For any enquiry concerning the program or related materials, please contact the following:

MIDASoft Inc.
38701 Seven Mile Road, Suite 260
Livonia, MI 48152, USA
MIDASoft
Modeling, Integrated Design & Analysis Software
Phone: 1-800-584-5541
E-mail: **MIDASoft@MidasUser.com**
http://www.MidasUser.com

PROGRAM VERIFICATION AND PRECAUTIONS BEFORE GETTING STARTED

MIDAS Family Programs produce accurate analysis results based on up-to-date theories and numerical techniques published in recognized journals. The program has been verified by thousands of examples and comparative analyses with other S/W during the development.

Since the initial development in 1989, **MIDAS Family Programs** have been accurately and effectively applied to over 4000 domestic and overseas projects.

A strict verification process of the Computational Structural Engineering Institute of Korea has scrutinized MIDAS Family Programs.

Due to the complexity of structural analysis and design programs which are based on extensive theories and design knowledge, the sponsors, developers and participating verification agencies do not assume any rights or responsibilities concerning benefits or losses that may result from using **MIDAS Family Programs**. The users must understand the bases of the program and the User's Guide before using the program. The users must also independently verify the results produced by the program.

DISCLAIMER

The developers and sponsors assume no responsibilities for the accuracy or validity of any results obtained from **MIDAS Family Programs** (MIDAS/Gen, MIDAS/SDS, MIDAS/Set, MIDAS/FEmodeler, MIDAS/Civil, MIDAS/FX+ and MIDAS/GTS, also referred to as "MIDAS Package" hereinafter).

The developers and sponsors shall not be liable for loss of profit, loss of business, or any other losses, which may be caused directly or indirectly by using the MIDAS package due to any defect or deficiency therein.

Preface

Welcome to the **MIDAS/Gen programs**.
MIDAS/Gen is a program for structural analysis and optimal design in the civil engineering and architecture domains. The program has been developed so that structural analysis and design can be accurately completed within the shortest possible time. The name **MIDAS/Gen** stands for ***Gen**eral structure design*.

About MIDAS/Gen and MIDAS Family Programs

MIDAS/Gen is a part of **MIDAS Family Programs** that have been developed since 1989.
MIDAS Family Programs are groups of Package Software that systematically integrates the entire design process generally encountered in the design of structures. **MIDAS Family Programs** consist of the following entities:

MIDAS/Gen	*General structure design system* Structural analysis and optimal design system for general structural engineering applications, especially in building design
MIDAS/SDS	*Slab & basemat Design System* Structural analysis and optimal design system for slabs and basemats
MIDAS/Set	*Structural Engineer's Tools* Collection of individual programs to expedite the design of structural units
MIDAS/FEmodeler	*finite element MESH generator* Program for automatic generation of finite element meshes
MIDAS/Civil	*CIVIL structure design system* Structural analysis and optimal design system for exclusive applications in civil engineering structures, especially in bridge design.
MIDAS/FX+	*General Pre & Post-processor for Finite Element Analysis* General purpose, FEA (Finite Element Analysis) pre & post-processing in CAE (Computer Aided Engineering)
MIDAS/GTS	*Geotechnical and Tunnel analysis System* Integrated solution for tunnel and geotechnical specific structures

Among **MIDAS Family programs**, “MIDAS/Gen”, “MIDAS/Civil”, “MIDAS/SDS”, “MIDAS/Set”, “MIDAS/FEmodeler” “MIDAS/FX+” and “MIDAS/GTS”, are currently in use and have been applied to over 5,000 projects.

Advantages and Features of MIDAS/Gen

MIDAS/Gen has been developed in Visual C++, an object-oriented programming language, in the Windows environment. The program is remarkably fast and can be easily mastered for practical applications. By using the elaborately designed GUI (*Graphic User Interface*) and the up-to-date Graphic Display functions, a structural model can be verified at each step of formation and the results can be directly set into document formats.

During the development process, **MIDAS/Gen** has been verified through numerous examples. Each of the functions has been verified by comparing the results with theoretical values and output from other similar programs. The program has been applied to over 5,000 projects and the reliability and effectiveness have been established.

Representative examples are in the Verification Manual. The latest theories form the bases for the finite element algorithm that determines the accuracy of analysis results. Excellent results are achieved compared to other similar programs.

Closing Remarks

MIDAS/Gen has been conceived as a result of the cooperation and efforts by a number of engineers and professors. We expect that MIDAS/Gen users will be pleasantly surprised with satisfying results. The users are encouraged to contact MIDAS IT to suggest any improvements that they feel can be implemented in subsequent versions.

In closing, we extend our gratitude to everyone who participated in the development of MIDAS/Gen.

About the User's Guide

The User's Guide for **MIDAS/Gen** consists of the following 3 volumes and the On-line Manual:

Volume 1	**Getting Started & Tutorials** Summary of the program contents and items to become familiarized before getting started with the tutorial examples
Volume 2	**Analysis** Explanation of the analysis backgrounds
Volume 3	**Verification Examples** Illustration of verification examples
On-line Manual	Detailed directions and explanations for each built-in function

Understanding the User's Guide is essential in effectively learning the characteristics and functions of **MIDAS/Gen**. The following is a recommended reading sequence before getting started with the program.

First, read the commentaries on the structural analysis and design functions of **MIDAS/Gen** in Volume 2. Volume 2 describes the fundamentals necessary to perform finite element analysis using **MIDAS/Gen.** Some technical journals have reported that the probability of incurring errors exceeds 90% when programs are used with poor knowledge of analysis theories and of the programs.

Install **MIDAS/Gen** following the procedure described in the "Installation" section of Volume 1. Read other parts of Volume 1, which outline the fundamental concepts necessary to run **MIDAS/Gen**. Also contained in Volume 1 are the following: the directions for various functions to run **MIDAS/Gen** efficiently, functions for modeling such as "Preferences setting", "Input Data", "Manipulation of Model Window", "Selection Functions and Activation/Deactivation Functions", and functions required for real analysis operations such as "Modeling", "Analysis", "Interpretation of Analysis Results", etc.

Detailed directions and explanations for each function are described in the On-line Manual that can be accessed from the Help Menu of **MIDAS/Gen**.

The "Tutorials" supply the modeling, analysis and results interpretation processes of simple structural examples. Subsequently, practice **MIDAS/Gen** following the procedures described in the "Tutorials" of Volume 1. The Tutorials are organized so that when all the step-by-step stages from modeling to the analysis and design of practical examples are followed, the user understands and acquires the capabilities of the program. If, at any time, some contents remain misunderstood, the user may refer to the relevant sections contained in the On-line Manual.

Volume 3 presents principal analysis functions where the results have been verified by comparisons with theoretical values and results from other programs. Because the verification examples are simple problems commonly introduced in the academic courses, these examples can be practically used by the novice in structural analysis as materials to understand the concepts related to the fundamentals of structural analysis. Representative examples have been selected and included in the Verification Manual. Contemporary theories have been applied to the finite element algorithm that determines the accuracy of analysis results. Compared to the results from other similar programs, MIDAS/Gen produces excellent results.

MIDAS/Gen 6

MIDAS/Gen

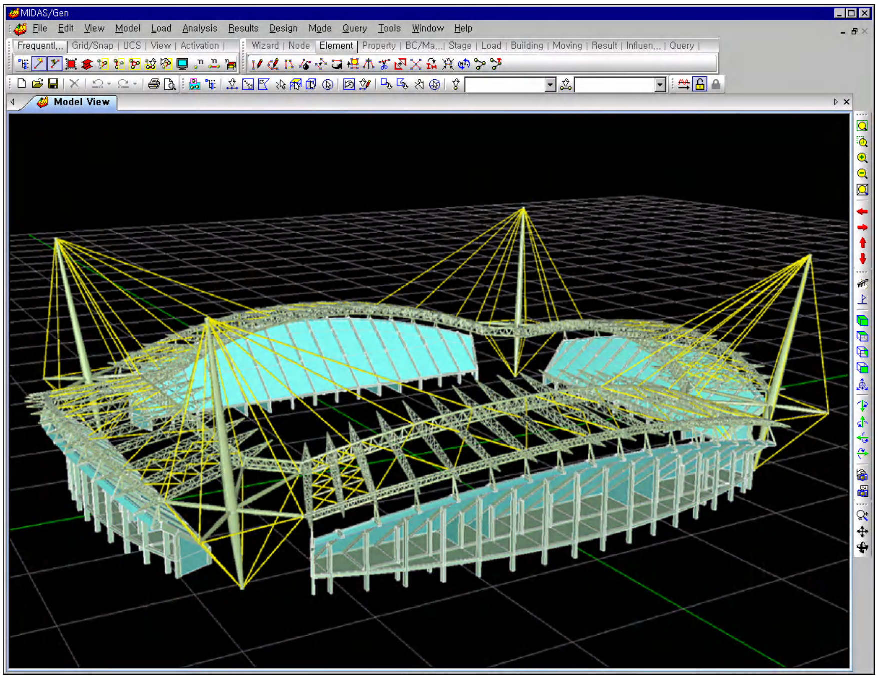
Getting Started

INDEX

About MIDAS/Gen

Summary

MIDAS/Gen stands for "**Gen**eral structure design system.", i.e., a Windows based integrated system for structural analysis and optimal design.

