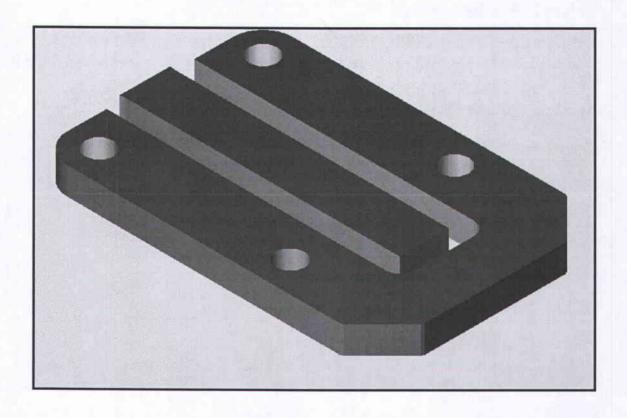
## **TUTORIAL SERIES FOR**

## Mastercam.X2

# TUTORIAL #1 LEVEL 1 - 2D GEOMETRY, DRILL & CONTOUR.





#### **Objectives:**

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

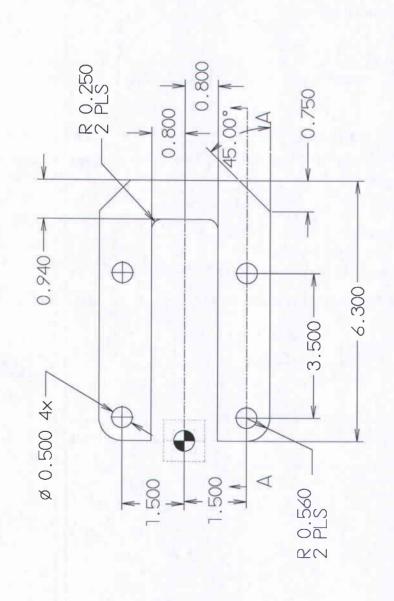
- > Creating a rectangle knowing the width and the height of the rectangle.
- > Creating parallel lines, by defining the offset direction and distance.
- > Trimming 1 entity to another existing entity.
- > Creating fillet radii.
- > Creating 45 degree chamfers.
- > Creating arcs knowing the diameter and the center point.
- > Mirroring existing geometry to complete a part.

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

- > 4 holes drilled through the part.
- > A 2-dimensional contour.
- > Lead in and lead out toolpath entry.
- ➤ Modifing the existing contour toolpath by adding 2 depth cuts, 2 roughing passes and 1 finish pass.

#### 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.



TUTORIAL 1 TITE

SECTION A - A

ALUMINUM T6067 MATERIAL

DATE: JUNE 12, 2005 | eMastercam.com



# **GEOMETRY CREATION**Setting the toolbar states

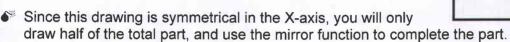
Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-5 in the **User Notes**.

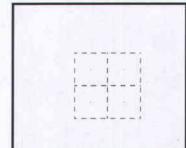
Select the double

arrow

● Toolpaths/Solids Manager to the left of the screen can be hidden to gain more space in the graphic area for design. Press Alt + O to remove it.

Before starting the geometry make sure that the Grid is enabled. It will show you at each moment where the part origin is. See Getting started page A-5 for details.





#### STEP 1:

#### CREATE HALF THE OUTSIDE BOUNDARY.

1.1 Create a rectangle knowing the width, the height and the base point.

#### Create

> Create Rectangular Shapes

> Select the radio button in front of Base Point.

- > Enter the Width and the Height as shown in the screenshot to the right.
- ➤ Select the double arrow to expand the **Rectangular Shapes Options** as shown.

Rectangular Shapes Options

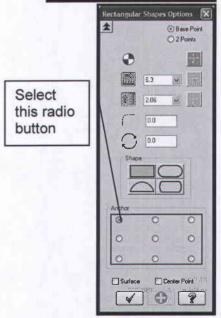
Base Point

2 Points

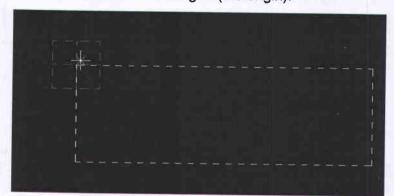
6.3

2.06

> Select the upper left corner radio button as the anchor.



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



➤ Select the OK button to exit the Rectangular Shapes Options 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 you can undo the last step using 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.

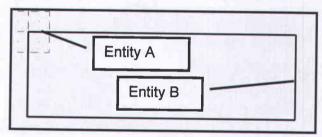
#### STEP 2: CREATE THE INSIDE GEOMETRY. 2.1 Create parallel lines.

#### Create

➤ Line

➤ Create Line Parallel

➤ [Select a line]: Select Entity A.



> [Select the point to place a parallel line through]: Pick a point lower than the selected line.

Note that the color of the geometry is cyan which means that the entity is "alive" and you can still change the line parameters.

> Enter the Distance 0.8 (Enter).

To continue using the same command select the **Apply** button. To exit the command you can either start a new command or select the **OK** button.

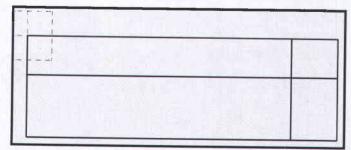
> Select the Apply button to continue.

➤ [Select a line]: Select Entity B.

> [Select the point to place a parallel line through]: Pick a point to the left of the selected line.

> Type the **Distance** 0.94 (Enter).

➤ Select the **OK** button. The drawing should look as shown to the right.



Point C

**Entity D** 

#### 2.2 Trimming one entity.

#### **Edit**

- > Trim/Break
- > Trim/Break/Extend

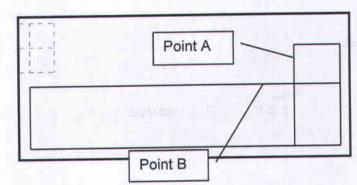


- > Select Trim 1 Entity. Trim 1 entity
  > [Select the entity to trim/extend]:
- > Select the entity at Point A (select all points exactly as shown in the drawing).
- > [Select the entity to trim/extend to]: Select Point B.
- > [Select the entity to trim/extend]: Select Point C.
- > [Select the entity to trim/extend to]: Select Point D.
- > Select the **OK** button.



#### Create

- > Fillet
- > Fillet Entities
- > Enter the fillet **Radius** ① 0.25.
- > [Select an entity]: Select Point A.



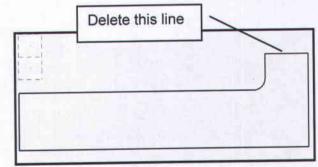
Point B

Point A

- Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities.
- > [Select another entity]: Select Point B.
- > Select the **OK** button.



- > Select the horizontal line as shown to the right.
- > Select the **Delete** entity icon.



#### STEP 3:

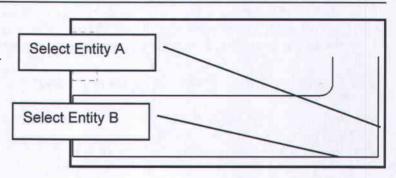
#### COMPLETE THE OUTSIDE PROFILE.

3.1 Create the 45 degree chamfer.

#### Create

- > Chamfer
- > Chamfer Entities
- > Make sure that 1 Distance | 1 Distance | is selected and the Trim option is enabled.

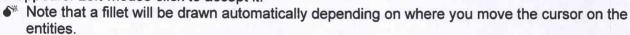
- > Enter the **Distance 1** 0.75 (Enter).
- > [Select line or arc]: Select Entity A.
- > [Select line or arc]: Select Entity B.
- > Select the OK button



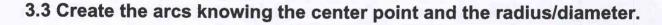
#### 3.2 Create a fillet.

#### Create

- > Fillet
- > Fillet Entities
- > Enter the fillet Radius 0.56 (Enter).
- [Select an entity]: Move the cursor in the corner area until the desired fillet appears. Left-mouse click to accept it.

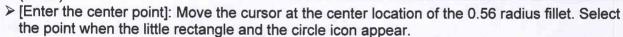


> Select the **OK** button.



#### Create

- > Arc
- > Create Circle Center Point
- > Enter the **Diameter** value 0.5 (Enter).

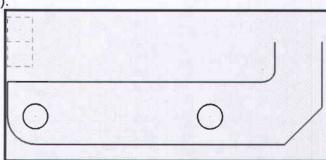


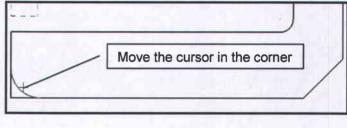
radius values will be highlighted in red.

> [Enter the center point]: Select the Fast Point icon.

> Enter the coordinates: 0.56+3.5, -1.5 (Enter).

> Select the **OK** button to exit







Note that Mastercam will perform basic math operations (+, -, \*, /). You can enter the values without any of the corresponding coordinate letters (X, Y, Z) as long as you enter them in this order and separate them by commas.

#### 3.4 Transform the geometry to represent the whole part.

#### **Xform**

> Xform Mirror

> [Select entities to mirror]: Click on the All button.



> Select the **OK** button in the **Select All** dialog box.



> Click on the End Selection button.

> Set the parameters to match the following dialog box (keep the original entities Copy, to mirror about X-axis, Fit the geometry to the screen).

01 3

End Selection

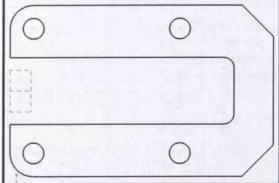
> Select the **OK** button to exit the **Mirror** window.



#### Screen

> Clear Colors

> The part should look as shown in the following picture.



#### STEP 4: SAVE THE FILE.

File

> Save As

➤ File name: "Your Name\_1"

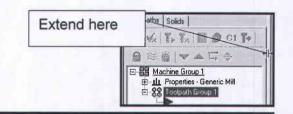
> Select the **OK** button.





# TOOLPATH CREATION STEP 5: SET UP THE STOCK TO BE MACHINED.

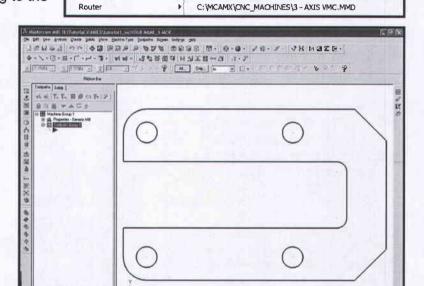
- > To display the Toolpaths Manager press Alt + O.
- > Extend the Toolpaths Manager if necessary.
- ➤ Machine type
- > Mill
- ➤ Default



C: WCAMX CNC MACHINES 3 - AXIS HMC.MMD

> Use the Fit icon to fit the drawing to the





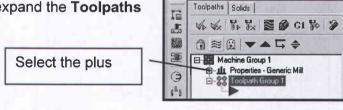
Screen Settings

Machine Type Toolpaths

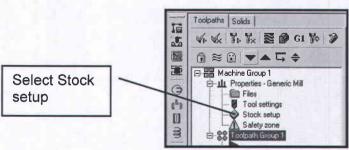
Mil

Lathe

➢ Select the plus in front of Properties to expand the Toolpaths Group Properties.



> Select Stock setup.



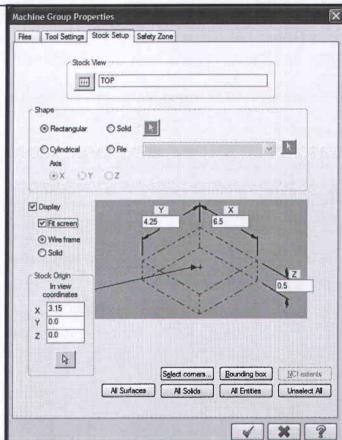


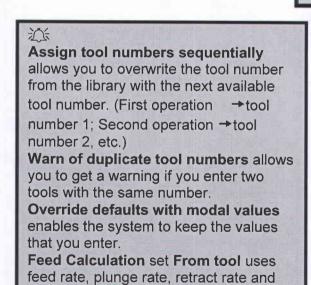
Change the parameters to match the screenshot to the right.

The Stock Origin values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finish part.

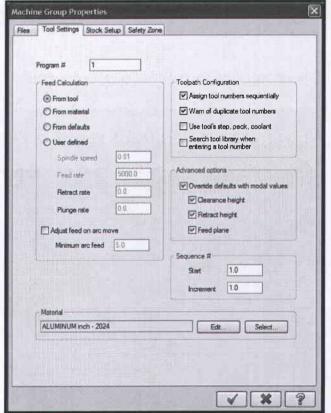
Display options allows you to set the stock as Wireframe and to fit the stock to the screen.(Fit Screen)

- Select the Tool Settings tab to set the tool parameters and the part material.
- Change the parameters to match the screenshot to the right.



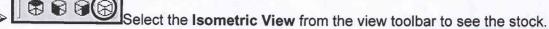


spindle speed from the tool definition.





Select the OK button to exit Toolpath Group Properties.



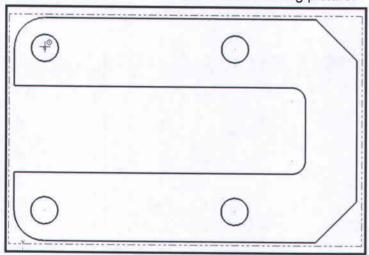
> Select the **Top View** from the view toolbar to see the part from the top.



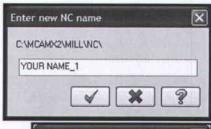
#### STEP 6: CENTER DRILL 4 X ½" DIAMETER HOLES.

#### **Toolpaths**

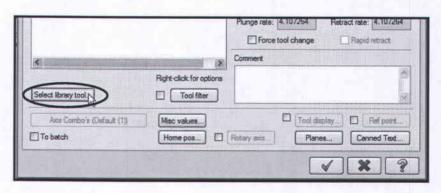
- **▶** Drill Toolpath
- > Select the **OK** button to accept the NC file name the same as the geometry name.
- > Select the 4 circle centers by moving the cursor at the center of the arcs as is shown in the following picture.



- > Select the **OK** button, once you select all four centers.
- > Select the Select library tool button.

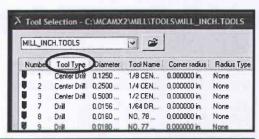




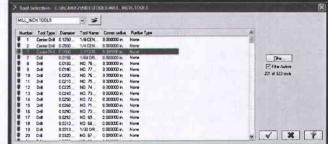




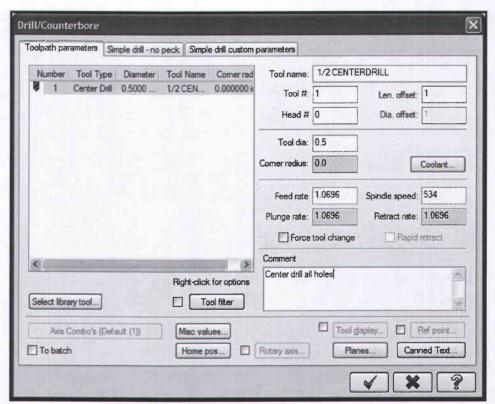
Click on the **Tool Type** to organize the tool list by the type.



- > Select the 1/2" Center Drill from the list.
- > Select the **OK** button.



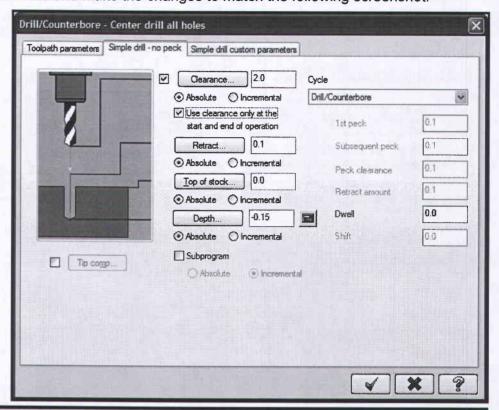
- > Make the changes in the Toolpath parameters as shown in the following screenshot.
- The Feed rate,
  Plunge rate,
  Retract rate and
  Spindle speed
  are based on the
  tool definition.
  Change them as
  desired.



> In the Comment field type a comment about the toolpath for future reference.



> Select the Simple drill tab and make the changes to match the following screenshot.



迩

Clearance value sets the height at which the tool rapids to or from the part.

Retract value sets the height at which the tool rapids/feed-rates up to, before the next step down.

Depth value sets the final machining depth for the drilling operation.

> Select the **OK** button to exit drilling parameters.

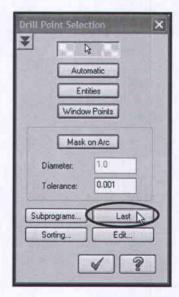


#### STEP 7:

#### DRILL 4 X 1/2" DIAMETER HOLES.

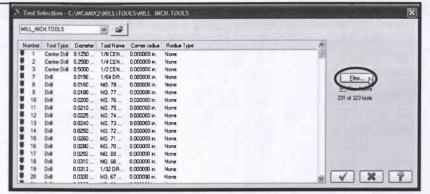
#### **Toolpaths**

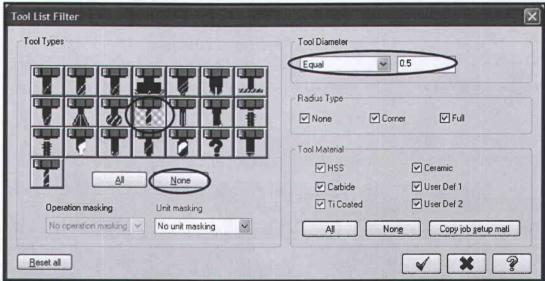
- ▶ Drill Toolpath
- > Select the Last button in the Drill Point Selection dialog box to select the same center points as you did before.
- > Select the OK button to exit Drill Point Selection.





- > Click on the Select library tool button.
- > Select the Filter button in the Tool Selection dialog box.





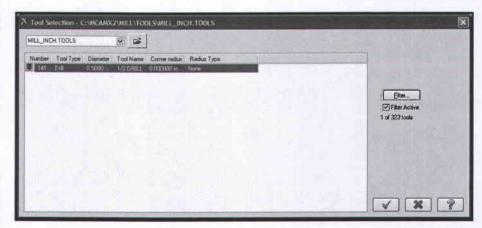
> Select

the None button to disable any previous tool selection as shown.

- > Select the **Drill** in the **Tool Types** list.
- > Select the drop-down arrow in the Tool Diameter field and select Equal.
- > Enter 0.5 in the Tool Diameter value box.
- > Select the OK button to exit Tool List

Filter.

> Make sure that the tool in the Tool Selection window is highlighted. Otherwise select it.

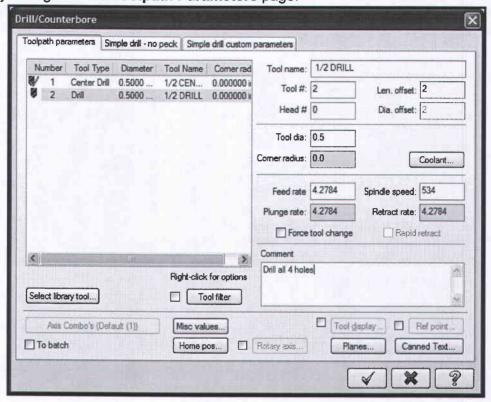


> Select the **OK** button to exit **Tool Selection**.



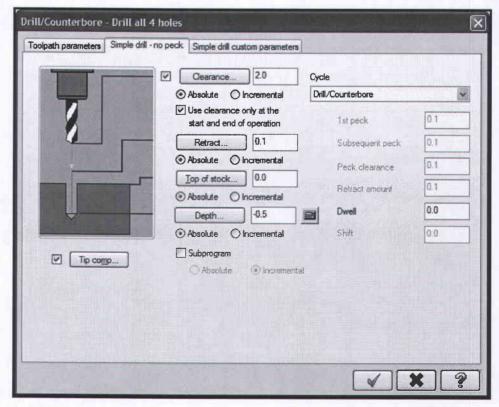


> Make any necessary changes in the Toolpath Parameters page.



> Select the Simple drill-no peck page and enter the Depth value as shown in the following

screenshot.

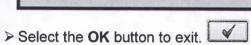




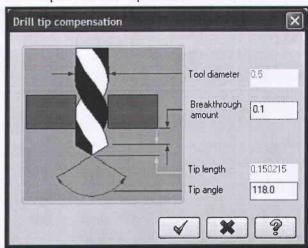
> Enable the **Tip comp...** to cut deeper than the final depth with the tip of the drill.

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 **Drill Parameters** page.



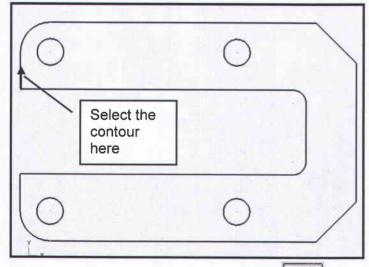
#### STEP 8:

#### CONTOUR THE OUTSIDE PROFILE.

#### **Toolpaths**

- **>** Contour Toolpath
- > Select the first entity in the contour, as shown.
- Be sure to chain the contour in a CW direction.
- > Otherwise select the **Reverse** button.



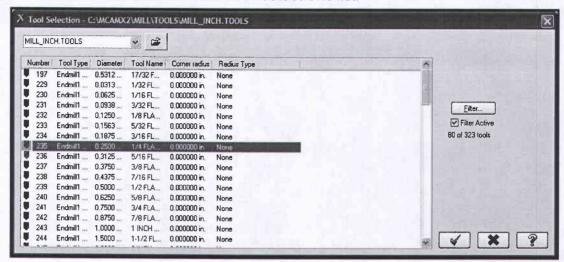


> Select the OK button to exit Chaining.





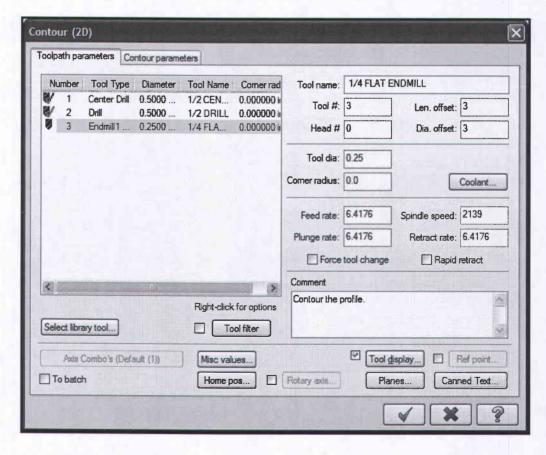
- > Click on the Select library tool button.
- > Select the 0.25 diameter Endmill from the Tool Selections list.



➤ Select

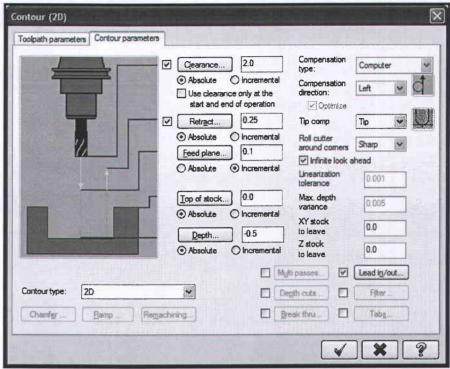
the **OK** button to exit **Tool Selection**.

