Assignment 6

Inventor

# Circles and Arcs

Objectives

When you complete this assignment you will:

1. sketch and dimension circles and arcs.
2. cut holes in the model using the cut feature of the extrusion command.
3. create Arcs using the trim and three-point arc commands.
4. create models using the precise input command.
5. locate hole positions with the assistance of construction lines.

Getting Started

In this assignment you will create a new model using many of the skills you have developed in Part 1 of this module. You will become familiar with circles and add holes to a simple part.
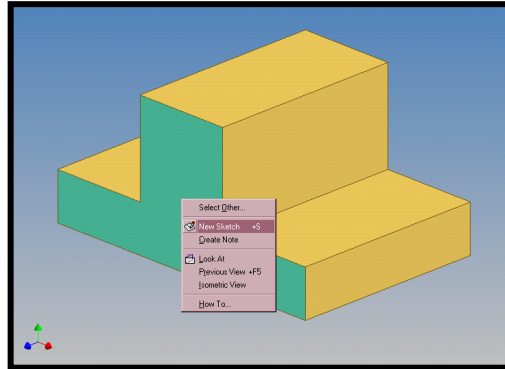
1. Launch Inventor 5.3 and launch the **Standard.ipt** template file. Create the following part.
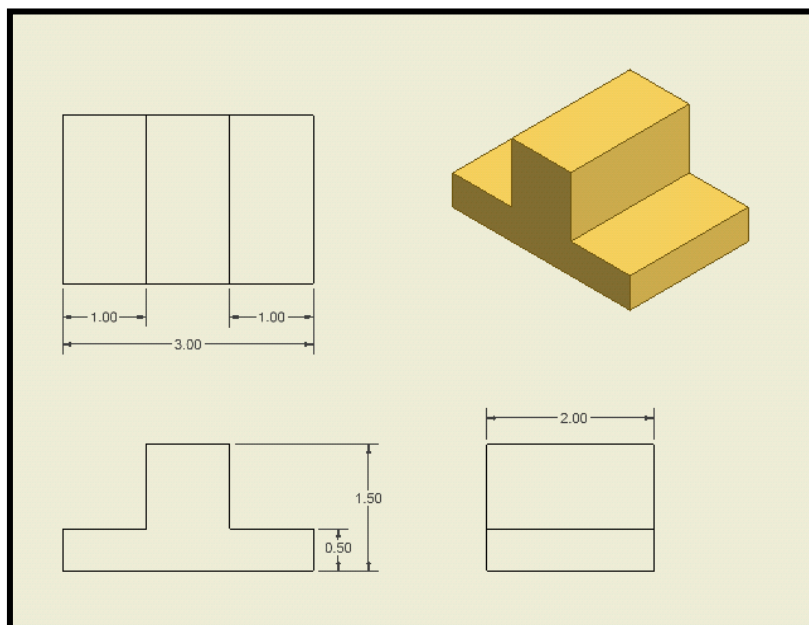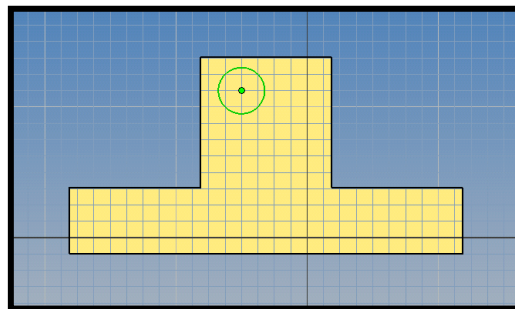
Extruding Holes

1. Right Click on the front face and select **New Sketch**.

2. Select the **Center Point Circle** tool from the sketch tab and draw a circle in the position shown below. Do not worry about exact size or position that will be adjusted later.


