



Chapter 1

Introduction

Learning Objectives

After completing this chapter, you will be able to:

- *Understand different modules of Autodesk Inventor.*
- *Understand how to open a new part file in Autodesk Inventor.*
- *Understand various terms used in sketching environment.*
- *Understand the important terms and definitions.*
- *Understand the use of various hotkeys.*
- *Customize hotkeys.*
- *Modify the color scheme in Autodesk Inventor.*

INTRODUCTION TO Autodesk Inventor 2011

Welcome to the world of Autodesk Inventor. If you are new to the world of three-dimensional (3D) design, then you have joined hands with thousands of people worldwide who are already working with 3D designs. If you are already using any other solid modeling tool, you will find this solid modeling tool more adaptive to your use. You will find a tremendous reduction in the time taken to complete a design using this solid modeling tool.

Autodesk Inventor is a parametric and feature-based solid modeling tool. It allows you to convert the basic two-dimensional (2D) sketch into a solid model using very simple, but highly effective modeling options. This solid modeling tool does not restrict its capabilities to the 3D solid output, but also extends them to the bidirectional associative drafting. This means that you only need to create the solid model. Its documentation, in the form of the drawing views, is easily done by this software package itself. You just need to specify the required view. This solid modeling tool can be specially used at places where the concept of “**collaborative engineering**” is brought into use. Collaborative engineering is a concept that allows more than one user to work on the same design at the same time. This solid modeling package allows more than one user to work simultaneously on the same design.

As a product of Autodesk, this software package allows you to directly open the drawings of the other Autodesk software like AutoCAD, Mechanical Desktop, AutoCAD LT, and so on. This interface is not restricted to the Autodesk software only. You can easily import and export the drawings from this software package to any other software package and vice versa.

To reduce the complications of design, this software package provides various design environments. This helps you capture the design intent easily by individually incorporating the intelligence of each of the design environments into the design. The design environments that are available in this solid modeling tool are discussed next.

Part Module

This is a parametric and feature-based solid modeling environment and is used to create solid models. The sketches for the models are also drawn in this environment. All applicable constraints are applied to the sketch automatically while drawing. You do not need to invoke an extra command to apply them. Once the basic sketches are drawn, you can convert them into solid models using simple, but highly effective modeling options. One of the major advantages of using Autodesk Inventor is the availability of the Design Doctor. The Design Doctor is used to calculate and describe errors, if any, in the design. You are also provided with the remedy for removing errors such that the sketches can be converted into features. The complicated features can be captured from this module and can later be used in other parts. This reduces the time taken to create the designer model. These features can be created using the same principles as those for creating solid models.

Assembly Module

This module helps you create the assemblies by assembling multiple components using assembly constraints. This module supports both the bottom-up approach as well as the top-down approach of creating assemblies. This means that you can insert external components into the **Assembly** module or create the components in the **Assembly** module itself. You are

allowed to assemble the components using the smart assembly constraints. All the assembly constraints can be added using a single dialog box. You can even preview the components before they are actually assembled. This solid modeling tool supports the concept of making a part or a feature in the part adaptive. An adaptive feature or a part is the one that can change its actual dimensions based upon the need of the environment.

Presentation Module

A major drawback of most solid modeling tools is their limitation in displaying the working of an assembly. The most important question asked by the customers in today's world is how to show the working of any assembly. Most of the solid modeling tools do not have an answer to this question. This is because they do not have proper tools to display an assembly in motion. As a result, the designers cannot show the working of the assemblies to their clients. In cases where it is necessary to show the animation, they have to take the help of some other software packages such as 3D Studio MAX or 3D Studio VIZ. However, keeping this problem in mind, this software package provides a module called the **Presentation** module. In this module, you can animate the assemblies created in the **Assembly** module and view their working. You can also view any interference during the operation of the assembly. The assemblies can be animated using easy steps.

Drawing Module

This module is used for the documentation of the parts or assemblies in the form of drawing views. You can also create the drawing views of the presentation created in the **Presentation** module. All parametric dimensions, added to the components in the **Part** module during the creation of the parts are displayed in the drawing views in this module.

Sheet Metal Module

This module is used to create the sheet metal component. When you invoke a sheet metal file, the sketching environment is active by default. You can draw the sketch of the base sheet in this module and then proceed to the sheet metal module to convert it into the sheet metal component.

GETTING STARTED WITH Autodesk Inventor

Install Autodesk Inventor on your system; the Autodesk Inventor Professional 2011 shortcut icon will automatically be created on the desktop. Double-click on this icon to start the software. You can also start Autodesk Inventor from the taskbar by choosing **Start > Programs** (or **All Programs**) > **Autodesk > Autodesk Inventor 2011 > Autodesk Inventor Professional 2011**, as shown in Figure 1-1.

The system will prepare for starting Autodesk Inventor by loading all required files. After all required files have been loaded, the initial screen of Autodesk Inventor Professional 2011 will be displayed, as shown in Figure 1-2. You can view the recent enhancement and information related to Autodesk Inventor 2011 by choosing the buttons displayed on the **Ribbon**.

