Chapter 2

Structural Modeling in STAAD.Pro

Learning Objectives

After completing this chapter, you will be able to:

- *Create structures using Editor*
- *Create structures using STAAD GUI*
- *Create structures using the Structure Wizard*

INTRODUCTION

Before erecting a structure, you need to run a stability check for the structure. To check the structural stability, you need to model the structure virtually. The virtual structural models comprise of various components such as beams, columns, walls, slabs, and so on. In STAAD.Pro, you can model a structure and then check its structural stability. A structural model in STAAD.Pro is represented as a line structure which consists of nodes and members. Before modeling a structure, you need to determine the nodes and the connection between them.

In STAAD.Pro, nodes are the joints which are capable to resist forces and moments. A node is located at the end of beams, columns, or plates. A node is always represented by a node number. To create nodes, you need to determine its coordinates in the XYZ space. After creating nodes, you can form members or plate elements by connecting the nodes. A member may be a column, beam, or truss, which is generated by connecting two nodes and is represented by a member number. Figure 2-1 shows the nodes and members in a portal frame structure.

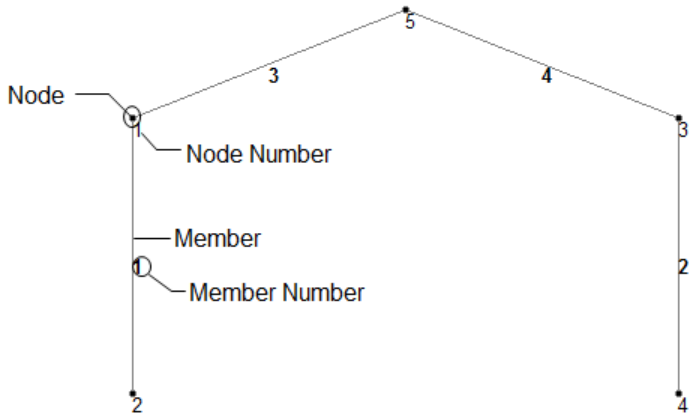


Figure 2-1 *Nodes and members in a portal frame structure*

A plate element is formed by connecting three or four nodes and is represented by a plate number. A plate element can be triangular or quadrilateral. These members and plate elements are represented by member number and element number.

In STAAD.Pro, you can create a structure model by using any of the following methods:

1) Using Editor
2) Using STAAD GUI
3) Using Structure Wizard
4) Building planner

These methods are discussed next.

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

STRUCTURAL MODELING USING STAAD EDITOR

In STAAD.Pro, the **STAAD Editor** window contains a list of commands required to model, analyze, and design a structure. These commands are the instructions related to the analysis and design of a structure and are executed consecutively. To open the **STAAD Editor** window, choose the **Command File** tool from the **Edit** panel of the **Utilities** tab. Figure 2- 2 shows the STAAD Editor window with default commands entered in it.

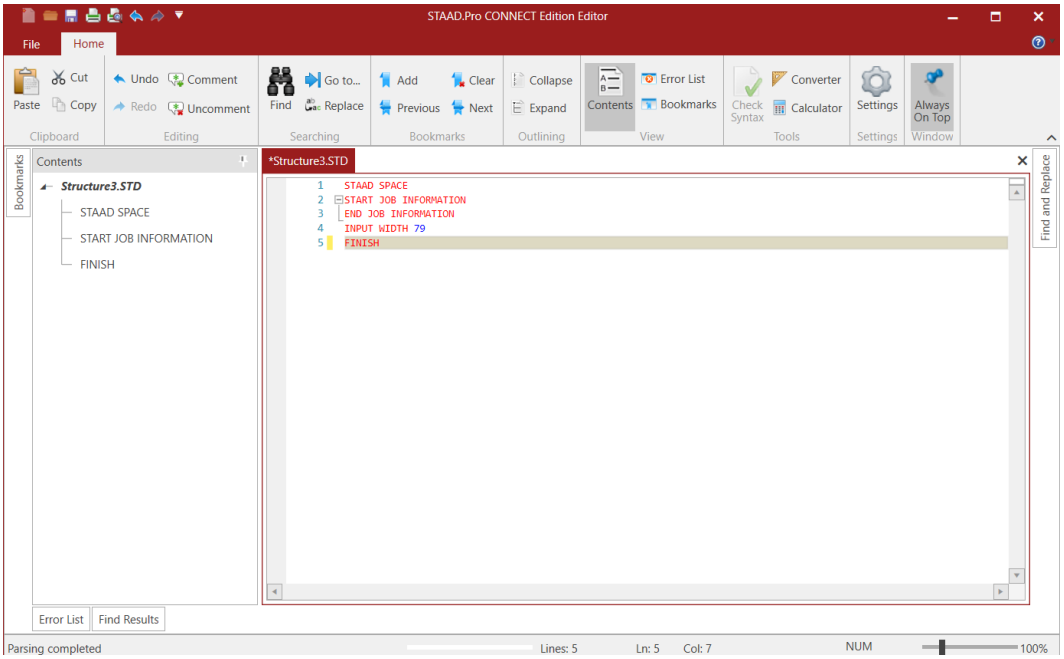


***Figure 2-2** The **STAAD Editor** window*

To start the structure geometry, the nodes must be created first and then other elements can be created like beams, plates, surfaces and solids. The command for creating the nodes is discussed next.

Creating Nodes

Nodes are created by specifying the X, Y, and Z coordinates of the joints in a structure. The command used for specifying the coordinates is **JOINT COORDINATES**. This command will be specified after specifying the unit command. The general format for specifying the coordinates is given below:

```
JOINT COORDINATES
n x y z
```

In the above command, **n** represents the node number and **x**, **y**, and **z** represent the coordinates of the joints nodes in a structure. To create nodes, first you need to specify the command **JOINT COORDINATES** in the **STAAD Editor** window. Next, you will specify the node number and coordinates for the first node. After specifying the first node, you need to put a semi-colon and then specify the node number and coordinates of the second node, and so on. After specifying

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

the command, choose the **Save** button from the toolbar in the **STAAD Editor** window and then close the window. You can view the node numbers of the created nodes by pressing SHIFT+N.



Note

*The **JOINT COORDINATE** command is not case sensitive. You can also write the initial three letters for the command. For example, enter **JOI COO** for this command. In case, you forget to choose the **Save** button before closing the window then the **Save** dialog box will automatically be displayed and you can save the changes by choosing the **Yes** button.*

You can also generate multiple nodes at a time by using the REPEAT and REPEAT ALL commands. These commands generate the nodes in a repetitive pattern.



Note

*The **REPEAT** command repeats the previous line of input n number of times with the increments specified in x, y, and z directions. The **REPEAT ALL** command repeats all the previously specified inputs. When you use **REPEAT** command after **REPEAT** or* **REPEAT ALL** *then the **REPEAT** command is only applicable for the last created node.*

The general format for specifying the REPEAT and REPEAT ALL commands is given below:

```
JOINT COORDINATES
n x y z
REPEAT n xi yi zi
REPEAT ALL n xi yi zi
```

In the above command, **n** represents the number of times the nodes will be repeated and $\mathbf{x_i}$, $\mathbf{y_i}$, and $\mathbf{z_i}$ represent increments in the **x**, **y**, and **z** coordinates.



Note

1. *In this chapter, you need to download the c02_staad_connect.zip file for the examples from https://www.cadcim.com. The path of the file is as follows: Textbook > Civil/GIS > STAAD.Pro > Exploring Bentley STAAD.Pro CONNECT Edition.*

2. *Before starting the examples, you need to create a folder with the name STAAD Examples in C: drive and then extract the downloaded zip folder to this folder.*

Example 1

In this example, you will create nodes for a portal frame structure. This structure will be created as a plane frame structure.

Steps required to complete this example are given below:

Step 1: Start STAAD.Pro CONNECT Edition and choose the **New** option from the left pane of the user interface screen; the **Model Information** page is displayed. In this page, specify the name *c02_staad_connect_ex1* in the **File Name** edit box and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box.

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

Step 2: Select **Analytical** in the **Type** area and **Metric** in the **Units** area. Choose the **Create** button from the left pane of the user interface screen; the file is loaded. Choose the **Command File** tool from the **Edit** panel of the **Utilities** tab.

Step 3: In this window, specify the commands, as shown in Figure 2-3.

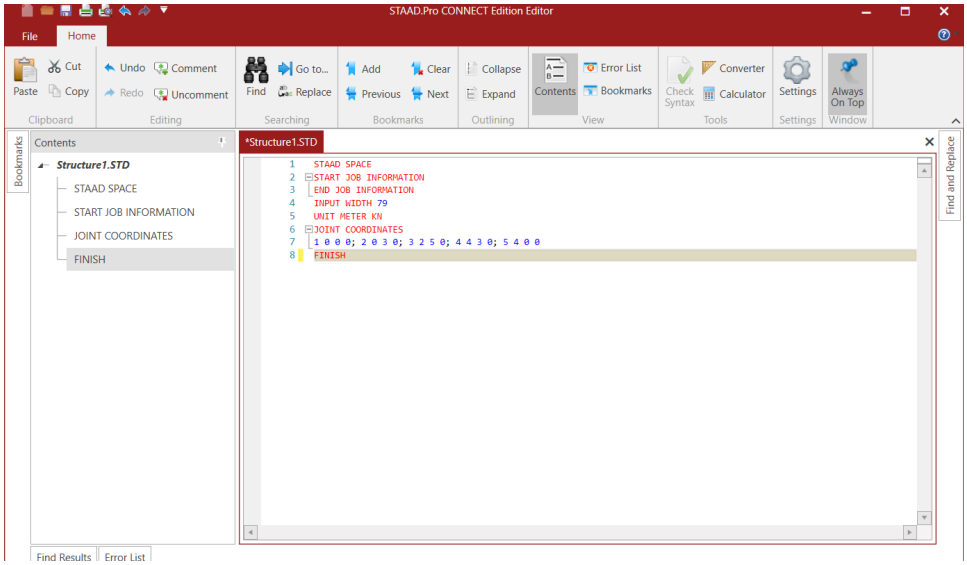


Figure 2-3 *Commands specified in the* ***Editor*** *window*

Step 4: Choose the **Save** button from the **File** menu in the **STAAD Editor** window and then close the STAAD Editor; the nodes are generated. Press SHIFT+N to view the node numbers, refer to Figure 2-4.

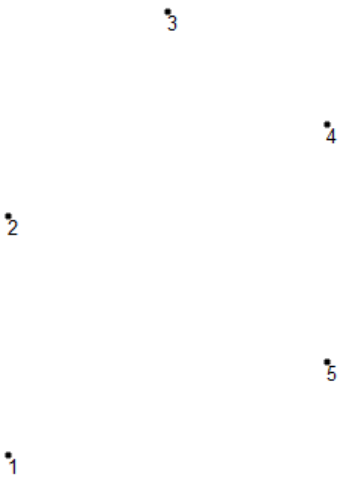


Figure 2-4 *Nodes created using commands*

Step 5: Choose the **Close** button from the **File** menu; the current file is closed.

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

Example 2

In this example, you will create the nodes for a structure using the **REPEAT** and **REPEAT ALL** commands. The structure will be created as a space frame structure.

Steps required to complete this example are given below:

Step 1: Start STAAD.Pro CONNECT Edition and choose the **New** option from the left pane of the user interface screen; the **Model Information** page is displayed. In this page, specify the name *c02_staad_connect_ex2* in the **File Name** edit box and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box.

Step 2: Select **Analytical** in the **Type** area and **Metric** in the **Units** area. Choose the **Create** button from the left pane of the user interface screen; the file is loaded. Choose the **Command File** tool from the **Edit** panel of the **Utilities** tab.

Step 3: In the **STAAD Editor** window, specify the commands, as shown in Figure 2-5.