3. Select the **dimensioning tool** from the sketch tab and click the **center point** of the circle and then the top left corner of the model. Add another dimension line from the center point to the top right corner of the model.
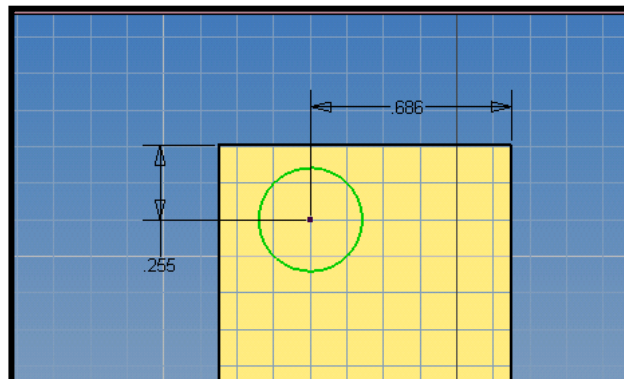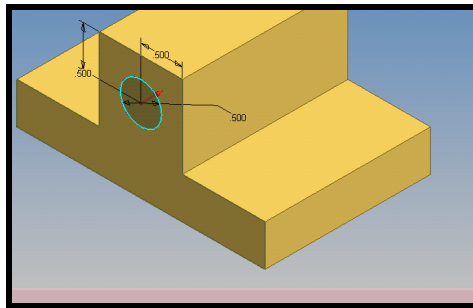

4. Edit both dimensions so they are set to 0.5.

5. Using the dimension tool again, click on the outer edge of the circle and drag out the diameter dimension. Set it to .5 as well. Your screen should look similar to the one below.
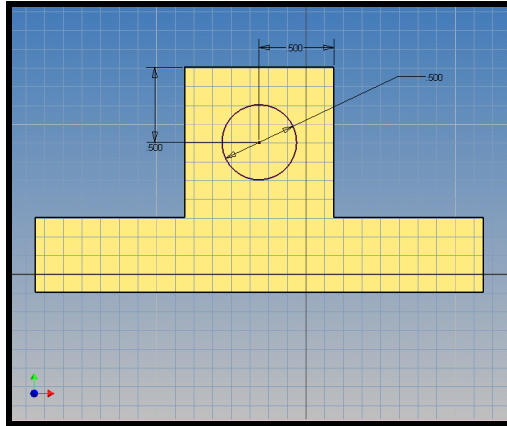
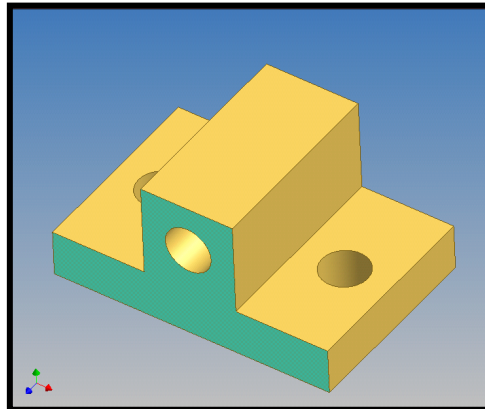6. **Finish the sketch** and **cut** the hole using the cut feature of the **extruding tool**.

7. Add two more holes to the model following the plan drawn below. Both holes should go **all the way through** the model.
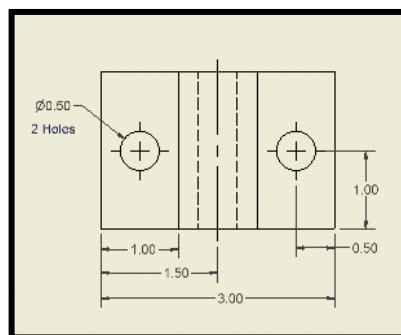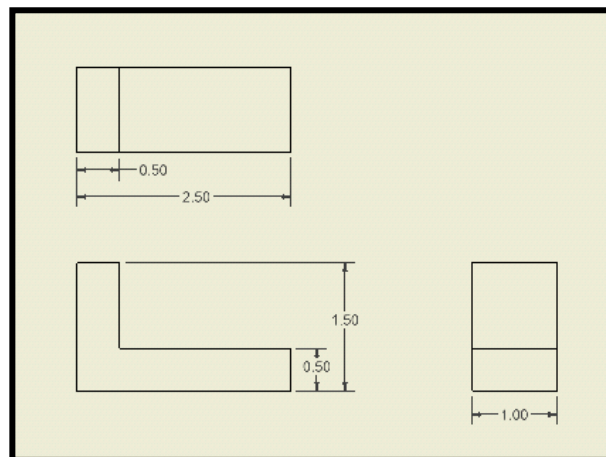
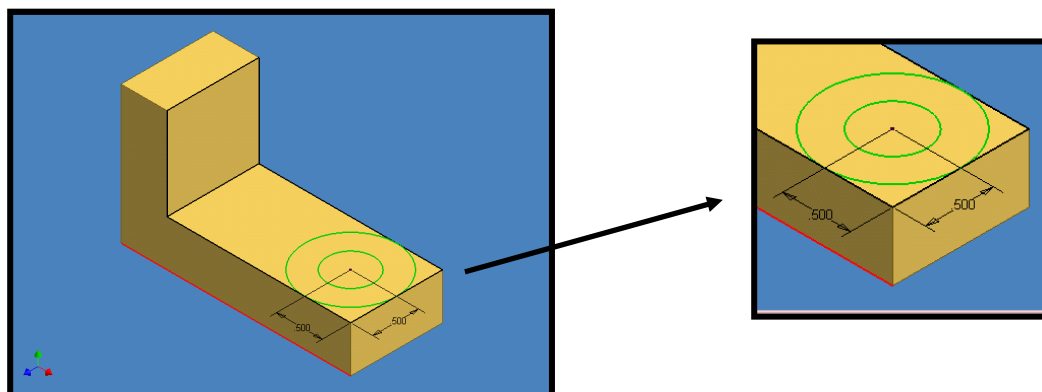8. Your completed model should look like the one below. Save the part in your server folder as IN-19.

