Tutorial 4

Arch Bridge

Civil

TUTORIAL 4. ARCH BRIDGE

TUTORIAL 4.
ARCH BRIDGE

Summary

This tutorial illustrates the modeling and interpretation of the analysis results of a single span arch bridge subjected to moving traffic loads.

The explanations for the basic functions of midas Civil ("Tutorial 1") are omitted. The Icon Menu is primarily used. Refer to "Tutorials 1 and 2" and the Online Manual for understanding the basic functions in midas Civil and the structural analysis processes.

The modeling and analysis processes presented in this example are as follows:

1. File Opening and Preferences Setting
2. Enter Material and Section Properties
3. Structure Modeling Using Nodes and Elements
4. Enter Structure Boundary conditions
5. Enter Moving Traffic Loads and Static Loads
6. Perform Structural Analysis
7. Verify and Interpret Analysis Results

Analysis Model and Load Cases

Figure 4.1 shows the arch bridge model. The specifications for the structure are as follows:

- Bridge Type: Arch bridge
- Bridge Class: First Class
- Span Length: 50 m
- Design Traffic Lanes: 4 Lanes
- Width: 14 m

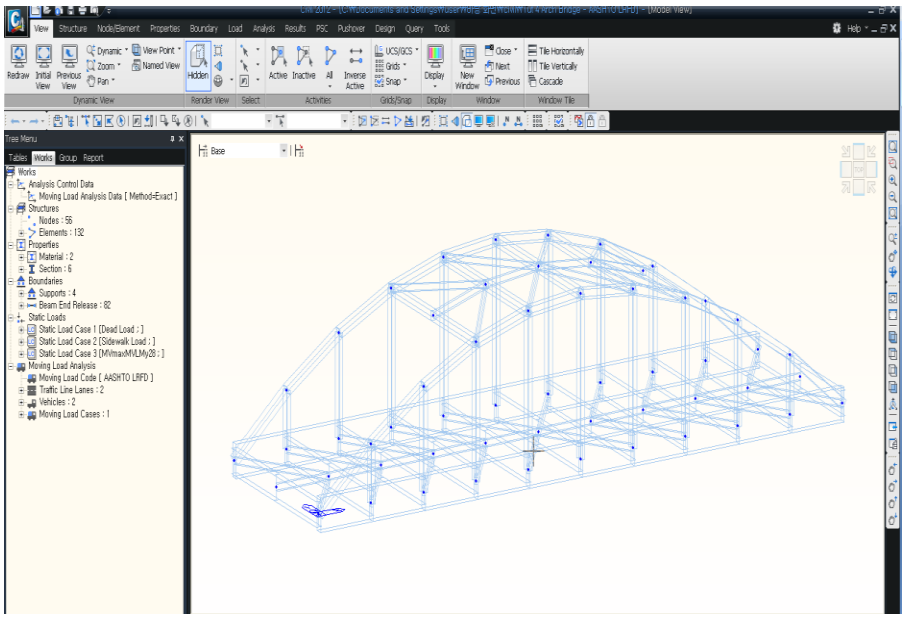

Figure 4.1 Final Arch Bridge Model

The following list describes the structural plan layout:

- Spacing of cross beams is 5 m.
- Stringers are placed longitudinally along the axis of the bridge.
- Main girders and arch ribs are placed 7 m from the centerline on both sides.

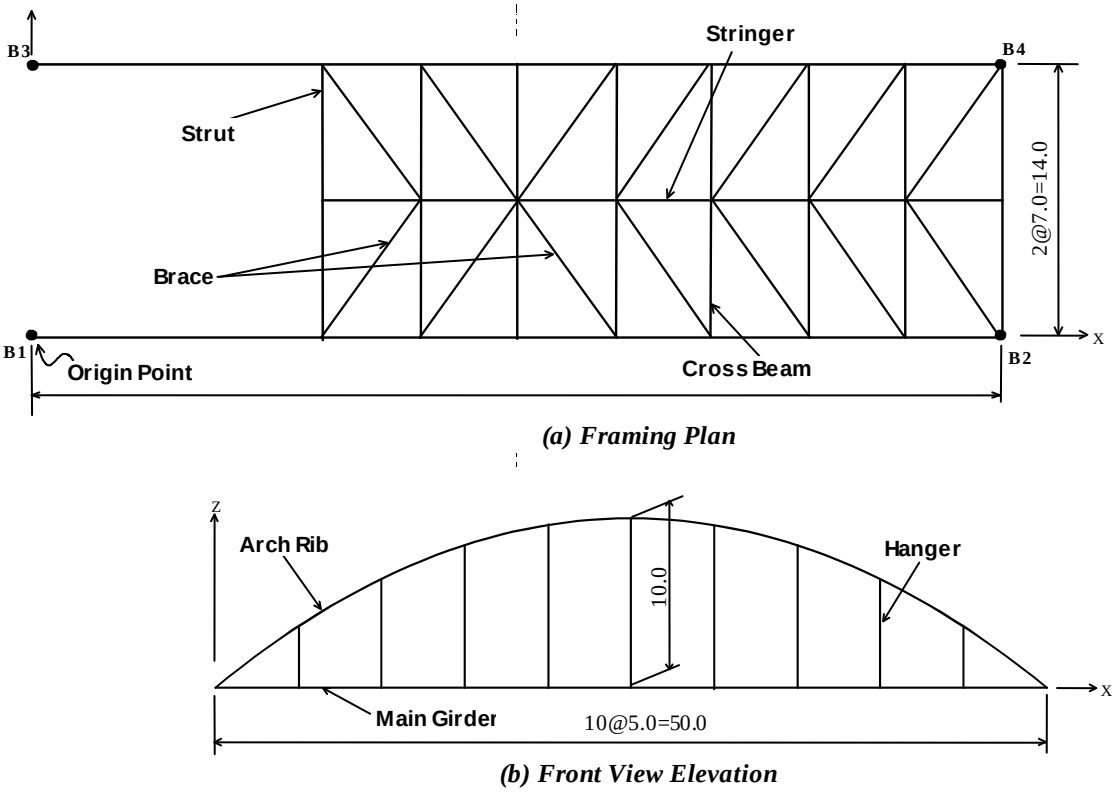

Figure 4.2 Framing Plan and Front View Elevation of the Arch Bridge [Unit: m]

For simplicity, only the following 3 load cases are considered:

- Load Case 1: 90 kN/m Dead Load (applied only on the main girders)
- Load Case 2: 6 kN/m Sidewalk Load (applied only on the main girders)
- Load Case 3: Vehicle Loads (HA + HB)

This example focuses on the explanation for the relevant functions in midas Civil. The assumptions made in this example may differ from those in practical applications.

File Opening and Preferences Setting

Open a new file (***New Project***) to model the bridge and save the file as "Arch BD37" (***Save***).

Click the unit system selection button in ***Status Bar*** at the bottom of the screen. Choose the unit system and select "**kN**" and "**mm**". Change the unit system as frequently as necessary for the convenience of data entry.

The structure is modeled using the Icon Menu instead of the Tree Menu or Main Menu to improve the modeling skills of the user.

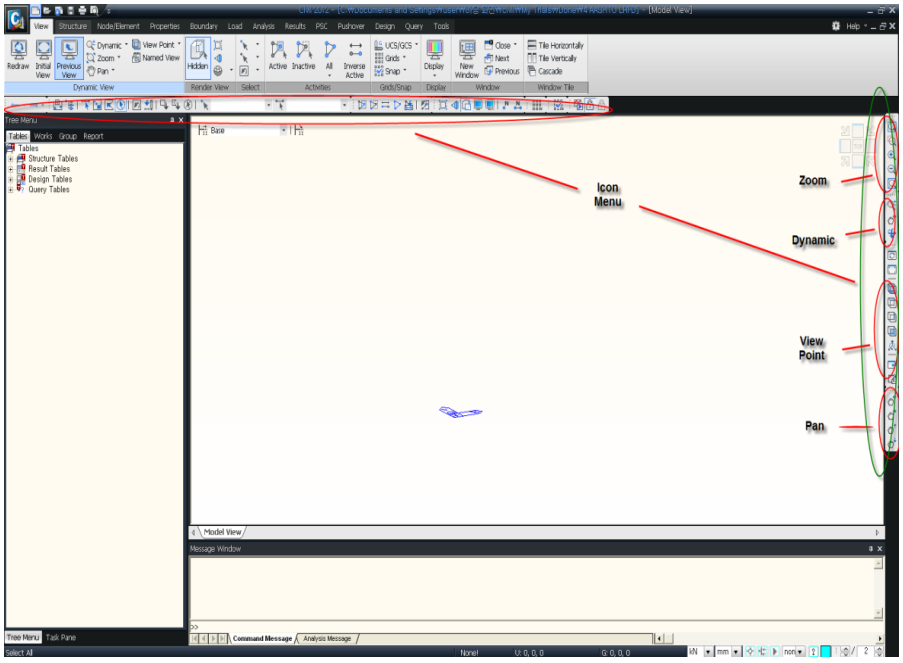

Figure 4.3 Model View Layout

Enter Material and Section Properties

Specify the following member material properties and section data.

- Material Properties
  1: A36 – cross beam, bracing
  2: A572-50 – main girder, arch rib, hanger

- Section Data
  1: TS 2100×600×10/10 – Main Girder (Box)
  2: I 1540×500×14/27 – Cross Beam (I-Section)
  3: TS 600×600×16/14 – Arch Rib (Box)
  4: I 600×400×12/16 – Hanger (I-Section)
  5: TS 600×500×10/14 – Strut (Box)
  6: W16×100 – Bracing & Stringer (W-Section)

Sections 1 to 5 are built-up sections. Use the ***User*** functions to enter the principal section dimensions. Use ***DB***, the AISC standard sections, contained in the program for Section 6.

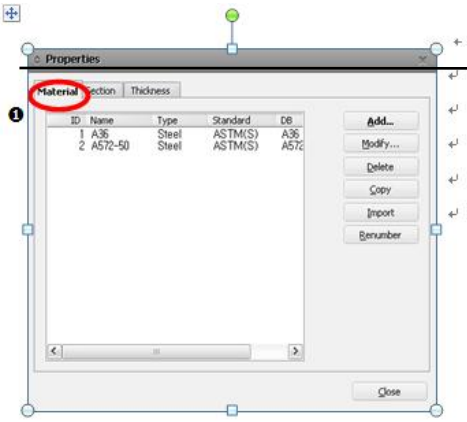

Figure 4.4 Sectional Attributes Dialog Box

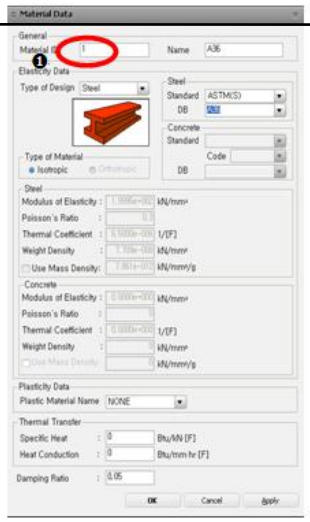

Figure 4.5 Material Properties Dialog Box

1. Select ***Properties>Material Properties*** in the Main Menu.
2. Click ***Material*** in the ***Properties*** dialog box (Figure 4.4❶).
3. Click Add.
4. Confirm "**1**" in the ***Material Number*** field of ***General*** (Figure 4.5❶).
5. Select "**Steel**" in the ***Type*** selection field.
6. Select "**ASTM(S)**" in the ***Standard*** selection field of ***Steel***.
7. Select "**A36**" in the ***DB*** selection field.
8. Click Apply.
9. Select "**2**" in the ***Material Number*** field of ***General***.
10. Select "**Steel**" in the ***Type*** selection field.
11. Select "**ASTM(S)**" in the ***Standard*** selection field of ***Steel***.
12. Select "**A572-50**" in the ***DB*** selection field.
13. Click OK.

1. Select the ***Section*** tab in the ***Properties*** dialog box (Figure 4.4) or select ***Property*** > ***Section*** from the Main Menu.
2. Click Add.
3. Confirm "**1**" in the ***Section ID*** field of the ***DB/User*** tab (Figure 4.6).
4. Type "**Main Girder**" in the ***Name*** field.
5. Select "**Box**" in the ***Section Shape*** selection field (Figure 4.6❶).
6. Select "**User**" in ***User*** or ***DB***.
7. Enter "**2100**" in the ***H*** field.
8. Enter "**600**" in the ***B*** field.
9. Enter "**10**" in the ***tw*** field.
10. Enter "**10**" in the ***tf1*** field.
11. Click Apply.
12. Repeat steps 3 to 11 for Sections 2 to 5 (See Page 6 for details).
    Note that for Sections 2 and 4, use ***I-section*** instead of ***Box***
13. Confirm "**6**" in the ***Section ID*** field.
14. Type "**Bracing & Stringer**" in the ***Name*** field.
15. Select "**I-Section**" in the ***Section Shape*** selection field (Figure 4.6❶).
16. Select "**DB**" in ***DB*** or ***User*** and select "**AISC**" in the field to the right.
17. Click the ***Sect. Name*** field and type "**W 16 ×100**" or use ***Scroll Bar*** to select the type.
18. Click OK.
19. Click Close.
20. Click the unit system selection button of the Status Bar and change "**mm**" to "**m**".

There are 2 methods to specify the Section Name:
1).Click button to the right of the field and select the desired section name with Scroll Bar.
2) Type in directly the desired section name.

Convert the unit system from "mm" to "m" for structural modeling.