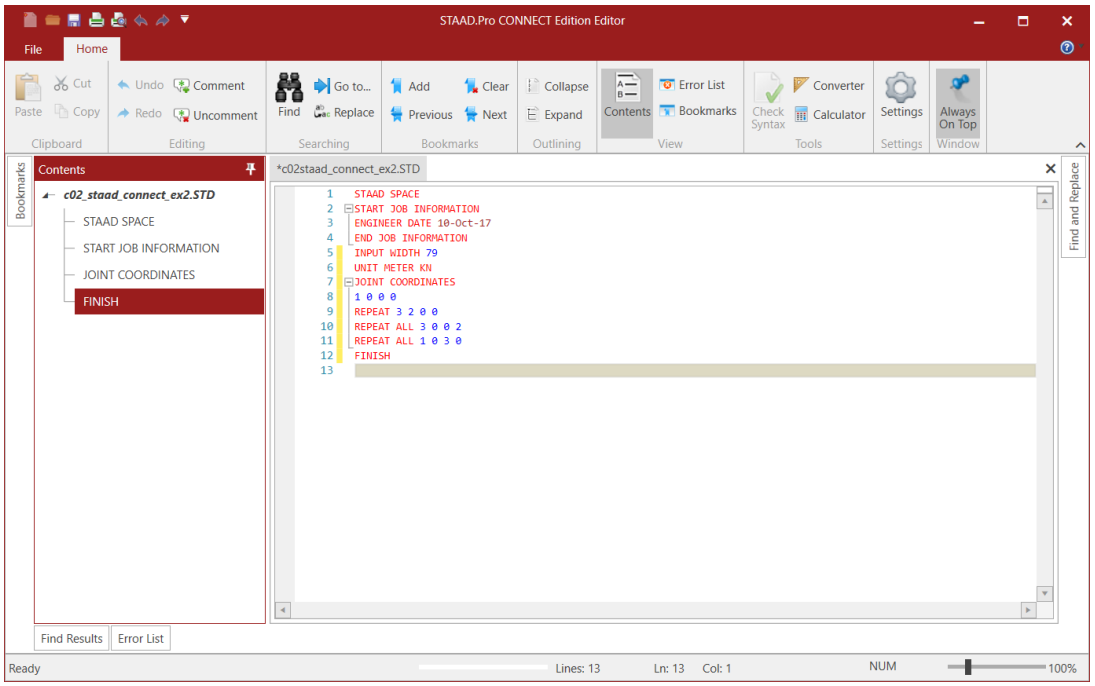


Figure 2-5 *Commands in the* ***STAAD Editor*** *window*

Step 4: Choose the **Save** button from the **File** menu in the **STAAD Editor** window and then close the window; the nodes are generated. Press SHIFT+N to view the node numbers, refer to Figure 2-6.

Step 5: Close the file by choosing the **Close** option from the **File** menu.

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



Note

*Whenever you will save a project in STAAD.Pro, the **REPEAT** and **REPEAT ALL** commands will be converted into comments and the coordinates of each node will be generated in the **STAAD Editor** window.*

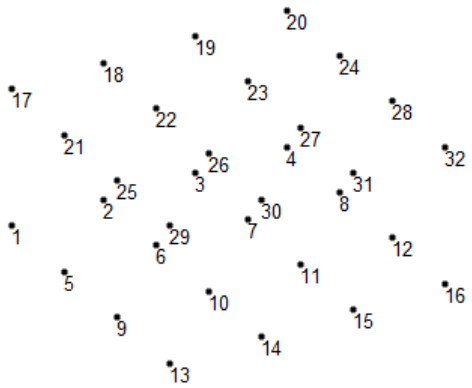


Figure 2-6 *Nodes of the space frame structure*

Creating Members

You can create members by specifying their start and end nodes. The command for creating member is **MEMBER INCIDENCES**. The general format for creating members is given below:

```
MEMBER INCIDENCES
m i j
```

Here, **m** represents the member number and **i**, **j** represents the start and end nodes, respectively. After creating nodes, you need to create members by specifying the connectivity between nodes. The **REPEAT** and **REPEAT ALL** commands can also be used for easy generation of the members. The general format to use the **REPEAT** and **REPEAT ALL** commands for creating members is given below:

```
MEMBER INCIDENCES
m i j
REPEAT  n i_i j_i
REPEAT ALL  n i_i j_i
```

In the above command, **n** represents the number of times, the previously created member will be repeated, $\mathbf{i}_i$ represents the member number increment and $\mathbf{j}_i$ represents the node number increment.

In this case, the **REPEAT** command will create the previously created member for specified number of times with specified increment in node numbers and member numbers. The **REPEAT ALL** command will create all the previously created members for specified number of times with specified increment in node numbers and member numbers.

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

Note

*To use the **REPEAT** and **REPEAT ALL** commands for creating members, you need to number the members in a consecutive order.*

Example 3

In this example, you will create the members to form a complete portal frame structure. The file used in this example is *c02_staad_connect_ex1.std*.

Steps required to complete this example are given below:

Step1: Choose the **Open** option from the left pane in the user interface of STAAD.Pro; the **Open** screen is displayed. In this screen, browse to the location: *C:\ STAAD Examples\c02_staad_connect* and select the *c02_staad_connect_ex1.std* file and then choose the **Open** button; the user interface screen is displayed with the nodes. Press SHIFT+N to view the node numbers, refer to Figure 2-7.

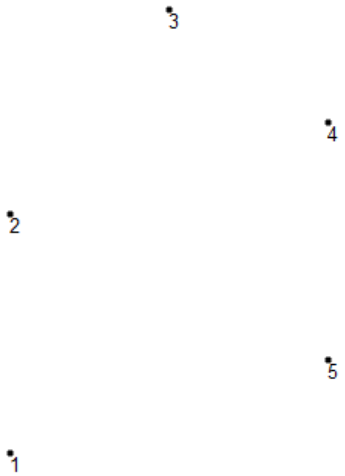


Figure 2-7 *Node numbers for the nodes*

Step 2: Invoke the **STAAD Editor** window and specify the commands for creating the members, as shown in Figure 2-8.

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

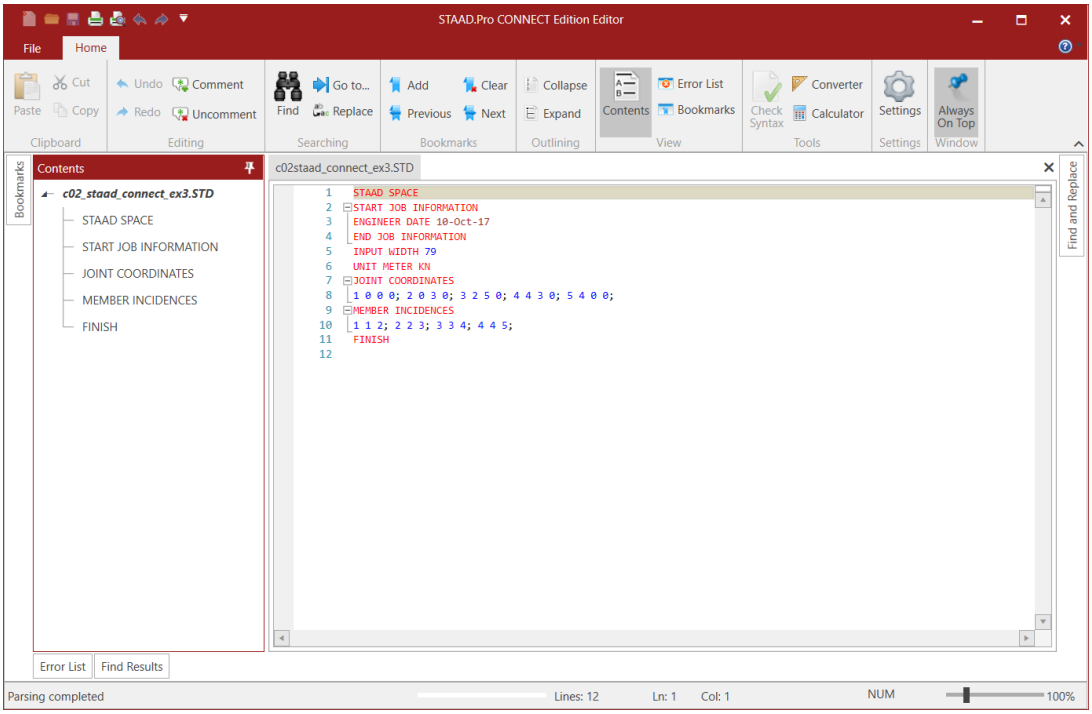


Figure 2-8 *Commands for creating the members in the* ***STAAD Editor*** *window*

Step 3: Choose the **Save** button from the **File** menu in the **STAAD Editor** window and then close the window; the members are generated. Press SHIFT+B to view the member numbers, refer to Figure 2-9.

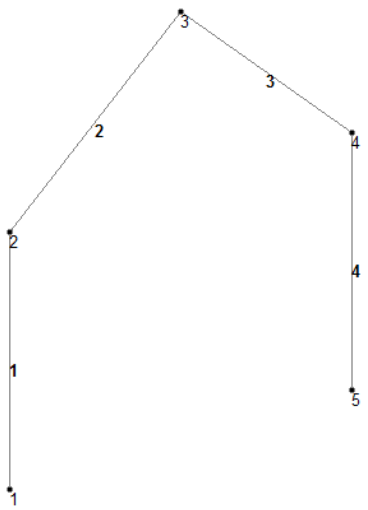


Figure 2-9 *Members created to complete the structure*

Step 4: Choose the **Save As** option from the **File** menu; the **Save As** page is displayed. In this page, specify the name *c02_staad_connect_ex3* in the **File name** edit box and save it at an appropriate location. Choose the **Save** button; the file is saved with the new name.

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

Step 5: Close the file by choosing the **Close** option from the **File** menu.

Example 4

In this example, you will create members using the **REPEAT** and **REPEAT ALL** commands to form a complete space frame structure. The file used in this example is *c02_staad_connect_ex2.std*.

Steps required to complete this example are given below:

Step 1: Open the *c02_staad_connect_ex2.std* file, as discussed in Example 3. Choose the **Geometry** tab and press SHIFT+N to view the node numbers.

Step 2: Invoke the **STAAD Editor** window and specify the commands given below after the **Joint Coordinates** command:

```
MEMBER INCIDENCES
1 1 17
```

Step 3: After specifying the above command, choose the **Save** button from the **File** menu and close the **STAAD Editor** window; the member 1 will be created, as shown in Figure 2-10. Click on working area. Press SHIFT+B to view the member number.

