Introduction to AutoCAD® 2000

Mark Dix and Paul Riley
CAD Support Associates

## PowerPoint Slide Materials:

Jack Leifer, Ph.D.

Department of Mechanical Engineering
Extended Campus Program at Paducah
University of Kentucky
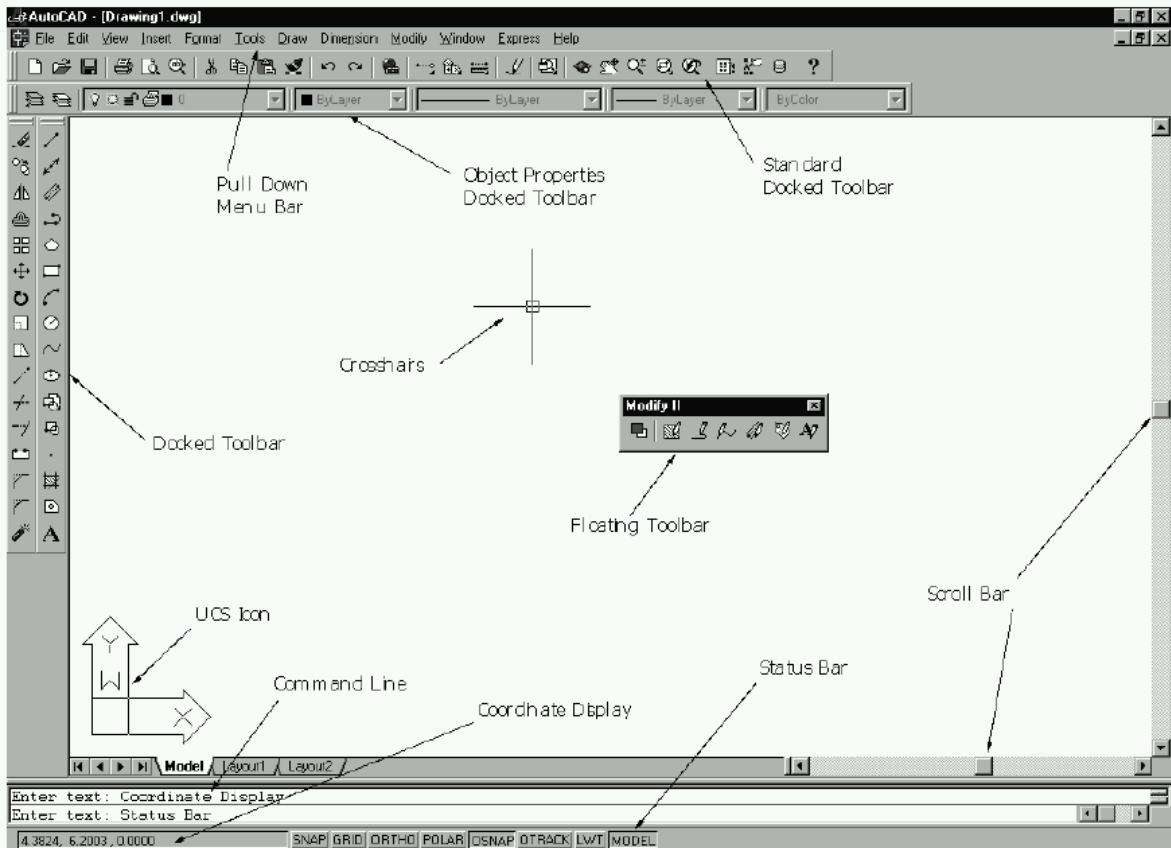
Introduction (Lines)

Beginning a New Drawing

- Start AutoCAD 2000 by using either the Start menu or the shortcut icon on the desktop

- The *startup screen* presents the user with a number of options:

  (a) Open an existing drawing

  (b) Start a new drawing from scratch

  (c) Use a drawing template

  (d) Use a Drawing Wizard (guides you through setup)

Introduction (Lines)
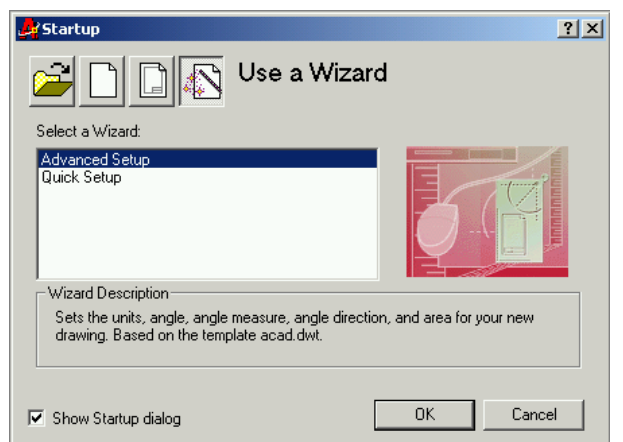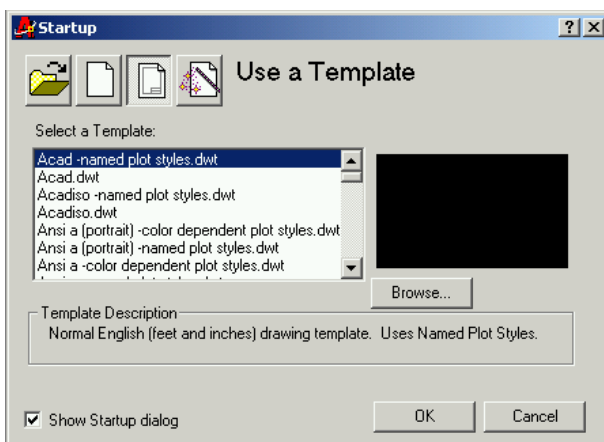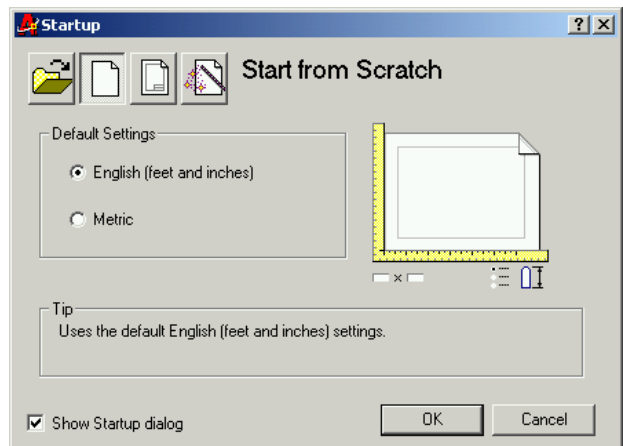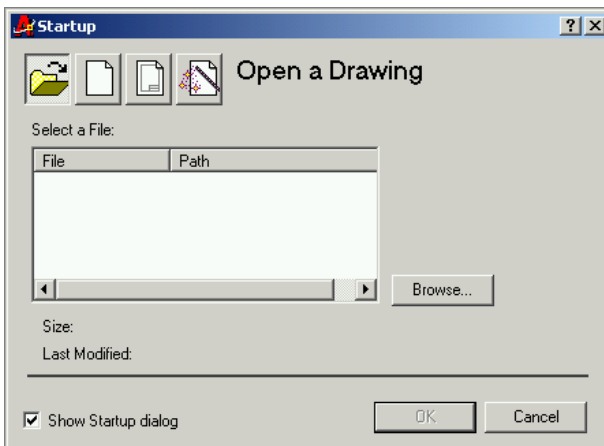
The Drawing Window

- The menus and options displayed on the drawing window screen can be modified, depending upon
  - the type of drawing being modified
  - personal preference
- Options are switched on and off manually via the mouse and function keys, and automatically by the program (context sensitive)
- Open a text window using the *F2* function key. Moving between open text and drawing windows is done by selecting the appropriate button on the task bar

Introduction (Lines)

AutoCAD Basics

- The two button *mouse* is the most common AutoCAD pointing device
  - Left button is used for point selection, object selection, and menu or tool selection. Called *pickbutton*
  - Right button functions as an alternative to the keyboard *enter* key, or calls pop up menus or help windows (depends on the situation)
  - Moving the mouse changes the position of the screen cursor
- Screen Cursors
  - Cross hairs: two perpendicular lines whose intersection marks the current point
  - Pick box: small box at the intersection of the cross hairs; used to select objects for editing
  - The cursor becomes an arrow (used to select menus, tools, and task bars) when you leave the drawing area
- Scroll Bars
  - Allows drawings too large to fit on the screen to be displayed
  - Position of the drawing changes relative to the screen but *not* relative to the UCS coordinate system

Introduction (Lines)

AutoCAD Basics (continued)

- The coordinate display
    - Located at the bottom left side of the status line
    - Keeps track of where the cross hairs are located
    - Measures the location of the cursor with respect to the drawing origin (0,0,0) (demarcated by the UCS icon)
    - Pressing F6 puts the coordinate display into *static mode*, where the coordinate display only changes when a new point is chosen with the mouse

Cursor

Gridpoints

Origin/UCS icon

Coordinate Display

GRID switch
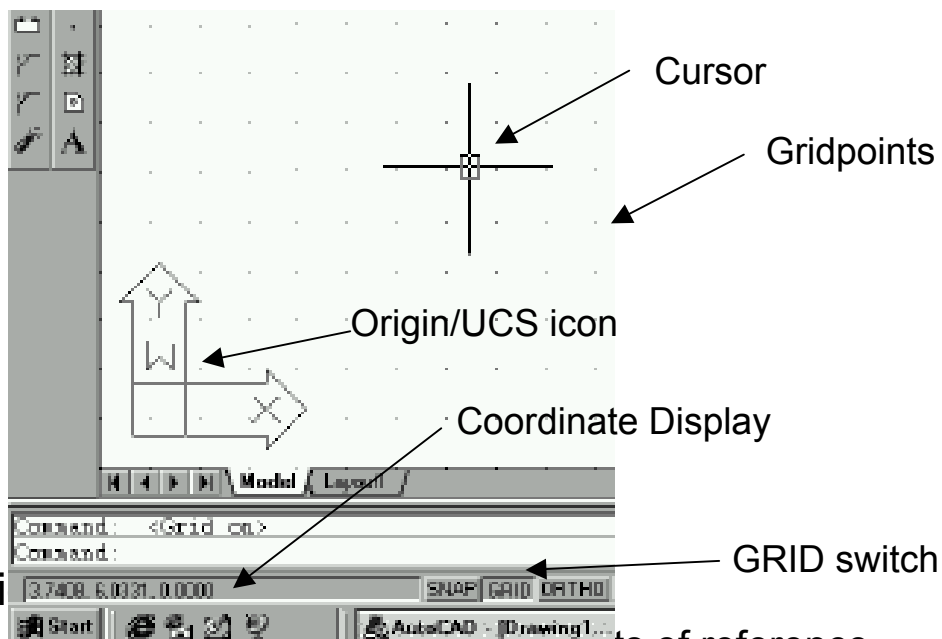
- Gri
    - Matrix of dots which provide points of reference
    - Gridpoints are not printed as part of a drawing
    - Pressing F7 or double clicking on the word GRID on the status bar toggles the gridpoints on and off

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Introduction (Lines)

AutoCAD Basics (continued)

- Snap
  - Prevents point selection at any location which is not on a gridpoint (or a specific point on an object)
  - Allows for a degree of accuracy which is not otherwise possible
  - Gridpoints and snap can be set at any distance; default setting is 0.5000 inches
  - Snap is toggled by using the F9 function key, or by double clicking on the word SNAP on the status bar
- UCS icon
  - Indicates the direction of the *X* and *Y* coordinates
  - Can be turned off through the use of the *ucsicon* command
    - Type *ucsicon*
    - Type *off* at the prompt
    - Can be turned on using the same command
- Undo and Redo
  - Undo commands by typing *U* at the command line, or by using the Undo toolbar button
  - Redo any undone commands by typing *R* at the command line, or by using the Redo toolbar button
- Cancel any in-progress command by hitting the *Esc* (escape) key
- Repeat any command by hitting *enter* key or *space bar* at the command prompt

Introduction (Lines)

Command Entry Methods

- The same command can often be executed by either typing a key word, or by selecting items from a toolbar, a pull down menu, or a tablet menu
- No one method is preferable to any other; AutoCAD users develop their own preferences as they work
- Keyboard/Command Line
    - Most primitive and fundamental way of entering commands
    - Commands are typed on the *command line* at the bottom of the screen and ended by pressing *enter*
    - Has changed little since early versions of the program
    - Some common commands are abbreviated by one or two letter aliases

| Alias	Command
A	ARC
C	CIRCLE
E	ERASE
F	FILLET
L	LINE
M	MOVE
O	OFFSET
P	PAN
R	REDRAW
S	STRETCH
Z	ZOOM

Introduction (Lines)

Command Entry Methods (continued)

