

Mill  
Level 1  
Training  
Tutorial

Metric

Mastercam

X<sup>4</sup>





## Mill Level 1- Metric Training Tutorials

To order more books:  
Call 1-800-529-5517 or  
Visit [www.inhousesolutions.com](http://www.inhousesolutions.com) or  
Contact your Mastercam Dealer

---



Mastercam® X4 Training Tutorials - Mill Level 1 Metric Applications  
Date: August 6, 2009  
Copyright © 1984 - 2009 In-House Solutions Inc. - All rights reserved.  
Software: Mill Mastercam X4  
Author: Mariana Lendel  
ISBN: 978-1-926566-52-8

### **Notice**

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

### **Disclaimer of All Warranties and Liability**

In-House Solutions Inc. makes no warranties, either expressed or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. This In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, or correction, and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

### **Copyrights**

This manual is protected under the copyright laws of Canada and the United States. All rights are reserved. This document may not, in whole or part, be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine-readable form without prior consent, in writing, from In-House Solutions Inc.

### **Trademarks**

Mastercam is a registered trademark of CNC Software, Inc.  
Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation; Mastercam Verify is created in conjunction with Sirius Systems Corporation; Windows 95, Windows NT; and Windows XP are registered trademarks of Microsoft Corporation.

### **Acknowledgements**

"Thanks to the In-House Solutions team for helping in the production and marketing of this book. It simply wouldn't be the same without all your efforts. Finally, thank you, the reader for making this purchase; I hope it serves you well."

### **Special Thanks to:**

Chris Lang and Andrew Bleau for their help revising this book.

Sincerely,  
Mariana Lendel



# TUTORIAL SERIES FOR

# *Mastercam* X<sup>4</sup>

## HOW TO USE THIS BOOK


This book provides a comprehensive step by step approach to learning Mastercam Mill. It contains pages of projects, helpful hints, as well as tool and material library instructions. The book covers ten part projects in all and an additional 17 exercises.


The material covered includes 2D Geometry and Solids Creation, Drilling, Tapping, Contouring, Pocketing, Circle Milling and Slot Milling. It also contains 2D High Speed Toolpaths and Feature Based Machining. Explanations are given for the use of WCS in multiple fixture applications as well as proper Stock Setup and Tool Settings.

The Mill Training Tutorials also include a General Notes chapter with useful tools and shortcuts that make the software easier to use. An alphabetic description of the 2D toolpath parameters, of the Solid menu and Operations Manager are also covered in the General Notes.

Each tutorial walks you through all the procedures from Geometry Creation to Toolpath instructions, Verification and G-Code Generation.

### LEGEND:

 Step to follow to complete the tutorial

 Additional explanation for the current step or warnings.



Callouts that give direction on how to complete the task



Callouts that describe the parameters used in the current step

[Select position for first corner]: Mastercam prompts Select position of base point

**Bold** text (usually) represents Mastercam terminology



## TABLE OF CONTENTS

<b>Getting Started</b>	<b>A-1</b>
Graphic User Interface .....	A-1
Navigate Through the System.....	A-2
Setting Mastercam to Metric.....	A-5
Setting the Toolbar States for Mill.....	A-7
Setting the Grid .....	A-9
 <b>TUTORIALS</b>	 <b>1-1</b>
<b>Tutorial #1</b> , 2D Geometry, Open Contour and Drilling.....	1-1
<b>Tutorial #2</b> , 2D Geometry, Drill, Open Pocket, Open Contour and Chamfer .....	2-1
<b>Tutorial #3</b> , 2D Geometry, Tapping, Contouring, Pocketing and Save Operations.	3-1
<b>Tutorial #4</b> , 2D Geometry, Drill, Slot Mill & 2D HS Core Mill .....	4-1
<b>Tutorial #5</b> , 2D Geometry, Dynamic Mill & Transform Toolpaths .....	5-1
<b>Tutorial #6</b> , 2D Geometry, Pocket with an Island & Circle Mill .....	6-1
<b>Tutorial #7</b> , 2D Geometry, Using WCS, Facing, Drilling, Pocketing, Contour .....	7-1
<b>Tutorial #8</b> , 2D & Solids Geometry, Machining the part using WCS .....	8-1
<b>Tutorial #9</b> , 2D & Solids Geometry, 2D HS Toolpaths, Import a SolidWorks file .....	9-1
<b>Tutorial #10</b> , Import a SolidWorks file, Orient the part, Dynamic plane & FBM, .....	10-1
 <b>General Notes</b>	 <b>B-1</b>
Default Key Assignments .....	B-2
Customizing.....	B-3
Key Mapping .....	B-7
Data Entry Shortcuts .....	B-9
2D Toolpath Parameters .....	B-10
2D HS Specific Parameters .....	B-23
Creating 3D Wireframe .....	B-28
Solids Menu Commands .....	B-33
Solids Manager.....	B-34
Chaining .....	B-36
Window Selection .....	B-39
Chaining and Window Options .....	B-40
Toolpaths Manager .....	B-42
Properties.....	B-47
Milling G-Codes.....	B-50
 <b>Creating or Editing a Tool in the Library</b>	 <b>C-1</b>
Create a New Tool.....	C-1
Edit an Existing Tool .....	C-6



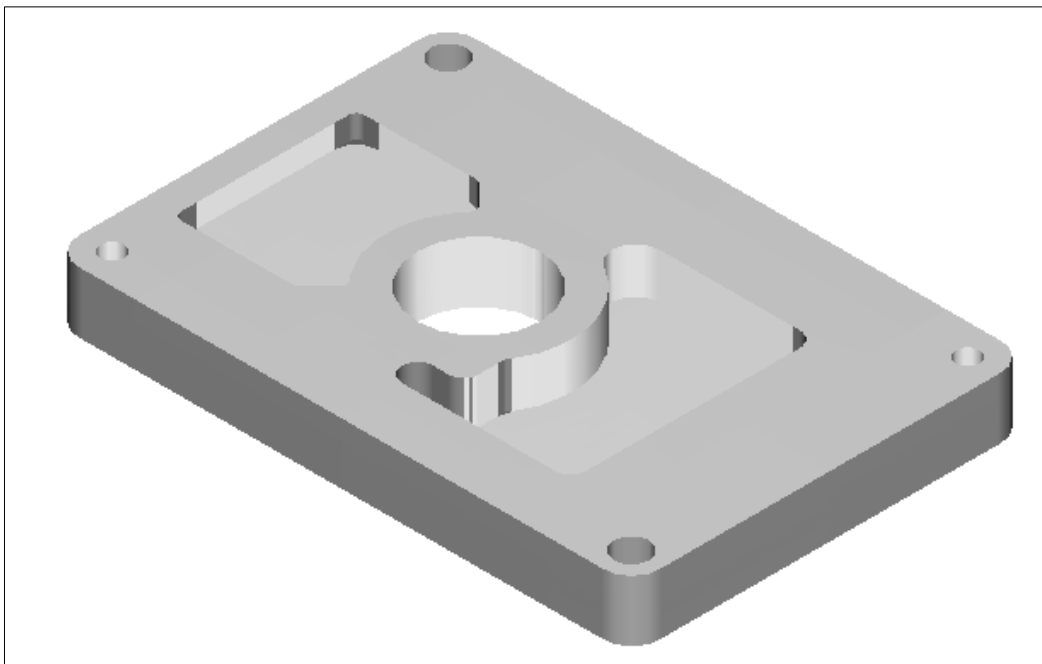
## TUTORIAL SERIES FOR

***Mastercam X<sup>4</sup>***

### TUTORIAL #7

#### LEVEL 1 – 3D WIREFRAME

USING WCS, FACING, CONTOURING, POCKETING, DRILLING,  
IMPORT FROM LIBRARY.



**Mill Level 1 - Metric**

*Objectives:*

***The Student will design a 3-dimensional wireframe drawing by:***

- Creating a rectangle.
- Creating parallel lines.
- Creating fillet radii.
- Creating arcs knowing the center location and the diameter.
- Creating an arc tangent to three entities.
- Using Translate to create a 3-dimensional wireframe.

***The Student will create a 2-dimensional milling toolpath consisting of:***

*First Setup*

- Facing the bottom of the part for an even surface.
- Importing from the library M16 x 2 holes.
- Drilling 10 mm holes.

*Second Setup*

- Facing the top of the part
- Machining simultaneous two pockets with different depths.
- Circle milling the 50 mm hole.
- Machining a 2D contour.

***The Student will check the toolpath using Mastercam's Verify module by:***

- Defining a 3-dimensional rectangular block the size of the workpiece.
- Running the Verify function to machine the part on the screen.

[illegible]

Technical drawing of a rectangular bar. The bar is oriented horizontally. It features a central section filled with diagonal hatching lines. The total length of the bar is indicated by a dimension line below, labeled 137. The width of the bar is indicated by a dimension line to the right, labeled 12. The drawing includes standard engineering notation for dimensions and hatching.

Technical drawing of a rectangular plate. The overall dimensions are 25 (width) and 19 (height). The plate has a central rectangular hole with a diameter of  $\varnothing 50$ . The hole is centered horizontally and vertically. The distance from the left edge to the left side of the hole is 10. The distance from the right edge to the right side of the hole is 10. The distance from the top edge to the top side of the hole is 10. The distance from the bottom edge to the bottom side of the hole is 10. The plate is shown with a hatched pattern on the left and right sides, indicating a cross-section or material type.

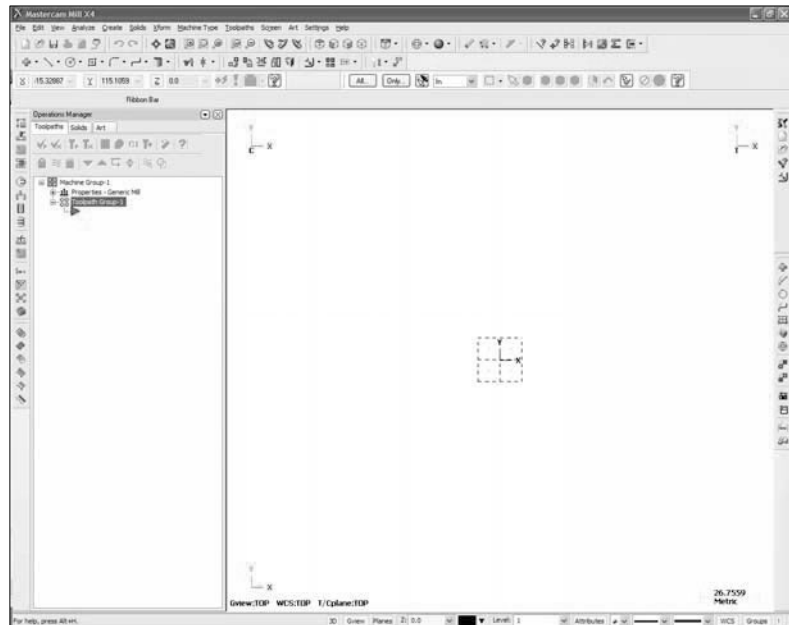
## GEOMETRY CREATION

### Setting up the Graphic User Interface

Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-7 for details.

Make sure that the **Grid** is enabled. It will show you where the part origin is. See **Getting started** page A-9 for further information.

- The **Operations Manager** to the left of the screen can be hidden to gain more space in the graphic area for design. From the keyboard, press **Alt + O** keys simultaneously to hide it. Repeat this command to make it visible again.



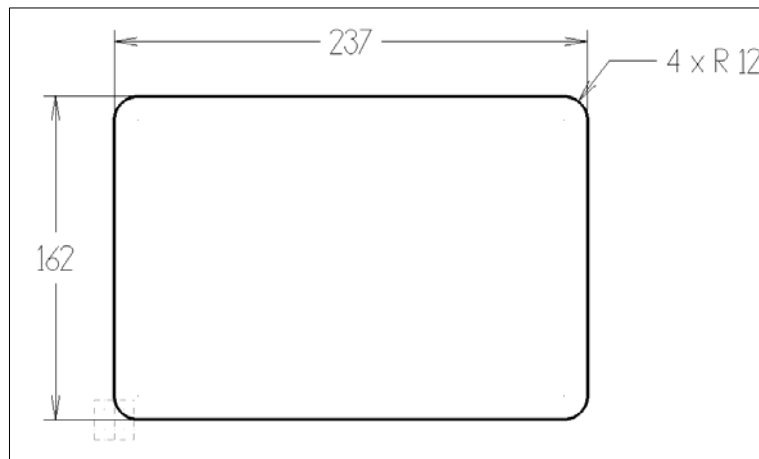
*Starting a new file*

### File

• **New**

## STEP 1: CREATE THE OUTSIDE PROFILE




*Step Preview:*



**Mill Level 1 - Metric**

**Create**

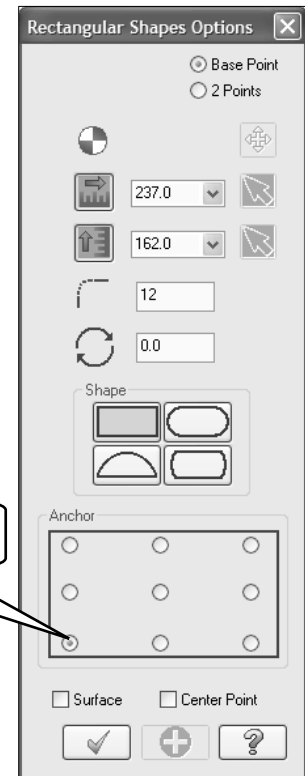
**Rectangular Shapes**

- ➔ Type the **Width** , the **Height**  and the **Radius of the fillet**  as shown in the following picture.

- ➔ Select the **Rectangular Shape**.
- ➔ Select the left bottom radio button as the anchor.

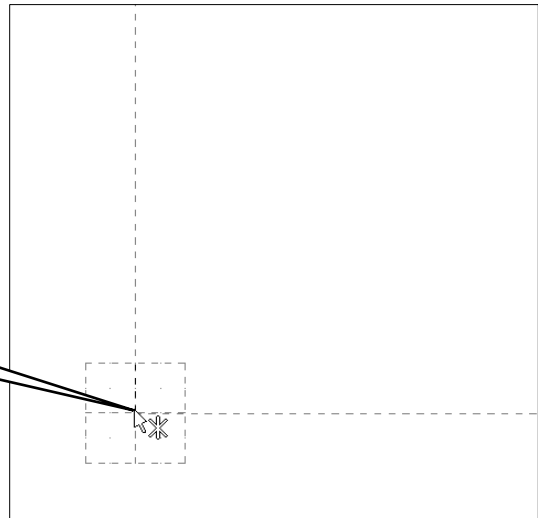
Select this radio button


- ➔ Do not exit the dialog box until you select the base point




- ➔ [Select position for the base point]: Select the center location of the grid (the origin).

Select the Origin



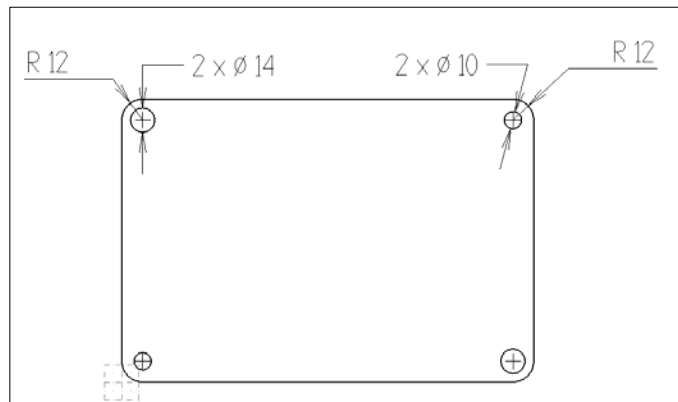
- ➔ Select the **OK** button to exit the rectangle dialog box. 

- ➔ Use the **Fit** icon to fit the drawing to the screen. 

- During the geometry creation of this tutorial, if you make a mistake, to undo the last step you can use the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.

## STEP 2: CREATE THE 14 MM AND 10 MM DIAMETER CIRCLES KNOWING THE CENTER POINT AND THE DIAMETER

*Step Preview:*



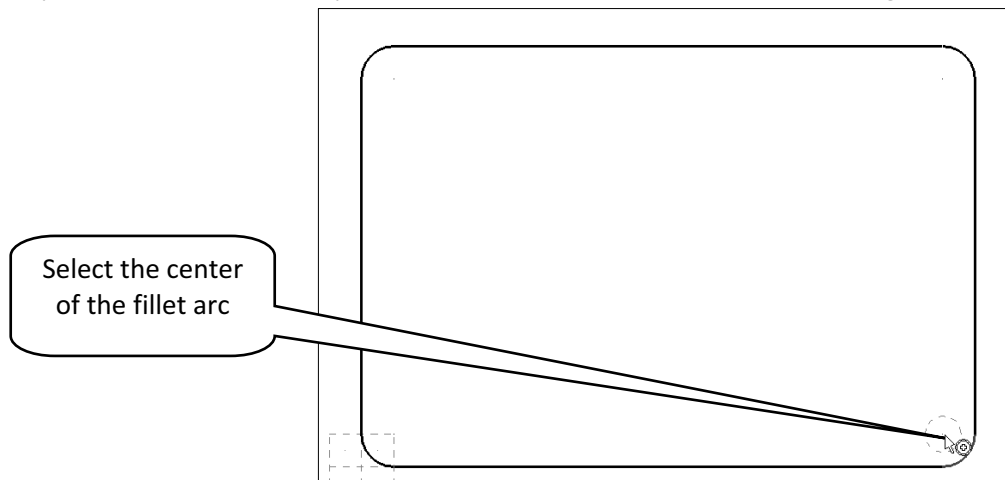
### 2.1 Create the two 14mm diameter circles

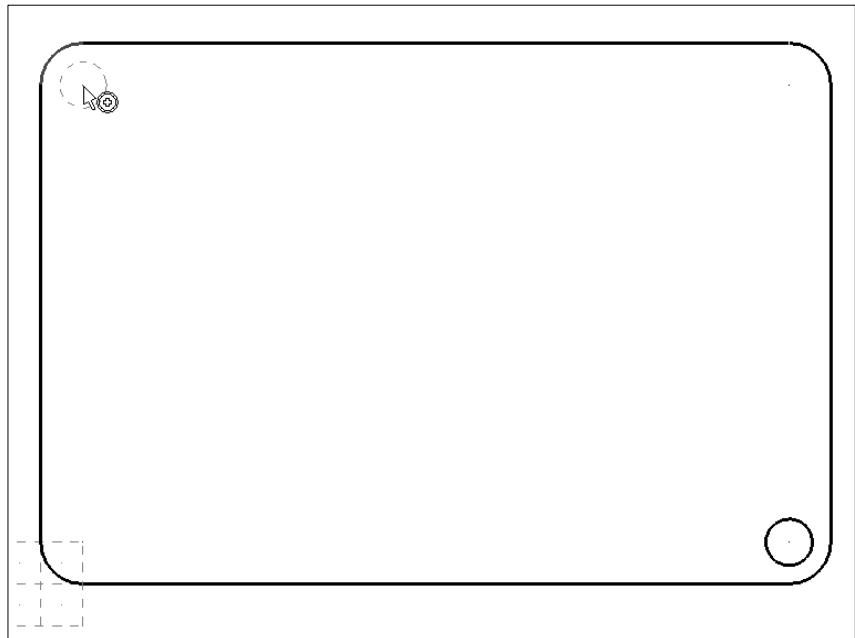
#### Create

##### Arc

##### Circle Center Point

- Enter the **Diameter** value 14 (Enter).
- To be able to create more than one circle with the same diameter, select the diameter icon to lock the value.
- [Enter the center point]: Select the center point of the fillet arcs as shown in the following two pictures.





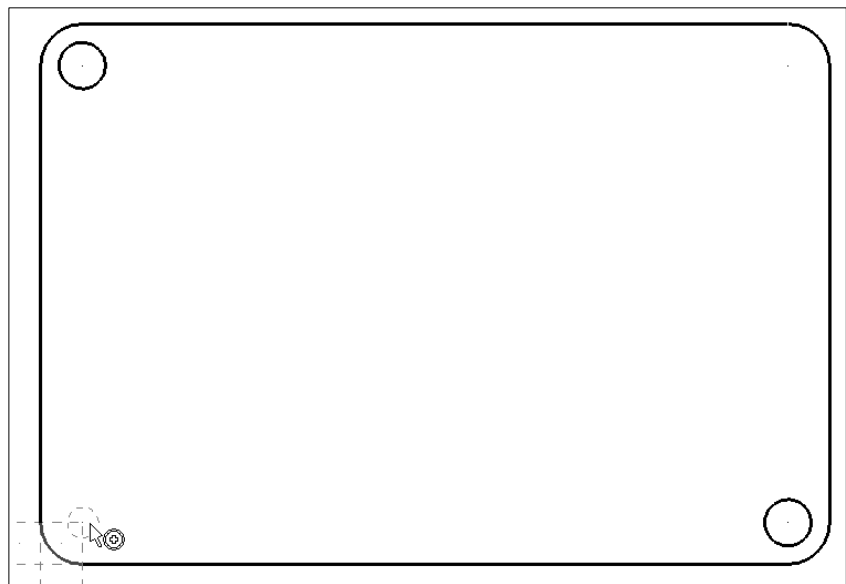
- ➔ Select the **Apply** button. 

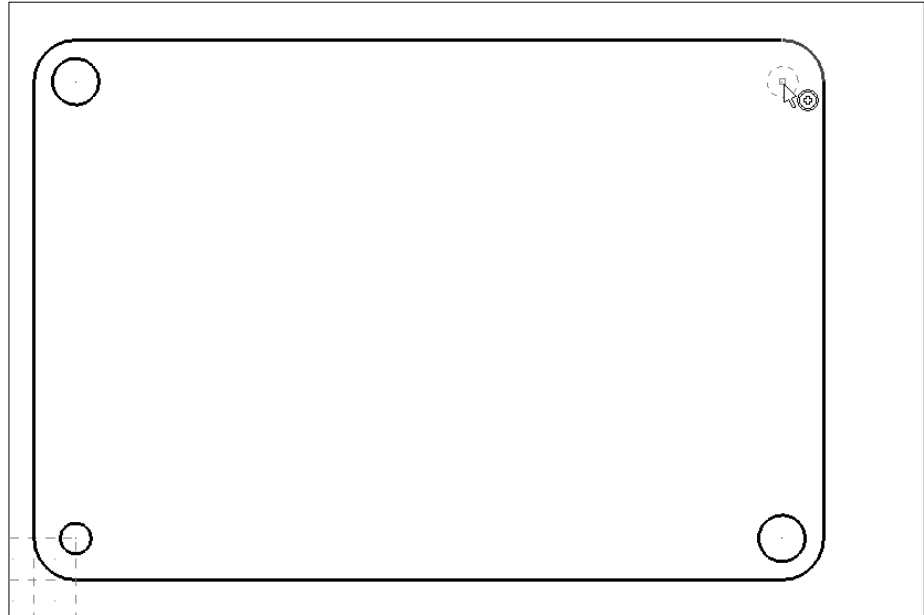
## 2.2 Create the two 10 mm diameter circles


- ➔ Change the diameter to 10 and leave it locked (Press Enter)



- ➔ [Enter the center point]: Select the center point of the fillet arcs as shown in the following two pictures.

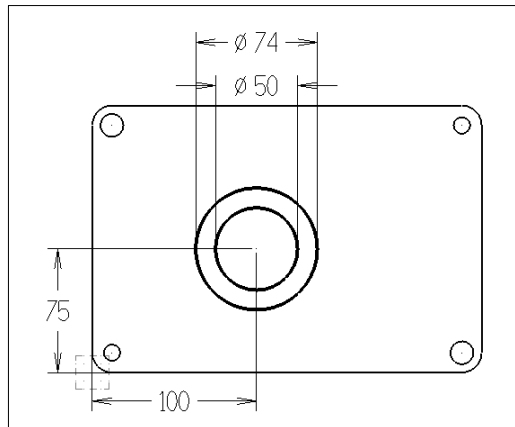








- Select the **Apply** button to continue using the same command. 

### STEP 3: CREATE THE CIRCLES WITH 74 MM AND 50 MM DIAMETERS

*Step Preview:*




#### 3.1 Create the 74 mm diameter circle.

- Unlock the value by reselecting the diameter icon. 
- Enter the **Diameter** value  74 (Enter).
- [Enter the center point]: Select the **Fast Point** icon. 
- Enter the values into the box to the left of the icon: 100, 75 (Enter).
- Select the **Apply** button to continue using the same command. 

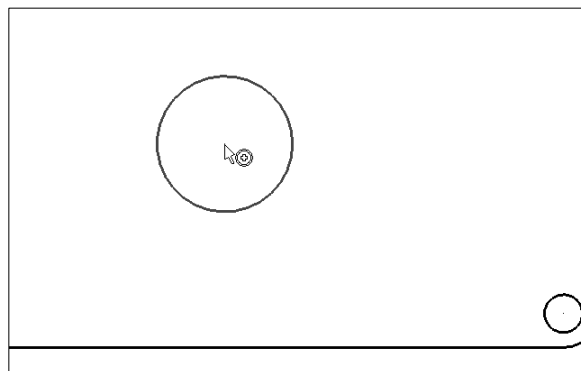


**Mill Level 1 - Metric**

**3.2 Create the 50 mm diameter circle.**

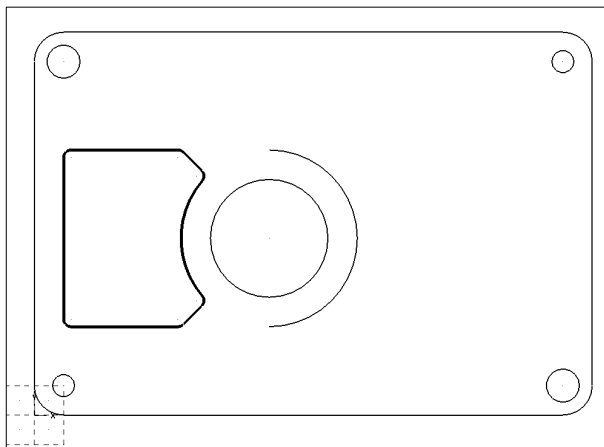
- Enter the **Diameter** value  50 (Enter).
- [Enter the center point]: Select the Center location of the 74 mm diameter.

- Select the **OK** button. 



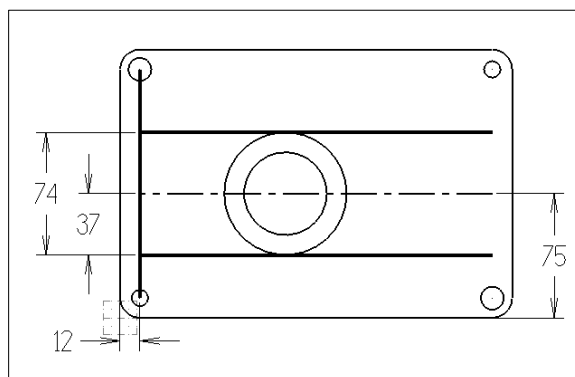
**STEP 4: CREATE THE LEFT SIDE POCKET**

*Step Preview:*



**4.1 Create parallel lines.**

*Sub Step Preview:*





**Create**

-  **Line**

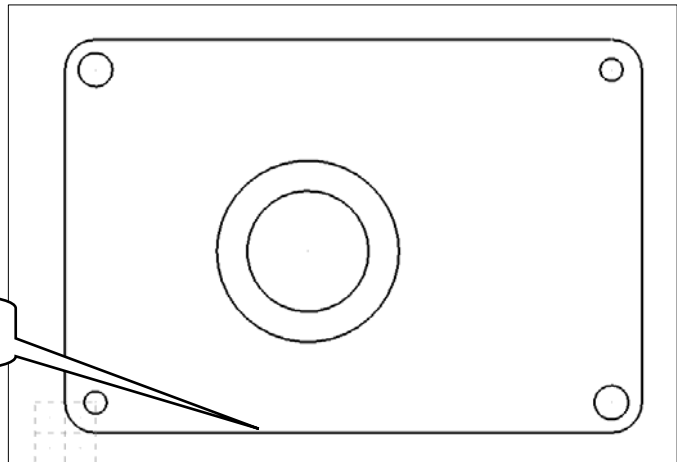
-  **Parallel**


**Mill Level 1 - Metric**

- [ Select a line ]: Select Entity A.
- [ Select the point to place a parallel line through ]: Pick a point above the selected line.
- Type the **Distance**  75-37 (Enter).
- [ Select a line ]: Select Entity A.
- [ Select the point to place a parallel line through ]: Pick a point above the selected line.
- Type the **Distance**  75-37+74 (Enter).

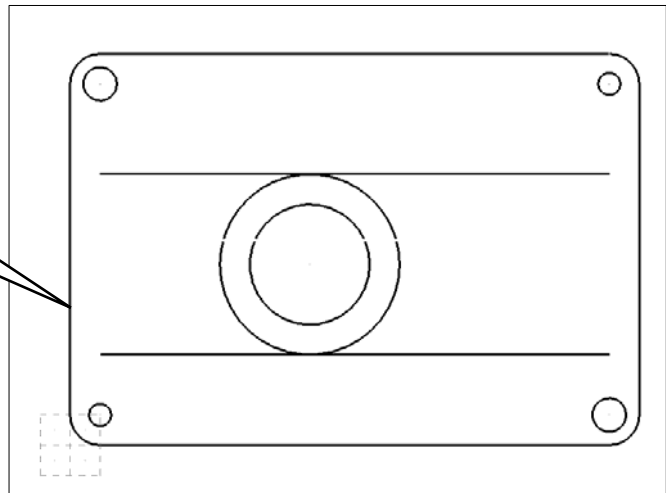
➤ Remember that Mastercam performs basic math functions.

Select Entity A



- [ Select a line ]: Select Entity B.
- [ Select the point to place a parallel line through ]: Pick a point to the right of the selected line.
- Type the **Distance**  12 (Enter).

Select Entity B

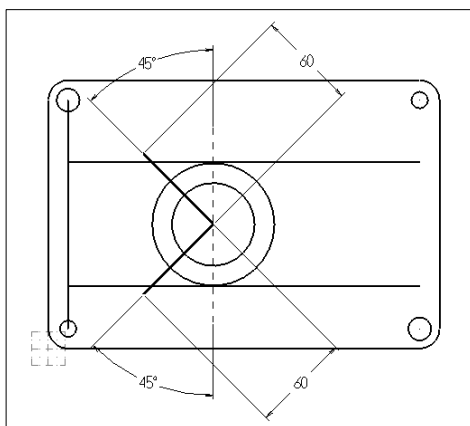


- Select the **OK** button to exit the command. 

**Mill Level 1 - Metric**

**4.2 Create the two polar lines using line endpoints command.**

Sub Step Preview:



**Create**

**Line**


**Endpoints**

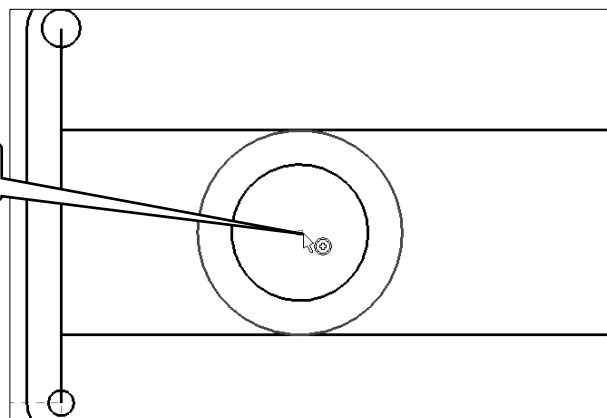
Enter the **Length** and the **Angle**:

[Specify the first endpoint]: Select the Center of the 50 mm diameter circle.



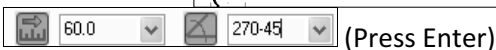
Select the center


Select the **Apply** button.  to create another line using the same command.

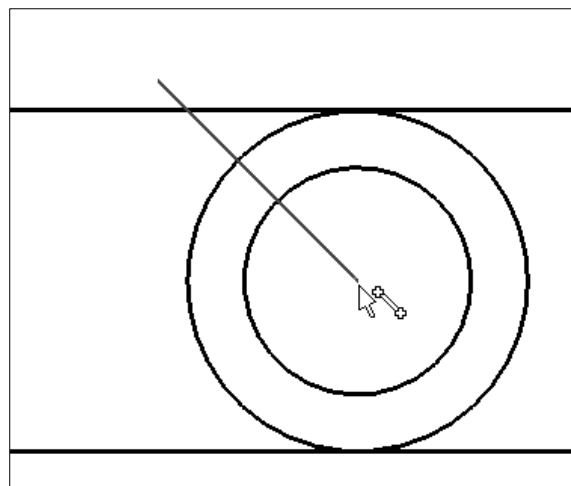


Enter the **Length** and the **Angle**:

[Specify the first endpoint]: Select the Endpoint of the line as shown.



Select the **OK** button. 

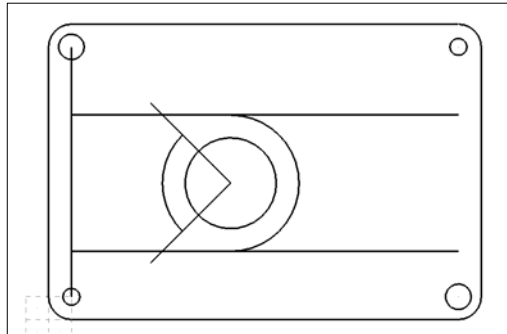


**Mill Level 1 - Metric**

**4.3 Remove parts of the 74 mm diameter circle that lie above and below the polar lines using Trim/Divide command**

- By dividing the arc we are maintaining the part of the arc that is needed for the right side pocket.

*Sub Step Preview:*



**Edit**

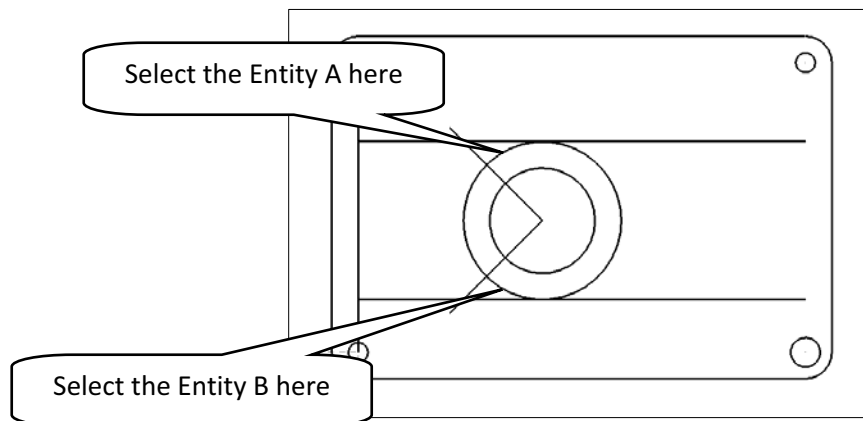
➔ **Trim/Break**

➔ **Trim/Break/Extend**



- ➔ Select the **Divide** icon from the trim ribbon bar.
- ➔ [Select the curve to divide]: Select Entity A here.
- ➔ [Select the curve to divide]: Select Entity B here.

