

SOEM 024:

Computer Aided Design

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3D Design with AutoCAD 2002

- Isometric Drawings
- 3D coordinates, views
- Wire-frame
- 3D modelling, extruding
- Primitive objects
- Boolean operators

Terminology

- **Boolean operations:** Commands that allow you to add, subtract or intersect solid objects in AutoCAD.
- **Complex surface:** Generally a curved surface. Examples: car fender, landscape contour.
- **Elevation:** The difference between an object being at zero on the Z-axis and the height that it is above zero.
- **Face:** The simplest true 3D surface.
- **Facet:** A three or four sided polygon that represents a piece (or section) of a 3D surface.
- **Hidden line removal:** A way of hiding lines that would not be visible if you were viewing the actual object you have drawn in AutoCAD. (Command: HIDE)
- **Isometric Drawing:** A simple way of achieving a '3D' appearance using 2D drawing methods.
- **Plan View:** Also known as the top view, a plan view looks directly down the WCS Z-axis to the X-Y axis.

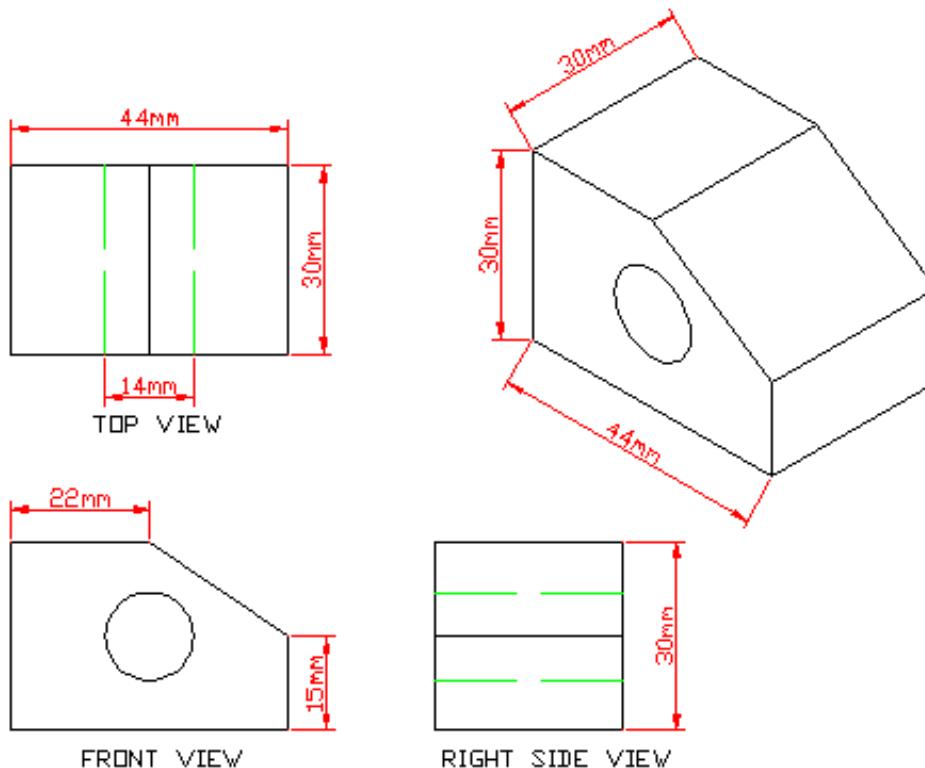
Terminology

- **Primitive:** A basic solid building block (cones, cylinders)
- **Rendering:** A complex way of adding photo-realistic qualities to a 3D model you have created.
- **Shading:** A quick way of adding colour to a 3D object you have drawn. (Command: SHADE)
- **Solid Model:** A 3D model creating using solid 'building blocks'. This is the most accurate way of representing real-world objects in CAD.
- **Surface Model:** A 3D model defined by surfaces. The surface consists of polygons. (See facets.)
- **Thickness:** A property of lines and other objects that gives them a 3D like appearance.
- **Wire-frame Model:** A 3D shape that is defined by lines and curves. A skeletal representation. Hidden line removal is not possible with this model.
- **Z-Axis:** The third axis that defines the depth.

ISOMETRIC DRAWING

- This is the simplest way to give a 3D representation while using only 2D commands. This has been the usual way of doing things before CAD allowed true 3D work to be done. Many times an isometric drawing is used to compliment a 3 view orthographic drawing.
- The basic isometric drawing of the object gives a very good idea of what it looks like. If this is all that is needed then isometric works well. As soon as you change anything, like height, then you'll need to redraw all four views.
- AutoCAD has a command called **ISOPLANE** which allows you to easily draw at a 30 degree angle as needed for an isometric drawing. You can switch between the three '*isoplanes*', by using this command or by pressing the **F5** key.

