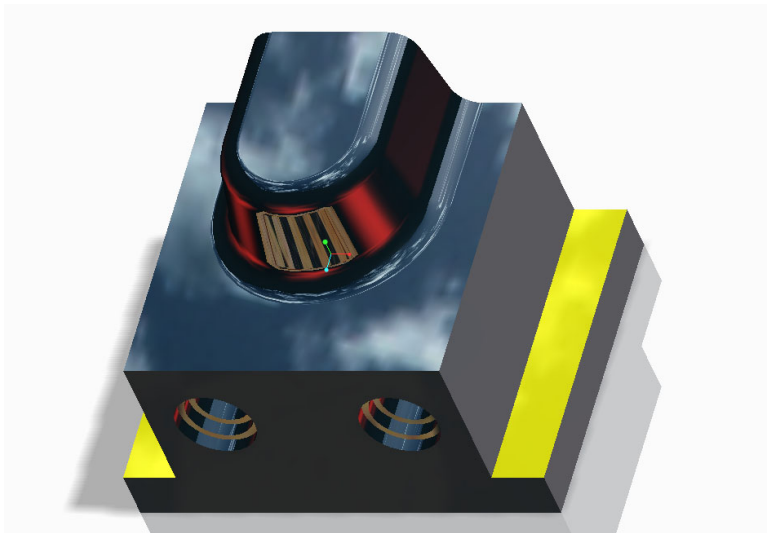


PART 6: SURFACE MILLING

CONVENTIONAL SURFACE MILL

Create a tool path to machine the top faces (shown in cyan below) of the provided mold insert.

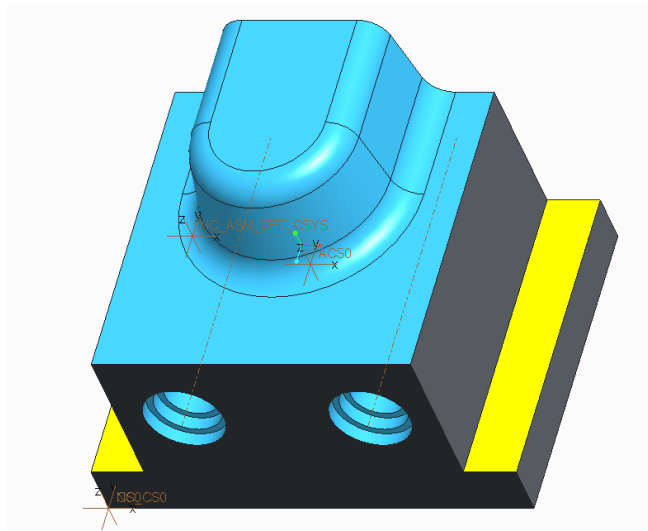


Before starting this tutorial:

Ensure that you have downloaded all CAM files from the write-protected drive (**P:\Courses\41617-CAM**) and placed them on your own drive (**M:**)

Set the Working Directory to the directory on your own drive where you placed the CAM files.

1. Open the manufacturing model called: **surf1.asm**



2. Use the existing NC Sequence.

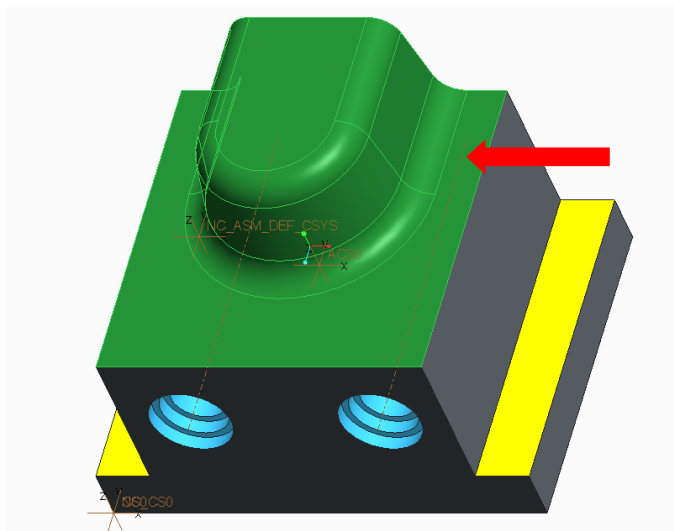
- In the Model tree, click **1. Surface Milling [OP 010]** and choose **Edit Definition**.
- In the **Menu Manager** choose **Seq Setup**.
- In the **Seq Setup** walk-through menu list, activate **Surfaces** and **Define Cut** (pre-chosen)
- Choose **Done**

*Note: This **NC Sequence** is setup using a 3-axis mill with a 25 mm diameter ball nose end mill tool. The other important machining parameters that are preset include:*

CUT_FEED	50
STEP_OVER	1.5
SCALLOP_HGT	1.25
CUT_ANGLE	0
SCAN_TYPE	TYPE_1
SPINDLE_SPEED	1000
COOLANT_OPTION	OFF
CLEAR_DIST	2.5

3. Select the surfaces that you want to machine.

- Choose **Done** to accept the default of **Model** (upon which to select the surfaces)
- Select all of the top surfaces on the insert

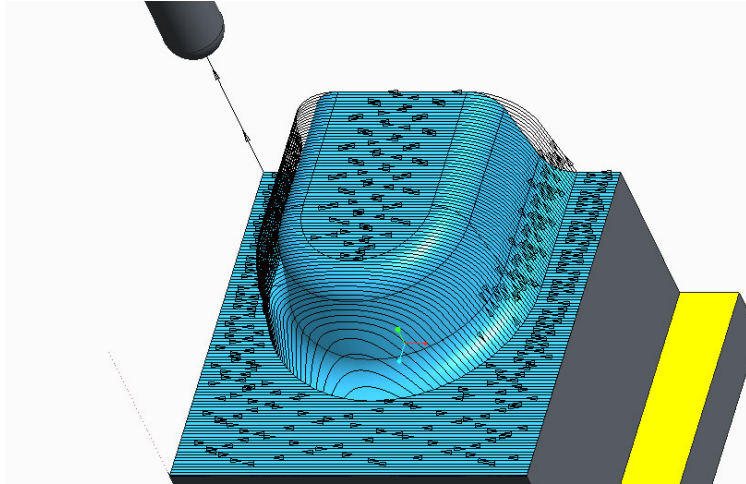


*The order in which you select them is not important. To select more than one surface, you must hold down the **CTRL** key as you select.*

Here all the top surfaces are chosen

- Choose **OK** in the **Select** box

4. Play the path.




The tool path should generate like this

CONVENTIONAL MILL PARAMETERS

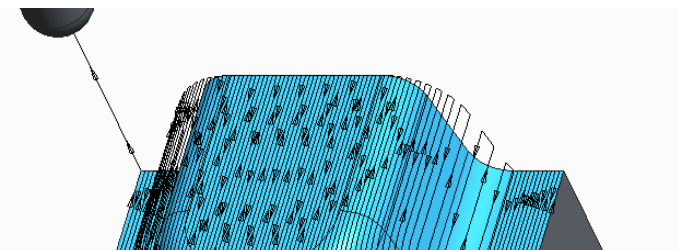
Alter the tool path on the insert you machined in the last Section by changing the **cut angle** and **scallop height** parameters

1. Change the **cut angle** to **90** degrees.

- Close the **PLAY PATH** dialog
- Select the **Step Parameters** icon  (On the **Mill** pane, on the right end of the ribbon)
- In the **Param Tree** window, select the **CUT_ANGLE** parameter and type **90** instead of **0**
- Choose **OK** to exit **Parameter Setup** window

2. Play the path..

- Choose **Screen Play**
- Choose the **Play Forward** action button



- The cut motion is now parallel to the "Y" axis of the machine coordinate system instead of parallel to the "X" axis.

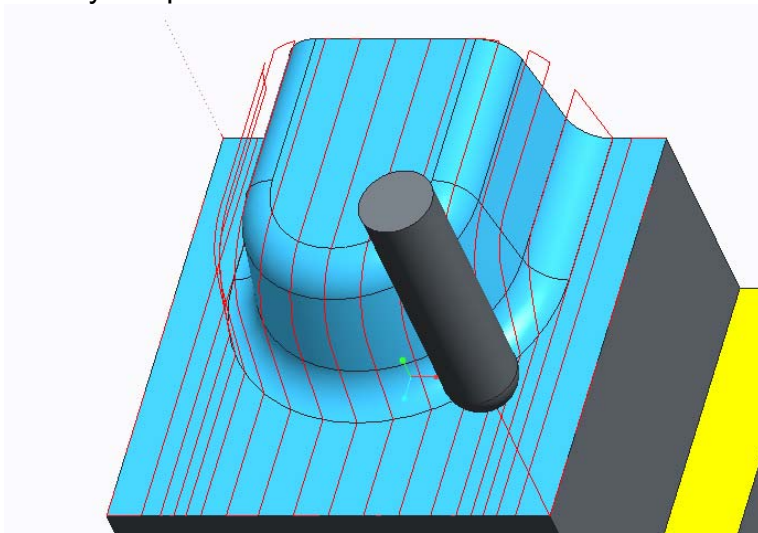
3. Now, change the scallop height to **5 mm**

- Choose the **Step Parameters** icon 
- Change **SCALLOP_HGT** to **5.0**

*Remember, the system checks the **Step Over** calculated from the **Scallop Height** specified in the **Param Tree** window against the **Step Over** distance specified and uses the smaller number. If you increase the **Scallop Height** and don't increase the **Step Over** number you may not see any change in the number of tool passes displayed. Increase the **Step Over** parameter so it will not play a part in the calculations.*

- Change **Step_Over** to **12 mm**
- Exit the **Parameter Setup** window

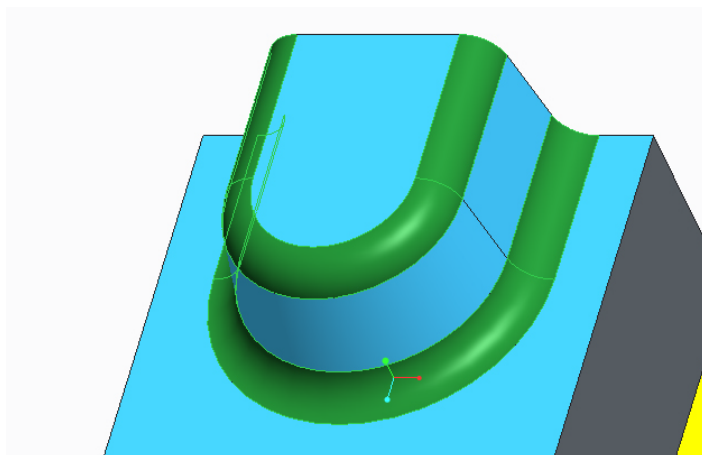
4. Play the path.