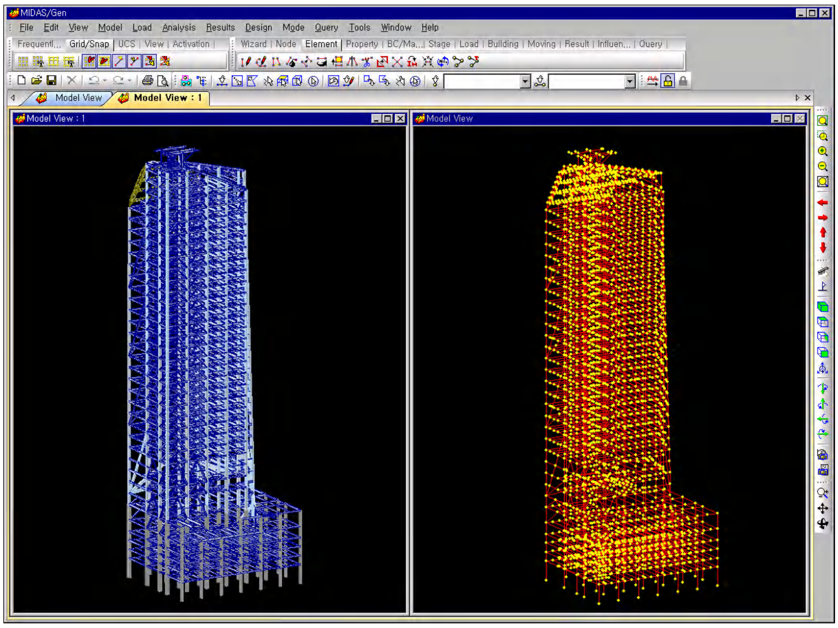


2002 FIFA World Cup Stadium (Jeonju)

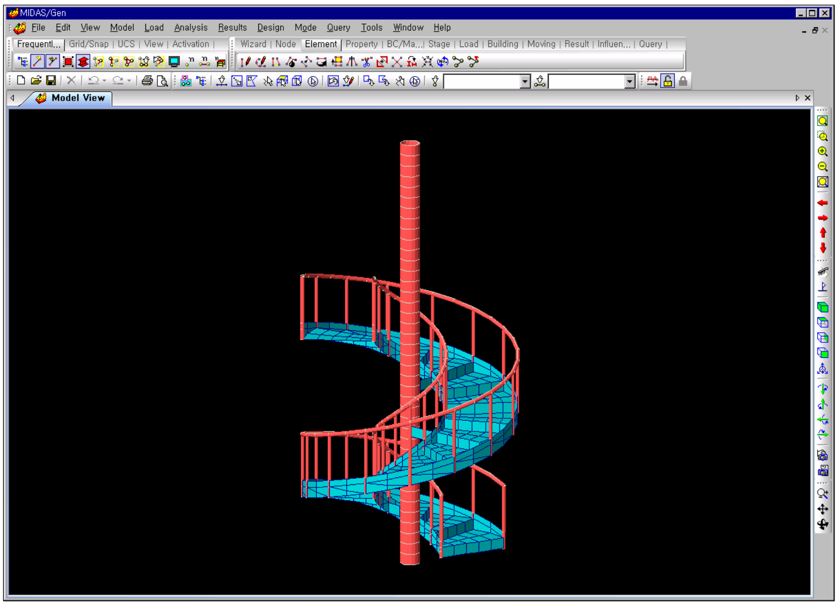
MIDAS/Gen, developed in the object-oriented programming language Visual C++, fully exploits the advantages and the characteristics of the 32bit Windows environment for the technical computations.

The user-oriented input/output functions are based on sophisticated and intuitive *User Interface* and up-to-date *Computer Graphics* techniques. They offer excellent facilities and productivity for the modeling and analysis of complex, large-scale structures.

The technical aspects of structural analysis functions necessary in a practical design process are substantially strengthened. Nonlinear elements such as ***Cable***, ***Hook***, ***Gap, Visco-elastic Damper, Hysteretic System, Lead Rubber Bearing Isolator and Friction Pendulum System Isolator*** are now included in the *Finite Element Library,* which will surely improve the accuracy and the quality of results. Construction stages, time dependent material properties and geometric/boundary nonlinear analyses are some of the new inclusions.



Surface View and Wire-frame View (Posteel)



Analysis model of a spiral staircase

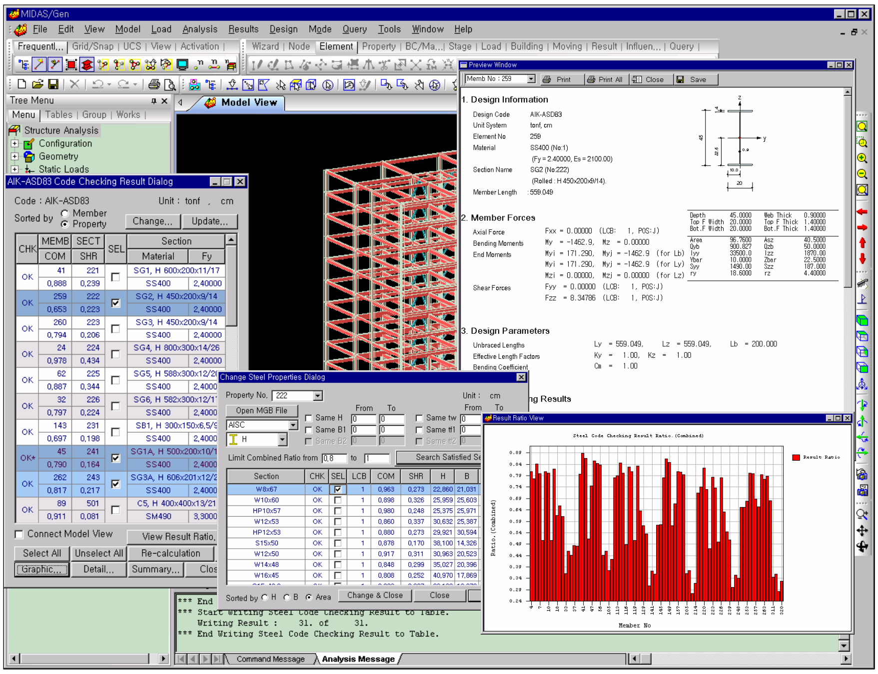
MIDAS IT's in-house researchers have developed an efficient CAD modeling technique, which is a totally new concept. Powerful automatic modeling functions such as ***Auto Mesh Generation*** (available as a separate module) and ***Structure Wizard*** are introduced. Also, a new Multi-Frontal Sparse Gaussian Solver has been added lately, which has accelerated the analysis speed dramatically.

Latest design standards are adopted in the design module. To list a few, they are ACI, AISC (ASD & LRFD), BS, Eurocodes, etc.

The ***Optimal Design*** function considers various design constraints and leads to weight optimization in the design of steel frame structures. It offers practical, convenient and accurate results.

Refer to "Appendix A. Principal Features of **MIDAS/Gen**" for more information.

🎧 After designing a plant structure, the detailed calculation for a particular member has been carried out. Automatic optimal design, the combined stress ratio and weight distribution by section properties of the structure are graphically displayed.



Results of strength verification according to AISC Design Standards

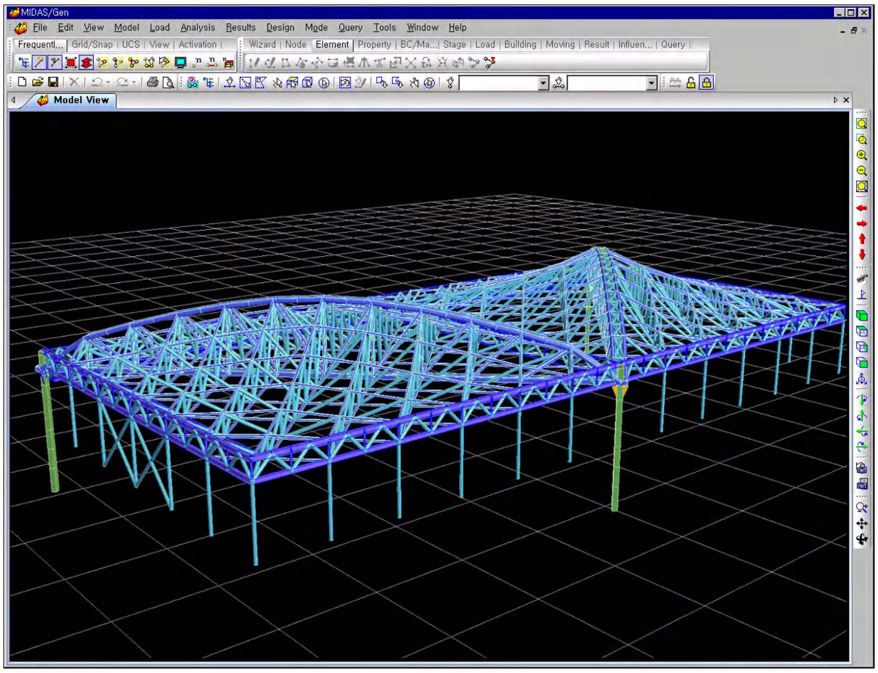
The domains of applications for **MIDAS/Gen** are as follows:

- **Civil engineering structures**
  Bridges, underground structures, water tanks, dams, etc.

