Exploded View
Extracting Drawings
ENGR 1182
SolidWorks 08
Today’s Objectives

- Formal Drawing Components:
  - Exploded View
  - Extracted Drawings

- SW07 In-Class Activity

- SW07 Out-of-Class Homework Assignment
Formal Drawings

Definition: First page of a **Working Drawing Set** – Later in SW-10

Formal Drawing Components:

1. **Extracted Drawings**
   - Exploded View
   - Bill of Materials
   - Balloons

   Later in SW-10
Formal Drawings

Definition: Detailed multi-view representations of a finished part

Formal Drawing Components:

1. **Extracted Drawings**
   - Extracted Views
   - Detailed Features
   - Title Block
Exploded View

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Converting an Assembly into an Exploded View
Converting an Assembly into an Exploded View

1. After creating a properly constrained assembly drawing, select Assembly/Exploded View

2. Note the Instructions in the upper pane of the drop down table

Converting an Assembly into an Exploded View

4. By holding the CTRL key down and left clicking on each part, select the two parts (Propeller and Generator) which are to be moved vertically to separate them from the Base and the Tower. The selection turns the parts BLUE.

5. Note that the arrow controlling the vertical movement is GREEN since it has not yet been selected.
Converting an Assembly into an Exploded View

6. Select the vertical arrow by left clicking on it WITHOUT releasing the mouse button. Note that the color changes to ORANGE indicating selection.

7. With the mouse button depressed, pull the selected parts upward as indicated. Release the mouse button to lock the parts in the desired position.
Converting an Assembly into an Exploded View

8. Repeat the procedure to adjust the position of the Tower.

9. After the tower has been locked in position, accept the Exploded View by clicking on the green arrow. The Exploded View can now be saved and later “Extracted” into a Drawing as described in the following slides.
Extracting Drawings

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Extracted Views

**Purpose:** create 2D engineering drawings from solid models

- Part File to Drawing File
- Place Base View
- For Multiview drawings:
  - Project Top, Right
  - Isometric View
- For Exploded Views, usually only the shaded Isometric is used
Detailed Features

• Automatically Aligned Features

• Hidden Lines
  • Orthographic Views
  • NOT on Isometric Views (unless necessary)

• Circular Features
  • Centerlines
  • Centermarks
Title Block

- Specific to Company
- Part Name
- Scale
- Units
- Author
- Date
- Any Other Necessary Information
A pre-made drawing sheet with an OSU title block is found on the website under content. Download this file and file out your name, instructor, etc. and save the file as:

**OSU_TitleBlock_Initials**

Open this file and always use “Save As” in order to keep the file for repeated use.
SolidWorks: Drawing File

Open a new Drawing file and use the “Model View” button to place a base view.

Select the correct part file from open documents or using the “Browse..” button to select closed files.
SolidWorks: Model View Options

Choose base view (usually front view)

Display hidden lines for orthographic views

Choose appropriate scale using the sheet scale or a custom option
SolidWorks: Extracted Views

Insert base view by left clicking once at the correct location

Place projected views by left clicking in the appropriate directions and spacing

Press “Esc” when finished
SolidWorks: Changing Options

The size of all the views can be changed by selecting the base view and changing the scale.

The isometric view can be selected in order to change its display properties to a solid part without edge lines.
Center Marks and Centerlines must be added to define circular features in Orthographic views.
SolidWorks: Title Block

Right click and select “Edit Sheet Format”

Note: The views in the main space will disappear

When finished right click and select “Edit Sheet” to see the views of the object again

Double click text to edit
Extracted Views
  • Part file to drawing file

Detailed Features
  • Aligned
  • Hidden
  • Circular

Title Block
  • Author
  • Units
  • Scale

Homework Assignment SW07-OUT:
Extract an Exploded View of OSU Phone (SW-06)
Important Takeaways

- Extracted drawings are used to show 3D parts as 2D drawings or to create an Exploded View as part of a set of Working Drawing. – see SW-10

- Titles blocks are used for identification and informative purposes.
What’s Next?

- Due Next Class: SW07 Out of Class HW
- Before next class, you will read about dimensioning in detail.
- Take SolidWorks 8 Quiz