Engineering Drawing (E 122)
Engineering Graphics (E 125)

AUTOCAD HANDOUT

1 No AutoCAD yet

2 No AutoCAD yet

3 (Lec: Ortho Views)

4 (Lec: Ortho Practice)

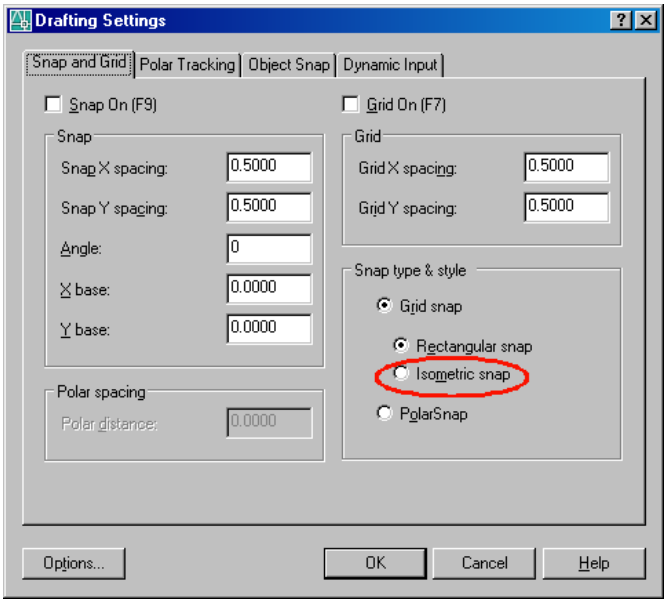
5 ORTHO PRACTICE II

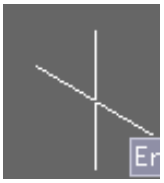
6 EXAM 2 ----------

7 PICKING VIEWS & BASIC DIMENSIONING

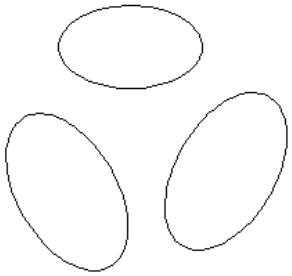
I. Isometric Ellipses (again)

A. Right click ‘snap’ in status bar, then pick ‘settings...’, to get drafting settings
B. Pick ‘isometric snap’ (notice cursor is now ‘skewed’)





C. Ellipse command (el), isocircle (i), pick center, pull out
   1. Note – “isocircle” option does NOT appear with rectangular snap setting.
D. Type F5 to move it to different planes. When it’s right, type value & <enter>



II. Rotate

A. Command = ro, Menu = modify/rotate, Toolbar (modify) 
B. Steps
   1. “ro” ↵
   2. Select objects ↵
   3. Pick base point ↵
   4. Enter value (in degees)
C. Rotating to a reference line
   1. After picking base point, “r” (Reference) ↵
   2. Specify reference angle – pick 2 points to be the reference line (we will make the line joining those 2 points parallel to the line joining 2 other points)
   3. Specify new angle or [Points]: “p” ↵
   4. Pick 2 points...

III. Offset – duplicate object // @ specified distance

A. Command = <o>, Menu = Modify/Offset, Toolbar (modify) 
B. Steps
   1. “o”
   2. Enter distance
   3. Pick object, pick direction (no need to hit <enter>)
   4. Pick object, pick direction...
   5. <enter> to exit

IV. Fillet

A. Command = f, Menu = modify/fillet, Toolbar (modify) = 
B. Steps – <f> ↵, <r> ↵, Enter value <0.000> ↵

V. Text

A. Single line text – “dt”, pick start point, type height, type angle, type TEXT 
B. Multi-line text – “mt”, pull out boundary box

VI. Hatch

A. “bh” or “hatch”, Draw/Hatch, 

VII. Styles

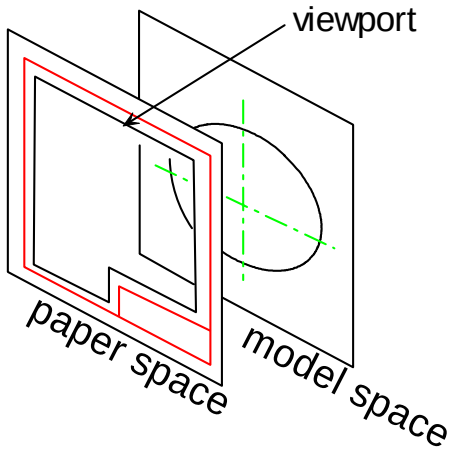
A. Styles are a collection of formatting that is given a name
B. There are styles for text, dimensions, etc.
C. I have already set them up in the template (don’t ALTER them, unless I tell you to)

VIII. Model vs. paper space

A. Model space – a type of level (different from the “layers” from before) where we place all drawing entities (lines, curves, dimensions, text, etc.). We have been drawing in model space up until now.
B. Paper space – another level that typically holds the sheet format (border, title block). Sometimes people will put notes, text, and dimensions on this level.

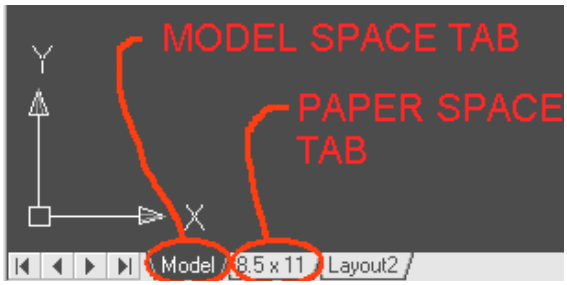
C. Concept – think of model space as a level that sits underneath paper space. The “viewport” is a hole that is torn out of paper space so that you can see thru to model space.