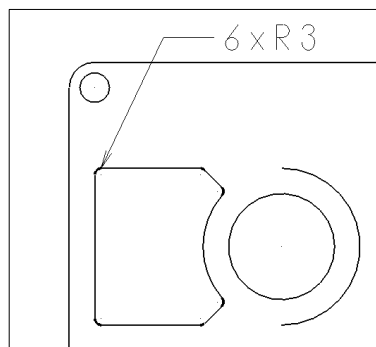
- ➔ Select the **OK** button.



**4.4 Creating the 3 mm radius fillets at all corners of the left side pocket**

- Note that the fillet command will allow us to automatically trim the geometry.

*Sub Step Preview:*




**Mill Level 1 - Metric**

**Create**

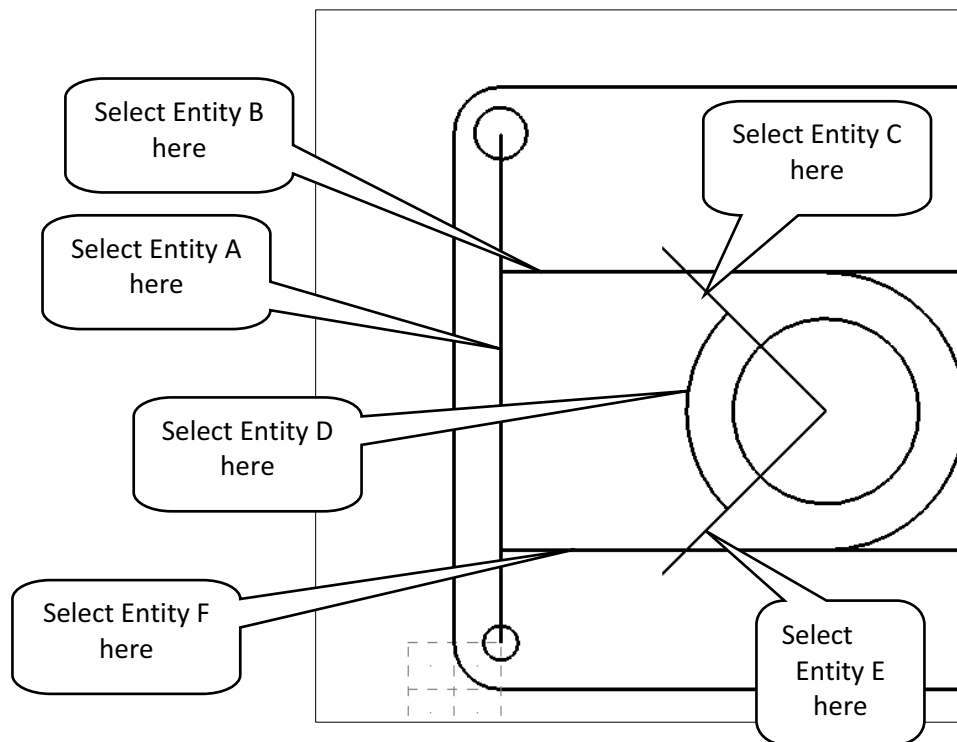
**Fillet**


**Entities**

Enter the fillet **Radius**  3

Enable the **Trim** icon from the Ribbon band. 

- [Fillet: Select an entity]: Select Entity A
- [Fillet: Select another entity]: Select Entity B
- [Fillet: Select an entity]: Select Entity B again
- [Fillet: Select another entity]: Select Entity C
- [Fillet: Select an entity]: Select Entity C again
- [Fillet: Select another entity]: Select Entity D
- [Fillet: Select an entity]: Select Entity D again
- [Fillet: Select another entity]: Select Entity E
- [Fillet: Select an entity]: Select Entity E again
- [Fillet: Select another entity]: Select Entity F
- [Fillet: Select an entity]: Select Entity F again
- [Fillet: Select another entity]: Select Entity A

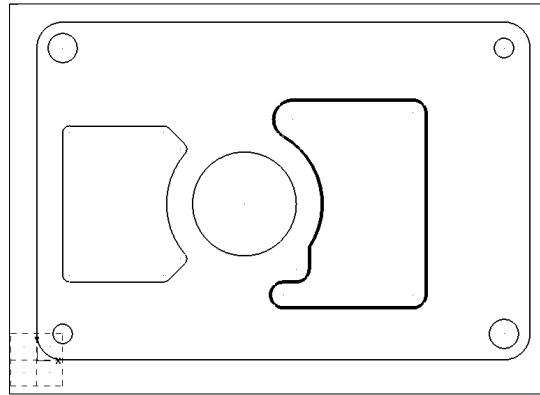


Select the **OK** button to exit fillet command. 

**Mill Level 1 - Metric**

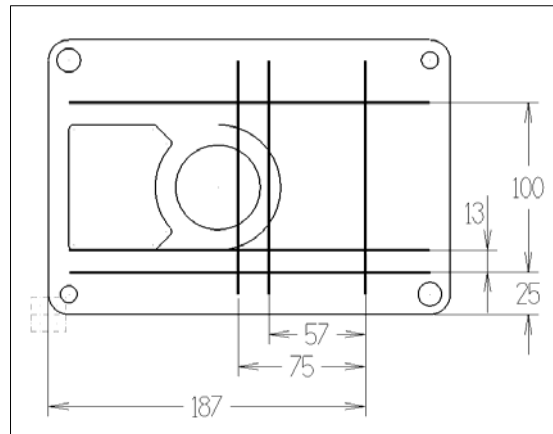
**STEP 5: CREATE THE RIGHT SIDE POCKET**

*Step Preview:*



**5.1 Create parallel lines**



*Sub Step Preview:*

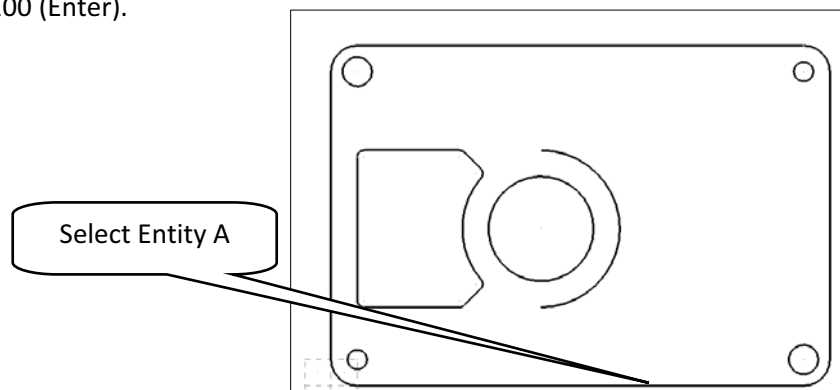


**Create**


➤ **Line**

➤ **Parallel**

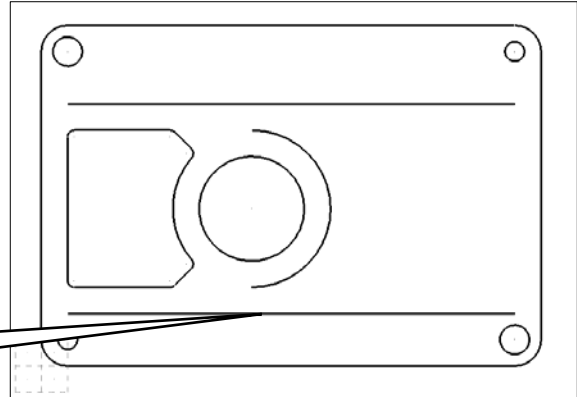
- [ Select a line ]: Select Entity A.
- [ Select the point to place a parallel line through ]: Pick a point above the selected line.
- Type the **Distance**  25 (Enter).
- [ Select a line ]: Select Entity A.
- [ Select the point to place a parallel line through ]: Pick a point above the selected line.
- Type the **Distance**  25 + 100 (Enter).




**Mill Level 1 - Metric**

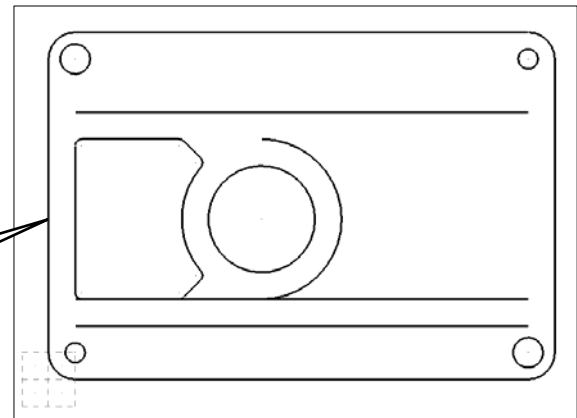
- [ Select a line ]: Select Entity B.
- [ Select the point to place a parallel line through ]: Pick a point above the selected line.
- Type the **Distance**  13 (Enter).


Select Entity B



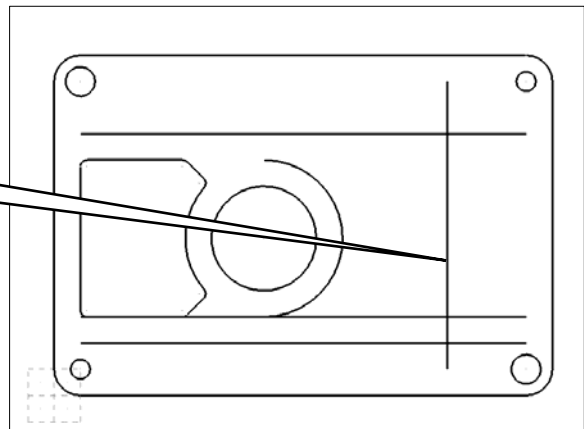
- [ Select a line ]: Select Entity C.
- [ Select the point to place a parallel line through ]: Pick a point to the right of the selected line.
- Type the **Distance**  187 (Enter)



Select Entity C



- [ Select a line ]: Select Entity D
- [ Select the point to place a parallel line through ]: Pick a point to the left of the selected line.
- Type the **Distance**  57 (Enter)

Select Entity D



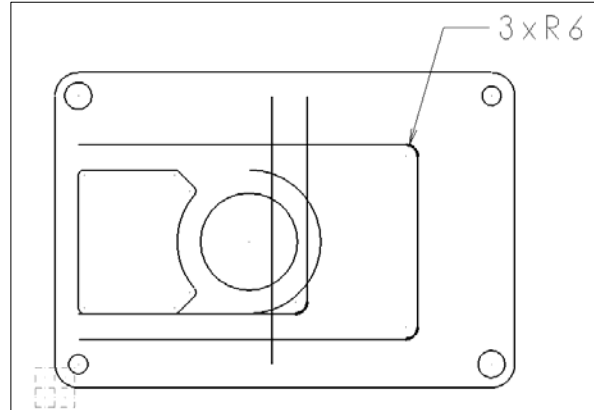
- [ Select a line ]: Select Entity D again
- [ Select the point to place a parallel line through ]: Pick a point to the left of the selected line.
- Type the **Distance**  75 (Enter)
- Select the **OK** button to exit the command. 

**Mill Level 1 - Metric**

**5.2 Creating the 6 mm radius fillets at 3 corners of the right side pocket**

Remember that the fillet command will allow us to automatically trim the geometry.



*Sub Step Preview:*

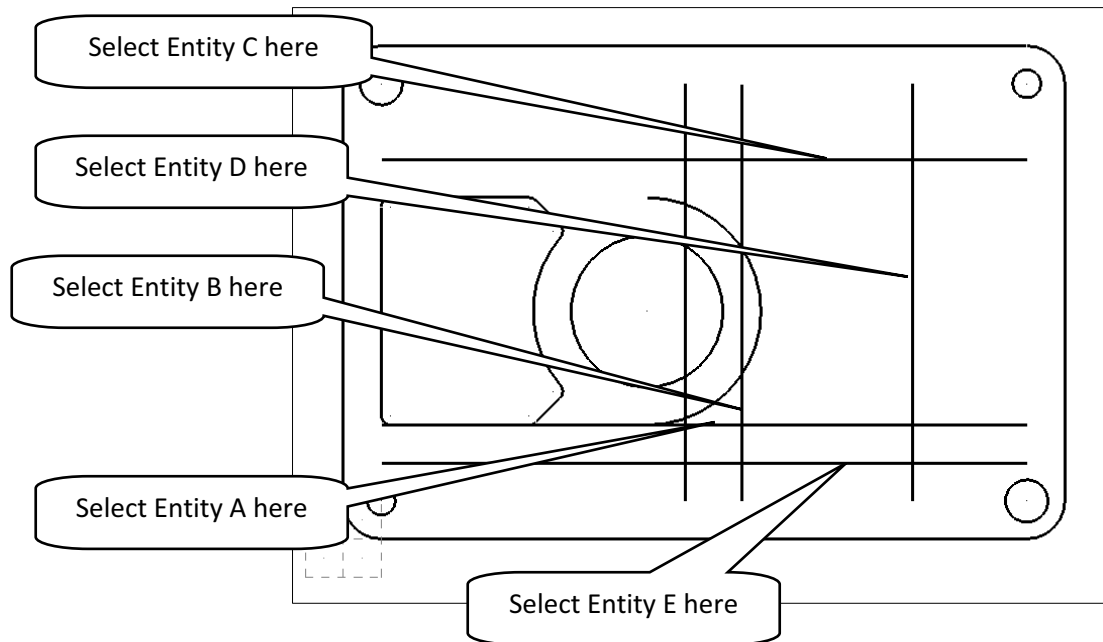


**Create**

**Fillet**

**Entities**

- Enter the fillet **Radius**  6
- Make sure that the **Trim** icon in the Ribbon band is enabled. 
- [Fillet: Select an entity]: Select Entity A
- [Fillet: Select another entity]: Select Entity B
- [Fillet: Select an entity]: Select Entity C
- [Fillet: Select another entity]: Select Entity D
- [Fillet: Select an entity]: Select Entity D again
- [Fillet: Select another entity]: Select Entity E



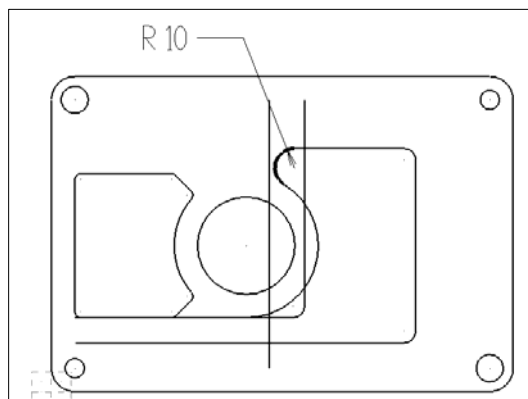
- Select the **Apply** button to finish the last fillet and continue to use the same command. 




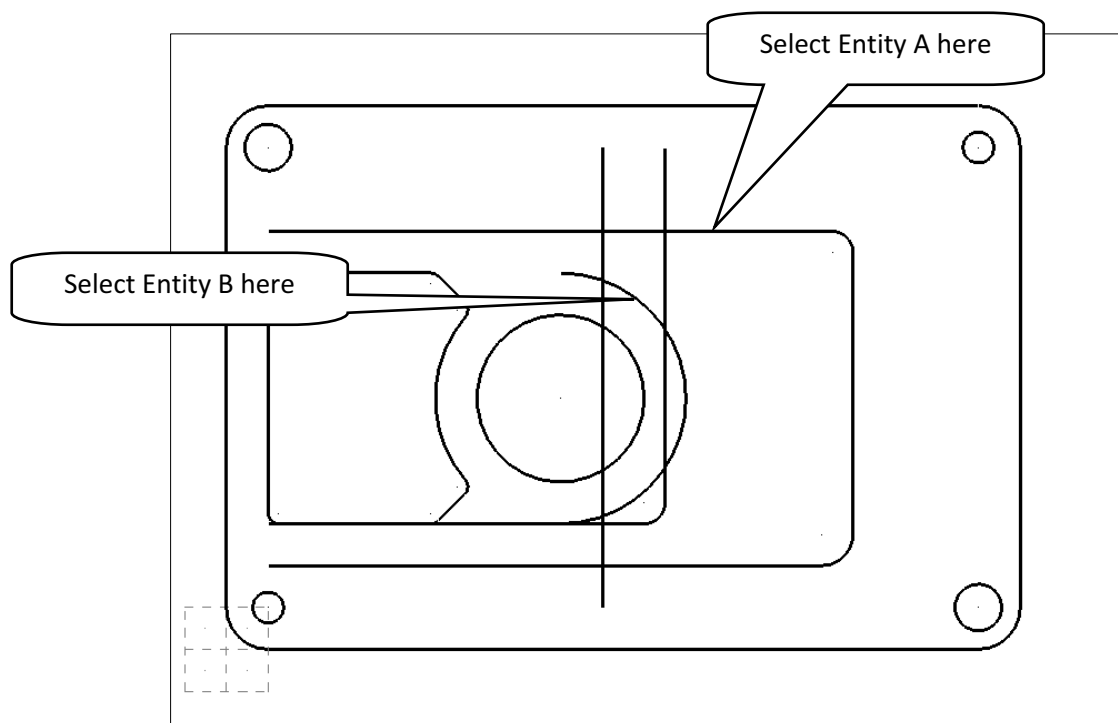
**Mill Level 1 - Metric**

**5.3 Creating the 10 mm radius fillet**

**Sub Step Preview:**



- Change the fillet **Radius**  10
- [Fillet: Select an entity]: Select Entity A
- [Fillet: Select another entity]: Select Entity B

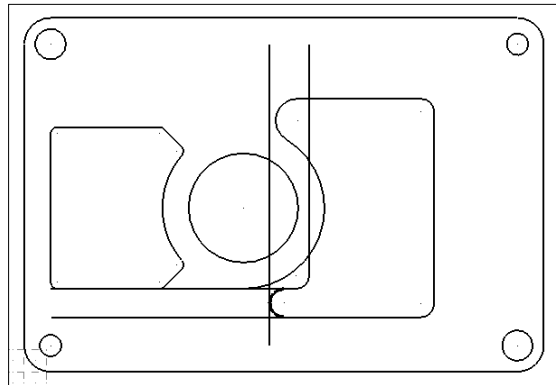


- Select the **OK** button. 

**Mill Level 1 - Metric**

**5.4 Creating an arc tangent to three entities**

**Sub Step Preview:**



**Create**

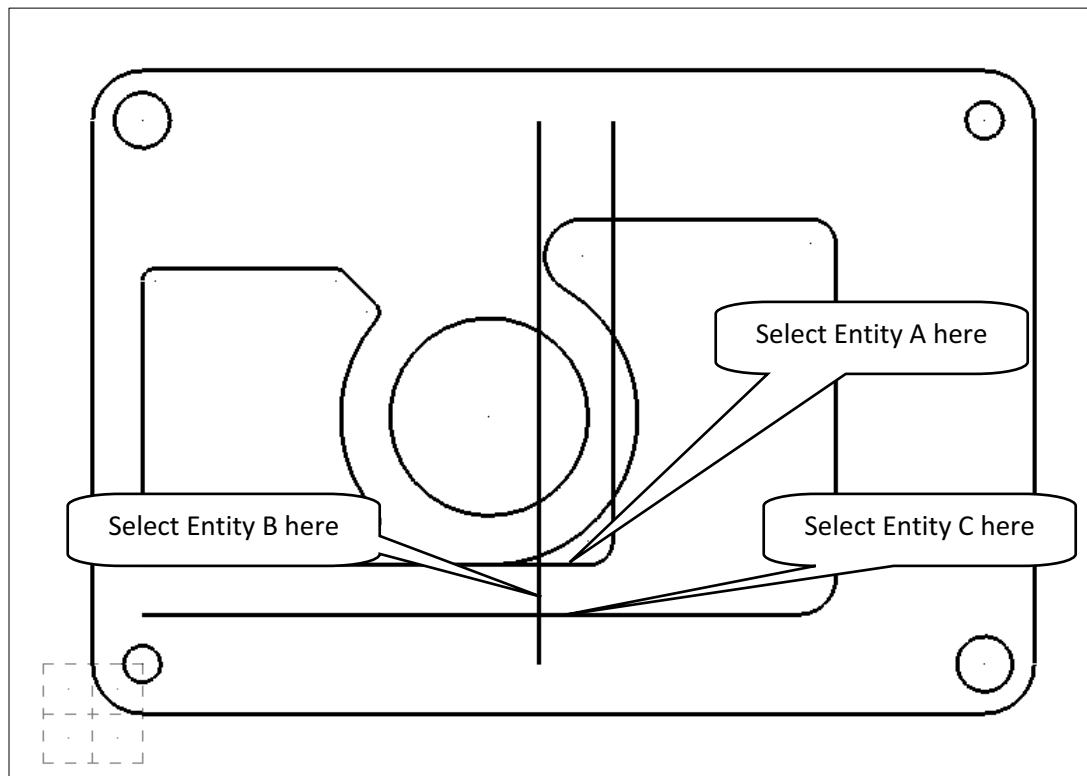
➤ **Arc**

➤ **Arc Tangent**

➤ Enable **Arc Tangent 3 Entities** icon in the **Ribbon Bar**.



- [Select the entity that the arc is to be tangent to]: Select Entity A
- [Select the entity that the arc is to be tangent to]: Select Entity B
- [Select the entity that the arc is to be tangent to]: Select Entity C

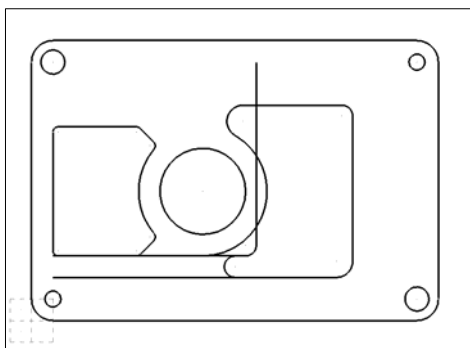


➤ Select the **OK** button. 

## Mill Level 1 - Metric

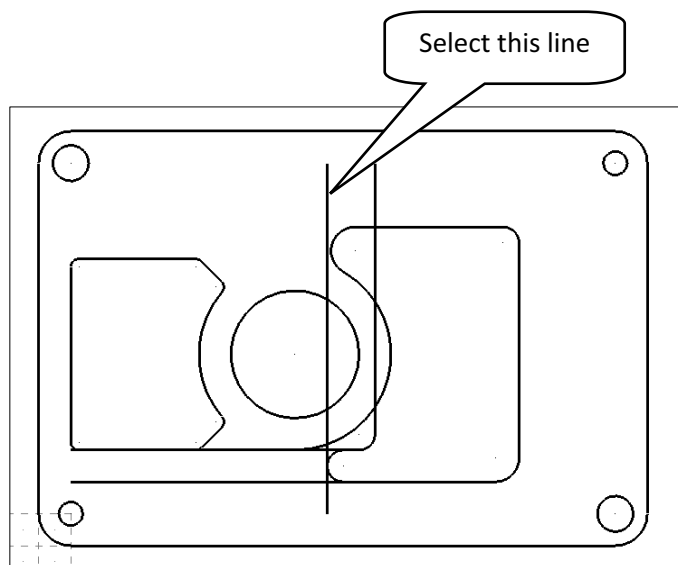
### 5.5 Delete the construction line.

Sub Step Preview:



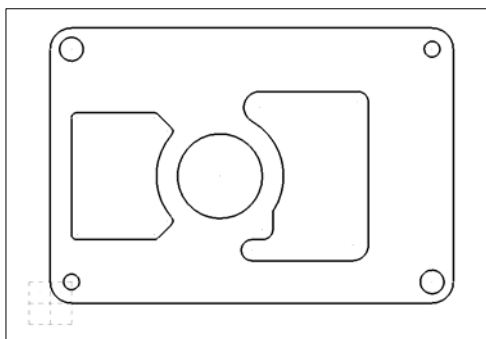
➔ Select the line as shown to the right.

➔ Select the **Delete** entity icon. 



### 5.6 Trim the geometry using Trim 2 entities and Trim 3 entities options

Sub Step Preview:

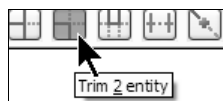


**Edit**

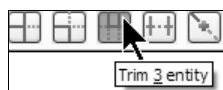
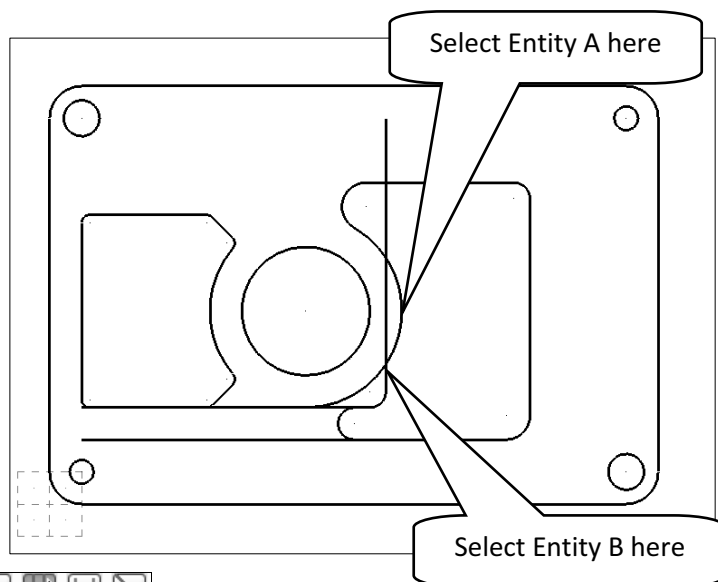
➔ Trim/Break

➔  Trim/Break/Extend

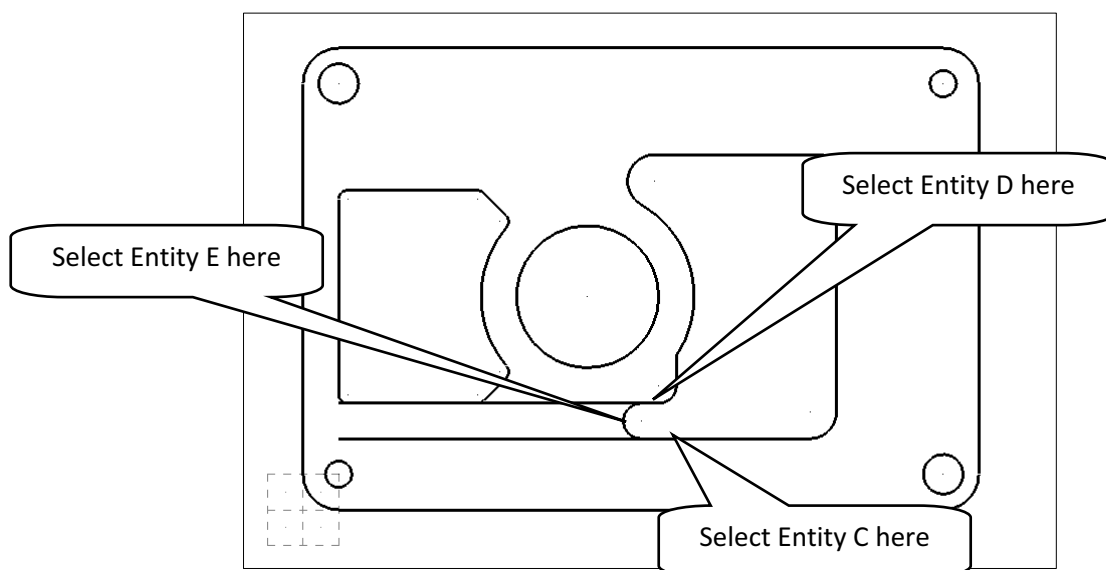
**Mill Level 1 - Metric**



- ➔ Select the **Trim 2 entities** button.
- ➔ [Select the entity to trim/extend]: Select Entity A.
- ➔ [Select the entity to trim/extend to]: Select Entity B.



- ➔ Select the **Trim 3 entities** button.
- ➔ [Select the first entity to trim/extend]: Select Entity C
- ➔ [Select the second entity to trim/extend]: Select Entity D
- ➔ [Select the entity to trim/extend to]: Select Entity E



- ➔ Select the **Repaint** button.

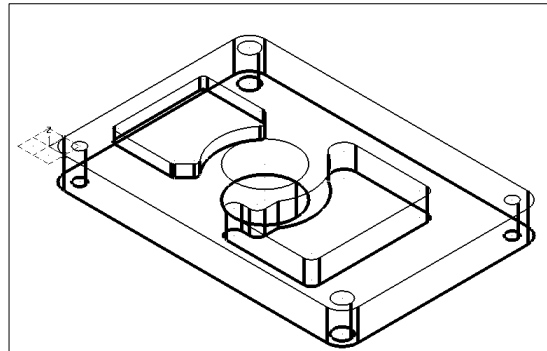


- ➔ Select the **OK** button.



## STEP 6: CREATE THE 3-D WIREFRAME

Step Preview:

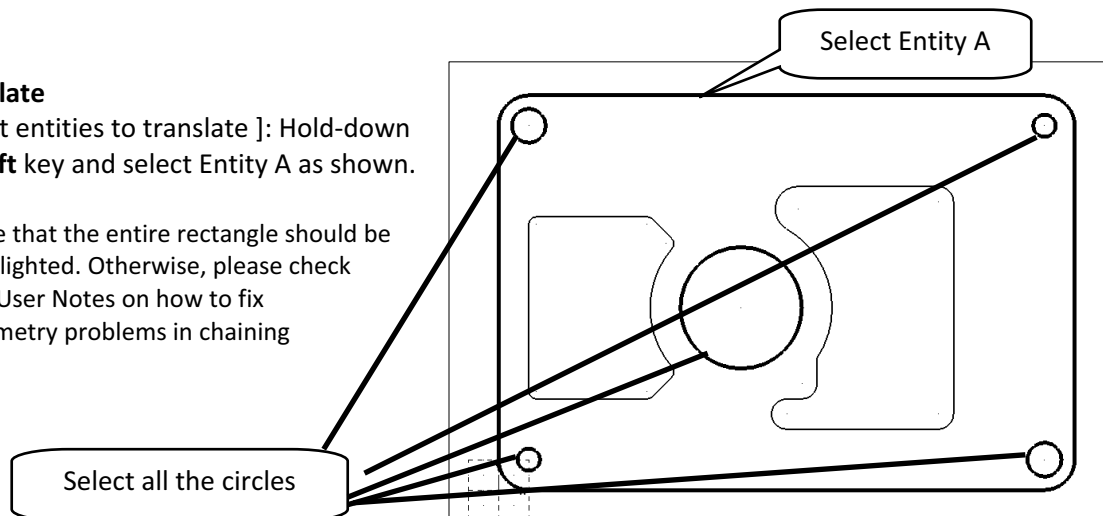


### Xform

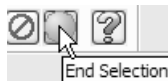
#### Translate

- [ Select entities to translate ]: Hold-down the **Shift** key and select Entity A as shown.

- Note that the entire rectangle should be highlighted. Otherwise, please check the User Notes on how to fix geometry problems in chaining



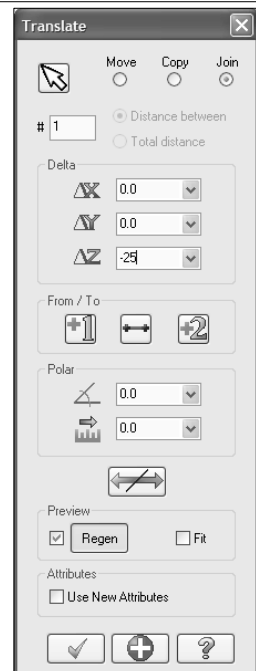
- Select all the circles as shown.



- Select the **End Selection** button.

- Enable **Join**.
- Set the number of translations to # 1.
- Change the **Delta** value on **Z** to -25

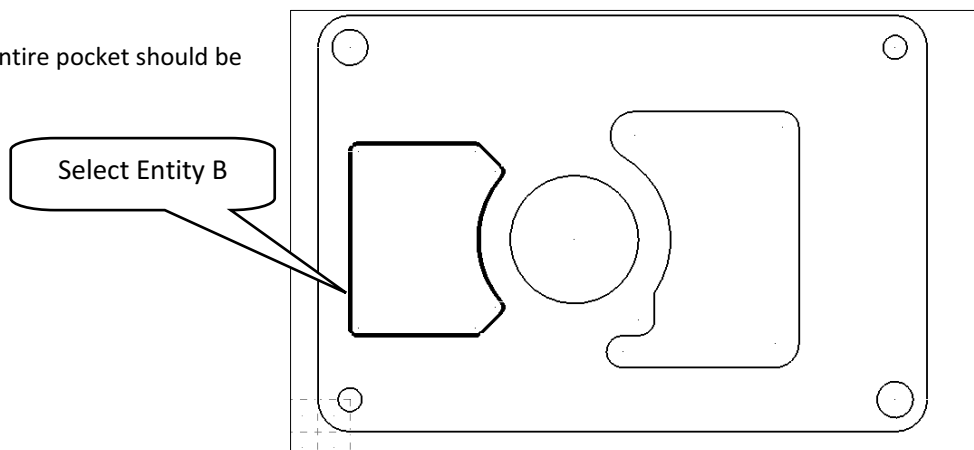
- Select the **Apply** button to continue.



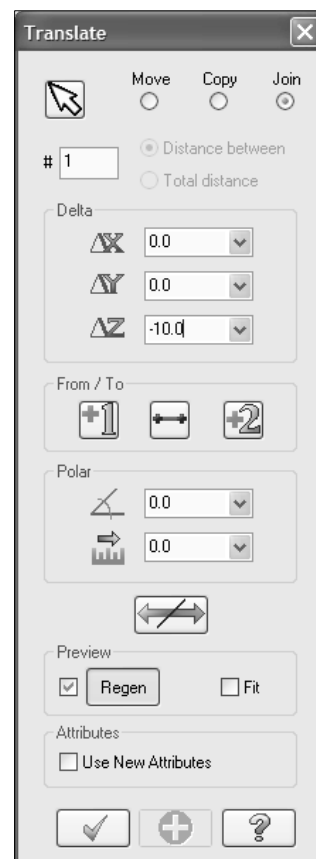
**Mill Level 1 - Metric**

- [ Select entities to translate ]: Hold-down the **Shift** key and select Entity B.

- Note that the entire pocket should be highlighted.



- Select the **End Selection** button.
- Enable **Join**.
- Set the number of translations to **# 1**.
- Change the **Delta** value on **Z** to **-10**.



