Chapter 2

Placed Features, Assembly

2-1 OVERVIEW

In this chapter we illustrate DesignModeler creation of features whose **shape** is **predetermined**. Among these are

- ♦ Holes

- ♦ Rounds

- ♦ Chamfers

- ♦ Patterns

In addition to these topics, at the end of the chapter we illustrate simple **assembly modeling** in ANSYS DesignModeler.

2-2 INTRODUCTION

Feature-based solid modeling involves the creation of part models by combining various features. The features illustrated in Chapter 1 are sometimes called **sketched features** because they were based upon sketched cross sections we created. Sketched features can have virtually any shape we desire. The basic parts of Chapter 1 can also be called **base features** since we started from scratch each time and created a new part.

We can add features to base features to create more complex parts. If these added features have predetermined shapes they are often called **placed features** because all we need to do is specify the size and location or placement of the new feature on an existing base feature.

The figure below shows the L-shaped Extrusion created earlier with a **hole**, a **round** and a **chamfer** added to it. This demonstrates the manner in which features can be added to a base feature in order to create more complicated and useful parts with the shapes desired for specific tasks.

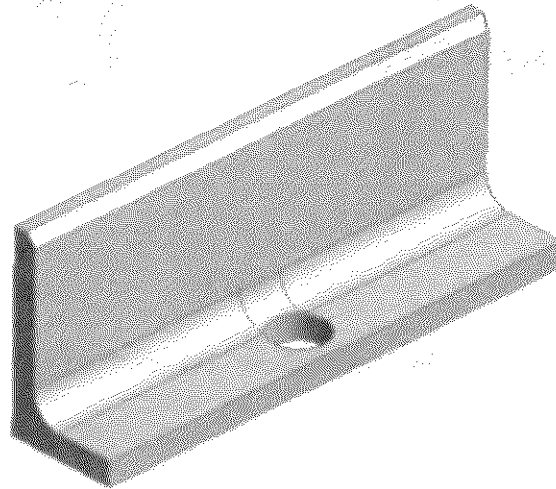The tutorials that follow illustrate how to add these placed features to the basic parts created in Chapter 1.

Figure 2-1 Extrusion with placed features.

2-3   TUTORIAL 2A – ADDING A HOLE TO THE EXTRUSION

Follow the steps below to cut a hole in the top face of the short leg of the Chapter 1 extrusion.

Start ANSYS Workbench and reload the L-section Extrusion

1.   **ANSYS > Workbench > DesignModeler Geometry > Browse > Tutorial1A** (or the file name you chose)

Now save this file under a new name for this tutorial.

2.   **File > Save As > Tutorial2A** (or another name you select)

We want to create the hole on the top surface of the short leg of the extrusion. We will create a new plane on which to place the circle that generates the hole.

3.   **Selection Filter: Model Faces (3D)**
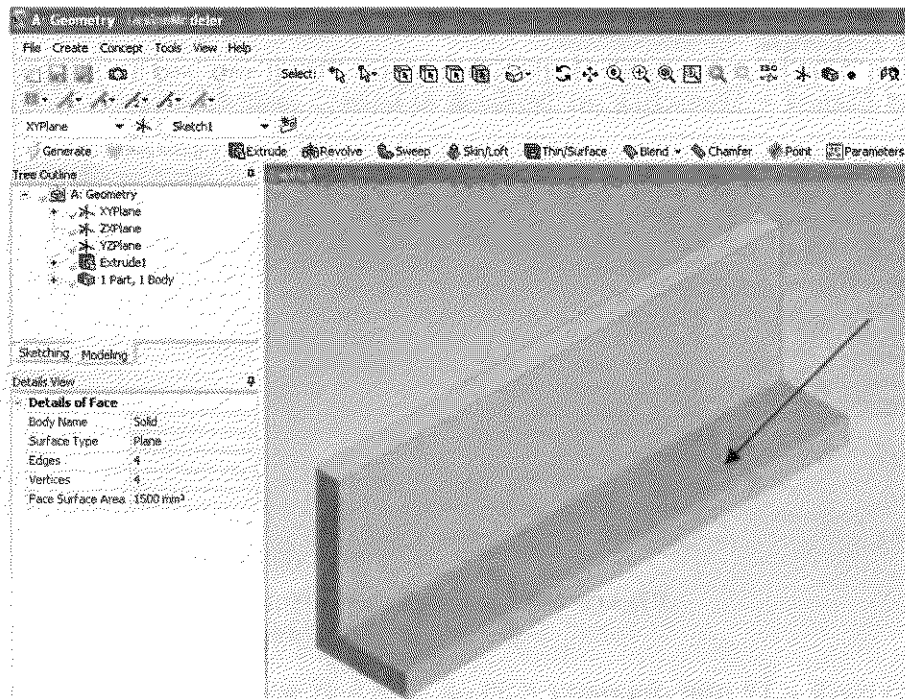
4.   **Click on the top surface of the short leg.**

Figure 2-2 Surface selected.

5.   **Create > New Plane** (from the top menu)

A new plane is added to the tree structure (Plane4 in this illustration; your plane number may be different) and an axis system is provided.
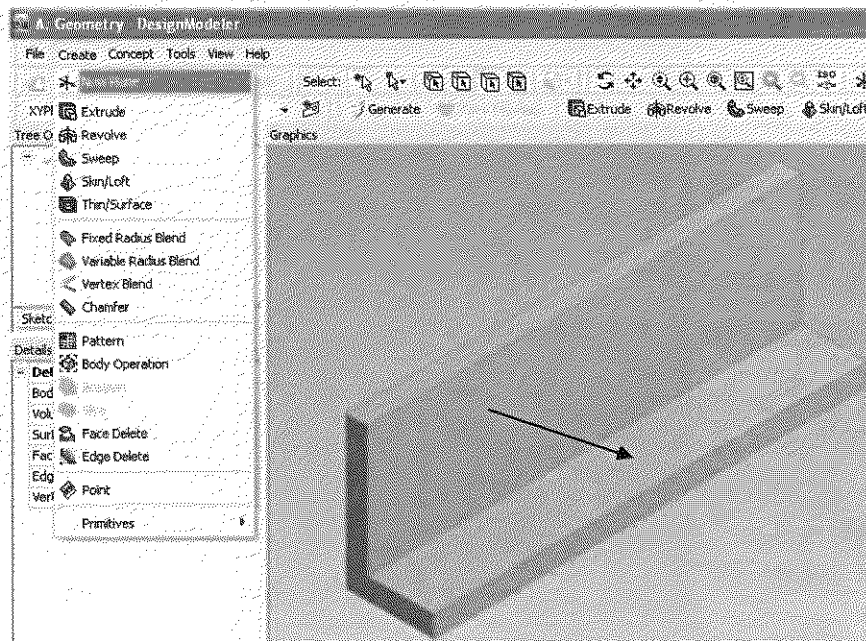

Figure 2-3 New plane is created.

