Chapter 2

Getting Started with a Structural Project

Learning Objectives

After completing this chapter, you will be able to:

- *Start a new structural project*
- *Set the units of various measurement parameters for a project*
- *Understand the concept of snaps, dimensions, and object snaps*
- *Save a project*
- *Close a project and exit Revit 2019*
- *Open an existing project*

INTRODUCTION

In this chapter, you will learn how to create a new structural project. Also, you will learn the methods of creating initial setups such as units and snaps.

STARTING A NEW STRUCTURAL PROJECT

File Menu: New > Project
Shortcut Key: CTRL+N

In Revit, a structural project is considered as a single database containing information related to a building design. This means that the project file created will contain all the information related to building design, from geometry to construction documentation. Generally, this information include the details of the components used to design building models (like walls, doors, windows, beams, columns, and other relevant information about the building design), different views of the model, working drawings created from model, and documentation related to design of the model.

In Revit, you can create different views such as plan, elevation, and sections from a single model created for a building project. These views are associated with each other. This means, when you change the building design in one view, it is propagated throughout the project.

In Revit, a new project file uses the default settings that are defined in the project template.

To start a new project, choose the **New > Project** from the **File** menu, as shown in Figure 2-1; the **New Project** dialog box will be displayed, refer to Figure 2-2. In this dialog box, the **Construction Template** is selected by default in the **Template file** drop-down list. As a result, the new project will adopt the settings of the template file. A template file has various project parameters saved in it such as units, views, and so on. When you apply the template file on a new project, it will adopt the same parameters as the template file. The difference between a template file format and a project file format is that the former has a *.rte* extension, whereas the latter has a *.rvt* extension. You can either select any of the template files provided in Revit or create your own template file.

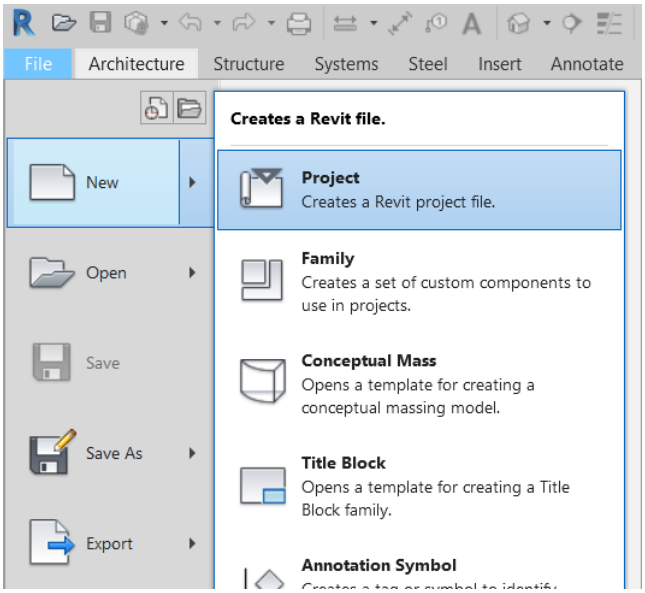

Figure 2-1 *Choosing the* ***Project*** *option from* ***File*** *Menu*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

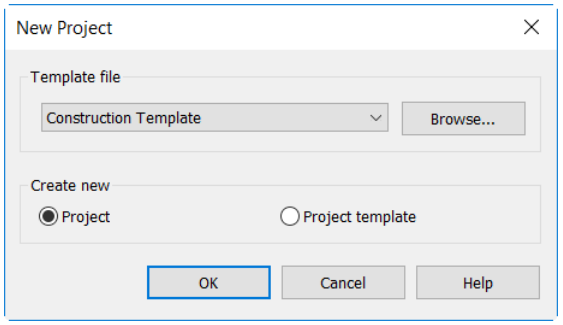

Figure 2-2 *The **New Project** dialog box*

You can also save any project file as a template file. You can select a file as template by first choosing the **Browse** button in the **Template file** area of the **New Project** dialog box and then selecting the required file from appropriate location. To create a new project, select the **Project** radio button and to create a template, select the **Project template** radio button in the **New Project** dialog box.

SETTING PROJECT UNITS

Ribbon: Manage > Settings > Project Units
Shortcut Key: UN



Units are important parameters of a project as they provide a standard for measurement of different entities. While installing Revit, you are prompted to set the Imperial (feet and inches) or Metric (meter) unit as default. Setting a default unit system helps you to start your project with a specific type of unit as per the conventions followed in the industry. To set units, choose the **Project Units** tool from the **Settings** panel of the **Manage** tab; the **Project Units** dialog box will be displayed, as shown in Figure 2-3. Alternatively, type **UN** to display the **Project Units** dialog box. Under the **Units** column in this dialog box, you can specify various units that are relevant to the building project. In the **Project Units** dialog box, units are grouped into six disciplines: **Common**, **Structural**, **HVAC**, **Electrical**, **Piping**, and **Energy**. Each discipline has a set of measurement parameters. You can select any of these disciplines from the **Discipline** drop-down list in the **Project Units** dialog box. The **Format** column in this dialog box displays the current unit format for the corresponding parameter in the **Units** column. You can preview and select the possible digit grouping and decimal separators from the **Decimal symbol/digit grouping** drop-down list, which is at the lower left corner of the dialog box. Different types of units and their parameters are discussed next.



Note

*The values displayed in the **Format** column of the **Project Units** dialog box may differ, depending upon the type of unit system, Imperial or Metric, selected for the project. In this textbook, the Imperial and Metric unit systems have been used in tutorials and illustrations.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

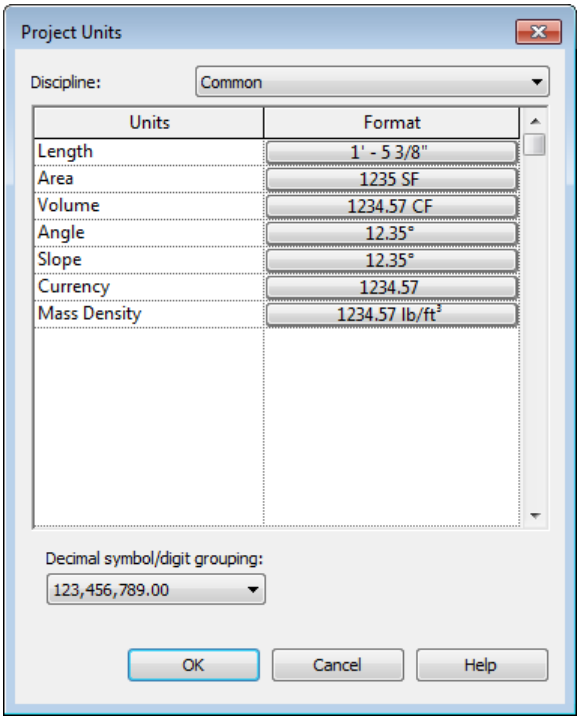

Figure 2-3 *The* ***Project Units*** *dialog box*

Common Unit Type

The **Common** unit type used in a structural project includes the parameters such as length, volume, area, angle, slope, mass density, and currency. In the **Project Units** dialog box, the **Common** option is selected by default in the **Discipline** drop-down list, refer to Figure 2-3. The **Common** unit type used in Revit is similar to those used in other Revit platforms. Moreover, the settings of the parameters of common units are similar to those used in other CAD programs. The methods of setting various parameters under the **Common** unit type are discussed next.

Setting Length Units

In a structural project, you can assign a unit for the measurement of length. To do so, click on the field corresponding to the **Length** parameter in the **Format** column of the **Project Units** dialog box; the **Format** dialog box will be displayed. This dialog box displays the units of length and their settings, as shown in Figure 2-4. Select the required unit from the **Units** drop-down list in the dialog box. The options that can be selected from this drop-down list are **Decimal feet**, **Feet and fractional inches**, **Decimal inches**, **Fractional inches**, **Meters**, **Decimeters**, **Centimeters**, **Millimeters**, and **Meters and centimeters**. By default, the **Feet and fractional inches** option is selected in this drop-down list if the **Imperial** is selected as the unit system at the time of installing Revit. But, if you have selected **Metric** as the unit system then the **Millimeters** option is selected in this drop-down list by default. The settings for rounding increment for the length unit can be made from their respective drop-down lists and edit boxes. To do so, select the desired option from the **Rounding** drop-down list. For certain units, you can specify custom rounding value using the **Rounding increment** edit box, which is placed next to the **Rounding** drop-down list. This edit box is inactive by default. To make it active, first select

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

any option from the **Units** drop-down list and then the **Custom** option from the **Rounding** drop-down list. Once the **Rounding increment** edit box is active, you can specify the desired rounding value in it for the selected unit.

Note

*The **Rounding increment** edit box is not available for the **Meters and Centimeters**, **Feet and fractional inches**, and **Fractional inches** options.*

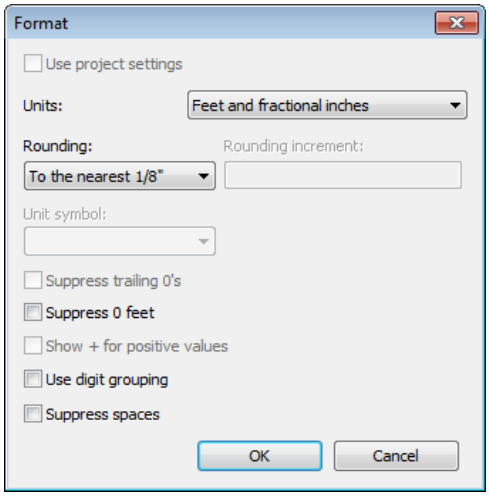

***Figure 2-4** The **Format** dialog box*

The **Unit symbol** drop-down list will be inactive if any of the **Feet and fractional inches**, **Fractional inches**, and **Meters and centimeters** options is selected in the **Units** drop-down list.

Once you make the **Unit symbol** drop-down list active, you can select the desired option from it to specify the measurement symbol to be used along with the unit of length in a project. For example, to use the symbol 'm' after all metric length measurements, you can select **m** option from the **Unit symbol** drop-down list as the measurement symbol. You can select the **Suppress spaces** check box to remove all spaces around the dash from the length strings. For example, you can remove spaces around the dash when a length string is expressed in feet and fractional inches to denote a particular measurement.

Note

*The **Suppress spaces** is only active when **Feet and Fractional inches** is selected in **units** drop down.*

Setting the Area and Volume Units

In the **Project Units** dialog box, you can assign a unit for the measurements of areas. To do so, click on the field corresponding to the **Area** parameter in the Format column of the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, you can set the unit for measuring the area by using the options in the **Units** drop-down list. This drop-down list contains various options for the units of area such as **Square feet**, **Square meters**, **Square inches**, **Square centimeters**, **Square millimeters**, **Acres**, and so on. By default, the **Square feet** option is selected in this drop-down list, if you had selected the **Imperial** units at the time of

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

installing Revit. But, if you have selected the **Metric** units then the **Square meters** option is selected in this drop-down list by default. The settings for rounding increment and unit symbol for the area units can be made from their respective drop-down lists and edit box.

Similar to set the units for the area, you can set the units for volume. To set unit for the volume measurement, click in the field of the **Format** column corresponding to the **Volume** parameter in the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, click in the **Units** drop-down list and select: **Cubic yards**, **Cubic feet**, **Cubic meters**, **Cubic millimeters**, or **Liters** and so on. After selecting a suitable option from the **Units** drop-down list, choose the **OK** button; the **Format** dialog box will be closed and the selected unit for the volume measurement will be displayed in the field of the **Format** column corresponding to the **Volume** parameter in the **Project Units** dialog box.

Setting Angle Units

In the **Project Units** dialog box, you can assign a unit for the measurement of angle. To do so, click on the field corresponding to angle parameter in **Format** column of the **Project Units** dialog box. The **Format** dialog box will be displayed. You can specify the unit for the angle by selecting the required option from the **Units** drop-down list in the **Format** dialog box for the **Angle** parameter. You can select **Decimal degrees**, **Degrees minutes seconds, Radians**, or **Grads** from the **Units** drop-down list to specify the unit for the angular measurement. By default, the **Decimal degrees** option is selected in this drop-down list, if Imperial is selected as a unit system at the time of installing Revit. Similarly, **12.35°** option is selected for Metric unit. The settings for rounding increment for the angle unit can be made from their respective drop-down list and edit box.

Setting Slope Units

In the **Project Units** dialog box, you can assign a unit for the measurement of slope. To do so, click on the field corresponding to angle parameter in **Format** column of the **Project Units** dialog box. The **Format** dialog box will be displayed. You can specify the unit for the slope by selecting the required option from the **Units** drop-down list in the **Format** dialog box for the **Slope** parameter. You can select options such as **1 : Ratio, Ratio : 12**, **Ratio : 10**, **Rise / 12"**, **Rise / 1'- 0"**, **Rise / 1000 mm**, **Decimal degrees**, or **Percentage** for the slope measurement from the **Units** drop-down list to specify the unit for the slope measurement. By default, the **Decimal degrees** option is selected in this drop-down list as the unit system while installing Revit. The settings for rounding increment for the slope unit can be made from their respective drop-down list and edit box.

Setting the Currency Units

In Revit, you can set the unit for the currency as well. To do so, click on the corresponding field for the **Currency** parameter in the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, select appropriate currency symbols and rounding values from the **Unit symbol** and **Rounding** drop-down lists, respectively.

Setting the Mass Density Units

In Revit, you can set the unit for mass density. To do so, click on the corresponding field for the **Mass Density** parameter in the Format column in the **Project Units** dialog box; the **Format**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

dialog box will be displayed. In this dialog box, you can specify the desired unit by selecting it from the **Units** drop-down list. The **Units** drop-down list contains options such as **Kilograms per cubic meter** and **Pound per cubic foot** for mass density measurement. By default, the **Pound per cubic foot** option is selected in this drop-down list for the **Imperial** unit system and **Kilograms per cubic meter** option is selected for the **Metric** unit system. The settings for the parameters of the rounding, rounding increment, and units symbol can be set by selecting the required option from the respective drop-down lists.



Note

*You can format only the display of units on the screen or for printout using the **Project Units** dialog box. The actual values for these units in the project may be different. For example, if you set the wall length rounding to the nearest value 1', the wall may show this rounded value, but the actual length of the wall might be in fractional feet. Similarly, for Metric, if you set the wall length rounding to the nearest value 10, the wall may show this rounded value, but the actual length of the wall might be different.*

Structural Unit Type

Structural units include the settings such as Force, Moment, Stress, Unit Weight, and so on. In Revit, you can set the structural units in the **Project Units** dialog box. To do so, invoke the **Project Units** dialog box and then select the **Structural** option from the **Discipline** drop-down list. Figure 2-5 displays the **Project Units** dialog box with the **Structural** option selected in the **Discipline** drop-down list. Some of the structural units are discussed next.

