SOLIDWORKS
3D CAD
Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

SOLIDXPERTS by solidxperience

Course Outlines for SOLIDWORKS

Here are the courses outilnes of all the training offered at SolidXperts. Also, here is the information you often need to make your request to the competent authorities.

- Training organization: SolidXperts Inc.
- Address: 2650 Marie-Curie, Saint-Laurent, Quebec, H4S 2C3, 1.877.876.5439
- Quebec enterprise number (NEQ): 1160447596
- Certificate of approval number (Quebec training body): 0054182

If you have any questions, do not hesitate to contact us, and thank you for trusting us to train your teams.

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SOLIDWORKS Essentials – 4 Days (28h) (2 sessions)
(1st session)

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
- Filleting
- Editing tools
- Detailing basics
- Drawing views
- Center marks
- Dimensioning
- Changing parameters

4. Symmetry and Draft

- Design intent
- 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 pattern
- Circular patterns
- Reference geometry
- Planes
- Mirror patterns
- Using pattern seed only
- Up to Reference
- Sketch driven patterns

6. Revolved Features

- Design intent
- Revolved features
- Building the Rim
- Building the Spoke
- Edit material
- Mass properties
- Files properties
- SOLIDWORKS SimulationXpress

➔ Continued…

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SOLIDWORKS Essentials (2nd session) (continued…)

7. Shelling and Ribs

- Shelling and ribs
- Analysing and adding draft
- Other options for draft
- 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. Configuration

- 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 variable and equations
- Renaming features and dimensions
- Design rules using global variables and equations
- Global variable
- Equations
- Using operators and functions

12. Using Drawings

- More about making drawings
- Removed section
- Detail Views
- Drawing sheets and sheet formats
- Model views
- Section view
- Annotations

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
- Sub-assemblies
- Smart Mates
- Inserting sub-assemblies
- 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 drawing

Appendix

- Options settings
- Document templates

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SOLIDWORKS Drawings ANSI – 3 Days (14h)

1. Review of the basics

- Review of the 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 view

2. Understanding Drawing Templates

- Structure of a Drawing Document
- Drawing Document
- Drawing Sheet
- Sheet Format
- Understanding Drawing Templates
- Drawing Template Design Strategy
- Design a Drawing Template
- Creating a Sample Model and Drawing

3. Customizing the Sheet Format

- Title Block Sketch
- 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 the Sheet Format Properties
- Understanding the Sheet Format Behavior
- Saving the Sheet Format
- Testing

5. Creating Additional Sheet Formats and Templates

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

6. Advanced Options for Drawing Views

- Advanced Drawing Views
- Showing Hidden Edges
- Broken-out Section View
- Auxiliary View
- 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

7. Understanding Annotation Views

- Understanding Annotation Behavior
- What are Annotation Views?
- Annotation Folder
- Default Annotation Views
- Annotation View Visibility
- Insert Annotation View
- Editing Annotation Views
- Annotation Update

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SOLIDWORKS Drawings ANSI (continued…)

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
- Dimension Styles
- Annotations in the Design Library
- Modifying a Table
- Flag Note Bank

10. Advanced Options for BOM Tables

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

11. Additional SOLIDWORKS Tables

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

12. Additional Drawing Tools

- Reusing Drawings
- Draw Compare
- SOLIDWORKS Design Checker
- SOLIDWORKS Task Scheduler

13. Managing Performance

- Performance Evaluation
- Detailing Practices
- System Options & Documents Properties
- Open Options
- Detached Drawings
- Hardware and Performance
- Additional Considerations

Important note regarding the exercises

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

  During the exercises that follow the different lessons of the training, you will build your drawing templates, personalize your sheet format, create your title blocks with your properties, produce your favorites of annotations, dimensions and finally produce your tables templates. (BOM, revision, etc.)

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SOLIDWORKS Sheet Metal – 2 Days (14h)

1. Basic Flange Features

- What are sheet metal parts?
- Sheet metal methods
- Unique sheet metal items
- Flange method
- Base flange/tab
- Sheet metal parameters
- 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 features
- Tab features
- Cuts in sheet metal

2. Working with the Flat Pattern

- Working with the flat pattern
- Flat pattern settings
- Feature for Manufacture
- Corner-trim feature
- Corners in the formed state
- Closed corner
- Corner relief
- Break corner / Corner trim
- Producing the flat pattern
- Sheet metal cut list properties
- Flat pattern drawing views
- Flat pattern view properties
- Drawing documents properties
- Sheet metal tables
- Cut list properties as a note
- Exporting the flat pattern

3. Additional Sheet Metal Techniques

- Additional sheet metal methods
- Designing from the flat
- Sketch bend feature
- Jog feature
- Adding features in an unfold state
- Unfold and fold
- Swept flange
- Swept flange flat pattern options
- Lofted bends
- Lofted bends in the design library

4. Converting to Sheet Metal

- Sheet metal conversion
- Insert bend method
- Imported Geometry to Sheet Metal
- Adding rips
- Insert bends
- Making changes
- Welded corner
- Converting cones and cylinders
- Convert to sheet metal

5. Multibody Sheet Metal Parts

- Multibody sheet metal parts
- Multi bodies 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 multi bodies
- Hiding and showing bodies
- Using split with sheet metal parts
- Patterning for multi bodies
- Using edge flange to merge bodies
- Interfering bodies
- Combining sheet metal with other bodies.

