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 2021

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 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 complicacies 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 automatically applied to a sketch 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 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 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 or they have to take the help of some other animation software packages. However, this software package provides a module called the **Presentation** module using which 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 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 a sheet metal component. You can draw the sketch of the base sheet in this Sketching environment and then proceed to the sheet metal module to convert it into a 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; a shortcut icon of Autodesk Inventor Professional 2021 will automatically be created on the desktop. Double-click on this icon to start Autodesk Inventor.

When Autodesk Inventor is started for the first time, the system prepares itself by loading all the required files and then the **Welcome to Inventor 2021** window gets displayed, as shown in Figure 1-1.

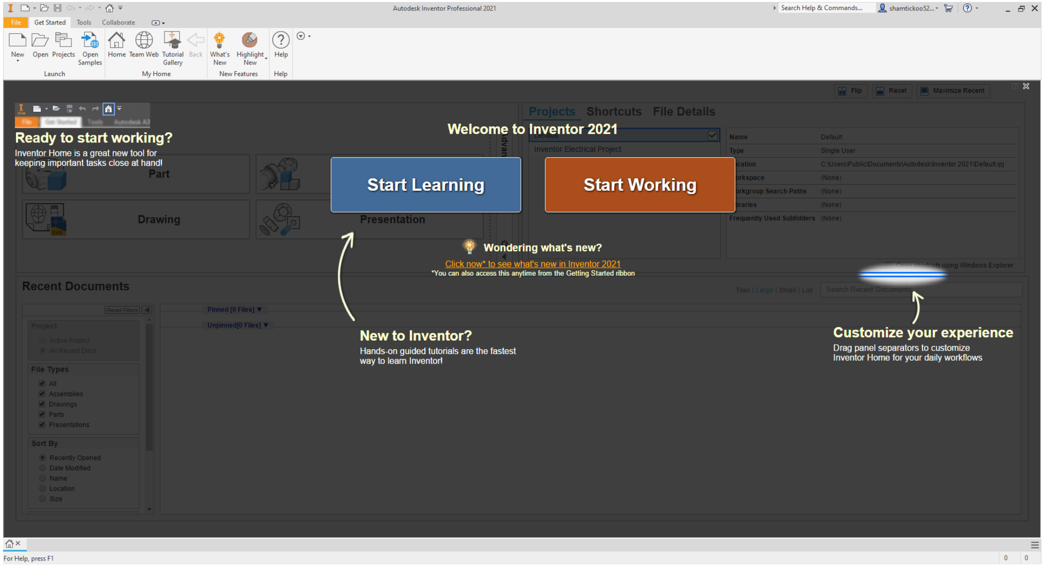

*Figure 1-1 The **Welcome to Inventor 2021** window*

On choosing the **Start Learning** button from this window, the **Tutorials** window will be displayed, as shown in Figure 1-2.