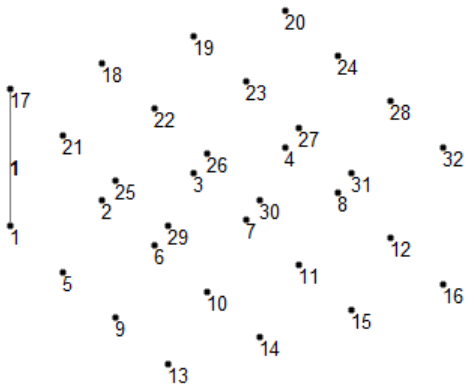


Figure 2-10 *Member 1 created using commands*

Step 4: Specify the **REPEAT** command after specifying the command in step 2 in the **STAAD Editor** window, as given below. After specifying the command, choose the **Save** button and close the **STAAD Editor** window; the members are created, as shown in Figure 2-11.

```
REPEAT 3 1 1
```

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

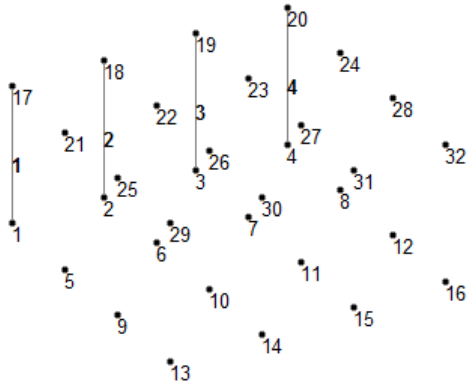


Figure 2-11 *Members created using the* ***REPEAT*** *command*

Step 5: Specify the next command given below in the **STAAD Editor** window. Next, choose the **Save** button and close the **STAAD Editor** window; the members are created, as shown in Figure 2-12.

```
REPEAT ALL 3 4 4
```

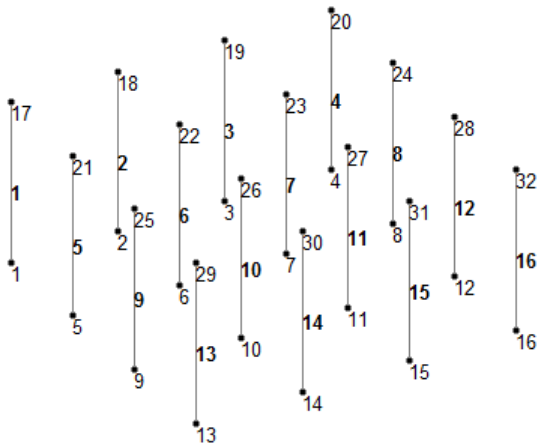


Figure 2-12 *Members created using the* ***REPEAT ALL*** *command*

Step 6: Next, create the horizontal members by specifying the command in the **STAAD Editor** which are given next.

```
17 17 18
REPEAT 2 1 1
REPEAT ALL 3 3 4
```

Step 7: Choose the **Save** button and close the **Editor** window; the members are created, as shown in Figure 2-13.

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

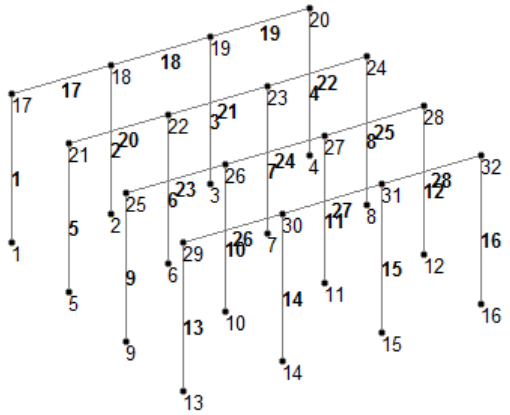


Figure 2-13 *Horizontal members created*

Step 8: Next, specify the command in the **STAAD Editor** window to create the remaining members as given below and the structure is completed, as shown in Figure 2-14.

```
29 17 21
REPEAT 3 1 1
REPEAT ALL 2 4 4
```

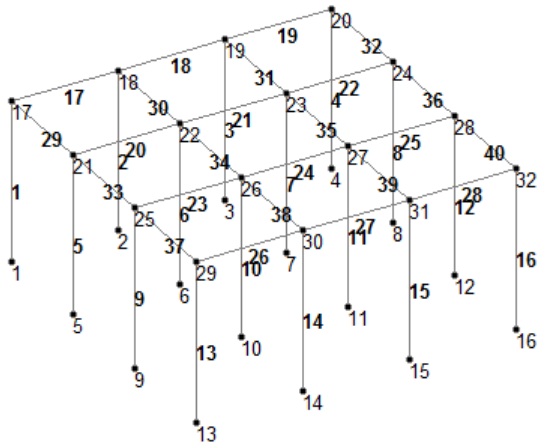


Figure 2-14 *The complete structure after creating members*

Step 9: Choose the **Save** option from the **File** menu and then close the **STAAD Editor** window.

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

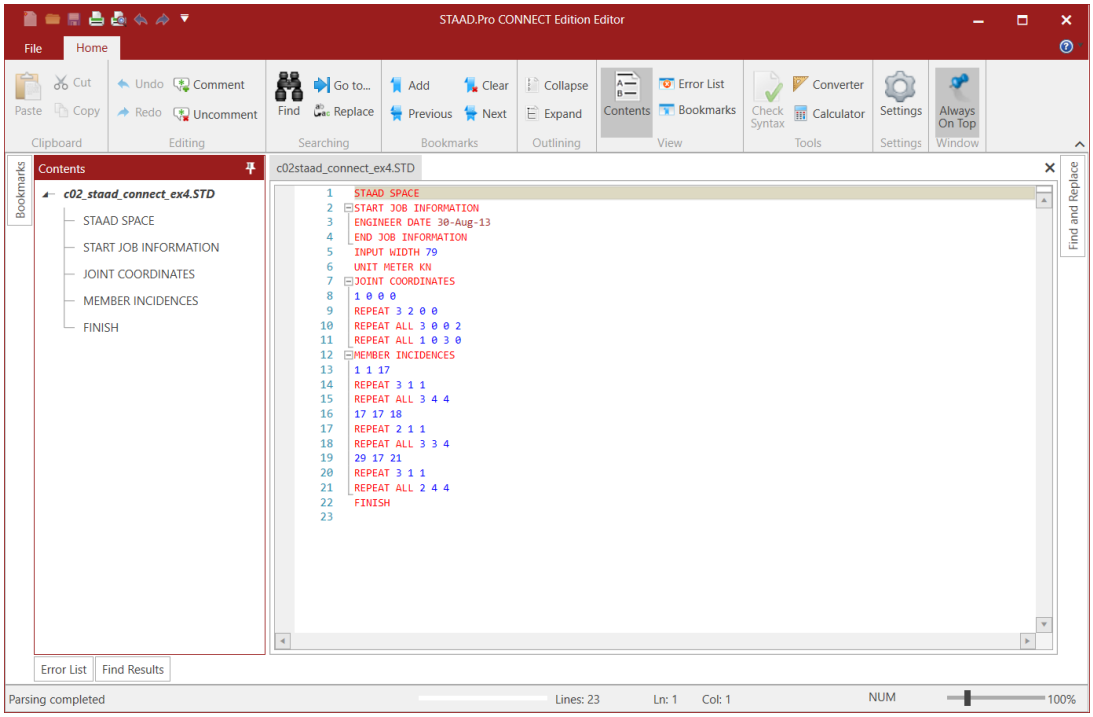


Figure 2-15 *Commands for creating members in the* ***STAAD Editor*** *window*

Step 10: Choose the **Save As** option from the **File** menu; the **Save As** page is displayed. In this page, specify the name *c02_staad_connect_ex4* in the **File Name** edit box and save it at an appropriate location.

The commands for creating members can also be written as given below:

```
MEMBER INCIDENCES
1 1 17 4
REPEAT ALL 3 4 4
17 17 18 19
REPEAT ALL 3 3 4
29 17 21 32
REPEAT ALL 2 4 4
```

Step 11: Choose the **Close** option from the **File** menu to close the open file.

In the first line of the above command, the span of the member 1 is from node 1 to node 17 and the new members will be generated with the member number increment and the node number increment as 1. Thus, the spans of the members numbered as 2, 3, and 4 are from nodes 2 to 18, 3 to 19, and 4 to 20, respectively.

Similarly, in the third and fourth line of the command, the members 17 to 19 and 29 to 32 will be formed.

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

Creating Plate Elements

In **STAAD.Pro**, a plate is a thin shell with multi-noded shape. It can have three or four nodes. A plate is used to model a floor slab, roof, or wall that does not need to be designed as a shear wall. For modeling a shear wall, you need to use the **Add Surface** tool. This tool will be discussed later in this chapter. But, to model a slab, roof, or wall, you need to create several plates which can be done by meshing. The process of meshing will be discussed later in this chapter.

To create a plate, first you need to create nodes. The process of creating nodes is already discussed earlier. Next, identify connectivity between the nodes and create the plates. The command for creating a plate is given next.

```
ELEMENT INCIDENCES
p i1 i2 i3 i4
```

In the above command, **p** represents plate number, $\mathbf{i_1}$, $\mathbf{i_2}$, $\mathbf{i_3}$, and $\mathbf{i_4}$ represent node numbers. The order of these node numbers can be clockwise or anti-clockwise

If the plate is 3-noded, then i_4 node is not needed. Note that, if you have created members in the project then the member numbers and element numbers must be distinct. You can also create plate elements by using the **REPEAT** and **REPEAT ALL** commands. The general format for using these commands is given below:

```
ELEMENT INCIDENCES
p i1 i2 i3 i4
REPEAT n ei ji
REPEAT ALL n ei ji
```

In the above command, n represents no. of times the plate will be repeated, $\mathbf{e_i}$ represents the element number increment and $\mathbf{j_i}$ represents the node number increment.

In this case, the **REPEAT** and **REPEAT ALL** commands will generate the previously created plate n number of times, with the specified plate and node number increment.

Example 5

In this example, you will create plate elements for the plane structure using the **REPEAT** and **REPEAT ALL** commands.

Steps required to complete this example are given below:

Step 1: Create a new file in STAAD.Pro with the name *c02_staad_connect_ex5* and then invoke the **STAAD Editor** window, as discussed earlier. Specify the commands to create nodes and members, as shown in Figure 2-16.

Step 2: Choose the **Save** button and close the **STAAD Editor** window; the nodes and members are created, as shown in Figure 2-17. Press Shift+N and Shift+B to view the no of nodes and beams respectively.

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

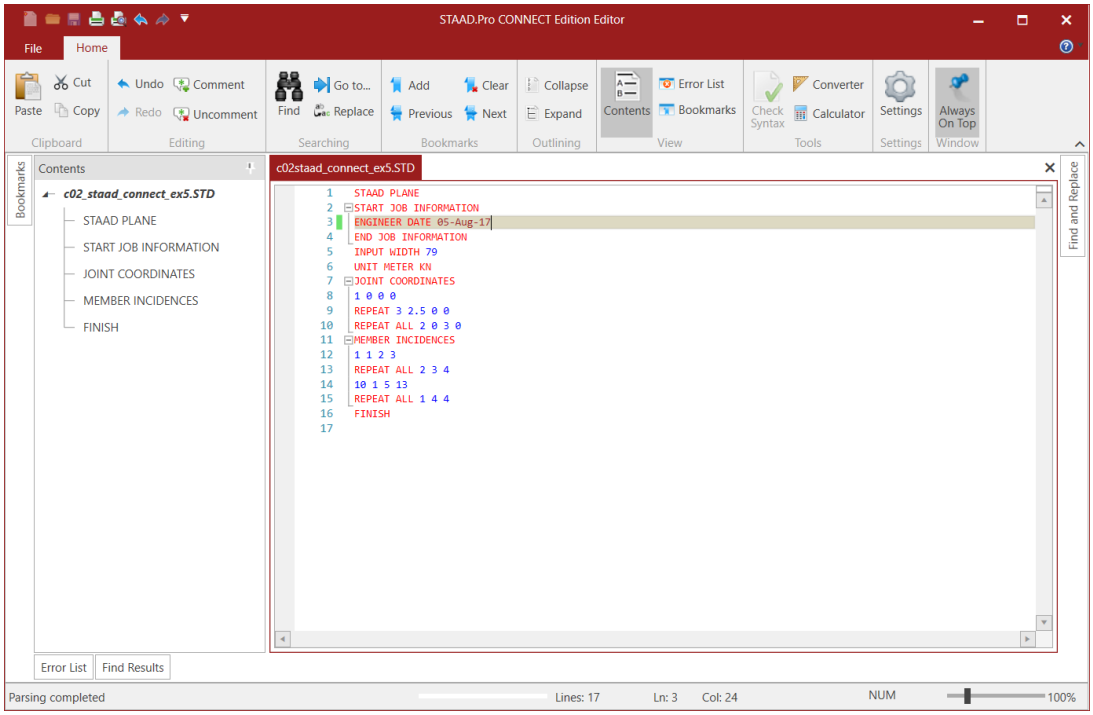


Figure 2-16 *Commands for creating nodes and members*

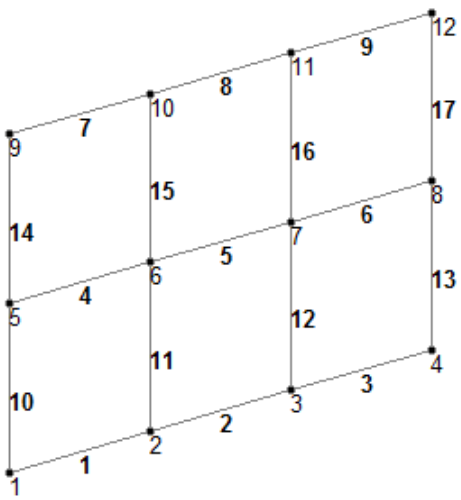


Figure 2-17 *Nodes and members created*

Step 3: Next, invoke the **STAAD Editor** window and specify the command for creating plates, as given below:

```
ELEMENT INCIDENCES
18 1 2 6 5
```

