Activity 5.5 CAD Model Features

Introduction

Two dimension sketches are nice, but parts have three-dimensional (3D) qualities that sketches can only imitate and communicate in an abstract manner. 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 do not require sketches, but they do require an existing three-dimensional object. Now that you have experience with the various two-dimensional sketch tools that a CAD modeling system has to offer, it is time to learn about some of the more common 3-D options that designers use to create computer models of design solutions.

As is the case with sketched geometry, 3D CAD features can be made to perfect dimensional accuracy. The ability to realize CAD models through sequentially adding and subtracting three-dimensional features is a critical skill that designers in multiple engineering disciplines use to make mental images into money-making products.

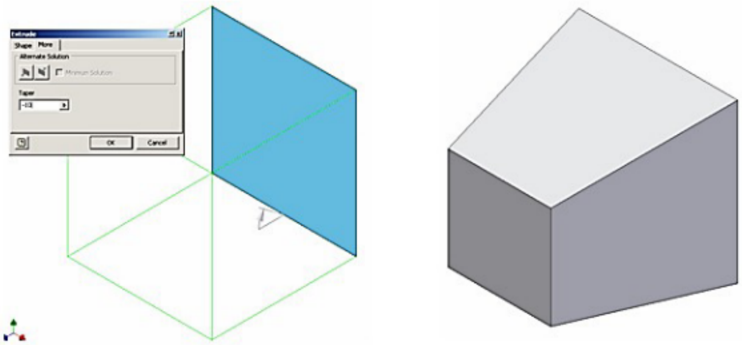
Equipment

- Computer with 3D CAD modeling software
- CAD files:
  - Bushing
  - Circular Pattern
  - Drilled Holes
  - Extrude-Taper
  - Fillet Chamfers
  - Intersect
  - Left Half
  - Loft
  - Paper Clip
  - Rectangular Pattern
  - Rib Support

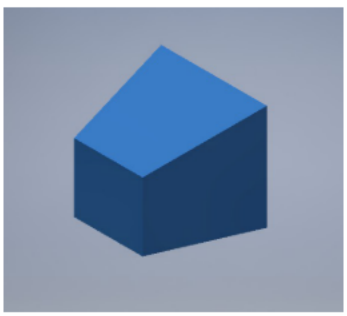
Procedure

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.

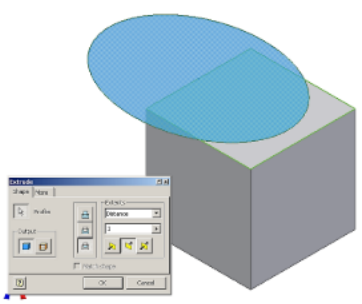
Tapered Extrusions

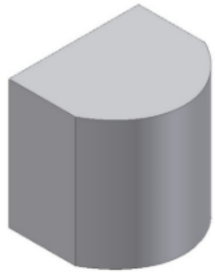


1. Extruded objects can be given a positive or negative taper angle. A common example of a tapered extrusion is the design of an ice cube. The sides of the ice cube are tapered with a draft angle to allow the cube to be easily removed from the ice cube tray. Open the file called **Extrude-Taper**, and extrude the square a distance of 1 inch with a taper angle of -10°.



Intersect Extrusions

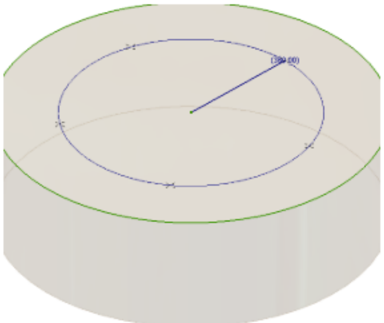
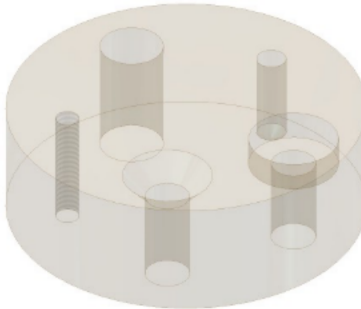




2. The intersect extrusion function will perform a Boolean addition and subtraction in one operation. Any part of the sketch profile that overlaps existing geometry will remain. The portion of the sketched profile and the existing geometry that do not overlap will be removed. Open the file called **Intersect** and perform an intersect extrusion on the sketch all the way through the existing object to observe what takes place. Note that the dialog box may appear different in other versions of Inventor.