*You can see how increasing the **Scallop Height** decreased the number of milling passes on the part. You notice however there is still a "bunching up" of tool passes on the radius surfaces at the bottom of the insert.*

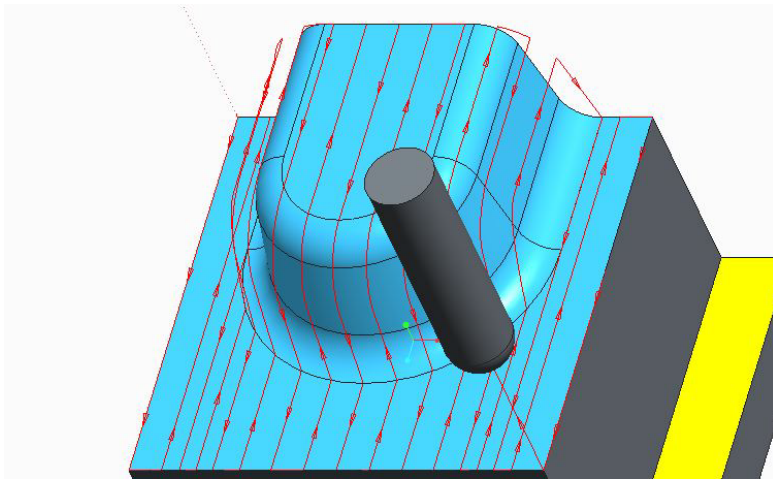
5. Remove the rounded surfaces on the insert from the scallop height calculation.

- Choose **Seq Setup**
- Choose **ScallopSrf**
- Choose **Done**
- Select the 6 rounded surfaces on the insert to **exclude** from the scallop height computation...remember, you must hold down the **CTRL** key to select several surfaces



- Choose **OK**
- Choose **Done/Return**

6. Play the path.



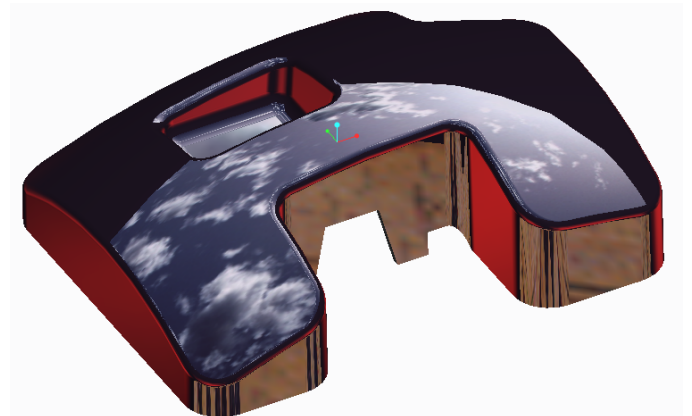
Notice that the spacing is "spread out" a little more at the bottom of the insert. Removing these surfaces from the scallop height calculation had this affect. The spacing is still closer however, on the side wall where the cuts have been spread out on the vertical walls in the **Z** direction, but not in the **X** direction.

- Save your work: **Choose File > Save as > Save a backup.** Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.



End of demonstration

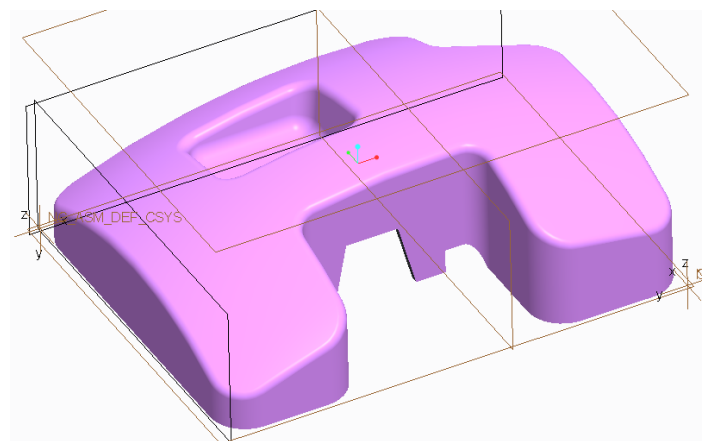
Machine the top surface of the housing shown left using two different scan types.



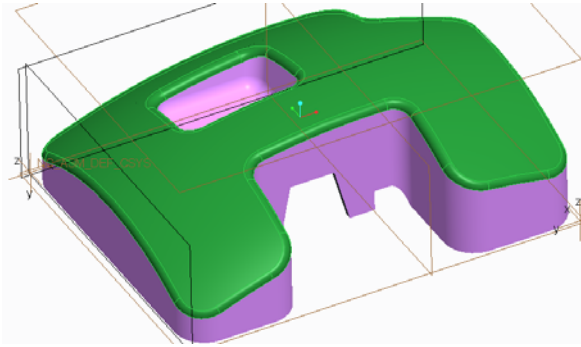
1. Open the **surf2.asm** manufacturing model.

2. Use the existing **NC Sequence**.

- In the **Model tree**, rightclick **1. Surface Milling (OP 010)** and choose **Edit Definition**. *In the appearing Walk-through Menu, **Surfaces** and **Define Cut** are pre-chosen*
- Choose **Done**
- Choose **Done**



3. Select the surfaces on the top of the housing. This includes the large surface, the round which runs around the outside top of the part and the round at edge of the recess. The surfaces are shown below.

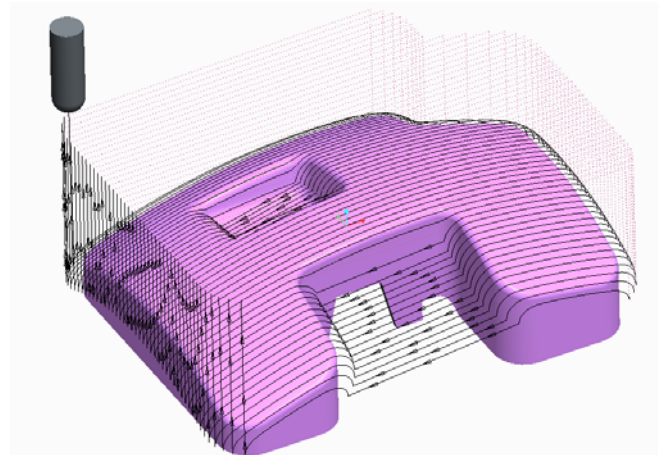


*The surfaces are collected one by one, using the **CTRL** button*

- Choose **OK** in the **Select** box

4. Play the path. The results should be as shown left.

*This path was created using the default scan type of **TYPE_1**. Notice that the tool moved straight across openings that it encountered. This may not be acceptable....change to scan type **TYPE_3**.*

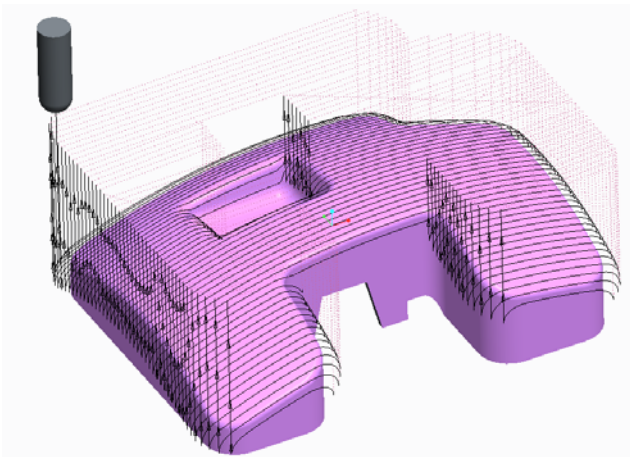


- Choose **Close**

5. Change to scan type **TYPE_3**.

- Choose the **Step Parameters** icon () to bring up the **Param Tree** window.
- Change the **SCAN_TYPE** to **TYPE_3**
- Choose **OK** to exit the **Parameter setup** window.

6. Play the path.



The tool now mills the "zone" on one side of the opening and then moves to the other side. Machining time is saved by not "machining air" as the cutter moves across the openings.

- Save your work: Choose **File > Save as > Save a backup**. Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.

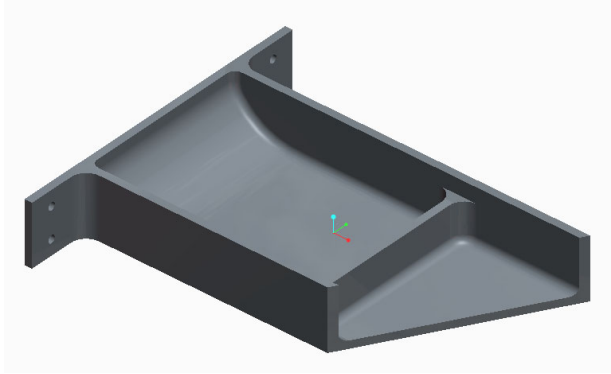
7. Return to the main menu, close all files.



