Tutorial 2

PLANT STRUCTURE

Civil

TUTORIAL 2. PLANT STRUCTURE

TUTORIAL 2.
PLANT STRUCTURE

Summary

This Tutorial presents an efficient method of modeling and analyzing a plant structure.

The fundamentals and the essential functions of **midas Civil** are covered in "Tutorial 1".

Some of the functions introduced in "Tutorial 1" will be revisited in 'Tutorial 2". Additional functions not covered in "Tutorial 1" will be introduced in "Tutorial 2"

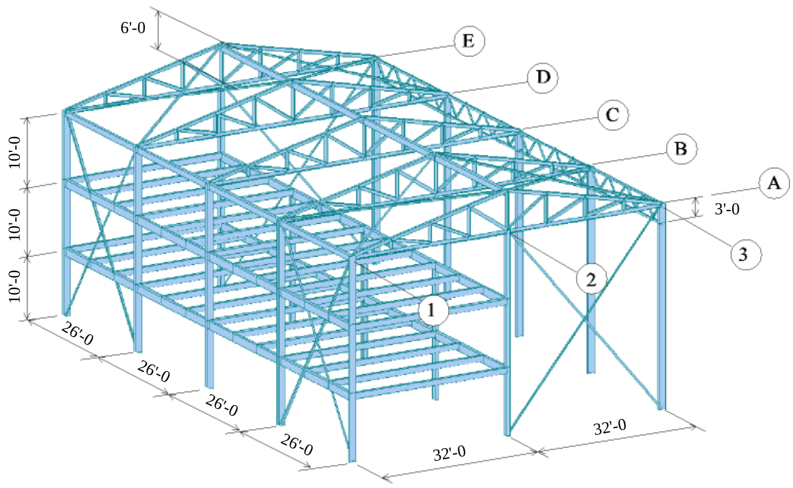
The step-by-step modeling and analysis processes presented in this example are the following:

1. File Opening and Preferences Setting
2. Enter Material and Section Properties
3. Structure Modeling using Nodes and Elements
4. Enter Structure Support conditions
5. Enter Loading Data
6. Confirm the status of model data input prepared using Works Tree
7. Perform Structural Analysis
8. Verify and Interpret Analysis Results

Analysis Model and Load Cases

The geometric shape, boundary conditions and members for the plant structure are shown in Fig.2.1. The load cases commonly encountered in real structures are considered for the model. The load combinations are generated in the post-processing stage.

- Load Case 1 – Self Weight
- Load Case 2 – Dead loads on the floors and the inclined roof
  0.1 ksf on the floors
  0.015 ksf on the inclined roof
- Load Case 3 – Live load, 0.1 ksf on the floors
- Load Case 4 – Live load, 0.02 ksf on the inclined roof
- Load Case 5 – Uniformly distributed load, 0.1 k/f on every member in the X-direction
- Load Case 6 – Concentrated load, 5 kips on every node in the Y-direction



Material Type A36

Section Type

1: W 16 × 67 : exterior columns, roof level girders
2: W 14 × 90 : interior columns
3: W 18 × 65 : floor girders / beams
4: W 4 × 13 : top-bottom chords of exterior trusses
5: WT 7 × 24 : top-bottom chords of interior trusses
6: WT 4 × 9 : vertical and diagonal members of roof trusses
7: L 4 × 4 × 5/16 : wind braces

Figure 2.1 Analysis Model

In this example, an identical section number has been attributed to the exterior columns and roof girders for simplicity. However, in practice, for efficiency in structural analysis and design, it is advisable to assign separate section numbers to members with different structural characteristics such as loadings and boundary conditions even if the sections are the same.

File Opening and Preferences Setting

Select ***File>New Project*** (or) to begin modeling and select ***File>Save*** (or) to save the work in the given file name.

Unit System

In this example, "ft" and "kip" are the basic units.

1. Select ***Tools>Unit System*** in the Main Menu.
2. Select "**ft**" in the ***Length*** selection field.
3. Select "**kips(kips/g)**" in the ***Force*** (***Mass***) selection field.
4. Click OK.

Grid Setup

Grid displays reference points or lines in the working window to help the user readily enter the nodes or elements.

To execute the grid function in **midas Civil**, use ***View>Grids>Point Grid*** and ***Line Grid***. If ***Grid*** and ***Snap*** are used in combination, the mouse cursor will automatically snap to the closest grid.

Point Grid and ***Line Grid*** can be used separately or jointly. In this case, ***Point Grid*** is used.

1. Click ***Define Point Grid*** in ***Main Menu > Structure > Grids***
2. Enter "**2, 2**" in the ***dx, dy*** field.
3. Click OK.

Model Boundary defines the Grid display boundary.

User Coordinate System (UCS) Setup

The conversion of the GCS X-Z plane into the UCS x-y plane is reviewed in "Tutorial 1".

Similarly, for this example, the GCS X-Z plane is assigned as the UCS x-y plane to model the columns and the roof truss on grid Ⓐ of the structure (see Fig.2.1).

1. Click ***X-Z*** in ***the Main Menu*** > ***UCS*** > ***X-Z***.
2. Confirm "**0, 0, 0**" in the ***Origin*** field.
3. Confirm "**0**" in the ***Angle*** field.
4. Check (✓) "**Change View Direction**".
5. Click OK

If you click OK after checking (✓) Change View Direction, View Point changes automatically so that the working window corresponds to the UCS plane.

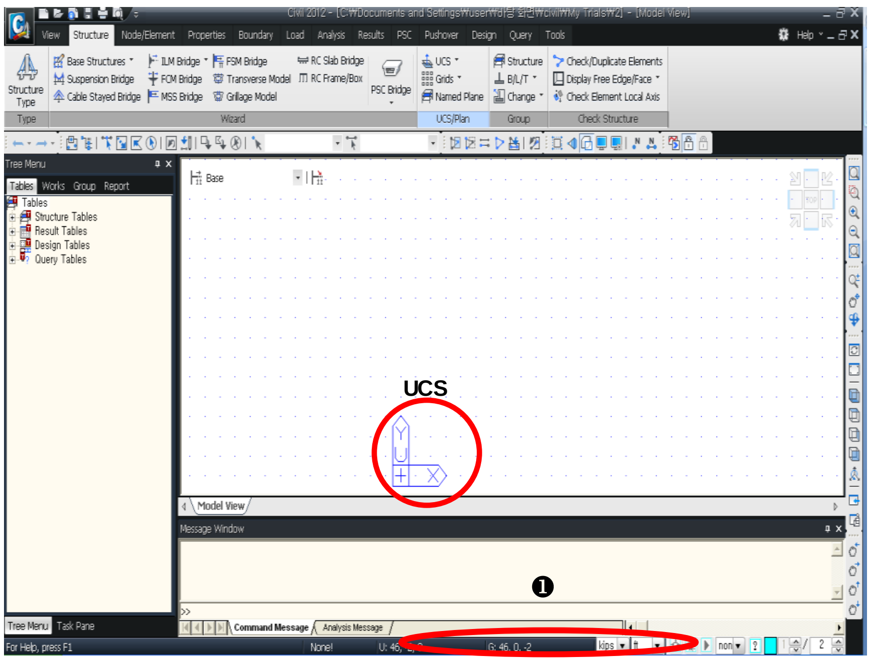


Figure 2.2 UCS Setup

Enter Material and Section Properties

Material properties and section data for members may be entered anytime, regardless of the generation of elements. However, it is most convenient to define such data prior to generating elements. Member sizes and shapes can then be viewed by the ***Hidden*** function as the structure is modeled.

Select ***Properties>Material Properties*** in the ***Main Menu*** to enter the material properties and section data.

The material properties and section data of the structural members are as follows:

- Material type A36

- Section type
  1: W 16 × 67 : exterior columns, roof level girders
  2: W 14 × 90 : interior columns
  3: W 18 × 65 : floor girders/beams
  4: W 4 × 13 : top and bottom chords of exterior trusses
  5: WT 7 × 24 : top and bottom chords of interior trusses
  6: WT 4 × 9 : vertical and diagonal members of roof trusses
  7: L 4 × 4 × 5/16 : wind braces

In the material and section types, the part in front of ":" represents the inherent member numbering, and the part after ":" represents the member designation.

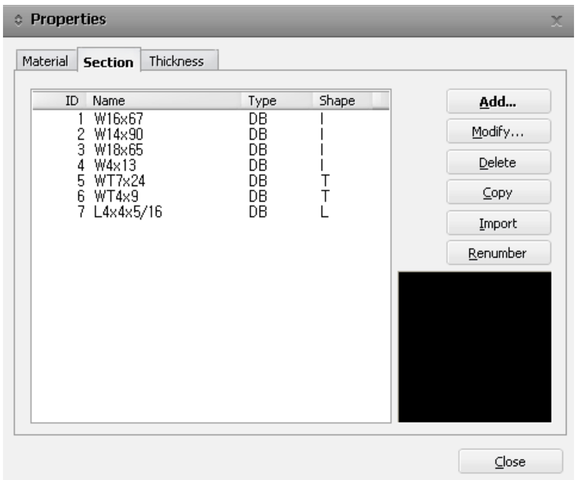


Figure 2.3 Section Data

For simplicity, all the member sections are chosen from the AISC sizes stored in **midas Civil**.

1. Select ***Properties>Material Properties*** in the ***Main Menu***
2. Click Add.
3. Confirm "**1**" in the ***Material Number*** field.
4. Confirm "**Steel**" in the ***Type*** selection field.
5. Select "**ASTM(S)**" in the ***Standard*** selection field.
6. Select "**A36**" in the ***DB*** selection field.
7. Enter "**Frame**" in the ***Name*** field.
8. Click OK.
9. Select the Section tab at the top of the ***Properties*** dialog box.
10. Click Add.
11. Confirm the ***DB/User tab*** at the top of the ***Section Data*** dialog box.
12. Confirm "**1**" in the ***Section ID*** field.
13. Confirm "**I-Section**" in the ***Section*** selection field.
14. Confirm "**AISC**" in the ***DB*** selection field.
15. Select "**W 16 × 67**" in the ***Sect. Name*** selection field.
16. Click Apply.
17. Enter "**2**" to "**4**" in ***Section ID*** (I-section) by repeating steps 15 and 16 (Fig.2.3).
18. Enter the remaining section data for T and Angle sections following the procedure similar to that for I-sections.
19. Finally, click OK in the dialog box.
20. Confirm if there is any error in the member selection and click Close.

A default in the Standard field can be assigned in advance by Tools>Preferences.

Name plays the role of Description to distinguish the material properties. It is not related to any physical properties. By selecting the DB type, an identical name is automatically attributed to the Name field.

By entering the first part of the member's name in the Section Name field, the member's name is automatically searched from the Drop List.

By using the Point Grid in the previously defined UCS x-y plane, position the elements located on grid A of the structure. Generate the elements by following the procedure below and by selecting ***Node/Element*** > ***Create Elements*** in the ***Main Menu***.

Structural Modeling Using Nodes and Elements

Generate Column Elements

1. Click ***Hidden*** in the Icon Menu or in ***View*** > ***Hidden***.
2. Click ***Node Number*** and ***Element Number*** in the Icon Menu (Toggle on).
3. Click ***Point Grid Snap*** in ***View*** > ***Grid*** (Toggle on).
4. Select ***Node/Element*** > ***Create Elements*** in the ***Main Menu***.
5. Confirm "**General beam/Tapered beam**" in the ***Element Type*** selection field.
6. Confirm "**1 : Frame**" in the ***Material Name*** selection field.
7. Confirm "**1 : W 16 × 67**" in the ***Section Name*** selection field.
8. Confirm "**0**" in the ***Beta Angle*** selection field.
9. Generate elements **1 & 2** (exterior columns) by linking the positions (**0, 0, 0**) and (**0, 30, 0**) and positions (**64, 0, 0**) and (**64, 30, 0**) relative to the UCS coordinates noted at the bottom of the screen (Fig.2.4–❶) with the mouse cursor.
10. Select "**2 : W 14 × 90**" in the ***Section Name*** selection field.
11. Select "**90**" in the ***Beta Angle*** selection field.
12. Assign successively positions (**32, 0, 0**) and (**32, 30, 0**) relative to UCS to generate element **3**.

While generating elements, if the window area is too small or the position is unsuitable, the user may adjust the window by using Zoom or Pan (Fig.2.4-❸).