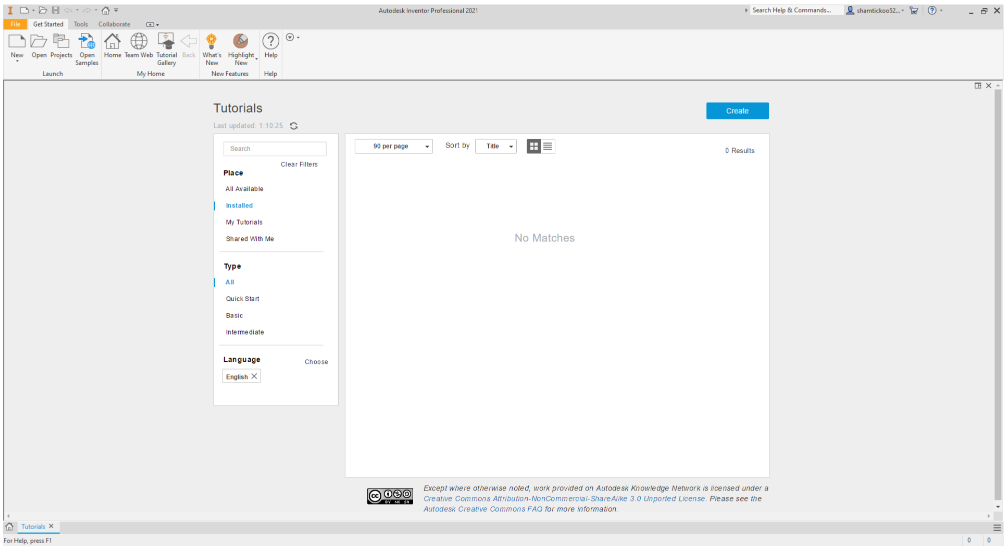

*Figure 1-2 The **Tutorials** window*

In this window, you can see the basic and intermediate level tutorials uploaded by Autodesk. Also, you can create tutorials and upload them in the cloud by choosing the **Create** button on right side of this window. On choosing the **Start Working** button from the **Welcome to Inventor 2021** window, the initial interface of Autodesk Inventor Professional 2021 will be displayed, as shown in Figure 1-3.

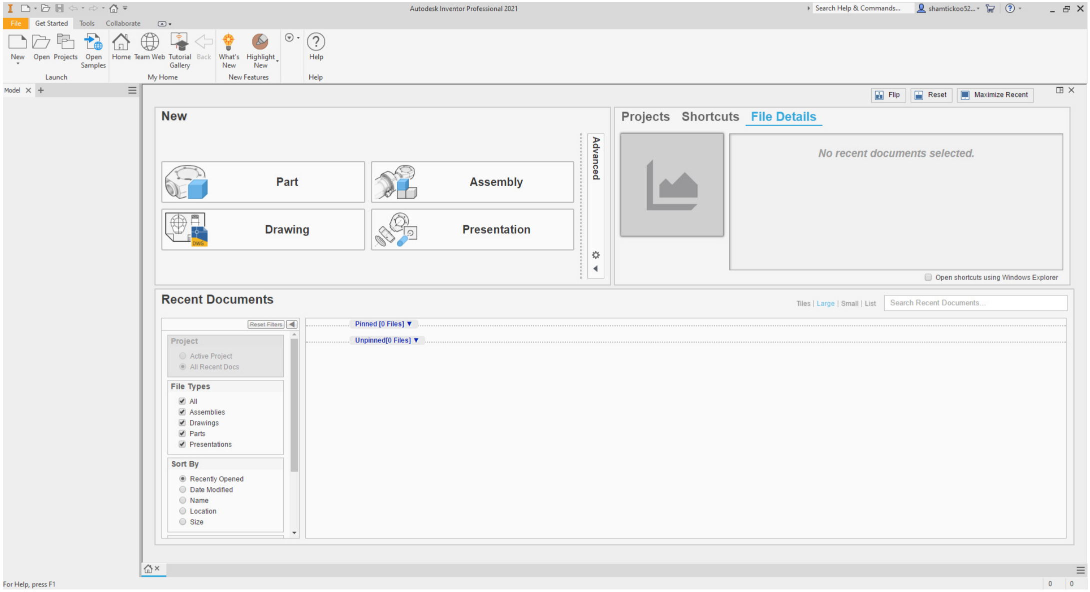

Figure 1-3 Initial interface of Autodesk Inventor Professional 2021

By using the tools available in the initial interface of Autodesk Inventor, you can view the recent enhancements and information related to Autodesk Inventor 2021, start new file, open an existing file, set a project, and so on. To view the enhancements and related information, choose the **What's New** tool available in the **Get Started** tab of the **Ribbon**. You will learn more about the **Ribbon** and respective tabs and tools available in it later in this chapter.

To start a new 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 the 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 interface.

Alternatively, to start a new part file, you can choose the **Part** button from the **New** area in the initial interface of Autodesk Inventor, refer to Figure 1-3. Note that on choosing this button, the part file will be invoked with the default template.

