Autodesk Inventor Essentials

Target Audience

The target audience for Autodesk Inventor Essentials includes mechanical engineers, product designers, CAD drafters, and manufacturing professionals who are new to Autodesk Inventor or transitioning from other CAD software. It is also ideal for students and educators in engineering and design fields, as well as professionals seeking to enhance their skills in 3D modeling, simulation, and design automation. This course caters to individuals aiming to create precise and innovative designs, assemblies, and technical drawings efficiently using Inventor's foundational tools and workflows.

Course Objective

Upon completion of this Autodesk Inventor Essentials course, participants will acquire a comprehensive understanding of the software, enabling them to proficiently utilize its powerful features for 3D design, simulation, and documentation. Participants will gain hands-on experience in creating parametric models, assemblies, and detailed drawings, as well as exploring advanced functionalities such as dynamic simulation, stress analysis, and parametric design automation.

Course Outcome

By the end of the course, students will be equipped with the skills needed to efficiently design, visualize, and analyze complex engineering projects using Autodesk Inventor, positioning them for success in various industries that rely on cutting-edge 3D design and engineering solutions.

Course Outline: The course comprises **40-hours** of theory and labs and is divided into **15** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Chapter 1. Introduction to Inventor

Installing Autodesk Inventor
Getting Started with Autodesk Inventor
Starting a New Inventor File
Invoking the Part Modeling Environment

- Ribbon
- File Menu
- Quick Access Toolbar
- InfoCenter
- Browser
- ViewCube
- Navigation Bar
- Status Bar

Invoking the Assembly Environment
Invoking the Drawing Environment
Invoking the Presentation Environment
Identifying Inventor Files
Creating and Accessing a Project
Invoking a Marking Menu
Customizing the Color Scheme
Choosing the User Interface Theme
Exporting Files to Other CAD Formats

Chapter 2. Drawing Sketches with Autodesk Inventor

Invoking the Part Modeling Environment
Invoking the Sketching Environment
Working with the Selection of Planes
Specifying Units
Specifying Grids and Snap Settings
Creating a Line Entity
Creating an Arc by using the Line Tool
Creating a Circle

- Creating a Circle by using the Center Point Circle Tool
- Creating a Circle by using the Tangent Circle Tool

Creating an Ellipse
Creating an Arc

- Creating an Arc by using the Three Point Arc Tool
- Creating an Arc by using the Tangent Arc Tool

- Creating an Arc by using the Center Point Arc Tool

Creating a Rectangle

- Creating a Rectangle by using the Two Point Rectangle Tool
- Creating a Rectangle by using the Three Point Rectangle Tool
- Creating a Rectangle by using the Two Point Center Rectangle Tool
- Creating a Rectangle by using the Three Point Center Rectangle Tool

Creating a Slot

- Creating a Slot by using the Center to Center Slot Tool
- Creating a Slot by using the Overall Slot Tool
- Creating a Slot by using the Center Point Slot Tool
- Creating a Slot by using the Three Point Arc Slot Tool
- Creating a Slot by using the Center Point Arc Slot Tool

Creating a Polygon

Creating a Spline

- Creating a Spline by using the Control Vertex Spline Tool
- Creating a Spline by using the Interpolation Spline Tool
- Creating an Equation Driven Spline

Creating a Bridge Curve

Editing a Spline

- Editing an Interpolation Spline
- Editing a Control Vertex Spline

Chapter 3. Editing and Modifying Sketches

Trimming Sketch Entities

- Trimming Sketch Entities Up to their Nearest Intersection
- Trimming Sketch Entities by Defining Boundaries
- Trimming Sketch Entities Dynamically

Extending Sketch Entities

- Extending Sketch Entities Up to their Nearest Intersection
- Extending Sketch Entities by Defining Boundaries

Splitting Sketch Entities

Offsetting Sketch Entities

Creating a 2D Fillet

Creating a 2D Chamfer

Patterning Sketch Entities

- Creating a Rectangular Pattern
- Creating a Circular Pattern

Creating Construction and Centerline Entities
Mirroring Sketch Entities
Moving a Sketch Entity
Creating a Copy of Sketch Entities
Rotating an Entity
Scaling Sketch Entities
Stretching Sketch Entities

Chapter 4. Applying Constraints and Dimensions

Applying Constraints

- Applying Horizontal Constraint
- Applying Vertical Constraint
- Applying Coincident Constraint
- Applying Collinear Constraint
- Applying Concentric Constraint
- Applying Parallel Constraint
- Applying Perpendicular Constraint
- Applying Tangent Constraint
- Applying Smooth Constraint
- Applying Symmetric Constraint
- Applying Equal Constraint
- Applying Fix Constraint

