

Tutorial 5

TWO COLUMN HAMMERHEAD PIER

Civil

TUTORIAL 5. TWO COLUMN HAMMERHEAD PIER

Summary	1
Analysis Model and Load Cases / 1	
Structural Modeling Using Nodes and Elements	3
Preferences Setting and Material Property Data Entry / 3	
Create the Pier Base with Plate Elements / 4	
Loading Data Entry / 21	
Perform Structural Analysis	25
Verification and Interpretation of Analysis Results	25
Load Combination / 25	
Check the Deformed Shape / 27	
Check the Stresses / 28	

TUTORIAL 5.

TWO COLUMN HAMMERHEAD PIER

Summary

This example presents a hammerhead pier commonly encountered in the design of bridge structures. This chapter has been organized so that the user can easily follow the instructions from the modeling to the interpretation of analysis results. It is assumed that the user has become familiar with the functions presented previously in “Tutorial 1”. In this example, the Icon Menu is mainly used, similar to “Tutorial 4”.

Analysis Model and Load Cases

The summary of the structural shape and model of the hammerhead pier is shown in Fig.5.1 and 5.2.

We will consider only the following two load cases for modeling:

- Load Case 1: Vertical load $P_1 = 430 \text{ kN}$
- Load Case 2: Seismic load $P_2 = 520 \text{ kN}$

It is assumed that the boundary condition at the base of the pier is completely fixed.

The present example focuses on the functions of **midas Civil**. Therefore, the engineering assumptions adopted here may be different from the practical applications. The basic items previously described concerning the functions of **midas Civil** have been omitted from this example.

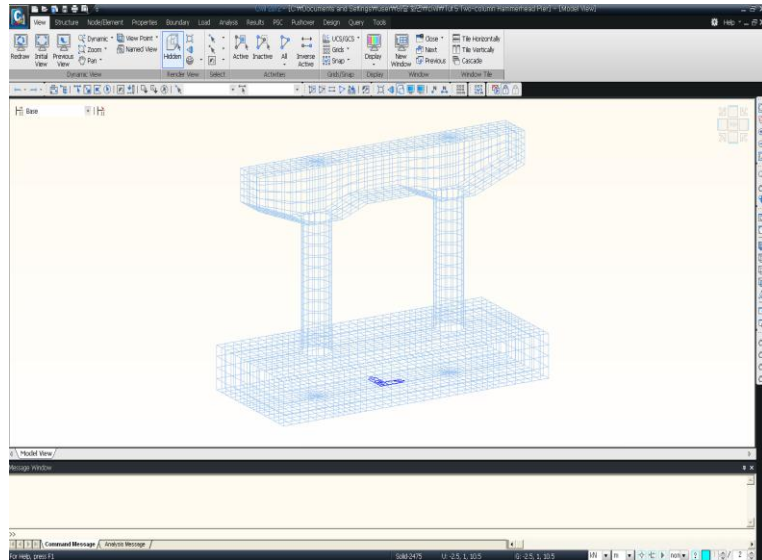


Figure 5.1 View of the Hammerhead Pier Model

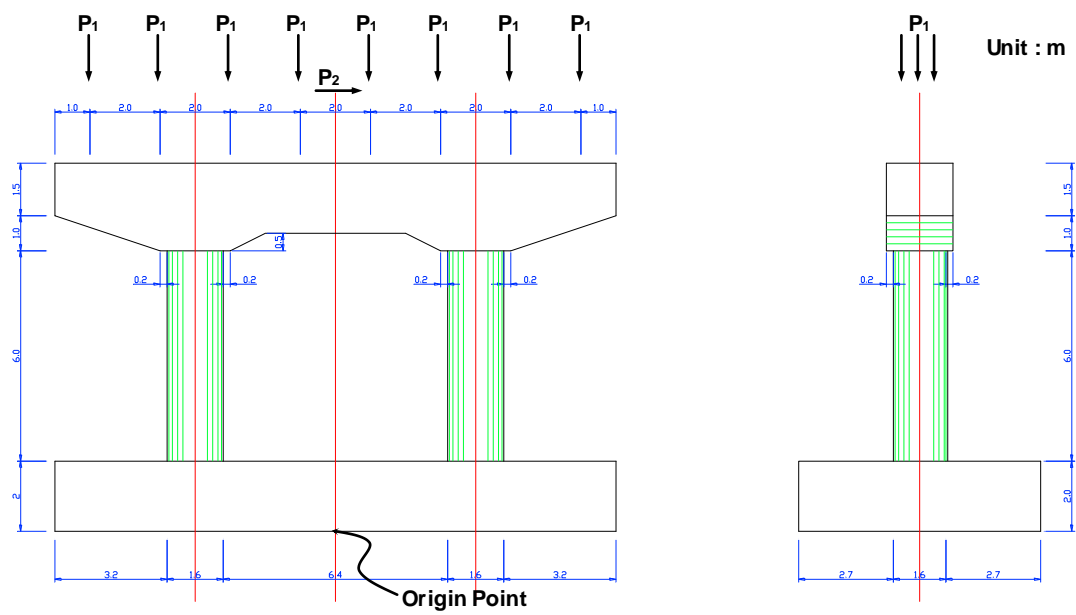





Figure 5.2 Front and Side Views of Hammerhead Pier

Structural Modeling Using Nodes and Elements

Preferences Setting and Material Property Data Entry

Open a new file ( **New Project**) to model the pier and save the file as “pier” ( **Save**).

Click the unit system selection button of **Status Bar** at the bottom of the screen  and select “**kN**” and “**m**”.

The modeling will be performed using principally the Icon Menu, similar to the previous “Tutorial 4. Arch Bridge”. Refer to “Tutorial 4” for the method of displaying the icons in the working window.

The material properties of the pier are as follows:

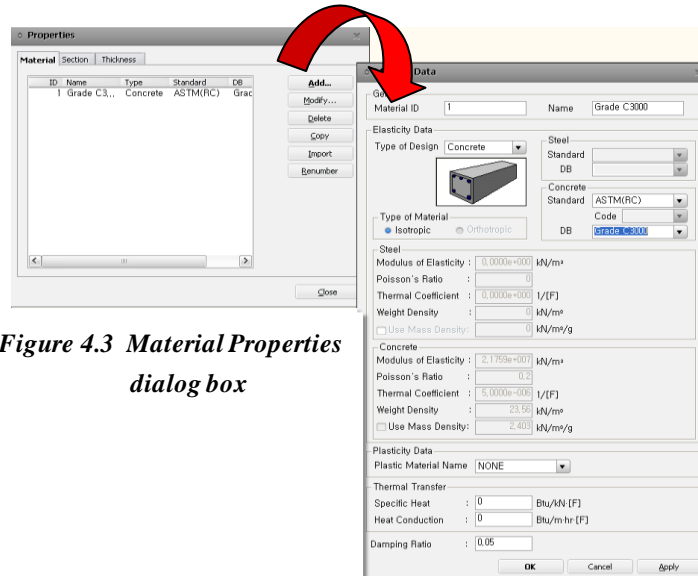






Figure 4.3 Material Properties dialog box

Figure 4.4 Material Data dialog box








-
1. Click  **Material Properties** in **Properties** from the Main Menu.
 2. Click  (Fig.5.3).
 3. Confirm “1” in the **Material Number** field of **General** (Fig.5.4).
 4. Select “**Concrete**” in the **Type of Design** selection field.
 5. Confirm “**ASTM (RC)**” in the **Standard** selection field of **Concrete**.
 6. Select “**Grade C3000**” in the **DB** selection field.
 7. Click .
 8. Click .
-


In this example, plate elements will be expanded to a specific direction to generate solid elements (by Extrude Elements) rather than modeling the pier directly with solid elements. The modeling procedure is as follows:

- Use rectangular plate elements to model the footing. Model the part that connects to a column with circular plate elements to reflect the circular shape of the column.
- Extrude the generated lower plane (plate elements) extending into the depth of the pier footing vertically.
- Select the circular-shaped plate intended for the column and extend the plate vertically to form the circular column by extruding it for the full height of the column.
- Move the relevant plate elements previously modeled upward to the top of the coping for modeling.
- Subdivide the above plate elements moved from the lower part, based on the coping depths. Project the plate elements vertically onto the lower-sloped planes to complete the coping model.

Create the Pier Base with Plate Elements

Use **Structure Wizard** to create the portion of the circular column within the lower plane of the footing (Fig.5.5).

1. Select **Structure>Wizard>Base Structures>Plate** from the Main Menu.
2. Select the circular plate () in the **Type1** selection field of the **Input** tab (Fig.5.5(a)).
3. Enter “**0.8**” in the **R** selection field.
4. Enter “**2**” in the **Material** selection field.
5. Enter “**1**” in the **Thickness** selection field.
6. Radio on **Number of Divisions** in the **Edit** tab (Fig.5.5(b)).
7. Enter “**16**” in the **m** selection field.
8. Enter “**4**” in the **n** selection field.
9. Enter “**-4,0,0**” in the **Insert tab > insert point**.
10. Enter “**-90**” in the **Alpha** field of **Rotations** (Fig.5.5(c)).
11. Check (✓) “**Show No.**” of **Origin Point** and select “**3(0.8,0,0.8)**” in the right selection field.
12. Click .
13. Click  **Auto Fitting**.
14. Click  **Top View**.
15. Click  **Point Grid** and  **Point Grid Snap** (Toggle off). 

