Chapter 1

Introduction to STAAD.Pro CONNECT Edition

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the basic features of STAAD.Pro CONNECT Edition*
- *Start STAAD.Pro CONNECT Edition*
- *Start new project in STAAD.Pro CONNECT Edition*
- *Explore different components of user interface*
- *Import different files to STAAD.Pro CONNECT Edition*

INTRODUCTION TO STAAD.PRO CONNECT EDITION

STAAD.Pro CONNECT Edition is used to create, analyze, and design any type of virtual structure through its flexible modeling environment. The three basic activities which are to be carried out to achieve this goal are: model generation, calculations to obtain the analytical results, and result verification. All these activities are discussed individually in different chapters. STAAD.Pro is designed for engineers who understand the process of modeling, analyzing, and designing a structure.

BASIC FEATURES OF STAAD.PRO CONNECT EDITION

The basic features of STAAD.Pro CONNECT Edition are listed below:

1) State-of-the art 2D/3D graphical environment.
2) Ability to perform structural analysis and design generate 2D/3D CAD models.
4) Modeling of truss and beam members, plates, solids, linear and non-linear cables, and curvilinear beams.
5) Advanced automatic load generation facilities for wind, area, floor, and moving loads.
6) Customizable structural templates for creating a model.
7) Toggle display of loads, supports, properties, and structural elements.
8) User-controlled numbering scheme for structural elements.

STARTING BENTLEY STAAD.PRO CONNECT EDITION

You can start STAAD.Pro CONNECT Edition by double-clicking on the STAAD.Pro CONNECT Edition icon available on the desktop. Alternatively, choose **Start > Bentley Engineering > STAAD.Pro CONNECT Edition** from the task bar (for Windows 10), refer to Figure 1-1; the start page screen of STAAD.Pro CONNECT Edition will be displayed, as shown in Figure 1-2.

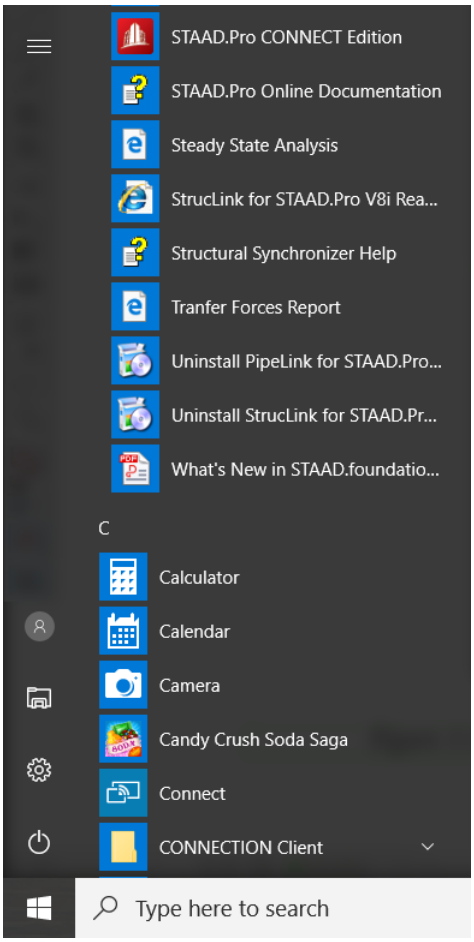

Figure 1-1 *Starting STAAD.Pro CONNECT Edition from the task bar*

The start screen of STAAD.Pro CONNECT Edition consists of two panes. The left pane contains five tabs: **Open**, **New**, **Help**, **License** and **Configure** and the right pane comprises three areas: **Properties**, **CONNECT Properties** and **Additional Licenses**, refer to Figure 1-2. These tabs and areas are discussed next.

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

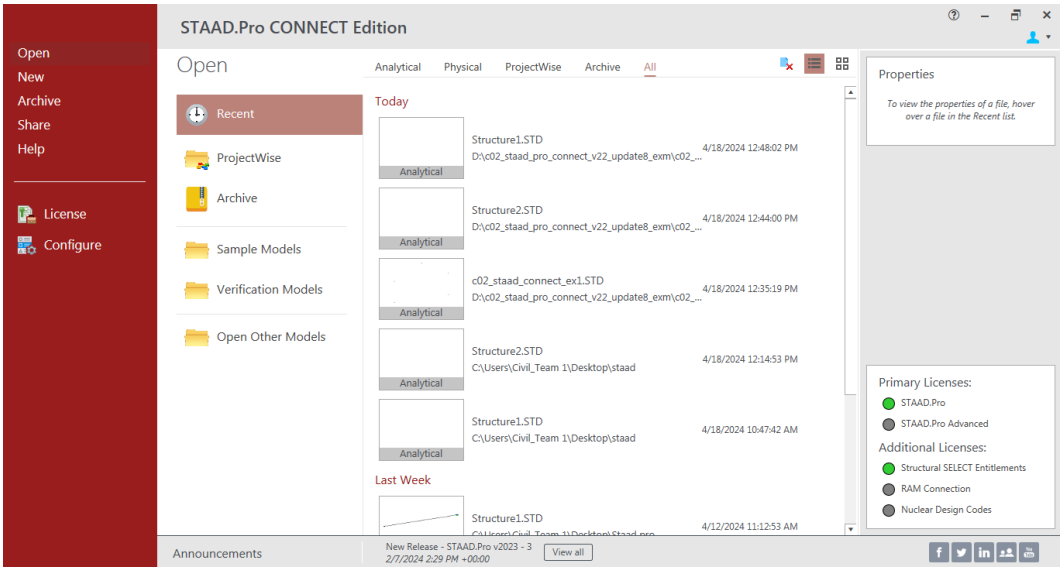

Figure 1-2 *The start page of STAAD.Pro CONNECT Edition*

The **Open** tab is used to open an existing STAAD input file. Note that you can also open STAAD projects from a ProjectWise data source. The **Open** tab in the left pane is chosen by default and the **Open** page is displayed in the start screen with four different options: **Recent**, **Projectwise**, **Archive**, and **Open Other Models**. The **Recent** option will be selected by default on choosing the **Open** tab and recently used files will be displayed on the screen. Choose the file either from the recent files or you can also open the model from the archieve files. The **Archive** option is used to create or extract STAAD archive files. Choose the **Open Other Models** option; the **Open** dialog box will be displayed. Browse to the required location and then select the required file from the available list of files. Next, choose the **Open** button; the project will be loaded and structure will be displayed in the **Whole Structure** Window. The **New** tab is used to create a new project. The process of creating a new project is discussed in detail later in this chapter.