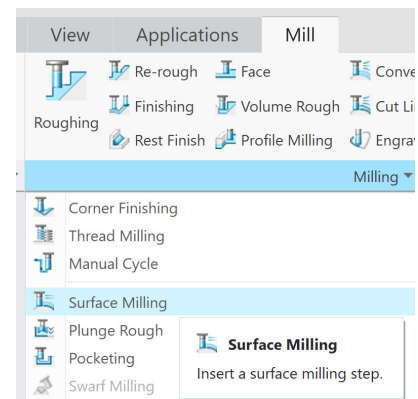
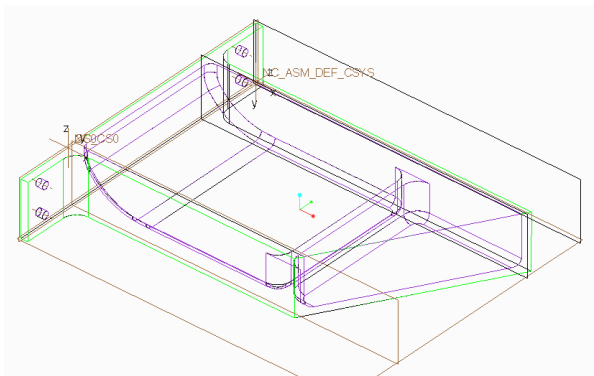
End of Practise

CUT LINE SURFACE MILL

Machine the curved surface on the aircraft frame support shown below using **By Edge** machining.



1. Open the manufacturing model: **surf4.asm**.



2. Start a new **NC Sequence**.

- In the top ribbon choose the **Mill** pane / **Surface Milling** - *The Menu Manager / Sec Setup appears showing Tool, Parameters, Retract Surf, Surfaces and Define Cut pre-chosen. If any of these are missing, activate them yourself.*
- Choose **Check Surfaces** additionally - *Since the surfaces you want to machine lie inside a pocket, you must specify Check Surfaces to keep the tool from hitting the sides of the pocket..*
- Choose **Done**

3. Select a tool.

- Under **Type**, choose **Milling**
- Click in the **Cutter Diameter** text field and enter **12.5**
- Click in the **Corner Radius** text field and enter **6.25**
- Click in the **Length** text field and enter **75**
- Choose the **Apply** action button
- Choose **Yes** to confirm the tool change
- Choose **OK** - *the Tool Dialogue closes, and the Edit Parameters Dialogue opens next...*

4. Retrieve the provided parameters.

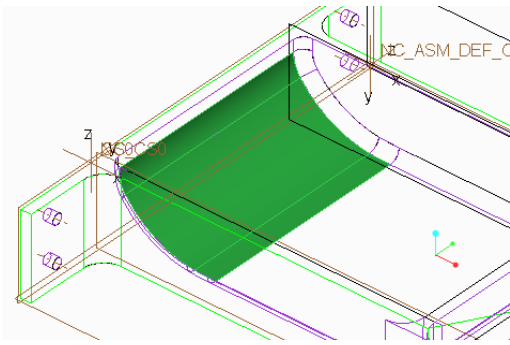
- In the **Edit Parameters** dialogue choose **File / Open**
- Choose **surf4.mil** from the list
- Choose **Open**
- Ignore and close the error dialogue, if it shows.
- Choose **OK** - *the Edit Parameters Dialogue closes, and the Retract Setup opens next...*

5. Setup the Retract Plane.

- If the **NC_CS0** Coordinate system is not preselected in the **Retract Setup** dialogue , select it in the **Model Tree**
- Set (offset) **Value** to **10**
- Choose **OK** - *the Retract Setup Dialogue closes, and the Define Cut opens next...*

6. Select the surfaces you want to machine. - *The system wants you to on the model to select which surfaces to mill.*

- Choose **Done**

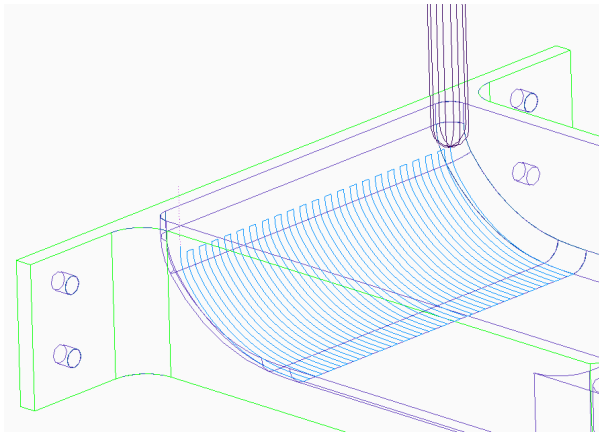


The selection order is unimportant...

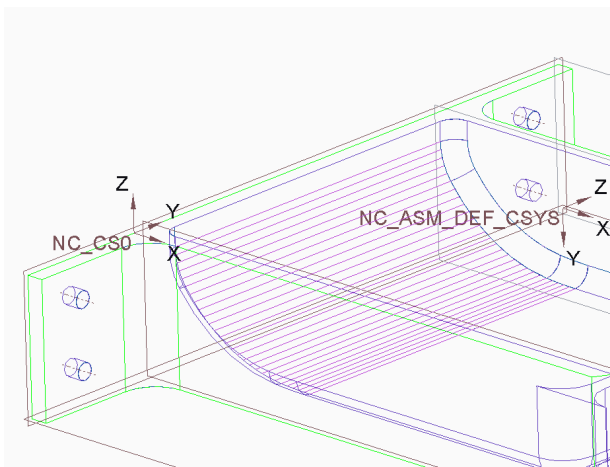
- Choose **OK** in the **Select** box

7. Play the path..

- Choose **Screen Play**
- Choose the **Play Forward** action button



- After closing the player, Choose **Seq Setup** once again
- Choose **Define Cut / Done** - *The system now displays the **Cut Definition** dialogue. Notice the default setting: **From Surface Isolines***
- Press the **Preview** button. - *The **From Surface Isolines** setting overrules the **Parameter** angle (0°)*

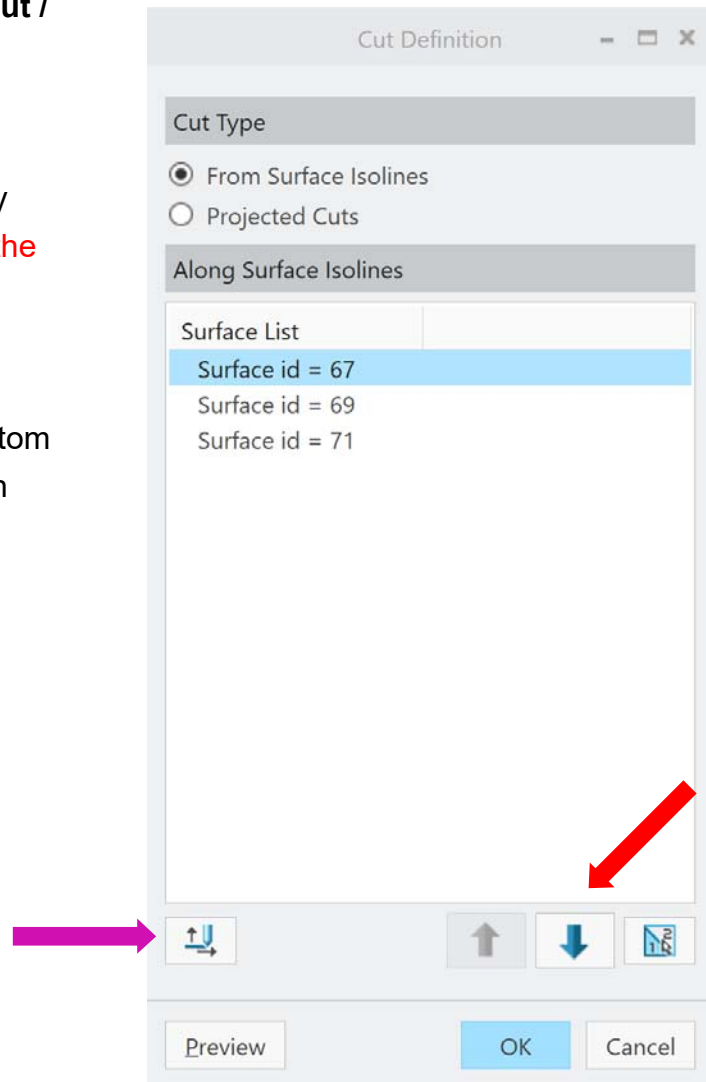


- Close the dialogue with an **OK**.

8. Play the path..

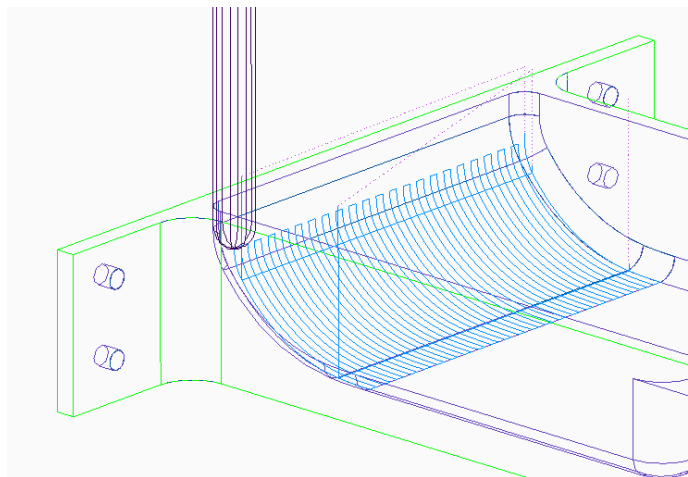
- Choose **Screen Play**
- Choose the **Play Forward** action button - *The milling starts on the upper surface*

- Enter the **Sec Setup / Define Cut / Done** once again
- In the **Cut Definition** dialogue, reverse the milling order of the 3 surfaces by choosing them one by one and change their order using **the arrow**
- Now reorient the **cut direction** by choosing them one by one and toggle direction with the tool in bottom left. Arrows, indicating the direction can be seen on the model



9. Play the path..

- Choose **Screen Play**

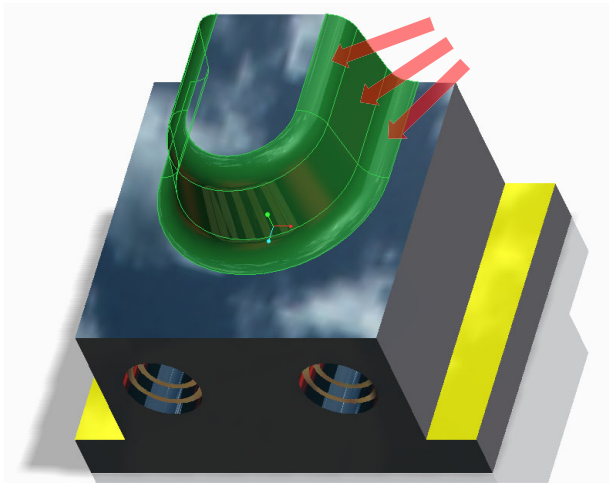


- Save your work: **Choose File > Save as > Save a backup.** Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.



