SOLID WORKS

DURATION
45 Days

1. INTRODUCTION TO SOLID WORKS

- Part Mode
- Assembly Mode
- Drawing Mode

SYSTEM REQUIREMENTS

- Getting Started with Solid Works
- Menu Bar and Solid Works menus

COMMAND MANAGER

- Part Mode Command Managers
- Assembly Mode Command Managers
- Drawing Mode Command Managers
- Customized Command Manager

TOOLBAR

- Pop-up Toolbar
- View (Heads-Up) Toolbar
- Shortcut Bar
- Mouse Gestures

DIMENSIONING STANDARD AND UNITS

IMPORTANT TERMS AND THEIR DEFINITIONS

- Feature-based Modeling
- Parametric Modeling

- Bidirectional Associativity
- Windows Functionality
- Geometric Relations
- Blocks
- Library Feature
- Design Table
- Equations
- Collision Detection

WHAT'S WRONG FUNCTIONALITY?

- 2D Command Line Emulator
- SimulationXpress
- Physical Dynamics
- Physical Simulation
- Seed Feature
- Feature Manager Design tree
- Absorbed Features
- Child Features
- Dependent Features
- Auto-Backup Option
- Selecting Hidden Entities
- Color Scheme
- Self-Evaluation Test

2. DRAWING SKETCHES FOR SOLID MODELS

- The Sketching Environment
- Starting a New Session of Solid Works
- Work flow customization Area

TASK PANES

- Solid Works Resources Task Pane
- Design Library Task Pane
- File Explorer Task Pane
- Solid Works Resources Task Pane
- Design Library Task Pane

- File Explorer Task Pane
- View Palette Task Pane
- Appearances, Scenes, and Decals Task Pane
- Custom Properties Task Pane

STARTING A NEW DOCUMENT IN SOLID WORKS

- Part
- Assembly
- Drawing

UNDERSTANDING THE SKETCHING ENVIRONMENT SETTING THE DOCUMENT OPTIONS

- Modifying the Drafting Standards
- Modifying the Linear and Angular Units
- Modifying the Snap and Grid Settings

LEARNING SKETCHER TERMS

- Origin
- Inferencing Lines
- Select tool
- Selecting Entities Using the Box Selection
- Selecting Entities Using the Cross Selection
- Selecting Entities Using the SHIFT and CTRL
- Keys Invert Selection Tool

DRAWING LINES

- Orientation Rollout
- Options Rollout
- Drawing Continuous Lines
- Drawing Individual Lines
- Line Cursor Parameters
- Drawing Tangent or Normal Arcs Using the Line Tool
- Drawing Construction Lines or Centerlines
- Drawing the Lines of Infinite Length

DRAWING CIRCLES

- Drawing Circles by Defining their Center Points Drawing
- Circles by Defining Three Points
- Drawing Construction Circles

DRAWING ARCS

- Drawing Tangent/Normal Arcs
- Drawing Center point Arcs
- Drawing 3 Point Arcs

DRAWING RECTANGLES

- Drawing Rectangles by Specifying their Corners
- Drawing Rectangles by Specifying the Center and a Corner
- Drawing Rectangles at an Angle
- Drawing Center point Rectangles at an Angle
- Drawing Parallelograms
    - Drawing Polygons
    - Drawing Splines
    - Drawing Slots
- Creating a Straight Slot
- Creating a Center point Straight Slot
- Creating a 3 Point Arc Slot
- Creating a Center point Arc Slot
- Placing Sketched Points
- Drawing Ellipses
- Drawing Elliptical Arcs
- Drawing Parabolic Curves
- Drawing Display Tools
- Zoom to Fit
- Zoom to Area
- Zoom In/Out
- Zoom to Selection
- Pan
- Previous View
- Redraw

- Deleting Sketched Entities
- Self-Evaluation Test

3. EDITING AND MODIFYING SKETCHES

EDITING SKETCHED ENTITIES

- Trimming Sketched Entities
- Extending Sketched Entities
- Filleting Sketched Entities
- Chamfering Sketched Entities
- Offsetting Sketched Entities
- Mirroring Sketched Entities
- Mirroring While Sketching (Dynamic Mirror Entities)
- Moving Sketched Entities
- Rotating Sketched Entities
- Scaling Sketched Entities
- Copying and Pasting Sketched Entities

