Technical Drawing

MEC1000

Spring 2006

Instructor: David Anderson

Topics

- Drawing Views
- Drawing Standards
- Best Practices
- Creating Drawings in SolidWorks

Drawing Views

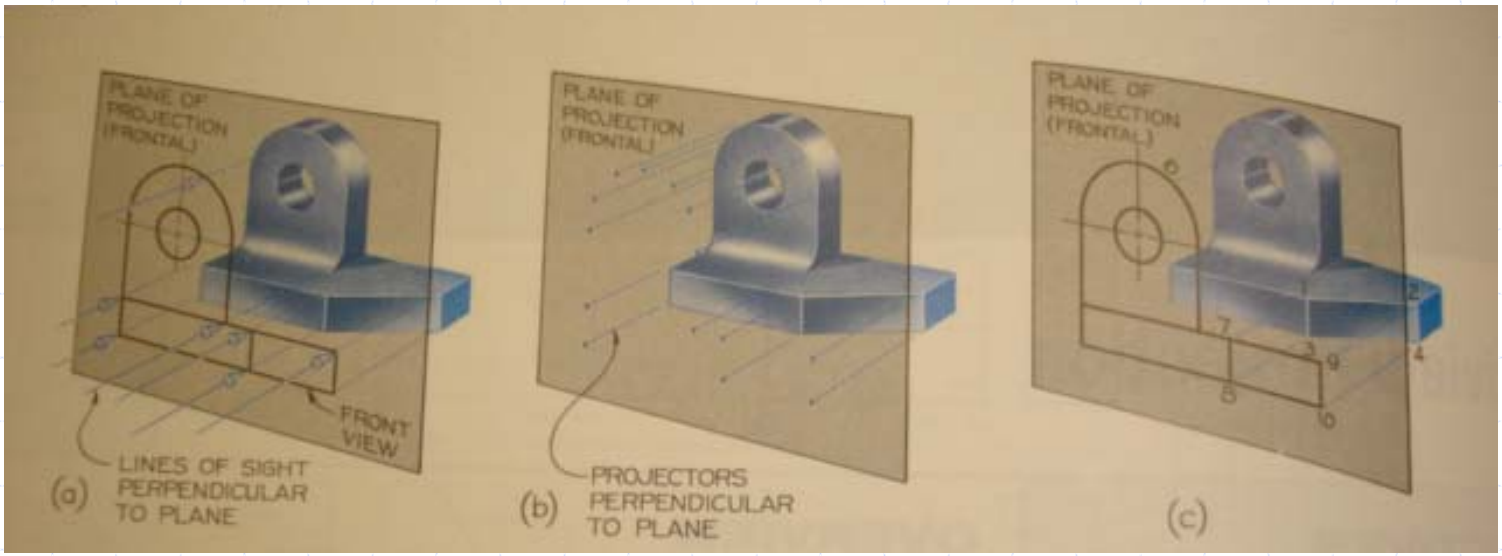
- Multi-View Projection - The Glass Box
- Third Angle Projection
- Two View Drawings
- Line Types
- Section Views
- Auxiliary Views
- Detail Views
- Broken-Out Section Views
- Partial Views, Cropped Views

Drawing Views – Multiview Projection

- A view of an object is know technically as a ***projection***
- A projection is a view conceived to be drawn or *projected* on to a plane, known as the ***plane of projection***
- Multiview or *orthographic projection* is a system of views of an object formed by projectors from the object perpendicular to the desired plane of projection. Huh?

Drawing Views – Multiview Projection

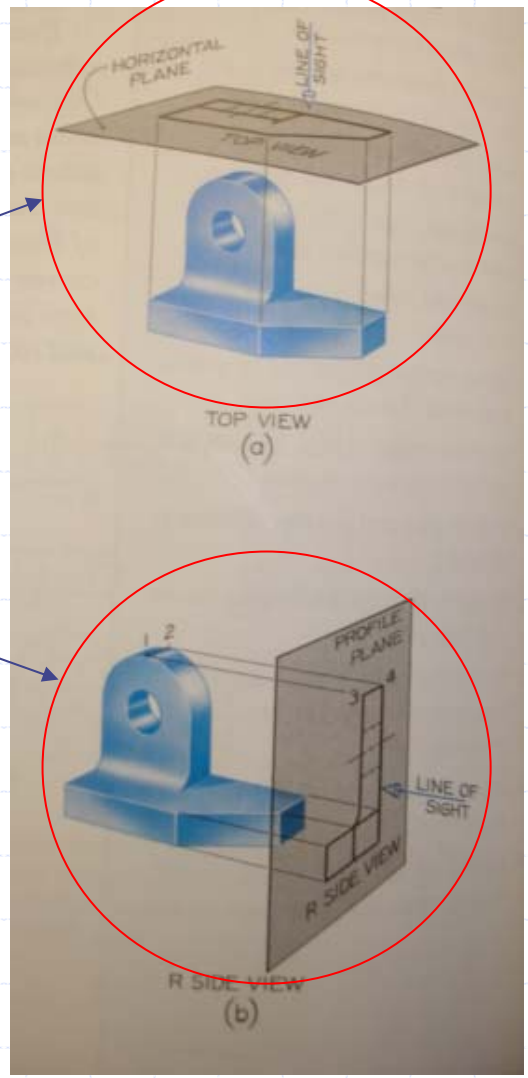
- The projection of an object.
  - Perpendicular lines or *projectors* are drawn from all points on the edges or contours of the object to the plane of projection.
- Shown below is the projection of an object onto the frontal plane.



Drawing Views – Planes of projection

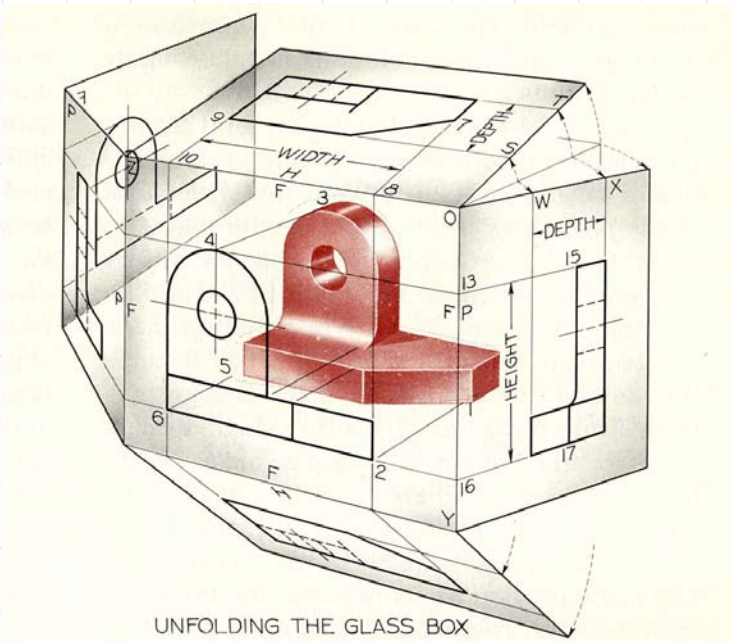
likewise,

- the top view is projected onto the ***horizontal plane***
- the side view is projected onto the ***profile plane***



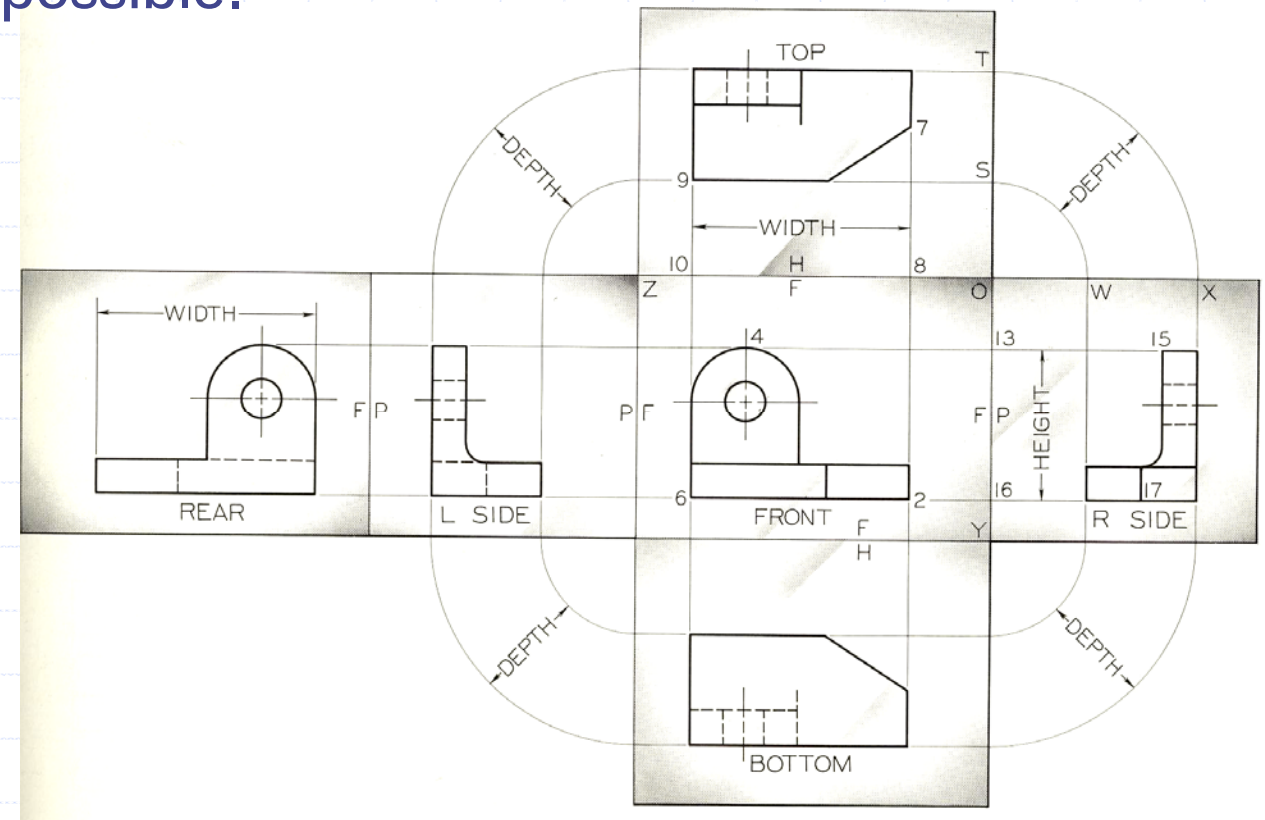
Multiview Projection – The Glass Box

- Placing parallel planes to the principal planes forms a *glass box* (always observed from outside the box)
- To show views of a 3D object on a 2D piece of paper, it is necessary to unfold the planes such that they lie in the same plane
- All planes except the rear plane are hinged to the frontal plane, which is hinged to the left-side plane



Multiview Projection – The Glass Box

- By unfolding the box, six views of the object are possible.



