



 **SOLIDWORKS**  
**SOLIDWORKS Training**

**Course Outline**

**SOLID**  **PERTS**  
by solidxperience

***ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS***

**TABLE OF CONTENTS - SOLIDWORKS TRAINING**

*SOLIDWORKS Essentials – 4 Days (28h)..... 5*  
*SOLIDWORKS Drawings ANSI – 3 Days (21h)..... 7*  
*SOLIDWORKS Sheet Metal – 2 Days (14h)..... 9*  
*SOLIDWORKS Assembly Modeling – 2 Days (14h)..... 11*  
*SOLIDWORKS Weldments – 1 or 2 Day(s) (7h or 14h)..... 13*  
*SOLIDWORKS Advanced Part Modeling – 2 Days (14h)..... 14*  
*SOLIDWORKS Surface Modeling – 2 Days (14h)..... 14*  
*SOLIDWORKS Mold Design – 2 Days (14h)..... 16*  
*SOLIDWORKS Routing: Piping and Tubing – 2 Days (14h)..... 19*  
*SOLIDWORKS What’s New – 1 Day (7h)..... 20*  
*SOLIDWORKS Refresher – 4 days (28h)..... 22*

## SOLIDWORKS Essentials – 4 days (28h) (2 sessions)

### 1. SOLIDWORKS Basics and the User Interface

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- What is the SOLIDWORKS Software?
- Design Intent
- File References
- Opening Files
- The SOLIDWORKS User Interface
- Using the Command Manager

### 2. Introduction to Sketching

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- 2D Sketching
- Stages in the Process
- Saving Files
- Sketching
- Sketch Entities
- Basic Sketching
- Rules that govern Sketches
- Design Intent
- Sketch Relations
- Dimensions
- Extrude
- Sketching Guidelines

### 3. Basic Part Modeling

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- Basic Modeling
- Terminology
- Choosing the best Profile
- Choosing the Sketch Plane
- Details of the Part
- Boss Feature
- Sketching on a Planar Face
- Cut Feature
- View Selector
- Using the Hole Wizard
- Filletting
- Editing Tools
- Detailing Basics
- Drawing Views
- Center Marks

- Dimensioning
- Changing parameters

### 4. Symmetry and Draft

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- Boss feature with Draft
- Symmetry in the Sketch
- Sketching inside the Model
- View Options
- Using Model Edges in a Sketch
- Creating Trimmed Sketch Geometry
- Copy and Paste Features

### 5. Patterning

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- Why use Patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern seed only
- Up to Reference
- Sketch Driven Patterns

### 6. Revolved Features

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- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- Files Properties
- SOLIDWORKS SimulationXpress
- Using SOLIDWORKS SimulationXpress
- The SimulationXpress Interface

*see Part 2 on next page »*

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PMT2200-ENG

## SOLIDWORKS Essentials (part 2)

### 7. Shelling and Ribs

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- Shelling and Ribs
- Analyzing and adding Drafts
- Other options for Drafts
- Shelling
- Ribs
- Full round Fillets
- Thin features

### 8. Editing: Repairs

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- Part Editing
- Editing Topics
- Sketch issues

### 9. Editing: Design Changes

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- Part Editing
- Design Changes
- Information from a Model
- Rebuilding Tools
- Sketch Contours
- Replace Sketch Entity

### 10. Configuration

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- Configurations
- Using Configurations
- Other methods to create Configurations
- Modeling strategies for Configurations
- Editing Parts that have Configurations
- Design Library

### 11. Global variables and equations

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- Using global Variables and Equations
- Renaming Features and Dimensions
- Design Rules using global Variables and Equations
- Global Variables
- Equations
- Using Operators and Functions

### 12. Using Drawings

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- Removed section
- Detail Views
- Drawing Sheets and Sheet Formats
- Model Views
- Section Views
- Annotations

### 13. Bottom-up Assembly Modeling

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- Bottom-up Assembly
- Creating a new Assembly
- Position of the first Component
- Feature Manager Design Tree and Symbols
- Adding Components
- Mating Components
- Using Part Configurations in Assemblies
- Subassemblies
- Smart Mates
- Inserting subassemblies
- Pack and Go

