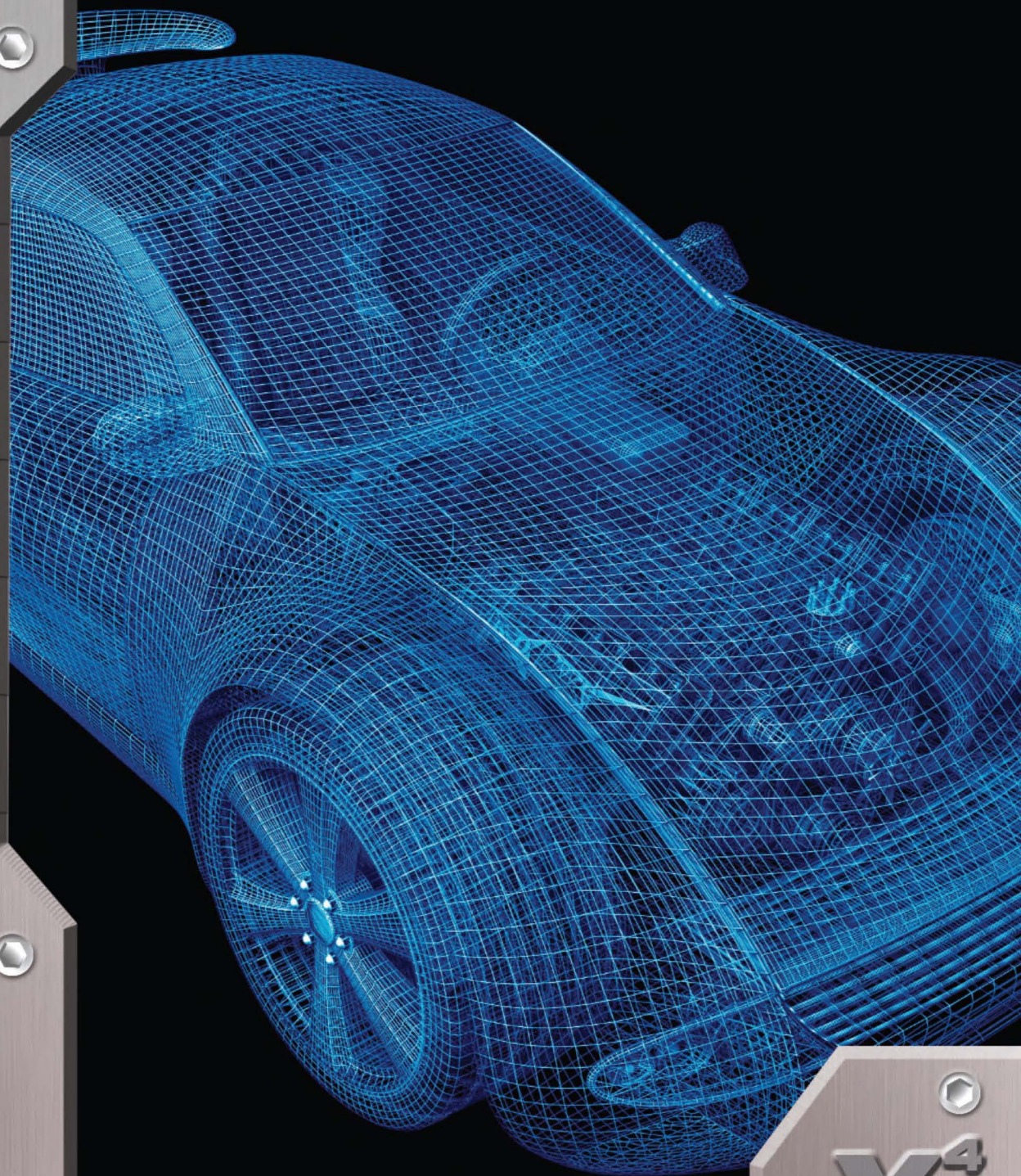


Design  
Training  
Tutorial

*Mastercam*®



*X<sup>4</sup>*



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## **Mastercam® X4 Training Tutorials - Design**

Date: June 18, 2009

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Software: Design Mastercam X4

Authors: Mariana Lendel

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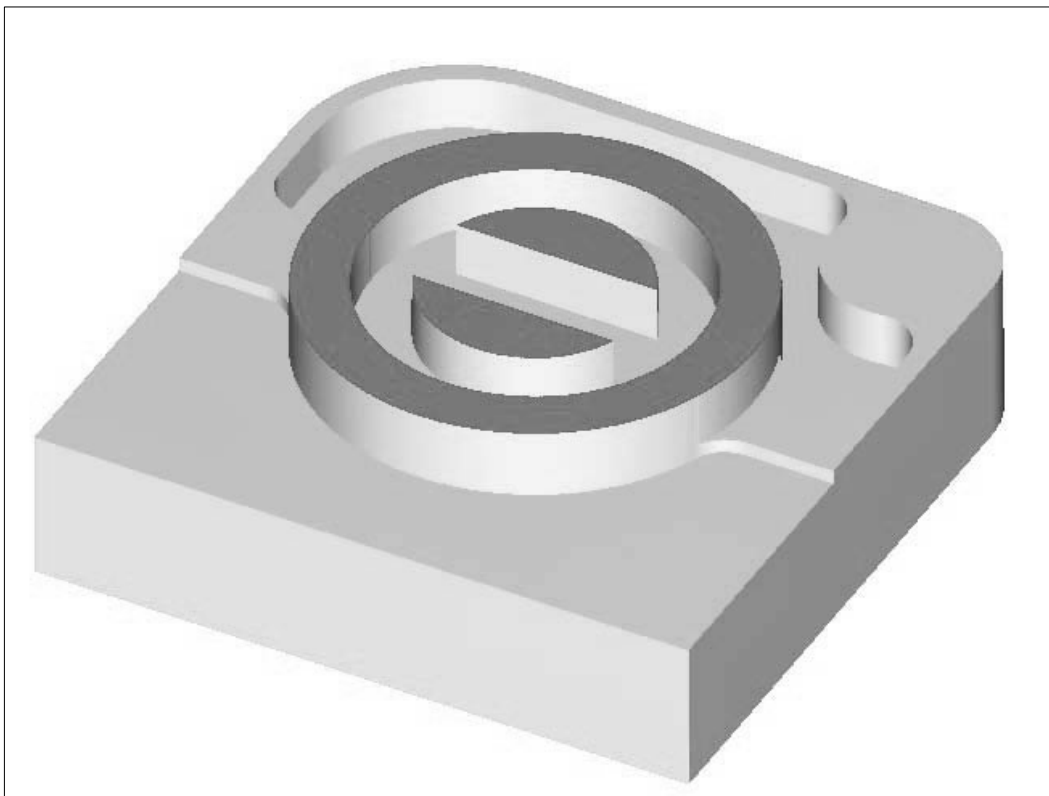
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## TUTORIAL SERIES FOR

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

### TUTORIAL #9 2D WIREFRAME & SOLID GEOMETRY



## **Design**

---

### *Objectives:*

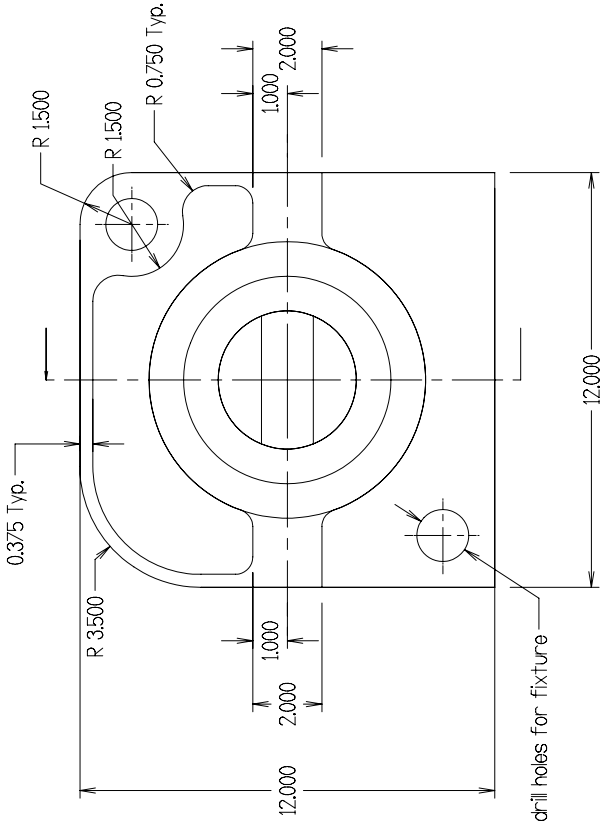
#### ***The Student will design 2-D wireframe in Top Cplane using:***

- Create arc knowing the center point & the diameter.
- Delete extra construction line.
- Create rectangle knowing the width and the height and a point placement.
- Fillet the two corners of the rectangle.
- Offset the rectangle with a given distance.
- Delete extra construction line.
- Rotate the rectangles to complete the wireframe.

#### ***The Student will create the solid using:***

- Change the main color and level.
- Create a solid by extruding a closed contour.
- Create the rest of the geometry using extrude cut body options.





TITLE	TUTORIAL 9	
MATERIAL	ALUMINUM T6061	
DATE: JUNE 12, 2008	eMastercam.com	



## GEOMETRY CREATION

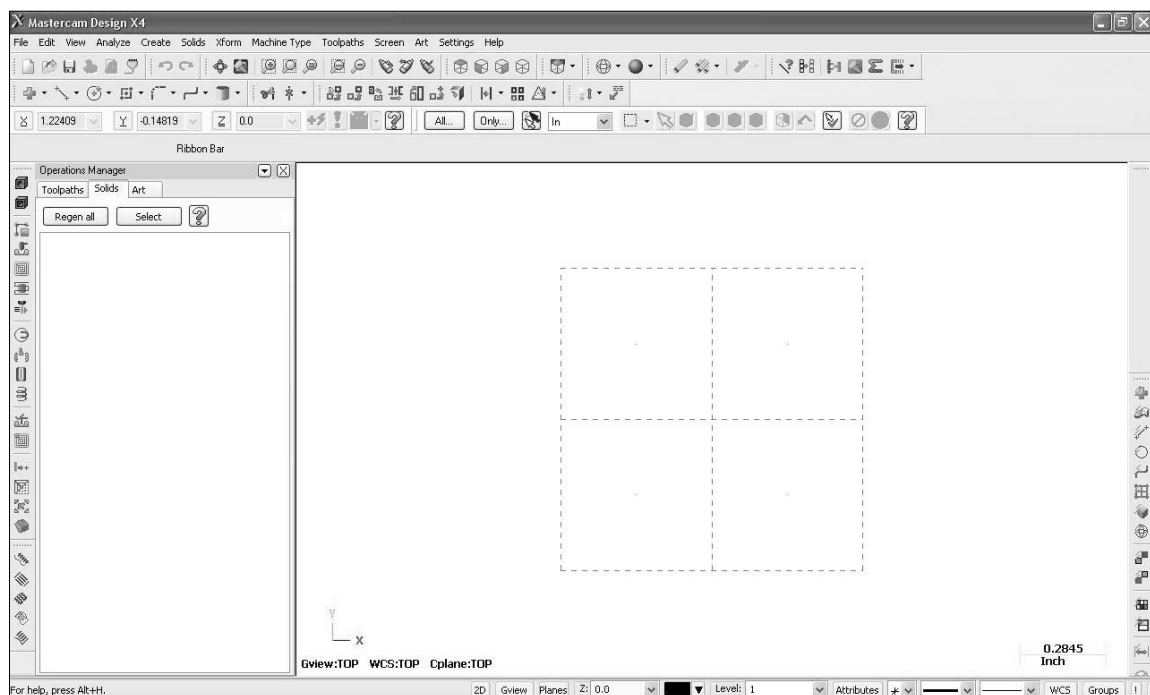
### CREATE THE WIREFRAME FOR THE SOLID

#### *Useful tools to start the geometry creation*

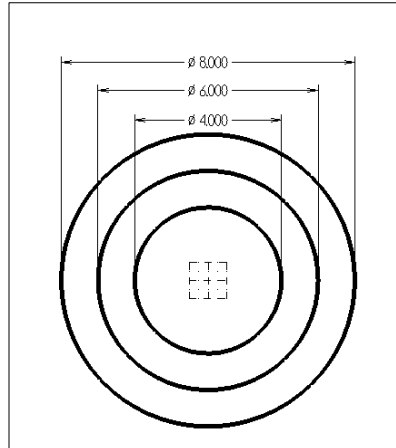
Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the solid geometry. See Getting started page A-4 for details.

Make sure that the **Grid** is enabled. It will show you where the part origin is. See **Getting started** page A-6 for further information.

- ➡ Select the Solids manager to the left of the screen. If it is hidden press **Alt + O** to display it.




- ➡ Note that during the geometry creation to start a command we will select the icons from the toolbar, instead of using the menu.

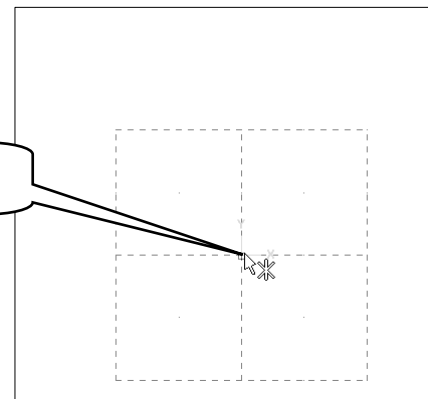
**STEP 1: CREATE THE CIRCLES WITH DIAMETERS 8.0, 6.0, AND 4.0, WHICH SHARE THE CENTER POINT THE ORIGIN***Step Preview:*








- ➔ From the **Sketcher** toolbar, select **Create Circle Center Point** icon



- ➔ Enter the **Diameter** value  8.0 (Enter)
- ➔ [Enter the center point]: Select the **Origin** (the center of the grid) as shown.

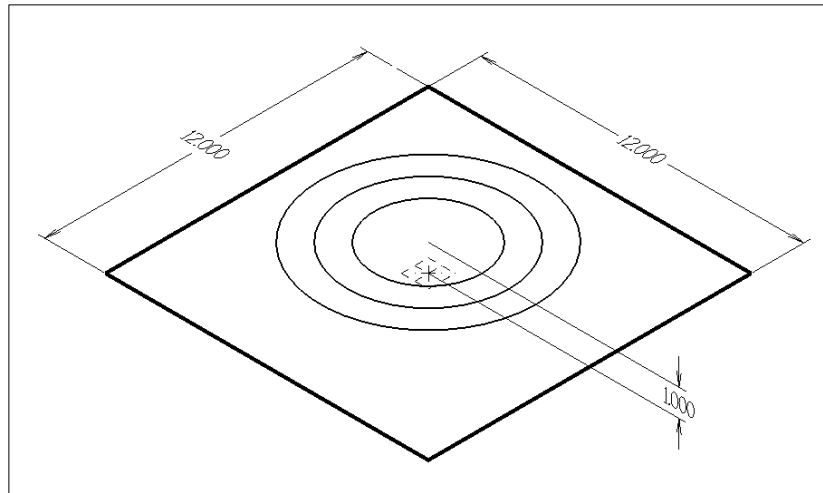
Select the Origin



- ➔ Select **Fit** icon to fit the drawing to the screen. 
- ➔ Select the **Apply** button to continue using the same command. 
- ➔ Enter the **Diameter** value  6.0 (Enter)
- ➔ [Enter the center point]: Select the **Origin** (the center of the grid) as shown before.
- ➔ Select the **Apply** button to continue using the same command. 
- ➔ Enter the **Diameter** value  4.0 (Enter)
- ➔ [Enter the center point]: Select the **Origin** as shown above.
- ➔ Select the **Apply** button to continue using the same command. 
- ➔ Select the **OK** button to exit the command. 

## STEP 2: CREATE THE 12 BY 12 RECTANGLE AT Z-DEPTH -1.0

*Step Preview:*

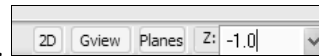


### 2.1 Change the construction mode to 2D and the construction plane depth to Z -1.0

- ➔ Click on **3D** mode and note that the construction mode changes to **2D**.



- ➔ Change the **Z** value in the **Status Bar** to -1.0 as shown.

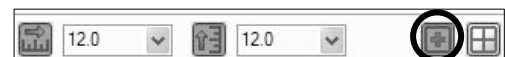


### 2.2 Create the rectangle

- ➔ From the **Sketcher** toolbar, select **Create rectangle** icon.