➔ Continued…

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SOLIDWORKS Sheet Metal (continued…)

6. Forming Tools and Gussets

- Sheet metal forming tools
- Standard forming tools
- Form tools features in the flat
- Part document properties
- Custom form tools
- Split line
- Forming tool
- Form tool in drawing
- Sheet metal gusset

7. Additional Sheet Metal Functions

- Additional sheet metal functions
- Cross-breaks
- Vent features
- Mirror part
- Tab and Slot
- Process plans
- Sheet Metal Costing

Appendix A : Sheet Metal Tables

- Tables
- Customizing Tables
- K-Factor

9. CutXperts (Tool from SolidXperts)

- Overview of CutXperts
- Automatic export in DXF
- Layers option
- Options

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SOLIDWORKS Assembly Modeling – 2 Days (14h)

1. Advanced Mate Techniques

- SOLIDWORKS Assemblies
- Assembly file structure
- File references
- Solving Mates
- Advanced mates Techniques
- Mate references
- Design library parts
- Capture mate references
- Multiple selection mate references
- Multiple mate mode
- Driven mates
- Using misaligned mates
- Copying multiple components
- Using Copy with mates
- Copy with mate options
- Fixed components
- Advance mate features
- Profile center mate
- Rack and pinion mate

2. Top-Down Assembly Modeling

- Top-down assembly modeling
- Stage in the process
- 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 & Smart Fasteners, and Smart Components

- Assembly features and smart fasteners
- Assembly features
- Hole series
- Smart fasteners
- 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

5. Using Configurations with Assemblies

- Using configurations with assemblies
- Creating configurations manually
- Configuration properties
- Using the modify configuration dialog
- Changing configurations using the context toolbar
- Managing tree display
- Assembly evaluation tools
- Controlling dimensions in an assembly
- Creating an equality
- Equations with functions
- Comments
- Sensors
- Using the mate controller

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SOLIDWORKS Assembly Modeling (continued…)

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 mode
- Using SpeedPak
- Using Simplified Configurations
- Defeature
- Modifying the structure of an assembly
- Envelope Publisher
- Large Design Review
- Comparison of Modes and Methods
- Tips for faster assemblies
- Drawing considerations

8. Facility Layout

- Facility layout
- Publishing an asset
- Using magnetic mates
- Modeling connection point geometry

9. Using SOLIDWORKS Treehouse

- SOLIDWORKS Treehouse
- Setting treehouse instances
- Exporting Treehouse data

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SOLIDWORKS Weldments – 1 Day (7h)

1. Weldments Features

- Weldments
- Structural members
- Groups vs. Structural members
- Adding plates and holes
- Gussets and end caps
- Using symmetry
- Advantage of a multibody part
- Limitation 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 of configured profiles
- Inserting existing parts
- When to use an assembly

3. Configuring and Detailing Weldments

- Weldment configuration
- Post-assembly machining features
- Weldments Drawings
- Drawing view of individual bodies
- Representing welds

4. Working with Bent Structural Members

- Working with bent structural members
- 3D Sketching

5. Introduction to Structure System

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

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SOLIDWORKS Advanced Part Modeling – 2 Days (14h)

1. Multibody Solids: How they Work

- 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

2. Saving Solid Bodies

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

3. Sketching with Splines

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

4. Introduction to Sweeping

- Sweeping
- Sweep with Guide Curves
- The SelectionManager

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

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SOLIDWORKS Advanced Part Modeling (continued…)

7. Advanced Sweeping

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

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
- Deform Feature
- Direct Editing

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SOLIDWORKS Surface Modeling – 2 Days (14h)

Training SOLIDWORKS Advanced Part Modeling* **is a prerequisite for this class.

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 away
- Why use surfaces
- Continuity explained
- Workflow with surfaces

2. Introduction to Surfacing

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

3. Solid-Surface Hybrid Modeling

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

4. Repairing and Editing Imported Geometry

- Importing data
- File translation
- Why do imports fail?
- 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 surface
- Corner blends

6. Complex Blends

- Complex blends
- Freeform Feature

7. Advanced Surface Modeling

- Stages in the process
- 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

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SOLIDWORKS Mold Design – 2 Days (14h)

Training - SOLIDWORKS Advanced Part Modeling* **is a prerequisite for this class.
***Training - SOLIDWORKS Surface Modeling* **is recommended**

11. Surface Concepts and Imported Geometry

- Course Overview
- Hide/Show Tree Items
- Accessing Commands
- Importing data
- 3D Model types
- Definitions
- Creating Solids from Surfaces
- Decomposing a Solid into Surfaces
- Additional Surface Concepts
- Importing and Mold Design
- SOLIDWORKS Import Options
- Comparing Geometry
- Addressing Translation Errors
- 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
- Manual Parting Surface Techniques

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

➔ Continued…

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SOLIDWORKS Mold Design (continued…)

7. Alternate Methods for Mold Design

- Alternate methods for mold design
- Using Combine and split
- Creating a cavity
- Using surfaces
- Techniques for Mold Tooling
- Using the up to surface method
- Using the split method

8. Reusable Data

- Reusing data
- Task pane
- SOLIDWORKS Resources
- Design library
- Files 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
- Completing the process

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

1. Fundamentals of Routing

- What is Routing?
- 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 Template
- Editing a Route
- Routing Along Existing Geometry

3. Piping Fittings

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

4. Tubing Routes

- Tubing routes
- Tubes and tubing components
- Flexible tubing with auto route
- Orthogonal tubing routes with auto route
- Bend and Spline Errors
- Starting a tube route on the fly
- Making Tubes
- Repairing a Bend Errors
- Tubing Drawings

5. Piping and Tubing Changes

- Piping and Tubing Changes
- Pipe Penetrations
- Flange to flange Connections
- Pipe Spools
- Copying Routes
- Adding Slope
- Editing Piping Routes
- Editing for Obstructions
- Piping Drawings

6. Creating Routing Components

- Routing library parts
- Libraries
- Creating Routing library parts
- Pipe & 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

7. Using P&ID Files

- Piping and instrumentation
- Adding a pipe
- Pipe with multiple lines
- Pipe with In-Line Fittings
- Process drawings

8. Electrical Ducting, Cable Tray, and HVAC Routes

- Electrical Ducting routes
- Cable tray routes
- HVAC routes

9. Piping Skids

- Piping Skids
- Grid system feature
- Weldments
- Walk-through animations
- Avatar paths

10. Using SOLIDWORKS content

- Using SOLIDWORKS content

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the learned features.
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SOLIDWORKS What's New – 1 Day (7h)

1. Welcome to SOLIDWORKS 2020

- Top Enhancements
- Performance Improvements

2. User Interface

- Markups
- Changes to the CommandManager Menu
- Display Feature Names in a Different Language
- Drag Multiple Contiguous Features to a Folder
- Enhanced Large Tooltips for Toolbars
- File Types List
- Searching for Materials
- Touch Mode
- Troubleshooting Enhancements
- Updated Help Menu
- View Recent Files
- Welcome Dialog Box