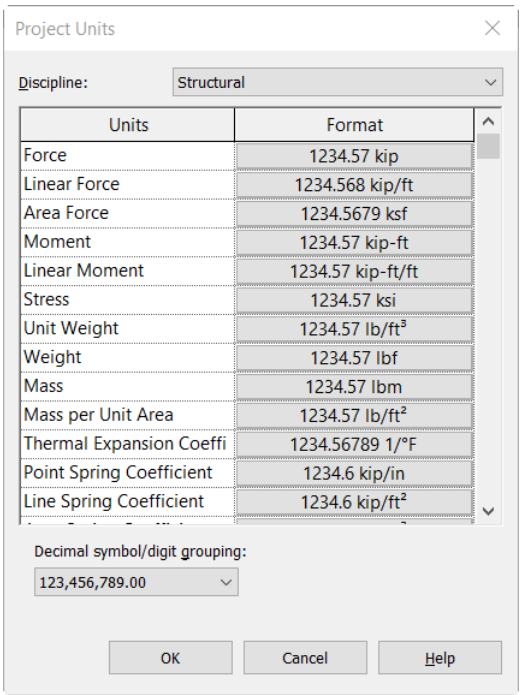

Figure 2-5 *The **Project Units** dialog box with the **Structural** option selected from the **Discipline** drop-down list*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Setting the Unit for Force

To specify the unit for the concentrated forces that act on the structural model, click on the field corresponding to the **Force** parameter in the **Format** column; the **Format** dialog box will be displayed. In this dialog box, specify a unit by selecting the required option from the **Units** drop-down list. The **Units** drop-down list contains the options such as **Newtons**, **Decanewtons**, **Kilonewtons**, **Meganewtons**, **Kips**, **Kilograms force**, and so on. For the Imperial unit system, **Kips** is the default option selected in the drop-down list. For the Metric unit system, **Kilonewtons** is the default option selected.

Setting the Unit for Linear Force

The linear force implies a uniformly distributed load that you can apply to any structural member. To specify unit for the linear force, click on the field corresponding to the **Linear Force** parameter in the **Format** column of the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, specify the desired unit for the linear force by selecting an option from the **Units** drop-down list. The **Units** drop-down list contains options such as **Newtons per meter**, **Decanewtons per meter, Kilonewtons per meter**, **Meganewtons per meter**, **Kips per foot**, **Kilograms force per meter**, and so on. The default unit selected for the **Imperial** unit system in is **Kips per foot**. The default unit selected for the **Metric** unit system in the drop-down list is **Kilonewtons per meter**.



Tip

*While selecting a rounding value from the **Rounding** drop-down list in the **Format** dialog box, you should consider the extent of detailing that may be required for the project, and then select it. For projects that require too much detailing, a lower rounding value may be set. This parameter, however, can be modified at any time during the project development.*

Setting the Unit for Area Force

Area forces denote the pressure forces that you can apply to any structural element (beams, columns, slabs, and so on). To specify unit for the area force, click on the field corresponding to the **Area Force** parameter in the **Format** column of the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, specify the desired unit for the area force by selecting an option from the **Units** drop-down list. In the **Units** drop-down list, you can select various options such as **Newtons per square meter**, **Decanewtons per square meter**, **Kilonewtons per square meter**, **Meganewtons per square meter**, **Kips per square foot**, **Kilograms force per square meter**, and so on. The default unit selected for the Imperial unit system in the drop-down list is **Kips per square foot** and for the Metric unit system is **Kilonewtons per square meter.**

Setting the Unit for Mass per Unit Area

The **Mass per Unit Area** parameter specifies the density that you can apply on any structural elements such as rebars, structural steel, and so on. To specify a unit for the mass per unit area, click on the field corresponding to the **Mass per Unit Area** parameter in the **Format** column of the **Project Units** dialog box; the **Format** dialog box will be displayed. In this dialog box, specify the desired unit for the mass per unit area by selecting an option from the **Units** drop-down list. In the **Units** drop-down list, you can select options such as **Kilograms per square meter** and

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Pounds per square foot. The default unit selected in the Imperial unit system for this parameter is **Pound per square foot** and in the Metric unit system is **Kilograms per square meter**.

Similarly, you can set the units for the other options of different discipline types.

STRUCTURAL SETTINGS

Ribbon: Manage > Settings > Structural Settings

In Autodesk Revit, you can specify various structural settings of a project such as settings of the symbolic representation of structural framing components, settings of various load cases to be applied to structural model, settings for the analysis to be performed on model, and settings for providing supports to structural model. You can specify all these settings by using the **Structural Settings** dialog box. The **Structural Settings** dialog box can be invoked by choosing the **Structural Settings** tool from the **Settings** panel of the **Manage** tab, as shown in Figure 2-6. This dialog box contains five tabs, namely **Symbolic Representation Settings**, **Load Cases**, **Load Combinations**, **Analytical Model Settings**, and **Boundary Conditions Settings**. In this dialog box, the **Symbolic Representation Settings** tab is chosen by default, refer to Figure 2-7.

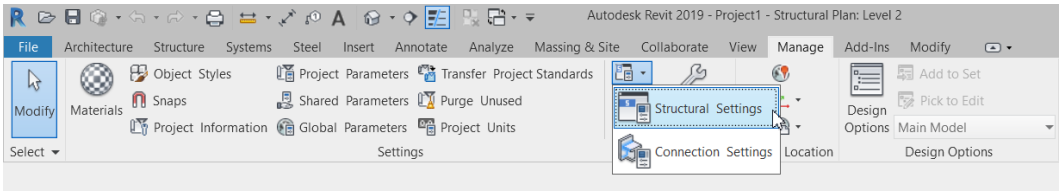

Figure 2-6 *Choosing the* ***Structural Settings*** *tool*

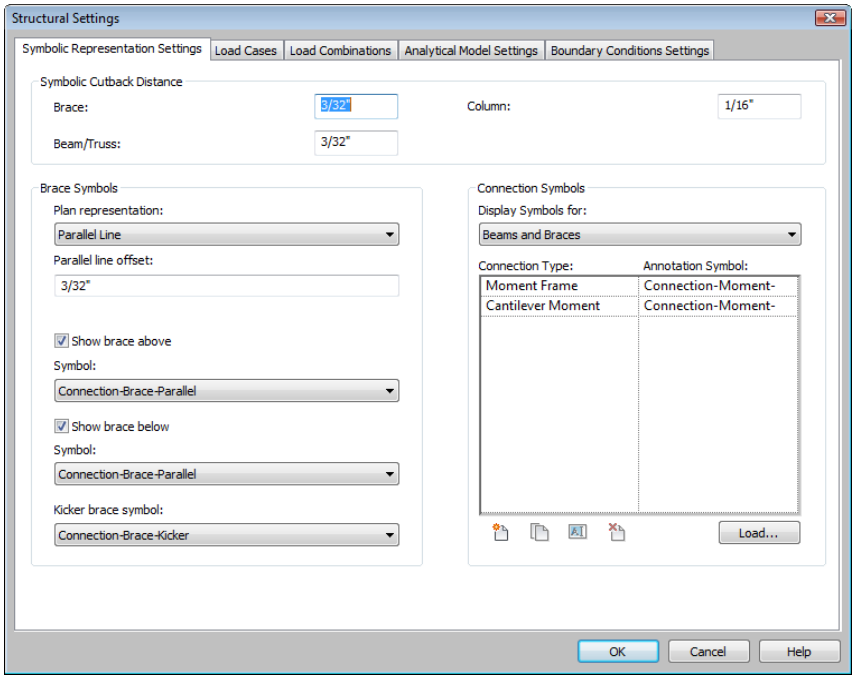

Figure 2-7 *The* ***Symbolic Representation Settings*** *tab chosen in the* ***Structural Settings*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

The settings that you specify in the tabs of the **Structural Settings** dialog box pertain to the appearance of the structural members. In the **Structural Settings** dialog box, you can also specify the load cases, boundary conditions, and the settings for the analytical model. In this chapter, you will learn about various options in the **Symbolic Representation Settings** tab. This will help you adjust the symbolic representation of various structural elements for documentation output.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Symbolic Representation Settings Tab

Whenever you invoke the **Structural Settings** dialog box, the **Symbolic Representation Settings** tab is chosen by default, refer to Figure 2-7. The options in this tab control the display and symbology of different structural conditions in plans, sections, and schedules of steel shapes. These options determine how certain elements of an object will display or allow you to use a symbol of your choice in different conditions. In this tab, there are three areas: **Symbolic Cutback Distance**, **Brace Symbols**, and **Connection Symbols**. The options in these areas are discussed next.

Symbolic Cutback Distance Area

The **Symbolic Cutback Distance** area, refer to Figure 2-7, contains three edit boxes: **Brace**, **Beam/Truss**, and **Column**. The values in these edit boxes control the cutback distances of the structural symbolic lines that represent beams, columns, and braces at a coarse detailed level of a structural project. These values are global settings that affect all beams, trusses, braces, and steel columns in a project.

In the **Symbolic Cutback Distance** area, you can enter a numeric value in the **Brace** edit box to control the symbolic cutback distance between a structural steel beam or column and a structural steel brace at a coarse detail level. In the Imperial unit system, the default value in this edit box is 3/32" whereas in Metric unit system, the default value in the edit box is 2.5000 mm. Figure 2-8 shows the symbolic cutback distance between a brace and a column at a coarse detail level.

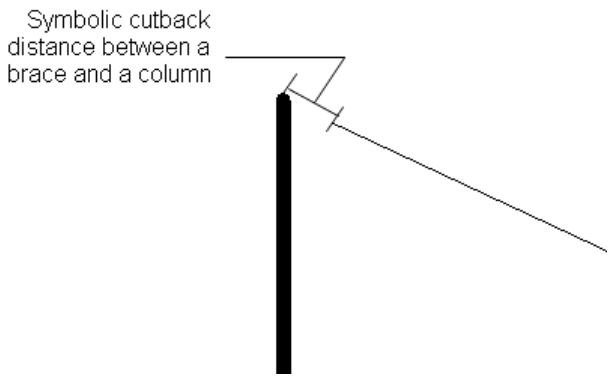

Figure 2-8 *The symbolic cutback distance between a column and brace*

In the **Beam/Truss** edit box of the **Symbolic Cutback Distance** area, you can enter any numeric value to control the symbolic cutback distance between beams and columns, or between beams, trusses, and columns. In the Imperial unit system, the default value in this edit box is 3/32"

whereas in Metric unit system, the default value in the edit box is 2.5000mm. Figure 2-9 shows the symbolic cutback distance between two beams and a column.

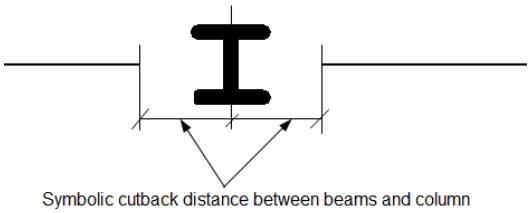

Figure 2-9 *The symbolic cutback distance between beams and column at coarse detail level*

The value in the **Column** edit box is used to increase or decrease the gap between the symbolic representations of joined columns in an elevation view. This gap will be visible in a coarse detail elevation view only if the **Top Connection Symbol** instance parameter of the bottom column is set to a value other than **None**. The default value in the **Column** edit box is 1/16" for Imperial unit system and in Metric system, the default value is 1.5000mm. Figure 2-10 shows the symbolic cutback distance between columns in an elevation view.

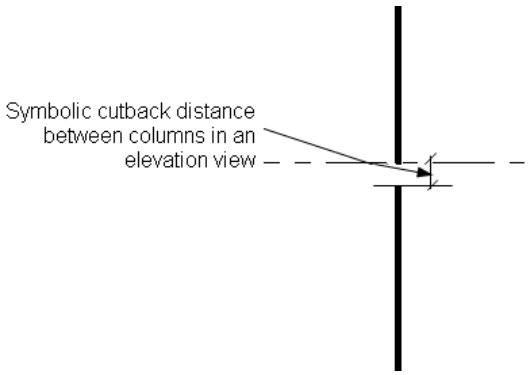

Figure 2-10 *The symbolic cutback distance between columns in an elevation view at a coarse detail level*

The Brace Symbols Area

The settings in the **Brace Symbols** area are used to control the symbolic display of bracing and kicker bracings that exist in a project. Figure 2-11 shows the settings in the **Brace Symbols** area in the **Symbolic Representation Settings** tab of the **Structural Settings** dialog box. The **Plan representation** drop-down list in this area contains two options: **Parallel Line** and **Line with Angle**. The **Parallel Line** option is selected by default. As a result, the symbolic representation of braces in a plan view is displayed with a line parallel to and offset from the actual brace. You can control the offset distance of this parallel line by entering a suitable value in the **Parallel line offset** edit box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

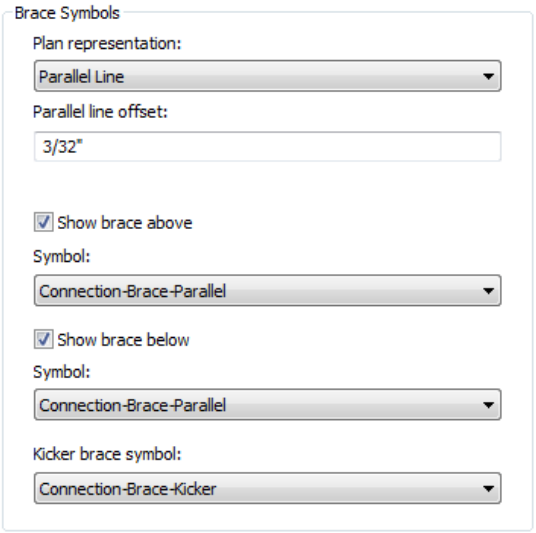

Figure 2-11 *Settings in the* ***Brace Symbols*** *area*

In the Imperial unit system, 3/32" is set as the default value in the **Parallel line offset** edit box and in the Metric unit system, default value is 2.5000 mm. Figure 2-12 shows the symbolic representation of a brace in a plan view with default offset distance. On selecting the **Line with Angle** option from the **Plan representation** drop-down list, the symbolic representations of braces in a plan view are displayed with an angled line. The angled line originates from the point of intersection of a brace and level line, if they intersect, or from the closest point to the level line on the brace. The angled line goes upward from the point of origin for the portion of the brace which is above the level or it goes downward for the portion of the brace which is below the level. Figure 2-13 shows a bracing symbol as an angled line.

