

Tutorial 4

Arch Bridge

Civil

TUTORIAL 4. ARCH BRIDGE

Summary	1
Analysis Model and Load Cases / 2	
File Opening and Preferences Setting	5
Enter Material and Section Properties	7
Structural Modeling Using Nodes and Elements	10
Generate the Arch Ribs / 10	
Generate the Hangers / 11	
Generate the Main Girder and Duplicate the Arch Frame / 13	
Generate the Cross Beams / 15	
Generate the Bracings / 16	
Enter Structure Boundary Conditions	21
Boundary Conditions for Beam End Connections / 22	
Generate the Cross Beam Group / 24	
Enter Moving Traffic Loads and Static Loads	26
Enter Load Cases / 26	
Define Static Loads / 27	
Define Moving Traffic Loads / 29	
Perform Structural Analysis	37
Verify and Interpret Analysis Results	37
Load Combinations / 37	
Verify Deformed Shape / 40	
Shear Force and Bending Moment Diagrams / 41	
View Influence Lines Results / 44	

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

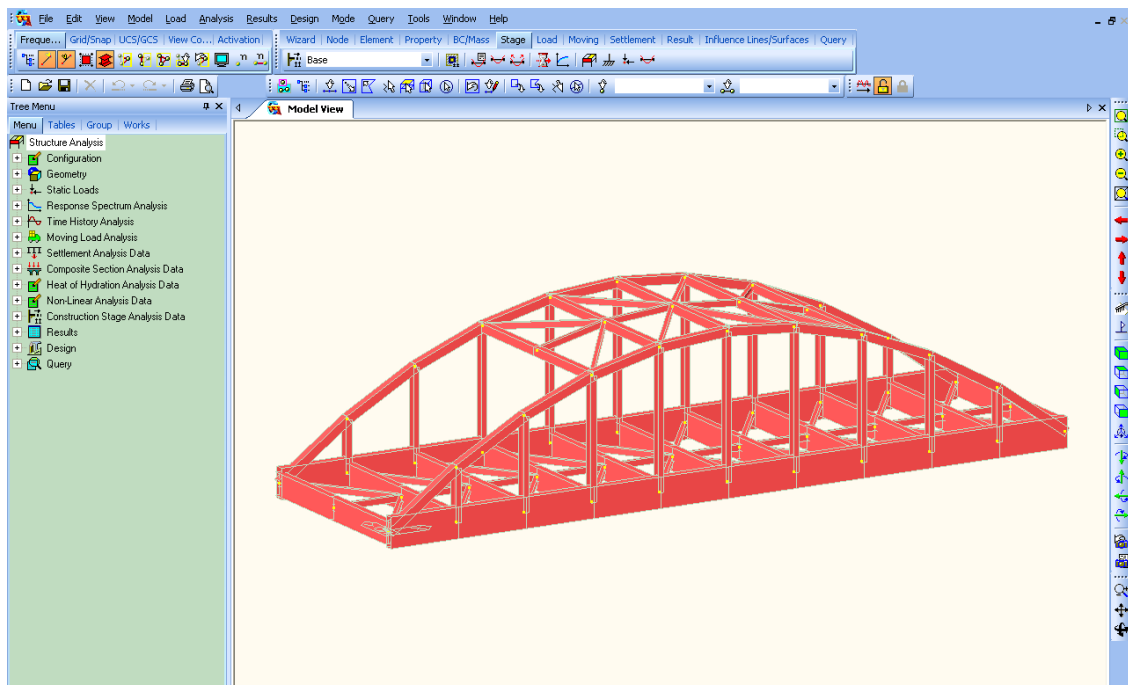
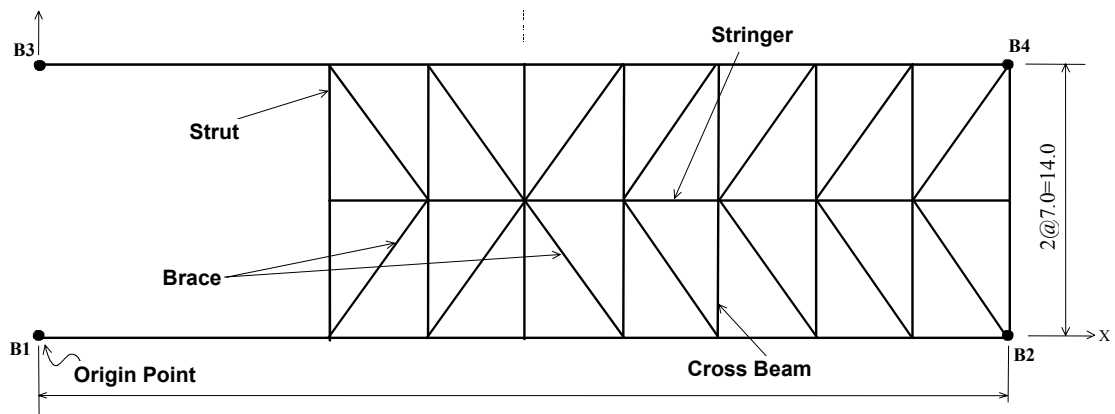


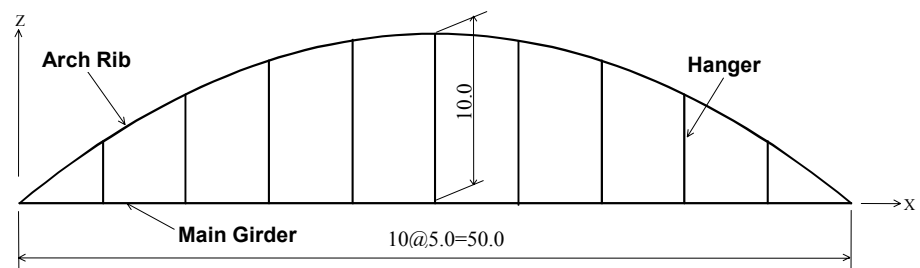
Figure 4.1 Arch Bridge Model

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.



(a) Framing Plan



(b) Front View Elevation



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.

The following procedure describes the method for displaying the icons required for modeling on the screen. This will help the user in making an efficient use of the Icon Menu.

1. Select **Tools>Customize>Customize>Toolbars** tab in the Main Menu.
2. Check (✓) all the items in the **Toolbars** selection field (Figure 4.3).
3. Click .

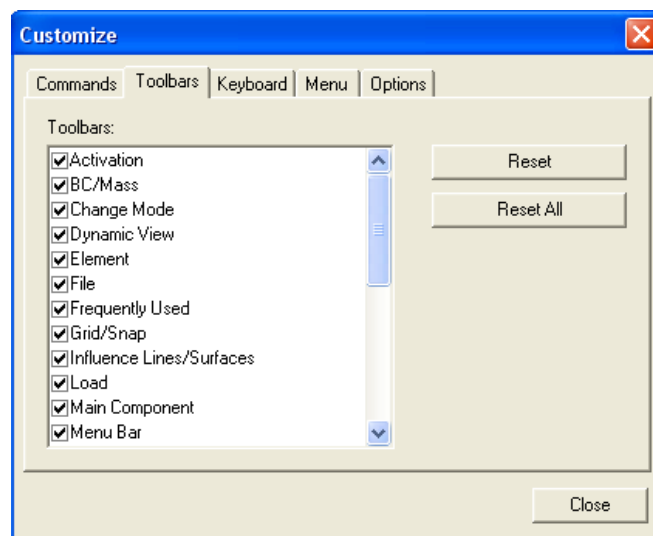


Figure 4.3 Toolbars Dialog Box

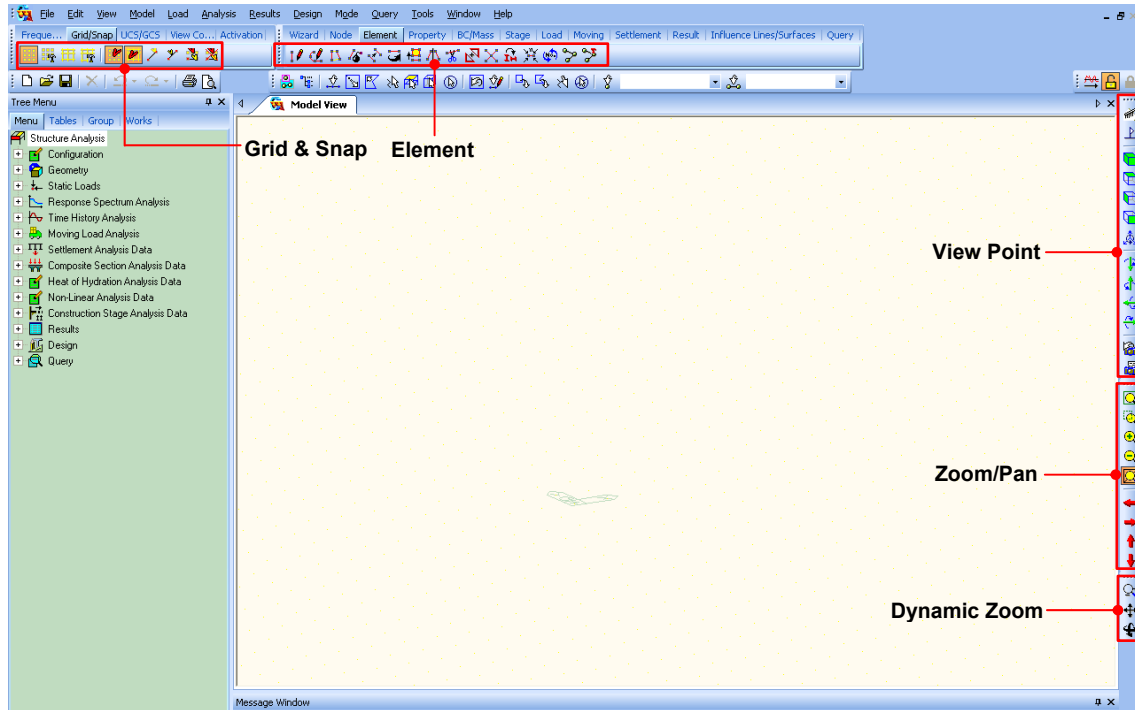


Figure 4.4 Model View Layout

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: ST 2100×600×10/10 – Main Girder
 - 2: I 1540×500×14/27 – Cross Beam
 - 3: ST 600×600×16/14 – Arch Rib
 - 4: I 600×400×12/16 – Hanger
 - 5: ST 600×500×10/14 – Strut
 - 6: W16×100 – Bracing & Stringer

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.