3. SOLIDWORKS Fundamentals

- 3D Printing
- Opening Documents
- Application Programming Interface
- Changes to System Options and Document Properties
- Closing a Document When Saving a Copy
- Selection Sets
- Specifying the Up Axis for View Orientation

4. Assemblies

- Envelope Publisher
- Flexible Components
- Mates
- Patterns
- Large Design Review
- Exploded Views
- External References Options
- Isolating Interferences
- Modify Configuration for Toolbox Components
- Performance Improvements for Assemblies
- Save as Copy with a Subset of Configurations

5. Detailing and Drawings

- Custom Scales for Drawing Sheets and Views
- Chain Dimensions
- Alternate Position Views
- Detailing Mode
- Render Pipeline in Drawings
- Hole Callouts
- Annotations
- Cosmetic Threads
- Creating Markups for Drawings
- Design Library
- Dimensions
- Link Child Views to Parent by Default
- Location for Saving a New Drawing
- Performance Improvements in Drawings

6. Model Display

- Body Compare
- Comparing Bodies
- Body Compare PropertyManager

7. Parts and Features

- Graphics Mesh and Mesh BREP Bodies
- Holes
- Repairing Missing References for Fillets and Chamfers
- Surfaces

8. Sheet Metal

- • Convert to Sheet Metal
- • Tab and Slot

➔ Continued…

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SOLIDWORKS What's new (continued…)

9. Sketching

- Applying Torsion Continuity Relations
- Baseline and Chain Dimensions in Sketches
- Importing 2D DXF or DWG Files as Reference Sketches
- Power Modify Tool
- Silhouette Entities

10. Structure System and Weldments

- Options for Creating Primary Members Based on Points
- Creating Curved Beams and Merging Tangent Members
- Split Members
- Pattern and Mirror Support
- Member and Corner Trim Enhancements
- Cut List Properties for Weldment and Structure System Features

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SOLIDWORKS Refresher – 4 days (28h)

** 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 min: https://www.solidworks.com/sites/default/files/ cswasampleexam.zip*

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
- Tips and Tricks in sketch mode

2. Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed Only.
- 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
- Replace Sketch Entity

5. Configurations

- Configurations
- Using Configurations
- Other Methods to Create Configurations
- Modeling Strategies for
- Editing Parts that Have
- Design Library
- In the Advanced Course

6. Bottom-Up Assembly Modeling

- Bottom-Up Assembly
- Creating a New Assembly
- Position of the First Component
- FeatureManager 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
- Bill of Materials
- Assembly Drawings

➔ Suite…

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SOLIDWORKS Refresher (continued…)

8. Multibody Design Techniques

- Multibody PartsFonction Grille
- 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
- File References
- Solving Mates
- Advanced Mate Techniques
- Mate References
- Design Library Parts
- Capture Mate References
- Multiple Selection Mate References
- Multiple Mate Mode
- Driven Mates
- Using Misaligned Mates
- Copying Multiple Components
- Fixed Components
- Summary: Inserting and Mating Components
- Advanced Mate Features
- Profile Center Mate
- Rack Pinion Mate

10. Assembly Editing

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

11. 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
- Comments
- 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
- Key Topics
- Assembly Modes
- Assembly Visualization
- Lightweight Components
- Large Assembly Mode
- Using SpeedPak
- Using Simplified Configurations
- Defeature
- Modifying the Structure of an Assembly
- Envelope Publisher
- Large Design Review
- Comparison of Modes and Methods
- Tips for Faster Assemblies
- Drawing Considerations

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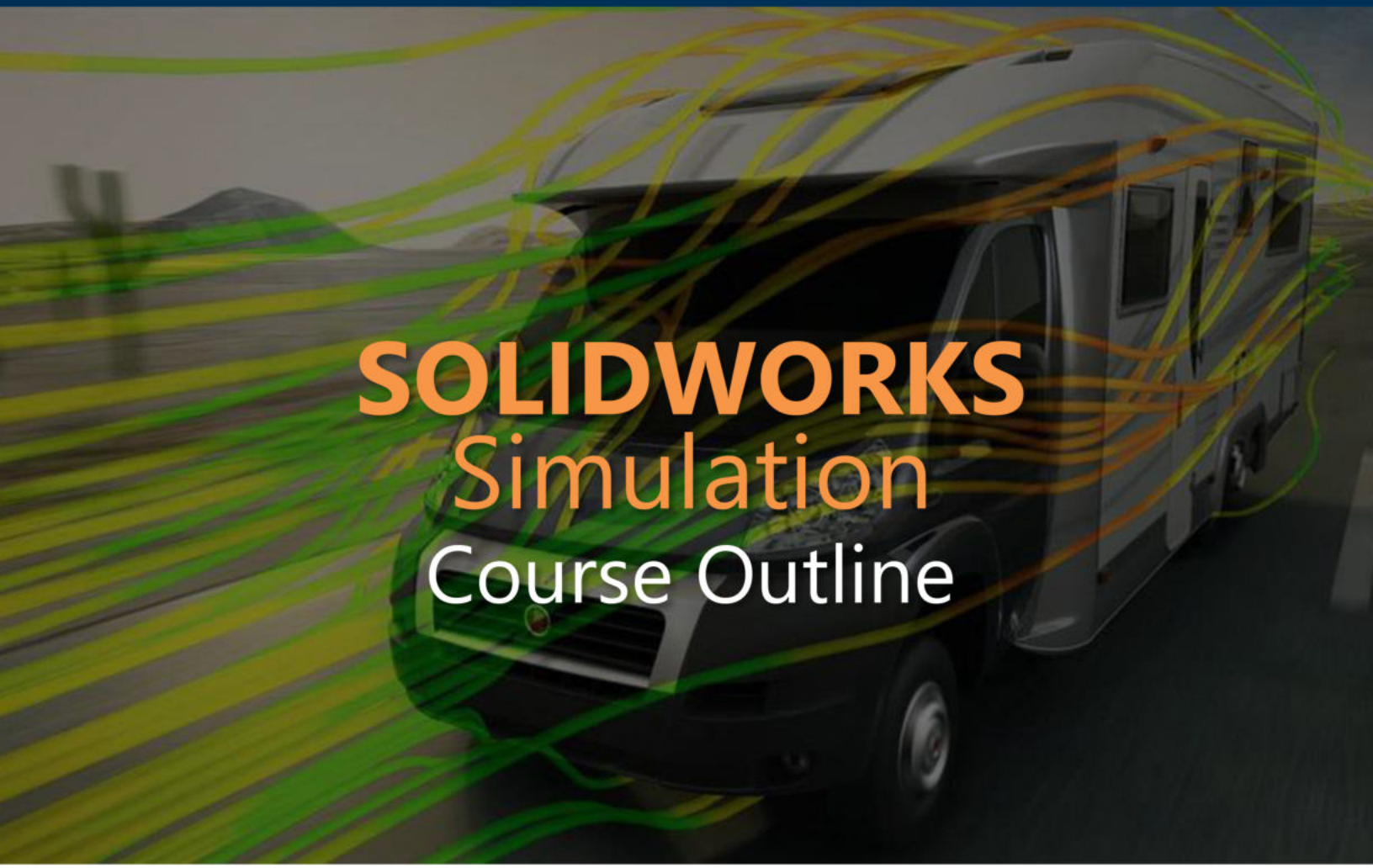