6.   Click Generate   ⟋ Generate

7.   Select Plane4 > Then **click** on the **Look at icon** to view the Plane4   🔲.
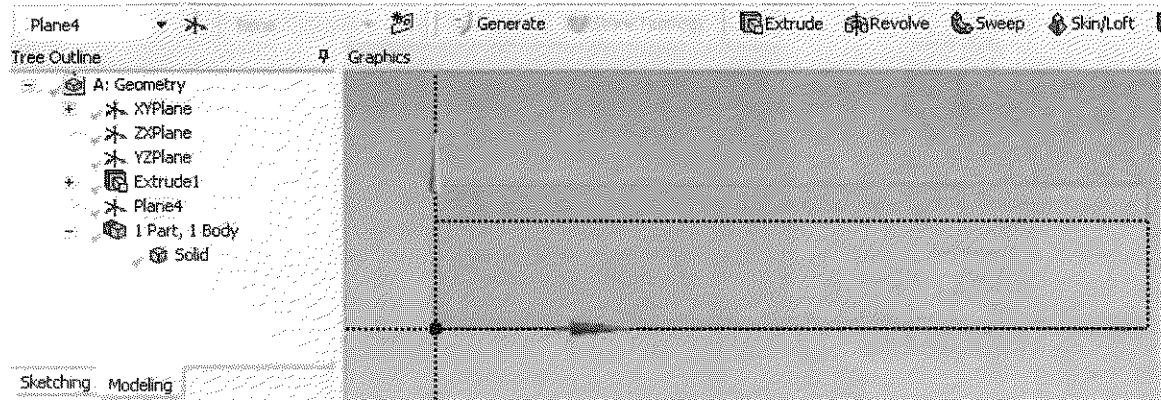
Figure 2-4 'Look at' new plane.

We want to place a **10 mm diameter** circular hole **half way** along the length of the 100 mm leg and **8 mm** from the edge.

8.   Sketching > Circle Draw a circle on the top face as shown in the figure below.
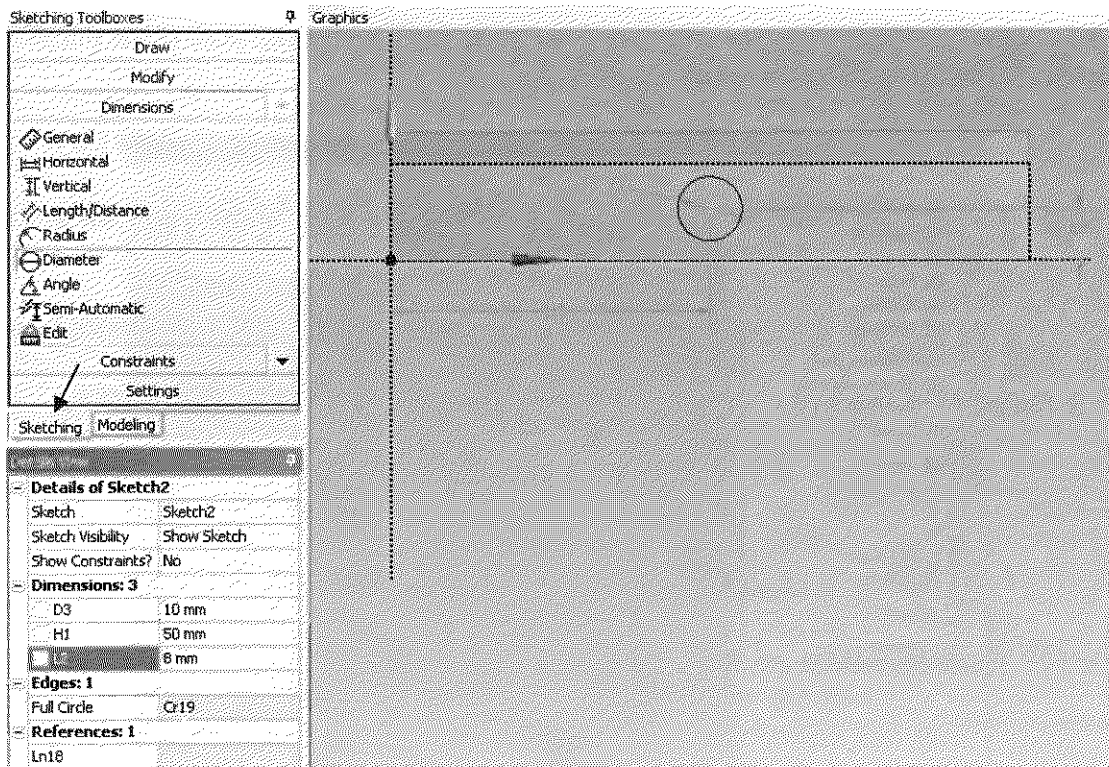
Figure 2-5 Circle sketch.

9.   **Dimensions > Diameter** – Place D3 as shown.

10.   **Dimensions > Horizontal** – Place H1 as shown.

11.   **Dimensions > Length** – Place L2 as shown.

Edit the dimension values to 10 mm, 50 mm, and 8 mm as shown in the figure above. (The number attached to each dimension, the 1 in H1, is an internal numbering scheme and depends upon the sketching sequence. Your numbers may be different.)

12.   **Modeling > Click** on **Sketch** in the tree structure (Sketch2 in the figure)

13.   **Extrude**   Extrude

14.   **Operation > Cut Material**

(Remove material instead of adding it.)

15.   **Type > Through All**
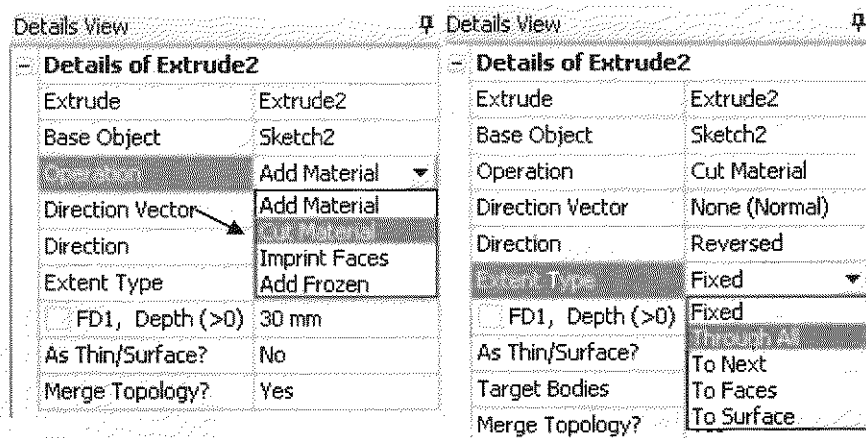
(Cut completely through the thickness, through all.)

Figure 2-6 Extrude details.

16.   **Generate** (to complete the feature.)   Generate

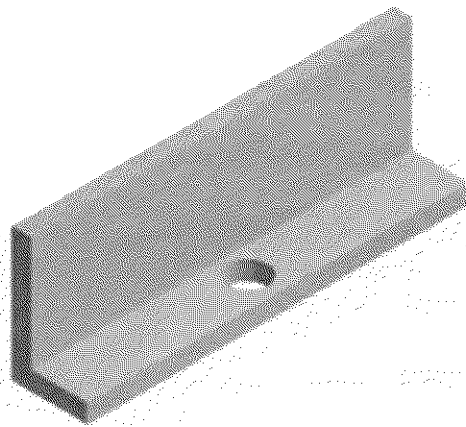The completed hole is shown in the next illustration.

Figure 2-7 Circular hole.

Keep this part in memory since we have more work to do on it.

2-4   TUTORIAL 2B – ADDING A ROUND TO THE EXTRUSION

A gradual transition between surfaces is variously called a **fillet**, a **round** or a **blend**. DesignModeler uses the blend terminology, and a blend is a placed feature. In this tutorial we will place a fixed radius blend at the inside corner of the L where the top surface of the short leg meets the inside vertical surface.
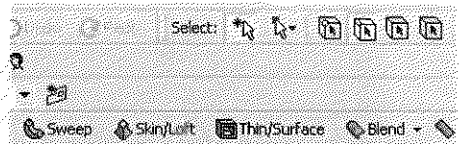
Save your part using a new name.