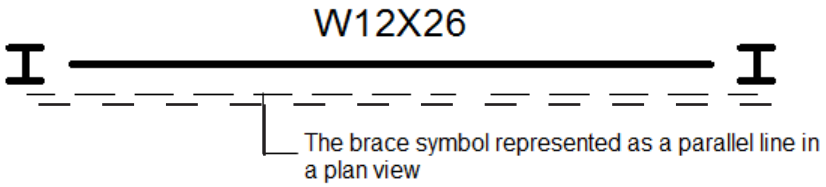

Figure 2-12 *The symbolic representation of a brace in a plan view*

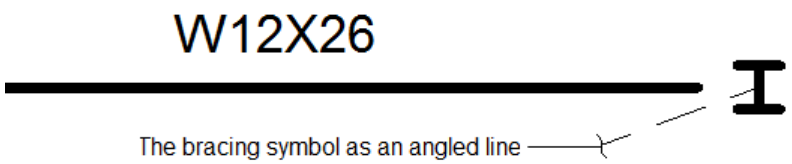

Figure 2-13 *The symbolic representation of a brace as an angled line*

The **Show brace above** check box in the **Brace Symbols** area is selected by default. As a result, the brace symbols representing the braces placed above the current view are displayed. When you select the **Show brace above** check box, the **Symbol** drop-down list below the check box

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

gets activated. From this drop-down list, you can select an option depending upon the option selected from the **Plan representation** drop-down list. You can select the **Connection-Brace -Parallel** option from the **Symbol** drop-down list, if the **Parallel Line** option is selected in the **Plan representation** drop-down list. Similarly, if the **Line with Angle** option is selected in the **Plan representation** drop-down list, you can select the **Connection-Brace-Angle** from the **Symbol** drop-down list. On clearing the **Show brace above** check box, the brace option placed above the current view will not be displayed.

Similar to the **Show brace above** check box, the **Show brace below** check box is selected by default. As a result, the brace symbols representing the braces placed below the current view are displayed. The **Symbol** drop-down list below the **Show brace below** check box becomes active when the **Show brace below** check box is selected. The options in the **Symbol** drop-down list located below the **Show brace below** check box are similar to those of the **Symbol** check box located below the **Show brace above** check box.

In the **Kicker brace symbol** drop-down list, the **Connection-Brace-Kicker** option is selected by default. As a result, the symbolic representation of a kicker brace will be a connection-brace in a plan view.



Note

*A normal bracing can be converted into a kicker bracing, if the **Structural Usage** instance parameter in the **Properties** palette of the bracing is set to **Kicker Bracing**, refer to Figure 2-14.*

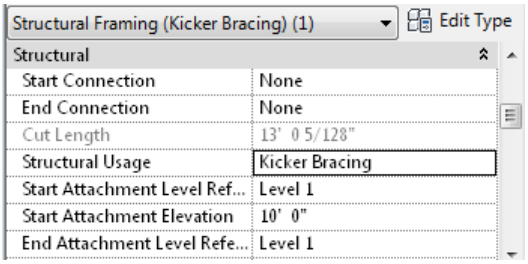

Figure 2-14** The Partial view of the **Properties** palette displaying the **Structural Usage** parameter set to **Kicker Bracing

Connection Symbols Area

The options in the **Connection Symbols** area, as shown in Figure 2-15, can be used to control the symbolic display of moment and shear connections for beams, braces, and columns that exist throughout the project. To select the type of framing member for which you want to modify the symbolic display for moment and shear connections, click on the **Display Symbols** drop-down list and then select any of the three options: **Beams and Braces**, **Column Base**, and **Column Top**. In the **Display Symbols** drop-down list, the **Beams and Braces** option is selected by default. As a result, the settings for this option will be displayed in a table located below the **Display Symbols** drop-down list, refer to Figure 2-15. In this table, you will notice two columns: **Connection Type** and **Annotation Symbol**. In the **Connection Type** column, two parameters are displayed: **Moment Frame** and **Cantilever Moment**.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

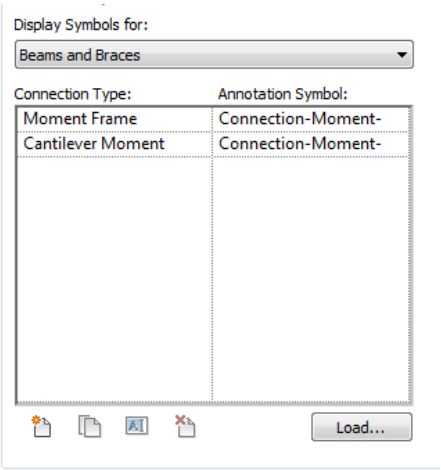

Figure 2-15 *The options in the* ***Connection Symbols*** *area*

The parameters displayed in the **Connection Type** column may vary depending on the option selected from the **Displayed Symbols** drop-down list. The values for these parameters are displayed in their corresponding fields in the **Annotation Symbol** column. To specify the annotation symbol for a beam or brace of **Moment Frame** parameter, click on the **Annotation Symbol** column corresponding to the **Moment Frame** parameter and select any of the three options: **None**, **Connection-Moment-Filled**, and **Connection-Moment-Not Filled**. Similarly, you can specify an option for **Cantilever Moment** parameter. To add a connection type parameter (for beams and braces) in the table, choose the **Add a connection type** button (first button from the left) displayed below the table, refer to Figure 2-15. On doing so, the **New Connection Type** dialog box will be displayed. In the **Connection Type** edit box of the **New Connection Type** dialog box, enter a name for the new connection type. Next, click in the **Annotation Symbol** drop-down list and select an option to assign an annotation symbol for the connection type. Optionally, in the **New Connection Type** dialog box, you can choose the **Load Symbol** button to load a symbol (with file format *.rfa* or *.adsk)* for the new connection type. Now, choose the **OK** button; the new connection type will be added as parameter in the table. Next, rename, delete, and create a duplicate for the connection type displayed in the table.



In the **Connection Symbols** area, you can specify the base plate symbol for the column. To do so, select the **Column Base** option from the **Display Symbols** drop-down list and specify a symbol for the base plate from the table displayed below the **Display Symbols** drop-down list. Similarly, to assign a symbol to the base plate of a column base, you can specify symbols for shear column connection and moment column connection by selecting the **Column Top** option from the **Display Symbols** drop-down list. Figure 2-16 shows a moment connection and cantilever connection symbols for a steel member.

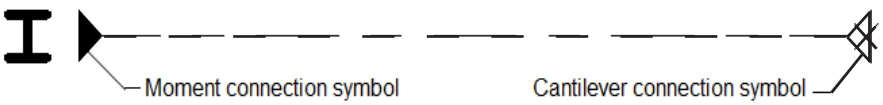

Figure 2-16 *The moment connection and cantilever connection symbols*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

CONNECTION SETTINGS

Ribbon: Manage > Settings > Structural Settings drop-down > Connection Settings

In Autodesk Revit, you can specify various connection settings for structural connections such as the **Beam end to end**, **Column - Beam**, **General bracings**, and so on. You can load different sets of connection in the project by using the **Structural Connection Settings** dialog box. The **Structural Connection Settings** dialog box can be invoked by choosing the **Connection Setting** tool from the **Setting** panel of the **Manage** tab, as shown in Figure 2-17.

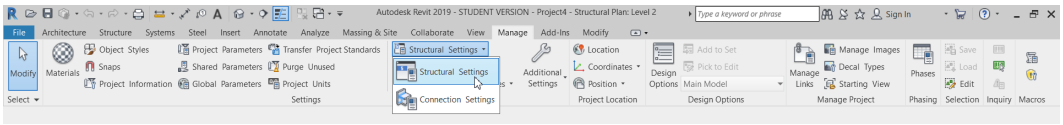

Figure 2-17 *Choosing the* ***Connection Setting*** *tool*

This dialog box contains two tabs, namely **Connections** and **Parameters**. In this dialog box, the **Connections** tab is chosen by default, as shown in Figure 2-18.

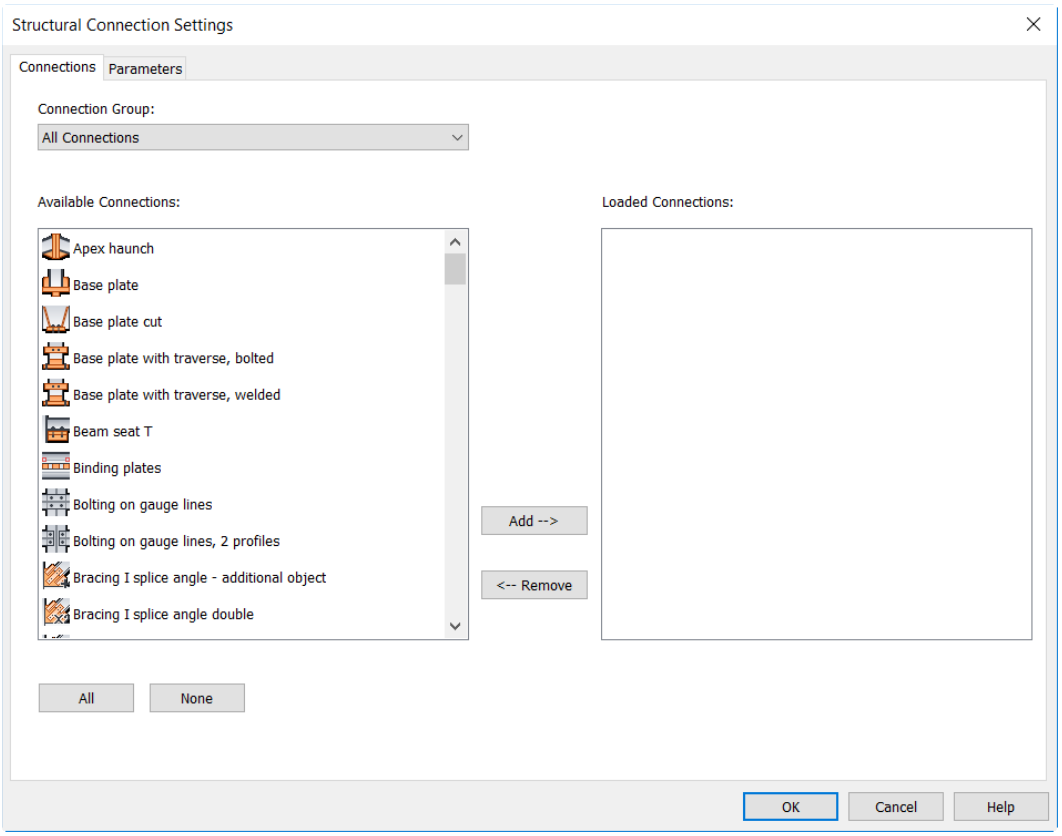

Figure 2-18 *The* ***Structural Connection Settings*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In the **Connections** tab of the **Structural Connection Settings** dialog box, the **All Connections** option is selected by default in the **Connection Group** drop-down. In the **Available Connections** area, some of the possible connections between different structural members are displayed. To load any of the connection types in the project, select the desired connection type from the **Available Connection** area and choose the **Add** button; the selected type of connection is added in the **Loaded Connections** area. To load more than one connection type in the project from the **Available Connections** area, select the connections and hold the **Ctrl** key to select other connection types and choose the **Add** button.

You can select different types of connection groups from the **Connection Group** drop-down. The **Connection Groups** available in the **Available Connections** area depends on the type of connection selected in the **Connection Group** drop-down. Some of the **Connection Groups** available in this drop-down are **Beam end to end**, **Column-Beam**, **General bracings**, and so on.

Once connections are added in the **Loaded Connections** area, choose the **OK** button, the **Structural Connection Settings** dialog box is closed and the type of connections is loaded in the project.

Now, to set a connection between the structural members, invoke the **Connection** tool from the **Connection** panel of the **Structure** tab. Select the type of connection from the **Type Selector** drop-down list in the **Properties** panel and select the structural member between which the connection is to be created. The connection loaded from the **Type Selector** drop-down list is displayed between the member in the 3D view. If the connection type is not displayed in the 3D view, choose the **Visibility/Graphics** option from the **Graphics** panel of the **View** tab; the **Visibility/Graphic Overrides for <View>** will be displayed. Select all the options available in the **Structural Connections** head from the **Visibility** column of the **Model Categories** tab in the displayed dialog box, as shown in Figure 2-19.

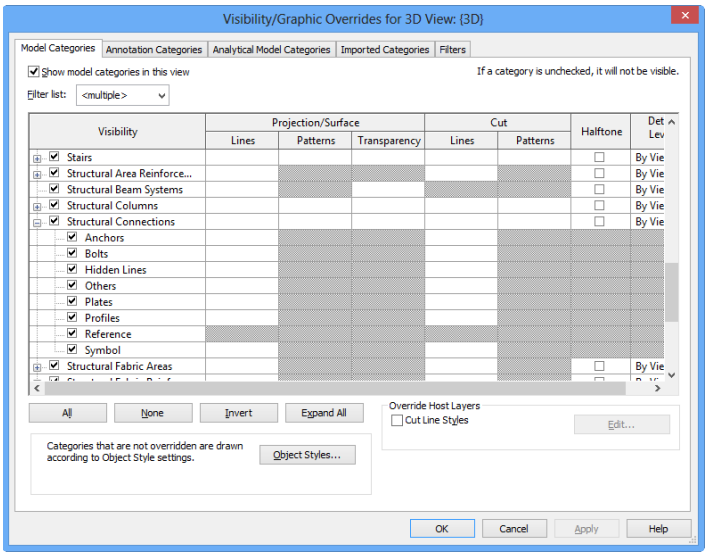

Figure 2-19 *The check box selected from the* ***Structural Connection*** *head*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

SNAPS TOOL

Ribbon: Manage > Settings > Snaps

The **Snaps** tool is one of the important tools used for creating and editing elements in a structural model. This tool is used to make the cursor snap or jump depending on the preset increments or specific object properties of elements such as endpoint, midpoint, and so on. Invoke the **Snaps** tool from the **Settings** panel of the **Manage** tab; the **Snaps** dialog box will be displayed, as shown in Figure 2-20. This dialog box has three areas, **Dimension Snaps**, **Object Snaps**, and **Temporary Overrides**. These areas are discussed next.

