HOUSATONIC COMMUNITY COLLEGE
Advanced Manufacturing Technology Center

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

Course Objectives: 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.**