Tutorial 1

Gen

TUTORIAL 1. 3-D SIMPLE 2–BAY FRAME

TUTORIAL 1.
3-D SIMPLE 2–BAY FRAME

Summary

This example is for those who never had an access to **midas Gen** 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 Gen** environment and to become acquainted with the procedure for using **midas Gen** within a very short time frame. The user will be introduced easily to **midas Gen** 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 and Ⓑ/① in the (+X) direction
- Load Case 4 – Uniformly distributed load, 1k/f applied to all the members on grid Ⓐ in the (+Y) direction

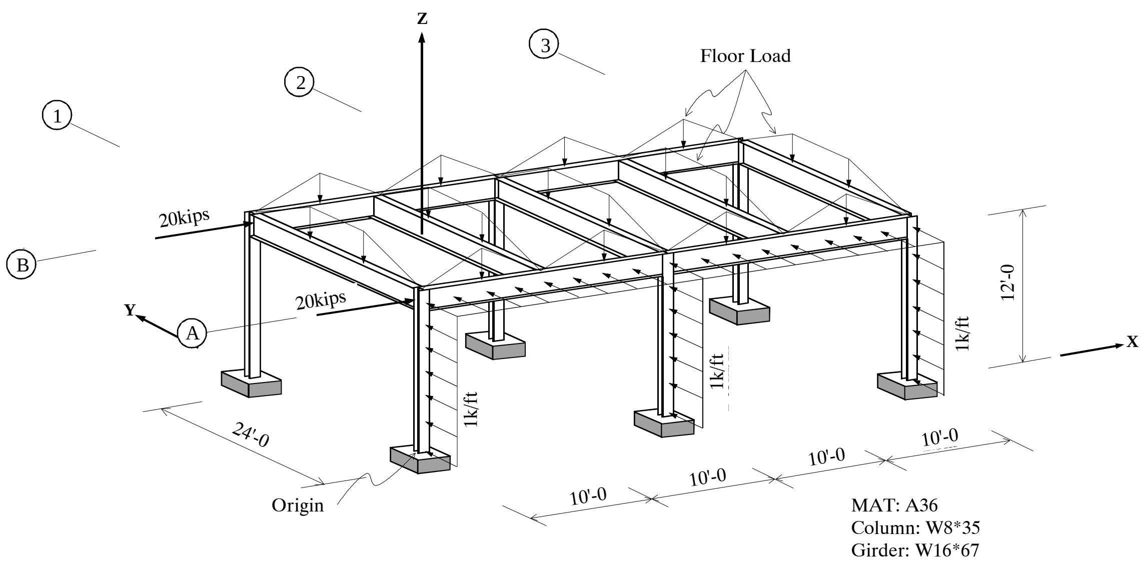


Figure 1.1 3–D Simple 2–Bay Frame

File Opening and Preferences Setting

First, double-click the **midas Gen** 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 Gen 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^2). 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>Setting>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 "ksi".

Works Tree allows the user to modify the data entries by the Drag & Drop

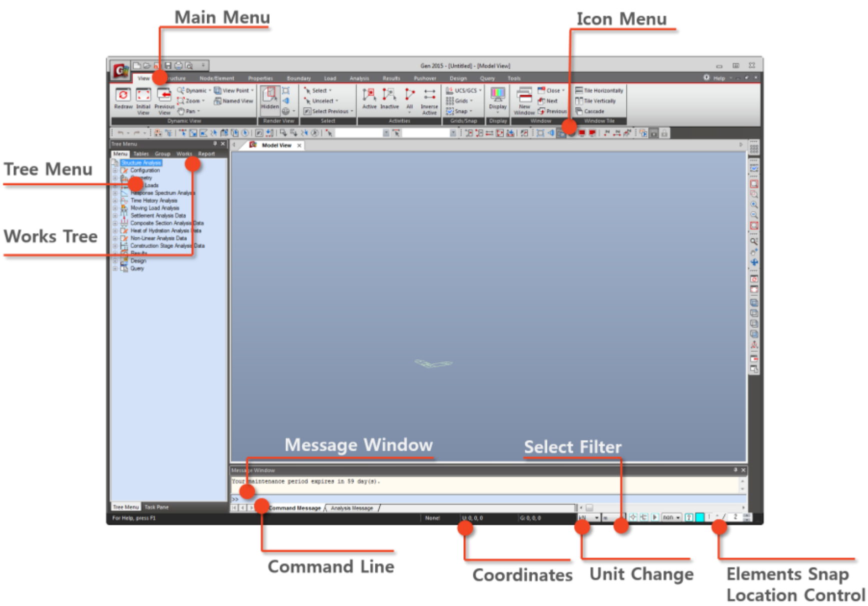


Figure 1.2 Default Window

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>Setting>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.

Toggle on

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

Menu System

midas Gen 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 Tree Menu and the Icon Menu.

In **midas Gen**, the user can modify the placement of toolbars as desired. All toolbars (Fig. 1.2) should be selected already for use in the working window. The user can click on the checkboxes to view where each toolbar is located in the working window.

1. Select ***Tools>Customize>Customize…*** from the Main Menu.
2. Select the ***Toolbars*** tab.
3. Confirm all toolbars are selected in the Toolbars List.
4. Icons will appear on the Main Menu. Different tools can be selected easily.
5. Click to exit ***Toolbars*** dialog box.

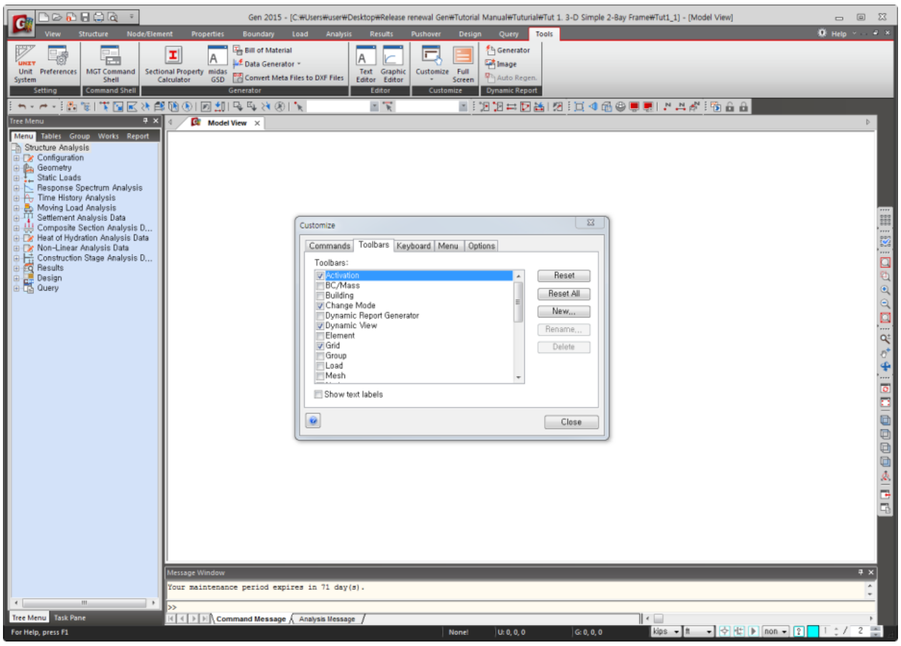


Figure 1.3 Placement of Toolbars

Coordinate Systems and Grids

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

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

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).

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 beam elements.

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

If UCS is not defined separately in **midas Gen**, it is assumed that the axes of UCS and GCS are identical. In addition, the default grids are laid out in UCS x-y plane.

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

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.

Assign the GCS X-Z plane containing the grid Ⓐ as UCS x-y plane to enter the 3 columns and 2 beams of the structure (Fig.1.1), by using ***X-Z*** (or ***Geometry***> ***User Coordinate System***>***X-Z Plane*** in the ***Menu*** tab of the Tree Menu).

1. Click ***X-Z*** from ***Structure*** > ***UCS/Plan*** > ***UCS*** in the Main Menu.
2. Confirm "**0,0,0**" in the ***Origin*** field.
3. Confirm "**0**" in the ***Angle*** field.
4. Click OK.

Toggle on

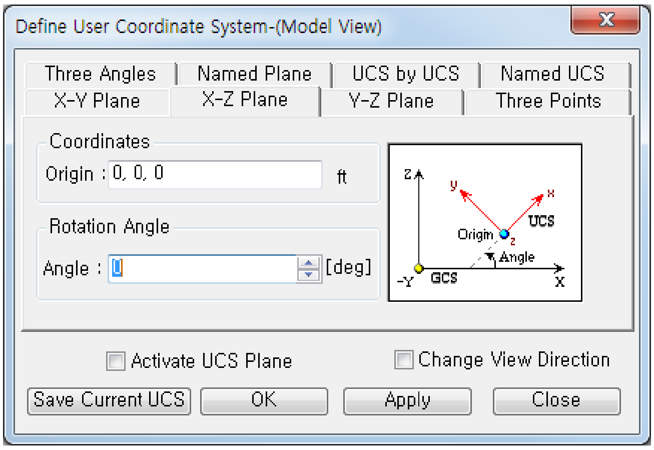


Figure 1.4 UCS Setting

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 ***Set Point Grid*** from ***Structure*** > ***UCS/Plan*** > ***Grids***.
2. Enter "**2,2**" in the ***dx, dy*** field.
3. Click OK.

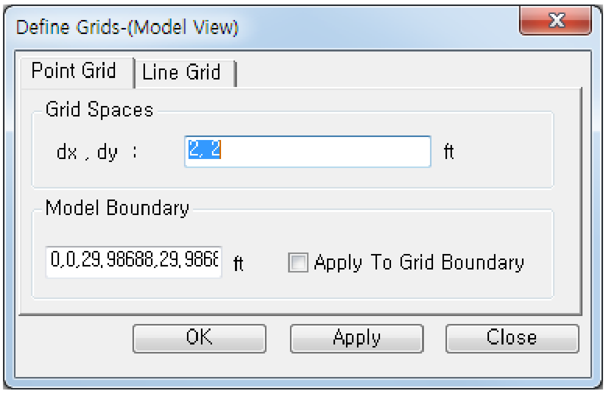


Figure 1.5 Point Grids Setting

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

When **midas Gen** 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.

