DATAK ENGINEERING COURSE MODULE

AUTODESK INVENTOR

GETTING STARTED WITH AUTODESK INVENTOR

- Starting Autodesk Inventor
- User Interface
- Ribbon
- File Menu
- Quick Access Toolbar
- Browser window
- Status bar
- Navigation Bar
- View Cube
- Shortcut Menus and Marking Menus
- Dialogs
- Mini toolbar
- Customizing the Ribbon, Shortcut Keys, and Marking Menus
- Color Settings

PART MODELING BASICS

- Creating a New Project
- Starting a New Part File
- Starting a Sketch
- Adding Dimensions
- Creating the Base Feature
- Adding an Extruded Feature
- Adding another Extruded Feature
- Saving the Part
- Starting a New Part File
- Sketching a Revolve Profile
- Creating the Revolved Feature
- Creating the Cut feature
- Creating another Cut feature
- Adding a Fillet
- Saving the Part
- Starting a New Part File
- Creating the Cylindrical Feature
- Creating Cut feature
- Saving the Part
- Start Extruded feature
- Applying Draft
- Saving the Part

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ASSEMBLY BASICS

- Top-Down Approach
- Bottom-Up Approach
- Starting a New Assembly File
- Inserting the Base Component
- Adding the second component
- Applying Constraints
- Adding the Third Component
- Checking the Interference
- Saving the Assembly
- Starting the Main assembly
- Adding Disc to the Assembly
- Placing the Sub-assembly
- Adding Constraints
- Placing the second instance of the Sub-assembly
- Saving the Assembly
- Starting a New Presentation File
- Creating a Storyboard Animation
- Animating the Explosion
- Taking the Snapshot of the Explosion

CREATING DRAWINGS

- Starting a New Drawing File
- Editing the Drawing Sheet
- Generating the Base View
- Generating the Section View
- Creating the Detailed View
- Creating Centermarks and Centerlines
- Retrieving Dimensions
- Adding additional dimensions
- Populating the Title Block
- Saving the Drawing
- Creating New Sheet Format
- Creating a Custom Template
- Starting a Drawing using the Custom template
- Adding Dimensions
- Creating a New Drawing File
- Generating Base View
- Generating the Exploded View
- Configuring the Parts list settings
- Creating the Parts list
- Creating Balloons
- Saving the Drawing

ADDITIONAL MODELING TOOLS

- Creating the First Feature
- Adding the Second feature
- Creating a Counterbore Hole
- Creating a Threaded hole
- Creating a Circular Pattern
- Creating Chamfers
- Creating the first feature
- Creating the Shell feature
- Creating the Third feature
- Creating a Cut Feature
- Creating the Rib Feature
- Creating the Coil
- Tutorial
- Creating the First Section and Rails
- Creating the second section
- Creating the Loft feature
- Creating the Extruded feature
- Creating the Emboss feature
- Mirroring the Emboss feature
- Creating Fillets
- Shelling the Model
- Adding Threads
- Creating a 3D Sketch
- Creating the Sweep feature
- Creating the Along Curve pattern
- Editing the Freeform Shape
- Create another Freeform box
- Start a new part file
- Creating the second feature
- Adding Threads
- Creating iParts
- Creating the First Feature
- Creating the Extruded surface
- Replacing the top face of the model with the surface
- Creating a Face fillet
- Creating a Variable Radius fillet
- Shelling the Model
- Creating the Boss Features
- Creating the Lip feature
- Creating the Grill Feature
- Creating Ruled Surface

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SHEET METAL MODELING

- Starting a New Sheet metal File
- Setting the Parameters of the Sheet Metal part
- Creating the Base Feature
- Creating the flange
- Creating the Contour Flange
- Creating the Corner Seam
- Creating a Sheet Metal Punch iFeature
- Creating a Punched feature
- Creating the Rectangular Pattern
- Creating the Bend Feature
- Applying a corner round
- Creating Countersink holes
- Creating Hem features
- Mirroring the Features
- Creating the Flat Pattern
- Creating 2D Drawing of the sheet metal part

TOP-DOWN ASSEMBLY AND JOINTS

- Creating a New Assembly File
- Creating a component in the Assembly
- Creating the Second Component of the Assembly
- Creating the third Component of the Assembly
- Adding Bolt Connections to the assembly
- Applying the constraint to the components
- Using the Search tool in the Bowser window
- Editing Values in the Bowser window
- Changing the Display Preferences of the Bowser window
- Using the Measure tool
- Creating the Slider Joint
- Creating the Rotational Joint
- Creating the Rigid Joint
- Adding more assembly joints
- Driving the joints
- Creating Positions
- Creating 3D PDF

DIMENSIONS AND ANNOTATIONS

- Creating Centerlines and Centered Patterns
- Editing the Hatch Pattern
- Applying Dimensions
- Placing the Feature Control Frame
- Placing the Surface Texture Symbols
- Modifying the Title Block Information

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MODEL BASED DIMENSIONING

- Geometric Dimensioning and Tolerancing
- Adding Tolerances to the Model dimensions
- Extracting the Model dimensions
- Adding Tolerance Feature