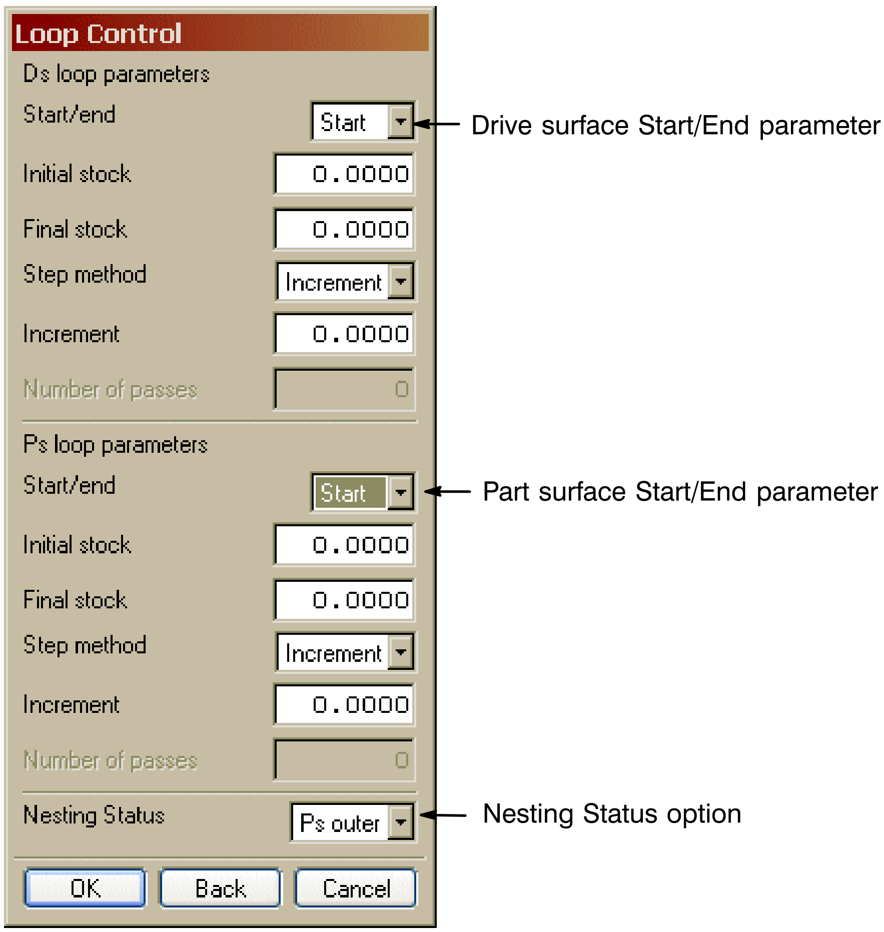
The tool path is then recomputed by adding the loop Stock and moving toward the part in a specified number of steps. The path will display in the graphics window. At this point, you can determine if the loop is correct. You can continue or change individual motions as required.

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

Nested Loops

A Drive surface and a Part surface loop within the same sub-operation or a later sub-operation is considered to be 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.



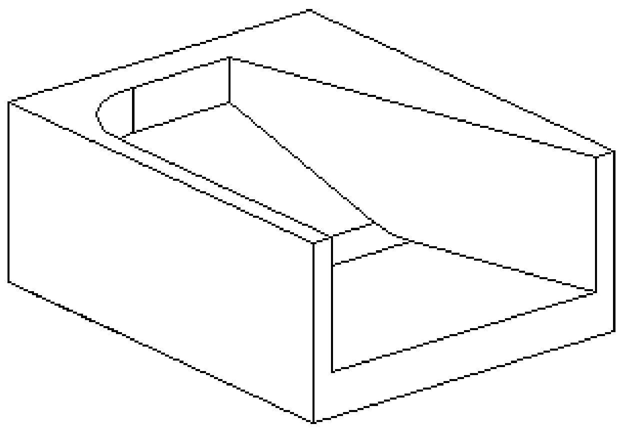
The next activity will familiarize you with some of the basic concepts of looping within Sequential Mill.

Activity 4—1: Sequential Mill — Using Loops

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

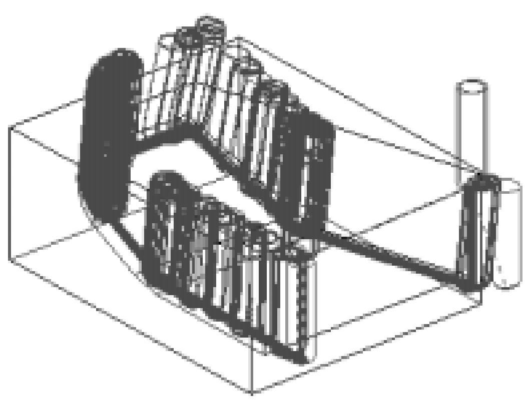
Step 1 Open a new part file and replay an existing operation.

- ❑ Open the file **mam_sq_3_loop.prt**.



- ❑ Enter the **Manufacturing application** by choosing **Application → Manufacturing.**

- ❑ From the Operation Navigator, highlight the **FINWALLS** operation, using **MB3**, **Replay** the 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 on the Sequential Mill dialog, the **Multiaxis Output** option is selected.

❑ Choose **OK.**

The Point to Point Motion dialog is displayed.

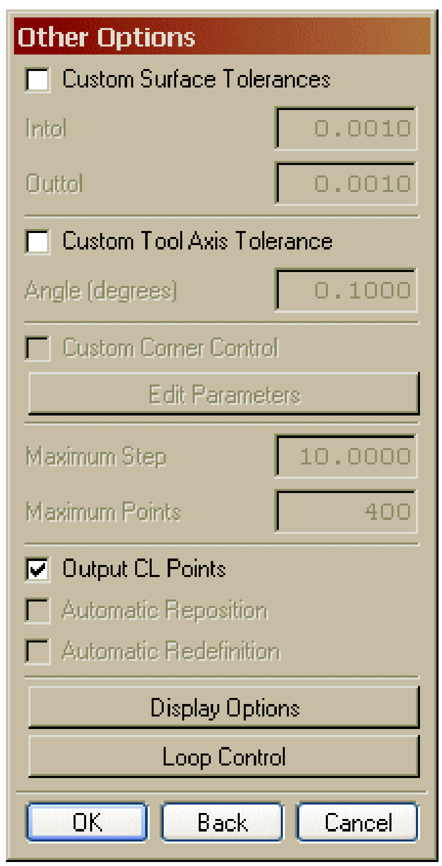
❑ Choose **OK**.

The Engage Motion dialog is displayed.

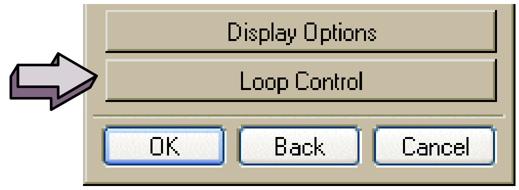
Normally, you start the looping process from within this dialog.

❑ Choose **Options**.

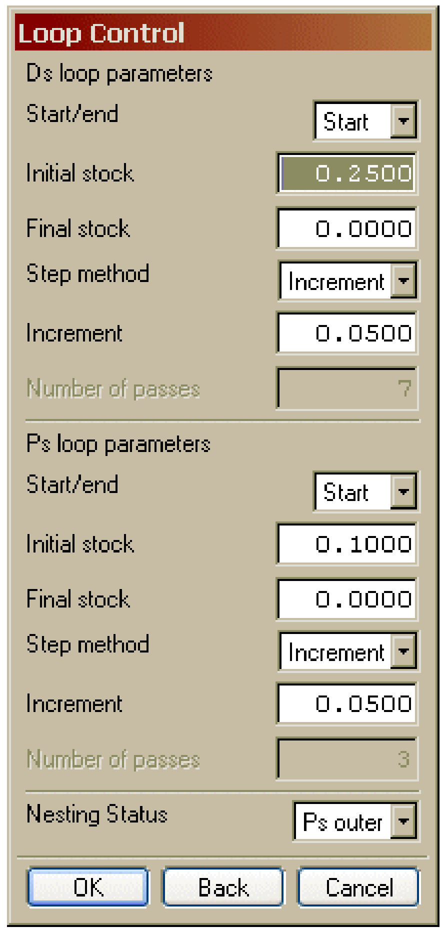
The Other Options dialog is displayed.



❑ Choose **Loop Control**.



The Loop Control dialog is displayed.



Note the Drive Surface (Ds) loop settings. The Start/End option is set to **Start**, the Initial stock is **.250,** the Step method is **Increment,** the Increment value is set to **.050**.

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

Note the Part surface loop settings as well.

❑ 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.

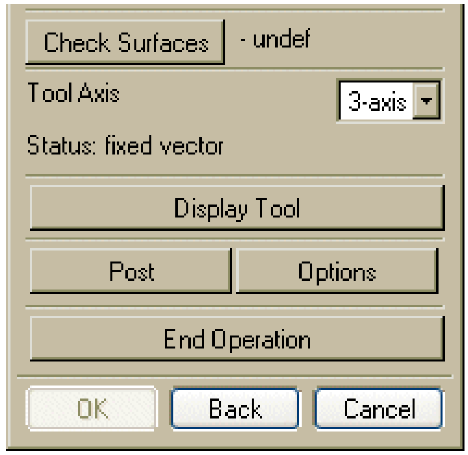
- ❑ Choose **End Operation** and then choose **OK** from the End Operation dialog to save the operation and return to the Operation Navigator.

The entire tool path is now displayed.

- ❑ Close the part.

Tool Axis Control

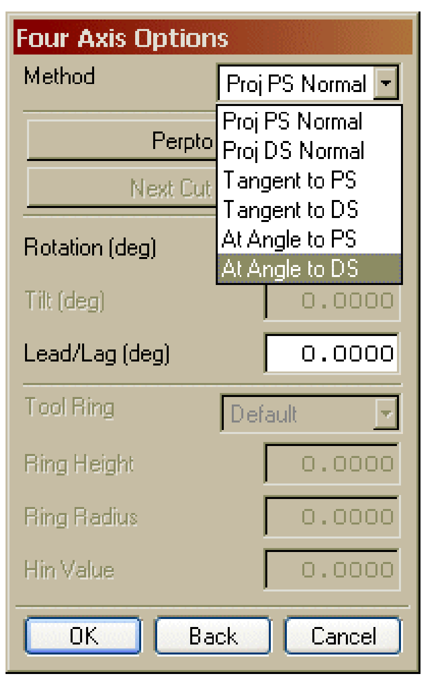
In Sequential Mill, you define the tool axis by first specifying 3, 4 or 5-axis tool positioning which is found on the Engage and Continuous Path Motion dialogs.



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)



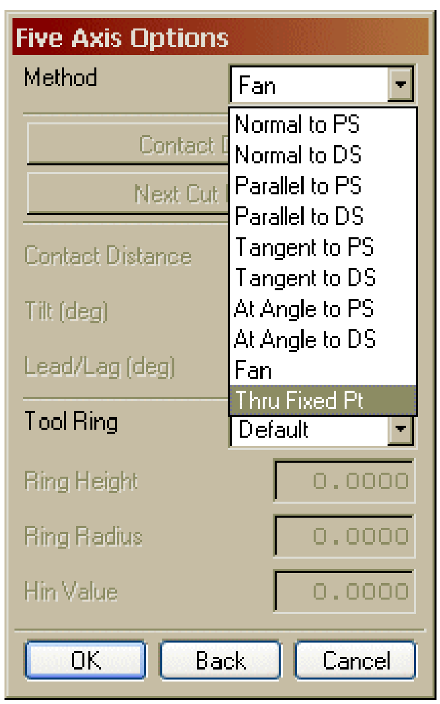
Project Part Surface (or **Drive Surface**) **Normal** indicates that the tool axis is obtained by rotating the surface normal by a lead or lag angle, projecting the resulting vector onto a plane perpendicular to the specified Perpto vector, and then rotating it in that plane by a specified angle. This 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.

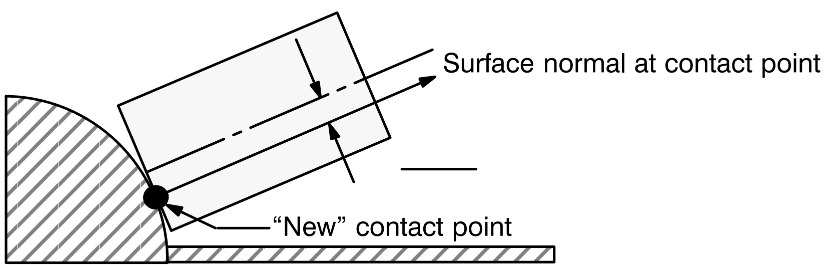
5-axis allows the tool axis to :

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



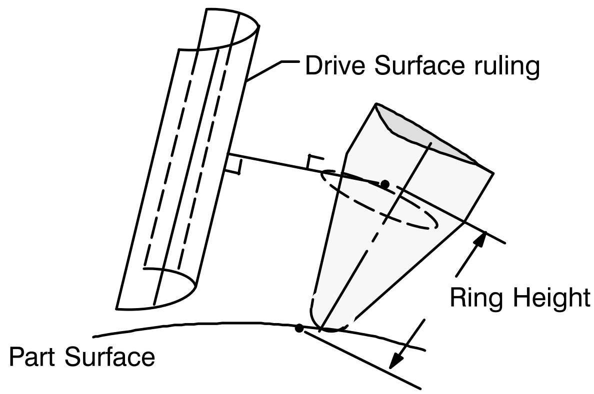
5-Axis Tool Axis Control	
Variable Contour	**Sequential Mill**
Toward or Away From Point	**Thru Fixed Point**
Normal to Part	**Normal To PS**
Normal To Drive	**Normal To DS**
Swarf Drive	**Parallel to PS** **Parallel to DS**
Relative To Drive	**At Angle to DS** **At Angle to PS**
—	**Tangent to PS**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

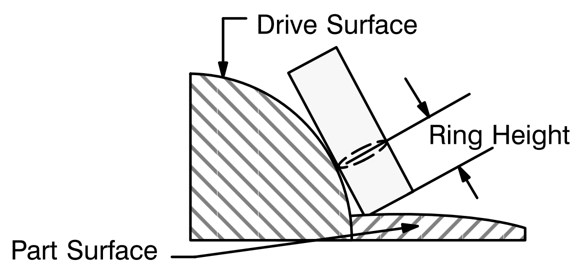


4

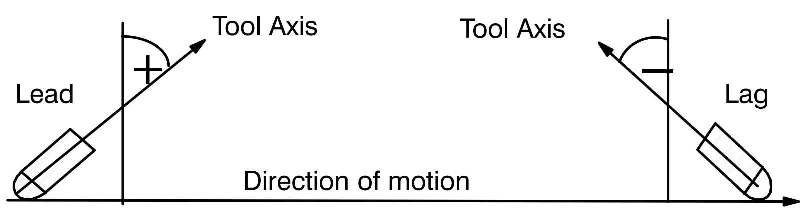
Parallel to Ps (or **Ds**) causes the side of the tool to be kept parallel to the surface rulings at the contact point. A ring on the tool must be specified to indicate where the side of the tool must touch the surface.



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.

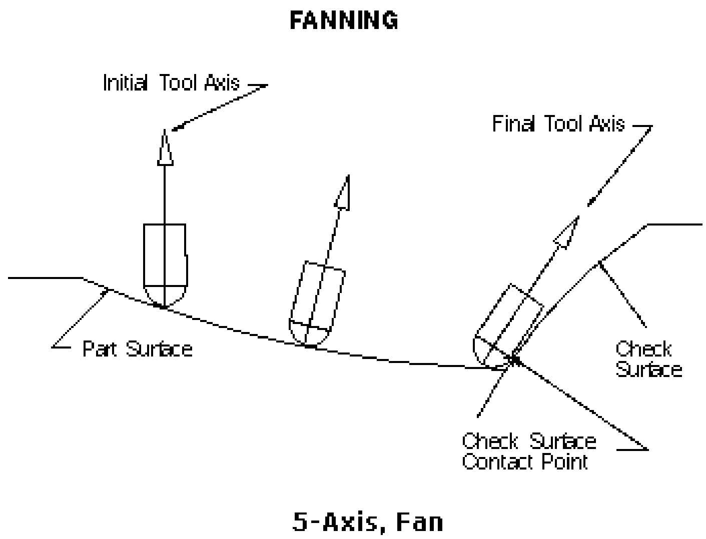


At Angle to Ps (or **Ds**) causes the tool axis to maintain a fixed angle (Tilt) with the surface normal and a fixed angle with the current direction of motion (a Lead or Lag angle).

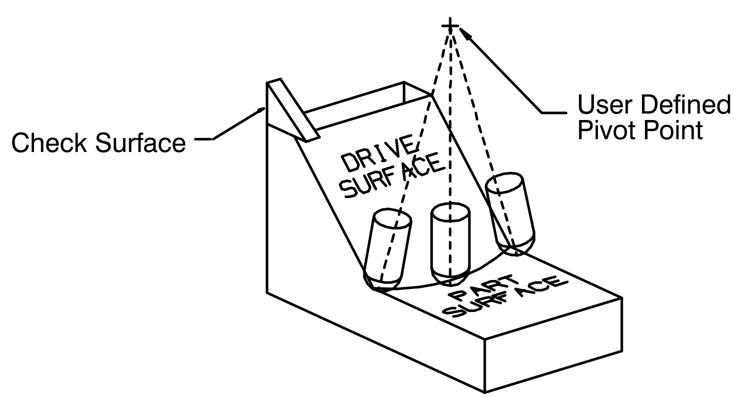


4

Fan 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.



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.

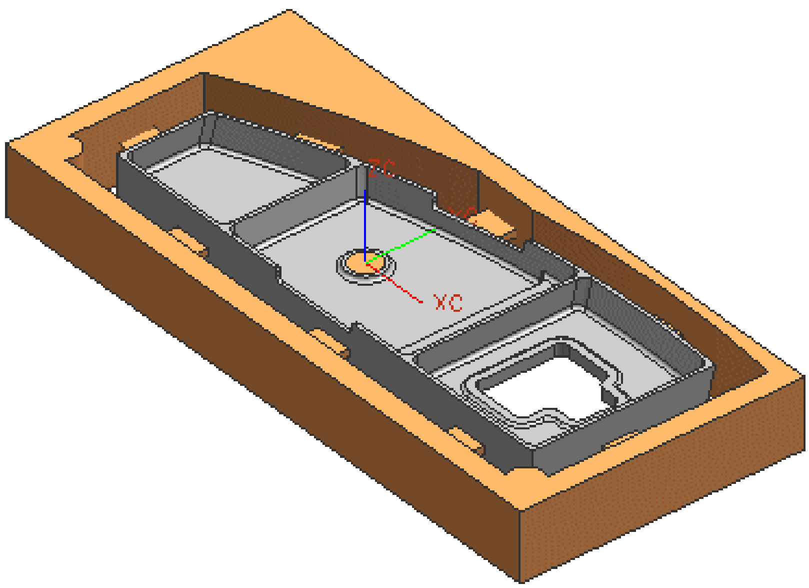


Activity 4–2: Sequential Mill Five-Axis Fan Motion

In this activity, you will create a Sequential Milling operation to finish the walls of a pocket on an aircraft structural component. The part has already been roughed out with Cavity Milling and the floors have been finished. In the next activity, you will create a semi-finish operation to remove excess stock.

Step 1 Open, rename and examine the part file.

❑ Choose **File → Open** and select the part file **mam_spar_mfg.prt.**



4

This is a “window frame” part. 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 accurately during the machining process.

The orange material represents the “window frame” portion of the block. Small tabs run from it to the part to secure it during the machining process. After the part is fully machined, the tabs are cut off, releasing the part from the frame.

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

❑ Rename the part ***_spar_mfg.prt using the **File** → ∇ℑ**Save As** option on the menu bar (*** represents your initials).

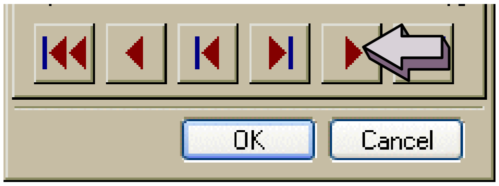
❑ Enter the **Manufacturing application** by choosing **Application** → **Manufacturing.**

❑ Choose the Operation Navigator tab from the tool bar.

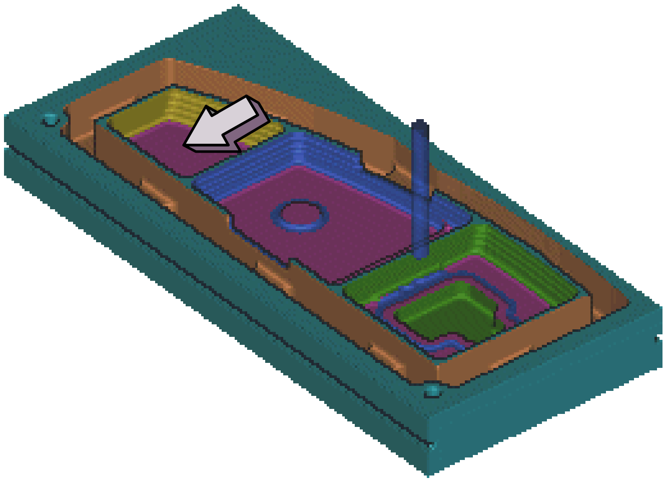
❑ Highlight the **SIDE_1** program object and then use **MB3,** select **Toolpath** and then **Verify.**

❑ Select the **Dynamic** tab from the Tool Path Visualization dialog.

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



The In-Process Workpiece of the part is represented. You will begin machining the leftmost pocket in the part.

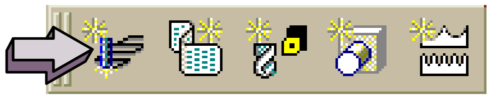


❑ Choose **OK** on the Tool Path Visualization dialog.

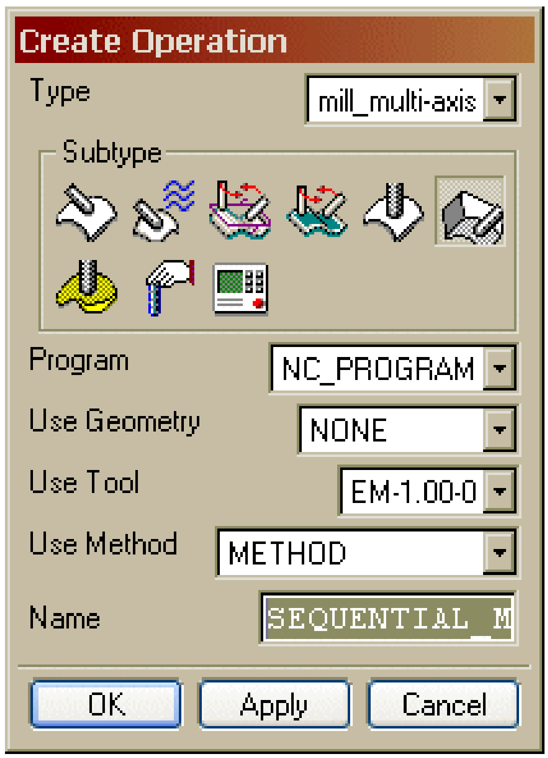
4

Step 2 Creat the Sequential Mill Operation.

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



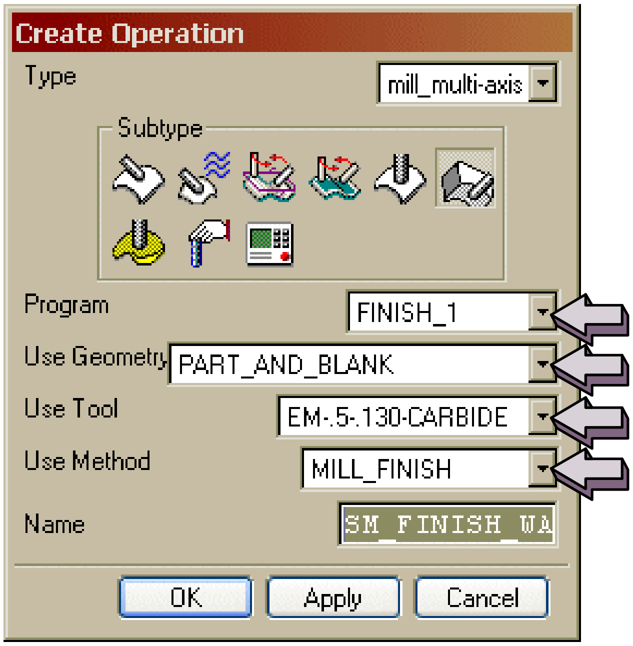
The Create Operation dialog is displayed.



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

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

❑ Set the Parent objects as follows:



❑ Key in **SM_FINISH_WALLS_POCKET_1** as the Name.

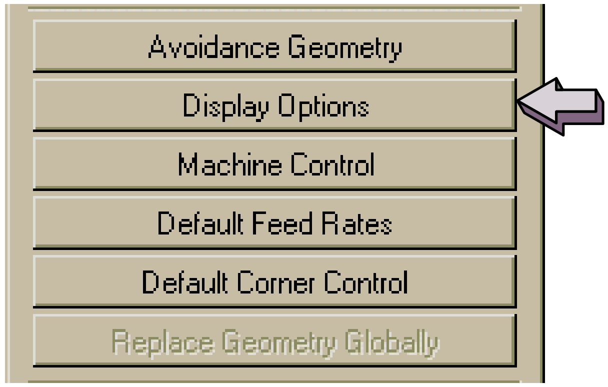
❑ Choose **OK.**

The Sequential Mill dialog is displayed.

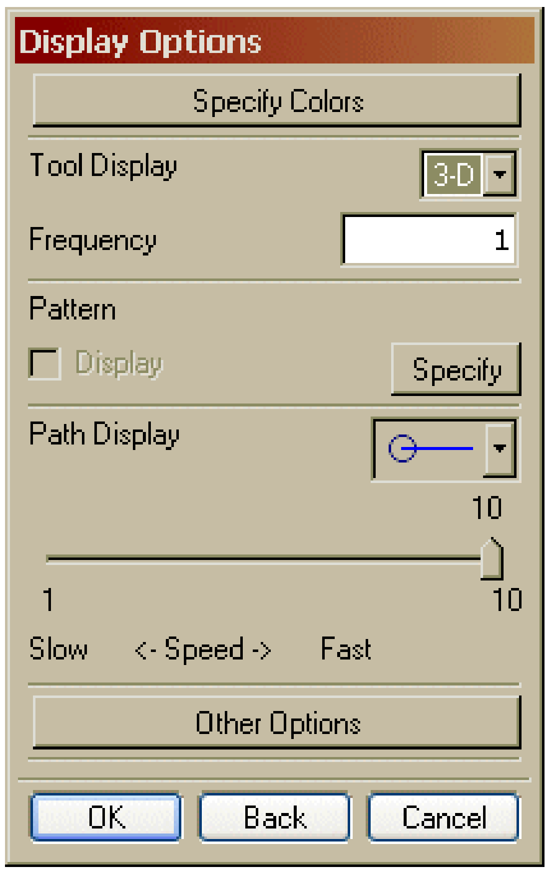
Step 3 Set Tool Display options and create a Point to Point Motion.

You will now set the tool display options, which will make the tool easier to visualize when you are creating tool motions.

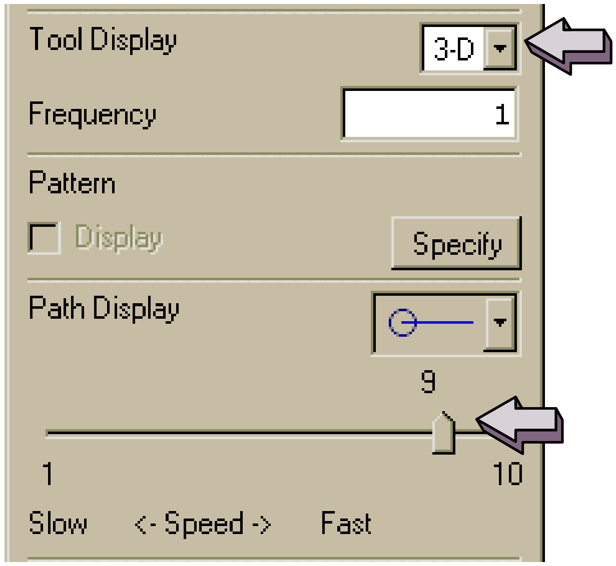
❑ Choose the **Display Options** button.



The Display Options dialog is displayed.



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

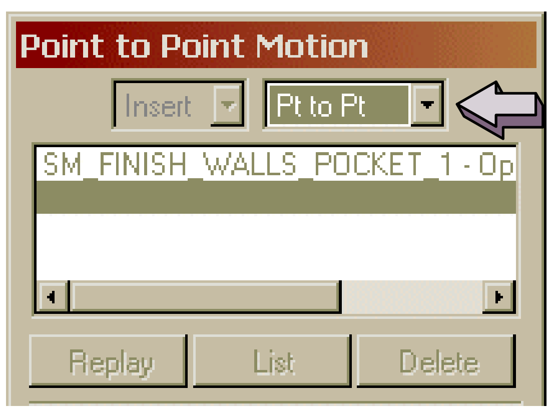


❑ Choose **OK**.

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**.

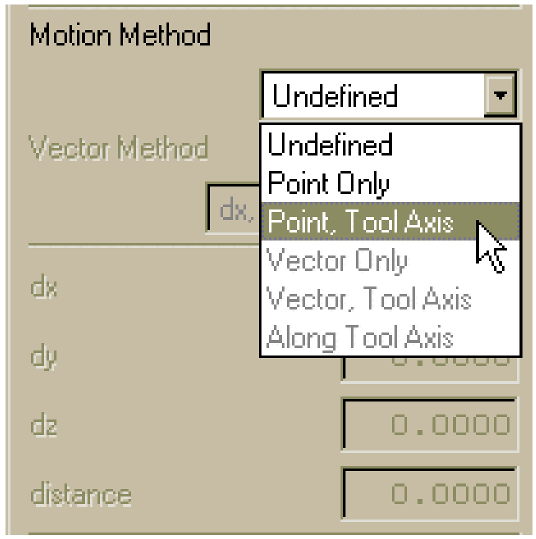


The corresponding dialog changes to match **Point to Point** motion.

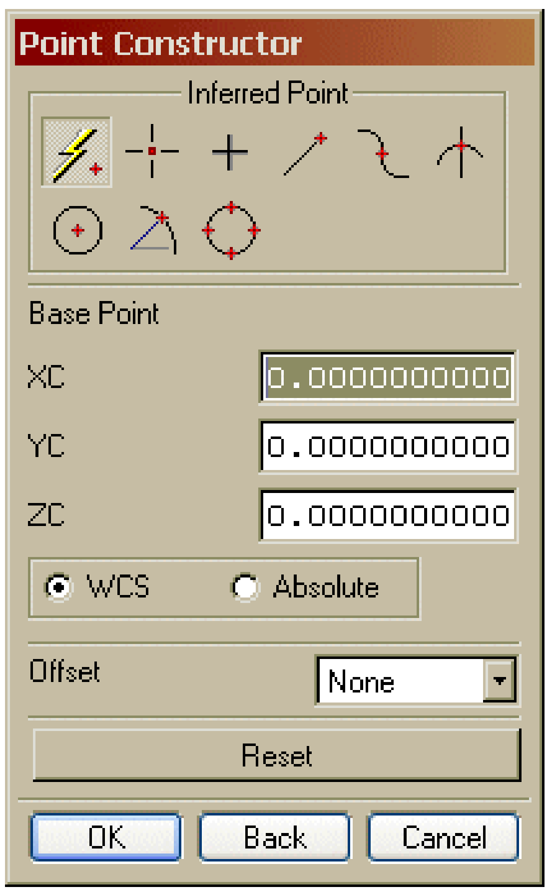
You will now establish the tool position, specifying both the position of the tool and the tool axis.

4

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

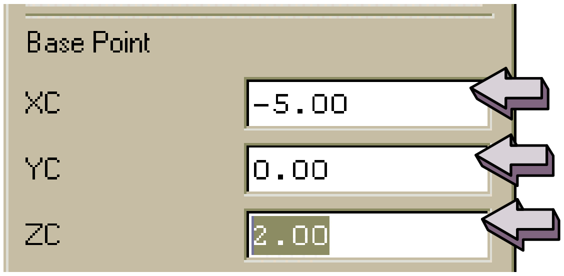


The Point Constructor dialog is displayed.



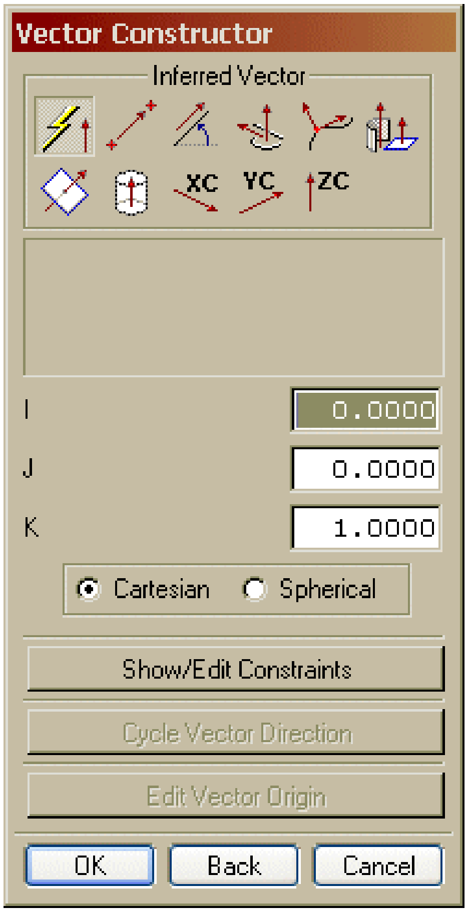
❑ Key in the following values for the Base Point:

X –5.00
Y 0.00
Z 2.00



❑ Choose **OK**.

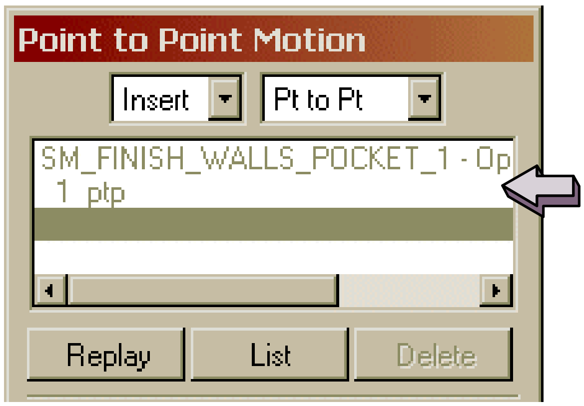
The Vector Constructor dialog is displayed.



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.

