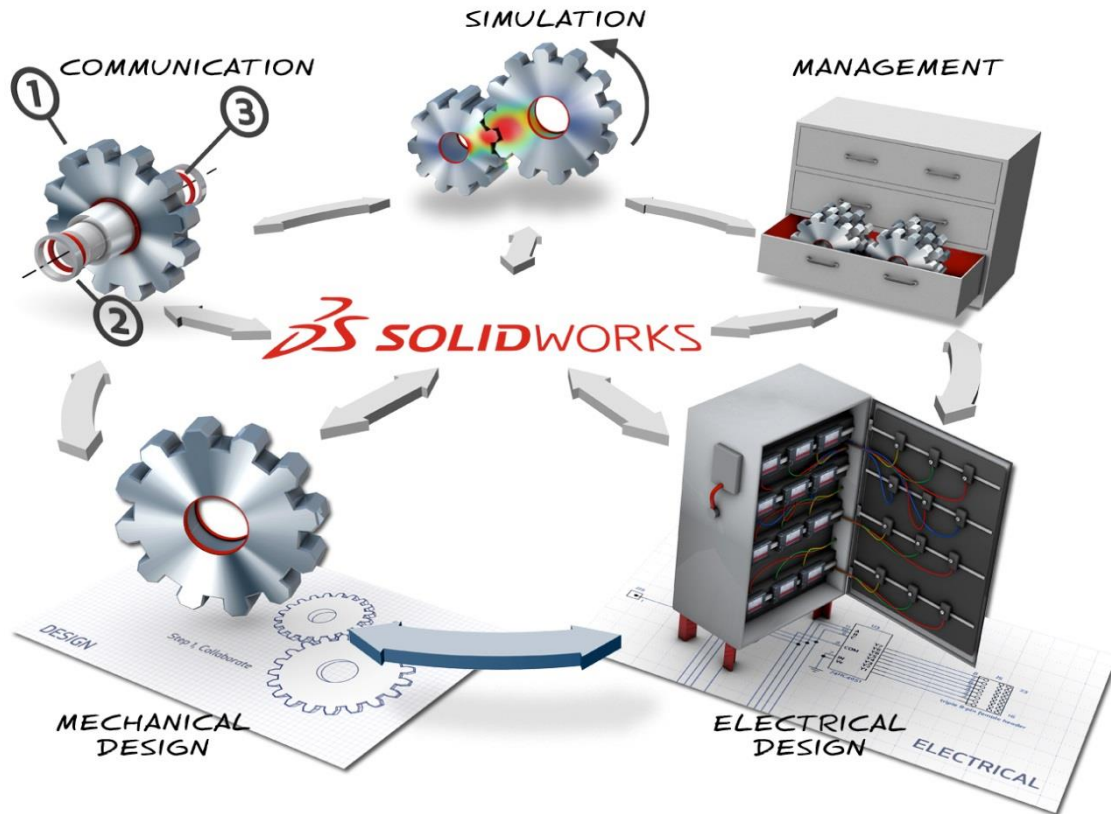


TRAINING COURSE PROSPECTUS





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Introduction

As a SOLID Applications Ltd customer, your design system can and will make a significant contribution towards increased efficiency and productivity. Effective training of all personnel involved with the system is a key factor in obtaining the maximum return on your investment. By continuing to invest in high quality, relevant training for your personnel you will:

- Ensure that your organisation gains the optimum benefit from your system
- Maximise users personal productivity through in-depth knowledge of system functionality
- Enhance your business operation through innovative and effective system usage
- Identify scope for system applications that may not have been part of the initial justification for your investment
- Deliver personal satisfaction and motivation to system users through increased skills and expertise.

It is our aim at SOLID Applications Ltd to assist you in achieving these goals through the provision of purpose-developed, specialised training courses designed to enhance the skill and expertise of your system users and to maximise the benefit you gain from your investment in a SOLID Applications Ltd solution. All SOLID Applications Ltd training courses are structured so that your staff may learn not only the theoretical but also the practical aspects of system usage.

SOLID Applications Ltd is able to provide training of the necessary high quality because:

- We listen to the needs of our customers and gain an understanding of your business and training needs so that training courses are relevant and develop effective skills
- We are able to respond to changing market demands
- Our instructors are highly professional, experienced tutors who are fully aware of product capabilities and how these may best be applied to meet your business needs
- Our training courses are designed to require total delegate participation in a distraction free training environment, conducive to effective learning.

Should you need further details about any of the courses outlined in this document, require assistance identifying the most appropriate courses to meet your training needs, or need any help in developing your training programme please contact us.

Training Details

Courses are scheduled on demand and the frequency of any course will reflect that demand. Therefore, although a course you desire may not be included in the current schedule, please do call and enquire about the course. If sufficient demand is shown then a course will be scheduled.

Training courses are to pre-set formats and each is designed to train delegates in specific aspects of the product as outlined in this prospectus.

Please check the pre-requisites of the course before you enrol delegates. Although specific SOLID Applications Ltd courses may be stated as having pre-requisites, delegates may have attained the required background via other channels. It is the delegates' responsibility to ensure that they have the required knowledge for a course before enrolling. If you have any queries regarding course pre-requisites we will be pleased to advise you.

Training at our site:

- All courses are conducted at SOLID Applications Ltd's fully equipped training rooms with dedicated facilities.
- On the first day of the course, delegates should report to the reception area for registration at 9.15am. Delegates are requested to arrive promptly and a map will be sent showing the location of our Oldbury Training Centre when course bookings are confirmed.
- Classes are conducted from 9.30am to 5.00pm each day with a break of approximately one hour for lunch. Lunch is provided each day of the training.
- Delegates are responsible for all personal expenses such as travel, accommodation and evening meals. Lunch will be provided each day. Details of local accommodation will be provided with confirmation of your booking.
- Courses include classroom lecture, demonstration and supervised "hands on" exercises.
- Training course manuals will be provided per workstation/laptop for use whilst on the course.

Training at your site:

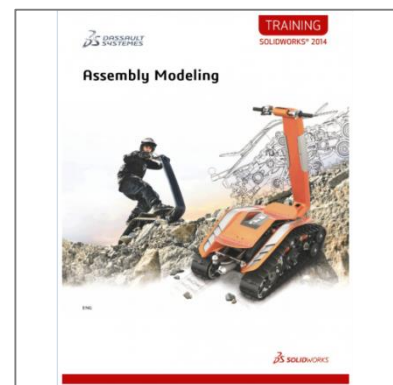
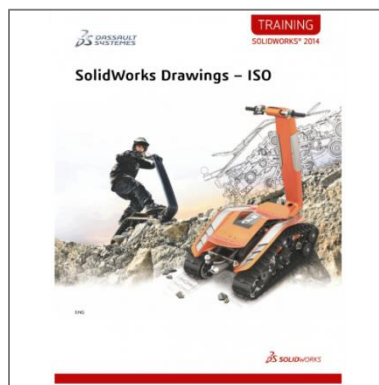
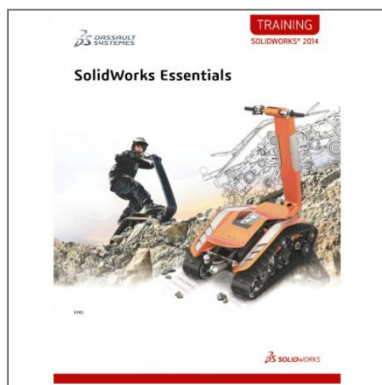
- The customer is responsible for providing a suitable training room, white-board, and projector.
- Unless specified, the customer is responsible for providing a suitable training workstations/laptops with the appropriate software installed and licensed.
- If SOLID Applications Ltd provides training workstations/laptops, the customer is responsible for securely storing them during non-training periods such as breaks or overnight, with the appropriate insurance in place.
- Classes can be conducted to start/end based on the customer's normal working hours each day, with a break of approximately one hour for lunch (subject to confirmation).
- It is assumed that the customer will provide lunches each day of the training.
- For standard courses, manuals will be provided per workstation/laptop for use whilst on the course.

Training Course Costs

The training rate for all training held at the SOLID Applications Ltd training centre in Birmingham is £375 per person, per day. This excludes any training offers or promotions, or special rates negotiated on a per client basis.

The training rate for all training held at the client's site is £1200 per day (excluding expenses).

Standard training courses at our offices include training manuals for the delegates, however, for client site training, the training manuals will be need to be added to the cost of the course.



All prices stated are subject to VAT.

Training Grants

SOLID Applications Ltd works with an impartial training broker who liaise closely with Train to Gain, Skills Funding Agency, European Social Funding, Manufacturing Advisory Services, Individual Regional Grant Funds and a variety of individual awarding bodies to distribute relevant training grants within ALL sectors nationally.

An example of some of the sectors that have had specialist grants are the Engineering, Manufacturing, Energy Related, Hi-tech, Automotive, Design, Architectural, Creative, Pharmaceutical and Construction industries and sectors (this list is not restrictive or exhaustive).

These Grants can really help elevate your business, staff and products as well as safe guard jobs and boost your company's image and output.

Grants change very frequently, and can be for Match funding, Training for Leaders, Managers and Key/Principle decision makers, and also Business Critical Training. The funding is dependent on the region of the UK you are in; If you would like to check if you are eligible for any funding, please contact us. Please be aware this is a FREE NO OBLIGATION SERVICE to check eligibility, and in most cases the application process is also free or on a no win no fee basis.

SOLIDWORKS Courses

SOLIDWORKS Transition

Code:	SA-SWT
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	Solid Edge experience
Description:	The aim of the course is to seamlessly transition existing Solid Edge users to SOLIDWORKS, ensuring minimal disruption to current working practices. This course will train the users the most commonly used tools to create some basic part components, to be then used in the assembly environment to create a finished product. This product will then be taken into the drafting environment to produce a detailed production drawing.

The topics covered in this course are:

- Generic User Interface
- Part Modelling
- Assembly modelling
- Drafting

Lesson 1: SOLIDWORKS user interface

- Overview of the user interface
- Opening files
- Part and Sheet Metal environment
- Basic settings
- Options to speed up your workflows

Lesson 2: Introduction to Part Modelling

Planes

- Manipulating FeatureManager Design Tree to access reference planes
- Use of design panes

Sketching

- Variants to create a sketch
- Solid Edge similarity settings
- Centre line and construction lines
- Trim command
- Constraining sketches
- Aligning to origin/planes
- Sketch dimensioning
- Sketch relationships

Part Modelling features

- Solid from existing sketch
- Solid from feature selection
- Solid from instant 3D
- Editing sketches
- Instant 3D edit
- Solid from open profiles

- Hole command
- Fillet command and fillet expert
- Draft command
- Revolve base /cut

Sheet Metal Modelling features

- Tabs
- Flanges
- Holes and Cutouts
- Flat pattern
- Convert to sheet metal

Lesson 3: SOLIDWORKS basic Assembly Modelling

- Insert components
- Mates/smart mates
- Movement
- Planar align with angle

Lesson 4: SOLIDWORKS Drawings

- Importing items – from environment or folder structure
- Creating views
- Sectioning and detail views
- Dimensioning
- Parts list and ballooning

Lesson 5: Data Migration

- Feature Recogniser
- Strategy for migrations

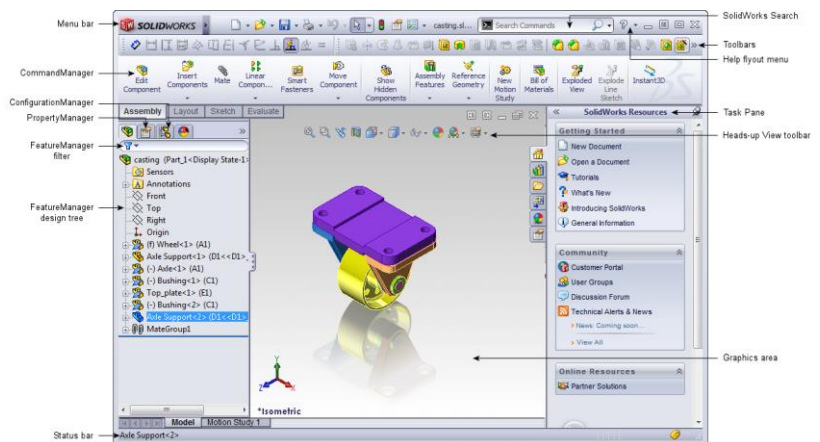
SOLIDWORKS Essentials

Code:	SA-TR40
Duration:	4 Days
Cost:	£1500 per person (includes course manual)
Prerequisites:	Mechanical design experience; experience with the Windows™ operating system
Description:	SOLIDWORKS Essentials teaches you how to use the SOLIDWORKS mechanical design automation software to build parametric models of parts and assemblies, and how to make drawings of those parts and assemblies.

