

Chapter 3

Creating Structures

Learning Objectives

After completing this chapter, you will be able to:

- *Add members*
- *Configure member settings*
- *Add stairs*
- *Add ladders*
- *Add railings*
- *Add footings*
- *Modify the structural members*

INTRODUCTION

In this chapter, you will learn to create a structural layout that is an integral part of an industrial plant. To create a structural layout, first you need to create a new drawing file under **Plant 3D Drawing** node using the **PROJECT MANAGER**. The procedure to create a new drawing has been discussed in Chapter 2. In this chapter, the tools used to create a structural layout will be discussed. These tools are available in the 3D Piping workspace.



Note

*If the file is opened in a workspace other than 3D Piping, you need to switch to this workspace. To do so, select the **3D Piping** option from the **Workspace** drop-down list in the **Quick Access Toolbar**.*

CREATING A GRID

Ribbon: Structure > Parts > Grid



Grid

A grid helps you to place structural components at appropriate location. To create a grid, choose the **Grid** tool from the **Parts** panel in the **Structure** tab; the **Create Grid** dialog box will be displayed, as shown in Figure 3-1.

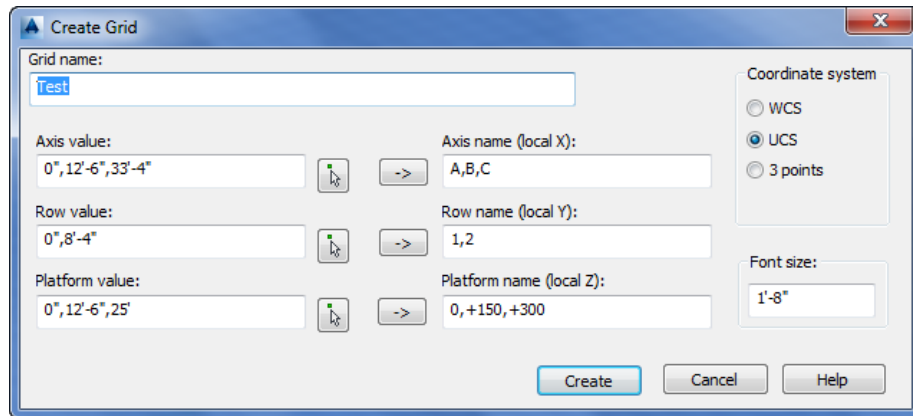


Figure 3-1 The *Create Grid* dialog box

In this dialog box, enter the name of the grid in the **Grid name** edit box. Next, specify the type of coordinate system to be used to set the orientation of the grid. You can specify the coordinate system by selecting anyone of the radio buttons available in the **Coordinate system** area. The **WCS** radio button is used to define the coordinate system of the grid using the world coordinate system. The **UCS** radio button is used to define a user coordinate system. If you select the **3 points** radio button, you will be prompted to specify the origin point. After specifying the origin point, you need to specify one point on the X-axis, one point on XY plane, and then one point on the Z-axis. These three points specify the directions of three axes in a grid. After specifying the coordinate system, enter values for the axis points in the **Axis value** edit box. You can use @ symbol in order to enter a value relative to the preceding value. You can also use the **Pick point** button adjacent to this edit box to directly specify the points in the workspace. Next, specify names of the axes in the **Axis name (local X)** edit box. Similarly,

enter values for the row points in the **Row value** edit box. Also, enter the names of the row points in the **Row name (local Y)** edit box. Specify the platform value points (in Z-direction) and their names in the **Platform value** and **Platform name (local Z)** edit boxes, respectively. Next, enter the font size in the **Font Size** edit box for the names displayed in the grid. Choose the **Create** button; a grid will be created in the model space, refer to Figure 3-2.

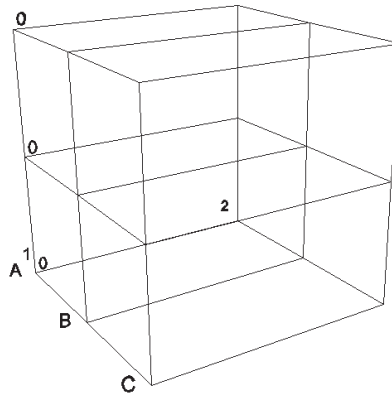


Figure 3-2 A 3D grid created

Editing Grids

To edit a grid, invoke the **Structure Edit** tool from the **Modify** panel and select the grid; the **Edit grid** dialog box will be displayed. The options in this dialog box are similar to the options in the **Create Grid** dialog box. Set the parameters in this dialog box as required and choose the **OK** button; the grid will be modified accordingly.

SETTING THE REPRESENTATION OF THE STRUCTURAL MEMBER

You can set the representation of the structural members as required by using the tools available in the **Line Model** drop-down of the **Parts** panel, refer to Figure 3-3. The different representations that you can set for the structural members are discussed next.

The **Line Model** is the default representation set for the member. In this mode, the members are represented as lines along with a symbol, as shown in Figure 3-4. This makes it easier to select the insertion point of a new member. In the **Symbol Model** mode, the members are represented as symbols, as shown in Figure 3-5. In the **Outline Model** mode, the members are represented as structure outlines, as shown in Figure 3-6. The **Shape Model** mode gives a real time look to the member with fillets added to the structural outline, as shown in Figure 3-7.

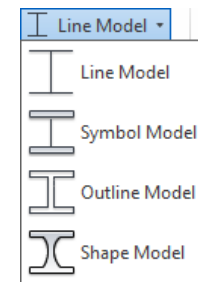


Figure 3-3 The **Line Model** drop-down

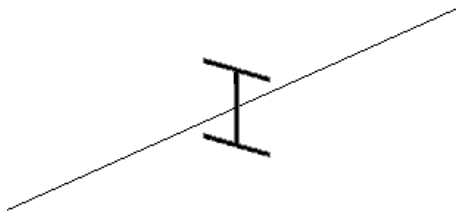


Figure 3-4 The member in the Line Model mode

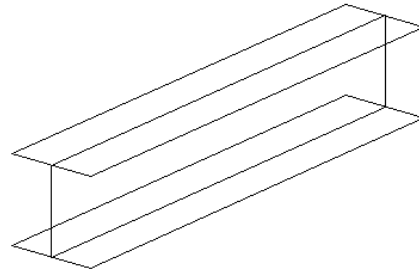


Figure 3-5 The member in the Symbol Model mode

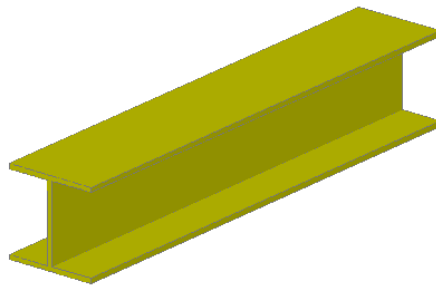


Figure 3-6 The member in the Outline Model mode

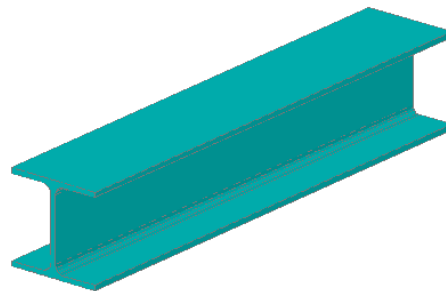


Figure 3-7 The member in the Shape Model mode

ADDING MEMBERS

In AutoCAD Plant 3D, a column or a beam is referred to as a structural member. A structural member can be placed in the plant 3D layout by specifying its start point and endpoint. Before adding a structural member, you need to specify the member settings such as shape, size, orientation, material standard, and so on. To add a member, choose the **Member Settings** tool from the **Settings** drop-down in the **Parts** panel of the **Structure** tab, refer to Figure 3-8; the **Member Settings** dialog box will be displayed, as shown in Figure 3-9.

