

Western Technical College

31606320 Solid Model Use for Manufacturing

Course Outcome Summary

Course Information

| | |
|----------------------------|--|
| Description | This course will develop basic solid modeling skills for manufacturing program students. Students will understand the basics of model design and creation and be able to interrogate part solid models to obtain needed feature information. |
| Career Cluster | Science, Technology, Engineering and Mathematics |
| Instructional Level | Technical Diploma Courses |
| Total Credits | 2 |
| Total Hours | 72 |

Textbooks

No textbook required.

Learner Supplies

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

Proper work boots - \$35.00-75.00. **Vendor:** To be discussed in class. Required.

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

Success Abilities

1. Apply mathematical concepts.
2. Demonstrate ability to think critically.
3. Demonstrate ability to value self and work ethically with others in a diverse population.
4. Use effective communication skills.
5. Use technology effectively.

Program Outcomes

1. MACH 2. Interpret industrial/engineering drawings

Course Competencies

1. Investigate the SolidWorks user interface.

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. Start and setup a Solidworks Session.

Learning Objectives

- 1.a. Discuss the advantages of using a parametric design software.
- 1.b. Investigate the SolidWorks user interface.
- 1.c. Explore the file format options for importing/exporting in SolidWorks i.e. IGES, STEP, STL, DWG, etc
- 1.d. Open, close, and save SolidWorks documents.
- 1.e. Identify dialog boxes at start-up.
- 1.f. Select proper template.

2. Create file folders for course and project files.

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.

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.

3. Apply System Options and Document Properties to part, assembly and drawing files.

Criteria

You will know you are successful when

- 3.1. you use proper commands to set options and properties.

Learning Objectives

- 3.a. Change document properties.
- 3.b. Set-up system options.
- 3.c. Activate the Toolbox and Toolbox Browser.
- 3.d. Configure the Toolbox for required standards including die companies.

4. Create a SolidWorks sketch.

Criteria

You will know you are successful when

- 4.1. you determine the geometry required to sketch parts.
- 4.2. you manipulate the software to produce a sketch outline (profile).
- 4.3. you fully constrain 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.

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.

Learning Objectives

- 5.a. Know options of each command.
- 5.b. Determine method of modeling to be used to create 3D part.
- 5.c. Extrude a profile to create a 3D part.
- 5.d. Revolve a profile to create a 3D part.
- 5.e. Use the Sweep command to create 3D part.
- 5.f. Create and solve a 2D path.
- 5.g. Create and solve a 3D path.
- 5.h. Create and utilize reference geometry, planes, axis, etc.
- 5.i. Utilize the following features: extrude-base, extrude-cut, extrude-boss, fillet, mirror, hole wizard, chamfer, and linear pattern.
- 5.j. View and rotate the 3D object using viewing and shading commands.
- 5.k. Edit features using the Browser or Feature Manager Tree.
- 5.l. Specify components from the SolidWorks Toolbox.
- 5.m. Create multiple configurations of a part model.
- 5.n. Fully constrain component parts.
- 5.o. Update or rebuild model.
- 5.p. Apply materials to components.

6. Create a SolidWorks assembly.

Criteria

You will know you are successful when

- 6.1. you produce an assembly of components models using correct mates and procedures.
- 6.2. you produce a model within an assembly (top down).

Learning Objectives

- 6.a. Differentiate between bottom up and top down assemblies.
- 6.b. Bring component parts into an Assembly file.
- 6.c. Analyze relationships between parts.
- 6.d. Differentiate types of constraints and conditions.
- 6.e. Use Browser or Feature-Manager Tree to list and control parts and assemblies.
- 6.f. Place components together using assembly mates and conditions.
- 6.g. Use advanced mates (limit) and mechanical mates (rack/pinion and hinge)
- 6.h. Edit mates.
- 6.i. Control the visibility of a part.
- 6.j. Display or hide work features of the current part, work objects and parts of the assembly.
- 6.k. Differentiate between six degrees of freedom: translational and rotational.
- 6.l. Create a new part based on existing parts. (top down assembly)
- 6.m. Check for interference.
- 6.n. Download component parts from the Intranet/Internet.
- 6.o. Simulate model motion.

7. Obtain needed feature and dimensional information from a part model or assembly.

Criteria

You will know you are successful when

- 7.1. you manipulate the model or assembly to gather information.

Learning Objectives

- 7.a. Utilize MEASURE tools.
- 7.b. Utilize DIMENSIONING tools.
- 7.c. Evaluate the model or assembly using various commands and techniques.
- 7.d. Obtain the Mass Properties of a model.

7.e. Explore the options under the Command Manager's EVALUATE tab.

8. Differentiate between dimensioning and tolerancing methods and understand the affect of each.

Criteria

You will know you are successful when

- 8.1. you compare and utilize different types of dimensioning and tolerancing methods.
- 8.2. you prepare correctly dimensioned drawings of part models.

Learning Objectives

- 8.a. Interpret and use ANSI Y14.5 drafting standards and symbology on all drawing layouts.
- 8.b. Identify dimensioning terminology.
- 8.c. Evaluate dimensioning systems.
- 8.d. Dimension a series of working drawings based on a required engineering function of the part.
- 8.e. Differentiate between unilateral, bilateral tolerancing and limits.
- 8.f. Interpret tolerancing methods.

9. Create a SolidWorks drawing from parts and assemblies.

Criteria

You will know you are successful when

- 9.1. you create all working drawings required in a project.

Learning Objectives

- 9.a. Set up a drawing layout for a sheet size and plotter device.
- 9.b. Identify types of views required to define part.
- 9.c. Insert standard views of a part model.
- 9.d. Create detail, exploded and section views.
- 9.e. Manipulate views (change, delete, move).
- 9.f. Hide and unhide drawing features.
- 9.g. Hide and move parametric dimensions.
- 9.h. Set up dimension styles.
- 9.i. Add reference dimensions.
- 9.j. Edit parametric and reference dimensions.
- 9.k. Use tolerance modeling.
- 9.l. Add titleblock, notes and other annotations.
- 9.m. Add centerlines.
- 9.n. Add symbols.
- 9.o. Add balloons and insert a parts list.
- 9.p. Edit balloons and parts list.
- 9.q. Create additional drawing sheets.