

## Activity 5.5b CAD Model Features Part 2

### Introduction

Two dimensional sketches are nice, but parts have three-dimensional (3D) qualities that sketches can only imitate and communicate in an abstract manner. CAD 3D solid modeling programs can bridge the gap between 2D representations and 3D artifacts by providing an electronic representation of an object that can be moved and rotated so that all aspects of the object can be viewed.

A sketch in a 3D Computer Aided Design (CAD) solid modeling program serves as the foundation for a three-dimensional feature. Some three-dimensional features require a sketch from which the 3D form is created. Other features do not require a sketch, but instead require a three-dimensional form.

In this activity you will learn about and use additional tools and features within a CAD program to enhance your knowledge of the capabilities of the program and improve your skill in producing 3D solid models.

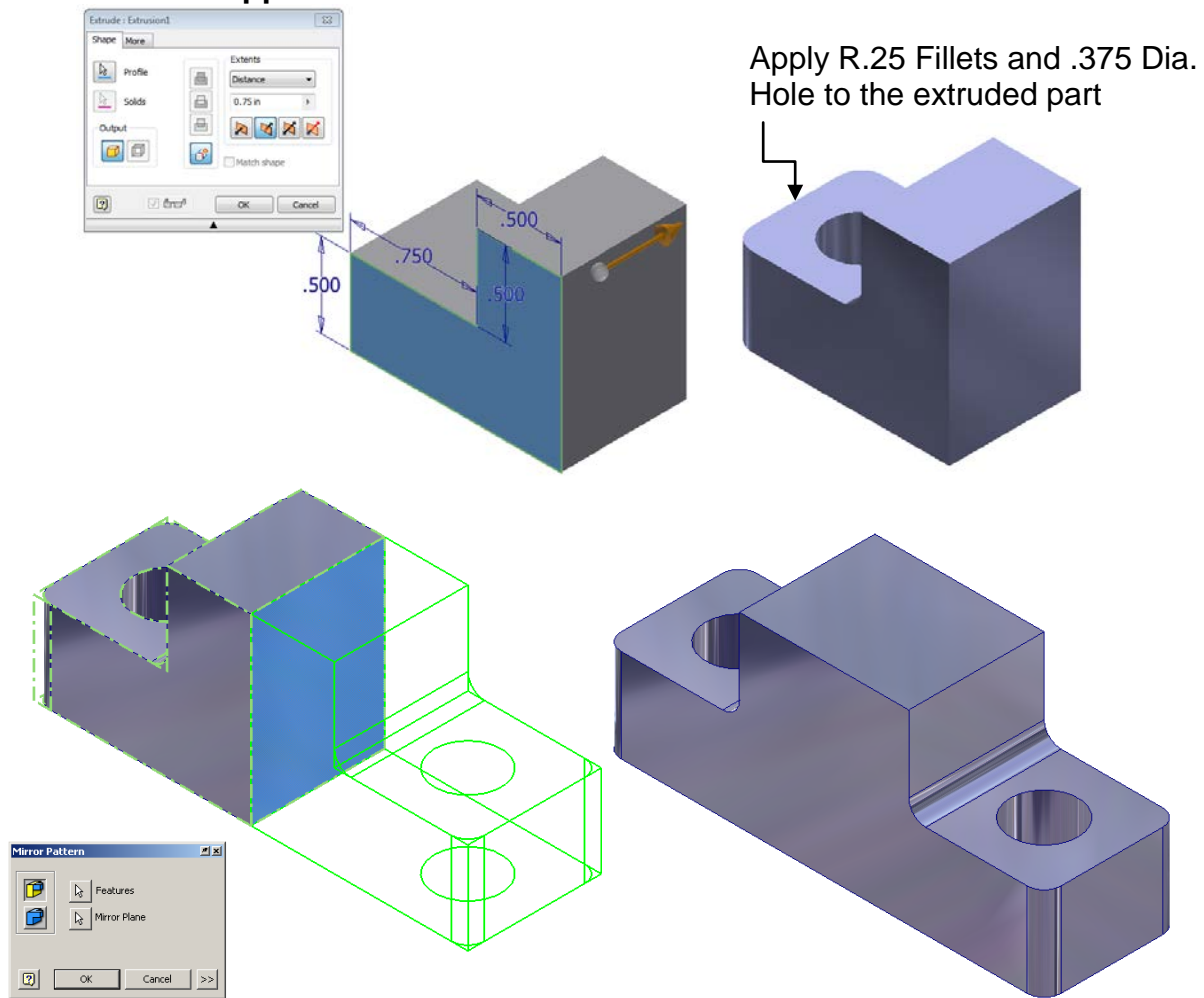
### Equipment

- Computer with 3D CAD modeling software

### Procedure

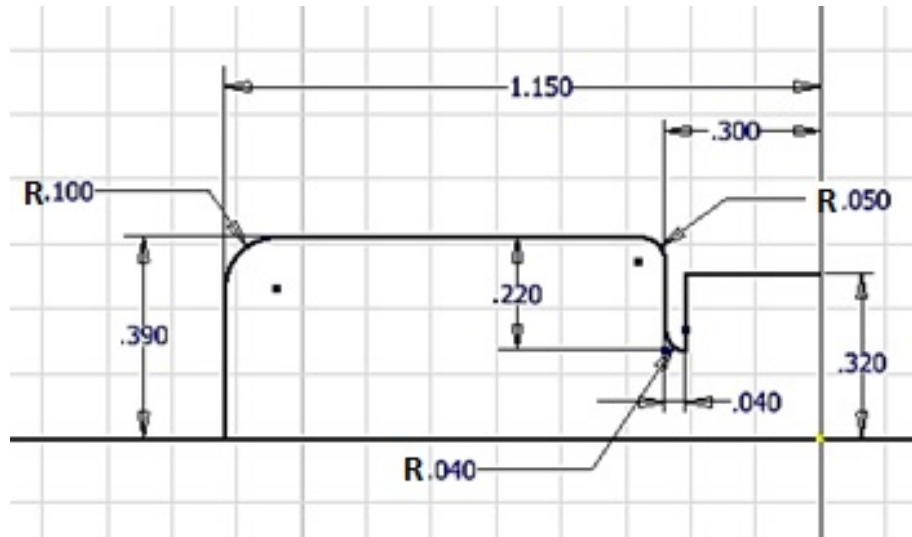
In order to effectively use a CAD program as a design tool, a designer must know what model features are available and how they work. This activity will help you to understand and utilize the feature tools that are common to most CAD programs.