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

Step 4: Specify the commands in the **STAAD Editor** window as given below to create the remaining plates, as shown in Figure 2-18. Press SHIFT + P to view the plate numbers.

```
REPEAT 2 1 1
REPEAT ALL 1 3 4
```

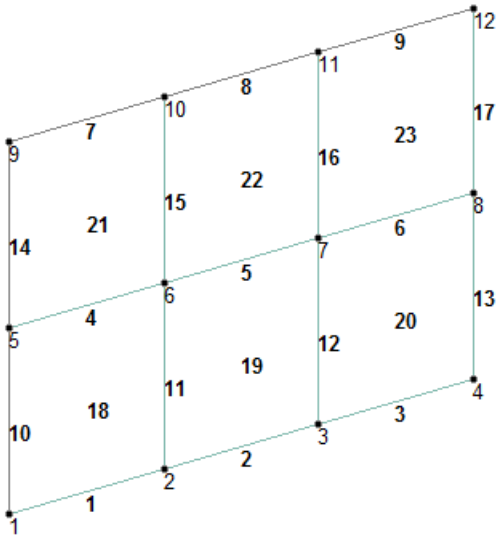


Figure 2-18 *Plates created using commands*

The commands used for creating plates can also be written as given below:

```
ELEMENT INCIDENCES
18 1 2 6 5 TO 20
REPEAT ALL 1 3 4
```

In the first line of the above command, the plate number 18 is formed by connecting the nodes 1, 2, 6, and 5. Plate number 19 and 20 are formed by connecting the node numbers 2, 3, 7, 6 and 3, 4, 8, 7. The node number and plate number increment is 1.

Step 5: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

Creating Solid Elements

In STAAD.Pro, solid elements are used to analyze the structures which include 3 dimensional stresses such as concrete gravity dams. A solid element is an eight-noded element and has three translational degrees of freedom per node. To create solid elements, first you need to create the nodes. The command for creating the solid elements is given below:

```
ELEMENT INCIDENCES SOLID
c i1 i2 i3 i4 i5 i6 i7 i8
REPEAT ni ei ji
REPEAT ALL ni ei ji
```

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

Here, **c** is the element number or solid number. $\mathbf{i}_1$ to $\mathbf{i}_8$ represents node numbers, n represents no. of times the solid will be repeated, $\mathbf{e}_i$ represents the element number increment and $\mathbf{j}_i$ represents the node number increment.

Example 6

In this example, you will create the solid block element in space frame by using the **STAAD Editor** commands.

Steps required to complete this example are given below:

Step 1: Create a new STAAD file with the name *c02_staad_connect_ex6.std* and invoke the **STAAD Editor** window from the **Utilities** panel of the **Utilities** tab. Specify the commands for creating nodes as given below:

```
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1.5 0 0; 3 3 0 0
4 0 0 2; 5 1.5 0 2; 6 3 0 2
REPEAT ALL 1 0 2 0
```

Step 2: Next, specify the commands for creating solid elements as given below:

```
ELEMENT INCIDENCES SOLID
1 4 5 11 10 1 2 8 7 TO 2
```

In the above command, 1 represents the solid number which will be created by connecting the eight nodes in a sequence and TO represents the solid number 2 which will be created automatically by connecting its corresponding nodes in the same manner.

Step 3: Choose the **Save** button and close the **STAAD Editor** window; the solid elements are created, as shown in Figure 2-19. Press SHIFT+C to view the solid element numbers.



Note

*You can view the element in the **Rendered View** window. The **Rendered View** window can be invoked by choosing the **3D Rendering** tool from the **Windows** panel of the **View** tab.*

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

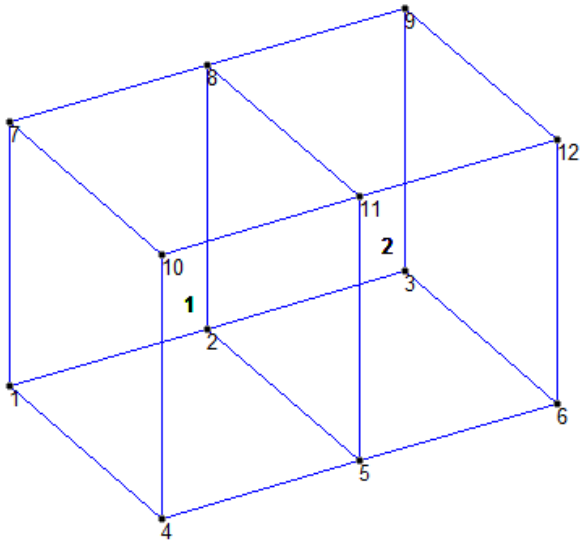


Figure 2-19 *Solid elements created using commands*

Step 4: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

STRUCTURAL MODELING USING STAAD GUI

You can model the structural geometry using the STAAD Graphical User Interface (GUI). The STAAD GUI comprises of various graphical tools that are used to generate the structure. Whenever you model the structure using the graphical tools, the **STAAD Editor** window will be automatically updated with the associated commands. Thus, the graphical model generation and the command file methods are interrelated. The changes made in the former will be reflected in the latter and vice-versa. It is essential for the users to learn both the methods so that they can easily make any change and update it in the model. There are various tools and methods available for modeling a structure. Some important methods are discussed next.

Creating Nodes and Members Using Snap Node/Beam Method

In the Snap Node/Beam method, the nodes and members are generated simultaneously. In this method, to create nodes and members, first you need to specify the grid and snap settings. The options related to the grid and snap settings will be available in the **Snap Node/Beam** window. You can invoke the **Snap Node/Beam** window by choosing the **Beam Grid tool** from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab.

In this window, the **Create** button is used to create new grids. The **Edit** button is used to configure the settings of an existing grid. The **Delete** button is used to remove an existing grid from the window. The **Copy** button is used to create a copy of an existing grid. You can change the name of the grid by using the **Rename** button. You can import a grid setting created in AutoCAD by using the **Import** button.

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

In the **Snap Node/Beam** window, you need to specify the settings of construction lines for creating nodes and members. In STAAD.Pro, there are three grid systems which can be used for specifying the settings of the construction lines. By default, the linear grid system is defined in the window, refer to Figure 2-20. In this window, the active grid system will be selected and highlighted. The three different grid systems available in STAAD.Pro are discussed next.

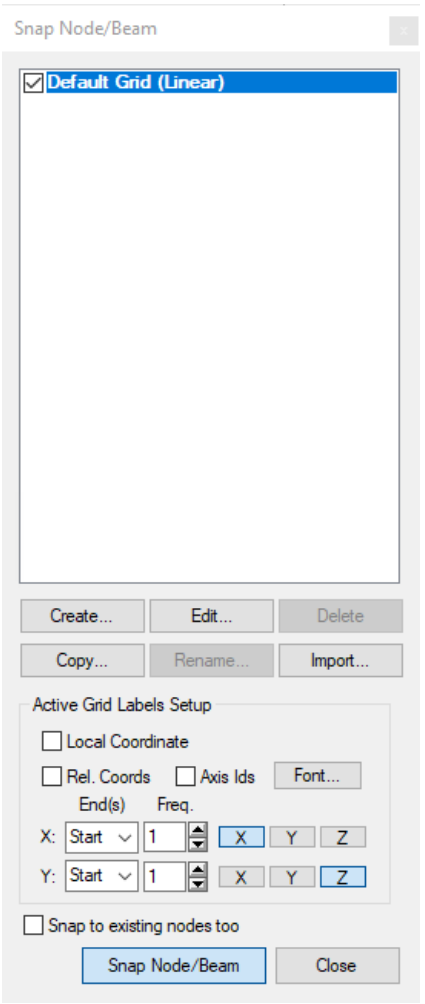


***Figure 2-20** The **Snap Node/Beam** window*

Linear Grid

In the linear grid system, the construction lines are perpendicular to each other. You can also create a new linear, radial, and irregular grid system. To create a linear grid system, choose the **Create** button in the **Snap Node/Beam** window; the **Linear** dialog box will be displayed, as shown in Figure 2-21.

In the **Linear** dialog box, the **Linear** option will be selected by default in the drop-down list available at the top of the dialog box. You can specify the name of the grid system in the **Name** edit box. Specify the required plane for the grid lines in which the structure will be drawn from the **Plane** area. For example, if a structure is to be drawn in the XY plane, select the **X-Y** radio button. Specify the angle of rotation of a plane about an axis in the corresponding edit box in the **Angle of Plane** area. Next, specify the coordinates of the origin of the grid in the **X**, **Y**, and **Z** edit boxes in the **Grid Origin** area. In the **Construction Lines** area, you can adjust the settings of the construction lines. To display the coordinates on the negative direction of X and Y axes, specify the required values in the **Left** column of the **X** and **Y** edit boxes by using the spinner. Specify the spacing between the grids in the **Spacing** edit box. To place the axes at an angle, specify the required value in the edit boxes corresponding to **Skew**. Figure 2-22 shows a linear grid system.

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

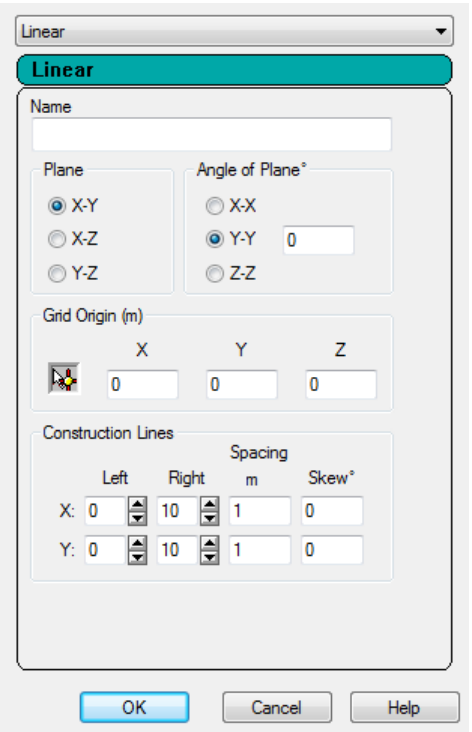


Figure 2-21 *The* ***Linear*** *dialog box*

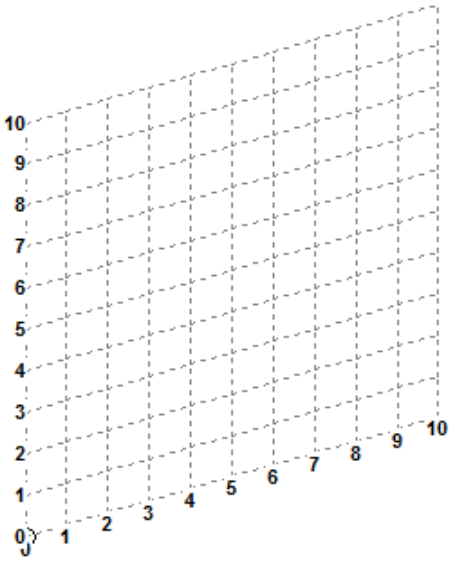


Figure 2-22 *The linear grid*

Radial Grid

In a radial grid system, the construction lines will appear in a spider-web style. In this grid style, you can create circular structures by creating small linear members. To create a radial grid system, choose the **Create** button from the **Snap/Node Beam** window; the **Linear** dialog box will be displayed, refer to Figure 2-21. Select the **Radial** option from the drop-down list available at the top; the **Radial** dialog box will be displayed and the options related to the radial grid system will be displayed, as shown in Figure 2-23.