Creating Arcs

1. Begin a new drawing with the Standard.ipt template file. Create the following model:

2. Add **circles** to the model as drawn below. The larger circle has a diamter of 1 and the smaller circle's diameter is 0.5. When you finish, **remain in sketch mode**.
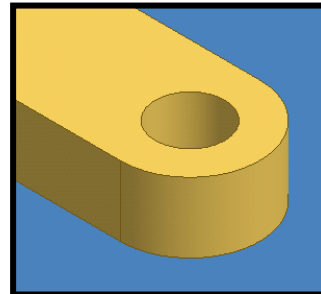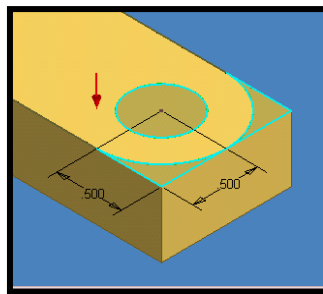
3. Select the **trim button** from the sketch tab on the ribbon and position the cursor over the back edge of the large circle. Left click to remove that section.
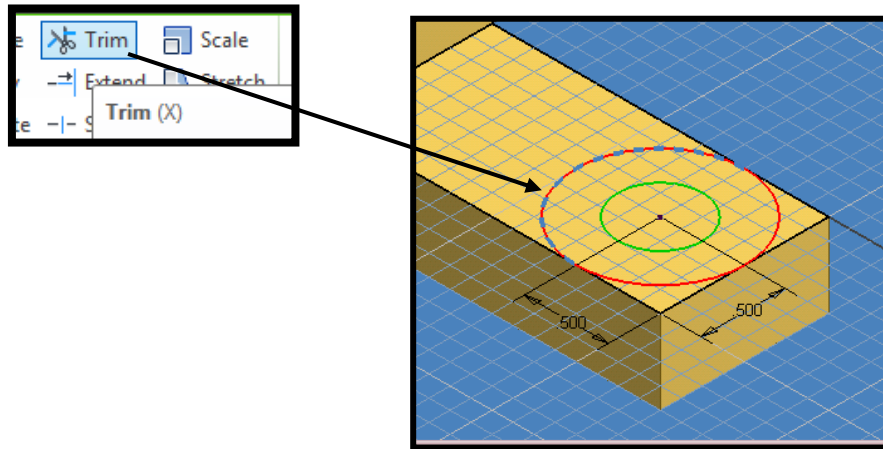
4. Finish the sketch and **extrude** the center hole and the front edge of the part. The result will be a hole and a rounded edge.


5. Create a new sketch on the back vertical face of the model. Position a circle with a diameter of .5 following the dimensions below.
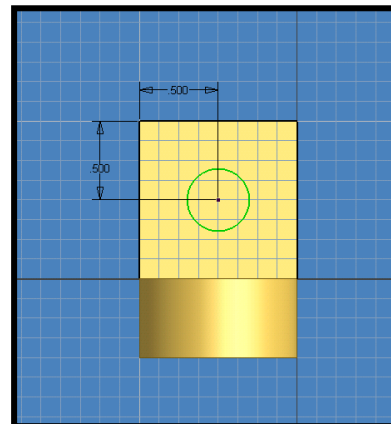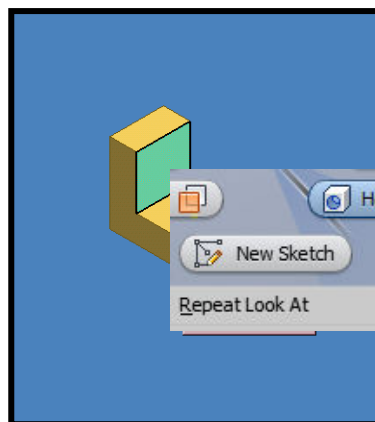
6. You used the **circle** and **trim** commands to create the curved front edge of this part in steps 2-4. Another method for creating that edge uses the **three-point arc** command.

