

# 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 usage of various hotkeys.*
- *Customize hotkeys.*
- *Modify the color scheme in Autodesk Inventor.*

## INTRODUCTION TO Autodesk Inventor 2014

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 and joints. All the

assembly constraints and joints 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.

## Mold Design Module

This module is used to create mold design by integrated mold functionality and content libraries using the intelligent tools and catalogs provided in mold design module. In this module, you can quickly generate accurate mold design directly from digital prototypes.

## GETTING STARTED WITH Autodesk Inventor

Install Autodesk Inventor on your system; the Autodesk Inventor Professional 2014 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 > All Programs > Autodesk > Autodesk Inventor 2014 > Autodesk Inventor Professional 2014**, refer to 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 2014 will be displayed along with the **Welcome** dialog box as shown in Figure 1-2. Using this dialog box, you can perform some of the tasks such as starting new file, opening existing file, and so on.

You will learn more about the **Welcome** dialog box in Chapter 2. You can exit the **Welcome** dialog box by choosing the **Close** button. Figure 1-3 shows the initial screen of Autodesk Inventor 2014 after closing the **Welcome** dialog box. You can view the recent enhancement and information related to Autodesk Inventor 2014 by choosing the **What's New** button from the **Ribbon**.

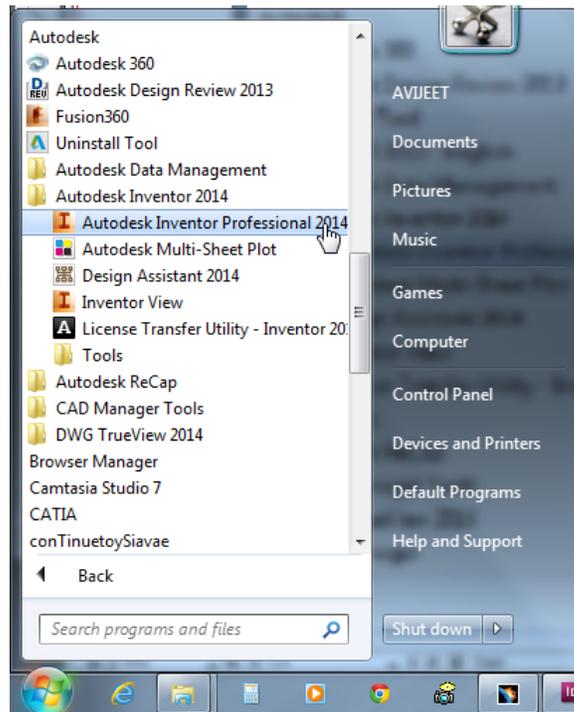


Figure 1-1 Starting Autodesk Inventor using the taskbar

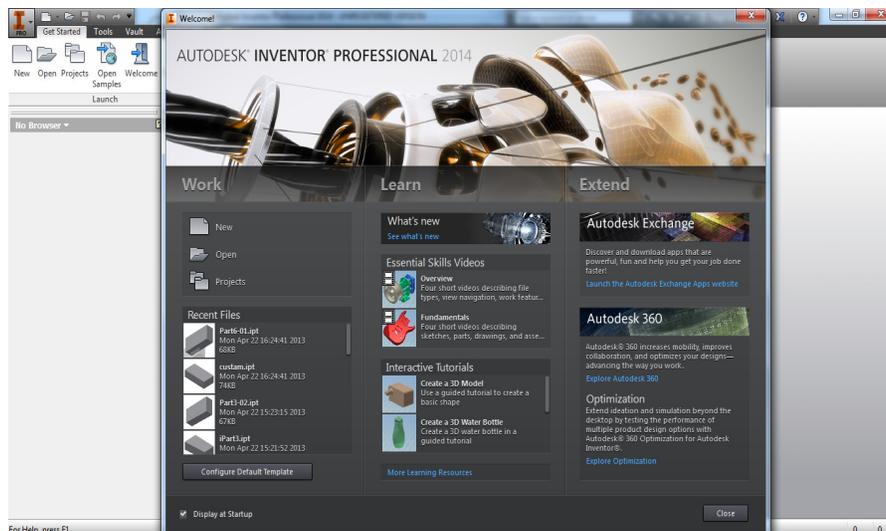
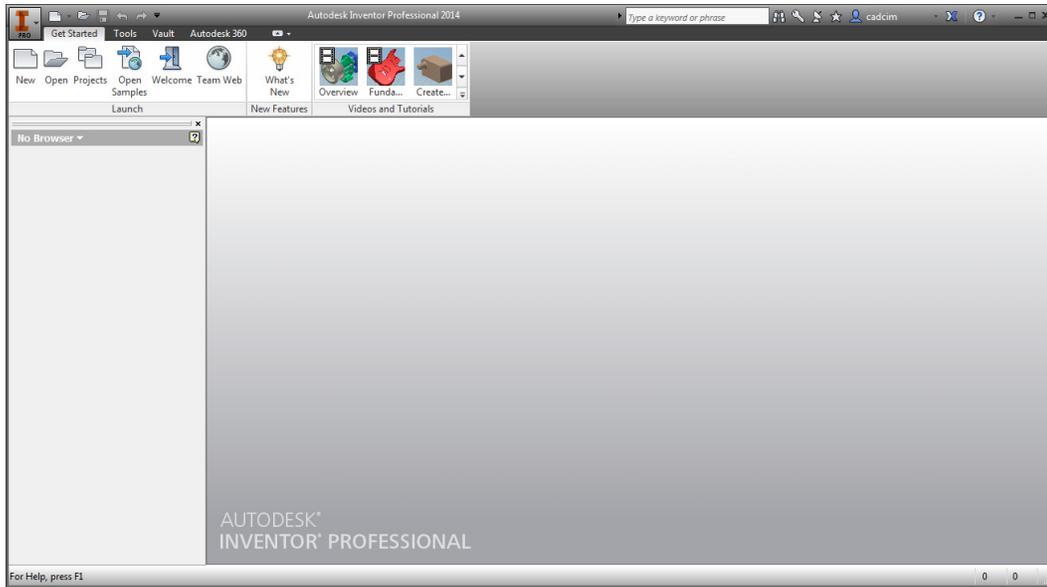
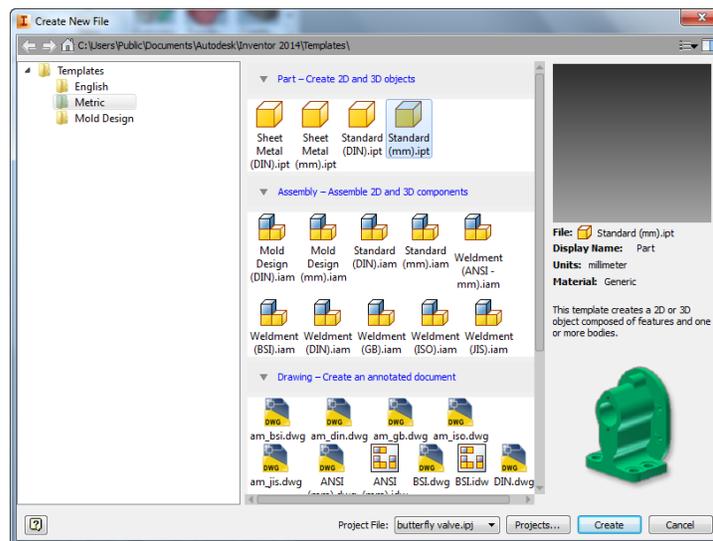


Figure 1-2 Initial screen of Autodesk Inventor Professional 2014 along with **Welcome** dialog box



