



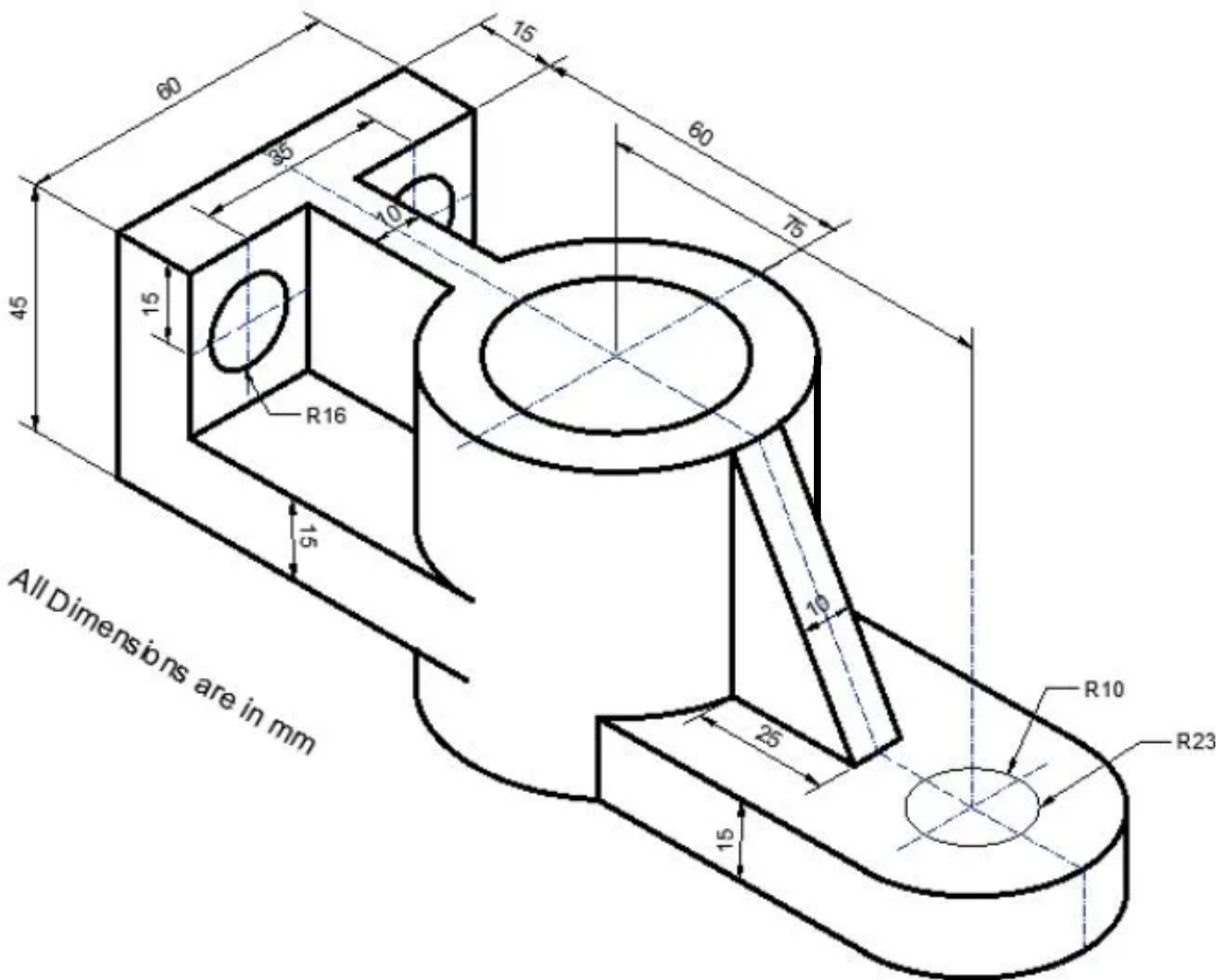
AutoCAD Level 3

Session 03

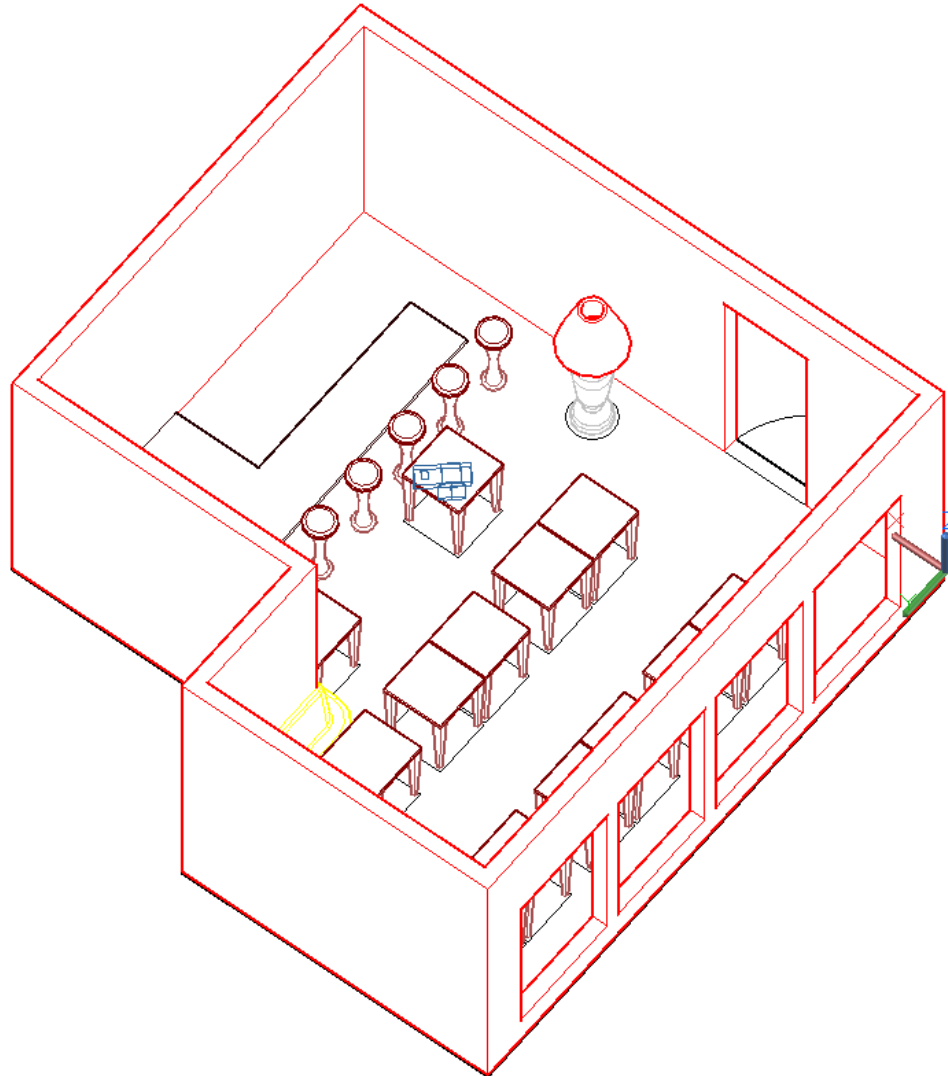
AGENDA

- **MORE 3D MODELLING EXERCISES**
- **PRESS PULL COMMAND**
