# SOLIDUNIVERSITY

## SOLIDWORKS ADVANCED PART MODELING

Pre-Requisites: SOLIDWORKS Essentials

Daily Schedule: 8:30 a.m. - 4:30 p.m.

Length: 3 Days

Advanced Modeling focuses on making free-form-type parts using sweeps, lofts, and blends. You'll go beyond the basics and also learn how to use guide curves, curve paths, and even continuity controls to help you create organic-style geometry in your SOLIDWORKS parts. Bridging, local operations, and other multi-body modeling techniques are also covered.

### Introduction

- » About This Course
- » Using This Book
- » Windows® 7
- » Use of Color
- » Hide/Show Tree Items

### Lesson 1: Multibody Design Techniques

- » Multibody Solids
- » Multibody Techniques
- » Case Study:Multibody Design
- » Introducing: Solid Bodies Folder
- » Feature Scope
- » Patterning Bodies
- » Tool Body Technique
- » Introducing: Insert Part
- » Introducing: Move/Copy Bodies
- » Combining Bodies
- » Common Bodies
- » Intersect with Solid Bodies
- » Introducing: Intersect
- » Calculating Internal Volume
- » Local Operations
- » Indent Feature
- » Introducing: Indent
- » Deleting Solid Bodies
- » Introducing: Delete Body

### Lesson 2: Saving Solid Bodies

- » Saving Solid Bodies as Parts and Assemblies
- » Using Cut to Create Multiple Bodies
- » Insert into New Part
- » Saving Bodies
- » Modeling for Rapid Tooling
- » Splitting a Part into Multiple Bodies
- » Introducing: Split
- » Creating an Assembly
- » Using Split Part with Legacy Data

### Lesson 3: Sketching with Splines

- » Curves in Sketches
- » Splines
- » Introducing: Spline Tools
- » Evaluating Curvature Combs
- » Introducing: The Control Polygon
- » Introducing: Spline Handles
- » Analyzing Solid Geometry
- » Introducing: Display Curvature
- » Introducing: Zebra Stripes
- » Style Spline
- » Sketch Picture
- » Fit Spline

## GSC

## SOLIDWORKS ADVANCED PART MODELING

### Lesson 4: Introduction to Sweeping

- » Sweeping
- » Case Study: Faux Raised Panel Door
- » Sweep with Guide Curves
- » Case Study: Bottle Body
- » Pierce Relation
- » Sweep with Guide Curves
- » The SelectionManager
- » Case Study: Hanger Bracket
- » Symmetrical Splines
- » Dome Feature

### Lesson 5: Working with Curves

- » Curve Features
- » Case Study: Modeling a Spring
- » Sweeping Along a 3D Path
- » 3D Sketching
- » Helix Curve
- » Introducing: Projected Curve
- » Composite Curve
- » Applying the Label Outline to the Bottle
- » Performance Considerations
- » Modeling Threads
- » Blower Housing
- » Sweeping Along Model Edges
- » Introducing: Split Line

### Lesson 6: Advanced Sweeping

- » Sweep Options
- » Orientation and Twist Control
- » Introducing: Intersection Curve
- » Face Curves
- » Align with End Faces
- » Solid Sweep
- » Defining Twist
- » Introducing: Equation Driven Curve

### Lesson 7: Boundary Feature and Lofting

- » Comparing Complex Features
- » How Lofting and Boundary Work
- » Loft vs Boundary Feature
- » Introducing: Loft
- » Boundary Feature
- » Using Derived and Copied Sketches
- » Copying a Sketch
- » Modify Sketch
- » Introducing: Modify Sketch
- » Derived Sketches
- » Introducing: Derived Sketch5
- » Boundary Preview Options
- » Sketch Block and Library Feature Profiles
- » Additional Curves in Loft and Boundary
- » Centerline Lofting
- » Loft Preview Options
- » Introducing: Split Entities
- » Cleaning Up a Model
- » Deleting Faces
- » Introducing: Delete Face
- » Evaluating Edges
- » Introducing: Deviation Analysis
- » Lofting with Guide Curves
- » Curve Influence
- » Hook Using Boundary
- » Conics
- » Introducing: Curve Through XYZ
- » Sketch Blocks

### Lesson 8: Other Advanced Tools

- » Advanced Filleting
- » Delete Face: Delete and Fill
- » Wrap Feature
- » Introducing: Wrap
- » Deform Feature
- » Introducing: Deform
- » Introducing: Knit Surface
- » Direct Editing
- » Introducing: Move Face

These courses are taught from the official course curriculum from SOLIDWORKS Corporation, with additional information from GSC instructors.