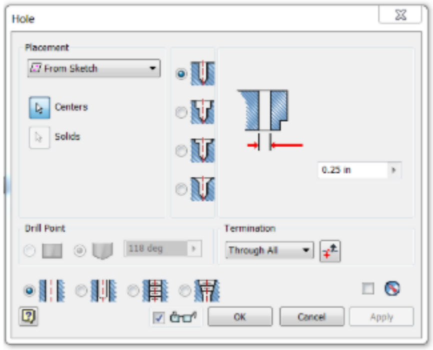
   

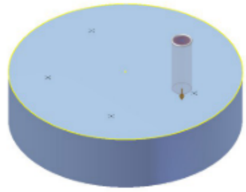
Hole

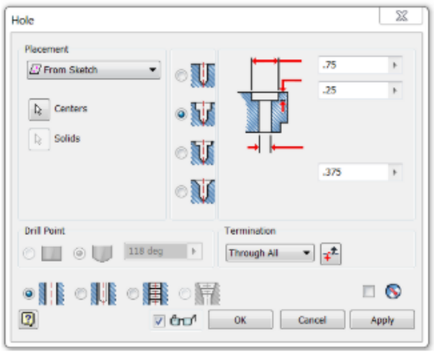
3. The **Hole** function requires a hole center for each instance. Open the file called **Drilled Holes**. The existing feature is a blind hole, and has been created for you. The following page shows the different function windows that are associated with a counterbore, countersink, threaded hole, and clearance hole. The center of each hole is a point placed in a sketch. All points in a sketch will be auto-selected by the computer when the **Hole** command is initiated. You will have to hold down the shift key to deselect the hole centers that you do not want – be sure the Centers button is depressed. Work your way around the block in a clockwise direction initiating the **Hole** function, selecting the appropriate hole center, and identifying the type of hole feature that is needed. The counterbore will have a major diameter of .75 inch that is recessed .25 inch. From there, the through hole will have a diameter of .375 inch. The countersink has a major diameter of .75 inch, with a taper of 83. The threaded through hole has a nominal diameter of .25 inch, and a 1/4-20 thread applied to its interior wall. The clearance hole goes through the disc, and has a diameter of .531 inch. Note: a shared sketch will not disappear after the first feature is created.

