Creo NC – A Practical Approach
Creo 3.0

www.caeuniversity.com

Creo NC – A Practical Approach
Creo 3.0

Copyright © 2018. All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the www.caeuniversity.com

CAE UNIVERSITY makes every effort to ensure the accuracy of the training material and assumes no responsibility or liability for errors or inaccuracies that may appear in this publication.

Contents

1. INTRODUCTION....1
    EXERCISE 1....5
2. FACE MILLING....19
    EXERCISE 1....21
    EXERCISE 2....34
3. VOLUME AND LOCAL MILLING....39
    EXERCISE 1....41
    EXERCISE 2....52
    EXERCISE 3....56
    EXERCISE 4....67
    EXERCISE 5....72
    EXERCISE 6....78
    EXERCISE 7....81
    EXERCISE 8....86
    EXERCISE 9....89
    EXERCISE 10....94
    EXERCISE 11....98
    EXERCISE 12....104
    EXERCISE 13....108
    EXERCISE 14....115
    EXERCISE 15....121
4. ROUGHING AND REROUGHING....125
    EXERCISE 1....126
    EXERCISE 2....132
    EXERCISE 3....136
    EXERCISE 4....145
    EXERCISE 5....151
    EXERCISE 6....155
5. PLUNGE MILLING....159
    EXERCISE 1....160
6. PROFILE MILLING....167
    EXERCISE 1....169
    EXERCISE 2....176
    EXERCISE 3....185
    EXERCISE 4....187
    EXERCISE 5....196
    EXERCISE 6....203
    EXERCISE 7....215
7. SURFACE AND CUT LINE MILLING....225
    EXERCISE 1....226
    EXERCISE 2....231
    EXERCISE 3....238
    EXERCISE 4....247
    EXERCISE 5....254
    EXERCISE 6....260
    EXERCISE 7....268
    EXERCISE 8....280

EXERCISE 9....288
EXERCISE 10....294
8. FINISHING....299
EXERCISE 1....301
EXERCISE 2....304
EXERCISE 3....307
EXERCISE 4....311
EXERCISE 5....317
EXERCISE 6....320
EXERCISE 7....323
9. REST FINISHING....327
EXERCISE 1....328
10. TRAJECTORY MILLING....333
EXERCISE 1....334
EXERCISE 2....340
EXERCISE 3....346
EXERCISE 4....354
EXERCISE 5....359
EXERCISE 6....363
EXERCISE 7....366
11. ENGRAVING....371
EXERCISE 1....372
12. HOLEMAKING....377
EXERCISE 1....378
EXERCISE 2....384
EXERCISE 3....388
EXERCISE 4....392
EXERCISE 5....397
EXERCISE 6....401
13. THREAD MILLING....403
EXERCISE 1....404
EXERCISE 2....411
14. TOOL MANAGER....419
EXERCISE 1....421
EXERCISE 2....427
EXERCISE 3....430
EXERCISE 4....436
EXERCISE 5....444
EXERCISE 6....446
EXERCISE 7....448
EXERCISE 8....454
EXERCISE 9....459
15. PROCESS MANAGER....463
Manufacturing Process Information....464
EXERCISE 1....465
EXERCISE 2....471
Creation of Steps....479
EXERCISE 1....480
XML Template....482
EXERCISE 1....483
16. MANUFACTURING PROCESS INFORMATION....491
EXERCISE 1....494
EXERCISE 2....507

EXERCISE 3.....513
EXERCISE 4.....520
17. MANUFACTURING TEMPLATE.....523
EXERCISE 1.....524
EXERCISE 2.....545
18. POSTPROCESSING.....555
EXERCISE 1.....557
EXERCISE 2.....566
EXERCISE 3.....580
EXERCISE 4.....585
EXERCISE 5.....589

1. INTRODUCTION

Not included in free download.

2. FACE MILLING

Not included in free download.

3. VOLUME AND LOCAL MILLING

Not included in free download.

4. ROUGHING AND REROUGHING

Not included in free download.

5. PLUNGE MILLING

Not included in free download.

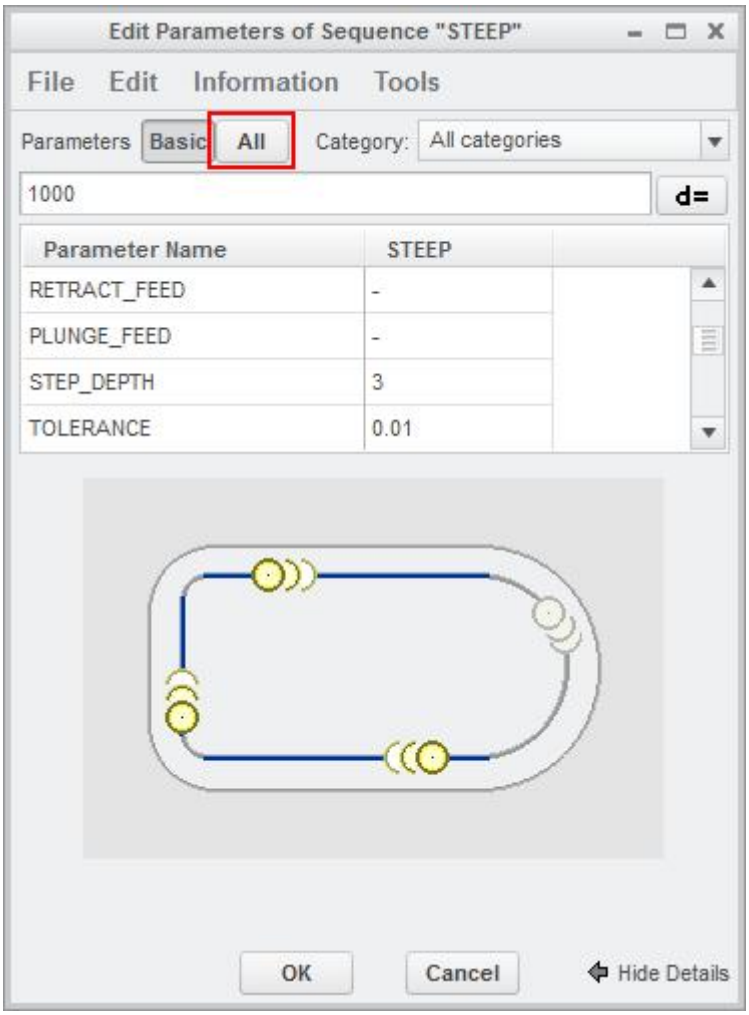
6. PROFILE MILLING

Profile milling NC Sequence is a 2.5 axis sequence. It is usually used to semi-finish or finish the vertical or slanted surfaces. The surface finish can be controlled with Step Depth adjustment or Scallop Height parameter.

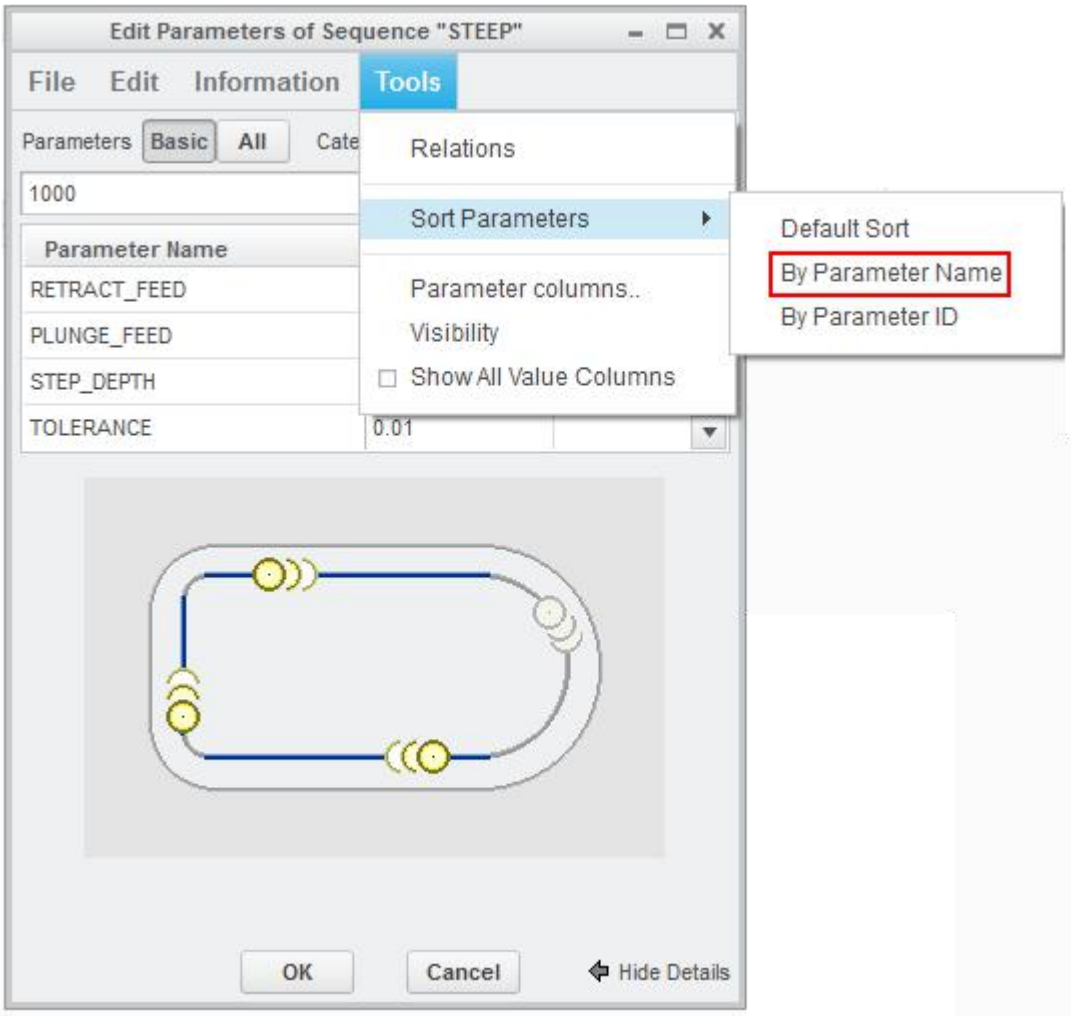
The required references are surfaces which may be the selected surfaces of either of the Reference model, Mill Surface or Mill Volume. It automatically degouges the tool path against the references selected for the cut geometry.

Locating Cutting Parameters

There are a lot of cutting parameters associated with the sequence. You can see all the parameters by picking All tab on the Edit Parameters dialog box.



If you don't know the exact location of a parameter then it is better to arrange the parameters alphabetically by picking **Tools > Sort Parameters > By Parameter Name** and then you can easily locate the desired parameters.


Edit Parameters of Sequence "STEEP"
File
Edit
Information
Tools
Parameters
Basic
All
Cate
1000
Parameter Name
RETRACT_FEED
PLUNGE_FEED
STEP_DEPTH
TOLERANCE
0.01
Relations
Sort Parameters
Parameter columns..
Visibility
Show All Value Columns
Default Sort
By Parameter Name
By Parameter ID
OK
Cancel
Hide Details

EXERCISE 1

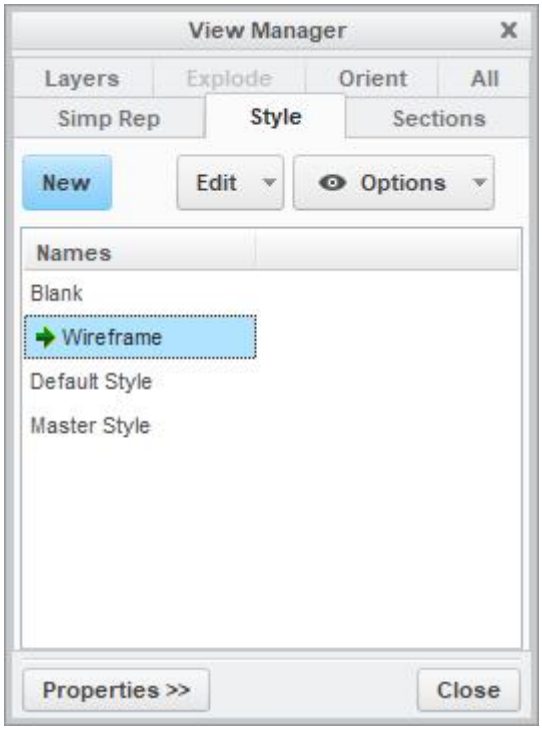
In this exercise we will define a profile milling NC Sequence to machine vertical walls of a block.

The topics covered in this exercise are as follows

- Defining a new Profile Milling NC Sequence
- Selecting the surfaces using Loop option
- Creating smooth entry/exit motions
- Optimizing the entry/exit conditions

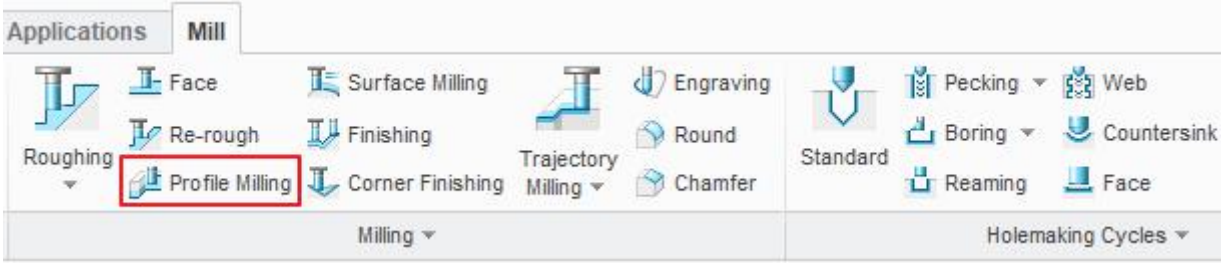
Set the working directory to INSERT and open the model insert.asm.

For better visualization it is good to display the workpiece in wireframe. So select **View Manager** icon to launch the View Manager Dialog box and double click the predefined style state **Wireframe**.



Select Close to exit the dialog box.

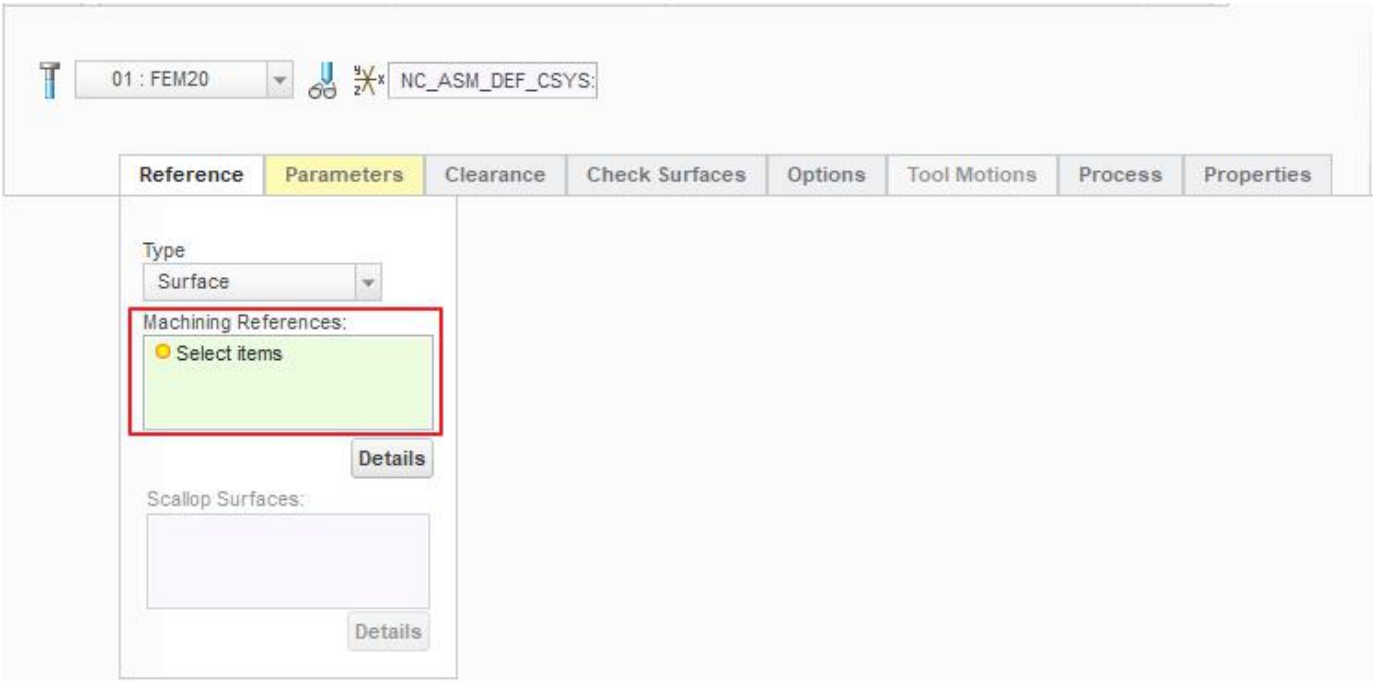
To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.



Pick the icon and Tools Setup dialog box will pop up.

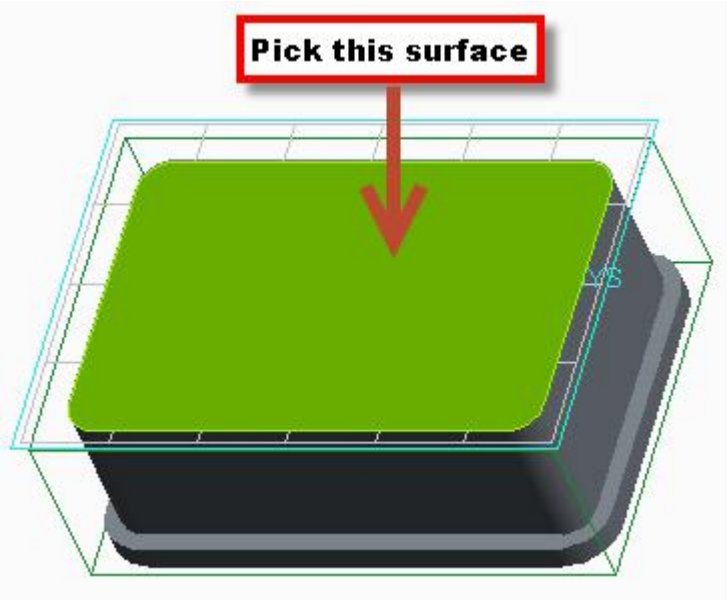
In the Tools Setup dialog box pick **FEM20** from the list and select OK.

Pick Reference tab and notice that the Machining Reference collector becomes active.

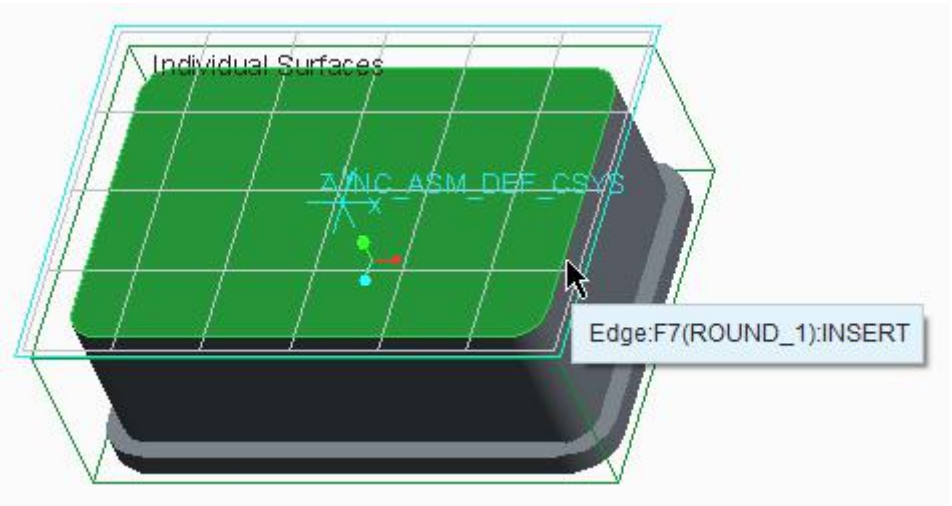


Here we will use the Loop selection technique to select the desired surfaces quickly.

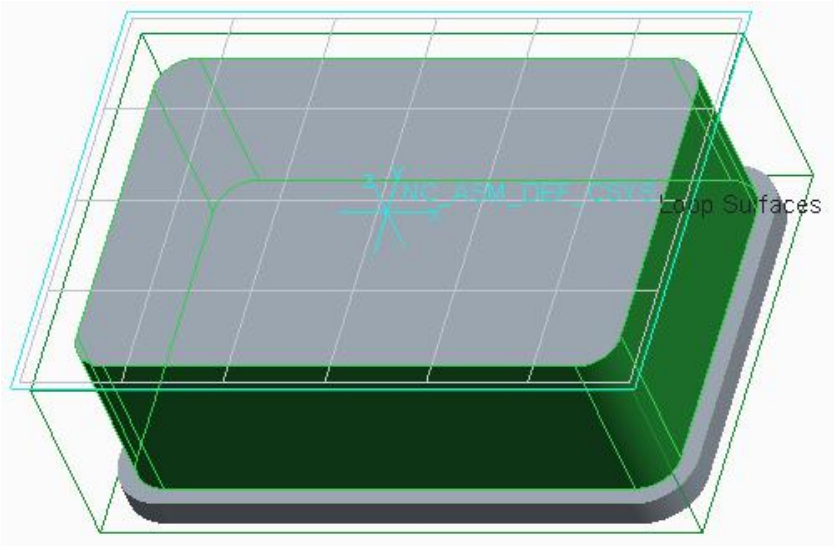
Now select the top face of the reference model as shown below.



Now hold down Shift key and place mouse pointer over any edge forming the boundary of the selected surface.



Pick on the edge with left mouse to select the Loop Surfaces while Shift key is held down. The selected surfaces will appear as shown below.



Loop option is used to select a closed loop of surfaces by choosing a face they surround.

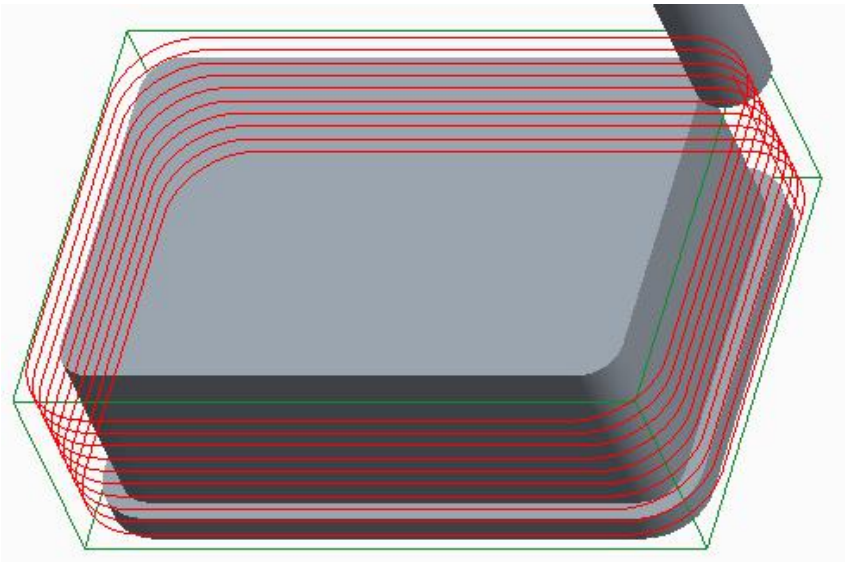
Now pick Parameters tab and enter the following values/relations and parameters.