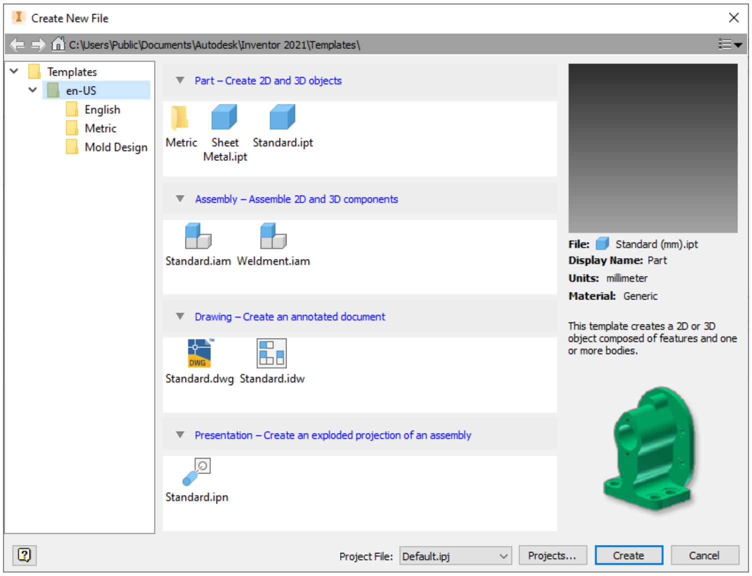

Figure 1-4 *The* ***Create New File*** *dialog box*

It is evident from Figure 1-5 that the interface 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 the 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.

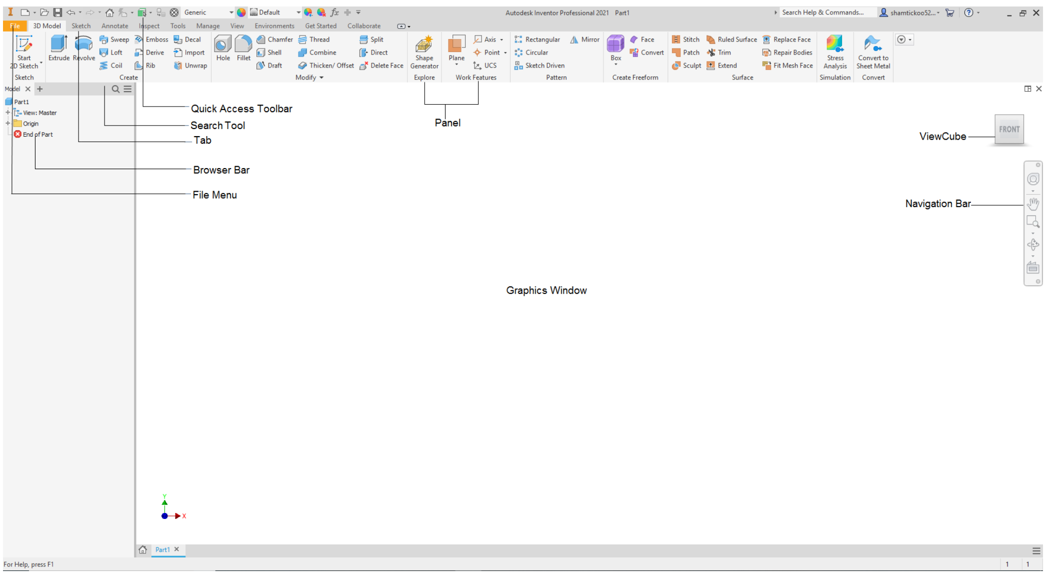

Figure 1-5 *Components of Autodesk Inventor interface*

Quick Access Toolbar

This toolbar is common to all the 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.