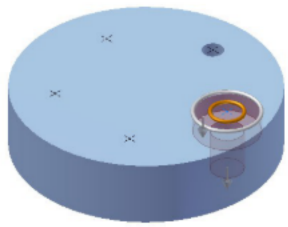
Through (THRU) Hole



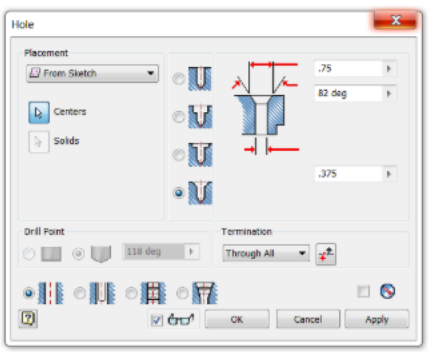


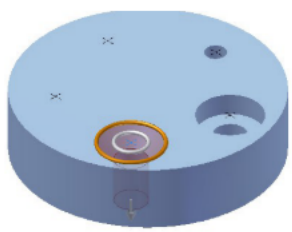
Counterbore



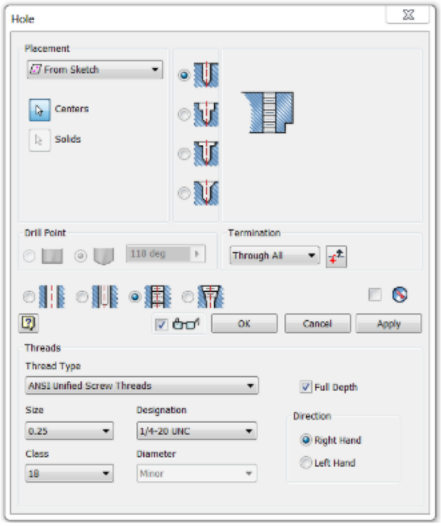


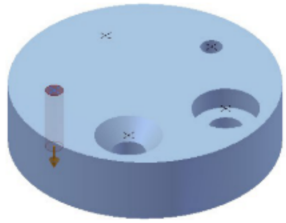
Countersink



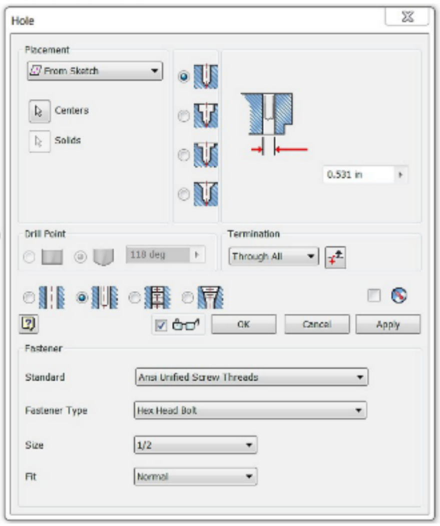


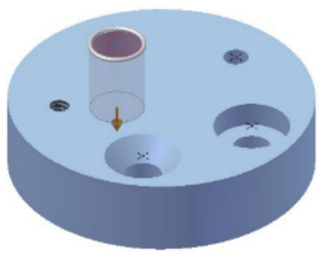
Threaded Hole

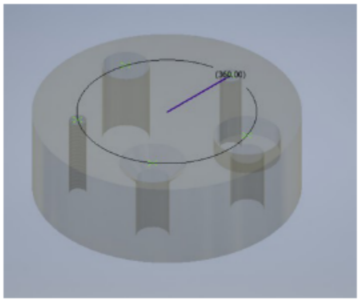




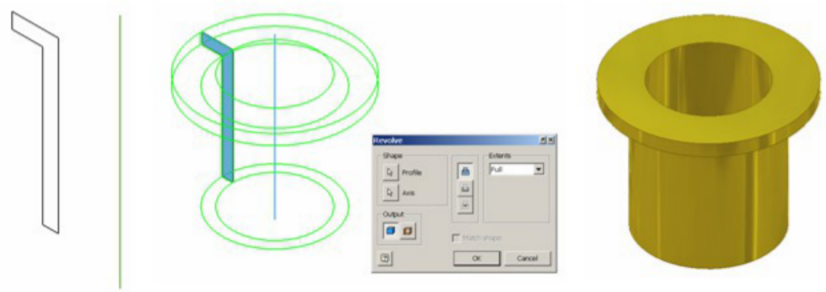
Clearance Hole



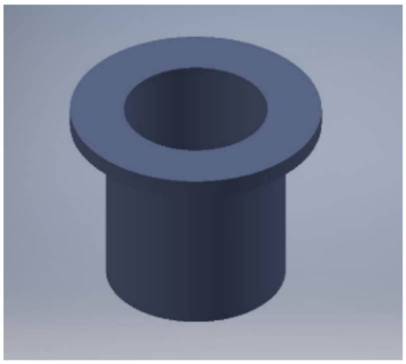



(360.00)

