Lesson 1

Parametric Modeling Fundamentals – Quick Start

Learning Objectives

When you have completed this lesson, you will be able to:

- Create simple parametric models
- Understand the Basic Parametric Modeling Process
- Create Rough Sketches
- Understand the "Shape before Size" Approach
- Use the view commands
- Create and modify dimensions

In Inventor, the parametric part modeling process involves the following steps:

1. Create a rough two-dimensional sketch.
2. Apply geometric constraints as needed
3. Apply dimensions
4. Extrude/revolve/sweep
5. Add additional features, such as holes, fillets, chamfers, and shells.
6. Analyse and refine the model
7. Create the drawing layout.

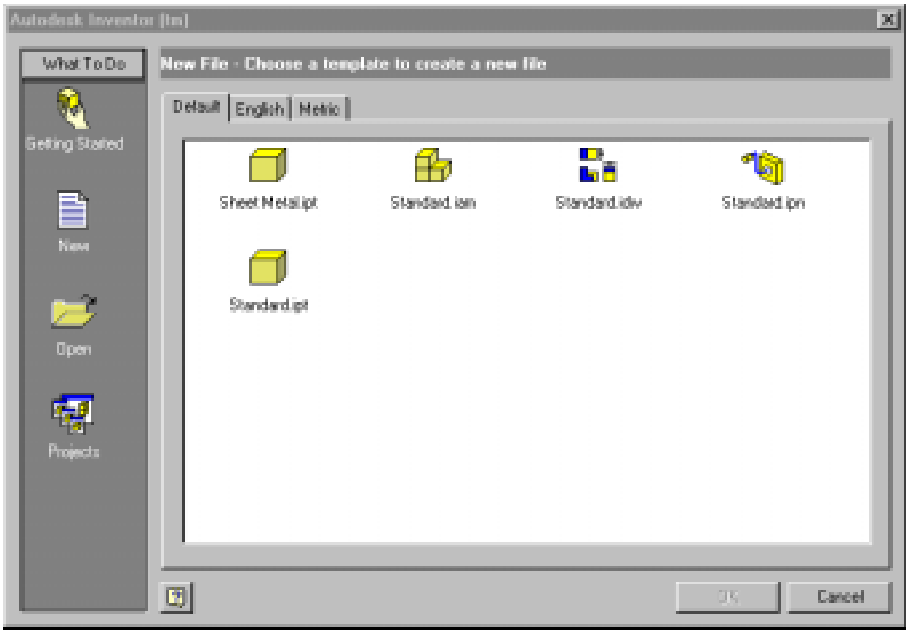
Start Inventor by double-clicking on the Inventor icon on the desktop.

A window will come up. On the left side, we see four icons: Getting Started, New, Open, and Projects. Depending on which left icon is highlighted, the options in the main window change. Verify that the 'New' icon is highlighted.

Inventor uses new file extensions. Refer to the table below to assist you in determining the type of file being created and accessed.

File Extension	File Type
*.ipt	Part File/Sheet Metal Part (3D model)
*.iam	Assembly File (3D model)
*.idv	Design View File
*.ipj	Project File
*.idw	Drawing Layout (2D paper space)
*.ipn	Presentation (3D model/Scene/Rendering)
*.ide	Design Element

Select the Standard.ipt icon and press 'OK'.

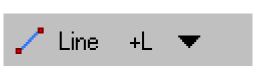


The files displayed and used in the Start-Up dialog are templates. These templates contain default drafting standards.

We will start with a rough sketch. Keep the following guidelines in mind when creating a rough sketch.

- ♦ Create a sketch that is proportional to the desired shape.
- ♦ Keep sketches simple. Leave out fillets, rounds, and chamfers…those can be added later.
- ♦ Exaggerate the geometric features of the desired shape. Remember your sketch is parametric - you can adjust the size when we start adding dimensions. It's easier to go from big to small than vice versa.
- ♦ Draw the geometry so it does not overlap. Inventor looks for a closed polygonal shape. Overlapping lines can create confusion and errors.

