Western Technical College

10606184 Solidworks

Course Outcome Summary

Course Information

Description	Introduction to Solid Works 3D parametric modeling software. Create 3D parts and use these 3D parts to create 3D assemblies and 2D drawings. Students will learn to preserve design intent using dimension-driven systems and geometric relationships.
Career Cluster	Science, Technology, Engineering and Mathematics
Instructional Level	Associate Degree Courses
Total Credits	2
Total Hours	72

Textbooks

No textbook required.

Learner Supplies

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

The software utilized in this course requires a Windows-based PC and Internet.

Success Abilities

1. Cultivate Passion: Expand a Growth-Mindset
2. Cultivate Passion: Increase Self-Awareness
3. Live Responsibly: Embrace Sustainability
4. Refine Professionalism: Act Ethically
5. Refine Professionalism: Improve Critical Thinking

Program Outcomes

1. Prepare detail and assembly drawings for documentation of mechanical components and products.
2. Create CAD geometry, parts, and assemblies.

3. Analyze mechanic engineering problems.
4. Demonstrate awareness of product liability and industry standards.
5. Meet design and production deadlines.
6. Interpret and communicate technical concepts, designs, and documentation.

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 create a part/drawing template.

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. Create a part/drawing template.
3.c. Set up system options.
3.d. Create and link custom properties in the template file.
3.e. Save a template to the correct 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 Sketch entities - lines, rectangles, circles, arcs, ellipses, centerlines and Sketch tools - offset, convert, trim, Dynamic Mirror
4.d. Apply the following geometric relationships to a sketch: horizontal, vertical, coincident, colinear, concentric, parallel, perpendicular, equal, tangent
4.e. Make Linear, circular, and fill patterns
4.f. Use constraints 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.
5.5. you have the ability to read orthographic projections
5.6. you have the ability to read dimensions

Learning Objectives

5.a. Create Boss and cut features - extrudes, revolves, sweeps, lofts
5.b. View and rotate the 3D object using viewing and shading commands.
5.c. Edit features using the Feature Manager Tree.
5.d. Apply materials to components.
5.e. Utilize Placed features - Fillets and chamfers, Linear, circular, and fill patterns
5.f. Make use of Feature conditions – start and end
5.g. Create Reference geometry – planes, axis, points
5.h. Create multiple configurations of a part model.

6. Create a SolidWorks assembly.

Assessment Strategies

6.1. Demonstration

Criteria

You will know you are successful when

6.1. you explain the difference between bottom up and top down assemblies.
6.2. you produce an assembly of parts and/or sub-assemblies using correct Mates and procedures.
6.3. you produce a model within an assembly (top down).
6.4. you edit Mates.

6.5. you add standard components to an assembly from the Toolbox.

Learning Objectives

6.a. Differentiate between bottom up and top down assemblies.
6.b. Insert components into an assembly
6.c. Place Standard mates - coincident, parallel, perpendicular, tangent, concentric, distance, angle
6.d. Use advanced Mates (limit) and mechanical Mates (rack/pinion and hinge).
6.e. Control the visibility of a part.
6.f. Display or hide work features of the current part, work objects and parts of the assembly.
6.g. Utilize Reference geometry – planes, axis, mate references
6.h. Differentiate between six degrees of freedom: translational and rotational.
6.i. Create a new part based on existing parts (top down assembly).
6.j. Check for interference.
6.k. Download component parts from a manufacturer's website
6.l. Simulate model motion.
6.m. Bring in standard parts from the Toolbox.

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

Assessment Strategies

7.1. Demonstration

Criteria

You will know you are successful when

7.1. you manipulate the model or assembly to gather information.
7.2. you create a 3D PDF that includes MBD.

Learning Objectives

7.a. Utilize MEASURE tools.
7.b. Evaluate the model or assembly using various commands and techniques.
7.c. Obtain the Mass Properties of a model.
7.d. Explore the options under the Command Manager's EVALUATE tab.
7.e. Examine model based definitions (MBD).

8. Differentiate between dimensioning and tolerancing methods.

Assessment Strategies

8.1. Demonstration

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.
8.3. you create a set of working drawings to ANSI Y14.5 standards.
8.4. you verify part or assembly is specified to the correct tolerance.

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.

Assessment Strategies

9.1. Demonstration

Criteria

You will know you are successful when

9.1. you include sufficient drawing views to fully define the part or assembly.
9.2. you apply correct dimensioning techniques to maintain the function of the part / assembly.

9.3. you Balloon and add a parts list (bill of materials) to an assembly drawing.
9.4. you add and complete a Title Block.
9.5. you add local and generalized Notes.
9.6. 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 annonations.
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.

10. Modify working drawings.

Assessment Strategies

10.1. Demonstration

Criteria

You will know you are successful when

10.1. you revise drawings.
10.2. you update models.
10.3. you document revisions needed / made.

Learning Objectives

10.a. Check drawings for accuracy.
10.b. Study marked up prints.
10.c. Update models.
10.d. Update working drawings.
10.e. Record revisions made.
10.f. Print drawings.