Inventor®

Introduction to Solid Modeling

The Autodesk® Inventor® Introduction to Solid Modeling course provides students with an understanding of the parametric design philosophy through a hands-on, practice-intensive curriculum. Students will learn the key skills and knowledge required to design models using Autodesk® Inventor®, starting with conceptual sketching, through to solid modeling, assembly design, and drawing production.

TOPICS OF DISCUSSION:

- Understanding the Autodesk® Inventor® software interface
- 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 parts in assemblies
- Manipulating the display of components in an assembly
- Obtaining model measurements and property information
- Creating Presentation files (Exploded views)
- Modifying and analyzing the components in an assembly
- Simulating motion in an assembly
- Creating parts and features in assemblies
- Creating and editing an assembly Bill of Materials
- Working with projects
- Creating and annotating drawings and views
- Customizing the Autodesk® Inventor® environment

PREREQUISITES:

- Experienced with Windows® Operating System
- Background in drafting of 3D Parts is recommended

- Instructor Led Training
- 5 Days, 9 am – 4 pm

Authorized Training Center:

1128 Roosevelt Avenue
Suite 100
York, PA 17404

Customized and on-site training available upon request.

To request a training, contact a Print-O-Stat specialist by phone at 1-844-435-7479 or email software@printostat.com.

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