CUT_FEED	1200
STEP_DEPTH	5
CLEAR_DIST	1
SPINDLE_SPEED	3000

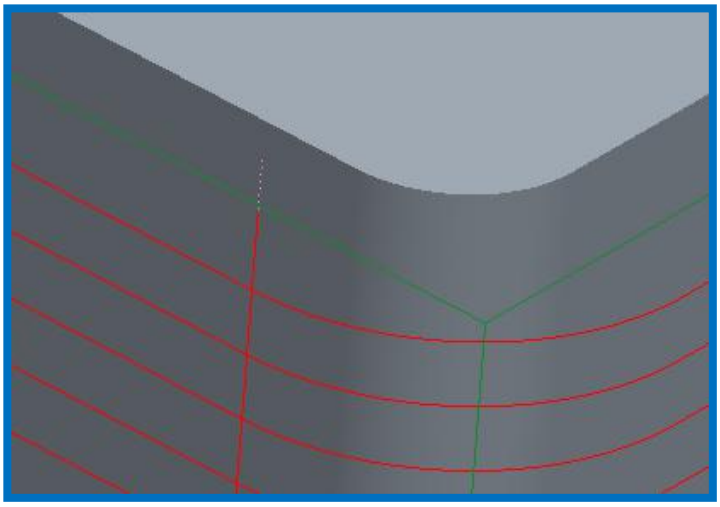
Now pick Properties tab and enter ***PROFILE*** as the name of the sequence.

To display the toolpath on screen pick icon. Play Path Dialog box will appear.

Click the Play Forward button and toolpath will appear as shown below.



Zoom to the toolpath and look at it carefully, you will notice that the tool plunges directly to the next slice.

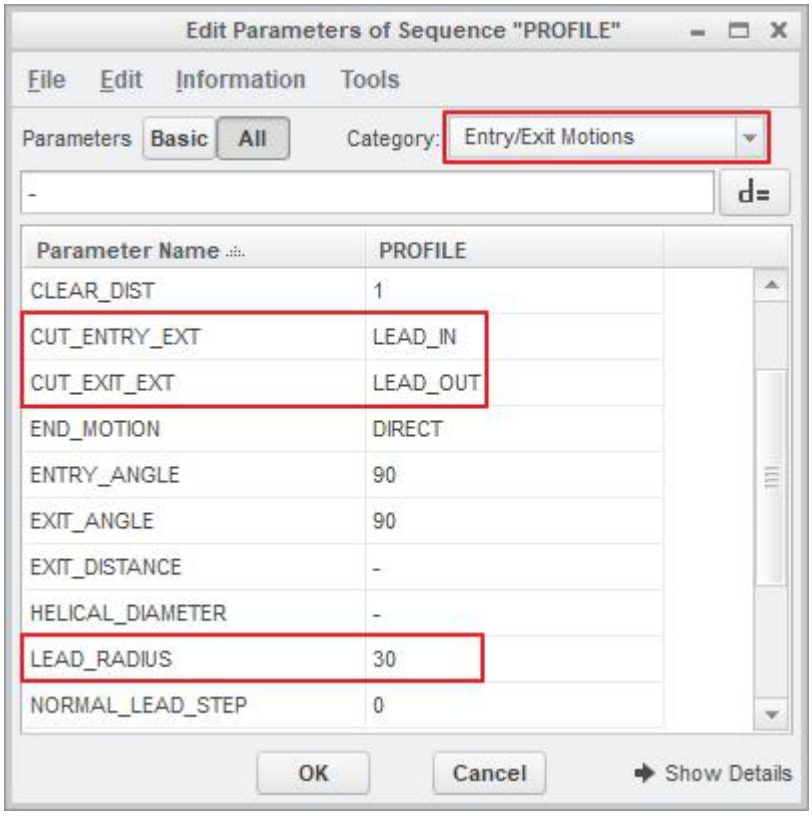


Direct plunge is not a good practice. To introduce smooth entry/exit into the cut, parameters need to be changed.

So pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

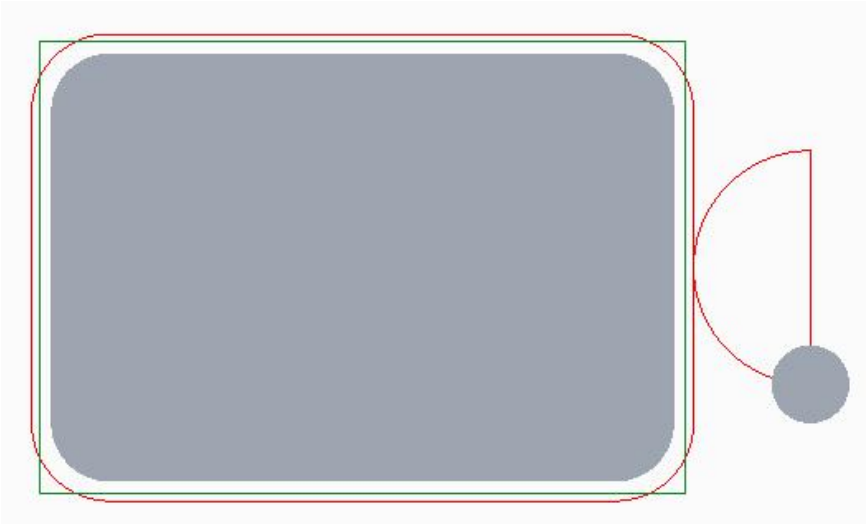
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	30

You can see all these parameters by picking All tab and changing to “Entry/Exit Motions” category as shown below.



Pick OK to apply the changes and exit the Edit Parameters Dialog box.

Now play the tool path using **PLAY PATH** dialog box. From TOP view it will appear as shown below.

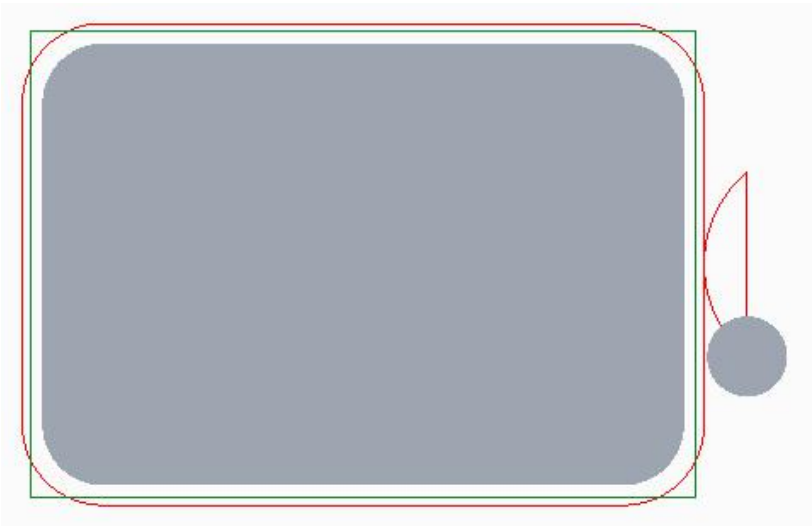


Although now tool enters and exits into the cut very smoothly but it is wasting a lot of time in non-cutting motion. To reduce this non-cutting time, again pick in Parameters panel and set the following parameters. You will find these by picking All tab and changing to “Entry/Exit Motions” category.

ENTRY_ANGLE	50
EXIT_ANGLE	50

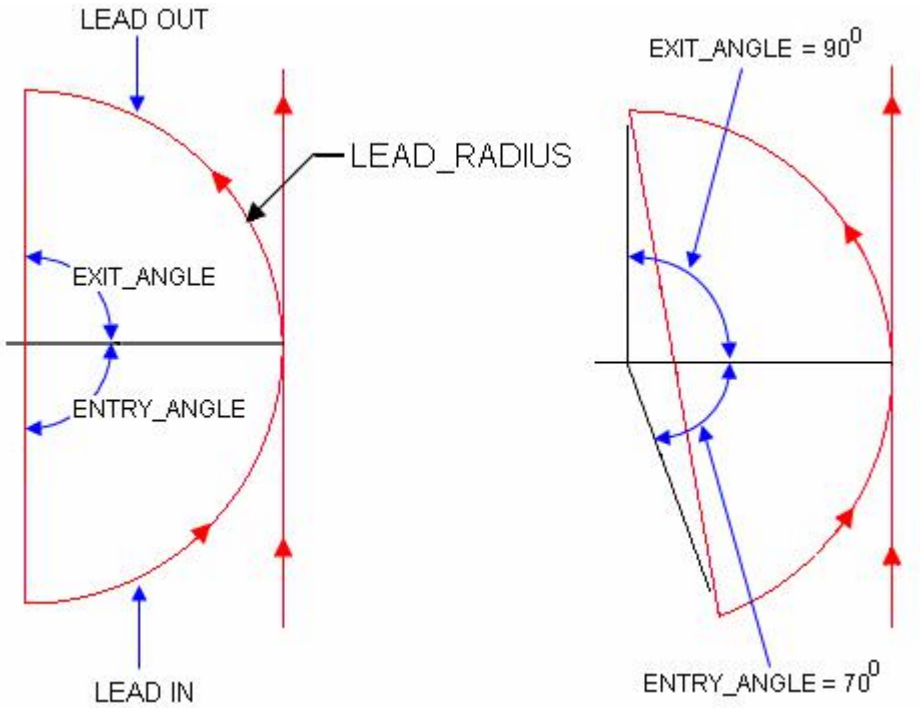
Pick OK to apply the changes and exit the Edit Parameters Dialog box.

Now play the tool path using **PLAY PATH** dialog box. From TOP view it will appear as shown below.



Notice that the non-cutting motion has been reduced significantly. The Entry and Exit angles must be sufficient large so that tool makes the plunge motion when it is out of the workpiece.

The entry/exit parameters can be readily understood from the figure below.



It is always advantageous to use a large LEAD_RADIUS because larger the radius, smoother the entry/exit into cut. But it also increases the idle time while the tool completes its non-cutting movement. To reduce this non-cutting time we have set the ENTRY_ANGLE and EXIT_ANGLE to a smaller value than default. By doing this we have achieved both i.e. smoother motion by setting a large LEAD_RADIUS value and reduction in non-cutting time by smaller entry/exit angles.

As the toolpath looks satisfactory so change the following parameters for final toolpath computation.

STEP_DEPTH	2

Pick to save the changes and close the dashboard.

Select **File > Save** to save the manufacturing model.

EXERCISE 2

In this exercise first we will define a Mill Surface to find the area which previous Ball End Mill cutter could not machine. Then we will machine this surface with a Profile Milling NC Sequence. The topics covered in this exercise are as follows

- Defining a Mill Surface as “Machinable Area”
- Creating smooth entry/exit motions

The **Machinable Area** Mill Surface is an advanced technique that calculates whether the selected surfaces can be entirely machined with the selected cutting tool, and creates explicit surface representations of the millable area (the portion of the selected surfaces that can be milled), as well as of the remainder area (the portion that can not be milled with the selected tool).

This technique allows to automate surface selection for subsequent machining with a smaller tool or with a different machining strategy.

You have to specify surfaces to be milled, additional surfaces for gouge checking (if any), the cutting radius of a ball end mill tool and the tool axis orientation. The system then calculates the machinable area based on tool contact and gouge avoidance.

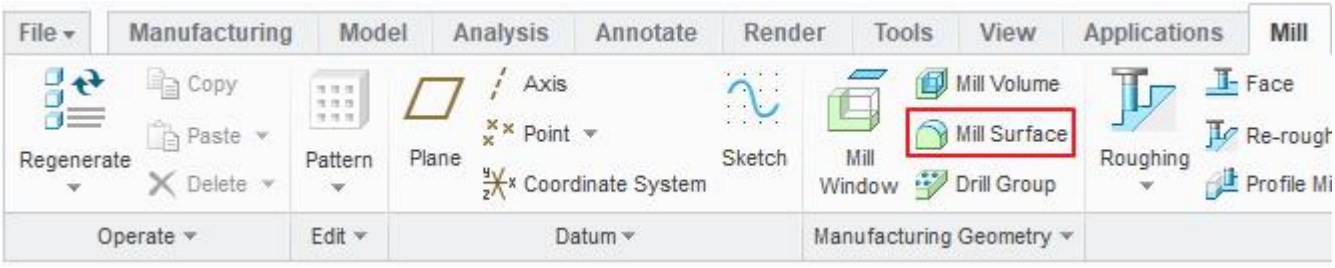
The manufacturing model, referred in this exercise, has been finish machined in the previous chapter “Volume and Local Milling” with a ball end mill of diameter 10mm. So for this exercise we assume that the reference model was previously finished with the ball end mill.

Set the working directory to TOP_COVER and open the manufacturing model top_cover.asm

For better visualization it is good to display the workpiece in wireframe. So pick icon to launch the View Manager Dialog box and double click the predefined style state **Wireframe**.

First of all we will define a Mill Surface to be later referenced in the Profile sequence.

So pick on the Mill tab to invoke the Mill Surface Tool.

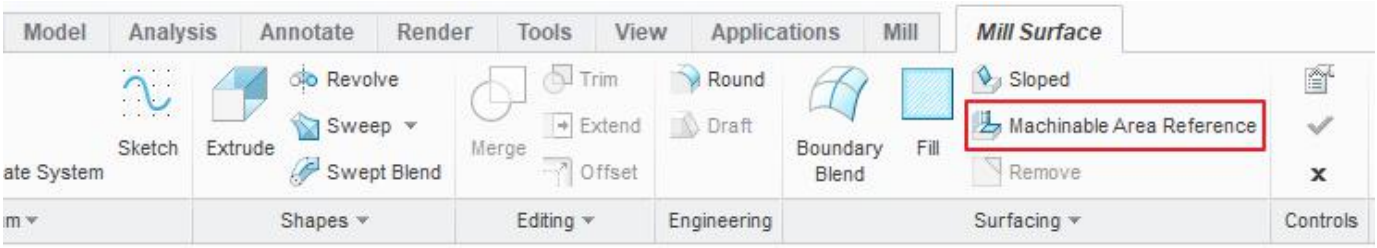


Pick icon to give a name to this volume.

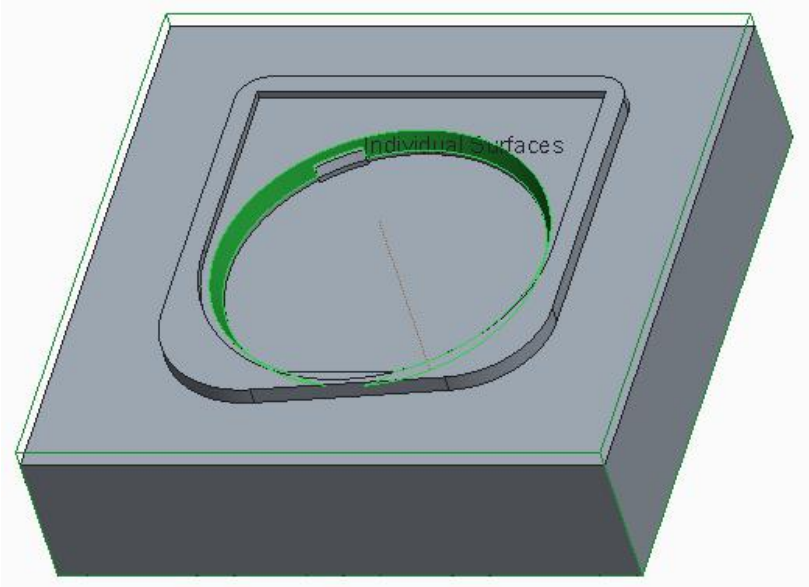
Enter ***MS_A*** as the name of the mill surface and pick

.

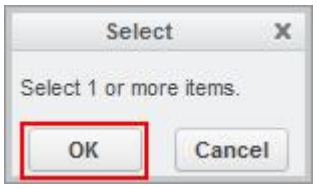
Pick icon to define a "Machinable Area Reference" surface.



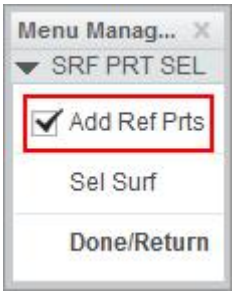
System will ask to select the surfaces for finding the machinable area. So pick the highlighted surface of reference model shown in the figure below.



Select OK or middle-click to proceed.



In the SRF PRT SEL menu, check the **Add Ref Prts** option then select **Done/Return**.



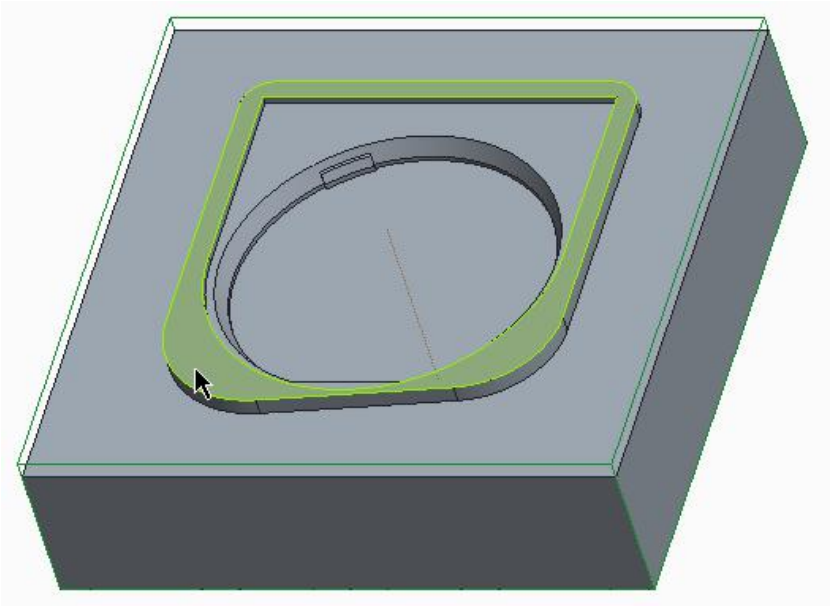
When Add Ref Prts (Add Reference Parts) option is checked, system uses all the surfaces of the reference model to check gouging against.

Now the system will ask to enter radius of ball end mill for calculating Machinable Area.

So enter ***5.2*** and pick icon or middle-click.

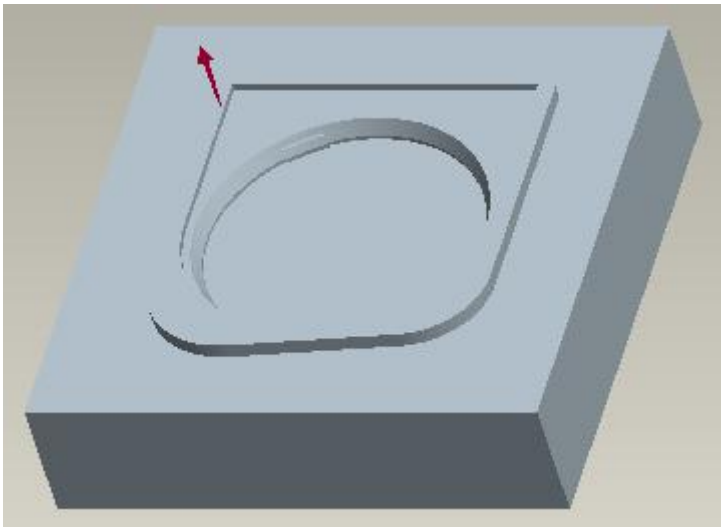
Although we should enter 5 as we used Ball end mill of diameter 10 to machine the cavity in a previous NC Sequence, we have entered a larger value just to make sure that we get continuous and smoother surface.

Now select the top surface of the reference model as the direction reference. The surface is highlighted in the figure below.



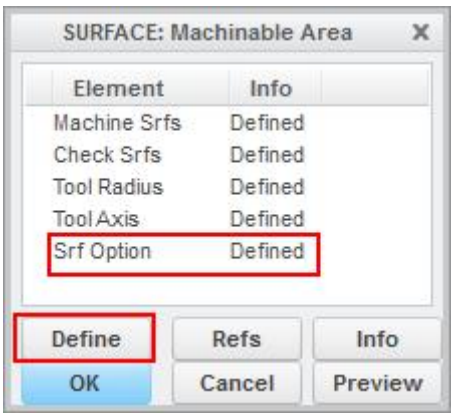
You may also pick the retract plane or any other surface or plane that is parallel to the retract plane.

Make sure that arrow points upward then select **Okay.**

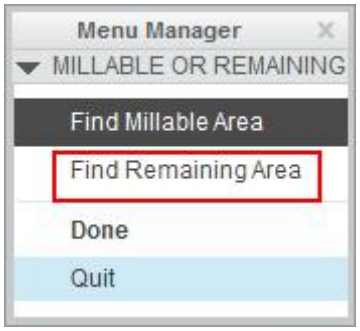


This direction of the arrow specifies the tool axis orientation.

Double-click the **Srf Option** in the dialog box.

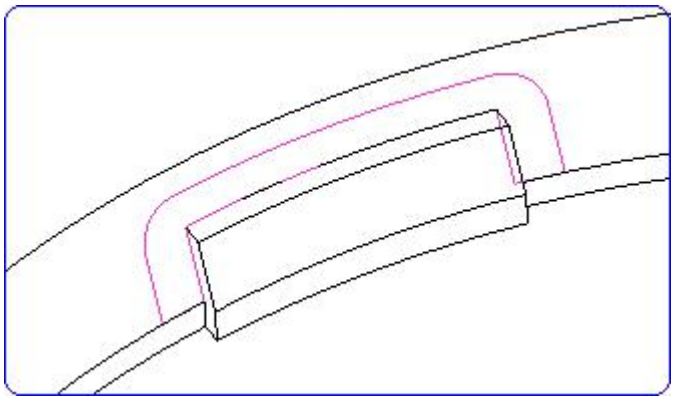


In the MILLABLE OR REMAINING menu select **Find Remaining Area > Done.**



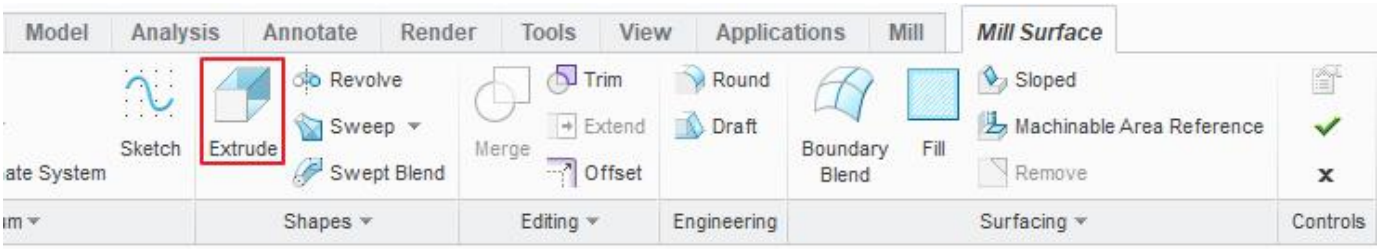
Find Remaining Area option finds the portion of the surface that can not be milled with the specified tool.