The topics covered in this course are:

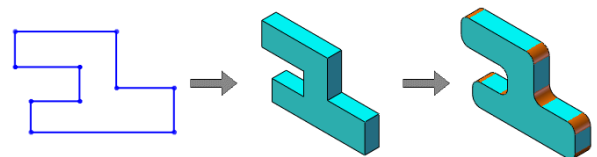
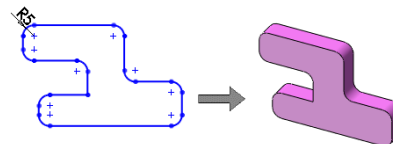
Lesson 1: SOLIDWORKS Basics and the User Interface

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



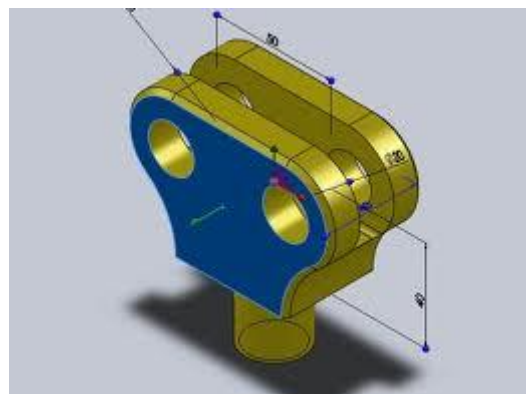
Lesson 2: Introduction to Sketching

- 2D Sketching
- Stages in the Process
- Saving Files
- What are We Going to Sketch?
- Sketching
- Sketch Entities
- Basic Sketching
- Rules That Govern Sketches
- Design Intent
- Sketch Relations
- Dimensions
- Extrude
- Sketching Guidelines



Lesson 3: Basic Part Modelling

- Basic Modelling
- Terminology
- Choosing the Best Profile
- Choosing the Sketch Plane
- Details of the Part
- Boss Feature
- Sketching on a Planar Face
- Cut Feature
- Using the Hole Wizard
- View Options
- Filleting
- Editing Tools
- Detailing Basics
- Drawing Views
- Center Marks
- Dimensioning
- Changing Parameters



Lesson 4: Symmetry and Draft

- Case Study: Ratchet
- 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

Lesson 5: Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed Only
- Up to Reference
- Sketch Driven Patterns

Lesson 6: Revolved Features

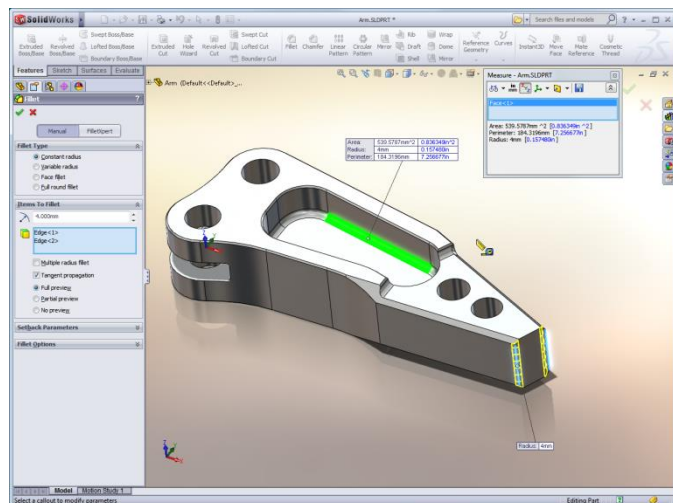
- Case Study: Handwheel
- Design Intent
- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- File Properties
- SOLIDWORKS SimulationXpress
- Using SOLIDWORKS SimulationXpress
- The SimulationXpress Interface

Lesson 7: Shelling and Ribs

- Shelling and Ribs
- Analyzing and Adding Draft
- Other Options for Draft
- Shelling
- Ribs
- Full Round Fillets
- Thin Features

Lesson 8: Editing: Repairs

- Part Editing
- Editing Topics
- Sketch Issues
- FilletXpert
- DraftXpert

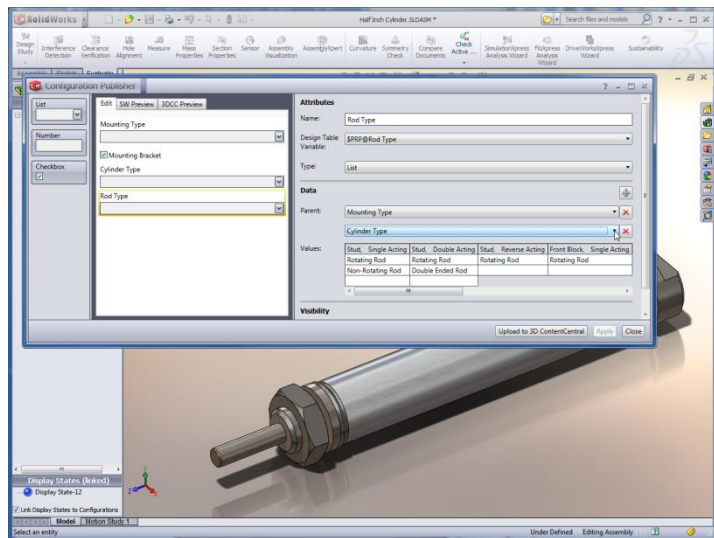


Lesson 9: Editing: Design Changes

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

Lesson 10: Configurations

- Configurations
- Using Configure Dimension/Feature
- Other Methods to Create Configurations
- Using Global Variables and Equations
- Creating Equalities
- Global Variables
- Defining the Overall Width
- Creating a Minimum Edge Distance
- Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library

**Lesson 11: Using Drawings**

- More About Making Drawings
- Section View
- Model Views
- Broken View
- Detail Views
- Drawing Sheets and Sheet Formats
- Projected Views
- Annotations

Lesson 12: Bottom-Up Assembly Modelling

- Case Study: Universal Joint
- 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
- Sub-assemblies
- Smart Mates
- Inserting Subassemblies
- Pack and Go

Lesson 13: Using Assemblies

- Using Assemblies
- Analyzing the Assembly
- Checking for Clearances
- Changing the Values of Dimensions
- Exploded Assemblies
- Explode Line Sketch

- Bill of Materials
- Assembly Drawings

Appendix A: Templates

- Options Settings
- Document Templates

SOLIDWORKS New Version Update Training

Code:	SA-SWU14
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	SOLIDWORKS Experience.

SOLIDWORKS 2016

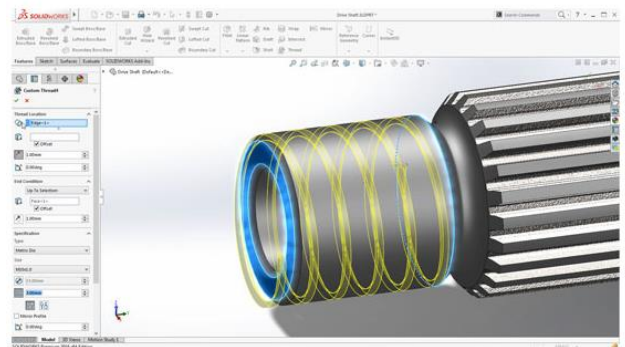
Make Great Design Happen

**DESIGN****FOCUS ON DESIGN, NOT SOFTWARE**

Work smarter and get the CAD system out of your way with fewer “picks and clicks”, increased modeling flexibility, a more intuitive interface, and easier access to commands

Topic covered include:

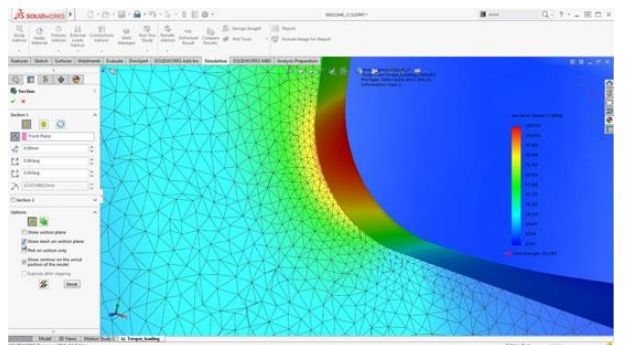
- User Interface
 - Bread crumbs
 - Confirmation corner shortcut
- Sketching
- Part Modeling
 - 3D Thread Feature
- Assembly Modeling
 - Simplified mate selection

**VALIDATE****SOLVE COMPLEX PROBLEMS QUICKLY**

Innovative design simulation makes analysis more efficient to help you solve complex problems, visualize and verify functionality, and find potential errors before they occur

Topic covered include:

- Complex Part Design
- Performance Validation
- Plastics Validation
- Flow Simulation



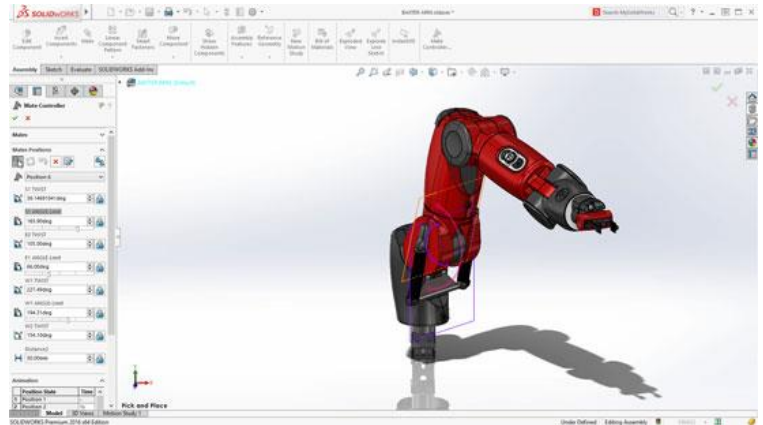
COLLABORATE

STREAMLINE YOUR PARALLEL DESIGN PROCESS

Communicate, collaborate, and work concurrently across teams, disciplines, customers, and vendors with mechatronic design, concurrent design, and streamlined electrical/mechanical design

Topic covered include:

- SOLIDWORKS PDM Standard & Professional
- SOLIDWORKS Electrical
- Routing
- Mate Controller
- Costing
- eDrawings
- Treehouse



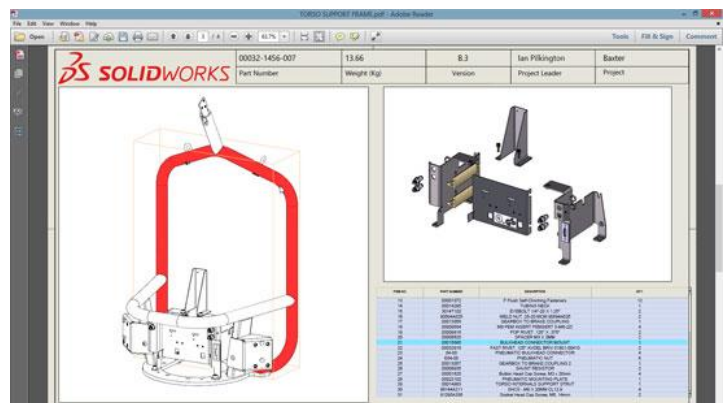
BUILD

FAST-TRACK YOUR DESIGNS THROUGH MANUFACTURING

Create more detailed outputs for manufacturing and shorten product development to manufacture while you save time and reduce errors

Topic covered include:

- PhotoView 360
- SOLIDWORKS Visualize
- 3D Printing
- Drawings
- Model Based Definition



SOLIDWORKS Drawings

Code:	SA-TR41
Duration:	2 Days
Cost:	£750 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials
Description:	This course teaches you how to make drawings of SOLIDWORKS parts and assemblies.

The topics covered in this course are:

Lesson 1: Drawing Sheets and Views

- Drawing Sheets and Views
- Sheets and Formats
- Terminology
- Drawing Views
- Sketching in Drawing Views
- View Settings
- Centermarks and Centerlines
- Model Edges in the View

Lesson 2: Dimensions

- Dimensions
- Manipulating Dimensions
- Dimension Properties

Lesson 3: Annotations

- Adding Annotations
- Annotation Types
- Blocks

Lesson 4: Assembly Drawing Views

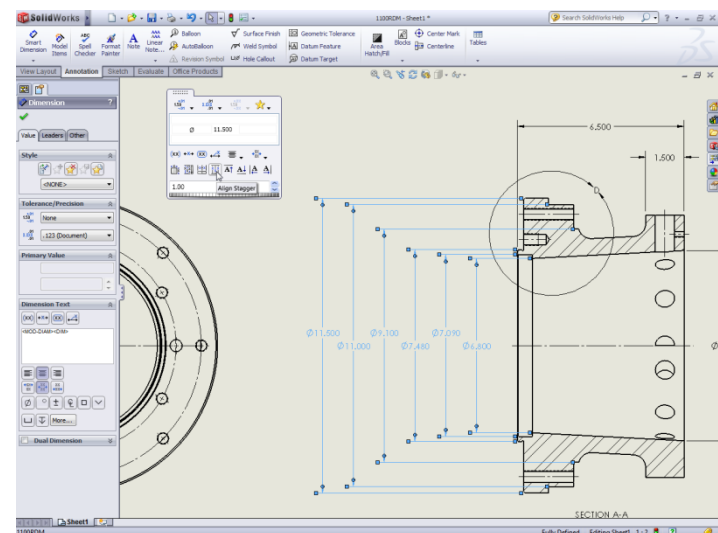
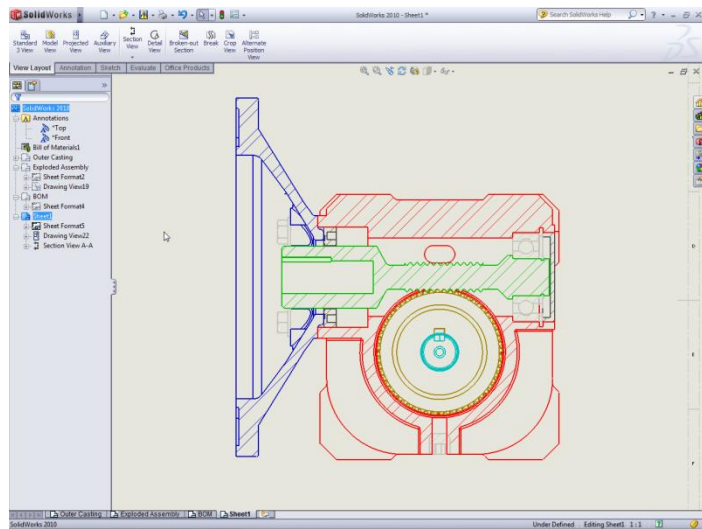
- Assembly Drawing Views
- Creating Views of Assemblies

Lesson 5: Sheet Formats and Templates

- Sheet Formats and Templates
- Drawing Templates
- Properties in the Template
- User Defined Properties
- Customizing a Sheet Format
- Define Title Block
- Updating Sheet Formats

Lesson 6: Bill of Materials and Tables

- Creating and Managing a Bill of Materials
- The Bill of Materials
- Adding a BOM
- Modifying the BOM
- Tabulated Bill of Materials
- Balloons
- Tables in the Drawing



Lesson 7: Performance and Display Issues

- Performance and Display Issues
- Large Assembly Mode
- Lightweight Drawings
- Detached Drawings
- Display Issues in Drawing Views

Lesson 8: Drawing References and Comparison

- Reusing a Drawing File
- Changing Drawing References
- Using DrawCompare
- SOLIDWORKS Design Checker

Appendix A: Preparations for Detailing

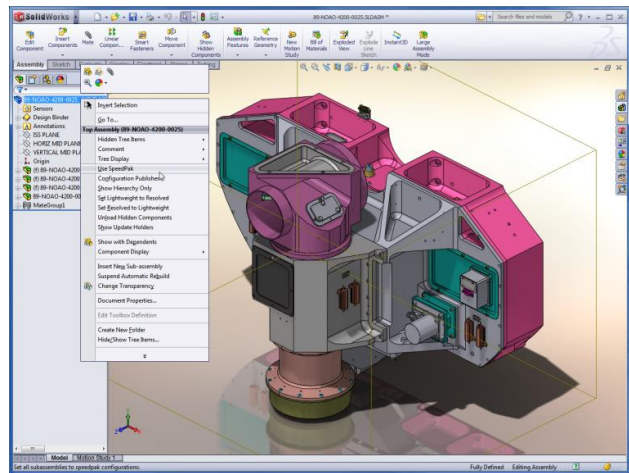
SOLIDWORKS Assembly Modelling

Code:	SA-TR42
Duration:	2 Days
Cost:	£750 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials
Description:	Assembly modelling teaches you how to maximize your use of the assembly modelling capabilities of SOLIDWORKS mechanical design automation software.