- Pull Down Menus
    - Work as they do in any Windows 95/98 application
    - Items followed by a triangle have "cascading" submenus (a)
    - Picking an item that is followed by an ellipsis (. . .) will call up a dialog box (b)
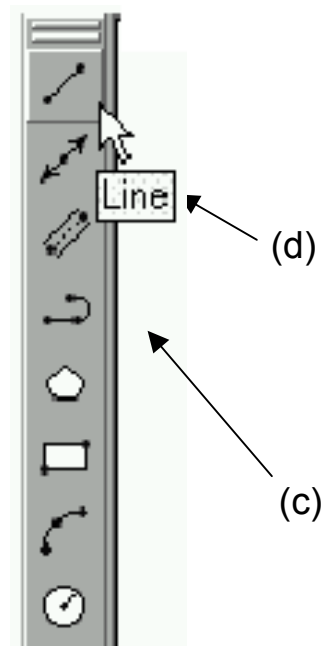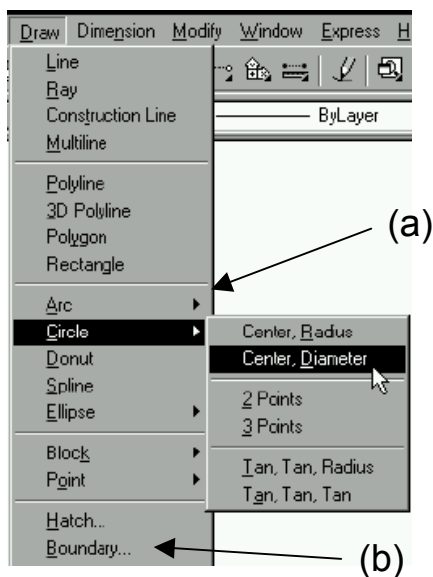
- Toolbars
    - Comprised of buttons (icons) that give one-click access to commands (c)
    - Seventeen toolbars are available; to avoid clutter, do not open more than are necessary
    - Toolbar icon labels ("tooltips") are shown by simply pointing to an icon without clicking (d)
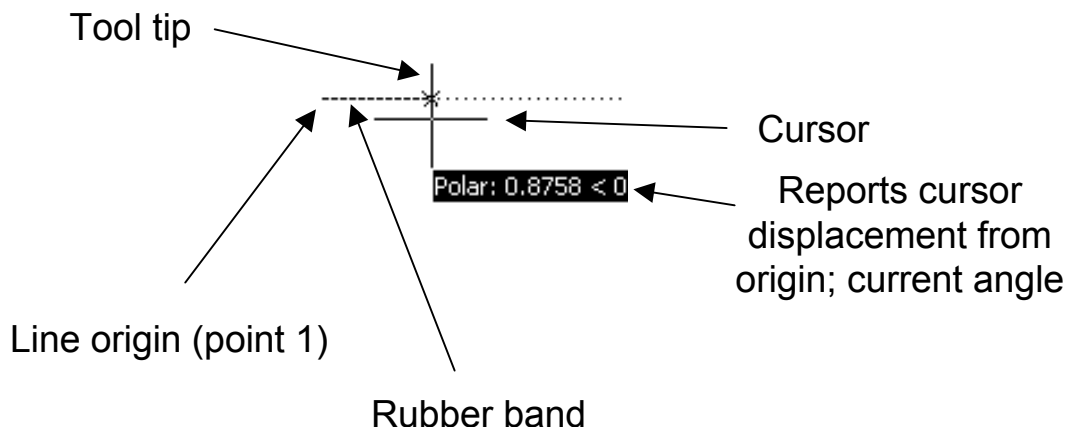
Introduction (Lines)

Drawing Lines

- Choose a method for entering the *Line* command
  - Type "L" at the command prompt (and hit *enter*)
  - Select the Line icon from the Draw toolbar
  - Select Line from the Draw pull down menu
- AutoCAD will say, *"Specify first point:"* Choose the initial point of the line by either…
  - Typing in the coordinates of a point and hitting enter
  - Moving the on-screen cursor with your mouse, and selecting the appropriate point by left clicking
- AutoCAD will say, *"Specify next point or [Undo]:"* Choose the second point in the same way you chose your first
- AutoCAD will continue to say, *"Specify next point or [Undo]:"* until *Enter* or the right mouse button is pushed. This will terminate the *line* command
- Some tips for entering lines:
  - A *rubber band* extends from the line start point to the position of the cursor on the screen. This indicates that AutoCAD expects you to provide an endpoint
  - Press *F6* to turn on the coordinate display, or to switch between absolute and polar coordinates
  - The *space bar* can be used in place of *enter* to terminate a command
  - Use *Ortho* mode (hit *F8* button, or double click on the word ORTHO) to force the pointing device to pick up points only along the horizontal and vertical quadrant lines from a given starting point

# Introduction (Lines)

Polar Tracking

- New to AutoCAD with this version

- Can be used in place of ORTHO mode

- To turn on, either press the *F10* button on the keyboard, or the *POLAR* button located on the Status Bar

  ```
  Polar Tracking
  SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL
  ```

- Polar Tracking and ORTHO mode are mutually exclusive

- In polar tracking, when a rubber band crosses a horizontal or vertical axis (i.e., when the rubber band is at 0, 90, 180, or 270 degrees), a dotted line appears that extends from the line origin to the end of the drawing area

- Polar tracking defaults to the orthogonal angles, but can be set to track at any angle

Tool tip

Cursor

Polar: 0.8758 < 0

Reports cursor displacement from origin; current angle

Line origin (point 1)

Rubber band


Introduction (Lines)

Saving and Opening Drawings

- Saving drawings
  - Invoke the *save* command either from the pull down menu, the toolbar, or by typing on the command line
  - *SAVE* saves an already named drawing under its current name. The current version of the drawing writes over the previous version
  - *SAVEAS* allows you to
    - Choose a name for your drawing file
    - Save a named drawing under a new name
    - Save as a file *type* other than AutoCAD 2000 (a)
  - All saved drawings will have a *.dwg* extension added to their names to identify them as AutoCAD files (b)
- Use the *OPEN* command to access previously saved drawings
  - Only *.dwg* files are displayed unless another file *type* is chosen in the dialog box
  - Files saved in another location can be accessed by changing directories (c)

(c)
(b)
(a)

Circles and Drawing Aids
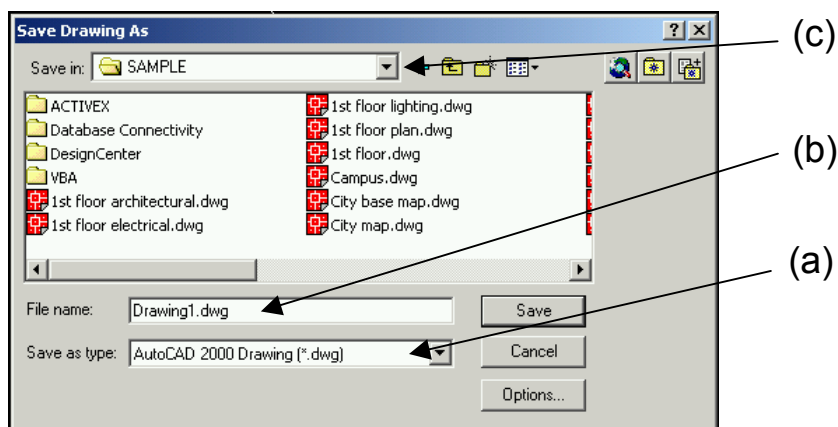
Grid and Snap

- Gridpoints can be toggled on and off by either
  - Pressing *F7*
  - Clicking on the word *GRID* in the status bar
- Snap can be toggled on and off by either
  - Pressing *F9*
  - Clicking on the word *SNAP* in the status bar
- Grid and snap parameters are easily changed via the *Drafting Settings Dialog Box*
  - Select *Tools*, then *Drafting Settings* from the pull down menu, or type *ds* at the command line to obtain the dialog box
 - Note that grid and snap spacing are not required to be the same

Circles and Drawing Aids
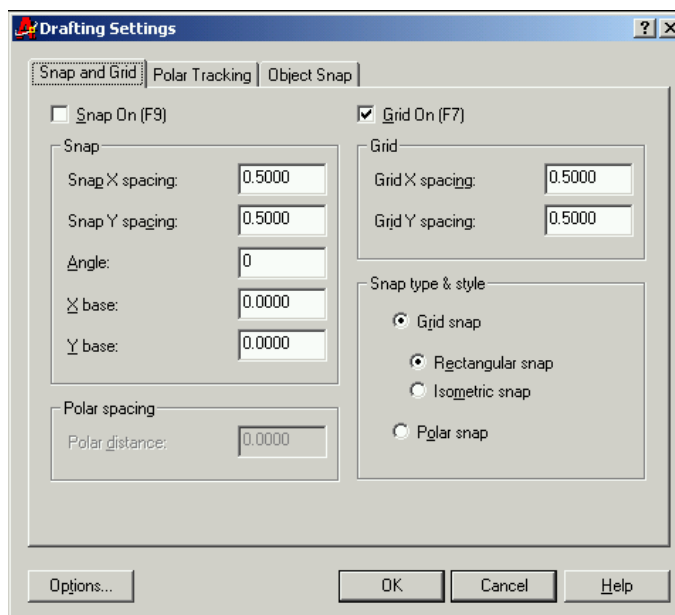
Units

- Units determine how coordinates and dimensions will be reported

Unit Type	Example
Scientific	1.55E + 01
Decimal	15.50
Engineering	1'-3.50"
Architectural	1'-3 ½"
Fractional	15 ½

- Access the *UNITS* dialog box by either typing *UNITS* on the command line or choosing *Units…* from the *Format* drop down menu
  - The dialog box allows unit formats (above) and number of significant digits (precision) to be specified
  - Units options are changed via the *pop-down* lists in the dialog box
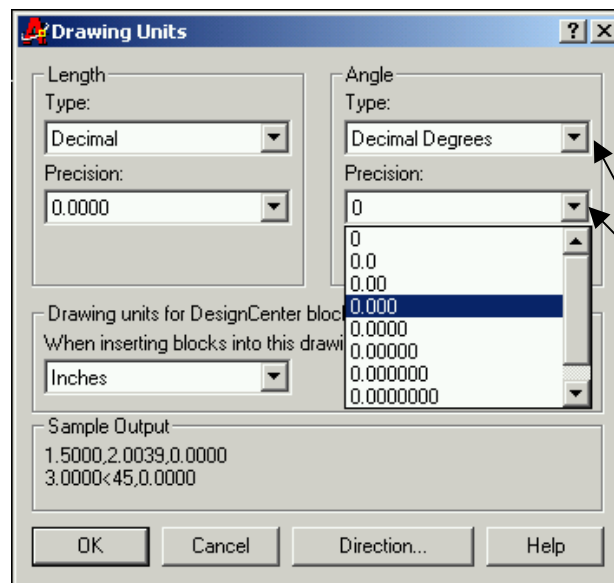

Access pop-down menus by clicking

Circles and Drawing Aids

Drawing Circles

- Access circles via the pull down menu, the toolbar button, or by typing the keyword *C* on the command line
- Circle dimensions are specified by giving AutoCAD one of the following:
    - Three points on the circle's circumference
    - The endpoints of the diameter
    - Two tangent lines and the length of the radius
    - The center point and the length of the radius
    - The center point and the length of the diameter
- Dimensions and point locations can be specified by typing in values, or by using the cursor and clicking with the mouse

AutoCAD Help

- Access help from the pull down menu, the toolbar button, typing F1, or by typing *help* on the command line
- If you access *Help* while in the middle of a command, you will get context sensitive assistance
- The *Help* function in AutoCAD works similarly to *Help* in most Windows 95/98 programs

Circles and Drawing Aids

Erase and OOPS

- Erase
  - Allows you to remove unwanted objects from your drawing
  - Type *e* on the command line, select *Erase* from the *Modify* pull down menu, or select the *Erase* button from the *Modify* toolbar
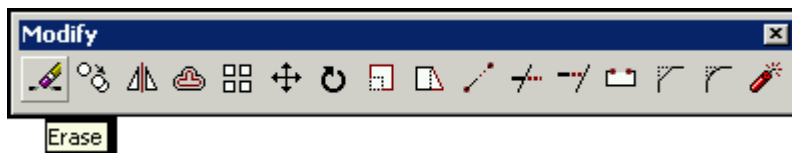

- OOPS
  - *Oops* is to *erase* as *redo* is to *undo*
  - *Oops* redraws the last object (or set of objects) that was erased
  - Only the most recent set of erased objects can be restored via *oops*

Command Sequence Order

- Verb/Noun: Choose the *action* first, then pick the objects. This is the "default" setting for AutoCAD
- Noun/Verb: Choose the *objects* first, then specify which *action* is to be taken

Circles and Drawing Aids

Selection Methods

- Objects (entities) are generally picked by using the cursor and pointing device
- There are many selection methods
  - Picking
    - Use the cursor to point to the desired object, and left click with the mouse
    - Selected object (a) will change from solid to dashed

(a)

(b)

(c)

