

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.

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.

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 →

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.

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

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.



SOLIDXPERTS TRAINING

COURSE OUTLINE

TABLE OF CONTENT

COURSE OUTLINE / SOLIDXPERTS TRAINING

SOLIDWORKS PERFORMANCES – 2 days (14h)

27

CAD ADMINISTRATORS – 2 days (14h)

28

SOLIDWORKS PERFORMANCE – 2 DAYS (14H)

Prerequisites : This training is intended for users with at least 1 year of experience
This course is given using digital files only (no physical book is provided)
This training is only available remotely

1. Structure of Default Templates

- What aspects can influence file size?
- Impact of Tessellation Quality
- Number of configurations in a file
- SOLIDWORKS last saved version
- Oldest features in a SOLIDWORKS file
- Number of in-context features

2. Work Methods

- Number of fixed components in an assembly
- Using the flexible feature
- Assembly Features
- Temporary update folder from external references
- Number of parts per assembly level
- Parts rebuild time
- Number of features in a part
- Errors in Solid Bodies
- Quantity of errors in features
- Quantity of reference errors
- Use of the freeze bar

3. Managing Imported Files

- Number of appearances in file
- Number of bodies in part
- Number of surfaces in a part
- How to reduce file size of imported files
- How to manage libraries from “McMaster, Parker etc.”

4. Workflow of Different Quick Open Modes

- Large design review
- Lightweight components
- SpeedPack
- Parts in simple version
- Detailing mode
- Open an assembly in simple version

5. Drawing Performance

- Impact on drawing performance
- View in draft quality vs. High quality
- Impact of Section Views
- Line format
- Automatic view updates
- Quick Open Tips
- Impact of Sketches in a Drawing
- Impact of configurations in views

Annex : ToolsXperts Demo



- DoctorXperts
- Explode Configurations
- ReferencesXperts

SPECIFIC NOTE FOR THIS TRAINING

For reasons of convenience, collaboration and interactivity and also to allow you to obtain concrete examples on your workstations and with your files, **this training is only available remotely.**

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.

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

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.

CAD ADMINISTRATORS – 2 DAYS (14H)

Prerequisites : Experience as an engineering manager or in IT.
Understanding of the Windows operating system.
This training is only available remotely

1. SOLIDWORKS Computer Requirements

- Analysis of computer hardware
- Review of computer needs to ensure good performance and stability
- Certified graphics card drivers

2. Installing SOLIDWORKS

- Review of installation techniques on a new workstation
- Review of installation techniques on a workstation with a previous SOLIDWORKS
- Creation of an administrative image for a large deployment
- Advanced uninstallation technique

3. File Management

- Network server maintainability and speed analysis
- Network directory naming best practices
- Antivirus Exceptions list
- Different types of network

4. System Settings and Document Templates

- Analysis of SOLIDWORKS system parameters
- Validity of the need for system parameters changes
- Analysis of the Sharing of system parameters between users
- Management of document templates
- Explanation and proof of document templates changes
- Property tab builder

5. Collaborative Environments

- Analysis of shareable elements (ex: Material database, Custom Bill of Materials, Toolbox, weldments, etc.)
- Maintenance of shareable items
- Files location management
- Multi-user environment
- SOLIDWORKS Search Routine
- Understanding the Pack and Go
- Design Library

6. Tools for CAD administrators

- CAD Admin Dashboard
- Design Checker
- Best practice document

SPECIFIC NOTE FOR THIS TRAINING

For reasons of convenience, collaboration and interactivity and also to allow you to obtain concrete examples on your workstations and with your files, **this training is only available remotely.**

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.

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

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.



3DEXPERIENCE WORKS TRAINING

COURSE OUTLINE

TABLE OF CONTENT

COURSE OUTLINE / 3DEXPERIENCE WORKS

<u>3DEXPERIENCE CAD ADMINISTRATOR – 2 days (14h)</u>	<u>31</u>
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<u>3DEXPERIENCE NON-CAD USER – 1 day (7h)</u>	<u>33</u>

3DEXPERIENCE CAD ADMINISTRATOR – 2 DAYS (14H)

Prerequisites : Students need to have their own 3DEXPERIENCE Platform
Students must be Admins of their own 3DEXPERIENCE Platform
This course is given using digital files only (no physical book is provided)

1. 3DX Platform Overview

- Navigating the User Interface

2. 3DDashboard

- What is a 3DDashboard?
- Platform Management
- Rename Platform

3. Validation of Roles / Licences

- Validating Roles
- Validating Quantity of Licences

4. Inviting Members

- Create User Groups
- Inviting Members
- Members' configurations
- Assigning Roles

5. 3DSwym

- 3DSwym Overview
- Managing Community Administrators
- Managing Community Parameters

6. 3DSpace

- Creating and Configuring a 3DSpace
- Locating the files
- 3DDrive vs. 3DSpace

7. Bookmark Editor

- Introduction to the Interface

8. Adding Content / File Management

- Adding Non-CAD Content to 3DX
- Adding CAD Content to 3DX
- File Preparation Assistant
- My Session

9. Custom Dashboards

- Creation and Allocation of Custom Dashboards
- Exercise : Creating a Custom Dashboard

10. 3DEXPERIENCE & SOLIDWORKS

- Connecting SOLIDWORKS to the 3DX Platform
- Exercise : Cloud Eligibility Tool
- Exercise : Installing xCAD PLM connector
- Exercise : Checking live status of the Platform

SPECIFIC NOTES FOR THIS TRAINING

Description : Basic training on the initial configuration and use of a 3DEXPERIENCE platform. The training and the concepts explained will be supported by a presentation on our platform, but the practical exercises will be done on the student's platform. The purchase of an access to the platform is a prerequisite to the training.

Objectives : To be able to understand how the platform works and then configure your own platform according to your organization's needs.

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.

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

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.

3DEXPERIENCE CAD USER – 2 DAYS (14H)

Prerequisites : Students need to have their own 3DEXPERIENCE Platform
Students must possess the following roles : “3DSwymmer”, “Collaborative Industry Innovator” and “Collaborative Designer for SOLIDWORKS”
This course is given using digital files only (no physical book is provided)

1. 3DX Platform Overview

- Navigating the User Interface

2. The Roles

- 3DSwymmer
- Collaborative Industry Innovator
- Collaborative Designer for SOLIDWORKS

3. 3DEXPERIENCE & SOLIDWORKS

- Connecting SOLIDWORKS to the 3DX Platform
- Exercise : Cloud Eligibility Tool
- Exercise : Installing xCAD PLM connector
- Exercise : Checking live status of the Platform

4. 3DDashboard

- What is the purpose of 3DDashboard?
- Exercise : Cockpits & Welcome App

5. 3DDrive vs. 3DSpace

- Locating the files
- Exercise : Saving a SOLIDWORKS Assembly
- Exercise : Share & Markup

6. SOLIDWORKS content lifecycle and access control

- Revisions and Maturity States
- Exercise : Using SOLIDWORKS content and collaborative space

7. Document management

- Bookmarks
- CAD Document Management
- Exercise : Create a Bookmark and Content
- Exercise : Editing, Managing and Deleting CAD & Non-CAD files

8. Search on the platform

- Adding and Using 6WTags
- Exercise : Practice searching for content

9. Collaborative Task

- Creating and Managing Tasks
- Exercise : Using Tasks in SOLIDWORKS

10. Product Structure Explorer App

- Export .csv for BOMs
- Pack and Go (Export As)

11. Relations

- In SOLIDWORKS
- In Browser

12. Compare

- Compare App
- Compare in 3DPlay

13. 3DSwym

- Working with a 3DSwym Community

SPECIFIC NOTES FOR THIS TRAINING

Description : Basic training on the usage of the 3DEXPERIENCE platform related to the «Collaborative Designer for SOLIDWORKS» role. The training and the concepts explained will be supported by a presentation on our platform, but the practical exercises will be done on the student's platform. The purchase of an access to the platform is a prerequisite to the training.

Objectives : Be able to use the platform properly and understand its main functions (Dashboards, saving files, collaboration tools, etc.).

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.

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

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.