1.   File > Save As > Tutorial2B

Set the selection filter.

2.   Selection Filter: Edges
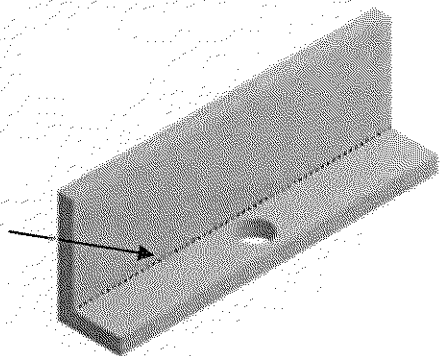
3.   Select the inside edge of the part.

Figure 2-8 Select the edge.

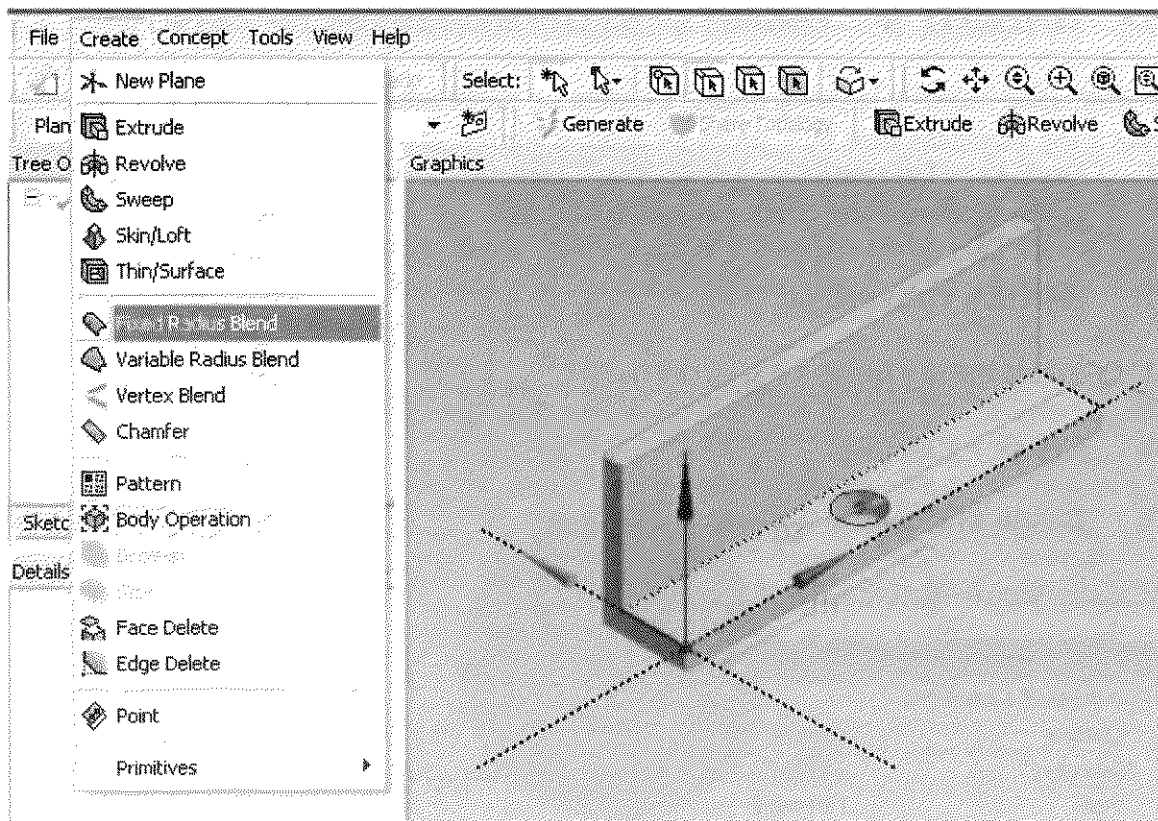**4.   Create > Fixed Radius Blend**

Figure 2-9 Selecting the blend option.

5. **Geometry > Apply**

(Use the default 3 mm radius.)   **Generate**
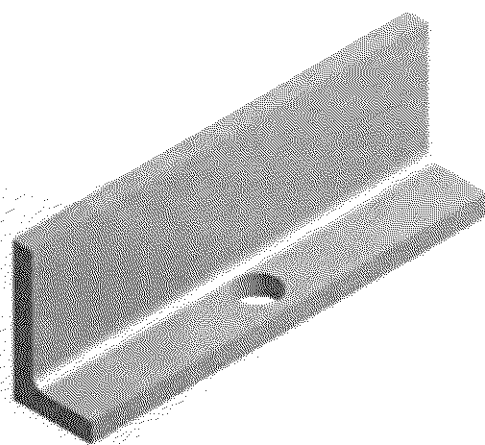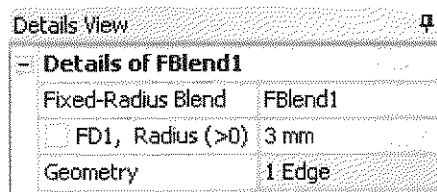

Figure 2-10 The blend completed.

2-5   TUTORIAL 2C – ADDING A CHAMFER TO THE EXTRUSION

Creating the chamfer is pretty much like creating the blend.

1.   **Save As > Tutorial2C**
      (Save the part under a new name if you wish)

2.   **Selection Filter: Edges** 🔲

3.   **Select the top inside edge of the part**.

4.   **Create > Chamfer**

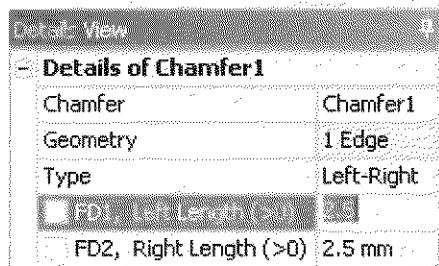5.   **Details of Chamfer > Geometry > Apply**

(Change the sizing to 2.5 mm.)

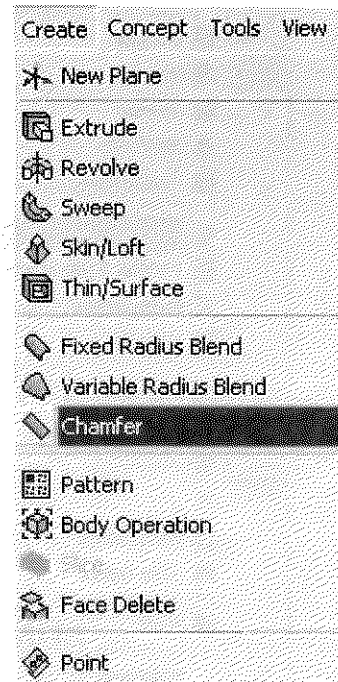| Details of Chamfer1	
Chamfer	Chamfer1
Geometry	1 Edge
Type	Left-Right
FD1, Left Length (>0)	
FD2, Right Length (>0)	2.5 mm

Figure 2-11 Select chamfer

6.   **Generate**   Generate
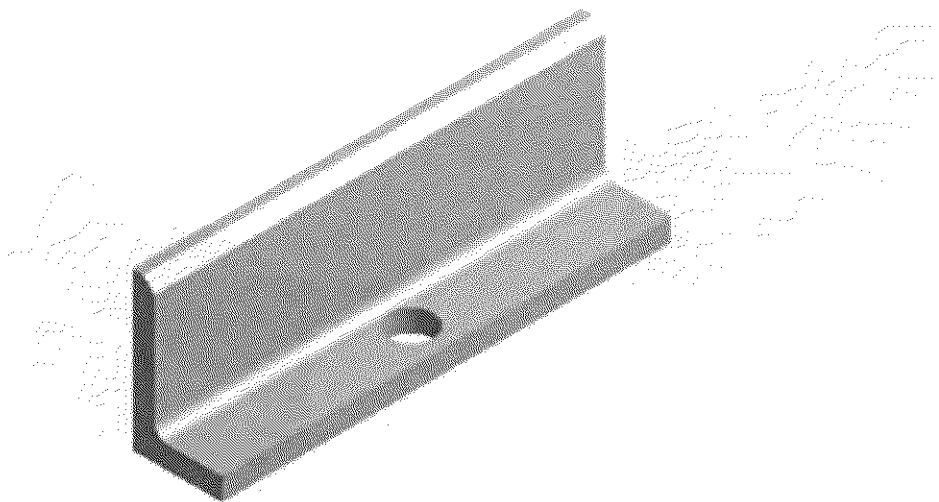
7.   **Save your work**

Figure 2-12 The chamfer completed.

