Table of Contents

Opening Solid Model File	2-4
Create the Contours	5
Step 1 - Extract Contours from Model	5-9
Step 2 - Set Contour Heights	9-23
Create the Toolpaths	24
Step 1 - Process plan	24
Step 2 - Set start point	24-26
Step 3 - Prepare for roughing	26-29
Step 4 - Rough outside profile	29-37
Step 5 - Rough counterbore	37-40
Step 6 - Rough bores	40-43
Step 7 - Rough inside	43-46
Step 8 - Finish outside profile	46-53
Step 9 - Finish bores	53-59
Step 10 - Finish counterbore	59-62
Step 11 - Finish inside	62-65
Step 12 - Machine radial slots	65-72
Step 13 - Spot drill holes	72-77
Step 14 - Drill holes	77-82
Step 15 - Drill holes	82-86
Step 16 - Create Material Stock	86-89
Step 17 - Solid Simulation	89-91
Step 18 - Output Nc code	91-92
Gear Housing Drawing	93
Index	94-95

1.

Opening Solid Model File

Download the accompanying STEP file - Gear Housing.STEP to a location of your choice.

Menu: File -> Open

Toolbar button: 旑

Shortcut: Ctrl + O

Choose the File type to open - STEP Files, as shown below:

2	Ope	en		×
Look in:	Public Documents	G 🤌 📂 🖽 -		
C	Name	Date modified	Туре	Size
Recent places	Gear Housing.STEP	19/10/2013 12:38	STEP File	254 KB
Desktop				
Libraries				
Market Contract Contr				
Network				
	File <u>n</u> ame:		v	<u>O</u> pen
	Files of type: STEP Files (*.step; *.stp)		~	Cancel
				<u>H</u> elp

Navigate to the downloaded file, then select the *Gear Housing*.*STEP* file and click the Open button.

SharpCam will read the units from the STEP file and set the units for the opened file as required.

Units

The units stored in any STEP file may not be correct and are arbitrary. It is always best to check the unit settings for the opened file:

Choose the Options... command from the Tools menu:

Too	Window Help	
	Snaps Fool Bars Fool Bars	
	Machine Developer Configure Machine Definition	
	RS232 Settings	
	Options	

Select the Active Part Settings node and make sure that the correct units are selected and click the OK button:

Options				
General Colours Colours Colours Colours Colours Concernal Concernal Concernal Concernation Defaults Concernation Defaults Concernation	 7 Arc Circle Smoothness 3 Model Image Quality Tools Folder Location: C: \Users _ \Documents \SharpCam Tools \Metric Machine Definition: C: \Users _ Documents \SharpCam Machine Definitions \Fanuc \Fanuc Part Units Inch Metric 			
OK Cancel				

A confirmation message box will be shown if the units are changed, click OK to accept.

Opening other types of CAD files

Dxf/dwg/STL file units cannot be read, the opened file will adopt the units set for a new Part, see below. Be sure to change the units if required, see above.

Choose the Options... command from the Tools menu:



Select the New Part Defaults node. The Part Units group indicates the units that other CAD files will adopt by default:

	Options
General Appearance Colours Colours Cons Cons Cons Cons Cons Cons Cons Con	Tools Folder Location: C:\Users\'pocuments\SharpCam Tools\Metric Machine Definition: C:\Users\'pocuments\SharpCam Machine Definitions\Fanuc\Fanuc Default Viewing Area 200 Top -200 Left 200 -200 Bottom
	OK Cancel

Other Settings

For this tutorial it is assumed that the following buttons are depressed - as indicated by the orange appearance:



Create the Contours

Step 1 - Extract Contours from Model

Rather than drawing the Contours, we will extract them from the Solid Model

Extract Contours using Extract Contours from Face

 Set the model view to isometric: Menu: View -> Standard View -> Isometric View Toolbar button:



 Extract Contours from top face: Menu: View -> Solids -> Extract Contours from Face Toolbar button: Highlight top face:



Left click mouse to extract Contours from the top face:



• Extract Contours from counter bore face - highlight face:



Left click mouse to extract Contours from the counter bore face:



The Contours have now been extracted.

Step 2 - Set Contour Heights

Set Contour Heights for machining depths

• Deselect all Contours - Press the Escape Key:



• Highlight outer Contour on top face, left click to select, repeat for four circles on top face also:



Set the Contour Heights: Menu: Modify -> Set Contour Height Toolbar button: 𝖓□¹

The default value for the Top of the selected Contours is zero, this is correct, we want Z0 on the top of the part. Click OK to accept:

	Тор
	E.
ſ	Bottom
	0
	ОК

We could fill in the Bottom value, but it is easier to pick the value from the model. Highlight a point on the bottom edge:



Left click to enter value:

Co	ntour Height
	Top 0
	ОК
	Bottom
	-25 OK

The Contour heights will be set at the same time:



• Press the escape key to deselect the Contours and exit the Set Contour Height command. Highlight and select the inner pocket Contour:



Choose the Set the Contour Heights command. Click OK to accept the default Top value:



Pick the Bottom value from the model. Highlight a point on the bottom edge of the pocket:



Left click to enter value:

Co	ntour Height
	Top 0 OK
	Bottom -20
l	

The Contour heights will be set at the same time:



The bottom of the Contour is hidden by the radius in the corner, you can set the view to Wire Frame to see the bottom of the Contour: Menu: View -> Wire Frame View Toolbar button:

Put the view back to Shaded View Menu: View -> Shaded View Toolbar button:

• Next delete the unwanted Contour in the bore. Press the escape key to deselect the Contours. Highlight and select the unwanted Contour:



Choose the Delete command to remove unwanted Contour: Menu: Edit -> Delete Toolbar button: X



• Highlight bore Contour on top face, left click to select:



Choose the Set the Contour Heights command Click OK to accept the default Top value:



Pick the Bottom value from the model. Highlight a point on the bottom edge of the bore:



Left click to enter value:

Co	ntour Height
	Тор
	0
	ОК
	Bottom
	-15
	OK

The Contour heights will be set at the same time:



 Press the escape key to deselect the bore Contour. Highlight and select the slots and holes in the bore: One of the slots is obscured, choose the Top View command to assist in selection. Menu: View -> Top View Toolbar button:



Switch back to Isometric View Menu: View -> Standard View -> Isometric View Toolbar button:

Choose the Set the Contour Heights command. Click OK to accept the default Top value: Menu: Modify -> Set Contour Height Toolbar button: $\rho^{\Box t}$

Co	ntour Height
ſ	Ten
	тор
	15
	OK
	Bottom
	-15
	OK

Pick the Bottom value from the model. Highlight a point on the bottom edge of the outside:



Left click to enter value:

ntour Height
Тор
-15
ОК
Bottom
-25
ОК

The Contour heights will be set at the same time:



Create the Toolpaths

Step 1 - Process plan

The Gear Housing is to be made from an aluminium rectangular billet 295mm x 225mm x 30mm. The billet is to be held in a machine vice, leaving 27mm protruding above the top of the jaws. The Gear Housing is machined completely, leaving an operation to remove the excess material that it was held with. This operation will not be covered during the tutorial. For the purpose of this tutorial assume that the top of the billet is at Z0 and does not require facing.

Operation Description	Operation Type	Tool	Comments
Rough outside profile	Pocket	T01, 20mm End Mill	0.5 finish allowance
Rough Ø120 counterbore	Pocket	T01, 20mm End Mill	0.5 finish allowance
Rough Ø50 and Ø70 bores	Pocket	T01, 20mm End Mill	0.5 finish allowance
Rough inside pocket	Pocket	T02, 12mm End Mill	0.5 finish allowance
Finish outside profile	Profile	T03, 20mm End Mill	
Finish Ø50 and Ø70 bores	Profile	T03, 20mm End Mill	
Finish Ø120 counterbore	Pocket	T03, 20mm End Mill	
Finish inside pocket	Pocket	T04, 12mm End Mill (2mm Corner Rad)	
Machine radial slots	Profile	T05, 8mm Slot Drill	
Spot Drill holes	Drilling	T06, 12mm Spot Drill	
Drill Ø8.5 holes	Drilling	T07, 8.5mm Drill	
Drill Ø10 holes	Drilling	T08, 10mm Drill	

Before starting, a process plan is required:

Step 2 - Set start point

Set start point on closed Contours using Set Start Point

Before creating any operations, it is recommended that the start point on closed contours be set first.

Choose the Top View command. Menu: View -> Top View Toolbar button: 🗗

Set the start point:

Menu: Modify -> Set Start Point

Toolbar button: 🕒

Choose the command and move the cursor near to the end of the line on the left:



Left mouse click to set the start point:



Step 3 - Prepare for roughing

Prepare for Roughing Outside Profile

In order to rough the outside using a pocketing operation, a rectangle needs to be created that will contain the toolpaths.

Choose the Rectangle command:

Menu: Draw -> Rectangle

Toolbar button: 🗖

Create a rectangle with a start point value of X-87, Y-107 and an end point value of X242, Y152:

Start (Corner Point
x	-87
Y	-107
	ОК
	ana Daiat
End C	
End Co	orner Point
End Co	242
End Co X	242 152

The Part should now look like this:



By default the Contour that forms the rectangle has a Z Top and Z bottom of zero. The outside profile and the rectangle will be used for the pocketing operation, they both must have the same Z Top and Z bottom.

Press the escape key to exit Rectangle mode.

Set the model view to isometric: Menu: View -> Standard View -> Isometric View Toolbar button: 🗇

Highlight and left click the rectangle to select:



Set the Contour Heights: Menu: Modify -> Set Contour Height

Toolbar button: \square^{\ddagger}

The default value for the Top of the selected rectangle is zero, this is correct. Click OK to accept:



Enter the value Z-25 in the Z Bottom and click OK

Contour Height
Тор 0 ОК
Bottom
-25
ОК

The contour height has been set:



Step 4 - Rough outside profile

Rough Outside Profile using Pocket

Before choosing the pocket command, first select the cutting tool to be used for the operation. The Tool Manager is used to select a tool:

Menu: Machine-> Tool Manger

Toolbar button: 🜌

Before selecting a tool, first check that the Tool Manager is pointing to the correct folder.

The folder is indicated by the Location of Tools box:

Tool Manager			×			
Name		🙆 Туре	Flat Mill 🔹			
Number	1	Diameter	0			
Direction	Clockwise 👻	Offset	1			
Pitch	0	Tool Length	0			
Corner Rad	0	Flute Length	0			
Diameter at Tip	0	Included Angle	0			
	Create Tool	Save Changes	Delete			
	Select Tool	Rename	Close			
Location of Tools: C:\Users\\Documents\SharpCam Tools\Metric						
TE	T H T T T T T T T					

As this is a metric Part the *Metric* tools folder, that was installed at the same time as SharpCam, should be used. This folder is located in Documents (Vista)/Documents Library (Windows 7, 8, 8.1), inside a folder called *SharpCam Tools*. If the location is wrong, click the browse button and navigate to 'SharpCam Tools' folder and select the *Metric* folder and click OK:

Browse For Folder	×
Please select where your Tools folder is located for the active Part	
SharpCam Machine Definitions	*
🔺 🍌 SharpCam Tools	
Inch Metric Metric Marginalis Marginali	4
4	
Make New Folder OK Cancel	

Choose the command to display the Tool Manager, click on 'T01, 20mm End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it.