Creating a rough sketch

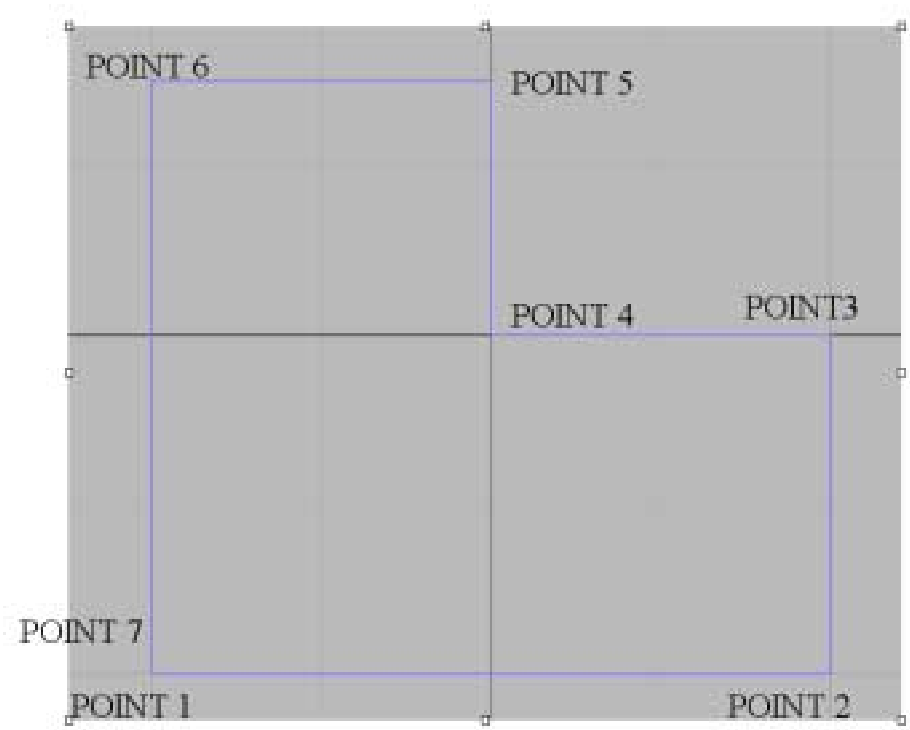


1. Select the LINE icon from the Sketch Toolbar by clicking on it with the left-mouse button. This will activate the Line command.

TIP: The +L next to the Line icon means that typing the letter L on the keyboard will also initiate the Line command. This similar to the command alias 'L' used in AutoCAD.

The Message box located in the top middle of the menu and in the lower left corner of the screen will change to indicate we are now in 'line' mode.   Use the Message box as a help. Inventor expects us to identify the starting location of a line. To switch to arc mode, merely hold down the left-mouse button and move the mouse to form an arc.

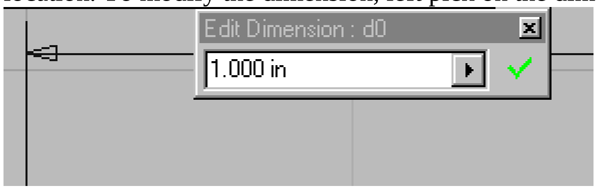
2. Move the mouse near the lower left corner of the Drawing Screen and create a freehand sketch as shown below. Create the sketch by starting at Point 1 and ending at Point 7. Do not be concerned by the actual size of the sketch. Do not worry about keeping the lines perfectly horizontal or vertical as Inventor automatically keeps lines orthogonal.



While in the LINE command, we can right-click the mouse to get the pop-up shown. When we are done with the sketch, we right-click the mouse and select 'Done'.   Notice that we have the option to select Midpoint, Center, Intersection or AutoProject to help us in our sketching.

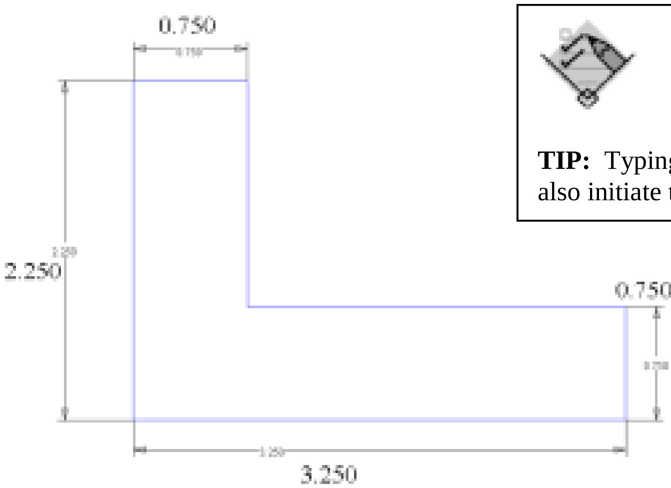
Dimensioning a Sketch

Select the dimension tool by left-picking the Dimension icon with the mouse. Then select the line to dimension. Finally move the mouse away from the line being dimensioned and select the dimension location. To modify the dimension, left pick on the dimension and an edit box appears.

