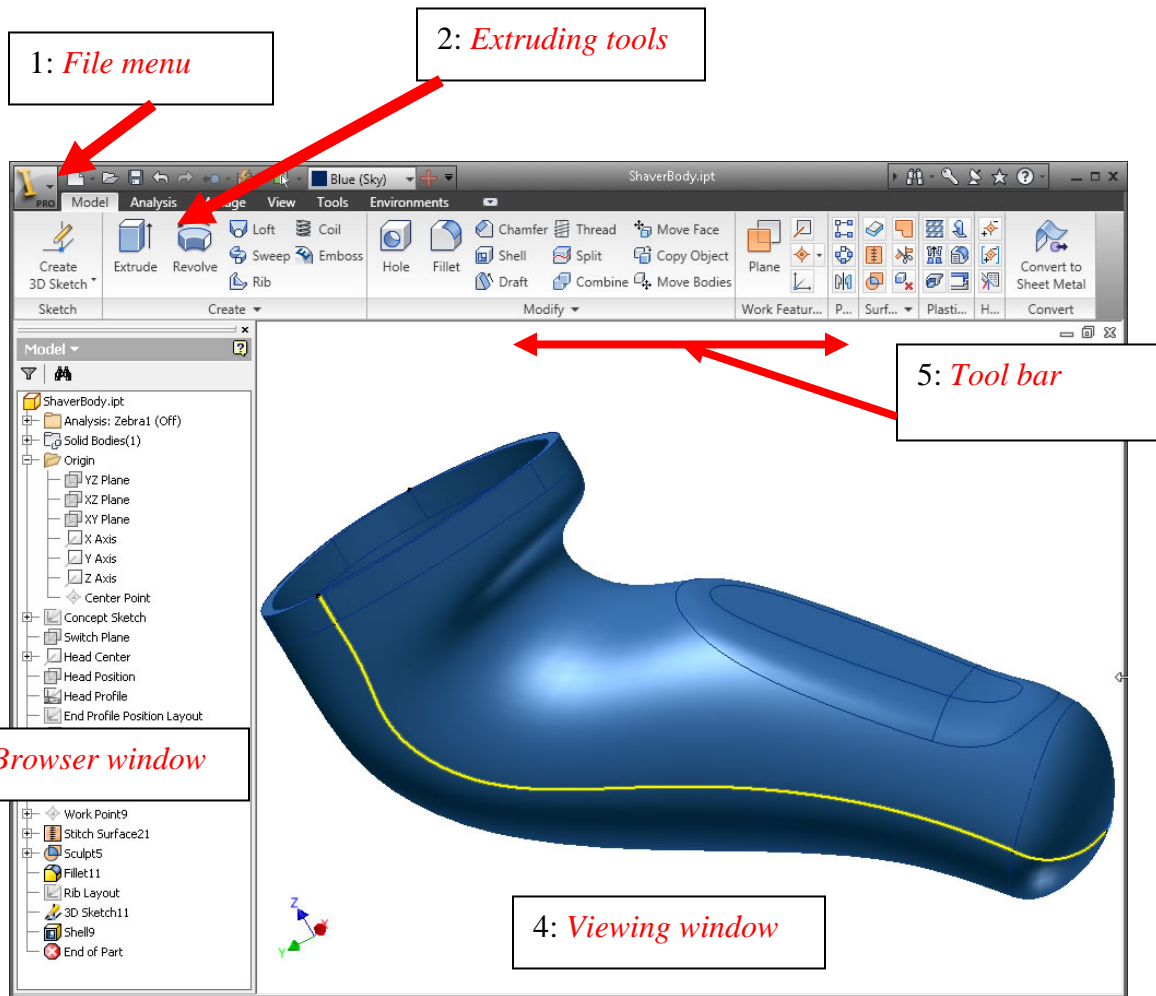


Introduction to Autodesk Inventor User Interface Student Manual

Fill in the blanks of the different tools available in the user interface of Autodesk Inventor as your instructor discusses them.