 Toggle off Point Grid as it is of no use in this example.

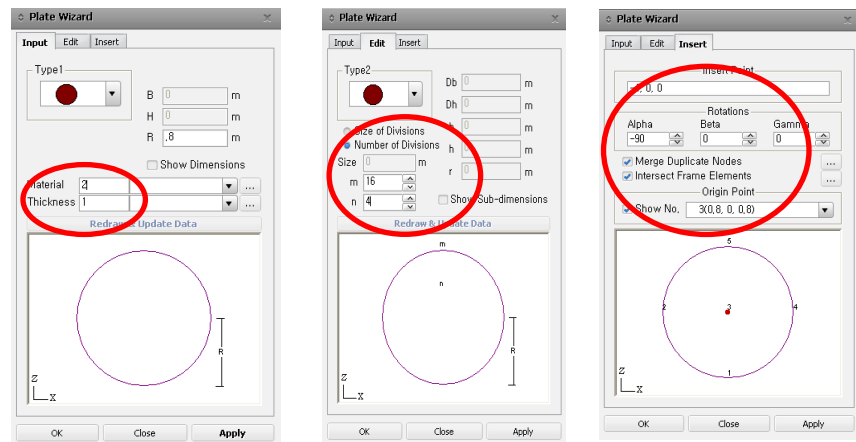


(a) **Input Tab**(b) **Edit Tab**(c) **Insert Tab**

Figure 5.5 Plate Wizard window

Use **Group** to attribute a name to the circular plate in advance for the sake of convenience later when the plate is selected and extruded to create the circular column.

1. Click  **Group**.
2. Right-click the mouse in the **Structure Group** to select “New”, and then enter “**Circular Column**”.
3. Click  **Select All**.
4. Drag the **Circular Column** into the Main Window to assign the selected elements in the group of the “**Circular Column**”

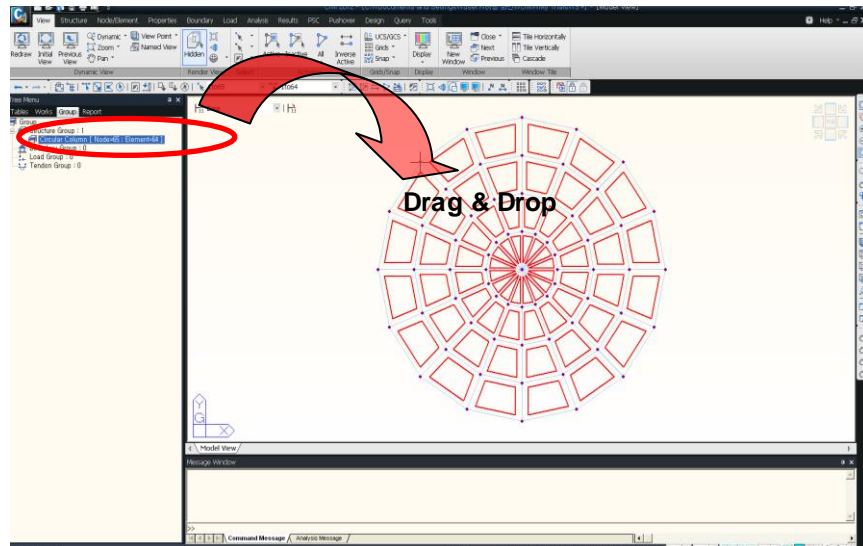









Figure 5.6 Group dialog bar

Now, create the rectangular plate elements in the vicinity of the circular plate to build up the footing.

-
1. Click  **Create Nodes**.
 2. Enter “**-3, 0, 0**” in the *Coordinates (x, y, z)* field.
 3. Enter “**1**” in *Number of Times* of the *Copy* selection field.
 4. Enter “**0, 1, 0**” in the *Distances (dx, dy, dz)* field.
 5. Click .
 6. Enter “**-4, 1, 0**” in the *Coordinates (x, y, z)* field.
 7. Enter “**0**” in *Number of Times* of the *Copy* selection field.
 8. Click .
 9. Click  **Divide In Node**.
 10. Confirm “**Equal Distance**” of *Divide*.
 11. Enter “**3**” in the *Number of Divisions* field.
 12. Click  **Node Number** and  **Element Number**(Toggle on).
 13. Use *Mouse Editor* in the *Nodes to Divide* field to successively assign nodes **66** and **67**, **67** and **68**.
 14. Click  **Create Elements**.
 15. Select “**Plate**” in the *Element Type* selection field and confirm “**4 Nodes**”.
 16. Confirm “**Thick**” in the *Type* selection field.
 17. Confirm “**1**” in the *No.* selection field of *Material*.
 18. Confirm “**1**” in the *No.* selection field of *Thickness*.
 19. Assign sequentially nodes **66, 69, 9, 5** to create plate element **65**.
 20. Assign sequentially nodes **69, 70, 13, 9** to create plate element **66**.
 21. Assign sequentially nodes **70, 67, 71, 13** to create plate element **67**.
 22. Assign sequentially nodes **13, 71, 72, 17** to create plate element **68**.
 23. Assign sequentially nodes **17, 72, 68, 21** to create plate element **69**.
-

Create temporary line elements along the right edge to extrude the line elements to generate the plate elements in the + X direction (Fig.5.7).

Click Query Elements and select the element for which you desire to find the element information which is displayed at the bottom of the screen in the Message window, or, toggle on Fast Query at the bottom of the screen (Fig.5.7-1) to get the information on the screen by placing the mouse on the desired element.

1. Select **"Truss"** in the *Element Type* selection field of the *Create Elements* dialog bar.
2. Confirm the check (✓) in **Node** of the *Intersect* selection field.
3. Assign successively nodes **66** and **67**.
4. Click *Select Recent Entities* (select truss elements **70**, **71** and **72**).
5. Click *Extrude Elements*.
6. Select **"Line Elem.→Planar Elem."** in the *Extrude Type* selection field.
7. Confirm the check (✓) in **"Remove"** in the *Source* selection field.
8. Select **"Plate"** in the *Element Type* selection field of *Element Attribute*.
9. Confirm **"Thick"** in the *Type* selection field.
10. Confirm **"Translate"** in the *Generation Type* selection field.
11. Confirm **"Equal Distance"** in the *Translation* selection field.
12. Enter **"0.5, 0, 0"** in the *dx, dy, dz* field and **"6"** in the *Number of Times* field.
13. Click

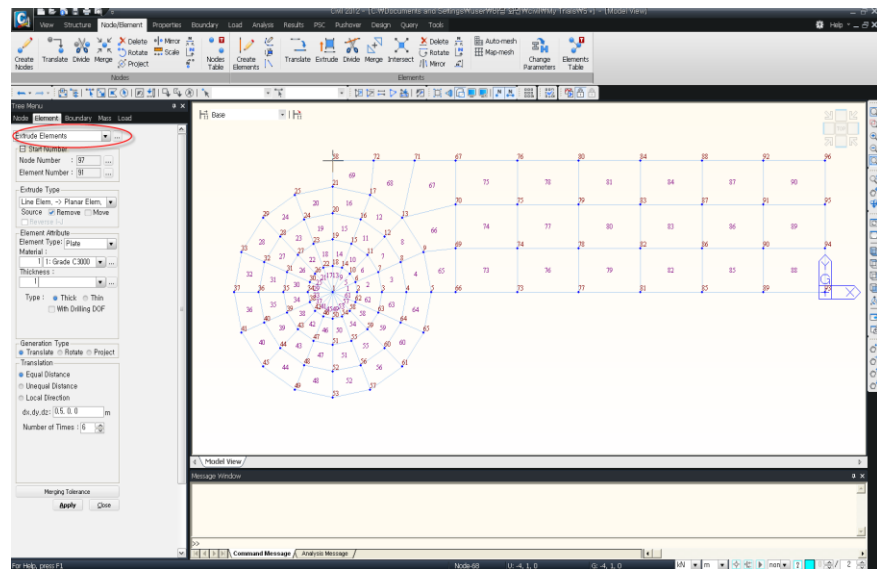








Figure 5.7 Creating Plate Elements

Use a procedure similar to the previous steps to create the plate elements along the width of the footing (Fig.5.8).

1. Click  **Create Elements**.
2. Confirm “**Truss**” in the **Element Type** selection field.
3. Confirm the check (✓) in **Node** of the **Intersect** selection field.
4. Use **Mouse Editor** to assign consecutively nodes **68** and **96**.
5. Click  **Node Number** and  **Element Number** (Toggle off).
6. Click  **Select Recent Entities**.
7. Click  **Extrude Elements**.
8. Select “**Line Elem.→Planar Elem.**” in the **Extrude Type** selection field.
9. Confirm the check (✓) in “**Remove**” in the **Source** selection field.
10. Confirm “**Plate**” in the **Element Type** selection field of **Element Attribute**.
11. Confirm “**Equal Distance**” in the **Translation** selection field.
12. Enter “**0, 0.5, 0**” in the **dx, dy, dz** field and “**5**” in the **Number of Times** field.
13. Click .