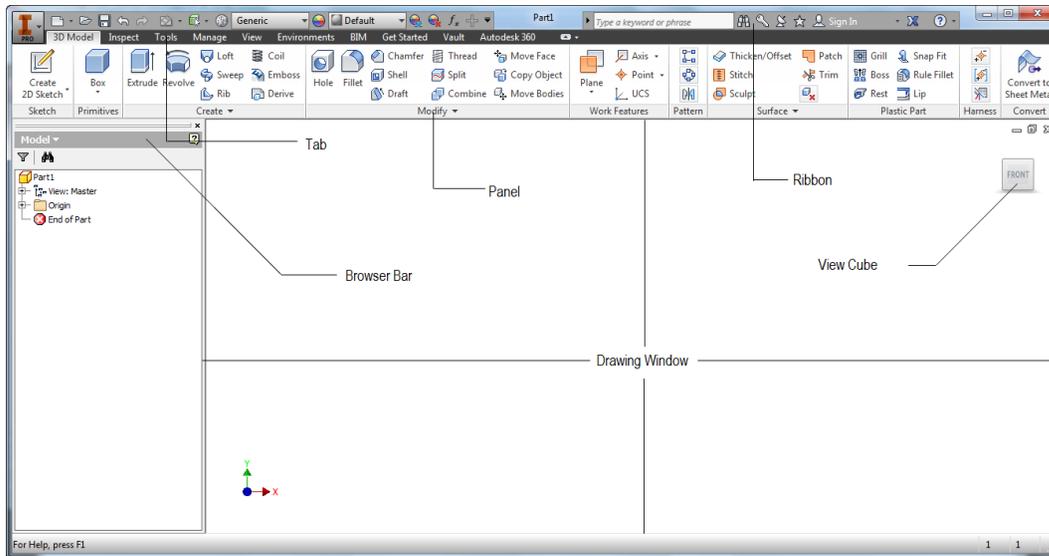
*Figure 1-3 Initial screen display of Autodesk Inventor Professional 2014*

To start a new part file, choose the **New** tool from the **Launch** panel of the **Get Started** tab in the **Ribbon**; the **Create New File** dialog box will be displayed, as shown in Figure 1-4. This dialog box is used to start a new file of Autodesk Inventor. Choose the **Metric** tab from the **Create 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, refer to Figure 1-5 and you can start working in this file. The figure also displays various components of the screen.



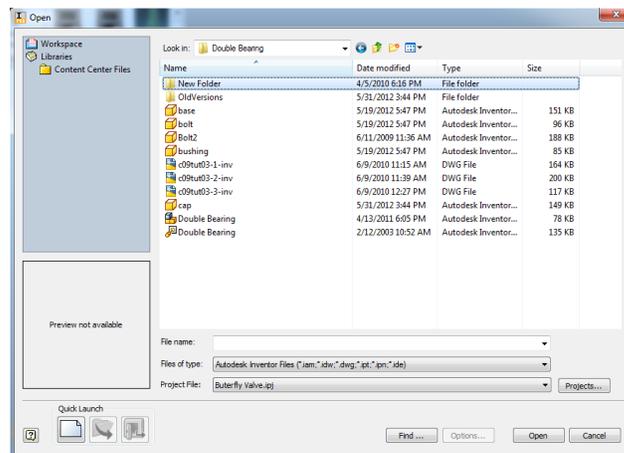
*Figure 1-4 The Create New File dialog box*

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 on right-clicking 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 1-5 The Sketching environment of Autodesk Inventor and its main components*

You can also start a new part file by using the **Quick Launch** area of the **Open** dialog box, see Figure 1-6. The **Open** dialog box can be invoked by choosing the **Open** button from the **Get Started** tab in the **Ribbon**. When you start a new session of Autodesk Inventor Professional 2014, only the **Start a new file** button will be activated in the **Quick Launch** area of the **Open** dialog box.



*Figure 1-6 The Open dialog box*

## 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 1-7. Some of the important options in this toolbar are discussed next.

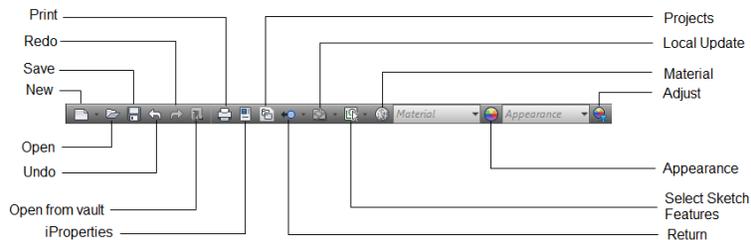


Figure 1-7 The Quick Access Toolbar

### Select

This tool is used to set the selection priority. If 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 become 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.