In the **Radial** dialog box, you can specify the plane, angle of plane, and grid origin for the radial grid system in the same way as discussed for the linear grid system. In the **Construction Lines** area, specify the start angle of the grid in the **Start Angle** edit box. Specify the total angle of sweep in the **Sweep** edit box. Specify the inner and outer radius in the **Radius 1** and **Radius 2** edit boxes, respectively. Specify the number of bays in the **Bays** edit box. Figure 2-24 shows the radial grid system.

Note

Increasing the number of bays in the first ***Bays*** *edit box produces a better circular structure.*

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

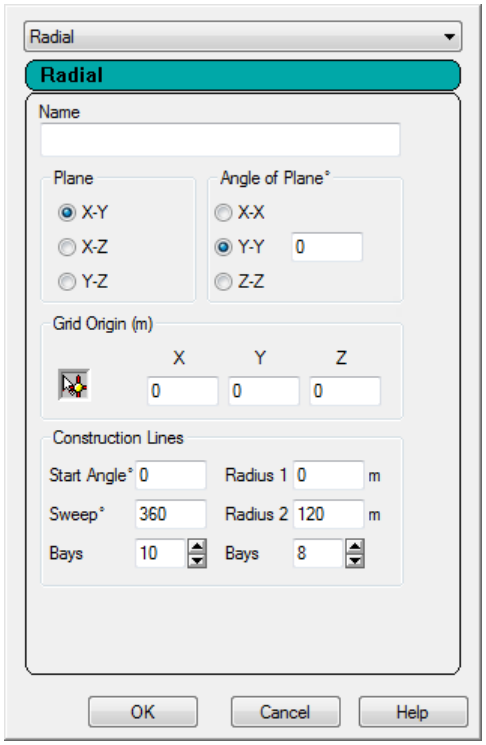


Figure 2-23 *The* ***Radial*** *dialog box*

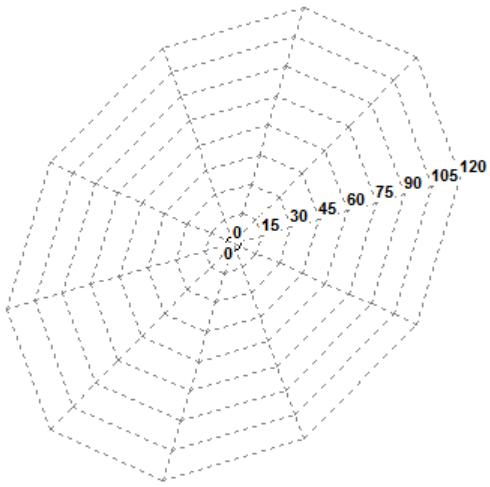


Figure 2-24 *The radial grid system*

Irregular Grid

In the irregular grid system, you can create the grid lines with unequal spacing. The process of creating an irregular grid is the same as discussed above for the linear and radial grid system. In this case, you can specify the relative grid distance in the X and Y edit boxes of the **Relative gridline distances** area in the **Irregular** dialog box, refer to Figure 2-25.

After creating the required grid system, choose the **OK** button; the **Snap Node/Beam** window will be displayed. Using this window, you can choose the desired grid system in which you want to work. For example, to create a linear structure, select the check box corresponding to the linear grid system. Next, choose the **Snap Node/Beam** button, if not chosen by default; a plus cursor will appear on the grids displayed in the main window area. Next, to create the nodes, click on the intersection point of the grids; the members will also be created along with the nodes. Figure 2-26 shows an irregular grid system.

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

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

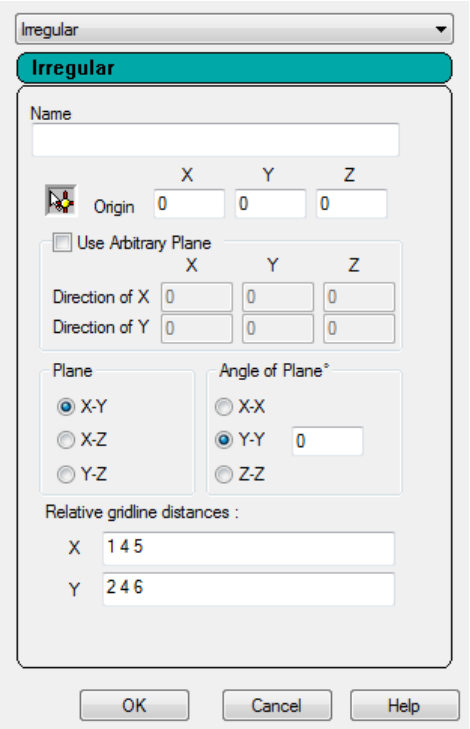


Figure 2-25 *The* ***Irregular*** *dialog box*

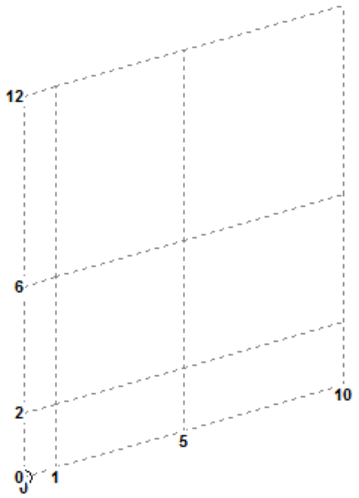


Figure 2-26 *The* ***Irregular Grid*** *System*

Example 7

In this example, you will create a structure, as shown in Figure 2-27, using the **Snap Node/ Beam** method.

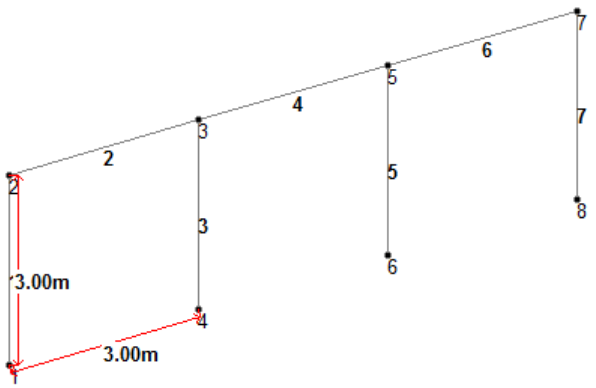


Figure 2-27 *Portal frame structure*

Steps required to complete this example are given next:

Step 1: Start STAAD.Pro CONNECT Edition and select the **New** option from the left pane; the **Model Information** page is displayed. In this page, specify the name *c02_staad_connect_ex7* in the **File Name** edit box and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box. Select **Analytical** from the **Type** area and **Metric** from the **Units** area. Next, choose the **Create** button from the left pane of the user interface screen; the file is loaded.

Step 2: Choose the **Beam Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab; the **Snap Node/Beam** window is displayed. In this window, choose the **Edit** button; the **Linear** dialog box is displayed.

Step 3: In this dialog box, ensure that spacing between grids is **1** and other default settings are retained. Choose the **OK** button to close the dialog box.

Step 4: Place the cursor at the origin and click; node1 is created and a member is attached to the cursor, refer to Figure 2-28.

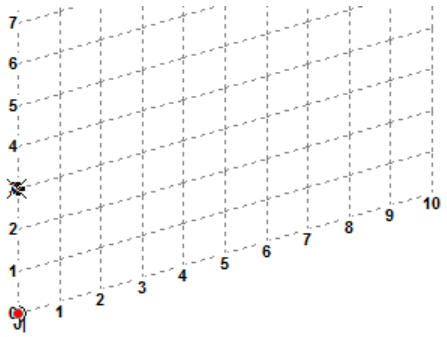


Figure 2-28 *Node 1 created at the origin*

Step 5: Next, move the cursor in the Y direction and click at (0,3) coordinates; the node 2 is created along with the member 1, refer to Figure 2-29.

Step 6: Now, move the cursor in the X direction and click at the coordinates (3,3); the node 3 is created along with the member 2, refer to Figure 2-29.

Step 7: Move the cursor in the negative Y direction and create node 4 at coordinates (3,0), along with member 3, refer to Figure 2-29.

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

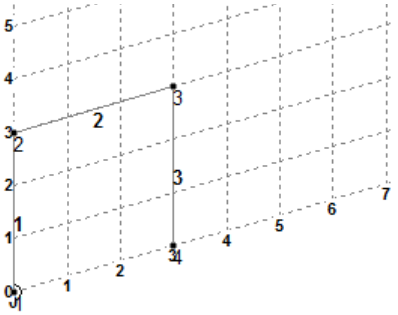


Figure 2-29 *Nodes 2, 3, and 4 created along with the members*

Step 8: Now, press the ESC key to deactivate the **Snap Node/Beam** mode.

Step 9: Again, choose the **Snap Node/Beam** button from the **Snap Node/Beam** window; the **Snap Node/Beam** mode will be activated and repeat the previous steps to create nodes 5, 6, 7, and 8 with members 4, 5, 6, and 7, refer to Figure 2-27.

Step 10: Next, close the **Snap Node/Beam** window; the structure is created, as shown in Figure 2-27.

Next, press SHIFT + N to show node numbers and SHIFT + B to show beam numbers.

Step 11: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

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

Creating Plate Elements Using the Snap Node/Plate Method

In the Snap Node/Plate method, you can create nodes and plates simultaneously. Before creating the nodes and the plates, first you need to specify the grid and snap settings. To do so, choose the **Quad Plate Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab; the **Snap Node/Plate** window will be displayed, as shown in Figure 2-30. In this window, you can specify the grid and snap settings, which are same as discussed above.

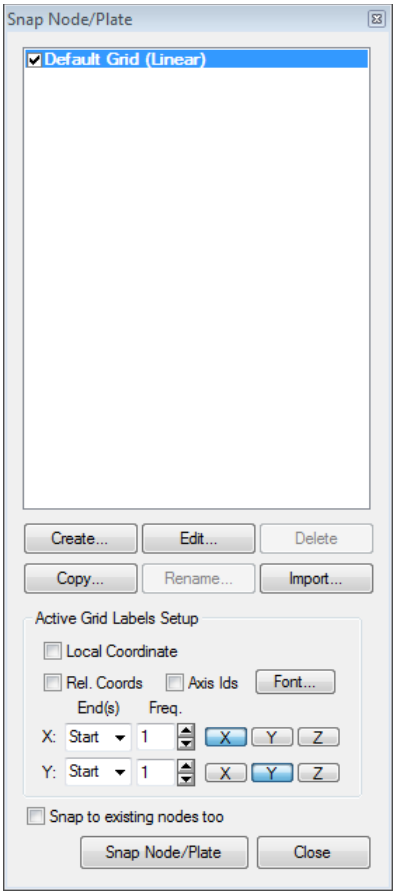


***Figure 2-30** The **Snap Node/Plate** window*

Note

*While working in an existing drawing, the **Snap Node/Plate** window will not be displayed. In that case, choose the **Quad Plate Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab.*

Now, choose the **Snap Node/Plate** button to create the plates. To specify the coordinates for the nodes, click at the appropriate places in the clockwise or anti-clockwise direction in the grid.

Note that the displayed **Snap Node/Plate** window can be used for the quadrilateral plates. To create triangular (3 noded) plates, choose the **Triangular Plate Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab; the **Snap Node/Plate** window for triangular plates is displayed. Now, using this window, you can create triangular (3 noded) plates.

Creating Solid Elements Using the Snap Node/Solid Method

You can create the solid elements by using the Snap Node/Solid method. To do so, choose the **Solid Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab; the **Snap Node/Solid** window is displayed. In this window, specify the required settings for the grid system and choose the **Snap Node/Solid** button; the plus cursor appears in the main window. Next, click at the appropriate places in the grid to create solid elements. Note that, if you click at the random places then the created solid element will be irregular in shape. So, you need to specify the nodes in a proper order. Figure 2-31 shows an irregular solid element which has been created by specifying the nodes randomly.