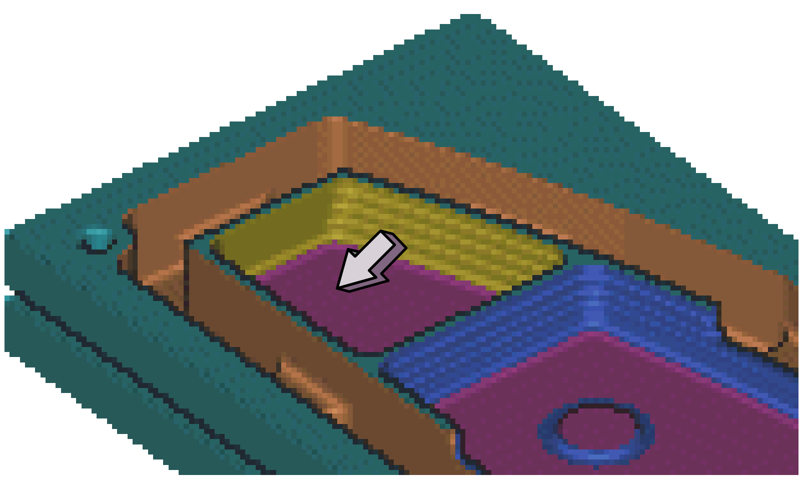
❑ Choose **OK** to accept the first sub-operation.



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

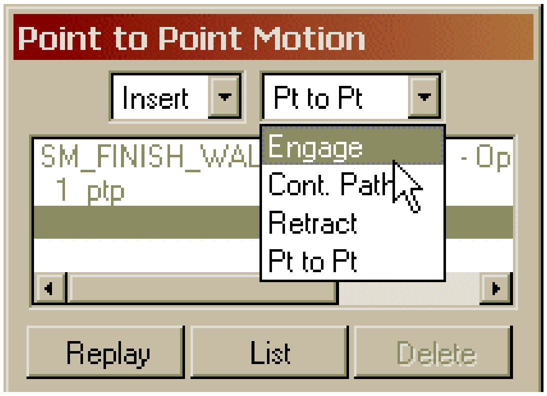
Step 4   Create the Engage Motion.

In Sequential Mill, it is a best practice to establish a cutting tool along a straight wall if possible when performing multi-axis machining as well as to feed into the wall away from a corner to eliminate tool chatter. Therefore, you will engage the wall as shown in the diagram below.

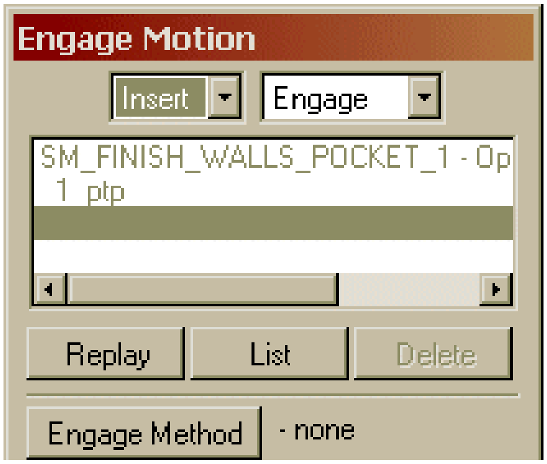


You will now define the Engage component.

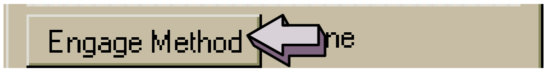
❑ Change to an **Engage** sub-operation.



The Engage Motion dialog is displayed.



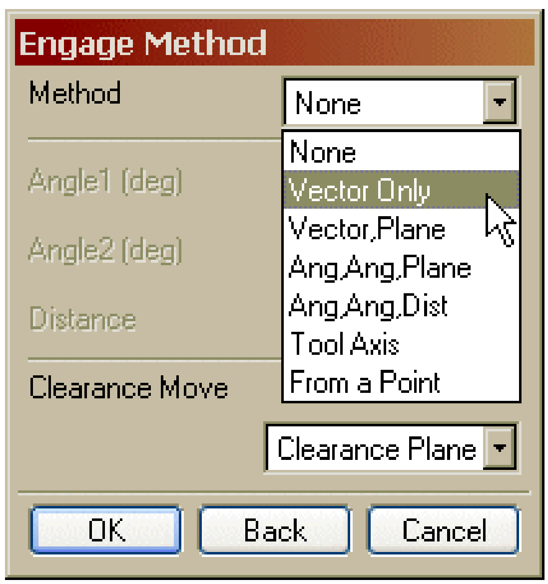
❑ Choose the Engage Method button.



4

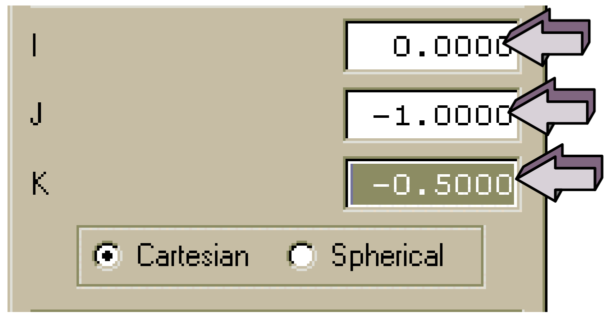
The Engage Method dialog is displayed.

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



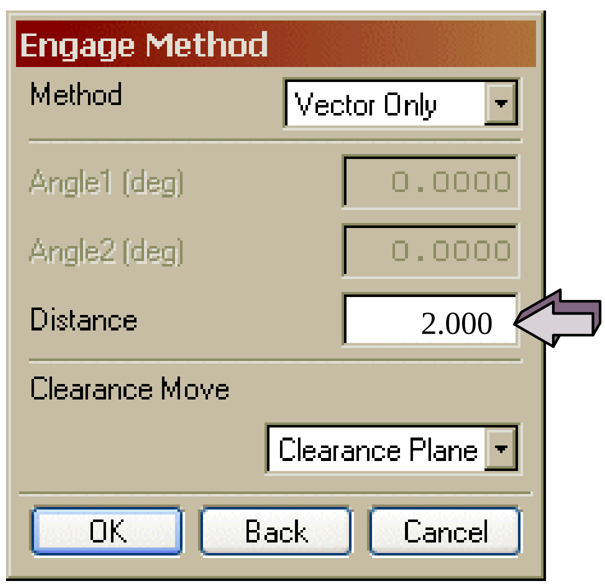
The Vector Constructor dialog is displayed.

- ❑ Key in the following values:
  I = 0.000
  J = –1.000
  K = –.500



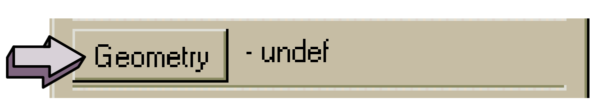
- ❑ Choose **OK**.

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

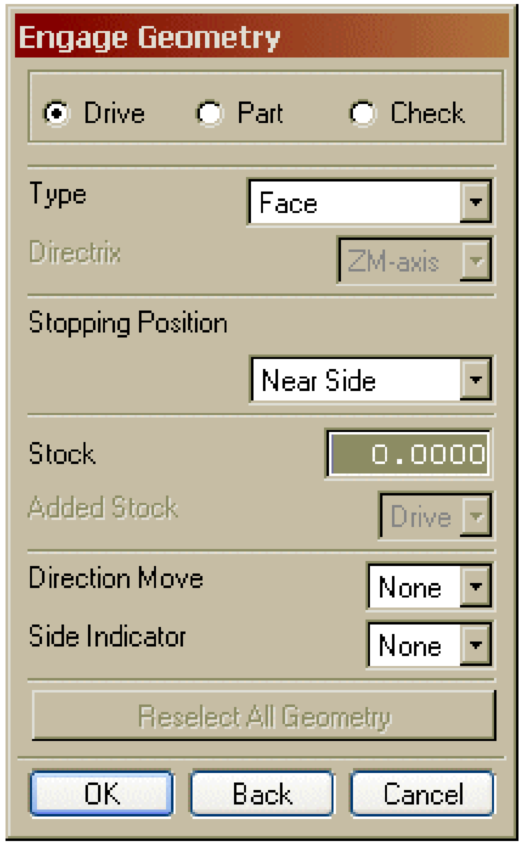


- ❑ Choose **OK.**

- ❑ Choose the Geometry button from the Engage Motion dialog.

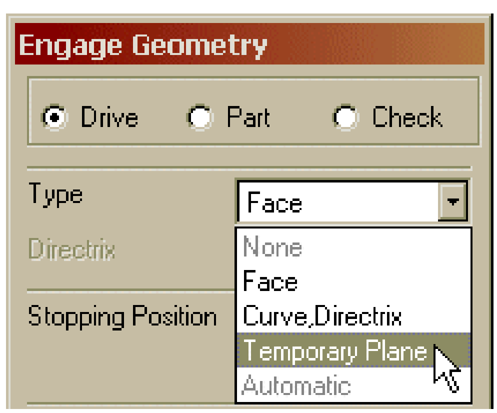


The Engage Geometry dialog is displayed.



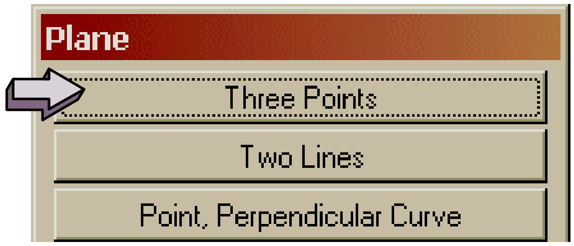
You will first create a temporary check plane as the Drive geometry using the Three Points option for plane creation.

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

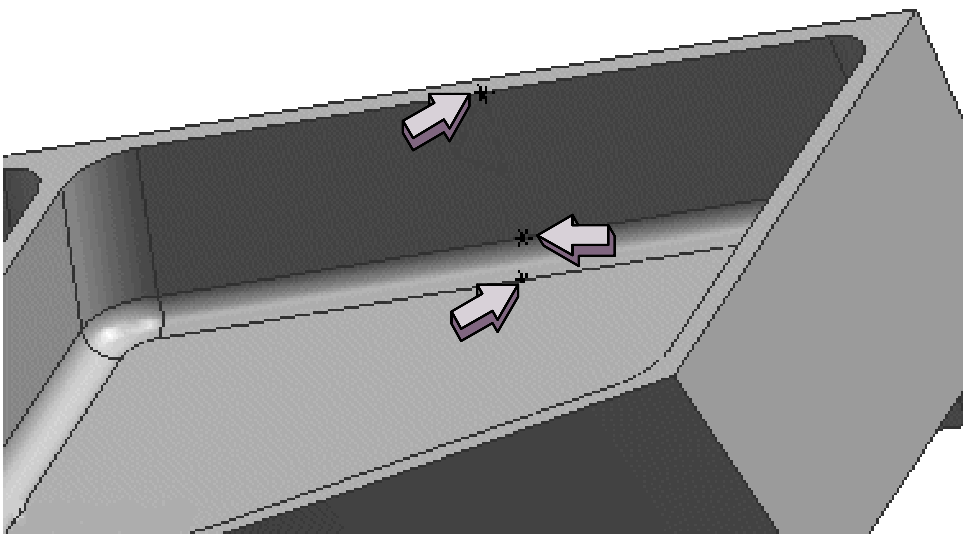


4

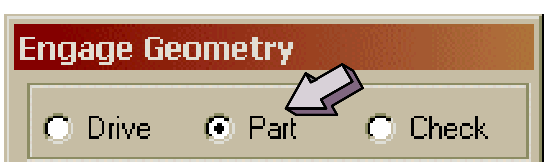
❑ Choose the Three Points option from the Plane dialog.



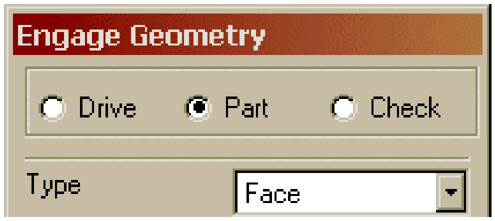
❑ Choose the three control points as shown below. Note that in the following views, the part has been rotated 180 degrees to facilitate looking at the wall being created. The stock and check geometry have been removed from the view for purposes of clarity.



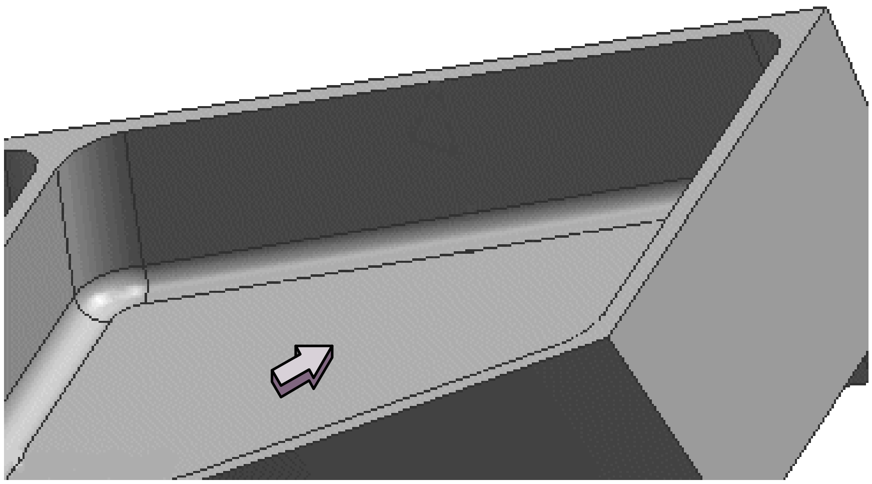
As the last point is selected, the geometry selection advances to Part geometry.



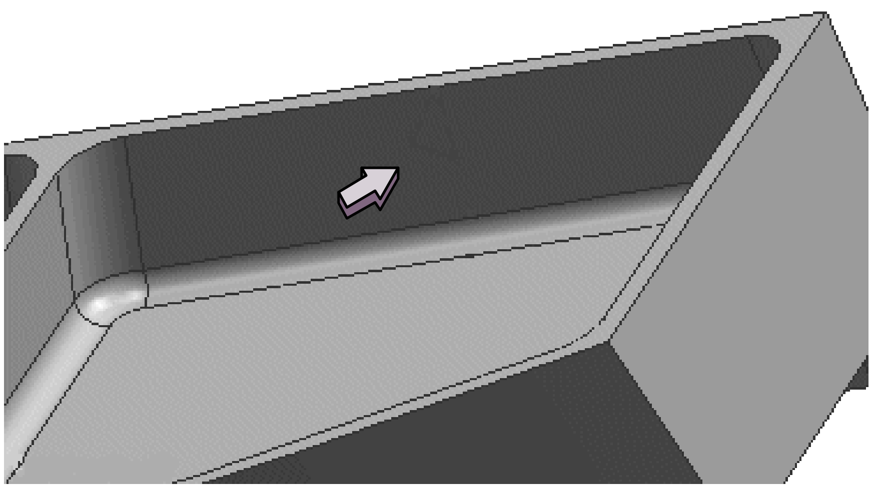
❑ Change the Type back to **Face**.



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

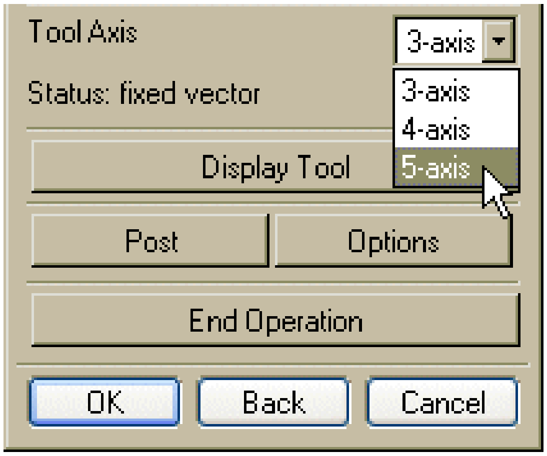


❑ Choose the wall face as the Check geometry.



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 desirable 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.

This part has been designed for manufacturing. 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. However, it does not mean that you can avoid selecting the corner fillets.

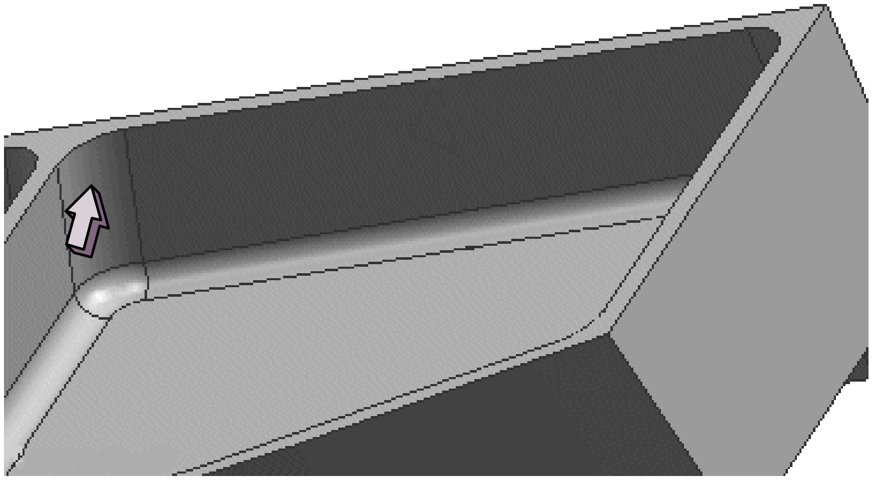
Sequential Mill has defaulted to Continuous Path Motion as the next sub-operation type. You will need to choose the fillet as the next Check surface.

❑ 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**.

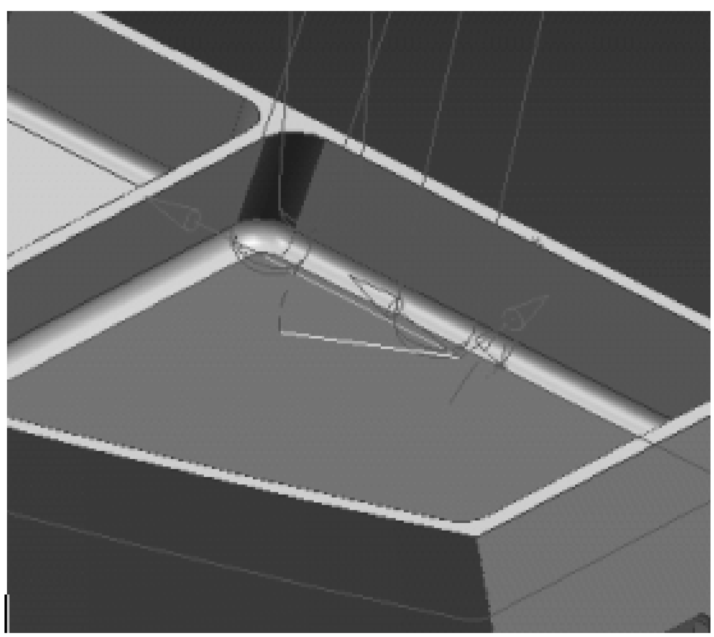
❑ Choose the corner fillet surface as shown.



❑ 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.

4

- ❑ 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.

  This concludes the activity. Do not close the part file. You will be using it in the next activity.

In the previous activity, you finish machined the wall of the pocket. One of the walls of the pocket is at an extreme closed angle. Due to this condition, extra stock was left on that wall.

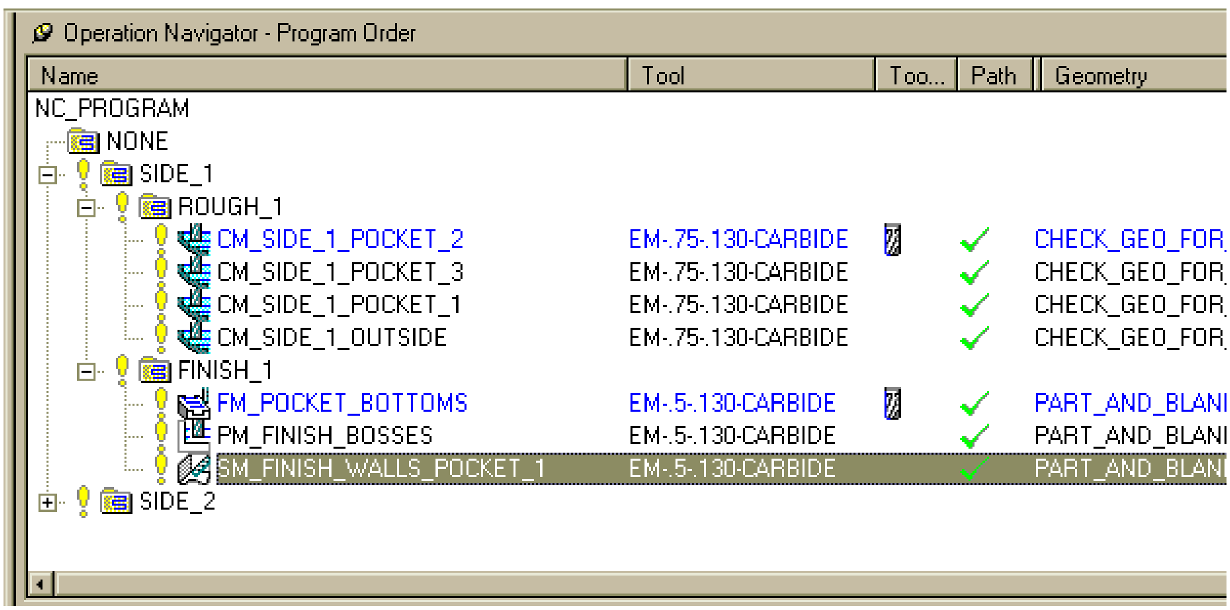
In the next activity, you will use Sequential Mill looping functionality, with five-axis motion, to remove that excess stock.

Activity 4–3: Removing Excess Stock from a Closed Wall

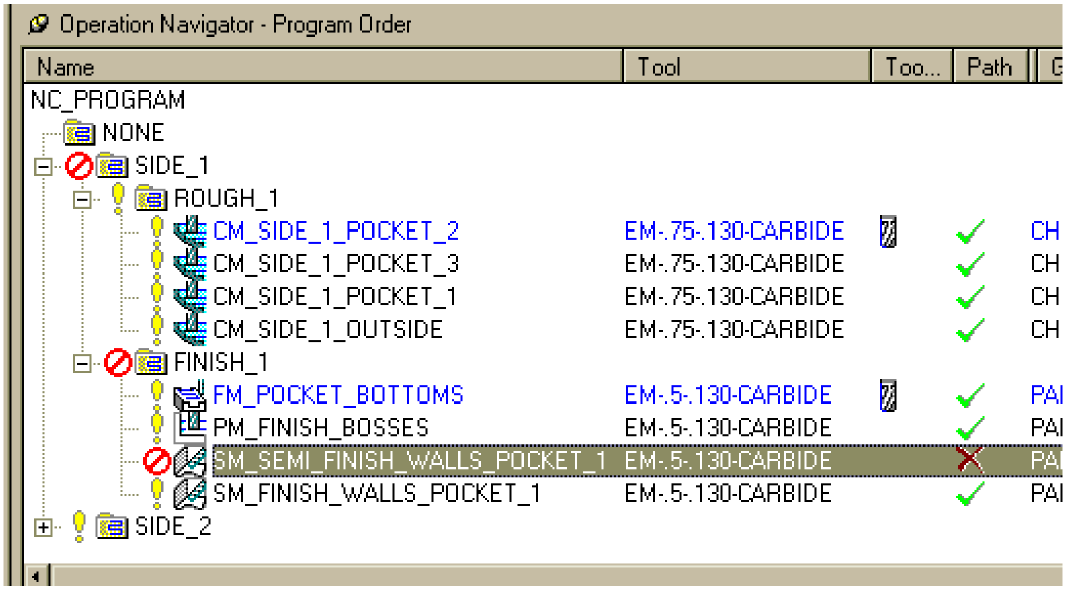
In this activity, you use the looping functionality of Sequential Mill to remove the excess stock on a undercut wall. You will make a copy of the previous operation that you created and modify that operation for doing looping activities.

Step 1 Copy a previous Sequential Mill operation.

- ❑ Continue using ***_**spar_mfg.prt**
- ❑ 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**.

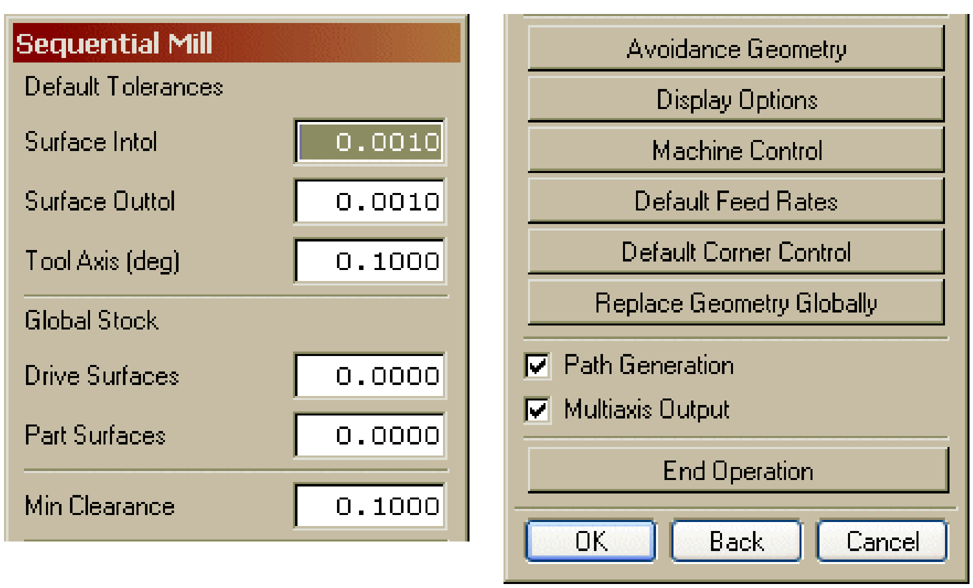


Step 2 Edit the operation.

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.

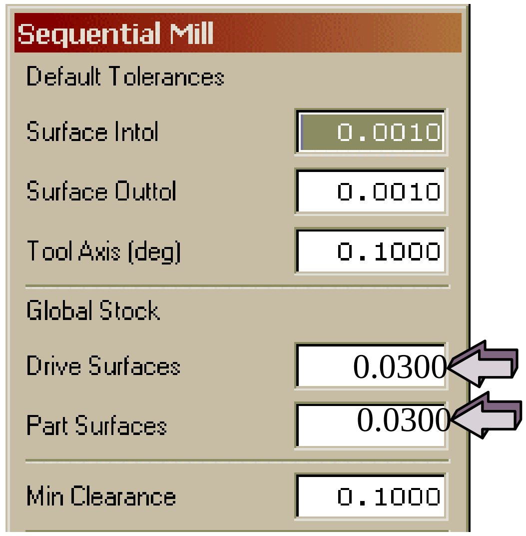
❑ Double click on the **SM_SEMI_FINISH_WALLS_POCKET_1** operation.

The Sequential Mill dialog is displayed.



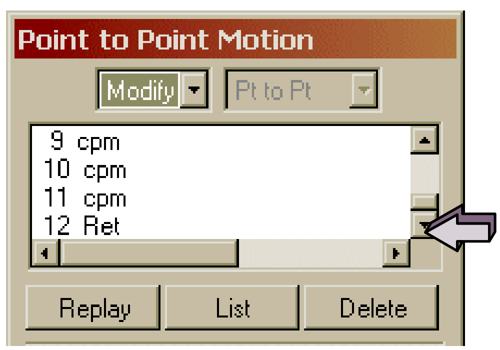
❑ Change the Global Stock on Drive Surfaces to **.030**.

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



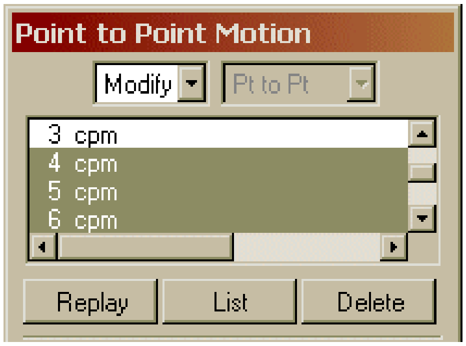
❑ Choose **OK** in the Sequential Milling dialog.

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

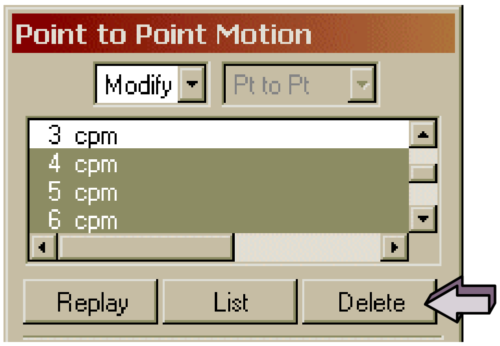


❑ Highlight the **12 Ret** sub-operation.

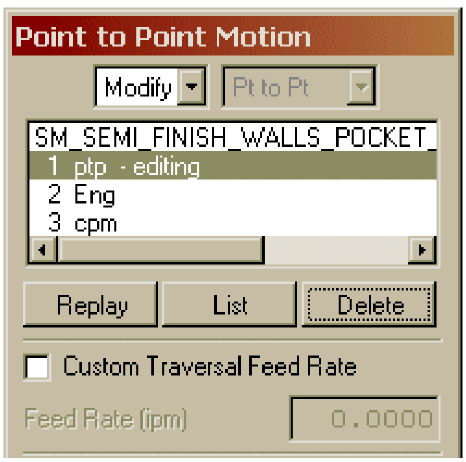
❑ Hold down the shift key, scroll back up in the dialog and choose the **4 cpm** sub-operation.



❑ 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 the one below.



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.

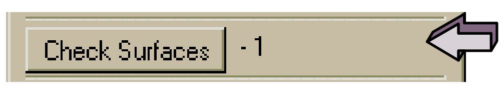
4

Step 3 Edit the sub-operation 3 cpm.

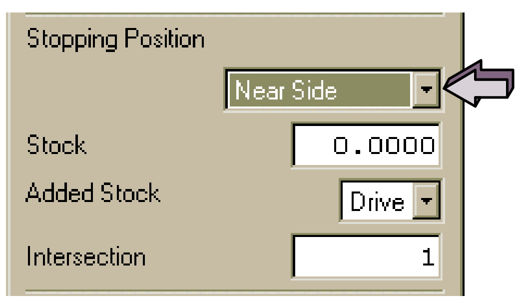
- ❑ Double click on sub-operation 3 in the sub-operation list box.

  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.

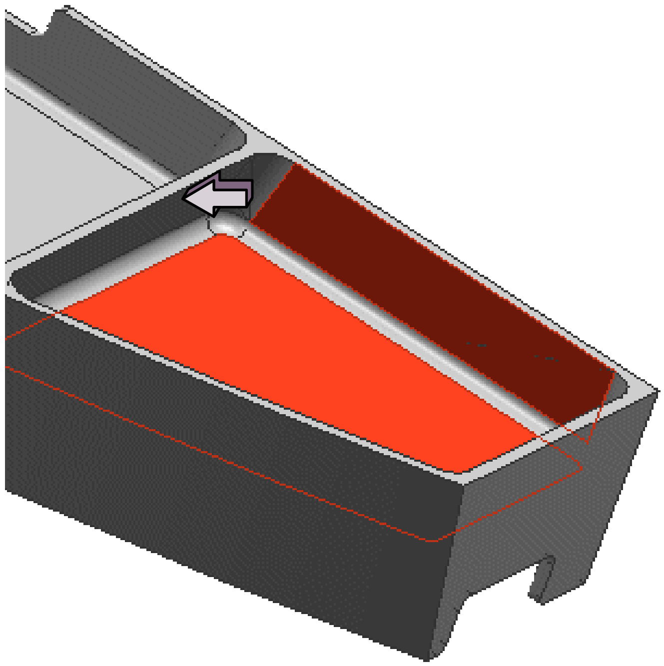


- ❑ Change the Stopping Position to Near Side.



4

- ❑ Choose the undercut wall as shown.



- ❑ Choose **OK** in the Check Surfaces dialog.

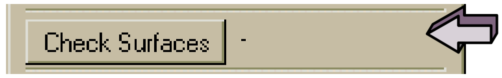
- ❑ Choose **OK** to accept the modified CPM sub-operation.

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

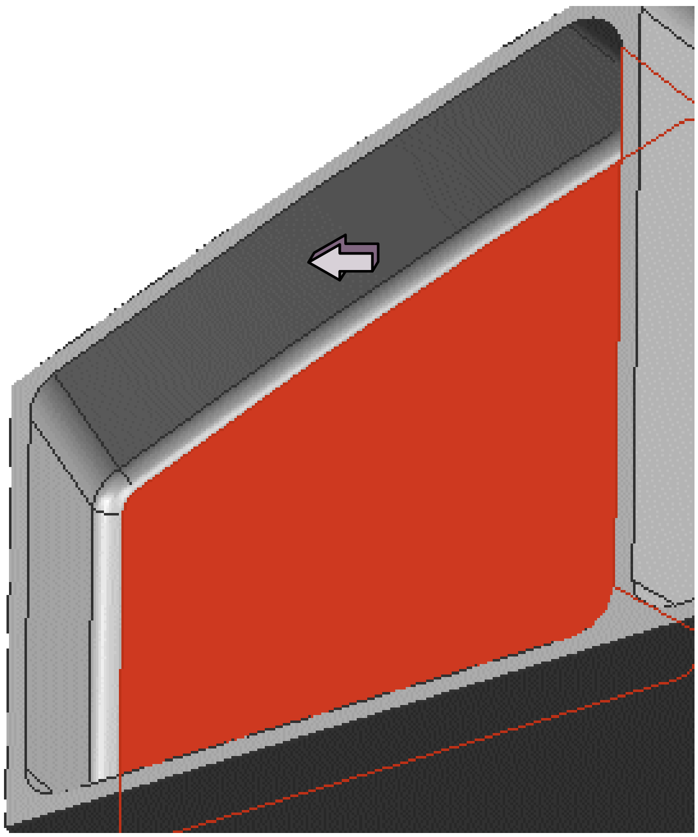
Step 4 Create additional sub-operations.

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.

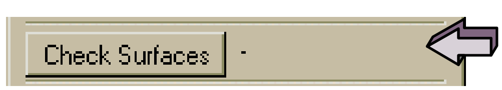


❑ Choose **OK** in the Check Surfaces dialog.

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

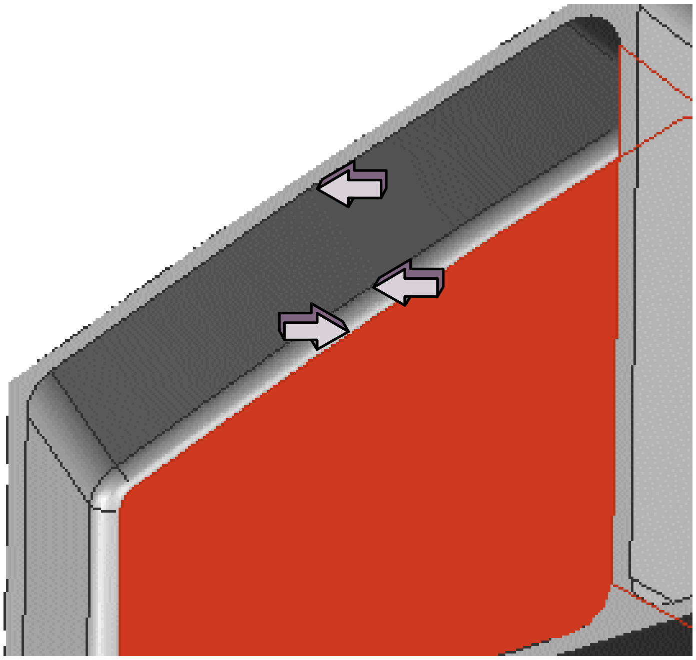
The sub-operation is created. You will now position the cutter to the middle of the Check surface which you previously selected and then will retract the tool.

