SETTING UP AUTOCAD TO WORK WITH ARCHITECTURAL DRAFTING STYLE

You will need to make some changes to AutoCAD to use it as a drafting tool for architectural drawings. AutoCAD "out-of-the-box" is set up primarily for mechanical drafting: drawing small parts for machinery using the metric system because that is the type of drafting is the most practiced in the world. However, making drawings of a building is quite different from making drawings of an automobile part. Not only is the scale of the object drawn very different, but the conventions of drafting are significantly different between architectural and mechanical parts. In order to make the drafting environment correct for architectural drafting conventions, several changes must be made. These changes are relatively simple to accomplish, and involve the following three types of setup:

CREATING A TEMPLATE FILE that has desirable variables set, as well as layers, text styles, dimension styles, layouts and plotting standards created specifically for architectural drafting purposes. Your Template file can then be used to open new drawings to eliminate having to set up these changes again in the future.

CREATING A USER PROFILE that will save the way you want the screen to look and where AutoCAD will search for files. Your personal Profile can be loaded when you start a new AutoCAD session at whatever computer you are working.

TEMPLATE FILES

When you begin a drawing using AutoCAD, you will be drawing in an environment which has pre-set variables setup by Autodesk for what they consider to be the most common use. However, the settings which come with the software "out of the box" are not good for drawings used in the architectural and interior design industry, so they must be changed by you to comply to standard drafting conventions in those fields.

USING TEMPLATE FILES:
To use a template file to begin a new drawing, select "File" from the pull-down menu, then "New." The "Select template" dialogue box will open up as shown in the following illustration:
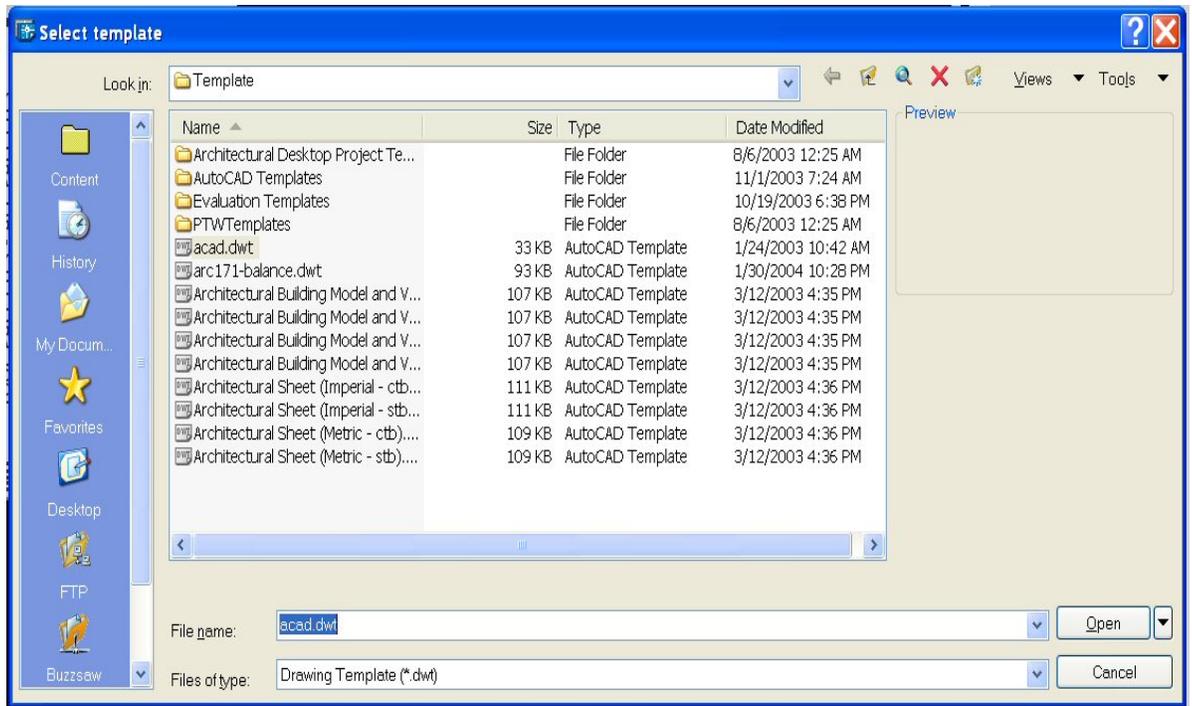
Templates have the filename extension of *.dwt. Select your template file from the list and pick "Open." To continue drawing, save it to any name and folder you want using the "Save" command. The Template file you started with will now be automatically saved as a DWG file.

The easiest way to make your standard settings permanent for later reuse is to create your own Template File. Begin a new drawing, set the drawing variables as you want them to be set, and then save the drawing as a "Template" file for reuse to begin with on future drawings. This new file will have no "entities" in it (lines, arcs, or circles), but it will include the following variables:

Your list of LAYERS, with associated linetypes, colors, and lineweights
Your preferred text styles
Your preferred dimension styles
Your preferred AutoCAD variable settings
Your layout settings
      Viewport(s)
      Name of each viewport
      Scale of each viewport
      Plotting parameters of each viewport, such as what printer to plot to, paper size, plot scale, drawing orientation, and plot style table (pen assignments)
      Title sheet information, such as a title block

To make a new Template file in AutoCAD, follow the steps in the order as listed below:

1. Change the variable Startup to 1 (turn on): type startup<RET>1<RET>

2. Type new<RET>

3. Select the "Start from Scratch" radio button to set up a new drawing.

4. Select Imperial (feet and inches). Click OK.

   Note that this unit type is known in the industry as "Imperial" units. This type of measurement system is used only in the United States, Liberia and Myanmar (Burma). All other countries in the world use the "System International" (or "SI") for drawing units. This is based on the metric system where all dimensions are measured in millimeters. The United States is slowly "inching" toward using the "*Système International d'unités*" (SI system) measurement, but because our industrial base is so entrenched in the Imperial system, it will take many years to accomplish. The U.S. Government has mandated that all drawings made for federal construction projects use both the Imperial and the SI system beginning in 1993. AutoCAD is flexible in that one may begin using Imperial units of measurement, and switch to metric units later by selecting "Decimal" units under the "Units Control" dialogue box and scaling the drawing up by a scale factor of 25.4 (since there are 25.4 millimeters in a inch).

5. Save your drawing. A blank drawing will appear (called "Drawing1.dwg). Immediately save it to your own assigned H: drive by clicking on the "Save" icon on the Toolbar (the one that looks like a floppy disk). Save it to your drive with your initials like this: [your initials]-A This will save it to H:\[your folder] as [your initials]-D.DWG. For instance in my case, I would save it to my H: drive as "FEH-D.DWG." The reason for the -D after your initials is to remind you that this is a template to be used for D-sized drawings (24" x 36")

6. Set the AutoCAD "Units" variable for use in architectural drafting

   Type UNITS ↵
   In the drop-down selection box "Type:" select: Architectural
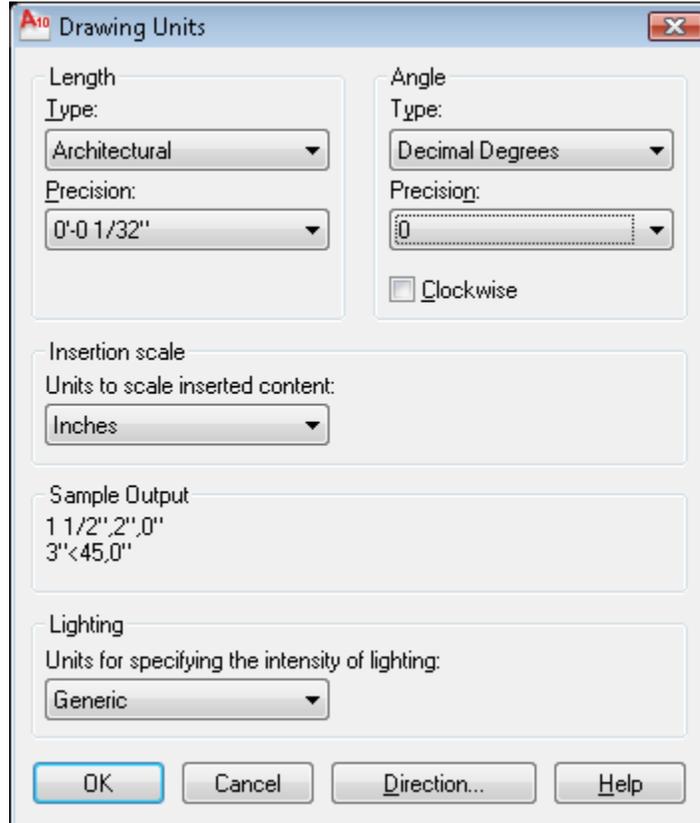   Precision: 1/32"
   Insertion Scale: select drop-down list and pick "Inches"
   Angle Type: Decimal Degrees
   Angle Precision: select drop-down list and pick 0
   Click the OK button

Below is how your Drawing Units dialogue box should look:

7. Set the following AutoCAD System Variables to their proper values

   a. Type VIEWRES <RET> <RET> 20000 <RET> This will make circles and arcs appear smooth. (The default value is 200 which makes circles look like octagons on the screen)

   b. Type UCSICON <RET> OR <RET> This will place the Model Space "User Coordinate System" (UCS) icon at the drawing or UCS "origin" point (0,0,0) in the Model and in each Viewport.

8. Set up the Status Bar:

The Status Bar is at the bottom of the screen on the left side and displays ten pictorial icons representing drawing status settings. If you prefer, you can turn the pictorial icons to words to help remember what they stand for. To do this, right-click one of the buttons and turn off "Use Icons." If you never want to see a particular button in the Status Bar, you can make it invisible by right clicking on any button and select the "Display" command, then uncheck the ones you don't want to see

on the status bar.

Set the Status Bar buttons to the following settings - buttons that have been pushed in are turned on and turned light blue color.