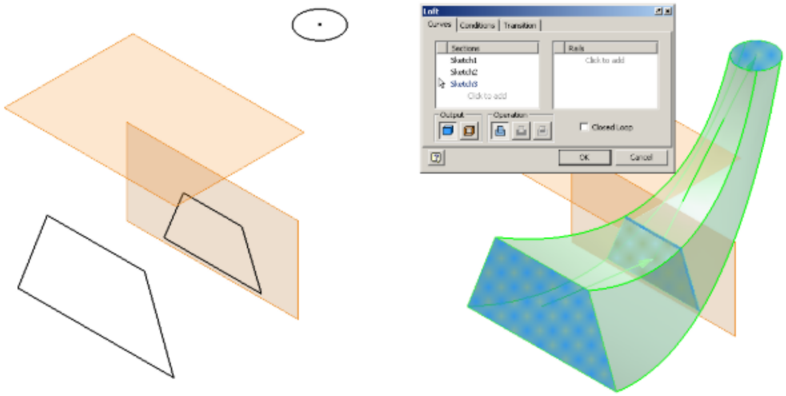
Revolve



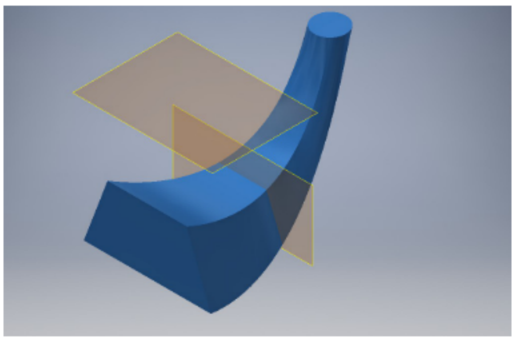
4. Revolve is a function that allows the user to extrude a closed profile around a fixed axis up to 360°. The axis can be part of the profile, an existing edge on a part, or one of the axes of the Cartesian coordinate grid. Grid axes may be selected from the Origin folder located in the Browser bar.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



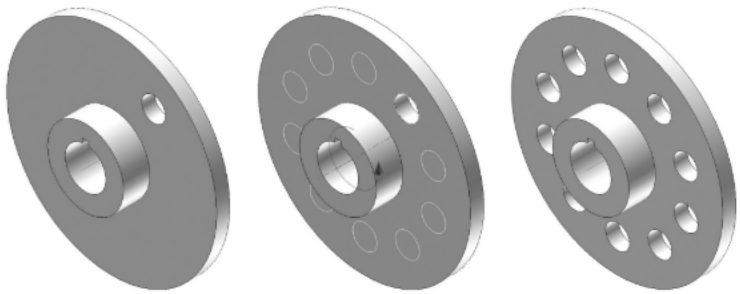
Loft



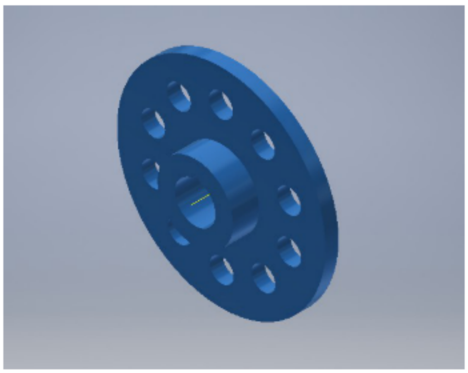
The loft function allows the user to create a solid or surface by blending two or more shapes that are located on different planes. Open the file called **Loft**. Use the Loft function to blend the three profiles into one solid object.