3DEXPERIENCE NON-CAD USER TRAINING – 1 DAY (7H)

Prerequisites : Students need to have their own 3DEXPERIENCE Platform
Students must possess the following roles : "3DSwymmer", "Collaborative Industry Innovator" and "Collaborative Designer for SOLIDWORKS"
*This course is given using digital files only (no physical book is provided)

1. 3DX Platform Overview

- Navigating the User Interface

2. The Roles

- 3DSwymmer
- Collaborative Industry Innovator

3. 3DDashboard

- What is the purpose of 3DDashboard?
- Platform Management
- Exercise : Cockpits & Welcome App

4. 3DDrive vs. 3DSpace

- Locating the files
- Install the 3DDrive Locally
- Exercise : Exporting non-CAD files

5. Bookmarks

- Exercise : Create a bookmark and add content

6. Search on the platform

- Practice searching for Content

7. 3DPlay

- Hidden Functions
- Validating 6WagTs

8. Collaborative Tasks

- Working with Tasks

9. 3DSwym

- Collaborating in a 3DSwym Community
- Access the platform and user's community

10. 3DMarkup

- Share & Markup

SPECIFIC NOTES FOR THIS TRAINING

Description : Basic training on the usage of the 3DEXPERIENCE platform for a non-CAD user, related to the «Collaborative Industry Innovator» role. The training and the concepts explained will be supported by a presentation on our platform, but the practical exercises will be done on the student's platform. The purchase of an access to the platform is a prerequisite to the training.

Objectives : Be able to use the platform properly and understand its main functions (Dashboards, saving files, collaboration tools, etc.).

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.

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

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Course Materials : This course is given using digital files only (no physical book is provided)

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.



SOLIDWORKS SIMULATION TRAINING

COURSE OUTLINE

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SOLIDWORKS SIMULATION STATIC – 3 DAYS (21H)

1. The Analysis Process

- The analysis process
- SOLIDWORKS Simulation options
- Preprocessing
- Meshing
- Processing
- Postprocessing
- Multiple studies
- Reports
- Summary
- References

2. Mesh Controls, Stress Concentrations, and Boundary Conditions

- Mesh Control
- Understanding the effect of Boundary Conditions

3. Assembly Analysis with Interactions

- Interaction Analysis
- Study Properties
- Contact or Bonded interaction
- Local Interaction

4. Symmetrical and Free Self- Equilibrated Assemblies

- Shrink Fit Parts
- Analysis with Soft Springs

5. Assembly Analysis with Connectors and Mesh Refinement

- Problem Statement
- Remote Load/Mass
- Connectors
- Mesh Control in an Assembly
- Mesh Plots

6. Bonded Mesh Options

- Bonded Mesh Options
- Centrifugal Force
- Cyclical Symmetry
- Bonding Options
- Bonding Formulation

7. Analysis of Thin Components

- Thin Components
- Mesh with Solid Elements
- Refined Solid Mesh
- Solid vs. Shell
- Creating Shell Elements
- Shell Elements - Mid-plane surface

8. Mixed Meshing - Shells & Solids

- Mixed meshing - Solids and Shells

9. Beam Elements- Analysis of a Conveyor Frame

- Beam and Truss elements

10. Mixed Meshing Solids, Beams & Shells

- Mixed Meshing
- Beam Imprint

11. Design Study

- Multiple load cases
- Geometry modification

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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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.

PMT2540-ENG

SOLIDWORKS SIMULATION STATIC (PART 2)

12. Thermal Stress Analysis

- Thermal stress analysis
- Saving model in a deformed shape

13. Adaptive Meshing

- Adaptive meshing
- h-Adaptivity study
- p-Adaptivity study
- h vs. p Elements – Summary

14. Large Displacement Analysis

- Small vs. Large displacement analysis
- Small displacement Linear analysis
- Large displacement Nonlinear analysis

Annex

- Meshing Strategy
- Geometry Preparation
- Meshing Quality
- Mesh Controls
- Meshing Stages
- Failure Diagnosis
- Tips for the Shell Elements Usage
- Hardware Requirements for Meshing
- Solvers in SOLIDWORKS Simulation
- Solver Selection

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PMT2540-ENG

SOLIDWORKS MOTION – 2 DAYS (14H)

1. Introduction to Motion Simulation and Forces

- Basic motion analysis
- Forces
- Results

2. Building a Motion Model and Post Processing

- Creating local mates
- Mates
- Local mates
- Power
- Plotting kinematic results

3. Introduction to Contacts, Springs and Dampers

- Contact and friction
- Contact
- Contact groups
- Contact friction
- Translational spring
- Translational damper
- Post-processing
- Analysis with friction (optional)

4. Advanced Contact

- Latch forces
- STEP function
- Contact: Solid bodies
- Geometrical description of contacts
- Integrators
- Instability points

5. Curve to Curve Contact

- Contact forces
- Curve to curve contact
- Solid bodies vs. Curve to curve contact
- Solid bodies contact solution

6. Cam synthesis

- Cams
- Trace path
- Exporting trace path curves

7. Motion Optimization

- Motion Optimization
- Sensors
- Optimization analysis

8. Flexible Joints

- Flexible joints
- System with Flexible Joints

9. Redundancies

- Redundancies
- How to check for redundancies
- Typical redundant mechanisms

10. Export to FEA

- Exporting results
- Export of loads
- Direct solution in SOLIDWORKS Motion

11. Event Based Simulation

- Event based simulation
- Servo motors
- Sensors
- Task

12. Design Projects (Optional)

- Design Project
- Self-guided problems
- Problem solution
- Creating the force function
- Force expression

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PMT2442-ENG

SOLIDWORKS SIMULATION PROFESSIONAL – 2 DAYS (14H)

*The "SOLIDWORKS Simulation Static" Training is required for this class.

1. Frequency Analysis of Parts

- Modal analysis basics
- Frequency Analysis with Supports
- Frequency Analysis without Supports
- Frequency Analysis with Load

2. Frequency Analysis of Assemblies

- All Bonded Interaction Conditions
- Bonded and Free Interactions

3. Buckling Analysis

- Buckling Analysis
- Linear vs Nonlinear Buckling analysis

4. Load Cases

- Load Cases

5. Submodeling

- Submodeling

6. Topology Analysis

- Topology Analysis
- Goals and Constraints
- Manufacturing Controls
- Mesh Effects
- Load Cases in Topology

7. Thermal Analysis

- Thermal Analysis Basics
- Steady-State Thermal Analysis
- Transient Thermal Analysis
- Transient Analysis with time varying Load
- Transient Thermal Analysis using a Thermostat

8. Thermal Analysis with radiation

- Steady State Analysis

9. Advanced Thermal Stress 2D Simplification

- Thermal Analysis
- Thermal Stress Analysis
- 3D model

10. Fatigue Analysis

- Fatigue
- Stress-life (S-N) based fatigue
- Thermal Study
- Thermal Stress Study
- Fatigue Terminology
- Fatigue Study
- Fatigue Study with dead load

11. Variable Amplitude Fatigue

- Fatigue Study

Part 2 →

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PMT2541-ENG

SOLIDWORKS SIMULATION PROFESSIONAL (PART 2)

12. Drop Test Analysis

- Drop Test Analysis
- Rigid Floor Drop Test
- Elastic Floor/Elasto-Plastic Material
- Elasto-Plastic Material Model
- Drop Test with Contact Interaction

13. Optimization Analysis

- Optimization Analysis
- Static and Frequency Analysis

14. Pressure Vessel Analysis

- Pressure esVsel Analysis
- Manhole Nozzle Flange and Cover

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PMT2541-ENG

SOLIDWORKS SIMULATION PREMIUM – 3 DAYS (21H)

*The "SOLIDWORKS Simulation Static" Training is required for this class.
**The "SOLIDWORKS Simulation Professional" is required for this class.

NON LINEAR

1. Large Displacement Analysis

- Linear static analysis
- Nonlinear static study
- Linear static study (Large displacement)

2. Incremental Control Techniques

- Incremental control techniques
- Linear analysis
- Nonlinear analysis – Force control
- Nonlinear analysis – Displacement control

3. Nonlinear Static Buckling Analysis

- Linear buckling
- Linear static study
- Nonlinear symmetrical buckling
- Nonlinear asymmetrical buckling

4. Plastic Deformation

- Plastic deformation
- Problem statement
- Linear elastic
- Nonlinear Study with Linear Material
- Nonlinear – von Mises
- Nonlinear – Tresca's
- Stress accuracy
- Using Nonlinear Elastic Material

5. Hardening Rules

- Hardening rules
- Isotropic hardening
- Kinematic hardening

6. Analysis of Elastomers

- Two constant Mooney-Rivlin (1 material curve)
- Two constant Mooney-Rivlin (2 material curves)
- Two constant Mooney-Rivlin (3 material curves)
- Six constant Mooney-Rivlin (3 material curves)

7. Nonlinear Interaction Analysis

- Connections
- Dynamic Solutions

8. Metal Forming

- Bending

DYNAMICS

1. Vibration of a Pipe

- Static analysis
- Frequency analysis
- Dynamic analysis (slow force)
- Dynamic analysis (Fast force)

2. Transient Shock Analysis According to MILS- STD-810H

- Run Frequency
- Damping

3. Harmonic Analysis of a Bracket

- Harmonic analysis of a bracket

4. Response Spectrum Analysis

- Response Spectrum Analysis
- Response Spectrum

5. Random Vibration Analysis According to MIL-STD-810G

- Random vibration analysis according to MIL-STD-810G

6. Random Vibration Fatigue

- Material properties, S-N curve
- Random vibration fatigue options

7. Nonlinear Dynamic Analysis of an Electronic Enclosure

- Linear dynamic analysis
- Nonlinear dynamic 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.