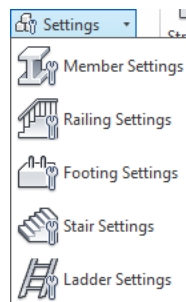


Figure 3-8 The Settings drop-down

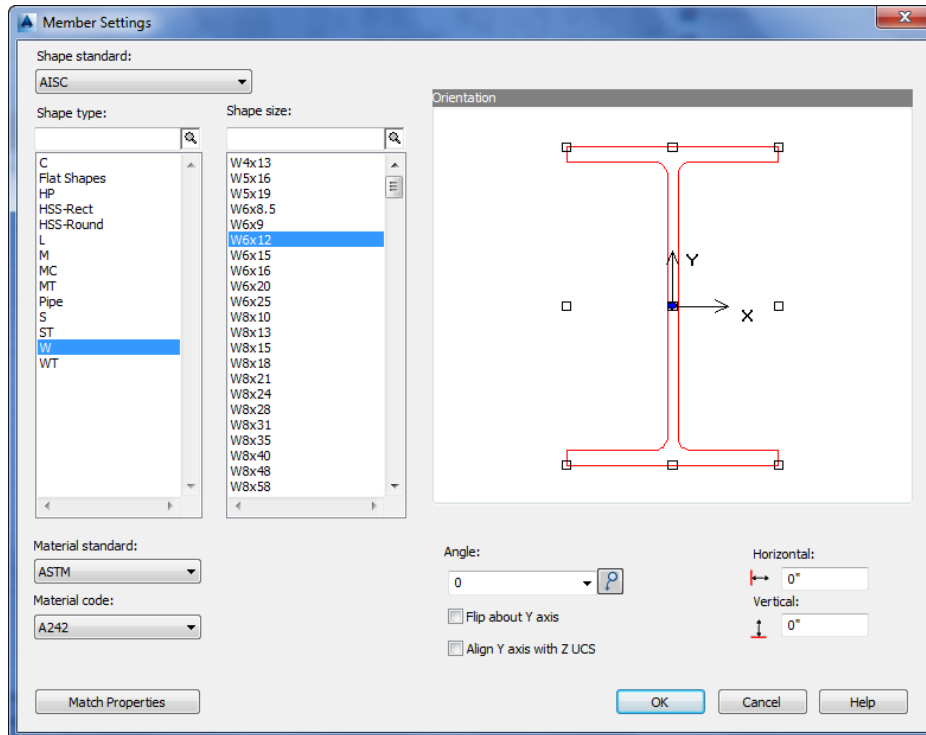


Figure 3-9 The Member Settings dialog box

In this dialog box, select the required standard for the shape of a member from the **Shape Standard** drop-down list. Next, select the shape type from the list box available in the **Shape type** area. You can search for a shape type using the search box available above the list box. After selecting the shape, you need to specify the size from the list box available in the **Shape size** area. You can search for a shape size using the search box available. Next, select the required material standard and material code from the respective drop-down lists. Specify the orientation angle using the **Angle** drop-down list. You can use the increment button available adjacent to this drop-down list to increase the angle by 90 degrees. Now, specify the justification of the member by selecting a point in the preview window. You can select the **Flip about Y axis** check box to flip the orientation about Y axis. Also, you can select the **Align Y Axis with Z UCS** check box to align the Y axis of the member with the Z UCS. After specifying all the parameters in the **Member Settings** dialog box, choose the **OK** button to set the specified member settings.

Next, choose the **Member** tool from the **Parts** panel; you will be prompted to specify the start point of the structural member. You can do so either by clicking in the drawing area or entering its coordinates at the Command prompt. After specifying the start point, you will be prompted to specify the endpoint of the member. Specify the endpoint where you want to end the member. At this point, you may continue to specify the points or terminate member creation by pressing ESC.





Tip: You can align the structural member with an already created line. To do so, enter **L** in the Command Bar; you will be prompted to select a line to align a member to. Select a line from the drawing area and then press **ENTER**; the member will be aligned to the selected line.

CREATING STAIRS

To create stairs, first you need to specify the settings. To do so, choose the **Stair Settings** tool from the **Settings** drop-down; the **Stair Settings** dialog box will be displayed, as shown in Figure 3-10. In this dialog box, you can specify the settings for creating stairs.

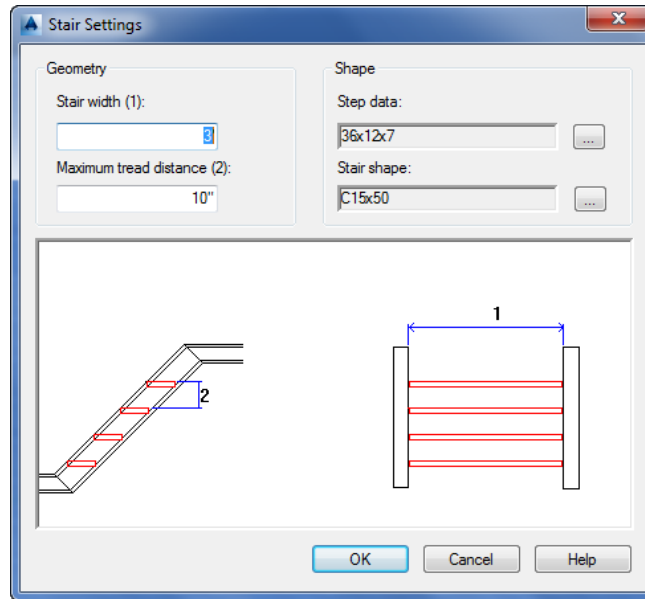


Figure 3-10 The *Stair Settings* dialog box

Specify the width of the stairs and the tread distance by entering desired values in the **Stair Width (1)** and **Maximum Tread Distance (2)** edit boxes, respectively. Next, you need to specify the step shape. To do so, click on the Browse button adjacent to the **Step data** field; the **Select Step** dialog box will be displayed. In this dialog box, select the required tread standard from the **Tread standard** list box; the tread shapes under the selected standard will be displayed in the **Tread shape** list box. Select the required shape type from the **Tread shape** list box and click **OK**; the **Select Step** dialog box will be closed. Next, you need to specify the stair shape. To do so, click on the Browse button adjacent to the **Stair shape** field; the **Select Stair Shape** dialog box will be displayed. The options in this dialog box are same as to that available in the **Member settings** dialog box. Specify the settings in this dialog box and choose the **Select** button to close this dialog box. Next, choose the **OK** button to close the **Stair Settings** dialog box.

After specifying the settings for the stairs, choose the **Stairs** tool from the **Parts** panel; you will be prompted to select the first point. Select the bottom point for the stairs; you will be prompted to select the next point, refer to Figure 3-11. Specify the top point of the stairs and press **ENTER**; the stairs will be created, as shown in Figure 3-12.



Stairs

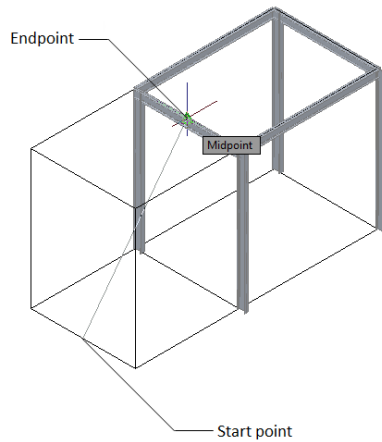


Figure 3-11 Selecting the start and end points of the stair

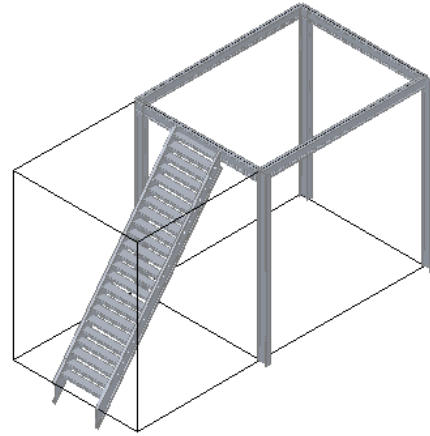


Figure 3-12 Stairs created from the selected points

Editing Stairs

To edit a stair, choose the **Structure edit** tool and then select the stair to be edited; the **Edit Stair** dialog box will be displayed. Modify the settings and choose **OK** to close the dialog box. You can also edit a stair using grips. To do so, select the stair to be edited; grips will be displayed on it, refer to Figure 3-13. You can use the center grips to change the location or length of the stairs and the side grips to increase or decrease the stair width.

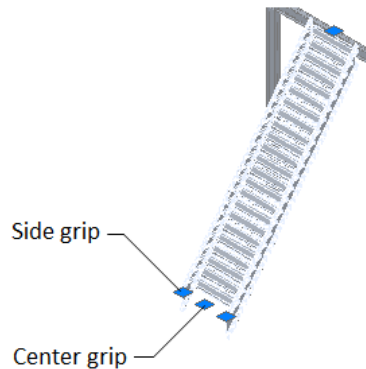


Figure 3-13 Stair with grips displayed on it

CREATING RAILINGS

To add railing to a stair, you need to specify parameters in the **Railing Settings** dialog box. To do so, choose the **Railing Settings** tool from the **Settings** drop-down; the **Railing Settings** dialog box will be displayed, as shown in Figure 3-14.