2-6 TUTORIAL 2D – PATTERNS

Next we will use a **pattern operation** to create a solid model of circular plate with a symmetric bolt hole pattern. First extrude a circle to create a **50 mm diameter** plate that is **10 mm** in **thickness** as shown in the next figure. Start a new part file.

1. **Sketch a 50 mm diameter circle** with center at the origin of the **XYPlane**. Click on the sketch then on **Extrude** and set the extrusion depth to be **10 mm**. Click Generate to complete the base feature disk.

2. **Sketching > Dimensions > Display >** check both **Name** and **Value**.
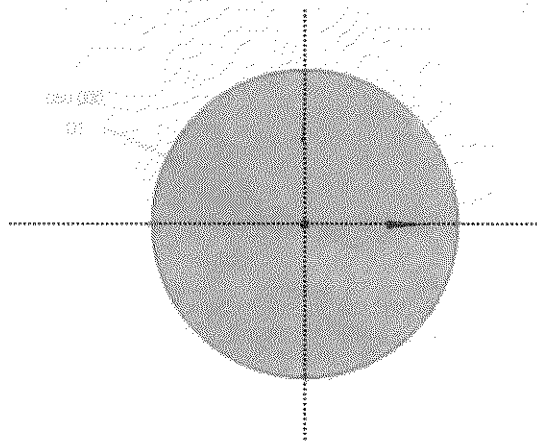
Figure 2-13 Base feature disk.

3. **Create > New Plane** Create a new plane for sketching on the **top or bottom surface** of the base disk

4. **Sketch an 8mm diameter circle** on this plane. Dimension as shown in the next figure.

5. **Sketch an 18 mm line** from the **center of the base feature** to the **center of the small circle.** Dimension as shown below. We'll use this line for angular reference.

6. **Locate the line with an angular dimension.** Click first on the horizontal axis, then on the line. Drag to place the dimension as shown.

7. **Switch to Modeling. Select the sketch and then Extrude > Cut Material > Through All** to create a hole.
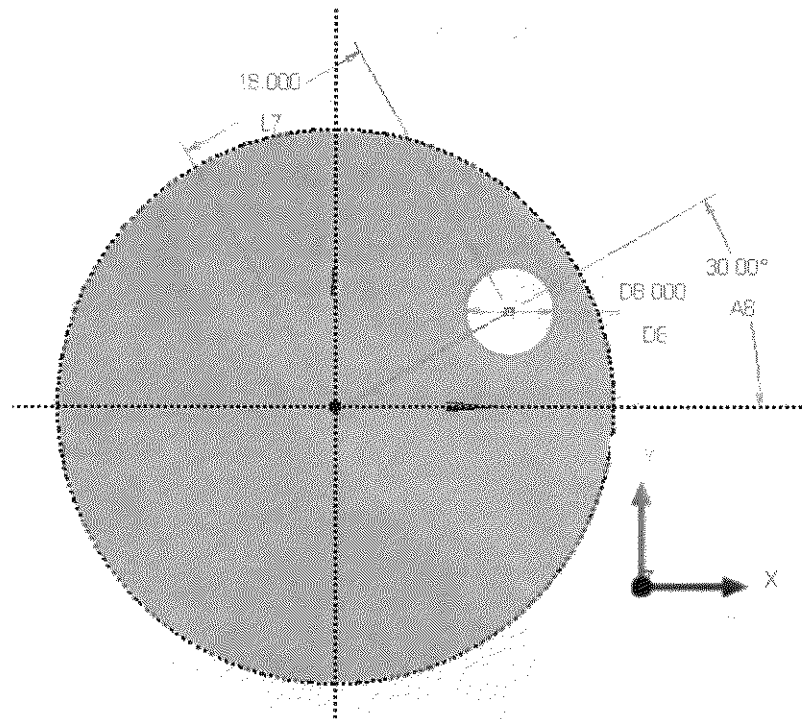
Figure 2-14 Placement of small hole.

8.   **Add a 1 mm chamfer to the top edge of the 8 mm hole.** See figure next page.

9.   **Sketch a short line along the Z axis** in the ZXPlane: Sketch3. We will use this for the pattern angular direction reference later.
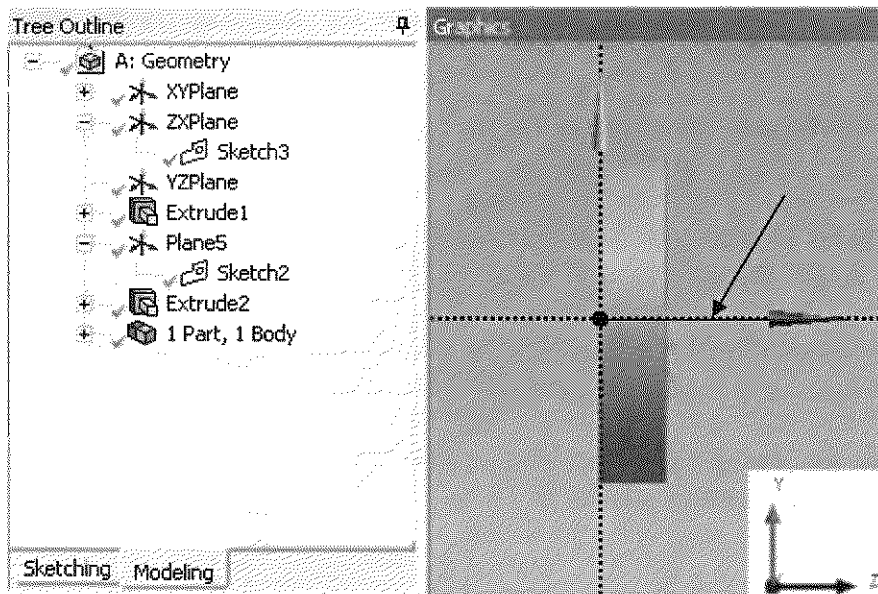

Figure 2-15 Create a line along the Z axis.