## Mirror – Feature Application

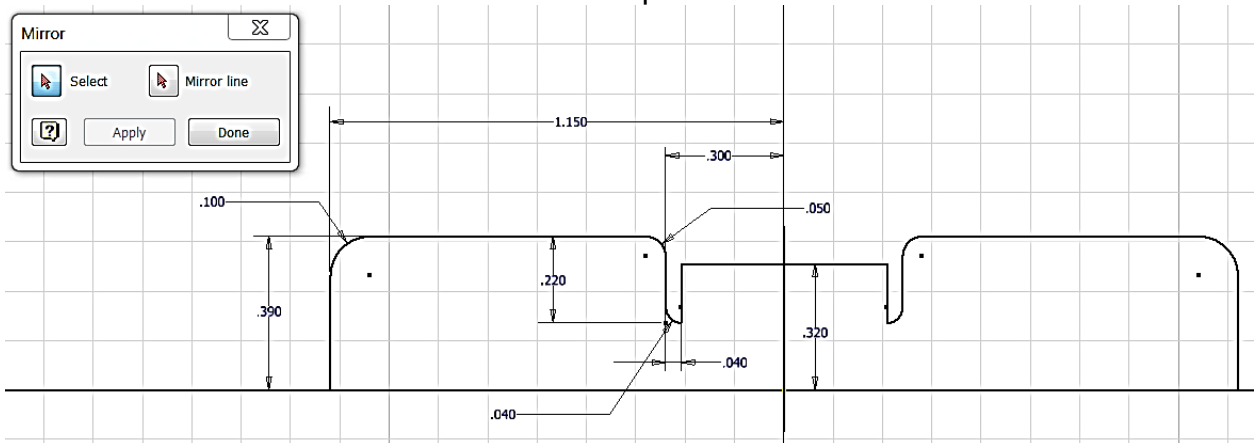


- I. Create a part according to the above images. Mirror is a function that allows the user to create a mirror image of existing geometry. This function requires an existing feature(s) and a surface or work plane to serve as the mid-plane of symmetry.
  1. Create the part using the provided sketch and extrude to a depth of .75. Add the fillets and hole as shown in the figure above.
  2. Use the mirror function to apply a mirror image of the existing geometry to the right face of the object as shown.
  3. Save the file as *Mirror.ipt* to your Practice project folder.
- II. Create a solid model to represent the Automoblox connector piece. This part can be modified later to include more details and features.
  1. Using your measurements (or the dimensions provided below), sketch the outline profile for one quarter of the face that includes the long rectangle as shown below. Sketch one quarter of the outline profile of the connector piece. **Do not sketch the rectangular hole.**

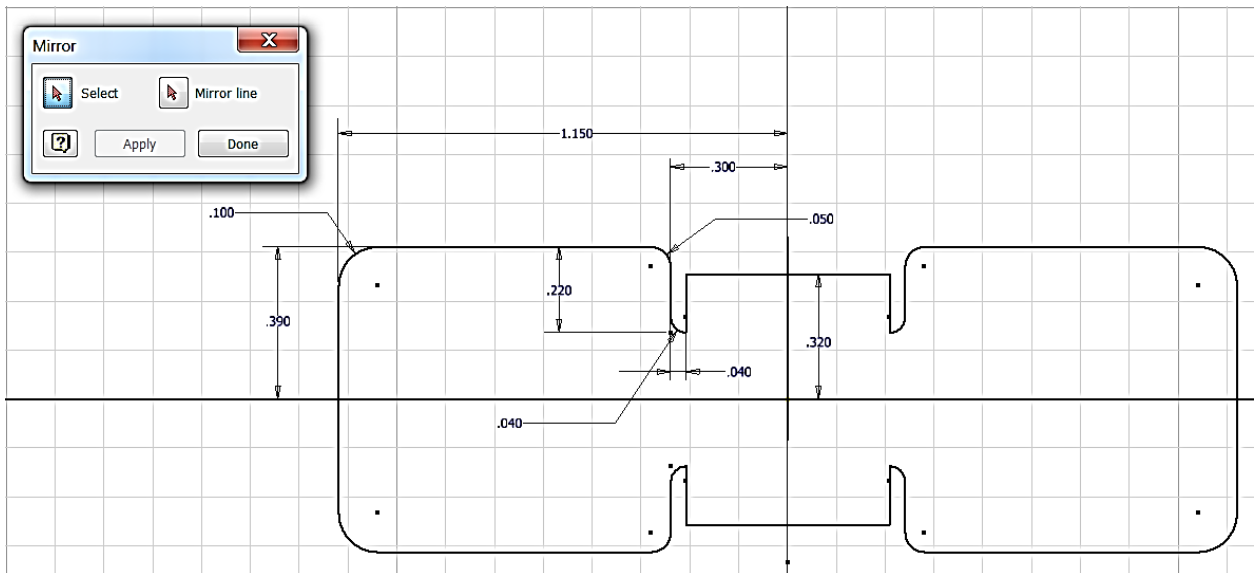
Face with long rectangle



2. Mirror the sketch to create half of the profile.

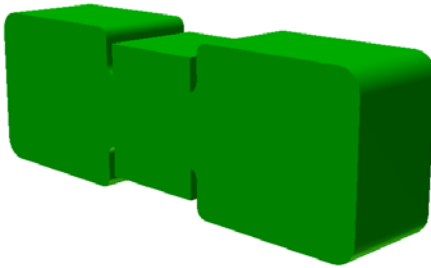


3. Mirror the sketch a second time to complete the section.



4. Extrude the sketch the appropriate depth according to your measurements. The connector will resemble the solid model below.

Note that if you have trouble extruding the sketch, you may need to use the Trim tool or Close Loop feature within the sketch to create coincident endpoints between adjacent lines and arcs. To close an open sketch loop, right click on a line and choose close loop, then follow the instructions.



