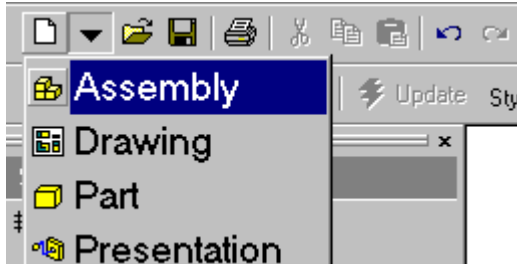


## Lesson 4

### The Standard Toolbar

The Standard toolbar looks similar to the Microsoft Windows standard toolbar. The first icon shows a blank piece of paper indicating that it opens a new file.

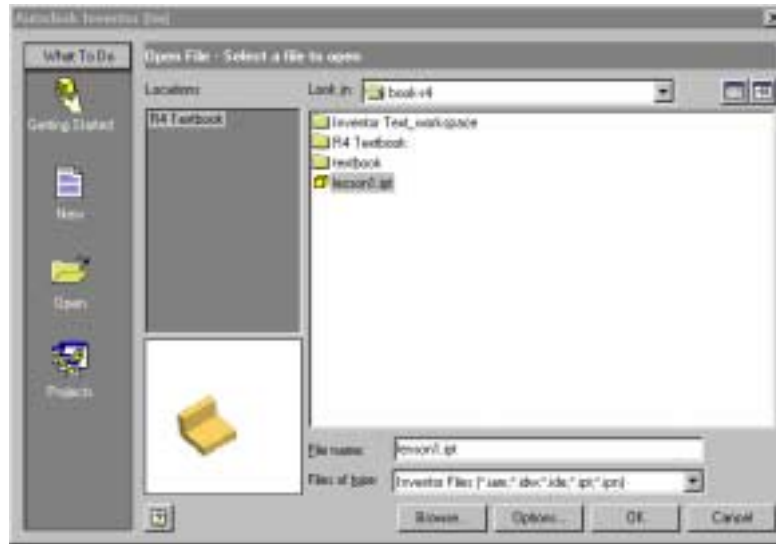


However, immediately to the right of the blank sheet of paper is an arrow. Pressing down on that arrow reveals four options: Assembly, Drawing, Part, or Presentation. So, when starting a new file, first specify whether to create an assembly, drawing, presentation or part file. Clicking on the blank sheet of paper will bring up Inventor's Start Up dialog box.



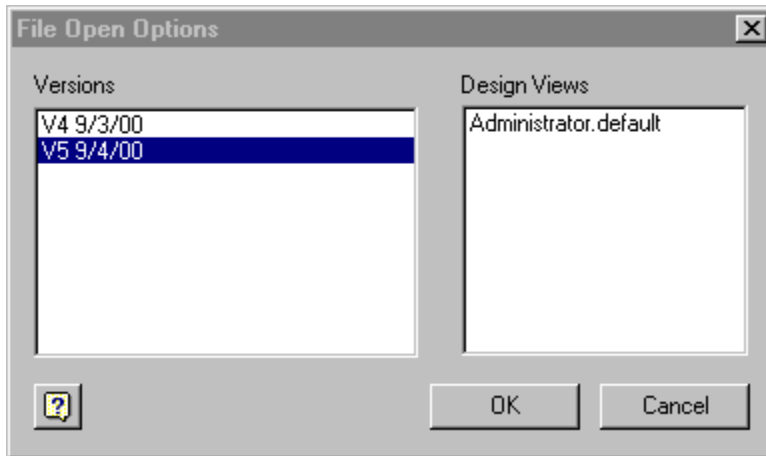
**TIP:** Using the pull-down arrow reduces the number of steps required to create a new file, so that is the preferred method when starting a new file.

The Open folder opens an existing file. Pressing this icon brings up a browser window that allows the user to locate the file.