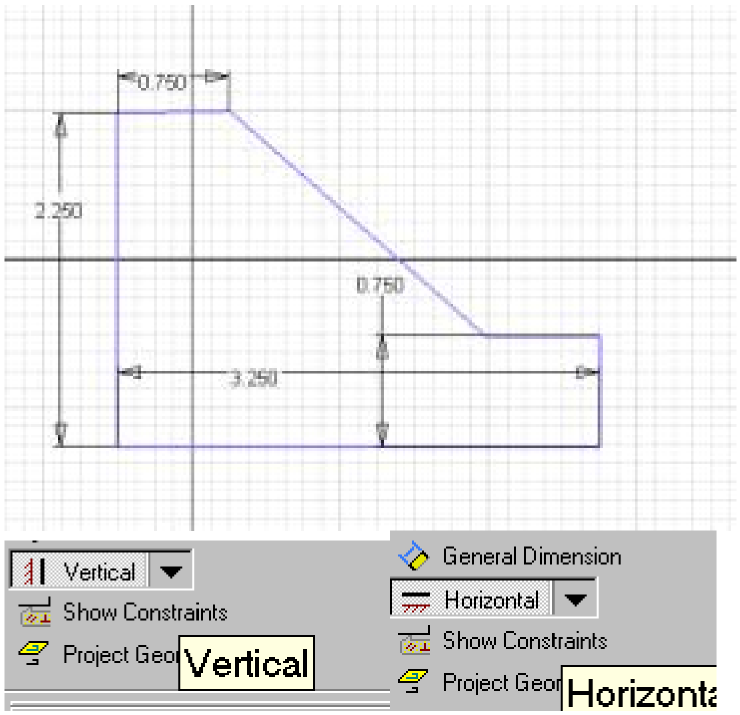


Place your mouse inside the text box and enter the desired dimension.
Select the green check mark when done editing.

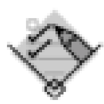
Repeat Step 3 applying the dimensions shown below.



TIP: Typing the letter D on the keyboard will also initiate the DIMENSION command.



What if your sketch turns out like this?
Time to use geometric constraints.
Add a vertical constraint to the slanted line.
Add a horizontal constraint to the upper line.
If your sketch is different, apply the constraints as needed.



TIP: If you do not wish constraints to automatically be added with your geometry, hold down the CONTROL key as you sketch.

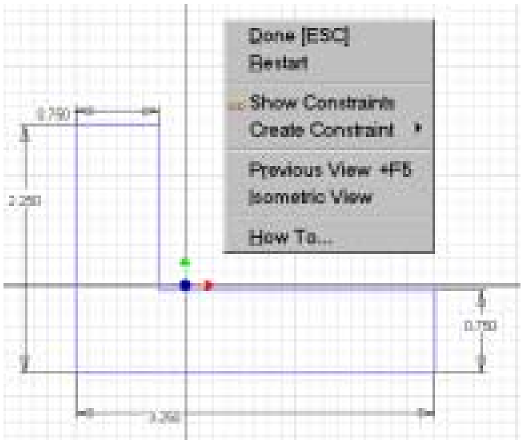
Completing the Sketch

Right click the mouse and Select 'Done' to finish adding dimensions/constraints. Right click the mouse and Select 'Finish Sketch' to indicate we are done defining our geometry.