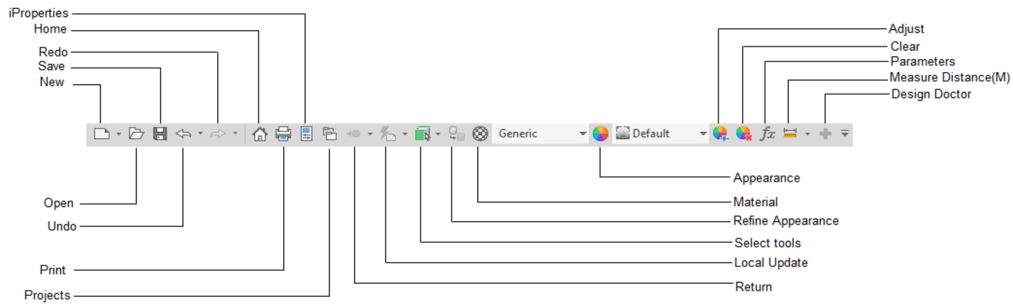

Figure 1-6** The **Quick Access Toolbar

Select

Select tools are used to set the selection priority. If you click on the down arrow on the right of the active select tool, a selection drop-down list will be displayed, refer to Figure 1-7. The **Select Bodies** tool is chosen to set the selection priority for 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 for faces and edges. The **Select Sketch Features** tool is chosen to set the priority for the sketched entities. The **Select Groups** and **Select Wires** tools will be activated in their respective environments when the different groups and wires become available.

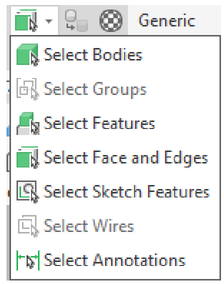

***Figure 1-7** The **Selection** drop-down list*

Return

This tool is activated in the sketching environment and is used to exit from the sketching environment. Once you have finished drawing a sketch, choose this tool to proceed to the **Part** module. In the **Part** module, you can convert the sketch into a feature using the required tools.

Note

*If the **Return** tool is not available in the **Quick Access Toolbar**, you need to add it. To do so, click on the down arrow on the right of the **Quick Access Toolbar**; a flyout is displayed. Next, choose the **Return** option from the flyout.*

Update/Local Update
This tool is chosen to update a design after modifying.

Appearance
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.

Material Drop-down List
You can use the options in this drop-down list to apply different types of materials to the selected features or component.

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.

In addition to the default tools available in a tab, you can also customize the tab by adding more tools. 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. Make sure that the **Ribbon** tab of the dialog box is chosen. Next, select the **All Commands** option from the **Choose commands from** drop-down list, if not selected by default; a list of all the commands/tools will be displayed on the left hand side in the dialog box. Next, select the required tool to be added from the list and then from the **Choose tab to add custom panel to** drop-down list, select the required tab to which the selected tool is to be added. Next, choose the **Add** button which is represented as double arrows and then choose the **Apply** button to add the tool. Similarly you can add multiple tools to the required tab of the Ribbon. Once you are done, close the **OK** button to exit the dialog box.

Tip
*In Autodesk Inventor, 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 the tools for creating the sketches of the parts are available in this tab. Most of the tools of the tab will be available on invoking the sketching environment. The **Sketch** tab is shown in Figure 1-8.

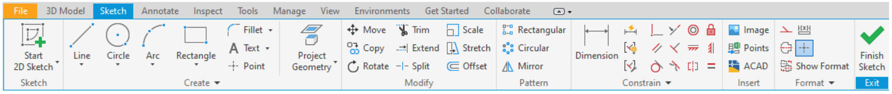

*Figure 1-8 The **Sketch** tab*

Inventor Precise Input Toolbar

Inventor provides you with the **Inventor Precise Input** toolbar to enter precise values for the coordinates of the sketch entities. This toolbar is also available in the **Drawing** and **Assembly** modules. The **Inventor Precise Input** toolbar is shown in Figure 1-9. Note that this toolbar is not available by default. You will learn more about this toolbar in Chapter 2.

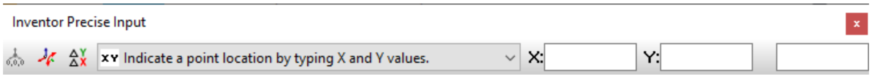

*Figure 1-9 The **Inventor Precise Input** toolbar*

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 the modeling tools that can be used to convert a sketch into a feature. The tools in the **3D Model** tab are shown in Figure 1-10.