- ➔ Enter the **Width** 12 and the **Height** 12 and enable **Anchor to center** icon as shown;



- ➔ [Select position of base point]: Select the **Origin**

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

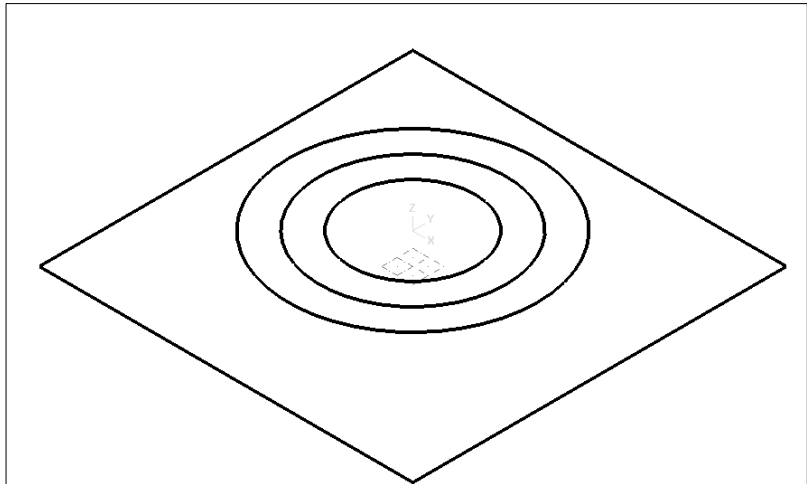
- ➔ Select the **OK** button to exit the command.

- ➔ Change the graphic view to **Isometric**.

**Design**

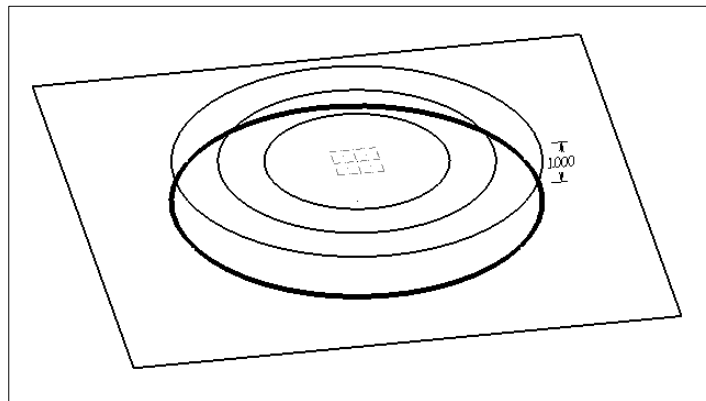
*The geometry should look as shown*

- Note that because of the 2D mode the grid moves down 1.0". With the construction mode set to 3D the rectangle would have been created at Z 0.



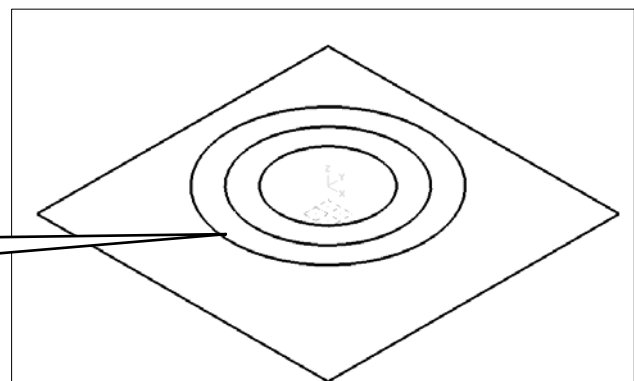
**STEP 3: TRANSLATE – COPY THE 8.0" DIAMETER CIRCLE AT Z -1.0**

*Step Preview:*



- Select the 8.0" diameter circle.

Select the circle

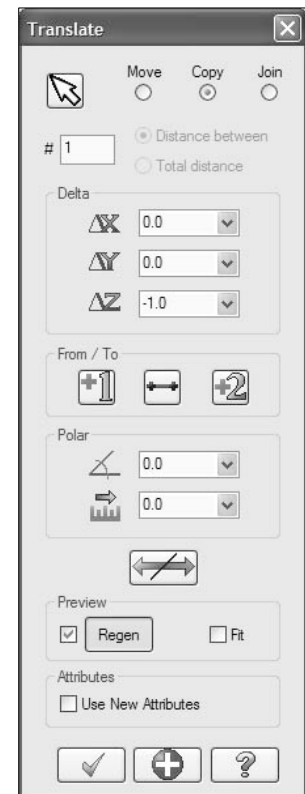
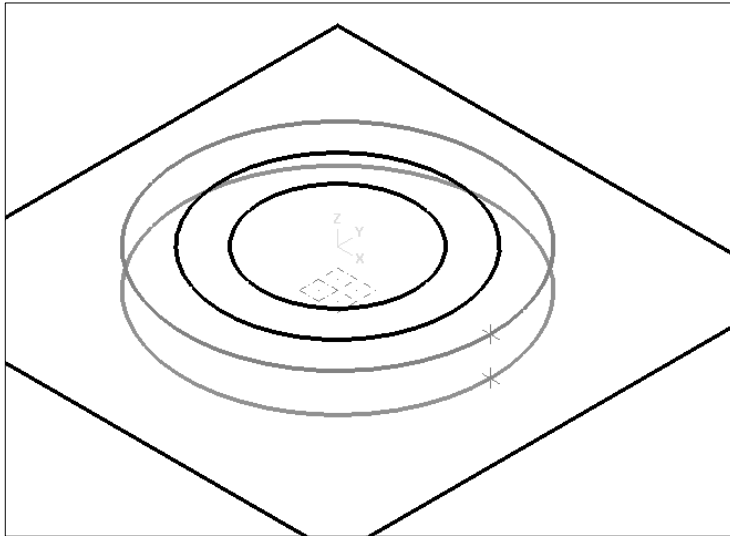



## Design

### Xform

#### Translate

- Change the parameter to create another circle at -1.0 " depth.
- Press enter once you type the delta Z value, and the preview of the translation should look as shown below;



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

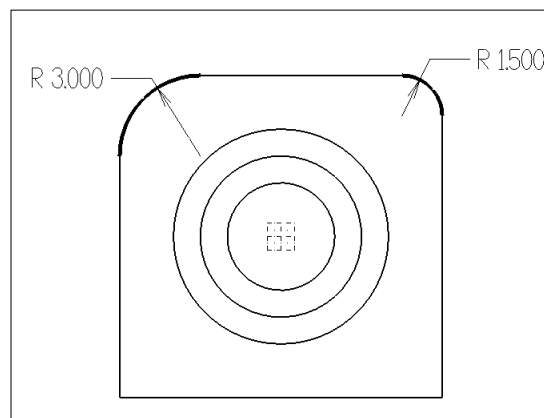
- Change the graphic view back to **Top**. 

- Select the **Clear color** button to remove the transform operation colors (red color for the original geometry and magenta color for the resulting geometry).




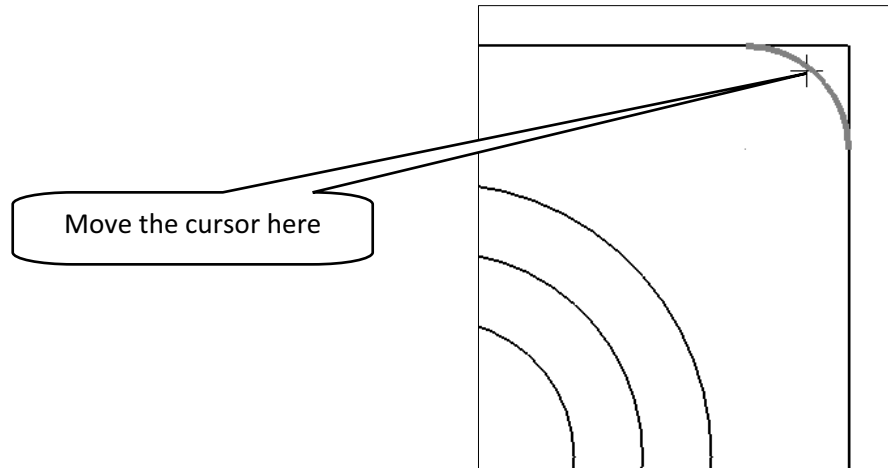
## STEP 4: FILLET THE CORNERS OF THE RECTANGLE



Step Preview:

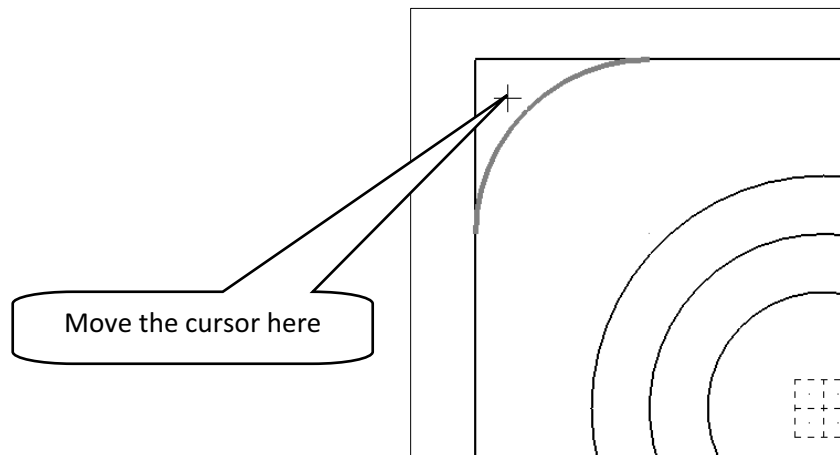





- ➔ From the **Sketcher** toolbar, select **Fillet entities** icon.
- ➔ Enter the **Radius**  1.5
- ➔ [Select an entity]: Move the cursor until you see a preview of the future fillet as shown, and then click to create it.



- ➔ Select the **Apply** button to continue using the same command. 
- ➔ Enter the **Radius**  3.0
- ➔ [Select an entity]: Move the cursor until you see a preview of the future fillet as shown, and then click to create it.

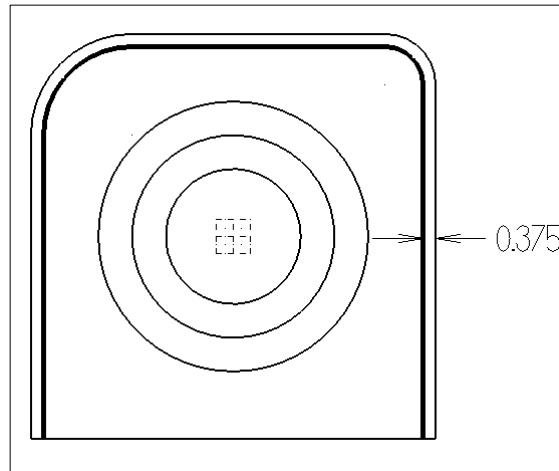


- ➔ Select the **OK** button to exit the command. 

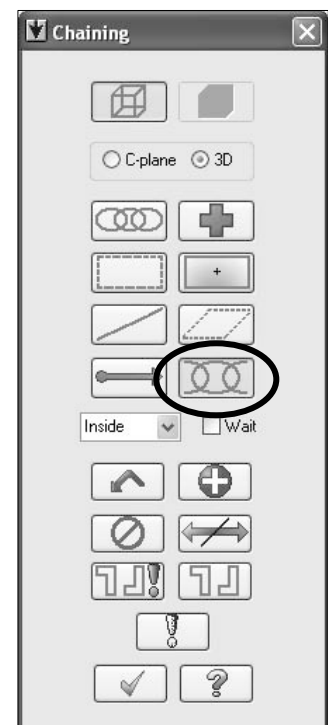
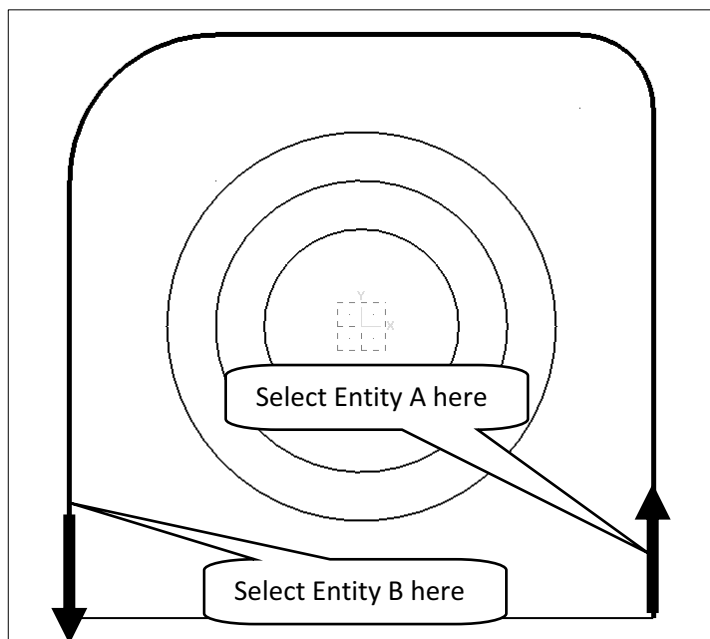
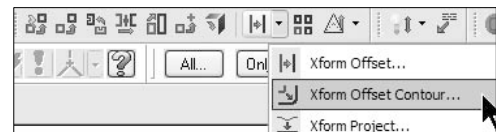
**Design**


**STEP 5: OFFSET PART OF THE RECTANGLE INSIDE WITH A 0.375 OFFSET DISTANCE**

*Step Preview:*



- From the **Xform** toolbar, select **Xform Offset Contour**.
- Enable **Partial** button in the **Chaining** dialog box.
- [Select the first entity]: Select Entity A as shown below.




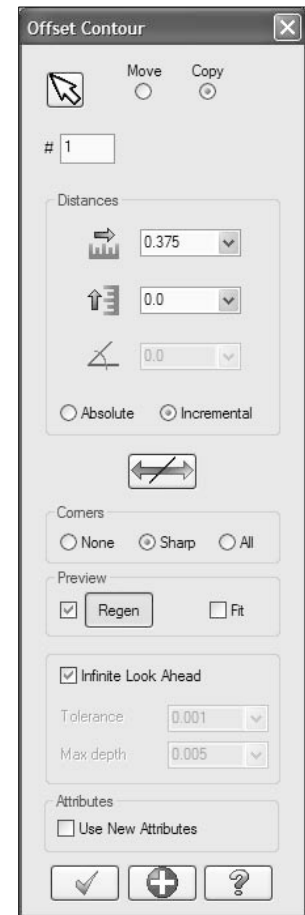
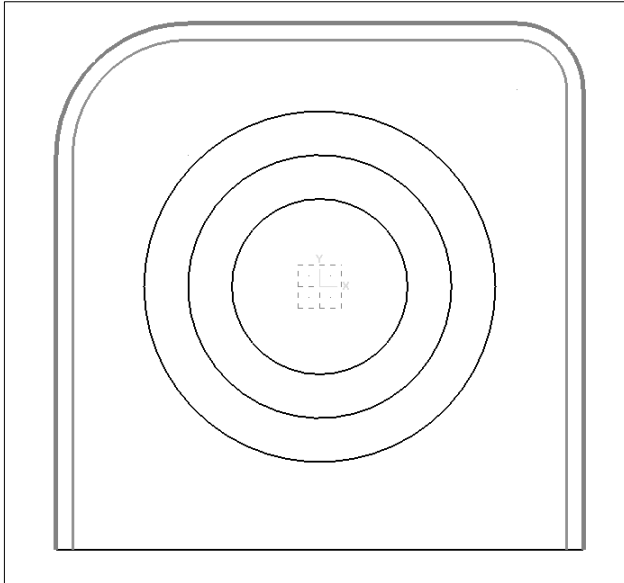
- [Select the last entity]: Select Entity B.
- Select the **OK** button to exit the **Chaining** dialog box. 



**Design**

➡ Modify the **Offset contour** dialog box to match the screenshot to the right.

➡ Make sure that the contour is created inside of the original chain as shown. Otherwise, select **Reverse** button. 



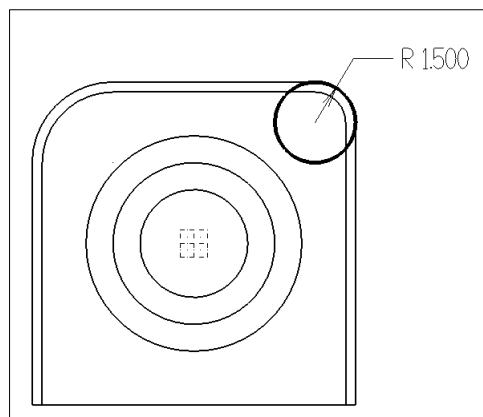
➡ Select the **OK** button to exit the **Offset Contour** dialog box. 