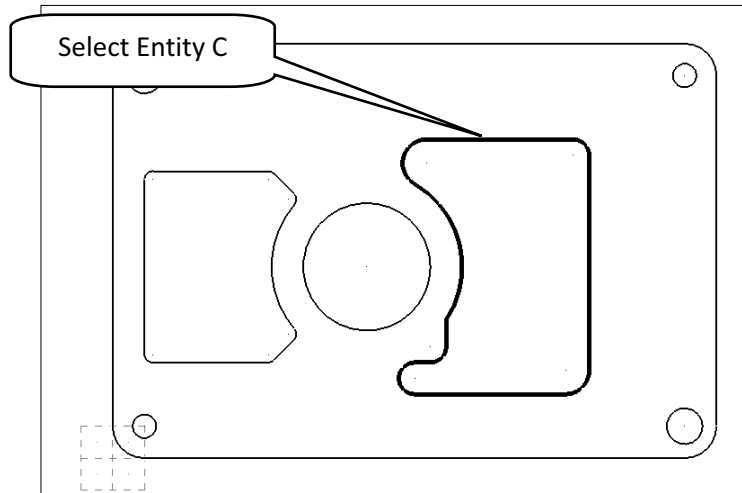
- Select the **Apply** button to continue.



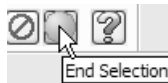
**Mill Level 1 - Metric**

➤ [ Select entities to translate ]: Hold-down the Shift key and select Entity C.

➤ Note that the entire pocket should be highlighted.



➤ Select the **End Selection** button.



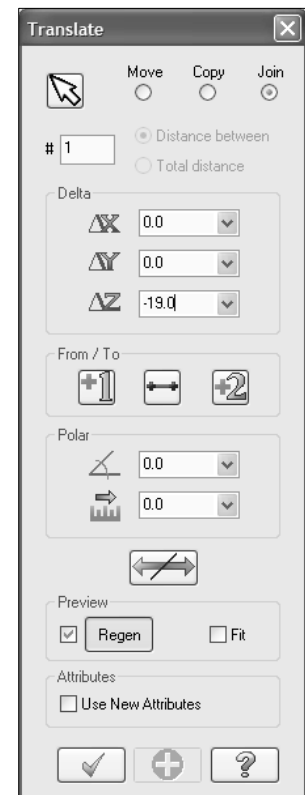
➤ Change the **Delta** value on **Z** to -19.

➤ Select the **OK** button to exit.

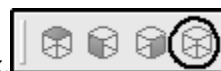


**Screen**

➤  **Clear colors**



➤ Select the **Isometric View** from the view toolbar to see the stock.

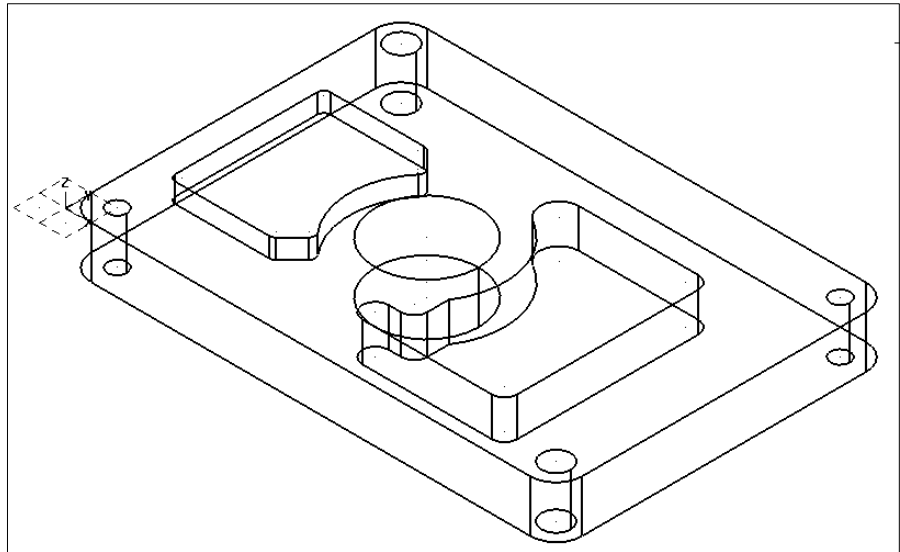


➤ Select **Fit** button.



**Mill Level 1 - Metric**

*The final geometry should look as shown below*

**STEP 7: SAVE THE FILE****File**

➤ Save as

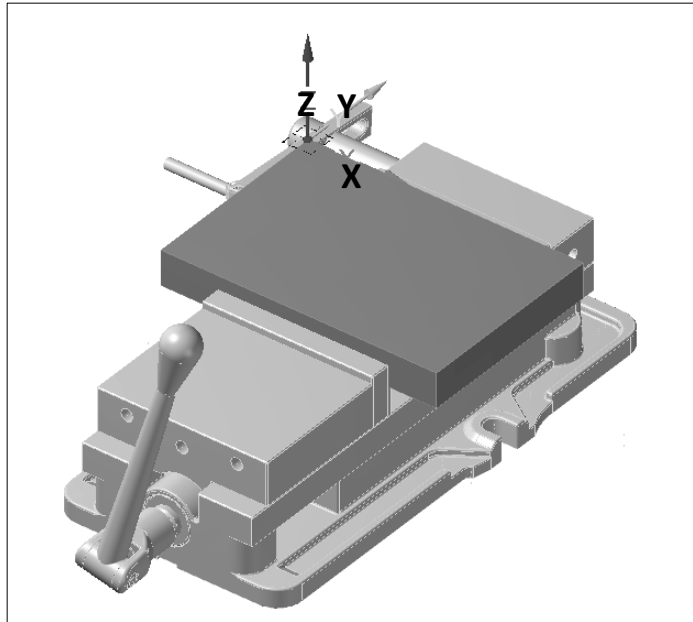
➤ File name: "Your Name\_7"

➤ Select the OK button. 








## TOOLPATH CREATION

### SUGGESTED FIXTURE FOR SETUP 1:




### SETUP SHEET FOR SETUP 1:

Tool List of TUTORIAL7.MCX																							
Proj./Part No.: 0		Date : 07/27/09		Customer : -		Programmer : 0																	
Drawing No. : 1																							
Prog. No. : 7																							
	Tool type :	75 Face mill	75 Face Mill																				
	Manufact.code :																						
	Chuck :																						
	Tool Number :	1			Feedrate :	8.0157																	
	Diameter :	75	RPM :	509	Plunge feed r.:	4.0079																	
	Corner radius :	0	Tip angle :	45	Diam. offset :	1																	
	Flute length :	10	Material :	ALUMINUM ...	Length offset :	1																	
	Overall length:	50	No flutes :	4																			
	Tool type :	20 Spot Drill	20. SPOT DRILL																				
	Manufact.code :																						
	Chuck :																						
	Tool Number :	2			Feedrate :	9.0157																	
	Diameter :	20	RPM :	1145	Plunge feed r.:	9.0157																	
	Corner radius :	0	Tip angle :	90	Diam. offset :	2																	
	Flute length :	50	Material :	ALUMINUM ...	Length offset :	2																	
	Overall length:	75	No flutes :	2																			
	Tool type :	12 Drill	12. DRILL																				
	Manufact.code :																						
	Chuck :																						
	Tool Number :	3			Feedrate :	4.507																	
	Diameter :	12	RPM :	954	Plunge feed r.:	4.507																	
	Corner radius :	0	Tip angle :	118	Diam. offset :	3																	
	Flute length :	50	Material :	ALUMINUM ...	Length offset :	3																	
	Overall length:	75	No flutes :	2																			
	Tool type :	14 Tap RH	14.00-2. TAP RH																				
	Manufact.code :																						
	Chuck :																						
	Tool Number :	4			Feedrate :	908																	
	Diameter :	14	RPM :	454	Plunge feed r.:	255.8036																	
	Corner radius :	0	Tip angle :	180	Diam. offset :	4																	
	Flute length :	50	Material :	ALUMINUM ...	Length offset :	4																	
	Overall length:	75	No flutes :	1																			
	Tool type :	10 Drill	10. DRILL																				
	Manufact.code :																						
	Chuck :																						
	Tool Number :	5			Feedrate :	5.4094																	
	Diameter :	10	RPM :	1145	Plunge feed r.:	5.4094																	
	Corner radius :	0	Tip angle :	118	Diam. offset :	5																	
	Flute length :	50	Material :	ALUMINUM ...	Length offset :	5																	
	Overall length:	75	No flutes :	2																			

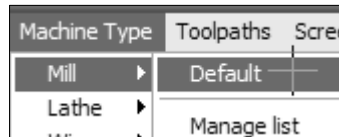
**Mill Level 1 - Metric**

**STEP 8: SELECT THE MACHINE AND SET UP THE STOCK TO BE MACHINED**

- ➡ To display the **Operations Manager** press **Alt + O**.
- ➡ Use the **Fit** icon to fit the drawing to the screen. 
- ➡ If a machine is already selected in the **Toolpaths Manager**, skip the machine selection step.

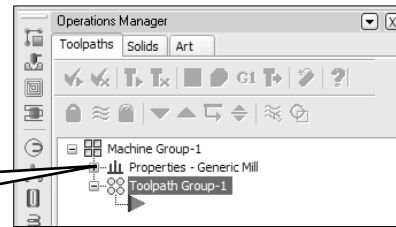
**Machine type**

- ➡ **Mill**
- ➡ **Default**



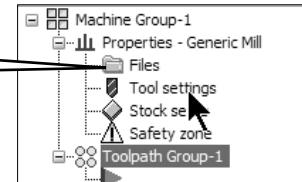
- ➡ Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus

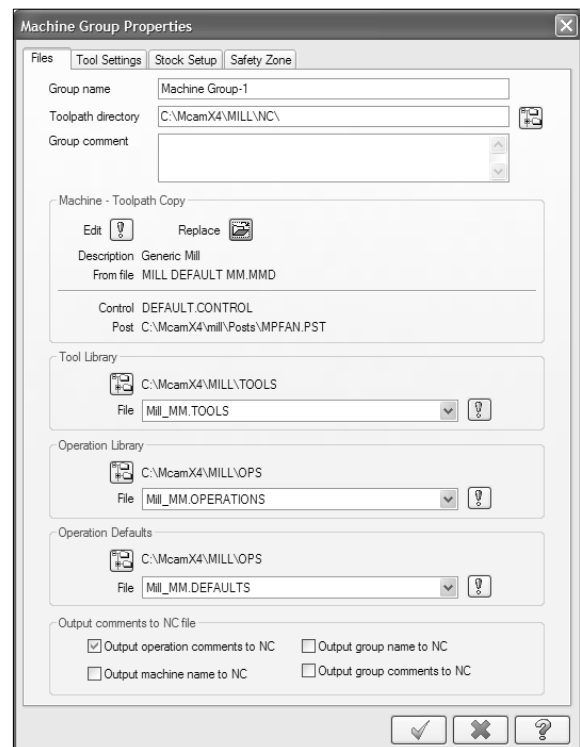


- ➡ Select **Files**.

Select Files

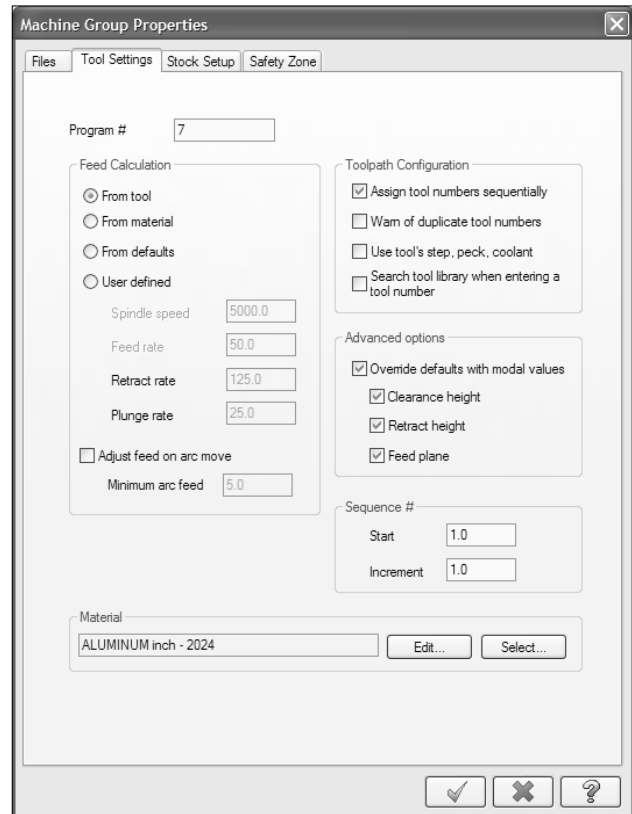


- ➡ Make sure that the machine definition is the **MILL DEFAULT MM.MMD** as shown.
- ➡ Otherwise, follow the instructions from **Tutorial 1** page 1-12 to 1-14 to replace it with the proper metric machine.
- ➡ All the other files used by this machine group are metric files as well.

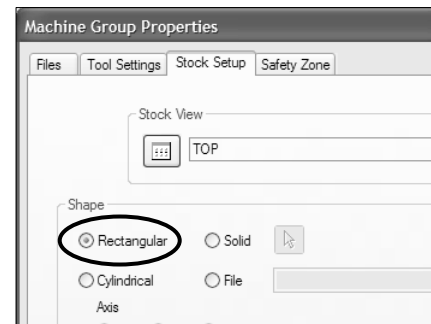


**Mill Level 1 - Metric**

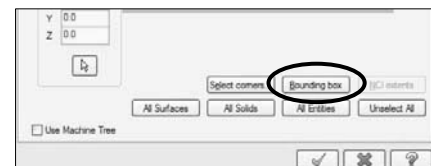
- ➡ Select **Tool Settings** to set the tool parameters
- ➡ Change the parameters to match the screenshot below.



- ➡ Select **Stock Setup** tab.
- ➡ Make sure that the **Shape** is set to **Rectangular**.



- ➡ Select the **Bounding box** button for the system to automatically find the part's overall dimensions.

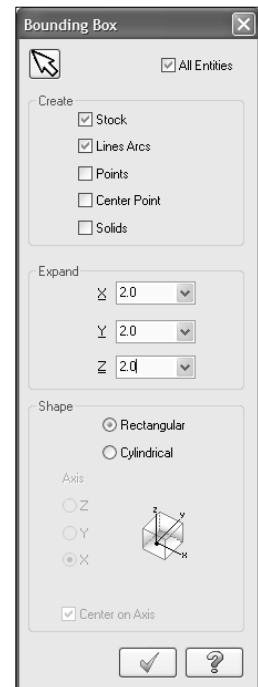


### Mill Level 1 - Metric

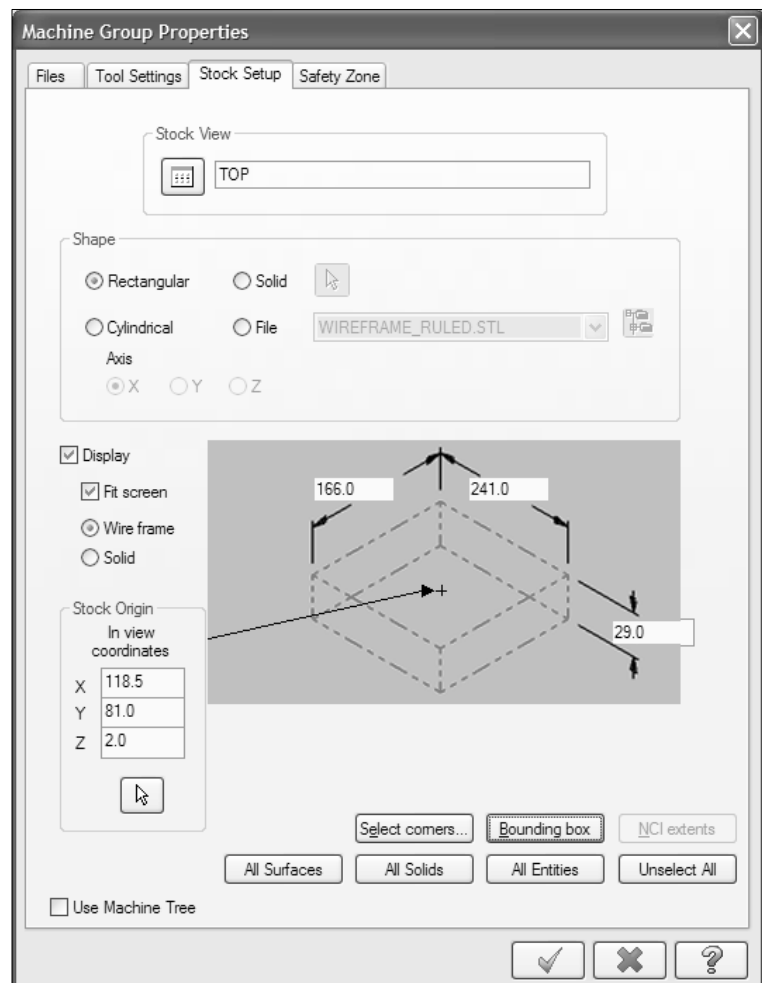
➔ Enable **Lines Arcs** to create the geometry of the stock.

➔ Expand X, Y and Z values with 2

➔ Select the **OK** button to exit the **Bounding Box** window.



➔ Select the **OK** button to exit **Machine Group Properties**.



**Mill Level 1 - Metric**

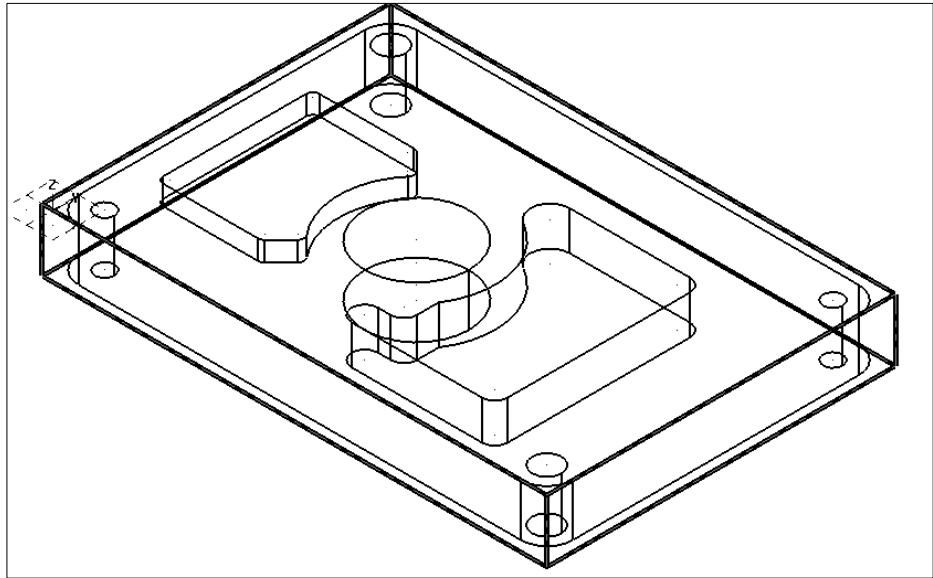
- Select the **Isometric View** from the view toolbar to see the stock.



- Select **Fit** button.



The stock should look as shown below.

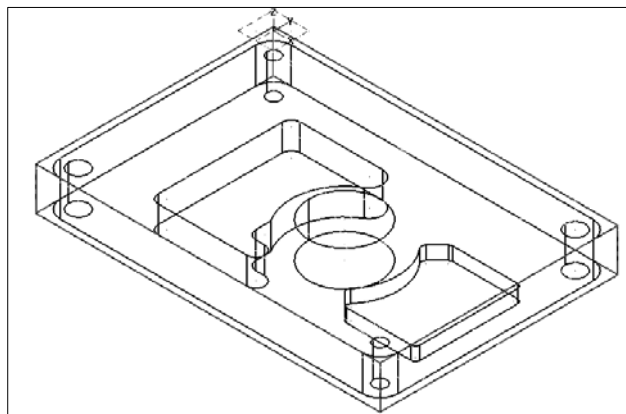


## STEP 9: USING WCS MANAGER FLIP THE PART. MACHINE THE BOTTOM OF THE PART FOR AN EVEN SURFACE FOR FUTURE OPERATIONS

Use the **View Manager** as a central point for selecting, editing, creating and managing views. You can select which views to use for the Cplane, Tplane, or the work coordinate system (WCS). You can also set a new origin of a view and assign a new work offset (G54, G55...) to a view.

### 9.1 Using WCS (Work Coordinate System) to define the new view

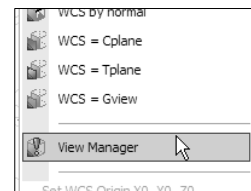
*Step Preview:*



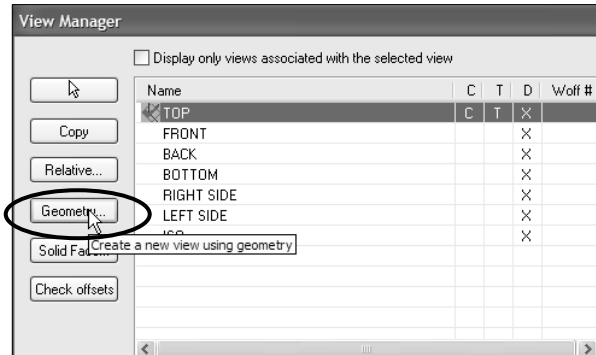
- Select **WCS** in the **Status Bar**.



- Select **View Manager**.



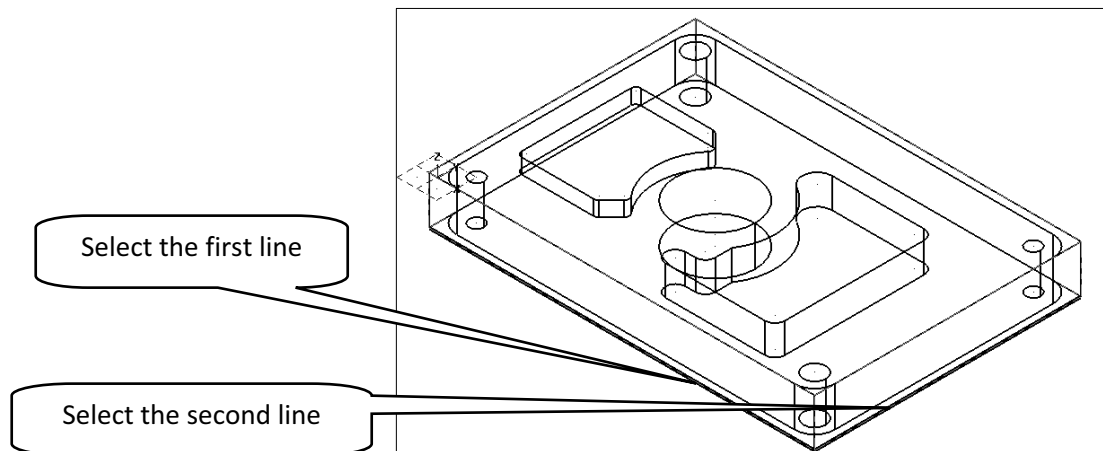
- Select **Geometry** button.



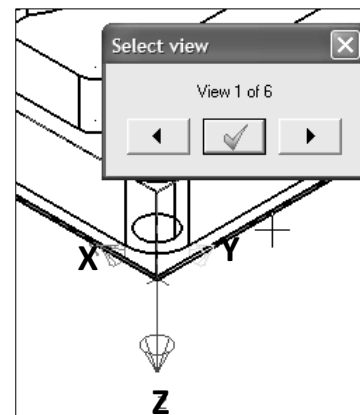
- You can define a new view based on existing geometry like 2 lines, a flat entity (circle or flat surface) or 3 points.


- Select the lines as shown below.

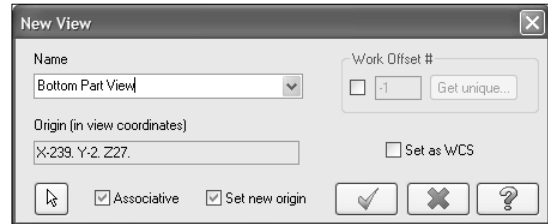
- Note that the first line that you select will determine the X axis orientation of the future view, and the second line will determine Y axis. Z axis should always be orientated towards the outside of the part.




- Select the **OK** button to accept the view.

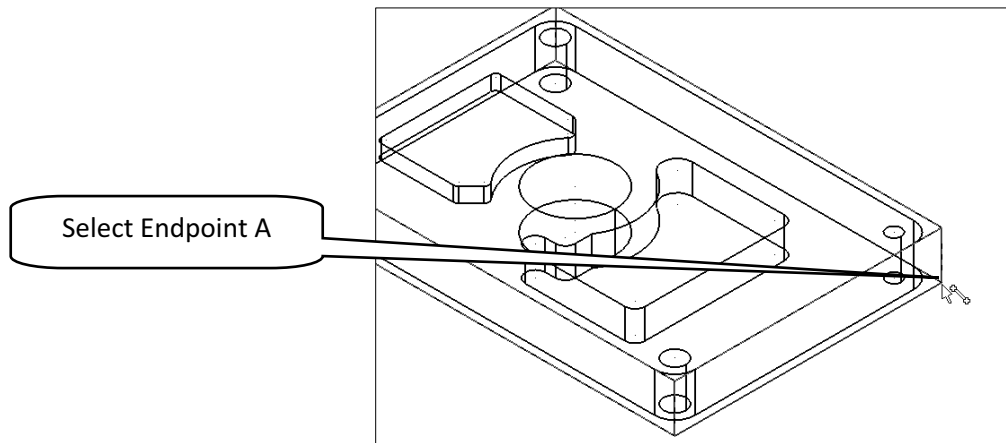
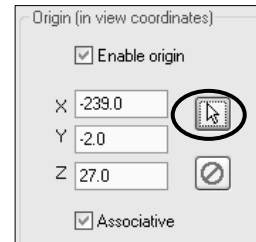


- ➡ Enter the **New View Name**: "Bottom Part View".
- ➡ Make sure that **Associative** and **Set new origin** are enabled.
- ➡ Select the **OK** button to exit **New View**. 

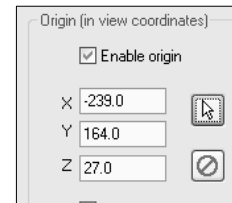


### 9.2 Using WCS to set a new origin for the view

- ➡ Click on **Select** button to select the new **Origin** from the graphic area. 
- ➡ [Select a point]: Select Endpoint A as shown.




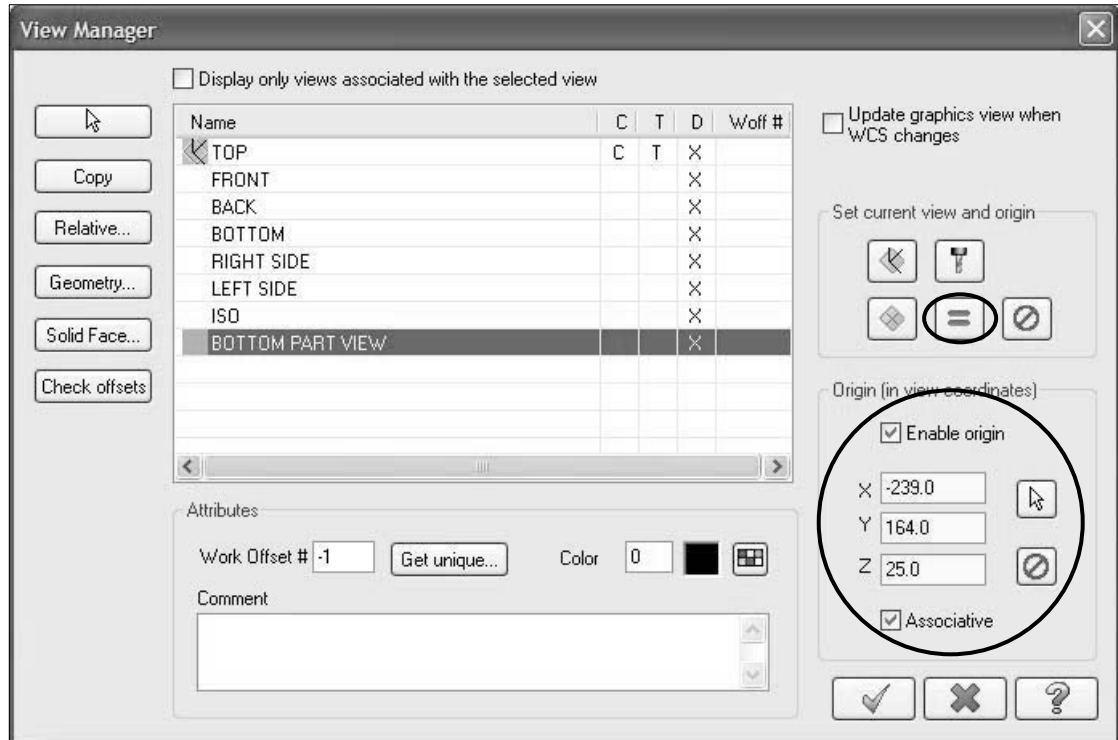
- ➡ The new origin values should look as shown.



**Mill Level 1 - Metric**




**9.3 Using WCS to set the new view as the current tool plane, construction plane and WCS**

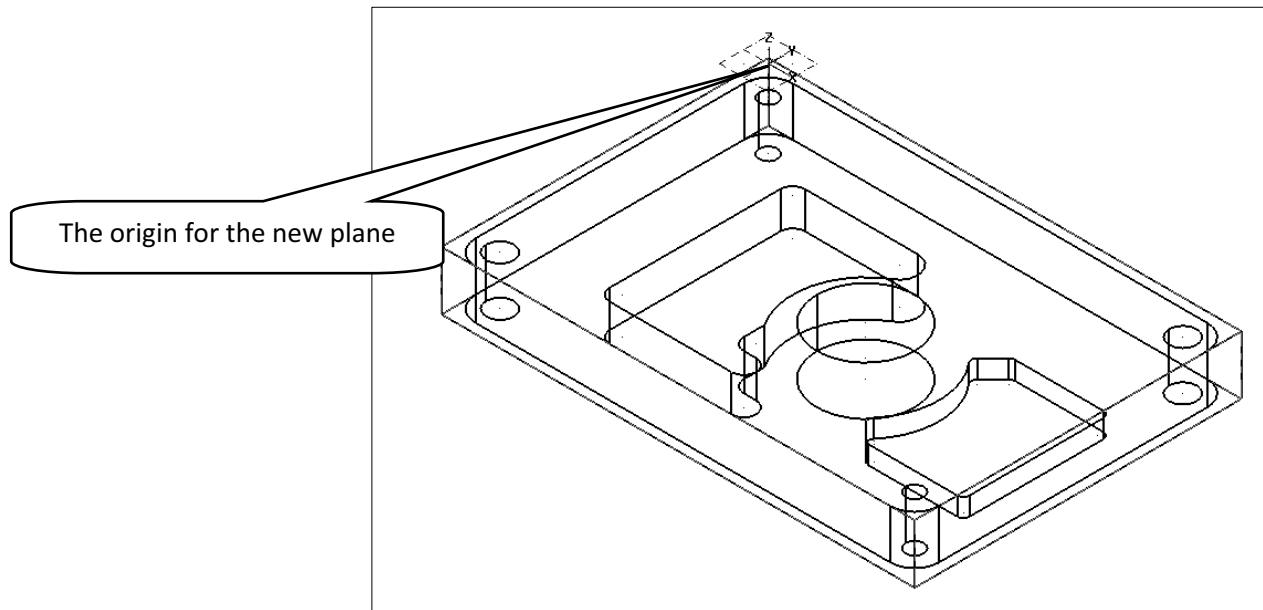
- ➔ Select **Bottom Part View** and click on the **Set your current WCS, construction plane and tool plane with their origins to the selected view** button. 
- ➔ Change the **Z** value in the **Origin** field to **25** to set **Program zero** at the finish size not at the top of the stock.





**Mill Level 1 - Metric**

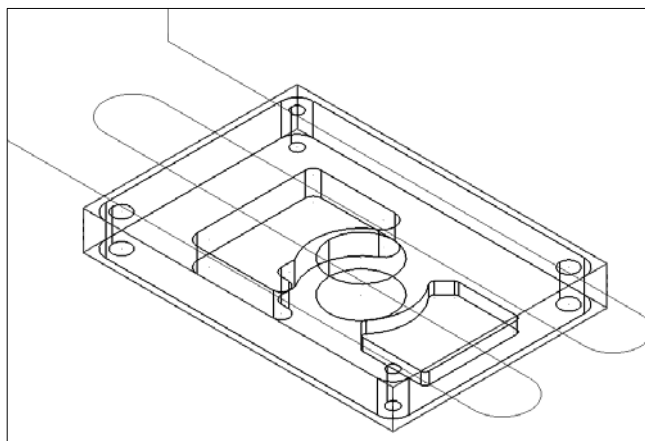
- Select the **OK** button to exit. 
- Select the Isometric view. 
- Select **Fit** button. 
- The part should look as shown below.



**STEP 10: FACE THE BOTTOM OF THE PART TO CREATE AN EVEN SURFACE FOR FUTURE OPERATIONS**

- Facing toolpath is used to quickly clean the stock from the top of a part, and create an even surface for future operations. You can base the toolpath on either chained geometry or on the current stock model.

*Toolpath Preview:*





### Mill Level 1 - Metric

#### 10.1 Selection of the top of the part

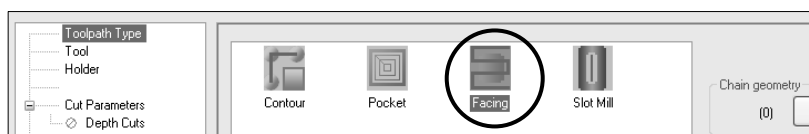
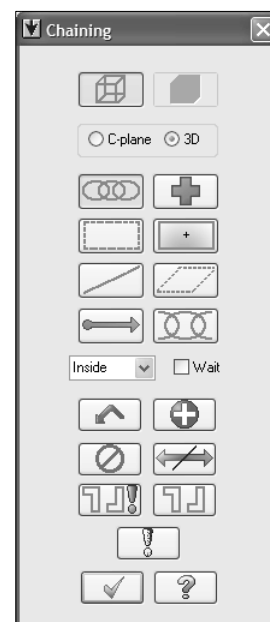
##### Toolpaths

##### ➔ Face

- ➔ Select the **OK** button to accept the NC file name. 