Drawing Views – Third Angle Projection

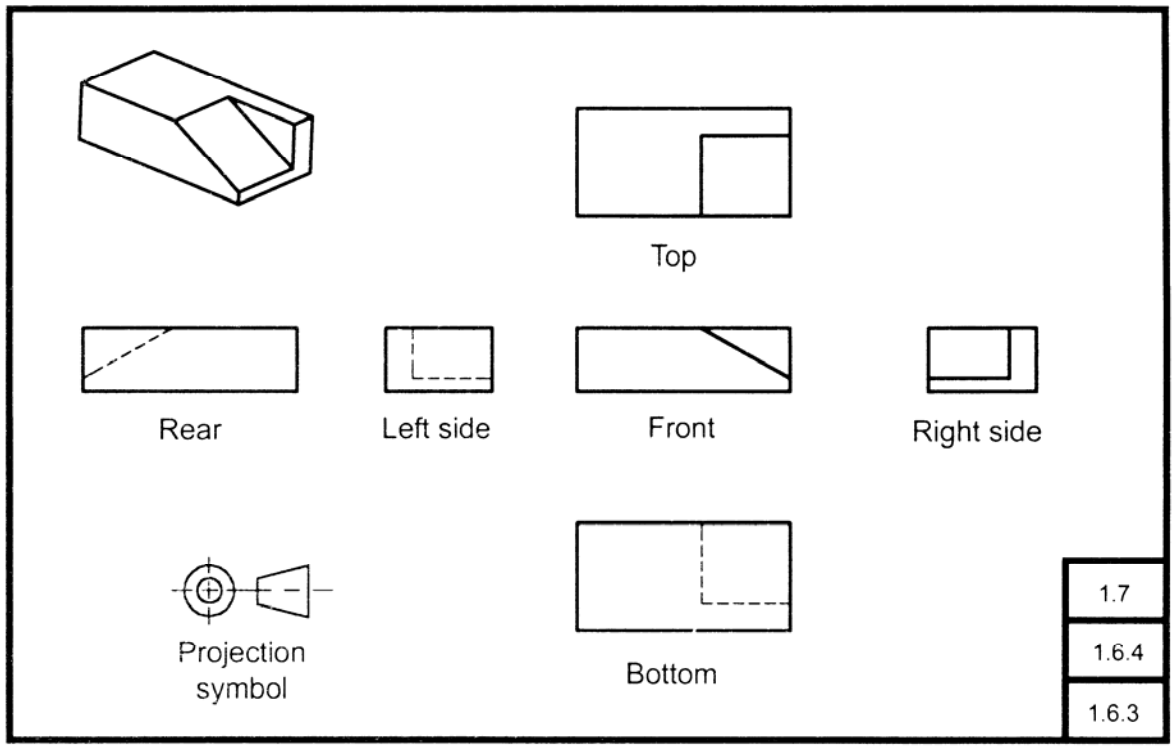


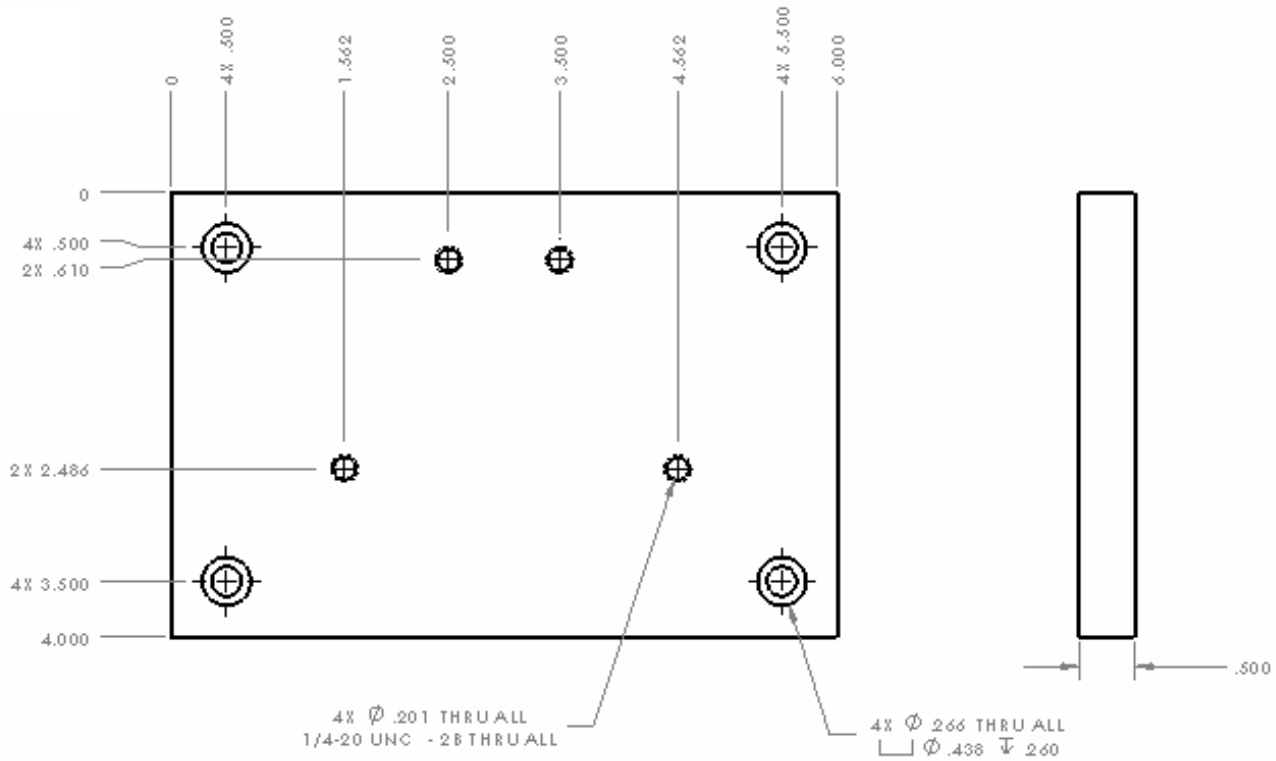
Fig. 4 Third Angle Projection Standard Arrangement of the Six Principal Orthographic Views

Multiview Projection – Proper number of Views

- It may not, be necessary to show all six views to completely describe the object.
- In fact, the minimum number of views is preferable.
- How many views are necessary to completely describe this plate?
- 1?
- 2?
- 3?
- 4?

Multiview Projection – Two View Drawings

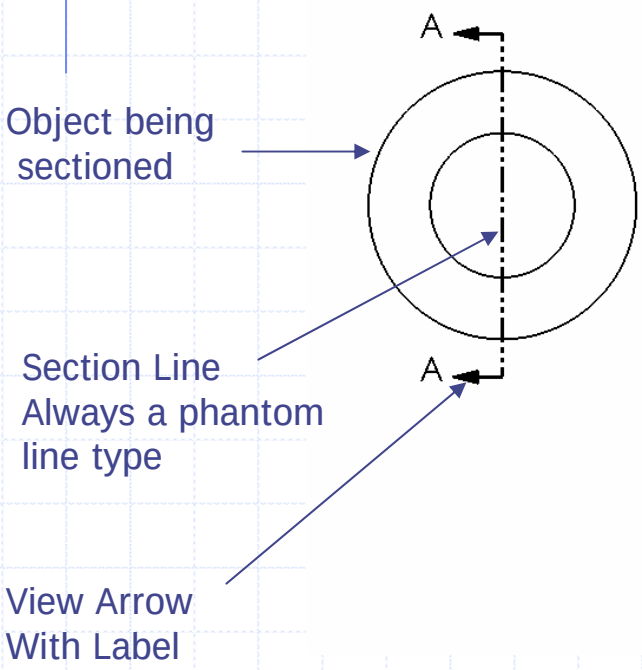
- The answer is 2!

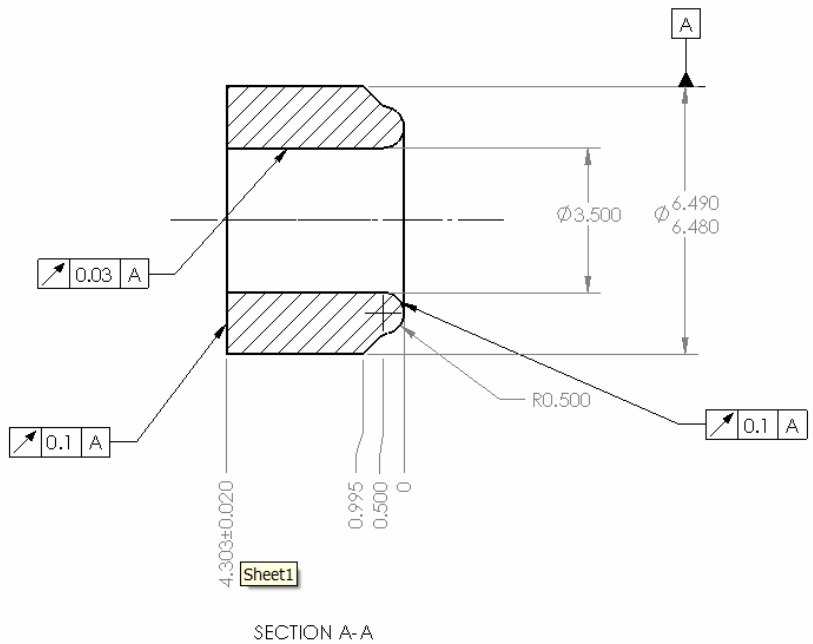


Drawing Views – Sectional Views

- We have covered the basic method of representing an object by projecting views. This allows us to see the external features of an object.
- Often times it is necessary to view the internal features, this is accomplished by slicing through the object and producing a *sectional or section view*

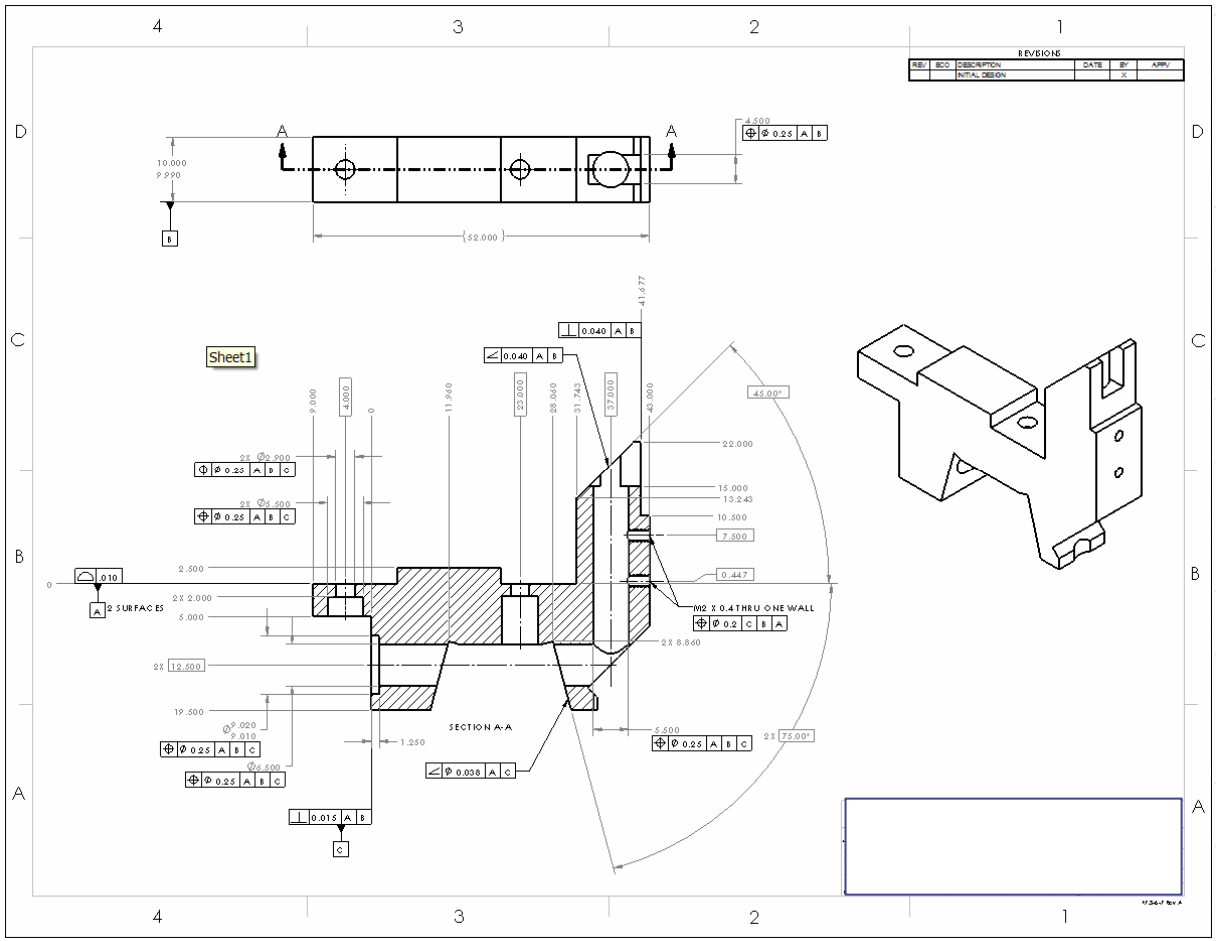
Section view is always placed BEHIND arrows