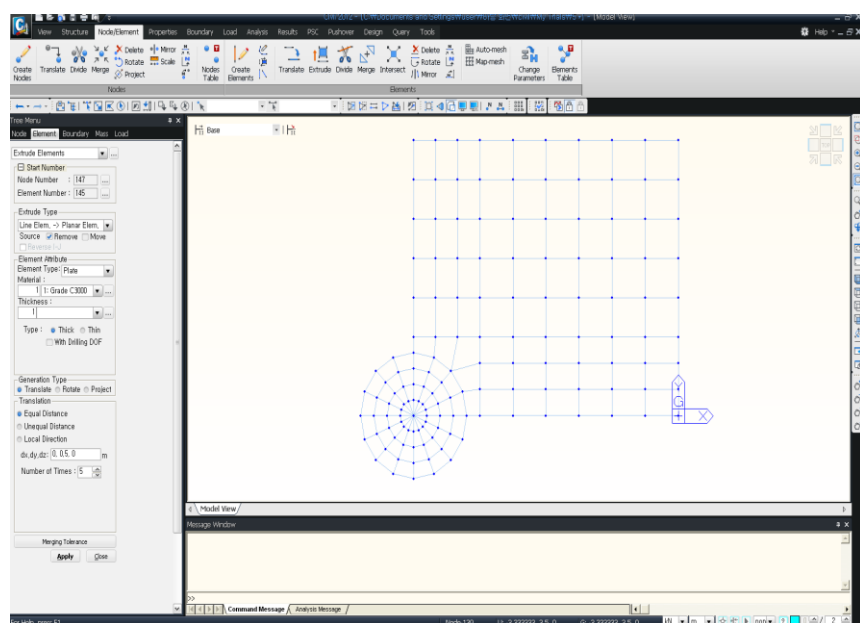


Figure 5.8 Creating Plate Elements

Use **Mirror Elements** and **Reflection** (symmetric duplication) to create the half of the footing plate.

1. Click **Group** and **Select All**.
2. Select “**Circular Column**” under **Structural Group** list on the left side of the screen and click **Unselect** with the mouse being right-clicked.
3. Click **Mirror Elements**.
4. Confirm “**Copy**” in the **Mode** selection field.
5. Select “**z-x plane**” in the **Reflection** selection field and confirm “**0**” in the **y** field.
6. Click **Apply**.
7. Click **Select Previous**.
8. Select “**y-z plane**” in the **Reflection** selection field and enter “**-4**” in the **x** field.
9. Click **Apply**.
10. Click **Select Recent Entities**.
11. Select “**z-x plane**” in the **Reflection** selection field and confirm “**0**” in the **y** field.
12. Click **Apply** (Fig. 5.9).

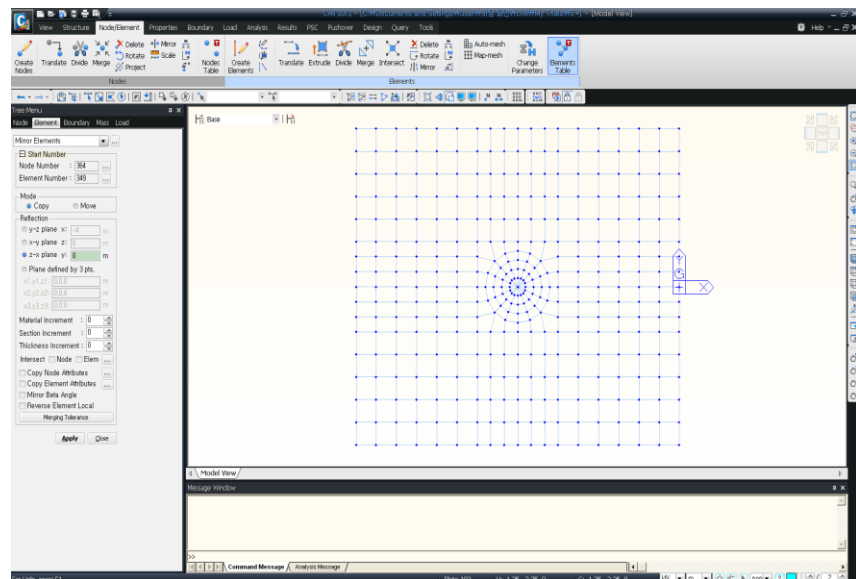




Figure 5.9 Completed Plate Elements for a half of the footing

Assign group names to different parts to facilitate the selection process during the creation of solid elements (footing, circular column, coping, etc.) extruded from the footing plate. Refer to Fig.5.10 to assign group names by areas.

1. Click  **Group**.
2. Right-click the mouse in the **Structure Group** to select “New...” and then “**Coping**” in name and “**1 to 5**” in **Suffix**.
3. Click  **Select Window** to select the relevant elements as shown in Fig.5.10.
4. From the Structure Group drag “**Coping 1**” with the mouse being left-clicked to the model window.
5. After selecting the relevant elements as per the figure, drag “**Coping 2**” with the mouse being left-clicked and drop it in the model window.
6. After selecting the relevant elements as per the figure, drag “**Coping 3**” with the mouse being left-clicked and drop it in the model window.
7. After selecting the relevant elements as per the figure, drag “**Coping 4**” with the mouse being left-clicked and drop it in the model window.
8. After selecting the relevant elements as per the figure, drag “**Coping 5**” with the mouse being left-clicked and drop it in the model window.

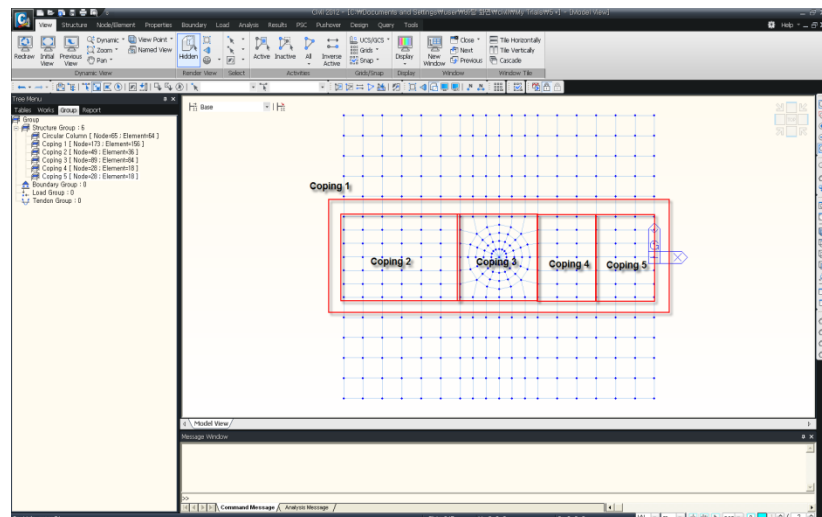









Figure 5.10 Group Definition

Use the footing plate created previously and  **Extrude Elements** to create the footing (Fig.5.11).

 **Remove Source** removes the existing source elements after using Extrude Elements. Remove Source must be unchecked if the source elements are to be used again.

1. Click  **Iso View**.
2. Click  **Select All**.
3. Click  **Extrude Elements**.
4. Select “**Planar Elem.→Solid Elem.**” in the **Extrude Type** selection field.
5. Check (✓) “**Move**” in the **Source** selection field. 
6. Confirm “**Solid**” in the **Element Type** selection field of the **Element Attribute**.
7. Confirm “**1: Grade C3000**” in the **Material** selection field.
8. Confirm “**Translate**” in the **Generation Type** selection field.
9. Confirm “**Equal Distance**” in the **Translation** selection field.
10. Enter “**0, 0, 0.5**” in the **dx, dy, dz** field and “**4**” in **Number of Times**.
11. Click  (Fig.5.11).

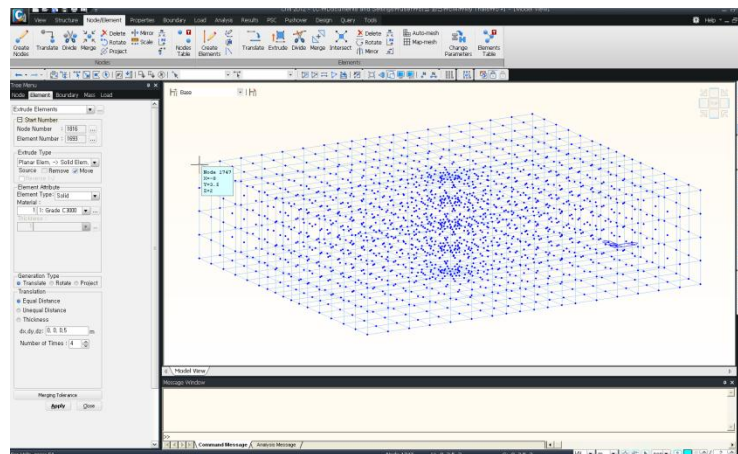





Figure 5.11 Completed Footing

Select the circular column assigned by **Group** and create the column with solid elements (Fig.5.12).