All the tools required for this tutorial were installed when SharpCam was installed for the first time.

Tool Manager			×
Name	T01, 20mm End Mill	🙆 Туре	Flat Mill 👻
Number	1	Diameter	20
Direction	Clockwise 👻	Offset	1
Pitch	0	Tool Length	50
Corner Rad	0	Flute Length	30
Diameter at Tip	0	Included Angle	0
	Create Tool	Save Changes	Delete
Location of Tools C:\Users\ TO1, 20mm B TO2, 12mm B TO3, 20mm B TO4, 12mm B TO5, 8mm SI TO5, 8mm SI TO5, 8mm SI TO6, 12mm B TO7, 8.5mm TO8, 10mm I TO9, M10 Ta	: Documents\SharpCam T T T T T T T T T T T T T T T T T T T	Tools\Metric	I & I

The tool currently selected is displayed in the Status Bar at the bottom:

Selected Tool: T01, 20mm End Mill

The outside is now ready to pocket, choose the command:

Menu: Machine-> Pocket

Toolbar button: 🔘

Enter the following values for the Z Positions:

Z Positions	
Abs 🗹 🛛 Initial Rapid	50
Feed From	2
Material Surface	0
Finish Depth	-1
Abs 🗹 🛛 Retract	2
Number of Passes	4
Depth of Cut	0.25

The Finish Depth is taken from the Bottom of the contour, which is -25, so the actual depth is -26

The Depth of cut will not be correct until the contours have been added to the operation.

Enter the following values for the Cutting Data:

Cutting Data	Entry	Advanced	$\prec \rightarrow$
Spindle	Speed		8000
Plunge Fee	d Rate		2000
Cut Fee	d Rate		1000
Finish Allowance			0.5
Step Over			10
Coolant		Flood	-
Sub Ro	outines		

Choose the Top View command. Menu: View -> Top View Toolbar button:

Select the outside profile and the outer rectangle by directly clicking them, the outer rectangle may already be selected:



To pocket the selected Contours click the Add button $\textcircled{\bullet}$. The toolpaths are created:



Looking at the resulting toolpaths a blue line can be seen in the top left hand corner. Blue indicates an Entry, the entry is configured using the Entry tab:

Cutting Data	Entry	Advanced	$\cdot \cdot \models$
Length	: Tool Ra	d x 1	
Radius	: Tool Ra	d x 1	
	Line/A	rc Angle	90
Smart Ramp	v	Ram	p 🔲

By default Smart Ramp is checked and the blue line is a result of this. To see what Smart Ramp is doing view the Part in an isometric view:

Menu: View -> Standard Views -> Isometric View

Toolbar button: 🔷

Choose the command and zoom in on the Smart Ramp by pointing the cursor at the Smart

Ramp and rotating the middle mouse button:



As can be seen the Smart Ramp, as the name implies, automatically ramps to depth at the shallowest angle possible. As it is the outside profile that is being machined Smart Ramp is not needed, because it is possible to plunge to depth outside of the billet. First start from the outside, click the Advanced tab:

En	ntry	Advanced	Program Stop 4	►
	Sta	art At		
	0	Inside	🔘 Outside	
	Cu	t Direction		
	0	Climb	Conventional	

At the moment the pocketing starts from the inside, click the outside radio button and the toolpaths will be recalculated to start from the outside:

Er	ntry	Advanced	Program Stop 4	Þ
	Sta	art At		
	0	Inside	⊙ Outside	
	Cu	t Direction		
	0	Climb	Conventional	



Choose the Entry tab and uncheck the Smart Ramp check box and check the Line check box:



Now the tool is plunging in fresh air:



To start further outside of the billet increase the length of the entry line. Change the entry line length, for example, to 2 (2 x the tool radius = 20):


To update this change click the Refresh button 🔊:



Changes to any value in a box are updated by clicking the Refresh button.

Step 5 - Rough counterbore

Rough Ø120 counterbore using Pocket

Menu: Machine-> Pocket

Toolbar button: 🔘

Choose the command and enter the following Z Positions:

Z Positions	
Abs 🗹 🛛 Initial Rapid	2
Feed From	2
Material Surface	0
Finish Depth	0.5
Abs 🗹 Retract	2
Number of Passes	2
Depth of Cut	0.25

Enter the following values for the Cutting Data:

Cutting Data	Entry	Advanced	$\prec \rightarrow$
Spindle Speed			8000
Plunge Feed Rate			500
Cut Feed Rate			750
Finish Allowance			0.5
Step Over			10
Coolant		Flood	-
Sub Routines			

Choose the Top View command. Menu: View -> Top View Toolbar button: 🗗

Select the Ø120 circle by directly clicking it:





To pocket the selected Contours click the Add button $\textcircled{ \baselinewidth}$. The toolpath is created:

This time Smart Ramp is required, this is still unchecked from the previous pocketing operation. Choose the Entry tab and check Smart Ramp:

Entry	Advanced	Program S	Stop 🔹 🕨
	Length: Tool	Rad x	2 🗵
	Radius: Tool	Rad x	1
	Line	/Arc Angle	90
Sma	art Ramp 🔽	1	Ramp 📃

At the moment the pocketing starts from the outside, but it is preferable to start from the inside. Choose the Advanced tab and select the Inside Radio button:

Entry	Advanced	Program Stop 4	×.
Cto	-+ .+		
518	TLAL		
0 1	inside	 Outside 	
Cut	Direction		
0	Climb	O Conventional	

The toolpath should look like this in isometric view:



Step 6 - Rough bores

Rough Ø50 and Ø70 bores using Pocket

Menu: Machine-> Pocket

Toolbar button: 🔘

Choose the command and enter the following Z Positions:



Enter the following values for the Cutting Data:

	Cutting Data	Entry	Advanced	$\cdot (\cdot \cdot) \models$
	Spindle Speed			8000
Plunge Feed Rate			200	
	Cut Fee	d Rate		500
	Finish Allowance			0.5
	Step Over			10
	C	oolant	Flood	-
	Sub Ro	outines		

Select the Ø50 and Ø70 circles by directly clicking them:



To pocket the selected Contours click the Add button 🔂. The toolpath is created:



Step 7 - Rough inside

Rough inside pocket using Pocket

Menu: Machine-> Pocket

Toolbar button: 🔘

Choose the command and THEN select the tool for this operation.

If a tool is selected, before choosing the Pocket command, SharpCam will assume that the current operation is to be created with a different tool and will regenerate the toolpath accordingly (do not forget that the Ø50 and Ø70 bores have just been roughed and are still editing the operation).

Choose the Tool Manager:

Menu: Machine-> Tool Manger

Toolbar button: 🛣

Choose the command to display the Tool Manager and click on 'T02, 12mm End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			X			
Name	T02, 12mm End Mil	😡 Туре	Flat Mill 🔹			
Number	2	Diameter	12			
Direction	Clockwise 🔹	Offset	2			
Pitch	0	Tool Length	30			
Corner Rad	0	Flute Length	25			
Diameter at Tip	0	Included Angle	0			
	Create Tool	Save Changes	Delete			
	Select Tool	Rename	Close			
Location of Tools	:					
C:\Users\i\	Documents\SharpCa	am Tools Metric				
) 🖉 🔳 🖉	ম হ দ			
T01, 20mm E	End Mill					
T02, 12mm End Mill						
103, 20mm E	nd Mill Ind Mill (2mm Corne	r Pad)				
T05, 8mm Sk	ot Drill	(Kuu)				
👿 T06, 12mm 9	Spot Drill					
🛛 🖉 T07, 8.5mm	Drill					
👩 T08, 10mm 🛛	Drill					
📱 Т09, М10 Та	p					
L						

Enter the following Z Positions:

Z Positions				
Abs 🗹 🛛 Initial Rapid	50			
Feed From	2			
Material Surface	0			
Finish Depth	1			
Abs 🗹 Retract	2			
Number of Passes	4			
Depth of Cut	0.25			

Enter the following values for the Cutting Data:

Cutting Data	Entry	Advanced	
Spindle Speed		i	10000
Plunge Feed Rate			1000
Cut Fee	d Rate		2500
Finish Allowance			0.5
Step Over			6
Coolant		Flood	-
Sub Routines			

Select the Contour on the inside by directly clicking it:



To pocket the selected Contour click the Add button $\textcircled{\bullet}$. The toolpath is created:



Step 8 - Finish outside profile

Finish outside profile using Profile

Menu: Machine-> Profile

Toolbar button: ${\mathfrak I}$

Choose the Profile command then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🜌

Choose the command to display the Tool Manager and click on 'T03, 20mm End Mill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			×		
Name	T03, 20mm End Mill	😡 Туре	Flat Mill 👻		
Number	3	Diameter	20		
Direction	Clockwise 🔹	Offset	3		
Pitch	0	Tool Length	50		
Corner Rad	0	Flute Length	35		
Diameter at Tip	0	Included Angle	0		
	Create Tool	Save Changes	Delete		
	Select Tool	Rename	Close		
Location of Tools	:				
C:\Users\;	Documents\SharpCam	Tools\/Metric			
TE	$\nabla \nabla \nabla$	y t	म स म		
T01, 20mm E	End Mill				
T02, 12mm E	End Mill	Filter - Rea	mer		
T03, 20mm E	End Mill				
104, 12mm E	End Mill (2mm Corner F	(ad)			
TO5, 8mm Si	Spot Drill				
E T07, 8,5mm	Drill				
हिं T08, 10mm [Drill				
🗑 Т09, М10 Та	ap .				

The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:



All operations now indicate that they are suppressed and the toolpaths are now hidden:



Click the Properties tab on the Part Manager to continue with the Profile command:



Enter the following Z Positions:



Enter the following values for the Cutting Data:

	Cutting Data	Entry	Exit	Proj 🔶 🕨
	Spindle Speed			8000
Plunge Feed Rate		Rate		500
	Cut Feed	Rate		1200
	Finish Allowance			0
	Coolant		Flood	•
	Cutter Radius Comp		None	•
	Sub Ro	utines		

Select the outside Contour by directly clicking it:



To profile the selected Contour click the Add button 🔂. The toolpath is created:



Add an entry and exit. Choose the Entry tab and change the Line/Arc Angle to 0 and check the Line check box:

Cutting Data	Entry	Exit	Proj	•
Line				
Length: Tool Rad x 📃 1 🔽				
Arc				
Radius	: Tool Ra	d x	1	
Ramp 🔲	Perp 🔲	Angle	<u>-</u>	0

Choose the Exit tab and check the Line and Arc check box:



The entry and exit have been added:



Cutter radius compensation is required for this profile operation. Choose the Cutting Data tab and select *Control Tool Centre*:



 When using Control Tool Centre, zero must be entered in the control offset table for the correct size tool. Adjust the size with small + or - values.
 If you prefer to enter the full radius in the control offset table, then choose Control.