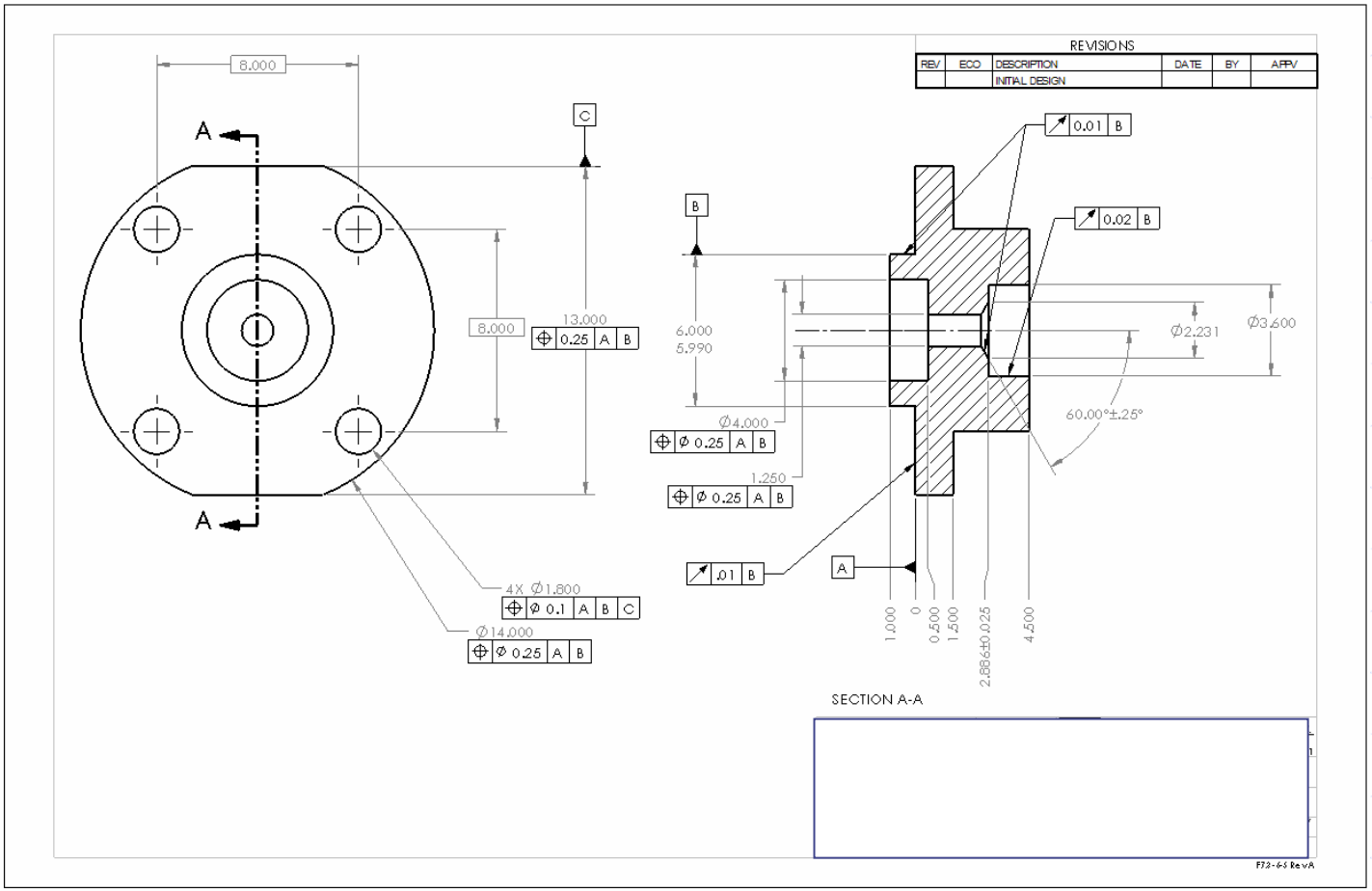
Drawing Views – Sectional Views

Sectional views are extremely useful in minimizing the number of projected views. How many views does this object require?



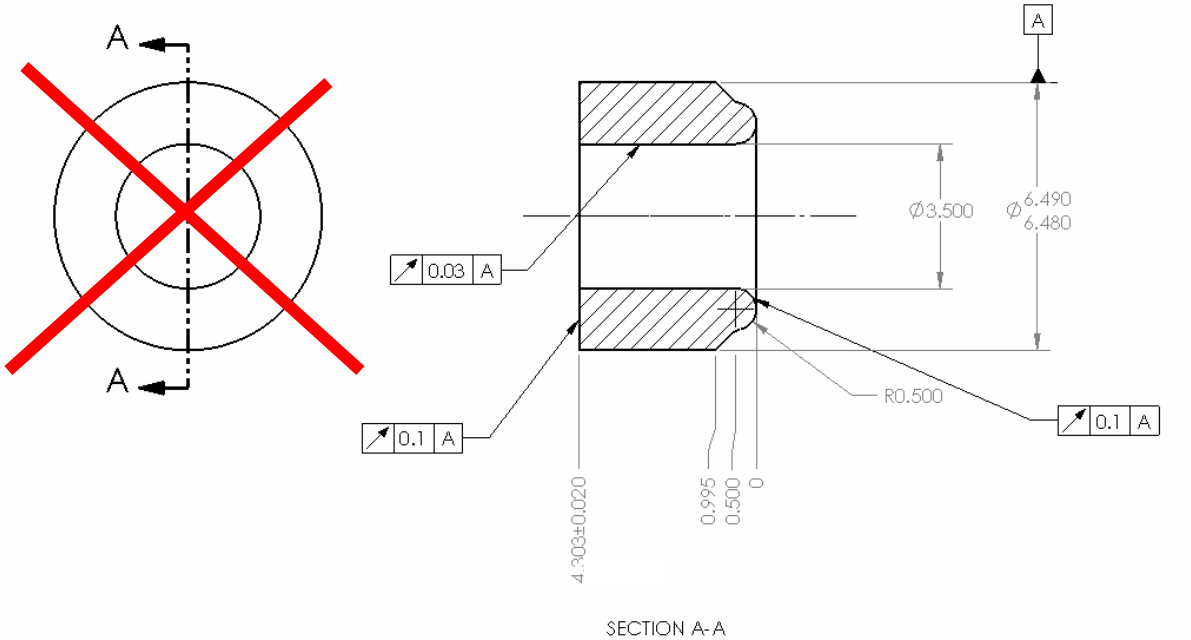
Drawing Views – Sectional Views

- Section views provide clear and unambiguous representation of internal features



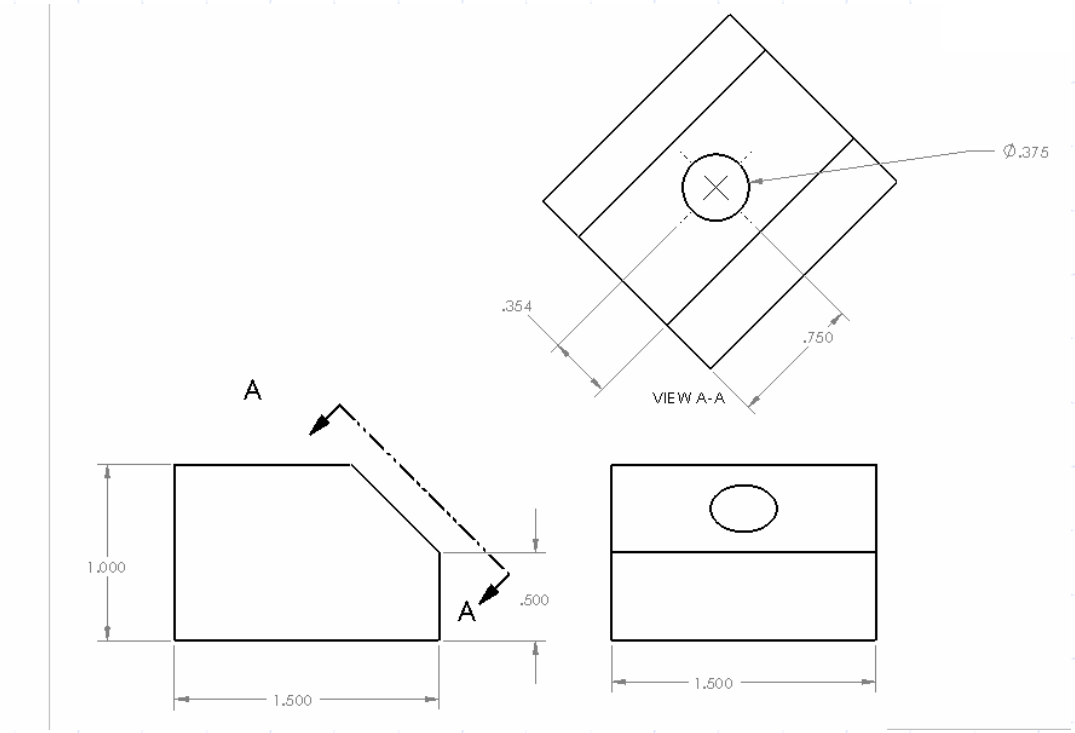
Drawing Views – Sectional Views

- Section views can reduced the number of views of many axisymmetric parts to a single view



Drawing Views – Auxiliary Views

- Inclined planes and oblique (neither parallel nor perpendicular) lines appear foreshortened when projected to the principle planes of projection.
- To obtain a true size view, *auxiliary* views are created using similar techniques as for creating standard views, unfolding about an axis...



