

Tutorial 3

WEB-OPENING DETAIL ANALYSIS

Civil

TUTORIAL 3. WEB-OPENING DETAIL ANALYSIS

Summary	1
Analysis Model and Load Cases / 2	
Preferences Setting	3
Unit System / 3	
Enter Material and Section Properties	4
Structural Modeling	5
Enter Structure Support Conditions	18
Enter Loading Data	21
Define Load Cases / 21	
Define Uniformly Distributed Load / 21	
Define Concentrated Loads / 22	
Perform Structural Analysis	24
Interpret Analysis Results	25
Verify Member Stresses / 25	
Auto-Compute Member Stresses / 26	

TUTORIAL 3.

WEB-OPENING DETAIL ANALYSIS

Summary

This tutorial presents the modeling and analysis processes for the reinforcement design of a beam with a circular web opening and explains the procedure for verifying results.

The essential contents for the user to experience in the example are the following:

- Detail modeling using plate elements to study the stress distribution around the vicinity of the opening
- Method of using **Rigid Link** for the structural link between the opening detail model and the model of the remaining parts with beam elements
- Method to extract the analysis results for plate elements

Extrude Elements (extension function which transforms nodes into line elements, line elements into plate elements and plate elements into solid elements) is used for the detail modeling of the opening. **Extrude Elements** is an extremely efficient tool to model complicated plate or 3-D models with minimal effort.

-
1. Preferences Setting
 2. Enter Material and Section Properties
 3. Structural Modeling
 4. Enter Structure Support conditions
 5. Enter Loading Data
 6. Perform Structural Analysis
 7. Interpret Analysis Results
-

Analysis Model and Load Cases

The summary and load cases for the structural model are shown in Fig.3.1.

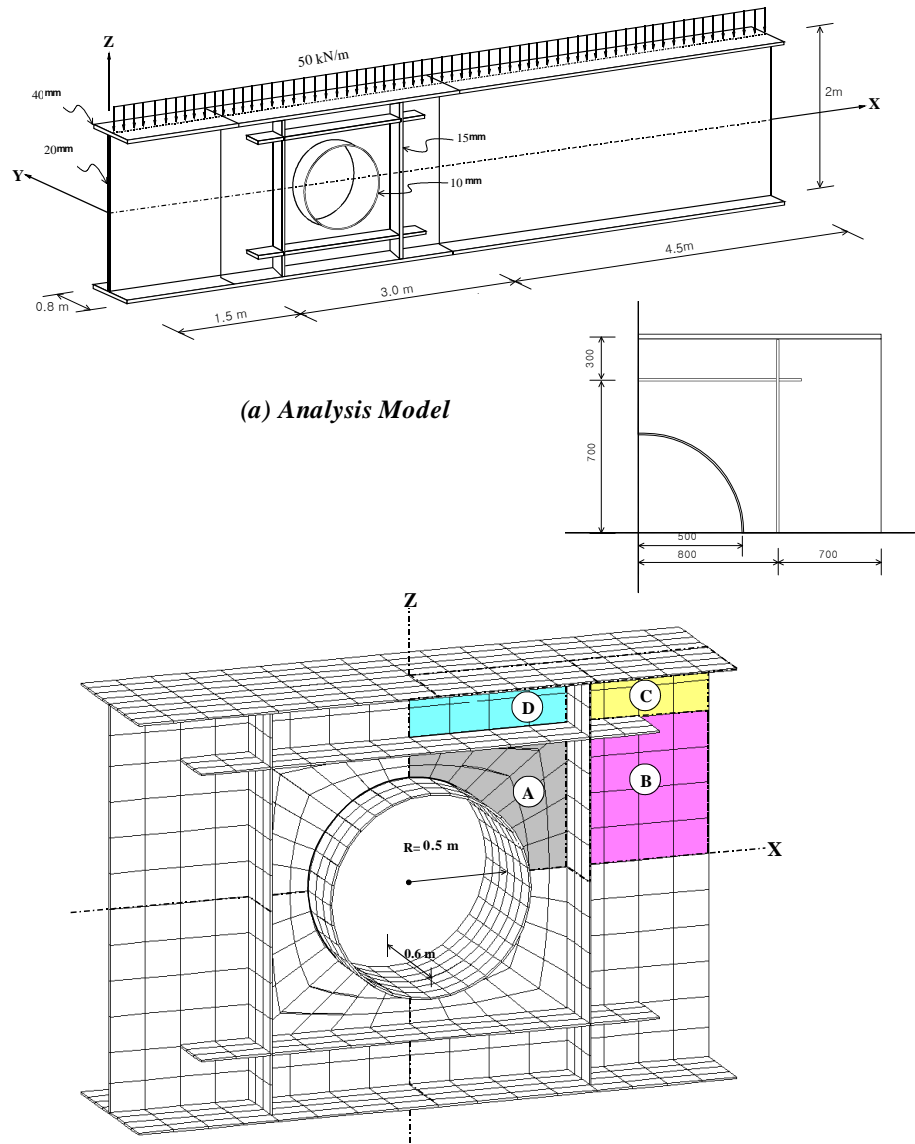





Figure 3.1 Beam Member with a Circular Web-Opening and a Detail Model


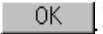


Preferences Setting



Unit System

First, open a new file. Then, use **Tools>Unit System** to specify the unit system adopted for the model.

1. Select **Tools>Unit System** in the main menu.
2. Select “**mm**” in the **Length** selection field.
3. Select “**N(kg)**” in the **Force (Mass)** selection field.
4. Click .

For data entry and results verification, model the structure such that the beam ECS corresponds to the GCS. In other words, set  **X-Z** to coincide with the web plane which is on the UCS x-y plane, and click  **Front View** to adjust the working plane to correspond to the UCS x-y plane.








1. Click  **X-Z** in **Structure > UCS > X-Z Plane** from the Main Menu.
2. Enter “**0, 0, 0**” in the **Origin** field.
3. Enter “**0**” in the **Angle** field.
4. Click  .
5. Click  **Front View** in the Icon Menu.


 If you check (✓)
Change View Direction
at the bottom of the X-Z
Plane field and click
, the step 5 can
be omitted.


Enter Material and Section Properties

Assign the material properties for the beam and the thickness for all the parts such as vertical and horizontal stiffeners, the flange of opening reinforcing, etc.

- Material Number 1: Steel (A36)
- Thickness Number 10: 10 mm (Pipe)
 15: 15 mm (Stiffeners)
 20: 20 mm (Web)
 40: 40 mm (Flange)

-
1. Select **Properties>Material Properties** in the Main Menu.
 2. Click  under the **Material** tab.
 3. Select “**ASTM(S)**” in the **Standard** selection field.
 4. Select “**A36**” in the **DB** selection field.
 5. Click .
 6. Select the **Thickness** tab at the top of the **Properties** dialog box.
 7. Click .
 8. Enter “**10**” in **Thickness ID** and “**10**” in **In-plane & Out-of-plane**.
 9. Click .
 10. Repeat steps 8 and 9 to enter successively thickness numbers “**15**”, “**20**” and “**40**”, and click .
 11. Select “**m**” in the unit system conversion window of the **Status Bar**.
 12. Click .

 When the unit system is changed, the existing unit system for thicknesses will reflect the new unit system. The screen will then display the change.

 Grid is not used in Tutorial 3. Toggle off all the Icons related to Grid.












Toggle on    

Structural Modeling

Generate 9 reference nodes in the UCS x-y plane to define the circular opening size and to locate the reinforcement (horizontal and vertical stiffeners).

The remaining zone including the circular opening is symmetrical about both axes. Only the upper-right quarter is modeled due to its symmetry (Fig. 3.1–©). The remaining 3 quarters are completed using symmetry copy (**Mirror Elements**).

When typing the coordinates or distances directly in the data field, insert “ ” (blank) or “,” to distinguish consecutive entries.

