

# **Tutorial 1**

---

**3-D SIMPLE 2-BAY FRAME**

Civil

# TUTORIAL 1. 3-D SIMPLE 2-BAY FRAME

## **Summary 1**

Analysis Model and Load Cases 2

## **File Opening and Preferences Setting 3**

Unit System 3

Menu System 4

Coordinate Systems and Grids 6

## **Enter Material and Section Properties 8**

## **Structural Modeling Using Nodes and Elements 11**

## **Enter Structure Support Conditions 18**

## **Enter Loading Data 20**

Define Load Cases 20

Define Self Weight 21

Define Floor Loads 21

Define Nodal Loads 23

Define UniformlyDistributed Loads 24

## **Perform Structural Analysis 28**

## **Verify and Interpret Analysis Results 29**

Mode 29

Load Combinations 30

Verify Reactions 32

Verify Deformed Shape and Displacements 35

Verify Member Forces 39

Shear Force and Bending Moment Diagrams 40

Verify Analysis Results for Elements 44

Verify Member Stresses and Manipulate Animation 46

Beam Detail Analysis 50

# TUTORIAL 1.

## 3-D SIMPLE 2-BAY FRAME

### Summary

This example is for those who never had an access to Midas Civil previously. Follow all of the steps from the modeling to the interpretation of analysis results for a 3-D simple 2-bay frame to get acquainted with the process.

This chapter is designed to familiarize the new user with the Midas Civil environment and to become acquainted with the procedure for using Midas Civil within a very short time frame. The user will be introduced easily to Midas Civil after practicing the program by following the tutorial.

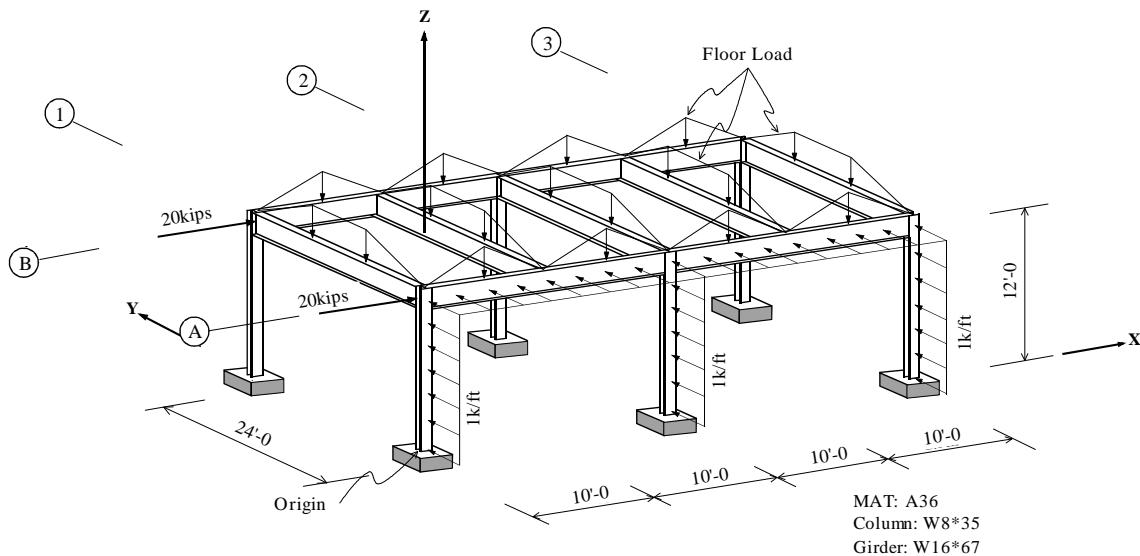
The step-by-step analysis process presented in this example is generally applicable in practice. The contents are as follows:

- 
1. File Opening and Preferences Setting
  2. Enter Material and Section Properties
  3. Structural Modeling Using Nodes and Elements
  4. Enter Structure Support Conditions
  5. Enter Loading Data
  6. Perform Structural Analysis
  7. Verify and Interpret Analysis Results
-

## Analysis Model and Load Cases

The structural shape and members used in the 3-D simple 2-bay frame are shown in Fig. 1.1. To simplify the example, consider the following 4 load cases.

- Load Case 1 – Floor load, 0.1 ksf applied to the roof and Self weight
- Load Case 2 – Live load, 0.05 ksf applied to the roof
- Load Case 3 – Concentrated loads, 20 kips applied to grids ①/① and ②/② in the (+X) direction
- Load Case 4 – Uniformly distributed load, 1k/f applied to all the members on grid ③ in the (+Y) direction



**Figure 1.1 3-D Simple 2-Bay Frame**

## File Opening and Preferences Setting

First, double-click the **Midas Civil** icon  in the relevant directory or on the background screen.

Select **File>New Project** on the top of the screen (or  ) to start the task. Select **File>Save** (or  ) to assign a file name and save the work.

### Unit System

**Midas Civil** allows a mixed use of different types of units. A single unit system may be used (example: SI unit system, i.e., m, N, kg, Pa) or a combined unit system may also be used (example: m, kN, lb, kgf/mm<sup>2</sup>). In addition, since the unit system can be optionally changed to suit the data type, the user may use “ft” for the geometry modeling and “in” for the section data. **The user can change the unit system by selecting the unit system change menu at the bottom of the screen (or Tools>Unit System from the Main Menu).** Even if the analysis has been performed in “kip” and “ft”, the units adopted for the stress results from the analysis can be converted to “ksf”.

#### Icon Menu

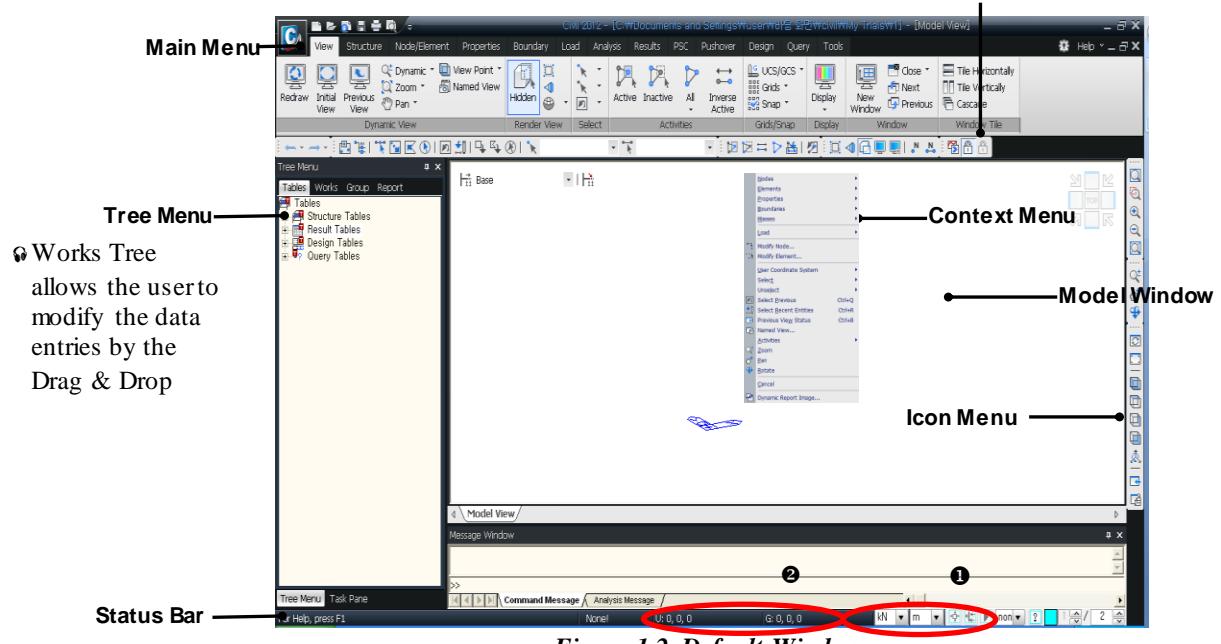


Figure 1.2 Default Window

The data input window and the unit display at the bottom of the screen (Status Bar – Fig. 1.2–❶) indicate the unit system in use and this reduces the possibility of errors. In this example, “ft” and “kip” units are used.

1. Select **Tools>Unit System** from 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**.

 The Toggle on/off status of the icon depends on the initial setting of Civil. It is advisable to toggle on the icons suggested in this tutorial to avoid any error.

Toggle on 

## Menu System

Midas Civil creates an optimal working environment and supplies the following 4 types of menu system for easy access to various features:

- Main Menu
- Tree Menu
- Icon Menu
- Context Menu

The Main Menu is a type commonly adopted in the Windows environment. It consists of Sub Menus that may be selected from the top of the screen.

The Tree Menu is located on the left of the Model Window. The menu has been organized systematically in a tree structure sequential to real problems. It presents the step-by-step order from the analysis to the design processes. This menu has been designed so that even novices can easily complete the analysis tasks just by following the sequence of the tree.

**Works Tree** displays all the input process in the form of hierarchical structure for easy recognition. Using the relevant categories, the modeling data can be entered or modified via **Drag & Drop**, in conjunction with the effective use of **Select** and **Activity**.

The Icon Menu represents the functions that are frequently used during modeling (all types of Model View or Selection).

The Context Menu has been designed to minimize the motion of the mouse on the screen. The user can access the frequently used menu simply by right-clicking the mouse at the current position.

The present example uses mainly the Main Menu, Tree Menu and Icon Menu.

## Coordinate Systems and Grids

For easy data entering, midas Civil provides NCS (Node local Coordinate System) and UCS (User Coordinate System) in addition to GCS (Global Coordinate System) and ECS (Element Coordinate System).

• In all dialog boxes, GCS is denoted by capital letters (X, Y, Z), and UCS and ECS are denoted by lower case letters (x, y, z).

GCS is the basic coordinate system that is used to define the entire geometric shape of the structure.

ECS is a coordinate system attributed to each element to reflect the element characteristics and is designed to readily verify the analysis results.

NCS is used to assign local boundary conditions or forced displacements in a specific direction to particular nodes linked to truss elements, tension-only elements, compression-only elements or beamelements.

UCS represents a coordinate system assigned additionally to GCS to simplify the modeling of complex shapes.

The coordinates of the nodes, grids and mouse cursor relative to GCS and UCS are displayed in the Status Bar (Fig.1.2–②).

Generally, structures modeled in practice are complex 3-D shapes. Therefore, it is convenient to work by setting 2-D planes to enter the basic shape data during the initial modeling stage.

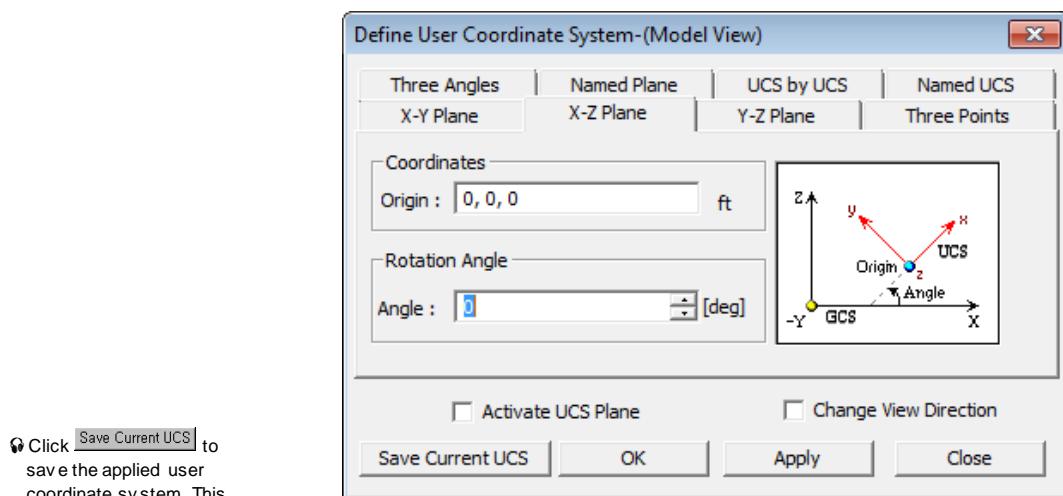
For complicatedly shaped structures, it is most efficient to assign the relevant planes as UCS x-y planes and lay out the **Point Grid** or **Line Grid** with **Snap**.

The structure in question is simple enough not to use Grid for element generation. However, UCS and **Grid** are used in this example in order to demonstrate the concept of the coordinate systems and Grid.

UCS is not set by default. Assign the GCS X-Z plane containing the grid A as UCS x-y plane to enter the 3 columns and 2 beams of the structure (Fig.1.1), by using **the Structure menu > UCS > X-Z**.

1. Click **X-Z** in **the Structure menu > UCS > X-Z**.
2. Confirm “**0,0,0**” in the **Origin** field.
3. Confirm “**0**” in the **Angle** field.
4. Click **OK**.

Click View > Snap, toggle on



**Figure 1.4 UCS Setting**

Click **Save Current UCS** to save the applied user coordinate system. This can be recalled at a later point as necessary when a number of UCS are interactively used.

For easy modeling, point grid is set with 2 ft interval in UCS x-y plane.

