

solidxperts

 **SOLIDWORKS**

COURSE OUTLINE

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

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Here are the course outlines for all trainings offered at SolidXperts. Training is always given on the latest software version. Also included is all relevant information necessary to apply for grants or accreditation with the appropriate competent authorities.

- **Training organisation :** SolidXperts Inc.
- **Address:** 2650 Marie-Curie, Saint-Laurent, Québec, H4S 2C3
- **Quebec enterprise number (NEQ):** 1160447596
- **Certificate of approval number (Quebec training body):** 0054182

If you have any questions, do not hesitate to contact us.

Thank you for trusting the SolidXperience team for your training needs.



SOLIDWORKS TRAINING

COURSE OUTLINE

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COURSE OUTLINE / SOLIDWORKS TRAINING

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SOLIDWORKS ESSENTIALS – 4 DAYS (28H) (2 SESSIONS)

1. SOLIDWORKS Basics and the User Interface

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

2. Introduction to Sketching

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

3. Basic Part Modeling

- 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

- 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

- Why use Patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Coordinate System
- Symmetric Patterns
- Up to Reference
- Sketch Driven Pattern

6. Revolved Features

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

Part 2 →

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PMT2500-ENG

SOLIDWORKS ESSENTIALS (PART 2)

7. Shelling and Ribs

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

8. Editing: Repairs

- Part Editing
- Editing Topics
- Sketch issues

9. Editing: Design Changes

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

10. Configurations

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

11. Global Variables and Equations

- 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

- Making Drawings
- Removed section
- Detail Views
- Drawing Sheets and Sheet Formats
- Model Views
- Section Views
- Annotations
- Inserting subassemblies
- Pack and Go

13. Bottom-up Assembly Modeling

- 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

- 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

15. Appendix

- Options Settings
- Document Templates

Annex : ToolsXperts Demo



- SaveXperts
- PropsXperts

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SOLIDWORKS DRAWINGS ANSI – 3 DAYS (21H)

1. Review of the Basics

- Review of Essentials
- Drawing System options
- 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 Views

2. Understanding Drawing Templates

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

3. Customizing the Sheet Format

- 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

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

5. Creating Additional Sheet Formats and Templates

- 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

- 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

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PMT2401-ENG

SOLIDWORKS DRAWINGS ANSI (PART 2)

7. Understanding Annotation Views

- 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

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

9. Using Layers, Styles, and the Design Library

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

10. Advanced Options for BOM Tables

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

11. Additional SOLIDWORKS Tables

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

12. Additional Drawing Tools

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

13. Managing Performance

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

Important note regarding the exercises

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 annotations and dimensions favorites, and finally produce your table templates (BOM, revision, etc.).

Annex : ToolsXperts Demo



- PrintXperts
- PropXperts
- Configurations to Drawings

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PMT2401-ENG

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
- Features 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
- Exporting a 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
- SOLIDWORKS Simulation Example

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

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PMT2406-ENG

SOLIDWORKS SHEET METAL (PART 2)

6. Multibody Sheet Metal Parts

- 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/DWG 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

7. Forming Tools and Gussets

- 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

8. Additional Sheet Metal Functions

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

Appendix A : Sheet Metal Tables

- Tables
- Customizing tables

Annex : ToolsXperts Demo



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

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PMT2406-ENG

SOLIDWORKS ASSEMBLY MODELING – 2 DAYS (14H)

1. Advanced Mate Techniques

- SOLIDWORKS Assemblies
- Assembly File Structure
- File References
- 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

- 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

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

4. Assembly Editing

- Assembly Editing
- Mate Errors
- Replacing and modifying Components
- Converting Parts and Assemblies
- Replacing Components Using Save As
- Reloading Components
- Component Patterns
- SOLIDWORKS Simulation Example

5. Using Configurations with Assemblies

- 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

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PMT2502-ENG

SOLIDWORKS ASSEMBLY MODELING (PART 2)

6. Display States and Appearances

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

7. Large Assemblies

- Large Assemblies
- Assembly Modes
- Assembly Visualization
- Lightweight Components
- Large Assembly Settings
- Using SpeedPak
- Using Simplified Configurations
- Automatic Assembly Loading
- 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

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

Annex : ToolsXperts Demo



- PropXperts
- ReferencesXperts
- SaveXperts
- Quantity Report

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PMT2502-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
- SOLIDWORKS Simulation Example

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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PMT2507-ENG

SOLIDWORKS ADVANCED PART MODELING – 2 DAYS (14H)

1. Multibody Design Techniques

- 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 Assembly
- SOLIDWORKS Simulation Example

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

- 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

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

7. Advanced Sweeping

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

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PMT2403-ENG

SOLIDWORKS ADVANCED PART MODELING (PART 2)

8. Introduction to Loft and Boundary Features

- 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

- 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

Annex : ToolsXperts Demo



- SaveXperts
- PropXperts
- Explode configurations

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Competences Evaluation : During the classwork, the instructor will correct the exercises on-demand and explain the solutions to the entire class if needed.

Instructor : SolidXperts trainers are Certified SolidWorks Instructors (CSWI) and authorized by Emploi-Québec.