SOLIDWORKS Simulation

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

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SOLIDWORKS Simulation Static – 3 Days (21h)

1. The Analysis Process

- The Analysis Process
- SOLIDWORKS simulation options
- Preprocessing
- Meshing
- Processing
- Post-processing
- Multiple studies
- Reports

2. Mesh Controls, Stress Concentrations and Boundary Conditions

- Mesh control
- Understanding the effect of boundary conditions

3. Assembly Analysis with Contacts

- Contact analysis
- No penetration or bonded contact
- Pliers with global contact
- No penetration local contact: Accuracy

4.Symmetrical and Free Self-Equilibrated Assemblies

- Shrink fit parts
- Analysis with soft springs

5. Assembly Analysis with Connectors and Mesh Refinement

- Connecting components
- Connectors
- Mesh Control in an Assembly
- Problem statement
- Draft quality coarse mesh analysis
- Mesh analysis

6. Compatible/ Incompatible Mesh

- Compatible /incompatible meshing
- Centrifugal force
- Cyclical symmetry

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
- Results Comparison

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

- Design study
- Multiple load cases
- Geometry modification

➔ Continued…

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SOLIDWORKS Simulation Static (continued…)

12. Thermal Stress Analysis

- Thermal stress analysis
- Saving models in its 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 non-linear analysis

Appendix A. Meshing, Solvers, and Tips & Tricks

- Meshing Strategies
- Geometry Preparation
- Mesh Quality
- Mesh Controls
- Meshing Stages
- Failure Diagnostics
- Tips for Using Shell Elements
- Hardware Considerations in Meshing.
- Solvers in SOLIDWORKS Simulation
- Choosing a Solver
- Email Notification Settings

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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

- Contact forces
- STEP function
- Contact: Solid bodies
- Geometrical description of contacts
- Integrators
- Instability points
- Modifying result plots
- Path Mate Motor

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 Optimisation

- Motion Optimisation
- Sensors
- Optimisation 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 load
- Direct solution in SOLIDWORKS motion

11. Event Based Simulation

- Event based simulation
- Servo motors
- Sensors
- Task

12. Design Projects (Optional)

- Design Project
- Self-guided problem – **Part 1**
- Self-guided problem – **Part 2**
- Problem solution – Part 1
- Creating the force function
- Force expression

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SOLIDWORKS Simulation Professional – 2 Days (14h)

SOLIDWORKS Simulation Static* **is a prerequisite 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 contact conditions
- Bonded and allow penetration contacts

3. Buckling Analysis

- Buckling analysis

4. Load Cases

- Load Cases

5. Sub-modeling

- Sub-modeling

6. Topology Optimization

- Topology Analysis
- Manufacturing Controls
- Mesh Effects
- Load Cases in Topology Studies
- Export Smoothed Mesh

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 stress analysis
- Thermal analysis
- 3D model
- Microchip Testing Assembly

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

12. Drop Test Analysis

- Drop test analysis
- Rigid floor drop test
- Elastic floor, Elasto-Plastic Material
- Elasto-plastic material model
- Drop test with contact (optional)

13. Optimization Analysis

- Optimization analysis
- Static and frequency analyses

14. Pressure Vessel Analysis

- Pressure vessel analysis
- Manhole nozzle flange and cover

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SOLIDWORKS Simulation Premium – 3 Days (21h)

SOLIDWORKS Simulation Static* **and *SOLIDWORKS Simulation Professional* **is a prerequisite 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. Non-Linear Static Buckling Analysis

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

4. Plastic Deformation

- Plastic deformation
- Problem statement
- Linear elastic
- Nonlinear – von Mises
- Nonlinear – Tresca's
- Stress accuracy
- Non-linear 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 Contact Analysis

- Problem statement

8. Metal Forming

- Bending

DYNAMIC

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-810G

- Model with remote mass

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 the course, the student will know the capabilities of the software and will be able to use the features.
Training Course: Training is given in class at SolidXperts or online where each student has access to a workstation or to an online 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 class work, the instructor will show the exercises and explain the solutions to the entire class.
Instructor: SolidXperts trainers are Certified SOLIDWORKS Instructors (CSWI) and authorized by Emplois Québec.
Course Materials: One or more training manuals are included with the training course (Exceptions apply to teachers and students from educational institutions).
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SOLIDWORKS Simulation Premium Composite – 1 Day (7h)

******SOLIDWORKS Simulation Static* **and** *SOLIDWORKS Simulation Professional* **is a prerequisite 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
- Summary
- Reference

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the features.
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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
- Noise Prediction
- Tangential faces of rotors
- Time step

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 trajectories

14. Supersonic Flow

- Supersonic flow

15. FEA Load Transfer

- FEA Load Transfer

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SOLIDWORKS Flow Simulation: Electronic Cooling Module – 1 Day (7h)

****SOLIDWORKS Flow Simulation* is a prerequisite for this class.**

1. Introduction to Electronics Module

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

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the features.
Training Course: Training is given in class at SolidXperts or online where each student has access to a workstation or to an online 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 Flow Simulation: HVAC Module – 1 Day (7h)

******SOLIDWORKS Flow Simulation* **is a prerequisite 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 the course, the student will know the capabilities of the software and will be able to use the features.
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SOLIDWORKS Plastics 1.5 Days (10h), 2 Days (14h) or 3 Days (21h)

The SOLIDWORKS Plastics training manual covers all the features and functions of the SOLIDWORKS Plastics application.
Lessons 1 through 8 cover the features of SOLIDWORKS Plastics Standard (1.5 day).
Lessons 1 through 14 cover the features of SOLIDWORKS Plastics Professional (2 day)
Lessons 1 through 19 cover the features of SOLIDWORKS Plastics Premium (3 day).