MODEL WINDOW

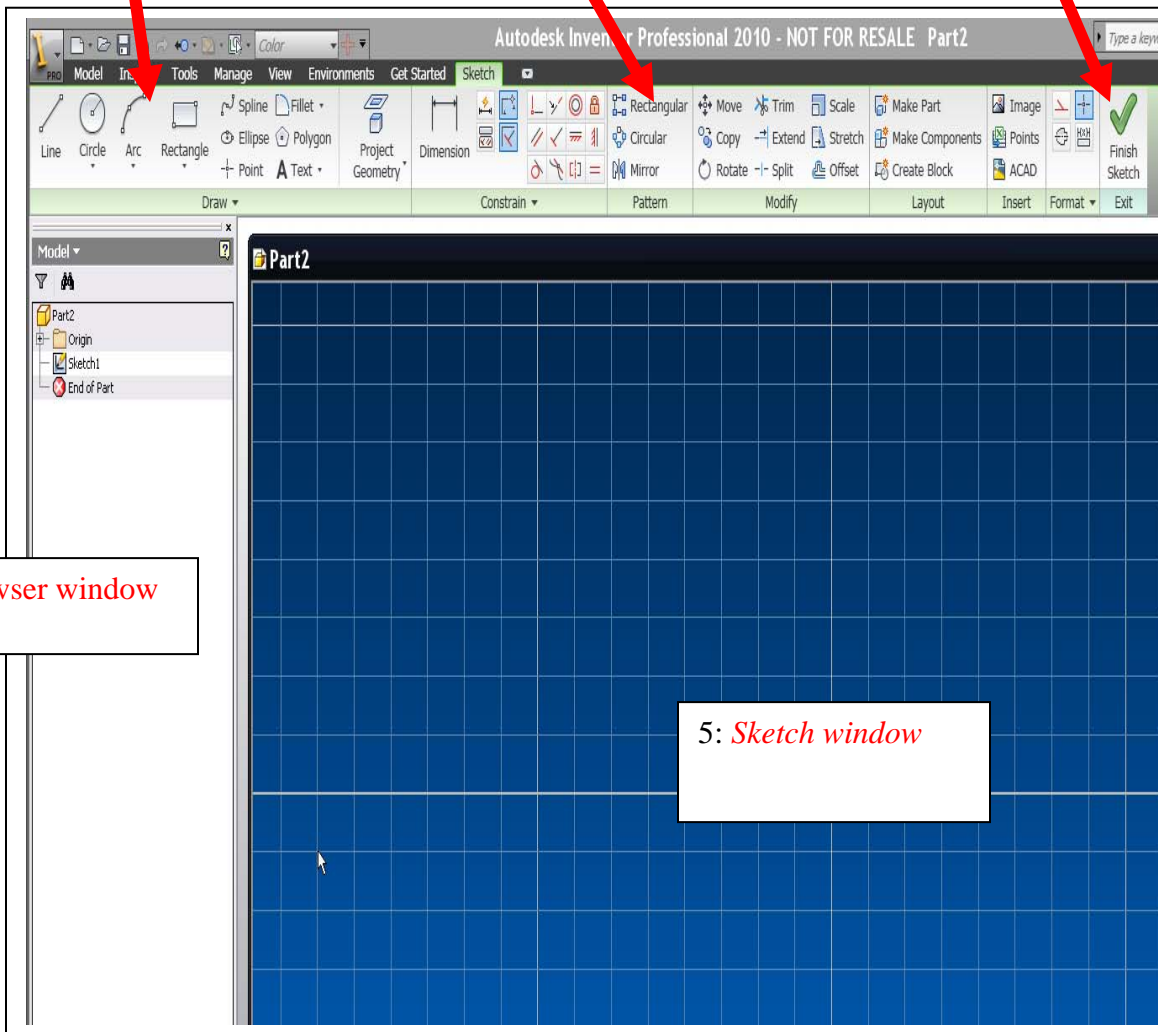


SKETCH WINDOW

1: *Drawing tools*

2: *Pattern tools*

3: *Return to model*



4: *Browser window*

5: *Sketch window*

2D Sketching

Student Manual

Materials:

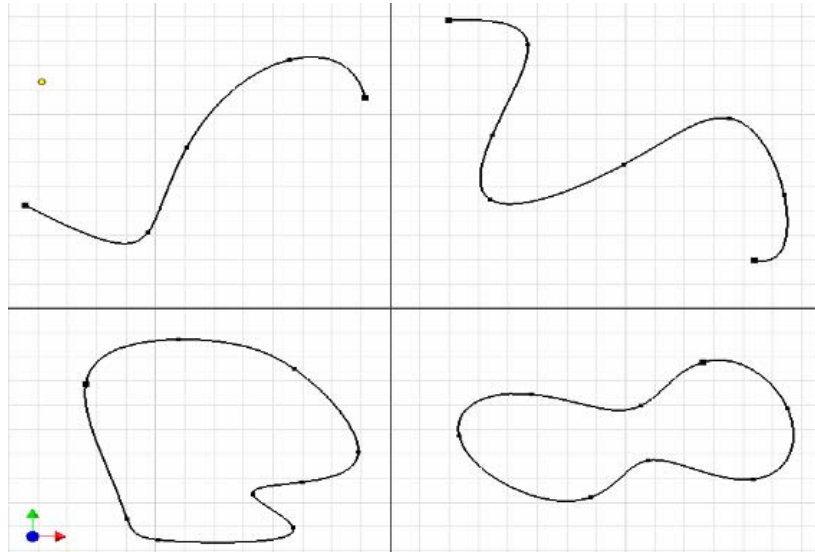
Computer with Autodesk Inventor installed

Procedure:

In order to effectively use Autodesk Inventor as a design tool, a designer must know what sketching tools are available and how they work. This activity will help you to understand and utilize the sketching tools that are available in Autodesk Inventor.

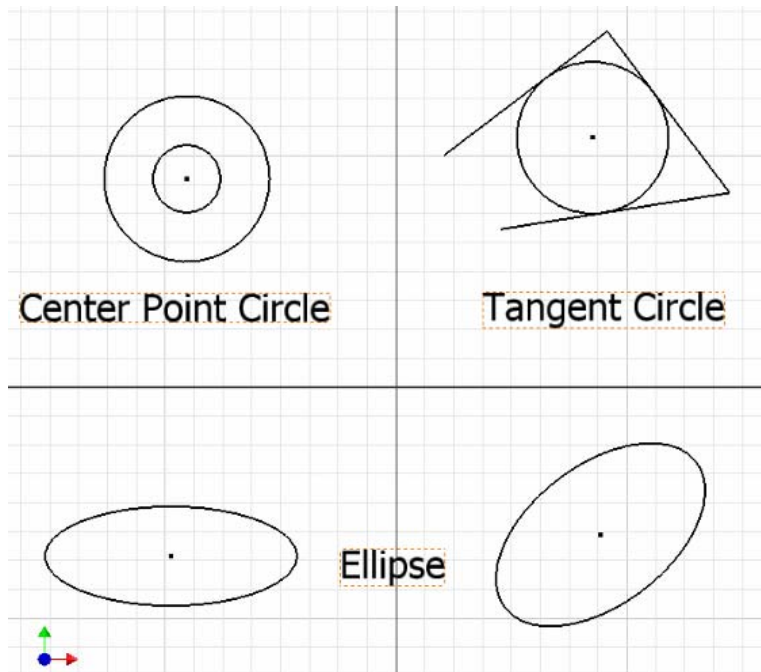
There are 13 exercises in this activity. These exercises require the creation of a new PART file, and the replication of the images pictured. As you finish each exercise, initial the graphic, save the PART file, document the file name and location on the line provided, and submit this activity to your instructor for evaluation. This exercise begins on the next page.

2D Sketching
Student Manual



The spline tool is used to create irregular curves, such as the contour of a car body surface. Create a new PART file, and use the spline sketch tool to draw two irregular curves and two closed shapes that approximate the figures pictured above. Note the locations and number of points in each spline.

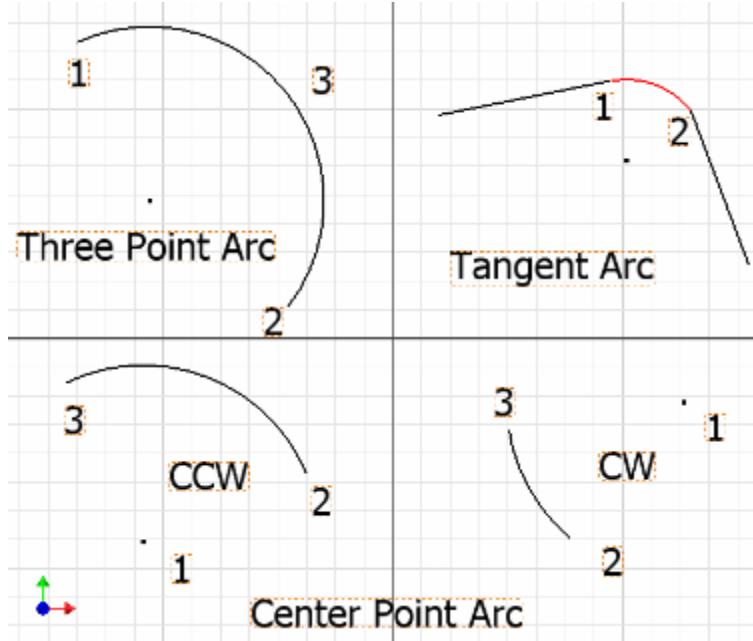
PART file name: _____



Create a new PART file and use the **circle** and **ellipse** sketch tools to replicate the figures shown above. Label the images as shown using the **text** tool.