Exercise 1, isometric drawing



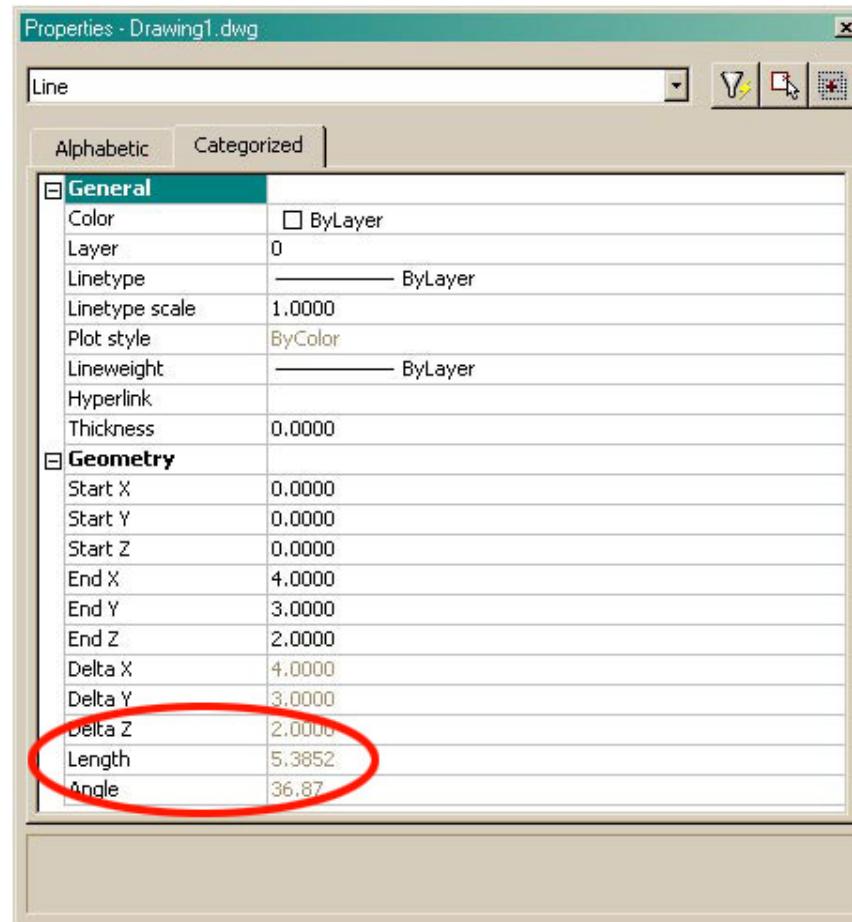
3 Dimensions

- When you entered points previously, you would enter them in the format: X,Y. By doing this, you let AutoCAD know that in these cases, Z was equal to zero.
- Entering 4,3 would be the same as entering 4,3,0.
- Now if you drew a line from the origin (0,0,0) to a point at 4,3,2, you would get a line that goes 4 drawing units (du) to the right, 3 du up and 2 du towards you.

3 Dimensions

Notice that the line is actually 5.3852 long. If you were to look at it from the plan view, it would look exactly like a line drawn from 0,0 to 4,3. The difference is that a line to 4,3,0 would only be 5 long.

Properties of the line

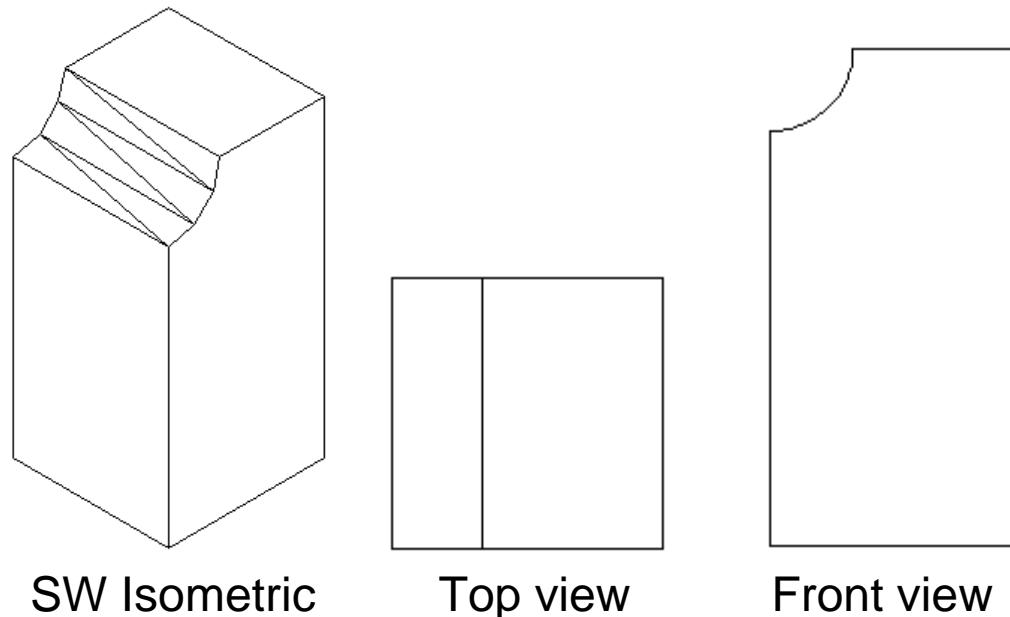


Angles in 3D

- You also have to know how AutoCAD measures angles of rotation in 3D. There is a somewhat simple rule for this called "The Right Hand Rule".
- To figure out which is the positive rotation angle, imagine that you are wrapping your right hand around the axis with your thumb pointing towards the positive end.
- **The direction that your fingers are wrapped is the positive direction.** This applies to all three axes.