1. Click ***Front View*** in the Icon Menu.
2. Click ***Grid Snap*** in the Icon Menu (Toggle on).
3. Click ***Line Grid Snap*** and ***Snap All*** from ***Structure*** > ***UCS/Plan*** > ***Grids*** in the Main Menu(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

❶

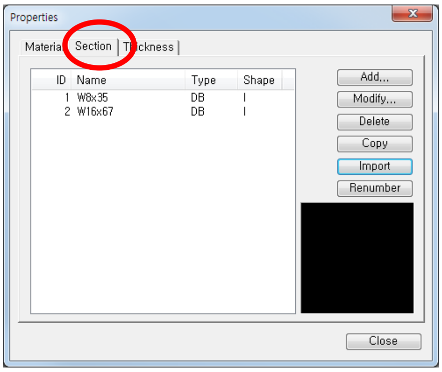


Figure 1.6 Dialog box for Section Properties

❷

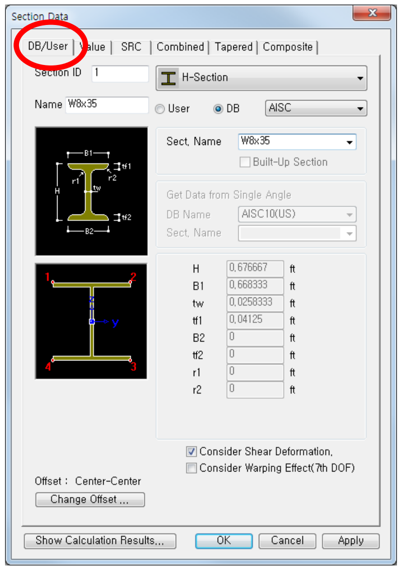


Figure 1.8 Section Data

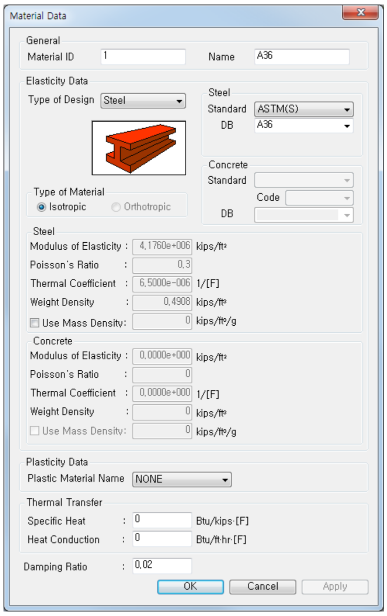


Figure 1.7 Material Data

Click [Show Calculation Results...] to verify the stiffness data of the specified section.

1. Select ***Geometry>Properties>Material*** from the ***Menu*** tab of the Tree Menu.
2. Click [Add] shown in Fig.1.6.
3. Confirm "**1**" in the ***Material ID*** 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.

7. Click OK.
8. Select the ***Section*** tab on the top of the ***Properties*** dialog box (Fig.1.6– ❶).

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

9. Click Add.
10. Confirm the ***DB/User*** tab on the top of the ***Section*** dialog box (Fig.1.8– ❷).
11. Confirm "**1**" in the ***Section ID*** field.
12. Confirm "**I-Section**" in the ***Section*** selection field.
13. Confirm "**AISC**" in the ***DB*** selection field.
14. Select "**W8 × 35**" from the ***Sect. Name*** selection field.
15. Click Apply.

OK closes the dialog box after completing the data entry. Apply completes the data entry and prompts the dialog box to remain. Click Apply when section data entry is repeated.

16. Confirm "**2**" in the ***Section ID*** field.
17. Select "**W16 × 67**" in the ***Sect. Name*** selection field.
18. Click OK.
19. Click Close in the ***Properties*** dialog box (Fig.1.6).

Structural Modeling Using Nodes and Elements

Before entering the data for structural members, toggle on ***Hidden*** (or ***View>Remove Hidden Lines*** in Main Menu) 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*** to verify the node and element numbers.

1. Click ***Hidden*** (Toggle on) in the Icon Menu.
2. Click ***Display*** in the Icon 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)).

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

3. Click OK.

Toggle on

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

---

1. Select ***Geometry>Elements>Create*** from the ***Menu*** tab of the Tree Menu.
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 Ⓑ by duplicating the elements already created above (Fig.1.1).

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.

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.

Check (✓) Align Top of Beam Section to Floor (X-Y Plane) for Panel Zone Effect/ Display in Structure > Type > Structure Type of Main Menu. Then, the effect of the beam/column panel zone will appear as ❶ in Fig.1.9.

By setting Auto Fitting Toggled on, **midas Gen** 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.

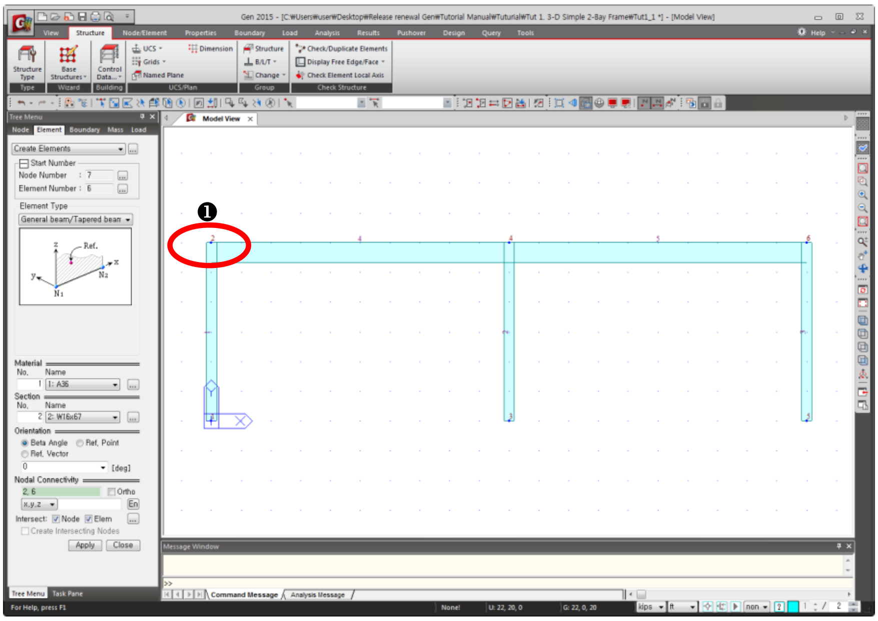


Figure 1.9 Generation of 2–D Frame

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.

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*** (or ***View>Select>Select All*** in the Main Menu). Then, duplicate the elements by ***Translate Elements***.

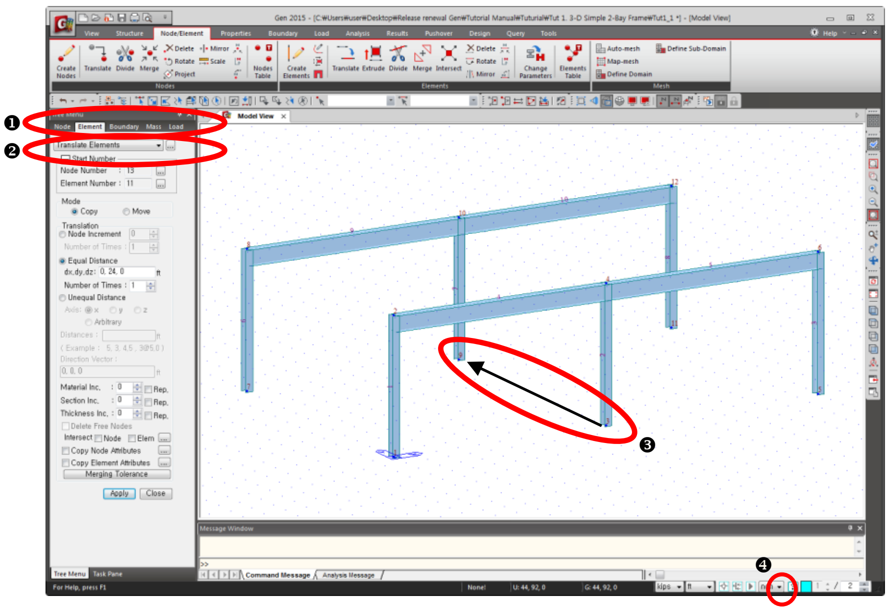
When switching from the current modeling function to another function, the Main Menu or Tree Menu can be used. In the case of mutually related functions (example: ***Create Elements***, ***Translate Elements***, etc.), **midas Gen** enables the user to switch directly using the functions selection field (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*** from ***View*** > ***Grids/Snap*** > ***UCS/GCS*** in the Main Menu.
2. Click ***Isometric View*** in the Icon Menu.
3. Click ***Select All*** in the Icon Menu.
4. Select ***Translate Elements*** from the functions selection field (Fig.1.10–❷).
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 ***Auto Fitting*** in the Icon Menu.
10. Click Apply.

Toggle on

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.1.10-❸).



In Fig.1.10:
❶: Model Entity tab
❷: 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.

Figure 1.10 Duplication of 2–D Frame

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

⊶ **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 ***Grid Snap***.

1. Click ***Point Grid*** and ***Grid Snap*** (Toggle off) in the Icon Menu.
2. Select ***Create Elements*** from the functions selection field (Fig.1.11–❶).
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 × 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.1.11-❷).

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.

midas Gen 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.1.11-❸).

1. Create element **14** by extending the centers of elements **4** and **9** with the mouse cursor.
2. Click ***Select Single*** and select element **14**.
3. Select ***Boundary*** from the **Model Entity** tab (Fig.1.11–❶).
4. Select ***Beam End Release*** from the functions selection field.
5. Click Pinned-Pinned and click Apply.
6. Select ***Element*** from the ***Model Entity*** tab (Fig.1.11–❶).
7. Select ***Translate Elements*** from the functions selection field.
8. Confirm "**Copy**" in ***Mode*** field.
9. Click ***dx,dy,dz*** field of ***Equal Distance*** once.

10. Click successively node **14** and the center of element **10** to enter “**20, 0, 0**” automatically.
11. Check (✓) **Node** and **Elem.** of ***Intersect***.
12. Check (✓) **Copy Element Attributes** and click [...] on the right.
13. Confirm the check (✓) in **Beam End Release** of ***Boundaries***.
14. Click [OK] in the ***Copy Element Attributes*** dialog box.
15. Click ***Shrink.***
16. Click ***Select Previous*** to select element **14**.
17. Click [Apply] of the ***Translate Elements*** dialog bar.
18. Click ***Display***.
19. Select ***Boundary*** tab (Fig.1.11–❹).
20. 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 Node/Element>Nodes> Nodes Table or Node/Element> Elements >Elements Table of Main Menu, the current status of nodes and elements can be verified and also modified.

