

SolidWorks Foundation Training

SolidWorks Foundation Training teaches you how to use SolidWorks mechanical design automation software to build parametric models of parts and assemblies, and how to make drawings of those parts and assemblies.

Overview of Topics Covered: Duration: 5 Days Total. 2 Days + 3 Days

Part 1 (Day 1)

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Lesson 1: SolidWorks Basics and the User Interface An explanation of "What is the SolidWorks Software?" and an overview of what it looks like.

Lesson 2: Introduction to Sketching How to create and dimension 2D sketches using SolidWorks with an explanation on how to use common sketch tools.

Lesson 3: Basic Part Modeling How to create multiple different feature to define the geometry of a part.

Lesson 4: Drawing Sheets and Views How to create a drawing sheet and add different views to that sheet.

Lesson 5: Dimensions Adding and manipulating dimensions on a drawing sheet.

Part 1 (Day 2)

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Lesson 1: Modeling a Casting or Forging Modelling parts with draft and using symmetry in sketches and features.

Lesson 2: Patterning Common pattern and mirror features in SolidWorks.

Lesson 3: Revolved Features How to model revolved solids using revolved features and some special sketch techniques for revolves. The use of SolidWorks SimulationXpress for design checking and comparative analysis.

Lesson 4: Shelling and Ribs Creating thin walled parts using the "Shell" feature and support structures using the "Rib" feature.

Part 2 (Day 3)

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Lesson 1: Editing: Repairs How to find and fix errors generated in SolidWorks part files and the "SWIFT" tools to aid efficient use of SolidWorks.

Lesson 2: Editing: Design Changes Making changes to designs and the factors that effect the ability of the part to change.

Lesson 3: Configurations of Parts Creating configurations using different methods.

Part 2 (Day 4)

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Lesson 1: Annotations Adding annotations to SolidWorks drawings.

Lesson 2: Sheet Formats and Templates Modifying drawing sheet formats and templates.

Lesson 3: Bottom-Up Assembly Modeling Creating an assembly in SolidWorks and adding part in different ways.

Part 2 (Day 5)

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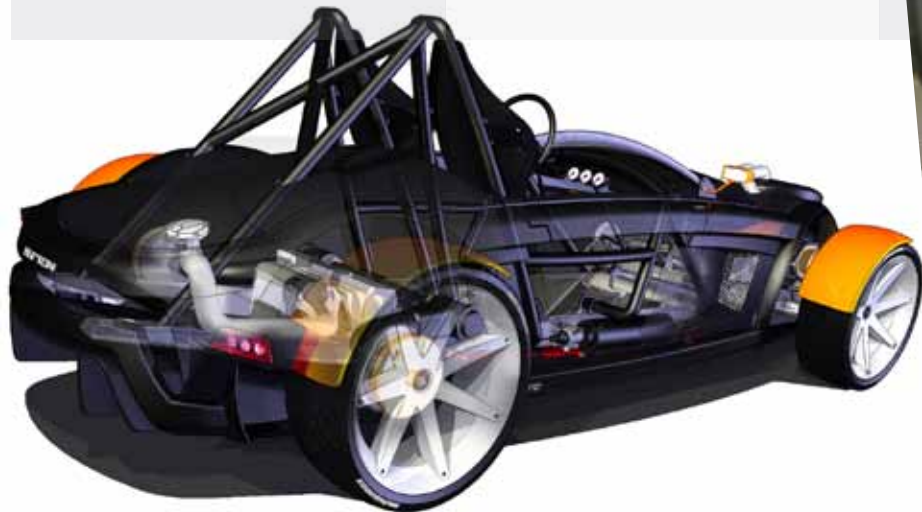
Lesson 1: Using Assemblies Working with the SolidWorks assembly tools.

Lesson 2: Assembly Drawing Views Adding assembly views to drawings.

Lesson 3: Bill of Materials and Tables Working with assembly BOMs and tables

Lesson 4: Performance and Display Issues Drawing performance issues.

Lesson 5: Drawing References and Comparison Drawing comparisons.



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LET'S GO DESIGN



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