INVENTOR COURSE OUTLINE

Inventor is **computer-aided design (CAD) software** developed by Autodesk, many used for 3D mechanical design, simulation and visualization. It uses the concept of parametric design, used primarily to create technical drawings for mechanical purposes.

Autodesk Inventor can be used for **mechanical design, product simulation, and tooling creation**. It can aid you greatly in simulation and visualization even before your products are built. Inventor is a dimension driven CAD application that is used in engineering designs, visualization simulation, and documentation.

Duration: 3 Weeks at 2 hours per day ot one and Half week at 4 hours per day

Reg Fee: 500, Once

COST: 10,000/=

Inventor boosts the following greta features:

Model-based definition

Include manufacturing information in your 3D model for downstream applications.

Content center

Choose your standard components from a comprehensive customizable library.

Shape Generator

Create and evaluate high-performing design options in minutes.

Flexible modeling

Use the right modeling tool for every job with parametric, freeform, and direct modeling tools.

Direct modeling

Use easy push/pull controls to move, rotate, resize, or scale features from imported geometries.

Freeform modeling

Freely sculpt the shape of your design by moving points, edges, and faces.

Mechanical concept and layout design

Open DWG™ files directly inside Inventor as the basis for your 3D model.

Component generators and calculators

Use built-in calculators to inform the design of common joints such as welds, clamps, and press fits.

Tube and pipe design

Use a combination of automated tools and full-control design functions in Inventor to build tube and pipe runs.

Dynamic simulation

Apply forces to evaluate the motion, speed, and acceleration of your design.

Stress analysis

Run quick checks on parts or perform in-depth analysis of the entire product at any stage.

Exploded views and animations

Use exploded views and animations of complex assemblies in product documentation, manuals, and assembly instructions.

Autodesk Inventor Essentials Training

Introduction:

Autodesk Inventor Essentials Training is designed for new and beginner Autodesk Inventor users who want to learn the essential tool and principles of 3D parametric part design, assembly design and creating production-ready part assembly drawings using Autodesk Inventor. The course is designed to give students an excellent foundation to understand and familiarize with the features and commands of Inventor.

Course Details	
Autodesk Inventor Essential Training	
Objective	Utilize the Design Support system and on line help Create, Constrain, and Edit sketched features Effectively work with Construction Planes, Axes and Points Fillets, Chamfers, Shells and Patterns Create Views, Sheets, Title blocks, and edit part drawings Use Assembly modeling to Create, Place and Constrain components Design and sketch adaptive parts with features and occurrences Manage model data using Templates and Design Assistant Exchange model data by Linking, Translating and Importing
Prerequisite	A working knowledge of basic design/drafting procedures and terminology. Working knowledge of Microsoft® Windows

Section 1: Introduction to Autodesk Inventor Autodesk Inventor Fundamentals Autodesk Inventor Interface Model Manipulation Chapter exercise	**Section 2: Creating the Base Feature** Creating a New Part File Sketched Base Features Primitive Base Features Chapter exercise
Section 3: Sketching Geometry Sketch Geometry Chapter exercise	**Section 4: Additional Sketching Tools** Advanced Editing Tools Rectangular Sketch Patterns Circular Sketch Patterns Over-Dimensioned Sketches Sketch Preferences Chapter exercise
Section 5: Sketched Secondary Features Extruded Secondary Features Revolved Secondary Features Using Existing Geometry Editing Sketched Secondary Features 3D Grip Modification Chapter exercise	**Section 6: Creating Pick and Place Features** Edge Chamfer Constant Fillets Variable Fillets Face Fillets Full Round Fillets Straight Holes Threads Editing Pick and Place Features Creation Sequence Chapter exercise
Section 7: Work Features Work Planes Work Axes Work Points Chapter exercise	**Section 8: Equations** Equations Parameters Chapter exercise

Section 9: Additional Features	**Section 10: Model and Display Manipulation**
Face Draft Splitting a Face or Part Shells Ribs Bend Part Chapter exercise	Reordering Features Inserting Features Suppressing Features Section Views Design Views Chapter exercise

Section 11: Fixing Problems	**Section 12: Sweep Features**
Sketch Failure Feature Failure Chapter exercise	Sweep Features Chapter exercise
Section 13: Loft Features	**Section 14: Duplication Tools**
Rail Lofts Center Line Lofts Advanced Loft Options Chapter exercise	Rectangular Feature Patterns Circular Feature Patterns Mirror Parts or Features Manipulate Patterns and Mirror Features Chapter exercise
Section 15: Feature Relationships	**Section 16: Assembly Environment**
Establishing Relationships Controlling Relationships Investigating Relationships Changing Relationships Chapter exercise	Assembling Components using Constraints Content Center Assembly Browser Saving Files Chapter exercise

Section 17: Joint Connections Assembling Components using Joints Chapter exercise	**Section 18: Manipulating Assembly Display** Moving and Rotating Assembly Components Suppressing Constraints Component Display Selection Options in Assemblies Chapter exercise

Section 19: Model Information Measurement Tools Model Properties Chapter exercise	**Section 20: Design Presentation and Animation** Exploded View Presentations Chapter exercise

SEction 21: Assembly Tools Replacing Components Restructuring Components Driving Constraints Contact Solver Interference Error Recovery Chapter exercise	**Chapter 22: Assembly Parts and Features** Assembly Parts Assembly Features Chapter exercise
Chapter 23: Assembly Bill of Materials Create Virtual Components Create Bill of Materials Chapter exercise	**Section 24: Working With Projects** New Projects Resolving Links The Vault Browser Chapter exercise

Section 25: Drawing Basics	Section 26: Detailing Drawings
New Drawing Views Manipulating Views Chapter exercise	Dimensions Drawing Sheets Parts List Balloons Styles and Standards Hatching Chapter exercise
Section 27: Drawing Annotations Text Symbols Hole and Thread Notes Chamfer Notes Center Marks and Center Lines Hole Tables Revision Tables and Tags Chapter exercise	**Section 28: Customizing Autodesk Inventor** Application Options Document Settings File Properties Changing Part Units Command Customization Chapter exercise