1. Basic Flow Analysis

- Basic Flow Analysis
- Element Types
- Meshing
- The PlasticsManager Tree
- Material
- Injection Location
- Running a Flow Analysis
- Flow Results

2. Detecting Short Shots

- Detecting Short Shots
- Fill Settings
- Flow Front Central Temperature

3. Automation Tools

- Automation Tools
- Duplicate Study
- Copying Settings
- Batch Manager

4. Injection Locations and Sink Marks

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

5. Materials

- Materials Properties
- User-Defined Database
- Resin Properties
- Heat Transfer Properties
- Viscosity
- PCT Data
- Mechanical Properties

6. Mesh Manipulation

- Local Refinement of Mesh
- Element Issues
- Mesh Editing
- Leader Lines
- Solid Mesh
- Solid Mesh types

7. Detecting Air Traps

- Detecting Air Traps
- Air Traps
- Venting

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
- CoolingTimes

10. Multiple Cavity Molds

- Multiple Cavity Molds
- Mold Layouts
- Channel Design
- Runner Channel Design
- Runner Wizard Channel Design
- Family Mold Layout
- Using Runner-Balancing

11. Symmetry Analysis

- Symmetry Analysis
- Symmetry Face

➔ Continued…

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the features.
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SOLIDWORKS Plastics (continued…)

12. Valve Gates and Hot Runners

- Valve Gates and Hot Runners
- Hot Runners
- Valve Gates

13. Reaction Injection Molding

- Reaction Injection Molding

14. Using Inserts

- Using inserts
- Cavities and Inserts
- Materials for Inserts

15. Multi Shot Mold

- Multi Shot Mold

16. Gas Assistance Molding

- Using Inserts
- Gas Assist

17. Cooling Analysis

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

18. Warpage Analysis

- Warpage Analysis
- Shrinkage
- Warpage
- Warp Settings
- Flow, Pack and Warp Analysis

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the features.
Training Course: Training is given in class at SolidXperts or online where each student has access to a workstation or to an online version.
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SOLIDWORKS Sustainability– 1 Day

1. SustainabilityXpress

- Evaluating the Environmental Impact of a Part
- Selecting a Material
- Using the Find Similar Material Tool
- Setting the Manufacturing Options
- Setting Use Options
- Using the Environmental Impact Dashboard
- Impact Assessment Methodologies
- Creating a Report
- Exporting Settings for a SOLIDWORKS Sustainability Study
- Adding Materials
- Updating the Materials Database

2. SOLIDWORKS Sustainability

- SOLIDWORKS Sustainability Overview
- Sustainability Task Pane Views
- Evaluating the Environmental Impact of a Part
- Selecting a Material
- Using the Find Similar Material Tool
- Setting the Manufacturing Options
- Setting Use Options
- Using the Environmental Impact Dashboard
- Impact Assessment Methodologies
- Creating a Report
- Exporting Settings for a SOLIDWORKS Sustainability Study
- Evaluating the Environmental Impact of an Assembly
- Preparing the Assembly to Be Evaluated
- Defining the Assembly Process
- Specifying Assembly Use Parameters
- Modifying Transportation Defaults
- Modifying End of Life Disposal Defaults
- Specifying Duration of Use
- Modifying Components
- Adding Materials
- Adding Sustainability Data to a Custom Material
- Assigning Financial Impact to a Custom Material
- Updating the Materials Database
- Requesting Materials
- Visualizing Sustainability Properties

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the features.
Training Course: Training is given in class at SolidXperts or online where each student has access to a workstation or to an online version.
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SOLIDWORKS PDM

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

PDM TRAINING INDEX

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Update SOLIDWORKS PDM Standard to Professional – 1 day (7h) 41
SOLIDWORKS PDM Professional API Fundamentals – 2 Days (14h) 42

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Administration of SOLIDWORKS PDM Professional – 3 Days (21h)

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 Cards Types
- Cards 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
- Bill of Materials Columns

7. Workflow

- Categories
- Creating some Workflows
- Revisions
- Revisions Tables and Drawings
- Transitions PDM Professional

8. Notifications and Tasks

- Notifications and Conditional Notifications
- Message System
- Various Conversion Tasks

9. Folder Templates

- Creating Folder Template

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 Setup
- CircuitWorks Setup

Course Objectives: At the end of the course, the student will know the capabilities of the software and will be able to use the learned features.
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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 Cards 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
- Revisions Tables and Drawings

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 Files Vaults

Appendix

- File Types and Settings
- Toolbox Setup

Not included:

- Installation process,
- Routing Setup
- CircuitWorks Setup

Course Objectives: At the end of the course, the student 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 to an online version.
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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

- 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
- See Notifications

8. Working in SOLIDWORKS*

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

** Lesson for SOLIDWORKS users only*

Appendix

- Use Bill of Materials (BOM)

SPECIFIC NOTES CONCERNING THIS TRAINING COURSE

***Target audience** : Users who do not work with SOLIDWORKS do not need to take the last lesson of the training.*
***Training Course**: This training is usually given to a larger number of participants. In order to accommodate customers, this training is held at the client's office or online rather than in the laboratory class at SolidXperts.*
***Methodology**: Training is based on case studies demonstrated by the instructor. Given the ease of use of PDM, there is no practical exercise at the end of each lesson.*
***Competences Evaluation**: There is no formal competences assessment at the end of the training given the ease of use of PDM.*

Course Objectives: At the end of the course, the student 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 to an online version.
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Update SOLIDWORKS PDM Standard to Professional – 1 day (7h)

*** The numbers which are not in sequential order, indicate the link in the chapters of the training: Administration of SOLIDWORKS PDM Professional**