The topics covered in this course are:

Lesson 1: Advanced Mate Techniques

- SOLIDWORKS Assemblies
- Assembly File Structure
- File References
- External Reference Search Order
- Solving Mates
- Advanced Mate Techniques
- Case Study: SmartMates and Mate References
- Mate References
- Capture Mate References
- Multiple Mate Mode
- Using Copy with Mates
- Copy with Mate Options
- Fixed Components
- Summary: Inserting and Mating Components
- Advanced Mate Features
- Profile Center Mate
- Rack Pinion Mate



Lesson 2: Top-Down Assembly Modeling

- Top-Down Assembly Modeling
- Stages in the Process
- Making Changes to Dimensions
- Case Study: Editing and Building In-context
- 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
- Machine_Vise Design Intent
- Removing External References

Lesson 3: Assembly Features, Smart Fasteners, and Smart Components

- Assembly Features and Smart Fasteners
- Assembly Features
- Case Study: Assembly Features
- Smart Fasteners

- Smart Components

Lesson 4: Assembly Editing

- Assembly Editing
- Key Topics
- Editing Activities
- Case Study: Assembly Editing
- Replacing and Modifying Components
- Troubleshooting an Assembly
- Replacing Components Using Save As
- Reloading Components
- Component Patterns

Lesson 5: Using Configurations with Assemblies

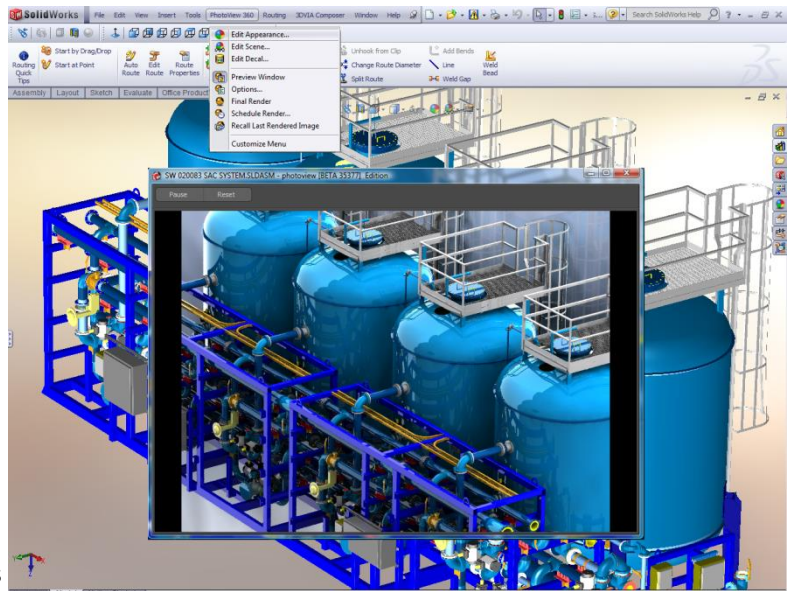
- Using Configurations with Assemblies
- Case Study: Assembly Configurations
- Creating Configurations Manually
- Configuration Properties
- Using the Modify Configurations Dialog
- Changing Configurations using the Context Toolbar
- Assembly Evaluation Tools
- Case Study: Hole Alignment
- Controlling Dimensions in an Assembly
- Creating an Equality
- Equations With Functions
- Comments
- Sensors

Lesson 6: Display States and Appearances

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

Lesson 7: Large Assemblies

- Large Assemblies
- Key Topics
- Lightweight Components
- Large Assembly Mode
- Case Study: Large Assembly Options
- Using SpeedPak
- Using Configurations with Large Assemblies
- Defeature
- Assembly Visualization
- Large Design Review
- Tips for Faster Assemblies
- Drawing Considerations



Lesson 8: Using SOLIDWORKS Treehouse

- SOLIDWORKS Treehouse
- Setting Treehouse Instances
- Exporting Treehouse Data

Lesson 9: Layout-based Assembly Design

- Key Topics
- Case Study: Clamp
- Blocks
- Inserting Blocks
- Creating a Part from a Block
- Gear and Pulley Motion in Blocks
- Case Study: Gears and Pulleys

SOLIDWORKS Advanced Part Modelling

Code:	SA-TR43
Duration:	3 Days
Cost:	£1125 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials
Description:	Advanced Part Modelling teaches you how to use multibody solids, sweeping and lofting features, and the more advanced shaping capabilities of SOLIDWORKS.

The topics covered in this course are:

Lesson 1: Multibody Solids: How They Work

- Multibody Solids
- Multibody Design Techniques
- Case Study: Multibody Design
- Contour Selection
- Introducing: Solid Bodies Folder
- Feature Scope
- Patterning Bodies
- Tool Body Technique
- Introducing: Insert Part
- Introducing: Move/Copy Bodies
- Combining Bodies
- Introducing: Combine
- Common Bodies
- Intersect with Solid Bodies
- Introducing: Intersect
- Local Operations
- Indent Feature
- Introducing: Indent
- Deleting Solid Bodies
- Introducing: Delete/Keep Body

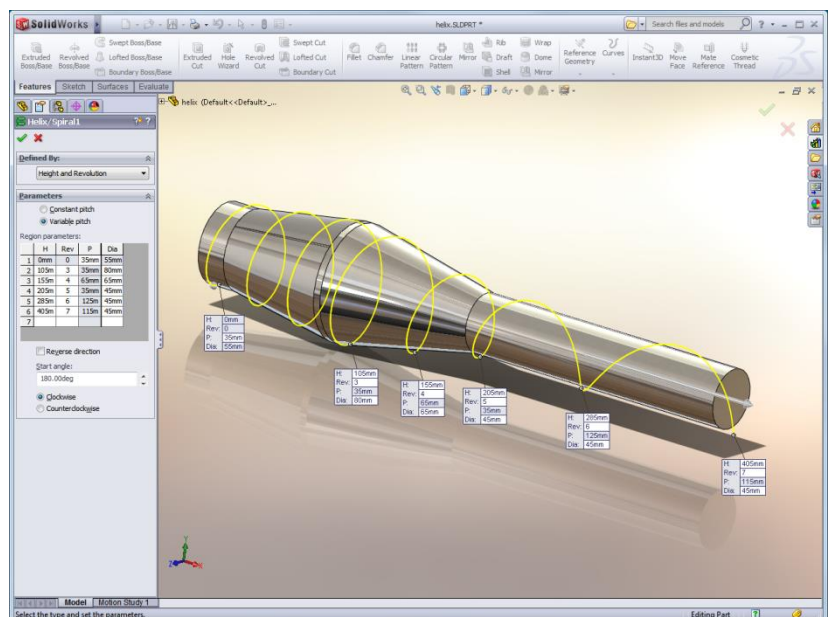


Lesson 2: Saving Solid Bodies

- Multibody Part vs. Assembly
- Saving Bodies Functions
- Using Cut to Create Multiple Bodies
- Insert into New Part
- Introducing: Insert into New Part
- Saving Bodies
- Introducing: Save Bodies
- Modeling for Rapid Tooling
- Splitting a Part into Multiple Bodies
- Introducing: Split
- Creating an Assembly
- Introducing: Create Assembly
- Using Split Part with Legacy Data

Lesson 3: Sketching with Splines

- Curves in Sketches
- Splines



- Introducing: Spline
- Introducing: Spline Tools
- Evaluating Curvature Combs
- Introducing: The Control Polygon
- Introducing: Spline Handles
- Analyzing Solid Geometry
- Introducing: Display Curvature
- Introducing: Zebra Stripes
- Surface Curvature Combs
- Sketch Picture
- Introducing: Sketch Picture
- Style Spline
- Introducing: Style Spline
- Case Study: Watering Can Handle
- Style Spline Tools
- Fit Spline
- Introducing: Fit Spline

Lesson 4: Introduction to Sweeping

- Sweeping
- Case Study: Faux Raised Panel Door
- Sweep with Guide Curves
- Case Study: Bottle Body
- Pierce Relation
- Sweep with Guide Curves
- The SelectionManager
- Introducing: SelectionManager
- Case Study: Hanger Bracket
- Symmetrical Splines
- Introducing: Dome

Lesson 5: Working with Curves

- Curve Features
- Case Study: Modeling a Spring
- Sweeping Along a 3D Path
- 3D Sketching
- Helix Curve
- Introducing: Helix and Spiral
- Introducing: Projected Curve
- Composite Curve
- Introducing: Composite Curve
- Adding Features to the Bottle
- Performance Considerations
- Modeling Threads
- Thread Profile
- Blower Housing
- Sweeping Along Model Edges
- Introducing: Split Line

Lesson 6: Advanced Sweeping

- Sweep Options
- Additional Sweep Settings

- Orientation and Twist Control
- Visualizing Sweep Sections
- Introducing: Face Curves
- Align with End Faces
- Solid Sweep
- Introducing: Equation Driven Curve

Lesson 7: Boundary Feature and Lofting

- Comparing Complex Features
- How Lofting and Boundary Work
- Loft vs Boundary Feature
- Introducing: Loft
- Boundary Feature
- Introducing: Boundary Feature
- Using Derived and Copied Sketches
- Copying a Sketch
- Modify Sketch
- Introducing: Modify Sketch
- Derived Sketches
- Introducing: Derived Sketch
- Boundary Preview Options
- Sketch Block and Library Feature Profiles
- Additional Curves in Loft and Boundary
- Centerline Lofting
- Loft Preview Options
- Adding Sketch Segments
- Introducing: Segment
- Introducing: Split Entities
- Cleaning Up a Model
- Deleting Faces
- Introducing: Delete Face
- Evaluating Edges
- Introducing: Deviation Analysis
- Lofting with Guide Curves
- Curve Influence
- Hook Using Boundary
- Conics
- Introducing: Conic
- Introducing: Curve Through XYZ
- Sketch Blocks
- Introducing: Sketch Blocks

Lesson 8: Advanced Filleting and Other Features

- Fillet Settings
- Fillet Parameters
- Constant Size Fillets
- Delete Face: Delete and Fill
- Fillet Options
- Variable Size Fillets
- Face Fillets
- Other Advanced Features
- Wrap Feature
- Introducing: Wrap

- Deform Feature
- Introducing: Deform
- Introducing: Knit Surface
- Direct Editing
- Introducing: Move Face

SOLIDWORKS Surface Modelling

Code:	SA-TR44
Duration:	2 Days
Cost:	£750 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials, Advanced Part Modelling
Description:	Surface Modelling teaches you how to build freeform shapes using SOLIDWORKS mechanical design automation software.

The topics covered in this course are:

Lesson 1: Understanding Surfaces

- Solids and Surfaces
- Working with Surface Bodies
- Why Use Surfaces?
- Continuity Explained
- Workflow with Surfaces

Lesson 2: Introduction to Surfacing

- Similarities Between Solid and Surface Modelling
- Basic Surfacing

Lesson 3: Solid-Surface Hybrid Modelling

- Hybrid Modelling
- Using Surfaces to Modify Solids
- Interchanging Between Solids and Surfaces
- Performance Implications
- Surfaces as Construction Geometry
- Alternative to Trim, Knit, and Thicken
- Making Copies of Faces

Lesson 4: Repairing and Editing Imported Geometry

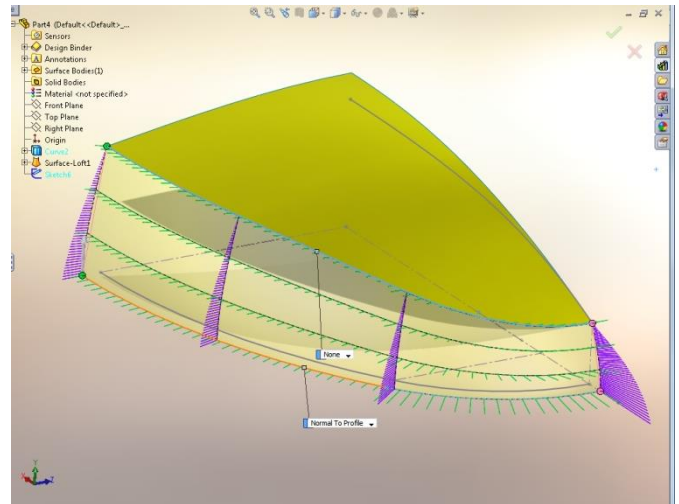
- Importing Data
- Repairing and Editing Imported Geometry

Lesson 5: Advanced Surface Modelling

- Stages in the Process
- Ruled Surfaces
- Lofting Surfaces
- Modeling the Lower Half
- Conclusion
- Design Changes

Lesson 6: Blends and Patches

- Complex Blends
- Smoothing Patches
- Boundary Surface
- Freeform Feature
- Corner Blends



Lesson 7: Master Model Techniques

- Introduction to Master Models
- Surface Master Model Technique
- Working with a Solid Master Model
- Specialized Features for Plastic Parts
- SOLIDWORKS Explorer

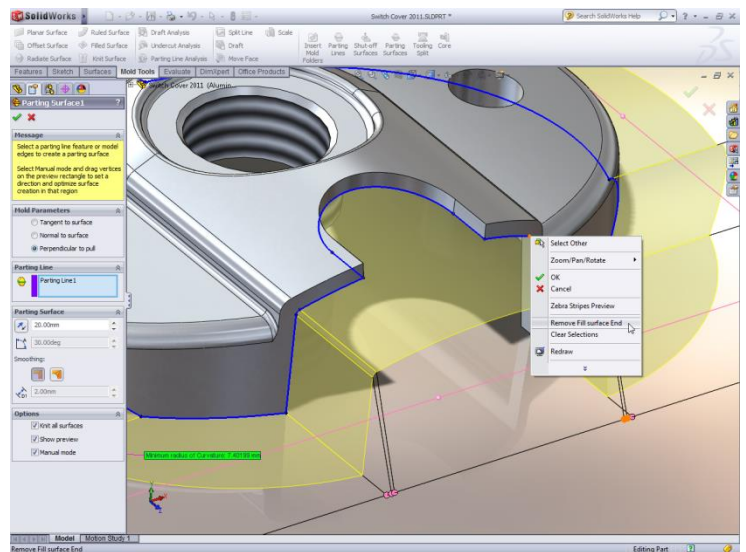
SOLIDWORKS Mould Design

Code:	SA-TR45
Duration:	2 Days
Cost:	£750 per person (includes course manual)
Prerequisites:	Advanced Part Modelling
Description:	Mould Design Using SOLIDWORKS teaches you several manual mould creation techniques and how to use the Mould Tools in SOLIDWORKS mechanical design automation software.