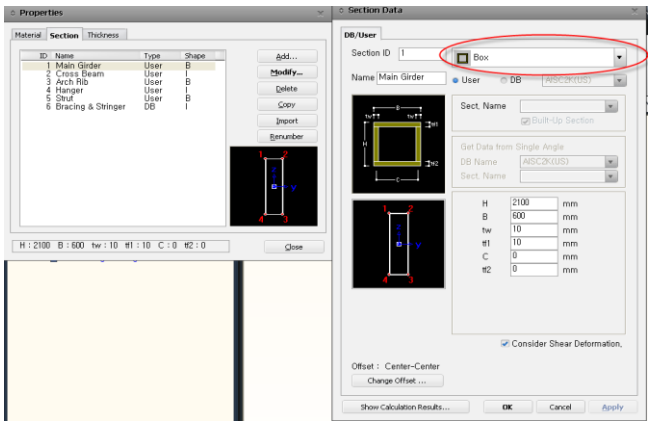

Figure 4.6 Section Properties and Section Data Dialog Box

Structural Modeling Using Nodes and Elements

Generate the Arch Ribs

Use ***Structure Wizard*** to generate the arch ribs (Figure 4.7).

1. Select ***Structure> Wizard>Base Structures > Arch*** in the ***Main Menu.***
2. Select "**Parabola1**" in the ***Type*** selection field of the ***Input & Edit*** tab.
3. Enter "**10**" in the ***Number of Segments*** field.
4. Enter "**50**" in the ***L*** field.
5. Enter "**10**" in the ***H*** field.
6. Select "**None**" in the ***Boundary Condition*** selection field.
7. Check (✓) "**Show Element No.**"
8. Select "**2 : A572-50**" in the ***Material*** selection field.
9. Select "**3 : Arch Rib**" in the ***Section*** selection field.
10. Enter "**0, 0, 0**" in the ***Insert Point*** field of the ***Insert*** tab.
11. Click OK.
12. Click ***Auto Fitting***.
13. Click ***Front View***.

The arch shape (parabola/ellipse with equal/ equal-projected spacing) can be selected in the Type field of the Input & Edit tab. Considering the hangers at an equal spacing, select "Parabola1" to set the nodes on the arch rib at an equal spacing projected on a horizontal line (Figure 4.8).

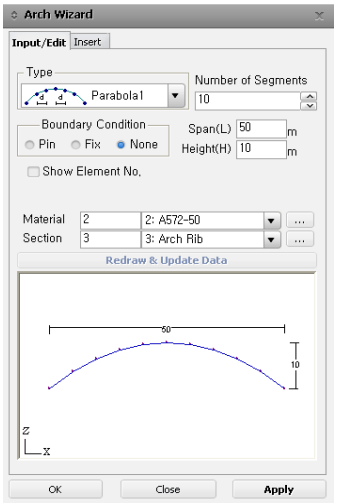

Figure 4.7 Concept of Parabola1 format and Arch Wizard Dialog Box

Generate the Hangers

Use ***Extrude Elements*** to generate the hangers. Extend the nodes generated on the arch rib by projecting them perpendicularly downward (Figure 4.8).

Extrude Elements generates geometrically 1-dimensional higher elements by following the moving path of the nodes or elements (node→line element, line →surface element, surface→solid element).

1. Click ***Extrude Elements*** in ***Node/Element*** > ***Elements*** > ***Extrude*** from the Main Menu (Figure 4.8❶).
2. Click ***Node Number*** (Toggle on) in the Icon Menu.
3. Click ***Select Window*** to select nodes **2** to **10** from which the hangers are generated.
4. Select "**Node→Line Elem.**" in the ***Extrude Type*** selection field.
5. Select "**Beam**" in ***Element Type*** of the ***Element Attribute*** selection field.
6. Select "**2 : A572-50**" in the ***Material*** selection field.
7. Select "**4 : Hanger**" in the ***Section*** selection field.
8. Select "**Project**" in the ***Generation Type*** selection field.
9. Select "**Project on a line**" in the ***Projection Type*** selection field.
10. Click the ***P1*** field of ***Base Line definition***. Once the background color turns to pale green, assign node **1** and then assign node **11** for ***P2*** filed.
11. Select "**Normal**" in the ***Direction*** selection field.
12. Click Apply.
13. Click ***Change Element Parameters*** (Figure 4.8❷).
14. Click ***Select Recent Entities*** (Figure 4.8❸).
15. Select "**Element Local Axis**" in ***Parameter Type*** selection field.
16. Select "**Assign**" and **"Beta Angle"** in the ***Mode*** selection field.
17. Enter "**90**" in the ***Beta Angle*** field.
18. Click Apply.

Base Line Definition requires 2 nodes of the line onto which it is projected.

The Direction choice in the Project function represents the projection direction of the element.

The hanger web direction is modified to be perpendicular to the bridge longitudinal axis as shown in Figure 4.10 (also refer to the section on Beta Angle in the Online Manual).

Click ***Shrink*** (Figure 4.9❶) and ***Hidden*** (Figure 4.9❷) (Toggle on) to check the entered ***Beta Angle***. Check the current data entries and click ***Shrink*** and ***Hidden*** to toggle off.

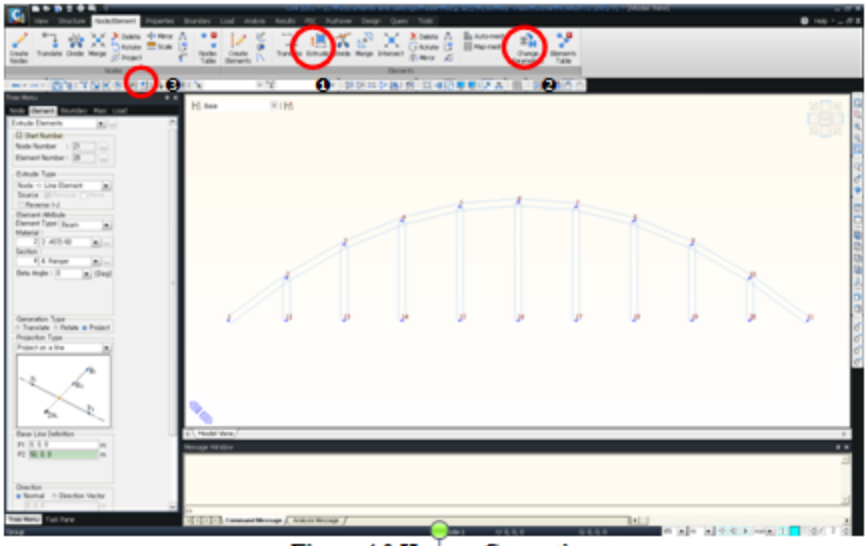

Figure 4.8 Hanger Generation

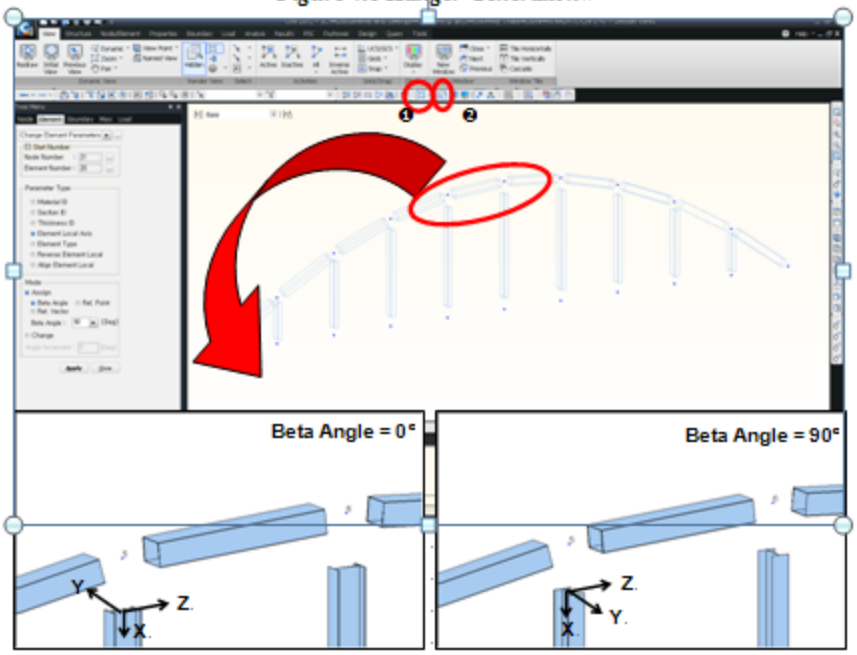

Figure 4.9 Modification of the Beta Angle for Hangers

Generate the Main Girder and Duplicate the Arch Frame

Create the main girder by connecting both ends of the arch. Duplicate the completed part of the arch frame including the main girder at the opposite side.

In this example, Point Grid is not used. To avoid confusion while assigning the nodes with the mouse, toggle off Point Grid and Point Grid Snap.

1. Click ***Point Grid*** and ***Point Grid Snap*** (Toggle off).
2. Click ***Iso View***.
3. Click ***Create Elements*** in ***Node/Elements*** > ***Create Elements*** from the Main Menu.
4. Select "**General beam/Tapered beam**" in the ***Element Type*** selection field.
5. Select "**2 : A572-50**" in the ***Material*** selection field.
6. Select "**1 : Main girder**" in the ***Section*** selection field.
7. Select "**0**" in the ***Beta Angle*** field of ***Orientation***.
8. Check (✓) in **Node** of the ***Intersect*** selection field.
9. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, assign nodes **1** to **11**.
10. Click ***Select All***.
11. Click ***Translate Elements***(Figure 4.10-❶).
12. Select "**Copy**" in the ***Mode*** selection field.
13. Select "**Equal Distance**" in the ***Translation*** selection field.
14. Enter "**0, 14, 0**" in the ***dx, dy, dz*** field.
15. Enter "**1**" in the ***Number of Times*** field.
16. Click Apply (Figure 4.10).

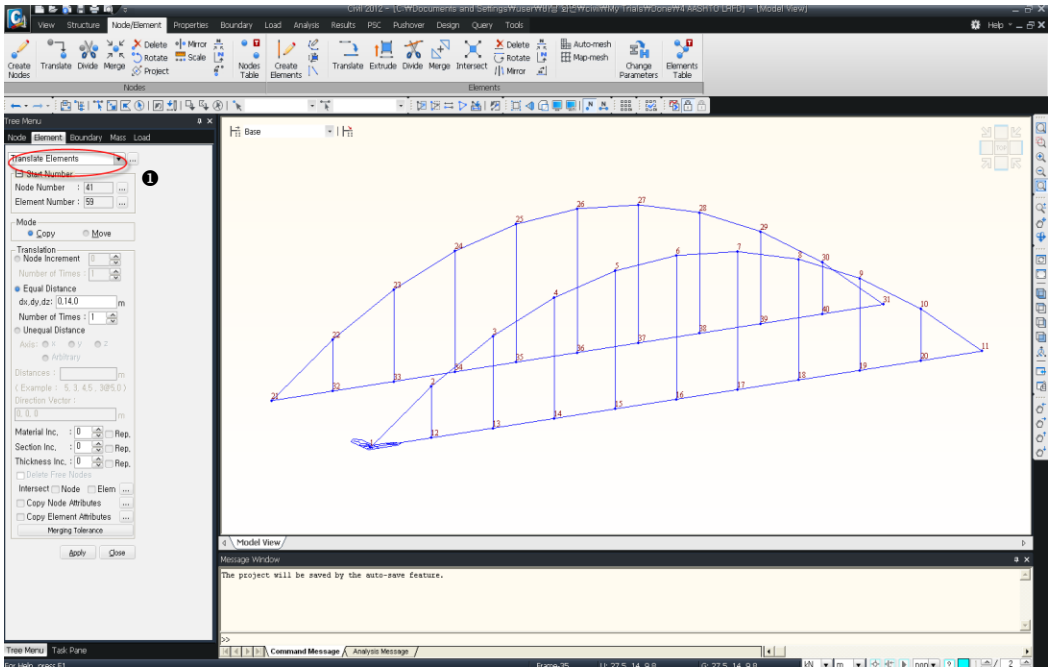

Figure 4.10 Completed Main Girders and Arches

Generate the Cross Beams

Use ***Extrude Elements*** to create the cross beams by extending the nodes on one of the main girders to the nodes on the opposite main girder.

1. Click ***Extrude Elements***. (Figure 4.11-❶)
2. Click ***Select Polygon*** and select nodes **1** and **11** to **20**.
3. Select "**Node→Line Element.**" in the ***Extrude Type*** selection field.
4. Select "**Beam**" in the ***Element Type*** selection field.
5. Select "**1 : A36**" in the ***Material*** selection field.
6. Select "**2 : Cross beam**" in the ***Section*** selection field.
7. Select "**Project**" in the ***Generation Type*** selection field.
8. Select "**Project on a line**" in the ***Projection Type*** selection field.
9. Click the ***P1*** in the ***Base Line Definition*** field. Once the background color turns to pale green, assign the nodes **21** and **31** consecutively.
10. Select "**Normal**" in the ***Direction*** selection field.[1]
11. Click Apply.

[1] The Direction represents the Direction of Projection.