(d)
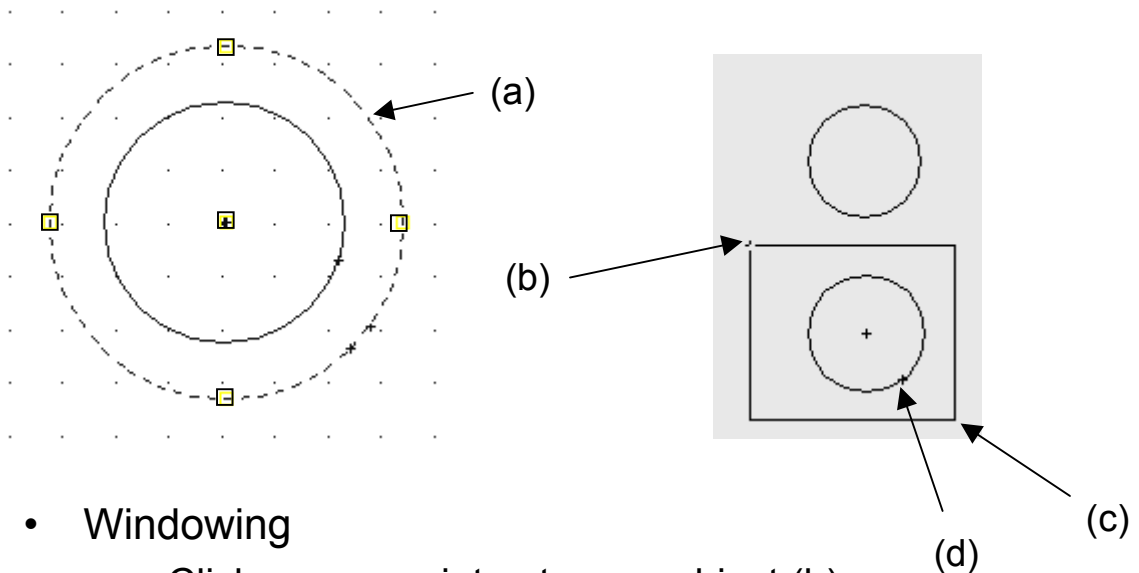
  - Windowing
    - Click on any point not on an object (b)
    - AutoCAD will ask for a second point
    - Click on any other point, to form a box (c)
    - All objects inside the box are selected (d)

Circles and Drawing Aids

Other Selection Methods

- <u>Crossing</u>: Entities within or crossed by the box are selected
- <u>Previous</u>: Entities that were previously picked are (again) selected
- <u>Last</u>: The entity that was drawn most recently (last) is picked
- <u>Remove</u>: Deselects objects one at a time from a group of selected entities
- <u>Add</u>: Adds objects to the group of entities already selected
- <u>All</u>: Selects all entities currently visible on the drawing
- <u>Fence</u>: Selects all objects crossed by a selection line (fence)
- <u>Wpolygon</u>: Selects all objects within a selection polygon
- <u>Cpolygon</u>: Selects all objects crossed or within a selection polygon

# Layers, Colors, and Linetypes

Layers

- Allow specialized groups of entities to be treated individually
- Individual layers can be "turned off" to render objects in that layer invisible. This reduces clutter, and allows specific details to be printed individually
- Objects in layers that have been turned off are still part of the drawing database
- Different line types and/or colors are often put on their own layers (e.g. dimensions, hidden lines)
- Use of layers is up to the AutoCAD designer
  - Their use is only for convenience
  - The final drawing will look the same regardless of whether layers were used

Layers, Colors, and Linetypes

Layer Properties Manager
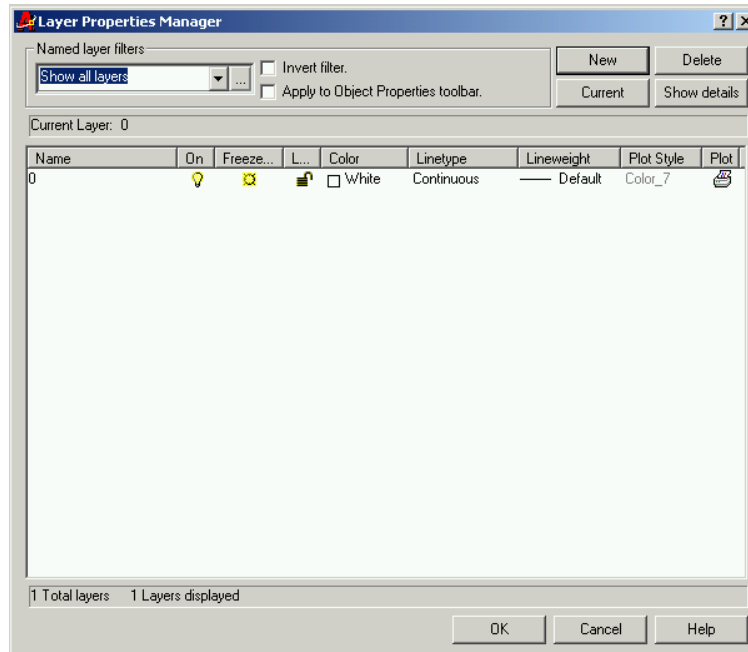
- Activate layer properties manager from the *Object Properties* toolbar or from the *Format* pull-down menu

- Layers are initially defined by number, and can be given unique names

- Define a new layer by clicking on `New`

- *Linetype, lineweight,* and *color* can be set for each layer that has been defined

- Quickly switch among layers by clicking on the *current layer list box* found on the *Object Properties* toolbar

Layers, Colors and Linetypes
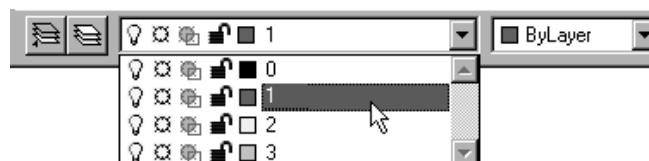
Fillets and Chamfers
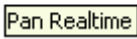
- Fillet
    - A rounded corner between two lines
    - Generally replaces a 90° angle
    - Start command by either
        - Typing *f* on the command line
        - Selecting the *Fillet* tool from the *Modify* toolbar
        - Choosing *Modify, Fillet* from the menu bar
    - Fillet radius (degree of rounding) can be changed by typing *R* at the fillet command prompt
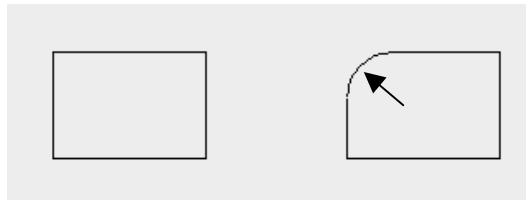
- Chamfer
    - A beveled edge used to replace a square (90°) angle
    - May be either symmetrical (a) or asymmetrical (b)
    - Start command by either
        - Typing *cha* on the command line
        - Choosing *Modify, Chamfer* from the menu bar
   - Selecting the *Chamfer* tool from the *Modify* toolbar

Layers, Colors and Linetypes
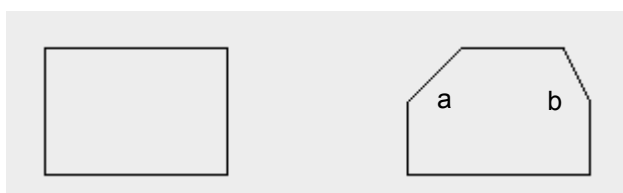
Zooming and Panning

- <u>Zoom</u>: allows you to control the magnification of the objects on the screen
    - More magnification means less workspace visible
    - Less magnification allows a wider view
    - Invoke *zoom* command by either typing *z* on the command line or selecting one of the Zoom tools from the *Standard* toolbar
    - At the *zoom* prompt (obtained using the *z* command)*:*
        - Type *a* (zoom all) to bring entire drawing into view
        - Type *p* (zoom previous) to return to the previous scale
        - Type *w* (zoom window) to define a view by specifying two points at the window corners. Specified view will fill available screen space
- Panning
    - Allows you to shift the area you are viewing in any direction
    - Similar to using window scrollbars
    - Does not change the view magnification
    - Invoke *pan* command by either typing *p* on the command line or selecting *Pan* tool from the *Standard* toolbar
    - When *pan* cursor (hand) is visible, drag drawing to a new position by moving the mouse while holding down its left button

Template Drawings

- Model Space
  - Space in which objects are created and edited
  - Objects in model space are always drawn full scale:
    1 screen unit = 1 unit of length in the real world
- Paper Space
  - Screen units represent units of length on a piece of drawing paper
  - Considered mostly when plotting multiple views of a 3D drawings
- Limits
  - Usually correspond to the size of the paper used
  - When limits are *on*, entities cannot be added to a drawing outside of the predetermined area
  - May be specified…
    - by typing *limits* on the command line
    - by choosing *Format, Drawing Limits* from the menu bar
    - when creating a new drawing by choosing the Advanced *Wizard* option in the *Create New Drawing* dialog box
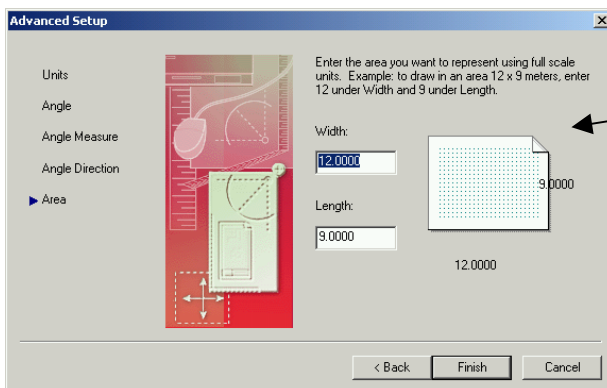

Limits are typed into the *Area* portion of the Advanced Setup Wizard dialog box. Units and angle parameters can be specified on the preceding Wizard screens

  - Limits are visually indicated in a drawing by the extent of the grid

Template Drawings

Creating a Template File

- A *template* is a special file that contains the AutoCAD settings you wish to use for your drawing

- Settings for the following may be contained in a template file: Grid, Snap, Units, Limits, Layers, Colors and Linetypes

- Generally, template files will contain only settings. No objects should be drawn on your screen when creating a blank template file

- Saving a template
    - Choose *File, Save As…* from the menu bar
    - Choose *Drawing Template File (*.dwt)* from the *Save as type* dialog
    - Your template file will automatically be saved in AutoCAD's template folder

Template Drawings
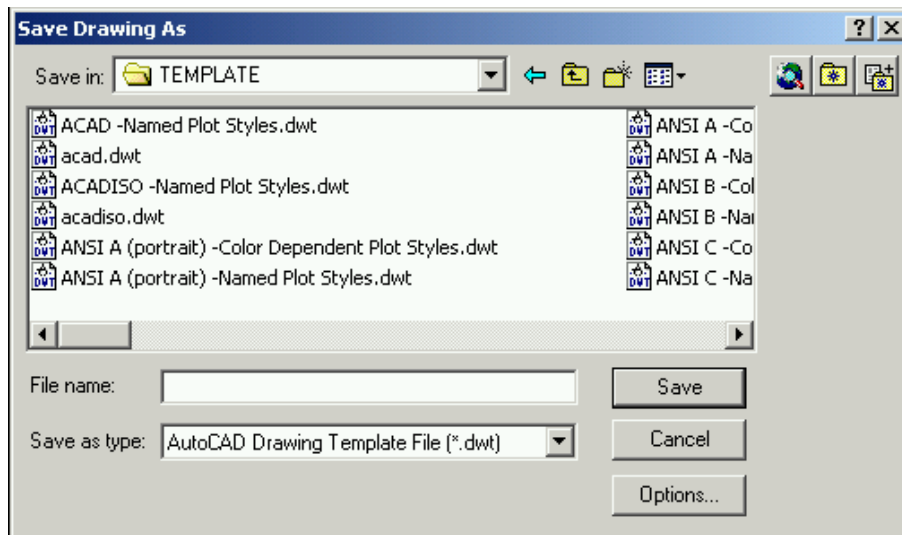
- Move Command
    - Changes the location of an object on a drawing
    - Invoke by typing *m* on the command line, selecting *Move* from the *Modify* menu or selecting the *Move* tool from the *Modify* toolbar
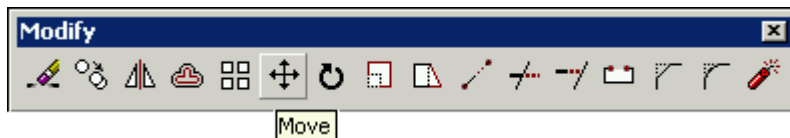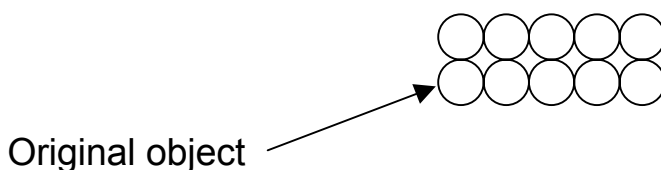

    - Select the object to be moved
    - Select a *base point* and a *displacement point*, which together define the vector that indicates the distance and direction of the object's movement