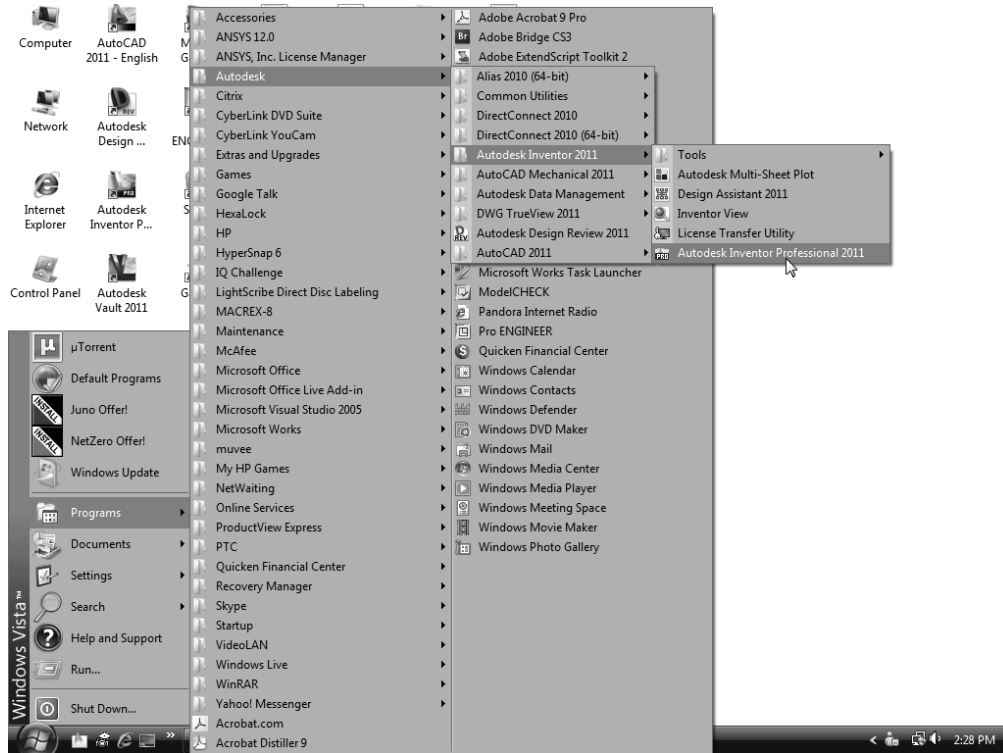


Figure 1-1 Starting Autodesk Inventor using the taskbar

To start a new part file, choose the **New** tool from the **Launch** panel of the **Get Started** tab in the **Ribbon**; the **New File** dialog box will be displayed, as shown in Figure 1-3. Alternatively, you can start a new part file by using the **Quick Launch** area of the **Open** dialog box, see Figure 1-4. The **Open** dialog box can be invoked by choosing the **Open** button from the **Get Started** tab of the **Ribbon**. When you start a new session of **Autodesk Inventor Professional 2011**, only the **Start a new file** button will be activated in the **Quick Launch** area of the **Open** dialog box. Choose the **Start a new file** button; the **New File** dialog box will be displayed, refer to Figure 1-3. Choose the **Metric** tab from the **New File** dialog box and then double-click on the **Standard (mm).ipt** template to open a default metric template. As a result, a new part file with the default name **Part1.ipt** will be opened and you can start working in this file. Also, the sketching environment will be invoked, as shown in Figure 1-5. This figure also displays various components of this screen.

It is evident from Figure 1-5 that the screen of Autodesk Inventor is quite user-friendly. Apart from the components shown in Figure 1-5, you are also provided with various shortcut menus, which are displayed upon right-clicking the mouse in the drawing area. The type of the shortcut menu and its options depend on where or when you are trying to access this menu. For example, when you are inside any command, the options displayed in the shortcut menu will be different from the options displayed when you are not inside any command. The different types of shortcut menus will be discussed when they are used in the textbook.