If the node numbering does not operate automatically, click the Nodal Connectivity field once and execute step 12.

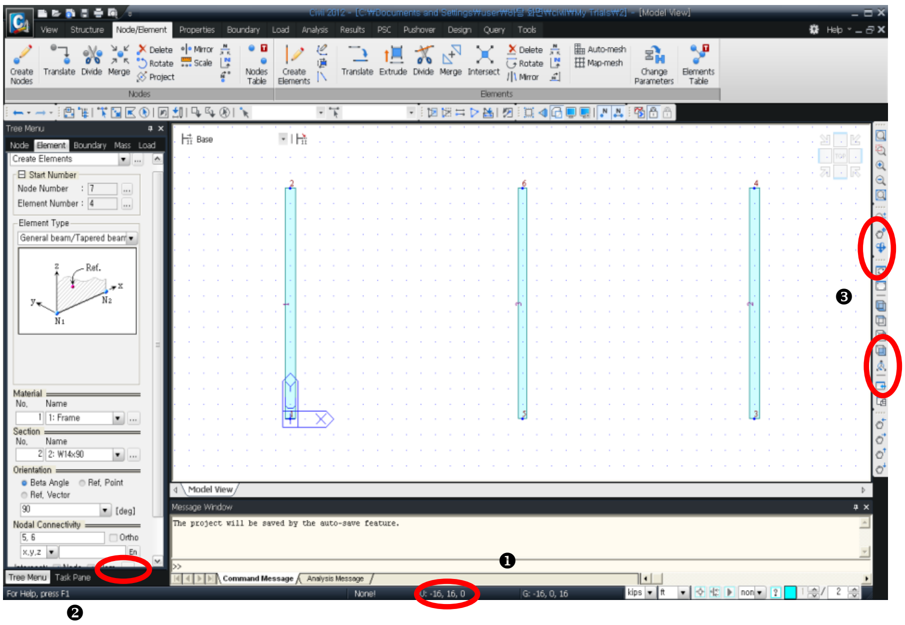


Figure 2.4 Generation of Column Elements

Generate Roof Truss

Civil supports Structure Wizard for various structures to expedite the modeling of standardized structures.

- Beam
- Column
- Arch
- Frame
- Truss
- Plate
- Shell
- Suspension Bridge
- Cable Stayed Bridge
- ILM Bridge
- FCM Bridge
- MSS Bridge
- FSM Bridge
- Transverse Model
- Grillage Model
- RC Slab Bridge
- RC Frame/Box
- PSC Bridge

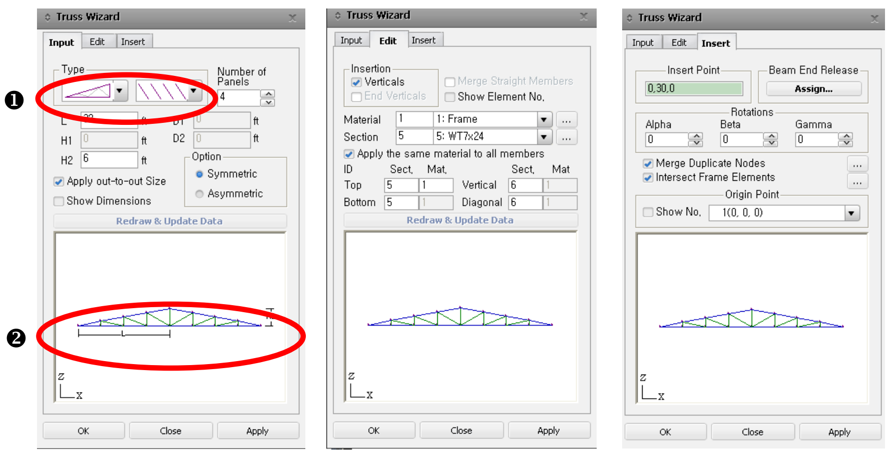
The procedure for generating the roof trusses in the UCS x-y plane is illustrated next. Even if the shape of the structure is complex such as the roof trusses, ***Structure Wizard*** can be a useful tool for standardized structures.

The ***Truss Wizard*** has 3 dialog boxes that serve the following purposes:

- ***Input*** tab: Assign the size and shape of the truss
- ***Edit*** tab: Determine whether the member is vertical and assign the material properties and sections of the truss
- ***Insert*** tab: Assign the position and orientation angles to set the defined truss in the model

When a truss is auto-generated by ***Truss Wizard***, top and bottom chords are classified as ***Beam Elements*** and vertical and diagonal members are classified as ***Truss Elements***.

First of all, click Close as shown in Fig.2.4–❷ and switch to the Main Menu. Then, execute ***Structure>Base Structures>Truss*** to start modeling.



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

Figure 2.5 Truss Wizard

1. Click Close as shown in Fig.2.4–❷.
2. Click ***Structure>Base Structures>Truss*** in the Main Menu.
3. Select the roof truss shape in the ***Type*** of the ***Input*** tab (Fig.2.5(a)–❶).
4. Confirm "**4**" in the ***Number of Panels*** field.
5. Enter "**32**" in the ***L*** field (length of truss).
6. Enter "**6**" in the ***H2*** field (height of truss).
7. Select "**Symmetric**" in the ***Option*** selection field. 🎧

🎧 Check (✓) Show Dimensions. Then the screen displays the values of L and H2 (Fig.2.5(a)-❷).

1. Select the ***Edit*** tab.
2. Check (✓) **Verticals** in ***Insertion***.
3. Select "**1: Frame**" in the ***Material*** selection field.
4. Select **5 (WT 7 × 24)** in the ***Top*** (top chord) selection field of ***Section***. 🎧
5. Select **5 (WT 7 × 24)** in the ***Bottom*** (bottom chord) selection field.
6. Select **6: (WT 4 × 9)** in the ***Vertical*** (vertical members) selection field.
7. Select "**6 (WT 4 × 9)**" in the ***Diagonal*** (diagonal members) selection field. 🎧

🎧 The material properties and section numbers can be directly entered by moving around the entry fields using the Tab key.

🎧 Check (✓) Show Member No. to display the member numbers in the Wizard Model Window.

1. Select the ***Insert*** tab.
2. Enter **(0,30,0)** relative to the UCS in the ***Insert Point*** field or click the field once and node **2** at the top end of element **1** with the mouse.
3. Confirm "**1 (0, 0, 0)**" in the ***Origin Point*** selection field.
4. Click on **Assign** button under **Beam End Release** and un-check all the checked Elements to remove beam end release.
5. Click OK.
6. Click Apply. 🎧
7. Click ***Zoom Fit***.
8. Click Close to exit the ***Truss Wizard*** dialog box.

🎧 If the truss data results are different from that in Fig.2.6, click Undo and modify the data for Structure Wizard Truss.

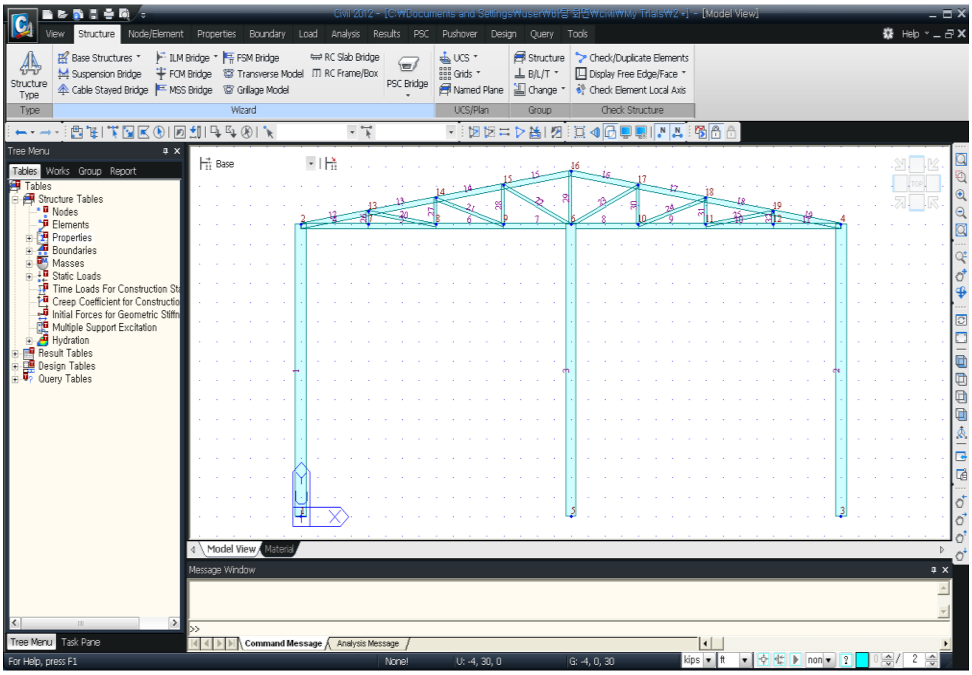


Figure 2.6 Generation of Roof Truss

Switch the Direction of Roof Truss Bottom Chords

The roof truss generated by ***Structure Wizard*** is shown in Fig.2.6.

The T-shape section members are used for the bottom chord of the roof truss (Fig.2.7– ❷) and the section is non-symmetrical about the strong axis. Click ***Iso View***, ***Shrink***, and ***Zoom Window*** to magnify the bottom chord of the truss. Observation of the section of the bottom chord will show that the web is oriented downward.

The joint detail at the bottom chord to which the vertical or diagonal members connect shows that the web of the bottom chord must be oriented upward for easier fabrication. The method that revises the section orientation such that the web is oriented upward is examined.

During the data entry, analysis of results or design steps, it is most efficient to assign the specific elements by group. Group is an extremely useful tool for data entry, results interpretation and design. Specific elements are grouped.

The bottom chord is selected by ***Group***, which selects and saves the entity as determined by the user in advance.

It is easier to select the bottom chord if 'x' is selected in the Filter field (Fig.2.7- ❶).

Double click the Roof Bottom Chord group, then the nodes and elements assigned as Roof Bottom Chord group will be selected as done in step 8.

1. Click ***Select Window*** in the Icon Menu and drag the mouse from left to right to select only the roof truss bottom chord.
2. In the ***Tree Menu*** > ***Group*** > ***Structure Group*** (or Click ***Group*** Icon), Right-click the mouse on the ***Structure Group*** and then select ***New*** to enter "**Roof Bottom Chord**".
3. From the ***Structure Group*** drag "**Roof Bottom Chord**" with the mouse and drop it to the model window.
4. Click ***Shrink*** in the Icon Menu (Toggle on).
5. Click ***Iso View*** in the Icon Menu.
6. Click ***Zoom Window*** in the Icon Menu (Toggle on).
7. Magnify ❷ shown in Fig.2.7 to confirm the orientation of the bottom chord.
8. Select the "**Roof Bottom Chord**" group and double-click the mouse.
9. Execute ***Node/Element>Elements>Change Element Parameters*** in the ***Main Menu***.
10. Select ***Element Local Axis*** in the ***Parameter Type*** selection field.
11. Select ***Assign*** in the ***Mode*** selection field.
12. Select ***Beta Angle*** in the ***Mode*** selection field.
13. Select "**180**" in the ***Beta Angle*** field.
14. Click Apply

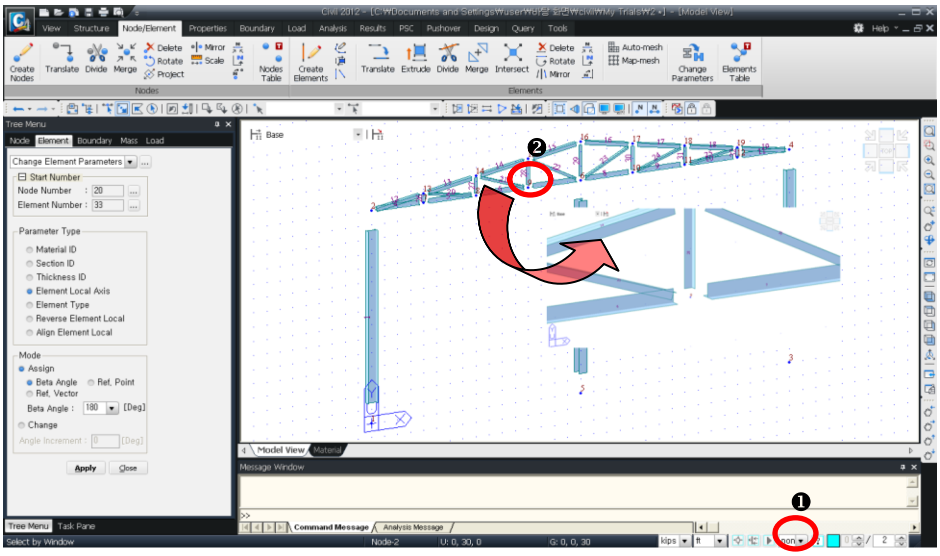


Figure 2.7 Switching the Bottom Chord Beta Angle

Generate Floor Story Girders/Beams

Use ***Point Grid Snap*** to generate the 2 intermediate floor girders.

