Western Technical College

10606217 Solid Model Use in Manufacturing 1 (CBE)

Course Outcome Summary

Course Information

Description	Requires the learner to develop fundamental solid modeling skills to create SolidWorks sketches and a parametric model.
Career Cluster	Manufacturing
Instructional Level	One-Year Technical Diploma
Total Credits	1
Total Hours	36

Textbooks

No textbook required.

Learner Supplies

Safety glasses with side eye protection that meet Z87 OSHA guidelines. **Vendor:** Campus Shop. Required.

Proper footwear - $35.00-75.00. **Vendor:** To be discussed in class. Required.

Scientific calculator (recommend T1-36x Solar). **Vendor:** Campus Shop. Required.

Three-ring binder. **Vendor:** Campus Shop. Required.

Clipboard. **Vendor:** Campus Shop. Required.

Pens/Pencils/Black Sharpie Marker. **Vendor:** Campus Shop. Required.

Minimum 4GB USB Flash Drive. **Vendor:** Campus Shop. Required.

Success Abilities

1. Live Responsibly: Embrace Sustainability
2. Refine Professionalism: Participate Collaboratively

Program Outcomes

1. Interpret industrial/engineering drawings.

Course Competencies

1. Investigate the SolidWorks user interface.

Assessment Strategies

1.1. Demonstration

Criteria

You will know you are successful when

1.1. you distinguish and manipulate the various components of the Graphical User Interface utilized in the software and operating system.
1.2. you apply technology to task: enters, modifies, retrieves, stores, and verifies data and other information in a computer.
1.3. you employ computers to acquire, organize, and communicate information.
1.4. you start and setup a Solidworks Session.

Learning Objectives

1.a. Discuss the advantages of using a parametric design software.
1.b. Examine the SolidWorks user interface.
1.c. Open, close, and save SolidWorks documents.
1.d. Identify dialog boxes at start-up.
1.e. Select proper template.

2. Create file folders for project and template files.

Assessment Strategies

2.1. Demonstration

Criteria

You will know you are successful when

2.1. you demonstrate good file management techniques.
2.2. you save part models, assemblies and drawings in proper project folders.
2.3. you import and/or export files from Solidworks.

Learning Objectives

2.a. Create file folders to manage projects.
2.b. List the types of files and file extensions in SolidWorks.
2.c. Save documents to multiple locations: hard drive, network pc, USB.
2.d. Explore the file format options for importing/exporting in SolidWorks i.e. IGES, STEP, STL, DWG,
2.e. Modify System Options to include different file search paths.

3. Apply Document Properties to files.

Assessment Strategies

3.1. Demonstration

Criteria

You will know you are successful when

3.1. you use proper commands to set options and properties.
3.2. you create a part/drawing template.
3.3. you create and link custom properties in the template.

Learning Objectives

3.a. Change document properties.
3.b. Set-up system options.
3.c. Configure the Toolbox for required standards including die companies.
3.d. Create a part/drawing template.
3.e. Create and link custom properties in the template folder.

4. Create a SolidWorks sketch.

Assessment Strategies

4.1. Demonstration

Criteria

You will know you are successful when

4.1. you choose the appropriate plane to start the sketch.
4.2. you determine the geometry required to sketch parts.
4.3. you fully define sketches to the correct size and shape using geometric relations and dimensional values.

Learning Objectives

4.a. Identify sketch entities and their icons.
4.b. Select the proper sketch plane to begin a part model.
4.c. Utilize rectangle, circle, arc, and line, trim, convert entities, offset and mirror sketch tools.
4.d. Apply the following geometric relationships to a sketch: horizontal, vertical, collinear, parallel, perpendicular, equal.
4.e. Make a linear pattern of sketch entities.
4.f. Explain the difference between Mirror and Dynamic Mirror when in sketch mode.
4.g. Use relations and dimensions to maintain design intent and fully define sketches.

5. Create a SolidWorks 3D parametric model.

Assessment Strategies

5.1. Demonstration

Criteria

You will know you are successful when

5.1. you create 3D parametric models from profiles.
5.2. you add features to 3D parametric models.
5.3. you view and manipulate 3D parametric models.
5.4. you create a multiple configuration part file using Design Tables.

Learning Objectives

5.a. Know options of each command.
5.b. Extrude a profile to create a 3D part.
5.c. Revolve a profile to create a 3D part.
5.d. Use the Sweep command to create 3D part.
5.e. Create and solve a 3D path.
5.f. Create and utilize reference geometry, planes, axis, etc.
5.g. View and rotate the 3D object using viewing and shading commands.
5.h. Edit features using the Browser or Feature Manager Tree.
5.i. Create multiple configurations of a part model.
5.j. Update or rebuild model.
5.k. Apply materials to components.