5. Save the file as Connector*YourInitials*.ipt in your Automoblox project folder.

## Extrude

**Sketch Tools**– These are suggested tools for this activity

- **Rectangle**
- Construction Line
- **Ellipse**
- **Project Geometry**
- **Offset**
- **Slice Graphics** (F7 and right mouse click functionality)
- **Arc**

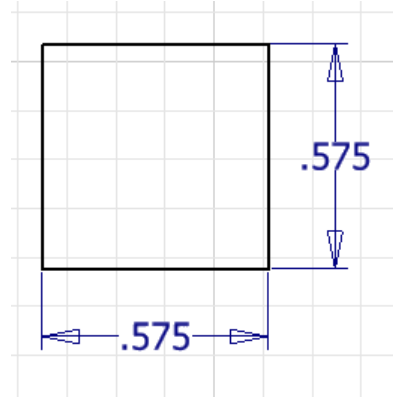
**Feature Tools** – These are suggested tools for this activity

- **Extrude** (Add/Join)
- **Extrude** (Tapered / Intersection)

- **Work Axis**
- **Extrude** Intersection (Mid-plane or Symmetric)
- **Revolve**

IV. Create the body of the male Automoblox figure with a square base.

1. Open a *standard.ipt* and create the sketch that is shown below.



2. Extrude the profile 0.30 inches as shown.

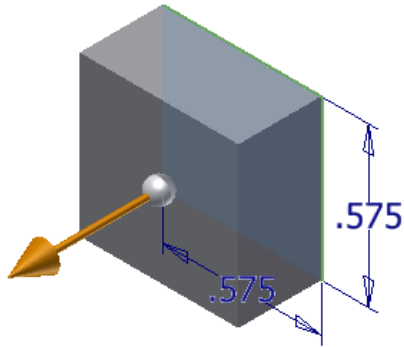
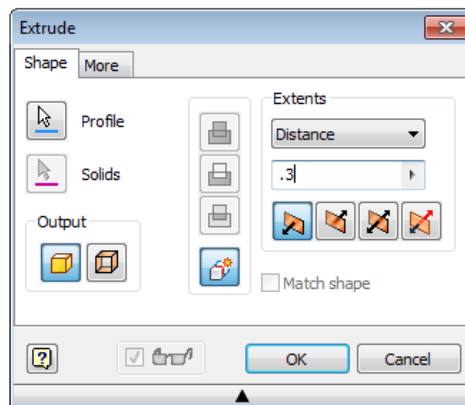
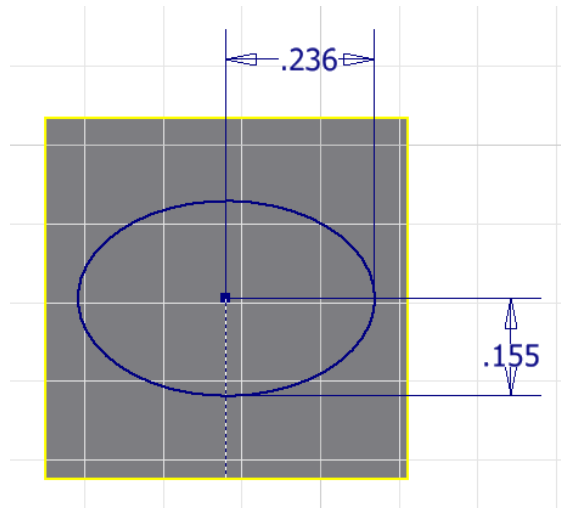