### 14. Using Assemblies

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- Using Assemblies
- Analyzing the Assembly
- Checking for clearances
- Changing the values of Dimensions
- Exploded Assemblies
- Rollback and Reorder Explode Steps
- Explode Line Sketch
- Bill of Materials
- Assembly Drawings

### Appendix

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- Options Settings
- Document Templates

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## SOLIDWORKS Drawings ANSI – 3 Days (21h)

### 1. Review of the Basics

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- Review of Essentials
- Drawing System Pptions
- Beginning a new Drawing
- View Palette and Model Views
- Detailing Techniques
- Section Views
- Detail Views
- Moving Drawing Views
- Moving Dimensions
- Center Marks & Centerlines
- Assembly Drawings Review
- Adding Sheets
- Standard 3 View

### 2. Understanding Drawing Templates

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- Structure of a Drawing Document
- Drawing Document
- Drawing Sheet
- Sheet Formats
- Understanding Drawing Templates
- Drawing Template Design strategy
- Design a Drawing Template
- Creating a sample Model and Drawing

### 3. Customizing the Sheet Format

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- Completing the Title Block Sketch
- Completing the Title Block Notes
- Tips for Locating Notes
- Adding a Company Logo
- Defining the Border
- Setting Anchors
- Title Block Fields

### 4. Saving and Testing the Sheet Format File

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- Understanding Sheet Format Properties
- Understanding Sheet Format Behavior
- Saving the Sheet Format
- Testing Sheet Format

### 5. Creating Additional Sheet Formats and Templates

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- Creating Additional Sheet Formats
- Drawing Templates with Sheet Formats
- Other Drawing Template Items
- Property Tab Builder
- Properties .txt File

### 6. Advanced Options for Drawing Views

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- Advanced Drawing Views
- Showing Hidden Edges
- Broken-out Section Views
- Auxiliary Views
- Rotating Views
- Crop View
- Understanding View focus
- Advanced Views for Assemblies
- Section Scope
- Alternate Position View
- Using Configurations
- Custom View Orientations
- New View
- Relative View
- 3D Drawing View

*see Part 2 on next page »*

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## SOLIDWORKS Drawings ANSI (suite)

### 7. Understanding Annotation Views

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- Understanding Annotation Behavior
- What are Annotation Views?
- Annotation Folders
- Default Annotation Views
- Annotations View Visibility
- Insert Annotation Views
- Editing Annotation Views
- Annotation Update
- Annotations Folder in Drawings

### 8. Advanced Detailing Tools

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- Detailing Tools
- Annotation Views vs. Model Items
- Parametric Notes
- Dimension Types
- Arranging Dimensions
- Location Labels

### 9. Using Layers, Styles, and the Design Library

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- Using Layers
- Reusing Dimension Properties
- Annotations in the Design Library
- Flag Note Bank

### 10. Advanced Options for BOM Tables

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- Tables in SOLIDWORKS
- Bills of Material Properties
- Displaying the BOM Assembly Structure
- Modifying a Table
- Saving a Table Template
- Properties in BOM Tables
- BOM Component Options
- Balloon Indicator

### 11. Additional SOLIDWORKS Tables

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- Inserting a Hole Table
- Splitting Tables
- Using a Revision Table
- Leader Annotation Options
- Design Tables in Drawings

### 12. Additional Drawing Tools

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- Reusing Drawings
- DrawCompare
- SOLIDWORKS Design Checker
- SOLIDWORKS Task Scheduler

### 13. Managing Performance

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- Managing Performance
- Performance Evaluation
- Detailing Practices
- System Options & Documents Properties
- Open options
- Detached drawings
- Hardware and performance
- Additional Considerations
- Quick Reference Guide

### Important note regarding the exercises

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- During this training, we'll replace the SOLIDWORKS offered exercise files with your documents. You are therefore asked to bring your own files to produce your drawing documents.

During the exercises that follow the training lessons, you will build your drawing templates, personalize your sheet formats, create title blocks with your custom properties, produce your annotation and dimension favorites, and finally produce your table templates (BOM, revision, etc.).