7. Choose the **three-point arc** tool from the sketch toolbar. Position the cursor on the left edge of the plane even with the center of the circle you just drew. Click once and then click at the same location on the right edge. Finally to finish the arc, click on the top line of the plane even with the center of the circle. That completes the arc.
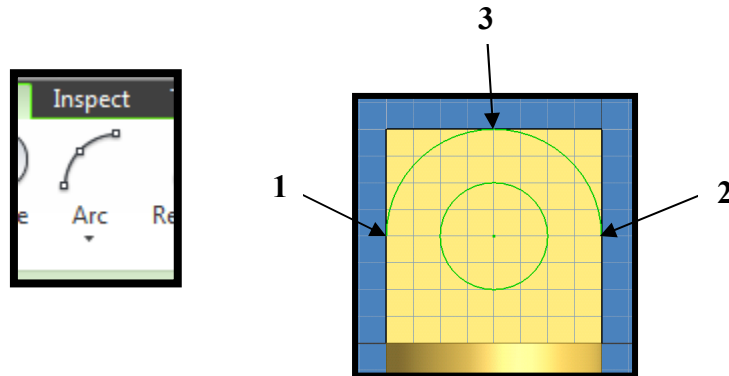

8. Cut the hole and top edge of the model with the **extrude** command exactly as you did in step 4.

9. Your completed model should look like the one below. **Save the part in your server folder as IN-20.**
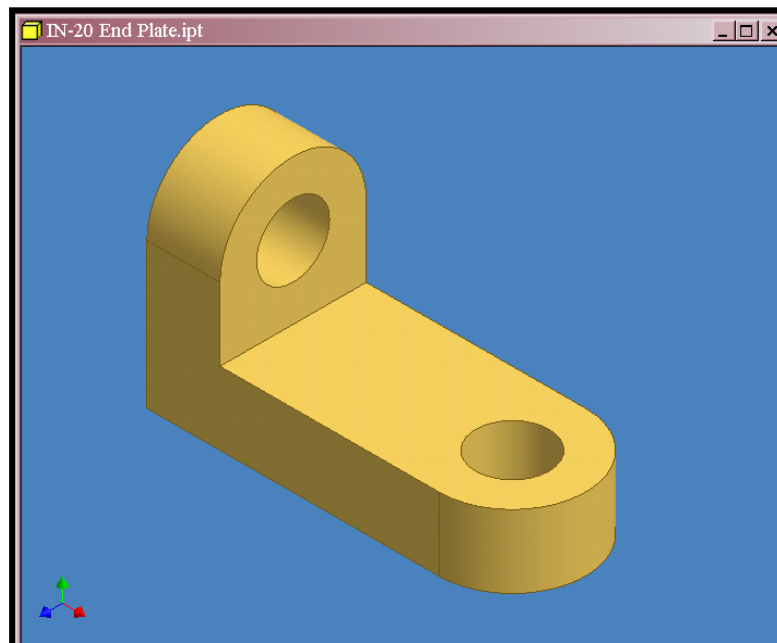
Precise Input and Construction Lines

1. Begin a new drawing with the Standard.ipt template file. Be sure the **precise input toolbar** is active. If it is not active, choose it by going to **View – Toolbars – Precise Input.**

2. Click on the **circle tool** on the sketch toolbar. Enter 0 in both the X and Y fields of the precise input toolbar. Notice that your pointer jumps to the **exact center**.
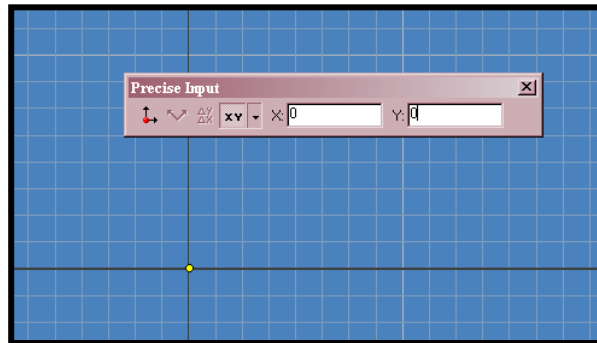

3. **Drag** out a circle. **Repeat** step 2 and create a larger circle with the same center point as the first one.