The topics covered in this course are:

Lesson 1: Surface Concepts and Imported Geometry

- Importing Data
- 3D Model Types
- Definitions
- Case Study: Solids vs. Surfaces
- Terminology
- File Translators
- Modeling Systems
- File Translation
- Why Do Imports Fail?
- Diagnosis and Repair
- Case Study: Repairing and Editing Imported Geometry
- Checking Solid Bodies
- Making Copies of Faces
- Case Study: Import Diagnostics
- Repairing Gaps
- Repairing Faces



Lesson 2: Core and Cavity

- Core and Cavity Mold Design
- Case Study: A Simple Two Plate Mold Design
- SOLIDWORKS Mold Tools
- Mold Analysis Tools
- Analyzing Draft on a Model
- Draft Analysis Colors
- Scale the Part to Allow for Shrinkage
- Establish the Parting Lines
- Manual Selection Of Parting Lines
- Automation
- Modeling the Parting Surfaces
- Smoothing the Parting Surface
- Surface Bodies
- Interlocking the Mold Tooling
- Creating the Mold Tooling

Lesson 3: Side Cores and EDM Design

- Multiple Parting Directions

- Trapped Molding Areas
- Side Cores
- Feature Freeze
- Lifters
- Core Pins
- Case Study: Electrode Design
- Electrode Clearances
- Keeping the Sharp Edges

Lesson 4: Advanced Parting Lines, Shut-Off Surfaces, and Cores

- Parting Lines and Shut-Off Surfaces
- Draft Analysis Options
- Parting Line Options
- Core and Cavity Surfaces
- Shut-Off Surfaces
- Parting Surface
- Tooling Split
- Seeing Inside the Mould

Lesson 5: Using Surfaces for Model Prep and Interlocks

- Surfaces in Mold Making
- Case Study: Cordless Drill Bezel
- Creating New Drafted Faces
- Interlock Surfaces

Lesson 6: Using Surfaces for Mold Design Features

- Surfaces in Mould Making
- Case Study: Router Bottom
- The Mixer
- Case Study: Mixer Upper Half
- Case Study: Manual Parting Surface
- Mold Split Folders

Lesson 7: Alternative Methods for Mold Design

- Alternate Methods for Mold Design
- Using Combine and Split
- Creating a Cavity
- Case Study: Cavity
- Using Surfaces
- Techniques for Mold Tooling
- Using the Up To Surface Method
- Using the Split Method
- Manually Creating Shut-off Surfaces

Lesson 8: Reusable Data

- Reusing Data
- Task Pane
- SOLIDWORKS Resources
- Design Library
- File Explorer
- Case Study: 3D ContentCentral
- Library Features

-
- Case Study: Create A Library Feature
 - Configurations in Library Features
 - Case Study: Water Line
 - Smart Components

Lesson 9: Completing the Mold Base

- Case Study: Mold Base
- Organizing the Assembly
- Modifying the Lifters
- Ejector Pins
- Cooling the Mold
- Making the Drawing
- Making Changes
- Completing the Process

SOLIDWORKS Sheet Metal

Code: SA-TR46

Duration: 2 Days

Cost: £750 per person (includes course manual)

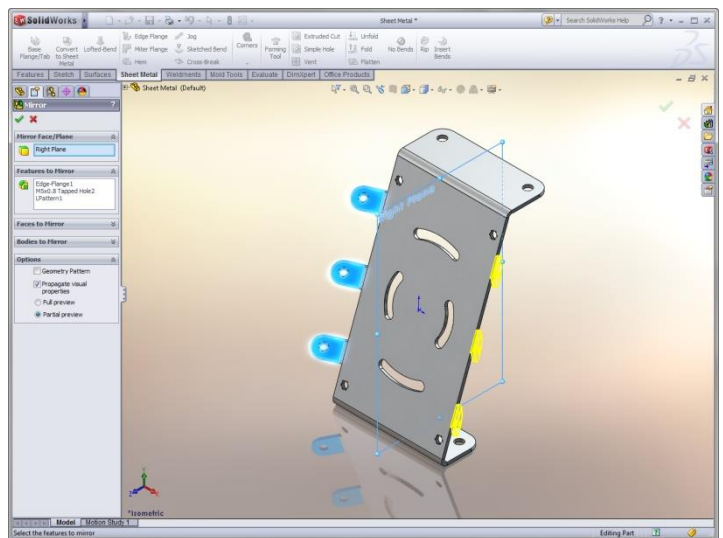
Prerequisites: SOLIDWORKS Essentials

Description: Sheet Metal teaches you how to build sheet metal parts using SOLIDWORKS mechanical design automation software. Building standalone sheet metal parts, and converting conventional parts to sheet metal, including in assembly context, are covered.

The topics covered in this course are:

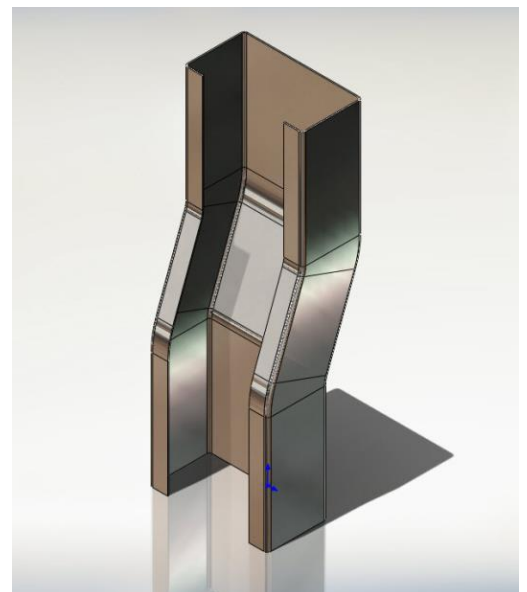
Lesson 1: Basic Flange Features

- What are Sheet Metal Parts?
- Sheet Metal Methods
- Unique Sheet Metal Items
- Flange Method
- Edge FlangesBase 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 Feature
- Tab Features
- Cuts in Sheet Metal



Lesson 2: Working with the Flat Pattern

- Working with the Flat Pattern
- Flat Pattern Settings
- Features for Manufacture
- Corner-Trim Feature
- Corners in the Formed State
- Closed Corner
- Corner Relief
- Break Corner/Corner Trim
- Producing the Flat Pattern
- Sheet Metal Cut List Properties
- Flat Pattern Drawing Views
- Flat Pattern View Properties
- Drawing Document Properties
- Sheet Metal Tables
- Cut List Properties as a Note



Lesson 3: Additional Sheet Metal Techniques

- Additional Sheet Metal Methods
- Designing from the Flat

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

Lesson 4: Converting to Sheet Metal

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

Lesson 5: Multibody Sheet Metal Parts

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

Lesson 6: Forming Tools and Gussets

- Sheet Metal Forming Tools
- Standard Forming Tools
- Form Tool Features in the Flat
- Part Document Properties
- Custom Form Tools
- Split Line
- Forming Tool
- Form Tools in Drawings
- Sheet Metal Gusset

Lesson 7: Additional Sheet Metal Functions

- Additional Sheet Metal Functions
- Cross-Breaks
- Vent Features
- Mirror Part

-
- Process Plans
 - Sheet Metal Costing

SOLIDWORKS Weldments

Code:	SA-TR47
Duration:	1 Day
Cost:	£375 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials
Description:	Weldments teaches you how to create welded structures with standard structural members. Weld beads are also covered.

The topics covered in this course are:

Lesson 1: Weldments 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

Lesson 2: Working with Weldments

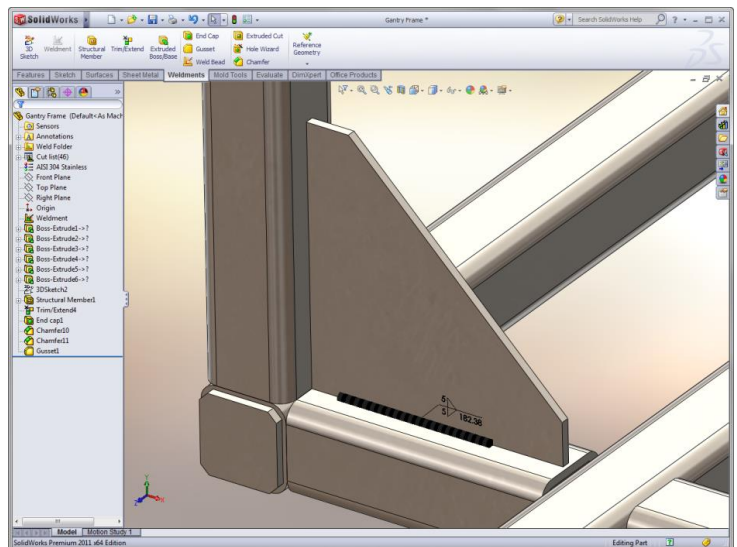
- Managing the Cut List
- Cut List Item Names
- Accessing Properties
- Cut-List Properties Dialog
- Structural Member Properties
- Adding Cut List Properties
- Bounding Boxes in Weldments
- Options for Generating Cut List Items
- Custom Structural Member Profiles
- Defining Material
- Creating Custom Profiles
- Standard or Configured Profiles
- Inserting Existing Parts
- When to Use an Assembly

Lesson 3: Configuring and Detailing Weldments

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

Lesson 4: Working with Bent Structural Members

- Working with Bent Structural Members
- 3D Sketching



SOLIDWORKS Routing – Electrical

Code: SA-TR48

Duration: 2 Days

Cost: £750 per person (includes course manual)

Prerequisites: SOLIDWORKS Essentials

Description: Routing - Electrical explains how to create, edit and manage Electrical routes, from the critical routing components and their design requirements to the sub-assemblies that contain the routes.

The topics covered in this course are:

Lesson 1: Fundamentals of Routing

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

Lesson 2: Basic Electrical Routing

- Basic Electrical Routing
- Adding Routing Components
- Start by Drag and Drop Connector
- Auto Route
- Save to External File

Lesson 3: Routing with Clips

- Routing with Clips
- Routing Through Existing Clips
- Adding Clips while Auto Routing
- Editing a Route
- Working with Clips
- Splitting a Route

Lesson 4: Electrical Routing Components

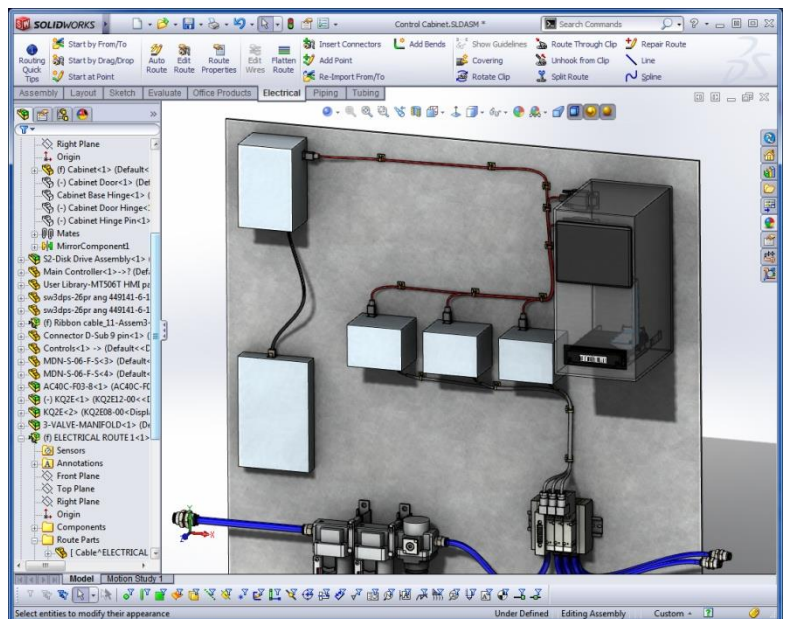
- Routing Library Parts Introduction
- Electrical Routing Library Parts
- Libraries
- Routing Component Wizard
- Routing Component Attributes
- Electrical Libraries

Lesson 5: Standard Cables

- Using Standard Cables
- Standard Cables Excel File
- Modifying Standard Cables
- Creating a Standard Cable
- Routing Templates

Lesson 6: Electrical Data Import

- Importing Data
- Routing Library Manager
- From/To Lists



- Route Properties
- Route Guidelines
- Using Guidelines and Clips

Lesson 7: Electrical Drawings

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

Lesson 8: Electrical Ribbon Cables

- Electrical Ribbon Cables
- Ribbon Cable Components
- Auto Routing Ribbon Cables
- Using Ribbon Cable Clips
- Through Connectors
- Ribbon Cable Libraries

Lesson 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

SOLIDWORKS Routing – Piping and Tubing

Code:	SA-TR49
Duration:	2 Days
Cost:	£750 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials
Description:	Routing - Piping and Tubing explains how to create, edit and manage Piping and Tubing routes, from the critical routing components and their design requirements to the sub-assemblies that contain the routes.

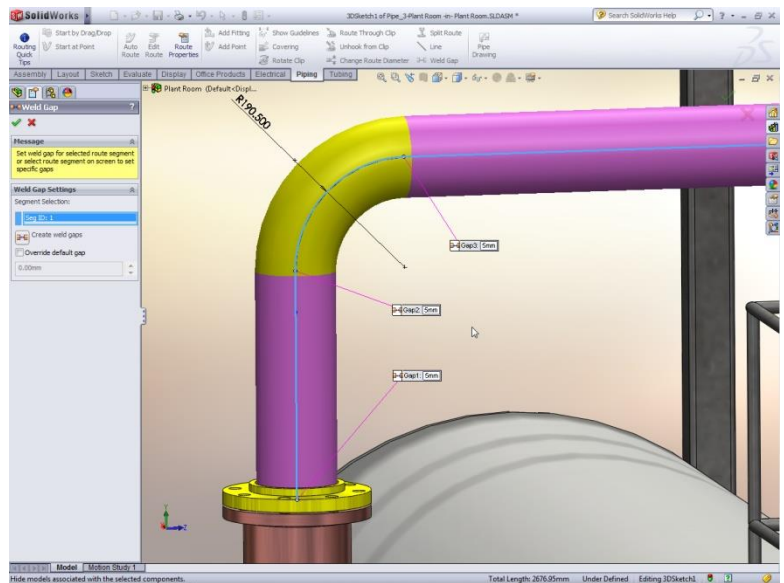
The topics covered in this course are:

Lesson 1: Fundamentals of Routing

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

Lesson 2: Piping Routes

- Piping Routes
- Pipes and Piping Components
- Routing Templates
- Creating a Piping Route
- Auto Route
- Using Routing Components with Auto Route
- Editing a Route
- Drag and Drop a Component
- Creating Custom Components
- Interferences and Clearances

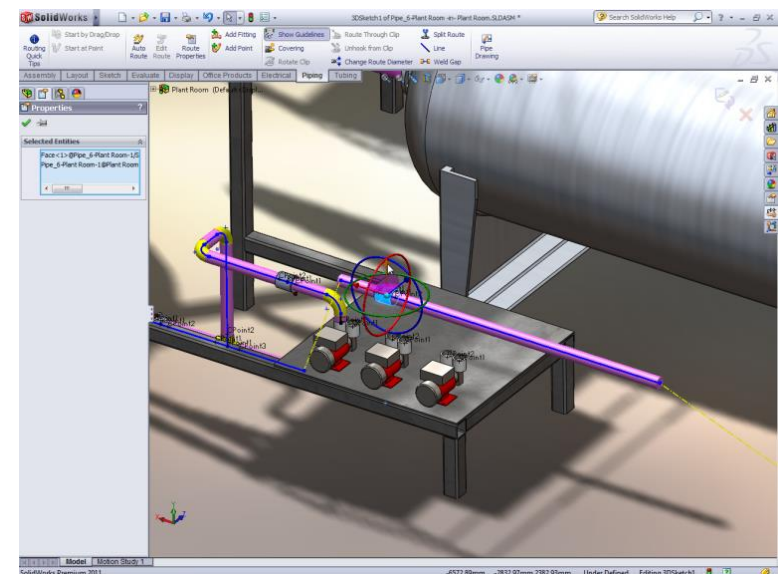


Lesson 3: Tubing Routes

- 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
- Tubing Drawings

Lesson 4: Piping and Tubing Changes

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



Lesson 5: Pipe and Tube Routing Components

- Routing Library Parts
- Libraries

- Creating Routing Library Parts
- Pipe and Tube Parts
- Pipe Parts
- Fitting Parts
- Routing Functionality Points
- Elbow Parts
- Tube Parts
- Multi-Body Components
- Equipment
- Assembly Fittings

Lesson 6: Using P&ID Files

- Piping and Instrumentation
- Adding a Pipe
- Pipe with Multiple Lines
- Pipes with In-line Fittings
- Process Drawings

Lesson 7: Piping Skids

- Piping Skids
- Grid System Feature
- Weldments
- Walk-through Animations
- Avatar Paths

Lesson 8: Using SOLIDWORKS Content

- Using SOLIDWORKS Content
- Creating a Routing Settings File

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

SOLIDWORKS File Management

Code:	SA-TR50
Duration:	1 Day
Cost:	£375 per person (includes course manual)
Prerequisites:	General experience with the SOLIDWORKS software. General experience with the Windows operating system.
Description:	SOLIDWORKS File Management teaches you how to manage files within SOLIDWORKS. This course should be taken before any of the SOLIDWORKS Enterprise PDM courses, and is also ideal for any SOLIDWORKS user not using SOLIDWORKS Enterprise PDM. It includes explanations of the SOLIDWORKS file structure, file references, file associativity, and how to manage a multi-user environment.