### Appearance 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 **Ribbon** tab from this dialog box; the list of all toolbars will be displayed. Select the required toolbar and then choose the **Add** button; the selected toolbar will appear on the screen. Choose the **Close** button to close the **Customize** dialog box.



**Tip.** In Autodesk Inventor Professional, 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 1-8.



Figure 1-8 The Sketch tab

## Inventor Precise Input Toolbar

Inventor provides you with the **Inventor Precise Input** toolbar to specify precise value. 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 1-9.

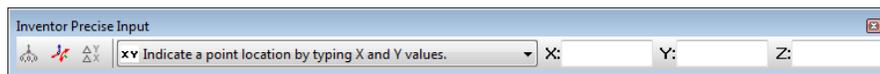


Figure 1-9 The Inventor Precise Input toolbar

## 3D 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 **3D Model** tab are shown in Figure 1-10.



Figure 1-10 The 3D Model tab

The **Create 2D Sketch** button in the **Sketch** panel of the **3D Model** tab is used to draw a 2D sketch in the Sketching environment. 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**. Whenever you need to draw the 2D sketch for another feature, choose this button; 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 **3D Model** tab in the **Ribbon**. The tools in the **Sheet Metal** tab are shown in Figure 1-11.



Figure 1-11 The Sheet Metal tab

## Assemble Tab

This tab will be available only when you open any assembly template (with extension *.iam*) from the **Create 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 1-12.

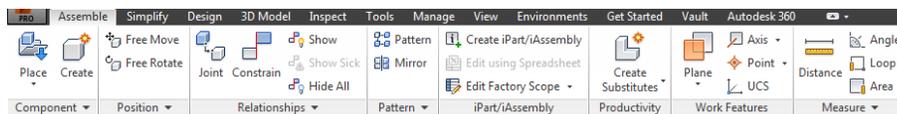


Figure 1-12 The Assemble tab

## 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 1-13.

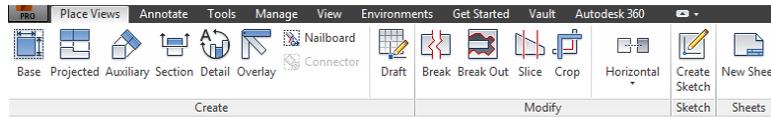


Figure 1-13 The Place Views tab

## 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 **Create New File** dialog box. The tools in the **Presentation** tab are shown in Figure 1-14.



Figure 1-14 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 1-15.

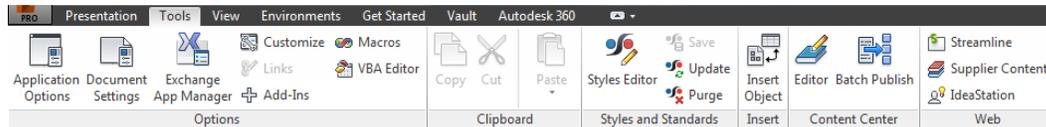


Figure 1-15 The Tools tab

## View Tab

The tools in this tab enable you to 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 1-16.