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PMT2544-ENG

SOLIDWORKS SIMULATION PREMIUM COMPOSITE – 1 DAY (7H)

*The "SOLIDWORKS Simulation Static" training is required for this class.
**The "SOLIDWORKS Simulation Professional" training is required for this class.

- Introduction to Composites
- Objectives
- Composite Materials
- Composite Lamina
- Composite Laminate
- SOLIDWORKS Simulation Premium: Composites
- Composite Post Processing
- Case Study: Mountain Board
- Project Description
- Stages in the Process
- Lamina Properties
- Experimental Measurements
- Micromechanics
- Required Parameters
- Strength Parameters
- Composite Options
- Composite Orientation
- Offset
- Shell Alignment
- Composite Post Processing
- Stresses
- Inter Laminar Shear
- Failure Criterion
- Shear Stresses

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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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.

SOLIDWORKS FLOW SIMULATION – 2 DAYS (14H)

1. Creating a SOLIDWORKS Flow Simulation Project

- Model Preparation
- Post-Processing

2. Meshing

- Computational Mesh
- Basic Mesh
- Initial Mesh
- Geometry Resolution
- Result Resolution/Level of initial Mesh
- Control Planes

3. Thermal Analysis

- Fans
- Perforated Plates

4. External Transient Analysis

- Reynolds Number
- External Flow
- Transient Analysis
- Turbulence Intensity
- Solution adaptive Mesh refinement
- Two-dimensional Flow
- Computational Domain
- Calculation control options
- Time animation

5. Conjugate Heat Transfer

- Conjugate Heat transfer
- Real Gases

6. EFD Zooming

- EFD Zooming

7. Porous Media

- Porous media
- Design modification

8. Rotating Reference Frames

- Rotating reference frame
- Averaging
- Noise Prediction
- Sliding Mesh
- Tangential faces of rotors
- Time step
- Axial Periodicity

9. Parametric Study

- Parametric analysis
- Steady state analysis

10. Free Surface

- Free Surface

11. Cavitation

- Cavitation

12. Relative Humidity

- Relative Humidity

13. Particle Trajectory

- Particle Trajectory

14. Supersonic Flow

- Supersonic Flow

15. FEA Load Transfer

- FEA Load Transfer

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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PMT2543-ENG

SOLIDWORKS FLOW SIMULATION: HVAC MODULE – 1 DAY (7H)

*The "SOLIDWORKS Flow Simulation" Training is required for this class.

1. Introduction to HVAC

- Objectives
- HVAC Module
- Case Study: Office
- Project Description
- Radiation
- Radiation Transparency
- Radiation Source
- Radiative Surface
- Discussion
- Comfort Parameters
- Conclusions

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.

SOLIDWORKS FLOW SIMULATION: ELECTRONIC COOLING MODULE – 1 DAY (7H)

*The "SOLIDWORKS Flow Simulation" training is required for this class.

1. Introduction to Electronics Module

- Objectives
- Electronic Module
- Case Study: Computer Box
- Conclusions

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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SOLIDWORKS PLASTICS – 1.5 DAY (10H), 2 DAY (14H) OR 3 DAY (21H)

1. Basic Fill Analysis

- Basic Fill Analysis
- Injection Process
- Element Types
- Units
- User Interface
- Injection Units
- Material
- Boundary Conditions
- Injection Location
- Create Mesh
- Simulation Type
- Running a Fill Analysis
- Fill Results

2. Detecting a Short Shot

- Detecting Short Shots
- Fill Properties
- Flow Front Central Temperature
- Configurations

3. Automation Tools

- Automation Tools
- Duplicate Study
- Plastics File Management
- Batch Manager

4. Injection Locations and Sink Marks

- Injection Locations and Sink Marks
- Injection Location Rules
- Visibility Commands
- Sink Marks

5. Materials

- Materials Properties
- User-Defined Database
- Resin Properties
- Temperature Properties
- Polymer types
- Thermal Properties
- Rheological Properties
- PVT Data
- Thermo-Mechanical Properties

6. Mesh Manipulation

- Mesh Manipulation
- Local Mesh Refinement
- Edit/Review
- Element Issues
- Leader Lines
- Edit Study
- Solid Mesh
- Solid Mesh Size

7. Detecting Air Traps

- Detecting Air Traps
- Air Traps
- Venting
- Solver settings

8. Gate Blush

- Gate Blush
- Runner Elements

9. Packing and Cooling Times

- Pack and Cooling
- Flow/Pack Switch
- Pack Stage
- Pack Analysis
- Pack Results
- X-Y Plot
- Clipping Plane Mode
- Isosurface Mode
- Cooling Times

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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PMT2539-ENG

SOLIDWORKS PLASTICS (PART 2)

10. Multiple Cavity Molds

- Multiple Cavity Molds
- Mold Layouts
- Runner System
- Runner Channel Design
- Clamping Force
- Family Mold Layout
- Using Runner-Balancing

11. Symmetry Analysis

- Symmetry Analysis
- Symmetry
- Cyclic Symmetry

12. Valve Gates and Hot Runners

- Hot Runners
- Valve Gates

13. Reaction Injection Molding

- Reaction Injection Molding

14. Using Inserts

- Using Inserts
- Inserts
- Metal Material Database

16. Co-Injection Molding

- Co-Injection Molding
- Thick Parts

17. Bi-Injection Molding

- Bi-Injection Molding
- Copy and Paste
- Bi-Injection
- Injection Start Value

18. Cooling Analysis

- Cooling Analysis
- Cooling
- Cooling Channels and Mold Bodies
- Baffle
- Bubbler
- Cooling Simulations
- Coolant
- Mold
- Cool Parameters
- Cool Analysis
- Cool Results

19. Warpage Analysis

- Warpage Analysis
- Shrinkage
- Warpage
- Warp Parameters
- Warp Results
- Reducing and Fixing Warped Parts

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.

PMT2539-ENG



SOLIDWORKS PDM TRAINING

COURSE OUTLINE

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COURSE OUTLINE / SOLIDWORKS PDM

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SOLIDWORKS MANAGE - 2 days (14h)

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ADMINISTRATION OF SOLIDWORKS PDM PROFESSIONAL – 3 DAYS (21H)

1. Installation Planning

- Planning for SOLIDWORKS PDM Professional
- Planning Process
- Training Scenario
- Installation Process

2. The Administration Tool

- PDM Professional Administration Tool
- Creating a File Vault
- Creating a Local File Vault View

3. Users and Groups

- Creating Users
- Creating Groups

4. Folder Card Creation

- Data Cards Types
- Card Editor
- Design a Folder Data Card
- Serial Numbers
- Card Lists

5. File and Search Cards

- Importing Data Cards
- Design a File Data Card
- Design a Search Data Card

6. Columns and Bill of Materials (BOM) Views

- File List Columns
- Search Columns
- Bills of Material Columns

7. Workflow

- Workflows
- Categories
- Revisions
- Drawings Revisions Tables

8. Notifications and Tasks

- Notifications
- Conditional Notifications
- Tasks
- Various Conversion Tasks

9. Folder Templates

- Creating Folder Templates

10. File Templates

- Creating File Templates

11. Data Migration

- Migrating Legacy Data
- Data Migration
- Migrating Revisions

12. Vault Backup

- Backing Up Files Vaults

Appendix

- File Types and Settings
- Data Import/Export
- Toolbox Setup

Not included :

- Installation Process
- Routing Configuration
- CircuitWorks Configuration

Annex : ToolsXperts Demo



- PDMxperts

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

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PMT2534-ENG

ADMINISTRATION OF SOLIDWORKS PDM STANDARD – 2 DAYS (14H)

1. Installation Planning

- Planning Process
- Data Management Plan
- Implementation Plan
- Fundamentals of PDM

2. The Administration Tool

- Administration Tool
- Creating a Files Vault
- Creating a Local Files Vault View

3. Users and Groups

- Creating Users
- Creating Groups

4. Folder Card Creation

- Data Card Types
- Cards Editor
- Design a Folder Data Card

5. File and Search Cards

- Importing Data Cards
- Design a File Data Card
- Design a Search Data Card

6. Columns and Bill of Materials (BOM) Views

- File List Columns
- Search Columns
- Bill of Materials Columns

7. Workflow

- Creating One Workflow
- Conditions
- Revisions
- Drawing Revisions Tables

8. Notifications and Tasks

- Notifications
- Task (conversion as PDF only)

9. Data Migration

- Migrating Legacy Data
- Data Migration
- Migrating Revisions

10. Vault Backup

- Backing up File Vaults

Appendix

- File Types and Settings
- Toolbox Setup

Not included :

- Installation Process
- Routing Setup
- CircuitWorks Setup

Annex : ToolsXperts Demo



- PDMxperts

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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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.

