# **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 tab on the Edit Parameters dialog box.

		- 🗆 X
File Edit Information	on Tools	
Parameters Basic All	Category: All categories	s 🔻
1000		d=
Parameter Name	STEEP	
RETRACT_FEED	12	
PLUNGE_FEED	28	11
STEP_DEPTH	3	
TOLERANCE	0.01	-
ê		

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.

	uence "STEEP" = 🗆 X	
File Edit Information	Tools	
Parameters Basic All Cate	Relations	1
1000	Sort Parameters	Default Sort
Parameter Name RETRACT_FEED PLUNGE FEED	Parameter columns Visibility	By Parameter Name By Parameter ID
STEP_DEPTH	Show All Value Columns	
TOLERANCE	0.01	4
	(ý	
2		

## **EXERCISE 1**

In this exercise we will define a profile milling NC Sequence to machine the 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**.

	V	iew M	lanage	E.		
Layers	8	Exploc	le	Orier	it	All
Simp Rep			Style >		Xs	ec
New	Edit	•	<b>@</b> 0	ptions	•	
Names	5					
Blank						
♦ Wiref	rame					
Default S	Style					
Default S Master S	Style Style					
Default S Master S	Style Style					
Default S Master S	Style Style					
Default S Master S	Style Style					
Default S Master S	ityle ityle					
Default S Master S	ityle ityle					

Select Close to exit the dialog box.

To define a new profile milling NC Sequence, select **Profile Milling** icon (

File •	Manufac	cturing	Model	Analysis	Annotat	te Render	Tools	View	Applications	Mill
<b>99</b>	e •	11 11		/. N	đ	🗐 Mill Volume	Ţ.	L Fac	e 🍒 Si rough 🛃 Fi	urface Milling nishing
Regenera	<sup>te</sup> × *	Pattern	Plane	¥× Sketch	Window	🖅 Drill Group	Roughing	Pro	file Milling 🗾 Co	orner Finishi
Oper	ate 🕶	Edit 🔻	[	Datum 🔻	Manufact	turing Geometry 🔻				Milling 🔻

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

	No. 2	ollo XZ				
	No tool		NC_ASM_DEF	CSYS:		
-						
		• • • • •				

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

Pick Reference tab and click inside the Machining References collector to activate it.

 Reference	Parameters	Clearance	Check Surfaces	Options	Tool Motions	Process	Propertie
Туре							
Surface	•						
Machining R	leferences:						
Select ite	ems						
-	Det	ails					
Scallop Sur	faces:						

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

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

### Chapter 6 **PROFILE MILLING**



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 tab and changing to "Entry/Exit Motions" category as shown below.

	Sequence "PROFILE"	>
File Edit Informatio	on Tools	
Parameters Basic All	Category: Entry/Exit I	Notions .
90		d=
Parameter Name	PROFILE	
HELICAL_DIAMETER	2	
START_MOTION	DIRECT	
END_MOTION	DIRECT	
CUT_ENTRY_EXT	LEAD_IN	
CUT_EXIT_EXT	LEAD_OUT	
		1
<u> </u>		

Pick 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.

PRT_CSYS_DEECSYS
------------------

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 and changing to "Entry/Exit Motions" category.

ENTRY_ANGLE	50
EXIT_ANGLE	50

Pick 0

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 the 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.



To complete the NC Sequence select

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.

File +	Manufac	cturing	Model	Analysis	Annotat	te Render	Tools	View A	pplications	Mill
Regenera	te La constante	Pattern	Plane	x Sketch	Mil	Mill Volum		E Face	h 🚺 Suri	face Milling
• Ope	rate 🕶	Edit 🕶	[	⊋ <b>X</b> * Datum ▼	Manufact	uring Geometry	p rodgining y▼	Profile N	Alling 🛂 Cori	Milling <del>•</del>

Pick icon to give a name to this volume.

Enter **MS\_A** as the name of the mill surface and pick

Pick b 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 or middle-click to proceed.

Sele	ct X
Select 1 or mo	re items.
ок	Cancel

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.

#### **Creo NC** - A Practical Approach

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 but we have entered a larger value just to make sure that we get smooth surface quality.

Now select the top surface of the reference model as the direction reference.



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.

SURFACE: I	Machinable	Area )
Element	Info	
Machine Srfs	s Defined	1
Check Srfs	Defined	
Tool Radius	Defined	
Tool Axis	Defined	
Srf Option	Defined	
Define	Refs	Info
ОК	Cancel	Preview

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 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.

