SPRINGFIELD TECHNICAL COMMUNITY COLLEGE

ACADEMIC AFFAIRS

Course Number: MET-159   Department: Mechanical Engineering Technology

Course Title: Solid Works for Sheet Metal Applications   Semester: Spring   Year: 2017

Objectives/Competencies

Course Objective	Competencies
1. Understand how to sketch, refine, and add parametric dimensions and relationships to profiles using Solid Works.	• Discuss the menu structure in Solid Works. • Discuss the use of the Design Tree/Feature Manager. • Recognize the three orthogonal planes. • Establish a plane or an existing face as a sketch plane. • Sketch the outline of a part feature. • Place appropriate parametric dimensions. • Establish appropriate geometric relationships. • Understand the difference between dimensions and relationships.
2. Manipulate the view of the part.	• Perform real time zoom and pan. • Rotate the view. • Create shaded and wire frame representations. • Create a plan view to the sketch plane.
3. Create part features for a 3D solid model.	• Explain the term "feature". • Create extruded solids—Base/Boss and Cut. • Create revolved solids. • Create fillets and chamfers as solid entities. • Create holes in a 3D model. • Create polar and rectangular arrays of solid features. • Understand how the Feature Manager/Design Tree is used for editing features.

4. Edit part features.	• Perform a feature edit. • Perform a sketch edit. • Add and delete dimensions and relationships. • Add and delete geometry. • Update the model.
5. Create basic flange features.	• Understand unique sheet metal FeatureManager tree items. • Create a sheet metal part using a base flange. • Flatten a sheet metal part to see the flat pattern. • Add edge flanges and miter flanges to sheet metal parts. • Use the hem feature. • Create Tab features. • Understand specialized sheet metal cut options.
6. Create and edit the flat pattern.	• Understand and modify flat pattern settings. • Add Corner-Trim features manufacturability. • Modify the corners of a formed sheet metal part using Closed Corner, Corner Relief, and Break Corner/Corner-Trim features. • Access and modify cut list item properties. • Create drawing of sheet metal parts. • Export a sheet metal flat pattern to DXF or DWG file formats.
7. Perform additional sheet metal techniques	• Design sheet metal parts from the flat pattern • Add features in an unfolded state. • Use the swept flange command. • Create sheet metal transitions using lofted bends.
8. Convert a solid to a sheet metal part.	• Use insert blends to add bend regions to a thin-walled part. • Rip the corners of a thin-walled part so that it can be unfolded. • Add a welded corner to a sheet metal part. • Use insert bends to unroll a cone or cylindrical part. • Use the convert to sheet metal command.