- Copy Command
    - Copies an object to another location while leaving the original intact
    - Invoke by typing *co* on the command line, selecting *Copy* from the *Modify* menu, or selecting the *Copy* tool from the *Modify* toolbar
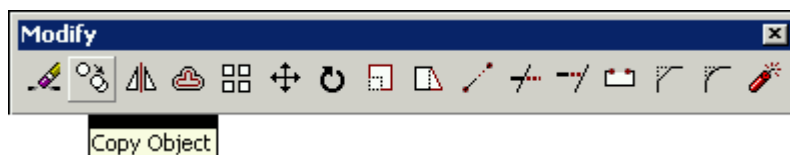

    - Select the object to be copied
    - Select a *base point* and a *displacement point* to indicate the location of the duplicate

Template Drawings

- Multiple Copies
    - Similar to the basic *copy* procedure
    - After selecting the object to be copied, type *m* (for *multiple* option) on the copy command line
    - Indicate a *base point* and *displacement point* for the first copy
    - AutoCAD will prompt you for additional *displacement points* that indicate the vector from the original base point to each new copy
    - Hit *enter* to exit the *multiple copy* command

- Rectangular Arrays
    - Arranges copies of a figure in a rectangular array
    - Powerful alternative to simple copying
    - Invoke by typing *ar* on the command line, selecting *Array* from the *Modify* menu, or selecting the *Array* tool from the *Modify* toolbar
    - Select the object to be copied
    - Type *r* to indicate *rectangular array*
    - AutoCAD will prompt for number of rows, number of columns, distance between rows, and distance between columns. The array is built *up and to the right* from the original object

Original object

Arcs and Polar Arrays

Polar Arrays

- Arranges copies of a figure in an arc
- Powerful alternative to simple copying
- Procedure:
    - Type *ar* on the command line, select *Array* from the *Modify* menu, or select the *Array* tool from the *Modify* toolbar
    - Select the object to be copied
    - Type *p* to indicate *polar array*
    - Click to specify the center point of the array
    - Specify number of items in the array
    - Specify angle to fill
        - Can be as large as 360º
        - positive angle = clockwise; negative angle = counter-clockwise
   - Indicate *yes (y)* if items are to be rotated as they are copied

Center point of array
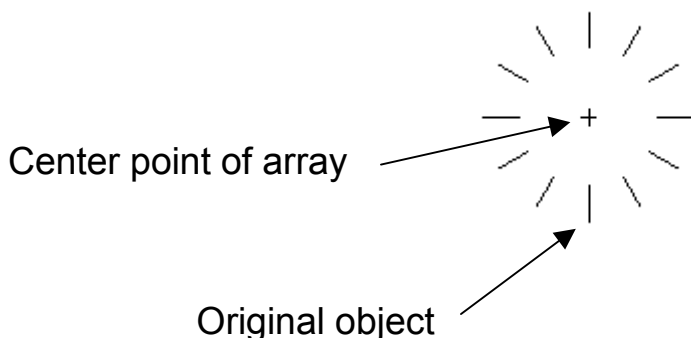
The objects in this array were rotated as they were coped

Original object

Arcs and Polar Arrays

Arcs

- AutoCAD provides at least eight different methods for drawing arcs

- Each method requires the same basic information: a start point, an end point, and information about what circle the arc is part of

- Simplest method: The *3-points* option (default)
 - Type *a* on the command line, or select the *Arc* tool from the *Draw* toolbar

  - Select points 1 and 2, the starting point, and a point on the arc
  - Point 3 is the arc endpoint. As you move your cursor, AutoCAD displays an arc that corresponds to the current location of point 3
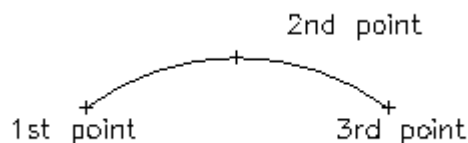
- Other methods include specifying start point, circle center, and end point; start point, circle center, and angle of sweep; and start point, circle center, and end point of the arc. These and others are shown in Figure 5.5 of the text


Rotate

- Used to rotate objects after they have been drawn
- It is often easier to draw an object in a vertical or horizontal position, and then use the *rotate* command to move it to a diagonal position
- To access the *Rotate* command:
    - Select an object to be rotated
    - Type *ro* on the command line, select *Rotate* from the *Modify* menu, or choose the *Rotate* tool from the *Modify* toolbar
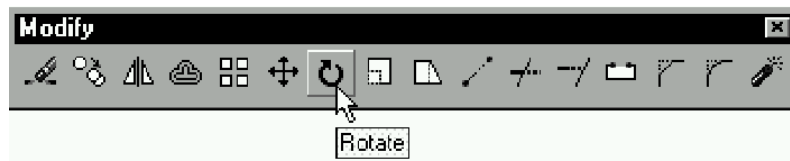
- Base Point
    - The point about which the object is rotated
    - The base point can be located anywhere on the object, and can be located on the object itself
- Angle of rotation
    - Positive angle provides a counter clockwise rotation
    - Generally specified with respect to 0°
    - Typing *r* after specifying the base point allows the rotation angle to be specified with respect to an angle other than 0°

Base point

This figure was generated by repeating the rotate command three times

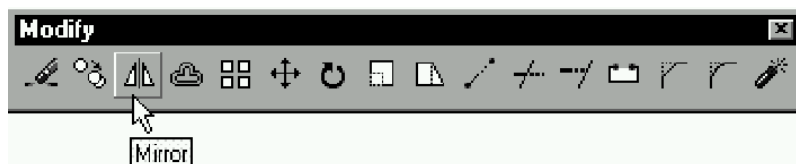Original object

Arcs and Polar Arrays
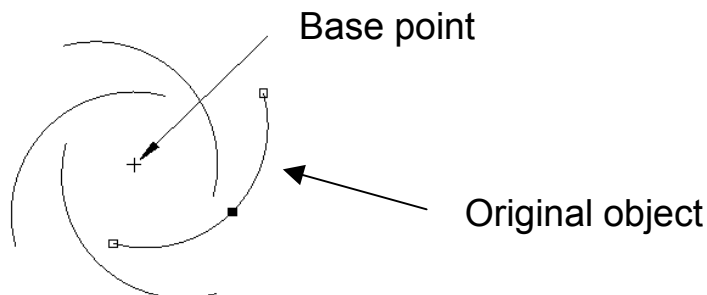
Mirror Images

- All points on an original object are reflected across a *mirror line* at an equal distance and opposite orientation

- Original object can be retained, or deleted from the drawing

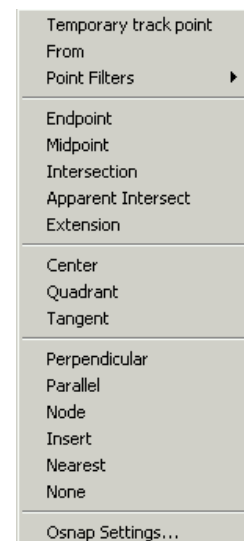Mirror line (not visible in finished drawing)
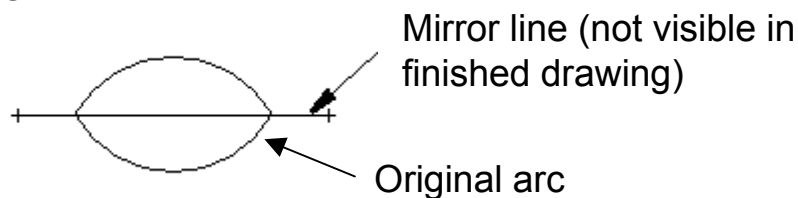
Original arc

- To access the *Mirror* command:
  - Select an object to be reflected
  - Type *mi* on the command line, select *Mirror* from the *Draw* menu, or choose the *Mirror* tool from the *Modify* toolbar

  - Choose the endpoints of the mirror line
  - Retain or delete the original object, when prompted
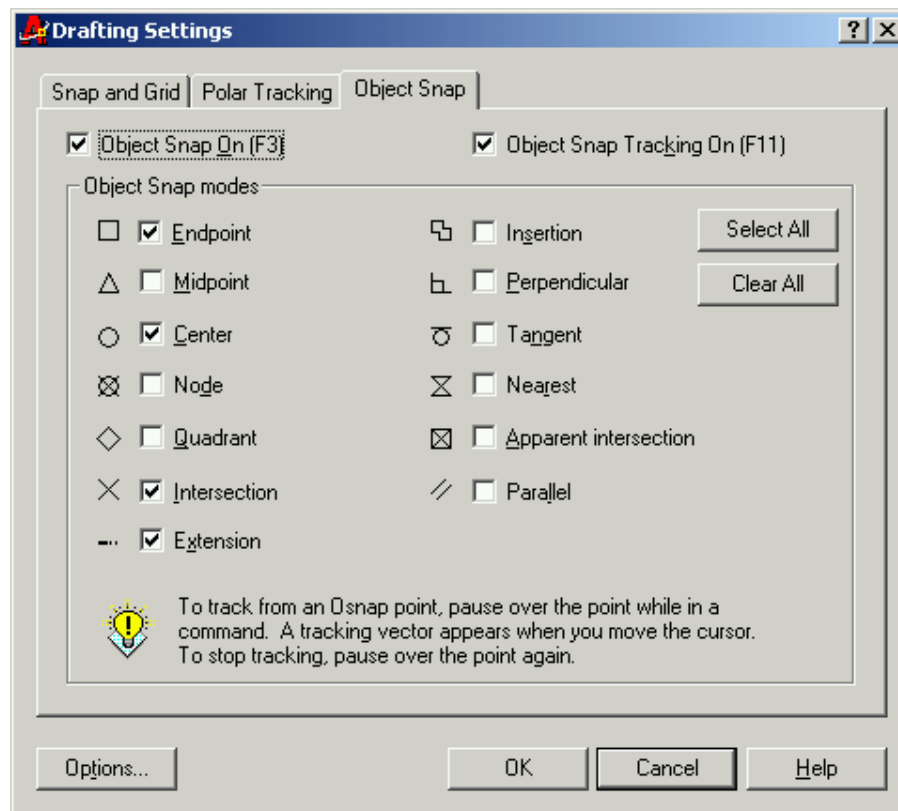
Object Snap

Selecting Points with Object Snap

- Snaps cursor to a specified point on a previously drawn object

- Overrides grid (coordinate system) snap function

- Increases accuracy of drawing where lines are to meet at one specific point

- Drawing an object using *Object Snap Mode:*

  - When AutoCAD asks for a point, turn on *object snap mode* by choosing a command from the pop-up *shortcut menu* (hold down shift or control key and right-click with the mouse)

  - Once a snap mode has been chosen, move the cursor near the desired snap point. When the *snap marker* indicates that AutoCAD recognizes an appropriate point, left-click to pick the point

  - Subsequent points can be picked using snap mode by repeatedly accessing the cursor menu

  - Specific object snap modes can also be chosen via the object snap toolbar (right-click on any toolbar and choose object snap from the pop-up menu)

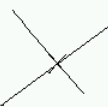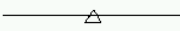| Temporary track point |
| From |
| Point Filters ▶ |
| Endpoint |
| Midpoint |
| Intersection |
| Apparent Intersect |
| Extension |
| Center |
| Quadrant |
| Tangent |
| Perpendicular |
| Parallel |
| Node |
| Insert |
| Nearest |
| None |
| Osnap Settings… |

Object Snap

Object Snap

Running Object Snap Mode

- Allows all points to be chosen using *object snap mode*

- Type *os* on the command line, or choose *Object Snap Settings* from the *pop-up shortcut menu* described previously
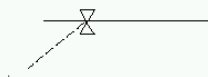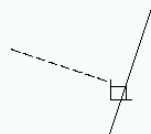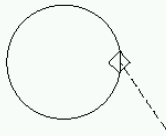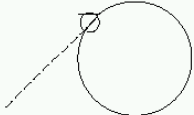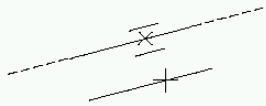

- Check boxes toggle various object snap (osnap) modes on and off

Object Snap

Description of Object Snap Modes

TYPE	APPEARANCE	DESCRIPTION
CENter		Snaps to the center of an arc, circle, ellipse, or elliptical arc.
ENDpoint		Snaps to the closest endpoint of an arc, elliptical arc, line, mline, polyline, ray, or to the closest corner of a trace, solid or 3D face.
INSertion	(See Chapter 10)	Snaps to the insertion point of a block, attribute, shape or text.
INTersection		Snaps to crossing or meeting point of arcs, lines, circles, ellipses, elliptical arcs, mlines, polylines, rays, splines, or xlines.
APParent Inter		Snaps to apparent crossing or meeting point of arcs, lines, circles, ellipses, elliptical arcs, mlines, polylines, rays, splines, or xlines. If apparent intersection and intersection are on at the same time, varying results may occur.
MIDpoint		Snaps to the midpoint of an arc, circle, ellipse, elliptical arc, line, mline, polyline, solid, spline, or xline.

