

SolidWorks Classroom Training

Essentials

Lesson 1

Basic Part Modeling

- Feature Based
 - Sketched Features
 - Applied Features
 - Parametric
 - Solid Modeler
 - Fully associative
 - Constraints
 - Design Intent
-
- Automatic (sketch) Relations
 - Added Relations
 - Equations
 - Link Values
 - Dimensions
-
- Menus
 - Keyboard Shortcuts
 - Toolbars
 - Feature Manager Design Tree
 - Mouse Buttons
 - Feedback
-
- Options
 - Applying Changes
 - Settings on the Default Options
 - Suggested Settings

Lesson 2

Basic Part Modeling

- Choose the best profile
 - Create a new part
 - Create a sketch
 - Extrude bosses and cuts
 - Add fillets
 - Changing dimensions
-
- Exercise 1- Plate
-
- Exercise 2- Changes-2
-
- Exercise 3- Bracket

Lesson 3

Modeling a Casting or Forging

- Design Intent
 - Base Feature with draft
 - Mid-Plane Extrusion
 - Cut using existing edges
 - Cut with trimmed sketch geometry
 - Using copy and paste
 - Filleting
 - Editing a feature's definition
 - Link Values
-
- Exercise 4- Base Bracket
 - Exercise 5- Changes-3
 - Exercise 6- Slotted Link
 - Exercise 7- Wedge Block
 - Exercise 8- Guide
 - Exercise 9- Idler Arm

Lesson 4

Revolved Features and Circular Patterns

- Revolved Features (*centerline*)
- Reorder
- Using Axes
- Circular Patterning (*axis or dimension*)
- Library Features
- Adding Equations
- Changes and Rebuild Problems

- Exercise 10- Flange
-
- Exercise 11- Changes-4

- Exercise 12- Wheel

- Exercise 13- Compression Plate

- Exercise 14- Tool Post

- Exercise 15- Copy and Paste Feature

- Exercise 16- Pulley

Lesson 5

Thin Walled Parts

- Shelling
 - Using Palette Features
 - Sketch Driven Patterns
 - Creating Ribs
 - Mirror Patterns
 - The Hole Wizard
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- Exercise 17- Clevis
 - Exercise 18- Motor Shield
 - Exercise 19- Thin Features
 - Exercise 20- Patterns
 - Exercise 21- Level
-
- Exercise 22- Pump Cover
 - Exercise 23- Ice Cube Tray
 - Exercise 24- Brace with Rib
 - Exercise 25- Blow Dryer
 - Exercise 26- Cap

Lesson 6

Configurations of a Part

- Suppress and Unsuppress features
- Create Part Configurations
- Part Design Tables
- Insert Design Table
- Insert *New* Design Table

- Exercise 27- Configurations

- Exercise 28- More Configurations

- Exercise 29- Part Design Tables

- Exercise 30- Part Design Tables Two

Lesson 7

Bottom Up Assembly Modeling

- Create a new assembly
 - Insert components into an assembly
 - Add Mates
 - Component configurations in an Assembly
 - Insert Subassemblies
 - Interference Detection
-
- Exercise 31- Basic Mates
 - Exercise 32- Changes-7
 - Exercise 33- Gearbox Assembly
 - Exercise 34- Part Design Tables in an Assembly
 - Exercise 35 Gripe Grinder

Lesson 8

Introduction to Detailing

- Create drawing sheets
 - Add drawing views
 - Standard 3 Views
 - Named View
 - Auxiliary View
 - Section Views
 - Detail Views
 - Driving Dimensions
 - Bill of Materials
 - Driven (Reference) Dimensions
 - Annotations
 - Alternate Position View
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- Exercise 36- Creating Views
 - Exercise 37- Views and Dimensions
 - Exercise 38- Views, Notes and Dimensions
 - Exercise 39- Assembly Drawings
 - Exercise 40- Annotations

Lesson 9

Part Editing

- Errors
 - Reattach and replace dimensions
 - Edit Sketch
 - Edit Sketch Plane
 - Edit Definition
 - Rollback and Reorder
 - Add and delete relations
 - Mirror Part
 - Library Parts and Palette Features
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- Exercise 41- Errors
-
- Exercise 42- Changes-9
-
- Exercise 43- Adding Draft
-
- Exercise 44- Editing

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Advanced Part Modeling Module