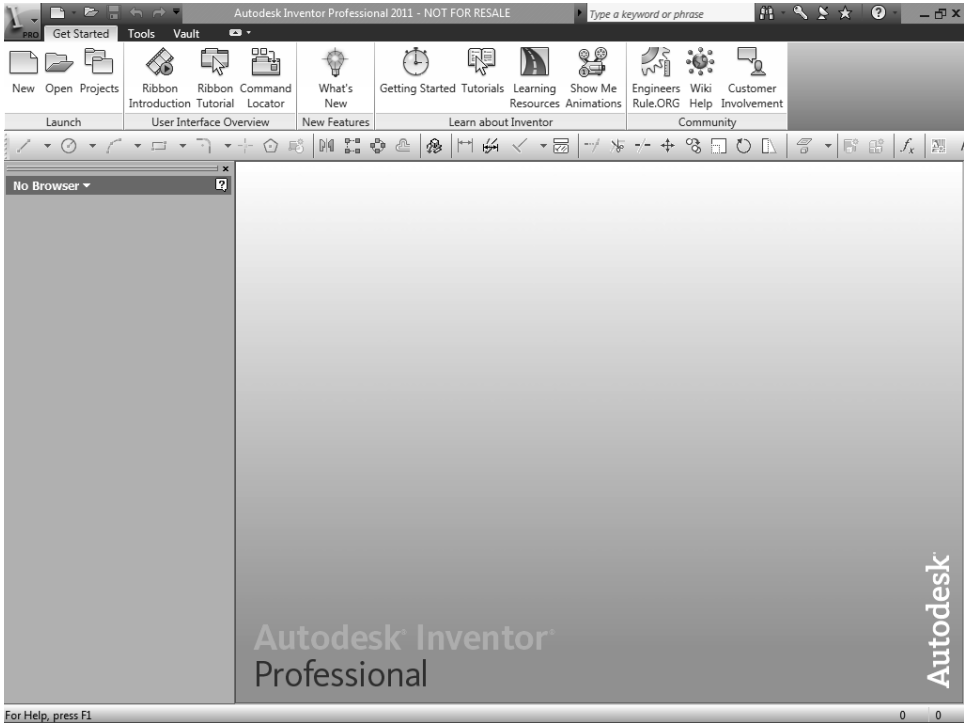


Figure 1-2 Initial screen display of Autodesk Inventor Professional 2011

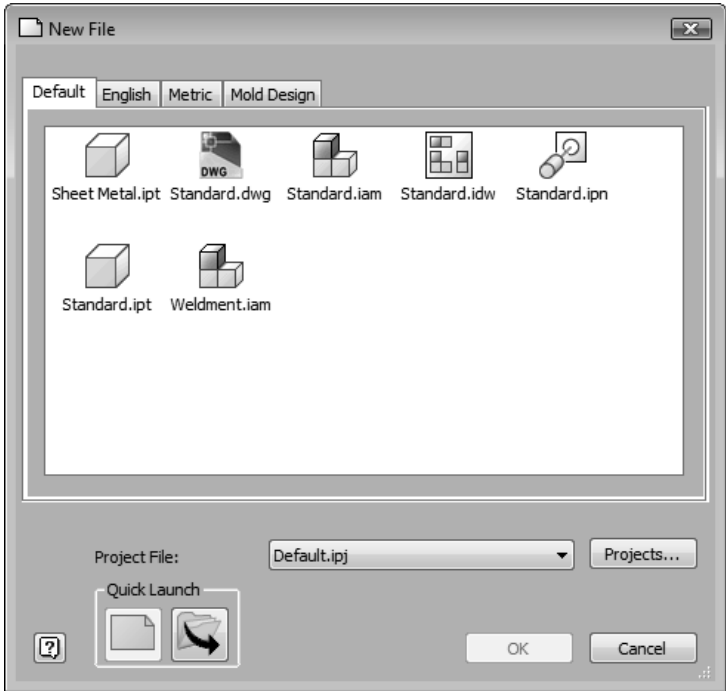


Figure 1-3 The New File dialog box

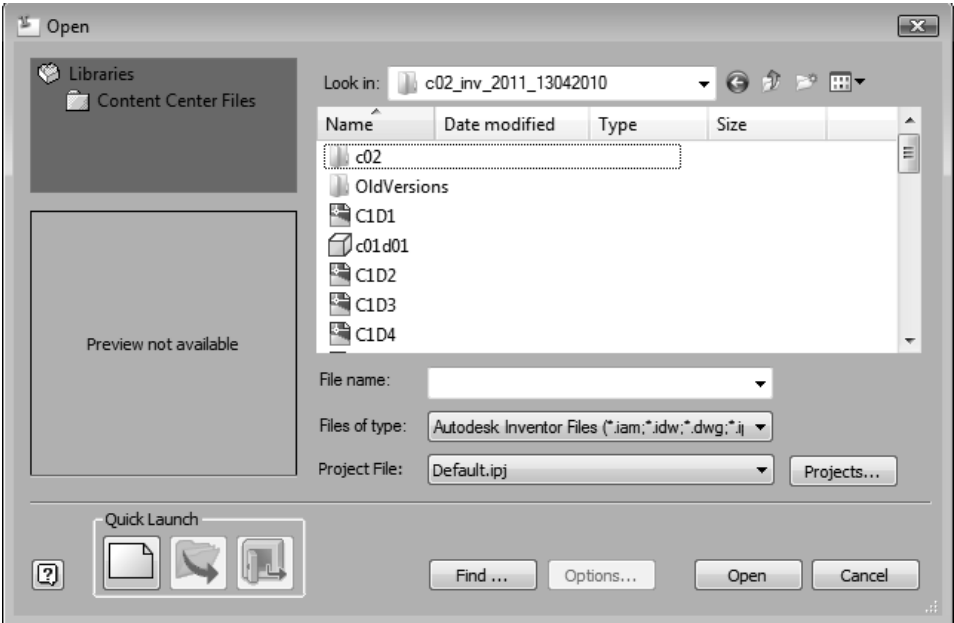


Figure 1-4 The Open dialog box

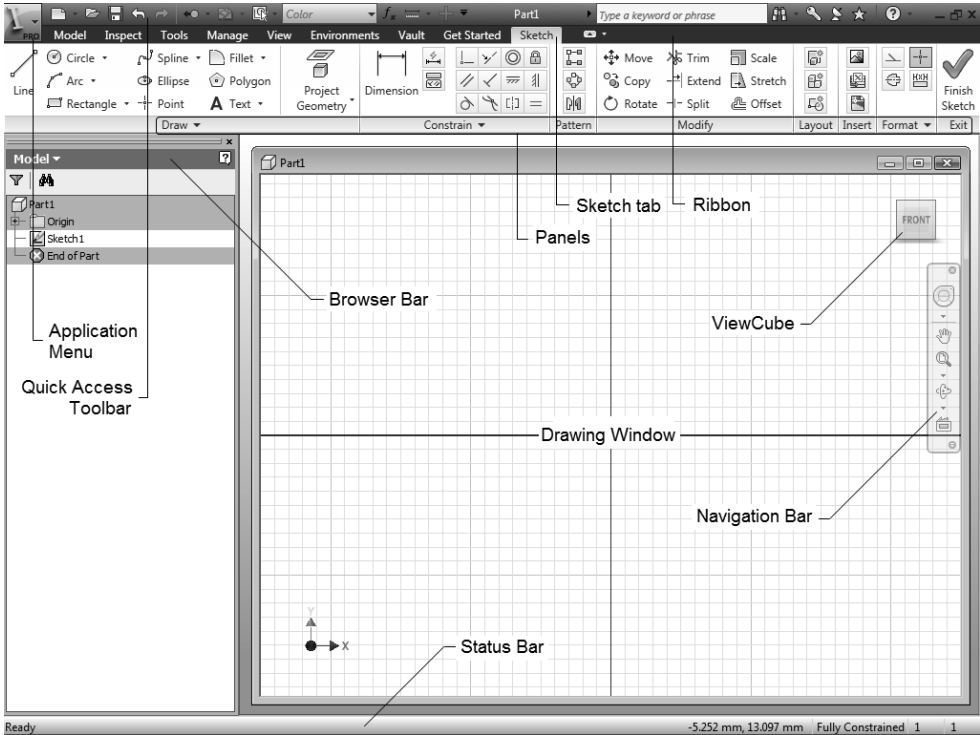


Figure 1-5 The sketching environment of Autodesk Inventor and its main components

Quick Access Toolbar

This toolbar is common to all design environments of Autodesk Inventor. However, some of these options will not be available when you start Autodesk Inventor for the first time. You need to add them using the down arrow given on the right of the **Quick Access Toolbar**, as shown in Figure 1-6. Some of the important options in this toolbar are discussed next.

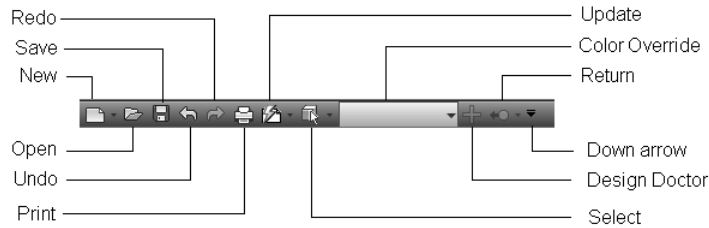


Figure 1-6 The Quick Access Toolbar

Select

This tool is used to set the selection priority. When you choose the down arrow on the right of this tool, six more tools are displayed. These tools are **Select Bodies**, **Select Groups**, **Select Features**, **Select Face and Edges**, **Select Sketch Features**, and **Select Wires**. The **Select Bodies** tool is chosen to set the selection priority to bodies. If this tool is chosen, you can select any individual body in the model. If you choose the **Select Features** tool, you can select any feature in the model. The **Select Faces and Edges** tool is chosen to set the priority to faces and edges. This tool is chosen by default, as a result you can select the faces and edges of the features. The **Select Sketch Features** tool is chosen to set the priority to sketched entities. The remaining two tools, **Select Groups** and **Select Wires** will be activated according to their respective environments when the different groups and wires are available. There are also other tools available in this area, which will be displayed in different modules.

Return

This tool is chosen to exit the sketching environment. Once you have finished drawing the sketch, choose this tool to proceed to the **Part** module where you can convert the sketch into a feature using the required tools.

Update/Local Update

This tool is chosen to update the design after editing.

Color Override

You can use this drop-down list to apply different types of colors or styles to the selected features or component to improve its appearance. It is much easier to identify different components, parts, and assemblies when proper color codes are applied to them.

RIBBON AND TABS

You might have noticed that there is no command prompt in Autodesk Inventor. The complete designing process is carried out by invoking the commands from the tabs in the **Ribbon**.