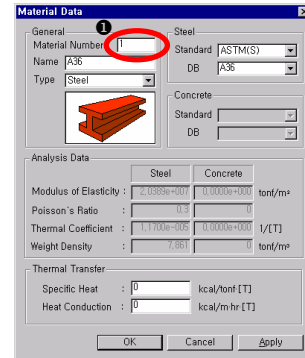
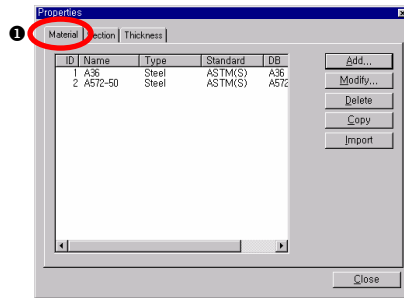












Figure 4.5 Sectional Attributes Dialog Box Figure 4.6 Material Properties Dialog Box

1. Select **Model>Properties>Material** in the Main Menu.
2. Click  **Material** in the **Properties** dialog box (Figure 4.5 ①).
3. Click .
4. Confirm “1” in the **Material Number** field of **General** (Figure 4.6 ①).
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 .
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 .

1. Select the **Section** tab in the **Properties** dialog box (Figure 4.5) or select  **Section** in the **Property** toolbar.
2. Click .
3. Confirm “1” in the **Section ID** field of the **DB/User** tab (Figure 4.7).
4. Type “**Main Girder**” in the **Name** field.
5. Select “**Box**” in the **Section Shape** selection field (Figure 4.7 ①).
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 .
12. Repeat steps 3 to 11 for sections 2 to 5.
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 4.7 ①).
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 .
19. Click .
20. Click the unit system selection button of the Status Bar and change “mm” to “m”.

① 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.

① Convert the unit system from “mm” to “m” for structural modeling.

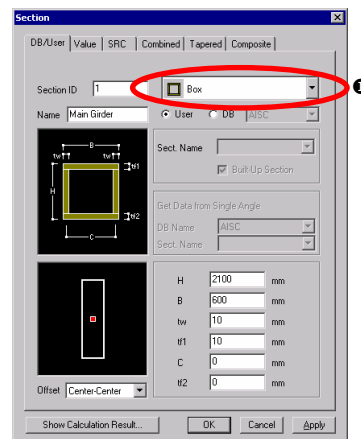


Figure 4.7 Section Data Dialog Box

Structural Modeling Using Nodes and Elements

Generate the Arch Ribs

Use *Structure Wizard* to generate the arch ribs (Figure 4.8).

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).

1. Select **Geometry>Structure Wizard>Arch** in the **Menu** tab of Tree 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**.

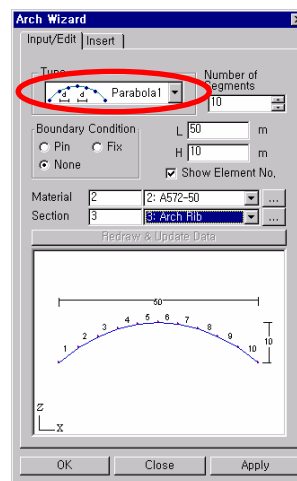






Figure 4.8 Concept of Parabola1 format and Arch Wizard Dialog Box


Generate the Hangers








Use  **Extrude Elements** to generate the hangers. Extend the nodes generated on the arch rib by projecting them perpendicularly downward (Figure 4.9).




 **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).

 **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).

1. Click  **Extrude Elements** in the **Element** toolbar (Figure 4.9①).
2. Click  **Node Number** (Toggle on).
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 nodes **1** to **11**.
11. Select “**Normal**” in the **Direction** selection field.
12. Click .
13. Click  **Change Element Parameters** (Figure 4.9②).
14. Click  **Select Recent Entities** (Figure 4.9③).
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 .

Click **Shrink** (Figure 4.10①) and  **Hidden** (Figure 4.10②) (Toggle on) to check the entered **Beta Angle**. Check the current data entries and click  **Shrink** and  **Hidden** to toggle off.

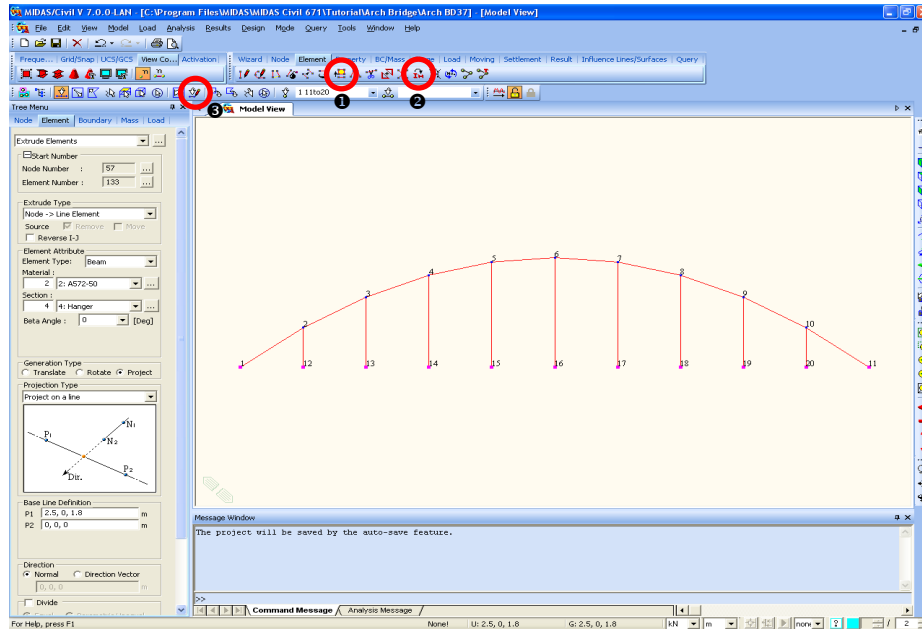


Figure 4.9 Hanger Generation

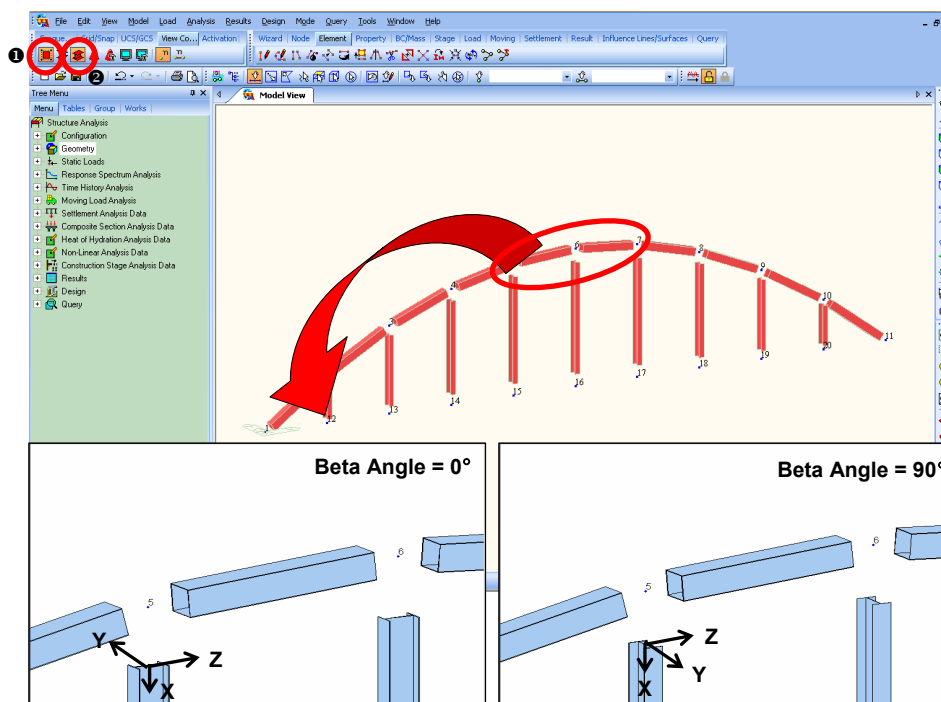













Figure 4.10 Modification of the Beta Angle for Hangers

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 the *Element* toolbar.
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** in the *Element* toolbar.
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  (Figure 4.11).

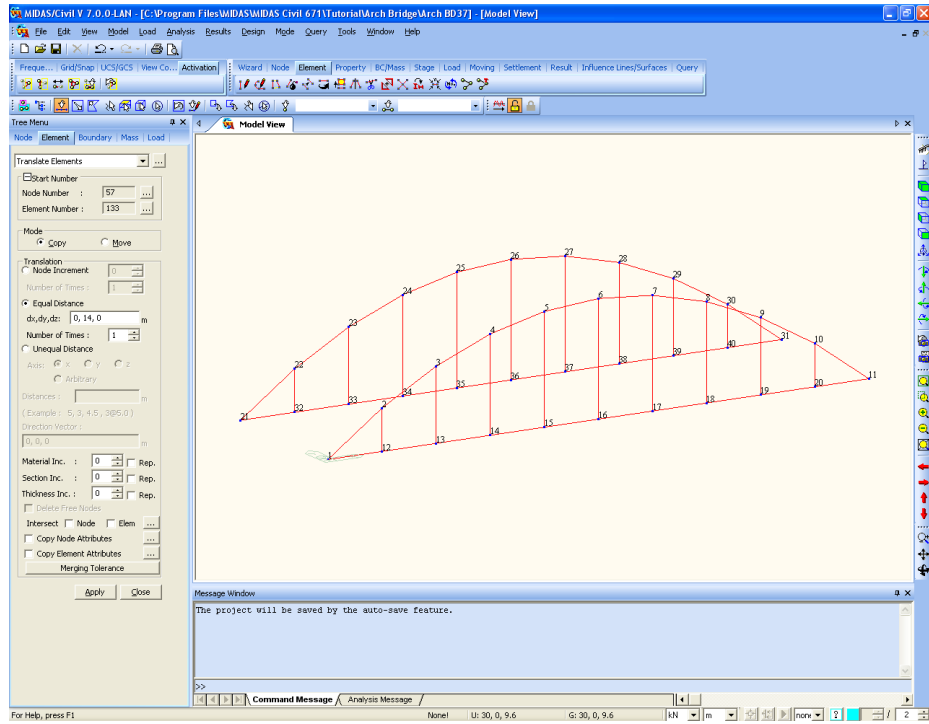





Figure 4.11 Completed Main Girders and Arches

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** in the **Element** toolbar.
2. Click  **Select Polygon** (Figure 4.12 ①) 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.
11. Click .