➡ Select **clear color** button to remove the transform operation colors.




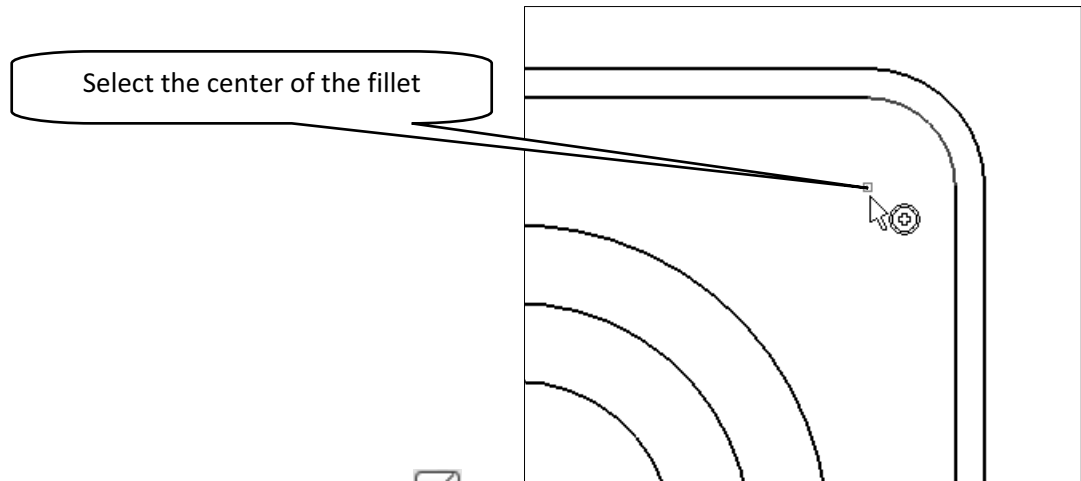
**STEP 6: CREATE THE CIRCLE WITH THE RADIUS 1.5 THAT HAS THE CENTER POINT OF THE FILLET WITH THE 1.5 INCH RADIUS**


*Step Preview:*



**Design**

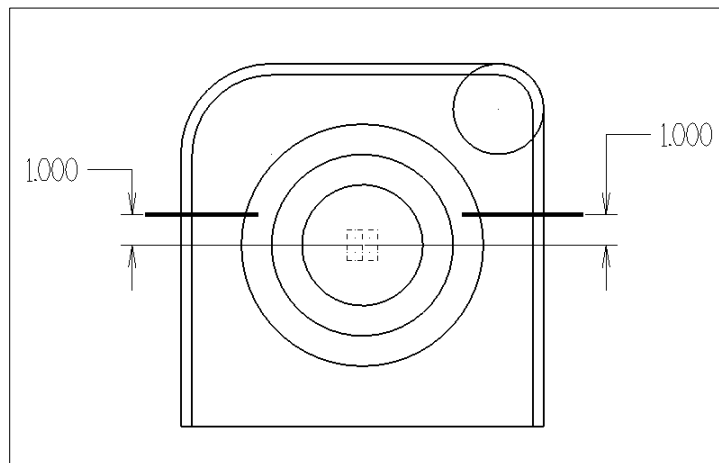
- From the **Sketcher** toolbar, select **Create Circle Center Point** icon.
- Enter the **Radius** value  1.5 (Enter)
- [Enter the center point]: Select the center of the fillet as shown;




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

**STEP 7: CREATE TWO HORIZONTAL LINES FOR THE POCKET**

*Step Preview:*



- From the **Sketcher** toolbar, select **Create Line Endpoint** icon.
- Enable the **Horizontal** icon in the **Ribbon Bar**. 



**Design**

- [Specify the first endpoint]: Sketch Point A approximately as shown.
- [Specify the second endpoint]: Sketch Point B.

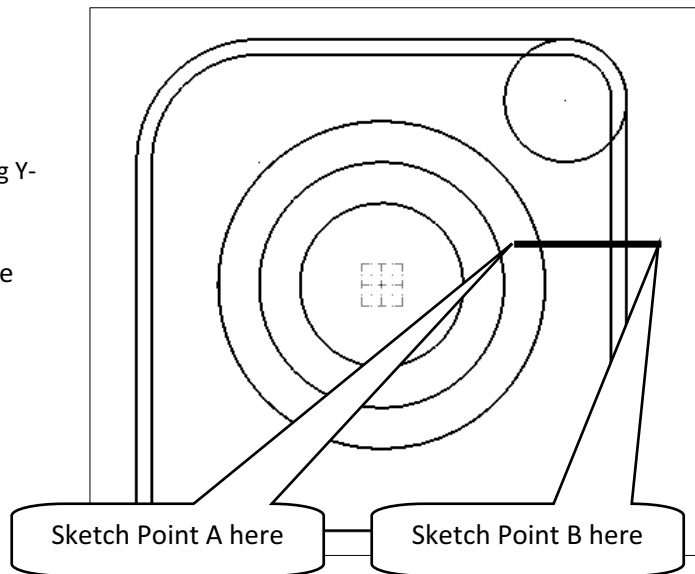
- [Enter the Y coordinate]: type 1.0



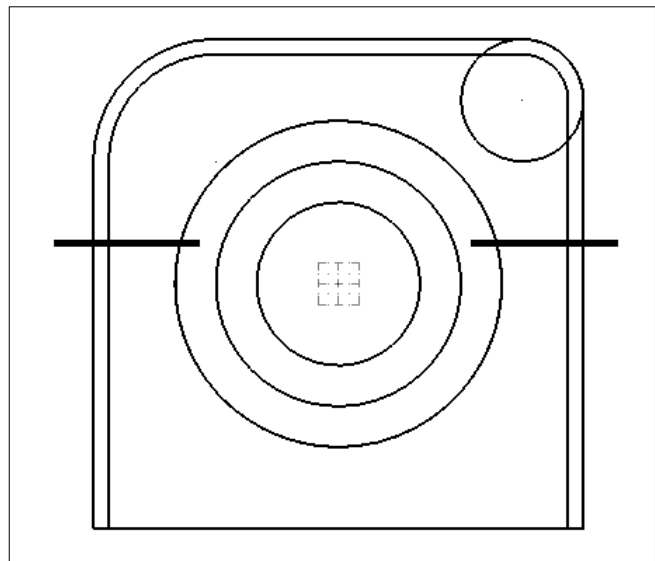
Press Enter


- Note the line will be moved 1.0" along Y-axis, measured from the origin.

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



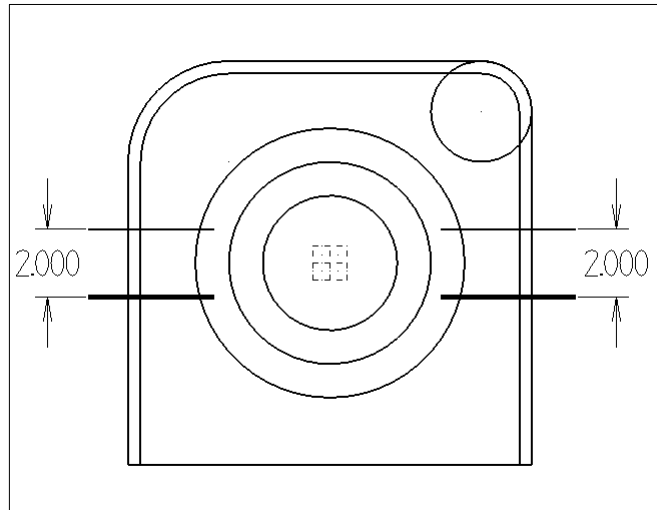
- Sketch a horizontal line in the same way as we did before; on the left side as shown below. Make sure that you enter the Y value 1.0



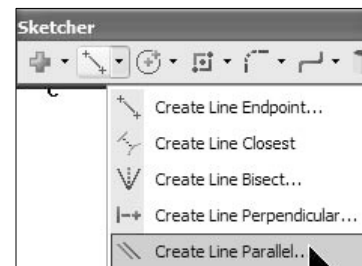
- Select the **OK** button to exit the command. 




## STEP 8: CREATE TWO LINES PARALLEL WITH THE HORIZONTAL LINES

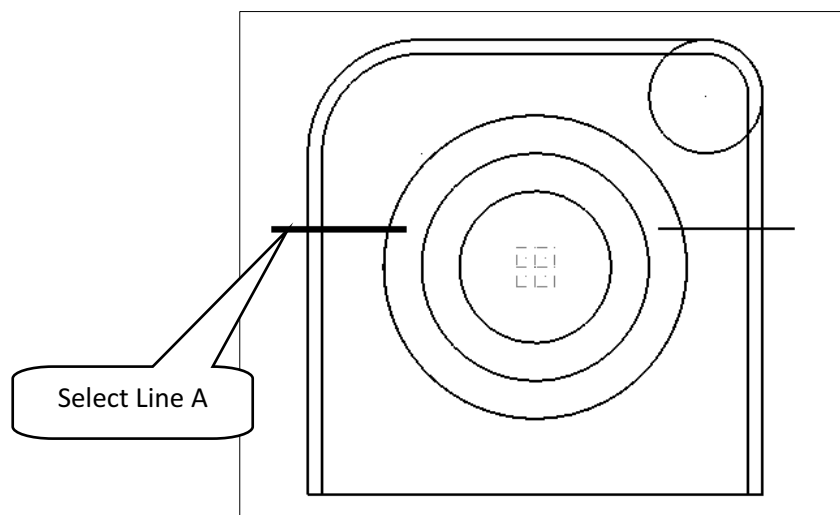
*Step Preview:*



- ➔ From the **Sketcher** toolbar, select the **drop-down arrow** next to **Create Line Endpoint** and select **Create Line Parallel** from the drop-down list

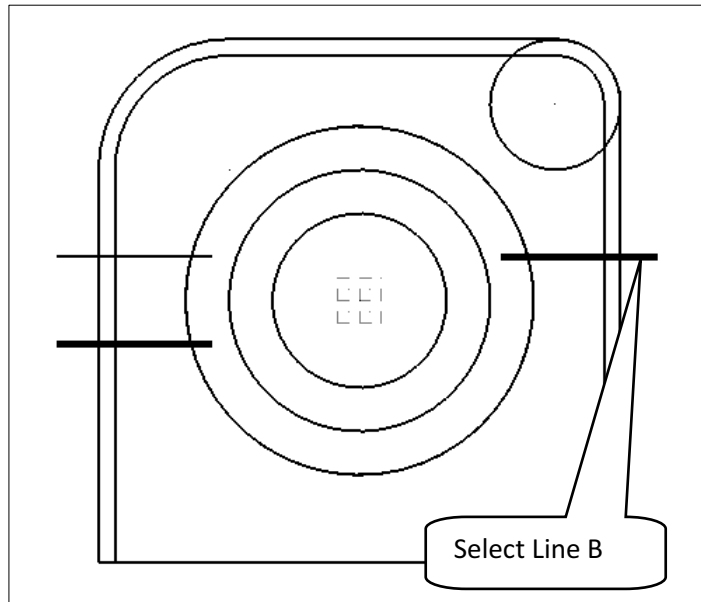



- ➔ Enter the **Distance**  2.0
- ➔ Click on the distance icon  to lock the value as shown  2.0
- ➔ [Select a line]: Select Line A
- ➔ [Indicate the offset direction]: Select a point below Line A



**Design**

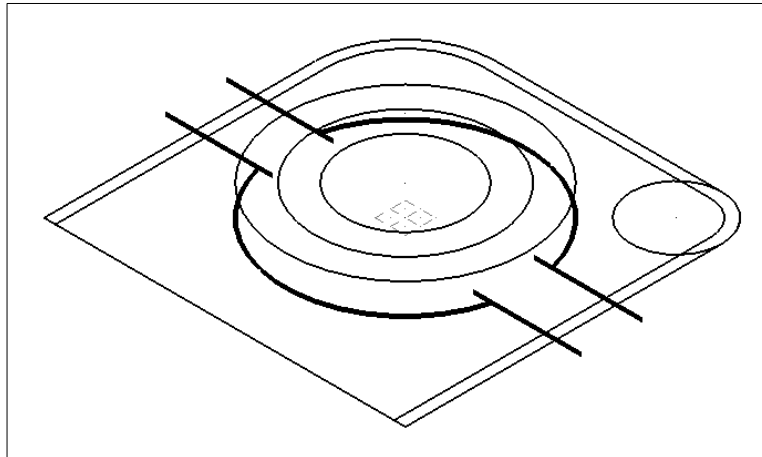
- [Select a line]: Select Line B
- [Indicate the offset direction]: Select a point below Line B



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

**STEP 9: TRIM - DIVIDE THE 8.0" CIRCLE THAT WAS CREATED AT Z -1.0**

*Step Preview:*



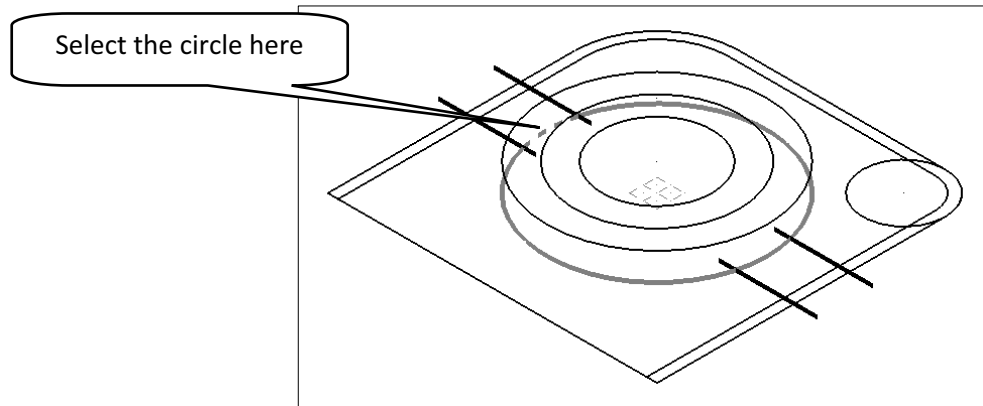
- From the **Trim/Break** toolbar, select **Trim/Break/Extend** icon. 

- Change the graphic view to **Isometric**. 

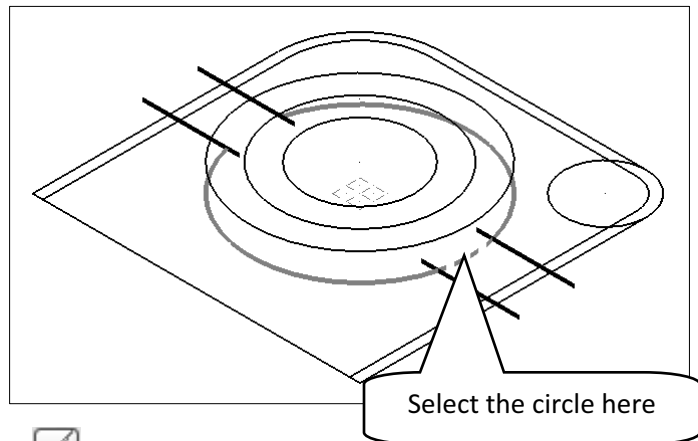
## Design


- ➔ Select **Divide** icon. 


- ➔ [Select the curve to divide/delete]: Select the 8.0" diameter circle as shown;



- ➔ [Select the curve to divide] Select the 8.0" diameter circle as shown;

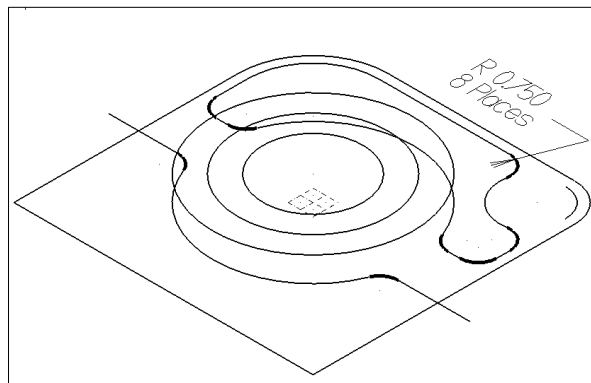


- ➔ Select the **OK** button to exit the command. 

- ➔ Change the graphic view back to **Top**. 