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## SOLIDWORKS Sheet Metal – 2 Days (14h)

### 1. 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 Edges
- Miter Flanges
- Hem Feature
- Tab Features
- Cuts in Sheet Metal

### 2. Working with Flat Pattern

- Working with the Flat Pattern
- Flat Pattern Settings
- Feature for Manufacture
- Corner-Trim Feature
- Corners in the Formed State
- Closed Corner
- Corner Relief
- Break Corner/Corner Trim
- Producing the Flat Pattern
- Sheet Metal Cut List Properties
- Sheet Metal Drawings
- Flat Pattern Drawing Views
- Flat Pattern View Properties
- Drawing document properties
- Cut List Properties as a Note
- Exporting a flat pattern

### 3. Standardizing Sheet Metal Designs

- Standardizing Gauge Numbers and Bend Radius
- Standardizing Bend Allowance
- Topics for Standardizing Parameters
- Using Tables
- Custom Sheet Metal Materials
- Sheet Metal Templates
- Sheet Metal Tables in Drawings
- Mapping DXF Output

### 4. Additional Sheet Metal Techniques

- Additional Sheet Metal Methods
- Designing from the Flat
- Sketched Bend Feature
- Jog Feature
- Adding Features in an Unfolded State
- Unfold and Fold
- Swept Flange
- Swept Flange Flat Pattern Options
- Lofted Bends
- Lofted Bends in the Design Library

### 5. Converting to Sheet Metal

- Sheet Metal Conversion
- Insert Bends Method
- Adding Rips
- Insert Bends
- Making Changes
- Welded Corners
- Converting Cones and Cylinders
- Convert to Sheet Metal

*see Part 2 on next page »*

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## SOLIDWORKS Sheet Metal (Part 2)

### 5. Multibody Sheet Metal Parts

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- Multibody Sheet Metal Parts
- Multibodies with Base Flange
- Sheet Metal Parameters for Multibodies
- Cut List Item Properties for Multibodies
- Flat Pattern Drawing Views for Multibodies
- Cut List Balloon Annotations
- Exporting to DXF/DWGs with Multibodies
- Convert with Multibodies
- Hiding and Showing Bodies
- Using Split with Sheet Metal Parts
- Patterning for Multibodies
- Using Edge Flanges to Merge Bodies
- Interfering Bodies
- Combining Sheet Metal with Other Bodies

### 6. Forming Tools and Gussets

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- Sheet Metal Forming Tools
- Forming Tools in the Design Library
- Form Tool Features in the Flat
- Part Document Properties
- Custom Forming Tools
- Split Line
- Forming Tool Feature
- Legacy Behavior for Forming Tools
- Form Tools in Drawings
- Sheet Metal Gusset

### 7. Additional Sheet Metal Functions

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- Additional Sheet Metal Functions
- Cross-Breaks
- Vent Features
- Fill Pattern
- Mirror part
- Tab and Slot
- Process plans

### Appendix A : Sheet Metal Tables

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- Tables
- Customizing tables



### Annex : ToolsXperts Demo

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- Overview of CutXperts
- Automatic export to DXF
- Layer Options
- Other Options

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PMT2306-ENG



## SOLIDWORKS Assembly Modeling – 2 Days (14h)

### 1. Advanced Mate Techniques

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- SOLIDWORKS Assemblies
- Assembly File Structure
- File References
- File References Example
- Solving Mates
- Advanced Mate Techniques
- Mate References
- Design Library Parts
- Capture Mate References
- Mates with Axes and Planes
- Multiple Selection Mate References
- Multiple Mate Mode
- Driven Mates
- Using misaligned Mates
- Copying multiple Components
- Fixed Components
- Advanced Mate Features
- Profile Center Mate
- Rack Pinion Mate

### 2. Top-Down Assembly Modeling

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- Top-Down Assembly Modeling
- Making Changes to Dimensions
- 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

### 3. Assembly Features and Smart Components

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- Assembly Features and Smart Fasteners
- Hole series
- Smart Components
- Flexible Components

### 4. Assembly Editing

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- Assembly Editing
- Key Topics
- Mate Errors
- Replacing and modifying Components
- Converting Parts and Assemblies
- Replacing Components Using Save As
- Reloading Components
- Component Patterns

### 5. Using Configurations with Assemblies

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- Using Configurations with Assemblies
- Creating Configurations Manually
- Configuration Properties
- Using the Modify Configurations Dialog
- Changing Configurations using the Context Toolbar
- Managing the Tree Display
- Assembly Evaluation Tools
- Controlling Dimensions in an Assembly
- Creating an Equality
- Equations with Functions
- Sensors
- Using the Mate Controller