The **Ribbon** is a long bar available below the **Quick Access Toolbar**. You can change the appearance of the **Ribbon** as per your need. To do so, right-click on it; a shortcut menu will be displayed. Choose **Ribbon Appearance** from this shortcut menu to invoke a cascading menu. Next, choose the required option from the cascading menu.

Autodesk Inventor provides you with different tabs while working with various design environments. This means that the tabs available in the **Ribbon** while working with the **Part**, **Assembly**, **Drawing**, **Sheet Metal**, and **Presentation** environments will be different.

You can also display the toolbars. To do so, choose the **Customize** button from the **Options** panel of the **Tools** tab in the **Ribbon**; the **Customize** dialog box will be displayed. Choose the **Toolbars** tab from this dialog box; the list of all toolbars will be displayed. Select the required toolbar and then choose the **Show** button; the selected toolbar will appear on the screen. Choose the **Close** button to close the **Customize** dialog box.



Tip. In Autodesk Inventor Professional 2011, the messages and prompts are displayed at the **Status Bar** which is available at the lower left corner of the Autodesk Inventor window.

Sketch Tab

This is one of the most important tabs in the **Ribbon**. All tools for creating the sketches of the parts are available in this tab. The **Sketch** tab is available only in the sketching environment. The **Sketch** tab is shown in Figure 1-7.

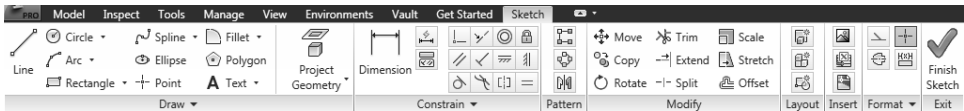


Figure 1-7 The Sketch tab

Inventor Precise Input Toolbar

You are now aware of the fact that Autodesk Inventor does not provide you with any command prompt. Because of this, you will be restricted from entering the precise values of the sketcher entities. But this problem has been taken care of in Autodesk Inventor by providing you with a very important toolbar called the **Inventor Precise Input** toolbar. This toolbar is used to enter the precise values for the coordinates of the sketcher entities. This toolbar is also available in the **Drawing** and **Assembly** modules for providing precise values. The **Inventor Precise Input** toolbar is shown in Figure 1-8.

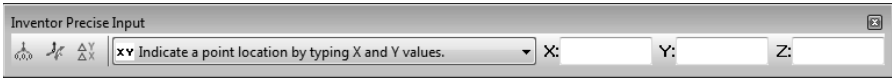


Figure 1-8 The Inventor Precise Input toolbar

Model Tab

This is the second most important tab provided in the **Part** module. Once the sketch is completed, you need to convert it into a feature using the modeling commands. This tab

provides all modeling tools that can be used to convert the sketch into a feature. The tools in the **Model** tab are shown in Figure 1-9.

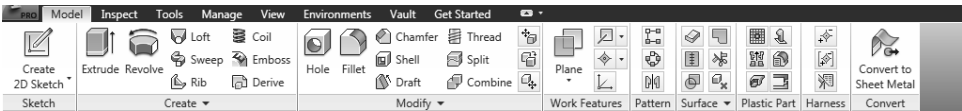


Figure 1-9 The Model tab

The **Create 2D Sketch** button in the **Sketch** panel of the **Model** tab is used to draw a 2D sketch in the sketching environment and is chosen by default when you start a new file in the **Part** module. As the first feature in most designs is a sketched feature, you can directly start working on the sketch of the feature. Once you have completed a sketch, you can choose either the **Return** button from the **Quick Access Toolbar** or the **Finish Sketch** button from the **Exit** panel of the **Sketch** tab in the **Ribbon** to exit the sketching environment. Whenever you need to draw the 2D sketch for another feature, choose this button again. You will be prompted to select the plane for sketching the feature. Once you define the new sketching plane, the sketching environment will be activated.

Sheet Metal Tab

This tab provides the tools that are used to create sheet metal parts. This toolbar will be available only when you are in the sheet metal environment. You can switch from the Modeling environment to the Sheet Metal environment by choosing the **Convert to Sheet Metal** tool from the **Convert** panel of the **Model** tab in the **Ribbon**. The tools in the **Sheet Metal** tab are shown in Figure 1-10.

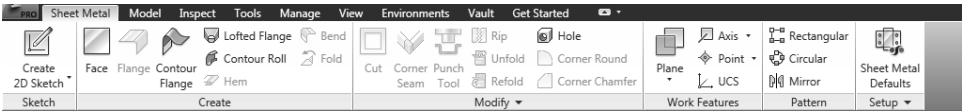


Figure 1-10 The Sheet Metal tab

Assemble Tab

This tab will be available only when you open any assembly template (with extension .iam) from the **New File** dialog box. This tab provides you all tools that are required for assembling components. The tools in the **Assemble** tab are shown in Figure 1-11.

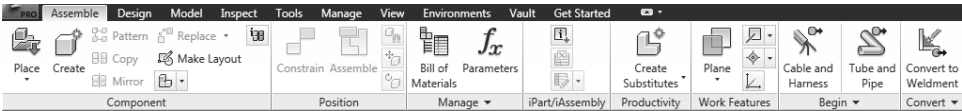


Figure 1-11 The Assemble tab

Place Views Tab

This tab provides the tools that are used to create different views of the components. This tab will be available only when you are in the Drafting environment. The tools in the **Place Views** tab are shown in Figure 1-12.

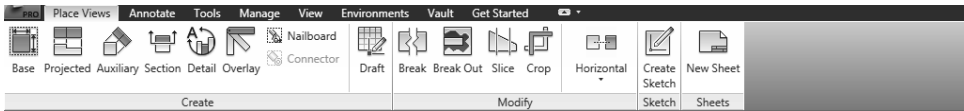


Figure 1-12 The Place Views tab

Presentation Tab

This tab provides the tools that are used to create different presentation views of the components. This tab will be available only when you open any presentation template (with extension .ipn) in the New File dialog box. The tools in the Presentation tab are shown in Figure 1-13.

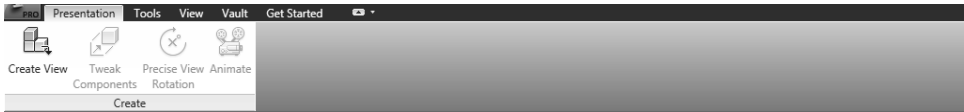


Figure 1-13 The Presentation tab

Tools Tab

This tab contains tools that are mainly used for setting the preferences and customizing the Autodesk Inventor interface. This tab is available in almost all environments. The tools in the Tools tab are shown in Figure 1-14.