Look in	Shows path of the active directory.
Locations	Shows file locations defined in the project file. When you pause the cursor over a path type, the ToolTip shows the path. Click to change to the folder. The folder tree and files are listed in the main window of the dialog box.
File name	Specifies the file to open, enter a file name or select a file from the listed files.
File of type	Filters file list to include only files of a specific type. Click the arrow to show list, then highlight to select a file type.
Open	Opens highlighted file.
Cancel	Cancels the file open operation and closes the dialog box.
Find	Opens the Find Files dialog box so that you can search for files based on file properties.
Options	Opens an options dialog box so that you can set the options for the selected file. If the file is an Autodesk Inventor file, you can specify the file version and design view (if applicable) to open. If you are importing a file, you can set import options specific to the type of file.
Reserved by	Shows the name of the person who has reserved the file or allows you to reserve the file. Available only if the Multi User option is on.
Reserve	When you open a file, click to reserve the file.
Clear	When you open a file, click to clear the name of the person who has reserved the file.

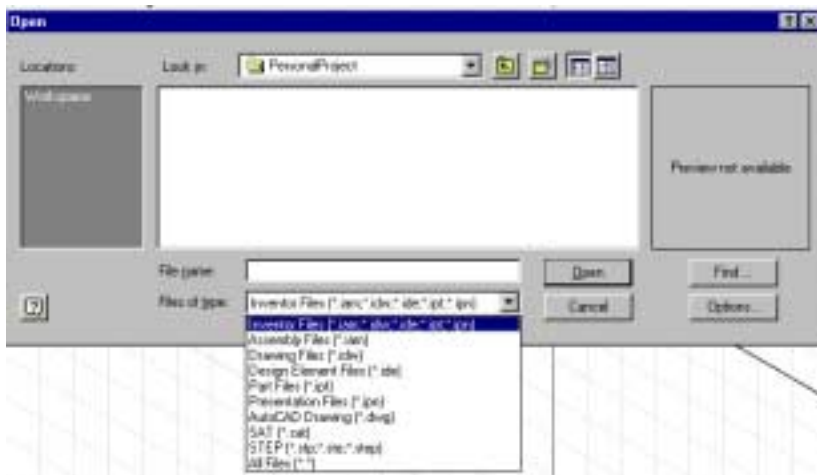




The File Open Options dialog box is accessed by highlighting a file in the Open dialog and then pressing the 'Options' button.

Versions	Lists the available versions of the selected file. Select a version from the list. Note: To increase or decrease the number of versions that are saved, specify the Number of Versions to Keep option in the Tools>Options>General tab.
Design Views	If the selected file is an assembly, lists the available design views. Select a design view from the list.

Inventor tracks up to ten different versions or revisions of each model and assembly. This allows the user to roll back to a prior version easily.

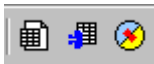




In the 'Files of type' drop-down window, the user can access a list of the file types Inventor can open. Drawings of types other than Inventor, i.e. .iam, .idw, .ide, .ipt, are not fully supported in Release 1, but Release 2 and above will fully support dwg files from AutoCAD or Mechanical Desktop and other file types as well. Release 4 allows users to open Pro/Engineer files. Pro/Engineer files can be modified and saved as Inventor files.



The next seven buttons are familiar to most users: Save, Print, Cut, Copy, Paste, Undo, and Redo.



The next three buttons are specific to Inventor. They are Parameters, iPart Table, and Create Interface.