Select OK to exit the dialog box. The surface will appear as shown below.

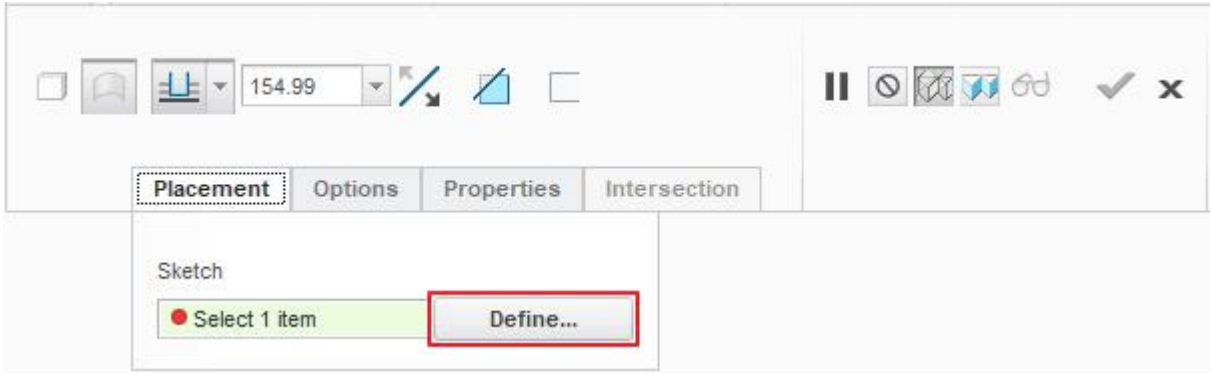


Now we will trim the unwanted part of the surface. We will use the extrude tool to trim the unwanted part of the surface.

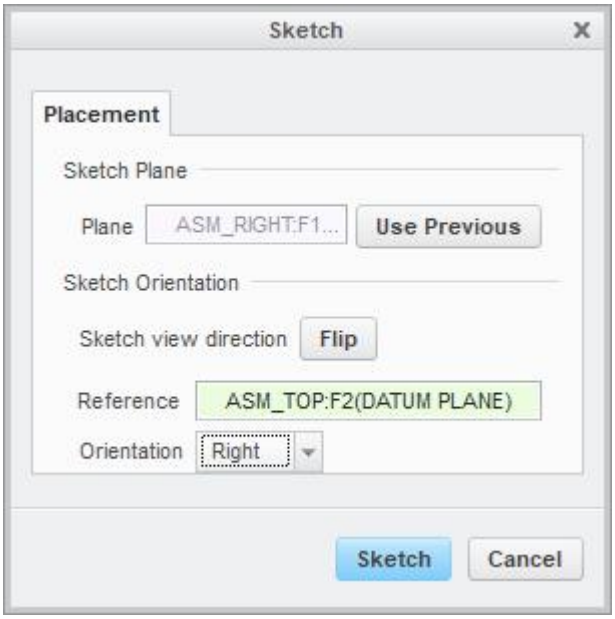
So pick icon to invoke the Extrude Tool.



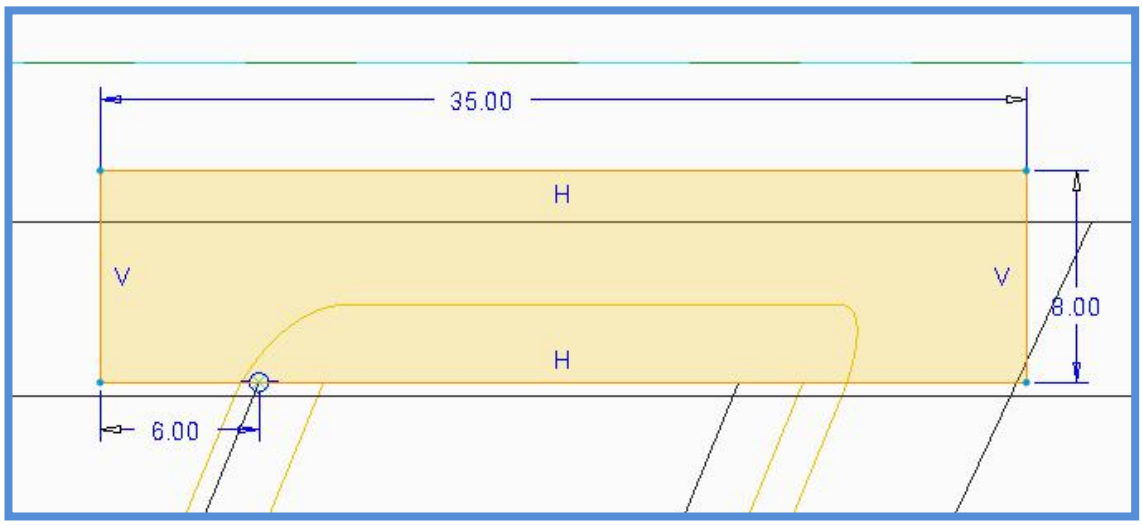
Pick Define... tab in the Placement panel.



Pick **ASM_RIGHT** as the Sketch Plane **ASM_TOP** datum as Right Orientation Reference.



Now sketch a rectangle after choosing suitable references as shown below.

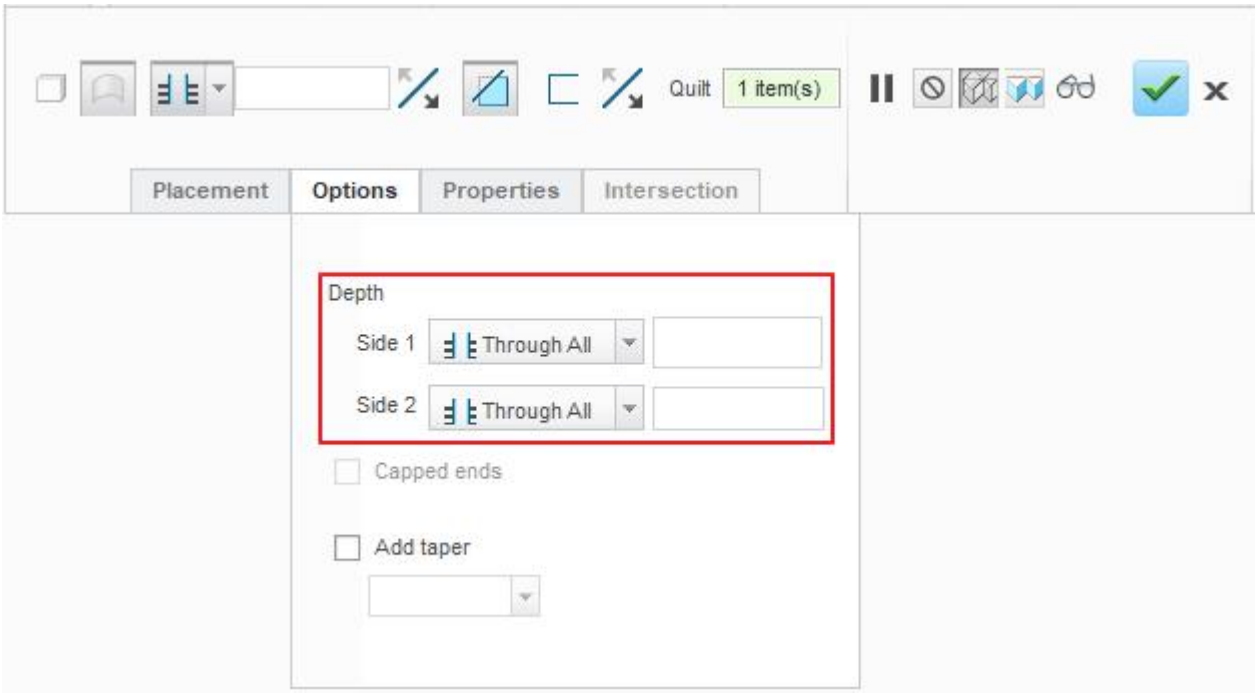


Exact dimensions are not important. The size of rectangle should be such that it encompasses the upper part of the visible surface.

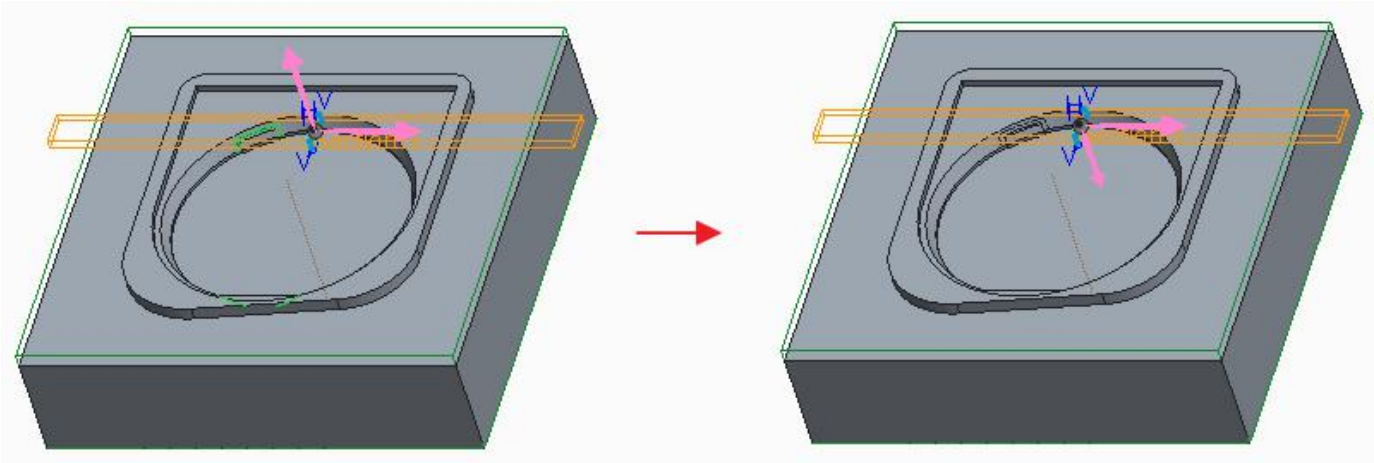
Pick to complete the section

Pick so that the extruded surface trims the existing quilt.

Select **Through All** for both the Side1 and Side 2 in the Options slide-up panel as shown below.



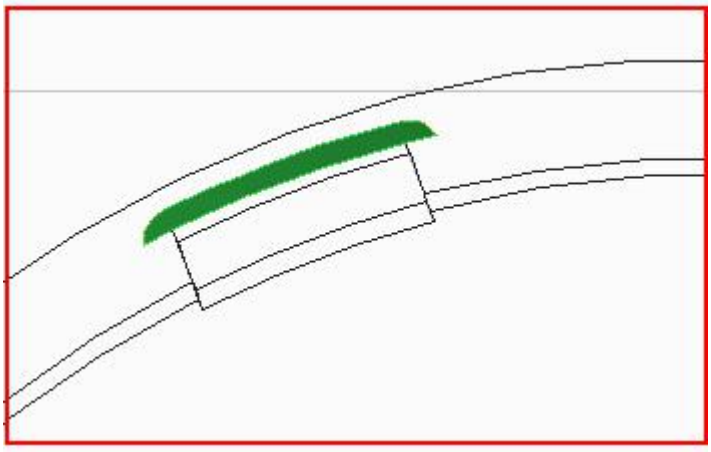
Pick on the Material Side arrow to flip the direction as shown below.



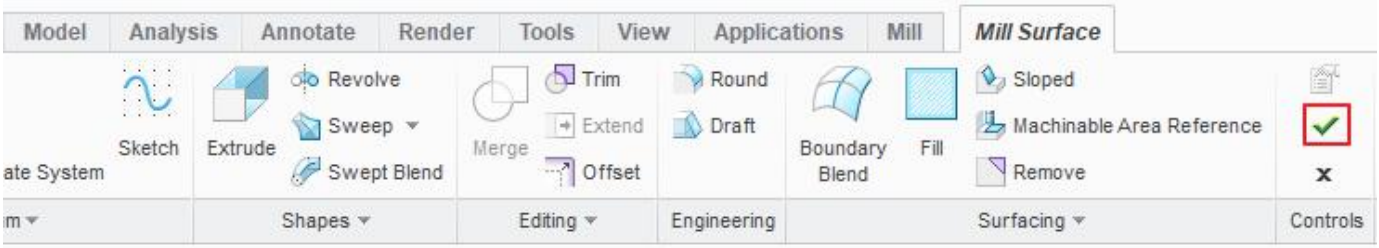
Now the system will keep the inner side of the sketch.

Pick the icon to exit the Extrude tool.

The mill surface will appear as shown below (after highlighting).



Pick and it completes the definition of the mill surface. if you don't get the shown result then you need to redefine the features and fix the problem otherwise you will not be able to do the remaining exercise.



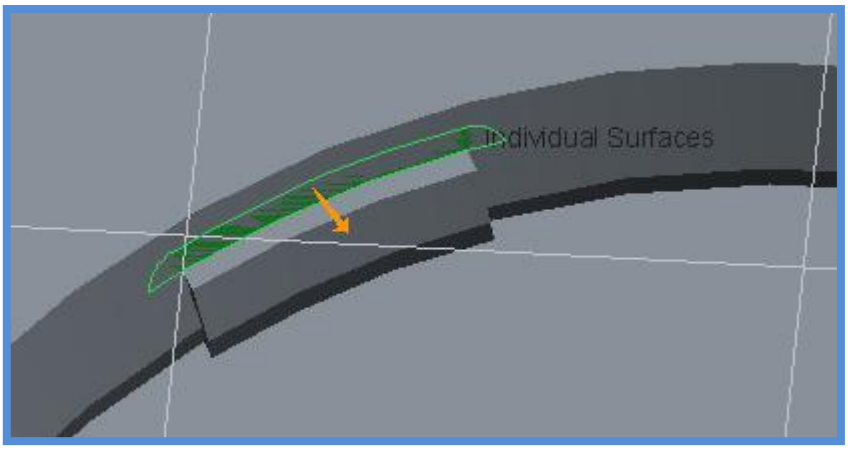
To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

Pick the icon and Tools Setup dialog box will pop up.

In the Tools Setup dialog box pick **FEM10** from the list, then select OK.

Pick Reference tab and notice that the Machining Reference collector becomes active.

Select the above defined mill surface MS_A.



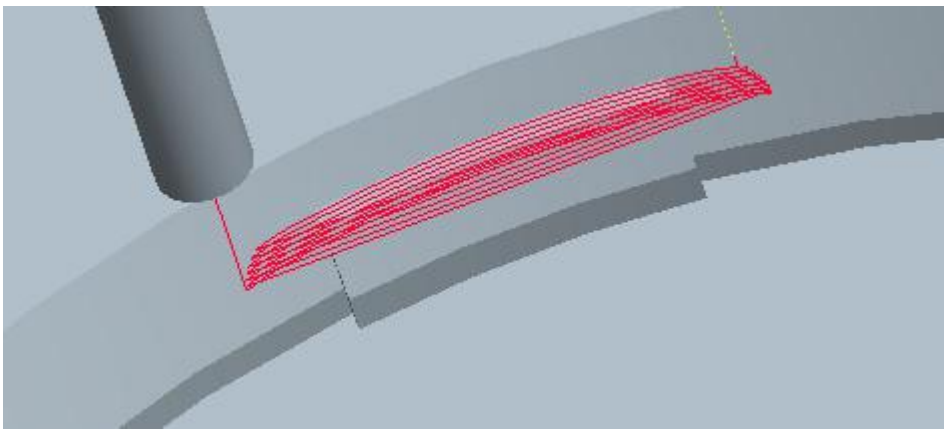
The arrow in the above figure should point away from the surface. This direction indicates side of surface to be machined.

Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

CUT_FEED	1200
STEP_DEPTH	0.5
CLEAR_DIST	1
SPINDLE_SPEED	4000
COOLANT_OPTION	ON
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	20
ENTRY_ANGLE	15
EXIT_ANGLE	15

Pick OK to apply the changes and exit the Edit Parameter Dialog box.

Play the toolpath using **PLAY PATH** dialog box**.** It should appear as shown below.



As the toolpath looks satisfactory so change the following parameters for good surface finish.

STEP_DEPTH	0.2

Remember never to use WALL_SCALLOP_HGT parameter for controlling the surface finish, while machining with a flat end mill.

Now pick Properties tab and enter ***CORNER*** as the name of the sequence.

Pick to save the changes and close the dashboard.

Select **File > Save** to save the manufacturing model.

EXERCISE 3

In this exercise we will define a profile milling NC Sequence to machine a sharp corner with a single pass of cutter.

Set the working directory to COVER_INSERT and open cover_insert.asm

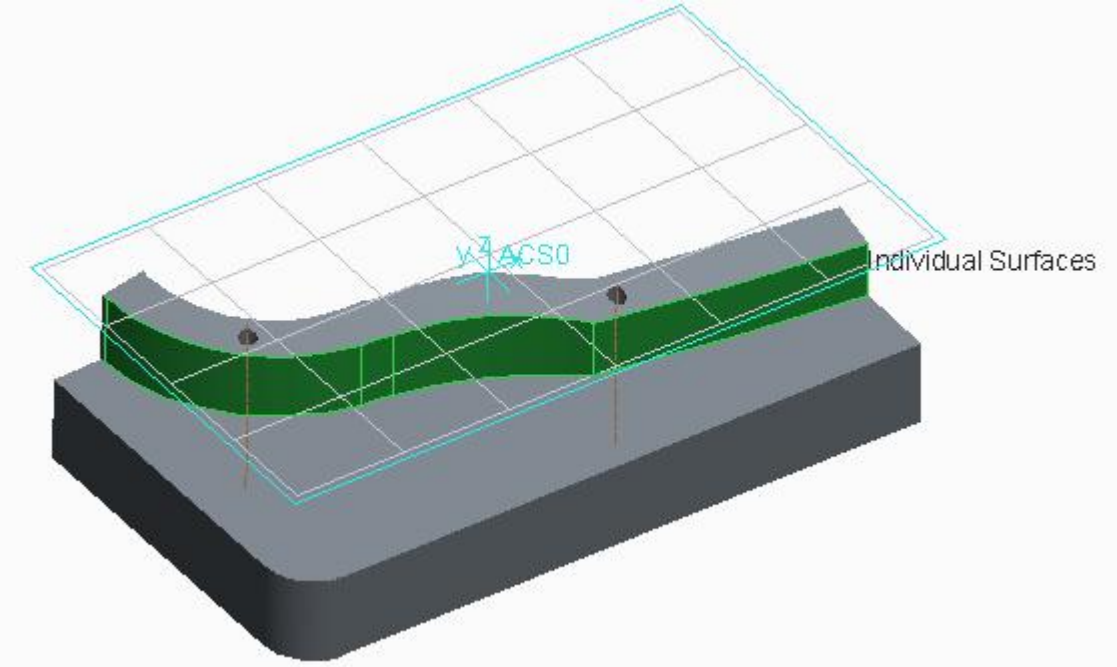
To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

From the drop-down list in front of icon, pick **FEM25**

Alternatively you can pick the icon to access Tools Setup dialog box and then select the tool.

Pick Reference tab and notice that the Machining Reference collector becomes active.

Select the model surfaces (while holding down the Ctrl key) as shown in the figure below.



Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

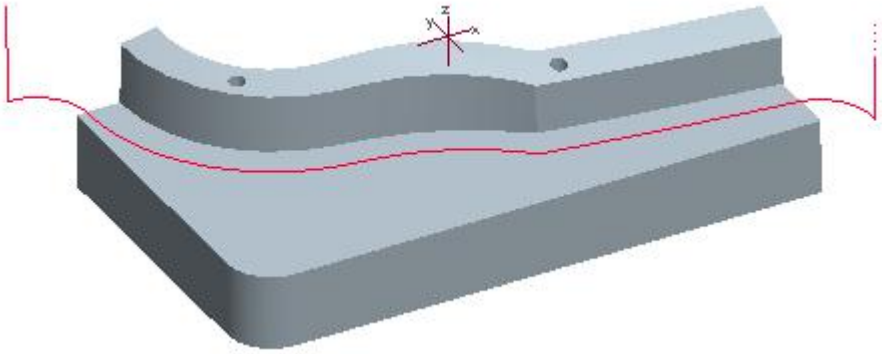
CUT_FEED	1200
STEP_DEPTH	50
CLEAR_DIST	1
SPINDLE_SPEED	3000
COOLANT_OPTION	ON
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	20

The large value of STEP_DEPTH will force the tool to machine only the last slice.

Pick OK to apply the changes and exit the Edit Parameter Dialog box.

Now pick Properties tab and enter ***CORNER*** as the name of the sequence.

Play the toolpath using **PLAY PATH** dialog box and notice that the tool is machining the complete surface with only a single pass as shown below.



Pick to save the changes and close the dashboard.

Select **File > Save** to save the manufacturing model.

EXERCISE 4

In this exercise we will use the profile milling NC Sequence to finish a hole. We will also learn the following

- How to control the plunge location of the tool
- How to perform multi-pass machining
- How to mirror a NC Sequence
- Playing the tool path for multiple Sequences

We assume that the hole has been previously drilled with the drill of diameter 12mm.

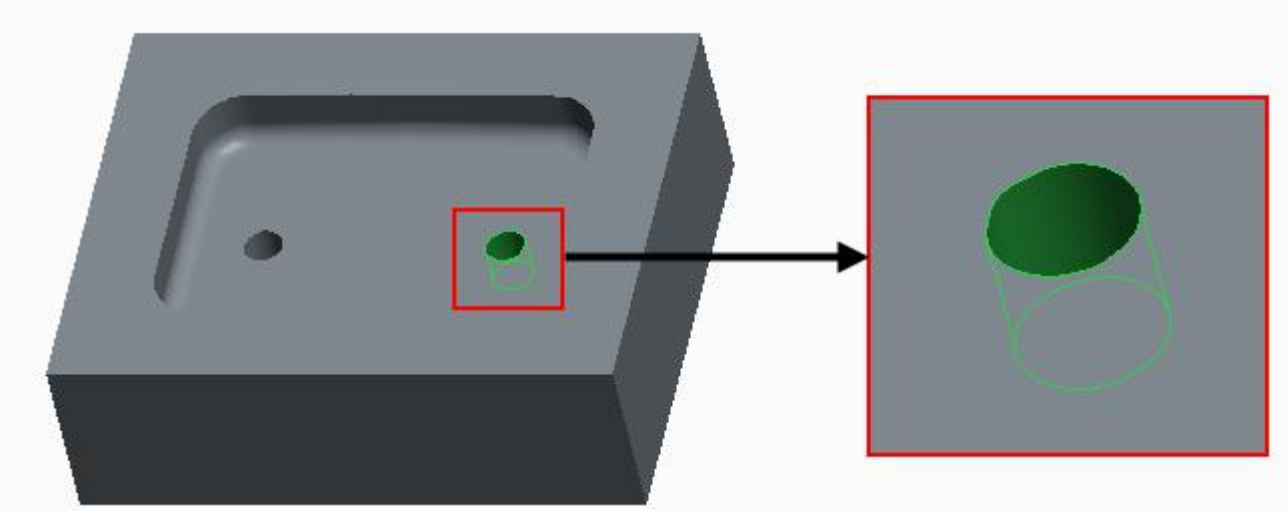
Set the working directory to LID_CAVITY and open the manufacturing model lid_cavity.asm.