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

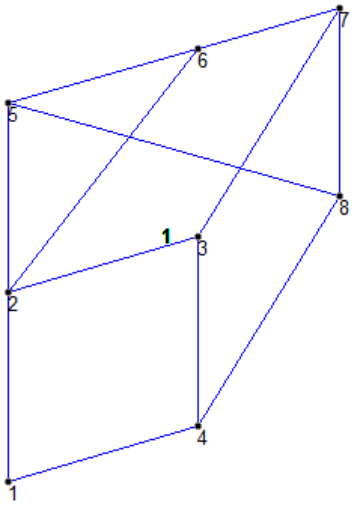


Figure 2-31 *An irregular solid element*

Example 8

In this example, you will create plate elements using the Snap Node/Plate method. Figure 2-32 shows the structure to be created in this example.

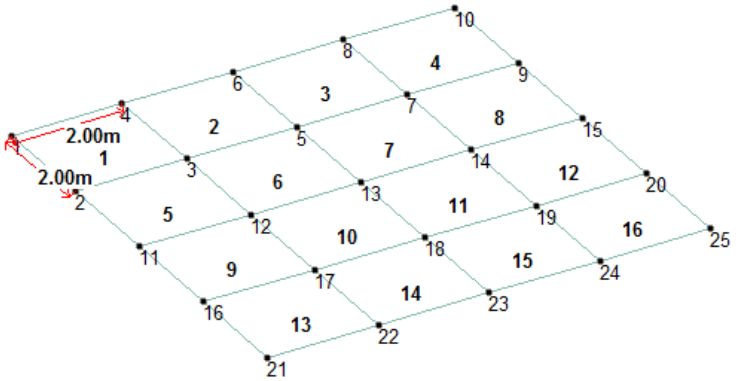


Figure 2-32 *The structure to be created for Example 8*

Steps required to complete this example are given below:

Step 1: Create a new file in STAAD.Pro with the name *c02_staad_connect_ex8.std* and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box. Select **Analytical** from the **Type** area and **Metric** from the **Units area**; choose the **Create** button; the user interface screen is displayed. Choose the **Quad Plate Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab. The **Snap Node/Plate** window is displayed in data area.

Step 2: In this window, choose the **Edit** button; the **Linear** dialog box is displayed. In this dialog box, select the **X-Z** radio button in the **Plane** area to activate the XZ plane.

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

Step 3: Next, enter **8** in the **X** and **Z** edit boxes in the **Right** column edit box/spinner of the **Construction Lines** area and choose the **OK** button to apply the changes.

Step 4: Now, place the cursor at the origin and click; the first node is created.

Step 5: Next, move the cursor in the Z direction and click at 2m; the node 2 is created.

Step 6: Move the cursor in the X direction and click at the coordinates (2,0,2); the node 3 is created.

Step 7: Again, move the cursor in the negative Z direction and click at the coordinates (2,0,0); node 4 and plate 1 is created, as shown in Figure 2-33. Press the ESC key to exit the tool selection. Press SHIFT+N and SHIFT+P to view the nodes and plates respectively.

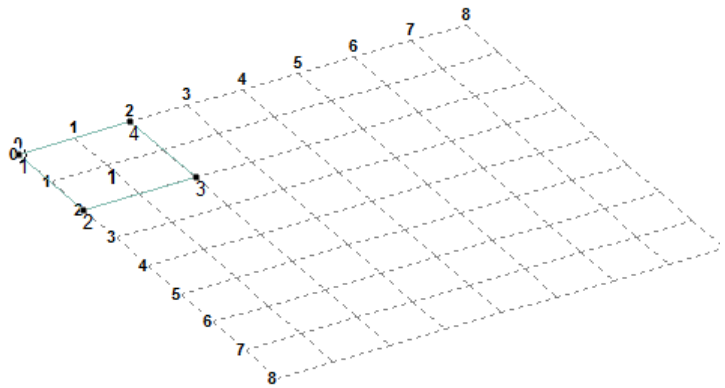


Figure 2-33 *Plate 1 created*

Step 8: Choose the **Snap Node/Plate** button and then and then click at (4,0,2); node 5 is created.

Step 9: Next, move the cursor in negative Z direction and click at the coordinate (4,0,0); node 6 is created.

Step 10: Move the cursor and click on node 2 and then move the cursor and click on node 3 respectively; plate 2 is created.

Step 11: Repeat the previous steps and create plates 2 to 16, refer to Figure 2-32.
Step 12: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

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

Example 9

In this example, you will create the solid block element using the Snap Node/Solid method. Figure 2-34 shows the concrete block to be created.

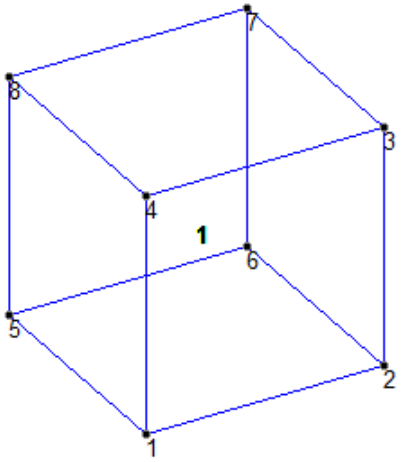


Figure 2-34 *The concrete block*

Steps required to complete this example are given below:

Step 1: Create a new file in STAAD.Pro with the name *c02_staad_connect_ex9.std* and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box. Select **Analytical** from the **Type** area and **Metric** from the **Units area** in the **Model Information** page and then, choose the **Create** button; the user interface screen is displayed. Choose the **Solid Grid** tool from the **Grids** drop-down in the **Structure** panel of the **Geometry** tab; the **Snap Node/Solid** window is displayed.

Step 2: Move the cursor at the coordinates (0, 0) and click to create node 1.

Step 3: Move the cursor in the X direction and click at the coordinates (4, 0) to create node 2.

Step 4: Similarly, click at the coordinates (4, 4) and (0, 4) to create nodes 3 and 4, respectively.

Step 5: Next, choose the **Edit** button from the **Snap Node/Solid** window; the **Linear** dialog box is displayed.

Step 6: In this dialog box, specify **-4** in the **Z** edit box under the **Grid Origin** area and then choose the **OK** button; the grid is moved to a new location, as shown in Figure 2-35.

Step 7: Now, repeat the steps 2 through 4 to create nodes 5, 6, 7, and 8; the solid element is created, refer to Figure 2-34. Press the ESC button to exit the command. Press SHIFT+N and SHIFT+C to view the node and solid number, respectively.

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

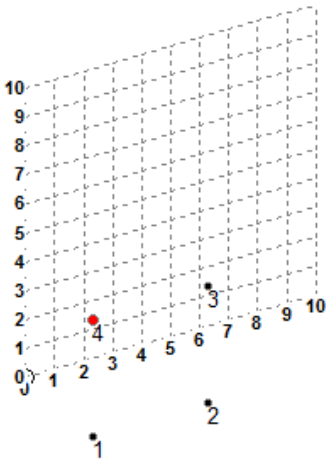


Figure 2-35 *Grids moved to the new location*

Step 8: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

STRUCTURAL MODELING USING THE STRUCTURE WIZARD

Structure Wizard contains pre-defined prototype models and templates such as truss models, frame models, surface models, solid models, and so on. Using these templates and prototype models, you can create a structure model by specifying the parameters such as length, width, height, radius, and so on. After creating the model in Structure Wizard, you can transfer or import it into STAAD.Pro to start a project. To access the Structure Wizard, choose the **Structure Wizard** option from the **Structure** panel of the **Geometry** tab; the **default.stp - StWizard** window will be displayed, as shown in Figure 2-36.

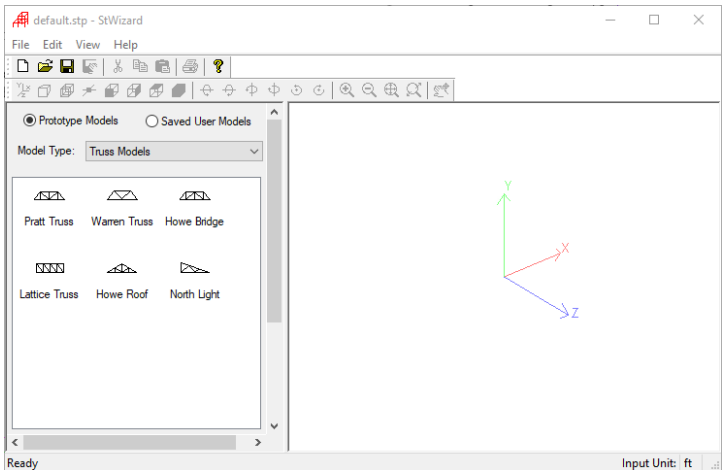


Figure 2-36 *The **default.stp - StWizard** window*

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

In this window, you can access both type of models, the prototype models and the models saved by the user. To access the prototype models, select the **Prototype Models** radio button. A list of different prototype models will be displayed in the **Model Type** drop-down list such as trusses, plates, solids, surfaces, and so on. To access the saved user models, select the **Saved User Models** radio button. Before generating a structure, you need to specify the units. To do so, choose the **Select Units** option from the **File** menu; the **Select Units** dialog box will be displayed. In this dialog box, select the appropriate unit and choose the **OK** button. The process of generating various types of structures is discussed next.

Truss Models

In the **Structure Wizard** window, you can create truss models. To do so, select the **Truss Models** option from the **Model Type** drop-down list of the **default.stp - StWizard** window; various prototype truss models will be displayed in the left pane of the window. In this pane, select an appropriate truss template. For example, select the **Howe Bridge** truss template and double-click on it; the **Select Parameters** dialog box will be displayed, as shown in Figure 2-37.

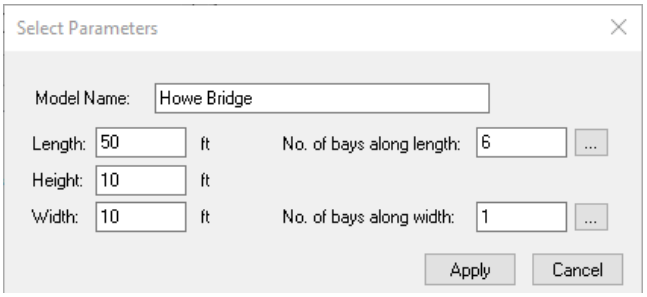


Figure 2-37 *The* ***Select Parameters*** *dialog box for* ***Howe Bridge***

In this dialog box, specify the name for the truss model in the **Model Name** edit box. Next, specify the length, width, height, and no. of bays in their corresponding edit boxes. Choose the **Apply** button; the structure will be generated and displayed in the **Structure Wizard** window. Figure 2-38 shows the Howe bridge model created with the default values.

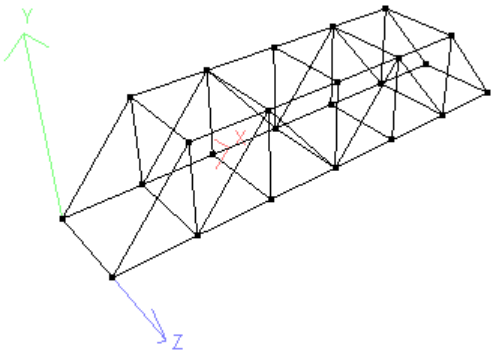


Figure 2-38 *The Howe Bridge model created*

Next, choose the **Merge Model with STAAD.Pro Model** option from the **File** menu; the **StWizard** message box will be displayed. In this message box, choose the **Yes** button; the **Paste Prototype Model** dialog box will be displayed, as shown in Figure 2-39.