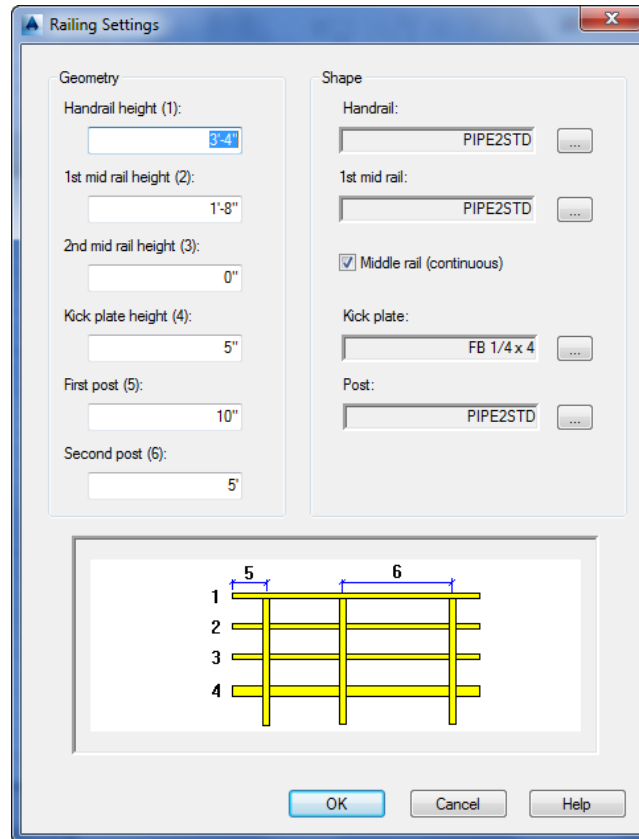


Figure 3-14 The Railing Settings dialog box

To specify different geometric values or parameters for the railing, enter desired values in the edit boxes available in the **Geometry** area in the dialog box. Refer to the preview window in Figure 3-14 for these parameters. Next, specify the shape properties by clicking on the Browse button available adjacent to the corresponding rails. Select the **Middle rail (continuous)** check box to make the mid rail continuous. Choose the **OK** button to close the dialog box.

After defining the settings for the railing, choose the **Railing** tool from the **Parts** panel; you will be prompted to select the start point. Select the bottom point of a stair and then select the endpoint of the railing. You will notice that as soon as you specify the endpoint of the first railing, you will be prompted to select the next endpoint. Press ENTER to terminate the command.



You can also add a railing to the stairs or members by selecting them from the drawing area. To do so, enter **Object** at the Command prompt; you will be prompted to select a stair or a structural member for applying the railing. Select the stairs or a structural member from the drawing area; the railing will be added to the selected member. Figure 3-15 shows a railing added to the stairs and other structural members.

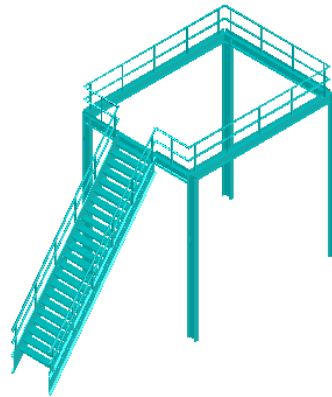


Figure 3-15 Railing added to the stairs and other structural members

CREATING LADDERS

To create a ladder, first specify the settings for the ladder. To do so, invoke the **Ladder Settings** dialog box by choosing the **Ladder Settings** tool from the **Settings** drop-down. In the **Ladder Settings** dialog box, choose the **Ladder** tab to display its options, as shown in Figure 3-16.

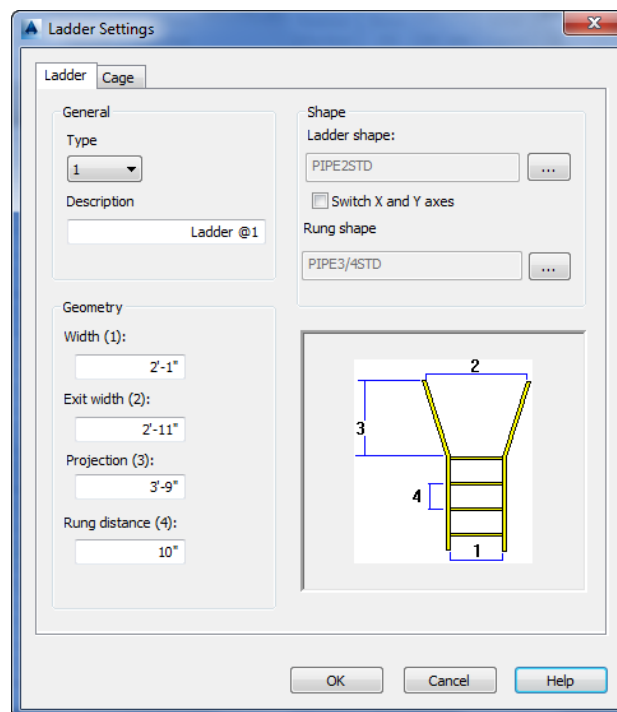


Figure 3-16 The Ladder Settings dialog box

Ladder Tab

The **Ladder** tab has three areas: **General**, **Geometry**, and **Shape**, refer to Figure 3-16. In the **General** area, you can specify the ladder type and description using the **Type** drop-down list and the **Description** edit box, respectively.

The options in the **Shape** area allow you to specify the settings for ladder shape and rung shape. To change the ladder shape settings, choose the Browse button adjacent to the **Ladder Shape** edit box; the **Select Ladder Shape** dialog box will be displayed. Modify the settings and choose the **Select** button to close the dialog box. Also, you can select the **Switch X and Y axes** check box to rotate the ladder shape by 90 degrees. Similarly, you can change the rung shape settings by invoking the **Select Rung Shape** dialog box.

The **Geometry** area contains options to define the geometry of the ladder. The preview window explains the use of the options available in this area.

Cage Tab

The **Cage** tab in the **Ladder Settings** dialog box contains the **General** and **Geometry** areas. You can see the preview of the ladder geometry in the preview area, refer to Figure 3-17.

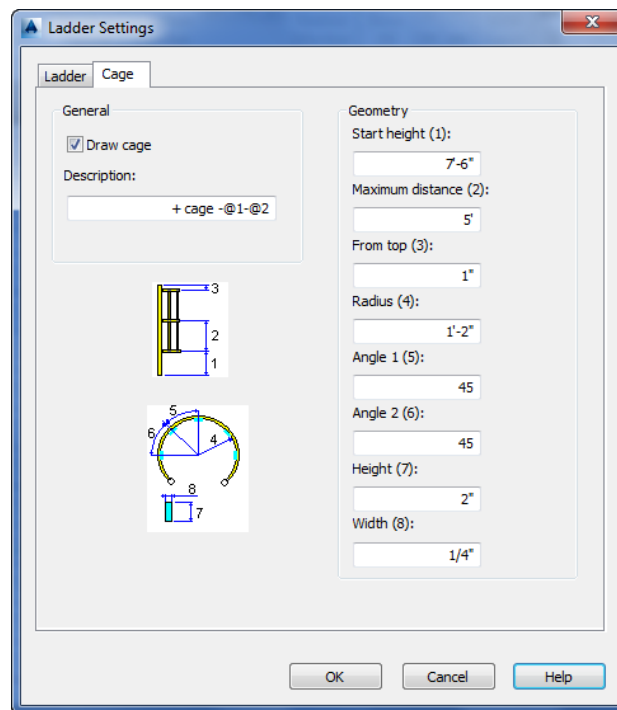


Figure 3-17 The **Cage** tab of the **Ladder Settings** dialog box

In the **General** area, select the **Draw cage** check box to create a ladder with a cage. You can enter the description of the cage in the **Description** edit box.

The **Geometry** area contains options to define the geometry of the cage. The preview window explains the use of the options available in this area. After specifying all the values, choose **OK**; the **Ladder Settings** dialog box will be closed.

After specifying the settings for the ladder, choose the **Ladder** tool from the **Parts** panel; you will be prompted to select the start point. Select the bottom point and then the top point as the endpoint of the ladder; you will be prompted to specify the directional distance. Move the cursor in the perpendicular direction and enter a distance at the Command prompt; a ladder will be created.



CREATING A PLATE/GRATE

Ribbon: Structure > Parts > Plate



To create a plate or a grate, choose the **Plate** tool from the **Parts** panel; the **Create Plate/Grate** dialog box will be displayed, as shown in Figure 3-18.

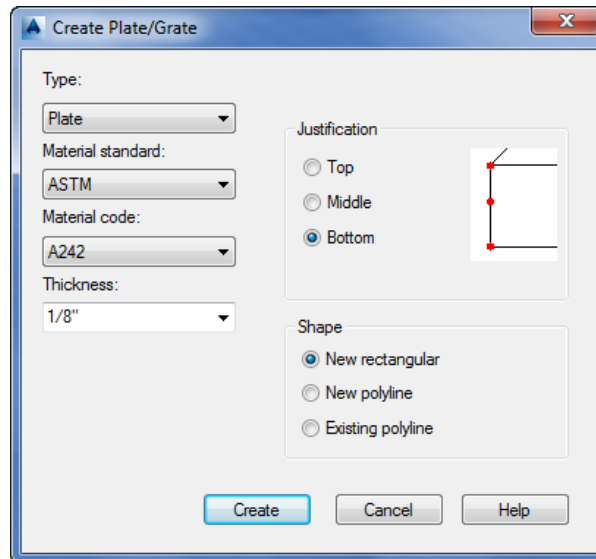


Figure 3-18 The Create Plate/Grate dialog box

Use the **Type** drop-down to specify whether you want to create a grate or a plate. Next, specify the material of the plate/grating by selecting the required material standard and material code from the respective drop-down lists. After specifying the material, select the plate/grating thickness from the **Thickness** drop-down list. If you have selected the **Grating** option from the **Type** drop-down list, you need to select the required hatch pattern and hatch scale of the grating from the **Hatch Pattern** and **Hatch Scale** drop-down lists, respectively.