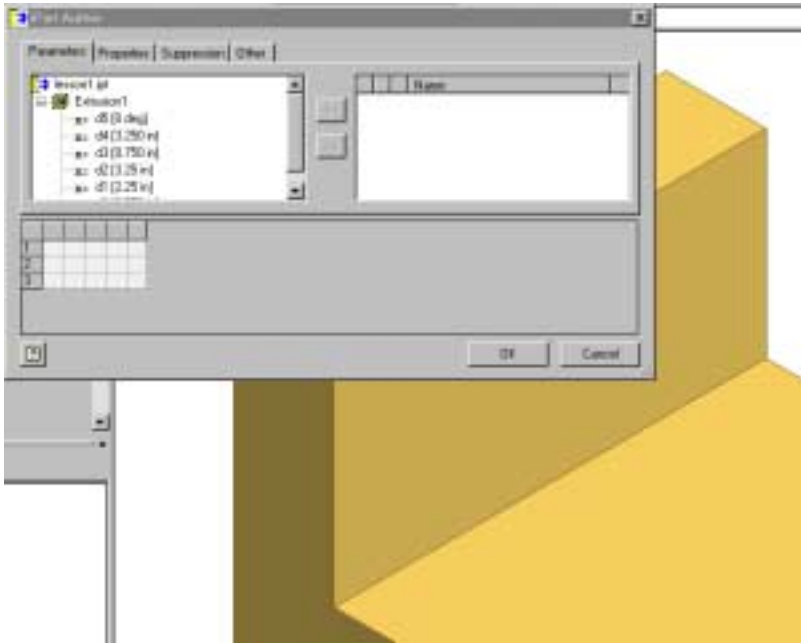
## Parameters

The parameters button, located between the Standard Windows icons and the Viewing icons, is used for table driven parts. More information on creation and management of table driven parts can be found in my intermediate level textbook.



## iPart Table

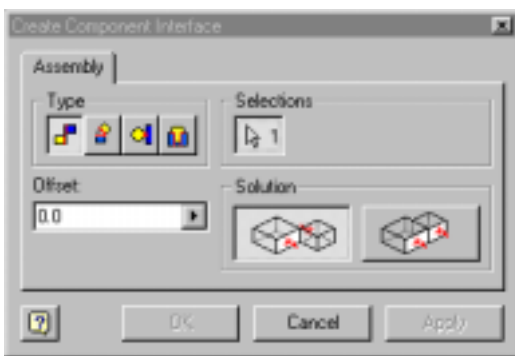
The iPart Table allows the user to convert the existing part into a part linked to an Excel-type spreadsheet.



The iPart Author allows the user to select existing dimensions and define them as variables. More information on how to use the iPart Author can be found in my intermediate level textbook.



## Create Interface



The Create Interface button launches the Create Component Interface. This allows the user to set edges, faces, or axes for use in assembling parts. By pre-selecting where assembly constraints are to be made, assembling parts together is faster and easier.



The next five buttons are grouped together because they all have to do with viewing options.



### **Zoom All**

The first button, which has four arrows pointing in each compass direction, is 'Zoom All'. Pressing the button expands or shrinks the view to encompass all drawn entities in the graphics window.



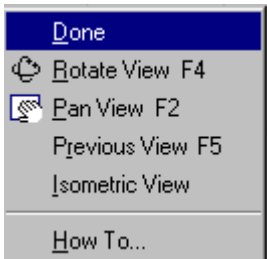
### **Zoom Window**

The next button is 'Zoom Window'. To create the window, pick a point with the left mouse button. This will be the upper or lower corner of the window. Drag the mouse across the area you wish to zoom in. Pick the next corner. The mouse will draw a light window to indicate the area that will be captured in the zoom.



### **Dynamic Zoom**

The middle button is a 'Dynamic Zoom' that allows the user to zoom in or out by moving the mouse. The user selects the Dynamic Zoom, then holds the left mouse down while moving the mouse up and down. A left click of the mouse selects the current view. If the user needs additional help adjusting the view, a simple right click of the mouse will bring up a submenu.



The submenu is similar to the drop-down menu in AutoCAD and provides the user with an abbreviated view menu.



### **Pan Mode**

The icon showing a hand on a piece of paper indicates 'Pan' mode. This allows the user to shift the view without changing the scale of the object. Hold down the left mouse button and move the mouse from left to right to adjust the view.



### **Zoom Selected**

The final view icon is 'Zoom Selected'. Press this button, then pick on the feature desired to zoom into and the window will automatically take you there. This is an excellent tool when working on complex parts or assemblies; something that other CAD users will envy. If Inventor is unable to determine which feature was selected, an alert box will appear and Inventor will take you to its best guess, which is usually close enough for you to continue working.



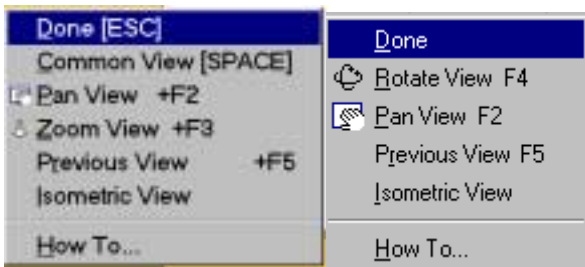
The next section of the Standard toolbar also comes under View menu topics.