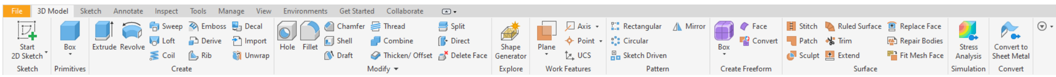

*Figure 1-10 The **3D Model** tab*

The **Start 2D Sketch** button in the **Sketch** panel of the **3D Model** tab is used to invoke the sketching environment to draw 2D sketch. As the first feature in most of the designs is a sketched feature, therefore you first need to create the sketch of the feature to be created. Once you have completed a sketch, you can choose either the **Finish Sketch** button from the **Exit** panel of the **Sketch** tab in the **Ribbon** or the **Return** button from the **Quick Access Toolbar**.

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**. If the **Convert** panel is not available in the **3D Model** tab, you need to customize to add it. You will learn more about customizing later in this book. The tools in the **Sheet Metal** tab are shown in Figure 1-11.

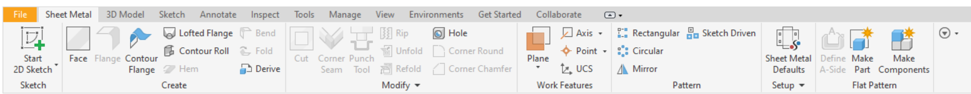

*Figure 1-11 The **Sheet Metal** tab*

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 the tools that are required for assembling components. The tools in the **Assemble** tab are shown in Figure 1-12.

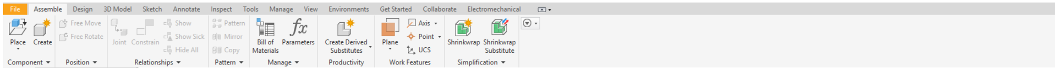

*Figure 1-12 The **Assemble** tab*

Place Views Tab

This tab provides the tools that are used to create different drawing 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 the environments. The tools in the **Tools** tab are shown in Figure 1-15.

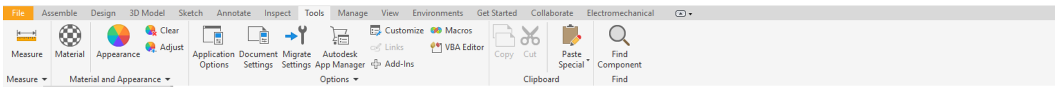

*Figure 1-15 The **Tools** tab*

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 the environments. The tools in the **View** tab are shown in Figure 1-16.

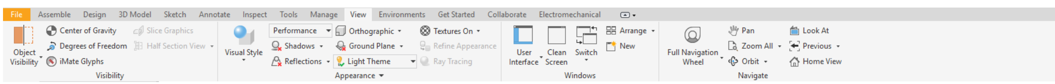

*Figure 1-16 The **View** tab*

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 1-17. 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.