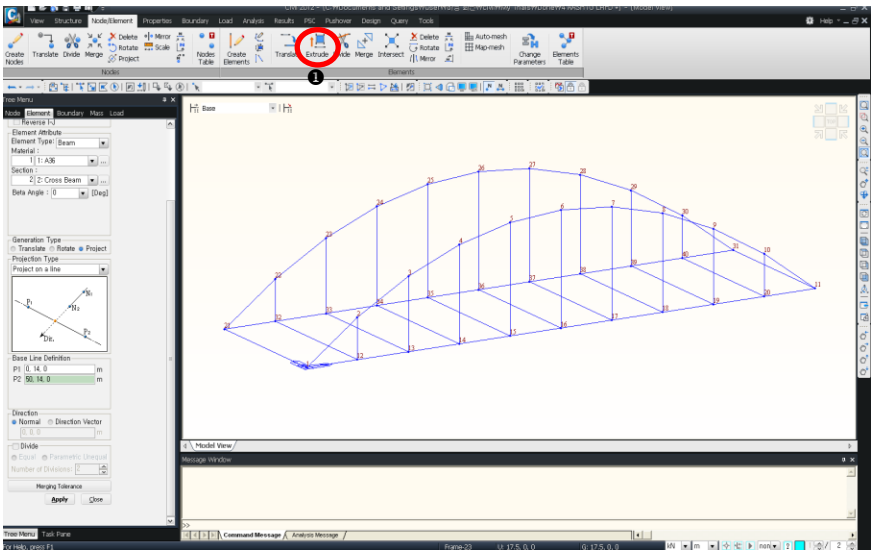

Figure 4.11. Creation of Cross Beams

Generate the Bracings

Activate only the newly created cross beams. Use ***Element Snap*** in conjunction with ***Create Elements*** to create the stringers.

1. Click ***Select Recent Entities*** in the Icon Menu.
2. Click ***Activate*** in the Icon Menu.
3. Click ***Element Number*** (Toggle on) in the Icon Menu.
4. Click ***Create Elements*** in ***Node/Element*** > ***Create Elements*** from the Main Menu.
5. Select "**General beam/Tapered beam**" in the ***Element Type*** field.
6. Select "**1 : A36**" in the ***Material*** field.
7. Select "**6 : Bracing & Stringer**" in the ***Section*** field.
8. Enter "**0**" in the ***Beta Angle*** field of ***Orientation***.
9. Check (✓) in **Elem** in the ***Intersect*** field.
10. Confirm that the location of ***Element Snap*** in ***Status Bar*** is 1/2 (Figure 4.12❶).
11. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, assign the middle of elements **59** and **60** consecutively.
12. Click ***Element Number*** (Toggle off) in the Icon Menu (Figure 4.12).

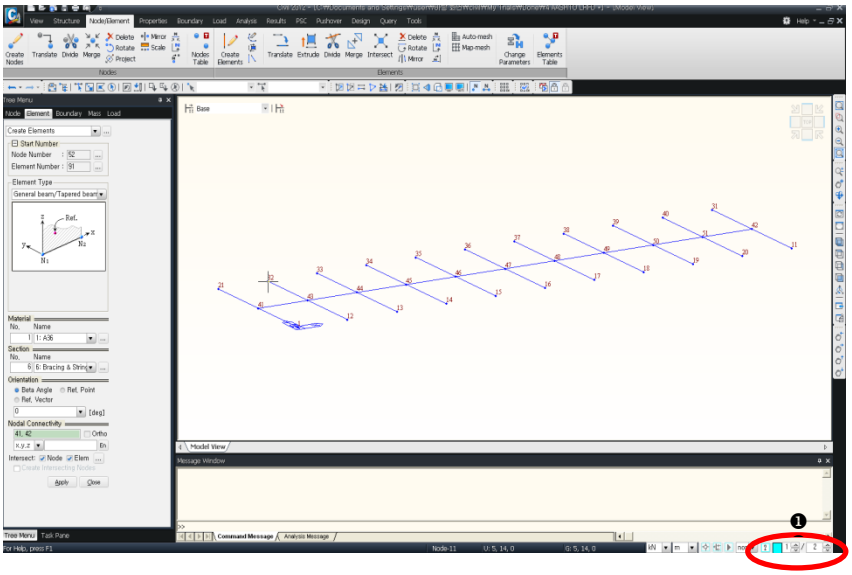

Figure 4.12 Completed Stringers

Generate the diagonal bracings on the floor plane.

1. Select ***Create Elements***. (Red Mark)
2. Select "**General beam/Tapered beam**" in the ***Element Type*** selection field.
3. Select "**1 : A36**" in the ***Material*** selection field.
4. Select "**6 : Bracing & Stringer**" in the ***Section*** selection field.
5. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, connect nodes **1** to **43** and nodes **43** to **21** to create two elements.
6. Select ***Translate Elements***. (Red Mark)
7. Click ***Select Single*** to select the two braces generated in step 4.
8. Select "**Copy**" in the ***Mode*** selection field.
9. Select "**Equal Distance**" in the ***Translation*** selection field.
10. Enter "**5, 0, 0**" in the ***dx, dy, dz*** field.
11. Enter "**4**" in the ***Number of Times*** field.
12. Click Apply.
13. Click ***Mirror Elements***.
14. Click ***Select Previous*** and ***Select Recent Entities*** to select all the diagonal bracings.
15. Select "**Copy**" in the ***Mode*** selection field.
16. Select ***y-z plane*** in ***Reflection*** and click the ***x*** field. Once the background color turns to pale green, assign node **16** or enter "**25**".
17. Click Apply (Figure 4.13).

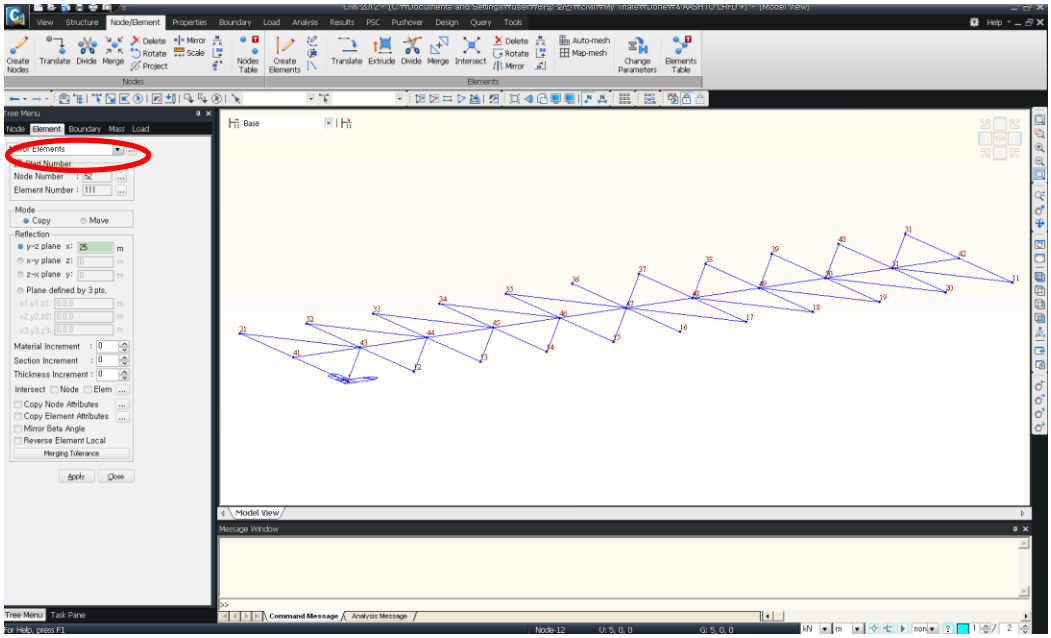

Figure 4.13 Completed Floor Plane

Create the bracings on the arch ribs located symmetrically on each side of the mid span.

Inverse Active function deactivates the nodes and elements displayed in the current window, and activates the formerly inactivated nodes and elements.

1. Click ***Inverse Activate*** (Figure 4.14❶) in the Icon Menu.
2. Click ***Create Elements***.
3. Select "**1 : A36**" in the ***Material*** selection field.
4. Select "**5 : Strut**" in the ***Section*** selection field.
5. Enter "**0**" in the ***Beta Angle*** field of Orientation.
6. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, connect separately nodes **4** and **24**, **5** and **25**, **6** and **26**, **7** and **27**, and **8** and **28** (Figure 4.14).

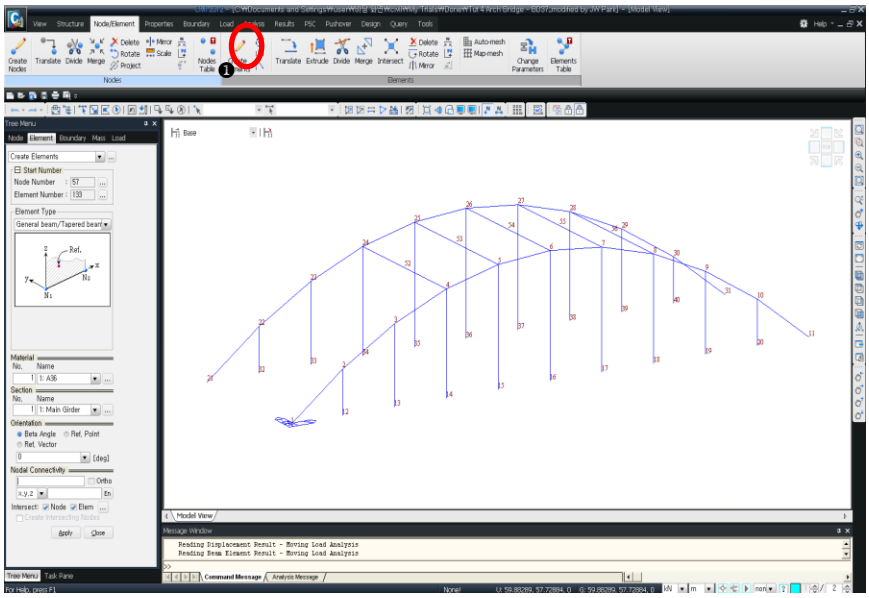

Figure 4.14 Completed Struts

When an undesirable location is selected during the data entry of elements, click the Esc key. Alternatively right-click the mouse and select Cancel at the bottom of the Context Menu to cancel the entry.

To create the bracings in the central portion of the arches, selectively activate the elements that are connected to the elements being generated.

1. Click ***Select Single*** to select the 5 struts generated in the previous step.
2. Click ***Activate***.
3. Click ***Element Number*** (Toggle on).
4. Click ***Create Elements***.
5. Select "**General beam/Tapered beam**" in ***Element Type***.
6. Select "**1 : A36**" in the ***Material*** selection field.
7. Select "**6 : Bracing & Stringer**" in the ***Section*** selection field.
8. Enter "**0**" in the ***Beta Angle*** field of ***Orientation***.
9. Check (✓) in **Elem** of the ***Intersect*** selection field.
10. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, connect successively the centers of elements **111** to **115** to create the bracings.
11. Click ***Element Number*** (Toggle off).
12. Click the ***Nodal Connectivity*** field. Once the background color turns to pale green, connect separately nodes **4** and **53**, **24** and **53**, **5** and **54**, **25** and **54**, **54** and **7**, **54** and **27**, **55** and **8**, and **55** and **28** (Figure 4.15).

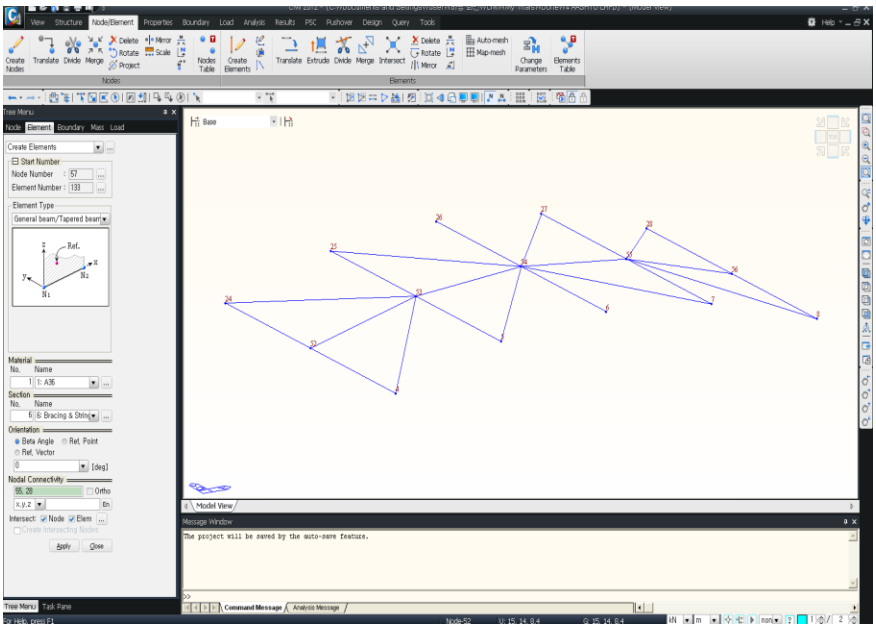

Figure 4.15 Completed Arch Bracings

Enter Structure Boundary Conditions

Once the structural configuration is created, specify the support conditions (Figure 4.2(a)).

1. Click  ***Activate All*** in the Icon Menu.
2. Select ***Boundary*** tab as shown in Figure 4.16, and select "**Supports**".
3. Select "**Add**" in the ***Options*** selection field.
4. Click  ***Select Single***.
5. Select node **1** and check (✓) "**D-ALL**".
6. Click Apply.
7. Select node **11** and check (✓) "**Dy** and **Dz**".
8. Click Apply.
9. Select node **21** and check (✓) "**Dx** and **Dz**".
10. Click Apply.
11. Select node **31** and check (✓) "**Dz**".
12. Click Apply.