ADMINISTRATION OF PDM

2. Administration Tool

- Administration Tool
- Color of the vaults
- Additional fields

3. Users and Groups

- Windows connection
- User settings: personalization

4-5. Folder dans File Datacards

- 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

- Creation of a folder template

10. File Templates

- Creation of a file template

12. Vault Backup

- Maintenance plan in SQL Server Management Studio

Appendix

- Import / Export of data
- Overview: Replication
- Overview: Web2
- Overview: Report generator
- Overview: PDMXperts tools, personalized programming, dispatch

USING OF PDM

2. 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

- Utilisation des nomenclatures nommées

Not included :

- Installation process,
- Routing Setup and CircuitWorks Setup

Course Objectives: At the end of the course, the student 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 Values
- SOLIDWORKS PDM Professional API Help
- Interface Versioning

2. Files, Folders, Items 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 Enterprise PDM 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 Enterprise PDM Task Add-Ins
- Task Interfaces
- Task Hooks
- The IEdm Search Interface
- The IEdm Workflow Mgr Interface
- The IEdm Workflow Interface

Course Objectives: At the end of the course, the student 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 to an online version.
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Technical Communication

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TECHNICAL COMMUNICATION - TRAINING INDEX

SOLIDWORKS Composer – 3 Days (21h) *45*
SOLIDWORKS Inspection – 1 Day (7h) *47*
SOLIDWORKS MBD – 1 Day (7h) *48*

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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 View
- 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 Bill of Materials

- Bill of Materials
- Vector Graphic Output
- Another BOM Table
- Assembly Level BOM
- Assembly Selection Mode

6. Creating a Marketing Image

- Selections
- Textures
- Lighting
- Scenes
- High Resolution

7. Creating an 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
- Adding Views to the Timeline

➔ Continued…

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SOLIDWORKS Composer (continued…)

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

- Product Files
- Product Orientation

13. Publishing from SOLIDWORKS Composer

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

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SOLIDWORKS Inspection – 1 Day (7h)

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

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

3. SOLIDWORKS Inspection Professional

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

Appendix A: Inspection Report Templates

- Inspection Report Templates
- Template Editor

Appendix B: Understanding Regular Expressions

- Regular Expressions

Appendix C: Glossary of Quality Terms

- Terminology

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SOLIDWORKS MBD – 1 Day (7h)

1. Introduction to SOLIDWORKS MBD

- What is SOLIDWORKS MBD?
- Course Layout
- 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
- Steps in the Process

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
- Supplemental Tutorials

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

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SOLIDWORKS Visualize

Course Outline

SOLIDXPERTS
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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

Visualize TRAINING INDEX

SOLIDWORKS Visualize – 2 Days (14h) 51

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SOLIDWORKS Visualize – 2 Days (14h)

1. CAD to SOLIDWORKS Visualize

- Rendering from CAD
- Importing to Visualize
- Render Selection
- Denoiser
- Appearances
- File Libraries
- Scenes
- 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 Type Parameters
- Merge Parts

3. Decals

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

4. Cameras

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

5. Backplates, Environments and Lights

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

6. Productivity tools

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

➔ Continued…

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SOLIDWORKS Visualize (continued…)

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 Images
- Panorama View
- Sun Study
- 360 Camera

11. CAD to SOLIDWORKS Visualize

- Simulated physics
- Shake simulation
- Simulation manager
- Simulation states
- Vehicle simulation

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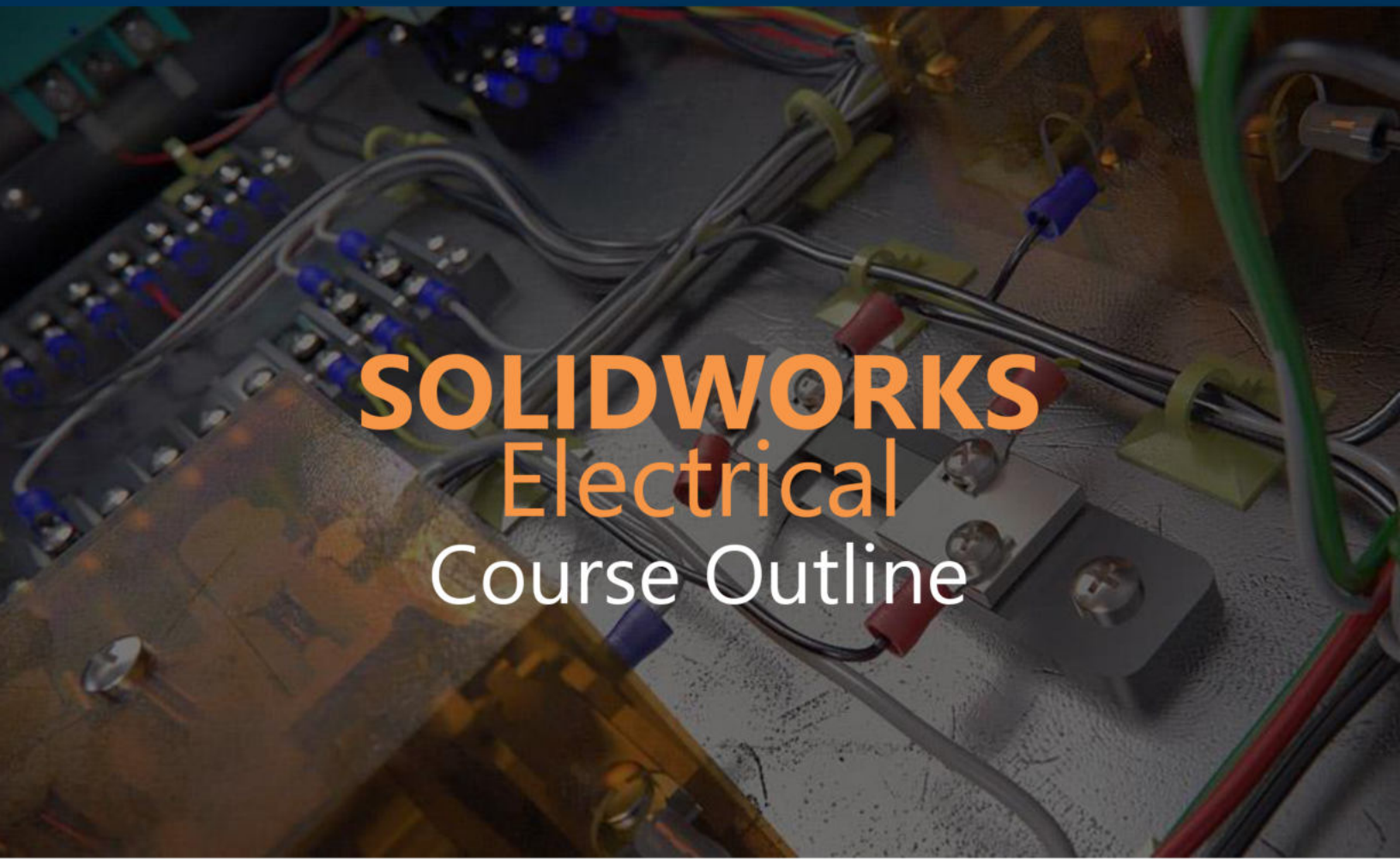