Step 9 - Finish bores

Finish Ø50 and Ø70 bores using Profile

Menu: Machine-> Profile

Toolbar button: 🕤

Choose the Profile command then select the tool and enter the following values for the Z Positions:

Z Positions	
Abs 🗹 Initial Rapid	2
Feed From	2
Material Surface	0
Finish Depth	-7
Abs 🗹 Retract	2
Number of Passes	1
Depth of Cut 🖉	7

Enter the following values for the Cutting Data:

Cutting Data	Entry	Exit	Proj 🔨 🕨
Spindle	Speed		8000
Plunge Feed Rate			2000
Cut Feed Rate			1000
Finish Allowance			0
Coolant		Flood	-
Cutter Radius Comp		None	-
Sub Routines			

Select the Ø50 and Ø70 circles by directly clicking them:



To profile the selected Contours click the Add button 🔂. The toolpaths are created:



As can be seen the toolpaths are on the outside and in a clockwise direction, this is the default. This needs to be changed to the inside in a counter-clockwise direction. The Contours must be selected in order to change the side and direction of the resulting toolpaths. When Contours are added to an operation they will be left selected. If the Contours are not selected they must be selected first, they can also be selected by clicking the name in *Contours* group box:

Contours: Top	o 0, Btm -2	5
+	2	×
Closed(2)	Open(0)	
Cirde4		
Selec	ted Co	ntours
Side		Direction
🔘 Inside		O CCW
🔘 Centre		⊙ CW
 Outside 		
External Corner Start		
Beginning of Arc O End of Arc		

To change the side and direction click the appropriate radio button, in this case *Inside* and *CCW*:

Contours: To	p 0, Btm -2	5
~	2	×
Closed(2)	Open(0)	
Cirde3 Cirde4		
Side		Direction
Inside		⊙ ccw
O Centre		○ cw
Outside		
External Co	orner Start	
🔘 Beginnir	ng of Arc	O End of Arc

The toolpath is now on the inside and in a counter-clockwise direction:



Change the entry and exit because the settings from the previous operation are still in effect. Choose the Entry tab and set the values (before checking the boxes) as below and check the line and arc check box:

Cutting Data	Entry	Exit	Proj 🔨 🕨	
Line		_		
Length: Tool Rad x 0.5 📝				
Arc		_		
Radius: Tool Rad x 0.5 🗹				
Ramp 🔲	Perp 🔲	Angle	90	

Choose the Exit tab and change the Length and Radius values and click the Refresh 🔊



The entry and exit have been added:



Cutter radius compensation is required for this profile operation. Choose the Cutting Data tab and select *Control Tool Centre*, if not already selected:

Cutting Data Entry	/ Exit
Spindle Speed	8000
Plunge Feed Rate	2000
Cut Feed Rate	1000
Finish Allowance	0
Coolant	Flood 👻
Cutter Radius Comp	Control Tool Ce 👻
Sub Routines	

Deselect the Contours in readiness for the next operation:

Menu: Edit -> Deselect All

Toolbar button: 📐

Step 10 - Finish counterbore

Finish Ø120 counterbore using Pocket

Menu: Machine-> Pocket

Toolbar button: 🔘

Choose the command and enter the following values for the Z Positions:

Z Positions	
Abs 🗹 Initial Rapid	2
Feed From	-12.5
Material Surface	-14.5
Finish Depth	0
Abs 🗹 Retract	2
Number of Passes	1
Depth of Cut	0.5

Enter the following values for the Cutting Data:

Cutting Data	Entry	Advanced	$\cdot \cdot \models$
Spindle Speed			8000
Plunge Feed Rate			1000
Cut Fee	d Rate		2000
Finish Allowance			0
Step Over			10
Coolant		Flood	-
Sub Routines			

First deselect the bores from the previous operation, if required, by directly clicking them, then select the counterbore Contour by directly clicking it:



To profile the selected Contour click the Add button 🔂. The toolpath is created:



Smart Ramp is applied, but is not particularly suitable as the cutter will plunge against the side wall. Choose the Entry tab, uncheck the Smart Ramp and change the line length to 1. Check the Line (if not already selected) and then the Ramp check box:

Cutting Data	Entry	Advanced 🔨 🕨		
Line				
Length: Tool Rad x 📃 1 📝				
Arc				
Radius: Tool Rad x 📃 🔲				
Line/Arc Angle 90				
Smart Ramp		Ramp		

This will create a smaller ramp from the centre of the counterbore:



Step 11 - Finish inside

Finish inside pocket using Pocket

Menu: Machine-> Pocket

Toolbar button: 🔘

Choose the command and then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🛣

Choose the command to display the Tool Manager and click on 'T04, 12mm End Mill (2mm Corner Rad)', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			×		
Name	T04, 12mm End Mil	(2mm) 💿 Type	Bull Nose 🔹		
Number	4	Diameter	12		
Direction	Clockwise 🔹	Offset	4		
Pitch	0	Tool Length	30		
Corner Rad	2	Flute Length	25		
Diameter at Tip	0	Included Angle	0		
	Create Tool	Save Changes	Delete		
	Select Tool	Rename	Close		
Location of Tools	:				
C:\Users\)	Documents\SharpCa	am Tools Metric			
TE	$\nabla \nabla \nabla$		म स म		
To 1, 20mm End Mill					
T02, 12mm End Mill					
T T03, 20mm End Mill					
T04, 12mm End Mill (2mm Corner Rad)					
105, 8mm Slot Drill					
TO7, 8,5mm	Drill				
T08. 10mm	Drill				
T09, M10 Ta	ID.				