> Make all the necessary changes as shown below.



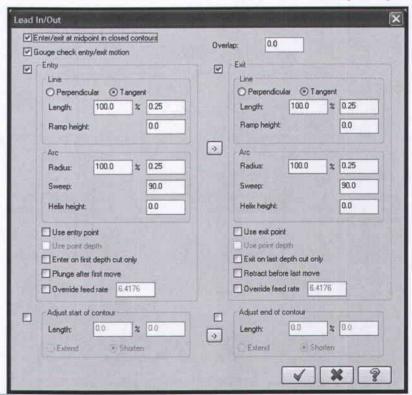


> Select the Contour parameter page and make the changes as shown below.



➤ Enable the box in front
of the **Lead in/out** button and select a combination of one **Line** and an **Arc** at the beginning

and/or end of the contour toolpath for a smooth entry/exit while cutting the part.



Select the OK button to exit the Lead In/Out dialog box.

> Select the **OK** button again to

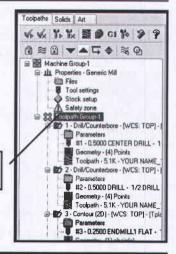
exit Contour parameters.



#### STEP 9: BACKPLOT THE TOOLPATH.

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

Select Toolpaths Group



> Select the Backplot selected operations button.

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

Display tool

Display rapid moves



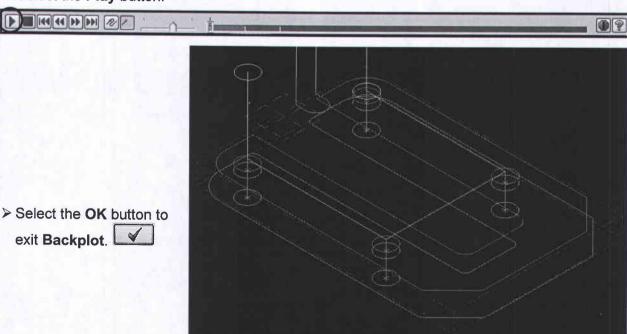
水 ≪ 乳 製 **配 ●** G1 乳 ●

Toolpaths Solids

> Select the Isometric View from the view toolbar to see the stock.

> Select the Fit button.

> Select the Play button.





#### **STEP 10:**

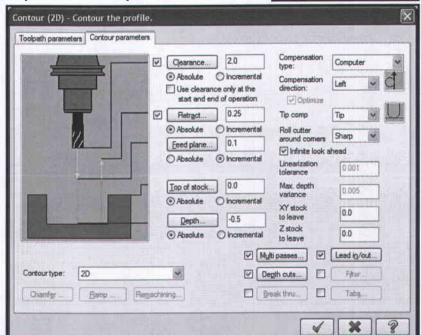
MODIFY THE EXISTING CONTOUR TOOLPATH (ADDING 2 DEPTH CUTS, 2 ROUGH PASSES AND 1 FINISH PASS).

> In the Toolpaths Manager select the contour Parameters.

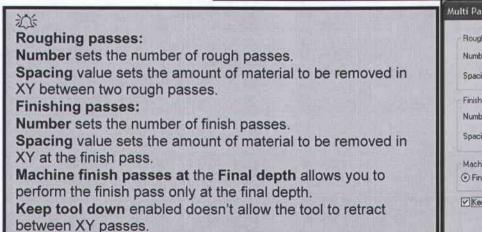
Select contour Parameters

Toolpaths | Solids | 4 4 1. 1 3 € G1 14 9 Toolpath Group 1 Properties - Generic Mill Tool settings
Stock setup A Safety zone 1 - Simple drill - no peck - [WCS: TO Parameters #1 - 0.5000 CENTER DRILL -Geometry - (4) Points
Toolpath - 4.5K - TOOLPATH 6 2 - Simple drill - no peck - [WCS: TO #2 - 0.5000 DRILL - 1/2 DRILL Geometry - [4] Points
Toolpath - 4,5K - TOOLPATH\_6 3 Contour (2D) - [WCS: TDP] - [Tpl Parameters #3 - 0.2500 ENDMILL1 FLAT -Geometry - [1] chain(s) Toolpath - 7.4K - TOOLPATH\_G

- > Select the Contour parameters page.
- > Check the boxes in front of Multi passes and Depth cuts.



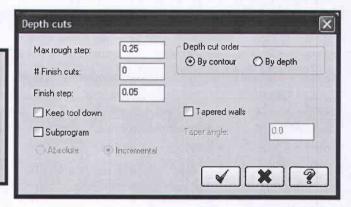
Select the Multi passes button to set the tool movements in the XY plane.



# Mastercam. TUTORIAL 1

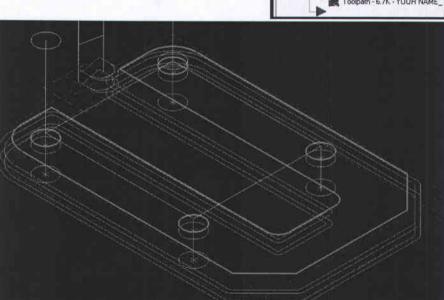
- > Select the **OK** button to exit.
- Select the **Depth cuts** button to set the Z-axis cuts.

Max rough step value sets the maximum amount of material removed in the Z-axis for each cut. The system will automatically divide the material in equal cuts until it reaches the final depth.

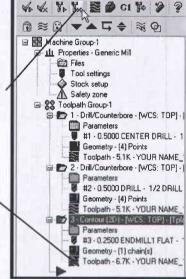


- > Select the **OK** button to exit.
- Select the **OK** button to exit the **Contour parameters** dialog boxes.
- Note the red X in front of the Toolpath description in the Contour operation. To update the toolpath with the new parameters you have to Regenerate it.
- > Select the Regenerate all dirty operations button.

Select Regenerate all dirty operations



Red X



Toolpaths Solids Art



#### VERIFY- TOOLPATH VERIFICATION STEP 11: VERIFY.

- > Expand the **Toolpaths Manager** if necessary by dragging the right side.
- > Select the **Select all operations** button.
- > Select the Verify selected operations button.



**沁** 

**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.





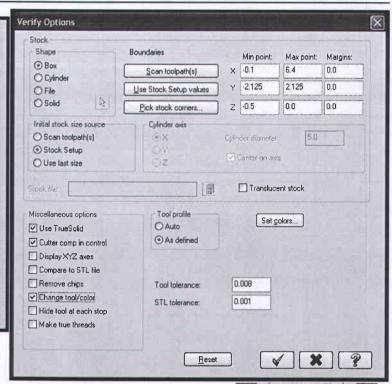
江

Initial stock size source should be set to Job 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.

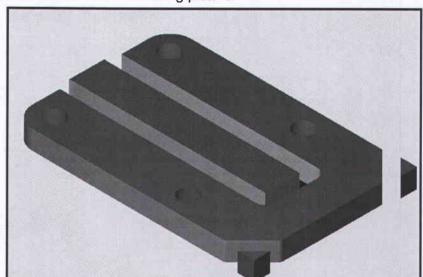
➤ Select the **OK** button to exit **Verify Options**.



> Set the Verify speed by moving the slider bar in the speed control bar.

> Select the Play button to start simulation.

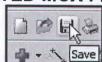
> The finished part should appear as shown in the following picture.



> Select the **OK** button to exit

#### **STEP 12:**

SAVE THE UPDATED MCX FILE.



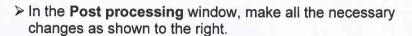
> Select the Save icon.



#### **STEP 13:**

#### POST THE FILE.

- > Make sure that all operations are selected, otherwise:
- > Select all operations
- ➤ Select the **Post selected operations** button from **Toolpath Manager**.



NC file enabled allows you to keep the NC file and to assign the same name as the MCX file.

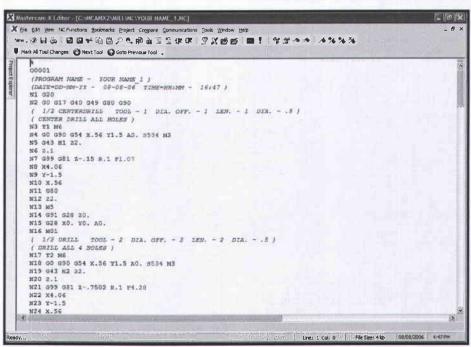
Edit enabled allows you to automatically launch the default editor.

- > Select the **OK** button to continue.
- Select the Save button to accept in the File name field the same name as the geometry name ("Your Name 1").

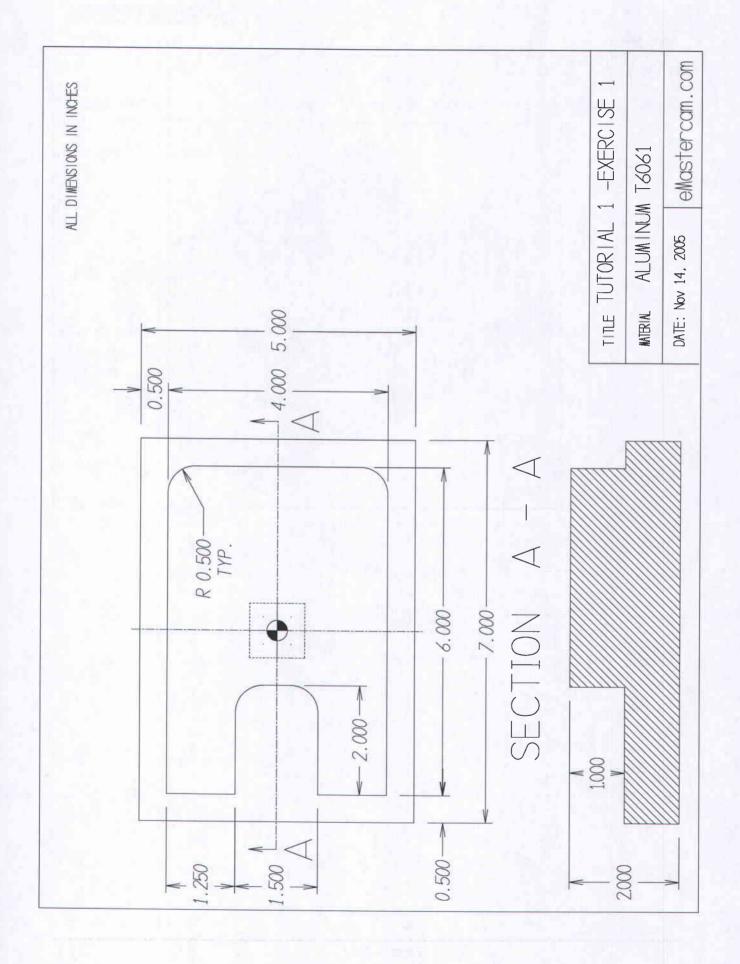








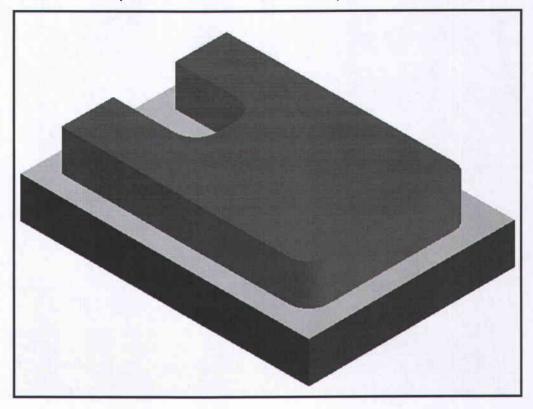
> Select the red X box at the upper right corner to exit the Editor.





#### **REVIEW EXERCISE 1.**

Student practise. Create the Toolpath for Exercise 1- Tutorial 1 as per the instructions below;



Tips:

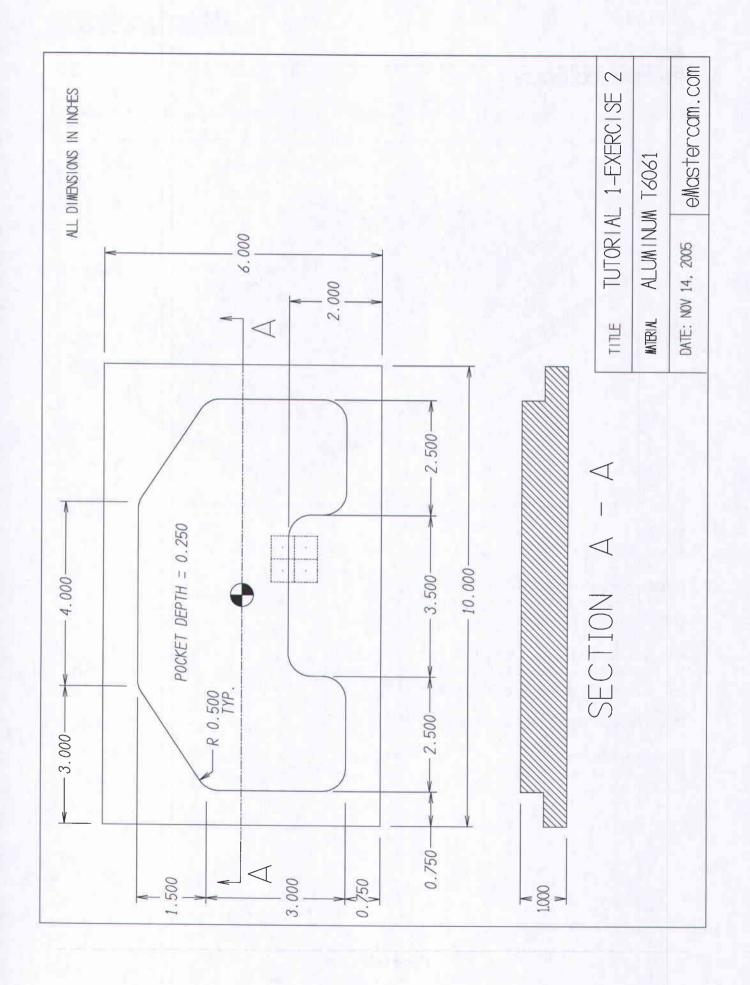
**Stock** size; X = 7.0, Y = 5.0 and Z = 2.0

Contour-rough the part using 1.0 " Flat End Mill; Max rough step =0.50" (Depth cuts) 2 roughing passes with spacing = 0.40" Stock to leave in XY and Z axis =0.05"

Contour -finish the part using the 0.50" Flat End Mill Stock to leave in XY and Z axis =0.0" 3 roughing passes with spacing = 0.20" Depth Cuts disabled

Backplot and Verify the toolpaths.

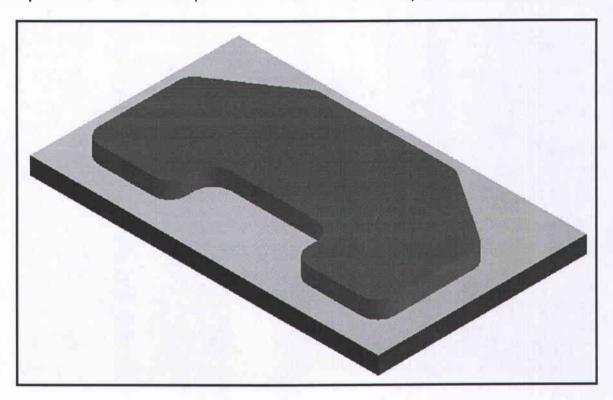
Post process the file.





#### **REVIEW EXERCISE 2.**

Student practise. Create the Toolpath for Exercise 2- Tutorial 1 as per the instructions below;



Tips:

**Stock** size; X = 10.0, Y = 6.0 and Z = 1.0

Contour-rough the part using 1.0 " Flat End Mill; 4 roughing passes with spacing = 0.45" Stock to leave in XY and Z axis =0.03"

Contour -finish the part using the 0.750" Flat End Mill Stock to leave in XY and Z axis =0.0" 6 roughing passes with spacing = 0.35"

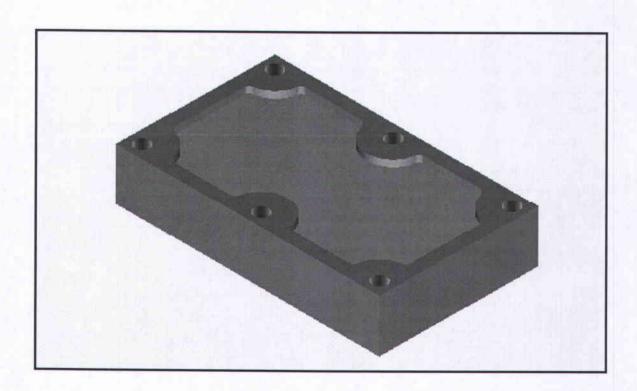
Backplot and Verify the toolpaths.

Post process the file.

### **TUTORIAL SERIES FOR**

## Mastercam, Xº

TUTORIAL 2
LEVEL 1 - 2D GEOMETRY, TAPPING, CONTOURING,
POCKETING, ASSOCIATIVITY, MODIFY THE POCKET
TOOLPATH, AND SAVE TAPPING OPERATIONS TO A
LIBRARY.





#### **Objectives:**

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

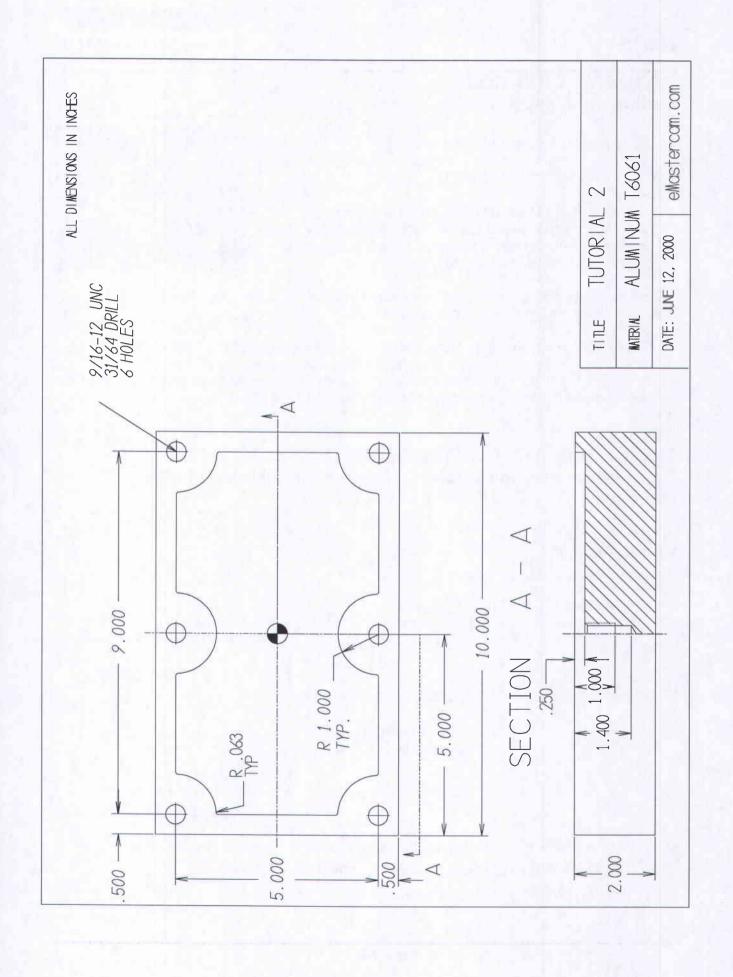
- > Creating a rectangle using the 2 points entry method.
- > Creating parallel lines by defining the offset direction and distance.
- > Creating lines using co-ordinate entry functions.
- > Creating arcs using center/radius functions.
- > Creating arcs using polar positioning.
- > Inserting fillets at the corners.
- > Mirroring existing geometry to complete a part.
- > Translating existing geometry to create a 3-dimensional block.

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

- > Machining 6 holes, spot drilled, drilled and tapped.
- ➤ Machining a 2-dimensional pocket.
- ➤ Machining a 2-dimensional contour.
- > Using lead in and lead out toolpath entry.
- > Modifying the pocket toolpath by creating a rough pocket, a remachining pocket and a finish pocket.
- ➤ Saving to library 9/16-12 tap holes.

#### 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.





# GEOMETRY CREATION Setting the toolbar states

To start a new file from Mastercam:

#### File

#### > New

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-5 in the User Notes.



Toolpaths/Solids manager to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.

- Before starting the geometry make sure that the **Grid** is enabled. It will show you at each moment where the part origin is. See **Getting started** page **A-5** for details.
- Since this drawing is symmetrical in the X-axis and Y-axis, you will only draw 1/4 of the total part, and use the mirror function to complete the part.

#### STEP 1:

#### CREATE 1/4 OF THE OUTSIDE GEOMETRY.

1.1 Create a rectangle knowing the width and the height of the rectangle.

#### Create

- > Create Rectangle
- > Enter the Width 5.0 (Tab).
- > Enter the **Height** 3.0 (Enter).
- > [Select position of first corner]: Select the Origin (center of the grid) as shown.
- > Select the OK button.

  > Use the Fit icon to fit the drawing to the screen.

  Select the center of the grid
- During the geometry creation of this tutorial, if you make a mistake you can undo the last step using 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 INSIDE GEOMETRY.

#### 2.1 Create parallel lines.

#### Create

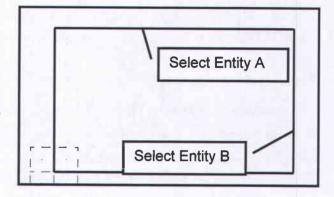
- ▶ Line
- > Create Line Parallel

> [Select a line]: Select Entity A.

[Select the point to place a parallel line through]: Pick a point below the selected line.

Note that the color of the geometry is cyan. This means that the entity is "alive" and you can still change the line parameters. To exit the command you can either start a new

command or select the **OK** button.



> Type the **Distance** 0.5.

> To continue using the same command, select the Apply button.

> To create more parallel lines with the same distance click on the distance icon. The distance

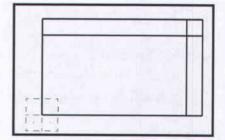
value will be highlighted in red.

> [Select a line]: Select Entity B

➤ [Select the point to place a parallel line through]: Pick a point to the left of the selected line.

> Select the **OK** button.

> The drawing should look as shown in the picture to the right.



# 2.2 Create polar arcs knowing the center location, initial and final angles.

#### Create

> Arc

> Create Arc Polar

➤ Enter the Radius 1.0 (Tab).

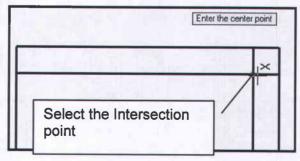
To set the other parameters of the arc use the Tab key. Note that the diameter value is automatically changed by the system based on the radius.