SOLIDWORKS Electrical

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

SOLIDWORKS ELECTRICAL TRAINING INDEX

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SOLIDWORKS Electrical: 3D – 1 Day (7h) 57
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SOLIDWORKS PCB Essentials – 1 Day (7h) 59

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SOLIDWORKS Electrical: Schematic – 2 Days (14h)

1. Project Templates

- SOLIDWORKS Electrical
- Stages in the Process
- 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 wire
- 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 attribute
- Splitting attribute data
- Add to library
- Copy, paste symbol

9. Macros

- What are Macros
- Creating and Adding Macros

➔ Continued…

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SOLIDWORKS Electrical: Schematic (continued…)

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?

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SOLIDWORKS Electrical: 3D – 1 Day (7h)

1. Assembly Creation

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

2. Cabinets, Ducts, Rails

- Cabinets, Ducts, Rails
- Insert Component
- 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

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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
- Save to External Files

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 Though 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 harness
- 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

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SOLIDWORKS PCB Essentials – 1 Day (7h)

1. SOLIDWORKS PCB Basics and the User Interface

- Overview
- SOLIDWORKS PCB Environment

2. Working with PCB Design Projects

- Understanding Projects
- Creating Project
- Creating Project Documents
- Basic Project Management Tasks

3. Creating Schematics Templates

- Understanding Templates
- Creating One Template from Another
- Inserting a Company Logo
- Setting Document Text Parameters
- Setting Template Project Parameters

4. Configuring the Schematic Preferences

- Optimizing Wires and Buses
- Breaking Wires at Auto junctions
- Displaying Cross-Overs
- Auto Panning

4. Creating Symbols

- Creating New Symbols
- Using the Symbol Wizard

6. Populating Schematics

- Using Symbol Placement Shortcuts
- Placing Library Components
- Placing Parts
- Inserting Power Ports
- Applying Supplier Links

7. Creating Schematic Connections

- Wiring Placement Modes
- Placing Wire Connections
- Creating Buses
- Using Net Labels

8. Using Schematic Annotations

- Processing Order
- Processing Location
- Matching Options
- Proposed Change List
- Engineering Change Order

9. Compiling and Verification

- Setting Design Violations
- Compiling and Realizing the Results
- Resolving Error Violations and Warnings

10. Collaborating with SOLIDWORKS

- Creating a PCB Board in SOLIDWORKS
- Pushing a Board to SOLIDWORKS PCB
- Creating a PCB Board in SOLIDWORKS PCB
- Pushing a Board to SOLIDWORKS

11. Configuring Layers and PCB Stacks

- Configuring PCB View Configurations
- Defining the Board Layer Stack
- Checking the Board Thickness in SOLIDWORKS

➔ Continued…

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SOLIDWORKS PCB Essentials (continued…)

12. Designing Rigid-Flex PCBs

- Using the Rigid-Flex Approach for Designing PCBs
- Understanding the Collaboration Workflow of Rigid-Flex PCBs
- Repositioning Components on a Rigid-Flex Board

13. Configuring the Outline, Cutouts and Keepouts

- Redefining the Board Shape
- Applying Cut outs
- Applying Keep outs
- Defining PCB Placement Constraints in SOLIDWORK

14. Configuring Origins and Grids

- Setting an Origin
- Creating a Cartesian Grid
- Creating a Polar Grid

15. Transferring Design Data

- Linking Components
- Updating Schematics
- Updating the PCB Layout

16. Creating Fooprints

- Creating New Footprints
- Using the IPC Footprint Wizard

17. Placing Footprints

- Positioning Footprints
- Reposition Footprints in SOLIDWORKS

18. Using Design Rule Checks

- Modifying the Existing Rules
- Creating New Rules

19. Routing

- Interactive Routing Preferences
- Interactive Routing Nets
- Quick Routing
- Adding Vias
- Multi-Trace Routing
- Auto routing
- Adjusting the Tracks to Fix Errors

20. Defining Polygon Pour

- Setting Polygon Pour Parameters
- Defining Polygon Pours Nets

21. Inspection – Global Edition

- Finding Similar Objects
- Modifying Multiple Objects

22. Outputting Data

- Configurating Output Files
- Generating Manufacturing Output

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SOLIDWORKS ADD-IN

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

ADD-IN AND 3D PRINTER TRAINING INDEX

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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 Statement

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
- 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 of

20. Windows

21. Operations on the 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)
Introduction to Visual Basic.net* **is a prerequisite 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 Option
- 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. Notification

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

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SOLIDWORKS Tolanalyst – 1 Day (7h)

*** No books currently exist for the TolAnalyst course. The duration of the training is about half a day, but we will complete it with an example on one of your models or coaching on your examples.**

***There is no book for this class.**

1. DimXpert

- DimXpert Overview
- Auto Dimension Scheme
- Datums
- Size Dimensions
- Location Dimensions
- Geometric Tolerances
- Tolerance Status
- DimXpert Options
- Using DimXpert Information in Drawings
- Pattern Feature

2. TolAnalyst

- TolAnalyst Overview
- Establishing the Measurement
- Assembly Sequence
- Assembly Constraints
- Analysing the Results
- Fixed/Floating Fasteners

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DriveWorks Solo – 3 Days (21h)

