Reading Working Drawing Packets and Section Views

ENGR 1182
Graphics 06
Why study graphics?

- Technical drawings are another major form of technical communication
- Formal drawings and Working Drawing Packets are a means to communicate between the engineer and the manufacturer
Putting It All Together

- How does it apply to engineering in real life?
  - Reading technical drawings is one of the most important skills to learn from graphics
  - Hand sketching is important during the brainstorming process, usually precedes CAD modelling
  - Employers look for visualization skills
    - Tests during interviews at times
Today’s Objectives

- Identify the purpose and components of a formal drawing packet
- Be able to collect and deduce information from an assembly and its working drawing packet
- Describe the basic characteristics of section views
- Distinguish between different types of section views
- GP06 Out-of-Class Homework Assignment
Formal Drawing

- **Definition**: Detailed multi-view representations of a finished part

- **Components**:
  - Detailed orthographic views
  - Isometric view of part
  - Dimensions
    - Size and type of features (diameter, radius, etc.)
  - Title Block
Working Drawing Packets

- Components:
  - Assembly Drawings
    - Assembly of parts (parts already fit together)
    - Exploded assembly (show how parts fit together)
  - Detail drawings of non-standard parts
  - Bill of Materials
Title Block

- Specific to each organization

Components:
- Part name
- Scale
- Units
- Author
- Date
- Any other necessary info

- ISOMETRIC SCALE
- SYMMETRY
- GROUP RADIUS SIZING
Assembly Drawings

- Fully assembled object is shown with all required pieces
- Multi-view drawing
  - With appropriate section view, if applicable
Exploded Assembly

- Parts are “exploded”, showing how they connect and fit into the overall assembly.

Exploded Assembly Drawing - SOLIDWORKS Drawing available
Assembly Drawing with Bill of Materials (BOM)

- Bill of Materials
  - Part numbers in bubbles
  - Part descriptions
  - Material
  - Quantity

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<th>PART NUMBER</th>
<th>DESCRIPTION</th>
<th>MATERIAL</th>
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3 OPTIONS FOR AN OUTLINE ASSEMBLY DRAWING

all with

• An ASSEMBLED VIEW

• The BILL OF MATERIALS

• BALLOONS

MULTI-VIEW ➔
3 ORTHOGRAPHICS +
PART DRAWINGS FOR ALL NON-STANDARD PARTS

3 ORTHOGRAPHICS with DIMENSIONS +

OBJECT IS SYMMETRICAL LEFT TO RIGHT

ALL UNDIMENSIONED ROUNDS HAVE A RADIUS = .025

The Ohio State University
First Year Engineering

Dwg. Title: WIND TURBINE GENERATOR
Scale: 1:4
Inst. DR. WHO
Units: INCH
Dwg. No.: 04

Drawn By: DR. WHO
Hour: XX
Set: XX
Date: 06/13/16
Section View Drawings

- **Definition:** Views of parts with cutouts to show inner details

- **Components:**
  - 1. Cutting plane line
  - 2. Viewing Direction
  - 3. Cutout View (Section A-A)
    - Previously hidden features are now visible in the section view
    - Cut materials are cross hatched
Full Section Views

- Object is cut completely in *half* by a cutting plane perpendicular to the viewing plane.
- This will show the *full* interior of the part.
Half Section Views

- **Half** of the interior is visible by cutting out a part of the object
- The section view shows both the exterior and interior
Important Takeaways

- Section views reveal details of the interior of a part
- Working drawings are a major form of technical communication
  - Primarily between the engineer and the manufacturer
- There are several components common to all working drawings
2D Sketching

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SolidWorks 01
2D Sketches

Profiles:
Closed loop shape that is drawn on a flat 2D plane (referred to as a datum) and is used to create 3D objects.

2D Profiles consist of:
- Points
- Lines
- Circles
- Arcs
- Etc.
Orientation of Sketch

• Pick Starting Plane

• Top planes
  • Consistent shapes from top to bottom

• Front and Right planes
  • Consistent shapes from left to right
Design Sketching

(a) corners of rectangle specified by user

(b) rectangle corrected by software
Acceptable Profiles

- Closed Loop
- Created from Lines, Circles, and Arcs
- Nested Loops
- Errors
SolidWorks: Main Interface

The tool bar ribbon options will update automatically to match the current status of the model.
SolidWorks: Sketching

Basic Tools

Lines and Circles are the most important tools

Also important are Arcs and Fillets

Most objects can be created using these options

Tool Ribbon

The dropdown arrows next to each icon contain more options for each tool type
SolidWorks: Sketching
Advanced Tools

Trim can be used to remove excess lines

Offset can be used to duplicate a profile either inside or outside the original

These tool options are not required but can help reduce the time it takes to create a sketch
SolidWorks: Sketching

The Line tool will continue to start a new line from the previous end point until ESC is pressed.

Circles and Arcs can be added to the profile.
SolidWorks: Trimming

“Trim to closest” is the option that was used to remove part of the circle.

Another option for creating the arc is to start with a full circle and then trim away the excess material.
SolidWorks: Trouble Shooting

Extra lines can be deleted by left-clicking and using the delete key.

Processes can be reversed by the Undo Function (Ctrl Z) or a list of operations can be viewed using the undo menu at the top of the user interface.
SolidWorks: 2D Fillets

Adding a fillet shortens the two lines and the sharp corner is replaced by a rounded corner.

Fillets can be added to two lines that originally intersect.
SolidWorks: Exiting a Sketch

Two options are available for Exiting a Sketch and saving any changes made.
2D Sketching Wrap-Up

- Profiles
  - closed loop shape that is drawn on a flat 2D plane

- Main Tools
  - Circles
  - Lines

- Acceptable Profiles
  - Closed Loop

- Trim
  - removes excess of lines
Re-create the shape from the lecture slides on the FRONT plane. Focus on shape and proportionality not dimensions (the fillet was chosen because of the relative size of the members, not for an exact dimension).
SolidWorks Files:
- Saving as PDFs or 3D PDFs
- Creating and Manipulating 3D PDFs

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2D Sketching
Saving SolidWorks Files as PDFs to Produce Part Image + Model Tree For Activities/Application Assignments and Exams

STEP 1 - Maximize the SolidWorks screen and then hit the Print Scrn key and paste image into a WORD document. Exams use only STEPS 1 & 4

STEP 2 – OPTIONAL* - Click on the image and then select PICTURE TOOLS FORMAT/Wrap Text/Square. Click again on Wrap Text/In Front of Text.

STEP 3 – OPTIONAL* - Enlarge the Image by using the mouse to pull on a corner of the Image and adjust the position

STEP 4 - Click on File/Save As and in the drop down list under Save as type select Adobe PDF.

OPTIONAL STEPS 2 & 3 illustrate how to make an anchored picture larger by releasing the positional constraints and ARE NOT REQUIRED on EXAMS

To Produce the PDF Part Image + Tree
Saving SolidWorks Files as 3D PDFs

Click on File/Print

In the drop down list under NAME select Adobe PDF

Select #D Printing and then OK and Save

To Produce the 3D PDF Part Image

Note that the 3D PDF can be manipulated by grabbing the colored arrows at the bottom left of the Image
What’s Next?

- Midterm 1
- SolidWorks
  - Extrude
  - Revolve