

# AutoCAD for BioMEMS

Eyad Hamad



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# What is AutoCAD?

- **AutoCAD** is a CAD (Computer Aided Design or Computer Aided Drafting) software application for 2D and 3D design and drafting.



# Introduction



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# Commands

- ⦿ Tool Bar
- ⦿ Pull Down
- ⦿ KEY COMMANDS
  - From the Key board



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# Key Sequences

- ⦿ Always look at the command line
  - At the bottom of the page
  - Always be as specific as possible



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# Layers

- In Auto CAD, we make individual layers...
  - For things like
    - Hidden lines
    - Center lines
    - Section lines
    - Break lines

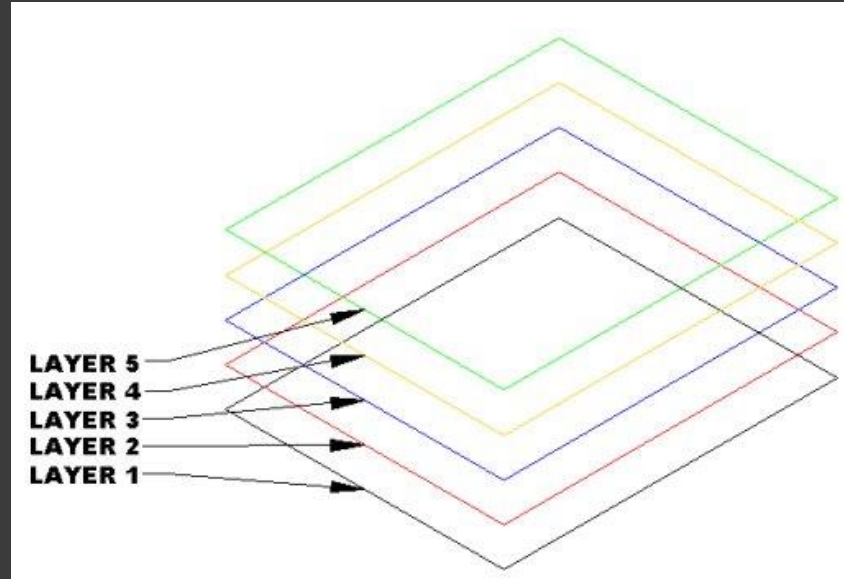


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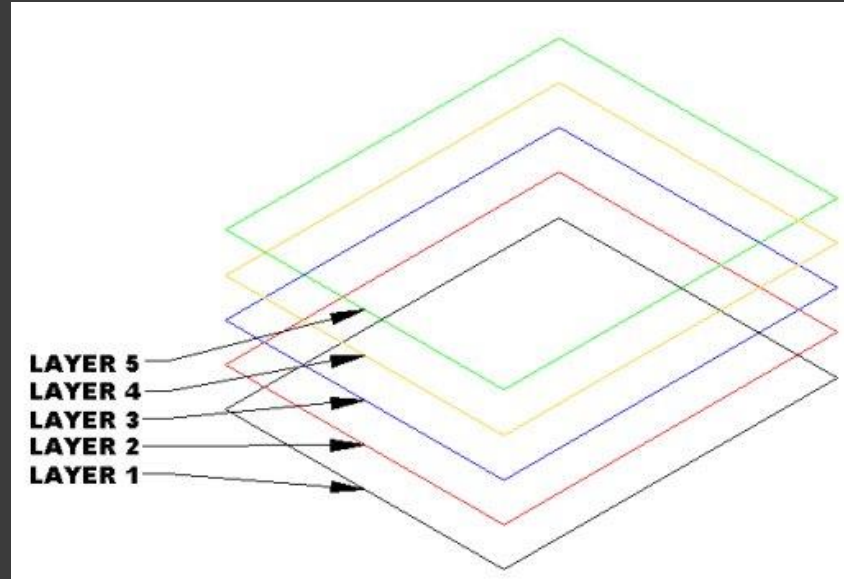
# Layers

- Layers are a way of managing, tidying and also controlling the visual layout of a drawing.



# Layers

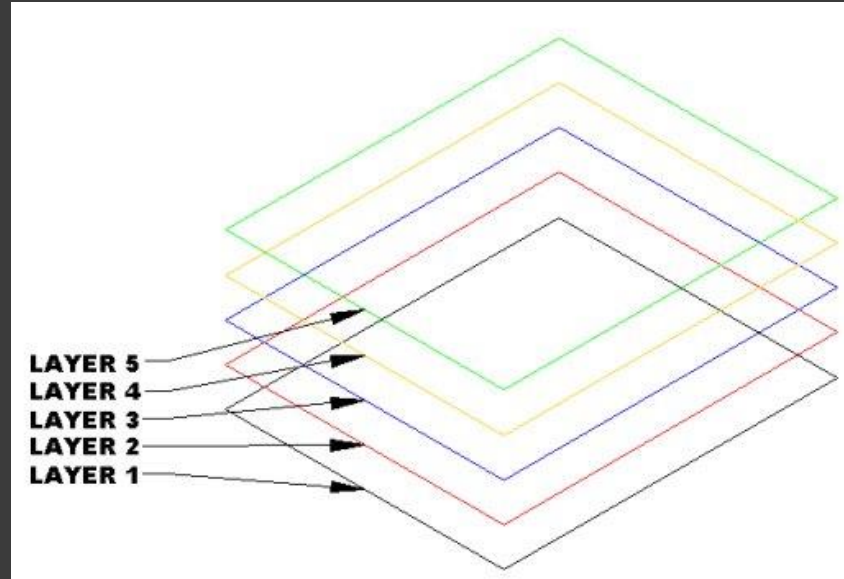
- A whole section of a drawing can be turned on or off, or simply one aspect can be controlled - text for example.





# Layers

- This is all done by using layers within Autocad.



# What is a Layer

- A layer can be thought of as a large piece of clear plastic, as infinitely large as the drawing area in Autocad.



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# What is a Layer

- When drawing in Autocad, everything is drawn on the default layer which is set current. Only the objects you are drawing are visible on the layer, the layer itself can never be seen - it is invisible.

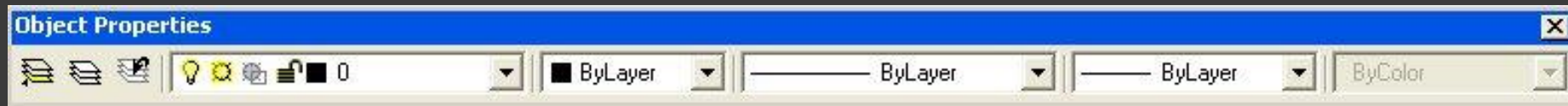


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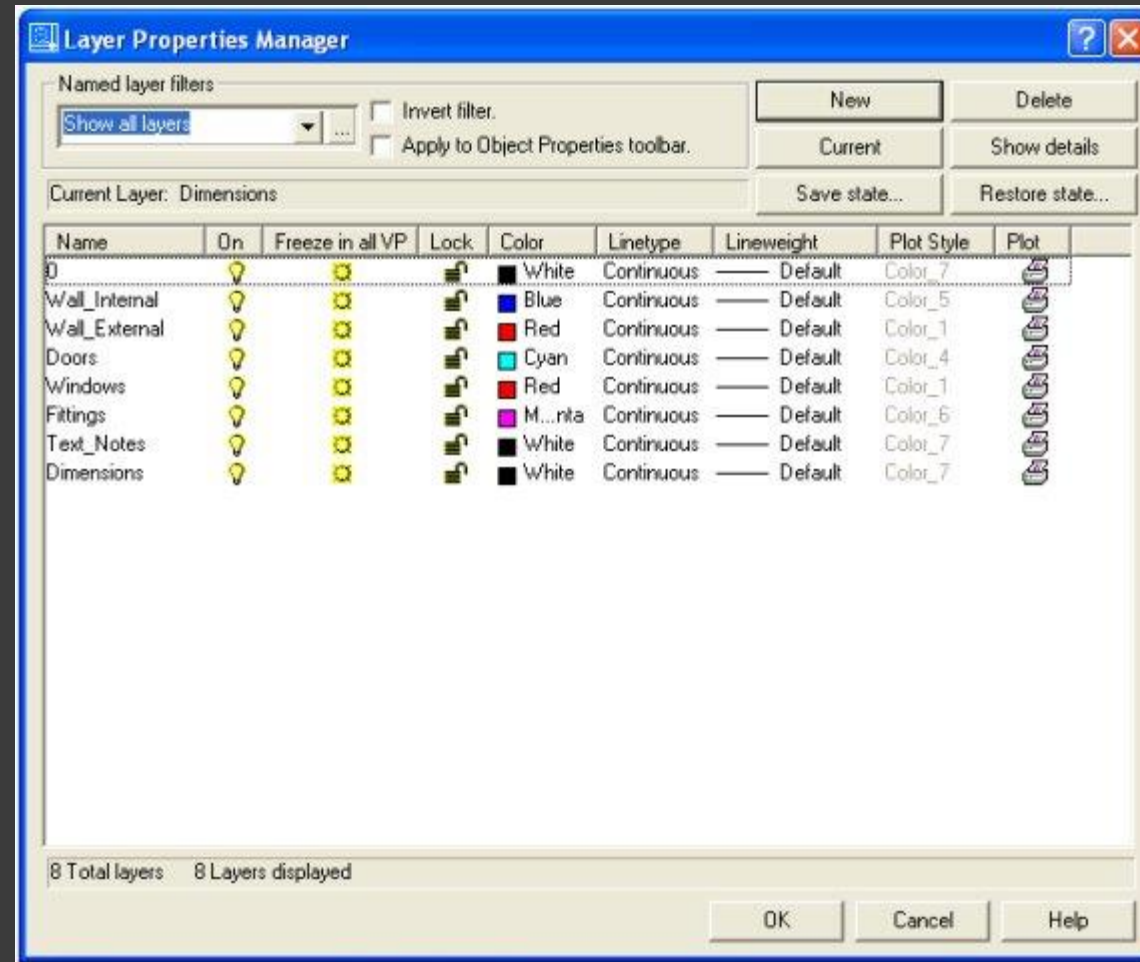
# What is a Layer

Layers are controlled by the **layer properties manager** button which is located on the object properties toolbar:



- Each new layer is created by you, the user. Normally, it is acceptable to have a layer for each different part of a drawing.

# Layer property manager



# In Layer Property manger

- ◉ Add a New Layer - Press the **New** button to create a new layer.

Delete a Layer - Press the **delete** button to delete the selected layer.

Set Current layer - Press the **current** button to set the selected layer current. All objects drawn will then be drawn on this current layer.

**Show Details** - Press the **show details** button to see more detailed information about the selected layer.

Each layer also has the following options against it:



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# Layer property Manger

- ◉ **Name** - Displays the layer name.
- ◉ **On** - Controls if the layer is on or off. Select the light bulb to turn the layer off on the drawing.
- ◉ **Freeze in all VP** - Pressing this will freeze the layer in all viewports as well as the current model view (see lesson 10 to learn about viewports)
- ◉ **Lock** - This handy feature locks a layer preventing any content of the layer from being modified.
- ◉ **Colour** - Change this to whichever colour you like. All objects drawn on the layer will display the chosen colour provided that the objects colour setting in the object properties toolbar (shown above) is set to 'By Layer'.
- ◉ **Linetype** - Set the default linetype for all objects drawn on the layer. i.e continuous, dashed, dotted etc.
- ◉ **Lineweight** - Set the thickness a line appears .Default is no thickness. This option can be toggled on/off on the display by the LWT button above the command console.
- ◉ **Plot** - Select if the layer will be shown when the drawing is plotted (printed).
- ◉ The current layer, layer colour, linetype and lineweight can all be controlled outside of the layer properties manager via the object properties toolbar.



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# Drafting Settings

- ⦿ Command DS
  - Changes Grid
  - Object Snaps
  - Polar Tracking



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# Object Snaps

- Midpoint
- End point
- Intersection
- Extension
- Center
- Quadrant
- Tangent
- Perpendicular
- Parallel
- Insert
- Node



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# Object Snaps

- The Object Snaps (Osnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately.



# Object Snaps

- Osnaps allow you to *snap* onto a specific object location when you are picking a point



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- When you are drawing, take a few moments to consider how you will construct each part of the drawing and which Osnaps you will use before starting work. There is always more than one way to draw anything in AutoCAD but the quickest, most accurate and the most efficient way always requires the use of one or more of the Osnap tools



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- Dock the Object Snap toolbar to your drawing window for quick access to the Osnaps. You will be using them all the time, one-click access is essential. See [Object Snaps](#) to find out how to display the Object Snap toolbar. To dock the toolbar, click on the toolbar title and drag it to the edge of the drawing window.



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# Modify Commands



- The modify tools (from left to right):  
**Erase, Copy, Mirror, Offset, Array, Move, Rotate, Scale, Stretch, Lengthen, Trim, Extend, Break at point, Break, Chamfer, Fillet, Explode**

# Erase

- Select this button then a drawing element to erase it permanently from the drawing.



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# Copy

- **Copy** - The copy command will copy any selected drawing elements and reposition where specified by the user, without affecting the original elements.



