

Course Title: Solid Works Course number: MFG 110

**Credits: 3** 

Course description: COURSE DESCRIPTION: SolidWorks design focuses on parametric modeling while introducing the student to the paperless computer based design process utilizing the modern parametric 3-D design software SolidWorks. The course reviews the following topics: design process, design engineering, assembly modeling, mechanism analysis, rapid prototyping, team design, geometric dimensioning and tolerancing, and the analysis of tolerance stackups. Students will participate in individual & team design projects

<u>Course Objectives:</u> At the conclusion of this course students will be able to...

- Demonstrate an understanding of the fundamentals of drawing.
- Demonstrate a basic understanding of the fundamentals of Part Modeling.
- Demonstrate an understanding of the extrude and revolve features.
- Demonstrate an understanding of Sweep, Loft & Additional Features.
- Demonstrate an understanding of the fundamentals of Assembly Modeling.
- Demonstrate an understanding of top-down assembly modeling

## Course Content: Students will learn to...

- Create drawing template with document properties and sheet properties.
- Create a sheet format with document properties & sheet properties.
- Create standard, auxiliary, detailed & section views.
- Insert, create and modify dimensions & annotations.
- Develop and incorporate bill of materials.
- Effectively use SolidWorks User Interface & Toolbars
- Create file folders for projects & templates
- Use System Options & Document Properties
- Create a 2D sketch.
- Incorporate design intent into sketches, features, parts, and assembly.
- Apply Document properties.
- Create custom part templates.
- Use the following features: Extruded Cut, Revolve Boss/Base, Revolved Cut, Dome, Shell, Circular Pattern, and Fillet.
- Use the Mold tools [Scale, Parting Lines, Parting Surfaces, Shut-off Surfaces, Tooling Split, and Draft Analysis] to create a simple Core and Cavity part.
- Use sweep & loft features in 3D modeling.

- Create & edit a design table.
- Create components.
- Use Bottom-Up assembly modeling techniques.
- Insert components; insert & edit Mates and Smart Mates in an assembly.
- Obtain & assemble components using the Content Central.
- Assemble components from the SolidWorks Design Library.
- Use the Resolved Based feature: the Save as a Copy option, and the Component Pattern feature
- Use top-down assembly method.
- Import parts. using the top-down assembly method
- Develop a sheet metal design table.
- Convert a solid part into a sheet metal part and insert sheet metal features.
- Configure drawing into 3D Formed & 2D Flat Pattern states.
- Use sheet metal features: Base, Edge, Miter Flange, Hem, and Flat Pattern.