Turn <u>On</u> only the following buttons on the Status Bar (turn off all the rest):

Polar Tracking
Object Snap

9. Set the following Object Snap functions. Right click on the Object Snap button in the Status Bar and select "Settings…"

    a. Endpoint
    b. Midpoint
    c. Center
    d. Node
    e. Quadrant
    f. Intersection
    g. Perpendicular
    h. Nearest

10. Set the following Polar Tracking functions. Right click on the Polar button in the Status Bar and select "Settings…" then select the Polar Tracking tab.

    a. Change Increment angle to 45 degrees and additional angles to 30 degrees
    b. In the "Polar Angle measurement" area, select the radio button "Absolute"

11. Set up Layers.

In the Home Tab of the Ribbon, there is a Layers panel that looks like this:
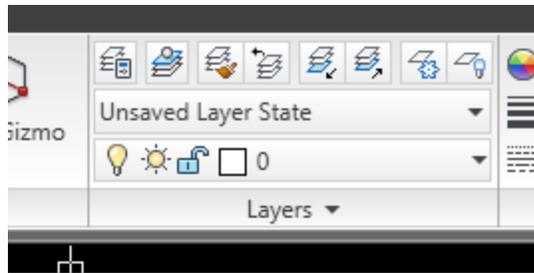
Create the following layers and assign color, linetype and lineweight.
Select the icon for the Layer Properties Manager (on the Home tab in

the Ribbon looks like this) on the toolbar.
Click on the "New Layer" icon to make a new layer.

These layer names are based on *CAD Layer Guidelines, Second Edition*,
Michael K. Schley, ed., published in 1997 by the American Institute of
Architects in Washington, D.C. I have assigned colors, linetypes, and
lineweights.

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
A-ANNO-DIMS	Dimensions	Magenta	Continuous	.1500 mm	
A-ANNO-NOTE	Notes	Cyan	Continuous	.3500 mm	
A-ANNO-SYMB	Symbols, Keys and Targets (Such as Door Numbers, Window Numbers, Furniture Numbers, Equipment Numbers, Toilet Accessory Numbers)	Cyan	Continuous	.3500 mm	
A-ANNO-VPRT	Viewports	Cyan	Continuous	.1000 mm	Non-Printing
A-AREA	Area Calculations, Hatch Patterns, and Numbers	60	Continuous	.1000 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
A-CEIL	Ceiling Information (Soffits, Holes, Skylights, Ceiling Grid)	10	Continuous	.1000 mm	
A-DOOR	Doors and Door Frames in Plan or Elevation (But Not Swing Arcs or Elevation Swing Indications of Doors)	Blue	Continuous	.2500 mm	
A-DOOR-GLAZ	Windows in Doors	Green	Continuous	.2500 mm	
A-DOOR-SWNG	Swing Arcs or Elevation Swing Indications of Doors	30	Hidden2	.1000 mm	
A-ELEV	Elevation Drawings, Interior or Exterior	blue	Continuous	.2000 mm	
A-ELEV-PATT	Hatch Patterns for Elevations	30	Continuous	.0500 mm	
A-ELEV-OTLN	Elevation Outline	yellow	Continuous	.5000 mm	
A-FLOR	Any Information Relating to Floors Which Would Not Be Shown on Another Layer, Such as Holes in Floors, Edges of Slabs, Guardrails Around Floor Openings, or Curbs	red	Continuous	.2500 mm	
A-FLOR-APPL	Appliances: Refrigerators, Ovens, Stoves, and	30	Continuous	.2000 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
	Any Fixed Element Which Requires a Plumbing or Electrical Connection to Make Operate				
A-FLOR-GARD	Guardrails	green	Continuous	.2000 mm	
A-FLOR-STRS	Stairs and Ladders in Plan or Elevation	red	Continuous	.2500 mm	
A-FLOR-STRS-RAIL	Stair Handrails in Plan or Elevation	green	Continuous	.2000 mm	
A-FLOR-STRS-AROW	Direction Arrows and Break Lines for Stairs	cyan	Continuous	.1500 mm	
A-FLOR-STRS-RISE	Risers of Stairs Should Be on a Separate Layer So That They Can Be Turned off on a Reflected Ceiling Plan	green	Continuous	.1500 mm	
A-FLOR-TPTN	Toilet Partitions, Doors, Door Swings	green, but door swing should be color 30	Continuous	.2500 mm	
A-FLOR-ELEV	Elevators Cars in Plan	green	Continuous	.2500 mm	
A-FLOR-RAMP	Ramps in Plan or Elevation	green	Continuous	.2500 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
A-FLOR-RAMP-RAIL	Ramp Handrails in Plan or Elevation	green	Continuous	.2000 mm	
A-FLOR-RAMP-AROW	Direction Arrows and Break Lines For Ramps	cyan	Continuous	.1500 mm	
A-FLOR-ACCS	Toilet Accessories	green	Continuous	.2000 mm	
A-FURN	Furniture	magenta	Continuous	.2000 mm	
A-FURN-PLNT	Interior Plants	60	Continuous	.2000 mm	
A-FURN-PATT	Furniture Finish Patterns	30	Continuous	.0500 mm	
A-GLAZ	Windows	blue	Continuous	.2500 mm	
A-ROOF	Roof, Gutters, Skylights	red	Continuous	.2500 mm	
A-ROOF-PATT	Hatch Patterns for Roofs	30	Continuous	.0500 mm	
A-SECT	Sections	red	Continuous	.3500 mm	
A-SECT-HIDD	Material Hidden in Section	30	hidden2	.1500 mm	
A-SECT-PATT	Hatch Patterns of Section	30	Continuous	.0500 mm	
A-WALL	Walls	yellow	Continuous	.7000 mm	
A-WALL-PATT	Wall Hatch Pattern (Poché)	30	Continuous	.0000 mm	
A-WALL-HEAD	Door and Window Headers	30	Continuous	.2500 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
A-WALL-HIDD	Walls behind other objects	30	Hidden 2	.1500 mm	
A-WALL-SILL	Door and window sills	30	Continuous	.2500 mm	
C-PKNG-CARS	Cars	green	Continuous	.1500 mm	
C-PKNG-STRP	Parking Lot Striping	30	Continuous	.0500 mm	
C-PROP	Property Lines	Magenta	Phantom2	.3500 mm	
C-COMM	Site communication poles and any other object above ground	140	Communications Linetype ----- C ----- C -----	.3500 mm	
C-WATR	Site Water Meter and any object above ground	140	Water Linetype ---- W ---- W -----	.3500 mm	
C-NGAS	Site Gas Meter and any object above ground	140	Gas Linetype ----- G ----- G -----	.3500 mm	
C-STRM	Site Storm Sewer Manholes	140	Storm Water Linetype ----- ST ----- ST -	.3500 mm	
C-FIRE	Site Fire Hydrants, pumps, and any object above ground	140	Fire Water Linetype ----- F ----- F -----	.3500 mm	
C-SSWR	Sanitary Sewer Manholes	140	Sanitary Sewer Linetype ----- S ----- S -----	.3500 mm	
C-ROAD-CURB	Curbs	10	Continuous	.2000 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
C-WALK	Sidewalks	10	Continuous	.2000 mm	
C-TOPO	Proposed Contours	35	Continuous	.2500 mm	
C-TOPO-DEMO	Contours, Existing to Be Changed	60	Hidden	.1500 mm	
C-TOPO-EXST	Contours, Existing to Remain	35	Continuous	.1500 mm	Screen 60%
E-COMM	Telephone Receptacles	red	Continuous	.3500 mm	
E-CTRL	Control Systems	red	Continuous	.1500 mm	
E-LITE	Lighting	30	Continuous	.2500 mm	
E-LITE-EMER	Emergency Lighting	30	Continuous	.2500 mm	
E-LITE-SWIT	Light Switch	red	Continuous	.2500 mm	
E-POWR	Electrical Power, Receptacles	red	Continuous	.2500 mm	
E-POWR-PANL	Power Panels	red	Continuous	.2500 mm	
L-PLNT	Landscape Materials	40	Continuous	.2500 mm	
L-PLNT-TREE-EXST	Trees - Existing to Remain	40	Continuous	.1000 mm	
L-PLNT-TREE-	Trees - to be removed	60	Continuous	.1000 mm	

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
DEMO					
L-PLNT-TREE-NEWW	Trees - New	40	Continuous	.3000 mm	
L-SITE-FNCE	Fences	yellow	Continuous	.1500 mm	
M-HVAC	HVAC System, Ductwork, Equipment	blue	Continuous	.2500 mm	
M-HVAC-CDFF	HVAC Ceiling Diffusers	30	Continuous	.2500 mm	
M-HVAC-ODFF	HVAC Wall and Other Diffusers	30	Continuous	.2500 mm	
P-FIXT	Plumbing Fixtures	Green	Continuous	.2000 mm	
S-BEAM	Structural Beams (Any Material)	Red	Continuous	.2500 mm	
S-BEAM-HIDD	Structural Beams Hidden Elements	30	Hidden2	.1000 mm	
S-COLS	Structural Columns (Any Material)	Yellow	Continuous	.3500 mm	
S-COLS-HIDD	Structural Columns Hidden Elements	30	Hidden2	.1000 mm	
S-JOIS	Structural Joists (Any Material)	Magenta	Continuous	.3500 mm	
S-JOIS-	Structural Joists	30	Hidden2	.1000 mm	Screen

Layer	Objects to Be Drawn on this Layer	Color	Linetype	Lineweight	Remarks
HIDD	Hidden Elements				50%
S-FNDN	Structural Foundation	Yellow	Continuous	.4000 mm	
S-FNDN-HIDD	Structural Foundation Hidden Elements	30	Hidden2	.1000 mm	
S-METL	Miscellaneous Metal Shapes & Plates	Magenta	Continuous	.3500 mm	
S-METL-HIDD	Miscellaneous Metal, Hidden Lines	30	Hidden2	.1000 mm	
S-DECK	Metal Deck	Magenta	Continuous	.2000 mm	
S-GRID	Structural Column Grid	20	Center2	.2500 mm	

Assign linetypes to layers:

Most layers will have the "Continuous" linetype (i.e. a solid, non-dashed line) assigned to them. However, you may note in the chart of standard layers that there are several layers which will need to have special linetypes assigned to them. To change the linetype assigned to a layer, click on the layer name to highlight it. Then click on the word "Continuous" at the right end of that line. This will bring up a list of linetypes. If the name of the linetype you want to assign to the layer does not appear on the list, click on the "Show Linetypes in File" button and select it from the list. Note that the new name of the linetype will now be shown in the list at the right end of the layer name.