❑ Choose the **Check Surfaces** button.

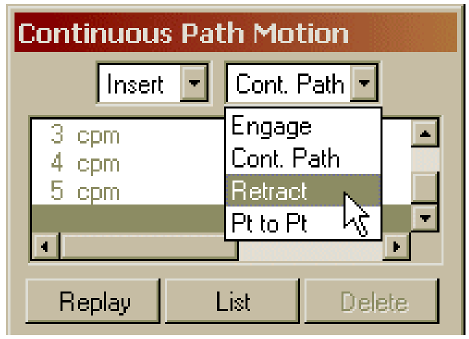


❑ Change the Type to **Temporary Plane**.

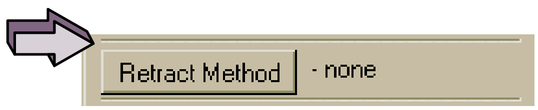
- ❑ Choose the Three Points method.
- ❑ In the Point Constructor dialog, choose the Control Point icon.
- ❑ Choose the three edges in the area as shown below.



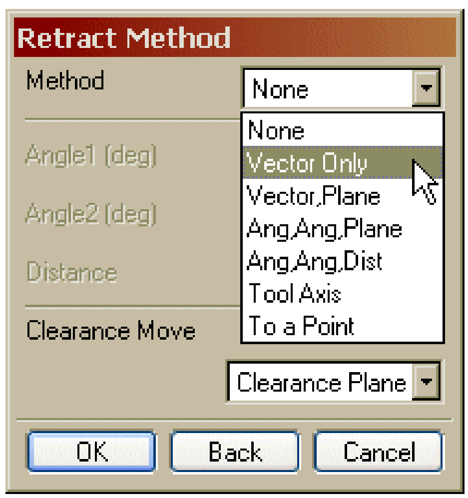
- ❑ Choose **OK** in the Check Surface dialog.
- ❑ Choose **OK** to accept the sub-operation.
- ❑ Change the motion type to **Retract**.



❑ Choose the **Retract Method** button.



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

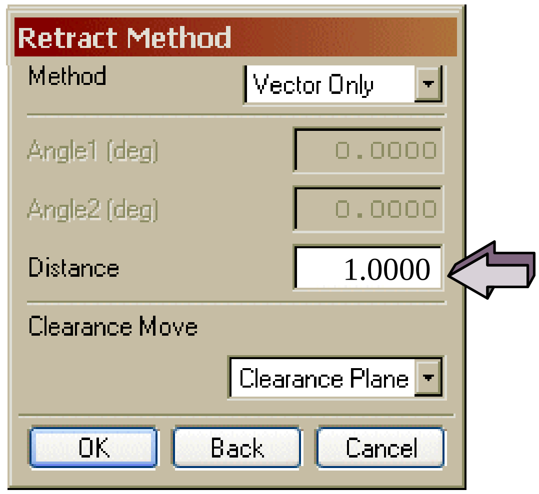


❑ Key in the following values to create the vector:

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.

This concludes this activity. Do not close the part file. You will be using it in the next activity.

In the next activity, you will apply looping to this operation to avoid the removal of excess stock in one operation.

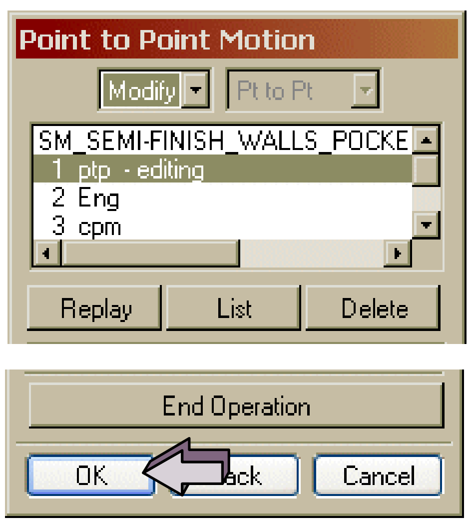
4

Activity 4—4: Using Looping to remove excess stock

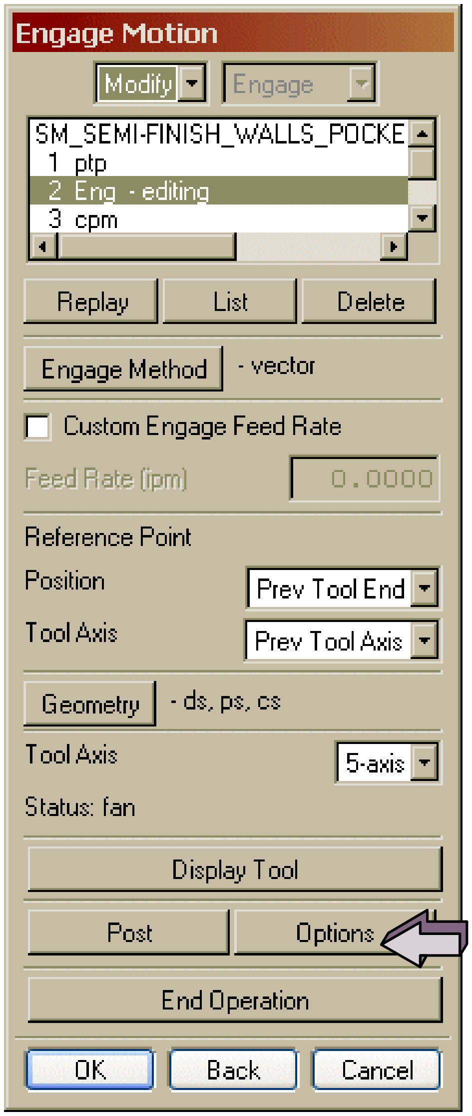
In this activity, you will edit the previous operation to clean-up the stock which remains. You will modify the operation by using the looping option of Sequential Mill, which will create a series of passes for stock removal.

Step 1 Edit the existing operation.

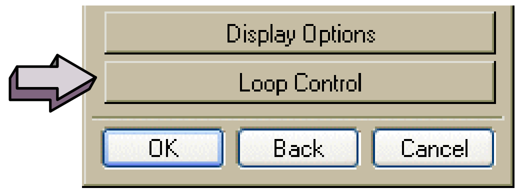
- ❑ Continue using ***_**spar_mfg.prt.**
- ❑ In the Operation Navigator, double click on the **SM_SEMI_FINISH_WALLS_POCKET_1** operation.
- ❑ Choose **OK** in the Sequential Mill dialog.
- ❑ Choose **OK** in the Point to Point Motion dialog to advance to sub-operation 2.



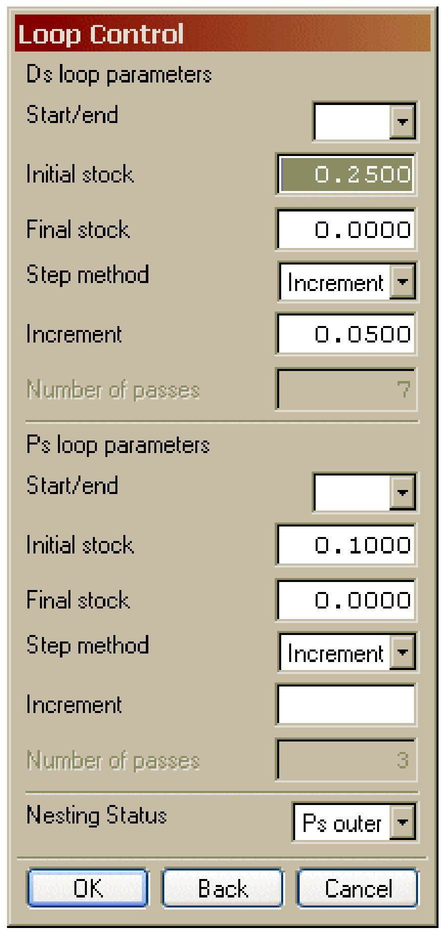
❑ In the Engage Motion dialog, choose the **Options** button.



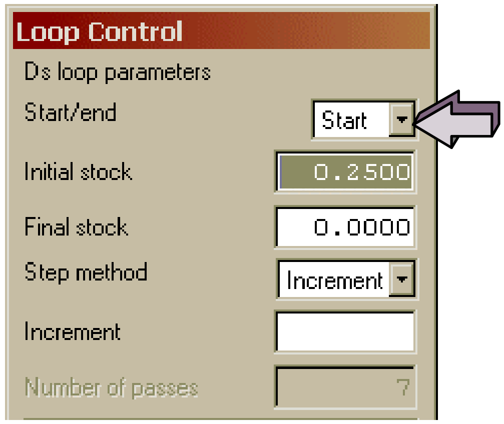
❑ Choose **Loop Control**.



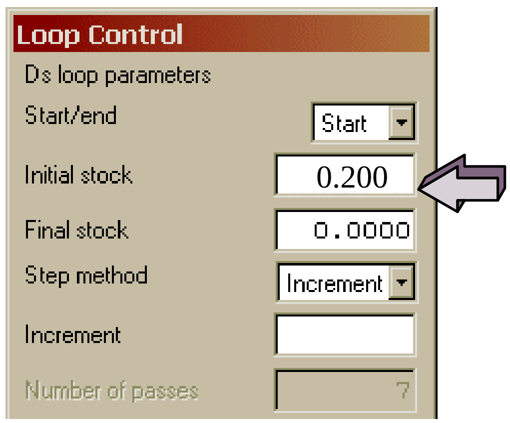
The Loop Control dialog is displayed.



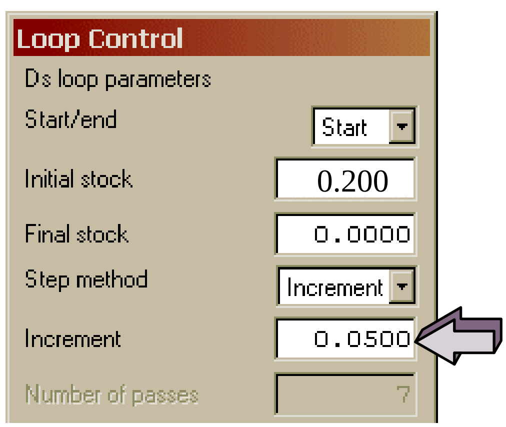
❑ Change the Ds loop parameters Start/End from **None to Start**.



❑ Key in 0.2 in the Initial stock field.



❑ Key in 0.05 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 crate 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** if you so desire.
- ❑ Save and close the part.

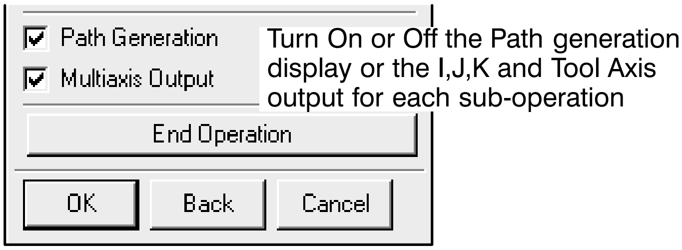
  You have completed this activity.

Additional Sequential Mill Options

The following are Sequential Mill options that you may not have 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.

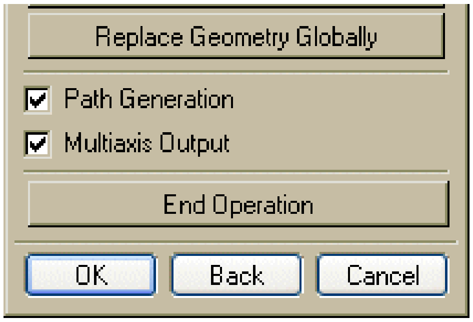


Also, note that the Multiaxis Output is an option. This outputs the I, J, and K components of the tool axis vector with each output point. This option must be active if 4 or 5-axis tool positioning is used. The default is **ON** (box is checked).

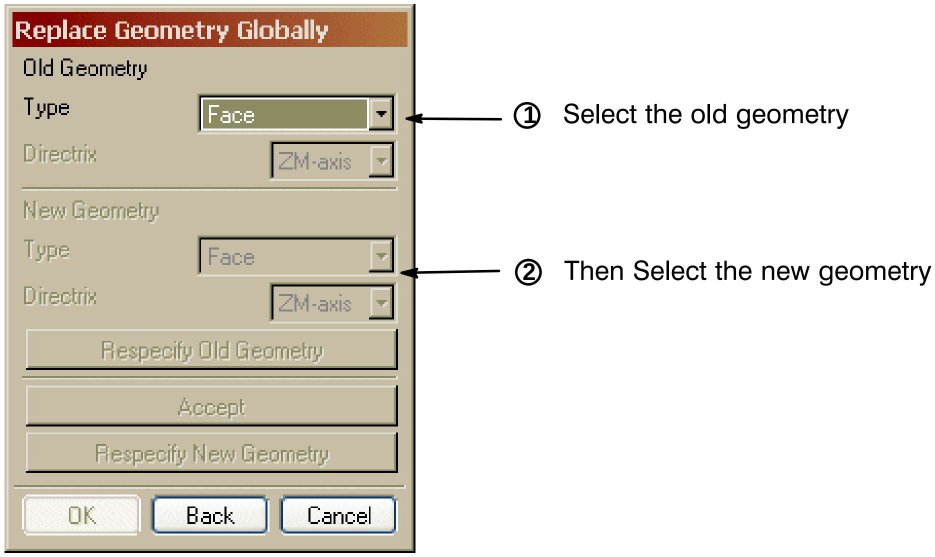
Replace Geometry Globally

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.

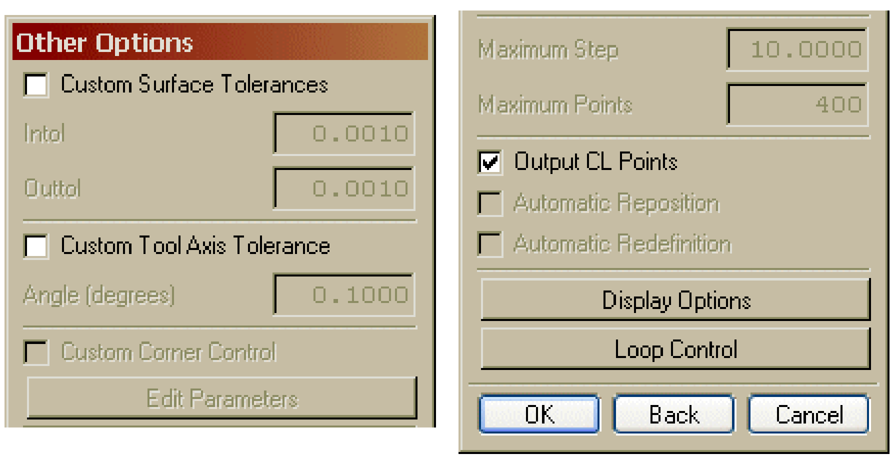


4



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 Tolerance — 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 Feedrate 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.

Sequential Milling Best Practices

Most Sequential Milling processor errors are caused by the tool being out of tolerance to the geometry. The following suggestions are presented to avoid such types of problems:

Engaging:

- use a reference point that is *near* the startup geometry
- when using the Fan tool axis, use **Tangent to Drive** as a better method
- 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.

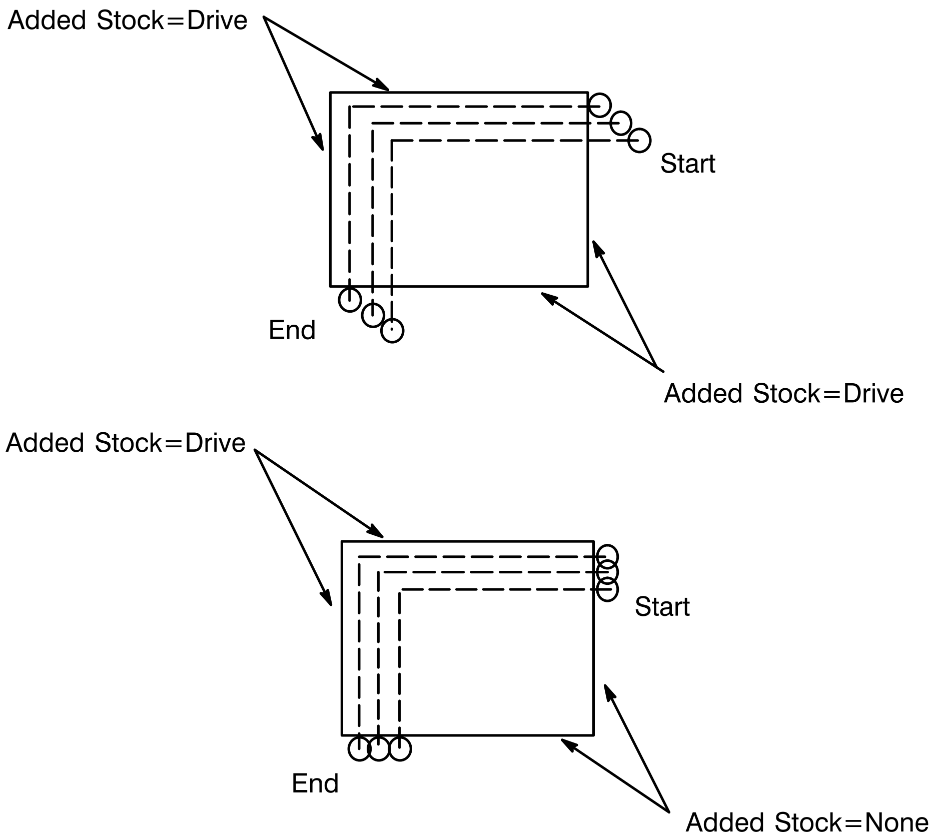
Continuous Path:

- 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 don't 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 illustration.



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 which allows for removal of excess stock
- fanning tool axis control
- complete control of tool positioning

4

Variable Contour — Basics

Lesson 5

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.

OBJECTIVES

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

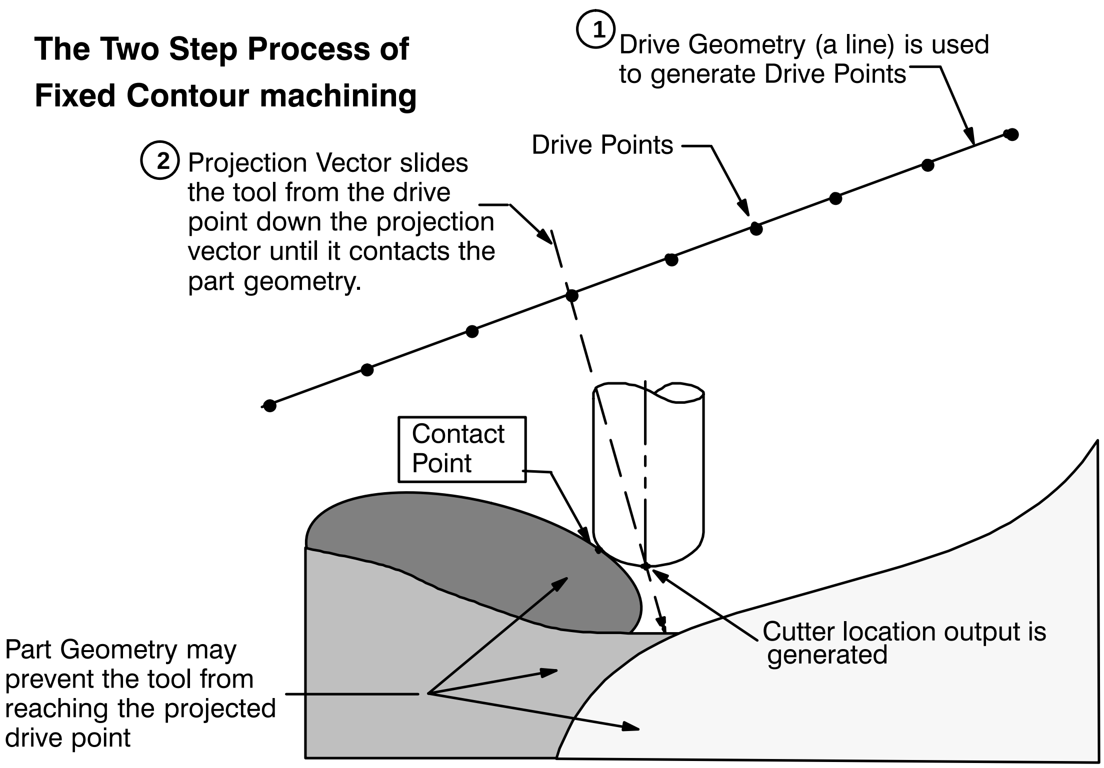
Variable Contour Operations

Variable Contour operations are used to finish areas formed by contoured geometry. Variable Contour tool paths are able to follow complex contours 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 some or all of the part geometry, or can be created from other geometry that is not associated with the part. The points are then projected to the part geometry.

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

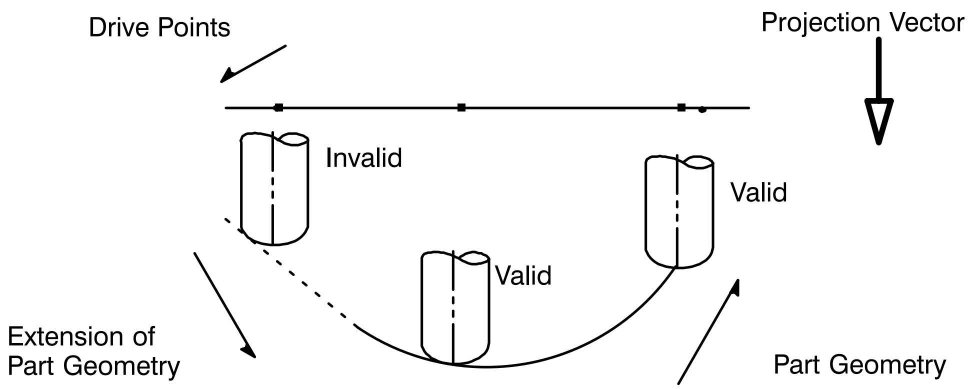


Variable Contour Tool Path Accuracy

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.



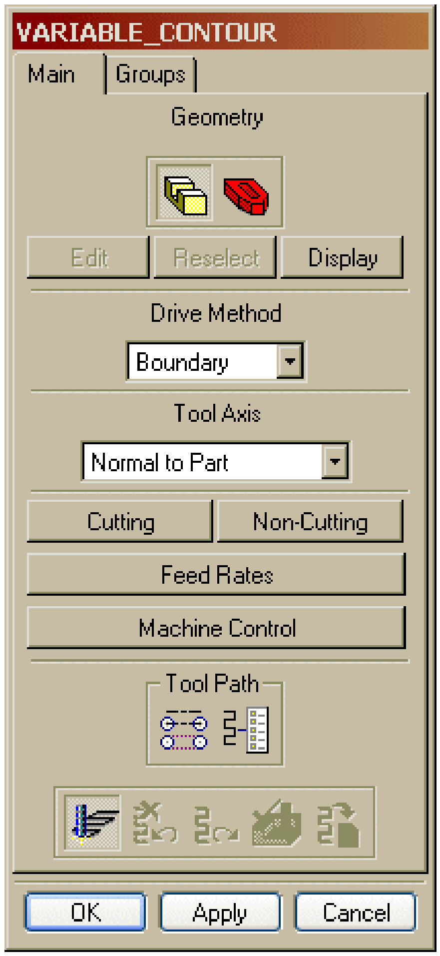
Terminology used in Variable Contour operations

- **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

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

The Variable Contour dialog

The Variable Contour dialog is similar to the Fixed Contour dialog.



The primary difference between the Fixed Contour and Variable Contour dialogs deals with the various methods of tool axis control.

Drive Methods for Variable Contouring

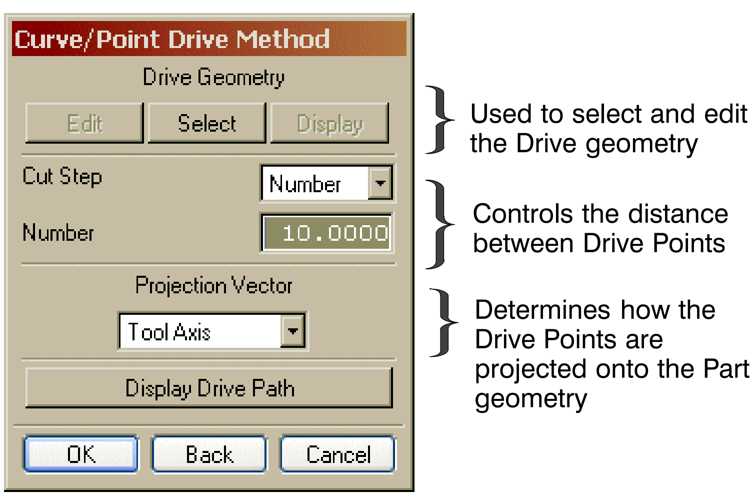
The Drive Method defines the method of creating Drive Points.

Curve/Point Drive Method

The Curve/Point Drive Method allows you to define drive geometry by specifying points and curves. When specifying points, the drive path is created as linear segments between the points. When specifying curves, drive points are generated along the selected curves. In each case, the drive geometry is projected on to the part surface(s) where the tool path is created. The curves may be open or closed, contiguous or noncontiguous, 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.

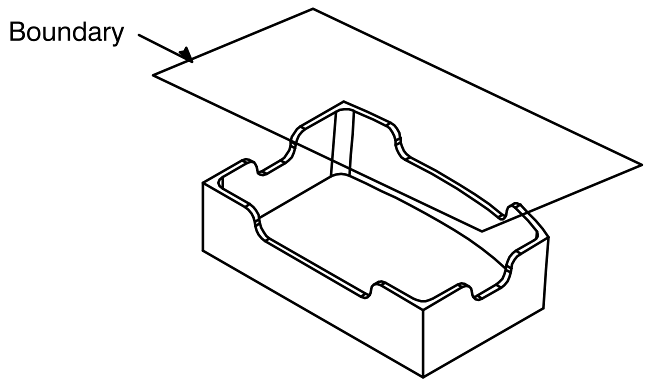


5

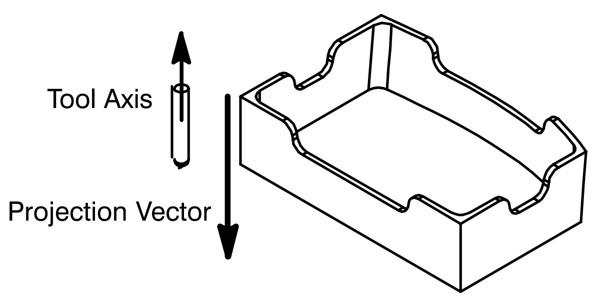
Boundary Drive Method

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.



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

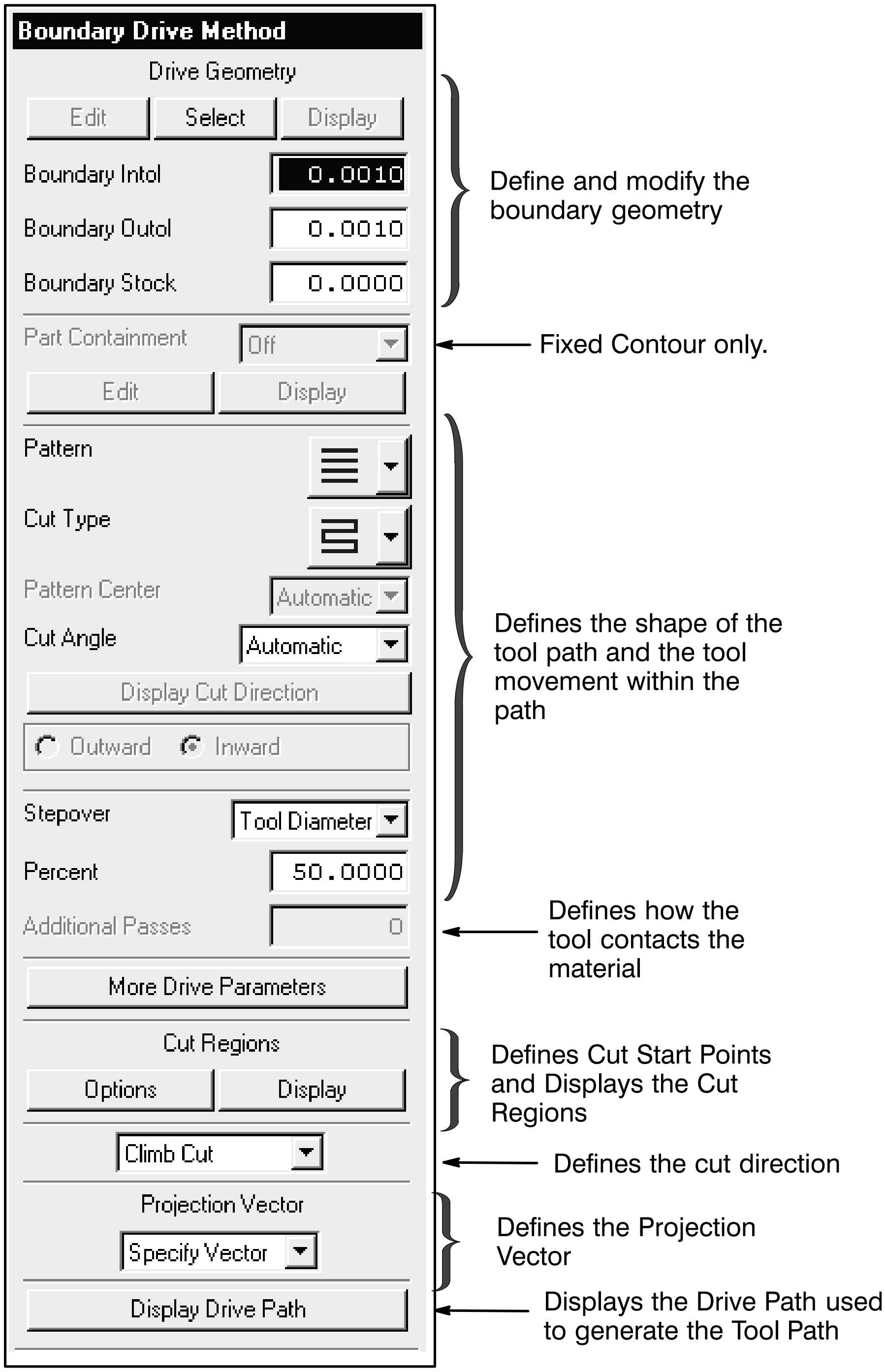


The Boundary Drive Method is preferred to the Surface Area Drive Method. You can quickly create a boundary and tool path without the surface design requirements of the Surface Area Drive Method.

However, the Boundary Drive Method does not allow as many choices of tool axis options that are available in the Surface Area Drive Method.

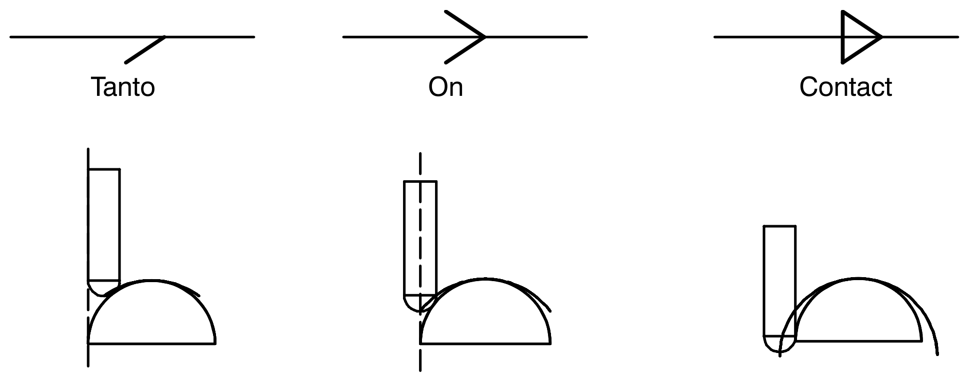
The Boundary method is better suited for roughing operations and the Surface Area Method is better suited for finishing operations.

The Boundary Method dialog, with description of options, follows:



Select allows you to initially define the boundaries. 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.

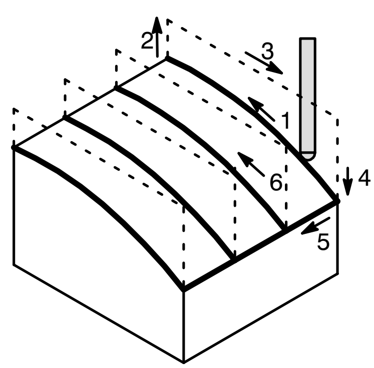
The boundary members graphically represent the associated tool positions as illustrated below:



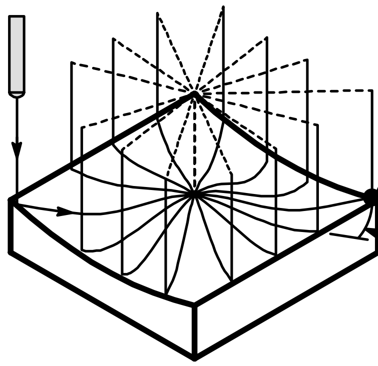
Pattern enables you to define the shape of the tool path. Some patterns cut the entire region, while others cut just around the perimeter of the region. Some patterns follow the shape of the cut region, while others are independent of it.

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

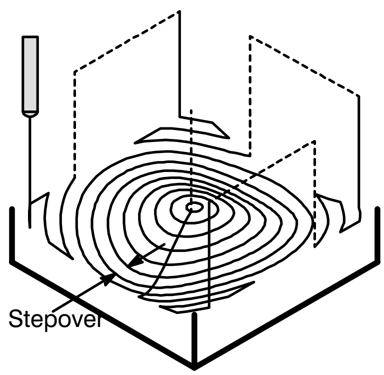
Parallel Lines creates a cut pattern defined by a series of parallel passes. This option requires you to specify a Cut Type of Zig-Zag, Zig, Zig With Contour, or Zig with Stepover and allows you to specify a Cut Angle.



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



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



Cut Type enables you to define how the cutter moves from one cut pass to the next. The options are used in combination with either the 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 you to interactively or automatically define the center point of Concentric Arcs and Radial Lines cut patterns.

Cut Angle determines the angle of rotation for the Parallel Lines Cut Patterns only.

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 allows you to specify 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 allows the system to determine the stepover distance based on the scallop height you enter.

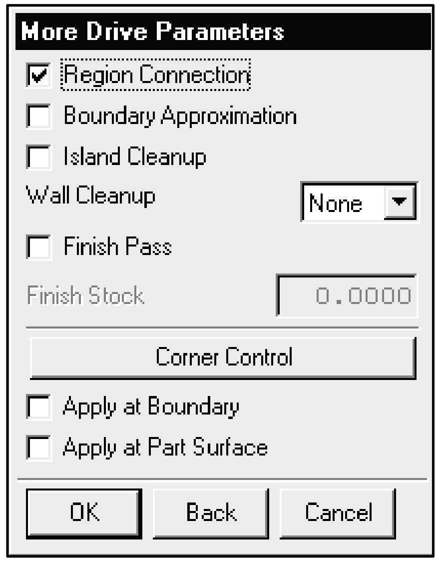
Tool Diameter allows you to define the stepover in terms of a percentage of the effective tool diameter.