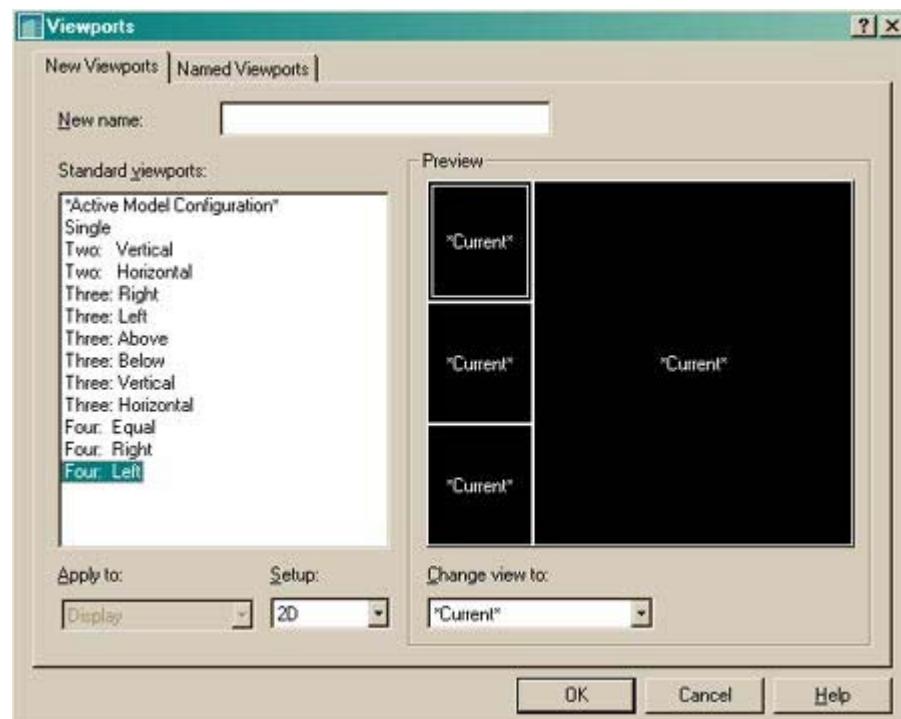
Viewing 3D objects

- For 3D views use the pull-down menu options. Choose **View>3D Views >** (then select one of the four bottom *isometric* options).
- It is most common to use the **Southwest** view, as this keeps your positive X and Y axes in a logical position. For most of your work in this course, stick with this view.