Object Snap

Description of Object Snap Modes (continued)

TYPE	APPEARANCE	DESCRIPTION
NEArest		Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, mline point, polyline, spline or xline.
NODe	⊗	Snaps to a point object.
PERpendicular		Snaps to a point on an arc, circle, ellipse, elliptical arc, line, mline, ray, solid, spline or xline.
QUAdrant		Snaps to nearest quadrant point of an arc, circle ellipse, or elliptical arc. 0, 90, 180, or 270 degrees
QUIck	(Modifies other modes)	Snaps to the first snap point found. Quick must be used in conjunction with other objects snap modes.
TANgent		Snaps to the tangent of an arc, circle ellipse, or elliptical arc.
EXTension		Snaps to the extension point of an object. Establish an extension path by moving the cursor over the endpoint of an object. A marker is placed on the endpoint. While the endpoint is marked, the cursor snaps to the extension path of the endpoint.
PARallel		Snaps to an extension parallel with an object. When the cursor is moved over the endpoint of an object, the endpoint is marked and the cursor snaps to the parallel alignment path to that object. The alignment path is calculated from the current "from point" of the command.

# Object Snap

Object Snap Tracking

- New AutoCAD 2000 feature
- Creates temporary construction lines from designated *object snap points* to the edge of the display
  - Similar to *polar tracking*
  - Construction lines can be drawn along orthogonal axes, or along polar tracking angles, through *acquired points* to the edge of the display
- By acquiring more than one point, AutoCAD can exactly locate points that could not be easily found any other way
- Example: a point whose vertical coordinate corresponds to the tangent point between a line and a circle

1: Initial point specified in *draw* command

2: Point acquired at corner of square

3: Point acquired tangent to circle

4: Point of interest at intersection of construction lines

Point 1

Point 4   Point 3

Endpoint: < 90°, Endpoint: < 180°

Point 2

Object Snap
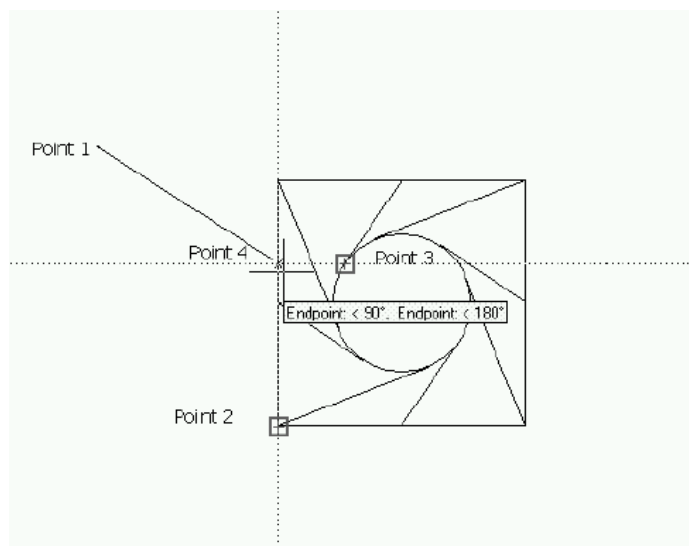
Object Offset

- Allows objects to be duplicated at specific distances away from the original. The new object is typically drawn parallel to the first

- Invoked by either selecting *Offset* from the *Modify* menu, or choosing the *Offset* tool from the *Modify* toolbar
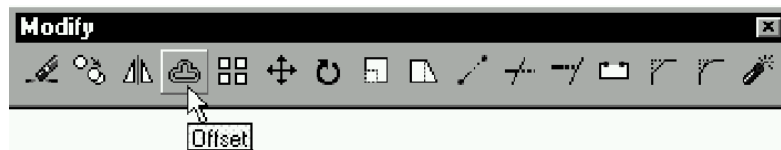

- Offset distances can be specified using points found via the *object snap tracking* method
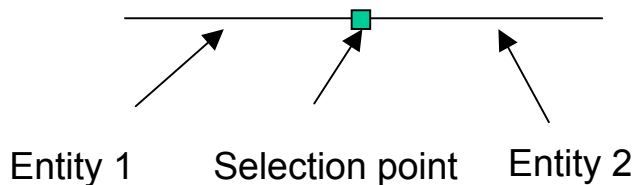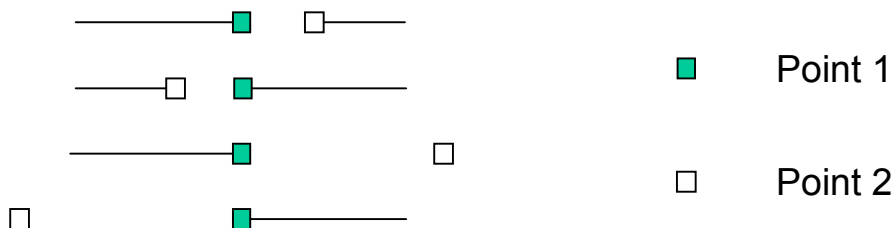
Object Snap

- Break
    - Allows a line to broken into two or more distinct entities
    - Allows unwanted sections of a line to be removed
- Invoke Break by either
    - Typing *break* on the AutoCAD command line
    - Selecting *Break* from the *Modify* drop down menu
    - Selecting the *Break* tool from the *Modify* toolbar
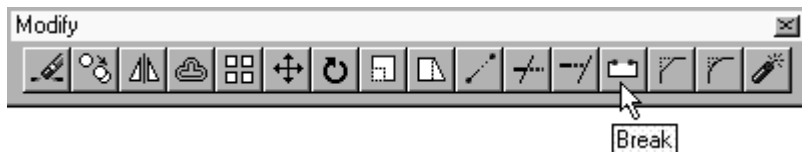
- After the break command has been selected, the line will be broken into two entities defined by the position of the point used to select the line

Entity 1   Selection point   Entity 2

- The position of the second point will determine which part of the line is deleted (4 cases shown)


■ Point 1

□ Point 2

Object Snap

- Trim
    - Allows objects to be shortened at their point of intersection with other objects
    - Works with lines, circles, arcs and polylines
- Invoke Trim by either
    - Typing *tr* on the AutoCAD command line
    - Selecting *Trim* from the *Modify* drop down menu
    - Selecting the *Trim* tool from the *Modify* toolbar

- After the trim command has been selected, choose objects which will serve as *cutting edges*
- Once all cutting edges have been selected, hit *Enter* to begin trimming objects
    - Click on the object to be trimmed on either side of a cutting edge
    - Portion of the object which is located on the side of the cutting edge chosen will disappear:

Cutting Edge
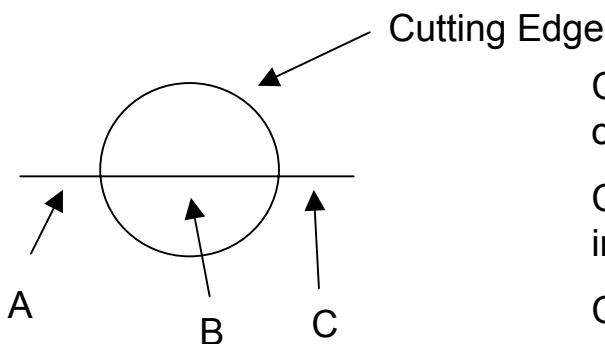
Choosing at A will cause portion of line on outside left of circle to disappear

Choosing at B will cause portion of line inside circle to disappear

Choosing at C will cause portion of line on outside right of circle to disappear

Object Snap

- Extend
    - Allows objects to be extended to an existing (or temporary) object or line
    - Similar in behavior to the *trim* command
- Invoke Extend by either
    - Typing *ex* on the AutoCAD command line
    - Selecting *Extend* from the *Modify* drop down menu
    - Selecting the *Extend* tool from the *Modify* toolbar

- After the extend command has been selected, choose objects which will serve as *boundary objects*
- Once all boundary objects have been selected…
    - Hit *Enter* to begin choosing objects to extend
    - Choose a point on the object to be extended that is closest to the boundary object

Points selected on original object

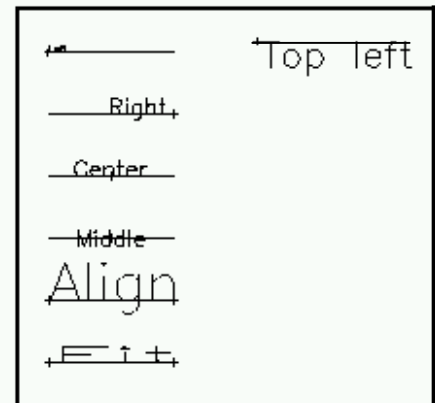Extended Lines

Boundary Line
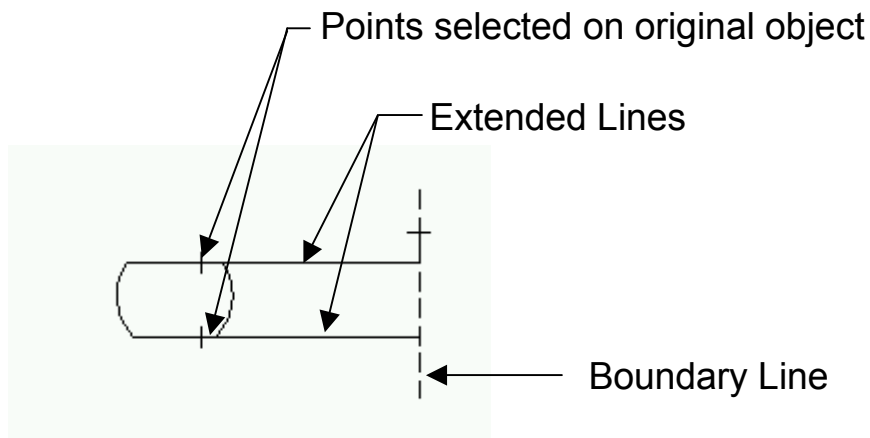
Text

- AutoCAD 2000 has two commands which allow the addition of text to a drawing:
  - DTEXT
    - Allows single lines of text to be entered
    - Shows the text on the screen as it is entered
    - Allows backspacing through text to correct errors
  - MTEXT
    - Allows you to type paragraphs of text in a dialog box
    - Window containing text is then positioned on the drawing
- Invoke DTEXT by either
  - Typing *dt* on the command line
  - Selecting *Text, Single Line Text* from the *Draw* menu
  - (Text tool on the *Draw* toolbar invokes MTEXT, not DTEXT)

Text

Using DTEXT

- Justification options: specified by entering *J* at the *DTEXT* command prompt, and then entering the *type abbreviation:*

| TEXT JUSTIFICATION	
<START POINT> TYPE ABBREVIATION	**TEXT POSITION** + INDICATES START POINT or PICK POINT
A	ALIGN
F	FIT
C	CENTER
M	MIDDLE
R	RIGHT
TL	TOP LEFT
TC	TOP CENTER
TR	TOP RIGHT
ML	MIDDLE LEFT
MC	MIDDLE +CENTER
MR	MIDDLE RIGHT
BL	BOTTOM LEFT
BC	BOTTOM CENTER
BR	BOTTOM RIGHT

Examples above shown with respect to a horizontal line; examples to the left shown with respect to text insertion point.

- You must specify text insertion point, height, and angle when prompted to by the *DTEXT* command
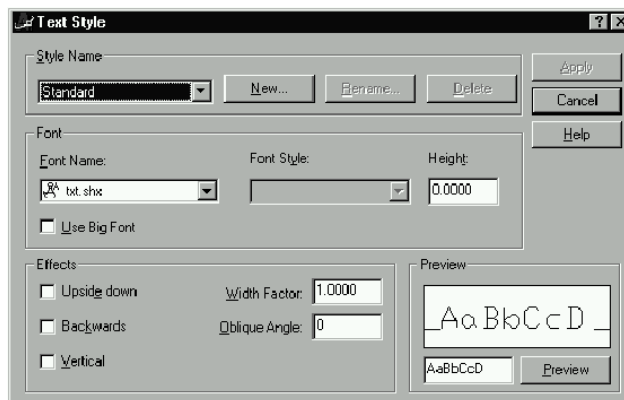

Text

Editing Text

- <u>DDEDIT</u>: Allows text to be edited in place on an AutoCAD drawing
  - Invoke DDEDIT by either
    - Typing *ddedit* on the command line
    - Selecting *Text* from the *Modify* menu
    - Selecting the *Edit Text* tool from the *Modify II* toolbar
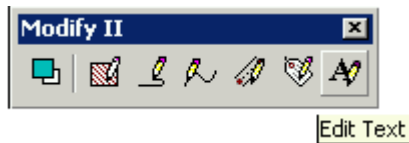
  - At the DDEDIT (select) prompt:
    - Choose a line of text (or other text object) to edit by clicking on it
    - Make appropriate changes in the dialog box that appears
    - Click *OK* button to make the changes
- <u>SPELL</u>: Checks the spelling of all text objects
  - Invoke SPELL by either
    - Typing *spell* on the command line
    - Selecting *Spelling* from the *Tools* menu
  - Select text to check by left clicking with the mouse, or type *all* to check every text object
  - Either *Change* or *Ignore* the suggested changes AutoCAD makes each time it notes an unrecognized word

---

Text

<u>Fonts</u> <u>and</u> <u>Styles</u>

- Font
    - Basic symbol or character shape used with DTEXT and MTEXT commands
    - Many to choose from, but difficult to create your own
    - Default font: *txt*
- Style
    - Refers to the size, orientation, and spacing of characters as they are typed in any specific font
    - STANDARD style is the default, but variations can be created by the user
- Invoke Style command by either
    - Typing *st* on the command line
    - Selecting *Text Style* from the *Format* menu
- Text Style Dialog Box
    - Allows previously defined styles to be selected
    - Allows new text styles to be created and stored for future use

Text

Changing Object Properties with MATCHPROP

- Allows some or all properties of one object to be changed (matched) to those of another

- Properties that can be transferred between objects include: layer, linetype, color, thickness, text style, dimension style, etc.