1. Click ***Auto Fitting*** in the Icon Menu (Toggle on).
2. Click ***Front View*** in the Icon Menu.
3. Select ***Create Elements*** in the function list (Fig.2.8–❶).
4. Select "**3: W 18 × 65**" in the ***Section Name*** selection field.
5. Confirm "**0**" in the ***Beta Angle*** field.
6. Check (✓) **Elem** and **Node** of ***Intersect***.

When connecting a new element to a particular node on a column, use Elem in the Intersect field to divide the column member automatically at the relevant node.

7. Click the ***Nodal Connectivity*** field once.
8. Assign positions (**0, 10, 0**) and (**32, 10, 0**) relative to the UCS to generate the first girder.
9. Assign positions (**0, 20, 0**) and (**32, 20, 0**) relative to the UCS to generate the second girder.

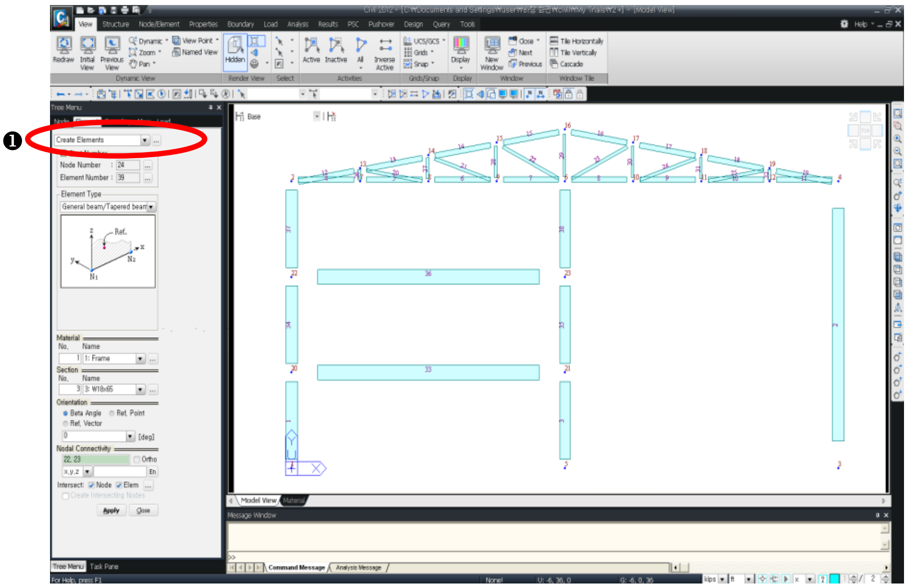


Figure 2.8 Generation of Floor Girders

Generate 3-D Frame

By using the completed 2-D frame, duplicate 2 frames in the GCS Y-direction at an interval of 26 ft.

First, switch the window to the 3-D state. When selecting elements to be duplicated, exclude the 2 floor girders. Two floor girders will be duplicated later with the small beams simultaneously.

1. Select ***GCS*** within ***View*** > ***UCS/GCS*** > ***GCS*** in the Main Menu.
2. Select ***Iso View*** in the Icon Menu.
3. Click ***Select All*** in the Icon Menu.
4. Click ***Select Single*** in the Icon Menu and click the girders (elements 33 and 36) so that they are not duplicated.
5. Select ***Translate Elements*** in the functions selection field (Fig.2.9–❶).
6. Confirm "**Copy**" in the ***Mode*** selection field.
7. Select "**Equal Distance**" in the ***Translation*** selection field.
8. Enter "**0, 26, 0**" in the ***dx, dy, dz*** field or use ***Mouse Editor*** to enter the data automatically.
9. Enter "**2**" in the ***Number of Times*** field.
10. Click Apply
11. Click ***Element Number*** in the Icon Menu (Toggle off).
12. Click ***Shrink*** in the Icon Menu (Toggle off).
13. Click ***Select Single*** in the Icon Menu and select the two girders with the mouse
14. Enter "**0, 26/3, 0**" in the ***dx, dy, dz*** field.
15. Enter "**6**" in the ***Number of Times*** field.
16. Click Apply

In Civil, numerical values as well as equations may be used simultaneously. Most operators used in engineering can be used.

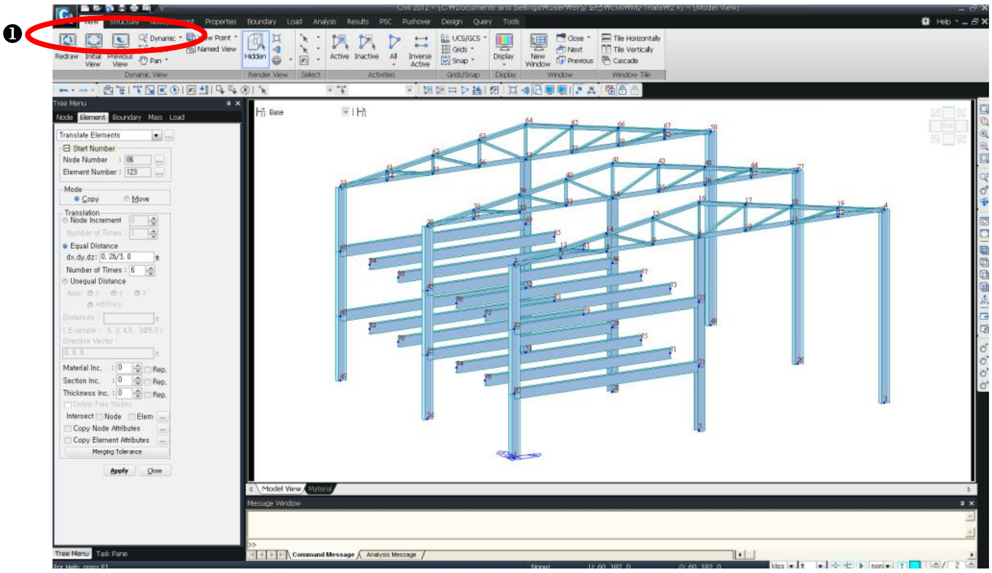


Figure 2.9 Duplicated 3-D Frame

Generate Floor Girders

Create the floor girders by connecting the longitudinal sides of the floors as shown in Fig.2.10. To avoid confusion between the existing point grid and the nodes, toggle off ***Point Grid*** and ***Point Grid Snap***.

1. Click ***Point Grid*** and ***Point Grid Snap*** in ***View > Grids*** and in ***View > Snaps***, respectively (Toggle off).
2. Select ***Create Elements*** in the functions selection field.
3. Confirm "**General Beam/Tapered Beam**" in the ***Element Type*** selection field.
4. Confirm "**1: Frame**" in the ***Material Name*** selection field.
5. Confirm "**3: W 18 × 65**" in the ***Section Name*** selection field.
6. Confirm "**0**" in the ***Beta Angle*** selection field.
7. Check (✓) **Elem** and **Node** of ***Intersect***.
8. Assign nodes **20** (**0, 0, 10**) to **48** (**0, 52, 10**) to generate the lower floor girders on grid ① of Fig.2.1.
9. Assign nodes **21** (**32, 0, 10**) to **52** (**32, 52, 10**) to generate the lower floor girders on grid ②.
10. Assign nodes **22** (**0, 0, 20**) to **68** (**0, 52, 20**) to generate the upper floor girders on grid ①.
11. Assign nodes **23** (**32, 0, 20**) to **69** (**32, 52, 20**) to generate the upper floor girders on grid ②.

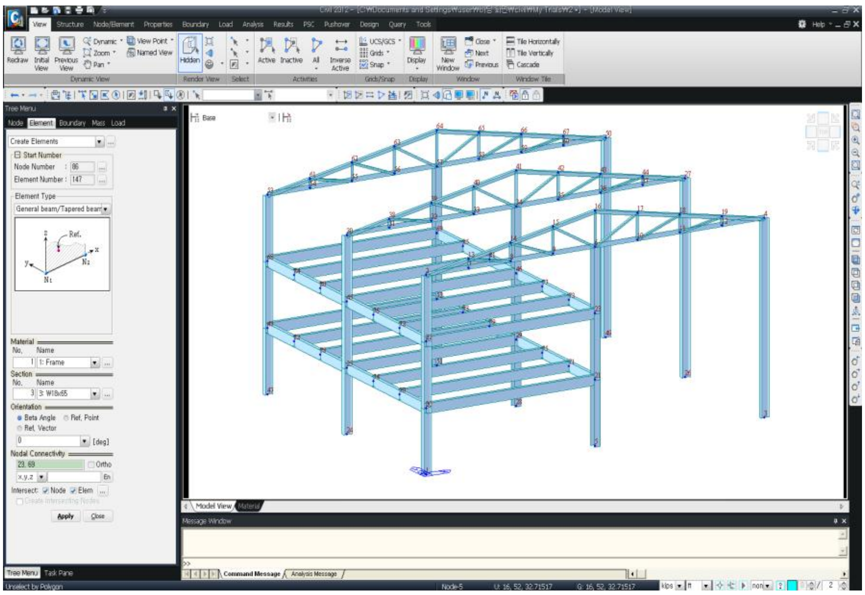


Figure 2.10 Generation of Floor Story Girders/Beams

Generate Roof Girders and Remove Columns

Use *Create Elements* to generate the roof-level girders and remove the generated column elements unnecessarily duplicated during the copy process.

1. Select "**1: W 16 × 67**" in the ***Section Name*** selection field.
2. Confirm "**0**" in the ***Beta Angle*** selection field.
3. Assign nodes **2 (0, 0, 30)** to **53 (0, 52, 30)** to generate the roof-level girders (Fig.2.11–❷)
4. Assign nodes **16 (32, 0, 36)** to **64 (32, 52, 36)** to generate the roof-level girders (Fig.2.11–❸).
5. Select ***Delete Elements*** in the functions selection field.
6. Confirm "**Picking**" in the ***Type*** selection field.
7. Click ***Rotate Dynamic*** to rotate the model to the desired orientation for a clear view of the model.
8. Confirm that ***Element Snap*** is on (in ***View*** > ***Snap*** from the Main Menu).
9. Remove the column elements in Fig.2.11–❹ by assigning with the mouse.