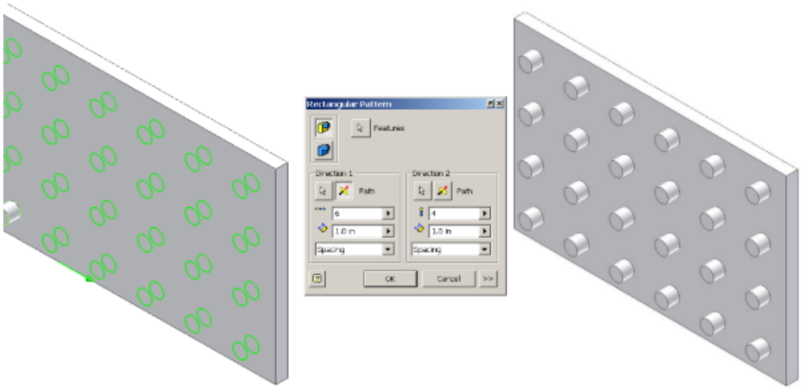
Circular Pattern



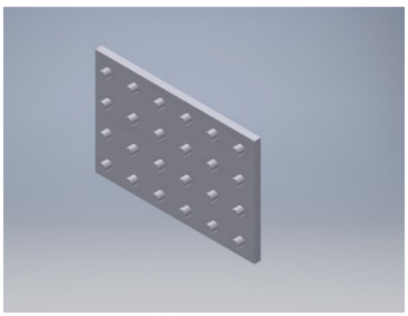
5. The pattern function allows the user to make multiple copies of an existing ***feature*** (as opposed to a sketch line as discovered in Activity 5.2 Making Sketches in CAD) in one of three ways. A circular pattern is often used to array a hole around a center axis. An edge on an existing feature can also serve as the center axis. Open the file called **Circular Pattern**, 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 work axis.



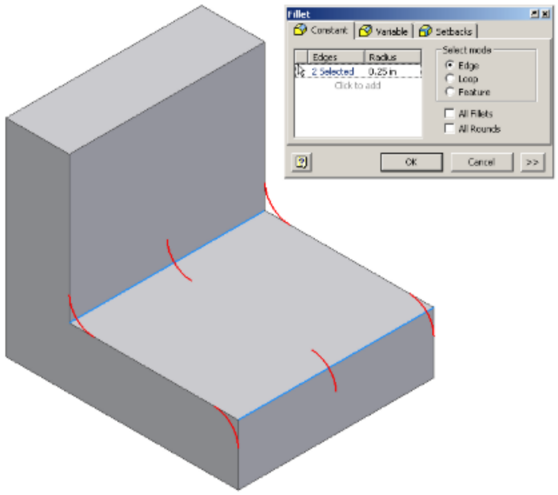
Rectangular Pattern



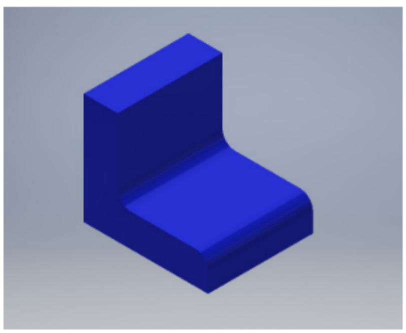
6. The rectangular pattern function allows the user to make copies of an existing feature in one direction or two directions simultaneously. Existing edges or the axes of the Cartesian coordinate grid must be selected to identify the desired direction(s). Open the file called **Rectangular Pattern.** Use the rectangular pattern to copy the existing cylindrical extrusion six times in the horizontal direction and four times in the vertical direction.



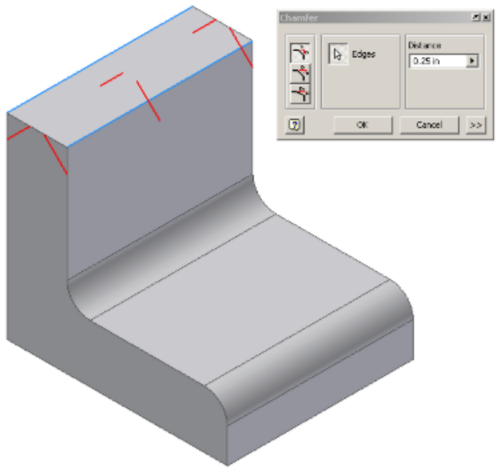
Fillet



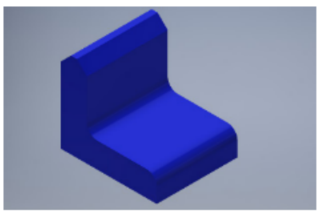
7. Fillet is a function that allows the user to create a rounded blend where two surfaces meet to form an edge. It should be noted that on an exterior corner, the resulting feature is known as a round. On an interior corner, the resulting feature is known as a fillet. Open the file called **Fillets Chamfers**.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