10.   **Click Selection Filter: Model Faces (3D)**

11.   **Select the inside surface of the hole**; then **Ctrl > Select** the **surface** of the **chamfer** so both items will be in the pattern.
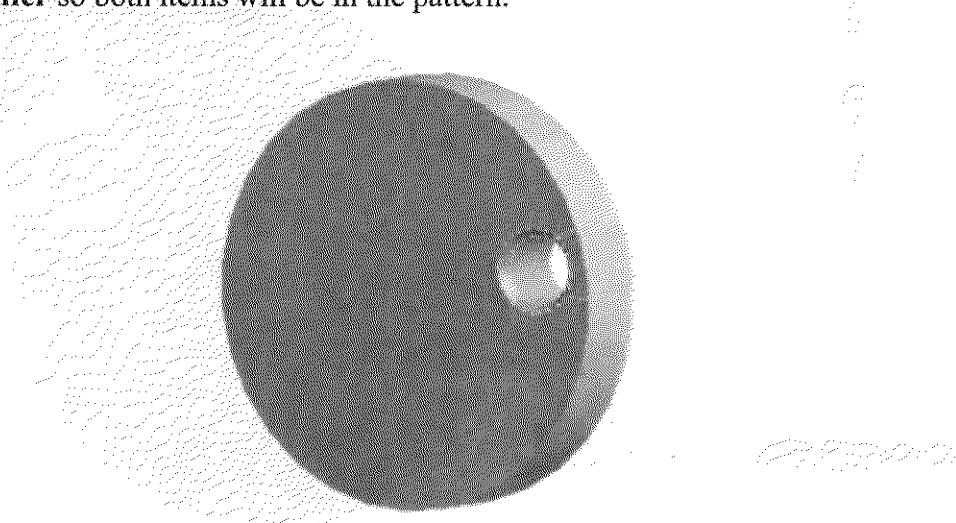

Figure 2-16 Select the chamfer and hole.

12.   **Create > Pattern**

13.   **Geometry > Apply** (in details of Pattern1.)

14.   **Pattern Type > Circular**

15.   **Selection Filter: Edges**

16.   **Axis > Click on Sketch3** and select the **short Z axis line > Apply**

17.   **Angle > Evenly Spaced**

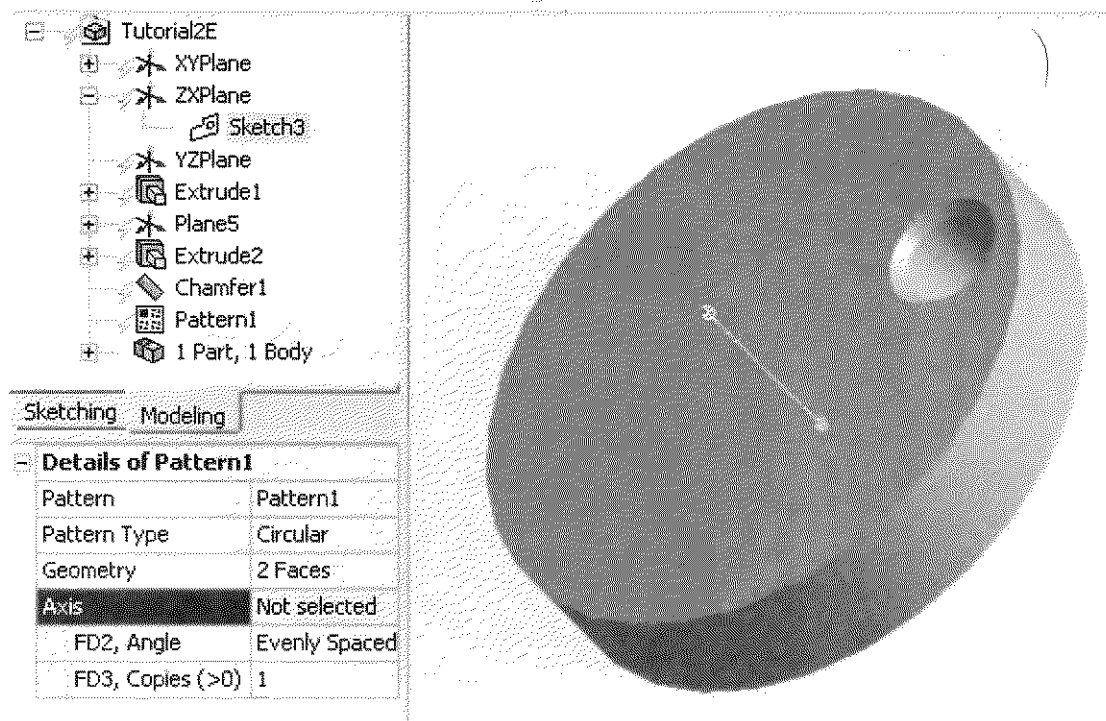18.   **Number of Copies > 6** (Creates 7 instances total.)

Figure 2-17 Pattern parameters.

19. Click Generate. Generate

The resulting hole pattern is shown next. The **selected edge** is used as the **axis** for determining the positive direction of incrementing the angular placement (taken according to the right-hand rule).
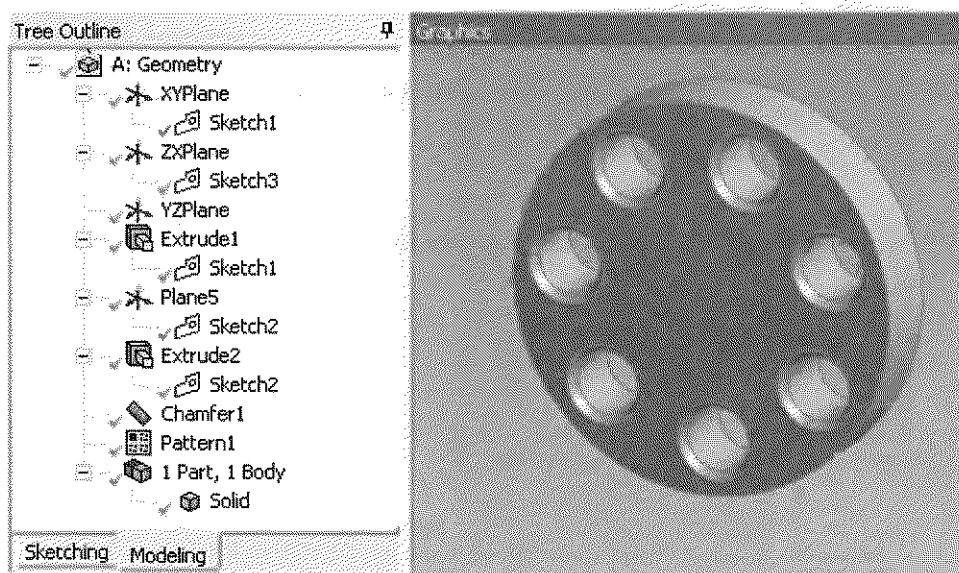

Figure 2-18 Circular pattern of chamfered holes.

Once again don't be surprised if the suffix numbers of the entities (Sketch, Extrude, etc) in your tree structures differ from those in the figures. Same with the lighting bolts indicating need for generation. Some experimentation with generation, views, etc. was conducted to obtain the figures presented here. If you have a problem, delete the problem object in the tree and start again.

(The positive direction for incrementing the angular placement is along the selected edge according to the right-hand rule. **Change Evenly Spaced to 35 degrees > 6 Copies** and see what solid is produced.)

Linear patterns are created using similar steps. The direction of the pattern can be along an existing edge or perpendicular to a surface.

2-7   TUTORIAL 2E – CLEVIS ASSEMBLY

The next figure shows an assembly model of the clevis device that is the subject of this final tutorial in this chapter.
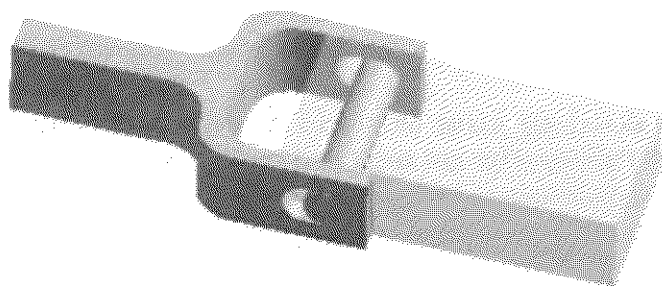

Figure 2-19 Clevis assembly.