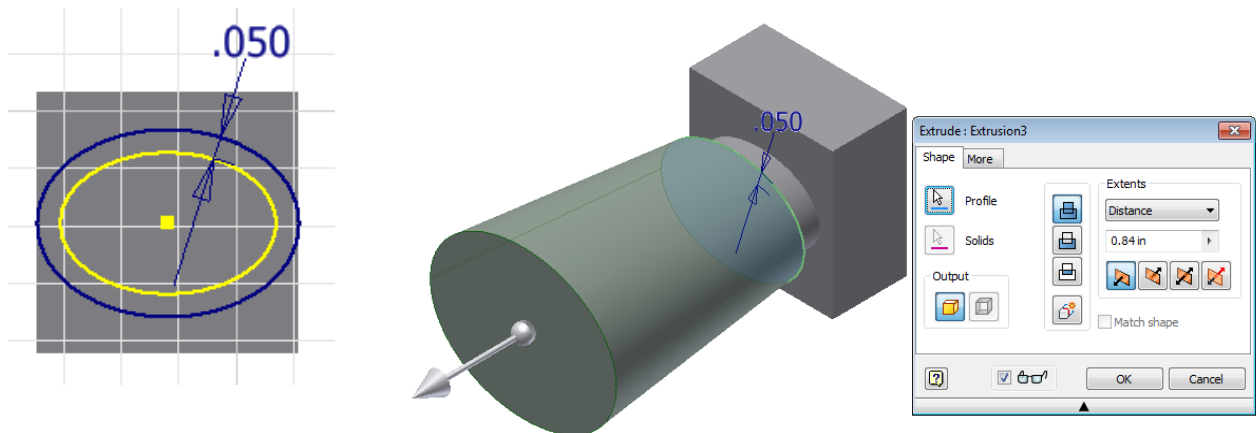
Fig. 2



3. Place a sketch on the top of the square base and sketch an ellipse. Apply the dimensions as shown in the image. Be sure to use a construction line to locate the ellipse in the center of the block. Extrude the ellipse to a value of .115.



4. Use a tapered extrusion to approximate the body of the figure. Use appropriate taper angle and extrusion depth based on approximations or measurements that result in a realistic visual representation. The part does not need to be created to exact dimensions as it is a visual representation used to develop



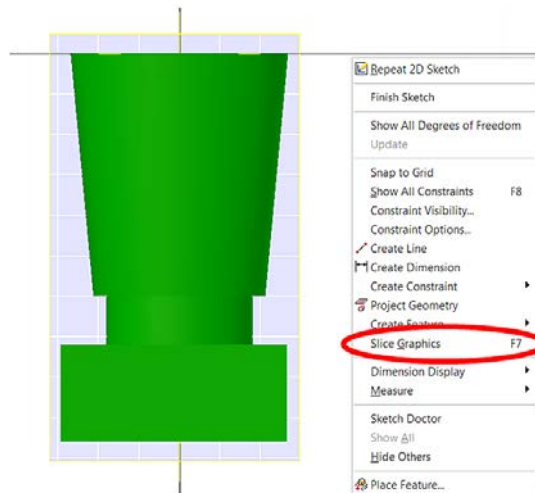
skills sets. Create a new sketch on the top surface and project the ellipse geometry. Create a larger offset ellipse using the **Offset** tool. Apply a dimension to the Offset as shown in the image below.

5. Save the file as Automoblox Male *YourInitials.ipt* to your Automoblox project folder.

## Intersect Extrusions

6. Place a work axis through the center point along the long axis of the ellipse of the body.
7. Choose a face of the square base that is parallel to the long axis of the ellipse that was used to create the body. Place a new sketch on this face of the square base.

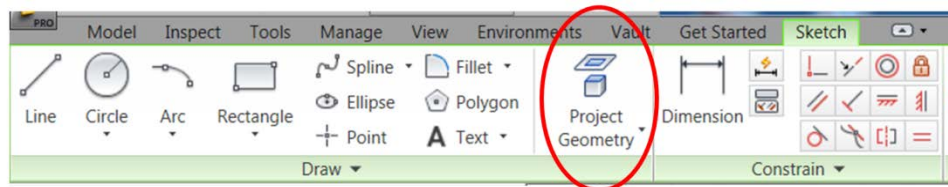
Right click on the sketch and Slice Graphics so that you can see sketch lines that intersect with the existing geometry (see image below).



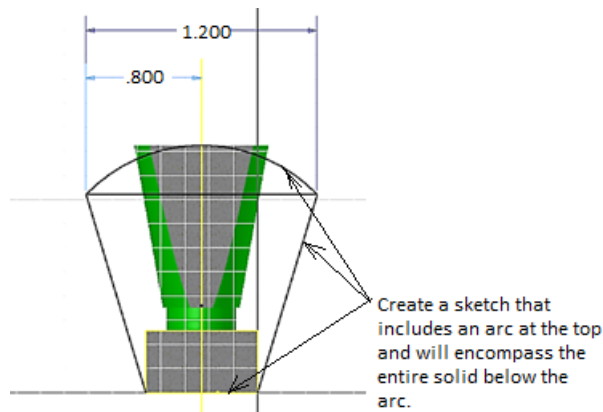
**Helpful Hint: Slice Graphics** will cut the part at the current sketch plane and remove the solid material in the foreground (similar to a section view). This will allow you to see the geometry at the sketch plane and can provide better visibility of the sketch plane by removing obstructing material.