1. Click  **Node Number** and  **Element Number** in the Icon Menu (Toggle on).
2. Click  **Auto Fitting** in the Icon Menu.
3. Select **Node/Element > Create Nodes** in the Main Menu.
4. Enter “**0, 0, 0**” in the **Coordinates** (x, y, z) field. 
5. Click .
6. Select **Translate Nodes** in the functions selection field (Fig.3.2–①).
7. Click  **Select All** in the Icon Menu.
8. Confirm “**Copy**” in the **Mode** selection field.
9. Select “**Unequal Distance**” from the **Translation** selection field.
10. Confirm “**x**” in the **Axis** selection field.
11. Enter “**0.8, 0.7**” in the **Distance** field.
12. Click .
13. Click  **Select All** in the Icon Menu.
14. Select “**y**” in the **Axis** selection field of **Unequal Distance**.
15. Enter “**0.7, 0.3**” in the **Distance** field.
16. Click .
17. Click  **Select Window** in the Icon Menu and select node **1**.
18. Select “**Move**” in the **Mode** selection field.
19. Select “**x**” in the **Axis** selection field.
20. Enter “**0.5**” in the **Distance** field and click .

Toggle on       

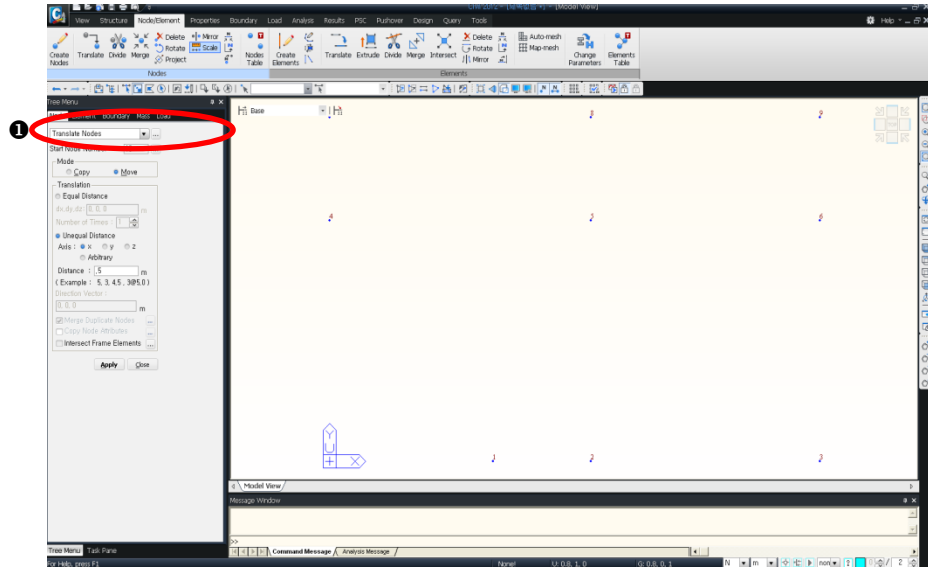





Figure 3.2 Generation of Nodes for Element Positions

While duplicating the nodes consecutively, use  **Extrude Elements** to generate elements concurrently to model beam elements for pipe-shaped stiffeners around the circumference of the opening.

These beam elements are used subsequently for the generation of the pipe-shaped stiffeners using **Extrude**, which expands the beam elements into plate elements.

1. Select **Element** tab in the Tree Menu or (Node/Element > Elements > Extrude from the Main Menu) (Fig.3.3–①).
2. Select  **Extrude Elements** in the functions selection field.
3. Confirm “**Node→Line Element**” in the **Extrude Type** selection field.
4. Click  **Select Window** in the Icon Menu and select node **1**.
5. Confirm “**Beam**” in the **Element Type** selection field.
6. Select “**1 : A36**” in the **Material** selection field.

The section number 999 for the beam elements is removed automatically after they have been extruded into plate elements. As such, it is not required to enter the section shape or dimensions.

7. Enter the section number “999” in the *Section* field.
8. Select “**Rotate**” in the *Generation Type* selection field.
9. Enter “8” in the *Number of Times* field.
10. Enter “90/8” in the *Angle of Rotation* field.
11. Select “**z-axis**” in the *Axis of Rotation* selection field.
12. Confirm “0, 0, 0” in the *1st Point* field.
13. Click **Apply**.

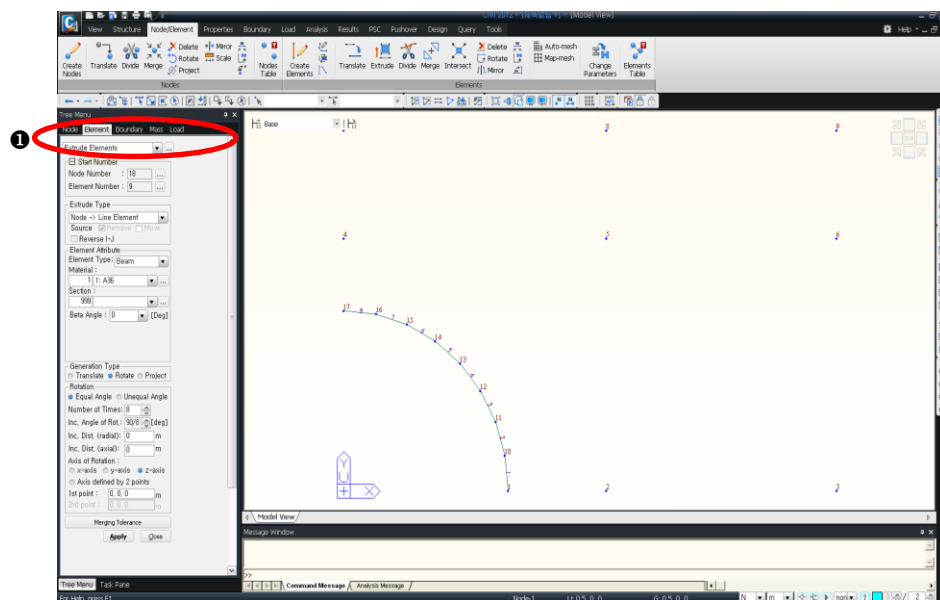



Figure 3.3 Generation of Temporary Beam Elements around the Opening Circumference

To create 8 plate elements in area \mathcal{A} of Fig.3.1(c), the lines between nodes 2 and 5 and nodes 4 and 5 are divided into 4 equal spacings.

1. Click  **Auto Fitting** (Toggle off).
2. Select the **Node** tab in the Tree Menu (Fig.3.4-①).
3. Select **Divide Nodes** in the functions selection field.
4. Enter “4” in the **Number of Divisions** field of **Equal Distance**.
5. Click the **Nodes to Divide** field once and click successively nodes **2** and **5** and nodes **4** and **5**.

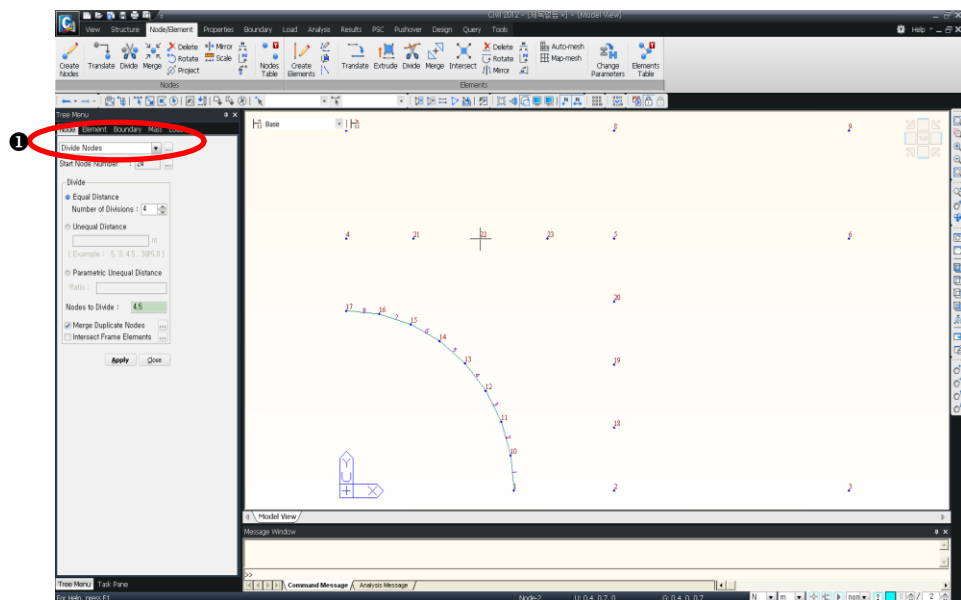






Figure 3.4 Division of nodes to create Plate Elements

Connect the generated nodes counter-clockwise to create the 8 plate elements in area ㉔ of Fig.3.1(c). The ECS thus-created consistently enables the user to use  **Divide Elements** effectively when dividing the elements afterwards.