To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

From the drop-down list in front of icon, pick **FEM10**

Pick Reference tab and notice that the Machining Reference collector becomes active.

Select the model surfaces (while holding down the Ctrl key) as shown in the figure below.



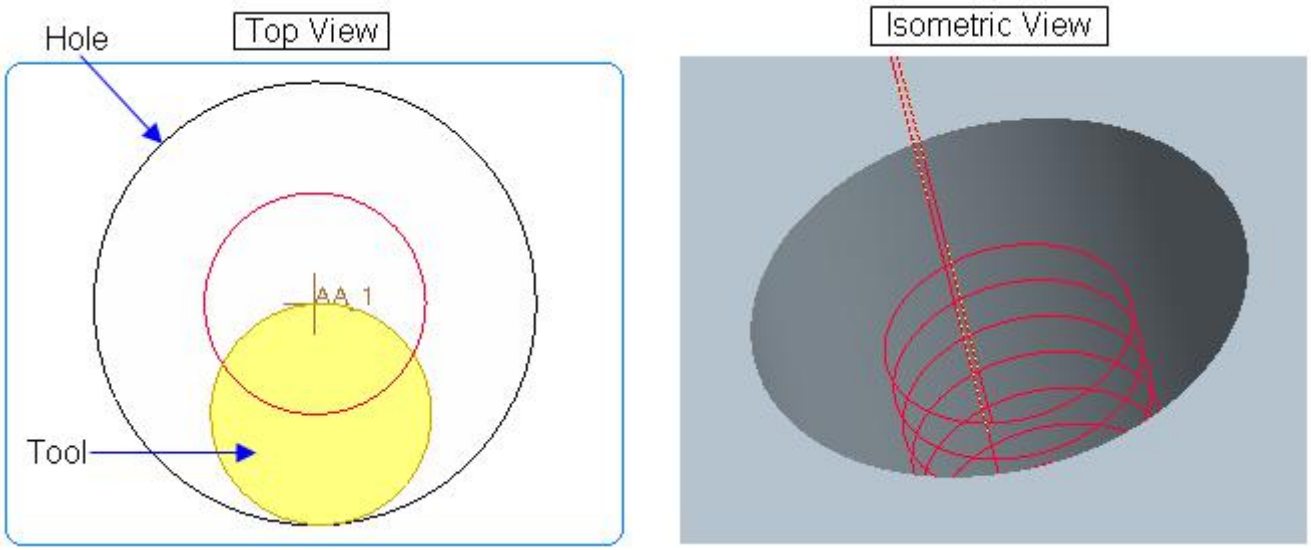
Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

CUT_FEED	1200
STEP_DEPTH	2
CLEAR_DIST	1
SPINDLE_SPEED	5000
COOLANT_OPTION	ON

CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	8

Pick OK to apply the changes and exit the Edit Parameter Dialog box.

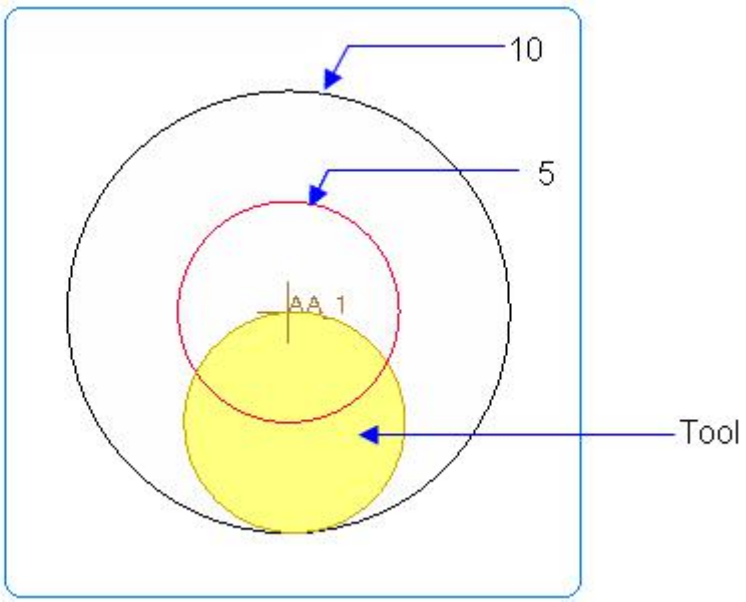
Play the toolpath using **PLAY PATH** dialog box**.** It should appear as shown below.



Notice that there is no lead in or lead out motion. System also warns about this by showing this message.

⚠WARNING: Lead out motion will cause gouging, it is ignored.

It is so because lead radius value is larger than the radius value toolpath consists of. This fact can be elaborated by the following figure.



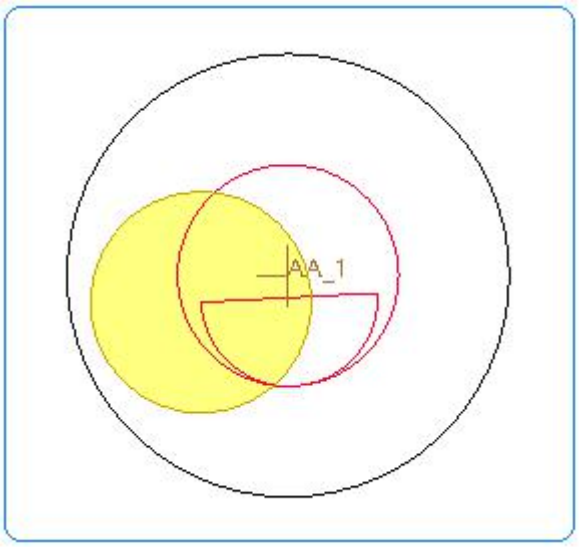
Please notice that “10” is the radius of the hole and 5 is the radius of the tool path arc. In this case tool will gouge the part while making lead in or lead out motion therefore system has ignored them.

So we should set the lead radius less than 5 (radius of toolpath arc) to make sure that tool makes lead in and lead out motions.

So set the following parameter in Edit Parameters Dialog box.

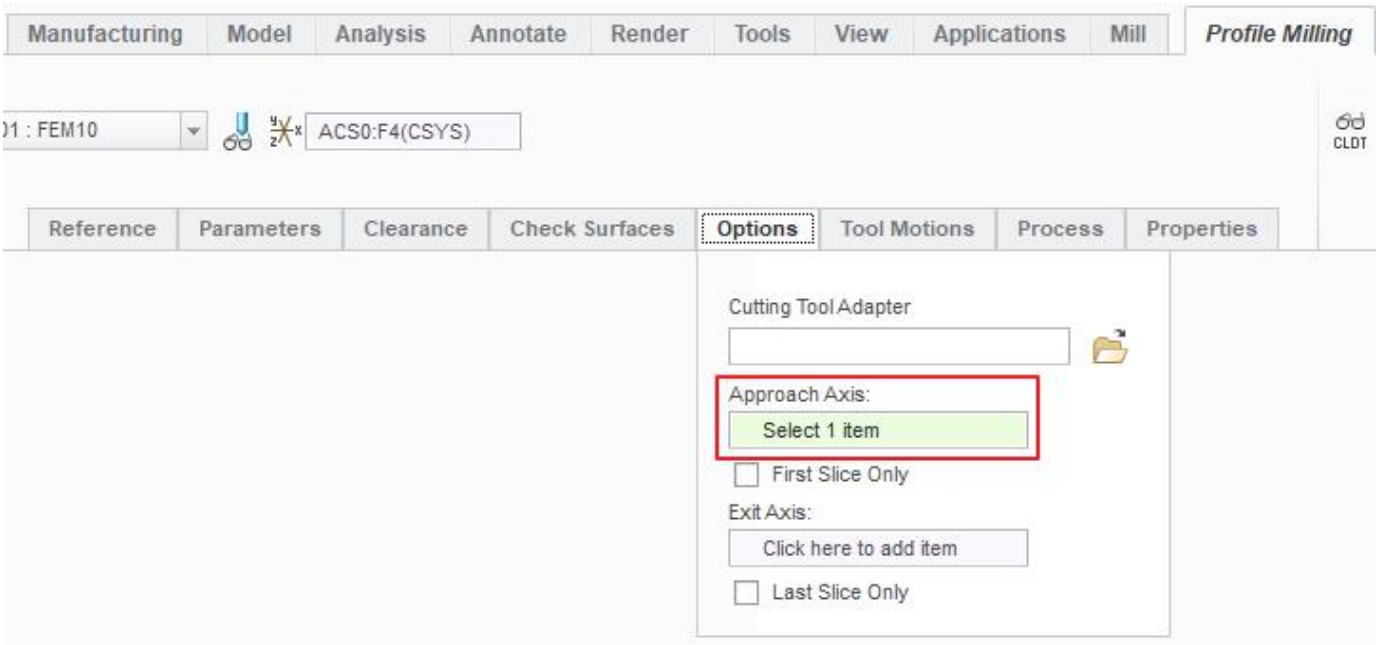
LEAD_RADIUS	4

Play the toolpath using **Screen Play.** It should appear as shown below.

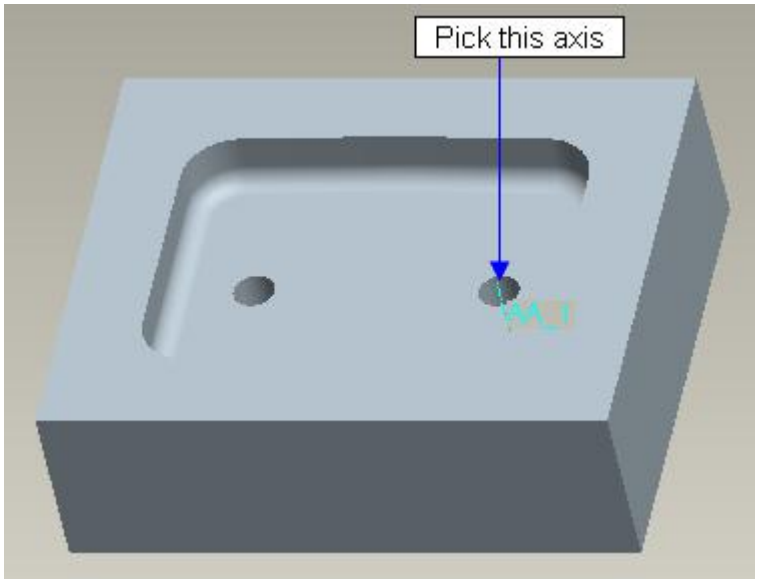


As we assumed at the start of the exercise that the hole was pre-drilled with the drill of 12mm diameter . So the tool will be cutting material during plunge motion because there is enough material at the walls of the hole. The tool must plunge to the next slice at the center of the hole to avoid cutting of material during plunge motion.

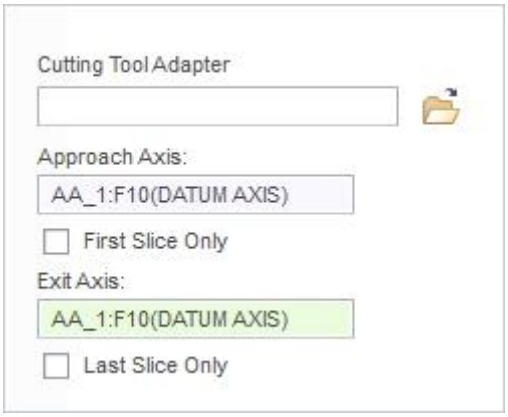
So pick Options tab and click inside the Approach Axis collector to activate it.



Select the axis **AA_1** as Approach axis as elaborated below.



Now pick in the Exit Axis collector to activate it and select the axis **AA_1** as exit axis.

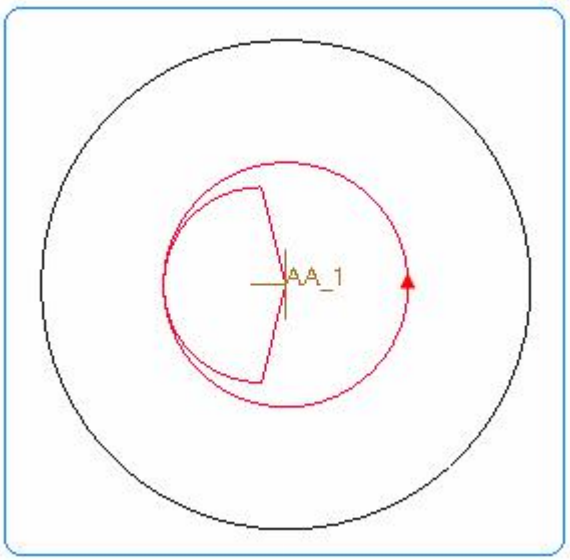


Pick Parameters tab and set the following parameters

PLUNGE_FEED	100

PLUNGE_FEED parameter controls the feed rate along the plunge motion. By default it is set to dash (-) in which case the CUT_FEED is used as PLUNGE_FEED.

Play the toolpath on screen using **PLAY PATH** dialog box and notice that the tool is plunging right on the axis AA_1 as shown below (tool not shown).



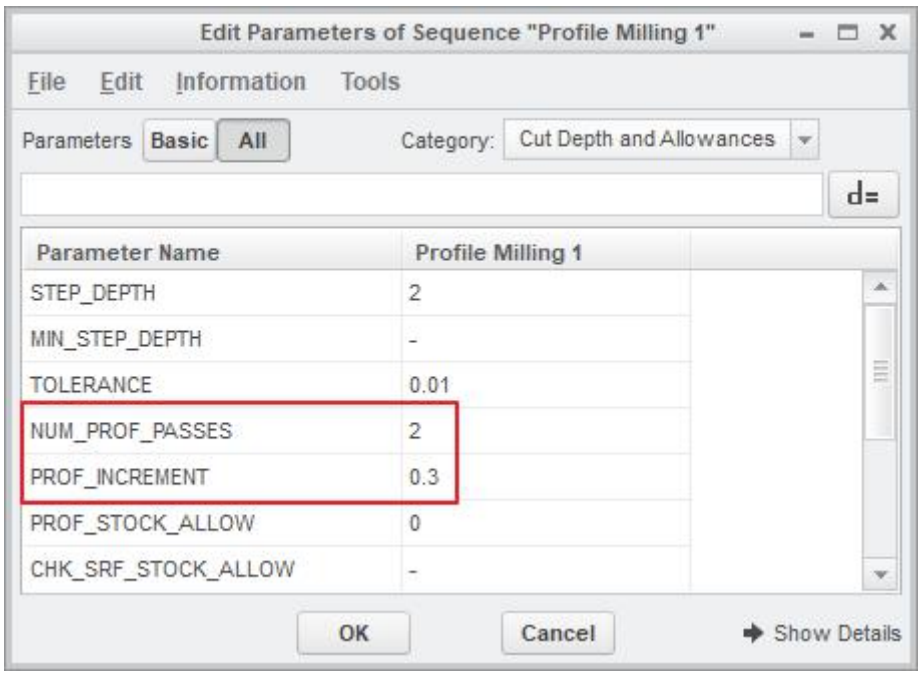
You should notice that tool is climb milling the hole surfaces.

We can machine the hole in multiple passes so that we don't have to define separate sequences for semi-finishing and finishing toolpaths.

Pick in the Parameters panel and set following parameters to machine the hole in two steps.

NUM_PROF_PASSES	2
PROF_INCREMENT	0.3

These parameters can be found in "Cut Depth and Allowances" Category as shown below.



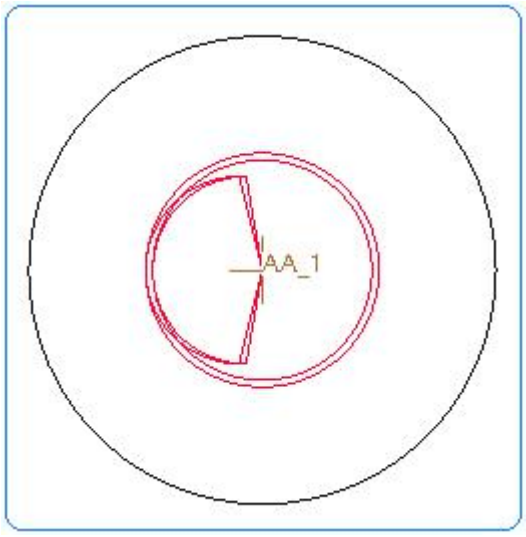
NUM_PROF_PASSES together with PROF_INCREMENT allows to create multiple profiling

passes horizontally offset from each other.

NUM_PROF_PASSES specifies the number of passes that will be generated.

PROF_INCREMENT specifies the horizontal distance between the passes generated according to NUM_PROF_PASSES.

Play the toolpath on screen using **PLAY PATH** dialog box. It will appear as shown below.



Notice that the tool first machines the walls from top to bottom of the hole at the offset specified by PROF_INCREMENT then retracts and again machines the walls from top to bottom without PROF_INCREMENT. This type of machining is called PASS_BY_PASS scanning. We can machine the hole by fist making the profile passes specified by the NUM_PROF_PASSES at the given z level and then plunging down to the next slice. This type of machining is called SLICE_BY_SLICE scanning. The scanning strategy is specified by the SLICE_PATH_SCAN parameter.

As the toolpath looks satisfactory so change the following parameters.

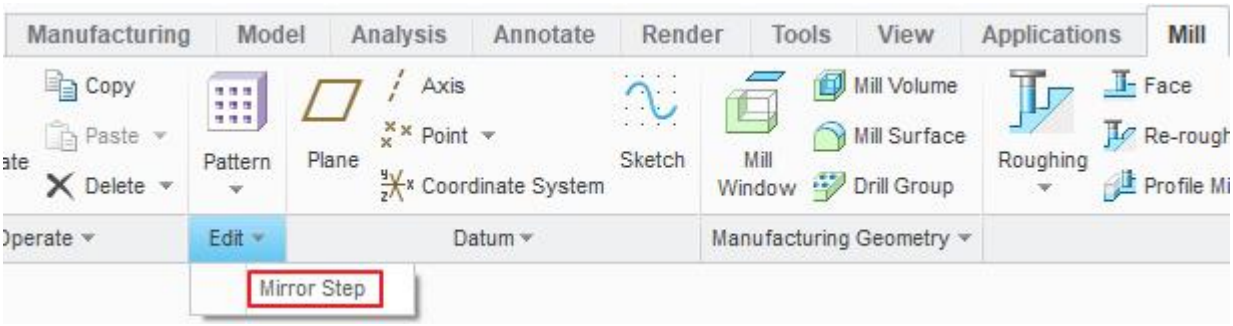
STEP_DEPTH	0.5

Now pick Properties tab and enter ***FINISH*** as the name of the sequence.

Pick to save the changes and close the dashboard.

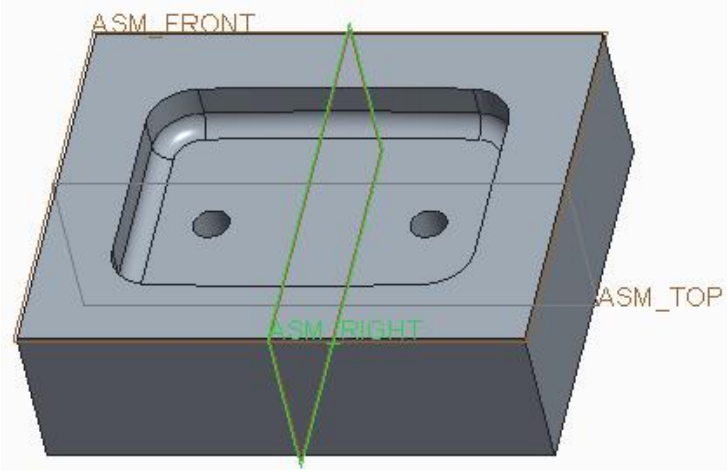
Now we will mirror this sequence to machine the second hole.

So select the FINISH sequence in the model tree and pick **Edit > Mirror Step** on the Mill tab.



In the NC Sequences Mirroring dialog box, pick Select...

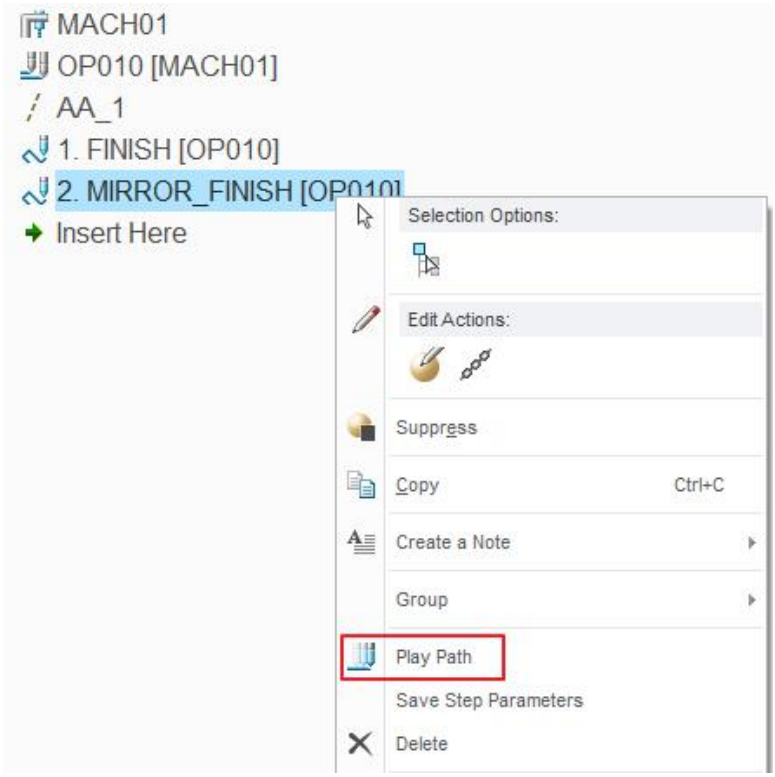
Select the datum ASM_RIGHT as mirroring plane.



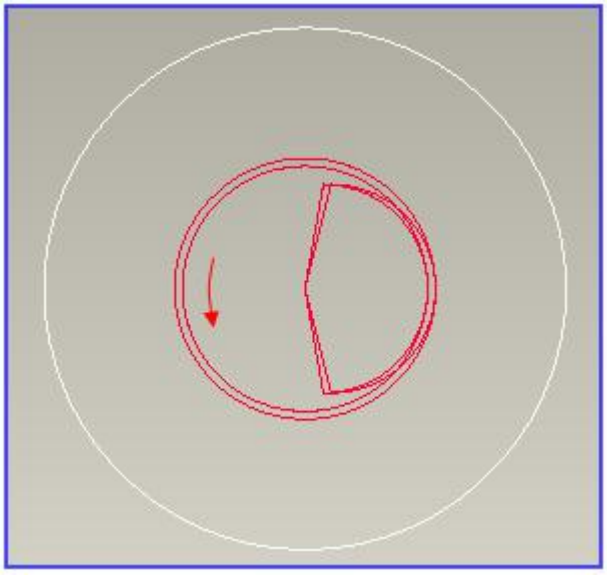
Notice that the option “Keep” is selected by default for Cutting Conditions in the dialog box

Pick OK to complete the mirroring operation.