1. Click  **Group**.
2. Select “**Circular column**” in the *Structure Group* and double-click the mouse.
3. Click  **Extrude Elements**.
4. Select “**Planar Elem.→Solid Elem.**” in the *Extrude Type* selection field.
5. Remove the check (✓) in “**Remove**” in the *Source* selection field.
6. Confirm “**Solid**” in *Element Type* of the *Element Attribute* selection field.
7. Confirm “**1: Grade C3000**” in the *Material* selection field.
8. Confirm “**Translate**” in the *Generation Type* selection field.
9. Select “**Thickness**” and confirm “**Equal**” in the *Translation* selection field.
10. Enter “**12**” in the *Number of Times* field.
11. Enter “**0.5**” in the *Thickness* field.
12. Confirm “**+z**” in the *Direction* selection field.
13. Click .

Among the Extrude functions, the Thickness of Translation extrudes plate elements in the thickness direction (ECS z-direction). This is an extremely convenient feature when extruding plate elements forming a curvature.

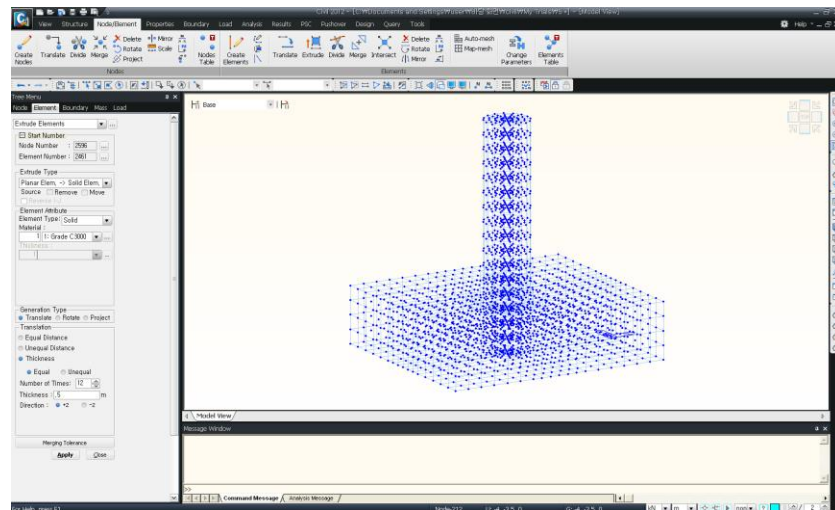





Figure 5.12 Completed Circular Column

Translate the plate elements at the top of the footing upward to the top level of the coping to extrude the coping.

1. Click  **Group**.
2. Double-click and select “**Coping 1**” in *Structure Group*.
3. Click  **Translate Elements**.
4. Select “**Move**” in the *Mode* selection field.
5. Confirm “**Equal Distance**” in the *Translation* selection field.
6. Enter “**0, 0, 8.5**” in the *dx, dy, dz* field.
7. Click .

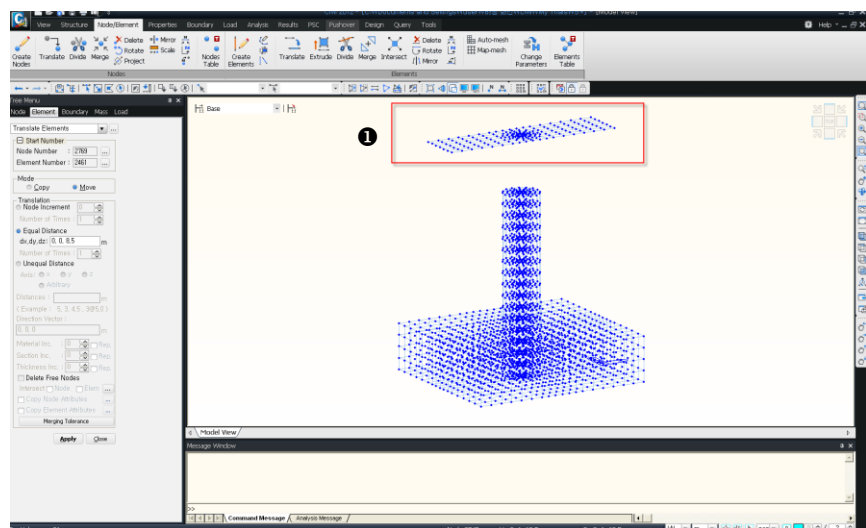




Figure 5.13 Coping Part at the Column

Activate Fig.5.13-① to initiate the modeling of the coping.

1. Click  **Select Window** and select Fig.5.13-①.
2. Click  **Active** (Fig.5.14).

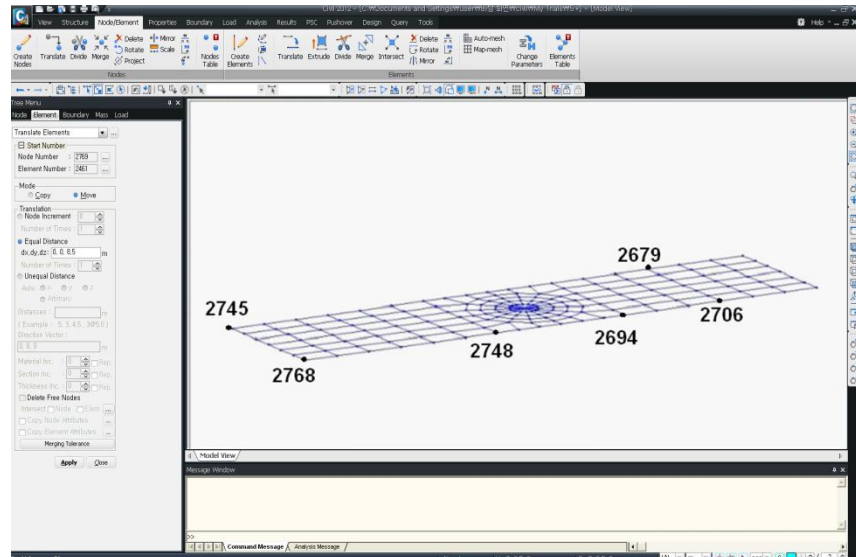



Figure 5.14 Plate Element at the top of Coping

In order to project the plate elements at the top of the coping onto the lower plane of the coping, copy the nodes corresponding to the boundaries to the level of the lower plane. The projection of the plate elements will create the solid elements (Fig.5.15).

1. Click **Translate Nodes**.
2. Click **Select Single** and select nodes **2768** and **2745** (Fig.5.14).
3. Confirm **“Copy”** in the **Mode** selection field.
4. Confirm **“Equal Distance”** in the **Translation** selection field.
5. Enter **“0, 0, -1.5”** in the **dx, dy, dz** field.
6. Confirm **“1”** in the **Number of Times** field.
7. Click **Apply**.
8. Click **Select Single** and select nodes **2748** and **2694** (Fig.5.14).
9. Confirm **“Equal Distance”** in the **Translation** selection field.
10. Enter **“0, 0, -2.5”** in the **dx, dy, dz** field.
11. Confirm **“1”** in the **Number of Times** field.
12. Click **Apply**.
13. Click **Select Single** and select nodes **2706** and **2679** (Fig.5.14).
14. Confirm **“Equal Distance”** in the **Translation** selection field.

15. Enter “**0, 0, -2**” in the dx, dy, dz field.
16. Confirm “**1**” in the *Number of Times* field.
17. Click  (Fig. 5.15 (Node Number is toggled on)).

☞ Extruding elements include Translate, Rotate and project. Translate extrudes elements in a straight line direction. Rotate extrudes elements in a circular or spiral path. Project extrudes elements about a line, plate, cylinder, cone, sphere, ellipsoid, element, etc.

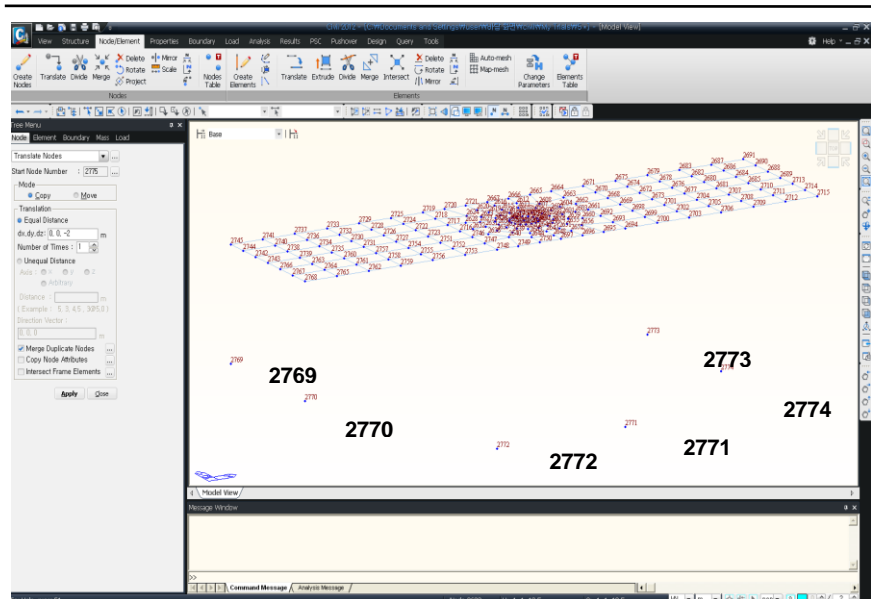













Figure 5.15 Copying Nodes for the Coping Modeling

Sort the plate elements at the top of the coping by different zones and project them onto the bottom of the coping (Fig.5.16).