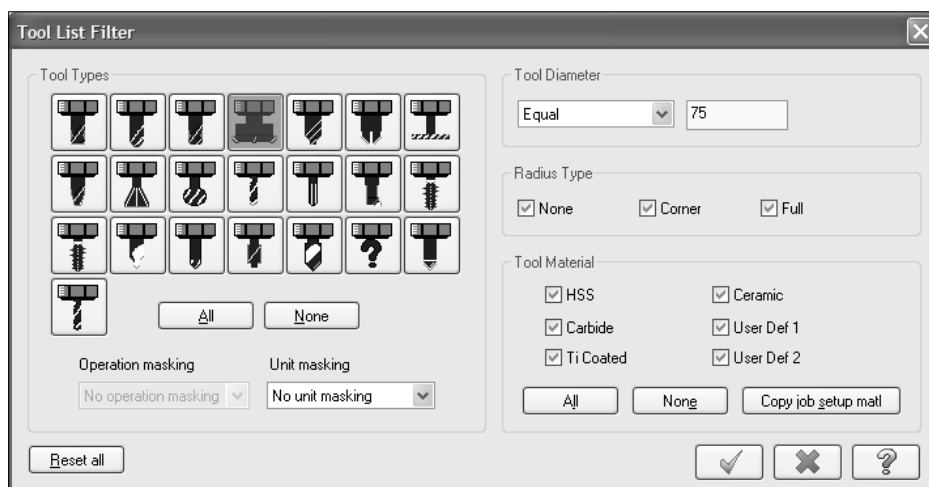
- ➔ [ Select OK to use defined stock or select chain1 ]: Select the **OK** button to use defined stock. 

- ➔ Note that in the **Toolpath Type** the **Facing** icon is already selected.





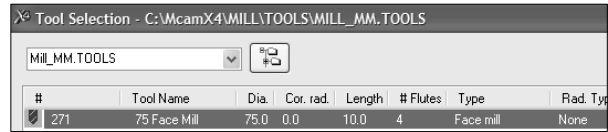
#### 10.2 Select a 75 mm diameter Face mill

- ➔ From the **Tree view list**, select **Tool**.
- ➔ Click on **Select library tool**. 
- ➔ Select the **Filter** button and make the changes as shown below.



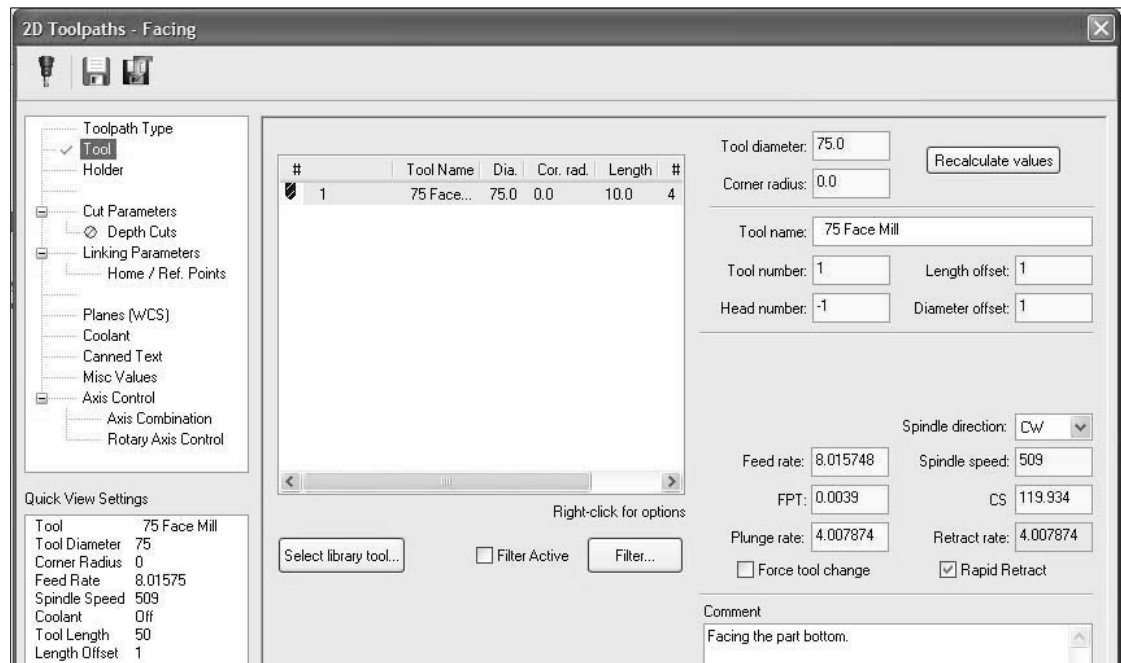
**Mill Level 1 - Metric**

- ➔ Select the **OK** button. 
- ➔ Highlight the 75 mm Face Mill in the **Tool Selection** window.
- ➔ Select the **OK** button. 



**10.3 Set the Tool parameters**

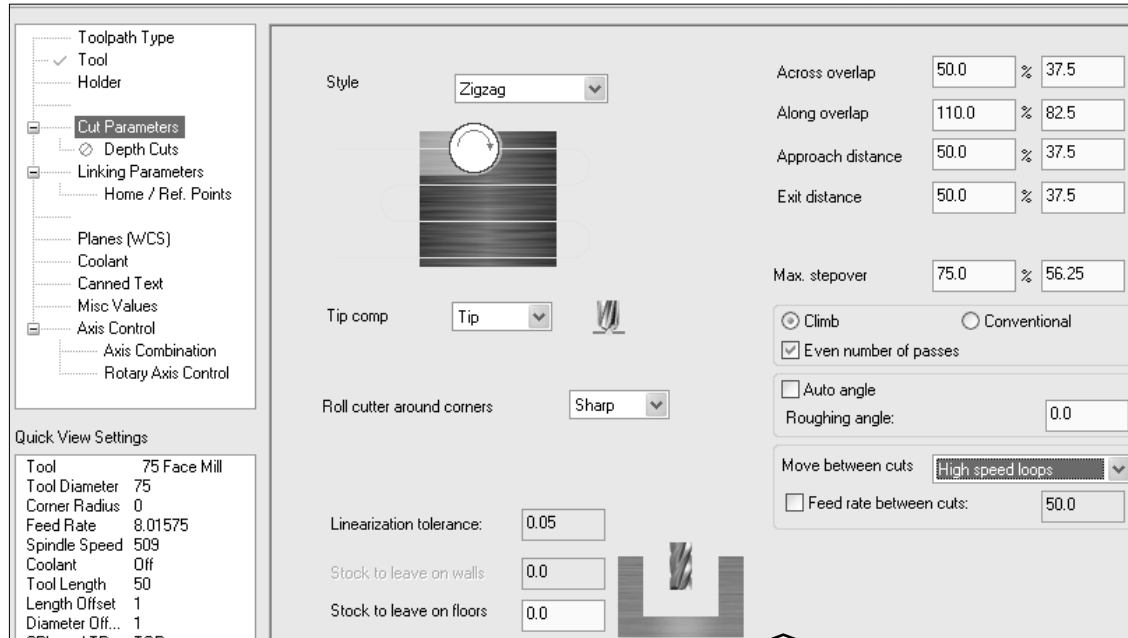
- ➔ Change the parameters in the **Tool** page as shown.



- ➔ Note that the **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition. Change them as needed.

## 10.4 Set the Cut Parameters page

- ➔ From the **Tree view list**, select **Cut Parameters** and change the parameters as shown.



**Cutting method** set to **Zigzag** for back and forth motion.

**Across overlap** sets the overlap amount for the edges of the stock perpendicular to the cut direction.

**Along overlap** sets the overlap amount for the edges of the stock parallel to the cut direction.

**Approach/Exit distance** sets an additional distance away from the stock on the first/last pass.

**Maximum stepover** set the distance between adjacent passes in the toolpaths.

**Even number of passes** (available only with Zigzag cutting method) sets an even number passes which ensures that the last pass reverses the cutting direction. This strategy prevents burring or chipping on the last pass.

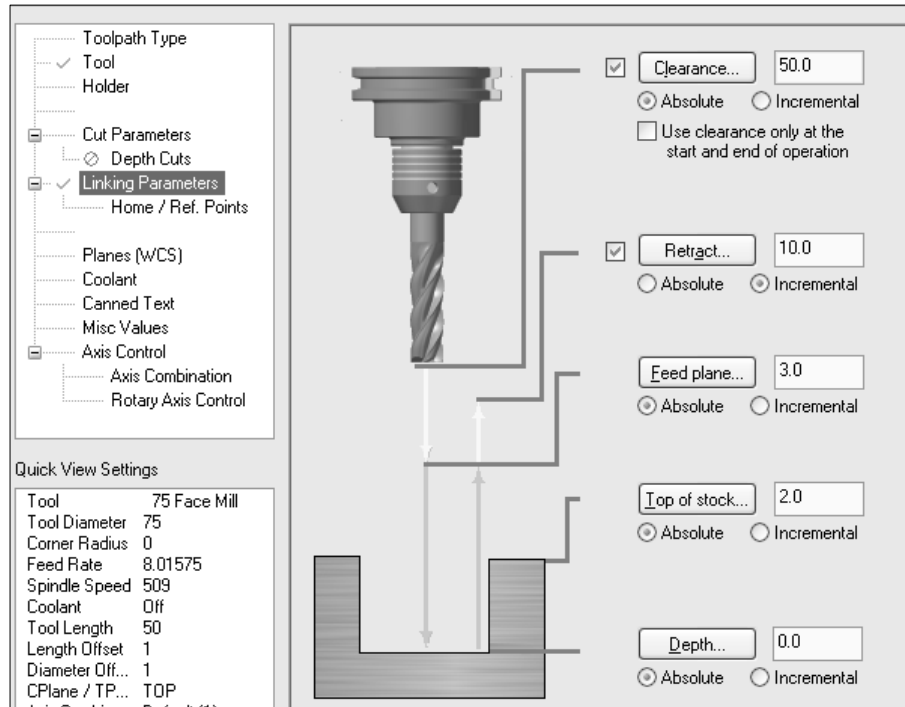
**Roughing angle** set to 0 will make the first pass along positive X axis.

**Move between cuts** set to **High speed loops** creates 180 degrees arcs between each cut.

**Mill Level 1 - Metric**

**10.5 Set the Linking Parameters page**

- ➔ From the **Tree view list**, select **Linking Parameters** and change the parameters as shown.



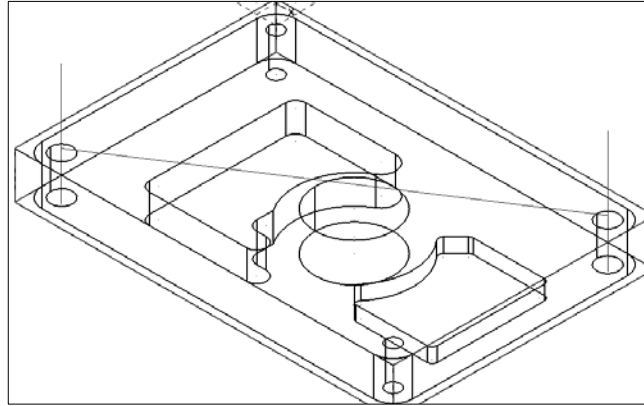
- ➔ Note that the **Depth** is set to absolute and **0** and the Top of stock is set to 2 mm. This is the result of setting up Z 0 to the top of the finish size while defining the Bottom View.

- ➔ Select the **OK** button to exit **2D Toolpaths – Facing**.

- ➔ Press **Alt+ T** to remove the toolpath display.

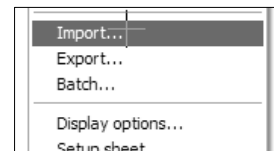
## STEP 11: IMPORT THE OPERATIONS FOR THE M14 X 2 TAP HOLES FROM THE LIBRARY

*Toolpath Preview:*

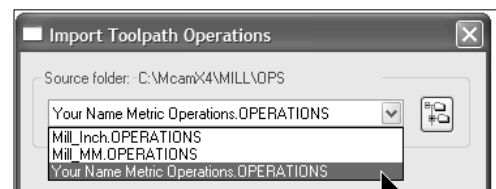


### 11.1 Import the operations

- ➡ **Right-mouse click in Toolpaths Operations Manager window.**
- ➡ **Select **Import...****

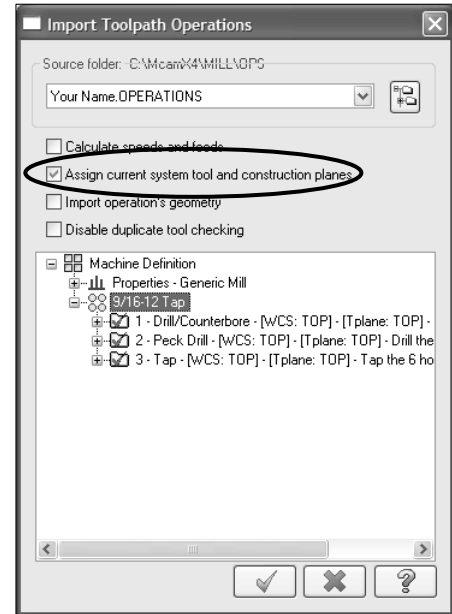


- ➡ **Select the drop-down arrow in the **Source** folder and select the **Your Name Metric Operations**.**



## Mill Level 1 - Metric

- ➡ Click on the **M 14 X 2 Tap** group to select all three operations.
- ➡ Make sure that **Assign current system tool and construction planes** is enabled. This option adjusts the imported operations to use the current tool plane and construction plane.



- ➡ Select the **OK** button. 

- ➡ Select the **NO** button to import the operation group.



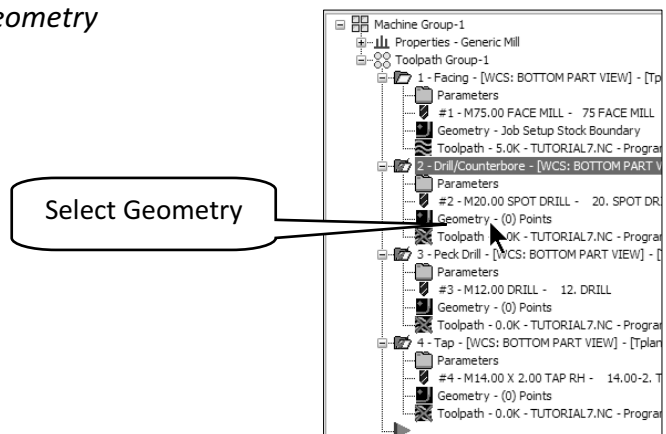
- ➡ Select the **OK** button.

- ➡ Close the **Import Toolpath Operations** dialog box. 

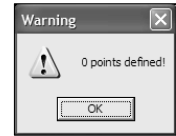
- ➡ Note that all the imported operations have no geometry. We are going to add the center points and regenerate the toolpaths as shown in the following steps.

### 11.2 Add the center points in the toolpath geometry

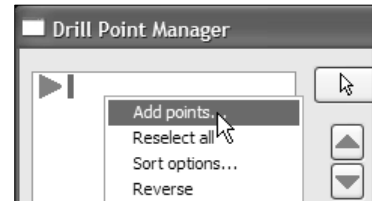
- ➡ Select the first drill operation, **Geometry**.



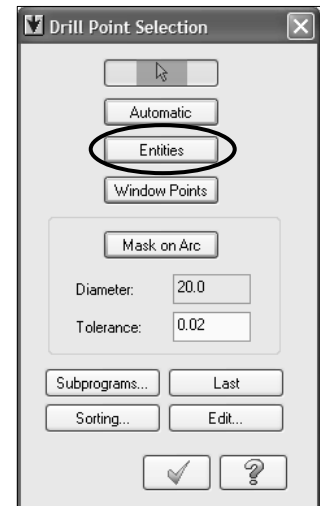
- ➡ Select the **OK** button.



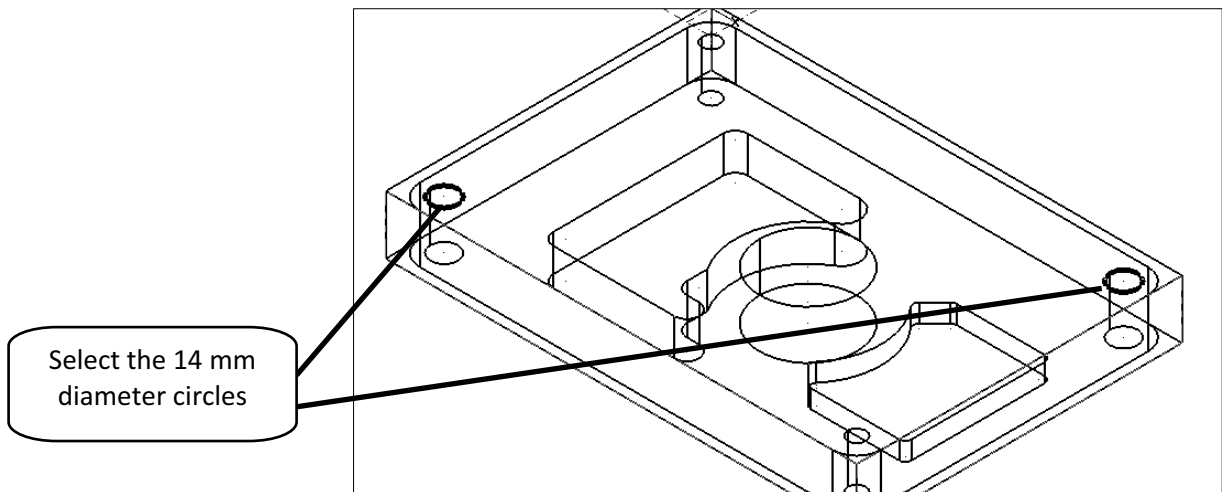
- ➡ Right-mouse click and Select **Add points...**



- ➡ Select the **Entities** button in the **Drill Point Selection**.



- ➡ [Select entities]: Select the 14 mm diameter circles as shown.

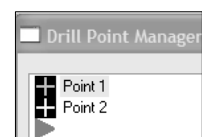


- ➡ Select the **OK** button to exit **Drill Point Selection**.



- ➡ Note that two points are selected in the **Drill Point Manager**.

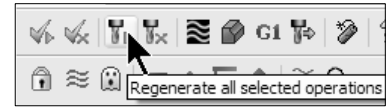
- ➡ Select the **OK** button to exit **Drill Point Manager**.





**Mill Level 1 - Metric**

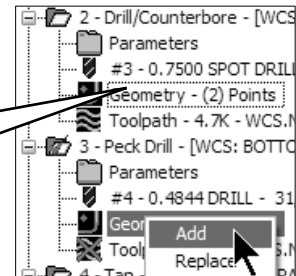
- Select the **Regenerate all selected operations** icon for the change to be applied only to this operation.



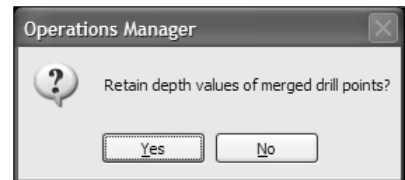
- Select **Alt +T** keys to remove the toolpath display.
- Holding the **right-mouse** button, drag the new **Geometry (2) Points** on top of Peck drill Geometry.

- Release and select **Add**.

Right-mouse click on Geometry, hold down and drag



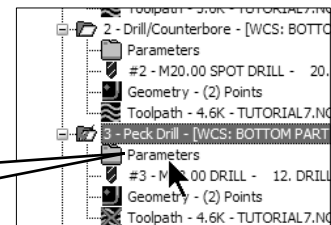
- Select **No** button, otherwise the holes will be drilled at the same depth as the depth of the spot drilling operation.



### 11.3 Modify the drill parameters

- Select the **Peck Drill Parameters** in the **Toolpaths Manager**.

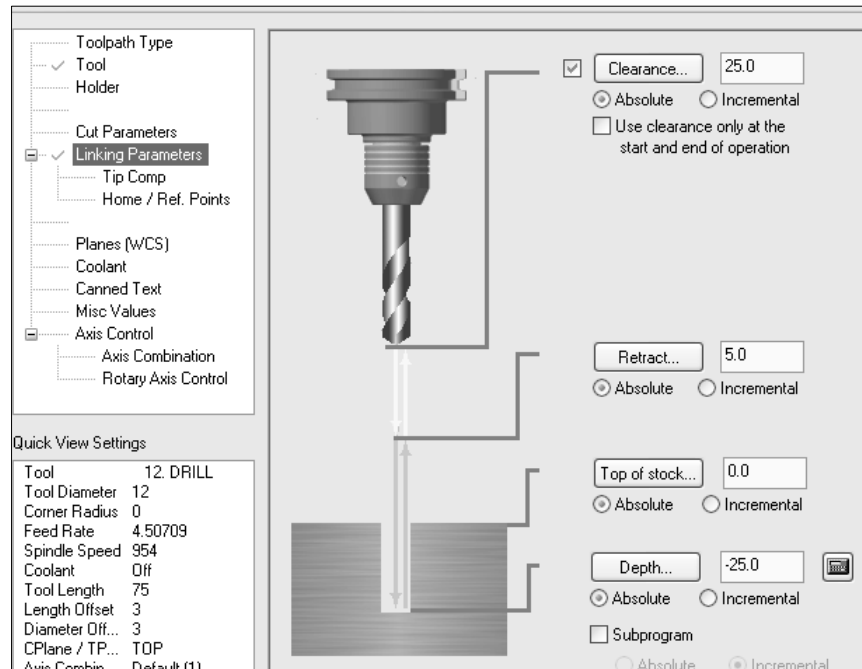
Select the Parameters



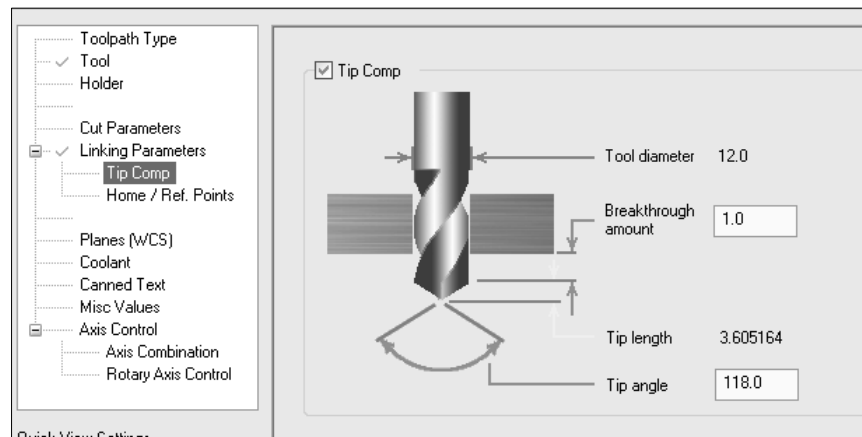
- From the **Tree view list**, select **Tool** and change the **Comment** to "Drilling all tap holes with 12 mm Drill".

**Mill Level 1 - Metric**

- From the **Tree view list**, select **Linking Parameters** and change the **Depth** as shown in the picture to the right.



- From the extended **Tree view list**, select the **Tip comp** and change the **breakthrough amount** value as shown.



**Breakthrough amount** value allows you to give an extra amount for the tool to go deeper than the final depth to prevent any remaining material for the cut-outs.

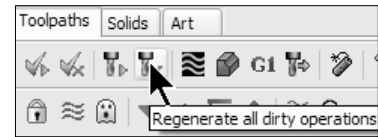
**Tip length** value is automatically calculated by the system based on the diameter and tip angle of the tool. The value is added to the final depth.

- Select the **OK** button to exit 2D Toolpaths - Drill parameters.

**Mill Level 1 - Metric**

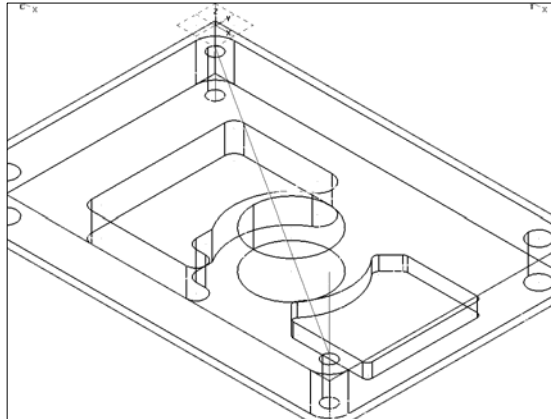
Copy the geometry in the tapping toolpath following the same steps shown for drilling geometry.

- Select the **Regenerate all dirty operations** icon.
- Press **Alt+ T** to remove the toolpath display.



**STEP 12: DRILL THE TWO 10 MM DIAMETER HOLES**

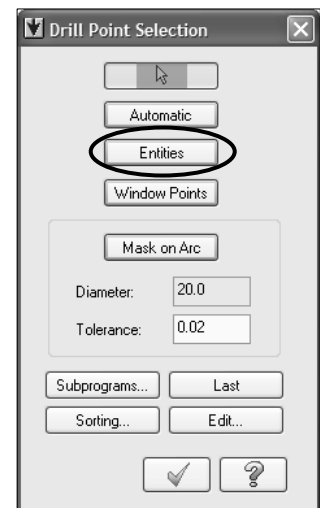
*Toolpath Preview:*



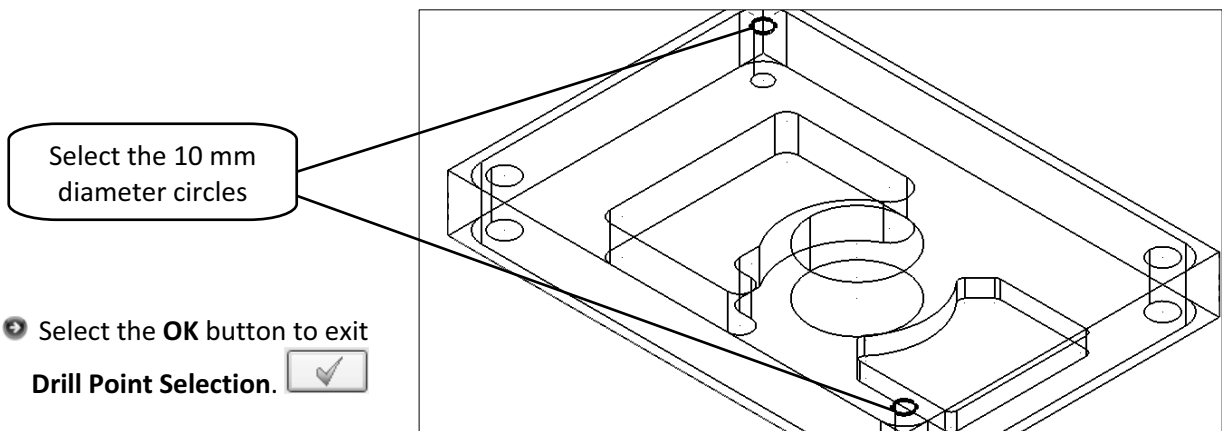
**12.1 Drill points selection**

**Toolpaths**

- **Drill**
- Select the **Entities** button.



- [Select entities]: Select the 10mm diameter circles.



## Mill Level 1 - Metric-

### 12.2 Set the Tool parameters

- ➔ From the **Tree view list**, select **Tool**.
- ➔ Click on **Select library tool**.
- ➔ Using **Filter** select the **10 mm Drill** as shown in the previous steps.
- ➔ Add the comment as shown below.

Toolpath Type

- ✓ Tool
- Holder
- Cut Parameters
- Linking Parameters
  - Tip Comp
  - Home / Ref. Points
- Planes (WCS)
- Coolant
- Canned Text
- Misc Values
- Axis Control
  - Axis Combination
  - Rotary Axis Control

Quick View Settings

Tool: 10. DRILL  
 Tool Diameter: 10  
 Corner Radius: 0  
 Feed Rate: 5.40945  
 Spindle Speed: 1145  
 Coolant: Off  
 Tool Length: 75  
 Length Offset: 5

#	Tool Name	Dia.	Cor. rad.	Length	#
1	75 Face...	75.0	0.0	10.0	4
2	20. SP...	20.0	0.0	50.0	2
3	12. DRI...	12.0	0.0	50.0	2
4	14.00-2...	14...	0.0	50.0	1
5	10. DRI...	10.0	0.0	50.0	2

Select library tool... ☐ Filter Active

Right-click for options

Tool diameter: 10.0

Corner radius: 0.0

Tool name: 10. DRILL

Tool number: 5 Length offset: 5

Head number: -1 Diameter offset: 5

Spindle direction: Cw

Feed rate: 5.409449 Spindle speed: 1145

FPT: 0.0024 CS: 35.9724

Plunge rate: 5.409449 Retract rate: 5.409449

☐ Force tool change ☐ Rapid Retract

Comment

Drill the two 10 mm diameter holes.

### 12.3 Set the Cut Parameters

- ➔ From the **Tree view list**, select the **Cut Parameters** page and change the **Cycle** to **Peck Drill** as shown.

Toolpath Type

- ✓ Tool
- Holder
- ✓ Cut Parameters
- Linking Parameters
  - Tip Comp
  - Home / Ref. Points
- Planes (WCS)
- Coolant
- Canned Text
- Misc Values
- Axis Control

Cycle: Peck Drill

1st peck: 5.0

2nd Peck: 0.0

Peck clearance: 0.0

Retract amount: 0.0

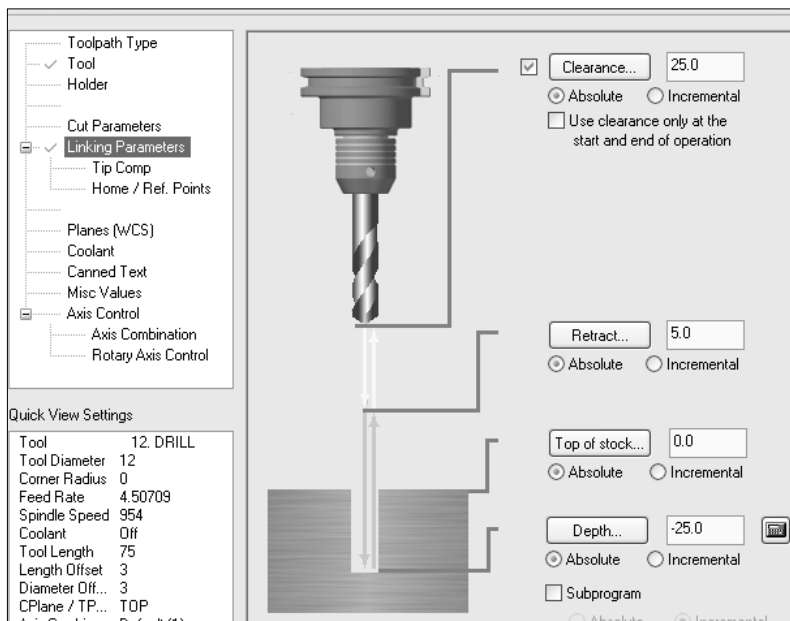
Dwell: 0.0

Chips: 0.0

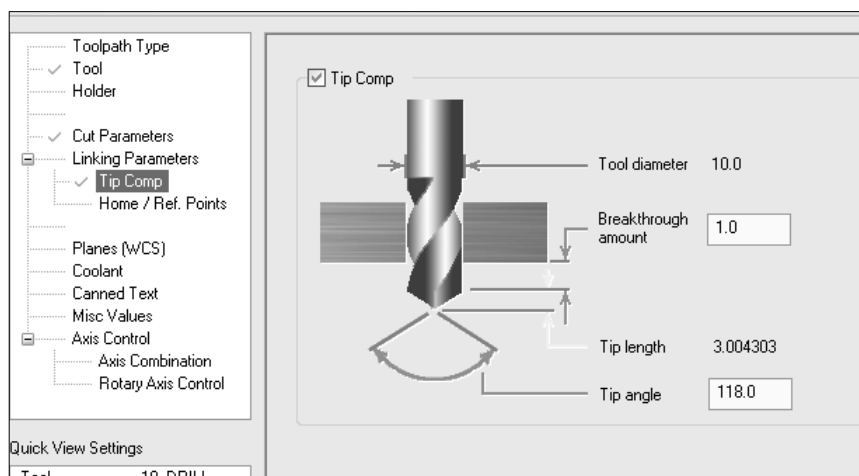
**Mill Level 1 - Metric-**

**12.4 Set the Linking Parameters**

- ➔ From the **Tree view list**, select the **Linking Parameters** page and change the **Depth** if needed as shown.



- ➔ From the extended **Tree view list**, select the **Tip comp**, enable it and add a **Breakthrough amount** as shown to ensure the complete machining of the through holes.



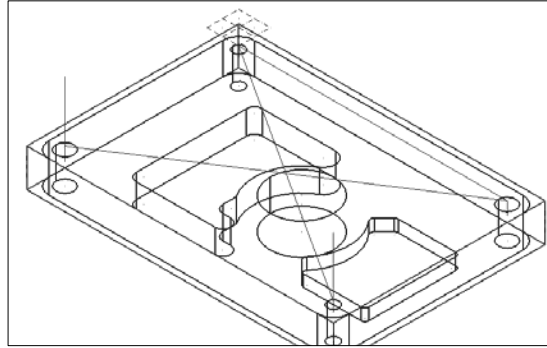
- ➔ Select the **OK** button to exit **2D Toolpaths – Drill** parameter pages.



## STEP 13: SPOT DRILL THE 10 MM DIAMETER HOLES

- Note that we have not yet spot drill the 10 mm diameter holes. In this step we will have to add the center points of the 10 mm diameter holes in the existing spot drilling operation geometry. We will also change the depth of the spot drill for the 10 mm diameter holes only.

*Toolpath Preview:*

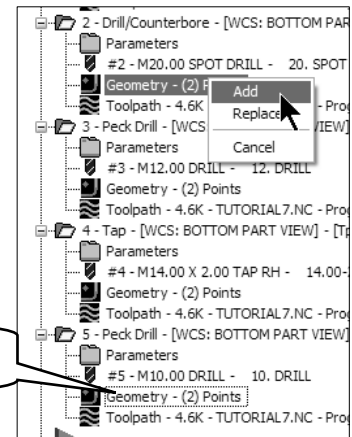


### 13.1 Add the center points of the 10 mm diameter holes in the existing spot drilling operation

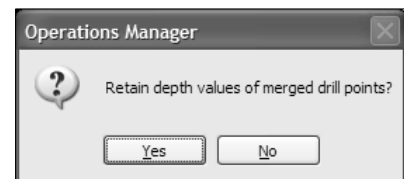
- In the **Toolpaths Operations Manager**, **Right mouse click** on the last Peck drill operation **Geometry** **hold** and **drag** the geometry above the Drill/Counterbore **Geometry**.

- Release the mouse and select **Add** from the list.

Right mouse click hold and drag



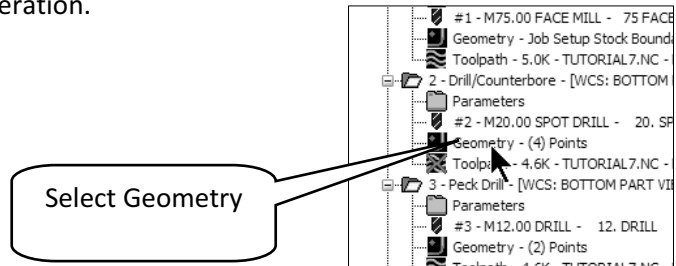
- Select **No** button to do not retain the depth from the 10 mm drilling operation.