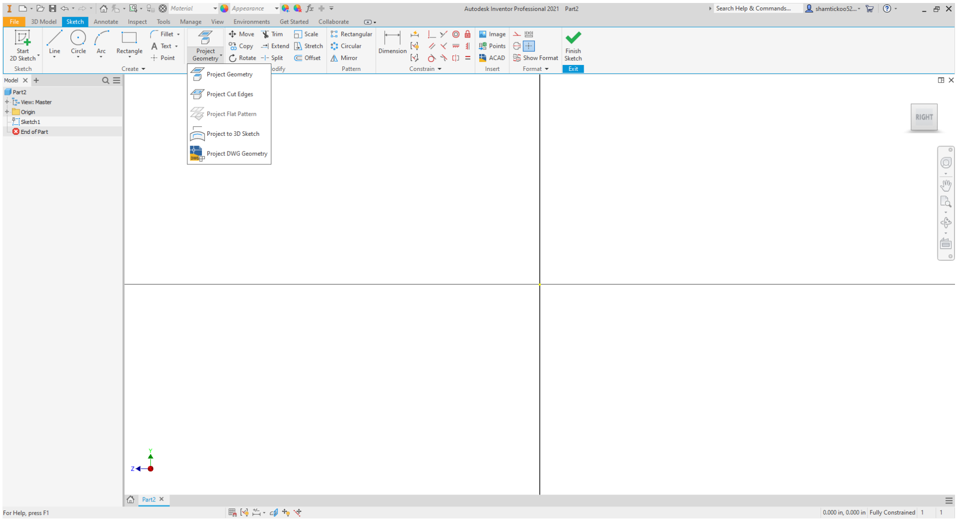

Figure 1-17 *More tools displayed on choosing the down arrow on the right of a tool in the* ***Ribbon***

Navigation Bar

The Navigation Bar is located on the right of the graphics window and contains tools that are used to navigate the model in order 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.

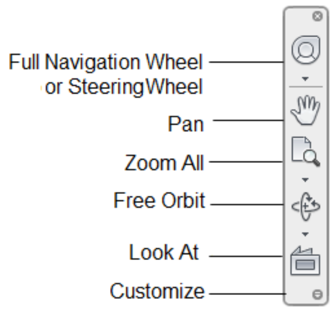

Figure 1-18 *The* ***Navigation Bar***

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. 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.

Search Tool

The **Search** tool is available at the top of the **Browser Bar**, refer to Figure 1-19. This tool is used to search fields such as Features, File nodes (both collapsed and expanded), Parts, Constraints, and so on.

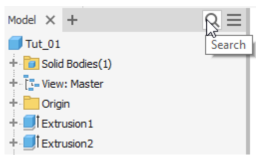

Figure 1-19 *The* ***Search*** *tool*

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 the 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 with in Autodesk Inventor, it is very important for you to understand the following terms 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 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 a new size or shape without considering its original size or shape. For example, a line of 20 mm that was initially drawn at an angle of 45 degrees can be derived to a line of 50 mm and its orientation can be changed to 90°. This property makes the designing process very easy as now you can draw a sketch with some relative dimensions and then can use this solid modeling tool to drive to the required actual values.

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 of the environments, it is automatically reflected in the other environments as well.

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.

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 to define its position with respect to some 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 sketch entities to relate them to the standard properties like collinearity, concentricity, perpendicularity, and so on. Autodesk Inventor automatically applies these geometric constraints to the sketch entities at the time of their creation. You do not have to use an extra command to apply these constraints on to the sketch entities. However, you can also manually apply these geometric constraints on to the sketch 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 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
This assembly constraint is used to make 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
This assembly constraint is used to place the selected faces of different components at some angle with respect to each other.

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

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

Symmetry
This assembly 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 automatically apply best suitable type of joints 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 from 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 with respect 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.

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 Professional, 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 goes beyond a permissible limit. Figure 1-20 shows the Displacement plot of leaf spring designed in Autodesk Inventor and analyzed using the analysis tools.

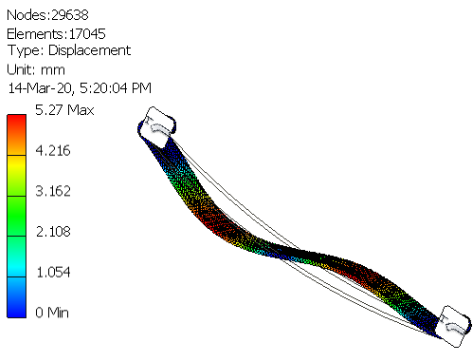

Figure 1-20 The resultant model with displacement

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 automatically displayed 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 1-21 shows the **Select Other** flyout displayed in the modelling environment. You can use this tool in all the modes and environments of Autodesk Inventor.