PMT2532-ENG

USING OF SOLIDWORKS PDM – 1 DAY (7H)

1. SOLIDWORKS PDM Concepts

- Fundamentals of PDM
- What is SOLIDWORKS PDM?
- SOLIDWORKS PDM Overview
- SOLIDWORKS PDM Modules
- SOLIDWORKS PDM Components

2. SOLIDWORKS PDM User Interface

- SOLIDWORKS PDM User Interface

3. Document Creation and Check In

- Document Creation and Check In
- Creating New Folders and Files
- Adding Existing Files
- Documents Check In
- Advanced Documents Check In

4. Versioning Files

- Versioning Files

5. File References

- File References
- Copying Files and References
- Move Files and References (PDM Professional Only)
- Sharing Files (PDM Professional Only)

6. Searching

- Searching in SOLIDWORKS PDM
- Favorite Searches (PDM Professional Only)

7. Workflow and Notification

- SOLIDWORKS PDM Workflow
- Change State
- Notifications

8. Working in SOLIDWORKS*

- Versioning SOLIDWORKS Files
- SOLIDWORKS Add-In
- Add-In Options
- Managing local cache

** Lesson for SOLIDWORKS users only*

Appendix

- Use of Named Bill of Materials (BOM)
- (PDM Professional only)

SPECIFIC NOTE TO THIS TRAINING

Target Audience : Users not working with SOLIDWORKS don't need to take the last lesson of the training;

Training Location : This training is generally given to a larger number of participants. In order to accommodate customers, this training is given at the customer's office or online rather than in a computer lab at SolidXperts.

Methodology : The training is based on case studies demonstrated by the trainer. Due to the ease of use of PDM, there is no practical exercise at the end of each lesson.

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.

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.

SOLIDWORKS Version : We always train on the latest product version.

PMT2531-ENG

UPDATE SOLIDWORKS PDM STANDARD TO PROFESSIONAL – 1 DAY (7H)

* Numbers are not in sequential order since they're referring to the chapters of the following trainings :
"Administration of SOLIDWORKS PDM Professional" & "Using of SOLIDWORKS PDM"

ADMINISTRATION OF PDM

2. Administration Tool

- Administration tool
- Color of the vaults
- Additional fields

3. Users and Groups

- Windows connection
- User settings: Personalization

4. Folder Card Creation

- Serial numbers
- Centralized card lists
- Lists from an SQL database
- Lists controlled by a variable
- Data card entry formulas

7. Workflow

- Categories
- Creation of several workflows
- Unlimited number of states
- Types of transition
- Unlimited revision schemes

8. Notifications and Tasks

- Message system
- Conditional notifications
- Various conversion tasks
- Additional task options

9. Folder Templates

- Creating Folder Templates

10. File Templates

- Creating file templates

11. Vault Backup

- Maintenance plan in SQL Server Management Studio

Appendix

- Import/Export de données
- Survol : Réplication
- Survol : Web2
- Survol : Générateur de rapport
- Survol : Outils PDMxperts, programmations personnalisées, dispatch

USING OF PDM

2. SOLIDWORKS PDM User Interface

- Multi-document preview
- Files in private state

5. File References

- Move files and references
- Share files

6. Searching

- Dedicated search tool
- Favorite searches
- Search in the content (indexing)
- Search in the "labels"

9. Working in SOLIDWORKS

- Add-in
- Marking with eDrawings Professional

Appendix

- Use of named nomenclatures

Not included :

- Installation process
- Routing Setup and CircuitWorks Setup

Annex : ToolsXperts Demo



- PDMxperts
- ReportXperts

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 PDM PROFESSIONAL API FUNDAMENTALS – 2 DAYS (14H)

1. Connecting to a Vault

- COM Programming
- Application Types
- Namespaces
- The IEdm Vault Interface
- Debugger feedback
- Logging into a Vault
- Handling HRESULT Return Codes
- SOLIDWORKS PDM Professional API Help
- Interface Versioning

2. Files, Folders and References

- The IEdm Object Interface
- The IEdm File Interface
- The IEdm Folder Interface
- The IEdm Pos Interface
- File References
- The IEdm Batch-Listening Interface
- The IEdm Clear-Local Cache Interface

3. Users and Groups

- The IEdm User Interface
- The IEdm User Group Interface
- The IEdm UserMgr Interface

4. Card Variables, Versions and Revisions

- Card Variables
- File Versions
- File Revisions
- The IEdm Dictionary Interface

5. Add-In Applications

- SOLIDWORKS PDM Professional Add-Ins
- The IEdm AddIn Interface
- The Implements Statement
- Simple Implementation
- COM Registration
- Get AddIn Info
- Minimum Version Required
- Additional Add-In Information
- Installing an Add_in
- Debugging a DLL
- The IEDm Vault Argument
- The IEDm CmdMGR Argument
- The IEDm AddIn 5. OnCmd
- EdmCmdData Members for EdmCmd
- EdmCmdData Members for EdmCmd_Serial No

6. Task Add-In Applications

- SOLIDWORKS PDM Professional Task Add-Ins
- Task Interfaces
- Task Hooks
- The IEdm Search Interface
- The IEdm Workfl ow Mgr Interface
- The IEdm Workfl ow Interface

Annex : ToolsXperts Demo



- PDMXperts

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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PMT2333-ENG

SOLIDWORKS MANAGE – 2 DAYS (14H)

*The "Administration of SOLIDWORKS PDM Professional" training is recommended for this class.

1. System Options

- Planning for SOLIDWORKS Manage
- System Architecture
- Configuration File
- Essential Administrative Options
- Numbering Schem

2. Connecting to SOLIDWORKS PDM Professional

- Record Object Types
- Requirements for Connecting
- Creating a PDM Object

3. Document and Record Object Types

- Record Object
- Permissions
- Document Object
- File Templates

4. Fields and Field Groups

- Fields
- Organizing Fields
- Field Groups
- Numbering by Field Group
- Global Variables

5. Users and Groups

- Importing Users
- Groups

6. Bills of Materials

- Bills of Materials
- Creating Bills of Materials

7. Backup, Restore, and Upgrade

- Backing Up
- Restoring the Environment
- Upgrading SOLIDWORKS Manage

8. The Web Interface

- The Web Interface
- Web Interface and PDM Objects

Annex : ToolsXperts Demo



- PDMXperts

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PMT2235-ENG



AUTOMATING TRAINING

COURSE OUTLINE

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COURSE OUTLINE / AUTOMATING

DRIVEWORKS SOLO – 3 days (21h)

DRIVEWORKS ADMINISTRATOR – 4 days (28h)

SWOOD DESIGN CREATOR – 3 days (21h)

SWOOD DESIGN USER – 1 day (7h)

SWOOD CAM – 2 days (14h)

SWOOD DESIGN ADVANCED CREATOR – 1 day (7h)

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DRIVEWORKS SOLO – 3 DAYS (21H)

1. Lesson 1

- Basic Setup
- Capturing your Models

2. Lesson 2

- Project Designer

3. Lesson 3

- Building Rules

4. Lesson 4

- Improving your Project

5. Lesson 5

- Static Replacement Files

6. Lesson 6

- Tables

7. Lesson 7

- Form Navigation

8. Lesson 8

- Enhancing your Forms
- Dynamic Replacement Files

9. Lesson 9

- Driving Custom Properties

10. Lesson 10

- Documents

11. Lesson 11

- Drawings

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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DRIVEWORKS ADMINISTRATOR – 4 DAYS (28H)

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

1. Lesson 1

- Creating a Group and Capturing Models

2. Lesson 2

- Building a user interface in DriveWorks Administrator

3. Lesson 3

- Building Rules

4. Lesson 4

- Running your Project

5. Lesson 5

- File Name and Relative Path Rules

6. Lesson 6

- Tables

7. Lesson 7

- Form Navigation
- Form Templates
- Static and Dynamic Control Properties
- Advanced form controls