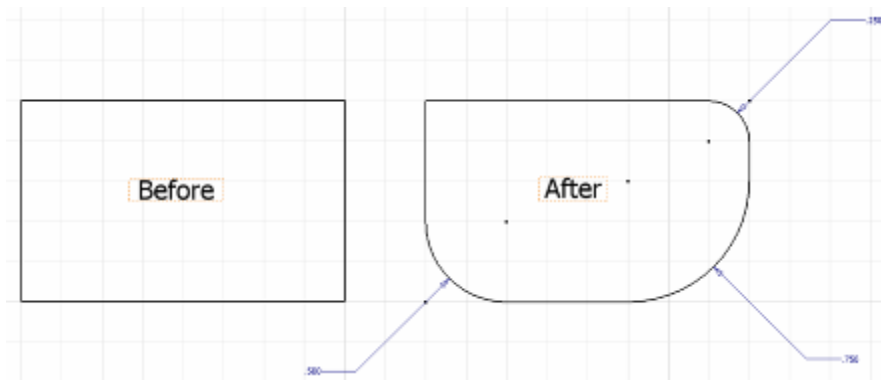
PART file name: _____

2D Sketching
Student Manual



Autodesk Inventor gives the user the ability to define arcs through several methods. These methods may include: defining the size of an arc by establishing three points of tangency, referencing two points of tangency, or identifying a center point and two points of tangency. Create a new PART file and use the line and arc sketch tools to replicate the figures shown above. Label the images as shown using the text tool.

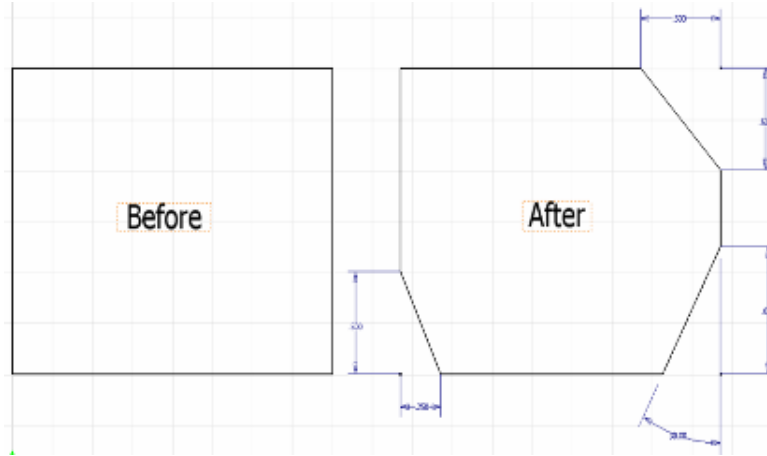
PART file name: _____



The fillet tool creates a round where two lines meet at a corner. The size of the round is identified as a radius value. Create a new PART file and draw a rectangle that is approximately 2 inches wide by 1.25 inches tall. Use the fillet sketch tool to round off the top right to .25 inch radius. Then, round off the bottom right corner with a .75 inch radius. Lastly, round off the bottom left hand corner with a .5 inch radius.

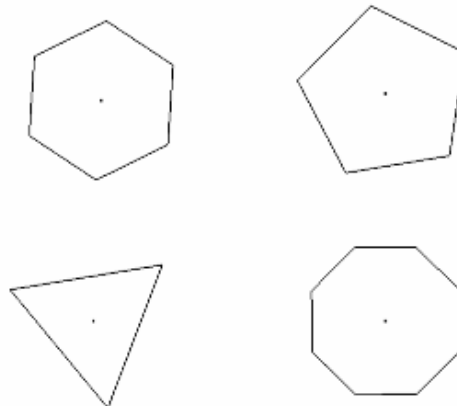
PART file name: _____

2D Sketching Student Manual



The chamfer tool is often used to “break” a corner, which results in a softer edge on a part. This process is typically done in one of three ways: by identifying the distance of one side of a 45° angle, by identifying the individual lengths of each side of the chamfer, or by identifying length of one side and the angle of the chamfer. Create a new PART file and draw a rectangle that is approximately 2 inches wide by 1.5 inches tall. Use the chamfer sketch tool and all three methods explained to create angles at the top right and bottom corners of the rectangle according to the dimensions given in the figure shown above.

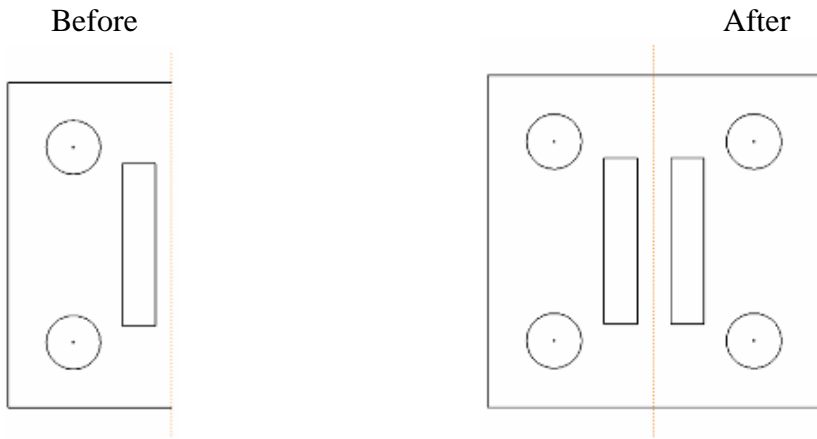
PART file name: _____



Regular polygons are multisided shapes that have sides of equal length. They may be inscribed or circumscribed within a given radius. The raw materials that are used to produce engineered objects are often manufactured in the shape of regular polygons. Create a new PART file and use the polygon sketch tool to draw the series of shapes pictured above. Use the text tool to label the names of each of the regular polygons.