1.   Start DesignModeler, Select Inches Units, and start sketching on the XYPlane

The yoke is 4.5 inches in overall length, 2.5 inches at its widest point, and the opening is 2.0 inches in width. Use the sketching tools to create the figure shown next with dimensions as indicated. **Arc by Tangent, Modify > Trim**, and other tools will come in handy. If you make a mistake, just delete the item in question and redraw, or just start over.
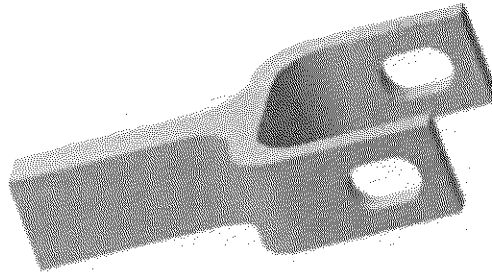
Figure 2-20 Clevis sketch.

2.   **Create the sketch** shown above and **extrude it symmetrically 0.5 inch**. (Total
      height will be 1.0 inch, 0.5 above the sketch plane, 0.5 inch below.)

3.   **Create a new sketching plane** on one of the yoke fingers and **sketch the opening**
      shown. The two semicircles are separated by 0.25 inch. Tangent line and trim will
      be useful.

Figure 2-21 Slot sketch.

4.   **Extrude this sketch through all, removing material.**

We obtain the solid model shown next.

Figure 2-22 Complete clevis.

To this we want to add the stem and pin to complete the assembly. First **hide the clevis**.

5.   **1 Parts, 1 Body > Solid > Right Click > Hide Body**

6.   We'll create the **stem** first. **View the XZ sketch plane** and **sketch the rectangle and circle** below for creation of the **brick-shaped stem extrusion.** Use **General** for the linear dimension definitions.
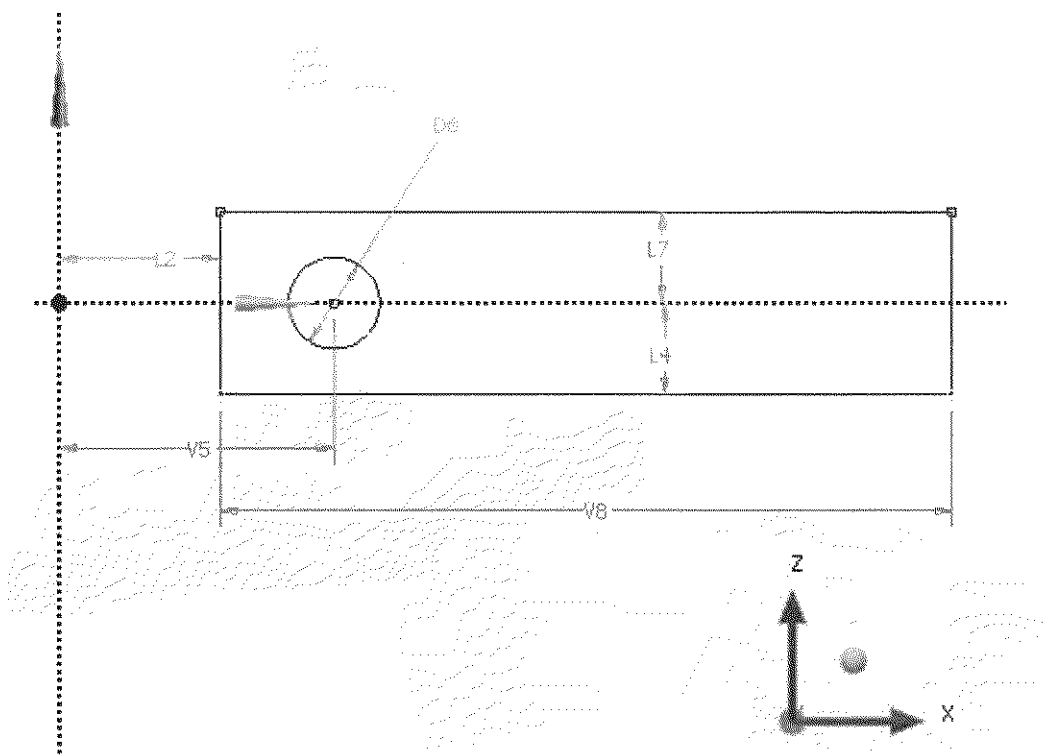

Figure 2-23 Stem sketch.

Refer to Figure 2-20 and give the L2 and V5 placements dimensions the values shown below.

Details of Sketch3	
Sketch	Sketch3
Show Constraints?	No
Dimensions: 6	
D6	0.5 in
L2	0.875 in
L4	0.5 in
L7	0.5 in
V5	1.5 in
	4 in

Figure 2-24 Stem sketch dimensions.

7. **Switch to Modeling**, **Select the sketch** of the **Rectangle** with **Circle**, Click **Extrude**.

8. **Details of Extrude > Operation > Add Frozen, Direction > Both – Symmetric**, and **Depth > 1.0** (See below.)

The **Add Frozen** option adds the stem as a new, separate object and does not merge the new geometry into the existing clevis. The two parts remain separate.

9. **Generate**   Generate

10. **2 Parts, 2 Bodies > Solid** (Clevis) **> Right Click > Show Body**

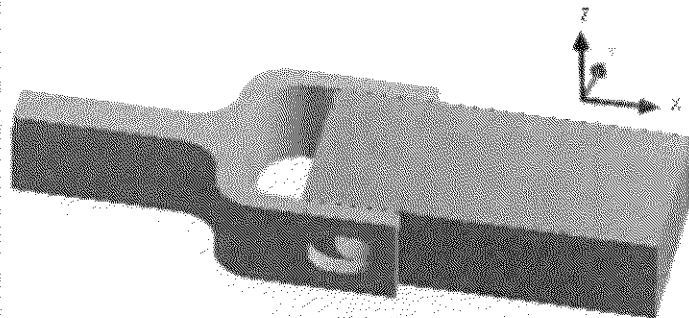| Details of Extrude10	
Extrude	Extrude10
Base Object	Sketch3
Operation	Add Frozen
Direction Vector	None (Normal)
Direction	Both - Symmetric
Type	Fixed
FD1, Depth (>0)	1 in
As Thin/Surface?	No
Merge Topology?	Yes

Figure 2-25 Stem extrusion and clevis.

Lastly we need to create the fastening pin that holds the assembly together. Create a new plane to sketch on.

11. **Click XZ Plane > Click** the **new plane icon** ✻ on the third line of icons >
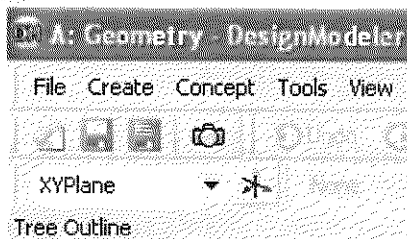   **Generate** ⬚ Generate

Figure 2-26 New plane icon.