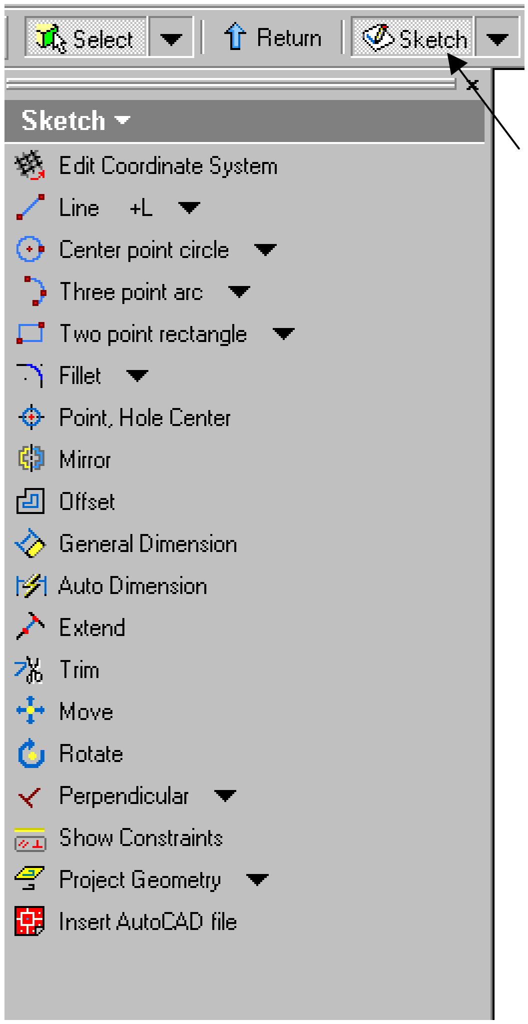
TIP: The +S next to the Finish Sketch icon means that typing the letter S on the keyboard will also initiate the Finish Sketch command.

Completing the Base Solid Feature

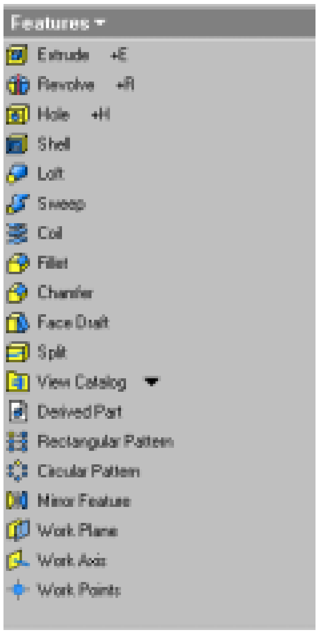
Now that the 2-D sketch is completed, we will proceed to the next step: create a 3D part from the 2D profile. Extruding a 2D sketch is the most common methods used to create 3D parts. We can extrude planar faces along a path. We can also specify a height value, direction and a tapered angle.

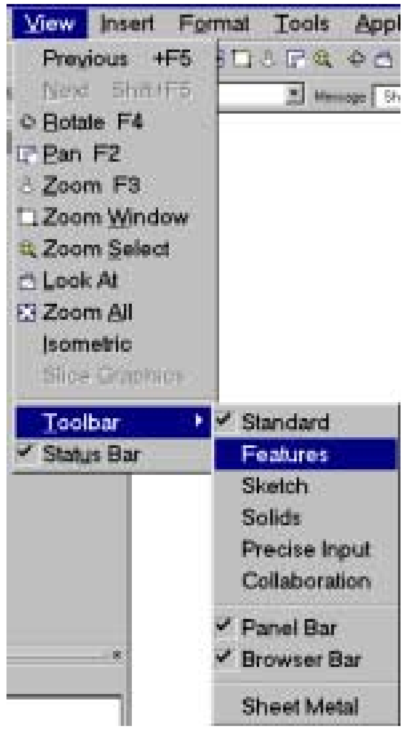


Before we extrude, let's switch to an Isometric View. That will allow us to see what is going to happen more easily. To switch to an Isometric View, right click the mouse anywhere in the Drawing Screen area and select 'Isometric View' from the pop-up menu. We select by highlighting 'IsometricView' and picking with the left mouse button.



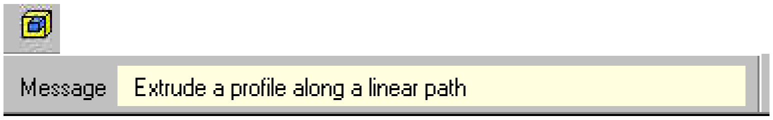
We still see the Sketch tools and we need to switch to the Feature tools. To do that, press the Sketch button located above the Sketch toolbar.



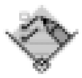


The Features toolbar should appear.