Next, you need to specify the justification of a grating or plate using the options in the **Justification** area. Now, select an option from the **Shape** area. The **New rectangular** option is used to create a rectangular shaped plate/grating. The **New polyline** option is used to specify the shape by drawing a closed sketch using the **Polyline** tool. The **Existing polyline** option is used to select an already existing polyline sketch to create the plate/grating.

After specifying the options in the **Create Plate/Grate** dialog box, choose the **Create** button; a prompt will be displayed at Command prompt depending upon the option selected in the **Shape** area. For example, if you select the **New rectangular** option, you will be prompted to specify the first corner point of the plate. Specify the first corner and then the second corner to create a plate or a grating.

CREATING FOOTINGS

Ribbon: Structure > Parts > Footing



Footing is one of the important components in a structural model. The function of footing in an industrial project is to provide support to the entire structural layout. To create a footing, first define the settings for it. To do so, invoke the **Footing Settings** dialog box by choosing the **Footing Settings** tool from the **Settings** drop-down list; the **Footing Settings** dialog box will be displayed. The **Footing Settings** dialog box has two areas: **Geometry** and **Material**, and a preview window, as shown in Figure 3-19.

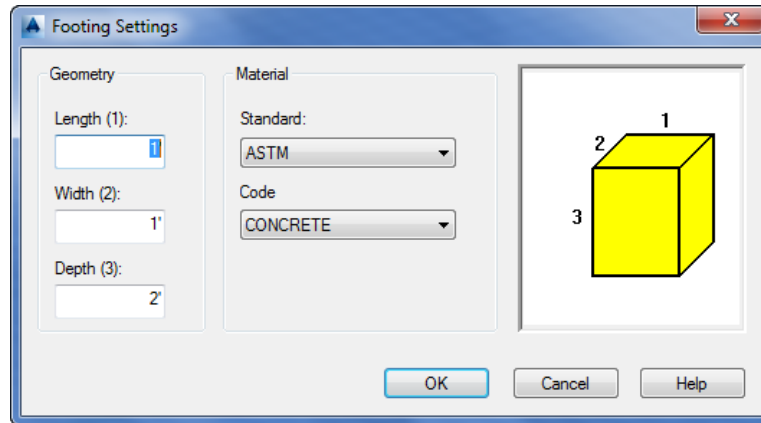


Figure 3-19 The Footing Settings dialog box

The **Geometry** area contains options to define the geometry of the footing. The preview window displays the different parameters of the **Geometry** area.

The options in the **Material** area allow you to specify the material standard and code of the footing.

After specifying the settings, choose the **Footing** tool from the **Parts** panel and specify a point to insert the footing.

EDITING THE STRUCTURAL MEMBERS

In the design process, you may need to edit the structural member. Various editing operations are discussed next.

Changing the Length of a Member

Ribbon: Structure > Cutting > Lengthen Member



You can change the length of a structural member. To do so, choose the **Lengthen Member** tool from the **Cutting** panel; you will be prompted to select a member and the following prompt sequence will be displayed at the Command prompt.

Command: PLANTSTEELLENGTHEN

Select a structural member or [Delta/Total]: *Select a member or enter an option*

The options in the Command prompt are discussed next.

Delta

This option allows you to enter a value of the length that is to be added to the existing length of a member. When you choose this option from the Command prompt, you will be prompted to specify a delta length. Enter a value and select a member; the length of the selected member will be changed. If you have specified a positive value, the length of the member will be increased. The length of the member will be decreased if you have entered a negative value.

Total

This option allows you to specify a value for the total length of a member.

Restoring the Member to its Original Length

Ribbon: Structure > Cutting > Restore Member



To restore the member to its original length, choose the **Restore Member** tool from the **Cutting** panel; you will be prompted to select the member. Select the member to restore it to its original length; the selected member will be restored to its original length.

Cutting Member at Intersections

Ribbon: Structure > Cutting > Cut Back Member



When two structural members intersect, you can remove the unwanted portion at the intersection using the **Cut Back Member** tool. To do so, invoke this tool from the **Cutting** panel; you will be prompted to select the first member. Also, the **cut Both** and **Gap** options will be displayed at the Command prompt. The **cut Both** option is used to cut both the intersecting members. The **Gap** option is used to create a gap at the intersection point. Note that you need to specify the required gap value at the Command prompt. Next, select both the intersecting members; the unwanted portion will be removed. Figure 3-20 shows a cut created using the **cut Both** option. Figure 3-21 shows a cut created by specifying a gap distance.

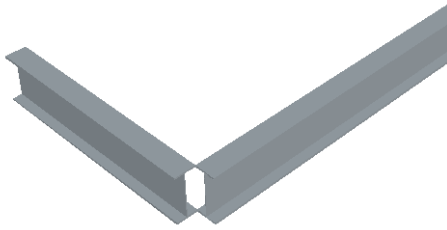


Figure 3-20 A cut created using the **cut Both** option

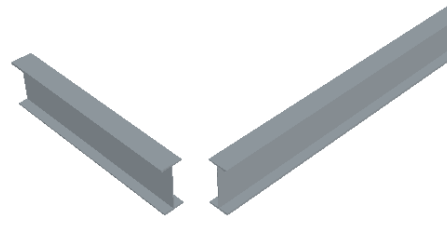


Figure 3-21 A cut created by specifying **gap value**

Creating Miter Joints

Ribbon: Structure > Cutting > Miter Cut Member



A miter joint is created at the corner of two structural members by bevelling them at an angle of 45-degrees. To create a miter joint between two structural members, choose the **Miter Cut Member** tool from the **Cutting** panel; you will be prompted to select the first structural member. Also, the **Align Edges** and **Gap** options will be displayed at the Command prompt. The **Align Edges** option allows you to create a miter joint with the edges of both the members aligned, as shown in Figure 3-22. The **Gap** option is used to create a miter joint with a gap between the edges, as shown in Figure 3-23. After choosing the required option, select the two intersecting members; the miter joint will be created.

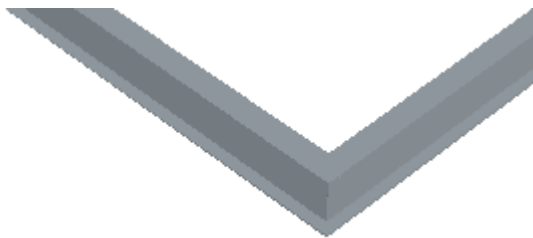


Figure 3-22 Miter joint with aligned edges

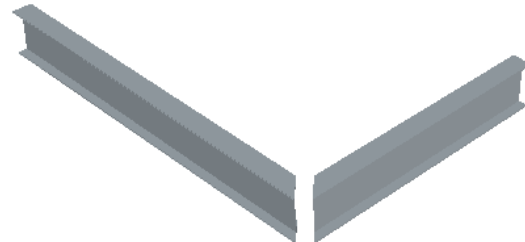


Figure 3-23 Miter joint with a gap

Trimming/Extending a Member

Ribbon: Structure > Cutting > Trim Member/Extend Member

You can trim or extend members upto a specified plane. To do so, choose the **Trim Member** tool from the **Cutting** panel; the **Trim to Plane** dialog box will be displayed, as shown in Figure 3-24. The options in this dialog box are discussed next.

Intersection plane Area

The options in this area are used to specify the plane at which the member is to be trimmed.

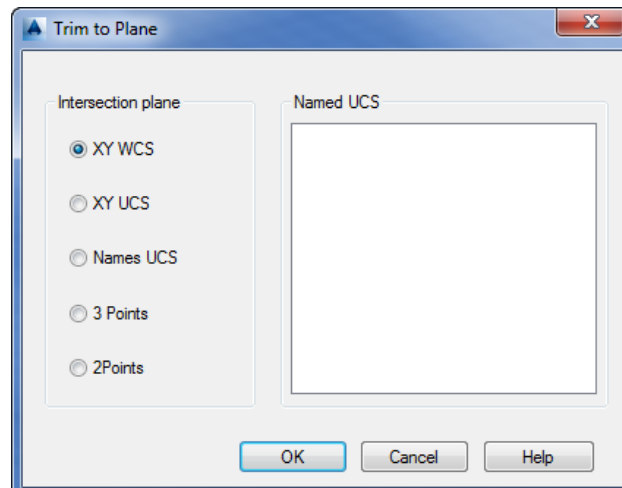


Figure 3-24 The Trim to Plane dialog box

XY WCS

This radio button is chosen by default. As a result, the XY plane of the World Coordinate System is selected as the cutting plane.

XY UCS

Select this radio button to specify XY plane of the User Coordinate System as the cutting plane.

Names UCS

Select this radio button to specify XY plane of the named User Coordinate System as the cutting plane.

3 Points

Select this radio button to specify XY plane of new coordinate system created by specifying 3 points.

2 Points

Select this radio button to create a cutting plane by drawing a line. The cutting plane is a vector of the line drawn.