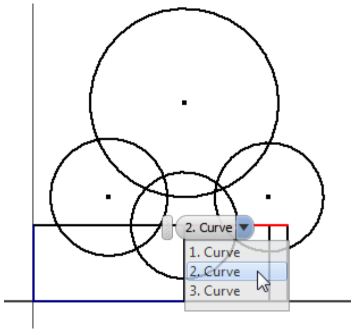

Figure 1-21 *Selecting the entities from the* ***Select Other*** *flyout*

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

The following hotkeys are used in the Sketching environment:

Hotkey	Function
L	Invokes the **Line** 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 **Constrain** tool
V	Invokes the **Free Move** tool
G	Invokes the **Free Rotate** 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	Displays the previous view

SHIFT+F5	Displays the 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; a list of all the available commands will be displayed, as shown in Figure 1-22. The options corresponding to the **Keyboard** tab are discussed next.

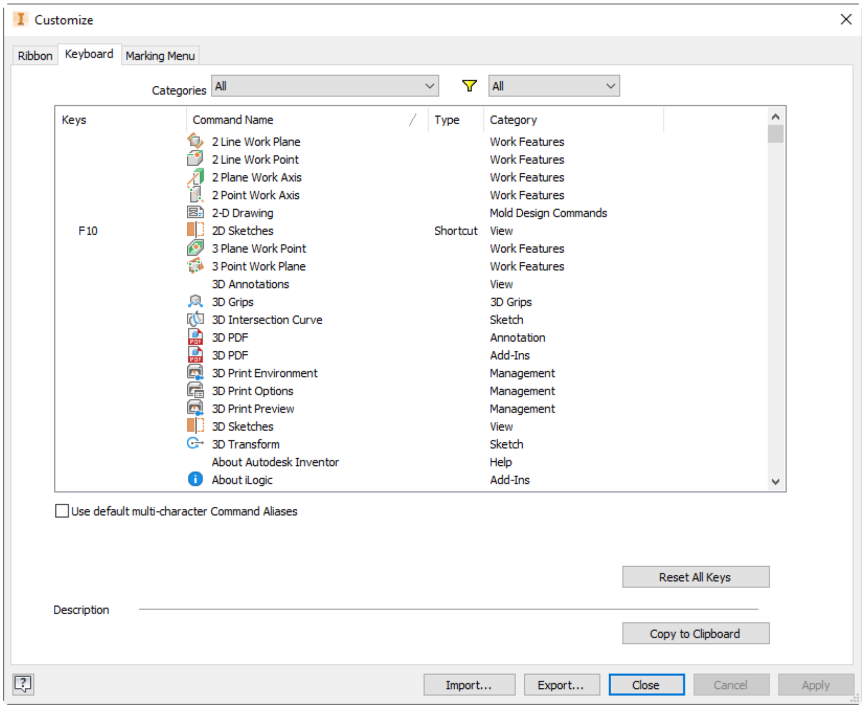

*Figure 1-22 The **Customize** dialog box displaying various commands in the **Keyboard** tab*

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 the 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, press the Enter key. 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 the customized hotkeys and restore the default hotkeys.

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

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

Export
Choose this button to save the customized settings in the .xml format. Make sure that all the 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 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 **Start 2D Sketch** tool from the **Sketch** drop-down in the **Sketch** panel of the **3D Model** tab, see Figure 1-23. On choosing this tool, the Sketching environment is invoked and you can create 2D sketches. If you choose the **Start 3D sketch** tool from the **Sketch** panel, you can create 3D sketches.

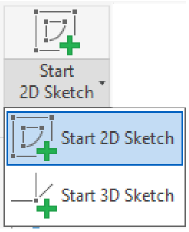

***Figure 1-23** Tools in the **Sketch** drop-down*

MARKING MENU
Marking menu is a type of menu that consists of 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 the 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 1-24 shows a Marking menu invoked in the Sketching environment and Figure 1-25 shows a Marking menu which is invoked in the Modeling environment.

