



Shounco Design Studios
Your Complete Engineering Resource Center

SolidWorks Part Modeling

The goal of this course is to teach you how to build multibody and complex shape solid models using SolidWorks.

The topics covered in this course are:

Lesson 1: Multibody Solids

- Multibody Solids
- Bridging
- Extrude From
- Local Operations
- Boolean Operations
- Common Bodies
- Tool Body
- Patterning
- Indent Feature
- Using Cut to Create Multibodies
- Saving Solid Bodies as Parts and Assemblies

Lesson 2: Sweeps

- Sweeping Along a 3D Path
- 3D Sketching
- Projected and Composite Curves
- Fit Spline
- Sweeping and Lofting: What's the Difference?
- Creating a Curve Through a Set of Points
- Sweep Options
- Working with a Non-planar Path
- Variable Radius Filleting
- Analyzing Geometry
- Curvature and Zebra Stripes
- Intersection Curves
- Performance Considerations
- Modeling Threads
- Orientation and Twist Control
- Align and End Faces
- Sweeping Along Model Edges
- SelectionManager
- Sweeping a Tool Body

Lesson 3: Lofts and Splines

- Basic Lofting
- Using Derived and Copies Sketches
- Split Entities
- Cleaning Up a Model
- Delete Face
- Deviation Analysis
- Spline Sketching
- Advanced Lofting
- Sketch Picture

Lesson 4: Other Advanced Tools

- Advanced Fillets
- Wrap Feature
- Deform Feature
- Knit Surface
- Move Face and Delete Face

Prerequisites: SolidWorks Essentials, One month experience using SolidWorks

Length: 2 Days

Cost: \$695