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# Mirror

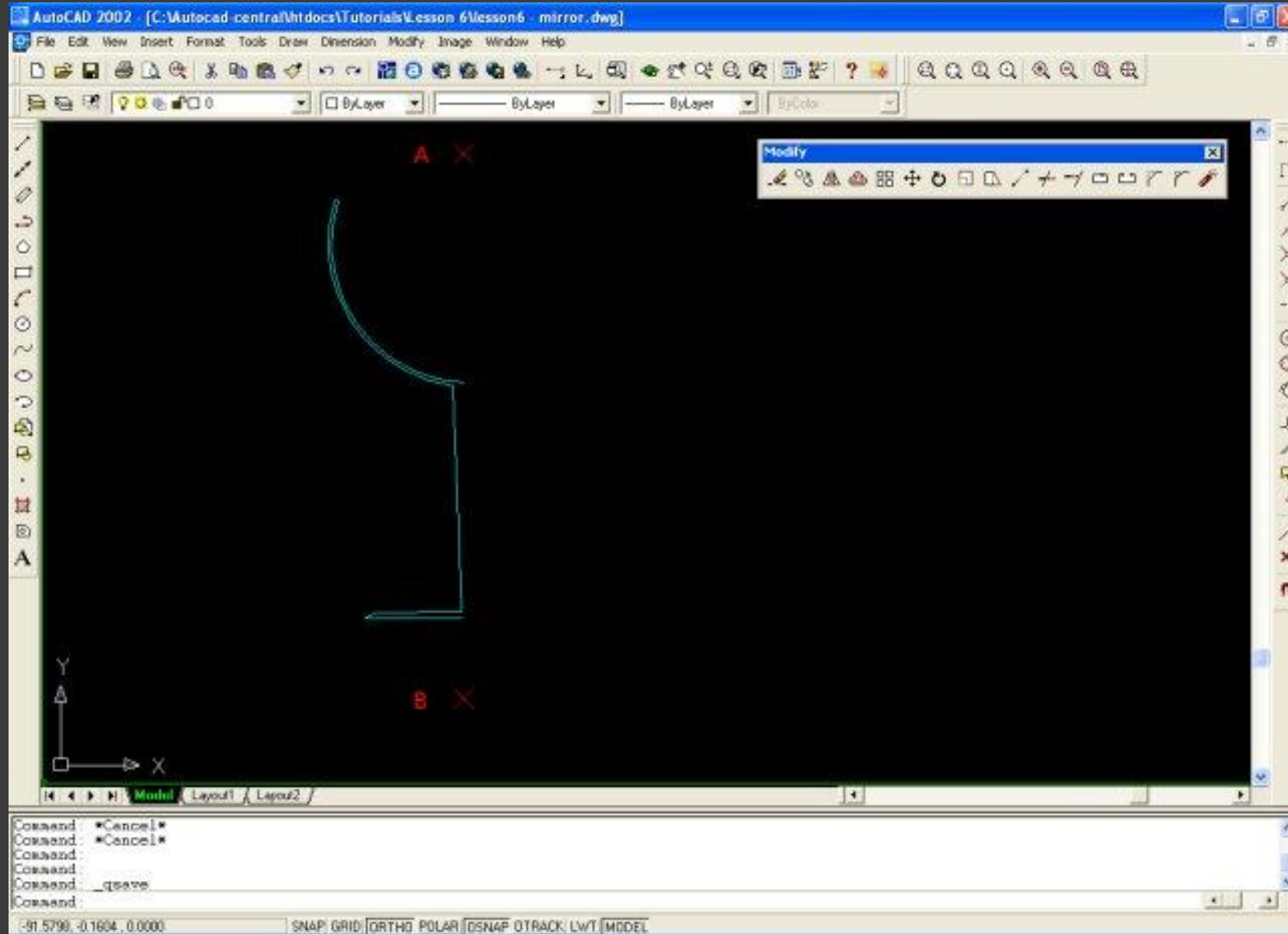
**Mirror** - The mirror command will create a mirror image of any selected drawing elements along any line of symmetry specified by the user.



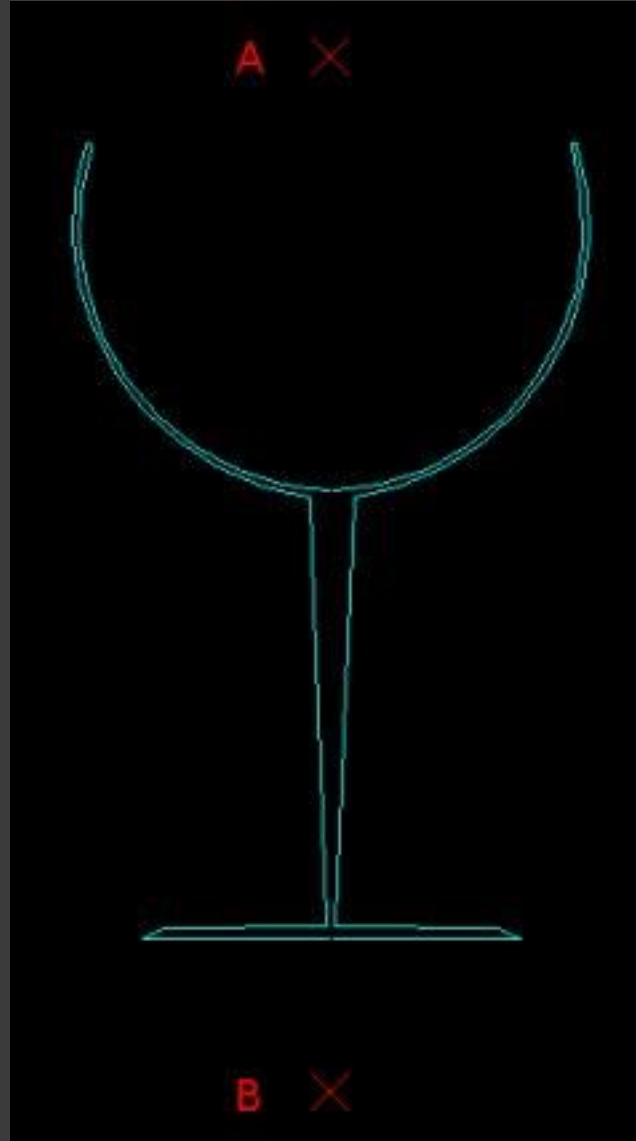
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# Mirror



# Mirror



# Offset

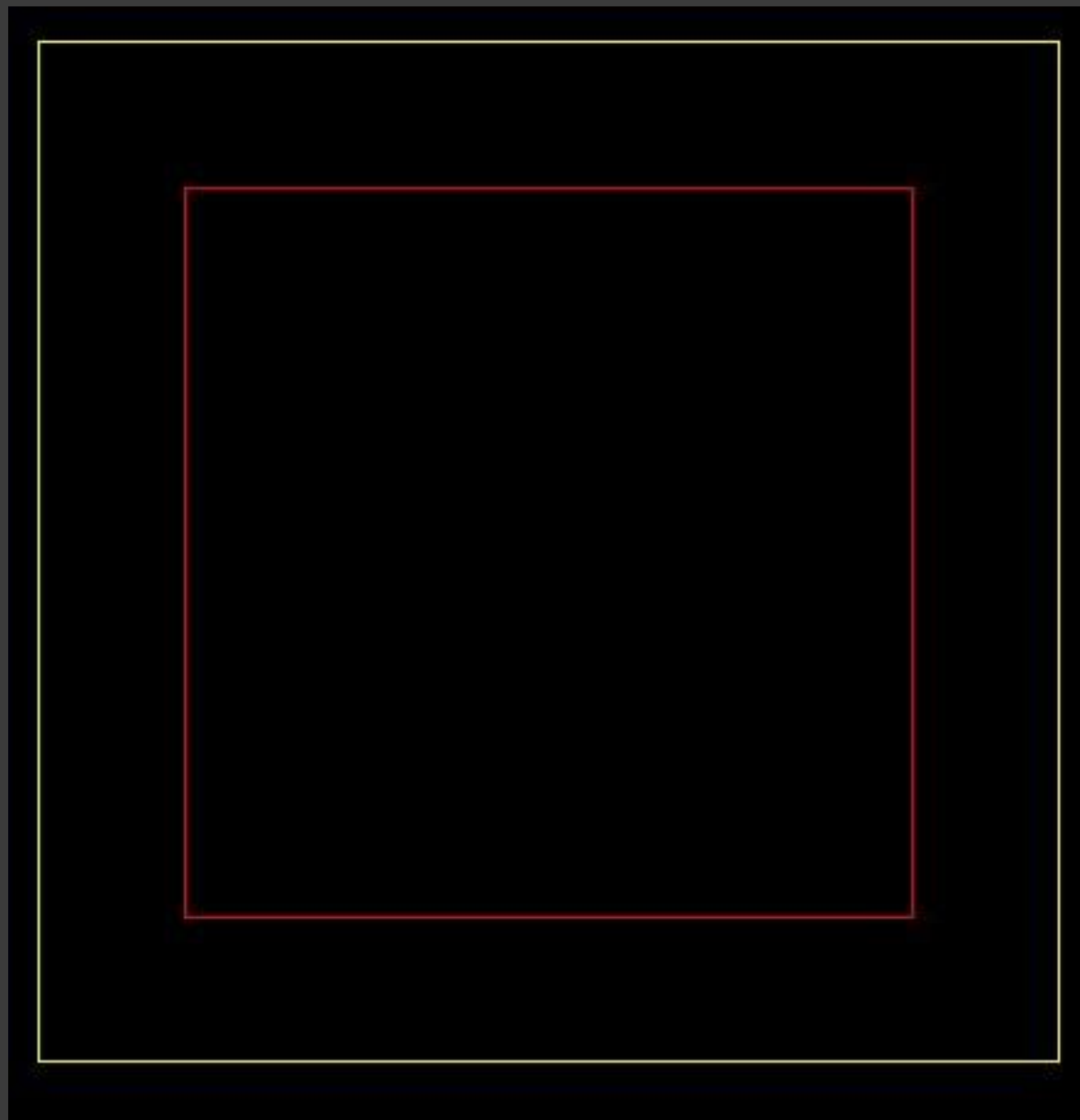
- Offset will make a copy of a line or series of selected lines by a specified distance in the direction specified.



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# Offset



# Array

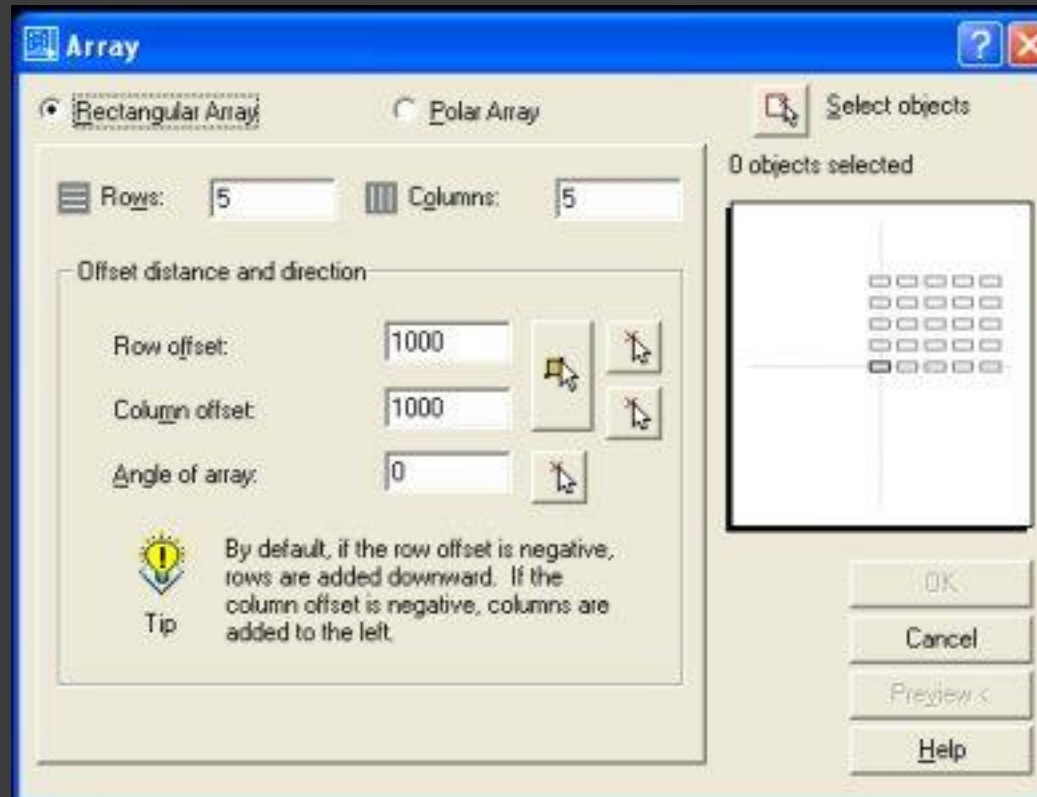
- The array command quickly creates copies of a selected object(s) to a specified spacing..