Assign Lineweights to each layer.

"Default" lineweight is 0.35mm. If you did not change the lineweight associated with a layer, that is what they would all be set to.

It seems to us a quaint fact that in AutoCAD, lineweights are measured by default in millimeters. For example, new layers created in the Layer Property manager have as their default lineweight 0.35mm. This derives from the traditional manual drafting technique of using technical pens for ink drafting on vellum such as

manufactured by the Rotring pen company in Hamburg, Germany (http://www.rotring.com/en/index.html) since the 1928. Because the design of these technical pens originated in Germany, they followed a European drafting tradition and system of dimensional measurement. A full set of technical pens had the following point sizes: 0.13, 0.18, 0.25, 0.35, 0.5, 0.7, 1.0, 1.5, and 2.0 mm. The International Organization for Standardization (ISO) called for four pen widths and set a color code for each: 0.25 (white), 0.35 (yellow), 0.5 (brown), 0.7 (blue). These point sizes produced lines that related proportionally to various text character heights and the ISO paper sizes. All ISO paper sizes have the same aspect ratio, 1 to √2. So any drawing made on one size of paper may be enlarged or reduced to any other size and will fit perfectly. The "zoom ratio" between any two adjacent size designations (e.g. A4 to A3) is also always 1 to √2. Given this ease of changing sizes, it is of course common to copy or print a given document on different sizes of paper, Text made with technical pen of 5 mm in height has a stroke or line thickness of 0.5 mm, and so requires a brown-nibbed 0.5 mm technical pen. If this text were drawn on an ISO-sized document (e.g. A0), and the document were reproduced at half its original size (A1), the text would be rendered 3.5 mm high with a stroke thickness of 0.35 mm - the yellow-nib size. Thus, changes to reductions or enlargements can be made easily as everything is in proportion. This standard ensures that drawings can always be legible even after microfilming, photocopying, and faxing.

Below is the European standard paper sizes and their relationship:
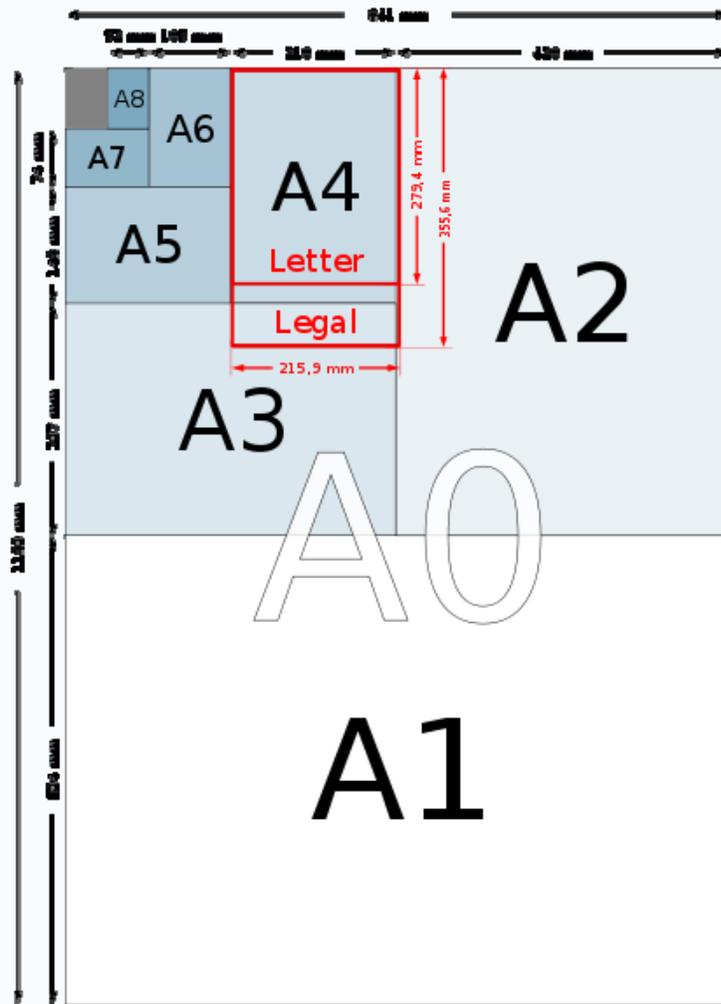
A technical pen that is still widely-used in America is the Rapidograph pen manufactured by the Koh-i-Noor Hardtmuth company located in České Budějovice in the Czech Republic. It has made fine drafting supplies since 1790. Their pen sizes have a number associated with a metric line width as follows:

Koh-i-Noor Rapidograph pen sizes

7 = 2.00mm
6 = 1.40mm
4 = 1.20mm
3 ½ = 1.00mm
3 = .80mm
2 ½ = .70mm
2 = .60mm
1 = .50mm

0 = .35mm
00 = .30mm
000 (3x0) = .25mm
0000 (4x0) = .18mm
000000 (6x0) = .13mm

When plotters were first manufactured for use with computers to make CAD ink plots, they made use of Rapidograph numbered pens and millimeter widths in fitted into an 8-pen carousel.

Set the current layer:

Highlight the layer on the list you want to make current and right click - pick "Make Current." You should make A-Wall layer current for the template, because that is the layer you would want to use first.

12. Create the following Text Styles

<u>Annotative Text Style:</u>

Click on the Annotate Tab in the Ribbon
Open the Text Style dialogue box by clicking the clicking on the arrow that points diagonally down and to the right on the "Text" panel
Select the "Standard" style
Font Name: Stylus BT
Check the box to the left of "Annotative"
Check the box to the left of "Match text orientation to layout
Set "Paper Text Height" to 3/32"
Click the "Apply" button
Click the "Close" button
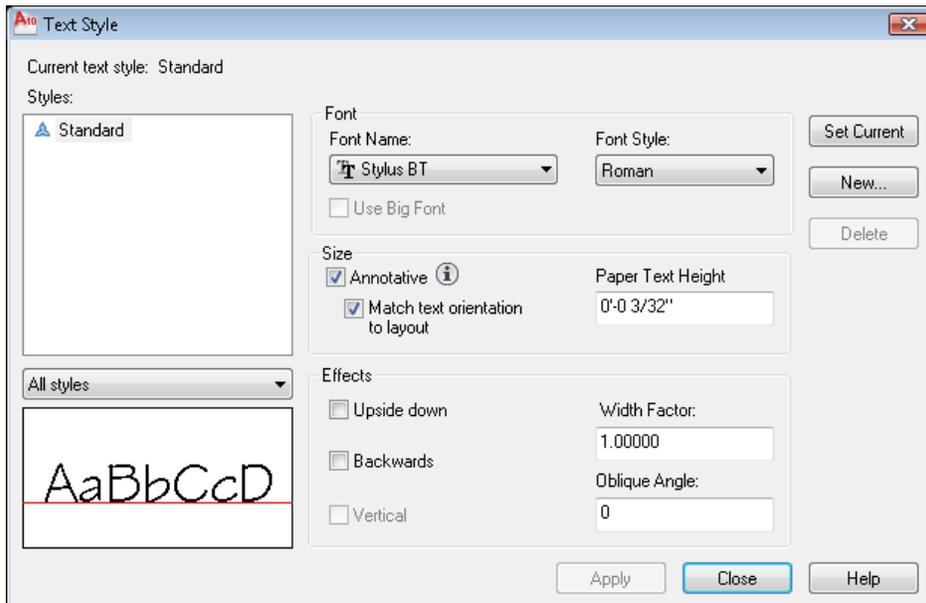
<u>Text style to assign to dimensions.</u>

This text style will not need to be "Annotative," but it will be compressed horizontally in order to facilitate placing dimension numbers between extension lines. It will also give dimension text a distinctive look so they will not be confused with notes.

Open the Text Style dialogue box by clicking the clicking on the arrow that points diagonally down and to the right on the "Text" panel

Select the "New…" button
Type the name of the new style: DIMENSIONS and Click the OK button
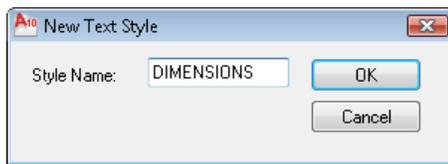
Font Name: Stylus BT
Leave the box to the left of "Annotative" unchecked
Set "Height" to 0
Set "Width Factor" to 0.75
Click the "Apply" button
Click the "Close" button

Your dialogue box should; look like this:
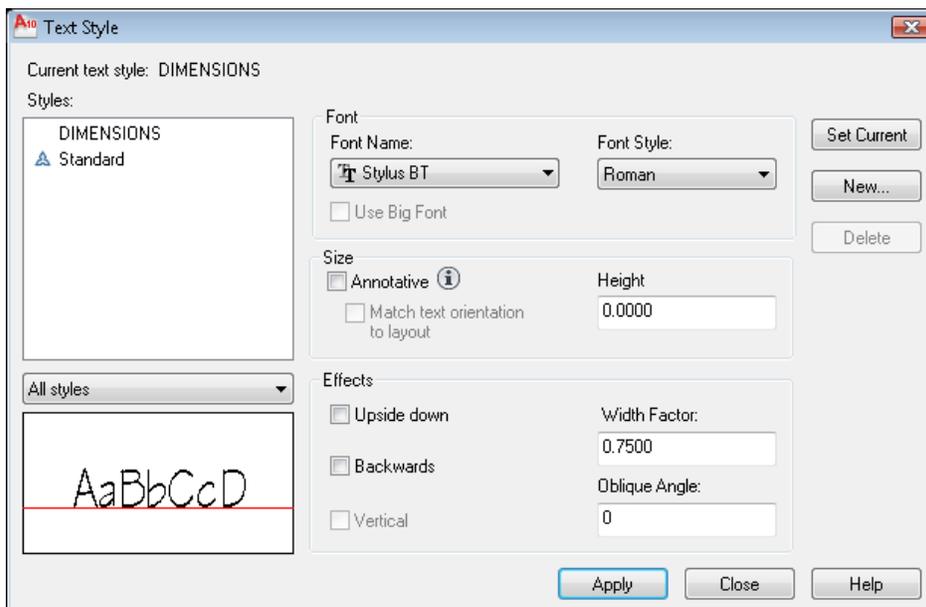
A variable height text style to use with the drawing titles in paper space