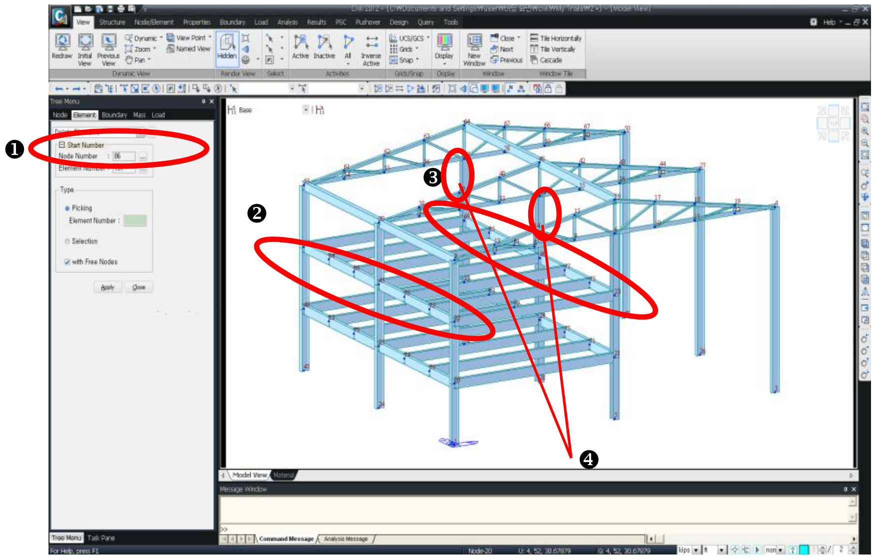
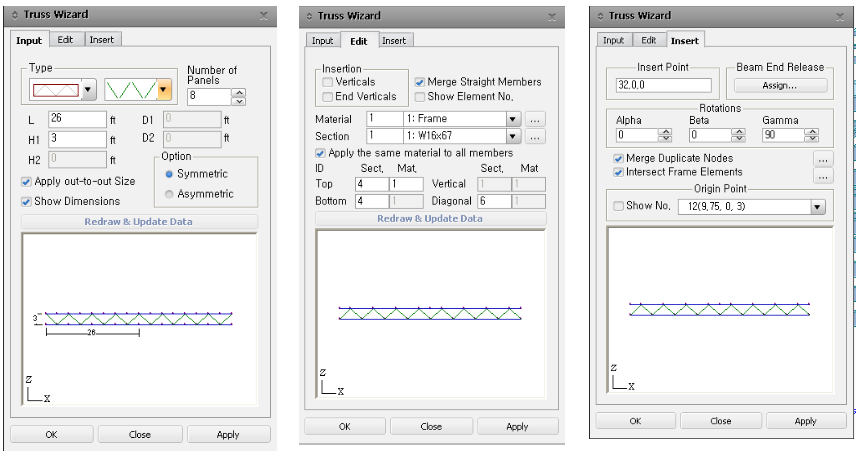


Figure 2.11 Generation of Roof Girders and Removal of Columns

Generate Longitudinal Truss

Use ***Structure Wizard*** to generate the longitudinal truss on grid ③ of the structure (Fig.2.1). Use ***Rotate Left (in View>View Point)*** or ***Rotate Dynamic (in the Icon Menu)*** to change the view by rotating the model to the desired orientation. The generation of the longitudinal truss is similar to that of the roof-level truss and the procedure is as follows:

1. Click ***Rotate Dynamic*** in the Icon Menu and rotate the model as shown in Fig.2.13.
2. Select ***Structure>Structure Wizard>Truss*** in the Main Menu.
3. Select the truss shape (Fig.2.12(a)) in the ***Type*** selection field of the ***Input*** tab.
4. Enter "**8**" in the ***Number of Panels*** (number of divisions of Top & Bot. Chords) field.
5. Enter "**26**" in the ***L*** field (length of truss).
6. Enter "**3**" in the ***H1*** field (height of truss).
7. Select "**Symmetric**" in the ***Option*** selection field.
8. Check (✓) **Show Dimensions** and confirm **L** and **H1**.



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

Figure 2.12 Truss Wizard

Select the ***Edit*** tab.

1. Select "**1 : Frame**" in the ***Material*** selection field.
2. Enter **4 (W 4 ×13)** in the ***Top*** (top chord) field.
3. Enter **4 (W 4 ×13)** in the ***Bottom*** (bottom chord) field.
4. Enter **6 (WT 4 × 9)** in the ***Diagonal*** (diagonal member) field.
5. Check (✓) ***Merge Straight Members***.

When Wizard Model is inserted into the main model, Alpha, Beta and Gamma represent the rotation angles which the partial model is to be rotated about GCS X, Y and Z-axes, respectively.

If Show No. is checked(✓), the node numbers will be displayed on the screen which will enable the user to find the Origin Point.

1. Select the ***Insert*** tab.
2. Enter "**64, 0, 30**" in the ***Insert Point*** field or use ***Mouse Editor*** to assign node **4**.
3. Enter "**90**" in the ***Gamma*** field of ***Rotations***.
4. Select "**12 (0, 0, 3)**" in the ***Origin Point*** selection field.
5. Click on **Assign** button under **Beam End Release** and un-check all the checked Elements to remove beam end release.
6. Click OK.
7. Click Apply (Figure 2-13-❶).
8. Confirm if the longitudinal truss is correctly generated and click Close in the ***Truss Wizard*** dialog box.
9. Click ***Iso View*** in the Icon Menu (Figure 2-13).

Comment 1. ……………………………………………

When using **midas Civil**, the use of ***Query*** is a handy tool to find the distance between nodes or the length of an element.

Selecting the ***Query***>***Query Nodes*** or ***Query Elements*** menus will display the ***Query*** dialog box. If the ***Node*** tab at the top is selected and if two nodes are selected with the mouse, the ***Message*** window will indicate the positions and the distance between the two nodes.

Similarly, select ***Element*** in the ***Query*** dialog box and assign the desired element with the mouse, then the element and length data will appear in the ***Message*** window.

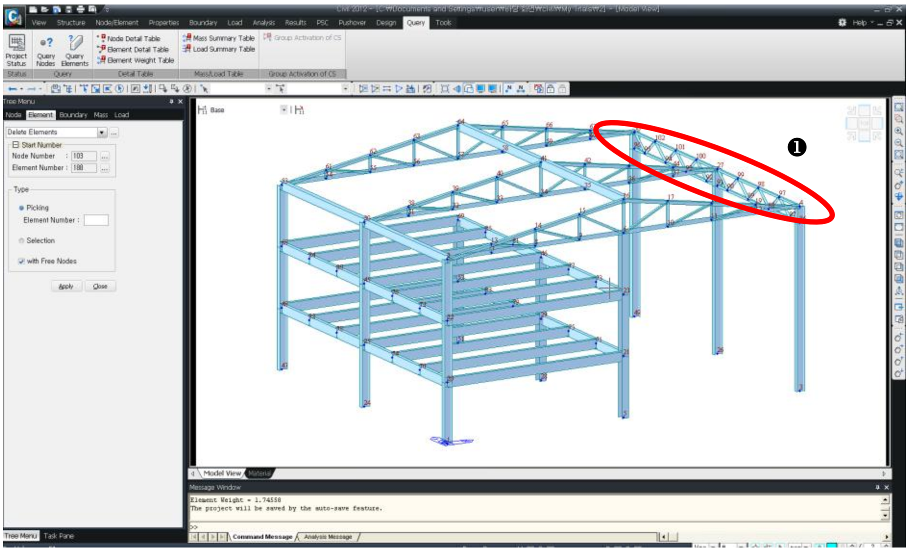


Figure 2.13 Generation of Longitudinal truss

Generate Wind Braces

1. Select *Create Elements* in the functions selection field (Fig.2.14–❶).
2. Select "**Truss**" in the ***Element Type*** selection field.
3. Confirm "**1 : Frame**" in the ***Material Name*** selection field.
4. Select "**7 : L 4 × 4 × 5/16**" in the ***Section Name*** selection field.
5. If **Node** and **Elem** of ***Intersect*** are already checked (✓), click once again to remove the check.
6. Connect nodes **5** (**32, 0, 0**) and **4** (**64, 0, 30**) of the X-Z plane to generate the wind brace.
7. Connect nodes **6** (**32, 0, 30**) and **3** (**64, 0, 0**) to generate the wind brace.
8. Repeat steps 6 and 7 to generate the remaining wind braces in the roof floor and the Y-Z plane.

If Elem of Intersect is checked (✓), remove the check to avoid the automatic division of elements at the intersection points of X-shape wind braces.

When nodes are to be identified for member generation, set Hidden to Toggle off or use Zoom and Rotate to change the View Point until it becomes easier to manipulate the model.

❶

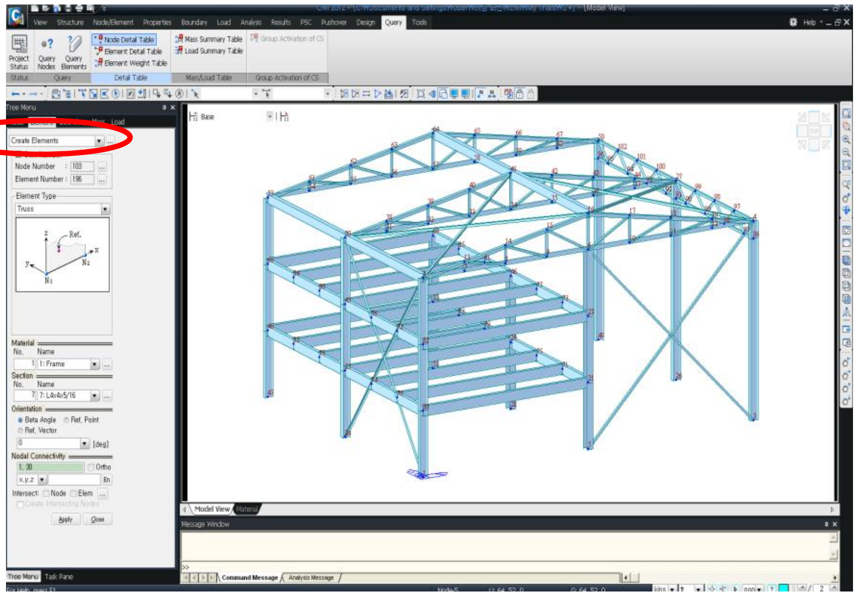


Figure 2.14 Generation of Wind Braces

Complete 3–D Modeling By Mirror Duplication

The structure is symmetrical about grid © (Fig.2.1). The structure modeled up to this point is mirror-duplicated with respect to grid ©, thereby completing the entire model.

1. Click **Node Number** in the Icon Menu (Toggle off).
2. Click ***Shrink*** in the Icon Menu (Toggle on).
3. Click ***Left View*** in ***View*** > ***View Point*** from the Main Menu.
4. Select ***Mirror Elements*** in the functions selection field (Fig.2.15–❶).
5. Click ***Select Window*** in the Icon Menu.
6. Drag the mouse from right to left as shown in Fig.2.15–❷ to select the nodes and elements to be duplicated.
7. Select "**Copy**" in the ***Mode*** selection field.
8. Select ***z-x plane*** in ***Reflection*** and assign any node in the plane containing grid C to specify "**52**" automatically (see Figure 2-1).
9. Click Apply
10. Click ***Iso View*** in the Icon Menu.

As Snap cannot be executed when Select Window is Toggled on, switch Select Window to Toggle off before executing step 8.

When selecting members for duplication, exclude the elements contained in the plane of symmetry (Fig. 2.15-❸). When using Select Window, only the elements completely included in the window are selected by dragging the mouse from left to right, and the elements intersecting the boundary of the window are also selected by dragging the mouse from right to left. For this reason the mouse is dragged from right to left.

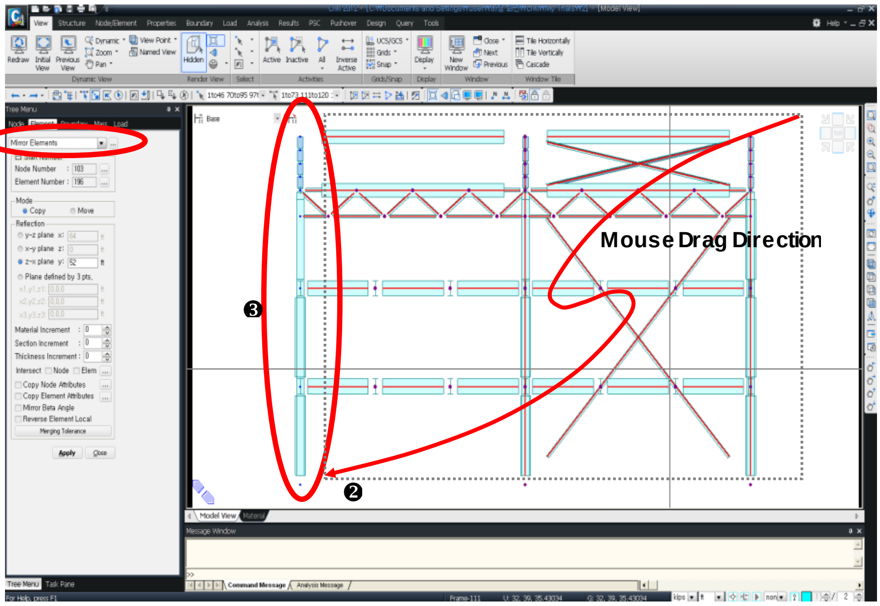


Figure 2.15 Selection of Target Elements of Mirror Elements

Enter Structure Support Conditions

When the entire model is complete, attribute the support conditions to the lower ends of the columns. In this example, assume that the lower ends of the columns are pinned (pin support).