D. Why use it? If you sheet format is in model space, then you must have many different sheet formats to accommodate different sized parts. When using paper space, you draw your views full size and scale model space in the “viewport” (the “viewport scale”).

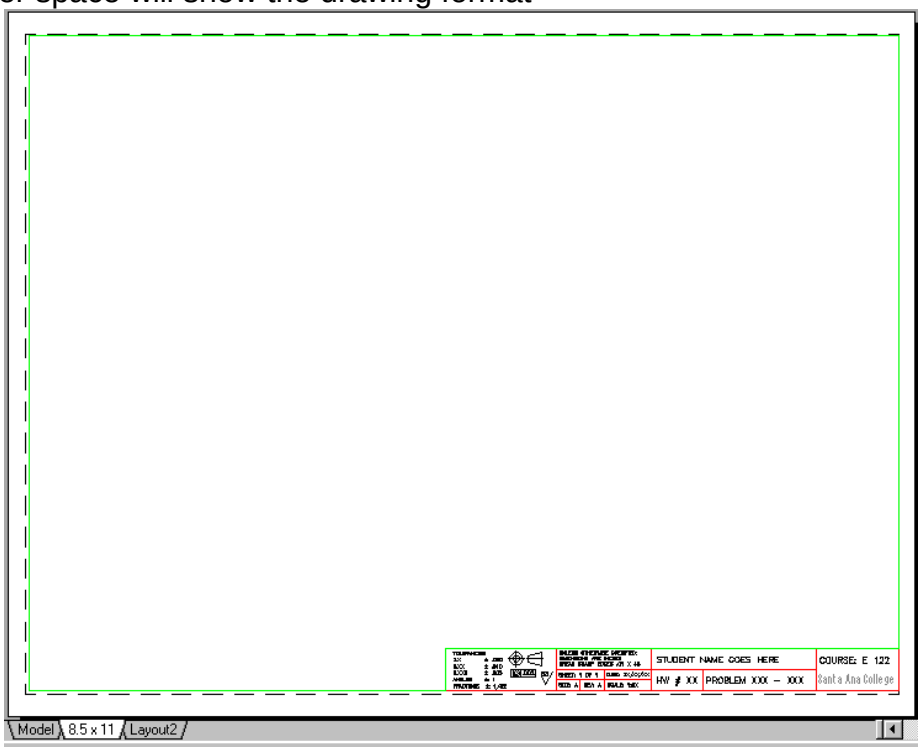


E. Going between model & paper space

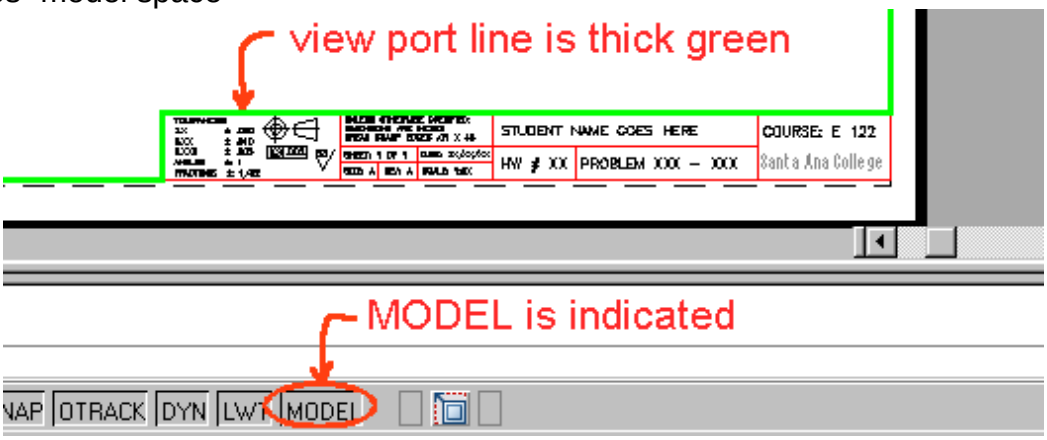
1. Toggle between the 2 spaces using the tabs at the bottom of the screen



2. The paper space will show the drawing format



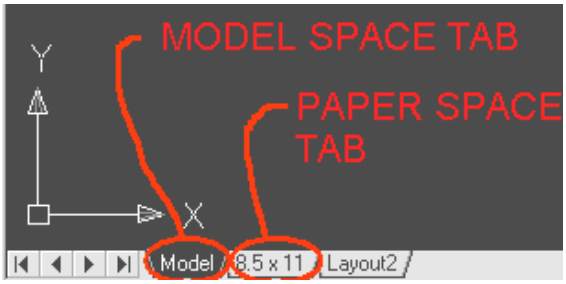
F. Alternate way to be on model space: While on the “8.5 x 11” tab, double-click in the white space in the middle. The viewport line (bright green) becomes thick, and the status bar indicates “model space”



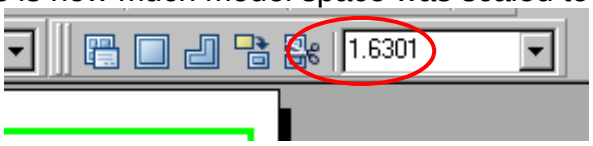
G. Set viewport scale with the paper-space template

1. In model space, draw all of your views FULL size (1:1). Do not dimension yet.
2. Turn on “viewport toolbar”
3. Dock the viewport toolbar on top (but NOT on right or left)

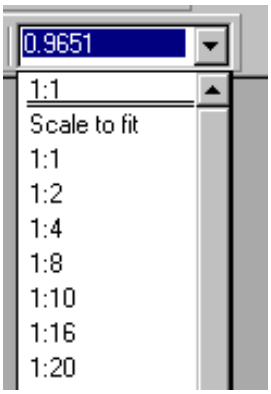
4. Get on paper space tab (8.5 x 11 tab).
5. Double click on white space in middle to activate model space (as shown above “Alternate way to be on model space”). The viewport line will be thick bright green.