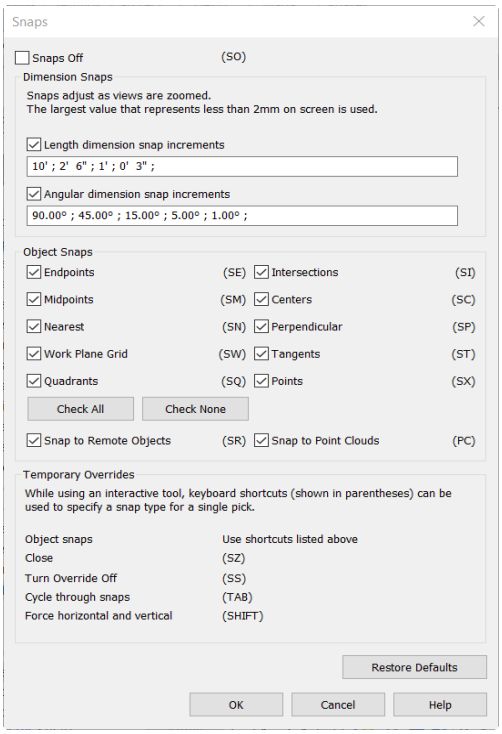

Figure 2-20 *The **Snaps** dialog box*



Note

The settings specified in this dialog box will be applied to all the projects opened in the session and will not be saved.



Tip

*The **Snaps** tool is used frequently not only while creating various building elements but also while editing and placing them. Besides making the modeling simpler, by using this feature efficiently, you can also improve the performance and accuracy of your project.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Dimension Snaps Area

In the **Dimension Snaps** area, you can set the length and angle dimension snap increments for placing various elements and components in a project. The dimension snap determines the increment by which the cursor will jump/snap along with length or angular dimension while placing elements and components in a project.

The **Length dimension snap increments** check box is selected by default. As a result, you can set the snap increment for the length dimension. On selecting the **Length dimension snap increments** check box, you can enter suitable value(s) in the edit box to set the snap value for length. The default values in this edit box are: 4'; 0'6"; 0'1"; 0'0 1/4", if the **US Imperial** template is selected. While, on selecting the **US Metric** template the default values in this edit box are: 1000; 100; 20; 5. Note that every incremental value is separated by a semicolon (;). You can also set increments by typing the values that are separated by a semicolon. For example, to create an interior layout plan in which the length of the partitions is in 5' (1524mm) modules, counter top width is 2' (610mm), and thickness of partitions is 4" (100mm), you can enter the values for the dimension snaps as **5'; 2';4"** (US Imperial) and **1524; 610; 100** (US Metric). This will enable the cursor to move in these increments and help to create the layout with relative ease. Similarly, you can set the angular dimension snap increments. However, ensure that the **Angular dimension snap increments** check box is selected and then enter suitable value(s) in the edit box below it. This parameter is quite useful for the projects that have radial geometry. To restore the default setting, choose the **Restore Defaults** button from the **Snaps** dialog box.



Note

The reason for specifying multiple length and angle snap increment values in the edit boxes is that the priority of increments may change on changing the zoom level of the drawing. Therefore, when you zoom in the drawing, Revit will use smaller increment values, whereas on zooming out, the larger increment values will be used.

Object Snaps Area

Object Snaps refer to the cursor's ability to snap to geometric points on an element such as its endpoint, midpoint, perpendicular, and so on. In the **Object Snaps** area of the **Snaps** dialog box, you can use various options to snap points of elements or objects in a project. The advantage of using these options is that you do not need to specify the exact point in a drawing. When object snaps are enabled, the suitable object snap is displayed as the cursor is moved close to an element. For example, it is virtually impossible to pick the exact endpoint to start a wall from the endpoint of an existing wall. But when you enable the endpoint object snap, the cursor automatically jumps or snaps to the endpoint of this wall. This helps to start the new wall from the endpoint. This, besides making the drawing accurate, later helps add dimensions to the project.



Tip

*The **Dimension Snaps** parameter should be set based on the scale and the amount of detailing required for the project. You may set smaller increments for working on a detailed or small portion of a building.*



Note

The object snaps tool works only with the objects that are visible on the screen. A tooltip, with the same name as the object snap, is also displayed when you bring the cursor close to a snap point.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In a structural model, you can use various **Object Snaps** modes such as **Endpoints**, **Midpoints**, **Nearest**, **Work Plane Grid**, **Quadrants**, **Intersections**, **Centers**, **Perpendicular**, **Tangents**, **Points**, **Snap to Point Clouds**, and **Snap to Remote Objects**.

The name of each object snap option resembles its usage in the project. For example, the **Work Plane Grid** snap option helps you snap the intersection points of grid lines in a work plane grid that is displayed for the current work plane.

Each object snap mode is represented by a geometrical shaped marker to identify it from other object snaps. For example, the endpoint object snap is represented by a square, midpoint by a triangle, nearest by a cross, and so on. To use an object snap mode, move the cursor on the object. As you move it close to the snap point, a marker appears. To select the appropriate snap point, click when the corresponding marker or tooltip is displayed.

In Revit, all enabled object snaps work simultaneously. You can turn off all snap options, including dimension snaps and object snaps by selecting the **Snaps Off** check box located at the top of the **Snaps** dialog box. Alternatively, you can type **SO** on the keyboard to turn them off and on while using a tool. The **Check All** and **Check None** buttons are used to select or clear all check boxes (except the **Snap to Remote Objects** and **Snap to point Clouds**) in the **Object Snaps** area.

Temporary Overrides Area

The options in the **Temporary Overrides** area provide you with an alternative of overriding snaps setting for a single use only. For example, if you have not selected the **Endpoints** check box in the **Snaps** dialog box and you want to use this option while working with a tool, you do not need to open the **Snaps** dialog box to select this option. Instead, you can type the shortcut, **SE** in this case, to temporarily activate the endpoint object snap. Once you have used this object snap option, snapping to the endpoint is automatically turned off.

Using overrides, you can toggle between various object snap options available at the same location. To do so, press the TAB key while snapping the points in the drawing. You can also use other overrides like pressing the SHIFT key to create elements vertically or horizontally. This restricts the movement of the cursor in the orthogonal directions only. Once you release the SHIFT key, the cursor resumes its movement in all directions. You can select the **Snaps Off** check box to disable all types of snapping.

OPTIONS DIALOG BOX



In Autodesk Revit, you can configure global settings by using the **Options** dialog box. This dialog box can be invoked by choosing the **Options** button from the **File Menu**. The **Options** dialog box, as shown in Figure 2-21, contains ten tabs: **General**, **Graphics**, **Hardware, File Locations**, **Rendering**, **Check Spelling**, **SteeringWheels**, **ViewCube**, **User Interface**, and **Macros**. These tabs are discussed next.

General Tab

The **General** tab is chosen by default and contains five areas: **Notifications**, **Username**, **Journal file cleanup**, **Worksharing update frequency**, and **View options**. These areas are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Notifications Area

Revit provides an option for setting reminders to save a work at regular intervals. To do so, select the desired option from the **Save reminder interval** drop-down list in the **Notifications** area. By default, 30 minutes is selected in this drop-down list. If you do not want a reminder to save your work in the project, select the **No reminders** option from this drop-down list. Similarly, you can select a value in the **Synchronize with Central reminder interval** drop-down list.

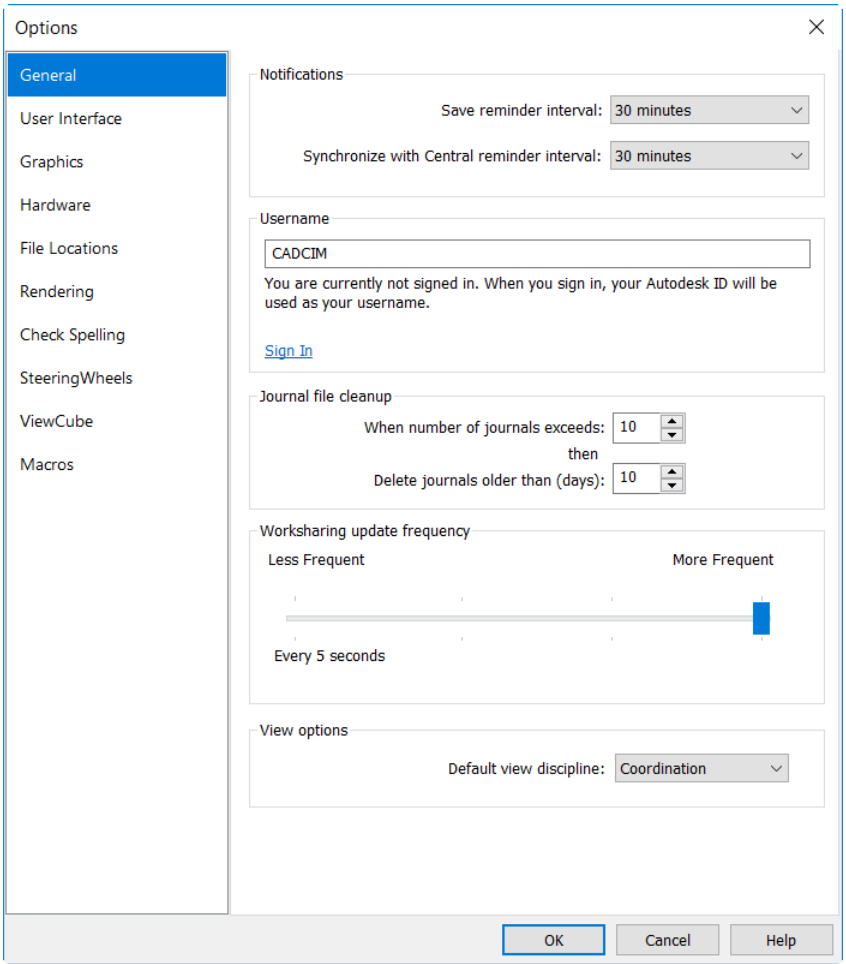

Figure 2-21 *Setting the file options using the* ***Options*** *dialog box*

Username Area

In Autodesk Revit, if you are not signed in as a user of Autodesk account, then the username will not be displayed. To enter a name in the **Username** area, you need to sign in to Autodesk 360. When you sign in, your Autodesk ID will be used as a username.



Note

If you are not logged into an Autodesk account, system username will be taken as the default username.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Journal File Cleanup Area

Journal files are the text files that are used to resolve the technical problems which occur during the Revit session. These files record every step during the session. Whenever you encounter a technical problem with the software, you can run this file to detect the problem or recover the lost files or to know the steps that may have caused the problem. In Autodesk Revit, these files are saved at the following default location: *C:\Users\<Username>\AppData\Local\Autodesk\Revit\Autodesk Revit 2019\Journals* for Windows 7 or Window Vista users and *C:\Documents and Settings\<Username>\Local Settings\Application* for Windows XP users. These files are saved each time you close the Revit session. Therefore, the number of these files keeps on increasing until you remove these files from their location. However to remove these files while retaining some of them, you can use the **Journal file cleanup** area in the **General** tab of the **Options** dialog box. This area contains two spinners: **When number of journal exceeds** and **Delete journals older than (days)**. You can set the required values in these spinners to retain the recently created files. The default value in both the edit boxes is 10. If you want to delete journal files if their number exceeds 15 and if they were created before 30 days, then in such a situation, set the value in the **When number of journals exceeds** spinner to **15** and the value in the **Delete journals older than (days)** spinner to **30**.

Worksharing Update Frequency Area

In this area, you can set the update frequency that indicates the time interval for updating the project in a worksharing environment. To set the limits for worksharing, you can set the slider between the **Less Frequent** and **More Frequent** limits.

View Options Area

In the **View options** area, you can specify the default view discipline to be used in the Revit project. To do so, select an option from the **Default view discipline** drop-down list in this area. In this drop-down list, the **Coordination** option is selected by default.

User Interface Tab

The **User Interface** tab contains three areas: **Configure**, **Ribbon tab switching behavior** and **Visual experience**. In the **Configure** area, you can use different options such as **Keyboard Shortcuts**, **Double-click Options**, and **Tooltip assistance** to change the appearance and operational settings. If you choose the **Customize** button corresponding to the **Keyboard Shortcuts** in the **Configure** area, the **Keyboard Shortcuts** dialog box will be displayed. In this dialog box, various commands with their respective shortcut keys are displayed. You can use the **Assign** and **Remove** buttons to add and remove the shortcut keys.

If you choose the **Customize** button corresponding to **Double-click Options** parameter; the **Customize Double-click Settings** diaog box will be displayed. In this dialog box, you can specify the response by double-clicking an element such as **family**, **Sketched Element,** and so on. To do so, select the desired option form the drop-down list corresponding to the elements available in the **Element Type** column of the **Customize Double-Click Settings** dialog box.

In the **Configure** area, you can select an option from the **Tooltip assistance** drop-down list to set the required level of assistance that a tool tip will display when a cursor is placed over a tool or an element in the project view. From the drop-down list, you can select any of the following options: **Normal**, **None**, **Minimal**, and **High**. By default, the **Normal** option is selected in the **Tooltip assistance** drop-down list.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In the **Ribbon tab switching behavior** area, you can specify the tab to be displayed once you clear a selection or exit a tool. In this area, the **Project Environment** drop-down list contains two options: **Stay on the Modify tab** and **Return to the previous tab**. Select the **Stay on the Modify tab** option to display the options in the **Modify** tab after exiting a tool or clearing a selection. Alternatively, you can select the **Return to the previous tab** option to display the last used tab after exiting a tool or clearing a selection. Similarly, you can use the settings in the **Family Editor** heading to specify the display behavior of the family editor mode. In the **Ribbon tab switching behavior** area, the **Display the contextual tab on selection** check box is selected by default. As a result, the contextual tab is displayed once you select a tool from the Revit interface.

To change the theme of Revit interface, choose the desired option from the **Active theme** drop-down list of the **Visual experience** area. There are two themes available in the **Active theme** drop down list: **Light** and **Dark**. **Light** theme is selected by default. To accelerate the graphical speed of any operation in software, select the **Use hardware graphics acceleration if available** check box from the **Visual experience** area in the **User Interface** tab.