8. **Project Geometry** of the work axis onto the sketch.

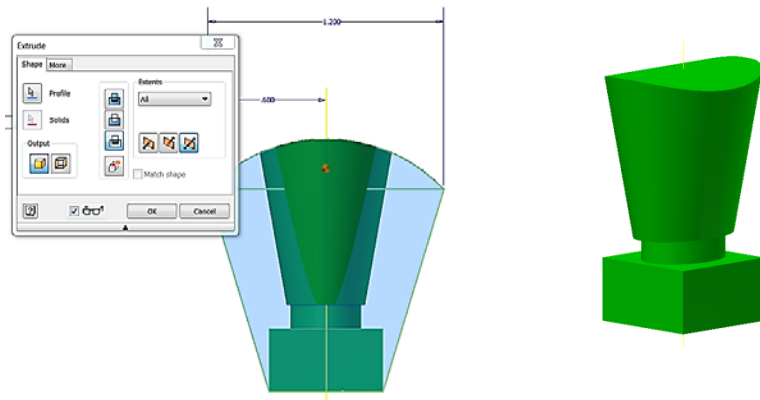
**Helpful Hint: Project Geometry** is a tool that will project the geometry of previously created features of a part onto the sketch plane so that you may use the geometry in the construction of the sketch. The Project Geometry tool is accessible in the Sketch Ribbon in the Draw panel.



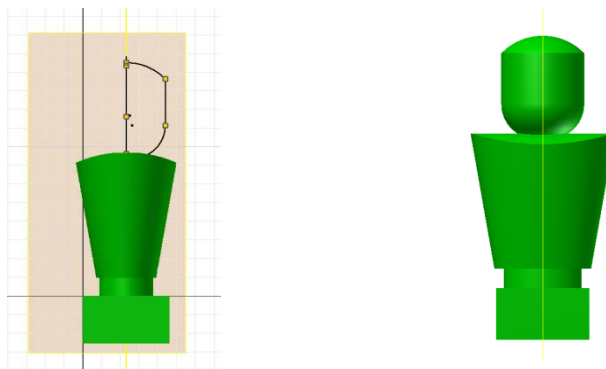
9. Sketch an arc and center it on the figure that will approximate the curve of the shoulders on the figure. Do not worry about exact dimensions. Provide a reasonable approximation of the figure shoulder contour (see image below).
10. Complete the sketch with lines that will encompass all of the existing geometry of the figure below the arc. Since you will be using the Intersect extrusion tool, include all of the lower geometry within this sketch.



11. Perform a **Mid-plane** (or Symmetric) **Intersect** extrusion. Note that the dialog box may appear different in other versions of Inventor.



12. Create a **Workplane** through the center of the figure and place a sketch on the work plane. Create a sketch to approximate the shape of the head of the figure – do not try to model the shape exactly. Your sketch should include a line on the work axis through the ellipse that you previously created. Then **Revolve** the shape around the work axis of the ellipse that you previously created.



13. Save the file to your Automoblox project folder.

## Loft

### Sketch Tools – These are suggested tools for this activity

- **Rectangle**
- 2D Sketch on new surface
- Home View (View Cube)
- View Constraints (F8)
- **Polygon**
- **Offset Workplane** w/sketch
- **Projected Geometry**

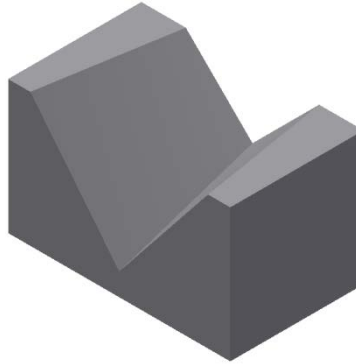
### Feature Tools – These are suggested tools for this activity

- **Extrude** (Add/Join)
- **Loft** (cut solution)