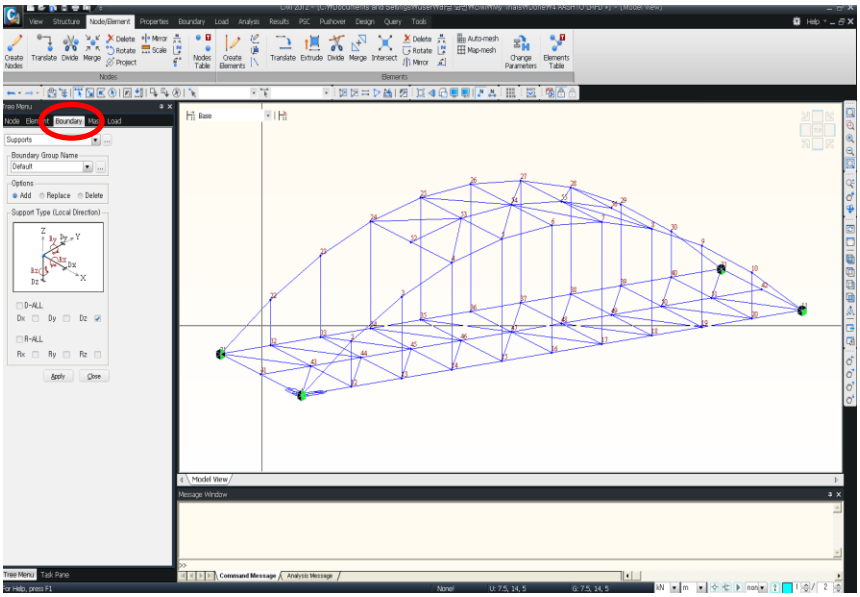

Figure 4.16 Structural Boundary Conditions

Boundary Conditions for Beam End Connections

Use ***Beam End Release*** to specify the boundary conditions at both ends of the beam elements (Figure 4.17).

- Both ends of hangers: Pin joint conditions about the ECS z-axis
- Both ends of bracings: Pin joint conditions about the ECS y- and z-axes
- Both ends of cross beams connected to the main girders: Pin joint conditions about the ECS y- and z-axes

---

1. Select "**Beam End Release**" in the ***Boundary*** tab.
2. Select "**Add/Replace**" in the ***Options*** selection field.
3. Click the ***Filter*** selection field (Figure 4.17❶) to select "**z**".
4. Click ***Select All***.
5. Check (✓) "**Mz**" of ***i-Node*** and ***j-Node*** in the ***General Types and Partial Fixity*** selection field.
6. Click Apply.
7. Click the ***Filter*** selection field (Figure 4.17❶) to select "**none**".
8. Click ***Select Identity-Elements*** (Figure 4.17❷).
9. Select "**Section**" in the ***Select Type*** field.
10. Select "**6 : Bracing & Stringer**" in the ***Section*** selection field.
11. Click Add.
12. Click the ***Pinned-Pinned*** button in the ***General Types and Partial Fixity*** selection field (or check (✓) "**My** and **Mz**" of ***i-Node*** and ***j-Node***).
13. Click Apply.
14. Select "**2 : Cross beam**" in the ***Section*** selection field of the ***Select Identity-Elements*** dialog box.
15. Click Add.
16. Click Close in the ***Select Identity-Elements*** dialog box.
17. Click ***Activate*** in the Icon Menu.
18. Click ***Element Number*** (Toggle on) in the Icon Menu.
19. Click ***Select by Intersecting*** to select elements **59** to **69**.
20. Click the ***Pinned-Fixed*** button in the ***General Types and Partial Fixity*** field.
21. Click Apply.

Refer to the Online Manual, Getting Started and Tutorials for detailed explanation on Filtering Selection.

Click Display and select Local Axis on the Element tab for checking the element coordinate axes.

Select Intersect selects the elements intersecting the specified lines drawn with the mouse.

22. Type "**80 to 90**" in the element selection window (Figure 4.17❷) and press [***Enter***].
23. Click the ***Fixed-Pinned*** button in the ***General Types and Partial Fixity*** selection field.
24. Click Apply.
25. Click ***Element Number*** (Toggle off) in the Icon Menu.
26. Click ***Activate All*** in the Icon Menu.
27. Click ***Node Number*** (Toggle off) in the Icon Menu.

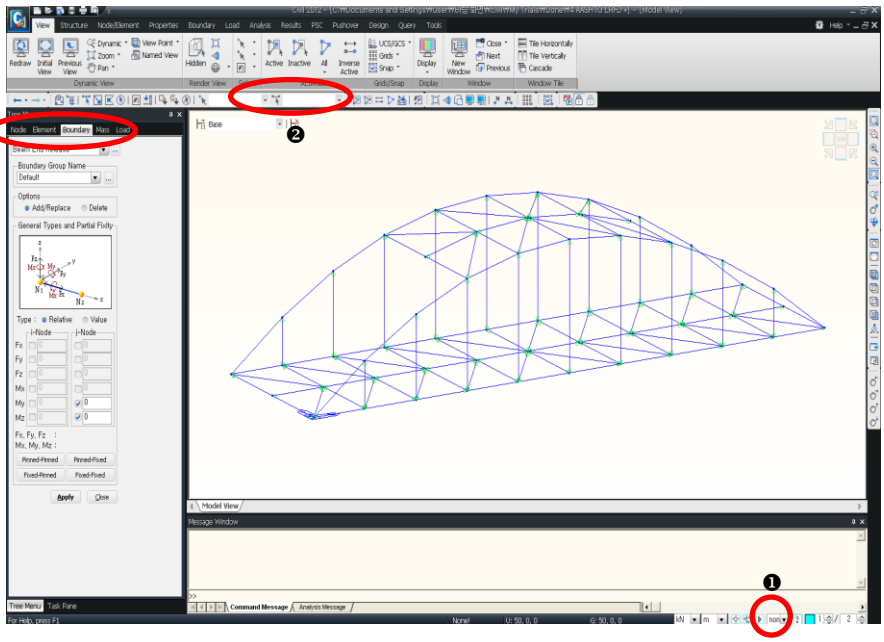

Figure 4.18 Beam End Release

Generate the Cross Beam Group

Generate the Cross Beam Group, which will be used to enter the moving loads.

1. Click ***Select Identity-Elements***.
2. Select "**Section**" in the ***Select Type*** field.
3. Select "**2 : Cross Beam**" in the ***Section*** field.
4. Click Add.
5. Click Close in the ***Select Identity-Elements*** dialog box.
6. Select the Group tab in the Tree Menu.
7. Click ***Activate*** in the Icon Menu.
8. Click ***Top View*** in the Icon Menu.
9. Right-click the mouse in the ***Structure Group*** and then select ***New*** to enter "**Cross Beam 1**".
10. From the ***Structure Group*** drag "**Cross Beam**" with the mouse and drop to the model window. (Figure 4.18-❶)
11. Click ***Activate All*** and ***Iso View*** in the Icon Menu.

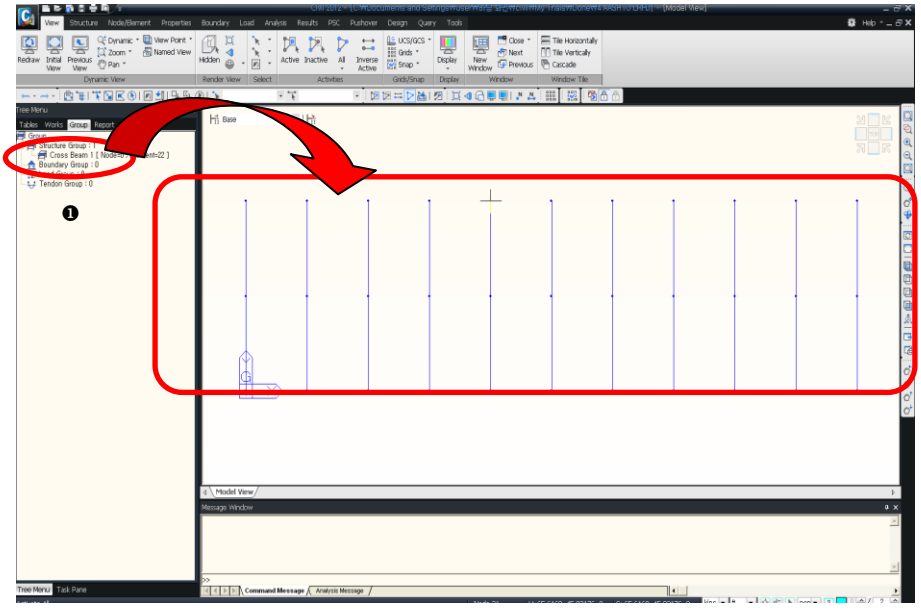

Figure 4.18 Cross Beam Group

Enter Moving Traffic Loads and Static Loads

Enter Load Cases

Set up the Load Cases prior to specifying the loads.

1. Select ***Load>Static Load Cases*** in the Main Menu.
2. Enter "**Dead Load**" in the ***Name*** field of the ***Static Load Cases*** dialog box (Figure 4.19).
3. Select "**Dead Load**" in the ***Type*** selection field.
4. Click Add.
5. Enter "**Sidewalk Load**" in the ***Name*** field.
6. Select "**Dead Load**" in the ***Type*** selection field.
7. Click Add.
8. Click Close.

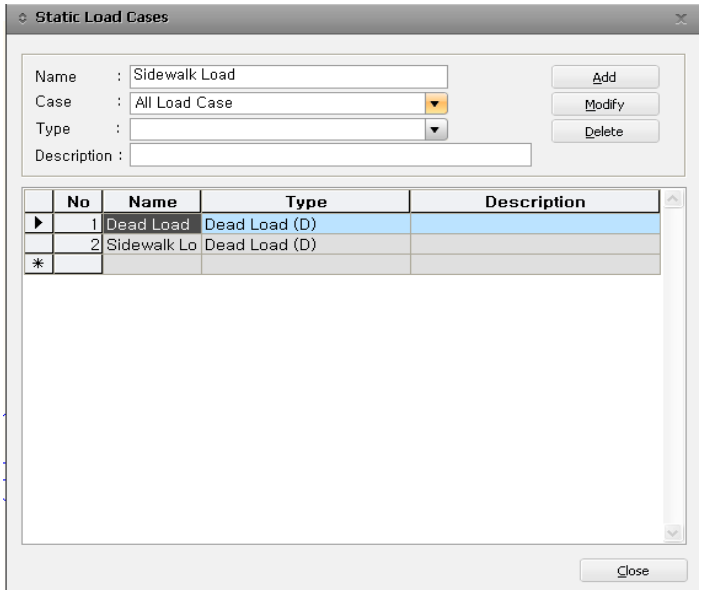

Figure 4.19 Static Load Cases Window

Define Static Loads

Specify the static load cases (Load Cases 1 and 2).

The dead and sidewalk loadings are assumed to be applied only on the main girders for simplicity (Figure 4.20).

1. Click ***Select Identity-Elements***.
2. Select "**Section**" in the ***Select Type*** field.
3. Select "**1 : Main Girder**" in the ***Section*** selection field.
4. Click Add.
5. Click Close in the ***Select Identity-Elements*** dialog box.
6. Select ***Load> Beam Load> Element*** in the Main Menu.
7. Select "**Dead Load**" in the ***Load Case Name*** selection field.
8. Select "**Add**" in the ***Options*** selection field.
9. Select "**Uniform Loads**" in the ***Load Type*** selection field.
10. Select "**Global Z**" in the ***Direction*** selection field.
11. Select "**No**" in the ***Projection*** selection field.
12. Select "**Relative**" in the ***Value*** selection field.
13. Enter "**0**", "**1**" and "**-90**" in the ***x1***, ***x2*** and ***w*** fields, respectively.
14. Click Apply.
15. Click ***Select Previous***.
16. Select "**Sidewalk Load**" in the ***Load Case Name*** field.
17. Select "**Add**" in the ***Options*** selection field.
18. Select "**Uniform Loads**" in the ***Load Type*** selection field.
19. Select "**Global Z**" in the ***Direction*** selection field.
20. Select "**No**" in the ***Projection*** selection field.
21. Select "***Relative***" in the ***Value*** selection field.
22. Enter "**0**", "**1**" and **"-6**" in the ***x1***, ***x2*** and ***w*** fields, respectively.
23. Click Apply.
24. Click Close.

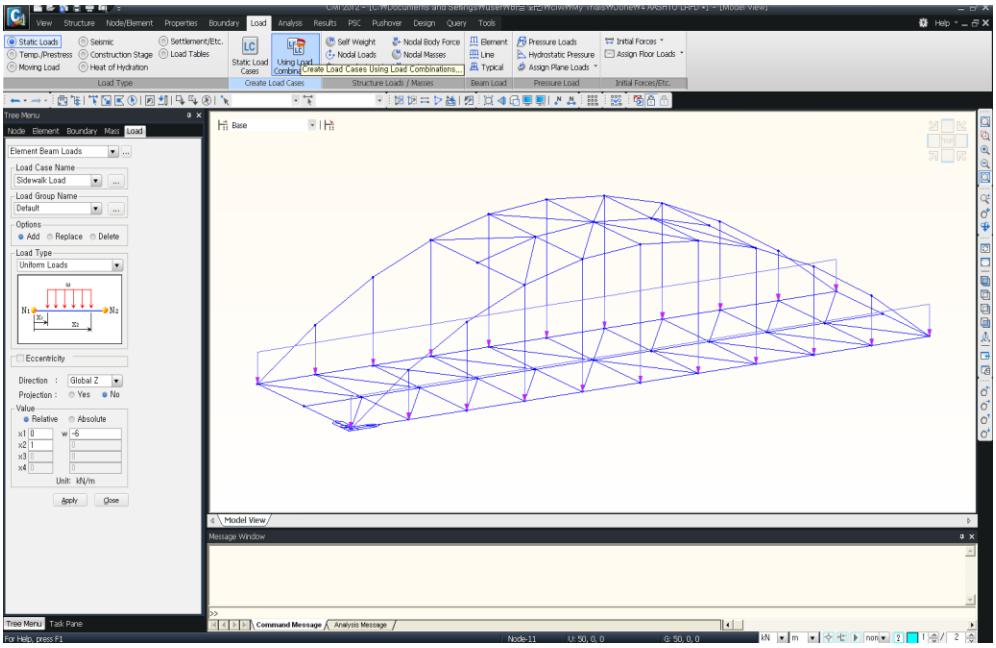

Figure 4.20 Current Loading Condition

Define Moving Traffic Loads

First, define the traffic line lanes (Figure 4.21).