1. Select the ***Boundary*** tab (in the Tree Menu) as shown in Fig.2.16–❶.
2. Confirm ***Supports*** in the functions selection field.
3. Click ***Select Plane*** in ***View*** > ***Select*** > ***Plane*** from the Main Menu.
4. Select "**XY Plane**".
5. Enter "**0**" in the ***Z Position*** field (Z-axis coordinate of the lower ends of the columns to receive support conditions). Click Apply or assign any node at the lower end of a column with the mouse and click Close.
6. Confirm "**Add**" in the ***Options*** selection field.
7. Check (✓) "**D-ALL**" in the ***Support Type*** selection field.
8. Click Apply

When the coordinates of a node has been defined by Node Local Axis, the restraints are also defined with respect to the Node Coordinate.

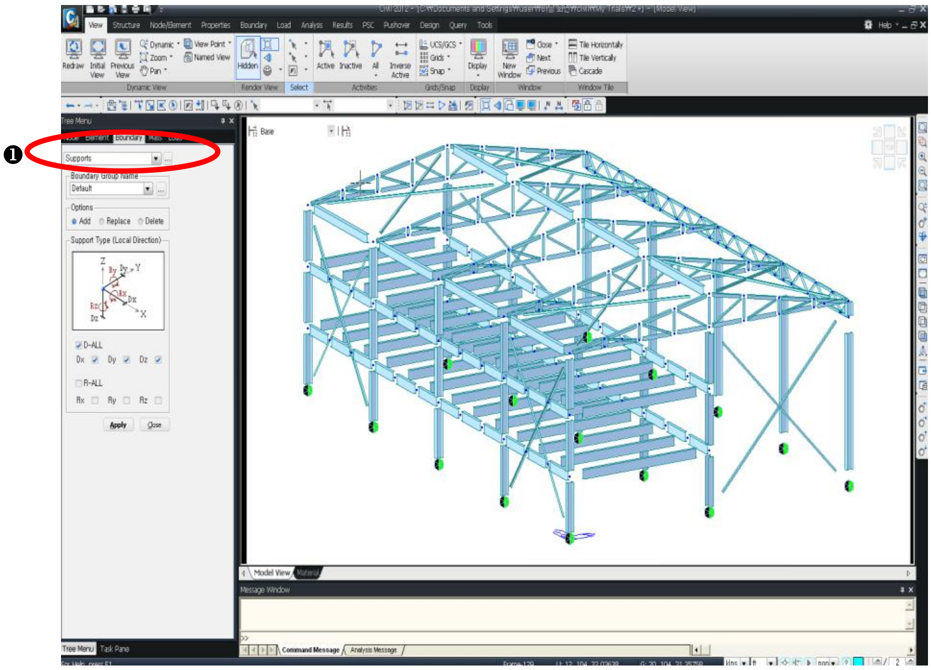


Figure 2.16 Data Entry of Support Conditions

Enter Loading Data

Setup Load Cases

Set up load cases before entering the loading data.

1. Select the ***Load*** tab within the Tree Menu.
2. Click the button [...] to the right of ***Load Case Name***.
3. Enter the load cases in the ***Static Load Cases*** dialog box as shown in Fig.2.17.

Refer to “Load Cases Setup” of “Tutorial 1” or On-line Manual.

Separate Roof Live Load from normal floor Live Load. A different live load factor is applied to roof live load in LRFD or Limit States Design.

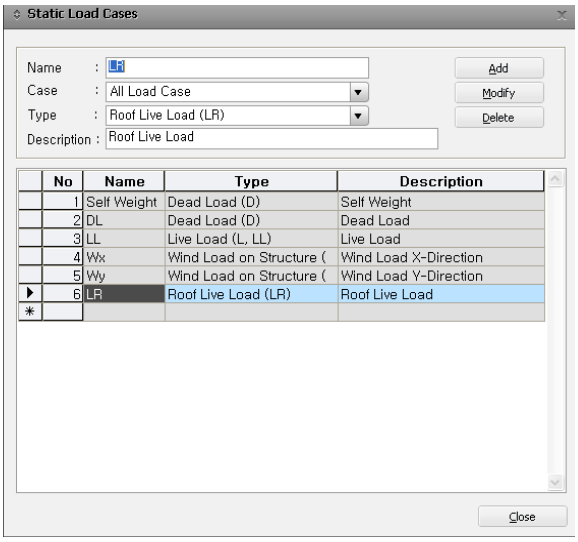


Figure 2.17 Data Entry for Load Cases

Define Self Weight

The ***Self Weight*** of the structure is computed automatically, reflecting the densities and the volumes, when the material properties and sections of members are defined. The ***self-weight*** of the structure is taken into account in the analysis with respect to the gravity direction or any other defined direction by the given factors through the ***Self Weight*** command.

Observe the following procedure to specify the self-weight:

1. Select ***Self Weight*** in the functions selection field (Fig.2.18–❶).
2. Confirm “**Self Weight**” in the ***Load Case Name*** selection field.
3. Enter “**-1**” in the ***Z*** field of ***Self Weight Factor***.
4. Click Add in the ***Operation*** selection field.

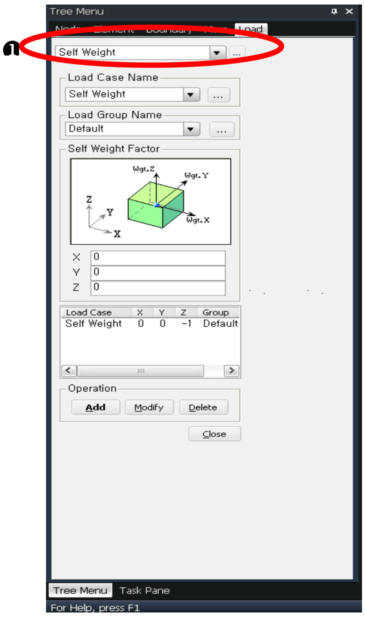


Figure 2.18 Data Entry of Self Weight

Define Floor Load

Specify the floor loads.

As the structure configurations and the magnitudes of floor loads are identical in this example, the floor loads on both floors are defined simultaneously.

Activate only the loaded planes to avoid any error during the data entry in ***Assign Floor Load***.

The size of Floor Load Label may be adjusted in the Size tab in Display Option.

❶

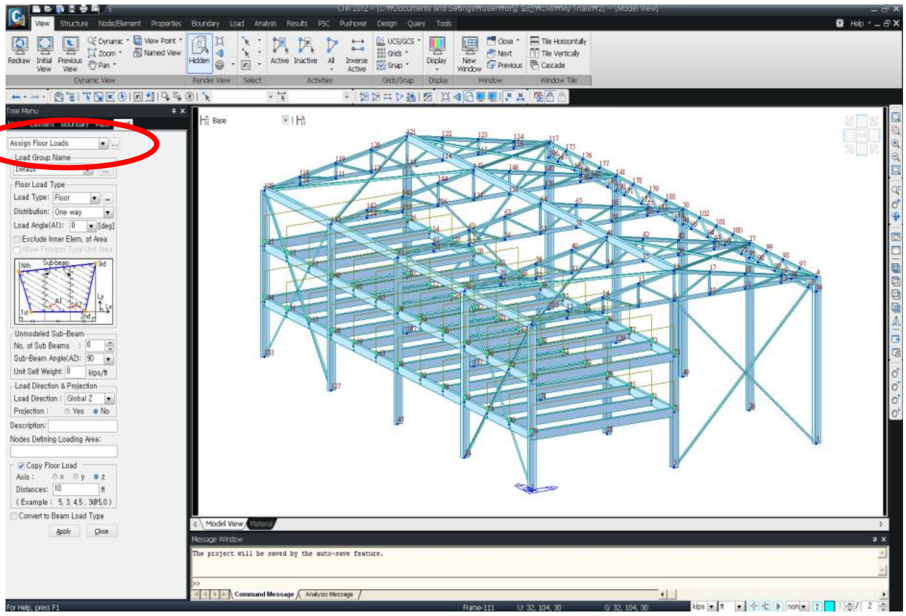


Figure 2.19 Data Entry for Floor Loads

1. Select ***Assign Floor Loads*** in the functions selection field (Fig.2.19–❶).
2. Click the button [...] to the right of ***Load Type***.
3. Enter "**Floor**" in the ***Name*** field of ***Floor Load Type & Description***.
4. Enter "**Top and Bottom identical**" in the ***Description*** field.
5. Select "**DL**" in the ***Load Case 1.*** selection field of the ***Floor Load & Load Case*** and enter "**- 0.1**" in the ***Floor Load*** field.
6. Select "**LL**" in the ***Load Case 2.*** selection field and enter "**- 0.1**" in the ***Floor Load*** field.
7. Click [Add].
8. Click [Close].
9. Select "**Floor**" in the ***Load Type*** selection field.
10. Select "**One Way**" in the ***Distribution*** selection field.
11. Check (✓) **Copy Floor Load** to enter the floor loads of the top and bottom floors simultaneously.
12. Confirm "**z**" in the ***Axis*** selection field.
13. Enter "**10**" in the ***Distances*** field.
14. Click ***Front View*** in the Icon Menu.
15. Click ***Select Window*** in the Icon Menu and drag the mouse from left to right to select only the bottom floor.
16. Click ***Activate*** in the Icon Menu.
17. Click ***Node Number*** in the Icon Menu (Toggle on).
18. Click ***Iso View*** in the Icon Menu.
19. Click the ***Nodes Defining Loading Area*** field once and assign sequentially the nodes (**104, 20, 21, 108, 104**) forming an irregular polygon plane defining the loaded area.
20. Click ***Shrink*** in the Icon Menu (Toggle off).
21. Click ***Activate All*** in the Icon Menu.
22. To see the applied floor load on the second floor, in ***View*** > ***Display*** > ***Load tab***, check ***floor load*** and then click ***OK***

If Sub Beam Weight is checked(✓), the value of Unit Self Weight specified for the unmodeled sub-beams are added to the floor load of the load cases in which the check (✓) is marked.

If Convert to Beam Load Type is not checked (✓), the load data will be saved as floor load and future modification will be easy.

During the floor load generation in Civil, the user is cautioned to properly define Load Angle (A1) and Sub-Beam Angle (A2) in specifying the loaded area. The angles are determined on the basis of the line defined by the 1st and 2nd nodes. (Refer to On-line Manual).

Define Inclined Roof Load

Apply the dead and live loads of the inclined roof on the top chords of the roof trusses (beam elements).

Floor Load may be applied to any plane in the model in **midas Civil**. Hence, dead and live loads acting on an inclined roof as well as snow loads and wind loads, may be specified by ***Floor Load***.

Use ***Floor Load*** to enter the dead and live loads acting on the inclined roof in the example.

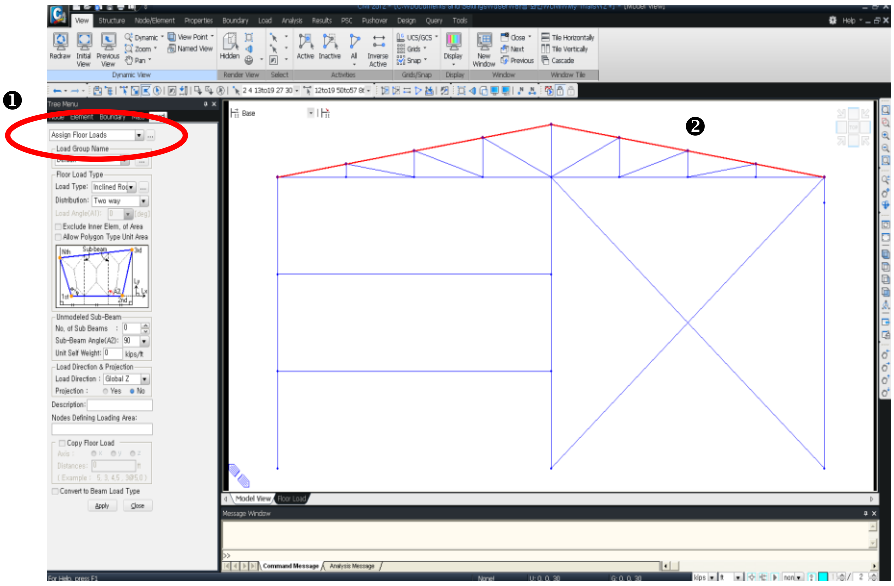


Figure 2.20 Data Entry for Inclined Roof Loads

1. Click *Front View* in the Icon Menu.
2. Click *Hidden* in the Icon Menu (Toggle off).
3. Confirm ***Assign Floor Loads*** in the functions selection field (Fig.2.20–❶).
4. Click the button [...] to the right of ***Load Type*** and enter "**Inclined Roof**" in the ***Name*** field.
5. Remove the contents in the ***Description*** field.
6. Enter "**- 0.015**" for ***Floor Load*** in ***Load Case 1*** from which ***DL*** has been selected.
7. Select "**LR**" from ***Load Case 2.*** and enter "**- 0.02**" for ***Floor Load***.
8. Click [Add].
9. Click [Close].
10. Select "**Inclined Roof**" in ***Load Type***.
11. Confirm "**One Way**" in the ***Distribution*** selection field.
12. Click ***Select Polygon*** in the Icon Menu and select only the upper chords of the roof trusses shown in Fig.2.20–❷
13. Click ***Activate*** and ***Iso View*** in the Icon Menu.
14. Enter "**3**" in ***No. of Sub Beams*** of ***Unmodeled Sub-Beam***.
15. Confirm "**90**" in ***Sub-Beam Angle (A2)***.
16. Enter "**0.033**" in ***Unit Self Weight***.
17. Remove the check (✓) in "**Copy Floor Load**".
18. Click the ***Nodes Defining Loading Area*** field once and click nodes **2, 16, 121, 109, 2** sequentially.
19. Click nodes **16, 4, 117, 121,16** successively.
20. Click ***Active All*** in the Icon Menu.
21. Click ***Node Number*** in the Icon Menu (Toggle off).