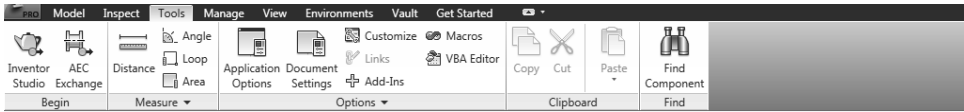


Figure 1-14 The Tools tab

View Tab

The tools in this tab enable you to the control the view, orientation, appearance, and visibility of objects and view windows. This tab is available in almost all environments. The tools in the View tab are shown in Figure 1-15.

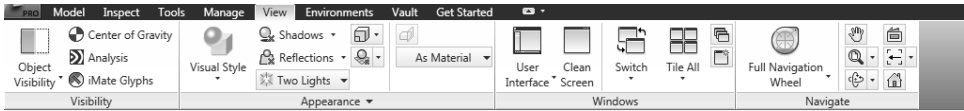


Figure 1-15 The View tab

The tools of a particular tab are arranged in different panels in the Ribbon. Some of the panels and tools have an arrow on the right, refer to Figure 1-16. These arrows are called down arrows. When you choose these down arrows, some more tools will be displayed as drop-downs, see Figure 1-16.

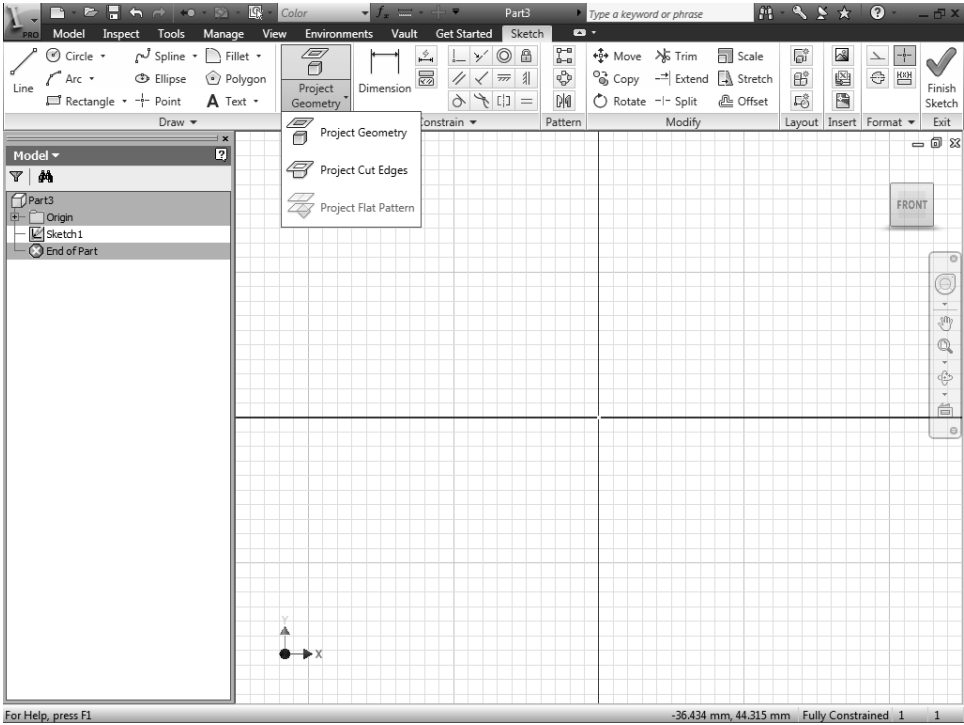


Figure 1-16 More tools displayed upon choosing the down arrow on the right of a tool in the Ribbon

Navigation Bar

The **Navigation Bar** is located on the right of the graphics area and contains tools that are used to make the designing process easier and quicker. The navigation tools also help you control the view and orientation of components in the drawing window. The **Navigation Bar** is shown in Figure 1-17

Browser Bar

The **Browser Bar** is available below the **Ribbon** on the left of the drawing window. It displays all the operations performed during the designing process in a sequence. All these operations are displayed in the form of a tree view. You can undock the **Browser Bar** by dragging it from its position to other position. The contents of the **Browser Bar** are different for different environments of Autodesk Inventor. For example, in the **Part** module, it displays various operations that were used in creating the part. Similarly, in the **Assembly** module, it displays all the components along with the constraints that were used to assemble them.

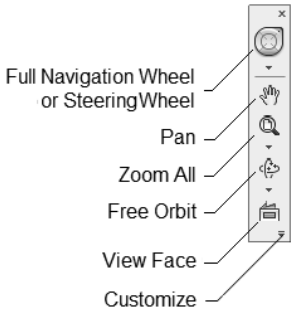


Figure 1-17 The Navigation Bar

UNITS FOR DIMENSIONS

In Autodesk Inventor, you can set units at any time by using the **Document Settings** dialog box. You can invoke this dialog box by choosing the **Document Settings** tool from the **Options** panel in the **Tools** tab. After invoking this dialog box, choose the **Units** tab in the dialog box; various areas related to units will be displayed. The options in the **Units** area are used to set the units. To set the unit for linear dimension, select the required unit from the **Length** drop-down. Similarly, to set the unit for angular dimension, select the required unit from the **Angle** drop-down. Next, choose the **OK** button to apply the specified settings and close the dialog box. If you want to apply the specified settings without closing the dialog box, choose the **Apply** button. If you choose the **Apply** button, the **OK** button is replaced by **Close**. Now, you can choose the **Close** button to close the dialog box.

IMPORTANT TERMS AND THEIR DEFINITIONS

Before you proceed further in Autodesk Inventor, it is very important for you to understand the following terms, which are widely used in this book.

Feature-based Modeling

A feature is defined as the smallest building block that can be modified individually. In Autodesk Inventor, the solid models are created by integrating a number of these building blocks. Therefore, the models in Autodesk Inventor are a combination of a number of individual features. These features understand their fit and function properly. As a result, these can be modified, whenever required. Generally, these features automatically adjust their values, if there is any change in their surroundings. For example, a feature created by cutting right through the base feature will automatically adjust its depth, if you increase the depth of the base feature. This provides greater flexibility to the design.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the standard properties or parameters to define the shape and size of a geometry. The main function of this property is to derive the selected geometry to the new size or shape without considering its original size or shape. For example, you can derive a line of 20 mm that was initially drawn at an angle of 45° to a line of 50 mm and change its orientation to 90°. This property makes the designing process very easy. This is because now you do not need to draw the sketch to the actual dimensions that are required. You just need to draw the sketch to some relative dimensions, and then this solid modeling tool will drive it to the actual values you require.