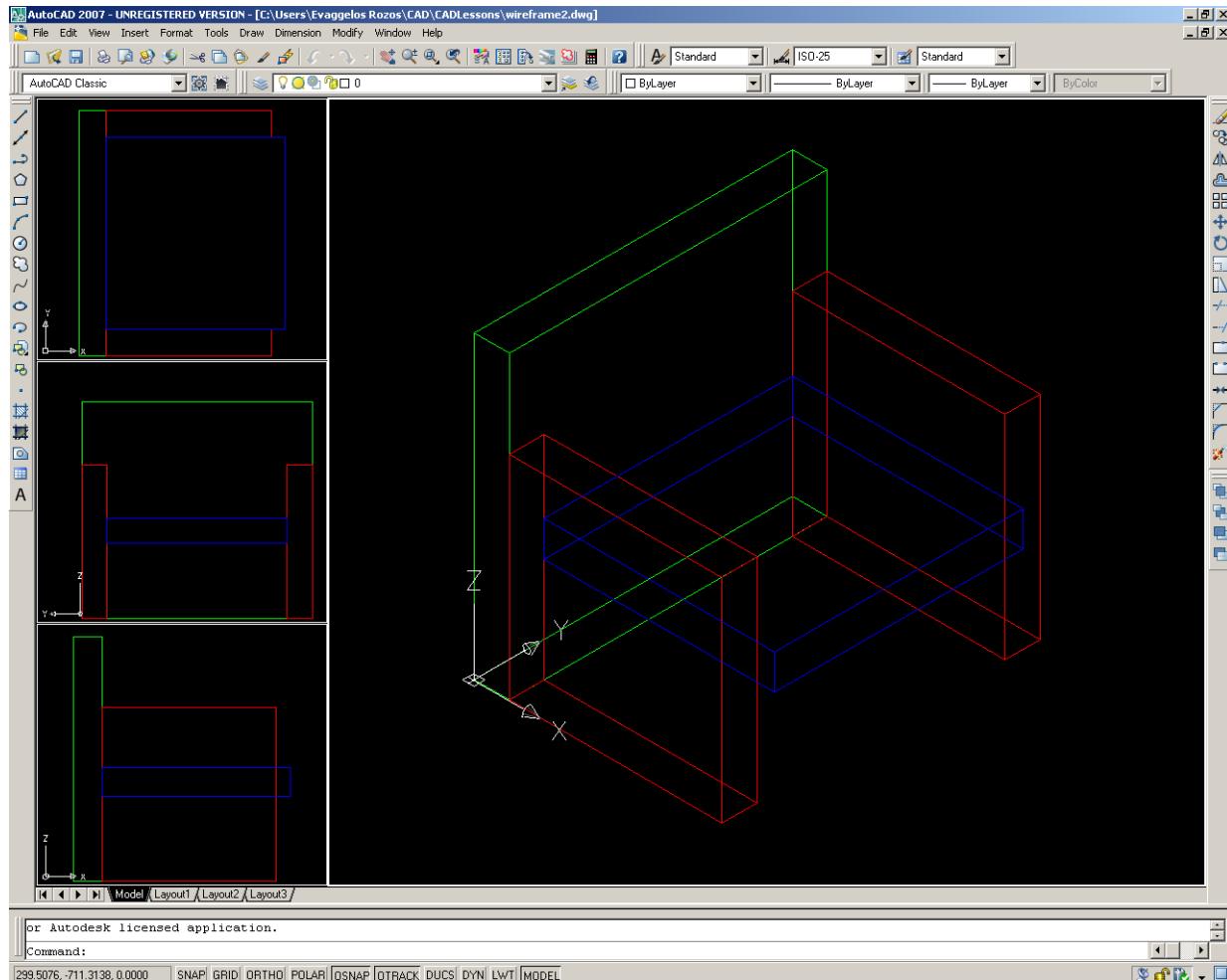
Views and viewports

- You can have viewports in either model space or paper space. You can use one of the prefixed viewport configurations (View → Viewports → New Viewports)
- When you have more than one viewport, click inside the one you want active. Every view operation (zoom, pan) will be restricted to the active viewport.
- Manage and save the current viewports with the VIEW command.

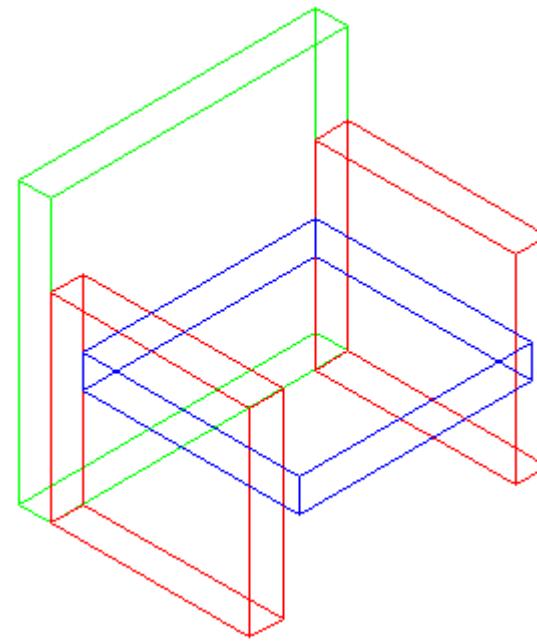
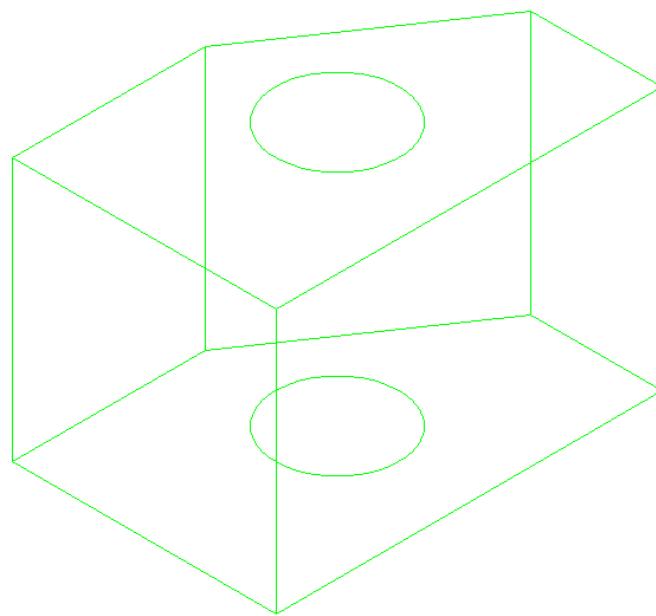


