Online Course: SE2025 Ordered Basics with Sheet Metal

Duration: Self-paced (estimated to take 5 days)

Version: SE 2025

At Course Completion

Students will have learned how to utilize Solid Edge to design production level parametric (ordered) models of parts, simple assemblies, detail drawings, and parametric (ordered) models of sheet metal parts. They will also be familiar with the Solid Edge user interface, adding features, sketching tools and various modeling techniques. This course **does not** cover sheet metal modeling, or synchronous modeling, or data management.

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.

- 18 Video Lectures (PowerPoints and Instructor lead video demonstrations).
- 71 practical activities to reinforce the lessons.

Topics:

Day 1

Lesson 1: Solid Edge – Getting Started

- Modeling paradigms in Solid Edge
  - Ordered
  - Synchronous
- First Time User Experience Startup page
- Solid Edge environments
- Creating, opening, and saving Solid Edge files
- Toolbars, Ribbons and Command Bars
- PathFinder
- Material table
- Solid Edge Help and learning tools
- Select Tool
- View manipulation commands
- Mouse Control

Lesson 2: Ordered - Reference Planes

- Reference Plane Types
- Reference Plane Creation Commands

Lesson 3: Ordered - Sketching Basics

- Introduction to Sketching
- Using IntelliSketch when drawing sketches
- Drawing Commands

Lesson 4: Sketching Constraints

- Placing and modifying of geometric relationships
- Placing and modifying of dimensional relationships
- Closed Sketch Indicator
- Relationship Colors

Day 2

Lesson 5: Sketching Editing and Helpful Tools

- Editing and Modifying Profiles
- Ordered Sketch Regions
- Accelerated 2D Sketching Workflow
- Relationship Assistant
- Dimension Formulas

Lesson 6: Ordered - Base Features

- Base Feature Types
- Extruded Protrusion
- Base Feature options
- Revolved Protrusion
- Swept Protrusion
- Lofted Protrusion

Lesson 7: Ordered-Profile Based Features

- Creating profile-based features
- Construct an extrusion or cutout: subsequent features
- Sidestep option
- Open profiles
- Holes
- Ribs
- Web networks
- Slots

Lesson 8: Ordered – Treatment Features

- Round
- Draft
- Chamfer
- Thin wall
- Lips
- Thin region
- Thicken
- Threaded
- Embossed text

Lesson 9: Ordered – Patterning & Feature Reuse

- Rectangular Patterns
- Circular Patterns
- Pattern along a curve
- Copying features
- Mirror copy features
- Mirror copy part
- Feature library
- Dynamic editing

Day 3

Lesson 10: Building assemblies

- Assembly Environment
- Placing Parts into Assemblies
- Applying relationships
- Most common relationships (Mate, Planer Align, Axial Align)
- Flash Fit
- Editing relationships
- Additional Relationships (Insert, Connect, Parallel, Angle, Tangent, Cam, and Center Plane)
- Displaying and Using reference elements
- Use Reduced Steps When Placing Parts
- FlashFit
- Capture Fit
- Pattern Parts in Assembly

Lesson 11: Manipulation of Assemblies

- Pathfinder review
- Ways to Select Parts
- Drag Component Command
- Editing Parts in Assembly
- Modeling Methods
- Create In Place

- Inter-Part Copy
- Light Weight Parts
- Part Colors
- Display Configurations
- Exploded Assemblies
- Working with synchronous Components

Lesson 12: Creating drawings of 3D models

- Draft Environment
- Sheet Types
- Draft Templates
- Drawing and Dimension Standards
- Adding Drawing Sheets
- Create drawing views of a part with the View Wizard
- View Wizard Options
- Create a drawing with the Drawing of Active Model command
- Additional Drawing View Creation
- Drawing View Display

Day 4:

Lesson 13: Dimensions and annotations

- Dimension Commands
- Dimension Options
- Dimension Alignment
- Modify Dimension Styles
- Annotation Commands
- Parts Lists

Lesson 14: Modifying draft documents and Design Manager

- Drawing View Selection command bar
- Repositioning views
- Change drawing view orientation
- Drawing View Display
- Drawing View Properties
- Manipulating Display
- Drawing View Alignment
- Profile vs. Drafting Dimensions
- In Place Activation (IPA)
- Drawing Revisions
- Track Dimension Changes
- Managing Unmanaged Solid Edge Data.
- Design Manager

Lesson 15: 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 5:

Lesson 16: Sheet Metal Design Additional features

- Flange command
- Flange options
- Partial Flanges
- Multi-edge Flange command
- Bend commands
- Jog Command

Lesson 17: Sheet Metal Features

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

Lesson 18: Deformation Features and Flat Patterns

- Dimple and Drawn Cutout
- Bead, Louvre, and Gussets
- Cross-Brake Feature
- Flat Patterns
- Save as Flat
- Etch command
- Stencil Fonts

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