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 allows you to define a constant stepover by keying in an angle. This option is used only in combination with the Radial Cut pattern.

Additional Passes enables you to specify 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 graphically displayed when the **Display** button is selected.

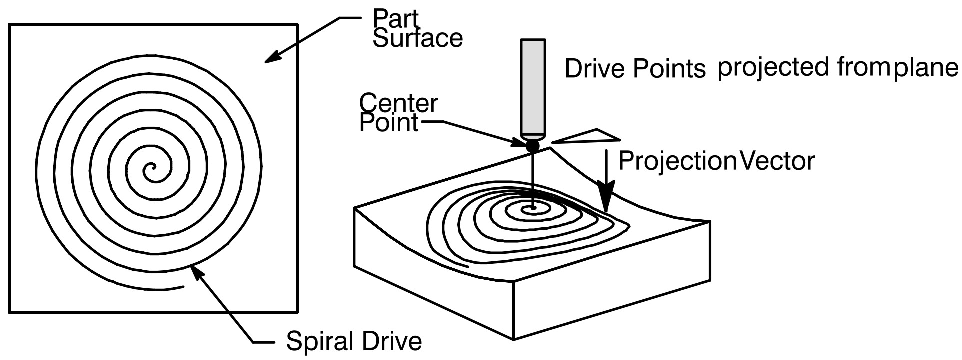
Display generates a temporary screen display of the cut regions for visual reference. The display is generated using the parameters specified under Cut Region Display options.

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

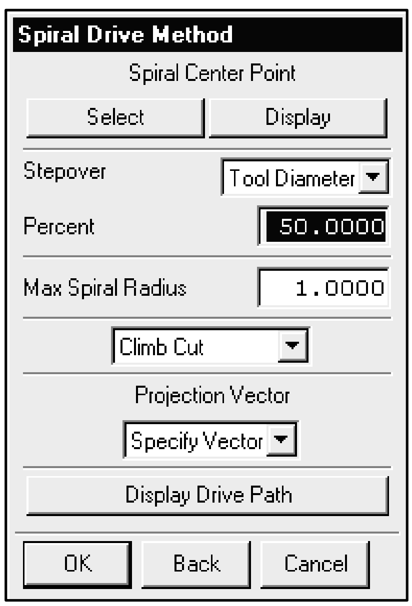
The Spiral Drive Method allows you to define drive points that spiral outward from a specified center point. The drive points are created within the plane normal to the projection vector and contain the center point. The drive points are projected on to the part surfaces along the projection vector.

Unlike other drive methods which require a sudden change in direction to stepover to the next cutting pass, Spiral Drive Method stepovers are a smooth, constant transition outward. Since this drive method maintains a constant cutting motion, it is preferred in high speed machining applications.



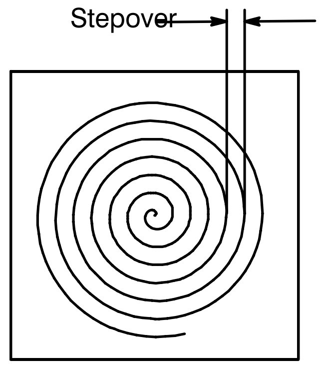
The Center Point defines the center of the spiral and is where the tool begins cutting. If you do not specify a center point, the system uses the (0,0,0) of the Absolute Coordinate System. If the Center Point is not on the part geometry, it follows the defined projection vector to the part geometry.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

The following dialog is displayed when you select the Spiral Drive method:



Stepover allows you to specify the distances between successive cut passes.

5

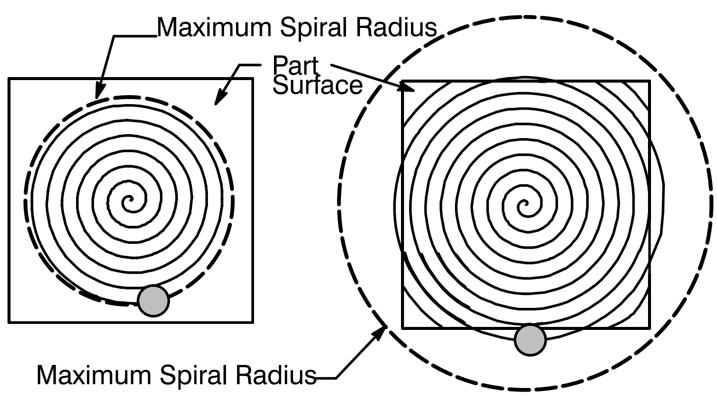


Spiral Drive method Stepovers are a smooth, constant transition outward and do not require an abrupt change of direction.

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.

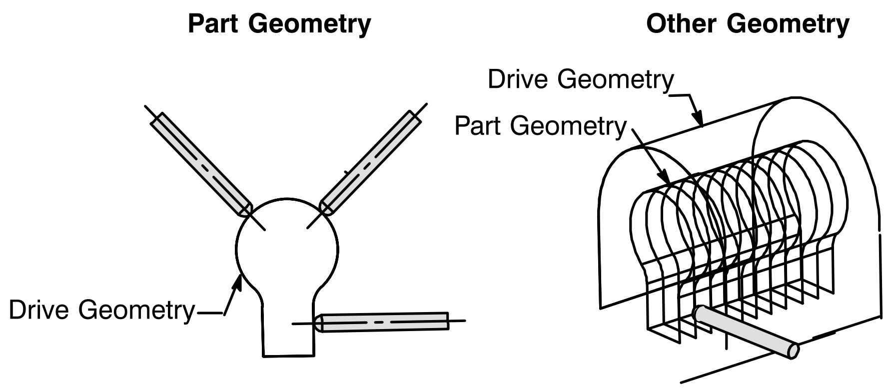


If the specified radius is contained within the part geometry, the center of the tool positions to the radius before retracting. If the specified radius exceeds the part geometry, the tool continues to cut until it can no longer position to the part geometry. The tool then retracts and engages when it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

5

Surface Area Drive Method

The 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.



When you want to generate Drive Points from part geometry, you select the surfaces as drive geometry and do not select any part geometry. The drive points are then generated on the drive geometry and used to create the tool path.

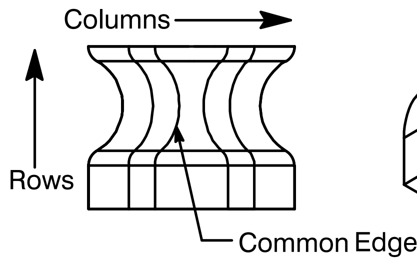
When you want to generate Drive Points from other geometry, you select the drive and part geometry. The Drive Points are then generated on the drive geometry and are projected onto the part geometry according to the Projection vector and are used to create the tool path.

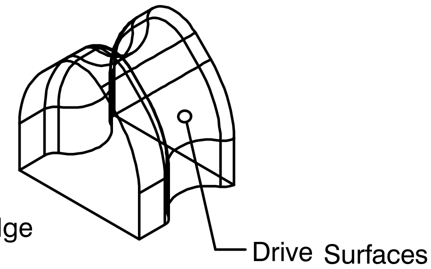
In either case, the tool axis can follow the drive geometry contour.

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.

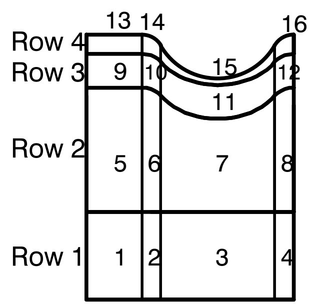
The Surface Area Drive method is much more accurate than its alternative, the Boundary Drive method for complex parts. You will find this method most useful for finishing types of operations.

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

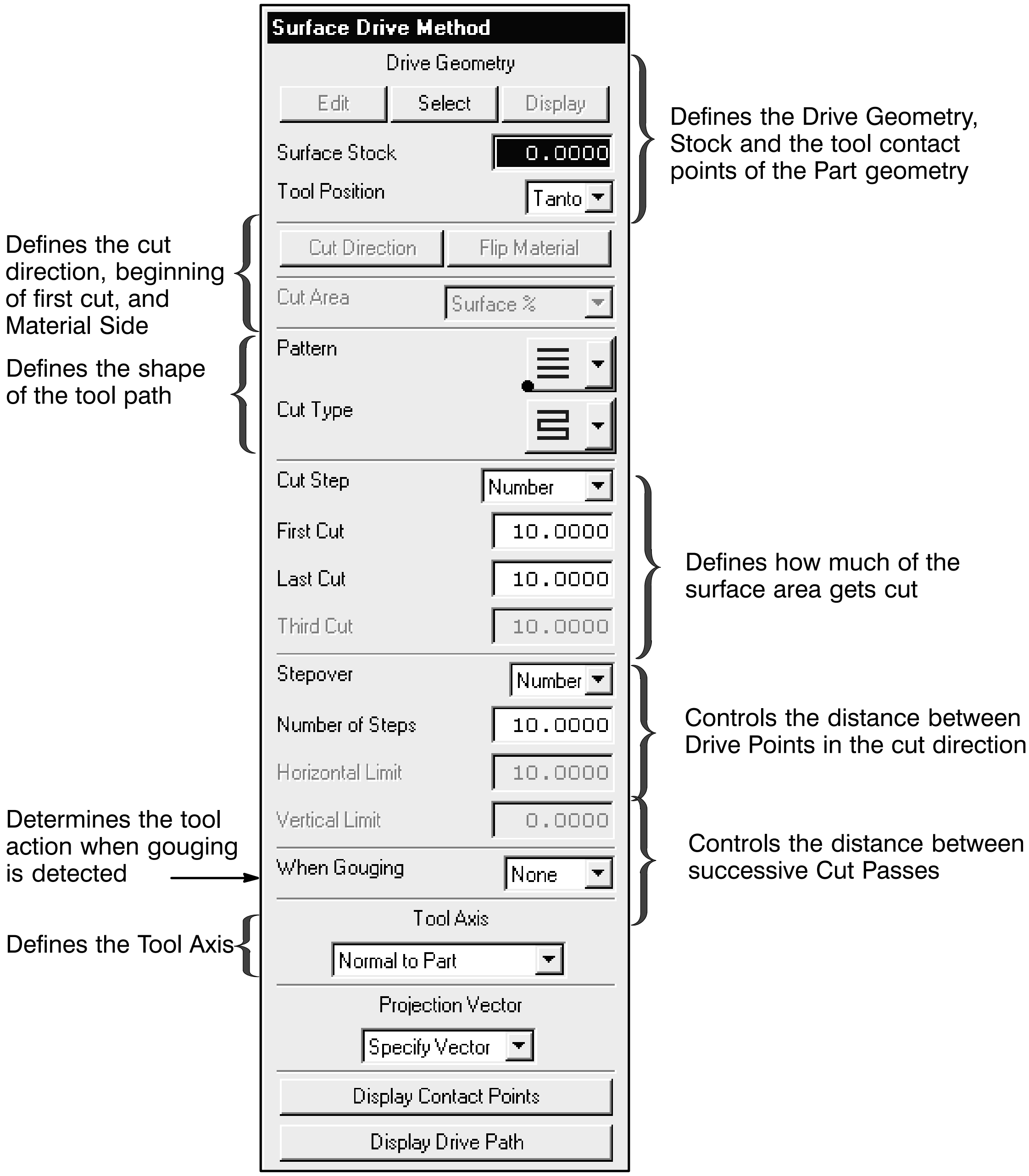




Drive geometries must be selected in an orderly sequence defining the rows.



The Surface Area Drive method dialog follows:



Select allows you to initially define the Drive Geometry.

Note that you can use part geometry as drive geometry.

5

Surface Stock allows you to offset 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 allows you to specify 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 allows you to reverse the direction of the Material Side Vector which determines the side of the surface the tool contacts when machining directly on the drive geometry. When machining on part geometry, the Projection vector determines the Material Side.

Cut Area enables you to define 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 % allows you to specify the drive geometry area to cut by specifying positive or negative percentage values for the beginning and end of first and last pass, and the first and last Stepover.

Diagonal Points allows you to use the cursor to indicate two diagonal points defining the area.

Pattern enables you to define the shape of the tool path as Follow Pocket or Parallel Lines.

Cut Type is used in combination with the Parallel Lines pattern to define how the cutter moves 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.

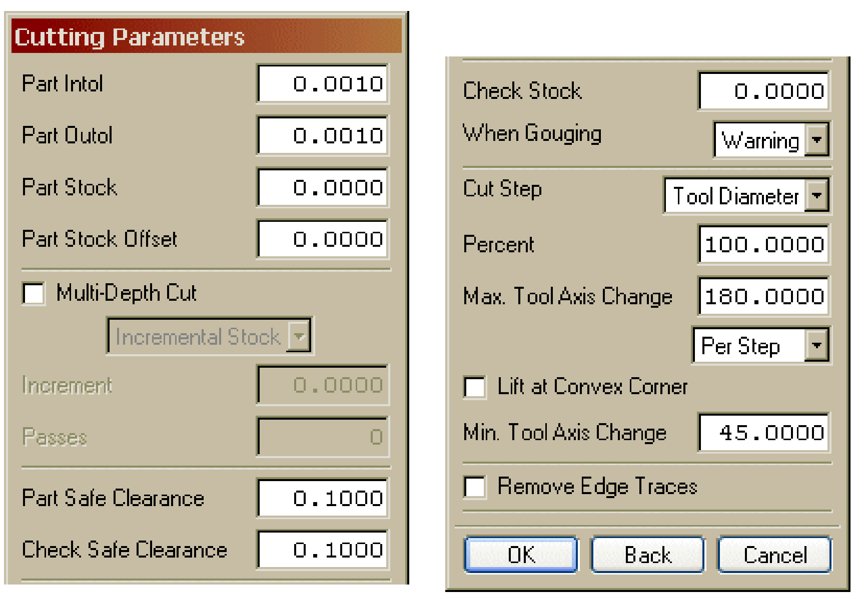
Stepover controls the distance between successive cut passes.

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 helps you avoid 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:



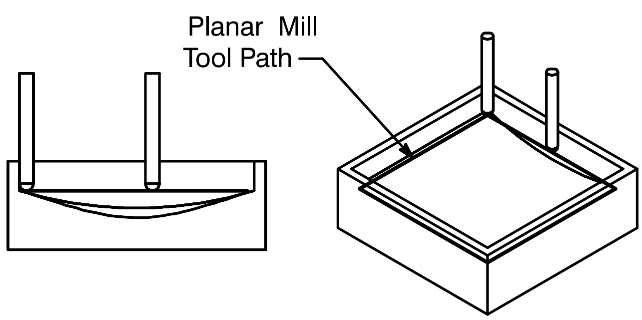
The **Part Stock** option defines an envelope of material surrounding the part geometry which is to remain on the part after machining. The stock specified only applies to those part entities which do not have Custom Stock specified (under Custom Data in the Part geometry dialog).

The **Check Stock** option is an envelope of material surrounding the Check geometry, which the tool will not gouge.

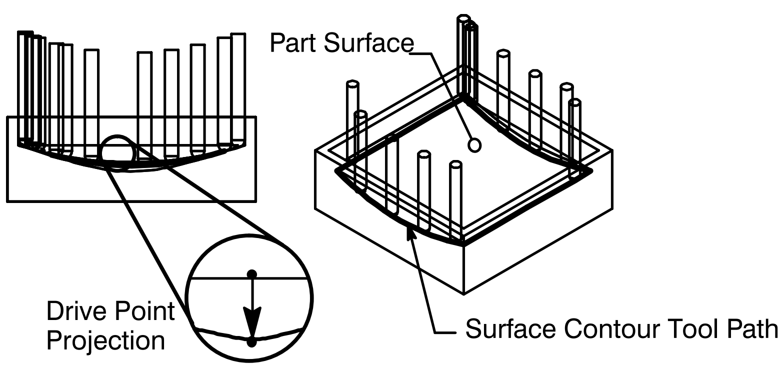
Tool Path Drive Method

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.

In the following illustration, a tool path was created using the Planar Mill, Profile cut type. This tool path can be used by the Tool Path Drive method to generate a new tool path that follows the contours of the part geometry.



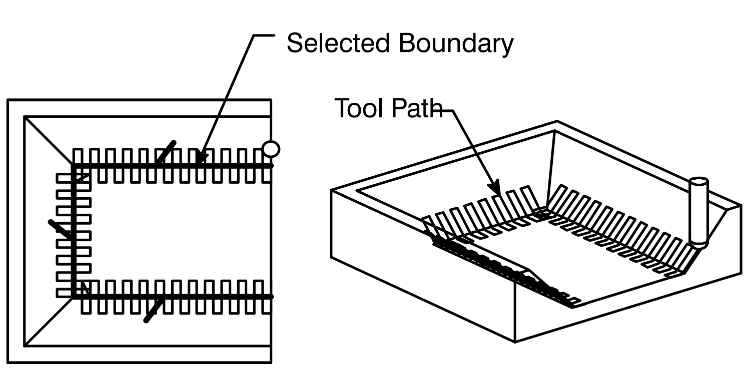
In the following illustration, the results of using the Tool Path Drive method is shown. The tool path in the Planar Mill operation has been projected on to the contoured part geometry in the direction of the project vector to create the Variable Contour tool path.



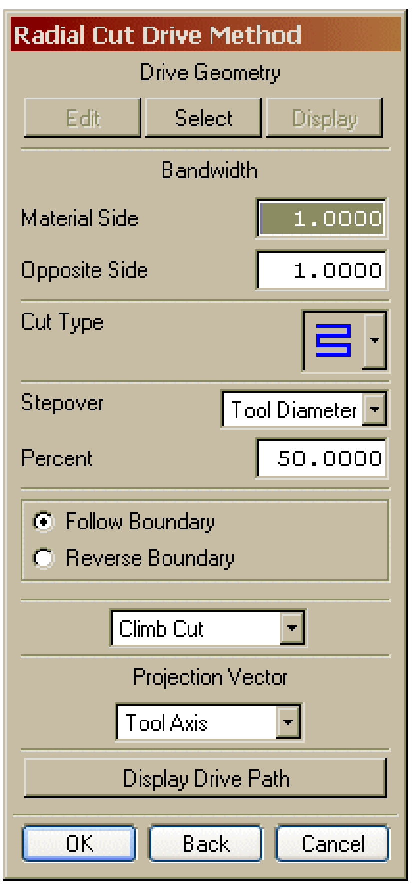
When you select Tool Path as the drive method, you must specify an existing CLSF to be used to generate drive points.

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.



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 dialog displays Radial Cut Drive Method options:

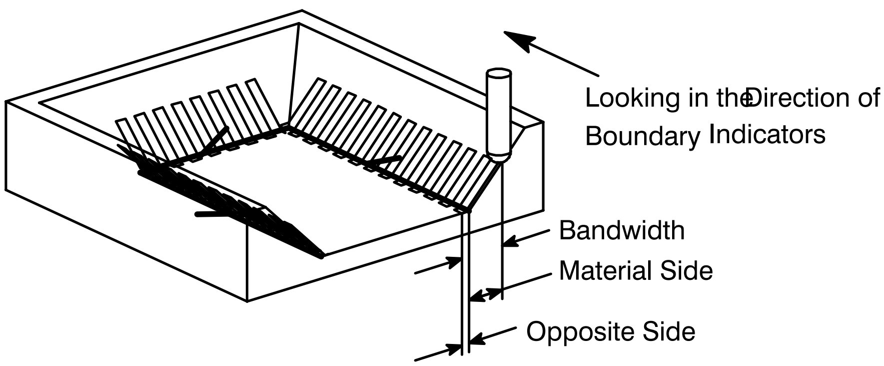


5

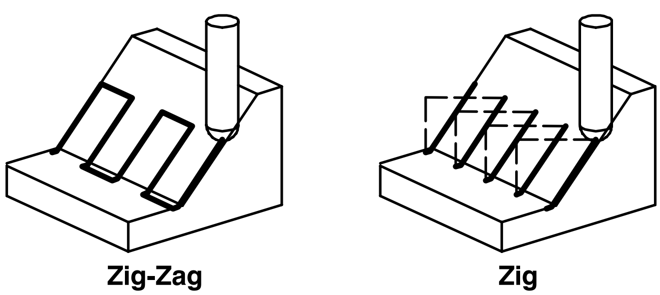
Select displays the *Permanent Boundary* or *Temporary Boundary* dialog allowing you to define the area to be cut. The Permanent Boundary dialog is only displayed if permanent boundaries currently exist. If multiple boundaries are defined, a lift is applied, allowing the tool to traverse from one boundary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



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



Stepover allows you to specify the distances between successive drive paths.

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

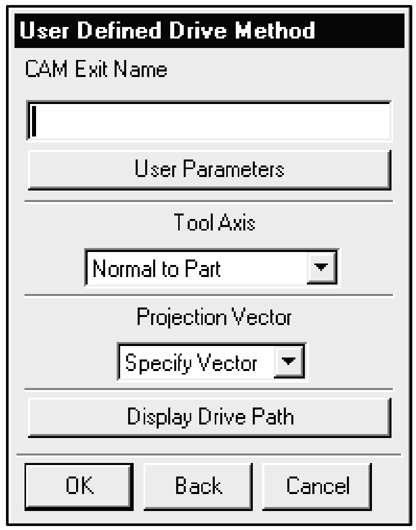
Follow Boundary and **Reverse Boundary** allow you to determine the direction the tool travels along the boundary.

User Function Drive Method

The 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.

The User Function menu option displays the following dialog:

Defines the operating system environment variable which contains the path name of the shared library containing the User Function Program



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

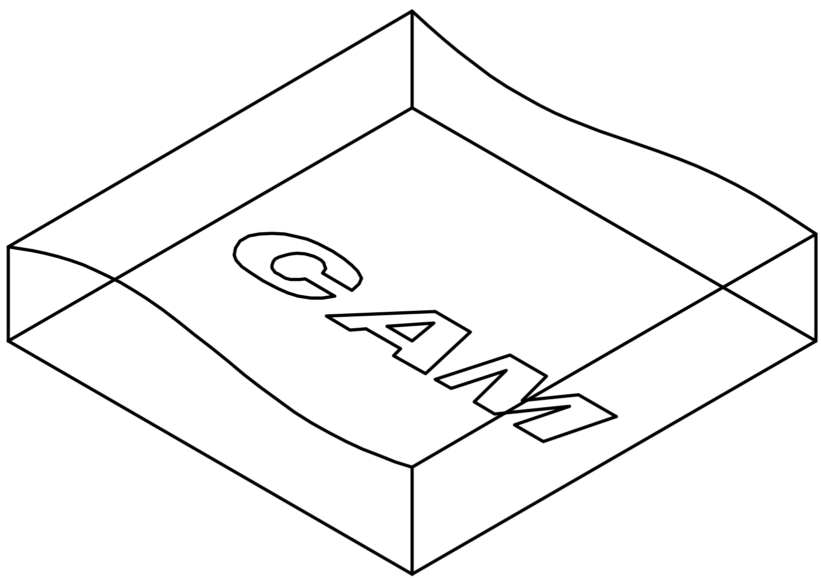
Users Parameters enables you to access a user exit specifying parameters for the drive path. The User Function program associates these parameters with the calling operation, using the name of the operation as the link.

Activity 5—1: Overview of Variable Contour Options

In this activity you will review the major options you are already familiar with in creating Fixed Contour operations. 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. The purpose of this activity is to review the basic methods that Variable Contour uses to create tool paths.

Step 1 Open an existing part file.

❑ Open the part file **mam_vx_0.prt**.



❑ Enter the **Manufacturing application** by choosing **Application → Manufacturing.**

❑ Select the Operation Navigator tab from the tool bar.

The Operation Navigator and the part model are displayed.

Step 2 Review an existing operation.

You will review the options by examining their settings.

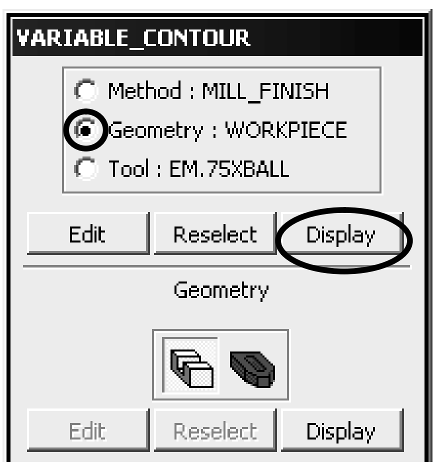
❑ In the Operation Navigator, expand the Program named **OVERVIEW** and double click on the operation named **REVIEW**.

The Variable Contour dialog is displayed.

Step 3 View the Variable Contour dialog options.

First you will review the option settings on the primary 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.

❑ At the top of the Variable_Contour dialog, choose **Geometry** and then select **Display**.



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

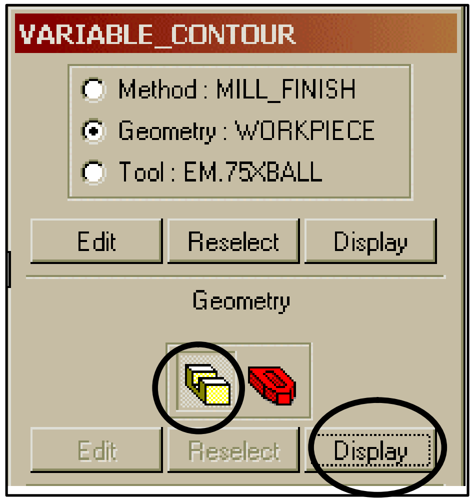
❑ **Refresh** the graphics window (Hint: Use the F5 key).

❑ Under the Geometry label, note that the Part icon is already chosen, and choose **Display**.

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

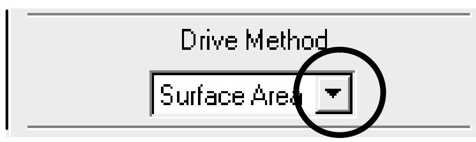
5

❑ Choose the **Check** icon.



Note that no previous Check geometry was selected.

❑ Under the Drive Method label, choose the arrow and view the Drive Methods that are available.



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

❑ Release the menu list and return to the Variable Contour dialog.

❑ On the Variable Contour dialog, under the Tool Axis label, choose the arrow and note the various tool axes which are available.

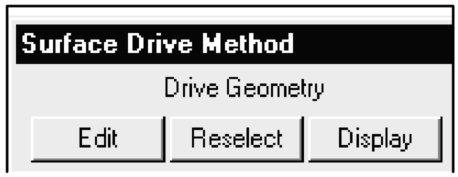
There are numerous choices of tool axes.

Step 4 View the Surface Area Drive Method settings.

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

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

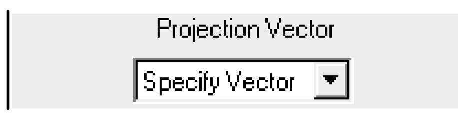
The Surface Drive Method dialog is displayed.



❑ 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 as specified by the Projection Vector.

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



Note that 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 for this part, is also their current location.

❑ Choose **Cancel**.

❑ Choose **Display Drive Path**.

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

❑ Choose **Display Contact Points.**

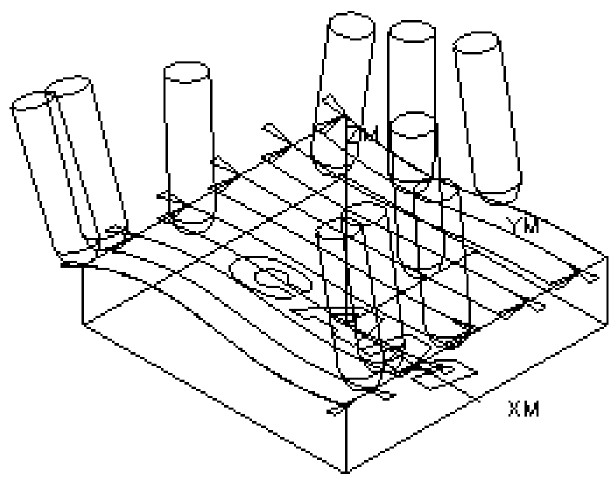
Note the surface normals are displayed at each tool contact point. The Surface Area Drive Method is the only Drive Method that allows you to display contact points.

❑ 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.

This completes this activity.

Tool Axis Control

The Variable Contour Tool Axes can be grouped with respect to the geometry that determines the tool axis.

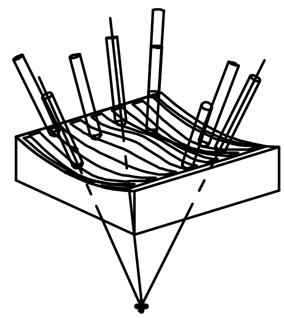
The choice of tool axis depends upon the Drive Method you choose. For instance, the Surface Area Drive Method allows you to specify many 4 and 5 axis tool positions that are not available by using any other Drive Method.

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
Normal To Part	x	x	x	x	x	x
Relative To Part	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
Interpolate	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	

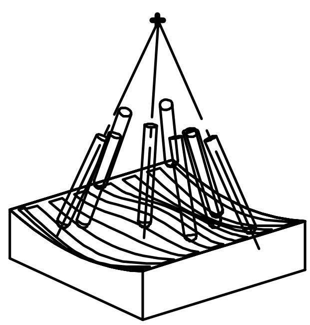
Point and Line Tool Axes

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

- Away From Point

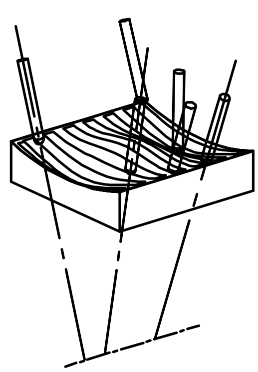


- Towards Point

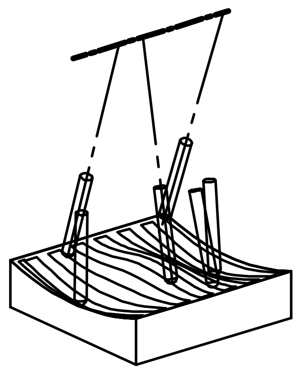


The following tool axis types use focal lines and can produce 4-axis movements:

- Away From Line



- Towards Line



Away and *Towards* refers to a vector direction.

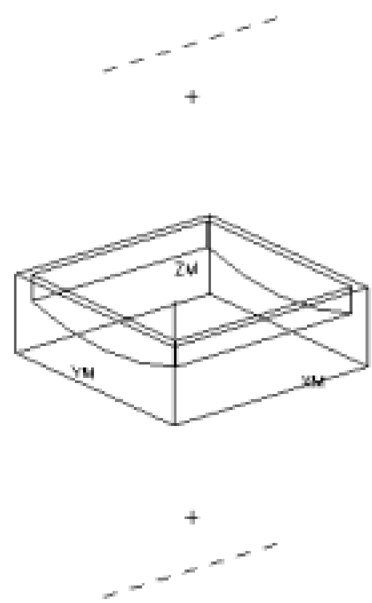
Note that consideration must be given to machine configuration, part fixturing and amount of swing or tilt of the table and or head when selecting tool axis types. It is advisable to select the method which minimizes the amount of table and or head tilt.

Activity 5—2: Point and Line Tool Axes 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.prt**.



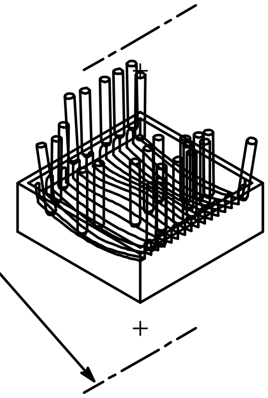
❑ If necessary, choose **Application** → **Manufacturing** and then select the Operation Navigator tab from the tool bar.

The Operation Navigator is displayed. There are four operations representing tool axis types with respect to lines and points.

Step 2 Replay the operations.

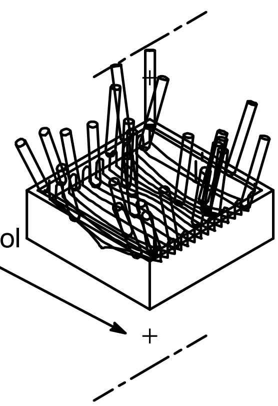
❑ **Edit** the **AWAYLINE** operation**.**

❑ **Replay** the tool path.



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

- ❑ **Cancel** the operation.
- ❑ **Edit** the **AWAYPT** operation**.**
- ❑ **Replay** the tool path.



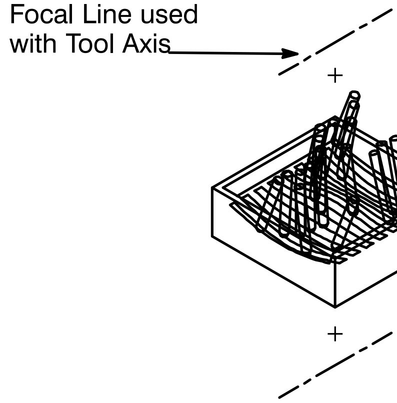
The tool path is replayed using the tool axis option **Away from Point.**

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.

- ❑ **Cancel** the operation.
- ❑ **Edit** the **TOWARDLINE** operation**.**

5

❑ **Replay** the tool path.

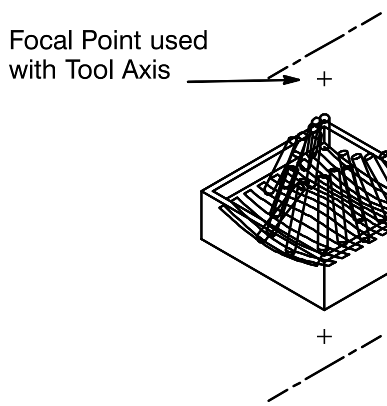


The tool path is replayed using the tool axis option **Towards Line.**

❑ **Cancel** the operation

❑ **Edit** the **TOWARDPT** operation**.**

❑ **Replay** the tool path.



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

Again, notice the difference in the amount of tool tilt. The method chosen, **towards** or **away** from a **point** or **line**, along with the placement of the **point** or **line** with respect to the geometry which you are cutting, gives you precise control of the tilt of the tool.

❑ **Cancel** the operation.

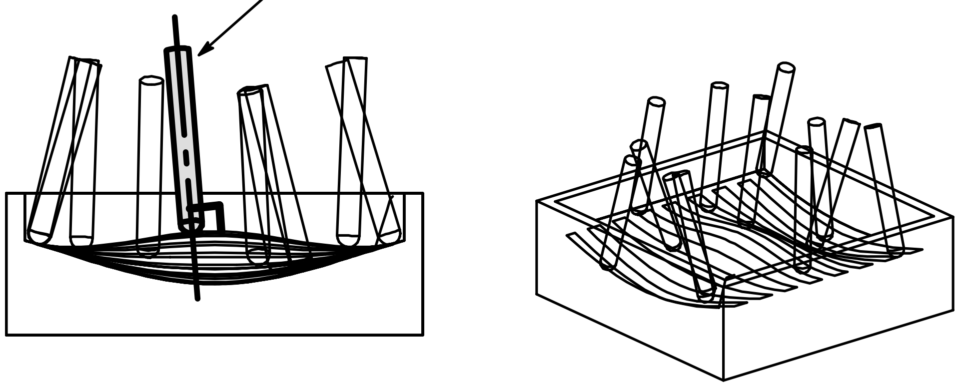
❑ **Close** the part file.