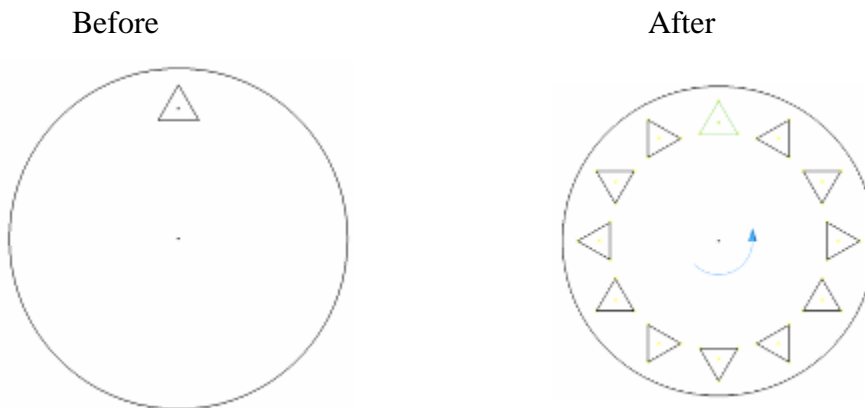
PART file name: _____

2D Sketching Student Manual



Autodesk Inventor allows the designer to mirror images across lines, which is a useful tool when designing parts that have high degrees of symmetry. Create a new PART file and use the line and circle sketch tools to create a similar figure to the one shown in the Before image. A regular vertical line may be used as the mirror line. The top and bottom horizontal edges must terminate at the vertical mirror line. Use the mirror sketch tool to mirror the figure across the mirror line.

PART file name: _____



The ability to pattern a shape or element allows the designer to save time and maintain accuracy. Create a new PART file and draw a circle with a diameter of approximately 2.25 inches. Use the polygon sketch tool to create an isosceles triangle that would fit within a .25 inch diameter circle. Orient the triangle so that it is pointing toward the top quadrant of the circle. The center of the triangle should be approximately 7/8 inch from the center of the circle. Use the pattern sketch tool to create a copy of the triangle 12 times (the number of instances includes the object being patterned) around the center of the circle.

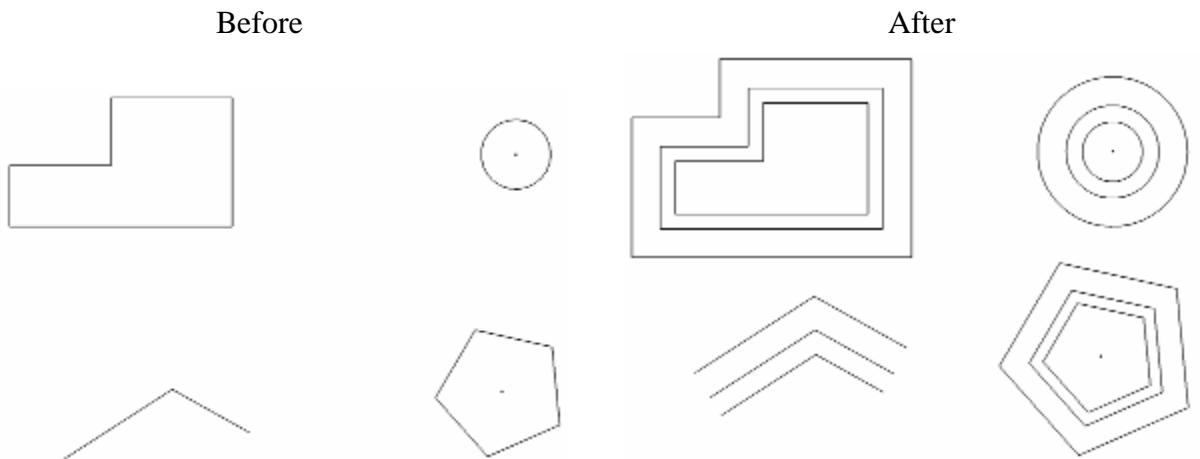
PART file name: _____

2D Sketching Student Manual



The Pattern sketch tool allows the designer to create a pattern from one or several objects. The direction or orientation of the pattern is derived from existing lines on the sketch. Create a new PART file and draw a rectangle that is approximately 4 inches wide by 3.25 inches tall. Create a $3/8$ inch diameter circle in the lower left hand corner. Locate the center of the circle approximately $3/8$ inch from the bottom and left edges. Use the pattern sketch tool to create multiple copies of the circle. The circle pattern must have seven columns and six rows, and fit within the boundaries of the rectangle.

PART file name: _____

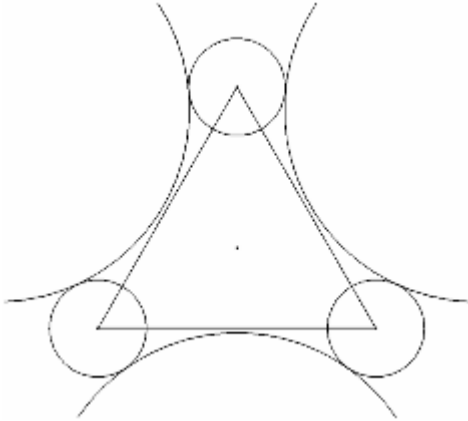


The need to create geometry that is identical in shape and parallel is very common in engineering design. The **Offset** sketch tool is used to make this process quick and accurate. Create a new PART file and draw the figures pictured in the before image. Use the Offset sketch tool to offset the geometry of each figure outward two times.