*see Part 2 on next page »*

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## SOLIDWORKS Assembly Modeling (Part 2)

### 6. Display States and Appearances

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- Display States
- Bulk Selection Tools
- Advanced Select
- Envelopes
- Appearances, Materials, and Scenes

### 7. Large Assemblies

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- Large Assemblies
- Key Topics
- Assembly Modes
- Assembly Visualization
- Lightweight Components
- Large Assembly Settings
- Using SpeedPak
- Using Simplified Configurations
- Defeature
- Modifying the Structure of an Assembly
- Envelope Publisher
- Large Design Review
- Comparison of Loading, Modes, and Methods
- Tips for faster Assemblies
- Drawing Considerations

### 8. Facility Layout

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- Facility Layout
- Publishing an Asset
- Using Magnetic Mates
- Modeling connection Point Geometry

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**PMT2302-ENG**

## SOLIDWORKS Weldments – 1 or 2 Day(s) (7h or 14h)

\*General Design & Multi-Bodies = 1 day

\*\*General Design, Multi-Bodies & Structure Systems for Buildings = 2 Days

### DAY 1 : General Design & Multi-Bodies

#### 1. 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

#### 2. Working 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
- Inserting Existing Parts
- When to Use an Assembly

#### 3. Configuring and Detailing Weldments

- Weldment Configurations
- Post-Assembly Machining Features
- Weldment Drawings
- Drawing Views of Individual Bodies
- Representing Welds

#### 4. Working with Bent Structural Members

- Working with Bent Structural Members
- 3D Sketching



#### Annex : ToolsXperts Demo

- ToolsXperts Presentation
- BeamCutXperts Introduction

### DAY 2 : Structure Systems for Buildings

#### 5. Introduction to Structure Systems

- Structure System
- Primary vs. Secondary Members
- Secondary Members

#### 6. Working with Structure Systems

- Managing the Cut List
- Gussets and End Caps
- Bounding Boxes in Structure System

#### 7. Defining and Inserting Connection Elements

- Connections
- Define Connection Element
- Insert Connection Element

#### SPECIFIC NOTES FOR THIS TRAINING

**General Design & Multi-Bodies :** This training mainly focused on the general design of mechanically welded frames and tubular or profiled structures.

This first part of the training is centered on design using the multi-bodies technique;;

The multi-bodies design technique is also appropriate when designing non-mechanically welded elements (ex : Design of furniture, cabinets, etc.)

**Structure Systems for Buildings :** Training mainly focused on the design of the structure of buildings.

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PMT2307-ENG

## SOLIDWORKS Advanced Part Modeling – 2 Days (14h)

### 1. Multibody Design Techniques

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- Multibody Parts
- Hide/Show Tree Items
- Multibody Design Techniques
- SolidBodies Folder
- Local Operations
- Feature Scope
- Patterning Bodies
- Tool Body Technique
- Combining Bodies
- Intersect with Solid Bodies
- Indent Feature
- Deleting Solid Bodies

### 2. Saving Solid Bodies

---

- Multibody Part vs. Assembly
- Saving Bodies Functions
- Insert into New Part
- Save Bodies
- Modeling for Rapid Tooling
- Splitting a Part into Multiple Bodies
- Automating an Assembly

### 3. Sketching with Splines

---

- Curves in Sketches
- Using Sketch Pictures
- Splines
- Adding Spline Relations
- Changing the Shape of a Spline
- Fully Defining Splines
- Evaluating Splines
- Analyzing Solid Geometry
- Style Spline
- Fit Spline

### 4. Introduction to Sweeping

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- Sweeping
- Sweep with Guide Curves
- The Selection Manager

### 5. 3D Sketching and Curve Features

---

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

### 6. Threads and Library Feature Parts

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- Bottle Features
- Saving a Library Feature Part
- Performance Considerations
- Creating the Sweep Path
- Sweeping Along Model Edges

### 7. Advanced Sweeping

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- Sweep Options
- Additional Sweep Settings
- Profile Orientation
- Intersection Curve Feature
- Visualizing Sweep Sections
- Solid Profile