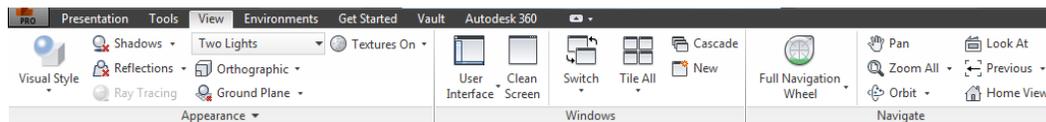
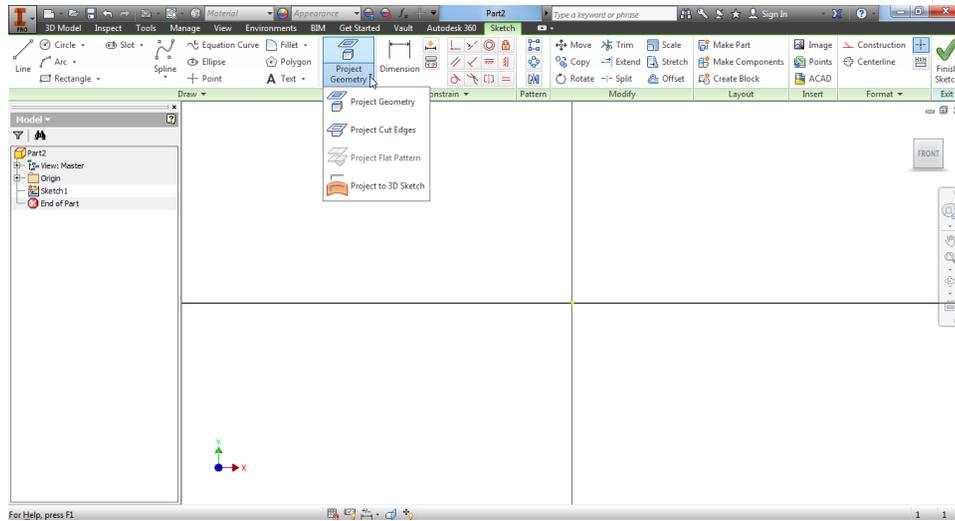


Figure 1-16 The View tab

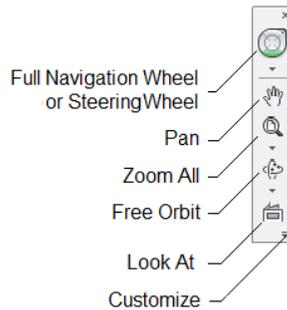
The tools of a particular tab are arranged in different panels in the **Ribbon**. Some of the panels and tools have an arrow on the right, refer to Figure 1-16. These arrows are called down arrows. When you choose these down arrows, some more tools will be displayed in the drop-downs, see Figure 1-17.



*Figure 1-17* 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 to control the view and orientation of the components in the drawing window. The **Navigation Bar** is shown in Figure 1-18.



*Figure 1-18* The **Navigation Bar**

## Browser Bar

The **Browser Bar** is available below the **Ribbon** on the left in 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.

## 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.

### Parametric Modeling

The parametric nature of a software package is 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 degrees 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 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 five 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 the same direction or 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.

**Symmetry**

The **Symmetry** constraint is used to make two selected components symmetric to each other about a symmetric plane so that both components remain equidistant from the plane.

**Assembly Joints**

The assembly joints are the logical operations performed on the components in order to join them together to create an assembly. These joints allow motion between the connected components or in the assembly. There are seven types of assembly joints which are discussed next.

**Automatic**

The Automatic joint is used to apply best suitable type of joints automatically between the connecting components of the assembly. The type of joint to be applied automatically will depend upon the selected geometry.

**Rigid**

The Rigid joint removes all the degrees of freedom of the component. As a result, the components after applying rigid joints, can not move in any direction. The Rigid joint is used to fix two parts rigidly. All the DOFs between the selected parts get eliminated and act as a single component when any motion will be applied to any of the direction.

**Rotational**

The Rotational joint allows the rotational motion of a component along the axis of a cylindrical component.

**Slider**

The Slider joint allows the movement of a component along a specified path. The component will be joined to translate in one direction only. You can specify only one translation degree of freedom in slider joint. Slider joint are used to simulate the motion in linear direction.

**Cylindrical**

The Cylindrical joint allows a component to translate along the axis of a cylindrical component as well as rotate about the axis. You can specify one translation degree of freedom and one rotational degree of freedom in the Cylindrical joint.

**Planar**

The Planar joint is used to connect the planar faces of two components. The components can slide or rotate on the plane with two translation and one rotational degree of freedom.

**Ball**

The Ball joint is used to create a joint between two components such that both the components remain in touch with each other and at the same time the movable component can freely rotate in any direction. To create a ball joint between two components, you need to specify one point from each component. The joints thus created will generate three undefined rotational DOFs and restrict the other three DOFs at a common point.

**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 that are discussed next.

**Rotation**

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

**Rotation-Translation**

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 UCSs.

