Online Course: Solid Edge Ordered Basics Plus Sheet Metal

Duration: 4 days

Version: SE 2020

At Course Completion

Students will have learned how to utilize Solid Edge to design production level parametric (ordered) models of parts, parametric (ordered) models of sheet metal parts, simple assemblies, detail drawings. They will also be familiar with the Solid Edge user interface, adding features, sketching tools and various modeling techniques.

Prerequisites

Here are the standard pre-requisites for the training course. Potential students should have or completed the following prior to the class:

- Mechanical Design Experience
- Windows Experience
- Previous 3D CAD experience

Course Content

Course consists of;

- 16 Video Lectures (PowerPoint's to support the Instructor's lecture).
- 147 Instructor lead video demonstrations.
- 53 practical activities to reinforce the lessons.
- Solution videos for each activity.

Topics:

Day 1

Lesson 1: Solid Edge – Getting Started

- Solid Edge environments
- Access to the Ordered modeling paradigm
- User Interface
- Creating, opening, and saving Solid Edge files
- Material table
- Solid Edge Help and learning tools
- Solid Edge menus, dialogs, and toolbars
- Solid Edge interface tools

Lesson 2: Ordered – Reference Planes

- Types of ordered reference planes
- Ordered reference plane creation commands
- Reference plane manipulation

Lesson 3: Ordered - Sketching Basics

- Creation of simple sketches
- Drawing Commands
- Using IntelliSketch when drawing sketches
- Editing sketches

Lesson 4: Ordered - Sketching Constraints

- Controlling the size and shape of sketches
- Placing and modifying of dimensional relationships
- Placing and modifying of geometric relationships

Day 2

Lesson 5: Base Features

- Creating base features.
    - Extruded Protrusion.
    - Revolved Protrusion.
    - Swept Protrusion.
    - Lofted Protrusion.

Lesson 6: Modeling Tools

- Creating profile-based features
    - Cutouts
    - Holes
    - Ribs
    - Web networks
    - Lips
    - Super Features
        - Vent
        - Mounting Bosses
        - Slots

Lesson 7: Ordered Features

- Creating treatment features
  - round
  - draft
  - chamfer
  - thin wall
  - thin region
  - thicken
  - threaded
  - embossed text

Lesson 8: Reusing Features

- Patterning features
- Copying features
- Mirror copy features
- Mirror copy part
- Feature library
- Dynamic editing

Lesson 8: Building assemblies

- Constructing an assembly document by placing parts into an assembly.
- Define the relationships between the parts.
- Using reference planes to control part placement.

Day 3

Lesson 10: Manipulating assemblies

- Editing assemblies
- Designing within an assembly
- Controlling part colors and properties
- Defining assembly display configurations
- Creating exploded assembly views

Lesson 11: Creating drawings of 3D models

- Creating 2D drawings from a part or assembly
- Placing multiple views of parts or assemblies
- Creating additional drawing sheets
- Modifying drawing views

Lesson 12: Dimensions and annotations

- Dimensioning of drawings
- Annotation of drawings
- Parts List
- Profile vs. drafting dimensions
- Tracking changes in model dimensions within a drawing

Lesson 13: Sheet Metal Design Introduction

- Introduction to the Sheet Metal Environment
  - Tab command
  - Contour Flange command
  - Lofted Flange command
    - Lofted bends and bend lines
  - Flange command

Day 4

Lesson 14: Sheet Metal Features

- Bend commands
- Jog Command
- Sheet Metal features
  - Closed Corner
  - Break Corner
  - Hems
  - Cutout features
  - Holes
  - Patterns
  - Etch command

Lesson 15: Deformation Features

- Deformation features
- Punch (Emboss) in Sheet Metal
- Deform Sheet Metal Features Across Bends

Lesson 16: Flattening and Drafting

- Flat Patterning
- Modeling in the Flat Pattern
- Save As Flat (DXF Output)
- Placing Flat Pattern in Draft
- Bend Tables

Note: The number of lessons covered on any given day could vary due to the progress of the student.