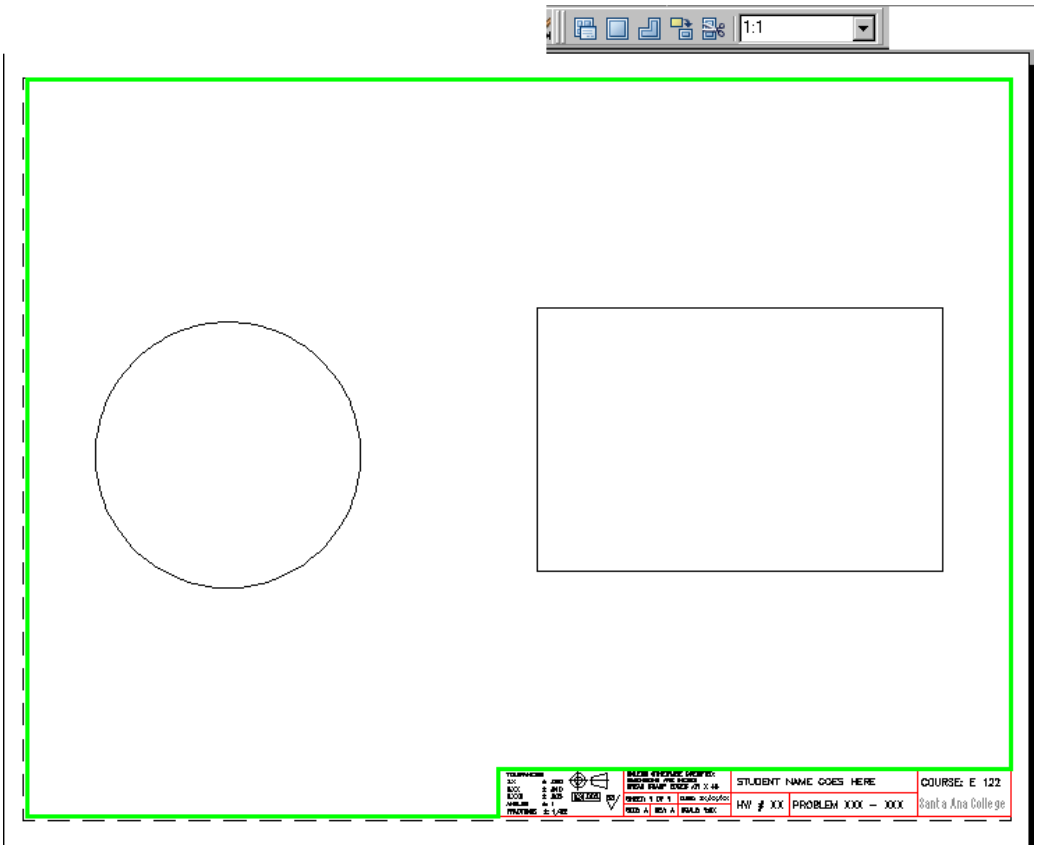
6. Zoom extents (z ↵ e ↵). All of you views should be fit onto the viewport area. But they will probably be a bit too big.
7. Observe the number in the pull down area of the viewport toolbar. This is the present “viewport scale” (this is how much model space was scaled to fit the viewport “hole”.



8. Reduce the viewport scale from its present value. Do this by accessing the pull down and selecting a value smaller than what you see presently. E.g., if you see 1.150, make it 1:1 (or 1.0000). You will see your view magnification shrink a bit.



9. Center the views on the screen using “pan”.
10. Be careful when panning with the scroll wheel. If you roll the wheel, then the scale is altered (but the scale shown on the viewport toolbar will not show it!).



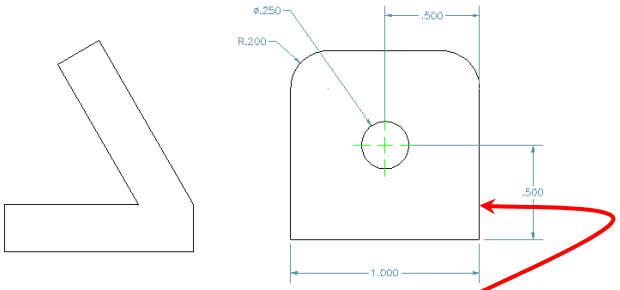
11. Once you are happy with the result, double click in the grey area outside the “sheet” to deactivate model space. The green viewport line will become thin.