## STEP 10: FILLET THE CORNERS OF THE POCKETS

*Step Preview:*



### Design

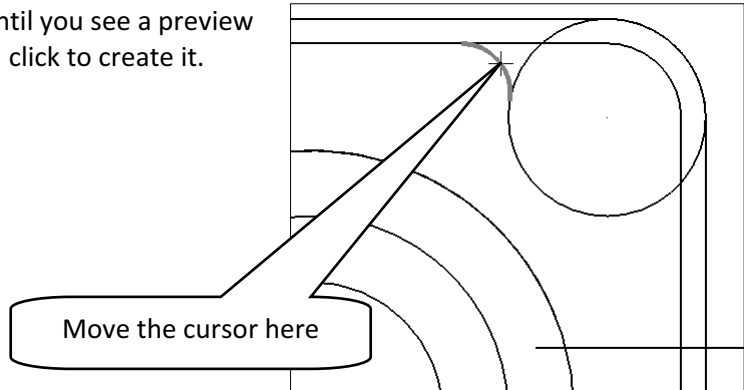
- From the **Sketcher** toolbar, select **Fillet entities** icon.



- Enter the **Radius**  0.75 (Press Enter)

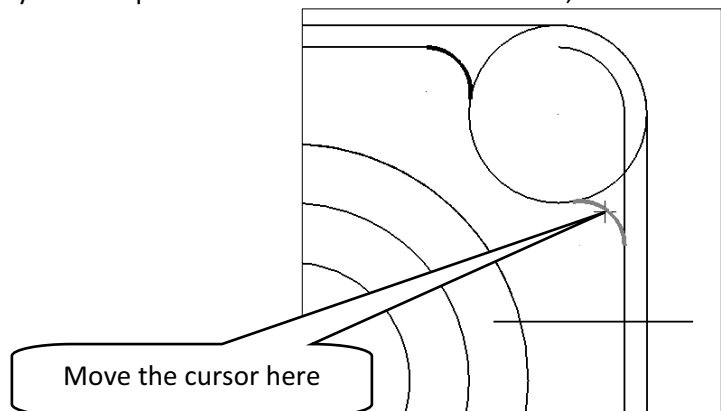
- Make sure that the **Trim** icon is enabled. 

- [Select an entity]: Move the cursor until you see a preview of the future fillet as shown, and then click to create it.

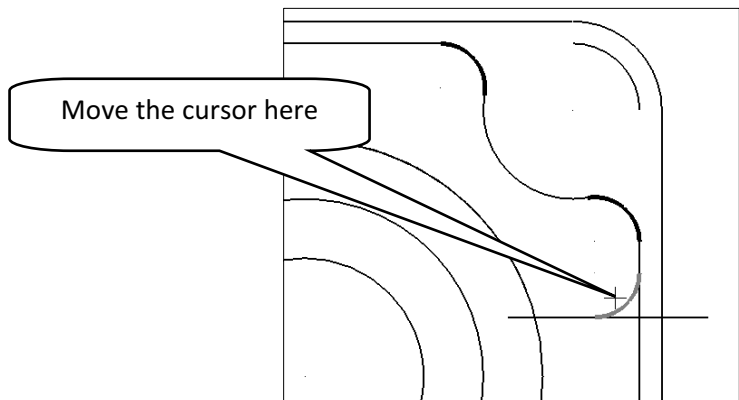


- Note: if the fillet does not appear, select the line and circle between which you are creating the fillet.

- [Select an entity]: Move the cursor until you see a preview of the future fillet as shown, and then click to create it.



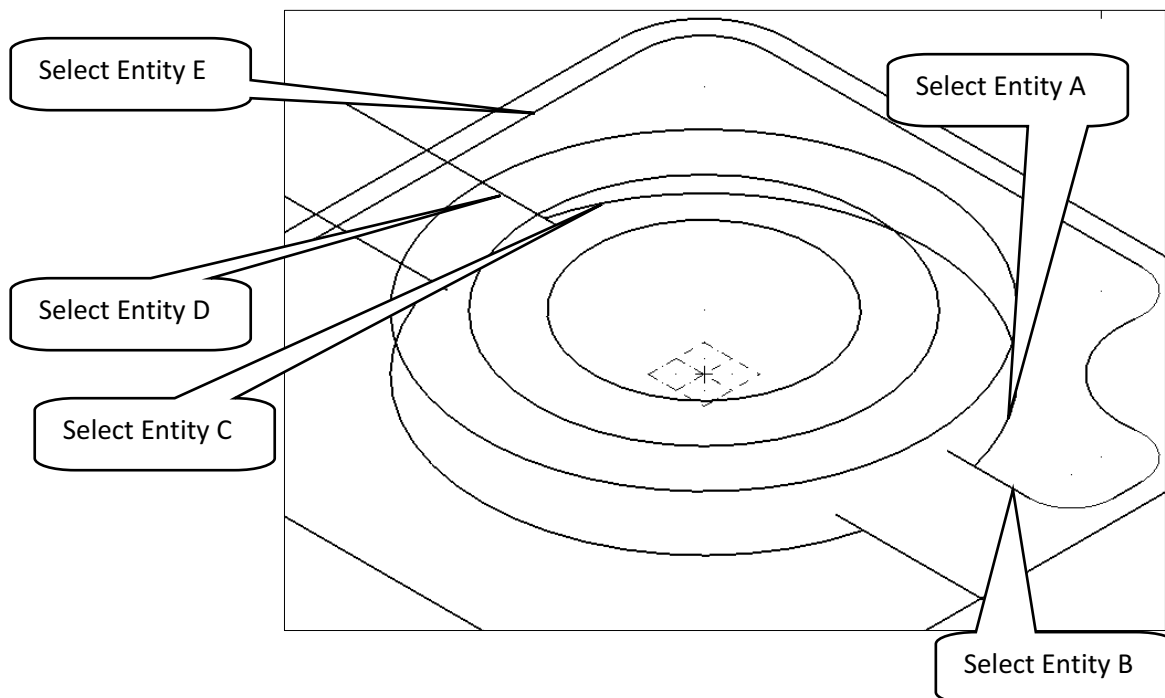
- [Select an entity]: Move the cursor until you see a preview of the future fillet as shown, and then click to create it.





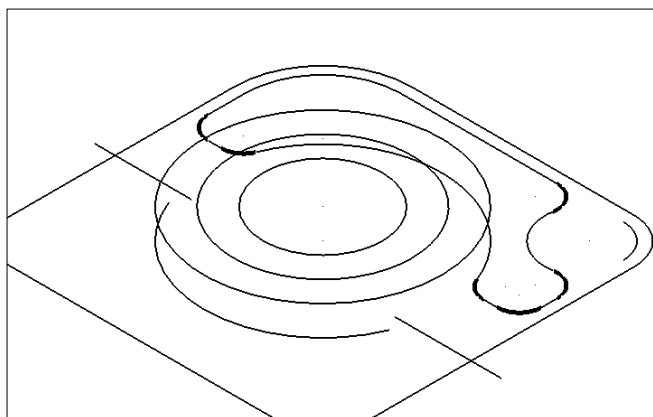
**Design**

- Change the graphic view to **Isometric**.
- [Select an entity]: Select Entity A
- [Select another entity]: Select Entity B



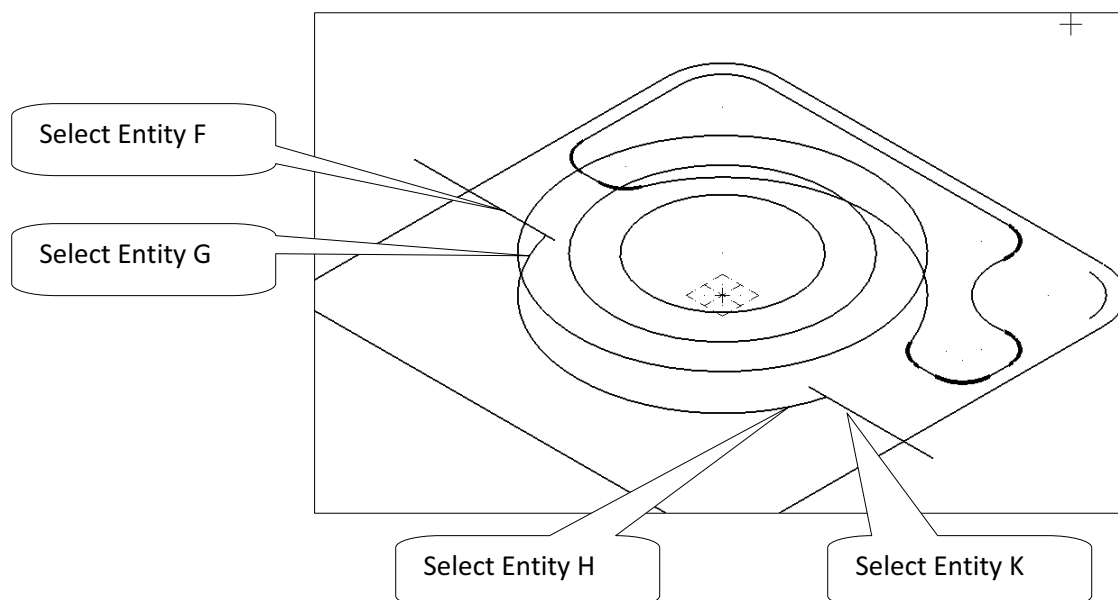
- [Select an entity]: Select Entity C
- [Select another entity]: Select Entity D
- [Select an entity]: Select Entity D
- [Select another entity]: Select Entity E


*The geometry should look as shown.*



**Design**

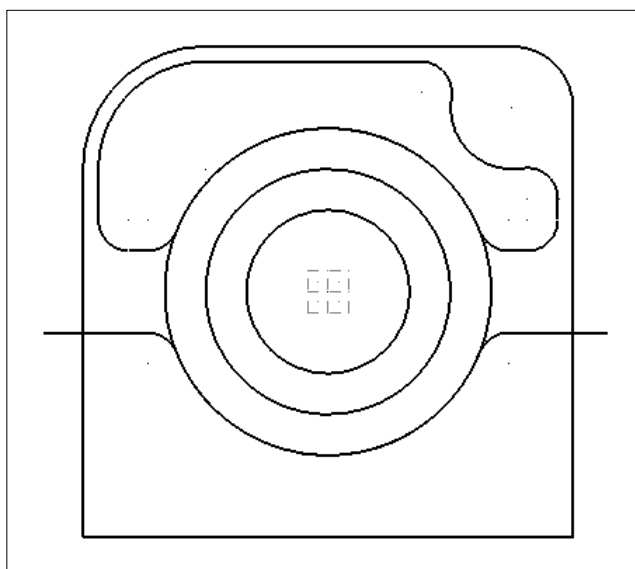
- [Select an entity]: Select Entity F
- [Select another entity]: Select Entity G
  
- [Select an entity]: Select Entity H
- [Select another entity]: Select Entity K




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

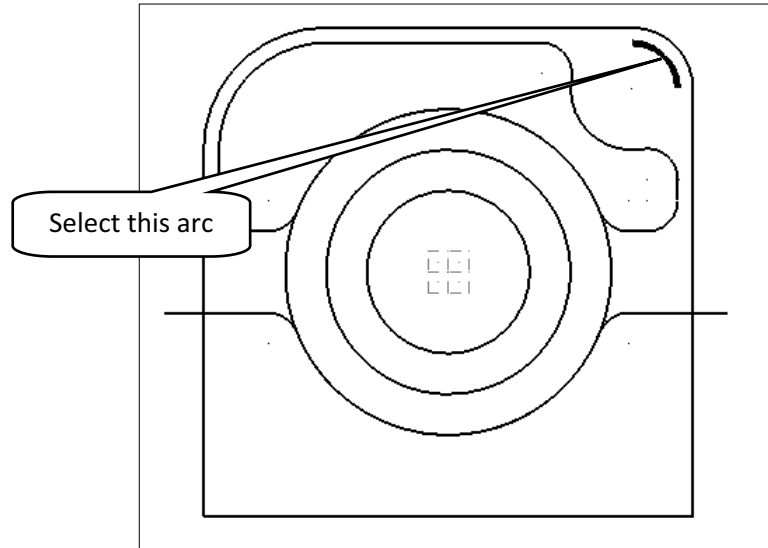
**STEP 11: DELETE THE EXTRA ARC**



*Step Preview:*



## Design

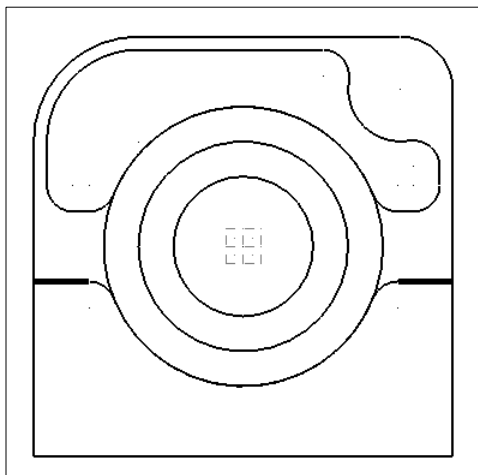
- Change the graphic view back to **Top**. 
- Select the arc as shown in the following picture.





- Select the **Delete Entity**  icon.
- Use **Repaint** toolbar icon to repaint the drawing. 

## STEP 12: TRIM THE TWO HORIZONTAL LINES

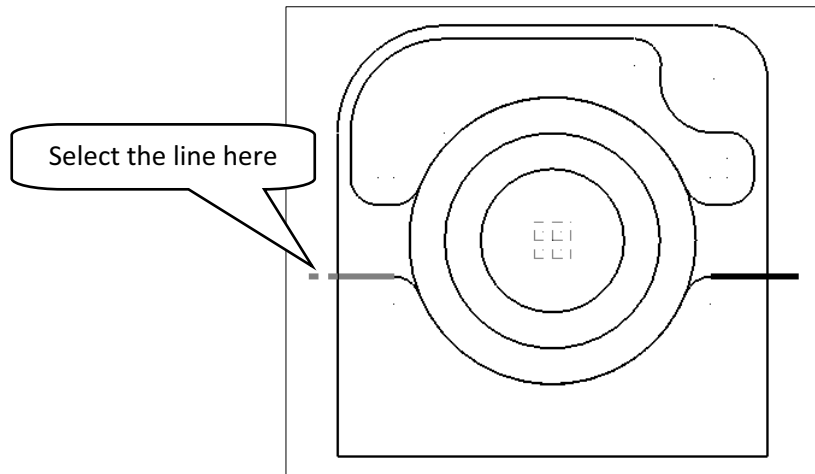
*Step Preview:*



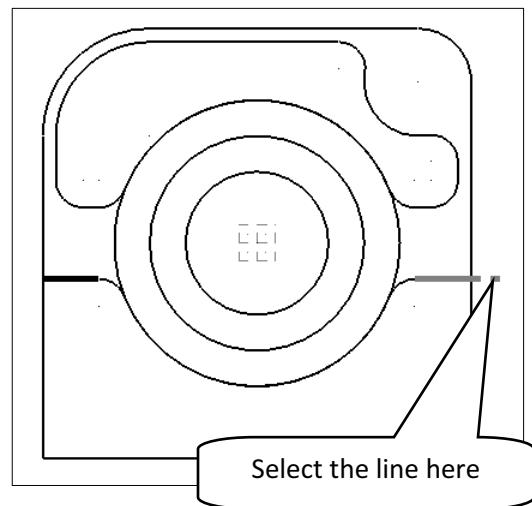
- From the **Trim/Break** toolbar, select **Trim/Break/Extend** icon. 
- Select **Divide** icon. 