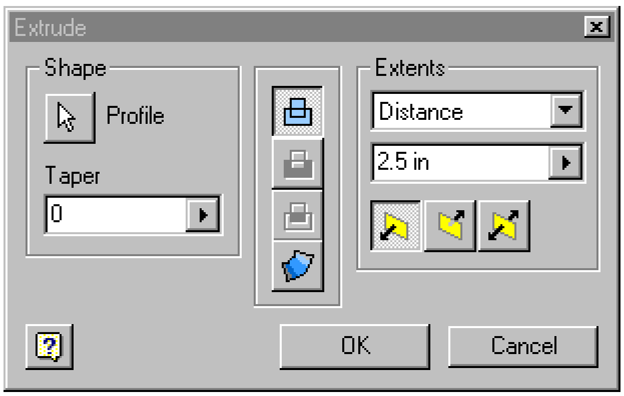
TIP: To hide the text next to the tool icons, switch to EXPERT mode. Select the panel bar, right click and select EXPERT.



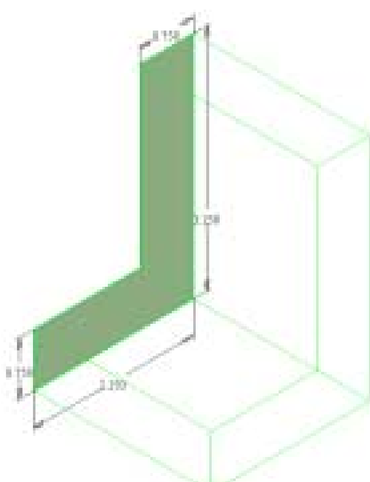
We select the Extrude icon located in the Features toolbar. Note that when we move the mouse over the Extrude icon, the message changes in the Message box to indicate what the Extrude icon does. The Extrude icon is selected by picking it with the left mouse button.



TIP: The +E next to the Extrude icon means that typing the letter E on the keyboard will also initiate the EXTRUDE command.



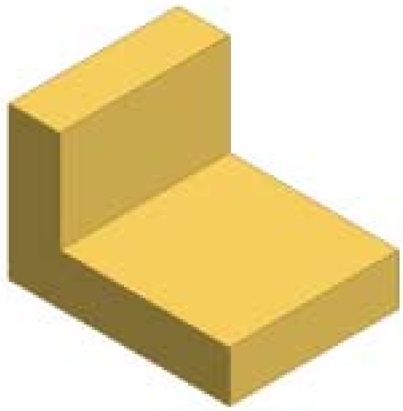
Select 'Distance' under Extents and enter '2.5' as the value.
Refer to the dialog box shown.



We see a preview of what the extrude will look like. Note that we can modify the distance by using the mouse in the Drawing Screen area.

To do this, simply place the mouse near one of the outer corners and hold down the left mouse button. Now drag the mouse back and forth. The value in the Extrude dialog box will automatically update.

Using your mouse, set your distance to 3.25 units. Then click on the 'OK' button.

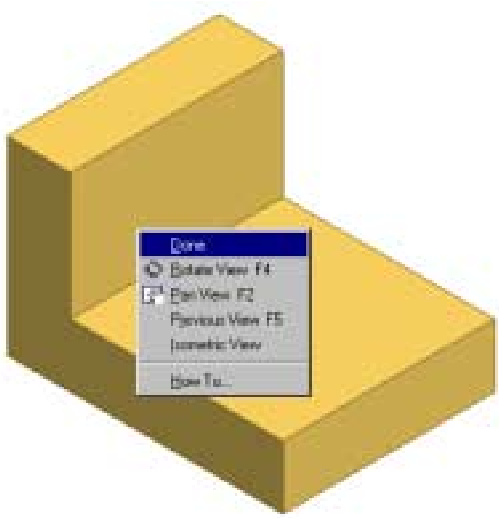


Our extruded part.