8 OTHER VIEWS

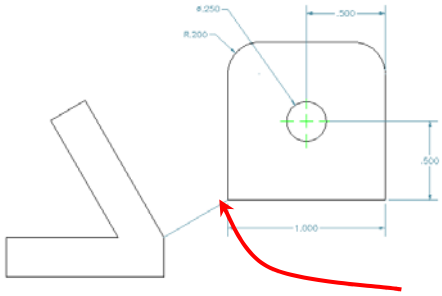
I. Auxiliary views

A. How create?

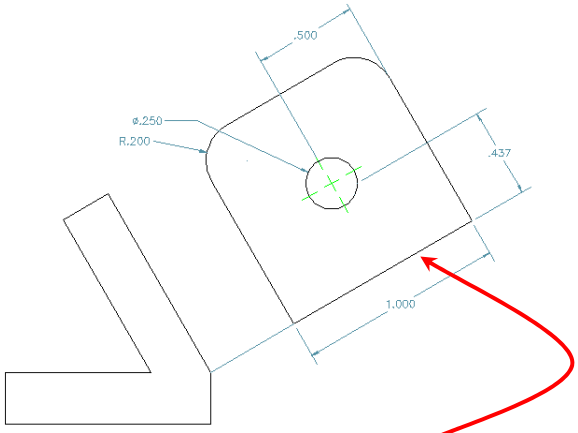
1. Draw view & all its annotations FIRST (including dimensions & centerlines).
2. Then rotate it LAST.



Auxiliary view drawn “upright”



Reposition auxiliary view (align it)



Auxiliary view after rotating

II. Sections

A. Draw the section view.
B. Get onto the “hatch” layer.
C. Hatch the view (do this BEFORE adding centerlines)
D. Cutting plane lines
   1. Get onto “cutting plane” layer (pink in my template)
   2. Draw a line from top to bottom.
   3. Pick the line and extend the top & bottom by .5” (use grips)
   4. Draw 2 horizontal lines at .5” each.
E. Arrows for cutting plane lines
   1. Create a horizontal dimension.
   2. Exploded it. “x” command.
   3. Delete everything except one arrow.
   4. Scale the arrow and then use it for the cutting plane lines

9 DIMENSIONING

I. Dimension toolbar

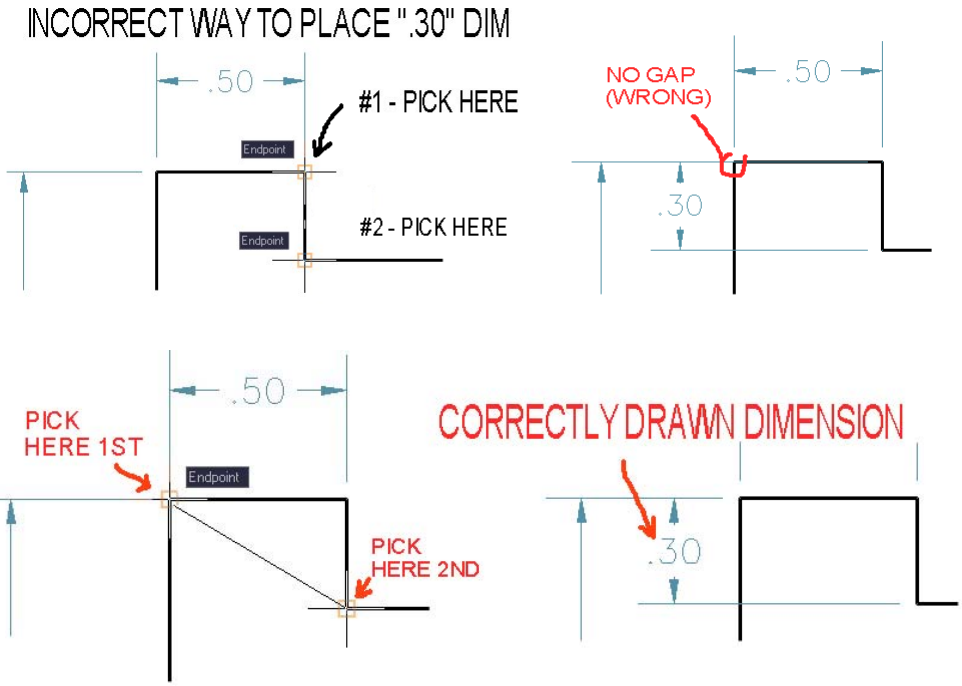
A. Need the dimension toolbar

B. Linear  , Aligned  , Angle  , Radius  , Diameter 

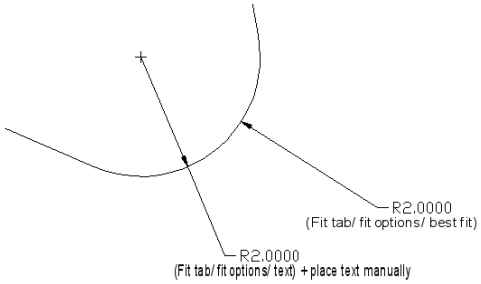
II. Dimensioning

A. Linear dimension
   1. Pick 2 points (zoom in so snap is correct)
   2. Then place location of dimension value
   3. AutoCAD will automatically leave a gap