Creo	1.0	
------	-----	--

Analy	sis Annotate Rend	er Tools Vie	w Applica	ations Mill	Mill Surface	
Sketch	Extrude	Merge	→ Round → Draft	Boundary Fill Blend	Sloped Machinable Area Reference Remove	<ul> <li>✓</li> <li>×</li> </ul>
	Shapes 🔻	Editing 🔻	Engineering		Surfacing 🔻	Controls

Pick

Define... tab in the Placement panel.

154.99	• % 💋		II O 🕅 🏹 85 🖌 🗙
Placement Option	s Properties	Intersect	
Sketch Select 1 item	Defi	ne	

Pick **ASM\_RIGHT** as the Sketch Plane **ASM\_TOP** datum as Right Orientation Reference.

	Sketch		х
Placement			
Sketch Plane			
Plane A	SM_RIGHT:F1(	Use Pre	vious
Sketch Orient	ation		
Sketch viev	direction Flip		
Reference	ASM_TOP:F2	(DATUM PLA	NE)
Orientation	Right 💌		

Now sketch a rectangle after choosing the suitable reference as shown below.



Exact dimensioning is not important; purpose is to cover the 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.

Placement	Options Properties Intersect	
	Depth	
	Side 1 🚽 E Through All 💌	
	Side 2 🚽 E Through All 🔻	
	Capped ends	
	Add Taper	

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

#### **Creo NC** - A Practical Approach



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.



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.

Analys	sis /	Annotate	Render	Tools	View	Applica	ations	Mill	Mill Surface	
Sketch	Extrud	e 🖓 Revolu Swee 🥜 Swep	ve p▼ t Blend		rim 📑 xtend 🔂	Round Draft	Boundary Blend	/ Fill	Sloped	×
		Shapes 🔻		Editing 🔻	Er	igineering			Surfacing 🕶	Controls

To define a new profile milling NC Sequence, select **Profile Milling** icon (

Pick the <sup>II</sup> icon and Tools Setup dialog box will pop up.

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

Pick Reference tab and click inside the Machining References collector to activate it then 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 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. It should appear as shown below.



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

	0.0
SIEP_DEPTH	0.2

Remember never to use WALL\_SCALLOP\_HGT parameter for controlling the surface finish, while machining with a flat end mill.

To complete the NC Sequence select

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 the manufacturing model cover\_insert.asm

To define a new profile milling NC Sequence, select **Profile Milling** icon (

From the drop-down list in front of <sup>1</sup> icon, pick **FEM25** 



Alternatively you can also pick the <sup>II</sup> icon to access the Tools Setup dialog box and then select or define a tool.

Pick Reference tab and click inside the Machining References collector to activate it.

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 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 a single pass only as shown below.



To complete the NC Sequence select

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 (

From the drop-down list in front of <sup>1</sup> icon, pick **FEM10** 

Pick Reference tab and click inside the Machining References collector to activate it.

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

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

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.

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

It is so because lead radius value we have specified 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 diameter 12mm. 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.

Reference	Parameters	Clearance	Check Surfaces	Options	Tool Motions	Process	Propertie
				Cutting 7	Cool A doptor		
				Cutting	oorAdapter		<b>1</b>
				Approad	ch Axis:	1	
				Selec	ct 1 item		
				Firs	t Slice Only	100	
				Exit Axis	ć.		

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.

	C C
Approach Axis:	
AA_1:F10(DATUM AXIS)	
First Slice Only	
Exit Axis:	
AA_1:F10(DATUM AXIS)	

Pick Parameters tab and set the following parameters

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).

### Chapter 6 **PROFILE MILLING**



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 the 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.

Edit Parameter	s of Sequence "	FINISH"	- 🗆 X
File Edit Information	Tools		
Parameters Basic All	Category: C	ut Depth and Allowances	•
-			d=
Parameter Name	FINISH		
MIN_STEP_DEPTH	24		
TOLERANCE	0.01		=
NUM_PROF_PASSES	2		
PROF_INCREMENT	0.3		
PROF_STOCK_ALLOW	0		•
	No image availab	le	

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 tool first machines the walls from top to bottom of hole at the offset of 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
------------	-----

To complete the NC Sequence select

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

NC Sequences Mirr	roring X
1: FINISH, Operation	n: OP010
Mirror Plane	Select
Cutting Conditions 💿 Ke	ep 🔿 Mirro
ОК	Cancel

Select the datum ASM\_RIGHT as mirroring plane.



Notice that the option "Keep" is selected 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.

### Chapter 6 **PROFILE MILLING**



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.