Enter the following values for the Z Positions:

Z Positions	
Abs 🗹 Initial Rapid	50
Feed From	-17
Material Surface	-19
Finish Depth	0
Abs 🗹 Retract	2
Number of Passes	1
Depth of Cut	19

Enter the following values for the Cutting Data.

Note that the step over value has been changed from the default value of 6 to 4 to ensure that pocket fully cleans up on the bottom, don't forget that the cutter has a 2mm corner radius:

Cutting Data	Entry	Advanced	$\prec \rightarrow$
Spindle Speed		:	10000
Plunge Feed Rate			1000
Cut Feed Rate			2000
Finish Allowance			0
Step Over			4
Coolant		Flood	-
Sub Routines			

First deselect the counterbore from the previous operation, if required, by directly clicking it, then select the Contour on the inside by directly clicking it:



To pocket the selected Contour click the Add button 🔂. The toolpath is created:



The entry settings from the previous operation are acceptable.

Step 12 - Machine radial slots

Machine radial slots using Profile

Menu: Machine-> Profile

Toolbar button: 🕤

Choose the Profile command then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🛣

Choose the command to display the Tool Manager and click on 'T05, 8mm Slot Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			X		
Name	T05, 8mm Slot Drill	😥 Туре	Flat Mill 👻		
Number	5	Diameter	8		
Direction	Clockwise 🔹	Offset	5		
Pitch	0	Tool Length	30		
Corner Rad	0	Flute Length	20		
Diameter at Tip	0	Included Angle	0		
	Create Tool	Save Changes	Delete		
	Select Tool	Rename	Close		
Location of Tools	:				
C:\Users\;	Documents\SharpCa	m Tools∖Metric			
TE			म स म		
To 1, 20mm End Mill					
TO2, 12mm End Mill					
T T03, 20mm End Mill					
T05, 8mm Slot Drill					
T06, 12mm Spot Drill					
🛃 T07, 8.5mm	Drill				
ឌ្ឌ T08, 10mm [Drill				
📱 Т09, М10 Тар					

Before machining the slots suppress the operation that machined the Ø120 counterbore, so it does not obscure the view.

Choose the Operations tab on the Part Manager and right click on the last operation created by 'T03, 20mm End Mill' and select Suppress:



Click the Properties tab on the Part Manager to continue with the Profile command:



Enter the following values for the Z Positions:



Enter the following values for the Cutting Data:

Cutting Data	Entry	Exit	Proj 🔨	►
Spindle Speed			12000)
Plunge Feed	Rate		100)
Cut Feed	Rate		500)
Finish Allov	wance		0)
C	oolant	Flood	-	,
Cutter Radius	Comp	None	-	
Sub Routines				

Select the slots by directly clicking them. Also unselect the Ø120 circle by directly clicking it, it was selected when it was suppressed:



To profile the selected Contours click the Add button 🗗. The toolpaths are created on the outside by default, so you will not be able to see them as they are obscured by the model.

With the Contours still selected change the side to *Inside* and the direction to *CCW*:

Contours: To	p -15, Btm	-25			
-	2	×			
Closed(4)	Open(0)				
Contour3 Contour4 Contour5 Contour6					
Side O Inside O Centre O Outside		Direction CCW CW			
External Corner Start Beginning of Arc O End of Arc					

The toolpath is now on the inside and in counter-clockwise direction:



Choose the Entry tab and set as below, changes the values before checking/unchecking the boxes:

Cutting Data	Entry	Exit	Proj 🔶 🕨
Line			
Length	: Tool Ra	d x 🛛	0.25 🔽
Arc			
Radius	: Tool Ra	d x	1
Ramp 🔲	Perp 🔲	Angle	90

Choose the Exit tab and set the same as the entry:

Cutting Data	Entry	Exit	Proj 🔹 🕨				
Line							
Length: Tool Rad x 0.25							
Arc							
Radius	: Tool Ra	d x	1				
Ramp 🔲	Perp 🔲	Angle	e 90				

The entry and exits have been added to all 4 slots:



Step 13 - Spot drill holes

Spot Drill holes using Drilling

Deselect radial slots by pressing the Escape key twice.

