Autodesk Inventor Advanced Part Modeling (IGTIAP)

Course Length	2 Days

This Course is mainly intended for students who acquired Inventor Basic training looking forAdvanced Commands in part Design when designing using Autodesk Inventor

Training Guide Contents

1. Tips & Tools
- 1.1 Design Philosophies
- 1.2 Sketching Tips
- 1.3 Display Options
- 1.4 Appearances

2. Sketching Tools
- 2.1 Splines
- 2.2 3D Sketches

3. Multi-Body Part Modeling
- 3.1 Multi-Body Part Modeling

4. Advanced Work Features
- 4.1 Grounded Work Points
- 4.2 User Coordinate Systems

5. Advanced Lofts, Sweeps, and Coils
- 5.1 Area Lofts
- 5.2 Advanced Sweeps
- 5.3 Coils

6. Analyzing a Model
- 6.1 Analysis Types
- 6.2 Analysis Procedures

7. Generative Shape Design
- 7.1 Shape Generator

8. Introduction to Surfacing
- 8.1 Introduction to Surfaces
- 8.2 Basic Surfaces
- 8.3 Patch Surfaces
- 8.4 Ruled Surfaces
- 8.5 Stitch Surfaces
- 8.6 Sculpting with Surfaces
- 8.7 Thickening & Offsetting a Surface

9. Additional Surfacing Options
- 9.1 Extend and Trim Surfaces
- 9.2 Replace Face with a Surface
- 9.3 Delete Faces
- 9.4 Copy Surfaces

10. Copying Between Parts (iFeatures)
- 10.1 Creating iFeatures
- 10.2 Inserting iFeatures
- 10.3 iFeatures vs. Copy Feature
- 10.4 Table-Driven iFeatures
- 10.5 Editing iFeatures

11. iParts
- 11.1 iPart Creation
- 11.2 iPart Placement
- 11.3 Editing an iPart Factory
- 11.4 Creating iFeatures from a Table-Driven iPart
- 11.5 Tables for Factory Members

12. Importing & Editing CAD Data
- 12.1 Importing CAD Data
- 12.2 Exporting Geometry
- 12.3 Editing the Base Solid
- 12.4 Direct Edit
- 12.5 Attaching Point Cloud Data

13. Working with Imported Surfaces
- 13.1 Importing Surfaces
- 13.2 Repairing Imported Surfaces

14. Working with AutoCAD Data
- 14.1 Opening AutoCAD Files
- 14.2 DWG File Underlays
- 14.3 Working with other Autodesk Product Files

15. Introduction to Freeform Modeling
- 15.1 Creating Freeform Geometry
- 15.2 Editing Freeform Geometry

Autodesk Inventor Advanced Assembly Modeling (IGTIAA)

Course Length	2 Days

This Course is mainly intended for students who acquired Inventor Basic training and Advanced Part Modeling looking for higher level of productivity when creating and working with assemblies

Training Guide Contents

1. **Working Effectively with Assemblies**
   - 1.1 General Assembly Tips
   - 1.2 Relationship Tips
   - 1.3 Motion Constraints
   - 1.4 Transitional Constraints
2. **Introduction to Top-Down Design**
   - 2.1 Top-Down Design Process
   - 2.2 Top-Down Design Tools
3. **Derived Components**
   - 3.1 Derived Components
   - 3.2 Modify Derived Components
4. **Multi-Body Part Modeling**
   - 4.1 Multi-Body Part Modeling
5. **Layout Design**
   - 5.1 Layout Design
6. **Associative Links and Adaptive Parts**
   - 6.1 Associative Links
   - 6.2 Adaptive Assembly Parts
7. **iMates**
   - 7.1 iMates
8. **Positional Representations**
   - 8.1 Introduction to Positional Representations
   - 8.2 Create and Edit Positional Representations
   - 8.3 Use Positional Representations
9. **Model Simplification**
   - 9.1 Shrinkwrap
   - 9.2 Assembly Simplification