8. Lesson 8

- Dynamic Replacement Files

9. Lesson 9

- Data Management

10. Lesson 10

- Documents

11. Lesson 11

- Drawings

12. Lesson 12

- Specification Flow
- Preparing your Models for Automation

13. Lesson 13 (Advanced)

- Advanced Form Controls

14. Lesson 14 (Advanced)

- Specification Control

15. Lesson 15 (Advanced)

- Linking to Data

16. Lesson 16 (Advanced)

- Rollup Data Tables

17. Lesson 17 (Advanced)

- Hierarchical properties

18. Lesson 18 (Advanced)

- Macro Buttons

19. Lesson 19 (Advanced)

- Generation Tasks

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SWOOD DESIGN CREATOR – 3 DAYS (21H)

1. SWOOD Design Presentation

- Introduction to SWOOD Design
- Configuring
- Integration of SWOOD into SOLIDWORKS
- User Interface

2. SWOOD Panel Creation

- Creating a Panel
- Editing a Panel
- Curved Panels
- Other Methods of Creation

3. SWOOD Frame Creation

- Demonstration of a SWOOD FRAME
- Creating a SWOOD Frame with a Panel
- Editing a Frame
- Adding Extra Parameters
- Creating a New Frame from an Existing Frame
- Finalising and Saving Frames to Library

4. SWOODBox Creation

- Introduction to SWOODBoxes
- Intention and Principles when Creating a SWOODBox
- Presentation of SWOODBox Task Pane
- Demonstration of SWOODBox Insertion
- Creation and Saving a SWOODBox to Library
- SWOODBox Machining Definition
- Insertion of a SWOODBox
- Introduction to SWOODBox Scripts

5. SWOOD Connector Creation

- Accessing Connectors Library
- Creating a Simple Connector
- Creating a Compound Connector
- Introduction to Rule Creation in Scripts
- Inserting a Connector

6. SWOOD Profiles

- Creating a new Profile
- Applying created profile to Profile Library
- Applying a Profile to an Edge

7. Edge Bands

- Applying an Edge Band to a Panel
- Creating a Machining Profile with Edge Band
- Applying an Edge Band with a Machining Profile

8. Materials

- Creating a New Material
- Applying a Material (Panel, frame, click, and drag with or without driving thickness)
- Managing Materials
- Managing Materials through Panel Interface

9. Creating a Project with Multiple Frames

- Project Creation
- Copying a Frame
- Modifying Dimensions of Frames
- Creating Layout Sketches
- Inserting Frames onto Layout Sketch
- Creating Magnetic Insertion Points
- Creating a Layout with Magnetic Mates
- Modifying Layout Sketch
- Generate a Report

Annex : ToolsXperts Demo



- CutXperts
- EdgeXperts

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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SWOOD DESIGN USER – 1 DAY (7H)

*Prerequisites : Basic knowledge of SOLIDWORKS Modeling

1. SWOOD and SOLIDWORKS

- About SWOOD and SOLIDWORKS
- Implementing SWOOD into SOLIDWORKS
- SOLIDWORKS Settings for SWOOD

2. Panel Design

- Panel Definition in SWOOD
- Edit Pannel Command and Library Features
- Manage Material Library
- Understanding the Material Library
- Edgebands and Shapes Management
- Panel Editing Interface

3. Frames

- What is a SWOOD Frame?
- Useful Interfaces

4. Connectors

- What is a SWOOD Connector
- Command and Library Interfaces
- Connectors Library Editing Window

5. SWOODBox

- What is a SWOODBOX?
- Useful Interfaces

6. Integration

- Integration Command Interface

7. SWOOD Reports

- Presentation
- Report Interfaces

Annex : ToolsXperts Demo



- CutXperts
- EdgeXperts

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SWOOD CAM – 2 DAYS (14H)

*The "SWOOD Design - Essential" Training is required for this class.

1. Integrating SWOOD CAM into SOLIDWORKS

- Add-ins
- SWOOD Settings

2. SOLIDWORKS Settings for SWOOD CAM

- Required Configurations
- Managing Views
- Complex Assemblies
- Customizing Command Bar
- Dynamic Highlight
- Custom Property Files

3. Tool Creation

- Presentation of Tool Library
- Presentation of Aggregate Library
- Aggregate Properties
- Properties of Drill Bits
- Simple Tool Creation
- Modifying a Aggregate/Drill Block
- Blade Management

4. Program Settings and Automatic Operations

- Configuring each Phase of a Part File
- Origin
- Tool Insertion
- Creating a Machining Definition (Automatic contour)
- Creating an Automatic Drilling Definition (without selection)
- Creating an Automatic Grooving Definition (without selection)
- Creating an Automatic Pocket Operation
- Creating an Automatic Sawing Operation

5. Manual Operations

- Pocket Milling and Machine Pocket Milling
- Creating a Contour with Wall Selection
- Creating a Contour for Grooving/Rebating Operation
- Creating a Contouring Operation with a Chamfering Tool
- Demonstration of Tool Simulation
- Creating an Operation on a Sketch

6. 4 & 5 Axis Operations

- Surfacing, Contouring and Sawing
- Guide Line for Inclined Plane
- Inclined Pocket Milling Operations
- Interpolate C-Axis
- Chamfering
- Creating a 5-Axis follow-up Operation in OPO
- Creating a 3D Roughing Operation (Roughing & Finishing)

7. SWOOD Design Panel Integration with SWOOD CAM Operations

- Template Creation
- Creating a Frame with Machinings
- Creating a Partial Contouring Operation
- Positioning by Mates in Assembly Machining
- Positioning by Offsets in Assembly Machining
- Positioning by Repetition in Assembly Machining
- Transforming a Part into an Assembly

8. Link with SWOOD DESIGN

- Profile Machining
- Calibrating with and without Edge Bands
- Stock Following Edge Bands and Laminate

Annex : ToolsXperts Demo



- CutXperts
- EdgeXperts

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.

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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.

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

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SWOOD DESIGN ADVANCED CREATOR – 1 DAY (7H)

*The "SWOOD Design - Essential" Training is required for this class.



1. Introduction to Script Programming

- Organization of scripts
- Introduction to script programming
- Different levels of script application

2. Advanced SWOODBox

- Advanced SwoodBox presentation
- Creation of the parameters of a SwoodBox
- Creation of the rules of a SwoodBox
- Automate a SwoodBox with a script

3. Using SWOODCenter

- Library opening
- Simple element creation
- Compound element creation
- Introduction of rules with script
- Insertion of links

4. SWOOD Report

- Data export

Annex : ToolsXperts Demo

- CutXperts
- EdgeXperts

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 ELECTRICAL TRAINING

COURSE OUTLINE

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SOLIDWORKS ROUTING: ELECTRICAL – 1 day (7h)

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SOLIDWORKS ELECTRICAL: SCHEMATIC – 3 DAYS (21H)

1. Project Templates

- SOLIDWORKS Electrical
- Starting SOLIDWORKS Electrical
- What are projects?
- Project templates
- Project configurations
- How is a project structured?

2. Modifying Project Templates

- What are environments?
- Draw multiples wires

3. Drawing Types

- What are drawing types?
- Existing and archived projects
- Line diagram symbols
- Adding cables
- Symbols panel
- Schematic symbols
- Symbol properties

4. Symbols and Components

- What is a component?
- Description columns
- Symbol component association

5. Manufacturer's parts

- What are manufacturer's parts?
- Circuits and terminals
- Finding manufacturer parts
- Electrical assemblies

6. Wires and Equipotentials

- Equipotentials and wires
- Wire style manager
- Replacing wires
- Equipotential numbering results
- Wire numbering results
- Using nodal indicators

7. Cabling

- What is cabling?
- Cables
- Detailed cabling
- Terminal strip
- Pin to pin connections
- Copy and paste

8. Symbol Creation

- Symbols and standards
- Symbols manager
- Symbol properties
- Circuits, terminals, types
- Multiple attributes
- Splitting attribute data
- Add to library
- Copy, paste symbol

9. Macros

- What are macros?
- Creating and adding macros

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.

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.