1. Select the **Element** tab, then **Create Elements** from the Tree Menu or select **Node/Element** >  **Create Elements** from the Main Menu (Fig.3.5—① and ②).
2. Select “**Plate**” in the **Element Type** selection field and confirm “**4 Nodes**”.
3. Confirm “**1: A36**” in the **Material Name** selection field.
4. Enter “**20**” in the **Thickness No.** field.
5. Click the **Nodal Connectivity** field and connect nodes **1, 2, 18, 10** to create plate element **9**.
6. Connect nodes **10, 18, 19, 11** to create plate element **10**.
7. Similarly, create successively the remaining plate elements **11** to **16**.
8. Click  **Shrink** in the Icon Menu (Toggle on).
9. Click  **Zoom out**.

Use the Size tab of Display Option to adjust Zoom In and Zoom Out Factor.

The default setting for midas Civil (Grid, Snap, DB, etc.) can be modified in the Tools> Preferences menu for user convenience.

Toggle on        

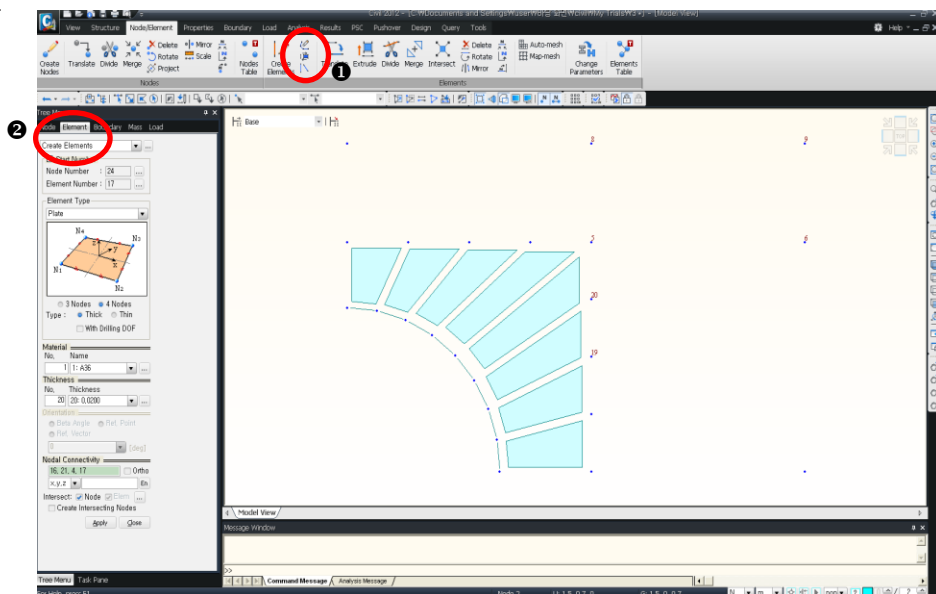


Figure 3.5 Generation of Plate Elements around the Circular Opening

Create 3 plate elements forming the boundaries of the ⑥, ⑦, ⑧ zones as shown in Fig.3.1(c) by connecting the corner nodes.

⚠ ECS is defined according to the order in which nodes are assigned during the generation of elements. It is advisable to follow a consistent order at all times. Refer to Model Numerical Analysis > Types of elements and related items > Plate Elements in Analysis & Design Manual for the ECS.

1. Click **Intersect Node** to remove the check (✓).
2. Connect nodes **2, 3, 6, 5** to create plate element **17**. ⚠
3. Connect nodes **5, 6, 9, 8** to create plate element **18**.
4. Connect nodes **4, 5, 8, 7** to create plate element **19**.

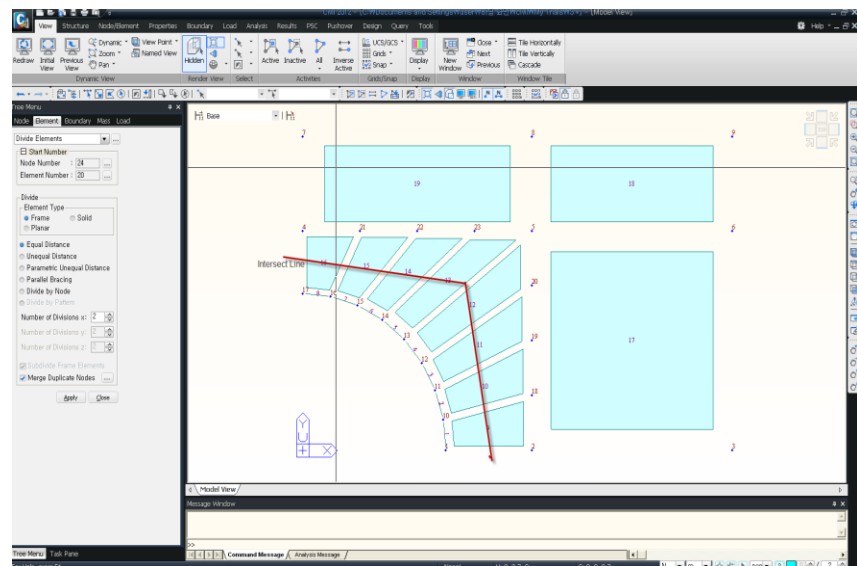






Figure 3.6 Generation of the Remaining Plate Elements of the Web

Divide the plate elements already created into appropriate sizes to form fine meshes.

⚠ To finish your selection of elements, using the Intersect Line command, double-click at the last point.

1. Select **Divide Elements** in the functions selection field (Fig.3.7–①).
2. Use **Select Intersect** in **View > Select > Intersect Line** from the Main Menu to select the plate elements **9 to 16** in area ④ (Fig.3.6) ⚠.

3. Select “**Planar**” in the *Element Type* selection field.
4. Confirm “**Equal Distance**” in the *Divide* selection field.
5. Enter “**3**” in the *Number of Divisions x* field (“3” means one element into three elements).
6. Enter “**1**” in the *Number of Divisions y* field (“1” means one element into one element).
7. Click .
8. Click  **Select Single** in the Icon Menu to select element **17** of area ⑧.
9. Enter “**4**” in both the *Number of Divisions x* and *y* fields.
10. Click .
11. Select elements **18** and **19** in areas ⑨ and ⑩ respectively.
12. Confirm “**4**” in the *Number of Divisions x* field.
13. Enter “**2**” in the *Number of Divisions y* field.
14. Click .

