1 Introduction to Autodesk Inventor®

The following section will give an overview of the Autodesk Inventor user interface and program basics in order to navigate the program as well as the part modeling environment.

1.1 File Types

The basic file references that exist in a typical Autodesk Inventor 3D design are represented in the following illustration.

1 Assembly files: *.iam files reference part files and are referenced by drawing files.
2 Part files: *.ipt files are referenced by assembly files and drawing files.
3 Drawing files: *.dwg files reference assembly files and part files.
4 Inventor Drawing files: *.idw files are interchangeable with *.dwg files in Inventor and reference assembly and part files.

Note: For the purposes of this section you will only be dealing with Part files.

1.2 User Interface

Autodesk Inventor uses a standard structure common in all Microsoft Windows applications. All environments share a common layout for tabs on a single toolbar across the top of the application window called the ribbon. The structure is context-sensitive based on the environment and mode you are using.

Below is the Model Tab used in the part modeling environment.
The following image illustrates the major areas of the Autodesk Inventor user interface.

1. Application Menu
2. Quick Access Toolbar
3. Ribbon
4. Ribbon Tabs
5. ViewCube
6. Navigation Bar
7. Browser
8. 3D Indicator
9. Graphics Window
Browser

The browser displays the Origin folder at the top of the list which contains the default X, Y, and Z planes, axes, and center point. There are two main versions of the browser depending on which operation you are preforming.

- The Model Browser displays all features you use to create the part.
- The Assembly Browser displays all the parts you use in the assembly.

Marking Menu and Overflow Menu

The Marking Menu is accessed by right-clicking anywhere in the graphics windows. The commands available change depending on the environment and command context. These commands indicate the next steps possible for the current process. The Marking Menu is offered as an alternative to selecting tools from the ribbon.
Although only eight items are available in the Marking Menu, there are many more options located below the Marking Menu in the Overflow Menu. These options are typically either specific to the operation being performed or new operations than can be started directly without exiting the current command first.
1.3 Viewing Tools

View manipulation is a key 2D drawing and 3D modeling skill. You are often required to view different areas of a design, and changing your view can help you visualize solutions for the current task. Many of the view manipulation tools are common to all environments.

Navigation Bar

Different view manipulation tools are available on the Navigation bar

1. ViewCube
2. SteeringWheel
3. Pan
4. View Face
Zoom Options

5. Zoom All

Orbit Options

6. Free Orbit
Function Key Shortcut Keys

<table>
<thead>
<tr>
<th>KEY</th>
<th>NAME</th>
<th>FUNCTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>F2</td>
<td>Pan</td>
<td>Pans the graphics window.</td>
</tr>
<tr>
<td>F3</td>
<td>Zoom</td>
<td>Zooms in or out in the graphics window.</td>
</tr>
<tr>
<td>F4</td>
<td>Rotate</td>
<td>Rotates objects in the graphics window.</td>
</tr>
<tr>
<td>F5</td>
<td>Previous View</td>
<td>Returns to the previous view.</td>
</tr>
<tr>
<td>F6</td>
<td>Isometric View</td>
<td>Display the isometric view of the model.</td>
</tr>
</tbody>
</table>

Mouse Viewing Tools

<table>
<thead>
<tr>
<th>MOUSE FUNCTION</th>
<th>NAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Click and Drag Mouse Wheel Button</td>
<td>Pan</td>
</tr>
<tr>
<td>Roll Mouse Wheel</td>
<td>Zoom</td>
</tr>
<tr>
<td>Shift + Click and Drag Mouse Wheel Button</td>
<td>Free Orbit</td>
</tr>
<tr>
<td>Double-Click Mouse Wheel Button</td>
<td>Zoom All</td>
</tr>
</tbody>
</table>
2 Project 1 - Clevis Mount Part

2.1 Instructions

The first project will introduce you to the Autodesk Inventor 2012 user interface and basic sketching and modeling commands. The main purpose of this project is to highlight component creation process and the interface of Autodesk Inventor.

1: Create a new part using the *Standard (mm).ipt* template.
   - On the *Quick Access* toolbar, click *New*.
   - In the *New File* dialog box, click the *Metric* tab.
   - Select *Standard (mm).ipt*.
Click OK.

2: Create a basic shape.
- Start the Line tool.
  **Sketch Tab | Draw Panel | Line**

- Select a start point near the origin.
  You will contain the sketch to the origin later.
Drag the cursor to the right, making certain the horizontal constraint glyph appears near the cursor. In the direct input field enter 150 mm for the line length. Press TAB to lock in the value. Press ENTER to accept this line segment.

A dimensional contain of 150 is added to the line segment.

With the Line tool still active, drag the cursor up, making certain the perpendicular constraint glyph appears near the cursor. In the direct input field enter 15 mm for the line length. Press TAB to lock in the value. Press ENTER to accept this line segment.

A dimensional contain of 15 is added to the line segment.

Drag the cursor back to the left, perpendicular to the last line segment, approximately 40 mm. Left-Click to accept line segment.
Drag the cursor up, perpendicular to the last line segment, approximately 50 mm. Left-Click to accept the line segment.

With the Line tool still active create an inline arc segment. Click-Hold the start point and Drag the endpoint of the line segment up and to the left to define the direction of tangency for the arc. Release the left mouse button when the endpoint of the arc is directly to the left of the start point.
Complete the sketch as shown

3: Investigate sketch.
   - Press the **F9** key to display the geometric contains that where automatically applied while sketching the line segments.

