

**On-line Course:** Solid Edge Fundamentals Plus Sheet Metal

**Duration:** 7 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, synchronous models of parts, assemblies, detail drawings. They will also be familiar with the Solid Edge user interface, adding features, sketching tools and various modeling techniques. Students will also have learned how to utilize Solid Edge Sheet Metal to design production level parametric (ordered) models of sheet metal parts, and synchronous models of sheet metal parts.

**Note:** This is our most comprehensive introductory course for Solid Edge.

### **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

### **Course Content**

Course consists of.

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

### **Topics:**

#### ***Day 1***

##### **Lesson 1: Getting started in Solid Edge 2020**

- Solid Edge environments
- Modeling paradigms in Solid Edge
  - Ordered
  - Synchronous
- 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: Reference planes**

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

## **Lesson 3: Sketching Basics**

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

## **Lesson 4: 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 (ordered).
  - Extruded Protrusion.
  - Revolved Protrusion.
  - Swept Protrusion.
  - Lofted Protrusion.

### **Lesson 6: Ordered 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 9: Synchronous Sketching**

- Reference planes in synchronous modeling
- Synchronous coordinate system
- Synchronous sketching
- Draw directly on faces of bodies
- Plane Locking
- Sketch View Command
- Sketch Elements in PathFinder
- Sketch Regions

## ***Day 3***

### **Lesson 10: Synchronous Base Features**

- Quick Shapes
  - Box
  - Sphere
  - Cylinder
- Synchronous base features
  - Extrude
  - Revolve
  - Swept and Loft

### **Lesson 11: Dynamic Editing of Synchronous Parts**

- Selection Tools
- Steering wheels
  - 3D steering wheel
  - 2D steering wheel
- Move/rotate face command
- Select Set Priority

### **Lesson 12: Design Intent (Live Rules)**

- Introduction to the:
  - Design Intent Panel
  - Live rules
  - Solution Manager

### **Lesson 13: 3D Dimensioning and Geometric Relationships**

- Synchronous 3D Dimensions
  - Placement
  - Locked and unlocked
  - Variable Table in Synchronous
- Relate commands
  - Placement
  - 3D Geometric constraints (persistent)
- Live Sectioning
  - Creating and editing
  - Revolved Feature - Auto-create Live Section

### **Lesson 14: Synchronous Features**

- Creating and editing Synchronous features
  - Rounds and blends
  - Reorder Blends
  - Draft
  - Chamfers
  - Thin wall
  - Holes – 3D centric
  - Threads

## **Day 4:**

### **Lesson 15: Re-using Synchronous Features**

- Feature Pattern
  - Circular
  - Rectangular
  - Pattern Along Curve
  - Fill Pattern
- Mirror faces
- Feature Library
- Cut, Copy or Ctrl+Drag, Paste
- Face Detach and Attach

### **Lesson 16: Integrated Part Modeling**

- Integrated part modeling
  - Move to Synchronous
  - PathFinder
  - Integrated Mode Patterns
  - Integrated Mode Save
  - Integrated Mode Modeling
  - Integrated Mode – Coord System and Ref Plane behavior
  - Editing Integrated Mode models

### **Lesson 17: 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

### **Lesson 18: Manipulating assemblies**

- Editing assemblies
- Designing within an assembly
- Controlling part colors and properties
- Defining assembly display configurations
- Creating exploded assembly views
- Assemblies with synchronous parts
  - Move Face
- Steering Wheel assembly options
  - Copy component(s)

## **Day 5**

### **Lesson 19: Creating drawings of 3D models**

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

### **Lesson 20: Dimensions and annotations**

- Dimensioning of drawings
- Annotation of drawings
- Parts List

### **Lesson 21: Modifying draft documents**

- Modifying drawing views
- Profile vs. drafting dimensions
- Tracking changes in model dimensions within a drawing
- Dimension alignment
- Modifying dimension styles
- Using Design Manager

### **Lesson 22: Sheet Metal Design Introduction**

Introduction to the Sheet Metal Environment

- Tab command
- Contour Flange command
- Lofted Flange command
  - Lofted bends and bend lines
  - Vertex mapping

## **Day 6**

### **Lesson 23: Sheet Metal Design Additional features**

Additional features

- Flange command
- Bend commands
- Jog Command

### **Lesson 24: Sheet metal features**

Sheet Metal features

- Closed Corner
- Break Corner
- Hems
- Cutout features
- Holes
- Patterns
- Etch command

### **Lesson 24: Deformation Features**

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

### **Lesson 26: Flattening and Drafting**

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

## **Day 7**

### **Lesson 27: Synchronous Sheet Metal**

Synchronous features commands

- Tab
- Flange
- Contour Flange
- Close bend corners
- Hem
- Jog
- Bend

## **Lesson 28: Synchronous Sheet Metal Features**

### Synchronous Sheet Metal Features

- Feature Origin
- Feature Profiles
- Louvers
- Dimple and Drawn Cutout
- Bead and Gusset Features
- Break Corner
- Cutout Across Bends

## **Lesson 29: Synchronous functions unique to Sheet Metal**

- Synchronous Sheet Metal Manipulation
- Flat Patterns
- Integrated modeling

## **Lesson 30: Sheet Metal Conversion Tools**

- Covert to Sheet Metal
- Rip Corners
- Ordered Part to Sheet Metal convert
- Create Blank – Flatten anything

**Note:** Daily progress is only a suggestion for the on-line course. The pace is dependent on the users of this material.