Chapter 1

Introduction to AutoCAD

Learning Objectives

After completing this chapter, you will be able to:

- *Start AutoCAD*
- *Understand the usage of the components of the initial AutoCAD screen*
- *Invoke AutoCAD commands from the keyboard, menus, toolbars, shortcut menus, Tool Palettes, and Ribbon*
- *Work with File Tabs*
- *Understand the usage of components of dialog boxes in AutoCAD*
- *Start a new drawing using the New tool, File Tabs, and the Startup dialog box*
- *Save the work using various file-saving commands*
- *Close a drawing*
- *Open an existing drawing*
- *Quit AutoCAD*
- *Understand various options of AutoCAD help*
- *Understand the concept of Design Feed*

Key Terms

- *Initial Setup*
- *AutoCAD Screen Components*
- *Ribbon*
- *Application Menu*
- *Tool Palettes*
- *Menu Bar*
- *Toolbar*
- *New*
- *Save*
- *Save As*
- *Close*
- *STARTUP*
- *Open*
- *Partial open*
- *Drawing Recovery Manager*
- *Workspaces*
- *Help*
- *Autodesk Exchange Apps*
- *Autodesk 360*
- *Autodesk Cloud*
- *Design Feed*
- *File Tabs*

AutoCAD SCREEN COMPONENTS

When you install AutoCAD 2017, the AutoCAD 2017 - English shortcut icon is created on the desktop. You can start AutoCAD by double-clicking on this icon. The initial AutoCAD screen comprises of drawing area, command window, menu bar, several toolbars, Model and Layout tabs, and Status Bar, as shown in Figure 1-1. A title bar that has an AutoCAD symbol and the current drawing name is displayed on top of the screen.

Start Tab

In AutoCAD, the **Start** tab is displayed in the AutoCAD environment when you close all the drawing templates or when there are no drawings open. The **Start** tab contains two sliding frames, **CREATE** and **LEARN**, see Figure 1-2. These frames are discussed next.

CREATE

When you click on **CREATE**, the **CREATE** page is displayed. In the **CREATE** page, you can access sample file, recent files, templates, product updates as well as connect with the online community. The **CREATE** page is divided into three columns: **Get Started**, **Recent Documents**, and **Connect**.

LEARN

When you click on **LEARN**, the **LEARN** page is displayed. The **LEARN** page provides tools to help you learn AutoCAD. It is divided into three columns: **What's New**, **Getting Started Videos**, and **Learning Tips/Online Resources**.

Drawing Area

The drawing area covers the major portion of the screen. In this area, you can draw the objects and use the commands. To draw the objects, you need to define the coordinate points, which can be selected by using your pointing device. The position of the pointing device is represented on the screen by the cursor. The window also has the standard Windows buttons such as close, minimize, scroll bar, and so on, on the top right corner. These buttons have the same functions as in any other standard window.

Command Window

The command window at the bottom of the drawing area has the Command prompt where you can enter the commands. It also displays the subsequent prompt sequences and the messages. You can change the size of the window by placing the cursor on the top edge (double line bar known as the grab bar) and then dragging it. This way you can increase its size to see all the previous commands you have used. You can also press the F2 key to display **AutoCAD Text Window**, which displays the previous commands and prompts.

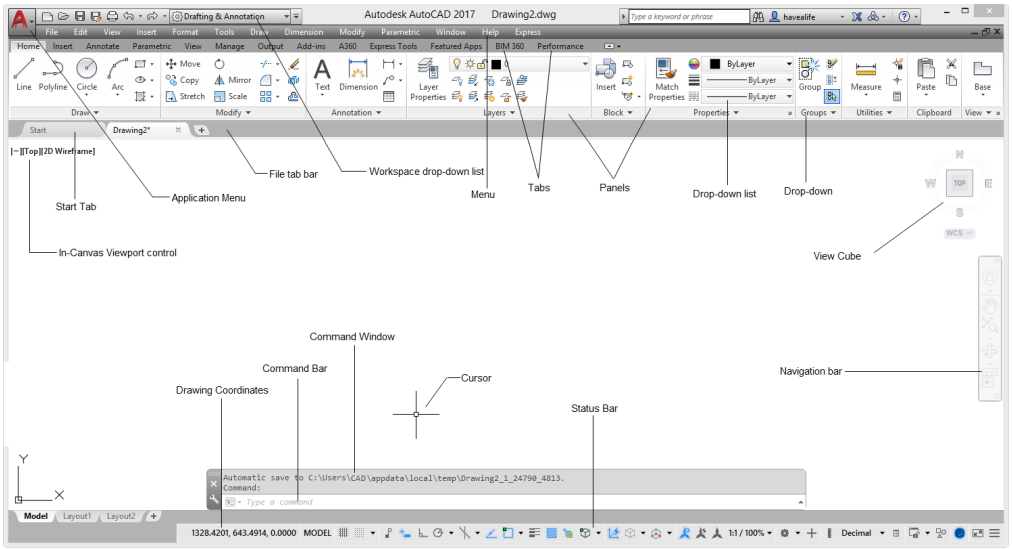

Figure 1-1 *AutoCAD screen components in AutoCAD* ***Drafting & Annotation*** *workspace*

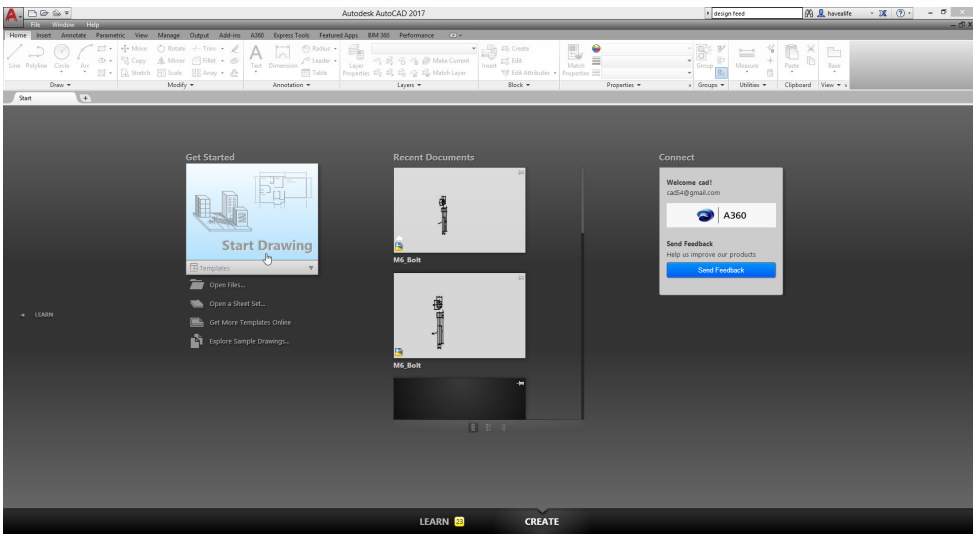

Figure 1-2 *The* ***Start*** *tab in AutoCAD 2017*

Tip

You can hide all toolbars displayed on the screen by pressing the CTRL+0 keys or by choosing ***View > Clean Screen*** *from the menu bar. To turn on the display of the toolbars again, press the CTRL+0 keys. You can also choose the* ***Clean Screen*** *button in the Status Bar to hide all toolbars.*

Auto Correct the Command Name

In AutoCAD, if you type a wrong command name at the Command prompt, a suggestion list with most relevant commands will be displayed, refer to Figure 1-3. You can invoke the desired command by selecting the required option from this list.

Auto Complete the Command Name

When you start typing a command name at the Command prompt, the complete name of the command will be displayed automatically. Also, a list of corresponding commands will be displayed, as shown in Figure 1-4. The commands that have not been used for a long time will be grouped in folders at the bottom of the list.

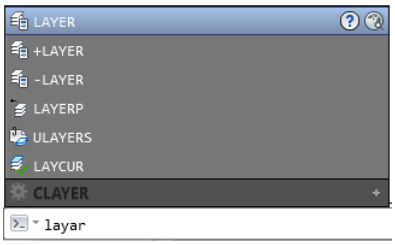

Figure 1-3 *Suggestion list with relevant commands*

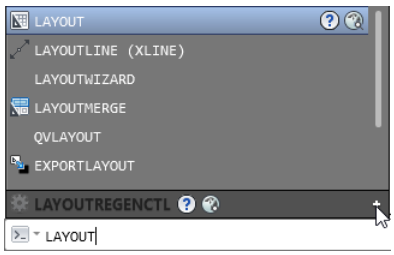

Figure 1-4 *Command bar displaying complete command name automatically*

Internet Search

You can get more information about a command by using the **Search in Help** and **Search on Internet** buttons available adjacent to the command name in the Command line, refer to Figure 1-5. If you choose the **Search in Help** button, the **Autodesk AutoCAD 2017-Help** dialog box will be displayed. In this dialog box, you can find information about the command. By using the **Search on Internet** button, you can find information about the command on internet. Note that these buttons will be available adjacent to the selected command name in the suggestion list.

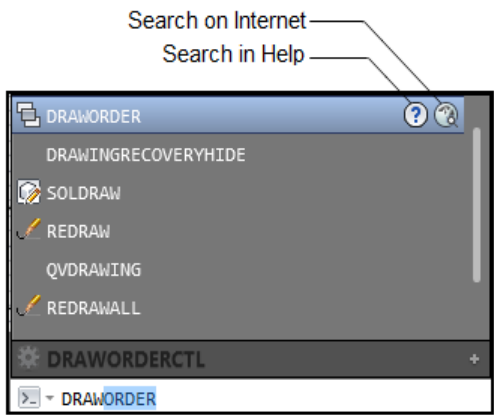

Figure 1-5 *The **Search in Help** and **Search on Internet** buttons displayed in the suggestion list*

Synonym Suggestions

In AutoCAD, you can invoke a command by entering synonyms of the command name. By entering the synonym at the Command prompt, the command related to the synonym entered will be displayed. For example, if you enter **ROUND** at the Command prompt, then the **FILLET** command will be displayed, as shown in Figure 1-6.

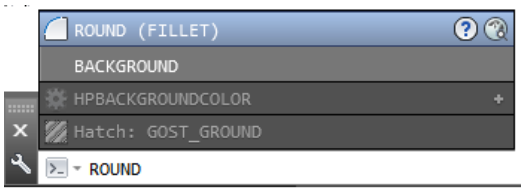

Figure 1-6 *Suggestion list displayed after entering a synonym of command name at the Command prompt*

Input Search Options

In AutoCAD, you can enable or disable the functions such as Auto Complete and Auto Correct by using the options available in the **Input Search Options** dialog box. To invoke this dialog box, right-click at the Command prompt; a shortcut menu will be displayed. Next, choose **Input Search Options** from the shortcut menu; the **Input Search Options** dialog box will be displayed, refer to Figure 1-7. You can now enable or disable the required functions by using this dialog box.

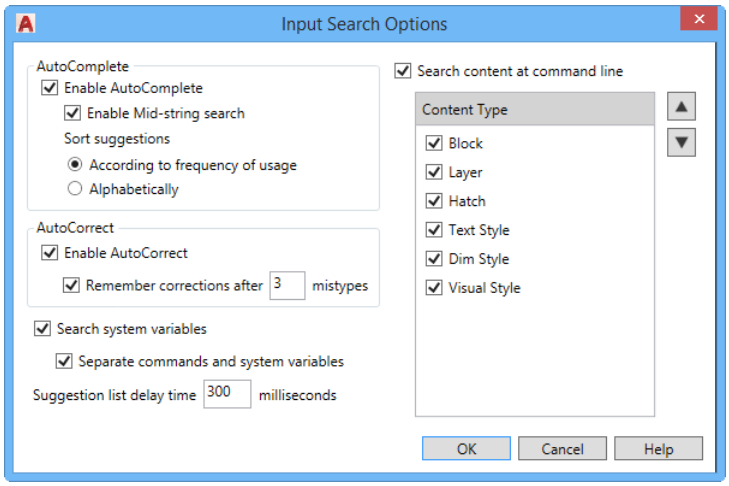

Figure 1-7 *The* ***Input Search Options*** *dialog box*

Navigation Bar

The **Navigation Bar** is displayed in the drawing area and contains navigation tools. These tools are grouped together, refer to Figure 1-8, and are discussed next.

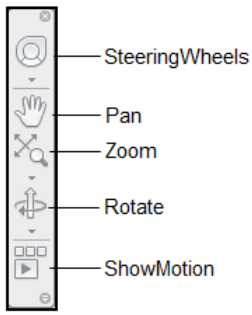

Figure 1-8 *Tools in the* ***Navigation Bar***

SteeringWheels

The **SteeringWheels** has a set of navigation tools such as pan, zoom, and so on. You will learn more about the **SteeringWheels** in the later chapters.

Pan

This tool allows you to view the portion of the drawing that is outside the current display area. To do so, choose this tool, press and hold the left mouse button, and then drag the drawing area. Press ESC to exit this command.

Zoom Tools

Zoom tools are used to enlarge the view of the drawing on the screen without affecting the actual size of the objects. These tools are grouped together in the **Zoom** drop-down. You will learn more about zoom tools in later chapters.

Rotate Tools

Rotate tools are used to rotate the view in 3D space. These tools are grouped together in the **Rotate** drop-down.

ShowMotion

Choose this button to capture different views in a sequence and animate them when required.

ViewCube

ViewCube is available on the top right corner of the drawing area and is used to switch between the standard and isometric views or roll the current view. The ViewCube and its options are discussed in later chapters.

In-canvas Viewport Controls

In-canvas Viewport Controls is available on the top left corner of the drawing screen. It enables you to change the view, visual style as well as the viewport.

Status Bar

The Status Bar is displayed at the bottom of the screen and is called Application Status Bar. It contains some useful information and buttons (see Figure 1-9) that make it easy to change the status of some AutoCAD functions. You can toggle between the on and off states of most of these functions by choosing them.

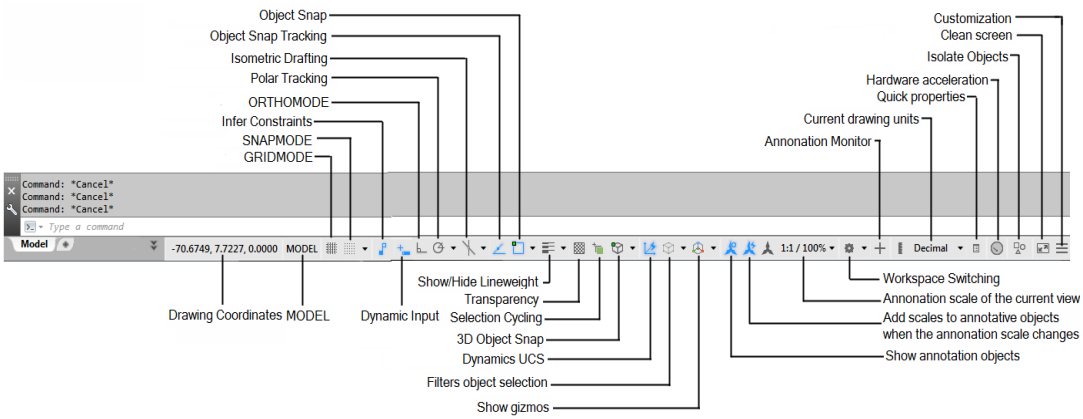

Figure 1-9 *The Status Bar displayed in the* ***Drafting & Annotation*** *workspace*

Drawing Coordinates

The information about the coordinates is displayed on the left corner of the Status Bar near the **Model** and **Layout** tabs. You can choose the coordinate button to toggle between the on and off states. The **COORDS** system variable controls the type of display of the coordinates. If the value of the **COORDS** variable is set to 0, the coordinate display is static, that is, the coordinate values displayed in the Status Bar change only when you specify a point. If the value of the **COORDS** variable is set to 1 or 2, the coordinate display is dynamic. When the variable is set to 1, AutoCAD constantly displays the absolute coordinates of the graphics cursor with respect to the UCS origin. The polar coordinates (length<angle) are displayed if you are in an AutoCAD command and the **COORDS** variable is set to 2. By default, the **COORDS** variable is set to 1. You can also click on the **Drawing Coordinates** area to change the coordinate status from on to off and vice-versa.

MODEL

The **MODEL** button is chosen by default because you are working in the model space to create drawings. You will learn more about the model space in later chapters.

GRIDMODE

In AutoCAD, the grid lines are used as reference lines to draw objects. If the **GRIDMODE** button is chosen, the Display drawing grid will turn on and the grid lines are displayed on the screen. The F7 function key can be used to turn the grid display on or off.