Named UCS Display Box

This box displays the Named User Coordinate System.

Choose the **OK** button after selecting the required option and select a member that is to be trimmed; the selected member will be trimmed using the defined plane.

You can also extend a structural member. To do so, choose the **Extend Member** tool from the **Cutting** panel in the **Structure** tab; the **Extend to Plane** dialog box will be displayed. The options in this dialog box are same as in the **Trim to Plane** dialog box. Select an option from the **Intersection plane** area to define the plane upto which the member should be extended,

and then choose the **OK** button; the dialog box will be closed and you will be prompted to select a member to extend. Select a structural member from the drawing area; it will be extended up to the specified plane.

Exploding a Structure

Ribbon: Structure > Modify > Structure Explode



You can explode structural objects into individual elements. Note that you can only explode stairs, railings, or ladders. To explode an object, choose the **Structure Explode** tool from the **Modify** panel; you will be prompted to select a structural object. Select a stair, railing, or ladder; the selected object will be exploded into individual elements that can be modified individually.

VISIBILITY OPTIONS

When you create a plant layout, whether it is large or small, you may need to toggle the visibility of various objects in the layout. You can do so by hiding the components at any stage. The method to toggle the visibility of an object is discussed next.

Hiding and Displaying Components

To hide a component placed in a plant layout, select the component from the graphics area and choose the **Hide Selected** tool from the **Visibility** panel in the **Structure** tab; the display of the component will be turned off. To hide all other components except the selected component, choose the **Hide Others** tool from the **Visibility** panel. On doing so, the selected component will be displayed while all other components will be hidden.

To show all hidden components, choose the **Show All** button; all hidden components will be displayed again in the layout.

EXCHANGING DATA WITH OTHER APPLICATIONS

Ribbon: Structure > Export > SDNF Export



SDNF allows the exchange of steel structure data between two applications. To create a SDNF report, choose the **SDNF Export** tool from the **Export** panel; the **SDNF Export** dialog box will be displayed, as shown in Figure 3-25. Specify the path for the output file by using the **Browse** button available at the right side of the **Output file** edit box in this dialog box.

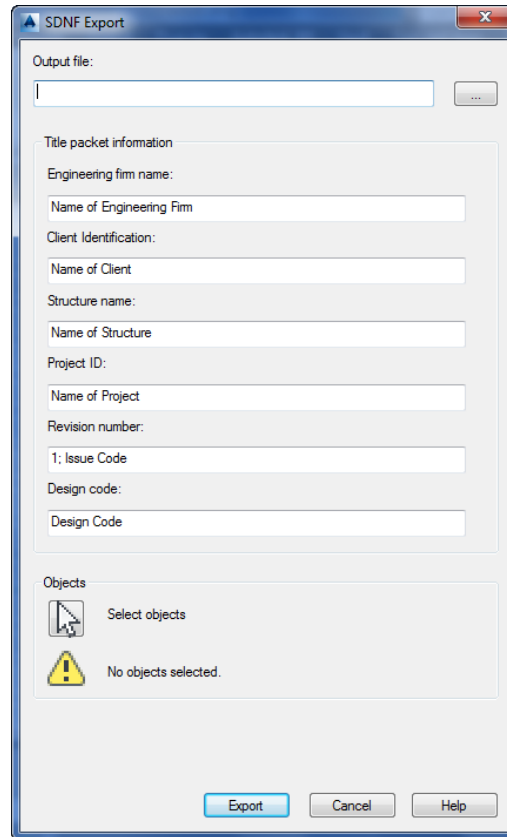


Figure 3-25 The *SDNF Export* dialog box

The **Title Packet Information** area contains options to specify the user data. The **Select Objects** button in the **Objects** area is used to select objects from the workspace. The selection status will be displayed in the **Objects** area. After specifying the parameters in this dialog box, choose the **Export** button to export the structure to SDN format.

TUTORIALS

Tutorial 1

In this tutorial, you will open the CAD/CIM project created in Chapter 2 and then to start a new AutoCAD Plant 3D file. In this file, you will create a pipe rack, as shown in Figure 3-26. You need to use W10 x 12 structural members to create this pipe rack. The dimensions for the model are given in Figure 3-27.

(Expected time: 45 min)



Note

You can also download the CAD/CIM project from www.cadcim.com by following the path: Textbooks > CAD/CAM > Plant 3D > AutoCAD Plant 3D 2015 for Designers.

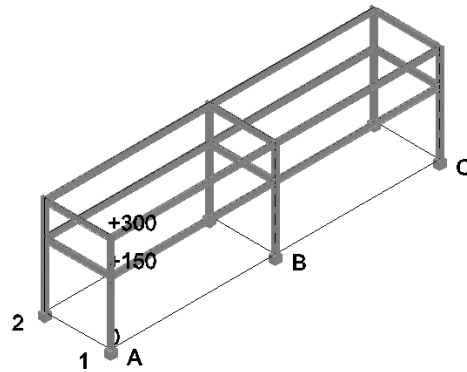


Figure 3-26 Model for Tutorial 1

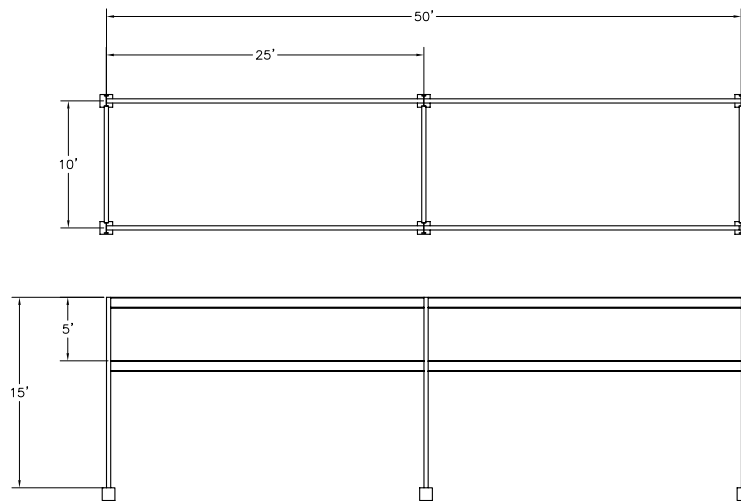


Figure 3-27 Dimensions for the model

Footing Dimensions:

Length=24 inch

Width=24 inch

Depth= 24 inch

The following steps are required to complete this tutorial:

- Open a new Plant 3D drawing file in the current project.
- Create a grid by specifying the axis, row, and platform values.
- Add footings at the bottom of the grid.
- Add columns and beams to the model.
- Cut members at intersections.

Opening a New Plant 3D File

1. Double-click on the **AutoCAD Plant 3D 2015 - English** icon from the desktop of your computer to start AutoCAD Plant 3D 2015.

Next, you need to create a new AutoCAD Plant 3D file.

2. Select the **CADCIM** project that you have created in Tutorial 1 of Chapter 2 from the **Current Project** drop-down list in the **PROJECT MANAGER**.
3. Select the **Plant 3D Drawings** node in the **Project** area and choose the **New Drawing** button; the **New DWG** dialog box is displayed.
4. Enter **Pipe_rack.dwg** in the **File name** edit box and choose **OK**; the new file is created.
5. Select the **3D Piping** option from the **Workspace** drop-down list located in the **Quick Access Toolbar**.

Creating the Grid

1. Choose the **Grid** tool from the **Parts** panel in the **Structure** tab; the **Create Grid** dialog box is displayed.
2. In this dialog box, specify the parameters as given below and retain the default settings for other parameters.

Axis value: 0, 25', 50'

Row value: 0, 10'

Platform value: 0, 10', 15'



Grid

3. Next, choose the **Create** button from the dialog box; the grid is created, as shown in Figure 3-28. Note that the view orientation in the figure is set to **SW Isometric**. You can set the view orientation to **SW Isometric** by choosing the **SW Isometric** option from the **3D Navigation** drop-down list in the **View** panel of the **Structure** tab.

Creating Footings

1. Choose the **Footling Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Footling Settings** dialog box is displayed.
2. Enter **24** in the **Length(1)**, **Width(2)**, and **Depth(3)** edit boxes. Next, accept the default values in the **Material** area and choose **OK**; the settings of the footings are changed and the **Footling Settings** dialog box is closed.
3. Choose the **Footling** tool from the **Parts** panel; you are prompted to select the insertion point. Place the footings at the bottom grid points, as shown in Figure 3-29.