Chamfer

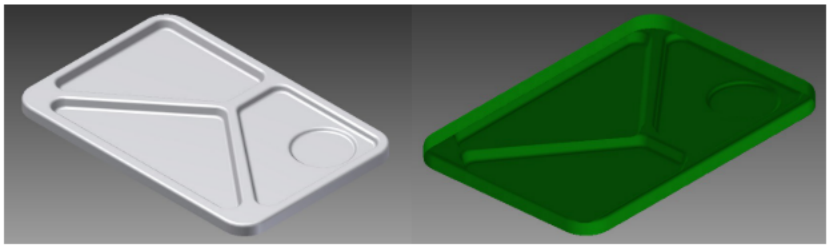


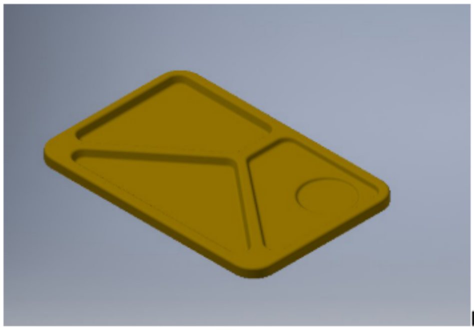
8. Chamfer is a function that allows the user to apply an angle surface where two existing surfaces meet to form an edge. Open the file called **Rib Support**. Use the chamfer function to apply a .125 inch x 45° chamfer to the edges shown above.



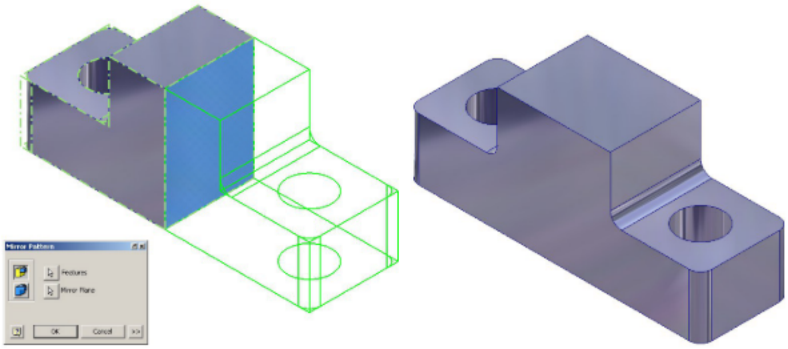
Shell

9. The Shell function allows the user to remove unnecessary mass from a feature. The resulting geometry will have a wall thickness that is specified by the user. Open lunch_tray.ipt. Use the **Shell** function to remove the excess material from the bottom of the tray to leave a wall thickness of 0.01 inch as shown. **Make sure to show the bottom of the tray.**