① The **Direction** represents the Direction of Projection.

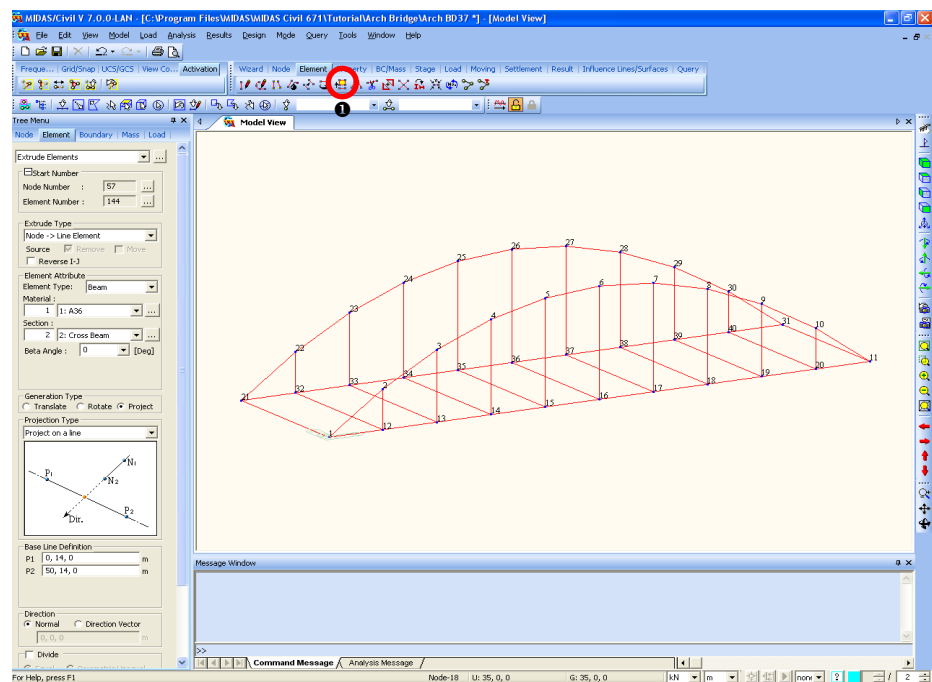








Figure 4.12. Creation of Cross Beams

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**.
2. Click  **Activate**.
3. Click  **Element Number** (Toggle on).
4. Click  **Create Elements** in the **Element** toolbar.
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 4.13①).
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) (Figure 4.13).

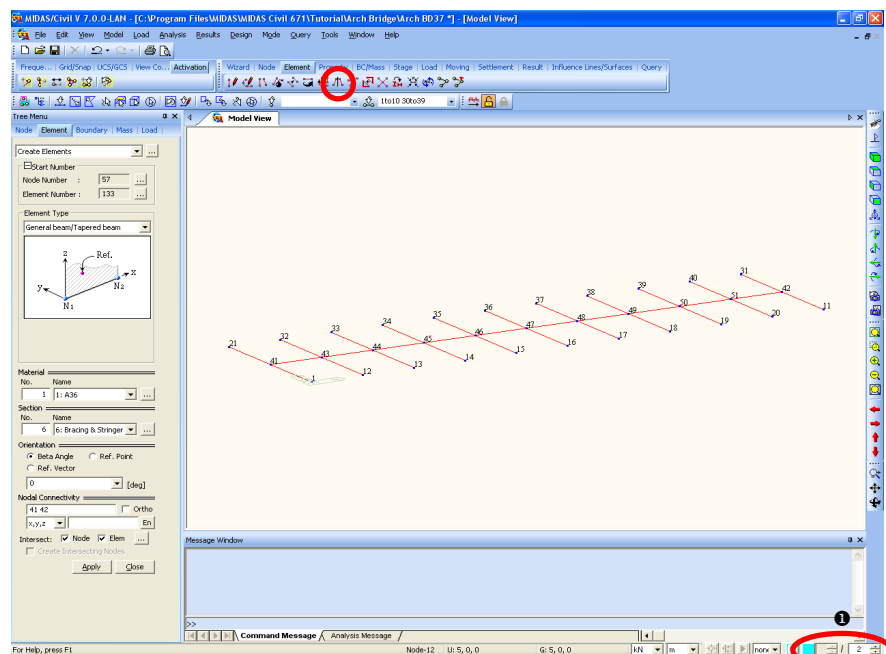










Figure 4.13 Completed Stringers

Generate the diagonal bracings on the floor plane.

1. Click  **Create Elements** in the *Element* toolbar.
 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. Click  **Translate Elements**.
 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 .
 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  (Figure 4.14).
-

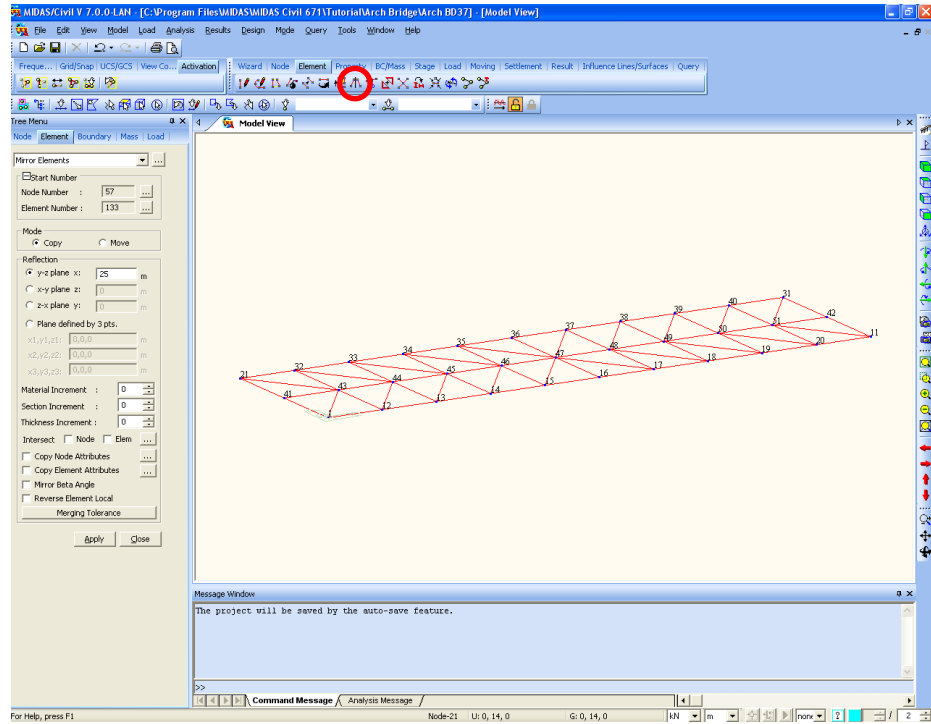






Figure 4.14 Completed Floor Plane

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 Active** (Figure 4.15 .
2. Click  **Create Elements** in the **Element** toolbar.
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 4.15).

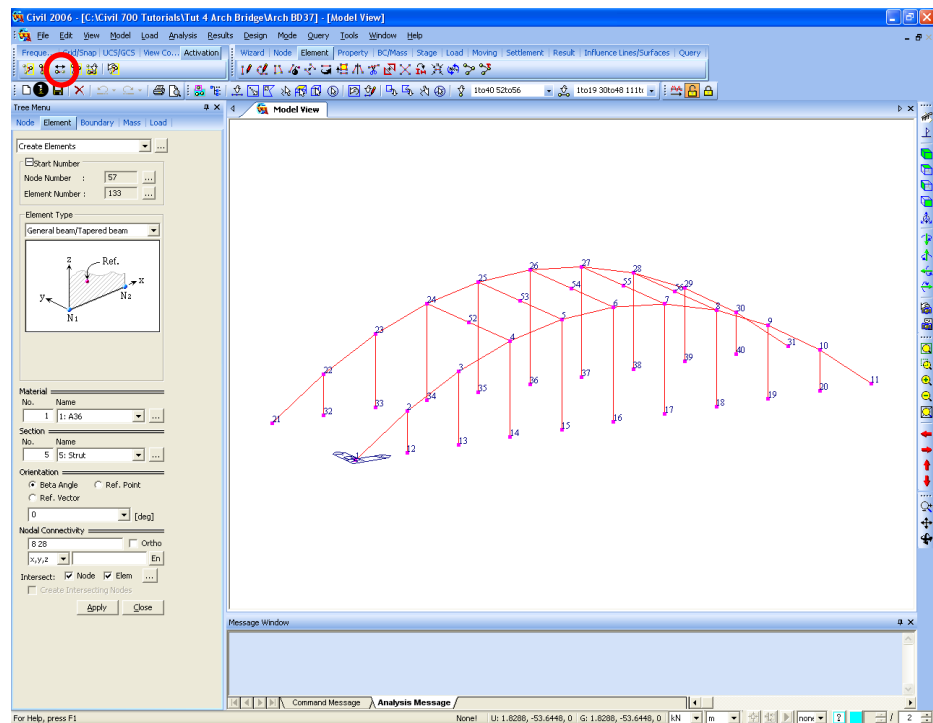







Figure 4.15 Completed Struts

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 5 struts.
2. Click  **Activate**.
3. Click  **Element Number** (Toggle on).
4. Click  **Create Elements** in the **Element** toolbar.
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 4.16).