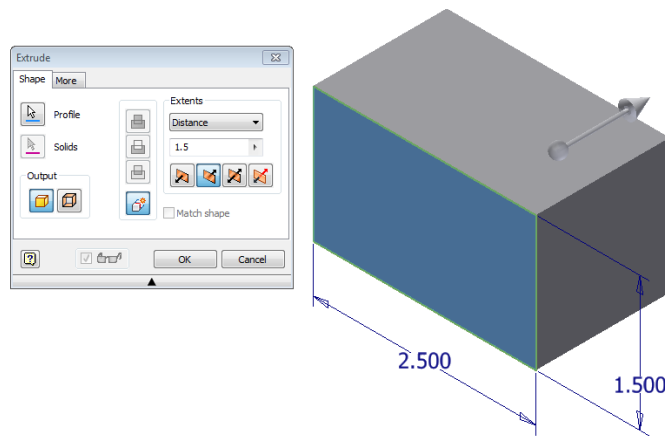


- **Revolve**
- **Work Axis**
- **Work Plane**

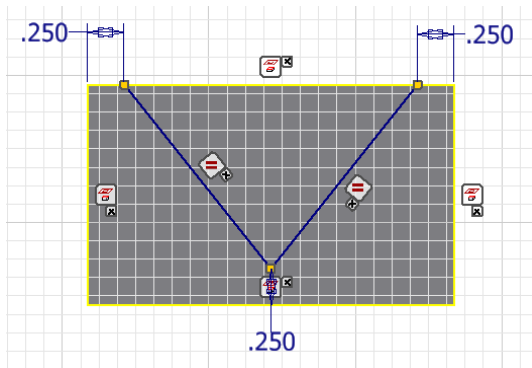
V. Model the part shown below. The loft function allows the user to create a solid surface by blending two or more shapes that are located on different planes.



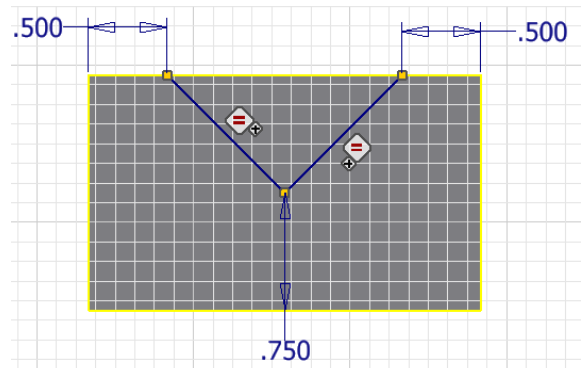
1. Create a rectangle and dimension it to the values shown in the image and then extrude it to a value of 1.5 deep.



2. Place a 2D Sketch on the front face of the block. Use the image below to draw and annotate the first of two sketches for the loft feature. Note that the sketch is showing the equal constraints that need to be applied to the sketch. Finish the sketch.
3. Apply a 2D sketch to the back face of the block and sketch the geometry shown in the right image below. Note that the sketches are placed on opposite faces of the rectangular solid. Finish the sketch.