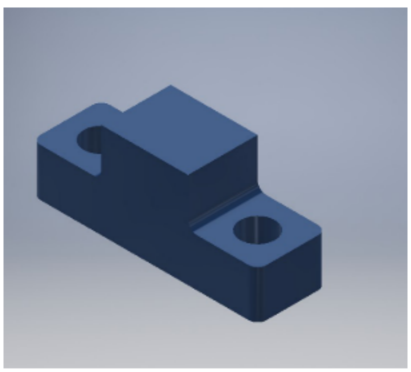




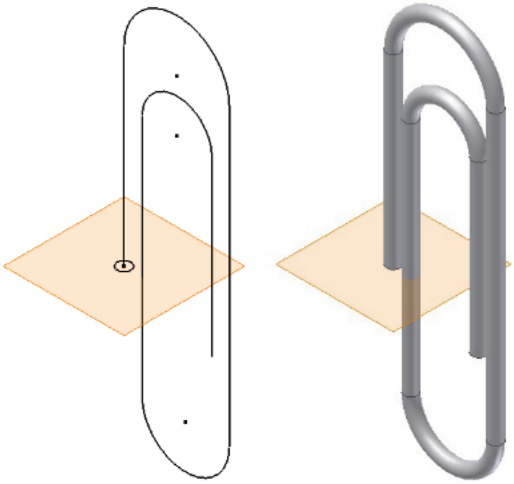
Mirror



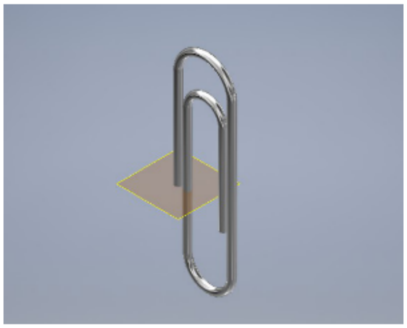
10. Mirror is a function that allows the user to create a mirror image of existing geometry. This function requires an existing feature(s) and a surface or work plane to serve as the mid-plane of symmetry. Open the file called **Left Half**. Use the mirror function to add a duplicate mirror image of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



Sweep



11. The sweep function allows the user to extrude a closed profile along a path. The path may be open or closed. The profile and the path must exist as two separate sketches. Open the file called **Paper Clip**. Use the sweep function to extrude the circle along the existing path to create the form of a paper clip.



Conclusion

1 What 3D CAD functions could be used to create a wire coat hanger?

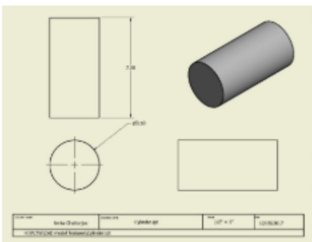
Loft, mirror, and sweep.

12. What feature would be used to create a 3D representation of a spindle that was created on a wood lathe?

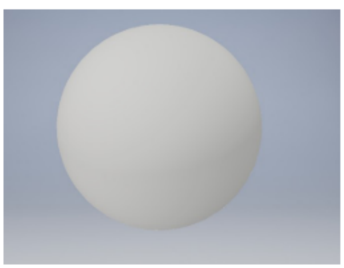
Revolve

13. For each of the following, sketch a 2D shape and an axis of rotation that you could use to model each of the following solid forms using the Revolve function within 3D modeling software. Dimension each sketch and indicate the axis of rotation that you would use. You may use software to check your answers.

    a. A solid cylinder with a diameter of 3.5 inches and a height of 7 in. Be sure to dimension the shape.



    b. A solid sphere using the **Revolve** function? What rotation angle would you use?



360 degrees

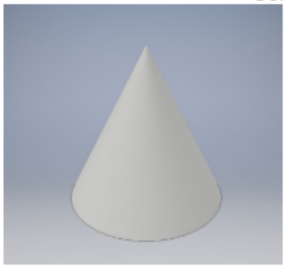
14. For each of the following, describe the solid form that would result from rotating the given shape about the axis of rotation by the given rotation angle. Then sketch the resulting 3D shape and indicate important dimensions.

a
15. .

a

How does 3D CAD solid model

Cone



b
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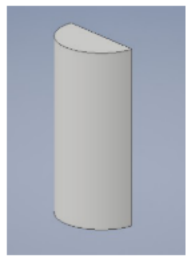

Half Cylinder

program display the progression of work involved in creating a model?

It shows step by step what to do.

16. If a mistake is made, how does the user make a correction without using the undo function?

    Going back to the toolbar and editing.