Dynamic Viewing Functions – Realtime Zoom

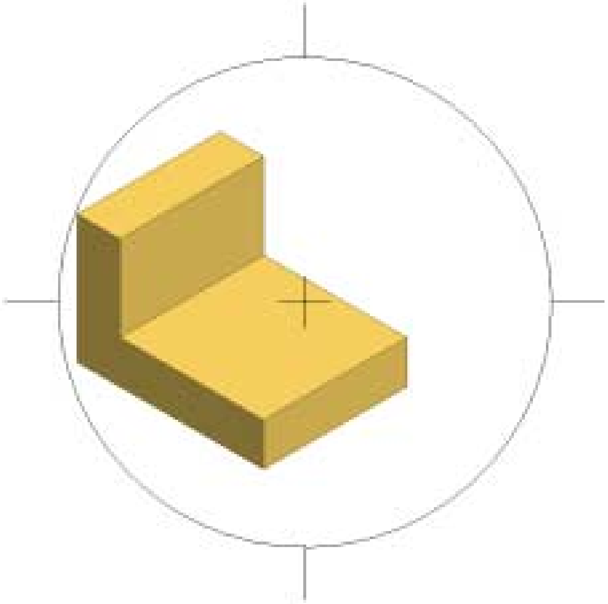
 Click on the Zoom Realtime icon in the Standard toolbar area.
Inside the Drawing Screen area, hold down the left mouse button and move the mouse upward. Next move the mouse downward. Note how the size of the object changes. We are not actually scaling or modifying the part size. We are changing our perspective view. Imagine holding a part at arms-length and then moving the part closer to our face. This is what the Zoom Realtime does.

Right-click the mouse.



A pop-up menu appears that provides several View Options. Select the 'Pan View' option.

Our mouse icon changes from an arrow to a small hand. Imagine a piece of paper on our desk and using our hand to move the paper around the tabletop. This is what 'pan' does. The size of our object does not change, but we can shift its position inside our Drawing Screen area.

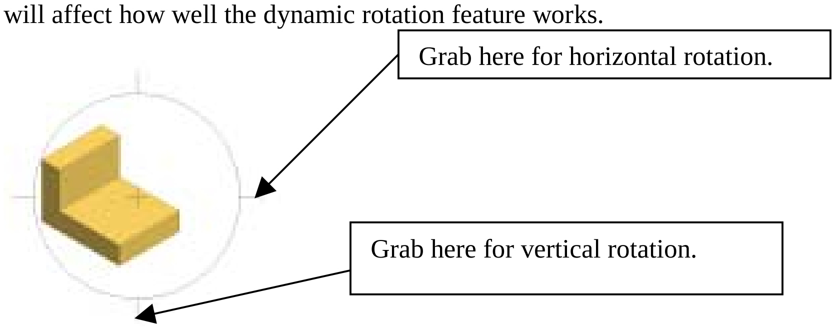


Right-click the mouse again and select ‘Rotate View’. The ‘Rotate View’ displays an arcball, which is a circle divided into four quadrants. This enables us to manipulate the view of 3D objects by clicking and dragging with the left-mouse button.

Inside the arcball, press down the left-mouse button and drag it up and down to rotate about the X-axis.

Move the cursor to different locations on the screen, such as outside the arcball or on one of the four small tickmarks, and experiment with the real-time dynamic rotation feature.

Note: The graphics card installed on the computer system as well as the amount of memory installed will affect how well the dynamic rotation feature works.

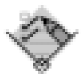


TIP: To rotate the model in the horizontal direction, grab one of the horizontal bars. To rotate the model in the vertical direction, grab one of the vertical bars.

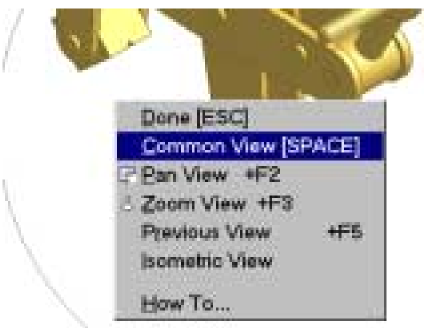
Use the 3D Rotate button on the Standard toolbar to:

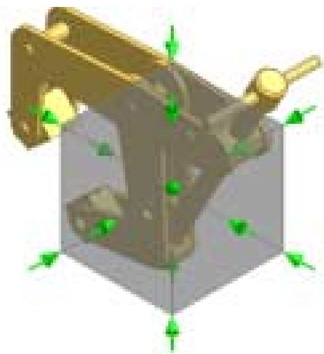
- Rotate a part or assembly in the graphics window
- Display standard isometric and orthographic views of a part or assembly.

Rotation can be around the center mark, free in all directions, or around the X or Y axes in the 3D Rotate symbol view. You can rotate the view while other tools are active.



TIP: The 3D Rotate tool remembers the last mode used when you exit the command. When the command is active, press the spacebar to switch modes.

