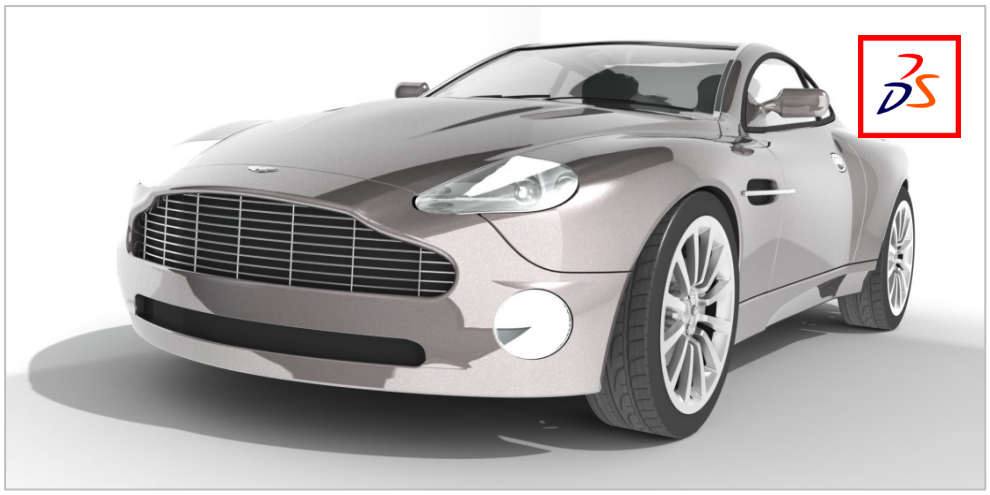
Introduction to SolidWorks

Training course outline

SolidWorks 3D design software helps teams around the world bring their ideas to life. It is easy to learn and use, letting you concentrate on your designs, not your CAD software.

Introduction to SolidWorks training provides a thorough grounding in SolidWorks for beginners. On completion, you will be able to use SolidWorks to build parametric models of parts and assemblies.



Course summary

This course teaches the basic principles of 3D part and assembly modelling in SolidWorks. Sessions include:

- SolidWorks basics
- Introduction to sketching
- Basic part modelling
- Patterning
- Shelling and ribs
- Editing: Design changes
- Configuration
- Using drawings
- Assembly modelling
- Templates

Duration

Three days.

Who should attend?

Newcomers to SolidWorks.

Pre-requisites

No previous CAD experience is required. Some mechanical design experience is beneficial but not essential.

In-class or live online

You can attend course in-person at any of our centres, or participate online from your place of work or home.

To read about our approach to online training, see armada.co.uk/live-online-training.

General information

Courses are hosted by highly experienced trainers from an engineering background with expert knowledge of SolidWorks

Introduction to SolidWorks training is arranged on request, i.e. one-to-one training or a course for your group. This means that the training can be:

- Provided when it suits you.
- Adapted to reflect your work

Whilst attending training at our centres, delegates have the use of a computer running SolidWorks to practice the techniques taught. Refreshments and lunch are provided.

Course fees can be paid by card or bank transfer. We accept purchase orders from UK-registered companies and public sector organisations.

If you're self-funding your training, you can pay in staged payments, interest-free, over 12 months.

Method of delivery

Training is designed for the busy professional, being short and intensive and combining lecture and demonstration. Practical exercises carried out under guidance help delegates to learn the techniques taught.

Delegates have ample opportunity to discuss specific requirements with the trainer.

Course materials and certificate

Delegates receive:

- A comprehensive training guide for SolidWorks.
- An e-certificate (PDF) confirming successful course completion.

After course support

Following SolidWorks training, you're entitled to 30 days' email support from your trainer.

Further information

For further details see armada.co.uk/course/solidworks-training. For a quote and details of our availability, please contact us.

Course syllabus

See over.

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SolidWorks

Course syllabus

Topics	Sub-topics
SolidWorks basics	What is SolidWorks? File references Opening files The SolidWorks user interface Using the Command Manager
Introduction to sketching	2D sketching Stages in the process What are we going to sketch? Sketching and sketch entities Basic sketching Rules that govern sketches Design intent Sketch relations Dimensions Extrude Sketching guidelines
Basic part modelling	Design intent Terminology Choosing the best profile Choosing the sketch plane Details of the part Extrude feature Revolved features Sketching on a planar face Cut feature Using the hole wizard View options Filleting Editing tools Detailing basics Drawing views Centre marks Dimensioning Changing parameters Edit material Mass properties File properties
Patterning	Why use patterns? Reference geometry Linear, circular and mirror patterns Using pattern seed only Sketch driven patterns

Topics	Sub-topics
Shelling and ribs	Shelling Planes Ribs Full round fillets Thin features
Editing: Design changes	Part editing Design changes Information from a model Rebuilding tools Sketch contours
Configuration	Creating configurations Using configure dimension/feature Using global variables, equations Global variables Equations Modelling strategies for configurations Editing parts that have configurations Design library
Using drawings	More about making drawings Section, model, broken and detail views Drawing sheets and sheet formats Projected views Annotations
Assembly modelling	Creating a new assembly Position of the first component FeatureManager design tree and symbols Adding components Using part configurations in assemblies Subassemblies Smart mates Inserting subassemblies Pack and go Analysing the assembly Checking for clearances Changing the values of dimensions Bill of materials Assembly drawings
Templates	Options settings Document templates