Right click the MIRROR_FINISH in the model tree and pick **Play Path.**



Play the toolpath on screen using **PLAY PATH** dialog box. It will appear as shown below.

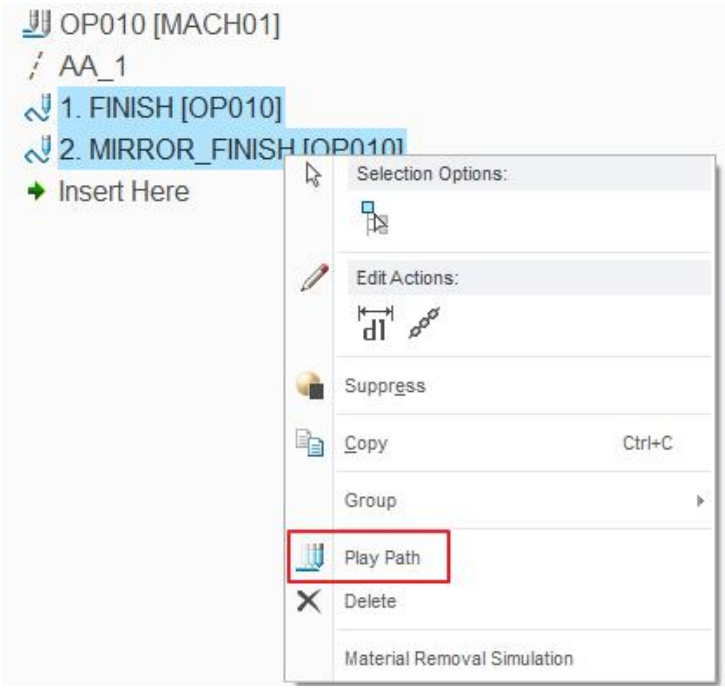


Notice that tool is Climb milling the hole surfaces. It is so because the option “Keep” is selected for Cutting Conditions in the NC Sequences Mirroring dialog box

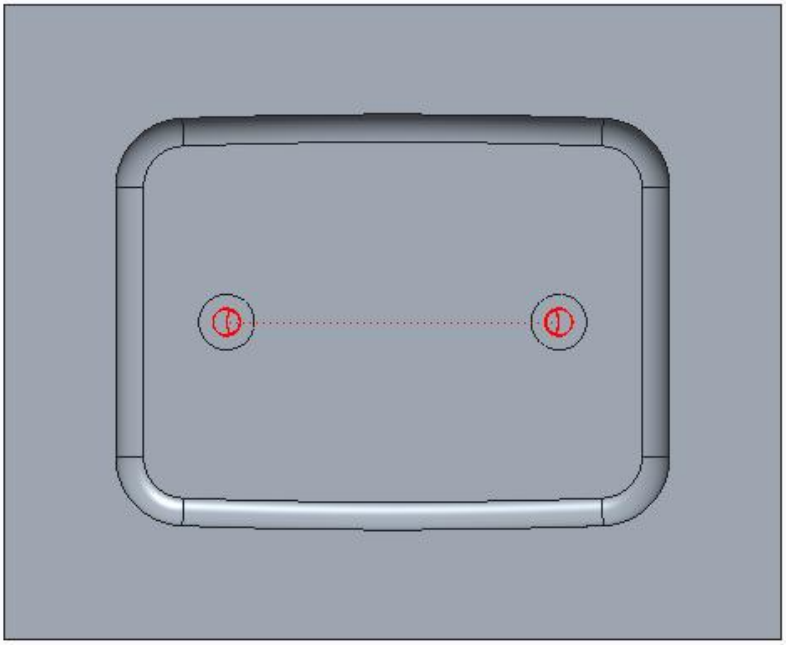
Now we will play the tool path for both of the sequences together.

Select the both sequences in the model tree while holding down the Ctrl key.

Right-click and select **Play Path.**



Click the Play Forward button in the Play Path Dialog box. Toolpath will appear as shown below.



Select **File > Save** to save the manufacturing model.

EXERCISE 5

In this exercise we will define a profile milling NC Sequence to machine an undercut, with a special round insert milling cutter.

The topics covered in this exercise are as follows

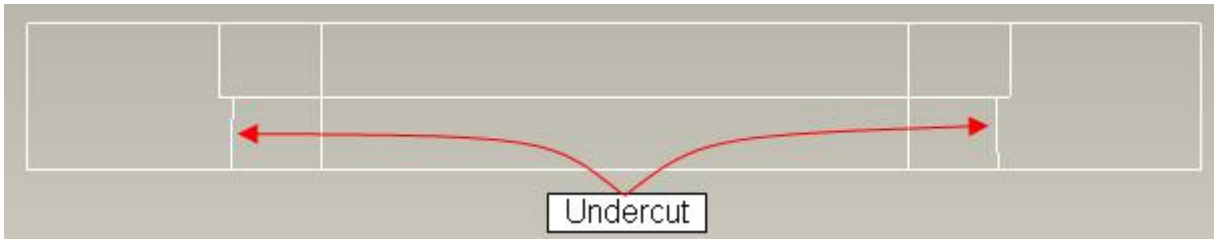
- Defining a side milling cutter
- Selection of surfaces by loop option
- Checking gouges visually, by placing the tool at desired location on the tool path

The cutter to be used in this exercise is shown below.

Round Insert Cutter from Sandvik

Set the working directory to BACK_PLATE and open the manufacturing model back_plate.asm.

Change the view to Front and zoom in to the reference part and see that lower surfaces of the plate form an undercut.

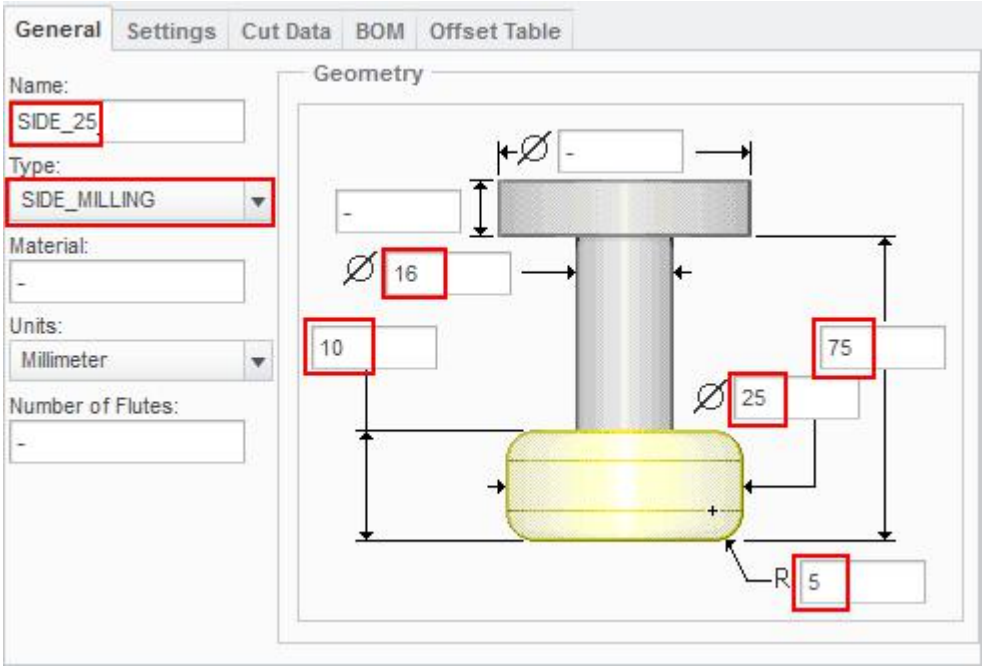


Conventionally, to machine this undercut we will need to machine first from top side and then invert the plate and machine the bottom side i.e. two separate operations in Creo NC. By using the special round insert cutter we will machine the complete plate with only one operation.

To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

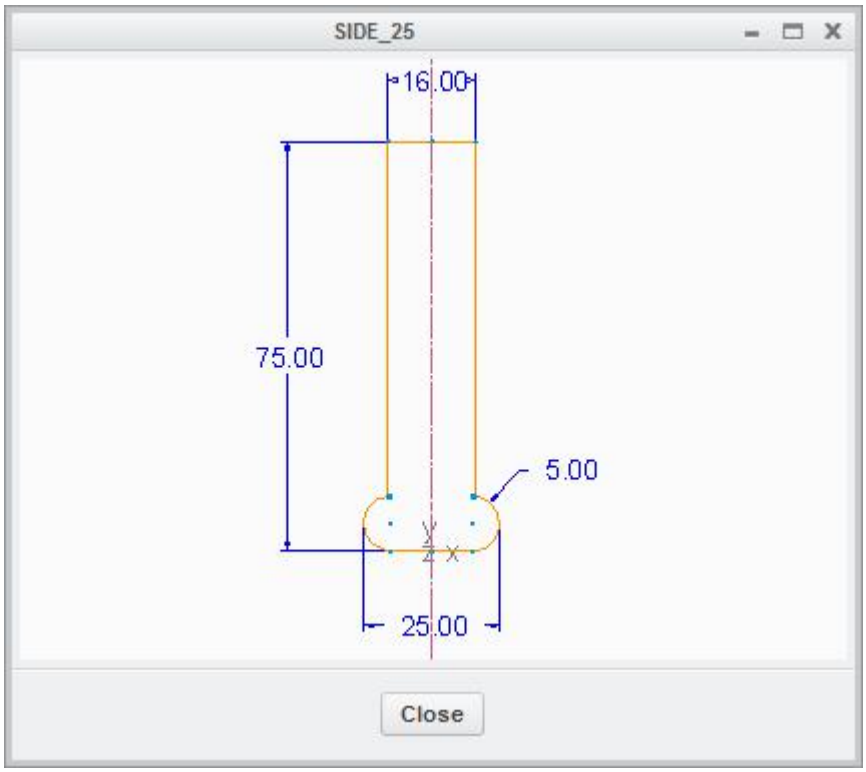
Pick the icon and Tools Setup dialog box will pop up.

In the Tools Setup dialog box, define a new “Side Milling” cutter of diameter 25. Other parameter of the tool should be as shown below.



Pick Apply tab and tool will appear in the Tools List.

To see the tool in separate window click in the dialog box and the detailed sketch of the tool will appear as shown below.



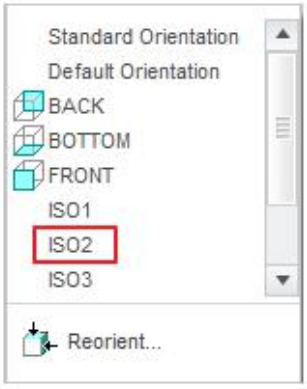
Pick Close to close this window.

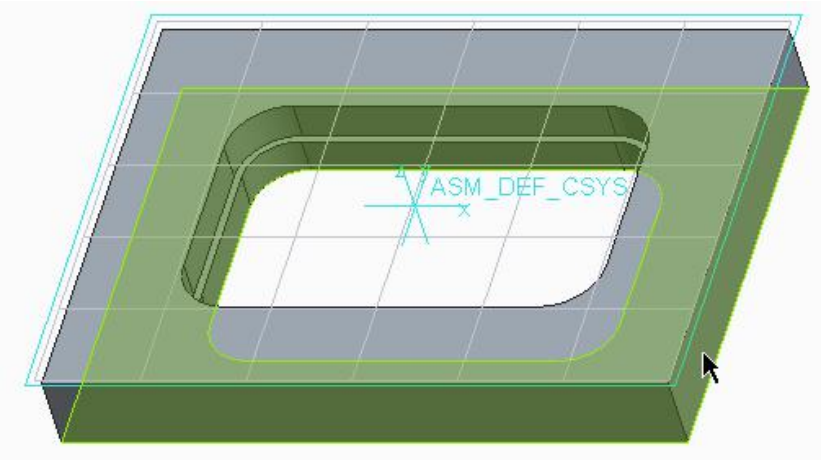
Select OK to complete the definition of the tool.

Pick Reference tab and notice that the Machining Reference collector becomes active.

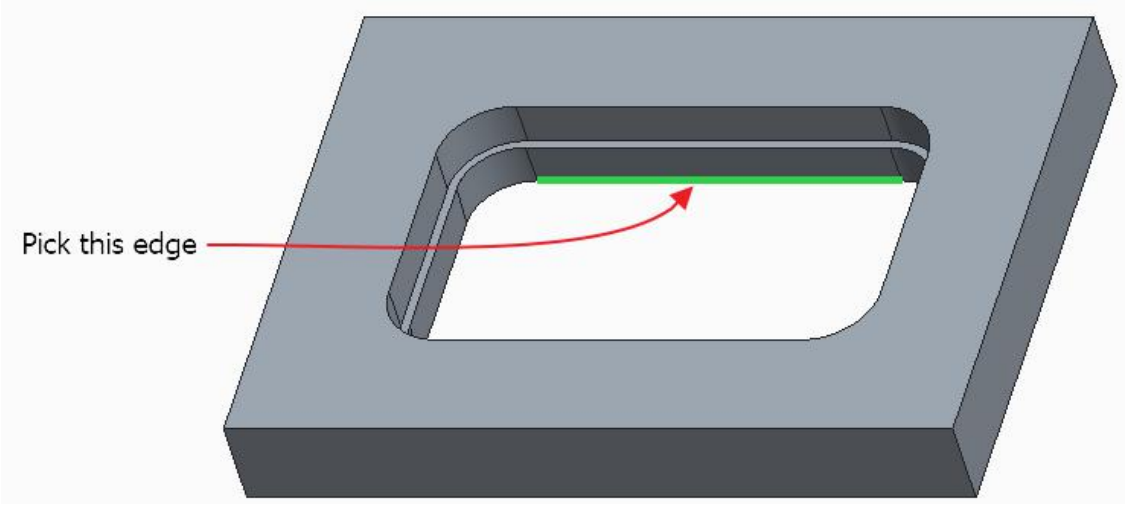
Here we will use the Loop selection technique to select the desired surfaces quickly.

Now select the bottom surface of the reference model as shown below. (Set the view to **ISO2** to avoid any ambiguity.)

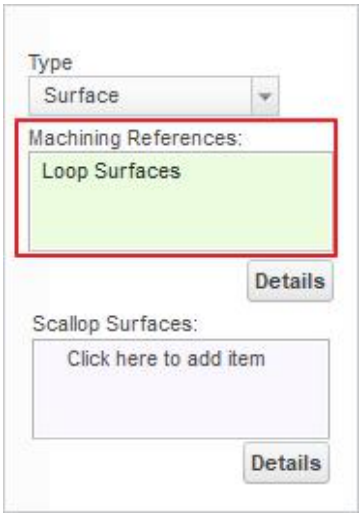


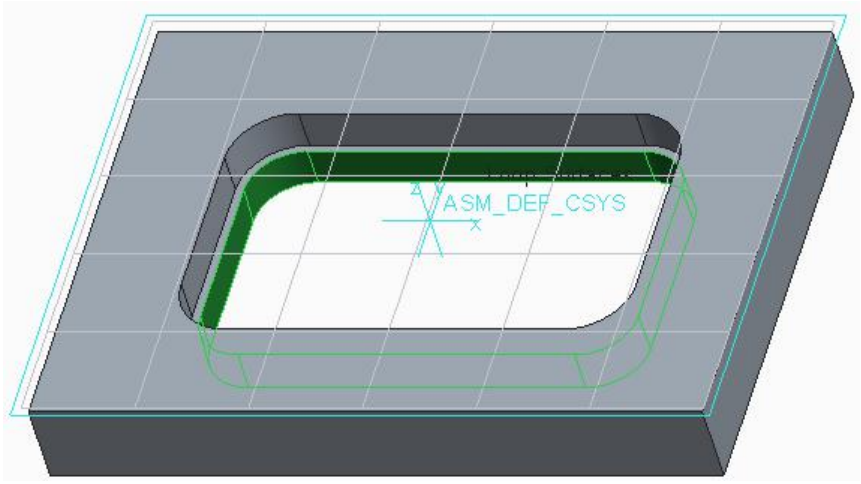


Now hold down Shift key and select any edge forming the inner loop of selected surface as shown below.



The selected loop of surfaces will be highlighted as shown below.





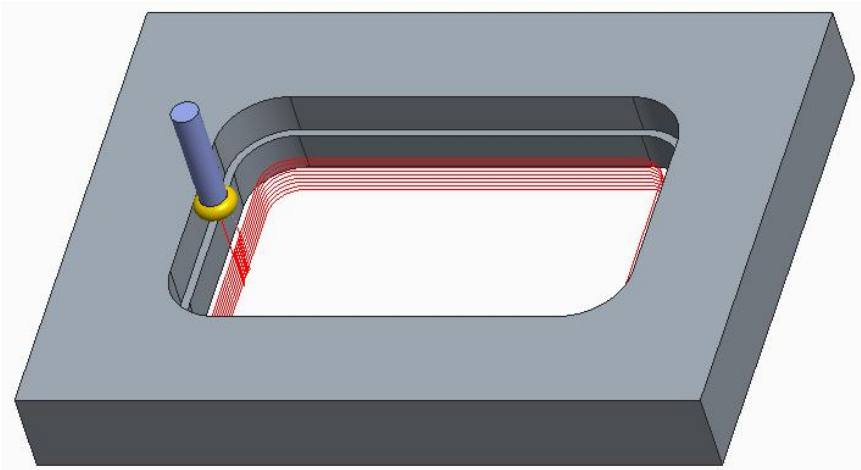
Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

CUT_FEED	1200
STEP_DEPTH	3
CLEAR_DIST	1
SPINDLE_SPEED	2000
COOLANT_OPTION	ON
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	20
ENTRY_ANGLE	20
EXIT_ANGLE	20

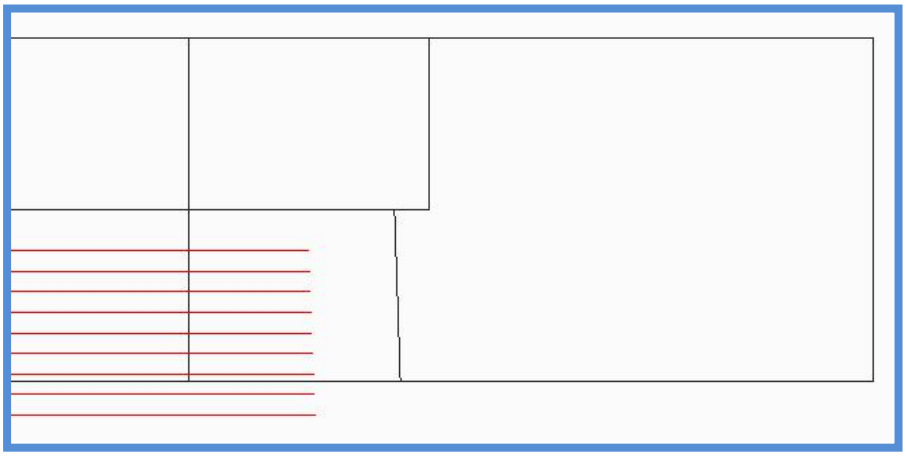
We have set the large value of STEP_DEPTH to make the computation of the toolpath faster. Later when the toolpath has been fine tuned, we will change it to suit the surface finish required.

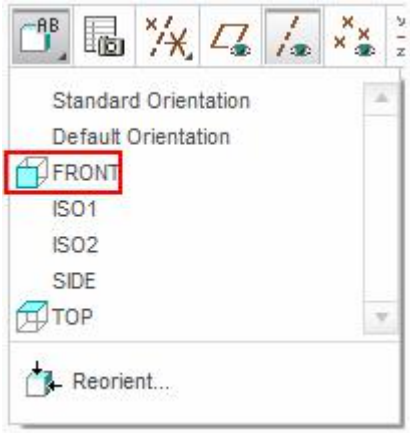
Pick OK to apply the changes and exit the Edit Parameter Dialog box.

Play the toolpath using **PLAY PATH** dialog box and it will appear as shown below.

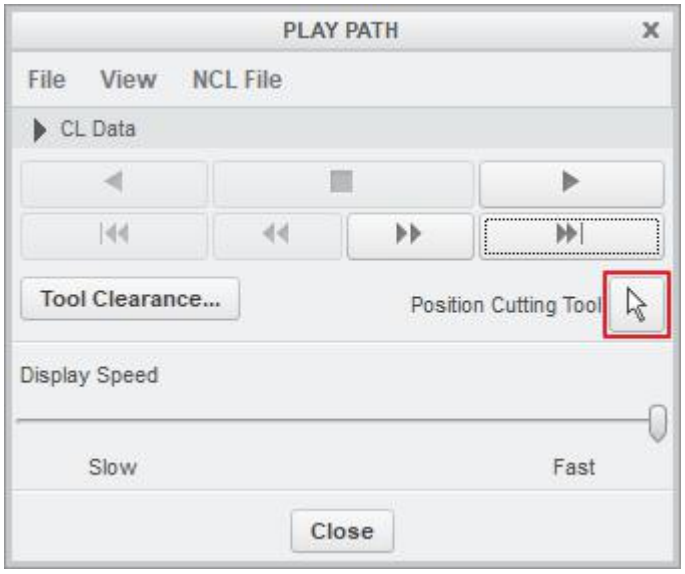


Change the view to Front and zoom-in to see that toolpath has followed the surface forming undercut as shown below (in Wireframe mode).