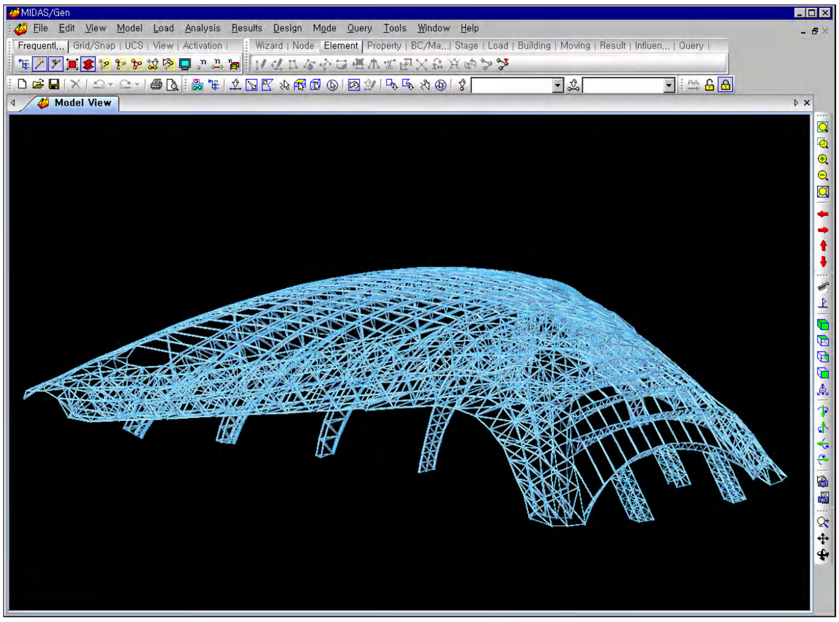
- **Architectural structures**
  Office buildings, residential buildings, commercial buildings, complex multi-use buildings, plants, maritime/offshore structures, etc.

- **Special structures**
  Stadiums, hangars, power plants, etc.

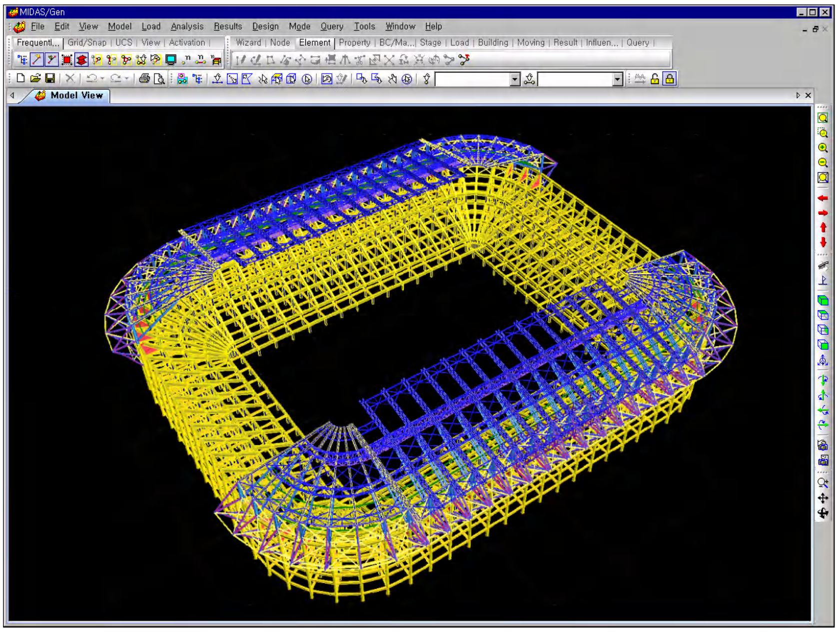
- **Other structures**
  Ships, airplanes, power line towers, cranes, pressurized vessels, etc.



Analysis model of KAL O/C hangar, Kimpo International Airport



Analysis model of Transportation Complex, Inchon Int. Airport



Analysis model of Daejeon 2002 World Cup Stadium

Installation

System Requirements

MIDAS/Gen operates on IBM compatible Personal Computer (PC) in Windows environment.

In addition, **MIDAS/Gen** requires the following minimum configuration:

- Pentium or better performing PC processor
- Minimum of 64MB RAM
- 500 MB of free space on HDD (**MIDAS/Gen** requires a minimum of 1 GB hard disk space for Construction Stage analysis)
- Microsoft Windows 95 or higher version or Windows NT Operating System
- Windows-supported Graphics card, Monitor with a minimum of 1024×768 resolution and a minimum of 16bit High Color display
- Windows compatible Printer or Plotter

Installation Sequence

Installing MIDAS/Gen

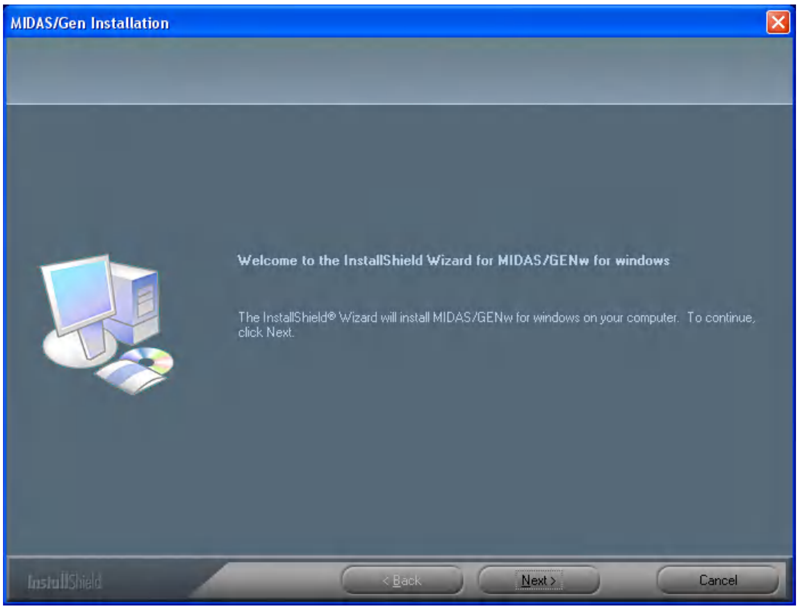
Follow the steps below to install **MIDAS/Gen**.

---

1. Insert **MIDAS/Gen** CD into the CD-ROM drive.

2. MIDAS Gen Installation
   When the automatic installation does not proceed, select the ***Run*** command in the ***Start*** menu of Windows. Once the CD-ROM drive is assigned, enter the following command:

   D:\setup
   (Note: this is the case where CD-ROM drive is assigned to the directory D)



Installation dialog box of MIDAS/Gen

3. Once the installation program is initiated, the dialog box shown in the figure above is displayed and the installation of **MIDAS/Gen** begins. The installation will proceed step-by-step to the subsequent phases following the displayed information. To proceed to the next step, click Next >. To return to the previous step, click < Back.

4. MIDAS/Gen will be installed only in the system where Internet Explorer version 5.0 or higher has been installed. Install Internet Explore if not already installed and install MIDAS/Gen.

MIDAS/Gen information dialog box

5. When the license agreement dialog box is displayed, read the agreement carefully. If the terms and conditions are agreeable click Yes, and the installation will continue.

6. Enter the user's registration information and click Next >.

7. The directory selection dialog box will appear. Select the folder in which **MIDAS/Gen** will be installed. **MIDAS/Gen** can be installed in the default folder by clicking Next >. To change the folder, click Browse... and choose the folder in which to install **MIDAS/Gen**.

8. Once the program folder selection dialog box is displayed, select a folder name for the registration of **MIDAS/Gen** icons and other related programs. Click the Next > button, and copying the files will begin.

9. Once the copying of the files is complete, the "installation completed" message dialog box will appear. Click Finish and the installation process now will be completed. If at this time "Run MIDAS/Gen Now" is checked and Finish is clicked, then the installation will be completed and MIDAS/Gen will be executed immediately.

Install Sentinel/pro Driver

The Sentinel Driver is used to drive the Lock key of Sentinel hardware. To run **MIDAS/Gen** and the Lock key the driver has to be installed. The Sentinel Driver is installed automatically during the installation process of **MIDAS/Gen**. For upgrading or replacing a damaged Lock driver, follow the procedure outlined below.

To install the Sentinel Driver manually follow these steps.

1. Press the left side ***Shift*** key and insert the **MIDAS/Gen** CD in your CD-ROM drive.

2. Select the ***Run*** command in the ***Start*** menu. Once the CD-ROM drive is assigned, enter the following command:

   D:\protection drivers\setup
   (Note: this is the case where CD-ROM drive is assigned to the directory D)

To uninstall the Sentinel Driver follow these steps.

1. Press the left side ***Shift*** key and insert the **MIDAS/Gen** CD in the CD-ROM drive.

2. Select the ***Run*** command in the ***Start*** menu. Once the CD-ROM drive is assigned, enter the following command:

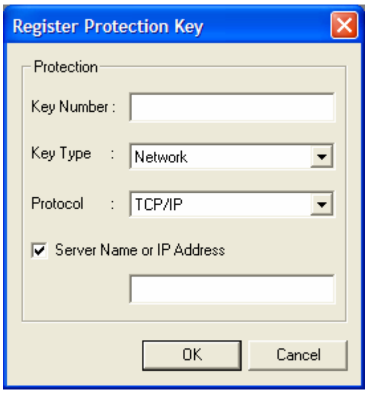
   D:\protection drivers\setup /u
   (Note: this is the case where CD-ROM drive is assigned to the directory D)

Registering the Protection Key

To operate **MIDAS/Gen** properly, register the serial number after connecting the protection key to the parallel port.

1. Connect the Protection Key to the Parallel Port.
2. Execute **MIDAS/Gen**.
3. Select ***Register Protection Key*** on the ***Help*** menu.
4. Enter the ***Protection Key ID*** provided in the Program CD Case in the Protection Key field.
5. Click 

.