**Note**

*The motion and transitional constraints are applied to 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.*

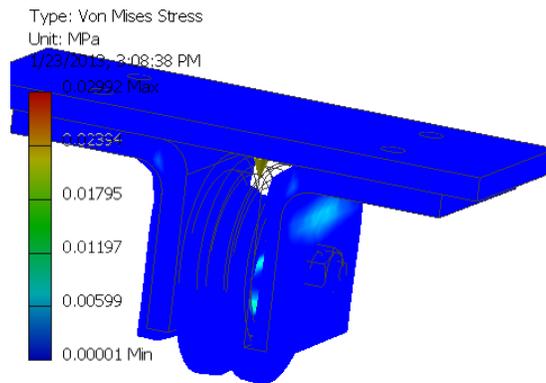
**Consumed Sketch**

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

**STRESS ANALYSIS ENVIRONMENT**

In Autodesk Inventor, you are provided with stress analysis environment which is an analysis tool to execute the static and model stress analysis. You can calculate the displacement and stresses developed in a component with the effect of material and various loading conditions applied on a model. A component fails when the stress applied on it reaches beyond a certain

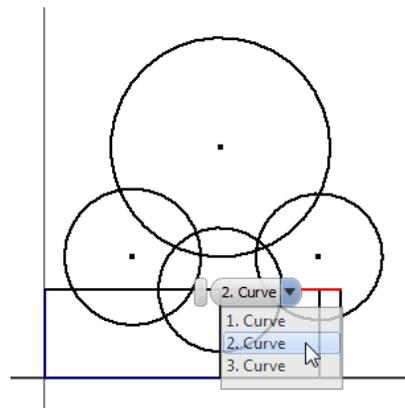
permissible limit. Figure 1-19 shows the Von Mises stress plot of model designed in Autodesk Inventor and analyzed using the analysis tools.



*Figure 1-19 The resultant model with Von Mises Stress*

## SELECT OTHER BEHAVIOR

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 do so, Autodesk Inventor provides you with the **Select Other** feature, which is displayed automatically when you hover the cursor at a point where more than one entity is available. To select any entity, click on the down arrow; a flyout will be displayed. Select the desired entity from the flyout; the selected entity will be displayed in blue. Figure 1-20 shows the **Select Other** flyout displayed in the modelling environment. You can use this tool in all the modes and environments of Autodesk Inventor.



*Figure 1-20 Selecting the entities from the Select Other flyout*

## 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 <b>Extrude</b> tool
R + ENTER	Invokes the <b>Revolve</b> tool
H + ENTER	Invokes the <b>Hole</b> tool
CTRL+SHIFT+L	Invokes the <b>Loft</b> tool
CTRL+SHIFT+S	Invokes the <b>Sweep</b> tool
F	Invokes the <b>Fillet</b> tool
CTRL+SHIFT+K	Invokes the <b>Chamfer</b> tool
D	Invokes the <b>Draft</b> tool
CTRL+SHIFT+R	Invokes the <b>Rectangular Pattern</b> tool
CTRL+SHIFT+O	Invokes the <b>Circular Pattern</b> tool
CTRL+SHIFT+M	Invokes the <b>Mirror</b> tool
J	Invokes the <b>Work Plane</b> tool
/	Invokes the <b>Work Axis</b> tool
.	Invokes the <b>Work Point</b> tool
CTRL+W	Invokes the <b>SteeringWheels</b>
F6	Invokes the <b>Home view</b>

The following hotkeys are used in the Sketching environment:

Hotkey	Function
L + ENTER	Invokes the <b>Line</b> tool
C	Invokes the <b>Center Point Circle</b> tool
D	Invokes the <b>Dimension</b> tool
X	Invokes the <b>Trim</b> tool
F7	Invokes the <b>Slice Graphics</b> 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 + ENTER	Invokes the <b>Place</b> tool
N	Invokes the <b>Create</b> tool
C + ENTER	Invokes the <b>Constraint</b> tool
V	Invokes the <b>Move Component</b> tool
G	Invokes the <b>Rotate Component</b> tool
A + ENTER	Invokes the <b>Analyze Interference</b> tool