**Design**

- [Select the curve to divide] Select the line at the location shown;



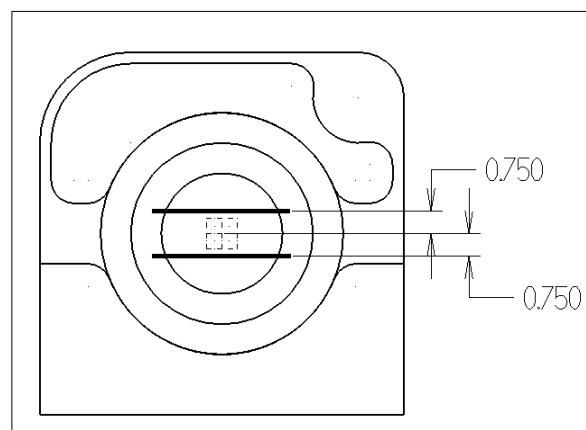
- [Select the curve to divide] Select the 8.0" diameter circle as shown;



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

**STEP 13: CREATE THE TWO HORIZONTAL LINES FOR THE 1.5" WIDTH SLOT**

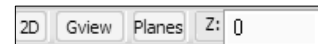
*Step Preview:*



## Design

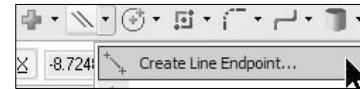
### 13.1 Change the Z – depth


- ➔ Change the Z depth of the construction plane to Z0

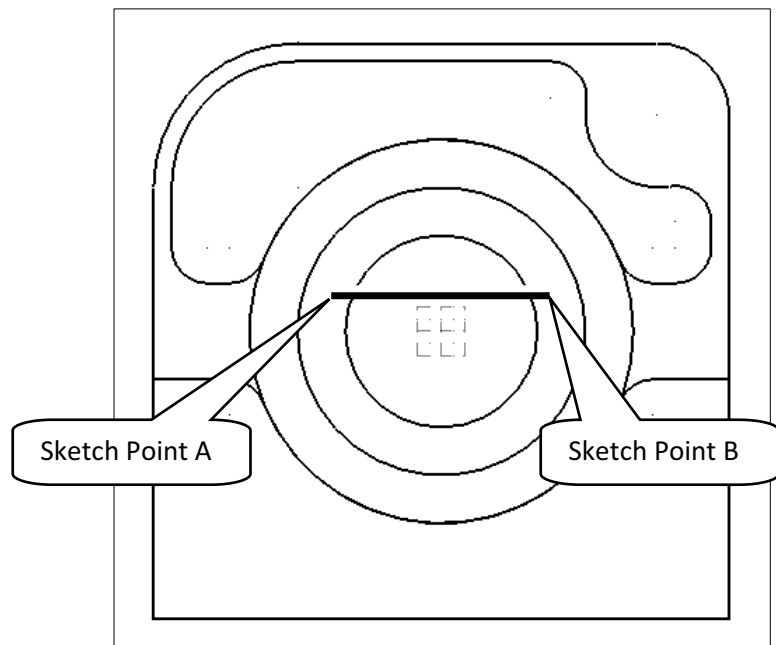



### 13.2 Create the two horizontal lines

- ➔ From the **Sketcher** toolbar, select **CreateLine Endpoint** icon



- ➔ Enable the **Horizontal** icon in the **Ribbon Bar** 
- ➔ [Specify the first endpoint]: Sketch Point A approximately as shown;
- ➔ [Specify the second endpoint]: Sketch Point B



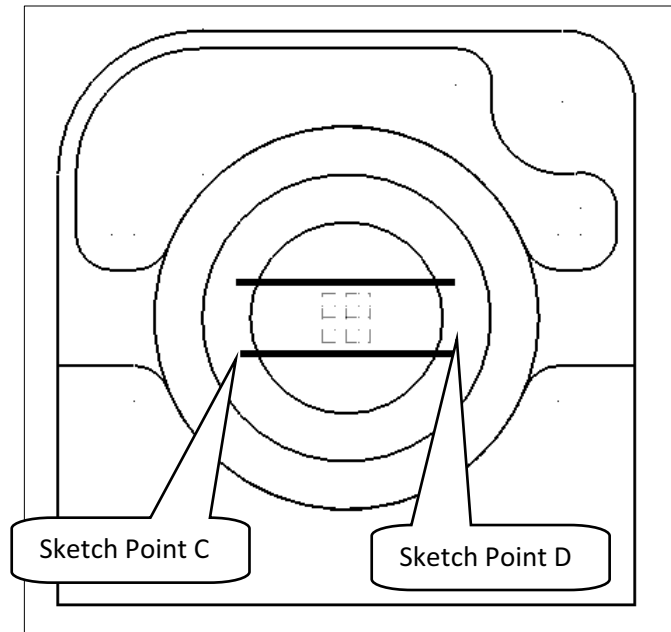
- ➔ [Enter the Y coordinate]: type 0.75  Press **Enter**

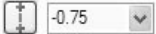
➔ Note that the line will be moved 0.75" along Y-axis, measured from the origin.

- ➔ Select the **Apply** button to continue in the same command. 


**Design**

- [Specify the first endpoint]: Sketch Point C approximately as shown;
- [Specify the second endpoint]: Sketch Point D



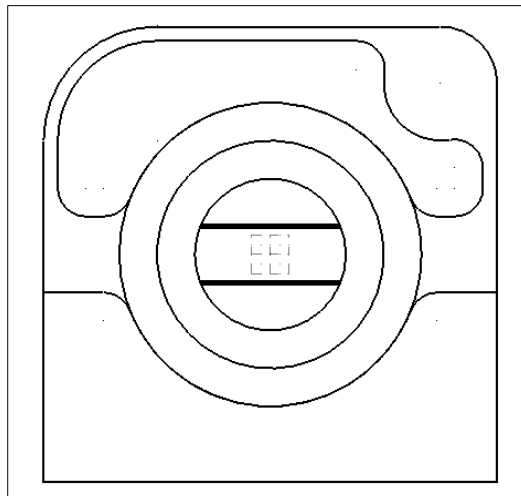
- [Enter the Y coordinate]: type -0.75  Press **Enter**

➤ Note that the line will be moved -0.75 " along Y-axis, measured from the origin.

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

**STEP 14: TRIM – DIVIDE THE TWO HORIZONTAL LINES**

*Step Preview:*



**Design**

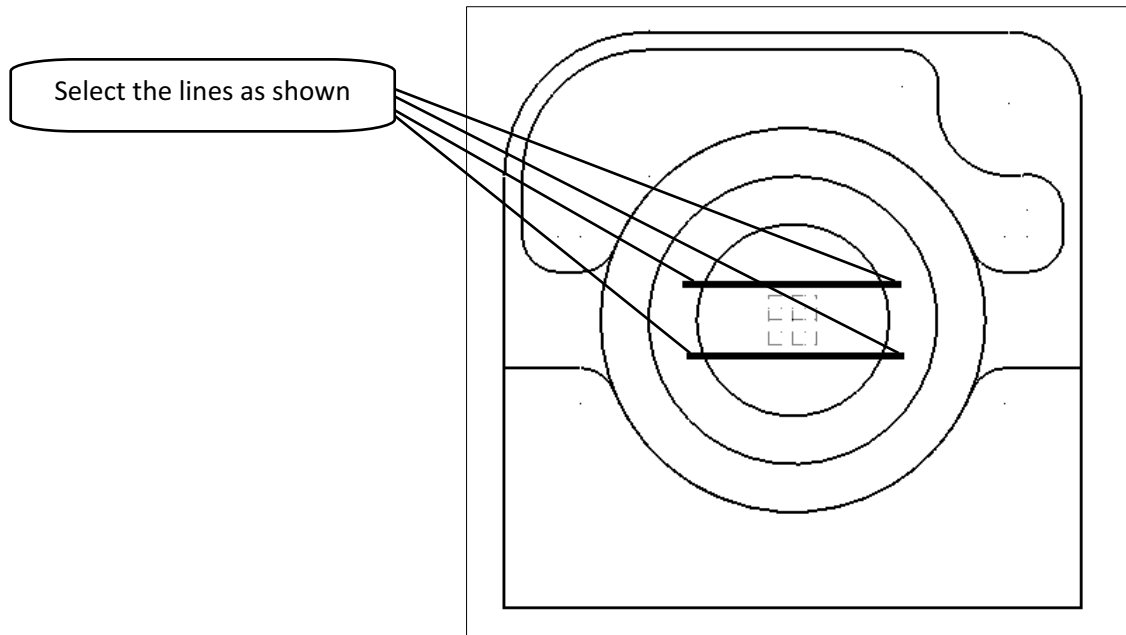
- From the **Trim/Break** toolbar, select **Trim/Break/Extend** icon.



- Select **Divide** icon.



- [Select the curve to divide] Select the lines on the side that you want to remove as shown;



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

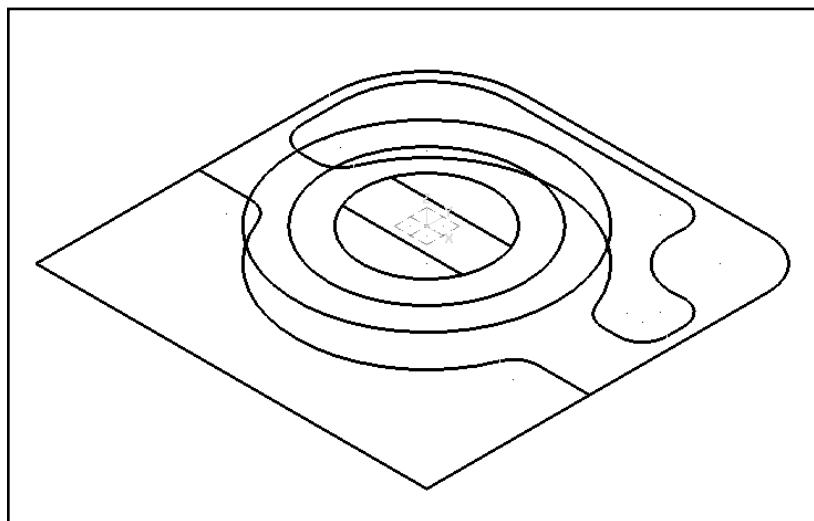


- Change the graphic view to **Isometric**.



*The geometry should look as shown.*

- Note that for toolpath creation purpose we don't need the solid geometry. Steps 16 to 25 are optional.





## Design


### OPTIONAL - CREATE THE SOLID

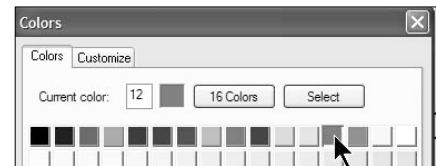
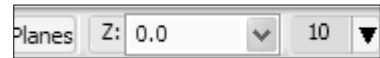
A **solid** is a geometric entity that occupies a region of space and consists of one or more faces, which define the closed boundary of the solid.

### STEP 15: CHANGE THE CURRENT COLOR TO RED (NUMBER 12)

You can set the current color in the **Status bar**. There are 256 color predefined. You can also use the Customize tab to create a custom color, based on one of the 256 standard colors.

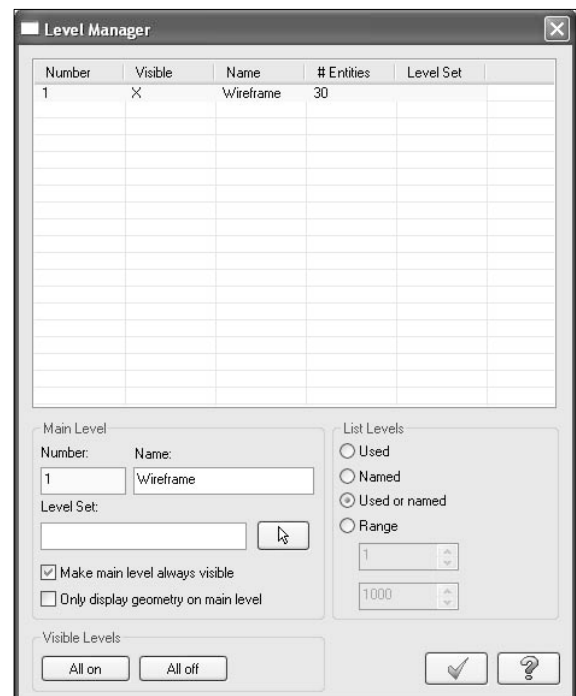
To change the color of an existing entity, right-click on the color in the **Status bar**, select the entities that you want to change and press Enter to finish the selection, and then, pick the desired color.

- ➔ Select **Color** from the **Status Bar**
- ➔ Select the color to **red (number 12)**.
- ➔ Select the **OK** button to exit. 



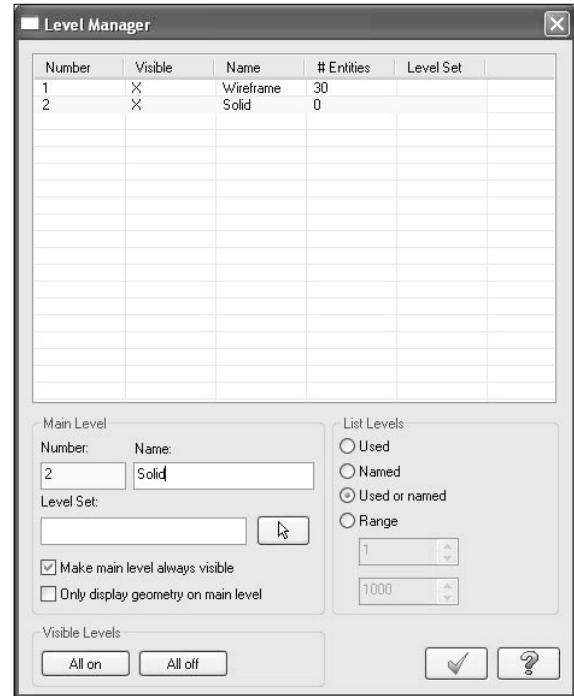
### STEP 16: CHANGE THE CURRENT LEVEL TO 2


- ➔ Select **Level** from the **Status Bar**
- ➔ Name **Level 1 Wireframe** as shown



## Design

- ➔ Change the **Main Level** number to **2** and name it **Solid** as shown.



- ➔ Note that the current level in which we will create the solid will be **Level 2**.
- ➔ To move existing entities on a different level; right-click on the level in the **Status Bar**, select the entities that you want to move and press enter, and then, enter the level number where you want to move them.
- ➔ Select the **OK** button to exit **Level Manager**. 

## STEP 17: SET THE SOLIDS TOOLBAR

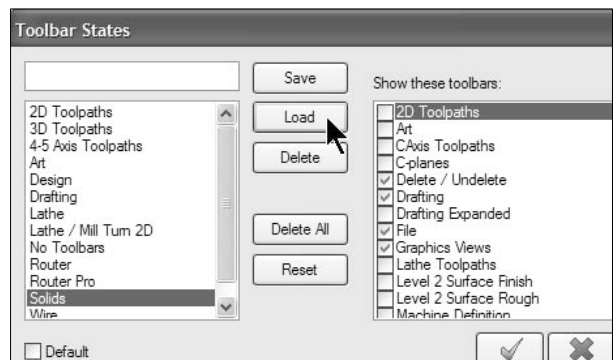
- ➔ To display the **Operations Manager** press **Alt + O**.

### Settings

#### ➔ Toolbar States

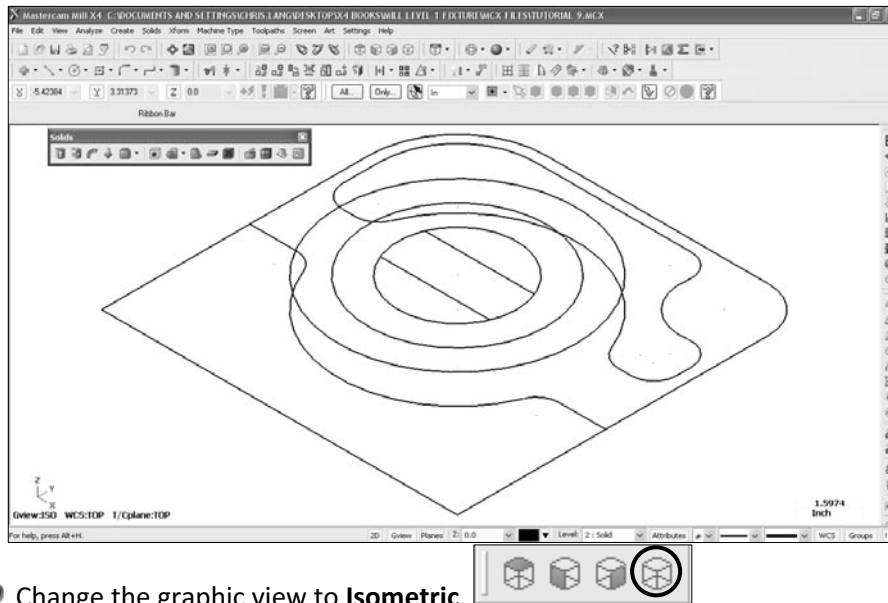
- ➔ Select **Solids** and **Load**.

- ➔ You should see the solids toolbar located at the left of the Operations Manager.



## Design

- ➡ Select the toolbar as shown (the cursor should be on the dividing top bar when you select it).
- ➡ Drop the toolbar in the graphic area.

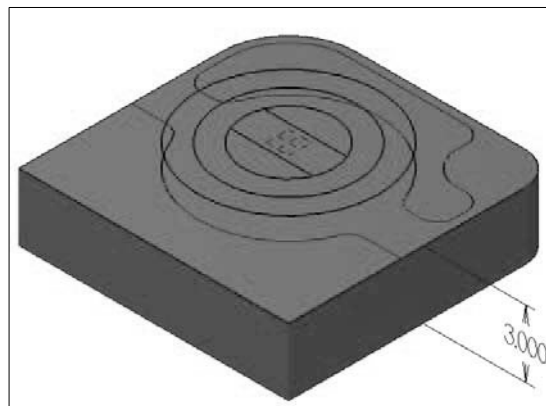


- ➡ Change the graphic view to **Isometric**.