SNAPMODE

If the **SNAPMODE** button is chosen, the snap mode will turn on. So, you can move the cursor in fixed increments. The F9 key acts as a toggle key to turn the snap on or off.

Infer Constraints

If this button is chosen then some of the geometric constraints will be automatically applied to sketch when it is drawn. You can use CTRL+SHIFT+I as a shortcut key to toggle this button.

Dynamic Input

The **Dynamic Input** button is used to turn the **Dynamic Input** on or off. Turning it on facilitates the heads-up design approach because all commands, prompts, and dimensional inputs will now be displayed in the drawing area and you do not need to look at the Command prompt all the time. This saves the design time and also increases the efficiency of the user. If the **Dynamic Input** mode is turned on, you will be allowed to enter the commands through the **Pointer Input** boxes, and the numerical values through the **Dimensional Input** boxes. You will also be allowed to select the command options through the **Dynamic Prompt** options in the graphics window. To turn the **Dynamic Input** on or off, use the F12 key.

ORTHOMODE

If the **ORTHOMODE** button is chosen, you can draw lines at right angles only. You can use the F8 function key to turn ortho on or off.

Polar Tracking

If you turn the polar tracking on, the movement of the cursor is restricted along a path based on the angle set as the polar angle. Choose the **Polar Tracking** button to turn the polar tracking on. You can also use the F10 function key to turn on this option. Note that turning the polar tracking on, automatically turns off the ortho mode.

Isometric Drafting

In AutoCAD, you can activate the required working plane. To activate the required working plane, choose the **Isometric Drafting** button from the Status Bar. On choosing this button, a flyout is displayed with the **isoplane Left**, **isoplane Top,** and **isoplane Right** options. You can choose the required option from this flyout to activate the respective work plane.

Object Snap Tracking

When you choose this button, the inferencing lines will be displayed. Inferencing lines are dashed lines that are displayed automatically when you select a sketching tool and track a particular keypoint on the screen. Choosing this button turns the object snap tracking on or off. You can also use the F11 function key to turn the object snap tracking on or off.

Object Snap

When the **Object Snap** button is chosen, you can use the running object snaps to snap on to

a point. You can also use the F3 function key to turn the object snap on or off. The status of **OSNAP** (off or on) does not prevent you from using the immediate mode object snaps.

Show/Hide Lineweight

Choosing this button in the Status Bar allows you to turn on or off the display of lineweights in the drawing. If this button is not chosen, the display of lineweight will be turned off.

Transparency

This button is available in the Status Bar and is chosen to turn on or off the transparency set for a drawing. You can set the transparency in the **Properties** panel or in the layer in which the sketch is drawn.

Selection Cycling

When this button is chosen, you can cycle through the objects to be selected, if they are overlapping or close to other entities. On selecting an entity when this button is chosen, the **Selection** list box with a list of the entities that can be selected will be displayed. You can use CTRL+W as a shortcut key to toggle this button.

3D Object Snap

When this button is chosen, you can snap the key point on a solid or a surface. You can also use the F4 function key to turn on or off the 3D object snap.

Dynamics UCS

Choosing this button allows or disallows the use of dynamic UCS. Allowing the dynamic UCS ensures that the XY plane of the UCS got dynamically aligned with the selected face of the model. You can also use the F6 function key to turn the **Dynamics UCS** button on or off.

Filters object selection

In AutoCAD, you can filter objects as per your requirement by using the **Filters object selection** button. If you want to select only Vertex, Edge, Face, Solid History, or Drawing View Components of a 3D object, then you can choose the required option from the flyout which is invoked by clicking on the small arrow at right side of the **Filters object selection** button. You can also select multiple objects through selection window.

The **Drawing View Component** option is used to select the components of an assembly or parts in a multi body. Using this option, you can select components either individually, or through window selection. You can also turn off the filters by choosing the **Filter object selection** button again.

Show gizmos

You can move, rotate and scale a 3D object by choosing the **Show gizmos** button from the Status Bar. When you click on the small arrow available at the right side of the Show gizmos button, a flyout will be displayed with the **Move Gizmo**, **Rotate Gizmo**, and **Scale Gizmo** options.

Show annotation objects

This button is used to control the visibility of the annotative objects that do not support the current annotation scale in the drawing area.

Add scales to annotative objects when the annotative scale changes

This button, if chosen, automatically adds all the annotation scales that are set current to all the annotative objects present in the drawing.

Annotation scale of the current view

The annotation scale controls the size and display of the annotative objects in the model space. The **Annotation Scale** button has a flyout that displays all the annotation scales available for the current drawing.

Workspace Switching

In AutoCAD, you can switch between different environments or workspaces by using the **Workspace Switching** button that is available at the right of the Status Bar. On clicking the **Workspace Switching** button, a flyout will be displayed with the list of all available workspaces. You can select the required workspace to invoke. You will learn more about workspaces later in this chapter.

Annotation Monitor

The **Annotation Monitor** button is used to turn the **Annotation Monitor** on or off. If it is turned on, all the non-associative annotations will get highlighted by placing a badge on them, as shown in Figure 1-10.

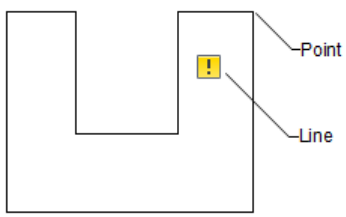

Figure 1-10 *The non-associative annotation*

Current drawing units

The **Current drawing units** button displays and controls the units of drawing. It has a flyout that displays all the unit systems available for drawing.

Quick Properties

If you select a sketched entity when this button is chosen in the Status Bar, the properties of the selected entity will be displayed in a panel. You can use CTRL+SHIFT+P as a shortcut key to toggle this button.

Hardware Acceleration

This button is used to set the performance of the software to an acceptable level.

Isolate Objects

This button is used to hide or isolate objects from the drawing area. On choosing this button, a flyout will be displayed with two options. Choose the **Isolate Objects** option from this flyout and then select the objects to hide or isolate. To end isolation or display a hidden object, click this button again and choose the **End Object Isolation** option.

Clean Screen

The **Clean Screen** button is available at the lower right corner of the screen. This button, when chosen, displays an expanded view of the drawing area by hiding all the toolbars except the command window, Status Bar, and menu bar. The expanded view of the drawing area can also be displayed by choosing **View > Clean Screen** from the menu bar or by using the CTRL+0 keys. Choose the **Clean Screen** button again to restore the previous display state.

Customization

The **Customization** button is available at right corner of the Status Bar. Using this button, you can customize to add or remove tools in the Status Bar.

Plot/Publish Details Report Available

This icon is displayed in the Status Bar when some plotting or a publishing activity is performed in the background. When you click on this icon, the **Plot and Publish Details** dialog box, which provides the details about the plotting and publishing activity, will be displayed. You can copy this report to the clipboard by choosing the **Copy to Clipboard** button from the dialog box.

Manage Xrefs

The **Manage Xrefs** icon is displayed in the Status Bar whenever an external reference drawing is attached to the selected drawing. This icon displays a message and an alert whenever the Xreffed drawing needs to be reloaded. To find detailed information regarding the status of each Xref in the drawing and the relation between the various Xrefs, click on the **Manage Xrefs** icon; the **External References Palette** will be displayed. The Xrefs are discussed in detail in Chapter 18, Understanding External References.

INVOKING TOOLS IN AutoCAD

On starting AutoCAD, when you are in the drawing area, you need to invoke AutoCAD tools to perform an operation. For example, to draw a line, first you need to invoke the **Line** tool and then define the start point and the endpoint of the line. Similarly, if you want to erase objects, you must invoke the **Erase** tool and then select the objects for erasing. In AutoCAD, you can invoke the commands using the Keyboard, Ribbon, Application Menu, Tool Palettes, Menu bar, Shortcut menu, and Toolbar.

Keyboard

You can invoke any AutoCAD command from the keyboard by typing the command name and then pressing the ENTER key. As you type the first letter of command, AutoCAD displays all available commands starting with the letter typed. If the **Dynamic Input** is on and the cursor is in the drawing area, by default, the command will be entered through the **Pointer Input** box. The **Pointer Input** box is a small box displayed on the right of the cursor, as shown in Figure 1-11. However, if the cursor is currently placed on any toolbar or menu bar, or if the **Dynamic Input** is turned off, the command will be entered through the Command prompt. Before you enter a command, the Command prompt is displayed as the last line in the command window area. If it is not displayed, you must cancel the existing command by pressing the ESC (Escape) key.

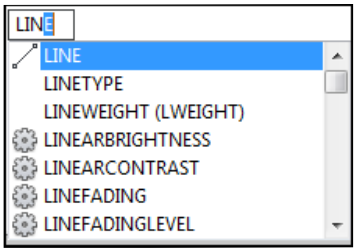

Figure 1-11 *The* ***Pointer Input*** *box displayed when the* ***Dynamic Input*** *is on*

The following example shows how to invoke the **LINE** command using the keyboard:

Command: **LINE** or **L** Enter (L is command alias)

Ribbon

In AutoCAD, you can also invoke a tool from the **Ribbon**. The tools for creating, modifying, and annotating the 2D and 3D designs are available in the panels instead of being spread out in the entire drawing area in different toolbars and menus, see Figure 1-12.

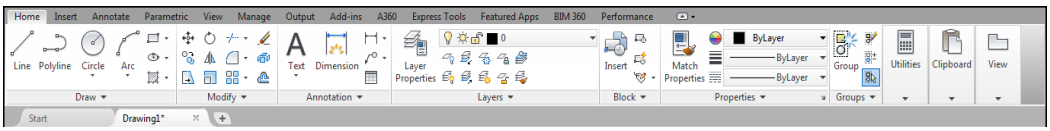

Figure 1-12 *The* ***Ribbon*** *for the* ***Drafting & Annotation*** *workspace*

When you start the AutoCAD session for the first time, by default the **Ribbon** is displayed horizontally below the Quick Access Toolbar. The **Ribbon** consists of various tabs. The tabs have different panels, which in turn, have tools arranged in rows. Some of the tools have small black down arrow. This indicates that the tools having similar functions are grouped together. To choose a tool, click on the down arrow; a drop-down will be displayed. Choose the required tool from the drop-down displayed. Note that if you choose a tool from the drop-down, the corresponding command will be invoked and the tool that you have chosen will be displayed in the panel. For example, to draw a circle using the **2-Point** option, click on the down arrow next to the **Center, Radius** tool in the **Draw** panel of the **Home** tab; a flyout will be displayed. Choose the **2-Point** tool from the flyout and then draw the circle. You will notice that the **2-Point** tool is displayed in place of the **Center, Radius** tool.

Choose the down arrow to expand the panel. You will notice that a push pin is available at the left end of the panel. Click on the push pin to keep the panel in the expanded state. Also, some of the panels have an inclined arrow at the lower-right corner. When you left click on an inclined arrow, a dialog box is displayed. You can define the setting of the corresponding panel in the dialog box.

You can reorder the panels in the tab. To do so, press and hold the left mouse button on the panel to be moved and drag it to the required position. To undock the **Ribbon**, right-click on the blank space in the **Ribbon** and choose the **Undock** option. You can move, resize, anchor, and auto-hide the **Ribbon** using the shortcut menu that will be displayed when you right-click on the heading strip. To anchor the floating **Ribbon** to the left or right of the drawing area in

the vertical position, right-click on the heading strip of the floating **Ribbon**; a shortcut menu is displayed. Choose the corresponding option from this shortcut menu. The **Auto-hide** option will hide the **Ribbon** into the heading strip and will display it only when you move the cursor over this strip.

You can customize the display of tabs and panels in the **Ribbon**. To customize the **Ribbon**, right-click on any one of the tools in it; a shortcut menu will be displayed. On moving the cursor over one of the options, a flyout will be displayed with a tick mark before all options and the corresponding tab or panel will be displayed in the **Ribbon**. Select/clear appropriate option to display/hide a particular tab or panel.

Application Menu

The **Application Menu** is available at the top-left of the AutoCAD window. It contains some of the tools that are available in the **Standard** toolbar. Click on the down arrow on the **Application Menu** to display the tools, as shown in Figure 1-13. You can search for tools or commands by using the search field on the top of the **Application Menu**. To search a tool or command, enter its complete or partial name in the search field; the list of related tools and commands will be displayed. If you click on a tool from the list, the corresponding command will get activated.

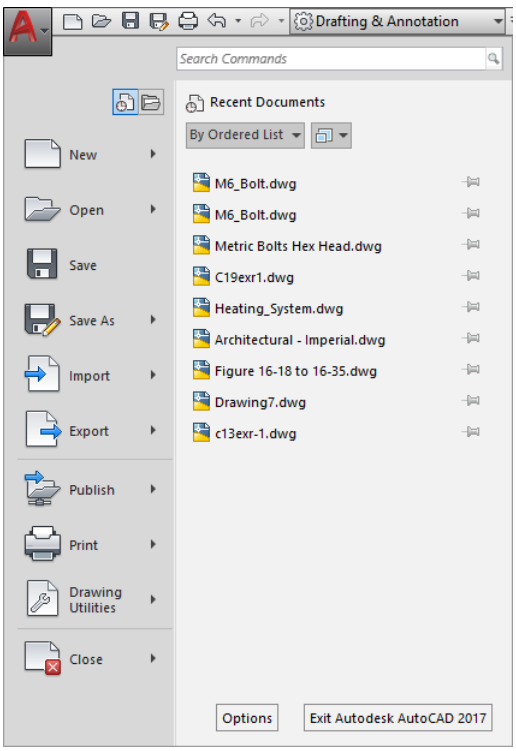

Figure 1-13** The **Application Menu

By default, the **Recent Document** button is chosen in the **Application Menu**. Therefore, the recently opened drawings will be listed. If you have opened multiple drawing files, choose the **Open File** button; the documents that are opened will be listed in the **Application Menu**. To

set the preferences of the file, choose the **Options** button available at the bottom-right of the **Application Menu**. To exit AutoCAD, choose the **Exit Autodesk AutoCAD 2017** button next to the **Options** button.

Tool Palettes

AutoCAD has provided **Tool Palettes** as an easy and convenient way of placing and sharing hatch patterns and blocks in the current drawing. By default, the **Tool Palettes** are not displayed. Choose the **Tool Palettes** button from the **Palettes** panel in the **View** tab or choose the CTRL+3 keys to display the **Tool Palettes** as a window in the drawing area. You can resize the **Tool Palettes** using the resizing cursor that is displayed when you place the cursor on the top or bottom extremity of the **Tool Palettes**. The **Tool Palettes** are discussed in detail in Chapter 12, Hatching Drawings.

Menu Bar

You can also select commands from the menu bar. Menu bar is not displayed by default. To display the menu bar, click on the down arrow in the Quick Access Toolbar; a flyout is displayed. Choose the **Show Menu Bar** option from it; the menu bar will be displayed. As you move the cursor over the menu bar, different tabs are highlighted. You can choose the desired item by left-clicking on it; the corresponding menu is displayed directly under the title. For example, to draw an ellipse using the **Center** option, choose the **Draw** menu and then choose the **Ellipse** option; a cascading menu will be displayed. From the cascading menu, choose the **Center** option. In this text, the command selection sequence will be referenced as choosing **Draw > Ellipse > Center** from the menu bar.

Toolbar

Toolbars are not displayed by default. To display a toolbar, choose **Tools > Toolbars > AutoCAD** from the **Menu Bar**; a list of toolbars will be displayed. Select the required toolbar. Figure 1-14 shows the **Draw** toolbar invoked.

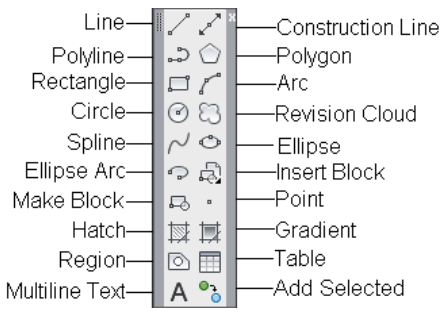

Figure 1-14 *The* ***Draw*** *toolbar*

Shortcut Menu

AutoCAD has provided shortcut menus as an easy and convenient way of invoking the recently used tools. These shortcut menus are context-sensitive, which means that the tools in shortcut menus are dependent on the place/object for which they are displayed. A shortcut menu is invoked by right-clicking and is displayed at the cursor location. You can right-click anywhere in the

drawing area to display the general shortcut menu. It generally contains an option to select the previously invoked tool again, apart from the common tools for Windows, refer to Figure 1-15.