This completes this activity.

5

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 an ideal method of tool axis control when the contoured geometry that is being machined does not change drastically in shape and or direction.

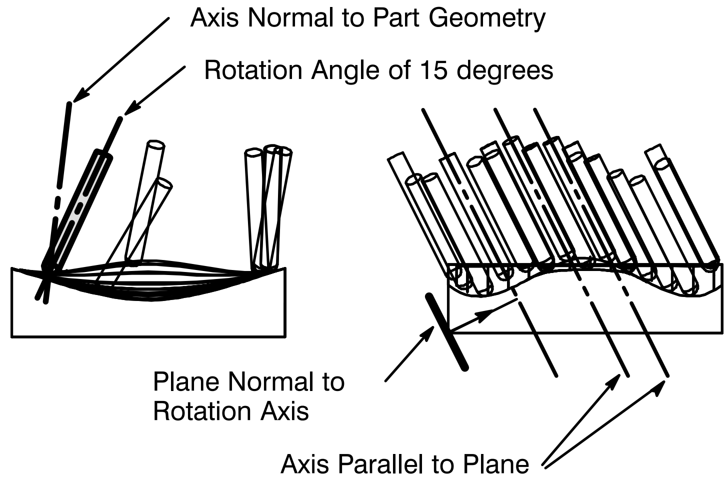


The following tool axis types use the Normal tool axis:

- 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.



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.

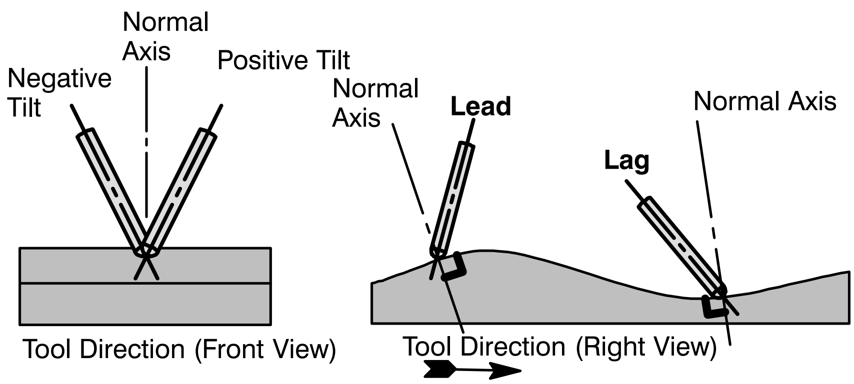
You can apply Lead or Tilt to the following tool axis types:

- 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.

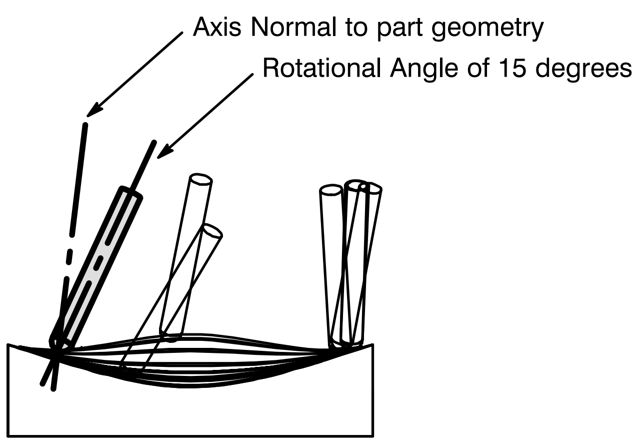
Tilt Angle defines the angle of the tool, side to side. A positive value tilts the tool to the right as you look in the direction of cut. A negative value tilts the tool to the left.



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. It is independent of the direction of the tool movement.

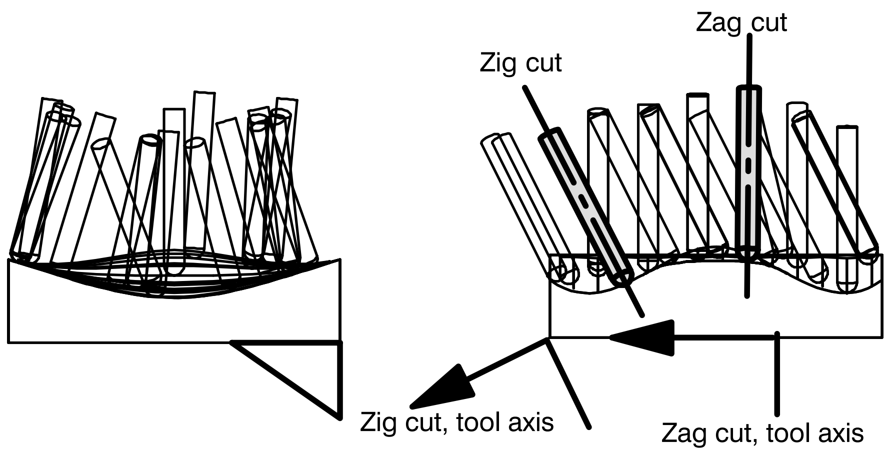
Notice that the rotational angle causes the tool axis to lean to the right of the part geometry normal axis in both the zig and zag moves. The tool moves within planes normal to the defined rotational axis.



Dual 4-Axis

Dual 4-Axis applies rotational and 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 effectively 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.



5

Activity 5–3: 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.

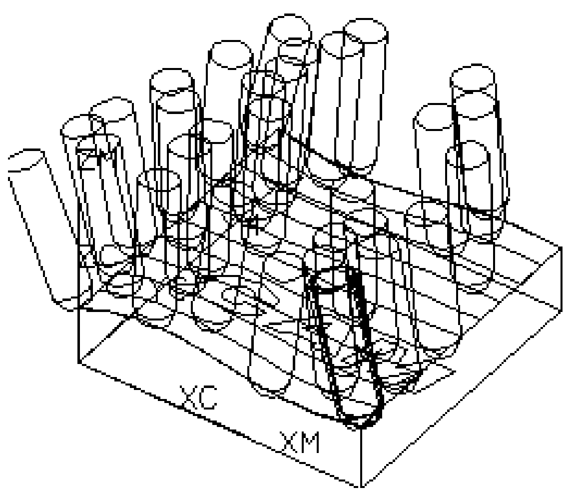
❑ Open the part file **mam_vx_0.prt.**

Step 2 View the tool path.

Note the tool axis in the first pass. The tool axis is **Normal to Part** which means that the tool is 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.

5

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

- ❑ **Edit** the operation **NORM_PART**.

  The Variable Contour dialog is displayed.

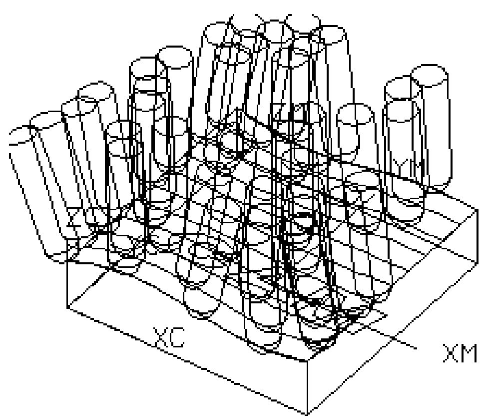
- ❑ Under the Tool Axis area of the dialog, select **Relative to Part** as the tool axis**.**

  Note that you are now prompted to change the Lead and Tilt angles.

  You will use the defaults of 0°.

- ❑ Choose **OK.**

- ❑ **Generate** the tool path.



Compare this tool path to the tool path on the previous page. 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**.

5

The Variable Contour dialog is displayed.

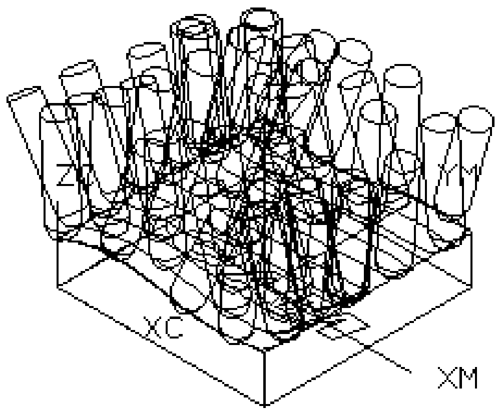
❑ Under the Tool Axis label, choose **Relative to Part**.

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.

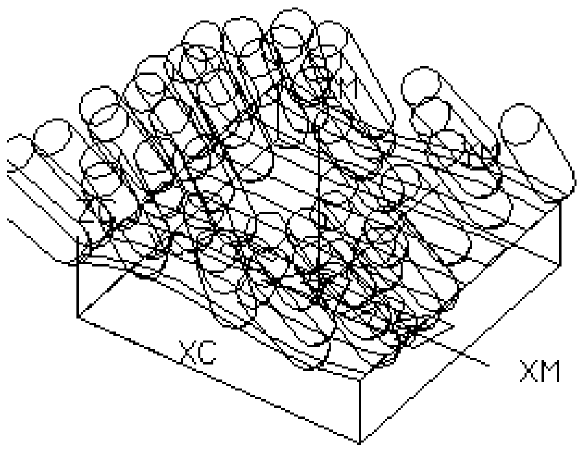
❑ Under the Tool Axis label, choose, **Relative to Part**.

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.

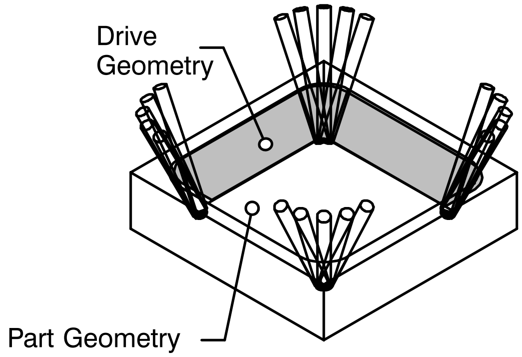
❑ Choose **Cancel.**

❑ Close the part file.

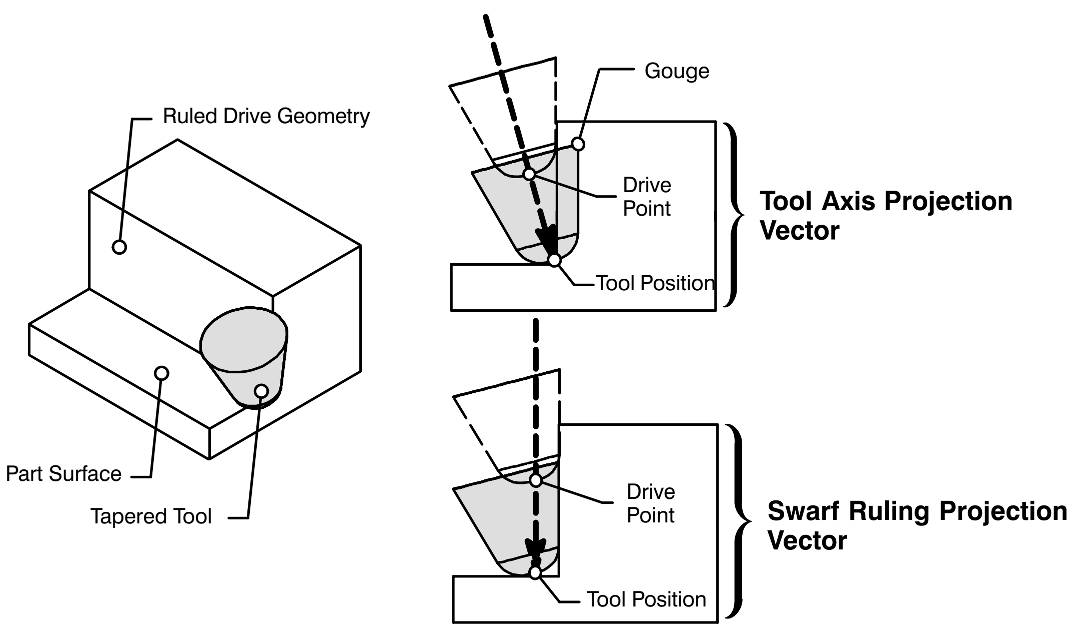
This completes this activity.

Swarf Drive Tool Axis

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



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 in the following illustration:



In this example, a comparison is made between the Swarf Drive Projection Vector and the Tool Axis Projection Vector. In either method, 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.

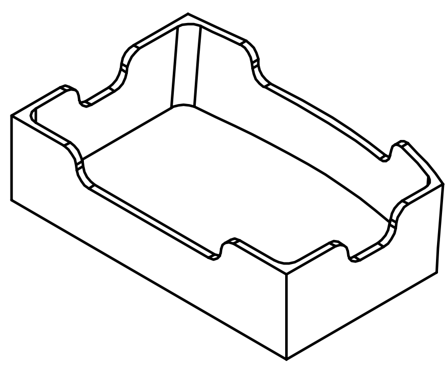
5

Activity 5—4: 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 modify in order to start the tool path at the center of one of the walls, which prevents the cutter from engaging and gouging the interior corner of the part.

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

- ❑ Open the part file **mam_vx_1.prt**.
- ❑ Rename the part to ***_vx_1.prt,** where *** represents your initials.

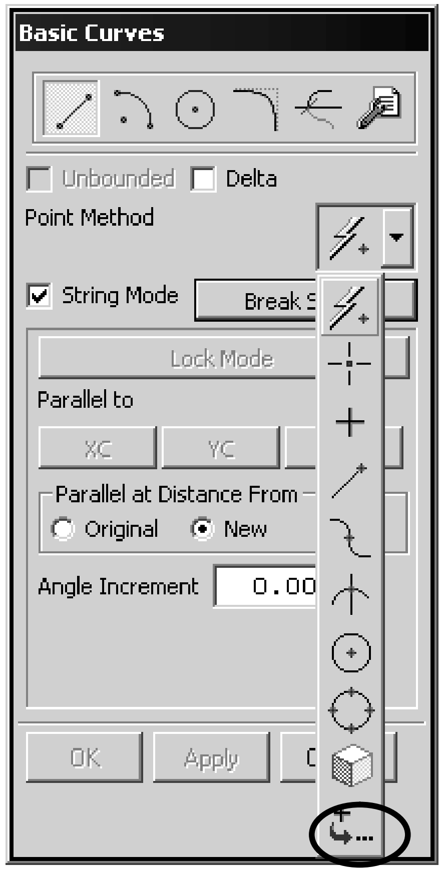


To prevent engaging and then gouging an interior corner of the part, you will engage the part from the middle of one of the long sides of the part. You must first subdivide one of the tapered side walls by creating a curve (in this case a line) before you can subdivide the face.

Step 2 Enter the Modeling application, create the curve for subdividing and subdivide the tapered sidewall face.

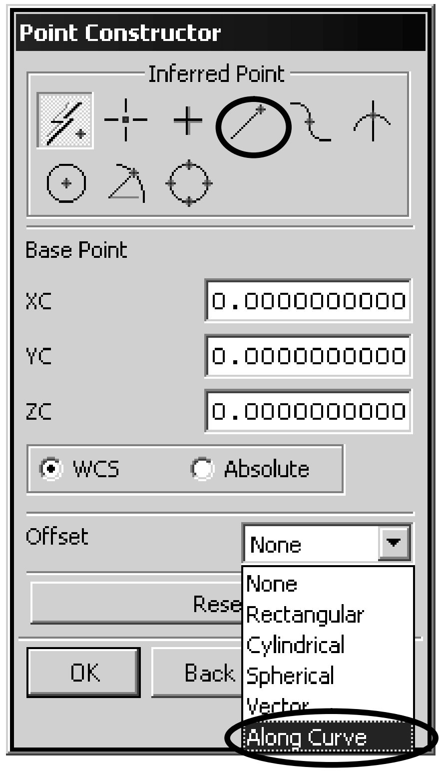
- ❑ Choose **Application** → **Modeling**.
- ❑ Choose **Insert** → **Curve** → **Basic Curves** from the Menu Bar.

The Basic Curves dialog is displayed.

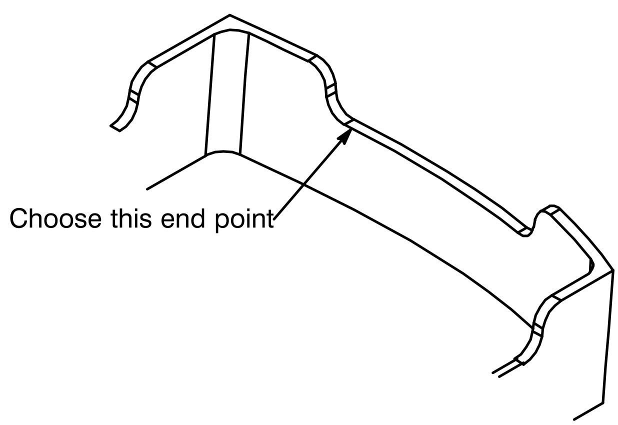


❑ As shown above, change the Point Method to **Point Constructor**.

The Point Constructor dialog is displayed.



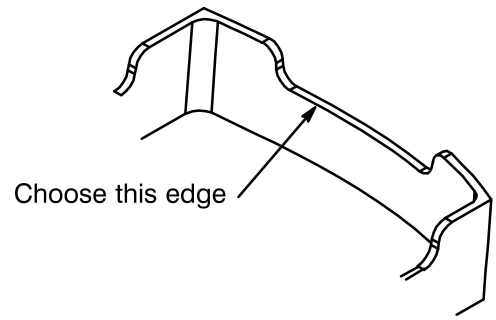
- ❑ As shown above, choose **End Point** as the Inferred Point and change the Offset to **Along Curve**.
- ❑ Choose the left end point of the top inside edge.



You will now define the offset curve.

5

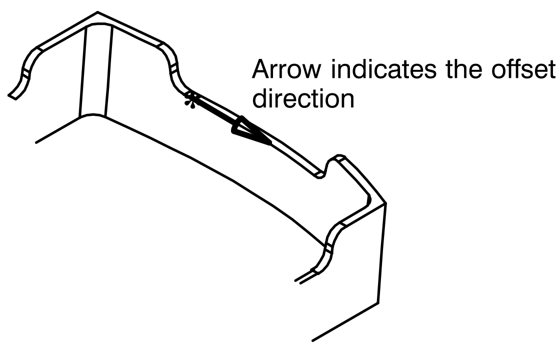
❑ Again, select the top inside edge of the curve, as shown.



The edge is highlighted.

❑ If necessary, choose **OK** from the Selection Confirmation dialog.

An arrow is displayed indicating the direction of the offset. The Point Constructor dialog is also displayed.



Notice that the dialog has changed. You can define the offset distance using a number or a percentage.

❑ Choose **Percent** from the Point Constructor dialog.

The percentage value field is displayed.

❑ Enter **50** into the percentage value field.

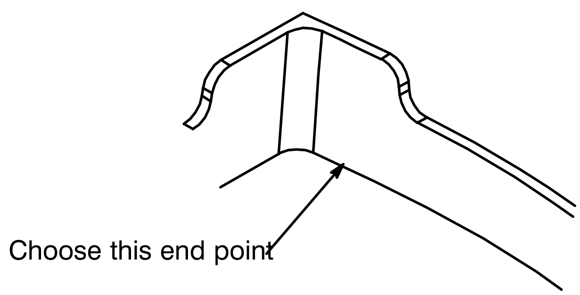
❑ Choose **OK**.

The Point Constructor dialog is displayed. You have defined the first end point for the subdividing line.

You must define the second end point for the line.

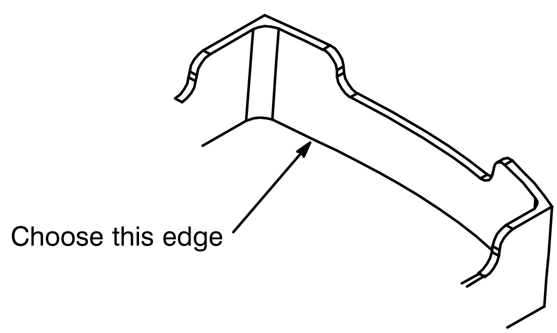
Repeat the same steps for the second line end point.

❑ With the Offset set to **Along Curve**, select the left end point of the bottom inside edge.



You must now define the offset curve.

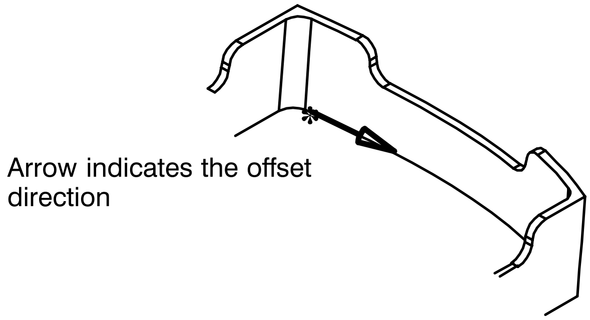
❑ Choose the bottom inside edge, again.



The edge curve is highlighted.

❑ If necessary, Choose **OK** from the Selection Confirmation dialog.

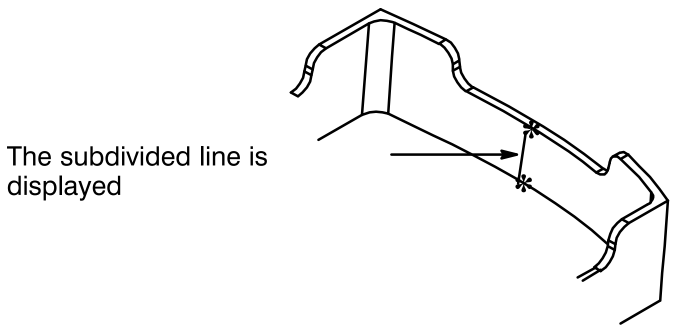
An arrow is displayed indicating the direction of the offset. The Point Constructor dialog is displayed.



The percent option is set at 50%.

❑ Choose **OK**.

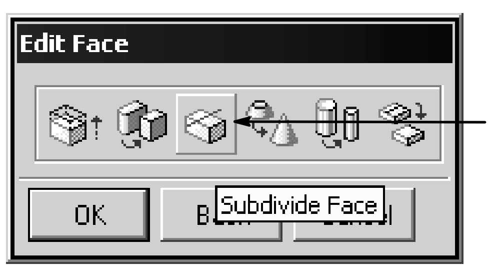
The subdivided line is displayed. You are now ready to subdivide the face.



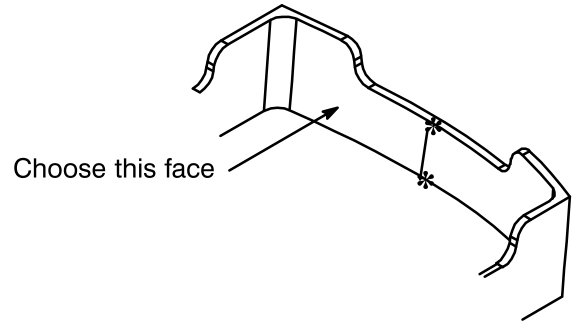
❑ Choose **Cancel** from the Point Constructor dialog.

❑ Choose **Edit** → **Face** from the menu bar.

❑ Choose the **Subdivide Face** icon from the Edit Face dialog.



❑ Select the face to subdivide.



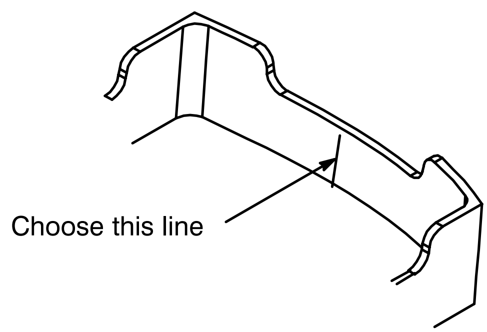
❑ Choose **OK** from the Selection Confirmation dialog.

The curve Subdivide Face dialog is displayed. You will want to blank the curve (the previously created line) that will be used to subdivide the face.

❑ Choose **Blank.**

You must select the curve you are going to use to subdivide the face.

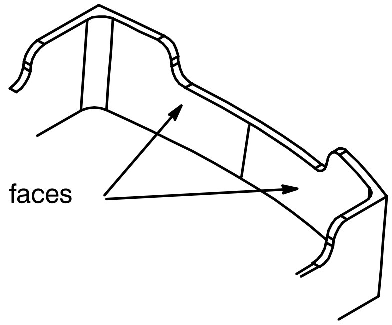
❑ Select the newly created line.



You have the option to add a second object if so desired (not required).

❑ 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**.

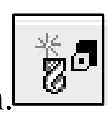
  You will now create the tool path to cut the part.

Step 3 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**.

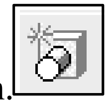
Step 4 Create an end mill needed to machine the part.

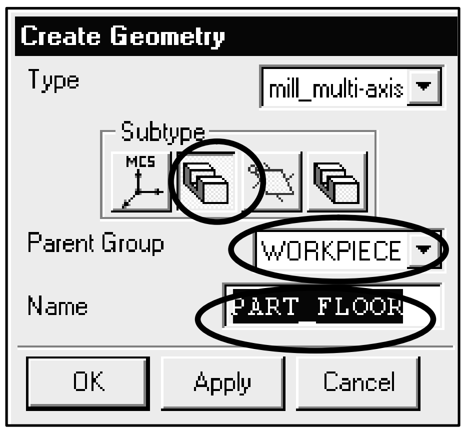
- ❑ Choose the **Create Tool** icon. 
- ❑ On the Create Tool dialog, choose the **Mill** tool icon.
- ❑ Name the tool **EM-.75-.06.**
- ❑ Choose **OK.**
- ❑ In the Milling Tool-5 Parameters dialog, enter a diameter of **.75** and a lower radius of **.06.**
- ❑ Choose **OK**.

5

Step 5 Specify the Part Geometry.

You will now select the floor of the part as the Part Geometry.

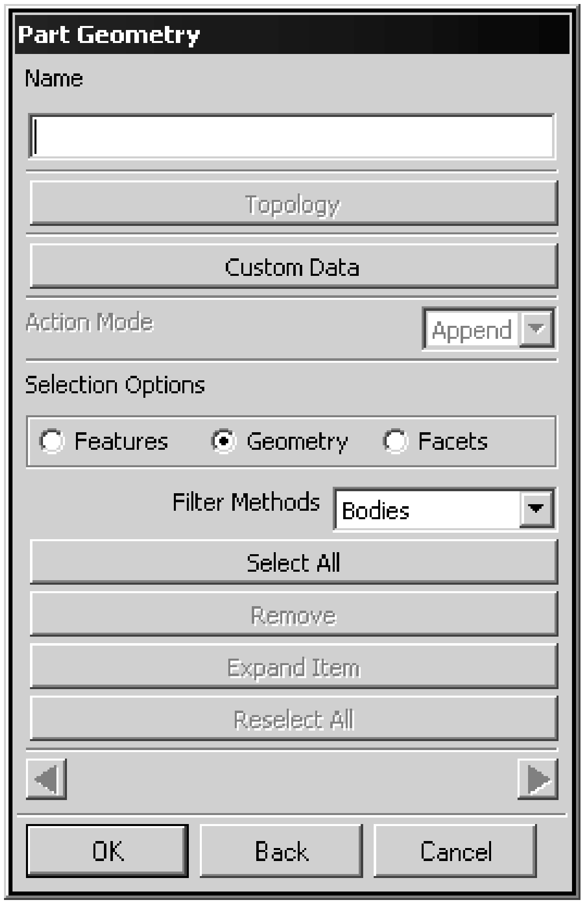
- ❑ Choose the **Create Geometry** icon. 
- ❑ Set the Operation Navigator to the **Geometry View**.
- ❑ Choose the **Mill_Geom** icon.
- ❑ Choose **WORKPIECE** as the Parent Group.
- ❑ **Name** the geometry Parent Group **PART_FLOOR**.



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**.

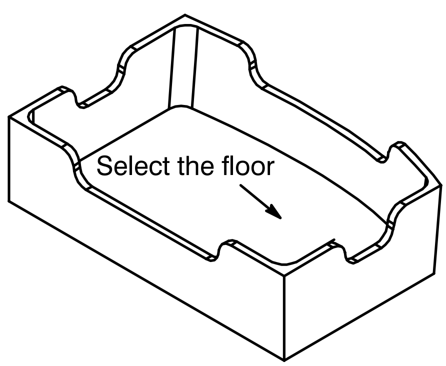
The Part Geometry dialog is displayed.



❑ 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.

5

❑ Verify that the Parent Group, **PART_FLOOR** was created in the Operation Navigator.

Step 6 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**

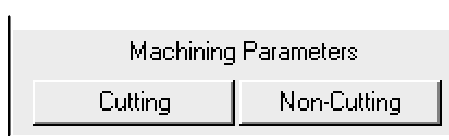
❑ Enter **fin-poc-walls** into the name field.

❑ Choose **OK**.

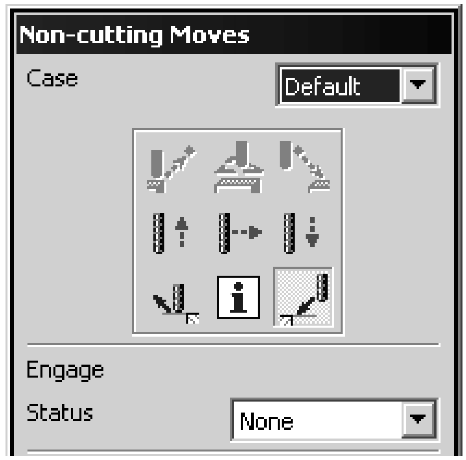
The Variable Contour dialog is displayed.

Step 7 Define an Engage and Retract move using the Non-cutting Moves option.

❑ Choose **Non-cutting** from the Machining Parameters area.

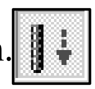


The Non-cutting Moves dialog is displayed.



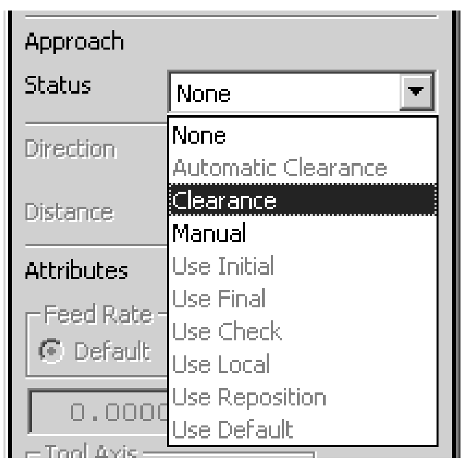
You will now define an Approach move for the Default Case.

You can also define different Approach moves for the Initial, Final Check, Local, and Reposition moves.

❑ Choose the **Approach** icon. 

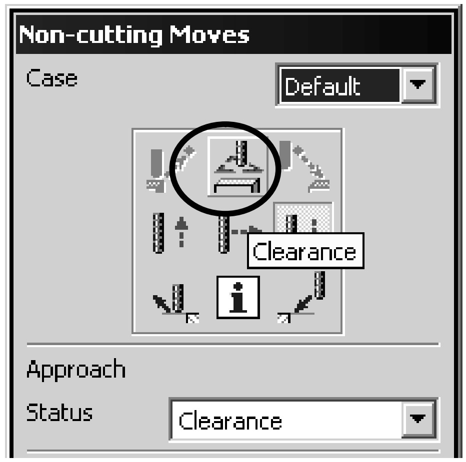
The options available have changed to reflect the Approach move options.

❑ Next to the Status label, change **None** to **Clearance**.

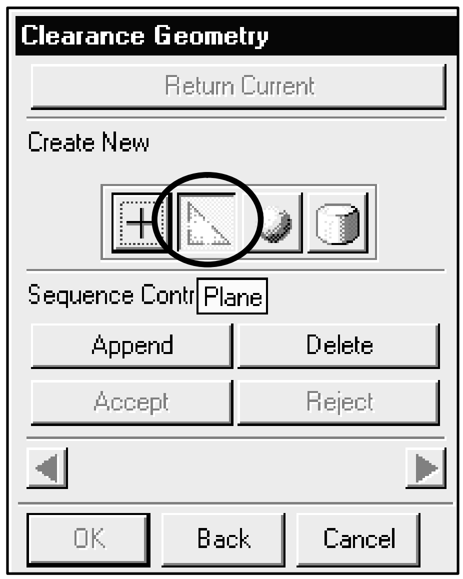


The Clearance icon is now active.

❑ Choose the **Clearance** icon located at the top of the Non-cutting Moves dialog.

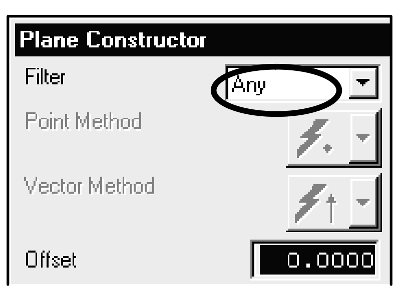


The Clearance Geometry dialog is displayed.



❑ Choose the **Plane** symbol icon.

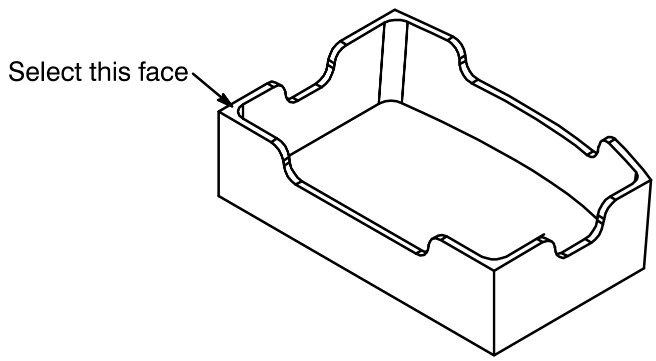
The Plane Constructor dialog is displayed.



❑ Change the Filter to **Face**.

❑ Enter **1** in the Offset field.

❑ Select the top face of the part.



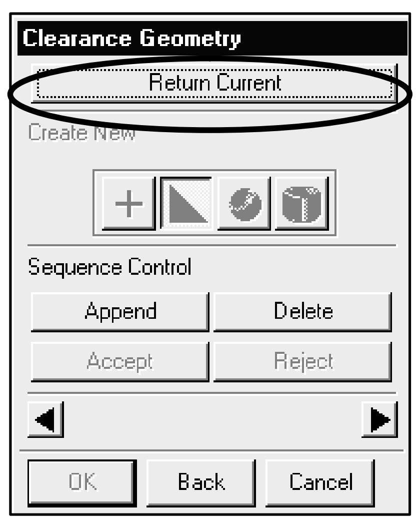
❑ Choose **OK**.

The Clearance Geometry dialog is displayed.

❑ Choose **Accept** from the Clearance Geometry dialog.

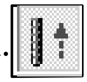
The Clearance geometry is accepted.

❑ Choose **Return Current**.



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.

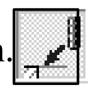
❑ Choose the **Departure** icon. 

❑ Next to the Status label, change **None** to **Clearance**.

Use the Clearance Plane that you just created for the Approach move.

Since you only have one plane defined, the departure moves will use that plane. This is all that is required for the departure move.

You will now define an Engage Move for the Initial Case.

❑ Choose the **Engage** icon. 

❑ Next to the **Case** label, change **Default** to **Initial**.