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# Rectangular Array

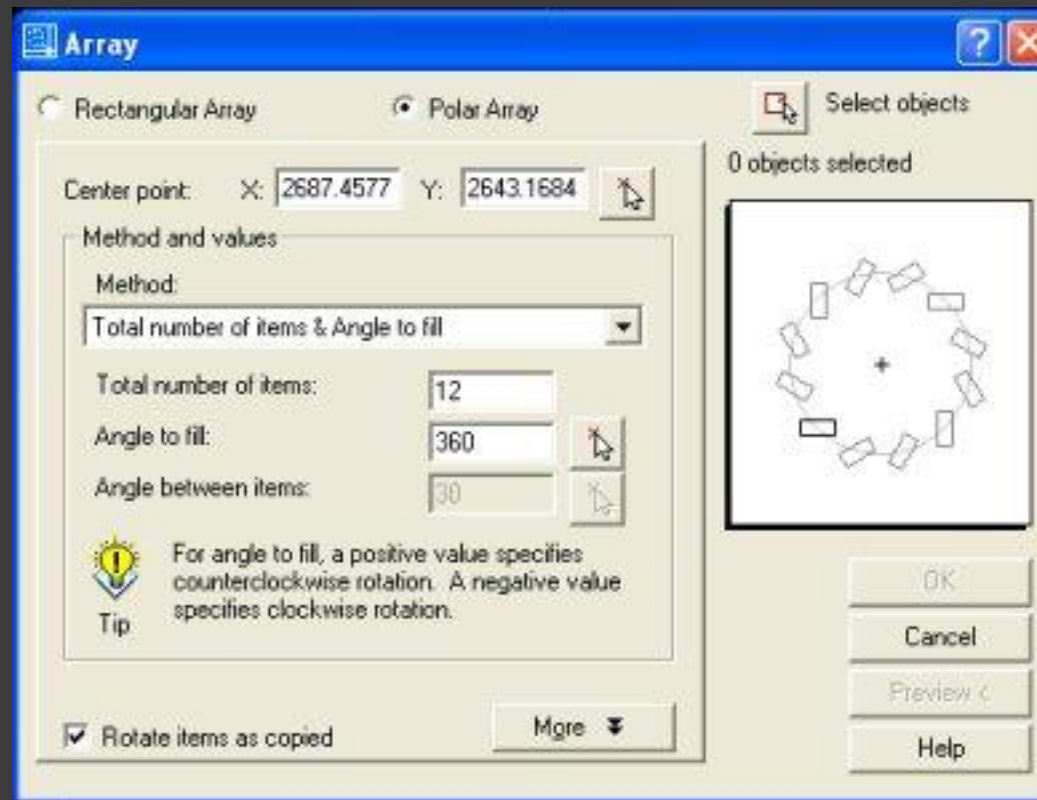


# Rectangular Array

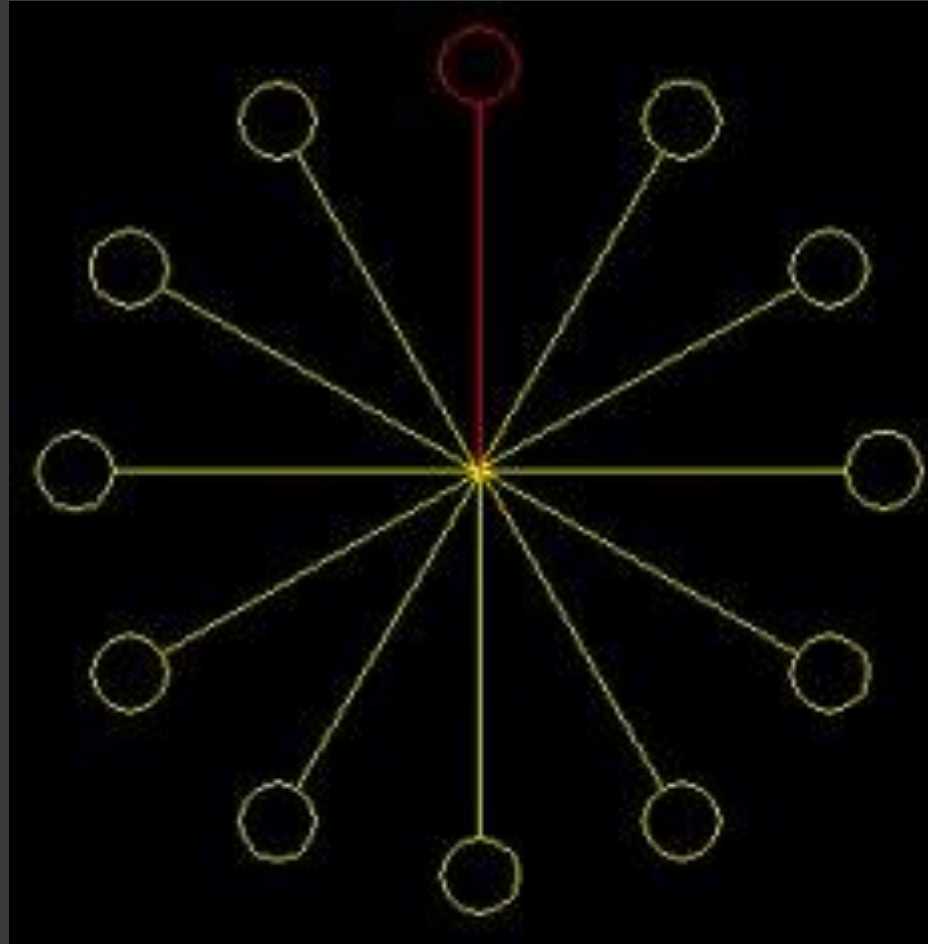




# Polar Array



# Polar Array



# Move

- The move command works exactly the same as the copy command described above, except instead of creating a copy of the selected objects, the second objects are moved.



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# Rotate

- The rotate command rotates any selected objects about a defined point by the angle specified. By default Autocad will rotate objects anticlockwise when an angle is entered.



# Scale

- The scale command scales the size of a selected object(s) by a defined scale factor from a selected base point. The selected objects can be scaled up to increase size or down to reduce the size.



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# Lengthen

- The lengthen command will lengthen a selected line.



```
Command:
Command:
Command: _lengthen
Select an object or [DElta/Percent/Total/DYnamic]:
```

# Lengthen

- **DE - Delta:** Autocad asks for a distance to lengthen the line by, when the line is selected it will then be lengthened by the specified distance **ONLY** to the side of the line where the line was selected. i.e If when you selected the line, you selected it just to the left of centre, then the left side of the line would be lengthened.



```
Command:  
Command:  
Command: _lengthen  
Select an object or [DElta/Percent/Total/DYnamic]:
```



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# Lengthen

- **P - Percent:** Autocad asks for a percent to lengthen by, then asks you to select the line. Specifying 50% would reduce the size of the line by half (The same effect as scaling by a factor of 0.5). Specifying 100% would result in no change in length. 200% would double the length of the line. When specifying the percentage to Autocad only the numerical figure has to be entered and not the percent (%) symbol.



```
Command:  
Command:  
Command: _lengthen  
Select an object or [DElta/Percent/Total/DYnamic]:
```



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# Lengthen

- T - Total: Autocad asks for the distance you want the entire line to be, when you select the line it will adjust the lines length to the distance specified.
- 

```
Command:  
Command:  
Command: _lengthen  
Select an object or [DElta/Percent/Total/DYnamic]:
```



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# Lengthen

- **DY - Dynamic:** Autocad adjusts the length of the line as the mouse is moved in the direction the line is to be lengthened. This is not an accurate technique, although is useful for quickly lengthening construction lines for example.



```
Command:  
Command:  
Command: _lengthen  
Select an object or [DElta/Percent/Total/DYnamic]:
```



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# Trim

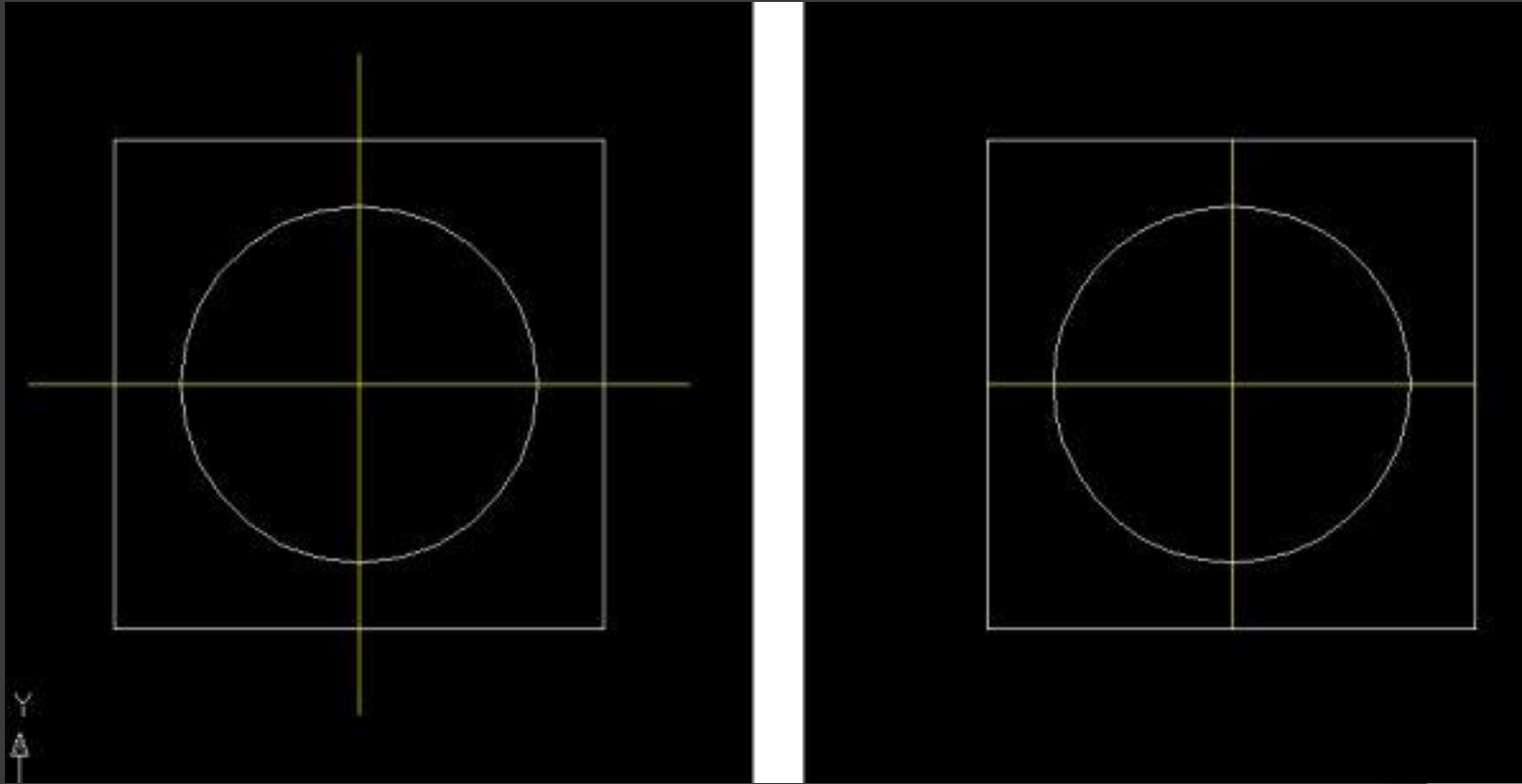
- The trim command is an extremely useful tool which will erase all parts of an object beyond or within its intersection with another object.



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# Trim



# Extend

- The extend command is similar to the trim command in how it functions, except it extends a selected line to a point of intersection of another selected object.



# Break at Point

- The break at point command enables the user to break an object at a specific point, creating two separate objects.



# Break

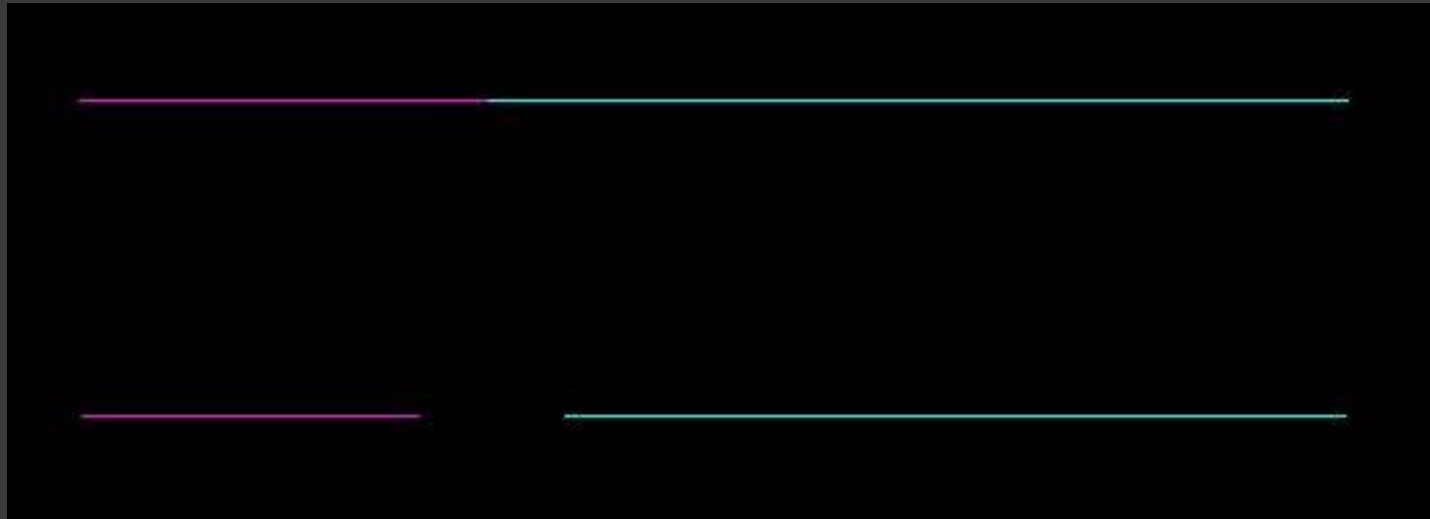
- ⦿ The break command is identical to the break at point command, except the break line isn't as neat as break at point.