4. Set smaller circle to a diameter of **1"** and the larger circle to a diameter of **2.5"**. When you finish dimensioning, right click and select **Done**.
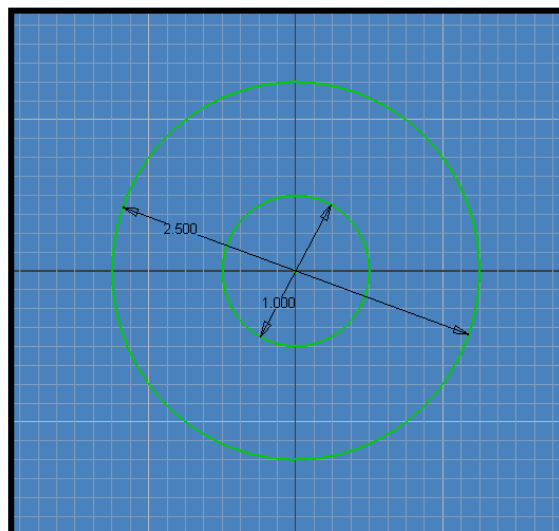
5. Locate the **style box** on the command bar and change the line style from **Normal** to **Construction**. Construction lines are used to "build with," that is to layout locations and positions for parts of the actual drawing. They are not intended to become part of the object.
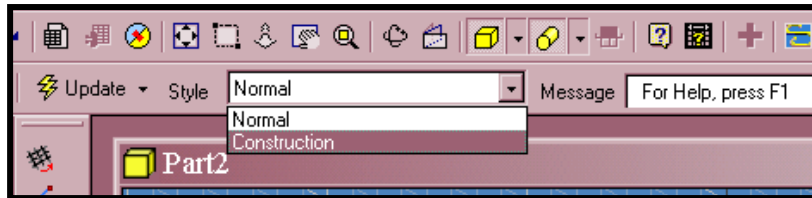

6. Use the circle tool on the sketch tool bar to draw a circle with a center point of (0,0) and a diameter of 1.75. **This circle is a construction line**. You will now use it to locate positions for 4 holes placed on the part.