## STEP 18: CREATES THE SOLID BODY BY EXTRUDING THE OUTSIDE PROFILE

- ➡ The **Extrude** function lets you extrude planar chains of curves to create one or more new solid bodies, cuts on an existing body, or bosses on an existing body. Mastercam extrudes chains of curves by driving the shape of the curves along a linear path using a specified direction and distance.

*Step Preview:*



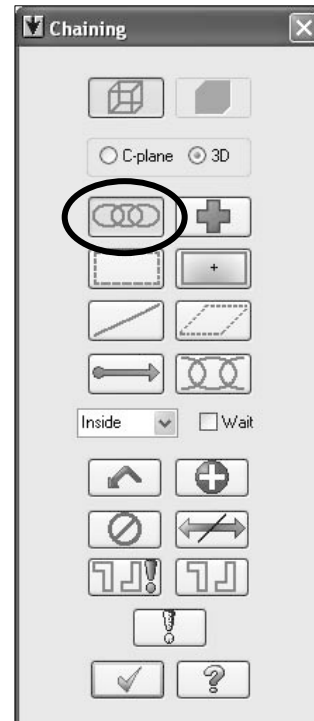
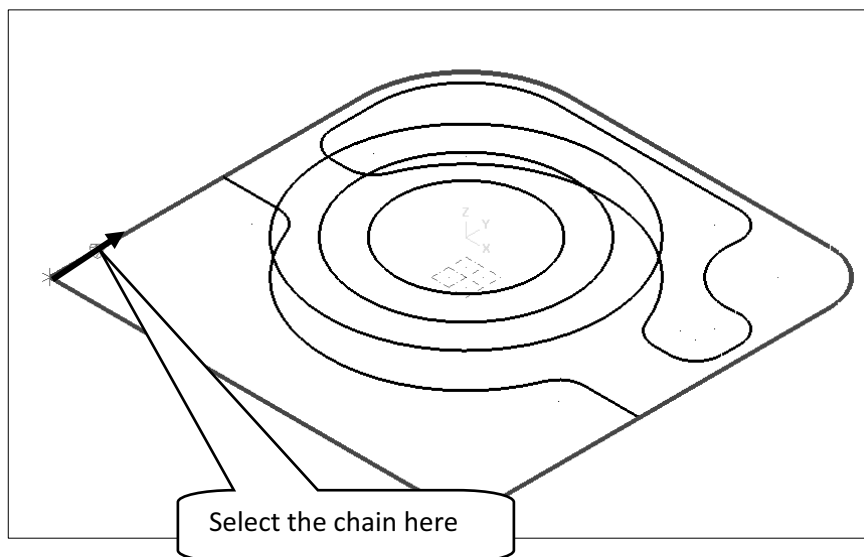
- ➡ Note that the construction plane doesn't affect the way in which the solid is generated.

- ➡ From **Solids** toolbar, select **Solids Extrude** icon.



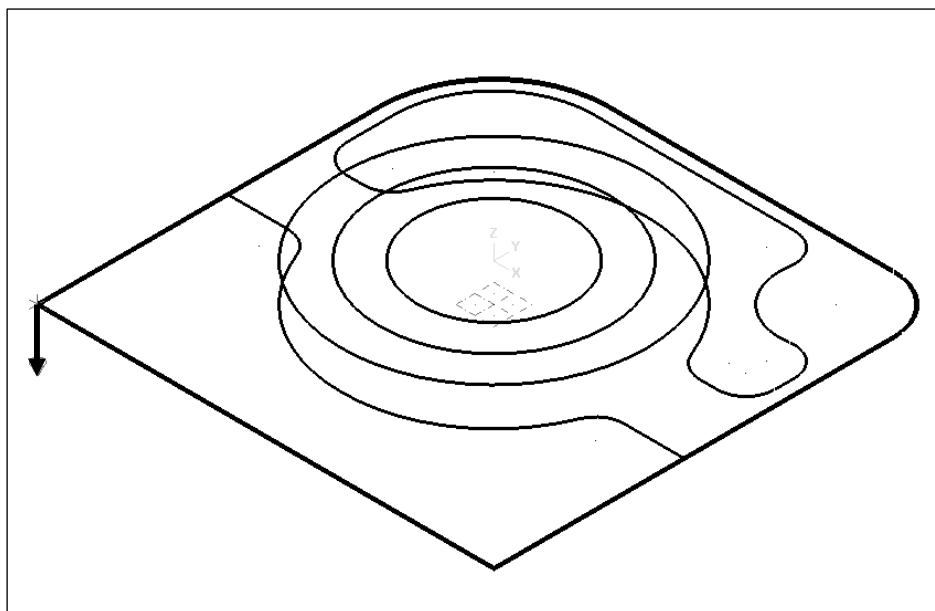
**Design**

- Select the circle in the **CW** direction.



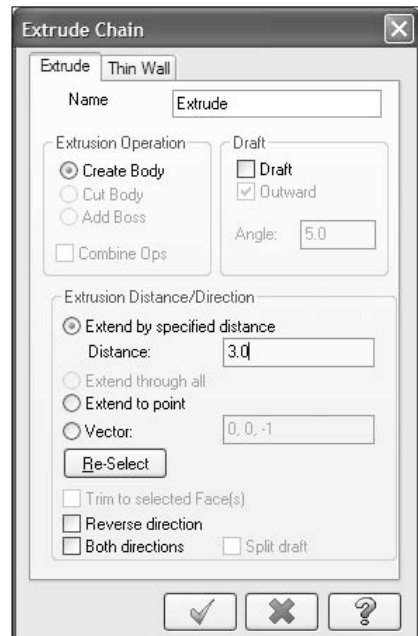
- Select the **OK** button to exit the **Chaining** dialog box.

- Make sure that the arrow is pointed inwards as shown in the following picture. If not, enable **Reverse Direction** in the **Extrude Chain** dialog box



## Design

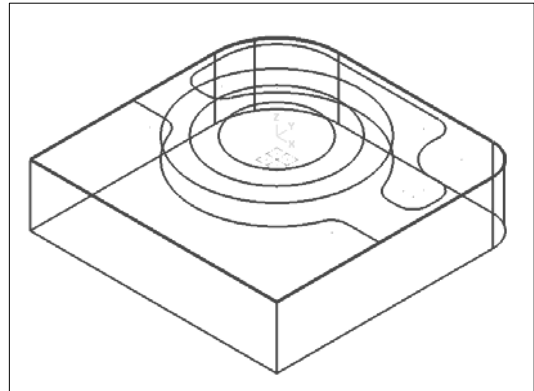
- ➡ Modify the **Extrude** dialog box as shown to create the solid body.



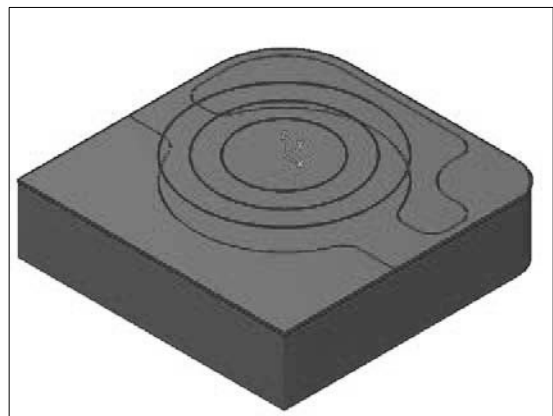
- ➡ Select the **OK** button.



*The drawing should look as shown in the following pictures.*



- ➡ You can press **Alt + S** simultaneous, at any time, to see the part shaded.



- ➡ Press **Alt + S** keys again to return to the wireframe display.

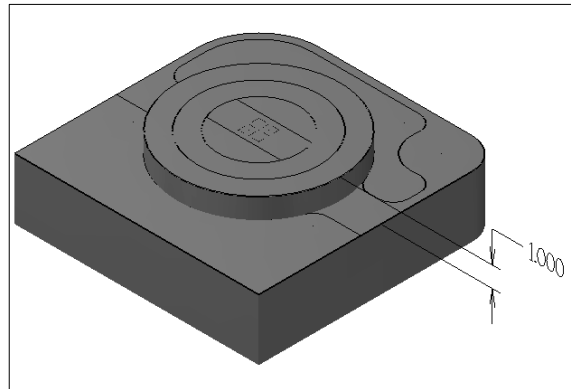
## Design

- Press **Alt + O** to see the **Operations Manager** if needed.
- Note that you can change the solid parameters at any point, by selecting the **Solids** tab in the **Operations Manager** and expand the solid tree. Select the **Parameters** and modify the settings as needed and then select the **Regen all** button.

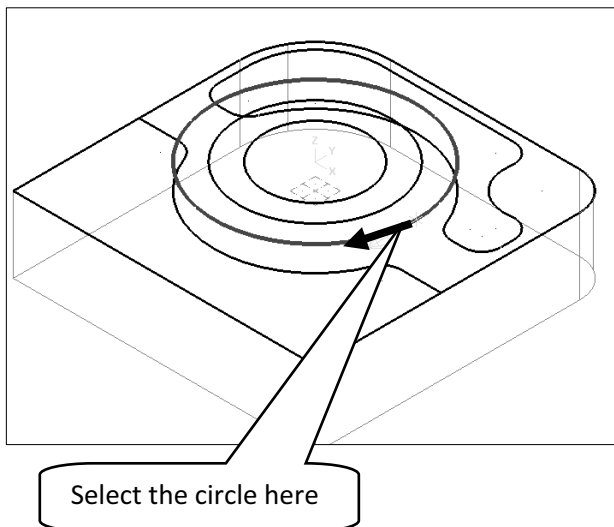


## STEP 19: CREATE THE BOSS WITH THE HEIGHT OF 1.0

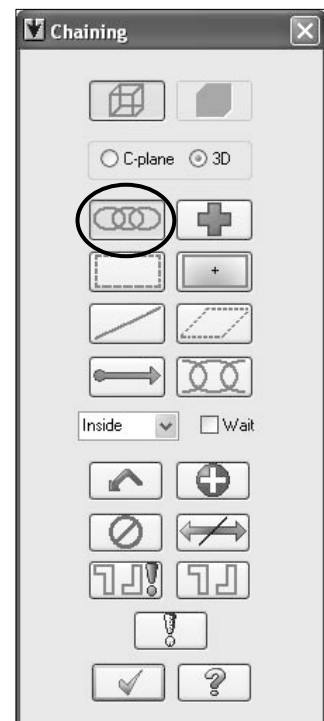
*Step Preview:*



- From **Solids** toolbar, select **Solids Extrude** icon.
- Select the 8.0 "diameter circle in the **CW** direction.

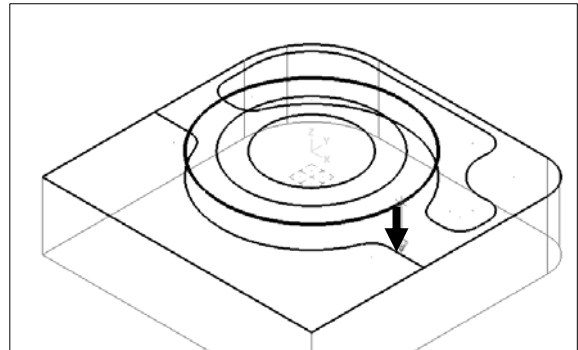


- Select the **OK** button to exit the **Chaining** dialog box.

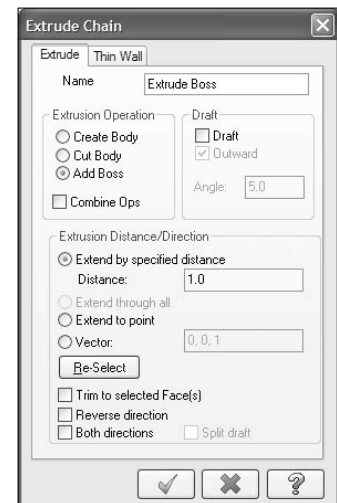


## Design

- Make sure that the arrow is pointed downwards as shown in the following picture. If not, enable **Reverse Direction** in the **Extrude Chain** dialog box



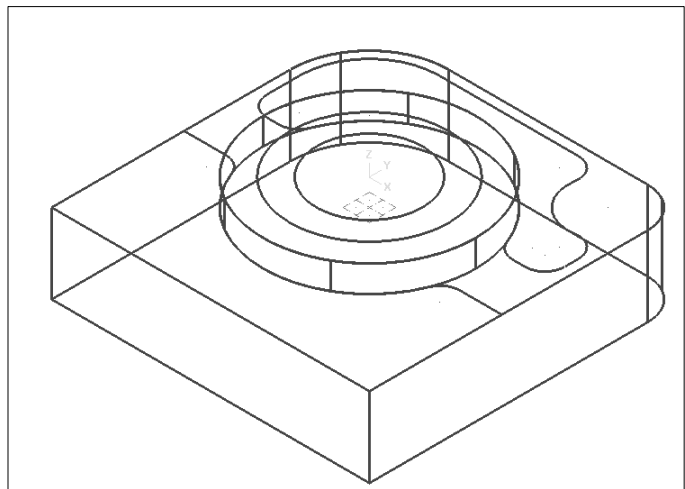
- Modify the **Extrude** dialog box to add a boss to the existing solid.



- Select the **OK** button.



The drawing should look as shown in the following picture.

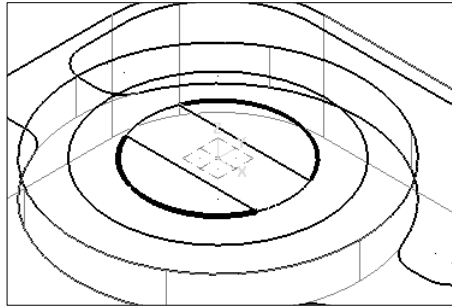


- You can press **Alt + S** simultaneously, at any time, to see the part shaded.
- Press **Alt + S** keys again to return to the wireframe display.

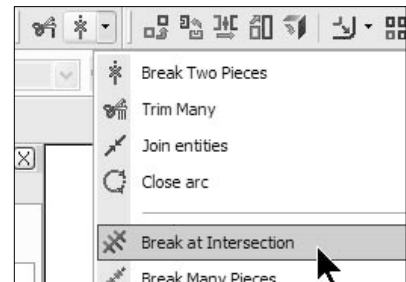
## STEP 20: BREAK AT INTERSECTION

- **Extrude** command requires closed chains. We need to break the 4.0 " circle at the intersection with the slot lines to be able to chain the geometry for the slot cut.

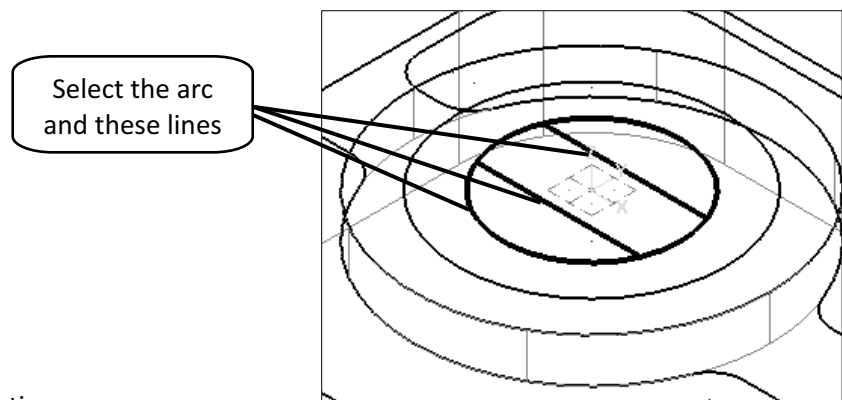
*Step Preview:*



- Click on the drop down arrow and select **Break at Intersection**



- [Select entities to break]: Select the circle and the two lines as shown



- Press **Enter** to finish the selection.