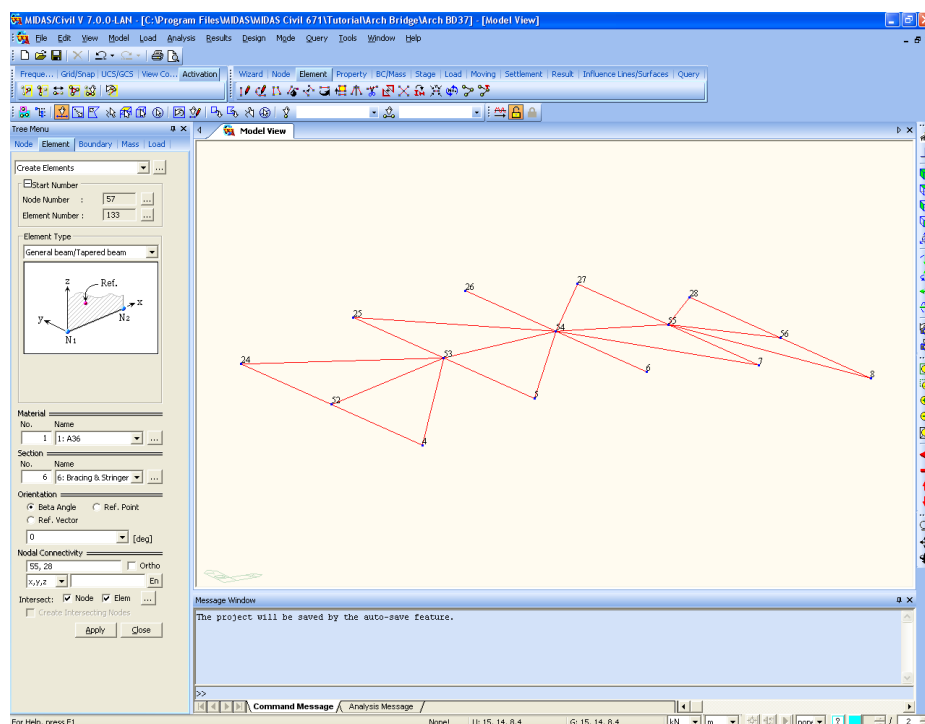








Figure 4.16 Completed Arch Bracings

Enter Structure Boundary Conditions

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

1. Click  **Activate All**.
2. Select **Boundary** tab as shown in Figure 4.17, 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 .
7. Select node **11** and check (✓) “**Dy** and **Dz**”.
8. Click .
9. Select node **21** and check (✓) “**Dx** and **Dz**”.
10. Click .
11. Select node **31** and check (✓) “**Dz**”.
12. Click .

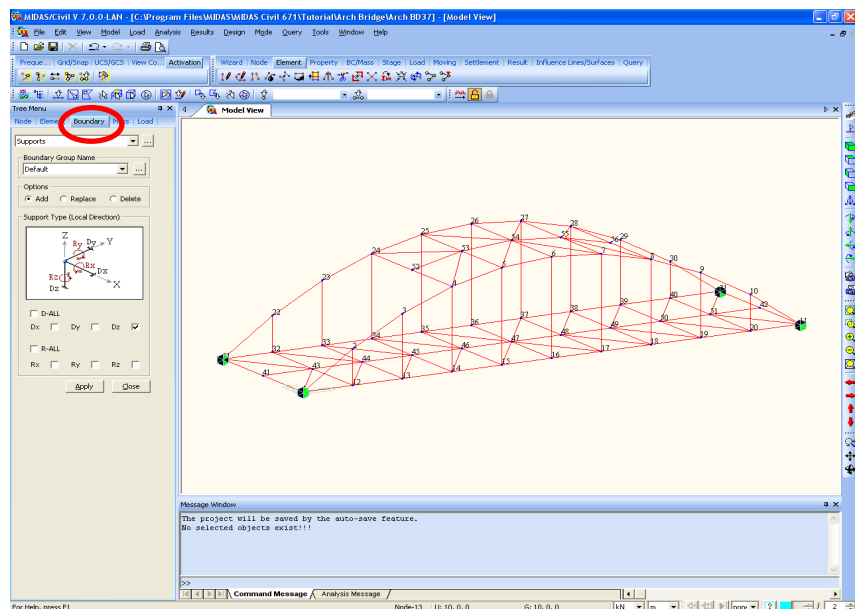



Figure 4.17 Structural Boundary Conditions












Boundary Conditions for Beam End Connections


Use **Beam End Release** to specify the boundary conditions at both ends of the beam elements (Figure 4.18).




- 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

🔊 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.

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 4.18🔊) 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 .
7. Click the *Filter* selection field (Figure 4.18🔊) to select “**none**”.🔊
8. Click  **Select Identity-Elements** (Figure 4.18🔊).
9. Select “**Section**” in the *Select Type* field.
10. Select “**6 : Bracing & Stringer**” in the *Section* selection field.
11. Click .
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 .
14. Select “**2 : Cross beam**” in the *Section* selection field of the *Select Identity-Elements* dialog box.
15. Click .
16. Click  in the *Select Identity-Elements* dialog box.
17. Click  **Activate**.
18. Click  **Element Number** (Toggle on).
19. Click  **Select Intersect** to select elements **59** to **69**.🔊
20. Click the *Pinned-Fixed* button in the *General Types and Partial Fixity* field.
21. Click .

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

22. Type “80 to 90” in the element selection window (Figure 4.18②) 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).
26. Click  **Activate All**.
27. Click  **Node Number** (Toggle off)

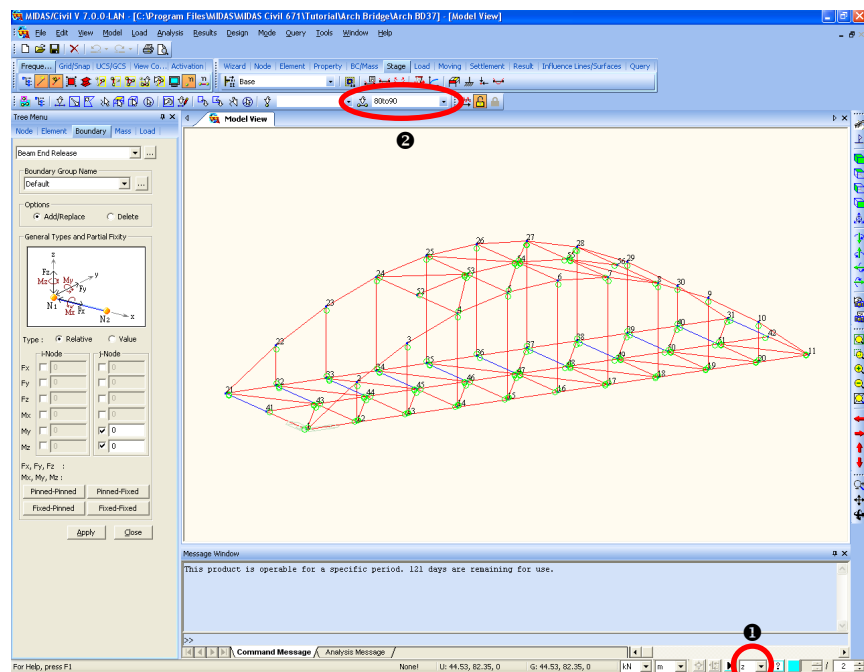












Figure 4.18 Beam End Release

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 .
 5. Click  in the *Select Identity-Elements* dialog box.
 6. Click  **Group** of Tree Menu.
 7. Click  **Activate**.
 8. Click  **Element Number** (Toggle on).
 9. Right-click the mouse in the *Structure Group* and then select *New* to enter “**Cross Beam 1**”.
 10. Follow the same procedure to enter “**Cross Beam 2**”.
 11. Click  **Select Intersect** to select elements **59** to **69**.
 12. From the *Structure Group* drag “**Cross Beam 1**” with the mouse and drop to the model window.
 13. Type “**80 to 90**” in the element selection window (Figure 4.19①) and press [Enter].
 14. Follow the same procedure to assign “**Cross Beam 2**”.
 15. Click  **Element Number** (Toggle off).
 16. Click  **Activate All** and  **Iso View**.
-

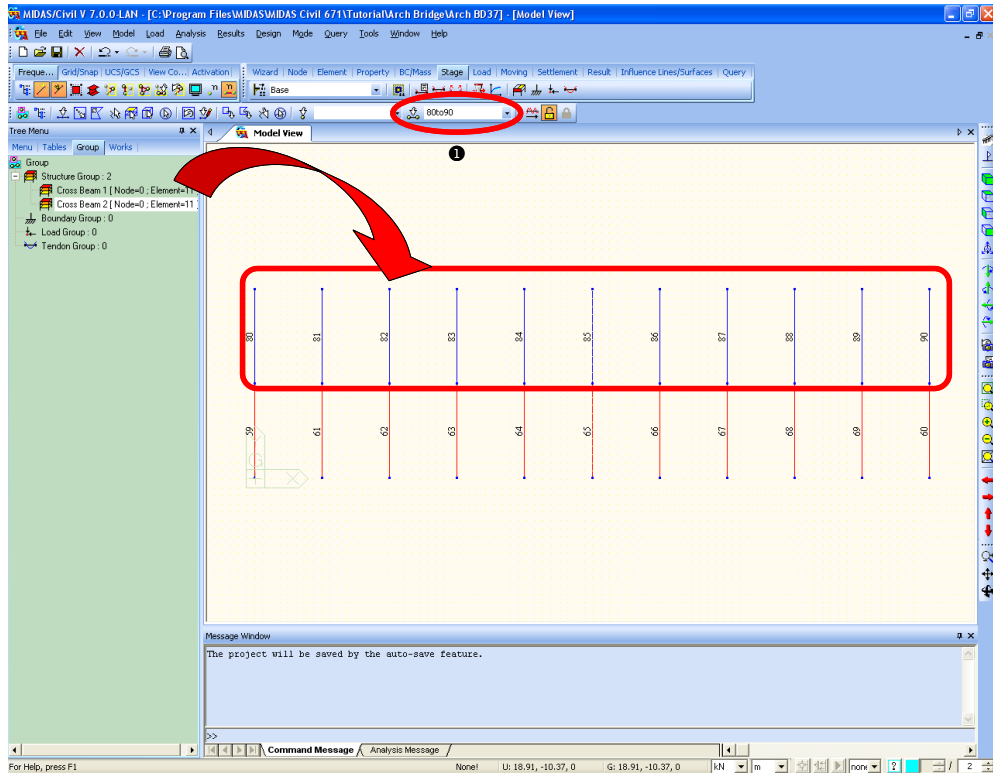


Figure 4.19 Cross Beam Group

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 4.20).
3. Select “**Dead Load**” in the *Type* selection field.
4. Click .
5. Enter “**Sidewalk Load**” in the *Name* field.
6. Select “**Dead Load**” in the *Type* selection field.
7. Click .
8. Click .