   - Click and drag the line segment, notice how they move.

   - Press **F9** to hide the geometric contain icons

4: Add geometric constraints to the sketch.
   - Add a Vertical geometric constraint.
     **Sketch Tab | Constrain Panel | Vertical Constraint**
■ Click the center point of the arc line segment, and then click the mid-point of the bottom line segment to place the Vertical constraint

Note: As you move the cursor along a line segment, the cursor will snap to the mid-point and will indicate the mid-point with a green circle.

■ Add a Collinear geometric Constraint.

Sketch Tab | Constrain Panel | Collinear Constraint

■ Click the two horizontal line segments shown to make them in line with each other.

■ Add a Coincident geometric Constraint.

Sketch Tab | Constrain Panel | Coincident Constraint
First click the Mid-Point of the bottom line segment and then click the projected Origin Center Point. The sketch will now be centered to the part Origin. By centering the sketch the part Origin planes can be used for features applied later.

5: Add dimensional constraints to the sketch.
   - Add a linear Dimensional constraint.

Sketch Tab | Constrain Panel | Dimension
■ Click bottom line segment and the arc segment center point. Enter 65 mm. Click OK

■ Add radial Dimensional constraint. Sketch Tab | Constrain Panel | Dimension

■ Click the arc segment. Enter 30 mm. Click OK

■ Notice the line colors have changed to indicate that the sketch is fully constrained.
Right-Click the Graphics Windows. Click **Finish 2D Sketch** from the Marking Menu to exit the sketch.

6: Reorient the Graphic Window using the Viewing Tools so the sketch looks like the image below.
- You can use the **View Cube** or **Orbit Tool** (F4) as optional Viewing Tools

7: Extrude the Base Feature.
- Start the **Extrude** Tool

![Extrude Tool Image]
Because there is only one closed loop exists in the sketch, the sketch profile is automatically selected

- Select the **Symmetric** direction from the Mini-Toolbar

- Drag the direct manipulation arrow, or enter **75 mm** into the Mini-Toolbar for the extrusion length. Click **OK**
8: Change Part Color
   - On the Quick Access Toolbar pick **Metal-Steel (Polished)** from the part color drop down list.

9: Create Secondary Feature
   - Right Click Graphics Windows, Click **New Sketch** on the Marking Menu
   - In the Browser expand the Origin folder. Click the YZ Plane to start the new sketch on this plane.
- Project reference part edges into the sketch at the point where the sketch cuts through the part. **Sketch Tab | Draw Panel | Project Cut Edges**

- Click **F7** (Slice Graphics) to show sketch.

- Draw a Two Point Rectangle  
  **Sketch Tab | Draw Panel | Rectangle**

- Pick top reference edge for the rectangle starting point. End **35 mm** and **65 mm** in the direct entry fields as shown. Click ENTER to except.

- Place a **Horizontal** Constraint  
  **Sketch Tab | Constrain Panel | Horizontal Constraint**

Between the mid-point of the rectangle to the projected Origin point as shown.
The sketch will now be fully constrained.

- Exit the Sketch.

10: Extrude cut through the part
- Start the Extrude Tool, set the following options:
  - Select the **Symmetric** direction
  - Select the **Cut** option from the Mini-Toolbar
Select the Through All option from the Mini-Toolbar.

11: Create a Hole Feature

- Start the Hole Tool
  - Model Tab | Modify Panel | Hole
  - In the Hole Dialog Box change the Placement option to Concentric.
  - Plane: Pick Front Face
  - Concentric Reference: Pick Top Arc
Set the following options in the Hole Dialog Box:

- **Termination**: Through All
- **Hole Diameter**: 35 mm

Click OK
12: Create Slot
   - Click the right hand top surface and select **Create Sketch** from the Mini-Toolbar
- Sketch a slot as shown
  Make sure that the Tangent glyph shows next to the cursor when you finish the second arc segment.

- Place a Horizontal Constraint between the mid-point of the slot and mid-point of the side reference edge.
■ Place a Linear Dimensional Constraint for the overall length of the slot. Pick the first arc segment, and the mid-point of the second arc segment. Make sure the Dimension To Arc Edge glyph is shown next to the cursor. Enter 45 mm for the slot overall length.

■ Place a Dimensional Constraint from the right reference edge to the center of the slot. Enter 20 mm.
Place a Dimensional Constraint for the slot width. Enter 18 mm

Exit the sketch

13: Extrude to cut out the slot
   - Start the Extrude Tool, pick inside the slot profile
Set the following options on the Mini-Toolbar:

- **Cut** option
- **Through All** option

Click **OK**

14: Turn on the XZ Origin Plane Visibility
In the browser Right-Click the **XZ Plane**, Select **Visibility** to turn it on
15: Mirror Feature

- Start the **Mirror** Tool
  
  **Model Tab | Pattern Panel | Mirror**

- Select the slot feature.
Select the visible XY Origin plane

Click OK

16: View Part
- Click **Home View** on the View Cube
  Notice that the default **Home View** setting displays the part lying on its back side.
■ Using the View Cube or other View Tools, rotate the part until it is in the position shown.

■ Right-Click the View Cube, Set the current view as Home
  Set Current View as Home | Fixed Distance

■ Click-Hold the View Cube and Drag the cursor to rotate the part to another position.

■ Click Home View on the View Cube once more, notice that the part returns to the up right position.
17: Save Part

- On the **Quick Access** toolbar, click **Save**.

- In the **Save As** dialog box, enter file name **ClevisMount.ipt**

- Click **Save**