1. Click  **Group**.
2. Double-click “**Coping 2**” under the *Structure Group*.
3. Click  **Extrude Elements**.
4. Select “**Planar Elem.→Solid Elem.**” in the *Extrude Type* selection field.
5. Confirm the check (✓) in “**Remove**” in the *Source* selection field.
6. Confirm “**Solid**” in *Element Type* of the *Element Attribute* selection field.
7. Confirm “**1: Grade C3000**” in the *Material* selection field.
8. Select “**Project**” in the *Generation Type* selection field. ☞
9. Select “**Project on a plane**” in the *Projection Type* selection field.
10. Use *Mouse Editor* in *Base Plane Definition* and assign Nodes **2769, 2770** and **2772** consecutively.
11. Select “**Direction Vector**” in the *Direction* selection field and enter “**0, 0, -1**”.

12. Select “**Divide**”.
13. Enter “**5**” in the *Number of Divisions* field.
14. Click .
15. Click  **Group**.
16. Double-click “**Coping 3**” under the *Structure Group*.
17. Click  **Extrude Elements**.
18. Select “**Planar Elem.→Solid Elem.**” in the *Extrude Type* selection field.
19. Confirm the check (✓) in “**Remove**” in the *Source* selection field.
20. Confirm “**Solid**” in *Element Type* of the *Element Attribute* selection field.
21. Confirm “**1: Grade C3000**” in the *Material* selection field.
22. Confirm “**Translate**” in the *Generation Type* selection field. [Ⓐ]
23. Confirm “**Equal Distance**” in the *Translation* selection field.
24. Enter “**0, 0, -0.5**” in the *dx, dy, dz* field.
25. Enter “**5**” in the *Number of Times* field.
26. Click .
27. Click  **Group**.
28. Double-click “**Coping 4**” under the *Structure Group*.
29. Click  **Extrude Elements**.
30. Select “**Planar Elem.→Solid Elem.**” in the *Extrude Type* selection field.
31. Confirm the check (✓) in “**Remove**” in the *Source* selection field.
32. Confirm “**Solid**” in *Element Type* of the *Element Attribute* selection field.
33. Confirm “**1: Grade C3000**” in the *Material* selection field.
34. Select “**Project**” in the *Generation Type* selection field.
35. Select “**Project on a plane**” in the *Projection Type* selection field.
36. Use *Mouse Editor* in the *PI* field of *Base Plane Definition* and assign nodes **2771, 2774** and **2773** consecutively. [Ⓐ]
37. Select “**Direction Vector**” in the *Direction* selection field and enter “**0, 0, -1**”.
38. Select “**Divide**”.
39. Enter “**5**” in the *Number of Divisions* field.
40. Click .
41. Click  **Group**.
42. Double-click “**Coping 5**” under the *Structure Group*.
43. Click  **Extrude Elements**.
44. Select “**Planar Elem.→Solid Elem.**” in the *Extrude Type* selection field.

[Ⓐ] Base Plane Definition refers to the plane onto which the elements are extruded.

45. Confirm the check (✓) in “**Remove**” in the *Source* selection field.
46. Confirm “**Solid**” in *Element Type* of the *Element Attribute* selection field.
47. Confirm “**1: Grade C3000**” in the *Material* selection field.
48. Confirm “**Translate**” in the *Generation Type* selection field.
49. Confirm “**Equal Distance**” in the *Translation* selection field.
50. Enter “**0, 0, -0.4**” in the *dx, dy, dz* field.
51. Confirm “**5**” in the *Number of Times* field.
52. Click **Apply**

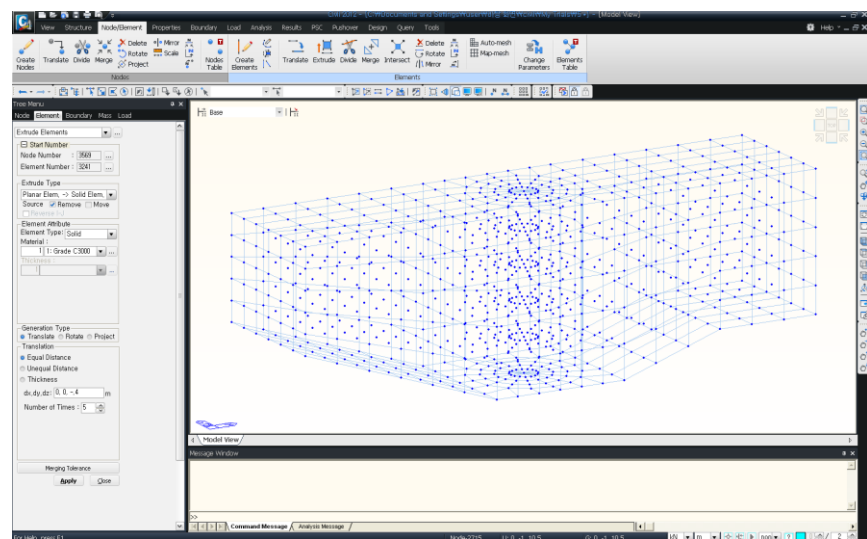


Figure 5.16 Completion of Coping

Delete all the plate elements used to create solid elements via extrude functions. Use **Mirror Elements** to duplicate the half model symmetrically to create the full model (Fig.5.18).

🔊 **Delete Elements** and the **Delete** button provide the same functional effect. However, **Delete Elements** removes the elements only by selecting the free nodes.

1. Click **Active All**.
2. Click **Select Identity-Elements**.
3. Select **"PLATE"** in the **Select Type** selection field.
4. Click **Add**.
5. Click **Close**.
6. Press **Delete** from the keyboard (Fig.5.17-1).
7. Click **Mirror Elements**.
8. Confirm **"Copy"** in the **Mode** selection field.
9. Confirm **"y-z plane"** in the **Reflection** selection field and confirm **"0"** in the **x** field.
10. Click **Select All**.
11. Click **Apply** (Fig. 5.18).

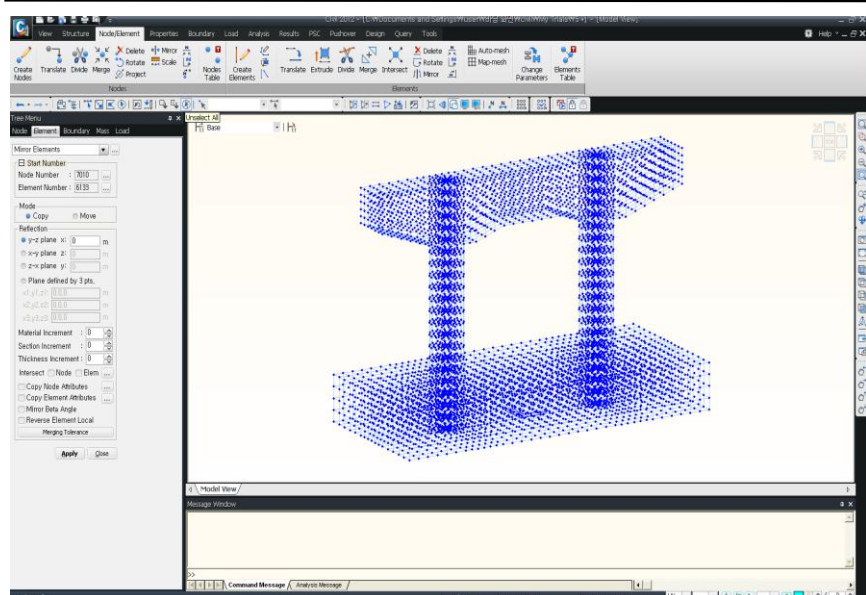


Figure 5.17 Creation of the Complete Structure

Check the current nodal connections between contiguous elements following the procedure outlined below.

☞ **Check and Remove Duplicate Elements** checks if elements are overlapped at the same locations. If this is the case, it keeps only one element and removes the redundant elements.

☞ **Free Edge** checks if the elements are properly created for the structure. If the elements split or overlap, Free Edge changes the boundary color for visual verification.

Check if elements have been overlapped at the same locations or contiguous elements sharing a common node have been miscreated during the element generation process. Remove such elements if detected.

1. Select **Structure>Check Structure>Check/Duplicate Elements** from the Main Menu.
2. Select **Structure>Check Structure>Display Free Edge/Face>Display Free Edge** from the Main Menu (Toggle on) (Fig.5.18).
3. Select **Structure>Check Structure>Display Free Edge/Face>Display Free Edge** from the Main Menu (Toggle off).