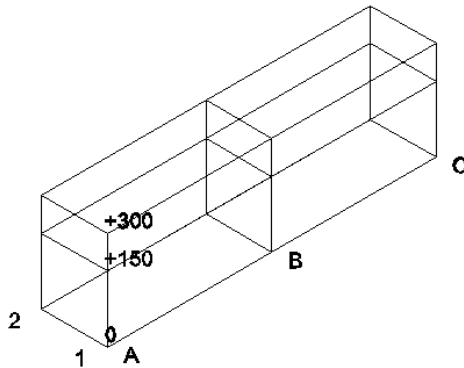


Figure 3-28 Model after creating the grid

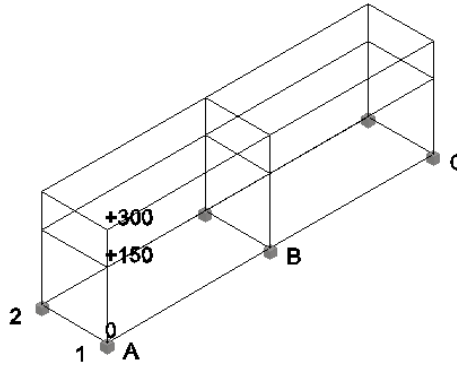


Figure 3-29 Model after adding footings

Creating Columns and Beams

To create structural members, first you need to set the properties of the member. Also, you need to turn on the object snap and 3D object snap in order to make the grid points easier.

1. Choose the **Member Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Member Settings** dialog box is displayed.
2. Select **W** and **W 10x12** from the **Shape type** and **Shape size** list boxes, respectively. Next, accept the default values of the other options and choose **OK**; the settings of the structural member are changed and the **Member Settings** dialog box is closed.
3. Choose the **Member** tool from the **Parts** panel; you are prompted to select the start point of the member.

Make sure that the **Ortho Mode** is turned on in the Status Bar.

4. Select the grid point, as shown in Figure 3-30; you are prompted to select the endpoint.

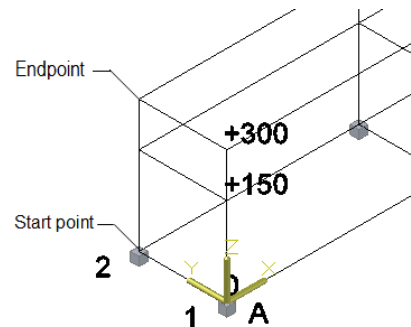


Figure 3-30 Selecting the start point and the endpoint of the column

5. Move the cursor vertically upward and snap to the grid point, refer to Figure 3-30. Next, click to select it; the column is created between the two selected points. Similarly, place columns on other footings, as shown in Figure 3-31. You can also use the **Copy** command to create rest of the columns.

Next, you need to add beams. But before that, you need to change the justification of the member to top.

6. Choose the **Member Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Member Settings** dialog box is displayed.

7. In the **Member Settings** dialog box, choose the top center justification point in the **Orientation** window. Next, choose **OK** to close the dialog box.
8. Invoke the **Member** tool; you are prompted to select the start point. Select the start point and the end point, as shown in Figure 3-32; a beam is created between the two selected points and a rubber-band line is displayed between the cursor and the specified point. Also, you are prompted to select the end point of the new member. Similarly, create rest of the beams, as shown in Figure 3-33.

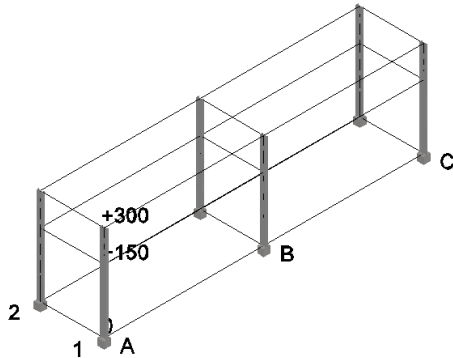


Figure 3-31 Model after creating columns

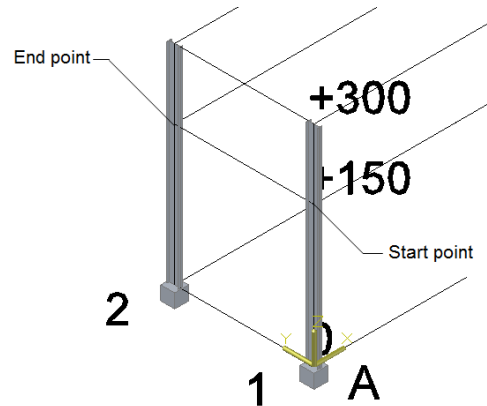


Figure 3-32 The start point and end point of the beam



Note

You can hide the columns such that only the grid points are visible while specifying the start and endpoints of the beam.

9. Similarly, create the second level. The model after placing all the beams is shown in Figure 3-34.



Tip: You can select all the beams at the first level and then use the **COPY** command to create the second level.

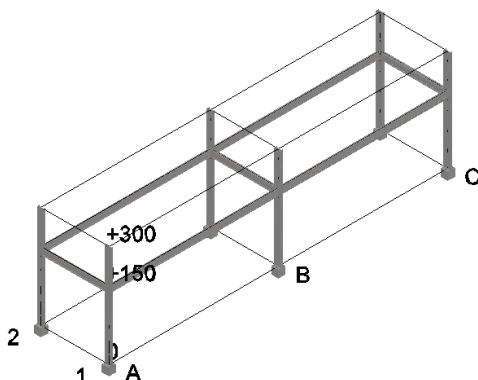


Figure 3-33 Model after creating the lower level

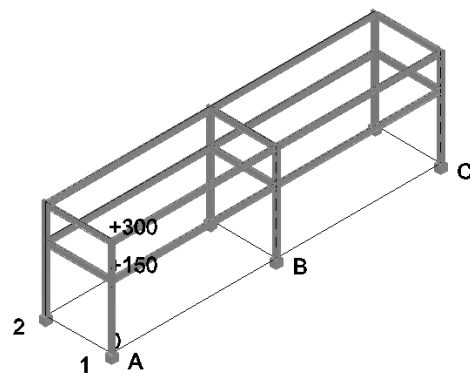


Figure 3-34 Model after adding two levels of beams

Next, you need to cut intersections between members.

10. Choose the **Cut Back Member** tool from the **Cutting** panel; you are prompted to select the limiting member.
11. Select the first column on the left side of the grid; you are prompted to select the structural member to be cut.
12. Select any one of the beam that is intersecting with the column; the beam is cut at the intersection. Also, you are prompted to select the limiting member. Similarly, cut beams at the intersections by using columns as the limiting members.

Saving the Model

1. Choose the **Save** tool from the **Quick Access Toolbar**; the file is saved.
2. Choose **Application Button > Close** to close the file.

Tutorial 2

In this tutorial, you will open the CADCIM project created in Chapter 2 and then start a new AutoCAD Plant 3D file. In this file, you will create a structural model, as shown in Figure 3-35. The layout uses W10 x 12 structural members. The dimensions of the model are shown in Figure 3-36. **(Expected time: 45 min)**



Note

You can also download the CADCIM project file of tutorial of chapter 2 of from www.cadcim.com by following the path: Textbooks > CAD/CAM > Plant 3D > AutoCAD Plant 3D 2015 for Designers.

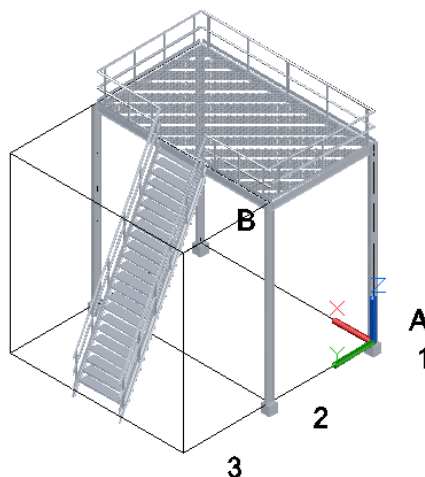


Figure 3-35 Structural model for Tutorial 2

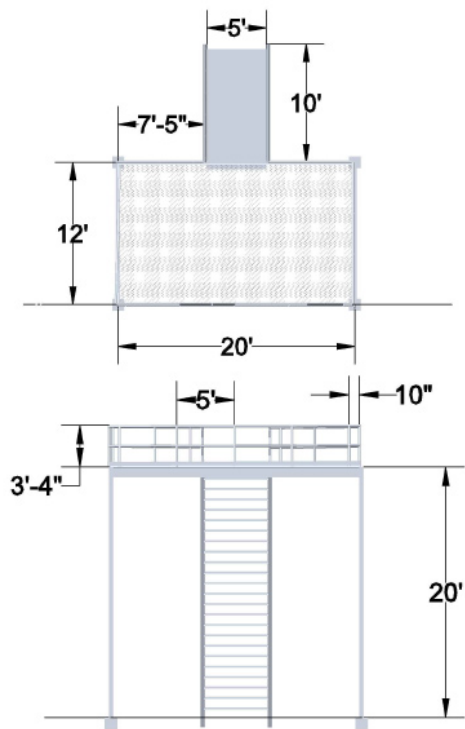


Figure 3-36 Dimensions of the model

The other specifications of the model to be created are as follows:

Grating Specifications

Material standard	ASTM
Material code	A242
Thickness	1"
Hatch pattern	ZIGZAG
Hatch scale	10"
Justification	Bottom
Shape	New rectangular

Footing Dimensions:

Length=12 inch
Width=12 inch
Depth= 12 inch

Railing Specifications

Handrail Height	40 inch
1st mid rail height	20 inch

2nd mid rail height	0
Kick plate height	5 inch
First post	10 inch
Second post	5 feet
Handrail Shape	PIPE2STD
Kick plate Shape	FB 1/4x4
Post Shape	PIPE2STD

Stair Specifications

Stair width	60 inch
Maximum tread distance	10 inch
Tread shape	36x12x7
Stair Shape	C15x50

The following steps are required to complete this tutorial:

- Open a new Plant 3D drawing file in the current project.
- Create a grid by specifying the axis, row, and platform values.
- Add footings at the bottom of the grid.
- Add columns and beams to the model.
- Cut members at intersections.
- Add stairs to the model.
- Create platforms by using the **Plate** tool.
- Add railings to stairs and platforms.