B. Offset locations
   1. Make sure you pick the correct 2 points (or the gap won't be there).



   2. Radius



C. Be careful – AutoCAD will let you pick the end of an extension line. Result – the dimension will be wrong.

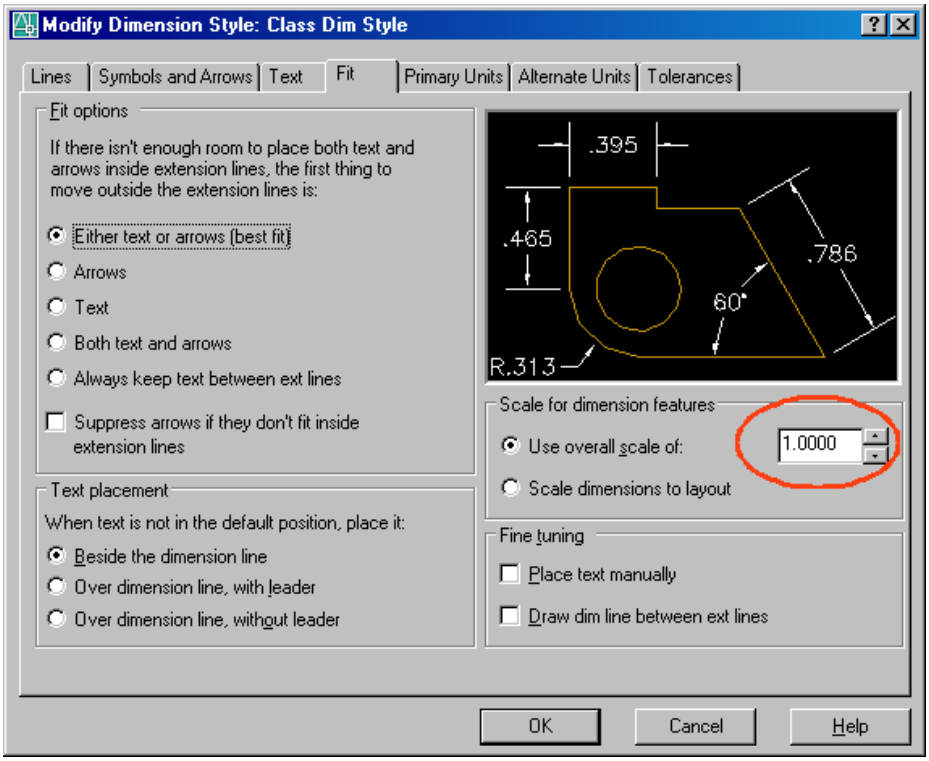
10 DIMENSIONING II

I. Dimensions and viewport scale

A. Note – we will be placing our dimensions in model space. When doing this, the dimension numbers will shrink or grow as we adjust the viewport scale. Sometimes they can become too big or too small, and we must compensate for this.

II. Steps

A. In model space, draw all of your views FULL size (1:1). Do not dimension yet.
B. Go to the 8.5 x 11 tab and adjust the viewport scale as shown above (& summarized below).
   1. Zoom extents (z ↵ e ↵).
   2. All views should fit the viewport area, but they will probably be a bit too big.
   3. Select a standard scale (using integer values) that is the next size smaller than what you presently have. This is necessary to make room for the dimensions
C. Get into the dimension style manager (type "d" ↵)
   1. Make sure "Class Dim Style" is highlighted.
   2. Press "Modify...".
   3. Click the "Fit" tab.
   4. Alter the value under "Use overall scale of:".
   5. Set the value to be the RECIPROCAL of the viewport scale.
   6. This will preserve the size settings in the dimension style as they are shown in paper space.



Examples:

Viewport scale	Dimension scale
1:1	1.000
1:2 (=.50)	2.000
2:3	1.500 (or 3:2)
3:4	1.333 (4:3)

III. Symmetrical centerlines

A. Extend centerline .5 beyond part
B. Add 2 thick lines on ea side – .3 long, .05 from end, .1 space between, .4 mm thick

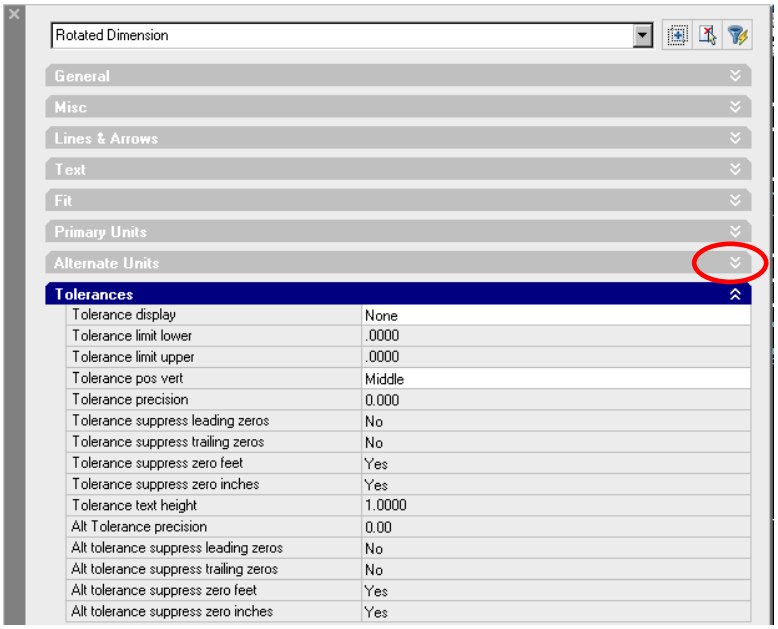
11 EXAM 2 ----------

12 TOLERANCING & ASSEMBLIES

I. Adding tolerances to dimensions

A. Steps

1. Access properties palette
   a. Older AutoCAD: double click the entity
   b. Newer AutoCAD: right click the entity, pick “properties” (double clicking opens “quick properties” instead now)
2. Expand or collapse subheadings as needed by clicking on the arrow (see below).