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# Break





# Chamfer

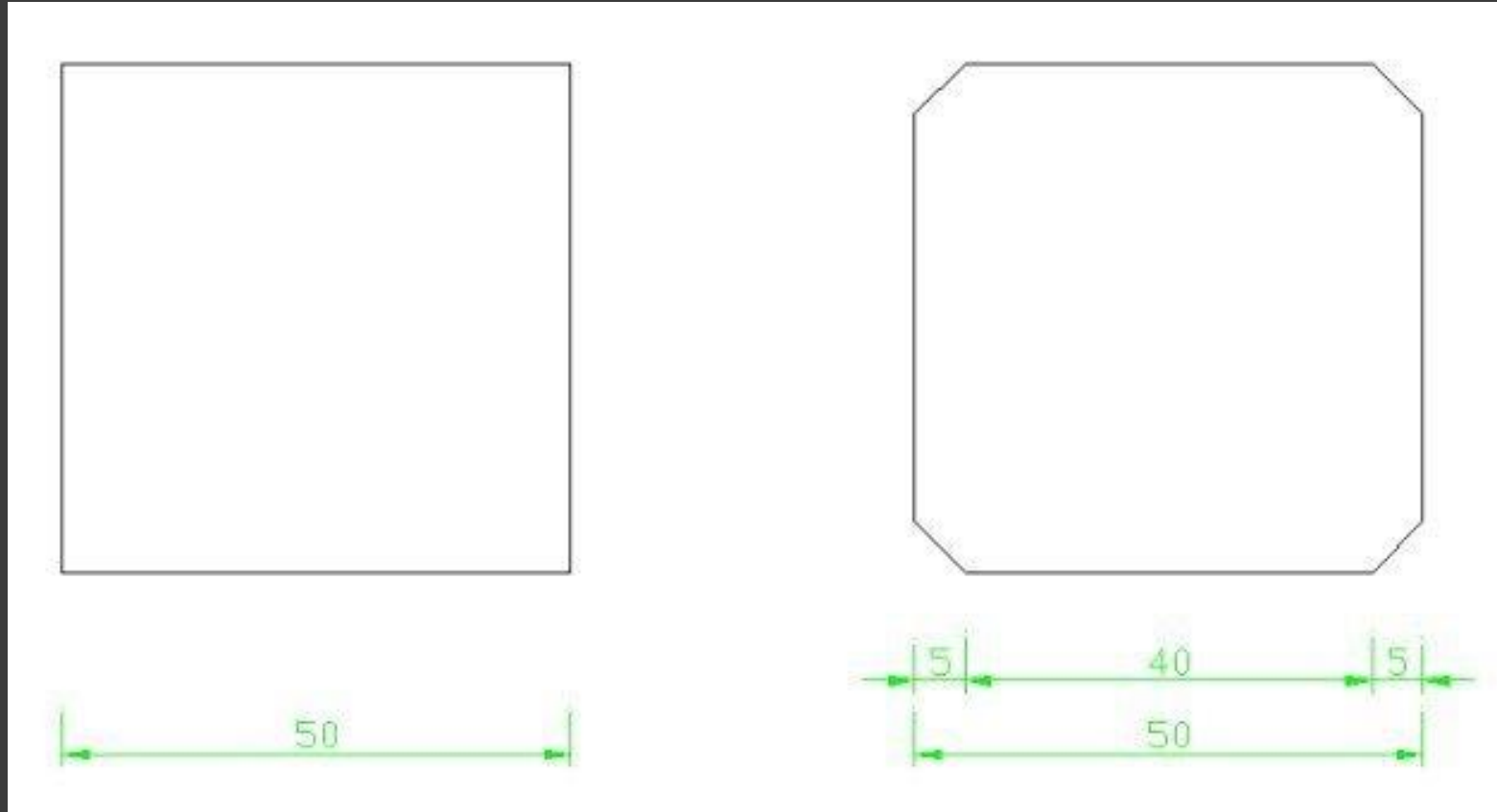
- The chamfer command will chamfer the intersection of two lines to a specified distance.



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# Chamfer



# Fillet

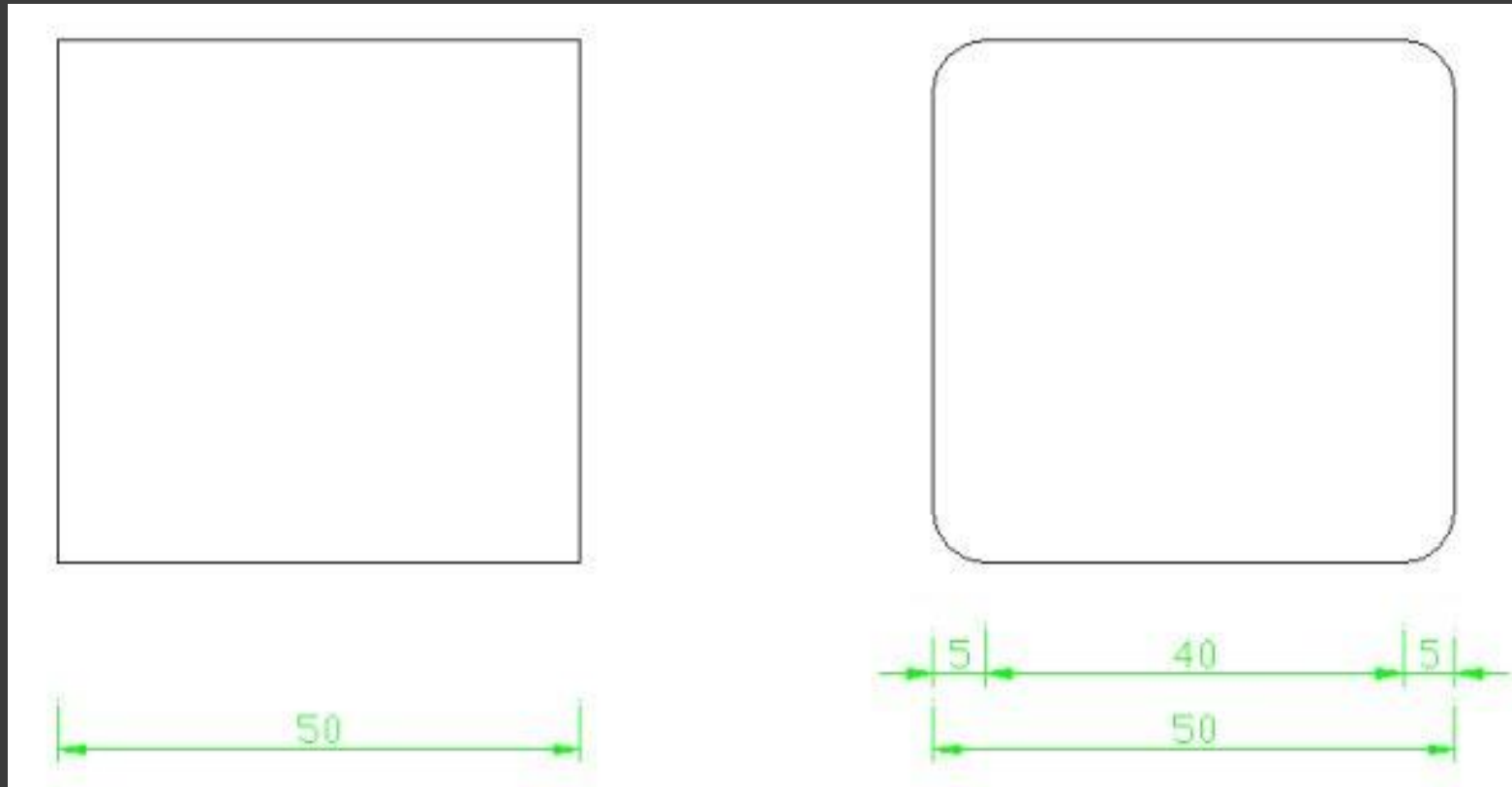
- The fillet command is very similar to the chamfer command above, except instead of creating a straight line chamfer, Autocad creates a radius between the two points.



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# Fillet



# Explode

- The explode command is very straight forward. It simply breaks down an object down to its basic line entities.



# ZOOM

- AutoCAD's zoom command toolbar:



# Zoom real time and Pan

- Pan Button



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# Zoom real time and Pan

- While holding the left mouse button, pull the mouse to the left, right, up or down. Notice how the pan tool reflects these mouse movements on screen so we can move around the screen effortlessly.



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# 2D AutoCAD

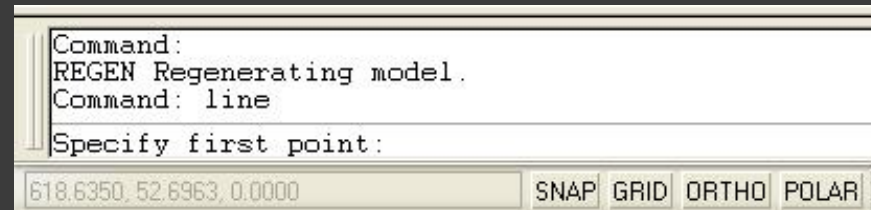


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# How do we give a command?

- Command line



- Toolbars  
(view/Toolbars)

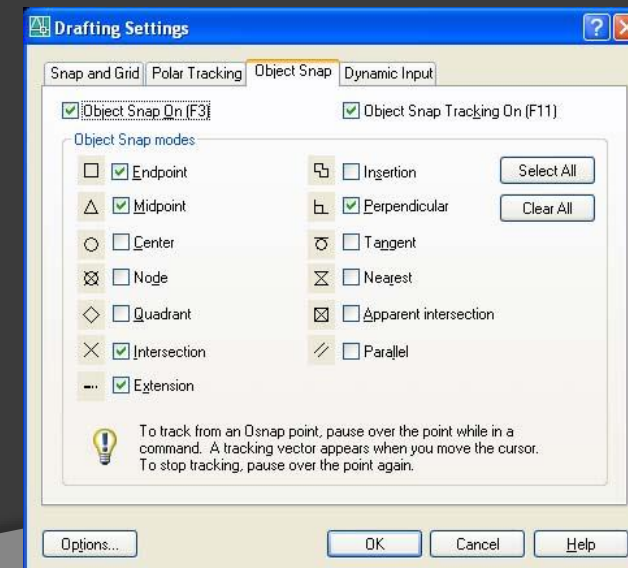


- Drop-down menus

You can pick any one(s) that you are comfortable with.

# What is OSNAP?

- Osnap (Object Snap) settings make it easier to select a 2d object's points
  - Endpoint
  - Midpoint
  - Perpendicular
  - Center
  - Intersection
- Osnap will be active when AutoCAD is expecting you to pick a point on the working area
- Type osnap on your command window:



# Zooming...

- You will need to zoom in and out while drawing with AutoCAD. This doesn't change your objects or UCS, only the way you see your working space. This can be done in many ways:

1. Scroll bars
2. Typing **z** or **zoom** in your command window.
  - All
  - Center
  - Dynamic
  - Extents
  - Previous
  - Scale
  - Window

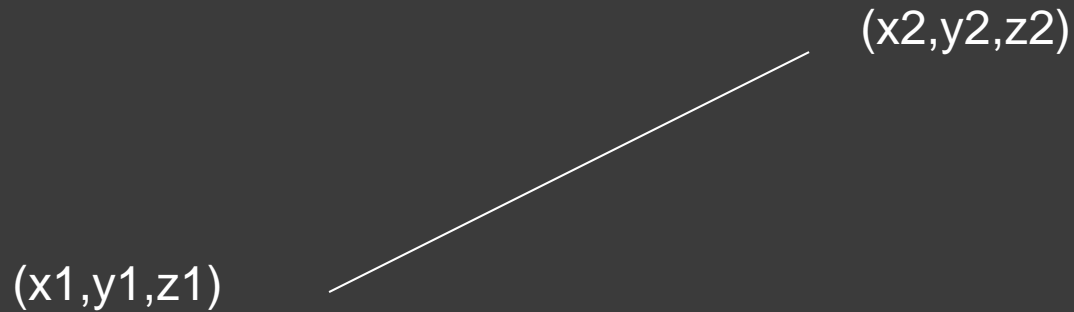


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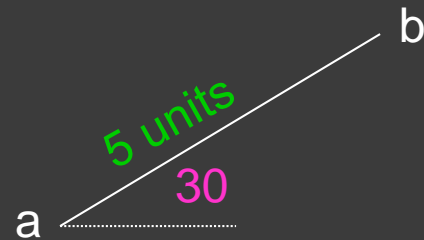
# Lets draw a LINE:

- remember that AutoCAD recognizes an object by its coordinates. You will need two given points to draw a line.