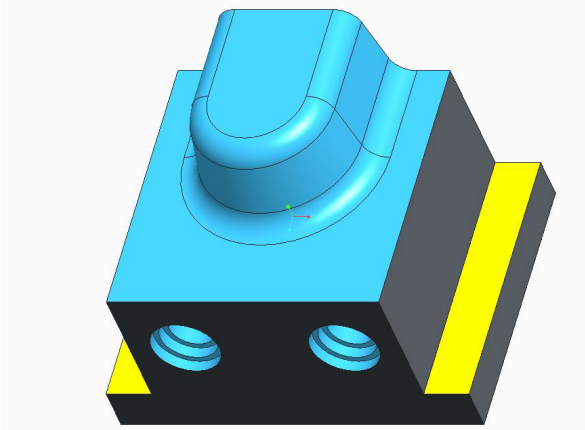
End of demonstration

Finish machine the contoured faces on the mold insert shown below.



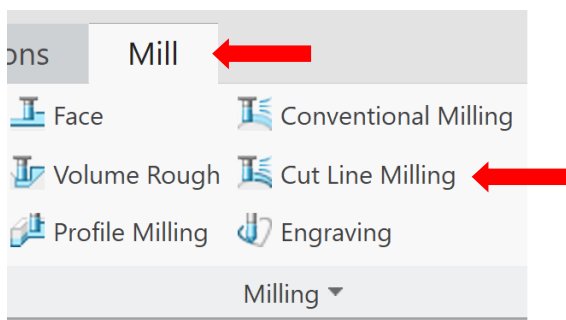
Finish machine these surfaces

1. Retrieve the manufacturing model called: **surf1.asm**

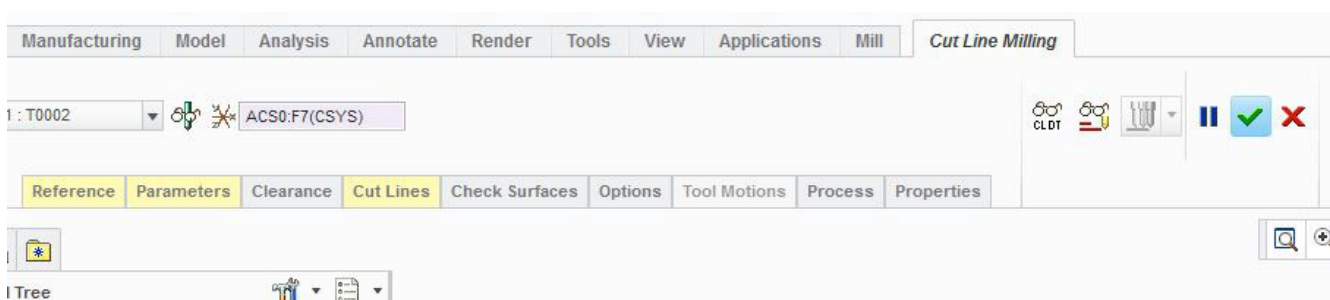


2. Start a new NC Sequence.

- In the top ribbon choose the **Mill** pane / **Cut Line Milling** - As shown:

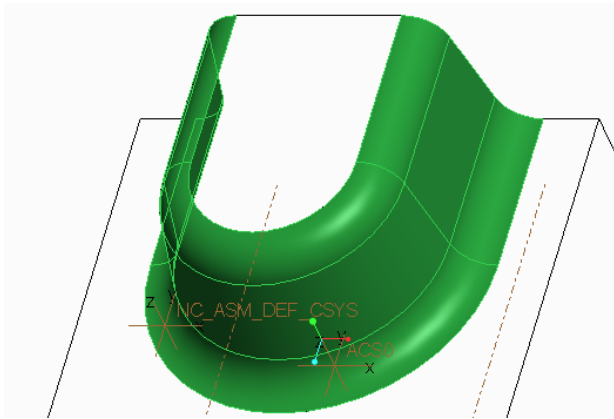


The Cut Line Milling interface has many entries, with the mandatory ones in yellow as usual:



3. Fill - in the **References**

- Open the **Reference** tab, verify - Under **Type** - that **Surface** is selected and choose the **Details** button in the lower right corner. - *The **Surface Sets** dialogue opens*
- Click in the **Select Items** area, and choose the faces on the model as shown:

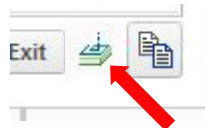


*The **Surface Sets** dialogue will list the selected faces - the sequence is unimportant*

- Close the **Surface Sets** dialogue with an **OK**.

4. Retrieve the provided **Parameters**.

- Choose the **Parameters** tab, followed by the **Parameters** tab bottom right of the list.



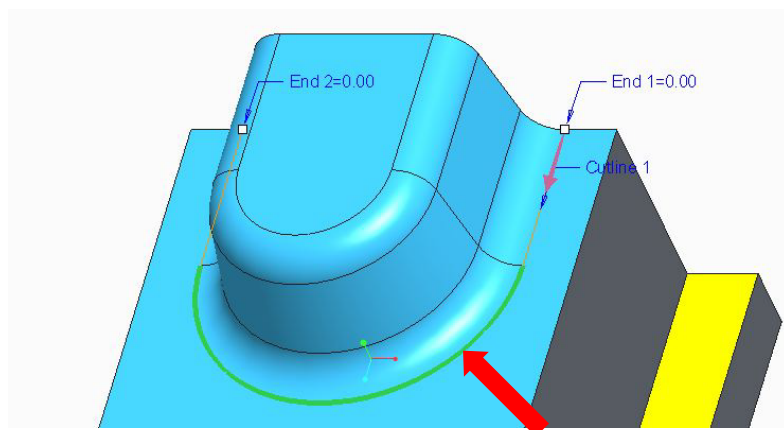
Edit Machining Param -

*- The familiar **Edit Parameters** dialogue opens.*

- In the **Edit Parameters** dialogue choose **File/Open** and doubleclick **Surf6.mil**.
- Close the **Information Window**.
- Close the **Edit Parameters** dialogue with an **OK**.

5. Define **Cut Line 1**

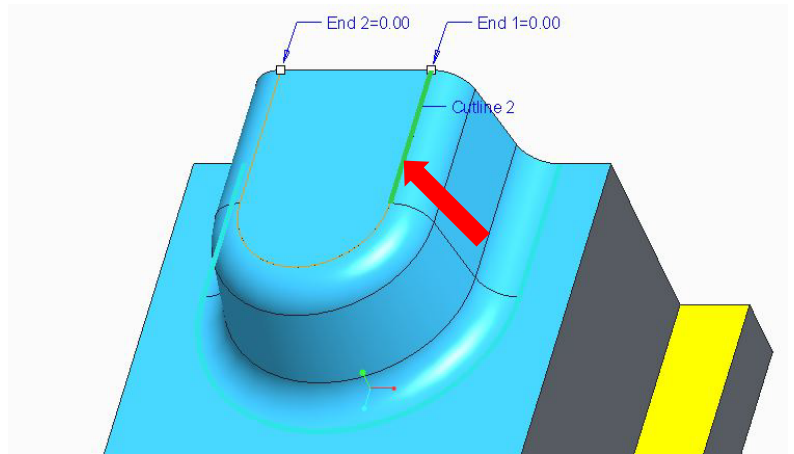
- Open the **Cut Lines** tab, and verify that **Cutline 1** is active. Click the *upper* one of the two **Details** buttons - *The **Chain** dialogue appears*
- In the **Chain** dialogue, under the **References** tab, choose the **Rule-based** radio button
- Verify that the **Tangent** radio button is selected.



- select the edges (one is enough) as shown highlighted in **green** as the **Cutline 1**. Notice the arrow indicating the direction.
- Close the **Chain** dialogue with an **OK**.

6. Define **Cut Line 2**

- Repeat the procedure, starting with entering the **Cut Lines** tab and activating **Cutline 2**.
- This time the upper line (yes, **green** again) has to be selected as **Cutline 2**.



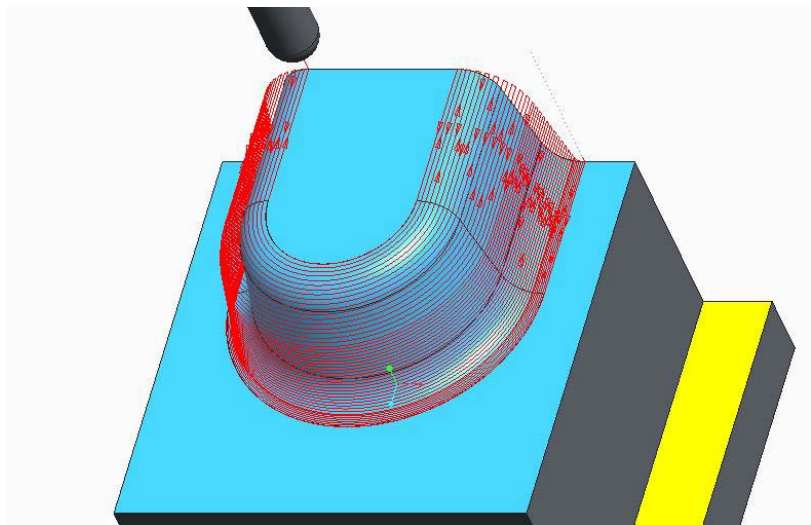
- Close the **Chain** dialogue with an **OK**



- The **Cut Line Milling** dialogue can now be closed with the **green OK** sign.