3. Go to: Tolerances / Tolerance display
   a. Choices: symmetrical, Deviation (unequal bilateral), Limits, Basic
   b. Pick the type, set values, & dropping leading zero (suppress leading zero)
   c. Tolerance precision – set # places on tolerance

II. Change the number of places on a dimension

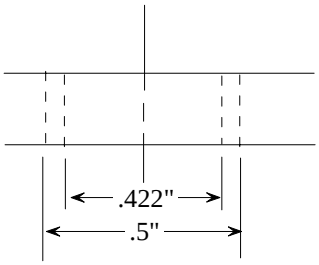
A. Use properties palette: Primary Units/ Precision
B. Note – pick ALL of the dimensions needing, say, 2 places, then get into the properties palette and then alter. There’s no need to do it 1 at a time.

III. Project

A. General – text height = .07 (smaller than standard, but usually dwg are on larger paper)
B. L-Bracket
   1. Symmetrical centerlines
      a. Extend .5 beyond part
      b. 2 Thick lines - .3 long, .05 from end, .1 space between, .4 mm thick
   2. Creating the “R.25 TYP” dimension
      a. First add the radius dimension.
      b. Bring up the properties palette for the dimension.

c. Under the “text” heading, under ‘text override’, type: “<> TYP”.
d. Note – the “<>” holds the original text information (in this case, “R .25”).

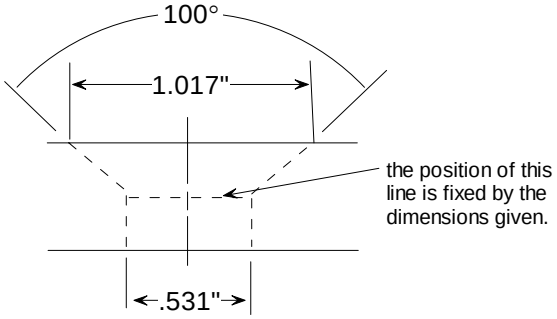
3. Creating the “4X ∅.500 THRU ALL” callout.
   a. First dimension the circle diameter.
   b. You can do the same as above, but the “hard returns” are signified by a “\P”.
   c. OR you can type “ed ↵” (edit command)
   d. Click the dimension text. The multi-line text formatting toolbar appears and you can format the dimension text as multi-line text.
   e. Symbols appear under the arrow down symbol (“∨”), & symbol.
4. General notes
   a. Use the text layer. Draw on paperspace (text height .07)
5. Centerlines for aligned holes
   a. Do center-marks first. Then draw centerlines to ends of extension lines
6. Threads (drawing the 1/2 – 13 UNC threads)
   a. In the circular (axial) view, make the smaller circle a visible line at ∅.422”.
   b. In the circular (axial) view, make the bigger circle a visible line at ∅.50”.
   c. In side view, draw lines aligned with quadrants of circles just drawn above



13 ANNOTATIONS & FEATURES LECTURE (AUTOCAD LEC = PROJECT)

I. Pillow block drawing

A. Creating the countersink symbol
   1. In the circular view, the smaller circle has a diameter of .531”.
   2. In the circular view, the larger circle has a diameter of 1.071”.
   3. In the side view, use the figure below.



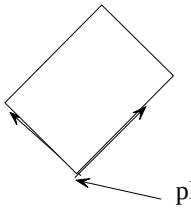
B. Adding the diameter symbol on a “side” view of a cylinder
   1. Use the same technique described above for the “4x <> THRU ALL” callout.
   2. Main difference is the diameter symbol (∅) is %%c.
   3. So type, “%%c <>” in the proper field.

C. Creating the limit tolerance callout
   1. Double click the dimension in question.
   2. Go to the tolerances heading, select “limit” for the ‘tolerance display’ field.

3. For the ‘tolerance limit upper’, type “.000”.
4. For the ‘tolerance limit lower’, type “.001”.
5. To create the diameter symbol (∅), follow the instructions above.

II. Bent arm drawing

A. Auxiliary view (technique 1) – as shown in class, draw the auxiliary view horizontally. Add dimensions. Then rotate the view and dimensions. Then position properly.
B. Auxiliary view (technique 2) – draw the auxiliary view at the appropriate angle. Then move the UCS (user coordinate system).
   1. Ensure the UCS toolbar is visible (RH click the toolbar area and check it).
   2. “Move the UCS” to the lower left corner of the auxiliary view using the 3-point UCS.
   3. The dimensions must be rotated afterwards. Click all of the dimensions, then double click to get the properties. Under text heading, rotate the dimensions.



   4. Dimensions can alternately be inserted as “rotated” dimensions. Click the 2 points to be dimensioned. Before laying down the dimension text, right click, and select ‘rotated’. Then pick 2 points that signify the direction of the dimension line.

III. Assembly Balloons (AutoCAD 2006)

A. Dimension/Leader
   1. Click where you want “arrow”
   2. Pull out diagonal leader line, click
   3. Click again to accept text width of .000
   4. Text, type “5”, to enter “5”
   5. Enter again to exit text entry
   6. Double click text and change horizontal alignment to center (leave vertical alignment alone at TOP)
   7. Double click leader to get properties palette (newer AutoCAD: right click, properties)
   8. Change text offset (under text) to 0.075
   9. Text height is per dimension style
B. Circle –
   1. make diameter .6
   2. Move so quadrant meets end of horizontal line
C. Change leader arrow style to “DOT” – dot small
D. Set dot size (.1 overall)

IV. Balloons with AutoCAD 2010

A. Dimension/ multi-leader
B. Click where you want “arrow”
C. Pull out diagonal leader line, click.
D. You get the multi-line text formatting window. Click OK (don’t enter any text).
E. Make a circle of .6 diameter.
F. Move circle so its quadrant aligns with the end of the horizontal part of the leader.
G. The line will cross into the circle. To fix, right click the leader/ properties (older AutoCAD: double click) leader to get properties palette.