*see Part 2 on next page »*

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PMT2203-ENG

## SOLIDWORKS Advanced Part Modeling (Part 2)

### 8. Introduction to Loft and Boundary Features

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- Comparing Complex Features
- How Lofting and Boundary Work
- Loft Feature
- Boundary Feature
- Copying a Sketch
- Modify Sketch
- Derived Sketches
- Boundary Preview Options
- Sketch Block and Library Feature Profiles

### 9. Advanced Loft and Boundary Features

---

- Additional Curves in Loft and Boundary
- Centerline Lofting
- Loft Preview Options
- Adding Sketch Segments
- Cleaning Up a Model
- Deleting Faces
- Evaluating Edges
- Face Fillets
- Curve Influence

### 10. Advanced Filleting and Other Features

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- Fillet Settings
- Fillet Parameters
- Constant Size Fillets
- Delete Face: Delete and Fill
- Partial Edge Parameters
- Fillet Options
- Variable Size Fillets
- Face Fillets
- FilletXpert
- Other advanced Features
- Wrap Feature
- Direct Editing

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## SOLIDWORKS Surface Modeling – 2 Days (14h)

*\*The "SOLIDWORKS Advanced Part Modeling" Course is required for this training.*

### 1. Understanding Surfaces

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- Solids and surfaces
- What is a solid?
- Behind the scenes
- Creating solids from surfaces
- Deconstructing a solid into surfaces
- Additional surface concepts
- Takeaways
- Why use surfaces?
- Continuity explained
- Workflow with surfaces

### 2. Introduction to Surfacing

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- Similarities between solid and surface modeling
- Basic surfacing
- Alternative to trim

### 3. Solid-Surface Hybrid Modeling

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- Hybrid modeling
- Using surfaces to modify solids
- Interchanging between solids and surfaces
- Performance implications
- Surfaces as construction geometry
- Copying faces
- Flattening surfaces

### 4. Repairing and Editing Imported Geometry

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- Importing data
- File translation
- SOLIDWORKS import options
- Importing a STEP file
- Comparing geometry
- Addressing translation errors
- Repairing and editing imported geometry
- Procedure for rebuilding fillets

### 5. Blends and Patches

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- Smoothing patches
- Boundary surfaces
- Corner blends

### 6. Complex Blends

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- Complex blends
- Freeform feature

### 7. Advanced Surface Modeling

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- Process stages
- Modeling the lower half
- Design changes

### 8. Master Model Techniques

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- Introduction to Master Models
- Surface Master Model technique
- Working with a solid Master Model
- Specialized features for plastic parts

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## Mold Design Using SOLIDWORKS – 2 Days (14h)

\*The "SOLIDWORKS Advanced Part Modeling" Course is required for this training.  
\*\*The "SOLIDWORKS Surface Modeling" is recommended prior to this course.

### 1. Surface Concepts and Imported Geometry

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- Surfaces in Mold Design
- 3D Model Types
- Geometry vs Topology
- Creating Solids from Surfaces
- Decomposing a Solid into Surfaces
- Additional Surface Concepts
- Surfaces Concepts Takeaways
- Importing and Mold Design
- File Translation
- SOLIDWORKS Import Options
- Addressing Translation Errors
- Repairing and Editing Imported Geometry
- Procedure for Rebuilding Fillets

### 2. Core and Cavity

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- Core and cavity mold design modeling
- SOLIDWORKS mold tools
- Mold analysis tools
- Analyzing drafts on a model
- Using the Draft Analysis tool
- Draft analysis options
- Adding drafts
- Scaling the model
- Establishing parting lines
- Shut-off surfaces
- Creating parting surfaces
- Surface bodies
- Creating the mold tooling
- Seeing inside the mold
- Interlocking the mold tooling
- Creating part and assembly files

### 3. Side Cores and Pins

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- Additional Mold Tooling
- Trapped Molding Areas
- Side Cores
- Feature Freeze
- Lifters
- Core Pins
- Manual Selection Techniques
- Modifying Shut-Off Surfaces
- Completing the Tooling

### 4. Advanced Parting Lines Options

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- Manual Parting Lines
- Splitting a Part

