

## Intermediate Engineering Graphics

### 6<sup>th</sup> Week Lecture Notes

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# Topic: Pictorial Drawings and Auxiliary View in AutoCAD

## 1<sup>st</sup> Subject: Isometric Drawing in AutoCAD

### Basic Strategy:

First set Drafting Settings to isometric with desired grids and snaps; use F5 to change Isometric Planes, turn on ORTHO and use DDE to draw isometric lines, then draw non-isometric lines using Endpoint Osnap.

### Basic skills:

(1) **Settings:** Type DS to open the Drafting Settings dialogue box, select Snap and Grid tab; check “Snap On (F9)” and enter a value in “Snap Y-spacing” line; check “Grid On (F7)” and enter a value in “Grid Y-spacing” line; select “Isometric Snap” in the “Snap type and style” section, press OK button.

(2) **Change isometric crosshairs orientation:** press F5 or Ctrl+E, or type ISOPLANE and press Enter (keep pressing Enter or type L, R, T), to change the isoplane. ISOPLANE helps to draw isometric ellipses correctly in the three isometric planes.

(3) **To draw isometric ellipses in the three isometric planes:** press F5 to select the corresponding isoplane, then select the Ellipse command by typing el or clicking on the Ellipse icon button in the Draw toolbar:

*Command: \_ellipse*

*Specify axis endpoint of ellipse or [Arc/Center/Isocircle]:* (Type I to select Isometric, press Enter)

*Specify center of isocircle:* (Pick a point on the drawing screen or type coordinate values, press Enter)

*Specify radius of isocircle or [Diameter]:* (Type a value for radius or to select Diameter, press Enter)

**Caution:** To maintain appropriate appearance of the ellipse in isometric planes when changing it from one isoplane to another, the rotation angle should be 120°.

(4) **Three steps in the construction of isocircles (ellipses) on any surface that is not parallel to an isometric plan**

1. Draw a parallelogram representing an imaginary square that bounds the circle; and locate the center of the circle. (type ds to open Drafting Settings/Object Snap, turn on Endpoints Object Snap)
2. Type mid to turn on Midpoint Object Snap while drawing an ellipse using Endpoints-of-Major-and-Minor-Axis method:

*Command: el ELLIPSE*

*Specify axis endpoint of ellipse or [Arc/Center]: mid of* [Type mid and click on the snap point on one side of the box]

*Specify other endpoint of axis: mid of* [Type mid and click on the snap point on the parallel side of the box]

*Specify distance to other axis or [Rotation]: mid of* [Type mid and click on a snap point on one neighboring side of the box]

3. Erase the parallelogram.

REMEMBER: an isocircles (ellipses) on any surface that is not parallel to an isometric plan needs only to appear correct. And it can not be drawn using regular F5 isometric plane method.

**(5) To draw Isometric arcs:** isometric arcs are used to create filleted corners. Two ways to draw isometric arcs: A. draw complete object first, trim away excess after locating the fillets and move the object lines; or B. draw the isometric arcs first and then draw the connecting lines. **The edge lines should be tangent to the isometric arcs. Use snaps to achieve this. Rounded edges, when viewed straight on, or not seen in the outer edge of the object, cannot be shown as complete-edge lines that extend to the end of the object, and need to be broken off from the edge).** To create isometric arcs: Type el for arc in the command line or go to Ellipse icon button on Draw toolbar (or go to Draw pull down menu to select Ellipse/Arc):

*Command: el*

*ELLIPSE*

*Specify axis endpoint of ellipse or [Arc/Center/Isocircle]:* (Type A to select Arc, press Enter)

*Specify axis endpoint of elliptical arc or [Center/Isocircle]:* (Type I to select Isometric, press Enter)

*Specify center of isocircle:* (Pick a point on the drawing screen or type coordinate values, press Enter)

*Specify radius of isocircle or [Diameter]:* (Type a value for radius or to select Diameter, press Enter)

*Specify start angle or [Parameter]:* (Type a value and press Enter)

*Specify end angle or [Parameter/Included angle]:* (Type a value and press Enter)

**Professional Tips:** to help locate the center of the iso arcs or ellipse, draw a point with an easily visible shape first.

**(6) Isometric Dimensioning (Using Oblique Dimensioning) in AutoCAD:** draw the regular dimensions first and than rotate it:

A. 30 degree for extension lines forming 30 degrees ccw with horizontal and

B. -30 degrees for extension line forming 30 cw with the horizontal.

To rotate the dimensions, type DIMEDIT or DED:

*Command: ded DIMEDIT*

Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: (Type o, and press Enter)

*Select objects: 1 found*

*Select objects:* (Press Enter)

*Enter obliquing angle (press ENTER for none):* (Type 30 or -30 depending on the dimensions, and press Enter)

## **2<sup>nd</sup> Subject: Four Steps in Oblique Drawing in AutoCAD**

1. Draw the “front view” first;
2. Draw the “receding lines” along the “receding axis”, using Endpoint Osnaps, @dist<angle;
3. Copy the “front view” to the “back”, using Endpoint Osnaps;
4. Trim off the unneeded portion of the copied “front view”.

## **3<sup>rd</sup> Subject: Auxiliary View in AutoCAD (Basic Strategy)**

Set the UCS to the slanted edge (ucs, Enter, 3p, Enter, pick 3 points using Endpoint Osnap), turn on ORTHO to allow Direct Distance Entry, use Inquiry>Distance tool (di,Enter, pick points) to borrow another set of true lengths from another view (with appropriate Osnaps for precision), or copy/move another view to right location and use Circle tool to transfer distance points.