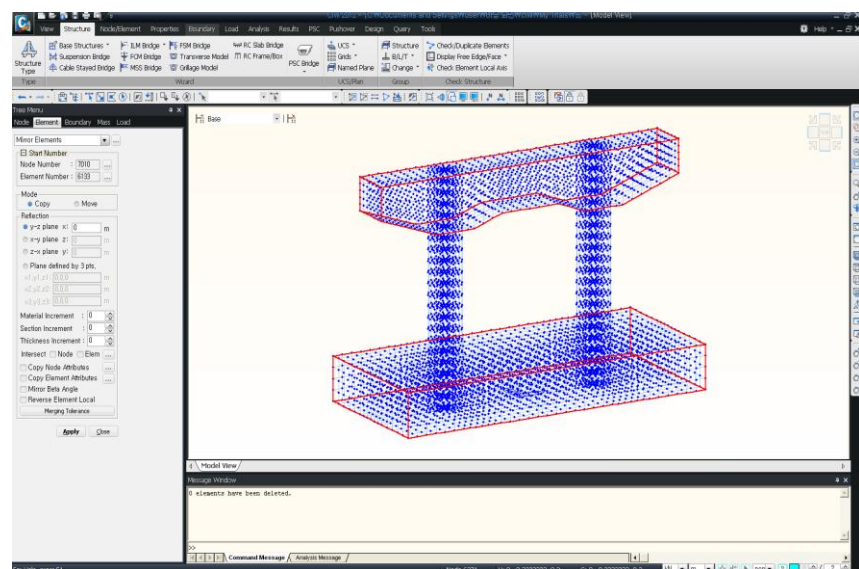





Figure 5.18 Free Edge (Toggle on)

Loading Data Entry

Prior to specifying the loads, set up the Load Cases.

1. Select **Load** tab.
2. Click the button  to the right of the **Load Case Name** dialog box.
3. Enter “**Self Weight**” in the **Name** field of the **Static Load Cases** dialog box (Fig.5.19).
4. Select “**Dead Load(D)**” in the **Type** selection field.
5. Click .
6. Enter the remaining load cases in the **Static Load Cases** dialog box as shown in Fig.5.19.
7. Click .

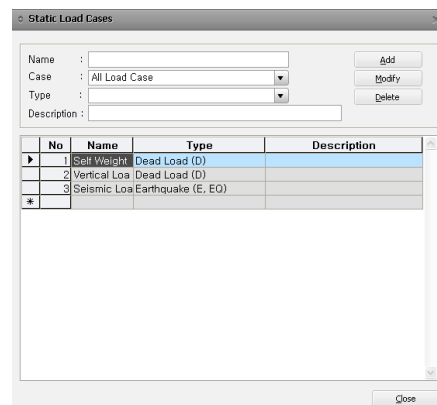










Figure 5.19 Static Load Cases dialog box

Specify the static load cases considered in this example.

1. Confirm “**Self Weight**” in the functions list of the **Load** tab.
2. Confirm “**Self Weight**” in the **Load Case Name** selection field.
3. Enter “**-1**” in the **Z** field of **Self Weight Factor**. 
4. Click  in the **Operation** selection field.

 -1 in the Z-direction in Self Weight Factor represents the action of the self-weight in the direction of gravity.

Select and activate only the nodes at the top of the structure to specify the vertical loads applied to the top (Fig.5.20).

1. Click  **Select Plane**.
2. Select “**XY Plane**” in the *Plane* tab and select a node at the top of the coping part.
3. Enter “**10.5**” in Z position.
4. Click  **Apply**.
5. Click  **Close**.
6. Click  **Active**.
7. Click  **Top View**.

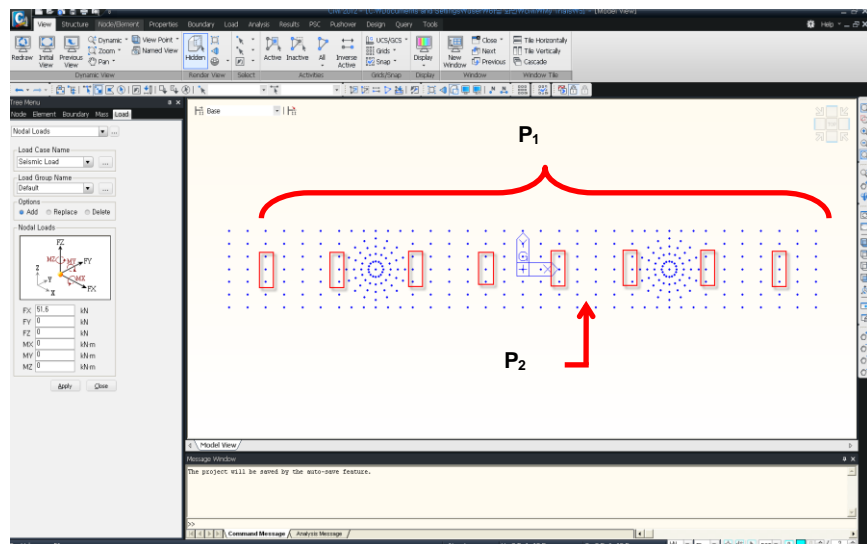








Figure 5.20 Loading Locations

The locations of the vertical and seismic loads are shown in Fig. 5.20.

1. Click  **Select Window** to select the parts loaded with P_1 (Fig.5.20).
2. Select “**Nodal Loads**” in the **Load** tab.
3. Select “**Vertical Load**” in the **Load Case Name** selection field.
4. Confirm “**Add**” in the **Options** selection field.
5. Enter “**-430**” in the **FZ** field of **Nodal Loads**.
6. Confirm “**0**” in the remaining fields of **Nodal Loads**.
7. Click .
8. Click  **Select Window** to select the parts loaded with P_2 (Fig.5.20).
9. Select “**Seismic Loads**” in the **Load Case Name** selection field.
10. Confirm “**Add**” in the **Options** selection field.
11. Enter “**520**” in the **FX** field of **Nodal Loads**.
12. Confirm “**0**” in the remaining fields of **Nodal Loads**.
13. Click .
14. Click  **Iso View**.
15. Click  **Display > Load Tab, then Check Nodal Load** (Fig.5.21).

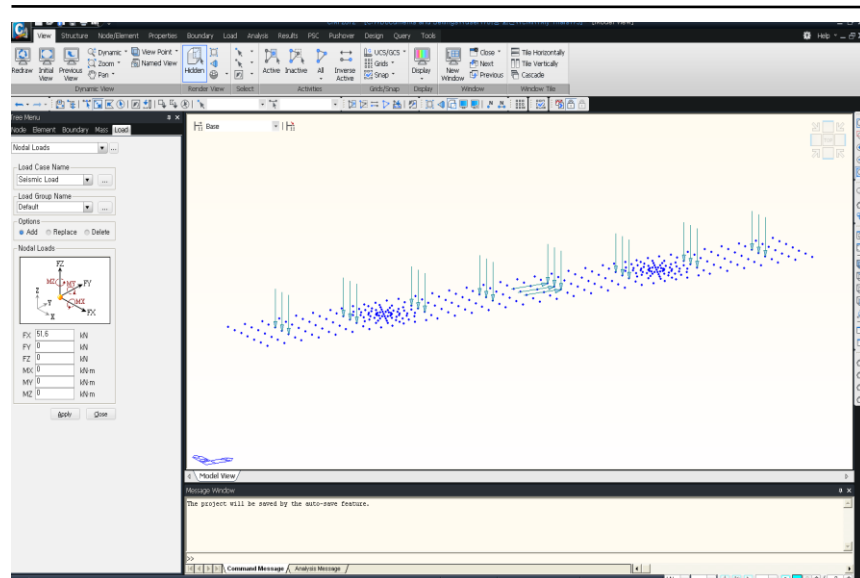








Figure 5.21 Display of Vertical and Seismic Loads

Enter the boundary conditions.

1. Click  **Active All**.
2. Click  **Select Plane**.
3. Select “**XY Plane**” in the *Plane* tab.
4. Enter “**0**” in *Z Position*.
5. Click .
6. Click .
7. Select “**Boundary**” tab and confirm “**Supports**”.
8. Confirm “**Add**” in the *Options* selection field.
9. Check (✓) “**D-All**” in the *Support Type (Local Direction)*.
10. Click  (Fig. 5.22).
11. Confirm the node entries for supports and click  **Redraw** (Fig.5.22).

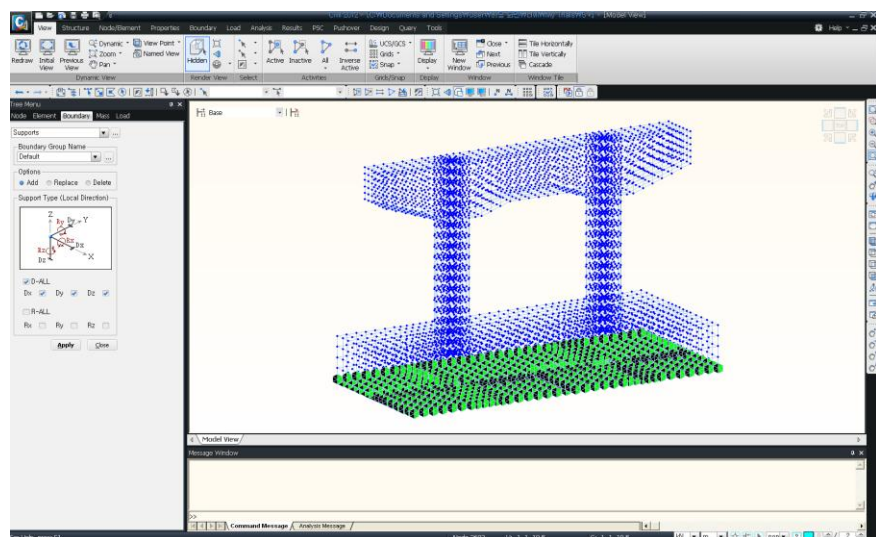




Figure 5.22. Completed Structure

Perform Structural Analysis

Analyze the structure with the load cases provided.