1. Select ***Load*** > ***Moving Load*** from the Main Menu.
2. Select **BS** for the ***Moving Load Code*** selection
3. Select ***Moving Load Analysis Data*** > ***Traffic Line Lanes.***
4. Click Add in the ***Traffic Line Lanes*** dialog box.
5. Enter "**Lane 1**" in the ***Lane Name*** field.
6. Enter "**-1.75**" in the ***Eccentricity*** field.
7. Enter "**1**" in the ***Wheel Spacing*** field.
8. Enter "**3.5**" in the ***Lane Width*** field.
9. Select "**Cross Beam**" in the ***Vehicle Load Distribution*** field.
10. Select "**Cross Beam 1**" in the ***Cross Beam Group*** field.
11. Select "**2 Points**" among the options of ***2 Points***, ***Picking*** and ***Element Number*** in ***Selection by*** and click the field below it. Once the background color turns to pale green, assign nodes **1** and **11**.
12. Click OK.
13. Click Add in the ***Traffic Line Lanes*** dialog box.
14. Enter "**Lane 2**" in the ***Lane Name*** field.
15. Enter "**-5.25**" in the ***Eccentricity*** field.
16. Enter "**1**" in the ***Wheel Spacing*** field.
17. Enter "**3.5**" in the ***Lane Width*** field.
18. Select "**Cross Beam**" in the ***Vehicle Load Distribution*** field.
19. Select "**Cross Beam 1**" in the ***Cross Beam Group*** field.
20. Select "**2 Points**" among the options of ***2 Points***, ***Picking*** and ***Element Number*** in ***Selection by*** and click the field to the right. Once the background color turns to pale green, assign nodes **1** and **11**.
21. Click OK.
22. Click Add in the ***Traffic Line Lanes*** dialog box.
23. Enter "**Lane 3**" in the ***Lane Name*** field.
24. Enter "**-8.75**" in the ***Eccentricity*** field.
25. Enter "**1**" in the ***Wheel Spacing*** field.

When a traffic lane is curved or when the lane data entry with 2 Points becomes awkward due to discontinuity, select "Number" and directly type in the element numbers. (In this case, even if you select "Number" and input "20 to 29", the same traffic lanes are selected)

26. Enter "**3.5**" in the ***Lane Width*** field.
27. Select "**Cross Beam**" in the ***Vehicle Load Distribution*** field.
28. Select "**Cross Beam 1**" in the ***Cross Beam Group*** field.
29. Select "**2 Points**" among the options of ***2 Points***, ***Picking*** and ***Element Number*** in ***Selection by*** and click the field to the right. Once the background color turns to pale green, assign nodes **1** and **11**.
30. Click OK.
31. Click Add in the ***Traffic Line Lanes*** dialog box.
32. Enter "**Lane 4**" in the ***Lane Name*** field.
33. Enter "**-12.25**" in the ***Eccentricity*** field.
34. Enter "**1**" in the ***Wheel Spacing*** field.
35. Select "**Cross Beam**" in the ***Vehicle Load Distribution*** field.
36. Select "**Cross Beam 1**" in the ***Cross Beam Group*** field.
37. Enter "**3.5**" in the ***Lane Width*** field.
38. Select "**2 Points**" among the options of ***2 Points***, ***Picking*** and ***Element Number*** in ***Selection by*** and click the field to the right. Once the background color turns to pale green, assign nodes **1** and **11**.
39. Click OK
40. Click Close

midas Civil contains the standard vehicle loads such as BS 5400, BS BD 37/01, AASHTO Standard, AASHTO LRFD, Caltrans, etc.

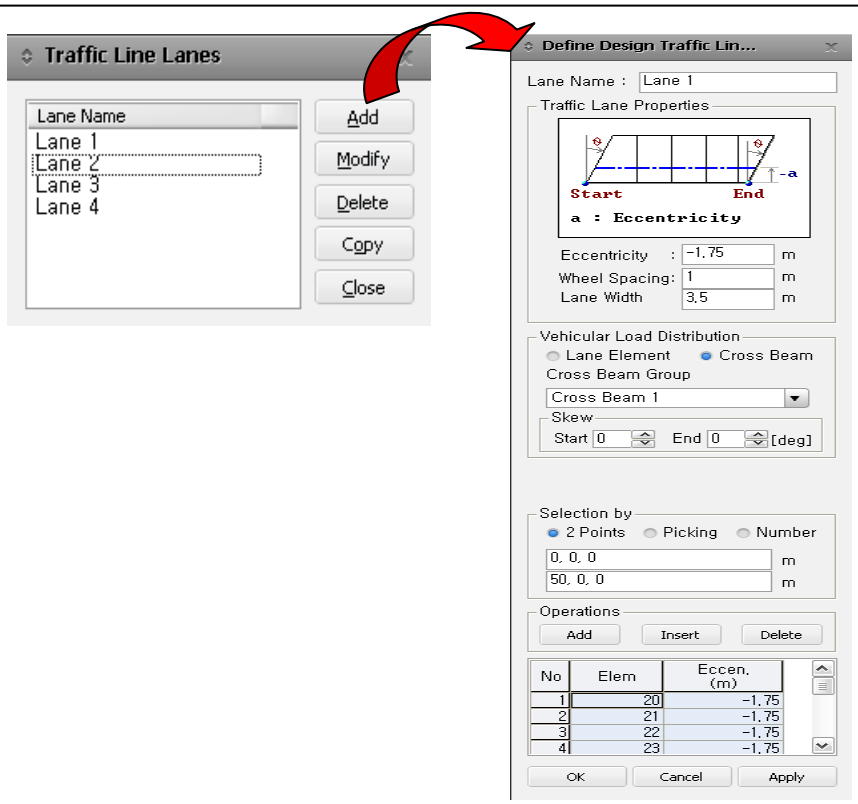

Figure 4.21 Traffic Line Lanes Dialog Box

The method for defining the moving traffic loads, HA and HB (Figure 4.22), is explained below.

1. Select ***Load*** > ***Moving Load Analysis Data*** > ***Vehicles*** from the Main Menu.
2. Click Add Standard in the ***Vehicles*** dialog box.
3. Select "**BS BD 37/01 Standard Load**" in the ***Standard Name*** field.
4. Confirm "**HA & HB (Auto)**" in the ***Vehicle Load Name*** and ***Vehicle Load Type*** fields.
5. Input **45** on **No. of Units** field.
6. Click OK.
7. Click Close.

- Use the Moving Load Cases function to define the vehicle loading cases such as the maximum / minimum number of lanes simultaneously subjected to the vehicle load, the type of vehicle and the lane onto which the load is applied, etc. (Refer to the Online Manual for details)

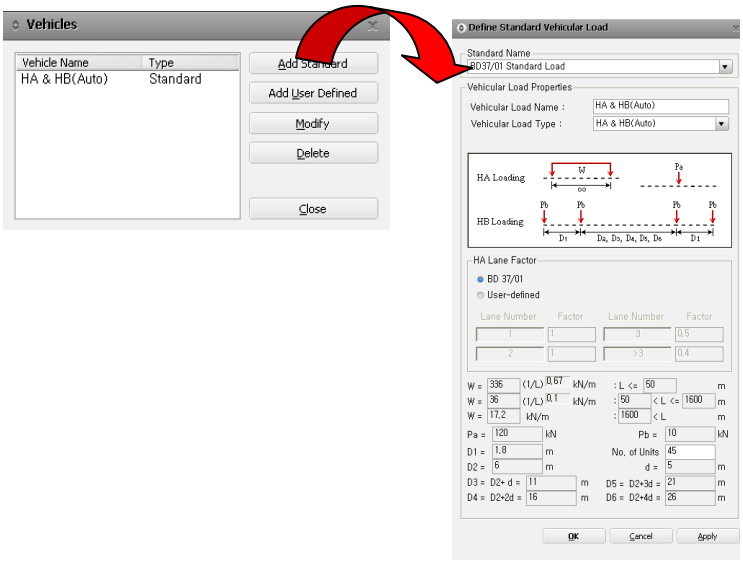

Figure 4.22 Definition of Standard Vehicle Loads

Use the Moving Load Cases Function to define the vehicle loading conditions, i.e. which vehicle loads are applied on which traffic lanes. Also define the maximum and minimum numbers of traffic lanes that can be loaded with vehicle loads simultaneously.

Load factors for HA loading for ULS, SLS, Combination 1 and Combinations 2 & 3 are taken from Section 6.2.7 of BD 37/01. Load factors for HB loading for ULS, SLS, Combination 1 and Combinations 2 & 3 are taken from Section 6.3.4 of BD 37/01. These load factors are automatically incorporated into moving load analysis results. Therefore, to avoid duplication, the user should not apply the load factors for moving loads while generating the Load Combinations.

Define the moving traffic load cases (Figure 4.23).

1. Select ***Load*** > ***Moving Load Analysis Data***>***Moving Load Cases*** from the Main Menu.
2. Click Add in the ***Moving Load Cases*** dialog box.
3. Enter "**MVL1**" in the ***Load Case Name*** field of the ***Moving Load Case*** dialog box.
4. Select "**Ultimate Limit State**" in ***Type of Design Combination Factor***.
5. Select "**Combination 1**" in ***Combination of Loads***.
6. Click Add in the ***Sub-Load Cases*** field.
7. Select "**HA & HB (Auto)**" in the ***Vehicle*** field.
8. Enter "**1**" in the ***Scale Factor*** field.
9. Enter "**4**" in the ***Number of Loaded Lanes*** field.
10. Select "**Lane 1, Lane 2, Lane 3** and **Lane 4**" in ***List of Lanes*** of ***Assignment Lanes*** and click -> to move to ***Selected Lanes***.
11. Select "**Lane 1** and **Lane 2**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
12. Select "**Lane 3** and **Lane 4**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
13. Click OK in the ***Sub-Load Cases*** dialog box.
14. Click OK in the ***Define Moving Load Case*** dialog box.
15. Click Add in the ***Moving Load Cases*** dialog box.
16. Enter "**MVL2**" in the ***Load Case Name*** field of the ***Moving Load Case*** dialog box.
17. Select "**Ultimate Limit State**" in ***Type of Design Combination Factor.***
18. Select "**Combination 2 or 3**" in ***Combination of Loads.***
19. Click Add in the ***Sub-Load Cases*** field.
20. Select "**HA & HB (Auto)**" in the ***Vehicle*** field.
21. Enter "**1**" in the ***Scale Factor*** field.
22. Enter "**4**" in the ***Number of Loaded Lanes*** field.
23. Select "**Lane 1, Lane 2, Lane 3** and **Lane 4**" in ***List of Lanes*** of ***Assignment Lanes*** and click -> to move to ***Selected Lanes***.
24. Select "**Lane 1** and **Lane 2**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.

25. Select "**Lane 3** and **Lane 4**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
26. Click OK in the ***Sub-Load Cases*** dialog box.
27. Click OK in the ***Define Moving Load Case*** dialog box.
28. Click Add in the ***Moving Load Cases*** dialog box.
29. Enter "**MVL3**" in the ***Load Case Name*** field of the ***Moving Load Case*** dialog box.
30. Select "**Serviceability Limit State**" in ***Type of Design Combination Factor.***
31. Select "**Combination 1**" in ***Combination of Loads.***
32. Click Add in the ***Sub-Load Cases*** field.
33. Select "**HA & HB (Auto)**" in the ***Vehicle*** field.
34. Enter "**1**" in the ***Scale Factor*** field.
35. Enter "**4**" in the ***Number of Loaded Lanes*** field.
36. Select "**Lane 1, Lane 2, Lane 3** and **Lane 4**" in ***List of Lanes*** of ***Assignment Lanes*** and click -> to move to ***Selected Lanes***.
37. Select "**Lane 1** and **Lane 2**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
38. Select "**Lane 3** and **Lane 4**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
39. Click OK in the ***Sub-Load Cases*** dialog box.
40. Click OK in the ***Define Moving Load Case*** dialog box.
41. Click Add in the ***Moving Load Cases*** dialog box.
42. Enter "**MVL4**" in the ***Load Case Name*** field of the ***Moving Load Case*** dialog box.
43. Select "**Serviceability Limit State**" in ***Type of Design Combination Factor.***
44. Select "**Combination 2 or 3**" in ***Combination of Loads.***
45. Click Add in the ***Sub-Load Cases*** field.
46. Select "**HA & HB (BD37/01)**" in the ***Vehicle*** field.
47. Enter "**1**" in the ***Scale Factor*** field.
48. Enter "**4**" in the ***Number of Loaded Lanes*** field.
49. Select "**Lane 1, Lane 2, Lane 3** and **Lane 4**" in ***List of Lanes*** of ***Assignment Lanes*** and click -> to move to ***Selected Lanes***.
50. Select "**Lane 1** and **Lane 2**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
51. Select "**Lane 3** and **Lane 4**" in ***Selected Lanes*** and click -> to move to ***HB Straddling Two Lanes***.
52. Click OK in the ***Sub-Load Cases*** dialog box.