Toggle on 

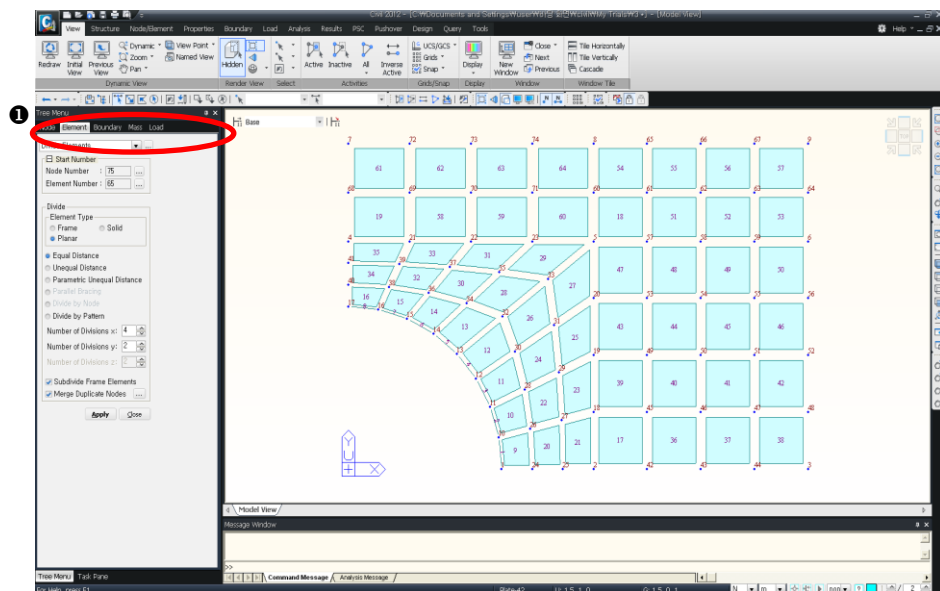


Figure 3.7 Division of Web Plate Elements

Create temporary beam elements at the locations of the reinforcing stiffeners and the flanges in order to generate the vertical and horizontal stiffeners and plate elements by extruding the beam elements into plate elements.

1. Select **Create Elements** in the functions selection field (Fig.3.8–❶).
2. Select “**General beam/Tapered beam**” in the **Element Type** selection field.
3. Enter section number “**998**” in the **Section No.** field.
4. Check (✓) **Intersect Node**.
5. Click the **Nodal Connectivity** field once and connect nodes **4** and **58** to generate the temporary beams.
6. Connect nodes **2** and **8** to generate the temporary beams.
7. Enter section number “**997**” in the **Section No.** field.
8. Click the **Nodal Connectivity** field once and connect nodes **7** and **9** to create temporary beams at the upper flange position.

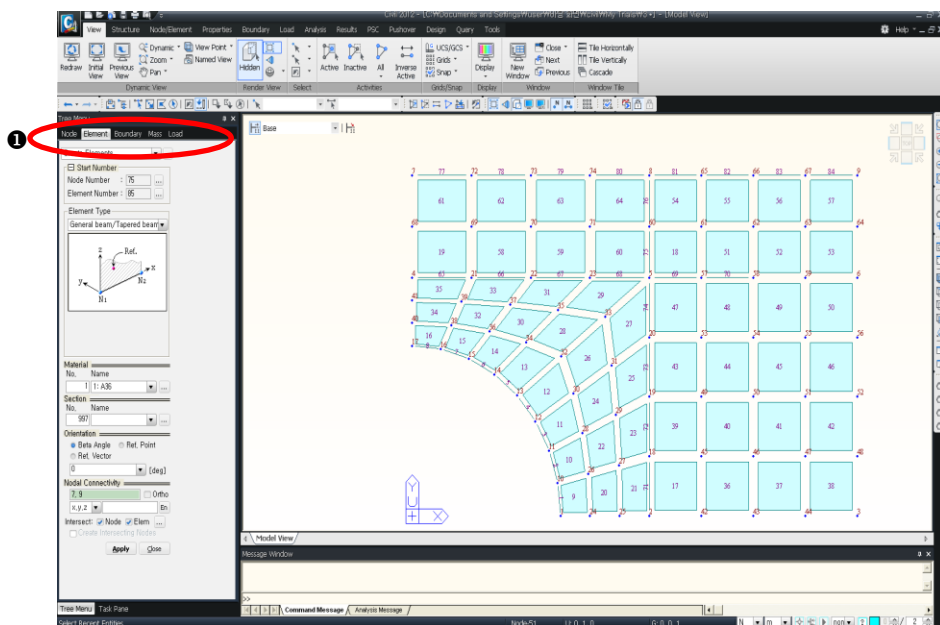


Figure 3.8 Generation of Beam Elements at the Reinforcing and Flange Plates

Use **Mirror Elements** to generate the elements in the remaining 3 quarters of the opening detail model.

1. Click **Node Number** and **Element Number** in the Icon Menu (Toggle off).
2. Click **Select All** and **Auto Fitting** in the Icon Menu.
3. Select **Mirror Elements** in the functions selection field (Fig.3.9-①).
4. Confirm “Copy” in the **Mode** selection field.
5. Select “z-x plane” in the **Reflection** selection field.
6. Confirm “y: 0” and click **Apply**.
7. Click **Select All** in the Icon Menu.
8. Select “y-z plane” in the **Reflection** selection field.
9. Confirm “x: 0” and click **Apply**.
10. Click **Close**.

⚠ when selecting plane that you are mirroring elements about, consider the plane as the perpendicular plane to the plane you want to mirror.

Ex) The first mirroring work here is to mirror elements in the x-y plane (UCS). For this case, the perpendicular plane is the x-z plane, therefore, Step 5 chooses the z-x plane in the reflection selection field.

Toggle on

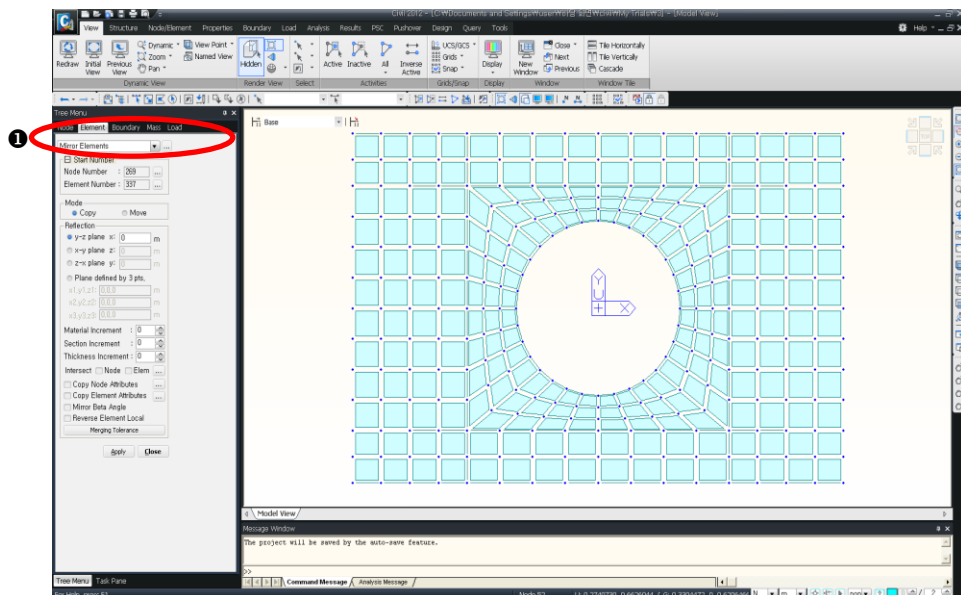











Figure 3.9 Complete Model of the Web

Extrude the temporary beam elements into plate elements to complete the reinforcing flange of the circular opening, the vertical and horizontal stiffeners and the flanges of the beam as shown in Fig.3.11.