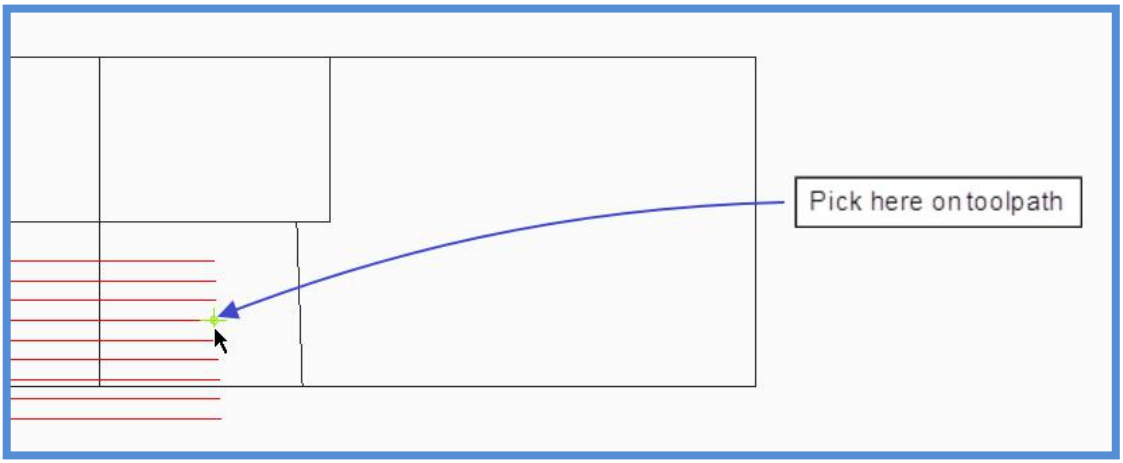




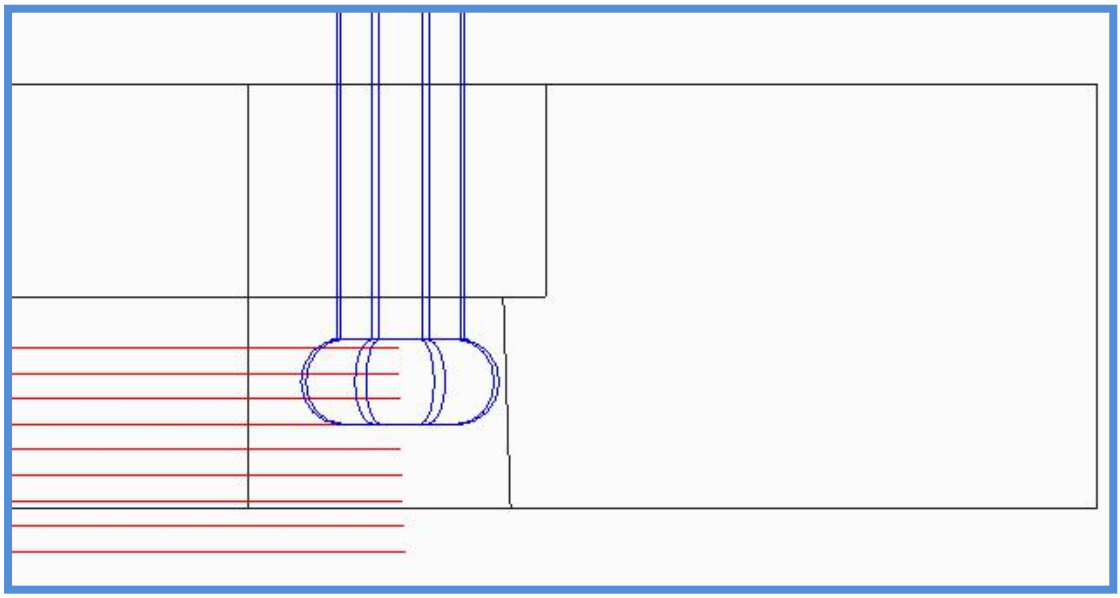
To check visually if the tool is gouging anywhere on the part, pick icon on the PLAY PATH dialog box.



Now pick on the toolpath as shown below.



The tool will move to the picked point as shown below.

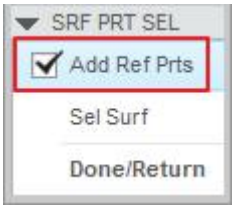


Notice that tool shank is clear of the reference part surfaces and does not gouges the part anywhere.

To be sure, that the tool does not violate the reference part, we should perform the gouge checking.

So select icon on the dashboard as shown below.

Now the system will ask to select the surfaces for checking gouges. As we want to check the gouges for the surfaces of complete part so check the **Add Ref Prts** option.



When Add Ref Prts option is checked, system adds all surfaces of the reference part for gouge checking.

Select **Done > Run.**

System will report that no gouges are detected.

It is always good practice while machining undercut to use larger values for ENTRY_ANGLE and EXIT_ANGLE parameter. So that on the retract path tool does not gouges the part.

As the toolpath looks satisfactory so change the following parameters for good surface finish.

STEP_DEPTH	0.2

Now pick Properties tab and enter ***UNDERCUT*** as the name of the sequence.

Pick to save the changes and close the dashboard.

Select **File > Save** to save the manufacturing model.

EXERCISE 6

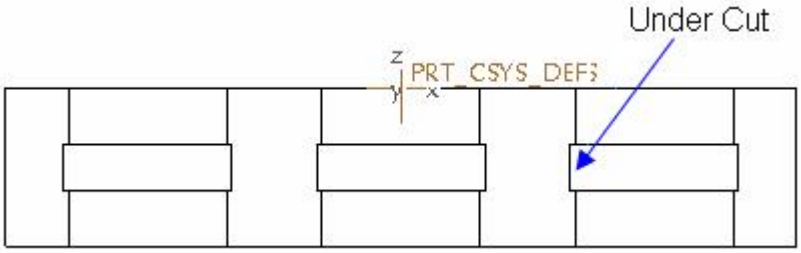
In this exercise we will define a profile milling NC Sequence to machine an undercut with a T-slot cutter.

The topics covered in this exercise are as follows

- Defining a new profile milling NC Sequence to machine and undercut
- Defining a new Key cutter
- Introduction to NORMAL_LEAD_STEP parameter
- Inserting a CL Command at the end of CL Data
- Patterning a NC Sequence
- Generating CL Data for a patterned NC Sequence
- Gouge checking patterned NC Sequence

Set the working directory to FLOAT_PLATE and open the manufacturing model float_plate.asm.

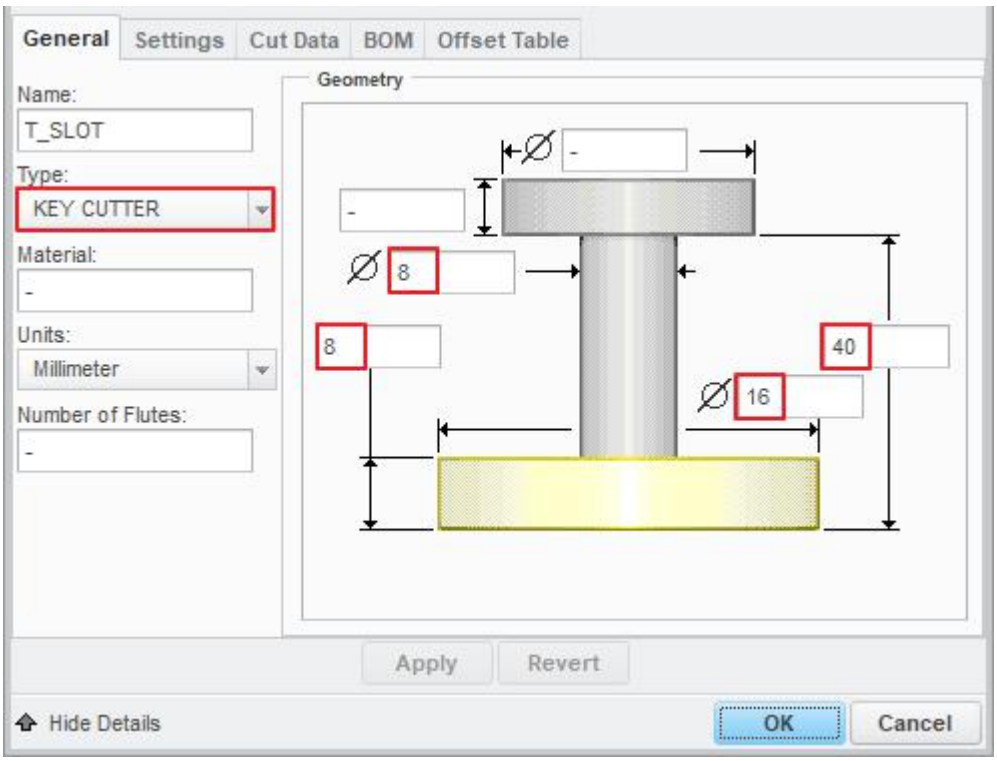
Change the view to FRONT and zoom-in to the reference part and see that each bore has an undercut.



To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

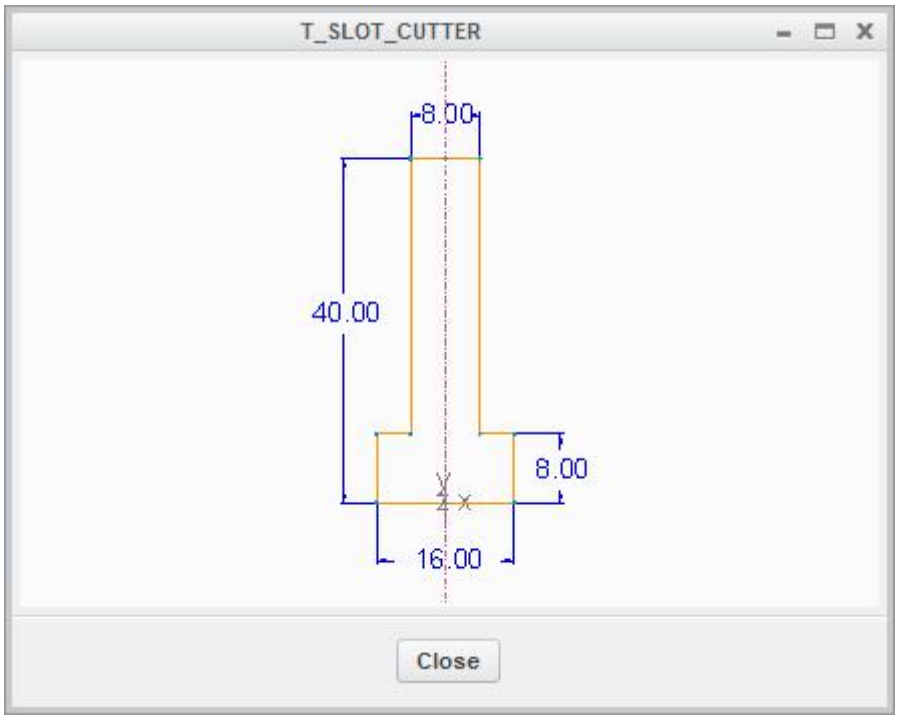
Pick the icon and Tools Setup dialog box will pop up.

In the Tools Setup dialog box, define a new Key Cutter of diameter 16. Other parameter of the tool should be as shown below.



Pick Apply tab and tool will appear in the Tools List.

To see the tool in separate window click and the detailed sketch of the tool will appear as shown below.

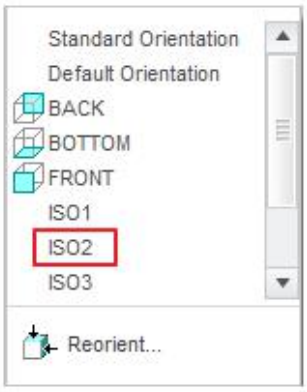


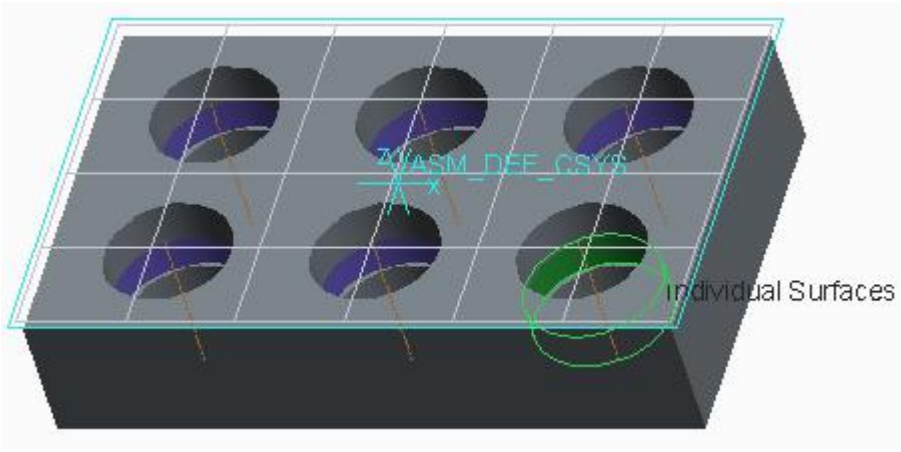
Pick Close to close this window.

Select OK to complete the definition of the tool.

Pick Reference tab and notice that the Machining Reference collector becomes active.

Select the surfaces forming undercut of first hole as shown in the figure below (view is set to ISO2). It is important to select the surfaces belonging to the first instance in the pattern as it will allow to reference pattern the NC Sequence.





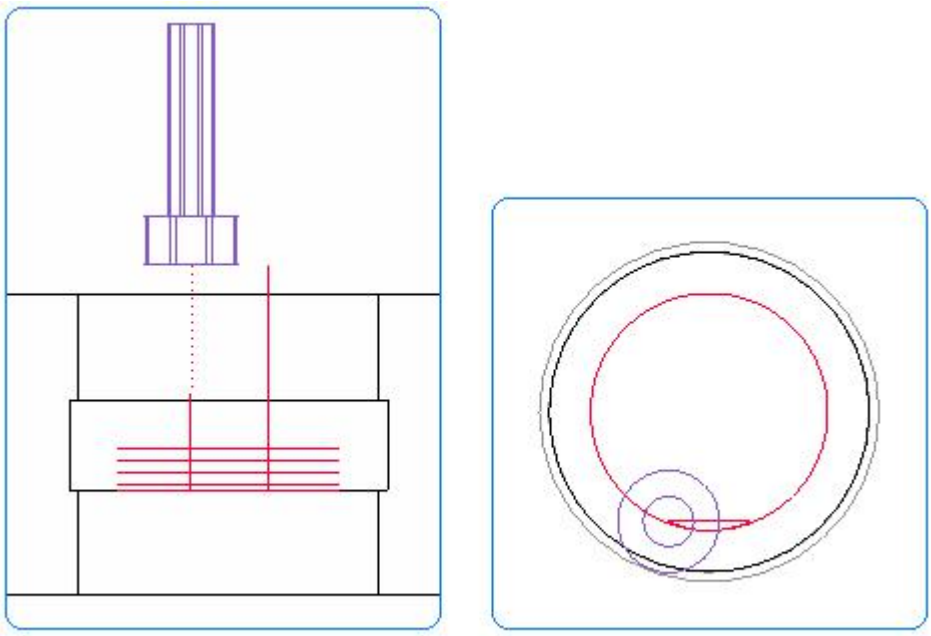
Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

CUT_FEED	800
STEP_DEPTH	2
CLEAR_DIST	1
SPINDLE_SPEED	2000
COOLANT_OPTION	ON
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	15
ENTRY_ANGLE	25
EXIT_ANGLE	25

It is always good practice while machining undercut to use larger values for ENTRY_ANGLE and EXIT_ANGLE parameter. So that on the retract path, tool does not gouges the part.

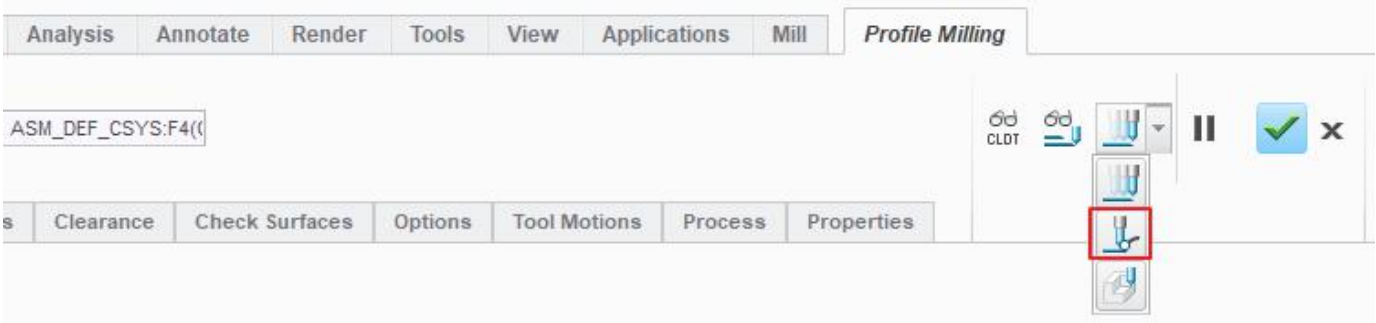
Pick OK to apply the changes and exit the Edit Parameter Dialog box.

Play the toolpath using **PLAY PATH** dialog box**.** It should appear as shown below.

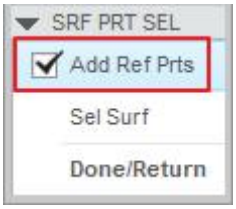


To be sure, that the tool does not violate the reference part, we should perform the gouge checking.

So select icon on the dashboard as shown below.



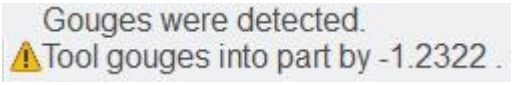
Now the system will ask to select the surfaces for checking gouges. As we want to check the gouges for the surfaces of complete part so check the **Add Ref Prts** option.



When Add Ref Prts option is checked, system adds all surfaces of the reference part for gouge checking.

Select **Done > Run.**

System will report gouges by the following message at message area.



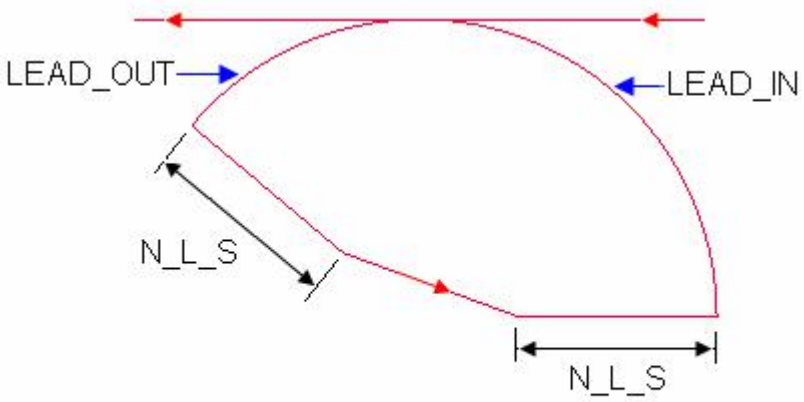
Tool is gouging the part during plunge and retract motion. So we should modify the toolpath so that tool clears the reference part during motions along z-axis.

So set the following parameter.

NORMAL_LEAD_STEP	5

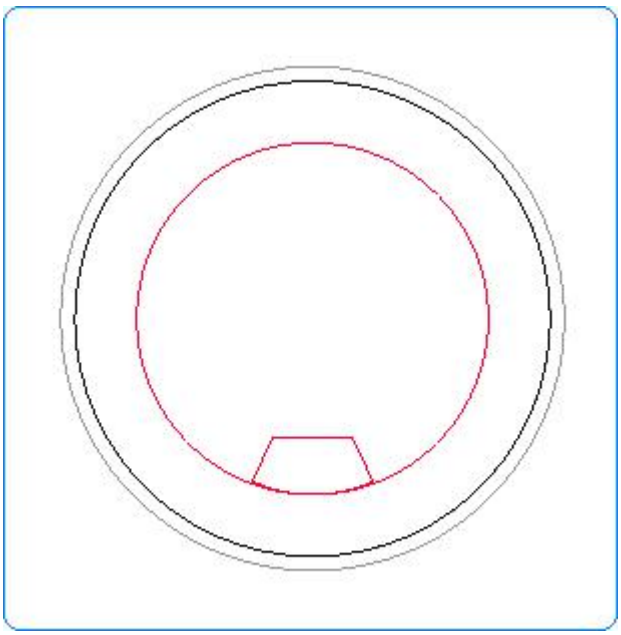
You can find this parameters by picking All tab and changing to “Entry/Exit Motions” category

NORMAL_LEAD_STEP is the length of the linear movement that is normal to the lead-in or lead-out motion.



N_L_S = NORMAL_LEAD_STEP

Play the toolpath using **PLAY PATH** dialog box. It should appear as shown below.

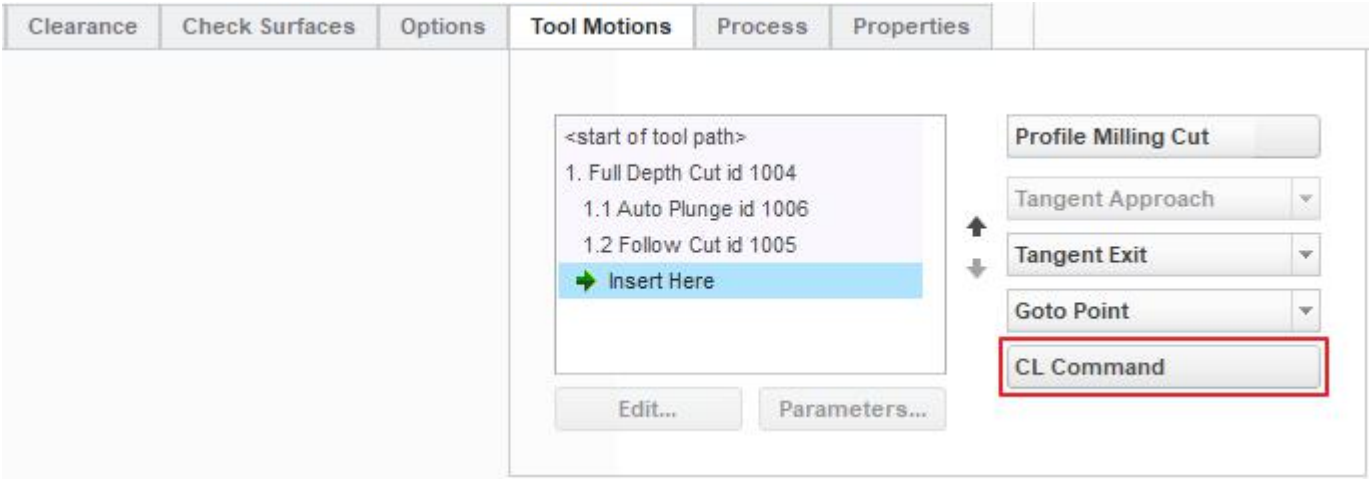


To perform gouge checking pick icon and check the **Add Ref Prts** option**.**

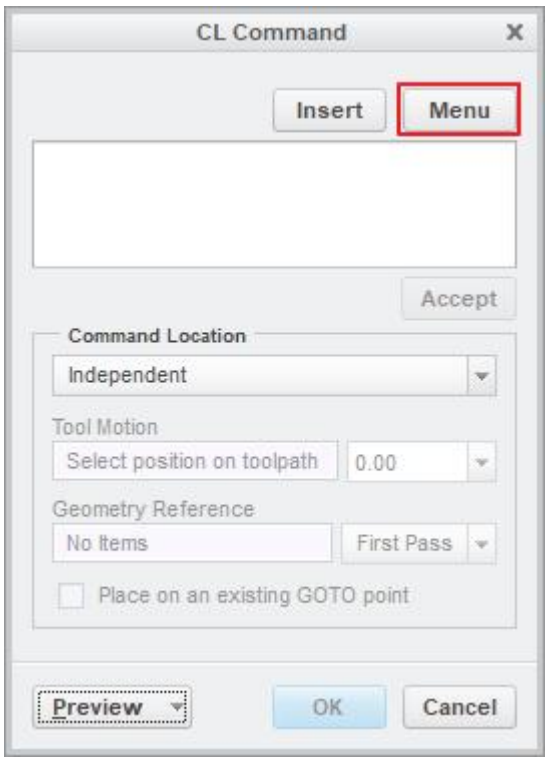
Select **Done > Run.** System will report that no gouges are detected.

Now we will insert an an Optional Stop at the end of the CL Data.