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

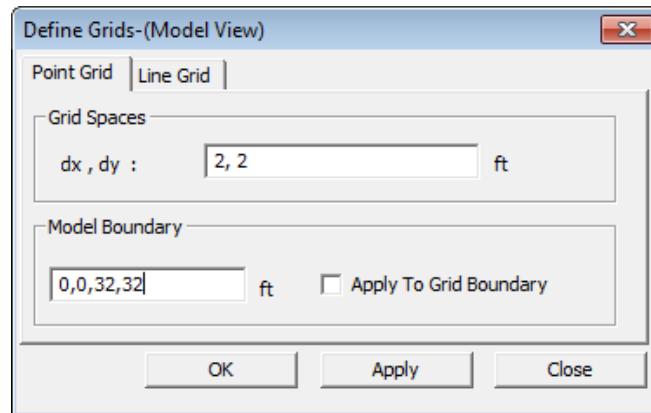


Figure 1.5 Point Grids Setting

When Civil is activated for the first time the default Grid Snap is automatically toggled on for user convenience. If Grid Snap is already toggled on it is not necessary to click it again.

**View Point** of the current window has been set to **Iso View**. Switch to **Front View** (**View Menu >View Point>Front (-Y)** or **Icon Menu >Front (-Y)**) to set the vertical and horizontal directions of **Point Grid** corresponding to the model window. Then, verify if **Point Snap Grid** is toggled on to automatically assign the click point of the mouse cursor to the closest grid point during the element generation.

- 
1. Click **Front (-Y)** in the View Point icon within the View menu.
  2. Click **Point** in the Snap icon within the View menu.  
(Toggle on).
  3. Click **Line Grid Snap** and **Snap All** (Toggle off).
- 

## Enter Material and Section Properties

Enter the material and section properties for the structural members which are assumed to be as follows:

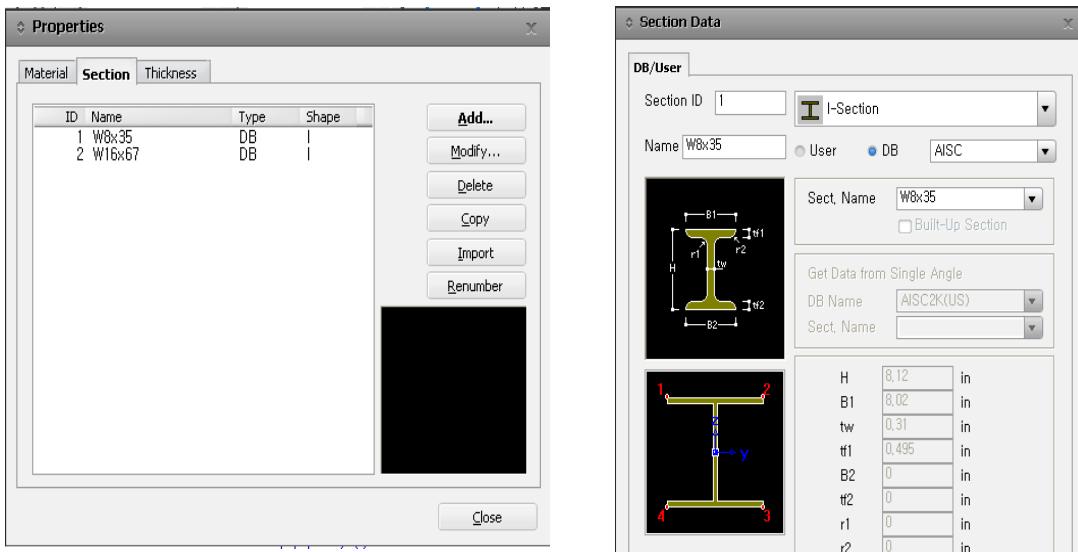
Material property ID	1: A36
Section ID	1: W8 × 35 – Columns
	2: W16 × 67 – Beams

☞ The section data can also be entered through Model>Properties>Section in Main Menu.

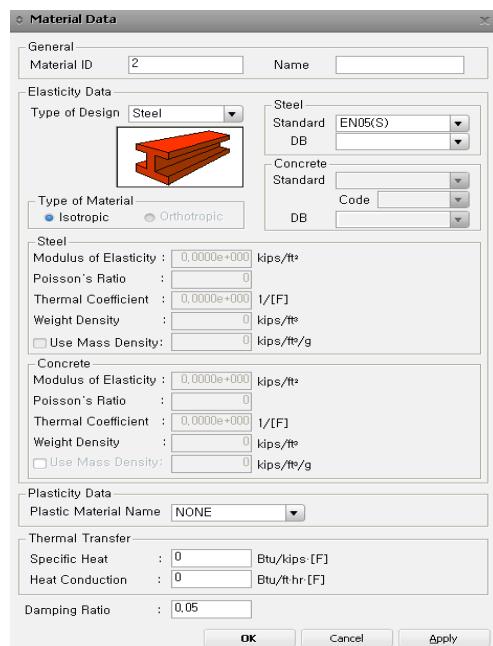
☞  closes the dialog box after completing the data entry.

 completes the data entry and prompts the dialog box to remain. Click  when section data entry is repeated.

1. Select **Properties>Material Properties** from the **main Menu**.
  2. Click  shown in Fig.1.6.
  3. Confirm “1” in the **Material Number** field of **General** (Fig.1.7).
  4. Confirm “**Steel**” in the **Type** selection field.
  5. Select “**ASTM(S)**” in the **Standard** selection field of **Steel**.
  6. Select “**A36**” from the **DB** selection field Click .
  7. Select the **Section** tab on the top of the **Properties** dialog box (Fig.1.6—①). ☞
  8. Click .
  9. Confirm the **DB/User** tab on the top of the **Section data** dialog box (Fig.1.8—②).
  10. Confirm “1” in the **Section ID** field.
  11. Confirm “**I-Section**” in the **Section** selection field.
  12. Confirm “**AISC**” in the **DB** selection field.
  13. Select “**W8 x 35**” from the **Sect. Name** selection field.
  14. Click  ☞
  15. Confirm “2” in the **Section ID** field.
  16. Select “**W16 x 67**” in the **Sect. Name** selection field.
  17. Click .
  18. Click  in the **Properties** dialog box (Fig.1.6).
-



**Figure 1.6** Dialog box for Section Properties



**Figure 1.7** Material Data

**Figure 1.8** Section Data

## Structural Modeling Using Nodes and Elements

Before entering the data for structural members, toggle on **Hidden In View > Hidden** to verify the current status of element generation and their section shapes simultaneously. If **Hidden** is toggled off, the members are displayed in Wire Frame without the section shapes.

Click **Node Number** and **Element Number** in the **Icon menu** to verify the node and element numbers.

---

The size and font of label can be adjusted by clicking **Display** Option in the **Icon Menu**.

1. Click **Hidden** (Toggle on) in the **View menu**.
2. Click **Display** in the **View Menu**, and check () **Node Number** in the **Node tab** and **Element Number** in the **Element tab** (or click **Node Number** and **Element Number** in the **Icon Menu** (Toggle on)).
3. Click **OK**.

Toggle on: under **Snap within View**  
 Under **UCS/GCS tab**  
 Under **View menu**

---

Using beam elements create the columns and beams on UCS x-y plane containing the grid ⑧ (Fig.1.1).

- 
1. Select **Node/Elements>Create Elements**.
  2. Confirm “**General Beam/Tapered Beam**” in the **Element Type** selection field.
  3. Confirm “**1: A36**” in the **Material Name** selection field.
  4. Confirm “**1: W8 × 35**” in the **Section Name** selection field.
  5. Select “**90**” in the **Beta Angle** selection field (☞Refer to Note 1).
  6. Create element **1** by clicking consecutively the positions **(0,0,0)** and **(0,12,0)** relative to UCS coordinates of **Status Bar** at the lower screen.
  7. Create element **2** by clicking consecutively the positions **(20,0,0)** and **(20,12,0)** relative to UCS.
  8. Create element **3** by clicking consecutively the positions **(40,0,0)** and **(40,12,0)** relative to UCS.
  9. Click  **Zoom Fit** in the Icon Menu.
  10. Select “**2: W16 × 67**” from the **Section Name** selection field.
  11. Select “**0**” in the **Beta Angle** selection field.
  12. Check () **Node** and **Element** in the **Intersect** selection field. ☞
  13. Create elements **4** and **5** by clicking consecutively nodes **2** and **6** with the mouse cursor.
- 

☞ In Nodal Connectivity field, the node number can be entered consecutively by placing “,” or “ ” (blank) in between the numbers.

☞ In Intersect field, if Node and Elem. are checked () and if a node already exists on the element to be created or if the element being created intersects an existing element, the newly created element is automatically divided at the intersecting points.

Generate the elements on UCS x-y plane containing the grid ⑧ by duplicating the elements already created above (Fig.1.1).

- Check () Align Top of Beam Section with Center Line (X-Y Plane) for Display in Model> Structure Type of Main Menu. Then, the effect of the beam/column panel zone will appear as ① in the bottom of Fig.1.9.

**Note 1** .....

**Beta Angle** represents the orientation of section of beam or truss elements.

In the case of columns having an I-section profile, Beta Angle has been preset to 0 where the plane of the web is parallel to the GCS X-Z plane. In this example, the plane of the column web is parallel to the GCS X-Y plane which is to be rotated by 90° by the right-hand-rule about the GCS Z-axis from the **Beta Angle = 0** position. For the beam/truss elements, Beta Angle has been preset to 0 where the plane of the web is parallel to the GCS Z-axis. Thus, all the beams in this example retain **Beta Angle = 0**.

- By setting  Auto Fitting Toggled on, Midas Civil automatically adjusts the scale. The screen fits the entire model including the newly generated elements, which eliminates the inconvenience of clicking  Zoom Fit every time.

- By switching to GCS, the position of Point Grid is automatically set to the GCS origin of the X-Y plane.

- During the data entry in an Iso View state, if Point Grid Snap is active, the node click may assign a node to a neighboring Grid Point contrary to the user's intention. To avoid visual mistakes, toggle off Grid Snap and activate Node or Element Snap.

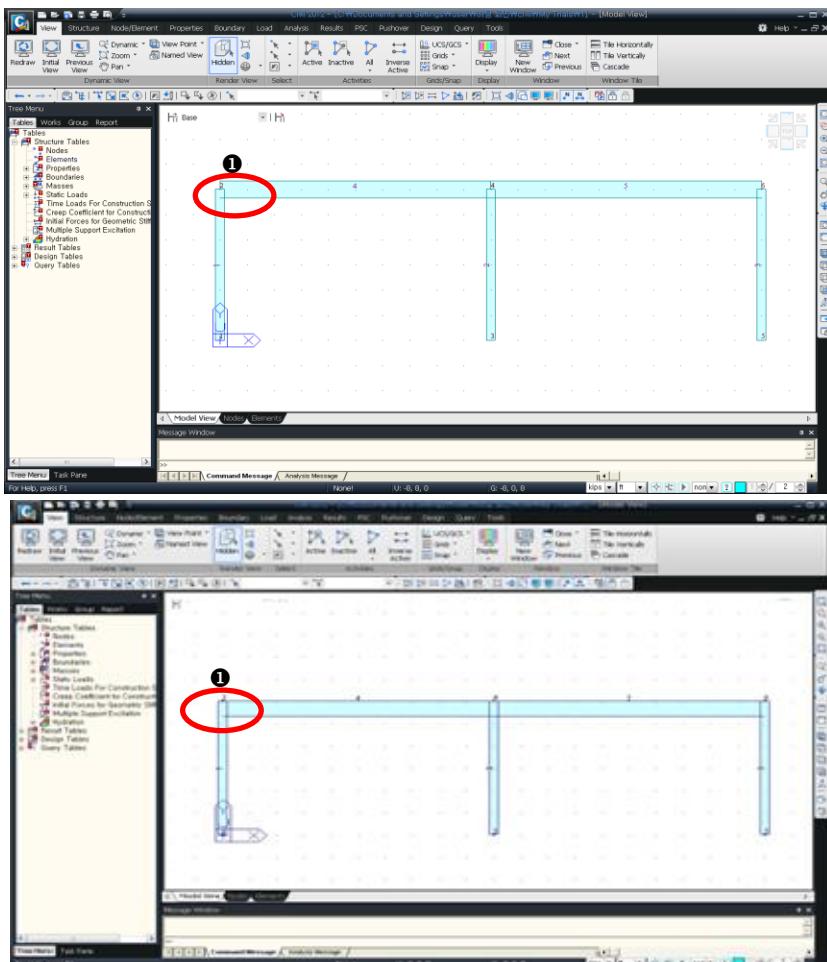


Figure 1.9 Generation of 2-D Frame

- Reference Point automatically computes Beta Angle, which is defined by specified coordinates of an arbitrary point located on the extension line of ECS z-axis.

Set the working environment to a 2-D UCS system for modeling on a plane. It may be more convenient to proceed to a 3-D model in **Iso View** state. Switch the coordinate system to GCS and select **Iso View** for **View Point**.

To define the elements to be duplicated, click  **Select All** (*View>Select>Select All* in the Main Menu). Then, duplicate the elements by Clicking **Translate with in the Elements field under the Node/Element menu.**

When switching from the current modeling function to another function, the Main Menu or Tree Menu can be used. In the case of mutually related functions (example: **Create Elements**, **Translate Elements**, etc.), midas Civil enables the user to switch directly using the functions selection field (Fig.1.10–❷).

Where the functions are remotely related or unrelated, it is recommended that the **Model Entity** tabs shown in Fig.1.10–❶ be used (**Node**, **Element**, **Boun...**, **Mass**, **Load**).

1. Click **GCS** in **View > UCS/GCS**.
2. Click **Iso** in **View > View Point**.
3. Click **Select All** in **View > Select**.
4. Select **Translate in Node/Element > Translate Elements**
5. Confirm “**Copy**” in **Mode** field.
6. Confirm “**Equal Distance**” in **Translation** field.
7. Enter “**0, 24, 0**” in the **dx, dy, dz** field (Refer to Note 2).
8. Confirm “**1**” in the **Number of Times** field.
9. Click .
10. Click **Auto Fitting** in the Icon Menu.