12. **Right click on graphics screen > View > Bottom** to get the view shown below.
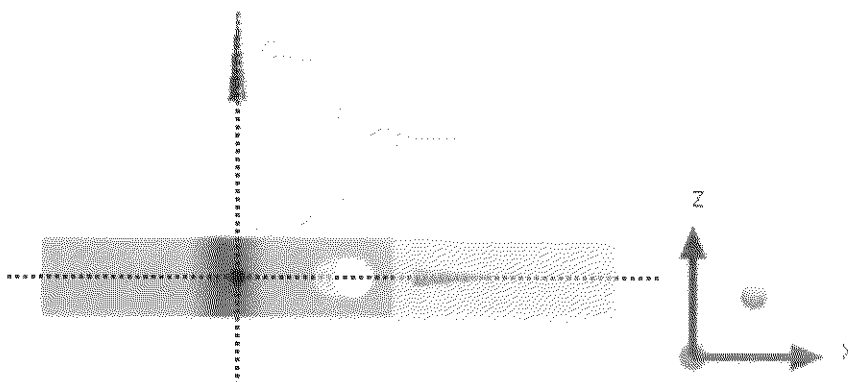

Figure 2-27 View for sketching pin.
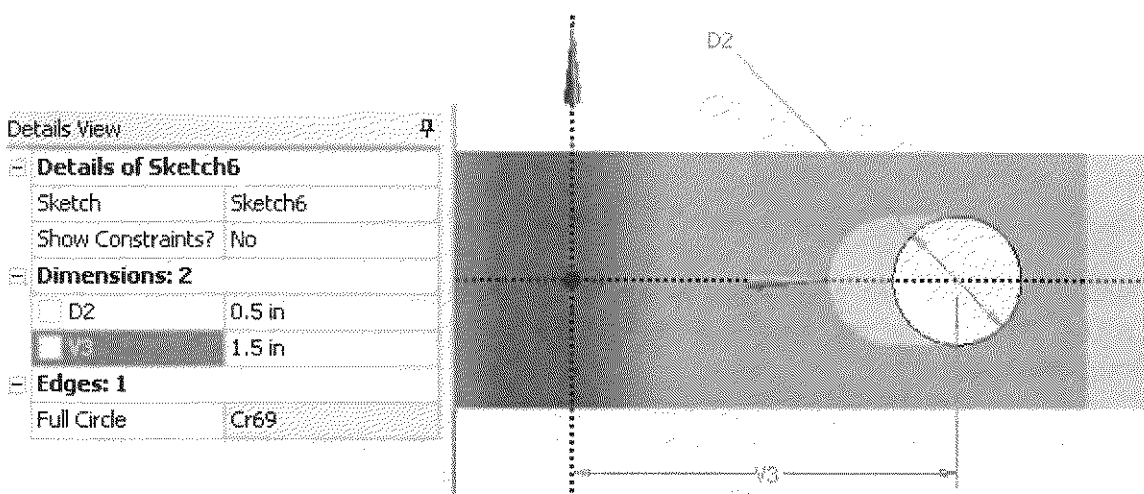
13. **Sketch > Circle**

Figure 2-28 Pin sketch.

Put the center of the circle on the X Axis; Dimension its diameter and location from the Z Axis. Set the diameter to **0.5 inches** and the horizontal distance from the Z Axis to **1.5 inches**. (See Figure 2-20.)

Now create the pin extrusion using this sketch.

14. **Switch to Modeling, Select the sketch** of the **Circle,** Click **Extrude.**

15. **Details of Extrude > Operation > Add Frozen; Direction > Both – Symmetric; Depth > 1.25**
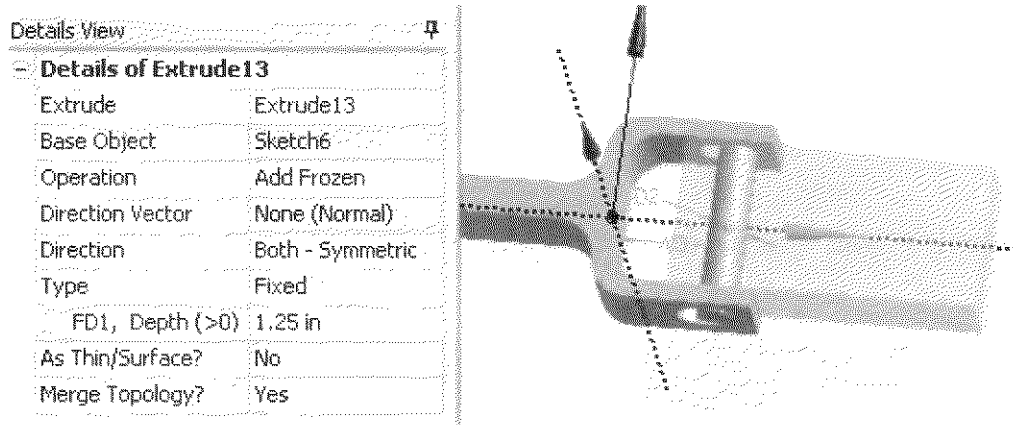

Figure 2-29 Final clevis, stem, pin assembly.

At any time use the middle mouse button to rotate the view so you can see the sketch plane with respect to the rest of the model and turn the axis and dimensions on/off by clicking the icon on the second row.

16. **Save your work.** We'll use an assembly such as this later.

2-8 TUTORIAL 2F – ALTERNATE SOLID MODELER

Finally we outline the procedures to utilize an alternate solid modeler such as Pro/E, CATIA, SolidWorks, etc. for use with ANSYS Workbench projects as an alternative to DesignModeler.

1. **Start the alternate solid modeler** and **create the clevis, stem,** and **pin parts** discussed above.

2. **Create** the **Clevis-Stem-Pin Assembly** in the **Alternate solid modeler.**

3. Using the **Alternate Solid Modeler, Save the Assembly** in the **STEP** or **IGES format**

STEP (Standard for the Exchange of Product Model Data) and **IGES** (Initial Graphics Exchange Specification) are industry agreed upon **neutral file** formats for the exchange of modeling information.
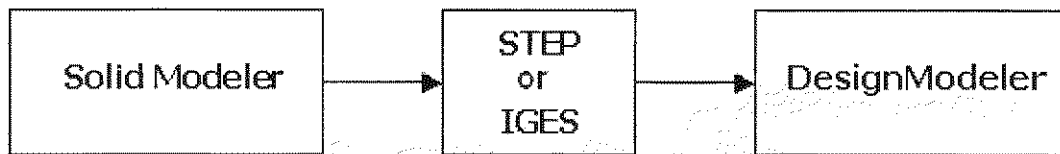

Figure 2-30 Neutral file transfer.

4.   **Start ANSYS DesignModeler**

5.   **File >** **Import External Geometry > Generate**
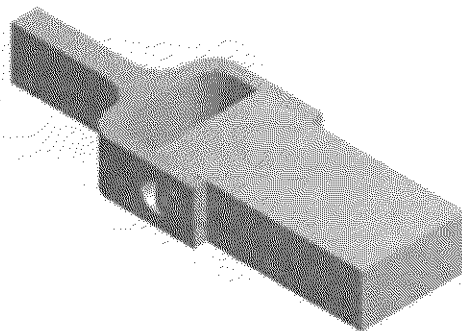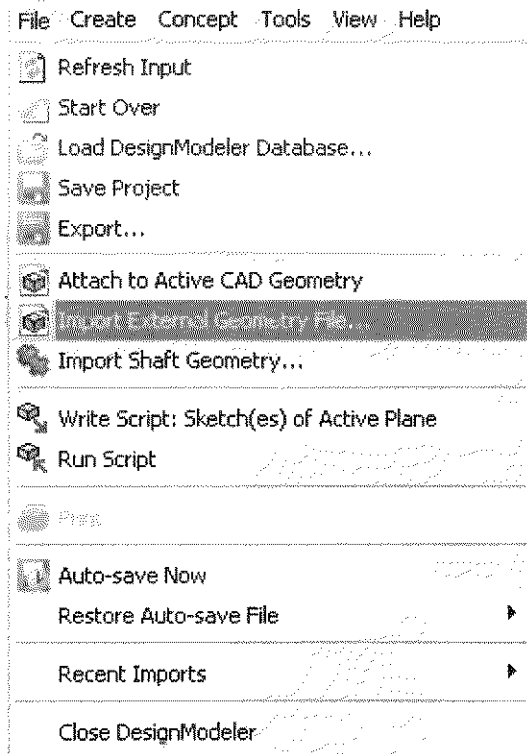

Figure 2-31 Import External Geometry using STEP or IGES.

