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Autodesk Inventor Introduction to Solid Modeling



Course Length: 5 days

The Inventor Introduction to Solid Modeling training guide instructs students in the best usage approaches for parametric design philosophy through a hands-on, practice-intensive curriculum. Students acquire the knowledge needed to complete the process of designing models from conceptual sketching, through to solid modeling, assembly design, and drawing production.

The main topics covered in the course include:

- The Autodesk® Inventor® software interface
- Creating 2D sketches
- Constraining and dimensioning sketches
- Generating 3D parts from sketches
- Part modeling, adding, and editing 3D features
- Work Features
- Creating equations and working with parameters
- Model geometry and model display manipulation
- Resolving feature failures
- Feature duplication techniques
- Establishing and working with model relationships
- Placing and constraining parts in assemblies
- Assembly component display
- Model Information

For the current course
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- Presentation files (Exploded views)
- Assembly tools
- Creating parts and features in assemblies
- Assembly bill of materials
- Working with projects
- Creating and annotating drawings and views
- Customization

Prerequisites:

As an introductory training guide, Inventor Introduction to Solid Modeling does not assume prior knowledge of any 3D modeling or CAD software. Students do need to be experienced with the Windows operating system and a background in drafting of 3D parts is recommended.

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