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

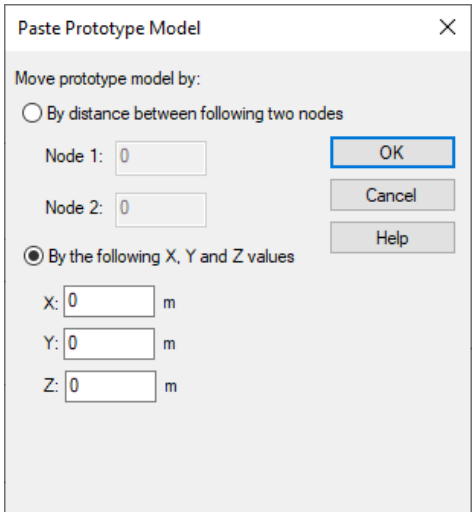


Figure 2-39 *The* ***Paste Prototype Model*** *dialog box for a new model*



Note

The ***Paste Prototype Model*** *dialog box will be displayed only when there is no existing structure in STAAD.Pro. If there is already a structure in STAAD.Pro, then the* ***Paste Prototype Model*** *dialog box will be displayed with the* ***Reference Pt*** *button added, as shown in Figure 2-40.*

In this dialog box, you can specify the coordinate values to move the prototype model. To move the model by a specified distance, select the **By distance between following two nodes** radio button; the **Node1** and **Node2** edit boxes will be enabled. Specify the desired values and choose the **OK** button to merge the model. To move the model to a specified coordinate, select the **By the following X, Y, and Z values** radio button; the **X**, **Y**, and **Z** edit boxes will be enabled. Specify the required values and choose the **OK** button to apply the changes. If you merge a new model with an existing one then the **STAAD.Pro CONNECT Edition** message box will be displayed, as shown in Figure 2-41.

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

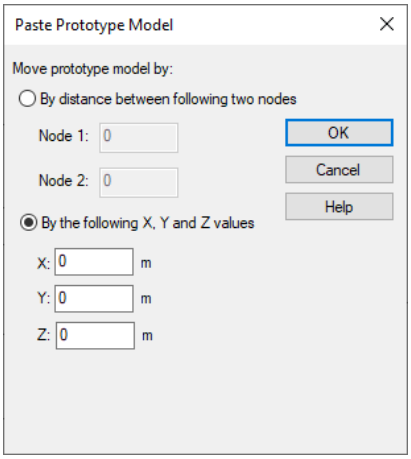


*Figure 2-40 The **Paste Prototype Model** dialog box for an existing model*

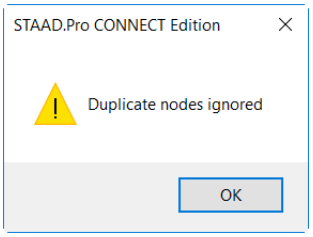


*Figure 2-41 The **STAAD.Pro CONNECT Edition** message box*

Choose the **OK** button to close the message box. You can choose the **Reference Pt** button in case of merging one model with other. In that case, you need to specify the node which will act as the reference point. Choose the **OK** button to close the dialog box. On doing so, all the members and nodes will be generated automatically.

Frame Models

You can create the frame models by selecting the **Frame Models** option from the **Model Type** drop-down list in the **Structure Wizard** window. On selecting this option, various prototype frame models will be displayed in the left pane of the window. Using these models, you can create different structures such as continuous beam, bay frame, cylindrical structure, circular beam, floor grid, and so on. In the left pane, double-click on the required prototype model; the **Select Parameters** dialog box for the selected model will be displayed, refer to Figure 2-42. In this dialog box, specify the desired values and choose the **Apply** button; the dialog box closes and the structure is created. Figure 2-43 shows a bay frame structure created with default values.

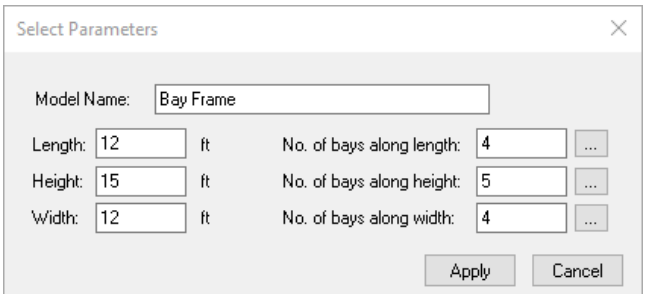


*Figure 2-42 The **Select Parameters** dialog box for the bay frame structure*

Next, you can transfer the model in STAAD.Pro by following the same method as discussed for the truss model.

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

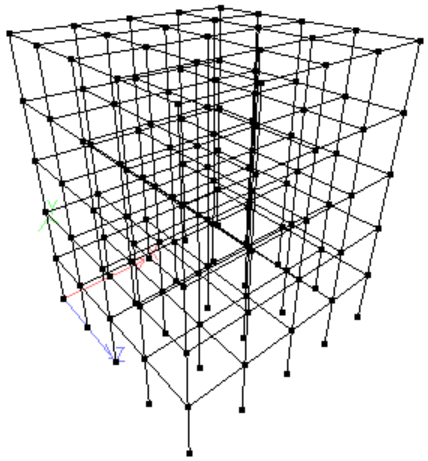


Figure 2-43 *The bay frame structure*

Surface/Plate Models

Using the **Structure Wizard** window, you can create surface/plates models like quadrilateral plates, cylindrical surfaces, polygonal plates with holes, spherical surface, cooling tower, and so on. To do so, select the **Surface/Plate Models** option from the **Model Type** drop-down list in the **Structure Wizard** window; various prototype models will be displayed in the left pane of the window. In the left pane, double-click on the required model; the dialog box related to the selected model will be displayed. Note that the dialog box displayed for each prototype model will be different. Figure 2-44 shows the dialog box for the **Polygonal Plate With Holes** model.

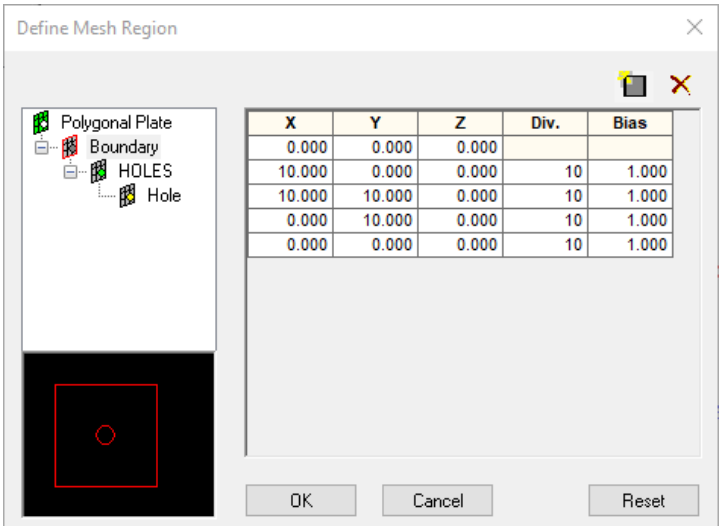


Figure 2-44 *The* ***Define Mesh Region*** *dialog box for the* ***Polygonal Plate With Holes*** *model*

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

In this dialog box, the parameters for the boundary will be displayed by default. You can consider the default values or can change them as per your requirement. Specify the locations for corners of the boundary, number of divisions for each side, and bias for each side division in their respective cells. You can also add a new row in the right pane by choosing the **Add New Row** button available at the top right in the dialog box. To delete a row, first select it and then choose the **Delete Row** button.

Next, click on the **Hole** sub node under the **HOLES** node in the left pane; the corresponding parameters will be displayed in the right pane. Using the options in this pane, you can create circular, polygonal, and elliptical holes. To do so, select the desired option from the **Region Type** drop-down list in the right pane of the dialog box; various parameters will be displayed in the left pane of the dialog box. The options displayed in the dialog box depend upon the options selected from the **Region Type** drop-down list. Specify the values as required and choose the **OK** button; the structure will be created and displayed in the right pane of the window. Figure 2-45 shows the polygonal plate with hole created using the default values. Now, merge the model in STAAD.Pro as discussed before.

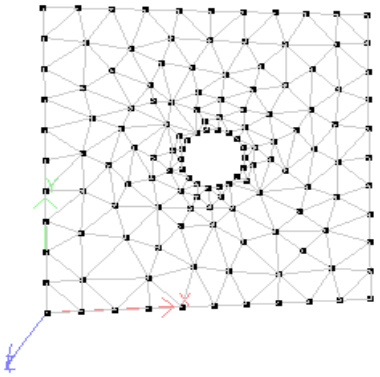


Figure 2-45 *Polygonal plate with hole*

Solid Models

You can also generate solid block models using the Structure Wizard. To do so, select the **Solid Models** option from the **Model Type** drop-down list; the **Solid Block** prototype model will be displayed in the left pane. Double-click on the **Solid Block** option; the **Select Meshing Parameters** dialog box will be displayed, as shown in Figure 2-46.

In this dialog box, specify the length and division along the axes in their corresponding edit boxes. Then, choose the **Apply** button; the solid block is created. Next, transfer it to STAAD.Pro by following the same process as discussed before. Figure 2-47 shows the solid block created using the default values.

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

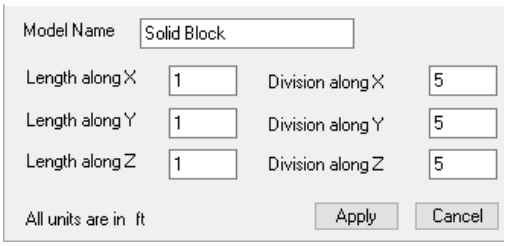


Figure 2-46 *Partial view of* ***Select Meshing Parameters*** *dialog box*

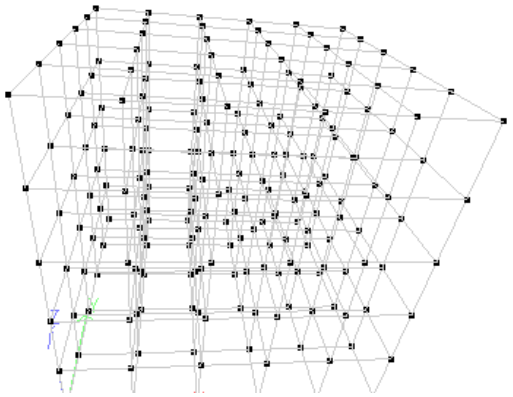


Figure 2-47 *The solid block created*

Composite Models

You can also generate composite models such as bunker or silo using Structure Wizard. To do so, select the **Composite Models** option from the **Model Type** drop-down list; the **Bunker** or **Silo** prototype model will be displayed in the left pane of the window. Double-click on the **Bunker** or **Silo** option; the **Select Meshing Parameters** dialog box will be displayed, as shown in Figure 2-48.

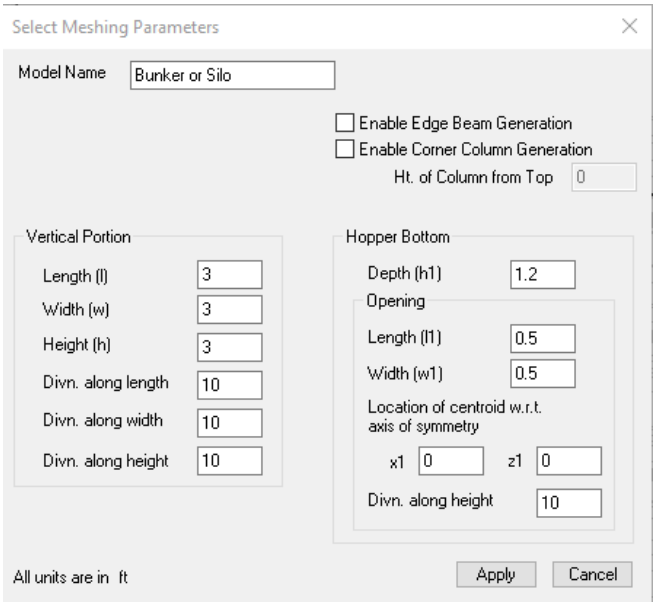


Figure 2-48 *The* ***Select Meshing Parameters*** *dialog box*

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

In this dialog box, you can define the geometry of the vertical portion. To do so, specify the required values in the corresponding edit boxes in the **Vertical Portion** area. Similarly, you can specify the depth of the hopper bottom in the corresponding edit boxes in the **Hopper Bottom** area. After specifying all the parameters, choose the **Apply** button; the structure will be created, as shown in Figure 2-49.