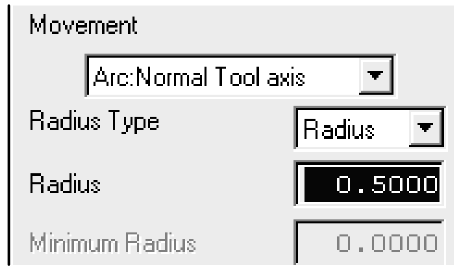
❑ Next to the **Status** label, change **Use Default** to **Manual**.

❑ Under the **Movement** label, change **Linear** to **Arc: Normal Tool axis**.

The Radius Type is Automatic.

❑ Change the Radius Type to **Radius.**

❑ Key in **.500** into the Radius value field.



You will now define a Retract Move for the Final Case.

❑ Next to the **Case** label, change **Initial** to **Final**.

The **Retract** icon is automatically specified.

❑ Choose the **Status** button and change it to **Manual.**

❑ Change the **Movement** to **Arc: Normal Tool Axis**.

❑ Change the Radius Type to **Radius.**

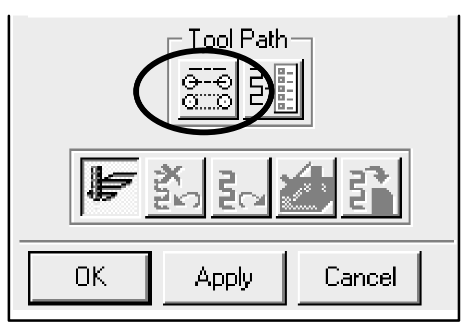
❑ Key in **.500** into the Radius value field.

❑ Choose **OK**.

The Variable Contour dialog is displayed.

Step 8 Define the Display options.

❑ Choose **Edit Display** in the Tool Path section.



The Display Options dialog is displayed.

- ❑ Set the following options:

Tool Display	**= 3D**
Frequency	**= 5**
Path Display Speed	**= 8**

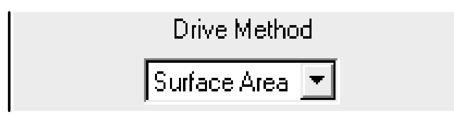
- ❑ Choose **OK**.

The Variable Contour dialog is displayed.

Now that 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. The Drive Method will then be used to create the drive points.

Step 9 Define the Drive Method.

- ❑ Under the Drive Method label, select **Surface Area.**

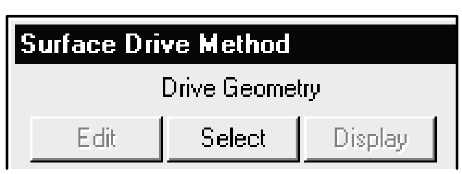


- ❑ If necessary, accept the warning message.

The Surface Drive Method dialog is displayed.

Step 10 Select the Drive Geometry.

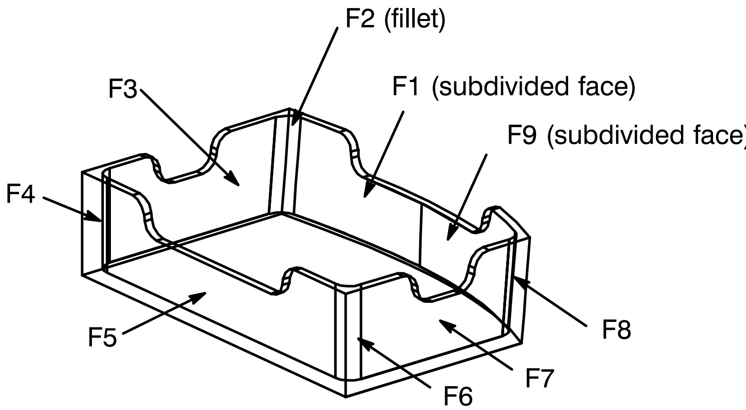
- ❑ Choose **Select** from the Drive Geometry area.



The Drive Geometry dialog is displayed.

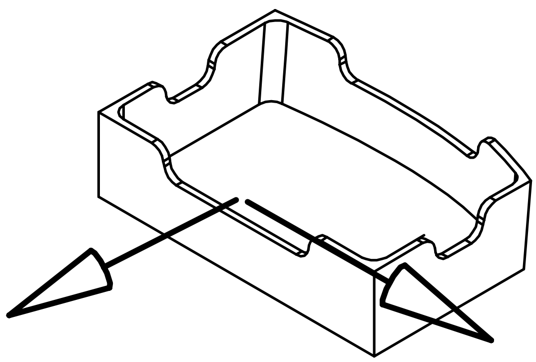
- ❑ Select the **interior** Faces F1 through F9 as shown.

5



❑ Choose **OK** from the Drive Geometry dialog when the face selection is complete.

The material side and direction indicator appears as follows:



❑ Choose **Flip Material** from the Surface Drive Method dialog.

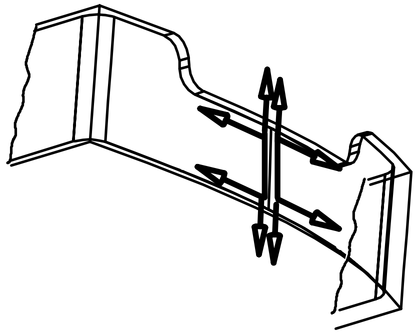
Step 11 Define the drive direction.

You must now define the direction of the cut.

❑ Choose **Cut Direction** from the Surface Drive Method dialog.

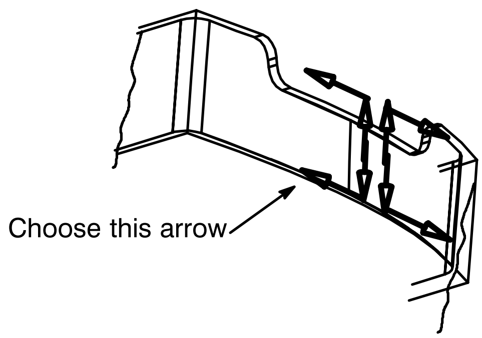
Cut direction arrows are displayed.

5



You will select the direction arrow that points in the direction that you will want to cut.

❑ Choose the lower left arrow.

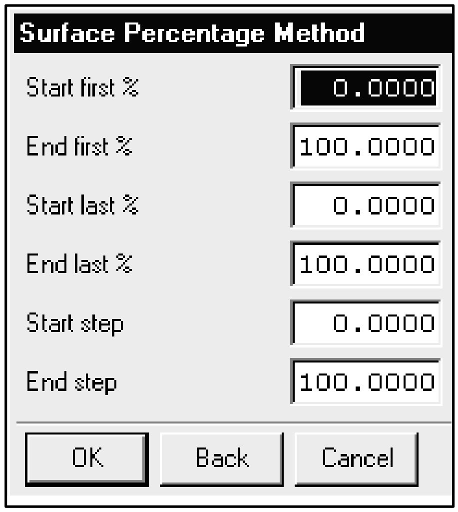


Step 12 Define the Cut Area.

The Cut Area option defines what percent of the surface area is going to be cut. You can use a percentage or use diagonal points to define the area.

❑ Next to the Cut Area label, choose **Surface %**.

The Surface Percentage Method dialog is displayed.



You can specify the cut area by specifying positive or negative percentage values for the beginning and end of the first pass, the beginning and end of the last pass, the first Stepover and the last Stepover.

❑ Enter the following values:

Start first % **= – 0.0100**

End first % **= 100.010**

Start step **= 0.000**

End step **= 0.000**

The Start first % and End first % values will assure overlapping cuts.

❑ Choose **OK**.

Step 13 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 14 Define the Gouge Action.

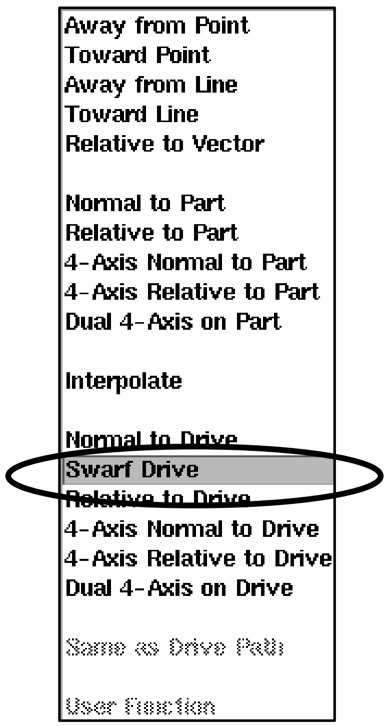
❑ Next to the When Gouging label, change **None** to **Skip**.

The Skip option will move the tool to the next non-gouging point if gouging is detected.

Step 15 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 16 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 complete.

❑ Choose **OK**.

The Variable Contour dialog is displayed.

Step 17 Create the tool path.

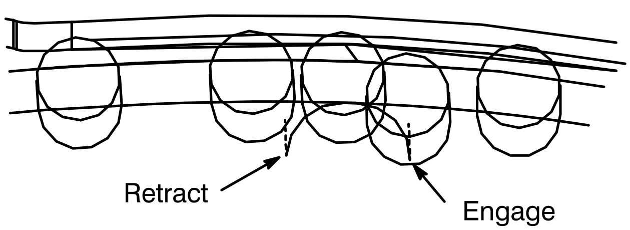
❑ Choose the **Generate** button.

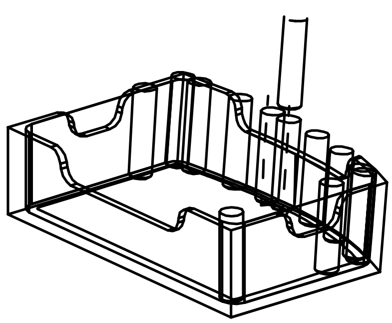
The tool path is generated and the option menu is displayed.

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.





❑ Choose **OK** from the option menu.

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

This completes this activity.

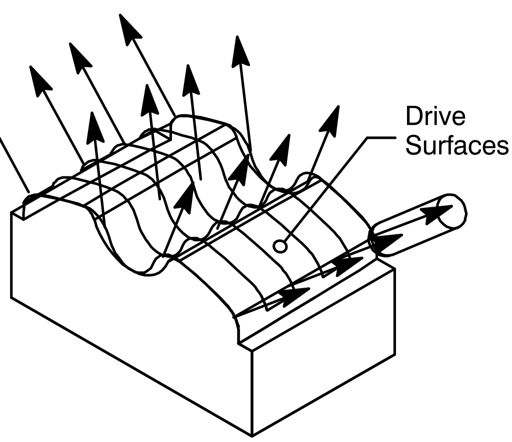
Interpolated Tool Axis

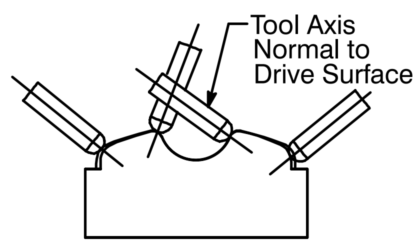
Interpolate tool axis enables the control of the tool axis at specific points by defining vectors. It allows for control of excessive change of the tool axis as a result of very complex drive or part geometry, without the construction of additional tool axis control geometry (e.g., points, lines, vectors, 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.

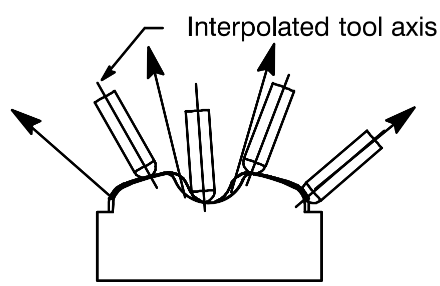
This option is available only when using the Curve/Point or Surface Area drive method.

User-Defined Controlling Vectors



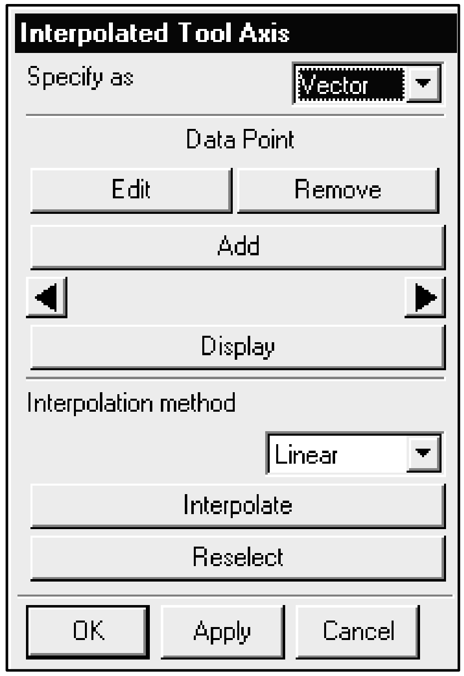


Excessive tool axis change



Smoother tool axis movement

Interpolate tool axis displays the following dialog:



Interpolated tool axis dialog options:

Specify as allows you to specify a method to define the vectors used to interpolate the tool axis. You can define as many vectors as necessary to control the tool axis.

Vector enables you to define vectors used to interpolate the tool axis by first specifying a data point on the drive geometry and then specifying a vector.

Angle/PS (or **DS**) enables you to define vectors used to interpolate the tool axis by specifying a data point on the drive geometry (using the Point Constructor dialog) 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 the -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. You must first specify a data point on the drive geometry using the Point Constructor dialog and then a vector direction. Initially, after specifying the data point, the system displays a vector normal to the drive geometry.

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).

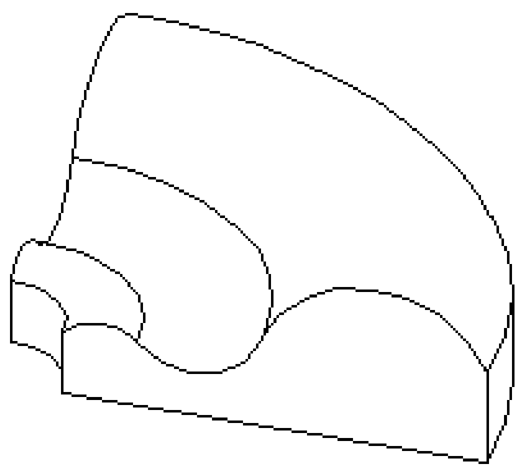
Reselect removes all defined data points.

Activity 5—5: 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 new part file, rename and enter the Manufacturing application.

❑ Open the part file **mam_vx_6.prt** and rename it to *****_vx_6.prt.**



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

❑ Choose the Operation Navigator icon from the tool bar.

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:	**NC_PROGRAM**
Use Geometry:	**WORKPIECE**
Use Tool:	**EM_2.000_1.000**
Use Method:	**MILL_FINISH**

❑ Key in the operation name**, interpolate** in the name field.

❑ Choose **OK**.

The Variable Contour dialog is displayed.

❑ Under the Machining Parameters label, select **Non-Cutting**.

❑ Specify the **Default Case, Clearance Planes** for:

Approach	=	ZC	**8.00**
Engage	=	ZC	**7.50**
Retract	=	ZC	**7.50**
Departure	=	ZC	**8.00**

Step 3 Define the Part Geometry.

In this activity, you do not need to specify part geometry because the drive geometry will be used as the part geometry once it is selected.

Step 4 Define the Drive Geometry.

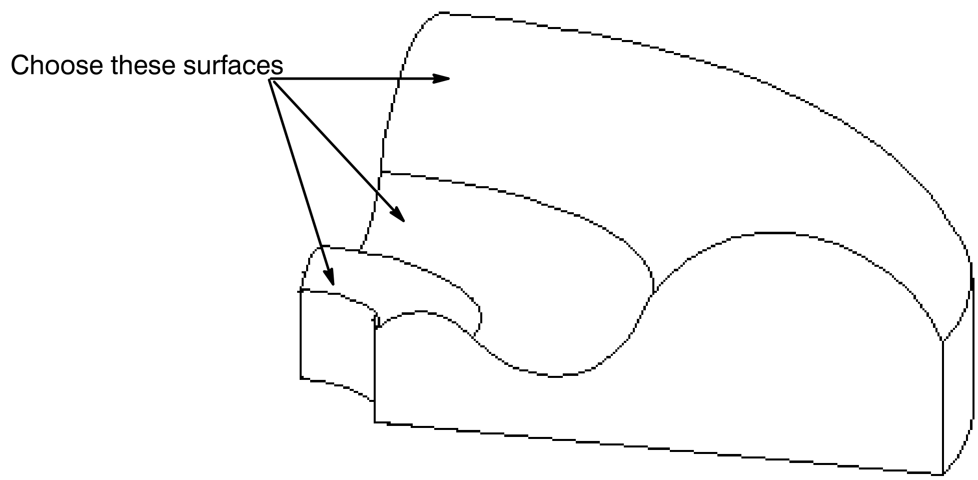
❑ Under the Drive Method label, choose **Surface Area**.

5

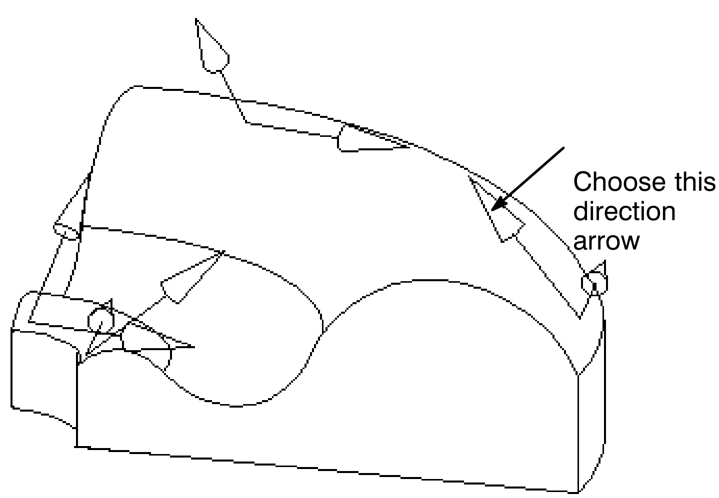
The Surface Drive Method dialog is displayed.

Step 5 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.



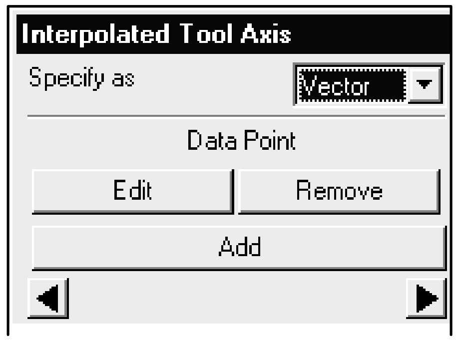
- ❑ Change the Cut Type to **Zig**.

❑ Change the Cut Step to **Tolerances**.

❑ Under the Stepover label, change the Number of Steps to **4**.

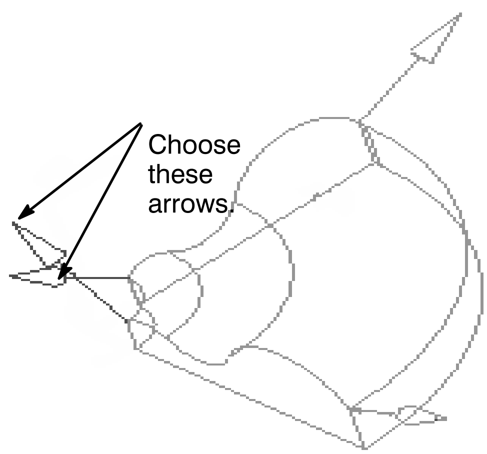
❑ Change the Tool Axis to **Interpolate**.

The Interpolated Tool Axis dialog is displayed.



Four default vector arrows show the current tool axis vector direction.

❑ As shown in the following illustration, choose the front arrows (using the cursor or the Selection Arrows) 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**.

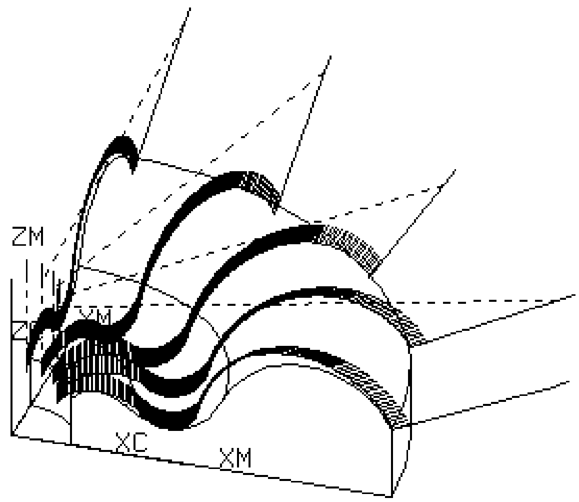
The Surface Drive Method dialog is displayed.

❑ 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.

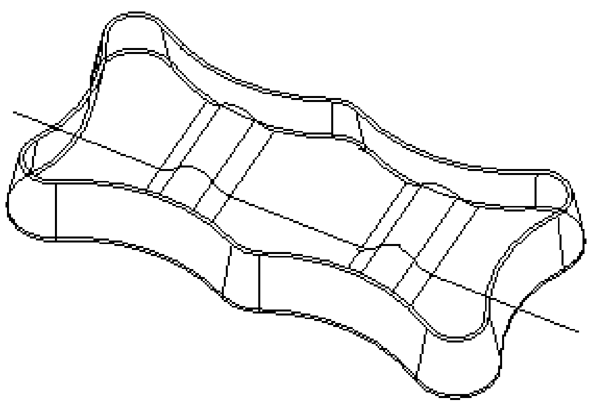
This completes this activity.

Activity 5—6: 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 a new part file, rename it and enter the Manufacturing application.

❑ Open the part file **mam_vx_7.prt** and rename it to *****_vx_7.prt**, where *** represents your initials.



❑ Choose **Application → Manufacturing**.

❑ Choose the **Create Operation** icon.

❑ Choose the **Variable Contour** icon.

❑ In the Create Operation dialog, set the following:

- Program: **NC_PROGRAM**
- Use Geometry: **MCS**
- Use Tool: **MILL**
- Use Method: **MILL_FINISH**

❑ Key the name**, proficiency** into the name field.

❑ 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.

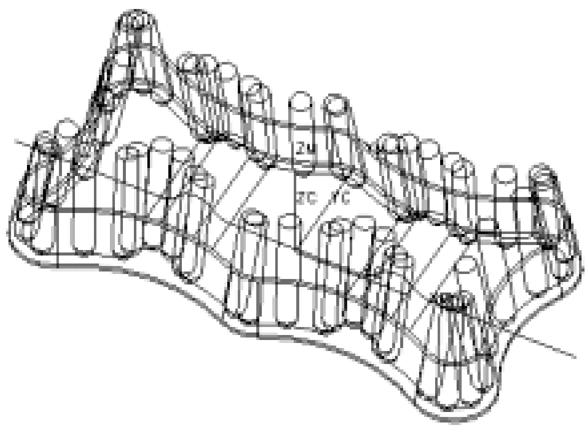
Tool Axis Review Questions

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? ______________________________

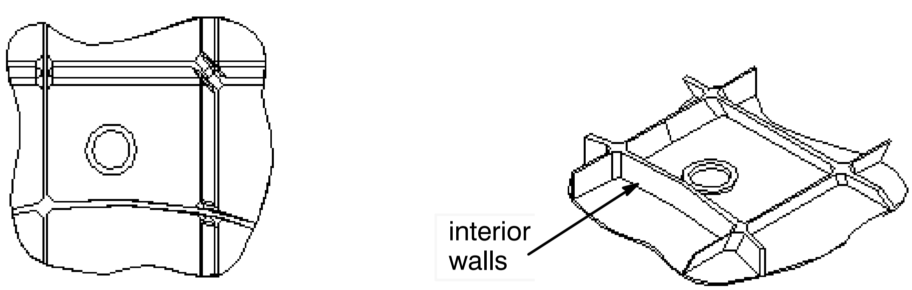
____

____

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? ______________________________



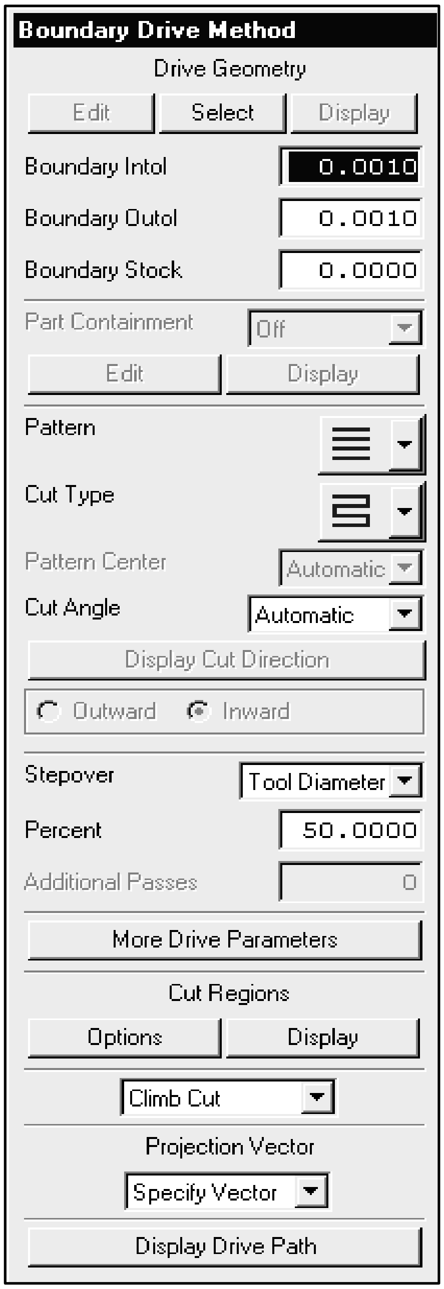
6. What Drive Method would you use? ______________________________

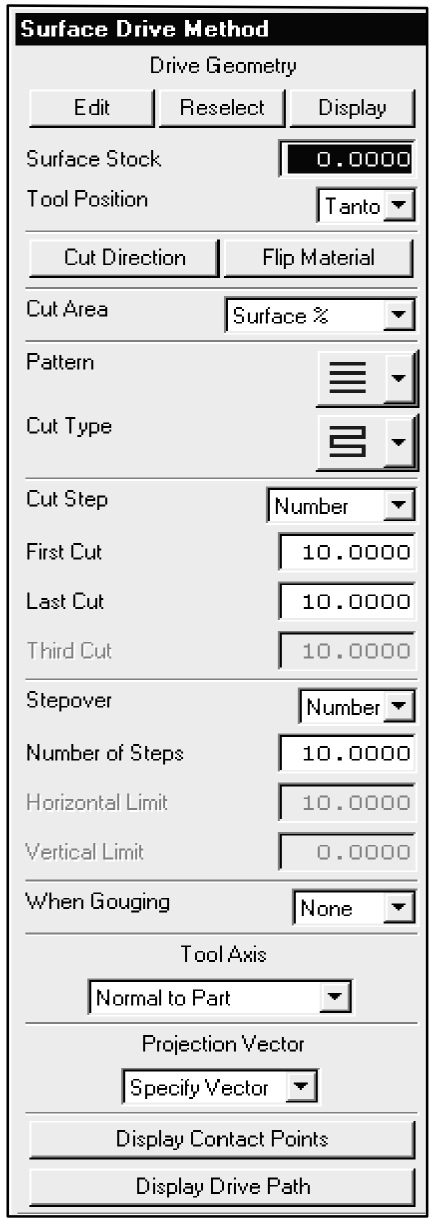
7. What Projection Vector would you use? ______________________________

Variable Contour Best Practices

Surface Area vs. Boundary Drive Methods

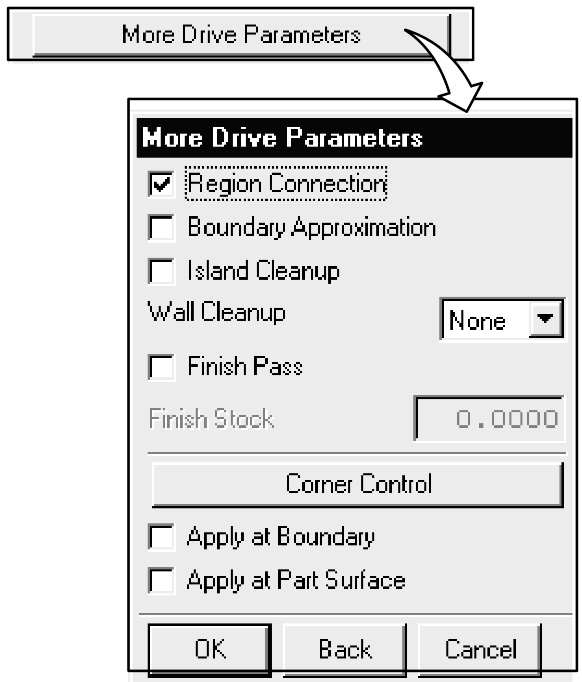
Surface Area and Boundary drive method dialogs are shown. You can compare the options available for each Drive Method.





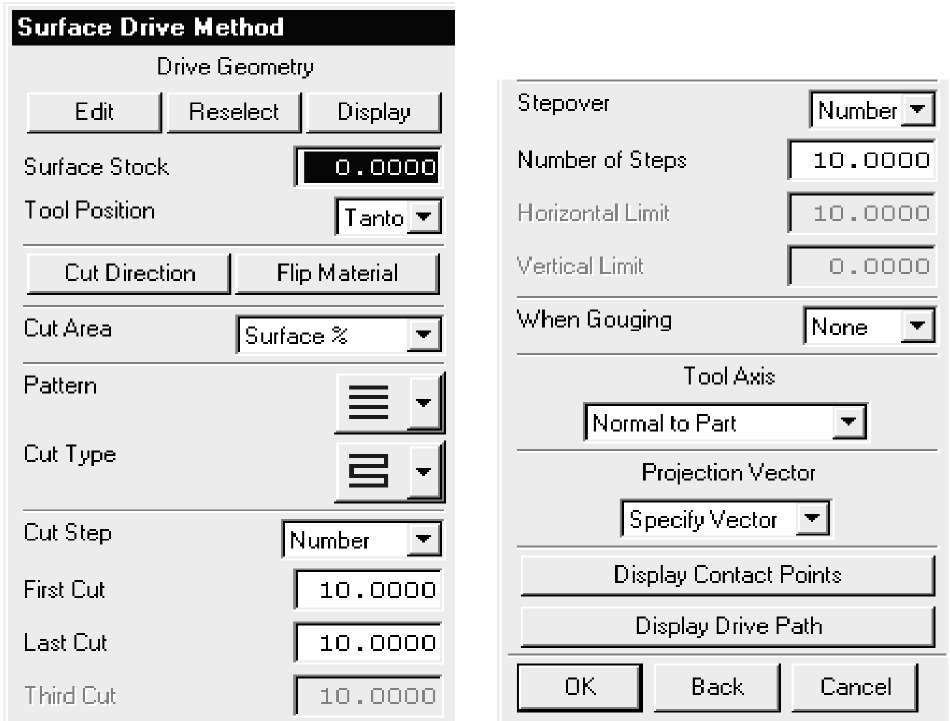
Either the Surface Area or Boundary Drive Method can be used for creating tool paths. There are some considerations to keep in mind in deciding which method is more conducive to use in multi-axis finish machining.

- 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, in Variable Contour, is used to create drive points that are projected to the Part geometry. Variable Contour allows you to 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, if specified, is typically used to prevent collision and gouging.

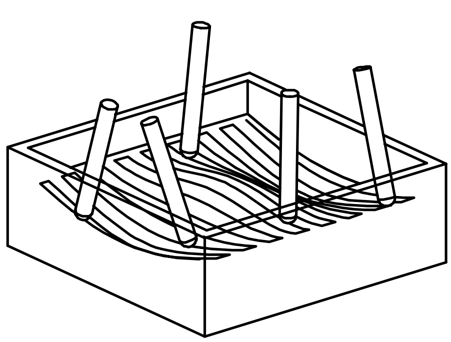
Sequential Mill requires selection of Check geometry. The Check geometry is used for tool positioning at the beginning of the next sub-operation and for preventing collision and gouging.

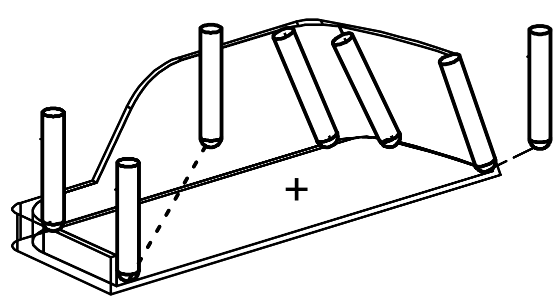
General Considerations

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
The preferred method for area milling	The preferred method for linear milling
Primary cutting with the bottom of the tool	Primary cutting with the side of the tool
Numerous drive methods for containing the tool path	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 respecify tool axis during operation
Edits apply to entire tool path	Edits applied to part of the tool path
Cuts convex walls best.	Cuts angled (overcut & undercut) walls best
Easier to create operation	Numerous steps to an operation
Easier for multi-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 DS / At Angle to PS
Dual 4-Axis on Part	—
Dual 4-Axis on Drive	—
—	Tangent to PS
—	Tangent to DS
—	Proj DS Normal
—	Proj PS Normal
5 Axis	
Away From Point	Thru Fixed Point
Toward 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
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 4 and 5-axis machining centers. 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

5

Variable Contour — Advanced

Lesson 6

6

PURPOSE

This lesson will introduce advanced concepts in conjunction with Variable Contour operations.

OBJECTIVES

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.

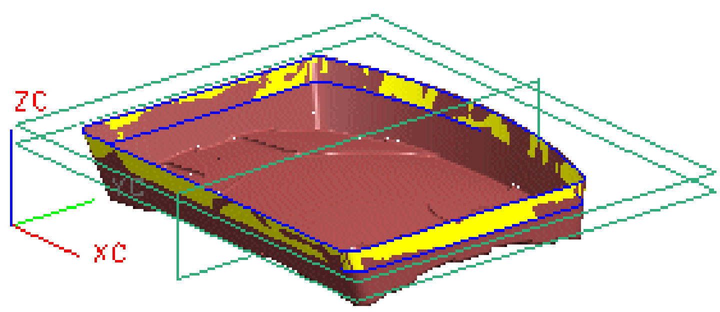
All surfaces which have not been intersected by parallel datum planes will be selected as part surfaces. The tool axis will be set to swarf drive and the projection vector will be the tool axis.

Activity 6–1: 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.prt**.