Instead of typing in the values for dx, dy, dz, the distance and direction of the position to be moved/duplicated can be defined with the mouse cursor using Mouse Editor (Fig.1.10-④).

Toggle on

In Fig.1.10:  
①: Model Entity tab  
②: list of related functions

dx, dy, dz are to be entered in UCS. If the UCS has not been defined, it is assumed to be identical to GCS.

Fast Query shows the attributes of the snapped nodes or elements which are off in Fig.1.10-③.

The attributes that can be verified by Fast Query are as follows:  
Node number,  
coordinates, element number, element type, material properties/section ID/thickness ID of element, Beta Angle, linked node numbers and length/area/volume of element.

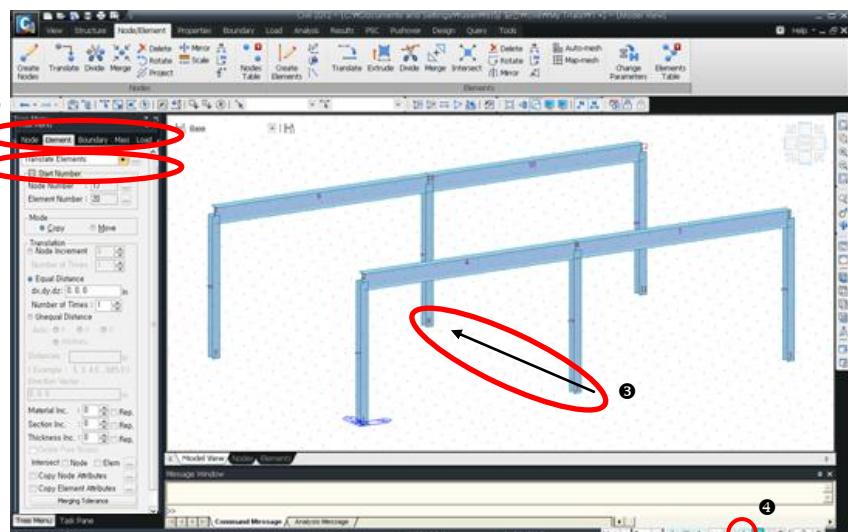


Figure 1.10 Duplication of 2-D Frame

Note 2

**Mouse Editor** is used in the copy distance field. **Mouse Editor** automatically enters the coordinates or distance when the user clicks a specific point on the working window with the mouse cursor instead of physically typing in the values. If **Mouse Editor** does not execute, click the related data entry field which turns to a pale green color and then enter the data.

Create elements for the girders on grids ①, ② and ③ of the structure (Fig.1.1).

Select **Create Elements**. To avoid any confusion between nodes and grids, toggle off  **Point Grid** and  **Point Grid Snap**.

1. Click  **Point Grid** in **View > Grid** and  **Point Grid Snap** in **View > Grid** (Toggle off).
2. Select **Create Elements** in **Node/Element > Create Elements**
3. Confirm “**General Beam/Tapered Beam**” in the **Element Type** selection field.
4. Confirm “**1: A36**” in the **Material Name** selection field.
5. Confirm “**2: W16 x 67**” in the **Section Name** selection field.
6. Confirm “**0**” in the **Beta Angle** selection field.
7. Create element **11** by extending nodes **2** and **8** with the mouse cursor.
8. Create element **12** by extending nodes **4** and **10** with the mouse cursor.
9. Create element **13** by extending nodes **6** and **12** with the mouse cursor.

Toggle on 

 Even if  Node Number is not toggled on, the attributes of snapped nodes can be easily verified using Fast Query (Fig.1.11-②).

Directly create an element for the beam located between elements 11 and 12 using **Element Snap** without entering nodes separately.

Beam end release conditions are assigned at both ends of the beam and the beam is duplicated rightward to the next bay. The subsequent task can be minimized if the beam end release conditions are duplicated simultaneously.

 Civil allows mouse snap at the centers of the elements as well as any particular point in the elements by using Snap located at the bottom of the screen (Fig.1.11-③).

1. Create element **14** by extending the centers of elements **4** and **9** with the mouse cursor. (by default, clicking the center of a member is possible)
2. Click  **Select Single** in **View > Select** and select element **14**.
3. Click  and click  **Apply** in **Boundary > Beam End Release**.
4. **Node/Element > Translate Elements**
5. Confirm “**Copy**” in **Mode** field.
6. Click **dx, dy, dz** field of **Equal Distance** once.

7. Click successively node **14** and the center of element **10** to enter “**20, 0, 0**” automatically.
8. Check () **Node** and **Elem.** of *Intersect*.
9. Check () **Copy Element Attributes** and click **...** on the right.
10. Confirm the check () in **Beam Release** of *Boundaries*.
11. Click **OK** in the *Copy Element Attributes* dialog box.
12. Click **Shrink** in the Icon Menu.
13. Click **Select Previous** to select element **14** (select previous)
14. Click **Apply** in the *Translate Elements* dialog bar.
- 15. View > Display > Boundary tab.**
16. Check () **Beam End release Symbol** and click **Apply**.

💡 If Shrink is toggled on, the linkage of members and nodes can be easily verified.

💡 By clicking the right button **...** of the function list or using Model>Nodes>Nodes Table or Model>Elements >Elements Table of Main Menu, the current status of nodes and elements can be verified and also modified.

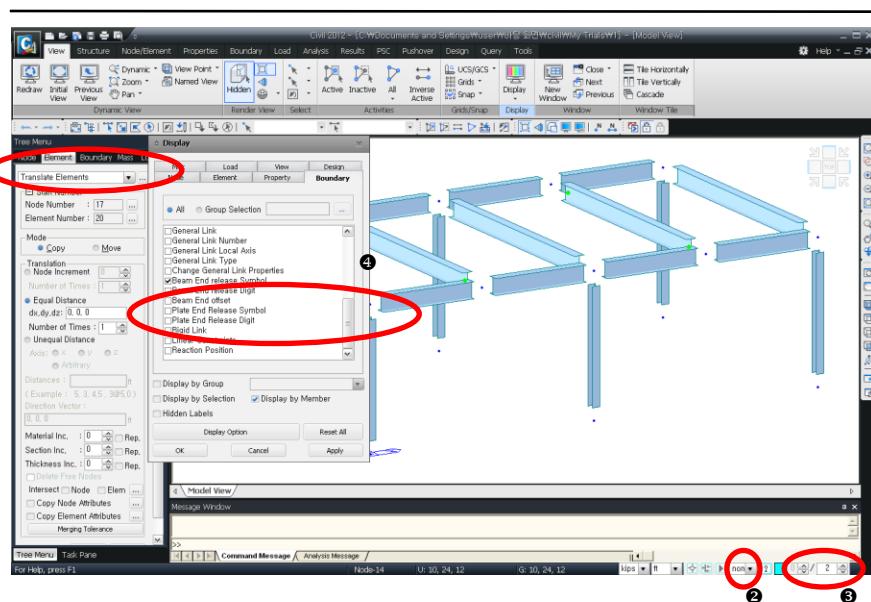


Figure 1.11 Generation of Girders and Beams

## Enter Structure Support Conditions

When the modeling of the structure shape is complete, provide the support conditions for the 6 columns.

In this example, it is assumed that the lower ends of the columns are fixed (restrain the 6 degrees-of-freedom).

Prior to defining the support conditions, select the plane that includes the lower ends of the 6 columns by **Plane** (*View>Select>Plane* from the Main Menu).

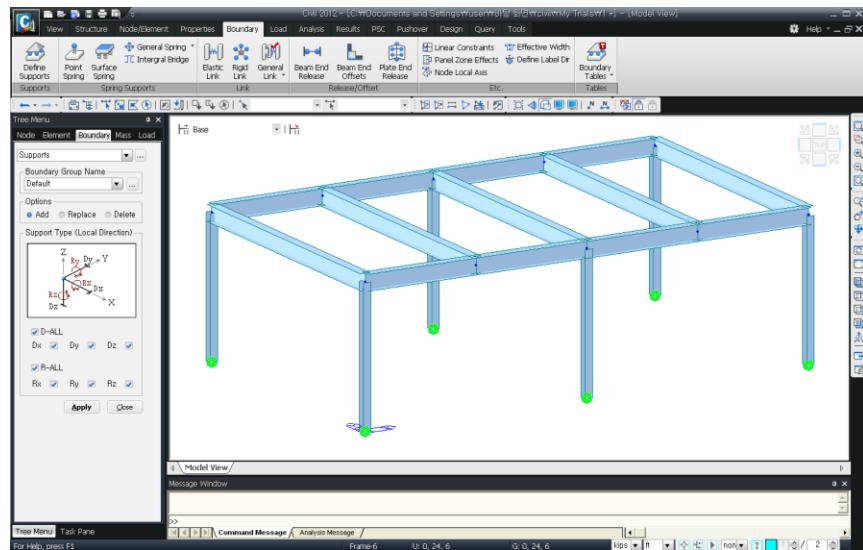
By toggling off Hidden in the Icon Menu, the selection of the nodes of the columns' lower ends can be easily verified by the change of color.

- 
1. Remove the check (✓) in **Beam End Release of Display**.
  2. Click .
  3. Click **Shrink** (Toggle off).
  4. Click **Plane** (*View>Select>Plane*)
  5. Select “**XY Plane**”.
  6. Click one node among the 6 column supports.
  7. Click .
- 

To specify the support conditions, access relevant function noted below.

- 
1. Select the **Boundary** menu (Fig.1.12–①).
  2. Select **Define Supports**.
  3. Confirm “**Add**” in the **Options** selection field.
  4. Check (✓) **D-ALL** and **R-ALL** in the **Support Type (Local Direction)** selection field.
  5. Click .
-

## Enter Structure Support Conditions



• Midas Civil supplies a variety of select functions.

- Select Identity
- Select Single
- Select Window
- Select Polygon
- Select Intersect
- Select Plane
- Select Volume
- Select All
- Select Previous
- Select Recent Entities

**Figure 1.12 Data Entry for Structure Supports**

## Enter Loading Data

### Define Load Cases

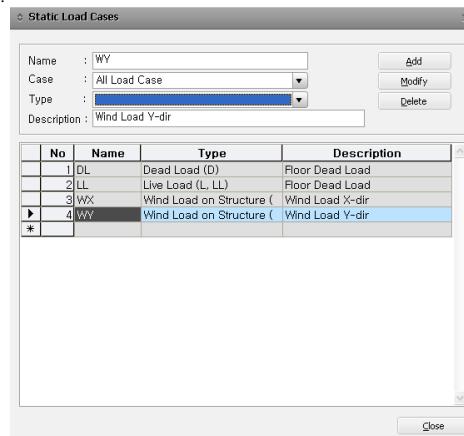
Define load cases before entering the loading data.  
Select **Load** in the menu for loading.

Click **Static Load Cases (Load>Static Load Cases)** in the Main Menu to access the **Static Load Cases** dialog box and enter the following load cases:

Note that in the new version of Midas Civil, **Case** drop-down menu has been added for easy access to the types of loads. Depending on which load case (**Case**) you choose, the options in the **Type** selection change.

1. Select **Load > Static Load Cases**
2. Enter “**DL**” in the **Name** field of the **Static Load Cases** dialog box (Fig.1.13).
3. Select “**Dead Load**” from the **Type** selection field.\*
4. Enter “**Floor Dead Load**” in the **Description** field.
5. Click **Add**.
6. Enter the remaining load cases in the **Static Load Cases** dialog box as shown in Fig.1.13.
7. Click **Close**.

\* The type of loadings (**Dead Load**, **Live Load**, **Snow Load**, etc.) selected in the **Type** selection field are used to generate automatically the load combination cases with respect to the specified design criteria assigned in the post-processing mode.



**Figure 1.13 Definition of Load Cases**

## Define Self Weight

Define the self-weight of elements.

1. Select **Self Weight** (**Main** menu > **Load** menu > **Self Weight**) or (**Tree** menu > **Load** > **Self Weight**).
2. Confirm “**DL**” in **Load Case Name**.
3. Enter “**-1**” in the **Z** field under **Self Weight Factor**.
4. Click **Add**.

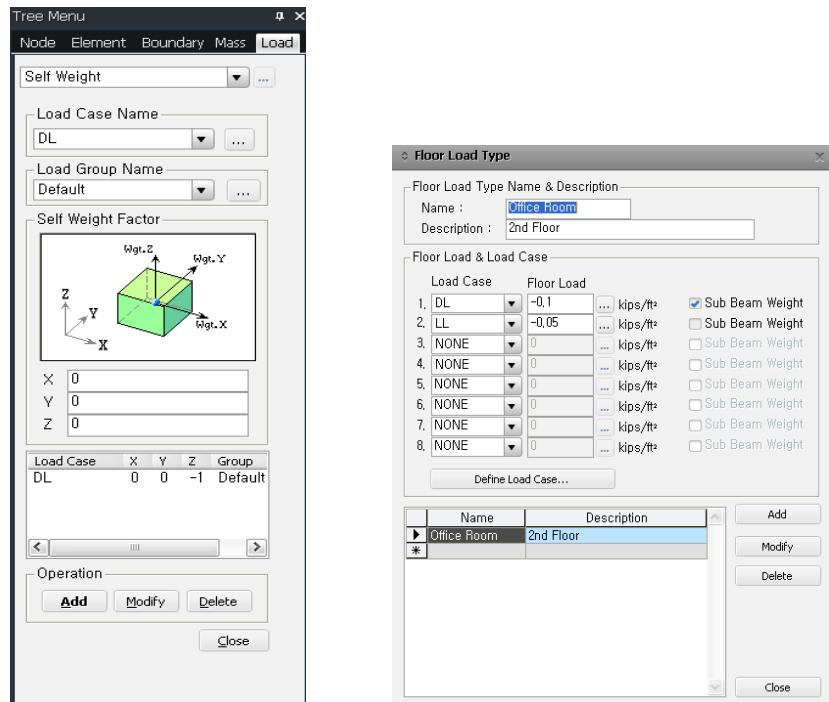


Figure 1.15 Definition of Floor Load Type

Figure 1.14 Self Weight Data

## Define Floor Loads

Select **Assign Floor Loads** in the functions selection field to enter gravity loads. To enter the floor loads, define the **Floor Load Type** first, then select the area to be loaded.