If you right-click in the drawing area while a command is active, a shortcut menu is displayed, containing the options of that particular command. Figure 1-16 shows the shortcut menu when the **Polyline** tool is active.

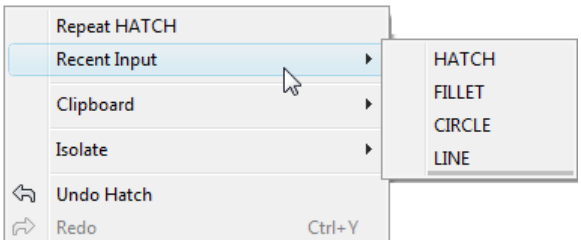

Figure 1-15 *Partial view of the shortcut menu with the recently used tools*

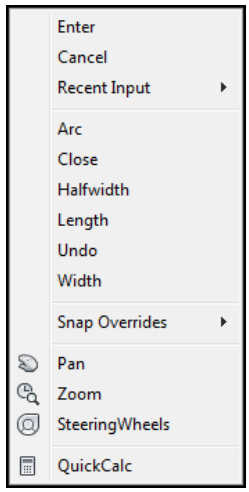

Figure 1-16 *Shortcut menu with the* ***Polyline*** *tool active*

File Tabs

The **File Tabs** button is available in the **Interface** panel of the **View** tab. It is used to toggle the display of the File tab bar which displays all opened files. You can easily switch between multiple opened drawings by clicking on them.

You can also create a new drawing file by clicking on the (**+**) sign available at the end of the File tab bar. When you click on the (**+**) sign, the **New Tab** will be displayed. You can create a new drawing by clicking on the **Start Drawing** icon on the left side of the **New Tab** in **Get Started** area. Figure 1-17 shows the **File Tabs** button chosen in the **Ribbon** and the File tab bar displayed at the bottom of the **Ribbon**.

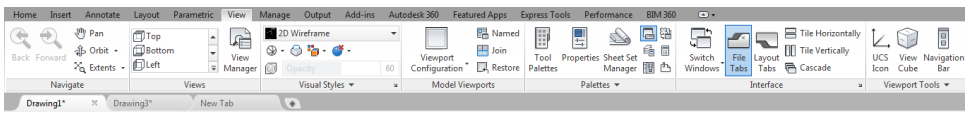

Figure 1-17 *The* ***File Tabs*** *button chosen in the* ***Ribbon*** *and File tab bar displayed at the bottom of the* ***Ribbon***

In the File tab bar, all the added tabs get arranged in the sequence in which the respective drawings are opened or created. You can change the sequence of tabs in the File tab bar by using the left mouse button. To do so, press and hold the left mouse button on any tab and drag it to the desired location, refer to Figure 1-18.

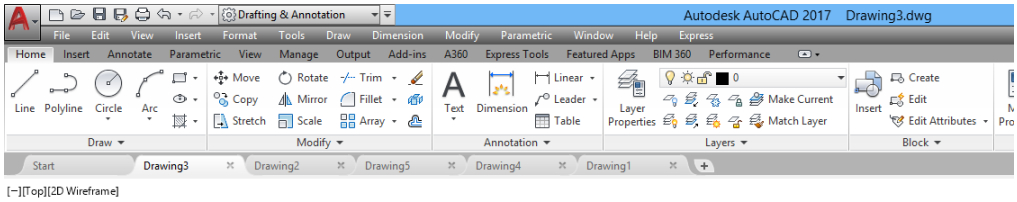

Figure 1-18 *Tab dragged in the File tab bar*

In AutoCAD if a large number of files are opened, some of the files will not be visible in the File tab bar and therefore an overflow symbol will be displayed on its right end, refer to Figure 1-19. To open any tab which is not visible in the File tab bar, click on the overflow symbol; the names of all the tabs will be displayed in a flyout, refer to Figure 1-19. Also, when you move the cursor on a tab name, previews of the Model, Layout1, Layout2, and so on will be displayed, refer to Figure 1-19. You can open the desired environment by clicking on its preview.

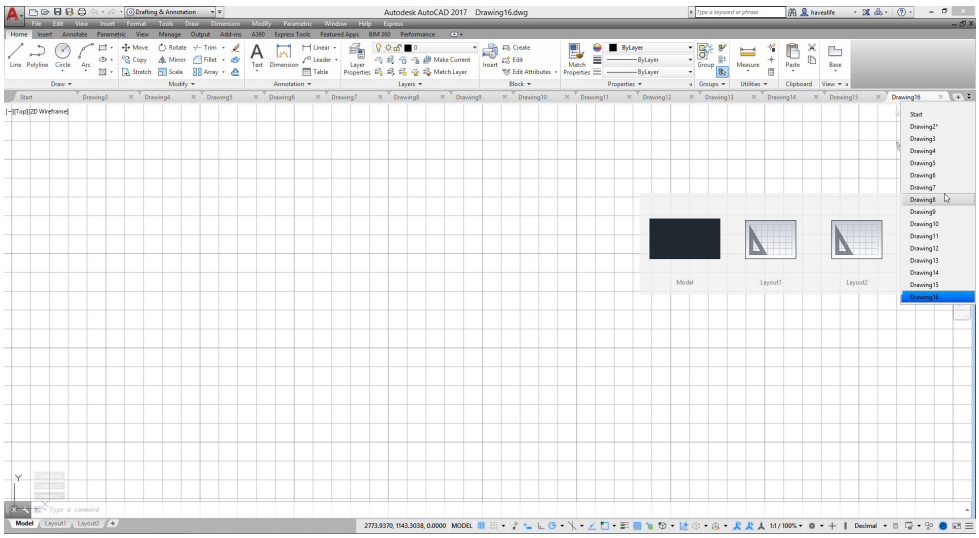

Figure 1-19 *Flyout with file tab names and preview of their respective drawings*

If you move the cursor over a file tab, the preview of the model and layouts will be displayed. When you move the cursor over any preview, the corresponding preview will be displayed in the drawing area, refer to Figure 1-20.

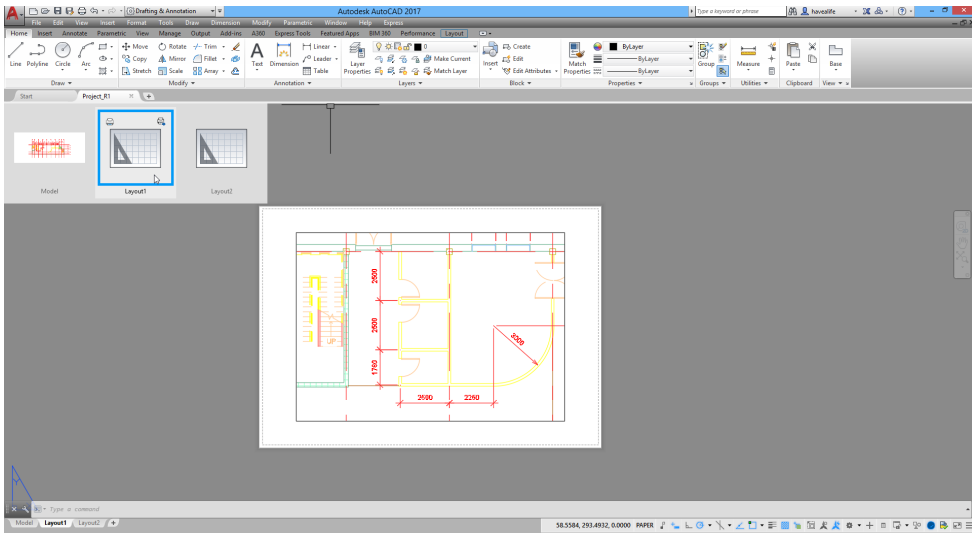

Figure 1-20 *Previews of model and layouts*

There are two buttons available on the top of preview window: **Plot** and **Publish**. By using **Plot**, you can plot the drawing, and by using **Publish**, you can publish the drawing. When you right-click on a file tab, a shortcut menu containing various options such as **New**, **Open**, **Save**, **Save As**, **Close**, and so on will be displayed, refer to Figure 1-21. You can choose the option from the shortcut menu as per your requirement.

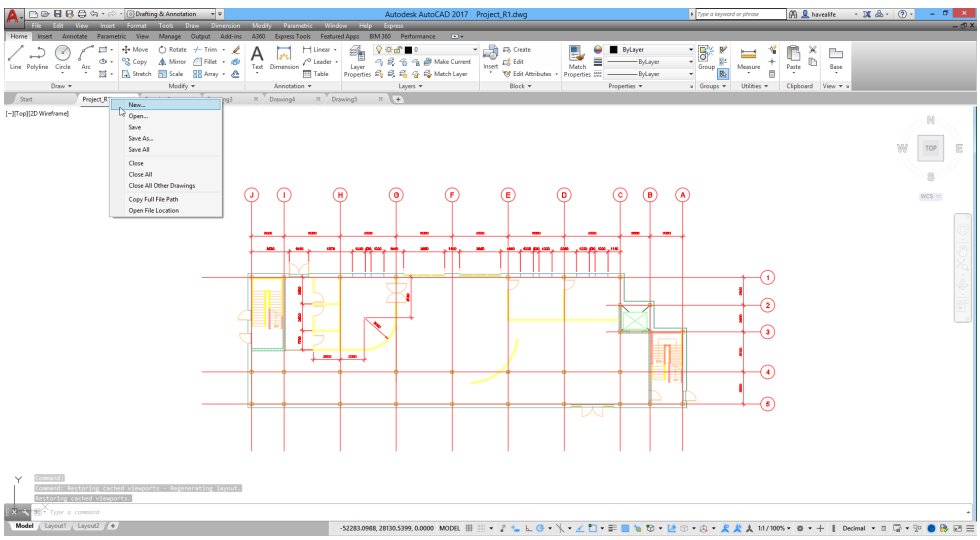

Figure 1-21 *Shortcut menu displayed on right-clicking on the File tab bar*

There are two icons displayed on the file tab: Asterisk icon and Lock icon. The Asterisk icon indicates that the file is modified but not saved. The Lock icon indicates that the file is locked and the changes cannot be saved with the original file name, although you can use the **Save As** tool to create another copy.

To open a drawing as a locked file, first choose the **Open** option from the shortcut menu displayed on right-clicking over the file tab; the **Select File** dialog box will be displayed. Select the desired file and then select the **Open Read-Only** option from the **Open** drop-down list, as shown in Figure 1-22. On doing so, the file will be opened as a locked file in the drawing area. You can also open the file as a locked file by using the **Open** button from the Quick Access Bar.

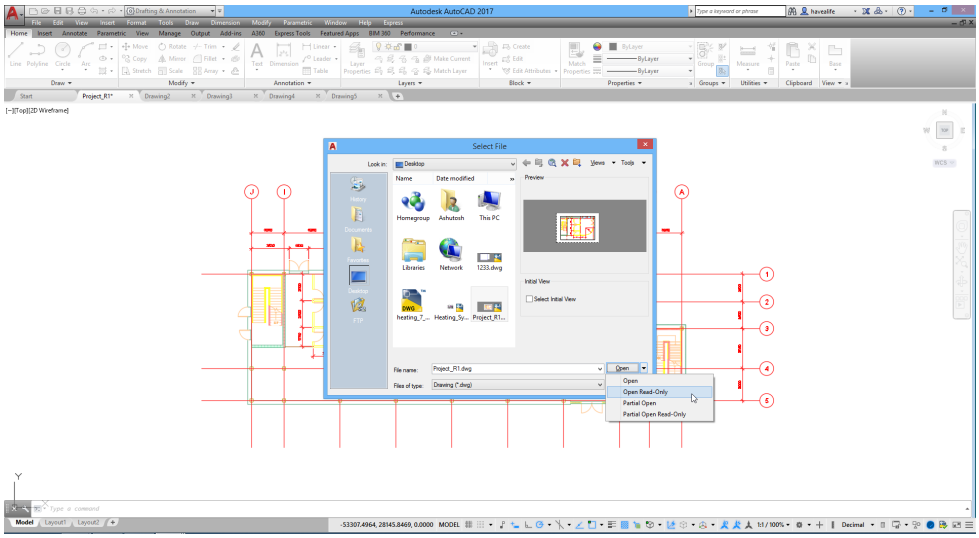

Figure 1-22 *Selecting the* ***Open Read-Only*** *option*

AutoCAD DIALOG BOXES

There are certain commands, which when invoked, display a dialog box. A dialog box is a convenient method of a user interface. When you choose an item in the menu bar with the ellipses [**...**], it displays the dialog box. For example, **Options** in the **Tools** menu displays the **Options** dialog box. A dialog box contains a number of parts like the dialog label, radio buttons, text or edit boxes, check boxes, slider bars, image boxes, and command buttons. These components are also referred to as tiles. Some of the components of a dialog box are shown in Figure 1-23.

You can select the desired tile of the dialog box by using the pointing device, which is represented by an arrow when a dialog box is invoked. The titlebar displays the name of the dialog box. The tabs specify the various sections with a group of related options under them. The check boxes are toggle options for making the particular option available or unavailable. The drop-down list displays an item and an arrow on the right which when selected displays a list of items to be chosen from.

You can make a selection in the radio buttons. Only one can be selected at a time. The image displays the preview image of the item selected. The text box is an area where you can enter a text such as a file name. It is also called an edit box because you can make changes to the text entered. In some dialog boxes, there is the [...] button, which displays another related dialog box. There are certain buttons (**OK**, **Cancel**, **Help**) at the bottom of the dialog box. The function of each of these buttons is implied in their name. The button with a dark border is the default button. The dialog box has the **Help** button for getting help on the various features of the dialog box.

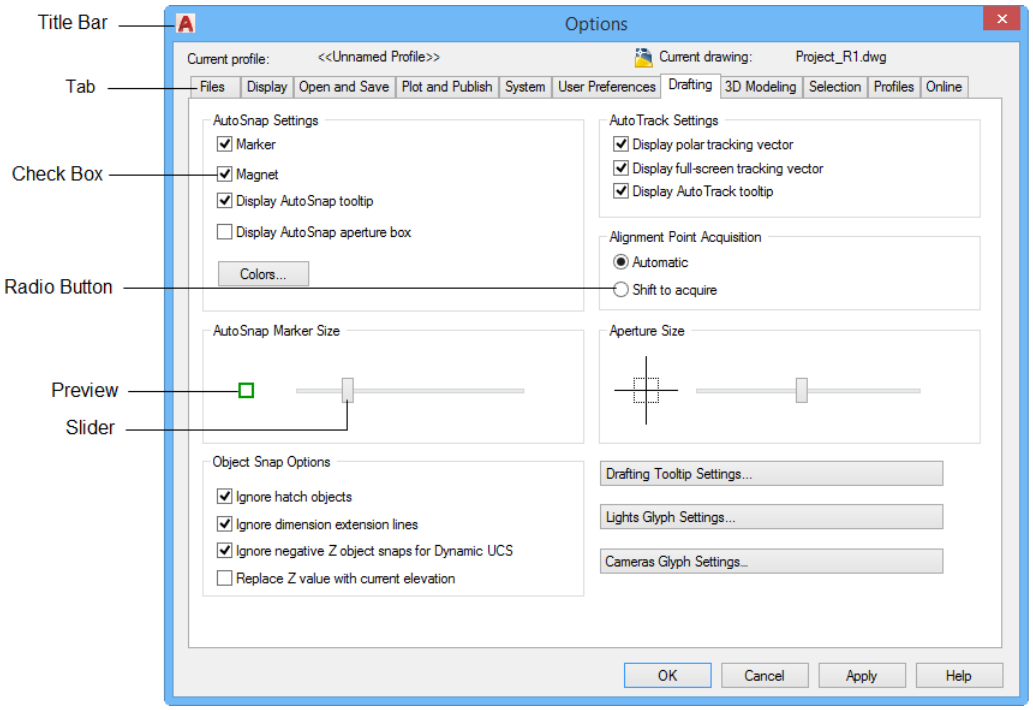

Figure 1-23 *The components of a dialog box*

STARTING A NEW DRAWING

Application Menu: New > Drawing **Menu Bar:** File > New
Quick Access Toolbar: New **Command:** NEW or QNEW

You can open a new drawing using the **New** tool in the Quick Access Toolbar. When you invoke the **New** tool, by default AutoCAD will display the **Select template** dialog box, as shown in Figure 1-24. This dialog box displays a list of default templates available in AutoCAD. The default selected template is ***acad.dwt***, which starts the 2D drawing environment. You can select the ***acad3D.dwt*** template to start the 3D modeling environment. Alternatively, you can select any other template to start a new drawing that will use the settings of the selected template. You can also open any drawing without using any template either in metric or imperial system. To do so, choose the down arrow on the right of the **Open** button and choose the **Open with no Template-Metric** option or the **Open with no Template-Imperial** option from the flyout.