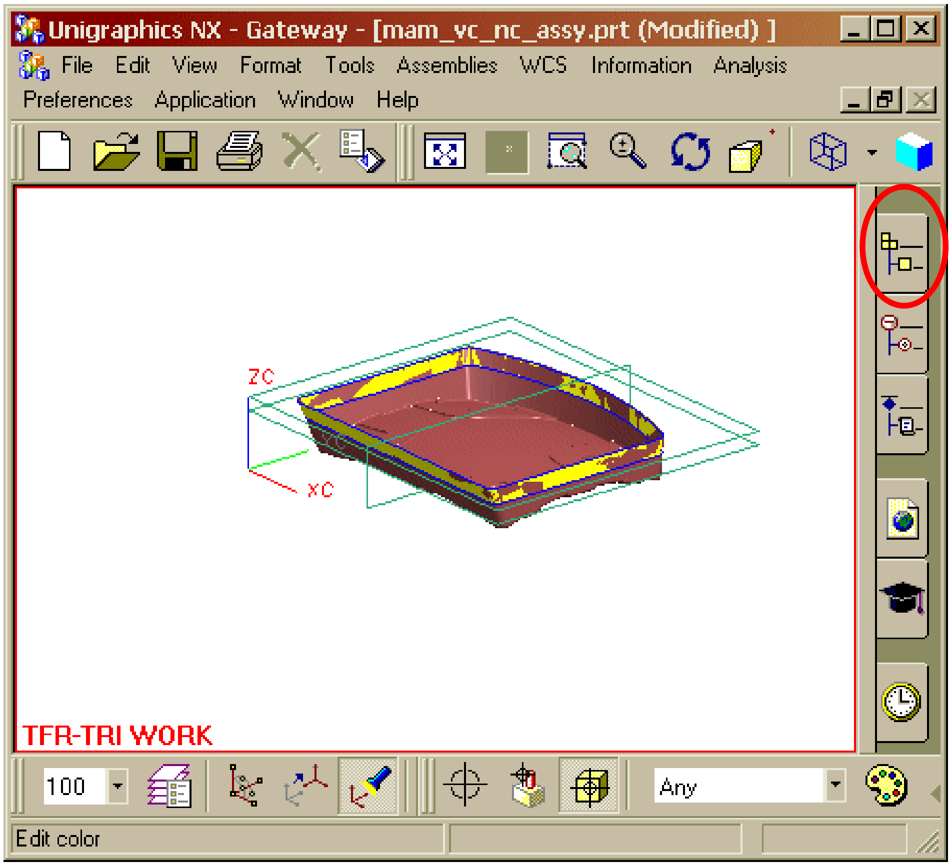


There are two existing sample operations that you will later examine and will then create like operations. However, first you will examine the various parts which comprise the assembly.

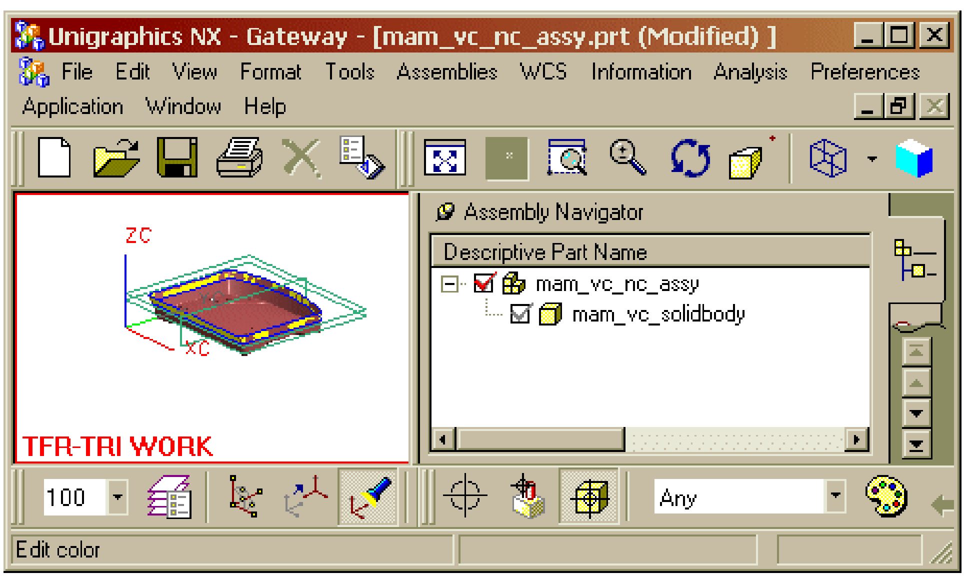
Step 2 Examine the assembly.

❑ Choose **Application** → **Modeling**.

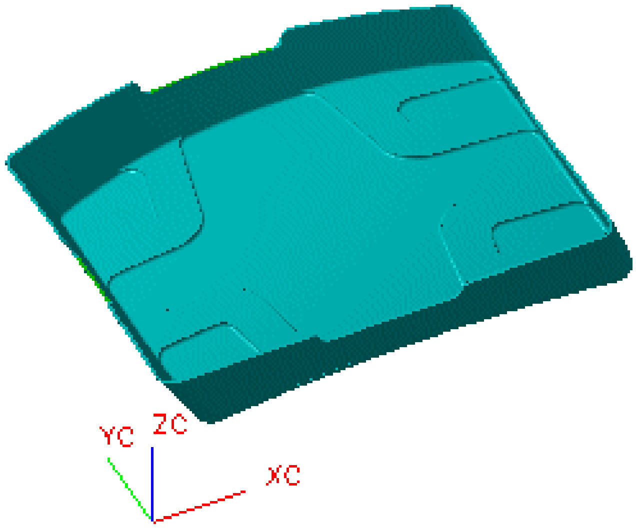
❑ Choose the Assembly Navigator tab from the tool bar.



The Operation Navigator and the part model are displayed.

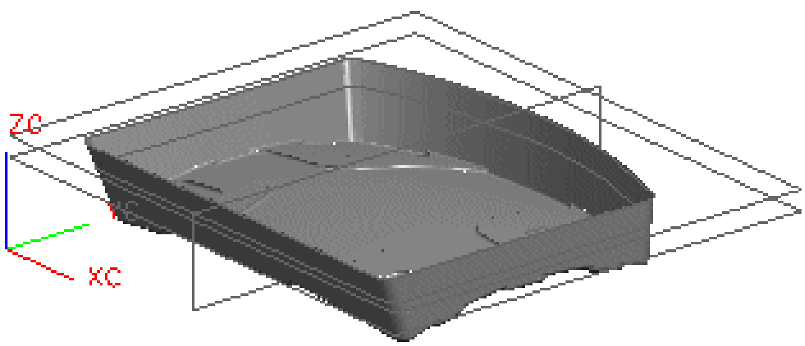


❑ Highlight the **mam_vc_solidbody** component, use **MB3** and choose **Make Displayed Part**.



Note the cut-out areas on top of the walls.

❑ Highlight the **mam_vc_solidbody** component, use **MB3** and choose **Display Parent** → **mam_vc_nc_assy**.



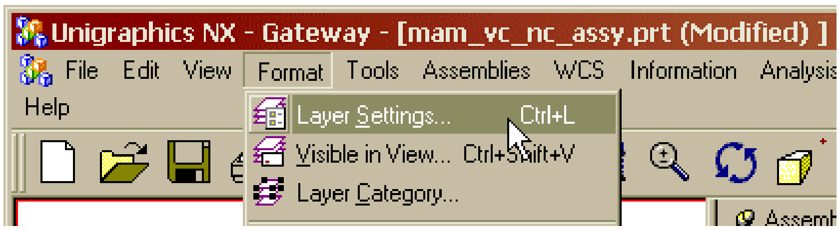
Note that the color of the part is gray.

❑ Highlight the **mam_vc_assy**, use **MB3** and choose **Make Work part**.

Note that the color of the part is returned to its original state.

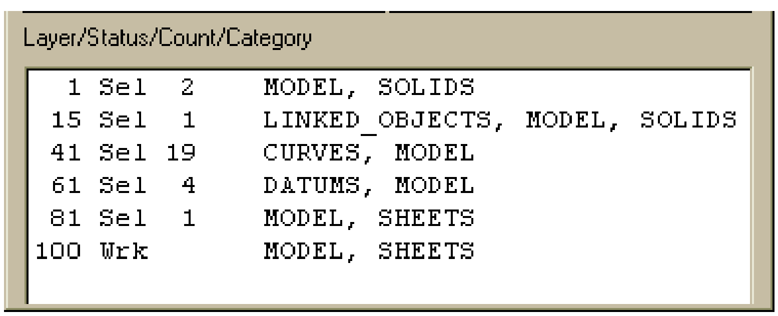
Step 3 Examine layers in the assembly.

❑ Choose **Format → Layer Settings** from the menu bar.



❑ Turn **Show Object Count** and **Show Category Names ON**.

Note the change in the Layer/Status/Count/Category area of the dialog.



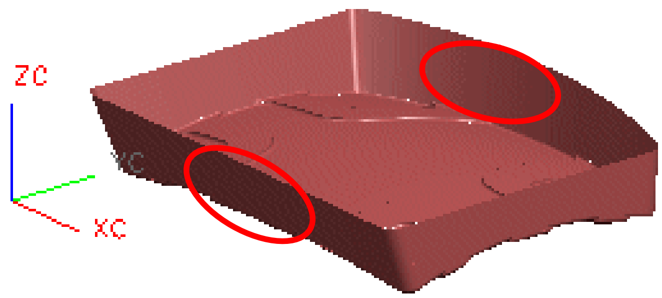
You will now examine the layers.

❑ Make all layers **Invisible**.

❑ Make **layer 15** Selectable.

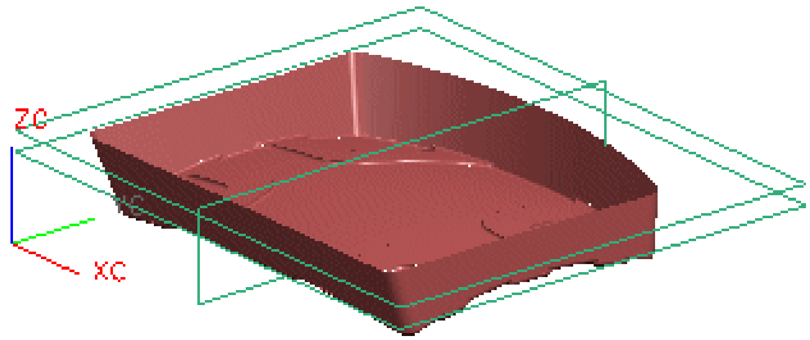
❑ Choose **OK**.

Now examine the WAVE Linked surfaces.



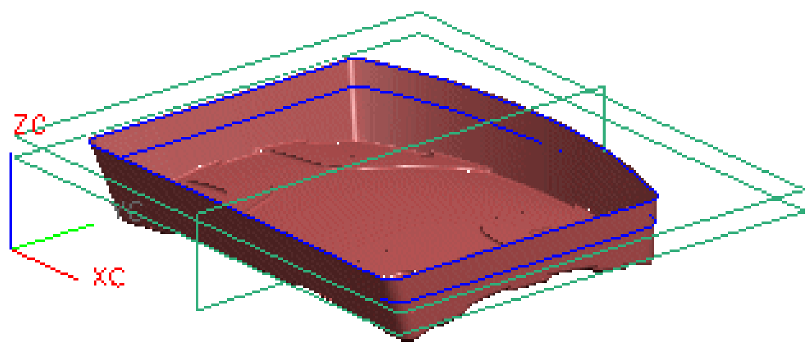
Note that the cut-outs were not passed to the WAVE Linked geometry due to the use and placement of the timestamp.

❑ Make **layer 61** Selectable.



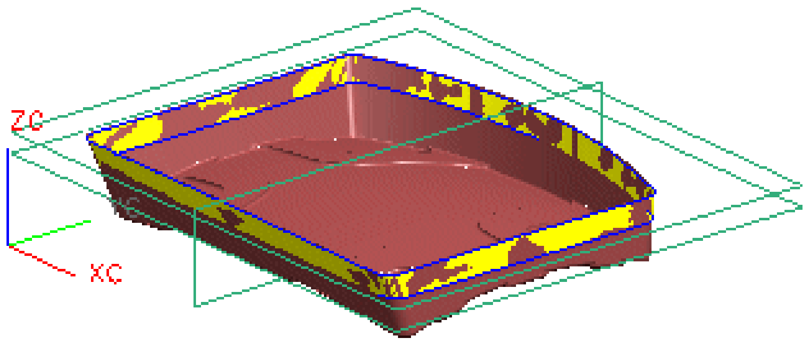
The view now shows the part with the Associative Datum planes that are used to create the necessary intersection curves.

❑ Make **layer 41** Selectable.



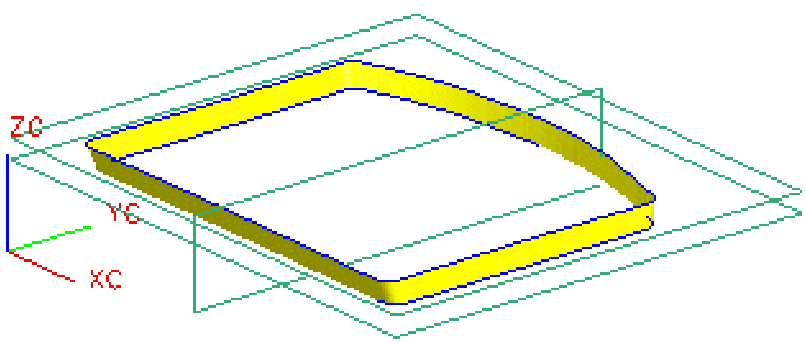
The view now shows the part with the Associative Curves that are used to create the necessary ruled surface.

❑ 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.

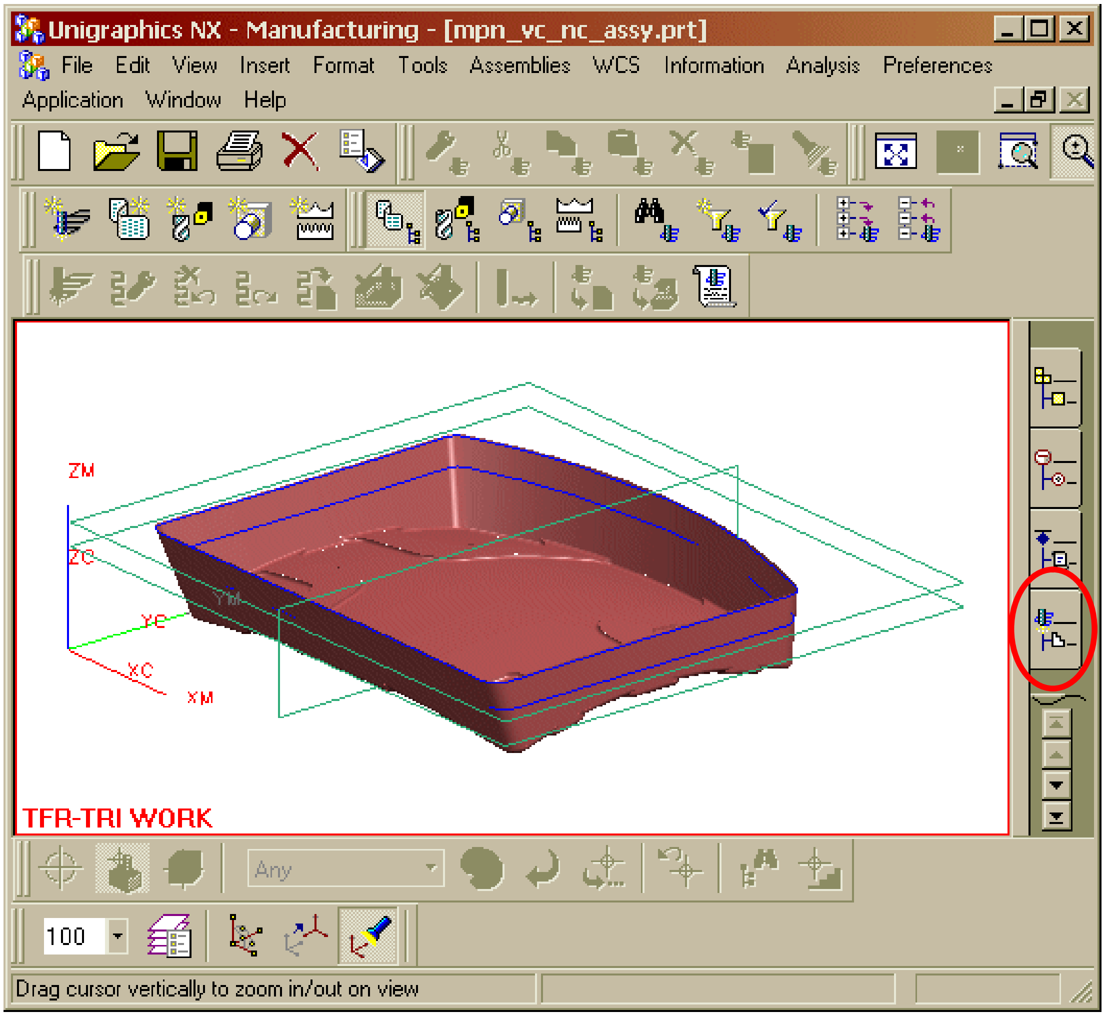
❑ Dismiss the Layer Settings dialog by choosing **Cancel**.

Step 4 Enter the Manufacturing Application and review the existing operations.

You will review the operations by examining their settings.

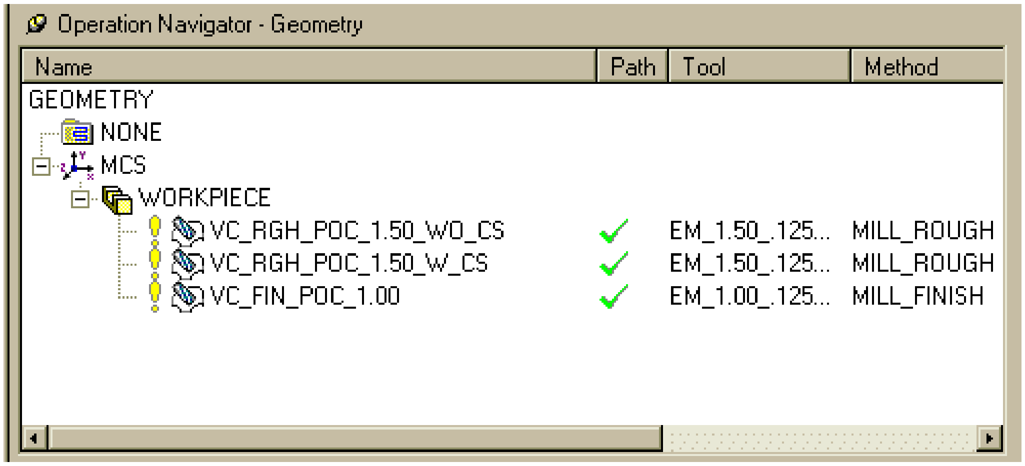
❑ Enter the **Manufacturing application** by choosing **Application → Manufacturing.**

❑ 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.

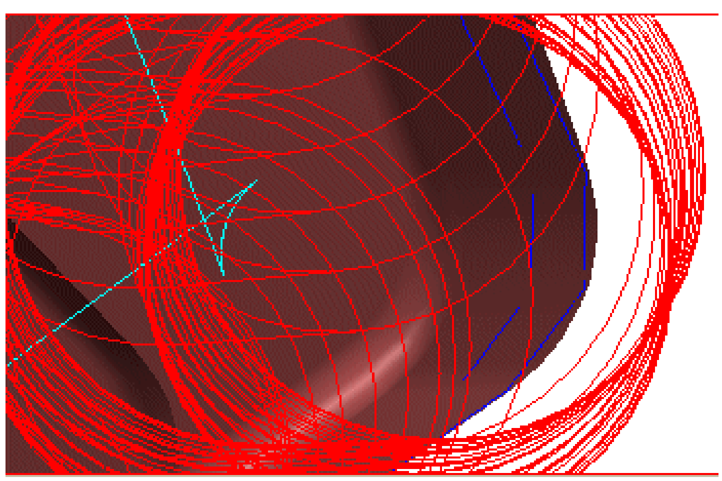
- ❑ Display the Part geometry.

- ❑ Now, display 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 how the cutter violates the drive surfaces.

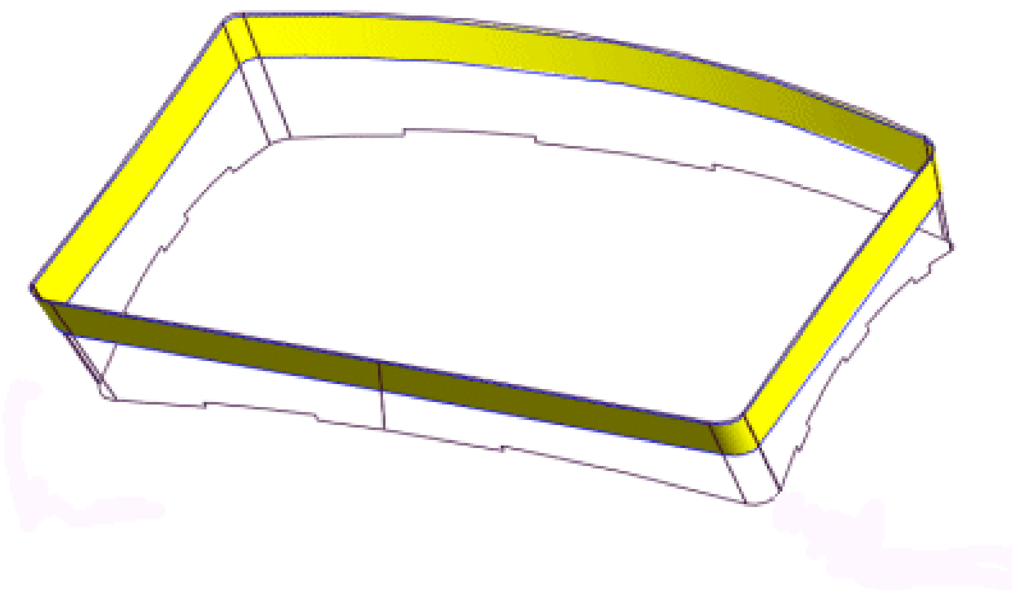


❑ Double click on the operation **VC_RGH_POC_1.50_W_CS**.

The Variable_Contour dialog is displayed.

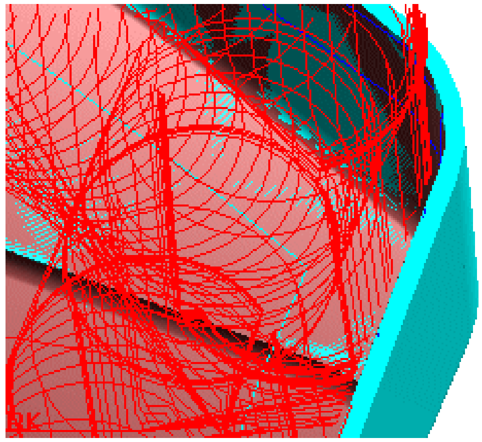
❑ Display the Part geometry.

❑ Now, display 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 now that 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.

This completes the activity and the lesson.

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.

(This Page Intentionally Left Blank)

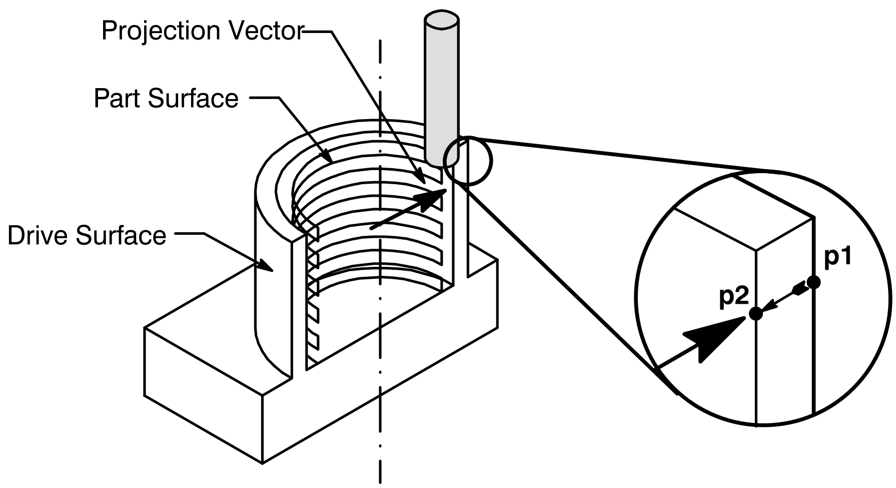
Projection Vectors

Appendix A

A

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.

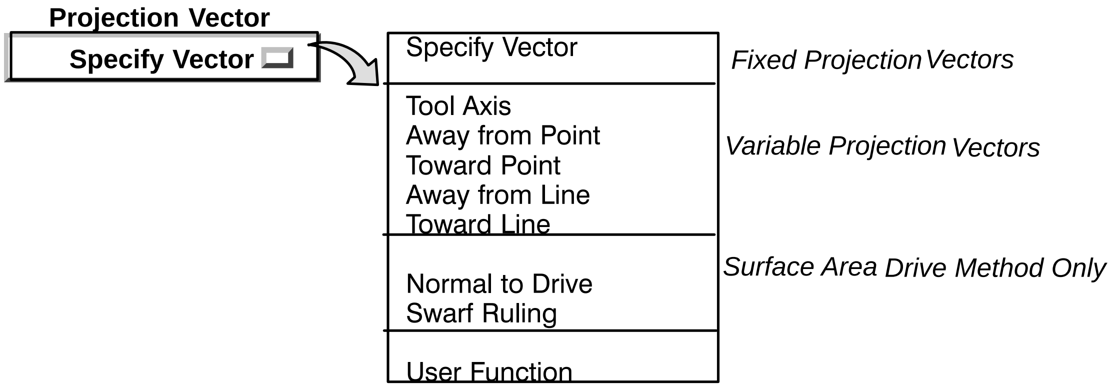
The following illustration shows a Projection Vector (defined as Away From Line, i.e. the centerline) indicating the side of the part surface to be cut. It also shows a drive point projected, along the projection vector, from the drive surface (P1) to the part surface (P2).



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.

NOTE 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

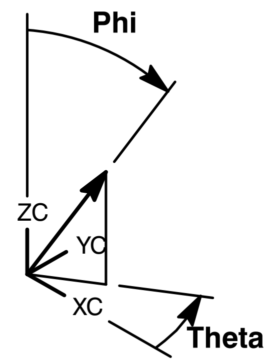
I, J, K allows you to define a Fixed Projection Vector by keying in values defining a vector relative to the origin of the Work Coordinate System.

Line End Points allows you to define a Fixed Projection Vector by defining two points, selecting an existing line, or defining a point and a vector.

2 Points allows you to define a Fixed Projection Vector by using the point Constructor to specify two points. The first point you specify defines the tail of the vector. The second point you specify defines the arrowhead of the vector.

Tangent to Curve allows you to define a Fixed Projection vector tangent to a selected curve. You will specify a point on the curve, select an existing curve, and select one of two displayed tangent vectors.

Spherical Coordinates allows you to define 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.



Variable Contour Projection Vectors

Tool Axis allows you to define a projection vector relative to the existing tool axis. When using tool axis, the projection vector always points in the *opposite* direction of the tool axis vector.

Away From Point allows you to create a projection vector extending away from a specified focal point toward the part surface. This option is useful in machining the inside spherical (or sphere like) surfaces where the focal point is the center of the sphere.

Towards Point allows you to create a projection vector extending from the part surface to a specified focal point. This option is useful in machining the outside spherical (or sphere like) surfaces where the focal point is the center of the sphere.

Away From Line allows you to create a projection vector extending from a specified line to the part surface.

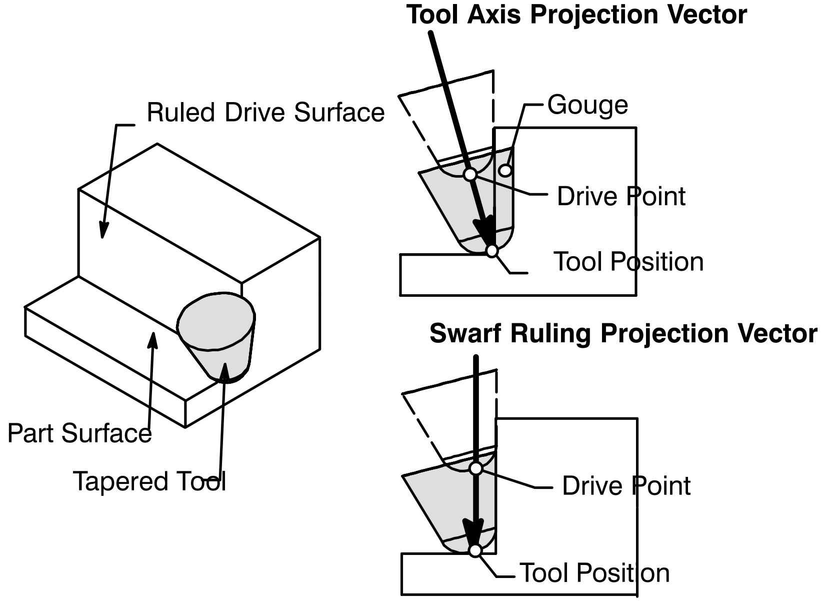
Towards Line allows you to create a projection vector extending from the part surface to a specified line.

Surface Area Drive Method Projection Vectors

Normal to Drive allows you to 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:



The above figure compares the Swarf Ruling projection vector to the Tool Axis projection vector (the Tool Axis projection vector is actually the reverse of the Tool Axis Vector). In each case, 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

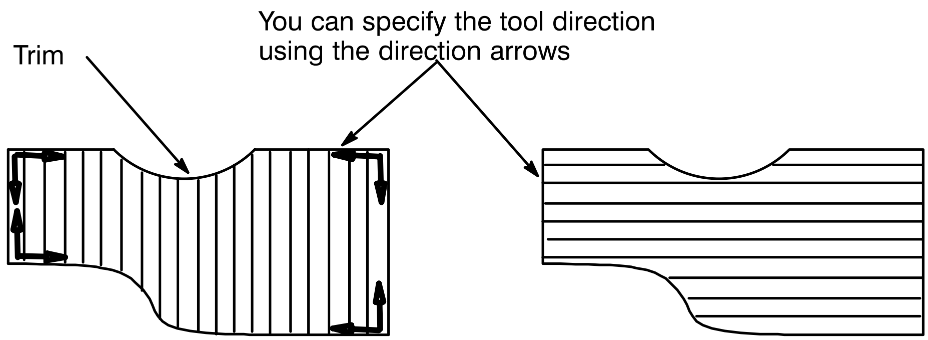
(This Page Intentionally Left Blank)

Zig-Zag Surface Machining

Appendix B

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.



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.

(This Page Intentionally Left Blank)

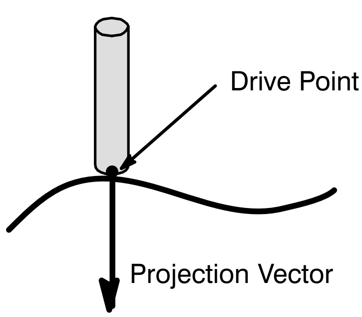
Advanced Surface Contouring

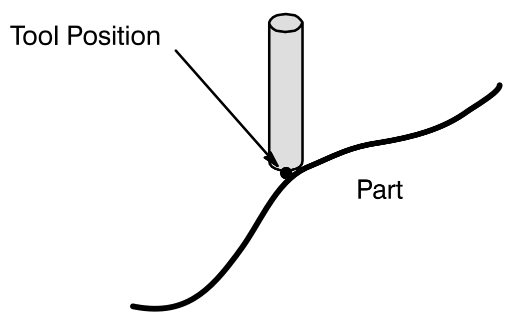
Appendix C

Projection

Mathematics of Projection:

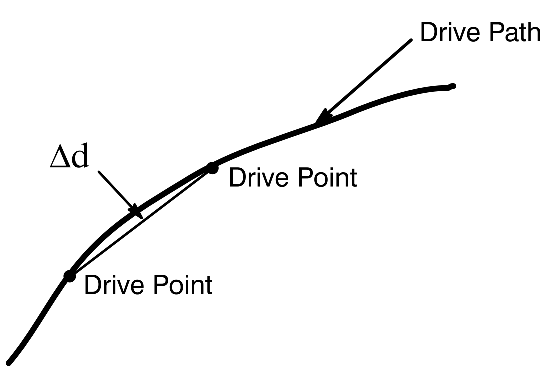
- Place tool end at drive point
- Project tool along projection vector
- Tool stops when making contact with part
- If necessary, adjust the tool axis and repeat the above steps until the tool axis is satisfied
- Add more intermediate drive points to satisfy the Intol/Outol with the part

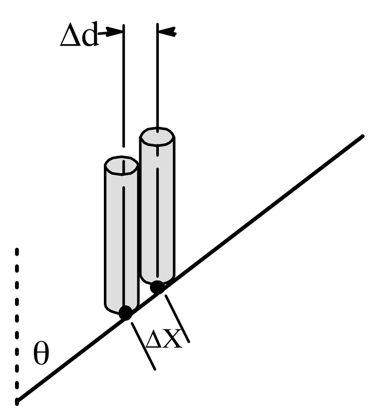




Projection and Steep Surface:

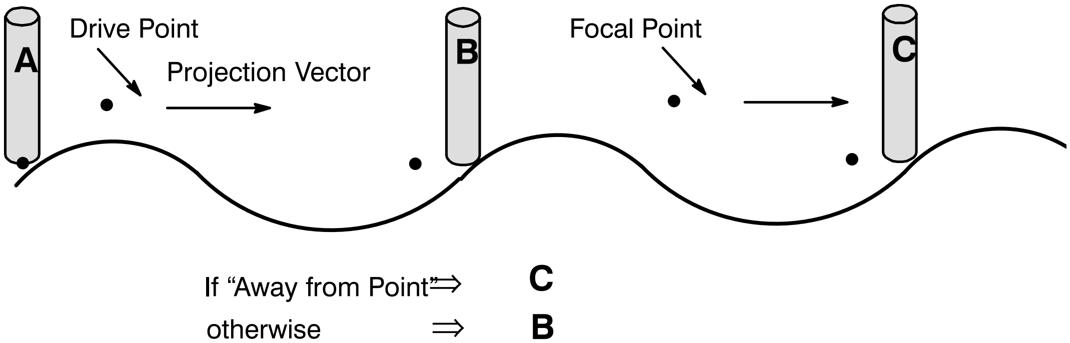
- $\Delta X = \Delta d / \sin\theta \cong \Delta d/\theta$
  ΔX becomes large if θ is very small (steep surface)
- The source of Δd is the chordal deviation of the drive path





Projection and Material Side:

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



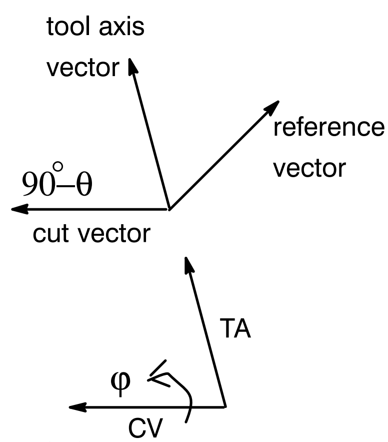
- In the case of Area Milling Drive (no projection vector), the tool axis vector is used to decide Material Side

C

Tool Axis

Definition of Lead/Tilt angles:

Lead = θ
Tilt = φ

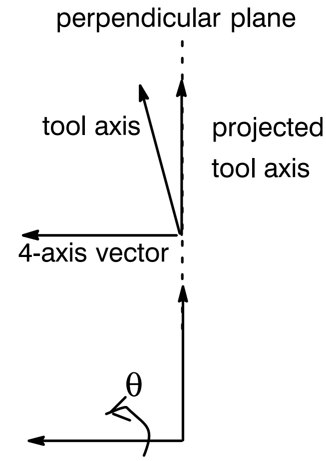


- Begin with cut vector, rotate it toward the Reference vector 90°−θ degrees
- Then rotate around the cut vector φ degrees (counter clockwise)
- Reference vector is the surface normal relative to the part/drive or a vector which is relative to a vector

Definition of 4-axis rotation angle:

Rotation angle= θ

- Compute tool axis vector without 4-axis constraint first
- Project this tool axis vector onto the perpendicular plane of the 4-axis vector
- Rotate the projected tool axis vector along 4-axis vector θ degrees (counterclockwise)

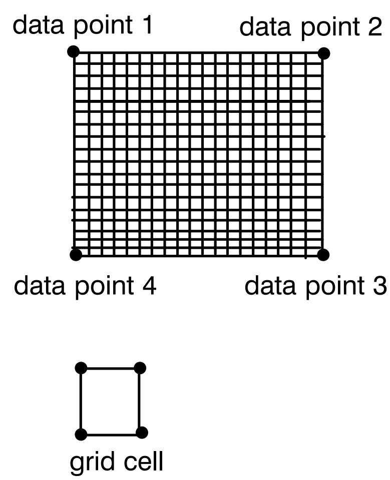


The unconstrained tool axis vector could be:

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

Interpolated tool axis algorithm:

- Divide the whole parameter (u,v) space for the drive surfaces by a 19x19 grid
- Compute the tool axis at each grid point using the data points weighted by the inverse of the distance square
- Inside each grid cell, compute the tool axis vector as the linear / spline interpolation of the tool axis vector at the four corners.