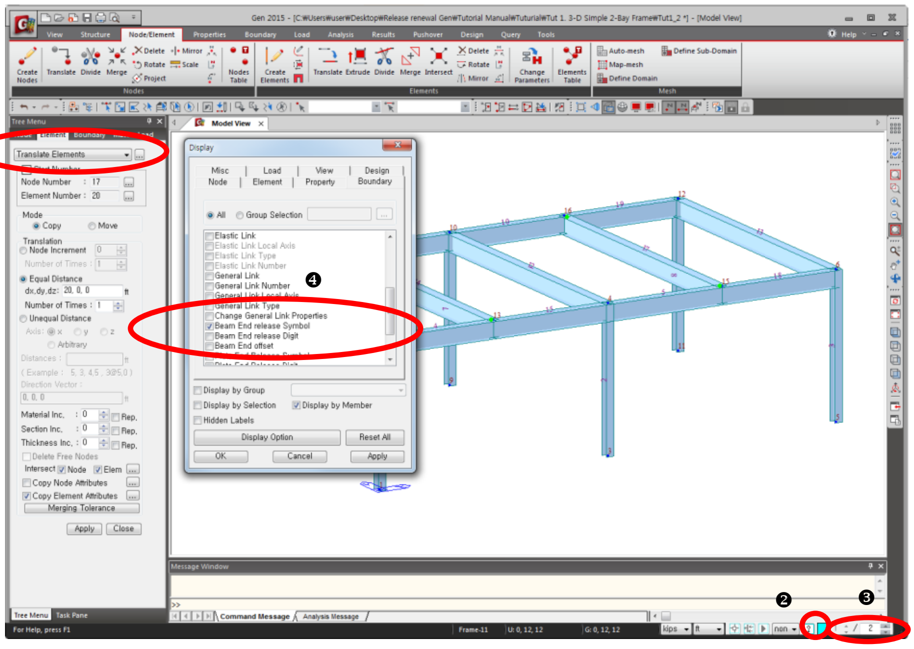


Figure 1.11 Generation of Girders and Beams

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 ***Select Plane*** (or ***View>Select>Plane*** from the Main Menu).

1. Remove the check (✓) in **Beam End Release** of ***Display***.
2. Click OK.
3. Click ***Shrink*** (Toggle off).
4. Click ***Select by Plane*** in the Icon Menu.
5. Select "**XY Plane**".
6. Click one node among the 6 column supports.
7. Click Close.

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.

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

1. Select ***Boundary*** in the ***Model Entity*** tab (Fig.1.12–❶).
2. Select ***Supports*** from the functions selection field.
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 Apply.

midas Gen supplies a variety of select functions.

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

❶

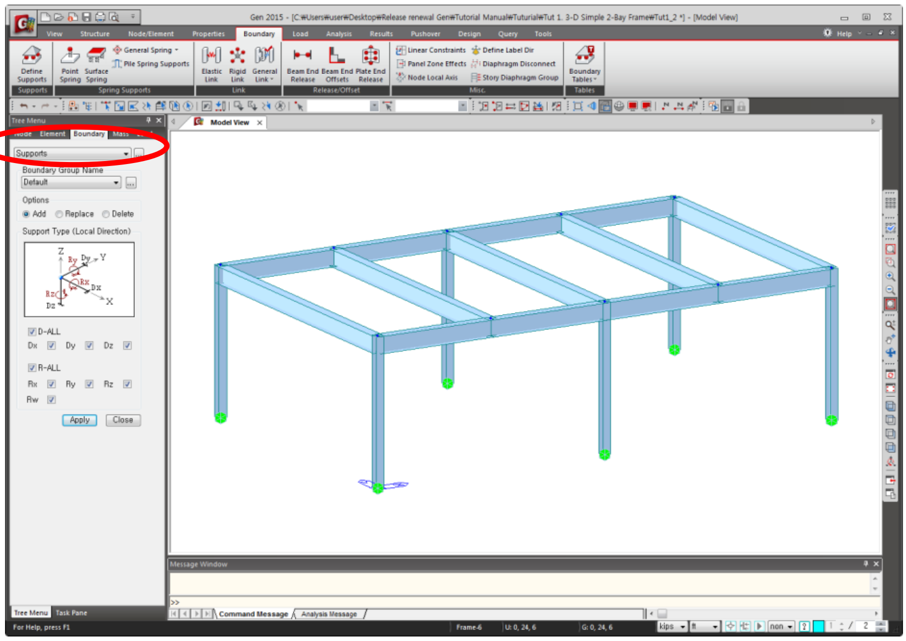


Figure 1.12 Data Entry for Structure Supports

Enter Loading Data

Define Load Cases

Define load cases before entering the loading data.
Select ***Load*** in ***Model Entity*** tab for loading (Fig.1.12–❶).

Click [...] on the right of the ***Load Case Name*** selection field (or ***Load>Static Load Cases*** in the Main Menu) to access the ***Static Load Cases*** dialog box and enter the following load cases:

1. Select ***Load*** from the ***Model Entity*** tab (Fig.1.12–❶).
2. Click [...] to the right of the ***Load Case Name*** selection field.
3. Enter "**DL**" in the ***Name*** field of the ***Static Load Cases*** dialog box (Fig.1.13).
4. Select "**Dead Load**" from the ***Type*** selection field. 🎧
5. Enter "**Floor Dead Load**" in the ***Description*** field.
6. Click [Add].
7. Enter the remaining load cases in the ***Static Load Cases*** dialog box as shown in Fig.1.13.
8. Click [Close].

🎧 Click the Type field once and type in "D", then Dead Load will be selected in Load Type. Similarly, Wind Load and Live Load can also be selected by typing in only the initials, i.e., "W" and "L".
When specifying Wind Load, be cautious to differentiate Wind Load on Structure from Wind Load on Live Load.

※ 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.

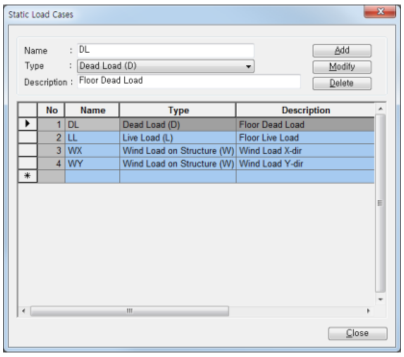


Figure 1.13 Definition of Load Cases

Define Self Weight

Define the self-weights of elements.

1. Confirm ***Self Weight*** in the functions selection field.
2. Confirm "**DL**" in ***Load Case Name***.
3. Enter "**-1**" in the ***Z*** field under ***Self Weight Factor***.
4. Click Add.

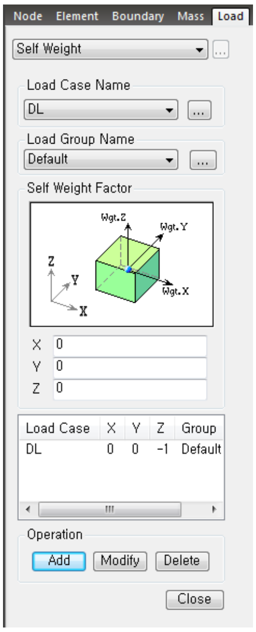


Figure 1.14 Self Weight Data

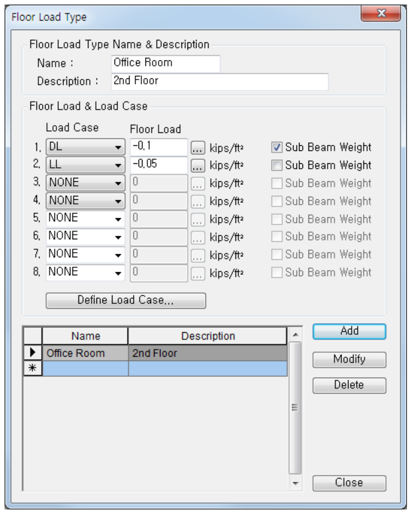


Figure 1.15 Definition of Floor Load Type

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 ***Assign Floor Loads*** from the functions selection field (Fig.1.16–❶).
2. Click ... to the right 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 Add.
8. Click Close.
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 Display Option. The size of the displayed Load Label can be adjusted likewise.

❶

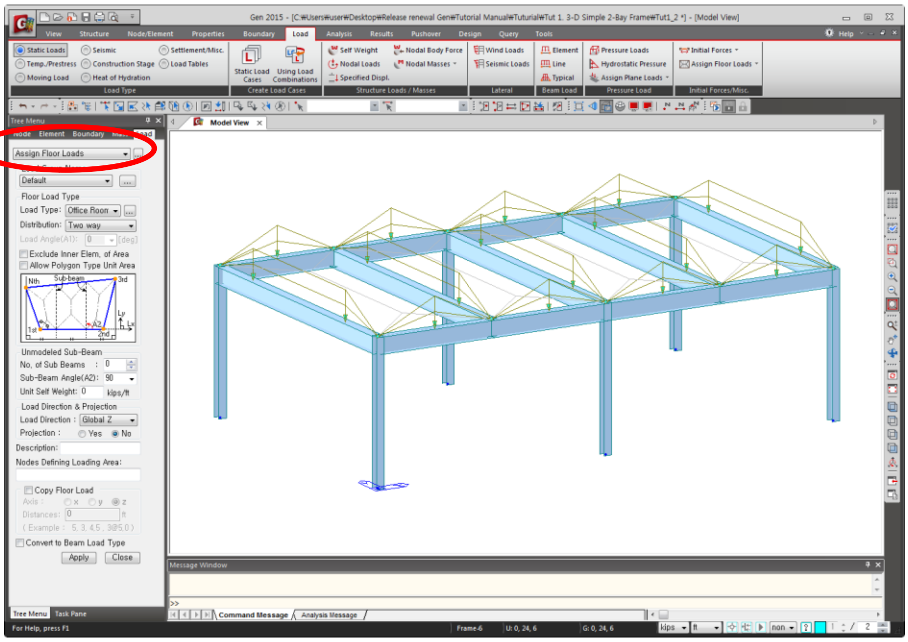


Figure 1.16 Entry of Floor Loads

Define Nodal Loads

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

1. Select ***Nodal Loads*** from the functions selection field. (Fig.1.17–).
2. Click ***Hidden*** (Toggle off) in the Icon Menu.
3. Click ***Select Window*** (Toggle on) in the Icon 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

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

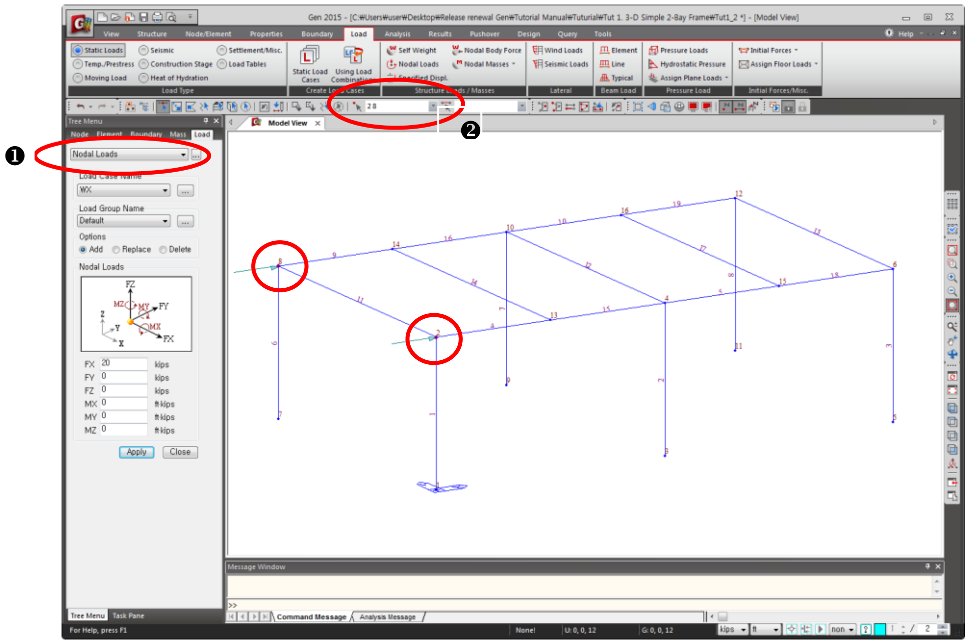


Figure 1.17 Entry of X-Direction Wind Load

Define Uniformly Distributed Loads

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

1. Click ***Select by Plane*** in the Icon Menu.
2. Select “**XZ Plane**”.
3. Click one point in grid Ⓐ (Fig.1.1). 🎧
4. Click Close.
5. Select ***Element Beam Loads*** from the functions selection field (Fig.1.18–❶).
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.