Course Materials : One or more training manuals are included with the training course.

Attestation : A certificate will be given to each student at the end of the course to attest to the successful completion of the requirements for the course.

PMT2403-ENG

SOLIDWORKS SURFACE MODELING – 2 DAYS (14H)

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

1. Understanding Surfaces

- Solids and Surfaces
- What is a solid?
- Behind the scenes
- Creating solids from surfaces
- Decomposing a Solid into Surfaces
- Additional surface concepts
- Take Aways
- Why use surfaces?
- Continuity explained
- Workflow with surfaces

2. Introduction to Surfacing

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

3. Solid-Surface Hybrid Modeling

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

3. Repairing and Editing Imported Geometry

- 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

- Smoothing patches
- Boundary surfaces
- Corner blends

6. Complex Blends

- Complex blends
- Freeform feature

7. Advanced Surface Modeling

- Process stages
- Modeling the lower half
- Design changes

8. Master Model Techniques

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

Annex : ToolsXperts Demo



- SaveXperts
- ScaleXperts

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.

Training Course : Training is given in class at SolidXperts or online where each student has access to a workstation or online product version.

Methodology : Training is based on case studies demonstrated by the instructor. At the end of each lesson, time will be given for exercises.

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PMT2504-ENG

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

- Course overview
- Surfaces in Mold Design
- Geometry vs Topology
- Creating Solids from surfaces
- Decomposing a solid into surfaces
- Additional surface concepts
- Surfaces and Takeaways
- Importing and Mold Design
- File translation
- SOLIDWORKS Import options
- Comparing geometry
- Addressing translation errors
- Repairing and editing imported geometry
- Procedure for rebuilding Fillets

2. Core and Cavity

- Core and cavity mold design
- SOLIDWORKS mold tools
- Mold analysis tools
- Analyzing draft on a model
- Using the Draft Analysis Tool
- Draft Analysis Options
- Adding Draft
- Scaling the Model
- Establish the parting lines
- Shut-Off Surfaces
- Creating the Parting Surface
- Surface bodies
- Creating the Mold Tooling
- Seeing Inside the Mold
- Interlocking the mold tooling
- Creating Part and Assembly Files

3. Side Cores and Pins

- 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

- Manual Parting Line
- Using Split Faces
- Using Entities to Split
- Creating Ruled Surfaces

5. Creating Custom Surfaces for Mold Design

- Surface modeling for Mold Design
- Manual Interlock Surfaces
- Using Select Partial Loop
- Ruled Surface Direction
- Creating the parting surface
- Organizing surfaces
- Manual Parting Surface Techniques
- Organizing manual shut-off surfaces
- Copying surfaces

6. Advanced Surfacing for Mold Design

- Surface Modeling for Mold Design
- Manual Parting Surface
- Insert Mold Folders
- Manual Shut-off Surfaces
- No Fill Shut-off Surfaces
- Manual Side Cores

Part 2 →

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Methodology : Training is based on case studies demonstrated by the instructor. At the end of each lesson, time will be given for exercises.

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PMT2505-ENG

SOLIDWORKS MOLD DESIGN (PART 2)

7. Alternative Methods for Mold Design

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

Annex : ToolsXperts Demo



- SaveXperts
- ScaleXperts

8. Reusable Data

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

9. Completing the Mold Base

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

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PMT2505-ENG

SOLIDWORKS ROUTING: PIPING AND TUBING – 2 DAYS (14H)

1. Fundamentals of Routing

- What is Routing?
- Types of Routes
- Routing Feature Manager
- Routing Setup
- Routing Library Manager
- General Routing Settings

2. Piping Routes

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

3. Advanced Piping Routes

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

4. Piping Fittings

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

5. Tubing Routes

- 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

- 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

- 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
- SOLIDWORKS Simulation Example

8. Electrical Ducting, Cable Tray, and HVAC Routes

- Electrical ducting routes
- Cable tray routes
- HVAC routes

9. Using SOLIDWORKS Content

- Using SOLIDWORKS content

Annex : ToolsXperts Demo



- BeamCutXperts
- Tubing
- Flow Simulation

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.

Training Course : Training is given in class at SolidXperts or online where each student has access to a workstation or online product version.

Methodology : Training is based on case studies demonstrated by the instructor. At the end of each lesson, time will be given for exercises.

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PMT2512-ENG

SOLIDWORKS WHAT'S NEW – 1 DAY (7H)

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

**The content of the training will be adjusted according to the experience of the audience and the topics covered according to the customer's needs.

***This training does not include practical exercises, so it is not necessary to have a workstation with a valid SOLIDWORKS license.

