



Introduction to SOLIDWORKS

Length: 1 Day

Prerequisite: None

Description: *Introduction to SOLIDWORKS* is an optional, preparatory course for the *SOLIDWORKS Essentials Class*. This course serves as an introduction for users who are new to CAD and focuses on understanding the basics of parametric modeling, sketching and extrusion techniques, as well as acquainting the user to the interface of SOLIDWORKS. This course prepares students by providing additional examples, hands-on exercises and definitions as preparation for the *SOLIDWORKS Essentials* class.

Course Syllabus

Lesson 1 - Fundamentals of SOLIDWORKS

- What is SOLIDWORKS?
- The Importance of Design Intent
- Factors of Design Intent
- User Interface Introduction
- Review of Options
 - Templates
- Customization Potential
- Mouse Basics (right click)
- Keyboard Shortcuts & Basic Commands

Lesson 2 - The Basics of Sketching

- Creating New Parts
- Default Planes
- Choosing the Best Plane
- Sketching Guidelines
- Cursor Feedback
- Relations & Dimensions
 - Driving Vs Driven Dimensions
- Sketch Mode & Statuses of Sketches
- The Modify Tool

Lesson 3 – Creating Multi-Feature Parts

- Common 3D Terminology
- Choosing the Best Initial Profile/Plane
- Boss/Cut Extrude Options
 - End Conditions
- Filleting Guidelines
- Rollback Bar
- Editing Sketches/Features
 - Deleting Relations
 - Adding new relations
 - Rebuilding the model
- View Selector

Lesson 4 – Diving Deeper: Multi-Feature Parts

- The Dynamics of building features on top of features
 - Design Intent in practice
- Relations Between Features
- Adding Draft
- Symmetry in Sketches
- Additional End Conditions
- Editing sketches from Toolbar



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