On-line Course: Solid Edge Fundamentals with Sheet Metal

Duration: Self-paced (estimated to take 7 days)

Version: SE 2025

At Course Completion

This course combines the SE2025 Fundamentals course with the SE2025 Sheet Metal Fundamentals course, providing the most complete fundamentals course available. At the completion of this course, students will have learned how to utilize Solid Edge to design production level parametric (ordered) models of parts, synchronous models of parts, assemblies, and 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.

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 CAD experience – if no previous CAD experience, the student **must** go through the Solid Edge tutorials prior to attending class.

Course Content

Course consists of.

- 30 Video Lectures (PowerPoints to support the instructor's lecture), with built in Instructor lead video demonstrations.
- 122 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: Synchronous Introduction

- What is Synchronous Technology
- Similarities with the ordered paradigm
- Setting up the synchronous templates
- Switching between the paradigms
- Synchronous selection tools
- Interface tools unique to the synchronous paradigm

Lesson 11: Synchronous Sketching

- Reference planes in synchronous modeling
- Synchronous coordinate systems
- Synchronous sketching

- Draw directly on faces of bodies
- Plane locking
- Sketch View command
- Sketch elements in PathFinder
- Dimension migration from sketch to model
- Sketch regions
- Shared commands

Lesson 12: Synchronous Geometry Creation

- Face Sets
  - System-Defined Sets
  - User-Defined Sets
- Quick Shapes
- Extrude Command
- Revolve Command
- Swept and Loft Commands

Lesson 13: Synchronous Geometry Editing

- 3D steering wheel
- 2D steering wheel
- Move/rotate face command
- Select Set Priority
- Design Intent Panel

Lesson 14: Synchronous Design Intent (Live Rules)

- Types of Live Rules
- Introduction to the Solution Manager

Day 4:

Lesson 15: 3D Dimensioning & 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 16: Additional Synchronous Geometry Creation

- Rounds and blends
- Chamfers
- Draft
- Thin wall
- Lip
- Rib
- Web Network
- Slot
- Holes – 3D centric
- Threads

Lesson 17: Reusing Synchronous Geometry

Patterning Features

- Circular
- Rectangular
- Pattern Along Curve
- Fill Pattern

- Mirror faces
- Feature Libraries
- Cut, Copy or Ctrl+Drag, Paste
- Face Detach and Attach

Lesson 18: Integrated Modeling

Integrated Mode Modeling

- Move to Synchronous
- PathFinder
- Integrated Mode Menus
- Integrated Mode Save
- Integrated Mode Models – Display Mode
- Integrated Mode Modeling
- Integrated Mode – Coord System and Ref Plane behavior
- Sync Sketch behavior in Ordered
- Editing Integrated Mode models

Lesson 19: 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

Day 5

Lesson 20: 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 21: 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

Lesson 22: Dimensions and annotations

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

Lesson 23: 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

Day 6

Lesson 24: 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

Lesson 25: Sheet Metal Design Additional features

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

Lesson 26: Sheet Metal Features

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

Lesson 27: 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

Day 7

Lesson 28: Synchronous Sheet Metal

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

Lesson 29: Synchronous Sheet Metal Features

- Feature Origin
- Feature Profiles
- Editing procedural features
- Louvers
- Dimple and Drawn Cutout
- Bead and Gusset Features
- Break Corner
- Cutout Across Bends
- Holes
- Mirror and Pattern commands

Lesson 30: Synchronous functions unique to Sheet Metal

- Synchronous Sheet Metal Manipulation
- Flat Patterns
- Integrated modeling

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