> Enter the Start Angle 180 (Tab).

> Enter the End Angle 270 (Enter).

➤ [Enter the center point]: Select the center point at the intersection as shown in the picture to the right.

> Select the Apply button to continue using the same command.





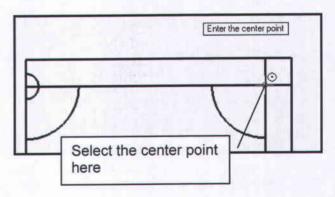
Enter the center point

- ➤ Enter the **Radius** 1.0 (Tab).
- > Enter the Start Angle 270 (Tab).
- > Enter the End Angle 4 0 (Enter).
- ➤ [Enter the center point]: Select the endpoint of the line as shown in the picture to the right.
- > Select the Apply button.
- > Enter the Radius 31/64/2 (Tab).
- > Enter the Start Angle 270 (Tab).
- > Enter the End Angle 4 90 (Enter).
- > [Enter the center point]: Select the same endpoint of the line as shown in the picture above.
- > Select the **OK** button.

#### 2.3 Create an arc knowing the center point and the radius/diameter.

#### Create

- > Arc
- > Create Circle Center Point
- > Enter the **Diameter** value 2 31/64 (Enter).
- ➤ [Enter the center point]: Select the point when the rectangle with the circle appears.
- > Select the **OK** button.



Select the Endpoint of

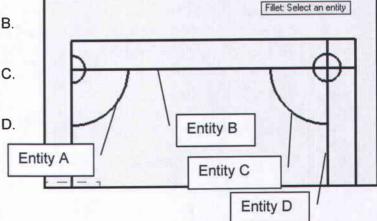
the line

#### 2.4 Create fillets.

#### Create

- **>** Fillet
- > Fillet Entities
- > Enter the fillet Radius 0.0625.
- > [Select an entity]: Select Entity A.
- ➤ [Select another entity]: Select Entity B.
- > [Select an entity]: Select Entity B.
- > [Select another entity]: Select Entity C.
- > [Select an entity]: Select Entity C.
- > [Select another entity]: Select Entity D.
- > Select the **OK** button.





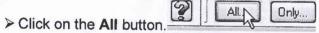


# 2.5 Deleting the construction lines. > Select the lines as shown to the right. > Select the Delete entity icon. Delete these two lines

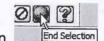
# STEP 3: TRANSFORM THE GEOMETRY TO REPRESENT THE WHOLE PART.

#### **Xform**

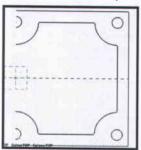
- **>** Mirror
- > [Select entities to mirror]



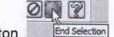
> Select the **OK** button.



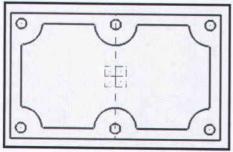
- ➤ Click on the End Selection button.
- ➤ Set the parameters in the dialog box as shown (enable **Copy** to keep the original entities, enable mirror about **X**-axis and **Preview** and **Fit** to see the result on the screen).



- > Select the Apply button to continue Mirror window.
- > [Select entities to mirror]: Click on the All button.
- > Select the **OK** button



- ➤ Click on the End Selection button.
- > Change the Axis to mirror about to Y-axis as shown to the right.







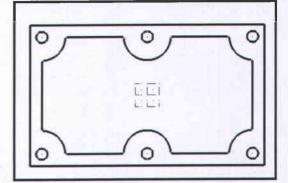
#### Mill X2



**V** > Select the **OK** button to exit.

#### Screen

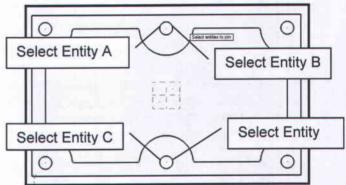
- > Clear Colors
- > The part should look as shown in the picture to the right.

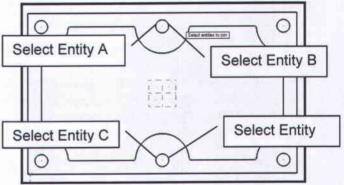


#### STEP 4: JOIN THE TWO ARCS.

#### Edit

- > Join entities
- > [Select entities to join]: Select Entity A.
- > [Select entities to join]: Select Entity B.
- > [Select entities to join]: Select Entity C.
- > [Select entities to join]: Select Entity D.





> Select End Selection button.



#### STEP 5: SAVE THE FILE.

#### File

- > Save as
- ➤ File name: "Your Name 2"
- > Select the **OK** button.



# TOOLPATH CREATION STEP 6:

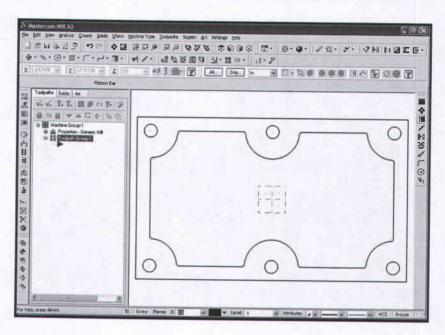
# SET UP THE STOCK TO BE MACHINED.

- > To display the Toolpaths Operation Manager press Alt + O.
- If a machine is already selected in the Toolpaths Manager, skip the next selection.

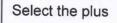
#### Machine type

- > Mill
- ▶ Default



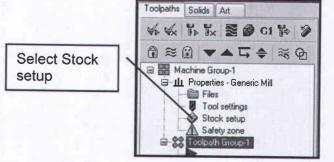


- > Use the Fit icon to fit the drawing to the screen.
- ➤ Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.





> Select Stock setup.





Change the parameters to match the screenshot to the right.

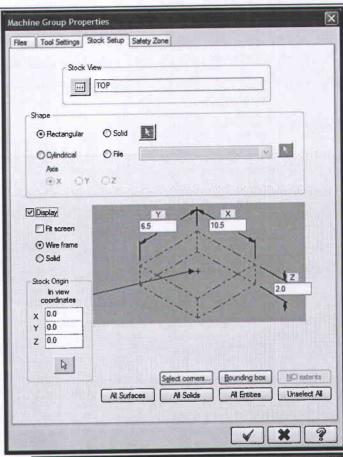
The Stock Origin values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finished part.

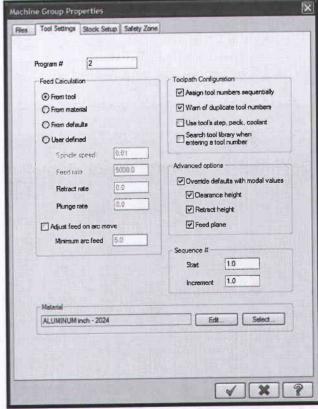
Display options allow you to set the stock as Wireframe and to fit the stock to the screen (Fit Screen).

- Select the Tool Settings tab to set the tool parameters and the part material.
- > Change the parameters to match the screenshot on the right.
- > See Tutorial #1 pages 1-9 for more info.

Select the OK button to exit Toolpath

Group Properties.







> Select the Isometric View from the view toolbar to see the stock.



> Select the **Top View** from the view toolbar to see the part from the top.

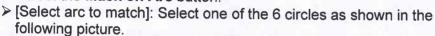


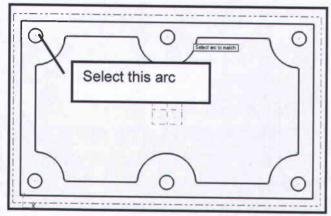
# STEP 7: SPOT DRILL 6 X 5/16" DIAMETER HOLES.

#### **Toolpaths**

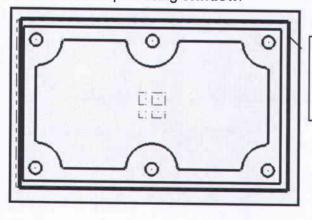
- > Drill Toolpath
- > Select the OK button to enter the same NC name as the geometry file name.







- The Diameter value should be now 0.48438.
- > Select the entire part using Window.

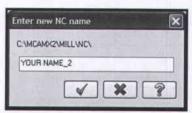


Select a window around the part

> Hit Enter when finish.

> Select the OK button to exit Drill Point Selection.

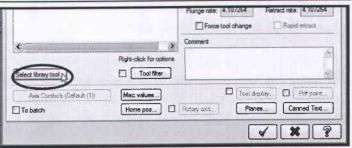






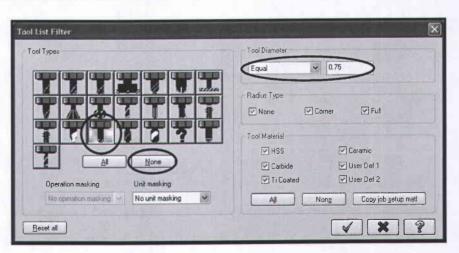


- > Click on the **Select library tool** button.
- > Select the **Filter** button in the **Tool Selection** dialog box.





- > Select the **None** button to disable any previous tool selection as shown.
- > Select the **Spot Drill** in the **Tool Types** list.
- > Select the drop-down arrow in the Tool Diameter field and select Equal.
- > Enter 0.75 in the Tool Diameter value box.
- > Select the **OK** button to exit **Tool List Filter**.

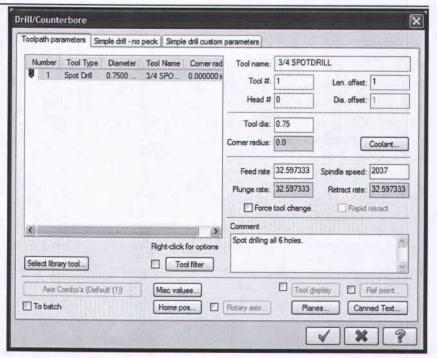


- ➤ Make sure that the tool is selected (highlighted) in the **Tool Selection** window.
- > Select the **OK** button to exit **Tool Selection**.





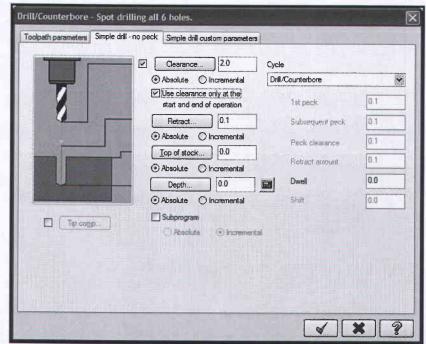
The Feed rate, Plunge rate, Retract rate and Spindle speed are based on the tool definition. Change them as desired.



> In the Comment field type a comment about the toolpath for future reference.

> Select the Simple drill-no peck tab and change the parameters to match the following

screenshot.



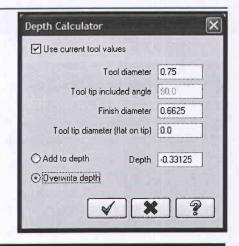
- We want to use the spot drilling to chamfer the holes.
- > To calculate the **Depth**, select the calculator icon.

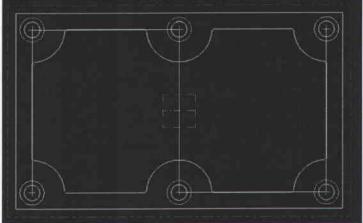


> Change the parameters as shown in the screenshot to the right.

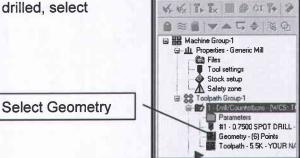
Enter the **Finish diameter** value to set the outside diameter of the chamfer. The system automatically calculates the machining depth for the spot drill operation.

- > Select the **OK** button to exit.
- ➤ The drilling Cycle should be set to the default, Drill/Counterbore (G81).
- > Select the **OK** button to exit drilling parameters.





> To change the order in which the holes are spot drilled, select **Geometry** in the **Toolpaths Manager**.



oolpaths Solids Art

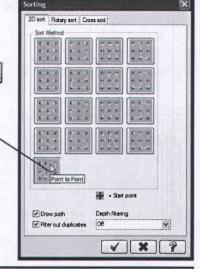
➤ Right-mouse click inside of the **Drill Point Manager** and select **Sort options...** 



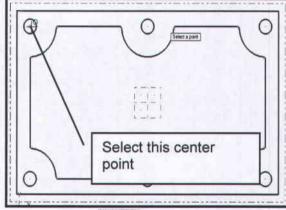


- > Select the Point to Point sort method
- ➤ Select the **OK** button to exit the **Point Sorting** dialog box.

Select Point to Point



> [Select sorting start point]: Select the center point of the first hole to drill.



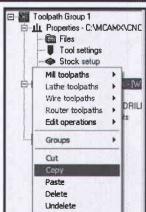
- > Select the OK button to exit the Drill Point Manager dialog box.
- Note the red X in front of the **Toolpath** description in the Drill operation. To update the toolpath with the new parameters you have to regenerate it.
- > Select the Regenerate all dirty operations button.

Select the Regenerate button



# STEP 8: DRILL THE 6 X 31/64" DIAMETER HOLES.

- > Toolpaths Manager
- > Right-Mouse Click and select Copy.
- > Right-Mouse Click again and select Paste.





@ ≈ Ø ▼ ▲ □ ♦ ≈ €

Toolpath - 5.5K - YOUR NA

Toolpath - 5.5K - YOUR NA

- Drill/Counterbore - [WCS, TI

#1 - 0.7500 SPOT DRILL

×

Toolpaths | Solids | Art

Machine Group-1

☐ 88 Toolpath Group-1

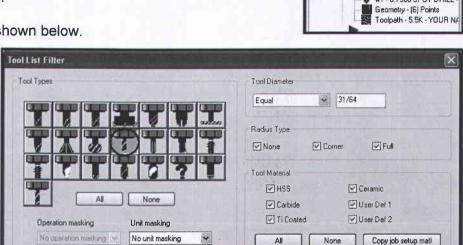
> The **Toolpaths Manager** will look as shown in the screenshot to the right.

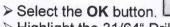
Reset all

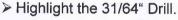
> Select the Parameters in the second drill operations.

Select Parameters

- > Select the Toolpath parameters tab.
- > Click on Select library tool.
- > Select the Filter button.
- > Make the changes as shown below.





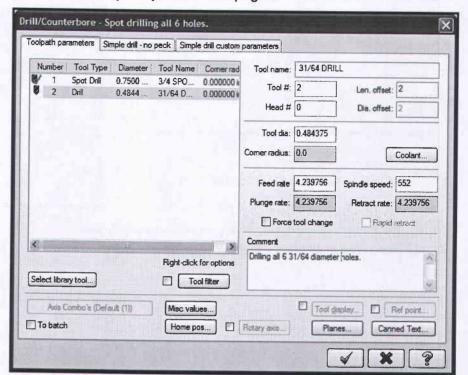




> Select the **OK** button.



> Make any necessary changes in the Toolpath parameters page.

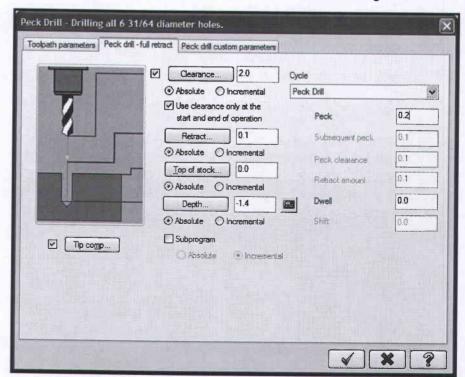


The Feed rate and Spindle speed are based on the tool definition. Change them as desired.

> In the Comment field type a comment about the toolpath for future reference.

> Select the Simple drill-no peck tab and change options as shown in the following

screenshot.





Tool diameter

Breakthrough

amount

Tip length

0.484375

0.145521

118.0

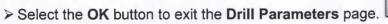
0.0

Drill tip compensation

> Enable the **Tip comp...** button and select it.

Tip comp... enabled allows you to cut deeper than the final depth with the tip of the drill. The system automatically calculates the tip length based on the drill diameter and angle.

- > Change the parameters as shown.
- > Select the **OK** button to exit the **Drill tip** compensation page.



> Select the Regenerate all dirty operations button.



Select the Regenerate button

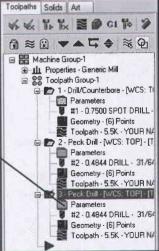
# STEP 9:

TAPPING THE 6 X 9/16-12 UNC DIAMETER TAP HOLES.

- > In the Toolpaths Manager, select only the Peck Drill operation.
- > Right-Mouse Click and select Copy.
- > Right-Mouse Click again and select Paste.
- > The **Toolpaths Manager** will look as shown in the screenshot to the right.
- > Select the **Parameters** in the second drill operations.

Select Parameters



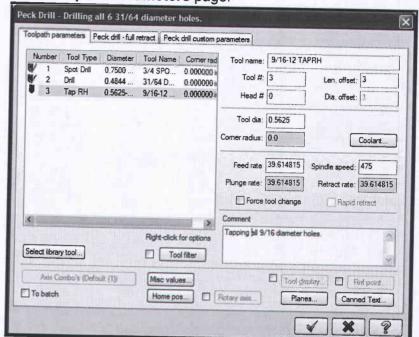




> Select the Toolpath parameters.

➢ Click on Select library tool and using the Filter, select the 9/16-12 Right Hand Tap from the library.

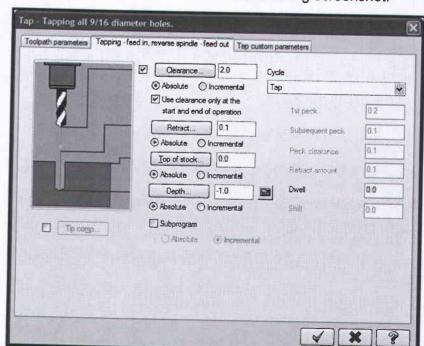
➤ Make any necessary changes in the Toolpath Parameters page.



Note that the feed rate is calculated based on the spindle speed and the number of threads/inch.

> In the Comment field type a comment about the toolpath for future reference.

> Select the second page and enter the **Depth** value as shown in the following screenshot.



- ➤ Disable Tip comp...
- Change the drill Cycle to Tap.
- > Select the **OK** button to exit the **Drill Parameters** page.

> Select the Regenerate all dirty operations button.

Select the Regenerate button



> If necessary, select the down arrow to move the arrow one step down to include the tapping operation.

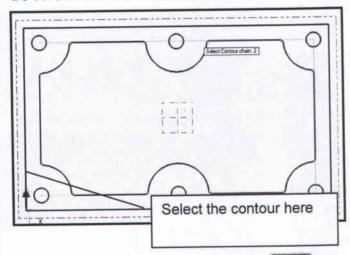
> Move insert arrow down one item

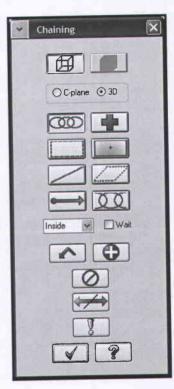


## **STEP 10:** CONTOUR THE OUTSIDE PROFILE.

#### **Toolpaths**

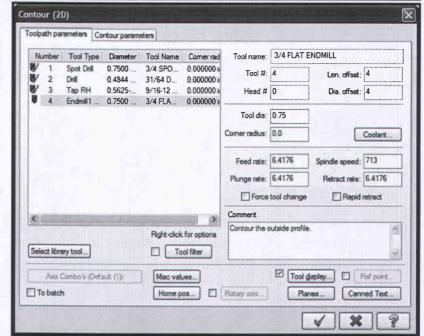
- > Contour Toolpath.
- > Select the first entity in the contour, as shown.
- > Be sure to chain the contour in a CW direction.





- > Select the **OK** button to exit **Chaining**.
- > Select the Toolpath parameters tab.
- > Click on the Select library tool button.
- ➤ Use the Filter button to select the ¾" Flat Endmill from the library.

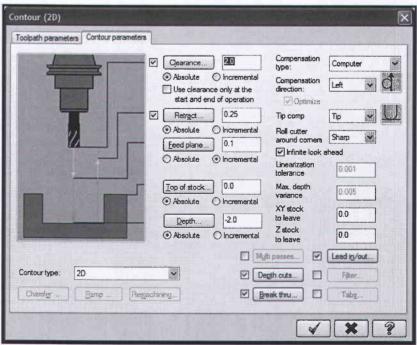




The Feed rate, Plunge rate, Retract rate and Spindle speed are based on the tool definition.
Change them as desired.

> In the Comment field type a comment about the toolpath for future reference.

> Make all the necessary changes as shown above and select the Contour parameter page.



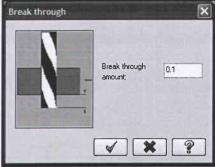
- > Enable **Depth** cuts and select the button to set the cuts along the Z-axis.
- > Change the Max rough step value as shown.
- ➤ Select the **OK** button to exit.





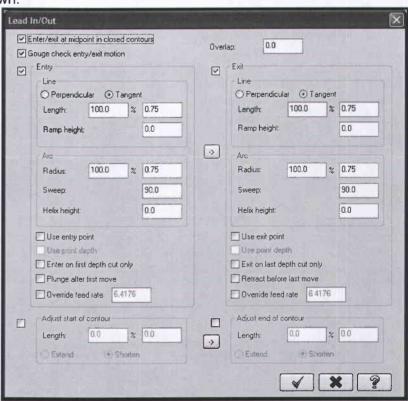
▶ If you want to cut through without leaving any burr enable Break through and select the button.

> Enter the Break through amount as shown.



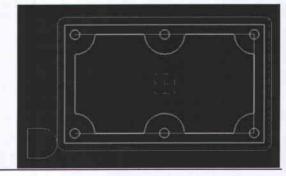
➤ Enable the box in front of **Lead in/out** and select a combination of a **Line** and an **Arc** at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting the part.

> Change the parameters as shown.



Select the OK button to exit the Lead In/Out dialog box.

> Select the **OK** button again to exit **Contour** parameters.



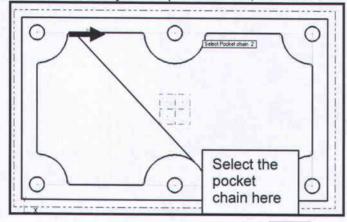


# STEP 11: POCKET THE INSIDE CONTOUR.

#### **Toolpaths**

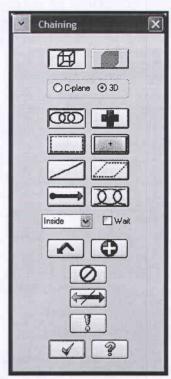
> Pocket Toolpath.

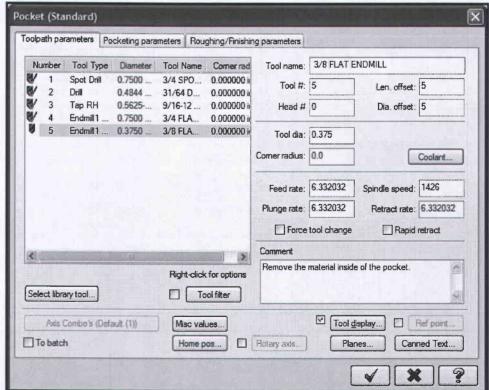
> Select one entity of the pocket chain, as shown.