1. Select Load > **Assign Floor Loads** from the functions selection field (Fig.1.16-①).
2. Click to the left of the **Load Type** selection field.
3. Enter “**Office Room**” in the **Name** field (Fig.1.15).
4. Enter “**2nd Floor**” in the **Description** field.
5. Select “**DL**” from the **Load Case 1.** selection field and type “**- 0.1**” in the **Floor Load** field.
6. Select “**LL**” from the **Load Case 2.** selection field and type “**- 0.05**” in the **Floor Load** field.
7. Click .
8. Click .
9. Select “**Office Room**” from the **Load Type** selection field.
10. Confirm “**Two Way**” in the **Distribution** selection field.
11. Click the **Nodes Defining Loading Area** field once and the background color turns to pale green. Then click sequentially the nodes (**2, 6, 12, 8, 2**) that define the loaded area in the model window.

The Description field may be left blank.

In order to verify a nodal position on the screen, enter the node number in Query>Query Nodes of the Main Menu and click Enter. The nodal position will be displayed on the screen and its coordinates will appear in Message Window. In addition, the currently snapped node or element number will be displayed in the Status Bar.

The size of Label Symbol is adjusted in the Size tab of . The size of the displayed Load Label can be adjusted likewise.

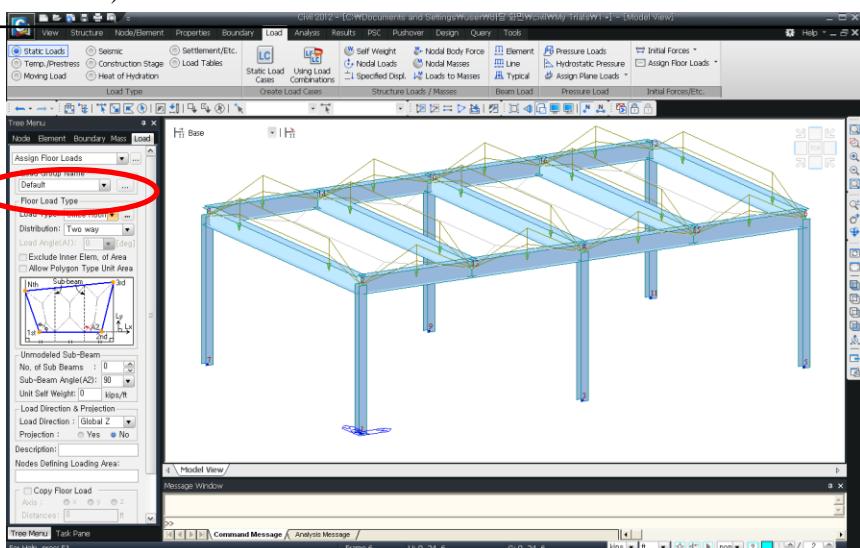


Figure 1.16 Entry of Floor Loads

## Define Nodal Loads

Enter the X-direction wind load (Load Case 3) as concentrated nodal loads.

The color of the selected nodes will change and nodes 2 and 8 can be verified in the Select-Identity Nodes in Fig.1.17-②.

1. Select **Nodal Loads** from the functions selection field. (Fig.1.17-①).
2. Click **Hidden** (Toggle off) in the View Menu.
3. Click **Single** (*View > Select > Single*) (Toggle on) in the Main Menu.
4. Select nodes **2** and **8** to apply concentrated loads with the mouse cursor.
5. Select “**WX**” from the **Load Case Name** selection field.
6. Confirm “**Add**” in the **Options** selection field.
7. Enter “**20**” in the **FX** field.
8. Click **Apply**.

Toggle on

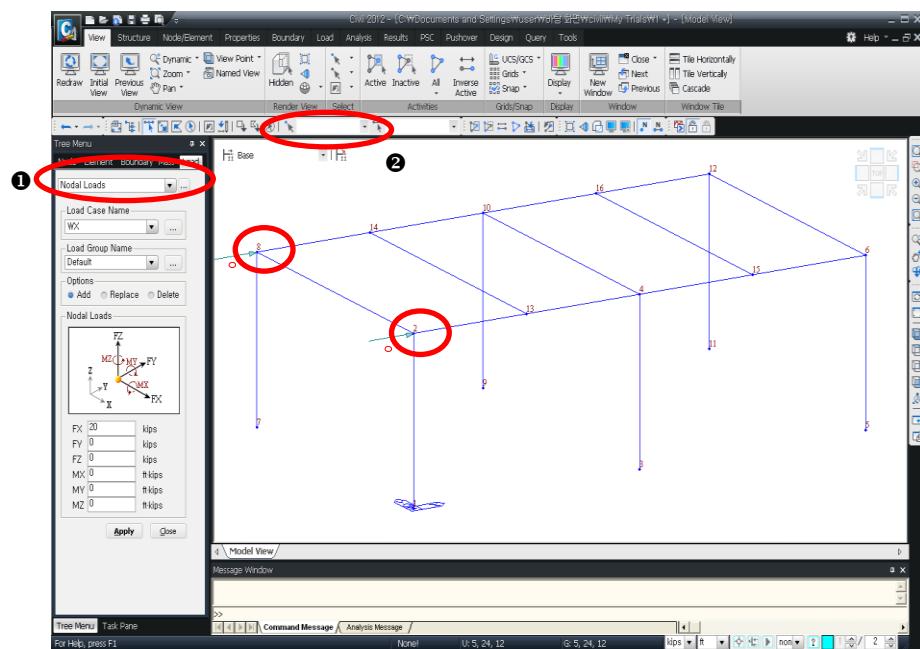


Figure 1.17 Entry of X-Direction Wind Load

## Define Uniformly Distributed Loads

Enter Y-direction wind load (Load Case 4) as Element Beam Load.

1. Click **Plane**(View > Select > Plane)  in the Main Menu.
2. Select “**XZ Plane**”.
3. Click one point in grid ① (Fig.1.1). 
4. Click **Close**.
5. Select **Element Beam Loads** from the functions selection field (Fig.1.18–①).
6. Select “**WY**” from 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 Y**” from the **Direction** selection field.
10. Confirm “**No**” in the **Projection** selection field.
11. Enter “**1.0**” in the **w** field.
12. Click **Apply**.

 After selecting relevant elements, all the data related to these elements can be verified by executing Query>Element Detail Table.

Element Detail Table allows the user to verify if there are any duplicating errors.

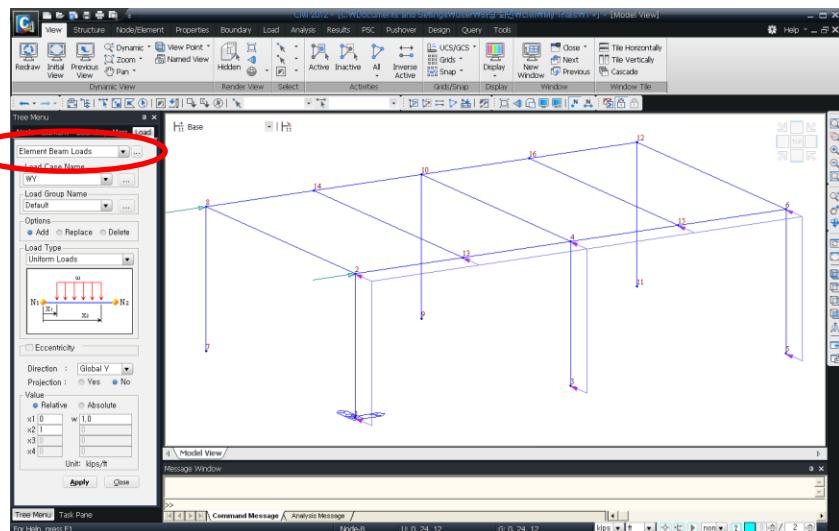


Figure 1.18 Entry of Y-Direction Wind Load

For easy reference, Midas Civil automatically displays the label for the latest data entry regardless of the user-selected display item. Such a label is automatically removed from the model window upon execution of subsequent data entry or a different display command.

Before analyzing the structure, change the **Display** status assigned during the modeling by the following procedure:

- 
1. Click **Display** (*View > Display*), select the **Node** tab and remove the check (✓) in **Node Number** (or click (Toggle off)).
  2. Select the **Element** tab and remove the check (✓) in **Element Number** (or click (Toggle off)).
  3. Click .
  4. Click in the **Element Beam Loads** dialog box.
  5. Select the **Works** tab in the Tree Menu.
-

Works Tree categorizes the entire model data entered up to now, which allows the user to glance through the modeling process. The Context Menu of **Works Tree** and the **Drag & Drop** method may be utilized to modify the current data or certain attributes.

At this point, we will examine the process of revising the column section dimension.

The Context Menu of Works Tree enables the user to access such functions as Assign, Select, Activity, Delete and Properties.

1. Click **Hidden**.
2. Under the **Properties>Section of Works Tree**, place the mouse cursor over “**1: W8x35**” and then right-click the mouse to select **Properties**.
3. Select “**W 36x300**” in the Section Name selection field.
4. Click **OK**.

The display on the model window reflects the change in section shapes and sizes if the section data are revised.

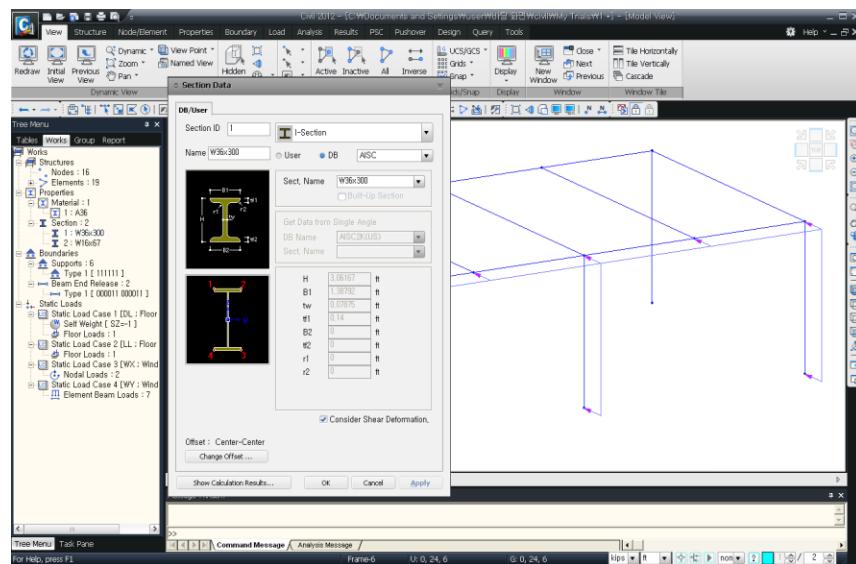


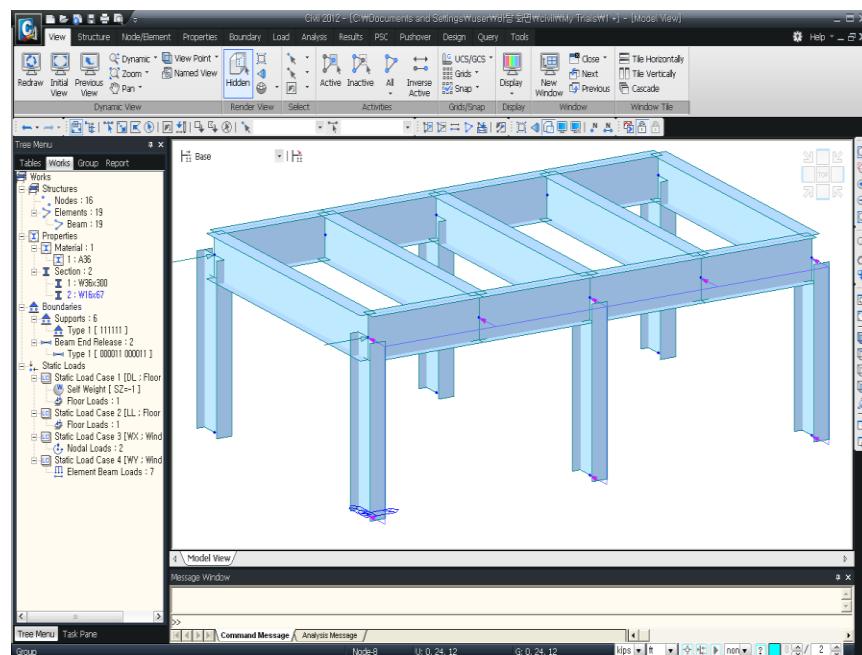
Figure 1.19 Section data revision using Works Tree

Next, we will demonstrate the procedure of modifying the model data using the **Drag & Drop** method provided by **Works Tree**.

ⓘ The color change of section number 2 into blue signifies that the section is not assigned to any one of the elements.