Another way to utilize a solid modeler other than DesignModeler is to establish a direct link between the DesignModeler and your preferred Solid Modeler. With the alternate solid modeler running and the part/assembly you created in the alternate modeler as the active session in that modeler, you can start **DesignModeler** and simply **Attach** to **Active CAD Geometry** and **Generate** the model. See below.

6.   **File >** **Attach to Active CAD Geometry > Generate**
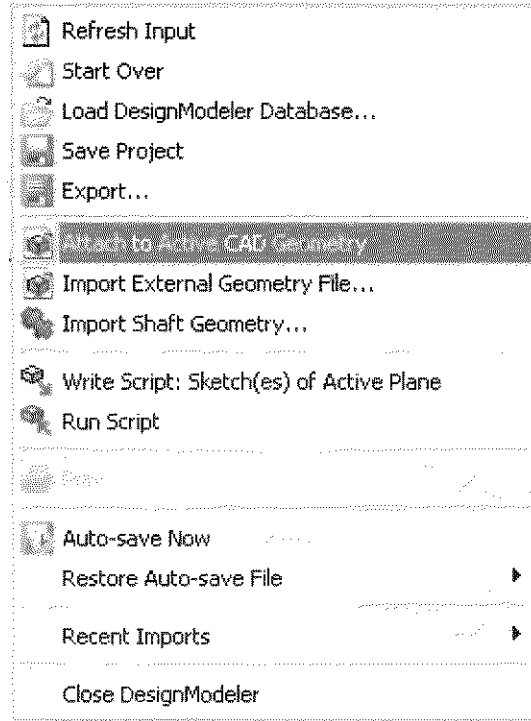

Figure 2-32 Assembly attached from an active Pro/E session.

This direct link also gives you the ability to initiate a Workbench session directly from your Solid Modeler. The geometry will be transferred automatically and need only be Generated in DesignModeler to become available.
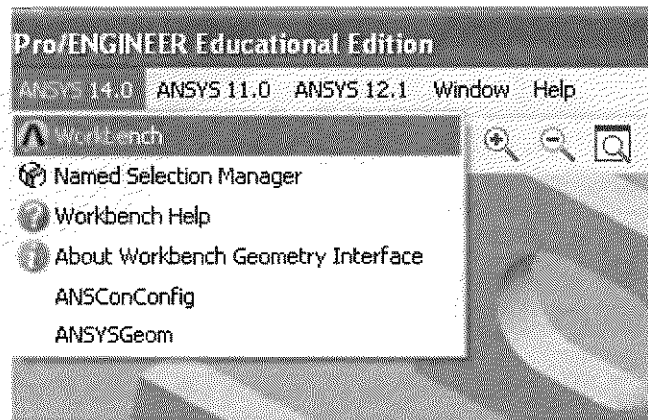

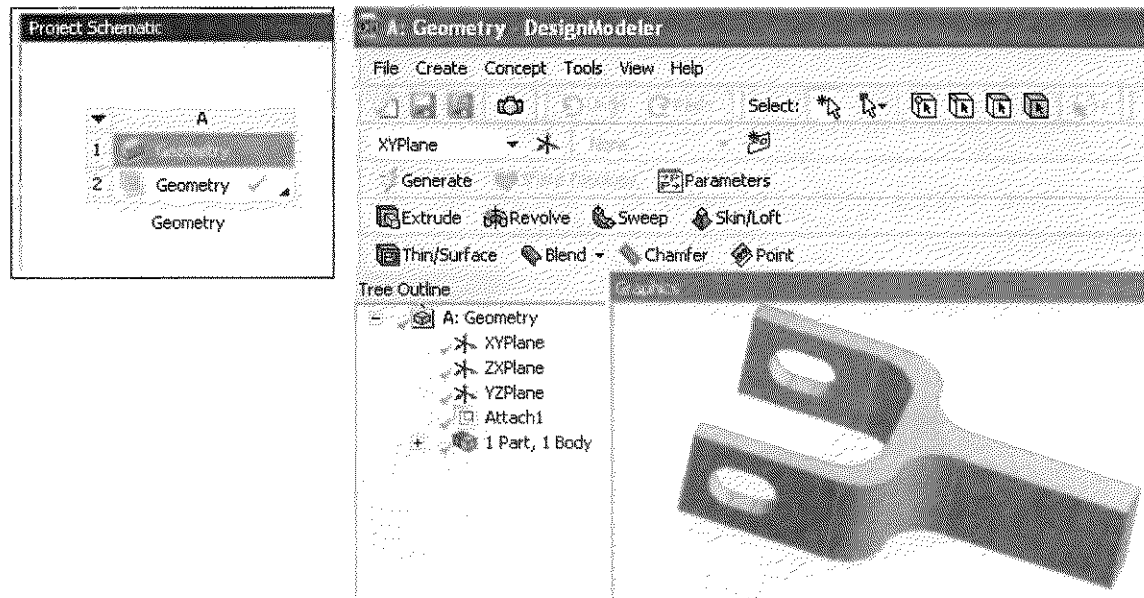**Figure 2-33** Start Workbench from Pro/E menu item.

Figure 2-34 The part from Pro/E shows up automatically as geometry object in DM.

Use the ANSYS utility CAD configuration manager to help install the geometry interfaces to the alternate solid modelers you wish to use.
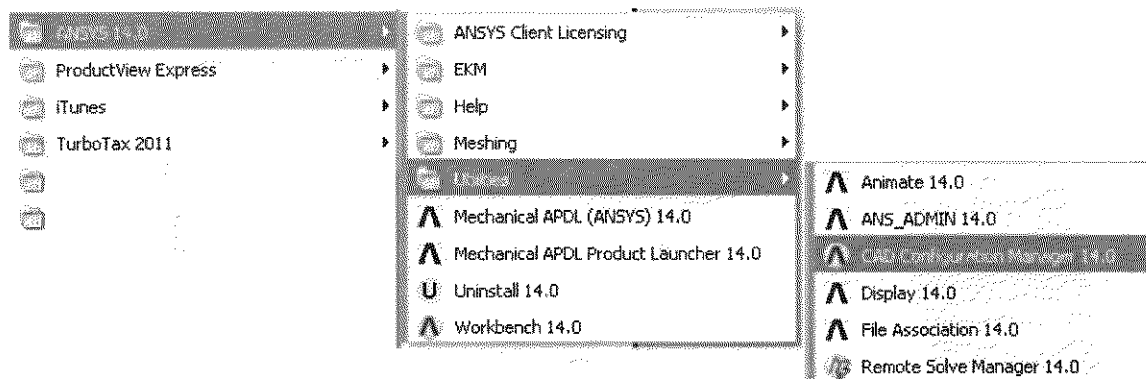

Figure 2-35 CAD configuration manager.

2-9 SUMMARY

The three tutorials in Chapter 2 illustrate basic placed feature creation and simple assembly modeling in ANSYS DesignModeler. In the next chapter we will extend these ideas to more complex parts and introduce additional solid modeling options.