Lesson 1

Modeling Advanced Shapes (I)

- Differences between sweep and loft
 - Curve from a Data Set
 - Projecting a curve onto a surface
 - Sweeping cuts and bosses
 - Using Library Features
 - Model threads using a helix
 - Creating Composite Curves
 - 3D Sketches
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- Exercise 1- Sweeps without Guides
 - Exercise 2- Attachment
 - Exercise 3- Hanger Bracket
 - Exercise 4- Offset Screwdriver
 - Exercise 5- Tire Iron
 - Exercise 6- 3D Sketching

Lesson 2

Modeling Advanced Shapes (II)

- Lofting Sketches
- Using Center Line in Loft
- Extrude up to surface with a Plane
- Advanced Face Fillets

- Exercise 7- Poker

- Exercise 8- Derived Sketch
- Exercise 9- Copy Sketch

- Exercise 10- Funnel

Lesson 3

Using Surfaces

- Surfaces
 - Extrude
 - Sweep
 - Planar
 - Knit
 - Trim
 - Extend
 - Fill
 - Mid Surface
- Exercise 11- Surface Modeling
- Exercise 12- Halyard Guide
- Exercise 13- Using Surfaces

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Advanced Assemblies Module

Lesson 1

Top Down Assembly Modeling

- Build parts in-context of the assembly
- Reference geometry in the assembly
- Create Assembly Patterns
- Remove external references from a copy

- Exercise 1- Top Down Assembly Modeling

- Exercise 2- Level Assembly

- Exercise 3- 3D Sketches in Top Down Assembly

Lesson 2

Working with Assemblies

- SmartMates
- Create a Assembly Configuration
- Assembly Design Tables
- Subassembly Configuration
- Available Properties:
 - \$CONFIGURATION@Bracket<7>
 - \$STATE@Plate<*>
 - \$SHOW@Lug<17-19>
 - \$COMMENT
 - \$USER_NOTES
 - \$PARTNUMBER
 - D1@Distance6

- Exercise 4- Mating and Assembly Motion

- Exercise 5- Fill Pipe

- Exercise 6- Configurations of an Assembly

- Exercise 7- Assembly Design Tables

- Exercise 8- Layout Assembly

Lesson 3

Assembly Editing

- Finding and fixing assembly errors
 - Information from an assembly
 - Assembly Features
 - SolidWorks Explorer
 - Previewing
 - Renaming
 - Copying
 - Replacing
 - Working with subassemblies
 - Mate Errors
 - Join components into a single part
 - Lightweight Assemblies
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- Exercise 9- Assembly Errors
 - Exercise 10- Assembly Features
 - Exercise 11- Assembly Equations
 - Exercise 12- Mirror Component
 - Exercise 13- SolidWorks Explorer
 - Exercise 14- Working with Subassemblies

Lesson 4

Core and Cavity

- Use Radiate Surface
- Create a Knitted Surface
- Use the cavity option in an assembly
- Base Part vs. Derived Part

- Exercise 15- Mold Cavity

- Exercise 16- Using Cavity and Surfaces

- Exercise 17- Using Cavity

- Exercise 18- Using a Base Part

- Exercise 19- Base Parts and Splitting

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Sheet Metal Module

Lesson 1

Modeling Sheet Metal Parts

- Insert Sheet Metal Features:
 - Base Flange
 - Edge Flange
 - Miter Flange
 - 3D Bends
 - Unfold/Fold
 - Tab
- Unfold the part
- Link features to the thickness
- Designing in the flat
- Formed Features

- Exercise 1- Sheet Metal Bends

- Exercise 2- Sheet Metal Relief

- Exercise 3- Sheet Metal from Flat

- Exercise 4- Sheet Metal Flanges and Bends

Lesson 2

Converting Parts to Sheet Metal

- Add bends
 - Convert Sheet Metal Parts
 - Add features in the flat
 - IGES importing
 - Using the Rip Feature
 - Process Planning
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- Exercise 5- Importing and Converting
 - Exercise 6- Process Planning
 - Exercise 7- Assorted Framing Hangers

Lesson 3

Modeling Sheet Metal in the Context of an Assembly

- Top Down Sheet Metal Design
- Edit Part
- Insert New Part

- Exercise 8- Ubolt in Context

- Exercise 9- Tube Brace

- Exercise 10- Sheet Metal in an Assembly