Open the Text Style dialogue box by clicking the clicking on the arrow that points diagonally down and to the right on the "Text" panel
Select the "New…" button
Type the name of the new style: TITLE and Click the OK button
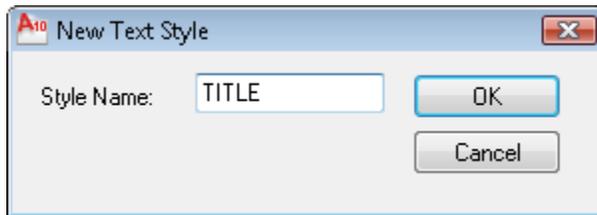
Font Name: Stylus BT
Leave the box to the left of "Annotative" unchecked
Set "Height" to 0
Set "Width Factor" to 1
Click the "Apply" button
Click the "Close" button

Your dialogue box should look like this:
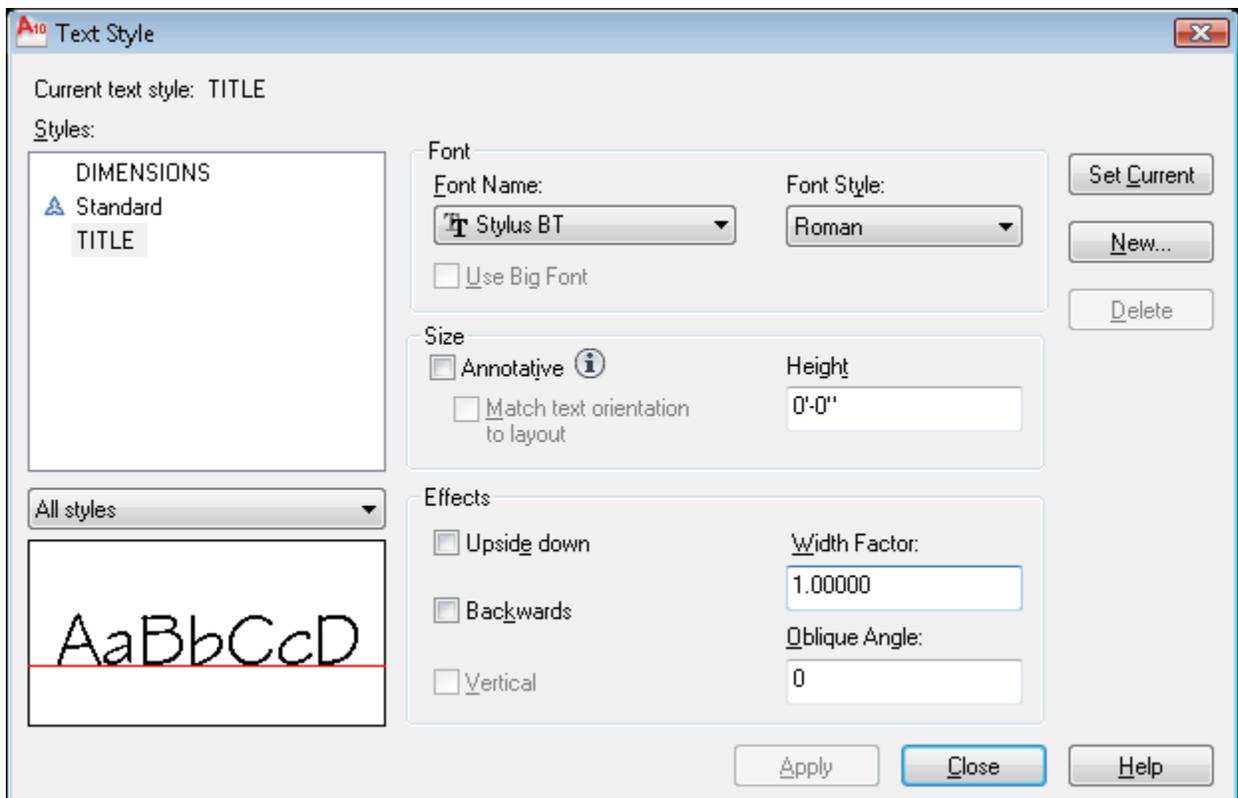
13. Create the following Dimension Styles

Click on the Annotate Tab in the Ribbon
Open the Text Style dialogue box by clicking the clicking on the arrow that points diagonally down and to the right on the "Dimensions" panel
Select the Standard style

Click the "Modify…" button to make the following changes to the Standard style:
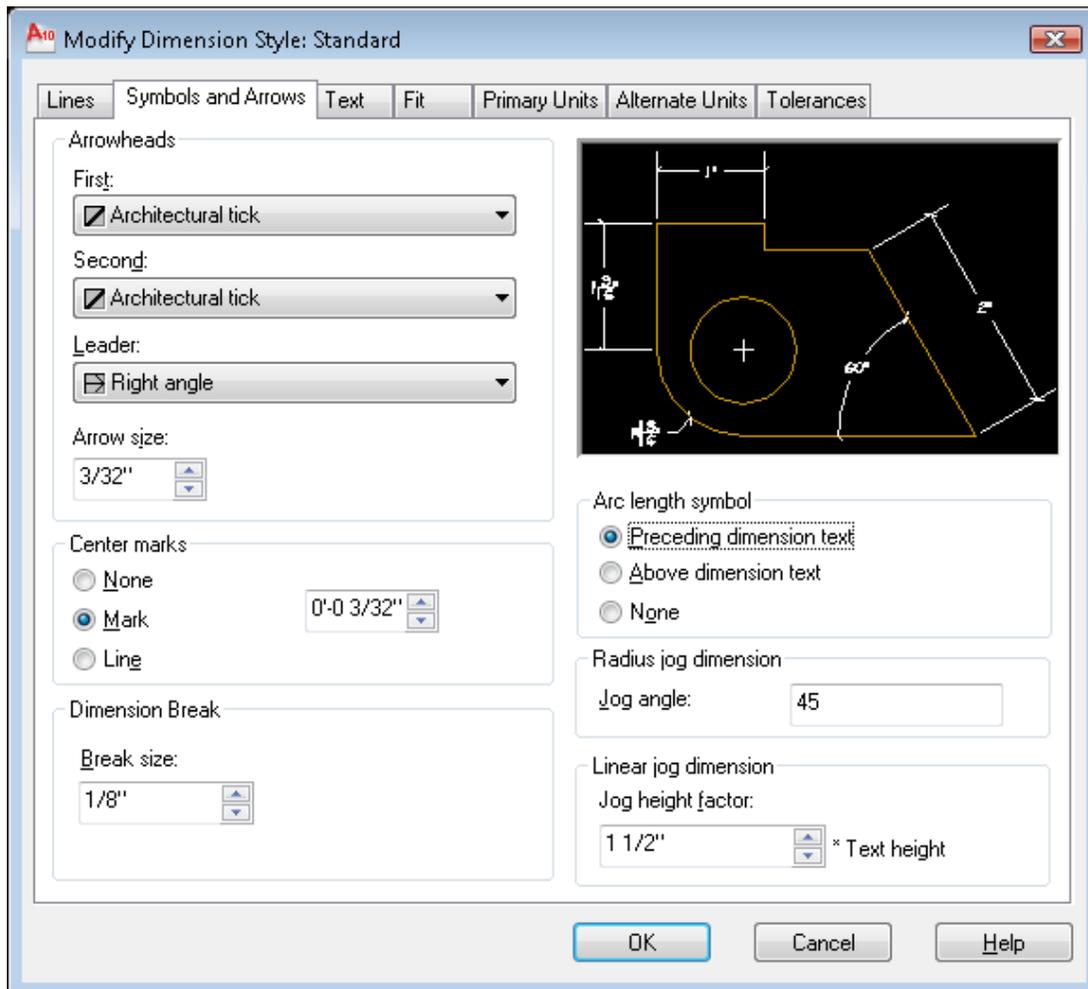
Select the "Symbols and Arrows" tab
Arrowheads First: select drop-down list and pick Architectural tick
Arrowheads Second: select drop-down list and pick Architectural tick
Arrowheads Leader: select drop-down list and pick Right angle
Arrow size: 3/32"

Your dialogue box should look like this:

Select the "Lines" tab
Color: Bylayer
Linetype: ByLayer
Lineweight: ByLayer
Extend beyond ticks: 1/16"
Baseline spacing: 0
Extension lines Color: ByLayer
Linetype ext line 1: ByLayer
Linetype ext line 2: ByLayer
Extension lines Linetype: ByLayer
Extend beyond dim lines: 1/16"
Offset from origin: 1/16"