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

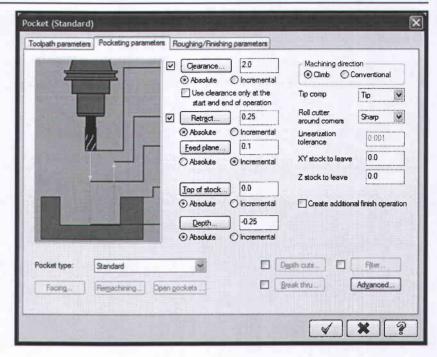
> Select the 0.375" Flat End Mill following the steps outlined before.





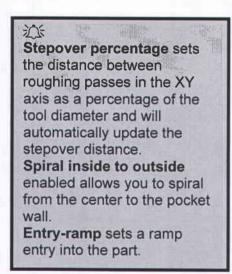


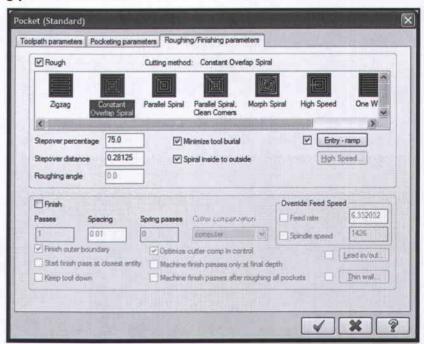
Make all the necessary changes as shown above and then select the Pocket parameter page.



➤ Enter the **Depth** value to set the final machining depth for the pocket operation (-0.25).

- > Select the Roughing/Finishing parameters tab.
- Select the Constant Overlap Spiral as the Cutting method.





- > Disable the Finish area.
- > Select the **OK** button to exit **Pocket parameters**.





V. V. V. Xx @ 0 G1 Yo 2

// W 🛎 🔒

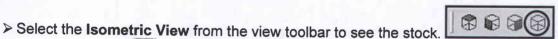
Toolpaths | Solids

# **STEP 12:** BACKPLOT THE TOOLPATH.

- > Click on the Toolpath Group-1 in the Toolpaths Manager to select all operations.
- > Select the Select All Operation button.
- > Select the Backplot selected operations button.
- > Make sure that you have the following buttons turned on (they will appear pushed down). ➤ Display tool X Backplot Display tool

Display rapid moves

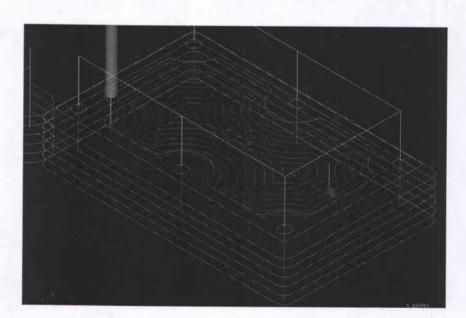
> Display rapid moves



- ➤ Select Fit button.
- > Select the Play button.



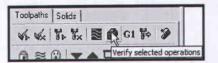
> Select the OK button to exit Backplot.





## VERIFY- TOOLPATH VERIFICATION STEP 13: VERIFY.

- > Expand the **Toolpaths Manager** if necessary by dragging the right side.
- > Select the Verify selected operations button.



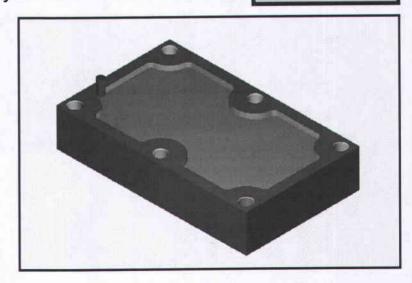
M M M M

Verify

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



- > Select the **Machine** button to start simulation.
- > The finished part should appear as shown in the following picture.
- ➤ Select the **OK** button to exit **Verify**.





#### **STEP 14:**

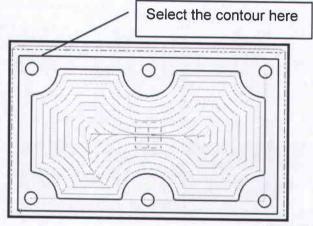
#### ASSOCIATIVITY.

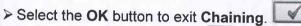
# 14.1 Modify the existing geometry. Chamfer the corners of the outside profile.

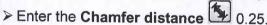
> Select the **Top View** from the view toolbar to see the part from the top. Create

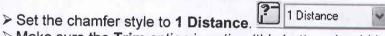


- ➤ Chamfer
- > Chamfer Chains
- > Select the outside contour.

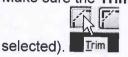




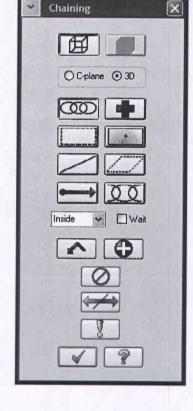




> Make sure the Trim option is active (this button should be



> Select the **OK** button.



# 14.2 Update the toolpath.

- Notice the red X in the contour toolpath. This indicates that the toolpath was affected and needs to be regenerated.
- > To regenerate the Contour toolpath select the Regenerate all dirty operations icon as shown.

Toolpaths | Solids | Regenerate all dirty operation

Select the Regenerate button



# 14.3 Modify the existing pocket toolpath. Rough out the pocket.

Note that the 3/8" Flat End Mill could not clean the .0625" radius fillets.

To improve the toolpath efficiency we will use a 1" Flat End Mill to rough out the part. Using a 1/8" Flat End Mill and the remachining pocket type we will remove the material in the areas where the 1" tool could not fit.

> In the Pocket Operation select the Parameters.

Select Parameters

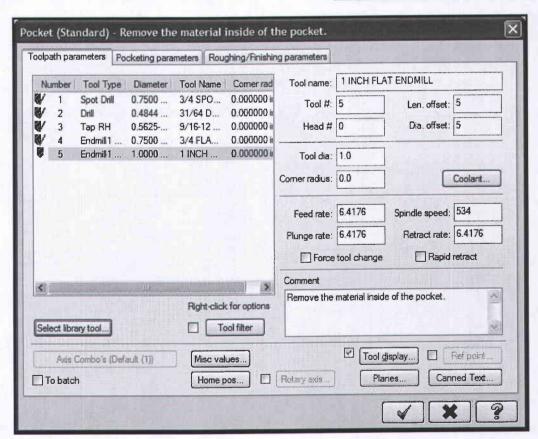
Select Parameters

Geometry · (1) chain(s)
Toolpath - 34.8K · YOU
Farameters

#5 · Pocket (Standard) · [WC
#5 · 20.750 ENDMILL1
Geometry · (1) chain(s)
Toolpath - 29.2K · YOU

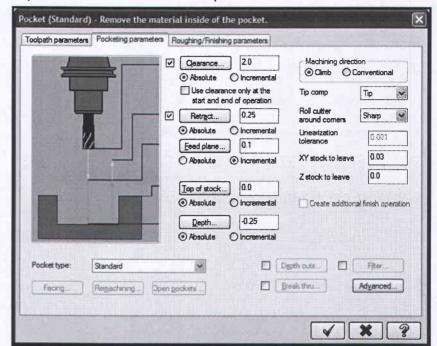
- > Select the Toolpath parameters tab.
- ➤ Click on 3/8" Flat End Mill, and select Delete from the keyboard.
- > Select the Yes button.
- > Click on Select library tool.
- > Use Filter to select the 1.0" Flat End Mill.







> Select the **Pocketing parameters** to enter the **XY stock to leave** amount, to instruct the system to leave stock along the pocket wall for future finish operations.



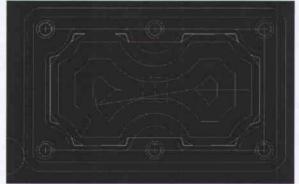
Select the OK button to exit the Pocket parameters.



> Select the Regenerate all dirty operations icon as shown to update the toolpath.

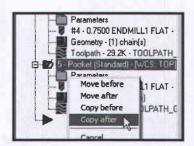
Select the Regenerate button



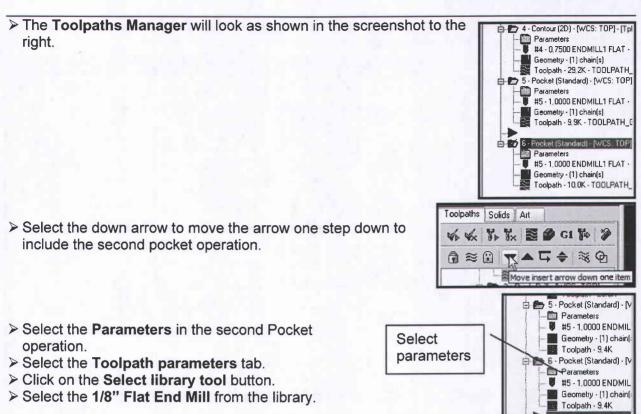


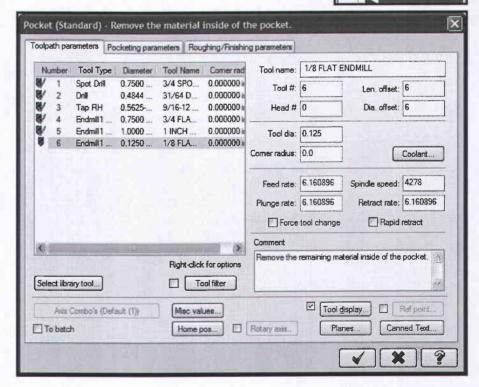
# 14.4 Remachine the pocket corners.

- ➤ In the **Toolpaths Manager**, select only the **Pocket** operation.
- > Right-mouse click on the Pocket, hold it down and drag, and select Copy after.





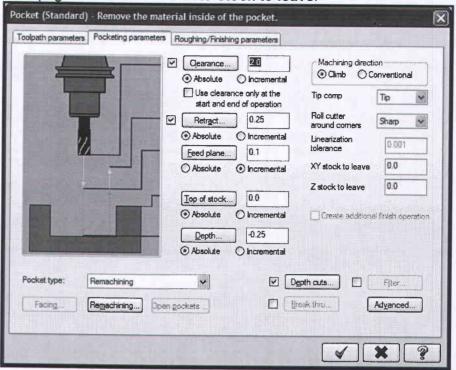




● The Feed rate, Plunge rate, Retract rate and Spindle speed are based on the tool definition. Change them as desired.



> Select the Pocket parameter page and enter 0 in the XY stock to leave.



➤ Change the **Pocket type** to **Remachining**, select the **Remachining** button and change the parameters as shown.

Compute remaining stock from: The previous operation enables the system to calculate the remaining stock for remachining by determining the stock left after the previous operation.

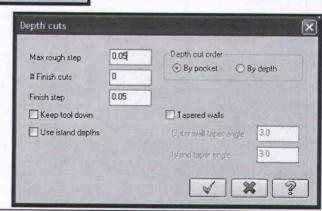
Clearance, set as a percentage of the tool diameter, allows you to expand the remachining area at the beginning and at the end to prevent a remaining cusp of material.

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.

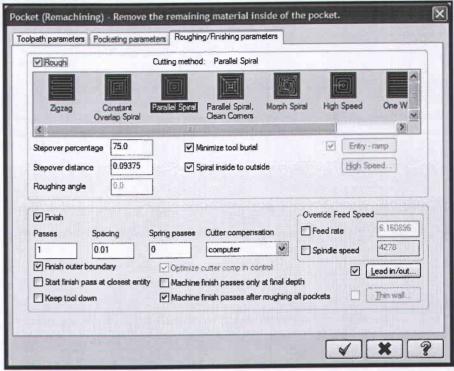
- > Select the **OK** button to exit the **Pocket** remachining dialog box.
- ➤ Enable the **Depth cuts** button and enter 0.05 in the **Max rough step**.
- > Select the **OK** button to exit.





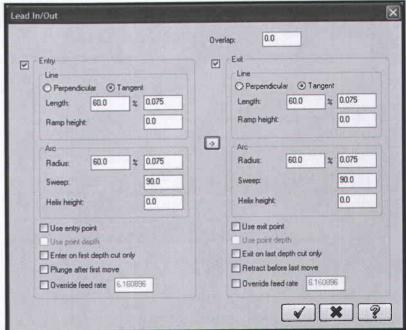


- > Select the Roughing/Finishing parameters.
- > Enable the Finish area and enable the Lead in/out.



> Select Lead In/Out and set the line and the arc at the beginning and/or end of the cuts.

> Select the **OK** button to exit **Lead In/Out**.



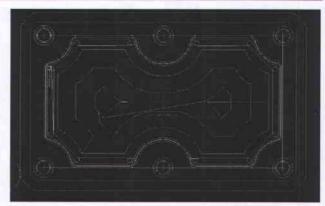
> Select the **OK** button to exit

Pocket parameters. 

➤ Select the Regenerate all dirty operations icon as shown to update the toolpath.

Select the Regenerate button



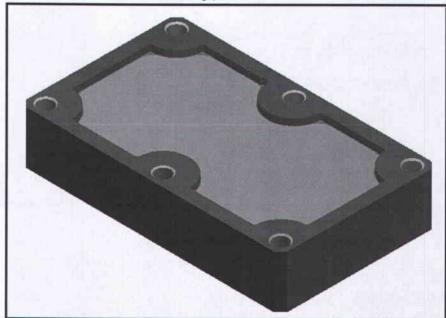


#### **STEP 15:**

#### **VERIFY THE CHANGED PART.**

- > Select the Select all operations button from Toolpath Manager.
- > Repeat STEP 13 to be sure that the changes have taken place.
- > The finished part should appear as shown in the following picture.





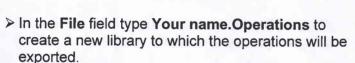
# STEP 16: EXPORT TO LIBRARY ALL DRILLING OPERATIONS.

- We are going to create an operations library by exporting all drilling operations from the current MCX file into a library (Ops folder). We are going to Import the operations saved in the library each time we need to machine 9/16 -12 tap holes.
- > Select only the **Spot drilling** operation.
- > Hold down the Shift key and select the Tapping toolpath.





- > Right click in the Toolpaths Manager window (below the operations).
- ➤ Select Export...

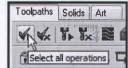


- > In the Group field enter the name of the operations group that will be saved in the new library.
- > Disable Export operation's geometry to export parameters only and no geometry.
- > Select the Export button.

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

# **STEP 17: POST THE FILE**

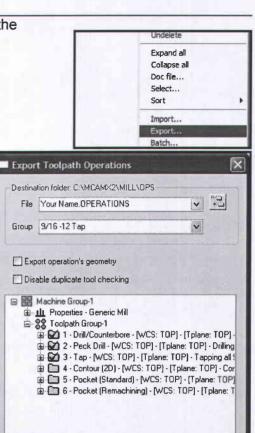
> Select the Select all operations button from Toolpath Manager.



Export toolpath operations X

3 operations exported successfully. OK

> Select Post selected operations.







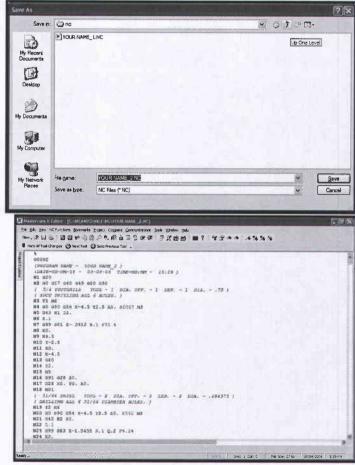
### Mill X<sup>2</sup>



- > In the **Post processing** window, make all the necessary changes as shown to the right.
- ➤ Enable NC file to keep the NC file assigning the same name as the MCX file.
- > Enable Edit to automatically launch the default editor.
- > Select the **OK** button to continue.



> Select the Save button.

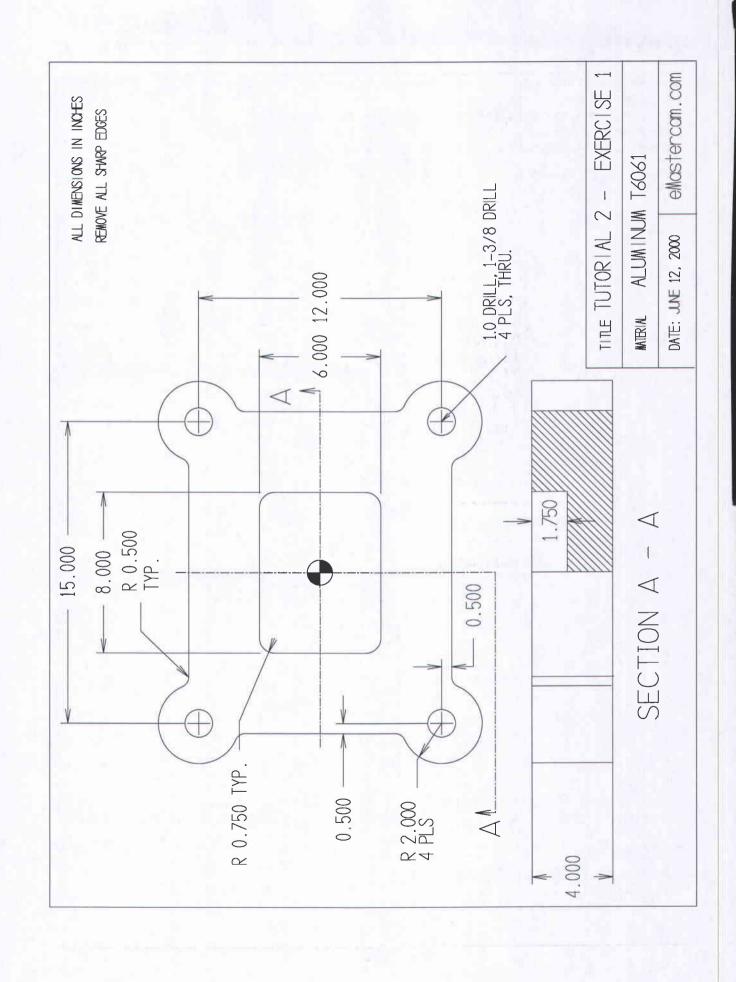


➤ Select the red X to exit Mastercam Editor

# STEP 18: SAVE THE UPDATED MCX FILE.

> Select the Save icon.

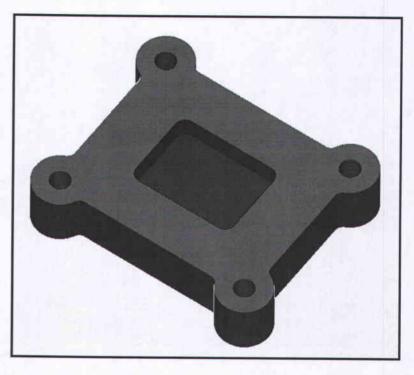






## **REVIEW EXERCISES.**

Student practise. Create the Toolpath for Exercise 1-Tutorial 2 as per the instructions below;



Tips:

**Stock** size; X = 19.5, Y = 16.5 and Z = 4

Spot Drill the holes using a 0.75" Spot drill

**Drill** through holes with a 1.0" drill (right-mouse click on the tool in the Toolpath parameters and **Edit Tool**: Flute = 4.125; Shoulder =4.25; Overall = 4.5)

Cycle Peck Drilling

Drill through holes using a 1-3/8" drill

Change the current library to Big.tools in the Tool selection window.

Edit the tool as in the previous step.

Cycle: peck Drilling

Contour the part using 1 " Flat End Mill; Edit the tool height to 4.5"

Max rough step =0.5" (Depth cuts)

2 roughing passes with spacing = 0.75"

1 finish pass 0.05" (Muti passes)

Pocket the part using the same 1" Flat End Mill

Max rough step = 0.5 " (Depth cuts)

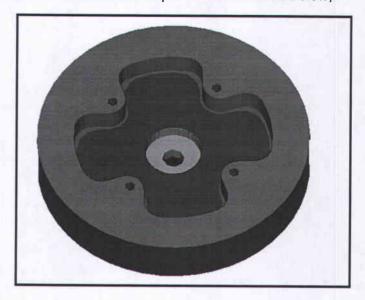
Cutting method = Parallel Spiral

Backplot and Verify the toolpaths.

Post process the file.



Student practise. Create the Toolpath for Exercise 2-Tutorial 2 as per the instructions below;



Tips:

Stock size use Bounding box/All/Entities and give

Z = 1"

Drill the 0.2 diameter holes using 0.2010 Drill

Cycle: Peck drilling

Contour the part using 1 " Flat End Mill;

Max rough step =0.5" (Depth cuts)

1 roughing passes with spacing = .75"

1 finish pass 0.05" (Muti passes)

Pocket (rough) the part using the same 1" Flat End Mill

Stock to leave XY = 0.05

Pocket (remachining) the part using the 3/8" Flat End Mill

Pocket (finish) the part using the 3/8" Flat End Mill

Drill the 3/4 diameter thru holes using 3/4 Drill

Cycle: Drill/conterbore

Top of stock = -0.500( abs)

Feed plane = 0.100 (incr)

Depth =-0.75

Circle milling the 1 1/2 diameter holes using 3/4 Flat End Mill

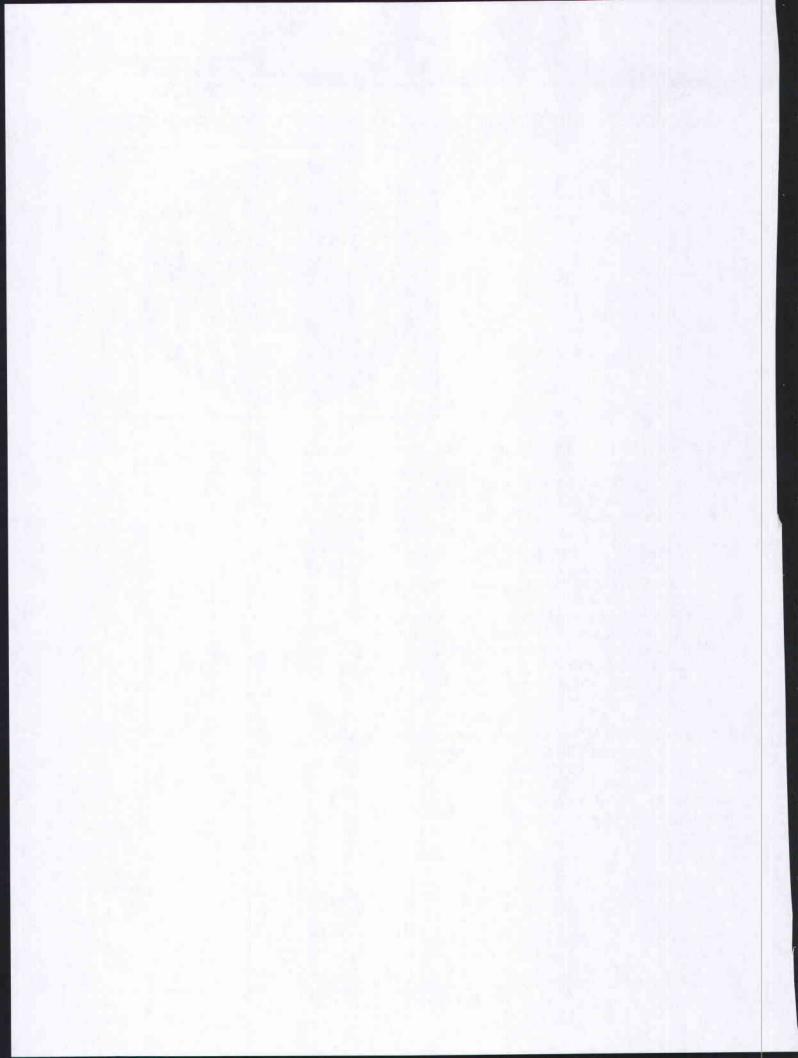
See Tutorial #5 Step 11

Top of stock = -0.500( abs)

Feed plane = 0.100 (incr)

Backplot and Verify the toolpaths.