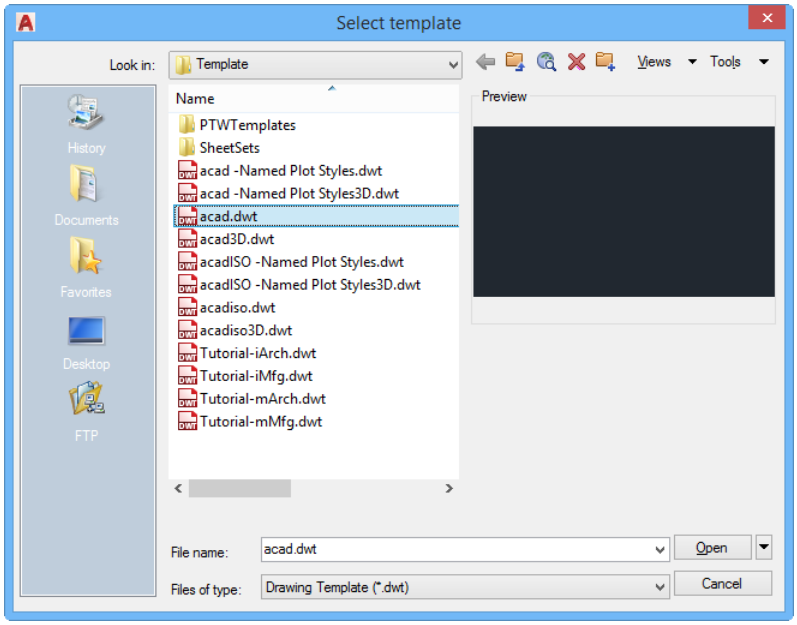

Figure 1-24 *The* ***Select template*** *dialog box*

You can also open a new drawing using the **Use a Wizard** and **Start from Scratch** options from the **Create New Drawing** dialog box. By default, this dialog box is not invoked. To invoke the **Create New Drawing** dialog box, enter **STARTUP** in the command window and then enter **1** as the new value for this system variable. After setting **1** as the new value for the system variable, whenever you invoke the **New** tool, the **Create New Drawing** dialog box will be displayed, as shown in Figure 1-25. The options in this dialog box are discussed next.

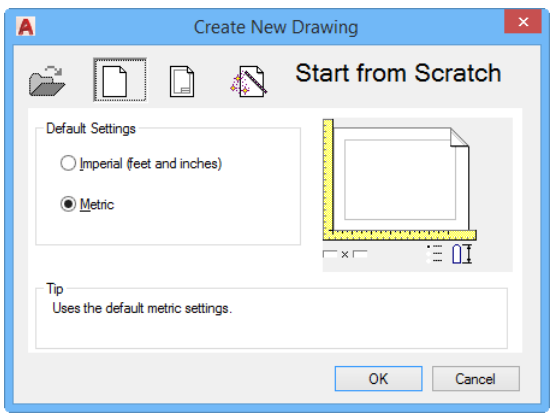

Figure 1-25 *The* ***Create New Drawing*** *dialog box*

Open a Drawing

By default, this option is not available. This option will be available when you start a new session of AutoCAD. This option is discussed later in this chapter.

Start from Scratch

When you choose the **Start from Scratch** button (Figure 1-25), AutoCAD provides you with options to start a new drawing that contains the default AutoCAD setup for Imperial (*Acad.dwt*) or Metric drawings (*Acadiso.dwt*). If you select the Imperial default setting, the limits are 12X9, text height is 0.20, and dimensions and linetype scale factors are 1.

Use a Template

When you choose the **Use a Template** button in the **Create New Drawing** dialog box, AutoCAD displays a list of templates, see Figure 1-26. The default template file is *Acad.dwt* or *Acadiso.dwt*, depending on the installation. You can directly start a new file in the 2D sketching environment by selecting the *acad.dwt* or *acadiso.dwt* template. If you use a template file, the new drawing will have the same settings as specified in the template file. All the drawing parameters of the new drawing such as units, limits, and other settings are already set according to the template file used. The preview of the template file selected is displayed in the dialog box. You can also define your own template files that are customized to your requirements (see Chapter 15, *Template Drawings*). To differentiate the template files from the drawing files, the template files have a *.dwt* extension whereas the drawing files have a *.dwg* extension. Any drawing file can be saved as a template file. You can use the **Browse** button to select other template files. When you choose the **Browse** button, the **Select a template file** dialog box is displayed with the **Template** folder open, displaying all the template files.

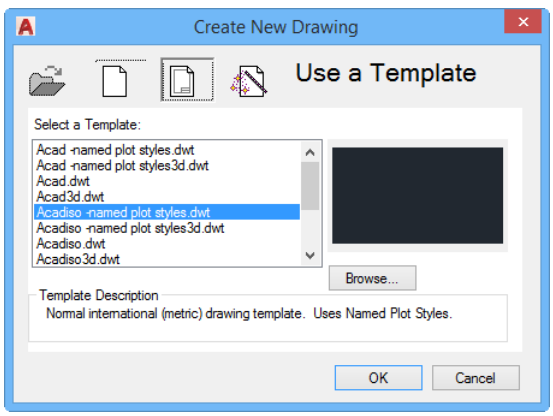

Figure 1-26 *The default templates displayed on choosing the* ***Use a Template*** *button*

Use a Wizard

The **Use a Wizard** option allows you to set the initial drawing settings before actually starting a new drawing. When you choose the **Use a Wizard** button, AutoCAD provides you with the option for using the **Quick Setup** or **Advanced Setup**, see Figure 1-27. In the **Quick Setup**, you can specify the units and the limits of the work area. In the **Advanced Setup**, you can set the units, limits, and the different types of settings for a drawing.

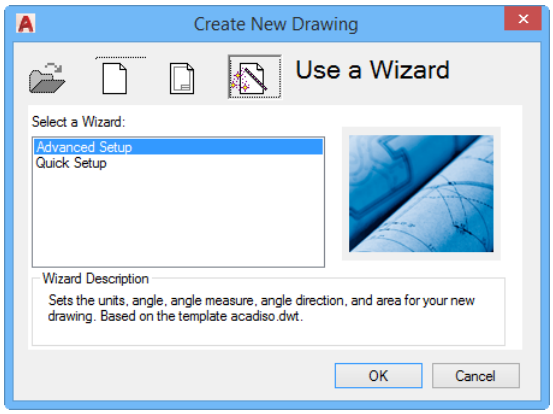

Figure 1-27 *The wizard options displayed on choosing the* ***Use a Wizard*** *button*

Advanced Setup

This option allows you to preselect the parameters of a new drawing such as the units of linear and angular measurements, type and direction of angular measurements, approximate area desired for the drawing, precision for displaying the units after decimal, and so on. When you select the **Advanced Setup** wizard option from the **Create New Drawing** dialog box and choose the **OK** button, the **Advanced Setup** wizard is displayed. In the wizard, the **Units** page is displayed by default, as shown in Figure 1-28.

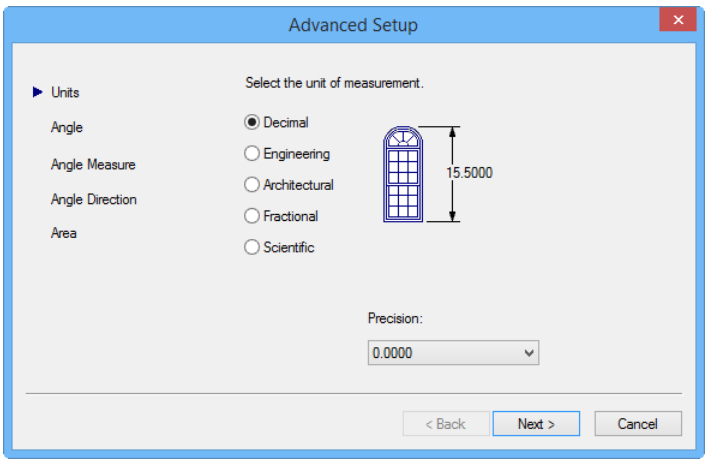

Figure 1-28 *The* ***Units*** *page of the* ***Advanced Setup*** *wizard*

This page is used to set the units for measurement in the current drawing. You can select the required unit of measurement by selecting the respective radio button. You will notice that the preview image is modified accordingly. The different units of measurement you can choose from are Decimal, Engineering, Architectural, Fractional, and Scientific. You can also set the precision for the measurement units by selecting the required option from the **Precision** drop-down list.

Choose the **Next** button to open the **Angle** page, as shown in Figure 1-29. You will notice that an arrow appears on the left of **Angle** in the **Advanced Setup** wizard. This implies that this page is current.

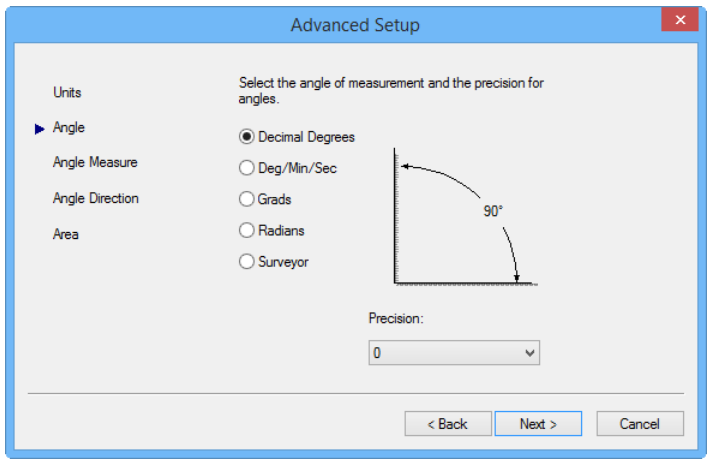

Figure 1-29 *The* ***Angle*** *page of the* ***Advanced Setup*** *wizard*

This page is used to set the units for angular measurements and its precision. The units for angle measurement are Decimal Degrees, Deg/Min/Sec, Grads, Radians, and Surveyor. The units for angle measurement can be set by selecting any one of these radio buttons as required. The preview of the selected angular unit is displayed on the right of the radio buttons. The precision format changes automatically in the **Precision** drop-down list depending on the angle measuring system selected. You can then select the precision from the drop-down list.

The next page is the **Angle Measure** page, as shown in Figure 1-30. This page is used to select the direction of the baseline from which the angles will be measured. You can also set your own direction by selecting the **Other** radio button and then entering the value in the corresponding edit box. This edit box is enabled when you select the **Other** radio button.

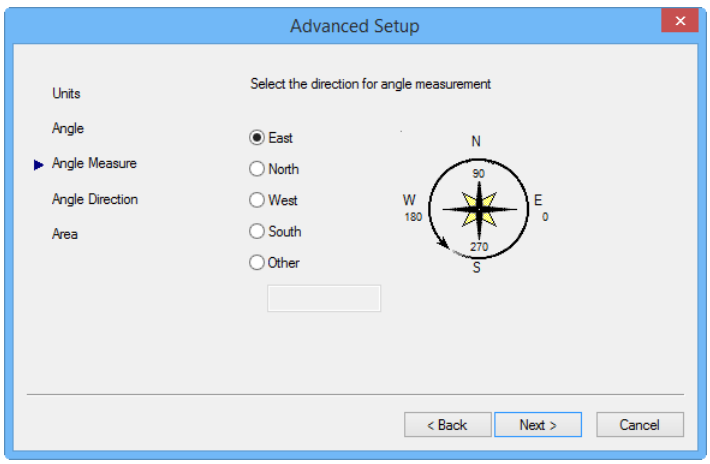

Figure 1-30 *The* ***Angle Measure*** *page of the* ***Advanced Setup*** *wizard*

Choose **Next** to display the **Angle Direction** page (Figure 1-31) to set the orientation for the angle measurement. By default, the angles are positive, if measured in a counterclockwise direction. This is because the **Counter-Clockwise** radio button is selected. If you select the **Clockwise** radio button, the angles will be considered positive when measured in the clockwise direction.

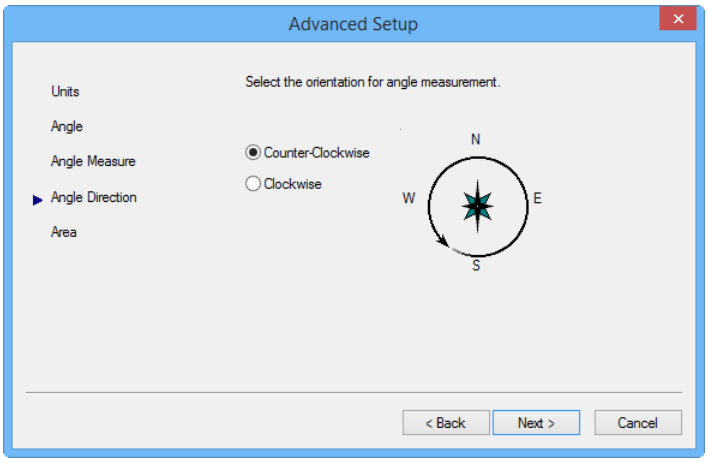

Figure 1-31 *The **Angle Direction** page of the **Advanced Setup** wizard*

To set the limits of the drawing, choose the **Next** button; the **Area** page will be displayed, as shown in Figure 1-32. You can enter the width and length of the drawing area in the respective edit boxes.

Note

*Even after you increase the limits of the drawing, the drawing display area is not increased. You need to invoke the **Zoom All** tool from the Navigation Bar to increase the drawing display area.*

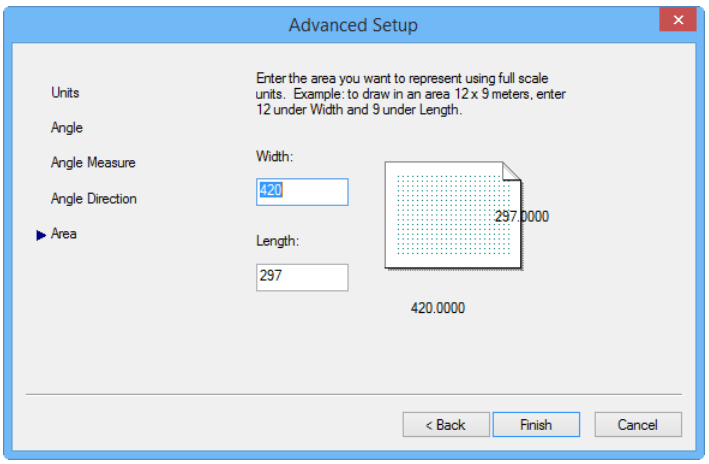

Figure 1-32 *The **Area** page of the **Advanced Setup** wizard*

Quick Setup

When you select the **Quick Setup** option from the **Create New Drawing** dialog box and choose the **OK** button, the **QuickSetup** wizard is displayed. This wizard has two pages: **Units** and **Area**.

The **Units** page is opened by default, as shown in Figure 1-33. The options in the **Units** page are similar to those in the **Units** page of the **Advanced Setup** wizard. The only difference is that you cannot set the precision for the units in this wizard.

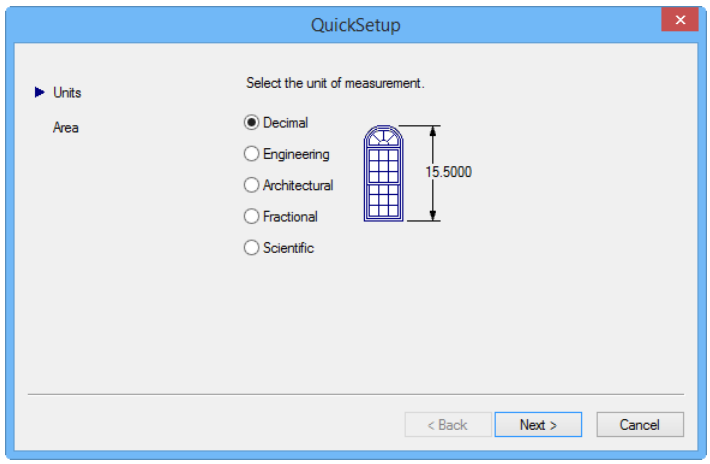

Figure 1-33 *The* ***Units*** *page of the* ***QuickSetup*** *wizard*

Choose **Next** to display the **Area** page, as shown in Figure 1-34. The **Area** page of the **Quick Setup** is similar to that of the **Advanced Setup** wizard. In this page, you can set the drawing limits.