- Invoke MATCHPROP by either
    - Typing *matchprop* on the command line
   - Selecting the *Match Properties* tool from the *Standard Toolbar*

Match Properties

- Source object: Object whose properties will be exported to destination object(s)

- Limit properties to be matched by typing *s* at the prompt. *Property Settings* dialog box will then appear
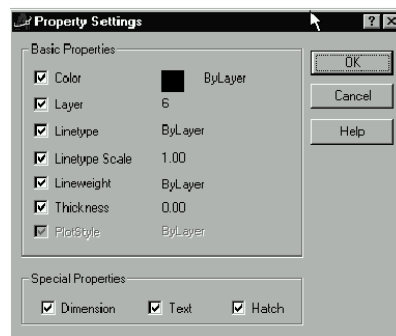

- Choose destination objects (by left clicking) to complete the command

Text

Scaling Previously Drawn Entities

- *SCALE* allows the size of previously-drawn entities to be changed
- Invoke Scale command by either
  - Typing *sc* on the command line
  - Selecting *Scale* from the *Modify* menu
 - Choosing the *Scale* tool from the *Modify* toolbar
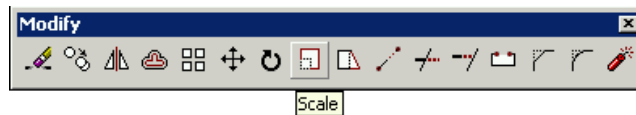
- Choose objects to *scale* by pointing or windowing
- The *base point* refers to the first point of a line that determines the *scale factor*.
  - If the line length specified is less than 1, the selected objects will decrease in size
  - If the line length specified is greater than 1, the selected objects will increase in size
- Location of the *base point* determines the location of the scaled object
- Scale factors can be entered by using the *reference (r)* option
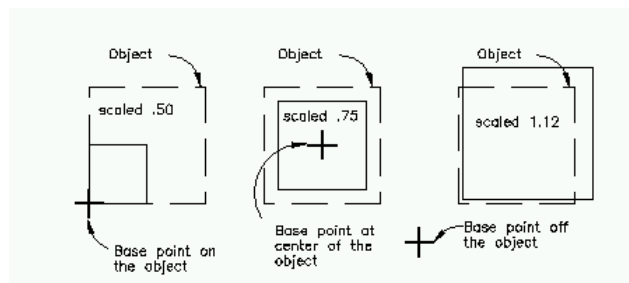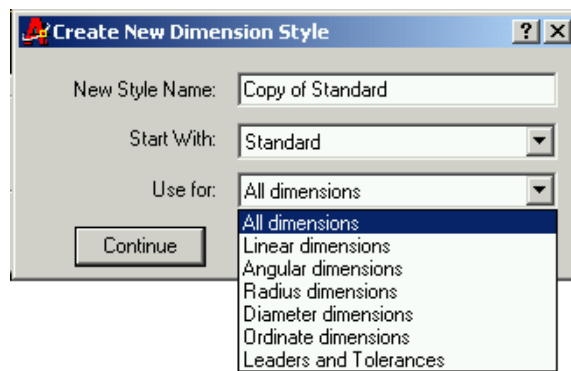

Dimensions

Dimension Style

- Set of variable settings that controls the text style and geometry of AutoCAD dimensions
- Dimension style should be created before dimensioning any AutoCAD objects
- Dimension style can be saved as part of the drawing template
- Dimension Style Manager Box: Allows dimension style to be set, created and previewed
- Access Dimension Style Manager Box by either
  - Typing *d* on the command line
  - Selecting *Dimension style* from the *Format* menu
  - Selecting *Style* from the *Dimension* menu
  - Selecting the *Dimension Style* tool from the *Dimension* toolbar
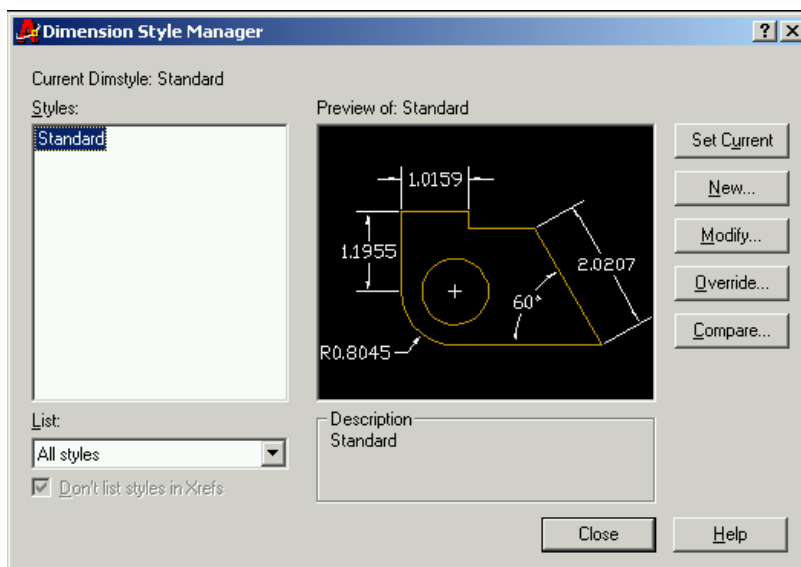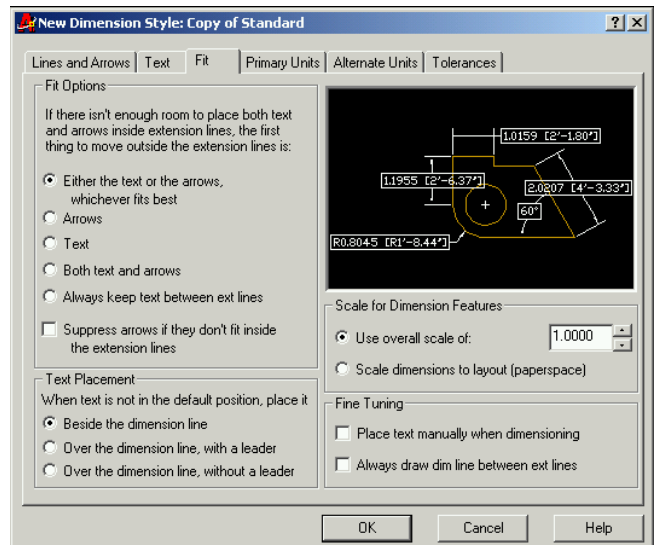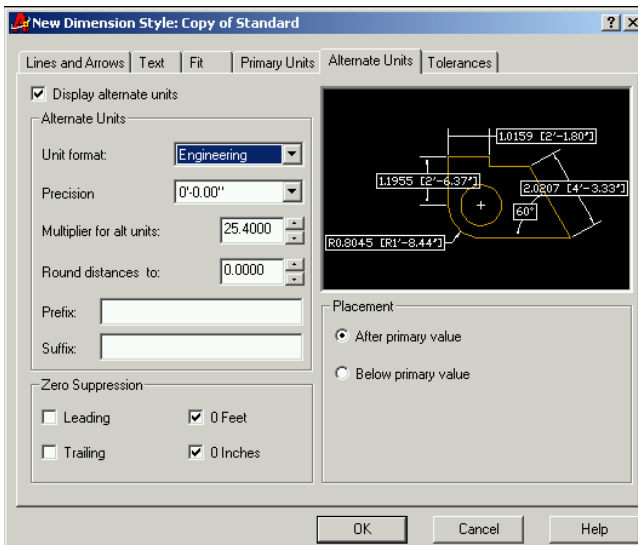
Dimensions

Creating a New Dimension Style

- Click on *New* button in Dimension Style Manager

- Specify name of new style, dimensions types (linear, angular, etc.) the new style should be used for, and "starting point" for new style, as shown

- Details of new dimension style are specified on the tabbed pages of the dialog box shown below
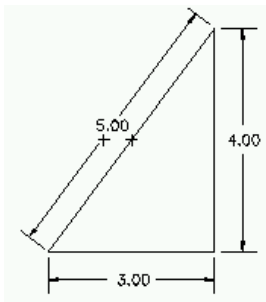

Dimensions

Dimension Types

- Linear dimensions: use *dimlinear* command

- Baseline format (multiple linear dimensions measured from the same origin): use *dimbaseline* command

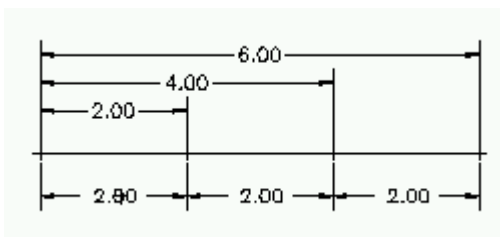- Continued format (a string of chained linear dimensions): use *dimcontinue* command

← Continued dimension

← Baseline dimension

- *QDIM:* New AutoCAD 2000 command that automates the creation of of certain types of multiple dimension formats
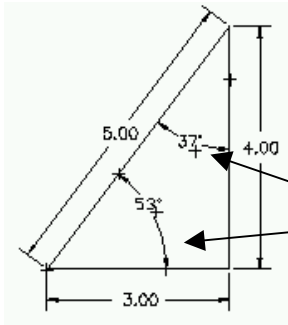

Dimensions

Dimension Types (continued)

- Angular dimensions: use *dimang* command

  Angular Dimension

  Angular Dimensions

- Insert Center Mark: use *dimcenter* command

  Center Mark

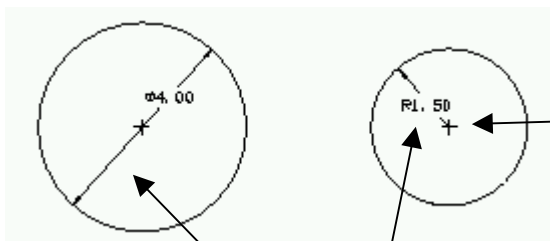- Dimension circle diameter: use *dimdia* command

  Diameter Dimension

- Dimension circle radius: use *dimrad* command

 Radius Dimension

  Center mark

  Diameter dimension   Radial dimension

Dimensions

Automated Hatching

- **BHATCH** (Boundary Hatch): Automatically hatches based on the nearest boundary to the specified points
- Access BHATCH by
  - Typing *bh* on the command line
  - Selecting *Hatch* from the *Draw* menu
 - Selecting the *Hatch* tool from the *Draw* toolbar

- Tabbed pages of Boundary Hatch dialog box allow…
  - Choice of predefined or user-defined hatch patterns
  - Hatch boundaries to be determined by picking objects
  - Hatch boundaries to be determined by picking points on the interior of a boundary
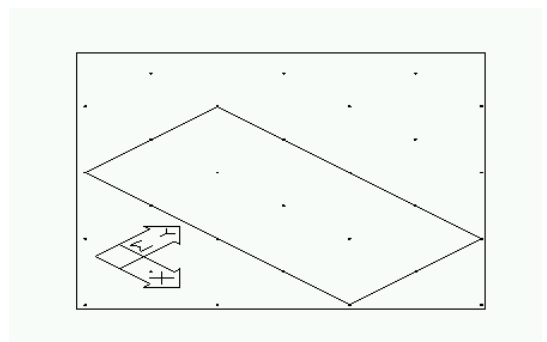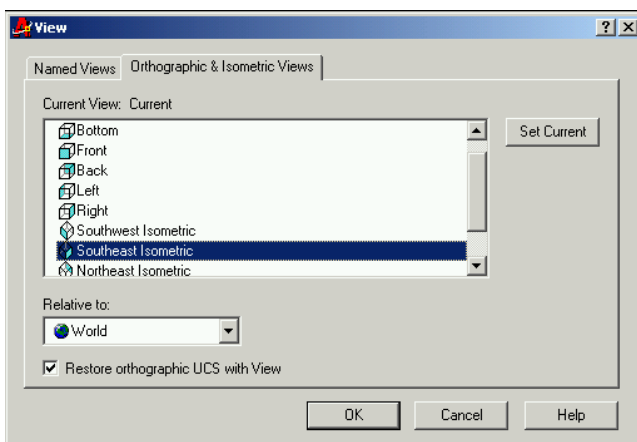 - Disposition of interior "islands"