PMT2513-ENG

SOLIDWORKS ELECTRICAL: SCHEMATIC (PART 2)

10. Cross Referencing

- What is Cross Referencing?
- Cross Referencing List
- Cross Referencing State colors
- Cross Referencing Text coding
- Cross Referencing Types
- Cross Referencing Location List

11. Managing Origin-Destination Arrows

- What are Origin-Destination Arrows?
- Origin-Destination Arrows

12. Dynamic Programmable Logic Control

- What is a PLC?
- Adding a New Scheme
- Adding a PLC Mark
- Inserting a PLC
- Editing a PLC

13. Automated Programmable Logic Control

- How are PLCs Automated?
- PLC mark part
- IO manager

14. Connectors

- Connectors
- Insert Connector
- Connector Insertion

15. 2D Cabinet Layouts

- What are Cabinet Layouts?
- Creating a 2D Layout
- Inserting Ducts and Rails
- Inserting Components
- Wire Cabling Order

16. Design Rule Checks

- What are Design Rule Checks?
- Unconnected Pins
- Equipotential Conflicts
- Max. Terminal Wires
- Duplicated Parent Symbols
- Child Symbols without Parent
- Empty Terminal Strip
- Duplicated Terminals

17. Reports

- What are Reports?
- Report Manager
- Report Columns
- Column Formula
- SQL Query Column Variable
- Sort and Break

18. Simple Reports

- What are Views?

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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PMT2513-ENG

SOLIDWORKS ELECTRICAL: 3D – 1 DAY (7H)

1. Assembly Creation

- What are Assemblies?
- Unarchiving a Project
- SOLIDWORKS Assembly

2. Cabinets, Ducts, & Rails

- Cabinets, Ducts, & Rails
- Inserting Components
- Inserting Rails
- Inserting Ducts

3. Component Intelligence

- What is a component?
- Component Intelligence
- The Electrical Component Wizard

4. Insert Components

- Insert Components
- Align Components
- Inserting Terminals

5. Routing Wires

- Routing Wires
- Routing Path
- Route Wires

6. Routing Cables

- Routing Cables
- Route Cables
- Set Origin Destination of Cables by Location

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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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.

PMT2514-ENG

SOLIDWORKS ROUTING : ELECTRICAL – 1 DAY (7H)

1. Fundamentals of Routing

- What is Routing?
- Routing Setup
- General Routing Settings

2. Basic Electrical Routing

- Basic Electrical Routing
- Adding Routing Components
- Start by Drag and Drop Connector
- Auto Route

3. Routing with Clips

- Routing with Clips
- Routing Through Existing Clips
- Adding Clips while Auto Routing
- Editing a Route
- Working with Clips
- Routing Through a Clip
- Splitting a Route
- Adding a Splice
- Multiple Routes Through a Clip

4. Electrical Routing Components

- Routing Library Parts Introduction
- Electrical Routing Library Parts
- Libraries
- Routing Component Wizard
- Routing Component Attributes
- Electrical Libraries

5. Standard Cables and Reusing Routes

- Using Standard Cables
- Standard Cable Excel File
- Modifying Standard Cables
- Creating a standard Cable
- Reuse route
- Delink route
- Routing Templates

6. Electrical Data Import

- Importing Data
- Routing Library Manager
- From/To Lists
- Route Properties
- Route Guidelines
- Using Guidelines and Clips

7. Electrical Drawings

- Route Flattening and Detailing
- Annotation Flattening
- Flatten Route
- Manufacture Flattening

8. Flex Cables

- Flex Cables
- Flex Cable Routes
- Flex Cable Auto Routing
- Using Flex Cables with Clips

9. Electrical Conduits

- Electrical Conduits
- Rigid Conduit
- Orthogonal Routing with Auto Route
- Electrical Data in Conduits
- Manual Sketch Routing
- Flexible Electrical Conduit

Appendix A: Review Section

- Review of Configurations
- A Note About File References
- Design Tables
- Review of Top Down Design
- Editing Options
- Review of Design Library Task Pane
- Review of 3D Sketching

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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PMT2411-ENG



SOLIDWORKS ADD-INS TRAINING

COURSE OUTLINE

TABLE OF CONTENTS

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SOLIDWORKS COMPOSER – 3 DAYS (21H)

1. Getting Started

- What is the SOLIDWORKS Composer Application?
- Notes on SOLIDWORKS Composer
- SOLIDWORKS Composer terminology
- SOLIDWORKS Composer user interface
- View
- Navigation Tools
- Updating Views
- Collaborative Actors
- Camera Views
- Transform
- Creating 2D Output
- View Mode/Animation Mode

2. Creating Cover and Detail Images

- Rendering tools
- Camera and Alignment tools
- Custom Rendering
- Digger

3. Creating an Exploded View

- Visibility tools
- Exploded Views
- Collaborative Actors
- Styles
- Vector Graphics Output

4. Creating Additional Exploded Views

- Importing Files
- Paper Space
- Update views With Selected Actors
- Align Actors
- Explode Lines
- Custom Views
- Linking Between Views

5. Creating Bills of Materials

- Bills of Materials
- Vector Graphics Output
- Another BOM Table
- Assembly Level BOM
- Assembly Selection Mode

6. Creating a Marketing Image

- Selections
- Textures
- Lighting
- Scenes
- High Resolution Image

7. Creating Animations

- Timeline Pane
- Location Keys

8. Creating Interactive Content

- Views for Animation
- Improving the Animation
- Digger Keys
- Selections in the Key Track
- Events
- Animating and Collaborative Actors

9. Creating a Walkthrough Animation

- Camera Keys
- Grids
- Additional Camera Functionality

Part 2 →

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PMT2590-ENG

SOLIDWORKS COMPOSER (PART 2)

10. Adding Special Effects to Animations

- Animation Library Workshop
- Animation Special Effects
- Assembly Selection Mode in Animations
- Scenarios

11. Updating SOLIDWORKS Composer Files

- Update an Entire Assembly
- Changing the Geometry of an Actor

12. Working with Projects

- What are Projects?
- Product Files
- Product Orientation

13. Publishing from SOLIDWORKS Composer

- Preparing a File for Publishing
- Publishing to PDF
- Publishing Inside Microsoft PowerPoint
- Publishing to HTML
- Linked SVG Files
- Publishing Multiple Views

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PMT2590-ENG

SOLIDWORKS VISUALIZE – 2 DAYS (14H)

1. CAD to SOLIDWORKS Visualize

- Rendering from CAD
- Importing to Visualize
- Render Selection
- Denoiser
- Appearances
- File Libraries
- Scenes
- Color Picker
- Rendering

2. Import Settings and Appearances

- Import Settings
- Appearances
- Project Description
- Part Grouping
- Structure and Organization
- Selection Tools
- Object Manipulation
- Split
- Copy and Paste
- Appearance Types
- Textures
- Texture Mapping
- Appearance Parameters
- Merge Parts

3. Decals

- Decals
- Decal Feature
- Blend Texture
- Multi-Layer Decal Process
- Decal Depth
- Decal Mapping

4. Cameras

- Cameras
- Aspect Ratio
- Keep Above Floor
- Perspective
- Camera Orientation
- Grid Overlay
- Depth of Field
- Filters

5. Backplates, Environments and Lights

- Scenes
- Backplates
- Environments
- New Cutting Plane
- Lights

6. Productivity tools

- Multiple Views
- Render All Cameras
- Time Limit Rendering
- Output Viewer
- Configurations
- Exports
- Render All Configurations
- Render Queue
- Patterns
- Visualize Boost

Part 2 →

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PMT2455-ENG

SOLIDWORKS VISUALIZE (PART 2)

7. Animation and Grouping

- Animations
- Groups
- Animation Output
- Motion Blur
- Keyframe Animation

8. Camera Animations

- Camera Animations
- Camera Movement with the triad
- Keyframe Properties

9. Animating Appearances, Scenes

- Appearance and Scene Animations
- Scene Animations

10. Alternative outputs

- Alternative Outputs
- Turntable
- Interactive View
- Panoramic View
- Sunlight Animation
- 360° Camera

11. Simulations

- Simulated physics
- Shake simulation
- Simulation manager
- Simulation states
- Vehicle simulations
- Physics Animations

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PMT2455-ENG

SOLIDWORKS INSPECTION – 1 DAY (7H)

*This training's duration is about half a day but we will complete it with personalized coaching on your models or examples files.

