SOLIDWORKS
Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

SOLID**X**PERTS
by solidxperience

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.

PRODUCT TRAINING MAIN INDEX

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SOLIDWORKS Training

Course Outline

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TABLE OF CONTENTS - SOLIDWORKS TRAINING

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

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. Introduction to Sketching

- 2D sketching
- Saving files
- Sketching
- Sketch entities
- Basic sketching
- Rules that govern sketches
- Design intent
- Sketch relations
- Dimensions
- Extruding
- 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

- 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
- 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

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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. 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 variables 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 views
- 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

- Option settings
- Document templates

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SOLIDWORKS Drawings ANSI – 3 Days (21h)

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 views

2. Understanding Drawing Templates

- Structure of a drawing document
- Drawing documents
- Drawing sheets
- Sheet formats
- Understanding drawing templates
- Drawing template design strategy
- Design a drawing template
- Creating a sample model and drawing

3. Customizing the Sheet Format

- Title block sketches
- 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 the sheet format
- Testing sheet format properties

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 views
- Auxiliary views
- Rotating views
- Crop views
- Understanding view focus
- Advanced views for assemblies
- Section scope
- Alternate position views
- Using configurations
- Custom view orientations
- New views
- Relative views
- 3D drawing views

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SOLIDWORKS Drawings ANSI *(suite)*

7. Understanding Annotation Views

- Understanding annotation behavior
- What are annotation views?
- Annotation folders
- Default annotation views
- Annotation view visibility
- Insert annotation views
- Editing annotation views
- Annotation updates
- 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
- Dimension styles
- Annotations in the Design Library
- Flag note bank

10. Advanced Options for BOM Tables

- Tables in SOLIDWORKS
- Bills of Material properties
- Displaying the BOM assembly structure
- Modifying a table
- Saving a table template
- Properties in BOM tables
- BOM component options
- Balloon Indicator

11. Additional SOLIDWORKS Tables

- Inserting a Hole Table
- Splitting tables
- 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

- Gestion des performances
- 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'll replace the SOLIDWORKS offered exercise files with your documents. You are therefore asked to bring your own files to produce your drawing documents.

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

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 Flat Patterns

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

3. Additional Sheet Metal Techniques

- Additional sheet metal methods
- Designing from the flat
- Sketch 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

4. Converting to Sheet Metal

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

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SOLIDWORKS Sheet Metal *(Part 2)*

5. Multibody Sheet Metal Parts

- Multibody sheet metal parts
- Multibodies with base flanges
- 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
- Split with sheet metal parts
- Patterning for multibodies
- Using edge flange to merge bodies
- Interfering bodies
- Combining sheet metal with other bodies

6. Forming Tools and Gussets

- Sheet metal forming tools
- Standard forming tools
- Form tool features
- Part document properties
- Custom form tools
- Split lines
- Forming tool
- Forming tool in drawings
- Sheet metal gussets

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

CutXperts (Tool from SolidXperts)

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

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SOLIDWORKS Assembly Modeling – 2 Days (14h)

1. Advanced Mate Techniques

- SOLIDWORKS assemblies
- Assembly file structure
- Opening assemblies
- 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
- Removing external references

3. Assembly Features and Smart Components

- Assembly features
- Hole series
- Smart fasteners
- Smart components
- Flexible components

4. 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

5. 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
- Equations with functions
- Sensors
- Using the Mate Controller

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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 sssemblies
- Key topics
- Assembly modes
- Assembly visualization
- Lightweight components
- Large assembly settings
- Using SpeedPak
- Using simplified configurations
- Defeature
- Modifying the structure of an assembly
- Envelope publisher
- Large design review
- Modes and methods comparison
- Tips for faster assemblies

8. Facility Layout

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

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SOLIDWORKS Weldments – 1 Day (7h)

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 box
- Structural member properties
- Adding Cut List properties
- Bounding boxes in weldments
- Options for generating Cut List items
- Custom structural member profiles
- Defining material
- Creating custom profiles
- Standard or configured profiles
- Inserting existing parts
- When to use an assembly

3. Configuring and Detailing Weldments

- Weldment configurations
- Post-assembly machining features
- Weldment drawings
- Individual body drawing views
- Representing welds