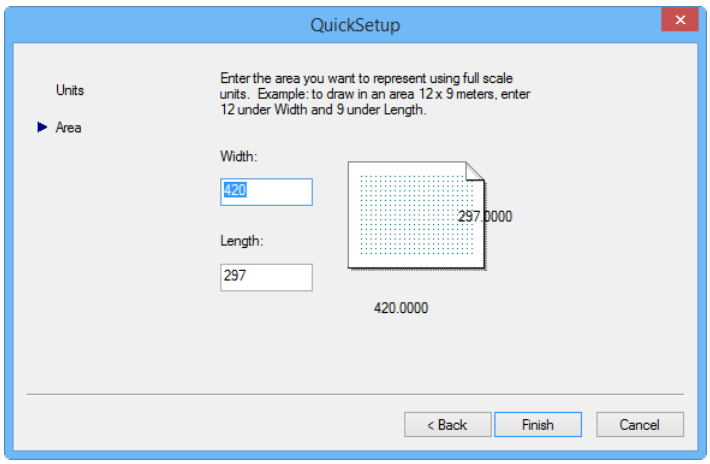

Figure 1-34 *The* ***Area*** *page of the* ***QuickSetup*** *wizard*

Tip

When you open an AutoCAD session, a drawing opens automatically. But you can open a new drawing using options such as ***Start from Scratch*** *and* ***Use a Wizard*** *before entering into AutoCAD environment using the* ***Startup*** *dialog box. As mentioned earlier, the display of the* ***Startup*** *dialog box is turned off by default. Refer to the section of* ***Starting a New Drawing*** *to know how to turn on the display of this dialog box.*

SAVING YOUR WORK

Application Menu: SAVE AS, SAVE **Menu Bar:** File > Save or Save As
Quick Access Toolbar: Save or Save As **Command:** QSAVE, SAVEAS, SAVE

You must save your work before you exit the drawing editor or turn off your system. Also, it is recommended that you save your drawings in regular intervals, so that in the event of a power failure or an editing error, all work saved before the problem started is retained.

AutoCAD has provided the **QSAVE**, **SAVEAS**, and **SAVE** commands that allow you to save your work. Also, these commands allow you to save your drawing by writing it to a permanent storage device, such as a hard drive or in any removable drive.

When you choose the **Save** tool from the Quick Access Toolbar or the **Application Menu**, the **QSAVE** command is invoked. If the current drawing is unnamed and you save the drawing for the first time in the current session, the **SAVEAS** command will be invoked and you will be prompted to enter the file name in the **Save Drawing As** dialog box, as shown in Figure 1-35. You can enter the name for the drawing and then choose the **Save** button. If you have saved a drawing file once and then edited it, you can use the **Save** tool to save it, without the system prompting you to enter a file name. This allows you to do a quick save.

When you choose **Save As** from the **Application Menu** or choose the **Save As** tool from the Quick Access Toolbar, the **Save Drawing As** dialog box is displayed, similar to that shown in Figure 1-35. Even if the drawing has been saved with a file name, this tool gives you an option to save it with a different file name. In addition to saving the drawing, it sets the name of the current drawing to the file name you specify, which is displayed in the title bar. This tool is used when you want to save a previously saved drawing with a different file name. You can also use this tool when you make certain changes to a template and want to save the changed template drawing but leave the original template unchanged.

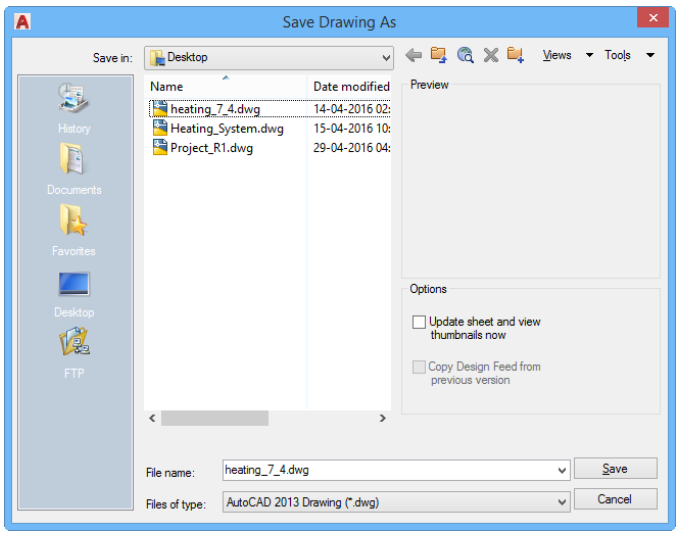

Figure 1-35 *The* ***Save Drawing As*** *dialog box*

Save Drawing As Dialog Box

The **Save Drawing As** dialog box displays the information related to the drawing files on your system. The various components of the dialog box are described next.

Places List

A column of icons is displayed on the left side of the dialog box. These icons contain the shortcuts to the folders that are frequently used. You can quickly save your drawings in one of these folders. The **History** folder displays the list of the most recently saved drawings. You can save your personal drawings in the **Documents** or the **Favorites** folder. The **FTP** folder displays the list of the various FTP sites that are available for saving the drawing. By default, no FTP sites are shown in the dialog box. To add a FTP site to the dialog box, choose the **Tools** button on the upper-right corner of the dialog box to display a shortcut menu and select **Add/Modify FTP Locations**. The **Desktop** folder displays the list of contents on the desktop. You can add a new folder in this list for an easy access by simply dragging the folder on to the **Places** list area. You can rearrange all these folders by dragging them and then placing them at the desired locations. It is also possible to remove the folders, which are not in frequent use. Right-click on the particular folder and then select **Remove** from the shortcut menu. Now, you can also save the document to a new location of Autodesk Cloud. The option for saving the document is discussed next.

File name Edit Box

To save your work, enter the name of the drawing in the **File name** edit box by typing the file name or selecting it from the drop-down list. If you select the file name, it automatically appears in the **File name** edit box. If you have already assigned a name to the drawing, the current drawing name is taken as the default name. If the drawing is unnamed, the default name *Drawing1* is displayed in the **File name** edit box. You can also choose the down arrow located on the right of the edit box to display the names of the previously saved drawings and choose a name here.

Files of type Drop-down List

The **Files of type** drop-down list (Figure 1-36) is used to specify the drawing format in which you want to save the file. For example, to save the file as an AutoCAD 2004 drawing file, select **AutoCAD 2004/LT2004 Drawing (*.dwg)** from the drop-down list.

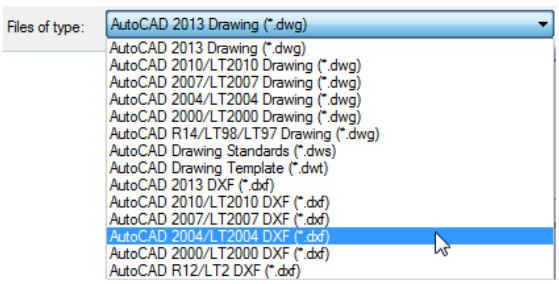

Figure 1-36 *The* ***Files of type*** *drop-down list*

Save in Drop-down List

The current drive and path information is listed in the **Save in** drop-down list. AutoCAD will initially save the drawing in the default folder, but if you want to save the drawing in a different folder, you have to specify the path. For example, to save the present drawing as *house*

in the *C1* folder, choose the arrow button in the **Save in** drop-down list to display the drop-down list. Select **C**: from the drop-down list; all folders in the C drive will be listed in the **File** list box. Double-click on the **C1** folder, if it is already listed there or create a folder C1 by choosing the **Create New Folder** button. Select *house* from the list, if it is already listed there, or enter it in the **File name** edit box and then choose the **Save** button. Your drawing (*house*) will be saved in the *C1* folder (*C:\C1\house.dwg*). Similarly, to save the drawing in the D drive, select **D:** in the **Save in** drop-down list.

Tip

The file name you enter to save a drawing should match its contents. This helps you to remember the drawing details and make it easier to refer them later. Also, the file name can be 255 characters long and can contain spaces and punctuation marks.

Views drop-down list

The **Views** drop-down list has the options for the type of listing of files and displaying the preview images (Figure 1-37).

List, Details, and Thumbnails Options

If you choose the **Details** option, it will display the detailed information about the files (size, type, date, and time of modification) in the **Files** list box. In the detailed information, if you click on the **Name** label, the files are listed with the names in alphabetical order. If you double-click on the **Name** label, the files will be listed in the reverse order. Similarly, if you click on the **Size** label, the files are listed according to their size in ascending order. Double-clicking on the **Size** label will list the files in descending order of size. Similarly, you can click on the **Type** label or the **Date Modified** label to list the files accordingly. If you choose the **List** option, all files present in the current folder will be listed in the **File** list box. If you select the **Thumbnails** option, the list box displays the preview of all the drawings, along with their names displayed at the bottom of the drawing preview.

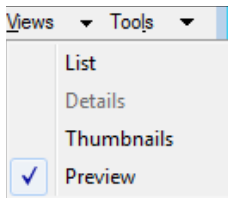

Figure 1-37 *The* ***Views*** *drop-down list*

Create New Folder Button

If you choose the **Create New Folder** button, AutoCAD creates a new folder under the name **New Folder**. The new folder is displayed in the **File** list box. You can accept the name or change it to your requirement. You can also use the ALT+5 keys to create a new folder.

Up one level Button

The **Up one level** button displays the folders that are up by one level. For example, if you are in the *Sample* subfolder of the *AutoCAD 2017* folder, then choosing the **Up one level** button will take you to the *AutoCAD 2017* folder. You can also use the ALT+2 keys to do the same.

Search the Web

It displays the **Browse the Web** dialog box that enables you to access and store AutoCAD files on the Internet. You can also use the ALT+3 keys to browse the Web when this dialog box is available on the screen.

Tools drop-down list

The **Tools** drop-down list (Figure 1-38) has an option for adding or modifying the FTP sites. These sites can then be browsed from the FTP shortcut in the **Places** list. The **Add Current Folder to Places** and **Add to Favorites** options add the folder displayed in the **Save in** edit box to the **Places** list or to the Favorites folder. The **Options** button displays the **Saveas Options** dialog box where you can save the proxy images of the custom objects. It has the **DWG Options** and **DXF Options** tabs. The **Digital Signatures** button displays the **Digital Signatures** dialog box, which is used to configure the security options of the drawing.

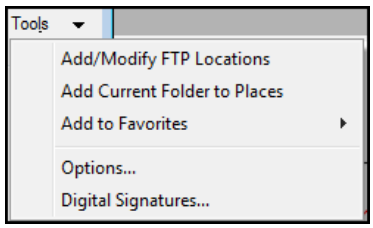

***Figure 1-38** The **Tools** drop-down list*

AUTOMATIC TIMED SAVE

AutoCAD allows you to save your work automatically at specific intervals. To change the time intervals, you can specify the intervals duration in minutes in the **Minutes between saves** text box in the **File Safety Precautions** area in the **Options** dialog box (**Open and Save** tab). This dialog box can be invoked by choosing the **Options** button from the **Application Menu**. Depending on the power supply, hardware, and type of drawings, you should decide on an appropriate time and assign it to this variable. AutoCAD saves the drawing with the file extension *.sv$*. You can also change the time interval by using the **SAVETIME** system variable.

Tip

*Although the automatic save feature saves your drawing after a certain time interval, you should not completely depend on it because the procedure for converting the ac$ file into a drawing file is cumbersome. Therefore, it is recommended that you save your files regularly using the **QSAVE** or **SAVEAS** commands.*

CREATING BACKUP FILES

If the drawing file already exists and you use **Save** or **Save As** tool to update the current drawing, AutoCAD creates a backup file. AutoCAD takes the previous copy of the drawing and changes it from a file type *.dwg* to *.bak*, and the updated drawing is saved as a drawing file with the *.dwg* extension. For example, if the name of the drawing is *myproj.dwg*, AutoCAD will change it to *myproj.bak* as a backup file and save the current drawing as *myproj.dwg*.

Changing Automatic Timed Saved and Backup Files into AutoCAD Format

Sometimes, you may need to change the automatic timed saved and backup files into AutoCAD format. To change a backup file into AutoCAD format, open the folder, in which you have saved the backup or the automatic timed saved drawing using **Computer** or **Windows Explorer**. Choose **Organize > Folder and search options** from the menu bar to invoke the **Folder Options** dialog box. Choose the **View** tab and clear the **Hide extensions for known file types** check box under the **Advanced settings** area, if selected. Exit the dialog box. Rename the automatic saved drawing or the backup file with a different name and also change the extension of the drawing from *.sv$* or *.bak* to *.dwg*.

Using the Drawing Recovery Manager to Recover Files

The files that are saved automatically can also be retrieved by using the **DRAWING RECOVERY MANAGER**. You can open the **DRAWING RECOVERY MANAGER** again by choosing **Drawing Utilities > Open the Drawing Recovery Manager** from the **Application Menu** or by entering **DRAWINGRECOVERY** at the Command prompt.

If the automatic save operation is performed in a drawing and the system crashes accidentally, the next time you run AutoCAD, the **Drawing Recovery** message box will be displayed, as shown in Figure 1-39. The message box informs you that the program unexpectedly failed and you can open the most suitable among the backup files created by AutoCAD. Choose the **Close** button from the **Drawing Recovery** message box; the **DRAWING RECOVERY MANAGER** is displayed on the left of the drawing area, as shown in Figure 1-40.

The **Backup Files** rollout lists the original files, the backup files, and the automatically saved files. Select a file; its preview will be displayed in the **Preview** rollout. Also, the information corresponding to the selected file will be displayed in the **Details** rollout. To open a backup file, double-click on its name in the **Backup Files** rollout. Alternatively, right-click on the file name and then choose **Open** from the shortcut menu. It is recommended that you save the backup file at the desired location before you start working on it.

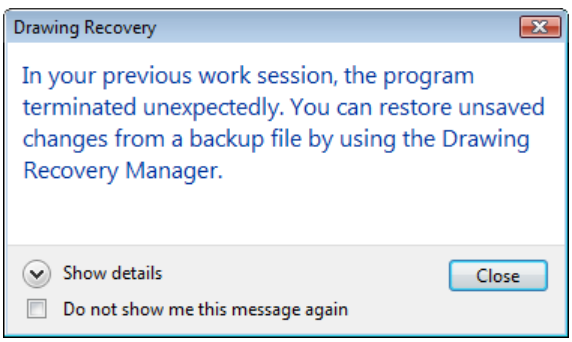

*Figure 1-39 The **Drawing Recovery** message box*

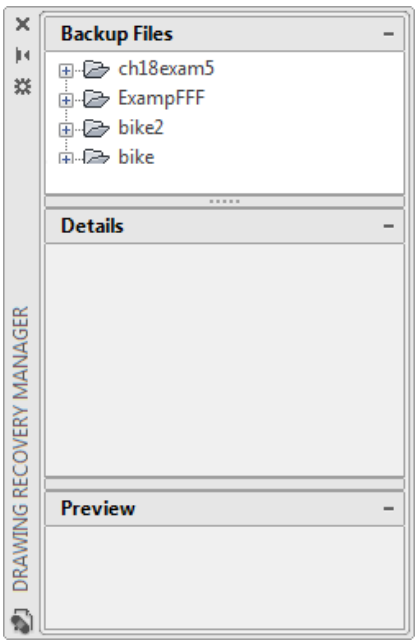

*Figure 1-40 The **DRAWING RECOVERY MANAGER***

CLOSING A DRAWING

You can close the current drawing file without actually quitting AutoCAD by choosing **Close > Current Drawing** from the **Application Menu** or by entering **CLOSE** at the Command prompt. If multiple drawing files are opened, choose **Close > All Drawings** from the **Application Menu**. If you have not saved the drawing after making the last changes to it and you invoke the **CLOSE**

command, AutoCAD displays a dialog box that allows you to save the drawing before closing. This box gives you an option to discard the current drawing or the changes made to it. It also gives you an option to cancel the command. After closing the drawing, you are still in AutoCAD from where you can open a new or an already saved drawing file. You can also use the close button (**X**) of the drawing area to close the drawing.

OPENING AN EXISTING DRAWING

Application Menu: Open > Drawing **Quick Access Toolbar:** Open
Menu Bar: File > Open **Command:** OPEN

You can open an existing drawing file that has been saved previously. There are three methods that can be used to open a drawing file: by using the **Select File** dialog box, by using the **Create New Drawing** dialog box, and by dragging and dropping.