53. Click OK in the ***Define Moving Load Case*** dialog box.
54. Click Close.

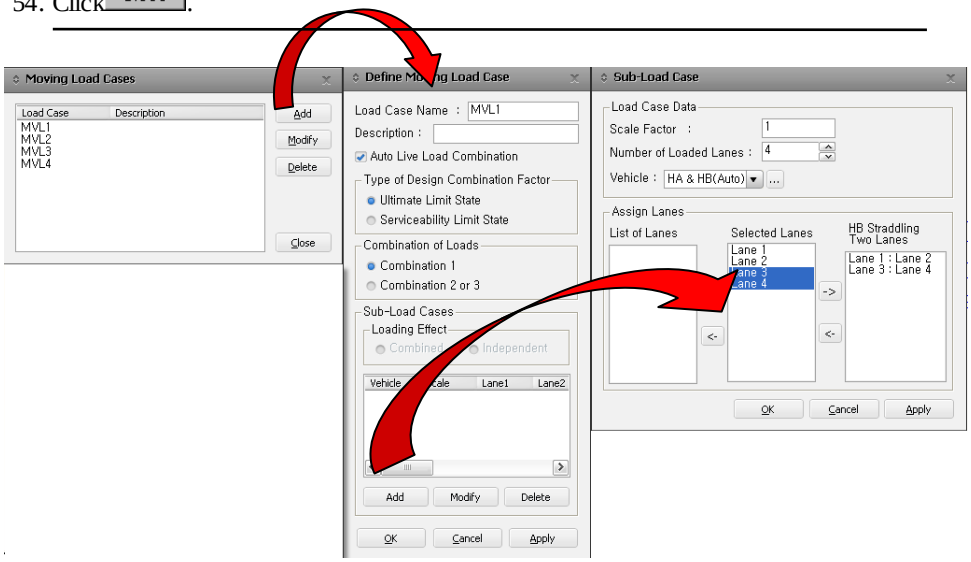

Figure 4.23 Definition of Moving Vehicle Load

Note (refer Section 4.4, BD 37/01):

Combination 1: For highway and foot/cycle track bridges, the loads to be considered are the permanent loads, together with the appropriate primary live loads, and, for railway bridges, permanent loads, together with the appropriate primary and secondary live loads.

Combination 2: For all bridges, the loads to be considered are the loads in combination 1, together with those due to wind, and, where erection is being considered, temporary erection loads.

Combination 3: For all bridges, the loads to be considered are the loads in combination 1, together with those arising from the restraint due to the effects of temperature range and difference, and, where erection is being considered, temporary erection loads.

Combination 4: It does not apply to railway bridges, except for vehicle collision loading on bridge supports. For highway bridges, the loads to be considered are the permanent loads and the secondary live loads, together with the appropriate primary live loads associated with them. Secondary live loads shall be considered separately and are not required to be combined. Each shall be taken with its appropriate associated primary live load.

Define the method of analysis for the moving vehicle load (Figure 4.24).

Use Moving Load Analysis Control to input the number of points on each line element where influence lines should be generated. For example, if "5" is inputted in Influence Generating Point No./Line Element field, it means that the concentrated axle load is applied successively at 5 equally spaced points on each line element, along the direction of the traffic lane. (Also refer to the Structural Analysis functions in the Online Manual for details)

Calculation Filter in Moving Load Analysis Control Data groups only the desired part of the results for review. The grouping reduces the computation time and the size of Results file for large structures.

1. Select ***Analysis*> *Analysis Control*> *Moving Load*** from the Main Menu.
2. Enter "**3**" in the ***Number/Line Element*** field of ***Influence Generating Points***.
3. Select "**Normal**" in ***Frame*** in the ***Analysis Results*** field.
4. Select "**All**" in ***Reactions, Displacements*** and ***Forces/Moments*** under ***Calculation Filters***.
5. Click OK.
6. Click ***Node Number*** (Toggle off).

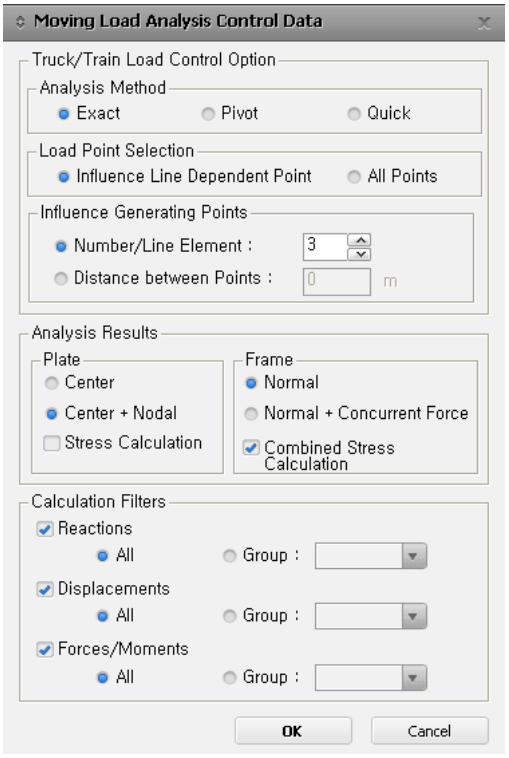

Figure 4.24 Moving Load Analysis Control Dialog Box

Perform Structural Analysis

Perform the structural analysis of the structure attributed with boundary conditions and load cases.

Click ***Analysis***.

Verify and Interpret Analysis Results

Load Combinations

We will now examine the Linear Load Combination method of the 3 load cases (dead load, sidewalk load and moving load) for which structural analyses have been completed.

In this example, we specify only one load combination as noted below, and check its results. The load combination case has been arbitrarily chosen and, as such, it may be irrelevant for any practical design application.

- Load Combinations (LCB): 1.05 Dead Load + 1.05 Sidewalk Load + 1.00 Moving Load

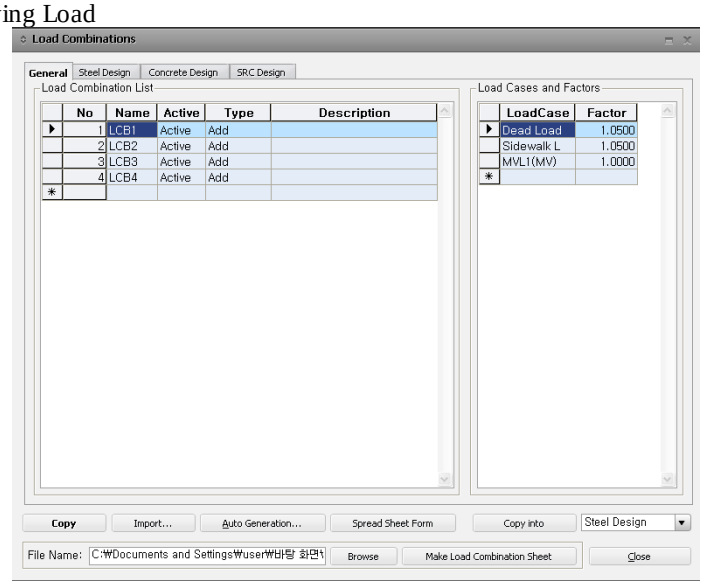

Figure 4.25 Load Combinations Dialog Box

Use ***Results>Combinations*** in the Main Menu to open the ***Load Combinations*** dialog box (Figure 4.25) and enter the following load combination:

1. Select ***Results>Load Combinations*** in the Main Menu.
2. Bring the cursor in the cell right below ***Active*** within the ***Load Combination List***. Click once to get a drop-down menu, then select ***Active***.

Selecting Active under *Active* in the Load Combinations dialog box prompts the load combination used by the design function of the program. (Refer to the Online Manual for details)

3. Enter "**LCB1**" in the ***Name*** field.
4. Select "**Add**" in the ***Type*** selection field.
5. Click the ***Load Case*** selection field and use to select "**Dead Load (ST)**" in the field.
6. Click the second selection field and use to select "**Sidewalk Load (ST)**" in the field.

For load factors, refer to Table 1 of BD 37/01. (Note: Sidewalk Load is also a type of Dead Load).

7. Click the third selection field and use to select "**MVL1(MV)**" in the field.
8. Enter "**1.05**" in ***Factor*** field for "**Dead Load (ST)"** and **"Sidewalk Load (ST)"**.

Load factors for moving loads are already incorporated into the moving load analysis results.

9. Enter "**1.00**" in ***Factor*** field for "**MVL1(MV)"** .
10. Enter "**LCB2**" in the ***Name*** field.
11. Select "**Add**" in the ***Type*** selection field.
12. Click the ***LoadCase*** selection field and use to select "**Dead Load (ST)**" in the field.
13. Click the second selection field and use to select "**Sidewalk Load (ST)**" in the field.
14. Click the third selection field and use to select "**MVL2(MV)**" in the field.
15. Enter "**1.05**" in ***Factor*** field for "**Dead Load (ST)"** and **"Sidewalk Load (ST)"**.
16. Enter "**1.00**" in ***Factor*** field for "**MVL2(MV)"**.
17. Enter "**LCB3**" in the ***Name*** field.
18. Select "**Add**" in the ***Type*** selection field.
19. Click the ***LoadCase*** selection field and use to select "**Dead Load (ST)**" in the field.
20. Click the second selection field and use to select "**Sidewalk Load (ST)**" in the field.
21. Click the third selection field and use to select "**MVL3(MV)**" in the field.

22. Enter "**1.00**" in ***Factor*** field for "**Dead Load (ST)"** and **"Sidewalk Load (ST)"**.
23. Enter "**1.00**" in ***Factor*** field for "**MVL3(MV)"**.
24. Enter "**LCB4**" in the ***Name*** field.
25. Select "**Add**" in the ***Type*** selection field.
26. Click the ***LoadCase*** selection field and use  to select "**Dead Load (ST)**" in the field.
27. Click the second selection field and use  to select "**Sidewalk Load (ST)**" in the field.
28. Click the third selection field and use  to select "**MVL4(MV)**" in the field.
29. Enter "**1.00**" in ***Factor*** field for "**Dead Load (ST)"** and **"Sidewalk Load (ST)"**.
30. Enter "**1.00**" in ***Factor*** field for "**MVL4(MV)"**.
31. Click 

Verify Deformed Shape

Use the following procedure to check the deformed shape (Figure 4.26):

1. Click ***Results*** > ***Deformations*** > ***Deformed Shape*** from the Main Menu (Figure 4.26-❶).
2. Select "**CBmin:LCB1**" in the ***Load Cases/Combinations*** selection field.
3. Select "**DXYZ**" in the ***Components*** selection field.
4. Check (✓) "**Undeformed**" and "**Legend**" in the ***Type of Display*** selection field.
5. Click the button ... to the right of ***Legend*** in the ***Type of Display*** selection field.
6. Select **"Fixed"** in ***Rank Value Type*** and enter **"2"** in the ***Decimal Point*** field.
7. Check (✓) in ***Apply upon OK***.
8. Click OK.
9. Click the button ... to the right of ***Deform*** in the ***Type of Display*** selection field.
10. Select "**Real Deform**" in the ***Deformation*** selection field.
11. Confirm Check (✓) in ***Apply upon OK***.
12. Click OK.
13. Click the unit system selection button of the Status Bar and change "**m**" to "**mm**".
14. Click ***Hidden*** (Toggle on).

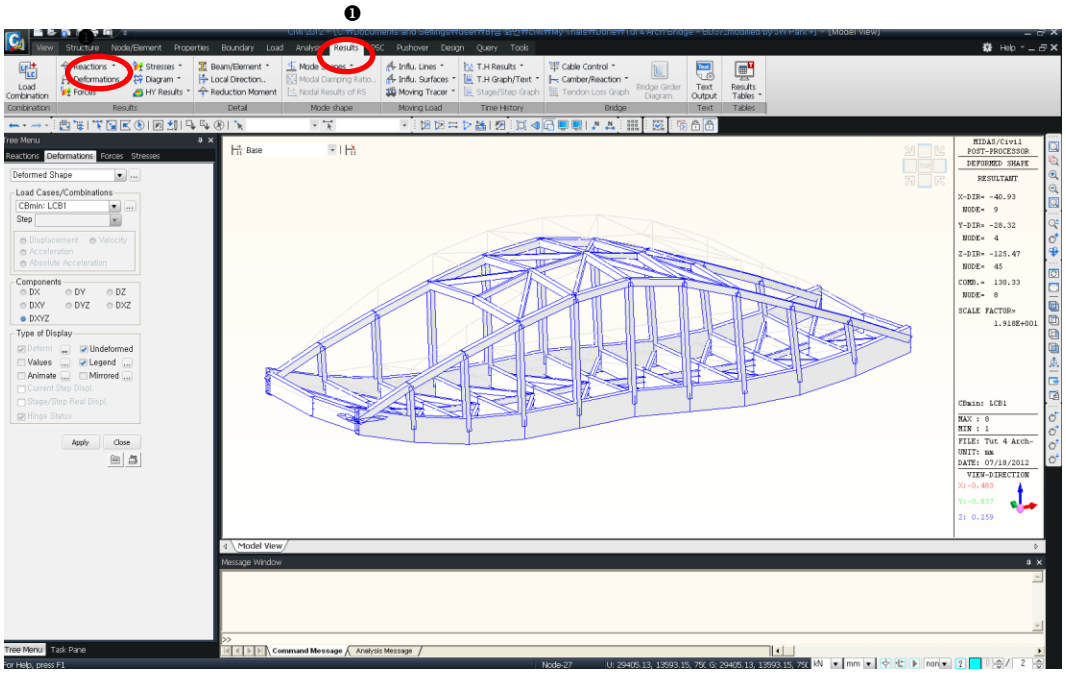

Figure 4.26 Deformed Shape

Shear Force and Bending Moment Diagrams

The method for reviewing the shear force and bending moment diagrams are quite similar. Therefore, only the method for displaying the bending moment diagram is reviewed in this case. This method is not intended to capture the bending moment diagram of the entire structure. The purpose is to display only the results related to a specific part of the structure. For instance, the following steps illustrate the procedure to display the bending moment diagram in the X-Z plane (Figure 4.27).