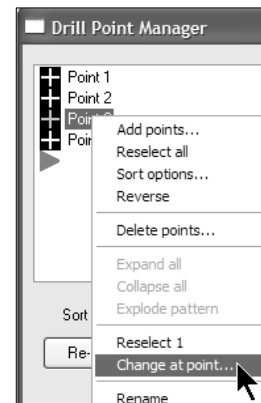
**Mill Level 1 - Metric**

**13.2 Modify the depth of spot drilling for the 10 mm holes**

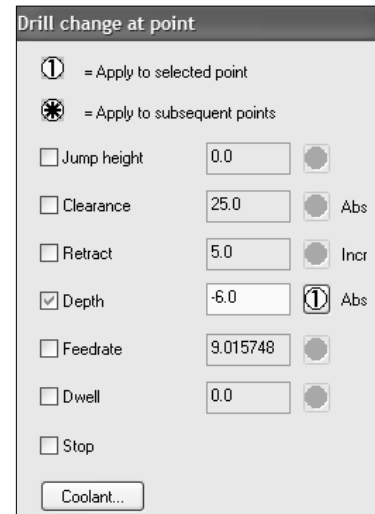
- ➔ Select the **Geometry** of the spot drilling operation.



- ➔ Click on the third Point and make sure that one of the 10 mm holes is selected.
- ➔ Then, **Right-mouse** click on the third point and select **Change at point...**



- ➔ In the **Drill change at point** dialog box, enable **Depth** and change the value to -6.



- ➔ Select the **OK** button to exit **Drill change at point** dialog box.



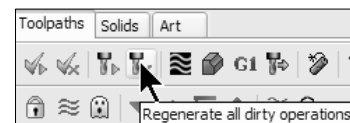
*Repeat the steps for the fourth point and change the depth to -6.*

- ➔ Make sure that first you select the point and then do a right-mouse click on it.

- ➔ Select the **OK** button to exit **Drill Point Manager** dialog box.



- ➔ Regenerate the toolpath.

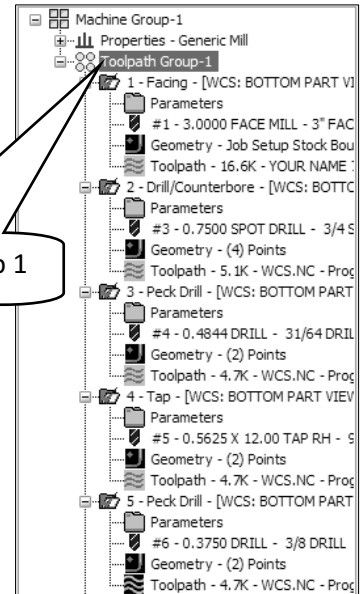


## STEP 14: BACKPLOT THE TOOLPATHS

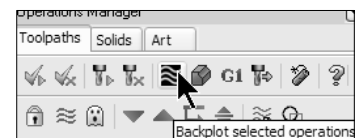
**Backplotting** shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

- Click on the **Toolpath Group** in the **Toolpaths Manager** to select all operations.

Select Toolpath Group 1

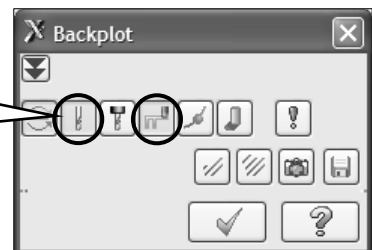


- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on (they will appear pushed down).

Display Tools and Rapid moves



- Select the **Isometric View** from the view toolbar to see the stock.



- Select the **Fit** button.



- You can adjust the speed of the backplot.

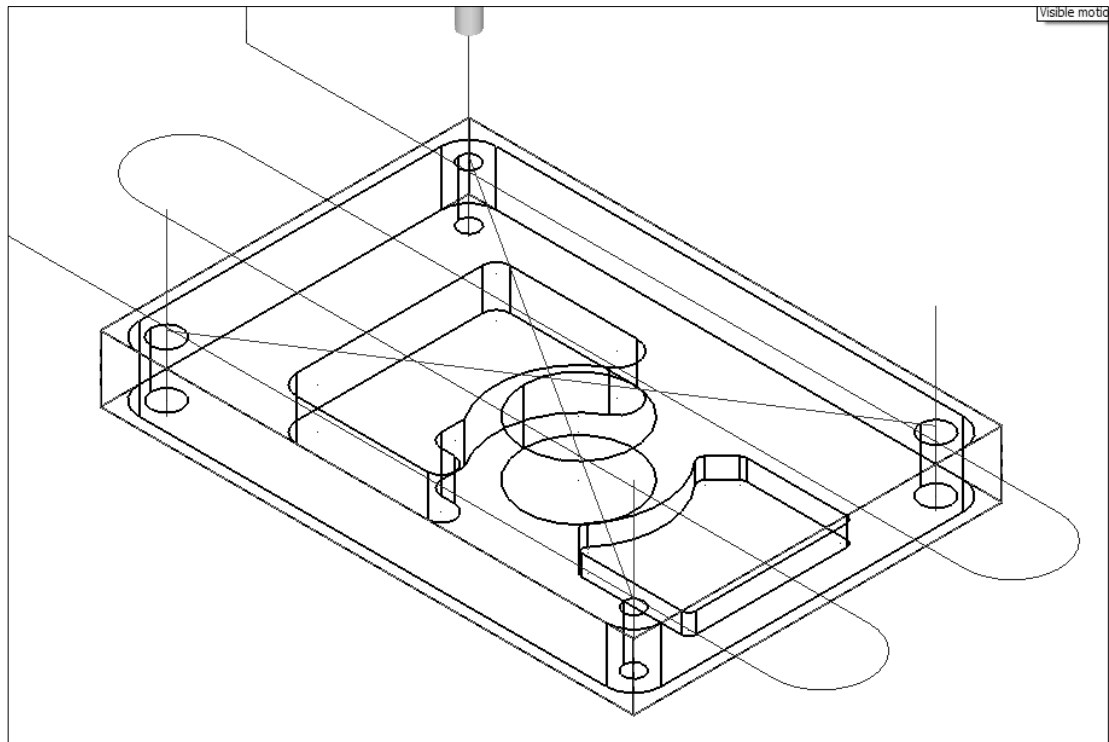
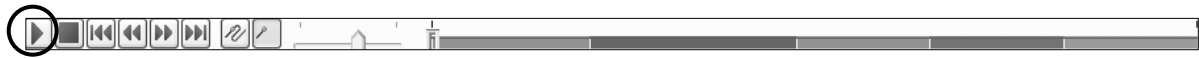


- You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.





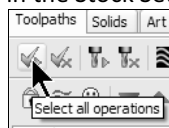
- Select the **Play** button in the **VCR** bar.



- Select the **OK** button to exit **Backplot**.


## STEP 15: VERIFY THE TOOLPATHS

- Verify simulates the machining of a part from a stock model display. The stock dimensions are based on the values that we specified in the Stock Setup.



- Select all operations button.
- Select the **Verify selected operations** button.

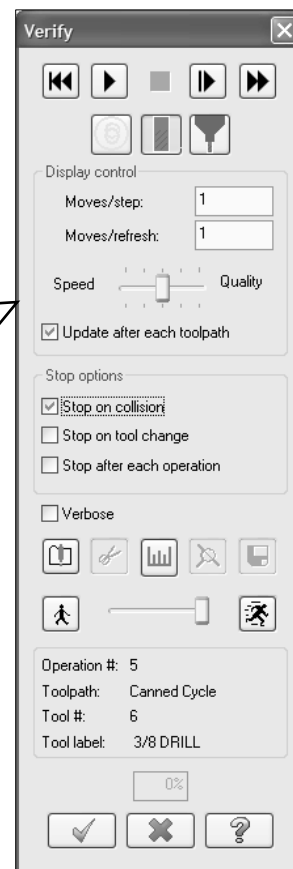


**Simulate tool**  simulates the toolpaths, displaying the solid tool without the holder

**Update after each toolpath** updates the stock after each operation.

**Stop on collision** pauses the verification when the tool touches the part with a rapid move.

➔ Select the **Configure** button.

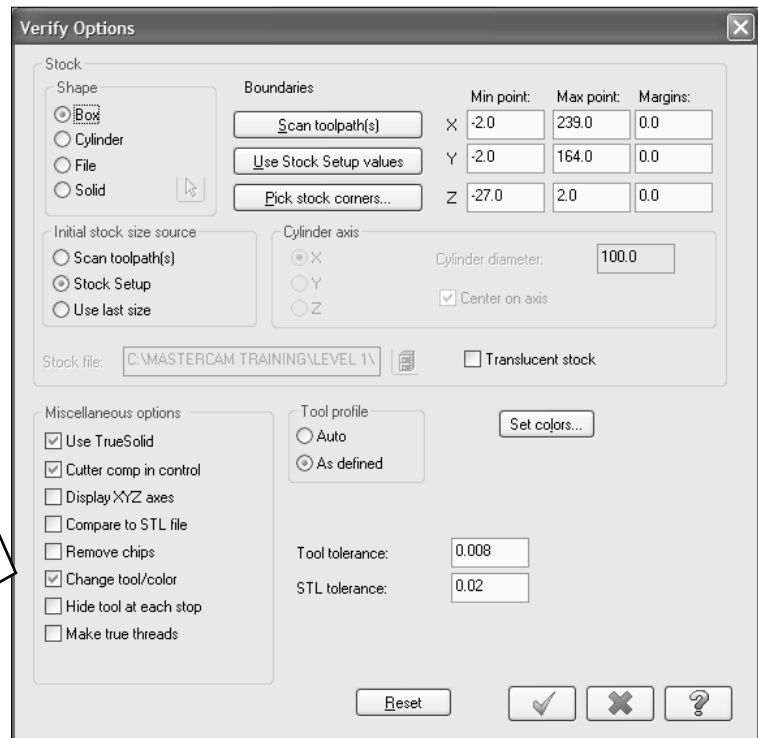


**Initial stock size source** should be set to **Stock Setup** to use the stock information from Stock Setup.

**Use True Solid** allows you, after verifying the part, to rotate and magnify the part to more closely check features, surface finish, or scallops.

**Cutter comp in control** allows Verify to use the information regarding the tool diameter and to simulate the cutter compensation.

**Change tool/color** to change the color of the cut stock to indicated tool changes in the toolpath.

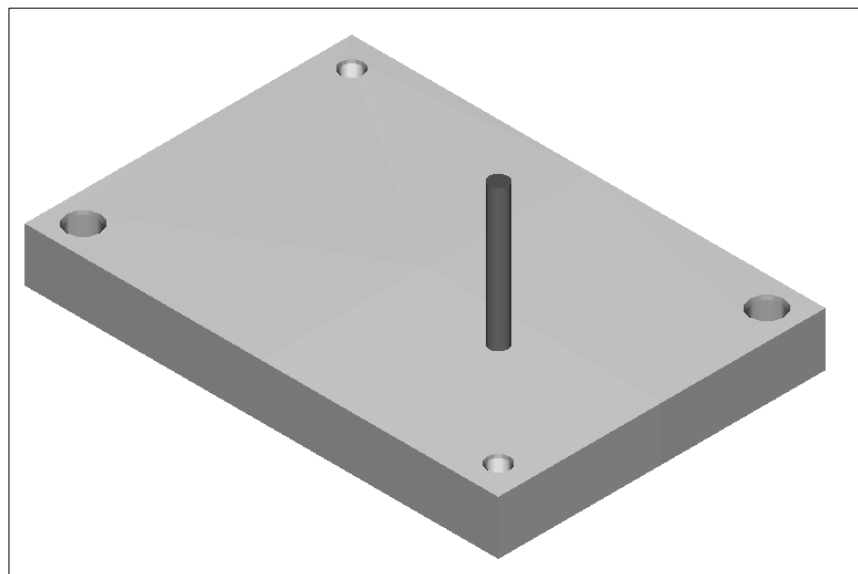


➔ Select the **OK** button to exit **Verify Options**.

➔ Set the **Verify speed** by moving the slider bar in the speed control bar.

➔ Select the **Play** button to start simulation.

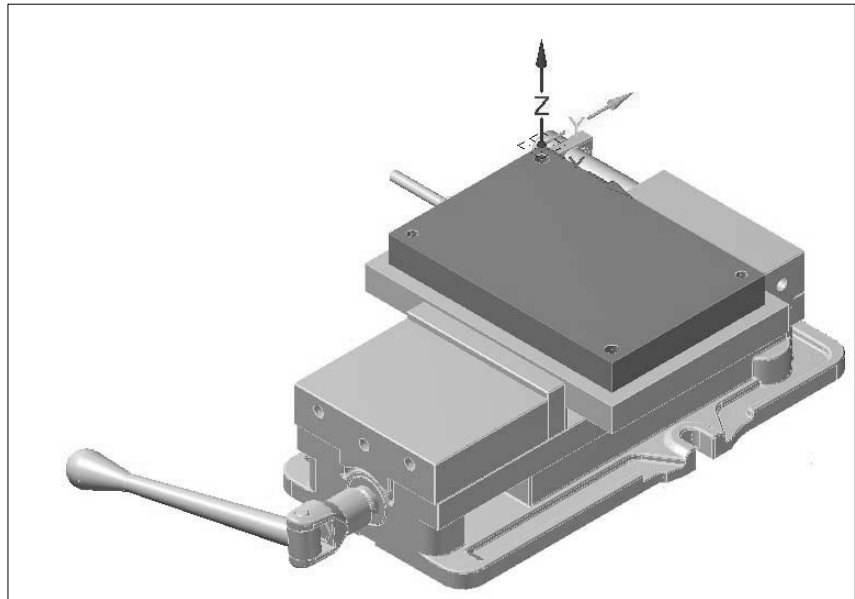
➔ The finished part should appear as shown in the following picture.



➔ Select the **OK** button to exit **Verify**.

#### SUGGESTED FIXTURE FOR SETUP 2:

- The stock is held on a fixture plate using the tapped and the drill holes. The two M14 bolts and two 10 mm dowels hold and position the part.



#### SETUP SHEET FOR SETUP 2:

Tool List of TUTORIAL7.MCX

Proj./Part No.: 0  
 Drawing No. : 1  
 Prog. No. : 7

Date : 07/27/09  
 Customer : -  
 Programmer : 0

Tool type :	75 Face mill	75 Face Mill		
Manufact.code :				
Chuck :				
Tool Number :	1		Feedrate :	8.0157
Diameter :	75	RPM :	509	Plunge feed r.:
Corner radius :	0	Tip angle :	45	Diam. offset :
Flute length :	10	Material :	ALUMINUM ...	Length offset :
Overall length:	50	No flutes :	4	

Tool type :	25 Endmill11 Flat	25. FLAT ENDMILL		
Manufact.code :				
Chuck :				
Tool Number :	6		Feedrate :	5.73
Diameter :	25	RPM :	3500	Plunge feed r.:
Corner radius :	0	Tip angle :	0	Diam. offset :
Flute length :	50	Material :	ALUMINUM ...	Length offset :
Overall length:	75	No flutes :	4	

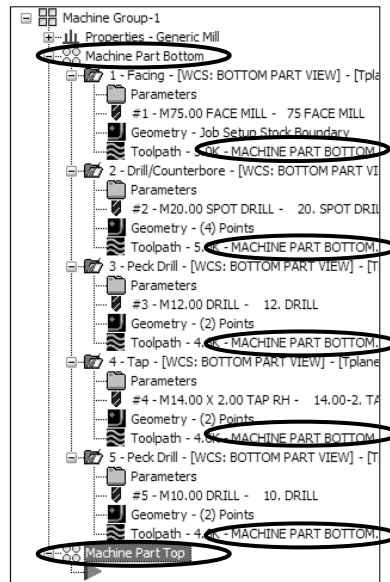
Tool type :	19 Endmill11 Flat	19. FLAT ENDMILL		
Manufact.code :				
Chuck :				
Tool Number :	7		Feedrate :	7.5395
Diameter :	19	RPM :	3500	Plunge feed r.:
Corner radius :	0	Tip angle :	0	Diam. offset :
Flute length :	50	Material :	ALUMINUM ...	Length offset :
Overall length:	75	No flutes :	4	

Tool type :	5 Endmill11 Flat	5. FLAT ENDMILL		
Manufact.code :				
Chuck :				
Tool Number :	8		Feedrate :	1.7906
Diameter :	5	RPM :	3500	Plunge feed r.:
Corner radius :	0	Tip angle :	0	Diam. offset :
Flute length :	50	Material :	ALUMINUM ...	Length offset :
Overall length:	75	No flutes :	4	

## STEP 16: CREATING AND RENAMING TOOLPATH GROUPS; RENAMING THE NC FILE

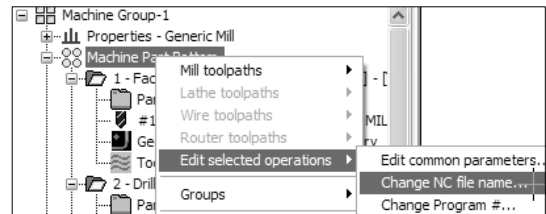
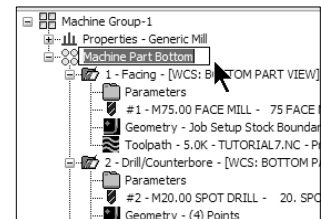
To machine the part in two different setups, we will need to have two separate programs. To be able to post process separately the operations of each setup, we will create them under different toolpath groups with different NC names.

*Step Preview:*



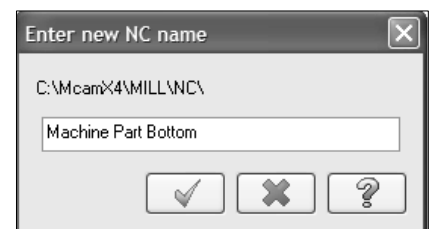
### 16.1 Rename the current Toolpath Group-1 and the NC file

- ➔ Click two times on the **Toolpath Group -1** to highlight it and rename it **Machine Part Bottom**.
- ➔ Right-mouse click on the toolpath group and select **Edit selected operations** and then, select **Change NC file name**.



- ➔ Enter the **new NC name**: "Machine Part Bottom".

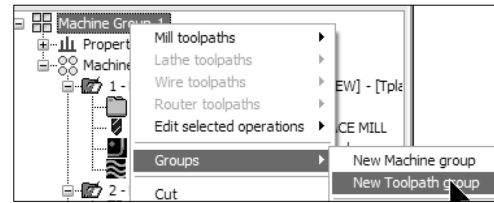
- ➔ Select the **OK** button.



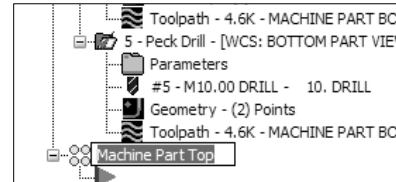
**Mill Level 1 - Metric**

**16.2 Create a new Toolpath Group**

- ➔ Right-mouse click on the **Machine Group-1**
- ➔ Select **Groups** from the drop down list and then select **New Toolpath group**.

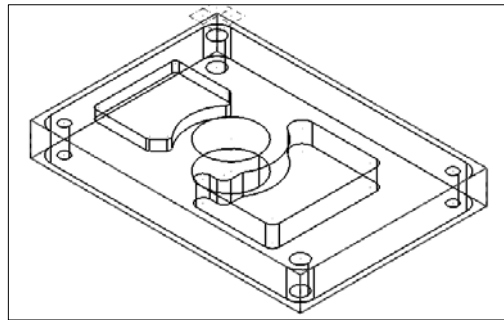


- ➔ Rename the toolpath group: "Machine Part Top".
- ➔ Make sure that the red insert arrow is below the Machining Part Top group; otherwise click on it and drag it below. This ensures that the next operation will be added in the proper group.



**STEP 17: SET THE TOP VIEW AS THE CURRENT WCS, TOOL AND CONSTRUCTION PLANE**

*Toolpath Preview:*

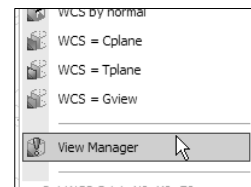


**17.1 Using WCS to set the Top view as the current tool plane, construction plane and WCS**

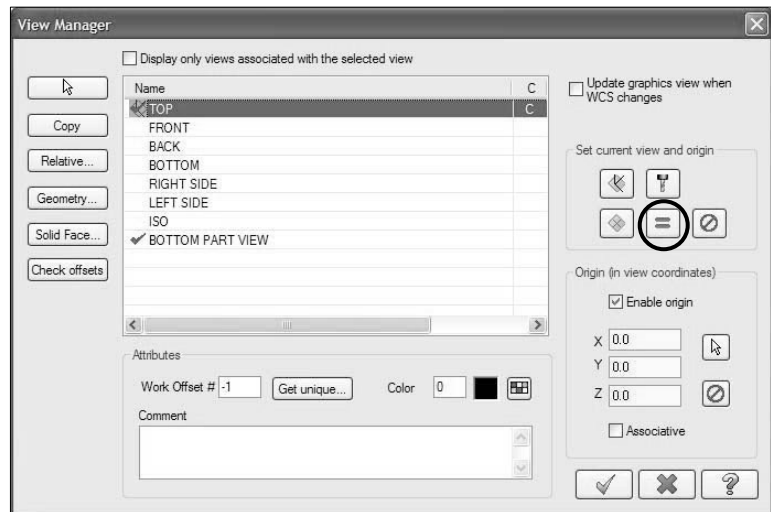
- ➔ Select **WCS** in the **Status Bar**.



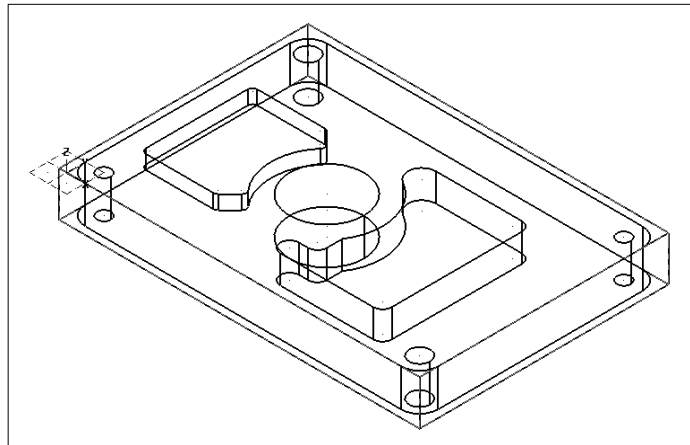
- ➔ Select **WCS Manager**.



- ➔ Select **Top View** and click on the **Set your current WCS, construction plane and tool plane with their origins to the selected view** button.

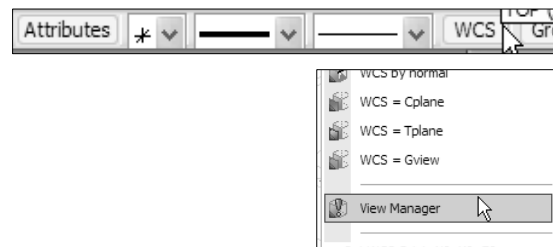


- ➔ Select the **OK** button.
- ➔ Select the **Isometric** view.
- ➔ Select **Fit** button.
- ➔ The part should look as shown to the right.
- ➔ Note that the origin is set at top of the part at the lower left corner. Follow next sub step if you want to move the origin at the upper left corner.



### 17.2 Using WCS to set the origin at the upper left corner

- ➔ Select **WCS** in the **Status Bar**.
- ➔ Select **View Manager**.



**Mill Level 1 - Metric**

- Click on the **Select** button in the **Origin** area as shown.

Origin (in view coordinates)

☒ Enable origin

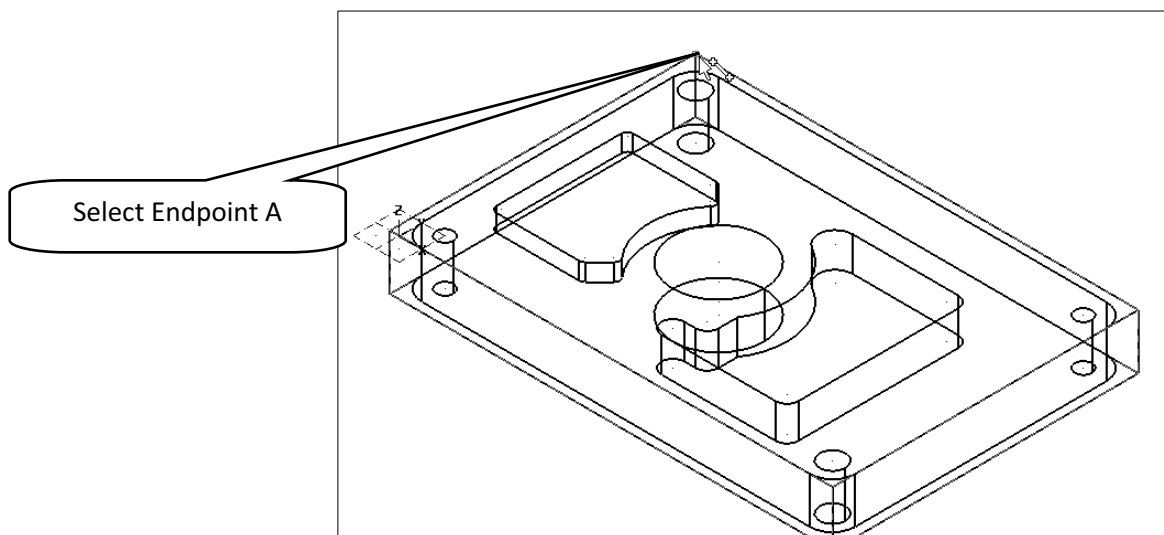
X 0.0

Y 0.0

Z 0.0

☐ Associative

- [Select a point]: Select Endpoint A.



- Change the **Z** value in the **Origin** area to 0 to set again the part zero at the finish size.
- The new origin coordinates should look as shown.

Origin (in view coordinates)

☒ Enable origin

X -2.0

Y 164.0

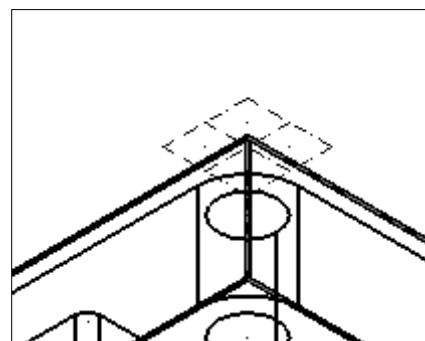
Z 0.0

☒ Associative

- Select the **OK** button to exit **View Manger**.



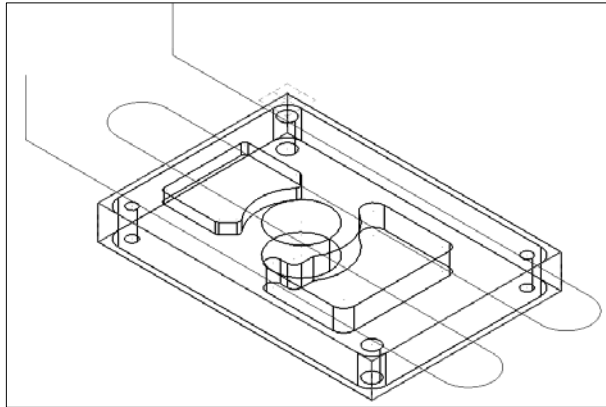
- Note that the grid moves at the upper left corner at the top of the part at the finished size.





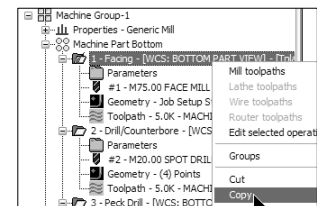
## STEP 18: FACE THE TOP OF THE PART

*Step Preview:*

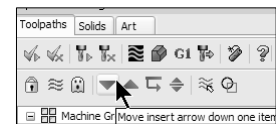


### 18.1 Copy the Facing operation in the Machine Part Top group

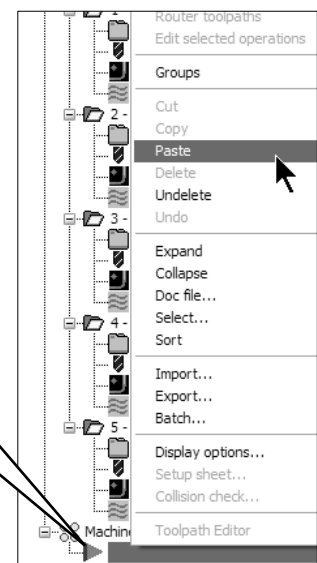
- ➡ Select only the Facing operation.
- ➡ **Right-mouse click** on the existing Facing operation and select **Copy** from the list.



- ➡ Move the red arrow below the Machine Part Top group if needed.
- ➡ **Right-mouse click** below **Machine Part Top** group and select **Paste** from the list.



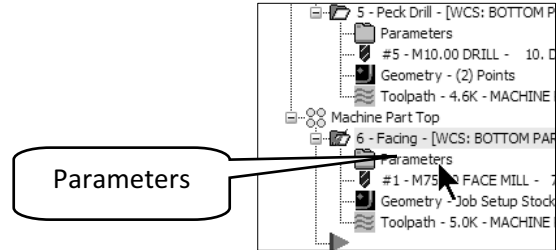
Right mouse click here



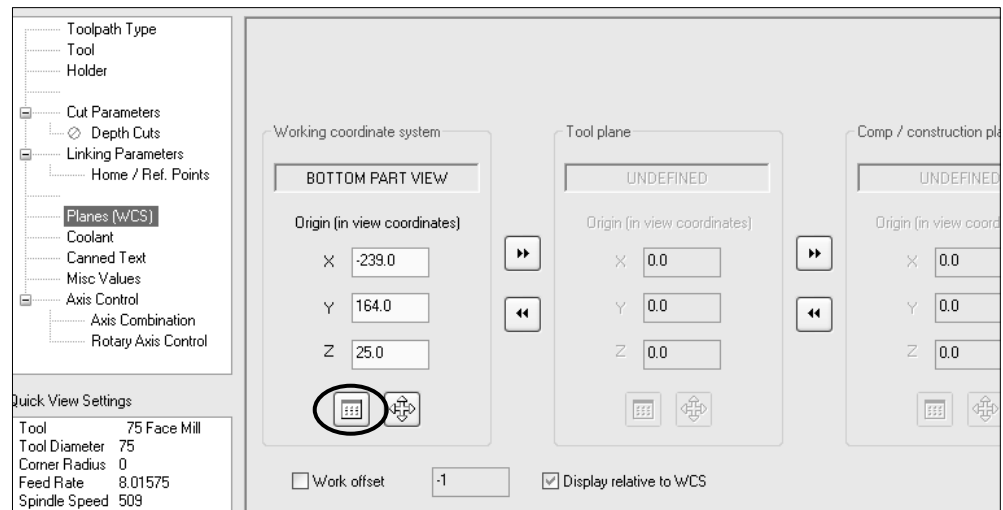
**Mill Level 1 - Metric**

**18.2 Change the planes to Top in the Facing operation**

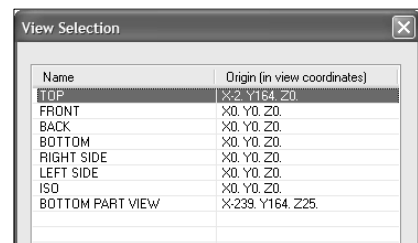
- ➔ From the **Toolpaths Operations Manager** select **Parameters** in the new Facing operation.



- ➔ From the **Tree view list** select **Planes**.
- ➔ Click on **Select WCS view** button as shown.



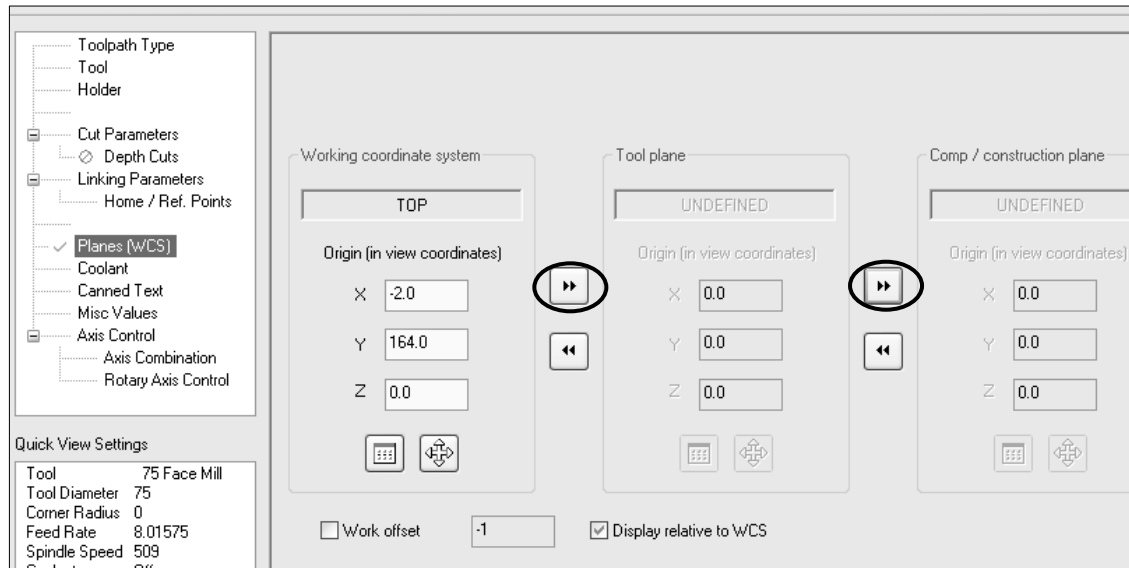
- ➔ From **View Selection** dialog box, select the **Top** view as shown.