4. Working with Bent Structural Members

- Working with bent structural members
- 3D sketching

5. Introduction to Structure Systems

- Structure systems
- Primary vs. Secondary members
- Secondary members

6. Working with Structure System

- Managing the Cut List
- Gussets and end caps
- Bounding boxes in structure systems

7. Defining and Inserting Connection Elements

- Connections
- Defining connection elements
- Inserting connection elements

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SOLIDWORKS Advanced Part Modeling – 2 Days (14h)

1. 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

2. Saving Solid Bodies

- Multibody parts vs. Assemblies
- Saving body functions
- Inserting into new part
- Saving 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 Selection Manager

5. 3D Sketching and Curve Features

- Curve features
- Sweeping along a 3D path
- 3D sketching
- Helix curves
- 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 a sweep path
- Sweeping along model edges

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SOLIDWORKS Advanced Part Modeling *(Part 2)*

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 boundaries work
- Loft feature
- Boundary feature
- Copying a sketch
- Modifying a 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

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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
- Deconstructing a solid into surfaces
- Additional surface concepts
- Takeaways
- 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
- Copying faces
- Flattening surfaces

4. 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

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SOLIDWORKS Mold Design – 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

- Surfaces in mold design
- 3D model types
- Geometry vs. Topology
- Behind the scenes
- Creating solids from surfaces
- Deconstructing a solid into surfaces
- Additional surface concepts
- Surfaces concept 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 modeling
- SOLIDWORKS mold tools
- Mold analysis tools
- Analyzing drafts on a model
- Using the Draft Analysis tool
- Draft analysis options
- Adding drafts
- Scaling the model
- Establishing parting lines
- Shut-off surfaces
- Creating parting surfaces
- Surface bodies
- Creating the mold tooling
- Seeing inside the mold
- Interlocking the mold tooling
- Creating part and assembly files

3. Side Cores and Pins

- Additional Mold Tooling
- Trapped molding areas
- Side cores
- Feature Freeze
- Lifters
- Core Pins
- Manual Selection Techniques
- Modifying Shut-Off Surfaces
- Tooling completion

4. Advanced Parting Lines Options

- Manual parting lines
- Splitting a part

5. Creating Custom Surfaces for Mold Design

- Surface modeling for mold design
- Drill bezel
- Router bottom

6. Advanced Surfacing for Mold Design

- Surface modeling for mold design
- The Mixer
- Mixer rear housing
- Mixer handle

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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

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
- Process completion

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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 tool
- 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

8. Electrical Ducting, Cable Tray, and HVAC Routes

- Electrical ducting routes
- Cable tray routes
- HVAC routes

9. Using SOLIDWORKS Content

- Using SOLIDWORKS content

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SOLIDWORKS What's New – 1 Day (7h)
This course is given using digital files only (no physical book is provided)(

1. Welcome to SOLIDWORKS 2022

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

2. Installation

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

3. Administration

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

4. SOLIDWORKS Fundamentals

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

5. User Interface

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

6. Sketching

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

7. Parts and Features

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

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SOLIDWORKS What's New *(Part 2)*

8. Model Display

- Model display performance improvements

9. Sheet Metal

- Edge flanges
- Etched contours on bends

10. Structure System and Weldments

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

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

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

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- Linear patterns
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- Using pattern seed only
- Sketch-driven patterns

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- Editing topics
- Sketch issues

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- 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

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- Bottom-up assembly
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- Positioning 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

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 drawing

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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

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
- 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
- Key topics
- Assembly modes
- Assembly visualization
- Lightweight components
- Large assembly settings
- Using SpeedPak
- Using simplified configurations
- Defeature
- Modifying the structure of an assembly
- Envelope publisher
- Large design review
- Modes and methods comparison
- Tips for faster assemblies

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SOLIDWORKS
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Training

Course Outline

SOLID**X**PERTS

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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENT - SOLIDXPERTS

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SOLIDWORKS Performance Training – 2 days (14h)

Prerequisites : This training is intended for users with at least 1 year of experience

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 holder from external references
- Number of parts per assembly level
- Parts rebuild time
- Number of features in a part
- Parts in simple version
- 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
- Detailling 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

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CAD Administrators – 2 days (14h)

Prerequisites : Experience as an engineering manager or in IT.
Understanding of the Windows operating system.