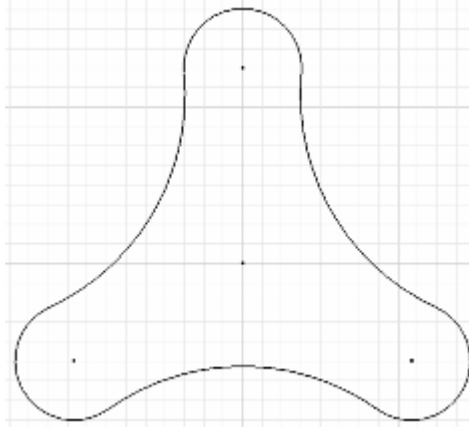
PART file name: _____

2D Sketching
Student Manual

Before



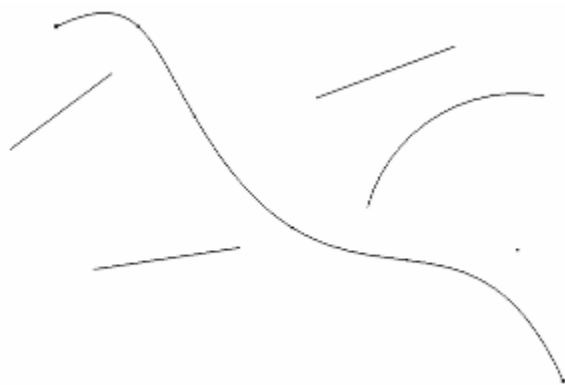
After



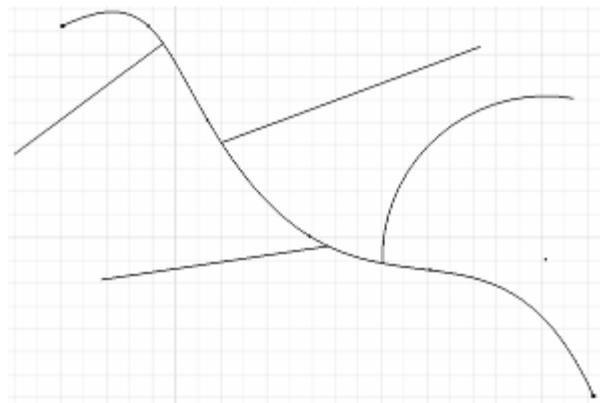
Create a new PART file and draw the figures pictured in the before image. Use the **Trim** sketch tool and delete keyboard function to make the before image look like the after image.

PART file name: _____

Before



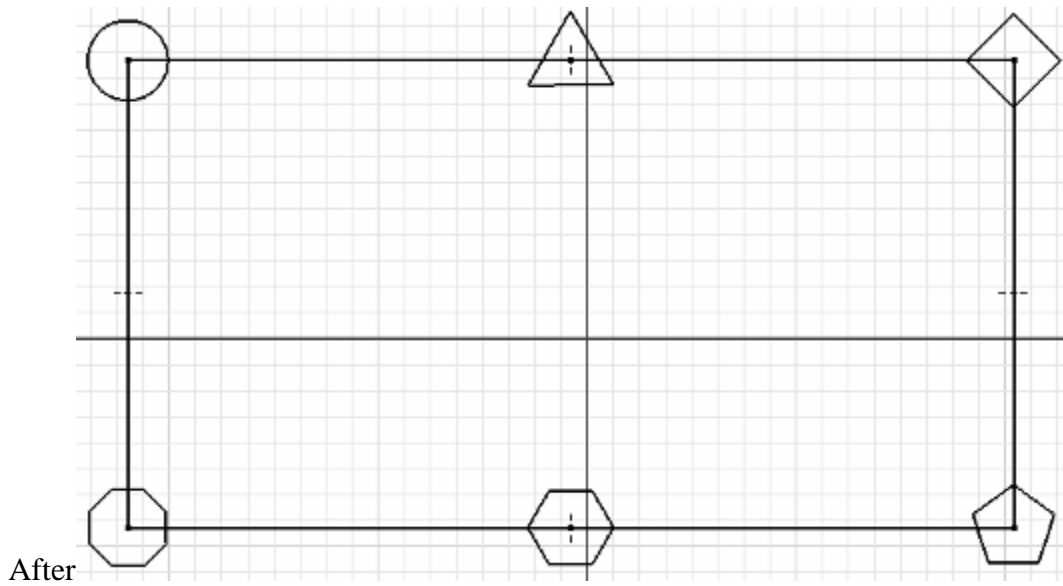
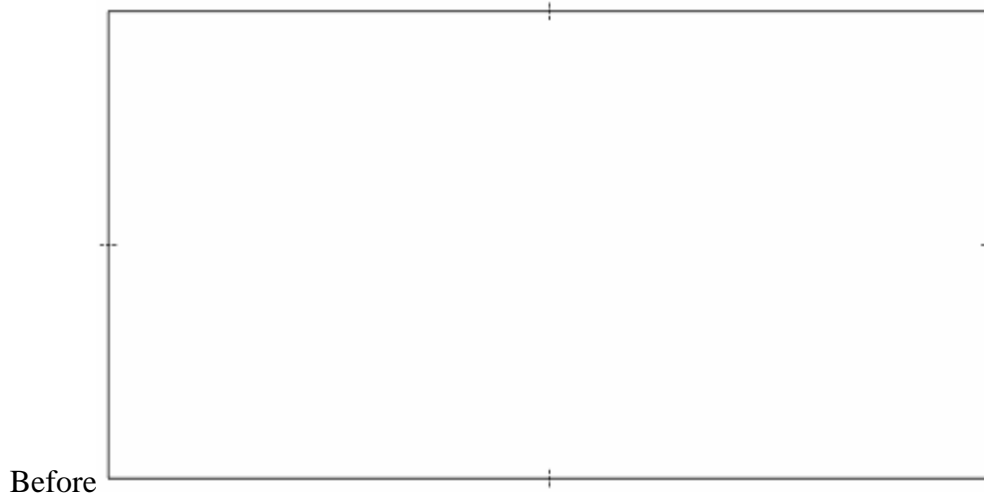
After



Create a new PART file and draw the figures pictured in the before image. Use the Extend sketch tool to extend the straight lines to the spline. When finished, the sketch should look like the after image.

PART file name: _____

2D Sketching
Student Manual



Create a new PART file and draw the figures pictured in the before image. Use the Move sketch tool to move the geometric shapes to the positions shown in the after image.

PART file name: _____

3D Modeling

Student Manual

Equipment

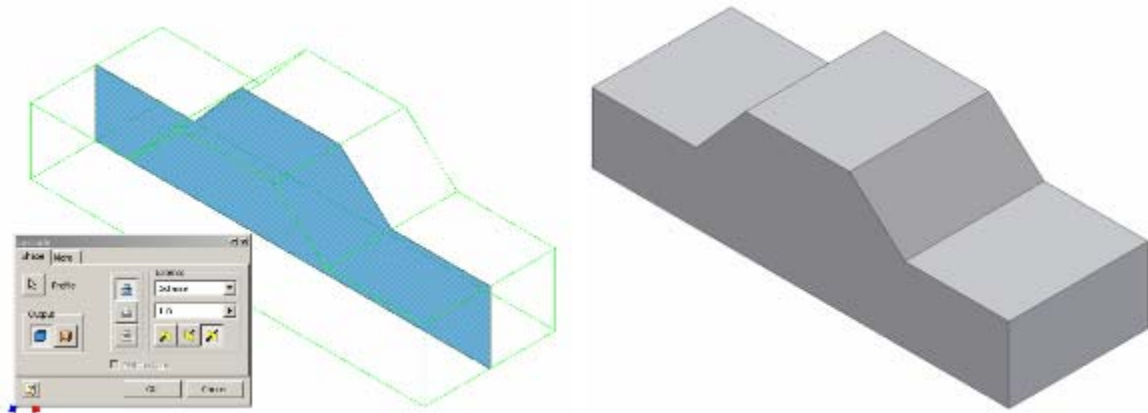
- Computer with Autodesk Inventor installed
- PART files located on Student Resource CD:
 - Mid-plane
 - Bushing
 - CircularPattern
 - RectangularPattern
 - Loft
 - Threaded rod
 - Fillets-chamfers
 - Left half

Procedure

This activity will help you to understand and utilize the feature tools used in Autodesk Inventor.

