

Intermediate Engineering Graphics

4th Week Lecture Notes

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Topic: Review of AutoCAD 2000 Interface, Settings, Options, and Tools

1st Subject: AutoCAD 2000 Interface ABC: Methods of Command Entry, Open/Exit, Save Files

1A. Four ways to Start AutoCAD:

- (1) **Open a Drawing:** Lets the user select a drawing from the most recently opened ones, or to “browse” for more files.
- (2) **Start from Scratch:** The user selects the measurement system, all settings set at default.
- (3) **Use a Template:** A template is a file that contains standard settings, which are applied to the new drawing. AutoCAD offers templates based on accepted industry standards. Users can create their own templates (.dwt).
- (4) **Use a Wizard:** provides step-by-step method of defining the drawing settings.
 - a. **Quick Setup** (includes two settings: Select units of measurement, Setup drawing area)
 - b. **Advanced Setup:** (includes five settings: Angle, Angle Direction, Angle Measurement, Select units of measurement, Setup drawing area)

1B. Four ways to create a new drawing: (1) click Standard toolbar icon button; (2) go to File/New... pull-down menu; (3) press Ctrl+N keys; (4) type new in Command line, press Enter or Space Bar.

1C. Four ways to open a file: (1) click Standard toolbar icon button; (2) go to File/Open pull-down menu; (3) press Ctrl+O keys; (4) type open in Command line, press Enter or Space Bar.

1D. Four ways to exit the AutoCAD program: (1) click the X closing box at the upper-right corner of the Screen (**The simplest way for closing all windows programs, menus, dialog box, toolbars, etc.**); (2) go to File/Exit pull-down menu; (3) press alt+F4 keys; (4) type exit or quit in Command line, then press Enter or Space Bar; (5) go to A-icon pull-down menu and select X Close, on the upper-left corner of Windows screen.

When there are unsaved files, AutoCAD will open a dialogue box with a warning message like Save changes to Drawing1.dwg? with three buttons: Yes for saving, No for not saving, Cancel for canceling the Exit command.

1E. Save and save as: (1) go to File/Save or Save as... pull down menu; (2) click Toolbar icon button; (3) press Ctrl+S keys; (4) type save in Command line to open the Save Drawing As dialogue Box.

File Formats include: .dwg (drawings), .dwt (templete), DXF (drawing interchange file, for exchanging AutoCAD files with other programs, import/export), .bak (backup file, generated by AutoCAD only, can be opened only with Microsoft Explorer)

Use of file name: File name can not use % or *.

To save a file in a drive, type the drive's code:file name, like

a:drawing 1 for saving the file as drawing 1 in the floppy disk drive (drive A:).

2A. The Three Most Important Methods of Entering Commands

(1) **Keyboard typing** to enter words or abbreviations in Command line and press Enter or Space Bar, or pressing special keys on the keyboard, is the most effective way. **To resize Command line box:** click on the edge and drag.

(2) **Pull-down menu:** click on the item names on the horizontal bar on top of the screen to “pull it down” and select the command. **Power Users shortcut to use pull down menus: hold down the Alt key, and press the underlined letter key to “pull it down”, and press the underlined letter key without holding down the Alt key to select the command.**

(3) **Toolbars:** 24 total, 4 are displayed by default: Standard, Object Properties, Draw, Modify. Toolbars contains icon buttons, and “tooltip” (move the cursor on the icon button to activate the tooltip) which help the user to know what command a tool icon button represents;

can be resized, floating or docked. **To add a new toolbar to the screen,** type toolbar in Command line or go to View Toolbars... pull-down menu to open the dialog box and check the corresponding box. **To close a toolbar:** uncheck the item in the dialogue box; or click on the X closing box at the upper-right corner of the toolbar.

To change the shape of a toolbar: place the cursor along the edge and drag. **To move a toolbar:** place the cursor on the toolbar and drag.

2B. Other Methods of Entering Commands

(4) **Shortcut Menus:** (right-click in the drawing area when there are words inside [...] on the command line), on the Status Bar buttons, or in the Command line.

(5) **Screen Side Menu:** (To activate: go to Options/Display/Windows Elements and check Display screen menu).

(6) **Tablet input:** (for Digitizers ONLY).

3. Functions of some special keys:

ESC Key Cancels the current command, menu, or dialogue box.

(This is your FIRST AID KEY!)

Enter Key Ends a command, or repeats the previous command if the command line is blank. Right-clicking the mouse can open the shortcut menu with Enter command or perform the Enter command.