ⓘ Fast Query occurs when cursor is placed on a certain element or node. It contains detailed information of node or element.

1. Under the **Properties>Section** of Works Tree double-click “**2: W16x67**” to select the beam elements.
2. From the section drag “**1: W36x300**” with the mouse left-clicked to the model window.
3. Notice the change of beam dimensions in the model window. ⓘ
4. Using the **Fast Query** ⓘ, we can confirm that the section number for the element 11 is changed to “**1**”.
5. Click ⓘ Undo List to the right of ⓘ Undo .
6. Select “**5. Modify Section**” to select all the items from 1 to 5.
7. Click ⓘ Undo .



**Figure 1.20 Change of model by Drag & Drop**

## Perform Structural Analysis

Select **Analysis>Perform Analysis** from the Main Menu to analyze the model with the load cases defined previously.

Since only **Linear Static Analysis** is carried out in the present example, no additional analysis data are required.

Once the structural analysis begins, the dialog box signaling the execution appears in the middle of the screen as shown in Fig.1.21. The overall analysis process, including the formation of the element stiffness matrix and the assembling process, is displayed step-by-step in the **Analysis Message Window** at the bottom of the screen (Fig.1.21–❶).

When the analysis is completed, the total time used for the analysis is displayed on the screen and the dialog box in the middle disappears.

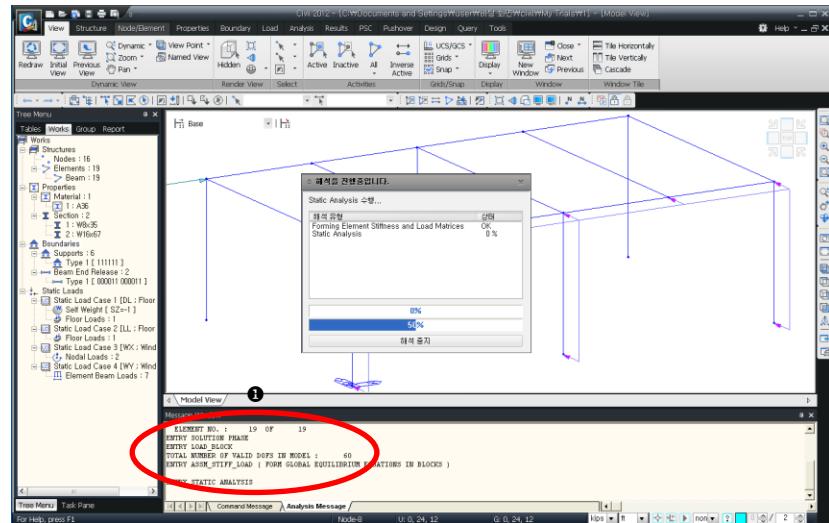


Figure 1.21 Execution Process of Structural Analysis

## Verify and Interpret Analysis Results

### Mode

For the sake of efficiency and convenience, midas Civil classifies the program environment into ***preprocessing mode*** and ***post-processing mode***.

All the data entry pertaining to the modeling is feasible only in the preprocessing mode. The interpretation of analysis results such as reactions, displacements, member forces, stresses, etc., is possible only in the post-processing mode.

In the analysis process, if the analysis is completed without any error, the ***Mode*** automatically switches from the preprocessing mode to the post-processing mode. Verification or modification/change of a part of the data can only be done in the preprocessing mode. Click ***Preprocessing Mode*** in the Icon Menu to revert to preprocessing mode.

 Be aware that the existing analysis results will be deleted if the data are altered after converting from post-processing mode to preprocessing mode.

Midas Civil supports the following post-processing functions for the verification of linear static analysis results.

- Extraction of maximum/minimum values (***Envelope***) of ***Load Combinations*** and grouped load combination cases
- Reactions verification, ***Search*** functions and ***Reaction Plots***
- Displacements verification, ***Search*** functions and deformation plots such as ***Deformed Shape*** and ***Displacement Contour***
- Member force plots such as ***Element Forces Contour***, ***BMD*** and ***SFD***
- Stress plots (***Element Stresses Contour***)
- Detail analysis results for beam elements (***Beam Detail Analysis***)
- Detail analysis results for individual elements (***Element Detail Results***)
- Calculation of member forces in a particular direction based on the nodal forces in plate or solid elements (***Local Direction Force Sum***)
- Spreadsheet tables related to the analysis results such as reactions, displacements, member forces, stresses, etc.
- Summarized or combined analysis results specified by the user in ***Text Output*** format

## Load Combinations

Static analysis has been performed for the 4 unit load cases, “DL”, “LL”, “WX” and “WY”, entered in the preprocessing step. The Linear Load Combinations of these 4 analyzed unit load cases are now examined.

Load combinations can also be defined in the post-processing mode in Midas Civil.

Specifying load combinations in the post-processing mode is efficient because the results are produced through a linear combination process on the basis of each unit load case.

☛ The load combinations for structural design can be auto-generated by selecting a design standard.

- Load Combination 1 (LCB1):                  1.0 DL + 1.0 LL
- Load Combination 2 (LCB2):                  1.2 DL + 0.5 LL + 1.3 WY

The load combination data are entered through the **Load Combinations** dialog box (Fig.1.22) in **Results>Load Combinations** in the Main Menu.

☛ ST stands for Static Load

☛ “1.0” is the default value in the Factor field.

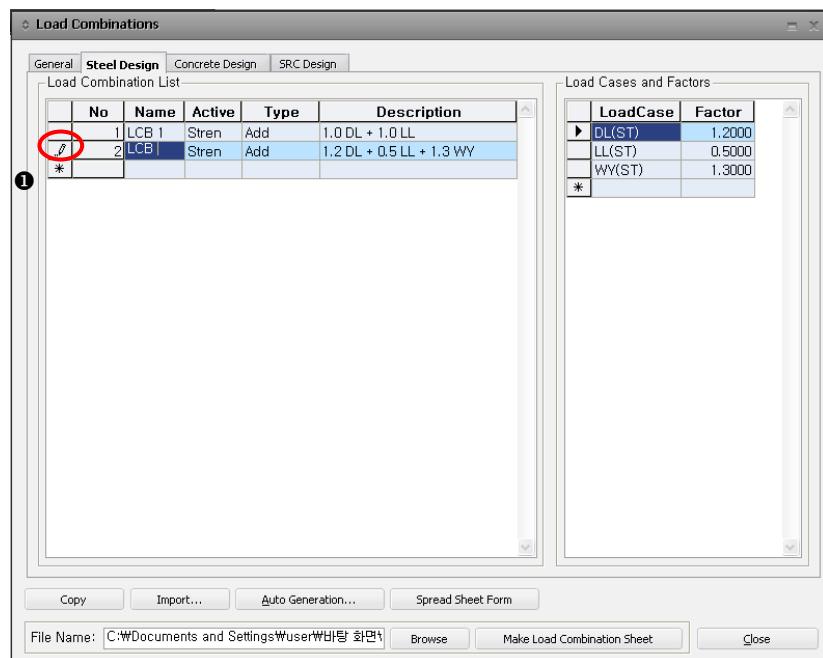
☛ When data entries are carried out in table, the symbol (Fig.1.22-①) has to disappear to complete the input. Select another cell to eliminate the ‘Edit-in – progress’ symbol and click Close.

1. Select **Results>Load Combinations** from the Main Menu.
2. Select **Steel Design** tab.
3. Type “**LCB1**” (Load Combination 1) in the **Name** field of **Load Combination List**.
4. Select “**Strength/Stress**” in the **Active** field.
5. Enter “**1.0 DL + 1.0 LL**” in the **Description** field.
6. Click the **Load Case** selection field of **Load cases and Factors**. Then, select “**DL(ST)**”. ☛
7. Confirm “**1.0**” in the **Factor** field. ☛
8. Select “**LL(ST)**” from the second line of the **Load Case** field.
9. Type “**LCB2**” in the second line of the **Name** field of **Load Combination List**.
10. Select “**Strength/Stress**” in the **Active** field.
11. Enter “**1.2 DL + 0.5 LL + 1.3 WY**” in the **Description** field.
12. Select “**DL(ST)**” from the **Load Case** selection field of **Load cases and Factors**.
13. Type “**1.2**” in the **Factor** field.

14. Similarly, enter “**LL(ST)**” and “**WY(ST)**” and the factors “**0.5**” and “**1.3**” respectively.

15. Click **Close**.

When data entries are carried out in table, the symbol(Fig.1.22-❶) has to disappear to complete the input. Select another cell to eliminate the ‘Edit-in – progress’ symbol and click **Close**.



**Figure 1.22 Load Combination Cases**

## Verify Reactions

To verify the reaction results at all the supports after the analysis, select **Results>Reactions>Reaction Forces/Moments** from the Main Menu and follow the steps below.

1. Click **Hidden** (Toggle on) in the Icon Menu.
2. Select **Results>Reactions>Reaction Forces/Moments** from the **Menu** tab of the Main Menu.
3. Select “**CBS:LCB1**” (Load Combination 1) from the **Load Cases / Combinations** selection field.<sup>6</sup>
4. Select “**FZ**” from the **Components** selection field.
5. Check () **Values** and **Legend** in the **Type of Display** selection field.
6. Click **Apply**.<sup>6</sup>

DS stands for the load combination cases produced from the Steel Design tab.

The decimal points of the reactions displayed on the screen can be adjusted by clicking on the right of Values in Type of Display. The part in red represents the support where the maximum reaction occurs.

By selecting Local Value (if defined) in Type of Display, nodal reactions are displayed in local axes if Node Local Axis has been attributed to the node.

Because the model shape is simple enough, the verification of reactions for the entire model is relatively easy. However, for a model with a complex geometric shape, the verification of reactions with the entire model is fairly cumbersome. It may be necessary to verify reactions selectively only at specific supports.

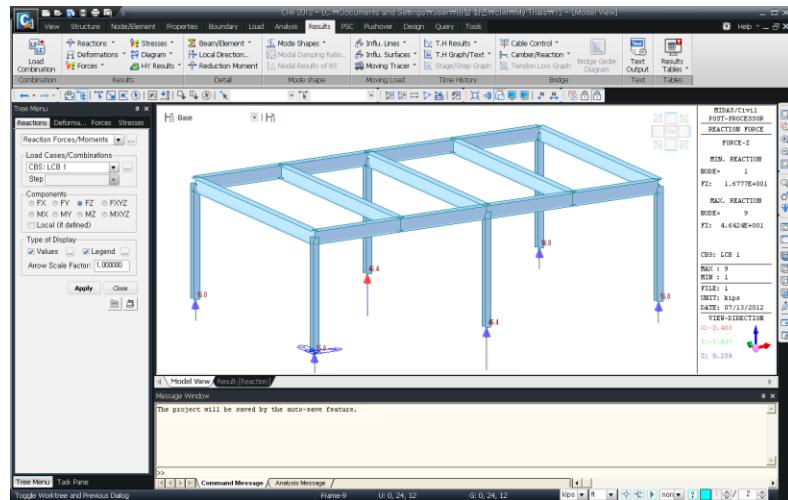


Figure 1.23 Reaction Forces

Now the method of selective verification of the reaction forces at specific supports is examined.

To easily select particular nodes, click **N Node Number** to display the node numbers on the screen.

1. Select **Search Reaction Forces/Moments** from the functions field (Fig.1.24-①).
2. Click **N Node Number** (Toggle on) in the Icon Menu
3. Click the **Node Number** field once.
4. Select nodes **1** and **3** with the mouse.

By clicking the desired node with the mouse, the reaction values in the 6 restraint directions are displayed in the Message Window (Fig.1.24-③).