- **REVOLVE COMMAND**
- **SWEEP COMMAND**
- **LOFT COMMAND**

Exercise 4- Mechanical part 3

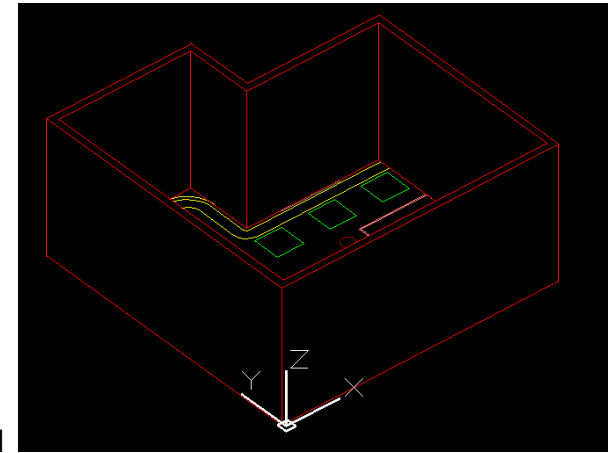
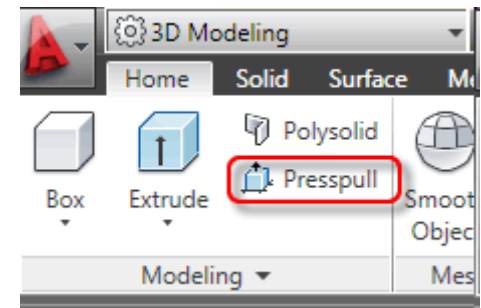