Graphics Tab



You can use this tab to assign colors to selections, highlights, and alerts, and to enable anti-aliasing for 3D views. In the **View navigation performance** area, the **Allow navigation during redraw** check box is selected by default. The selection of the check box allows you to navigate the model in a 2D view or a 3D view, and also pan, zoom and orbit while sketching the drawing. On clearing the check box, you will not be able to perform any of the above actions before the completion of drawing. In this area, the **Simplify display during view navigation** check box is cleared by default. If you select this check box, it will suspend various graphic effects and details such as fill and line, shadows, hidden lines, underlays, etc during camera manipulation. In the **Graphics mode** area, you can select the **Smooth line with anti-aliasing** check box to improve the quality of lines in 3D views. In the **Colors** area of the **Graphics** tab, choose the button corresponding to the **Background** parameter to change the color of the background. The **Selection** parameter refers to a color that an element acquires when it is selected. The default color is **RGB 000-059-189**. To use any other color, click the button on the right of the **Selection** parameter and select the desired color from the **Color** dialog box. The **Pre-selection** parameter is used to specify the color of the highlighted elements. To use any other color, click the button on the right of the **Selection** parameter to display the **Color** dialog box and then select the desired color from this dialog box. Revit uses the **Alert** button to highlight elements when an error occurs. In the **Colors** area, you can select the **Semi-transparent** check box to make the selected elements semi-transparent. Semi-transparency of a selected element allows you to view the elements behind it. In the **Temporary dimension text appearance** area, you can select an option from the **Size** drop-down list to specify the size of temporary dimension text. You can also set the background of the temporary dimensions text. To do so, select the **Opaque** or **Transparent** option from the **Background** drop-down list. **Transparent** is selected by default.

Hardware Tab



The options in the **Hardware** tab enables you to view information related to hardware setup and change related information for better navigation and rendering view. In the **Hardware setup** area of this tab, the **Use hardware acceleration** check box is selected by default. As a result, the hardware accelerators are enabled. Hardware accelerators help display

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the larger models faster on refreshing the views. In addition, the hardware accelerators help you speed up the process of switching between the windows of views.

File Locations Tab

By using the **File Locations** tab, you can set the path for various files and directories that are accessed frequently, refer to Figure 2-22. The path for these files is set while installing Revit. However, you can modify the location of a file by choosing the corresponding **Browse** button and specify a new location. You can also change the default template file location, in case, you wish to use a custom made template file for your projects. In Revit, the default path to save or open a project can be specified in the **Default path for user files** edit box. You can also specify the default path for family template files and for the point clouds by using the corresponding **Browse** button.

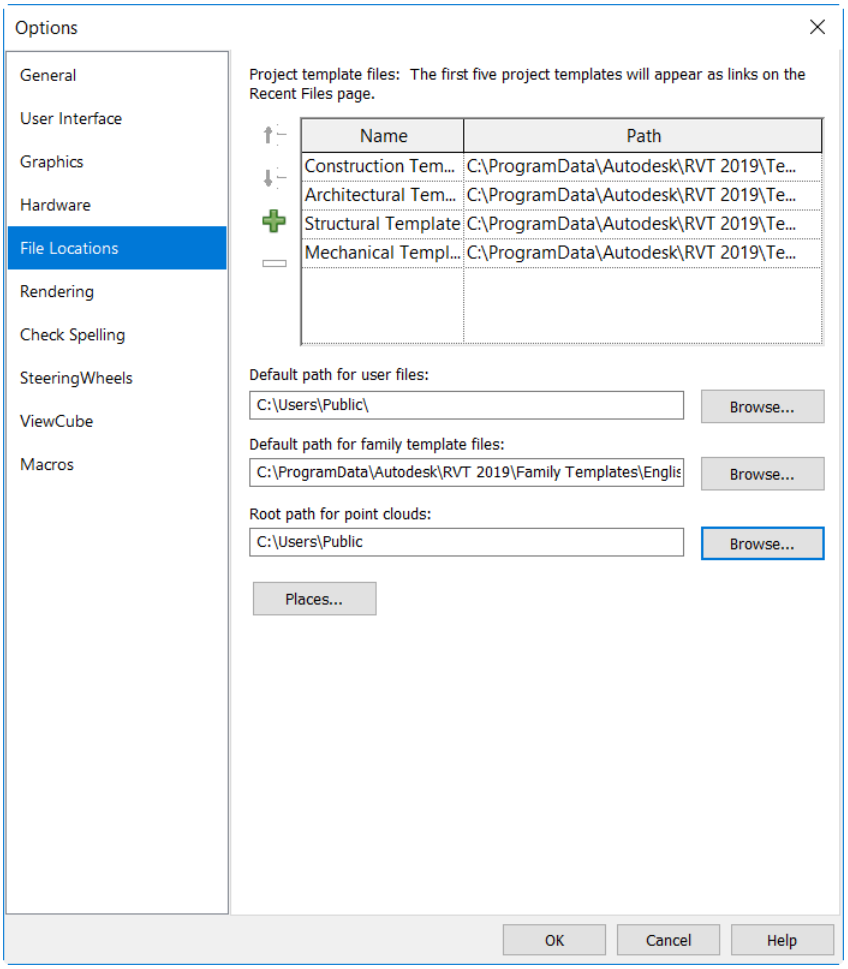

Figure 2-22 *The* ***File Locations*** *tab of the* ***Options*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Rendering Tab

Autodesk Revit uses the mental ray rendering engine for its rendering process. The mental ray has its own library, Render Appearance Library. This library stores information about render appearances for materials, default RPC contents in the software, and other information relevant to the rendering process. The Render Appearance Library is a read-only library and is loaded into the following default location while installing the Revit software: *C:\Program Files (x86)\Common Files\Autodesk Shared\Materials\2019\assetlibrary_base.fbm.*

In addition to the default Render Appearance Library location, you can also specify paths for the additional image files defining texture, bump map, and custom color for the render appearance that you can use in the project. These image files are not present in the software and therefore, you need to specify their paths to use them. To do so, choose the **Add Value** button in the **Additional render appearance paths** area and specify the required path in the displayed field or choose the **Browse** button; the **Browse For Folder** dialog box will be displayed. In this dialog box, select the desired path and choose the **Open** button to add the path in the field.

Check Spelling Tab

Revit provides you with the option to run spell check to find out spelling errors in text and then rectify them. You can choose the **Check Spelling** tab from the **Options** dialog box to display its options. Figure 2-23 shows various options in the **Check Spelling** tab. In the **Settings** list box of this tab, you can specify various self-explanatory settings by selecting their respective check boxes. In the **Main Dictionary** area, you can select the type of dictionary to be used as main dictionary for the spell check from the **Autodesk Revit** drop-down list. Apart from the main dictionary, you can also use additional dictionaries available in this dialog box such as the personal and building industry dictionaries. This facilitates the use of various personal and industrial terms in the text. There are many words that are not included in the main dictionary but are frequently used in the building industry. For example, the abbreviation 'conc' for the word concrete is not available in the main dictionary. The additional building industry dictionary has many such words and abbreviations that can be used in the text of the project and therefore, it does not prompt you to check for spelling errors when it finds such terms/words. You can also add or remove words from your personal and building industry dictionary. To do so, choose the **Edit** button next to the required option in the **Additional dictionaries** area to view the list of words and then enter or remove any word from these lists by using the cursor and keyboard. To run spell check in your drawing, choose the **Check Spelling** tool from the **Text** panel of the **Annotate** tab; the **Spelling** dialog box will be displayed, wherein you can rectify spelling errors in the text by selecting the correct spelling and then choosing the **Change** button in the dialog box. Alternatively, you can press the F7 key to display the **Check Spelling** dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

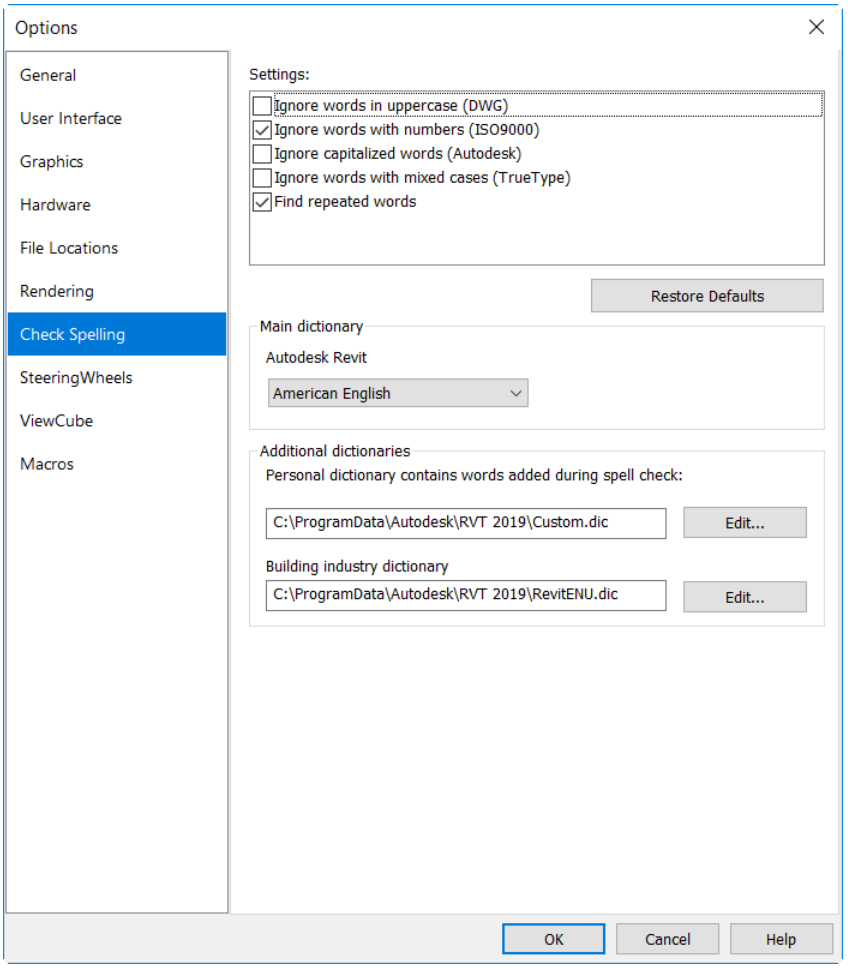

Figure 2-23 *The* ***Check Spelling*** *tab of the* ***Options*** *dialog box*

SteeringWheels Tab

The **SteeringWheels** tab in the **Options** dialog box has options to control the visibility, appearance, and operational tools of different types of SteeringWheels. This tab has seven different areas, as shown in Figure 2-24. These areas are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

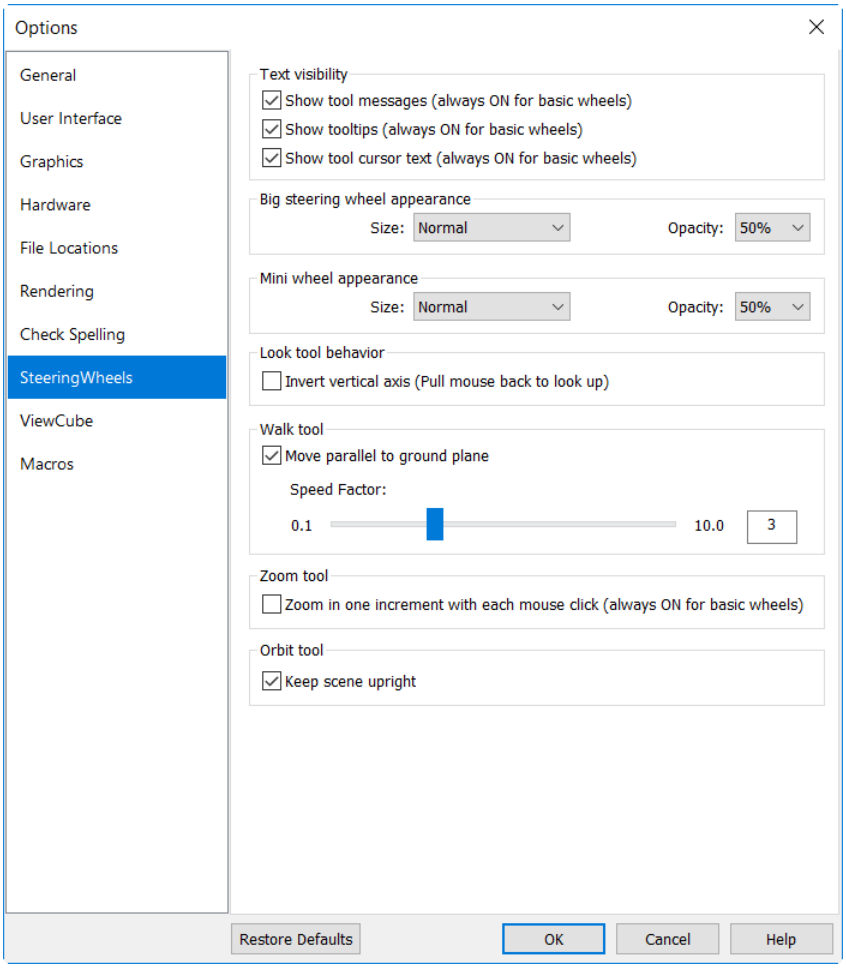

Figure 2-24 *The* ***SteeringWheels*** *tab of the* ***Options*** *dialog box*

Text Visibility Area

You can control the display of tool messages, tooltips, and tool cursor of SteeringWheels by using the options in the **Text visibility** area of the **SteeringWheels** tab. You need to select the **Show tool messages** check box to enable the visibility of tool messages in SteeringWheels. To display tooltips along with SteeringWheels, select the **Show tooltips** check box in this area. Similarly, to control the display of the cursor text when a tool is active, select the **Show tool cursor text** check box.

Big Steering Wheel Appearance and Mini Wheel Appearance Areas

The options in these areas are used to set the size and transparency of the SteeringWheels. To set the size of SteeringWheels, select the required option from the **Size** drop-down list in the corresponding areas and set its size to small, normal, or large. Similarly, you can set the transparency of SteeringWheels by selecting the required option from the **Opacity** drop-down list.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Look Tool Behavior and Walk Tool Areas

In the **Look tool behavior** area of the **SteeringWheels** tab, select the **Invert vertical axis** check box to change the vertical axis movement of the view. Selecting this check box enables the view to move in the same direction as the cursor does in the vertical axis.

The **Speed Factor** slider in the **Walk tool** area is used to change the speed of walk while using the **Walk** tool of the SteeringWheels. You can also select the **Move parallel to ground plane** check box in the **Walk tool** area to constrain the angular movement of the walk to ground plane.

Zoom Tool and Orbit Tool Areas

Select the **Zoom in one increment with each mouse click** check box in the **Zoom tool** area to enable the zooming operation with a single click.

In the **Orbit tool** area, select the **Keep scene upright** check box to maintain perpendicularity between the sides of the model and the ground plane while using the **Orbit** tool.

ViewCube Tab

The **ViewCube** tab in the **Options** dialog box is used to edit various settings of the ViewCube. It has four different areas: **ViewCube appearance**, **When dragging the ViewCube**, **When clicking on the ViewCube**, and **Compass**, as shown in Figure 2-25. These areas are discussed next.