🎧 This plane can also be selected by assigning 3 nodes on the plane with 3 Point.

🎧 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 duplicating errors.

❶

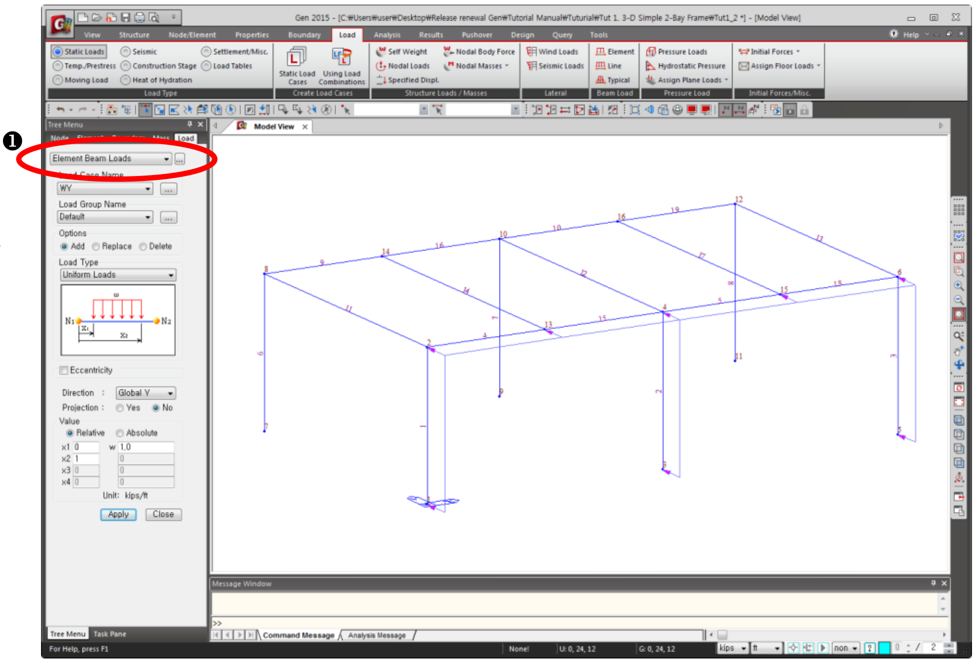


Figure 1.18 Entry of Y-Direction Wind Load

For easy reference, **midas Gen** 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*** in the Icon Menu, 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 OK.
4. Click Close in the ***Element Beam Loads*** dialog box.
5. Select the ***Works*** tab.

Works Tree categories all the 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.

Right-clicking in the 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: W8×35**" and then right-click the mouse to select ***Properties***.
3. Click the ***Sect. Name*** field once and enter "**W3**".
4. Select "**W 36×300**".
5. Click OK.

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

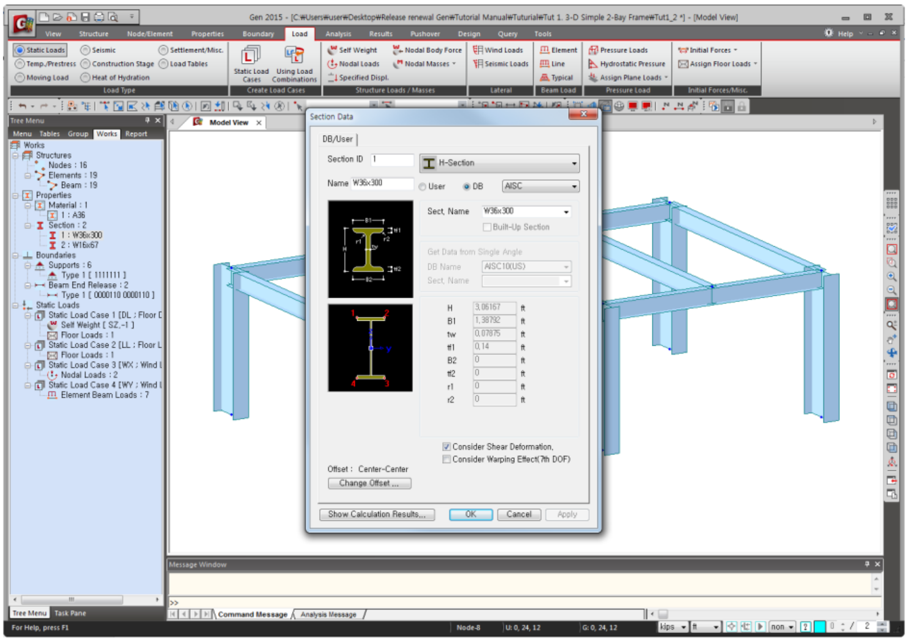


Figure 1.19 Section Data Revision using Works Tree

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

1. Under the ***Properties>Section*** of Works Tree double-click "**2:W16×67**" to select the beam elements.
2. From the section drag "**1:W36×300**" 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.

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

Basic attributes of nodes or elements can be easily viewed with ***Fast Query***. Placing the mouse cursor over a node or element will display a bubble tip.

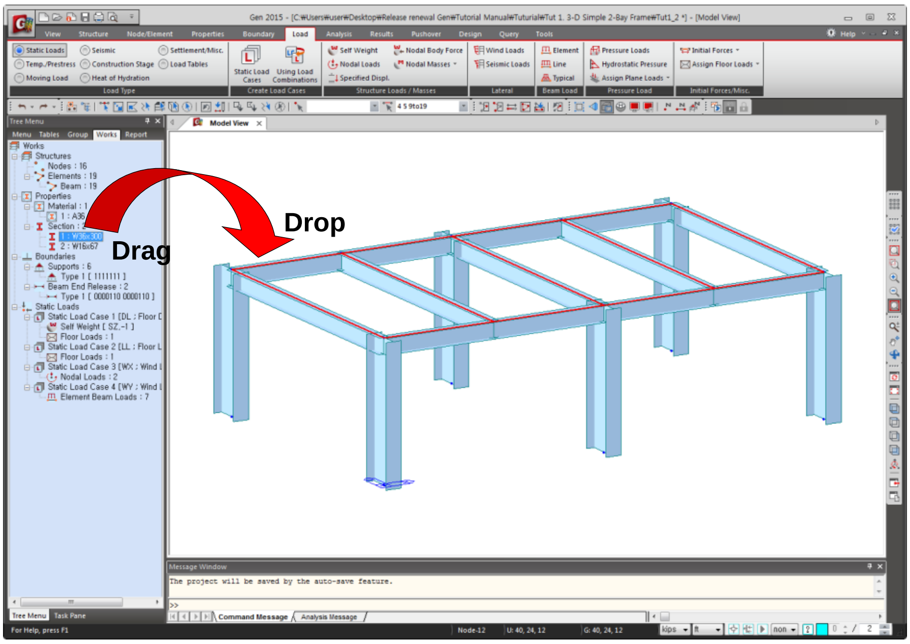


Figure 1.20 Change of model by Drag & Drop

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.

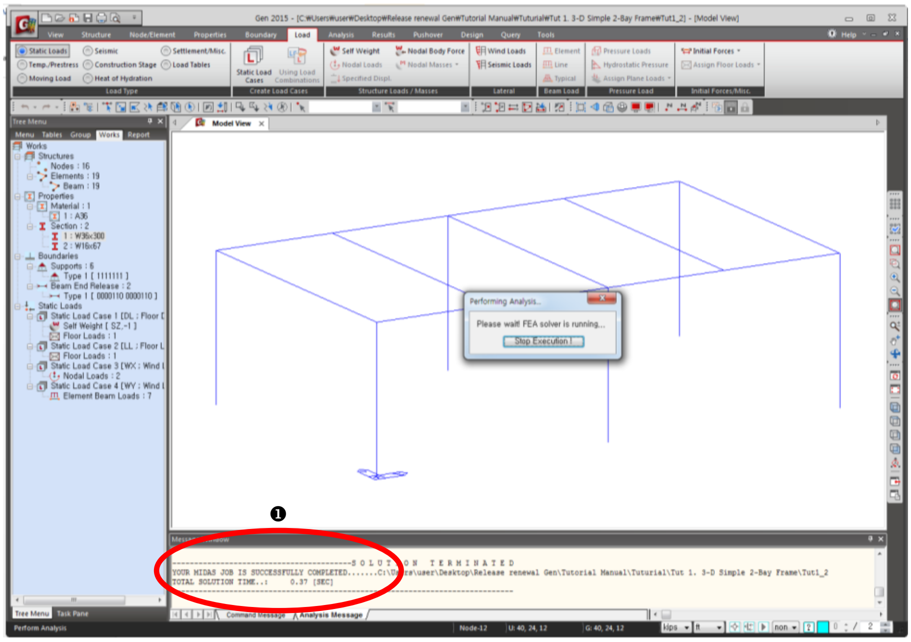


Figure 1.21 Execution Process of Structural Analysis

Verify and Interpret Analysis Results

Mode

For the sake of efficiency and convenience, **midas Gen** 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 or ***Mode> Preprocessing Mode*** in the Main 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 Gen 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 Gen**.

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 results obtained from 2 simple load combinations are analyzed. The selected load combinations are arbitrary, which do not reflect the real conditions of the structure.

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>Combinations*** of the Main Menu.

---

1. Select ***Results>Load Combination*** 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 of ***Load Combination List***.
5. Enter “**1.0 DL + 1.0 LL**” in the ***Description*** field.
6. Click the ***Load Case*** selection field of ***Loadcases 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 of ***Load Combination List***.
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***.