Opening an Existing Drawing Using the Select File Dialog Box

If you are already in the drawing editor and you want to open a drawing file, choose the **Open** tool from the Quick Access Toolbar; the **Select File** dialog box will be displayed, see Figure 1-41. You can select the drawing to be opened using this dialog box. This dialog box is similar to the standard dialog boxes. You can choose the file you want to open from the folder in which it is stored. You can change the folder from the **Look in** drop-down list. You can then select the name of the drawing from the list box or you can enter the name of the drawing file you want to open in the **File name** edit box. After selecting the drawing file, you can choose the **Open** button to open the file. Here, you can choose *Drawing1* from the list and then choose the **Open** button to open the drawing.

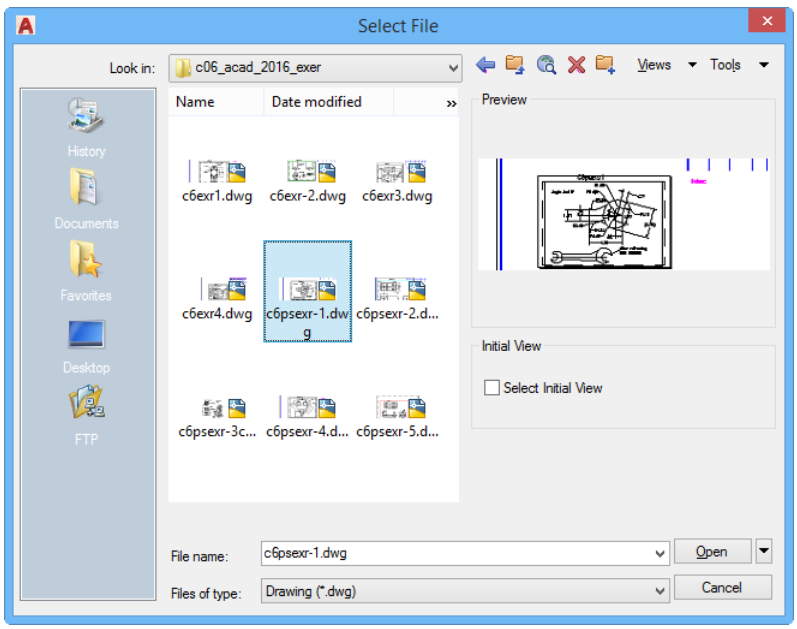

Figure 1-41 *The* ***Select File*** *dialog box*

When you select a file name, its image is displayed in the **Preview** box. If you are not sure about the file name of a particular drawing but know the contents, you can select the file names and

look for the particular drawing in the **Preview** box. You can also change the file type by selecting it in the **Files of type** drop-down list. Apart from the *dwg* files, you can open the *dwt* (template) files or the *dxf* files. You have all the standard icons in the **Places** list that can be used to open drawing files from different locations. The **Open** button has a drop-down list, as shown in Figure 1-42. You can choose a method for opening the file using this drop-down list. These methods are discussed next.

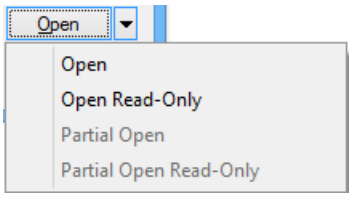

Figure 1-42 *The* ***Open*** *drop-down list*

Open Read-Only

To view a drawing without altering it, you must select the **Open Read-Only** option from the drop-down list. In other words, read only protects the drawing file from changes. AutoCAD does not prevent you from editing the drawing, but if you try to save the opened drawing with the original file name, AutoCAD warns you that the drawing file is write protected. However, you can save the edited drawing to a file with a different file name using the **Save Drawing as** dialog box. This way you can preserve your drawing.

Partial Open

The **Partial Open** option enables you to open only a selected view or a selected layer of a selected drawing. This option can be used to edit small portions of a complicated drawing and then save it with the complete drawing. When you select the **Partial Open** option from the **Open** drop-down list, the **Partial Open** dialog box (Figure 1-43) is displayed, which contains different views and layers of the selected drawing.

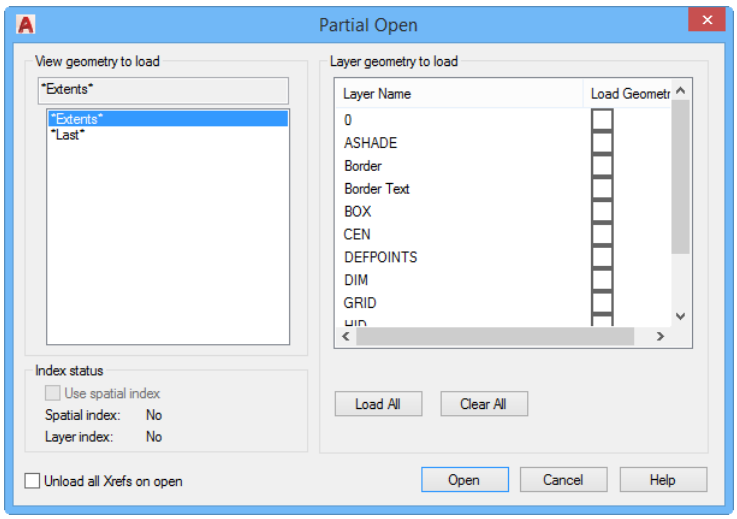

Figure 1-43 *The* ***Partial Open*** *dialog box*

When you select a check box for a layer and then choose the **Open** button, only the objects drawn in that particular layer for the drawing are displayed in the new drawing window. You can make the changes and then save it. For example, in the *C:/Program Files/Autodesk/AutoCAD 2017* folder, double-click on the **Sample** folder and then **Mechanical Sample** and select *Mechanical - Xref* from the list. Now, choose the down arrow on the right of the **Open** button to display the drop-down

list and choose **Partial Open**. All the views and layers of this drawing are displayed in the **Partial Open** dialog box (click **Yes**, if AutoCAD prompts for read-only files). Select the check box on the right of the layer that you want to open. When you choose the **Open** button, after selecting the layers, only the selected layers of the drawing will be opened.

Note

The concept of layers is discussed in Chapter 4, Working with Drawing Aids.

Loading Additional Objects to Partially Opened Drawing

Once you have opened a part of a drawing and made the necessary changes, you may want to load additional objects or layers on the existing ones. This can be done by using the **PARTIALOAD** command, which can be invoked by choosing **File > Partial Load** from the menu bar or by entering **PARTIALOAD** at the Command prompt. This command displays the **Partial Load** dialog box, which is similar to the **Partial Open** dialog box. You can choose another layer and the objects drawn in it will be added to the partially loaded drawing.

Note

1. *The **Partial Load** option is not enabled in the **File** menu unless a drawing is partially opened.*
2. *Loading a drawing partially is a good practice when you are working with objects on a specific layer in a large complicated drawing.*
3. *In the **Select File** dialog box, the preview of a drawing which was partially opened and then saved is not displayed.*

Tip

If a drawing was partially opened and saved previously, it is possible to open it again with the same layers and views. AutoCAD remembers the settings so that while opening a previously partially opened drawing, a dialog box is displayed prompting for an option to fully open it or restore the partially opened drawing.

Select Initial View

A view is defined as the way you look at an object. Select the **Select Initial View** check box if you want to load a specific view initially when AutoCAD loads the drawing. This option will work, if the drawing has saved views. This is generally used while working on a large complicated drawing, in which you want to work on a particular portion of the drawing. You can save that particular portion as a view and then select it to open the drawing next time. You can save a desired view by using AutoCAD's **VIEW** command (see "Creating Views" topic, Chapter 6). If the drawing has no saved views, selecting this option will load the last view. If you select the **Select Initial View** check box and then the **OK** button, AutoCAD will display the **Select Initial View** dialog box. You can select the view name from this dialog box, and AutoCAD will load the drawing with the selected view displayed.

Tip

*1. Apart from opening a drawing from the **Startup** dialog box or the **Select File** dialog box, you can also open a drawing from the **Application Menu**. By default, the **Recent Documents** option is selected in the **Application Menu**. As a result, the most recently opened drawings will be displayed and you can open the required file from it.*

2. It is possible to open an AutoCAD 2000 drawing in AutoCAD 2017. However, when you save this drawing, it is automatically converted and saved as an AutoCAD 2017 drawing file.

Opening an Existing Drawing Using the Startup Dialog Box

If you have configured the settings to show the **Startup** dialog box by setting the **STARTUP** system variable value as **1**, the **Startup** dialog box will be displayed every time you start a new AutoCAD session. The first button in this dialog box is the **Open a Drawing** button. When you choose this button, a list of the recently opened drawings will be displayed for you to select from it, see Figure 1-44. Note that this button is activated only when this dialog box is displayed by default on starting a new session of AutoCAD. The **Browse** button displays the **Select File** dialog box, which allows you to browse another file.

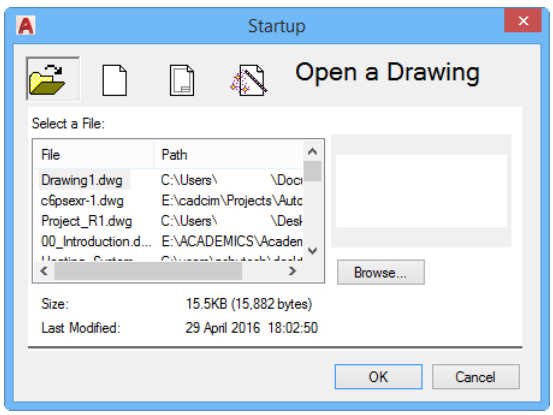

Figure 1-44 *List of the recently opened drawings*

Note

*The display of the dialog boxes related to opening and saving drawings will be disabled, if the **STARTUP** and the **FILEDIA** system variables are set to 0. The initial value of these variables is 1.*

Opening an Existing Drawing Using the Drag and Drop Method

You can also open an existing drawing in AutoCAD by dragging it from the Window Explorer and dropping it into AutoCAD. If you drop the selected drawing in the drawing area, the drawing will be inserted as a block and as a result, you cannot modify it. But if you drag the drawing from the Window Explorer and drop it anywhere other than the drawing area, AutoCAD will display the selected drawing.

QUITTING AutoCAD

You can exit the AutoCAD program by using the **EXIT** or **QUIT** commands. Even if you have an active command, you can choose the **Exit Autodesk AutoCAD 2017** from the **Application Menu** to quit the AutoCAD program. In case the drawing has not been saved, it allows you to save the work first through a dialog box. Note that if you choose **No** in this dialog box, all the changes made in the current list till the last save will be lost. You can also use the **Close** button (**X**) of the main AutoCAD window (present in the title bar) to end the AutoCAD session.

CREATING AND MANAGING WORKSPACES

A workspace is defined as a customized arrangement of **Ribbon**, toolbars, menus, and window palettes in the AutoCAD environment. You can create your own workspaces, in which only specified toolbars, menus, and palettes are available. When you start AutoCAD, by default, the **Drafting & Annotation** workspace is displayed as the current workspace. You can select any other predefined workspace from the **Workspace** drop-down list available in the title bar, next to the Quick Access Toolbar, see Figure 1-45. You can also set the workspace from the flyout that will be displayed on choosing the **Workspace Switching** button on the Status Bar or by choosing the required workspace from the Quick Access Toolbar.

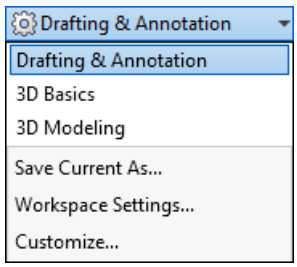

Figure 1-45 *The predefined workspaces*

Creating a New Workspace

To create a new workspace, customize the **Ribbon** and invoke the palettes to be displayed in the new workspace. Next, select the **Save Current As** option from the **Workspace** drop-down list in the titlebar; the **Save Workspace** dialog box will be displayed, as shown in Figure 1-46. Enter the name of the new workspace in the **Name** edit box and choose the **Save** button.

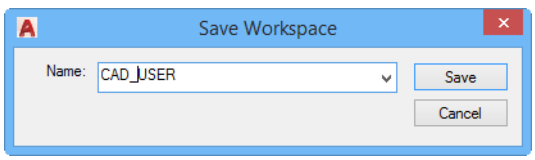

Figure 1-46 *The **Save Workspace** dialog box*

The new workspace is now the current workspace and is added to the drop-down list in the title bar. Likewise, you can create workspaces based on your requirement and switch from one workspace to the other by selecting the name from the drop-down list in the **Workspaces** toolbar or the drop-down list in the title bar.

Modifying the Workspace Settings

AutoCAD allows you to modify the workspace settings. To do so, select the **Workspace Settings** option in the **Workspace** drop-down list in the title bar; the **Workspace Settings** dialog box will be displayed, as shown in Figure 1-47. All workspaces are listed in the **My Workspace** drop-down list. You can make any of the workspaces as My Workspace by selecting it from the **My Workspace** drop-down list. You can also choose the **My Workspace** button from the **Workspaces** toolbar to change the current workspace to the one that was set as My Workspace in the **Workspace Settings** dialog box. The other options in this dialog box are discussed next.

Menu Display and Order Area

The options in this area are used to control the display and the order of display of workspaces in the **Workspace** drop-down list. By default, workspaces are listed in the sequence of their creation. To change the order, select a workspace and choose the **Move Up** or **Move Down** button. To control the display of the workspaces, you can select or clear the check boxes. You can also add a separator between workspaces by choosing the **Add Separator** button. A separator is a line that is placed between two workspaces in the **Workspace** drop-down list in the title bar, as shown in Figure 1-48.

When Switching Workspaces Area

By default, the **Do not save changes to workspace** radio button is selected in this area. This ensures that while switching the workspaces, the changes made in the current workspace will not be saved. If you select the **Automatically save workspace changes** radio button, the changes made in the current workspace will be automatically saved when you switch to the other workspace.

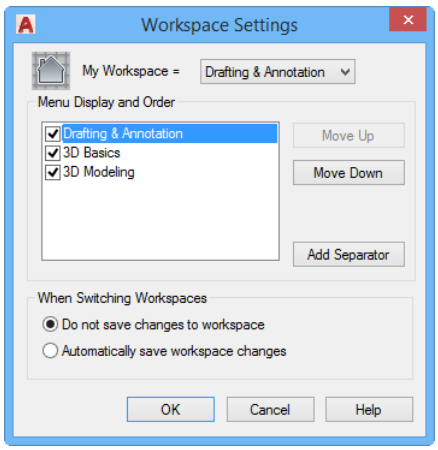

*Figure 1-47 The **Workspace Settings** dialog box*

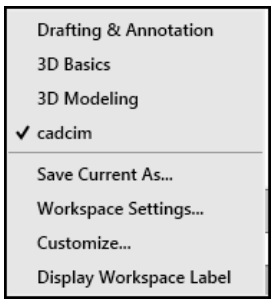

*Figure 1-48 The **Workspace** drop-down list after adding separators*

AutoCAD HELP

Titlebar: ? > Help **Shortcut Key:** F1 **Command:** HELP or ?

You can get the on-line help and documentation about the working of AutoCAD 2017 commands from the **Help** menu in the title bar, see Figure 1-49. You can also access the **Help** menu by pressing the F1 function key. An **InfoCenter** bar is displayed at the top right corner in the title bar that will help you to sign into the Autodesk Online services, see Figure 1-50. You can also access AutoCAD community by using certain keywords. Some important options in the **Help** menu are discussed next.

In the **Autodesk AutoCAD 2017 - Help** window, refer to Figure 1-51, you can choose the **Help Home** button to explore new features in AutoCAD 2017. In this window, you can browse videos, tutorials, and documentation for beginners and advanced users in the **Learn** and **Resources** sections. You can download sample files and offline help database from the **Downloads** section. You can also connect to other users of Autodesk community and discussion groups using the **Connect** section.

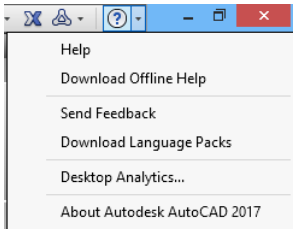

*Figure 1-49 The **Help** menu*

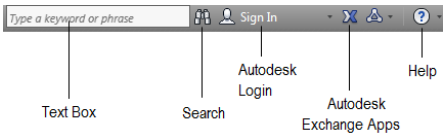

*Figure 1-50 The **InfoCenter** bar*

*Figure 1-51 The **Autodesk AutoCAD 2017 - Help** window*

In the **Autodesk AutoCAD 2017 - Help** window, you can search about any keyword related to AutoCAD by using the Search edit box, as shown in Figure 1-52. You can further refine your search by selecting the desired link using the **REFINE BY** section of the help window.