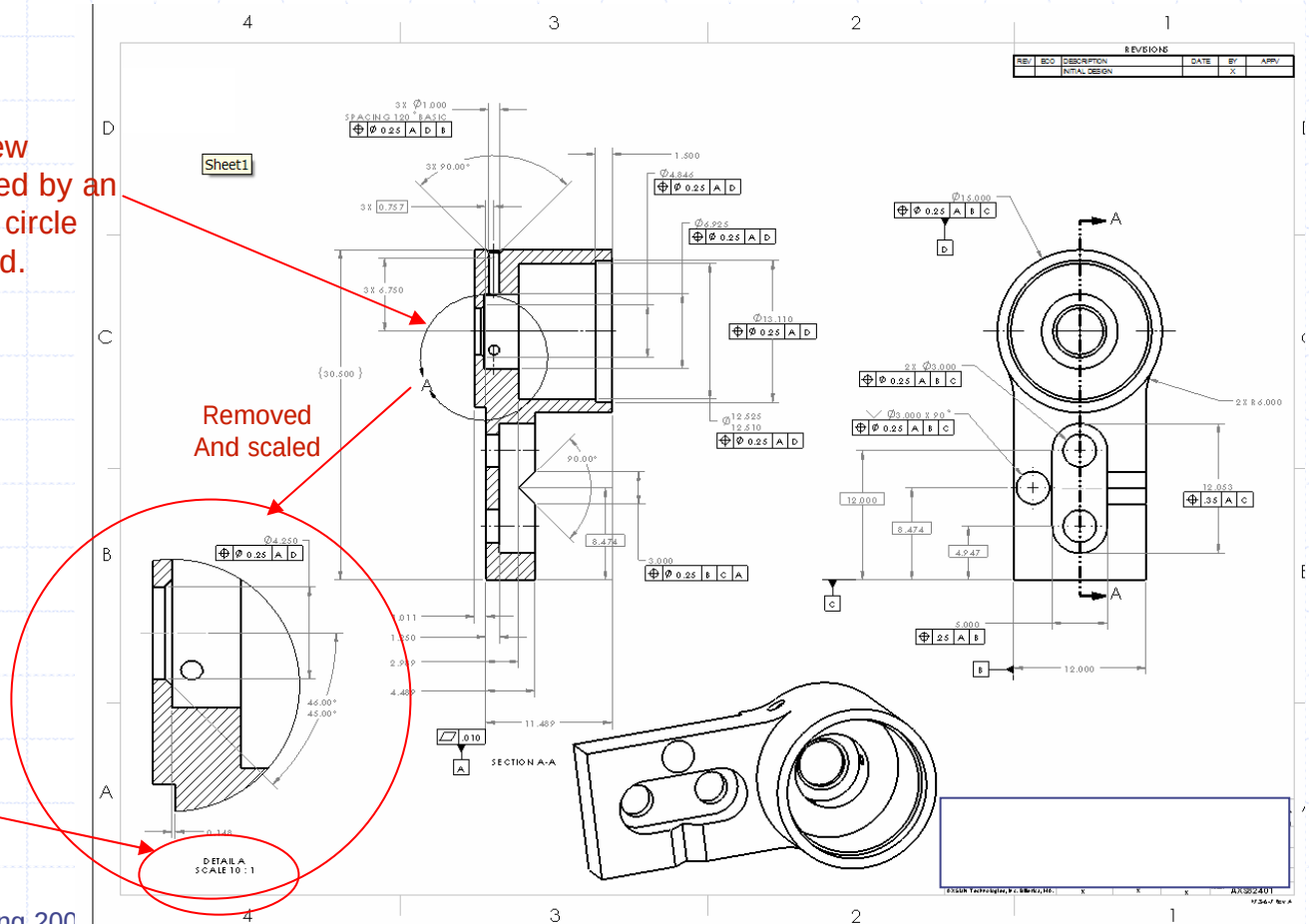
Drawing Views – Detail Views

When there is a great disparity between feature size, or views are overcrowded with dimensions, a *detail* view can be used to capture the feature(s) of interest and display them in a *removed view of greater scale*.

Detail View Designated by an Enclosed circle and labled.

Removed And scaled

Labeled and scale noted

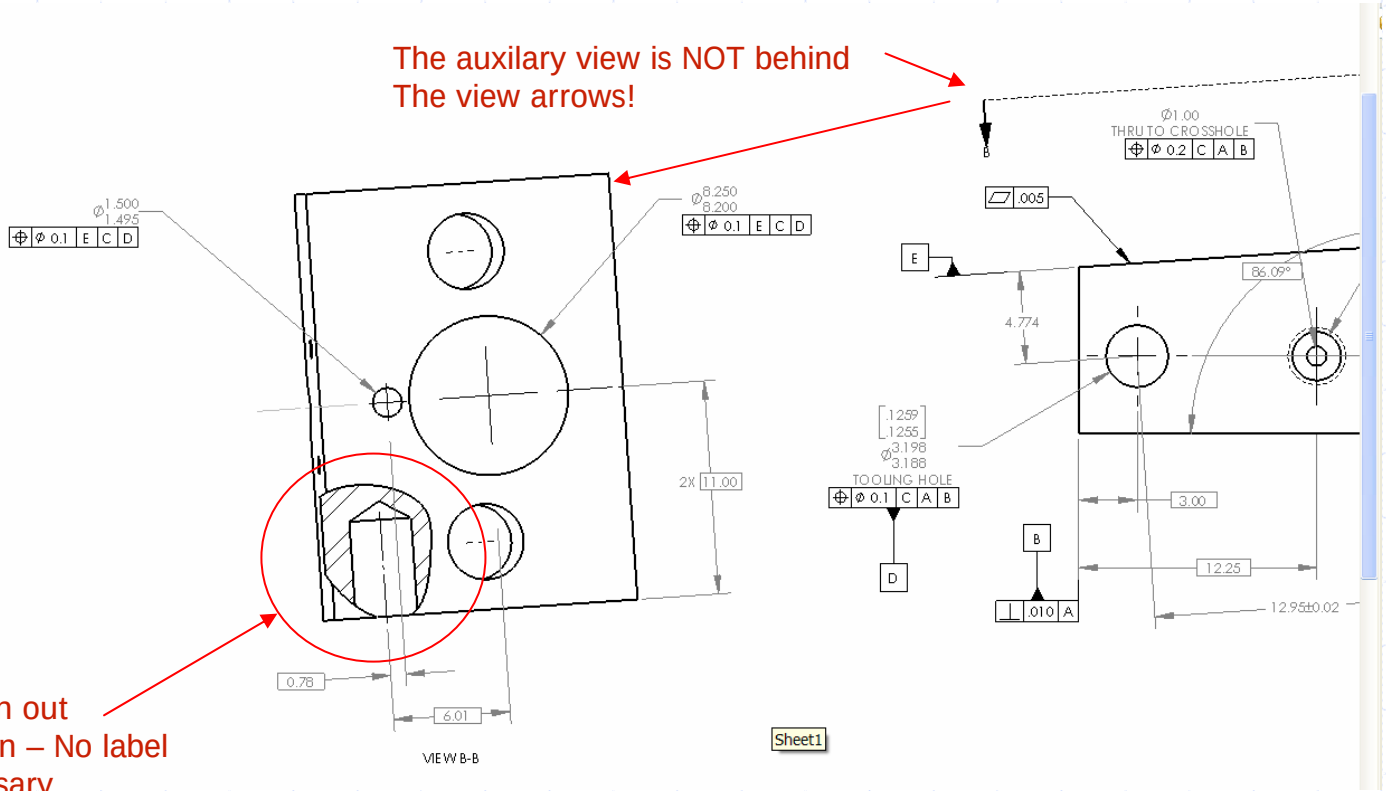


Drawing Views – Broken-Out Section

Broken-out Section views are essentially partial section views with out the section arrow. Often times they are used to expose a feature of interest while eliminating the need to create another view.

What is wrong with this drawing?

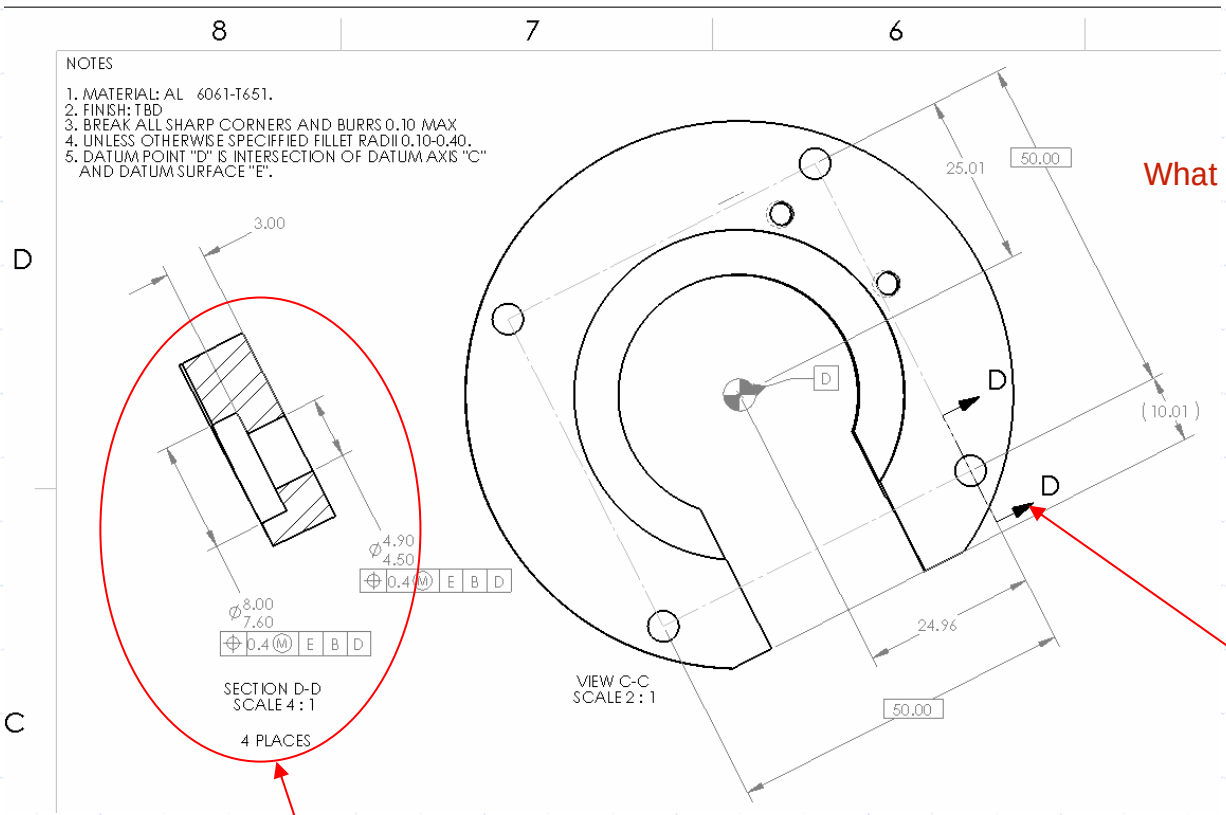
The auxilary view is NOT behind The view arrows!



Broken out Section – No label necessary

Drawing Views – Partial Views

Partial views are removed views and are established in a similar manner as section views, that is they *require view arrows to establish viewing direction*. However, they do not have to section an entire object, rather can simply display a partial view of a projection at a larger scale if desired.



What is wrong with this drawing?

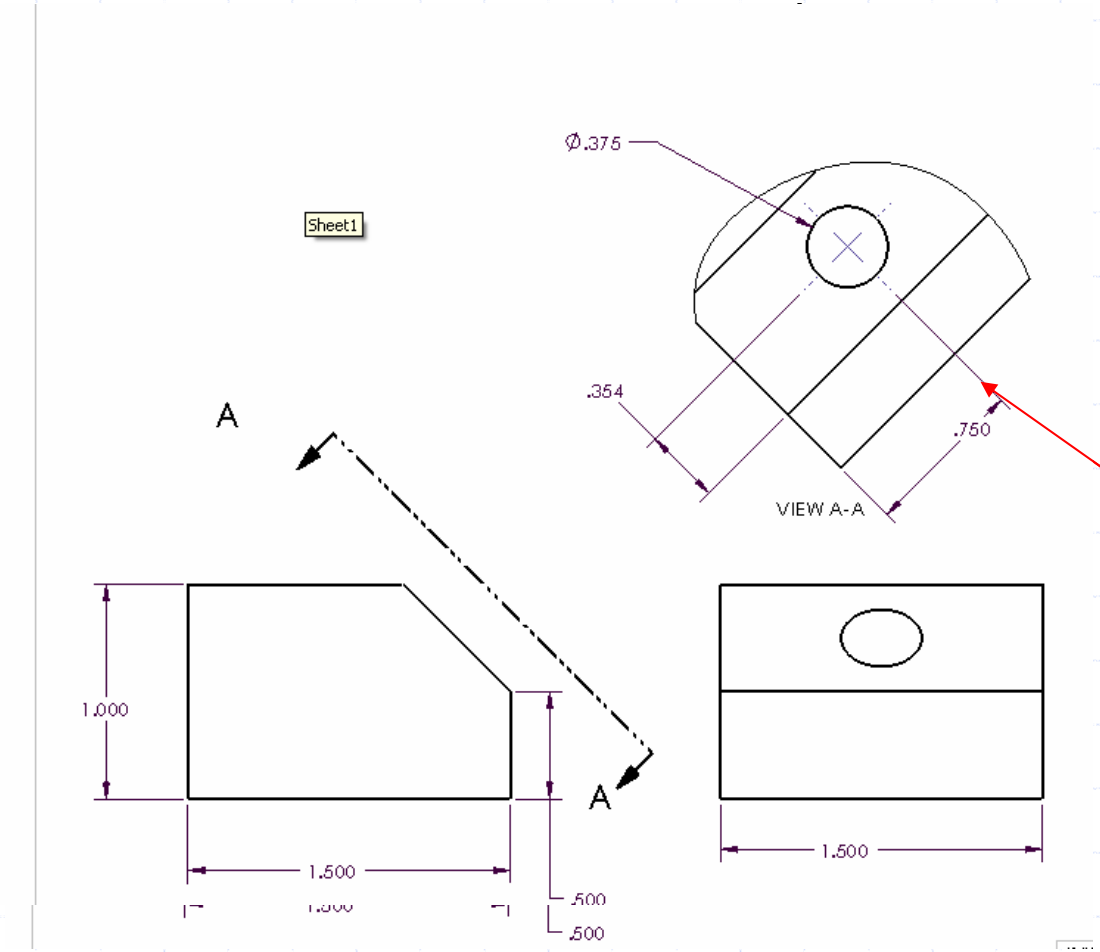
Nothing!