When using Select Polygon, double click the last node which defines the selected area for nodes or elements.

Where a load such as snow load is applied vertically on a projected plane of the roof, select 'Yes' in the Projection field.

If Fast Query of the Status Bar is used, the attributes of the snapped nodes or elements can be easily verified.

Define X-Direction Wind Load

Specify the X-direction wind load as a uniformly distributed load.

In practice, the wind load must be applied to wall and roof planes considering the members' orientation and the tributary areas. To simplify the problem, assume that a uniformly distributed load of 0.1 k/f is applied to only the columns and girders on grid ① of the structure.

1. Click *Plane* in View > Select > Plane from the Main Menu
2. Select "**YZ Plane**".
3. Click any point on grid ① plane of the structure (Fig.2.1) to enter "**0**" automatically in the ***X Position*** field.
4. Click Close
5. Select "**Element Beam Loads**" in the functions selection field (Fig.2.21–❶).
6. Select "**WX**" in the ***Load Case Name*** selection field.
7. Confirm "**Add**" in the ***Options*** selection field.
8. Confirm "**Uniform Loads**" in the ***Load Type*** selection field.
9. Select "**Global X**" in the ***Direction*** selection field.
10. Confirm "**No**" in the ***Projection*** selection field.
11. Enter "**0.1**" in the ***w*** field.
12. Click Apply
13. Toggle on ***Hidden.***

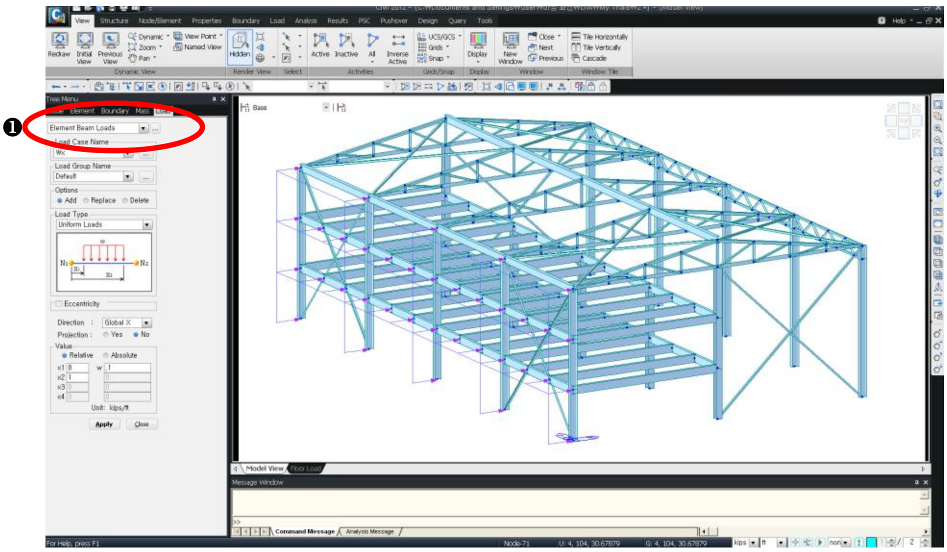


Figure 2.21 Data Entry for X-Direction Wind Load

Click the button on the right of the functions selection field (Fig.2.22–❶) to confirm the loads applied to beam elements in ***Beam Loads Table***.

When confirming the data entries such as load, release, etc. relative to a specific element, use ***Element Detail Table*** for convenience.

1. Click ***Select Previous*** in the Icon Menu.
2. Select ***Query>Element Detail Table*** in the Main Menu.
3. Click the ***Bmld*** tab at the bottom of the ***Element Detail Table*** window (Fig.2.22–❷) and confirm the current load of the selected element.
4. Click as shown in Fig.2.22–❸.
5. Click ***Unselect All*** in the Icon Menu.

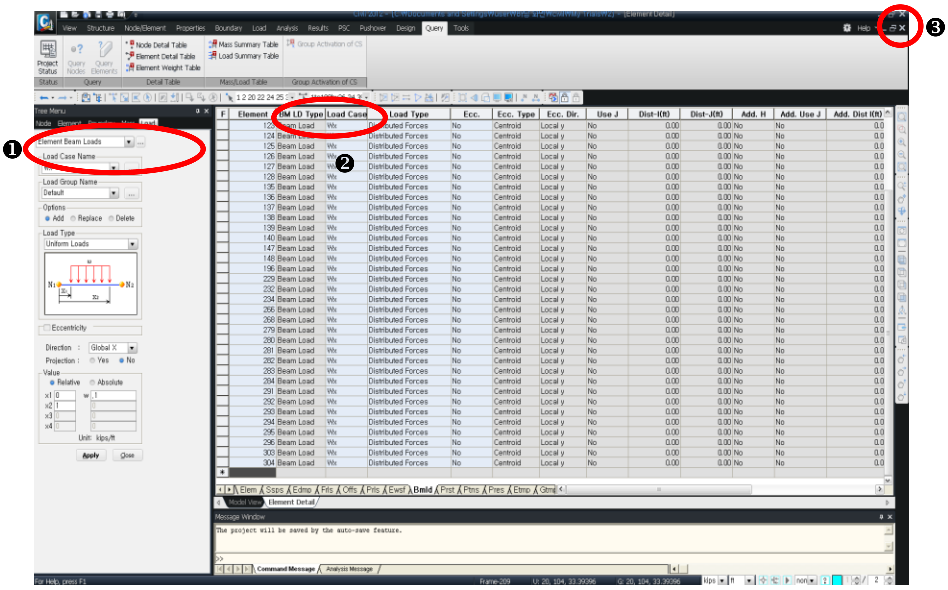


Figure 2.22 Element Detail Table

Define Y-Direction Wind Load

Specify the Y-direction wind load simply as concentrated loads.

1. Click ***Plane*** in View > Select > Plane from the Main Menu.
2. Select "**XZ Plane**".
3. Click any point on plane Ⓐ of the structure for the ***Y Position*** field.
4. Click Close
5. Select ***Nodal Loads*** in the functions selection field.
6. Select "**WY**" in the ***Load Case Name*** selection field.
7. Confirm "**Add**" in the ***Options*** selection field.
8. Enter "**5**" in the ***FY*** field.
9. Click Apply

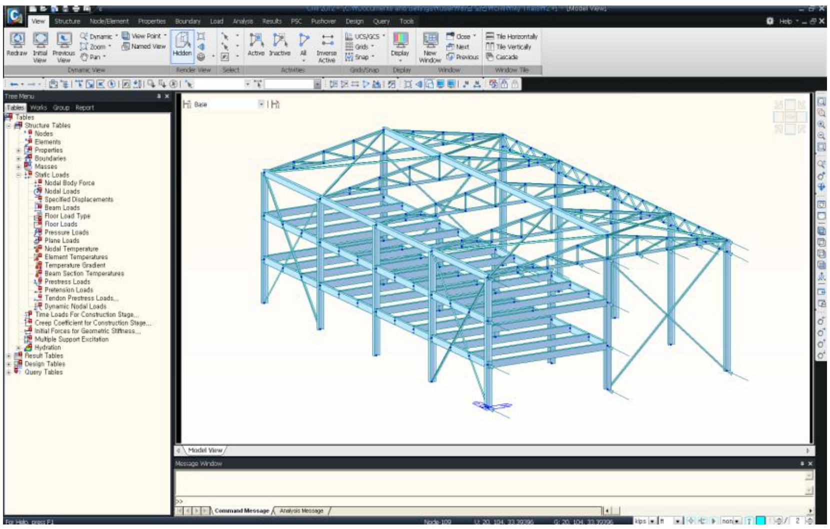


Figure 2.23 Data Entry for Y-Direction Wind Load

Verify the entered nodal loads.

1. Click ***Select Previous*** in the Icon Menu.
2. Select ***Query>Node Detail Table*** in the Main Menu.
3. Click the ***Cnld*** tab at the bottom of the ***Node Detail Table*** window (Fig.2.24) and confirm the current load of the selected element.
4. Click as shown in Fig.2.24–❶.
5. Click ***Unselect All*** in the Icon Menu.

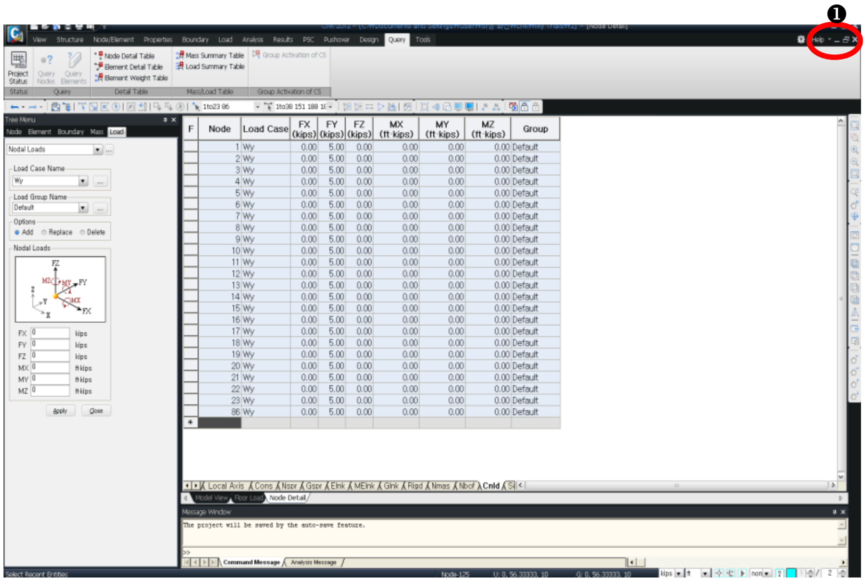


Figure 2.24 Node Detail Table

Confirm the status of model data input prepared using Works Tree

Midas Civil provides the state-of-the-art modeling capability, ***Works Tree***, which systematically summaries and maintains the process of input data construction.

From the record of data entry process contained in ***Works Tree***, previously entered attributes may be modified, deleted, activated or deactivated. Moreover, appropriate attributes can be intuitively assigned through ***Drag & Drop***.

1. Select the ***Works*** tab as shown in Fig.2.25–❶.
2. Select "**Type 1[111000]**" in ***Boundaries>Supports*** and right-click the mouse.
3. Select "**Display**" From the context menu of ***Works Tree***.
4. Confirm the entered support condition of the model as shown in Fig.2.26.
5. Select "**Floor Loads: 4**" in ***Static Loads>Static Load Case2 [DL:Dead Load]***.
6. Select "**Display**" from Context Menu.
7. Confirm the entered floor load as shown in Fig.2.27.
8. Click ***Display*** within ***View*** from the Main Menu and uncheck ***Support*** under the ***Boundary*** tab.
9. Click OK.