Your dialogue box should look like this:
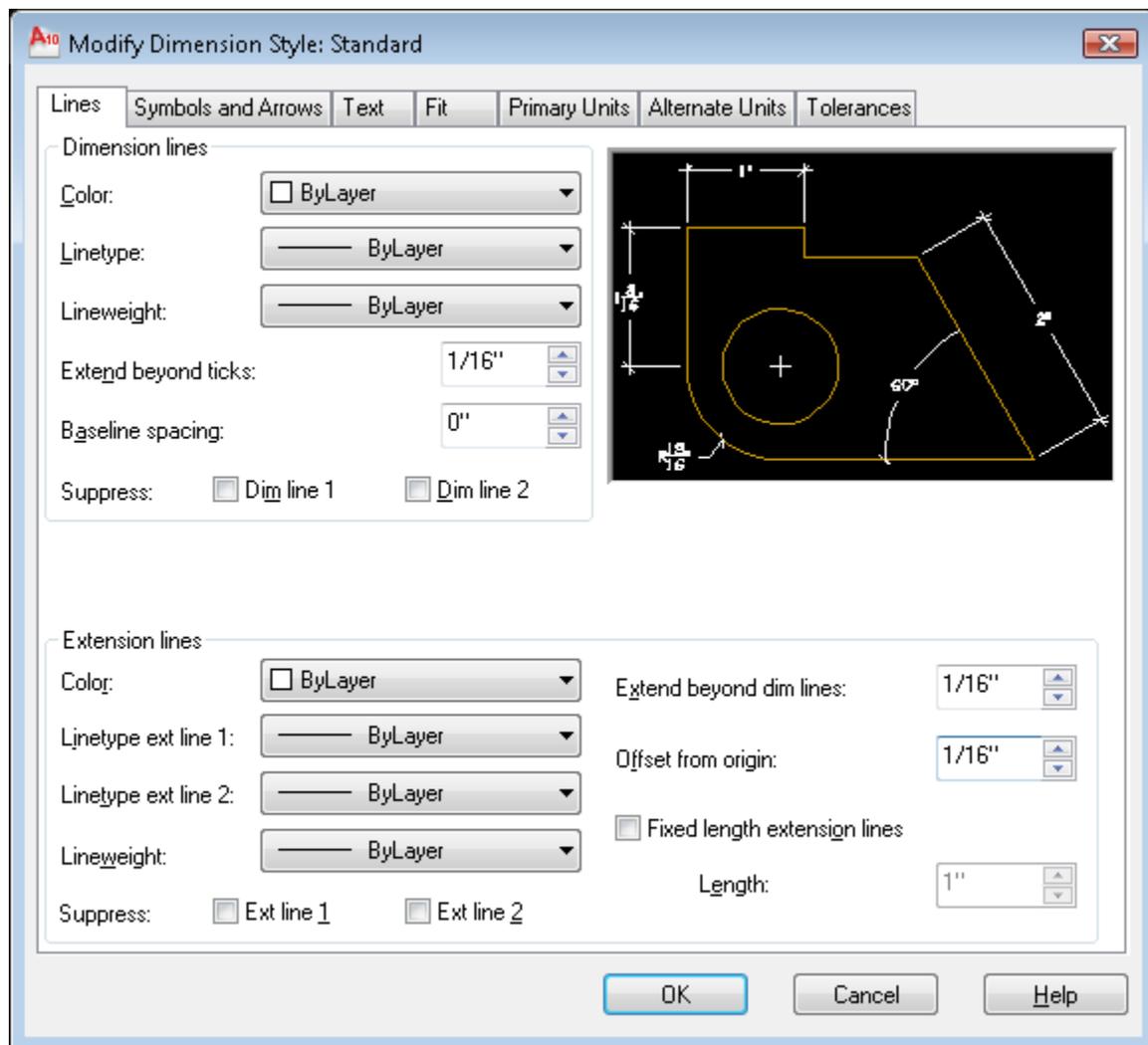
Select the "Text" tab
Text style: Dimensions (you made this text style earlier in this exercise)
Text color: ByLayer
Text height: 3/32"
Fraction height scale: 0.75 (this is the height proportion of numerator and denominator in stacked fractions)
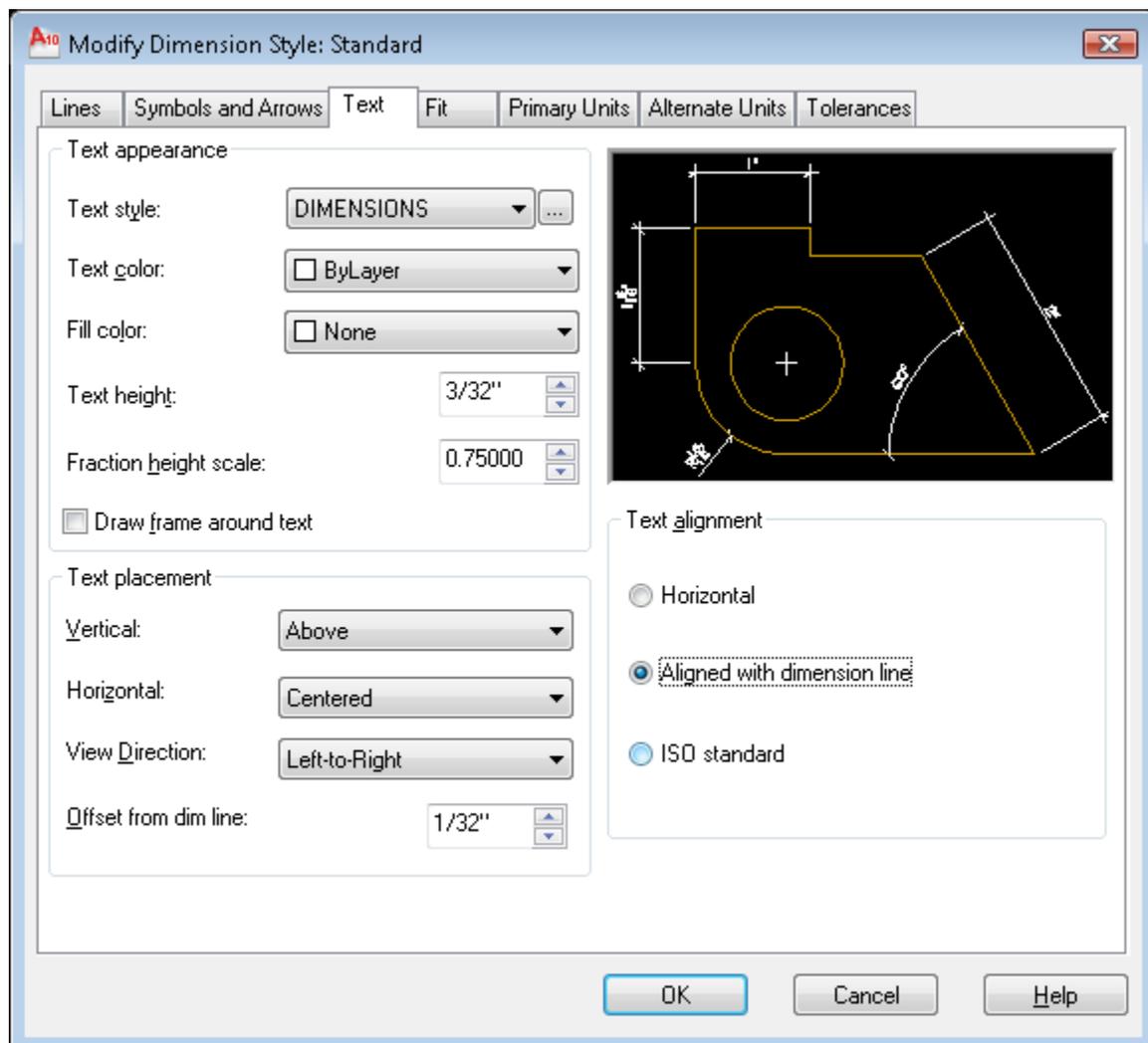Text placement Vertical: select drop-down list and pick "Above"
Text placement Horizontal: select drop-down list and pick "Centered"
Offset from dim line: 1/32"
Text alignment: select the radio button "Aligned with dimension line"

Your dialogue box should look like this:
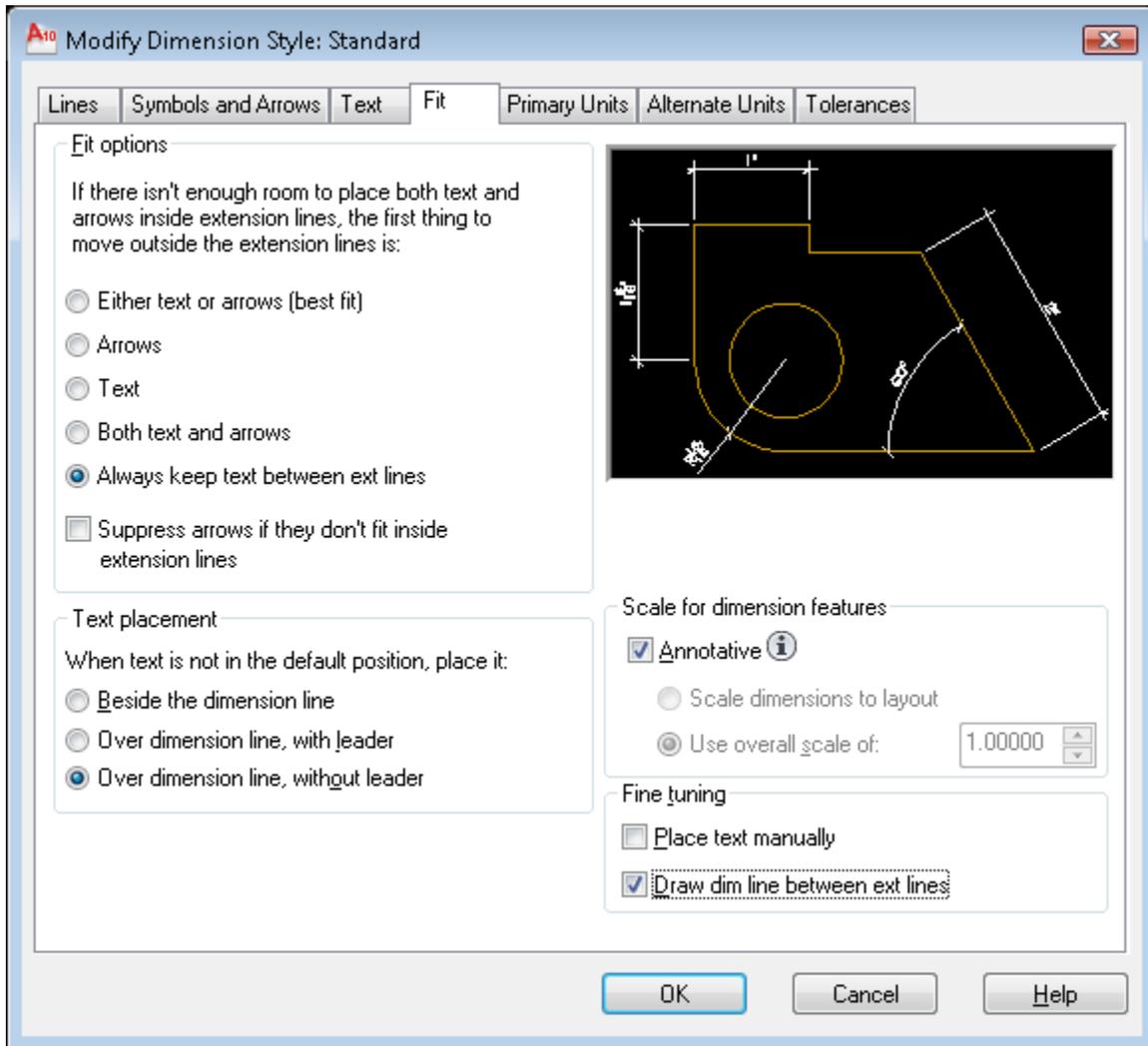
Select the "Fit" tab
Fit options: "Always keep text between ext lines"
Text placement: "Over dimension line, without leader"
Scale for dimension features: check the box to the left of "Annotative"
Fine tuning: check the box to the left of "Draw dim line between ext lines"