The **Help** tab contains several options that enables you to access help documents online or offline, technical support services, product news, and so on. You can access the offline help by clicking on the **Help Contents** option.

The **License** tab is used to check the activation status of currently installed products.

In STAAD.Pro, there are two base unit systems: English Imperial and metric. The default unit system will be the one which you have selected while installing the program. You can also change the base units in the program. To do so, choose the **Configure** option; the **Application Configuration** dialog box will be displayed, refer to Figure 1-3. In this dialog box, the **General** tab and **Base unit** area will be chosen by default. In this page, the current base unit will be displayed at the top. You can change the unit by selecting the **English** or **Metric** radio button from the **Base Unit** area. The **Configure** tab is also used to configure the program settings such as units, color, input or output file formats, default design codes, and so on.

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

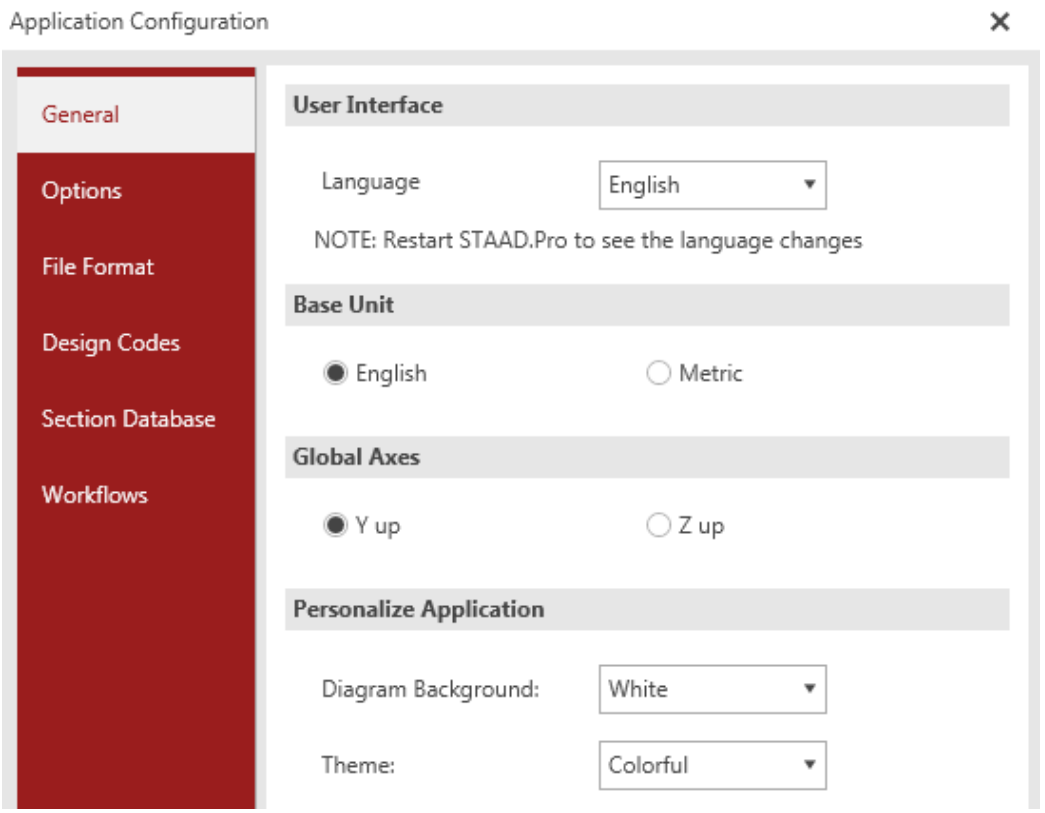

Figure 1-3 *The* ***Application Configuration*** *dialog box*

When you hover the cursor on any of the recently used files, its preview and location will be displayed in the **Properties** or **CONNECT Properties** area in the right pane of the user interface. The **Additional Licenses** area consists of three check boxes: **STAAD.Beava (Bridge Codes), STAAD.Planwin**, and **RAM Connection**. You can select the required code before opening a file by selecting the corresponding check box. If the license for the corresponding code is available, the circle besides the check box will be green otherwise will be red.

STARTING A NEW PROJECT

To start a new project, choose the **New** option from the left pane; the **Model Information** page will be displayed, as shown in Figure 1-4. Enter name of the file in the **File Name** edit box in the **Model Information** page. To specify the file location, click on the **Browse** button available on the right of the **Location** edit box; the **Select Folder** dialog box will be displayed. Select the folder where you want to save the file and then choose the **Select Folder** button; the path of the location will be displayed in the **Location** edit box. Next, choose **Analytical** in the **Type** area and **Metric** in the **Units** area.