Bidirectional Associativity

As mentioned earlier, this solid modeling tool does not restrict its capabilities to the 3D solid output. It is also capable of highly effective assembly modeling, drafting, and presentations. There exists a bidirectional associativity between all these environments of Autodesk Inventor. This means that every time, there exists a link between all the environments of Autodesk Inventor. This link ensures that if any modification is made in the model in any one environment, it is automatically reflected in the other environments.

Adaptive

This is a highly effective property that is included in the designing process of this solid modeling tool. In any design, there are a number of components that can be used in various places with a small change in their shape and size. This property makes the part or the feature adapt to its environment. It also ensures that the adaptive part changes its shape and size as soon as it is constrained to other parts. This considerably reduces the time and effort required in creating similar parts in the design.

Design Doctor

The Design Doctor is one of the most important parts of the designing process used in the Autodesk Inventor software. It is a highly effective tool to ensure that the entire design process is error free. The main purpose of the Design Doctor is to make you aware of any problem in the design. The Design Doctor works in the following three steps:

Selecting the Model and Errors in the Model

In this step, the Design Doctor selects the sketch, part, assembly, and so on and determines the errors in it.

Examining Errors

In this step, it examines the errors in the selected design. Each of the errors is individually examined and the required solution is provided.

Providing Solutions for Errors

This is the last step of the working of the Design Doctor. Once it has individually examined each of the errors, it suggests solutions for them. It provides you with a list of methods that can be utilized to remove the errors from the design.

Constraints

These are the logical operations that are performed on the selected design to make it more accurate or define its position with respect to the other design. There are four types of constraints in Autodesk Inventor. All these types are explained next.

Geometric Constraints

These logical operations are performed on the basic sketching entities to relate them to the standard properties like collinearity, concentricity, perpendicularity, and so on. Autodesk Inventor automatically applies these geometric constraints to the sketcher entities at the time of their creation. You do not have to use an extra command to apply these constraints on to the sketcher entities. However, you can also manually apply these geometric constraints on to the sketcher entities. There are twelve types of geometric constraints.

Perpendicular Constraint

This constraint is used to make the selected line segment normal to another line segment.

Parallel Constraint

This constraint is used to make the selected line segments parallel.

Coincident Constraint

This constraint is used to make two points or a point and a curve coincident.

Concentric Constraint

This constraint forces two selected curves to share the same center point. The curves that can be made concentric are arcs, circles, or ellipses.

Collinear Constraint

This constraint forces two selected line segments or ellipse axes to be placed in the same line.

Horizontal Constraint

This constraint forces the selected line segment to become horizontal.

Vertical Constraint

This constraint forces the selected line segment to become vertical.

Tangent

This constraint is used to make the selected line segment or curve tangent to another curve.

Equal

This constraint forces the selected line segments to become equal in length. It can also be used to force two curves to become equal in radius.

Smooth

This constraint adds a smooth constraint between a spline and another entity so that at the point of connection, the line is tangent to the spline.

Fix

This constraint fixes the selected point or curve to a particular location with respect to the coordinate system of the current sketch.

Symmetric

This constraint forces the selected sketched entities to become symmetrical about a sketched line segment, which may or may not be a center line.

Assembly Constraints

The assembly constraints are the logical operations performed on the components in order to bind them together to create an assembly. These constraints are applied to reduce the degrees of freedom of the components. There are four types of assembly constraints which are discussed next.

Mate

The **Mate** constraint is used to make the selected faces of different components coplanar. The model can be placed facing in the same direction or in the opposite direction. You can also specify some offset distance between the selected faces.

Angle

The **Angle** constraint is used to place the selected faces of different components at some angle with respect to each other.

Tangent

The **Tangent** constraint is used to make the selected face of a component tangent to the cylindrical, circular, or conical faces of the other component.

Insert

The **Insert** constraint forces two different circular components to share the same orientation of the central axis. It also makes the selected faces of the circular components coplanar.

Motion Constraints

The motion constraints are the logical operations performed on the components that are assembled using the assembly constraints. There are two types of motion constraints:

Rotation

The **Rotation** constraint is used to rotate one component of the assembly in relation to the other component.

Rotation-Translation Constraint

The **Rotation-Translation** constraint is used to rotate the first component in relation to the translation of the second component.

Transitional Constraints

The transitional constraints are also applied on the assembled components and are used to ensure that the selected face of the cylindrical component maintains contact with the selected faces of the other component when you slide the cylindrical component.

UCS to UCS Constraint

This constraint is used to constrain two components together by their UCS.

**Note**

The motion and transitional constraints are applied on the components that have already been assembled using the assembly constraints. Therefore, these constraints work along the degrees of freedom of the components that are not restricted using the assembly constraints.

Consumed Sketch

A consumed sketch is a sketch that is utilized in creating a feature using tools such as **Extrude**, **Revolve**, **Sweep**, **Loft**, and so on.

CYCLING THROUGH ENTITIES

While working on the complicated models, sometimes you may need to select the entities that are not visible in the current view or are hidden behind other entities. To select these types of entities, Autodesk Inventor allows you to cycle through entities using a cycling tool, as shown in Figure 1-18. This tool is displayed automatically when you hold the cursor at a point where more than one entity is available. This cycling tool consists of two arrows at each end and a rectangle in between. The left arrow is used to cycle through the previous entities, the right arrow is used to cycle through the next entities, and the rectangle is used to select the highlighted entity. The current entity will be highlighted and displayed in red. Once the required entity is highlighted, move the cursor over the rectangle in the cycling tool and select it using the left mouse button. The highlighted entity will be selected and displayed in blue. Figure 1-18 shows the cycling tool displayed in the sketching environment to cycle through the sketched entities. You can use this tool in all modes and environments of Autodesk Inventor.

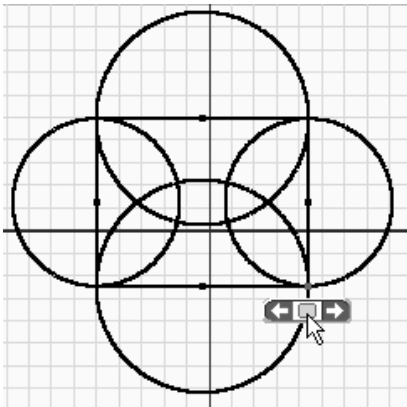


Figure 1-18 Cycling through entities

HOTKEYS

As mentioned earlier, there is no command prompt in Autodesk Inventor. However, you can use the keys on the keyboard to invoke some tools. The keys that can be used to invoke the tools are called hotkeys. Remember that the working of the hotkeys will be different for different environments. The use of hotkeys in different environments is given next.