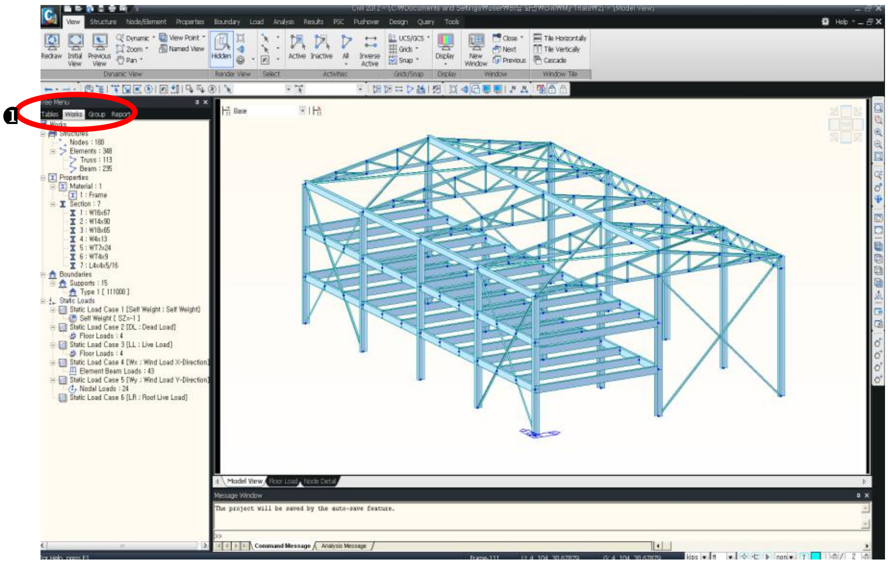


Figure 2.25 Selection of the Works tab of Tree Menu

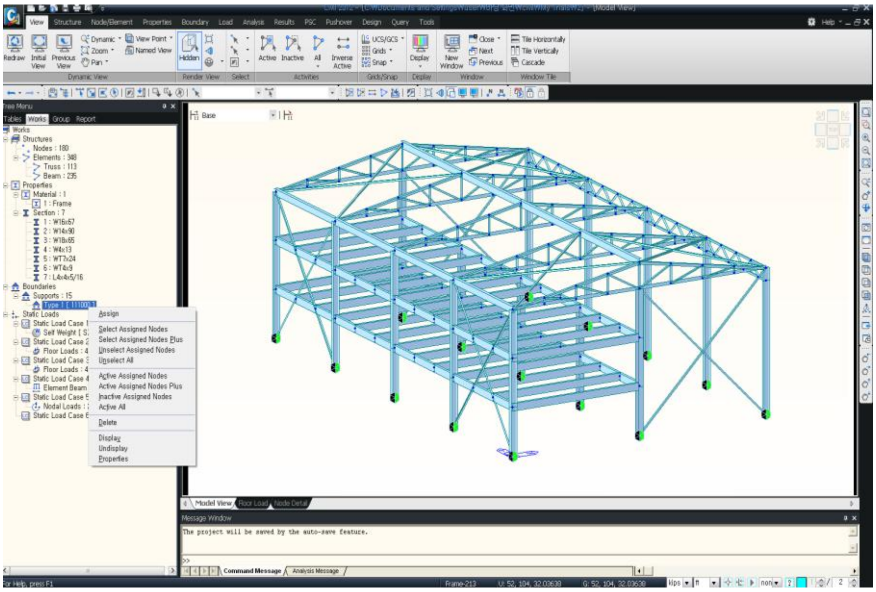


Figure 2.26 Confirmation of Support Condition using Works Tree

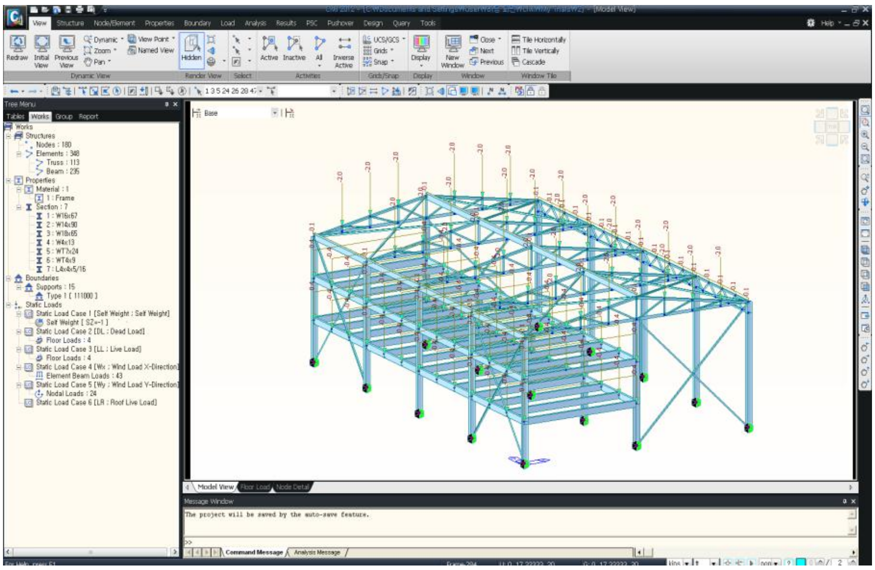


Figure 2.27 Confirmation of Floor Load input using Works Tree

Perform Structural Analysis

Click ***Analysis*** in the Icon Menu or select ***Analysis>Perform Analysis*** in the Main Menu to analyze the model. Once the analysis is completed, the program switches automatically to the ***post-processing*** mode, which provides access to the analysis and design results.

Click ***Preprocessing Mode*** in the Icon Menu when the ***preprocessing*** mode has to be restored to modify the data.

Verify and Interpret Analysis Results

Load Combinations

Use ***Results>Load Combinations*** in the Main Menu to specify the load combinations prior to verifying the analysis results.

The load combinations, in conformity with the ***Limit State Design Method***, are auto-generated by ***Auto Generation***. Additional ***Service Load*** combinations may be entered to examine displacements and reactions.

1. Select the ***Results> Load Combinations*** in the Main Menu.
2. Select the ***Steel Design*** tab.
3. Click Auto Generation... .
4. Confirm "**Add**" in the ***Option*** selection field.
5. Select "**AISC-LRFD 2k** in the ***Design Code*** field.
6. Click OK .
7. Click Close in the ***Load Combinations*** dialog box.

For details concerning the data entry, refer to "Load Combination" of "Tutorial 1".

During the auto-generation of load combinations, the load combination description reflects the load case names assigned by the user such as wind and earthquake.

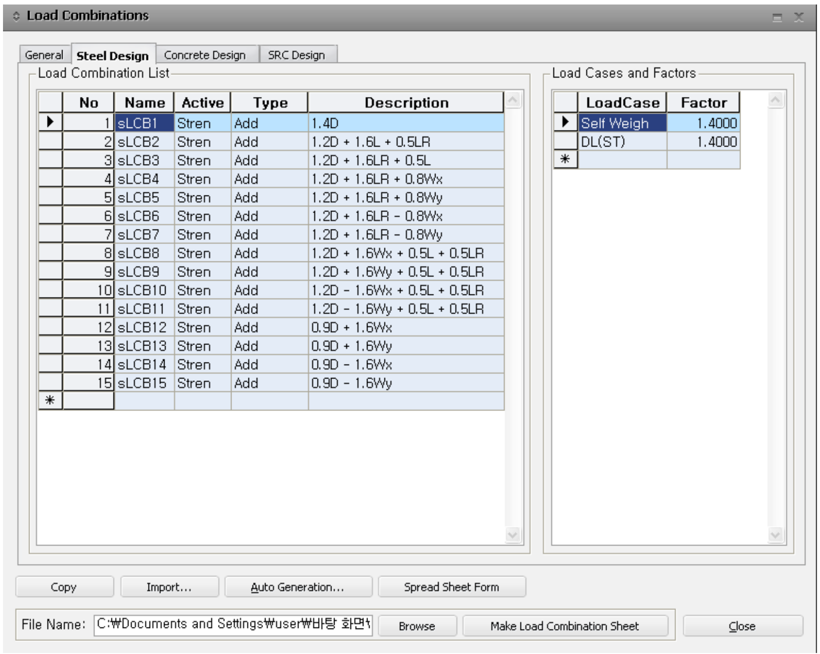


Figure 2.28 Load Combinations

Verify Reactions

1. Select ***Results>Reactions>Reaction Forces/Moments*** in the Main Menu.
2. Select "**CBS: sLCB1**" in the ***Load Cases/Combinations*** selection field.
3. Select "**FZ**" in the ***Components*** selection field.
4. Check (✓) **Values** and **Legend** in the ***Type of Display*** selection field.
5. Click Apply

It will be more convenient to place the Result Toolbar in the post-processing mode. Refer to Tutorial 1 for the placing of Result Toolbar.

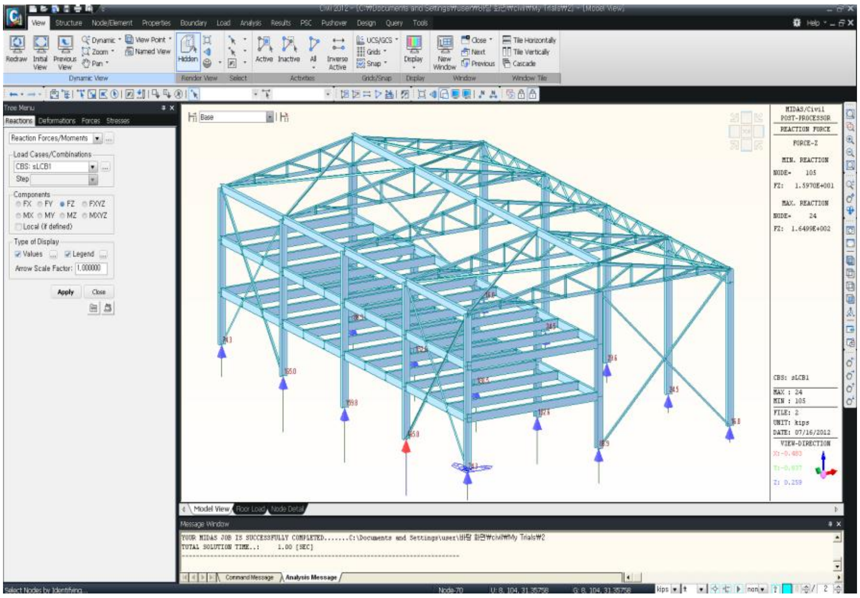


Figure 2.29 Reactions due to Vertical Loads

Displacement Contour

1. Select **Deformations** in the post-processing functions tab (Fig.2.30–❶).
2. Select ***Displacement Contour*** in the functions selection field.
3. Select "**CBS: sLCB8**" in the ***Load Cases/Combinations*** selection field.
4. Confirm "**DXYZ**" in the ***Components*** selection field.
5. Check (✓) **Contour**, **Value**, **Deform** and **Legend** in the ***Type of Display*** selection field.
6. Click the button ... on the right of ***Contour***.
7. Select "**18**" in the ***Number of Colors*** selection field.
8. Check (✓) "**Gradient Fill**".
9. Click "**Apply upon OK**" to remove the check (✓).
10. Click OK.
11. Click the button ... to the right of ***Deform***.
12. Select "**Real Deform**" in the ***Deformation*** selection field and click OK.
13. Select "**in**" in the unit conversion window of ***Status Bar*** and click Apply.
14. Click ***Render View*** in ***View*** > ***Render View*** from the Main Menu.

With the appropriate use of ***Render View*** and ***Perspective*** in the post-processing stage, diversified and interesting displays may be viewed on the screen.