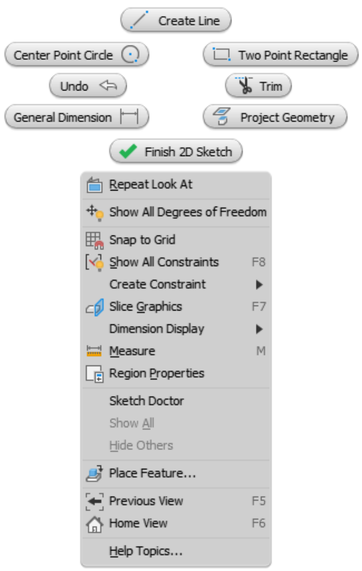

Figure 1-24 *Marking menu available in the Sketching environment*

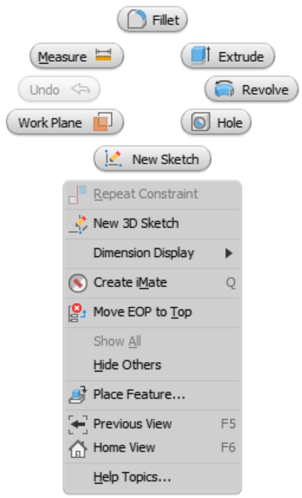

Figure 1-25 *Marking menu available in the Modeling environment*

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

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 **In-canvas Color Scheme** list box in the **Colors** tab. Select **1**

Color from the drop-down list in the **Background** area, refer to Figure 1-26. 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.

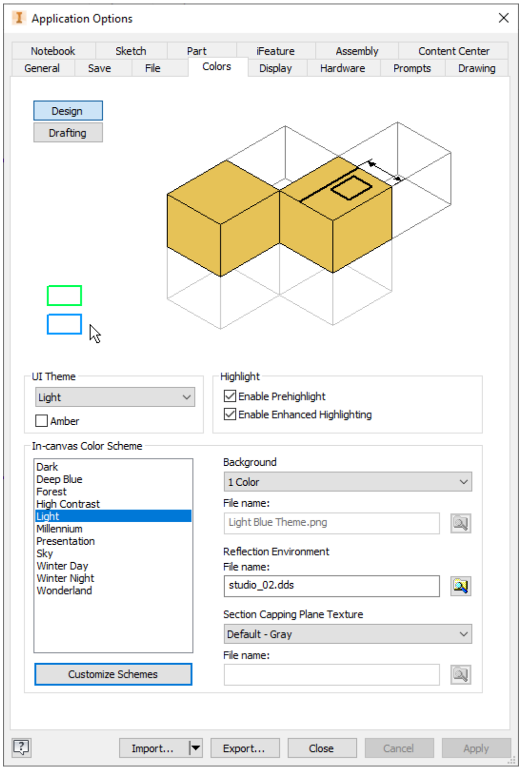

***Figure 1-26** The **Application Options** dialog box with the required options set in the **Colors** tab*

Self-Evaluation Test

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

1. You can invoke the **Line** tool by using the __________ hotkey.

2. Press __________ to invoke the recently used tool.

3. Choose the __________ button from the **Customize** dialog box to restore the customized settings in the .xml format.

4. When you start a new session of Autodesk Inventor Professional 2021, only the **Start a new file** button will be available in the **Quick Launch** area of the **Open** dialog box. (T/F)

5. The **Inventor Precise Input** toolbar is used to specify the precise values for the coordinates of the sketch entities. (T/F)

6. The tools in the **3D Model** tab enable you to control the view, orientation, appearance, and visibility of objects and view windows. (T/F)

Review Questions

Answer the following questions:

1. You can use the __________ drop-down list to apply different types of colors or styles to the selected feature or component to improve its appearance.

2. You can invoke the **Analyze Interference** tool from the **Assembly** module by pressing the __________ key.

3. You change the color scheme by choosing the __________tool from the **Options** panel of the **Tools** tab in the **Ribbon.**

4. There are twelve types of geometric constraints in Autodesk Inventor. (T/F)

5. Design Doctor works in five steps. (T/F)

6. You can invoke the **Trim** tool by pressing the X key. (T/F)

Answers to Self-Evaluation Test

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