In-Class Activity 7 - Solidworks Assemblies

ME 3001-002- Mechanical Engineering Analysis - Fall 2021

Overview:

You will develop your mini-project design in CAD and begin to construct a Solidworks assembly.

Learning Objectives:

- Practice creating fully-defining 2D sketches
- Practice using extrude and revolve to create 3D features
- Learn to produce assemblies from multiple part files using mates

Required Materials:

- **Computer**: This activity requires a computer with Solidworks installed.

Activity:

1. Choose at least two different parts from your mini design project. Create a representative part model for each piece in Solidworks and save the part file as **<USERNAME>_partA.sldprt** and **<USERNAME>_partB.sldprt**. Additional parts may be included.
2. Create a Solidworks assembly from the individual parts. Click *New -> Assembly* to open a new assembly and save the file as **<USERNAME>_assemblyAB.sldasm**.
3. Use different *mates* to fully define the position of the parts in your assembly. If there are moving parts (not required), there must be at least one part that is fixed.
4. *If time allows*, save a copy of your work and begin to make revisions for improvement. Alternatively, begin working on the design of the remaining components of the project.

Submit:

Submit the most complete version of the following files to the Activity 7 ilearn folder before the posted due date. Additional parts may be included.

- **<USERNAME>_partA.sldprt**
- **<USERNAME>_partB.sldprt**
- **<USERNAME>_assemblyAB.sldasm**