Controlling the Display of Contraints
Controlling Constraint Settings
Applying Dimensions

- Applying a Horizontal Dimension
- Applying a Vertical Dimension
- Applying an Aligned Dimension
- Applying an Angular Dimension
- Applying a Diameter Dimension
- Applying a Radius Dimension
- Applying a Linear Diameter Dimension

Controlling Dimension Settings
Modifying/Editing Dimensions
Working with Different States of a Sketch

- Under Constrained Sketch
- Fully Constrained Sketch

Displaying Available Degrees of Freedom

Chapter 5. Creating Base Feature of Solid Models

Creating an Extrude Feature
Creating a Revolve Feature
Navigating a 3D Model in Graphics Area

- Navigating a 3D Model by Using the Navigation Tools
- Navigating a 3D Model by Using the Mouse Buttons
- Navigating a 3D Model by Using the ViewCube

Changing the Visual Style of a Model

- Realistic
- Shaded
- Shaded with Edges
- Shaded with Hidden Edges
- Wireframe
- Wireframe with Hidden Edges
- Wireframe with Visible Edges Only
- Monochrome
- Watercolor
- Sketch Illustration
- Technical Illustration

Chapter 6. Creating Work Features

Creating Work Planes

- Creating a Work Plane Through Selected Geometries
- Creating a Work Plane at an O set Distance
- Creating a Work Plane Parallel to a Face/Plane
- Creating a Work Plane in the Middle of Two Faces/Planes
- Creating a Work Plane through the Midplane of a Torus
- Creating a Work Plane at an Angle
- Creating a Work Plane Passing Through Three Points
- Creating a Work Plane Passing Through Two Coplanar Edges
- Creating a Work Plane Tangent to a Face through an Edge
- Creating a Work Plane Tangent to a Face through a Point
- Creating a Work Plane Tangent to a Face and Parallel to a Plane
- Creating a Work Plane Normal to an Axis Through a Point
- Creating a Work Plane Normal to a Curve

Creating Work Axes

- Axis Tool

- On Line or Edge Tool
- Parallel to Line through Point Tool
- Through Two Points Tool
- Intersection of Two Planes Tool
- Normal to Plane through Point Tool
- Through Center of Circular or Elliptical Edge Tool
- Through Revolved Face or Feature Tool

Creating Work Points

- Point Tool
- Grounded Point Tool
- On Vertex, Sketch point, or Midpoint Tool
- Intersection of Three Planes Tool
- Intersection of Two Lines Tool
- Intersection of Plane/Surface and Line Tool
- Center Point of Loop of Edges Tool
- Center Point of Torus Tool
- Center Point of Sphere Tool

Creating a User Coordinate System

Chapter 7. Creating Work Features

Using Advanced Options of the Extrude Tool
Using Advanced Options of the Revolve Tool
Working with a Sketch having Multiple Profiles

- Projecting Geometries
- Projecting Geometries onto the Sketching Plane
- Projecting Intersecting Edges onto the Sketching Plane
- Projecting 2D Sketch onto a Face
- Projecting Geometries of a .DWG File

Creating a Section View

- Creating a Half Section View
- Creating a Quarter Section View
- Creating a Three-Quarter Section View

Editing a Feature and its Sketch

- Editing a Feature and its Sketch

Displaying Earlier State of a Model
Re-ordering Features of a Model
Measuring the Distance between Entities
Assigning an Appearance

- Assigning Appearance by using the Appearance Tool
- Assigning Appearance by using the Appearance Drop-down List
- Copying and Pasting an Appearance by Using the Adjust Tool

Applying a Material

- Applying a Material by Using the Material Tool
- Applying a Material by Using the Material Drop-down List

Assigning Finishes/Manufacturing Data
Calculating Physical Properties

Chapter 8. Advanced Modeling – II

Creating a Sweep Feature

- Creating a Sweep Feature Along a Path
- Creating a Sweep Feature Along a Path with Fixed Orientation
- Creating a Sweep Feature Along a Path and a Guide Rail
- Creating a Sweep Feature Along a Path and a Guide Surface
- Creating a Sweep Feature by Sweeping a Toolbody Along a Path

Creating a Loft feature

- Creating a Loft Feature with Sections
- Creating a Loft feature with Sections and Guide Rails
- Creating a Loft feature with Sections and a Centerline
- Creating an Area Loft feature