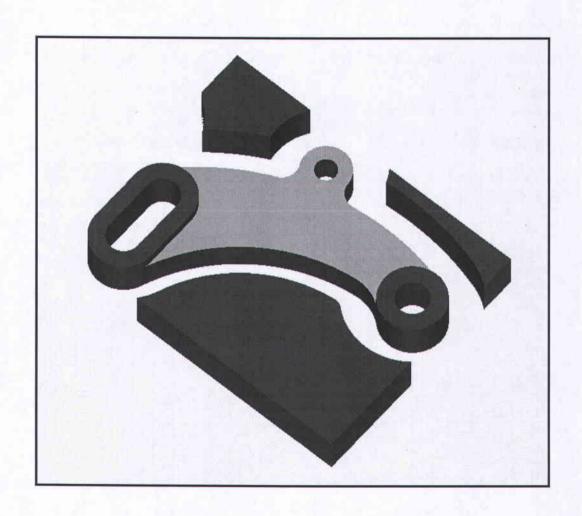
Post process the file.



# **TUTORIAL SERIES FOR**

# Mastercam.X2

# TUTORIAL 3 LEVEL 1 - 2D GEOMETRY, ISLAND POCKETING, IMPORT FROM THE LIBRARY.





## **Objectives:**

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

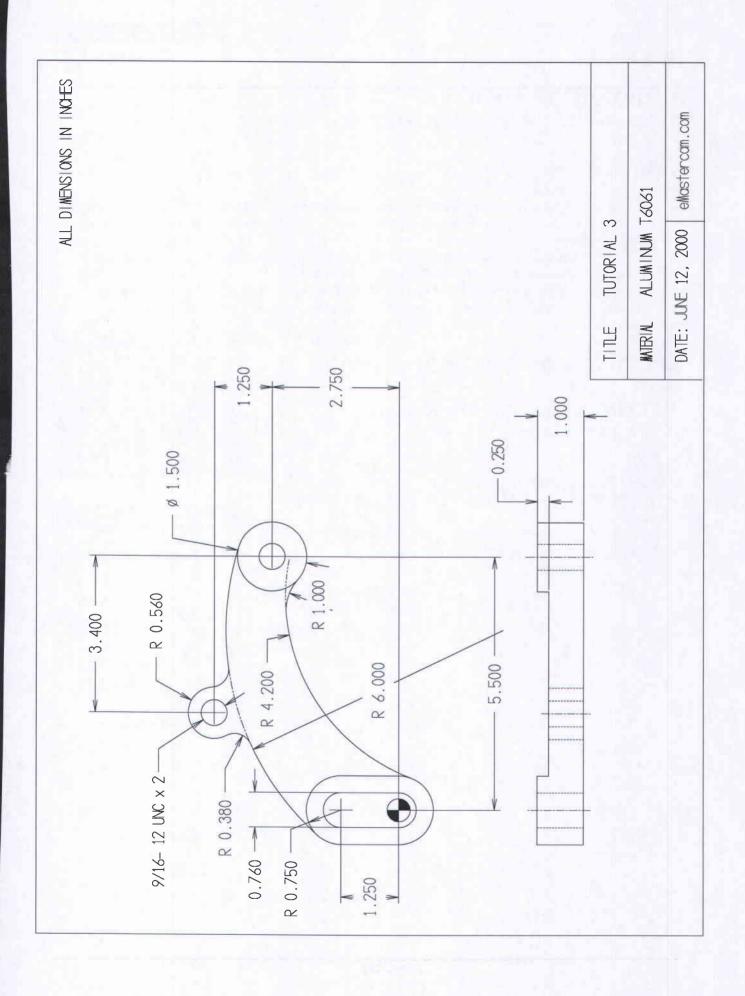
- > Creating arcs using polar positioning.
- > Creating lines using co-ordinate entry.
- > Creating arcs tangent to 2 existing entities.
- > Trimming 1 entity to another existing entity.
- > Creating fillet radii.
- > Creating parallel lines, by defining the offset direction and distance.
- > Breaking existing geometry into 2 sections.
- > Translating existing geometry to create a 3-dimensional block.

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

- > 3 holes drilled through the part.
- > A 2-dimensional pocket.
- > A 2-dimensional contour.
- > Lead in and lead out toolpath entry.

#### 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.





# GEOMETRY CREATION To start a new file from Mastercam:

#### File

> New

Setting the toolbar states

- 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-5 in the **User Notes**.
- Toolpaths/Solids manager to the left of the screen can be hidden to gain more space in the graphic area for design. Press Alt + O to remove it.
- Before starting the geometry make sure that the **Grid** is enabled. It will show you at each moment where the part origin is. See **Getting started** page A-5 for details.

## STEP 1: CREATE THE OUTSIDE PROFILE OF THE SLOT.

1.1 Create an obround shape using rectangular shapes.

#### Create

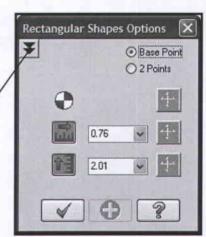
> Create Rectangular Shapes

> Type the Width and the Height as shown in the following picture.

> Select the **double arrow** to expand the **Rectangle Options** as shown in the screenshot below if needed.

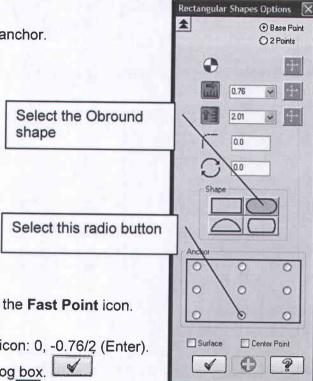
Select the Double arrow



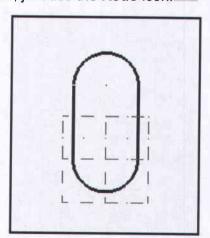




- > Select the Obround Shape.
- > Select the middle bottom radio button as the anchor.



- > [Select the position for the base point]: Select the **Fast Point** icon.
- > Enter the values into the box to the left of the icon: 0, -0.76/2 (Enter).
- > 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.

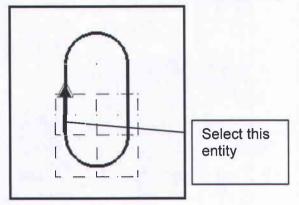


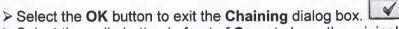


## 1.2 Create the outside obround shape using offset contour.

#### Xform

- > Xform Offset Contour.
- > Select the chain as shown in the following screenshot.



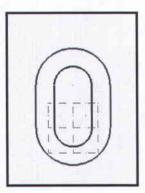




> Enter the **Distance** in the dialog box: 0.37.

- > Enable the Preview and Fit to be able to see the result in the graphic
- Note that the resulting offset contour should be outside of the original one.
- > Select the direction arrow button if you need to change the side of

the offset contour.



> Select the **OK** button to exit.



#### Screen

> Clear color





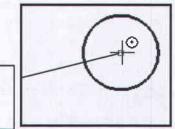


#### STEP 2:

## CREATE THE 9/16" AND 1.5" DIAMETER ARCS KNOWING THE CENTER POINT AND THE DIAMETER.

#### Create

- > Arc
- > Create Circle Center Point
- > Enter the **Diameter** value 9/16 (Enter).
- > [Enter the center point]: Select the Fast Point icon.
- Enter the values into the box to the left of the icon : 5.5, 2.75 (Enter).
- > Use the Fit icon to fit the drawing to the screen.
- > Select the Apply button.
- > Enter the **Diameter** value 1.5 (Enter).
- > [Enter the center point]: Select the center point of the created circle.



- ➤ Select the **OK** button.
- > Use the Fit icon to fit the drawing to the screen.



Select the

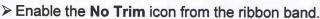
center point

## STEP 3:

## CREATE THE TANGENT ARCS WITH 6" AND 4.2" RADII USING THE FILLET COMMAND.

#### Create

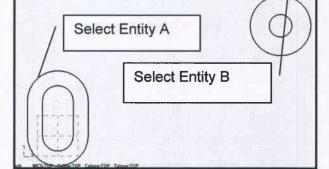
- **>** Fillet
- > Fillet Entities
- > Enter the fillet Radius 6.0







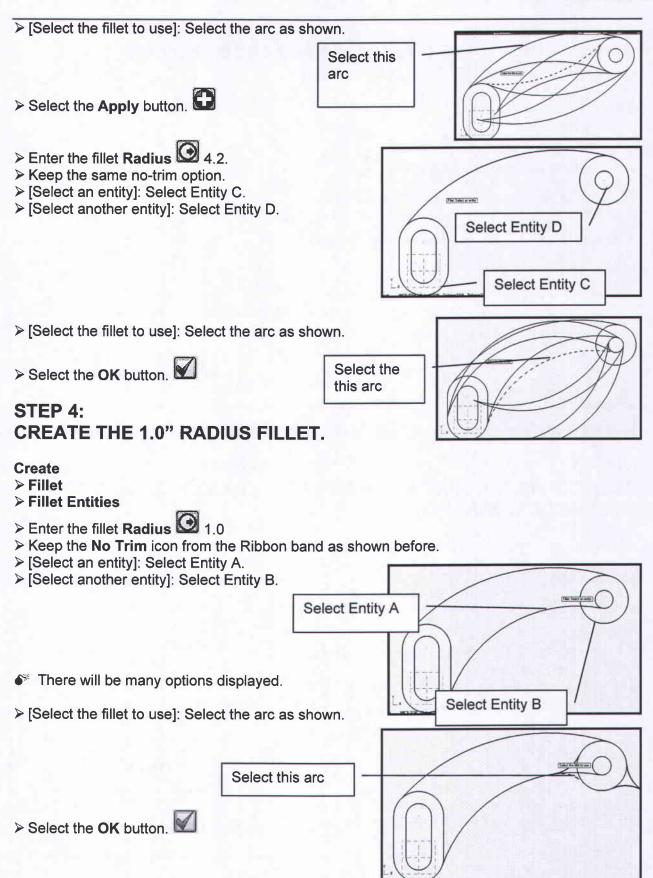
> [Select another entity]: Select Entity B.



There will be many options displayed.

### Mill X<sup>2</sup>







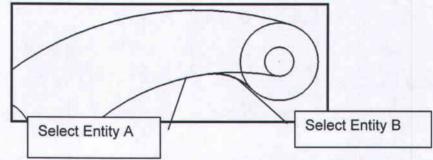
## STEP 5: TRIM ONE ENTITY.

#### Edit

- > Trim/Break
- > Trim/Break/Extend



- > Select the Trim 1 entity button. Trim 1 entity
- > [Select the entity to trim/extend]: Select Entity A.
- > [Select the entity to trim/extend to]: Select Entity B.
- ➤ Select the **OK** button.

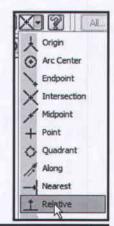


## STEP 6:

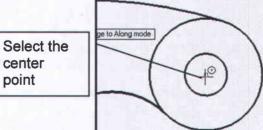
## CREATE THE 0.56" RADIUS POLAR ARC KNOWING THE CENTER LOCATION, INITIAL AND FINAL ANGLES.

#### Create

- > Arc
- > Create Arc Polar
- The center location of the arc is measured relative to the center of the 1.5 diameter circle.
- ➤ [Enter the center point]: Click on the drop-down arrow as shown and select **Relative**.



> [Enter a known point...]: Select the center point of the 1.5 diameter circle as shown in the picture below.





> Enter the Radius 0.56.

To set the other parameters of the arc use the Tab key. Note that the diameter value is automatically changed by the system based on the radius.

> Enter the Start Angle 0 (Tab).

> Enter the End Angle 4 180 (Enter).

> Use the Fit icon to fit the drawing to the screen.

> Select the **OK** button.

## STEP 7: CREATE THE 0.38" RADIUS FILLET.

#### Create

**>** Fillet

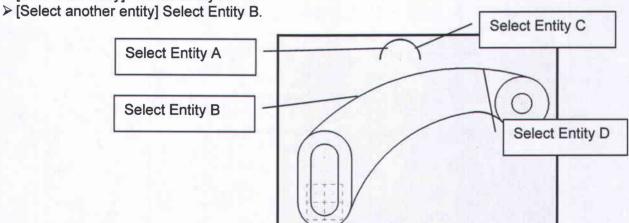
> Fillet Entities

> Enter the fillet Radius 0.38.

> Enable the No Trim icon from the ribbon band (should be already enabled).



➤ [Select an entity]: Select Entity A.



> [Select an entity]: Select Entity C.

➤ [Select another entity]: Select Entity D.

Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities. You can select the desired fillet.

> Select the **OK** button.



#### STEP 8:

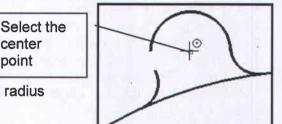
## CREATE THE 9/16" DIAMETER CIRCLE KNOWING THE CENTER POINT AND THE RADIUS/DIAMETER.

center

point

#### Create

- > Arc
- > Create Circle Center Point
- ➤ Enter the **Diameter** value 9/16 (Enter).
- > [Enter the center point]: Select the center of the 0.56 radius
- > Select the OK button.



#### STEP 9:

## TRIMMING THE ENTITIES.

### 9.1 Trim two entities.

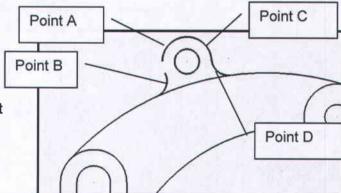
#### Edit

- > Trim/Break
- > Trim/Break/Extend

> Select the Trim 2 Entities button.



- > [Select entity to trim/extend]: Select Point
- > [Select the entity to trim/extend to]: Select Point B.
- > [Select the entity to trim/extend]: Select Point C.
- > [Select the entity to trim/extend to]: Select Point D.

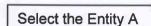


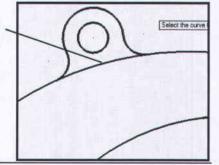
## 9.2 Remove the arc that lies between the two fillet arcs using divide.

Select the Divide icon from the trim ribbon bar.



- > [Select the curve to divide]: Select Entity A.
- ➤ Select the **OK** button.

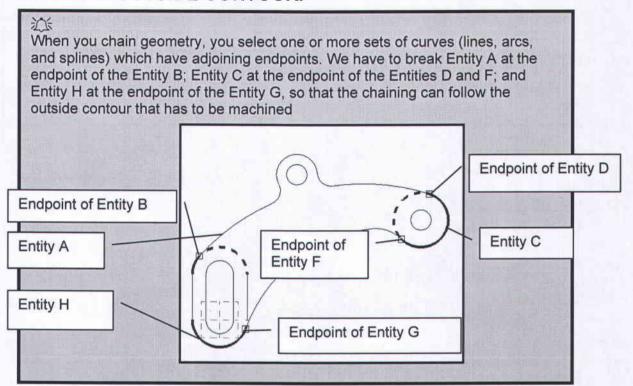






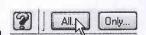
#### **STEP 10:**

## BREAK THE EXISTING ARCS AT INTERSECTION TO BE ABLE TO CHAIN THE OUTSIDE CONTOUR.



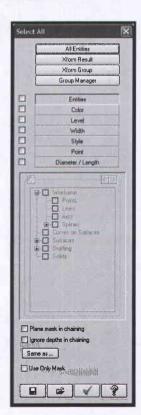
#### **Edit**

- > Trim/Break
- > Break at Intersection
- > [Select the entities to break]: Select the All button.



➤ Select the **OK** button inside the **Select All** dialog box.





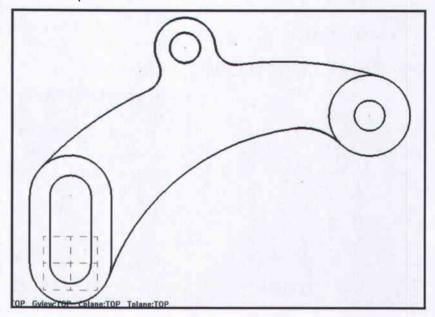


> Select the End Selection icon.



> Select the **Repaint** icon.

The geometry should look as shown in the picture below.



## STEP 11: SAVE THE FILE.

File

> Save as

> File name: "Your Name\_3"

➤ Select the **OK** button.



# TOOLPATH CREATION STEP 12: SET THE MACHINE TYPE.

#### Machine type

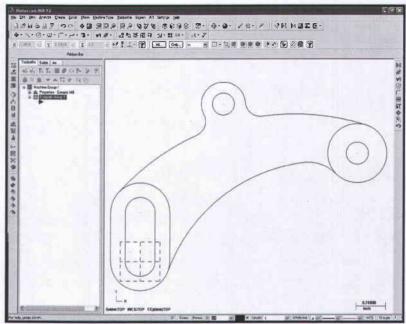
- **> Mill**
- > Select the Default

Machine Type Toolpaths Screen Art

Mill Default

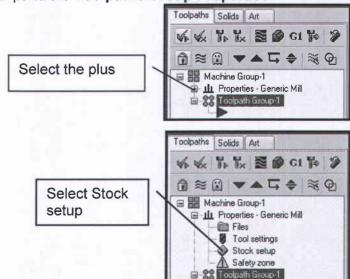
Lathe 1 C:\MCAMX2\CNC\_MAR

> To display the Toolpaths Manager press Alt + O.



## STEP 13: SET UP THE STOCK TO BE MACHINED.

> Select the plus in front of Properties to expand the Toolpaths Group Properties.

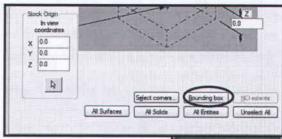


> Select Stock setup.

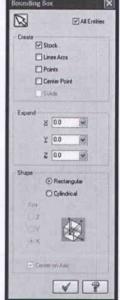


> Select the **Bounding box** button for the system to automatically find the part's overall

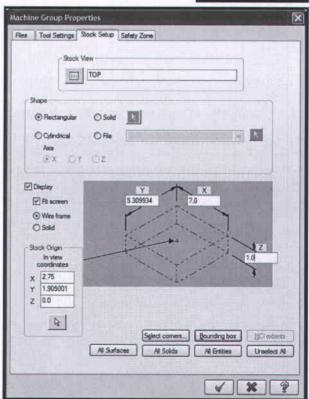
dimensions.



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

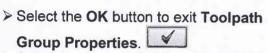


➤ Enter the Z value 1.0".





- > Select the Tool Settings tab and change the parameters to match the screenshot to the left.
- Tutorial #1.



> Select the Isometric View from the view 6 6 6 toolbar to see the stock

Machine Group Properties Files Tool Settings Stock Setup Safety Zone Program # Feed Calculation Toolpath Configuration Assign tool numbers sequentially (a) From tool O From material Warn of duplicate tool numbers O From defaults Use tool's step, peck, coolant Search tool library when entering a tool number O User defined Spindle speed Advanced options Feed rate Ovenide defaults with modal values Retract rate ☑ Clearance height Retract height Adjust feed on arc move Feed plane Minimum arc feed 5.0 Sequence # 1.0 Start Increment 1.0 ALUMINUM inch - 2024 Edit. Select. ×

> Select the **Top View** from the view toolbar to see the part from the top.



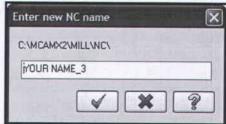
> Use the Fit icon to fit the drawing to the screen.



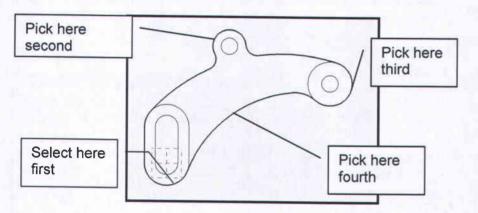
## STEP 14: CONTOUR THE OUTSIDE PROFILE.

#### **Toolpaths**

- **➢ Contour Toolpath**
- > Select the **OK** button to accept the NC file name.

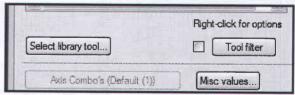


- > Select the first entity in the contour, as shown.
- Be sure to chain the contour in a CW direction.

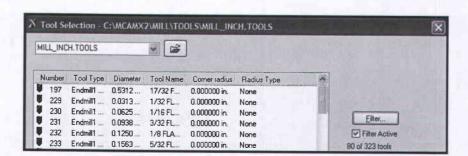




- > Select the **OK** button to exit **Chaining**.
- > Select the 0.75" Flat End Mill following the steps outlined below.
- > Click on the Select library tool button.



> Select the Filter button in the Tool Selection window.





₩ Full

☑ Ceramic

User Def 1

User Def 2

Copy job selup mall

30

w 0.75

**☑** Corner

Tool Diamete

Radius Type

Tool Material

No unit masking

No operation masking w

₩ HSS

☑ Carbide

☑ Ti Coated

AJI

Equal

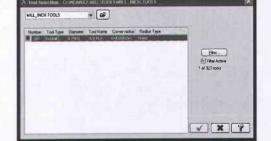
- Select the None button to disable any previous tool selection as shown.
- Select the Flat Endmill button in the Tool Types list.
- Select the drop-down arrow in the Tool
   Diameter field and select
   Equal.
- > Enter 0.75 in the **Tool Diameter** value box.
- > Select the **OK** button to exit **Tool List**
- ➤ Make sure that the tool is selected (highlighted) in the **Tool Selection** window.

Tool List Filter

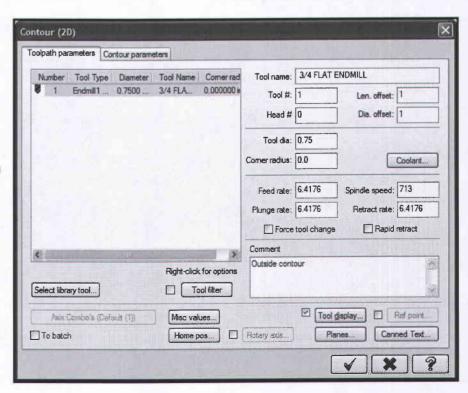
Tool Types

Reset all

> Select the OK button to exit Tool Selection.



