



Transferring Technology...

SOLIDWORKS Basics

COURSE



CERTIFIED
Training & Support
Provider

 **SOLIDWORKS**



Contact us:

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COURSE DESCRIPTION : SolidWorks Basics

COURSE DURATION : 50 hrs.

SYLLABUS

Module 1: SolidWorks Basics and the User Interface

- Design Intent
- File References
- Opening Files
- The SolidWorks User Interface
- Using the Command Manager

Module 2: Introduction to Sketching

- 2D Sketching.
- Saving Files
- What are We Going to Sketch?
- Sketching
- Sketch Entities
- Basic Sketching
- Rules That Govern Sketches
- Design Intent
- Sketch Relations
- Dimensions
- Extrude
- Sketching Guidelines

Module 3: Basic Part Modeling

- Basic Modeling
- Terminology
- Choosing the Best Profile
- Choosing the Sketch Plane
- Details of the Part
- Boss Feature
- Sketching on a Planar Face
- Cut Feature
- View Selector
- Using the Hole Wizard
- Filletting
- Editing Tools
- Detailing Basics
- Drawing Views
- Center Marks
- Dimensioning



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- Changing Parameters

Module 4: Symmetry and Draft

- Boss Feature with Draft
- Symmetry in the Sketch
- Sketching Inside the Model
- View Options
- Using Model Edges in a Sketch
- Creating Trimmed Sketch Geometry
- Creating Views of Assemblies

Module 5: Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed Only
- Sketch Driven Patterns

Module 6: Revolved Features

- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- File Properties
- SolidWorks SimulationXpress
- Using SolidWorks SimulationXpress
- The SimulationXpress Interface

Module 7: Shelling and Ribs

- Shelling and Ribs
- Analyzing and Adding Draft
- Other Options for Draft
- Shelling
- Ribs
- Full Round Fillets
- Thin Features

Module 8: Editing - Repairs

- Part Editing
- Editing Topics
- Sketch Issues



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- Freezing Features
- FilletXpert

Module 9: Editing - Design Changes

- Part Editing
- Design Changes
- Information From a Model
- Rebuilding Tools
- Replace Sketch Entity
- Sketch Contours

Module 10: Configurations

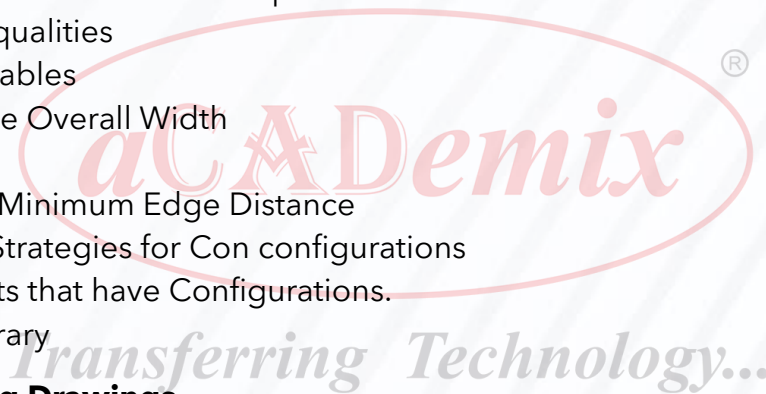
- Configurations
- Using Con configurations
- Other Methods to Create Configurations
- Using Global Variables and Equations
- Creating Equalities
- Global Variables
- Defining the Overall Width
- Equations
- Creating a Minimum Edge Distance
- Modeling Strategies for Con configurations
- Editing Parts that have Configurations.
- Design Library

Module 11: Using Drawings

- More About Making Drawings
- Section View
- Model Views
- Broken View
- Detail Views
- Drawing Sheets and Sheet Formats
- Projected Views
- Annotations

Module 12: Bottom-Up Assembly Modeling

- Modeling Creating a New Assembly
- Position of the First Component
- Feature Manager Design Tree and Symbols
- Adding Components
- Mating Components
- Using Part Configurations in assemblies
- Sub-assemblies
- Smart Mates
- Inserting Sub-assemblies



- Pack and Go

Module 13: Using Assemblies

- Using Assemblies
- Analyzing the Assembly
- Checking for Clearances
- Changing the Values of Dimensions
- Exploded Assemblies
- Explode Line Sketch
- Bill of Materials
- Assembly Drawings

Certificate:

On successful completion of the course and evaluation, the Certificate will be issued by aCADemix.



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OTHER COURSES

SolidWorks 3D CAD

- SolidWorks-Advanced
- SolidWorks-Mold Design
- SolidWorks- Routing
- SolidWorks- Piping & Tubing
- SolidWorks-Composer
- SolidWorks-MBD

SolidWorks Simulation

- SW Simulation-Advanced
- SW Simulation-Fatigue FEA
- SW Simulation-Drop Test FEA
- SW Simulation-Dynamic FEA
- SolidWorks Motion
- SolidWorks Plastics

SolidWorks Flow Simulation

- SolidWorks Flow Simulation

Design For Quality (DFQ)

- GD&T-Introduction
- GD&T-Advanced
- Tolerance Stack-Up Analysis
- DFMEA

SolidWorks Automation

- SolidWorks API - Basics
- SolidWorks API - Advanced

SolidWorks PDM

- SolidWorks PDM- User
- SolidWorks PDM - Admin

SolidWorks PDM Automation

- SolidWorks PDM- Automation

Abaqus CAE

- Abaqus CAE - Linear Static Analysis