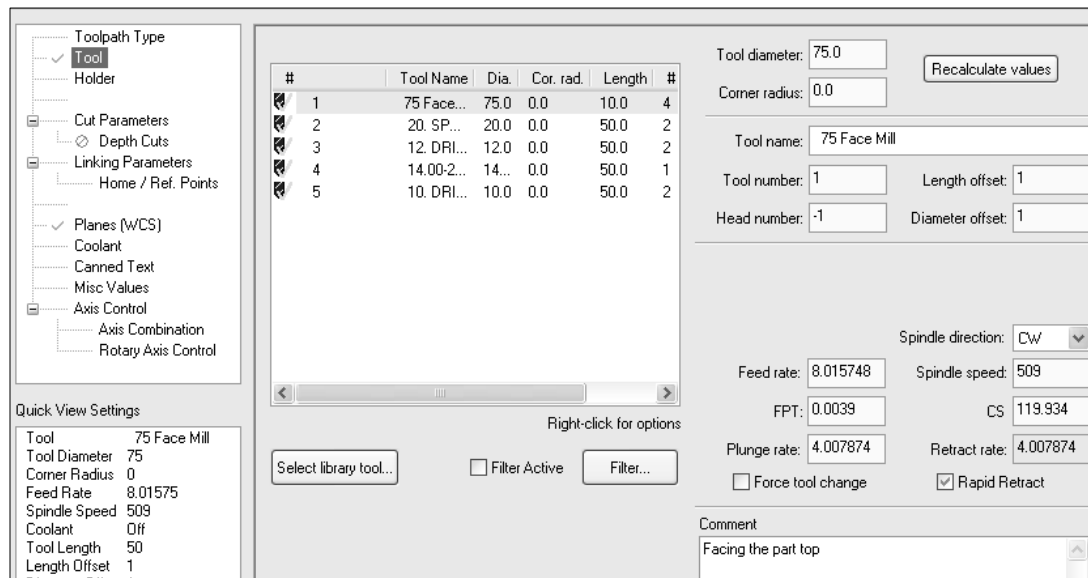
- ➔ Select the **OK** button to exit **View Selection** dialog box.



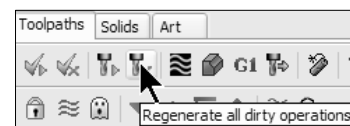
- ➔ Select **Copy to tool plane** and **Copy to construction plane** buttons to set all the planes to the **Top** plane.



- ➔ From the **Tree view list**, select the **Tool** page and change the comment as shown.

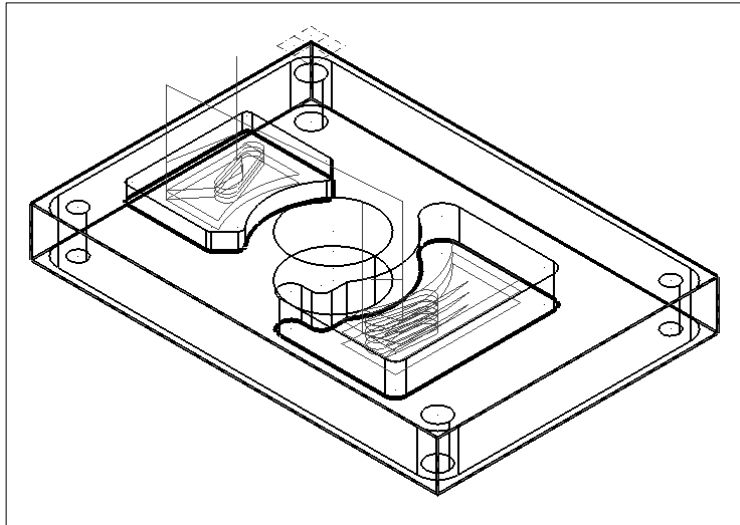


- ➔ Select the **OK** button to exit.
- ➔ Regenerate the toolpath.
- ➔ Press **Alt+ T** to remove the toolpath display if needed.



## STEP 19: ROUGH MACHINE THE TWO POCKETS IN THE TOP VIEW

*Step Preview:*



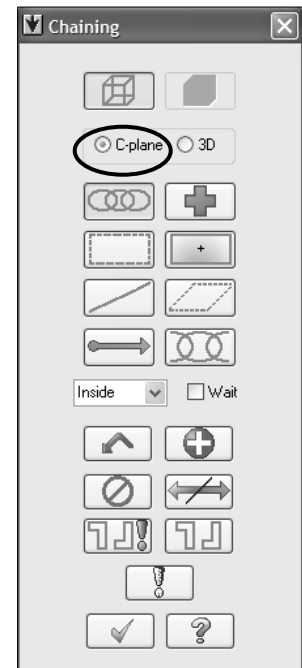
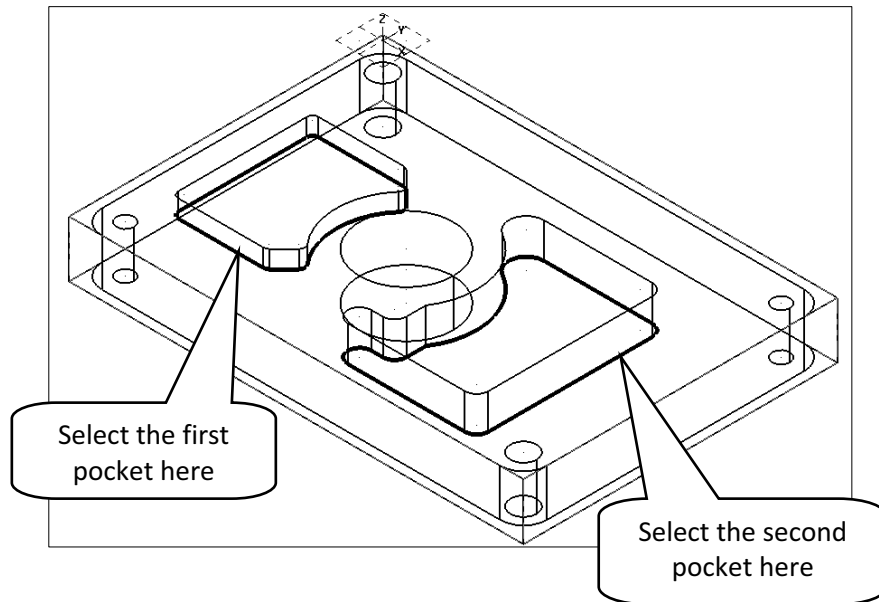
### 19.1 Select the bottom chains of the two pockets


- To be able to machine both pockets in the same operation, although they each have different depths, we need the 3D wireframe to select the pockets at the bottom.

#### **Toolpaths**

##### ➤ **Pocket**

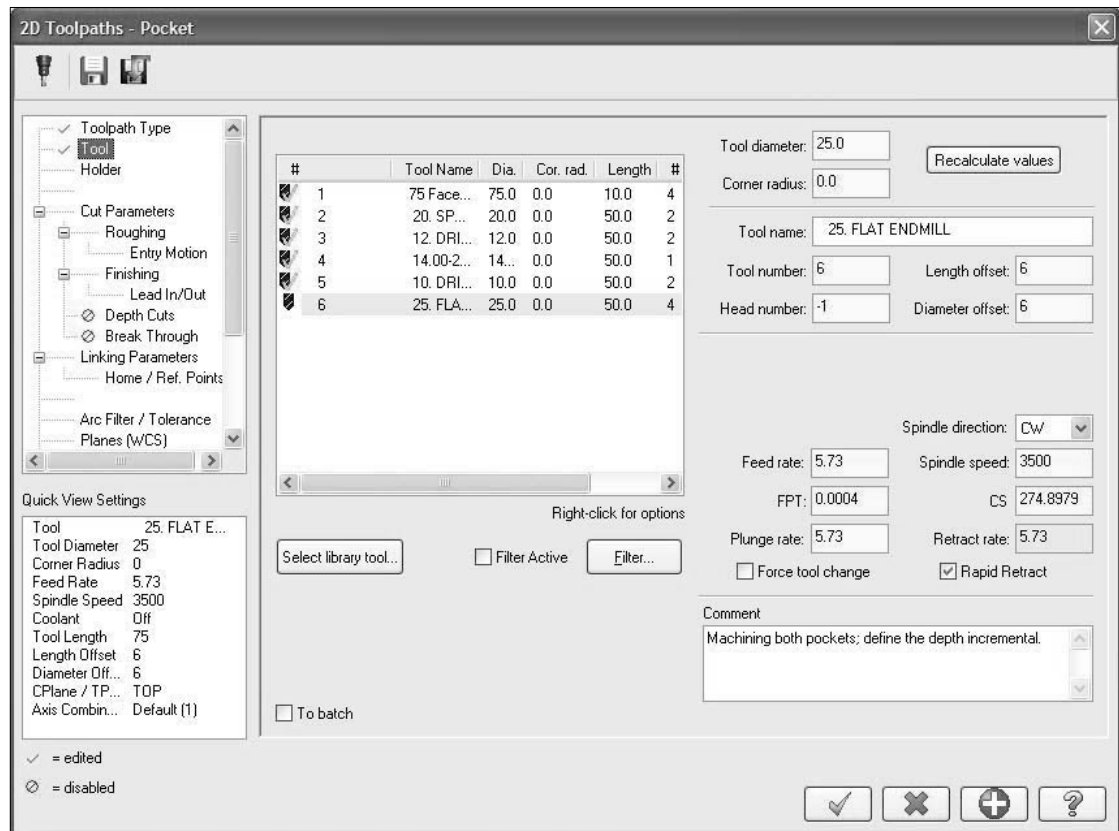
- Enabled **C-plane** in **Chaining** to be able to select the chains without stopping at the branches.
- Select the two pockets at the bottom, as shown.



- Note that both pockets are highlighted.
- Select the **OK** button to exit **Chaining**. 

### 19.2 Select a 25 mm Flat Endmill and set the parameters in the Tool page

- From the Tree view list, select **Tool**.
- Select the 25 mm Flat Endmill using the **Filter** options as shown in the facing operation.
- Make the necessary changes to match the parameters with the screenshot below.

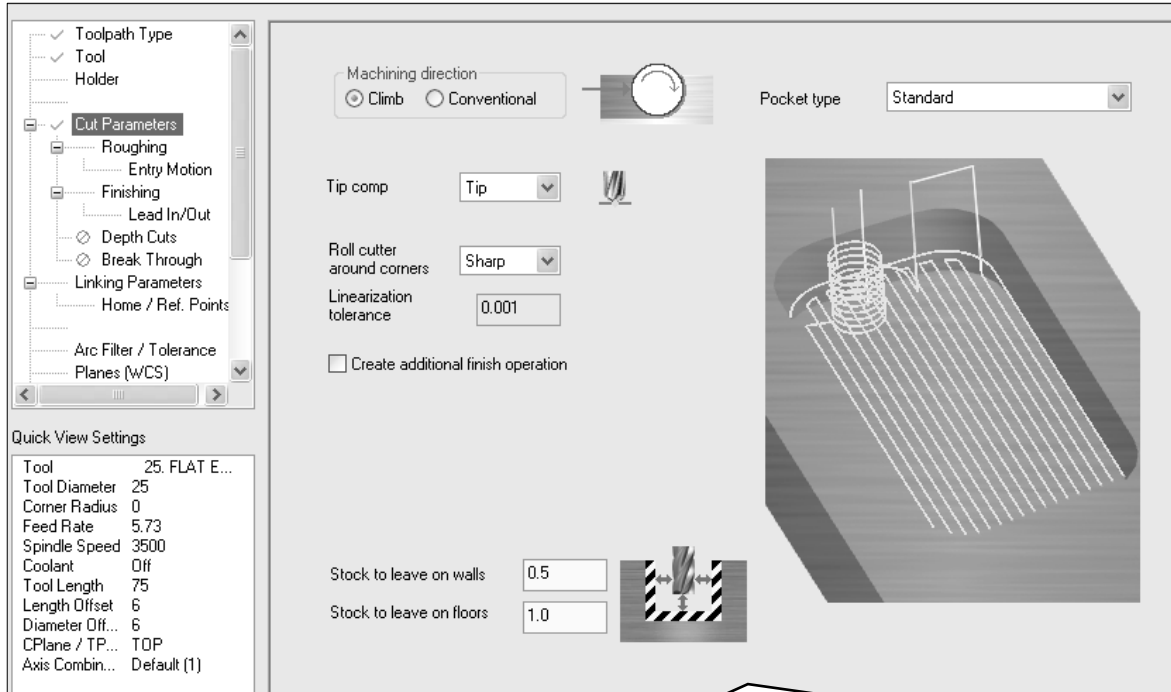


- Remember that the **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** values are all define in the parameters of the tool definition. Change them as needed.

**Mill Level 1 - Metric**

**19.3 Set the Cut Parameters**

- ➡ From the Tree view list, select the **Cut Parameters** page and change the parameters as shown.



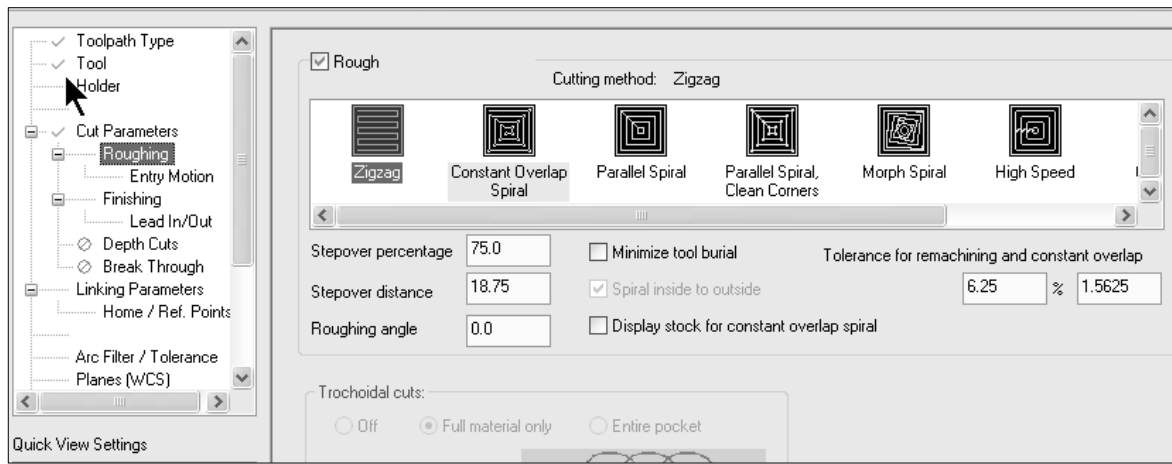
**Pocket type** set to **Standard** can be used only with closed chains. The tool will be constrained inside of the closed chain.

**Machining Direction** set to **Climb** cuts in one direction with the tool rotating in the opposite direction of the tool motion. See the graphic below.

**Tip Comp** set to the **Tip** sets the tool offset to the tool tip.

**Roll cutter around corners** inserts arc moves around the corners in the toolpath for a smoother tool movement. Set to Sharp (135 degrees or less) will roll the tool at the sharp corners.

- ➡ From the extended **Tree view list**, select **Roughing** and select as cutting method **Constant Overlap Spiral**.



**Stepover percentage** sets the distance between roughing passes in the XY axis as a percentage of the tool diameter and will automatically update the stepover distance.

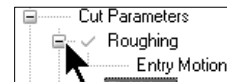
**Minimize tool burial** clears the area around pocket islands (semi islands in our case) to avoid tool damage due to removing too much stock at once.

**Spiral inside to outside** enabled allows you to spiral from the center to the pocket wall.

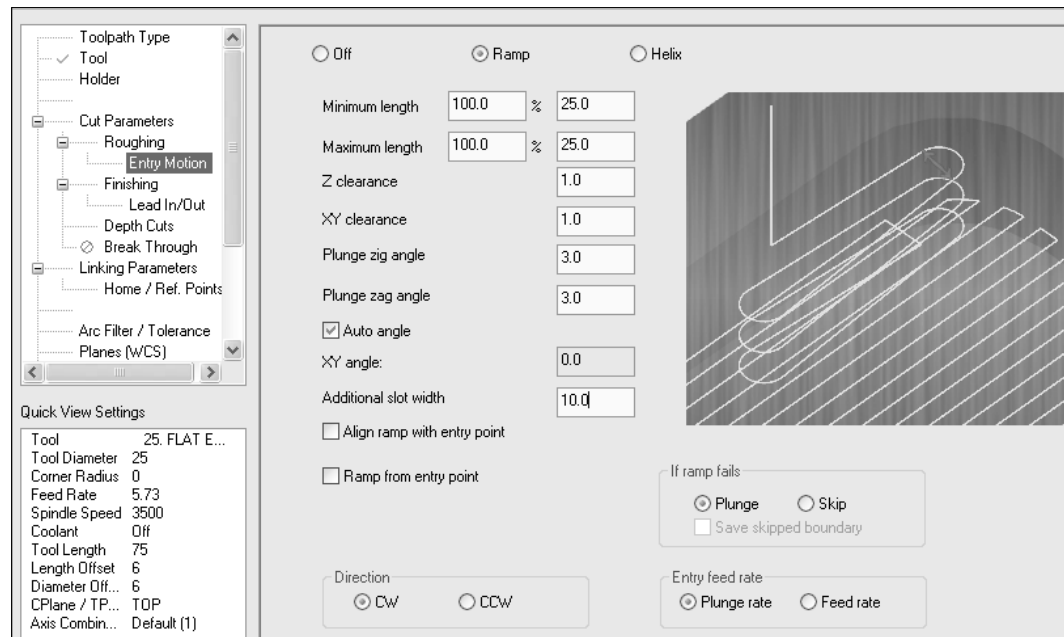
**Display stock for constant overlap spiral** previews the amount of stock removed by the roughing passes.

**Tolerance for remachining and constant overlap spiral** determines the accuracy of the toolpath when using one of the two options. Recommended tolerance is the default.

- ➡ Select the plus sign in front of the **Roughing** to expand the **Tree view list**, and select **Entry Motion** (if needed).



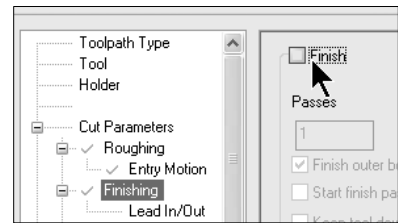
- Enable **Ramp** options and change the parameters to match the following screenshot.



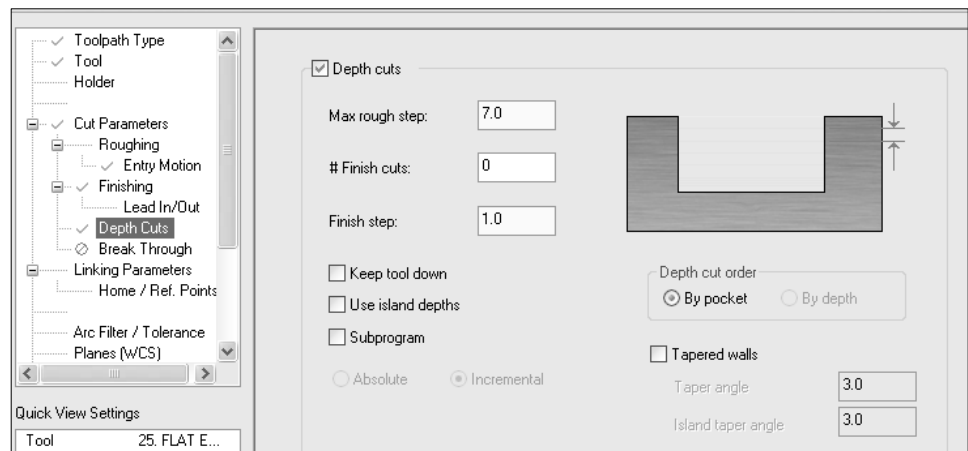
The ramp length value will be determined by Mastercam and can not be smaller than the **Minimum length** or bigger than the **Maximum length**.  
**Z clearance** sets the height above the stock where the entry ramp will start.  
**XY clearance** sets the minimum distance in the XY plane between the ramp and the pocket wall.  
**Plunge zig/zag angles** set the ramp angles as the tool zig-zags to the bottom of the pocket before beginning the cutting pass.  
**Auto angle** sets automatically the angle in the XY plane based on the longest area of the pocket.  
**Additional slot width** adds a fillet at the end of each ramp for a smoother tool movement.



- Select **Finishing** and disable the option as with this operation we want to rough only the part.



- From the **Tree view list**, select **Depth Cuts** and enable the option.



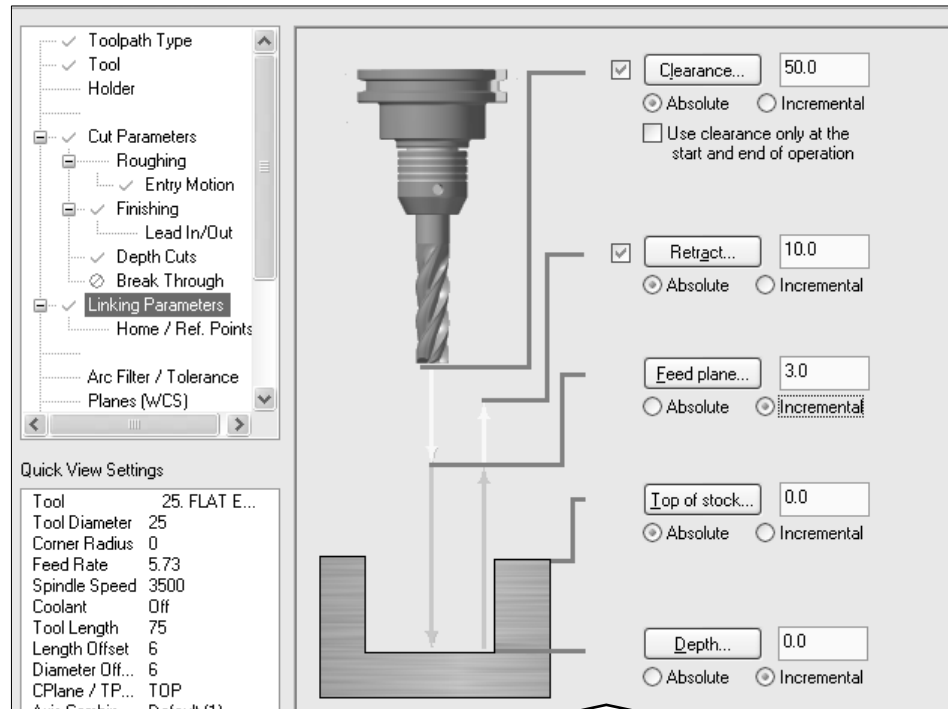
**Depth cuts** are the Z axis cuts that the tool makes in a pocket toolpath.  
**Max rough step** sets the maximum amount that can be machined in one step.  
**By pocket** enable the system to complete all depth cuts in the first pocket before moving to next one.

- Mastercam never performs unequal depth cuts. For example, in our case, the left side pocket is 10 mm deep; the step is calculated:  $(\text{Final depth} - \text{Stock on the floor}) / 2 = (10 - 1) / 2 = 4.5$  which is smaller than 7. The same way the system calculates the step for the second pocket;  $(19 - 1) / 3 = 6$

**Mill Level 1 - Metric**


**19.4 Set the Linking Parameters**

- ➡ From the **Tree view list**, select **Linking Parameters** and enable **Clearance** height and set the Top of the stock and the final **Depth** as shown.



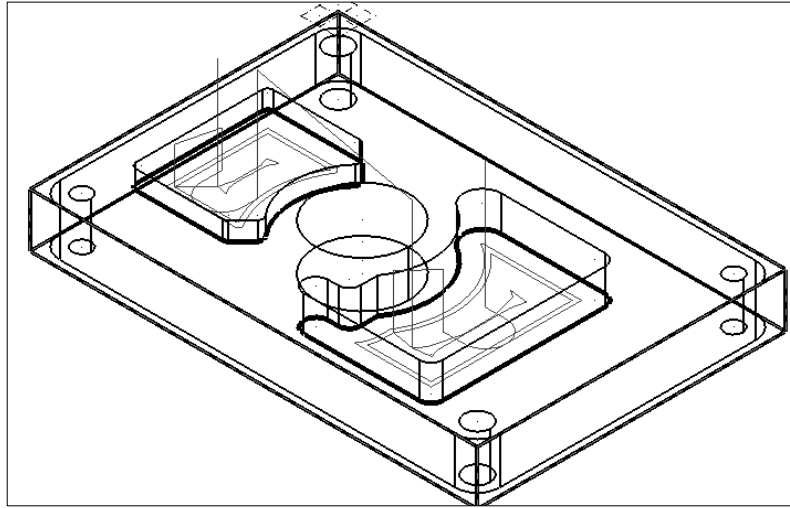
**Depth** value sets the final machining depth for the pocket operation. The value is set to **0** and **incremental** and is measured from the two geometry chains that we selected. This insures that both of them are going to be machined to the appropriate depth.

Choosing **Incremental** tells the system to calculate the value relative to either the current top of stock (as with **Clearance** parameter), relative to the selected geometry (as with **Top of stock** and **Depth** parameters), or relative to the depth of each cut (as with **Feed plane** and **Retract** ).

- ➡ Select the **OK** button to exit **2D Toolpaths Pocket** parameters. 
- ➡ Press **Alt+ T** to remove the toolpath display.

## STEP 20: FINISH THE TWO POCKETS

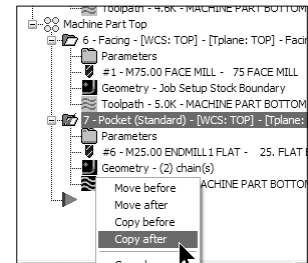
*Toolpath Preview:*



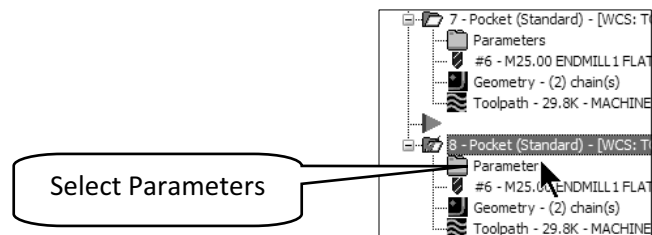
### 20.1 Copy the existing pocket operation

- You can copy an operation when machining the same geometry and using the same type of toolpath. Advantages of using a copy of an existing operation is that we do not need to select the geometry chains and we can use some of the parameters set up in the original operation.

- **Right-mouse click** and hold it down on the folder icon in front of the **Pocket** toolpath.
- Drag the mouse down and release it.
- Select **Copy after**.



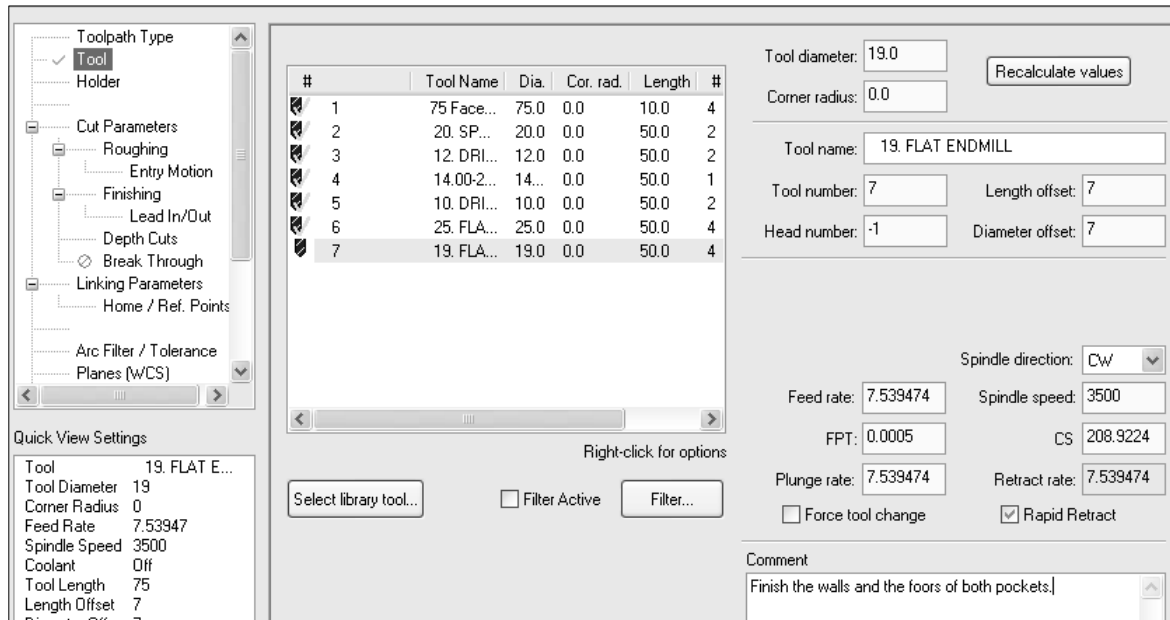
- You should now have two pocket toolpaths.
- Left-click on the second pocket **Parameters**.



## Mill Level 1 - Metric

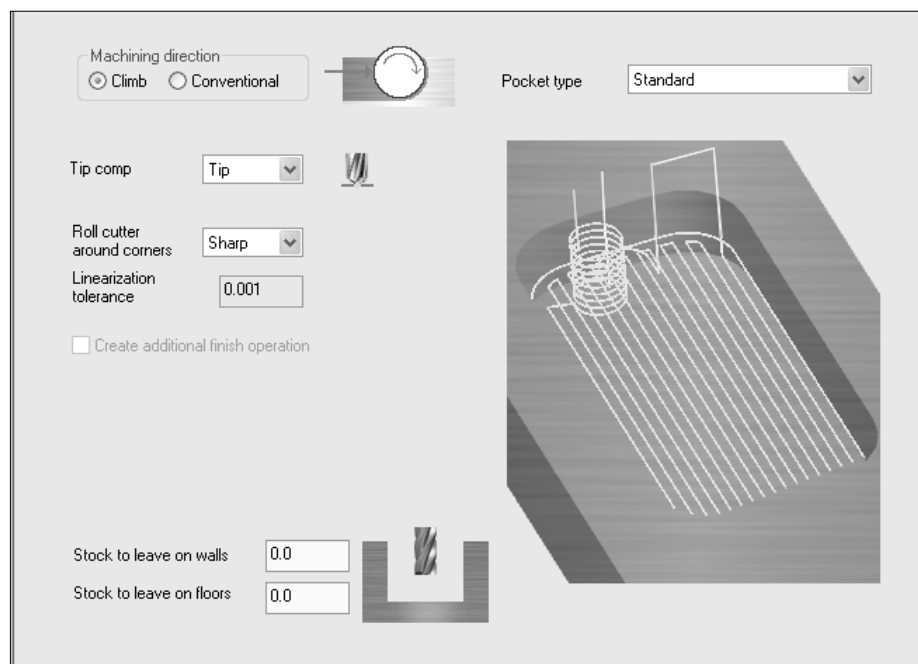
### 20.2 Select a 19 mm Flat endmill and set the Tool page parameters

- ➔ From the **Tree view list**, select the **Tool** page.
- ➔ Click on **Select library tool**.
- ➔ Following the steps outlined earlier, using the **Filter** option, select the 19 mm Flat Endmill.



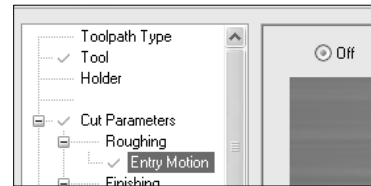
### 20.3 Set the Cut Parameters

- ➔ From the **Tree view list**, select the **Cut Parameters** and remove the stock to leave on the walls and on the floors.

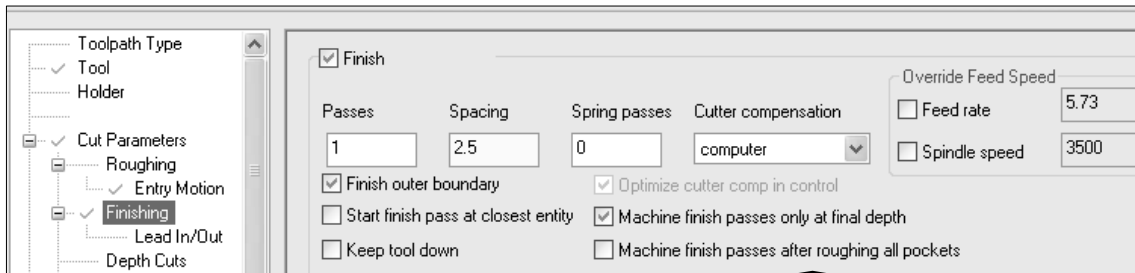


### Mill Level 1 - Metric

- From the **Tree view list**, select **Entry Motion** and disable the entry as shown.



- From the **Tree view list**, select **Finishing** and enable the **Finish** to finish the walls. Set the rest of parameters to match the following screenshot.

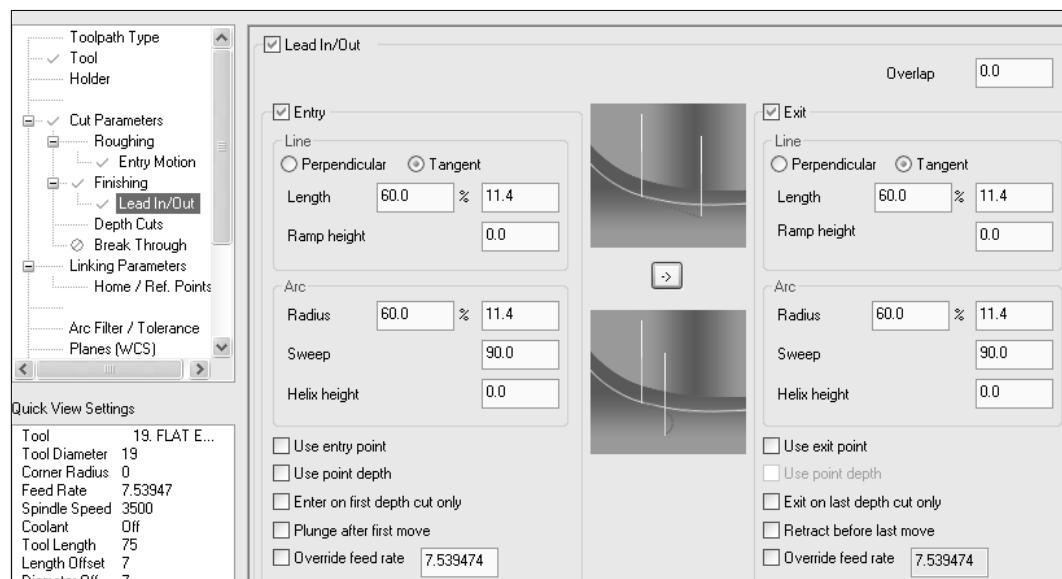


**Passes** sets the number of finish passes.

**Spacing** sets the amount of stock to be removed with each cut. Because we set the number of passes to 1, the amount of stock to be remove is the stock to leave on the walls amount set in the roughing operation.

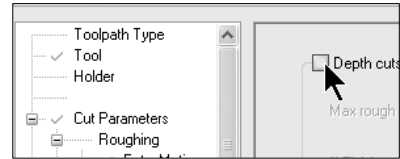
**Finish outer boundary** needs to be enabled to finish the walls of the pocket.

- From the **Tree view list**, select the plus sign in front of the **Finishing** (if needed) and set the parameters for the **Lead In/Out** as shown to make smooth entry/exit tool moves in and out of the remaining stock.