- The Feed rate, Plunge rate, Retract rate and Spindle speed are based on the tool definition. Change them as desired.
- ➤ In the **Comment** field type a comment for future reference.

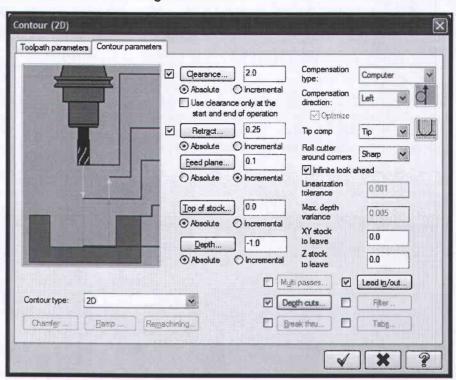




> Change the parameters as shown in the following screenshot.

For more info on the parameters see Tutorial # 1.

You can increase the Depth value if you want to cut through without leaving any burr.



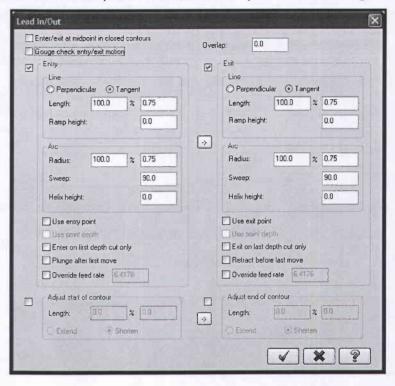
- > Enable **Depth cuts** and select the button to set the cuts along the Z-axis.
- > Select the **OK** button to exit.





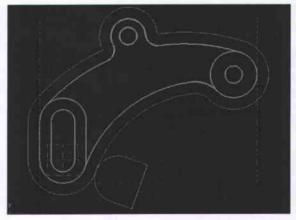
> Enable the box in front of the Lead in/out button and select a combination of a Line and an Arc at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting

the part.



Select the **OK** button to exit the **Lead In/Out** dialog box.

> Select the **OK** button again to exit **Contour parameters**.





## STEP 15: CHANGING THE STARTING POINT.

> Select the Geometry.

Select the Geometry

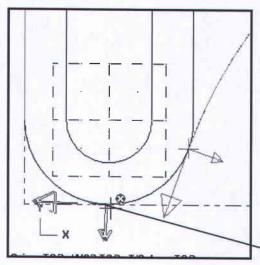
> Right-mouse click on Chain1, and select the Start Point.



Select the Dynamic button

> Select the **Dynamic** button.

> Move the arrow to the desired position and click as shown in the following picture.





Select the point here

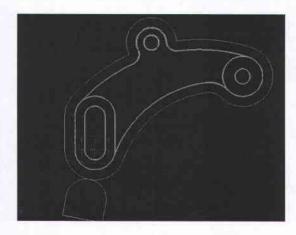
Select the **OK** button to exit the **Chaining** dialog box.

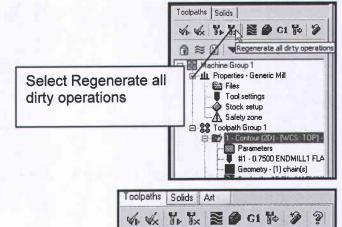
**V** 

> Select the OK button again to exit Chain Manager.



- > Select the Toolpaths Manager to activate it.
- > Select the Regenerate all dirty operations icon for the change to be applied.





商業園▼▲耳◆◎●

Toggle tooloath display on selected operations

Select Toggle toolpath display on selected operations to remove the toolpath display

### **STEP 16:**

CREATE EXTRA GEOMETRY TO BE ABLE TO OVERLAP THE OPEN AREAS OF THE TOP OF THE PART.

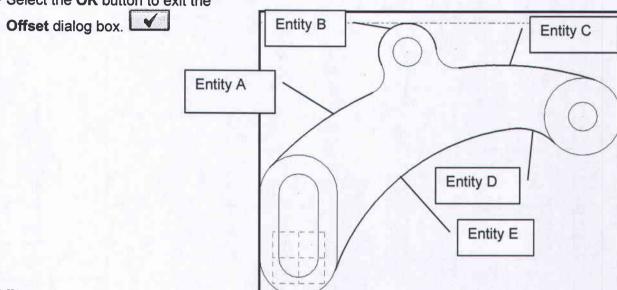
#### **Xform**

- > Xform Offset
- > Enter the distance value 0.5.





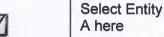
- > [Select the line, arc, spline or curve to offset]: Select Entity A.
- > [Indicate the offset direction]: Select a point outside the part.
- Repeat this procedure for Entities B-E.
  Select the **OK** button to exit the

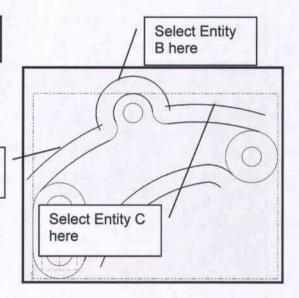


Toolpaths | Trim 2 entity

#### **Edit**

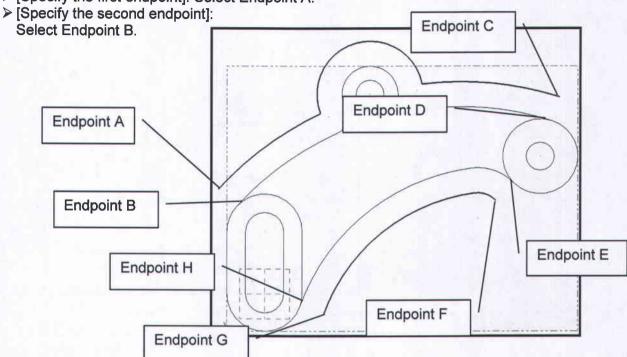
- > Trim/Break
- > Trim/Break/Extend
- > Select the Trim 2 Entities button.
- > [Select entity to trim/extend]: Select Entity A.
- > [Select the entity to trim/extend to]: Select Entity B.
- > [Select the entity to trim/extend]: Select Entity B.
- > [Select the entity to trim/extend to]: Select Entity C.
- ➤ Select the **OK** button.





#### Create

- **>** Line
- > Create Line Endpoint
- > [Specify the first endpoint]: Select Endpoint A.



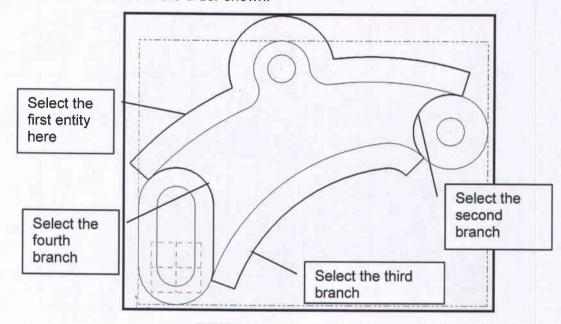
- ➤ [Specify the first endpoint]: Select Endpoint C.
- > [Specify the second endpoint]: Select Endpoint D.
- > [Specify the first endpoint]: Select Endpoint E.
- > [Specify the second endpoint]: Select Endpoint F.
- > [Specify the first endpoint]: Select Endpoint G.
- > [Specify the second endpoint]: Select Endpoint H.
- > Select the **OK** button to exit the command.



## STEP 17: POCKET THE INSIDE OF THE PART.

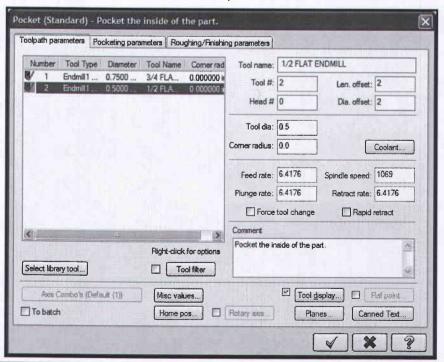
#### **Toolpaths**

- **> Pocket Toolpath**
- > Select the first entity of the pocket chain, as shown.
- > Select the rest of the branches in the order shown.



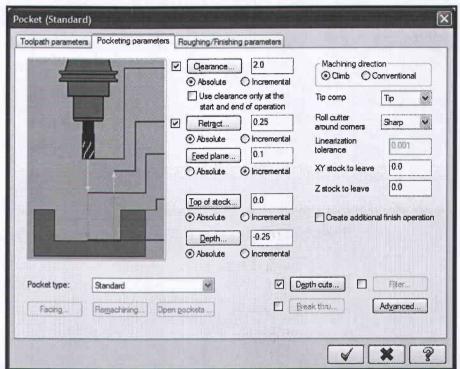
> Select the **OK** button to exit **Chaining**.

➤ Select the 1/2" diameter Flat End Mill using Select library tool and Tool Filter as shown in Step 14. In the Comment field type a comment about the toolpath for future reference.



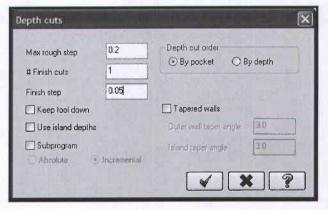


> Make all the necessary changes as shown to the right and select the **Pocket parameters** page.



See Tutorial #1 for more info.

- > Enable the **Depth cuts** and select the button to set the cuts along the Z-axis. Mastercam divides the total depth into equal steps based on the **Max rough step** value.
- > Select the **OK** button to exit **Depth cuts**.

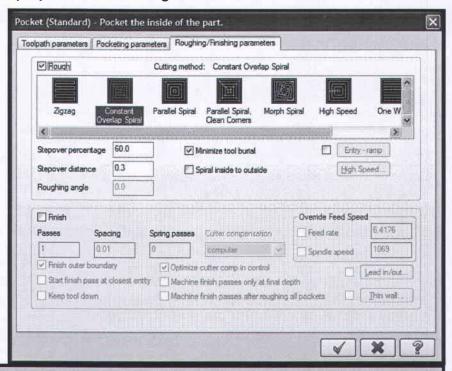




Select the Roughing/Finishing parameters tab and make the necessary changes.

> Select the Constant Overlap Spiral as the Cutting method.

> Disable the Finish area.

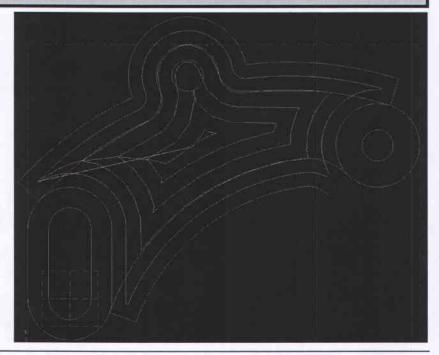


沚

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. Finish area enabled allows the tool to make another cut around the pocket walls to "contour" the walls. Used when Zig-zag or One way was selected as cutting method, to remove the scallops left by these cutting methods. It can be also used to finish only the pocket walls.

> Select the **OK** button to exit

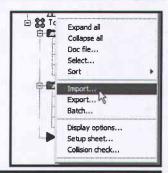
Pocket parameters.



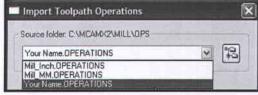


## STEP 18: IMPORT FROM LIBRARY 9/16 TAP HOLES.

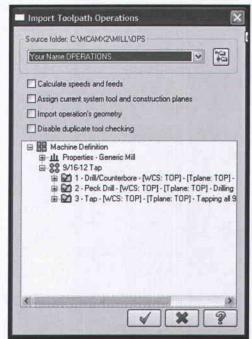
> Right-mouse click in Toolpaths Manager window.



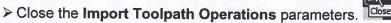
- > Select Import...
- > Select the drop-down arrow in the **Source** folder and select the **Your name. Operations**.



- > Click on the 9/16-12 Tap group.
- Make sure that all three operations (spot drilling, drilling and tapping) are highlighted.
- > Select the **OK** button.



- > Select the **Yes** button to import the operation group.
- > Select the **OK** button.



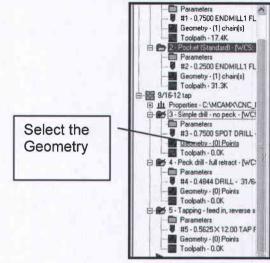


Note that all the imported operations have no geometry. We are going to add the center points and regenerate the toolpaths as shown in the following steps.





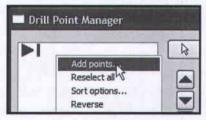
> Select the first drill operation, Geometry.



> Select the OK button.



➤ Right-mouse click and Select Add points...



> Select the Mask on Arc button in the Drill Point Selection.





> [Select the arc to match]: Select the 9/16 " diameter arc as shown. Select the 9/16 diameter arc > Select a window around the part. Select a window around the part > Hit Enter when finished. > Select the **OK** button twice to exit. > Select the Regenerate all selected operations icon for the change to be applied. Toolpaths | Solids | % % ₹ ₹ 8 @ G1 % > Regenerate all selected operations ➤ Holding the **right-mouse** button, drag the new **Geometry** (2) Points on top of Peck drill Geometry. ☐ 
☐ 3 - Simple drill - no peck - [WC!] Parameters > Release and select Add. #3 - 0.7500 SPOT DRILL -Right-mouse Geometry · (2) Points click on Toolpath - 4.4K

Geometry

> Select No button to do not retain the depth values of the imported operation.



4 · Peck drill · full retract · [WC:

Took Add

Par

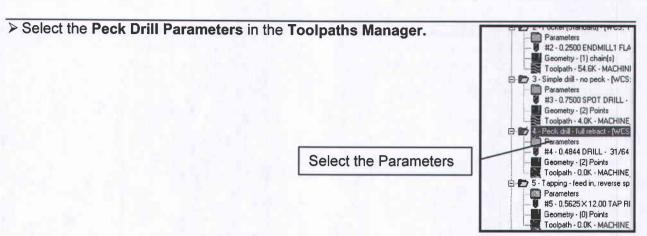
#4 - 0.4844 DRILL - 31/6-

Replace

#5 0.0020 A TE. 00 TAP F

rse s

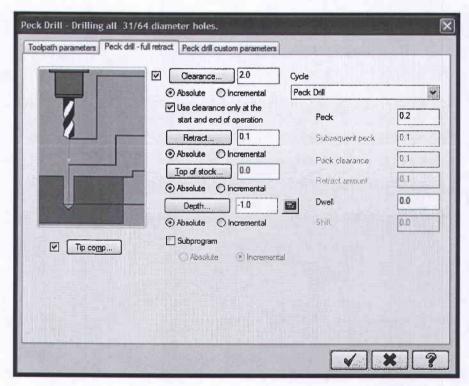




> Change the Comment in the Toolpath parameters page to "Drilling all 31/64 diameter holes."

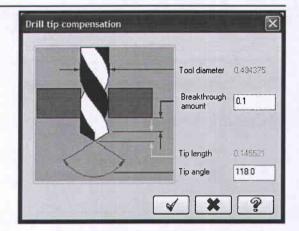
> In the second parameter page change the drilling cycle and the **Depth** as shown in the picture

to the right.





- Select the Tip comp button and change the breakthrough amount value as shown.
- > Select the **OK** button to exit the **Drill tip** compensation window.



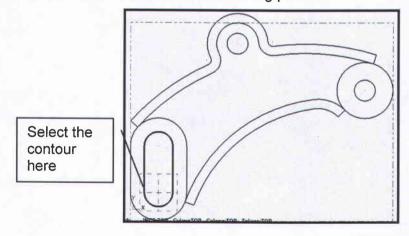
- > Select the **OK** button to exit drilling parameters.
- > Copy the geometry in the tapping toolpath following the same steps shown for drilling geometry.
- > Select the Regenerate all dirty operations icon.



## STEP 19: CREATE A POCKET TOOLPATH FOR THE 1.25 LONG SLOT.

#### **Toolpaths**

- ➤ Circle Paths
- > Slot Mill Toolpath
- > Select the chain as shown in the following picture.

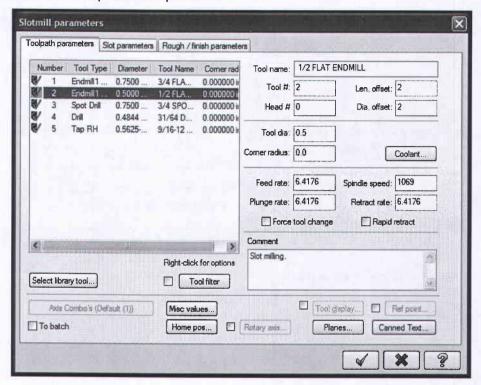


> Select the **OK** button to exit the **Chaining** dialog box.

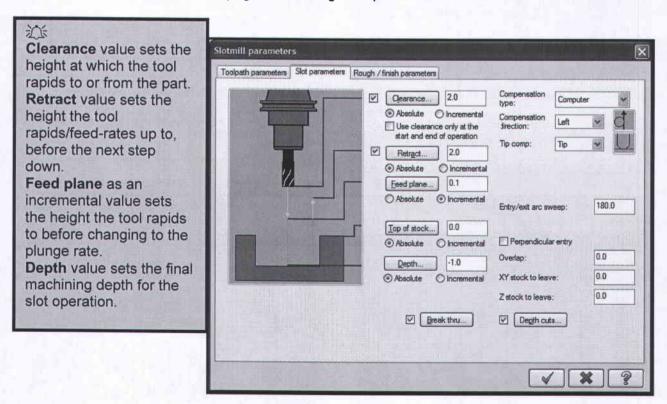




> Select the 1/2" End mill used in the previous operation.



> Select the Slot parameters page, and change the parameters as shown:





➤ Enable the **Depth cuts** and select the button to set the parameters as shown.

沇

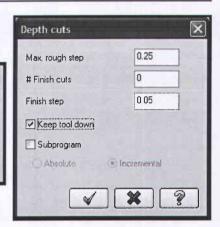
**Depth cuts** set the cuts along the Z-axis. Mastercam divides the total depth into equal steps based on the **Max rough step** value.

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

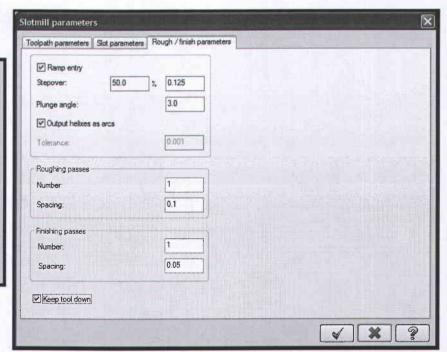
> Select the **OK** button to exit the **Depth cuts** window.

Select the Rough/Finish parameters page and check if you

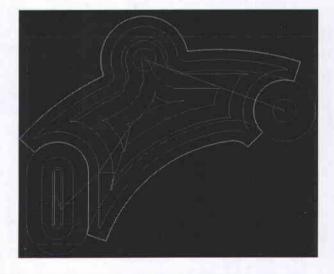
have the same settings as shown in the next picture.



Ramp entry sets the tool to ramp down to the cutting depth on the first pass. The next passes are performed in the XY plane based on the stepover amount. The Number of Finishing passes and the Spacing amount set passes that the tool will remove last.



> Select the **OK** button to exit slot parameters pages.



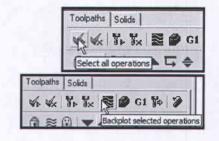


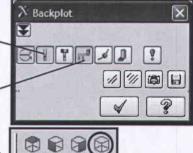
## TOOLPATH VERIFICATION STEP 20: BACKPLOT THE TOOLPATH.

- > Click on Select all operations in the Toolpaths Manager.
- > Select the Backplot selected operations button.
- > Make sure that you have the following buttons turned on (they will appear pushed down).
- ▶ Display tool
- > Display rapid moves

Display tool

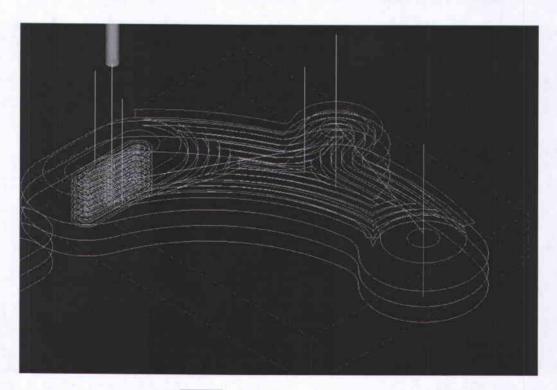
Display rapid moves





- > Select the Isometric View from the view toolbar to see the stock.
- ➤ Select Fit button.
- > Select the Play button.





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

Page 3-34



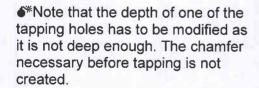
### STEP 21: VERIFY

- > Select the **Verify selected operations** button.
- > Make sure that you have the same parameters enabled as shown.
- > Check Tutorial #1 for more details.

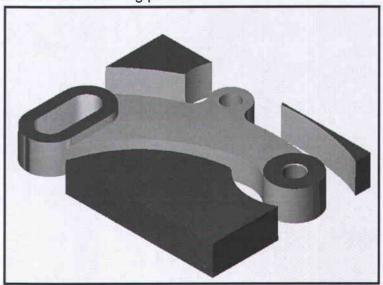




- > Set the **Verify speed** by moving the slider bar in the speed control bar.
- Select the **Machine** button to start simulation.
- > The finished part should appear as shown in the following picture.



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



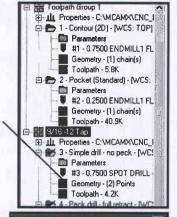




## STEP 22 EDIT THE DEPTH IN A DRILLING OPERATION.

Select the Spot drilling toolpath Geometry in the Toolpaths Manager.

> Select the Geometry





- > Right-mouse click on Point 2. > Select Change at point.
- > Enable **Depth** and enter the value as shown to the right (-.5).
- > Select the OK button to exit Drill change at point.
- Select the OK button to exit Drill Point Manager.



Toolpaths | Solids |

Vo Va To Tr

- > Select the Regenerate all dirty operations button.
- > Select Verify and check the toolpaths again.

## **STEP 23:** POST THE FILE.

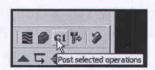
- > Make sure that all operations are selected, otherwise
- > Select all operations button.



€ 61 %

Regenerate all dirty operations

> Select the Post selected operations button from Toolpath Manager.



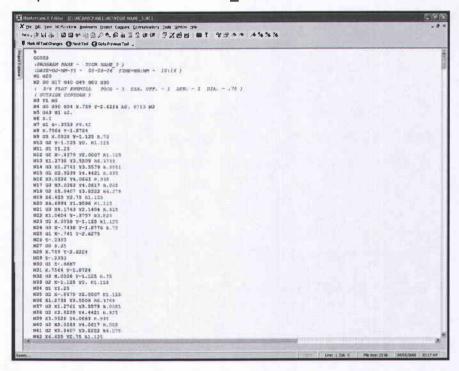


➤ In the **Post processing** window, make all the necessary changes as shown to the right.