## Drawing Module

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

Hotkey	Function
B	Invokes the <b>Balloon</b> tool
D	Invokes the <b>Dimension</b> tool
O	Invokes the <b>Ordinate Set</b> tool
F	Invokes the <b>Feature Control Frame</b> 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 <b>Help</b> command
F2	Invokes the <b>Pan</b> tool
F3	Invokes the <b>Zoom</b> tool
F4	Invokes the <b>Free Orbit</b> tool
F5	Previous view
SHIFT+F5	Next view
ESC	Aborts the current command
SPACEBAR	Invokes the recently used tool
T (In <b>Presentation</b> module)	Invokes the <b>Tweak Components</b> 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 1-21. The options corresponding to the **Keyboard** tab are discussed next.

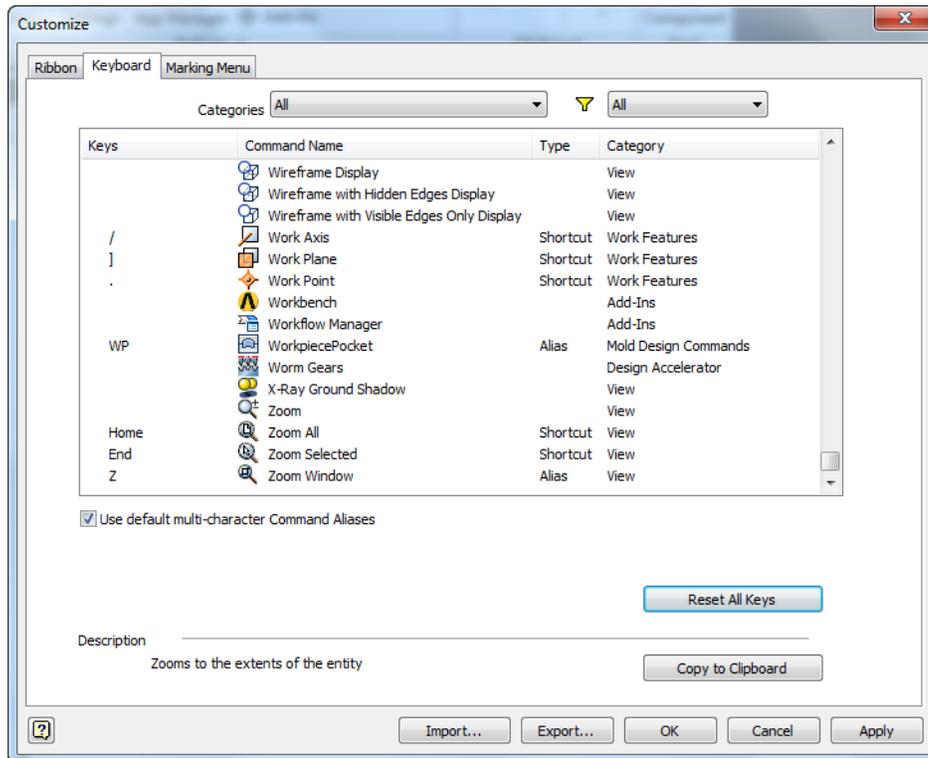


Figure 1-21 The **Customize** dialog box displaying various commands in the **Keyboard** tab

### 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.

## 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.

## 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. To do so, choose the **Create 2D Sketch** tool from the **Sketch** drop-down in the **Sketch** panel of the **3D Model** tab, see Figure 1-22. 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.

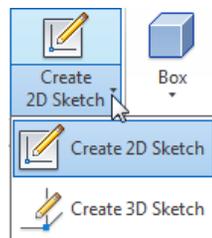


Figure 1-22 Tools in the **Sketch** drop-down

## MARKING MENU

Marking menu is a type of menu that consists of some of the tools and options which are commonly used in Autodesk Inventor software in different environments. Marking menu replaces the conventional right-click context menu. The Marking menu consists of different tools in different environments. For example, in the Sketching environment, the Marking menu consists of commonly used tools such as **Create Line**, **Two Point Rectangle**, **Done [ESC]**, **Trim**, **General Dimensions**, and so on. In the Modeling environment, it consists of tools and options such as **Extrude**, **Fillet**, **Hole**, **New Sketch**, and so on.