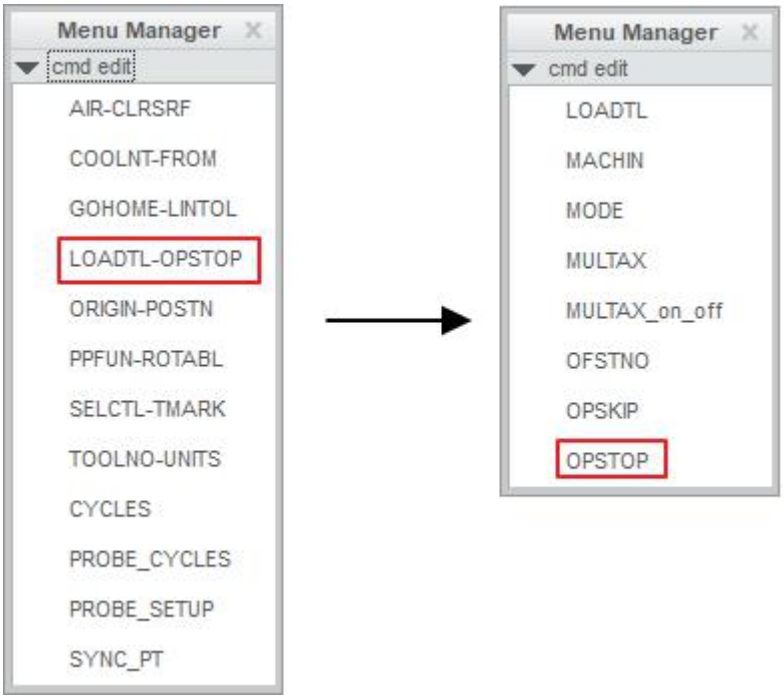
So select Tool Motions tab and pick on CL Command icon.



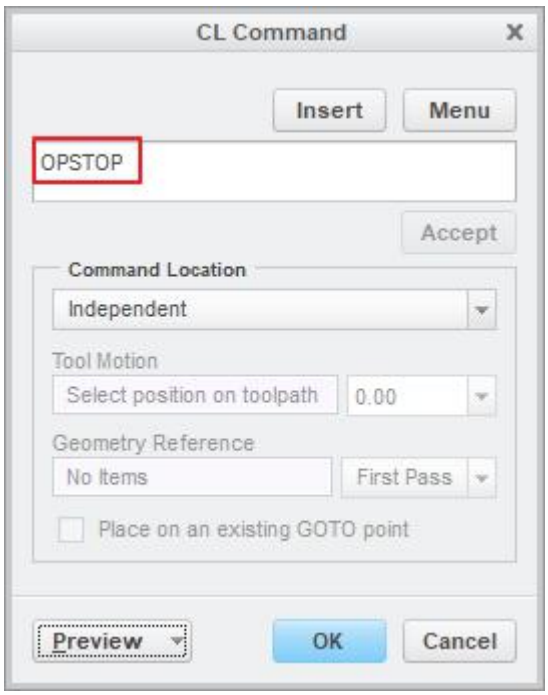
In the CL Command dialog box pick Menu icon.



In the “cmd edit” menu select **LOADTL-OPTSTOP > OPTSTOP.**



OPTSTOP command will appear in the dialog box as shown below.



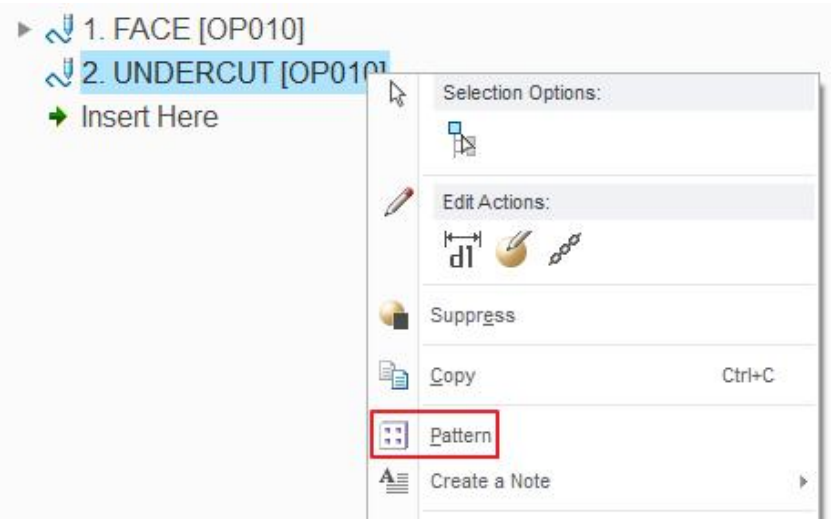
Select OK to exit the CL Command dialog box.

Now pick Properties tab and enter ***UNDERCUT*** as the name of the sequence.

Pick to save the changes and close the dashboard.

Now we will pattern the sequence to machine all the undercuts. We will use the Reference Pattern technique.

So right-click the NC Sequence in the model tree and pick **Pattern**

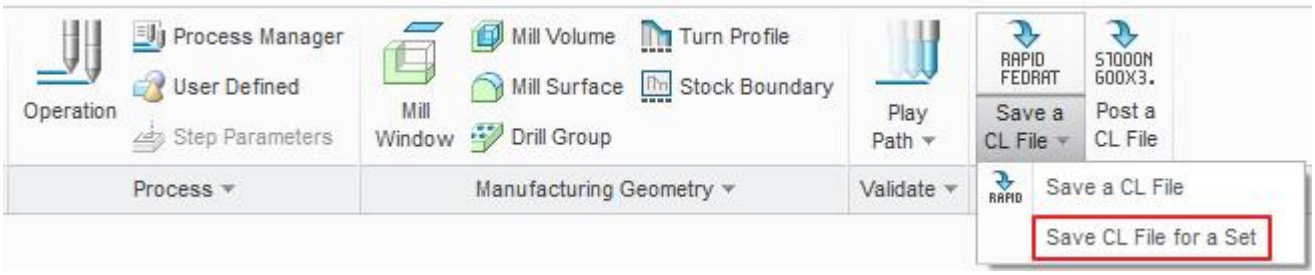


Notice that Reference is selected by default as shown below.

Pick icon or middle-click to complete the pattern.

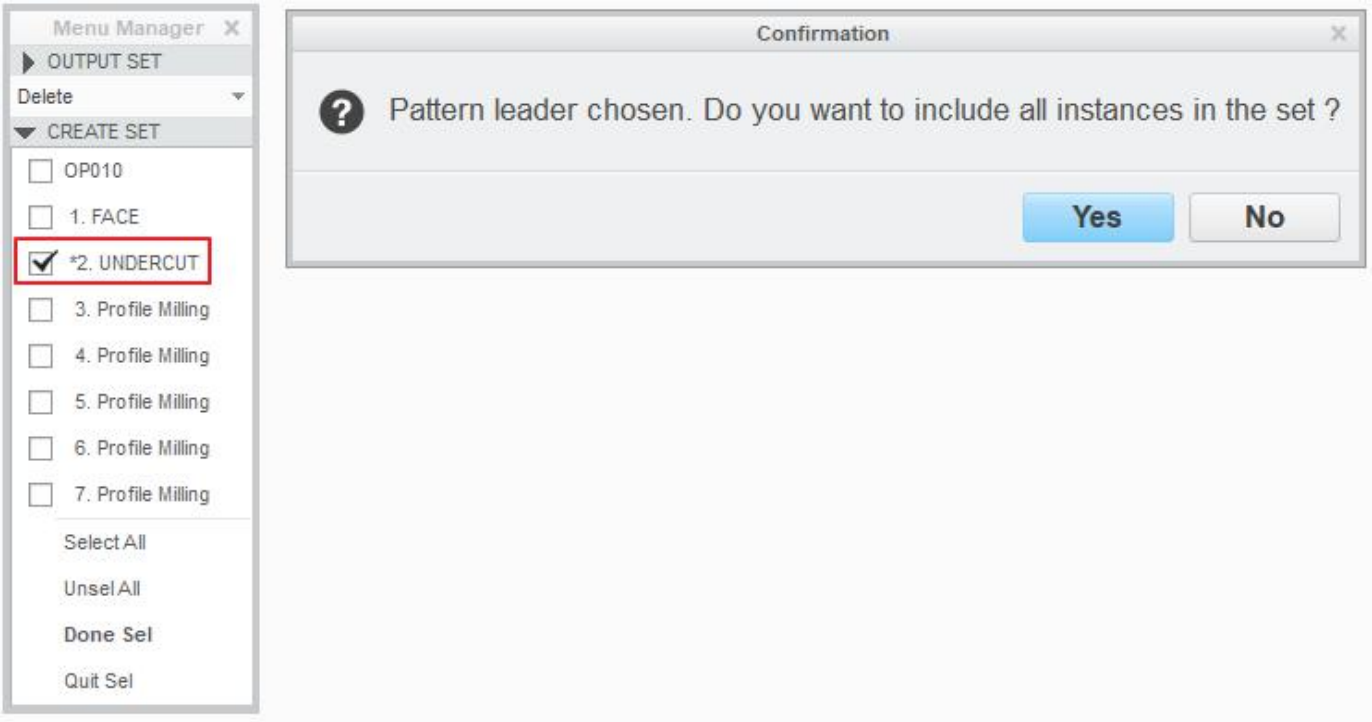
Now we will create the CL Data file for the patterned toolpath. To do this first we will create a set of patterned sequences.

Select Save CL File for a Set on the Manufacturing tab as shown below.



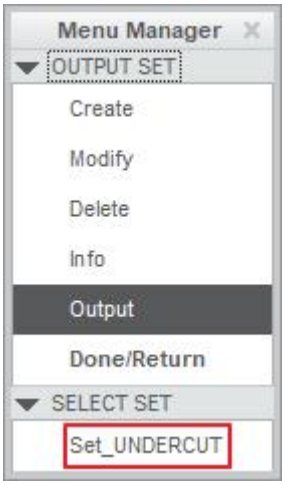
Pick **Create** in the OUTPUT SET menu and enter ***Set_UNDERCUT*** as the name of the set and hit Enter key.

In the CREATE SET menu, select the first instance of the pattern and system will ask to include all instance in the set. Pick Yes to confirm as shown below.

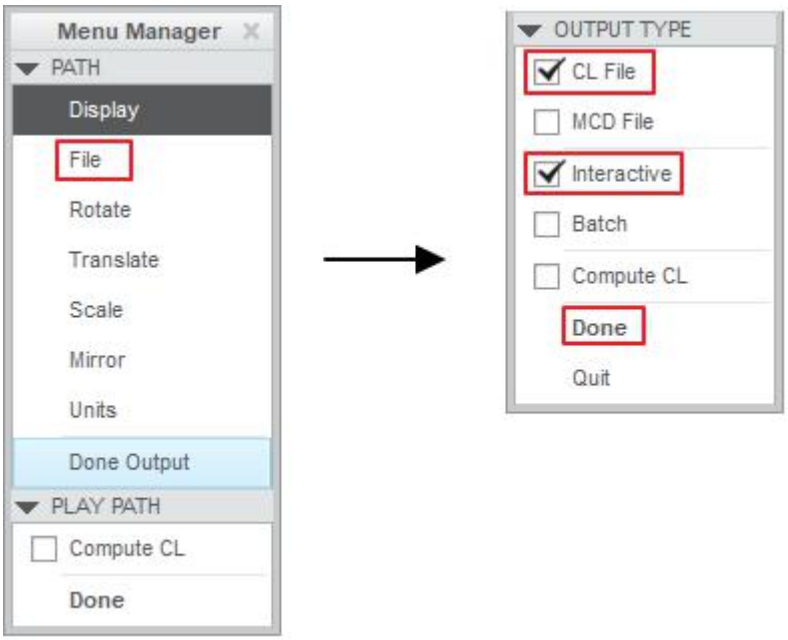


Select **Done Sel** in the CREATE SET menu.

Now select the **Set_UNERCUT** in the SELECT SET menu.

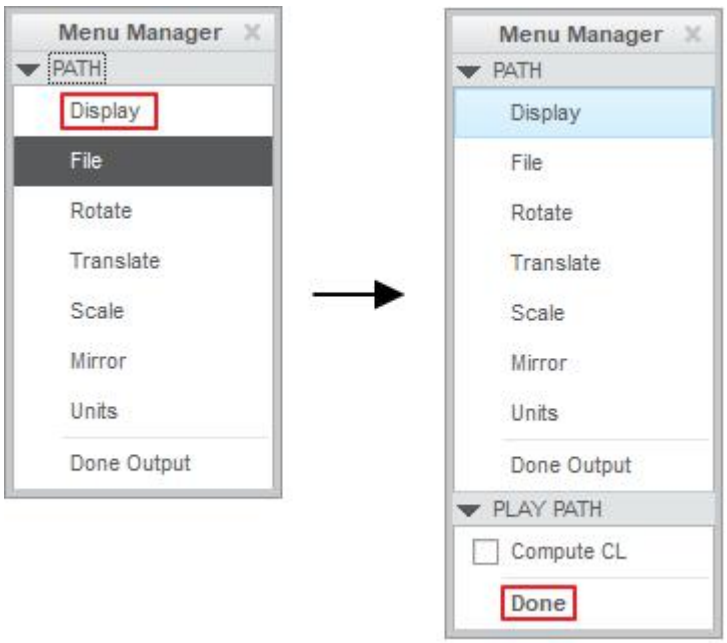


In the PATH menu pick **File** and then **Done**

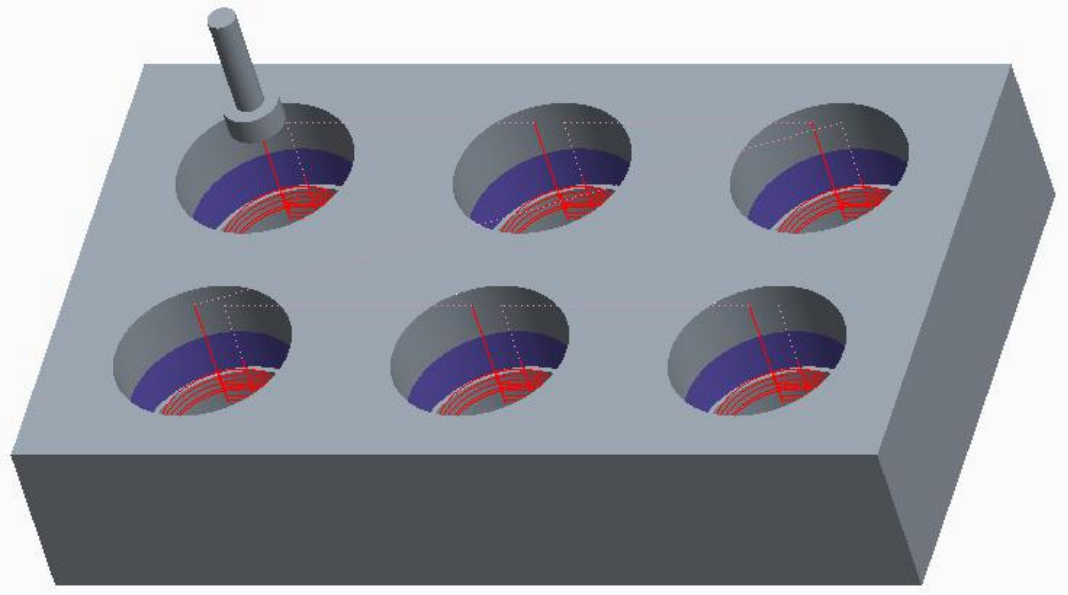


Pick OK in the Save a Copy dialog box. The file set_undercut.ncl.1 will be created.

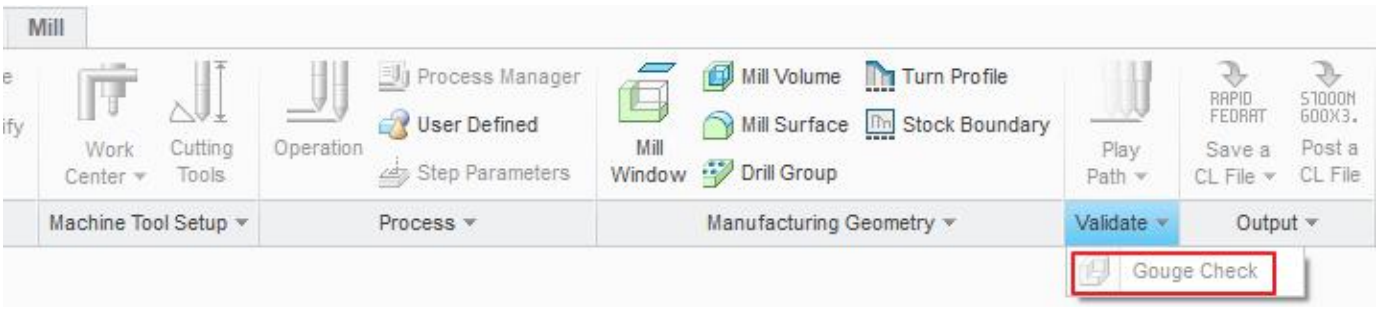
To display the toolpath on screen select **Display > Done** on the PATH menu.

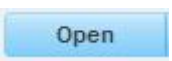


Play the toolpath using **PLAY PATH** dialog box**.** It should appear as shown below.



To perform gouge checking pick **Validate > Gouge Check** on the Manufacturing tab as shown below.

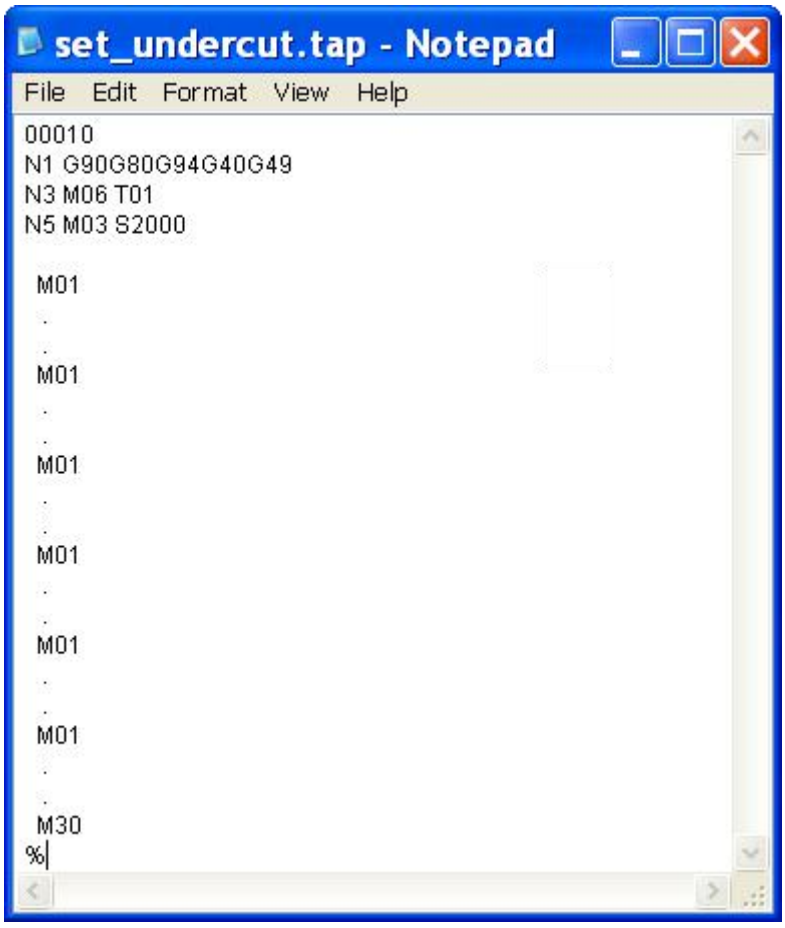


Select the newly created file set_undercut.ncl and pick 

Check the box **Add Ref Prts** and select **Done > Run.**

System will report that no gouges are detected.

Although we could insert an optional stop in Edit Parameters Dialog box, but it will only appear at the end of patterned MCD data i.e. once in the tape file. By inserting CL Command at the end of CL Data, the Optional Stop will appear at the end of each patterned instance i.e. 6 times in tape file for above case as shown in the figure below.



So the machine operator may have the option to stop the machine at the end of machining each bore and check the cutter condition or may perform some other task.

Select **File > Save** to save the manufacturing model.

EXERCISE 7

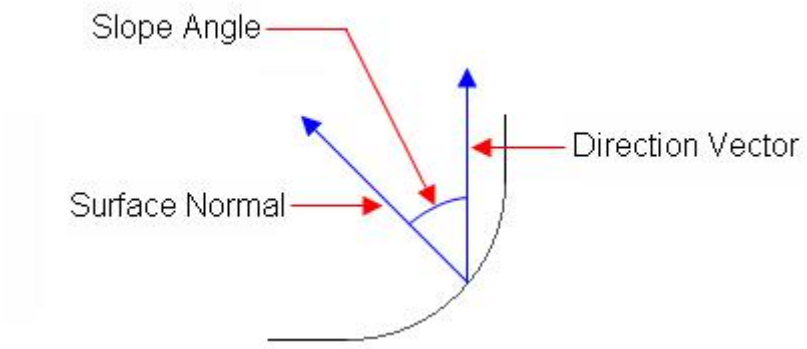
In this exercise first we will define a “Sloped” mill surface and then use the profile milling NC Sequence to machine it.

The topics covered in this exercise are as follows.

- Defining a Sloped Mill Surface
- Introduction to Check Surfaces functionality
- Setting the “Configuration file” option related to check surfaces.

While finish machining a part it is preferable to machine the steep walls with a constant Z type toolpath (as Profile and Volume Milling NC Sequence offer) and shallow surfaces with a little step over (as Surface Milling NC Sequence offers). Sloped Mill Surface is used to classify surfaces into Steep (near vertical) and Shallow (near horizontal) areas.

The criterion to classify surfaces into Steep and Shallow areas is the Slope Angle. At any given point, Slope angle is defined as the angle between Surface Normal and the Direction Vector as elaborated in the following figure.



Direction vector is always parallel to the Z-axis and Surface Normal depends upon the topology of the surface.

So a Flat surface has Slope angle of 0^0 and a Vertical surface has Slope Angle of 90^0.

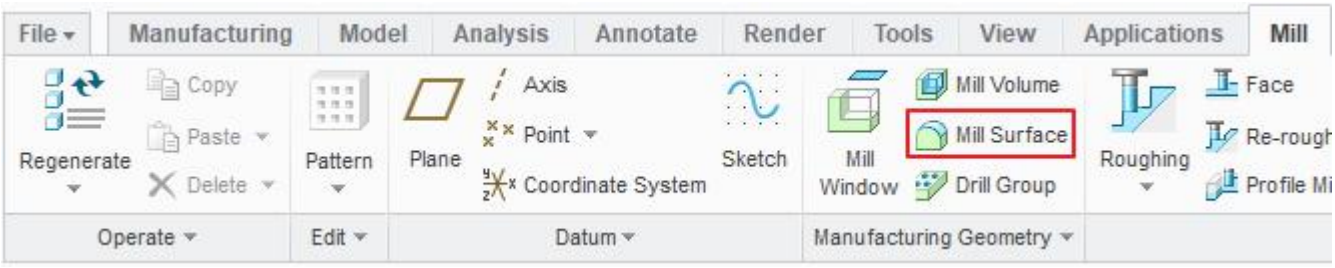
During the definition of Sloped Surface, we specify a Draft Angle. If a surface patch has Slope angle greater than this Draft Angle it is considered a Steep area and if its value is lesser then it is considered as Shallow area.