Opening a New Plant 3D File

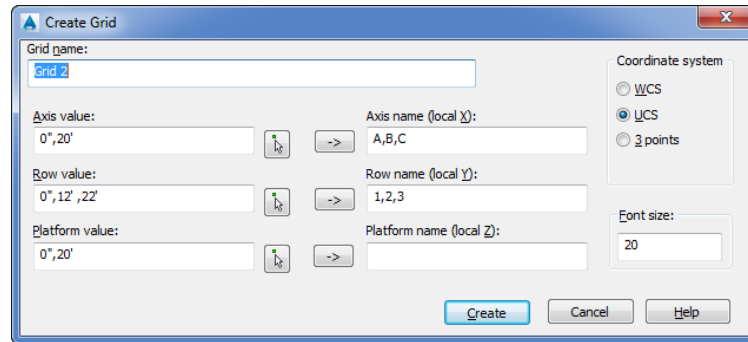
- Double-click on the **AutoCAD Plant 3D 2015 - English** icon from the desktop of your computer to start AutoCAD Plant 3D 2015.

Next, you need to create a new AutoCAD Plant 3D file.

- Select the CADCIM project that you created in Chapter 2 from the **Current Project** drop-down list in the **PROJECT MANAGER**.
- Select the **Plant 3D Drawings** node in the **Project** area and choose the **New Drawing** button available; the **New DWG** dialog box is displayed.
- Enter **c03tut02.dwg** in the **File name** edit box and choose **OK**; the new file is created.

Creating the Grid

- Choose the **Grid** tool from the **Parts** panel in the **Structure** tab; the **Create Grid** dialog box is displayed.
- In this dialog box, specify the values, as shown in Figure 3-37.



*Figure 3-37 The values in the **Create Grid** dialog box*

- Next, accept default values for other options and choose the **Create** button; the grid is created, as shown in Figure 3-38. Next, change the view orientation to **SW Isometric**.

Creating Footings

- Choose the **Footing Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Footing Settings** dialog box is displayed.
- In the **Footing Settings** dialog box, specify the following values:

Length(1): 1'

Width(2): 1'

Depth(3): 1'

Next, accept the default values in the **Material** area and choose **OK** to close the dialog box.

- Invoke the **Footing** tool and place the footings at the bottom grid points, as shown in Figure 3-39.

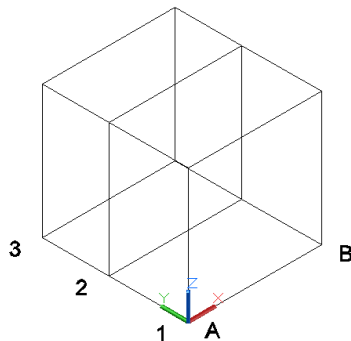


Figure 3-38 Model after creating the grid

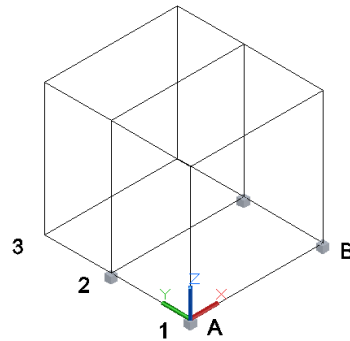


Figure 3-39 Model after adding footings

Creating Columns and Beams

To create structural members, first you need to set the properties of the member and then place a member. Also, you need to turn on the object snap and 3D object snap so that you can easily select the grid points.

1. Choose the **Member Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Member Settings** dialog box is displayed.
2. In the **Member Settings** dialog box, select **W** and **W 10x12** from the **Shape type** and **Shape size** edit boxes, respectively. Next, choose the middle center justification point in the **Orientation** window and accept default values of the other options. Choose **OK**; the settings of the structural member are changed and the **Member Settings** dialog box is closed.
3. Choose the **Member** tool from the **Parts** panel and then place columns on all footings, as shown in Figure 3-40. Use grid points for the precise placement of columns.

Next, you need to add beams. But before that, you need to change the justification of the member to top.

4. Invoke the **Member Settings** dialog box and choose the top center justification point in the **Orientation** window. Next, choose **OK** to close the dialog box.
5. Choose the **Member** tool and place beams on both levels of the grid. The model after placing all the beams is shown in Figure 3-41.

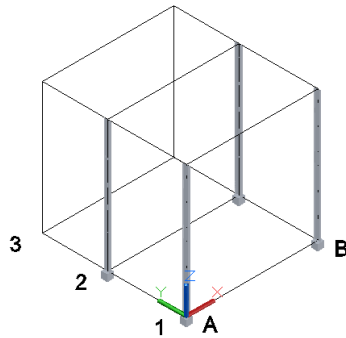


Figure 3-40 Model after adding columns

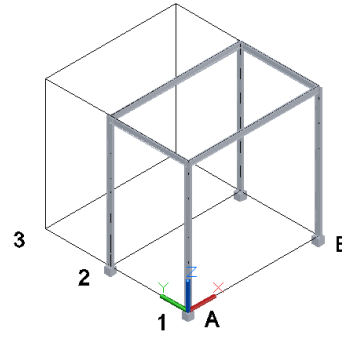


Figure 3-41 Model after adding columns and beams

6. Trim the intersections between the members.

Creating Stairs

Next, you need to add stairs to the model. To do so, first you need to set the stair settings and then place stairs.

1. Choose the **Stair Settings** tool from the **Settings** drop-down in the **Parts** panel; the **Stair Settings** dialog box is displayed.

2. Enter **5'** in the **Stair width(1)** edit box and **10"** in the **Maximum tread distance(2)** edit box. Next, specify the **Step data** and **Stair shape** as given in the **Stair Specification**. Choose the **OK** button in the **Stair Settings** dialog box to close it.

Next, you need to create stairs. Make sure that the **Object Snap** and the **3D Object Snap** are turned ON.

3. Choose the **Stairs** tool from the **Parts** panel; you are prompted to select the first point of the stair.
4. Rotate the model and select the midpoint on the top edge of the member, as shown in Figure 3-42; you are prompted to select the second point.
5. Select the midpoint of the bottom grid line, as shown in Figure 3-43; a line is placed between the two specified points. Press ENTER to create the stairs on the specified line, as shown in Figure 3-44.

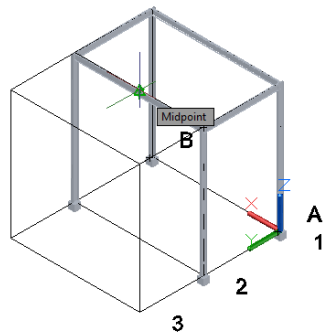


Figure 3-42 Model after creating the grid

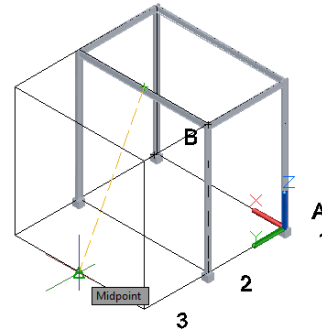


Figure 3-43 Model after adding footings

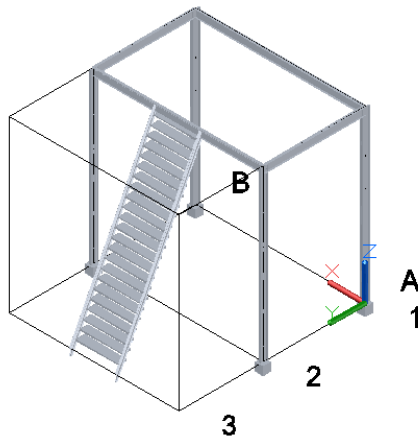


Figure 3-44 Model after creating stairs

Adding Gratings

You need to create platforms by adding gratings.

1. Choose the **Plate** tool from the **Parts** panel; the **Create Plate/Grate** dialog box is displayed.
2. Specify the parameters for the grating as given below and then choose the **Create** button; you are prompted to specify the first corner of the grate.

Type	Grating
Material standard	ASTM
Material code	A242
Thickness	1"
Hatch pattern	ZIGZAG
Hatch scale	10"
Justification	Bottom
Shape	New rectangular

3. Select the corner points, as shown in Figure 3-45; a rectangular grating is created on the platform, as shown in Figure 3-46.