1. Inspection Add-in

- What is SOLIDWORKS Inspection?
- Inspection project
- SOLIDWORKS Inspection manager
- Export inspection data
- Design revision
- Manual Ballooning
- Working with 3D documents

2. Standalone Application

- User interface
- Inspection project
- Manual Extract
- General characteristic tools
- Table manager
- Managing of characteristics
- Grids
- Multiple Documents
- Publishing reports
- Drawing revisions

3. SOLIDWORKS Inspection Professional

- Loading the inspection professional Add-in
- Measurements Input
- Publishing reports with inspection results
- CMM data import

4. Appendix A: Inspection Report Templates

- Inspection Report Templates
- Template Editor

5. Appendix B: Understanding Regular Expressions

- Regular Expressions

6. Appendix C: Glossary of Quality Terms

- Terminology

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PMT2526-ENG

SOLIDWORKS MBD – 1 DAY (7H)

1. Introduction to SOLIDWORKS MBD

- What is SOLIDWORKS MBD?
- MBD Using Feature Dimensions
- Sharing 3D Views
- 3D PDF Capabilities
- MBD Using DimXpert
- DimXperts Capabilities
- eDrawings and MBD
- eDrawings Capabilities
- STEP 242 Files
- MBD and Assemblies

2. Using Feature Dimension and Annotation Views

- Using Feature Dimensions with MBD
- Default Annotations Views
- Optimizing Settings
- Adding and Organizing Annotations
- Adding Reference Dimensions
- Modifying Dimensions
- Creating a Section Annotation View
- Editing an Annotation View
- Unassigned Items
- Creating an Annotation View
- Note Area
- Using Tables

3. Capturing 3D Views

- 3D Views
- 3D Views Tab
- Capture 3D View
- Activating and Modifying 3D Views
- Using 3D Views Options
- Publishing PMI
- Special 3D View Types
- Model Break View

4. 3D PDF Template Editor

- 3D PDF Template Editor
- Areas of the Template
- Text Types
- Other Template Aspects
- Building a Custom Template
- Saving and Storing Custom Templates
- Testing the Templates

5. Using DimXpert

- What is DimXpert?
- DimXpert Settings
- DimXpert Block Settings
- DimXpert Dimension Settings
- How DimXpert works
- Auto Dimension Scheme
- DimXpertManager
- Show Tolerance Status
- Modifying DimXpert Annotations
- Combining Dimensions
- Creating Multiple Schemes
- Manual DimXpert Annotations
- Feature Selector Toolbar
- Using DimXpert Dimension Tools
- Unique DimXpert Options

6. MBD and Assembly Models

- Assembly Models and MBD
- Assembly Level Dimensions
- Optimizing Settings in Assemblies
- Adding Assembly Annotations
- BOM Tables and Balloons
- Publishing Assembly PMI
- Additional MBD Tools

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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PMT2525-ENG

SOLIDWORKS CAM STANDARD – 2 DAYS (14H)

1. SOLIDWORKS CAM Basics and User Interface

- What is SOLIDWORKS CAM?
- SOLIDWORKS CAM User Interface
- Overview
- SOLIDWORKS CAM Feature Tree
- Operation Plans
- Toolpaths

2. Automatic Feature Recognition (AFR) and Operation Modification

- Working with Features, Operations, and Toolpaths
- Automatic Feature Recognition
- Feature Strategy
- Modifying Operations
- Modifying Parameters
- Design Changes

3. Interactive Feature Recognition (IFR)

- Interactive Feature Creation
- 2.5 Axis Features
- Part Perimeter Feature
- Mill Part Setup
- Work Coordinate Offsets
- Selection Filters

4. Interactive Operations

- Interactive 2.5 Axis Mill Operations
- Create operations
- Save Operation Plan

5. Merging Features and Operations

- Machining Similar Features
- Create Group
- Combine Operations
- Link Operations

6. Avoid and Contain Areas

- Adding “Avoid” and “Contain” Areas

7. Pattern Features and Mirror Toolpaths

- Patterning
- Mirror Toolpaths

8. Advanced Features and Operations

- Advanced Feature Creation
- Engrave Feature
- Curve Feature
- Multi-stepped Hole
- Tap and Thread Mill Hole Operation
- Corner Round and Chamfer Machining
- Multi Surface Feature

9. Customizing the Technology Database

- SOLIDWORKS CAM Technology Database (TechDB)
- User Defined Tool Creation
- Mill Machine
- Mill Tool
- Tool Crib
- Strategies

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.

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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.

PMT2565-ENG

SOLIDWORKS CAM PROFESSIONAL – 1.5 DAY (10H)

*The "SOLIDWORKS CAM Standard" training is required for this class.

**Since our clients generally use only 50% of the training's content, this course is shortened to 1 day and is adjusted with personalized coaching based on your manufacturing process.

***If you want to get the whole course content, we can offer this class on a 2 days schedule. Please contact your account manager.

1. SOLIDWORKS CAM Configurations

- SOLIDWORKS CAM Product Review
- SOLIDWORKS CAM Configurations
- Working With CAM Configurations

2. High Speed Machining (VoluMill™)

- VoluMill Overview
- VoluMill Settings
- VoluMill Technology Expert

3. Assembly Machining

- SOLIDWORKS CAM Assembly Mode
- Part Manager
- Assembly Machining - Programming with Subroutines
- Machining - Multiple Parts
- Machining Split Instance
- Split Setup

4. 3 Plus 2 Machining

- 3 Plus 2 machining (Indexing)
- Assembly machining with a tombstone

5. Turning Basics

- SOLIDWORKS CAM Turning
- Setup
- Chuck/Fixture
- Stock
- Machinable Features
- New Turn Feature

6. Chucks, ID Features and Operations

- Section Method
- Using plane section
- Double Chucking
- Modifying Feature and Operation Parameters
- Editing toolpaths

7. Modifying Feature and Operation Parameters

- Custom chuck, OD and thread features
- Editing toolpaths

8. Probing

- Introduction to Probing

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.

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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.

PMT2566-ENG

SOLIDSTEEL PARAMETRICS ADVANCED ENGINEERING PACKAGE / 2,5 DAYS (18H)

Prerequisites: Students should have a general knowledge of SOLIDWORKS.
Students should have experience creating 3D sketches, parts and assemblies.

1. Introduction

- Introduction to the use of the software
- User-specific setting

2. Profiles & Connections

- Profiles and profile cuts
- Creation of individual profile cross-section and sizes
- Typical steel connections
- Bolted connections

3. Assembly Manager

- Introduction to the Assembly Manager
- Working with the Assembly Manager

4. Stairs & Railings

- Working with stairs and railings
- Creation of individual railing templates

5. Administration & Management

- Standard parts management
- Administration of the bolt manager
- Teaching in bolt sets

6. BOM & Drawings

- Changing the drawing templates
- Creation of BOM templates
- Detection of identical parts
- DXF export for all sheet metal parts

7. Advanced Tools

- License server administration

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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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.

SOLIDSTEEL PARAMETRICS DESIGN PACKAGE / 3 DAYS (21H)

Prerequisites: Students should have a general knowledge of SOLIDWORKS.
Students should have experience creating 3D sketches, parts and assemblies.

1. Introduction

- Introduction to the use of the software
- User-specific setting

2. Profiles & Connections

- Profiles and profile cuts
- Creation of individual profile cross-section and sizes
- Typical steel connections
- Bolted connections

3. Assembly Manager

- Introduction to the Assembly Manager
- Working with the Assembly Manager

4. Stairs & Railings

- Working with stairs and railings
- Creation of individual railing templates

5. Administration & Management

- Standard parts management
- Administration of the bolt manager
- Teaching in bolt sets

6. BOM & Drawings

- Changing the drawing templates
- Creation of BOM templates
- Detection of identical parts
- DXF export for all sheet metal parts

7. Industrial Facades & Roof

- Workflow overview

8. Advanced Tools

- SDNF interface
- Bidirectional data exchange with 3D structural analysis software
- License server administration

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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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.

DSTV ASSISTANT / 0,5 DAY (4H)

Prerequisites: Students should have a general knowledge of SOLIDWORKS.
Students should have experience creating 3D sketches, parts and assemblies.