Views and viewports

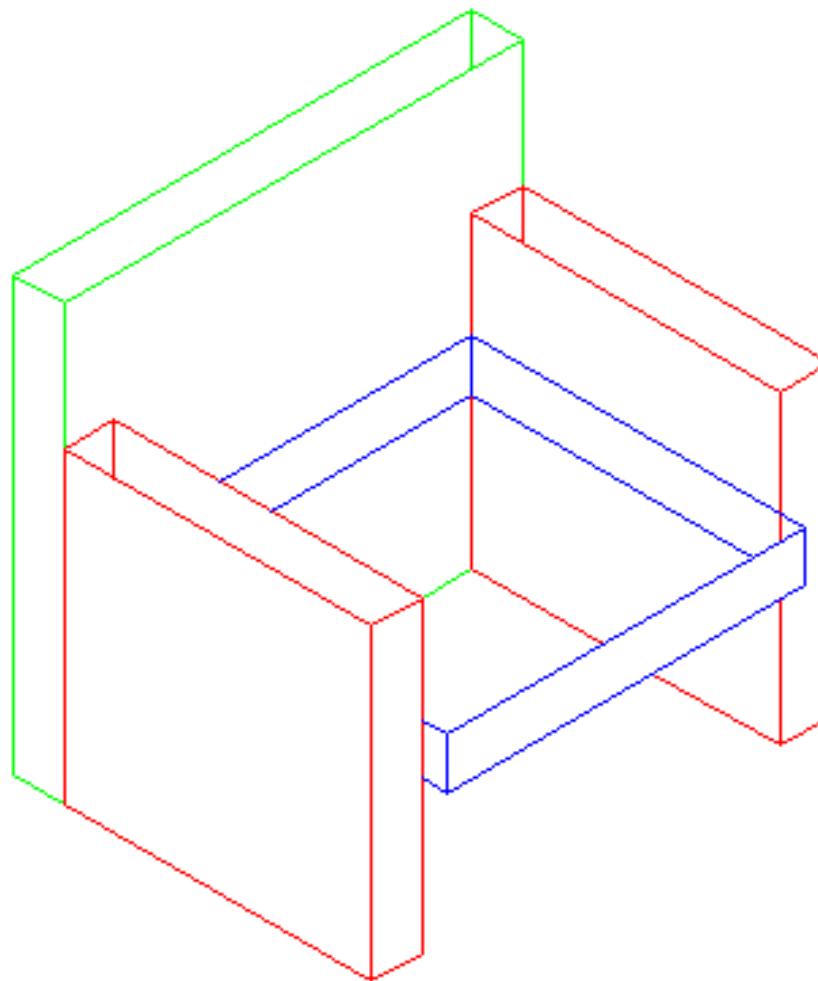
In 3D design
any
modification to
the object
automatically
reflects to all
views.