> Select the **OK** button to continue.

> Select the Save button to accept the File Name "Your Name\_3.NC"

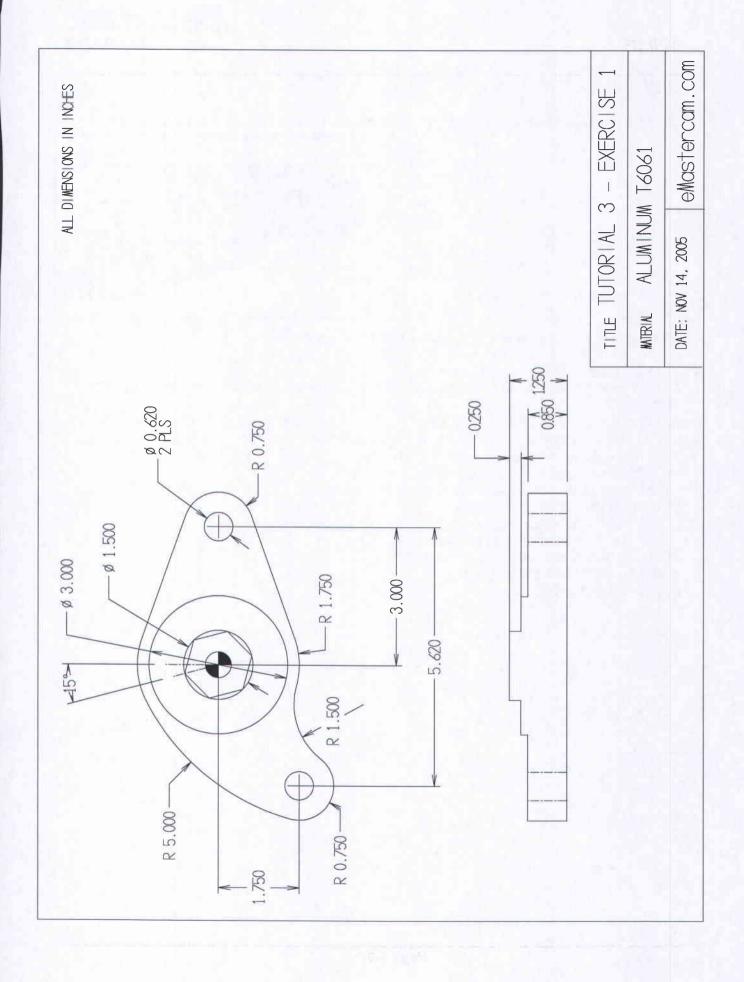


> Select the red X to exit Mastercam Editor.

## STEP 24: SAVE THE UPDATED MCX FILE.

> Select the Save icon.

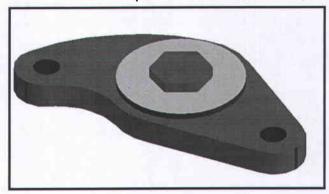






#### **REVIEW EXERCISES.**

Student practise. Create the Toolpath for Exercise 1-Tutorial 3 as per the instructions below;



Tips:

**Stock** size use Bounding box to establish X & Y sizes and give Z = 1.0"

Center Drill all holes using ½ Center Drill

**Drill** all holes through using 5/8 Drill (copy the previous toolpath and get the new tool and the depth)

Contour the part using 2 " Flat End Mill

Max rough step =.5" (Depth cuts)

1 roughing passes with spacing =1.5"

1 finish pass .05" (Muti passes)

Offset the ouside contour geometry with .5 distance

Pocket the part using ½ " Flat End Mill

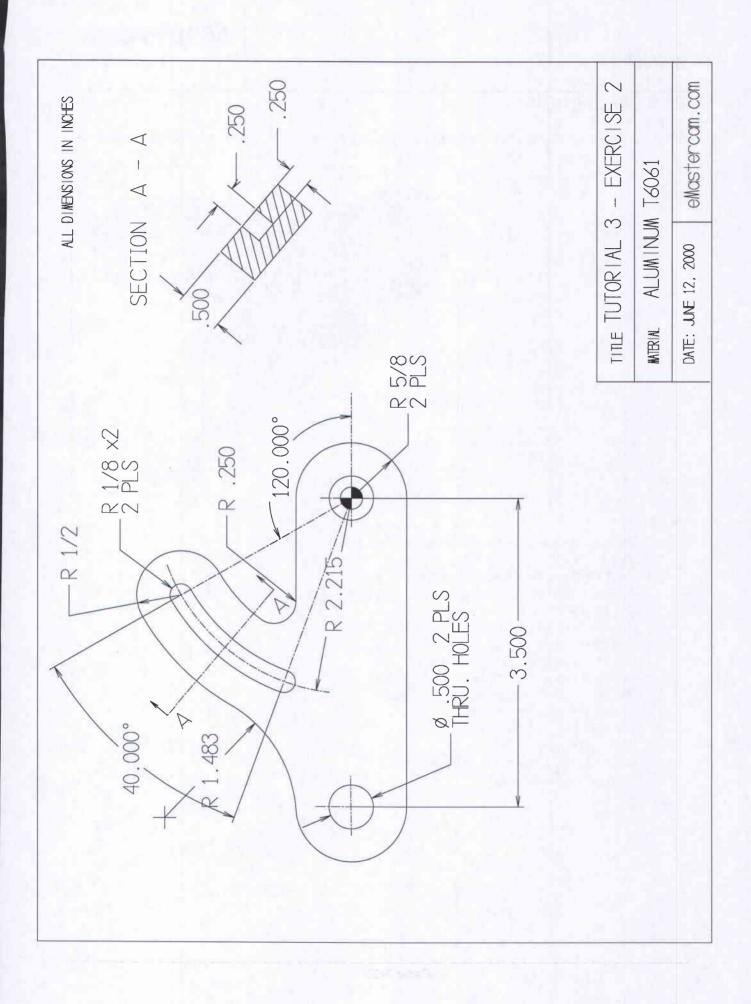
Select the ouside contour first and then the 3" diameter.

Use True spiral cutting method

Contour the hexagon using 2 " Flat End Mill

Backplot and Verify the toolpaths.

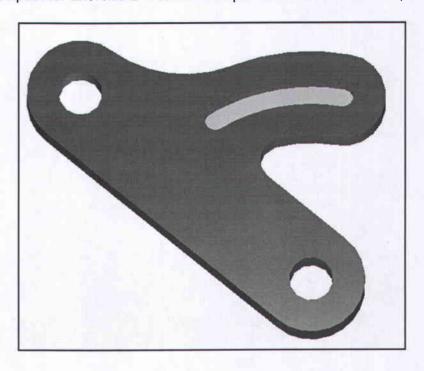
Post process the file.





### **REVIEW EXERCISE.**

Student practise. Create the Toolpath for Exercise 2- Tutorial 3 as per the instructions below;



### Tips:

**Stock** size use Bounding box to establish X & Y sizes and give Z = 1.0"

Spot Drill the two holes using 3/4 Spot Drill

**Drill** all holes through using 1/2 Drill (copy the previous toolpath and get the new tool and the depth)

Pocket the slot using 1/8 " Flat End Mill

Use depth cuts

Use Parallel Spiral cutting method

Contour the part using 3/8 " Flat End Mill

Max rough step = 0.1875 " (Depth cuts)

1 roughing passes with spacing =0.1"

1 finish pass .05" (Muti passes)

Break thru amount 0.1

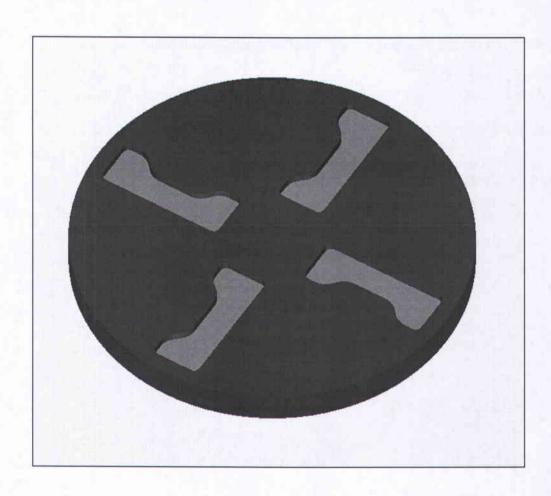
Backplot and Verify the toolpaths.

Post process the file.

### **TUTORIAL SERIES FOR**

### Mastercam.X°

# TUTORIAL 4 LEVEL 1 – 2D TRANSFORM TOOLPATHS USING ROTATE & MIRROR.





### **Objectives:**

### The Student will design a 2-dimensional drawing by:

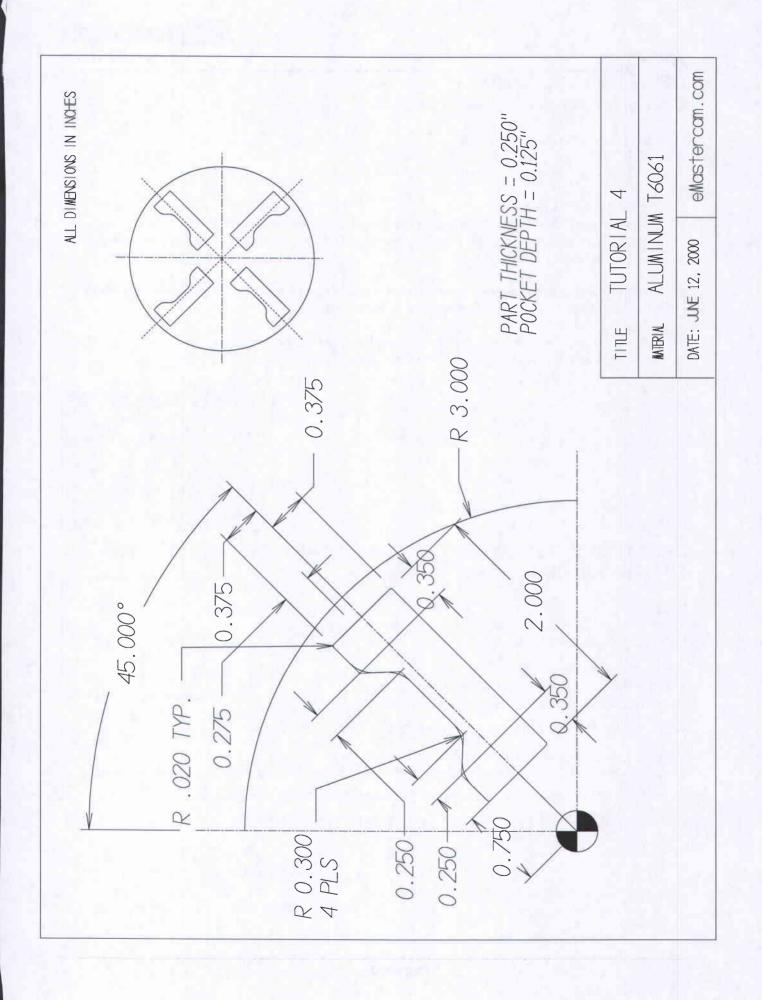
- > Creating a circle knowing the center and the diameter.
- > Creating horizontal and vertical lines.
- > Creating lines using co-ordinate positioning.
- > Creating parallel lines, by defining the offset direction and distance.
- > Creating rectangular shape.
- > Trimming the existing geometry.
- ➤ Creating fillet radii.
- Mirroring existing geometry to complete a part.
- > Rotating one shape two times.

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

- > A 2-dimensional pocket.
- > The rest of the pockets created using Transform toolpaths, Rotate and Mirror.

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

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





# **GEOMETRY CREATION**Setting the toolbar states

To start a new file from Mastercam:

File

> New

Setting the toolbar states

- 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-5 in the **User Notes**
- **Toolpaths/Solids manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.
- Before starting the geometry make sure that the **Grid** is enabled. It will show you at each moment where the part origin is. See **Getting started** page A-5 for details.

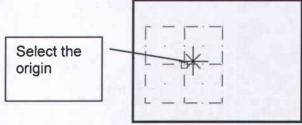


### STEP 1:

CREATE THE OUTSIDE ARC KNOWING THE CENTER POINT AND THE RADIUS/DIAMETER.

#### Create

- > Arc
- > Create Circle Center Point
- ➤ Enter the **Diameter** value 6.
- > [Enter the center point]: Move the cursor at the center location of the grid to select the **Origin**.
- > Use the Fit icon to fit the drawing to the screen.
- > Select the **OK** button to exit the command.
- During the geometry creation of this tutorial, if you make a mistake you can undo the last step using 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. 
   □





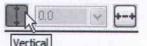
### STEP 2:

# CREATE THE CENTER LINES USING VERTICAL AND HORIZONTAL LINES.

- **♠**\*\*Change the color of the center lines to avoid confusion.
- > Select the Color icon in the Status bar.
- > Select the color red (No 12).
- > Select the **OK** button to exit the command.

### Create

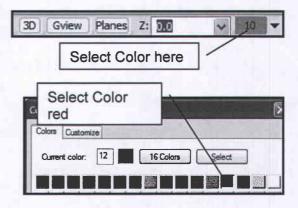
- **>** Line
- > Create Line Endpoint

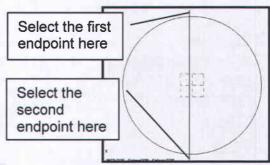


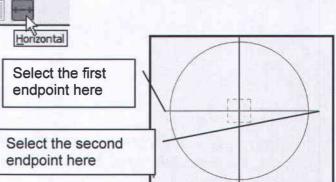
lo.d

0.0

- > Select the Vertical icon.
- ➤ [Specify the first endpoint]: Select a point outside the circle at approximately 90 degrees
- ➤ [Specify the second endpoint]: Select a point outside the circle at approximately 270 degrees as shown to the right.
- > [Enter the X coordinate]:
- > Select the Apply button.
- > Select the Horizontal icon.
- ➤ [Specify the first endpoint]: Select a point outside the circle at approximately 0 degrees.
- ➤ [Specify the second endpoint]: Select a point outside the circle at approximately 180 degrees as shown in the picture to the right.







> [Enter the Y coordinate]:



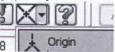
➤ Disable the Horizontal button



### STEP 3:

# CREATE POLAR LINES KNOWING AN ENDPOINT, THE ANGLE AND THE LENGTH.

- > Enter the line **Length** 4.0 (Tab).
- > Enter the **Angle** in degrees 45 (Enter).
- > [Specify the first endpoint]: Select the drop-down arrow and pick **Origin** as shown.



- > Select the Apply button.
- > Enter the line **Length** 4.0 (Tab).
- ➤ Enter the **Angle** in degrees 135 (Enter).
- > [Specify the first endpoint]: Select the drop-down arrow and pick Origin as shown above.
- ➤ Select the **OK** button.

### STEP 4:

### CREATE A PARALLEL LINE.

Select the Color icon in the Status bar.



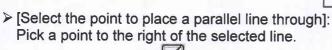
Select Entity A

- > Select the color green (No 10).
- > Select the **OK** button to exit the command.

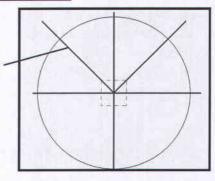
# Colors Customize Current color: 10 16 Colors Select Select color green

### Create

- **▶** Line
- > Create Line Parallel
- ➤ Enter the **Distance** 1 0.75.
- > [Select a line]: Select Entity A.



➤ Select the **OK** button.





### STEP 5:

# CREATE A RECTANGLE IN THE PROPER ORIENTATION, KNOWING THE WIDTH, THE HEIGHT AND THE BASE POINT.

- ➤ Create
- > Create Rectangular Shapes
- > Select the radio button in front of Base Point.
- > Type the Width and the Height as shown in the picture to the right.
- > Select the double arrow to expand the Rectangle ShapesOptions if needed.

Select the rectangle shape

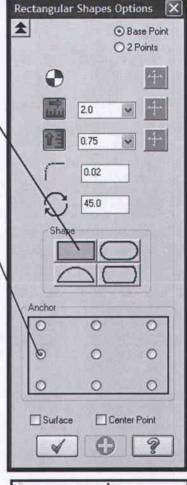
- > Enter the fillet Radius 0.02.
- > Enter 45 degrees for Rotation.

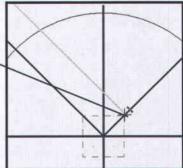
Select this radio button

> Select the radio button shown, as the **Anchor**.

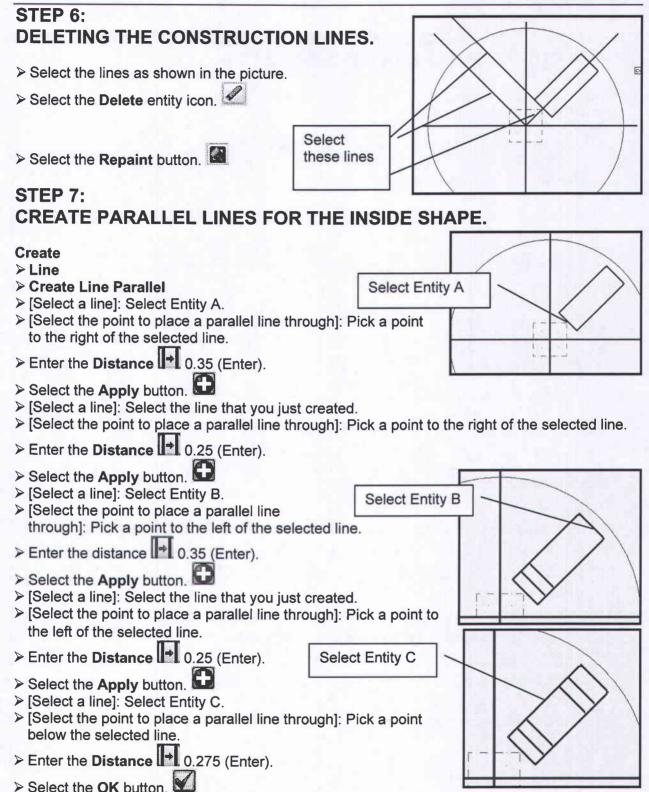
- ➤ [Select position of base point]: Select the endpoint as shown.
- > Select the **OK** button to exit the rectangle dialog box.

Select the endpoint here

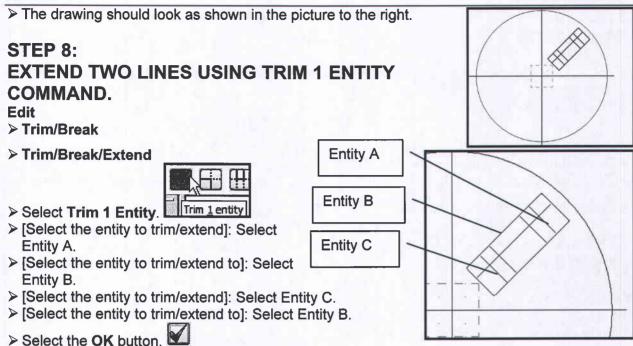












### STEP 9:

### CREATE LINES KNOWING THE TWO ENDPOINT COORDINATES.

Create

**>** Line

➤ Create Line Endpoint

♠ By moving the cursor exactly on the endpoint of the entity the system will display a small

square and the endpoint symbol. For the second endpoint the system will display the intersection symbol.

Select Endpoint A

Select Endpoint D

Select Endpoint B

> [Specify the first endpoint]: Select Endpoint A.

> [Specify the second endpoint]: Select Endpoint B.

➤ [Specify the first endpoint]: Select Endpoint C.

➤ [Specify the second endpoint]: Select Endpoint D.

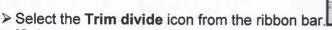
> Select the **OK** button.



### **STEP 10:** TRIM DIVIDE.

### Edit

- > Trim/Break
- > Trim/Break/Extend



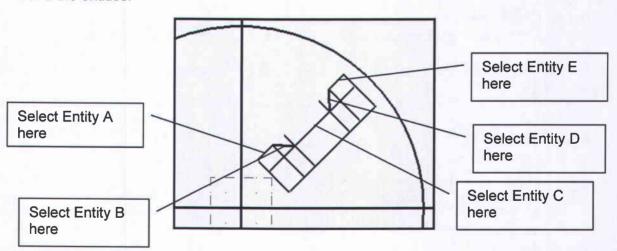
> [Select the curve to divide]: Select Entity A.

> Select the **OK** button to exit.

### **STEP 11:** CREATE FILLET.

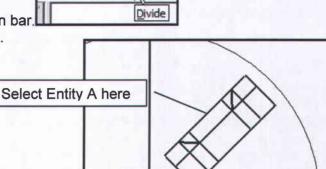
### Create

- > Fillet
- > Fillet Entities
- > Enter the fillet Radius (0.3.
- > [Select an entity]: Select Entity A.
- > [Select another entity]: Select Entity B.
- Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities.



- > [Select an entity]: Select Entity B.
- > [Select another entity]: Select Entity C.
- > [Select an entity]: Select Entity C.
- > [Select another entity]: Select Entity D.
- > [Select an entity]: Select Entity D.
- > [Select another entity]: Select Entity E.
- ➤ Select the **OK** button.





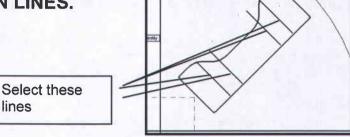
# Mastercam. TUTORIAL 4

### **STEP 12:**

### **DELETE THE CONSTRUCTION LINES.**

> Select the lines as shown to the right.

> Select the **Delete** entity icon.



### STEP 13: MIRROR THE PART ABOUT Y- AXIS.

### Xform

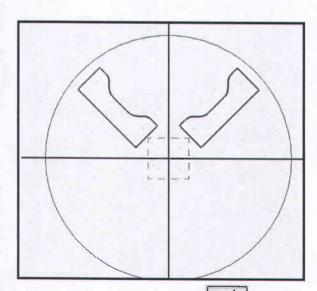
- > Xform Mirror
- > [Select entities to mirror ]:
- ➤ Hold-down the Shift key and select one entity of the profile as shown to the right.



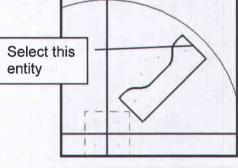
Click on the End Selection button.

> Set the parameters in the Mirror dialog box as shown.

The preview should look as shown in the picture below.



> Select the **OK** button to exit.







### **STEP 14:**

### ROTATE THE PART ABOUT THE ORIGIN.

### **Xform**

- > Xform Rotate
- > [Rotate: Select entities to rotate]: Select the All

button.

> Select **Xform Result** to select the shape resulting after mirroring.

