SOLIDWORKS Advanced Topics
Course Outline

DURATION/TIME 5 Days 9:00 am - 5:00 pm	**METHODOLOGY** Practical hands-on with using computers, lecturing, discussions and case studies.
PREREQUISITES Experience with the Windows™ Operating System and Mechanical design. Completed SOLIDWORKS Essentials training	**TARGET** Application Engineer, R&D Engineer, Product Designer or Engineer, and Industrial Engineer

INTRODUCTION

SOLIDWORKS Advanced Topics is a collection of material from several "advanced" CAD training manuals. It includes selected lessons and exercises from: SOLIDWORKS Assembly Modeling, Advanced Part Modeling, Surface Modeling, Sheet Metal and Weldments. NOTE: This course may not be available in your area.

OBJECTIVE

At the end of this program participants are expected to:

- Perform advanced part and assembly modelling techniques.
- Understand and utilize the general commands of surfacing, sheet metal, and weldments.

CONTACT

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SOLIDWORKS Advanced Topics

DAY 1

Course Outline:

Lesson 1: Top-Down Assembly Modeling

- Top-Down Assembly Modeling
- Stages in the Process
- Making Changes to Dimension
- Case Study: Editing and Building In-context
- Adding features In-context
- Inserting a New Part into an Assembly
- Building In-context Features
- Propagating Changes
- Saving Virtual Parts as External
- External References
- Breaking and Locking External References
- Assembly Design Intent
- SOLIDWORKS File Utilities
- Removing External References

Lesson 2: Assembly Features and Smart Components

- Assembly Features and Smart Fasteners
- Case Study: Assembly Features
- Assembly Features
- Hole Series
- Smart Fasteners
- Smart Components
- Case Study: Smart Component

Lesson 3: Assembly Editing

- Assembly Editing
- Key Topics
- Case Study: Assembly Editing
- Mate Errors
- Replacing and Modifying Components
- Converting Parts and Assemblies
- Replacing Components Using Save As
- Reloading Components
- Component Patterns

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DAY 2

Course Outline:

Lesson 4: Large Assemblies

- Large Assemblies
- Key Topics
- Assembly Loading
- Assembly Modes
- Assembly Visualization
- Lightweight Components
- Large Assembly Settings
- Case Study: large Assembly Options
- Using SpeedPak
- Using Simplified Configurations
- Automatic Assembly Loading

Lesson 5: Multibody Design Techniques

- Multibody Parts
- Hide/Show Tree Items
- Multibody Design Techniques
- Case Study: Multibody Design
- Solid Bodies Folder
- Local Operations
- Feature Scope
- Patterning Bodies
- Tool Body Technique
- Combining Bodies
- Case Study: Protective Screen
- Intersect with Solid Bodies
- Case Study: Bowl

Lesson 6: Sketching with Splines

- Curves in Sketches
- Using Sketch Pictures
- Case Study: Guitar Body
- Spline
- Adding Spline Relations
- Changing the Shape of a Spline
- Fully Defining Splines
- Evaluating Splines
- Case Study: Two Point Spline
- Analyzing Solid Geometry
- Case Study: Torsion Continuity
- Style Spline
- Case Study: Watering Can Handle
- Fit Spline
- Case Study: Coffee Cup

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DAY 3

Course Outline:

Lesson 7: Introduction to Sweeping

- Sweeping
- Case Study: Faux Raised Panel Door
- Sweep with Guide Curves
- Case Study: Bottle Body
- The SelectionManager
- Case Study: Hanger Bracket

Lesson 8: 3D Sketching and Curve Features

- Curve Features
- Case Study: Spring
- Sweeping Along a 3D Path
- 3D Sketching
- Helix Curve
- Creating a 3D Curve from Orthogonal Views
- Projected Curve Feature
- Combining Curves
- Smoothing Transitions

Lesson 9: Introduction to Loft and Boundary Features

- Comparing Complex Features
- How Lofting and Boundary Work
- Case Study: Defroster Vent
- Loft feature
- Boundary Feature

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DAY 4

Course Outline:

Lesson 10: Understanding Surfaces

- Solids and Surfaces
- What is a Solid?
- Behind the Scenes
- Creating Solids from Surfaces
- Decomposing a Solid into Surfaces
- Additional Surface Concepts

Lesson 11: Introduction to Surfacing

- Similarities Between Solid and Surface Modeling
- Basic Surfacing
- Alternative to Trim

Lesson 12: Solid-Surface Hybrid Modeling

- Hybrid Modelling
- Using Surfaces to Modify Solids
- Interchanging Between Solids and Surfaces
- Performance Implications
- Surfaces as Construction Geometry
- Intersection Curve Feature
- Flattening Surfaces

Lesson 13: Repairing and Editing Imported Geometry

- SOLIDWORKS Import Options
- Importing a STEP files
- Comparing Geometry
- Addressing Translation Errors
- Repairing and Editing imported Geometry
- Procedure for Rebuilding Fillets

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DAY 5

Course Outline:

Lesson 14: Basic Flange Features

- What are Sheet Metal Parts?
- Sheet Metal Methods
- Unique Sheet Metal Items
- Flange Method
- Base Flange/Tab
- Sheet Metal Parameters
- Sheet Metal Thickness and Bend Radius
- Bend Allowance
- Auto Relief
- Editing Sheet Metal Parameters
- Sheet Metal Bend Features
- Flat-Pattern Feature
- Additional Flange Features
- Edge Flanges
- Editing the Flange Profile
- Edge Flanges on Curved Edge
- Miter Flanges
- Summary of Flange Features

Lesson 15: Converting to Sheet Metal

- Sheet Metal Conversion
- Insert Bends Method
- Adding Rips
- Insert Bends
- Making Changes
- Convert to Sheet Metal

Lesson 16: Weldment Features

- Weldments
- Structural Members
- Groups vs. Structural Members
- Adding Plates and Holes
- Gussets and End Caps
- Using Symmetry
- Advantages of a Multibody Part
- Limitations of a Multibody Part

Lesson 17: Woking with Weldments

- Managing the Cut List
- Cut List Item Names
- Accessing Properties
- Cut-List Properties Dialog
- Structural Member Properties
- Adding Cut List Properties
- Bounding Boxes in Weldments
- Options for Generating Cut List Items
- Custom Structural Member Profiles
- Defining Material
- Creating Custom Profiles
- Standard or Configured Profiles