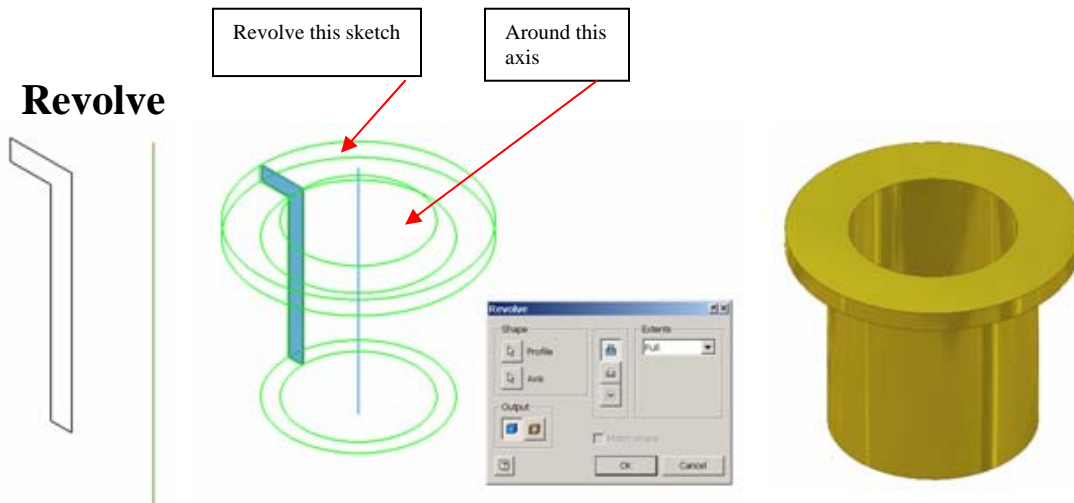
There are 8 exercises in this activity which represent the different feature tools available to use in Autodesk Inventor. You will need to open the files provided from the student resource CD. As you finish each exercise, initial the graphic, save the PART file, document the file name and location on the line provided, and submit this activity to your instructor for evaluation.

Mid plane Extrusions



The mid plane extrusion function will join, cut, or intersect the selected sketch equal distances in both directions from the selected profile. Open the file Mid-plane, and perform a 1 inch mid plane extrusion on the sketch. (note: the extrusion can occur in one direction or the other if a mid plane is not desired) Save the file as a different name.

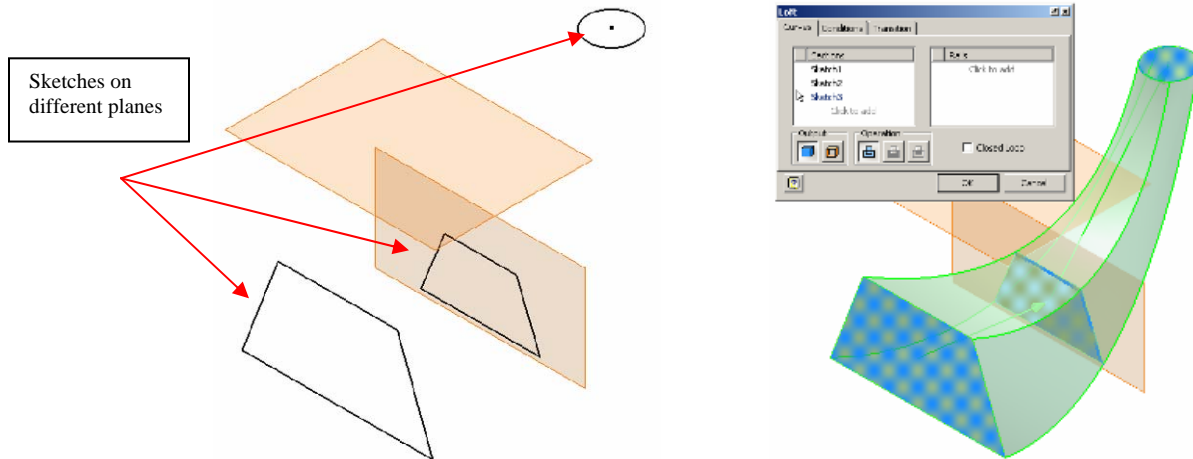
PART file name: _____



Revolve is a function that allows the user to extrude a closed profile around a fixed axis up to 360°. The axis can be part of the profile, an existing edge on a part, or one of the axes of the Cartesian coordinate grid. Grid axes may be selected from the Origin folder located in the Browser bar. Open the file called Bushing. Use the revolve function to revolve the sketch around the existing axis a full 360°. Save the file as a different name.