(This is a VERY IMPORTANT key for Command line entry!)

Space Bar Same as the Enter Key except when entering text.

(This is also a VERY IMPORTANT key for Command line entry!)

Alt Key **used for pull down menus:** hold down the Alt key, and press the underlined letter key to “pull it down”, and press the underlined letter key without holding down the Alt key to select the command.

4. Using Function Keys:

F1 Help Explains the Command

F2 Flipscreen Toggles from Text Screen to Graphics screen

F3 Osnap Toggles Osnap On and Off

F4 Tablet Toggles the Tablet On and Off (used ONLY with a digitizer)

F5 Isoplane Changes the Isoplane from Top to Left to Right
(REMEMBER THIS!)

F6 Coordinate Display Toggles the Coordinate Display On and Off

F7 Grid Toggles the Grid On and Off

F8	Ortho	Toggles Ortho On and Off
F9	Snap	Toggles Increment Snap On or Off
F10	Polar	Toggles Polar Tracking On or Off
F11	Otrack	Toggles Object Snap Tracking On and Off

2nd Subject: Settings and Options in AutoCAD

1. **To change units:** type units or ddunits in Command line or go to Format/ units... pull-down menu to open the Drawing Units dialogue box.

2. **To set drawing limits:** Type “limits” in Command line, press Enter or Space Bar; or go to Format/Drawing Limits pull down menu. The drawing limits set the boundaries of the drawing to match the size of a sheet of paper.

3. **AutoCAD Options: Options window includes the following tab:**

Files, Display, Open and Save, Plotting, User Preferences, Drafting, Selection Profiles

To open the **Options dialogue box:** type op or options in Command line, press Enter or Space Bar; or go to Tools/Options pull down menu.

4. **AutoCAD Drafting Settings: Options window includes the following tab:**

Snap and grid, Polar Tracking and Object Snap.

To open the **Drafting Settings dialogue box:** type ds or dsettings in Command line, press Enter or Space Bar; or go to Tools/ Drafting Settings... pull down menu; or right click on the settings buttons in the Status line.

Grid settings places a dotted grid background on the drawing screen as visual reference points for sizing and locating points and lines. Can be turned on/off by clicking on status line GRID button or pressing F7.

Snap settings limits the movement of the cursor to predefined points on the screen. Can be turned on/off by double clicking on status line SNAP button or pressing F9.

3rd Subject: Cartesian Coordinate System and Coordinate Entry, Division, Measurement and Calculation

1. **The Cartesian Coordinates System includes:** the origin, the x-axis, the y-axis, and the z-axis, with positive values (above, to the right of, towards the user from the origin) and negative values (below, to the left, into the screen from the origin). Use the **“Right Hand Rule”** (Index finger for Y-axis, Thumb for X-axis, and Middle finger for Z-axis)

2. **Using Coordinate Entry in AutoCAD:**

Absolute Coordinates: come from the origin (0,0,0) (X-axis coordinate value, Y-axis coordinate value, Z-axis coordinate value). The format: X,Y

Relative Coordinates: come from the last point entered.

(1). **XY coordinates:** The format: @X,Y.

(2A). **DDE (Direct Distance Entry) or Ortho-On Direct Distance Entry for vertical and horizontal lines:** is a combination of keyboard entry and mouse movement. The format: turn ORTHO On, type distance value, move the mouse, press Enter or Space Bar.

- (2A). Polar Tracking DDE **Direct Distance Entry for slanted lines:** is a combination of keyboard entry and mouse movement. The format: go to Drafting Settings dialog box → Polar Tracking tab → Increment angle and select an angle, choose the Line tool, set a starting point, move the mouse, and as soon as the dotted line “tracking” shows up, type distance value, and press Enter or Space Bar.
- (3). **Polar Coordinate Input** (Direct Distance and Angle Entry): the format:
- relative: @distance value<angle value, press Enter or Space Bar (from the last point entered. This is the preferred method), or
 - absolute: distance<angle (from the Origin), press Enter or Space Bar.

3A. To divide a line or arc into equal parts: Type div or divide, go to Draw/Point/Divide pull down menu, or to Draw toolbar.

Command: div

DIVIDE

Select object to divide: [Pick the line or arc object]

Enter the number of segments or [Block]:

Requires an integer between 2 and 32767, or option keyword.

Enter the number of segments or [Block]: 5

This command divides an object by the number of segments designated. The object divided is NOT broken into segments.

