

Certified SolidWorks Essentials Part & Assembly Modelling			
Course Code: CAD01	Level: 1	Duration: 3 Days	Price in Training Vouchers: 3
Manuals Supplied	SolidWorks Essentials		
Description	This course teaches you how to use SolidWorks mechanical design automation software to build parametric models of parts and assemblies, and how to make drawings of those parts and assemblies.		
Prerequisites	Mechanical design experience; experience with the Windows™ operating system. To gain the most value from this course attendees should also complete the 'Getting started' Tutorials. These can be found by selecting 'SolidWorks tutorials' from the list under the 'Help' pull down menu in SolidWorks.		
The topics covered in this course are:			
<p>Introduction About This Course Windows® XP Use of Colour</p> <p>Lesson 1: SolidWorks Basics and the User Interface What is the SolidWorks Software Design Intent File References Opening Files The SolidWorks User Interface</p> <p>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</p> <p>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 Filletting Detailing Basics Drawing Views Centre Marks Dimensioning Changing Parameters</p> <p>Lesson 4: Modelling a Casting or Forging 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 Using Copy and Paste</p>	<p>Lesson 5: Patterning Why Use Patterns? Reference Geometry Linear Pattern Circular Patterns Mirror Patterns Using Pattern Seed Only Sketch Driven Patterns</p> <p>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</p> <p>Lesson 7: Shelling and Ribs Shelling and Ribs Analyzing and Adding Draft Other Options for Draft Shelling Ribs Full Round Fillets Thin Features</p> <p>Lesson 8: Editing: Repairs Part Editing Editing Topics Sketch Issues FilletXpert DraftXpert</p> <p>Lesson 9: Editing: Design Changes Part Editing Design Changes Information From a Model Rebuilding Tools Sketch Contours Editing with Instant 3D</p>	<p>Lesson 10: Configurations Configurations Using Configurations Creating Configurations Link Values Equations Configure Dimension / Feature Modelling Strategies for Configurations Editing Parts that Have Configurations Design Library</p> <p>Lesson 11: Using Drawings More About Making Drawings Section View Model Views Broken View Detail Views Drawing Sheets and Sheet Formats Projected Views Annotations</p> <p>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 Using Part Configurations in Assemblies Sub-assemblies Smart Mates Inserting Sub-assemblies Pack and Go</p> <p>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</p> <p>Appendix A: Templates Options Settings Document Templates</p>	