### 5. Creating Custom Surfaces for Mold Design

---

- Surface Modeling for Mold Design
- Manual Interlock Surfaces
- Manual Parting Surface Techniques
- Organizing Manual Shut-Off Surfaces

### 6. Advanced Surfacing for Mold Design

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- Surface Modeling for Mold design
- Manual Parting Surface
- Manual Shut-Off Surfaces
- No Fill Shut-Off Surfaces
- Manual Side Cores

see Part 2 on next page »

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PMT2305-ENG

## SOLIDWORKS Mold Design (Part 2)

### 7. Alternative Methods for Mold Design

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- Alternate Methods for Mold Design
- Using Combine and Split
- Creating a Cavity
- Techniques for Mold Tooling

### 8. Reusable Data

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- Reusing Data
- Task Pane
- SOLIDWORKS Resources
- Design Library
- File Explorer
- Library Features
- Configurations in Library Features
- Smart component

### 9. Completing the Mold Base

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- Organizing the Assembly
- Modifying the Lifters
- Lifter Motion
- Ejector Pins
- Cooling the Mold
- Making the Drawing
- Making Changes
- Process Completion

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## SOLIDWORKS Routing: Piping and Tubing – 2 Days (14h)

### 1. Fundamentals of Routing

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- What is Routing?
- Routing setup
- Routing Library Manager
- General Routing Settings

### 2. Piping Routes

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- Piping Routes
- Pipes and Piping Components
- Routing Assembly Templates
- Creating a Piping Route
- Auto Route
- Route Specification Templates

### 3. Advanced Piping Routes

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- Advanced Piping Routes
- Editing a Route
- Routing along existing Geometry

### 4. Piping Fittings

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- Piping Fittings
- Drag-and-Drop a Fitting
- Creating Custom Fittings

### 5. Tubing Routes

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- Tubing Routes
- Tubes and Tubing Components
- Flexible Tubing with Auto Route
- Orthogonal Tubing Routes with Auto Route
- Bend and Spline Errors
- Tubing Drawings

### 6. Piping and Tubing Changes

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- Piping and tubing changes
- Pipe penetrations
- Flange-to-flange connections
- Pipe spools
- Copying routes
- Adding slopes
- Editing piping routes
- Editing for obstructions
- Piping drawings

### 7. Creating Routing Components

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- Routing library parts
- Libraries
- Creating routing library parts
- Pipe and tube components
- Copying routing components
- Routing Library Manager
- Fitting components
- Routing functionality points
- Routing geometry
- Part validity check
- Design table check
- Component attributes
- Elbow components
- Valve components

### 8. Electrical Ducting, Cable Tray, and HVAC Routes

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- Electrical ducting routes
- Cable tray routes
- HVAC routes

### 9. Using SOLIDWORKS Content

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- Using SOLIDWORKS content

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PMT2312-ENG

## SOLIDWORKS What's New – 1 Day (7h)

*\*This course is given using digital files only (no physical book is provided)*

### 1. Welcome to SOLIDWORKS 2022

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- Top enhancements Performance
- SPR fixes
- More information
- Legal notices

### 2. Installation

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- Standard, remote, and compressed administrative images
- Allow subgroups in the Administrator Image Option Editor
- Improved SOLIDWORKS Electrical client installation
- Set up a PDM vault view when creating an administrative image

### 3. Administration

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- Copy settings when options are locked
- Pack and Go includes split and save body features in parts
- Controlling the display of graphics warnings in the Settings Administrator tool

### 4. SOLIDWORKS Fundamentals

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- Changes to system options and document properties
- Application Programming Interface
- Quick Copy settings

### 5. User Interface

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- Dismissed messages
- Message bar
- Redesigned notifications
- Shortcut bar and command search
- General usability
- Reference geometry
- Junk characters in Beta Unicode View resolved
- Component name and description

### 6. Sketching

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- Linear sketch entity as direction reference Pattern and copy text in sketches

### 7. Parts and Features

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- Coordinate systems
- Cosmetic threads
- Draft across parting lines
- External Threaded Stud Wizard
- Hole Wizard slots
- Hybrid modeling
- Mirroring about two planes
- Rotating a section view about a hole or axis
- Thickness analysis resolution
- Redo support for parts

*see Part 2 on next page »*

**Course Objectives :** At the end of each course, students will know the capabilities of the software and will be able to use the learned features.