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

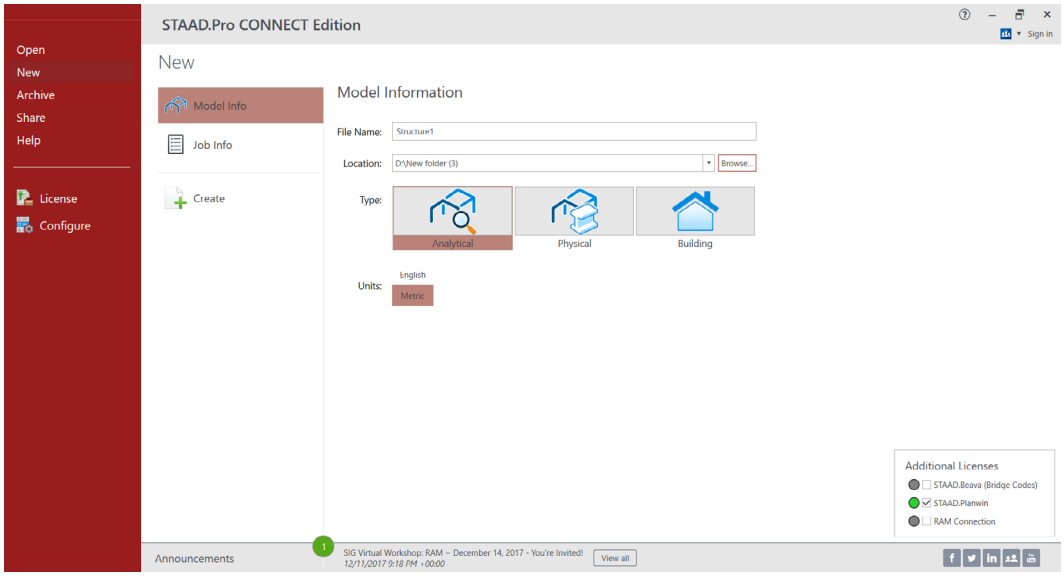

Figure 1-4 *The start screen of STAAD.Pro CONNECT Edition with the* ***Model Information*** *page*

Choose the Job Info option from the New area; the Job Information page will be displayed. In this page, enter the job name, client name, job number, engineer name, project ID, and project name in the respective edit boxes. Choose the **Associate Project** button to save the details before creating the model, refer to Figure 1-5. After specifying all the data, choose the **Create** option in the **New** area; the user interface will be displayed, as shown in Figure 1-6.

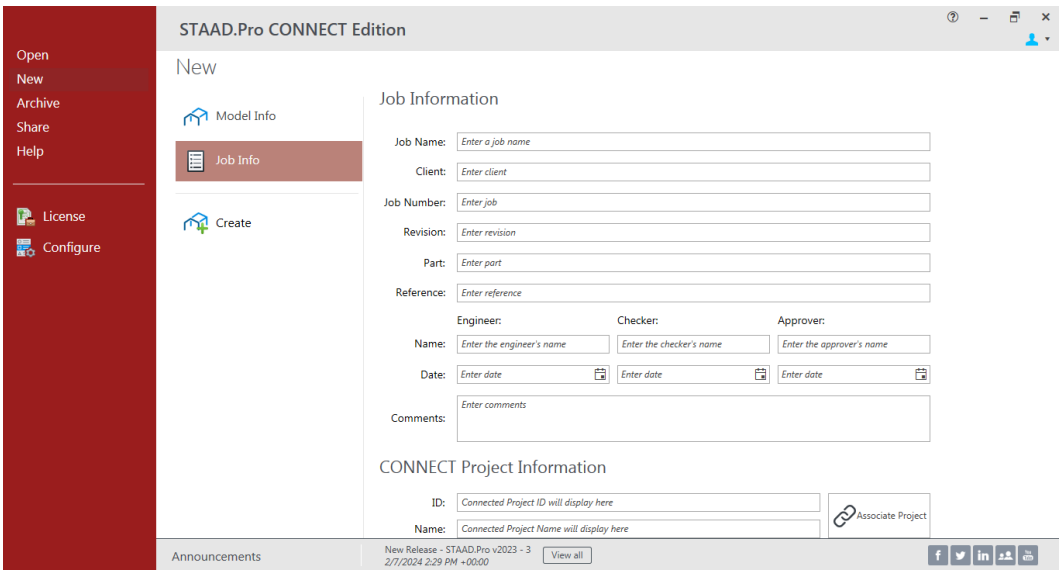

Figure 1-5 *The start screen of STAAD.Pro CONNECT Edition with the* ***Job Information*** *page*

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

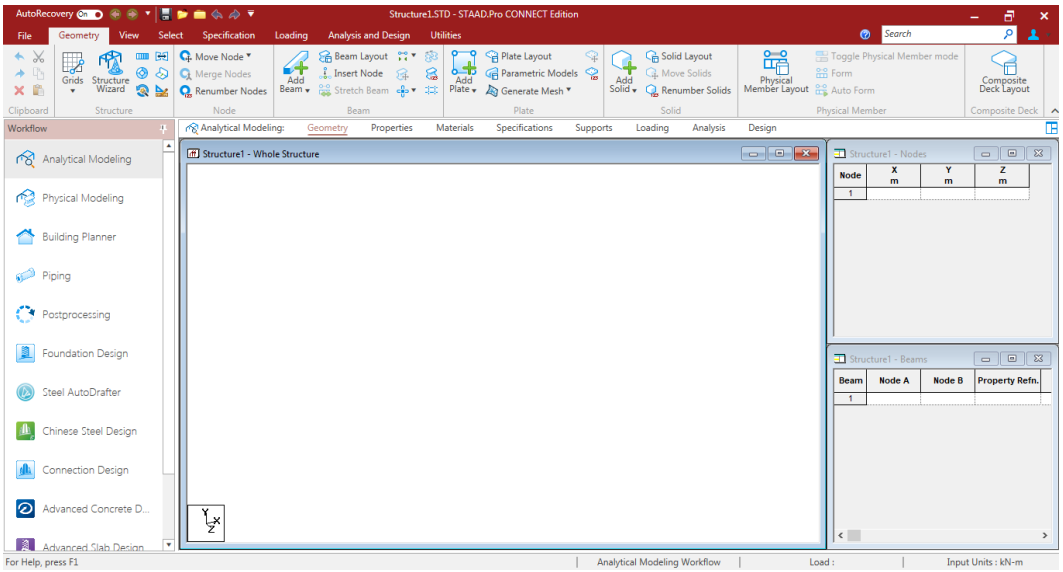

Figure 1-6 *The user interface screen of STAAD.Pro CONNECT Edition*

WORKING IN USER INTERFACE

The user interface of STAAD.Pro CONNECT Edition elements are exclusively designed to provide an easy access to the tools and windows. The interface comprises several elements such as the Ribbon, Quick Access Toolbar, Workflows, Data area, View window, Tool Search, Status Bar, and so on, refer to Figure 1-6. Some of these elements are discussed next.