-
1. Click  **Iso View** in the Icon Menu.
 2. Click  **GCS** in **View > Grids/Snap > UCS/GCS** from the Main Menu.
 3. Select the **Works** tab (Fig.3.10–①).
 4. Double-click section number ‘**999**’ (pipe-shaped stiffener) in **Properties > Section**.
 5. Click  **Extrude Elements** in the **Main Menu > Node/Element > Extrude**.
 6. Select “**Line Elem.→Planar Elem.**” in the **Extrude Type** selection field.
 7. Select “**10: 0.010000**” in the **Thickness** selection field.
 8. Confirm “**Translate**” in the **Generation Type** selection field.
 9. Type “**0, -0.1, 0**” in the **dx, dy, dz** field of **Equal Distance**.
 10. Enter “**3**” in the **Number of Times** field.
 11. Click .
 12. Click  **Select Identity-Elements** in the Icon Menu.
 13. Select “**Section**” in the Select Type field.
 14. Click section number “**998**” (vertical, horizontal stiffeners).
 15. Click .
 16. Click .
 17. Select “**15: 0.015000**” in the **Thickness** selection field.
 18. Click .
 19. Repeat steps 12 to 16 to enter section number “**997**” (flange of the beam).
 20. Select “**40: 0.040000**” in the **Thickness** selection field.
 21. Enter “**4**” in the **Number of Times** field.
 22. Click .
-

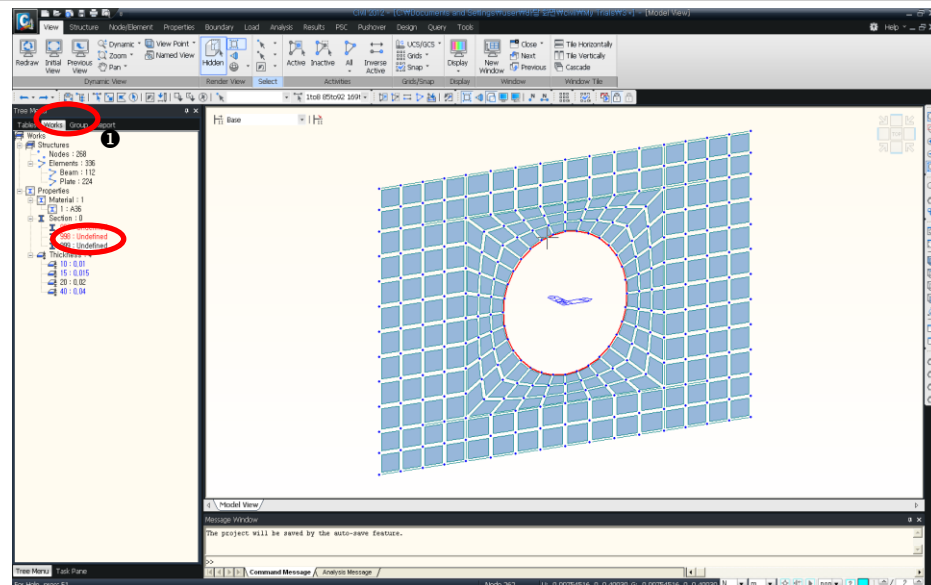


Figure 3.10 Section selection using Works Tree

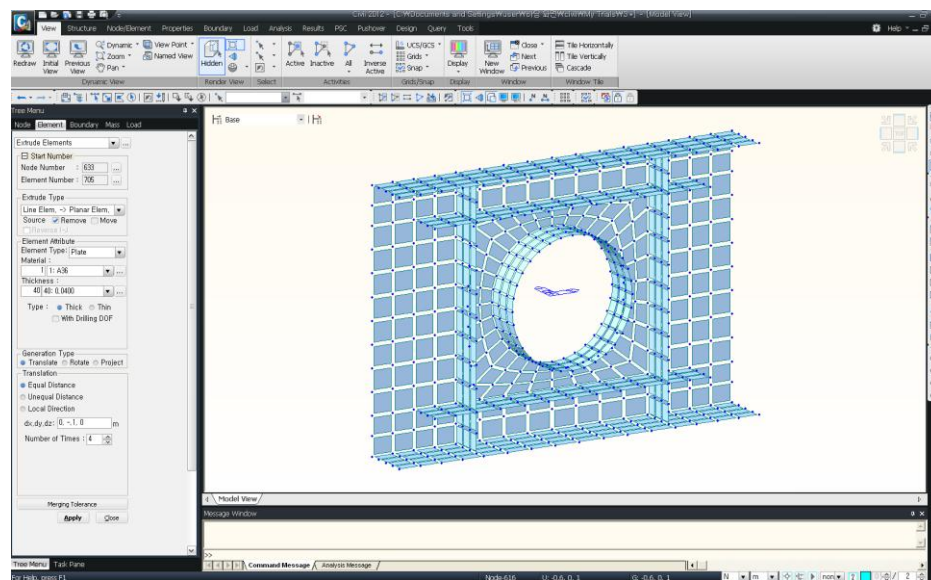


Figure 3.11 Complete One Side of the Opening Detail Model

To generate the flanges and stiffeners of the opposite face, select all the parts, except for the web, and use **Mirror Elements** to complete the opening detail model.

1. Click **Select All** in the Icon Menu.
2. After selecting the thickness number “**20: 0.02**” in the **Tree Menu > Works tab > Properties>Thickness**, right-click the mouse.
3. Select **Unselect** from the Context menu.
4. Select **Mirror Elements** in the **Main Menu > Node/Element > Elements > Mirror**.
5. Select “**z-x plane**” in the **Reflection** field.
6. Click **Apply**.

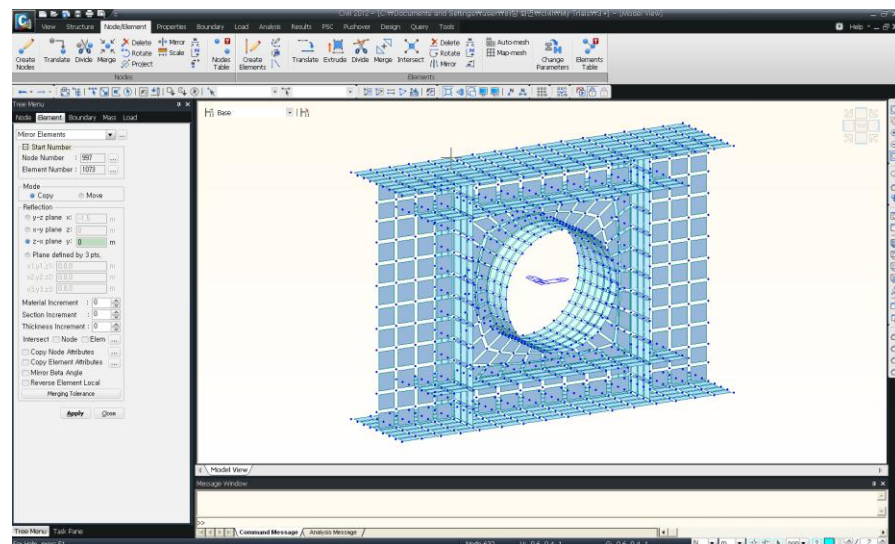




Figure 3.12 The Complete Opening Detail Model

After completing the opening reinforcing detail model, extend both ends of the beam elements to the supports to specify the support conditions.

Before creating the beam elements, create the nodes where support conditions are to be assigned.

⚠ The unspecified axis coordinates are recognized as 0.

1. Select  **Create Nodes** in the **Main Menu > Node/Element > Nodes > Create Nodes**.
2. Enter “-3” in the **Coordinates (x, y, z)** field. ⚠
3. Enter “1” in the **Number of Times** field.
4. Enter “9” in the **Distances (dx, dy, dz)** field.
5. Click .