Exercise 5- Café 3D



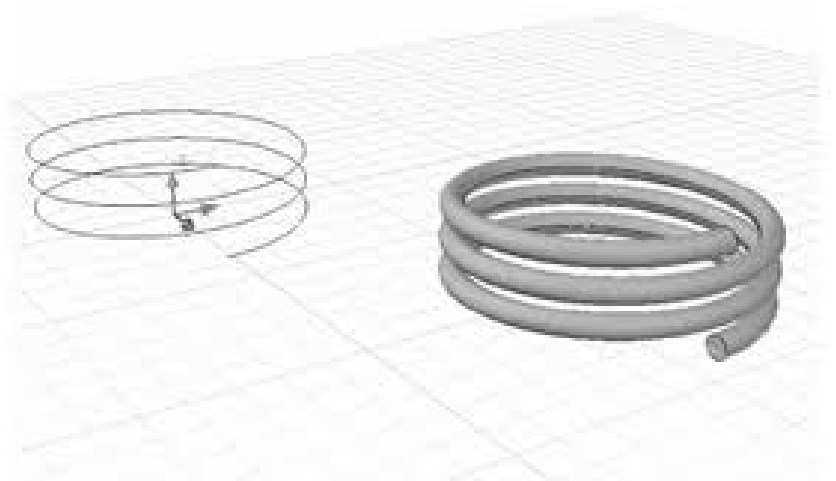
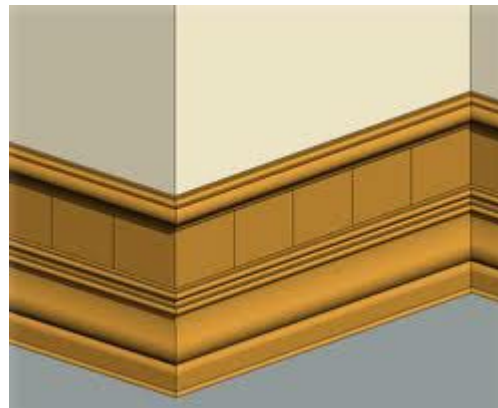
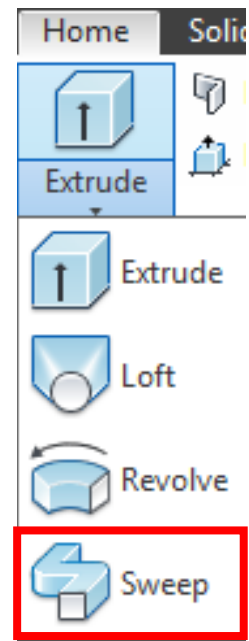
Presspull command

- Open the “cafe3d.dwg” file from previous week
 - Similar to the extrude command in result
 - Contrary to popular opinion, it is not necessary to create a single, contiguous object such as a pline or spline to serve as the boundary for an extrusion.
 - The PressPull command simply asks you to pick a point within the area to be extruded. The area can be defined by overlapping line segments, arcs, circles, or whatever.
 - It then searches outward from that point and defines its own boundary.
 - Now you can drag the profile to the desired height, or you can input a specific value.
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- Lets continue with the cafe3d.dwg
 - Use the presspull command to give the walls a height of 10’.
 - Make sure to remove any 2d lines representing doors and windows before extruding the walls.