During the definition of Sloped Surface, we have the option either to keep the steep or shallow side of the selected surfaces. When we select **Keep Steep Side**, the system keeps only those portions of the selected surfaces where the Slope Angle is greater than the specified Draft Angle value. And when we select to **Keep Shallow Side**, the system keeps only those portions of the selected surfaces where the Slope Angle is less than the specified Draft Angle value.

Set the working directory to BASE_PUNCH and open the manufacturing model base_punch.asm.

First of all we will define a Mill Surface to be later referenced in the Profile sequence.

So pick on the Mill tab to invoke the Mill Surface Tool.



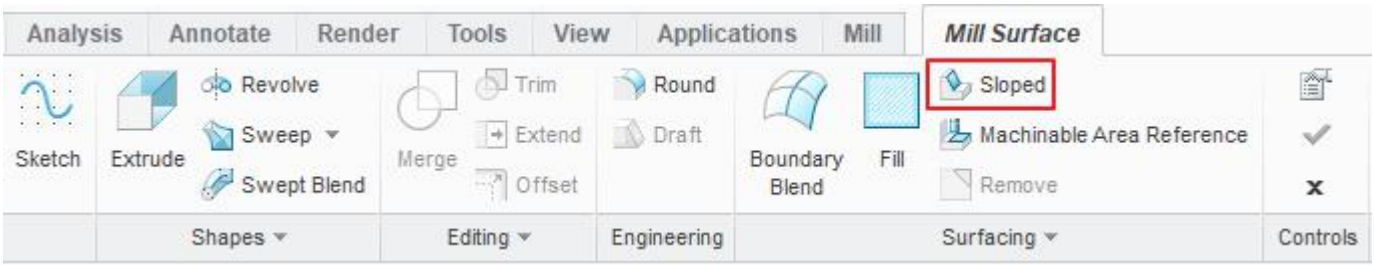
Pick icon to specify a name to this surface.

Enter ***MS_A*** as the name of the mill surface and pick

.

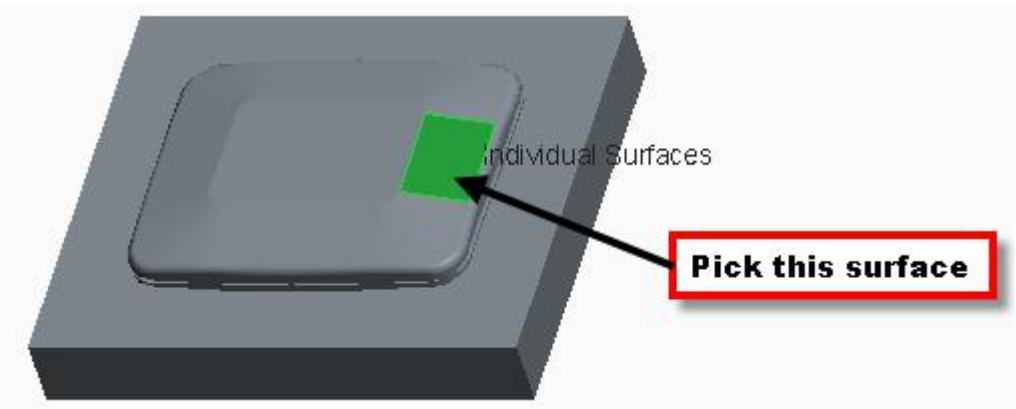
Pick

icon to create a Sloped mill surface.

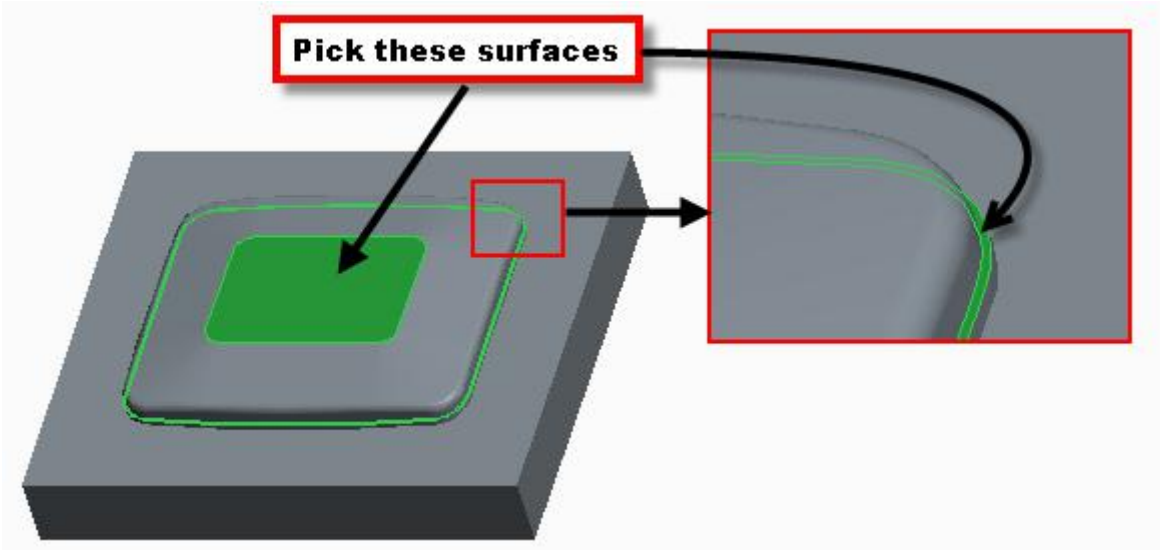


We will use the Seed and Boundary surface selection method to speed up the surface selection process.

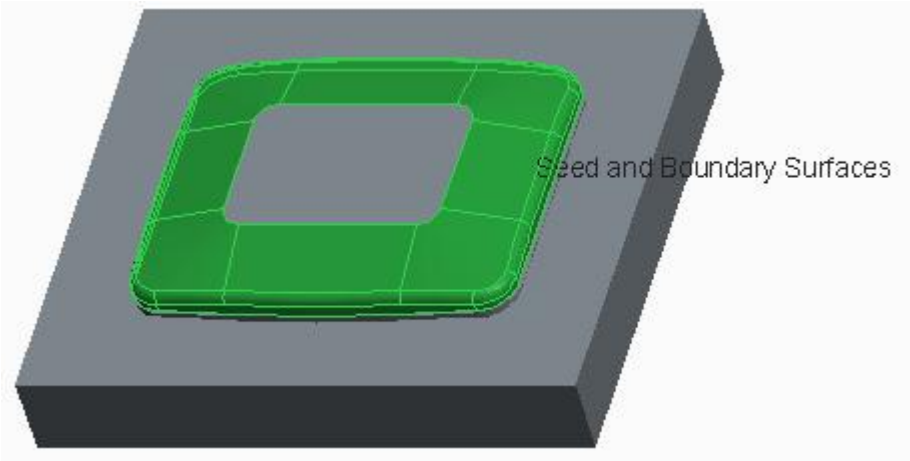
Select the model surface as shown in the figure below as the seed surface.



Now hold down the Shift key and select the following two highlighted surfaces as Boundary surfaces.

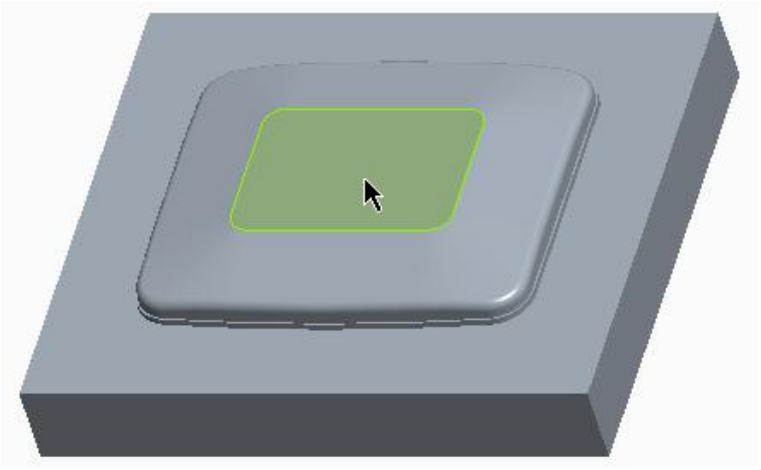


After relieving the Shift key the system will highlight the selected surfaces as shown below.

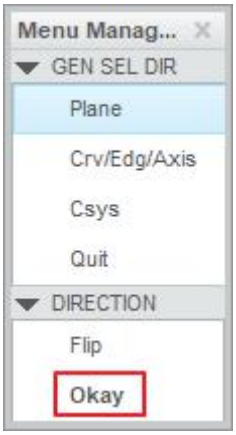


Select OK or middle-click.

Now select the top surface of the reference model as the direction reference. Make sure that arrow points upward. This direction is used for measuring the slope angle.



Then select **Okay.**



In the message area system will ask to enter the Draft angle value. This value is used to classify surfaces as steep or shallow.

Enter ***60*** and hit Enter key or middle-click.

Now select **Keep Steep Side > Done**

Select

to exit the dialog box.

Pick and it completes the definition of the mill surface.

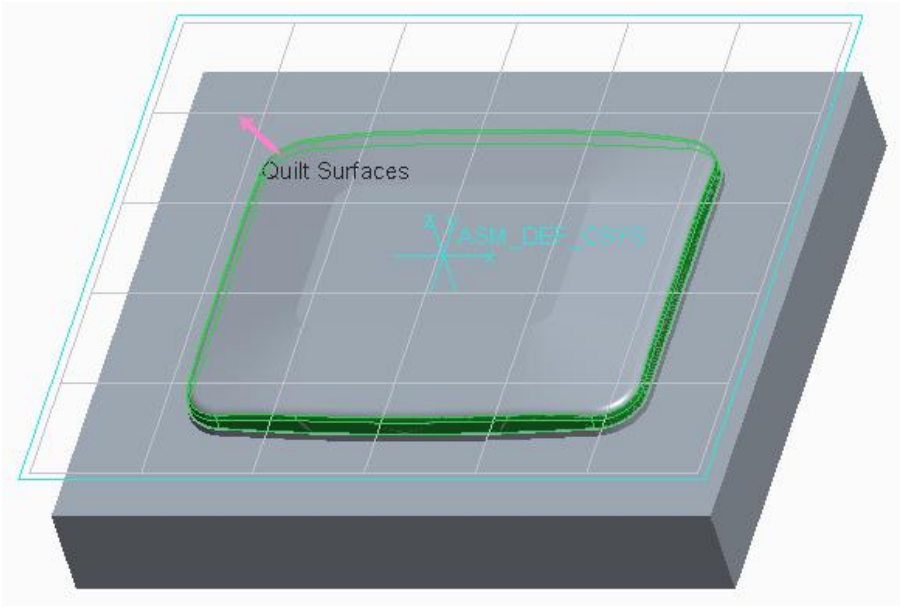
To define a new profile milling NC Sequence, select **Profile Milling** icon ( ) on Mill tab.

From the drop-down list in front of icon, pick **BEM10**

You can also pick the icon to access the Tools Setup dialog box and then select the tool.

Pick Reference tab and notice that the Machining Reference collector becomes active.

Pick the currently defined mill surface MS_A and make sure that arrow points outward as shown in figure below.

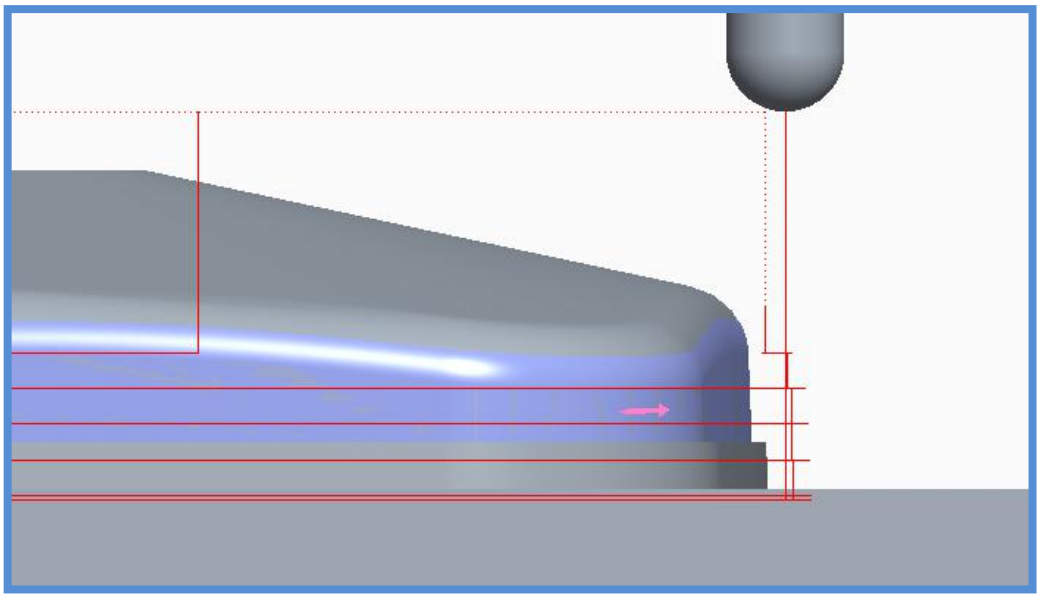


Now pick Parameters tab and then icon on the Parameters panel to open the Edit Parameters Dialog box . Enter the following values/relations and parameters.

CUT_FEED	1000
STEP_DEPTH	3
CLEAR_DIST	1
SPINDLE_SPEED	2000
COOLANT_OPTION	ON
CUT_ENTRY_EXT	LEAD_IN
CUT_EXIT_EXT	LEAD_OUT
LEAD_RADIUS	20
ENTRY_ANGLE	15
EXIT_ANGLE	15

Pick OK to apply the changes and exit the Edit Parameter Dialog box.

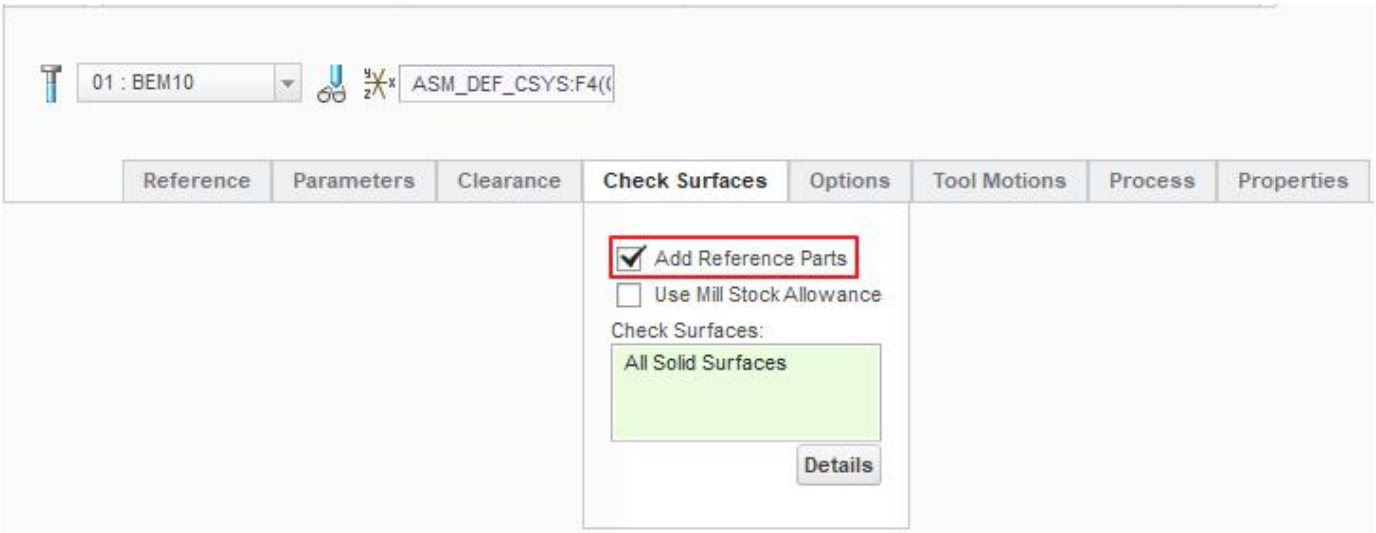
Play the toolpath using **PLAY PATH** dialog box and notice that tool is gouging the part as shown below. (You can also use the Gouge Check functionality i.e. by picking to confirm gouging)



To avoid the gouging we will declare the reference part to act as check surfaces.

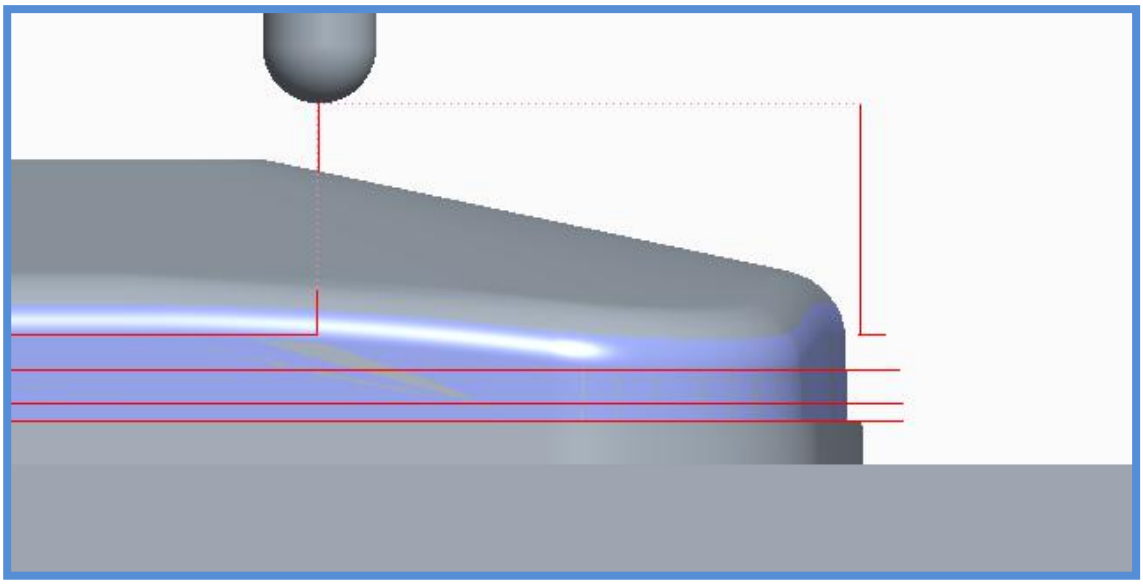
To avoid the gouging Creo NC provides the function of Check Surfaces. Although system automatically checks gouging against the surfaces selected as cut geometry (Mill Surface in this case) but it does not perform gouge checking against the reference part surfaces while computing the toolpath. So by declaring a surface to be Check Surface we explicitly tell the system to avoid gouging them while computing the toolpath.

Pick Check Surfaces on the dashboard and check the **Add Reference Parts** option as shown below.



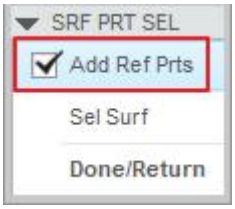
When Add Ref Prts (Add Reference Parts) option is checked, system uses all the surfaces of the reference model to check gouging against.

Play the toolpath using **PLAY PATH** dialog box**.** It should appear as shown below.

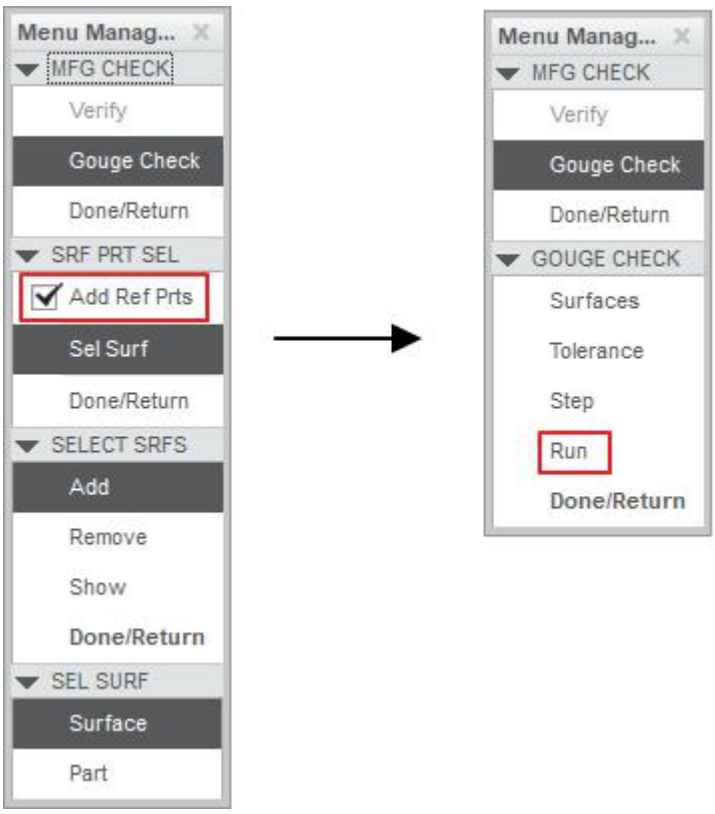


To perfrom gouge checking, pick icon on the dashboard as shown below.

Now the system will ask to select the surfaces for checking gouges. As we want to check the gouges for the surfaces of complete part so check the **Add Ref Prts** option**.**



Now the system will ask to select the surfaces for checking gouges. As **Add Ref Prts** is already checked so select **Done > Run.**



System will report that no gouges are detected.

As the toolpath looks satisfactory so change the following parameters for good surface finish.

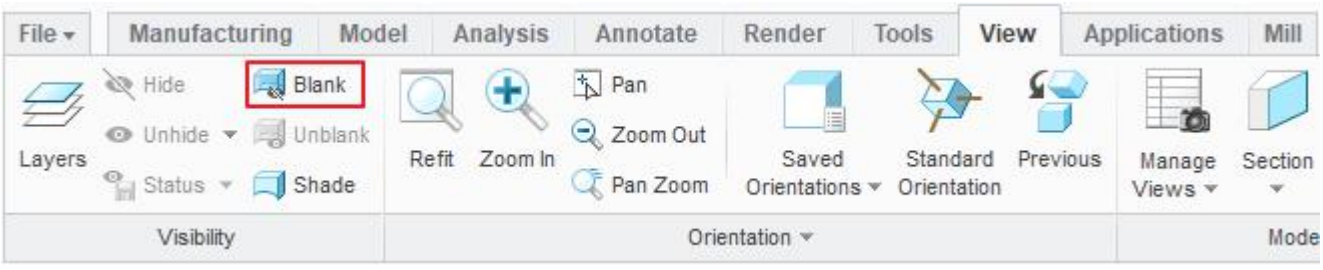
STEP_DEPTH	0.2

Now pick Properties tab and enter ***STEEP*** as the name of the sequence.

Pick to save the changes and close the dashboard.

Note: We can set the Configuration file option **chk_part_surfs_profpock** to Yes to force the system to add all the surfaces of the reference part as Check surfaces for profiling sequences.

To blank the Mill Surface MS_A, select Blank icon on the View tab.



Pick the MS_A quilt on the screen and then middle-click.

Select **File > Save** to save the manufacturing model.

7. SURFACE AND CUT LINE MILLING

Further material is not included in the free download. You can buy the complete tutorials at http://www.caeuniversity.com/buy. Thanks for your review of the training material.