1. Introduction

- Activation of the SOLIDWORKS add-in
- Overview of the interface

2. Database

- Files locations / Path settings
- Naming schematics
- Database settings (Part 1)

3. Creation of NC Data

- Preparation of parts & assemblies for the Export
- Export NC files

4. Analyzed Parts

- SolidSteel parametric profiles
- Identified/Unidentified solid bodies
- Change / Provide information
- Quickstart & Quick export for assembly
- Export of analyzed parts
- Signatures, markings & powder lines

5. Costing Module

- Configuring database
- Creating / Changing a machine
- Creating setup rules & new tools
- Creating a costing analysis

6. General Settings

- Dimension references
- Contour settings
- Filter settings
- Export template settings

7. Administration

- Database settings (part 2)
- Database management

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INTRODUCTION TO VISUAL BASIC.NET – 2 DAYS (14H)

1. Introduction

2. The Basic Interface

3. Code

- Types of variables
- Definition of variables
- The message box

4. Compilation

5. VB Tools

- Benchmarks
- Breakpoints
- Keyboard Shortcuts

6. Loops

- For...Next
- Do...Loop
- Force output

7. If Statements

8. The Select Case Statement

9. The Interface of Your Programs

- Interface Language
- Activation checks
- Other interface parameters

10. Posts

- Simple message
- Message to capture the response

11. Goto

12. Conventions

- Variable names
- Method names
- Argument names

13. Variables List

- The array
- The collection

14. The Timer Control

15. Operators

16. The Text File

- Reading a text file
- Writing to a text file

17. File Manager

- File operations
- Operations on files

18. Creating a Method

- Sub-type method
- The method of type function

19. Manipulate the Registry

- Management options in the registry

20. Windows

21. Operations on Text

- Text comparison
- Conversion values
- Extracting a portion of text
- Extracting data from a file path
- Splitting text
- Replacement values

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SOLIDWORKS API FUNDAMENTALS – 2 DAYS (14H)

*The "Introduction to Visual Basic.net" training is required for this class.

1. Using the Macro Recorder

- Macro Recording
- Macro Toolbar
- Understanding How Macro Code Works
- Understanding How to Call Members on API interfaces
- Passing Parameters
- Cleaning Up Code
- Adding Forms to a Macro

2. The API Object Model

- SOLIDWORKS API Object Model
- Application Objects
- Connecting to New Documents
- Connecting to Existing Documents

3. Setting System Options and Document Properties

- User Preferences – System Options
- User Preferences – Document Properties
- Locating the Correct APIs and Enumeration Values
- User Preferences Tables for System Option, Document Properties and Menu Items

4. Automating Part Design

- Automation Tool for Parts

5. Assembly Automation

- Automation Tool for Assemblies

6. Drawing Automation

- Automating Drawing Creation

7. Selection and Traversal Techniques

- Programming with a Selected Object
- The SOLIDWORKS BREP Model
- Body and Face Traversal
- Feature Manager Traversal

8. Adding Custom Properties and Attributes

- Custom Properties
- Configurations with Custom Properties
- File Summary Information
- Document Attributes
- The Attribute Objects
- Face Attributes

9. The SOLIDWORKS API SDK

- The API SDK
- Creating a VB.NET Add-In
- Creating a C# Add-In
- C++ Add-Ins
- Choosing a Programming Language

10. Customizing the SOLIDWORKS User Interface

- Customizing the UI With VB.NET
- Understanding The Add-in Code
- Property Pages
- Property Page Groups and Controls
- Removing Menus and Toolbars
- Other Areas of Customization

11. Notifications

- Notifications
- Notifications in VBA
- Simple Notification
- Using Notifications in .NET

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PMT2515-ENG

DRAFTSIGHT ESSENTIALS / 1,5 DAYS (10H)

*This course is aimed at new users of 2D CAD drawing software.

*Not suggested for experienced users of software such as AutoCAD, TurboCAD, etc.

1. User Interface

- What is DraftSight?
- The User Interface
- Stages in the Process
- Pointing Device
- Title Bar
- Toolbars
- Dock and Undock
- Tool palettes - Customizing

2. View Commands

- Viewing Drawings in DraftSight
- Stages in the Process
- Dynamic Pan
- Dynamic Zoom
- Zoom Window
- Zoom Fit
- Rebuild
- Display Quality
- Zoom Previous
- Named Views
- View Tiles
- Viewports

3. Coordinates

- Coordinated System
- Cartesian Coordinates
- Inquiries
- Units
- Base Angle
- Coordinate Entry
- Quick Input
- Entity Snap
- Drawing Boundary

4. Creating a Simple Drawing

- DraftSight Drawing Basics
- New Drawings
- Lines
- Delete
- Selection Methods
- Rectangle
- Circle
- Undo / Redo
- Arc
- Polygon

5. Modifying Entities

- Modifications
- Move
- Rotate
- Mirror
- Stretch
- Copy
- Scale
- Quick Modify
- Power Trim

6. Properties and Layers

- Layers
- Layer States
- Locked Layers
- Layer Manager
- Entity Properties
- Property Painter
- Merge Layers

7. Dimensions

- Dimensions
- Dimension Styles
- Smart Dimension
- Linear Dimension
- Angular Dimension
- Diameter Dimension
- Dimension Palette
- Continue Dimensions
- Split Dimension

Part 2 →

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PMT2480-ENG

DRAFTSIGHT ESSENTIALS (PART 2)

8. Text

- Text
- Text Styles
- Simple Notes
- Notes
- Note Formatting
- Smart Leaders
- Curved Text
- Find and Replace
- Spell Check

9. Blocks

- Blocks
- Stages in the Process
- Make Blocks
- Paste as Block
- Insert Block
- Edit Component
- Assign Attributes
- Update Block Attributes
- Block Attributes Display Mode
- Edit Attributes

10. Custom Blocks

- What are Custom Blocks?
- Edit Blocks
- Mirror Elements
- Mirror Activity
- Visibility States

11. Entity Groups

- Creating Entity Groups
- Edit Entity Groups

12. Printing

- Printing in DraftSight
- Print Previews
- Print Configurations
- Quick Print
- Manual setup
- Additional Options
- Style Table

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PMT2480-ENG

POWER SURFACING / 1 DAY (7H)

1. Basic Operations

- Basic Manipulations
- Extrude
- Insert Loops

2. Operations (Continued)

- Hard Line
- Define Boundary

3. SOLIDWORKS Body References

- Import References
- Constrain to...
- Auto Update All Constrains

4. Spatial Constrains

- Retain Offset Constrain
- Retain Ratio Constrain

5. Advanced Operations

- Advanced Manipulations
- Thicken
- Symmetry

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3D PRINTING TRAINING

COURSE OUTLINE

TABLE OF CONTENTS

TABLE OF CONTENTS / 3D PRINTING

MARKFORGED (INSTALLATION) – 1 day

DESIGN FOR ADDITIVE MANUFACTURING (DFAM) – 1 day

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MARKFORGED (INSTALLATION) – 1 DAY

1. Preparation

- Unboxing
- Verifying Boxes Content
- Printer Installation
- Plugging the Printer on the Network

2. Introduction

- SolidXperts Introduction
- Good Use of 3D Printing
- Comparison Against ABS
- Useful Web Pages
- Eiger Account Creation
- Introduction to STL files

3. Maintenance and Calibration

- 3D Printer Components
- The Use of USB Key
- Leveling Technic
- Fiber Nozzle Adjustment
- Test Prints
- Nozzle Cleaning
- Plastic and Fiber Nozzle Replacement
- Wet Plastic Purge
- XY Adjustment
- Strap Tensioning

4. Informations

- Part Glue
- Print Information
- Mechanical Properties

5. Advanced Operations

- Menu Options
- Fiber/Sandwich Technic
- Type of Fiber Filling
- Part View and Internal View
- Visibility Options
- Completely Filling a Part of Fiber
- Completely Filling a Part of Plastic
- Helping the Fiber Pathing by Changing the Geometry
- Helping the Fiber Pathing by Changing the orientation of the Part
- Brim
- Opening a Request to MarkForged
- Saving the Logs

6. Questions

- Questions
- Starting a Print with a Custom Part

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DESIGN FOR ADDITIVE MANUFACTURING (DFAM) – 1 DAY

1. What is Additive Manufacturing?

- Brief history of additive manufacturing
- Examples of uses

2. Basic Principle of Technology

- Mechanical operation
- Special features of the FFF process
- Strengths and weaknesses of the technology

3. Presentation of Printing Materials

- ABS and PLA
- Onyx
- Continuous fiber

4. Overview of Printing Software

- Creation of an STL file
- Example of printing software

5. Producing Efficiently

- Choose the right orientation
- Limit the use of support material
- Limit weaknesses (sense of impression)
- Limit printing time
- Support behavior

6. Questions to Ask Yourself Before Producing a Part

- Purpose of manufacture
- Usage environment
- Duration of use
- Number of parts to manufacture
- Technologie available

7. Adaptation of the design according to the type of manufacture and use

- Machining mode of thinking vs. Additive manufacturing

8. Design Optimization for FFF Additive Manufacturing

- Precision and tolerances
- Wall thicknesses
- Minimum dimensions
- Reduce stress
- Chamfer vs. Rounding
- Limit fragility
- Surface quality
- Cost and manufacturing time

9. Tips for Greater Durability

- Wear parts and technology integration
- Use of purchased parts
- Thread
- Pause while printing

10. Scenarios

- Prototyping
- Tools

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