Ribbon Tab

The Ribbon tab is located at the top of the interface. It comprises different tabs: **File**, **Geometry**, **View**, **Specification**, **Loading**, **Analysis and Design**, and **Utilities**. These menus will be available in the **Analytical Modeling** mode. When you switch to a different mode, such as **Postprocessing**, different tabs will appear in the Ribbon tab. These modes are discussed in the later chapters. The different tabs in the Ribbon tab are discussed next.

File

The **File** menu contains the options which are used for performing different file operations such as creating new structure, opening an existing model, saving files, printing files, and so on. Figure 1-7 shows different options of the **File** menu.

The selection of the **Info** option from the **File** menu will display the **Info** page. You can use this page to specify general information about the structure including a job description, job number, persons responsible for creating, checking, and approving the structure, and so on. The **New** option will allow you to create a new STAAD project.

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

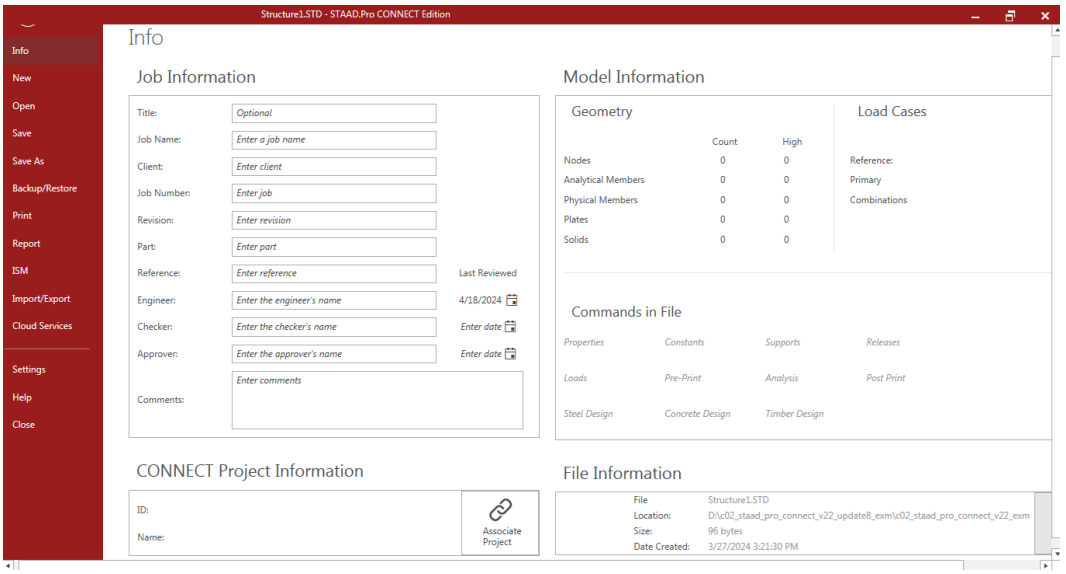

Figure 1-7 *Options of the* ***File*** *menu*

The Open option is used to open an existing STAAD input file or archive. The **Save** and **Save As** options are used to save changes to the current model or to save the current model as a different file, respectively. The **Print** option is used to print input, output, and reports for the current STAAD project. The **Report** option is used to configure, review, and export STAAD reports. The **ISM** option contains tools for working with the Integrated Structural Modeling repositories.

The **Import/Export** option will allow you to import data from other formats and also export data to other formats such as DXF, VRML, CIMsteel STEP format, and so on. The **Cloud Services** option opens the Bentley Cloud Services page for using the cloud services. The **Settings** option is used to control some of the application settings.

The **Tools** option displays the **Tools** page which contains tools for managing file backups. The **Help** option displays the **Help** page that has links for getting additional assistance and information about STAAD.Pro. The **Close** option is used to close STAAD graphical environment.

Geometry

The **Geometry** tab contains the Selection, Clipboard, Structure, Node, Beam, Plate, Solid, Physical Member, and Composite deck panels. The tools available in the panels of the **Geometry** tab are shown in the Figure 1-8 and some of these are discussed next.

The **Node Cursor**, **Beam Cursor**, and **Plate Cursor** tools are used to select nodes, beams, and plate element in a structure, respectively.

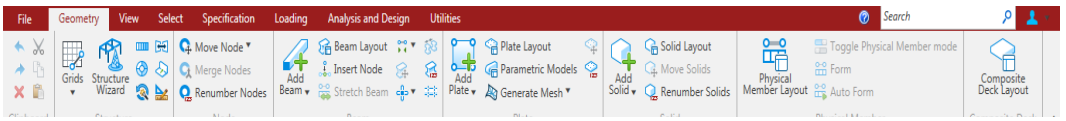

Figure 1-8 *Options in the* ***Geometry*** *tab*

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

The **Snap/Grid Node** tool is used to specify the grid and snap settings for creating nodes, beams, plates, and solids. The **Insert Node** tool is used to insert node on an existing member. The **Add Plate** tool is used to add plates by connecting nodes. The **Create Infill Plates** tool is used to automatically generate the floor slab by selecting some or all beams in a structure. The **Translational Repeat** tool is used to copy the entire structure linearly. The **Circular Repeat** tool is used to copy the entire structure in circularly. The **Generate Surface Meshing** tool is used to create a finite element mesh. The **Rotate** tool is used to rotate the selected portions of the structure or the entire structure about the specified axis through a specified distance. The **Merge Selected Members** tool is used to merge two members and replace them with one. The **Renumber** tool is used to renumber nodes, members, or plates starting with a specified number. The **Run Structure Wizard** tool is used to display the **StWizard** window which is discussed in later chapter.

View

The **View** tab consists of the **Labels**, **Tools**, **Views, Options,** and **Windows** panels. The **Labels** panel consist of tools which are used to customize the view of the structure by setting different view-related parameters. The **Tools** panel is used to perform various view related tasks such as zooming, panning, setting colors and fonts, and so on. Figure 1-9 shows different options in the **View** tab. The **3D Rendering** option is used to view the rendered view of the structure.

