SOLIDWORKS Mechanical Design (CSWA) Certification Prep Course

Target Audience

This course is designed for students, diploma holders, engineers, designers, and working professionals who want to prepare for and pass the SOLIDWORKS Certified SOLIDWORKS Associate (CSWA) Mechanical Design Certification exam. It is suitable for beginners as well as users who want to validate their SOLIDWORKS skills through certification and build a strong foundation in mechanical design concepts aligned with the exam requirements.

Course Outcomes

- Understand core SOLIDWORKS concepts required for CSWA certification
- Apply sketching and part modeling techniques to create accurate 3D models
- Create and manage assemblies using mates and relationships
- Generate engineering drawings with proper dimensions and annotations
- Calculate and analyze mass properties of models
- Prepare effectively and successfully attempt the CSWA Mechanical Design Certification exam

Course Objectives

- Develop structured knowledge of SOLIDWORKS tools and workflows
- Build proficiency in sketching, modeling, assemblies, and drawings
- Enable learners to work with real-world mechanical design scenarios
- Strengthen understanding of mass properties and advanced modeling features
- Reinforce learning through hands-on exercises aligned with certification requirements
- Prepare participants comprehensively for the CSWA certification exam

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.

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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
- Lines
- Rectangles
- Circles
- Arcs
- Slots
- Polygons
- Ellipse

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
- Trim
- Offset
- Fillet
- Chamfer

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
- Horizontal
- Vertical
- Collinear
- Tangent
- Parallel
- Perpendicular
- Coincident
- Equal
- Symmetric
- Smart Dimension
- Fully Defined Sketch

Chapter 5. Creating Base Features of Solid Models

- Creating an Extruded Feature
- Creating a Revolved Feature
- Creating Cut Features
- Fillet
- Chamfer
- Shell
- Draft
- Rib
- Hole Wizard
- Linear Pattern
- Circular Pattern
- Mirror
- 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

- 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
- Density
- Volume
- Mass
- Center of Mass

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

- 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
- Mates
- Standard Mates
- Advanced Mates

Chapter 13. Assembly Advanced Tools

- 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
- Standard Views
- Section View
- Dimensions