# SOLIDWORKS ESSENTIALS

## Target Audience:

This course is tailored for diploma holders, vocational students, technicians, and working professionals in design, manufacturing, or related industries who are looking to upskill or transition into CAD-based roles using SolidWorks. It is also beneficial for educators and trainers aiming to include 3D design tools in their teaching curriculum.

## Course Objective:

Upon completion of this course, participants will acquire a comprehensive understanding of SolidWorks, a leading 3D computer-aided design (CAD) software. The course is designed to equip students with the fundamental skills necessary to proficiently navigate, model, and analyze components and assemblies using SolidWorks.

## Course Outcome:

By the end of this course, participants will have a solid foundation in SolidWorks, enabling them to confidently create, modify, and analyze 3D models. The acquired skills will empower students to leverage SolidWorks for various engineering and design applications, enhancing their capabilities in the field of computer-aided design and manufacturing.

## Prerequisites

Participants should have basic computer literacy and an understanding of engineering or manufacturing principles. While no prior experience with SolidWorks is necessary, a background in fields like mechanical engineering, product design, or industrial technology will help in grasping the concepts more effectively.

## Course Outline:

The course comprises **40-hours** of theory and labs and is divided into **14** different chapters. Each chapter will be followed by **hands-on lab exercises** to reinforce learning and gauge understanding of the topics covered.

## Chapter 1. Introduction to SOLIDWORKS

- Installing SOLIDWORKS
- Getting Started with SOLIDWORKS
- Invoking the Part Modeling Environment
- Invoking the Assembly Environment
- Invoking the Drawing Environment
- Identifying SOLIDWORKS Documents
- Invoking a Shortcut Menu
- Customizing the Context Toolbar
- Customizing the Command Manager
- Working with Mouse Gestures
- Saving Documents
- Opening Existing Documents

## Chapter 2. Drawing Sketches with SOLIDWORKS

- Invoking the Part Modeling Environment
- Specifying Units
- Invoking the Sketching Environment
- Working with the Selection of Planes
- Specifying Grids and Snap Settings
- Drawing a Line Entity
- Drawing a Centerline
- Drawing a Midpoint Line
- Drawing a Rectangle
- Drawing a Circle
- Drawing an Arc
- Drawing a Polygon
- Drawing a Slot
- Drawing an Ellipse
- Drawing an Elliptical Arc
- Drawing a Parabola
- Drawing Conic Curves
- Drawing a Spline
- Editing a Spline
- Modifying the Tangency Direction of Arc/Spline

## Chapter 3. Editing and Modifying Sketches

- Trimming Sketch Entities
- Extending Sketch Entities
- Offsetting Sketch Entities
- Mirroring Sketch Entities
- Patterning Sketch Entities

- Creating a Sketch Fillet
- Creating a Sketch Chamfer
- Adding Text
- Moving a Sketch Entity
- Creating a Copy of Sketch Entities
- Rotating an Entity
- Scaling Sketch Entities
- Stretching an Entity

## Chapter 4. Applying Geometric Relations and Dimensions

- Working with Geometric Relations
- Applying Geometric Relations
- Controlling the Display of Geometric Relations
- Applying Dimensions
- Modifying/Editing Dimensions
- Working with Different States of a Sketch

## Chapter 5. Creating Base Features of Solid Models

- Creating an Extruded Feature
- Creating a Revolved Feature
- Navigating a 3D Model in the Graphics Area
- Manipulating View Orientation of a Model
- Changing the Display Style of a Model
- Changing the View of a Model

## Chapter 6. Creating Construction Geometries

- Creating Reference Planes
- Creating a Reference Axis
- Creating a Reference Coordinate System
- Creating a Reference Point
- Creating a Bounding Box

## Chapter 7. Advanced Modeling - I

- Using Advanced Options of the Extruded Boss/Base Tool
- Using Advanced Options of the Revolved Boss/Base Tool
- Creating Cut Features
- Working with Different Types of Sketches
- Working with Contours of a Sketch
- Displaying Shaded Sketch Contours
- Projecting Edges onto the Sketching Plane
- Editing a Feature and its Sketch

- Importing 2D DXF or DWG Files
- Displaying the Earlier State of a Model
- Reordering Features of a Model
- Measuring the Distance between Entities
- Assigning an Appearance/Texture
- Applying a Material
- Calculating Mass Properties

## Chapter 8. Advanced Modeling - II

- Creating a Sweep Feature
- Creating a Sweep Cut Feature
- Creating a Lofted feature
- Creating a Lofted Cut Feature
- Creating a Boundary Feature
- Creating a Boundary Cut Feature
- Creating Curves
- Splitting Faces of a Model
- Creating 3D Sketches

## Chapter 9. Patterning and Mirroring

- Patterning Features/Faces/Bodies
- Mirroring Features/Faces/Bodies

## Chapter 10. Advanced Modeling - III

- Working with the Hole Wizard
- Creating Advanced Holes
- Adding Cosmetic Threads
- Creating Threads
- Creating a Stud Feature
- Creating Fillets
- Creating Chamfers
- Creating Rib Features
- Creating Shell Features
- Creating Wrap Features

## Chapter 11. Working with Configurations

- Creating Configurations by using the Manual Method
- Creating Configurations by using the Design Table
- Saving Configurations as a Separate File

- Suppressing and Unsuppressing Features

## Chapter 12. Working with Assemblies - I

- Working with Bottom-up Assembly Approach
- Working with Top-down Assembly Approach
- Creating an Assembly by using Bottom-up Approach
- Working with Degrees of Freedom
- Applying Relations or Mates
- Hiding Faces while Applying a Mate
- Moving and Rotating Individual Components
- Working with SmartMates

## Chapter 13. Working with Assemblies - II

- Creating an Assembly by using the Top-down Approach
- Creating Flexible Components
- Editing Assembly Components
- Editing Mates
- Patterning Assembly Components
- Mirroring Components of an Assembly
- Creating Assembly Features
- Suppressing or Unsuppressing Components
- Inserting Parts having Multiple Configurations
- Creating and Dissolving Sub-Assemblies
- Publishing Envelopes
- Creating an Exploded View
- Collapsing an Exploded View
- Animating an Exploded View
- Editing an Exploded View
- Adding Explode Lines
- Detecting Interference in an Assembly
- Creating Bill of Material (BOM) of an Assembly

## Chapter 14. Working with Drawings

- Invoking the Drawing Environment
- Creating the Base View of a Model
- Invoking Drawing Environment from the Part or the Assembly Environment
- Creating a Model View
- Creating Projected Views
- Creating 3 Standard Views
- Working with the Angle of Projection

- Defining the Angle of Projection
- Editing the Sheet Format
- Creating other Drawing Views
- Applying Dimensions
- Modifying the Driving Dimension
- Modifying Dimension Properties
- Controlling the Default Dimension/Arrow Style
- Adding Notes
- Adding a Surface Finish Symbol
- Adding a Weld Symbol
- Adding a Hole Callout
- Adding Center Marks
- Adding Centerlines
- Creating the Bill of Material (BOM)
- Adding Balloons
- Detailing Mode