Part Module

The hotkeys that can be used in the **Part** module and their functions are given next.

Hotkey	Function
E	Invokes the Extrude tool
R	Invokes the Revolve tool
H	Invokes the Hole tool
CTRL+SHIFT+L	Invokes the Loft tool
CTRL+SHIFT+S	Invokes the Sweep tool
F	Invokes the Fillet tool
CTRL+SHIFT+K	Invokes the Chamfer tool
D	Invokes the Draft tool

CTRL+SHIFT+R	Invokes the Rectangular Pattern tool
CTRL+SHIFT+O	Invokes the Circular Pattern tool
CTRL+SHIFT+M	Invokes the Mirror tool
]	Invokes the Work Plane tool
/	Invokes the Work Axis tool
.	Invokes the Work Point tool
CTRL+W	Invokes the SteeringWheels
F6	Invokes the Home view

The following hotkeys are used in the sketching environment:

Hotkey	Function
L	Invokes the Line tool
C	Invokes the Center Point Circle tool
D	Invokes the Dimension tool
X	Invokes the Trim tool
F7	Invokes the Slice Graphics tool
F8	Displays all constraints
F9	Hides all constraints

Assembly Module

In addition to the hotkeys of the part modeling tool, the following hot keys can also be used in the **Assembly** module:

Hotkey	Function
P	Invokes the Place tool
N	Invokes the Create tool
C	Invokes the Constraint tool
CTRL+H	Invokes the Replace tool
CTRL+SHIFT+H	Invokes the Replace All tool
V	Invokes the Move Component tool
G	Invokes the Rotate Component tool
A	Invokes the Analyze Interference tool

Drawing Module

The hotkeys that can be used in the **Drawing** module are given next.

Hotkey	Function
B	Invokes the Balloon tool
D	Invokes the Dimension tool
O	Invokes the Ordinate Set tool
F	Invokes the Feature Control Frame tool

In addition to these keys, you can also use some other keys for the ease of designing. Note that you will have to hold some of these keys down and use them in combination with the pointing device. These hotkeys are given next.

Hotkey	Function
F1	Invokes the Help command
F2	Invokes the Pan tool
F3	Invokes the Zoom tool
F4	Invokes the Free Orbit tool
F5	Previous view
SHIFT+F5	Next view
ESC	Aborts the current command
SPACEBAR	Invokes the recently used tool
T (In Presentation module)	Invokes the Tweak Components tool

Customizing Hotkeys

You can customize the settings of hotkeys. To do so, choose the **Customize** tool from the **Options** panel of the **Tools** tab in the **Ribbon**; the **Customize** dialog box will be displayed. Next, choose the **Keyboard** tab; the list of all available commands will be displayed, as shown in Figure 1-19. The options corresponding to the **Keyboard** tab are discussed next.

Categories

Select the required category of command from this drop-down list; the commands related to the selected category will be listed in the list box.

Filter

You can further shortlist the displayed commands from this drop-down list. If you select the **All** option, all commands related to the selected category will be displayed. If you select the **Assigned** option, then the commands to which the hotkeys are assigned will be displayed. Similarly, if you select the **Unassigned** option, then the commands to which the hotkeys are not assigned will be displayed.

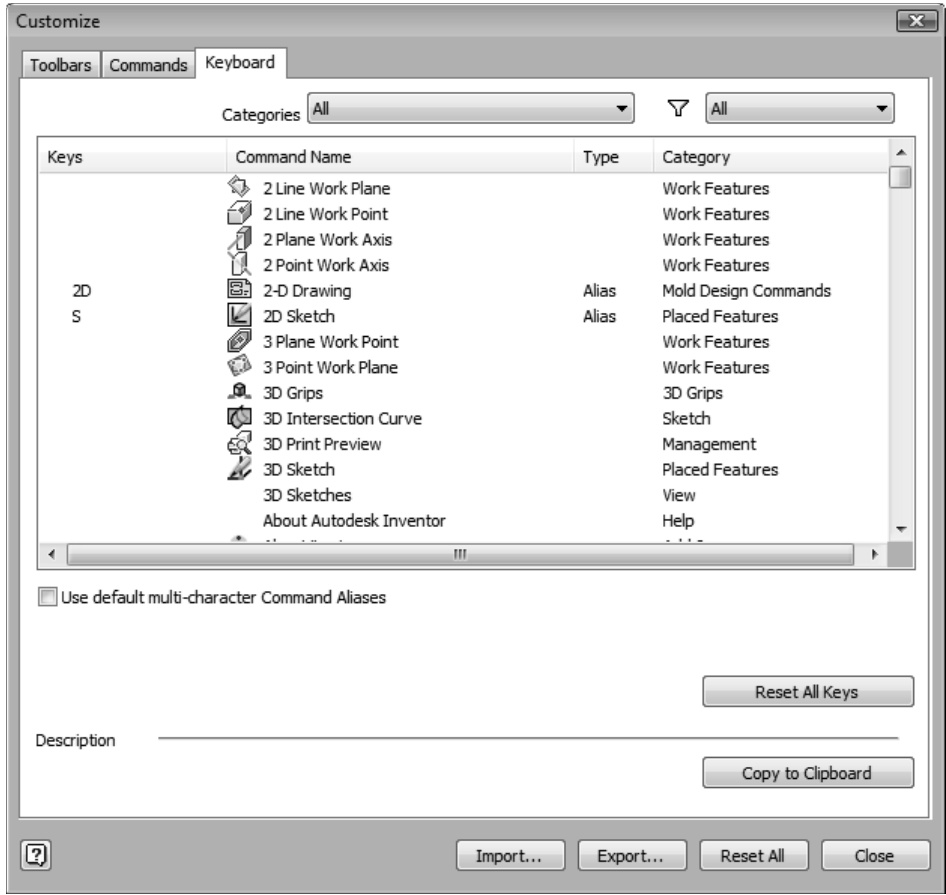


Figure 1-19 The *Customize* dialog box displaying various commands in the **Keyboard** tab

List Box

The list box has four columns: **Keys**, **Command Name**, **Type**, and **Category**. The **Key** column displays the hotkeys assigned to the commands. The name of the command, its type, and category will be listed in the **Command Name**, **Type**, and **Category** columns, respectively.

To assign hotkeys to a tool, click in the **Keys** column that is associated to the command; an edit box will be displayed. In this edit box, enter the shortcut key that you want to assign. To accept the settings, click on the tick-mark provided at the right side of this edit box. Else, click on the cross-mark provided next to the tick-mark.