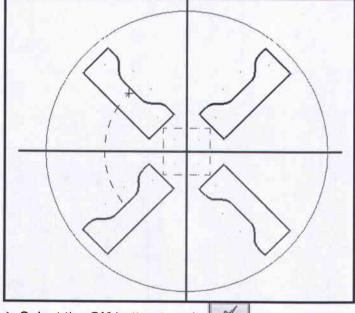


- > Click on the End Selection button.
- > Make sure that you make all the changes as shown in the screenshot.

ON ?

End Selection

- > Change the number (#) to 2.
- > Enter the angle as 90 degrees.
- If the Preview is active you should be able to see the result.



> Select the **OK** button to exit.



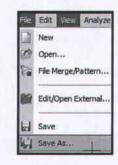
> Clear colors

### STEP 15: SAVE THE GEOMETRY.

### File

- > Save As
- ➤ File Name: "Your Name\_4"
- > Select the **OK** button.



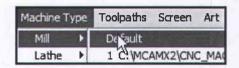


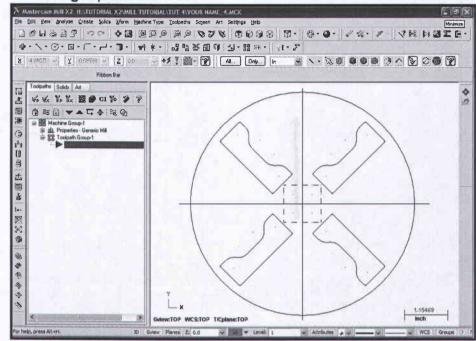


# TOOLPATH CREATION STEP 16: SET UP THE STOCK TO BE MACHINED.

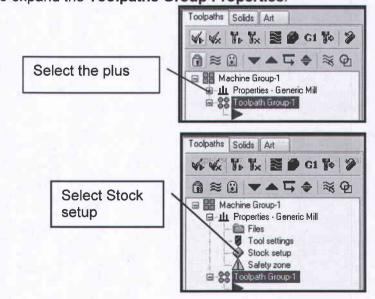
### Machine type

- > Mill
- > Default
- > To display the Toolpaths Manager press Alt + O.





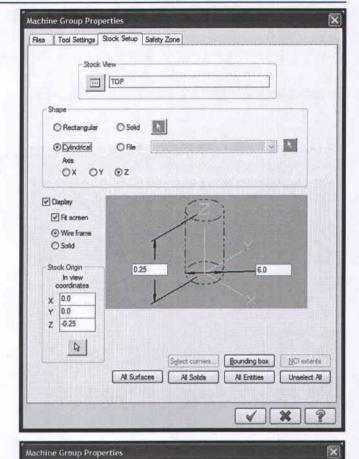
> Select the plus in front of Properties to expand the Toolpaths Group Properties.



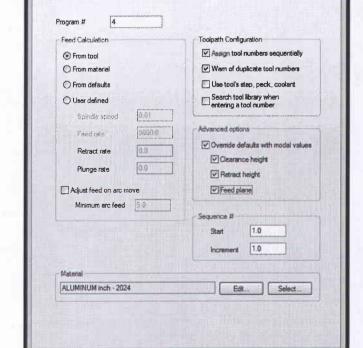
> Select Stock setup.



Change the parameters to match the screenshot to the right.



➤ Select the **Tool Settings** and change the parameters to match the following screenshot.



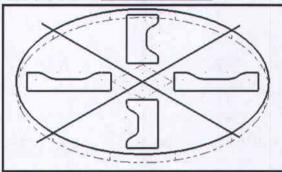
×

Files | Tool Settings | Stock Setup | Safety Zone |

➤ Select the **OK** button to exit **Toolpath Group Properties**.

> Select the Isometric View from the view toolbar to see the stock.





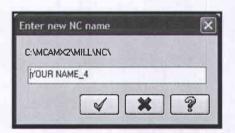
> Select the **Top View** from the view toolbar to see the part from the top.

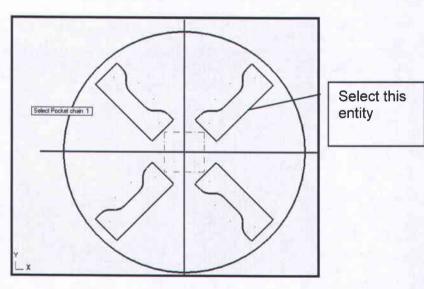


# STEP 17: POCKET ONE OF THE INSIDE SHAPES.

### **Toolpaths**

- ➤ Pocket Toolpath
- > Select one entity of the pocket chain, as shown below.
- > Select the **OK** button to accept the **NC** file name.





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





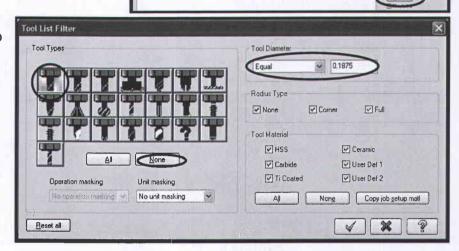
Right-click for options

Misc values...

Tool filter

- > Click on the Select library tool button.
- > Select the Filter button in the Tool Selection window.
- ➤ In the Tool Types field select the None button to disable all tools.
- Select the Flat Endmill button in the Tool Type list as shown.
- ➤ In the Tool Diameter field click the pull-down arrow and select Equal.
- ➤ Enter the **Tool Diameter** value: 0.1875.
- Select the OK button to exit Tool List Filter.





Select library tool...

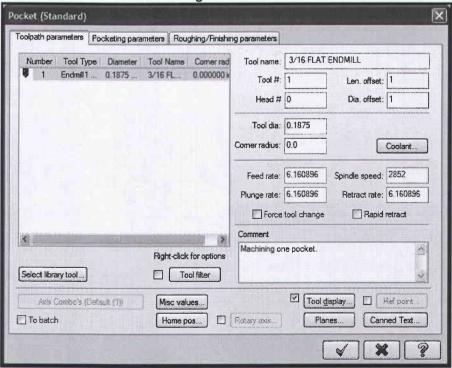
Mile State Tool Type Dismeter Tool None Corner today Roday Type Dismeter Tool None Corner today Roday Type 229 England 05000 1/2 FLA 0000000 n. None

Axis Combo's (Default (1))

- ➤ Make sure that the tool is selected (highlighted) in the Tool Selection window.
- > Select the **OK** button to exit **Tool Selection** window.

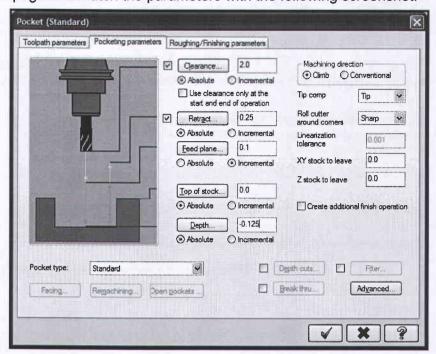
> Make the changes as shown in the screenshot to the right.

The Feed rate,
Plunge rate, Retract
rate and Spindle
speed are based on
the tool definition.
Change them as
desired.



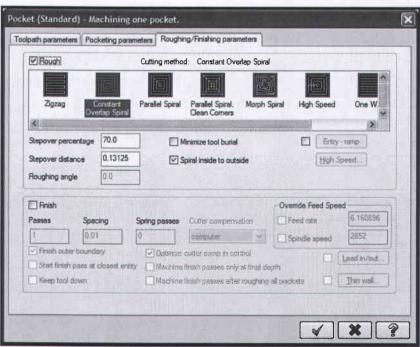


> Select the Pocket parameter page and match the parameters with the following screenshot.



> Select the Roughing/Finishing parameters tab.

Finish area enabled allows the tool to make another cut around the pocket walls to "contour" the walls. Used when Zigzag or One way was selected as cutting method, to remove the scallops left by these cutting methods. It can be also used to finish only the pocket walls.



- > Select the Constant Overlap Spiral as the Cutting method.
- > Enable Spiral inside to outside to spiral from the center to the pocket wall.
- Disable the Finish area.



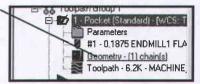
> Select the **OK** button to exit **Pocket** parameters.

### **STEP 18:**

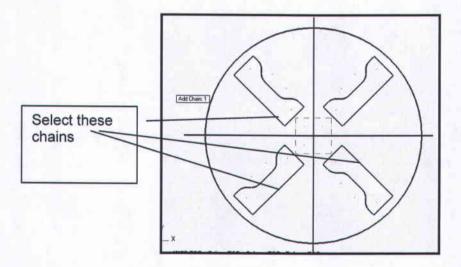
# MODIFY THE TOOLPATH TO COMPLETE THE REMAINING POCKETS. 18.1 Option 1.

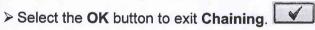
> Select Geometry in the Toolpaths Manager.

Select Geometry

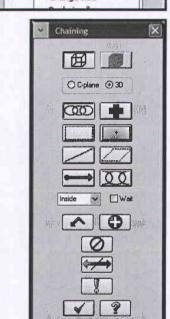


- > Right-mouse click in the Chain Manager.
  > Select Add chain.
- > Select all remaining pockets.





- > Select the OK button to exit Chain Manager.
- > Select the Toolpaths Manager.
- > Select the Regenerate all dirty operations button.





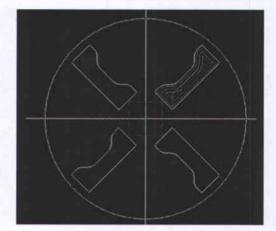


### 18.2 Option 2. Using Transform toolpaths.

- Transform toolpath allows you to copy and change the orientation of previously created operations. You can use this function to translate, rotate, or mirror operations. You must create an operation before using the Transform toolpath. You can also transform a transformed operation. Transform toolpath is associative.
- > Select Geometry in the Toolpaths Manager again.
- > Right-mouse click on Chain 2 in the Chain Manager.
- > Select Delete chain.
- > Repeat these steps to delete Chain 3 and Chain 4.
- > Select the OK button to exit Chain Manager.
- > Select the Regenerate all dirty operations button.

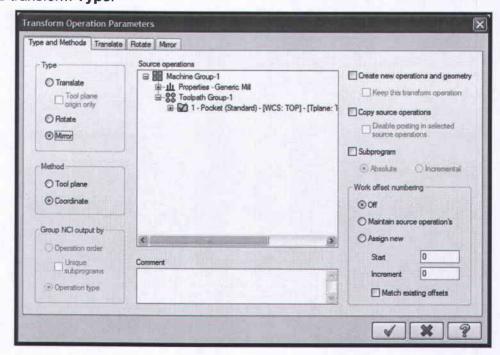






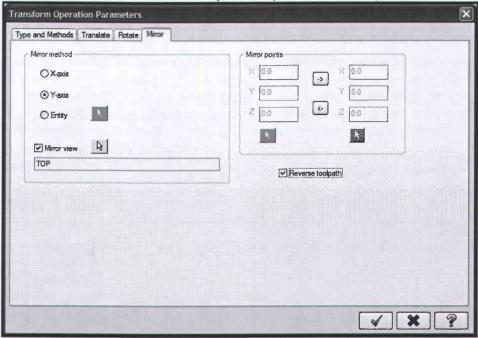
### **Toolpaths**

- > Transform Toolpath
- > Select Mirror in the transform Type.





Select the Mirror tab to set the Y-axis about which the system replicates the toolpath.

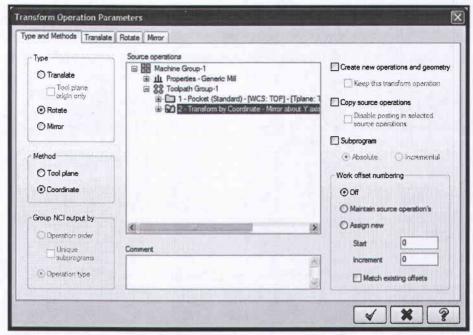


- > Enable Reverse toolpath for the system to cut the second pocket in the same machining direction (climbing or conventional).
- > Select the **OK** button to exit **Transform Operation Parameters**.

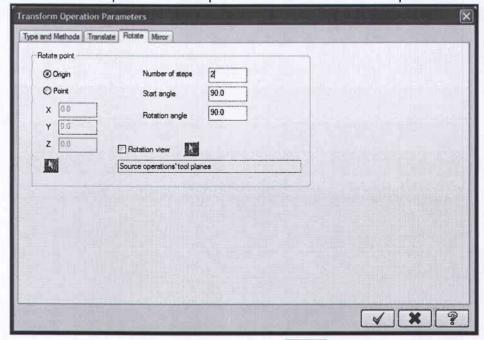


### **Toolpaths**

- > Transform Toolpath
- Select Rotate as transform Type.



- > Enable Coordinate as the Method to allow the system to rotate the toolpath in the same tool plane.
- > Enable the previous **Transform** operation in the **Source** operation field.
- > Select the Rotate tab to establish the parameters required to obtain the other two pockets.



> Select the **OK** button to exit **Transform operation parameters**.



### **STEP 19:** BACKPLOT THE TOOLPATH.

> Select the Select all operations button.

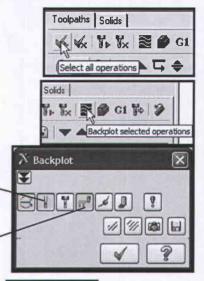
> Select the Backplot selected operations button.

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

➤ Display tool

> Display rapid moves

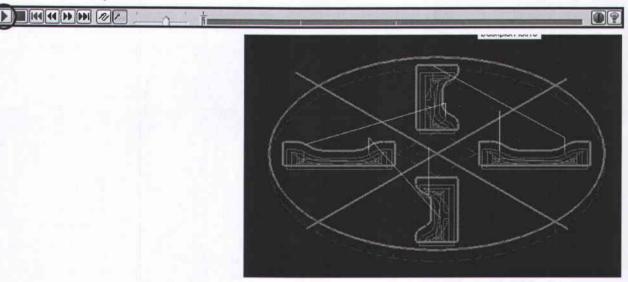
Display tool Display rapid moves



**8 6 6** 

> Select the Isometric View from the view toolbar to see the stock.

> Select the Play button.



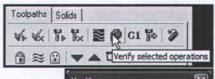
> Select the **OK** button to exit **Backplot**.





### TOOLPATH VERIFICATION STEP 20: VERIFY.

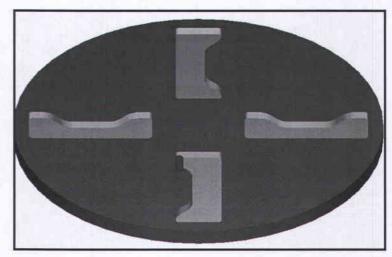
- > Expand the **Toolpaths Manager** if necessary by dragging the right side.
- > Select the Verify selected operations button.





- > Set the Verify speed by moving the slider bar in the speed control
- > Select the **Machine** button to start simulation.
- > The finished part should appear as shown in the following picture.

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





Post processing

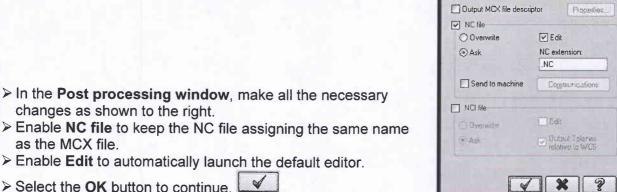
HERAN PST

### **STEP 21:** POST THE FILE.

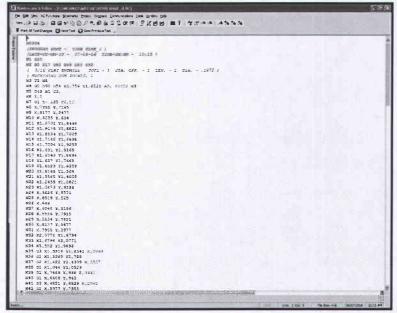
- Make sure that all operations are selected.
- > Select the Post selected operations button from Toolpath Manager.



Select Fost



- changes as shown to the right.
- > Enable Edit to automatically launch the default editor.
- > Select the **OK** button to continue.
- > Select the Save button to accept the file name.

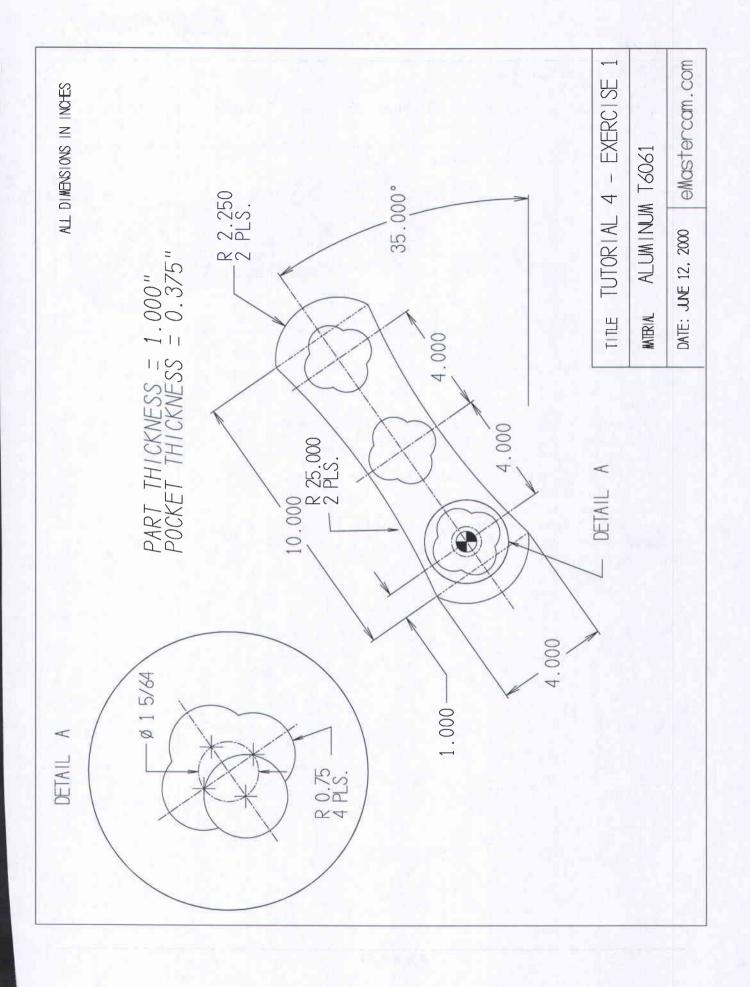


> Select the red X at the upper right side to exit.

### **STEP 22:** SAVE THE UPDATED MCX FILE.

> Select the Save icon.

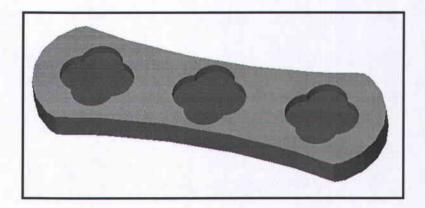






### **REVIEW EXERCISES.**

Student practise. Create the Toolpath for Exercise 1-Tutorial 4 as per the instructions below;



### Tips:

Stock size use Bounding box to establish X &Y sizes and give Z = 1" Contour the part using 1.5" Flat End Mill

Max rough step =0.5" (Depth cuts)

1 roughing passes with spacing =0.5"

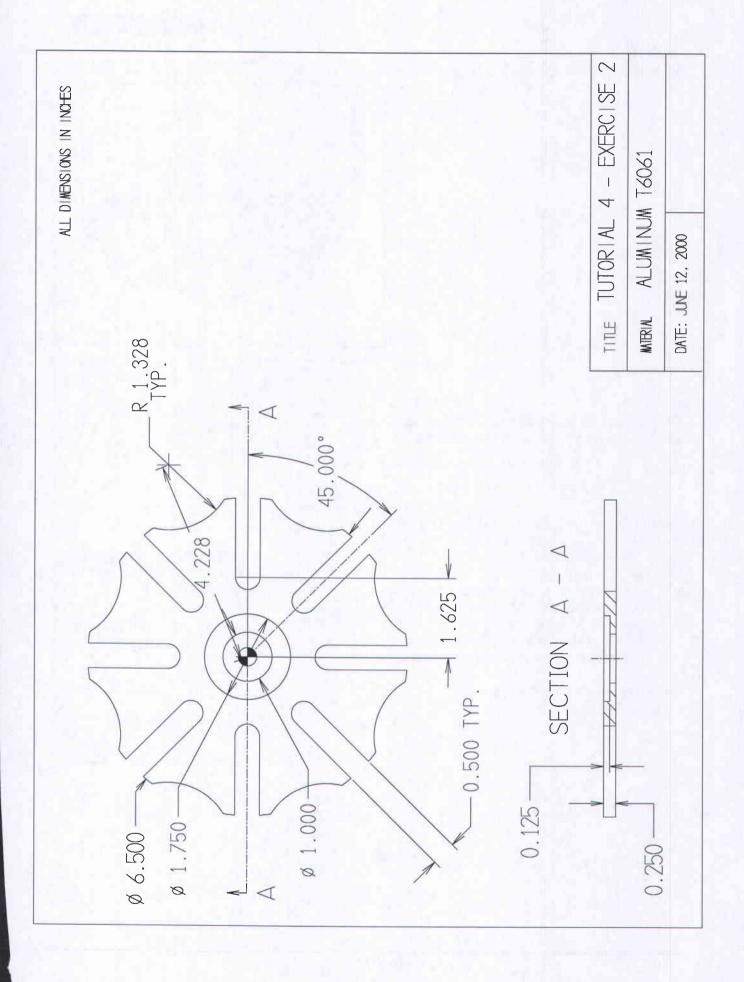
1 finish pass 0.05" (Multi passes)

Pocket the part using 1/2 " Flat End Mill

Use Constant overlap spiral cutting method

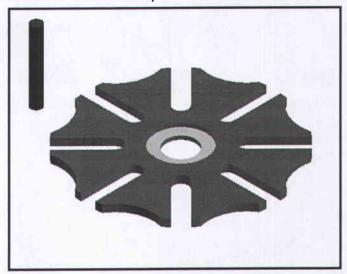
Backplot and Verify the toolpaths.

Post process the file.



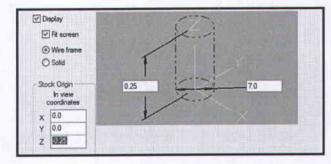


Student practise. Create the Toolpath for Exercise 2-Tutorial 4 as per the instructions below;



Tips:

**Stock** change to **Cylindrical** shape; enable Z–Axis; Height= 0.25" & Diameter = 7.0"



Center Drill & Drill the 1.0" diameter thru hole. Pocket the 1.75"diameter using 1" Flat End Mill

Use True spiral cutting method

Contour the part using the 1.0" Flat End Mill
1 roughing passes with spacing = 0.1"
1 finish pass 0.05" (Multi passes)

Contour remachining the part using 0.375 " Flat End Mill Enable Remachining in the Contour Type and set the parameters as shown in the screenshot.

Backplot and Verify the toolpaths. Post process the file.