ASM_FRONT	
X ACS0	
ESETRO	
ADTM4	
MACH01	
AA_1	
AA_1	
AA_1 U 1. FINISH [OP010] 2. MIRROR_FINISH	
AA_1 1. FINISH [OP010] 2. MIRROR_FINISH Insert Here	H [QP010] Delete
<ul> <li>✓ AA_1</li> <li>↓ 1. FINISH [OP010]</li> <li>↓ 2. MIRROR_FINISH</li> <li>◆ Insert Here</li> </ul>	T [OP010] Delete Group
AA_1 U. FINISH [OP010] U. MIRROR_FINISH Insert Here	Delete Group Suppress
<ul> <li>✓ AA_1</li> <li>↓ 1. FINISH [OP010]</li> <li>↓ 2. MIRROR_FINISH</li> <li>◆ Insert Here</li> </ul>	T [OPO10] Delete Group Suppress Edit

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 (

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 <sup>60</sup> in the diloag box and the detailed sketch of the tool will appear as shown below.



Pick Close this window.

Select **OK** to complete the definition of the tool.

Pick Reference tab and click inside the Machining References collector to activate it.

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.





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.

Reference	Parameters	Clearance	Check Surfaces	Options	Tool Motions	Process	Properties
Type Surface Machining R Loop Surfa	eferences: ces						
Scallop Sur	Det faces: e to add item	ails		E		_DEF_CS	
	Deta	ails		4	_/ _/		

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 defined satisfactorily, we will change it to suit the surface finish required.

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

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

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 **b** icon on the PLAY PATH dialog box.

	PLAY P	ATH	×
File View NO	CL File		
CL Data			
-			
44		bb-	alet a
Tool Clearance Display Speed	)	Position	n Cutting Tool 📐
Slow	0		Fast
	Cle	2	
	CIC	30	

Now pick on the toolpath as shown below.

	Pick here on toolpath

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.

#### 🕅 No gouges were 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
----------------

To complete the NC Sequence select

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 (

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.

Type:		<b>+</b> ∅ -	
KEY CUTTER	•	T	
Material:	Øs		+
Units: Millimeter	8		40
Number of Flutes:	1	<u> </u>	Ø 16

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 this window.

Select

to complete the definition of the tool.

Pick Reference tab and click inside the Machining References collector to activate it.

#### **Creo NC** - A Practical Approach

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

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

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.

ം∲ ¥	ASM_DEF_CS	SYS:F4(C					ප්	<u>\</u>	11	~	×
imeters	Clearance	Check Surfaces	Options	Tool Motions	Process	Properties	1				
								Ø			

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.

👻 SRF PRT SEL	8
🗹 Add Ref Prts	]
Sel Surf	
Done/Return	1

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

Select Done > Run.

#### **Creo NC** - A Practical Approach



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

Gouges were detected.
A Tool gouges into part by -1.2322 .

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 LALL 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 leadout motion.



N\_L\_S = NORMAL\_LEAD\_STEP

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

### Chapter 6 **PROFILE MILLING**



To perform gouge checking pick **I** 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.

o se	elect	Tool	Motions	tab ar	nd pick	on CL Comm	hand		icon		
	Clear	ance	Check S	urfaces	Options	Tool Motions	Process	Proper	ties		
						<start of="" td="" tool<=""><td>path&gt;</td><td></td><td></td><td>Profile Mill Cut</td><td></td></start>	path>			Profile Mill Cut	
					1. Full Depth Cut id 1004				Tangent Approach	Ŧ	
						2. Auto Plunge id 1006				Tangent Exit	-
						<ul> <li>Insert Here</li> </ul>			4	Goto Point	
										CL Command	
						Entit	Deres	motoro			
						COltin	Para	meters.,	*C		

In the CL Command dialog box pick Menu icon.

Insert						
	Accept					
0.00	*					
First P	ass 🔻					
	sert					

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



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

### Chapter 6 **PROFILE MILLING**

CL Comm	and		X
	Insert	Me	enu
OPSTOP			
· · ·			
		Acc	ept
Command Location			
Independent			Ŧ
Tool Motion			
Select position on toolpa	ath 0.00		Ψ.
Geometry Reference			
No Items	First	t Pass	Ŧ
Place on an existing	GOTO po	oint	
	Art		
	20		X

Select ✓ to exit the CL Command dialog box.

To complete the NC Sequence select



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

Insert Here	Delete
	Group
	Suppress
	Rename
	Edit
	Edit Definition
	Edit References
	Edit Step Parameters
	Edit Step Tool
	Pattern
	Play Path
	Save Step Parameters

Notice that Reference is selected by default as shown below.

Referen	ce 🔻 Referen	ce type Group 🔻					II 🛛 🕅 80 🔽 🗙
	Dimensions	Table Dimensions	References	Tables	Options	Properties	

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.