Exercise 2, wireframe modelling



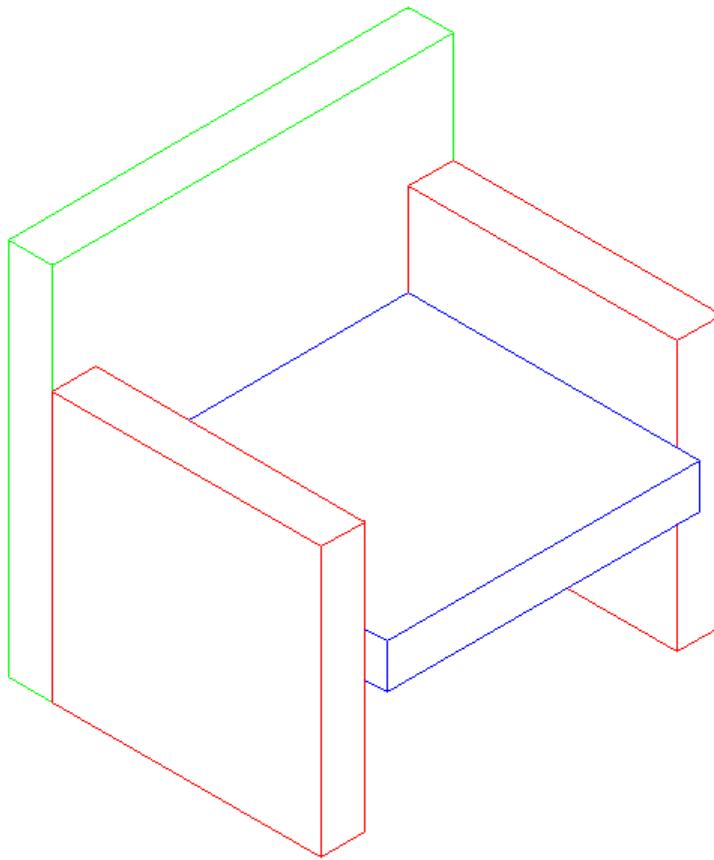
Exercise 3, line thickness



Regions and 3D surfaces

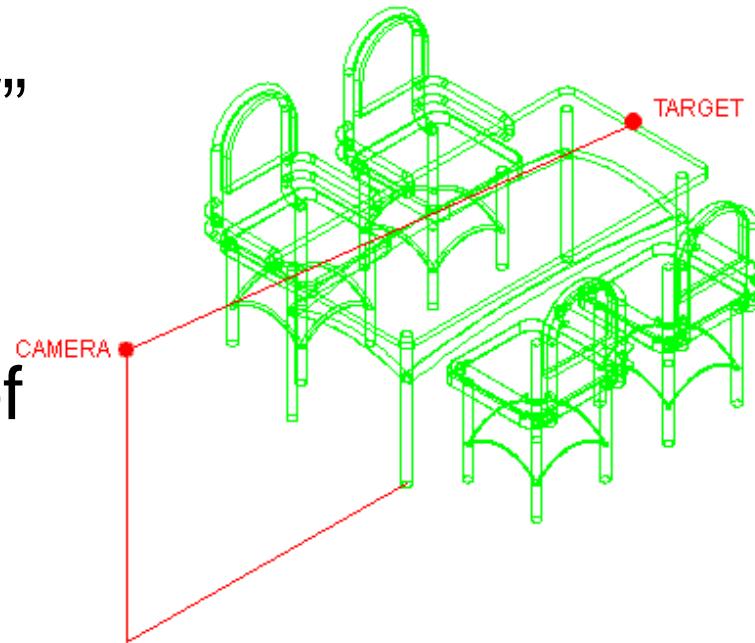
We will now begin using true 3D commands. First you will create a 2D **region** and then **extrude** it into a 3D solid.