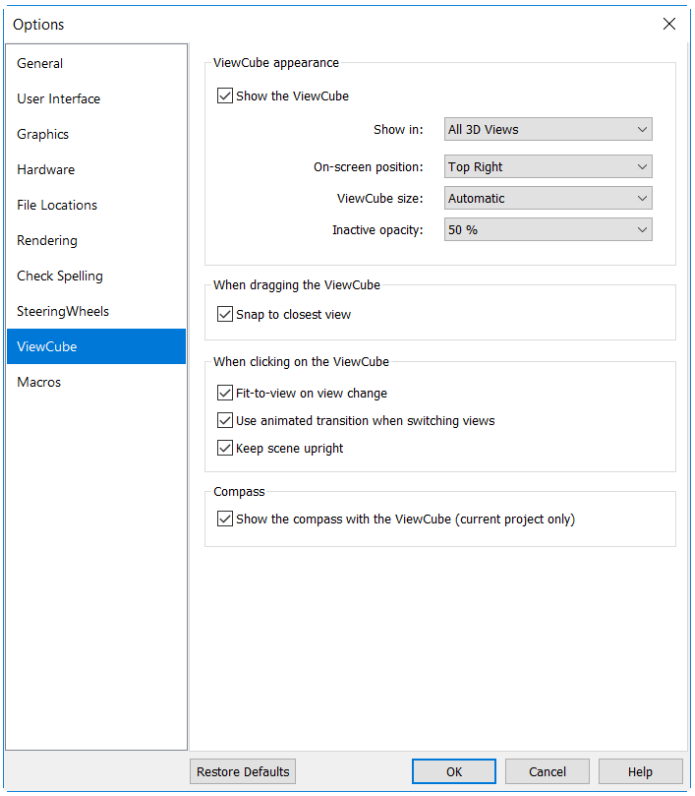

Figure 2-25 *The **ViewCube** tab of the **Options** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

ViewCube Appearance Area

This area is used to control the appearance and display of the **ViewCube**. In this area, the **Show the ViewCube** check box is selected by default. As a result, the ViewCube will be visible. If you clear this check box, the ViewCube will disappear and all options in the **ViewCube** tab will be deactivated. In the **ViewCube appearance** area, you can use various drop-down lists to align, resize, and change the transparency of the ViewCube.

Select the options from the **On-screen position** drop-down list to align the ViewCube on the screen. Similarly, if you want to resize the ViewCube, select the required option from the **ViewCube size** drop-down list. You can also set the opacity of the inactive ViewCube by selecting an option from the **Inactive opacity** drop-down list.

When Dragging the ViewCube Area

Select the **Snap to closest view** check box in this area to enable the snap to select the closest view in the ViewCube.

When Clicking on the ViewCube Area

Select the **Fit-to-view on view change** check box in the **When clicking on the ViewCube** area to fit the view on the screen while changing the viewing direction. In this area, the **Use animated transition when switching views** check box is selected by default. As a result, the animated transition occurs while switching the views. If you clear this check box, the animated transition will not occur while using the ViewCube. Select the **Keep scene upright** check box to keep the sides of ViewCube and the sides of the view perpendicular to the ground plane. Clear the check box to turn around the model in full 360-degree swing. Clearing this check box can be useful when you are editing a family.

Compass Area

In this area, the **Show the compass with the ViewCube** check box is selected by default. As a result, the compass along with the ViewCube is visible in the drawing. In the **ViewCube** tab, you can choose the **Restore Defaults** button to restore the default settings that were changed in its different areas.

Macros Tab

Revit provides you with the option to run macros within the projectI and then rectify the problems occurred. To do so, choose the **Macros** tab from the **Options** dialog box; various options will be displayed in the window, refer to Figure 2-26. The options in the **Macros** tab can be used to set the security level for the macros used in the project. There are two options in this area, **Application macro security settings** and **Document macro security settings**, which are discussed next.

Application Macro Security Settings Area

In this area, the **Enable application macros** radio button is selected by default. As a result, the application macros in the project are enabled. To disable the application macros, select the **Disable application macros** radio button from the **Application macro security settings** area.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Document Macro Security Settings Area

In this area, the **Ask before enabling document macros** radio button is selected by default. As a result, the document macros are disabled but you will be prompted to enable them whenever you open a project that contains a macro. To enable the document macros, select the **Enable document macros** radio button. Similarly, to disable a document macros in the project, select the **Disable document macros** radio button in the **Document macro security settings** area.

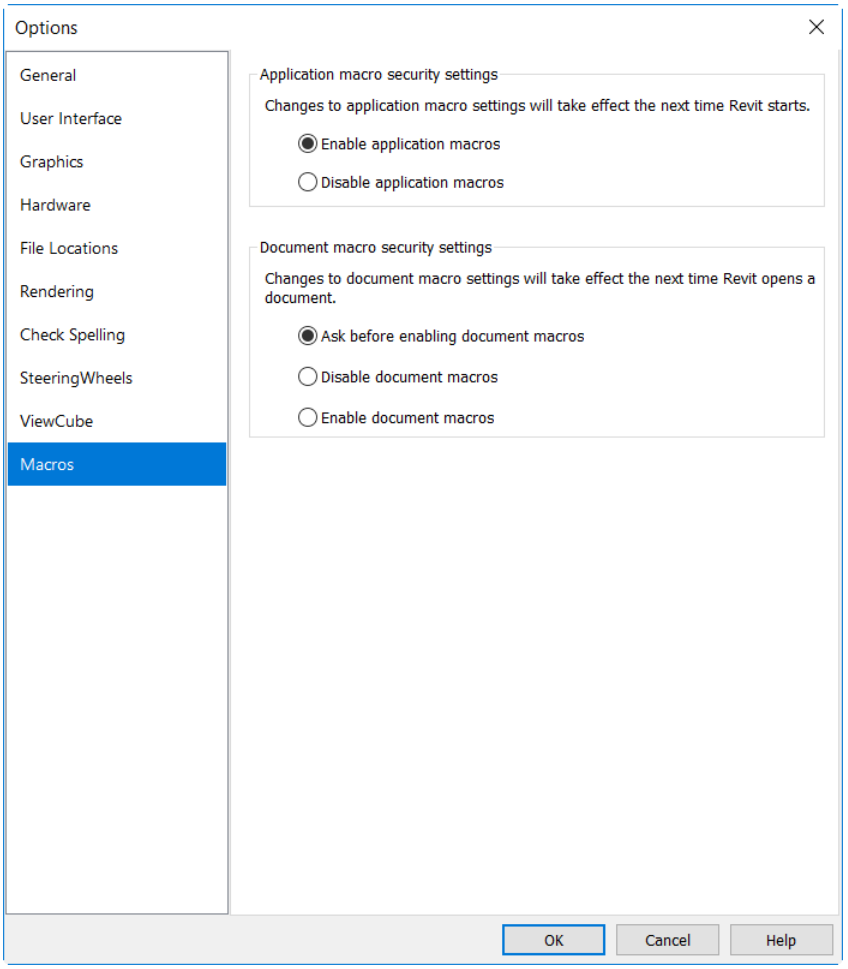

Figure 2-26 *The* ***Macros*** *tab of the* ***Options*** *dialog box*

SAVING A PROJECT

Before you close or exit a Revit session, it is recommended to save the project file. You can save a project file in a permanent storage device, such as a hard disk or a removable storage device like CD or USB. Also, you must save your work at regular intervals to avoid data loss due to any error in the computer's hardware or software.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Using the Save As Tool

Application Menu: Save As > Project

In Revit, you can save your project file at the desired location. To do so, choose the **File** menu and then choose **Save As > Project**; the **Save As** dialog box will be displayed, refer to Figure 2-27. Alternatively, you can save the project file by choosing the **Save** button from the **Quick Access Toolbar.**

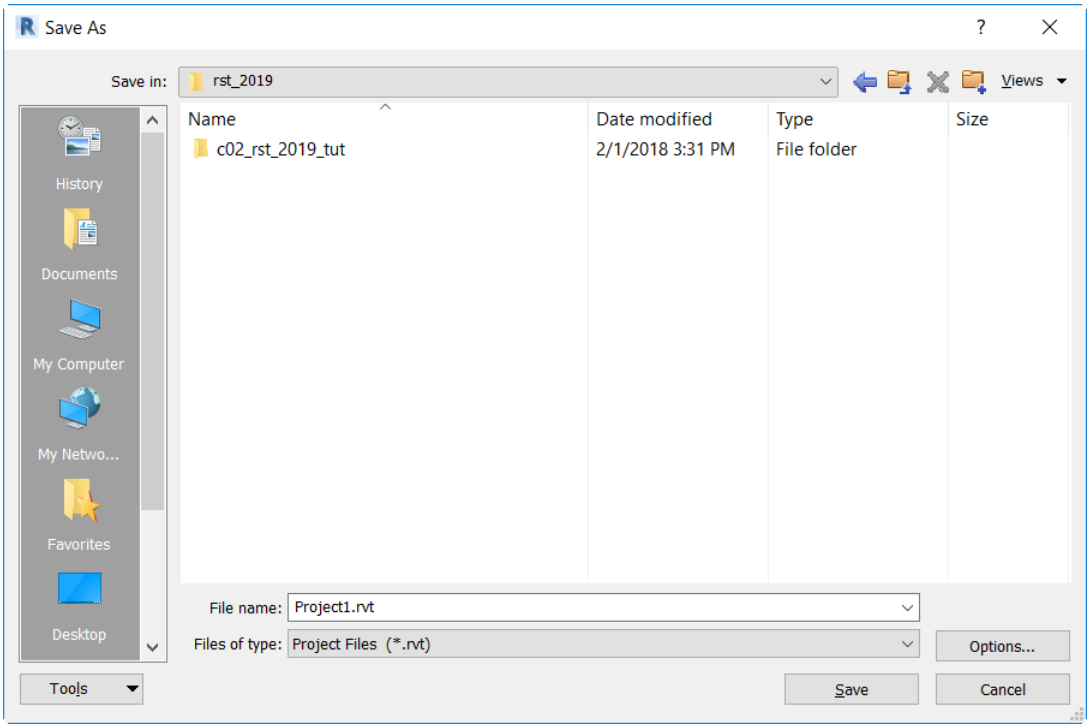

Figure 2-27 *The* ***Save As*** *dialog box*

In the **Save As** dialog box, the **Save in** drop-down list displays the current drive and path in which the project file will be saved. The list box below the **Save in** drop-down list shows all folders available in the current directory. The **File name** edit box is used to specify the name of the file to be assigned to the project. The **Places List** area on the left of the **Save As** dialog box contains shortcuts for the folders that are frequently used.

You can use different file saving features by choosing the **Options** button from the **Save As** dialog box. On choosing this button, the **File Save Options** dialog box will be displayed, as shown in Figure 2-28. Using the **Maximum** edit box from this dialog box, you can specify the maximum number of backup files that you need to store for a project. In Autodesk Revit, the non-workshared projects have three backup files and the workshared projects have twenty backup files by default. The options in the **Worksharing** area are inactive for non-workshared projects. You can use the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

options in this area to make the current workshared file central compact and select the default workset. The options in the **Thumbnail Preview** area enable you to specify the image to be used as the preview of the project file. This image is used at the time of opening a project file. You can specify a view of the model as a preview image by selecting the corresponding option from the **Source** drop-down list in the **Thumbnail Preview** area of the **File Save Options** dialog box. By default, the **Active view/sheet** option is selected in the **Source** drop-down list of the **Thumbnail Preview** area for the preview of a project file. For example, to make **Structural Plan: Level 1** as the preview image, select it from the drop-down list. As a result, when you invoke the **Open** dialog box and select a file to open, the view displayed in the **Preview** area will correspond to the selection that is made from the **Source** drop-down list in the **File Save Options** dialog box.

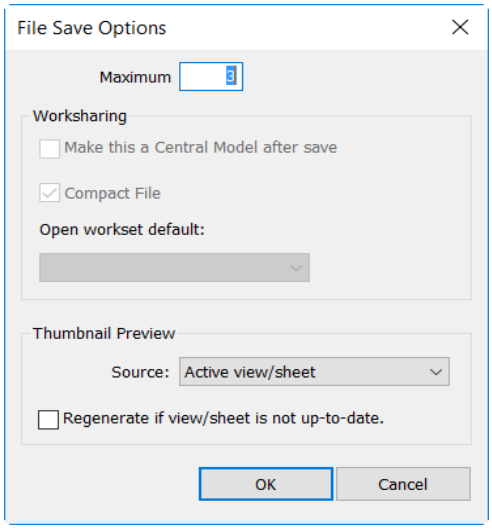

Figure 2-28 *The* ***File Save Options*** *dialog box*

In the **File Save Options** dialog box, you can select the **Regenerate if view/sheet is not up-to-date** check box to see the preview with the latest modifications. On selecting this check box, the preview image will be updated when you close the project file.



Note

Revit updates the preview image continuously. Therefore, selecting the ***Regenerate if view/sheet is not up-to-date*** *check box can consume considerable resources.*

Using the Save Tool

Once a project has been saved using the **Save As** tool, you do not need to re-enter file parameters to save it again. To save a project to the hard disk, choose the **File** menu button and then choose the **Save** option from **File Menu**. If you are saving the project for the first time, the **Save As** dialog box will be displayed, even if you invoke the **Save** tool.



Tip

By default, the preview of a file is the last active view or sheet at the time it was last saved. You can set the preview to a particular view using the ***Options*** *button in the* ***Save As*** *dialog box.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Alternatively, you can save your project by choosing the **Save** button from the **Quick Access Toolbar**. As you save your project file, Revit updates it automatically without prompting you to re-enter the file name and path.

CLOSING A STRUCTURAL PROJECT

To close a project, choose the **File** menu button and then select the **Close** option. If you have already saved the latest changes, the project file will be closed. Otherwise, Revit will prompt you to save the changes through the **Save File** dialog box. You can save the changes by choosing the **Yes** button or discard them by choosing the **No** button. You can also choose the **Cancel** button to return to the interface and continue working on the project file. You can also use the **Close** button (X) in the drawing area to close the project.

EXITING A STRUCTURAL PROJECT

To exit a Revit session, choose the **Exit Revit** button from **File** menu. Even if the project is open, you can choose the **Exit Revit** button to close the file and exit Revit. If the project has not been saved once, the **Save File** dialog box will be displayed on choosing the **Exit Revit** button. In this dialog box, if you choose the **No** button, all unsaved changes will be lost. You can also use the **Close** button (**X**) in the main Revit window (in the title bar) to end the Revit session.