7. **Change** the **line style** back to **normal** and select the circle tool again. Place 4 circles with a diameter of .25 at each **quadrant** of the construction circle you just drew (the center point of each circle will snap to the quadrant points). Use the drawing below as a guide.
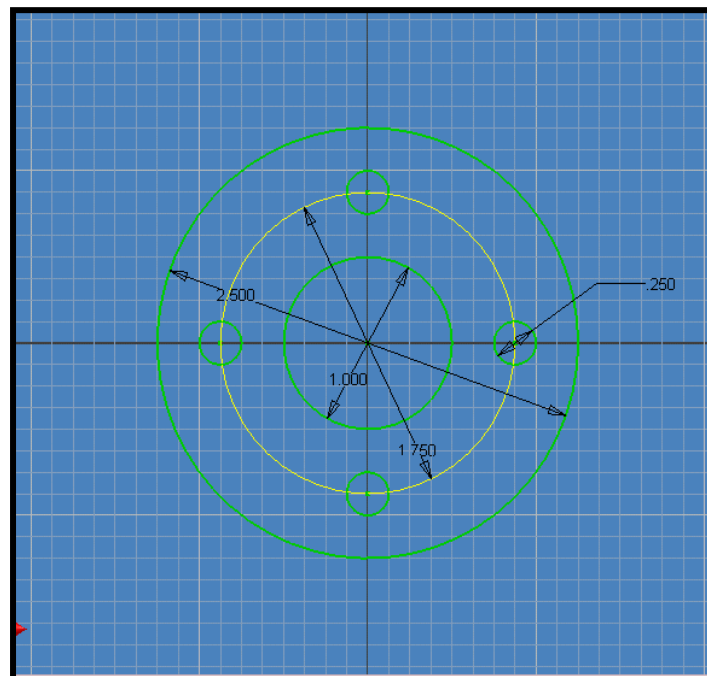
8. Finish the sketch and click on the **extrude** tool on the features tool bar. Click on the area **between** the **inner circle** and **outer circle** to select which section you want to extrude. Notice the smaller circles that are arrayed around the part will remain holes. Set the extrusion distance to .02 and complete the extrusion.
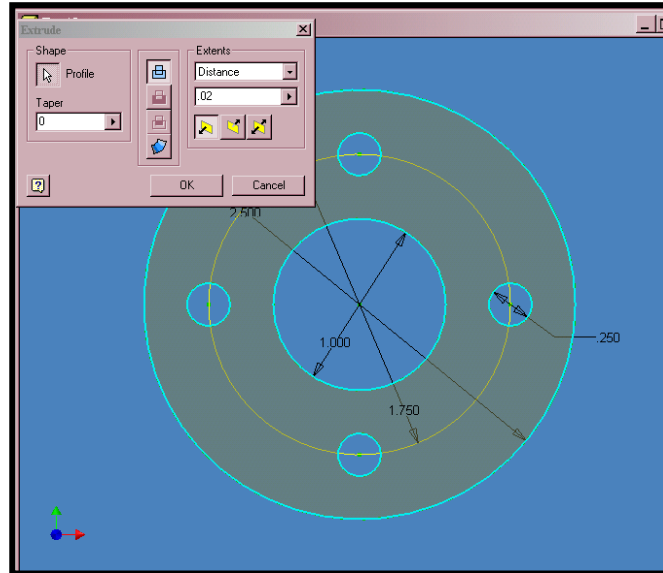
9. Your completed model should look like the one below. **Save the part in your server folder as IN-21.**
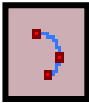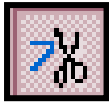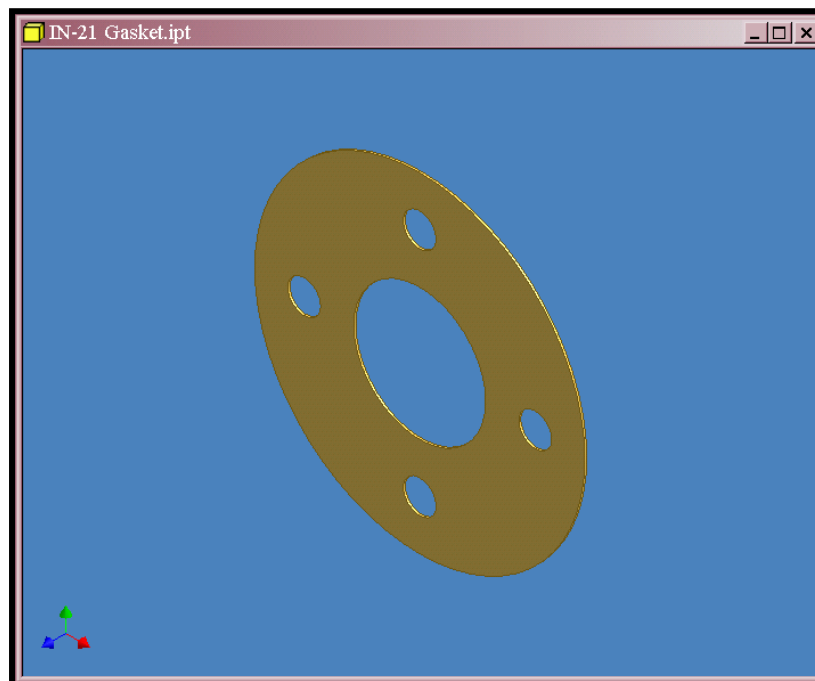
Finishing Up

1. Congratulations! You've finished three more drawings. Pick one of the drawings in this activity and create a set of engineering drawings like you did in Assignment 5.

2. Print a copy and save the drawing using the same file name as you did for the model. These drawings have a different file extension so you will not have to worry about saving over your previous work.

What Do You Know?

Load Assignment 6 of module 81.40 and answer the following questions:

1. Which tool is used to draw circles?

   a. Center Point Circle   b. Create Circle
   c. Round   d. Arc

2. Which of the following icons represents the trim command?

   a.   b.

   c.   d.

3. The two methods used to create the rounded edges in model IN-20 were:

   a. Cut & Round   b. Trim & Paste
   c. Three-Point Arc or Circle& Trim   d. Extrude and Revolve

4. Which tool can be used to locate a specific point on a drawing?

   a. Specific Input   b. Precise Input
   c. Point Input   d. Zoom

5. True or False? Construction lines are used to layout locations and positions for parts of the actual drawing.