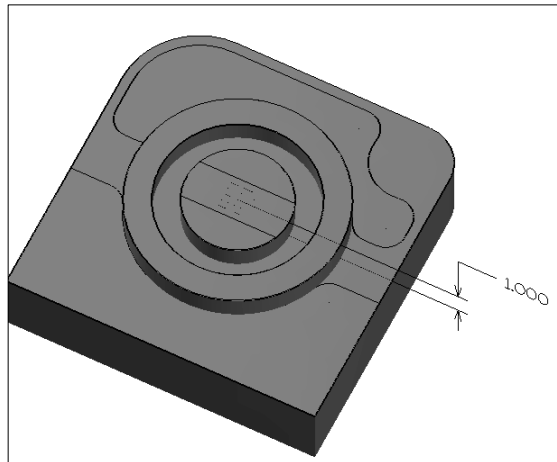
- Move the cursor along the circle that we broke. The system will highlight the different arcs. Note that the circles are always broken at 0 degree (3 o'clock).



**Design**

**STEP 21: EXTRUDE – CUT THE CIRCULAR CHANNEL**

*Step Preview:*

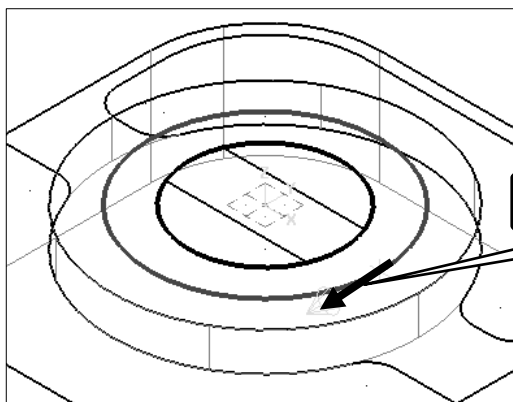


➡ Use the mouse wheel to zoom in/out as needed.

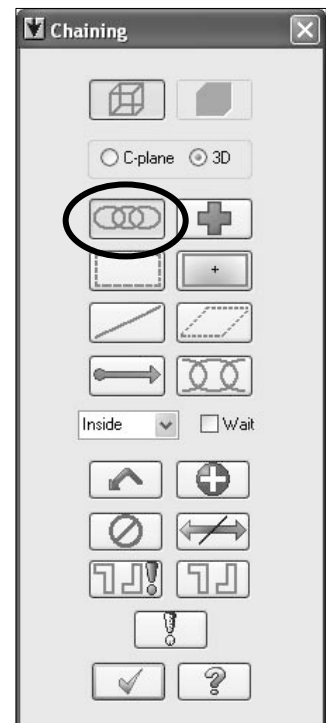
➡ From **Solids** toolbar, select **Solids Extrude** icon.



➡ Select the 6.0 " diameter circle in the **CW** direction.




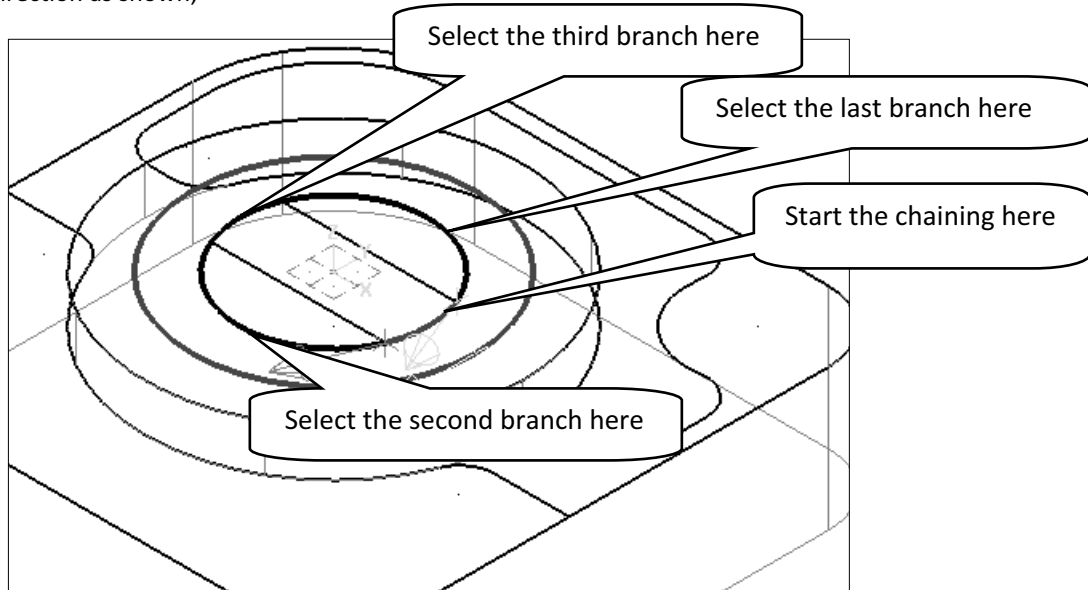
Select the circle here



**Design**

- Select the first arc with the 4.0 " diameter in the CW direction.

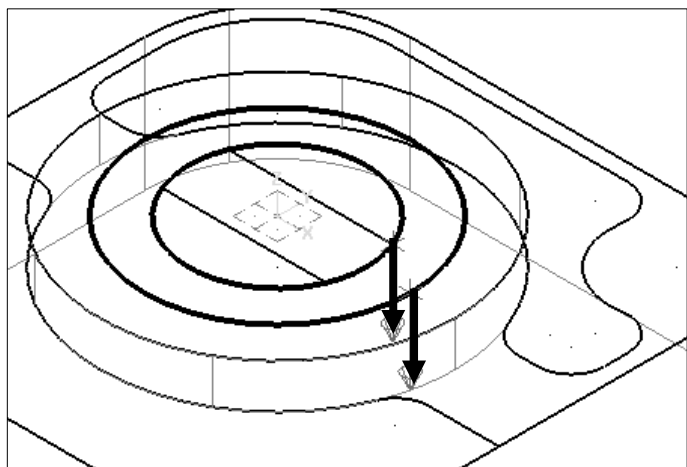
- Select **Reverse** button  in the Chaining dialog box if the arrows are not pointing in the CW direction as shown;



- Branches points are points in a chain where the endpoints of three or more entities meet. You will be prompted to select the branch along which the chaining should continue.
- Note that you will have to guide the red color arrow along the circle to make sure that you select the entire circle as one chain.

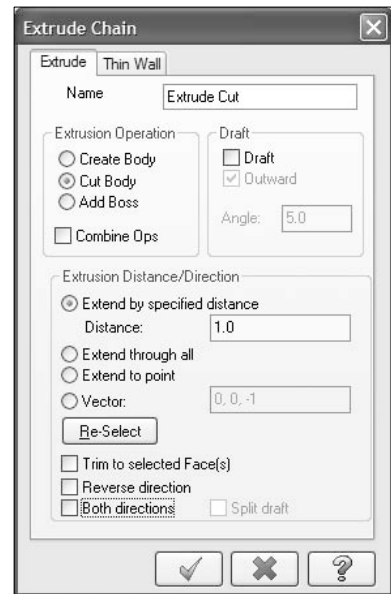
- Select the **End** button in the **Chaining** dialog box to finish the selection. 
- Select the **OK** button to exit the **Chaining** dialog box. 

- Make sure that all arrows are pointed in the same direction. (Downwards).



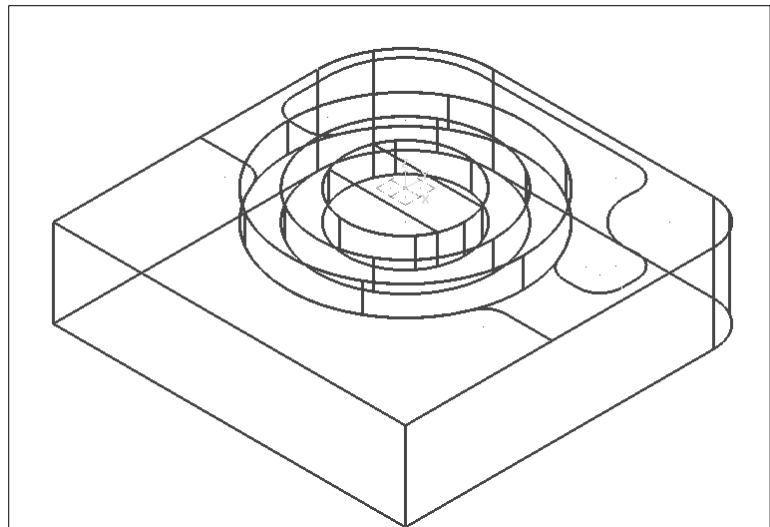
**Design**

- ➡ Modify the **Extrude** dialog box to cut the holes in one direction.



- ➡ Select the **OK** button. 

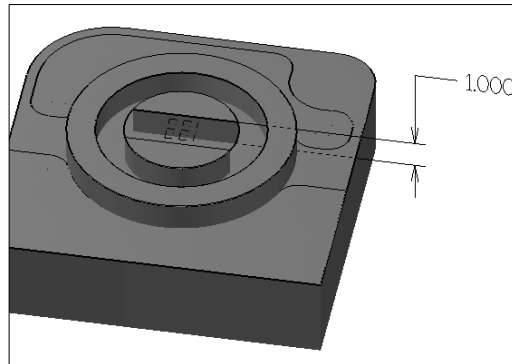
*The drawing should look as shown in the following picture.*



**Design**

**STEP 22: EXTRUDE - CUT THE 1.5" SLOT**


*Step Preview:*

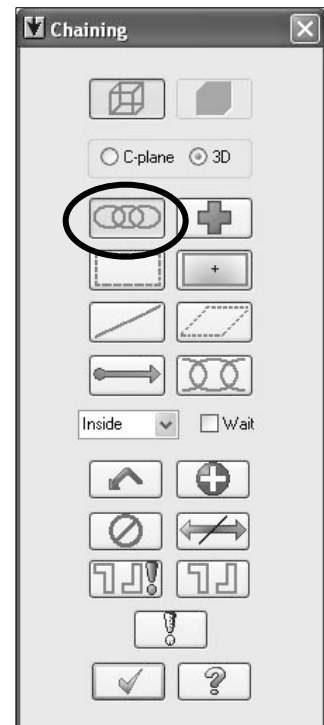
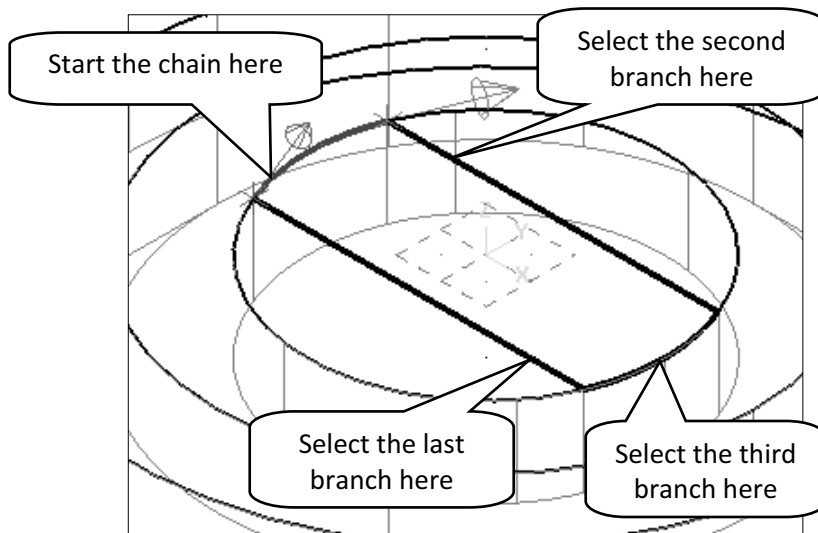


- From **Solids** toolbar, select **Solids Extrude** icon





- Select the first arc with the 4.0" diameter in the CW direction.

- Select **Reverse** button  in the **Chaining** dialog box if the arrows are not pointing in the CW direction as shown;

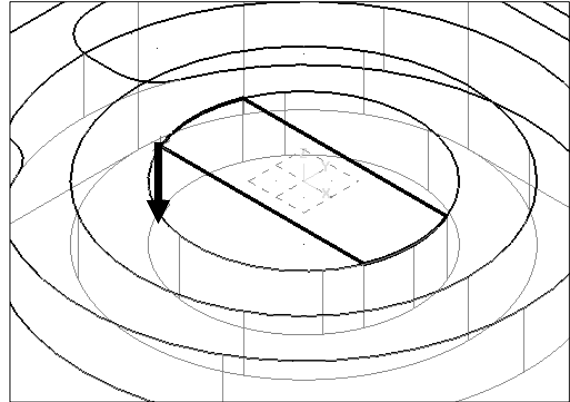



- Note that you will have to guide the red color arrow along the circle to make sure that you select the entire circle as one chain.

- Select the **End** button in the **Chaining** dialog box to finish the selection. 
- Select the **OK** button to exit the **Chaining** dialog box. 

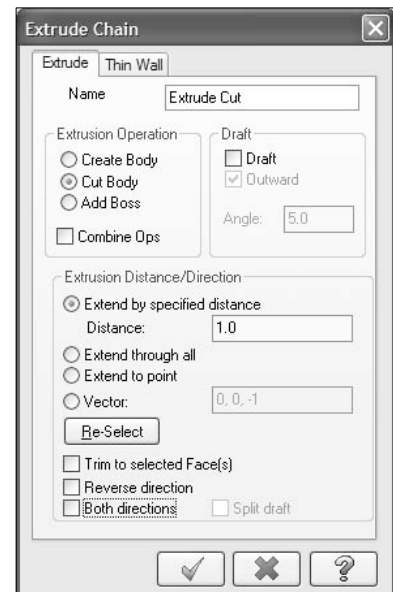
## Design

- Make sure that all arrows are pointed in the same direction. (Downwards).



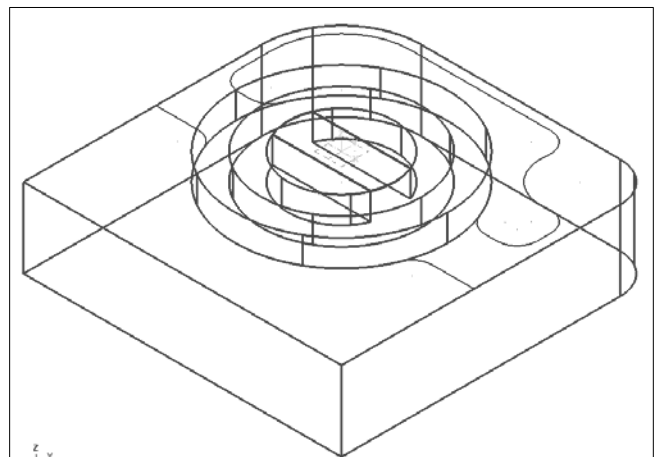
- Select the **OK** button to exit the **Chaining** dialog box. 

- Modify the **Extrude** dialog box to cut the pockets 1" deep.



- Select the **OK** button. 

The drawing should look as shown in the following picture.

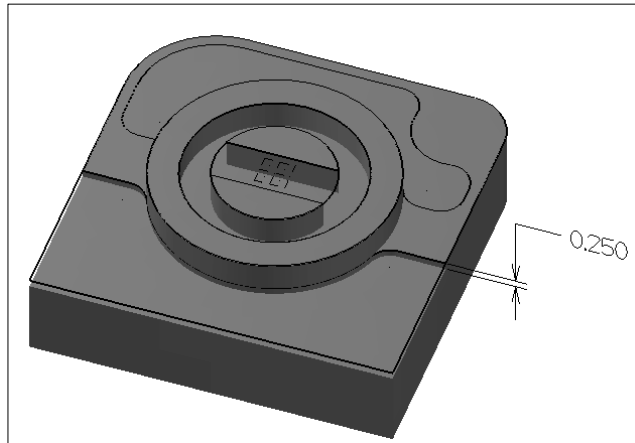


## Design

### STEP 23: EXTRUDE - CUT THE OPEN POCKET 0.25" DEEP

- Remember that extrude requires closed chains as geometry. We will have to break the geometry to have the appropriate geometry to chain.

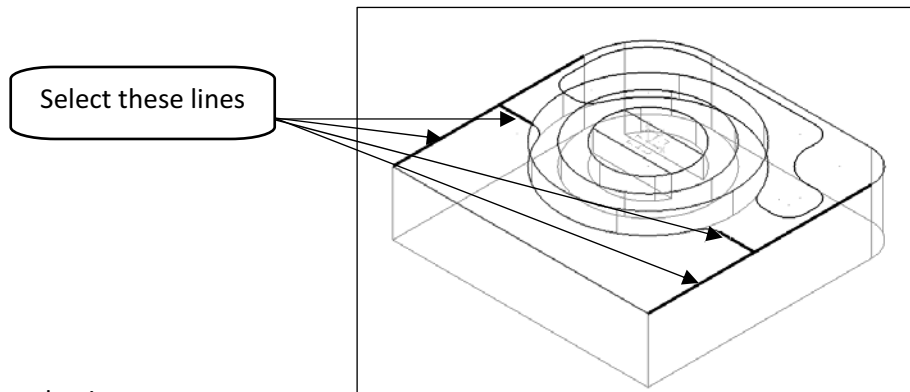
*Step Preview:*



#### 23.1 Break the lines at intersection

**Break at Intersection** 

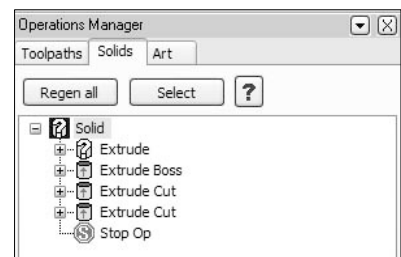
- [Select entities to break]: Select the four lines as shown



- Press **Enter** to finish the selection.