Your dialogue box should look like this:

Select the "Primary Units" tab
Unit format: select drop-down list and pick "Architectural"
Precision: select drop-down list and pick 0'-0 1/16"
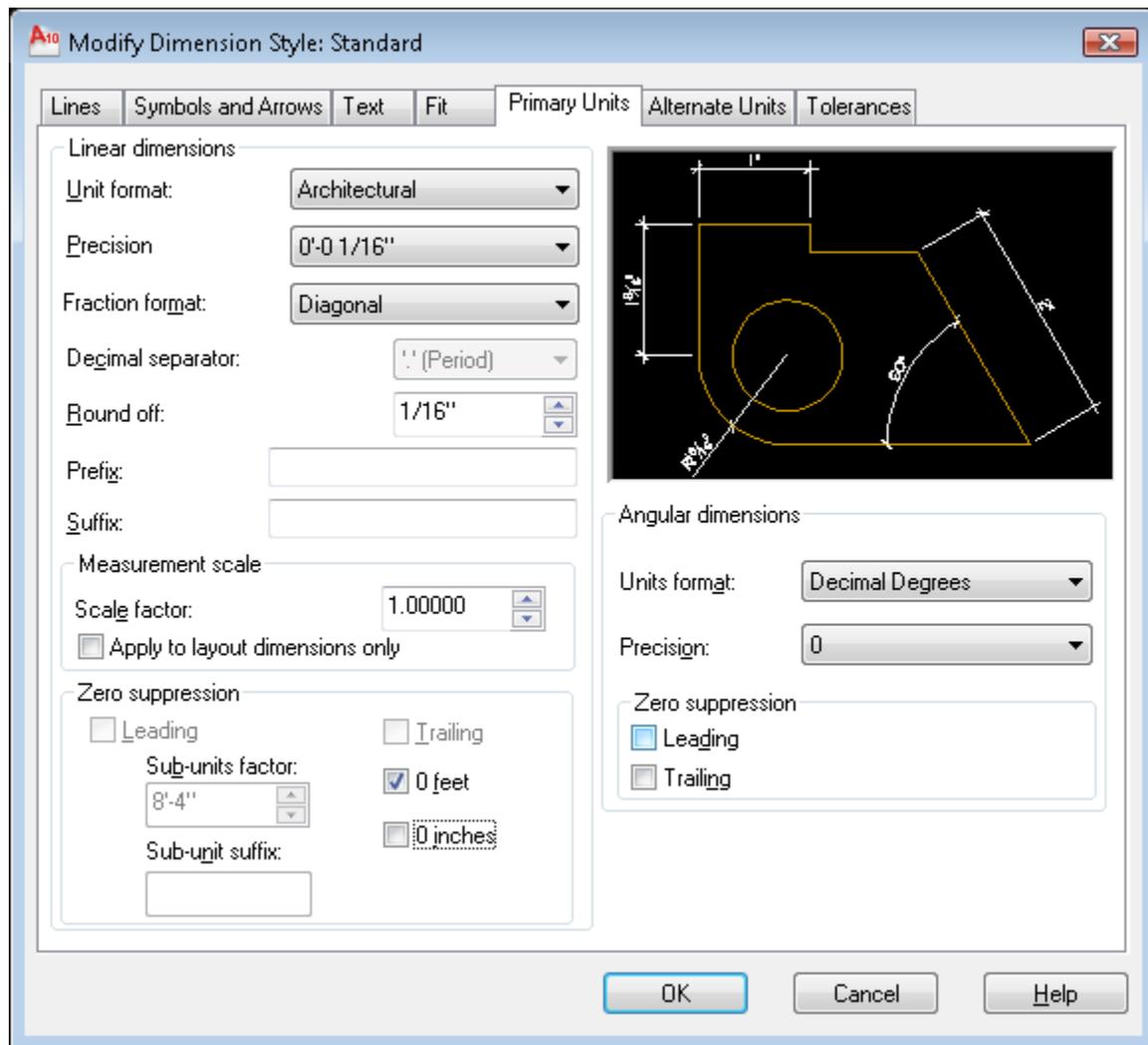Fraction format: select drop-down list and pick "Diagonal"
Round off: 1/16"
Zero suppression: check the box to the left of 0 feet
Angular dimensions: select drop-down list and pick "Decimal Degrees"
Angular dimensions Precision: select drop-down list and pick 0

Your dialogue box should look like this:

You do not need to change anything in the "Alternate Units" or "Tolerances" tabs
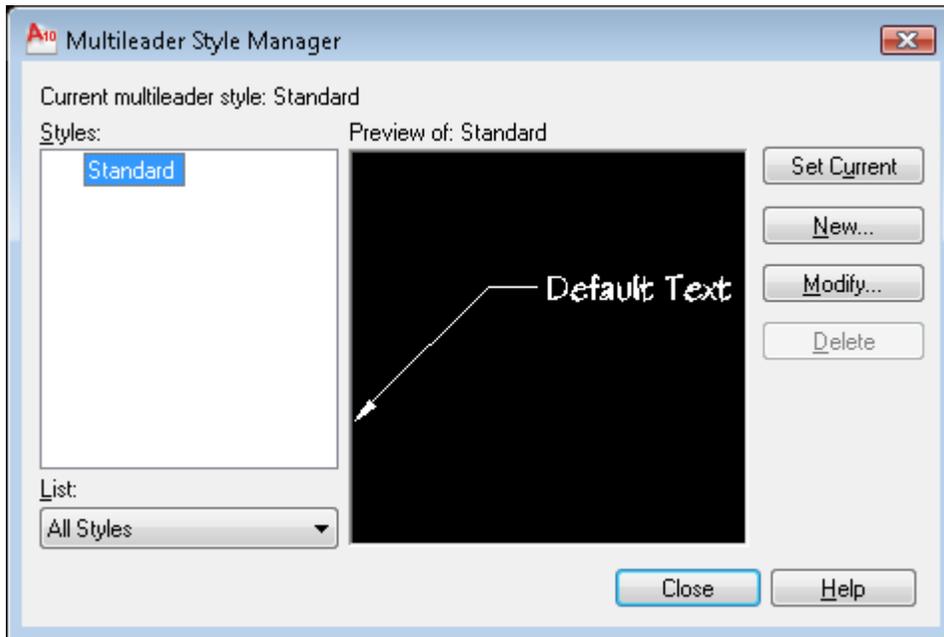
Click "OK" button
Click "Set Current" button
Click "Close" button

Click on the Annotate Tab in the Ribbon
Open the Multileader Style Manager by clicking on the arrow that points
diagonally down and to the right on the "Leaders" panel

In the Multileader Style Manager dialogue box, select the Standard style:

Click the "Modify…" button

Under the Leader Format tab, change the following:

Type: Straight
Color: By layer
Linetype: By layer
Lineweight: By layer
Arrowhead Symbol: Right Angle
Arrowhead Size: 3/32"
Leader Break Size: 1/8" (a Leader Break is a gap created in the leader line where
leader lines cross model geometry)

Your dialogue box should look like this:

Modify Multileader Style: Standard

Leader Format | Leader Structure | Content

General

Type: Straight

Color: ByLayer

Linetype: ByLayer

Lineweight: ByLayer

Arrowhead

Symbol: Right angle

Size: 3/32"

Leader break

Break size: 1/8"