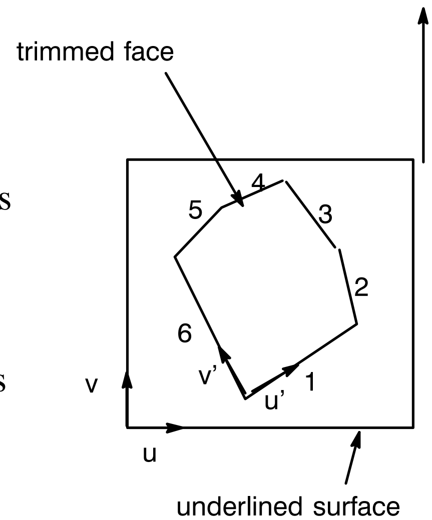


C

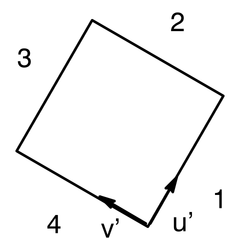
Drive Surface

Remap of drive surface:

Remap algorithm:

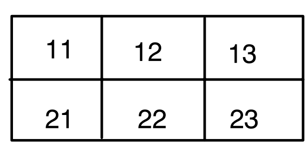


- Merge the exterior edges of the trimmed face to 4 sides
- Re-proportion the parameters of the exterior edges according to the arc length
- Use the arc length proportional edge parameters to construct the new (u',v') space for the trimmed face (Coon's Mapping)
- Finally, align the multiple drive surfaces into a rectangular grid formation



Limitations of remap:

- Fails on 3-sided faces
- Fails on faces that do not have rectangular shape
- May fail on faces with too many edges
- Multiple drive surfaces must be in grid formation

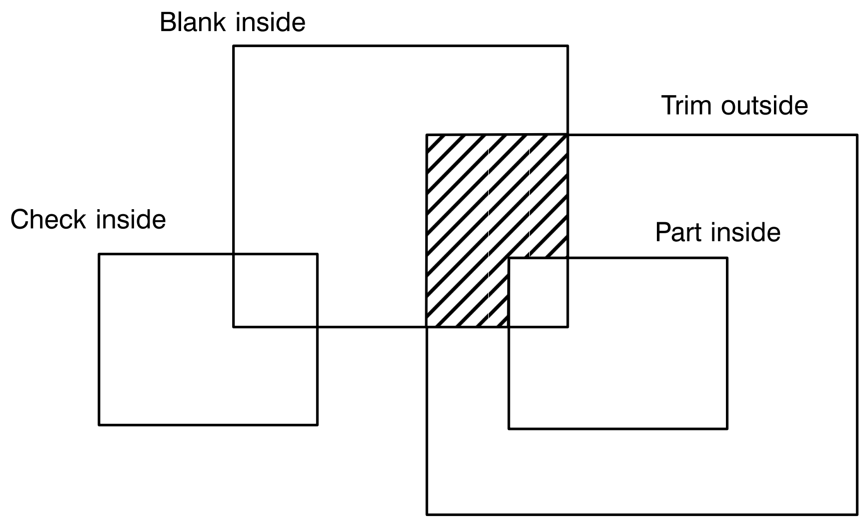


Swarf developable surface:

- Developable surfaces are special kinds of ruled surfaces when the surface normal vectors on any given rule line are the same (ruled surface without twisting)
- Only developable surfaces can be milled by swarfing without undercut or overcut

Planar Milling

- Blank – the region to be included
- Part – the region that can not be violated
- Check – the additional region that can not be violated
- Trim – as a final step, the region to be trimmed away



Boolean Logic

Boundary Drive

- Drive boundary – similar to "blank" if no part containment, otherwise it is like "part"
- Part containment – similar to "blank"

Area Milling Drive

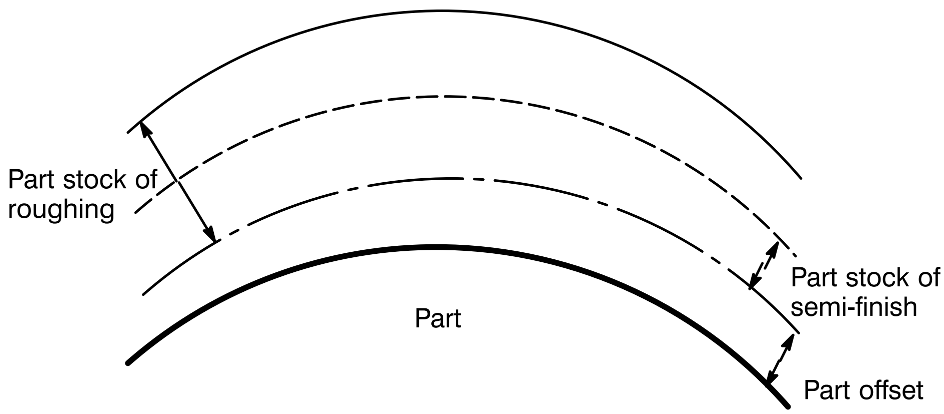
- Cut area – similar to "blank"
- Trim – behaves slightly different from planar milling

Stock

Part offset and part stock

	What	Where
Part Offset	Offset of part as the permanent definition of the final shape of the product	Geometry Group
Part Stock	Leftover material on part by a given operation	Operation

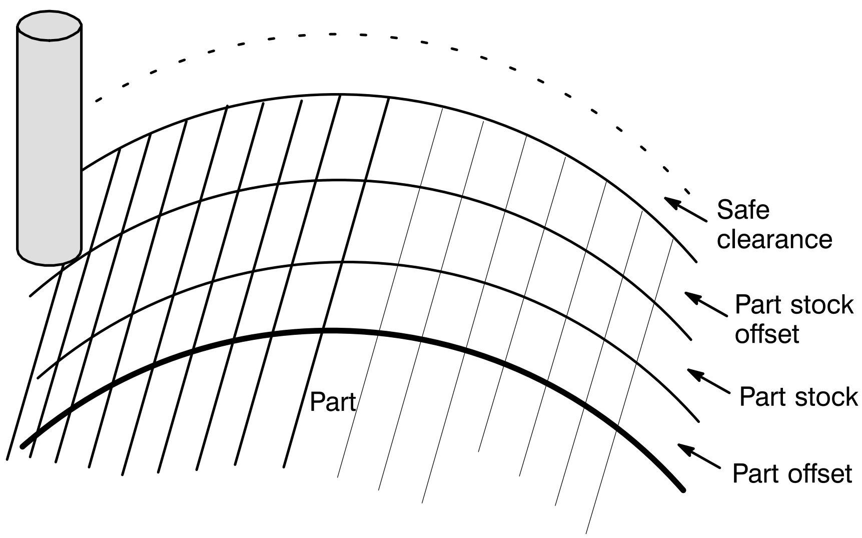
- Part stock is defined on "top" of part offset



Safe clearance and part stock offset

	What	Where
Part Stock Offset	Difference between the part stock from the previous operation and the part stock of the current operation	Operation
Safe Clearance	The additional safety zone for collision checking	Operation

- Safe clearance is defined on “top” of part stock offset



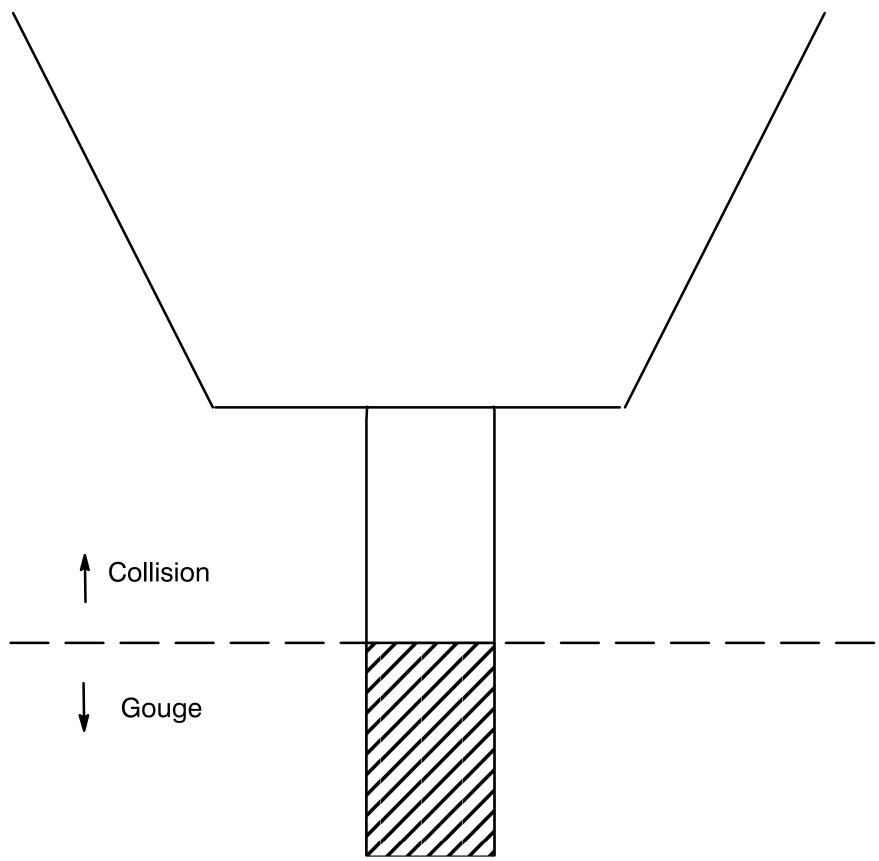
- Part stock offset is used in multiple pass, engage / retract and collision checking
- Safe clearance is used in engage / retract and collision checking

Gouge / Collision

Definitions:

	Rapid moves	**Feed moves**
Cutting part of tool assembly	Collision	Gouge
Non-cutting part of tool assembly	Collision	Collision

- Usually gouge check against part offset + part stock
- Usually collision check against part offset + part stock + part stock offset + safe clearance



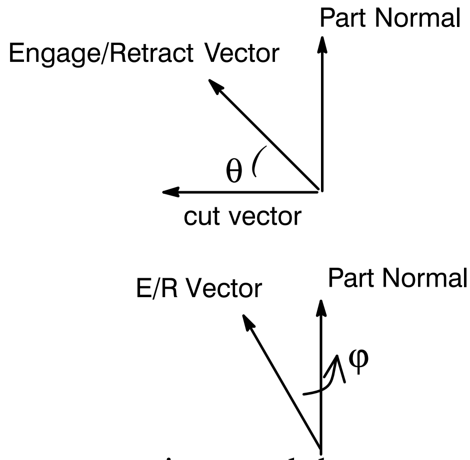
Usage:

	Collision check	Gouge check
Tool path generation	No	Yes on Part
Drive Path Generation	No	Optional on Drive
Engage / Retract	No	Optional on Part
Transfer moves	Optional on Part	Optional on Part
Cut Region computation (Cut Area)	Optional (holder) on Part / Check	Yes on Part
Check Geometry	No	Optional on Check
Gouge Check (Operation Navigator)	No	(No part stock)

Noncut Moves

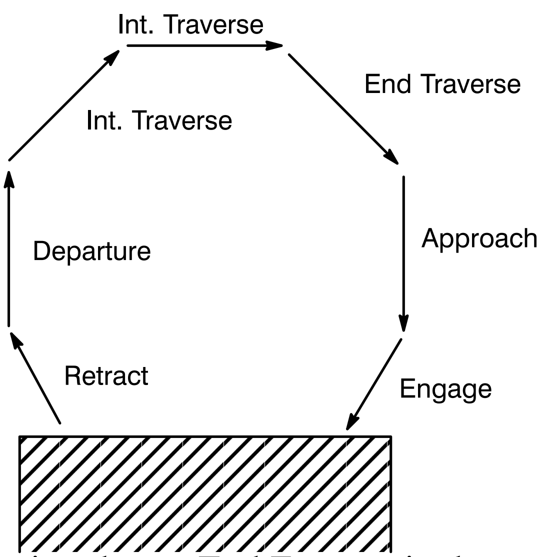
Azimuth / Latitude:

Latitude= θ
Azimuth= φ



- Begin with cut vector, rotate it toward the part normal θ degrees
- Then rotate around the part normal φ degrees (counter clockwise)

End / Intermediate traverse:



- There is only one End Traverse in the sequence, but there may be zero or multiple Int Traverse
- The Start and End positions of the End Traverse move are determined by other moves in the sequence

Glossary

ABS – Absolute coordinate system.

Absolute Coordinate System – Coordinate system in which all geometry is located from a fixed or absolute zero point.

active view – One of up to 49 views per layout in which you can directly work.

angle – In Unigraphics, an angle measured on the X-Y plane of a coordinate system is positive if the direction that it is swept is counterclockwise as viewed from the positive Z axis side of the X-Y plane. An angle swept in the opposite direction is said to be negative.

arc – An incomplete circle; sometimes used interchangeably with the term "circle."

ASCII – American Standard Code for Information Interchange. It is a set of 8-bit binary numbers representing the alphabet, punctuation, numerals, and other special symbols used in text representation and communications protocol.

aspect ratio – The ratio of length to height which represents the change in size of a symbol from its original.

assembly – A collection of piece parts and sub-assemblies representing a product. In Unigraphics, an assembly is a part file which contains components.

assembly part – A Unigraphics part file which is a user-defined, structured combination of sub-assemblies, components and/or objects.

associativity – The ability to tie together (link) separate pieces of information to aid in automating the design, drafting, and manufacture of parts in Unigraphics.

attributes – Pieces of information that can be associated with Unigraphics geometry and parts such as assigning a name to an object.

block font – A Unigraphics character font which is the default font used for creating text in drafting objects and dimensions.

body – Class of objects containing sheets and solids (see solid body and sheet body).

bottom-up modeling – Modeling technique where component parts are designed and edited in isolation of their usage within some higher level assembly. All assemblies using the component are automatically updated when opened to reflect the geometric edits made at the piece part level.

boundary – A set of geometric objects that describes the containment of a part from a vantage point.

CAD/CAM – Computer Aided Design/Computer Aided Manufacturing.

category, layer – A name assigned to a layer, or group of layers. A category, if descriptive of the type of data found on the layers to which it is assigned, will assist the user in identifying and managing data in a part file.

chaining – A method of selecting a sequence of curves which are joined end-to-end.

circle – A complete and closed arc, sometimes used interchangeably with the term "arc."

component – A collection of objects, similar to a group, in an assembly part. A component may be a sub-assembly consisting of other, lower level components.

component part – The part file or "master" pointed to by a component within an assembly. The actual geometry is stored in the component part and referenced, not copied, by the assembly. A separate Unigraphics part file that the system associates with a component object in the assembly part.

cone direction – Defines the cone direction using the Vector Subfunction.

cone origin – Defines the base origin using the Point Subfunction.

half angle – The half vertex angle defines the angle formed by the axis of the cone and its side.

constraints – Refer to the methods you can use to refine and limit your sketch. The methods of constraining a sketch are geometric and dimensional.

construction points – Points used to create a spline. Construction points may be used as poles (control vertices), defining points, or data points. See POLES, DEFINING POINTS, and DATA POINTS.

control point – Represents a specific location on an existing object. A line has three control points: both end points and the midpoint of the line. The control point for a closed circle is its center, while the control points for an open arc are its end and midpoints. A spline has a control point at each knot point. A control point is a position on existing geometry. Any of the following points: 1. Existing Points 2. Endpoints of conics 3. Endpoints and midpoints of open arcs 4. Center points of circles 5. Midpoints and endpoints of lines 6. Endpoints of splines.

convert curve – A method of creating a b-curve in which curves (lines, arcs, conics or splines) may be selected for conversion into a b-curve.

Coordinate System – A system of axes used in specifying positions (CSYS).

counterclockwise – The right-hand rule determines the counter- clockwise direction. If the thumb is aligned with the ZC axis and pointing in the positive direction, counterclockwise is defined as the direction the fingers would move from the positive XC axis to the positive YC axis.

current layout – The layout currently displayed on the screen. Layout data is kept in an intermediate storage area until it is saved.

curve – A curve in Unigraphics is any line, arc, conic, spline or b-curve. A geometric object; this may refer to a line, an arc, a conic, or a spline.

defaults – Assumed values when they are not specifically defined.

defining points – Spline construction points. Splines created using defining points are forced to pass through the points. These points are guaranteed to be on the spline.

degree-of-freedom arrows – Arrow-like indicators that show areas that require more information to fully constrain a sketch.

design in context – The ability to directly edit component geometry as it is displayed in the assembly. Geometry from other components can be selected to aid in the modeling. Also referred to as edit in place.

dimensional constraint – This is a scalar value or expression which limits the measure of some geometric object such as the length of a line, the radius of an arc, or the distance between two points.

directory – A hierarchical file organization structure which contains a list of filenames together with information for locating those files.

displayed part – The part currently displayed in the graphics window.

edit in place – See design in context.

emphasize work part – A color coding option which helps distinguish geometry in the work part from geometry in other parts within the same assembly.

endpoint – An endpoint of a curve or an existing point.

expression – An arithmetic or conditional statement that has a value. Expressions are used to control dimensions and the relationships between dimensions of a model.

face – A region on the outside of a body enclosed by edges.

feature – An all-encompassing term which refers to all solids, bodies, and primitives.

file – A group or unit of logically related data which is labeled or "named" and associated with a specified space. In Unigraphics, parts, and patterns are a few types of files.

filtering – See object filtering.

font box – A rectangle or "box" composed of dashed line objects. The font box defines the size, width and spacing of characters belonging to a particular font.

font, character – A set of characters designed at a certain size, width and spacing.

font, line – Various styles of lines and curves, such as solid, dashed, etc.

free form feature – A body of zero thickness. (see body and sheet body)

generator curve – A contiguous set of curves, either open or closed, that can be swept or revolved to create a body.

geometric constraint – A relationship between one or more geometric objects that forces a limitation. For example, two lines that are perpendicular or parallel specifies a geometric constraint.

grid – A rectangular array of implied points used to accurately align locations which are entered by using the "screen position" option.

guide curve – A set of contiguous curves that define a path for a sweep operation.

virtual intersection – Intersection formed by extending two line segments that do not touch to the position that they cross. The line segments must be non-parallel and coplanar.

inflection – A point on a spline where the curve changes from concave to convex, or vice versa.

interactive step – An individual menu in a sequence of menus used in performing a Unigraphics function.

isometric view (Tfr-ISO) – Isometric view orientation – one where equal distances along the coordinate axes are also equal to the view plane. One of the axes is vertical.

knot points – The defining points of a spline. Points along a B-spline, representing the endpoints of each spline segment.

layer – A layer is a partition of a part. Layers are analogous to the transparent material used by conventional designers. For example, the user may create all geometry on one layer, all text and dimensions on a second, and tool paths on a third.

layout – A collection of viewports or window areas, in which views are displayed. The standard layouts in Unigraphics include one, two, four or six viewports.

layouts – Standard layouts are available to the user. These include:
L1 – Single View,
L2 – Two Views,
L3 – Two Views,
L4 – Four Views,
L6 – Six Views.

Information window – The window used in listing operations, such as **Info**.

loaded part – Any part currently opened and in memory. Parts are loaded explicitly using the *File→Open* option and implicitly when they are used in an assembly being opened.

menu – A list of options from which the user makes a selection.

model space – The coordinate system of a newly created part. This is also referred to as the "absolute coordinate system." Any other coordinate system may be thought of as a rotation and/or translation of the absolute coordinate system.

name, expression – – The name of an expression is the single variable on the left hand side of the expression. All expression names must be unique in a part file. Each expression can have only one name. See expression.

objects – All geometry within the Unigraphics environment.

offset face – A Unigraphics surface type created by projecting (offsetting) points along all the normals of a selected surface at a specified distance to create a "true" offset.

options – A number of various alternatives (functions, modes, parameters, etc.) from among which the user can choose.

origin – The point X = 0, Y = 0, Z = 0 for any particular coordinate system.

parametric design – Concept used to define and control the relationships between the features of a model. Concept where the features of the model are defined by parameters.

part – A Unigraphics file containing a .prt extension. It may be a piece part containing model geometry, a sub-assembly, or a top-level assembly.

part or model – A collection of Unigraphics objects which together may represent some object or structure.

partially loaded part – A component part which, for performance reasons, has not been fully loaded. Only those portions of the component part necessary to render the higher level assembly are initially loaded (the reference set).

point set – A distribution of points on a curve between two bounding points on that curve.

Point Subfunction Menu – A list of options (methods) by which positions can be specified in Unigraphics.

read-only part – A part for which the user does not have write access privilege.

real time dynamics – Produces smooth pan, zoom, and rotation of a part, though placing great demand on the CPU.

Refresh – A function which causes the system to refresh the display list on the viewing screen. This removes temporary display items and fills in holes left by ***Blank*** or ***Delete.***

right-hand rule, conventional – The right-hand rule is used to determine the orientation of a coordinate system. If the origin of the coordinate system is in the palm of the right fist, with the back of the hand lying on a table, the outward extension of the index finger corresponds to the positive Y axis, the upward extension of the middle finger corresponds to the positive Z axis, and the outward extension of the thumb corresponds to the positive X axis.

right-hand rule for rotation – The right-hand rule for rotation is used to associate vectors with directions of rotation. When the thumb is extended and aligned with a given vector, the curled fingers determine the associated direction of rotation. Conversely, when the curled fingers are held so as to indicate a given direction of rotation, the extended thumb determines the associated vector.

screen cursor (cursor) – A marker on the screen which the user moves around using some position indicator device. Used for indicating positions, selecting objects, etc. Takes the form of a full-screen cross.

sheet – A object consisting of one or more faces not enclosing a volume. A body of zero-thickness. Also called sheet body.)

sketch – A collection of geometric objects that closely approximates the outline of a particular design. You refine your sketch with dimensional and geometric constraints until you achieve a precise representation of your design. The sketch can then be extruded or revolved to obtain a 3D object or feature.

Sketch Coordinate System (SCS) – The SCS is a coordinate system which corresponds to the plane of the sketch. When a sketch is created the WCS is changed to the SCS of the new sketch.

solid body – An enclosed volume. A type of body (see Body).

spline – A smooth free-form curve.

stored layout – The last saved version of a layout.

stored view – The last saved version of a view.

string – A contiguous series of lines and/or arcs connected at their end points.

sub-assembly – A part which both contains components and is itself used as a component in higher-level assemblies.

surface – The underlying geometry used to define a face on a sheet body. A surface is always a sheet but a sheet is not necessarily a surface (see sheet body). The underlying geometry used to define the shape of a face on a sheet.

system – The Unigraphics System.

temporary part – An empty part which is optionally created for any component parts which cannot be found in the process of opening an assembly.

top-down modeling – Modeling technique where component parts can be created and edited while working at the assembly level. Geometric changes made at the assembly level are automatically reflected in the individual component part when saved.

trim – To shorten or extend a curve.

trimetric view (Tfr-Tri) – A viewing orientation which provides you with an excellent view of the principal axes. In Unigraphics II, the trimetric view has the Z-axis vertical. The measure along the X-axis is 7/8 of the measure along Z, and the measure along the Y-axis is 3/4 of the measure along Z.

Unigraphics – A computer based turnkey graphics system for computer-aided design, drafting, and manufacturing, produced by UGS.

units – The unit of measure in which you may work when constructing in Unigraphics. Upon log on, you may define the unit of measure as inches or millimeters.

upgraded component – A component which was originally created pre-V10 but has been opened in V10 and upgraded to remove the duplicate geometry.

version — A term which identifies the state of a part with respect to a series of modifications that have been made to the part since its creation.

view — A particular display of the model. View parameters include view orientation matrix; center; scale; X,Y and Z clipping bounds; perspective vector; drawing reference point and scale. Eight standard views are available to the user: Top, Front, Right, Left, Bottom, Back, Tfr-ISO (top-front-right isometric), and Tfr-Tri (top-front-right trimetric).

view dependent edit — A mode in which the user can edit a part in the current work view only.

view dependent modifications — Modifications to the display of geometry in a particular view. These include erase from view and modify color, font and width.

view dependent geometry — Geometry created within a particular view. It will only be displayed in that view.

WCS — Work Coordinate System.

WCS, work plane — The WCS (Work Coordinate System) is the coordinate system singled out by the user for use in construction, verification, etc. The coordinates of the WCS are called work coordinates and are denoted by XC, YC, ZC. The XC-YC plane is called the work plane.

Work Coordinate System — See WCS.

work layer — The layer on which geometry is being constructed. You may create objects on only one layer at a time.

work part — The part in which you create and edit geometry. The work part can be your displayed part or any component part which is contained in your displayed assembly part. When displaying a piece part, the work part is always the same as the displayed part.

work view — The view in which work is being performed. When the creation mode is view dependent, any construction and view dependent editing that is performed will occur only in the current work view.

XC axis — X-axis of the work coordinate system.

YC axis — Y-axis of the work coordinate system.

ZC axis — Z-axis of the work coordinate system.

Index

IN

T

U

V

W

IN

X

Y

Z

Reference Chart Tear Outs

These tear out reference charts are provided for your convenience.

(This Page Intentionally Left Blank)

Unigraphics NX
Multi–Axis Machining Course Agenda

Day 1 **Morning**

Overview of Variable Contour

Workbook Introduction *The Manufacturing Process*

Lesson 1 Introduction to Four and Five-Axis Machining

Workbook Section 1 *Drilling the Top Flange*

Lesson 2 Sequential Mill Basics

Afternoon

Lesson 3 Sequential Mill Intermediate

Lesson 4 Sequential Mill Advanced

Workbook Section 2 *Sequential Mill – Cutting the Manifold Flange*

Day 2 **Morning**

Lesson 5 Variable Contour – Basics

Workbook Section 3 *Variable Contour – Cutting the Manifold Flange*

Afternoon

Lesson 6 Variable Contour – Advanced

(This Page Intentionally Left Blank)

STUDENT PROFILE

In order to stay in tune with our customers we are requesting that you take a little time to answer these questions. This information will be kept confidential, and will not be shared with anyone outside of Education Services. PLM Solutions Education Services thanks you for your participation and hopes your training experience will be an outstanding one. Please print the following:

Your Name____**Date**___________________________

Course Number__________________**Course Title**____

Employer____

When is your planned departure time?_______________ **U.S. citizen?** ❑ **YES** ❑ **NO**

1. Please select the job that best describes your present one.

❑ Designer MFG	❑ Designer Product	❑ Designer Tooling	❑ Quality Engineer
❑ Engineer MFG	❑ Engineer Product	❑ Engineer Tooling	❑ Process Planner
❑ NC Dies	❑ NC Other	❑ NC Tooling	❑ Supplier Quality
❑ Drafter MFG	❑ Drafter Product	❑ Drafter Tooling	❑ Workflow Admin.
❑ Manager Design	❑ Manager Engineering	❑ Manager Shop Floor	❑ Collaboration Admin.
❑ Sys. Admin Local Network	❑ Sys. Admin Enterprise	❑ Systems Manager	❑ Configuration Analyst

2. Your job responsibilities are:____
3. Reason for training____
4. What other CAD/CAM/CAE /PDM software have you used?___________________________
5. What types of parts/data do you model/analyze or store?___________________________

    ____

6. Have you ever had any other instructor–lead/ on–line or self–paced classes for the following:

Subject	From Whom	When	Course Name
❑ **Unigraphics**			
❑ **I–deas**			
❑ **Imageware**			
❑ **Teamcenter Manufacturing**			
❑ **Teamcenter Engineering (I–Man)**			
❑ **Team Center Enterprise (Metaphase)**			
❑ **Dimensional Management / Visualization**			

Please list your CAD/CAM/CAE/PDM experience and indicate the level of understanding by using the terms novice, intermediate or advanced.

Tool Axis Control

The Variable Contour Tool Axes can be grouped with regards to the geometry that determines tool axis.

Your 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.

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
Normal To Part	x	X	x	x	x	x
Relative To Part	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
Interpolate	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	

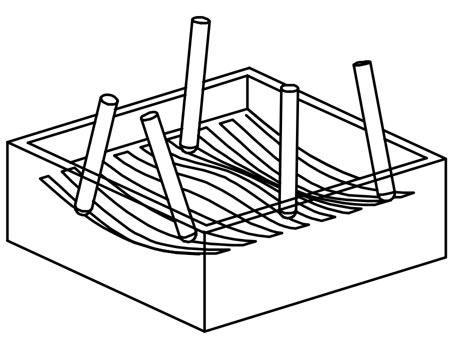
5-Axis Tool Axis Control	
Variable Contour	**Sequential Mill**
Toward or Away From Point	**Thru Fixed Point**
Normal to Part	**Normal To PS**
Normal To Drive	**Normal To DS**
Swarf Drive	**Parallel to PS** **Parallel to DS**
Relative To Drive	**At Angle to DS** **At Angle to PS**
—	**Tangent to PS**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

Comparison between variable contour and sequential mill

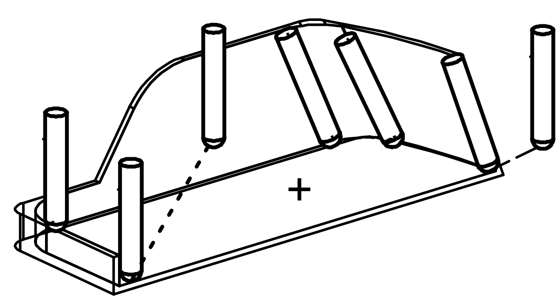
The overriding consideration in choosing between Variable Contour and Sequential Mill is “Which method creates the best tool path.”

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
The preferred method for area milling	The preferred method for linear milling
Primary cutting with the bottom of the tool	Primary cutting with the side of the tool
Numerous drive methods for containing the tool path	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 respecify tool axis during operation
Edits apply to entire tool path	Edits applied to part of the tool path
Cuts convex walls best.	Cuts angled (overcut & undercut) walls best
Easier to create operation	Numerous steps to an operation
Easier for multi-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 DS / At Angle to PS
Dual 4-Axis on Part	—
Dual 4-Axis on Drive	—
—	Tangent to PS
—	Tangent to DS
—	Proj DS Normal
—	Proj PS Normal
5 Axis	
Away From Point	Thru Fixed Point
Toward 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
Interpolate	—
Same as Drive Path	—
User Function	—
—	Tangent to PS
—	Tangent to DS
—	Fan

Projection Methods

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

(This Page Intentionally Left Blank)

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 Intentionally Left Blank)

Hot Key Chart

Hot Key	Function	Hot Key	Function
Ctrl-A	Assembly Navigator	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		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	
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	Model Navigator	Ctrl-Shift-Z	View, Operation, Zoom (full menu)

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

Level 1 Evaluation

Course Code_______ Course__________________ Date________ Instructor____________

1. STRONGLY DISAGREE	2. DISAGREE	3. SOMEWHAT AGREE	4. AGREE	5. STRONGLY AGREE

Content:

1. The course objectives were clear_______________________________	1	2	3	4	5
3. The goals and objectives for this course were met_______________	1	2	3	4	5
4. The course flowed in a logical and meaningful manner___________	1	2	3	4	5
5. The course was the appropriate length for the info taught________	1	2	3	4	5

Materials:

1. The training materials supported the course objectives__________	1	2	3	4	5
2. The training materials were logically sequenced________________	1	2	3	4	5
3. Enough information was provided to complete workshops________	1	2	3	4	5
4. The course materials provided clear and descriptive directions____	1	2	3	4	5
5. The exercises and workshops supported the learning experience___	1	2	3	4	5
6. The materials were easy to read and understand________________	1	2	3	4	5

Instructor:

1. The instructor was knowledgeable about the subject					
2. He/She answered my questions appropriately__________________	1	2	3	4	5
3. He/She was well prepared to deliver the course________________	1	2	3	4	5
4. He/She was organized and well spoken_______________________	1	2	3	4	5
5. The instructor encouraged questions in class__________________	1	2	3	4	5
6. The instructor made good use of the training time______________	1	2	3	4	5
7. The instructor completed the course_________________________	1	2	3	4	5
8. The instructor used examples relevant to the course____________	1	2	3	4	5
9. The instructor provided enough time to complete the exercises___	1	2	3	4	5
10. The instructor used review tests and gave feedback_____________	1	2	3	4	5

Student Self–Evaluation:

1. I met the prerequisites for the class (I had the skills I needed)_____	1	2	3	4	5
2. I will be able to use the skills I have learned on my job___________	1	2	3	4	5
3. My expectations for this course were met_____________________	1	2	3	4	5
4. I am confident that with practice I will become proficient________	1	2	3	4	5

Facilities:

1. The computer equipment was reliable_________________________	1	2	3	4	5
2. The software performed properly____________________________	1	2	3	4	5
3. The overhead projection unit was clear and working properly_____	1	2	3	4	5
4. The registration and confirmation process was efficient_________	1	2	3	4	5
5. The training facilities were comfortable and clean and provided a good learning environment_________________________________	1	2	3	4	5

(Over)

Hotels: (Optional)

1. **Was this hotel recommended during your registration process?___________** ☐ YES ☐ NO

   If not how was it chosen____

2. **Was the hotel clean?____** ☐ YES ☐ NO

3. **Did it have a full–serve restaurant?__________________________________** ☐ YES ☐ NO

4. **Overall impression of the Hotel______________________________**

1	2	3	4	5

5. **What was the name of the hotel?______________________________**

☐ **Please check this box if you would like your comments below featured in our training publications. (Your name is required at the bottom of this form)**

☐ **Please check this box if you would like to receive more information on our other courses and services. (Your name is required at the bottom of this form)**

We would appreciate it if you would take a minute to add some personal comments on this form. These comments help us to better understand the ever–changing student needs.

Thank you for choosing EDS PLM Education Services, and we hope we can be of help to you in the future.

Comments:____

Name Optional:_________________________________ **Location**_____________________

Software Version__________________