Partial Section Line w/Labled Arrows

Removed partial section view Labled and scale noted

Drawing Views – Cropped Views

Cropped views reduce the size of a view such that only necessary information is displayed. Cropped views also maximize the sheet area by reducing view size.



Cropped View

Drawing Standards

- ASME responsible for mechanical drawing standards
  - Sheet Formats
  - Line Types
  - Dimensioning Rules and Schemes

Drawing Standards - ASME

- There exists standards and practices for creating technical drawings of mechanical parts and assemblies. The governing agency responsible for setting the standards is ASME. There are a number of documents published by ASME that cover various aspects of mechanical drawings, here are a few of them...

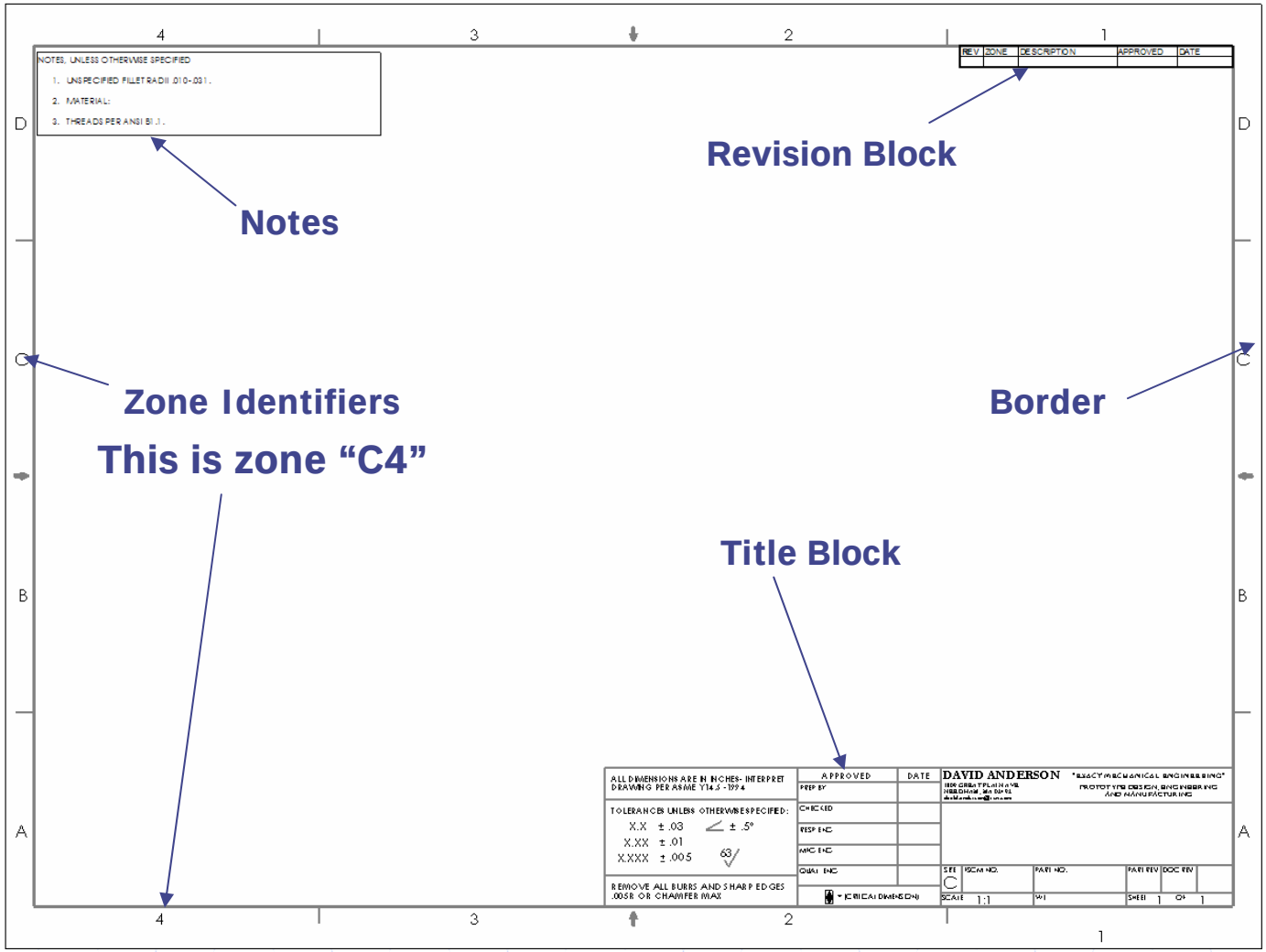
- ASME Y14.100 -2004 Engineering Drawing Practices
- ASME Y14.4M - 1989 Pictorial Drawing
- ASME Y14.3M – Multi and Sectional View Drawings
- ASME Y14.1 - 1995 Decimal Inch Drawing Sheet Size and Format
- ASME Y14.5M – 1994 Geometric Dimensioning and Tolerancing
- ASME Y14.13M - 1981 Mechanical Spring Representation

- It is important to follow these standards to ensure your drawings are interpreted correctly by others.
- **Always consult the standard when it doubt!**

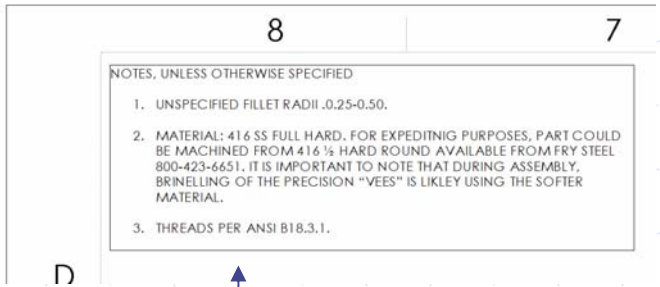
Drawing Standards – Sheet Formats

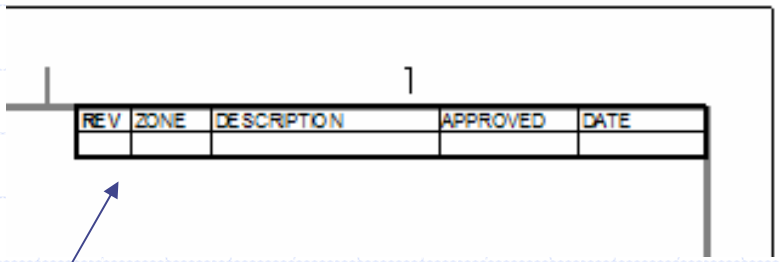
- There exist standardized sheet formats for creating engineering drawings.
- American National Standard
  - A - 8.5” x 11”
  - B – 11” x 17”
  - C – 17” x 22”
  - D – 22” x 34”
  - E – 34” x 44”
- International Standard ISO (mm)
  - A4 – 210 x 297
  - A3 – 297 x 420
  - A2 – 420 x 594
  - A1 – 594 x 841
  - A0 – 841 x 1189

Drawing Standards – Sheet Format Example C-Size



Drawing Standards – Sheet Formats





Revision Block

Drawing Notes

TEXT IS ALL CAPS! NO LOWER CASE.

Default Tolerance

Tolerance Block

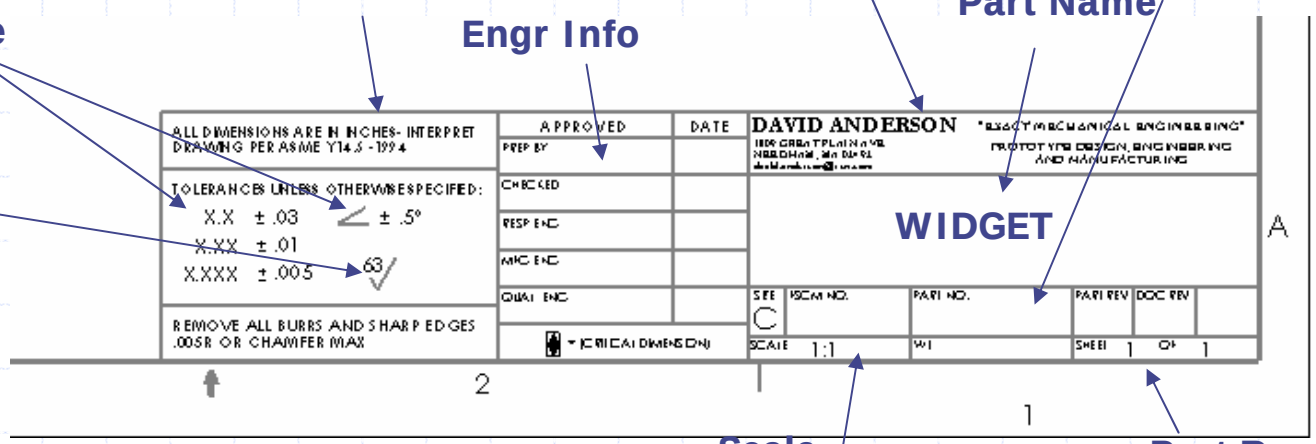
Engr Info

Company Name

Part #

Part Name

Default Surface Finish



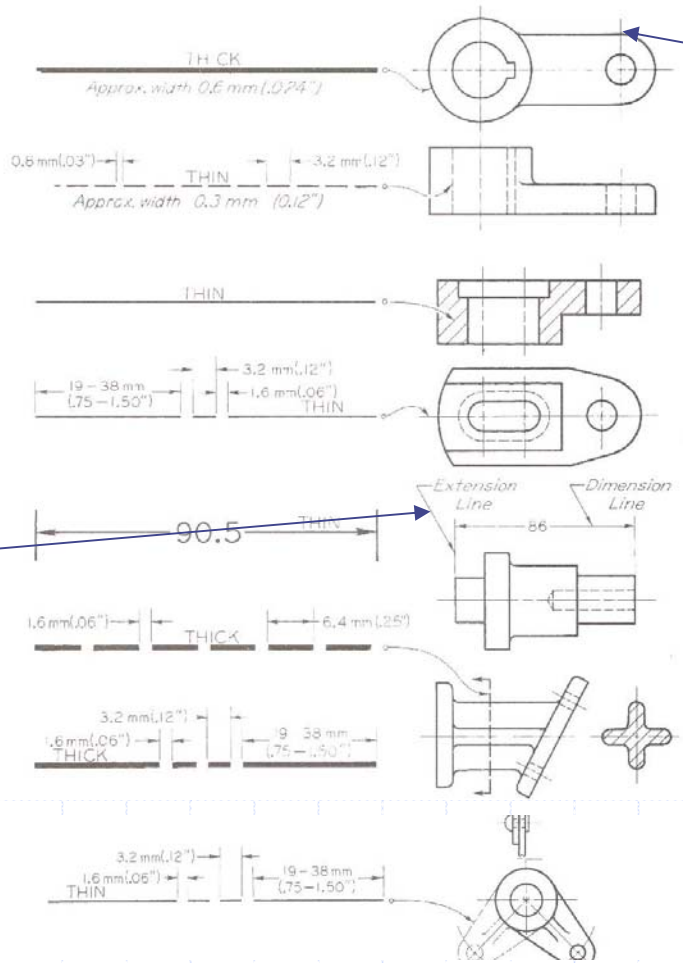
Scale

Part Rev

# of Shts

Drawing Standards - Line Types

- There exist many line types here are but a few...