OPENING AN EXISTING STRUCTURAL PROJECT

Application Menu:	Open > Project
Shortcut Keys:	CTRL+O

In Autodesk Revit, there are several options available to open an existing project. These options are discussed next.

Opening an Existing Project Using the Open Tool

To open an existing project file, choose **Open > Project** from **File Menu**, as shown in Figure 2-29. Alternatively, you can open the project file by choosing the **Open** button from the **Quick Access Toolbar** or by pressing the CTRL+O keys. On invoking the **Open** tool, the **Open** dialog box will be displayed, as shown in Figure 2-30. Using the **Look in** drop-down list in this dialog box, you can access the desired folder and open the desired file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

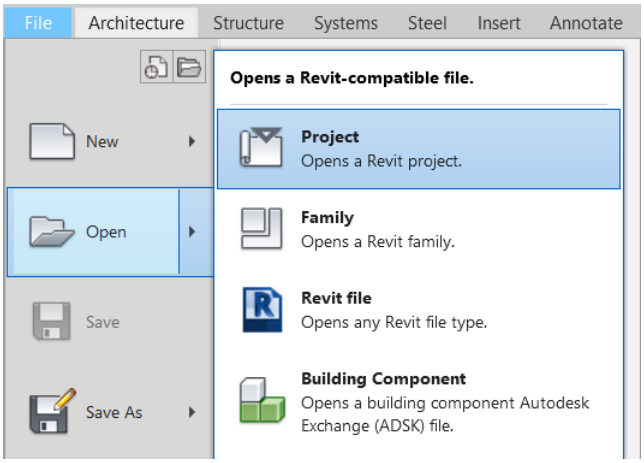

Figure 2-29 *Choosing the* ***Project*** *option from* ***File*** *menu*

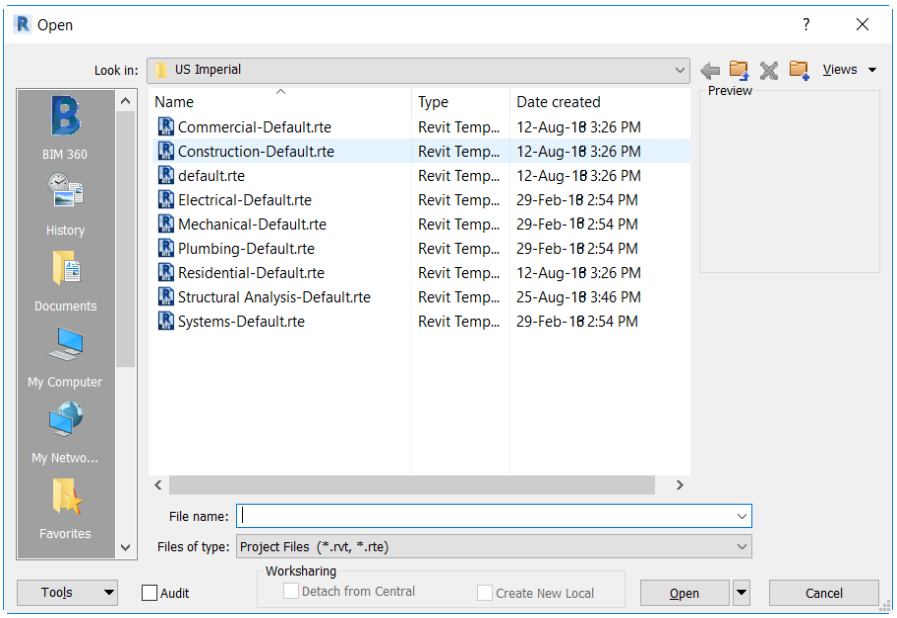

Figure 2-30 *The* ***Open*** *dialog box*

The **Preview** area of the **Open** dialog box shows the preview of the selected project file. It helps you select a particular file by viewing its contents, even if you are not sure about the name of the file. The window icons such as the **Views** menu, placed on the right of the **Look in** drop-down list, helps you select a project file based on its size, type, or date when it was last saved. On choosing the **Thumbnails** option from the **Views** menu, you can preview the contents of the project files inside the selected folder in the file list area. In the **Open** dialog box, you can browse to important locations from the **Places** list. The **Places** list is located on the left side of the **Open** dialog box. In this list, you can add or remove folders as per your requirement. To do so, first close the **Open** dialog box and then choose the **Options** button from **File** menu; the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Options dialog box will be displayed. In this dialog box, choose the **File Locations** tab and then the **Places** button from it; the **Places** dialog box will be displayed, as shown in Figure 2-31. The **Places** dialog box contains two columns: **Library Name** and **Library Path**.

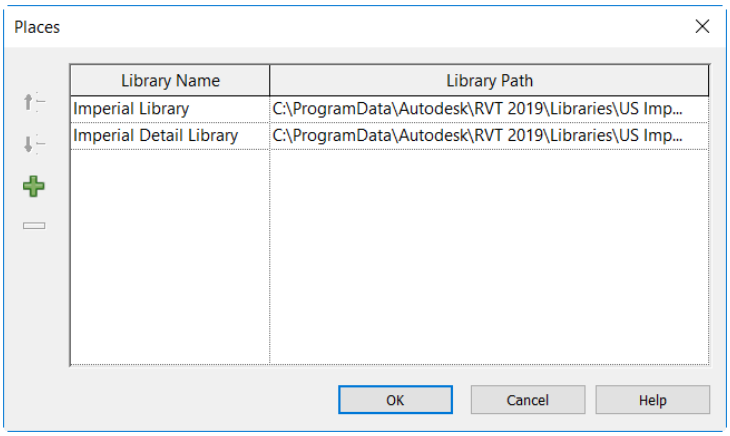

Figure 2-31 *The* ***Places*** *dialog box*

You can add or remove folders in the libraries list to create a list of frequently accessed folders. The four buttons on the left side of the **Places** dialog box can be used to create or delete a library, or move it up and down in the list. To create a new library, choose the **Add Value** button, which is the third button from the top; a new library will be added to the defined path. By default, the name of the new library in the **Library Name** section will be **New Library1**. Change the name of the new library and then click in the **Library Path** column to display the **Browse** button. Choose the **Browse** button and select the folder to be added in the libraries list by using the **Browse For Folder** dialog box. Next, choose the **Open** button; the new folder will get added to the list. If required, choose the upward arrow button from the **Places** dialog box to move the folder up to the top of the list. Similarly, you can choose the down arrow button to move it down. To delete a library, select it and choose the **Remove Value** button. Choose the **OK** button in the **Places** dialog box to exit, and then close the **Options** dialog box. When you invoke the **Open** tool next time, the new folder icon will be displayed in the places list.

Once the file to be opened has been selected, its name will be displayed in the **File Name** edit box of the **Open** dialog box and its preview will be displayed in the **Preview** area.



Note

If you try opening an already opened file that has been modified in the Revit session, a message box appears, prompting you to close the file first and then reopen it. In case, you open a file that has been created using an older version of Revit, the ***Program Upgrade*** *message box is displayed. This message box informs that the file is being upgraded to the latest file format and that this is a onetime process. Once the file is opened, choose the* ***Save*** *button from* ***Application*** *menu to save it to upgraded Autodesk Revit version.*

Using the Windows Explorer to Open an Existing Project

Apart from using the **Open** tool from the Revit interface to open a file, you can also open files directly from the **Windows Explorer** by using the methods discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

A file can be opened by double-clicking on its icon in the **Windows Explorer**. It opens the project file in the latest Revit session. If Revit is not running, double-click on the file icon to start Revit and then open the file.

Another method of opening a project file is by dragging the project file icon from the **Windows Explorer** and dropping it in the drawing window of the Revit interface. You can also select, drag, and drop more than one file in the drawing window. In this case, Revit prompts you to open the files in separate windows. Choose the **OK** button to open all files in the same Revit session.

TUTORIALS

In this tutorials, you will create two structural projects as tutorials: a commercial project of a concrete structure and an industrial project of steel and concrete structure. These tutorials will be created in a sequence according to the topics discussed in the chapter.

Tutorial 1 Commercial Complex

In this tutorial, you will create a revit file for the *Commercial Complex* project with the following parameters and project specifications: **(Expected time: 15 min)**

1. Template file:
   For Imperial **US Imperial > Structural Analysis-Default.rte**
   For Metric **US Metric > Structural Analysis-DefaultMetric.rte**

2. Project Units:
   For Imperial **Feet and fractional inches**, Rounding- **To the nearest 1/2"**
   For Metric **Millimeters**, Rounding- **To the nearest 10**
3. Length dimension snap increment:
   For Imperial **5' ; 2' 6" ; 0' 3" ; 0' 01/2"**
   For Metric **1500 ; 700 ; 70 ; 10**

4. File name to be assigned:
   For Imperial *c02_Commercial-Complex_tut1.rvt*
   For Metric *M_ c02_Commercial-Complex_tut1.rvt*

The following steps are required to complete this tutorial:

a. Start a Revit session.

b. Open a new project.

c. Use the template file for the project,
   For Imperial **Structural Analysis-Default.rte**.
   For Metric **Structural Analysis-DefaultMetric.rte**

d. Select project units.

e. Set **Endpoint**, **Midpoint**, **Nearest**, **Perpendicular**, **Work Plane Grid**, and **Intersections** as the object snaps in the **Snaps** dialog box for both the systems (Imperial & Metric).

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

f. Save the project by using the **Save** tool.
For Imperial *c02_Commercial-Complex_tut1.rvt*
For Metric *M_c02_Commercial-Complex_tut1.rvt*

g. Close the project by using the **Close** tool.

Starting Autodesk Revit 2019

1. Start Autodesk Revit by choosing **Start > All Programs > Autodesk > Revit 2019**. On doing so, the Revit interface window is displayed.

Opening a New Project

1. Choose **New > Project** from menu; the **New Project** dialog box is displayed.

Selecting the Template File

To select the template file for the project, you need to access the appropriate folder and select the required template file from it.

1. In the **New Project** dialog box, choose the **Browse** button from the **Template file** area; the **Choose Template** dialog box is displayed. This dialog box shows a list of template files available in the **US Imperial** folder, if **Imperial** unit system is selected at the time of Revit installation. (On selecting **Metric** unit system, list of metric template is displayed).

2. In the **Choose Template** dialog box, the template file **Structural Analysis-Default.rte**, as shown in Figure 2-32, is selected as the default template file in the **US Imperial** folder. In **US Metric** folder, default template file is **Structural Analysis-DefaultMetric.rte**. Choose the **Open** button to assign the selected template file to the **New Project** dialog box.

3. Select the **Project** radio button (default) to create a project file and then choose the **OK** button; the selected template file of selected unit system is loaded. Note that the **Project Browser** shows different levels and views that have already been created in the selected template file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

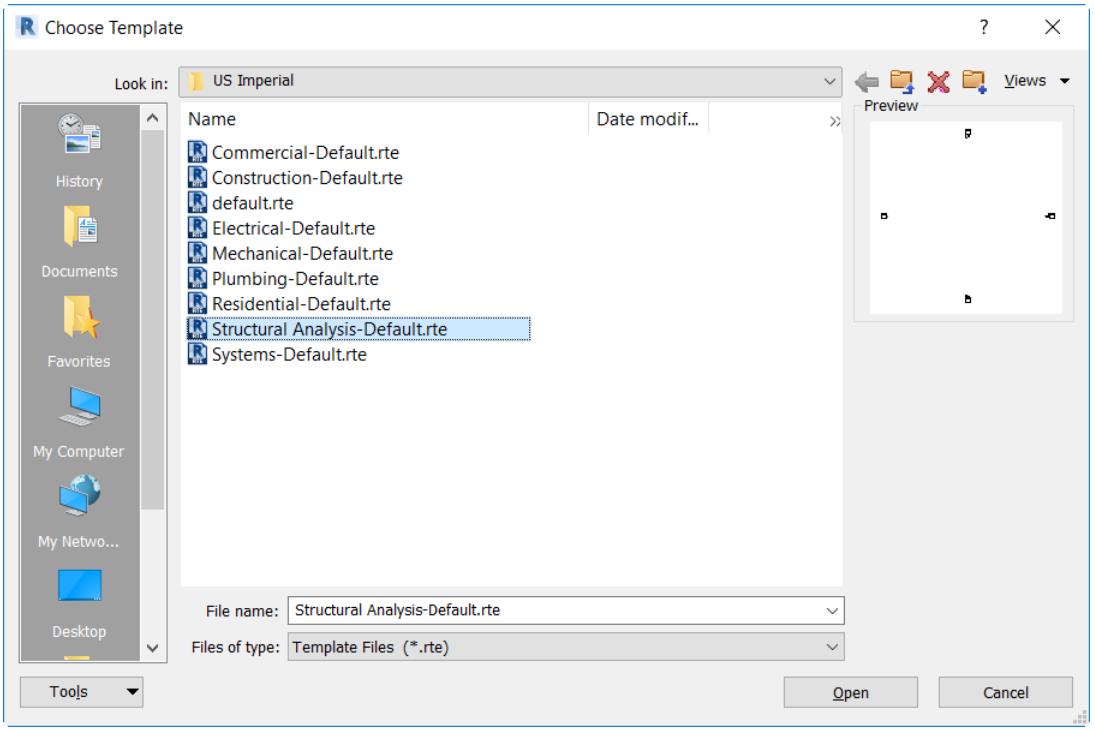

Figure 2-32 *The* ***Choose Template*** *dialog box*

Setting Project Units

1. To set units for the project, choose the **Project Units** tool from the **Settings** panel of the **Manage** tab; the **Project Units** dialog box is displayed.

2. Make sure that the **Common** option is selected in the **Discipline** drop-down list of the **Project Units** dialog box. Now, click in the **Format** column corresponding to the **Length** unit; the **Format** dialog box is displayed.

3. In this dialog box, select the option by clicking on the **Units** drop-down list.
   For Imperial **Feet and fractional inches**
   For Metric **Millimeters**

4. Select the option from the **Rounding** drop-down list.
   For Imperial **To the nearest 1/2"**
   For Metric **To the nearest 10**

5. Choose the **OK** button in the **Format** dialog box and then in the **Project Units** dialog box to close them.

Setting Dimension and Object Snaps

You need to set dimension snaps and object snaps by using the **Snaps** tool. These settings are based on the type of the project and the amount of detailing required.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

1. Choose the **Snaps** tool from the **Settings** panel of the **Manage** tab; the **Snaps** dialog box is displayed, as shown in Figure 2-33.

2. In the **Dimension Snaps** area of this dialog box, ensure that the **Length dimension snap increments** check box is selected.