You can invoke a tool in Marking menu by using two modes: Marking mode and Menu mode. To invoke the Marking menu using the Menu mode, right-click anywhere in the graphic window; all menu items surrounding the cursor will be displayed. After invoking the Marking menu, you can choose the desired tool or option from it. To do so, move the cursor toward the desired tool; the tool is highlighted along with a marker ray. Next, choose the highlighted tool to invoke it.

The other mode, Marking mode, is also known as gesture behavior. It helps you to mark a trail and choose the desired tool. To choose a tool in the Marking mode, right-click and drag the cursor immediately in the direction of the desired tool.

Figure 1-23 shows a Marking menu invoked in the Sketching environment and Figure 1-24 shows a Marking menu which is invoked in the Modeling environment.

## 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 1-25. 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.



**Tip.** You can modify the tools listed in the Marking Menu. You can also turn the Marking Menu feature on or off using the option in the **User Interface** flyout in the **Windows** panel of the **View** tab in the **Ribbon**.

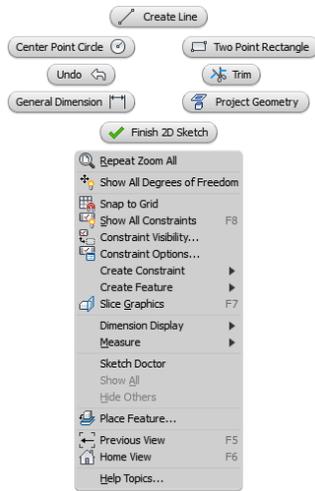


Figure 1-23 Marking menu available in the Sketching environment

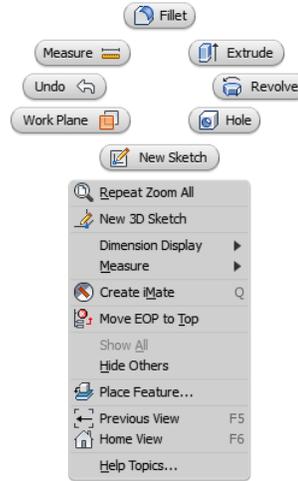


Figure 1-24 Marking menu available in the Modeling environment

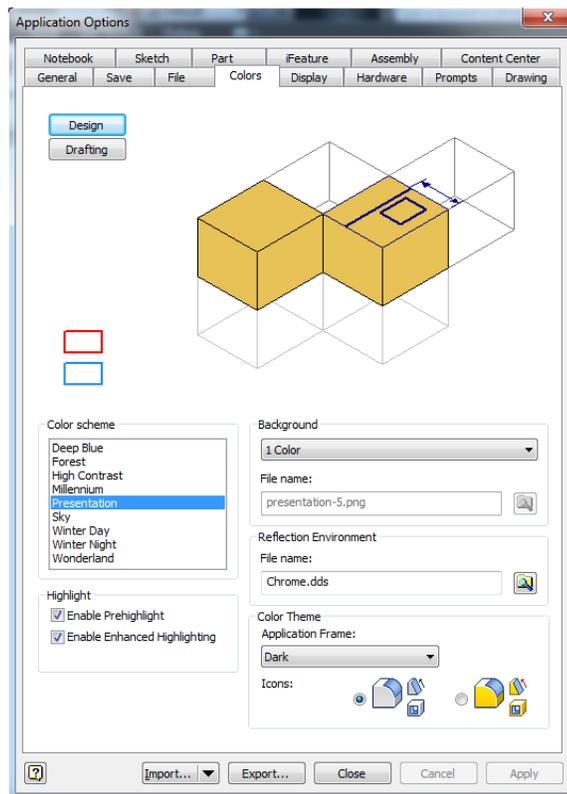


Figure 1-25 The Application Options dialog box with the required options set in the Colors tab

### 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 2014, 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 the sketcher entities. (T/F)
3. The tools in the **3D 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.
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