- You can start at a random point on your WCS for your FIRST POINT, but you should specify the coordinates of your SECOND POINT.

# Lets draw a LINE (1):



1. Give the command
  - Type “line” on the command window, OR
  - Click on the line icon on the Draw toolbar, OR
  - Select Line on the Draw menu
2. Specify the first point (a)
  - Click on a random point on your working area (black space)
3. Specify the second point in relation to the first point
  - @distance<degrees
  - @5<30

# Lets draw a LINE (2):

a 7 units b

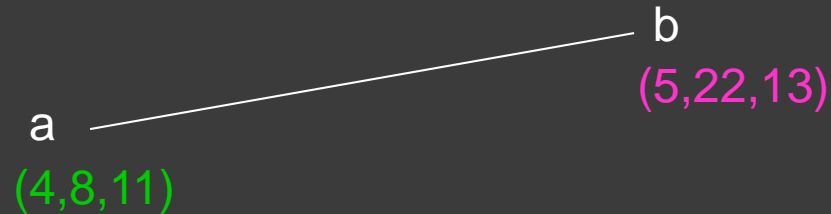
1. Hit F8 (ortho on)
2. Give the command
3. Specify the first point
4. Specify the second point in relation to the first point
  - Point the cursor to the left hand side. You will see that the cursor snaps only to  $0^{\circ}$ - $90^{\circ}$ - $180^{\circ}$ - $270^{\circ}$
  - Type 7 and hit enter



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# Lets draw a LINE (3):



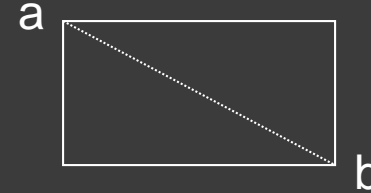
If we know the coordinates of the line we want to draw, we can simply type them into the command line. (However, this mostly is not the case.)

1. Give the command
2. Specify the first point (4,8,11)
3. Specify the second point (5,11,23)

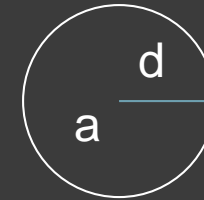


# What else?

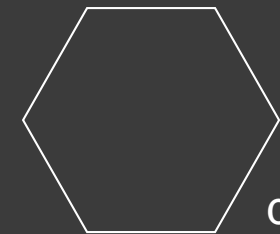
- Rectangle: two diagonal lines  
(pick first point, select the second one with relation to the first. @5<-33)



- Circle/Arc: center and radius



- Polygon: specify the number of edges and length of a side



**A small tip:** you can use the EXPLODE command for the tool to split the object into its components or lines

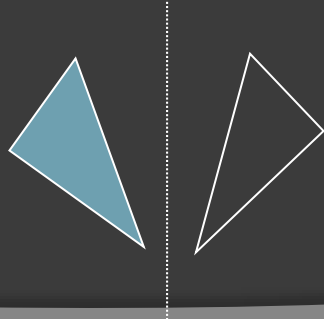
# Editing...

## ● COPY

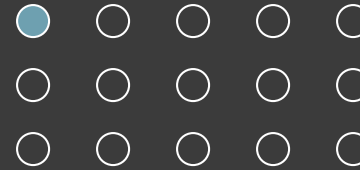


## ● MOVE

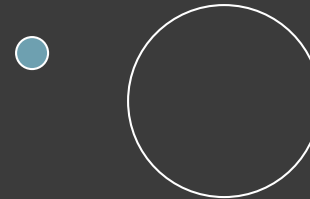
## ● MIRROR



## □ ARRAY



## □ SCALE



## □ STRETCH



# More editing...

☐ OFFSET



☐ ROTATE



☐ ERASE

☒ STRETCH

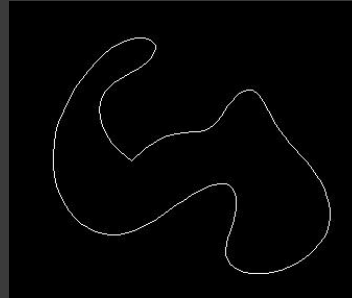
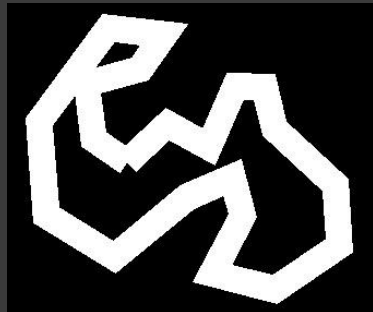
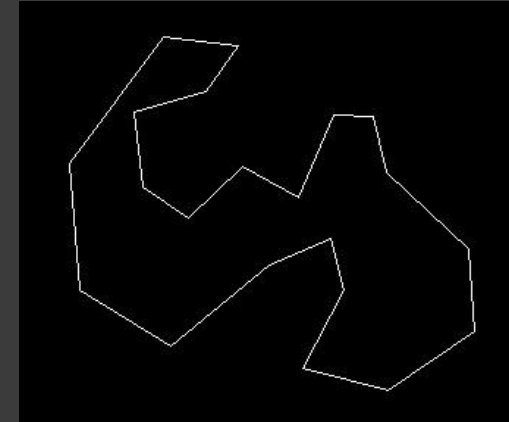


☒ EXPLODE

# Polyline (pline)

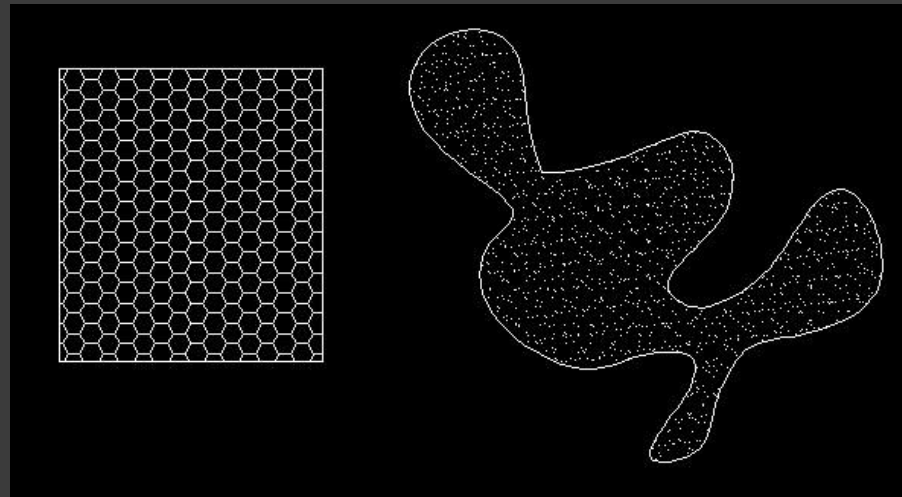
- The **PLINE** command creates a chain line with multiple vertexes and straight or circular segments between the vertexes
- Draw a closed shape with the pline command:
- Then type **PEDIT** (**p**olyline **e**dit) on the command line and see what you can do with it:  
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]:

Try **Width** and **Spline**



# Hatching

- Hatching is used to add shaded patterns to objects and shapes within an Autocad drawing. Hatch patterns can be used to indicate a material to be used, such as a concrete hatch. Alternatively it could be used to make an area of a drawing stand out.
- You will pick:
  - Pattern
  - Scale
  - Angle
  - points



# Drawing SOLIDS

- Solids contain the “mass properties” of 3D objects.
- You can use the Solids toolbar for readily accessible objects
  - Box
  - Cylinder
  - Wedge
  - Torus
  - Cone
  - Sphere
- You can use the **Boolean** operations of more complicated shapes.
  - Union (join two solids)
  - Subtract (carve out the second solid from the first)
  - Intersection (only the common area)

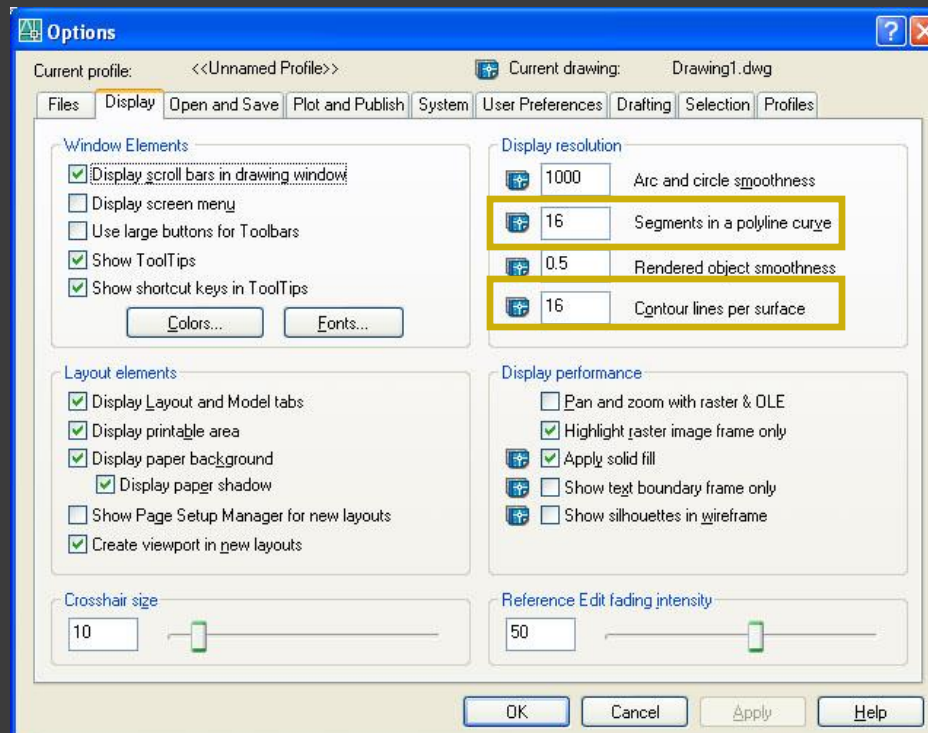


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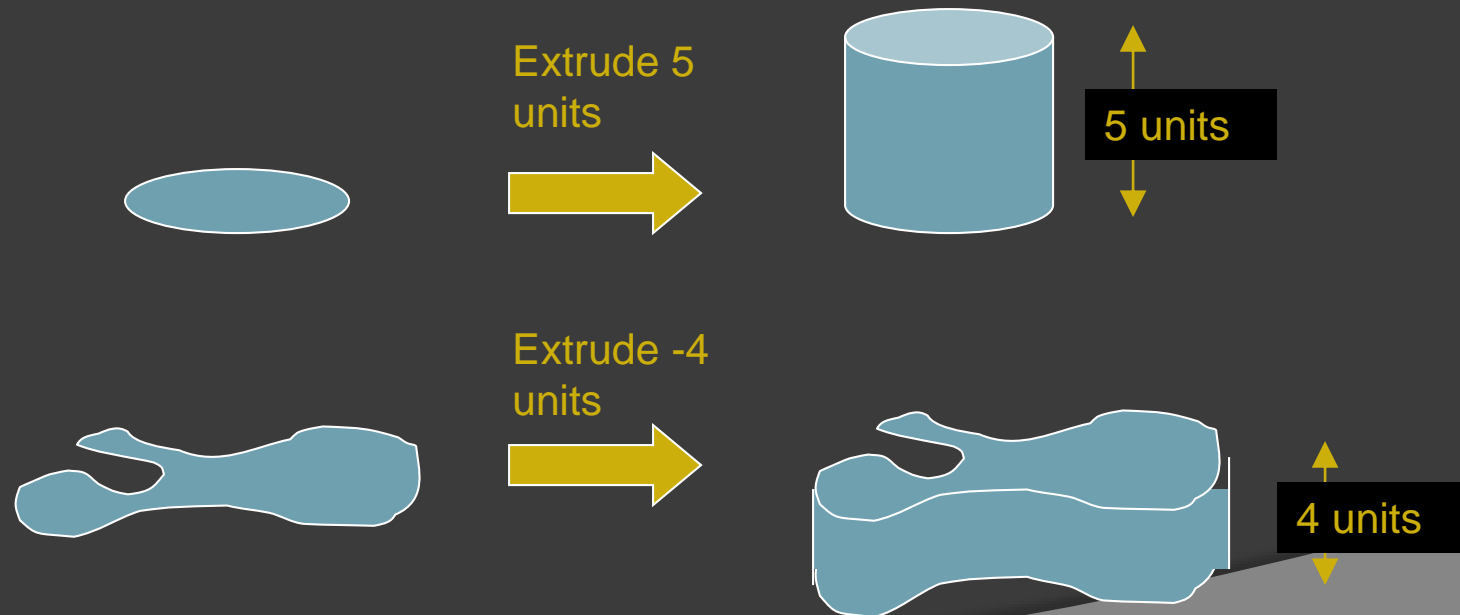
# To increase the number of faces

- Tools > Options > Display tab >



## We can create solids by extruding as well

- If you “**extrude**” a surface into the third dimension, you simply add a thickness in section. This basically is same as creating a “solid” object





# UCS and WCS

- The AutoCAD world is 3 dimensional. However, if we want to draw a 2d object, such as a plan or a section, we will use only 2 dimensions (x and y).
- WCS (world coordinate system) is the imaginary plane that is parallel to the ground. It is the default coordinate system.
- Modifications made to the World Coordinate System (WCS) result in a User Coordinate System (UCS). It is the plane that you work on. It enables the user to draw 3 dimensional objects.
- To create a new UCS, type ucs on the command window, then say New and specify 3 points on your new UCS plane.



