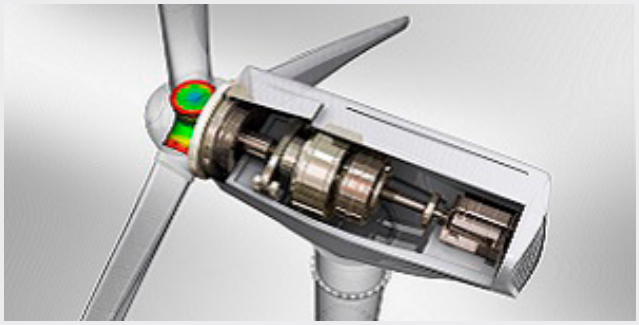
man machine

AUTODESK
Authorised Training Centre

Autodesk Inventor Introduction to Solid Modelling

Course Description

This course teaches new Inventor users how to approach parametric design using Autodesk Inventor. The course will provide a hands-on and practice-intensive learning experience ensuring that you will acquire the knowledge needed to design models from conceptual sketching, through to solid modelling, assembly design, and drawing production.



Prerequisites

Inventor Introduction to Solid Modelling does not assume prior knowledge of any 3D modelling or CAD software, although a background in drafting of 3D parts is recommended, plus an understanding of MS-Windows.

Topics Include

- Creating, constraining and dimensioning 2D sketches
- Creating and editing the solid base 3D feature from a sketch
- Creating and editing secondary solid features that are sketched and placed
- Creating equations and working with parameters
- Manipulating the display of the model
- Resolving feature failures
- Duplicating geometry in the model
- Placing and constraining/connecting assembly parts
- Manipulating display of components in an assembly
- Obtaining model measurements
- Obtaining property information
- Creating Presentations (Exploded views)
- Modifying and analyzing components in an assembly
- Simulating motion in an assembly
- Creating parts and features in an assembly
- Creating and editing an assembly
- Bill of Materials
- Working with projects
- Creating and annotating drawings and views
- Customizing the Autodesk Inventor environment

Course Duration: 2 + 2 Days

Next Step: Inventor Sheet Metal Design

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