**Mill Level 1 - Metric**

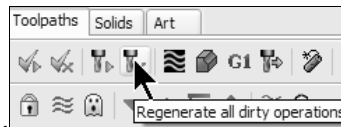
- From the **Tree view list**, select **Depth Cuts** and disable the option to remove the remaining stock in one step at the final depth.



- Select the **OK** button to exit the **2D Toolpaths- Pocket**.
- Press **Alt+ T** to remove the toolpath display.

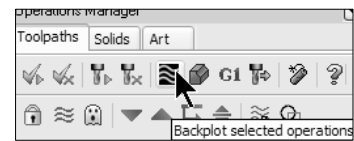


- Regenerate the toolpath.

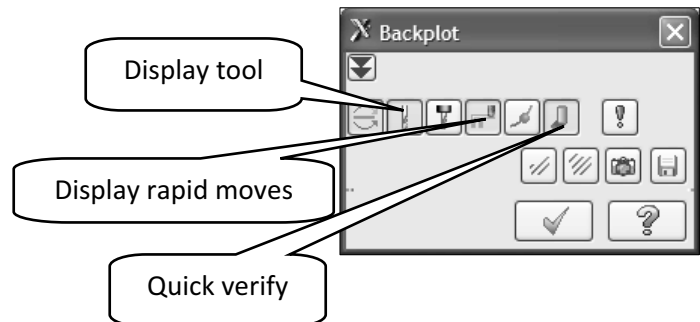


## STEP 21: BACKPLOT THE POCKET TOOLPATHS

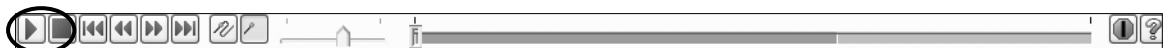
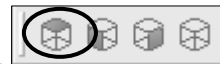
- In the **Toolpaths Operations Manager**, click on the first pocket operation.
- Holding down the **Shift** key select the second pocket.
- Click on **Backplot selected operations** icon.



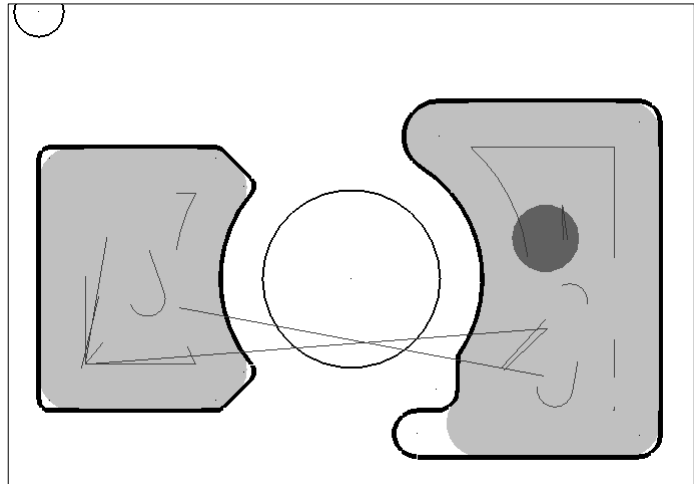
- Make sure that you have the following buttons turned on (they will appear pushed down).



- Select the **Top View** from the view toolbar to see the stock.
- Select the **Fit** button.
- Select the **Play** button in the **VCR** bar.



- You can adjust the speed of the backplot.

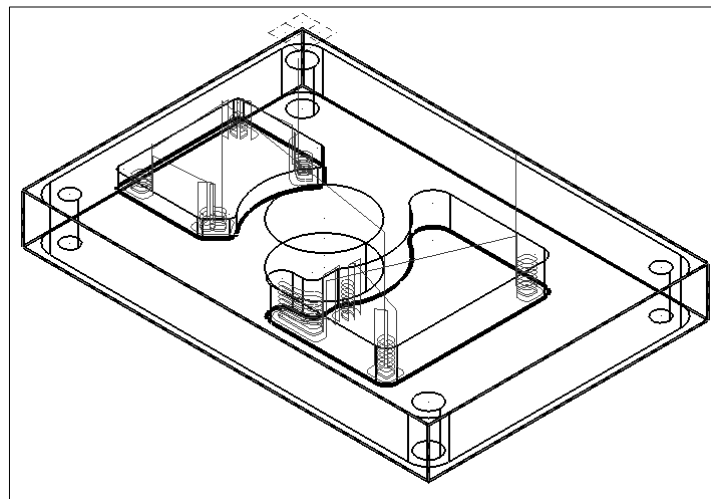


- Note the remaining material in the corners. The 19 mm Flat Endmill can not clean up the stock. In the next operation we will remachine the corners using a smaller tool.
- Select the **OK** button to exit **Backplot**.

## STEP 22: REMACHINE THE TWO POCKETS

- Note that the 19 mm Flat End Mill could not clean the 3 mm radius fillets. Using the 5 mm diameter Flat Endmill to remove all the material inside the pocket will not be efficient. We will remove the remaining material only by using a 5 mm Flat Endmill tool with the remachining pocket style.

*Toolpath Preview:*

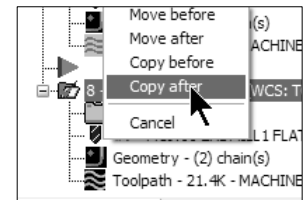


## Mill Level 1 - Metric

### 22.1 Copy the existing last pocket operation

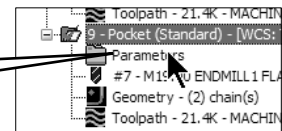
- You can copy an operation only when machining the same geometry using the same toolpath. Advantages of using a copy of an existing operation is that we don't need to select the geometry chains and we can use some of the parameters set up in the original operation.

- Select only the second Pocket (operation 8).
- Right-mouse click** and hold it down on the folder icon in front of the second **Pocket toolpath**.
- Drag the mouse down and release it.
- Select **Copy after**.



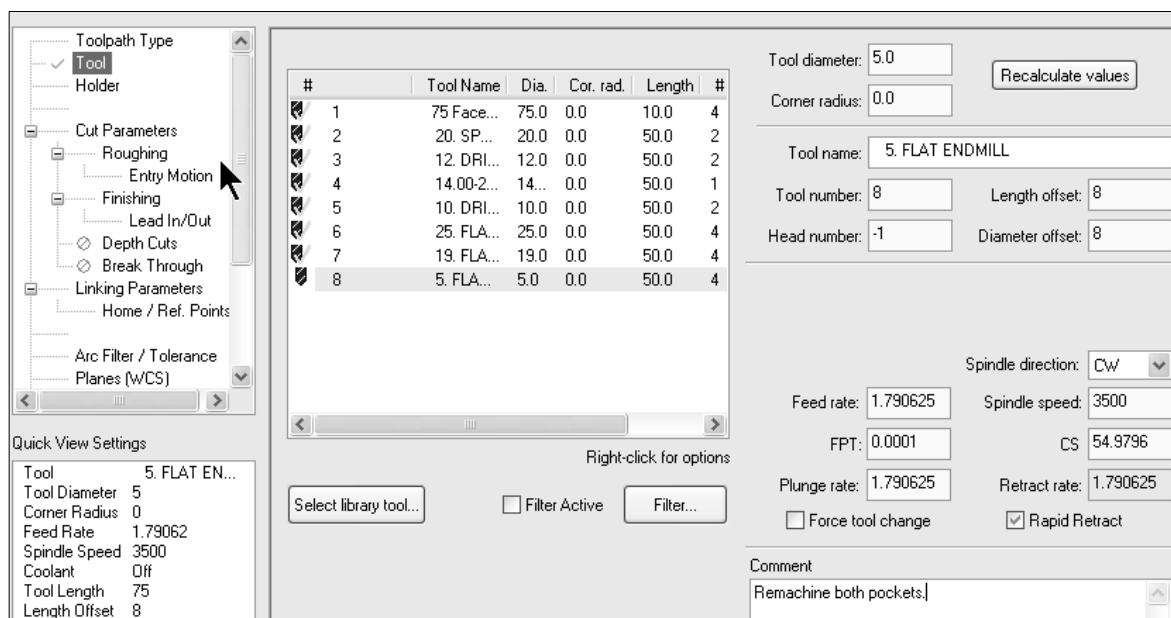
- You should now have three pocket toolpaths.
- Left-click on the last pocket **Parameters**.

Select Parameters



### 22.2 Select a 5 mm Flat Endmill and set the Tool parameters

- From the Tree view list, select the **Tool** page.
- Click on **Select library tool**.
- Following the steps outlined earlier, using the **Filter** option, select the **5 mm Flat Endmill**.

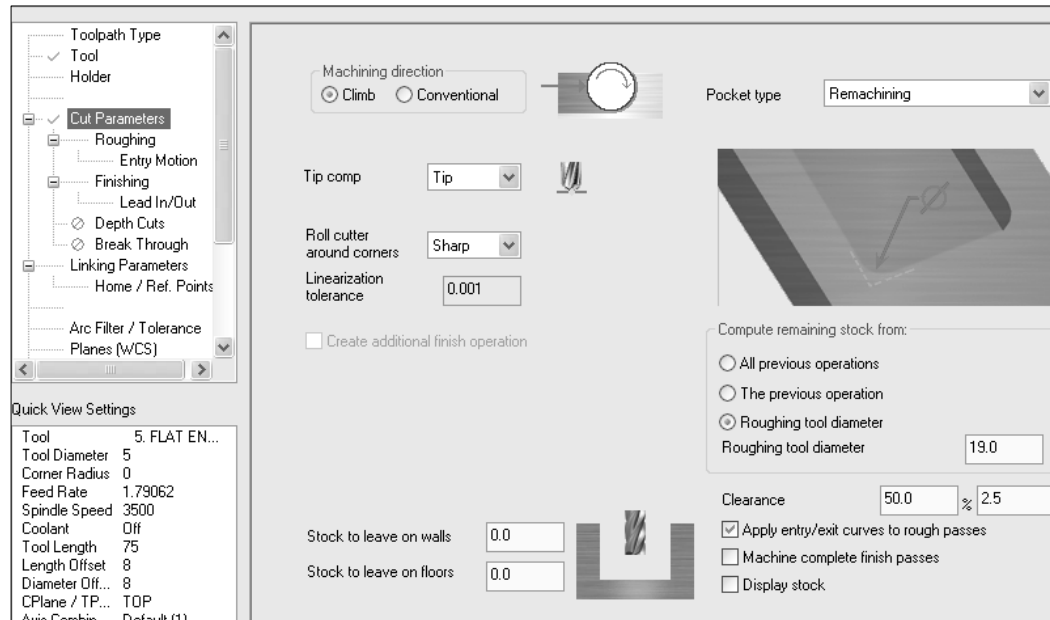




**Mill Level 1 - Metric**

**22.3 Set the Cut Parameters**

- ➔ Select **Cut Parameters** and change the **Pocket type** to **Remachining** as shown below.
- ➔ Set the **Compute remaining stock from the Roughing tool diameter** and set the parameters as shown.



**Compute remaining stock from:**

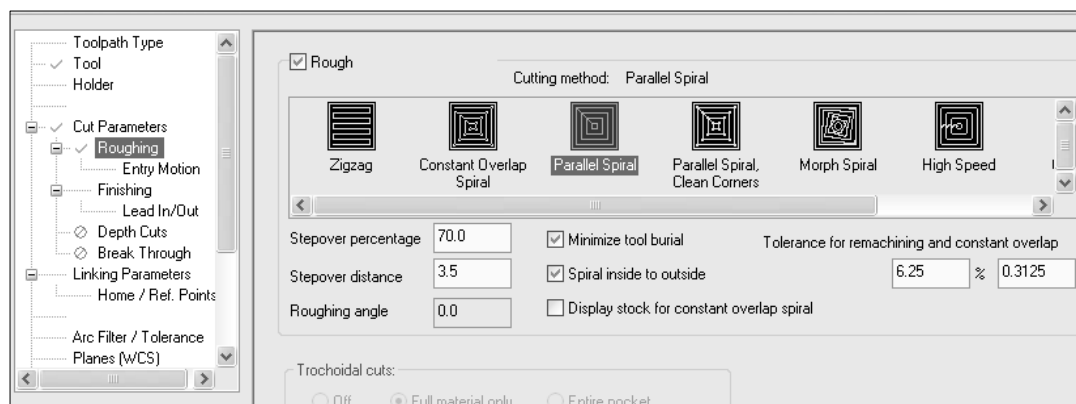
**Roughing tool diameter** enables the system to calculate the remaining stock for remachining based on the size of the tool diameter that you enter.

**Clearance**, set as a percentage of the tool diameter, allows you to expand the remachining area at the beginning and at the end to prevent a cusp of material remaining.

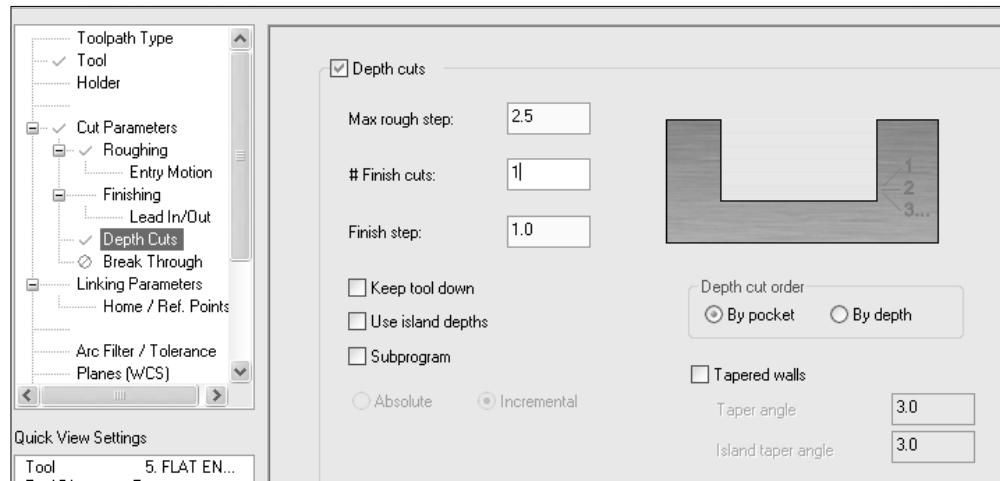
**Apply entry/exit curves to rough passes** allows you to use the Lead in/out parameters.


**Machine complete finish passes** allows you to finish the entire part.

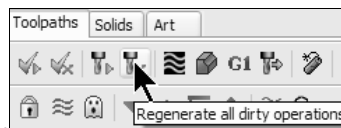
- ➔ From the **Tree view list**, select **Roughing** and change the **Stepover percentage** to 70 % to ensure a complete removal of the stock.



- From the **Tree view list**, select **Depth Cuts** and enable it.
- Change the **Max rough step** as shown.



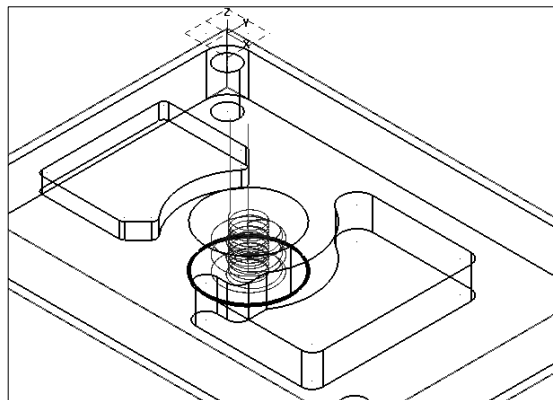
- Select the **OK** button to exit from the **2D Toolpaths - Pocket**. 



- Regenerate the toolpath.
- Press **Alt + T** to remove the toolpath display.

## STEP 23: CIRCLE MILL THE 50 MM DIAMETER HOLE

*Toolpath Preview:*



### Mill Level 1 - Metric

#### 23.1 Select the toolpath geometry

- ➔ From the **Toolpaths Operations Manager**, select **Move insert arrow down one item** until the insert arrow is below the last operation.

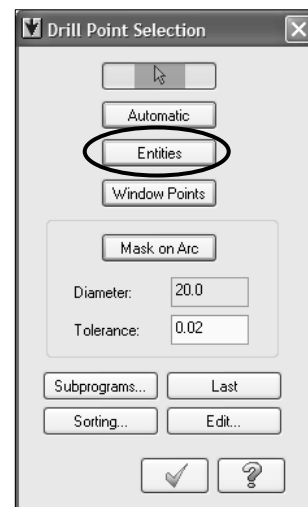
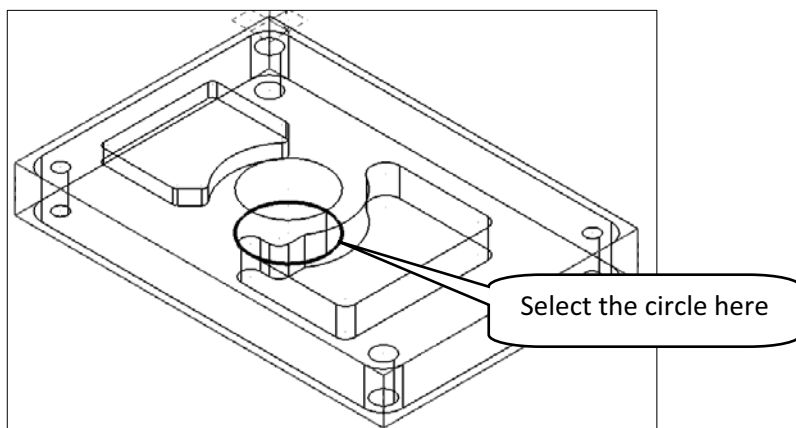


#### Toolpaths

##### ➔ Circle Paths

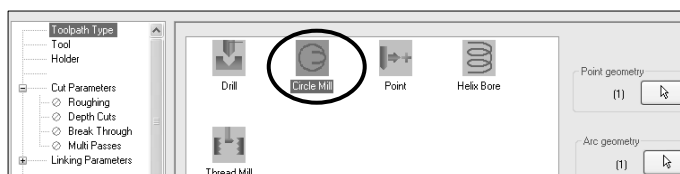
##### ➔ Circmill

- ➔ From the **Drill Point Selection**, select **Entities** button.
- ➔ [Select entities]: Select the bottom of 50 mm diameter hole as shown.



- ➔ Select the **OK** button to exit **Drill Point Selection** dialog box.

- ➔ Note that in the **Toolpath Type** the **Circle Mill** icon is automatically selected.



**Mill Level 1 - Metric**

**23.2 Set the Tool parameters**

- ➔ From the **Tree view list**, select the **Tool** page.
- ➔ Select the existing 19 mm Flat Endmill from the tool list window as shown.

The screenshot shows the Mastercam X4 interface with the Tool List and Tool Parameters dialog open.

**Tool List Window:**

#	Tool Name	Dia.	Cor. rad.	Length	#
1	75 Face...	75.0	0.0	10.0	4
2	20. SP...	20.0	0.0	50.0	2
3	12. DRI...	12.0	0.0	50.0	2
4	14.00-2...	14...	0.0	50.0	1
5	10. DRI...	10.0	0.0	50.0	2
6	25. FLA...	25.0	0.0	50.0	4
7	19. FLA...	19.0	0.0	50.0	4
8	5. FLA...	5.0	0.0	50.0	4

**Tool Parameters Dialog:**

Tool diameter: 19.0  
 Corner radius: 0.0  
 Recalculate values

Tool name: 19. FLAT ENDMILL

Tool number: 7  
 Length offset: 7

Head number: -1  
 Diameter offset: 7

Spindle direction: CW  
 Feed rate: 7.539474  
 Spindle speed: 3500  
 FPT: 0.0005  
 CS: 208.9224

Plunge rate: 7.539474  
 Retract rate: 7.539474

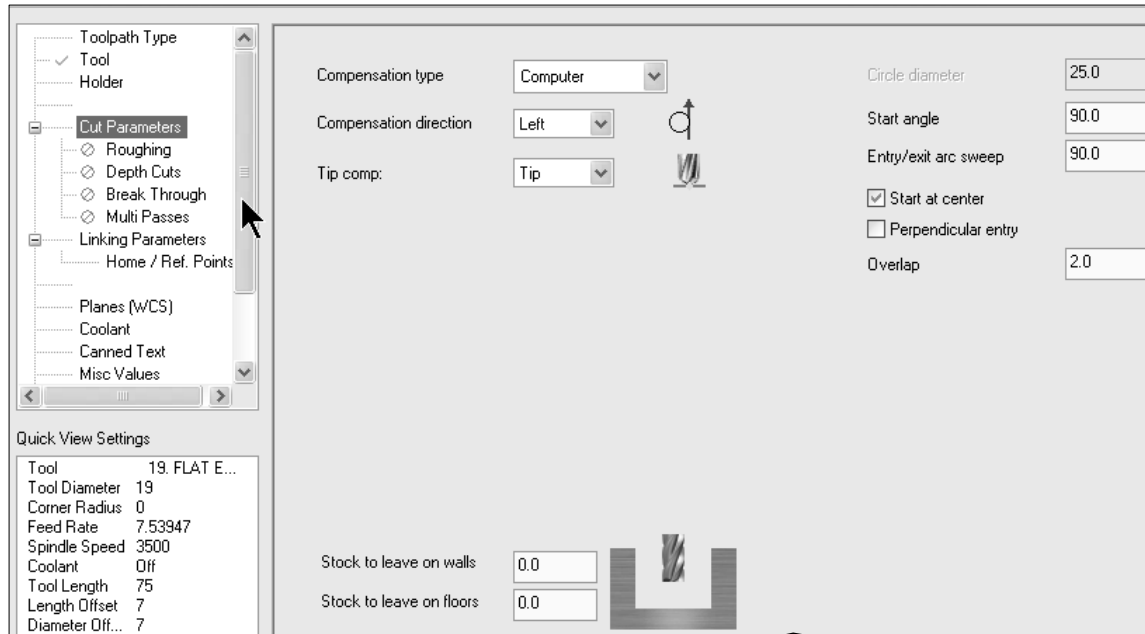
☐ Force tool change ☒ Rapid Retract

Comment: Circle mill the 50 mm diameter hole.

**Mill Level 1 - Metric-**

**23.3 Set the Cut Parameters**

➔ From the **Tree view list**, select the **Cut Parameters** and change the parameters as shown.



**Compensation type** set to **Computer** allows Mastercam to compensate the toolpath based on the tool diameter and does not output G41/G42 in the code.

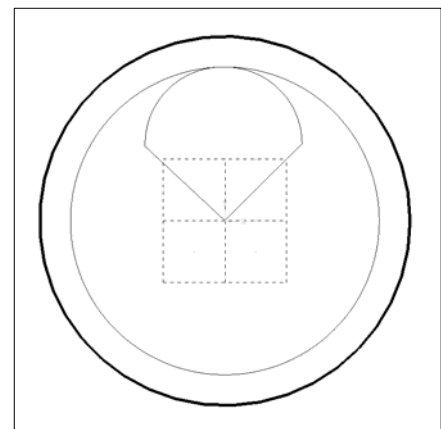
**Compensation direction** set to **Left** compensates the toolpath to the left of the chain in the CCW direction.

**Tip Comp** set to the **Tip** sets the tool offset to the tool tip.

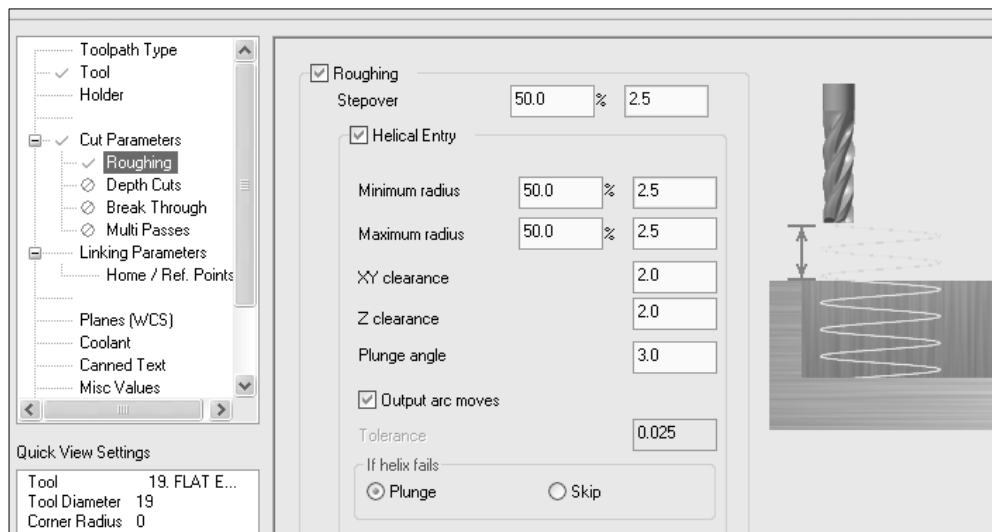
**Entry/exit arc sweep** set to 90 degrees makes a line between the plunge point and the entry/exit arc movement of a  $\frac{1}{4}$  of a circle, at each entry/exit. See the picture below.

**Overlap** sets how far the tool goes past the end of the toolpath before exiting. This ensures a clean finish.

*Entry/exit arc sweep set to 90 degrees will generate this result.*



- From the **Tree view list**, select **Roughing** to add roughing passes to completed machine the inside of the hole.
- Enable **Roughing** and set the parameters as shown.



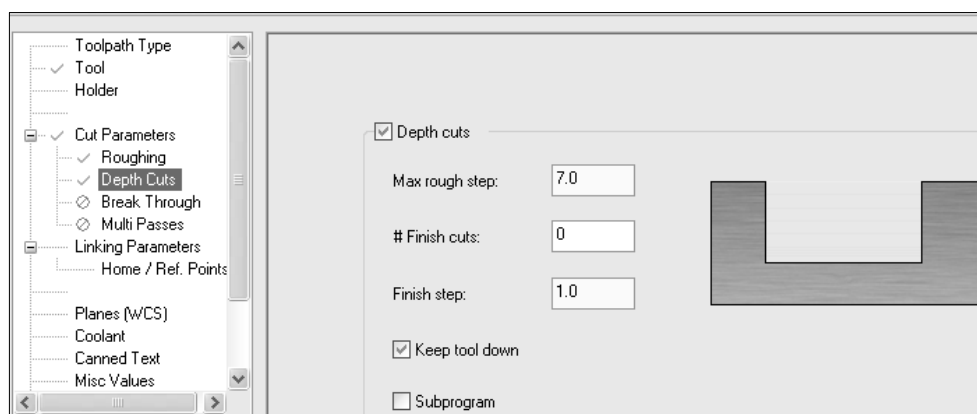
**Circle mill roughing** creates roughing passes using tangent arcs. The result provides a smooth motion for the tool, a short NC program, and good cleanout.

**Stepover** is set as a percentage of the tool diameter.

**Helical entry** parameters create the roughing motion tangent to a helical entry.

**Output arc moves** generates arc movements instead of small linear moves which make the program smaller.

- From the **Tree view list**, select **Depth Cuts** and set the parameters as shown.

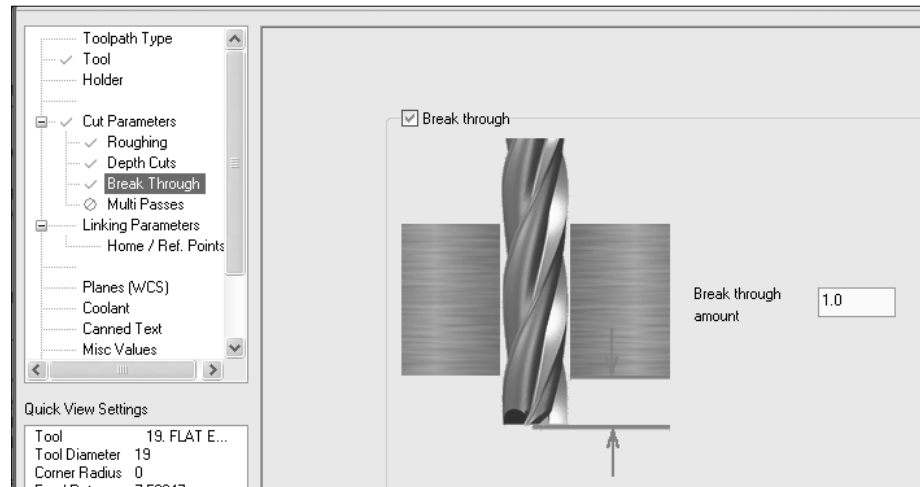


**Depth cuts** set the cuts along the Z-axis. Mastercam divides the total depth into equal steps based on the **Max rough step** value.

**Keep tool down** enabled doesn't allow the tool to retract between cuts.

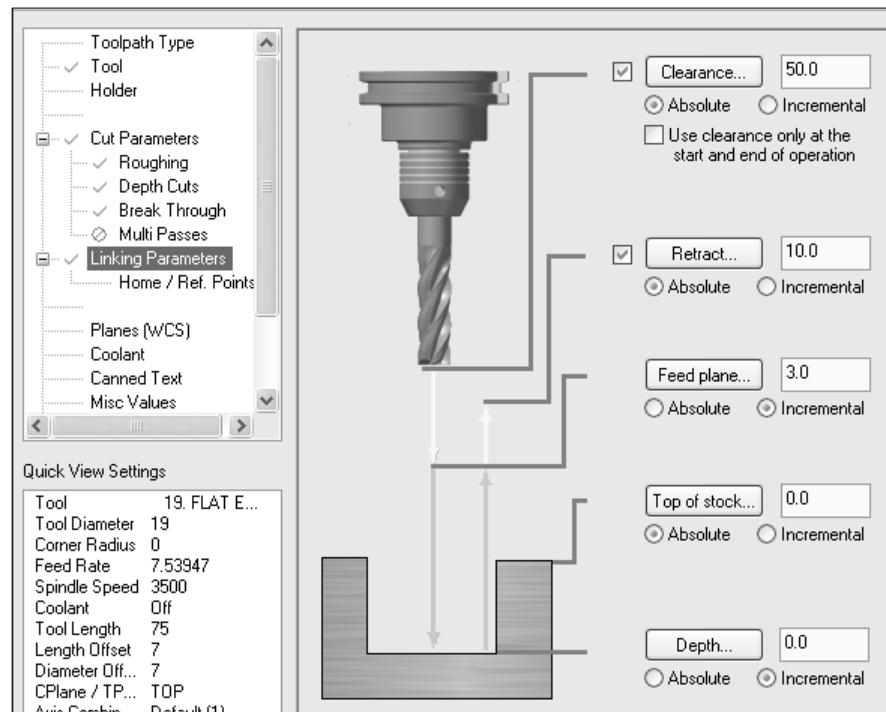
**Mill Level 1 - Metric**


- From the **Tree view list**, select **Break Through** to cut completely through the material by the amount specified.



**23.4 Set the Linking Parameters**

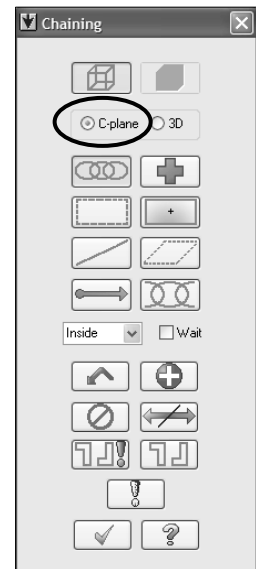
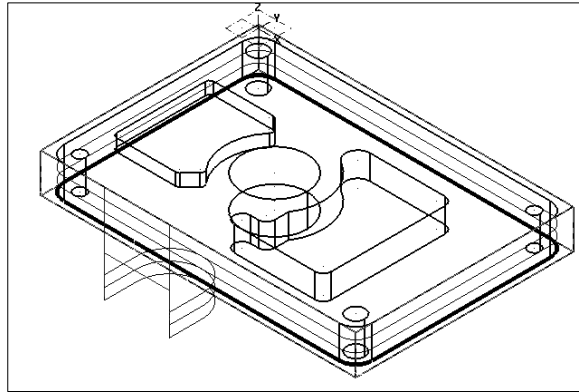
- From the **Tree view list**, select the **Linking Parameters** and change the parameters as shown.



- Note that the **Depth** is set to **Incremental** and **0** because we select the circle mill geometry at the bottom (final depth). Setting the **Depth** to **Absolute** and **-25** will give the same result.
- Select the **OK** button from the **2D Toolpath – Circle Mill** parameter screen. 
- Press **Alt + T** to remove the toolpath display.

## STEP 24: CONTOUR THE OUTSIDE PROFILE

*Toolpath Preview:*

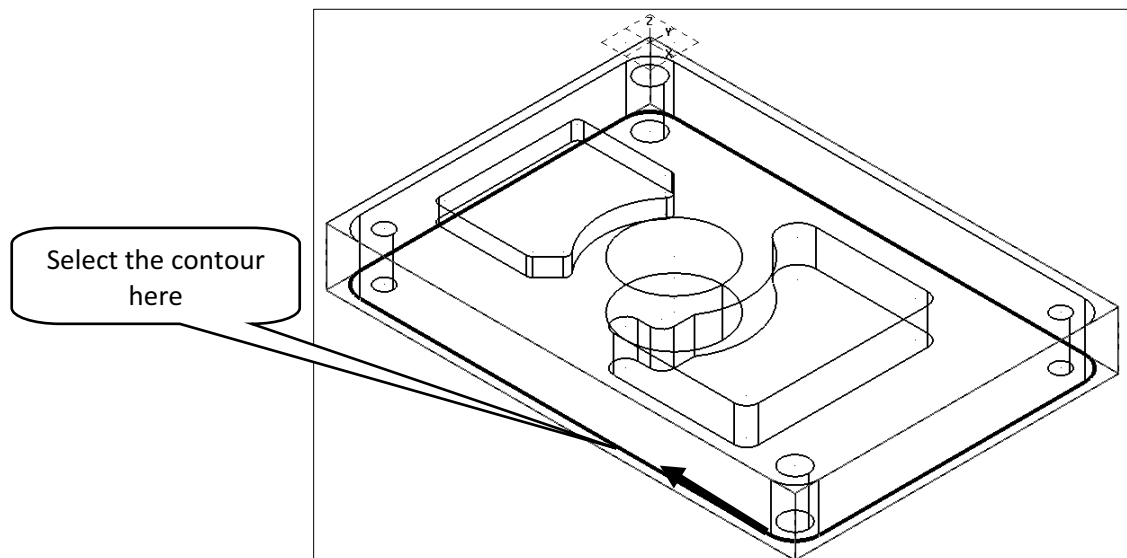


### 24.1 Select the toolpath geometry

#### Toolpaths

##### ➤ Contour

- Make sure that **C-plane** is selected in the **Chaining** dialog box.
- Select the contour at the bottom.
- Select the first entity in the contour, as shown.
- Be sure to chain the contour in a **CW** direction. Otherwise select the **Reverse** button.