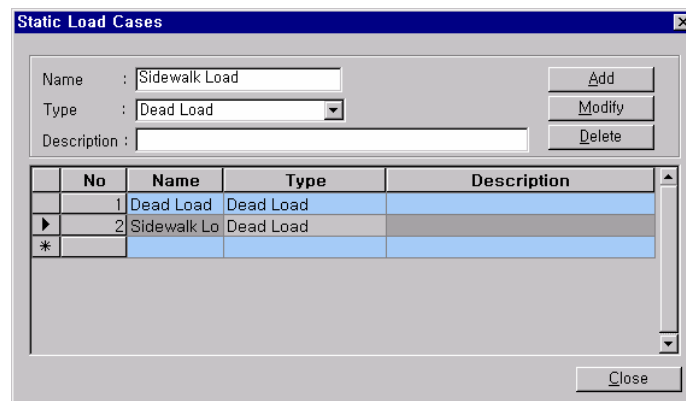









Figure 4.20 Static Load Cases Window

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 4.21).

-
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 .
 5. Click  in the *Select Identity-Elements* dialog box.
 6. Select **Load>Element Beam Loads** 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 .
 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 .
 24. Click .
-

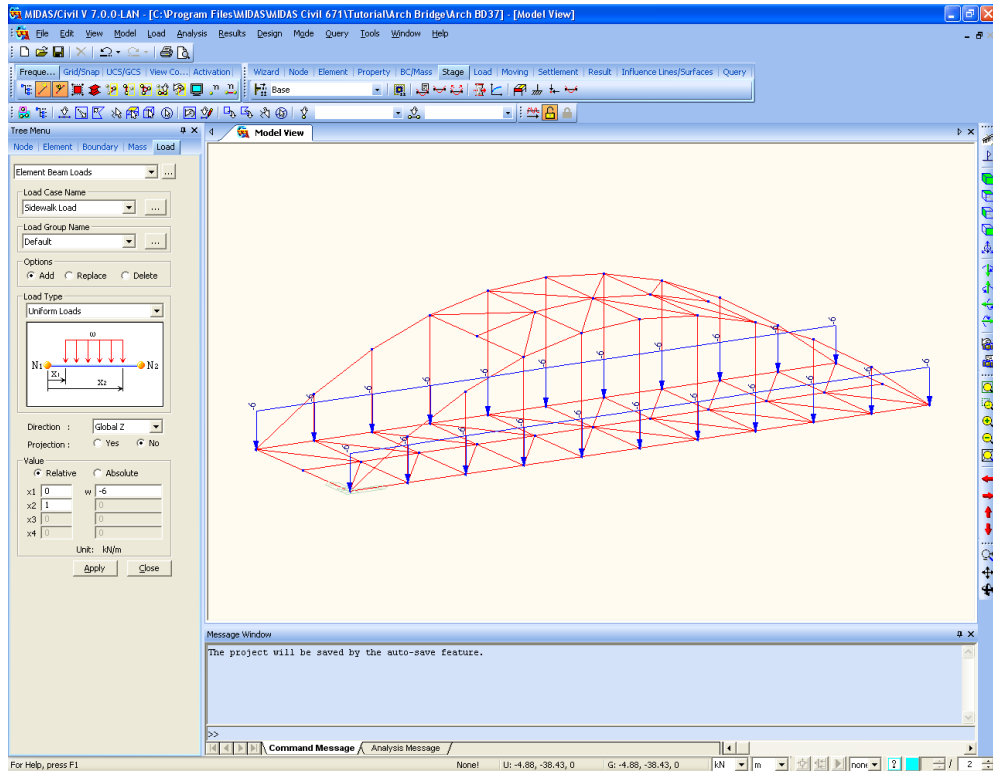








Figure 4.21 Current Loading Condition





Define Moving Traffic Loads

First, define the traffic line lanes (Figure 4.22).

🔊 Line lane width is defined as per Section 3.2.9 of BD 37/01. Number of lanes in this model is 4 and overall width is 14 m. The line lane width is calculated as $14 \div 4 = 3.5$ m. Central reserve, verge, hard strip, footway, etc., shown in Figure 1 of BD 37/01, are neglected. MIDAS/Civil uses this line lane width for calculating the HA lane factor as per Table 14 of BD 37/01, and then automatically incorporates the lane factor into the results from moving load analysis.

🔊 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)

1. Select **Moving Load Analysis>Moving Load Code** in the **Menu** tab of the Tree Menu.
2. Select "**BS**" in the **Select Moving Load Code** dialog box.
3. Click .
4. Select **Moving Load Analysis>Traffic Line Lanes** in the **Menu** tab of the Tree Menu.
5. Click  in the **Traffic Line Lanes** dialog box.
6. Enter "**Lane 1**" in the **Lane Name** field.
7. Enter "**-1.75**" in the **Eccentricity** field.
8. Enter "**1**" in the **Wheel Spacing** field.
9. Select "**Cross Beam**" in the **Vehicle Load Distribution** field.
10. Select "**Cross Beam 1**" in the **Cross Beam Group** field.
11. Enter "**3.5**" in the **Lane Width** field. 🔊
12. Select "**2 Points**" among **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**. 🔊
13. Click .
14. Click  in the **Traffic Line Lanes** dialog box.
15. Enter "**Lane 2**" in the **Lane Name** field.
16. Enter "**-5.25**" in the **Eccentricity** field.
17. Enter "**1**" in the **Wheel Spacing** field.
18. Select "**Cross Beam**" in the **Vehicle Load Distribution** field.
19. Select "**Cross Beam 1**" in the **Cross Beam Group** field.
20. Enter "**3.5**" in the **Lane Width** field.
21. Select "**2 Points**" among **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**.
22. Click .
23. Click  in the **Traffic Line Lanes** dialog box.

24. Enter "**Lane 3**" in the *Lane Name* field.
 25. Enter "**-8.75**" in the *Eccentricity* field.
 26. Enter "**1**" in the *Wheel Spacing* field.
 27. Select "**Cross Beam**" in the *Vehicle Load Distribution* field.
 28. Select "**Cross Beam 2**" in the *Cross Beam Group* field.
 29. Enter "**3.5**" in the *Lane Width* field.
 30. Select "**2 Points**" among *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**.
 31. Click .
 32. Click  in the *Traffic Line Lanes* dialog box.
 33. Enter "**Lane 4**" in the *Lane Name* field.
 34. Enter "**-12.25**" in the *Eccentricity* field.
 35. Enter "**1**" in the *Wheel Spacing* field.
 36. Select "**Cross Beam**" in the *Vehicle Load Distribution* field.
 37. Select "**Cross Beam 2**" in the *Cross Beam Group* field.
 38. Enter "**3.5**" in the *Lane Width* field.
 39. Select "**2 Points**" among *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**.
 40. Click .
 41. Click .
-

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

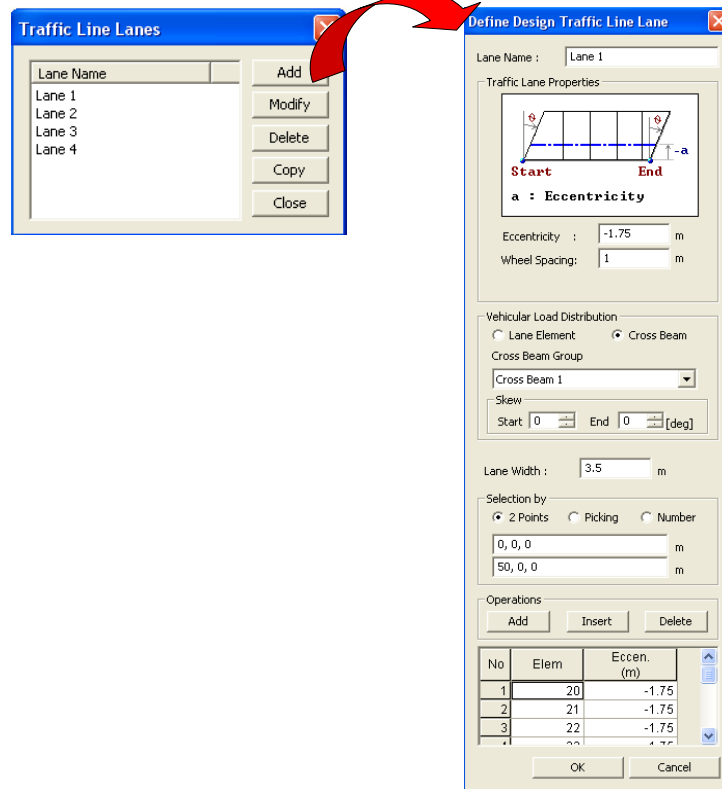


Figure 4.22 Traffic Line Lanes Dialog Box

The method for defining the moving traffic loads, HA and HB (Figure 4.23), is explained below. ⚙

1. Select **Moving Load Analysis > Vehicles** in the **Menu** tab of the Tree Menu.
2. Click **Add Standard** in the **Vehicles Load Type** dialog box.
3. Select “**BS BD 37/01 Standard Load**” in the **Standard Name** field.
4. Confirm “**HA & HB (BD37/01)**” in the **Vehicle Load Name** and **Vehicle Load Type** fields.
5. Click **OK**.
6. 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)

