Course: Solid Edge Fundamentals
Days: 5 days
Version: SE 2020

Description:

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. This course does not cover sheet metal modeling.

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.
• Completion of the Solid Edge Tutorials delivered with the product

Lack of Mechanical Design and Windows Experience is very difficult to overcome in the classroom. Students who lack one or both should realize the impact it will have on their learning experience. Just a little exposure to Solid Edge prior to the class can make all the difference.

Course Content

Course consists of;

• PowerPoint's to support the Instructor's lecture.
• Instructor lead demonstrations.
• Instructor lead activities.
• Independent practical activities to reinforce the lessons.

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

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

Lesson 5: Base Features

• creating base features (ordered)

Day 2

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
- Thicken
- threaded
- embossed text

Lesson 8: Reusing features

- patterning features
- feature library
- copying features
- mirror copy features
- mirror copy part

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

➢ 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 Cut, Copy & Paste
  • 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

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