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# Lets change the UCS

- —Prompt: **Command:** Type **ucs** and press **Enter**.
- —Prompt: **Enter and option [New/Move/ orthoGraphic /Prev/ Restore/Save/Del/ Apply/?/World] <World>**: Type in **n** for the **New** option and press **Enter**. —Prompt: **Specify origin of new UCS or [Zaxis/ 3point/ OBject/ Face/ View/X/Y/Z <0,0,0>**: Type **3** to select the **3point** option and press **Enter**.
- —Prompt: **Specify a new origin point <0,0,0>**: Use an **Osnap** or **coordinate** to select the position for a new **0,0,0** point (this selects the junction of the X, Y, and Z axes). **NOTE:** The **intersection Osnap** will NOT work with solid models.
- —Prompt: **Specify point on positive portion of X-axis <1.000,0.000,0.000>**: Use an **Osnap** or **coordinate** to select the direction of the desired **X-axis**. **NOTE:** The numbers inside the brackets **< >** will not necessarily be the same as those shown in this example.
- —Prompt: **Specify point on positive-y portion of the UCS XY plane <1.000,0.000,0.000>**: Use an **Osnap** or **coordinate** to select the direction of the **Y-axis**. The UCS icon should change position to indicate the new UCS orientation.



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# 3D AutoCAD



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# 3D AutoCAD (Advantages of 3D models)

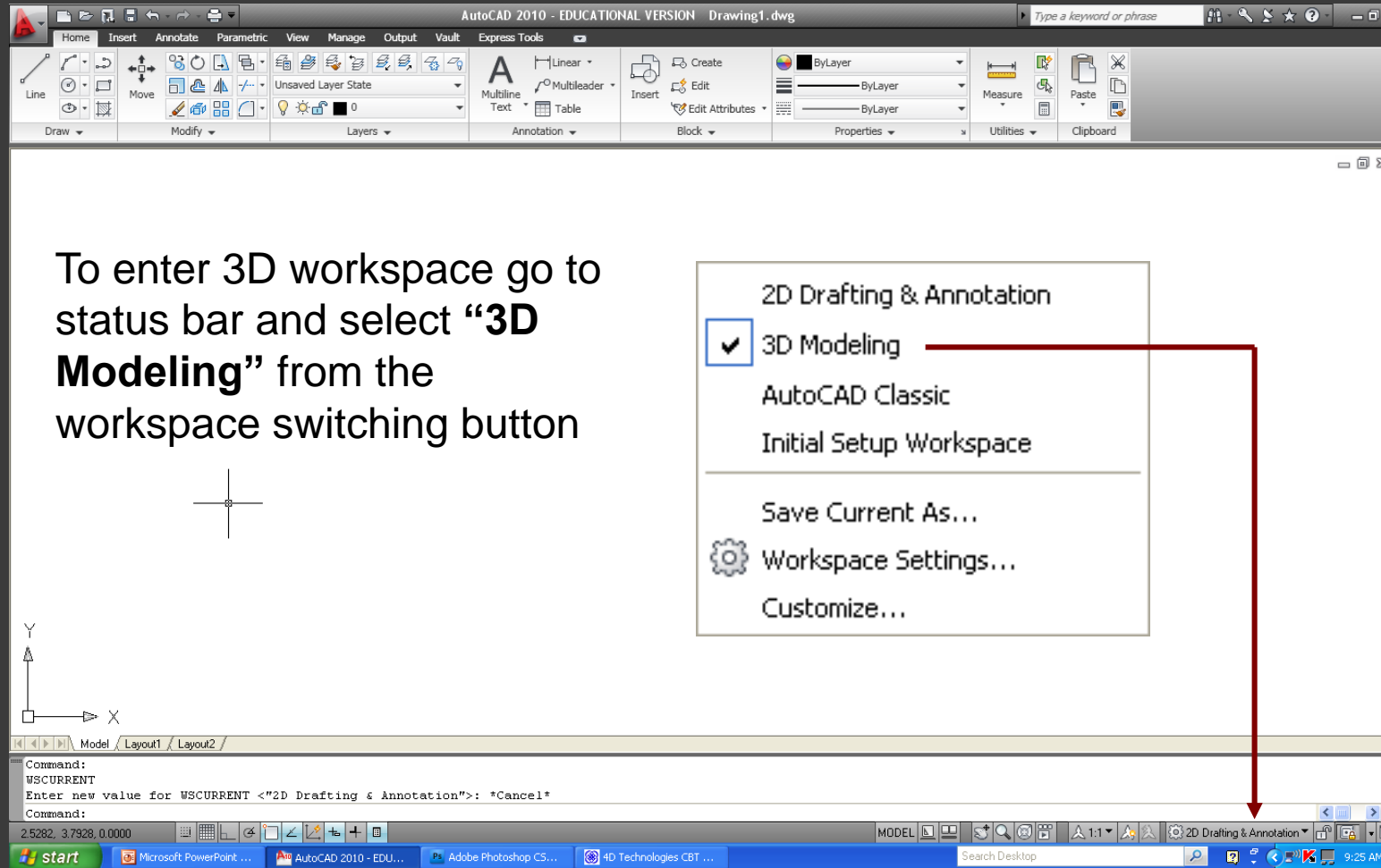
- Models can be rotated or viewed from any position
- You can generate **section** and **auxiliary views**
- You can **remove hidden lines** and do **realistic shading**
- You can run **part interference checks** for **engineering analysis**
- You can add **lighting** and **realistic rendering** to models
- You can **create animations**
- You can **extract manufacturing data** for making the part



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# 3D AutoCAD (The 3D Workspace)



# 3D AutoCAD (3D Viewing & Navigation Tools)

## ViewCube

**Compass ring**- rotate

**Faces**- standard orthographic view

**Corners**-  $\frac{3}{4}$  view

**Edges**-  $\frac{1}{2}$  view

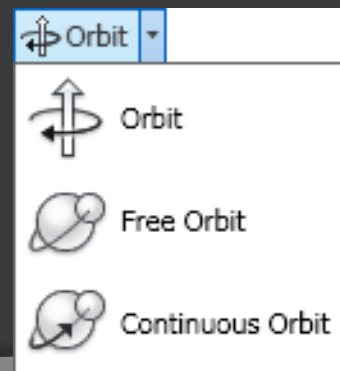


## 3D Orbit (View Menu)

Uses cursor

Camera moves, not the object

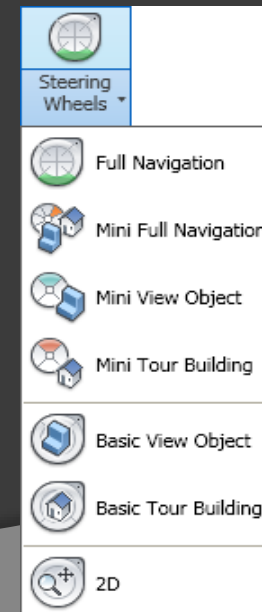
To use hold down “**Shift Key**” and **middle mouse button** together



## Steering Wheel

4 different kinds

Can place anywhere on screen



# 3D AutoCAD (Visual Styles)

Group of settings that control edges and shading in the viewport

Opens and docks on left of screen

There are five default styles:

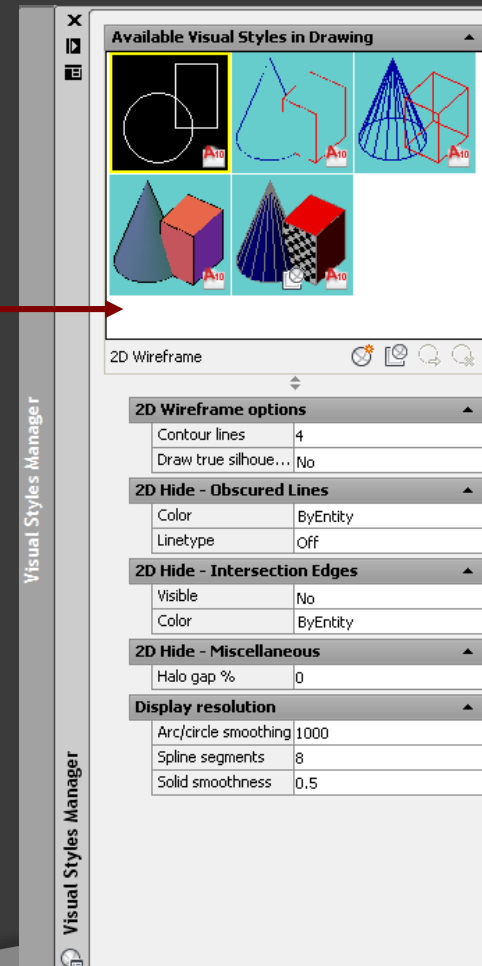
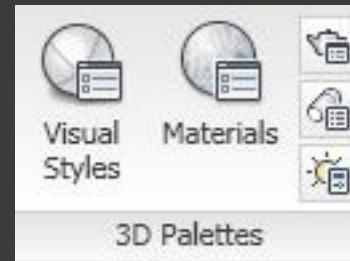
**2D Wireframe**- NOT use full in 3D

**3D Hidden**- Hides back faces

**3D Wireframe**- XYZ planes shaded

**Conceptual**-Smooths polygons and uses warm colors to contrast faces

**Realistic**- Shades faces and applies any preselected materials (i.e., brick)



**NOTE:** You can create your own visual style and add to list

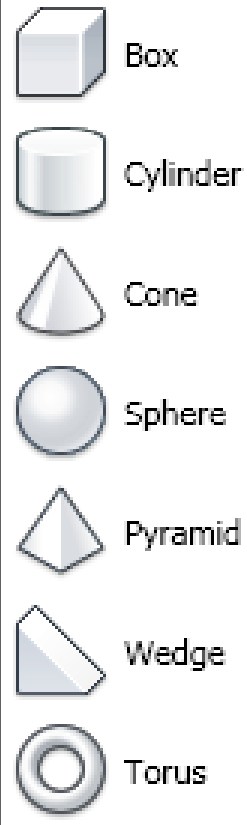


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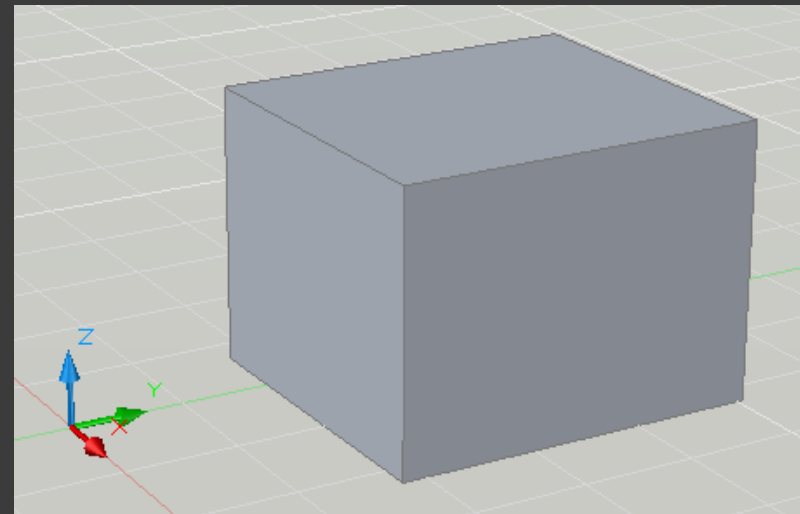
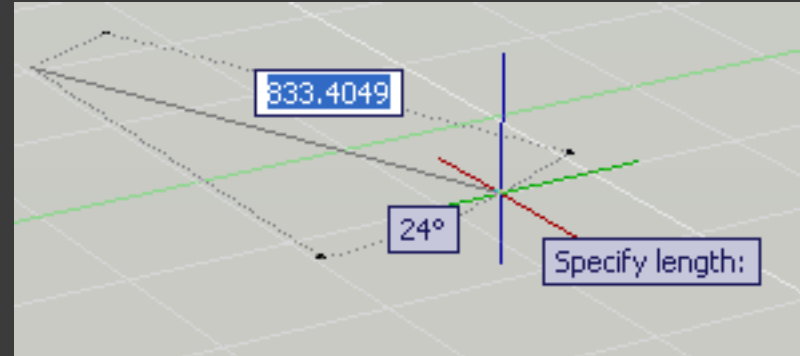
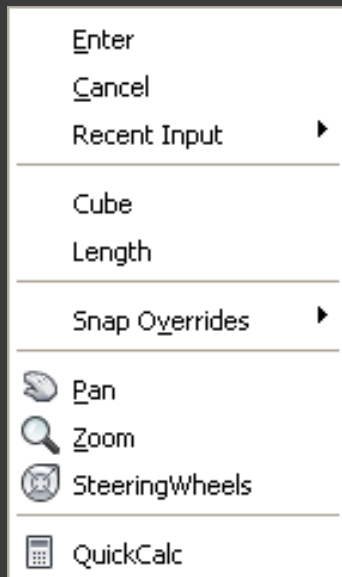
This project is funded by the European Union.