The topics covered in this course are:

Lesson 1: SOLIDWORKS File Structure

- Understanding SOLIDWORKS Files
- SOLIDWORKS File Structure
- Case Study: Reducing File Size
- File Shadowing
- Configuration Bodies
- Case Study: External References
- File References
- File Reference Example
- SOLIDWORKS File Conversion
- Case Study: File Conversion
- Opening Files
- Case Study: RAM Resident
- Reload
- Case Study: Quick View
- Case Study: Read-only

Lesson 3: File References

- External Reference Search Order
- Case Study: Searching for References
- Recursive Searches
- Changing References
- Case Study: In-context Features
- Locking and Breaking References
- SOLIDWORKS Explorer
- Case Study: SOLIDWORKS Explorer

Lesson 2: Saving Files

- Saving Files
- Case Study: Save Options
- Editing References
- Automatic File Backup
- Case Study: Backup/Recover
- File Properties
- Property Tab Builder
- Case Study: File Properties
- Additional Data
- Design Binder

Lesson 4: Shared Files

- Working in a Collaborative Environment
- Production Files
- Case Study: Sharing Files
- Reload
- Multiple In-context References To The Same Part
- Case Study: Motor Mounting Bracket
- Support Files
- Case Study: Shared File Locations
- SOLIDWORKS Add-Ins
- Toolbox
- Case Study: Toolbox Parts
- PhotoWorks Files
- Case Study: PhotoWorks Files
- SOLIDWORKS Explorer For Revision Management
- Case Study: Revision Management

SOLIDWORKS API Fundamentals

Code:	SA-TR51
Duration:	3 Days
Cost:	£1125 per person (includes course manual)
Prerequisites:	SOLIDWORKS Essentials, Visual Basic programming
Description:	API Fundamentals teaches you how to use the SOLIDWORKS API (Application Programming Interface) to automate and customize SOLIDWORKS.

The topics covered in this course are:

Lesson 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

Lesson 2: The API Object Model

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

Lesson 3: Setting System Options and Document Properties

- User Preferences - System Options
- User Preferences - Document Properties
- Locating the Correct APIs and Enumeration Values
- UserPreference Tables For System Options, Document Properties, and Menu Items

Lesson 4: Automating Part Design

- Case Study: Automation Tool for Parts

Lesson 5: Assembly Automation

- Case Study: Automation Tool for Assemblies

Lesson 6: Drawing Automation

- Case Study: Automating Drawing Creation

Lesson 7: Selection and Traversal Techniques

- Case Study: Programming With a Selected Object
- The SOLIDWORKS BREP Model
- Case Study: Body and Face Traversal
- Case Study: Feature Manager Traversal

Lesson 8: Adding Custom Properties and Attributes

- Case Study: Custom Properties
- Case Study: Configurations With Custom Properties
- Case Study: File Summary Information

- Case Study: Document Attributes
- The Attribute Objects
- Case Study: Face Attributes

Lesson 9: The SOLIDWORKS API SDK

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

Lesson 10: Customizing the SOLIDWORKS User Interface

- Case Study: 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

Lesson 11: Notifications

- Notifications
- Notifications in VBA
- Case Study: Simple Notification
- Case Study: Using Notifications in .NET

Appendix

- Macro Feature
- Batch Conversion 1
- Batch Conversion 2
- Assembly Traversal
- Custom Model View

SOLIDWORKS Electrical Courses

SOLIDWORKS Electrical Schematic

Code: SA-TR60

Duration: 2 Days

Cost: £750 per person (includes course manual)

Prerequisites: Completed SOLIDWORKS Essentials training, Electrical Engineering Experience

Description: The goal of this course is to teach you how to use SOLIDWORKS Electrical to optimize your drawings and designs for manufacturability so you can maximize quality, avoid rework and decrease time to market.

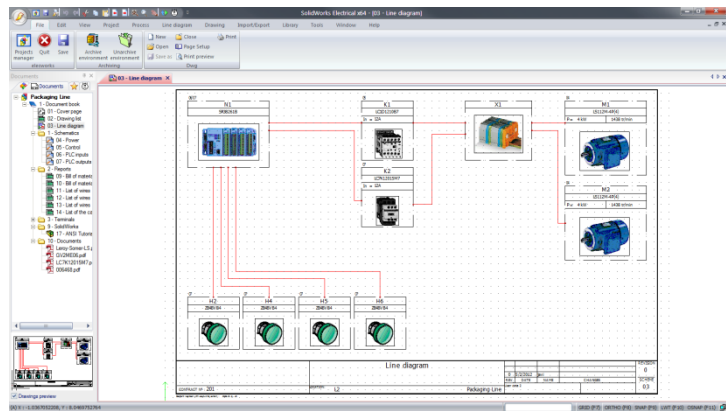
The topics covered in this course are:

Lesson 1: Projects

- Projects
- Locations
- Zooming and Scrolling

Lesson 2: Single Line Diagrams

- Archived projects
- Opening an existing project
- Line diagram symbols
- Adding cables
- Drawing tools



Lesson 3: Cabling

- Detailed cabling
- Adding manufacturer parts
- Terminal Strip
- Pin to Pin Connections
- Using Copy and Paste

Lesson 4: Creating Schematics

- Schematics
- Drawing multiple wires
- Schematic symbols
- Symbol properties
- Creating new symbols

Lesson 5: Cross Referencing the Single Line Diagram

- Cross referencing drawings
- Inserting multiple terminals
- Location outlines

Lesson 6: Control Drawings

- Drawing single wires
- Using the cross reference list
- Using functions
- Inserting single terminals
- Function outlines

Lesson 7: Managing Origin-Destination Arrows

- Replacing wires
- Origin-destination arrows

Lesson 8: Programmable Logic Controllers

- Adding a new scheme
- Adding a PLC mark
- Inserting a PLC
- Editing a PLC

Lesson 9: Macros

- Creating macros
- Using macros

Lesson 10: Editing the PLC Drawing

- Associate cable cores
- Adding a part manually
- Assigning a part
- Wire style manager
- Numbering wires

Lesson 11: Reports

- Using the report manager
- Drawing terminal strips

SOLIDWORKS Electrical 3D

Code: SA-TR61

Duration: 1 Days

Cost: £375 per person (includes course manual)

Prerequisites: Completed SOLIDWORKS Essentials training, Electrical Engineering Experience

Description: The goal of this course is to teach you how to use SOLIDWORKS Electrical to optimize your drawings and designs for manufacturability so you can maximize quality, avoid rework and decrease time to market.

The topics covered in this course are:

Lesson 1: Adding a Cabinet

- Adding a cabinet
- Insert component
- Inserting rails and ducts

Lesson 2: Inserting Components in the Cabinet

- Inserting electrical components
- Converting a part to an electrical component
- Inserting terminals

Lesson 3: Adding Door Components

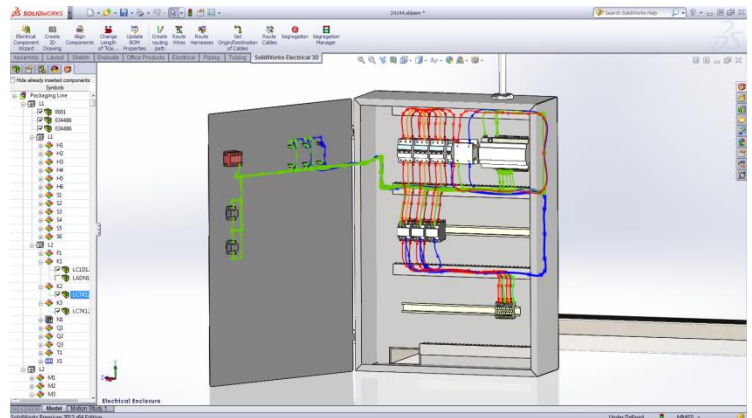
- Smart components
- Inserting smart features

Lesson 4: Propagating 2D Changes to 3D

- Changes in SOLIDWORKS Electrical 2D
- Changes at the assembly level

Lesson 5: Routing Wires

- Routing path
- Route wires



SOLIDWORKS Simulation Courses

SOLIDWORKS Plastics Professional

Code:	SA-TR62
Duration:	1 Day
Cost:	£375 per person (includes course manual)
Prerequisites:	Completed SOLIDWORKS Essentials training and a fundamental knowledge of plastic materials, plastic part design, and/or injection mould design
Description:	The SOLIDWORKS Plastics course teaches you how to use specialized simulation software tools to predict how melted plastic flows during the injection moulding process. Predicting how the plastic will flow enables you to predict manufacturing defects such as weld lines, air traps, short shots, and sink marks. By predicting these defects, you can change the part or mould geometry, the processing conditions, or the plastic material itself to eliminate or minimize them, saving energy, material, time, and money.

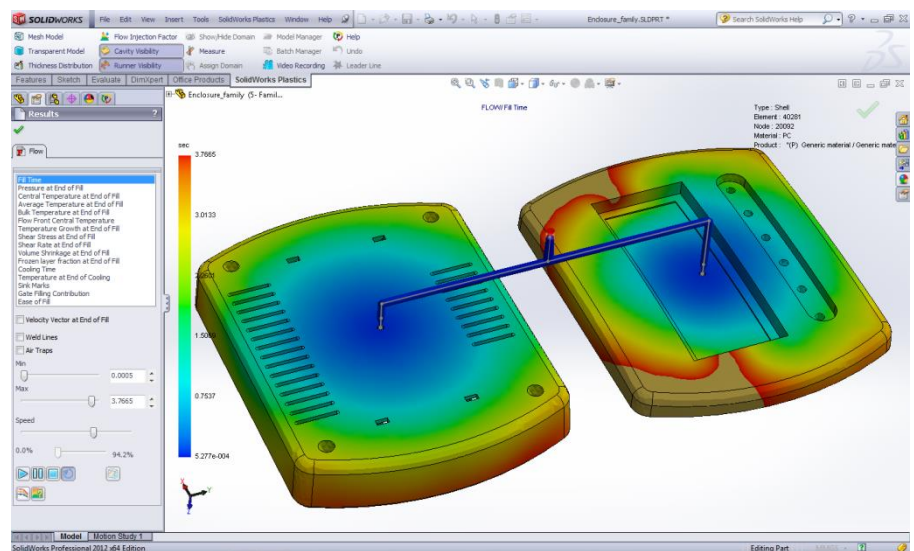
The topics covered in this course are:

Introduction

- About This Course
- Use of Colour

Lesson 1: Basic Flow Analysis

- Basic Flow Analysis
- Stages in the Process
- Element Types
- Meshing
- The PlasticsManager Tree
- Input Options
- Gates
- Running a Flow Analysis
- Flow Results



Lesson 2: Detecting Air Traps

- Detecting Air Traps
- Stages in the Process
- Air Traps
- Switching Modes for Design Changes
- Design Changes
- Parts Created Using Mold Design Tools

Lesson 3: Detecting Short Shots

- Detecting Short Shots
- Stages in the Process
- Flow Settings
- Flow Front Central Temperature

Lesson 4: The Model Manager

- The Model Manager
- Stages in the Process
- Using the Model Manager
- Copying Parts and Results
- Batch Manager
- Summary and Report

SOLIDWORKS Plastics Premium

Code:	SA-TR63
Duration:	1 Day
Cost:	£375 per person (includes course manual)
Prerequisites:	Completed SOLIDWORKS Essentials and SOLIDWORKS Plastics Professional training, and a fundamental knowledge of plastic materials, plastic part design, and/or injection mould design
Description:	The SOLIDWORKS Plastics course teaches you how to use specialized simulation software tools to predict how melted plastic flows during the injection moulding process. Predicting how the plastic will flow enables you to predict manufacturing defects such as weld lines, air traps, short shots, and sink marks. By predicting these defects, you can change the part or mould geometry, the processing conditions, or the plastic material itself to eliminate or minimize them, saving energy, material, time, and money.