Default Text

Learn about Multileader Styles

OK | Cancel | Help
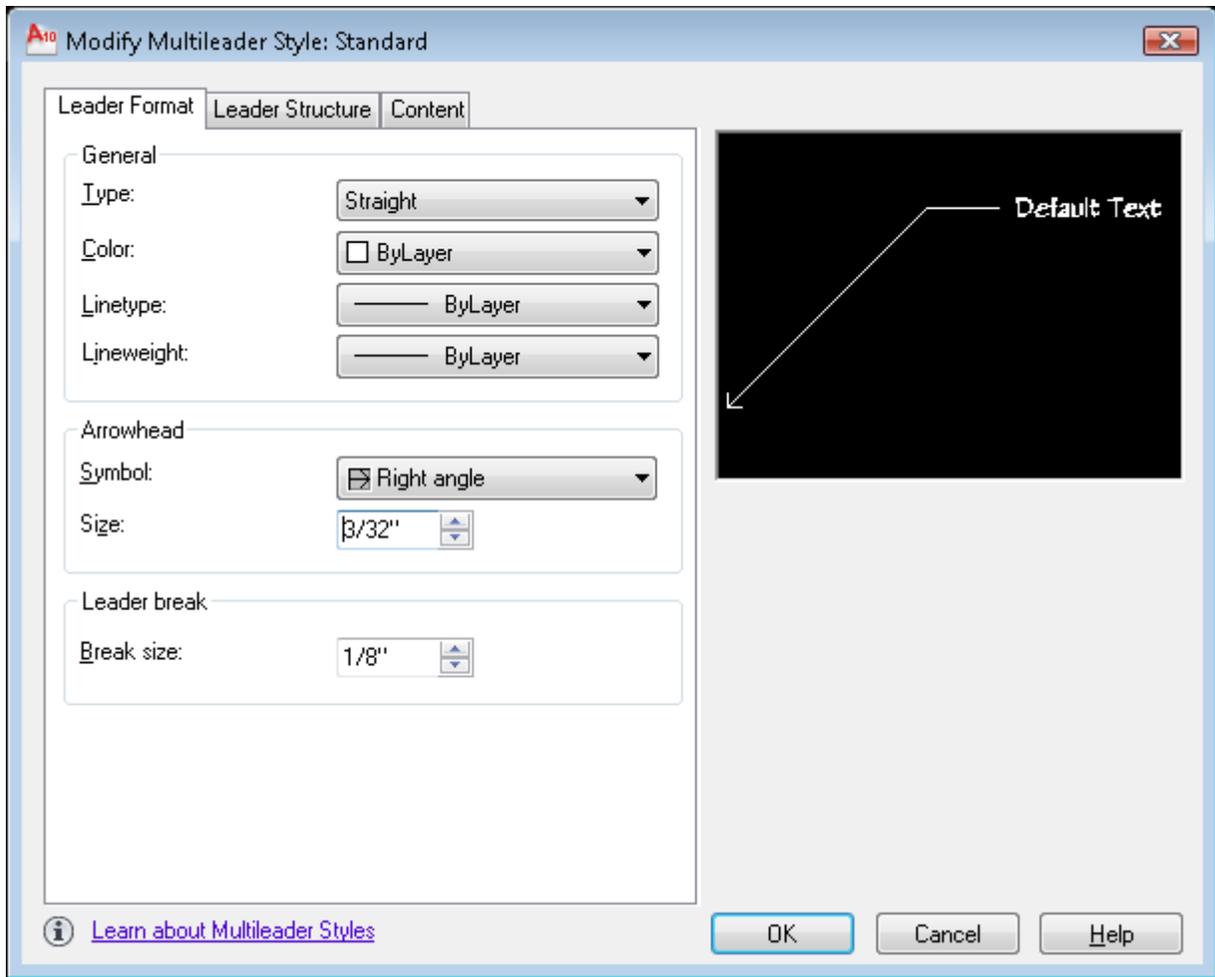
Click on the "Leader Structure" tab
Check the box to the left of Maximum leader points and set to 2
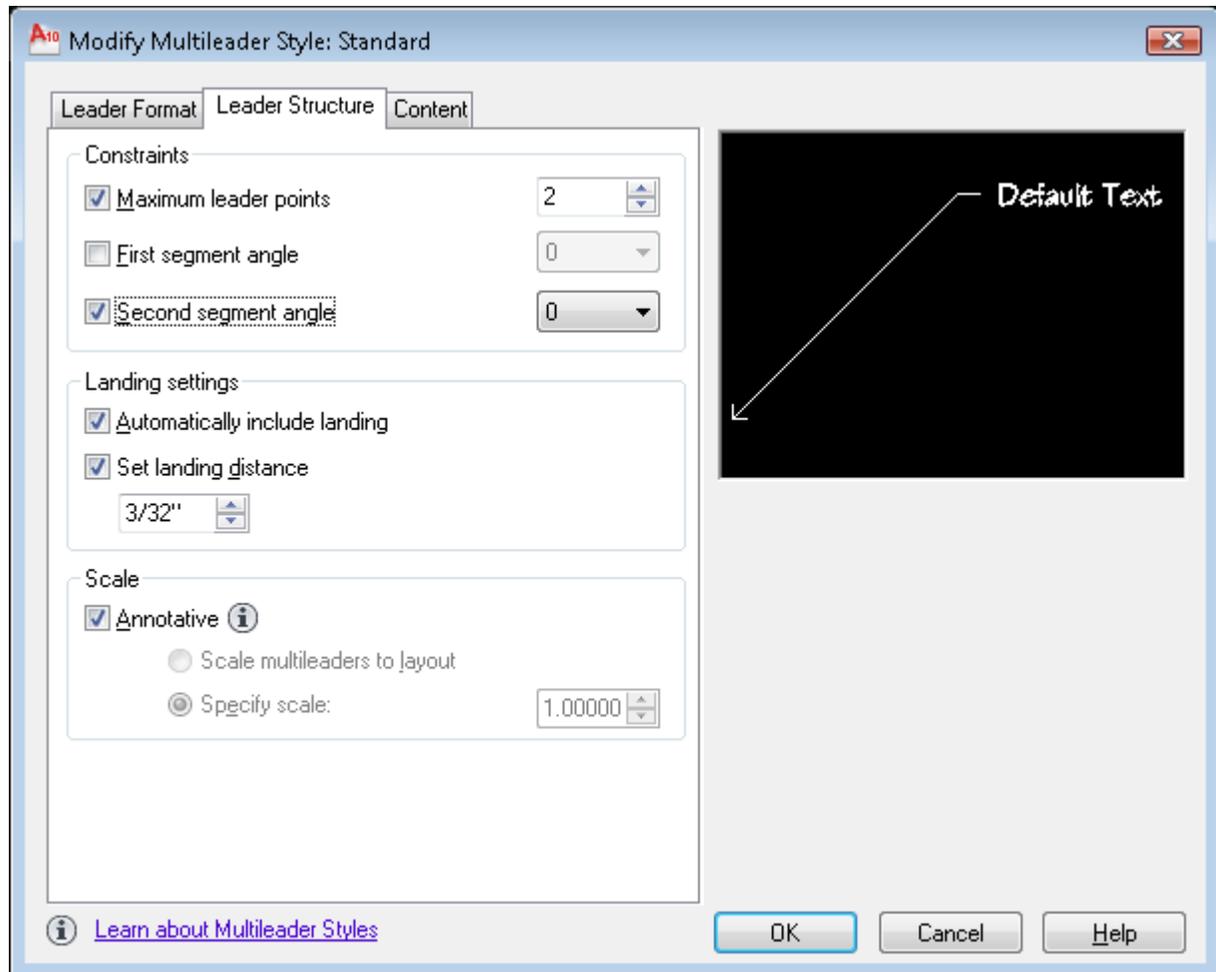Uncheck the box to the left of "First segment angle"
Check the box to the left of "Second segment angle" and set to 0
Check the box to the left of "Automatically include landing"
Check the box to the left of "Set landing distance" and set it to 3/32""
Check the box to the left of "Annotative"

Your dialogue box should look like this:

Click the Content tab
Multileader type = Mtext
Text style: Standard
Text angle: Keep horizontal
Text color: ByLayer
Check the box to the left of "Always left justify"
Pick the radio button "Horizontal attachment"
Left attachment: Middle of top line
Right attachment: Middle of bottom line
Landing gap: 3/32"
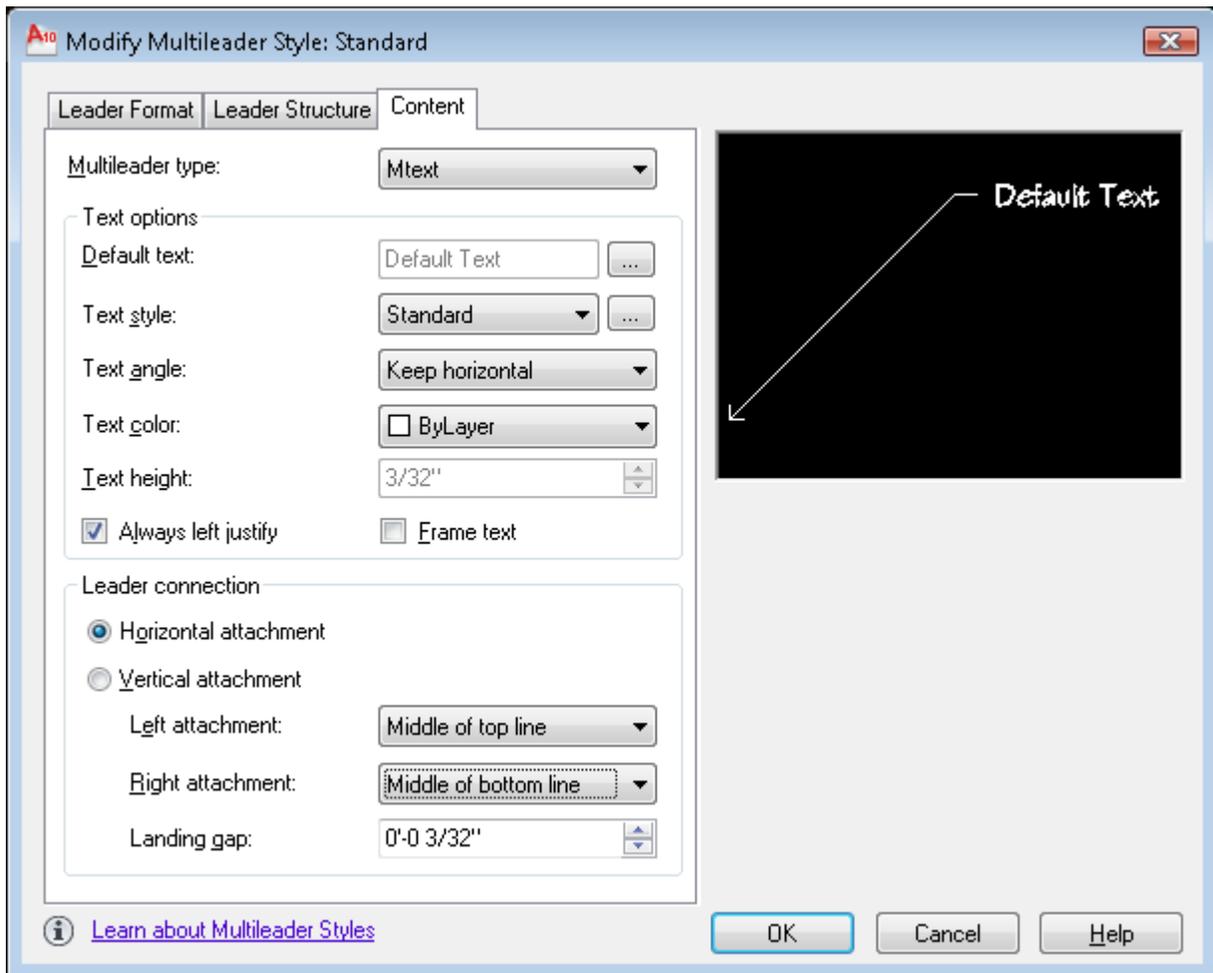
Click "OK" button
Click "Set Current" button
Click "Close" button