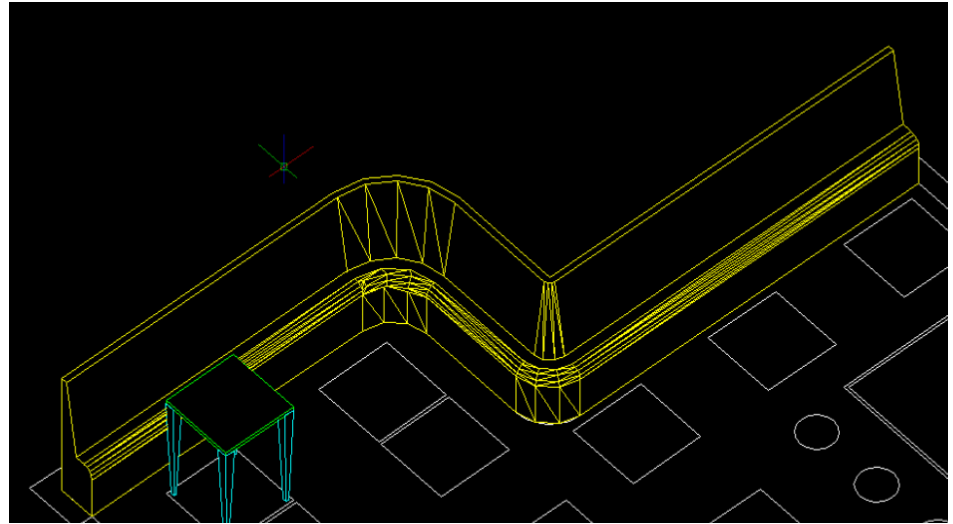
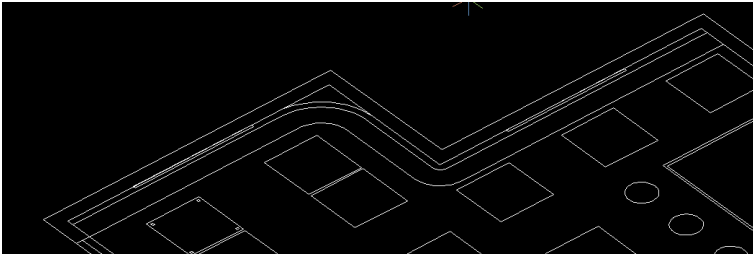
Sweep Command

- The sweep command is very similar to the extrude along a path command.
- The main advantage of the sweep command over the extrude command is the sweep command can extrude along a closed path. The extrude command can only create a solid along an open path.



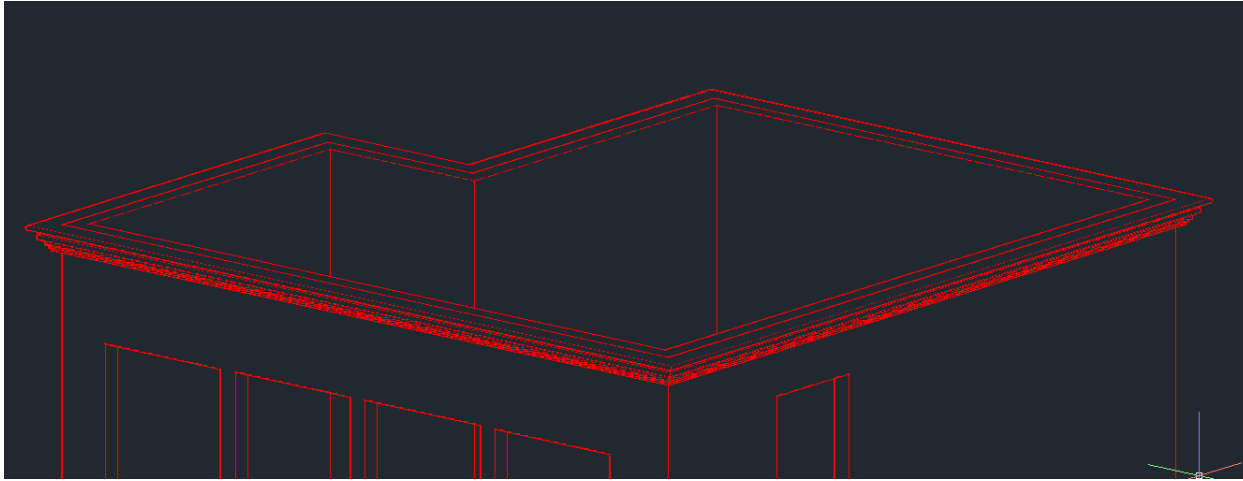
Sweep command to create banquette seating

- Open the file entitled “cafe3d.dwg”.
- Another way to extrude an object is along a path.