Download Offline Help

This option is used to read the help contents and topics when the user has no internet access. (AutoCAD prompts you to download help file, user can save this file and then use it later on.)

Send Feedback

You can also directly contact Autodesk regarding support, discussion, or any consulting work. When you click on this option, a page will be displayed asking for the required queries and contact details. After entering the contact details, you can press **Submit** to exit.

Download Language Packs

Once AutoCAD is installed, you can also install the **AutoCAD language pack** to run AutoCAD in your preferred language by choosing this option.

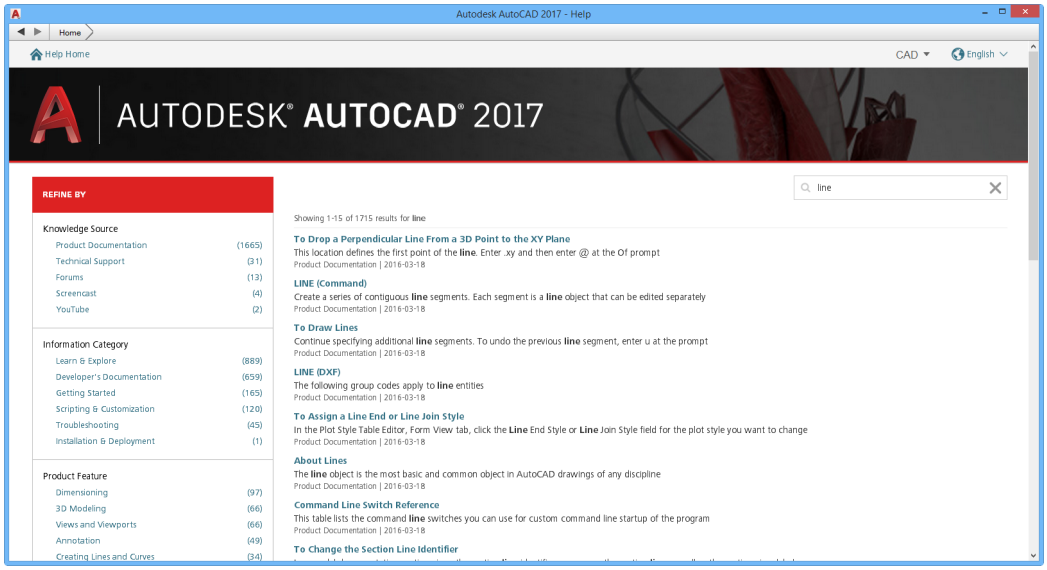

Figure 1-52 *Searching the keyword using the Search edit box*

Customer Involvement Program

This option is used to share your configuration information and uses of Autodesk products with Autodesk. The collected information is used by Autodesk for the improvement of Autodesk softwares.

Desktop Analytics

When you choose this option, the **Data collection and use** dialog box will be displayed. In the dialog box, Autodesk will prompt you to participate in a program to simply collect nonpersonal product usage information from the Autodesk software.

About Autodesk AutoCAD 2017

This option gives you information about the release, serial number, licensed to, and also the legal description about AutoCAD.

Autodesk Cloud

The Autodesk Cloud is used to save and share the documents online. Using this technology, you can also view and update other user's documents available on Autodesk Cloud. To share a document, you first need to login to the Autodesk account. To do so, choose the **Sign In to Autodesk account** option from the **Sign In** flyout in the **InfoCenter** bar; the **Autodesk-Sign In** window will be displayed. Now, login to the account using the Autodesk ID and password; the **Default A360 Settings** window will be displayed. Choose the **OK** button from the window; your account name will be displayed in place of **Sign In** in the **Sign In** flyout. Next, choose the **A360** option from the **Sign In** flyout; the **A360 Drive** window will be displayed in the default browser, refer to Figure 1-53. To upload a document on Autodesk Cloud, choose the **Upload** button from the top right corner of the browser window; the **Open** dialog box will be displayed. Next, select the document to be uploaded and choose the **Open** button; the document will get uploaded in the cloud and a preview of the uploaded document will be displayed in the **All Data** area. Now,

you can share the document in the Autodesk Cloud by selecting suitable options. There are two ways to share a document: Private Sharing and Public Sharing. To share a file, hover the cursor over that file and choose the **Share** button from the highlighted options, refer to Figure 1-54; the **Share** window will be displayed, as shown in Figure 1-55. In the **Get Link** tab, you can specify the settings for sharing the file.

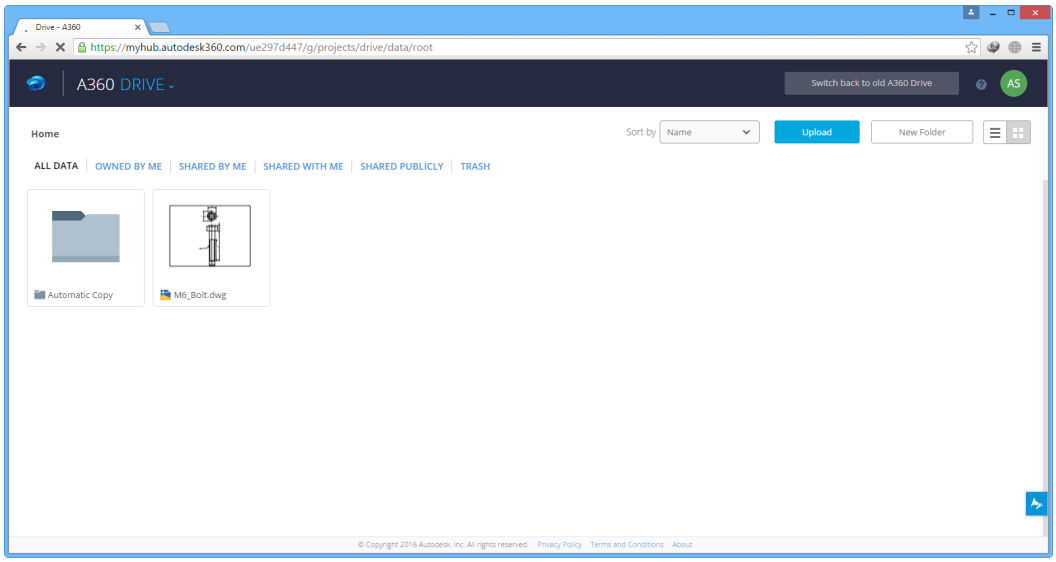

Figure 1-53 *The* ***Autodesk Cloud Documents*** *window*

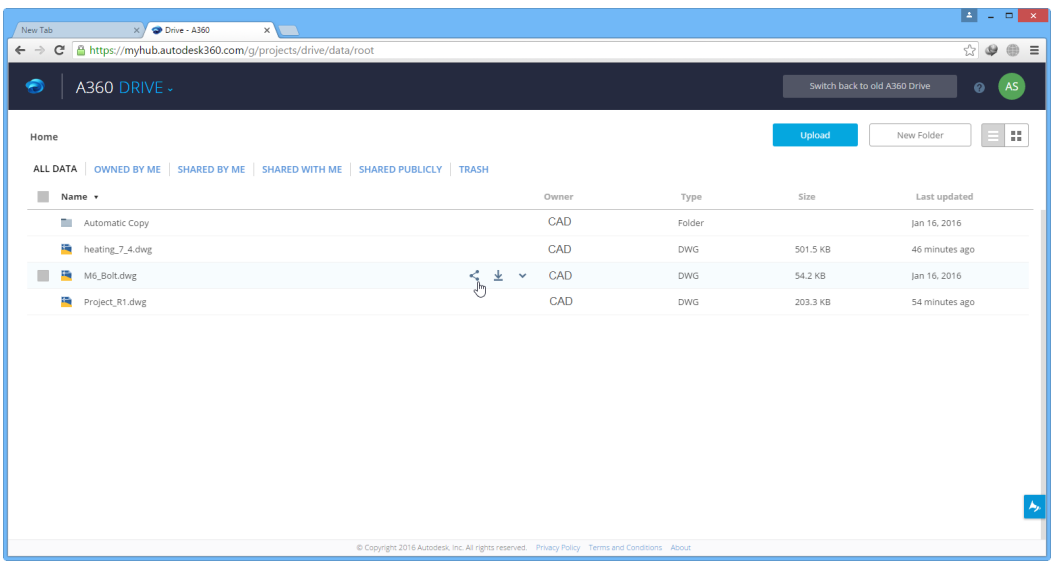

Figure 1-54 *Options highlighted on hovering the cursor on a file*

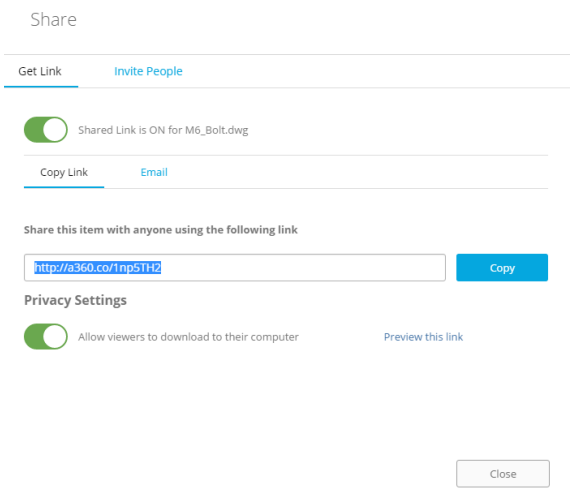

Figure 1-55 *The* ***Share*** *window*

You can also invite other users to view, download, or update the files in the A360 Drive. To do so, choose the **Invite People** tab in the **Share** window and enter a valid email address of the user whom you want to invite. Next, select the required options from the **Permission** drop-down list to specify whether the user can view, view and download, download and update, or have a full access of the shared file, refer to Figure 1-56.

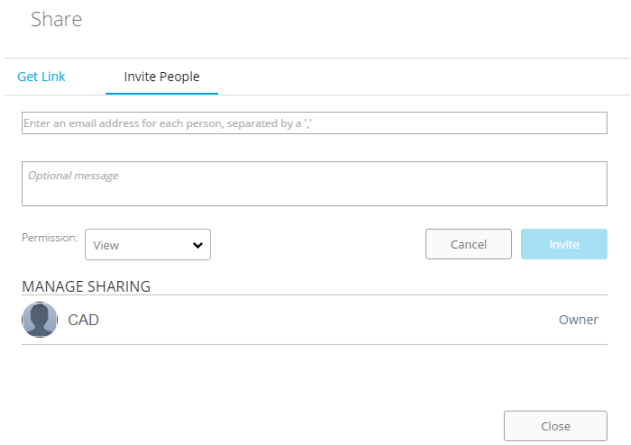

Figure 1-56 *The* ***Invite People*** *tab in the* ***Share*** *window*

Autodesk Exchange Apps

Autodesk Exchange Apps helps you to download various applications for AutoCAD, get connected to the AutoCAD community, share information and designs, and so on. You can download the AutoCAD apps from the **Featured Apps** panel of the **Featured Apps** tab in the **Ribbon**, refer to Figure 1-57. To download the AutoCAD apps, choose the required apps from the **Featured Apps** panel of the **Featured Apps** tab; the default browser will open with icons of apps that you can download, as shown in Figure 1-58. Now, you can download the apps by clicking the **Download** button.

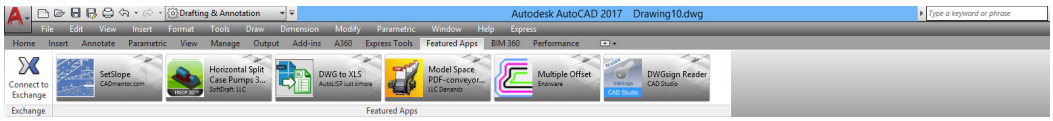

Figure 1-57 *The* ***Featured Apps*** *panel of the* ***Featured Apps*** *tab in the* ***Ribbon***

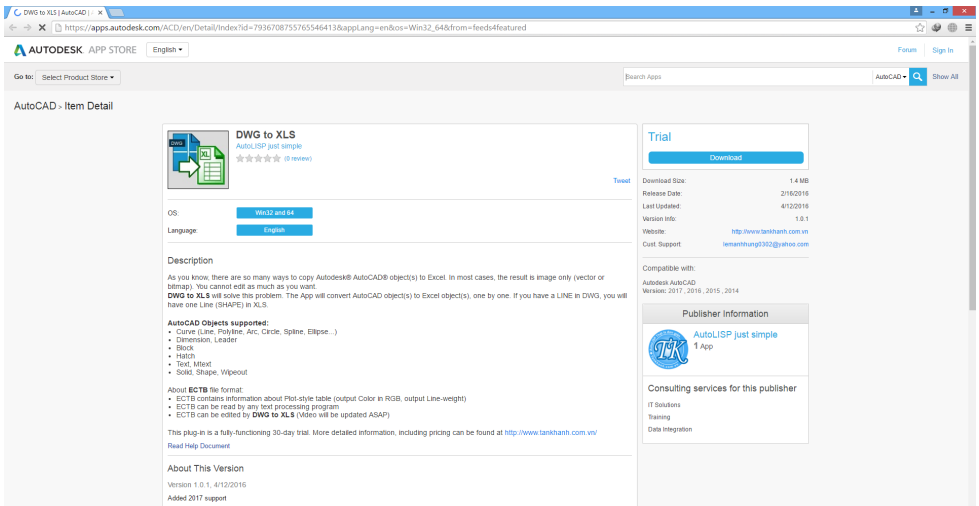

Figure 1-58 *The* ***AUTODESK APP STORE*** *window open with apps to be downloaded*

You can also download the apps other than the apps available in the **Featured Apps** panel. To do so, choose the **Connect to Exchange** button from the **Exchange** panel of the **Featured Apps** tab; the **AUTOCAD APP STORE** window will be displayed, as shown in Figure 1-59. Now, you can download the required apps from the window. Some of the apps are paid and some of them are free to download. You can download various Autodesk apps such as UDS Tools Blocks, Bronze Valves 3D, and so on from this page. You can also publish your own Autodesk products for other users of Autodesk 360.

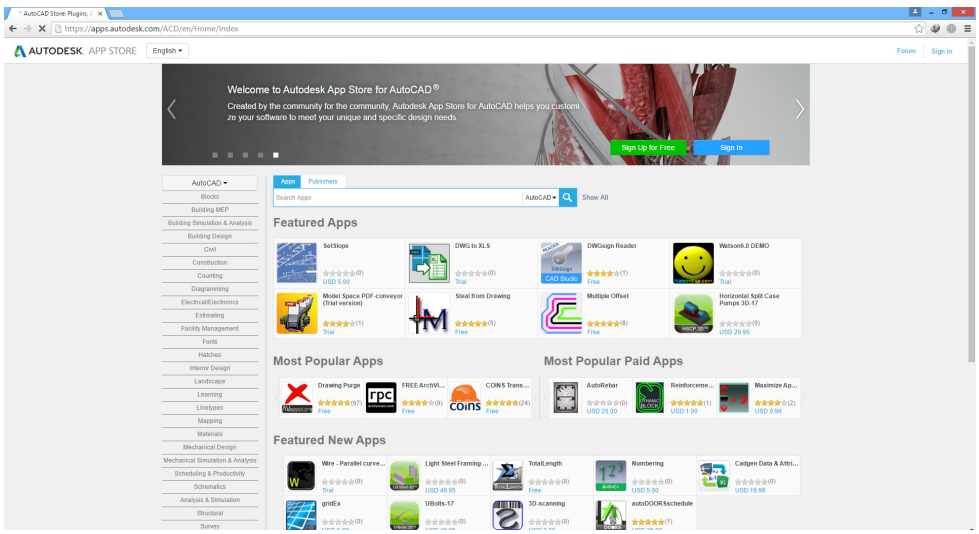

Figure 1-59 *The* ***AUTODESK APP STORE*** *window*

In addition to AutoCAD apps, you can also download apps for software other than AutoCAD such as Autodesk Alias, Revit, Simulation, and so on. You can also search for the apps by entering the name of the app in the **Search Exchange Apps** text box.

After downloading and installing the apps, the apps will be available in the **Plug-ins** tab. You can choose the required app to work with. Also, you can manage the apps by choosing the **Exchange App Manager** button available in the **App Manager** panel of the **Plug-ins** tab. To manage the app, choose the **Exchange App Manager** button; the **Autodesk Exchange App Manager** dialog box will be displayed. In the dialog box, all the installed apps are displayed. To manage a particular app, right-click on it; a shortcut menu will be displayed, as shown in Figure 1-60. Now, you can choose the required option to manage it.