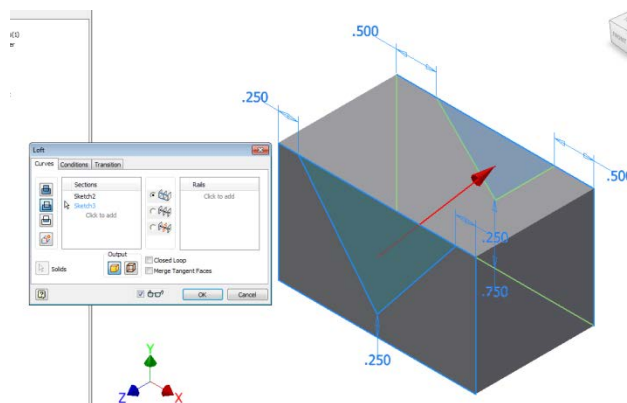


Front Face Sketch

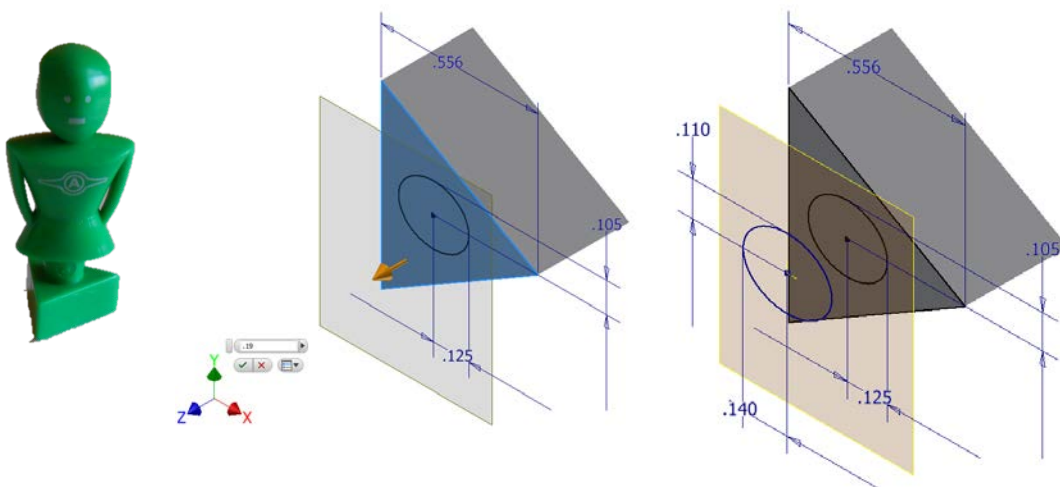


Back Face Sketch

4. Select the **Loft** command from the 3D Modeling ribbon bar. Select the front sketch and then pick the deep “V” profile. This will highlight the shape that will start the loft. You must then press “Click to add” on the Loft palette then select the back sketch and then pick the small “V” profile on that sketch. Choose the Cut option by depressing the cut button. Then click OK.

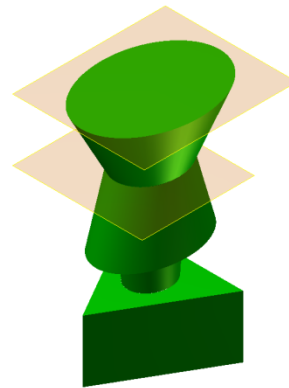
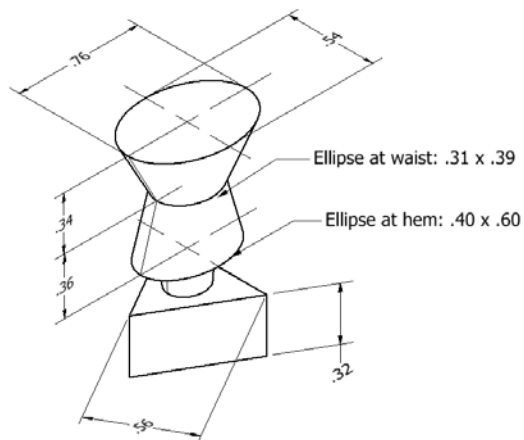


5. Save the part as *LoftCut.ipt* in the Practice project folder.



VI. Create the Automoblox female figure with the triangular base. **Loft** can be used to create most of the female figure. If you have not done so, measure the part and record the dimensions on a sketch.

1. Create a sketch for the base of the figure as shown below. If you have not done so already, measure the height of the base with dial calipers and **Extrude** the triangle base the appropriate distance.
2. Place a sketch on the surface of the triangular base and sketch an **Ellipse** (centered on the triangle center point). Be sure to orient the ellipse properly. Use your own measurements – the dimensions provided are for reference only.
3. Create an **Offset Workplane** on which to create a new sketch of a larger ellipse. Be sure to **Project Geometry** of the small ellipse from sketch 1 so that you can locate the large ellipse on sketch 2. Use your measurements of the part – the measurements shown below are for reference only.



4. Use the **Loft** feature to create the remainder of the body of the triangular figure. Compare two different methods of creation. First, try using a **Loft** involving three sketches. Then try using two separate lofts. In either case you will need another **Offset Workplane** at the shoulders. Choose the method that most closely resembles the actual figure.
5. To finish the part, you will need to add the head to this figure. You may use the method detailed for the male figure in a previous exercise, or you may use another method.
6. Save the file to your Automoblox project folder as Automoblox Female *YourInitials.ipt*.

## Conclusion

1. What CAD feature would be used to most efficiently model a baseball bat (Extrude, Loft, Revolve, Sweep)? Explain your answer.
2. How does a 3D CAD solid model program display the progression of work involved in creating a model?
3. If a mistake is made, how does the user make a correction without using the undo function?