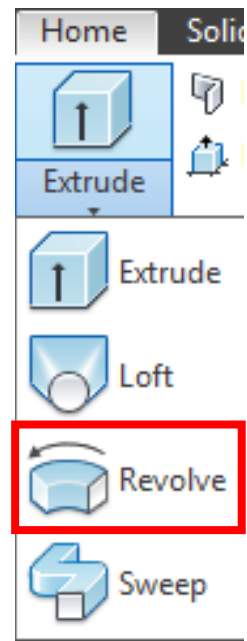
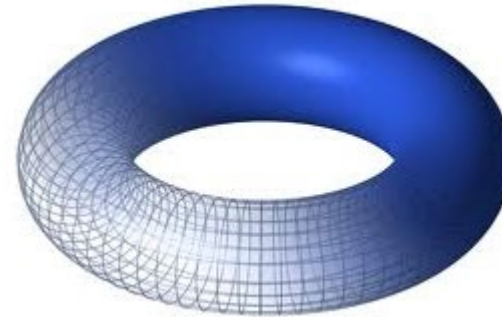
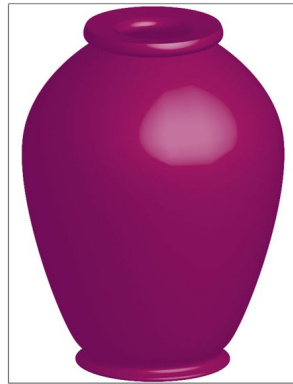
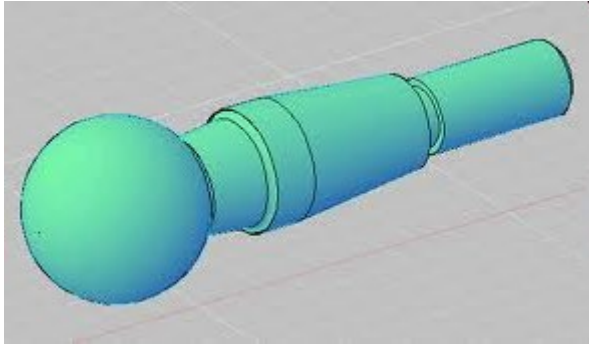


- Anytime you need to draw the profile of an object on a perpendicular plane from the ground, draw a line and give it a thickness using the change>thickness command. Then set your UCS>3pt on the side of the new plane.
- To view the ucs straight on, use the Plan> current ucs command.
- To draw the path on the ground, set your ucs back to the ground (world ucs)
- The path must be an open polyline. Fillet the corners to create a smoother path. But be careful not to make the path too complex. If the fillets are too large or too small, the sweep may not work.
- You can use the sweep command to create objects such as banquette or bench seating.

Sweep Command



Revolved Command



- The **REVOLVE** command allows you to create a curved solid from a 2D Polyline object by sweeping it around an axis. Revolve can be used with an object made from Plines, Polygons, Circles, Ellipses, and Splines.
- Similar to the extrude command, the profile to be revolved must be the following:
 - 1) must be a one entity object (polyline)
 - 2) must be a closed entity
 - 3) must be non self intersecting
- Pick two points to define the axis of revolution.
- If the axis is away from the object, a hole trough the centre is created.

Using Revolve To create Lamp

- Open the “cafe3d.dwg” drawing
- Draw a line through the centre of the circle from quadrant to quadrant. Give the line a thickness of 6' to create a new drawing surface.
- Set the UCS along the new plane.
- Draw the profile of the lamp base and shade using a closed polyline.
- Revolve the profiles using the steps outlined below

Command: Revolve

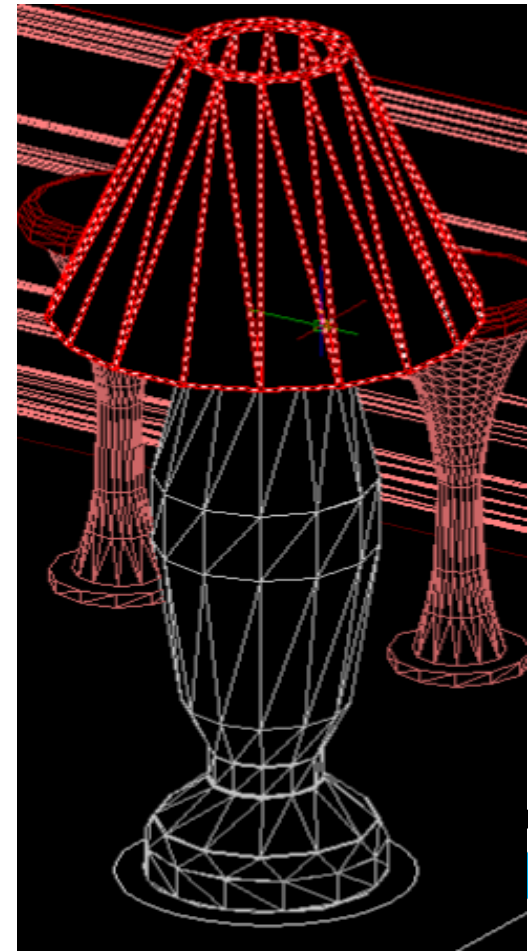
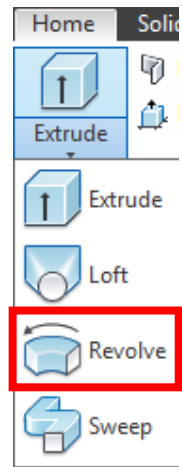
Prompt: Current wire frame density: ISOLINES=4