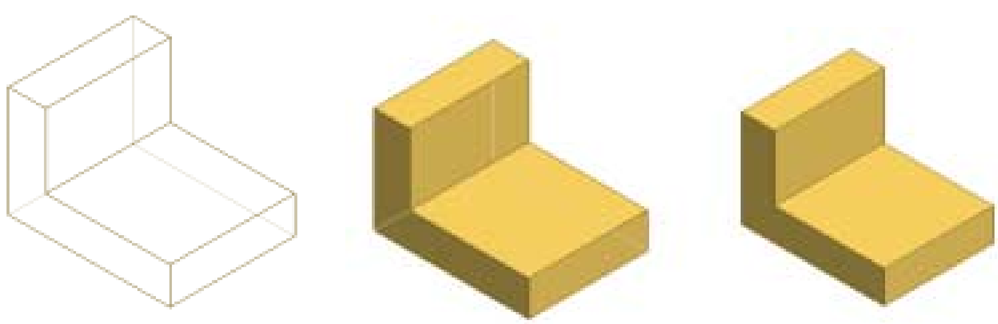




To switch to Common View mode, right click and select Common View from the menu or press the space bar. Common View mode allows the user to quickly switch view orientations. Simply pick on the arrow to select the desired view.

Changing the Appearance of the Model

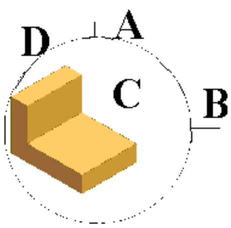
Inventor comes with three modes for model appearance: Wire frame, Hidden, and Shaded. To change the mode, use the left mouse button to select the desired mode.



Our model in all three modes: Wire frame, Hidden Edge, and Shaded.

Save the file as ‘lesson1.ipt’.

Review Questions



1. Select the area to rotate the model in a horizontal direction.

2. The shortcut key to place a General Dimension is:

   A. I
   B. G
   C. D
   D. GD

3. To edit or modify a dimension:

   A. Left pick on a dimension and type in the edit box.
   B. Right pick on a dimension and select Edit from the menu.
   C. Select the Edit Dimension tool.
   D. Select Edit Dimension from the Menu.

4. To quickly switch to an Isometric View:

   A. Right click anywhere in the drawing window and select ‘Isometric View’ from the menu.
   B. Select the Isometric View tool from the View toolbar.
   C. Select the 3D Orbit tool, right click and select ‘Isometric View.’
   D. Select Isometric View from the View menu.

5. To draw a line, you should be in this mode:

   A. Sketch
   B. Features
   C. Solids
   D. Assembly

6. The three modes of appearance for a model are:

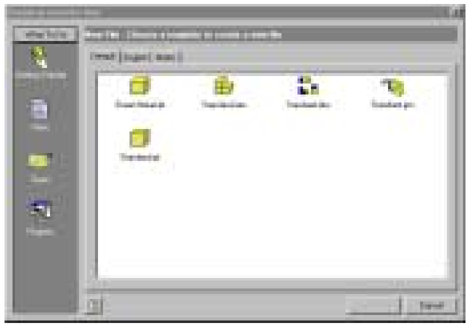
   A. Wire Frame, Hidden, Shaded with Edges On
   B. Wire Frame, Shaded, Colored
   C. Wire Frame, Hidden, Rendered
   D. Wire Frame, Hidden, Shaded

7. To prevent constraints from automatically being added to a sketch:

   A. Turn off constraint mode under Options.
   B. Hold down the CONTROL key while sketching.
   C. Hold down the ALT key while sketching.
   D. You can not prevent constraints from being added.

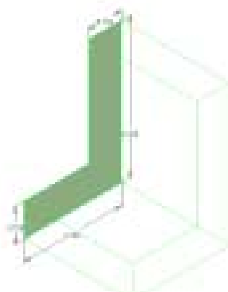
8. The file extension for an Inventor part file is:

    A. ipt
    B. dwg
    C. iam
    D. ipn



9. The files displayed in the start up dialog are:

    A. Dummy files
    B. Blank files
    C. Templates with default settings
    D. Transparent files – they don't exist, they are just icons used for starting a new file



10. To change the extrusion distance:

    A. Modify the value in the distance edit box of the Extrude dialog box
    B. Use the mouse to drag the extrusion into position
    C. Right click on the extrusion and enter a value.
    D. A & B but not C.

ANSWERS: 1) B; 2) C; 3) A; 4) A; 5) A; 6) D; 7) B; 8) A; 9) C; 10) D