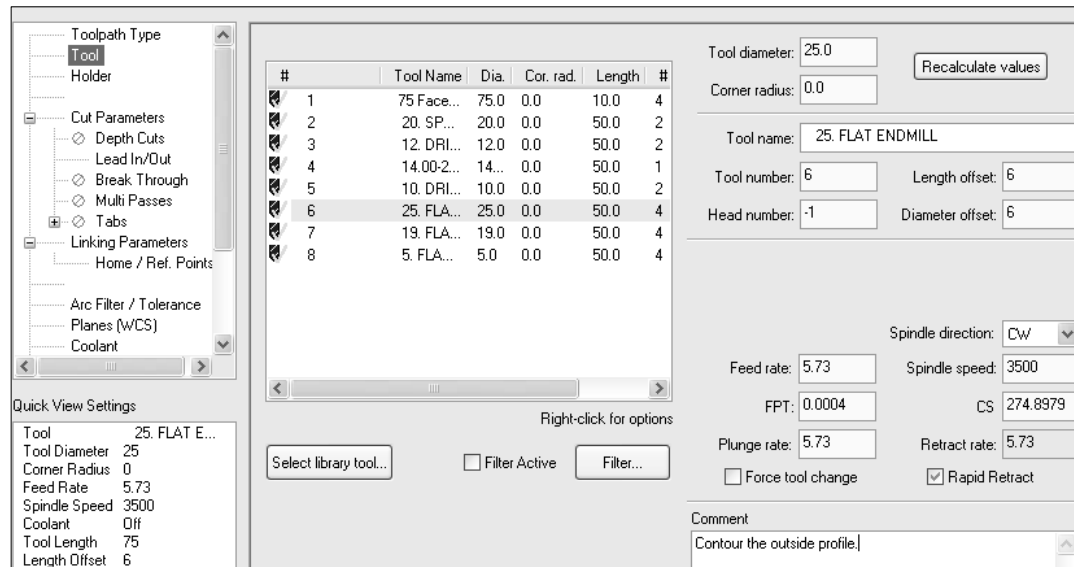
- Select the **OK** button to exit **Chaining**.



**Mill Level 1 - Metric**

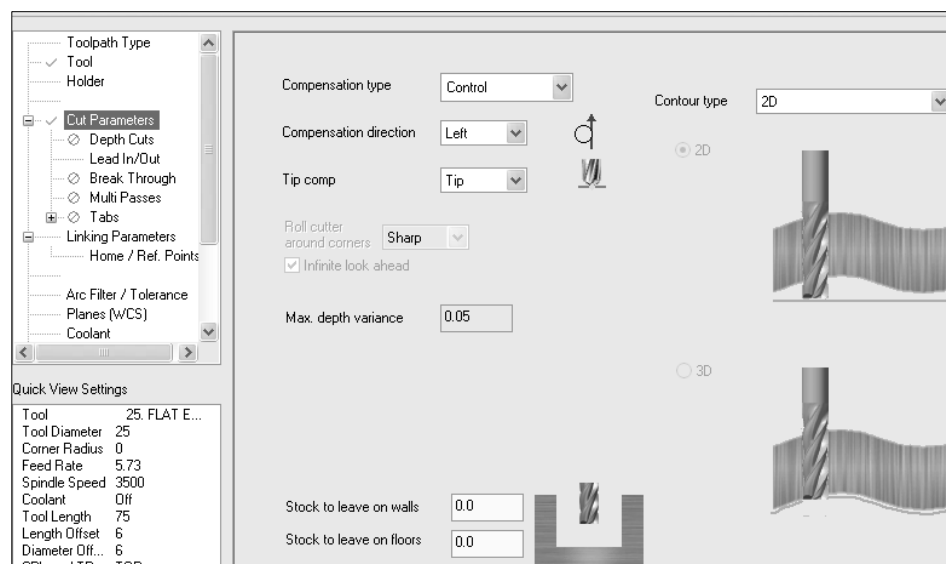
**24.2 Select the 25 mm diameter Flat End Mill and set the parameters in the Tool page**

- ➡ From the **Tree view list**, select the **Tool**.
- ➡ Select the existing **25 mm Flat Endmill** and make all the necessary changes in the **Tool** page.



**24.3 Set the Cut Parameters**

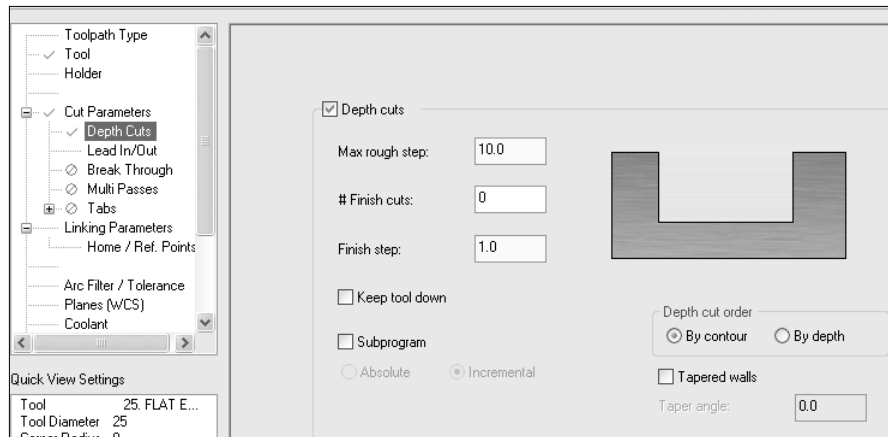
- ➡ Select the **Cut Parameters** from the **Tree view list** and change the **Compensation type** to **Control**.
- ➡ Make sure that the parameters match the screenshot below.



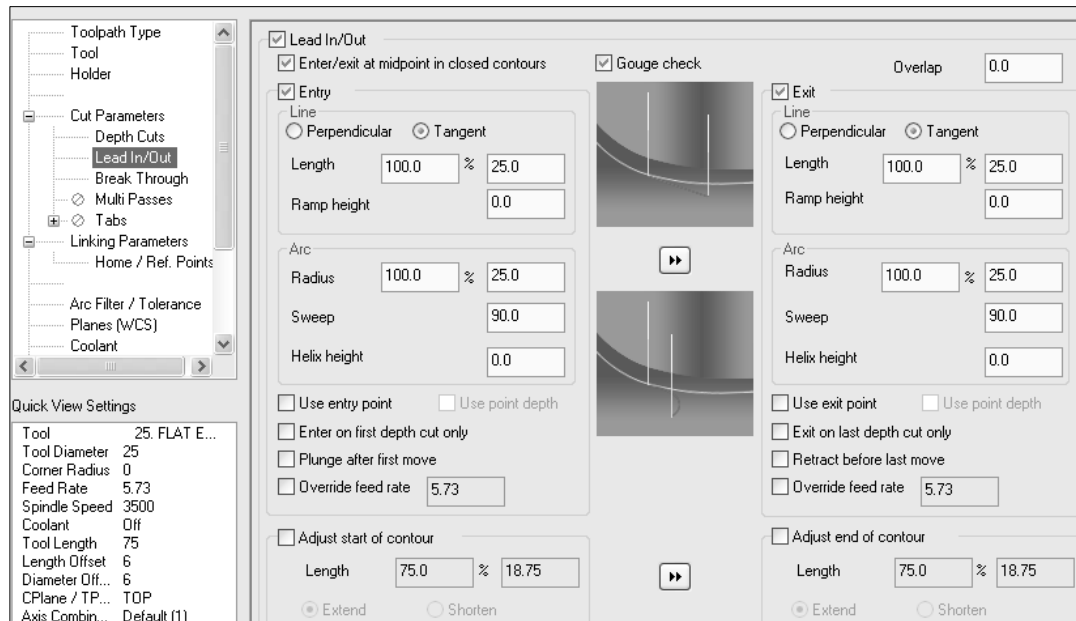
Cutter compensation set in Control outputs G41 or G42 codes to turn the compensation on. The compensation amount or tool wear adjustment is set at the control. Using this method you are programming the part edge.

### Mill Level 1 - Metric

- From the **Tree view list**, select **Depth Cuts**.
- Enable **Depth cuts** and set the **Max rough step** to 10 mm.

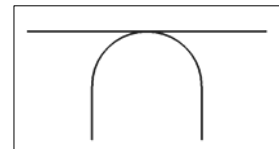


- From the **Tree view list**, select **Lead In/Out**.
  - Note that **Cutter compensation in control** requires the **Lead in/out** parameter to be turned on; most of the CNC machines need a linear move at the beginning of the program to compensate the cutter diameter. If you use the arc option too, make sure that the radius is larger than the radius of the tool.



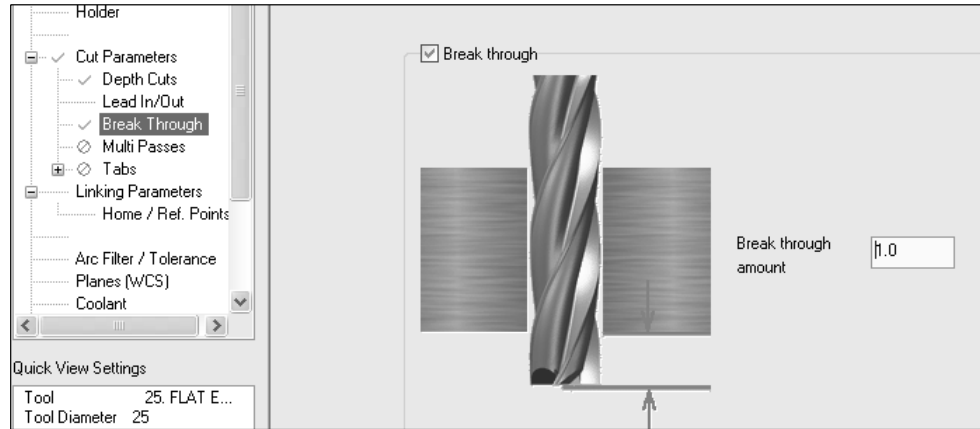
**Entry/exit at midpoint in closed contours** will move the start of the contour toolpath at the midpoint of the line that we selected when chaining.

**Line /Tangent** will be tangent to the arc and machined first in the entry combination. When exiting the part the arc will be machined first. See the graphic below.



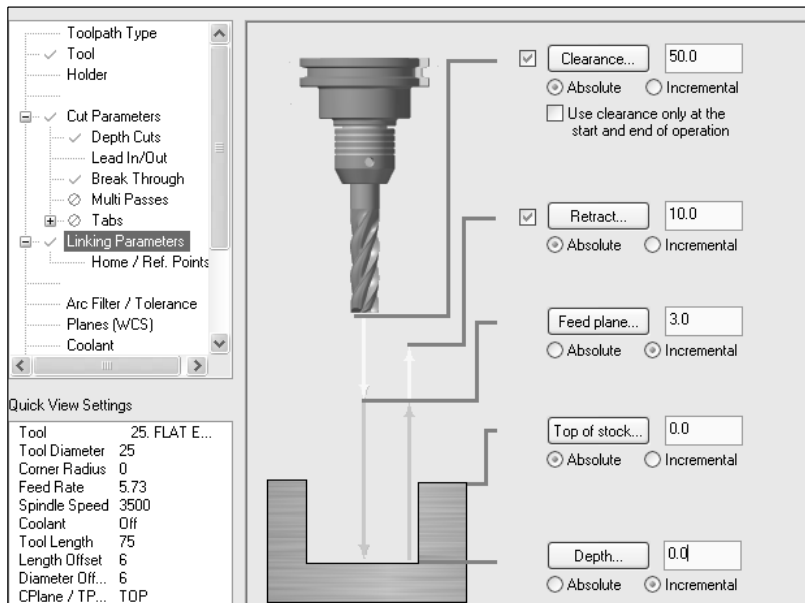
**Mill Level 1 - Metric-**

- From the **Tree view list**, select the **Break Through** and enter a **Break through amount** to completely machine the profile.



**24.4 Set the Linking Parameters**

- Select the **Linking Parameters** and make the changes as shown.

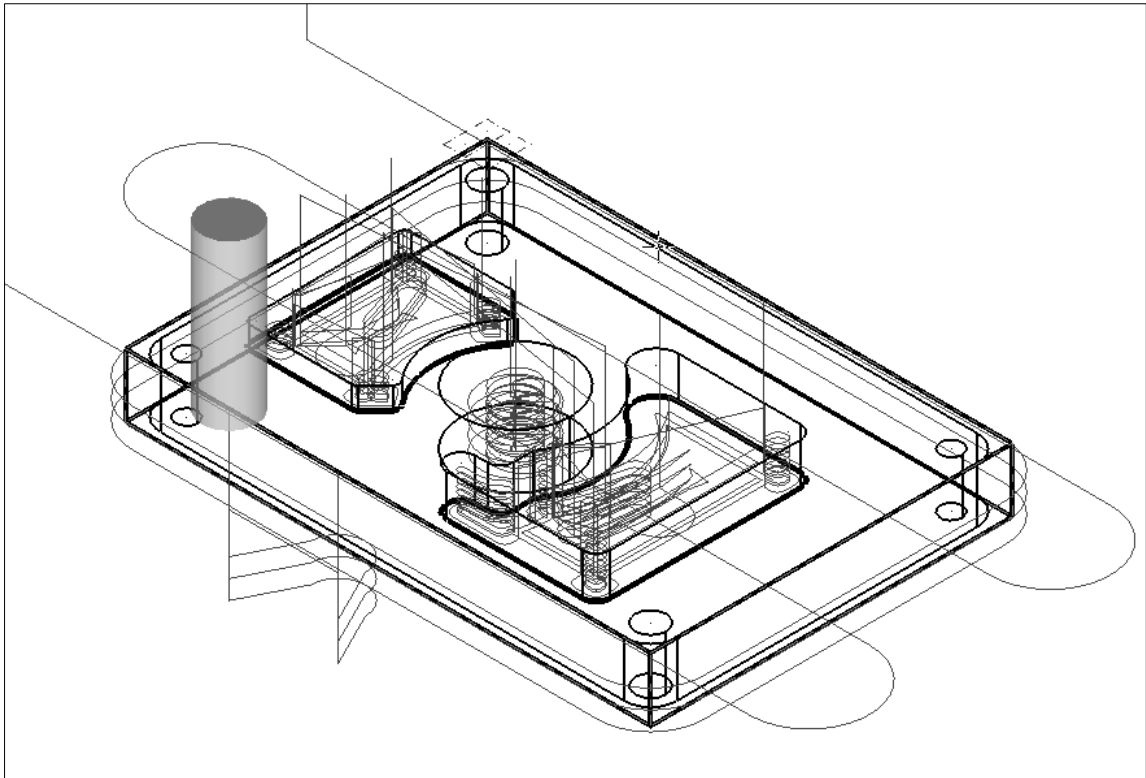


- Note that the final **Depth** can be set to **Incremental** and **0** due to the fact that we select as the contour geometry the bottom of the contour.

- Select the **OK** button to exit **2D Toolpaths – Contour** parameters. 

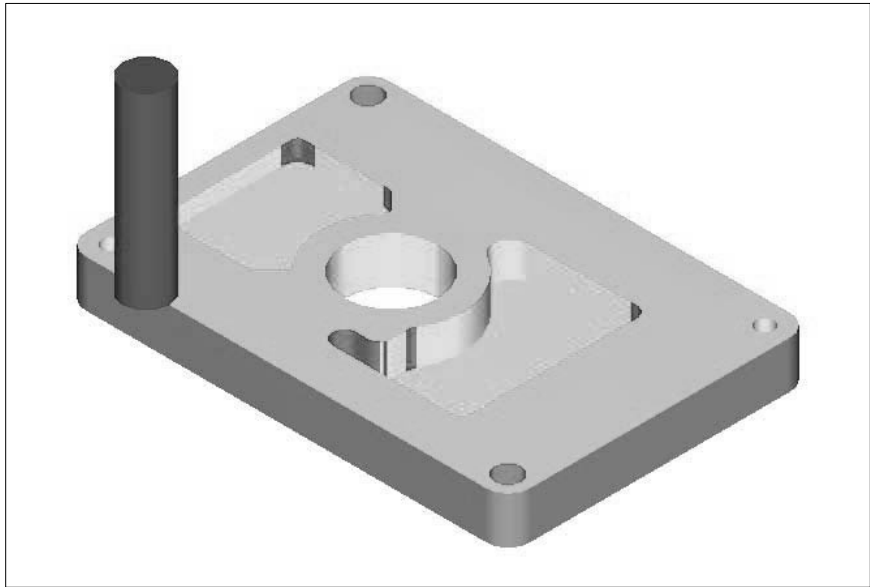
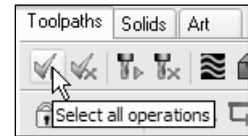
## **STEP 25: BACKPLOT THE TOOLPATHS**

- From **Toolpaths Operations Manager** click on the Machine Part Top group to select all the operations done in the second setup.
- Repeat **STEP 14** to be sure that the changes have taken place.
- Disable the **Quick verify** button.



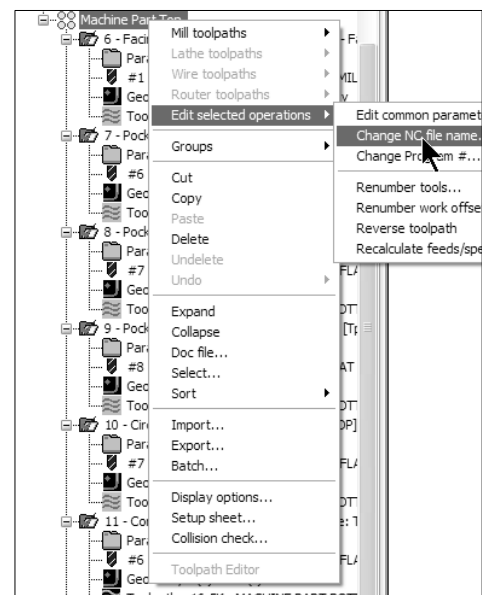
## STEP 26: VERIFY THE CHANGED PART

- Select the **Select all operations** button from **Toolpath Manager** if needed.
- Repeat **STEP 15** to be sure that the changes have taken place.
- The finished part should appear as shown in the following picture.



## STEP 27: POST THE FILES

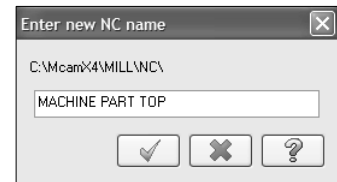
- Right-mouse click on the **Machine Part Top** group and select **Edit selected operations** and then, select **Change NC file name**.



### Mill Level 1 - Metric

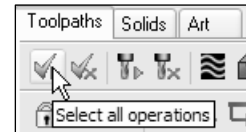
- Enter the new NC name: Machine Part Top

- Select the **OK** button. 

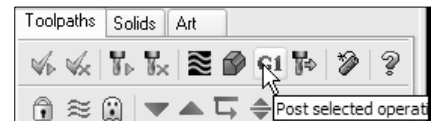


**Post processing**, or **posting**, refers to the process by which the toolpaths in the Mastercam part files are converted to a format that can be understood by the machine tool's control (for example, G-codes). Generally, every machine tool or control will require its own post processor, customized to produce code formatted to meet its exact requirements.

- Make sure that all operations are selected, otherwise, **Select all operations**.





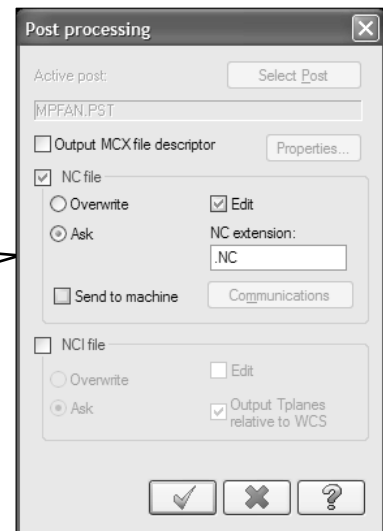
- Select the **Post selected operations** button from **Toolpath Manager**.



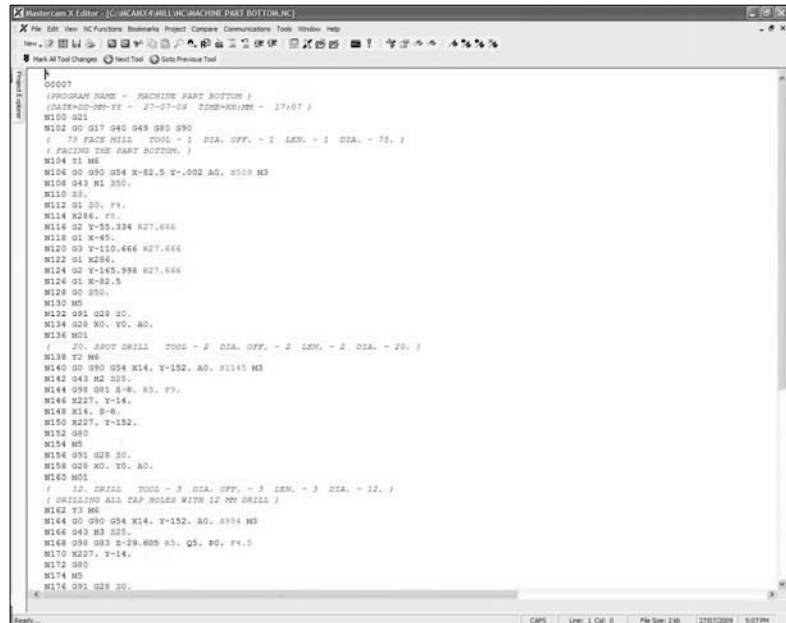
- In the **Post processing** window, make all the necessary changes as shown to the right.

**NC file** enabled allows you to keep the NC file and to assign the same name as the MCX file.  
**Edit** enabled allows you to automatically launch the default editor.

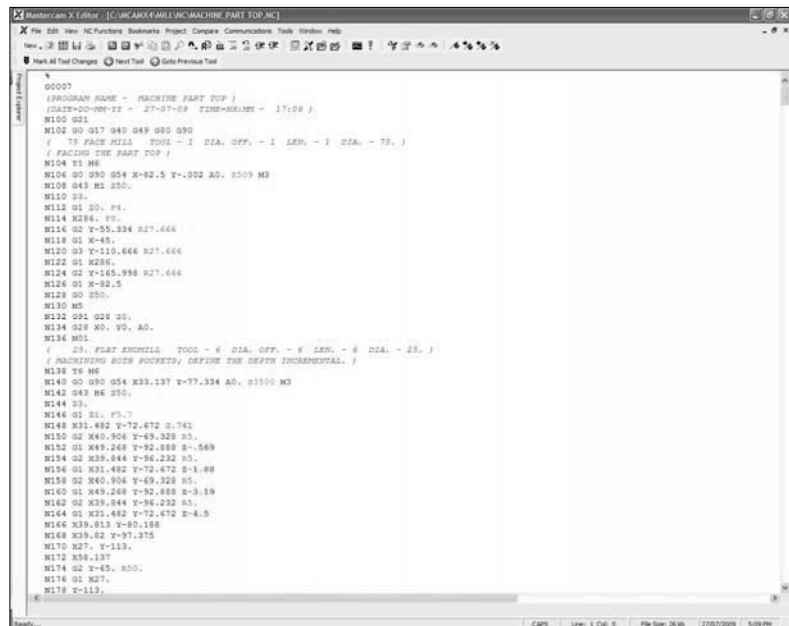
- Select the **OK** button to continue. 
- Select the **OK** button to post the first group as the geometry name ("Machine Part Bottom"). 



### Mill Level 1 - Metric



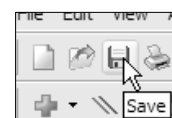
- ➔ Select the red **X** to close the first program.
- ➔ Select the **OK** button to start the second program ("Machine Part Top").



- ➔ Select the red **X** to close the editor.

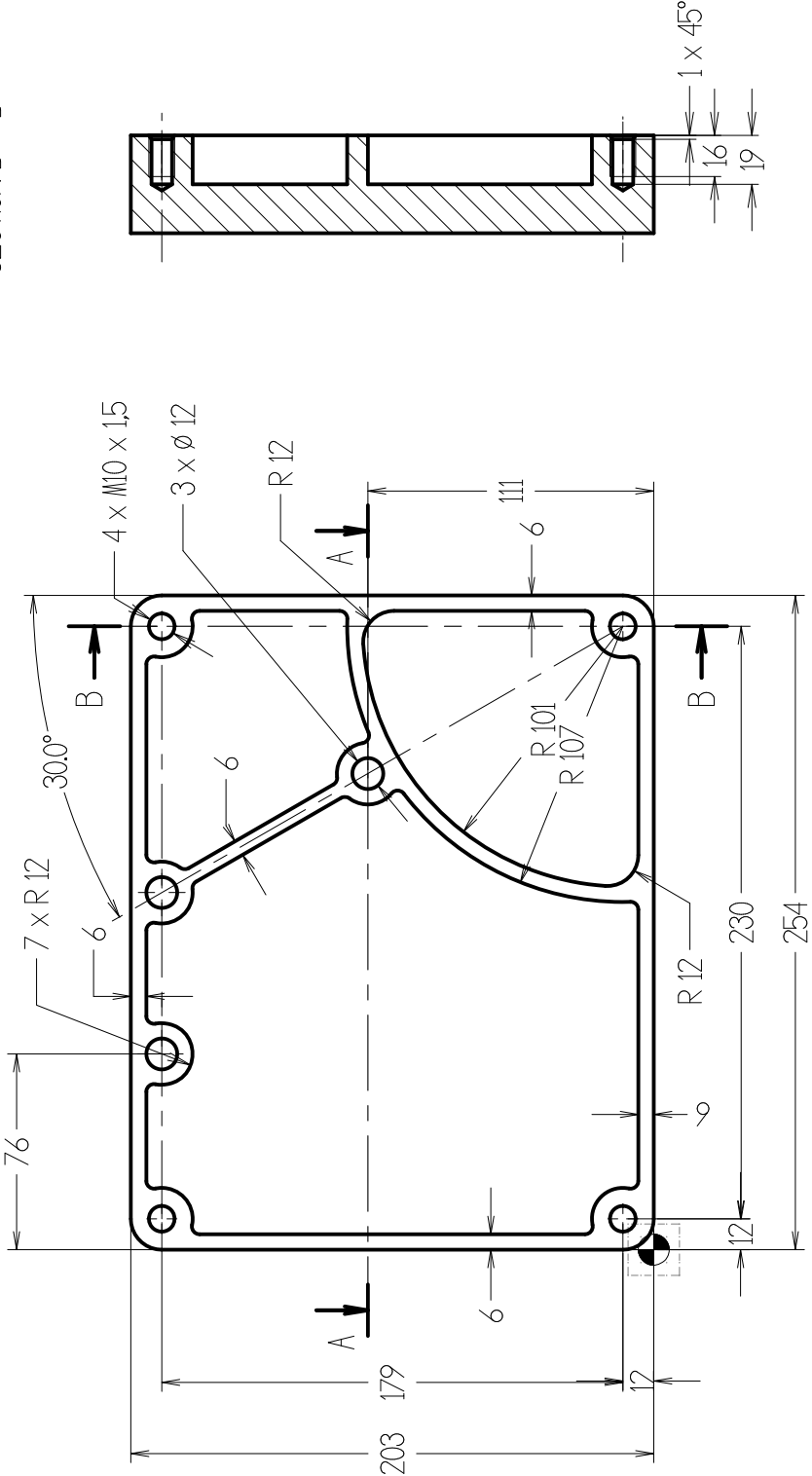
### STEP 28: SAVE THE UPDATED MCX FILE

- ➔ Select the **Save** icon.



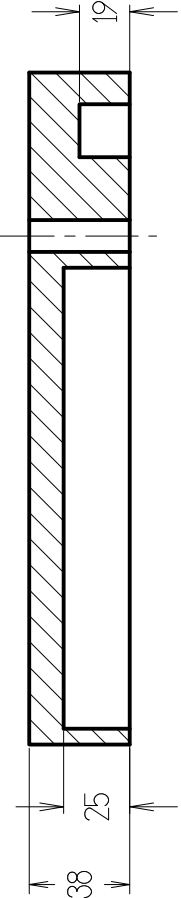
ALL DIMENSIONS IN MILLIMETERS

SECTION B - B



ALL FILLETS R3 UNLESS OTHERWISE SPECIFIED

SECTION A-A



TITLE TUTORIAL 7 - EXERCISE 1

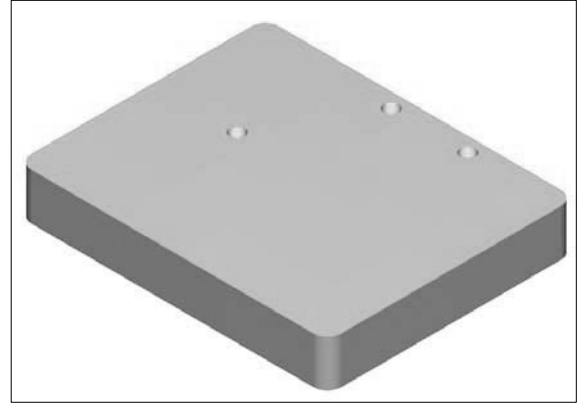
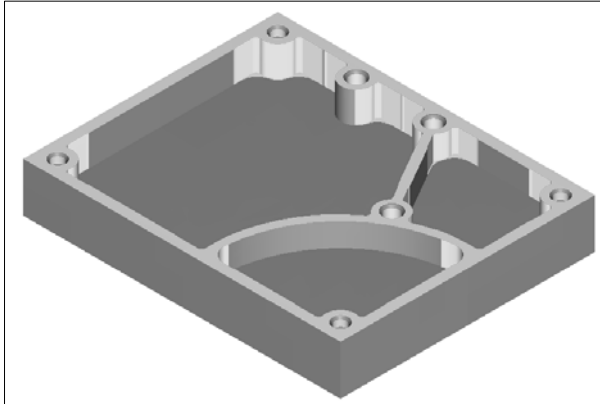
MATERIAL ALUMINUM

DATE: JULY 29, 2009 eMastercam.com



## REVIEW EXERCISES

*Student practice: Create the Toolpath for Tutorial 7 - Exercise 1 as per the instructions below;*



### Tips:

**Create the 3D geometry** using Xform Translate.

Create each pocket with a different depth.

**Stock** size use **Bounding box** to establish X, Y & Z sizes and give extend X, Y, and Z 2 mm

### Setup 1

**Face** the top of the part using 75 mm Face Mill

**Spot Drill** the 10 mm diameter holes using 15 mm Spot Drill

**Drill** the 10 mm diameter holes using 8.5 Drill (Create the new tool)

**Tap** the 10 mm diameter holes using 10 x 1.5 Tap

**Drill** the 12 mm diameter holes using 12 mm Drill

**Add** the 12 mm diameter holes center points in the **Spot Drill** and edit the depth of the holes. (See Tutorial 4)

**Rough Pocket** the part using 25 mm Flat Endmill

Select each pocket at the bottom

Stock to leave XY = 0,5

Use Parallel spiral clean corners cutting method

Depth = 0 (incr)

Max rough step (depth cuts) = 10 mm

Finish step 1 mm

**Remachining Pocket** the part using 5 mm Flat Endmill

Max rough step (depth cuts) = 3 mm

Finish step 1 mm

### Setup 2

**Face** the bottom of the part using 75 mm Face Mill

**Contour** the part using 25 mm Flat Endmill

**Backplot and Verify** the toolpaths.

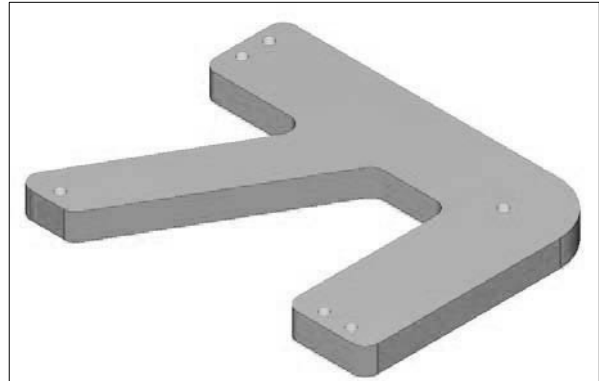
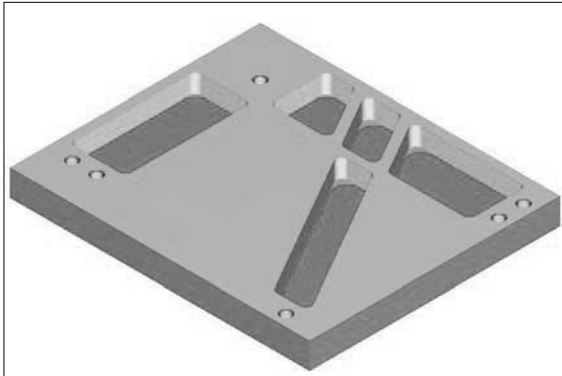
**Post process the file.**

[illegible]

DATE: JULY, 30 2009

**Mill Level 1 - Metric**

**Student practice:** Create the Toolpath for Tutorial 7 - Exercise 2 as per the instructions below;

**Tips:**

**Create the 3D geometry** using Xform Translate.

**Stock** size use **Bounding box** to establish X & Y sizes and give expand Z 2 mm. Create the stock geometry too.

**Setup 1**

**Face** the top of the part using 75 mm Face Mill

**Spot Drill** the 9 mm diameter holes using 15 mm Spot Drill; 6 mm deep

**Drill** the 9 mm diameter through holes using 9 mm Drill

**Rough all Pockets** using 25 mm Flat Endmill

Select each pocket at the bottom

Depth = 0 (incr)

Max rough step (depth cuts) = 10 mm

Stock to leave XY = 0,5

Use Parallel spiral cutting method

**Remachining Pocket** the part using 10 mm Flat Endmill

**Finish Pocket** the part using 10 mm Flat Endmill

**Setup 2**

**Face** the bottom of the part using 75 mm Face Mill

**2D High Speed Core Mill** using 25 mm Flat Endmill to machine the outside profile

Select the outside rectangle and the outside profile.

**Backplot and Verify** the toolpaths.

**Post process the file.**

**NOTES:**

## **TUTORIAL 7 QUIZ**

- ➡ What command and what options were enabled to create the 3D wireframe?
  
- ➡ What does WCS stand for?
  
- ➡ What entities can be selected to define a new plane, using the Geometry option in the View Manager?
  
- ➡ What is facing toolpath used for?
  
- ➡ What would you require as toolpath geometry, and how would you set the final Depth value when machining pockets with different depths in the same operation?
  
- ➡ How does the system calculate the stock for remachining when “Compute remaining stock from Roughing tool diameter” is enabled?
  
- ➡ How is the tip length of the drill calculated in the drilling operation?