Select objects: Select the object or objects you wish to revolve and press Enter.

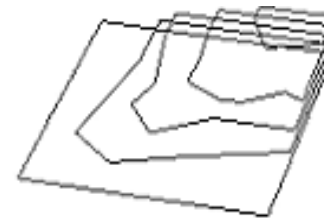
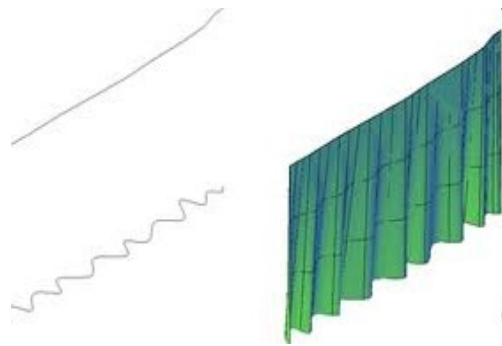
—Prompt: Specify start point for axis of revolution or define axis by [Object / X (axis)/ Y (axis)]: Select one end of the axis that the object or objects will be revolved around.

—Prompt: Specify end point of axis: Select the other end of the axis that the object or objects will be revolved around.

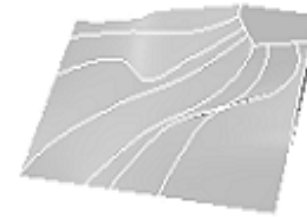
—Prompt: Specify angle of revolution <360>: Type a value for the degrees of revolution and Enter or press Enter to accept the default of full circle.



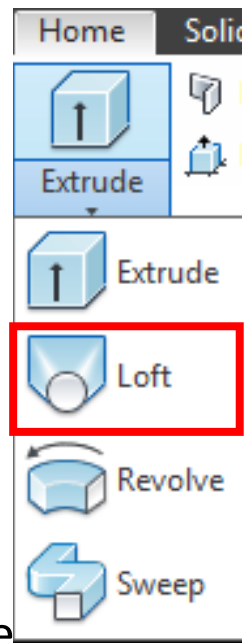
Loft Command



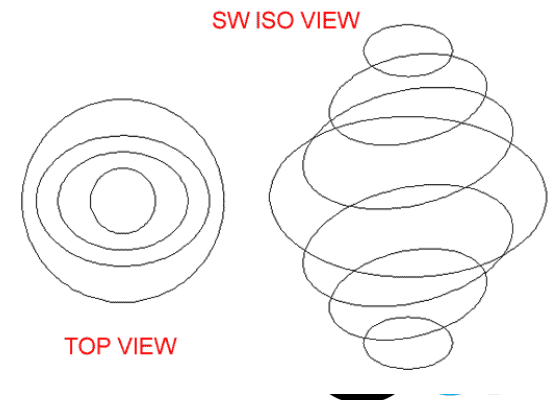
Cross sections
for lofting



Lofted solid



- The loft command is similar to the extrude command, but much more versatile. Instead of extruding a single shape, the loft command allows you to extrude several shapes and make one continuous object.
- When you connect closed entities together, a solid is created.
- When connecting open entities together, a surface is created.



TOP VIEW

SW ISO VIEW

Loft to create the stool legs

- Open the “cafe3d.dwg” drawing
- Offset the circle inwards 2” twice.
- Change the elevation of the largest circle to 30”
- Change the elevation of the smallest circle to 15”
- Use the loft command to connect the 3 shapes.