Menu: Machine-> Drilling

Toolbar button: 📲

Choose the Drilling command then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🛣

Choose the command to display the Tool Manager and click on 'T06, 12mm Spot Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:
Tool Manager			×			
Name	T06, 12mm Spot Dri	l 😡 Type	Drill 👻			
Number	6	Diameter	12			
Direction	Clockwise 👻	Offset	6			
Pitch	0	Tool Length	50			
Corner Rad	0	Flute Length	10			
Diameter at Tip	0	Tip Angle	90			
	Create Tool	Save Changes	Delete			
	Select Tool	Rename	Close			
Location of Tools	:					
C:\Users'	Documents\SharpCa	m Tools (Metric				
TE	$\nabla \nabla \nabla$	e T F	म स म			
T01, 20mm E	End Mill					
T02, 12mm E	End Mill					
T03, 20mm E	End Mill	- 0				
T04, 12mm E	End Mill (2mm Corner	Rad)				
T05, 8mm Si	ot Drill Spot Drill					
a T07, 8,5mm	Drill					
हिं T08, 10mm [Drill					
🗑 Т09, М10 Та	ip					

Enter the following Z Positions and Cutting Data as below, remember the *Finish Depth* is from the bottom of the Contour which is -25:

Z Positions					
Abs 🗹 Initial Rapid	i 50				
Feed From	2				
Material Surface	. 0				
Finish Depth	20.5				
Retract To:	 Initial Rapid Feed From 				
Cutting Data Program Stop					
Spindle Speed	8000				
Feed Rate	200				
Peck Amount	0				
Dwell 0					
Coolant	Flood 🔻				
Sub Routines					

Select the Circles by directly clicking them:



To drill the selected Circles click the Add button 🔂. The toolpaths are created:



As the Z position of the Ø10 holes are different a new drilling operation is required. Menu: Machine-> Drilling

Toolbar button: 📲

Choose the Drilling command then enter the following values for the Z Positions:

Z Positions				
Abs 🗹 Initial Rapi	d 2			
Feed From	n 2			
Material Surface	e 0			
Finish Dept	h 5.5			
Retract To: O Feed From				
Cutting Data Prog	ram Stop			
Spindle Speed	8000			
Feed Rate	200			
Peck Amount	0			
Dwell	0			
Coolant	Flood 🔻			
Sub Routines				

Select the Circles by directly clicking them:



To drill the selected Circles click the Add button 🔂. The toolpaths are created:



Step 14 - Drill holes

Drill Ø8.5 holes using Drilling

Menu: Machine-> Drilling

Toolbar button: 📲

Choose the Drilling command then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🜌

Choose the command to display the Tool Manager and click on 'T07, 8.5mm Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			×				
Name	T07, 8.5mm Drill	🕖 Туре	Drill 👻				
Number	7	Diameter	8.5				
Direction	Clockwise 👻	Offset	7				
Pitch	0	Tool Length	70				
Corner Rad	0	Flute Length	50				
Diameter at Tip	0	Tip Angle	118				
	Create Tool	Save Changes	Delete				
	Select Tool	Rename	Close				
Location of Tools	:						
C:\Users\ \	Documents\SharpCar	m Tools\Metric					
TEC		t T T	ज र ज				
T01, 20mm E	End Mill						
T02, 12mm E	End Mill						
T T04, 12mm E	na Mill End Mill (2mm Corner	Rad)					
T05, 8mm Sk	ot Drill	(Loo)					
👩 T06, 12mm S	Spot Drill						
👩 T07, 8.5mm	Drill						
हु T08, 10mm E	Drill						
Т09, М10 Та	p						

The Part now has a number of toolpaths. In order to prevent the view from being obscured, suppress them so they are hidden. Choose the Operations tab on the Part Manager and click the Suppress All button:



Click the Properties tab on the Part Manager to continue with the Drilling command:



Enter the following Z Positions and Cutting Data as below, note that the type is Peck Drill:

Type Peck Drill 🔻					
Z Positions					
Abs 🗹 Initial Rapid	50				
Feed From	2				
Material Surface	0				
Finish Depth	-10				
Retract To:	Initial Rapid Feed From				
Cutting Data Program St	ор				
Spindle Speed	8000				
Feed Rate	600				
Peck Amount	10				
Dwell 0					
Dwell	0				
Dwell Coolant Flood	0				

Select the Circles by directly clicking them:



To drill the selected Circles click the Add button $\textcircled{\blacksquare}$. The toolpaths are created:



Step 15 - Drill holes

Drill Ø10 holes using Drilling

Menu: Machine-> Drilling

Toolbar button: 📲

Choose the Drilling command then select the tool for this operation:

Menu: Machine-> Tool Manger

Toolbar button: 🜌

Choose the command to display the Tool Manager and click on 'T08, 10mm Drill', then click the 'Select Tool' button. Alternatively double click a tool to select it:

Tool Manager			×				
Name	T08, 10mm Drill	🔘 Type	Drill 👻				
Number	8	Diameter	10				
Direction	Clockwise 👻	Offset	8				
Pitch	0	Tool Length	80				
Corner Rad	0	Flute Length	60				
Diameter at Tip	0	Tip Angle	118				
	Create Tool	Save Changes	Delete				
	Select Tool	Rename	Close				
Location of Tools	:						
C:\Users'	Documents\SharpCa	am Tools Metric					
TE) T T	ज र ज				
T01, 20mm E	End Mill						
T02, 12mm E	End Mill						
T03, 20mm E	End Mill	- D1)					
T T05 8mm Sl	at Drill	r Rad)					
T06, 12mm 9	Spot Drill						
2 T07, 8.5mm	Drill						
👩 T08, 10mm (Drill						
😈 T09, M10 Ta	ip						

Enter the following Z Positions and Cutting Data as below, note that the type is Drilling:

Type Drilling 👻					
Z Positions					
Abs 🗹 Initial Rapid	50				
Feed From	2				
Material Surface	0				
Finish Depth	-10				
© Initial Rapid Retract To: O Feed From					
Cutting Data Program Stop					
Spindle Speed	7000				
Feed Rate	700				
Peck Amount 10					
Dwell 0					
Coolant Flood 💌					
Sub Routines 🔲					

Select the Circles by directly clicking them:



To drill the selected Circles click the Add button $\textcircled{\blacksquare}$. The toolpaths are created:



All toolpaths have now been created.