1. Under Text – make “Left attachment” & “right attachment” = “middle of text”.
2. The line should end right at the side of the circle now.
3. Set Leaders/Arrowhead to “Dot small”

H. Under “Leaders/ Landing Distance” – set this to .25 (the length of the horizontal line)
I. Landing gap = space
J. Type “dt” for single line text. Set the location at circle’s center. Set height to be .21, rotation angle 0, and type the number desired (e.g., “5”). Enter, enter.
K. Right click the number. Go to properties. Set text/justification to “middle center”.

V. BOM

A. Draw on paperspace.
B. Text – on text layer, .07 high
C. Table – 3” wide, 1.5 high, .25 high each row,
   1. Columns - .3, .3, 1, 1.4 wide

14 FASTENERS, HOLES, & THREADS

I. Fasteners

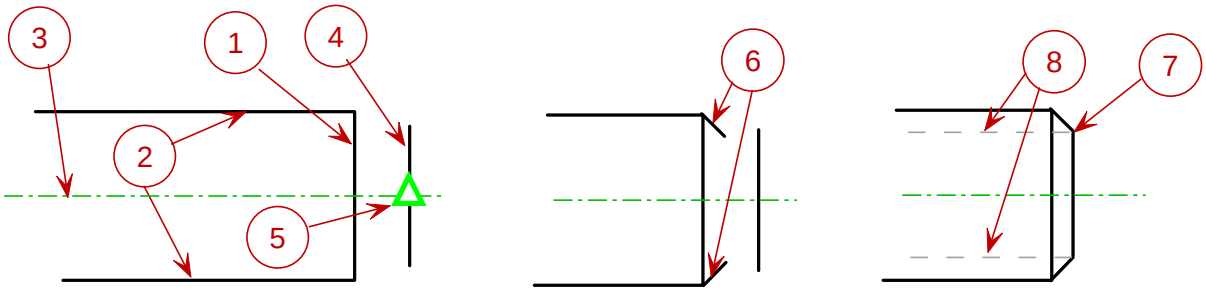
A. Fastener dimensions are standardized – must look up in a table

II. Holes

A. Holes dimensions are standardized – must look up in a table
B. Blind holes (do not go all the way thru) – draw a “tip” (angle = 120°).

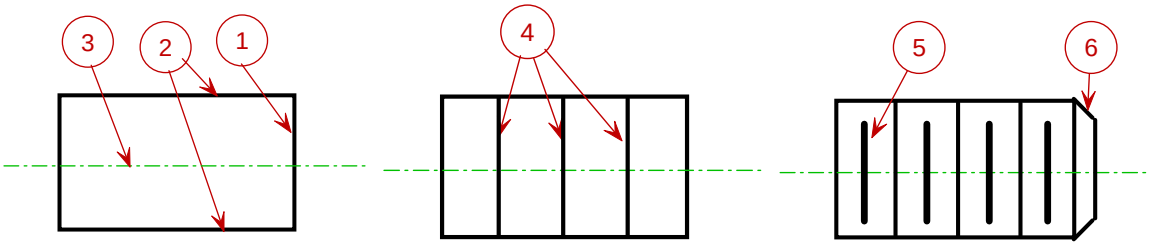
III. Threads

A. Simplified – steps
   1. Draw a line for the major diameter (its length = major diameter value).
   2. Draw perpendicular lines for the thread length
   3. Draw a centerline thru midpoint of major diameter line & parallel to “length” lines
   4. Draw a line for the minor diameter.
   5. Vertically align its midpoint with that of the major diameter line
   6. Draw a 45 degree chamfer off the major diameter.
   7. Move the minor diameter line.
   8. Draw hidden lines parallel to centerline to represent the minor diameter

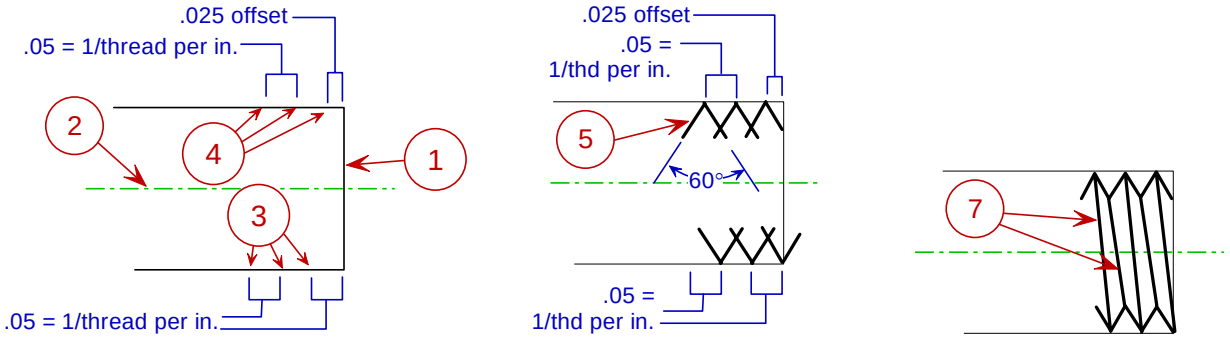


B. Schematics – steps
   1. Draw a line for the major diameter
   2. Draw lines for the thread length.
   3. Draw a centerline.
   4. Divide the length into convenient equal increments
   5. Draw “minor” diameter lines (THICK).