Each Layout that you create will be carry with it the following elements:

> A unique name of each Layout as displayed on a Tab at the bottom of the drawing

> One or more Viewports including a scale for each viewport, locked or unlocked status for each viewport, and plotting parameters for each viewport, such as what printer to plot to, paper size, plot scale, drawing orientation, and plot style table (pen assignments)

> Drawing entities you want to have visible and printable in the Layout, such as drawing title information and a border

Click on the "Layout1" tab at the bottom of the drawing. This will bring you into "Paper Space" and allow you to set up viewports and draw a border. By default you always are drawing in so-called "Model Space," which is where you should be to draw the 3-dimensional "model" of the building. However, the border of the sheet should be drawn in so-called "Paper Space." "Paper Space" is a special 2D drawing mode in which you can group various "views" of your 3D "model" of the building for plotting purposes. These views are typically plan, elevation, section, and perspective and are created in "viewports" drawn in the Paper Space Layout. Since the model of the building is the only drawing, when the model is changed, the plans, elevations, and sections are automatically changed in their viewports. The great advantage of working this way is that you can visualize the building or space as it will actually exist in three dimensions. This tool will virtually eliminate the old difficulty of "turning the corner" in visualizing the building design. Delete the viewport entity that AutoCAD creates by default - you may have to zoom to extents to see it

Set the current layer to A-ANNO-VPRT

Create a new paper space viewport. Type MV (alias for the command MVIEW which means "Make Viewport"). Type 0,0<RET>35,23<RET> This will establish a viewport size as well as a maximum size which can be plotted on D-sized paper with the HP Design Jet 500 plotter located in the studio. The viewport size can readily be changed later by clicking on the viewport and moving the grips.

Double click inside the viewport and set the scale to 1/4" = 1'-0 (the list of scales appears at the bottom of the screen. Then lock the viewport scale by clicking on the gold lock at the bottom of the screen.

Right-click on the Layout1 tab at the bottom of the drawing. Select "Page Setup..." from the menu that will appear. This will bring up the "Page Setup"

dialogue box. Select the "Plot Device" tab first. After the word Name: there is a drop-down list. Pick the HPDJ500 plotter from the list.

Select from the "Plot style table (pen assignments)" drop down list the plot style "monochrome.ctb."

Click on the "Layout Settings" tab.

Select "letter" paper size and "landscape" drawing orientation. Under "Plot Area" select "Extents." Under "Plot Scale" select 1:1.

Right-click on the Layout1 tab at the bottom of the drawing again. Select "Rename" from the menu. Type "A101" in the Rename Layout dialogue box and pick OK.

Return to "Model Space: Click on the Model tab at the bottom of the drawing. Note that the word PAPER now reads MODEL at the bottom of the drawing.

16. Set the current layer to A-WALL.

17. Change Startup to start with a template file: Type startup <RET> 0 <RET>

18. Save the file you have just created as a "Template" file, which can be used as a start for your other drawings. To do this, pick "File" from the pull-down menu, then "Save As". When the "Save Drawing As" dialogue box appears on the screen , click on the drop-down box labeled "Files of Type." Pick the "AutoCAD Drawing Template (*.dwt)" choice. You will need to change the drive to which it will be saved to your H: drive by clicking in the "Save in" drop down box at the top, then pick the "Save" button. You should create a "Templates" folder on your H: drive in which to keep your AutoCAD template files. Next, a "Template Description" dialogue box will pop up. In this box, you can type in a description for your Template file for future reference. You could put in the fact that it is to be used for A-sized drawings, the date it was created, and the course it was made for, as an example. Then click on the "OK" button.

19. Copy the template file to your flash drive for safe keeping.

This Template file may be edited at any time to add other items or change variables which you want to use in your future drawings. To open the template file for editing, in AutoCAD, either type new<RET> or select the icon in the tool bar that looks like a blank sheet of paper (the QNew command). A list of template files which are stored in the default Template File folder location will appear, but your template file will not be on that list. Pull-down the down arrow to the right of the "Look in:" box and navigate to your folder on your H: Drive. Select your template file to open it. Make the changes you want and then re-save-as a Template file. To do this, pick "File" from the pull-down menu, then "Save As".

When the "Save Drawing As" dialogue box appears on the screen, click on the drop-down box labeled "Files of Type." Pick the "AutoCAD Drawing Template (*.dwt)" choice. Change the folder to which it will be saved to your Template folder on your H: drive by clicking in the "Save in" drop down box at the top, then pick the "Save" button.

20. You can make several template file versions. The one you just created will be used to plot D-sized drawings at 36" x 24" size on the large plotter. You could do another similar template (using this one as a starting point) for other plot formats, such as 18" x 24" (C-sized drawings) and 8 1/2" x 11" (A-sized drawings).

PROFILES

A "profile" is a personalized file with a filename extension of **.arg** that contains information about how you want the screen to look and information on where AutoCAD will search for files while you are working in AutoCAD. The Profile file can be saved and then loaded in the future when you begin a new AutoCAD session. This will automatically restore your personal settings.

How to create a Profile:

1. Right-click in Command: Line & select "Options..." from right-click menu

2. Click on the "Files" Tab

Click on the + sign to the left of "Support File Search Path" on the list. This will open the list of folders which are on the path of searching for files that will be inserted or externally referenced into the drawing. You can add your folder - select the "Browse" button and find your folder on your drive H:
The "Move Up" button will allow you to reposition the folders to make the ones at the top of the list be searched first - therefore if you have the same file name in two folders, the one in the folder at the top of the list will be found first.

Click on the + sign to the left of the "Template Settings" on the list. Then click on the + sign to the left of the "Drawing Template File Location." Click the Browse button and find the folder where your template file is located.

Click on the + sign to the left of the "Automatic Save File Location." Click on the "Browse" button and browse for the folder where you want to save your temporary save files. An automatic save will have the filename extension of ___.sv$ which is created automatically by AutoCAD and by default is saved to the folder C:\Documents and Settings\AT150\Local Settings\Temp\. These files are automatically deleted after you turn the computer off and on again, so it would be better to save them to your folder on your network drive (drive H:). Then you can keep them and always go back to them later.

3. Click on the "Display" tab.

Click on the "Colors..." button. Select "Layout Tabs Background (paper). Pick Black for the color. Click Apply and Close button.

Under the section labeled "Layout Elements" uncheck "Display printable area " and "Display paper background"

Slide "Crosshair size" bar to 100%
Note that most of the choices on the right side of this tab "Display resolution" and "Display performance" have an blue AutoCAD icon to the left side of the choices.

This means that you can change these variables for this session, but the permanent values are contained within the drawing you are working with. Therefore, you should make these changes in your Template file, not in the profile.

4. Click on the "Open and Save" tab.

Under the section labeled "File safety precautions" change the Automatic Save to 10 minutes between saves. This will create an automatic save with the filename extension of ___.sv$ which is created automatically by AutoCAD at 10 minute intervals and saved to your folder that you set in step 2 above.

Under the File Open section, change the number of recently-used files to list to 9 and check box labeled "Display full path in title."

5. Click on the "Plot and Publish" tab.

Under the section "Default plot settings for new drawings" select the radio button "Use last successful plot settings."

Click on "Plot Style Table Settings..." button in lower right hand corner. Select the radio button "Use color dependent plot styles." Select the monochrome.ctb plot style from "Default plot style table" drop-down list. Click OK.

6. Click on the "User Preferences" tab.

Under the "Windows Standard Behavior" section, check box labeled "Shortcut menus in drawing area."

Click the button "Right-Click Customization" and check box labeled "Turn on time-sensitive right click:" Click "Apply and Close." Click OK.

7. Click on the "Drafting" tab.

Under the "AutoSnap Settings" section, check boxes labeled "Marker," "Magnet," and "Display Autosnap Tooltip."

Slide Autosnap marker size slightly to the right to increase its size.

Under the "Object Snap Options" section, check box marked "Ignore hatch objects."

8. Click on the "Selection" tab.

Slide the Pickbox size slightly to the right to increase its size.

9. Click on the "Profiles" tab.

Select the "Add to List..." button. Type in a profile name that you want to create in the "Add Profile" dialogue box. Type in a description. The name of that new profile will appear in the "Available Profiles" list.

Click on the newly added profile on the list and select the button "Set Current." This will make your new profile the current one.

Select the "Export" button. Browse for your H: drive and type in the name you want to give to your profile. It should be the same name you gave it when you added it to the list of available profiles (although it does not have to be). AutoCAD will automatically append the filename extension *.arg to whatever name you give it when you export it. The purpose of exporting the profile is to be able to use it at another computer and at a future drawing session.

When you want to use this profile again in the future, right-click in Command Line & select "Options..." Select the "Profiles" tab. Select the "Import" button and find your profile (the .arg file). Double-click on its name. This will insert the name of the profile in the "Import Profile" dialogue box. Select "Apply and Close" button. The imported profile name will now appear in the "Available Profiles" list. Click on it and then click the Set Current" button. Click OK.