Creating a Coil feature
Creating an Emboss Feature
Creating a Mark Feature
Creating a Rib Feature

- Creating a Rib Feature Normal to the Sketching Plane
- Creating a Rib Feature Parallel to the Sketching Plane

Applying an Image on a Face of a Model
Creating a Shell Feature

Chapter 9. Patterning and Mirroring

Creating a Rectangular Pattern
Creating a Circular Pattern
Creating a Sketch Driven Pattern
Suppressing Features and Pattern Occurrences

Unsuppressing Features and Pattern Occurrences
Mirroring a Feature or a Body

Chapter 10. Advanced Modeling - III

Creating Holes
Creating Threads
Creating Fillets
- Creating Edge Fillets
- Creating Face Fillets
- Creating Full Round Fillets

Creating Chamfers
Splitting a Face and a Solid Body
- Splitting a Faces of a Model
- Splitting a Solid Body

Creating 3D Sketches and Curves
- Creating a 3D Sketch by Using the Line Tool
- Creating a Helical Curve
- Creating a 3D Intersection Curve
- Creating a Silhouette Curve
- Creating a 3D Projected Curve
- Creating a 3D Curve on a Face
- Creating 3D Curves by Projecting Existing Geometries

Chapter 11. Working with Assemblies – I

Working with Bottom-up Assembly Approach
Working with Top-down Assembly Approach
Creating an Assembly using Bottom-up Approach
Inserting Components in the Assembly Environment
Working with Degrees of Freedom
Applying Constraints
- Applying Assembly Constraints
- Applying Motion Constraints
- Applying Transitional Constraints
- Applying Constraint Set Constraints

Applying Joints
- Applying a Rigid Joint
- Applying a Rotational Joint
- Applying a Slider Joint

- Applying a Cylindrical Joint
- Applying a Planar Joint
- Applying a Ball Joint

Editing Constraints and Joints
Deleting Constraints and Joints
Moving and Rotating Individual Components

- Moving a Component by using the Free Move Tool
- Rotating a Component by using the Free Rotate Tool

Chapter 12. Working with Assemblies – II

Creating an Assembly by using the Top-down Approach
Editing Assembly Components

- Editing Assembly Components within the Assembly Environment
- Editing Assembly Components in the Part Modeling Environment

Patterning Assembly Components

- Creating an Associative Pattern
- Creating a Rectangular/Circular Pattern

Mirroring Components of an Assembly
Copying Components of an Assembly
Creating Bill of Material (BOM) of an Assembly

Chapter 13. Creating Animation and Exploded Views

Invoking the Presentation Environment
Capturing Actions on the Timeline
Capturing Tweaks on the Timeline
Editing Time and Properties of a Tweak
Deleting a Tweak
Creating a Snapshot View

- Creating a Linked Snapshot View
- Creating an Independent Snapshot View

Editing a Snapshot View
Renaming a Snapshot View
Deleting a Snapshot View
Publishing a Snapshot View to a Raster Image
Creating an Exploded View in a Drawing File
Creating a New Storyboard
Creating a New Scene
Playing Animation of a Storyboard

Publishing Animation to a Video File

Chapter 14. Working with Drawings

Invoking the Drawing Environment
Editing the Sheet Size
Editing/Creating the Title Block
Editing the Drafting Standard
Creating the Base View of a Model
Creating Projected Views
Working with Angle of Projection
Defining the Angle of Projection
Creating Other Drawing Views

- Creating an Auxiliary View
- Creating a Section View
- Creating a Detail View
- Creating an Overlay View
- Creating a Break View
- Creating a Break Out View
- Creating a Slice View
- Creating a Crop View

Deleting a Drawing View
Applying Dimensions

- Applying Drawing Dimensions
- Applying Model Dimensions

Adding a Text/Note
Adding a Surface Finish Symbol
Adding a Weld Symbol
Adding a Hole and Thread Note
Adding the Parts List / Bill of Materials (BOM)
Editing the Parts List / Bill of Materials (BOM)
Adding Balloons

- Adding Balloons Automatically
- Adding Balloons Manually

Chapter 15. Sheet Metal Design and Simulation

Modify sheet metal defaults
Create a face feature
Create a Flange
Create a Contour Flange
Create a Corner Seam
Create Punches
Create a Bend Feature
Create Corner Rounds
Flat Patterns
Starting a stress analysis
Defining parts materials
Constraints for your part
Defining Input Forces
Run the stress analysis