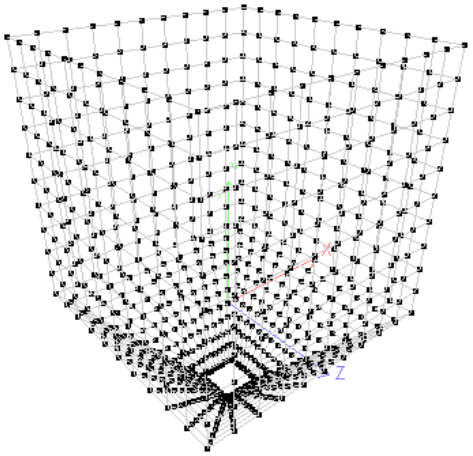


Figure 2-49 *The* ***Bunker*** *or* ***Silo*** *model*

Import CAD Models

Using the **Import CAD Models** option, you can import the AutoCAD models, which are saved in DXF format. To do so, select the **Import CAD Models** option from the **Model Type** drop-down list; the **Scan DXF** and **STAAD Model** options will be displayed in the left pane of the window. To import the AutoCAD models, double-click on the **Scan DXF** option in the right pane; the **Open** dialog box will be displayed. In this dialog box, browse to the required location, select the .dxf file, and then choose the **OK** button; the model will be displayed in the right pane. Using this option, you can import line, 3D-Polyline, and 3D-Face.

To import STAAD models, double-click on the **STAAD Model** option in the right pane; the **Open** dialog box will be displayed. In this dialog box, browse to the required location, select the file, and then choose the **OK** button; the file will be displayed in the window.

VBA Macro Models

Using the **VBA Macro Models** option, you can create models such as stadium roof. To do so, select the **VBA-Macro Models** option from the **Model Type** drop-down list; the **Stadium Roof** and **A Simple Tower** options will be displayed in the left pane of the window. Using these two options, you will be able to model a stadium roof and tower. The procedures to create a stadium roof and a tower are discussed next.

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

To create a stadium roof, select the **Stadium Roof** option from the left pane of the **default.stp - StWizard** window and drag it to the right pane; the **Stadium Roof** dialog box will be displayed, as shown in Figure 2-50. In this dialog box, you can specify the length, width, depth, number of panels along length and number of panels along width in their corresponding edit boxes. Next, choose the **OK** button; the stadium roof is created and displayed in the window. Figure 2-51 shows the stadium roof model created using default values.

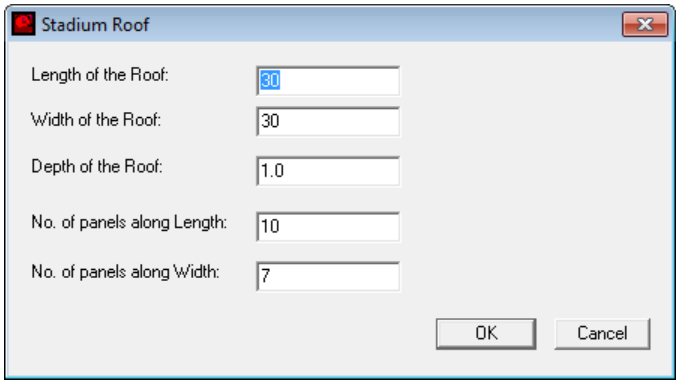


Figure 2-50 *The* ***Stadium Roof*** *dialog box*

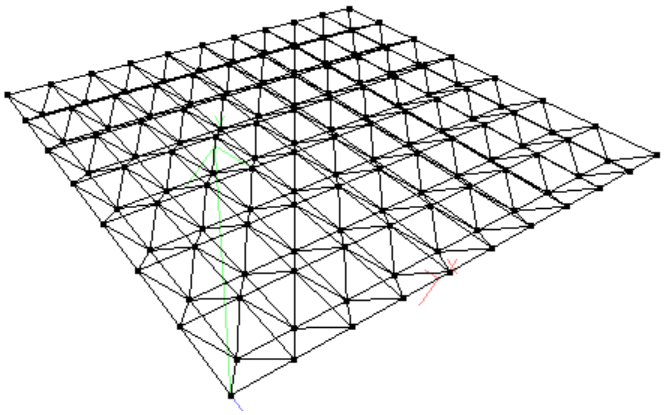


Figure 2-51 *Stadium roof created with default values*

To create a tower, select the **A Simple Tower** option from the left pane of the **default.stp - StWizard** window and drag it to the right pane; the **Tower Parameters** dialog box will be displayed, as shown in Figure 2-52. In this dialog box, you can specify the base dimension, top dimension, height, and number of bays along height in their corresponding edit boxes. Next, choose the **OK** button; the dialog box is closed and the tower will be created. Figure 2-53 shows a simple tower modeled using default values.

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

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

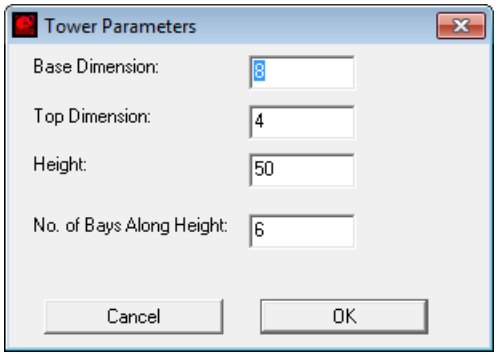


Figure 2-52 *The* ***Tower Parameters*** *dialog box*

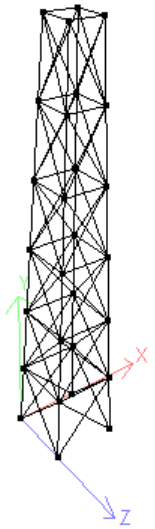


Figure 2-53 *A simple tower created using default values*

Example 10

In this example, you will create Howe Bridge in Structure Wizard and transfer it to STAAD.Pro. Figure 2-54 shows the structure of Howe Bridge to be created in this example.

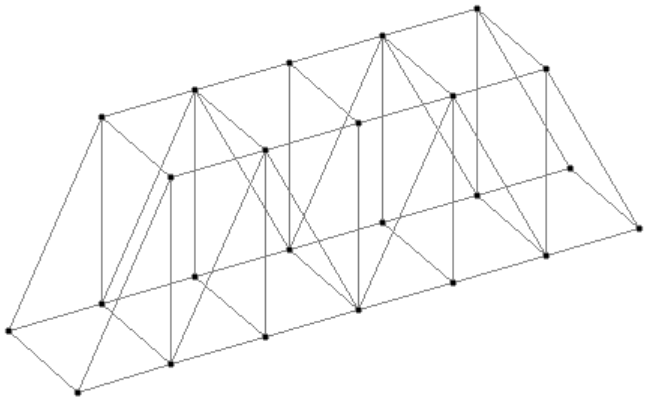


Figure 2-54 *The Howe Bridge model*

Steps required to complete this example are given below:

Step 1: Create a new file in STAAD.Pro with the name *c02_staad_connect_ex10.std* and browse to the location *C:\STAAD Examples\c02_staad_connect* by clicking the button next to the **Location** edit box. Select **Analytical** from the **Type** area and **Metric** from the Units area in the **Model Information** page.Next, choose the **Create** button; the user interface screen is displayed. Select

Structure Wizard from the **Structure** panel of the **Geometry** tab; the **default.stp - StWizard** window is displayed.

Step 2: In this window, ensure that the **Truss Models** option is selected in the **Model Type** drop-down list.

Step 3: Double-click on the **Howe Bridge** option in the left pane; the **Select Parameters** dialog box is displayed.

Step 4: In this dialog box, specify the length, height, width, and number of bays parameters, as shown in Figure 2-55.

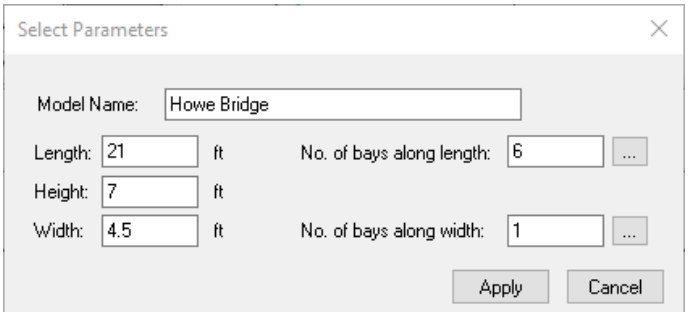


Figure 2-55 *The* ***Select Parameters*** *dialog box*

Step 5: Choose the **Apply** button; the model is created and displayed in the window, as shown in Figure 2-56.

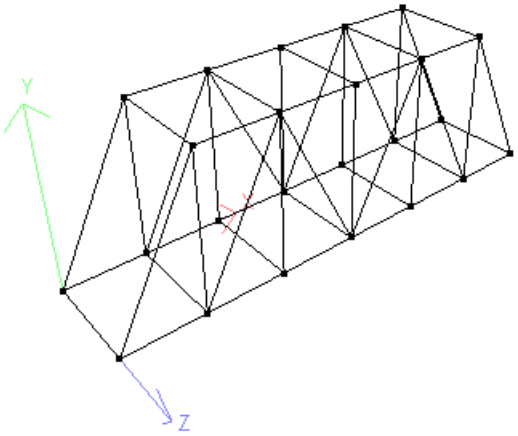


Figure 2-56 *The Howe bridge model*

Step 6: Choose the **Add/Paste Model in STAAD.Pro** option from the **Edit** menu in the **StWizard** window; the **StWizard** message box appears.

Step 7: Choose the **Yes** button; the **Paste Prototype Model** dialog box is displayed. Choose the **OK** button; the model is displayed in the main area of the STAAD.Pro window, refer to Figure 2-54.

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

Step 8: Choose the **Save** option from the **File** menu to save the file and then close it by choosing the **Close** option from the **File** menu.

Self-Evaluation Test

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

1. The command used for creating the nodes is _____.

2. The _____ command repeats the previous line of input specified in the **STAAD Editor** window.

3. In the _____ grid system, the construction lines are perpendicular to each other.

4. The command used for creating solid elements is _____.

5. In STAAD.Pro, plates are used to model slabs and shear walls. (T/F)

6. You can create nodes and members simultaneously using the Snap/Node Beam method. (T/F)

7. In STAAD.Pro, the Structure Wizard contains the prototype models. (T/F)

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

Review Questions

Answer the following questions:

1. Which of the following commands is used to create members?

   a) **Joint Coordinates** b) **Element Incidences**
   c) **Member Incidences** d) None of these

2. Which of the following commands repeats all the previously defined inputs?

   a) **Finish** b) **Repeat**
   c) **Repeat All** d) All of the above

3. Which of the following methods is used to create nodes and members simultaneously?

   a) Snap/Node Plate b) Snap/Node Beam
   c) Snap/Node Solid d) None of these

4. Which of the following grid styles can be used to create circular structures?

   a) **Radial** b) **Linear**
   c) **Irregular** d) All of these

5. In STAAD.Pro, you can create members before creating the nodes. (T/F)

6. Solid elements are the eight-noded elements. (T/F)

7. The **Truss Models** template contains various prototype truss models. (T/F)

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

Answers to Self-Evaluation Test

1. **Joint Coordinates**, 2. **Repeat**, 3. Linear, 4. **Element Incidences Solid**, 5. T, 6. T, 7. T

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