7. Play the **Path**

- In The **Model Tree**, right-click the **Cut Line Milling** sequence and choose **Play Path**:



- Save your work: **Choose File > Save as > Save a backup**. Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.



ATTENTION

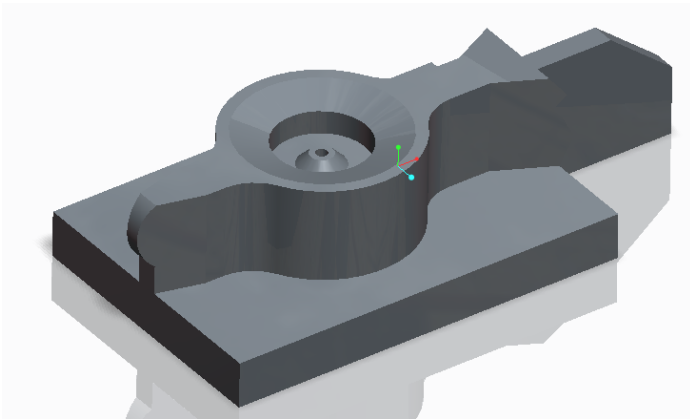
The final step of the above exercise must be **REVIEWED** and **APPROVED** by your **INSTRUCTOR** to make you eligible for a signature on your approval sheet confirming your successful completion of this tutorial.

Leave this file open and continue to the next exercises. Please complete both **ATTENTION** - marked **Day-3** exercises before requesting review and approval

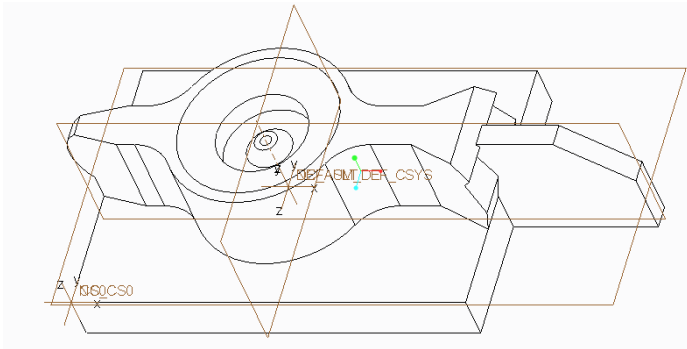
PART 7: CUSTOMIZED MILLING

APPROACH & EXIT CUTS

Customize the two NC Sequences provided to profile the outline of the valve body armature shown below.



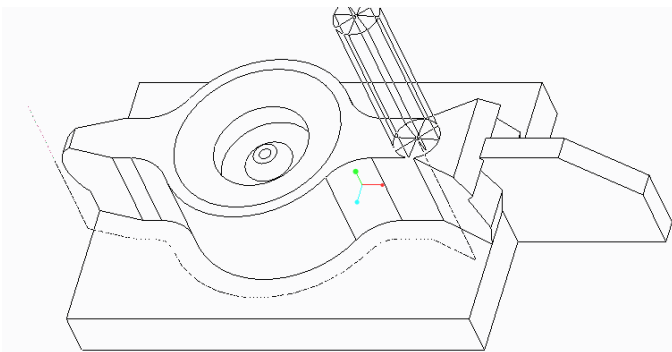
1. Retrieve the manufacturing model: **cust2_manf.asm**.



2. Activate the existing NC Sequence.

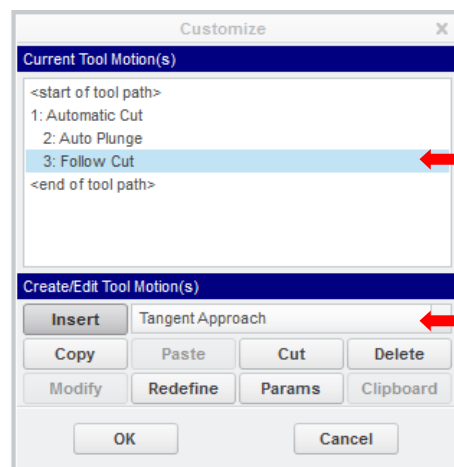
- In the **Model Tree** click **1. Pocket Milling [OP010]** and choose **Edit Definition**

3. Play the path



4. Customize this cutter path to add a **Tan - gent Approach** and a **Tangent Exit** move.

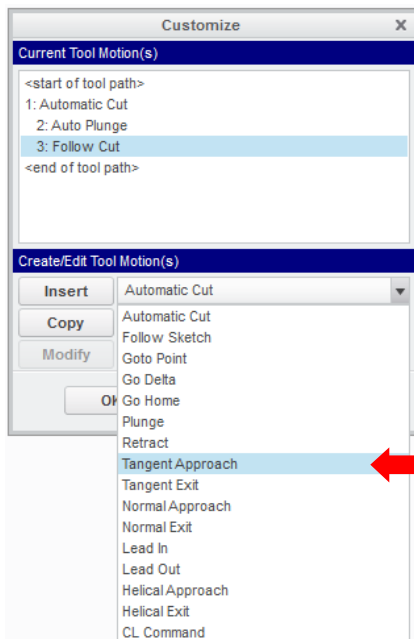
- Choose **Customize**
- In the **Customize** dialog, highlight item **3: Follow Cut** in the **Current Tool Motion(s)** list as shown right:



Choose this one

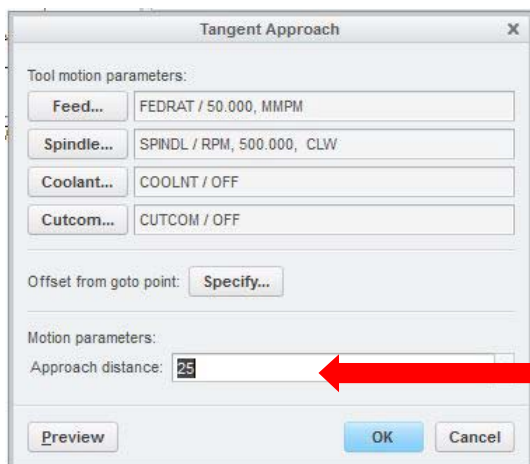
Choose Tangent Approach

- Choose **Tangent Approach** from the option menu as shown below



As soon as the **Tangent Approach** option is selected, the system displays the **Tangent Approach** dialogue as shown below. For the **Tangent Approach**, you can specify **Feed Rate**, **Spindle Speed**, **Coolant On/Off**, a **Cutcom** command, an **X, Y, Z Offset Distance** and/or an **Approach Distance**.

- Select in the **Approach Distance** text field and type **25**

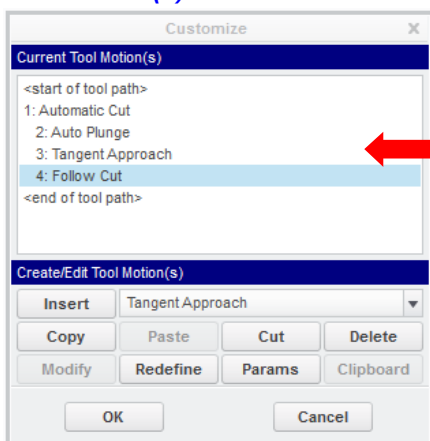
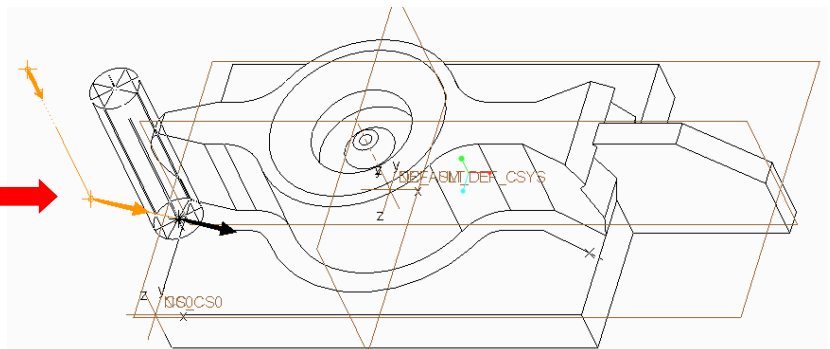


Value 25

- Choose the **Preview** action button

25 mm Tangent Approach added

The system immediately displays the **Tangent Approach** on the graphic screen and inserts it into the **Current Tool Motion(s)** list:



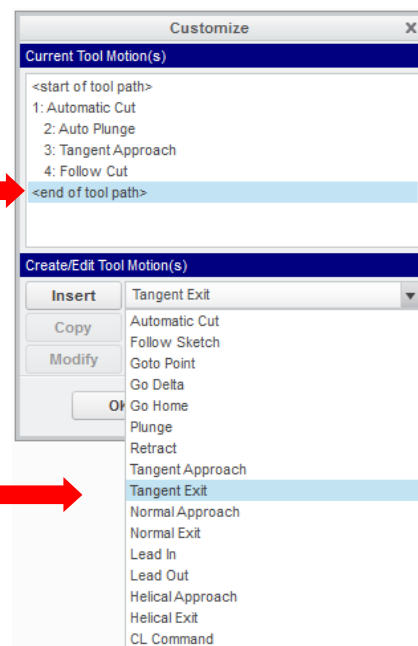
Tangent Approach now shows on the list

5. Add a **Tangent Exit** move of **25.0**.

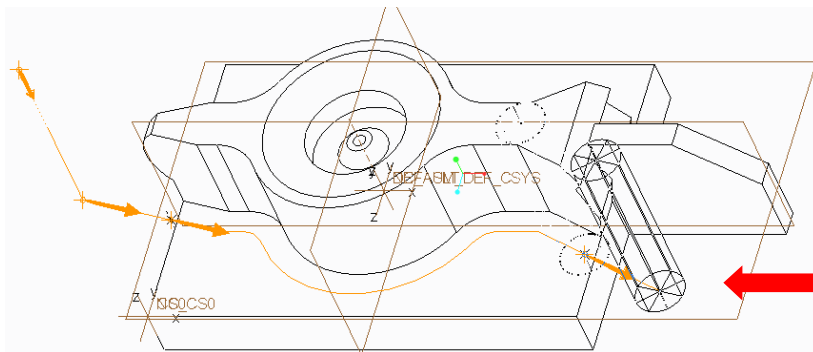
- Highlight **<end of tool path>** in the **Current Tool Motions** list
- Choose **Tangent Exit** from the option menu

You should see the **Customize** dialog as shown below.