1. SOLIDWORKS Computer Requirements

- Analysis of computer hardware
- Review of computer needs to ensure good performance and stability
- Certified graphics 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 an Go
- Design Library

6. Tools for CAD administrators

- CAD Admin Dashboard
- Design Checker
- Best practice document

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SOLIDWORKS

3DEXPERIENCE Works

Course Outline

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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENT - 3DEXPERIENCE WORKS

Configuration of the 3DEXPERIENCE – 1 day (7h)

1. **3DDashboard Platform Management**

2. **Validation of the roles and the qty (licenses)**

3. **Invitation and configuration of the members**

   - Roles
   - Applications
   - Additional options

4. **Adding and managing 3DSwym communities**

   - Manage communities
   - Manage administrators
   - Manage parameters

5. **Creating and configuring 3DSpace**

   - Drive Control Center
   - Collaborative Spaces Control Center
   - Collaborative Spaces Configuration Center
   - Attribute management
   - Maturity graph
   - PLM xCAD
   - Derived formats

6. **Creation and allocation of 3DDashboard**

   - Creation of an enterprise 3DDashboard
   - Sharing the 3DDashboard
   - Definition off the 3DDashboard by default

7. **3DSearch and 6WTags configuration**

   - Search parameters
   - 6WTags

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User Training of the 3DEXPERIENCE – 1 day (7h)

1. **What is the 3DEXPERIENCE platform?**

2. **Explanation of the cloud structure**

3. **The roles**

4. **Connecting SOLIDWORKS to the 3DEXPERIENCE platform**

   - Exercise | Cloud Eligibility Tool
   - Exercise | Installing the xCAD PLM connector and prerequisites

5. **Where are my files?
   3DDrive vs 3DSpace**

   - Exercise | Registering a SOLIDWORKS assembly on the 3DEXPERIENCE platform

6. **What is the purpose of the 3DDashboard?**

   - Exercise | Create your first 3DDashboard

7. **SOLIDWORKS content lifecycle and access control**

   - Exercise | Using SOLIDWORKS content and collaborative space

8. **SOLIDWORKS content management**

   - Exercise | Edit, manage and delete files

9. **Search on the platform**

   - Exercise | Practice searching for content

10. **Bookmarks**

    - Exercise | Create a bookmark and add content

11. **Collaborative Task**

    - Exercise | Using Tasks in SOLIDWORKS

3D Creator – 1 or 2 days (7h or 14h)
Prerequisites : Knowledge of data management in the 3DEXPERIENCE platform, parametric modelingand mechanical concepts.
Note : You must have a license for the 3D Creator role.

1. **Presentation of xDesign**
2. **Creating and opening designs**
3. **Design management in xApps**
4. **Sketches**
5. **Flexibility of the functionalities**
6. **Sketch-based features and use of «super features»**
7. **Reference geometry**
8. **Applied features**
9. **Rehearsals**
10. **Addition of holes**
11. **Use of equations**
12. **Multiple body parts**
13. **Direct editing**
14. **Design changes**
15. **Design Guide**
16. **Design of assemblies**
17. **2D sketch movement**
18. **Component repeats**
19. **External references**
20. **Derived formats**
21. **xCAD PLM connector**

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SOLIDWORKS Simulation Training

Course Outline

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TABLE OF CONTENTS - SOLIDWORKS SIMULATION

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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

- Objectives
- Mesh control
- Project description
- Understanding the effect of boundary conditions

3. Assembly Analysis with Interactions

- Interaction analysis
- Contact or bonded interaction
- Pliers with 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
- 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

- Multiple load cases
- Geometry modification

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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 non-linear analysis

Annex

- Meshing Strategy
- Geometry Preparation
- Meshing Quality
- Meshing Parameters
- Meshing Steps
- Failure Diagnosis
- Tips for the Shell Elements Usage
- Requirements for Meshing
- Solvers in SOLIDWORKS Simulation
- Solver Selection
- Help and Customer Support

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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)
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 contact conditions
- Bonded and free interactions

3. Buckling Analysis

- Buckling analysis

4. Load Cases

- Load Cases

5. Submodeling

- Submodeling

6. Topology Analysis

- 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

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

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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 analyses

14. Pressure Vessel Analysis

- Pressure vessel analysis
- Manhole nozzle flange and cover

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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 – von Mises
- Nonlinear – Tresca's
- Stress accuracy
- Using Nonlinear Elastic Material