6. Draw 45° chamfer on end.



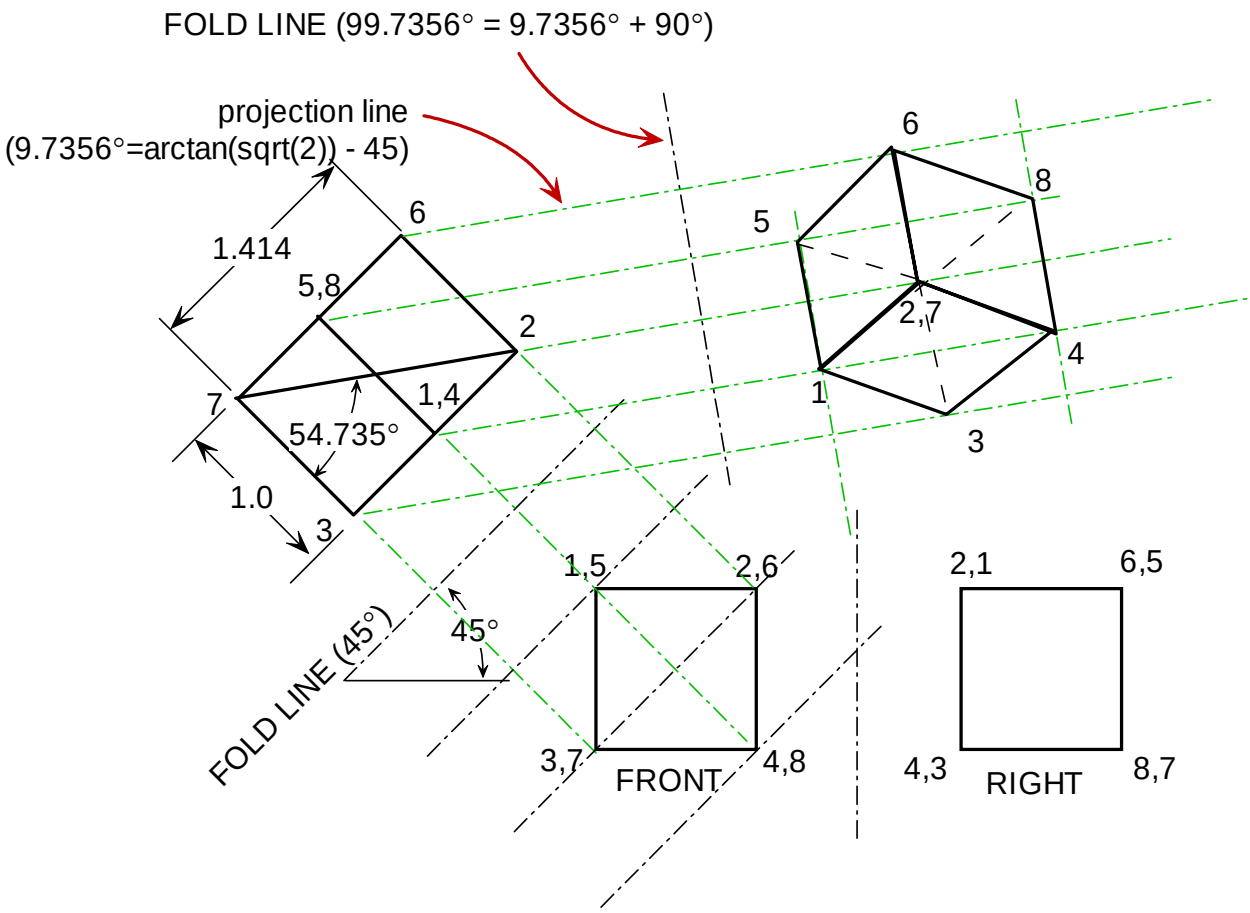
C. Detailed – steps
   1. Establish major diameter & thread "length"
   2. Draw centerline (thru midpoint of major diameter line)
   3. Establish crest positions for side 1– for single thread, distance between crests should be 1/threads per inch. (e.g., 1/4 – 20 threads... 20 = 20 threads per inch, so distance between crests is 1/20)
   4. Establish crest positions for side 2 – spaced as above, but offset by half distance.
   5. Set thread angle = 60° (for machine threads).
   6. Trim. Note the trimmed lines establish the minor diameter.
   7. Join crests and roots with straight lines.
   8. Note – this is just an estimate!



15 NOTE – ISOMETRICS VIA DESCRIPTIVE GEOMETRY

I. Use Descriptive Geometry techniques

   A. Use "fold lines" at different angles to create new orthographic views
   B. Keep track of the positions of all vertices.
   C. A vertex must remain on its projection line
   D. Distance along the project line is determined by the adjacent view.


FOLD LINE (99.7356° = 9.7356° + 90°)
projection line
(9.7356°=arctan(sqrt(2)) - 45)
1.414
6
5,8
2
7
1,4
54.735°
1.0
3
6
8
5
2,7
4
1
3
FOLD LINE (45°)
45°
1,5
2,6
3,7
4,8
FRONT
2,1
6,5
4,3
8,7
RIGHT