# 3D AutoCAD (Creating 3D Objects)

## 3D Primitives



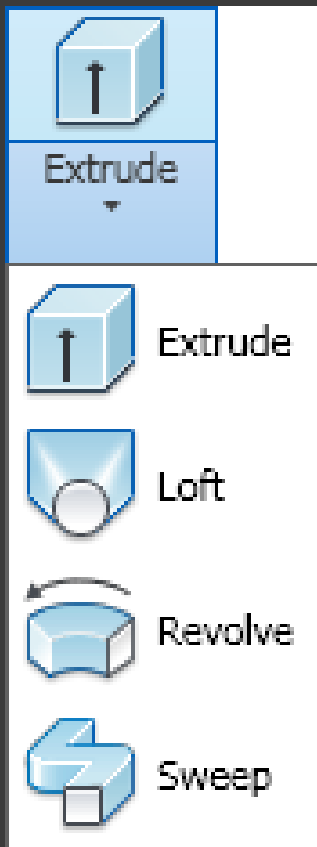
With “**Dynamic Input**” turned on you can right-click and select different options that allow you to change the angle of the primitive



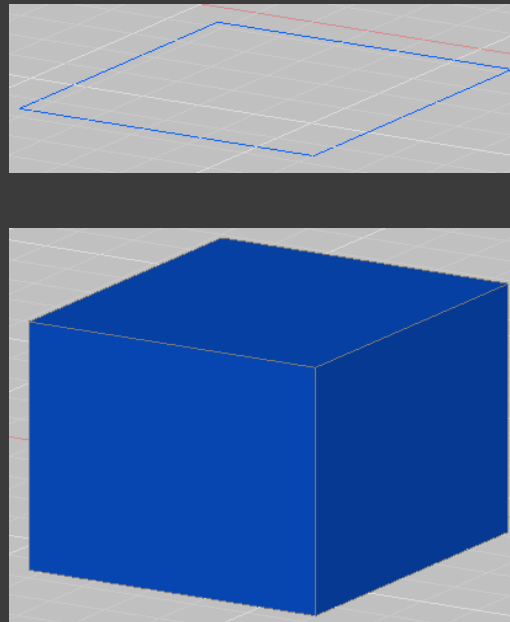


# 3D AutoCAD (Creating 3D Objects)

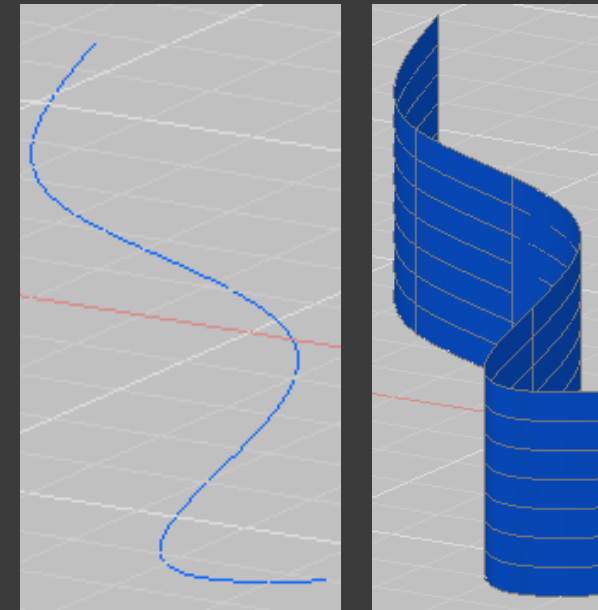
## Extrude Commands



A closed polyline will create a 3D Solid Model

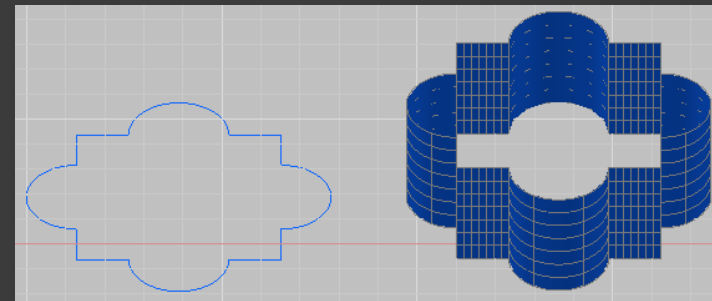
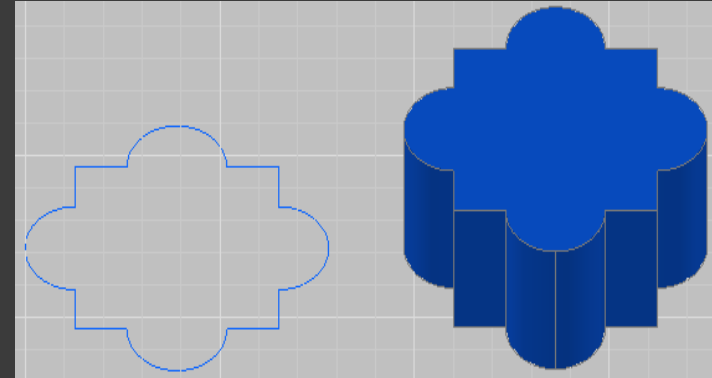


An open polyline will create a surface model



# 3D AutoCAD (Convert 2D Line Segments into 3D Models)

NOTE: In order for 2D lines and/or arcs to be extruded into a solid 3D model you must use the “**Edit Polyline Command (PEDIT)**” to convert the separate line segments into one **closed** polyline.



# 3D AutoCAD (Extrude Options)

## Direction

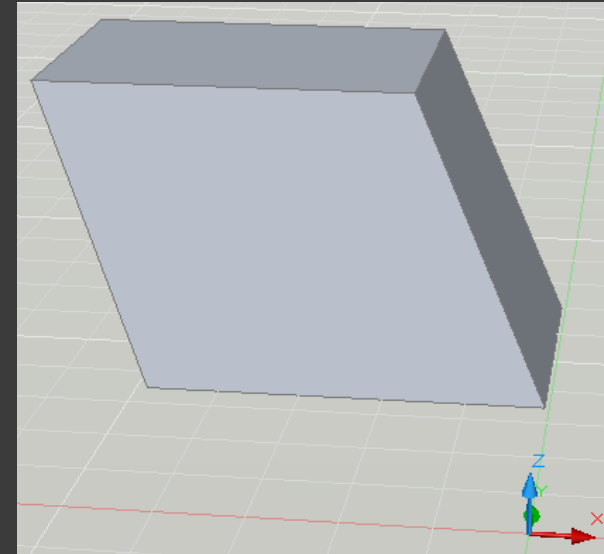
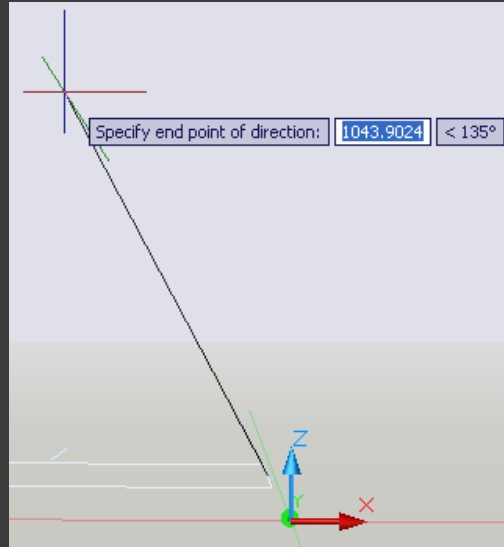
Specifies the length and direction of the extrusion with two points you specify. (The direction cannot be parallel to the plane of the sweep curve created by the extrusion.)

*Start point of direction.*

Specifies the first point in the direction vector.

*End point of direction.*

Specifies the second point in the direction vector.

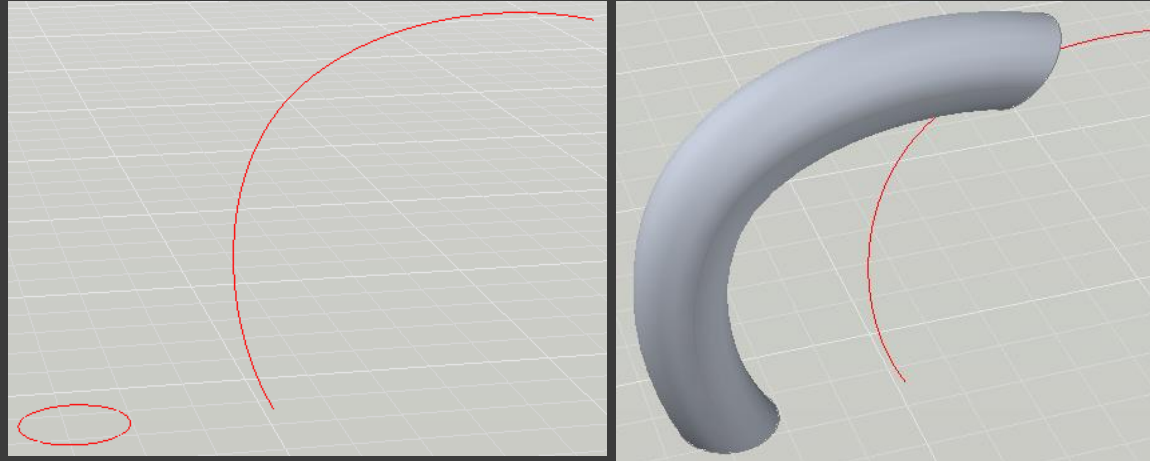


# 3D AutoCAD (Extrude Options)

## Path

Specifies the extrusion path based on an object that you select. The path is moved to the **centroid** of the profile. Then the profile of the selected object is extruded along the chosen path to create solids or surfaces.

The path should not lie on the same plane as the object, nor should the path have areas of high curvature.



# 3D AutoCAD (Extrude Options)

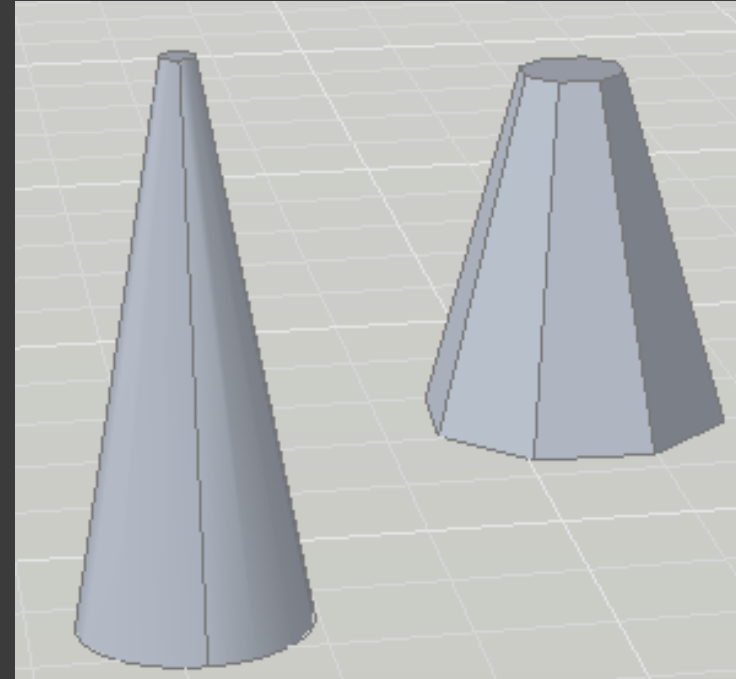
## Taper Angle

Sets an angle of taper for an extrusion. Positive angles taper in from the base object. Negative angles taper out. The default angle, 0, extrudes a 2D object perpendicular to its 2D plane. All selected objects and loops are tapered to the same value.

Specifying a large taper angle or a long extrusion height can cause the object or portions of the object to taper to a point before reaching the extrusion height.

Individual loops of a region are always extruded to the same height.

When an arc is part of a tapered extrusion, the angle of the arc remains constant, and the radius of the arc changes.



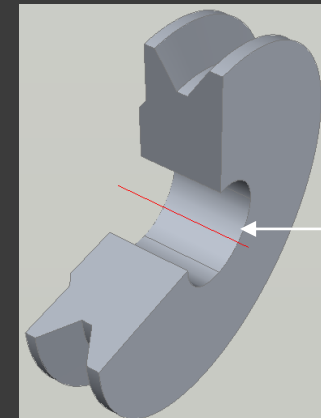
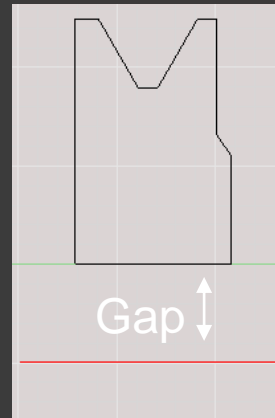
# 3D AutoCAD (Revolve Tool)

## Revolve Tool

You can revolve closed objects to create 3D solid models and open objects to create surface models.