Reset All Keys

The **Reset All Keys** button is used to remove all customized hotkeys and restore the default hotkeys.

Copy to Clipboard

Choose this button to copy the contents of the **Keyboard** tab and paste them in other document.

Import

Choose this button to restore the customized settings from the .xml format. Note that before importing the file, all Autodesk Inventor files must be closed.

Export

Choose this button to save the customized settings in the .xml format. Make sure that all Autodesk Inventor files are closed before choosing this button.

Reset All

This button is used to reset all the customized settings of environments, toolbars, and menus. Make sure that all Autodesk Inventor files are closed before choosing this button.

Close

Choose this button to close the **Customize** dialog box.

CREATING THE SKETCH

After starting Autodesk Inventor, you can start creating the model in the Part environment. But before creating the model, you need to create its sketch in the Sketching environment. The Sketching environment is invoked automatically when you start a new part file. But if you want to invoke the Sketching environment manually, choose the **Create 2D Sketch** tool from the **Sketch** drop-down in the **Sketch** panel of the **Model** tab, see Figure 1-20. On choosing this tool, the sketching environment is invoked and you can create the 2D sketches. If you choose the **Create 3D sketch** tool from the **Sketch** panel, you can create the 3D sketches.

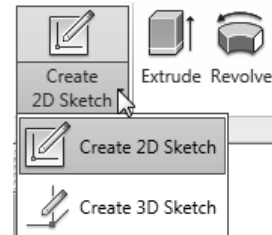
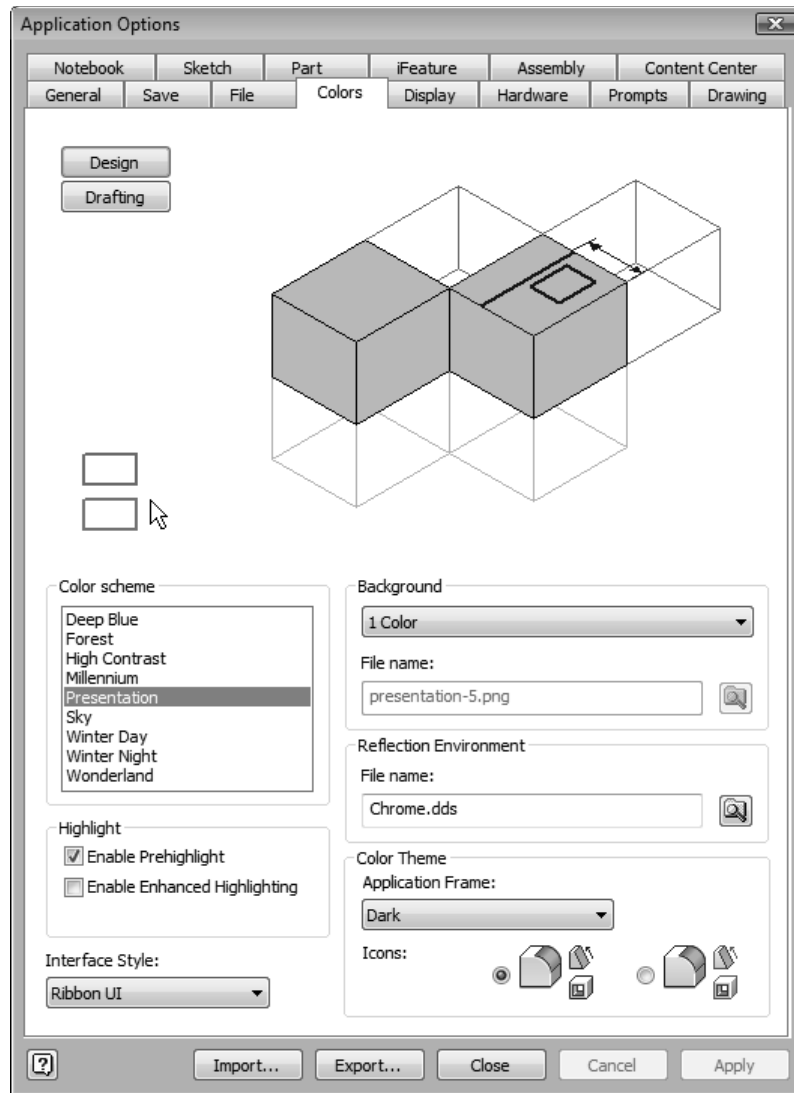


Figure 1-20 Tools in the Sketch drop-down

COLOR SCHEME

Autodesk Inventor allows you to use various color schemes to set the background color of the screen and for displaying the entities on the screen. Note that this book uses the **Presentation** color scheme with a single color background. To change the color scheme, choose the **Application Options** tool from the **Options** panel of the **Tools** tab in the **Ribbon**; the **Application Options** dialog box will be displayed. Choose the **Colors** tab to display the predefined colors. Next, select the **Presentation** option from the **Color scheme** list box in the **Colors** tab. Select **1 Color** from the drop-down list in the **Background** area, refer to Figure 1-21. Choose **Apply** to apply the color scheme to the Autodesk Inventor environment, and then choose **Close**. Note that all the files you open henceforth will use this color scheme.



*Figure 1-21 The **Application Options** dialog box with the required options set in the **Colors** tab*

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. When you start a new session of Autodesk Inventor Professional 2011, only the **Start a new file** button will be available in the **Quick Launch** area of the **Open** dialog box. (T/F)

2. The **Inventor Precise Input** toolbar is used to specify the precise values for the coordinates of sketcher entities. (T/F)
3. The tools in the **Model** tab enable you to control the view, orientation, appearance, and visibility of objects and view windows. (T/F)
4. You can invoke the **Line** tool by using the _____ hotkey.
5. Press _____ to invoke the recently used tool.
6. Choose the _____ button from the **Customize** dialog box to restore the customized settings in the .xml format.

Review Questions

Answer the following questions:

1. There are twelve types of geometric constraints in Autodesk Inventor. (T/F)
2. Design Doctor works in five steps. (T/F)
3. You can invoke the **Trim** tool by pressing the X key. (T/F)
4. You can use the _____ drop-down list to apply different types of color or style to the selected feature or component to improve its appearance. (T/F)
5. You can invoke the **Analyze Interference** tool in the **Assembly** module by pressing the _____ key.
6. The _____ button is used to draw a 2D sketch in the sketching environment and is chosen by default when you start a new file in the **Part** module.

Answers to Self-Evaluation Test

1. T, 2. T, 3. F, 4. L, 5. SPACEBAR, 6. Import