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 known technically as a projection.
- A projection is a view conceived to be drawn or projected onto 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.

Object being sectioned

Section Line
Always a phantom line type

View Arrow
With Label

A

A

Sheet1

SECTION A-A
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
Section views can reduce 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 labeled.

Removed And scaled

Labeled and scale noted
Drawing Views – Broken-Out Section

Broken-out Section views are essentially partial section views without 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 auxiliary view is NOT behind The view arrows!

Broken out Section – No label necessary
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!

Removed partial section view
Labeled and scaled noted
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.
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 Y14.5M – 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

- Notes
- Revision Block
- Zone Identifiers
  This is zone “C4”
- Border
- Title Block
Drawing Standards – Sheet Formats

Notes, unless otherwise specified:
1. UNSPECIFIED FLEET RADIUS 0.250 ± 0.05
2. MATERIAL 4140 SS FULL HARD, FOR EXPEDITING PURPOSES, PART COULD BE MACHINED FROM 4163 HARD ROUND AVAILABLE FROM FRY STEEL 600-203-6651. IT IS IMPORTANT TO NOTE THAT DURING ASSEMBLY, BRINGING OF THE PRECISION "YESS" IS UNLILY USING THE SOFTIER MATERIAL
3. THREADS PER ANSI B32.3.1

Default Tolerance

Default Surface Finish

Tolerance Block

Engr Info

Company Name

Part Name

Part #

Widget

Scale

Part Rev

# of Sheets

Spring 2006 MEC1000 Technical Drawing - D. Anderson

TEXT IS ALL CAPS! NO LOWER CASE.
Drawing Standards - Line Types

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

Visible Line

Hidden Line

Section Line

Center Line

Dim & Extension

Leaders

Cutting Plane

Viewing Plane

Leaders

Center Mark
Drawing Standards - Dimensions

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

(a) CORRECT  
(b) NO!

(a) CORRECT  
(b) NO!

(a) CORRECT  
(b) NO!
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

<table>
<thead>
<tr>
<th>TAG</th>
<th>X LOC</th>
<th>Y LOC</th>
<th>SIZE</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>.50</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A2</td>
<td>.50</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A3</td>
<td>1.50</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A4</td>
<td>1.20</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A5</td>
<td>2.20</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A6</td>
<td>2.50</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A7</td>
<td>3.50</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A8</td>
<td>3.50</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A9</td>
<td>4.50</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A10</td>
<td>4.50</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A11</td>
<td>5.50</td>
<td>.30</td>
<td>Ø.500 THRU</td>
</tr>
<tr>
<td>A12</td>
<td>5.50</td>
<td>2.50</td>
<td>Ø.500 THRU</td>
</tr>
</tbody>
</table>

2X .500" X 45.00°
Drawing Standards – Hole Callouts

SIMPLE HOLE - END VIEW CALLOUTS

SIMPLE HOLE - SECTION VIEW CALLOUTS

SECTION A-A
SECTION B-B
SECTION C-C
SECTION D-D
SECTION E-E
Drawing Standards – Threaded Hole Callouts

THREADED HOLE - END VIEW CALLOUT

THREADED HOLE - SECTION VIEW CALLOUT

SECTION A-A
SECTION B-B
SECTION C-C
SECTION C-C

INTERNAL FEATURE CALLOUTS
Drawing Standards – Misc Callouts

- **Chamfer Callout**
- **Equal Leg Chamfer Callout**
- **External Thread Callout**
- **Radius Callout**

**External Feature Callouts**

- Dimensions and tolerances provided in the drawing.
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 label 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!