### 3D Rotate

The first icon, which resembles a sphere, is Inventor's 3D orbiter. This tool allows us to rotate our object in 3D space seamlessly.

When first activated, a crosshair target figure will appear. The user can grab any pole and rotate by that pole. Additionally, a right mouse click will bring up a submenu.



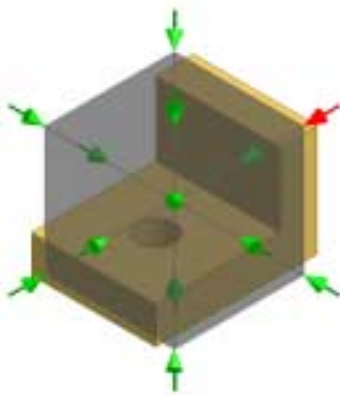
The Rotate Submenu

The Dynamic Zoom Submenu

Compare the submenus depending on whether you are in Rotate mode or Dynamic Zoom mode.

When the object is positioned as desired, a left mouse click sets the view.

Starting in Inventor R2, a new orbit feature called Common View was introduced.



Selecting this option brings up a box with arrows at the vertices and centered on each plane. Selecting an arrow will automatically orient the view accordingly. This can be a quick way of rotating and orienting your model.



## Look At

The second icon, which resembles a projected plane, is the 'Look At' tool. This tool will automatically switch the view to a plan view of the selected plane or feature of an object, allowing for quick editing.



## Display

The last icon is a drop-down that controls the appearance of our objects, Wire Frame, Hidden Edge, or Shaded. Modifying the appearance can affect the ease of editing.



**TIP:** Pressing the SPACE bar when the Rotate command is active switches from COMMON mode to FREE ROTATE mode.



## Section All Parts

This tool allows the user to create a sectioned view that can be used in a drawing.



The last three icons in the Standard toolbar are Design Support System tools.



## Help

The balloon with the question mark brings up 'Help'. This can also be brought up by pressing F1.



## Visual Syllabus


















The icon with a table and a question mark brings up the DesignProf Visual Syllabus, an on-line learning tool for new users. The syllabus will walk the beginner through just about any task they want to do.











**Design Doctor**

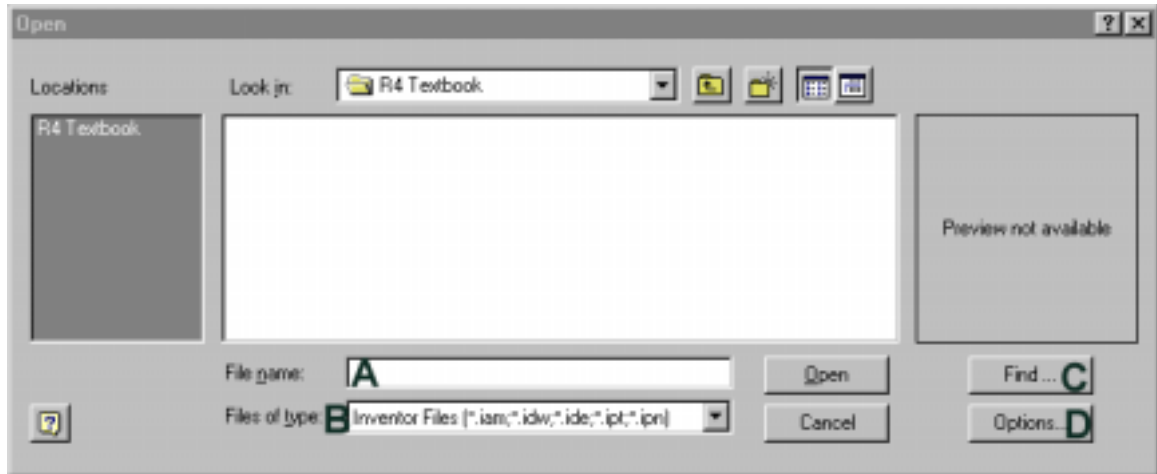
The final icon button is normally grayed out. The cross indicates the Design Doctor. This is a remarkable tool to walk the user through any problems or conflicts, which may occur during the design process. The icon is only enabled if Inventor detects a modeling or assembly error.