The verification method for reaction forces at specific supports with the mouse has been presented. The verification of reactions for each support and the method of their graphic representation is as follows:

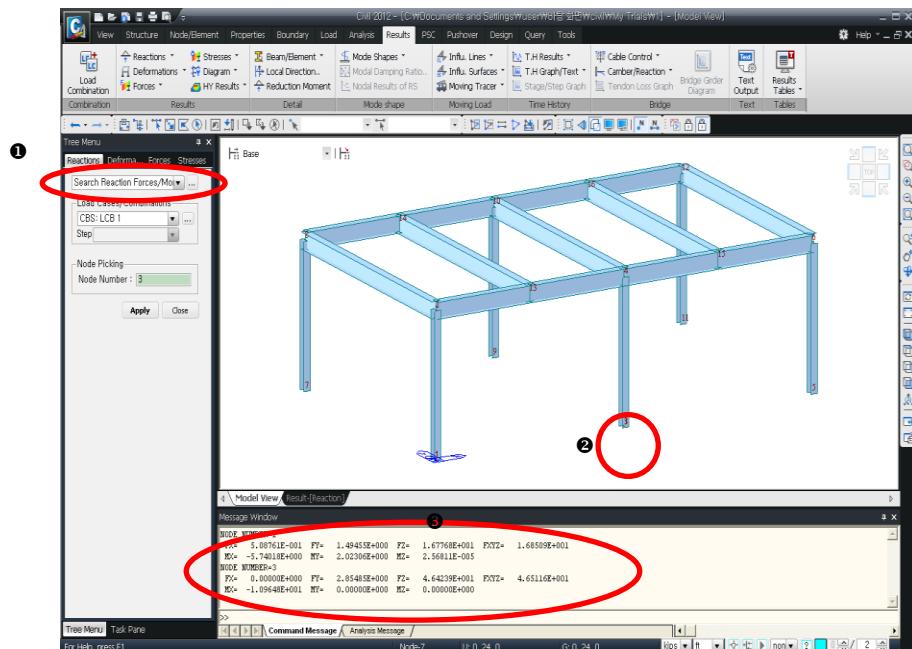
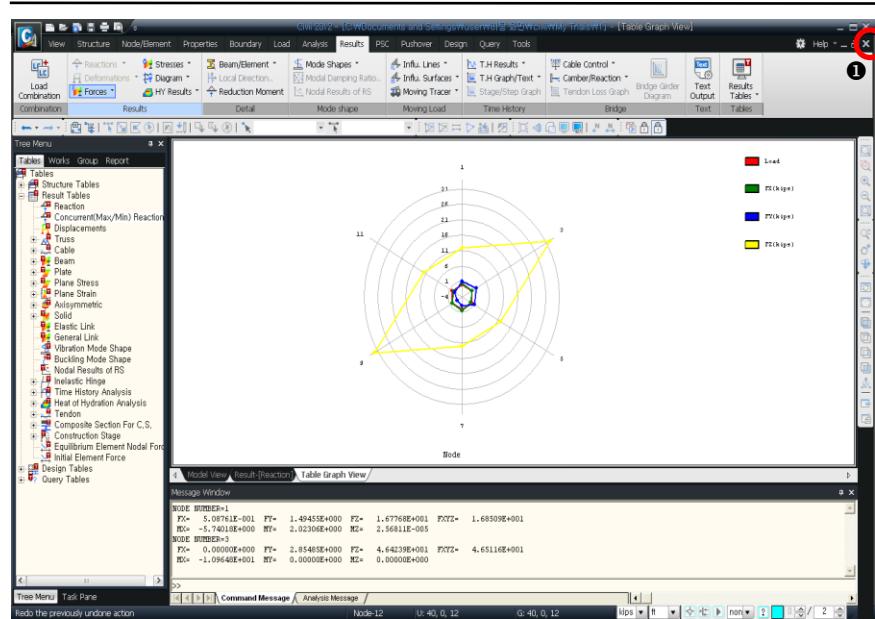


Figure 1.24 Verification of Reaction Forces at Specific Supports

1. Select **Results>Result Tables>Reaction** from the Main Menu.
2. Check () in **DL** in the **Records Activation Dialog** box.
3. Click **OK**.
4. Select each of the **Node**, **FX**, **FY** and **FZ** fields by dragging them with the mouse in the **Result-[Reaction]** table window while pressing the **[Control]** key.
5. Select **Show Graph** by right-clicking the mouse.
6. Select “**Web Chart**” from the **Graph Type** selection field.
7. Confirm “**Node**” in the **X Label (Index)** selection field.
8. Click **OK**.
9. Click to magnify **Table Graph View Window**.



**Figure 1.25 Web Chart showing Reaction Forces**

## Verify Deformed Shape and Displacements

For complex structures, the verification of deformed shape in Wire Frame is easier to view on the screen. For the present example, the deformed shape is verified in a **Hidden** state.

1. Click of Fig.1.25-① to close the **Table Graph View** and **Results [Reaction]** windows.
2. Click **Node Number** (Toggle off) in the Icond Menu
3. Select **Results > Deformations**.
4. Select **Deformed Shape** from the functions selection field.
5. Select “**ST:DL**” from the **Load Cases/Combinations** selection field.
6. Confirm “**DXYZ**” in the **Components** selection field.
7. Check () **Undeformed, Values** and **Legend** in the **Type of Display** selection field.
8. Click
9. Click to the right of **Deform** in the **Type of Display** selection field.
10. Select “**Real Deform**” from the **Deformation Type** selection field.
11. Click

$$\text{DXYZ} = \sqrt{DX^2 + DY^2 + DZ^2}$$

In the current state, the deformed shape reflects only the nodal displacements.

In the current state, the real deformed shapes of the members are displayed. Because reanalysis of the internal deformation is performed along the lengths of all the elements, Real deform takes much longer computation time compared to that of Nodal Deform. Therefore, it is more efficient to select Nodal Deform for a model with many elements.

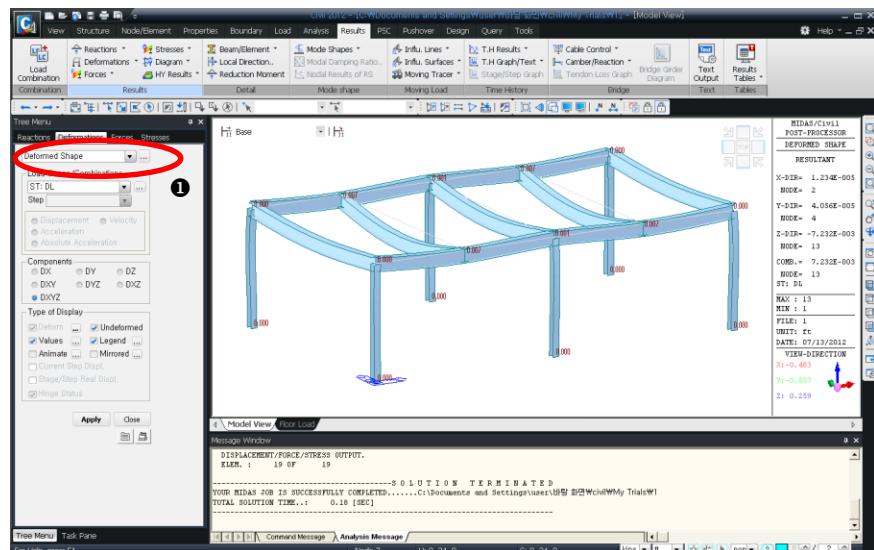


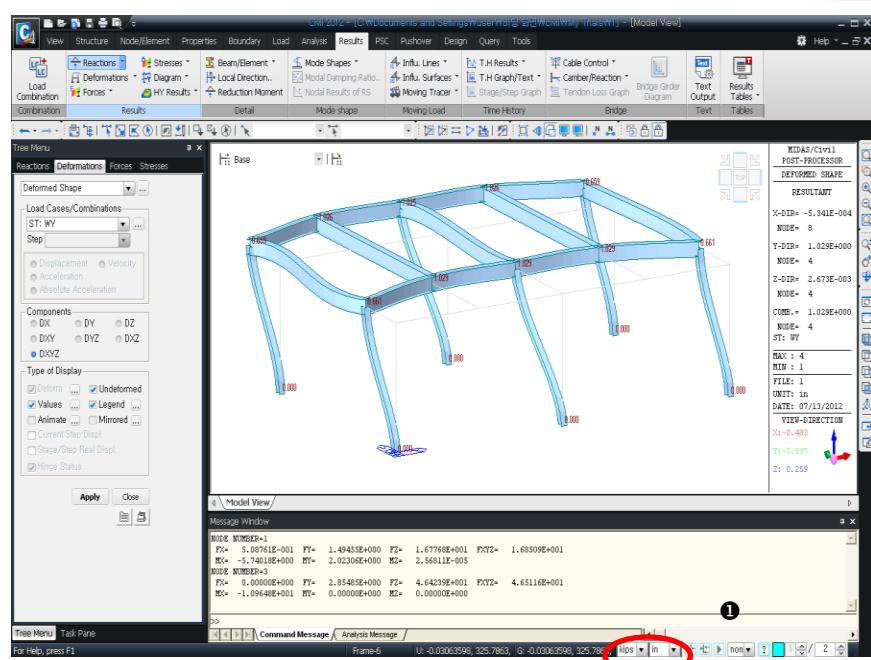
Figure 1.26 Deformed Shape

The magnitude of deformation displayed in Fig.1.26 depends on the magnification Scale Factor in the right margin. However, the numerical values of the displacements displayed for each node are true numbers.

To verify the deformation behavior displayed on the screen more closely, magnify the current deformation scale by 5 times. The following process illustrates the change of unit system. Convert the unit from “ft” to “in”. Then, observe the screen change and revert to “ft” unit.

1. Select “**ST:WY**” from the **Load Cases/Combinations** selection field.
2. Click  to the right of **Deform** in the **Type of Display** selection field.
3. Enter “**5**” in the **Deformation Scale Factor** field.
4. Click .
5. Click  in the unit conversion button at the bottom of the window (Fig.1.27–①) and select “**in**”.

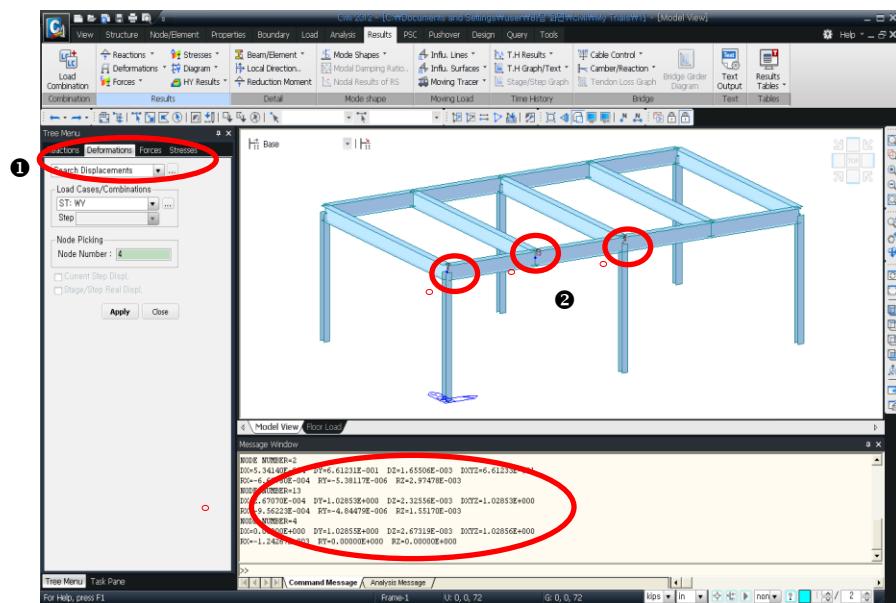
② Click  to the right of **Values** in **Type of Display** to adjust the decimal points of the v values displayed.



**Figure 1.27 Deformed Shape (Scale Factor = 5.0)**

The procedure for the verification of displacements at specific nodes is similar to that of the verification of reactions. The procedure is as follows:

1. Select **Search Displacement** from the functions selection field (Fig.1.28–❶).
2. Click the **Node Number** field once.
3. Select nodes **2, 4** and **13** with the mouse (Fig.1.28–❷).

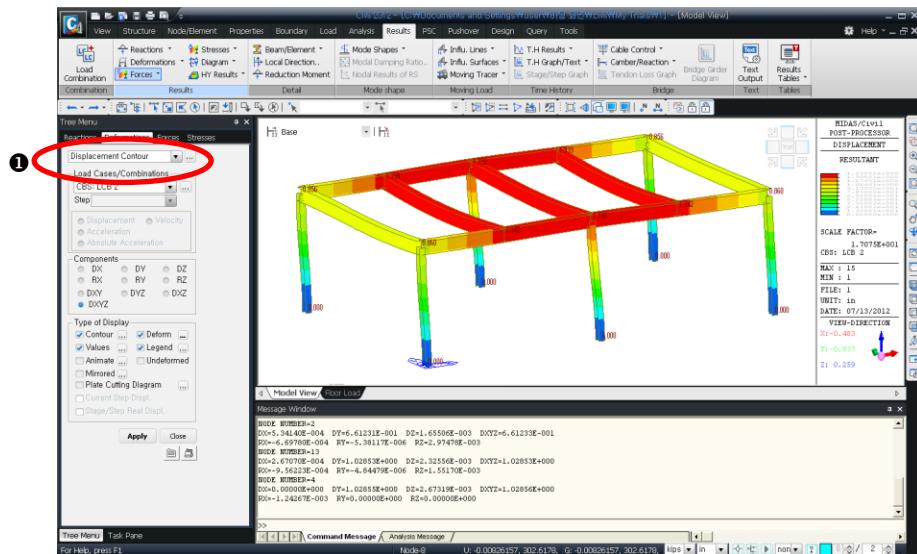


**Figure 1.28 Verification of Displacements at Specific Nodes**

**Displacement Contour** displays the displacements of each member in a series of contour lines. The procedure for the verification of deformation using contour lines is outlined as follows:

1. Select **Displacement Contour** from the functions selection field (Fig.1.29–❶).
2. Select “**CBS:LCB2**” from the **Load Cases/Combinations** selection field.
3. Confirm “**DXYZ**” in the **Components** selection field.
4. Check (**✓**) **Contour, Deform, Values** and **Legend** in the **Type of Display** selection field.
5. Click **Apply**.

ST: Static Load Case  
CB: General tab  
CBS: Steel Design tab  
CBC: Concrete Design tab  
CBR: SRC Design tab

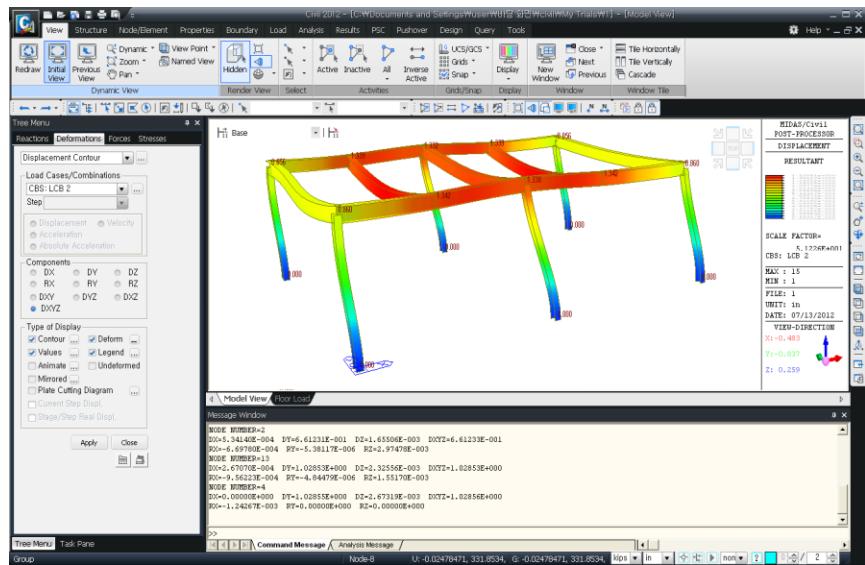


**Figure 1.29 Deformed Shape (Contour lines)**

The Gradation method is a tool to smoothen the contour distribution shown in Fig.1.29. In addition, the model is displayed in **Perspective View**.