The topics covered in this course are:

Lesson 1: Gate Locations and Sink Marks

- Gate Locations and Sink Marks
- Stages in the Process
- Gate Selection Rules
- Sink Marks

Lesson 2: Multiple Cavity Moulds

- Multiple Cavity moulds
- Stages in the Process
- Mirroring Cavities
- Sketching Runners
- Runner Design
- X-Y Plots

Lesson 3: Runner-Balancing

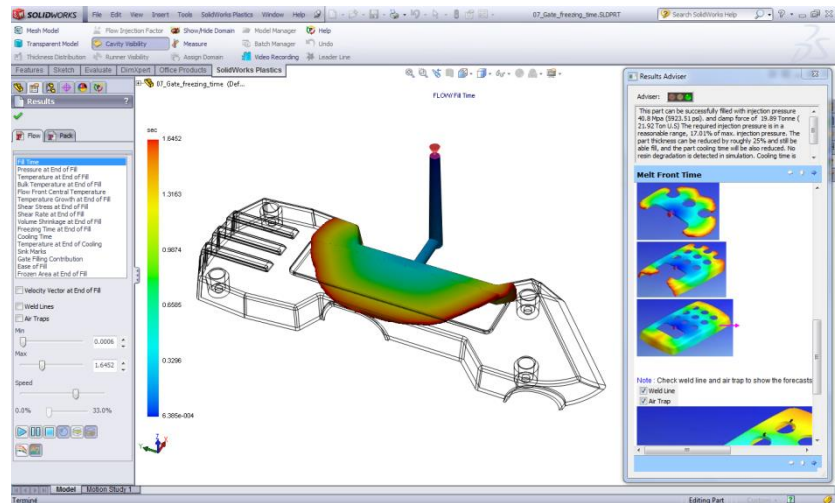
- Runner-Balancing
- Stages in the Process
- Local Refinement of Mesh
- Using Runner-Balancing

Lesson 4: Gate Freeze

- Gate Freeze
- Stages in the Process
- Solid Mesh
- Pack Settings
- Flow and Pack Analysis
- Pack Results

Lesson 5: Optimizing Cooling Time

- Optimizing Cooling Time
- Stages in the Process
- Multiple Gates



Lesson 6: Using Inserts

- Using Inserts
- Stages in the Process
- Cavities and Inserts
- Materials for Inserts

Lesson 7: Mesh Repairs

- Mesh Repairs
- Stages in the Process
- Element Issues
- Edit Mesh

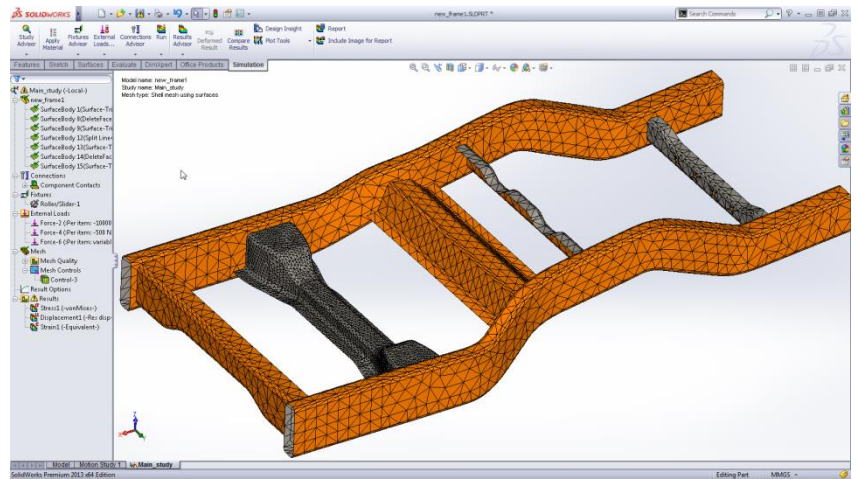
SOLIDWORKS Simulation

Code:	SA-TR64
Duration:	3 Days
Cost:	£1,125
Prerequisites:	Completed SOLIDWORKS Essentials training and basic mechanical engineering concepts is recommended.
Description:	This course is designed to make SOLIDWORKS users productive more quickly with the SOLIDWORKS Simulation Bundle. This course will provide an in-depth coverage on the basics of Finite Element Analysis (FEA), covering the entire analysis process from meshing to evaluation of results for parts and assemblies. The class discusses linear stress analysis, gap/contact analysis, and best practices.

The topics covered in this course are:

Introduction

- About This Course
- What is SOLIDWORKS Simulation?
- What Is Finite Element Analysis?
- Build Mathematical Model
- Build Finite Element Model
- Solve Finite Element Model
- Analyze Result
- Errors in FEA
- Finite Elements
- Degrees of Freedom
- Calculations in FEA
- Interpretation of FEA Results
- Units of Measurement
- Limitations of SOLIDWORKS Simulation



Lesson 1: The Analysis Process

- Objectives
- The Analysis Process
- Case Study: Stress in a Plate
- Project Description
- SOLIDWORKS Simulation Options
- Preprocessing
- Meshing
- Postprocessing
- Multiple Studies
- Reports

Lesson 2: Mesh Controls, Stress Concentrations and Boundary Conditions

- Objectives
- Mesh Control
- Case Study: The L Bracket
- Project Description
- Case Study: Analysis of Bracket with a Fillet
- Case Study: Analysis of a Welded Bracket
- Understanding the Effect of Boundary Conditions

Lesson 3: Assembly Analysis with Contacts

- Objectives
- Contact Analysis
- Case Study: Pliers with Global Contact
- Pliers with Local Contact

Lesson 4: Symmetrical and Free Self-Equilibrating Assemblies

- Objectives
- Shrink Fit Parts
- Case Study: Shrink Fit
- Project Description
- Analysis with Soft Springs

Lesson 5: Assembly Analysis with Connectors

- Objectives
- Connecting Components
- Connectors
- Case Study: Vise Grip Pliers

Lesson 6: Compatible/Incompatible Meshes

- Objectives
- Compatible / Incompatible Meshing
- Case Study: Rotor

Lesson 7: Assembly Analysis Mesh Refinement

- Objectives
- Mesh Control in an Assembly
- Case Study: Cardan Joint
- Problem Statement
- Part 1: Draft Quality Coarse Mesh Analysis
- Part 2: High Quality Mesh Analysis

Lesson 8: Analysis of Thin Components

- Objectives
- Thin Components
- Case Study: Pulley
- Part 1: Mesh with Solid Elements
- Part 2: Refined Solid Mesh
- Solid vs. Shell
- Creating Shell Elements
- Part 3: Shell Elements - Mid-plane Surface
- Results Comparison
- Case Study: Joist Hanger

Lesson 9: Mixed Meshing Shells & Solids

- Objectives
- Mixed Meshing Solids and Shells
- Case Study: Pressure Vessel

Lesson 10: Mixed Meshing Solids, Beams & Shells

- Objectives

- Mixed Meshing
- Case Study: Particle Separator

Lesson 11: Design Scenarios

- Objectives
- Design Study
- Case Study: Suspension Design
- Part 1: Multiple Load Cases
- Part 2: Geometry Modification

Lesson 12: Thermal Stress Analysis

- Objectives
- Thermal Stress Analysis
- Case Study: Bimetallic Strip
- Examining Results in Local Coordinate Systems
- Saving Model in its Deformed Shape

Lesson 13: Adaptive Meshing

- Objectives
- Adaptive Meshing
- Case Study: Support Bracket
- h-Adaptivity Study
- p-Adaptivity Study
- h vs. p Elements - Summary

Lesson 14: Large Displacement Analysis

- Objectives
- Small vs. Large Displacement Analysis
- Case Study: Clamp
- Part 1: Small Displacement Linear Analysis
- Part 2: Large Displacement Nonlinear 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

Appendix B: Customer Help and Assistance

- Customer Help and Assistance



SOLIDWORKS Simulation Professional

Code:	SA-TR65
Duration:	1 Day
Cost:	£375 per person (includes course manual)
Prerequisites:	Students must have attended the introductory SOLIDWORKS Simulation course (3 days) or must have working knowledge of the SOLIDWORKS Simulation software. Knowledge of SOLIDWORKS and basic mechanical engineering concepts is recommended.

Description: This course is designed to make SOLIDWORKS Simulation users productive with the SOLIDWORKS Simulation Professional extension. This 1 day course will provide an in-depth coverage on the advanced topics in Finite Element Analysis (FEA) including heat transfer analysis, frequency analysis, fatigue, stability analysis based on the linear buckling concepts, 2D simulations (plane stress, strain and axisymmetry) and pressure vessel modulus. Example of parts and assemblies including those with various gap/contact conditions are reviewed. (1 day)

The topics covered in this course are:

Lesson 1: Frequency Analysis of Parts

- Objectives
- Modal Analysis Basics
- Case Study: The Tuning Fork
- Project Description
- Frequency Analysis With Supports
- Frequency Analysis Without Supports
- Frequency Analysis with Load

Lesson 2: Frequency Analysis of Assemblies

- Case Study: The Engine Mount
- Project Description
- All Bonded Contact Conditions
- Bonded and Allow Penetration Contacts

Lesson 3: Buckling Analysis

- Buckling Analysis
- Case Study: Particle Separator
- Project Description

Lesson 4: Thermal Analysis

- Thermal Analysis Basics
- Case Study: Microchip Assembly
- Project Description
- Steady-State Thermal Analysis
- Transient Thermal Analysis
- Transient Analysis with Time Varying Load
- Transient Thermal Analysis using a Thermostat

Lesson 5: Thermal Analysis with Radiation

- Case Study: Spot Light Assembly
- Project Description
- Steady State Analysis

Lesson 6: Advanced Thermal Stress2D Simplification

- Thermal Stress Analysis
- Case Study: Metal Expansion Joint
- Project Description
- Thermal Analysis
- Thermal Stress Analysis
- 3D model

Lesson 7: Fatigue Analysis

- Fatigue
- Stress-life (S-N) Based Fatigue
- Case Study: Pressure Vessel
- Thermal Study
- Thermal Stress Study
- Fatigue Terminology
- Fatigue Study
- Fatigue study with dead load

Lesson 8: Advanced Fatigue Analysis

- Case Study: Suspension
- Project Description
- Fatigue Study

Lesson 9: Drop Test Analysis

- Drop Test Analysis
- Case Study: Camera
- Project Description
- Rigid Floor Drop Test
- Elastic Floor Drop Test
- Elasto-Plastic Material Model
- Drop Test with Contact

Lesson 10: Optimization Analysis

- Optimization Analysis
- Case Study: Press Frame
- Project Description
- Static and Frequency Analyses
- Optimization Analysis
- Design Study

Lesson 11: Pressure Vessel Analysis

- Case Study: Pressure Vessel
- Project Description
- Pressure Vessel Analysis
- Manhole Nozzle Flange and Cover

SOLIDWORKS PDM Courses

Using SOLIDWORKS PDM Professional

Code:	SA-TR52
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	Basic experience with the SOLIDWORKS software, experience with the Windows™ operating system, and knowledge of SOLIDWORKS file structure and referencing.
Description:	The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS PDM. The intended audience for this course is anyone who will manage files with SOLIDWORKS PDM.

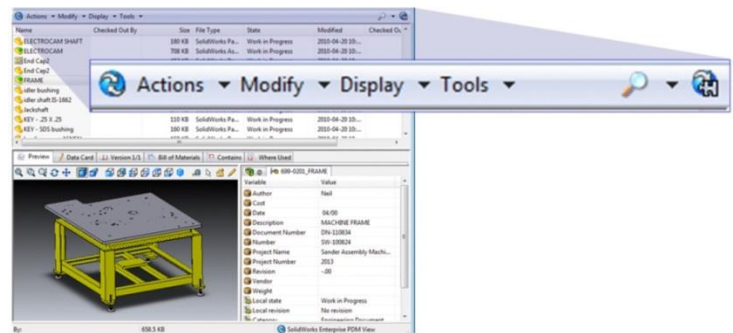
The topics covered in this course are:

Introduction

- About This Course

Lesson 1: SOLIDWORKS PDM Concepts

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



Lesson 2: SOLIDWORKS Enterprise PDM User Interface

- SOLIDWORKS PDM User Interface
- Case Study: Exploring SOLIDWORKS PDM

Lesson 3: Document Creation and Check In

- Document Creation and Check In
- Case Study: Checking in Documents
- Advanced Document Check In
- Case Study: Checking in Documents with References

Lesson 4: Versioning Files

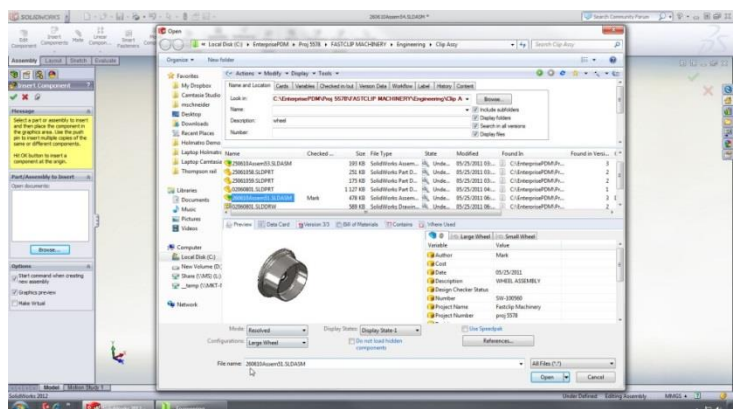
- Versioning Files
- Case Study: Versioning Files

Lesson 5: File References

- File References
- Case Study: File References
- Case Study: Copy File and References

Lesson 6: Searching

- Searching SOLIDWORKS PDM
- Case Study: Search



Lesson 7: Workflow and Notification

- SOLIDWORKS PDM Workflow
- Case Study: Workflow

Lesson 8: Managing Local Cache

- Managing Local Cache

Lesson 9: Working in SOLIDWORKS

- Versioning CAD Files
- Case Study: Versioning Files

Appendix A: Working with Bill of Materials

- Bill of Materials (BOM)
- Case Study: Modify a Computed BOM
- Case Study: Create and Modify a Named BOM

Administering SOLIDWORKS PDM Professional

Code: SA-TR54

Duration: 3 Days

Cost: £1125 per person

Prerequisites: Using SOLIDWORKS PDM course, basic experience with the SOLIDWORKS PDM software, experience with the Windows™ operating system, and knowledge of SOLIDWORKS PDM file structure and referencing.

Description: The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS PDM. The intended audience for this course is anyone who will setup and/or administer SOLIDWORKS PDM.