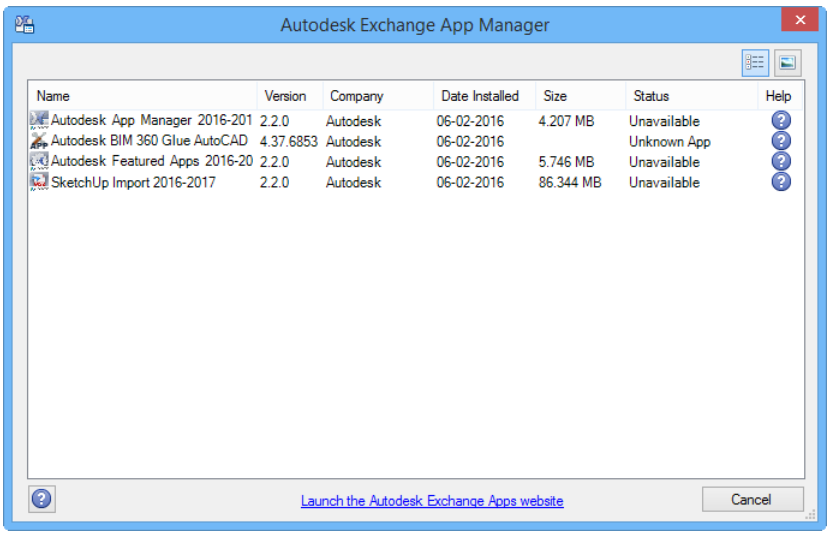

Figure 1-60 *The* ***Autodesk Exchange App Manager*** *dialog box*

Design Feed

In AutoCAD, the **DESIGN FEED** palette is used to share a document with the people who have Autodesk 360 account. It can be invoked by entering **DESIGNFEEDOPEN** in the Command bar. The **DESIGN FEED** palette is shown in Figure 1-61. In this palette, click on the **Login** link in the palette; the **Autodesk Sign-In** window will be displayed. Enter your Autodesk ID and password to sign in. Next, click the **Invite People** link to display the **Design Feed - Invite People** dialog box as shown in Figure 1-62. In this dialog box specify the mail address with you want to share the file. Also, the drawing file will be saved in **A360**. The other options of the **DESIGN FEED** palette are discussed next.

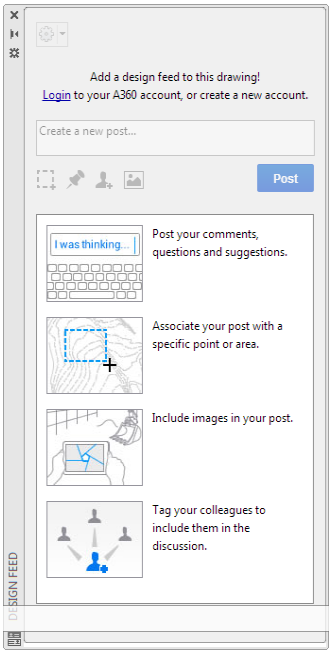

*Figure 1-61 The **DESIGN FEED** palette*

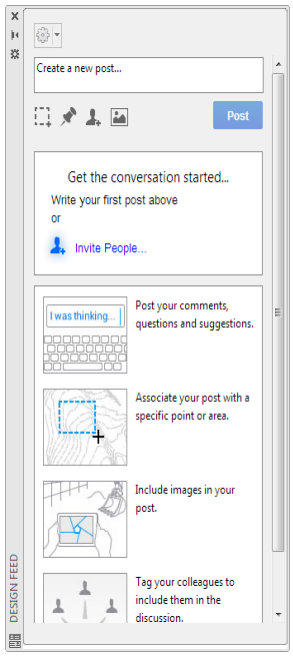

*Figure 1-62 The **DESIGN FEED** palette with all options activated*

Create a new post Text Box

This text box is used to enter a message for the post. You can enter any query related to any specific area or point in a drawing by using this text box and then share it. Note that everyone can view your post but only the Autodesk A360 registered users can reply to your post.

Associate this post to an area in the drawing

This button is used to associate the post entered in the **Create a new post** text box to the desired area in a drawing. To post a message/query associated with a specific area in the drawing, choose this button and then specify the required area by using the cross selection method; a blue dashed rectangle enclosing the selected portion along with the design feed bubble is displayed in the drawing area, refer to Figure 1-63. Once the area has been specified for the post, you need to post it so that other users of Autodesk A360 account working on the same drawing, can view and reply to your post. To post the text, choose the **Post** button from the **DESIGN FEED** palette. Figure 1-64 shows a drawing with the added post. Note that in the drawing area, every post is represented with its serial number assigned in the **DESIGN FEED** palette.

Associate this post to a point in the drawing

This button is used to associate the post entered in the **Create a new post** text box to the desired point in the drawing. To do so, choose this button and then specify the point by clicking the left mouse button in the drawing area; the post will be assigned to the specified point and a design feed bubble will be displayed in the drawing area. Once the point has been specified for the post, you need to publish the post so that other users of Autodesk A360 account working on the same drawing can get updated with the post. To publish the post, choose the **Post** button from the **DESIGN FEED** palette. Note that in the drawing area, every post is represented with its serial number assigned in the **DESIGN FEED** palette.

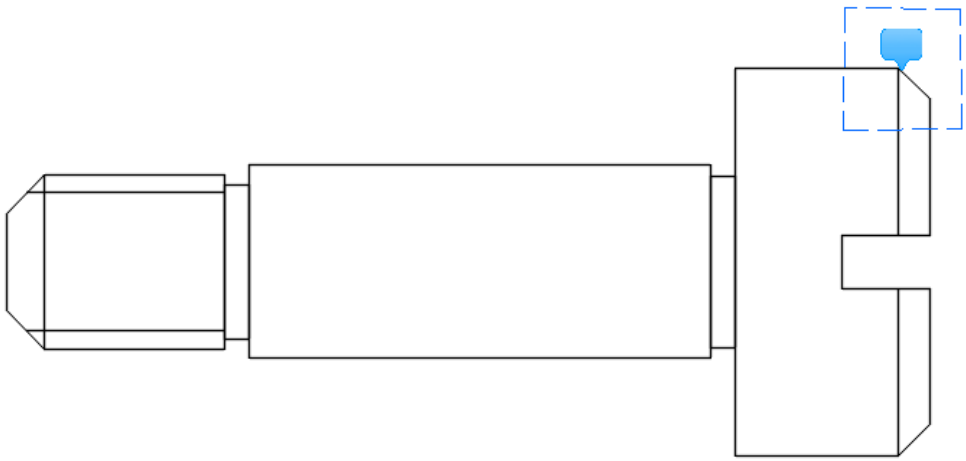

Figure 1-63 *Drawing with enclosed area and design feed bubble*

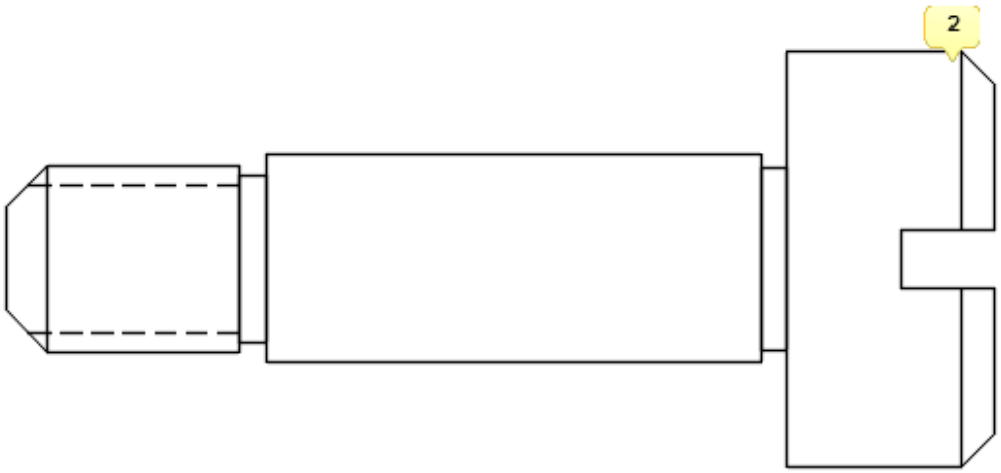

Figure 1-64 *Post with the number assigned in the* ***Design Feed*** *palette*

Tag in this post

This button is used to tag the people with whom you want to share the post through Autodesk A360. To add people to the sharing list, you need to choose this button before posting the message. On doing so, the **Invite People** button will be displayed in the **DESIGN FEED** palette. If you already have the connections with whom you want to share the post, select the check boxes adjacent to their E-mail ids from the **DESIGN FEED** palette and then choose the **Post** button.

You can also share the post with the new connections that are not listed in the **DESIGN FEED** palette by choosing the **Invite People** button. To do so, choose the **Invite People** button; the **Design Feed - Invite People** dialog box will be displayed, refer to Figure 1-65. To add people in sharing list, enter their E-mail ids in the **To** text box and then choose the **Invite** button adjacent to the text box; the E-mail ids will be added in your connection and will be displayed in the **DESIGN FEED** palette. Choose the **Post** button to publish the post.

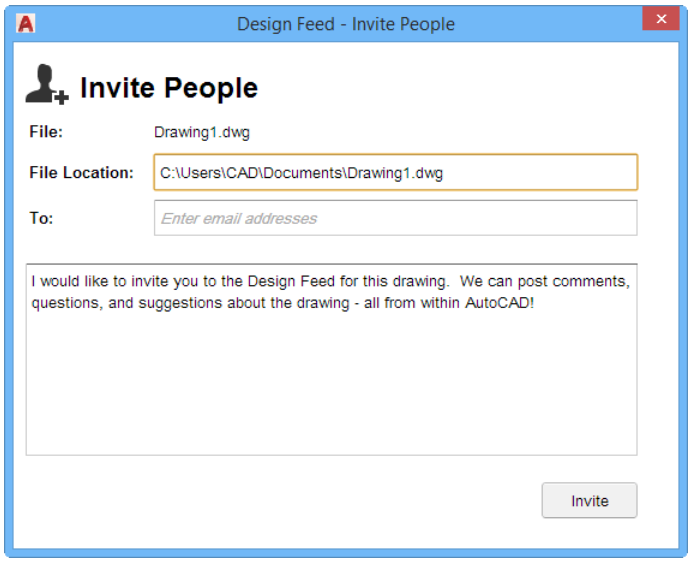

Figure 1-65 *The* ***Design Feed - Invite People*** *dialog box*

Attach image(s) to this post

This button is used to attach images to your posts. Using this button, you can attach different image file formats such as *BMP*, *DIB*, *JPEG*, *JPG*, *JPE*, *JFIF*, *JIF*, *GIF*, *TIF*, *TIFF*, and *PNG* to your post. To attach an image with the post, choose the **Attach image(s) to this post** button; the **Select File** dialog box will be displayed. Browse to the desired file, select it and then choose the **Open** button from the dialog box; the selected image file will be attached. Now, you can publish your post with the attached image file.

Design Feed Settings

The options in this drop-down list are used to display and hide the design feed bubbles and resolved posts from the drawing. These options are discussed next.

Hide All

This option is used to hide all design feed bubbles and posts attached in the drawing area.

Show Bubbles

This option is used to display the design feed bubbles in the drawing.

Note

When you click on the bubble, the post will be displayed along with the blue colored dashed rectangle in the drawing area. Note that if you press the ESC key, the post gets disappeared but the bubble remains available.

Show Extended bubbles

This option is used to display the post with the preview of the message in the drawing area along with its bubble. Note that if you press the ESC key, the preview of the post remains available in the drawing area.

Show Resolved

This check box is used to toggle the display of resolved posts and their respective design feed bubbles in the drawing area.

BIM 360

In AutoCAD, **BIM 360** tab is available in the **Ribbon**, as shown in Figure 1-66. This tab provides tools for sharing models amongst the users of **Autodesk BIM 360 Glue** online service and also allows to view the clash results in the models.

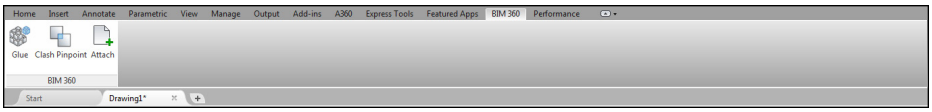

Figure 1-66 *The **BIM 360** tab in the **Ribbon***

ADDITIONAL HELP RESOURCES

1. You can get help for a command while working by pressing the F1 key. The help window containing information about the command is displayed. You can exit the window and continue with the command.

2. You can get help about a dialog box by choosing the **Help** button in that dialog box.

3. Autodesk has provided several resources that you can use to get assistance with your AutoCAD questions. The following is a list of some of the resources:

   a. Autodesk website *http://www.autodesk.com*
   b. AutoCAD technical assistance website *http://knowledge.autodesk.com*
   c. AutoCAD discussion groups website *http://forums.autodesk.com*

4. You can also get help by contacting us at *techsupport@cadcim.com* and *sales@cadcim.com*.

5. You can download AutoCAD drawings, programs, and special topics by registering yourself at the faculty's website by visiting: *http://cadcim.com/Registration.aspx*

Self-Evaluation Test

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

1. The _____ button is used to set the performance of the software at an acceptable level.

2. If the _____ variable is set to 1 and you invoke the **New** tool, the **Create New Drawing** dialog box will be displayed.

3. If you want to work on a drawing without altering the original drawing, you must select the _____ option from the **Open** drop-down list in the **Select File** dialog box.

4. The _____ option enables you to open only a selected view or a selected layer of the current drawing.

5. You can use the _____ command to close the current drawing file without actually quitting AutoCAD.

6. The _____ system variable can be used to change the time interval for automatic save.

7. By using the _____ button, you can find information about a command on internet.

8. You can enable or disable the functions such as AutoComplete, AutoCorrect, and so on by using the options available in the _____ dialog box.

9. The _____ button is used to toggle the display of the File tab bar which displays all opened files.

10. The _____ palette is used to share a document with the users who have Autodesk 360 account.

11. You can press the F3 key to display the **AutoCAD** text window, which displays the previous commands and prompts. (T/F)

12. If you do not have internet connection, you cannot access the Help files. (T/F)

13. If a drawing was partially opened and saved previously, it is not possible to open it again with the same layers and views. (T/F)

14. If the current drawing is unnamed and you save the drawing for the first time, the **Save** tool will prompt you to enter the file name in the **Save Drawing As** dialog box. (T/F)

Review Questions

Answer the following questions:

1. Which of the following combination of keys should be pressed to hide all toolbars displayed on the screen?

(a) CTRL+3 (b) CTRL+0
(c) CTRL+5 (d) CTRL+2

2. Which of the following combination of keys should be pressed to turn on or off the display of the **Tool Palettes** window?

(a) CTRL+3 (b) CTRL+0
(c) CTRL+5 (d) CTRL+2

3. Which of the following commands is used to exit the AutoCAD program?

(a) **QUIT** (b) **END**
(c) **CLOSE** (d) None of these

4. Which of the following options in the **Startup** dialog box is used to set the initial drawing settings before actually starting a new drawing?

   (a) **Start from Scratch** (b) **Use a Template**
   (c) **Use a Wizard** (d) None of these

5. When you choose **Save** from the **File** menu or choose the **Save** tool from the **Quick Access** Toolbar, which of the following commands is invoked?

   (a) **SAVE** (b) **LSAVE**
   (c) **QSAVE** (d) **SAVEAS**

6. AutoCAD has provided _____ as an easy and convenient way of placing and sharing hatch patterns and blocks in the current drawing.

7. By default, the angles are positive if measured in the _____ direction.

8. The _____ check box is used to toggle the display of resolved posts.

9. The shortcut menu invoked by right-clicking in the command window displays the most recently used commands and some of the window options such as **Copy**, **Paste**, and so on. (T/F)

10. You can open an AutoCAD 2002 drawing in AutoCAD 2017. (T/F)

11. The file name that you enter to save a drawing in the **Save Drawing As** dialog box can be 255 characters long, but cannot contain spaces and punctuation marks. (T/F)

12. You can close a drawing in AutoCAD even if a command is active. (T/F)

Answers to Self-Evaluation Test

1. Hardware Acceleration, **2. STARTUP**, **3. Open Read-Only**, **4. Partial Open**, **5. CLOSE**, **6. SAVETIME**, **7. Search on Internet**, **8. Input Search Options**, **9. File Tabs**, **10. Design Feed**, **11.** F, **12.** F, **13.** F, **14.** T