CREATING PATTERNS

- Creating Linear Sketch Patterns
- Creating Circular Sketch Patterns

EDITING PATTERNS

Writing Text in the Sketching Environment Modifying Sketched Entities

- Modifying a Sketched Line
- Modifying a Sketched Circle
- Modifying a Sketched Arc
- Modifying a Sketched Polygon
- Modifying a Spline
- Modifying the Coordinates of a Point
- Modifying an Ellipse or an Elliptical Arc
- Modifying a Parabola
- Dynamically Modifying and Copying Sketched Entities
- Splitting Sketched Entities

4.ADDING RELATIONS AND DIMENSIONS TO SKETCHES

- Applying Geometric Relations to Sketches
- Applying Relations Using the Add Relations Property Manager
- Design Intent
- Dimensioning a Sketch Horizontal/Vertical Dimensioning
- Aligned Dimensioning
- Angular Dimensioning
- Diameter Dimensioning
- Radius Dimensioning
- Linear Diameter Dimensioning
- Ordinate Dimensioning
- Concept of a Fully Defined Sketch
- Fully Defined
- Over defined
- Under defined
- Dangling
- No Solution Found
- Invalid Solution Found
- Deleting Over defining Dimensions
- Displaying and Deleting Relations
- Opening an Existing File

- Address Bar
- File name
- Type Drop-down List
- Open as Read-Only
- Quick view
- References
- Configurations
- Display States Area

5. ADVANCED DIMENSIONING TECHNIQUES AND BASE FEATURE OPTIONS

- Advanced Dimensioning Techniques Fully Defining the Sketches
- Dimensioning the True Length of an Arc
- Measuring Distances and Viewing Section Properties
- Measuring Distances
- Determining the Section Properties of Closed Sketches
- Creating Base Features by Extruding Sketches
- Creating Thin Extruded Features
- Creating Base Features by Revolving Sketches
- Creating Solid Revolved Features
- Creating Thin Revolved Features
- Determining the Mass Properties of Parts
- Dynamically Rotating the View of a Mode
- Rotating the View Freely in 3D Space
- Rotating the View around a Selected Vertex, Edge, or Face
- Modifying the View Orientation
- Changing the Orientation Using the Reference Triad
- Restoring the Previous View
- Displaying the Drawing Area in Viewports
- Displaying the Drawing Area in Two Horizontal Viewports
- Displaying the Drawing Area in Two Vertical Viewports
- Displaying the Drawing Area in Four Viewports
- Display Modes of a Model

- Wireframe
- Hidden Lines Visible
- Hidden Lines Removed
- Shaded With Edges
- Shaded
- Additional Display Modes
- Shadows in Shaded Mode Perspective
- Assigning Materials and Textures to Models
- Assigning Materials to a Model
- Changing the Appearance of the Model
- Editing the Appearances

6: CREATING REFERENCE GEOMETRIES

- Importance of Sketching Planes
- Reference Geometry
- Reference Planes
- Creating New Planes
- Creating Reference Axes
- Creating Reference Points
- Creating Reference
- Coordinate Systems
- Advanced Boss/Base Options
- From
- End Condition
- Direction of Extrusion
- Modeling Using the Contour Selection Method
- Creating Cut Features
- Creating Extruded Cuts
- Handling Multiple Bodies in the Cut Feature
- Creating Revolved Cuts
- Concept of the Feature Scope

7. ADVANCED MODELING TOOLS-I

- **Advanced Modeling Tools**
  - Creating Simple Holes
  - Creating Standard Holes Using the Hole Wizard
  - Adding External Cosmetic Threads
  - Creating Fillets
  - Selection Methods
  - Creating Fillets Using the FilletXpert
  - Creating Chamfers
  - Creating Shell Features
  - Creating Wrap Features

8. ADVANCED MODELING TOOLS-II

- **Advanced Modeling Tools**
  - Creating Mirror Features
  - Creating Linear Pattern Features
  - Creating Circular Pattern Features
  - Creating Sketch Driven Patterns
  - Creating Curve Driven Patterns
  - Creating Table Driven Patterns
  - Creating Fill Patterns
  - Creating Rib Features
  - Displaying the Section View of a Model

9. EDITING FEATURES

EDITING THE FEATURES OF A MODEL

- Editing Using the Edit Feature Option
- Editing Sketches of the Sketch-based Features
- Changing the Sketch Plane of the Sketches
- Editing by Selecting an Entity or a Feature
- Editing Using the Instant3D Tool
- Editing Features and Sketches by Cut, Copy, and Paste
- Cutting, Copying, and Pasting Features and Sketches from One Document
- To the Other
- Copying Features Using Drag and Drop

- Deleting Features
- Deleting Bodies
- Suppressing Features
- Unsuppressing the Suppressed Features
- Unsuppressing Features with Dependents
- Hiding Bodies
- Moving and Copying Bodies
- Reordering the Features
- Rolling Back the Feature
- Renaming Features
- Creating Folders in the Feature Manager Design tree
- What's Wrong Functionality?