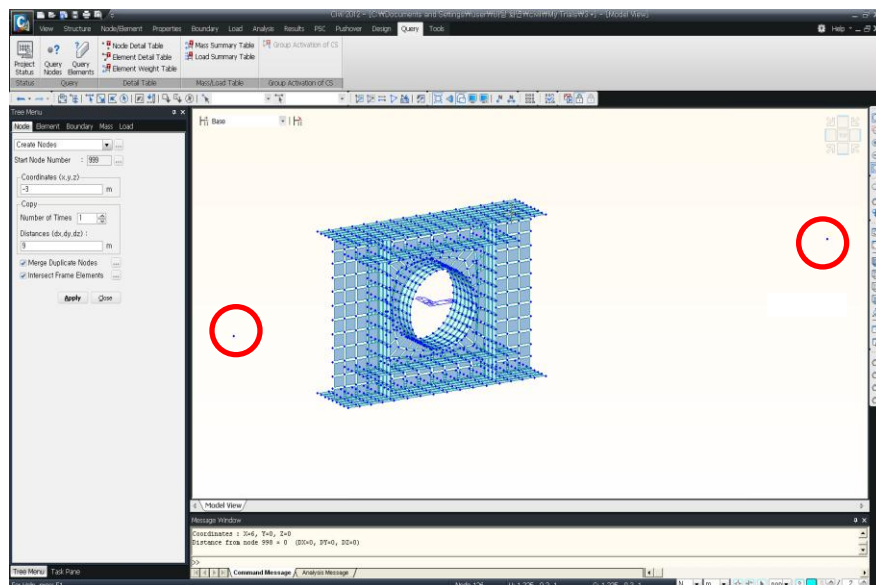







Figure 3.13 Creation of Nodes at the Beam Supports

1. Select  **Create Elements** in the **Main Menu > Node/Element > Create Elements**.
2. Select **“General beam/Tapered beam”** in the **Element Type** selection field.
3. Confirm **“1: A36”** in the **Material Name** selection field.
4. Click the button  to the right of the **Section Name** selection field.
5. Select **“mm”** in the unit system conversion window of **Status Bar**.
6. Click .
7. Confirm **“I-Section”** in the **DB/User** tab.
8. Select **“User”**.
9. Enter **“I 2000×800×20/40”** in the **Name** field.
10. Enter **“2000”**, **“800”**, **“20”** and **“40”** in the **H**, **B_I**, **t_w** and **t_{fI}** fields, respectively.
11. Click .
12. Click .
13. Select **“m”** in the unit system conversion window of **Status Bar**.
14. Select **“1: I 2000×800×20/40”** in the **Section Name** selection field.
15. Click the **Nodal Connectivity** field once.
16. Connect nodes **997** and **183** and nodes **3** and **998** (Fig.3.14) to create beam elements **1073** and **1074** respectively.

Enter Structure Support Conditions

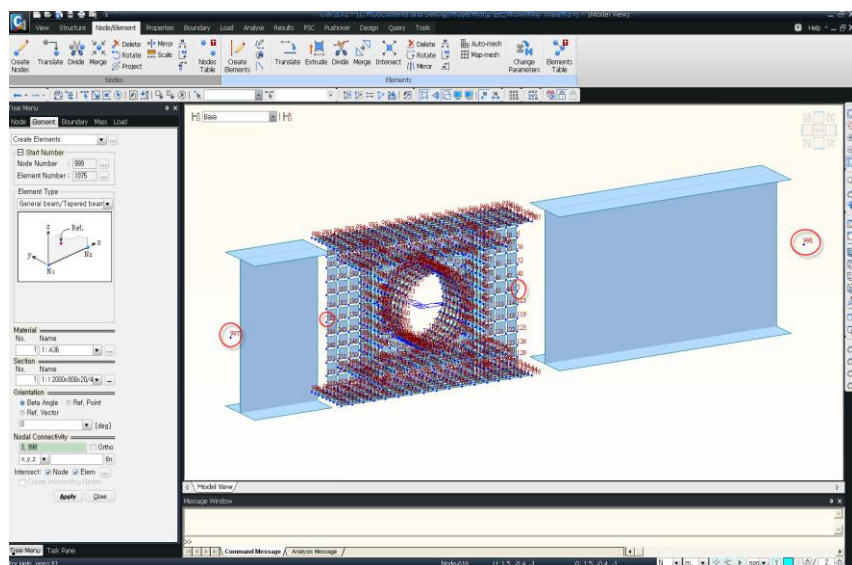




Figure 3.14 Creation of Beam Elements at Both Ends of the Opening Detail Model

Specify the pin joint support conditions at both ends of the beam

1. Select **Boundary** in the tab (Fig.3.15–❶).
2. Confirm **Supports** in the functions selection field.
3. Check (✓) “D-All” and “RX” for boundary conditions.
4. Click  **Select Window** in the Icon Menu.
5. Select both ends of the beam (nodes **997, 998**).
6. Click 

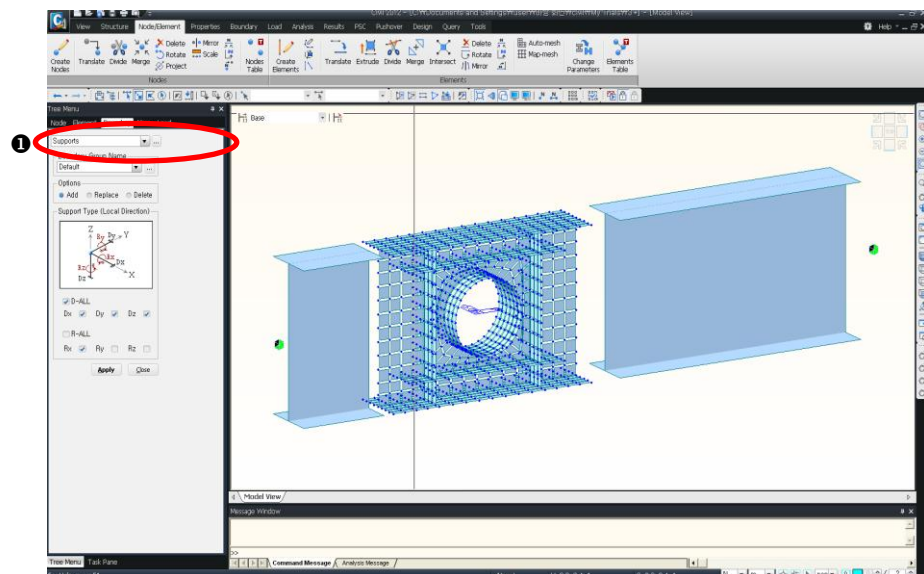


Figure 3.15 Definition of Support Conditions

Use **Rigid Link** to attribute the continuity conditions between the beams modeled as line elements and the detail model composed of plate elements.

1. Click **Zoom Window** (Toggle on) to magnify the opening detail model and click **Zoom Window** once again to Toggle off.
2. Select **Rigid Link** in the functions selection field.
3. Click the **Master Node Number** field once and click the node (Fig. 3.16–①) to which the left beam extends in the Model window to enter “183” automatically.
4. Click **Rigid Body** in the **Typical Types** selection field.
5. Click **Select Plane** in the **Main Menu > View > Select**.
6. Select “**YZ Plane**”.
7. Click the node at the left-end of the opening detail model.
8. Click **Close**.
9. Click **Apply**.
10. Repeat the steps 3~9 to specify the rigid body connection condition of the master node/slave nodes at the right end of the detail model (Fig. 3.16–②).

Toggle on      

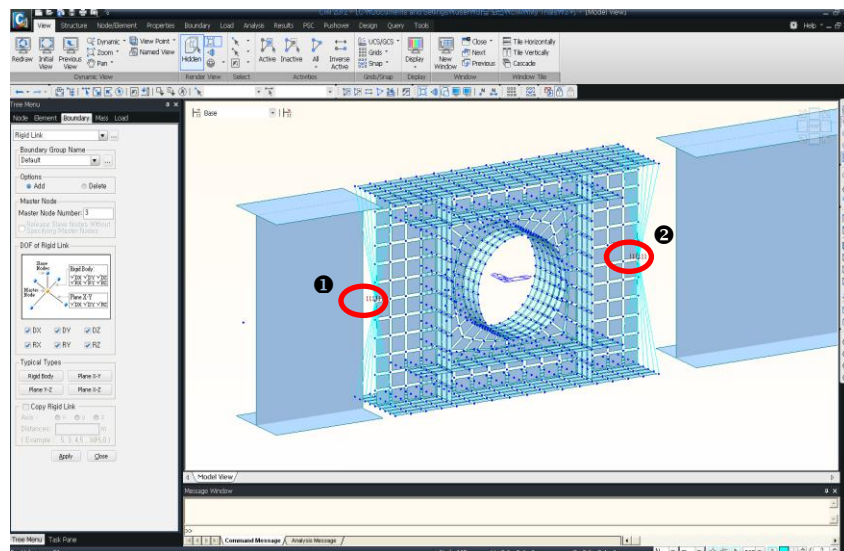





Figure 3.16 Rigid Link Setup

Enter Loading Data

Define Load Cases

1. Select **Load** in the tab (Fig.3.18–❶).
2. Click the button  to the right of **Load Case Name**.
3. Enter the contents shown in Fig.3.17 in the **Static Load Cases** dialog box.
4. Click .
5. Click .