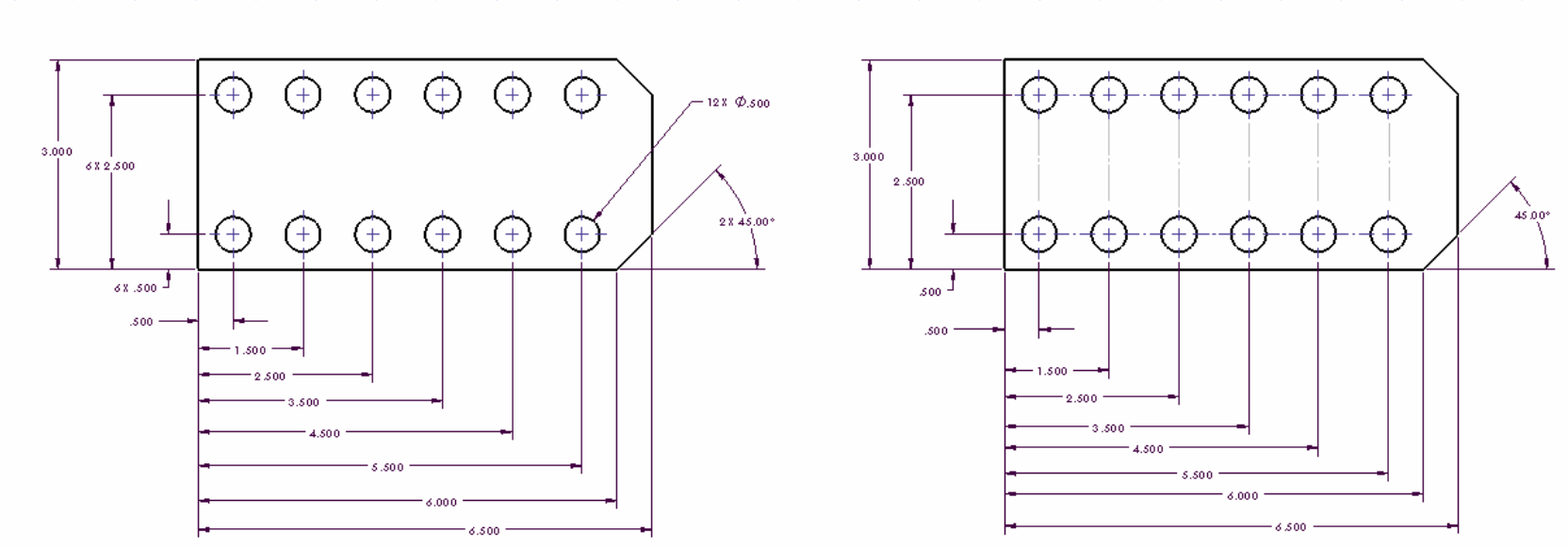


Drawing Standards - Dimensions

- There exist a number of dimension types
  - Linear
    - Coordinate Dimensions
    - Coordinate without dimension lines (Ordinate)
  - Angular
  - Radial/Diametrical
  - Tabular
  - Dimension Placement

Drawing Standards – Coordinate

Are these 2 drawings the same? **YES!**

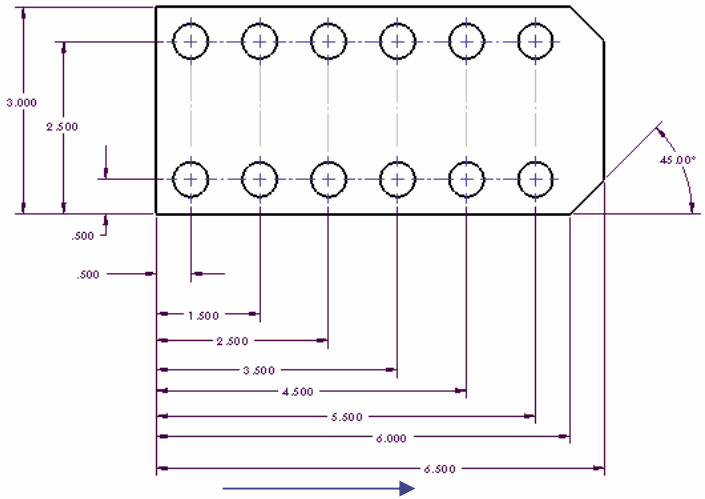


Which one would you rather detail?

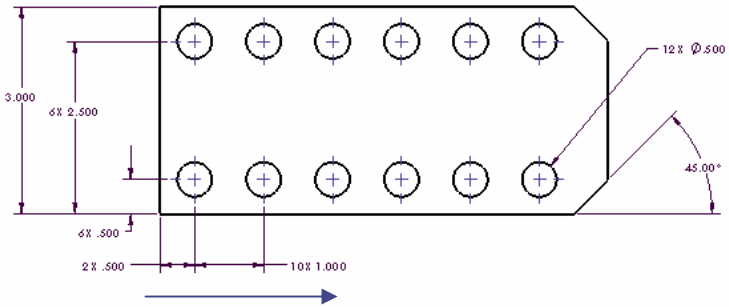
Which one would you rather make?

Drawing Standards – Coordinate

Are these 2 drawings the same? **NO!**



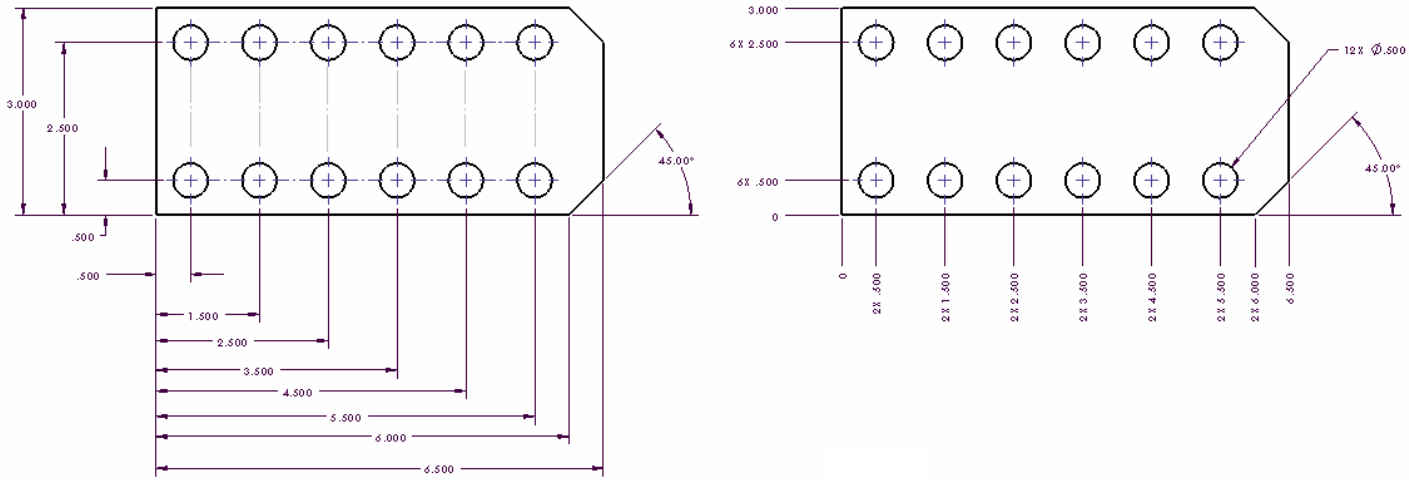
The hole-to-tolerance increases
The hole to edge tolerance is constant



The hole-to-tolerance is constant
The hole to edge tolerance increases

Drawing Standards – Ordinate

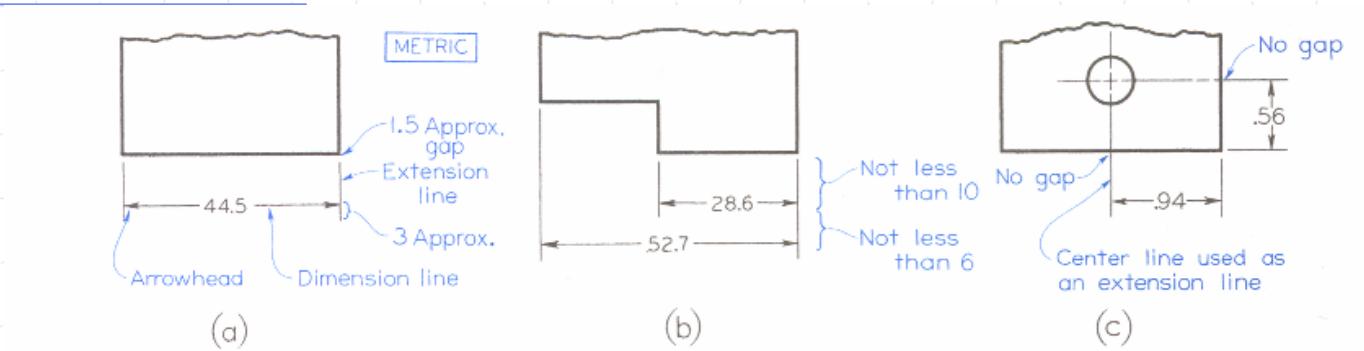
Are these 2 drawings the same? **YES!**

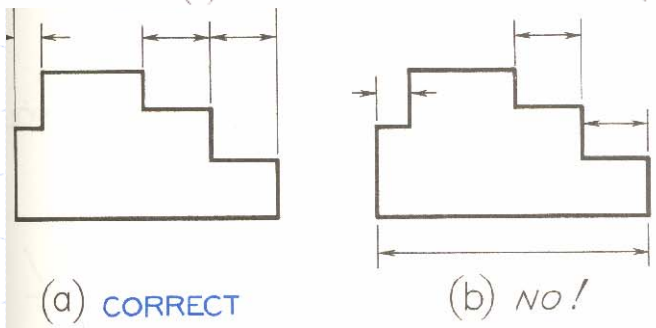


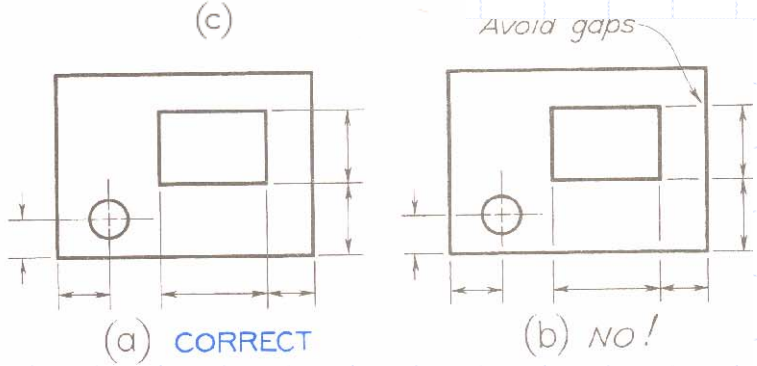
Which one would you rather detail?