Register Protection Key

Before Getting Started

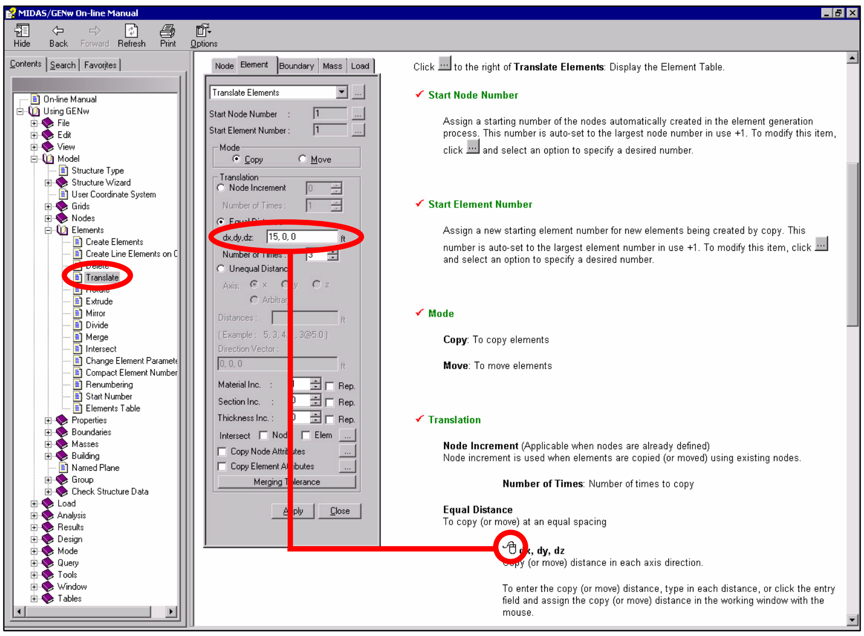
How to Use the On-line Manual

When using **MIDAS/Gen**, pressing F1 key or clicking the Help menu can always allow us to access the On-line Manual.

Every category of help is connected to related keywords by hyperlink, and all the detailed explanations and information in connection with the keyword may be obtained.

A summary of the help contents and an index of the main keywords are arranged systematically in the ***On-line Manual*** of **MIDAS/Gen**. Read it as a reference in the order presented in the summary. Alternatively, the information regarding the desired item may be directly obtained using the ***Search*** function of the keywords.

Symbol in On-line manual signifies that the Mouse editor is supported for the corresponding data entry field. The Mouse editor replaces the keyboard function for defining materials, distances, etc. on the screen.



On-line Manual of MIDAS/Gen

If the ***Midas on the Web*** feature of **MIDAS/Gen** is used, the website of MIDASoft (http://www.MidasUser.com, MIDASoft@MidasUser.com) can be directly connected, and e-mails can be sent.

Recognition of Input/Output Files

The types of files, their purposes and the generation process are as follows:

Data Files

Refer to Tools>MGT Command Shell in On-line Manual.

fn.mgb	Binary	The basic data file of **MIDAS/Gen** During the initial generation, use ***File>New Project***. When opening an existing file, use ***File>Open Project***.
fn.mgt	Text	The basic data file of **MIDAS/Gen** If necessary, it can be modified using ***Text Editor***. The user may transform the data generated by **MIDAS/Gen** into a format suitable for other S/W. The data file can also be used for ***MGT Command Shell***. ***File>Export>Gen MGT File*** creates a file and ***File>Import>Gen MGT File*** recalls the file in the format used by MIDAS/Gen model data.
fn.wpf	Text	Wind loading data file that MIDAS/Gen automatically calculated Click [Make Wind Load Calc. Sheet] in ***Load>Wind Loads>Add/Modify Wind Load Code>Wind Load Profile*** to create this file.
fn.spf	Text	Equivalent static seismic loading data file that **MIDAS/Gen** automatically calculated Click [Make Seismic Load Calc. Sheet] in ***Load>Static Seismic Loads>Add/Modify Seismic Load Design Code>Seismic*** to create this file.

Analysis Output Files

fn.ga1	Binary	Data file obtained from a static/dynamic analysis process File generated ***automatically*** by ***Analysis>Perform Analysis***
fn.ga2	Binary	Analysis results generated for each time step from a time history analysis and a heat of hydration analysis File generated automatically by ***Analysis>Perform Analysis***
fn.ga4	Binary	File for all the analysis data generated in the process of a geometric nonlinear analysis File generated automatically by ***Analysis>Perform Analysis***
fn.ga5	Binary	File for all the analysis data generated in the process of a pushover analysis File generated automatically by ***Design>Perform Pushover Analysis***
fn.ga6	Binary	File for all the analysis data generated in the process of construction stage analysis File generated automatically by ***Analysis>Perform Analysis***
fn.anl	Text	File containing structural analysis results (reactions, displacements, element forces, stresses, etc.) which has been arranged by the user's preference This file is useful for verifying analysis results and preparing calculation sheets. File generated automatically by ***Results>Combinations*** or ***Envelope***
fn.out	Text	All kinds of messages or related data produced during a structural analysis process File generated automatically by ***Analysis>Perform Analysis***.

Design Output Files

fn.gd1	Binary	Design of steel frame elements and all the related data File generated automatically by ***Design>Steel Code Check***
fn.gd2	Binary	Design of RC (reinforced concrete) elements and all the related data File generated automatically by ***Design>Concrete Code Design*** (or ***Concrete Code Check***)
fn.gd3	Binary	Design of footings and all the related data File generated automatically by ***Design>Footing Design***
fn.gd4	Binary	Design of SRC elements and all the related data File generated automatically by ***Design>SRC Code Check***
fn.acs	Text	Data file that contains a summary of structural steel member design results and the detail calculations Click Detail... or Summary... in the design results dialog box after a design.
fn.rcs	Text	Data file that contains a summary of reinforced concrete member design results and the detail calculations Click Detail... or Summary... in the design results dialog box after a design.
fn.src	Text	Data file that contains a summary of structural steel/reinforced concrete composite member design results and the detail calculations Click Detail... or Summary... in the design results dialog box after a design.

Graphic Files

fn.color	Binary	Color data file of MIDAS/Gen Click Save in ***Color*** and ***Print Color*** tabs from the ***View>Display Option***.
fn.emf	Binary	Graphic data file of the model window in the EMF (Enhanced Meta File) format File generated automatically by ***Files>Windows Meta File***.
fn.bmp	Binary	Graphic data file of the model window in the BMP (Bitmap) format File generated automatically by ***Files>Windows Bitmap File***.
fn.mgf	Binary	Graphic data file produced by ***Graphic Editor*** of **MIDAS/Gen** File generated automatically by the ***Save*** function of ***Tools>Graphic Editor***.

Refer to "File>Import/Export/Data Conversion" of On-line Manual.

Data Transfer Files

Fn.mgt	Text	**MIDAS/Gen** text file
Fn.dxf	Text	**AutoCAD DXF** file compatible with data for **MIDAS/Gen**
Fn.s90	Text	Data file of **SAP90** compatible with data for **MIDAS/Gen**
Fn.s2k	Text	Data file of **SAP2000** compatible with data for **MIDAS/Gen**
fn.std	Text	Data file of **STAAD** compatible with data for **MIDAS/Gen**
fn.gti	Text	Data file of **GT STRUDL** compatible with data for **MIDAS/Gen**

Other Files

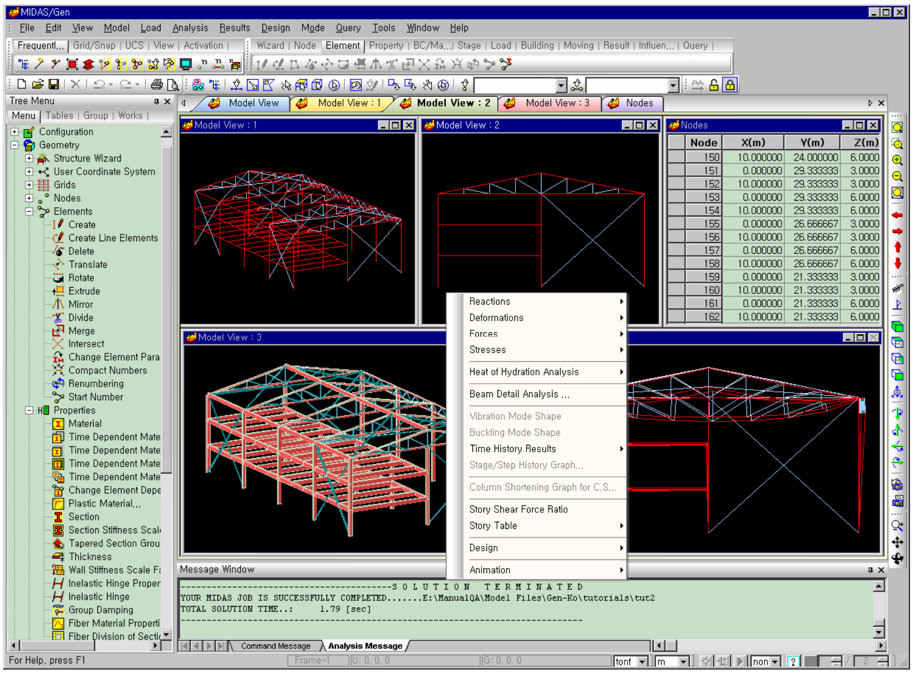
fn.bak	Binary	Back-up data file of MIDAS/Gen Select ***Make Backup File*** in ***Tools>Preferences*** to create the file automatically while saving the model data in progress.
fn.bom	Text	Weight data file of every element included in the modeling and bill of material File generated automatically by ***Tools>Bill of Material***.
fn.sgs	Text	Seismic data file produced by the seismic acceleration and response spectrum generation module of **MIDAS/Gen** It uses ***Tools>Seismic Data Generator***.
fn.spd	Text	Response spectrum data file required for a response spectrum analysis File produced by ***Load>Response Spectrum Analysis Data>Response Spectrum Functions***.
fn.thd	Text	***Time Forcing Function*** data file required for a time history analysis File produced by ***Load>Time History Analysis Data> Time Forcing Functions***.
fn.bog	Binary	File containing the data entered in the ***Batch Output Generation*** dialog box Among the checking features of analysis results of the ***Results*** menu, the  button of the Batch Output Generation dialog box generates the file, which can be accessed by the  button.