PART file name: _____

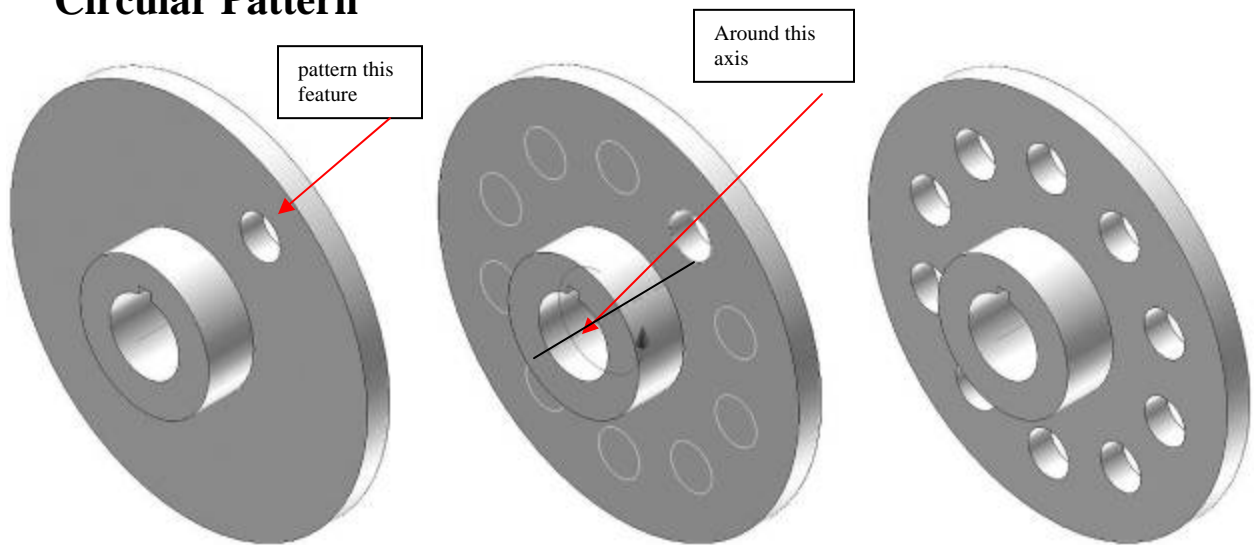
Loft



The loft function allows the user to create a solid by blending two or more closed sketches that are located on different planes. Open the file called Loft. Use the loft function to blend the three sketched profiles into one solid object. Save the file as a different name.

PART file name: _____

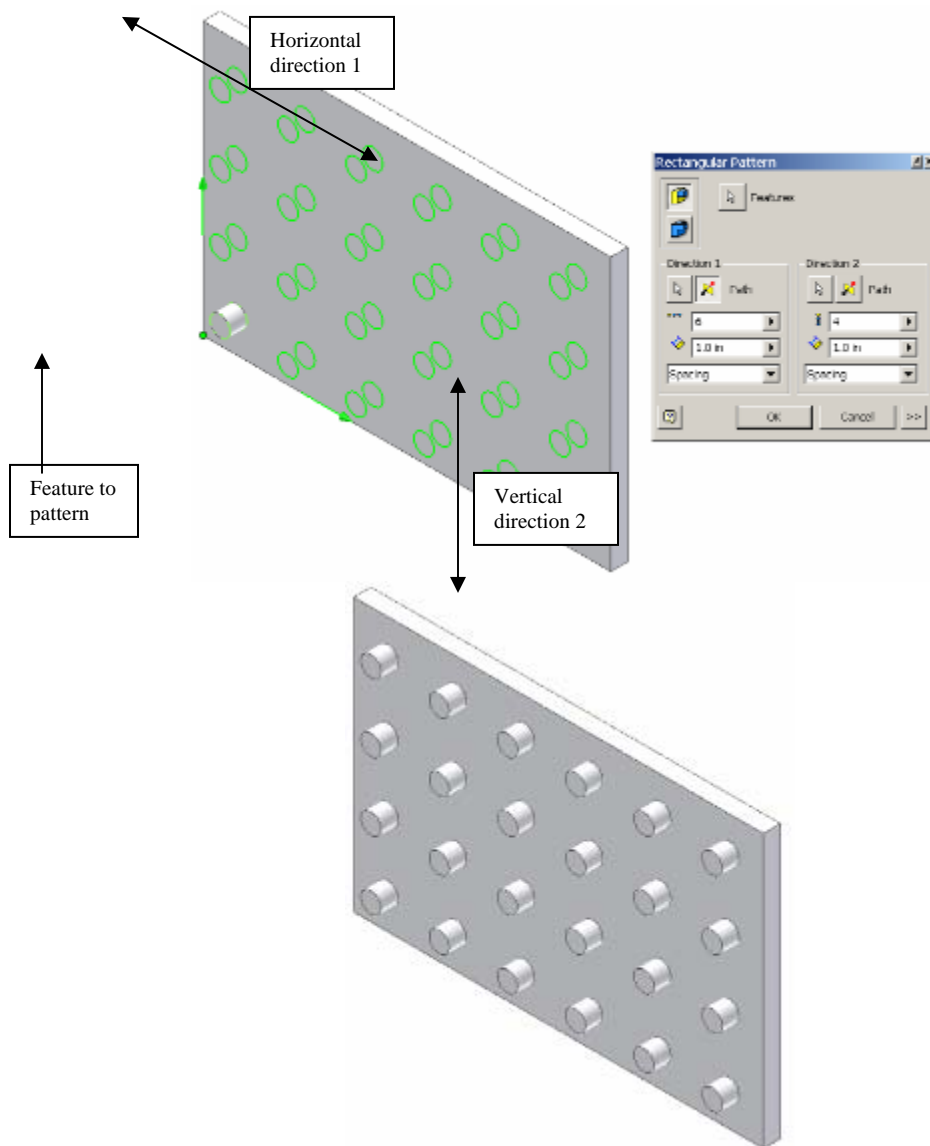
Circular Pattern



The pattern function allows the user to make multiple copies of an existing feature in one of three ways. A circular pattern is often used to pattern a hole around a center axis. An edge on an existing feature can also serve as the center axis. Open the file called CircularPattern, and use the circular pattern function to copy the existing hole on the flange plate a total of 10 times (the first hole must be represented in the count) around the existing axis. Save the file as a different name.

PART file name: _____

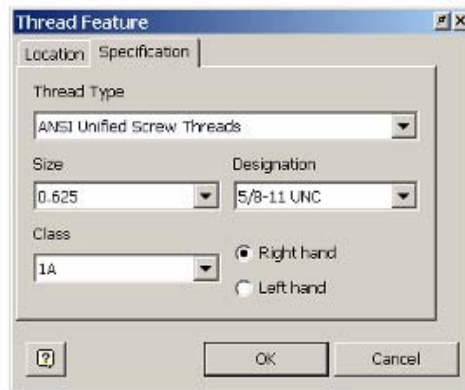
Rectangular Pattern



The rectangular pattern function allows the user to make copies of an existing feature in one direction, or two directions simultaneously. Existing edges must be selected to identify the desired direction(s). Open the file called RectangularPattern. Use the rectangular pattern to copy the existing cylindrical extrusion six times in the horizontal direction and four times in the vertical direction. Save the file as a different name.