10. ADVANCED MODELING TOOLS-III

ADVANCED MODELING TOOLS

- Creating Sweep Features
- Creating Cut-Sweep Features
- Creating Loft Features
- Adding a Section to a Loft Feature
- Creating Lofted Cuts
- Creating 3D Sketches
- Creating Grid Systems
- Editing 3D Sketches
- Creating Curves
- Extruding a 3D Sketch
- Creating Draft Features

11. ADVANCED MODELING TOOLS-IV

ADVANCED MODELING TOOLS

- Creating Dome Features
- Creating Indents
- Creating Deform Features
- Creating Flex Features

CREATING FASTENING FEATURES

- Creating the Mounting Boss
- Creating Snap Hooks
- Creating Snap Hook Grooves
- Creating Vents
- Creating a Lip/Groove Feature

CREATING FREEFORM FEATURES

- Face Settings Rollout
- Control Curves Rollout
- Control Points Rollout
- Display Rollout

DIMENSIONING A PART USING DIMXPERT

- Specifying the Datum
- Pop-up Toolbar
- Adding Dimensions
- Specifying the Location of a Feature
- Adding Geometric Tolerance to the Features
- Collecting Pattern Features
- Adding Dimensions Automatically

12. ASSEMBLY MODELING-I

- **Assembly Modeling**
  - Types of Assembly Design Approach
- **Creating Bottom-Up Assemblies**
  - Placing Components in the Assembly Document
  - Assembling Components
- **Creating Top-down Assemblies**
  - Creating Components in the Top-down Assembly
- **Moving Individual Components**
  - Moving Individual Components by Dragging
  - Moving Individual Components Using the Move Component Tool

- **Rotating Individual Components**
  - Rotating Individual Components by Dragging
  - Rotating Individual Components Using the Rotate Component Tool
  - Moving and Rotating Individual Components Using the Triad
  - Assembly Visualization

13. ASSEMBLY MODELING-II

- **Advanced Assembly Mates**
  - Applying the Symmetric Mate
  - Applying the Width Mate
  - Applying the Distance Mate
  - Applying the Angle Mate
  - Applying the Path Mate
- **Mechanical Mates**
  - Applying the Cam Mate
  - Applying the Gear Mate
  - Applying the Rack Pinion Mate
  - Applying the Screw Mate
  - Applying the Hinge Mate
- **Creating Sub-assemblies**
  - Bottom-up Sub-assembly Design Approach
  - Top-down Sub-assembly Design Approach
  - Inserting a New Sub-assembly
- **Deleting Components and Sub-assemblies**
  - Editing Assembly Mates
  - Replacing Mated Entities
  - Editing Components
  - Editing Sub-assemblies
  - Dissolving Sub-assemblies
  - Replacing Components
  - Creating Patterns of Components in an Assembly
  - Feature Driven Pattern
  - Local Pattern

- Copying and Mirroring Components
- Copy a Component with Mates
- Simplifying Assemblies using the Visibility Options
- Hiding Components
- Suppressing and Unsuppressing the Components
- Changing the Transparency Conditions
- Changing the Display States
- Checking Interferences in an Assembly
- Checking the Hole Alignment
- Creating Assemblies for Mechanism
- Analyzing Collisions Using the Collision Detection Tool
- Creating the Exploded State of an Assembly
- Creating the Explode Line Sketch

14. WORKING WITH DRAWING VIEWS-I

- The Drawing Mode
- Starting a Drawing Document
- Starting a New Drawing Document Using the New Solid Works Document
- Dialog Box
- Starting a New Drawing Document from the Part/Assembly Document

TYPES OF VIEWS

- Model View
- Projected View
- Section View
- Aligned Section View
- Auxiliary View
- Detail View
- Broken View
- Broken-out Section View
- Crop View
- Alternate Position View

GENERATING STANDARD DRAWING VIEWS

- Generating Model Views
- Using the View Palette to Place the Drawing Views
- Generating the Three Standard Views
- Generating Standard Views Using the Relative View Tool
- Generating Standard Views Using the Predefined View Tool