- Choose the **<end of tool part>** line to make it the active one.
- In the drop down list, Choose **Tan-**
gent Exit
- In the **Tangent Exit** box, for **Exit**
Distance type 25

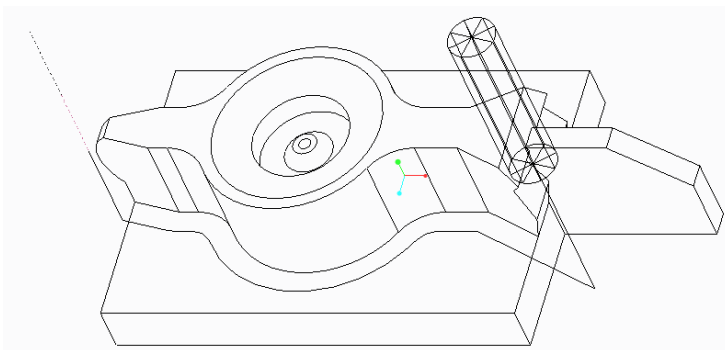


- Choose the **Preview** action button



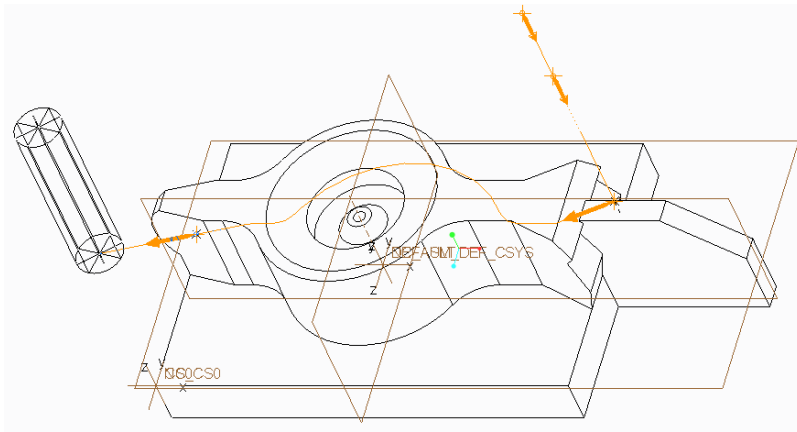
25 mm Tangent Exit added

- Choose the **OK** action button from the **Customize** dialog



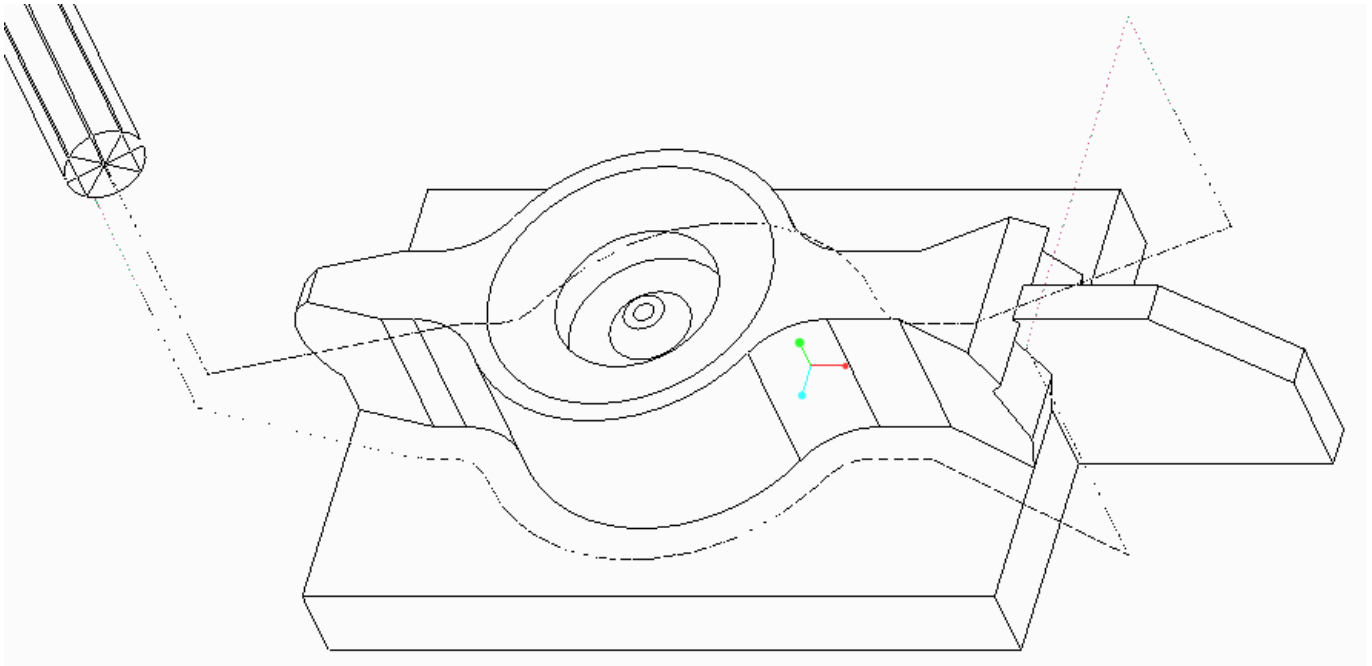
- Choose **Done Seq**

6. Repeat steps 2 through 5 using the second existing NC Sequence: **2. Pocket Milling [OP010]**



7. Use the **CL Data** option to replay the complete operation **OP010**.

- In the Model Tree, highlight (singleclick) **OP010 [MACH01]**
- In the model tree, right-click on **OP 010[MACH01]** and choose **Play Path** - *(Similarly you can run **Play Path** for one of the individual NC Sequences, if you pre-highlight it)*
- In the **Menu Manager**, choose **Done**
- Run the **Play Path** - *The complete **Operation** will display*

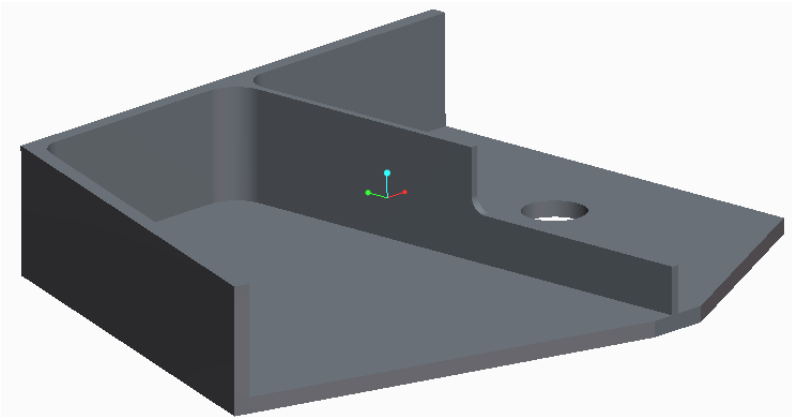


The full tour...

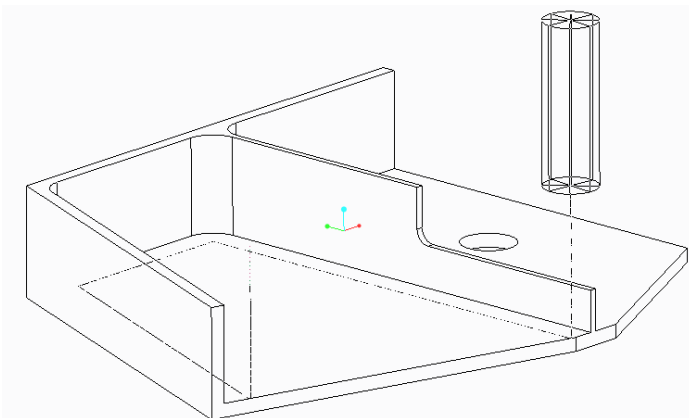
- Save your work: **Choose File > Save as > Save a backup**. Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.

APPROACH & EXIT CUTS

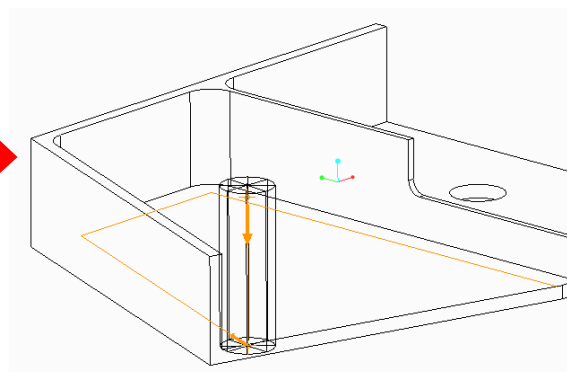
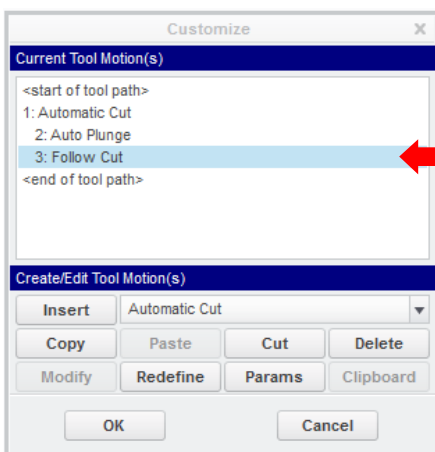
Customize the tool path that profiles the left side of this aircraft bracket to add an acceptable approach and exit motion.