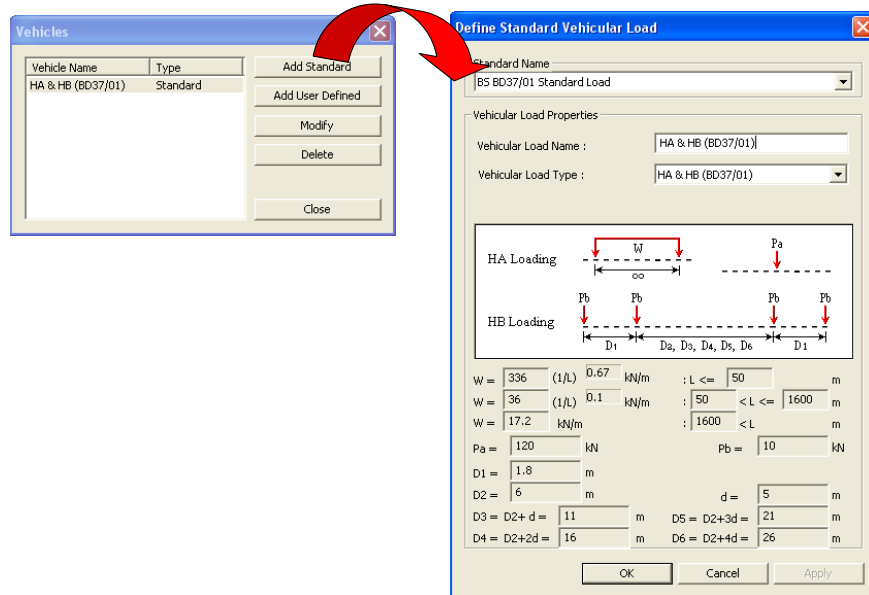



















Figure 4.23 Definition of Standard Vehicle Loads

🔊 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 4.24). 🔊

1. Select **Moving Load Analysis>Moving Load Cases** in the Tree 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 (BD37/01)"** 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 (BD37/01)"** 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  in the *Sub-Load Cases* dialog box.
 27. Click  in the *Define Moving Load Case* dialog box.
 28. Click  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  in the *Sub-Load Cases* field.
 33. Select “**HA & HB (BD37/01)**” 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  in the *Sub-Load Cases* dialog box.
 40. Click  in the *Define Moving Load Case* dialog box.
 41. Click  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  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  in the *Sub-Load Cases* dialog box.
 53. Click  in the *Define Moving Load Case* dialog box.
 54. Click .
-

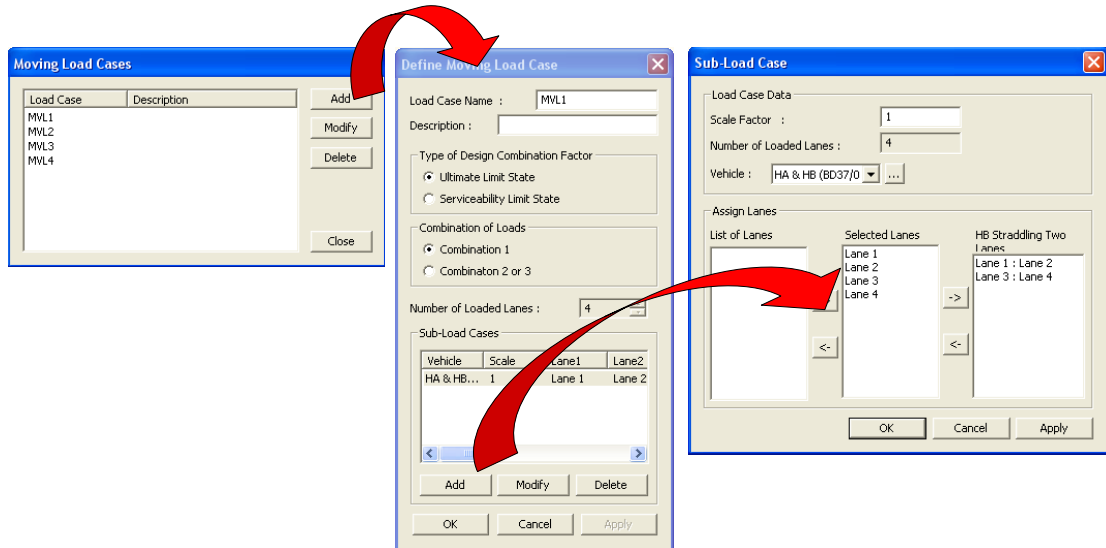


Figure 4.24 Definition of Moving Vehicle Load

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 4.25).

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>Moving Load Analysis Control** from the Main Menu.
2. Enter “5” in the **Influence Generating Point No./Line Element** field.
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).

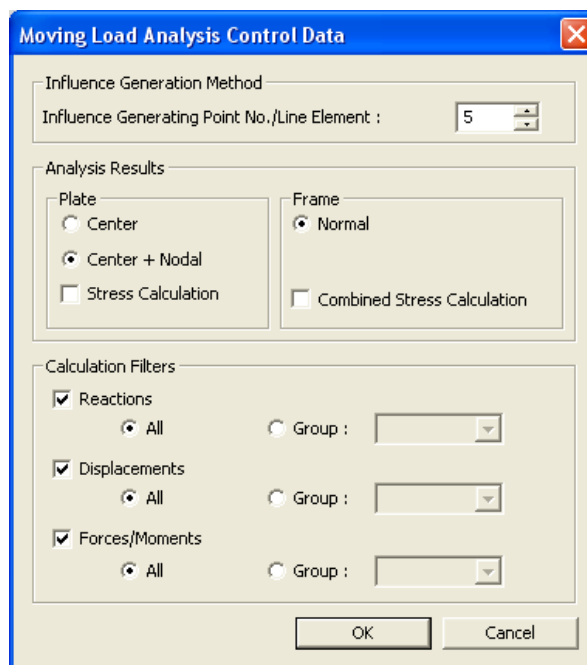



Figure 4.25 Moving Load Analysis Control Dialog Box

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

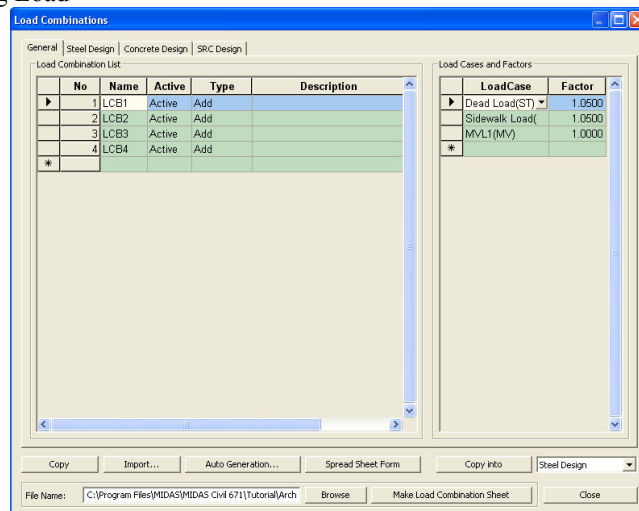


Figure 4.26 Load Combinations Dialog Box





Use **Results>Combinations** in the Main Menu to open the **Load Combinations** dialog box (Figure 4.26) and enter the following load combination:

☞ 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)

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







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

1. Select **Results>Combinations** in the Main Menu.
2. Select **“Active”** under **Active** in **Load Combination List**.[☞]
3. Enter **“LCB1”** in the **Name** field.
4. Select **“Add”** in the **Type** selection field.
5. Click the **LoadCase** 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.
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)”**.[☞]
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 4.27):

-
1. Click  **Deformed Shape** on the **Result** tab in toolbar (Figure 4.27①).
 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 .
 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. Check (✓) in **Apply upon OK**.
 12. Click .
 13. Click the unit system selection button of the Status Bar and change “**m**” to “**mm**”.
 14. Click  **Hidden** (Toggle on).
-

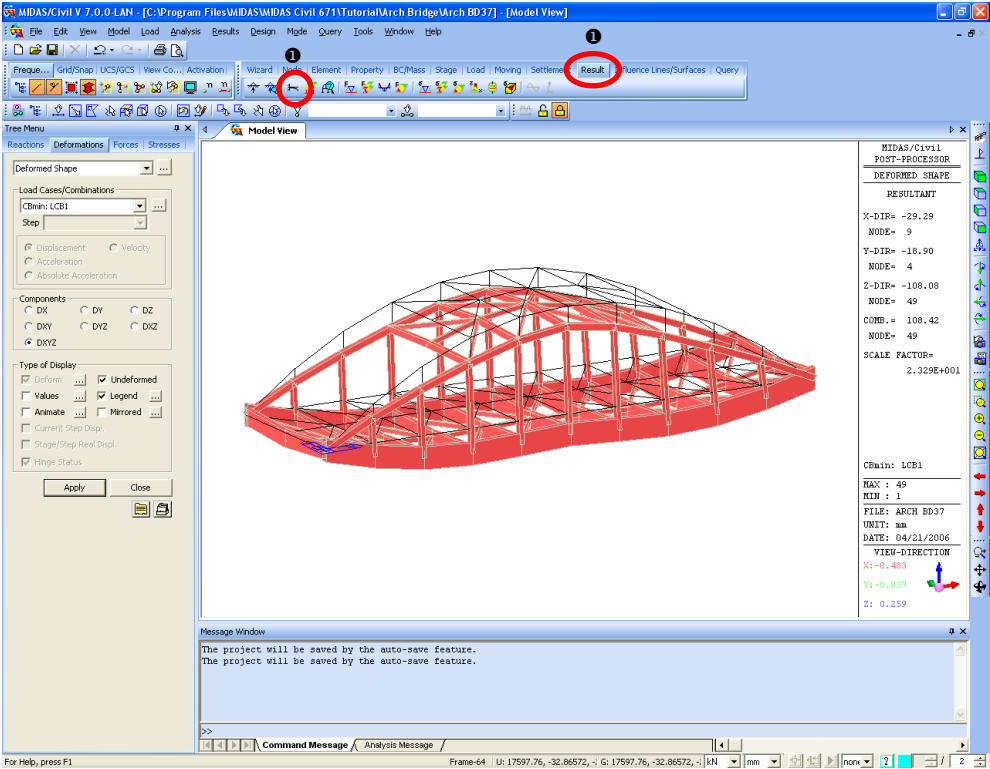


Figure 4.27 Deformed Shape







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 4.28).