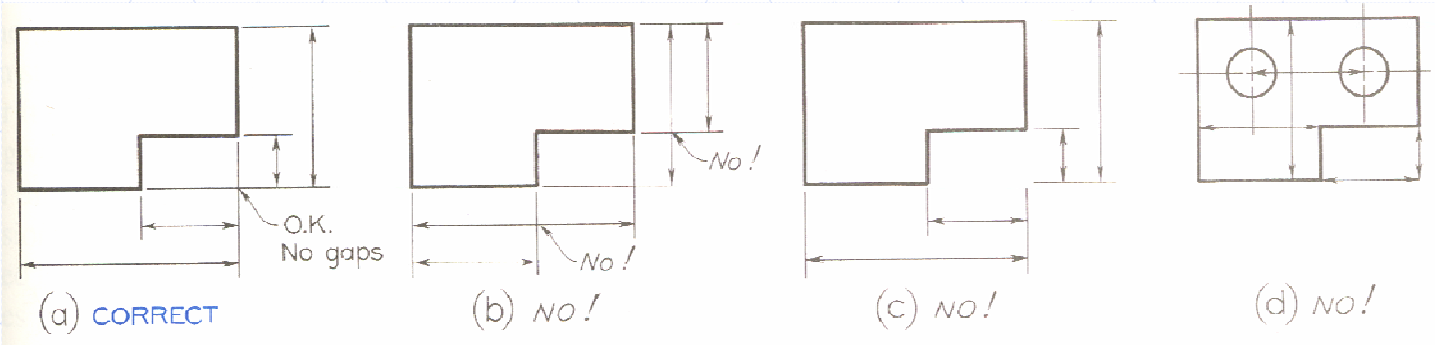
Which one would you rather make?

Drawing Standards – Proper Dimension Placement



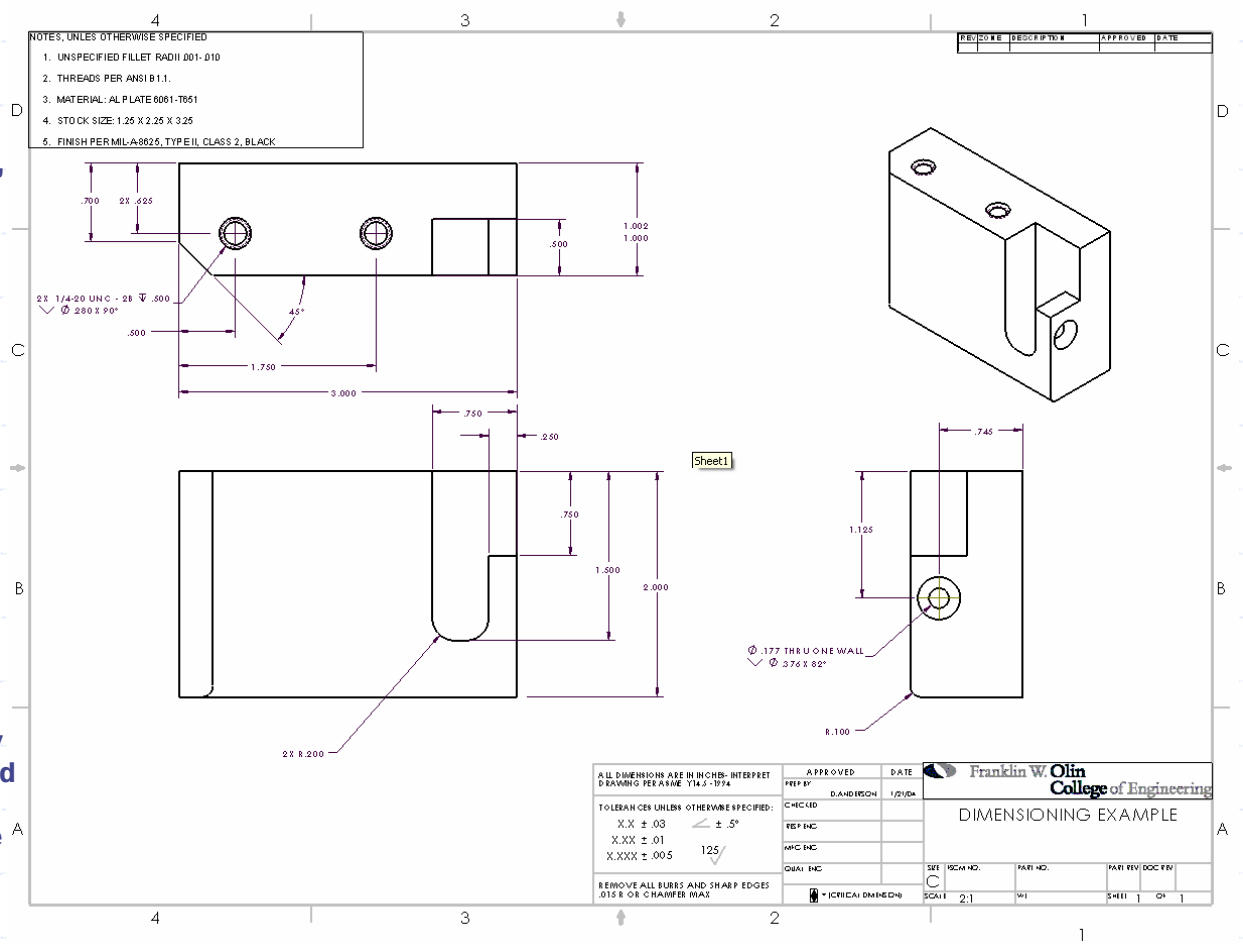




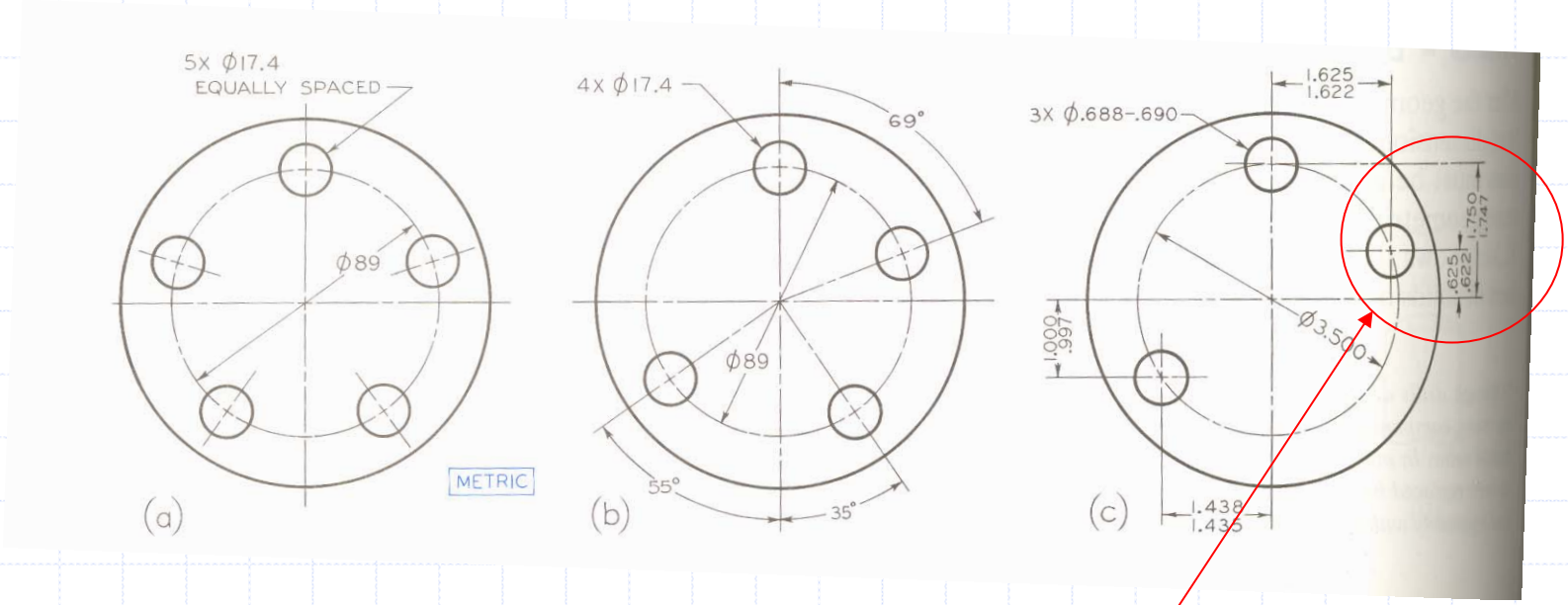


Drawing Standards – Dimensioning Rules

1. All CAPS!
2. All Decimals
3. Select a front view that best describes the part
4. Remove hidden lines always, unless absolutely necessary
5. Do not duplicate dimensions
6. Do not dimension to hidden lines
7. Place dims between views if possible
8. No dims allowed on body of part. Offset .38” inch from object outline
9. Place all dims for feature in one view if possible
10. Dim lines cannot cross dim lines
11. Dim lines should not cross extension lines
12. Extension lines can cross extension lines
13. Center marks in view(s) only where feature is dimensioned only
14. Centerlines in view(s) where feature is dimensioned