1. Open the manufacturing file: **cust1_manf.asm**.
2. Activate the existing **NC Sequence**.
 - In the **Model Tree** click on **1. Pocket Milling [OP010]** and choose **Edit Definition**
3. Play the path

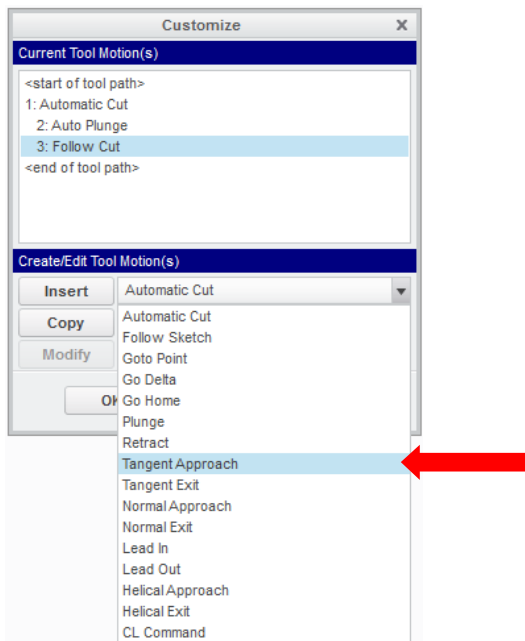


3. Customize this cutter path to add a **Tangent Approach** and a **Tangent Exit** move.
 - Choose **Customize**
 - In the **Customize** dialog, highlight item **3: Follow Cut** in the **Current Tool Motion(s)** list as shown below

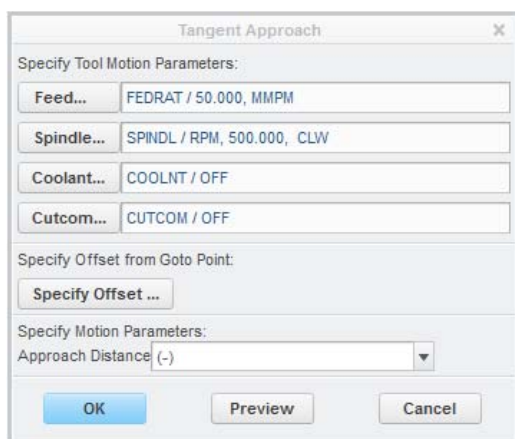


The Follow Cut motion displays on the model

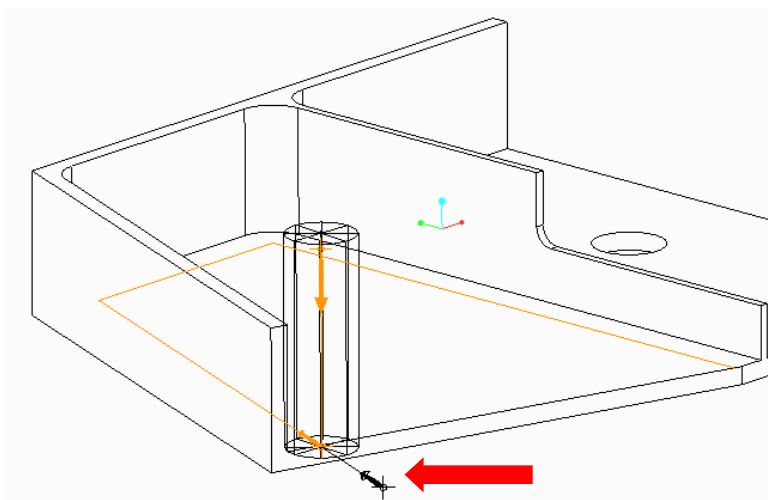
- Choose **Tangent Approach** from the option menu as shown below



As soon as the **Tangent Approach** option is selected, the system displays the **Tangent Approach** dialogue as shown below. For the **Tangent Approach**, you can specify **Feed Rate**, **Spindle Speed**, **Coolant On/Off**, a **Cutcom** command, an **X, Y, Z Offset Distance** and/or an **Approach Distance**.



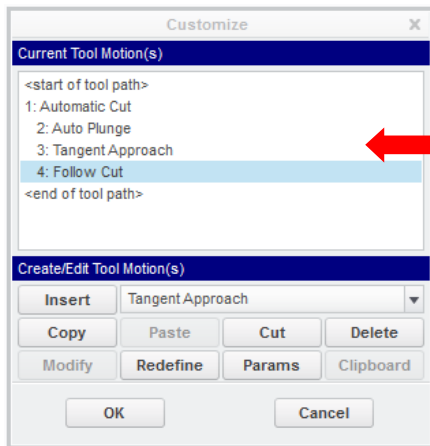
- Select in the **Approach Distance** text field and type **50**
- Choose the **Preview** action button



Preview of the 50 mm Approach Distance

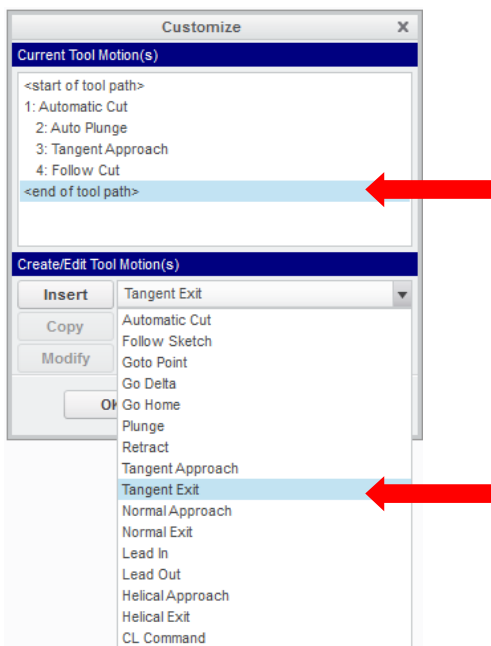
- Choose the **OK** action button

The system immediately displays the **Tangent Approach** on the graphic screen and inserts it into the **Current Tool Motion(s)** list.



Tangent Approach now shows on the list

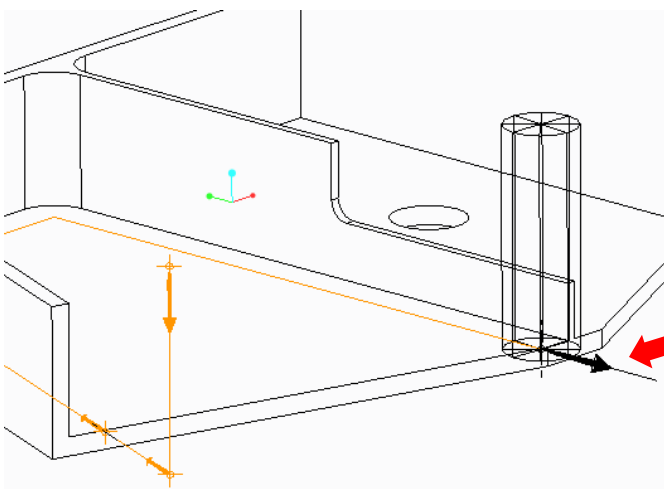
- Highlight <end of tool path> in the **Current Tool Motions** list
- Choose **Tangent Exit** from the option menu



The system now displays the **Tangent Exit** dialog with the same options as in the **Tangent Approach** dialog.

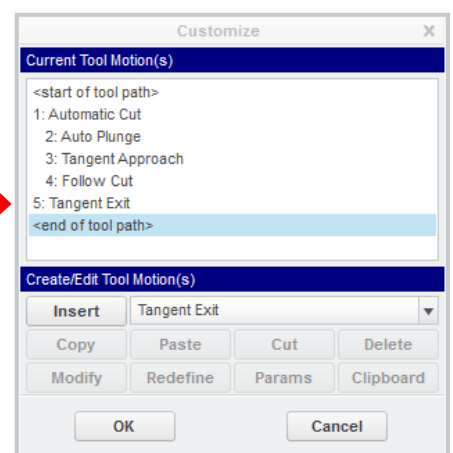
- Select in the **Exit Distance** text field and type **50** and choose the **OK** action button

Again, the system immediately displays the **Tangent Exit** motion on the graphic screen and has inserted the **Tangent Exit** statement in the **Current Tool Motion(s)** list.



Tangent Exit inserted

The 50 mm Exit Distance

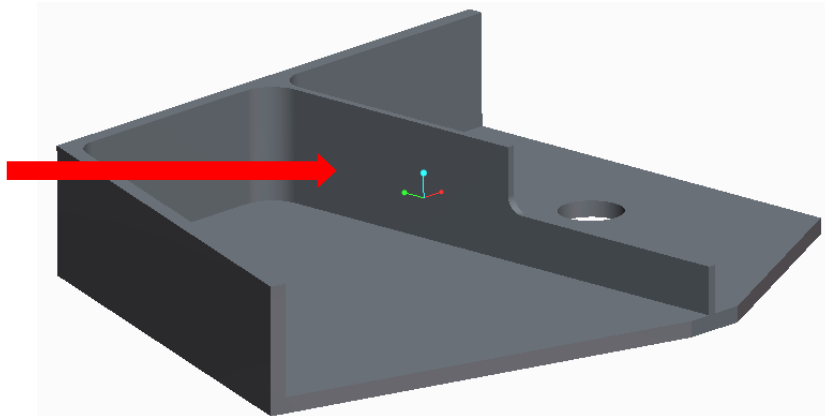


Previously you have **Customized** a tool path by building an additional path with the new motions in it, thus ending up having to delete one of them. This time you simply customized the original cutter path by inserting the new requirements.

POSTPROCESSOR COMMANDS

Add a block of text at the beginning of the file you have been working on. Also, you need to slow down the feed rate of the tool along the thin walled rib which lies along the center of the bracket. Add a feed rate command to affect the slow down.