So 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 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** and then <sup>12</sup> icon on the Manufacturing tab as shown below.

Operation	Vser Defined	Mill Window	Mill Volume Im Turn Profile Mill Surface Im Stock Boundary Drill Group	Play Path ▼	RAPID FEDRAT Save a CL File ▼	S7000N G00X3. Post a CL File
	Process 🕶		Manufacturing Geometry 🔻	Validate 🔻	Outpu	ut 🕶
				🗐 Gouge	Check	
the newl	v created file set	under	cut nol and pick			

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.

set_undercut.tap - Notepad	
File Edit Format View Help	
00010 N1 G90G80G94G40G49 N3 M06 T01 N5 M03 S2000	~
M01	
9. (c)	
M01	ł
MO1	
100-10 10	
МП1	
3	
M01	
M01	
M20	
%	~
<u>×</u>	2

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<sup>0</sup> and a Vertical surface has Slope Angle of 90<sup>0</sup>.

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 Mfg Geometry toolbar or pick **Mill Surface** icon on the Mill tab to invoke the Mill Surface Tool.

ill	ations	Applic	View	Tools	Render	Annotate	Analysis	Model	turing	Manufac	File •
Milling	Karfac ∭r Finishir	e ough	L Fac	<b>J</b>	Mill Volume Mill Surface		Sketch	Plane ×	Pattern	e	Regenerate
inishi ng 🔻	Corner M	ile Milling	gli Prof	Roughing	Drill Group	Window 🎒	¥× atum ▼	Da	▼ Edit ▼	× *	• Operati

Pick icon to give a name to this surface.

Enter **MS\_A** as the name of the mill surface and pick

Analys	sis A	nnotate	Render	To	ols	View	Applic	ations	Mill	Mill Surface	
Sketch	Extrude	ola Revol 🟠 Swee 🖉 Swep	lve ⊧p ▼ I ot Blend	(Jerge	Fr Ex Of	im ttend ffset	🔊 Round 🔊 Draft	Boundar Blend	y Fill	Sloped Machinable Area Reference Remove	✓
		Shapes 🔻		Edi	ting 🕶		Engineering			Surfacing 🔻	Controls

Pick <sup>b</sup> 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.



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.

### Chapter 6 **PROFILE MILLING**





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  $\bigcirc K \bigcirc$  to exit the dialog box.

Pick  $\checkmark$  and it completes the definition of the mill surface.

To define a new profile milling NC Sequence, select **Profile Milling** icon (

From the drop-down list in front of <sup>1</sup> icon, pick **BEM10** 

01 -	BEM10	- Alo W	ACH DEE CO	VEEAIC			
01.	DENTO	108 34	ASM_DEF_C3	51.5.F4(C			
-							
						-	-

You can also pick the <sup>1</sup> icon to access the Tools Setup dialog box and then select or define a tool.

Pick Reference tab and click inside the Machining References collector to activate it.

Pick the currently defined mill surface MS\_A.



Now make sure that arrow points outward as shown in figure below and select  $\checkmark$ .



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

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

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 it to confirm gouging)

#### **Creo NC** - A Practical Approach



To avoid the gouging Creo NC provides the function of Check Surfaces. Although system checks gouging against the surfaces selected as cut geometry (Mill Surface in this case) automatically 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 are explicitly telling 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.

Reference	Parameters	Clearance	Check Surfaces	Ontions	Tool Motions	Process	Properties
				an Darta			
			Use Mill Stock	ce Parts k Allowance			
			Check Surfaces:				
			All Solid Surface	S			

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.

### Chapter 6 **PROFILE MILLING**



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

o∲r <del>}×</del>	ASM_DEF_CS	SYS:F4(C					<u>60</u>	-	 ~	×
imeters	Clearance	Check Surfaces	Options	Tool Motions	Process	Properties				
								Ø		

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.

🕅 No gouges were detected.

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

STEP_DEPTH 0	).2
--------------	-----

To complete the NC Sequence select

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.

To blank the Mill Surface MS\_A, select Real icon on the View tab.

File *	Manufacturing	Model	Analysis	Annotate	Render	Tools	View	Applications	Mill
Layers	Image: Arrow of the second	ank C	fit Zoom In	Pan ⊇ Zoom Out ♀ Pan Zoom	Named Views •	Standa Standa	urd Orientat us nt	tion Manage Views ▼	Appearanc Gallery •
	Visibility			Orie	ntation 🔻			M	lodel Display

Pick the MS\_A quilt on the screen and then middle-click.

Select File > Save to save the manufacturing model.

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.