PART file name: _____

Thread

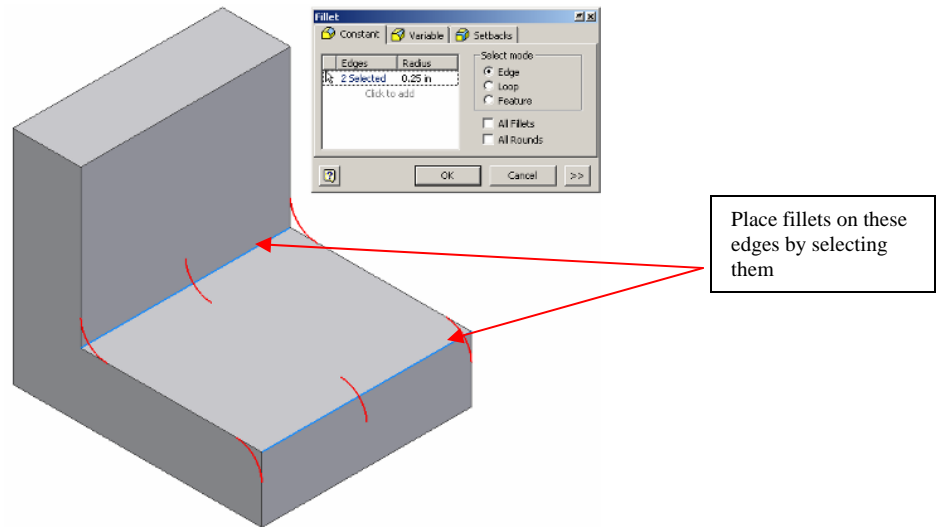


Click on the face of the cylinder

The thread function allows the user to simulate the appearance of threads on the curved surface of either a cylinder or a hole. This will aid in calling out the dimensions later in the documentation process. The diameter of the cylinder or hole must match the nominal diameter of the desired thread form. Open the file called ThreadedRod. Use the thread function to place a right-hand 5/8-11 UNC thread along the entire length of the cylinder's curved face. Save the file as a different name.

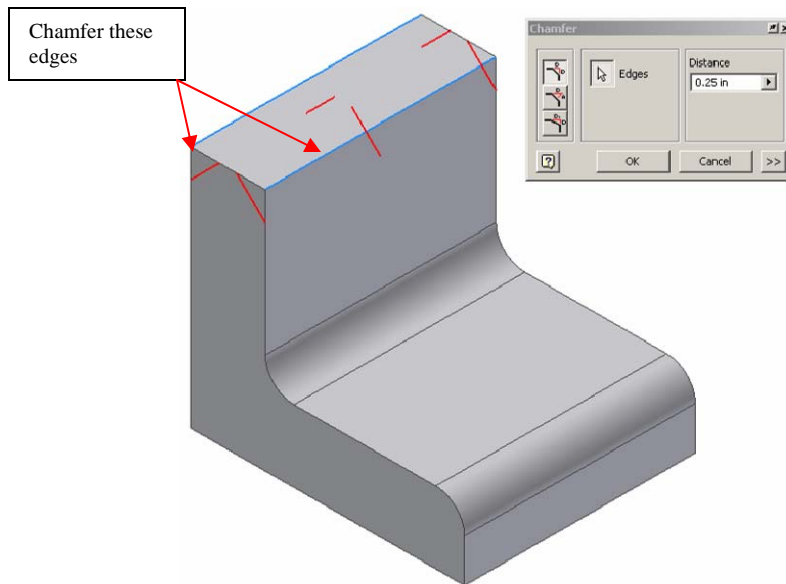
PART file name: _____

Fillet



Fillet is a function that allows the user to create a rounded edge where two surfaces meet to form an edge. It should be noted that on an exterior corner, the resulting feature is known as a round. On an interior corner, the resulting feature is known as a fillet. The fillet tool is used for both features. Open the file called Fillets Chamfers. Use the fillet function to apply a .25 radius to the corners shown above. This model will be used in the next exercise.

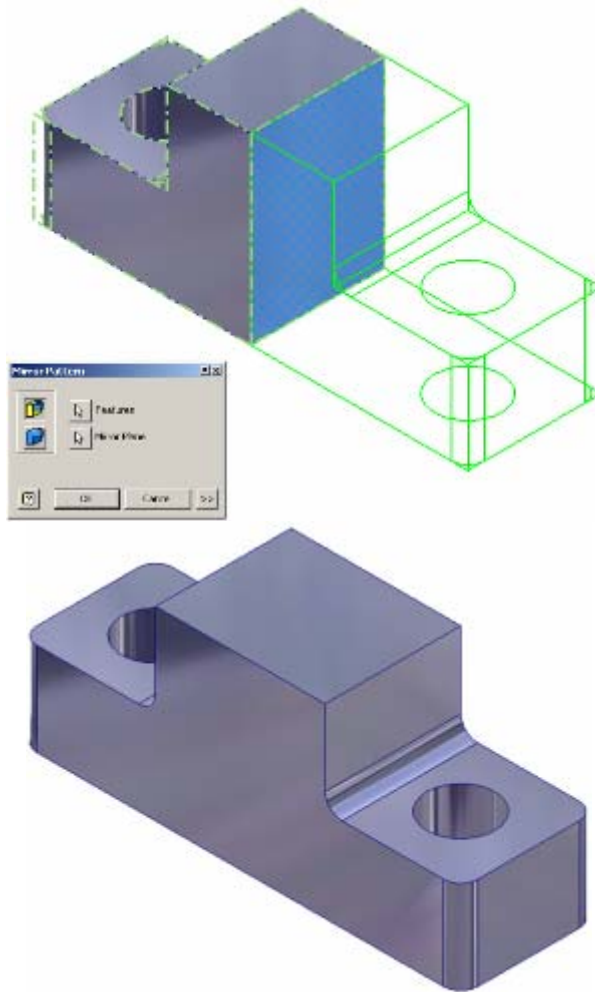
Chamfer



Chamfer is a function that allows the user to apply an angle surface where two existing surfaces meet to form an edge. Use the chamfer function to apply a .25 inch x 45° chamfer to the edges shown above on the file from before. Save the file as a different name.

PART file name: _____

Mirror



Mirror is a function that allows the user to create a mirror image of existing geometry. This function requires an existing feature(s) and a surface or work plane to serve as the mid-plane of symmetry. Open the file called LeftHalf. Use the mirror function to add a duplicate mirror image of the existing geometry on the other side of the right face of the object. Save the file as a different name.

PART file name: _____

Assessment

Student Guide

Materials:

Computer with Autodesk Inventor installed
Student Manual

Procedure:

In this assessment, you will create computer models of different objects. You will write step-by-step instructions on how to create one of your objects and give them to a classmate to create from your instructions. The teacher will assign the object for which you will need to write the directions.

- Create nine of the eleven objects located below using Autodesk Inventor.
- Create a set of step-by-step instructions on how an object was created using Autodesk Inventor. The instructor will determine the object for which you will write the instructions.
- Trade your instructions with a classmate and create his or her object.