Organization of Windows and Menu System

The Menu System of **MIDAS/Gen** permits an easy access to all the functions related to the entire process of input, output and analysis and minimizes the mouse movement.

The ***Works*** tab of ***Tree Menu*** systemizes the entire design process, which allows us to review the status of input at a glance while the ***Drag & Drop*** type of modeling capability allows us to readily modify the data during the modeling process.

The organization of the working windows of **MIDAS/Gen** and the Menu system are as follows:



Organization of the working windows and the Menu system of MIDAS/Gen

Main Menu

When running MIDAS/Gen for the first time, the use of Main Menu is recommended to understand the built-in functions and the working environment. Once the user becomes familiar with MIDAS/Gen, the use of Icon Menu or Context Menu will be more effective.

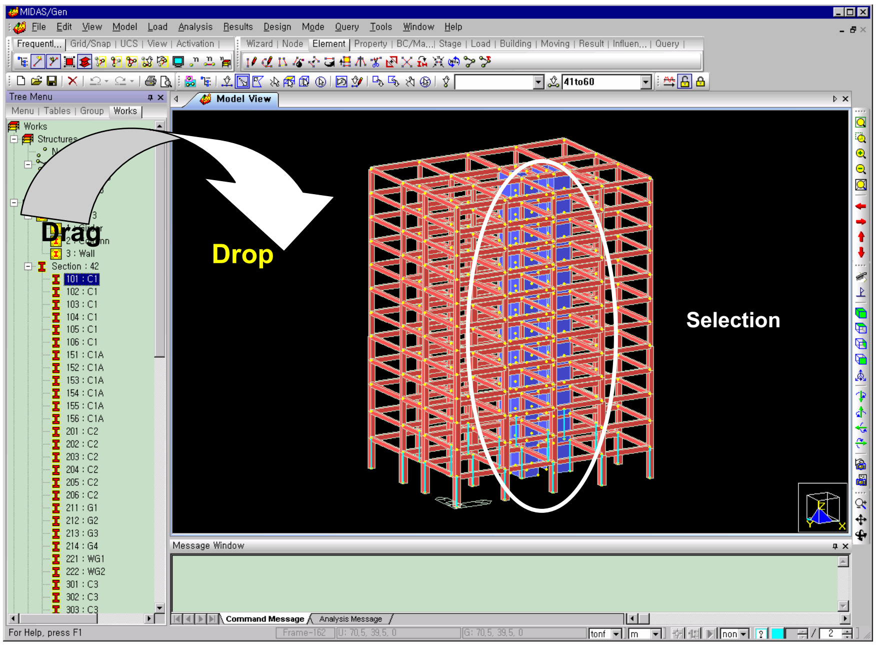
The commands and shortcut keys for all the functions necessary to run **MIDAS/Gen** are built-in.

File	File, print, data transfer and related functions
Edit	***Undo/Redo*** functions and functions related to editing in spreadsheet table window formats
View	Visual presentation method and manipulation functions, selection functions, Activation/Deactivation functions, etc.
Model	Entering model data and automatic generation of grids, nodes, elements, section properties, boundary conditions, masses, etc.
Load	Enter all types of static loads, dynamic loads, thermal loads, automatic generation functions, etc.
Analysis	Enter all types of control data necessary for analysis process and analysis execution functions
Results	Enter load combinations, plotting analysis results (reactions, displacements, member forces, stresses, vibration modes, buckling modes, etc.), verification and analysis functions, etc.
Design	Automatic design of structural steel, SRC, RC and footings, code checking, etc.
Mode	Switch functions between preprocessing and post-processing modes
Query	Status verification functions for nodes, elements and related data
Tools	Assignment of unit system and preferences setting, ***MGT Command Shell***, computation of bill of material, extraction of seismic data, ***Sectional Property Calculator***, etc.
Window	Control functions for every window within the main window and arrangement functions
Help	Help functions and access to MIDAS IT homepage and e-mail.

Tree Menu

The entire procedure for modeling from data entry to analysis, design and preparation of calculations are systemically organized. An expert as well as a novice can efficiently work without making errors by accessing the related dialog boxes, which provide the procedural guidance.

Also, ***Works Tree*** allows the user to glance over the input status of the current model data, which can be revised by the ***Drag & Drop*** capability.



Drag & Drop capability of Works Tree tab

Context Menu

In order to minimize the physical motions of the mouse, simply right click the mouse. **MIDAS/Gen** automatically selects a menu system, which offers related functions or frequently used functions reflecting the working circumstances of the user.

Model Window

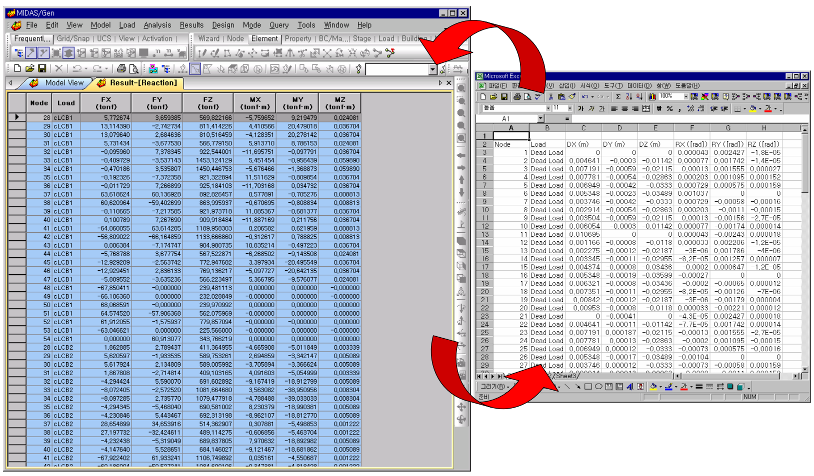
The working window deals with the modeling, interpretation of analysis results and design by means of **GUI** (Graphic User Interface) of **MIDAS/Gen**.

The Model Window may present several windows simultaneously on the screen. Because every window operates independently, different user coordinate systems can be assigned to the individual windows to create a model. In addition, each window shares the same database and as such, the work performed in a window updates the other windows simultaneously.

The Model Window can represent common model shapes as well as shapes generated by up-to-date features such as hidden lines, removal of hidden surfaces, shading, lighting, dispersion of color tone, etc. The model, analysis and design results may be displayed in rendering views. The input status of the model or each type of analysis and design results can be visually verified by "walking through or flying over" the interiors of structures using the ***Walk Through Effect.***

Table Window

Table Windows display all types of data entry, analysis and design results in the Spread Sheet format. Various kinds of data modification, additional input, compilation, arrangement for different characteristics and searching capabilities are provided in Table Windows. They allow transfers with common database S/W or Excel.



Data exchange with Microsoft Excel

History Window

History Window displays the contents of data entry such that the user may verify previous activities or the status of analysis and design process.

Message Window

Message Window displays all types of information necessary for modeling, warnings and error messages.

Status Bar

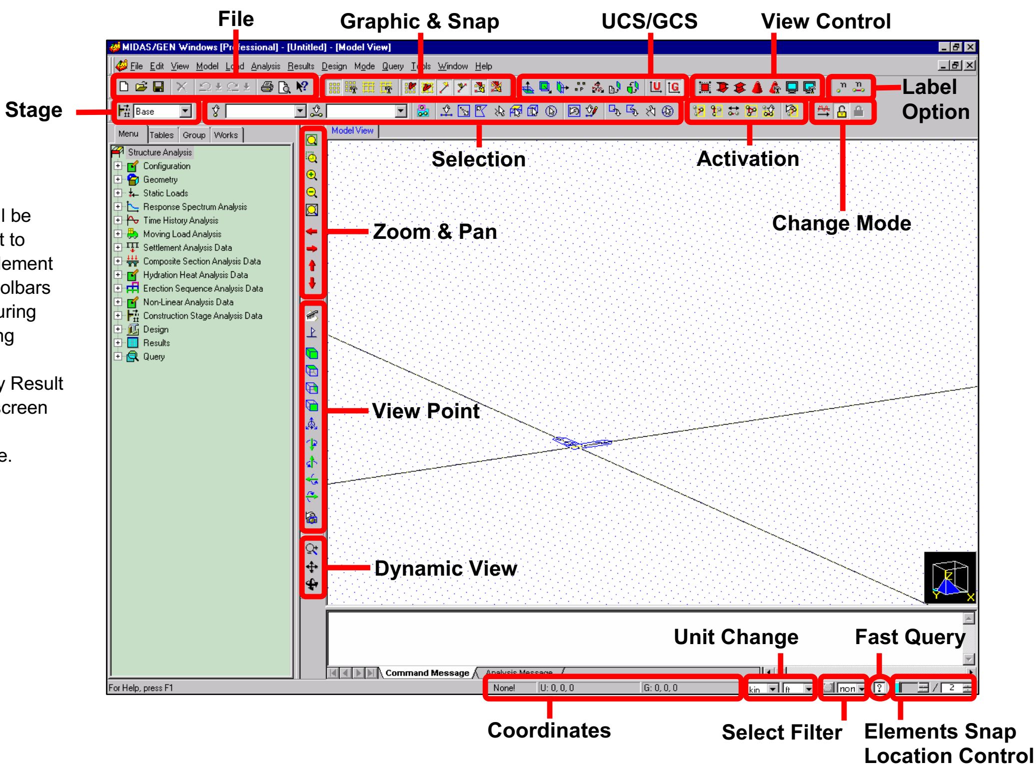
Status Bar presents matters related to all kinds of coordinate systems, unit systems conversion, select filtering, fast query, element snap control, etc., which enhance the work efficiency.