*Slow the Feed Rate to 15.0
MMPM along this thin Rib*

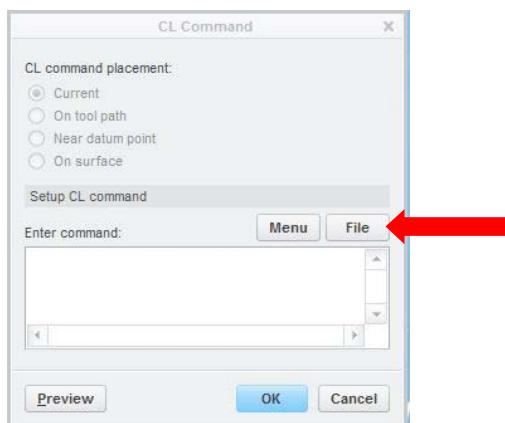


1. Add the block of text at the beginning of the sequence.

- In the **Customize** box Select **<start of tool path>** in the **Current Tool Motion(s)** list.

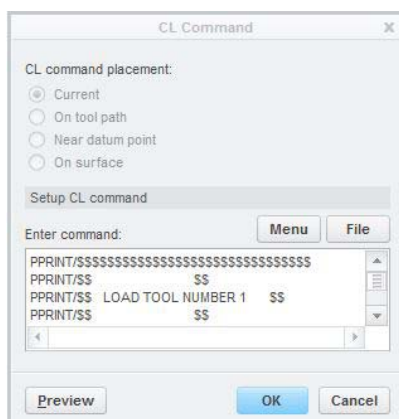
*You want the block of text to go as close to the front of the file as possible. You will also notice that the only parameter you can enter is a **CL Command**. A **CL Command** is the only option that can be inserted at the beginning of a cutter file.*

- Choose **CL Command** from the option menu to the right of the **Insert** button



- Choose **File** in the **CL Command** dialog
- Choose **loadtl1.cmd** from the list
- Choose the **Open** action button

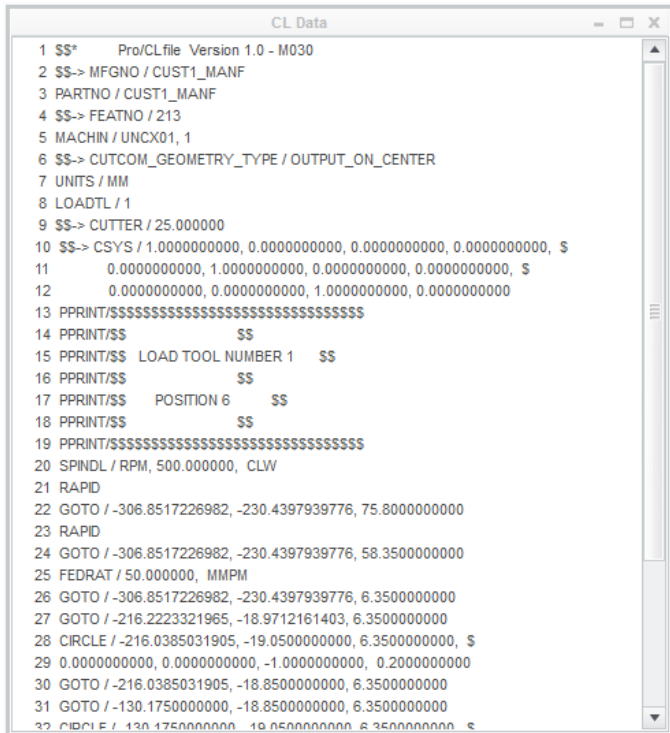
*The file is read in and is displayed in the **Command** text field of the **CL Command** dialog.*



- Choose the **OK** action button

One line of text from the file is shown in the **Current Tool Motion(s)** listing, but to see the complete text just look at the **CL Data window**, which now appears as a result of the **Open** command. The **CL Data window** is only for viewing and it responds when you highlight one of the **Tool Motions** in the **Customize** box.

Please notice that the **CL Data window** may **hide** behind the whole Creo screen. (use ALT - TAB to display it)



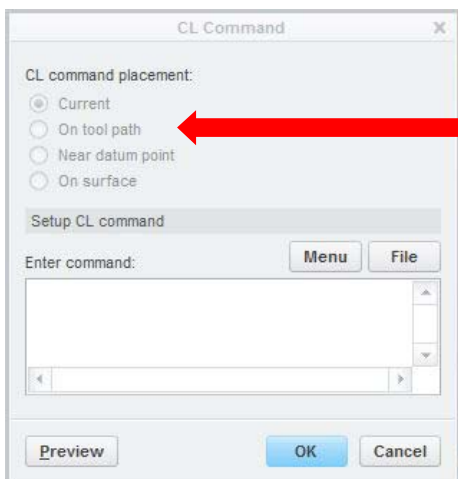
- Scroll down in the window until you can see the entire block of text you added

2. Now add the feed rate command to slow down the tool as it machines the thin rib.

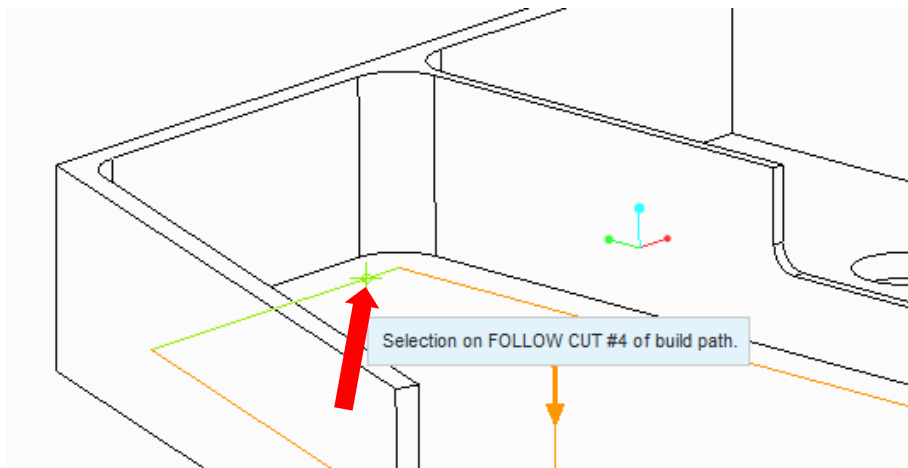
You now need to add a **Cut Motion** to hold the new post command and its associated **GOTO** point....

- in the **Customize** dialog highlight item **5: Tangent Exit** on the **Current Tool Motion(s)** list
-You probably noticed that in the **CL Data window** line 36 highlighted too.
- Click in the text field to the right of the **Insert** action button in the **Customize** dialog
- Choose **CL Command** from the list

The system displays the **CL Command** dialog as shown:

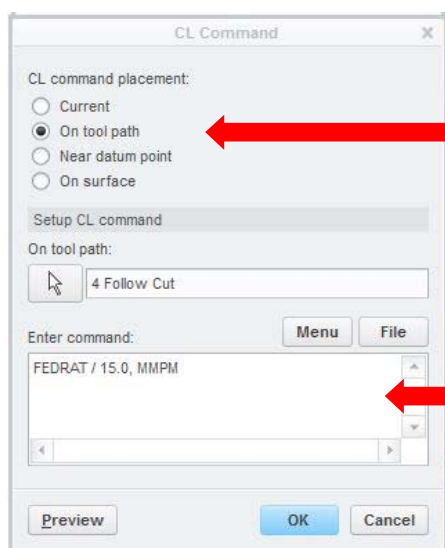


- Choose the **On Tool Path** radio button (Arrow)
- Select the position approximately as shown next page



Click on the Tool Path a little bit before the Tool meets the corner...

- Place the Cursor in the **Enter Command** text field and click the **LMB**

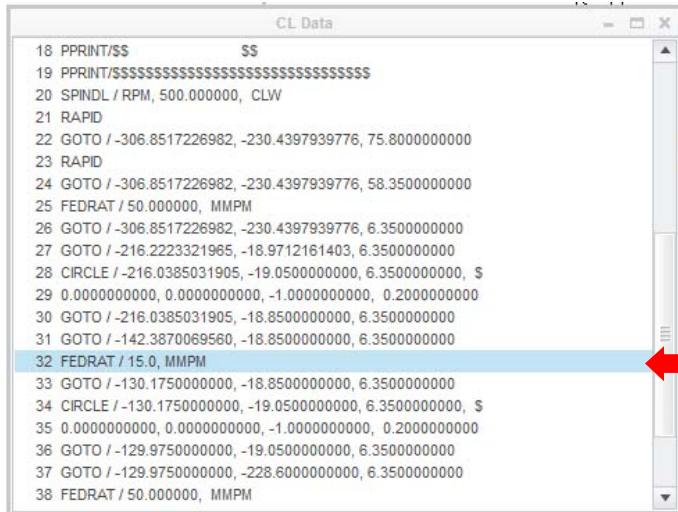


Remember this...

Click for typing...

- Type **FEDRAT / 15.0, MMPM** exactly as shown above
- Choose the **OK** action button

*You should now be able to see the new command and **GOTO** Point in the CL-File. In the **Customize** box. you may highlight line 5 (**FEDRAT...**) to see the change in the CL-File...*



- Choose the **OK** action button in the **Customize** dialog

3. You can see the relative affect of the new feedrate on the tool path by playing it with a slower speed from the **Play Path** CD player.

- Choose **Play Path**

Notice how the tool changes speeds depending on the commands that you just added...

- Save your work: Choose **File > Save as > Save a backup**. Use **Organize**, and create a new folder. This action puts all the files necessary for running a machining simulation in this folder and enables you at a later time to view or continue your work .
- The folder must be handed in to the 41617 home page > **Assignments > Cam Week 3**, following instructions here.



ATTENTION

The final step of the above exercise (the tool speed slowdown) must be **REVIEWED** and **APPROVED** by your **INSTRUCTOR** to make you eligible for a signature on your approval sheet confirming your successful completion of this tutorial.