Quite often, analysis results for the structural behavior of specific parts are required in practice. Use the Select Plane to separately extract the results at the desired planar section.

1. Click ***Hidden*** (Toggle off).
2. Click ***Initial View*** in the Icon Menu**.**
3. Click ***Select by Plane.***
4. Select "**XZ Plane**" in the ***Plane*** tab.
5. Click in the ***Y Position*** field and select a point with the mouse, which defines the desired X-Z plane (the color of the selected plane changes).
6. Click Close.
7. Click the unit system selection button of the Status Bar and change "**mm**" to "**m**".
8. Click ***Activate***.
9. Click ***Front View***.
10. Select ***Beam Diagram*** in ***Results*** > ***Forces*** from the Main Menu (Figure 4.27-❶).
11. Select "**MVall: MVL1**" in ***Load Cases/Combinations*** selection field.
12. Select "**My**" in the ***Components*** selection field.
13. Select "**5 Points**" and "**Line Fill**" in ***Display Options*** selection field.
14. Enter "**1.0**" in the ***Scale*** field.
15. Check (✓) "**Legend**" in the ***Type of Display*** selection field.
16. Click Apply.

MVmin: The minimum member force resulting from the vehicle load applied to the structure.
MVmax: The maximum member force resulting from the vehicle load applied to the structure.

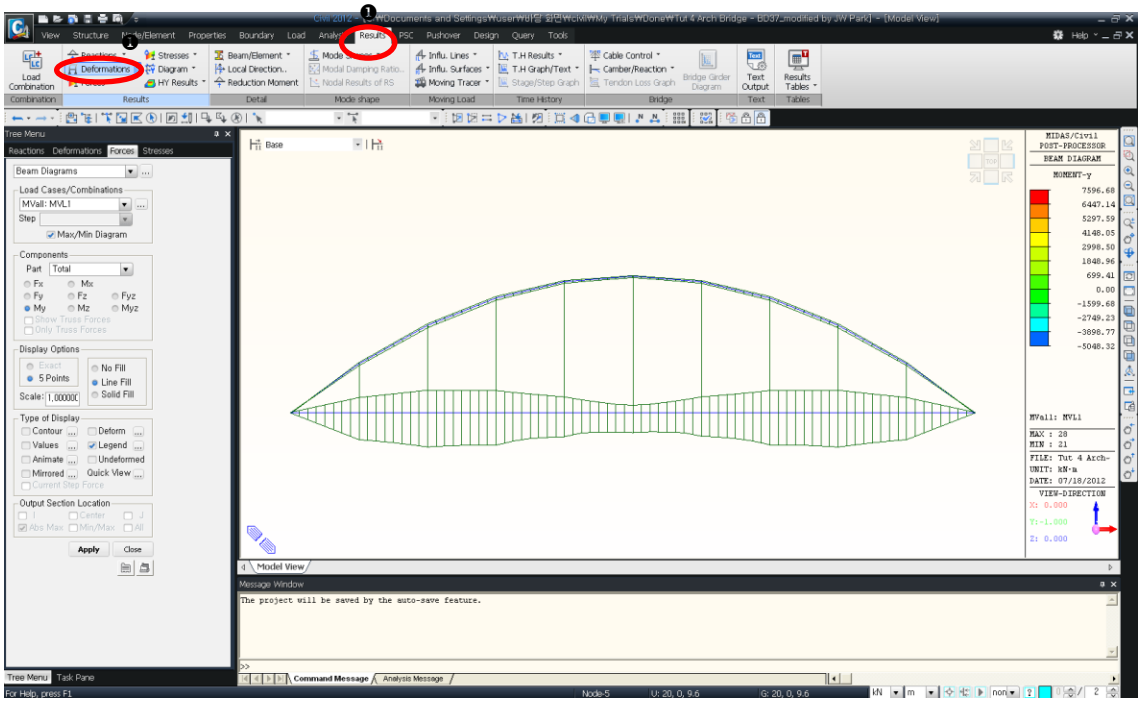

Figure 4.27 My diagrams (XZ Plane) for the Beam

Refer to the Online Manual for more details on Influence Lines.

View Influence Lines Results

First we will examine the influence lines for a support reaction. Figure 4.28 shows the results for support B1 (node 1).

1. Click *Activate All*
2. Click *Reactions* in ***Result*** > ***Moving Load*** > ***Influ. Lines*** from the Main Menu. (Figure 4.28-❶)
3. Select "**Lane 1**" in the ***Line/Surface Lanes*** field.
4. Enter "**1**" in the ***Key Node*** field.
5. Enter "**1.0**" in the ***Scale Factor*** field.
6. Select "**FZ**" in the ***Components*** field.
7. Check (✓) "**Legend**" in the ***Type of Display*** field.
8. Click Apply.
9. Click *Iso View*.

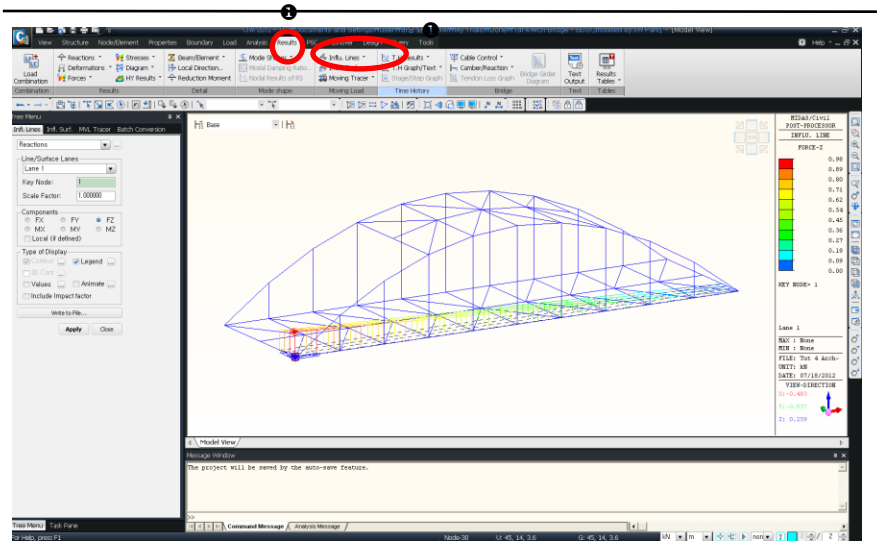

Figure 4.28 Reaction Influence Line

Use animation to investigate the results of the support reaction influence line (Figure 4.30).

1. Check (✓) "**Legend**" and "**Animate**" in the ***Type of Display*** field.
2. Click Apply.
3. Click ***Record*** (Figure 4.29❶).
4. Click ***Close*** (Figure 4.29–❷).

🎧 After reviewing the animation, click Close to restore the original screen (Figure 4.30❷).

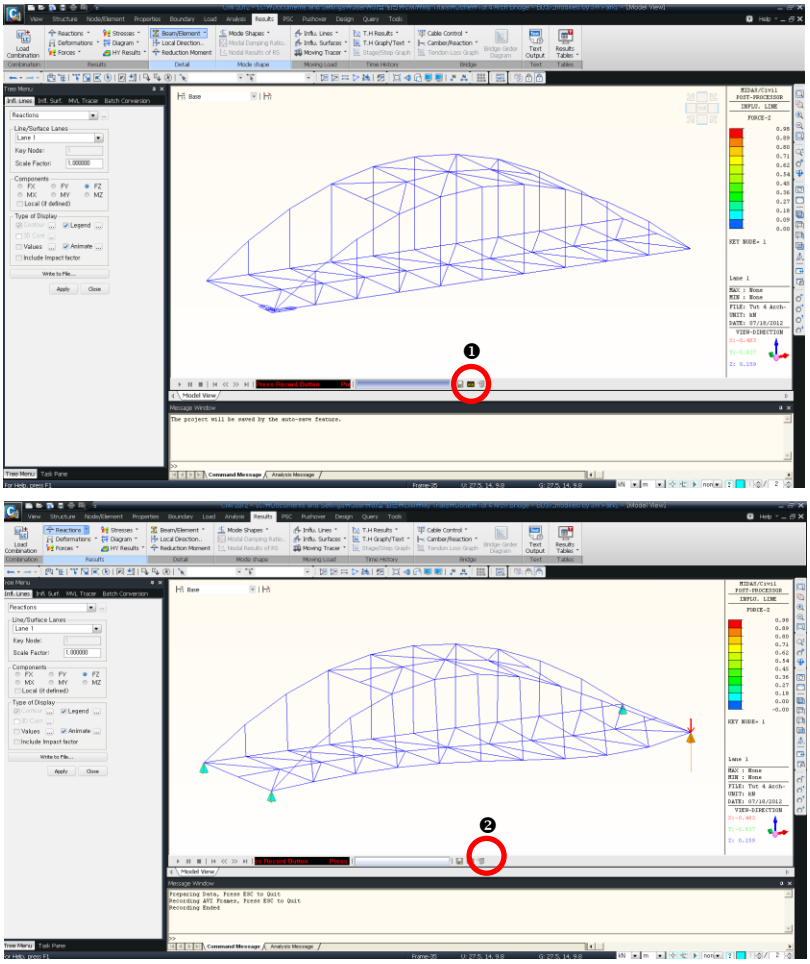

Figure 4.29 Animation of Support Reaction Influence Line

Figure 4.30 shows the deflection influence line.

1. Click the unit system selection button of the Status Bar and change "**m**" to "**mm**".
2. Click ***Displacements*** in ***Result > Moving Load > Influ. Lines*** from the Main Menu. (Figure 4.30-❶).
3. Confirm "**Lane 1**" in the ***Line/Surface Lanes*** selection field.
4. Enter "**15**" in the ***Key Node*** field.
5. Enter "**1.0**" in the ***Scale Factor*** field.
6. Select "**DZ**" in the ***Components*** selection field.
7. Check (✓) "**Legend**" in the ***Type of Display*** selection field.
8. Click the button ... to the right of ***Legend*** in the ***Type of Display*** selection field.
9. Select **"Fixed"** in ***Rank Value Type*** and enter **"4"** in the ***Decimal Point*** field.
10. Click Apply.

Mouse Editor may be used for the Key Node field to select the nodes directly.

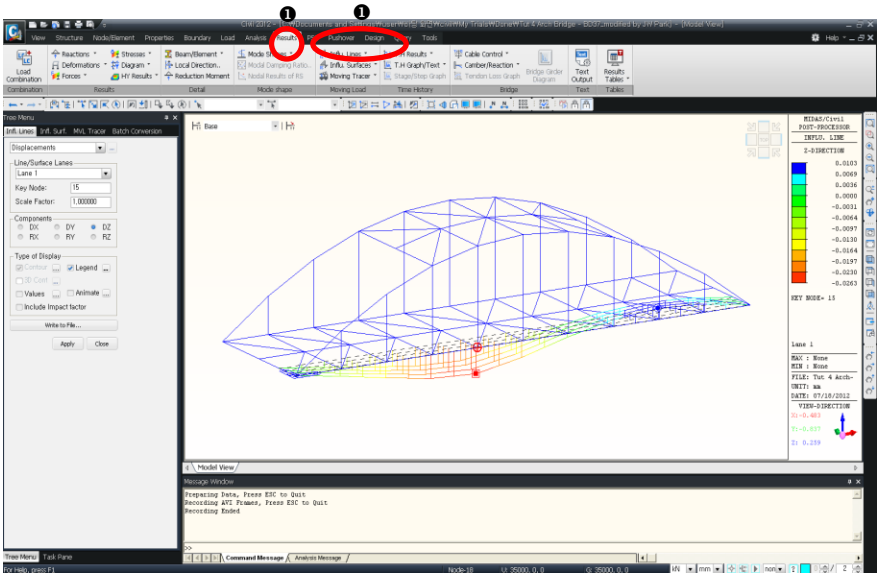

Figure 4.30 Deflection Influence Line

Figure 4.31 shows the moment influence line.

1. Click the unit system selection button of the Status Bar and change "**mm**" to "**m**".
2. Click *Beam Forces/Moments* in ***Result*** > ***Moving Load*** > ***Influ. Lines*** from the Main Menu. (Figure 4.31-❶).
3. Confirm "**Lane 1**" in the ***Line/Surface Lanes*** selection field.
4. Enter "**23**" in the ***Key Element*** field.
5. Enter "**1.0**" in the ***Scale Factor*** field.
6. Select "**i**" in the ***Parts*** selection field.
7. Select "**My**" in the ***Components*** selection field.
8. Check (✓) "**Legend**" in the ***Type of Display*** selection field.
9. Click Apply.