GENERATING DERIVED VIEWS

- Generating Projected Views
- Generating Section Views
- Generating Aligned Section Views
- Generating Broken-out Section Views
- Generating Auxiliary Views
- Generating Detail Views
- Cropping Drawing Views
- Generating Broken Views
- Generating Alternate Position Views
- Generating Drawing Views of the Exploded State of an Assembly
- Working with Interactive Drafting in Solid Works
- Editing and Modifying Drawing Views
- Changing the View Orientation
- Changing the Scale of Drawing Views
- Deleting Drawing Views
- Rotating Drawing Views
- Manipulating the Drawing Views
- Modifying the Hatch Pattern in Section Views
- Properties Rollout
- Options Rollout

15. WORKING WITH DRAWING VIEWS-II

ADDING ANNOTATIONS TO DRAWING VIEWS

- Generating Annotations Using the Model Items Tool
- Adding Reference Annotations
- Aligning the Dimensions
- Editing Annotations

ADDING THE BILL OF MATERIALS (BOM) TO A DRAWING

- Table Template Rollout
- Table Position Rollout
- BOM Type Rollout
- Configurations Rollout
- Part Configuration Grouping Rollout
- Keep Missing Item Rollout
- Item Numbers Rollout
- Setting Anchor Point for the BOM
- Linking Bill of Materials
- Adding Balloons to the Drawing Views
- Adding Balloons using the Auto Balloon tool
- Adding New Sheets to the Drawing Views
- Editing the Sheet Format
- Creating User-Defined Sheet Formats

16. EQUATIONS, CONFIGURATIONS, AND LIBRARY FEATURE

- Equations and Configurations Working with Equations Linking Dimensions Working with Configurations
- Creating Configurations by Using Design Tables
- Changing the Suppression State by Using the Design Table Editing the Design Table
- Deleting the Design Table
- Changing the Suppression State of a Component without Invoking the Design Table
- Changing the Visibility of Components in Different Configurations of an Assembly
- Library Features
- Creating a Library Feature Placing Library Features in a Part Editing the Library Features Dissolving the Library Features

17. WORKING WITH BLOCKS

- **Introduction to Blocks**
- Blocks Toolbar
- Saving a Sketch as a Block in the design Library Creating Mechanisms by using Blocks
- Creating the Rack and Pinion Mechanism Creating the Cam and Follower Mechanism
- Applying Motion to Blocks Creating Parts from Blocks
- Selected Blocks
- Block to Part Constraint

18. SHEET METAL DESIGN

- **Sheet Metal Design**
- Designing the Sheet Metal Components by Creating the Base Flange Creating the Base Flange
- Understanding the Feature Manager Design tree of a Sheet Metal Component Creating the Edge Flange
- Creating Tabs
- Creating the Sketched Bend Creating the Miter Flange Creating Closed Corners Creating Hems
- Creating the Jog Bend Breaking the Corners
- Creating Cuts on the Planar Faces of the Sheet Metal Components Creating Lofted Bends
- Creating a Flat Pattern View of the Sheet Metal Components Creating Sheet Metal Components from a Flat Sheet
- Creating a Sheet Metal Component From a Flat Part
- Converting a Part or a Flat Part into Sheet Metal by Adding Bends Adding Bends to the Flattened Sheet Metal Component Unbending the Sheet Metal Part Using the No Bends Tool
- Creating a Sheet Metal Component By Designing it as a Part Types of Bends
- Converting a Solid Body into a Sheet Metal Part Designing a sheet Metal Part from a Solid Shelled model
- Ripping the Edges

- Creating Cuts in Sheet Metal Components Across the Bends
- Creating Cuts in a Sheet Metal Component Created from a Solid Model
- Creating Cuts in a Sheet Metal Component Created Using the Base Flange Creating Cylindrical and Conical Sheet Metal Components
- Generating the Drawing View of the Flat Pattern of the Sheet Metal Components

19. SURFACE MODELING

- **Surface Modeling**
- Creating an Extruded Surface
- Creating a Revolved Surface
- Creating a Swept Surface
- Creating a Lofted Surface
- Creating a Boundary Surface
- Creating a Planar Surface Creating a Fill Surface
- Creating a Radiated Surface Offsetting Surfaces
- Trimming Surfaces
- Untrimming Surfaces
- Extending Surfaces
- Knitting Surfaces
- Filleting Surfaces
- Creating a Mid-Surface
- Deleting Holes from Surfaces Replacing Faces
- Deleting Faces
- Moving and Coping Surfaces
- Mirroring Surface Bodies
- Adding Thickness to Surface Bodies
- Creating a Thicken Surface Cut
- Creating a Surface Cut