Exercise 4, daw 3D objects

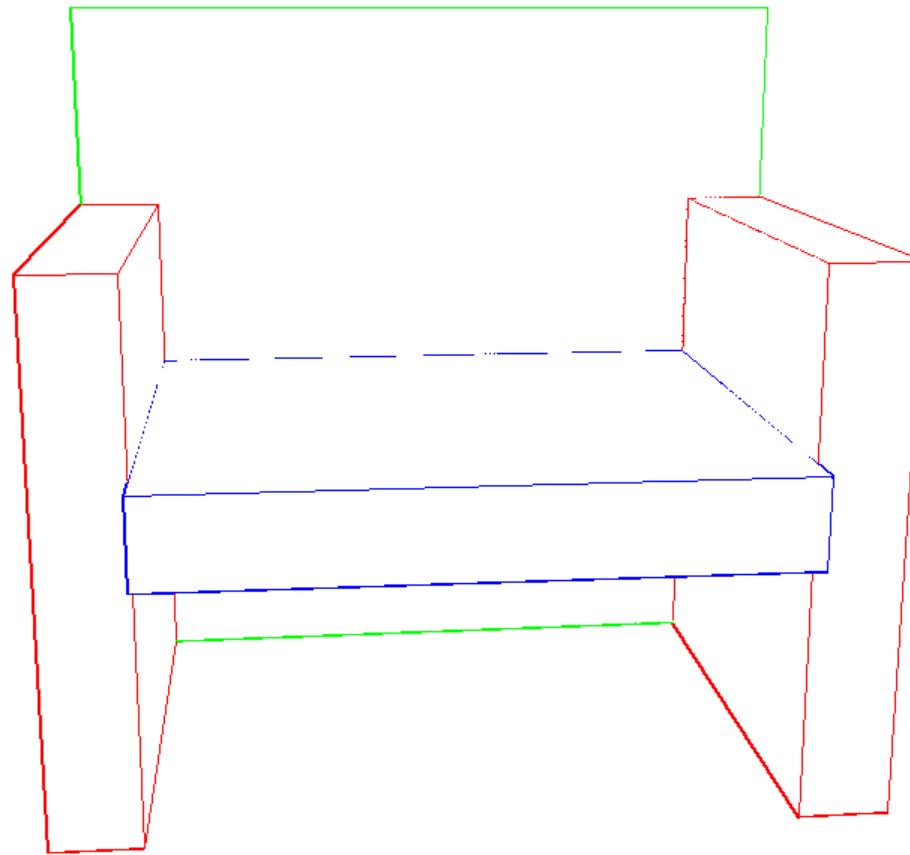


Perspective views

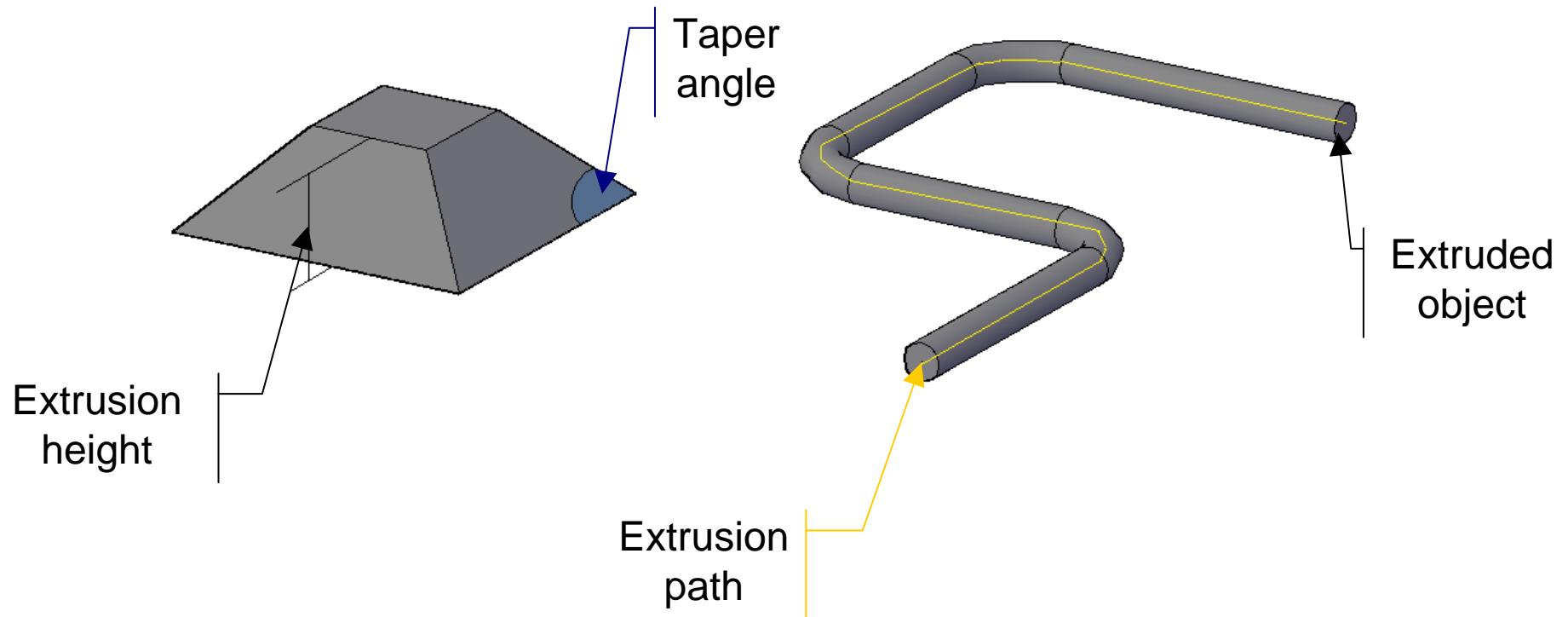
Perspective view is controlled with the “DVIEW” command. The basis of generating a perspective view is to have a virtual camera and target. Think of where you would like to 'stand' (the camera) and what direction you want to look at (the target).



Exercise 5, 3D object viewing

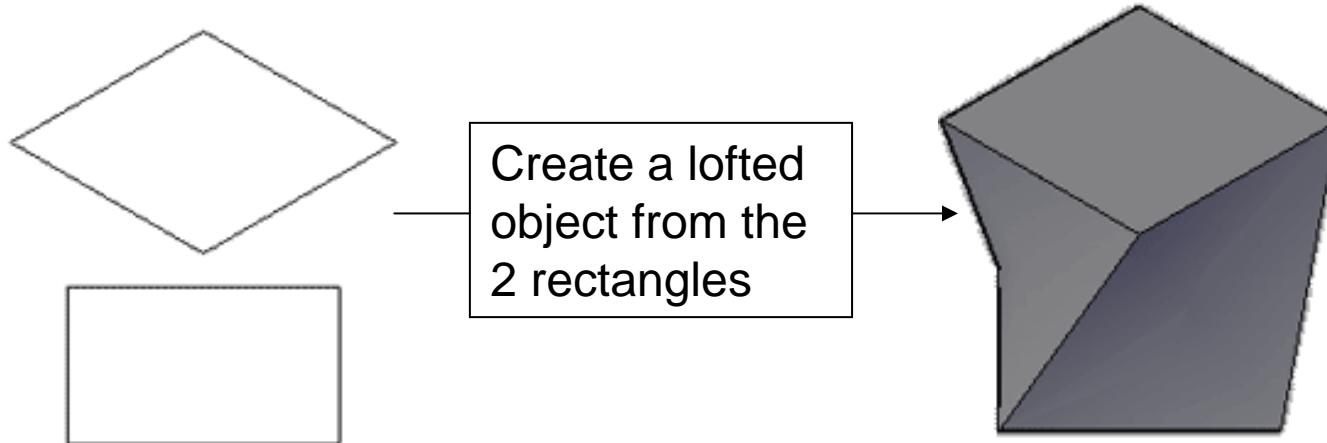


Exercise 6, advanced extruding



Lofting

The loft command was introduced in AutoCad 2007. The loft command allows you to extrude several shapes and make one continuous object.