Toolbar and Icon Menu

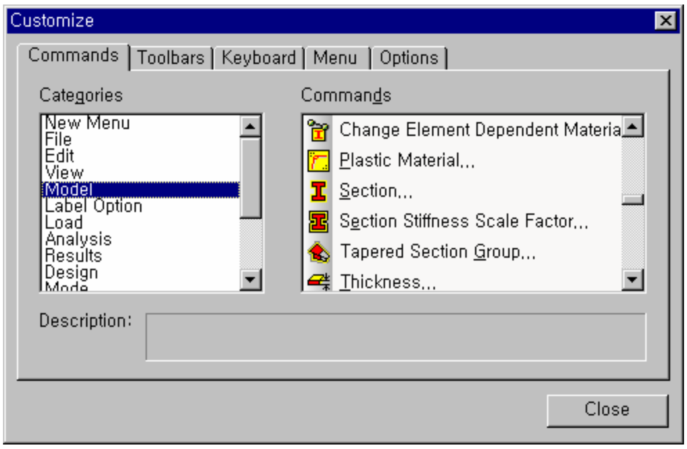
Icon Menu helps the user promptly invoke functions frequently used in **MIDAS/Gen**. Each icon is regrouped with the icons of similar purposes in various Toolbars. Each Toolbar may be easily dragged with the mouse to the desired position on the screen. They may be edited to appear selectively on the screen or modified by using ***Tools>Customize***. For more information on any icon in the Toolbar, place the mouse cursor on the icon in question and ***tool tip*** will provide a short description.

Refer to "***APPENDIX B. TOOLBAR AND ICON MENU***" for more information regarding the Toolbars and the corresponding Icons.

Applying Tools> Customize, it will be more convenient to display Node, Element and Property Toolbars on the screen during the preprocessing stage.
Similarly, display Result Toolbar on the screen during the post-processing stage.



Default positions of the Toolbars and status tabs in the window



Dialog box of Tools>Customize

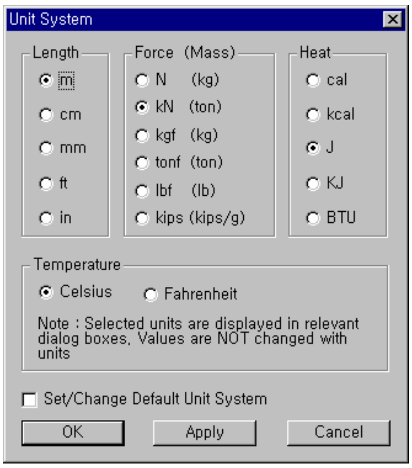
Preferences Setting

Assignment of Unit System and Conversion

In practice, there are diverse working conditions and forms of data entry. **MIDAS/Gen** is designed to operate concurrently under a specific system of units or a combination of several types of unit systems. For instance, “m” unit for the geometry data and “mm” unit for section data may be used in the same model. The “SI” unit system used in the data entry process can be converted into the “Imperial” unit system for the analysis and design results.

The thermal unit system requires a consistent unit system for the data. The units for moment, stress or modulus of elasticity which combine length units and force units are automatically adjusted by the program according to the types of length and force units selected by the user.

The user may use ***Tools>Unit System*** or the unit system conversion function of **Status Bar** located at the bottom of the screen to assign or convert the system of units.



Dialog box of Unit System Setting

Preferences Setting

Generally, each project is unique. The size and the material characteristics of a structure differ from one another, and it is convenient to define the modeling environment in advance when starting a new project.

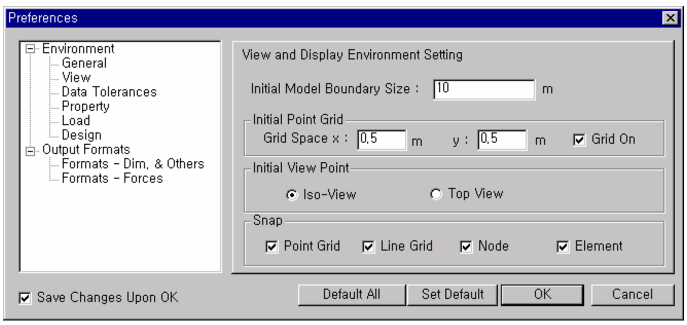
As the scale of the structure becomes apparent during the initial stage of a new project, it is possible to assign the grid spacing using ***Grid*** in advance. This will avoid additional and cumbersome adjustments of the screen dimensions.

Tools>Preferences of **MIDAS/Gen** allows the setting of the basic data required to run the program in advance.

When the ***Preferences*** function is selected, the dialog box shown below is displayed. Select the entities desired from ***Tree Menu*** on the left side and enter the required data.

Environment

General	Provide the user's name, company logo & set the automatic file saving defaults
View	Set the default window and its size
Data Tolerance	Assign the bounds of nodal combination and the upper limit of numerical values to be recognized as zero (0)
Property	Assign the basic database for materials and sections
Design	Assign applicable design standards for different material types properties
Load	Save the database for the floor loads



Dialog box for Preferences

Output Formats

Formats Assign the effective number of decimal points for the model data and analysis results

Refer to ***On-line Manual*** for detail information regarding each of the above-mentioned ***Preferences***. The ***View*** function is necessary to set the working window at the initial stage of the work as described below.

Initial Model Boundary Size

Assign the size of the working window. For example, if the length unit is set to “m” and “10” is entered, the vertical length of the new window will be set to 10m.

Initial Point Grid

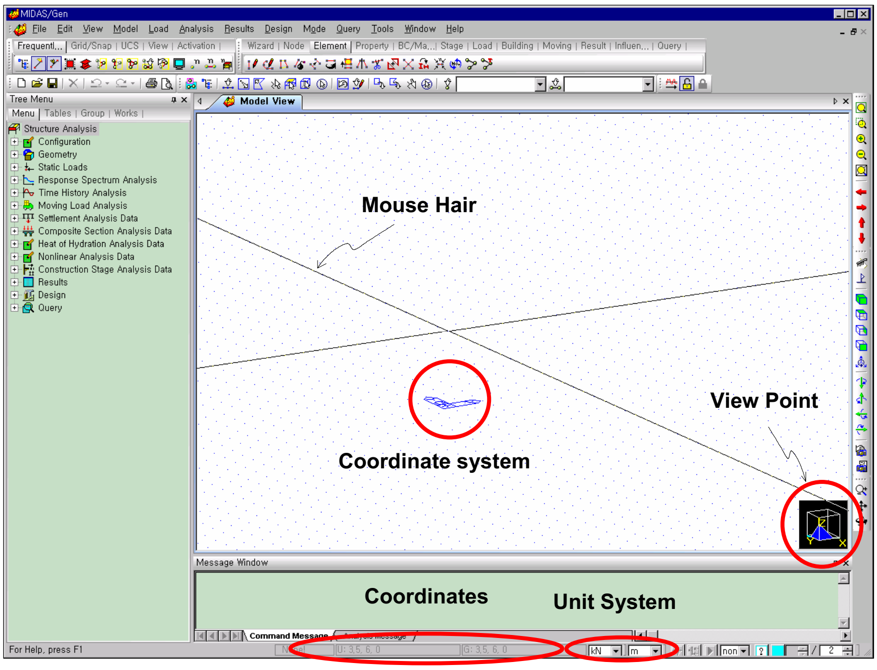
Refer to “Preferences Setting for Modeling” in Getting Started & Tutorials.

Assign the spacing of point grids to display in the window.

Grid Space x Spacing of point grids in x-direction in user coordinate system

Grid Space y Spacing of point grids in y-direction in user coordinate system

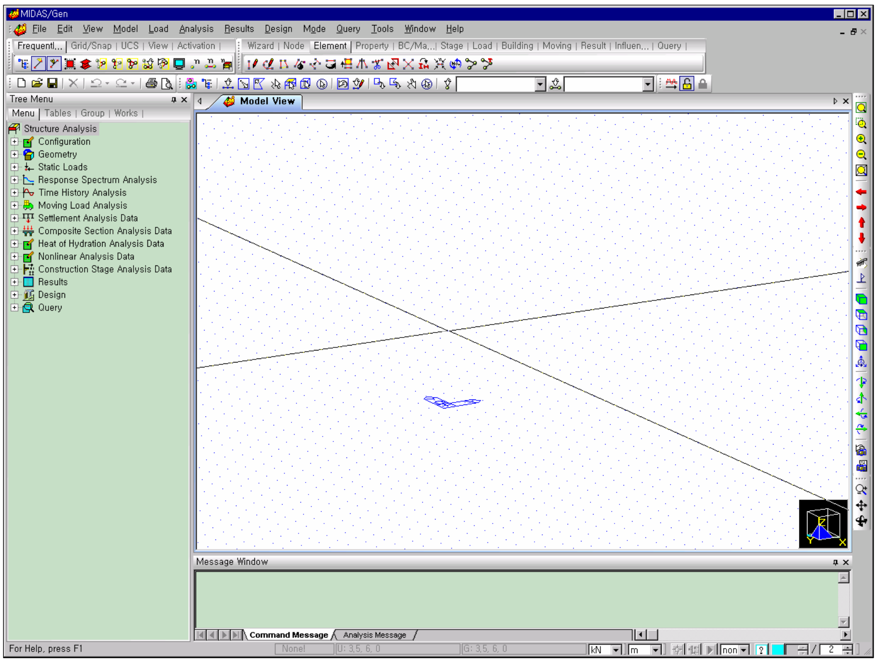
Grid On Option to display the point grids in the window



Default window of MIDAS/Gen

Initial View Point

Assign the window coordinate system to correspond to either an isometric view (Iso View) or the global X-Y plane coordinate system.