The topics covered in this course are:

Introduction

- About This Course

Lesson 1: Installation Planning

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

Lesson 2: The Administration Tool

- SOLIDWORKS PDM Administration Tool
- Creating a SOLIDWORKS PDM File Vault
- Case Study: Create a New File Vault
- Create a Local File Vault View
- Setup Overview

Lesson 3: Users and Groups

- Users
- Case Study: Creating Users
- Groups
- Case Study: Create a New Group

Lesson 4: Folder Card Creation

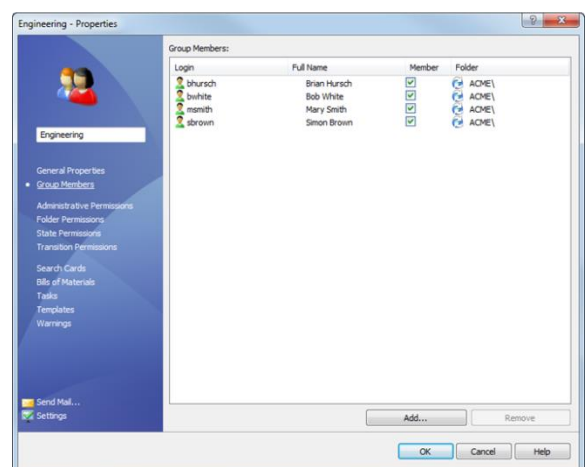
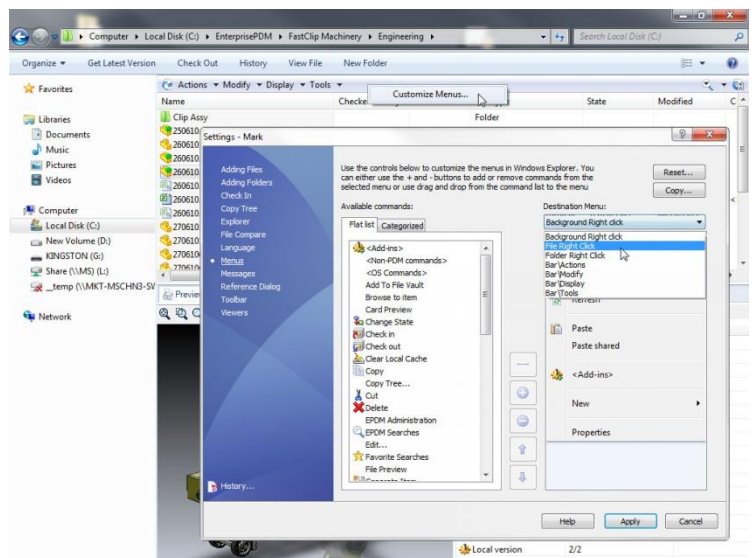
- Data Cards
- Anatomy of a Data Card
- Case Study: Design a Folder Data Card

Lesson 5: File and Search Cards

- Importing Data Cards
- Case Study: Design a File Data Card
- Case Study: Design a Search Data Card

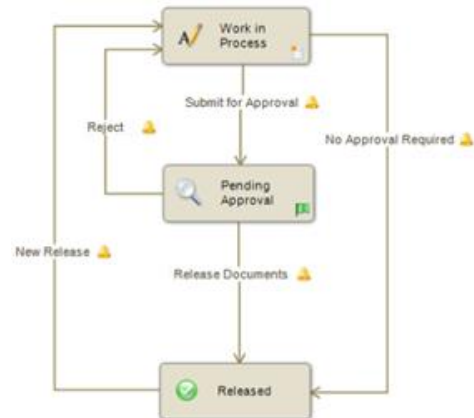
Lesson 6: Column and Bill of Materials (BOM) Views

- Columns
- Case Study: Column
- Bill Of Material



Lesson 7: Workflow

- Categories
- Case Study: Create Categories
- Case Study: Create New Workflows
- Revisions
- Case Study: Create New Revision Schemes
- Case Study: CAD Files Revision Scheme

**Lesson 8: Notifications and Tasks**

- Notifications
- Case Study: Assign Notifications
- Case Study: Convert Task

Lesson 9: Folder Templates

- Templates
- Case Study: Folder Templates

Lesson 10: File Templates

- File Templates
- Case Study: File Template

Lesson 11: Data Migration

- Migrating Legacy Data
- Case Study: Data Migration
- Migrating Revisions

Lesson 12: Vault Backup

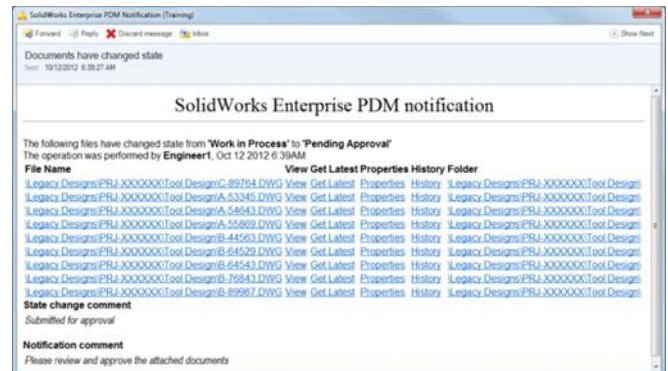
- Backing Up File Vaults

Appendix A: File Types and Settings

- File Types
- User Settings

Appendix B: Data Import

- Import and Export ERP Data
- Case Study: Alias Set
- Case Study: Export Rule
- Case Study: Import Rule

Appendix C: Toolbox Setup**Appendix D: Routing Setup****Appendix E: CircuitWorks Setup**

Administering SOLIDWORKS PDM Standard

Code: SA-TR58

Duration: 2 Days

Cost: £750 per person

Prerequisites: Using SOLIDWORKS PDM course, basic experience with the SOLIDWORKS PDM software, experience with the Windows™ operating system, and knowledge of SOLIDWORKS PDM file structure and referencing.

Description: The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS PDM. The intended audience for this course is anyone who will setup and/or administer SOLIDWORKS PDM Standard.

The topics covered in this course are:

Introduction

- About This Course

Lesson 1: Installation Planning

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

Lesson 2: The Administration Tool

- SOLIDWORKS PDM Administration Tool
- Creating a SOLIDWORKS PDM File Vault
- Case Study: Create a New File Vault
- Create a Local File Vault View
- Setup Overview

Lesson 3: Users and Groups

- Users
- Case Study: Creating Users
- Groups
- Case Study: Create a New Group

Lesson 4: Folder Card Creation

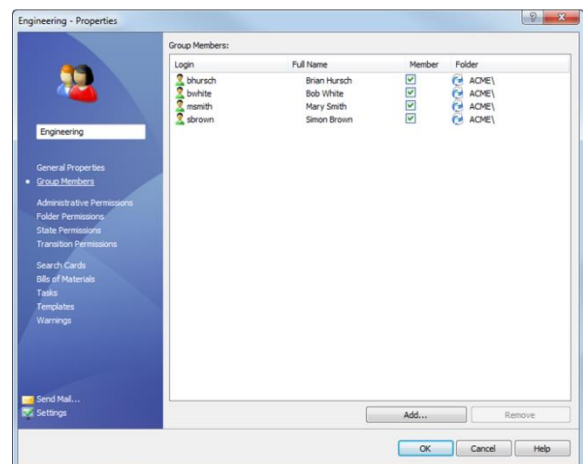
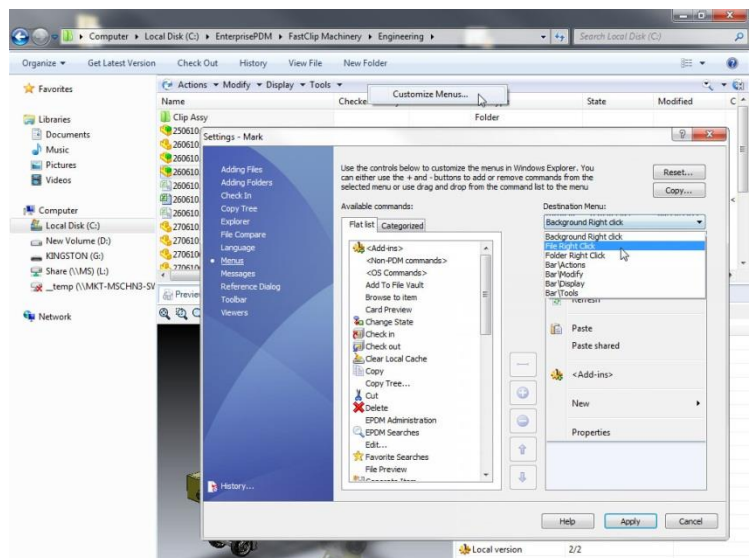
- Data Cards
- Anatomy of a Data Card
- Case Study: Design a Folder Data Card

Lesson 5: File and Search Cards

- Importing Data Cards
- Case Study: Design a File Data Card
- Case Study: Design a Search Data Card

Lesson 6: Column and Bill of Materials (BOM) Views

- Columns
- Case Study: Columns
- Bill Of Material
- Case Study: BOM



Lesson 7: Workflow

- Workflows
- Case Study: Create New Workflow
- Revisions
- Case Study: Create New Revision Schemes
- Case Study: CAD Files Revision Scheme

Lesson 8: Notifications

- Notifications
- Case Study: Assign Notifications

Lesson 9: Data Migration

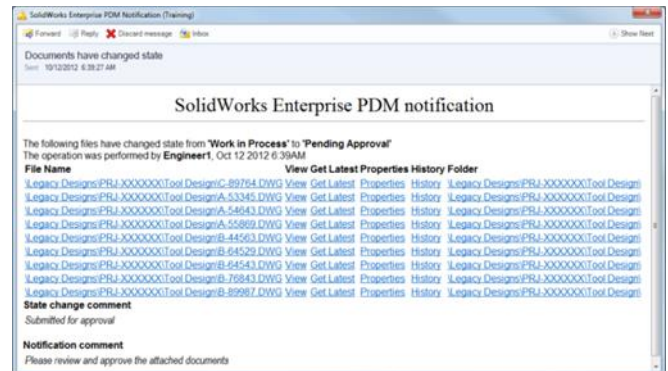
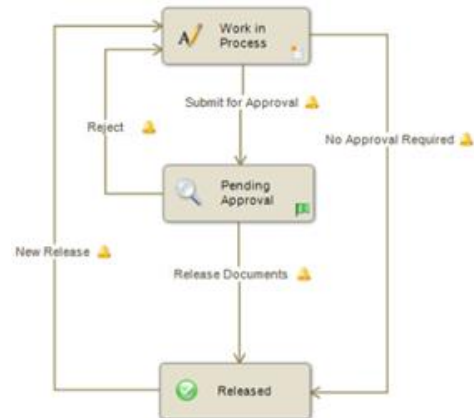
- Migrating Legacy Data
- Case Study: Data Migration
- Migrating Revisions

Lesson 10: Vault Backup

- Backing Up File Vaults

Appendix A: File Types and Settings

- File Types
- User Settings

Appendix B: Toolbox Setup**Appendix C: Routing Setup****Appendix D: CircuitWorks Setup**

SOLIDWORKS Workgroup PDM Courses

SOLIDWORKS Workgroup PDM Contributor

Code:	SA-TR55
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	Basic experience with the SOLIDWORKS file structure and referencing, and the Windows™ operating system.
Description:	The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS Workgroup PDM. The intended audience is people who do not use SOLIDWORKS software, but still manage SOLIDWORKS files.

The topics covered in this course are:

Introduction

- About This Course
- Windows®
- Use of Colour

Lesson 1: PDM Concepts and File Management

- The Four Pillars of PDM
- What is the SOLIDWORKS Workgroup PDM Software?
- SOLIDWORKS Workgroup PDM Overview
- SOLIDWORKS Workgroup PDM Components
- Basic Rules
- SOLIDWORKS File Management
- File References
- File Reference Example
- Opening Files
- Saving Files
- File Properties
- External Reference Search Order

Lesson 2: SOLIDWORKS Explorer and SOLIDWORKS Workgroup PDM Add-In

- SOLIDWORKS Workgroup PDM Contributor
- Case Study: SOLIDWORKS Explorer
- Renaming Components
- SOLIDWORKS Workgroup PDM Add-In
- Case Study: SOLIDWORKS Workgroup PDM Add-In

Lesson 3: Managing Files with SOLIDWORKS Workgroup PDM Add-In

- Document Check In Process
- Case Study: Check in Documents
- Revision Scheme
- Case Study: Change Project
- Working on Vaulted Documents
- Case Study: Working on Documents
- Reporting
- Searching
- Searching for Documents

-
- Case Study: Windows Search
 - Searching for Documents in the Vault
 - Case Study: Searching the Vault
 - Case Study: Renaming Documents
 - Explain Access

Lesson 4: SOLIDWORKS Workgroup PDM Viewer

- SOLIDWORKS Workgroup PDM Viewer
- Case Study: Accessing the SOLIDWORKS Workgroup PDM Viewer

SOLIDWORKS Workgroup PDM CAD Editor

Code:	SA-TR56
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	Basic experience with the SOLIDWORKS file structure and referencing, and the Windows™ operating system.
Description:	The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS Workgroup PDM. The intended audience for this course is SOLIDWORKS users.

The topics covered in this course are:

Introduction

- About This Course
- Windows®
- Use of Colour

Lesson 1: PDM Concepts and File Management

- The Four Pillars of PDM
- What is the SOLIDWORKS Workgroup PDM Software?
- SOLIDWORKS Workgroup PDM Overview
- SOLIDWORKS Workgroup PDM Components
- Basic Rules
- SOLIDWORKS File Management
- File References
- File Reference Example
- Opening Files
- Saving Files
- File Properties
- External Reference Search Order

Lesson 2: Document Check In

- SOLIDWORKS Workgroup PDM User Interface
- Starting SOLIDWORKS Workgroup PDM
- SOLIDWORKS User Interface
- Getting Help
- Setting up SOLIDWORKS Workgroup PDM
- Users Accounts
- Case Study: Checking In SOLIDWORKS Documents
- User Options
- Document Check In Process
- Revision Scheme
- Lifecycle
- Checking in References
- Logging In/Out
- Creating Sub-projects
- Reporting

Lesson 3: Accessing Documents

- Working on Vaulted Documents
- Case Study: Modify an assembly

-
- Non-SOLIDWORKS Documents

Lesson 4: SOLIDWORKS Workgroup PDM Environment

- Searching for Documents
- Case Study: Setting Up Windows Search
- Searching for Documents in the Vault
- Case Study: Advanced Search
- Other SOLIDWORKS Workgroup PDM Activities
- Reports
- Custom Properties
- Case Study: Adding Properties
- Drawing Revisions
- Case Study: Drawing Revisions
- Collaboration
- Case Study: Collaboration
- Printing Drawings From The Vault
- Case Study: Batch Print
- Exporting SOLIDWORKS Workgroup PDM Files
- Automated File Properties
- Case Study: Custom Properties in Templates
- Bulk Loading Files
- Alternate Methods to Access the Vault
- Case Study: Using SOLIDWORKS Explorer
- Functional Differences
- User Options
- SOLIDWORKS Workgroup PDM Viewer
- Case Study: Accessing the SOLIDWORKS Workgroup PDM Viewer
- Software Updates

Lesson 5: Toolbox and Files that are Not Revision Managed

- Toolbox Parts
- Toolbox Case Studies
- Case Study: Adding Toolbox Parts
- Files That are Not Revision Managed
- Case Study: Not Revision Managed Files
- Case Study: The Toolbox Flag
- Standard Parts
- Drag and Drop
- Case Study: Common Parts Project
- Case Study: Design Library
- Revision Bumping
- Adding Notes
- Case Study: Revision Bumping
- Removing Files From the Vault
- Explain Access
- Case Study: Remove Files From The Vault
- Managing Configuration Revisions
- Case Study: Configuration Revisions

Administering SOLIDWORKS Workgroup PDM

Code:	SA-TR57
Duration:	1 Day
Cost:	£375 per person
Prerequisites:	Basic experience with the SOLIDWORKS software and the Windows™ operating system, and knowledge of SOLIDWORKS file structure and referencing.
Description:	The focus of this course is on the fundamental skills and concepts central to the successful use of SOLIDWORKS Workgroup PDM. The intended audience for this course is vault administrators.