1. Select **Analysis>Analysis Options** in the Main Menu.
2. Confirm “**Multi Frontal Sparse Gaussian**” in the **Equation Solver**.
3. Click .
4. Click  **Analysis**.

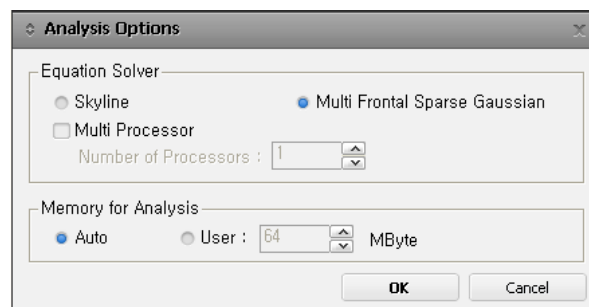


Figure 5.23 Selection of Analysis Method

Verification and Interpretation of Analysis Results

Load Combination

We will examine the **Linear Load Combination** method for the 3 load cases (self-weight, vertical loads and seismic loads) after the structural analysis has been completed.

In this example, specify only one load combination case for simplicity and check the results thereof. The load combination case has been arbitrarily chosen and may differ from practical design applications.

- Load Combination 1 (LCB1): 1.0 (Self-Weight + Vertical Loads + Seismic Loads)

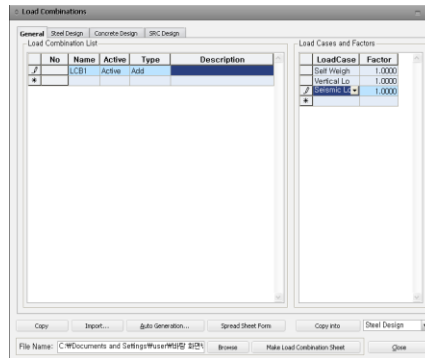


Figure 5.24 Load Combinations dialog box

Use **Results>Load Combinations** from the Main Menu to open the **Load Combinations** dialog box (Fig.5.24) and specify the load combination as below.

1. Select **Results>Load Combinations** in the Main Menu.
2. Enter “**LCB1**” in the **Name** field of **Load Combination List**.
3. Confirm “**Add**” in the **Type** selection field.
4. Click the **LoadCase** selection field and use ▼ to select “**Self Weight(ST)**” and confirm “**1.0**” in the **Factor** field.
5. Click the second selection field and use ▼ to select “**Vertical Load(ST)**” and confirm “**1.0**” in the **Factor** field.
6. Click the third selection field and use ▼ to select “**Seismic Load(ST)**” and confirm “**1.0**” in the **Factor** field.
7. Click **Close**

Check the Deformed Shape

Use the following procedure to review the deformed shape (Fig.5.26):

1. Select **Results>Deformations>Displacement Contour** in the Main Menu.
2. Select “**CB: LCB1**” in the **Load Cases/Combinations** selection field.
3. Confirm “**DXYZ**” in the **Components** selection field.
4. Check (✓) “**Contour**”, “**Deform**” and “**Legend**” in the **Type of Display** selection field.
5. Click **Apply**

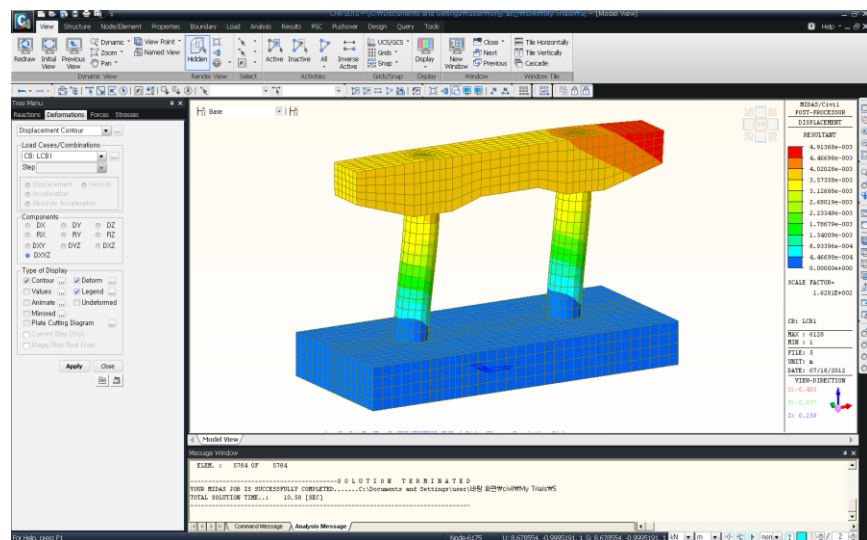




Figure 5.25 Displacement Contour

Check the Stresses

Check the stresses in solid elements.

1. Select  **Solid Stresses** in the *Stresses* tab of the Tree Menu.
2. Confirm “**CB: LCB1**” in the *Load Cases/Combinations* selection field.
3. Confirm “**UCS**” and “**Avg. Nodal**” in the *Stress Options* selection field.
4. Select “**Sig-Pmax**” in the *Components* selection field.
5. Check (✓) “**Contour**” and “**Legend**” in the *Type of Display* selection field.
6. Click 

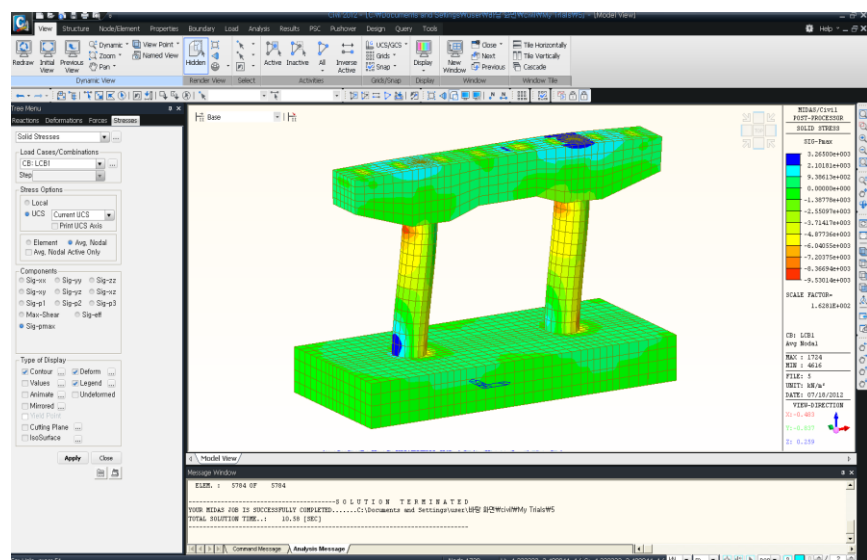






Figure 5.26 Resulting Stresses

Use  **Zoom Dynamic**,  **Rotate Dynamic**,  **Render View** and  **Perspective** to select the display of the resulting stresses with different view ports (Fig.5.27).

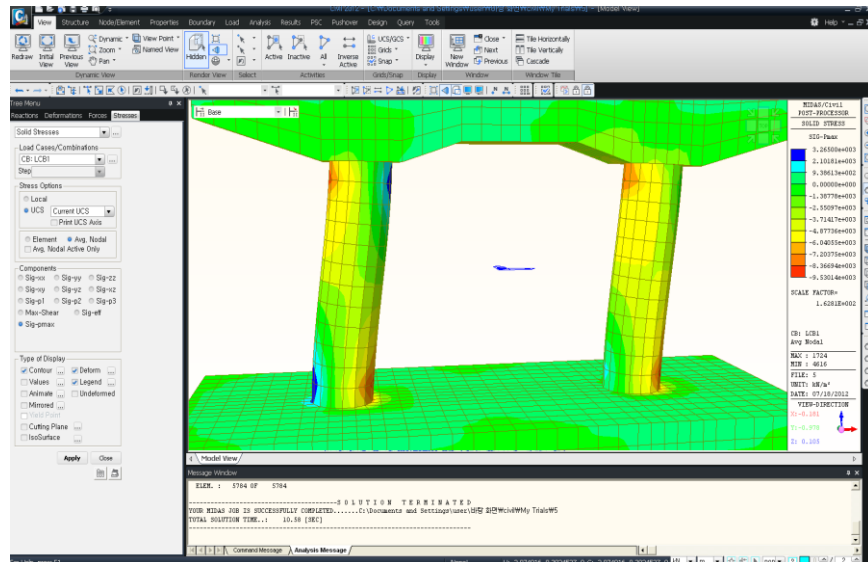




Fig.5.27 View from the Bottom of the Pier

Check the stress distribution relative to a specific cutting plane of the solid elements.