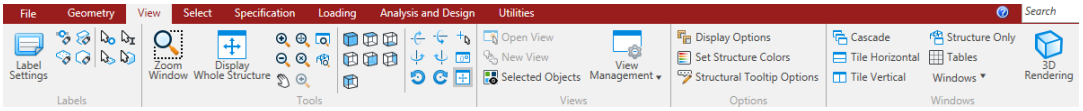

Figure 1-9 *Options in the* ***View*** *tab*

Specification

The **Specification** tab consists of the **Beam Profiles**, **Plate Profiles**, **Materials**, **Specifications**, **Supports,** and **Tools** panels. The various options in the **Specification** tab are shown in the Figure 1-10.

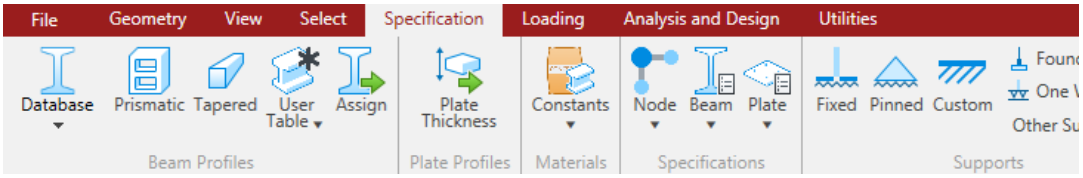

Figure 1-10 *Options in the* ***Specification*** *tab*

The **Assign** option is used to assign sections, materials, material properties, supports, and specifications to the structural members. The **Plate Thickness** option is used to specify the plate thickness. The **Constants** option is used to define and assign material constants such as Density, Elasticity, Poisson's Ratio, Coefficient of Thermal Expansion, and so on. The **Section Wizard** option is used to display the **Section Wizard** window. In this window, you can calculate section property values such as area, moment of inertia, and so on for cross sections. These cross sections can be assembled from pre-existing standard shapes, user-created shapes, and parametric shapes.

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

Loading

The **Loading** tab is used to define and apply loads on a structure. The **Loading** tab consists of the **Loading Specifications**, **Load Generation**, **Define Load Systems**, **Dynamic Specifications**, and **View** panels. The options in these panels are shown in Figure 1-11.

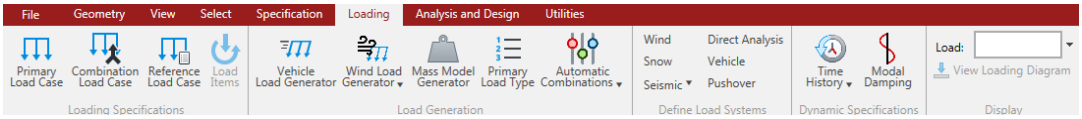

***Figure 1-11** Options in the **Loading** tab*

Analysis and Design

The **Analysis and Design** tab consists of the **Analysis Data**, **Analysis**, **Cloud Analysis Services,** and **Design Commands** panels. The various options in these panels are shown in Figure 1-12.

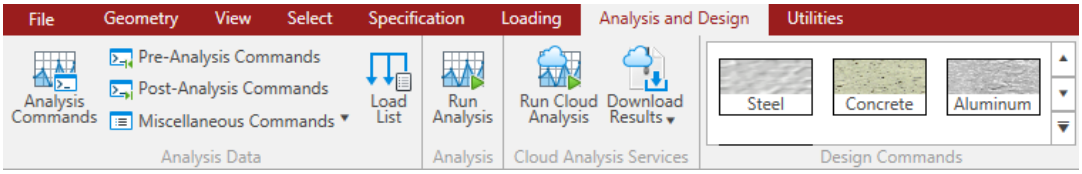

***Figure 1-12** Options in the **Analysis and Design** tab*

The **Analysis Commands** and **Pre Analysis Commands** options open the **Analysis/Print Commands** dialog box which is used to define the analysis and pre-analysis print commands to be included in the input command file, respectively. The **Post Analysis Commands** option opens the **Post Analysis Print -- Whole Structure** dialog box, which is used to define the post-analysis print commands to be included in the input command file. The **Miscellaneous Commands** option is used to add commands such as Input Width, Output Width, Set Z Up, and so on. The **Run Analysis** option performs the STAAD analysis as directed by input commands. The **Design Commands** panel is used to specify design parameters and commands.

Utilities

The **Utilities** tab consists of the **Geometry Tools**, **Physical Model**, **Query, Display, Edit, View**, **Tools**, **Developer**, and **User Tools** panels. The options in these panels are shown in Figure 1-13. The options available in the **Utilities** panel are discussed next.

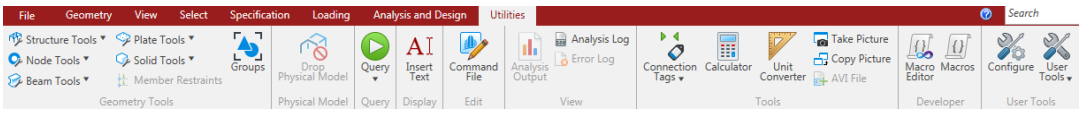

***Figure 1-13** Options in the **Utilities** tab*

The **Editor** option opens the current input command file (file extension .std) in the **STAAD.Pro CONNECT Edition Editor** window. On selecting the **STAAD Output** option, the results of analysis and design will get displayed in the **STAAD Output Viewer** window. **Unit Converter** opens the **STAAD.Pro Converter** window which is used to convert data from one unit system to another. **AVI File** opens the **Create AVI File** dialog box which is used to create a video file recording for animated deflection, section displacement, mode shape, and plate stress contour diagrams.

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

Tool Search

You can search for any tool by typing a part of the name in the Search field. In the search results drop-down, click **Show Details** to display the tool tip description for each tool in the results list. The tool search box is placed at the right extreme corner of the Ribbon tab, refer to Figure 1-6.

Quick Access Toolbar

The Quick Access Toolbar is located just above the Ribbon tab. It contains some of the most frequently used tools in a convenient location. Figure 1-14 shows the tools in the Quick Access Toolbar.



Figure 1-14 *Tools in the* ***Quick Access Toolbar***



The **Save** tool saves any change made to the current model.



The **Open** tool opens the start page screen of STAAD.Pro CONNECT Edition with the **Open** tool selected, which is used to select a model to open in the program.



The **Close** tool closes the current model and returns to the start screen of STAAD.Pro CONNECT Edition.