5. Hardening Rules

- Hardening rules
- Isotropic hardening
- Kinematic hardening

6. Analyse de topologie

- 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

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SOLIDWORKS Simulation Premium Composite – 1 day (7h)

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**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
- Summary
- Reference

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SOLIDWORKS Flow Simulation – 2 days (14h)

1. **Creating a SOLIDWORKS Flow Simulation Project**

   - Model preparation
   - Postprocessing

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

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

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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

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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
- Project Description
- Conclusions

SOLIDWORKS Plastics – 1.5 day (10h), 2 day (14h) or 3 day (21h)

1. Basic Flow Analysis

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

2. Detecting Short Shots

- 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
- Predict fill pattern
- Injection location advisor

5. Materials

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

6. Mesh Manipulation

- Mesh Manipulation
- Local Refinement of Mesh
- Edit/Review
- Element Issues
- Leader Lines
- 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
- CoolingTimes

see Part 2 on next page »

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SOLIDWORKS Plastics *(Part 2)*

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
- Cyclic symmetry
- Cyclic

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

15. Multi Material Overmolding

- Multi-material overmolding
- Assigning injection units

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

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SOLIDWORKS PDM
Training

Course Outline

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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENTS - SOLIDWORKS PDM

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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
- Bills of Material Columns

7. Workflow

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

8. Notifications and Tasks

- Notifications
- Conditional Notifications
- Message System
- 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 :

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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

- Migrating Legacy Data
- Data Migration
- Migrating Revisions

Appendix

- File Types and Settings
- Toolbox Setup

Not included :

- Installation Process
- Routing Setup
- CircuitWorks Setup

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 Bills of Material (BOM)

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Update SOLIDWORKS PDM Standard to Professional – 1 day (7h)

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

12. 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»

8. Working in SOLIDWORKS

- Add-in
- Marking with eDrawings Professional

Appendix

- Use of named nomenclatures

Not included :

- Installation process
- Routing Setup and CircuitWorks Setup

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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

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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 Schemes

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 Material
- Creating Bills of Material

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

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SOLIDWORKS

Automating
Training

Course Outline

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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENTS - AUTOMATING

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.
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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

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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

13. **Lesson 14 (Advanced)**

    • Specification Control

13. **Lesson 15 (Advanced)**

    • Linking to Data

13. **Lesson 16 (Advanced)**

    • Rollup Data Tables

13. **Lesson 17 (Advanced)**

    • Hierarchical properties

13. **Lesson 18 (Advanced)**

    • Macro Buttons

13. **Lesson 19 (Advanced)**

    • Generation Tasks

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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

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

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 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 Configuration
- 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 OP0
- 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

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SWOOD Advanced – 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

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![SOLIDWORKS logo]

SOLIDWORKS Electrical Training

Course Outline

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ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENTS - SOLIDWORKS ELECTRICAL

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SOLIDWORKS Electrical: Schematic – 3 days (21h)

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 attributes
- Splitting attibute data
- Add to library
- Copy, paste symbol

9. Macros

- What are macros?
- Creating and adding macros

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?

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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
- Set Origin Destination of Cables by Location

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 Add-Ins Training

Course Outline

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENTS - SOLIDWORKS ADD-INS

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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 Bills of Material

- Bills of Material
- 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

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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

- 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 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

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 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

SOLIDWORKS Inspection – 1 day (7h)

This training's duration is about half a day but we will complete it with personnalized 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

- 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

7. Appendix C: Glossary of Quality Terms

- Terminology

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

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 TolAnalyst - 1 day (7h)
This training's duration is about half a day but we will complete it with personnalized coaching on your models or examples files.
**No books currently exist for the TolAnalyst course.*

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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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

SOLIDWORKS CAM Professional – 1 day (7h)

*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 personnalized 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
- 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
- Modifying Feature and Operation Parameters
- Editing toolpaths

7. Probing

- Introduction to Probing

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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
    - La 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 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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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 Commands

- 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. Using Schematic Annotations

- 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. Advanced Operations

- Advanced Manipulations
- Thicken
- Symmetry

SOLIDWORKS

3D Printing Training

Course Outline

SOLIDXPERTS
by solidxperience

ENSURE YOUR SUCCESS IN 3D DESIGN WITH SOLIDWORKS

TABLE OF CONTENTS - 3D PRINTING

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