3. Now, click in the edit box below the **Length dimension snap increments** check box and delete the existing values and then enter the new values.

   For Imperial **5’ ; 2’ 6” ; 0’ 3” ; 0’ 0 1/2”**
   For Metric **1500 ; 700 ; 70 ; 10**

   Figure 2-33 shows the value entered for length snaps increment.

4. In the **Object Snaps** area, clear the **Quadrants**, **Centers**, **Tangents**, and **Points** check boxes to disable them. Leave the other options such as **Endpoints**, **Midpoints**, **Nearest**, **Perpendicular**, **Work Plane Grid**, **Snap to Remote Objects**, **Snap to Point Clouds**, **Perpendicular** and **Intersections** enabled, refer to Figure 2-33. Choose the **OK** button to apply settings and to close the **Snaps** dialog box.

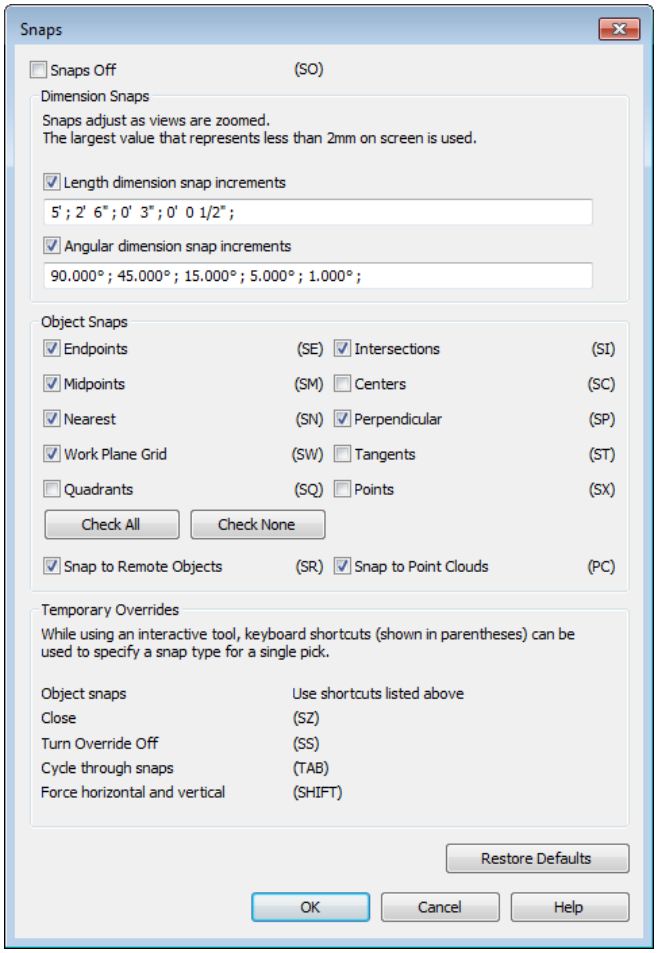

Figure 2-33 *The* ***Snaps*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Saving the Project

The project parameters set in the previous steps are the global settings of the project file. To save this project file, use the **Save As** tool.

1. Choose the **Save** option from the **File** menu. As you are saving the project for the first time, the **Save As** dialog box is displayed.

2. In this dialog box, browse to the *C* drive and then create a folder with the name **rst_2019**.

3. Open the *rst_2019* folder and then create a sub-folder with the name *c02_rst_2019_tut*. Next, open the created folder and save the file with the name *c02_Commercial-Complex_tut1.rvt* (For Imperial), refer to Figure 2-34. For Metric, save the file with the name *M_c02_Commercial-Complex_tut1.rvt* .

4. Choose the **Save** button; the **Save As** dialog box closes and the project file is saved.

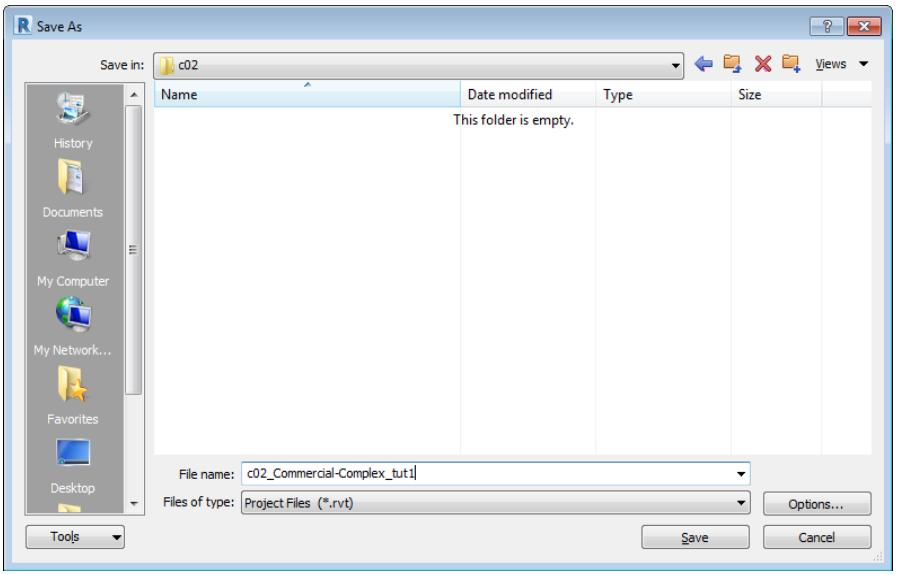

Figure 2-34 *The* ***Save As*** *dialog box*

Closing the Project

1. To close the project, choose the **Close** option from **File** menu in the Ribbon.

Tutorial 2 Industrial Complex

In this tutorial, you will create a new project file for the *Industrial Complex* project by using the following parameters and project specifications. **(Expected time: 15 min)**

1. Template file:

For Imperial **US Imperial > Structural Analysis-Default.rte**
For Metric **US Metric > Structural Analysis-DefaultMetric.rte**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

2. Project Units:
   For Imperial **Feet and fractional inches,** Rounding- **To the nearest 1/4"**
   For Metric **Millimeters**, Rounding- **To the nearest 10**

3. Length dimension snap increment:
   For Imperial **10';2'6";1'; 0'3"**
   For Metric **3000;760;300; 70**

4. Object snaps to be set- all available object snaps.

5. File name to be assigned:
   For Imperial *c02_Industrial-Complex_tut2.rvt*
   For Metric *M_c02_Industrial-Complex_tut2.rvt*

The following steps are required to complete this tutorial:

a. Start the Revit session.

b. Open a new project.

c. Use the template file.
   For Imperial *Structural Analysis-Default.rte*
   For Metric *Structural Analysis-DefaultMetric.rte*

d. Set the project units by using the **Project Units** dialog box.
   For Imperial **Feet and fractional inches**
   For Metric **Millimeters**

e. Set the length dimension snap increment in the **Snaps** dialog box.
   For Imperial **10';2'6";1'; 0'3"**
   For Metric **3000;760;300; 70**

f. Enable all object snaps using the **Snaps** dialog box.

g. Save the project by using the **Save As** tool.
   For Imperial *c02_Industrial-Complex_tut2.rvt*
   For Metric *M_c02_Industrial-Complex_tut2.rvt*

h. Close the project by using the **Close** tool.

Starting Revit and Opening a New Project

1. Start a new Revit session by double-clicking on the Revit 2019 shortcut icon on the desktop. The program is loaded and the user interface screen is displayed. If the Revit session is already running, you can ignore the step and open the project file directly.

2. Choose **New > Project** from the **File** menu; the **New Project** dialog box is displayed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Using the Template File

As given in the project parameters, you need to use the template file for this project. You can select this template file from the **New Project** dialog box.

1. In the **New Project** dialog box, choose the **Browse** button in the **Template file** area; the **Choose Template** dialog box is displayed. Select the **Structural Analysis-Default.rte** template file from the **US Imperial** folder or the **Structural Analysis-DefaultMetric.rte** template file from the **US Metric** folder and then choose the **Open** button; the **Choose Template** dialog box is closed. Make sure that the **Project** radio button is selected in the **Create new** area. Next, choose **OK** to exit the **New Project** dialog box; the template file is loaded for this project.

Setting Project Units

You can set units for various measurement parameters by using the **Project Units** dialog box.

1. Choose the **Project Units** tool from the **Settings** panel of the **Manage** tab; the **Project Units** dialog box is displayed.

2. Click in the **Format** column corresponding to the **Length** unit; the **Format** dialog box is displayed. In this dialog box, select the option from the **Units** drop-down list.
   For Imperial **Feet and fractional inches**
   For Metric **Millimeters**

3. Click on the **Rounding** drop-down list and select the desired option, as shown in Figure 2-35.
   For Imperial **To the nearest 1/4”**
   For Metric **To the nearest 10**

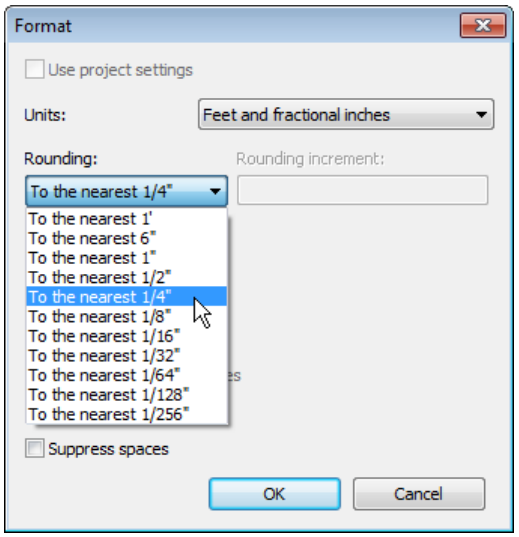

Figure 2-35 *The* ***Format*** *dialog box*

4. Choose **OK** to return to the **Project Units** dialog box. Next, choose the **OK** button; the dialog box is closed and the user interface screen is displayed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Setting Dimensions and Object Snaps

Next, you need to access and modify settings in the **Snaps** dialog box. You need to specify the dimension snap increment and enable all object snap options.

1. Choose the **Snaps** tool from the **Settings** panel of the **Manage** tab; the **Snaps** dialog box is displayed.

2. In this dialog box, enter values in the edit box below the **Length dimension snap increments** check box in the **Dimension Snaps** area.
   For Imperial **10';2'6";1'; 0'3"**
   For Metric **3000; 760; 300; 70**

3. In the **Object Snaps** area, ensure that all the object snaps options are selected and then choose the **OK** button to apply settings and exit the **Snaps** dialog box.

Saving the Project

In this section, you need to save the project and the settings using the **Save As** tool.

1. Choose **Save As > Project** from the **File** menu. The **Save As** dialog box is displayed.

2. Browse to the *C:\rst_2019\c02_rst_2019_tut* and for imperial enter **c02_Industrial-Complex_tut2** in the **File name** edit box (for Metric enter **M_c02_Industrial-Complex_tut2**). Notice that in the **File of type** drop-down list, the **Project Files (*.rvt)** option is selected by default.

3. Choose the **Save** button to save the project as *c02_Industrial-Complex_tut2.rvt* for Imperial *(M_c02_Industrial-Complex_tut2.rvt* for Metric*)*. The project is saved at the specified location.

Closing the Project

1. Choose the **Close** option from the **File** menu; the project gets closed.

Self-Evaluation Test

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

1. The options in the _____ drop-down list are used to control the symbolic display of moment connections for cantilever framing that exists throughout a project.

2. The _____ drop-down list in the **Structural Settings** dialog box contains two options: **Parallel Line** and **Line with Angle**.

3. In Autodesk Revit, you can set the symbolic representation of structural framing components by using the _____ dialog box.

4. To specify the location of the default template file, you can use the _____ tab in the **Options** dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

5. The _____ button in the **Save As** dialog box can be used to specify the maximum numbers of backup(s) for a project file.

6. The _____ option of Revit enables you to override snap settings for a single pick only.

7. You can add folders to the _____ in the **Save As** dialog box to access the frequently used folders directly.

8. You can open only one Revit project at a time. (T/F)

9. In Revit, all enabled object snaps work together. (T/F)

10. A project file can be opened by double-clicking on its file name in the Windows Explorer. (T/F).

Review Questions

Answer the following questions:

1. Which of the following options is not an object snap option?

a) **Endpoints** b) **Work Plane Grid**
c) **Dimension** d) **Centers**

2. Which of the following keys is used to toggle between the object snap options available at the same point?

a) TAB b) CTRL
c) ALT d) F3

3. You can specify the settings for **ViewCube** and **SteeringWheels** in the _____ dialog box.

4. You can select any of the three options, namely **Beams and Braces**, **Column Base**, and **Column Top** from the _____ drop-down list.

5. You can modify the project unit settings anytime during a project. (T/F)

6. The name and path of a project file should be specified each time you save the project. (T/F)

7. You cannot control the visibility of the tooltip assistant. (T/F)

8. The **Save reminder interval** drop-down list available in the **General** tab of the **Options** dialog box is used to specify the time interval between reminder prompts to save a project file. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

9. If you choose the **Close** button without saving the changes made in a project file, Revit will prompt you to save the changes before closing it. (T/F).

10. You can add words to additional dictionaries for checking spelling errors. (T/F)

EXERCISES

Exercise 1 Academic Institution

In this exercise, you will create a new project file for the *Academic Institution* project with the following parameters. **(Expected time: 15 min)**

1. Template file-
   For Imperial **US Imperial > Structural-Analysis Default.rte**
   For Metric **US Metric > Structural-Analysis DefaultMetric.rte**

2. Project Units-
   For Imperial **Feet and fractional inches**
   For Metric **Millimeters**

3. File name to be assigned:
   For Imperial *c02_Academic-Institution_exer1.rvt*
   For Metric *M_c02_Academic-Institution_exer1.rvt*

Exercise 2 Factory Shed

In this exercise, you will create a new project file for the *Factory Shed* project by using the following parameters. **(Expected time: 15 min)**

1. Template file:
   For Imperial **US Imperial > Structural-Analysis Default.rte**
   For Metric **US Metric > Structural-Analysis DefaultMetric.rte**

2. Project Units:
   For Imperial **Feet and fractional inches**
   For Metric **Millimeters**

3. File name to be assigned:
   For Imperial *c02_Factory-Shed_exer2.rvt*
   For Metric *M_c02_Factory-Shed_exer2.rvt*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. Cantilever connection, **2. Plan representation**, **3. Structural Settings**, **4. File Locations**, **5. Options**, **6. Temporary Overrides**, **7. Places list**, **8.** F, **9.** T, **10.** T