The **Undo** tool undo some previous operation.



The **Redo** tool redo some previous undo operation.



The **Command File Editor** tool opens the current input command file (file extension .std) in STAAD.Pro CONNECT Edition Editor. If any change made in the structure is unsaved, you are prompted to save the structure first.



The **STAAD Output** tool opens the results of a successful analysis and design of the current model in the **STAAD Output Viewer** window.

Workflows

The STAAD.Pro user interface allows you to switch between the **Analytical Modeling**, **Physical Modelling**, **Building Planner**, **Piping**, **Bridge Deck**, **Postprocessing**, **Foundation Design, Chinese Steel Design**, **Connection Design**, **Advanced -Concrete Design**, **Advanced Slab Design,** and **Earthquake** tabs, refer to Figure 1-15. Some of these tabs are discussed next.

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

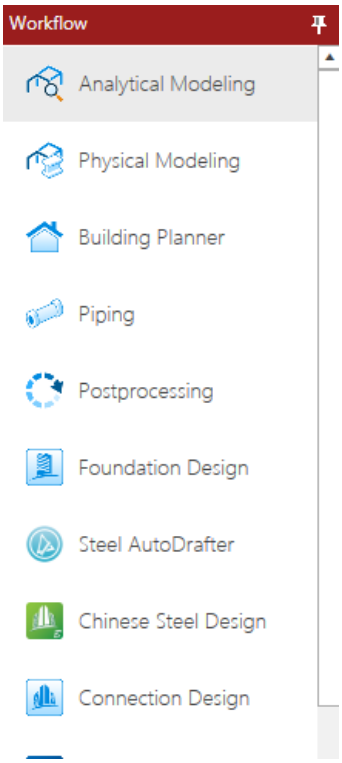

Figure 1-15 *Partial view of the* ***Workflows*** *tab*

Analytical Modeling

The **Analytical Modeling** tab is the first tab of the **Workflows**, as shown in Figure 1-15. This tab contains different pages which are used for creating structure, assigning properties, assigning supports, defining loads, and so on, refer to Figure 1-16. The pages that are associated with analytical modelling are discussed in detail in the later chapters.

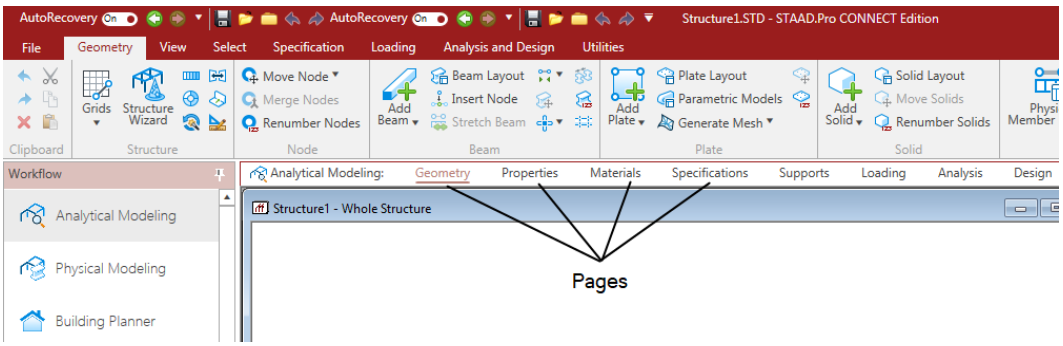

Figure 1- 16 *Pages associated with the analytical modeling*

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

Physical Modeling

STAAD.Pro Physical Modeler is user interface for modeling structures using physical concepts. Beams and surfaces are placed in the model on the same scale as they would appear in the physical world. You can model structure geometry, specifications, and many loads with the STAAD.Pro Physical Modeler.



Note

Existing analytical models cannot be opened in the physical modeler. When you open a model that was created using the physical modeler, you will be asked if you want to open the model in the physical modeler environment. Select the ***Physical Modeling*** *workflow. The* ***STAAD.Pro Physical Modeler*** *window will be displayed.*

When a structure is to be analyzed, an analytical model is generated from the physical model in a process referred to as decomposition. Surface entities will mesh into plate elements and beams will be automatically subdivided for analysis as required by intersecting members. Physical modeling is discussed in detail in the later chapters. Figure 1-17 shows the user interface screen in physical modeling.

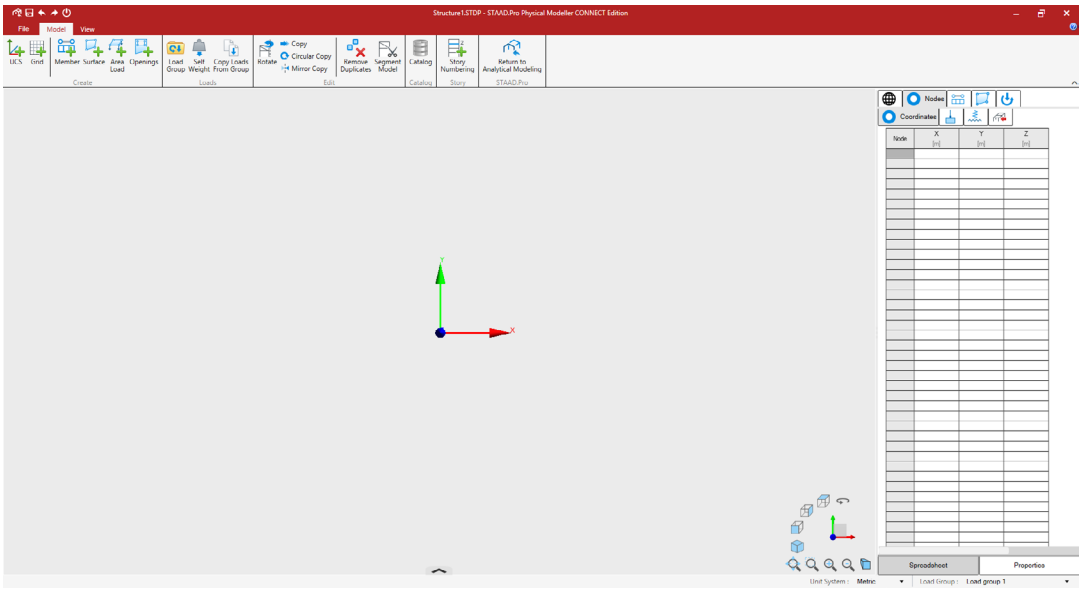

Figure 1-17 *User interface screen in Physical modeling*

Postprocessing

This workflow offers graphical result verification and visualization facilities. A comprehensive custom report generation facility is also incorporated. The customized reports may be in both tabular and graphic form. The **Postprocessing** tab has the **Displacements**, **Reactions**, **Beam Results**, **Plate Results**, **Solid Results**, **Dynamics**, and **Reports** pages, refer to Figure 1-18. Some of these are discussed next.