Wire Frame and Surface Models
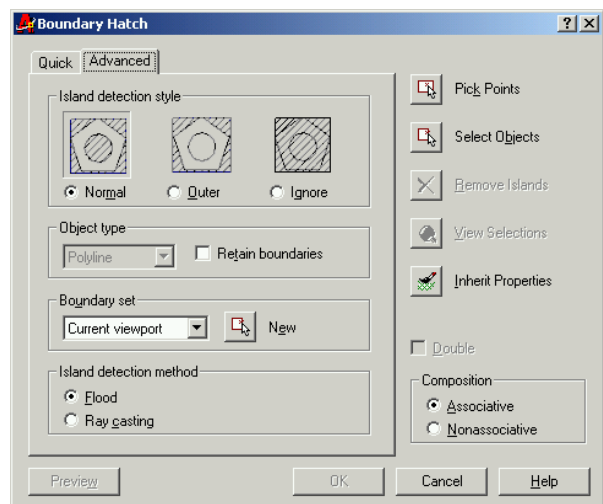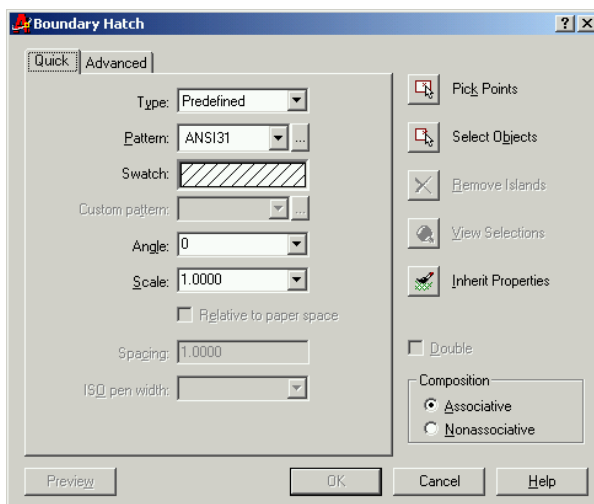
Creating and Viewing 3D Objects

- **UCS**: User Coordinate system

- **World Coordinate System**
    - AutoCAD's *absolute* coordinate system
    - All other UCS's are defined with respect to the WCS
    - WCS is indicated by a *W* in the UCS icon
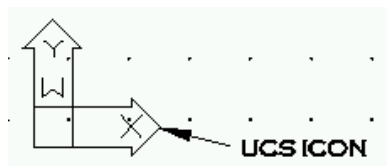
    - WCS icon corresponds to (0,0,0) in 3D space
    - Z-Axis is indicated here as positive perpendicular to the screen (2D view)

- Viewpoint
    - The point from which you are viewing the objects within AutoCAD. Access via the *Named Views* tool from the *Standard Toolbar*
   - **Example**: *Southeast Isometric* viewpoint

---

Wire Frame and Surface Models

Creating and Viewing 3D Objects (continued)

- Coordinate system icons indicate current viewpoint

ICON	DESCRIPTION
Y, W, X	WCS (World Coordinate System) "W" appears on "Y" arm
Y, X	UCS (User Coordinate System) NO "W" appears on "Y" arm
Y, W, +, X	+ Appears in box and "W" appears on the "Y" arm if the current UCS is the same as the WCS
Pos. "Z"	Box appears at the base of ICON if viewing UCS from above its X—Y plane
Neg. "Z"	Box is missing if viewing UCS from below its X—Y plane
(pencil icon)	Broken pencil ICON appears if viewing direction is "EDGE ON" or near "EDGE ON" X—Y plane of current UCS

- New UCS can be defined by choosing New *UCS, Origin* from the *Tools* menu
- Same menu can be used to switch among various stored user coordinate systems

---

Wire Frame and Surface Models

Creating and Viewing 3D Objects (continued)

- Pointing devices (left mouse button, etc.) cannot be used to enter 3D point locations. Methods of pointing are limited to either
    - Typing 3D coordinates from keyboard
    - Using X/Y/Z point filters
    - Snapping to objects
- Viewport mode
    - Allows the simultaneous display of up to four different views
    - Any work done in one view will be reflected in all others
    - Views defined in dialog box by choosing *Viewports, New Viewports* from the *View* menu
    - Note that viewport arrangement is shown schematically in the *Preview Pane*

Wire Frame and Surface Models
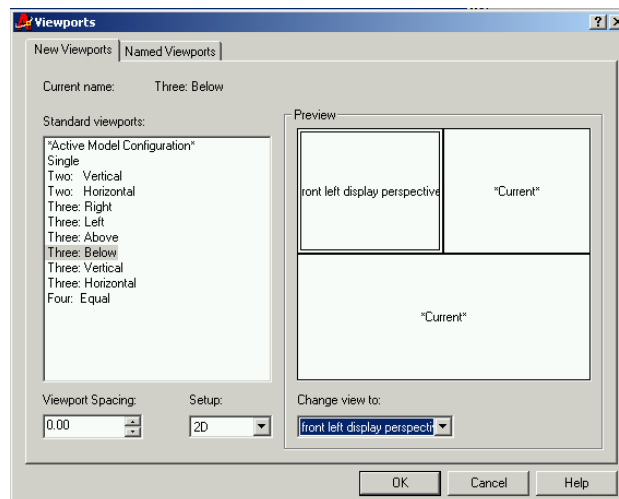
Wire Frame versus Surface Models
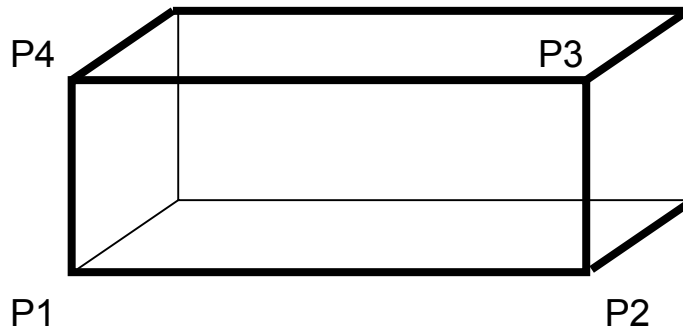
- Wire Frame Model: Model consists of only the points and lines connecting the points
- The surfaces of the object "do not exist" within AutoCAD unless they are specifically defined
- Triangular and quadrilateral faces may be defined using the *3DFACE* command
  - Invoke command by choosing *Surfaces, 3D Face* from the *Draw* menu
  - Enter points P1, P2, P3, P4 in order as indicated on the above diagram (counterclockwise)
  - Points can be typed on the command line, or can be picked in any view port
- Hidden Lines
  - Lines that would not be seen when "hidden" behind solid surfaces
  - In the above surface model, the "thin" lines are hidden, while the thick lines are visible

---

Wire Frame and Surface Models

## Wire Frame versus Surface Models (continued)

- Hidden lines can be temporarily removed from <u>specific</u> <u>views</u> by using the *HIDE* command
- Hidden lines are returned to view whenever the screen is regenerated
- Type *hi* on the command line, or choose *View, Hide* from the menu bar to access the *hide* command
- Hidden line removal has no effect on wire frame models (because there are no surfaces present to block the view of any lines)
- This command may act strangely if all of your layers are not on and unfrozen:
    - Frozen layers are ignored by the *Hide* command
    - Layers that are off are treated like visible layers, but are not displayed. Surfaces that have been created on a layer that is not turned on can therefore "hide" lines that are on a visible layer.
- Example of hidden line removal (Two viewports):

Wire Frame and Surface Models
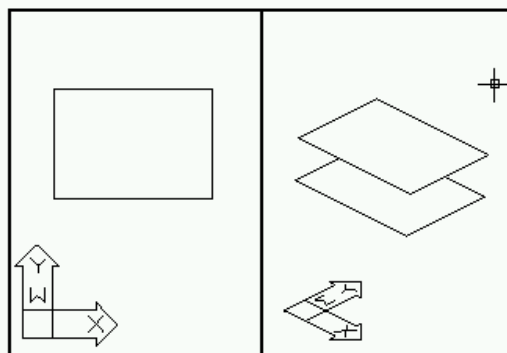
Other Surface Generation Techniques

- TABSURF (tabulated surface) requires…
  - Line or curve to define the shape of the surface
  - Vector to show surface size (length) and direction
  - Surface is generated by repeating the shape of the original curve at every point along the path specified by the vector
  - Two viewport example is shown below (note that curve and surface is contained within the XY plane)
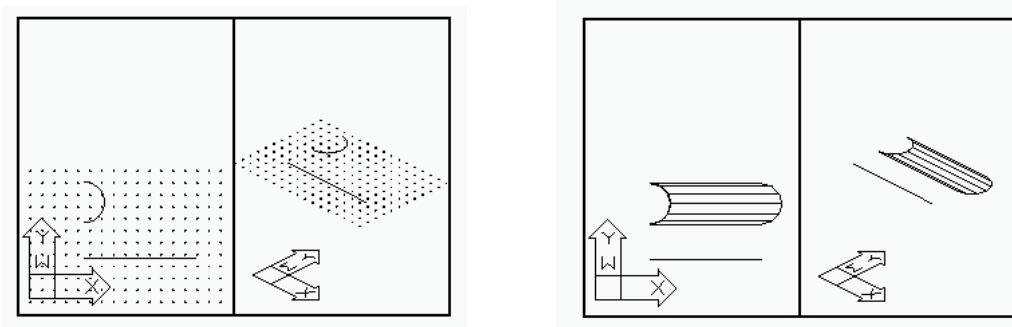

- RULESURF
  - Allows a surface to be generated between two different paths
  - Example below shows surface generated between the line and the curve shown to the above left

Wire Frame and Surface Models
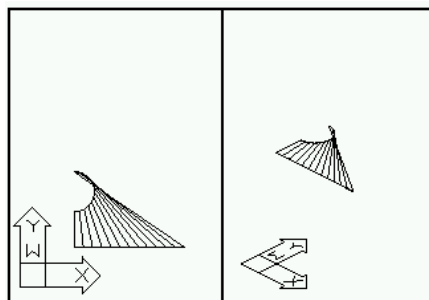
Other Surface Generation Techniques (continued)

- **EDGESURF**
    - Generates a surface bounded by four curves
    - Restriction is that the four edges must create a bounded surface – they must enclose a flat or curved area
    - Example of curves and surface formed using EDGESURF command
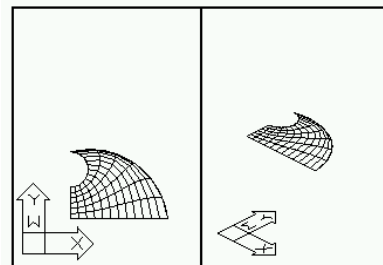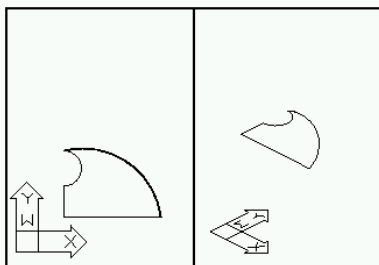
- **REVSURF**
    - Creates surfaces by spinning a curve through a given angle around an axis of revolution
    - Surfaces always follow an arc-shaped path – they generate fully three-dimensional surfaces!
    - Example below shows surface generated using the figure shown to the above left

Wire Frame and Surface Models
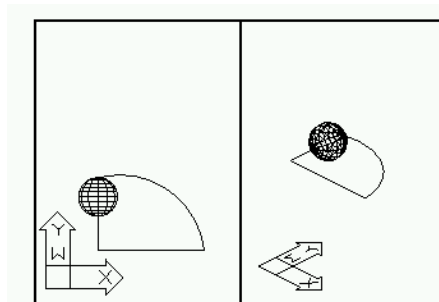
Introduction to Solid Modeling

- Complex 3D objects are created by combining or subtracting basic solid shapes
- The basic solid shapes used are known as *primitives* and are found on the *Solids* toolbar
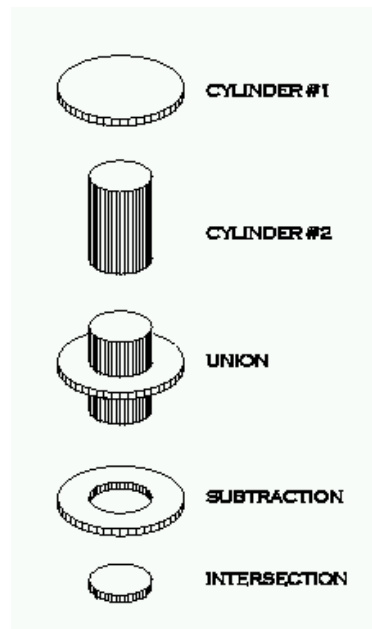

- Solids are created from the primitives using the three basic Boolean operators found on the *Solids Editing* toolbar

- Example: Washer created from the subtraction of two cylinders

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Plotting (Chapter 2)

Simple Method for Plotting

- In AutoCAD 2000, generating a hardcopy of your drawing is referred to as *plotting*, regardless of whether a printer or a plotter is used