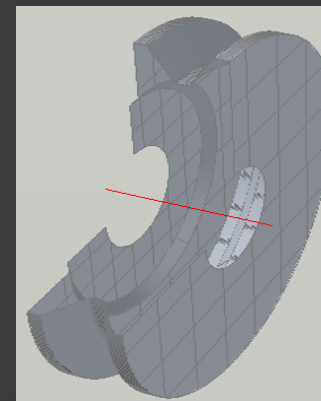
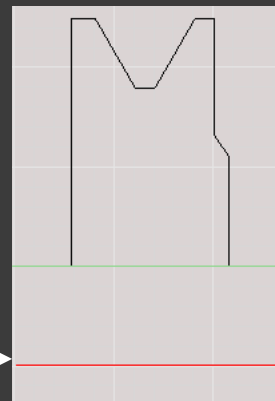
Objects can be revolved **360 degrees** or another specified angle.

You can revolve more than one object.



Solid

Gap  
creates  
hole



Surface

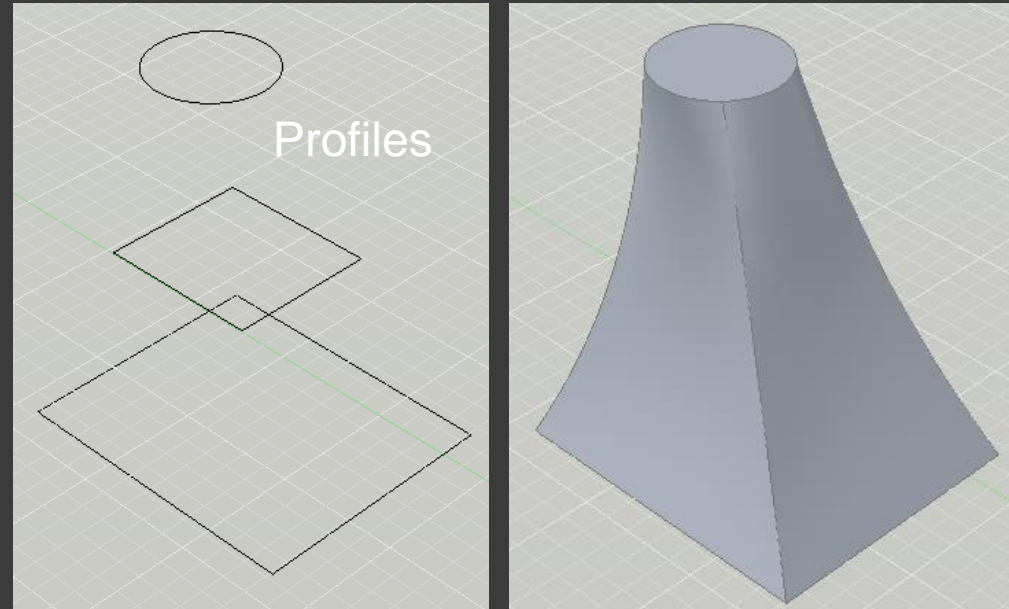
# 3D AutoCAD (Loft Tool)

## Loft Tool

Create a 3D solid or surface by lofting a profile through a set of two or more cross-section profiles.

The cross-section profiles define the shape of the resulting solid or surface object. You must specify at least two cross-section profiles.

The cross sections that you use when lofting must be all open or all closed. You cannot use a selection set that includes both open and closed curves.

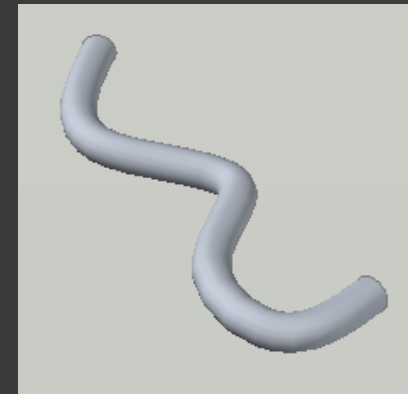
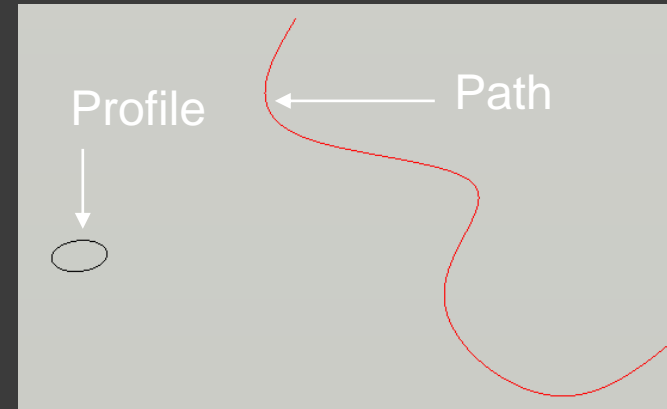


# 3D AutoCAD (Sweep Tool)

## Sweep Tool

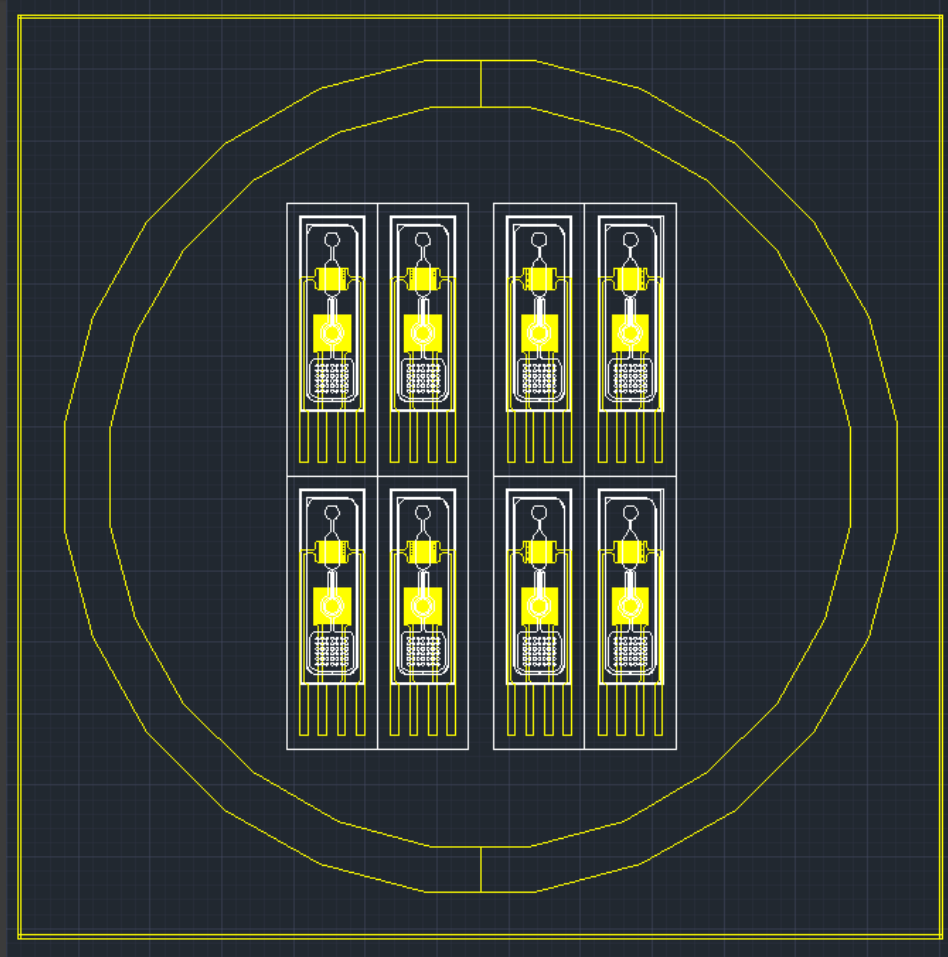
Create a new solid or surface by sweeping a planar curve (profile) along a path.

The **SWEEP** command draws a solid or surface object by extending a profile shape (the swept object) along a specified path. When you sweep a profile along a path, the profile is moved and aligned normal (perpendicular) to the path. If you sweep a closed curve along a path, the result is a solid. If you sweep an open curve along a path, the result is a surface.

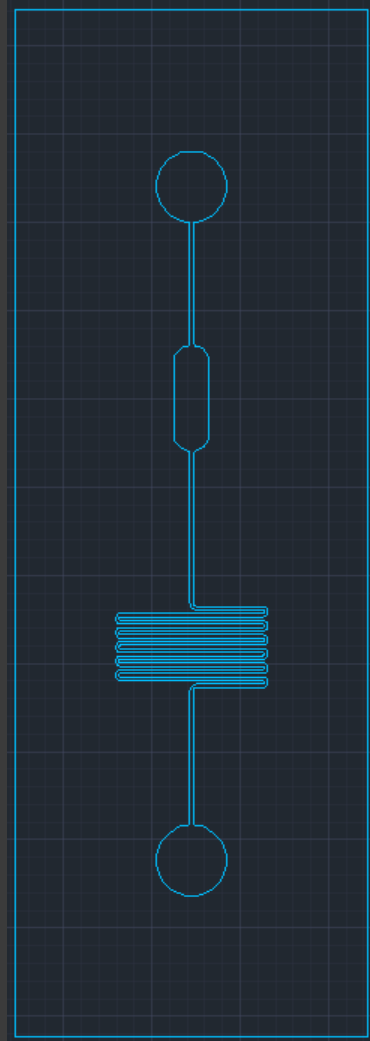




# Use of AutoCAD: Mask design for Microfluidics



# Simple Microfluidics



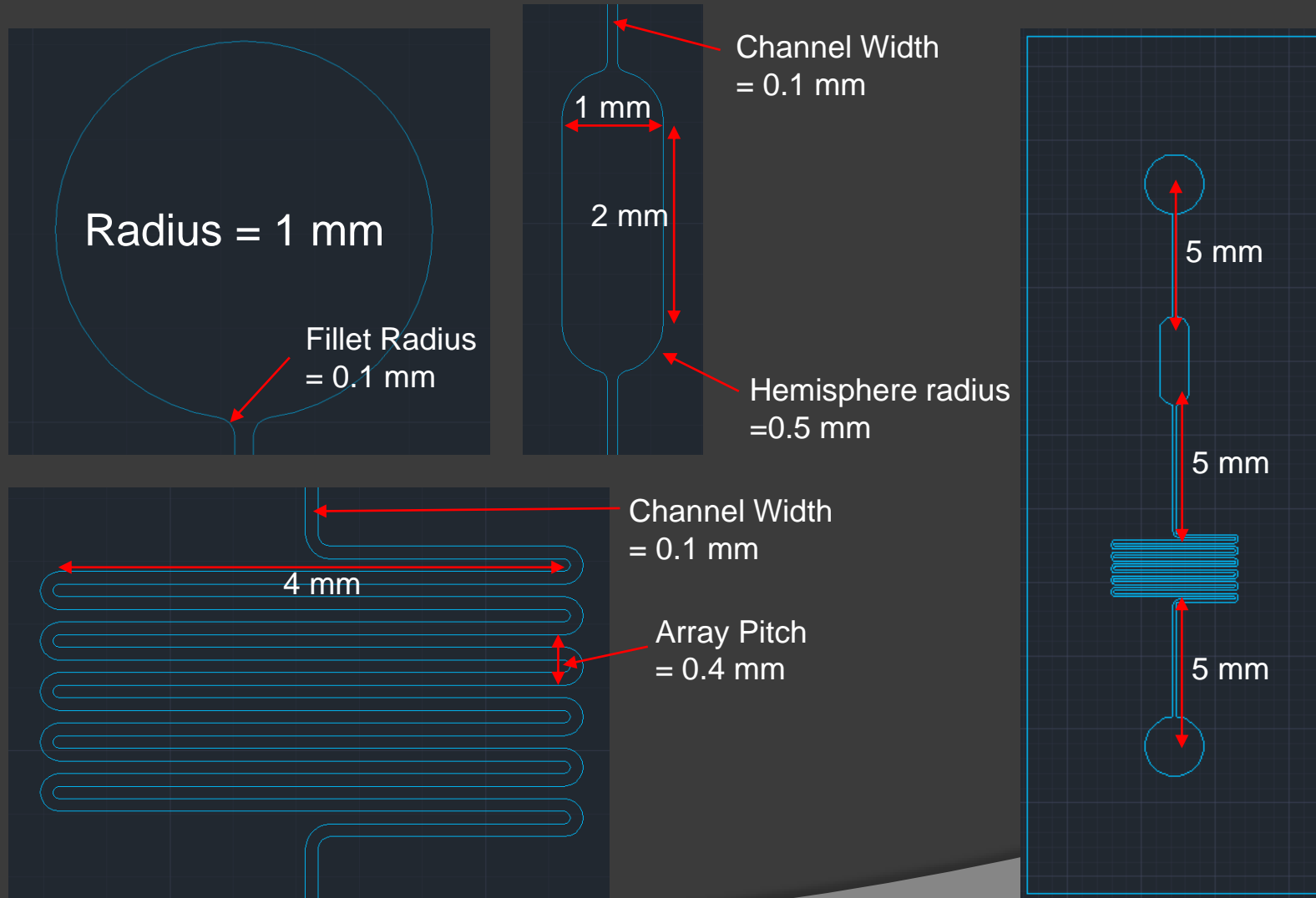
Inlet

Reaction Chamber

Mixer

Outlet

# Characteristics



# HOME WORK



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*This project is funded by the European Union.*