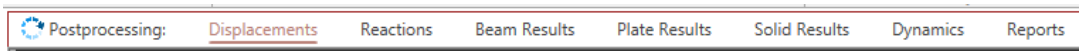

Figure 1-18 *Pages in the* ***Postprocessing*** *tab*

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

Displacements

The **Displacement** page allows you to view the nodes and beam relative displacements. You can view them graphically in the view window or in tabular form in the data area.

Beam Results

The **Beam** page allows you to view the member end forces and member stresses in both graphic and tabular form. It also allows you to view the bending moment diagram and shear force diagram of individual members at a time. These pages are discussed in detail in the later chapters.

Reports

The **Reports** page allows you to create customized report. This report will include structural elements, properties, load cases, mode shapes, numerical and graphical results, and so on.

Data Area

In STAAD.Pro, the data area is located on the right side in the interface. This area contains different windows and tables. In this area, you can provide the coordinates for creating nodes and members, define member properties, define supports, define loads, and so on. This area will be displayed on creating a new file or opening an existing file. Figure 1-19 shows partial view of the data area which contains the **Nodes** and **Beams** tables.

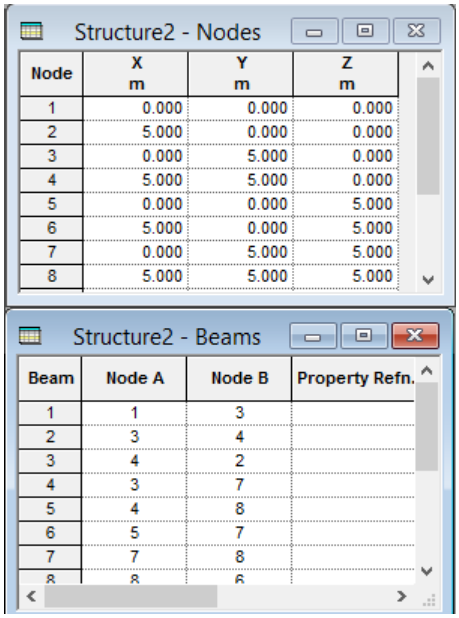

Figure 1-19 *Partial view of the* ***Nodes*** *and* ***Beams*** *tables in the data area*

Status Bar

Status Bar displays the current workflow along with tips on next actions based on the current tools. You can also set the current load case and input units in the status bar.

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

View Window

The View window covers the largest area in the interface. In the View window, the created structure along with the assigned properties, loads, supports, and specifications will be displayed. The analysis results and diagrams will also be displayed in this area.

KEYBOARD SHORTCUTS

In STAAD.Pro, keyboard shortcuts are assigned to some of the frequently used commands. These shortcuts can be typed using the keyboard to invoke the corresponding command. These shortcut keys are also available in the **Labels** tab of the **Diagrams** dialog box, as shown in Figure 1-20. To invoke the **Diagrams** dialog box, right-click in the main window; a shortcut menu will be displayed. Choose the **Labels** option; the **Diagrams** dialog box will be displayed. Table 1-1 shows some of the frequently used shortcut keys in STAAD.Pro.

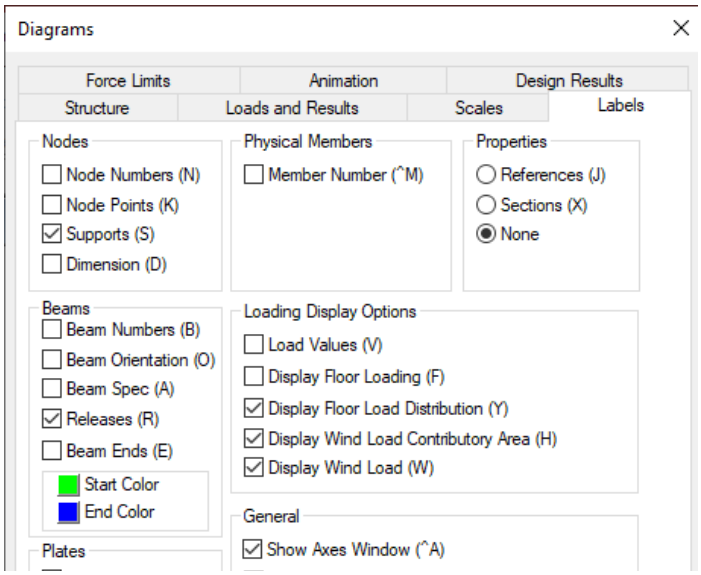

Figure 1-20 *Partial view of the shortcut keys displayed in the* ***Labels*** *tab of the* ***Diagrams*** *dialog box*

Table 1-1 *Various shortcut keys used in STAAD.Pro*

Keyboard Shortcut	Description
SHIFT+N	Displays node numbers
SHIFT+B	Displays beam numbers
SHIFT+S	Displays support icons
SHIFT+E	Displays member ends
SHIFT+O	Displays beam orientation
SHIFT+P	Displays plate numbers
SHIFT+L	Displays master slave node

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

SHIFT+V	Displays load values
SHIFT+C	Displays solid numbers
SHIFT+M	Displays material
SHIFT+I	Displays axes at origin

IMPORTING AND EXPORTING A MODEL IN STAAD.Pro

In STAAD.Pro, data can be imported from the following file types: DXF, QSE ASA, and CIS/2. From the CIS/2 file type, data such as member properties, material constants, member end conditions, support conditions, and loading information can be imported. To import data from any of the files, choose the **Import/Export** option from the **File** menu; the **Import/Export** screen will be displayed, as shown in Figure 1-21.