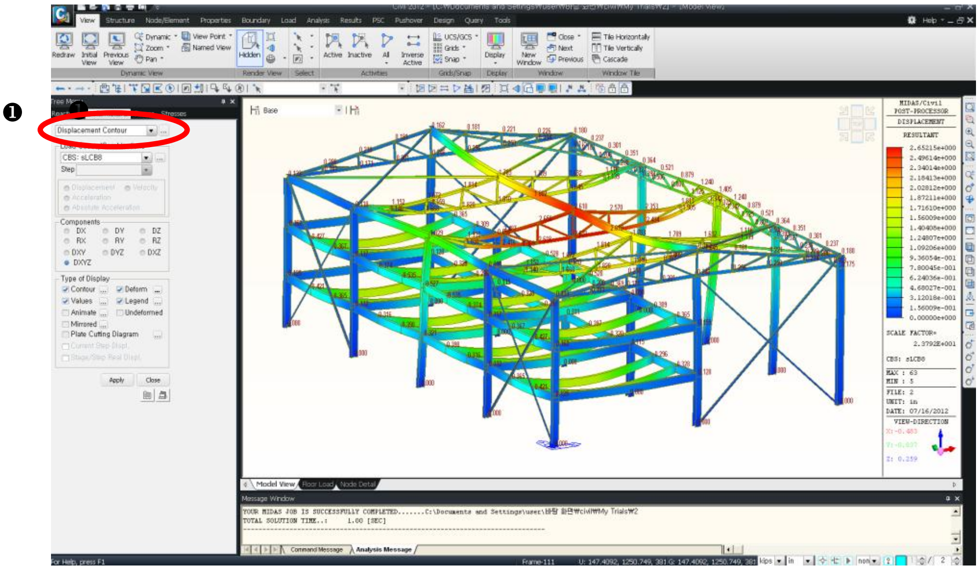


Figure 2.30 Contour of the Deformed Shape

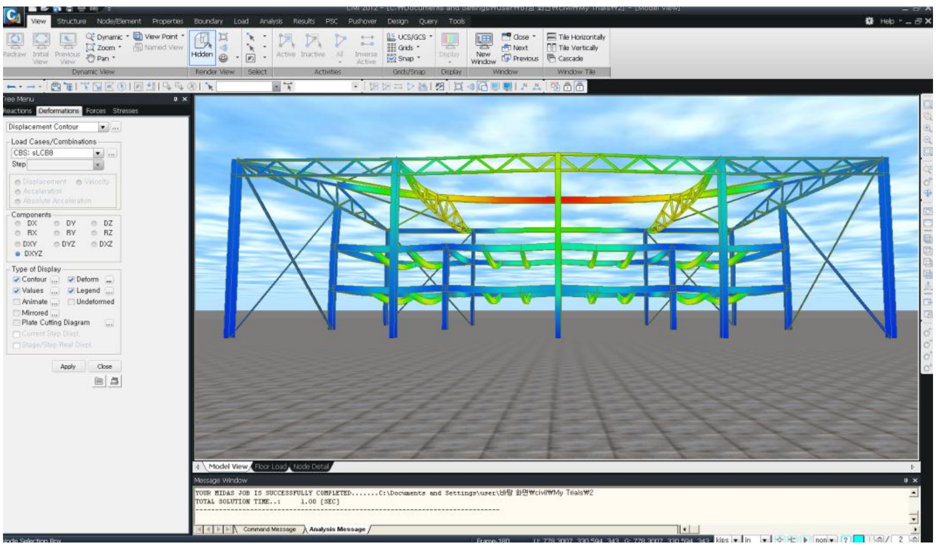


Figure 2.31 Rendering View of the Deformed Shape

1. Click ***Render View*** in ***View*** > ***Render View*** from the Main Menu (Toggle off)
2. Select ***Forces*** in the post-processing functions tab (Fig.2.32–❶).
3. Select ***Beam Diagrams*** in the functions selection field.
4. Confirm "**Myz**" in the ***Components*** selection field.
5. Select "**Exact**" and "**Line Fill**" in the ***Display Options*** selection field.
6. Check (✓) **Contour**, **Values** and **Legend** in ***Type of Display***.
7. Click the button ... to the right of ***Values***.
8. Enter "**1**" in ***Decimal Points*** and click OK.
9. Change the unit from 'in' to 'ft' (Fig.2.32–❷).
10. Click Apply.

If there are many elements as is the case shown in Fig.2.32, the verification of bending moments from the diagram becomes next to impossible. In such a case use ***Select Plane*** to selectively ***activate*** only the plane of interest.

1. Click ***Plane*** in View > Select > Plane from the Main Menu.
2. Select "**XZ Plane**".
3. Click any point on plane Ⓐ to enter "**0**" automatically in the ***Y Position*** field.
4. Click Close in the ***Plane & Volume Select*** dialog box.
5. Click ***Active*** in the Icon Menu.
6. Click ***Front View*** in the Icon Menu.

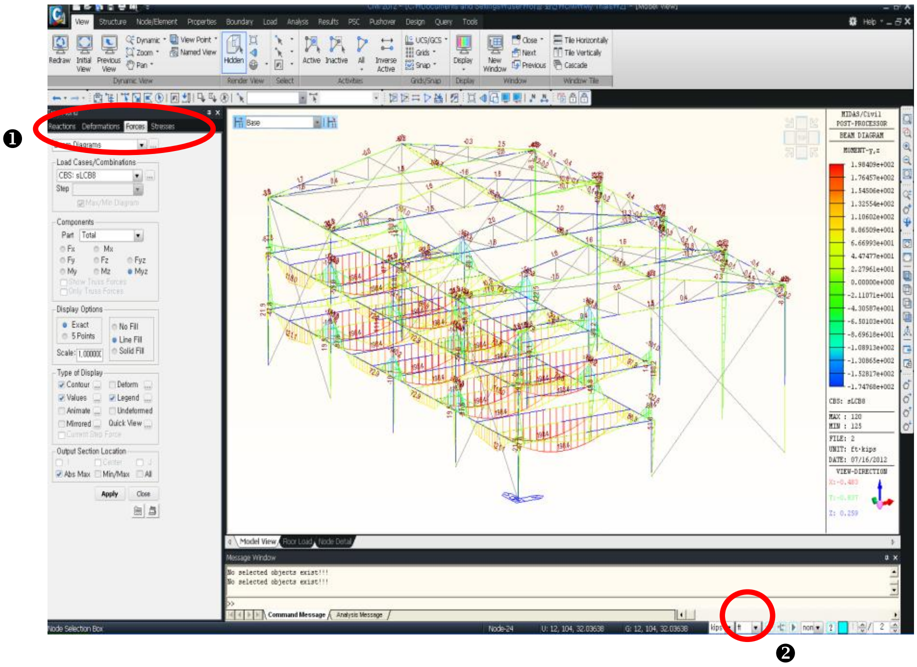


Figure 2.32 Bending Moment Diagram of the Total Model

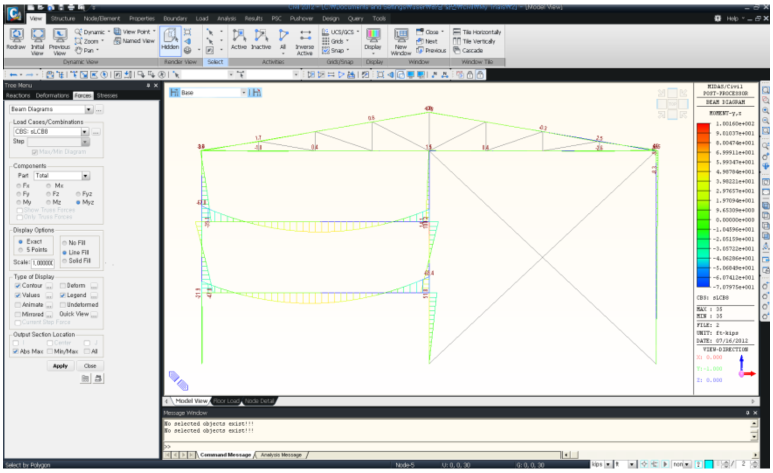


Figure 2.33 Bending Moment Diagram of Grid ① plane of the Structure (Fig.2.1)

Verify Member Stresses and Process Animation

1. Click ***Iso View*** in the Icon Menu.
2. Click ***Perspective*** (Toggle on) in the Icon Menu.
3. Click ***Active All*** in the Icon Menu.
4. Select the ***Stresses*** in the post-processing functions tab (Fig.2.34–❶).
5. Select ***Beam Stresses*** in the functions selection field.
6. Select "**CBS: sLCB8**" in the ***Load Cases/Combinations*** selection field.
7. Confirm "**Combined**" in the ***Components*** selection field.
8. Check (✓) **Contour**, **Deform** and **Legend** in ***Type of Display***.
9. Select "**in**" in the unit conversion window.
10. Click Apply

1. Check (✓) **Animate** in ***Type of Display*** (Fig.2.36) and remove the check in **Legend**.
2. Click Apply
3. Click ***Record*** as shown in Fig.2.35–❶.

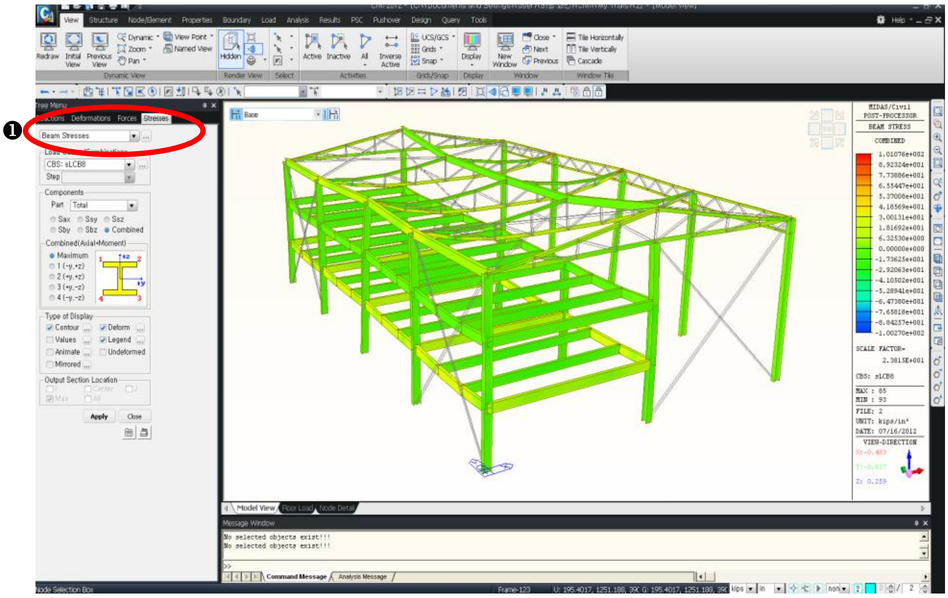


Figure 2.34 Combined Stress Diagram of Beams

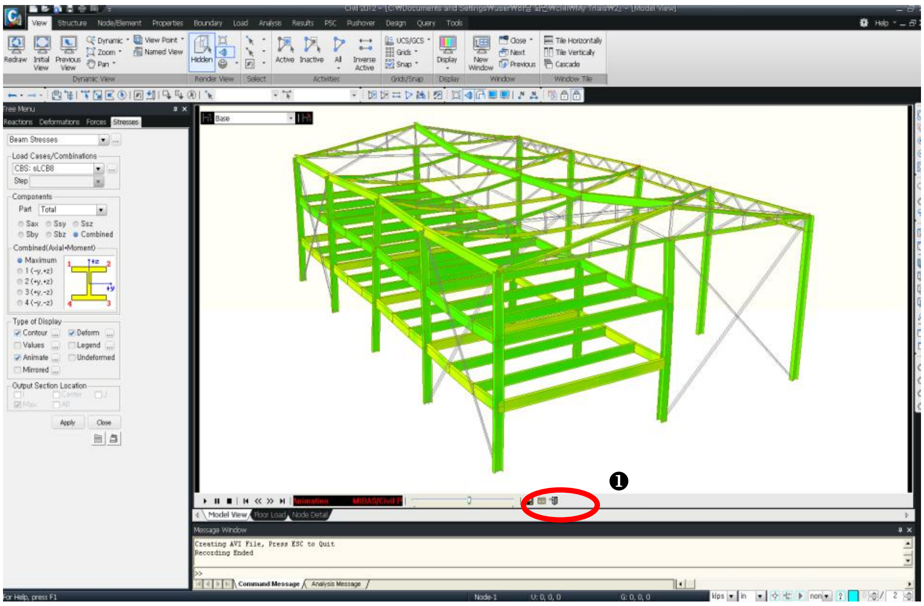


Figure 2.35 Animation Window

As explained in ***Tutorial 1*** and ***Tutorial 2***, the structural analysis results may be verified by ***Graphic*** window or by text format using ***Text Output***.

Use ***Results>Text Output*** in the Main Menu to execute ***Text Output***. ***Text Output*** is organized such that the user may directly select the ***Load Set*** (Load Combinations for output), the output contents (reaction, displacement, member force, stress, etc.), the entities to output (type of element, element number, section number, material property number, story, etc.) and the output format (maximum/minimum values by sectional properties, etc.).

The output file (fn.ANL) may be printed by the ***Text Editor*** in **midas Civil,** which enables the user to add appropriate headings and footings as necessary for documentation.

Refer to ***On-line Manual*** and ***Application Examples*** or "***Tutorial 2. Animation***" of the Install CD for details on Text Output options.