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**PMT2308-ENG**

**SOLIDWORKS What's New (Part 2)****8. Model Display**

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- Model display performance improvements

**9. Sheet Metal**

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- Edge flanges
- Etched contours on bends

**10. Structure System and Weldments**

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- End cap support
- Custom properties architecture
- Complex corner PropertyManager
- Secondary members
- Connection element for structure systems
- Properties dialog box

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## SOLIDWORKS Refresher – 4 days (28h) (2 sessions)

\* Prerequisite : this course is intended for users who have used SOLIDWORKS or other 3D CAD software for many years.  
\*\*To self-assess your competence to take this training, here is an exemple where at least you should be able to take the CSWA practical exam in less than 90 minutes : <https://www.solidworks.com/sites/default/files/cswasampleexam.zip>

### 1. SOLIDWORKS Basics and the User Interface

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- What is SOLIDWORKS?
- Design intent
- File References
- Opening Files
- The SOLIDWORKS User Interface
- Using the Command Manager

### 2. Patterning

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- Why use Patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed only
- Sketch-Driven Patterns

### 3. Editing: Repairs

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- Part Editing
- Editing Topics
- Sketch Issues

### 4. Editing: Design Changes

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- Part Editing
- Design Changes
- Information from a Model
- Rebuilding Tools
- Sketch Contours
- Replacing Sketch Entities

### 5. Configurations

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- Configurations
- Using Configurations
- Other methods to create Configurations
- Modeling strategies for Configurations
- Editing part Configurations
- Design Library

### 6. Bottom-up Assembly Modeling

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- Bottom-up Assembly
- Creating a new Assembly
- Positioning the first Component
- Feature Manager Design Tree and Symbols
- Adding Components
- Mating Components
- Using part Configurations in Assemblies
- Subassemblies
- Smart Mates
- Inserting Subassemblies
- Pack and Go

### 7. Using Assemblies

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- Using Assemblies
- Analyzing the Assembly
- Checking for clearances
- Changing the values of Dimensions
- Exploded Assemblies
- Rollback and reorder Explode Steps
- Explode Line Sketch
- Bills of Material
- Assembly Drawings
- Collisions and Clearance

see Part 2 on next page »

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## SOLIDWORKS Refresher (Part 2)

### 8. Multibody Design Techniques

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- Multibody parts
- Hide/Show tree items
- Multibody design techniques
- Solid bodies folder
- Local operations
- Feature scope
- Patterning bodies
- Tool Body Technique
- Combining bodies
- Intersect with solid bodies
- Indent feature
- Deleting solid bodies

### 9. Advanced Mate Techniques

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- SOLIDWORKS assemblies
- Assembly file structure
- Opening assemblies
- File references
- Solving mates
- Advanced mate techniques
- Mate references
- Design library parts
- Capturing mate references
- Mates with axes and planes
- Multiple selection mate references
- Multiple Mate mode
- Driven mates
- Using misaligned mates
- Copying multiple components
- Fixed components
- Advanced mate features
- Profile center mate
- Rack pinion mate

### 10. Assembly Editing

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- Assembly Editing
- Information from an Assembly
- Mate errors
- Replacing and Modifying Components
- Converting Parts and Assemblies
- Replacing Components using Save As
- Reloading Components
- Components Patterns

### 11. Using Configurations with Assemblies

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- Using Configurations with Assemblies
- Creating Configurations manually
- Configuration Properties
- Using the Modify
- Configurations Dialog Box
- Changing Configurations using the Context Toolbar
- Managing the Tree Display
- Assembly Evaluation Tools
- Controlling Dimensions in an Assembly
- Equations with Functions
- Sensors
- Using the Mate Controller

### 12. Display States and Appearances

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- Display States
- Bulk Selection Tools
- Advanced Select
- Envelopes
- Appearances, Materials, and Scenes

### 13. Large Assemblies

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- Large Assemblies
- Assembly Modes
- Assembly Visualization
- Lightweight Components
- Large Assembly Settings
- Using SpeedPak
- Using simplified Configurations
- Defeature
- Modifying the structure of an Assembly
- Envelope Publisher
- Large Design Review
- Modes and methods comparison
- Tips for faster Assemblies

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