☞ 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.

☞ **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.

-
1. Click  **Select by Plane**.
 2. Select “**XZ Plane**” in the *Plane* tab.
 3. Click in the *Z Position* field and select a point with the mouse, which defines the desired X-Z plane (the color of the selected plane changes).
 4. Click .
 5. Click the unit system selection button of the Status Bar and change “**mm**” to “**m**”.
 6. Click  **Activate**.
 7. Click  **Front View**.
 8. Select  **Beam Diagram** on the *Result* tab in toolbar (Figure 4.28).^①
 9. Select “**MVall: MVL1**” in *Load Cases/Combinations* selection field.^②
 10. Select “**My**” in the *Components* selection field.
 11. Select “**5 Points**” and “**Line Fill**” in *Display Options* selection field.
 12. Enter “**1.0**” in the *Scale* field.
 13. Check (✓) “**Legend**” in the *Type of Display* selection field.
 14. Click .
-

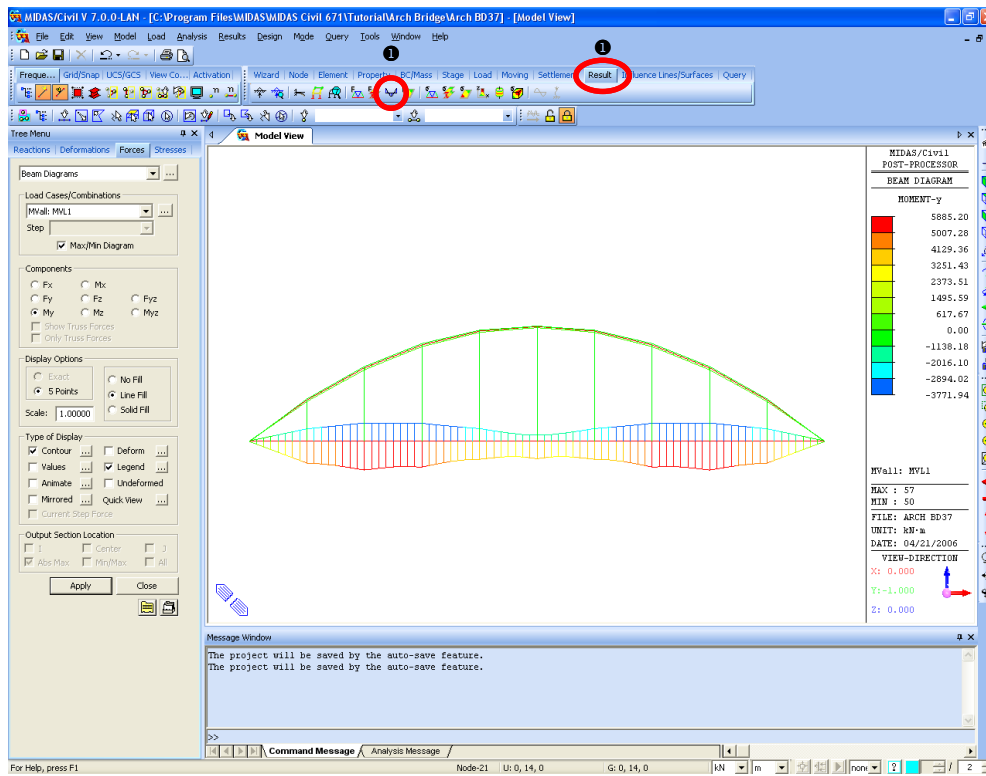







Figure 4.28 My diagrams (XZ Plane) for the Beam

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 4.29 shows the results for support B1 (node 1).

1. Click  **Hidden** (Toggle off).
2. Click  **Activate All**
3. Click  **Reactions** on the **Influence Lines/Surface** tab in the toolbar (Figure 4.29).
4. Select "**Lane 1**" in the **Line/Surface Lanes** field.
5. Enter "**1**" in the **Key Node** field.
6. Enter "**1.0**" in the **Scale Factor** field.
7. Select "**FZ**" in the **Components** field.
8. Check (✓) "**Legend**" in the **Type of Display** field.
9. Click .
10. Click  **Iso View**.

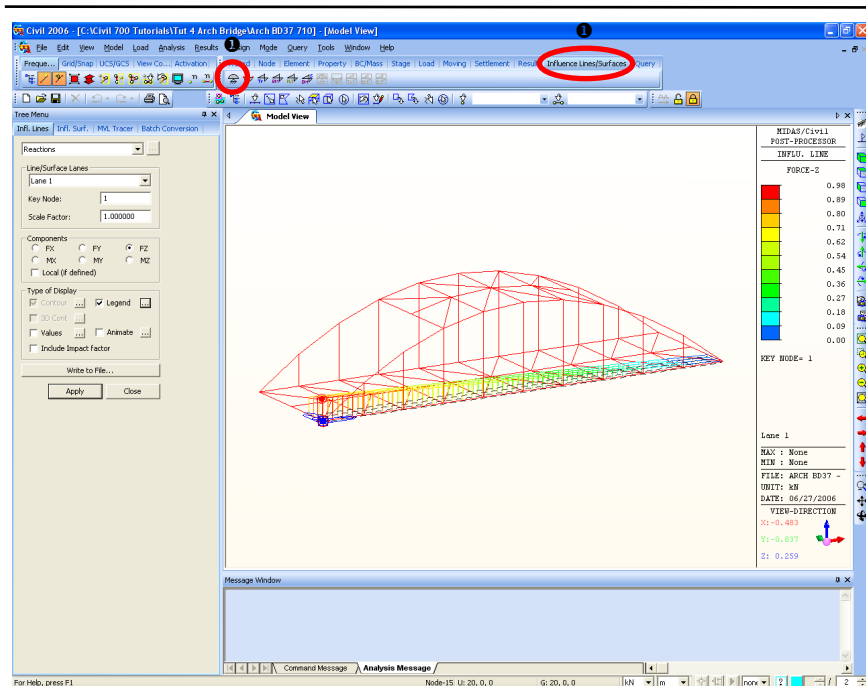


Figure 4.29 Reaction Influence Line

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

After reviewing the animation, click Close to restore the original screen (Figure 4.30).

1. Check (✓) “**Legend**” and “**Animate**” in the *Type of Display* field.
2. Click **Apply**.
3. Click **Record** (Figure 4.30 1).

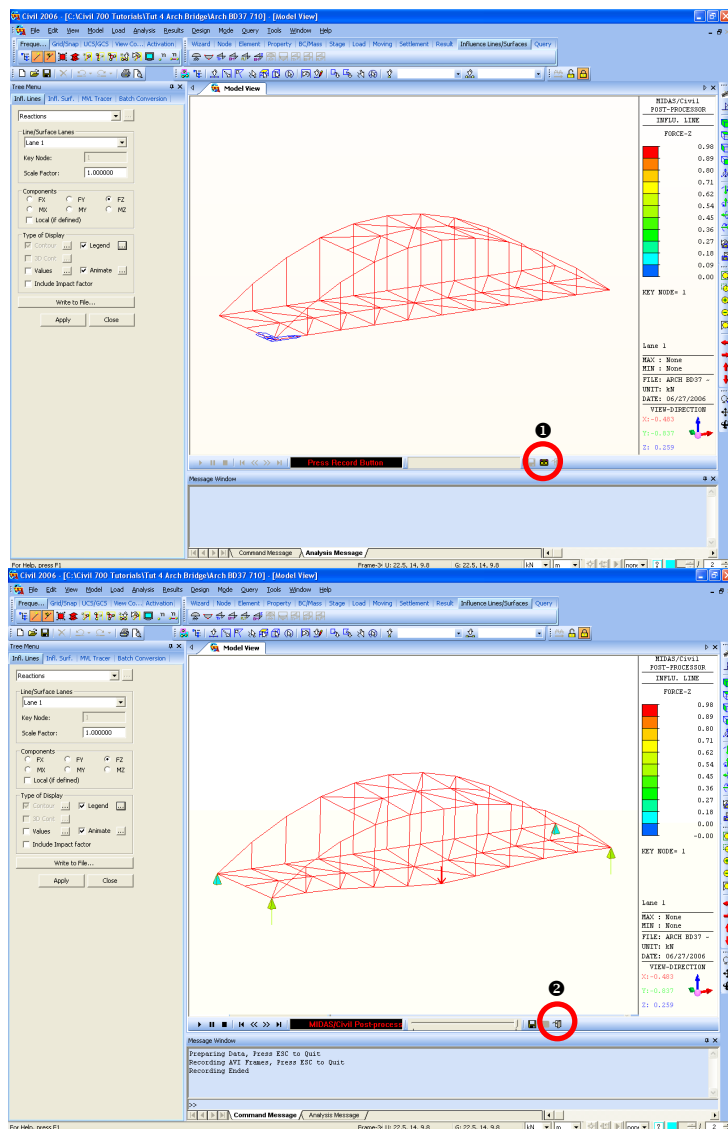







Figure 4.30 Animation of Support Reaction Influence Line

Figure 4.31 shows the deflection influence line.

1. Click the unit system selection button of the Status Bar and change “m” to “mm”.
2. Click  **Displacements** on the *Influence Lines/Surfaces* tab in the toolbar (Figure 4.31 ①).
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 .