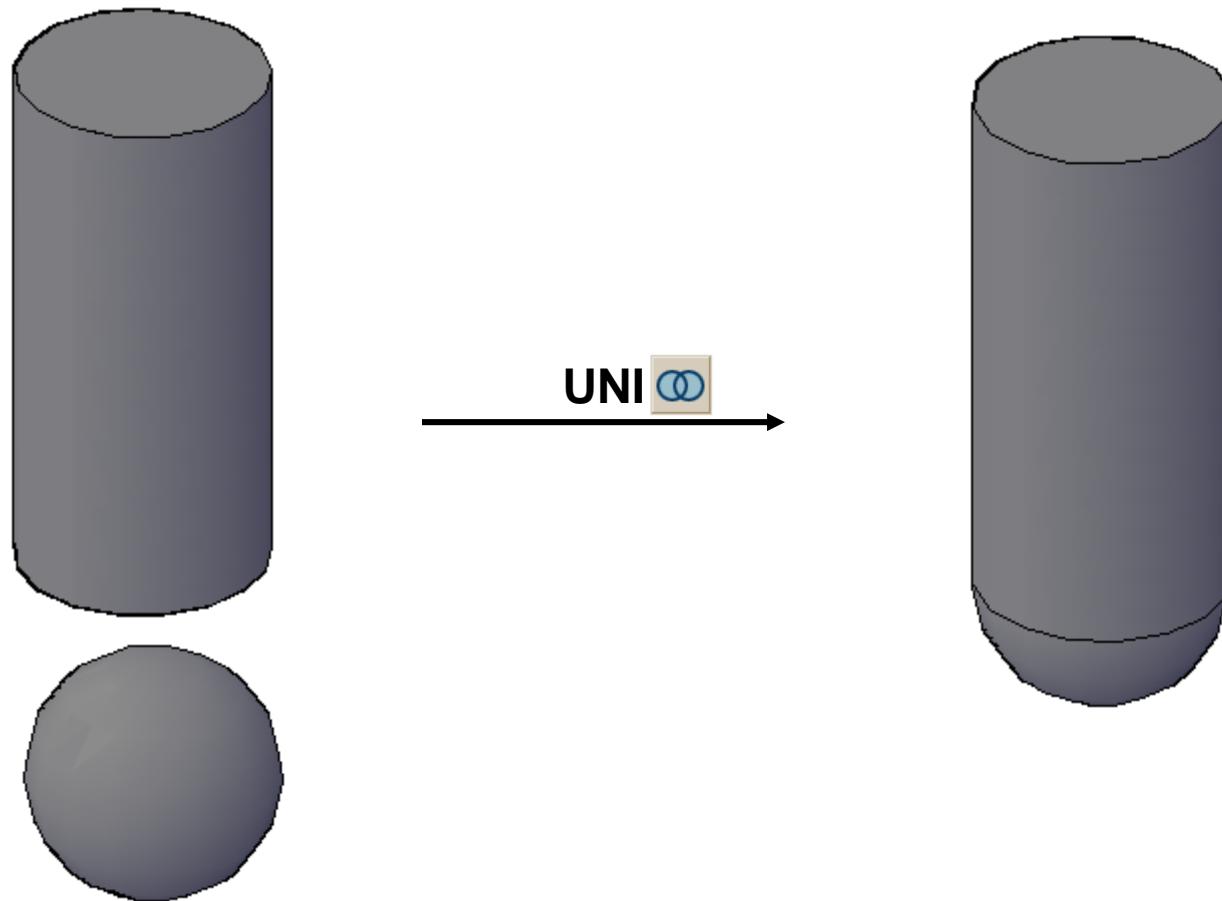
Primitive objects

SHAPE	COMMAND	ICON	DESCRIPTION
BOX	BOX		Creates a solid box after you provide 2 opposite corners.
SPHERE	SPHERE		Creates a solid sphere from a center point and radius.
CYLINDER	CYLINDER		Creates a straight cylinder from a center point, radius and height.
CONE	CONE		Creates a tapered cone from a center point, radius and height.
WEDGE	WEDGE		Creates a triangular wedge from 2 opposite points.
TORUS	TORUS		Creates a torus (donut shape) based on center point, radius and tube radius.
POLYSOLID	PSOLID		Draws a solid object with width and height as would draw a polyline.

Boolean operators

COMMAND	INPUT	ICON	DESCRIPTION
UNION (Boolean)	UNI		Joins two or more solids into creating one based on the total geometry of all.
SUBTRACT (Boolean)	SU		Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT (Boolean)	IN		Creates a single solid from one or more solids based on the intersected geometry.
EXTRUDE FACE	SOLID EDIT		Allows you to increase the size of a solid by extruding out one of its faces.
SLICE	SLICE	none	Slices a solid along a cutting plane.
3D ALIGN	3DALIGN		Aligns 2 3D Objects in 3D space.

Exercise 7, 3D boolean operations



Assignment 3

Design an aeroplane