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.1.22-❶) has to disappear to complete the input. Select another cell to eliminate the 'Edit-in – progress' symbol and click Close.

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.

❶

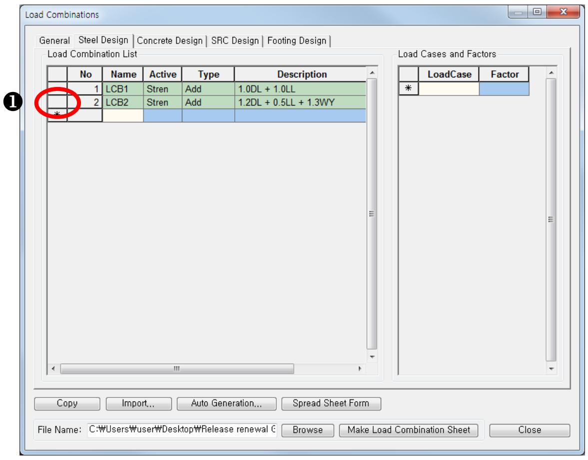


Figure 1.22 Load Combination Cases

Verify Reactions

To verify the reaction results at all the supports after the analysis, select ***Results>Reactions>Reaction Forces/Moments*** from the Tree Menu (or ***Result>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 Tree Menu.
3. Select "**CBS:LCB1**" (Load Combination 1) from the ***Load Cases / Combinations*** selection field.
4. Select "**FZ**" from the ***Components*** selection field.
5. Check (✓) **Values** and **Legend** in the ***Type of Display*** selection field.
6. Click Apply.

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.

To verify the analysis results in the post-processing mode, it is easier to use Result Toolbar rather than Node and Element Toolbar (Fig.1.23-❶).

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.

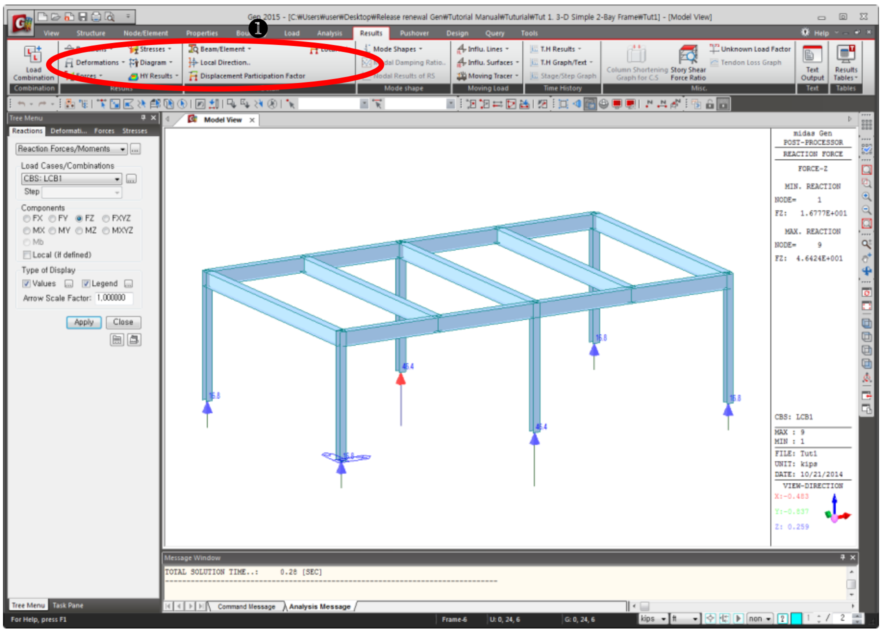


Figure 1.23 Reaction Forces

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

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

---

1. Select ***Search Reaction Forces/Moments*** from the functions field (Fig.1.24–❶).
2. Click ***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.1.24-❸).

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:

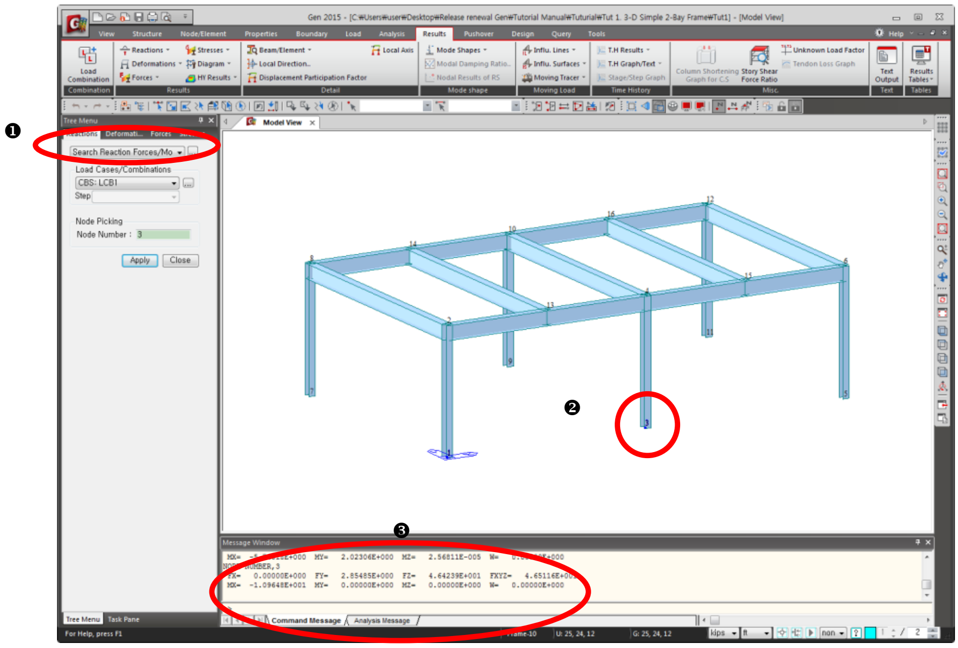


Figure 1.24 Verification of Reaction Forces at Specific Supports

1. Select ***Results>Result Tables>Reaction*** from the Main Menu.
2. Remove the check (✓) in **LL**, **WX** and **WY** in the ***Records Activation Dialog*** box and check (✓) only in **LCB1(CBS)**.
3. Click OK.
4. Select each of the ***Node***, ***FX***, ***FY*** and ***FZ*** fields by clicking 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***.

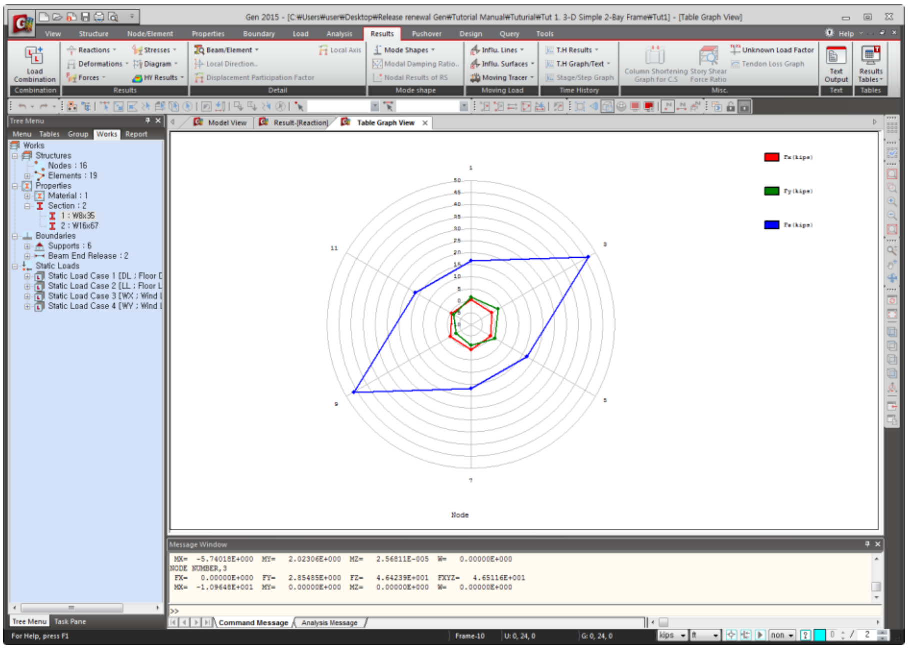


Figure 1.25 Web Chart showing Reaction Forces

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.1.25-❶ to close the ***Table Graph View*** and ***Result-[Reaction]*** windows.
2. Click ***Node Number*** (Toggle off) in the Icon Menu.
3. Select ***Deformations*** from the functions tab (Fig.1.26–❶).
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 Apply .
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 OK .

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.

❶

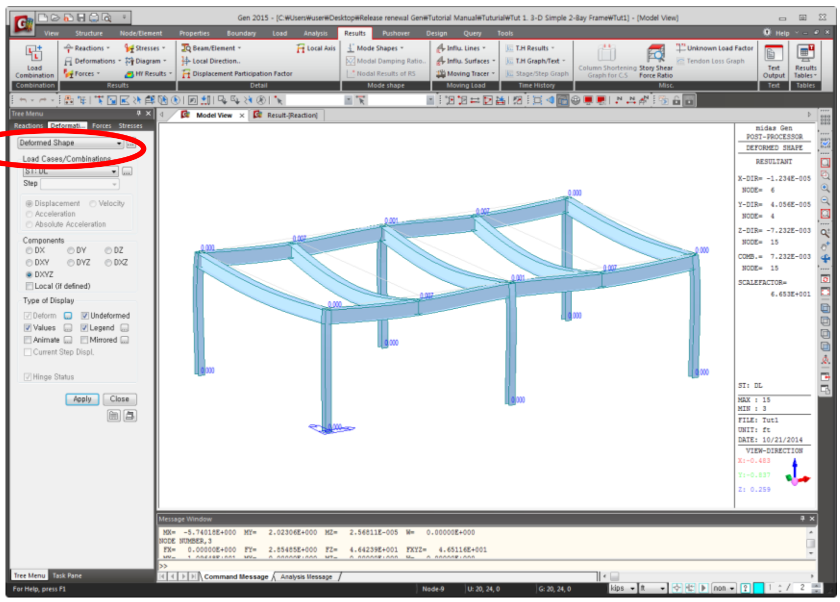


Figure 1.26 Deformed Shape

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 [OK].
5. Click [▼] in the unit conversion button at the bottom of the window (Fig.1.27–❶) and select "**in**".

🎧 Click [...] to the right of Values in Type of Display to adjust the decimal points of the values displayed.