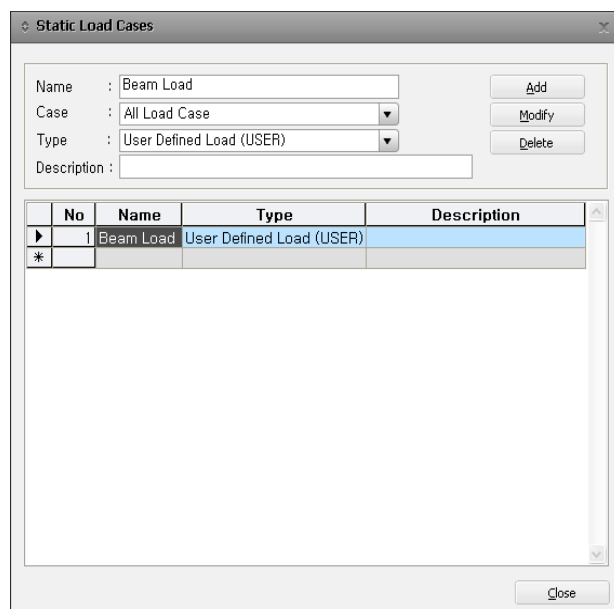





Figure 3.17 Load Cases

Define Uniformly Distributed Load

1. Click  **Zoom Fit** in the Icon Menu.
2. Click  **Select Single** in the Icon Menu.
3. Select the beams at both ends of the opening detail model (Fig.3.18–❷).
4. Select **Element Beam Loads** in the functions selection field.

5. Confirm “**Beam Load**” in the *Load Case Name* selection field.
6. Enter “**-50000**” in the *w* field of *Value*.
7. Click 
8. Toggle off the *Hidden* icon

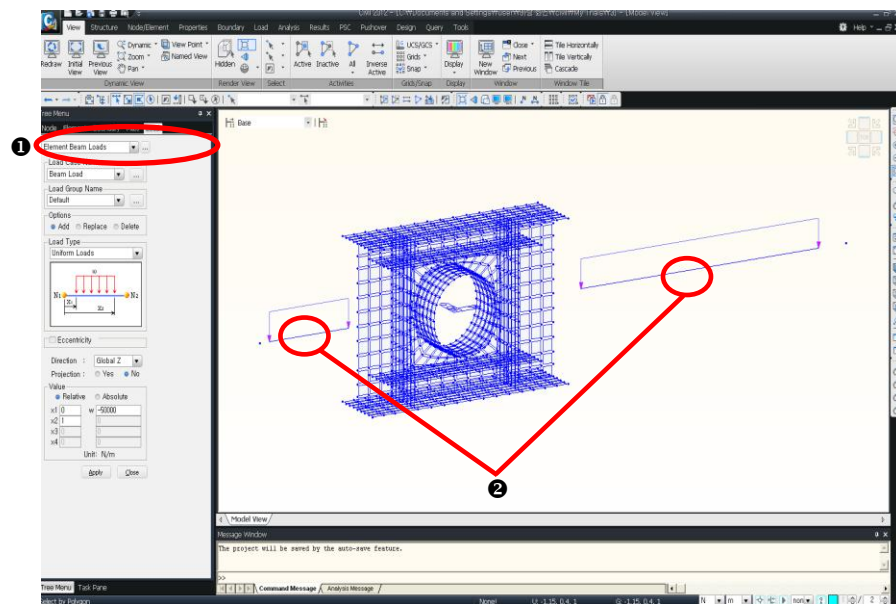









Figure 3.18. Assigning Uniformly Distributed Load on the top of the Beams

Define Concentrated Loads

When selecting elements by  Select Polygon or  Select Intersect in the Icon Menu, double-click to end the selection.

1. Click  *Select Plane* in the *Main Menu > View > Select*.
2. Select “**XZ Plane**”.
3. Select any node in the plane of the web of the opening detail model.
4. Click 
5. Click  *Activate* in the Icon Menu.
6. Click  *Zoom Window* in the Icon Menu to magnify the detail model.
7. Click  *Select Polygon* in the Icon Menu.

8. Select the nodes where concentrated loads are applied as shown in Fig.3.19–①.
9. Select **Nodal Loads** in the functions selection field.
10. Confirm “**Beam Load**” in the **Load Case Name** selection field.
11. Enter “**-50000*3/16**” in the **FZ** field.
12. Click .
13. Click to select 2 unloaded nodes at both ends of the detail model.
14. Enter “**-50000*3/16/2**” in the **FZ** field.
15. Click (Fig.3.19).
16. Click in the Icon Menu.
17. Click .
18. After Selecting “**Element Beam Load**” in the **Tree menu > Works tab > Static Load>Static Load Case**, right-click the mouse.
19. Select **Display Loads** from the Context Menu.
20. Confirm the Element Beam Element Load input.
21. Confirm the Point Load similarly following the steps 17 to 19 (Fig. 3.20).

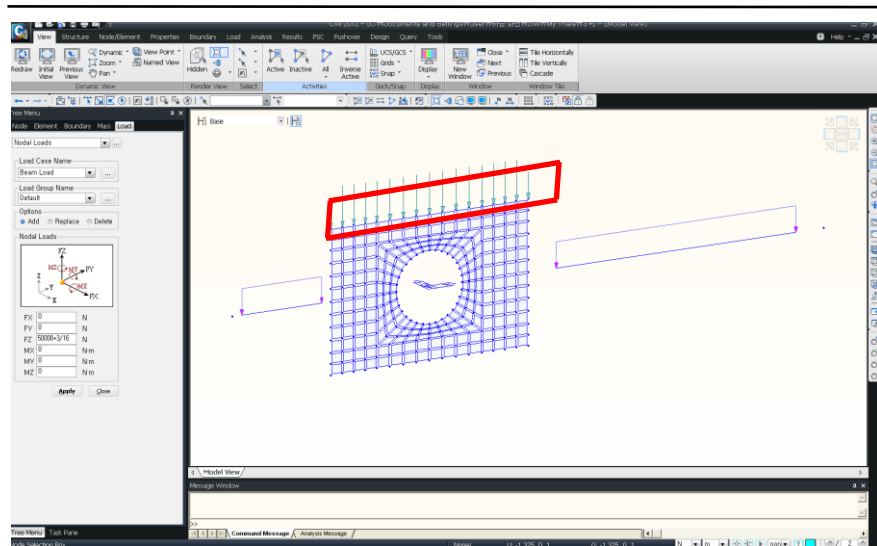


Figure 3.19 Concentrated Loads on the Opening Detail Model

Fig.3.20 shows the screen display after checking the uniform distributed and point loads above using *Works Tree*.

Works Tree systematically organizes the model data by attributes for easy manipulation of data.