Considerable time is required if Gradient Fill is selected and the output is formatted as a Windows Meta File. Therefore, it is not generally recommended.

1. Click **Perspective** (Toggle on) in the Icon Menu or (*View > Render View > Perspective*) in the Main Menu.
2. Click to the right of **Contour** in the **Type of Display** selection field.
3. Select “18” from the **Number of Colors** selection field.
4. Check **Gradient Fill**.
5. Remove the check in **Apply upon ok**.
6. Click .
7. Click to the right of **Deform**.
8. Enter “3” in **Deformation Scale Factor**, check **Real Deform** and click .
9. Click .



**Figure 1.30 Deformed Shape (Contour lines—Gradient Fill)**

## Verify Member Forces

The procedure for the verification of member forces is shown in terms of the moments about y-axis in the ECS.

1. Click the unit selection button  of Fig.1.31–❷ and select “ft”.
2. Click Perspective (Toggle off) in the Fixed Icon Menu.
3. Select **Tree Menu > Forces or Main Menu > Results > Forces**
4. Select **Beam Forces/Moments** from the functions selection field (Fig.1.31–❶)..
5. Confirm “My” in the **Components** selection field.
6. Check () **Contour, Values** and **Legend** in the **Type of Display** selection field.
7. Click  to the right of **Values** and modify **Decimal Points** to “1”.
8. Click .
9. Check () **All** in the **Output Section Location** selection field.
10. Click .

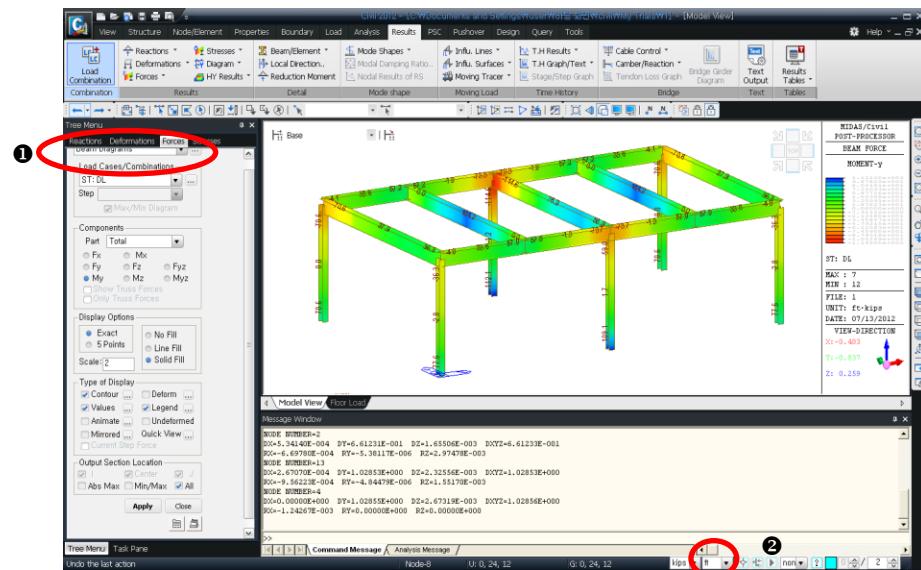


Figure 1.31 Member Forces Contour Lines  
(Bending moments about y-axis in the ECS)

## Shear Force and Bending Moment Diagrams

As the drawing procedures for the shear force and bending moment diagrams are similar, only the verification procedure for a bending moment diagram is examined.

1. Select **Beam Diagrams** from the functions selection field (Fig.1.32–①).
2. Select “**ST:DL**” from the **Load Cases/Combinations** selection field.
3. Confirm “**My**” in the **Components** selection field.
4. Select “**Exact**” and “**Solid Fill**” from the **Display Options** selection field and enter “**2**” in the **Scale** field.
5. Check (✓) **Contour**, **Values** and **Legend** in the **Type of Display** selection field.
6. Confirm the check(✓) in **All** in the **Output Section Location** selection field.
7. Click **Apply**.

Midas Civil can produce the bending moments about the weak and strong axes separately as well as depicting the bending moment diagrams about both axes in the same window concurrently.

The procedure for displaying the bending moment diagrams about the weak/strong axes pertaining to a part of the model in the same window is as follows:

1. Select “Myz” from the **Components** selection field.
2. Select “Line Fill” from **Display Options**.
3. Click **Apply**.
4. Magnify partially node 2 in Fig.1.32 by **Zoom Window**.
5. Confirm the bending moment diagram and click **Zoom Fit**.

When Both is selected, the larger of the two bending moments relative to both axes is displayed as Value.

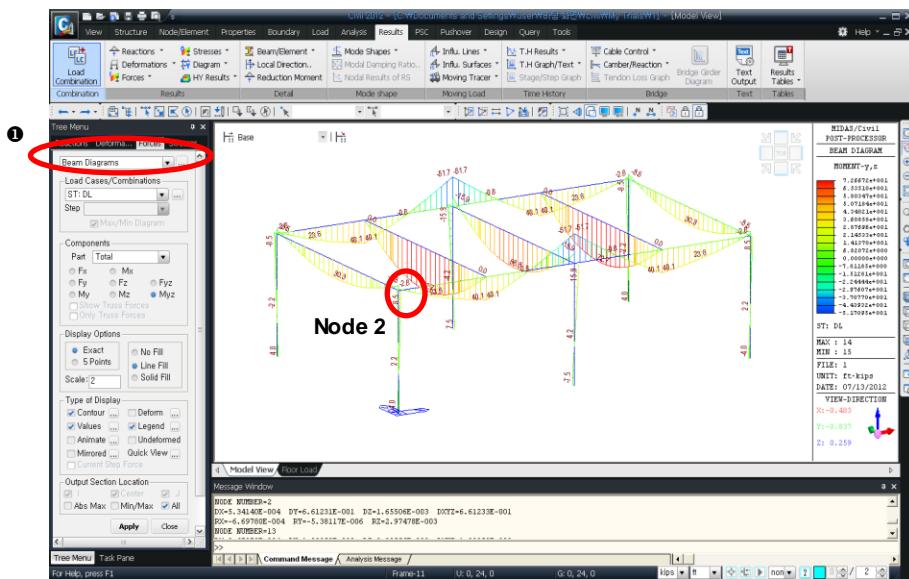
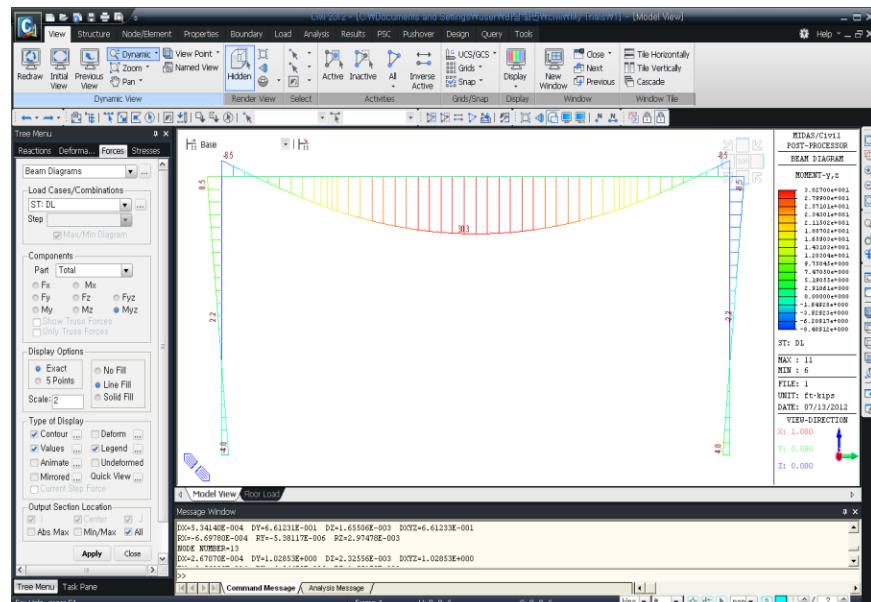


Figure 1.32 Bending Moment Diagram

In practice, it is common to select the interpretation results for structural behavior pertaining to specific parts and to include them in a report.

The procedure for selecting the bending moment diagram of the plane containing grid ① (Y-Z plane) in Fig.1.1 is as follows:

1. Click **Plane** in the **Main Menu > View > Select > Plane**.
2. Select “**YZ Plane**”.
3. Click a node located on the plane containing ① in Fig.1.1.
4. Click **Close**.
5. Click **Activate(or F2)** in the Icon Menu.
6. Click **Right View** in the Icon Menu.



**Figure 1.33 Bending Moment Diagram in Y-Z Plane**

Using midas Civil's manipulative capabilities, **Selection** and **Active/Inactive**, the user can select and color-process a specific part of the model.

Next, restore the window to the state prior to the activation of that particular area.

- 
1. Click  **Active All** the Icon Menu.
  2. Click  **Iso View** in the Icon Menu.
-

## Verify Analysis Results for Elements

The previous exercises showed analysis results that focused on specific components such as reactions, displacements, member forces, etc. When the member forces or stresses for a specific element are sought for the purpose of overall design review, use **Element Detail Results**.

- 
1. Click  **Initial View** in the Icon Menu.
  2. Click  **Element Number** (Toggle on) in the Icon Menu.
  3. Select **Results> Beam/Element > Element Detail Results** from the Main Menu.
  4. Select “**CBS:LCB1**” from the **Load Case** selection field.
  5. Click the **Element Number** field once and select element **11**.
  6. Confirm the element attributes in the **Information** tab and select successively the **Force** tab and **Stress** tab to check the analysis results.
  7. Click  to exit the **Element Detail Results** dialog box.
-

**Element Detail Results-(Model View)**

Element ID :	11	Type :	BEAM
Material ID :	1	Name :	A36
Section ID :	2	Name :	W16x67
LC :	LCB 1	Type :	St
Type : DB			

**Element Detail Results-(Model View)**

Element Number	Load Case	Step
11	CBS: LCB 1	
<b>Information</b>		
<b>Force</b>		
<b>Stress</b>		

PT	AXIAL	SHEAR-y	SHEAR-z	TORSION	MOMENT-y	MOMENT-z
I	-1,5	0,0	-7,9	0,0	-12,2	0,0
1/4	-1,5	0,0	-4,9	0,0	28,8	0,0
CNT	-1,5	0,0	0,0	0,0	43,5	0,0
3/4	-1,5	0,0	4,9	0,0	28,8	0,0
J	-1,5	0,0	7,9	0,0	-12,2	0,0

**Element Detail Results-(Model View)**

Element Number	Load Case	Step
11	CBS: LCB 1	
<b>Information</b>		
<b>Force</b>		
<b>Stress</b>		

PT	AXIAL	SHEAR-y	SHEAR-z	+y	(BENDING) -y	+z
I	-10,9	0,0	-195,2	-0,8	0,8	180,2
1/4	-10,9	0,0	-120,7	-0,8	0,8	-426,2
CNT	-10,9	0,0	0,0	-0,8	0,8	-643,7
3/4	-10,9	0,0	120,7	-0,8	0,8	-426,2
J	-10,9	0,0	195,2	-0,8	0,8	180,2

Figure 1.34 Element Detail Results

## Verify Member Stresses and Manipulate Animation

Midas Civil provides axial stress, shear force and bending moment diagrams in weak/strong directions of members. A combined stress is generated by combining the axial and flexural stresses on the basis of directional components.

For this example, the combined stresses due to LCB 2 (Load combination 2) in the model are examined. Then, by combining the relevant stresses and the deformed shapes, the procedure for the animation representation is illustrated below.

1. Select **Results>Stresses>Beam Stresses** from the Main Menu.
2. Select “**CBS:LCB2**” from the **Load Cases/Combinations** selection field.
3. Confirm “**Combined**” from the **Components** selection field.
4. Confirm the check () in **Contour, Values** and **Legend** in **Type of Display**.
5. Check () **Max** in the **Output Section Location** field.
6. Click **Element Number** (Toggle off) in the Icon Menu.
7. Click **Apply**.