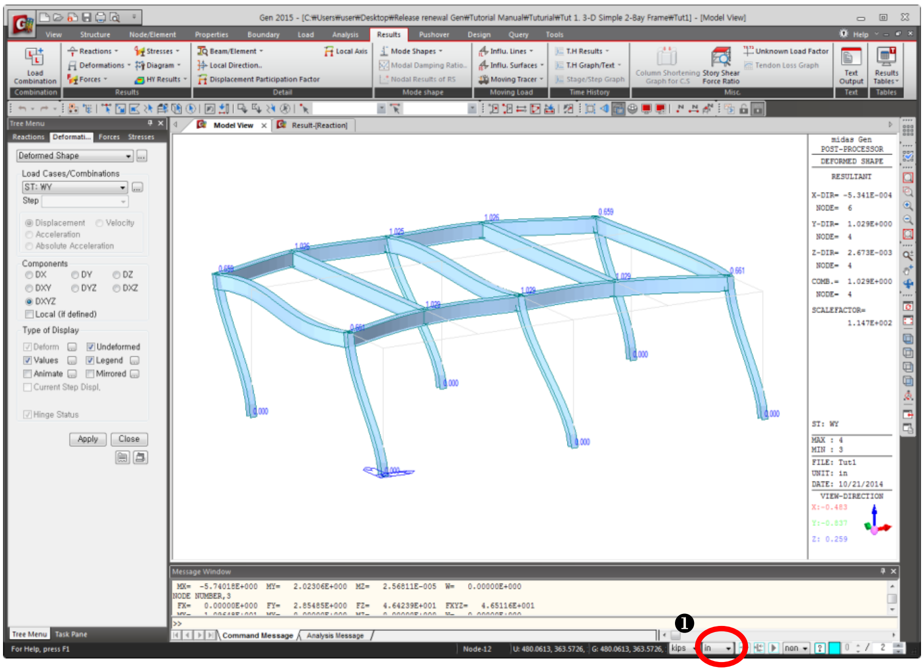


Figure 1.27 Deformed Shape (Scale Factor = 5.0)

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).

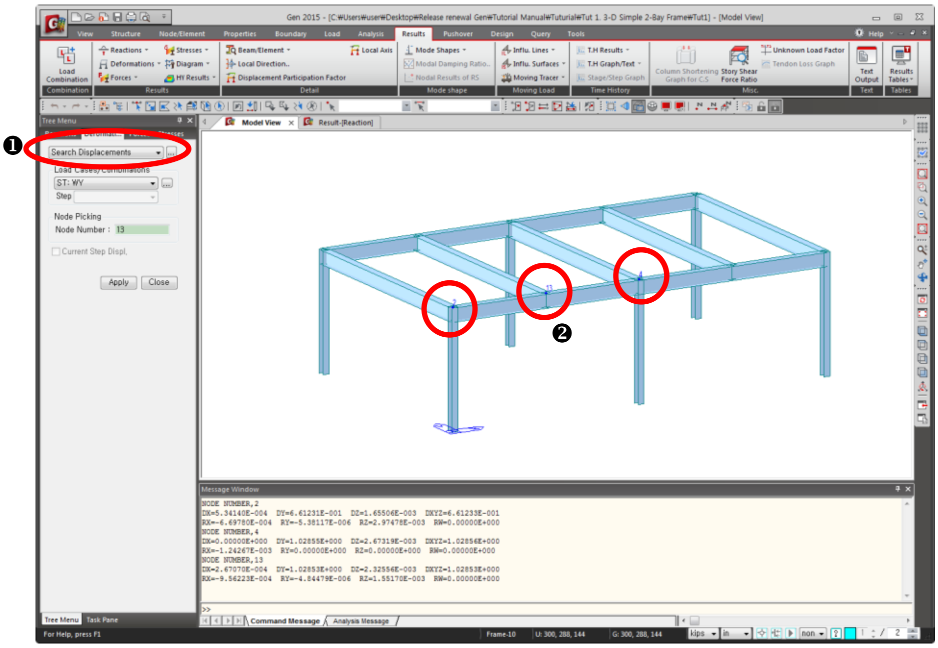


Figure 1.28 Verification of Displacements at Specific Nodes

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
DS: Steel Design tab
DC: Concrete Design tab
DF: Footing Design tab
DR: SRC Design tab

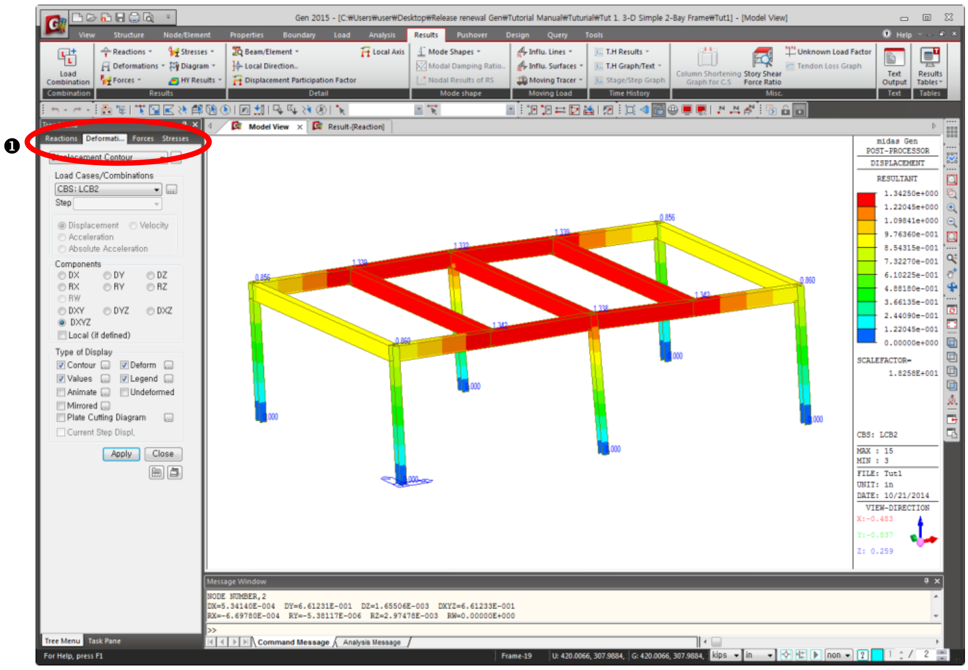


Figure 1.29 Deformed Shape (Contour lines)

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***.

1. Click ***Perspective*** (Toggle on) in the Icon 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 [OK].
7. Click [...] to the right of ***Deform***.
8. Enter “**3**” in ***Deformation Scale Factor*** and click [OK].
9. Click [Apply].

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.

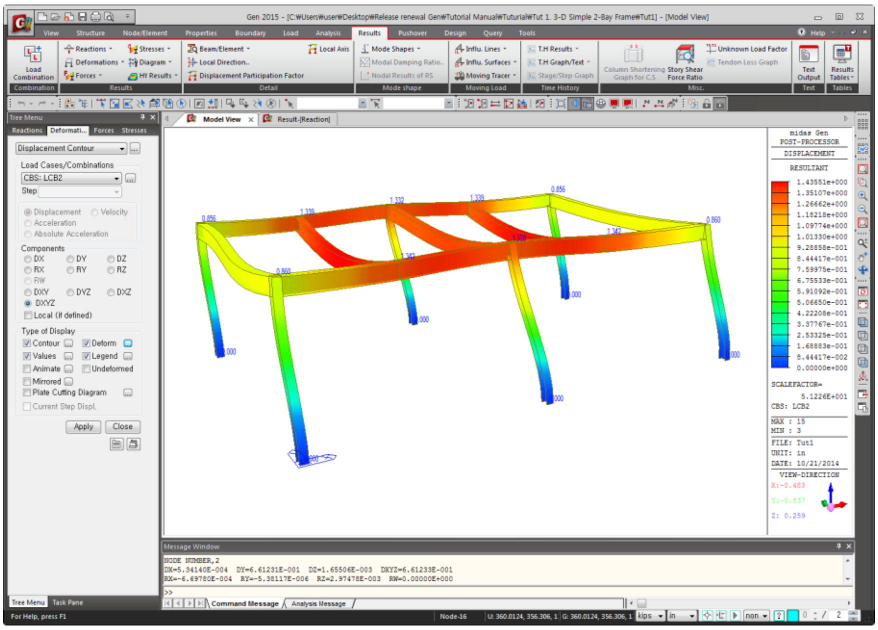


Figure 1.30 Deformed Shape (Contour lines–Gradient Fill)

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 Icon Menu.
3. Select ***Forces*** from the functions tab (Fig.1.31–❶).
4. Select ***Beam Forces/Moments*** from the functions selection field.
5. Confirm "**My**" in the ***Components*** selection field.
6. Check (✓) **Contour**, **Values** and **Legend** in the ***Type of Display*** selection field.
7. Click the button ... to the right of ***Values*** and modify ***Decimal Points*** to "**1**".
8. Click OK.
9. Check (✓) **All** in the ***Output Section Location*** selection field.
10. Click Apply.

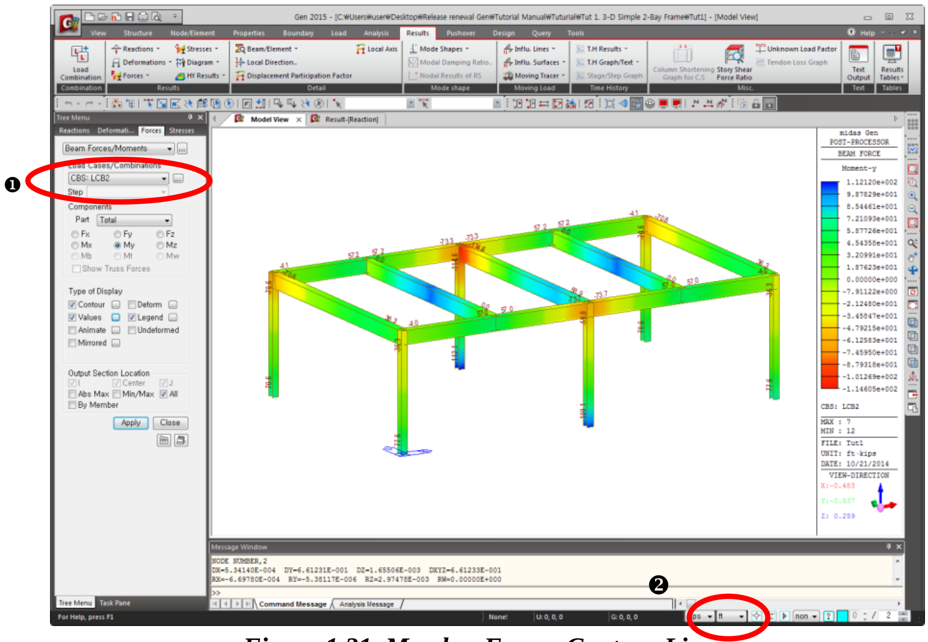


Figure 1.31 Member Forces Contour Lines (Bending moments about y-axis in the ECS)

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.1.32–❶).
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 Gen 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:

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

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***.