The topics covered in this course are:

Introduction

- About This Course
- Windows
- Use of Colour

Lesson 1: Planning for PDM

- Planning for SOLIDWORKS Workgroup PDM
- The Planning Process
- Data Management Plan
- Transition Plan
- The VaultAdmin Tool

Lesson 2: SOLIDWORKS Workgroup PDM Setup

- Administrator Topics
- The Vault
- Vault Backup
- Vault Restoration
- Case Study: Replace The Vault
- VaultAdmin Tool
- VaultAdmin Settings
- User Accounts
- Case Study: User Accounts
- Case Study: Groups
- Projects
- Project Access
- Project Settings
- Case Study: Copy Project Settings
- Revision Scheme
- Lifecycles
- Case Study: Create Lifecycles
- Standard Libraries
- Case Study: Common Parts
- Case Study: Toolbox Parts
- Vault Settings
- Case Study: Global Settings
- Custom Properties
- Case Study: Adding Custom Properties
- Search
- Case Study: Index the Vault

-
- Revision Table
 - Case Study: Revision Table Properties
 - Triggers

Lesson 3: Maintaining the Vault

- Maintaining The Vault
- Case Study: Vault Backup
- Case Study: Log Files
- Tree Options
- Maintaining User Accounts
- Case Study: Modify Users
- Case Study: Passwords
- Regaining Disk Space
- Case Study: Obsolete Folders
- Archiving Files
- Case Study: Archiving Files
- Update Software
- File Conversion
- Case Study: Converting Files In The Vault

Lesson 4: Administration Tasks

- Administrator Tasks
- Mapping Custom Properties
- Case Study: Property Mapping
- Copy Project
- Case Study: Copy Project
- Bulk Loading Files
- Case Study: Bulk Check In Files
- Transferring Projects between Vaults
- Case Study: Exporting Projects

Appendix

- Best Practices

SOLIDWORKS Composer Courses

Using SOLIDWORKS Composer

Code:	SA-TR70
Duration:	3 Days
Cost:	£1,125 per person
Prerequisites:	Experience with the Windows operating system.
Description:	Teaches you how to use the SOLIDWORKS Composer software to create 2D and 3D output from CAD design data. This course is compatible with versions V6R2013.

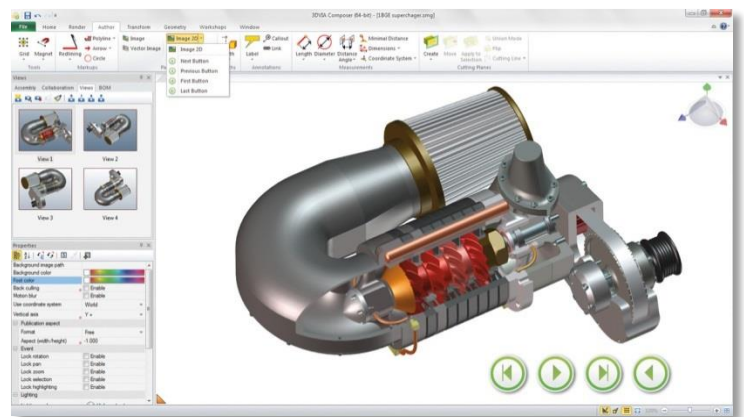
The topics covered in this course are:

Lesson 1: Quick Start

- Quick Look at SOLIDWORKS Composer

Lesson 2: Getting Started

- What is the SOLIDWORKS Composer Application?
- Starting SOLIDWORKS Composer
- SOLIDWORKS Composer Terminology
- SOLIDWORKS Composer User Interface
- Preferences and Document Properties
- Creating 2D Output
- Exercise 1: User Interface
- Exercise 2: Terminology

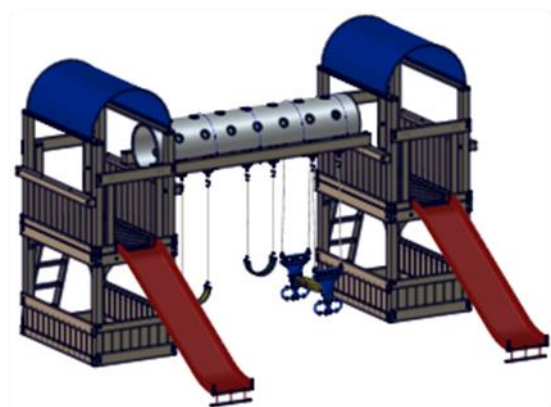


Lesson 3: Creating Cover and Detail Images

- Overview
- Views
- Rendering Tools
- Navigation Tools
- Camera Alignment Tools
- Create 2D Output
- Digger
- Exercise 3: Navigation Tools
- Exercise 4: Digger
- Exercise 5: Updating Views

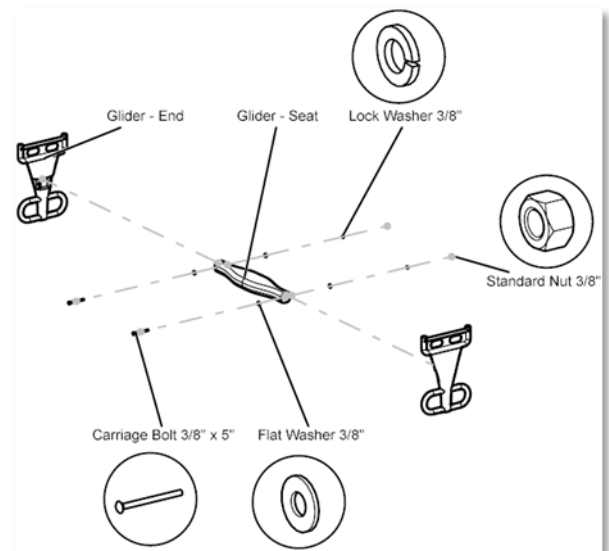
Lesson 4: Creating an Exploded View

- Overview
- Visibility Tools
- Exploded Views
- Collaborative Actors
- Styles
- Vector Graphics Output
- Exercise 6: Exploded View
- Exercise 7: Styles
- Exercise 8: Markups and Annotations
- Exercise 9: Visibility and Rendering Tools



Lesson 5: Creating Additional Exploded Views

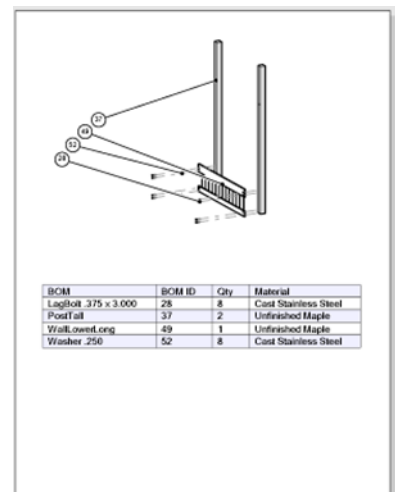
- Overview
- Paper Space
- Importing Files
- File Types
- Update Views with Selected Actors
- Align Actors
- Explode Lines
- Custom Views
- Linking Between Views
- Exercise 10: Importing Assemblies
- Exercise 11: Merging and Aligning Actors
- Exercise 12: Custom Views

**Lesson 6: Creating Bills of Materials**

- Overview
- Bill of Materials
- Vector Graphics Output
- Another BOM Table
- Assembly Level BOM
- Exercise 13: Exploded Views, BOMs, and Callouts
- Exercise 14: Assembly Level BOMs
- Exercise 15: Vector Graphics Files

Lesson 7: Creating a Marketing Image

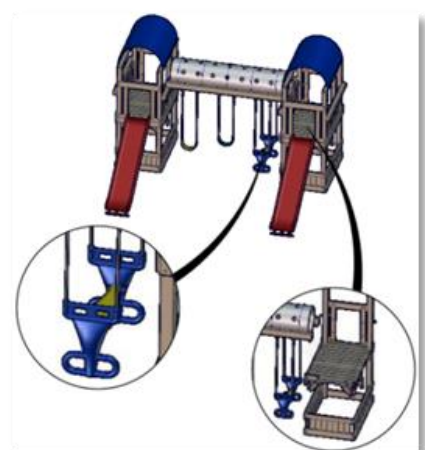
- Overview
- Selections
- Textures
- Lighting
- Scenes
- High Resolution Image
- Exercise 16: Lights and Textures
- Exercise 17: Render Effects

**Lesson 8: Creating an Animation**

- Overview
- Timeline Pane
- Position Keys
- Exercise 18: Creating an Explode Animation

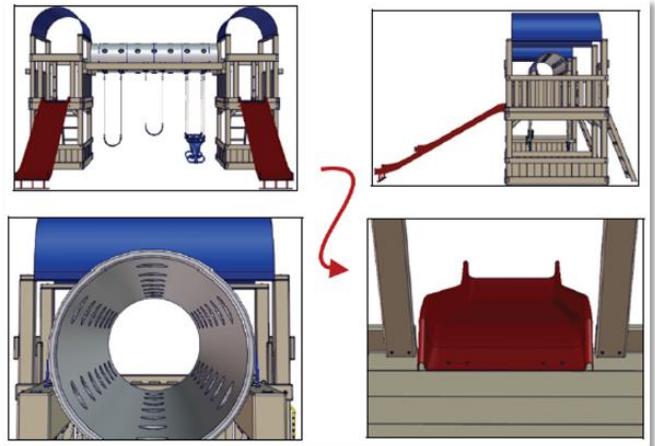
Lesson 9: Creating Interactive Content

- Overview
- Selections in the Key Track
- Position and Property Keys
- Filters
- Animating Collaborative Actors
- Digger Keys
- Events
- Exercise 19: Managing the Timeline Pane
- Exercise 20: Animating Changes to Actors
- Exercise 21: Events



Lesson 10: Creating a Walkthrough Animation

- Overview
- Camera Keys
- Adding Views to the Timeline
- Grids
- Additional Camera Functionality
- Exercise 22: Camera Keys 1
- Exercise 23: Camera Keys 2



Lesson 11: Creating an Explode and Collapse Animation

- Overview
- Assembly Selection Mode
- Scenarios
- Exercise 24: Animating Actors and the Digger
- Exercise 25: Compound Motion and Scenarios

Lesson 12: Updating 3DVIA Composer Files

- Overview
- Update an Entire Assembly
- Changing the Geometry of an Actor
- Exercise 26: Updating from CAD

Lesson 13: Publishing from 3DVIA Composer

- Overview
- Preparing a File for Publishing
- Publishing to PDF
- Publishing inside Microsoft Word
- Publishing to HTML
- Exercise 27: Publishing to PDF
- Exercise 28: Publishing inside Microsoft Word
- Exercise 29: Publishing inside HTML

Appendix A: Additional Applications and Workshops

- Overview
- SOLIDWORKS Sync and SOLIDWORKS Enterprise Sync
- SOLIDWORKS Composer Check
- SOLIDWORKS Composer Path Planning
- Simplification Workshop

KeyShot Courses

KeyShot Training

Code:	SA-KST
Duration:	1 Day
Cost:	£850 per day (for up to 5 attendees)
Prerequisites:	None
Description:	KeyShot Training teaches you how to use KeyShot in conjunction with any CAD package to bring 3D models in and apply the correct materials before creating photo realistic renderings.

The topics covered in this course are:

Lesson 1: Importing Models

- Techniques for preparing the CAD models ready for import.
- Importing options for single parts and assemblies
- Once imported look at the structures within the KeyShot environment
- Manipulate the model for positioning, rotating and zooming.

Lesson 2: Assigning Materials

- Assign materials from the predefined library with a simple drag and drop.
- Assign material directly to components in the Project.
- Create new materials and add to the Library
- Understand and work with textures and bumps.
- Add decals directly to the model and explain which graphics file to use



Lesson 3: Adjusting Environments

- Using Environments to create the desired lighting effect.
- Manipulate the Environments to rotate the light and shadow around the model.
- Work with or without the environments within KeyShot to create stunning shots

Lesson 4: Position the Camera

- Look at photographic techniques used by studio photographers that can be applied in KeyShot.
- Adding Perspective
- Other creative camera techniques

Lesson 5: Tune Backgrounds

- Use photographs as backdrops in the image to add realism to shot.

Lesson 6: Save the Shot

- Speed options for test renderings/images
- Queuing options for remote rendering
- Quality settings

Registration Procedure

Choose the appropriate course, or courses, using the information provided in this prospectus.

Telephone us on 0121 544 1400 to confirm the course availability. Once a date has been agreed, a place on the course will be provisionally booked for you. This will remain provisional until receipt of the Training Course Registration Form enclosed with this prospectus and a purchase order or cheque to confirm the booking.

Fees and Invoicing

In order to confirm all provisional bookings, we will require either a purchase order, or alternatively, a cheque along with your Training Course Registration forms (enclosed with this prospectus). If you submit a purchase order, you will receive an invoice for the required sum and payment in full is due by return.

Terms and Conditions

SOLID Applications Ltd reserve the right to cancel the enrolment of any delegate for whom payment has not been received one week prior to the start of the course. In such circumstances our standard cancellation terms will apply.

SOLID Applications Ltd reserves the right to cancel a course:

- Through any circumstance beyond its control
- Through any circumstances deemed by SOLID Applications Ltd to affect its ability to provide training to the normal standard
- If it does not attract the minimum number of delegates

In such circumstances, delegates will be given the maximum possible notice and offered enrolment on the next available course.

Courses are scheduled on demand and the frequency of any course will reflect that demand.

Cancellation Terms

When a confirmed booking has been accepted by SOLID Applications Ltd and it is subsequently cancelled or transferred to a later course at the request of the customer, SOLID Applications Ltd reserves the right to apply the charges set out below:

<i>Where the request is received in writing</i>	<i>Cancellation</i>	<i>Transfer to later course</i>
More than five weeks before commencement	No charge	No charge
Three or Four weeks before commencement	25% of fee	No charge
One or Two weeks before commencement	100% of fee	50% of fee

Training Course Registration Form

PLEASE COMPLETE THIS FORM IN BLOCK CAPITALS AND RETURN TO SOLID APPLICATIONS LTD BY FAX (0121 544 1600) OR EMAIL (training@solidapps.co.uk)

Company:		Contact:	
Address:		Position:	
		Department:	
		Tel. No.:	
Postcode:		E-mail:	

<i>Delegate Name (s)</i>	<i>Course Code/Title</i>	<i>Date and Location</i>	<i>Purchase Order Number *</i>

Special dietary requirements?	
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YOUR PLACE CANNOT BE CONFIRMED UNTIL WE HAVE A PURCHASE ORDER. ONCE WE RECEIVE YOUR COMPLETED REGISTRATION FORM AND PURCHASE ORDER NUMBER, WE CAN SECURE YOUR BOOKING AND SEND CONFIRMATION AND JOINING INSTRUCTIONS.

**WE REGRET WE CANNOT ACCEPT ATTENDANCE ON ANY SOLID APPLICATIONS TRAINING COURSE WITHOUT A PURCHASE ORDER.*

Purchase Order No(s):		Order Value:	
Name:			
Company:			
Signature:			
Date:			

