Extra Credit Advanced Assemblies
ENGR 1182
Solidworks 7
Advanced Assemblies In The Real World

Engineers at companies like Honda build a comprehensive model of each new vehicle design so that each design group can verify the fit of their components in the completed product.
Today’s Objectives

- Identify relationship of advanced assemblies to manufacturing
- Understand how to apply materials
- Putting it all together – using all techniques learned up to this point
- SW07X Out-of-Class Assignment
Advanced Assemblies in Real World Applications

- Solidworks or solid modeling in general is used in many disciplines
  - Aerospace Engineering
  - Biomedical Engineering
  - Civil Engineering
  - Electrical Engineering
  - Mechanical Engineering
Material Properties

- All parts created in SolidWorks can be identified with a material type with known material properties.

- This allows the designer to know the estimated weight and strength of parts created by using the analysis tools.

- This is useful in manufacturing where many different methods are used for the different materials.
To Restore Material in Model Tree:

- In SW_01 your instructor may have had you hide the “Material” display in the Model Tree using the following instructions:
  
  Right click anywhere in the Model Tree area and select "Hide /Show Tree Items". Change the values as indicated below to hide the Sensor, Annotations and Material fields in the Model Tree. This setting will be maintained in all future SolidWorks usage.

- If so, to restore the display of “Material” change “Automatic” back to “Show” and click “OK”.

![Diagram of Model Tree settings change](image)
SolidWorks: Applying Materials

In the Model Tree:
Right Click on Material and select “Edit Material”

Choose the correct Material and click Apply
Putting it All Together

- This topic will put all the concepts learned in CAD up to this point together.
- You as the designer will be able to create 4-5 individual parts and then put them together into one assembly.
SW07 Assignment Options

- Landing Gear
- Replacement Hip
- Aerospace
- Biomedical
- Replacement Hip
- Mechanical
SW07 Assignment Options

Monorail Support

Electromagnetic Brake

Civil

Electrical
Advanced Assemblies Assignment

- Model the parts in your selected discipline and create the assembly
- Check the assignment sheet for information on how to submit the assignment
- Follow the guides in assignment document for creating the parts
Important Takeaways

- CAD is used in many disciplines for real-world engineering applications
- Materials are a helpful addition to CAD models because it allows for a more useful model
What’s Next?

▪ Due SW-09 Class: SW07X Out-of-Class HW

▪ Extracting Drawings
  ▪ Final file type for SolidWorks

▪ Basic Dimensioning
  ▪ Overall Size
  ▪ Size and Location of All Features

▪ Take SolidWorks 8 Quiz on readings
Advanced Assembly Techniques
For Student Review
ENGR 1182
Solidworks 7
Advanced Assemblies Techniques

- How should we constrain the plate to the center of the bar?
  - Calculate half length of bar
  - Add half width of plate
  - Do distance constraint from edge of both

- Is there a better way?
Advanced Technique – Width Constraint

- Width constraints in Solidworks space a component equally from two faces on another part.
  - Can we use that on the previous example?

- Requirements
  - Two width faces
  - Two tab faces
  - Which part is used for each set of faces is arbitrary
    - Plate could be tabs or width faces and vice versa for bar
Advanced Technique – Width Constraint

1. Select mate tab
2. Click on “Advanced Mates”
3. Select “Width”
   a. Width selection will be plate
   b. Tab selection will be bar

Don’t forget that these can be reversed and the effect is the same.
Advanced Technique – Width Constraint

- Plate is equally spaced or centered on the bar
- Note other advanced mates such as the symmetric mate or linear/linear coupler mate
- Width constraints are the most common advanced constraint
Other Advanced Techniques

- Construction lines can be used to constrain two parts together when a concentric mate won’t work
  - Ex: constraining a spring to a cylinder

- Concentric constraint cannot be applied to bar and spring shown.
  - There is no circular feature in a single plane on the spring that can be used for a concentric mate

- Solution?
Other Advanced Techniques

- Construction line through center of spring used for concentric mate.

- To select construction line, right click sketch in part file and select “show”
  - Also make sure to select line close to part otherwise line will not highlight or be selected during mate window