10. **Level of Detail Representations**
    - 10.1 Level of Detail Representations
    - 10.2 System-Defined Level of Detail Representations
    - 10.3 User-Defined Level of Detail Representations
    - 10.4 Using Level of Detail Representations
    - 10.5 Substitute Level of Detail Representations
    - 10.6 LOD Productivity Tools
11. **Design Accelerator**
    - 11.1 Design Accelerator
    - 11.2 Generators
    - 11.3 Calculators
    - 11.4 Engineer's Handbook
12. **Advanced File Management**
    - 12.1 Design Assistant
    - 12.2 Pack and Go
    - 12.3 Purging Old Files
13. **Inventor Studio**
    - Rendering
    - Animation
    - Video Producer
    - Creating a Standard Room
14. **iAssemblies**
    - Introduction
    - Create Basic iAssemblies
    - Create Multi-Level iAssemblies
    - Create iAssemblies Using Existing Assemblies
    - Place iAssemblies
    - Edit iAssemblies
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Autodesk Inventor Tube & Pipe Design (IGTITP)

Course Length	2 Days

This Course is designed for experienced Autodesk Inventor users. Those who attending the training should have completed the Autodesk Inventor Basic training or have an equivalent understanding of the Autodesk Inventor software user interface and working environment. This course is mainly intended for designing Tube & Pipe in Inventor software

Training Course Contents

1. Introduction to Tube and Pipe
- Tube and Pipe Designs
- Tube and Pipe Environment
- Setup for Routes and Runs
- Tube and Pipe Files
- Creating a Tube and Pipe Assembly
- Tube and Pipe Styles and Content
- Placing Library & Authored Fittings
- Connecting and Editing Fittings

2. Routes and Runs
- Creating Rigid Routes
- Autoroute Regions
- Creating Derived Routes
- Defining Gravity for a Self-Draining Route
- 3D Orthogonal Route Tool
- Placing Points Along An Axis
- Creating Bends in Routes
- Snapping to Points and Rotations
- Aligning the Axes
- Construction Lines
- Editing Routes
- Controlling Tube and Pipe Updates
- Adding and Changing Fittings
- Applying a New Style
- Deleting Routes and Runs
- Creating Flexible Hose Routes
- Inserting Nodes
- Editing Hose Routes
- Check Minimum Bend Radius
- Reuse of Routes and Runs

3. Fittings and Components
- Adding a Custom Library
- Configuring the Content Center File Path
- Copying Content to a Custom Library
- Creating Custom Content
- Authoring Tube and Pipe Content
- Publishing Tube and Pipe Content
- Document Settings for Published Parts
- Adding and Editing Family Column Values
- Adding a Custom Display Name
- Creating a New Style
- Configuring a Self-Draining Style
- Configuring Styles in Templates

4. Documenting Tube and Pipe Assemblies
- Creating, Saving, and Activating Level of Detail Representations
- Creating a Substitute Level of Detail
- Tube and Pipe Design
- Creating Drawing Views
- Including Route Centerlines
- Customizing a Parts List
- Creating Bending Machine Files
- Exporting ISOGEN Files

Autodesk Inventor Professional Simulation (IGTIS)

Course Length	2 Days

This course is designed to introduce users about the fundamentals of Inventor Professional's simulation tools (FEA). The following below topics were covered under the course. People who attending this course should have Completed the Inventor Fundamental Course.

Training Course Contents

1. **Essentials Understanding FEA Creating a Simulation Exploring the FEA User Interface Setting Up a FEA Analysis**

   - Understanding File Structure
   - Assigning Materials
   - Managing Material Libraries
   - Applying Constraints
   - Applying Basic Loads
   - Applying Advanced Loads
   - Assigning Prescribed Displacements
   - Creating a Mesh Running a Simulation

2. **Interpreting results**
   - Generating Stress Plots
   - Generating Displacement Plots
   - Determining Safety Factor
   - Finding Maximum and Minimum Results
   - Probing Results
   - Animating Results
   - Configuring Display Settings
   - Generating a Report
   - Copying a Simulation.

3. **Dynamic simulation Environment**
   - Navigating the Dynamic Simulation Environment
   - Creating Joints
   - Convert Assembly constraints to joints
   - Insert Joints - standard Joints
   - Rolling Joints - sliding joints
   - 2D contact Joints
   - Force Joints
   - Screw and worm Joints
   - Joint Properties
   - edit joints

4. **Mechanism status and redundancies**
   - Color mobile groups
   - Creating Forces and Torques

5. **Dynamic simulation player**
   - Player settings
   - Dynamic Part Motion
   - Using the Output Grapher
   - Creating AVI animation
   - Detecting Collision During Operation
   - Export Dynamic Simulation Results to FEA
   - Analyzing and Interpreting Results

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