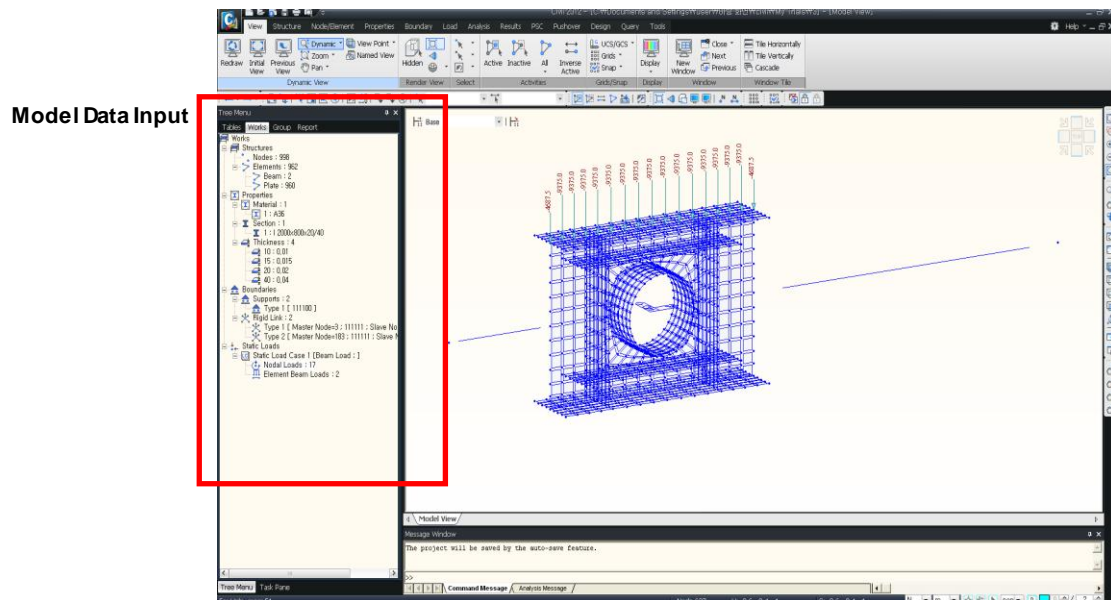



Figure 3.20 Complete Model




Perform Structural Analysis

Click  **Analysis** in the Icon Menu to analyze the model. After completing the analysis, the program switches automatically to the *post-processing* mode, which provides access to the interpretation of analysis and design results.

Interpret Analysis Results

Verify Member Stresses

The opening detail model is modeled with plate elements. The analysis results and interpretation of results focus on the deformed shape and the variation of stresses in the vicinity of the opening.

1. Click  **Hidden** (Toggle on) in the Icon Menu.
2. Click  **Shrink** (Toggle off) in the Icon Menu.
3. Select **Results>Stresses>Plane-Stress/Plate Stresses** in the Main Menu.
4. Select “**Sig-XX**” in the **Components** selection field.
5. Check (✓) “**Contour**” and “**Legend**” in the **Type of Display** selection field.
6. Convert to “**kN**” and “**cm**” in the unit conversion window.
7. Click .

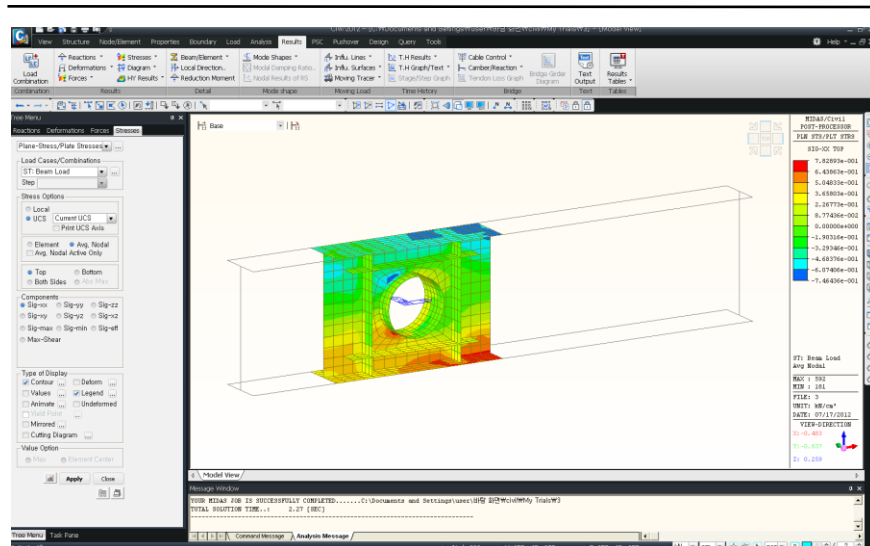








Figure 3.21 Stress Contour for Plate Elements

Auto-Compute Member Stresses

It is necessary to compute the element forces from the internal forces at each node in plate or solid elements for design purposes.

Use **Local Direction Force Sum** to compute the element forces automatically at the boundaries between the beam elements and the detail model.

🔊 Use Plate Edge Polygon Select to assign a polygon which includes the section of interest for verification.

-
1. Click  **Initial View** in the Icon Menu.
 2. Convert to “m” in the unit conversion window.
 3. Click  **Zoom Window** in the Icon Menu to magnify the boundary of the detail model and the right side line element (Fig. 3.22).
 4. Select **View > Display > Display Option > Draw > Hidden Option (Model)**.
 5. Select **Outline** in **Type of Option Value** and click .
 6. Select **Results>Local Direction Force Sum** in the Main Menu.
 7. Select **Plate Edge Polygon Select** in **Mode**.[🔊]
 8. Confirm “**ST: Beam Load**” in the **Load Case** selection field.
 9. Click  **Hidden** (Toggle off) in the Icon Menu.
 10. Click nodes **980, 971, 607, 616, 980** successively as shown in Fig.3.22.
 11. Click  **Hidden** (Toggle on) in the Icon Menu.
 12. Click  in the **Local Direction Force Sum** dialog box.
-

The sum of all the nodal forces, contained in the specified section, is computed at the centroid of the section according to the local coordinates (Fig. 3.22–**1**) defined on the section for which element forces are to be computed. The computed value of the strong axis bending moment, M_y , for the member at the right end of the detail model is 506.25 kN·m.

The member forces computed by **Local Direction Force Sum** are compared with the member forces of the linear element on the right side.

1. Select **Results>Forces>Beam Forces/Moments** in the Main Menu.
2. Click **Apply**.
3. Click the button **...** to the right of **Contour** in **Type of Display** and check (✓) “**Reverse Contour**”.
4. Click **OK** in the **Contour Details** dialog box.
5. Move the mouse cursor to the middle of element **1074** and snap. Use **Fast Query** to confirm “**My 506.25 kN-m**” at the **i** end.
6. Change **Components** in the **Beam Forces/Moments** dialog bar to compare the member forces of **Local Direction Force Sum** with those of **Bubble Tip**.

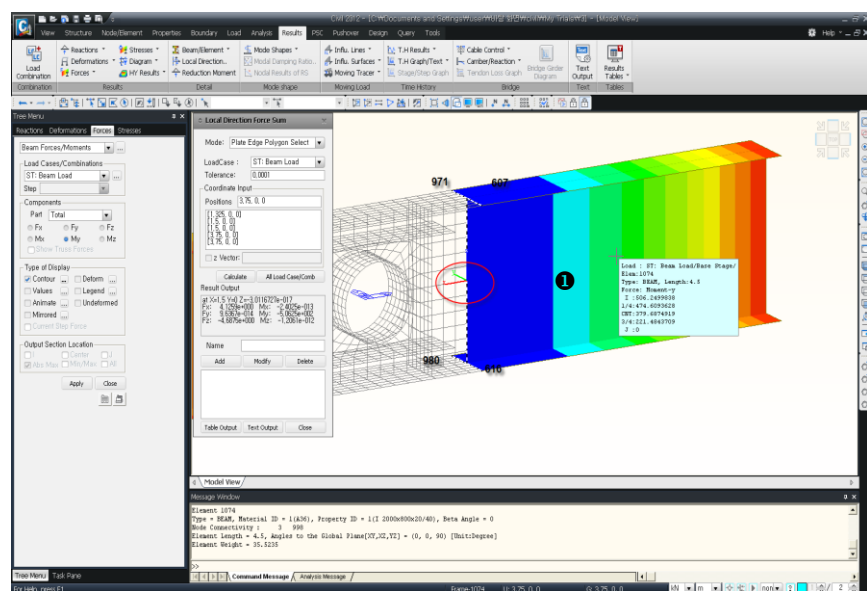


Figure 3.22 Local Force Sum