****The duration of the training can vary from half a day to a full day. Please confirm with your account manager.

1. Welcome to SOLIDWORKS 2025

- Top Enhancements

2. Using SOLIDWORKS on the 3DEXPERIENCE Platform

- Removal of Option to Generate 3D Format
- Visibility of Quantity Column
- Linking Configuration Properties of Physical Products to Representations
- Licensing Support for SOLIDWORKS CAM, Inspection, and MBD Add-Ins
- Task Pane

3. Installation

- Convert SolidNetWork License Server to 64-Bit
- Installing the SOLIDWORKS Manage Web API

4. Administration

- Inheriting Default File Locations When Upgrading to SOLIDWORKS 2025
- SOLIDWORKS Login Manager

5. SOLIDWORKS Fundamentals

- Changes to System Options and Document Properties
- Application Programming Interface
- Specifying a Z-Up Template
- Saving SOLIDWORKS Inspection Files Using Bookmarks

6. User Interface

- Reorganize Components
- Usability
- Hole Wizard
- Save and Auto Save Progress
- Create Document Group
- Creating Multiple Files as a Document Group
- Updating a Document Group

7. Sketching

- Repairing Dangling Relations
- Linear and Circular Sketch Patterns

Part 2 →

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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SOLIDWORKS WHAT'S NEW (PART 2)

8. Parts and Features

- Defeature Silhouette Method for Parts
- Patterning Reference Geometry
- Converting Mesh BREP to Standard BREP
- Segment Mesh Enhancements
- Move/Copy Body Features
- Variable Size Fillets
- Curve Through XYZ Points Enhancements

9. Sheet Metal

- Bend Notches
- Creating Bend Notches
- Bend Notch PropertyManager
- Tab and Slot
- Tab and Slot PropertyManager
- Multi Length Edge Flanges and Automatic Flange Length Dimensions
- Performance Improvements in Cosmetic Thread Features
- Performance Improvements in Rebuilding Drawings

10. Structure System and Weldments

- Accessing and Working with Favorite Profiles
- Complex Corner PropertyManager and Structure System
- Trimming Attached Members
- Groove Beads
- Creating Groove Beads
- Groove Bead PropertyManager

11. Assemblies

- Assembly Visualization
- SpeedPak Instances
- Interference Detection in Large Design Review Mode
- Performance Evaluation
- Linking Display State to the Patterned Seed Component
- Inserting Assemblies with Rolled-Back Features
- Copy with Mates
- Performance When Calculating Mass Properties
- Controlling the Visibility of Part Sketches in Assemblies

12. Detailing and Drawings

- Additional Tolerance Types for Chamfer Dimensions
- BOM Quantity Override for Detailed Cut Lists
- Creating Surface Finish Symbols in Conformance with ISO 21920
- Performance Improvements in Drawings

13. Configurations

- Display State Tables

14. Import/Export

- Exporting Custom Properties to IFC Files
- Importing Extended Reality Files

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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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 example 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

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

2. Patterning

- Why use Patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Coordinate System
- Up To Reference
- Sketch-Driven Patterns

3. Editing: Repairs

- Part Editing
- Editing Topics
- Sketch Issues

4. Editing: Design Changes

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

5. Configurations

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

6. Bottom-up Assembly Modeling

- 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

- 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

Part 2 →

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.

Training Course : Training is given in class at SolidXperts or online where each student has access to a workstation or online product version.

Methodology : Training is based on case studies demonstrated by the instructor. At the end of each lesson, time will be given for exercises.

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PMT2408-ENG

SOLIDWORKS REFRESHER (PART 2)

8. Multibody Design Techniques

- 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

- 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

- 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

- 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
- Creating an Equality
- Equations with Functions
- Sensors
- Using the Mate Controller

12. Display States and Appearances

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

13. Large Assemblies

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

Annex : ToolsXperts Demo



- PropXperts
- SaveXperts
- Configuration to Drawing

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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PMT2408-ENG

CERTIFIED SOLIDWORKS PROFESSIONAL TRAINING - 0.5 DAY (3.5H)

*This course is useful to any SOLIDWORKS user who is seeking to earn their Certified SOLIDWORKS Professional Certification.

**Topics covered include redeeming your voucher, areas of study, and methods of study.

***This course does not train attendees on the skills required to pass, prior knowledge is required (see SOLIDWORKS Refresher)

1. How To Access The Certification

- Requirements to utilize complementary vouchers
- Creating a 3D Experience Certification Account
- Obtaining a 3D Experience ID
- Redeeming your voucher
- Downloading Tangix TesterPro Client
- Launching the test

3. How To Prepare To Pass

- SOLIDWORKS Tutorials
- Available Classes (Refresher)
- YouTube Channels / Tutorials
- Online Resources
- Basic in Class Demos

2. Topics Of The Tests

- Parts
 - Equations
 - Anaylasis
 - Modifications
- Configurations
- Assemblies
 - Mates
 - Assembly Modeling
 - Analysis

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.

Training Course : Training is given in class at SolidXperts or online where each student has access to a workstation or online product version.

Methodology : Training is based on case studies demonstrated by the instructor. At the end of each lesson, time will be given for exercises.

Competences Evaluation : During the classwork, the instructor will correct the exercises on-demand and explain the solutions to the entire class if needed.

Instructor : SolidXperts trainers are Certified SolidWorks Instructors (CSWI) and authorized by Emploi-Québec.

Course Materials : One or more training manuals are included with the training course.

Attestation : A certificate will be given to each student at the end of the course to attest to the successful completion of the requirements for the course.