3B. To bisect an angle

Command: xl XLINE [type xl and press Enter to invoke the Extension Line tool]

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: b [type b for Bisect option and press Enter]

Specify angle vertex point: [click the vertex point of the angle]

Specify angle start point: [click an endpoint of the line that represents one side of the angle]

Specify angle end point: [click another endpoint of the line that represents another side of the angle]

3C. To divide an angle into equal parts

Use the Circle tool to draw a circle with any convenient radius with the vertex of the angle as the center; use the Trim tool and the lines that represent the two sides of the angle as cutting edges to trim the circle into an arc; use the Division tool to divide the arc into equal parts; draw lines that connect the division points and the vertex.

4A. Measurement: measures an object by the length designated. The object divided is NOT broken into segments. Type me (DO NOT TYPE measurement, it is a system variable), go to Draw/Point/Measurement pull down menu, or to Draw toolbar.

Recommended: select Point Style first (Format/Point Style). The side where the selection point is placed is also where the measurement will start.

4B. To change Point Style when using Division and Measurement commands: select Point Style first (Format/Point Style).

5. Using the Command line calculator: Type cal, Enter, type the Expression using these keys to designate arithmetic function: * (multiply), + (plus), - (minus), / (division); press Enter or Space Bar to get the result.

4th Subject: Basic AutoCAD Commands: Selection, Copy/Paste, Block/Explode, Erase, Undo/Redo,

1. Selection: click individual object or “windowing” (right-to-left windowing will select partially enclosed objects with dashed selection window, left-to-right windowing only selects completely enclosed objects, with a solid lined selection window. **To remove an object from selection:** hold Shift key and click on the object to remove it from selection.

2A. To Copy objects: the Copy command creates duplicate sets of objects selected (single or multiple copies), needs selection of the object, selection of base point (usually on the object) and of new location (point of displacement).

To Copy: type co and press Enter or Space Bar; or press Modify/Copy toolbar icon button; or go to Modify/Copy pull down menu, then follow up on the Command line prompts.

To draw multiple copies, type m and press Enter or Space Bar at *Specify base point or displacement , or [Multiple]:* Command prompt.

2B. Using Copy to Clipboard and Paste commands: press Ctrl+C keys; select objects, press Enter or Space Bar; then press ctrl+V keys, type values for insertion point, press Enter or Space Bar; or left-click to paste the object; then repeat right-click and left-click to paste more.

2C. To Cut and Paste objects: (1) press Ctrl+X, select the objects, and right-click or press Enter or Space Bar; (2) select the objects, press Ctrl+X keys (faster method); (3) go to Edit/Cut pull down menu. Cut objects can be pasted using Paste command.

3. To Erase objects: (1) type e and press Enter or Space Bar, then click on objects and press Enter or Space Bar; or (2) select objects, then type e and press Enter or Space Bar, or click on the Modify/Erase Toolbar icon button; or (3) click Modify/Erase Toolbar icon button; or go to Modify/Erase pull-down menu, and follow Command line prompts to select objects and erase. (4) Use the Delete key: select objects and press Delete.

4A. To Undo: type u in Command line and press Enter or Space Bar; or press Ctrl+Z, or click the Standard/Undo toolbar icon button; or go to Edit/Undo pull down menu.

4B. To Redo: press Ctrl+Y, or click the Standard/Redo toolbar icon button; or go to Edit/Redo pull down menu.

5th Subject: Constructions of Lines and Polylines

1. To draw a line: type “l” or “line” in Command line, or click Toolbar/Line icon button; or go to Draw/Line pull-down menu; follow the Command line prompt; press “Enter” to end the Command; press “Enter” again to restart Line command. Lines can be drawn by picking points on the drawing screen or by typing coordinate value input in the Command line:

Five ways of using Cartesian Coordinate System Input to draw lines in AutoCAD:

(1) Absolute Coordinates: come from the origin (0,0) (X-axis coordinate value, Y-axis coordinate value). The format: X,Y

(2) Relative Coordinates: come from the last point entered.