- Move the cursor along the lines that we broke. The system will highlight the different lines.

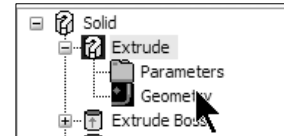
- Note the question mark on the first Extrude operation and on the Solid in the Solid Manager. The reason behind it is that the original geometry used to create the solid body is modified because we broke to lines.



**Design**

**23.2 Fix the Extruded geometry**

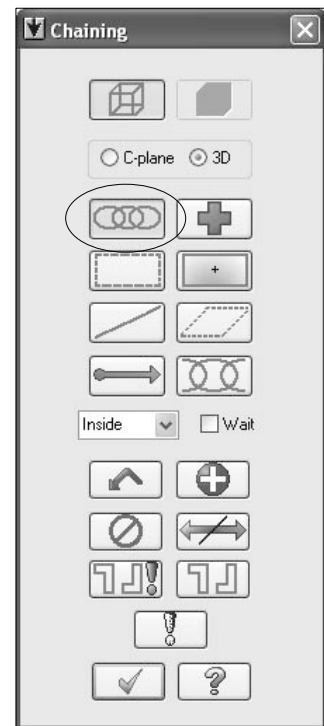
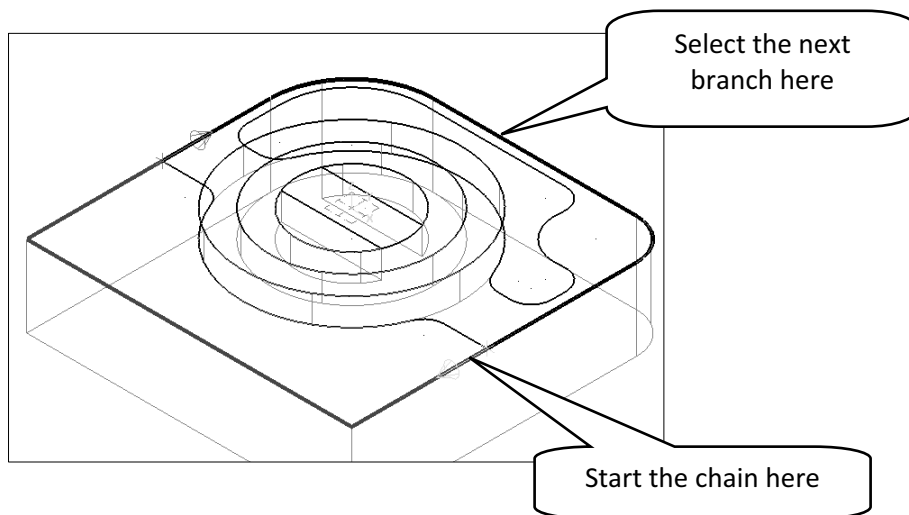
- ➔ Click on the plus sign in front of the **Extrude** to expand the solid tree.
- ➔ Select **Geometry**



- ➔ **Right-mouse** click on the **Base Chain** and select **Rechain**



- ➔ Reselect the outer geometry CW. Make sure that you guide the red color arrow along the entire chain as shown.



- ➔ Select the **End** button in the **Chaining** dialog box to finish the selection.



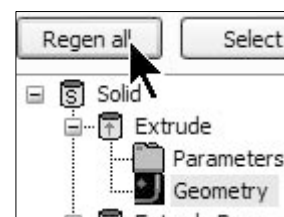
- ➔ Select the **OK** button to exit **Chaining** dialog box.



- ➔ Select the **OK** button to exit the **Solid Chain Manager**.



- ➔ Select **Regen all** to regenerate the solid.



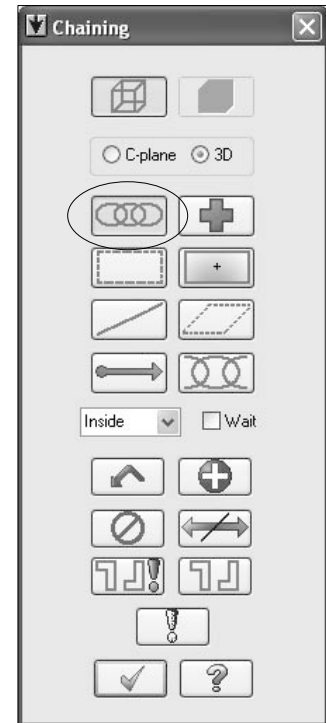
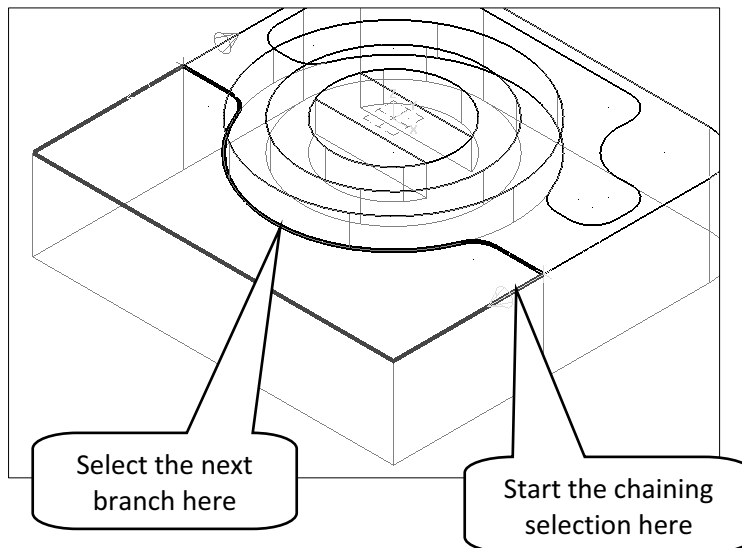
## Design

### 23.3 Extrude-cut the open pocket

- ➔ From **Solids** toolbar, select **Solids Extrude** icon.



- ➔ Select the pocket in the **CW** direction as shown and select the next branch as shown to complete the chain.



- ➔ Select the **End** button in the **Chaining** dialog box to finish the selection.

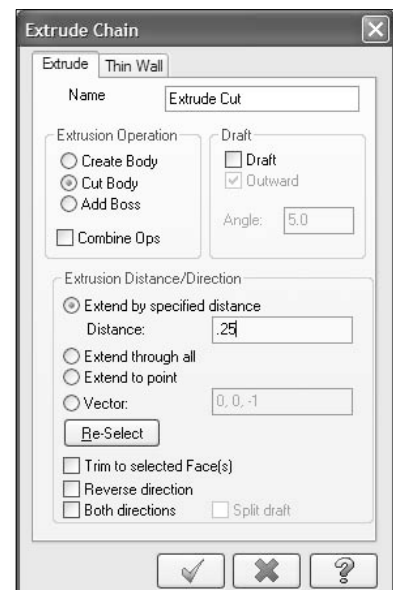


- ➔ Select the **OK** button to exit the **Chaining** dialog box.



- ➔ Make sure that the arrow is pointed downwards.

- ➔ Modify the **Extrude** dialog box to cut the pockets 0.25" deep.



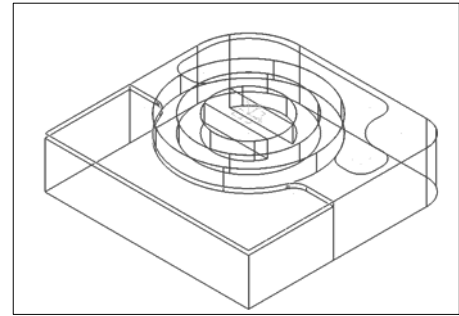
- ➔ Select the **OK** button.





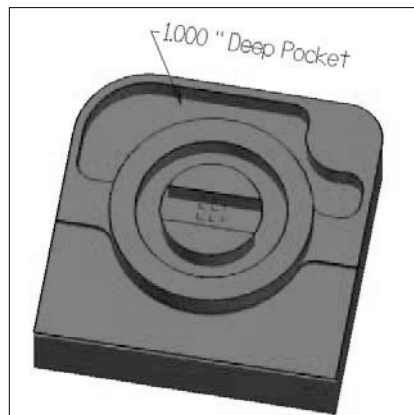
## Design

The drawing should look as shown in the picture.



### STEP 24: EXTRUDE - CUT THE POCKET 1.0" DEEP

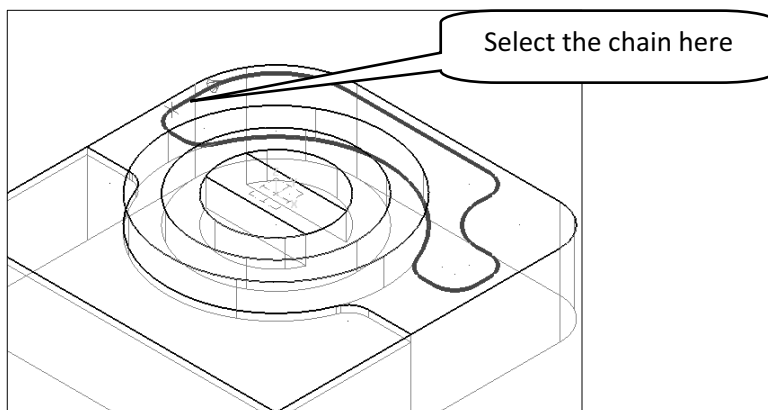
Step Preview:



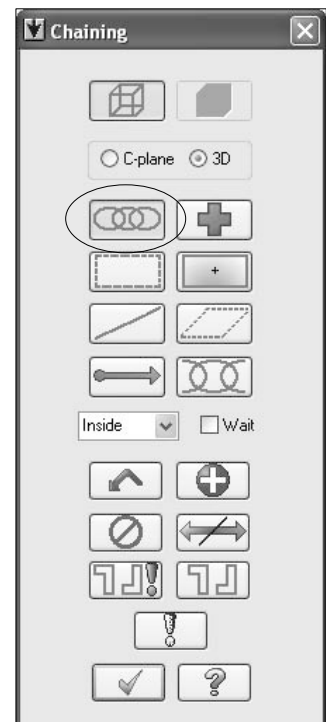
- From **Solids** toolbar, select **Solids Extrude** icon



- Select the pocket in the **CW** direction as shown;

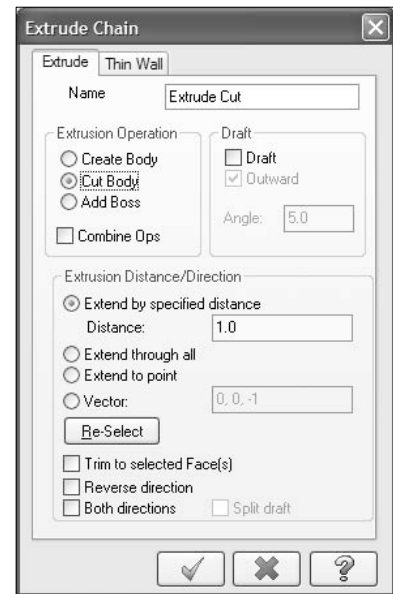


- Select the **OK** button to exit the **Chaining** dialog box.



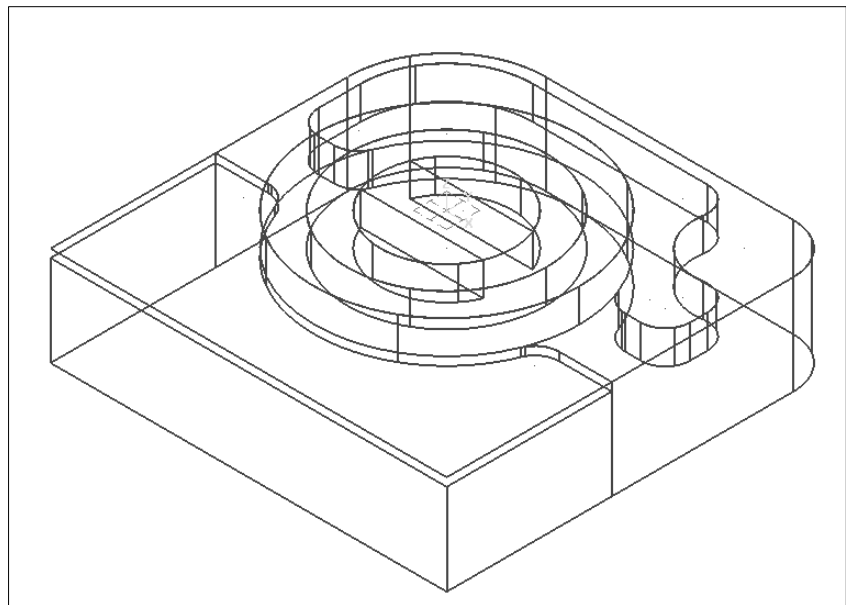
## Design

- Make sure that the arrow is pointed downwards.
- Modify the **Extrude** dialog box to cut the pockets 1.0" deep.




- Select the **OK** button. 

*The final drawing should look as shown in the following picture.*



- Note that we didn't create the through holes as they were predrilled for fixture purpose. We are not going to machine them.

## File

- **Save As**
  - File name: "Your Name\_9"
  - Select the **OK** button. 

## Design

### STEP 25 IMPORTING A SOLIDWORKS FILE

#### File

##### ➔ New

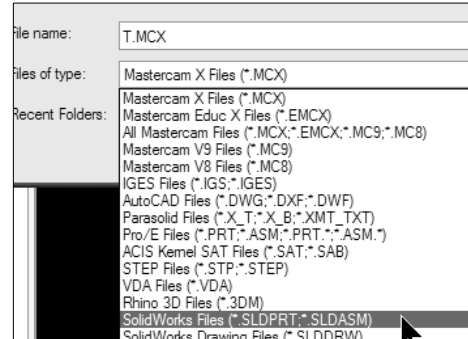
Download the file from:

[www.emastercam.com/files/x4\\_mill\\_l1.htm](http://www.emastercam.com/files/x4_mill_l1.htm)

#### File

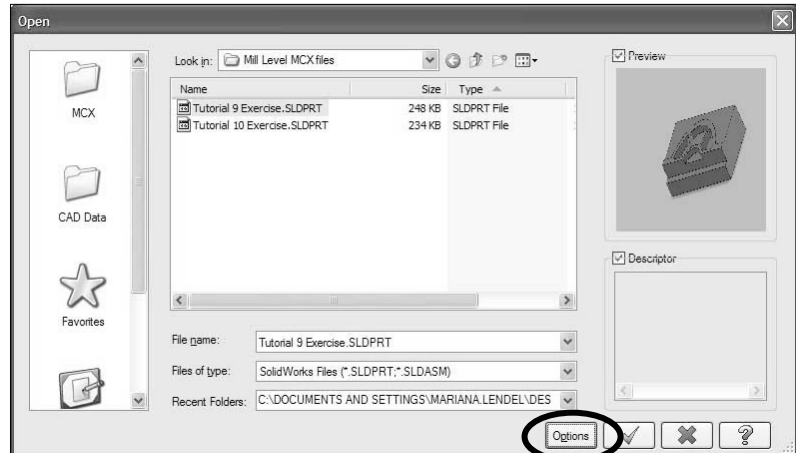
##### ➔ Open




- ➔ In **Files of type** field select the SolidWorks File (\*.SLDPRT; \*.SLDASM)
- ➔ Select the file Tutorial 9 Exercise.sldprt.



- ➔ You can only enable Import Solids History if you have SolidWorks installed on your computer. Skip the Options step if you don't.

- ➔ Select the **Options** button.



- ➔ Enable Import Solids History to be able to see it in Solids Manager.
- ➔ Select the **OK** button twice. 
- ➔ Change the graphic view to **Isometric**. 
- ➔ Select **Fit** icon to fit the drawing to the screen. 
- ➔ Press Alt +S to shade the part if needed.

The part should look as shown to the right