Define the plane first.

1. Select **Structure>Named Plane** from the Main Menu.
2. Enter “**plane 1**” in the **Plane Name** field.
3. Select “**Y-Z plane**” in the **Plane Type** selection field.
4. Enter “**-4**” in the **X Position** field.⁶
5. Confirm “**0.001**” in the **Tolerance** field.
6. Click  in the **Operations** selection field (Fig. 5.28).

 Use Mouse Editor to define the desired Y-Z plane among an infinite number of Y-Z planes.

Named Plane

Plane Name : plane 1

Plane Type

☐ 3 Points

☐ X-Y Plane

☐ X-Z Plane

☒ Y-Z Plane

X Position : -4 m

Tolerance : 0,001 m




Plane Name	Plane Type
plane 1	Y-Z Plane

Operations

Add Modify Delete

Close

Figure 5.28 Named Plane

-
1. Select **Results>Stresses>Solid Stresses** in the Main Menu.
 2. Confirm “**CB: LCB1**” in the **Load Cases/Combinations** selection field.
 3. Select “**UCS**” and “**Avg. Nodal**” in the **Stress Options** selection field.
 4. Select “**Sig-Pmax**” in the **Components** selection field.
 5. Check (✓) “**Contour**”, “**Legend**” and “**Cutting Plane**” in the **Type of Display** selection field.
 6. Click the button  to the right of the **Cutting Plane** selection field (Fig.5.30).
 7. Check (✓) “**plane 1**” and “**Current UCS x-z Plane**” in the **Named Planes for Cutting** selection field.
 8. Select “**Free Face**”.
 9. Click  in the **Cutting Plane Detail Dialog** window to exit.
 10. Click .
-

Click

Ani. Option Show

(Fig.5.29-1) to select the motion direction of the animation which illustrates the resulting stresses on the cutting planes.

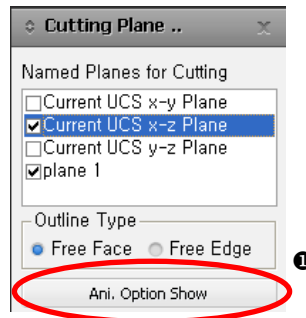


Figure 5.29 Cutting Plane Detail dialog box

Selecting On Cutting Plane allows the user to check the sectional results visually on the defined planes as opposed to reviewing the results on the elements.

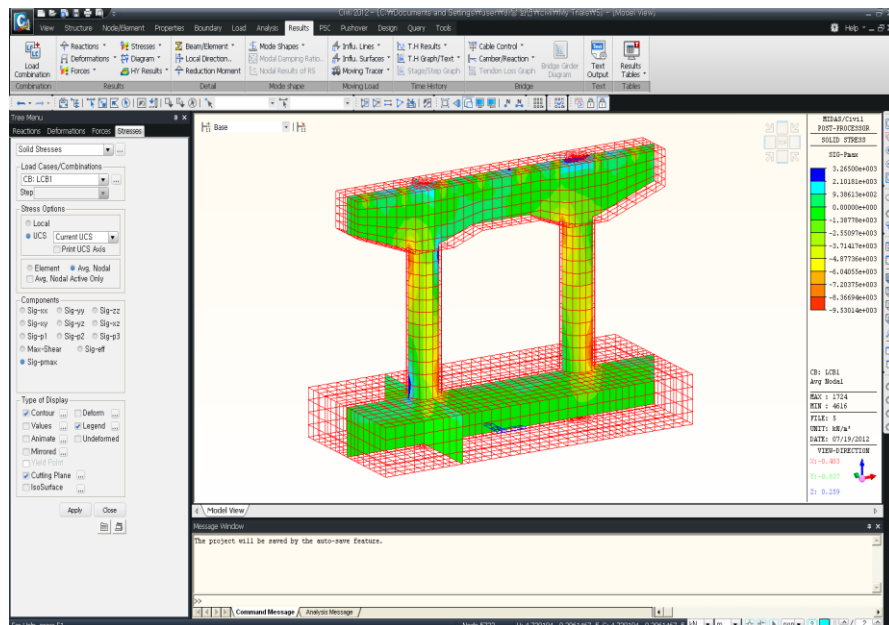


Figure 5.30 Resulting Stresses On Cutting Planes

Local Direction Force Sum displays the member forces of a specific plane using the nodal results. It is effective when checking the member forces of solid or plate elements.

Finally, check the results of **Local Direction Force Sum**.

1. Click  **Initial View**.
2. Click  **Front View**.

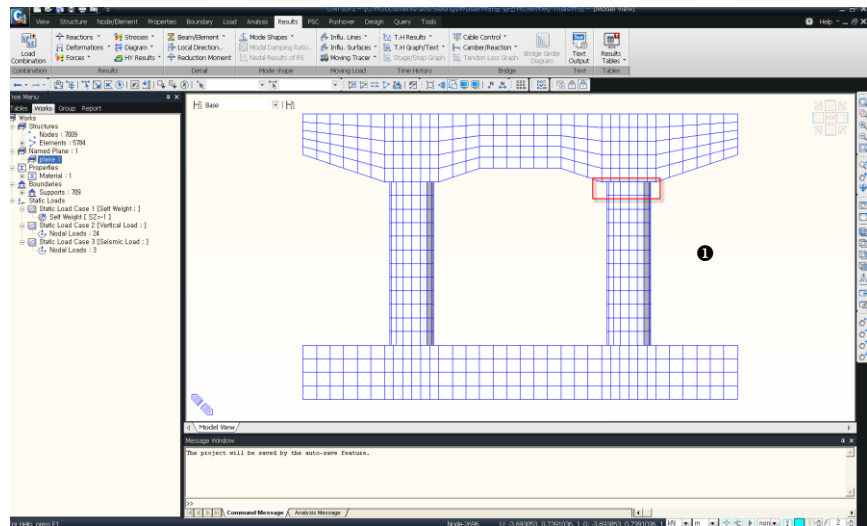






Figure 5.31 Structure with GCS

1. Click  **Select Window** to select the relevant elements (Fig.5.31-①).
2. Click  **Active**.
3. Click  **Iso View**.
4. Select **Results>Local Direction Force Sum** in the Main Menu.
5. Select “**Solid Face Polygon Select**” in the **Mode** selection field.
6. Confirm “**CB: LCB1**” in the **LoadCase** selection field.
7. Enter “**1**” in the **Tolerance** field.
8. Use **Mouse Editor** in the **Coordinate Input** field to mark a closed polygon, including the relevant section, in the counterclockwise direction.
9. Confirm the removal of the check (✓) in “**z Vector**”.
10. Click .

☞ The direction of creating a closed polygon becomes the x-direction of the new coordinate system.

☞ If z-Vector is not checked (✓), the direction of the first edge of the polygon becomes the z-direction. If z-Vector is checked (✓), the z-direction can be defined separately on the relevant plane.

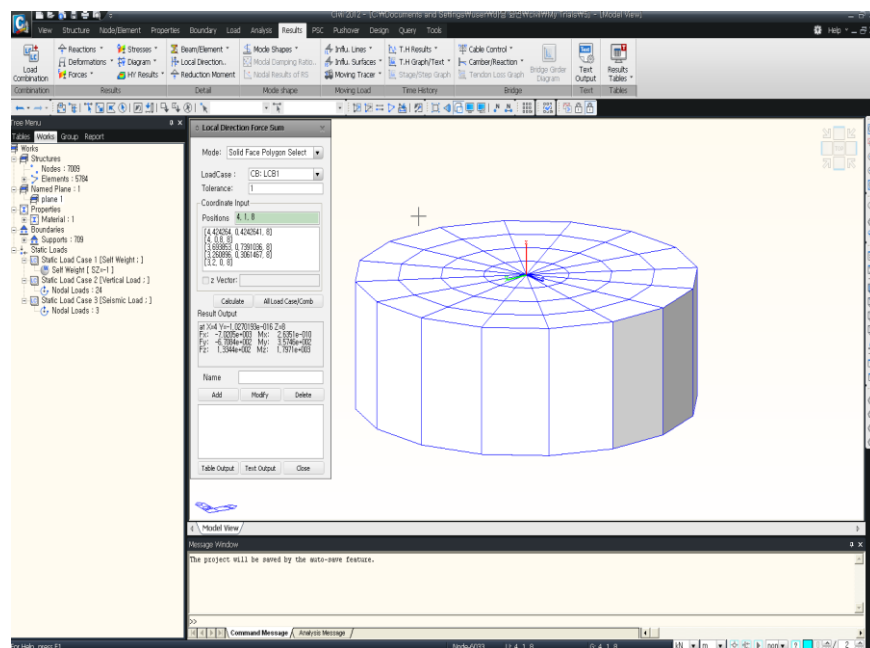


Figure 5.32 Member Forces at the Coping-Column Joint Iso View