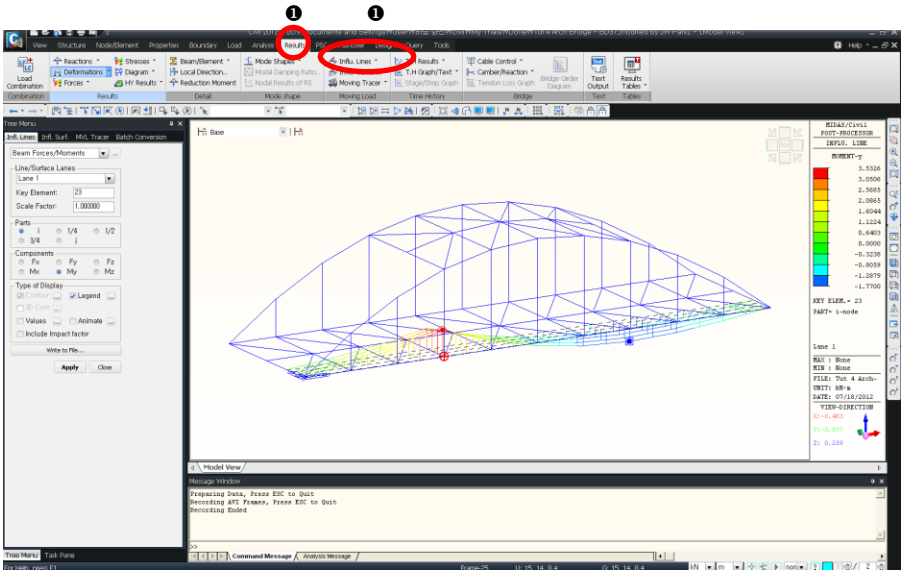

Figure 4.31 Moment Influence Line

Moving Load Tracer can be applied to the results obtained from the structural analysis related to Moving Vehicle Load. It displays the results similar to an influence line or influence surface diagram by tracking the location of the vehicle loading.

Use ***Moving Load Tracer*** to check the reactions on the structure resulting from the movement of vehicular traffic (Figure 4.32).

1. Select ***Initial View***.
2. Click ***Select Plane***.
3. Select "**XY Plane**" in the ***Plane*** tab, click in the ***Z Position*** field, and select node **1** with the mouse.
4. Click Close.
5. Click ***Activate***.
6. Select ***Results*** > ***Moving Load*** > ***Moving Tracer***>***Reactions*** in the Main Menu.
7. Select "**MVmax : MVL1**" in the ***Moving Load Cases*** selection field.
8. Enter "**1**" in the ***Key Node*** field.
9. Enter "**1.0**" in the ***Scale Factor*** field.
10. Select "**FZ**" in the ***Components*** field selection.
11. Check (✓) "**Contour**", "**Legend**" and "**Applied Loads**" in the ***Type of Display*** selection field.
12. Click Apply.

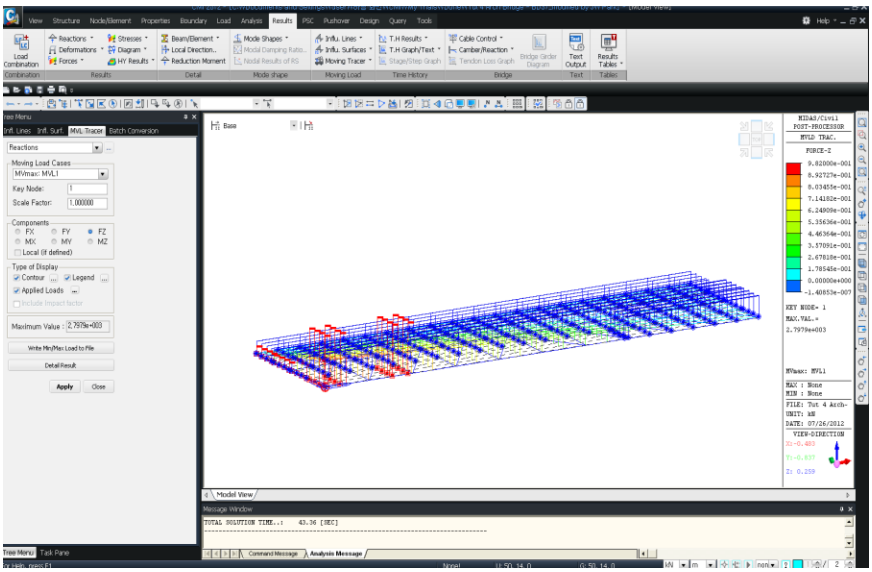

Figure 4.32 Checking the Loading points of a Vehicle using Moving Load Tracer

Moving Load Tracer generates a particular loading condition, which produces specific results due to a vehicle moving load. The traced moving load condition is expressed in terms of an influence line or surface.

Using ***Moving Load Tracer***, we can now check the moving load location, which causes the movement at the i[th] end of element 28.

1. Select ***Results>Moving Load Tracer>Beam Forces/Moment*** in the Main Menu.
2. Select "**MVmax : MVL1**" in the ***Moving Load Cases*** selection field.
3. Enter "**28**" in the ***Key Element*** field.
4. Enter "**1.0**" in the ***Scale Factor*** field.
5. Select "**i**" in the ***Parts*** selection field.
6. Select "**My**" in the ***Components*** selection field.
7. Check (✓) "**Contour**", "**Legend**" and "**Applied Loads**" in the ***Type of Display*** selection field.
8. Click Apply.

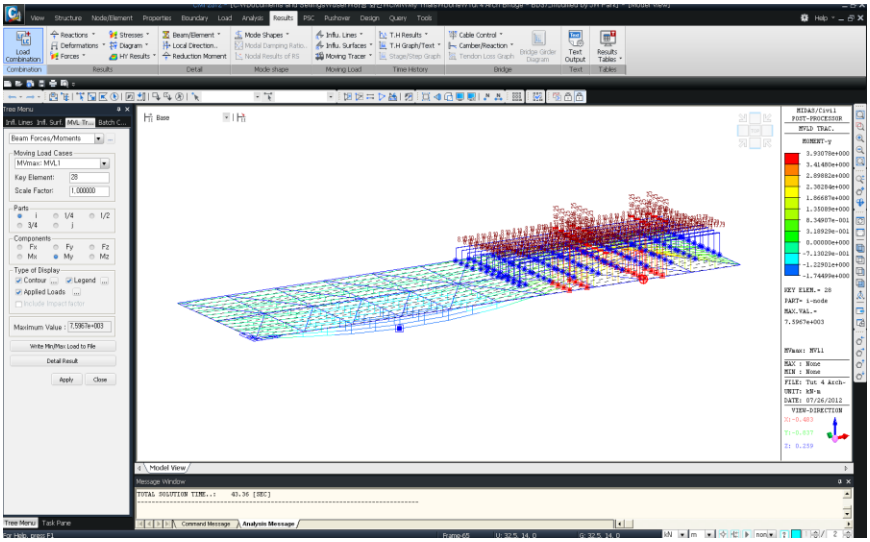

Figure 4.33 Checking the Loading points of a Vehicle using Moving Load Tracer

Having determined the moving load location by the ***Moving Load Tracer***, we will now examine the method of converting the live load into a static load. If we click the Write Min/Max Load to File button of the ***Moving Load Tracer Function***, the converted static load is saved in an MCT file. When we execute the MCT file using the ***MCT Command Shell*** in the model file already generated, the static load will be entered in the model. (For details on ***MCT Command Shell,*** refer to the Online Manual).

1. Click Write Min/Max Load to File.
2. Click OK in ***Moving Load Converted to Static Load*** dialog box.
3. Select ***File>Exit*** in the ***MIDAS/Text Editor***.
4. In the Main Menu, select ***Tools>MCT Command Shell>Open***  ***>File Name (*MVmaxMVL1My28.mct*)>Open***.
5. Click Run in the ***MCT Command Shell*** dialog box.
6. Click Yes when prompted for ***"Analysis/design results will be deleted; Continue?"*** in the ***CVLw*** dialog box.
7. Click Close in the ***MCT Command Shell*** dialog box.
8. Select ***Load>Static Load Cases*** in the Main Menu.
9. Confirm that **"MVmaxMVL1My28.mct"** is generated under the ***Name*** column in the ***Static Load Cases*** dialog box.
10. Click Close in the ***Static Load Cases*** dialog box.
11. Click  ***Analysis***.

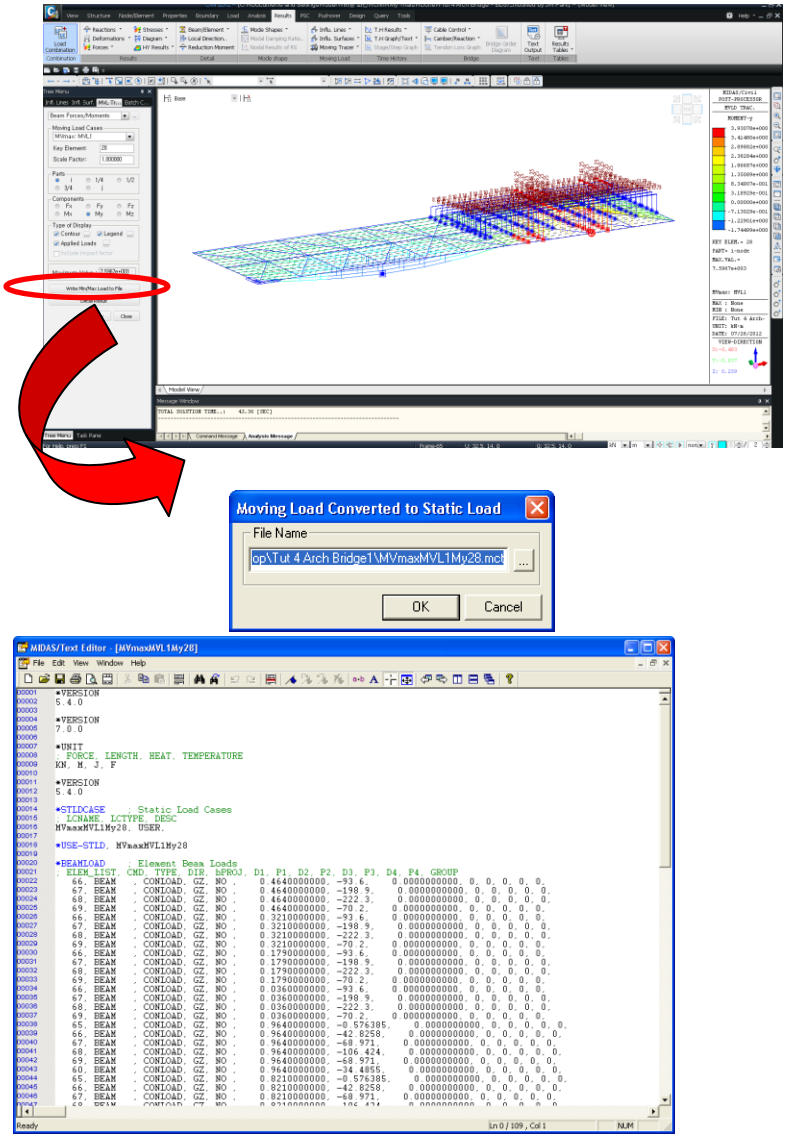

Figure 4.34 Live load automatically converted into static load

We can now check the bending moment due to the static load that was generated from the live load which caused the movement at the ith end of element 28.

1. Select ***Beam Diagram*** in ***Result > Force*** (Figure 4.35❶).
2. Select **"ST:MVmaxMVL1My28"** in the ***Load Cases/Combinations*** selection field.
3. Select "**My**" in the ***Components*** selection field.
4. Select "**5 Points**" and "**Solid Fill**" in ***Display Options*** selection field.
5. Enter "**1.0**" in the ***Scale*** field.
6. Check (✓) "**Legend**" in the ***Type of Display*** selection field.
7. Click Apply.

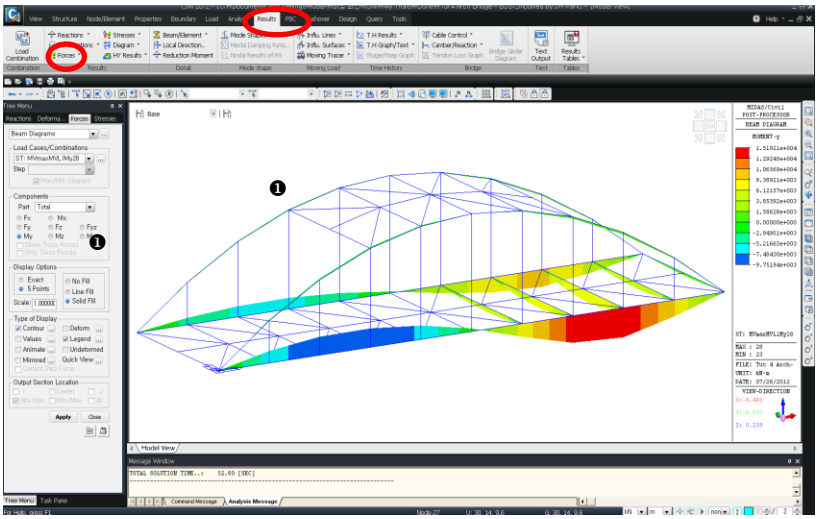

Figure 4.35 My beam diagrams for the live load converted into static load