Lesson 1

- Basic Setup
- Capturing your Models

Lesson 2

- Project Designer

Lesson 3

- Building Rules

Lesson 4

- Improving your Project

Lesson 5

- Static Replacement Files

Lesson 6

- Tables

Lesson 7

- Form Navigation

Lesson 8

- Enhancing your Forms
- Dynamic Replacement Files

Lesson 9

- Driving Custom Properties

Lesson 10

- Documents

Lesson 11

- Drawings

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DriveWorks Administrator – 4 Days (28h)

Lesson 1

- Creating a Group and Capturing Models

Lesson 2

- Building a user interface in DriveWorks Administrator

Lesson 3

- Building Rules

Lesson 4

- Running your Project

Lesson 5

- File Name and Relative Path Rules

Lesson 6

- Tables

Lesson 7

- Form navigation
- Form templates
- Static and dynamic control properties
- Advanced form controls

Lesson 8

- Dynamic replacement files

Lesson 9

- Data management

Lesson 10

- Documents

Lesson 11

- Drawings

Lesson 12

- Specification flow
- Preparing your models for automation

Lesson 13 (Advanced)

- Advanced form controls

Lesson 14 (Advanced)

- Specification control

Lesson 15 (Advanced)

- Linking to data

Lesson 16 (Advanced)

- Rollup data tables

Lesson 17 (Advanced)

- Hierarchical properties

Lesson 18 (Advanced)

- Macro buttons

Lesson 19 (Advanced)

- Generation tasks

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DraftSight – 1 Day (7h)

1. User Interface

- Pointing Device -DraftSight Window
- Title Bar - Pull-Down Menu
- Toolbars - Graphic Windows
- Commande Line - Status Bar
- Shortcut Menus - Dialog Boxes
- Keyboard Options - Help Menu

2. Creating a Simple Drawing

- New - Line
- Delete Command - Rectangle
- Circle -Drafting Settings
- Arc - Undo and Redo
- Polygon- Drawing Setup

3. Coordinates

- Coordinated System- CCS Icon
- Inquiry - Units
- Coordinate Entry - Snap from Entity Snap
- Drawing Boundary

4. Modify Commandes

- Selecting Entities
- Move Command - Copy Command
- Offset Command - Mirror Command
- Rotate Command - Scale Command
- Trim Command - Extend Command

5. Properties and Layers Toolbars

- Layer Control - Activate Layer Command
- Color Control Command - LineStyle Control Command
- LineWeight Control Command
- Layers Manager Command
- Property Painter
- Properties Command

6. Drawing Files

- New – Save - Open
- File Management - File Utilities

7. Advanced Commands

- Point - Point Format
- Mark Divisions - PolyLine
- Explode- Edit PolyLine
- Blocks -Insert Block
- Clean -ExportDrawing
- Hatch - Pattern
- Stretch - Change Length
- Split - Fillet- Chamfer
- EntityGrips- Egrip Settings

8. View Commands

- Dynamic Pan - Dynamic Zoom
- Zoom Window - Zoom Previous
- Zoom Options - Rebuild
- Named Views - Multiple ViewTiles

9. Text and Dimensions

- Simple Note – Note
- Text Style – Edit Annotation
- Find and Replace – Spell Check
- Dimensions – Dimensions Style
- Edit Dimension Location - Edit Dimension Text
- Edit Dimension Properties

10. Drawing Output and Layout

- Drawing Layout – View Tiles
- Working with Viewports on Sheets- Print

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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. Advance Operations

- Advance Manipulations
- Thicken
- Symmetry

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SOLIDWORKS CAM Standard – 2 Days (14h)

1. SOLIDWORKS CAM Basics and User Interface

- What is SOLIDWORKS CAM?
- SOLIDWORKS CAM User Interface
- Process 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
- Save Operation Plan

5. Merging Features and Operations

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

6. Avoid and Contain Areas

- Add Avoid and Contain Areas

7. Pattern Features and Mirror Toolpaths

- Patterning 160
- Mirror Toolpaths

8. Advanced Features and Operations

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

9. Customizing the Technology Database

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

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SOLIDWORKS CAM Professional – 1 Day (7h)

*** It is not possible to give this course outline in 1 day, but since the majority of our clients only use 50% of the content, the training offered is one day in duration and focused on your manufacturing methods. If you need all of the content, it is possible to have this training on 2 days. Check with 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
- Machine Setup
- Part Manager
- Stock Manager
- Assembly Machining -Programming with Subroutines
- Machining - Multiple Parts
- Import Part Data
- Split Instance
- Split Setup

4. 3 Plus 2 Machining

- 3 Plus 2 Machining (Indexing)
- Indexing

5. Turning Basics

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

6. Chucks, ID Features and Operations

- Section Method
- Double Chucking
- 7. Modifying Feature and Operation Parameters
- Editing toolpaths

7. Probing

- Introduction to Probing

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SWOOD Design – Essential – 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

7. 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

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SWOOD CAM – 2 Days (14h)

SWOOD Design – Essential* **is a prerequisite for this class.

1. Integrating SWOOD CAM into SOLIDWORKS

- Add-ins
- SWOOD Settings

2. SW Settings for SWOOD CAM

- Required Configuration
- Managing Views
- Complex Assemblies
- Customising 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 OP0
- Creating a 3D Roughing Operation (Roughing & Finishing)

8. 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

9. Link with SWOODDESIGN

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

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SWOOD Advanced – 1 Day (7h)

****SWOOD Design – Essential* is a prerequisite 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

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3D Printer
Installation and how to use
Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

3D PRINTER - TRAINING INDEX

Markforged (Installation) – 1 Day 77
Design for Additive Manufacturing (DFAM) – 1 Day 78

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

Course Objectives: At the end of the course, the student 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 to an online 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 class work, 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 Emplois Québec.
Course Materials: One or more training manuals are included with the training course.
Attestation: A certificate will be given to the student at the end of the course to attest the successful completion of the requirements for the course.

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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. Produce 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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The training offered by SolidXperts emphasizes the skills and concepts essential to your success in the field of 3D design.

By being an authorized SOLIDWORKS training center, SolidXperts offers the highest quality basic and advanced training. Our trainers are certified by Dassault Systèmes SOLIDWORKS Corp. Each training includes a training book and takes place in a room or online where each student has access to a workstation. SolidXperts is also a training center approved by the CPMT on behalf of the Minister of Employment and Social Solidarity. We can therefore issue certificates in accordance with the "Law promoting the development and recognition of workforce skills" (Law of 1%).