Step 16 - Create Material Stock

Create Material Stock for Solid Simulation

First a rectangle must be created that represents the Material Stock. As mentioned in Step 1 the Gear Housing is to be made from an aluminium rectangular billet 295mm x 225mm x 30mm.

Choose the Rectangle command:

Menu: Draw -> Rectangle

Toolbar button: 🗖

Create a rectangle with a start point value of X-70, Y-90 and an end point value of X225, Y135:

Start C	Corner Point
Ditar e e	Jorner Forne
x	-70
v	-90
	OK
End Co	orner Point
End Co	orner Point
End Co X	orner Point 225
End Co X	orner Point 225 135

Set the model view to isometric: Menu: View -> Standard View -> Isometric View Toolbar button:

Press the Escape Key to exit Rectangle mode.

Select the rectangle by directly clicking it:



Set the Contour Heights:

Menu: Modify -> Set Contour Height Toolbar button: $eq^{\Box i}$

Click OK to accept the default value of zero for the Z Top, then enter -27 in Z Bottom and click OK:

Contour Height
Тор
ОК
Bottom
-27 OK

Create the Material Stock by perform the Material From Contours command:

Menu: View -> Material Stock - >From Contours

Toolbar button: \square



Step 17 - Solid Simulation

Before simulating the part the operations must be unsuppressed. Choose the Operations tab on the Part Manager and click the Unsuppress All button:



Simulate part being machined

Menu: View-> Solid Simulation

Toolbar button: 퇷

There are many options and you are encouraged to experiment, but for now you can press either the Forwards button \triangleright or the Fast Forwards button \triangleright . The part is simulated until the end:



Simulating the Part is optional and does not need to be performed to create the Nc code.

Step 18 - Output Nc code

Outputting Nc Code

Choose the Nc Code tab on the Part Manager :

	<u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>D</u> raw	<u>M</u> achine	M <u>o</u> dify
) 🔁			8 9	6 X	-]: 🖻 [
2	ŝ	2	F.	adal		• 1 6
1	peratio	% 01 (GEAR H	HOUSING	INCH)	
Ν	ō	(CYC (T07	LE TIN , 0.31	ME 0.73 125INCH	MINUTES DRILL)	5)
	5	(T08 G40	, 0.37 G17 G8	5INCH	DRILL)	
0	pertie	G91 M01	G28 Z0)		
ŏ	Ero	T7 M G0 G	6 (T07 90 G54	7, 0.31 4 X5.15	25INCH I Y-2.75	RILL) M3 S8000
Ō		G43 683	Z2.0 H G98 R0	H7 M8).1 Z-1	.2 Q0.4	F25.0
0	Code	X-1. 680	95			
5	ž	691 101	G28 Z0	M09		
	2	С8 М G0 G	6 (TO8 90 G54	8, 0.37 4 X1.17	SINCH DF 5 YO.0 M	(ILL) 13 S7000
A	Tools	G43 G81	Z0.1 H G98 R-	H8 M8 -0.52 Z	-1.2 F30	.0
		X0.5 X-0.	875 Y1 5875	0176		
	ers	X-1. X-0.	175 YC 5875 Y).0 7-1.017	6	
Ø	Laj	X0.5 G80	875			
Æ		G91	G28 Z0)		

There are two choices:

- Click the 🛃 button and save the Nc code to disk.
- Click the $\ensuremath{^{1}}$ button and send directly to the machine, using RS232 communications.

Gear Housing Drawing



1 Index

Create the Contours Step 1 - Extract Contours from Model, 5-9 Step 2 - Set Contour Heights, 9-23 Creating the Toolpaths Step 1 - Process plan, 24 Step 10 - Finish counterbore, 59-62 Step 11 - Finish inside, 62-65 Step 12 - Machine radial slots, 65-72 Step 13 - Spot drill holes, 72-77 Step 14 - Drill holes, 77-82 Step 15 - Drill holes, 82-86 Step 16 - Create Material Stock, 86-89 Step 17 - Solid Simulation, 89-91 Step 18 - Output Nc code, 91-92 Step 2 - Set start point, 24-26 Step 3 - Prepare for roughing, 26-29 Step 4 - Rough outside profile, 29-37 Step 5 - Rough counterbore, 37-40 Step 6 - Rough bores, 40-43 Step 7 - Rough inside, 43-46 Step 8 - Finish outside profile, 46-53 Step 9 - Finish bores, 53-59 Gear Housing Drawing, 93 **Getting Started** Opening Solid Model File, 2-4 Opening Solid Model File, 2-4 Step 1 - Extract Contours from Model, 5-9 Step 1 - Process plan, 24 Step 10 - Finish counterbore, 59-62 Step 11 - Finish inside, 62-65 Step 12 - Machine radial slots, 65-72 Step 13 - Spot drill holes, 72-77 Step 14 - Drill holes, 77-82 Step 15 - Drill holes, 82-86 Step 16 - Create Material Stock, 86-89 Step 17 - Solid Simulation, 89-91 Step 18 - Output Nc code, 91-92 Step 2 - Set Contour Heights, 9-23

- Step 2 Set start point, 24-26
- Step 3 Prepare for roughing, 26-29
- Step 4 Rough outside profile, 29-37
- Step 5 Rough counterbore, 37-40
- Step 6 Rough bores, 40-43
- Step 7 Rough inside, 43-46
- Step 8 Finish outside profile, 46-53
- Step 9 Finish bores, 53-59