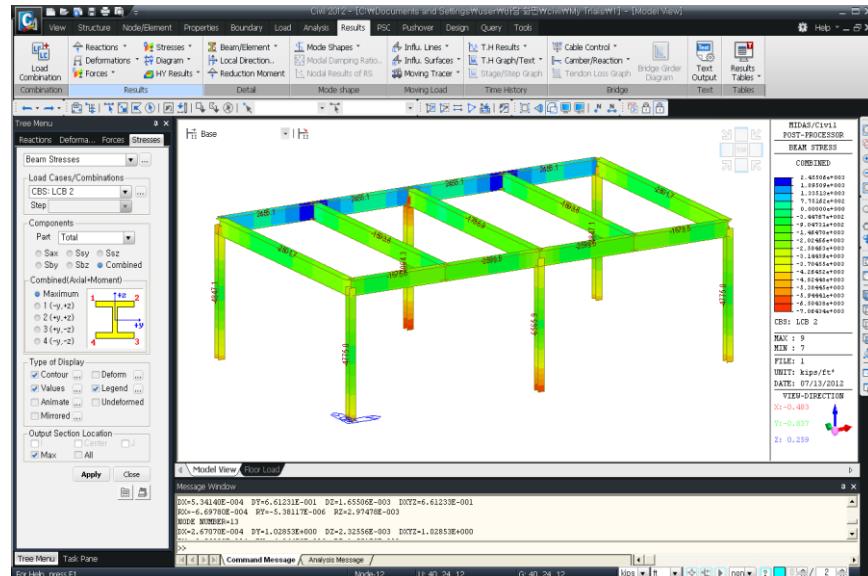


Figure 1.35 Combined Stresses in Beam Elements

In order to depict the results display window realistically, Midas Civil supports **Dynamic View** and **Animation**.

The summary of **Dynamic View** supplied by Midas Civil is as follows:

**Dynamic View** comprises  **Zoom Dynamic**,  **Pan Dynamic** and  **Rotate Dynamic**, which supplies realistic representations of the structure with respect to the desired view point.

If **Zoom** and **Rotate** are applied in connection with **Render View**, the user is drawn to the effects of walking through (Walking Through Effect) the structure or flying over the structure.

Use **Dynamic View Toolbar** (Fig. 1.36–①), located vertically on the right of the Model Window, as directed below.

Click  **Zoom Dynamic** and move the mouse cursor to the Model Window. Then, left-click and hold to magnify the model by dragging to the right (upward) or reduce the model by dragging to the left (downward).

Click  **Pan Dynamic** and move the mouse cursor to the Model Window. Then, left-click and hold to move the model to the desired direction by dragging to the left, right, upward or downward.

Click  **Rotate Dynamic** and move the mouse cursor to the Model Window. Then, left-click and hold to rotate the model to the desired direction by dragging to the left, right, upward or downward.

Observe the combined stresses of the structure by using the above-mentioned **Dynamic View** functions according to the following procedure:

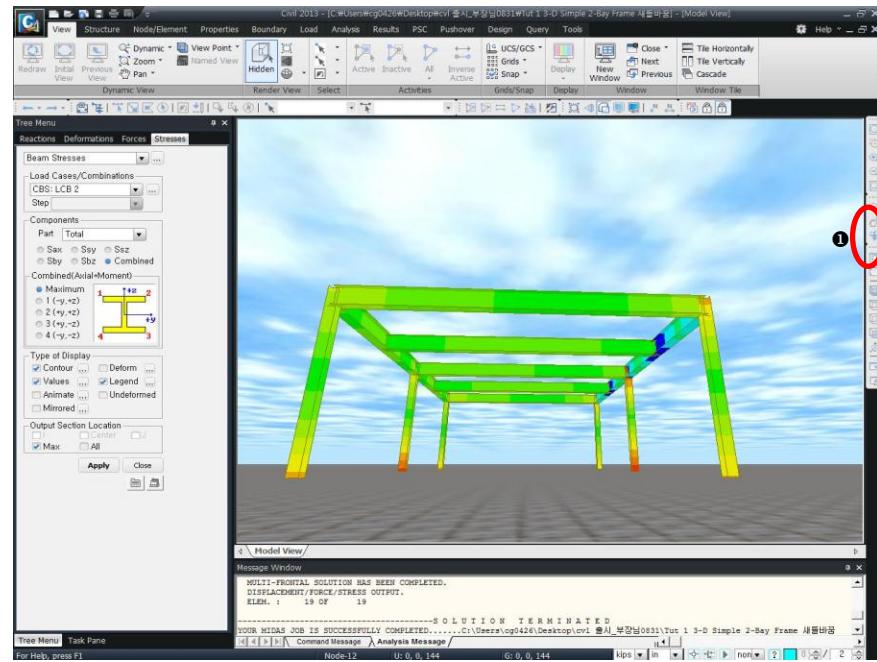


Figure 1.36 Render View

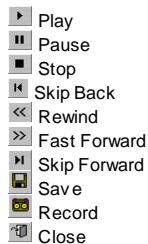
- 
1. Click  **Render View** in **View > Render View** in the Main Menu (Toggle on).
  2. Use left and right buttons of your mouse to control view.
  3. Click  **Render View** in **View > Render View** to switch from  **Render View** to **Model View** (Toggle off).
- 

Create an animation combining the relevant stresses and the deformed shapes in the current window.

For easier assessment of the deformation trend due to LCB 2 (Load Combination 2), rotate the model as shown in Fig.1.36 by using  **Rotate Dynamic**.

When the desired window is selected, adjust the window by means of  **Zoom Fit** and  **Perspective**. The procedure to create an animation is as follows:

 The representative icons controlling the animation are listed below.



1. Click  **Perspective** in the Icon Menu (Toggle on).
  2. Click  **Rotate Dynamic** in the Icon Menu and adjust to the desired **View Point**.
  3. Check () **Contour, Deform, Legend, Animate** in the **Type of Display** selection field.
  4. Click the button  to the right of **Deform**.
  5. Select “**Real Deform**” in **Deformation Type** of the **Deformation Details** dialog box.
  6. Click .
  7. Click  **Record** as shown in Fig.1.37–①.
- 

Once the above procedure is completed, wait a while. The animation reflecting the effects of combined stresses and deformed shapes appears on the screen as shown in Fig.1.37.

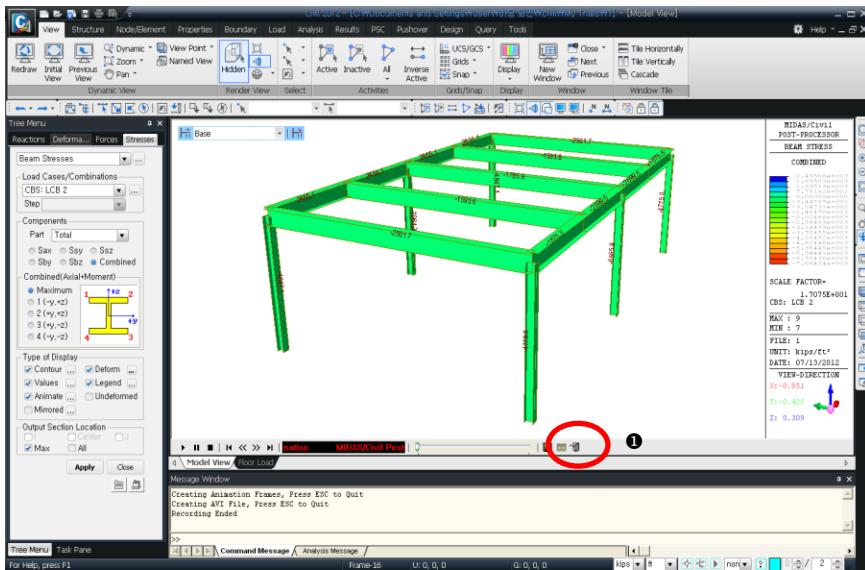


Figure 1.37 Animation Window

## Beam Detail Analysis

Midas Civil provides detail displacements and shear force/bending moment diagrams for both axes of beam elements. A detail analysis process also provides the stress distribution relative to a specified section.

The execution of **Beam Detail Analysis** by selecting **Results>Beam/Element>Beam Detail Analysis** from the Main Menu results in the following contents:

➲ The detail numerical values in each distribution diagram can be verified by moving the scroll bar located at the bottom of the dialog box.

- The detail displacement/shear force/moment distribution plots relative to the weak and strong axes and the corresponding numerical values
- The maximum stress distribution plot relative to a specific position in the element length direction
- The stress distribution plot and sectional stress diagram for the weak and strong axes relative to a specific section

1. Click **Close** shown in Fig.1.37-①.
2. Select **Results>Beam/Element > Beam Detail Analysis** from the Main Menu.
3. Select “**ST:DL**” from the **Load Cases/Combinations** selection field.
8. Click **Element Number** (Toggle on) in the Icon Menu.
4. Click the **Element Number** field once, then select element **11** in the **Model View** window (Fig.1.37).
5. Click to magnify the **Beam Detail Analysis** window.
6. Verify the analysis results by selecting consecutively the **DISP/SFD /BMD z-dir**, **DISP/SFD/BMD y-dir** and **Section** tabs shown in Fig.1.38-②.

① The windows currently opened in the Window of the Main Menu can be automatically assigned in diverse formats.

② The z-dir tab displays Dz, Fz and My.

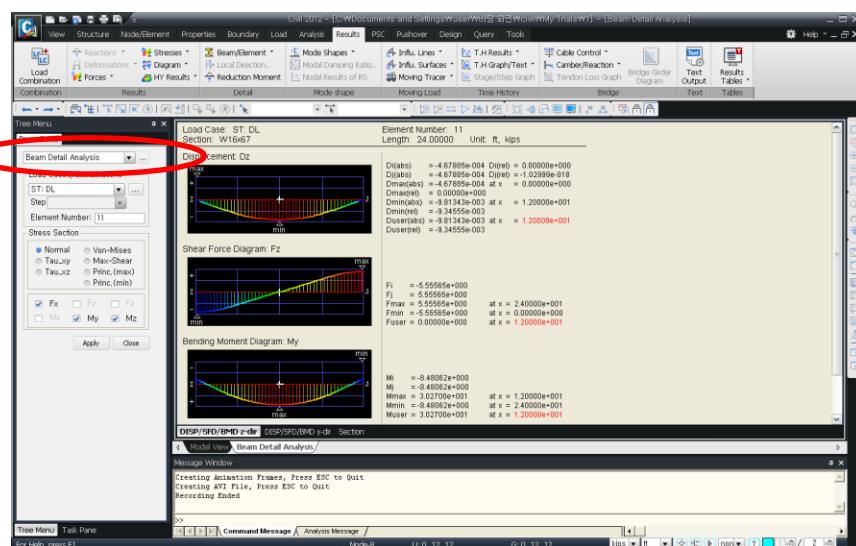
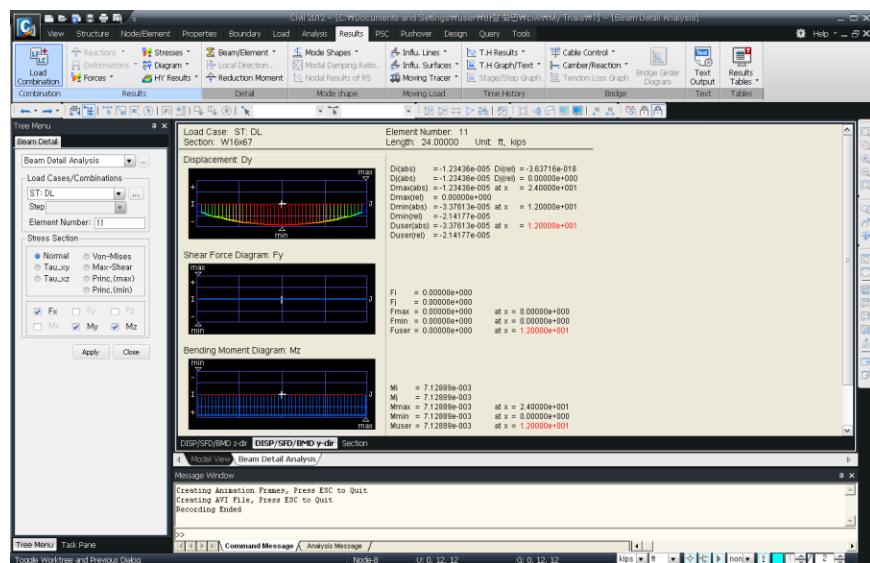
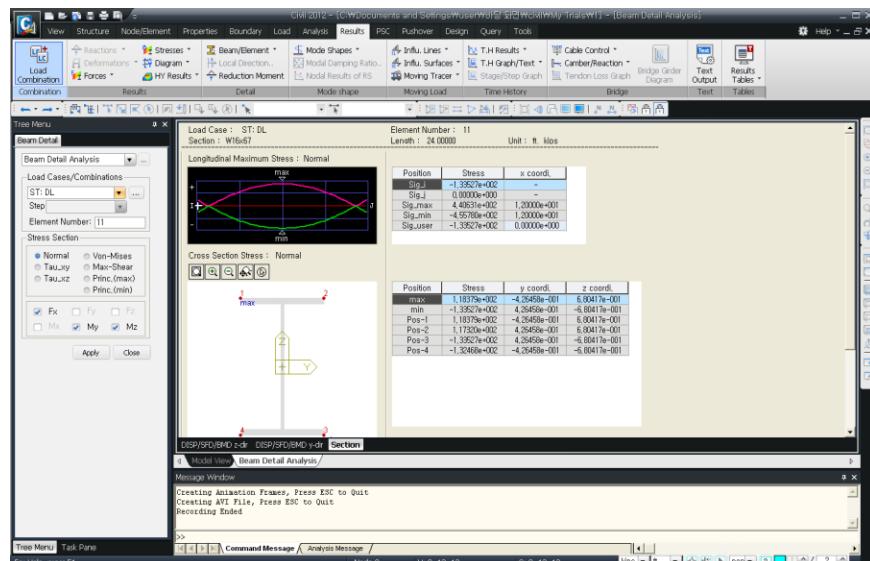


Figure 1.38 Beam Detail Analysis (DISP/SFD/BMD z-dir)



**Figure 1.39 Beam Detail Analysis (DISP/SFD/BMD y-dir)**



**Figure 1.40 Beam Detail Analysis (Section)**