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

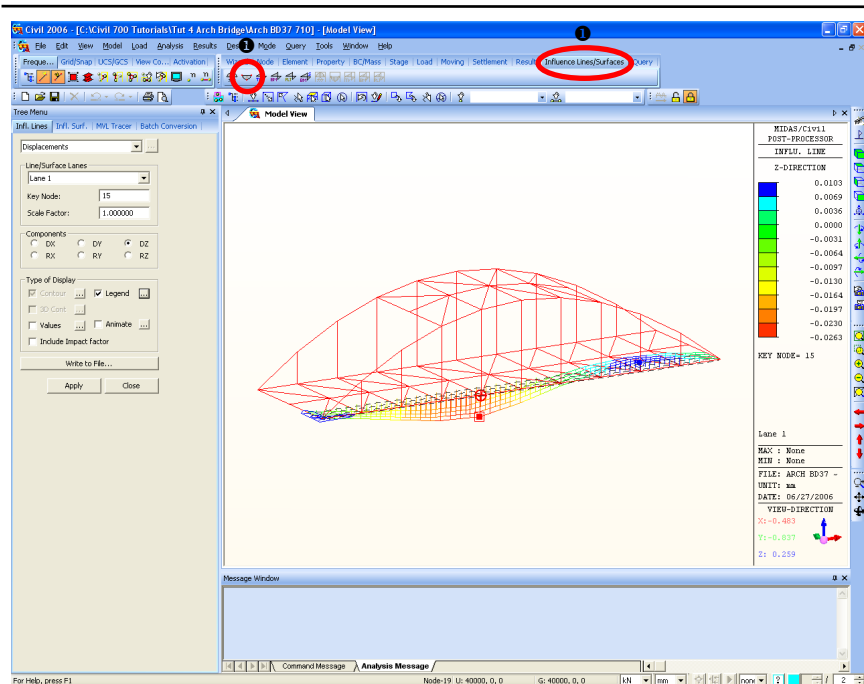




Figure 4.31 Deflection Influence Line

Figure 4.32 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** on the *Influence Lines/Surfaces* tab in the toolbar (Figure 4.32 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 .

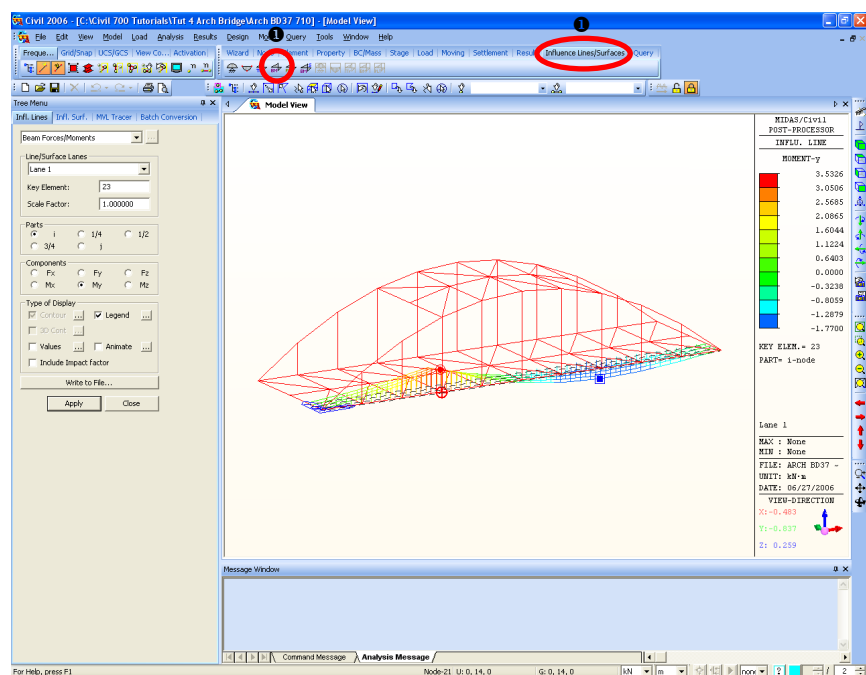


Figure 4.32 Moment Influence Line

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 4.33).

1. Select **Iso 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 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**.

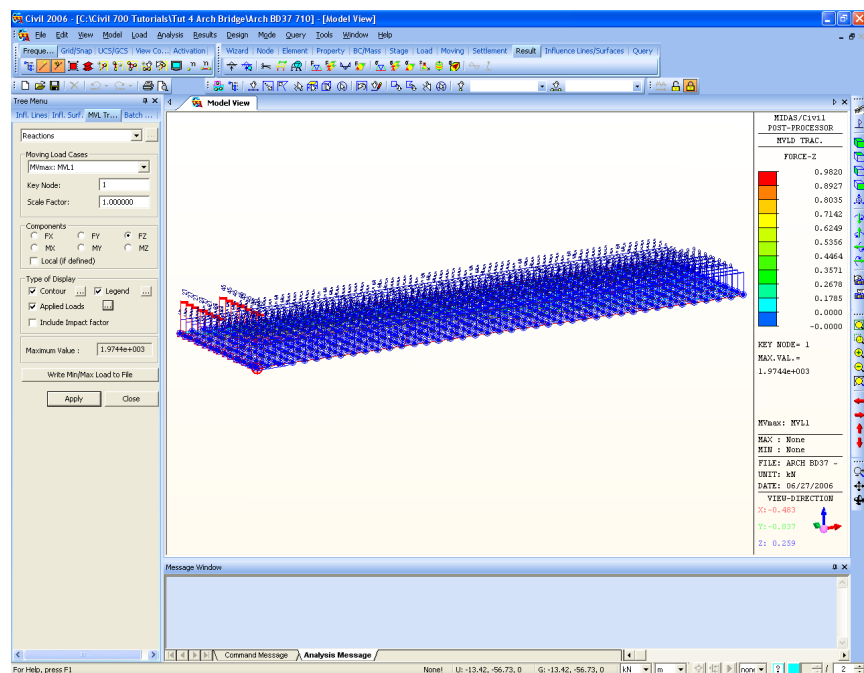


Figure 4.33 Checking the Loading points of a Vehicle using Moving Load Tracer

🔊 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 1th 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**.

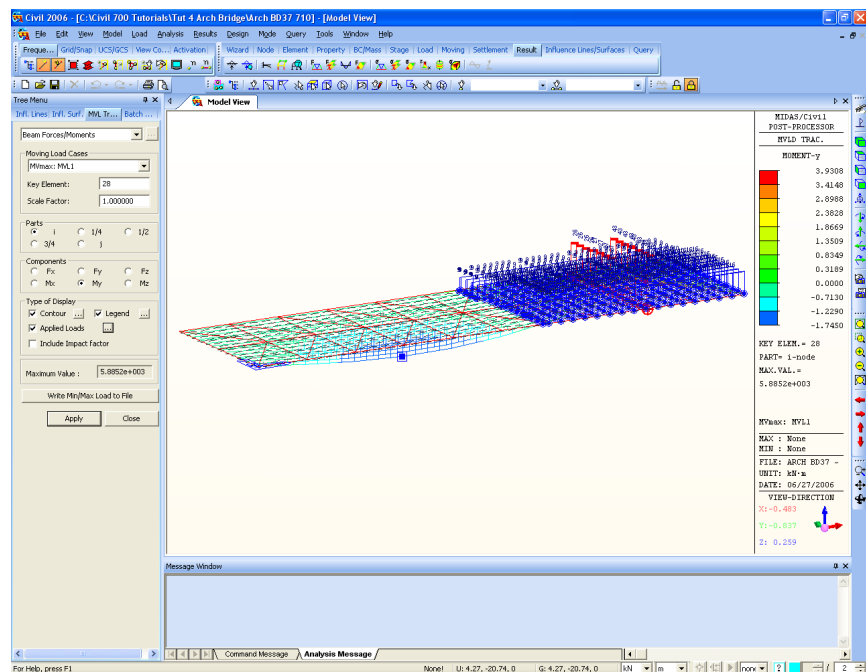




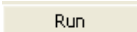
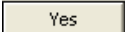





Figure 4.34 Checking the Loading points of a Vehicle using Moving Load Tracer

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  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 .
 2. Click  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  in the *MCT Command Shell* dialog box.
 6. Click  when prompted for “*Analysis/design results will be deleted; Continue?*” in the *CVLw* dialog box.
 7. Click  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  in the *Static Load Cases* dialog box.
 11. Click  **Analysis**.
-

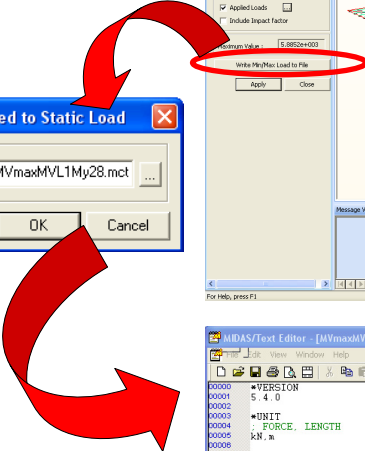



Figure 4.35 *Live load automatically converted into static load*

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 i^{th} end of element 28.

1. Select  **Beam Diagram** on the **Result** tab in toolbar (Figure 4.36 ❶).
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**.

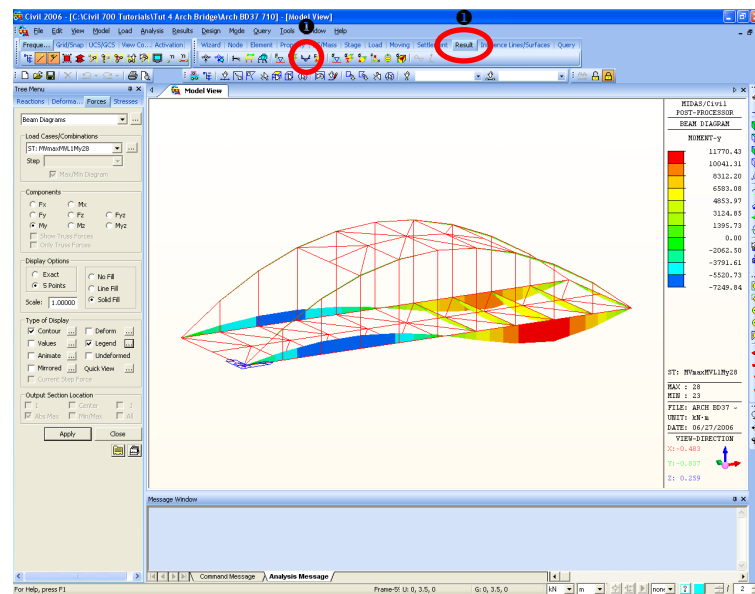


Figure 4.36 My beam diagrams for the live load converted into static load