- Simplest method: directly plot from current model space to paper

- Type *Ctrl-P,* select the Plot tool from the standard toolbar, or select *Plot…* from the pull down menu to view *Plot* dialog box

- *Plot Settings* Tab: Contains options which determine how the drawing will appear on paper when plotted

Plotting (Chapter 2)
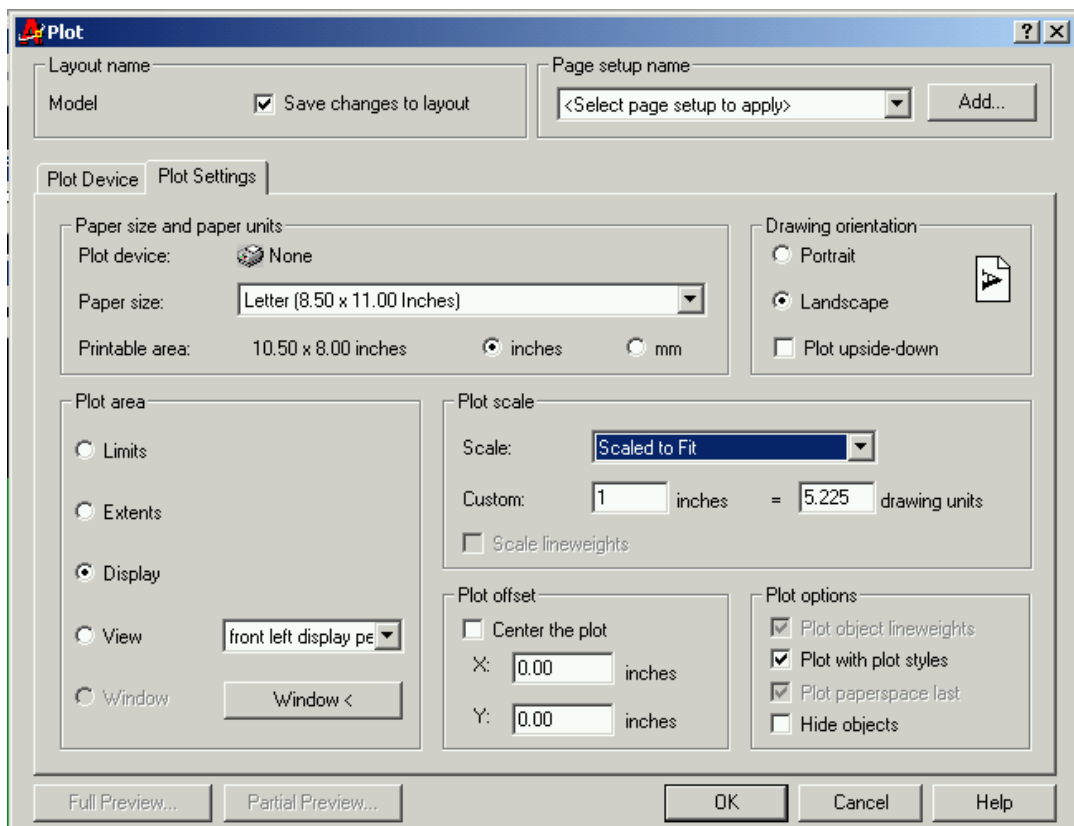
Plot Window

- Dialog box option; simplest method of defining an area to be printed or plotted

- In the Plot Settings Dialog Box, click | Window < |

- Use the mouse to define the plot window as shown:

Corner 1

Corner 2
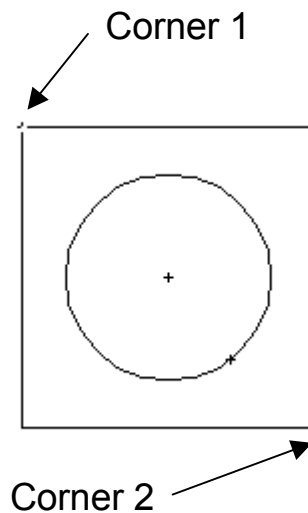
- Make sure that *Scaled to Fit* is visible in the *Plot Scale* area of the dialog box, so that the object fits correctly on the paper when plotted

Plot scale

Scale: Scaled to Fit

Custom: 1 inches = 5.225 drawing units

☐ Scale lineweights

Plotting (Chapter 3)
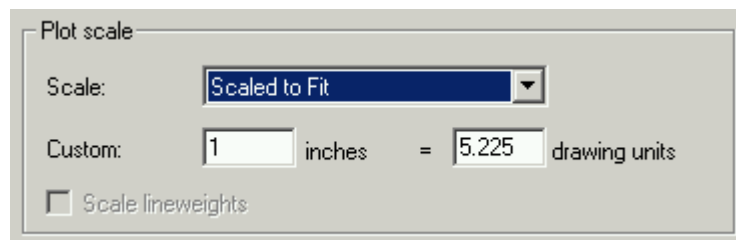
Plot Preview

- Allows drawings to be previewed before they are plotted on paper
- *Plot Preview* is found under the *Plot Device* tab of the *Plot* dialog box
    - *Partial Preview* shows an outline of the effective plotting area in relation to the printable area, as well as a triangular origin icon
    - *Full Preview* shows an image of the drawing in relation to its actual position on the paper
- Choose *OK* to print if drawing placement looks satisfactory

Plotting (Chapter 4)
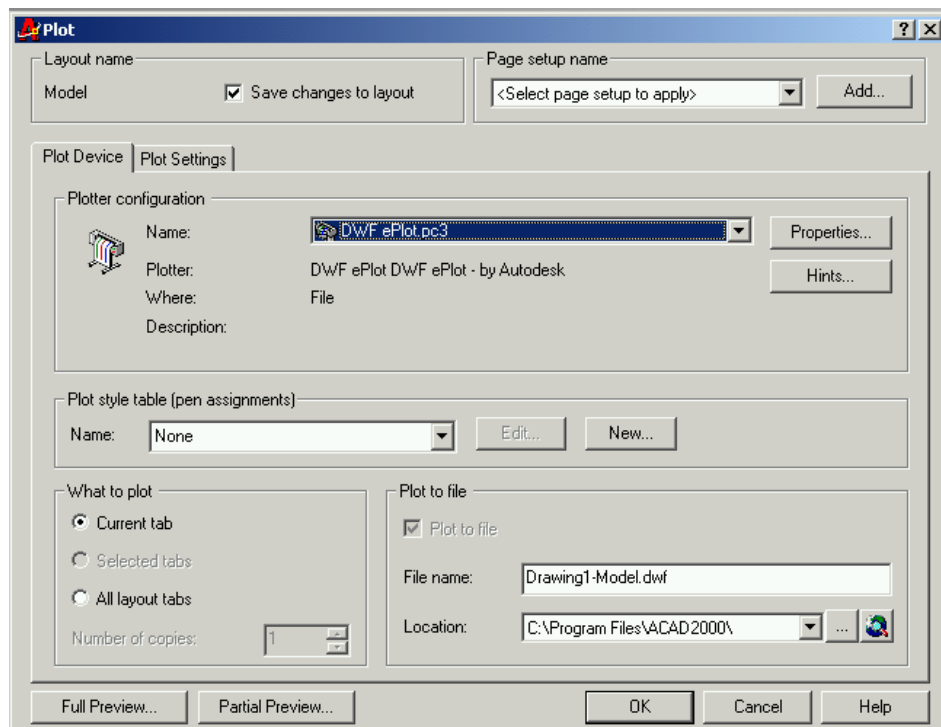
Plot Device

- *Add a Plotter Wizard*
    - Installs a plotter driver
    - Accessed by choosing *Plotter Manager* from the *Files* menu
    - If no plotter is available, choose the *DWF ePlot* utility, which simulates the behavior of a plotter
- *Plot Styles*
    - Accessed under the *Plot Device* tab of the *Plot* dialog
    - Allows a drawing to be plotted differently from how it appears on the screen
- *What to Plot*
    - Accessed under the *Plot Device* tab of the *Plot* dialog
    - Allows you to choose among the various model and layout views that comprise your AutoCAD drawing
- *Plot to File*
    - Accessed under the *Plot Device* tab of the *Plot* dialog
    - Creates a *plot file* that allows the drawing to be plotted at a later time or at another site
    - Plot file has the same name as the AutoCAD drawing, with a *.plt* extension

Plotting (Chapter 4)
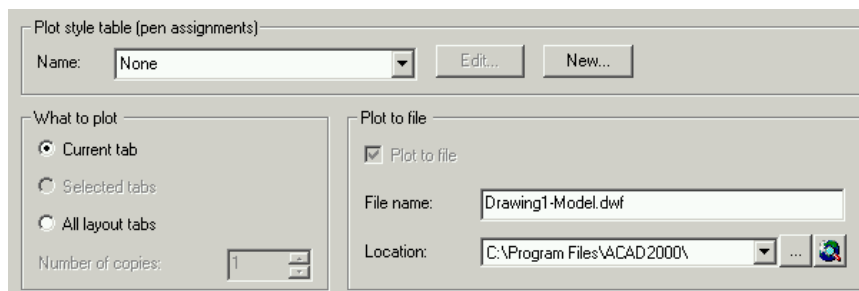
Plot Settings

- Drawing Orientation
  - Accessed under the *Plot Settings* tab of the *Plot* dialog
  - Portrait: paper is oriented with the short edge parallel to the horizontal direction
  - Landscape: paper is oriented with the long edge parallel to the horizontal direction
  - Default mode is generally set to *landscape*, which corresponds to how a drawing is depicted on the screen

- Plot Area
  - Accessed under the *Plot Settings* tab of the *Plot* dialog
  - Determines what is plotted as compared to what is displayed on the screen
  - Display: Plots only what is on the screen
  - Extents: Plots the entire drawing area
  - Limits: Plots the drawing limits specified using the LIMITS command
  - View: Only active when multiple views are used
  - Window: Only active if a plot window has been selected, as described in Chapter 2

Plotting (Chapter 4)
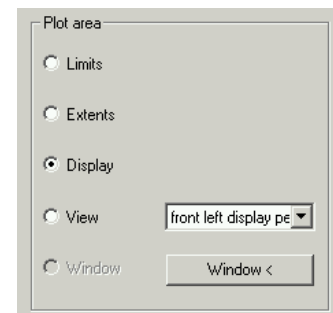
Plot Settings (continued)

- Plot Scale
    - Accessed under the *Plot Settings* tab of the *Plot* dialog
    - Scaled to fit
        - Maximizes the size of the drawing when plotted
        - No relation between size of objects as depicted and actual size
    - Specific scales can be chosen by using the scale pull-down menu, when required
- Plot Offset
    - Sets the location of the drawing origin on the plot
    - Location of this origin corresponds to the placement of the triangular origin icon on the partial plot preview

Plot offset			Plot scale		
☐ Center the plot			Scale:	Scaled to Fit	
X:	0.00	inches	Custom:	Scaled to Fit 1:1 1:2	drawing units
Y:	0.00	inches	☐ Scale linewe	1:4 1:8 1:10 1:16 1:20 1:30 1:40 1:50	

Plotting (Chapter 5)
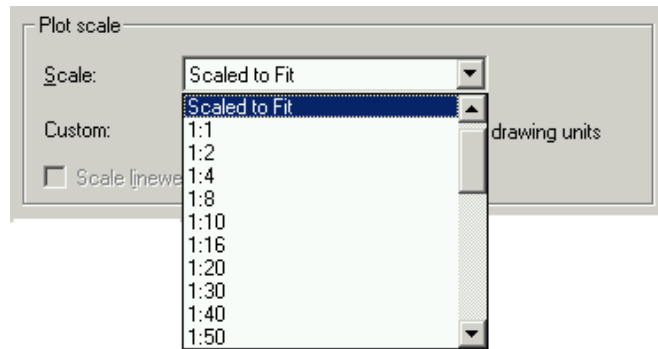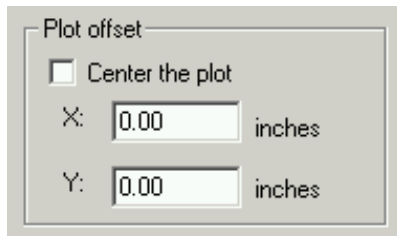
Page Setup

- Allows a group of plot settings to be saved in a file
- Once saved, any desired page setup may be recovered and used without manually repeating the setup process
- Page setups are accessed and stored via the *Plot* dialog box
    - Choose *Add* to store the current plot settings
    - Previously saved setups may be chosen from the drop-down list
    - Only page setups created for a particular drawing will be accessible, though page setups created for other drawings may be *imported*
    - Page setups are particular to either model space or paper space; layouts created in one will not be visible when a plot is begun from the other

```
┌─ Page setup name ──────────────────────────────┐
│  ┌──────────────────────────────┬─┐  ┌────────┐ │
│  │ <Select page setup to apply> │▼│  │ Add... │ │
│  └──────────────────────────────┴─┘  └────────┘ │
└──────────────────────────────────────────────┘
```