



SOLIDWORKS for Mechanical Engineers

Duration: 50 Days

Syllabus Content

Module 1: Solidworks Fundamental

a) Introduction

- Understand solidworks Software
- What is the versions?
- Menus and Toolbars
- Selecting Objects with the Mouse
- Planes orientation
- What is the Compass?
- Graphic Properties
- Changing the Graphic Properties

b) Profile Creation

- Create a new Part
- Select and Appropriate Sketch Support
- Create Sketched Geometry
- Constrain the Sketch
- Sketch Analysis
- Manipulating Profile

Module 1: Part Design 3D

- Create Extrude boss/base Feature
- Create Extrude cut Feature
- Create Multi-profile Sketch Features
- Create Feature Profiles and Axis System
- Create Revolve boss/base Features
- Create Revolve Cut Features
- Create Sweep boss/base Features
- Sweep cut Feature
- Lofted boss/base Feature
- Lofted Cut Feature
- Boundary boss/base Feature
- Boundary Cut Feature
- Rib (Parallel & Perpendicular)
- Wrap Feature

- Helix & Spiral

c) Dress-up Features

- Shell the Model
- Apply a Draft
- Create Threads and Taps
- Fillets And Chamfer
- Edit Features
- Mirror Features
- Pattern Features

d) Finalizing Design Intent

- Apply Material Properties
- Analyze the Model
- Create Advanced Drafts
- Planes
- Points 2D & 3D

PRACTICE MODULE FOR PART DESIGN

Module 2: Assembly Design

- Create a new Assembly
- Assemble in the Base Component
- Manipulate the position of the Component
- Assemble and Fully Constrain Components
- Save the Assembly
- Manage the Sub assembly
- How to edit the existing part in an Assembly

PRACTICE MODULE FOR ASSEMBLY

Module 3: Drafting

- Introduction to Drafting
- Start a New Drawing
- Setting the Drawing Sheet Format and Drafting Standards
- Starting a Drawing with a Blank Sheet
- Sheet Properties
- Standard 3 view
- Model view
- Create Views
- Types of Views (Projection, Auxillary, Section)
- About Projection Plane
- Drafting Toolbars and Objects
- Title Blocks
- Create Dimensions and Annotations

- Section Views
- Adding a Detail View, Clipping View, Broken View, Breakout View
- View Modifications
- Save the Drawing
- Print the Drawing
- Drafting of An Assembly
- BOM and Ballooning

PRACTICE MODULE FOR DRAFTING

Module 4: Surface Design

a) Introduction to Surface Design

- Introduction to Surface Design
- Why create 3D Wireframe Geometry?
- Creative Points, Lines, Planes and Curves in 3D
- Creating Extrude Feature
- Creating Revolve Feature
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b) Creating Surfaces

- Why create Surface Geometry?
- Creating Basic Surfaces
- Creating a Swept Surface
- Creating a Surface Offset from a Reference
- Creating a Surface from Boundaries
- Creating a Loft Surface

c) Performing Operations on the Geometry

- Splitting/Trimming
- Creating Fillets
- Transforming Elements
- Extending Elements
- Disassembling Elements
- Joining Elements
- Additional Methods for Operations

d) Completing the Geometry in Part Design

- Why complete geometry in part design?
- Creating a surface to solid model
- Completing Geometry recommendations

PRACTICE MODULE FOR SURFACING

Module 5: Sheetmetal Products

a) Introduction

- Entering the Workbench
- Defining the Sheet Metal Parameters
- Flange/Tab
- Creating a Cutout
- Editing the Sheet and Tool Parameters
- Modifying the Bend Extremities
- Computing the Bend Allowance
- Creating Bends on Walls
- Extruding
- Creating a Flange
- Creating a Hem
- Creating a Tear Drop
- Creating a User Flange
- Creating a Flanged Hole, Bead ,Circular Stamp , Surface Stamp, Bridge , Flanged Cutout, Rip , Curve Stamp , Louver
- Creating User-Defined Stamping Features
- Mirroring
- Creating Corners, Creating Chamfers
- Patterning
- Looking For Sheet Metal Features
- Saving As DXF

PRACTICE MODULE FOR SHEETMETAL

Module: 6. SIMULATION

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