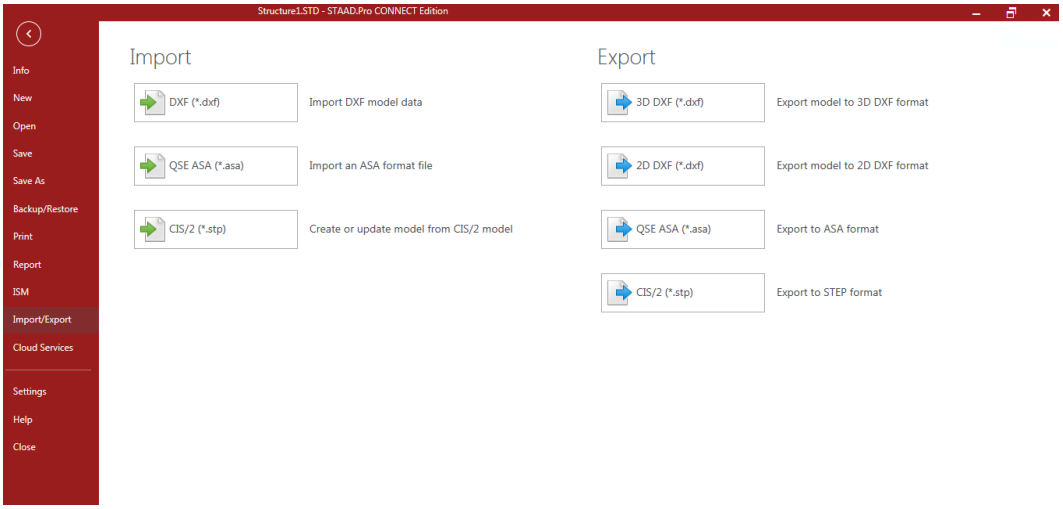

Figure 1-21 *Options in the* ***Import/Export*** *screen*

In this screen, select the required file format from the **Import** area by selecting the corresponding radio button. For example, choose the **DXF (*.dxf)** button to import the file from a .dxf file; the **Open** dialog box will be displayed. In this dialog box, browse to the required location, select the file, and choose the **Open** button; the **DXF Import** dialog box will be displayed, as shown in Figure 1-22.

In this dialog box you will define the axis of gravity in your STAAD.Pro model. If the axis system is same in the STAAD.Pro model and the dxf file then choose the **No Change** radio button. Now, choose the **OK** button; the file will be imported to STAAD.Pro.

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

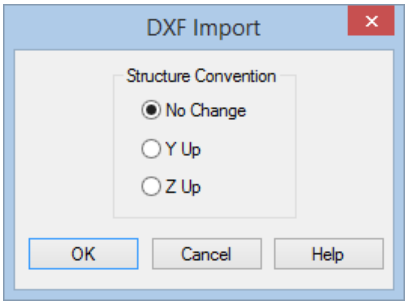

Figure 1-22 *The **DXF Import** dialog box*

Similarly, you can export the model to the following file types: 3D DXF, 2D DXF, QSE ASA, VRML, and CIS/2. From the 3D DXF file type, only line diagrams can be exported. From the CIS/2 file type, data such as member properties, material constants, member end conditions, support conditions, and loading information can be exported. To export data to any of the files, choose the **Import/Export** option from the **File** menu; the **Import/Export** screen will be displayed, as shown in Figure 1-21. In this screen, select the required file format by choosing the corresponding button. For example, choose the **3D DXF (*.dxf)** button to export the file in a .dxf format; the **Save As** dialog box will be displayed. In this dialog box, browse to the required location, enter the name in the **File name** edit box, and choose the **Save** button; the **DXF Export** dialog box will be displayed, as shown in Figure 1-23. In this dialog box, you will define the axis of gravity in your STAAD.Pro model. If you want the axis system to be same in the STAAD.Pro model and the dxf file, then select the **No Change** radio button and choose the **OK** button; the file will be exported from STAAD.Pro.

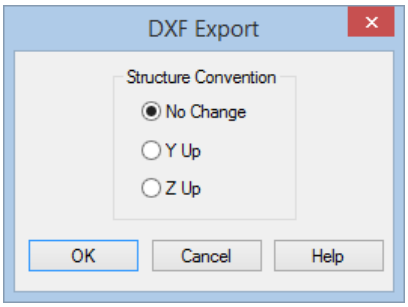

Figure 1-23 *The **DXF Export** dialog box*

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

Self-Evaluation Test

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

1. Which of the following tabs is used to open an existing STAAD Input file?

   (a) **Configure** (b) **Open**
   (c) **New** (d) None of these

2. Which one of the following tools is used for selecting the nodes, members, and elements of the structure at the same time?

   (a) **Node Cursor** (b) **Beam Cursor**
   (c) **Geometry Cursor** (d) None of these

3. The _____ shortcut key is used to display the load values.

4. The _____ shortcut key is used to display the member ends.

5. STAAD files are saved in the _____ format.

6. Job information is provided in the **Job Info** dialog box. (T/F)

7. In the **Postprocessing** mode, you can verify the analysis results both graphically and numerically. (T/F)

Review Questions

Answer the following questions:

1. Which of the following shortcut keys is used for displaying the master slave node?

   (a) SHIFT+L (b) SHIFT+N
   (c) SHIFT+B (d) SHIFT+V

2. Which of the following commands can be added using the **Miscellaneous Commands** option in the **Analysis and Design** tab?

   (a) Perform Analysis (b) Run Analysis
   (c) Input Width (d) Pre Analysis

3. Which of the following panels is not a part of the **Geometry** tab?

   (a) Selection (b) Structure
   (c) Node (d) Geometry Tools

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

4. Postprocessing Mode is used to model the structure. (T/F)

5. Status bar is present in the Ribbon tab. (T/F)

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

Answers to Self-Evaluation Test

1. c, **2.** b, **3.** SHIFT+V, **4.** SHIFT+E, **5.** .std, **6.** T, **7.** T