Initial window after setting the preferences

Notice that the initial window appears as shown in the figure above after specifying the following: The length unit is set to “m” in ***Tools>Unit System***. The size of the default window is 10m in ***View*** of ***Tools> Preferences***. The grid spacings in the *x* & *y* directions of the coordinate system are set to 1m and 2m respectively.

Snap

Snap is used to assign the snap state. Multiple ***Snap*** functions may be assigned at a time. When nodes or elements are being entered with the mouse, ***Snap*** automatically sets the mouse-click point to the closest grid, node or element.

Refer to “Snap” in “Nodes and Elements Generation” of the “Modeling” section.

The types of the ***Snap*** functions supported by **MIDAS/Gen** are as follows:

Point Grid Snap

Search the point grid contiguous to the mouse cursor.
Set the point grid by ***Set Point Grid***.

Line Grid Snap

Search the intersection of line grids contiguous to the mouse cursor.
Set the line grid by ***Set Line Grid***.

Node Snap

Search the node contiguous to the mouse cursor.

Element Snap

Search the mid point of the element contiguous to the mouse cursor.

In the case of a line element, the position of the snap may be adjusted by using the Snap point assignment function to the right of the status bar located at the bottom of the window. For example, the user may locate the snap at the third points of an element (1 / 3). This is an extremely convenient feature when a line element is already set up and another line element has to be connected to a particular point on that existing element.

Line Element means elements of Line Type constituted by two nodes such as truss or beam elements.

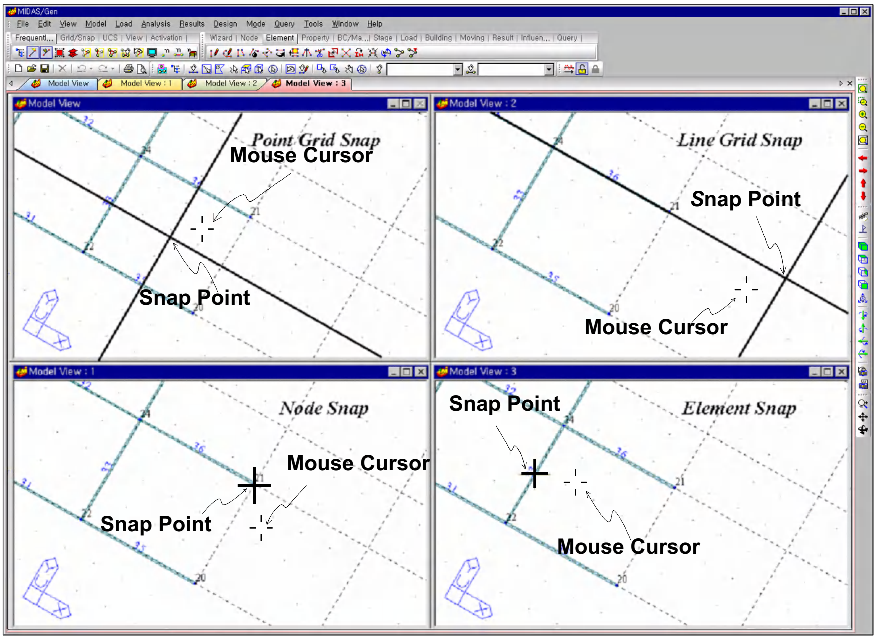
Snap All

Select all the above-mentioned snap functions.

Snap Free

Release all the snap functions.

To release Snap types separately, click the relevant Icon so that it switches to Toggle Off state.



Examples of Snap applications

Modeling Preferences Setting

Coordinate Systems

The coordinate systems used in **MIDAS/Gen** are as follows:

- Global Coordinate system ***(GCS)***
- Element Coordinate System ***(ECS)***
- Node local Coordinate System ***(NCS)***

Refer to "Structural Analysis>Numerical Analysis Model> Coordinate Systems and Nodes" of On-line Manual.

The GCS uses the X, Y and Z-axes of the ***Conventional Cartesian Coordinate System*** with the right-hand rule. The axes are denoted by the capital letters (X, Y, Z). Nodal data and the majority of data entry related to nodes, nodal displacements and nodal reactions are in GCS.

The GCS is used for the geometric data for the structure. The Reference Point is automatically set to the coordinates X=0, Y=0 and Z=0.

In **MIDAS/Gen**, because the vertical direction of the screen is set parallel to the Z-direction of the global coordinate system, it is more convenient to coincide the vertical direction of the structure (the direction opposite to the direction of gravity) with the GCS Z-direction.

Refer to "Structural Analysis>Numerical Analysis Model>Types of Elements and Important Considerations" of On-line Manual.

The ECS uses the x, y and z-axes of the Conventional Cartesian Coordinate System with the right-hand rule. The axes are denoted by the lowercase letters. (x, y, z)

Element internal forces, stresses and the majority of data entry related to elements are in ECS.

The NCS is used to assign Inclined Support Condition at a particular node. NCS uses the x, y and z-axes of the ***Conventional Cartesian Coordinate System*** with the right-hand rule. The axes are denoted by the notations x, y and z.

Once the Node Local Axes define the node coordinates, the following boundary conditions and forced displacements are entered according to the defined node coordinates:

- ***Supports***
- ***Point Spring Supports***
- ***General Spring Supports***
- ***Surface Spring Supports***
- ***Specified Displacements of Supports***

User Defined Coordinates and Grids

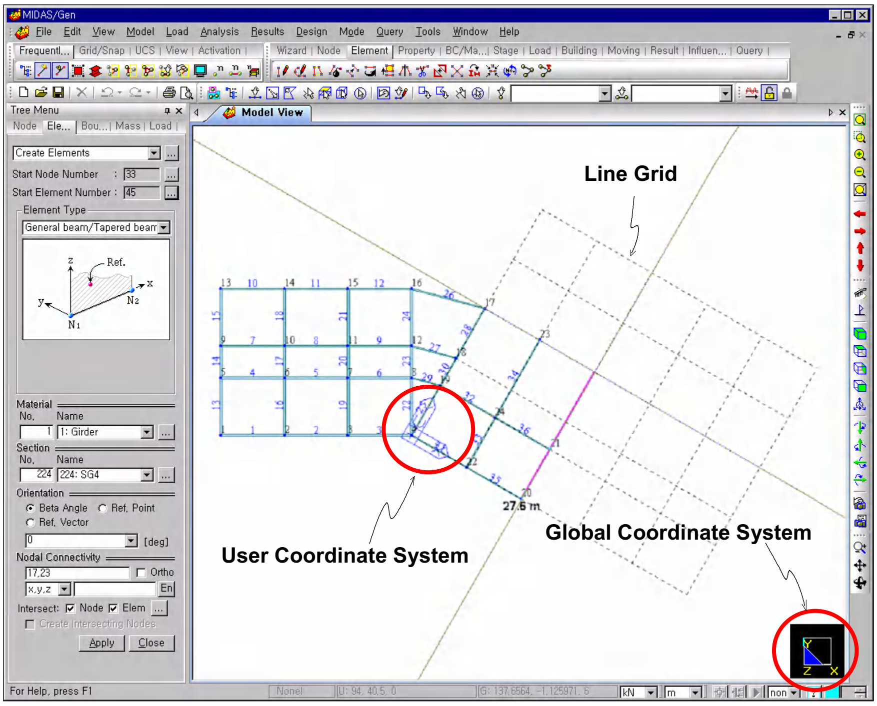
Refer to "Open File and Setting of Preferences> Working Plane and Grids" in Tutorial 1 to understand this procedure.

The ***User Coordinate System*** (***UCS***) is the coordinate system additionally defined by the user to ease the modeling task. The UCS is defined relative to the GCS and can be useful when the geometry is complex.

Generally, the majority of structures in practice are constituted in 3-D with various unit-planar structures. The structure is decomposed into a number of planes. For each plane, apart from the GCS, a coordinate system convenient for the modeling task is assigned. Once the individual segments are modeled, these planes are reassembled with respect to the GCS, and the overall 3-D shape now becomes effectively complete. The UCS is used mainly for such purpose and assigns a local coordinate system for each unit-planar structure.

User-defined Coordinate System may be saved with pre-defined titles (Named UCS), which can be recalled interchangeably with GCS.

An example of UCS and Grid Line assignment for entering beam elements located at different angles



UCS and the grid layout

When entering coordinates or elements, assign the grids to coincide with the UCS x-y plane. Such technique is extremely convenient for modeling.

MIDAS/Gen supports the following two types of grid system:

- ***Point Grid***
- ***Line Grid***

The point grid represented by a series of points on the UCS x-y plane is parallel with the x & y-axes, and each point is set equally apart. Generally, during the initial stage of modeling, set the point grid by ***Tools>Preferences***. Depending on the work conditions, use ***View>Grids>Define Point Grid*** to reassign the grid.

The line grid, as a grid represented by lines at right angles on the UCS x-y plane, is positioned parallel with both x and y directions. The spacing may be unequal.

Set the line grid by  ***Set Line Grid***.

Each grid system can be positioned at the same time, and it is convenient to use ***Snap*** to automatically locate the mouse cursor to a contiguous grid.