A. XY coordinates: The format: @X,Y.

- B. DDE or Ortho-On **Direct Distance Entry**: is a combination of keyboard entry and mouse movement. The format: type distance value, move the mouse, press Enter or Space Bar.
 - C. **Polar Coordinate Input** (Direct Distance and Angle Entry): the format:
 - c. @distance value<angle value, press Enter or Space Bar (from the last point entered. This is the preferred method), or
 - b. distance<angle (from the Origin), press Enter or Space Bar.
2. **To draw construction lines:** type “xl” or “xline”, click Draw/Construction Line Toolbar icon button, go to Draw/Construction Line pull down menu. Construction line extent to infinity; and need two through points to draw. Settings include: Horizontal, Vertical, Angle, Offset (need object, distance and side), Bisect (used to bisect an angle, need vertex, start and end points). The @distance<angle formula can be used.
3. **To draw polylines:** a polyline is a line made from a series of connected individual line segments that act as a single entity; is used to generate curves and splines, and to produce solid objects in three-dimensional applications. To access the polyline command, type pl or pline in Command line, or click Draw/Polyline toolbar icon button; or go to Draw/Polyline pull-down menu. In the Command line prompt, W option is used to change the width of the line, CL option will join the last point drawn to the first point of the polyline. CE activates the Center option, allowing the user to define center point, angle, chord length or endpoint of the arc. Polyline command can be used to draw mixed line and arc segments by typing L and A alternately in the Polyline command prompt.
- 4. **To change line type, color and layer location:** go to Object Properties bar.
 - 5. **To change line width:** type w in Command Line and enter values at prompts.
 - 6. **To change line type scale:** type ltscale, Enter and type a value.

6th Subject: Constructions of Rectangles and Circles

1. **To draw a rectangle:** the Rectangle command generates rectangular blocks as whole entity, but the entity can be Exploded into segments. Chamfer, Fillet, Width of line values can be changed in Command line. (Elevation and Thickness are for 3D and will NOT be covered in these courses). **Three ways to access Rectangle tool:** (1) Type rec or rectangle in Command line; (2) Click Draw/Rectangle toolbar icon button; (3) go to Draw/Rectangle pull-down menu.

Three ways to draw rectangles:

- (1) **corner-to-corner** mouse pick points:

Command: rec [Type “rec”]

Command: _rectang

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: [Pick a point on the drawing screen]

Specify other corner point or [Dimensions]: [Pick a point on the drawing screen]

- (2) Typing **length and width dimensions** values in Command line:

Command:

RECTANG

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: [Pick a point and press Enter key]

Specify other corner point or [Dimensions]: d [Type d for Dimensions]

Specify length for rectangles <0.0000>: 6 [Type a value]

Specify width for rectangles <0.0000>: 5 [Type a value]

(3) **using** absolute and relative coordinates.

Command: `_rectang`

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 2,1 [Type absolute coordinate values and press Enter to start with]

Specify other corner point or [Dimensions]: @3,5 [Type relative coordinate values and press Enter to complete]

2. To draw circles:

Three ways to access Circle tool: (1) Type `c` or `circle` in Command line;

(2) Click Draw/Circle toolbar icon button; (3) go to Draw/Circle pull-down menu.

Five ways to draw circles: (1) Center-Radius; (2) Center-Diameter; (3) 3-Points; (4) 2-Points; (5) Tangent-Tangent-Radius. Need center plus radius or diameter, or 2 to 3 points on circumference; or tangencies to two existing lines or objects plus radius. (TTR method is available through Draw Toolbar menu only).

Command: `c CIRCLE` *Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:*

Specify radius of circle or [Diameter]:

Circle Centerlines: It is standard drawing convention to include it. It is used to locate the circle and the location of the center for drill during manufacturing. To set up center lines:

Use Dimcen system variable: type “`dimcen`” in Command line, Enter, type `-0.1` for inch units, or `-2` for metric, Enter, select circles.

Use Center Mark tool in Dimension toolbar: press the tool icon button, then pick the circle object.

To change centerline to center mark: select Dimension styles tool on Dimension toolbar (at right corner) to access Dimension Styles Manager, press `Override...` button to access `Override Current Style: Standard` dialog box, and change it in the `Center Mark for Circle` box.

7th Subject: Constructions of Polygon, Ellipse, Donut, and Point

1. **Polygon:** a closed figure bound by straight lines. The polygon tool can only draw equilateral polygons. Need number of sides (3-1024), center, inscribed (inside an imaginary circle)/circumscribed (outside an imaginary circle), radius; or number of sides, and edge/edge length)

To access the polygon command, type `pol`, go to Draw toolbar, or to Draw/polygon menu.

2. **Ellipse:** there are three options related to the Ellipse tool

1. Length of major and minor axis;

2. An angle of rotation about the major axis (an ellipse with 0° rotation is a circle, with 90° rotation is a line);

3. An included angle.

Type el or ellipse, go to Draw toolbar, or go to Draw/Ellipse menu (Center, Axis, End, Arc) to access the tool.