❶

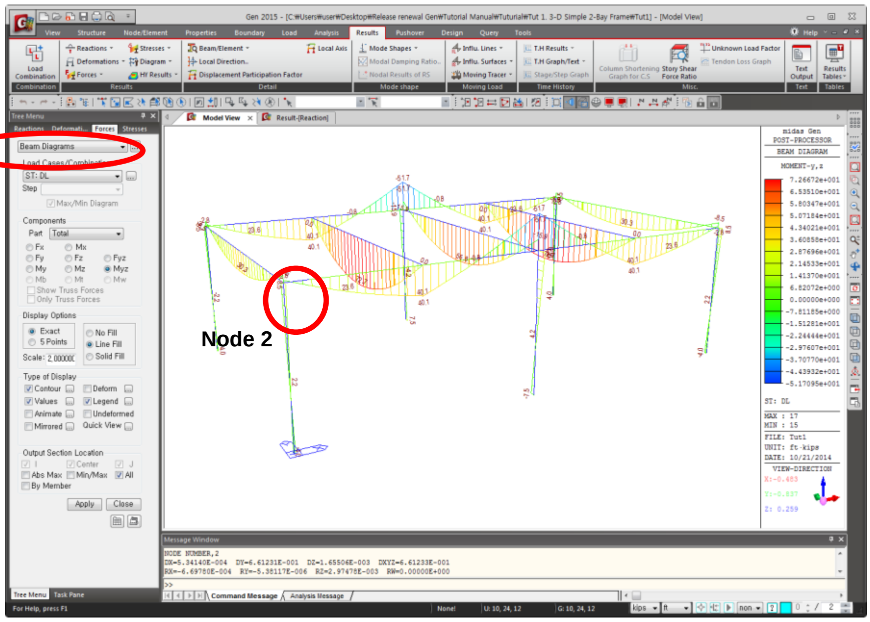


Figure 1.32 Bending Moment Diagram

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 ***Select Plane*** in the Icon Menu.
2. Select "**YZ Plane**".
3. Click a node located on the plane containing ① in Fig.1.1.
4. Click Close.
5. Click ***Activate*** in the Icon Menu.
6. Click ***Right View*** in the Icon Menu.

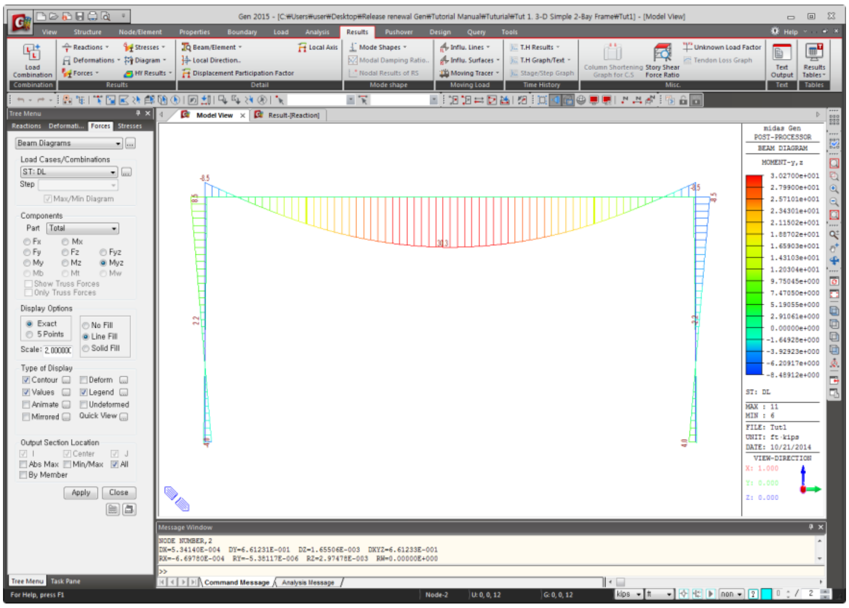


Figure 1.33 Bending Moment Diagram in Y-Z Plane

Using **midas Gen**'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 ***Activate All*** in the Icon Menu.
2. Click ***Isometric 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 ***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 enter 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.

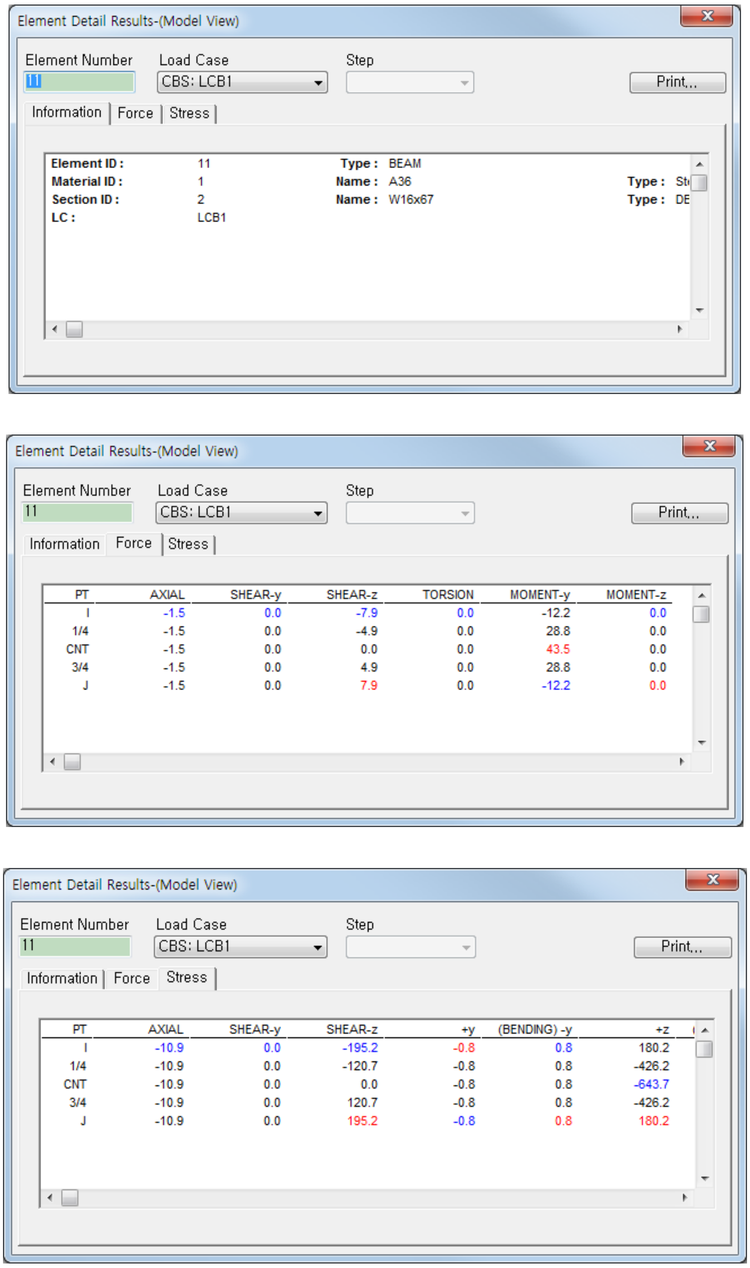


Figure 1.34 Element Detail Results

Verify Member Stresses and Manipulate Animation

midas Gen 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.

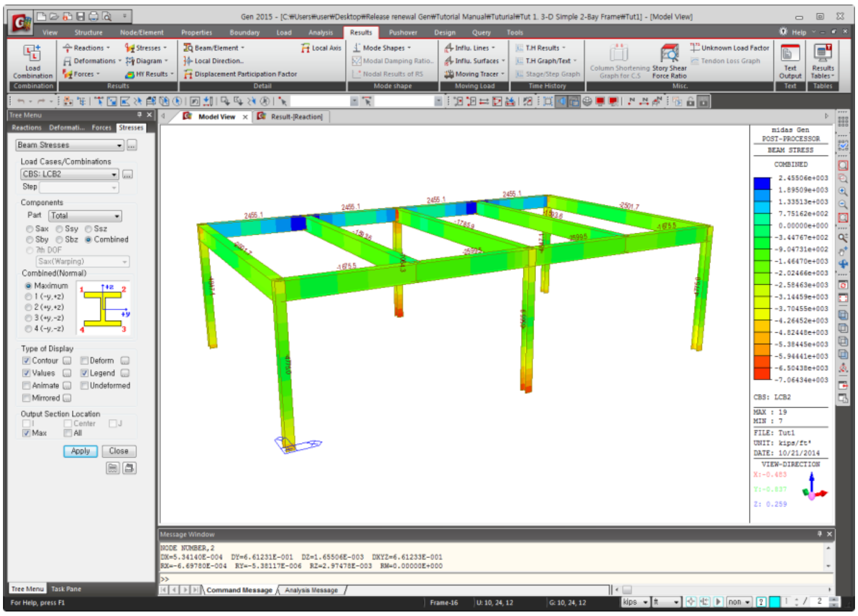


Figure 1.35 Combined Stresses in Beam Elements

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

The summary of ***Dynamic View*** supplied by **midas Gen** 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.1.36–❶), located vertically on the bottom-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:

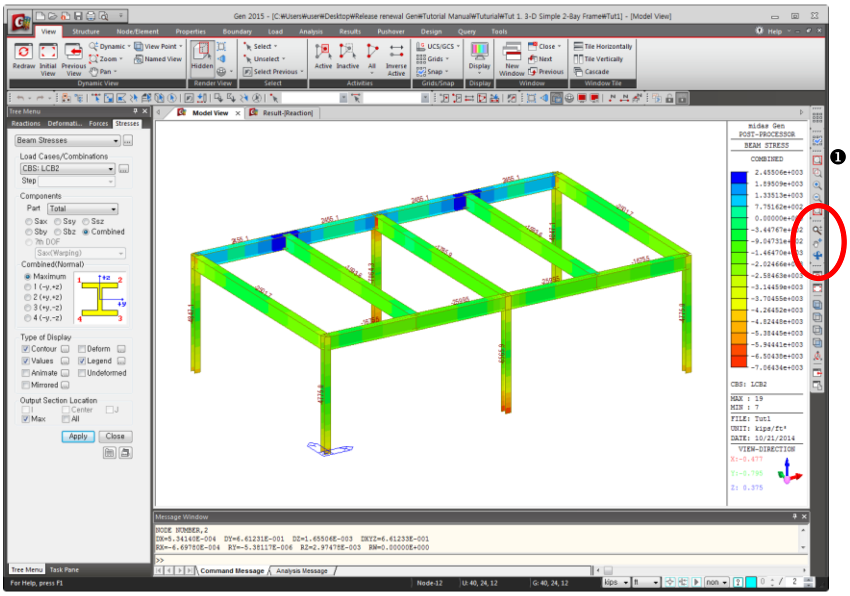


Figure 1.36 Eye Level View

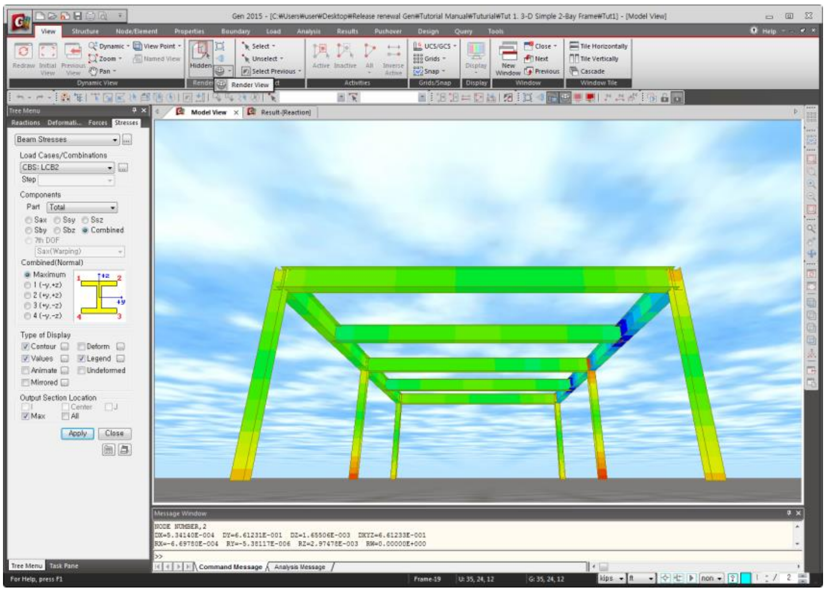


Figure 1.37 Render View

1. Click ***Render View*** in the Icon 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:

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 OK.
7. Click ***Record*** as shown in Fig.1.38–❶.

The representative *icons* controlling the animation are listed below.

Play
Pause
Stop
Skip Back
Rewind
Fast Forward
Skip Forward
Save
Record
Close

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.1.38.

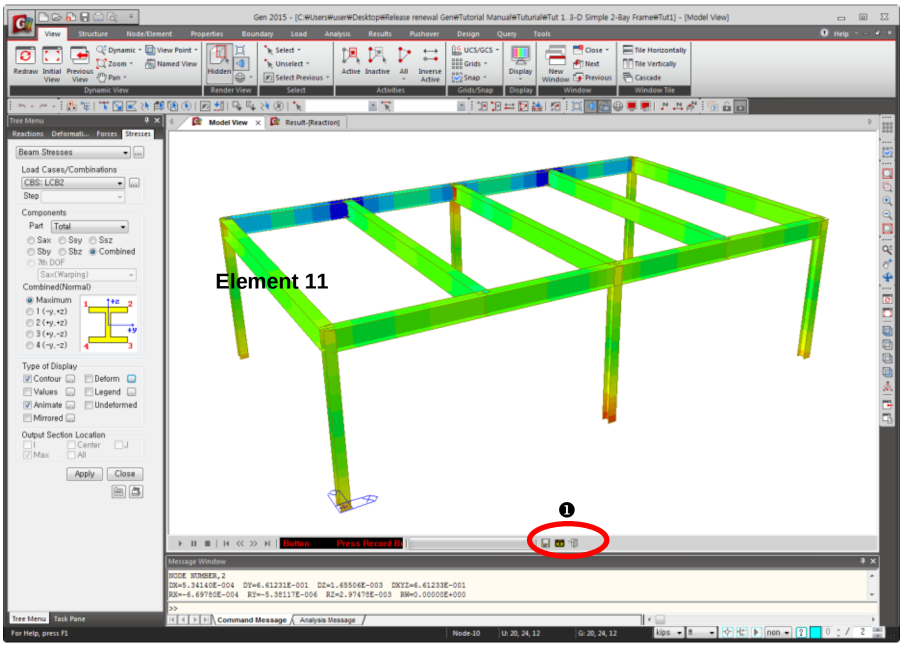


Figure 1.38 Animation Window

Beam Detail Analysis

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

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

- 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 [icon] **Close** shown in Fig.1.38–❶.
2. Select ***Results>Beam Detail Analysis*** from the Main Menu.
3. Select "**ST:DL**" from the ***Load Cases/Combinations*** selection field.
4. Click the ***Element Number*** field once, then select element **11** in the ***Model View*** window (Fig.1.38)
5. Click [icon] 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.1.39–❶.

If the entire screen does not appear in the Model Window, use the right scroll bar.

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

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

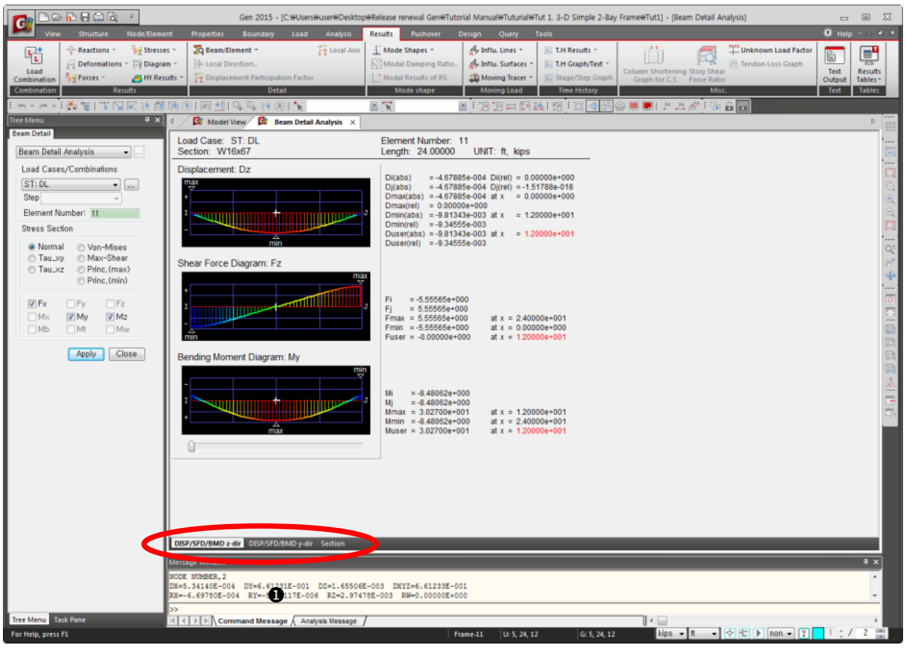


Figure 1.39 Beam Detail Analysis (DISP/SFD/BMD z-dir)

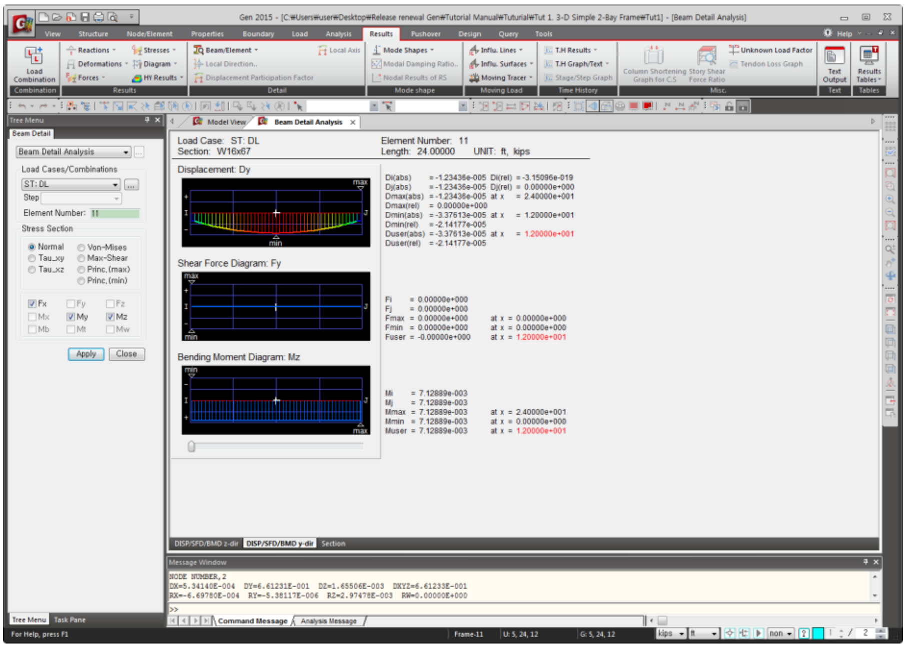


Figure 1.40 Beam Detail Analysis (DISP/SFD/BMD y-dir)

🎧 Picture of the lower flange of a section after selecting Normal in Stress Section.

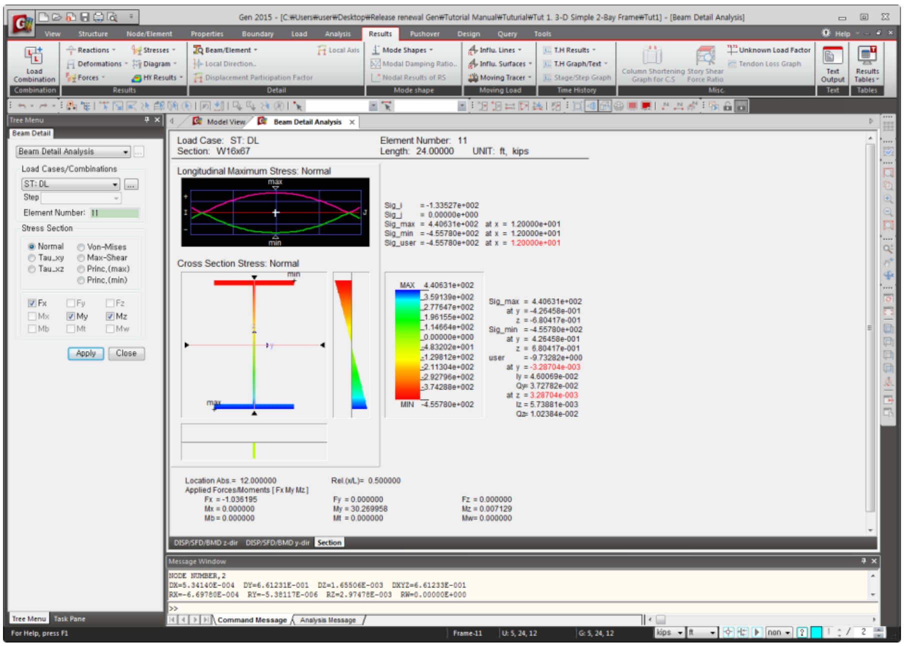


Figure 1.41 Beam Detail Analysis (Section)