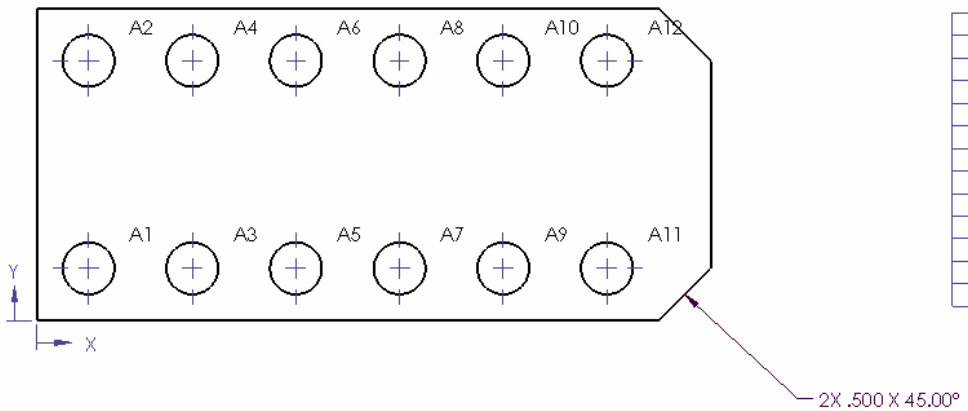


Drawing Standards – Bolt Holes



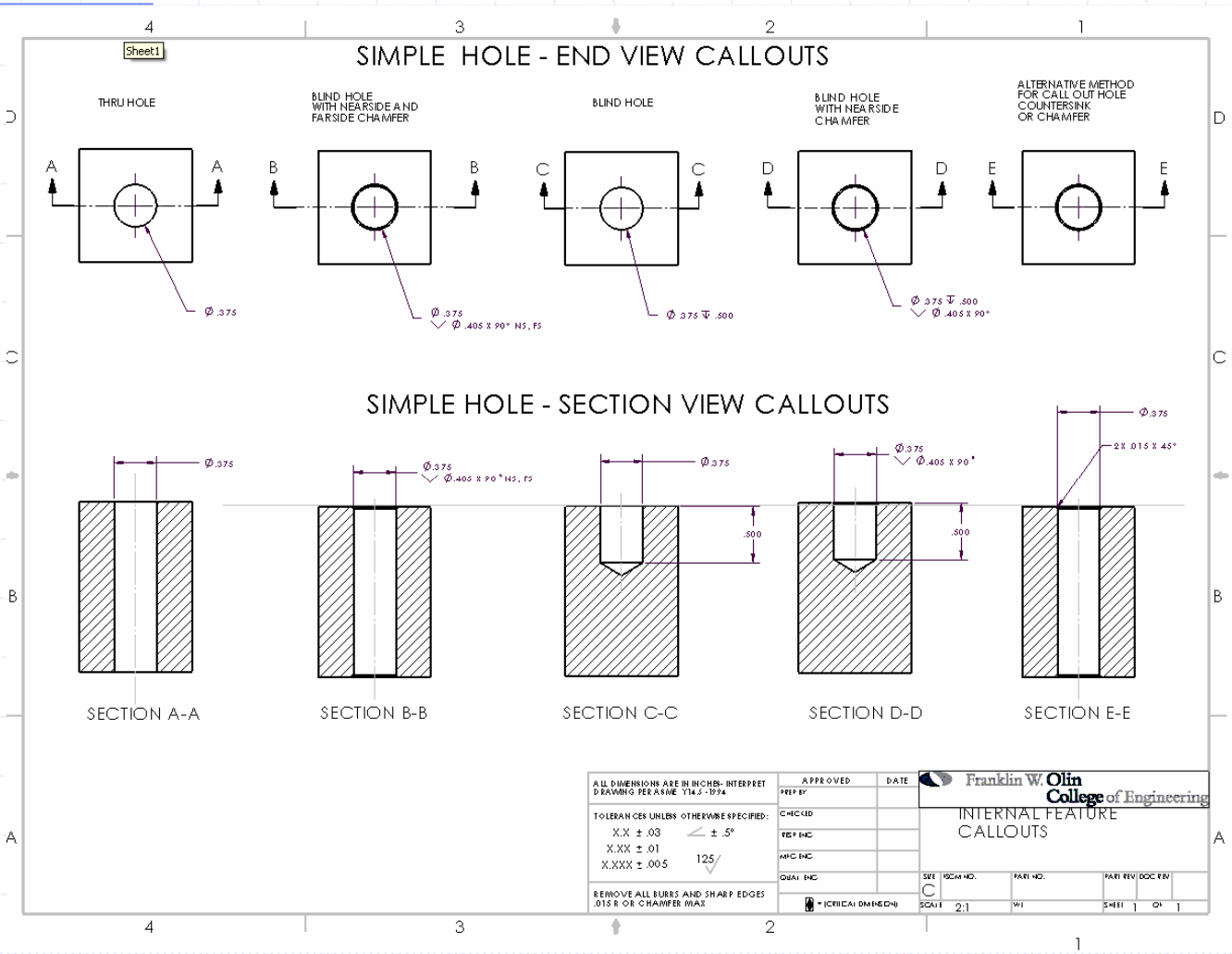
Poor practice, dims should all be horizontal

Drawing Standards – Hole Tables

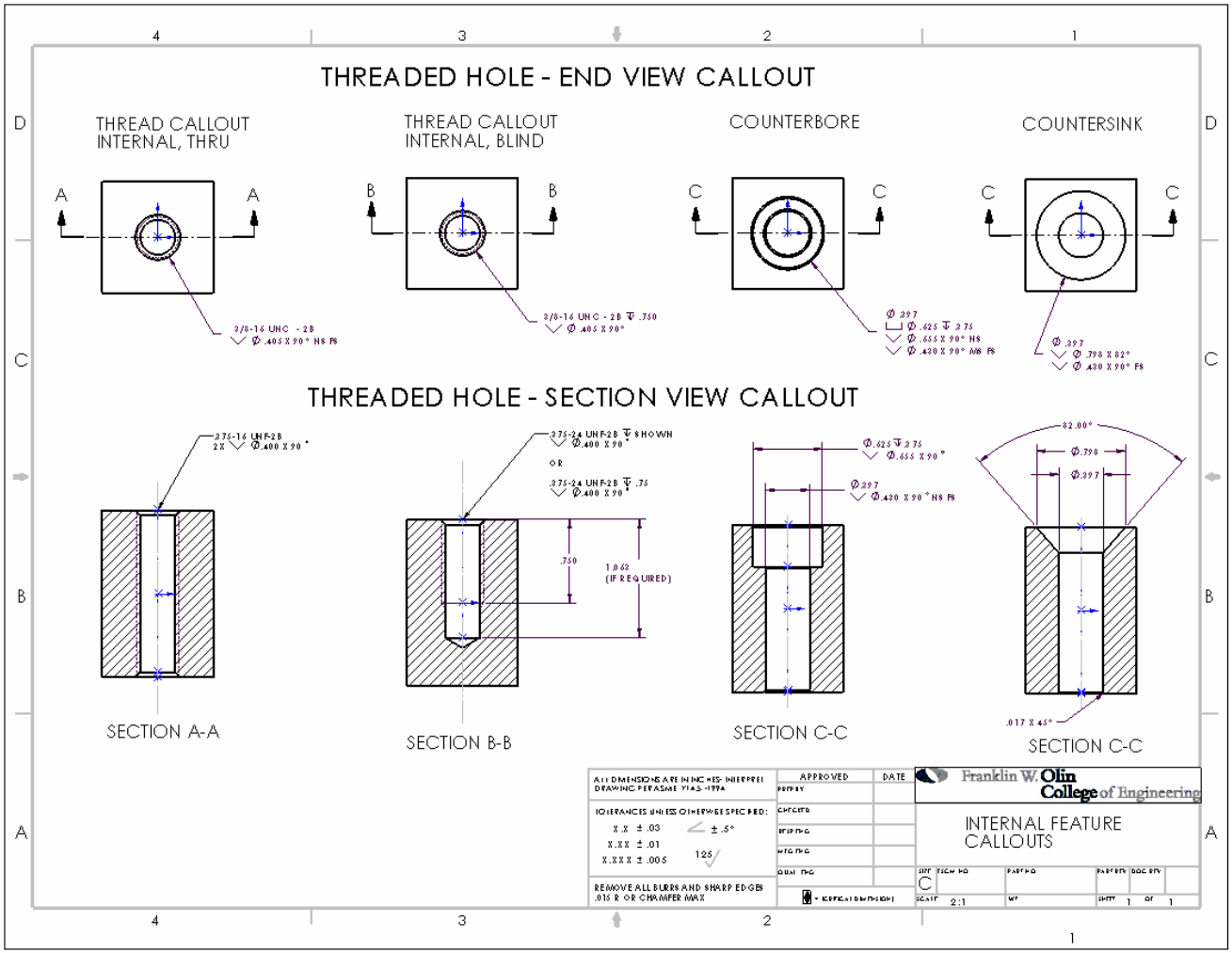


TAG	X LOC	Y LOC	SIZE
A1	.50	.50	Ø.500 THRU
A2	.50	2.50	Ø.500 THRU
A3	1.50	.50	Ø.500 THRU
A4	1.50	2.50	Ø.500 THRU
A5	2.50	.50	Ø.500 THRU
A6	2.50	2.50	Ø.500 THRU
A7	3.50	.50	Ø.500 THRU
A8	3.50	2.50	Ø.500 THRU
A9	4.50	.50	Ø.500 THRU
A10	4.50	2.50	Ø.500 THRU
A11	5.50	.50	Ø.500 THRU
A12	5.50	2.50	Ø.500 THRU

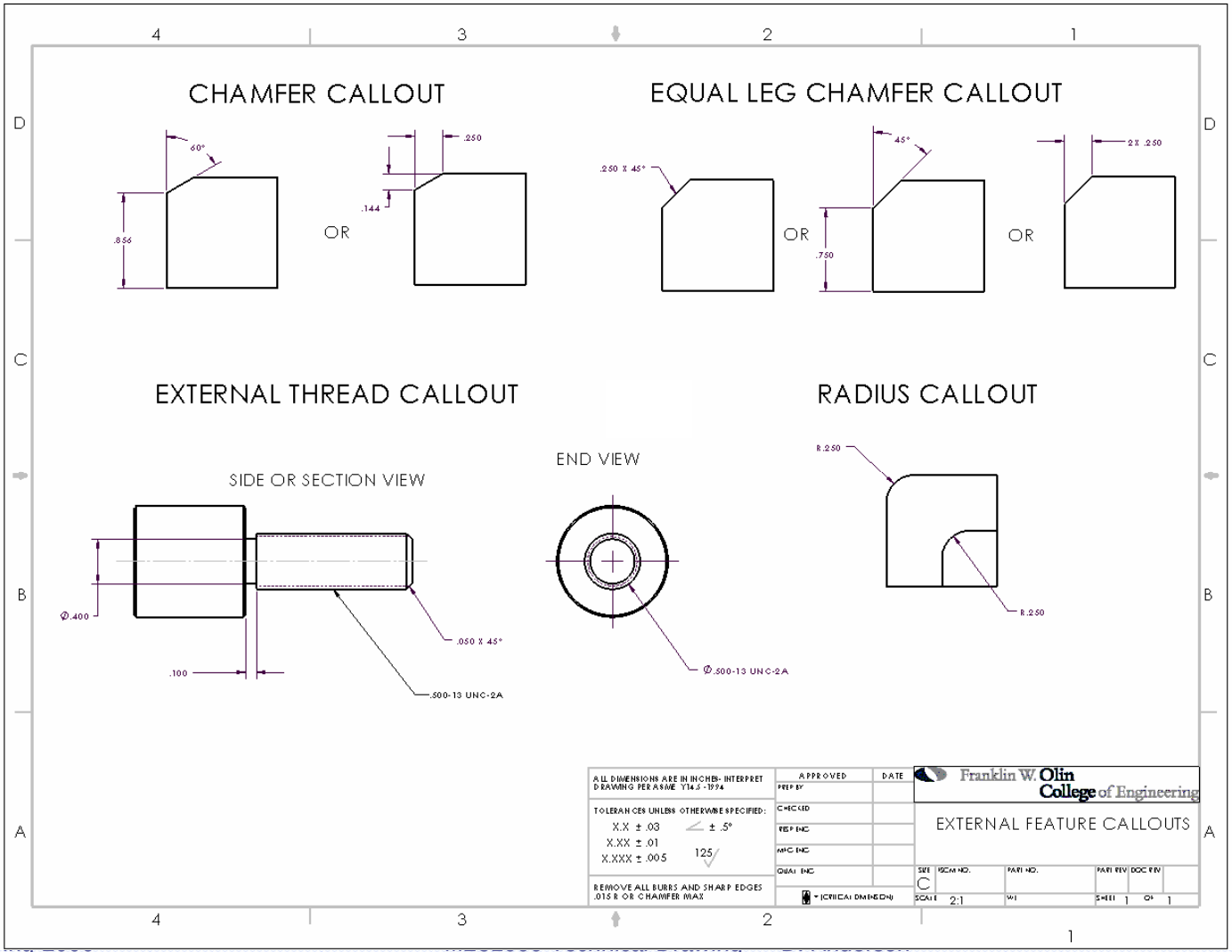
Drawing Standards – Hole Callouts



Drawing Standards – Threaded Hole Callouts



Drawing Standards – Misc Callouts



Best Practices/Basic Rules

1. All CAPS!
2. All Decimals
3. Select a front view that best describes the part
4. Remove hidden lines unless absolutely necessary to describe the shape of the object
5. Consider datums and dimensioning scheme based on
    1. Feature relationship
    2. Manufacturability and inspection
    3. Reduce math for machinist
6. Do not duplicate dimensions, use reference dims if necessary to duplicate
7. Do not dimension to hidden lines
8. Place dims between views if possible
9. No dims on body of part. Offset .38" inch from object outline
10. Place all dims for same feature in one view if possible
11. Dim lines cannot cross dim lines
12. Dim lines should not cross extension lines
13. Extension lines can cross extension lines
14. Use center marks in view(s) only where feature is dimensioned
15. Use centerlines and center marks in views only if feature is being dimensioned or referenced otherwise omit.
16. When multiples of the same feature exists in a view, dimension only one of the features and lable the dim as "NumberX" DIM meaning that the feature exists in that view"Number" times. For example, "4X .250" implies that in the view, there exists 4 like dimensions for the dimensioned feature
17. Minimize use of centerlines between holes etc, they add little value and clutter the object being drawn.

SolidWorks Custom Properties

DEMO!