3. Donuts: is a circle with width. Need to define the inside and outside diameter. The appearance is controlled by the “Fill” mode (On = donut width is solid; Off = donut fill with lines like spokes of a wheel).

To access Donut tool: type do; go to Draw/Donut menu.

To control Fill appearance: type fill, Enter, type On or Off.

4. Point: a point is an object with location but no dimension; the default style is a dot; to change the shape of points:

1. Type “pdmode” type corresponding number, Enter. (See “Purple Book” Figure 2-32 for numbers). or

2. Type “ddptype”, or go to Format/Point Style pull-down menu to open the Point Style dialog box, and pick a style. (This is the easiest way to remember)

To access Point Tool: type po; go to Draw/Point menu, or to Draw Toolbar.

8th Subject: Constructions of Arcs, Spirals, Elliptical Arcs, and Splines

1A. Ten ways to draw ARCS in AutoCAD: an arc is a segment of a circle less than 360 degrees. Most arcs are drawn counterclockwise. An arc can be measured by angle or chord length (the distance between the two end points of the arc). Ten ways to draw arcs:

1. 3 Points;
2. Start-Center-End (draws only ccw);
3. Start-Center-Angle (+Angle = ccw, -Angle = cw);
4. Start-Center-Length (- Length = Large Segment/ccw, + Length = Small Segment/ccw);
5. Start-End-Angle (+Angle = ccw, -Angle = cw);
6. Start-End-Direction;
7. Start-End-Radius (+ Radius = Small Segment/ccw, -Radius = Large Segment/ccw);
8. Center-Start-End (ccw only);
9. Center-Start-Angle (+Angle = ccw, -Angle = cw),
10. Center-Start-Length (+ Length = Small Segment/ccw, - Length = Large segment/ccw).

An arc can be drawn within the Polyline command. To switch back and forth from Arc to Line, type A or L within the Polyline Command.

1B. To create isometric arcs (See handouts on isometrics)

2. To draw a spiral using arc command: draw the first Arc using SCE (180 degree angle draws a “regular” looking spiral), Enter, Enter, place end of second arc at a location with set distance, Enter, Enter, and repeat the process.

3. To draw Spline: a spline is a curved line created through a series of predefined points. A **closed spline** forms an enclosed area; an **open spline** does not form an enclosed area. Go to Draw toolbar, or to Draw/Spline menu, or type spl or spline in Command line.

9th Subject: To Set Up a Scaled Drawing

In AutoCAD, you can draw everything in full scale in the Model Space, and create a scaled up or scaled down drawing in the Layout Space using the Viewport.

Read the following for more details:

About File Layout and Printing Management

Create your drawing in Model Space:

Set drawing Limits (limits, paper size, and Grid coverage) and Units (ddunits)

Create the following Layers:

- Object Thick Continuous line White
- Hidden Thin Hidden Line White
- Dimension Thin Continuous Line White
- Centerline Thin Centerline White
- Notes Thin Continuous Line White
- Construction Thin Continuous Line Cyan Not Printing

Complete the drawing (scale: 1=1)

Note: different colors can be used in the drawing; when you need to print them black, choose monochrome.ctb option in the Plot style table (pen assignments): Name pull-down menu in the Plot dialog box.

Go to Layout (Paper Space):

Create the following objects on the Layers:

- Viewport Frame
Viewport Frame Thin Continuous Line Cyan Not Printing
- Title Block
Title Block Thin Continuous Line White

Double-click inside the Viewport Frame to enter the Model Space and make needed changes; Go to View>Toolbar... → Viewport toolbar: Scale text field, change the drawing scale; Change drawing scale notation on the Title Block. Zoom tool with Scale option can be used to accomplish this task too:

Command: z ZOOM [type z, press Enter]

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: s [type s, press Enter]

Enter a scale factor (nX or nXP): .5x [type a scale factor value followed by letter x, and press Enter]

Command:

Print the file.

10th Subject: 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 ds to open Drafting Settings/Object Snap, turn on Midpoint Object Snap, turn off other Snaps; draw an ellipse using Center-endpoint-distance method:

Command: el ELLIPSE

Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: c [Type “c” to select Center option]

Specify center of ellipse: [Pick the center]

Specify endpoint of axis: [Move the cursor to the midpoint of one side of the parallelogram, until it reaches the Midpoint Snap, and click]

Specify distance to other axis or [Rotation]: [Move the cursor to the midpoint of the neighboring side of the parallelogram, until it reaches the Midpoint Snap, and click]

3. Erase the parallelogram and other construction lines.