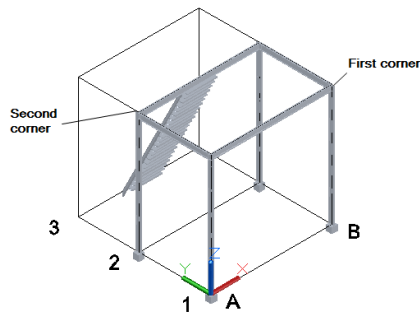


Figure 3-45 Selecting corner points to create a grating

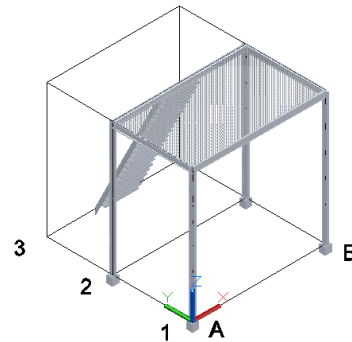


Figure 3-46 Selecting corner points to create other grating

Adding Railing

Next, you need to add railing to the stairs and the platforms.

1. Choose the **Railing Settings** tool from the **Settings** drop-down list; the **Railing Settings** dialog box is displayed. Next, specify settings as given in the **Railing Specifications**.
2. Choose the **Railing** tool from the **Parts** panel; you are prompted to select the start point of the railing. Enter **Object** at the Command prompt; you are prompted to select an object to align railing.
3. Select the stairs; a railing is added to the stairs, as shown in Figure 3-47.

Next, you need to add railing to the beams.

- Invoke the **Railing** tool again and select the points, as shown in Figure 3-48; a railing is created between the selected points. Next, press ENTER.

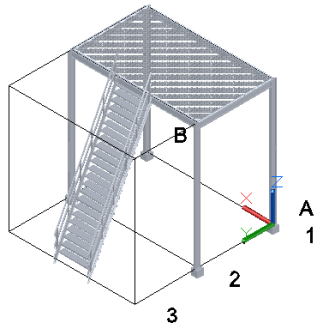


Figure 3-47 Railing added to the stairs

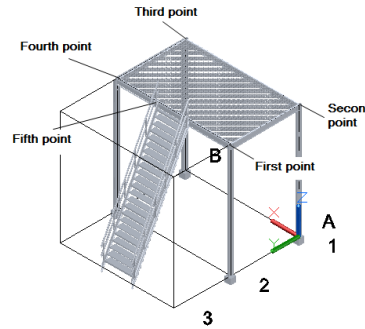


Figure 3-48 Points to be selected to create the railing

- Invoke the **Railing** tool again and select the start and end points, as shown in Figure 3-49. The model after adding the railing is shown in the Figure 3-50.

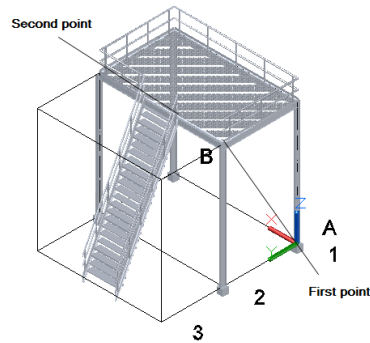


Figure 3-49 Points to be selected to create the railing

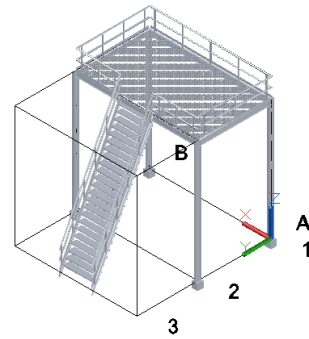


Figure 3-50 Model after adding the railing

Saving the Model

- Choose the **Save** tool from the **Quick Access Toolbar**; the file is saved.
- Choose **Application Button > Close** to close the file.

Self-Evaluation Test

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

- In the _____ mode, the members are represented as lines with a shape symbol.

2. The _____ mode gives a 3D look to the members with fillets added to the structural outline.
3. You can add a railing to a structural member or a stair by invoking the _____ option and selecting them directly.
4. The _____ allows you to exchange steel structure data between two applications.
5. You need to specify _____ to place a ladder at some distance from the specified location.
6. The _____ option creates a miter joint with a gap between the edges.
7. You need to select the _____ check box in the **Ladder Settings** dialog box to create a ladder with a cage.
8. You can use the _____ button to match the properties of an already existing member with a new member.
9. You can modify individual elements of a stair, railing, or a ladder after exploding them. (T/F)
10. You can also convert a line into a structural member by invoking the **Member** tool and then selecting it. (T/F)

Review Questions

Answer the following questions:

1. The _____ check box in the **Railing Settings** dialog box is selected to make the middle rail continuous.
2. The _____ are used to increase or decrease the width of the stairs.
3. You can hide all other components, except the selected one by using the _____ button.
4. You can select an existing polyline to convert it into a grating or a plate, if the _____ radio button is selected in the **Create Plate/Grate** dialog box.
5. The _____ check box in the **Member Settings** dialog box is used to orient the Y axis in the opposite direction.

EXERCISE

Exercise 1 **Skid**

In this exercise, you will create the model shown in Figure 3-51. Its orthographic views are given in the drawing shown in Figure 3-52. The layout uses W10 x12 structural members.
(Expected time: 30 min)

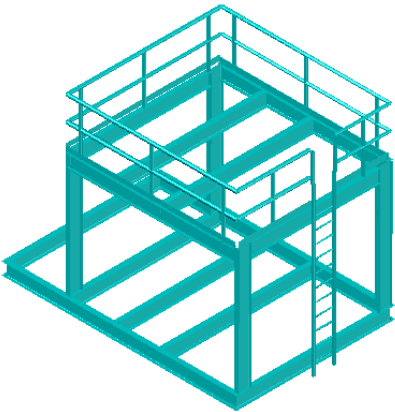


Figure 3-51 The model for Exercise 1

Railing Specifications

Handrail Height	40 inch
1st mid rail height	20 inch
2nd mid rail height	0
Kick plate height	5 inch
First post	10 inch
Second post	5 inch
Handrail Shape	PIPE2STD
Kick plate Shape	FB 1/4x4
Post Shape	PIPE2STD

Ladder Specifications

Width	25 inch
Exit width	35 inch
Projection	45 inch
Rung Distance	10 inch
Ladder shape	PIPE2STD
Rung Shape	PIPE3/4STD

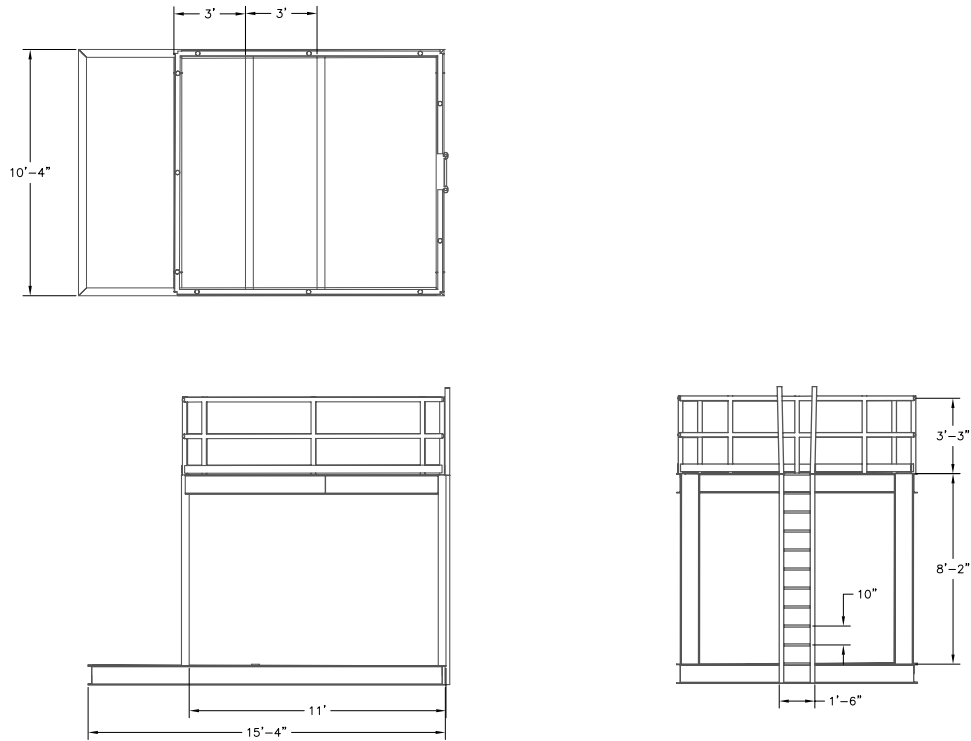


Figure 3-52 Orthographic views for Exercise 1

Answers to Self-Evaluation Test

1. Line Model, 2. Shape Model, 3. Object, 4. SDNF, 5. Directional distance, 6. Gap, 7. Draw cage, 8. Match Properties, 9. T, 10. T