### Standard Toolbar

Button	Tool	Function	Special Instructions
	Parameters	Allows the user to create a part using a spreadsheet to control dimensions and features	
	iPart Author	Creates a spreadsheet for an existing part	
	Create Interface	Allows the user to pre-set edges, faces, or axes to be used for assembly constraints	
	Zoom All	Zoom in or out so everything is visible in the graphics window	
	Zoom Window	Zoom in so the selected viewing area fills the graphics window	Pick two corners to designate the window
	Dynamic Zoom	Drag to zoom in or out	Use F3 to activate
	Pan	Drag to reposition the model in the graphics window – does not zoom in or out	Use F2 to activate
	Zoom Selected	Zooms in or out so selected geometry fits in graphics window	
	3D Rotate	Change the viewing perspective of the model	Use F4 to activate. Use spacebar to toggle rotation modes
	Look At	Select geometry for a PLAN view	
	Wire Frame display	Display the model as a wire frame	
	Hidden Edge display	Display the model as a shaded solid with hidden edges visible	
	Shaded Display	Display the model as a shaded solid	Default display setting
	Section All Parts	Allows the user to create a sectioned 3D view of a part or assembly.	
	Help	Brings up Help	Use F1 to activate
	Visual Syllabus	Animated tutorials	
	Design Doctor	Diagnose and fix part and assembly errors and problems	Is grayed out unless errors occur

## Review Questions

1. The three display modes in Inventor are:
  - A. Shaded, Wire Frame, Hidden Edge
  - B. Shaded, Wire Frame, Shaded with Edges On
  - C. Wire Frame, Hidden, Rendered
  - D. Wire Frame, Shaded, Rendered
2. The function key to activate Dynamic Zoom is:
  - A. F1
  - B. F2
  - C. F3
  - D. F4
3. The function key to bring up Help is:
  - A. F1
  - B. F2
  - C. F3
  - D. F4
4. The tool to select geometry and then automatically switch to a PLAN view of that geometry is:
  - A.  Look At
  - B.  Zoom Selected
  - C.  Zoom Window
  - D.  Dynamic Zoom
5. To switch from Common to Free Rotate when Rotate is active:
  - A. Use TAB key
  - B. Use Control key
  - C. Use Shift key
  - D. Use SPACE bar
6. The toolbar to select to bring up Visual Syllabus:
  - A. 
  - B. 
  - C. 
  - D. 



7. The button to select to search for a file.
8. The button to select to see an AutoCAD dwg file
9. The user highlights a file and presses the Options button. The Options button allows the user to:
  - A. Select which version of the file to open
  - B. Which version of Inventor to use to edit the file
  - C. Enable partial file loading
  - D. Set the default settings for units, physical properties, and display
10. The Design Doctor tool is:
  - A. Normally inactive
  - B. Normally active
  - C. Used to check a design for interference
  - D. Used to recover corrupted files

**Notes:**

3D orbiter, 4-5  
 Common View, 4- 6  
 Design Doctor, 4- 8  
 Design Support System, 4- 7  
 DesignProf, 4- 7  
 Dynamic Zoom, 4- 4  
 file types, 4- 3  
 Help, 4- 7  
 Look At, 4- 6

Pan, 4- 5  
 parameters, 4- 4  
 Shaded, 4- 6  
 Standard toolbar, 4- 1  
 Wire Frame, 4- 6  
 Zoom All, 4- 4  
 Zoom Selected, 4- 5  
 Zoom Window, 4- 4

<a href="#">The Standard Toolbar</a> .....	1
<a href="#">Parameters</a> .....	4
<a href="#">Zoom All</a> .....	6
<a href="#">Zoom Window</a> .....	6
<a href="#">Dynamic Zoom</a> .....	6
<a href="#">Pan Mode</a> .....	6
<a href="#">Zoom Selected</a> .....	6
<a href="#">3D Rotate</a> .....	7
<a href="#">Look At</a> .....	8
<a href="#">Help</a> .....	8
<a href="#">Visual Syllabus</a> .....	8
<a href="#">DesignProf</a> .....	<b>Error! Bookmark not defined.</b>
<a href="#">Design Doctor</a> .....	9
<a href="#">Review Questions</a> .....	11