SOEM 024: Computer Aided Design

E. Rozos
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.
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.
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, draw 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

Taper angle

Extrusion height

Extrusion path

Extruded object
Lofting

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

Create a lofted object from the 2 rectangles
## Primitive objects

<table>
<thead>
<tr>
<th>SHAPE</th>
<th>COMMAND</th>
<th>ICON</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>BOX</td>
<td>BOX</td>
<td><img src="image" alt="Box Icon" /></td>
<td>Creates a solid box after you provide 2 opposite corners.</td>
</tr>
<tr>
<td>SPHERE</td>
<td>SPHERE</td>
<td><img src="image" alt="Sphere Icon" /></td>
<td>Creates a solid sphere from a center point and radius.</td>
</tr>
<tr>
<td>CYLINDER</td>
<td>CYLINDER</td>
<td><img src="image" alt="Cylinder Icon" /></td>
<td>Creates a straight cylinder from a center point, radius and height.</td>
</tr>
<tr>
<td>CONE</td>
<td>CONE</td>
<td><img src="image" alt="Cone Icon" /></td>
<td>Creates a tapered cone from a center point, radius and height.</td>
</tr>
<tr>
<td>WEDGE</td>
<td>WEDGE</td>
<td><img src="image" alt="Wedge Icon" /></td>
<td>Creates a triangular wedge from 2 opposite points.</td>
</tr>
<tr>
<td>TORUS</td>
<td>TORUS</td>
<td><img src="image" alt="Torus Icon" /></td>
<td>Creates a torus (donut shape) based on center point, radius and tube radius.</td>
</tr>
<tr>
<td>POLYSOLID</td>
<td>PSOLID</td>
<td><img src="image" alt="PolySolid Icon" /></td>
<td>Draws a solid object with width and height as would draw a polyline.</td>
</tr>
</tbody>
</table>
## Boolean operators

<table>
<thead>
<tr>
<th>COMMAND</th>
<th>INPUT</th>
<th>ICON</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNION (Boolean)</td>
<td>UNI</td>
<td><img src="image1.png" alt="Icon" /></td>
<td>Joins two or more solids into creating one based on the total geometry of all.</td>
</tr>
<tr>
<td>SUBTRACT (Boolean)</td>
<td>SU</td>
<td><img src="image2.png" alt="Icon" /></td>
<td>Subtracts one or more solids from another creating a solid based on the remaining geometry.</td>
</tr>
<tr>
<td>INTERSECT (Boolean)</td>
<td>IN</td>
<td><img src="image3.png" alt="Icon" /></td>
<td>Creates a single solid from one or more solids based on the intersected geometry.</td>
</tr>
<tr>
<td>EXTRUDE FACE</td>
<td>SOLID EDIT</td>
<td><img src="image4.png" alt="Icon" /></td>
<td>Allows you to increase the size of a solid by extruding out one of its faces.</td>
</tr>
<tr>
<td>SLICE</td>
<td>SLICE</td>
<td>none</td>
<td>Slices a solid along a cutting plane.</td>
</tr>
<tr>
<td>3D ALIGN</td>
<td>3DALIGN</td>
<td><img src="image5.png" alt="Icon" /></td>
<td>Aligns 2 3D Objects in 3D space.</td>
</tr>
</tbody>
</table>
Exercise 7, 3D boolean operations
Assignment 3

Design an aeroplane