Entering Data

General

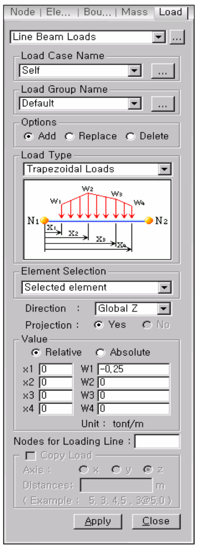
All the data are entered with the Dialog Box, Table Window, MGT Command Shell and Model Window in **MIDAS/Gen**. Using the Dialog Box, the data can be entered by both mouse and keyboard. The keyboard is mainly used for the Table Window and MGT Command Shell, and the mouse is mainly used for the Model Window.

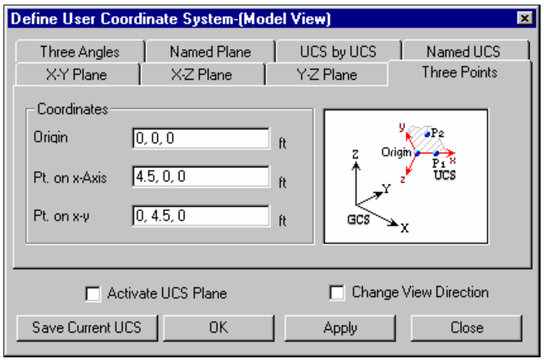
In the Dialog Box, the following buttons are used to reflect or cancel the data entry in the model.

OK	Reflect the data entry in the model and, at the same time, close the corresponding operation and the dialog box.
Apply	Reflect the current data entry in the model and continuously accept any additional data entry and modification maintaining the dialog box active.
Cancel	Cancel the current data entry and close the dialog box.
Close	Close the dialog box.

When shifting the focus from one data entry to another in a Dialog Box, use the ***Tab*** key on the keyboard to move successively from one data field to the next, or directly specify data by placing the mouse cursor over the desired data field.

If the ***Shift***+***Tab*** key is used, the input sequence will be reversed.





Dialog box

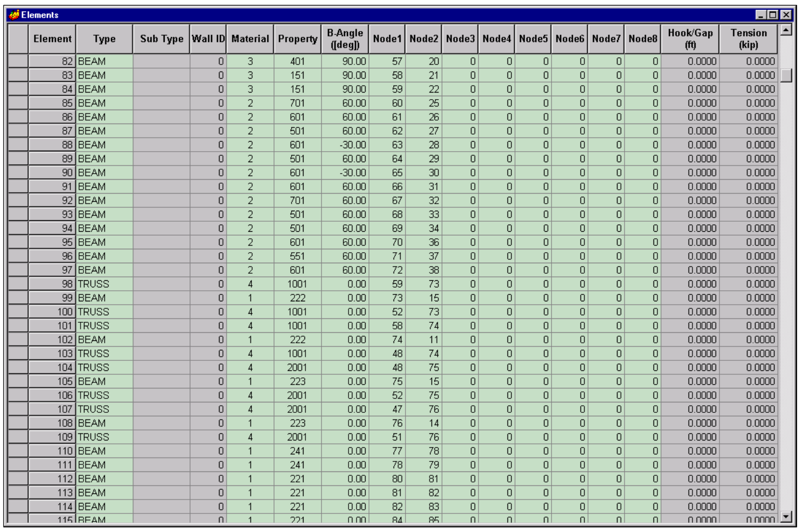
Dialog box in the form of Dialog Bar

🎧 Table Window of MIDAS/Gen offers data input/output and modification capabilities. In addition, it provides all types of selection functions, Filtering, Sorting and Graph functions, data exchange with Excel, etc.

The Table Window is a Spread Sheet type window where all the data entry and design results can be viewed at a glance. It allows the user to make any additional data entry or modification.

MGT Command Shell is a unique modeling feature, which allows the user to enter data by text type commands.

For more details concerning the applications, refer to the ***On-line Manual***.



Elements

Element	Type	Sub Type	Wall ID	Material	Property	B-Angle ([deg])	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8	Hook/Gap (ft)	Tension (kip)
82	BEAM		0	3	401	90.00	57	20	0	0	0	0	0	0	0.0000	0.0000
83	BEAM		0	3	151	90.00	58	21	0	0	0	0	0	0	0.0000	0.0000
84	BEAM		0	3	151	90.00	59	22	0	0	0	0	0	0	0.0000	0.0000
85	BEAM		0	2	701	60.00	60	25	0	0	0	0	0	0	0.0000	0.0000
86	BEAM		0	2	601	60.00	61	26	0	0	0	0	0	0	0.0000	0.0000
87	BEAM		0	2	501	60.00	62	27	0	0	0	0	0	0	0.0000	0.0000
88	BEAM		0	2	601	-30.00	63	28	0	0	0	0	0	0	0.0000	0.0000
89	BEAM		0	2	501	60.00	64	29	0	0	0	0	0	0	0.0000	0.0000
90	BEAM		0	2	601	-30.00	65	30	0	0	0	0	0	0	0.0000	0.0000
91	BEAM		0	2	601	60.00	66	31	0	0	0	0	0	0	0.0000	0.0000
92	BEAM		0	2	701	60.00	67	32	0	0	0	0	0	0	0.0000	0.0000
93	BEAM		0	2	501	60.00	68	33	0	0	0	0	0	0	0.0000	0.0000
94	BEAM		0	2	501	60.00	69	34	0	0	0	0	0	0	0.0000	0.0000
95	BEAM		0	2	601	60.00	70	36	0	0	0	0	0	0	0.0000	0.0000
96	BEAM		0	2	551	60.00	71	37	0	0	0	0	0	0	0.0000	0.0000
97	BEAM		0	2	601	60.00	72	38	0	0	0	0	0	0	0.0000	0.0000
98	TRUSS		0	4	1001	0.00	59	73	0	0	0	0	0	0	0.0000	0.0000
99	BEAM		0	1	222	0.00	73	15	0	0	0	0	0	0	0.0000	0.0000
100	TRUSS		0	4	1001	0.00	52	73	0	0	0	0	0	0	0.0000	0.0000
101	TRUSS		0	4	1001	0.00	58	74	0	0	0	0	0	0	0.0000	0.0000
102	BEAM		0	1	222	0.00	74	11	0	0	0	0	0	0	0.0000	0.0000
103	TRUSS		0	4	1001	0.00	48	74	0	0	0	0	0	0	0.0000	0.0000
104	TRUSS		0	4	2001	0.00	48	75	0	0	0	0	0	0	0.0000	0.0000
105	BEAM		0	1	223	0.00	75	15	0	0	0	0	0	0	0.0000	0.0000
106	TRUSS		0	4	2001	0.00	52	75	0	0	0	0	0	0	0.0000	0.0000
107	TRUSS		0	4	2001	0.00	47	76	0	0	0	0	0	0	0.0000	0.0000
108	BEAM		0	1	223	0.00	76	14	0	0	0	0	0	0	0.0000	0.0000
109	TRUSS		0	4	2001	0.00	51	76	0	0	0	0	0	0	0.0000	0.0000
110	BEAM		0	1	241	0.00	77	78	0	0	0	0	0	0	0.0000	0.0000
111	BEAM		0	1	241	0.00	78	79	0	0	0	0	0	0	0.0000	0.0000
112	BEAM		0	1	221	0.00	80	81	0	0	0	0	0	0	0.0000	0.0000
113	BEAM		0	1	221	0.00	81	82	0	0	0	0	0	0	0.0000	0.0000
114	BEAM		0	1	221	0.00	82	83	0	0	0	0	0	0	0.0000	0.0000
115	BEAM		0	1	221	0.00	84	85	0	0	0	0	0	0	0.0000	0.0000

Elements table window

Data Input Commands

For convenience, **MIDAS/Gen** provides the following data entry options:

- Where several numerical data are entered consecutively in a data field, these data can be distinguished by a "," (Comma) or a " " (Blank).
- <Example> '333, 102, 101' or '333 102 101'
- Position data, element sections and properties and other relevant data can be entered by simple assignments in the Model Window.
- Length or directional increments can be specified using the mouse by choosing the relevant origin and ending points in the Model Window rather than typing these data directly on the keyboard.
- Where the same length is repeated, the entry can be simplified by "*number of repetition @ length*" instead of repeating the same number.
- <Example> 20, 25, 22.3, 22.3, 22.3, 22.3, 22.3, 88 → 20, 25, 5@22.3, 88

The keyboard may be used to enter selected data directly. The related node numbering or element numbering may be an arithmetic progression in series or the progression may be incremental. Then, the data entry can be simplified by "*start number* to (t) *final number*" or "*start number* to (t) *final number* by *increment*".

< Example> 21, 22, … , 54, 55, 56 → "21 to 56", "21 t 56"

< Example> 35, 40, 45, 50, 55, 60 → "35 to 60 by 5", "35 t 60 by 5"

- Numbers and mathematical expressions can be used in combination. The majority of the operators and parentheses applied in engineering computation can be used.

<Example> $\pi \times 20^2$ → PHI * 20^2

<Example> $35 + 3 \times \left(\sin 30^\circ + 2\sqrt{\cos^2 30^\circ + \sin^2 30^\circ} \right)$

→"35 + 3 * (sin(30) + 2 * SQRT(cos(30)^2+sin(30)^2))"

Notation	Content	Remarks
(	Open parenthesis	–
)	Close parenthesis	–
^	Power of n (^2→square, ^3→cube)	Ex.: 2^3 = 2 ^ 3
+	Addition	–
–	Subtraction	–
*	Multiplication	–
/	Division	–
PI	π	3.141592653589793
SQRT	$\sqrt{\ }$	Ex.: $\sqrt{2}$ = SQRT(2)
SIN	Sine	Unit: Degree
COS	Cosine	Unit: Degree
TAN	Tangent	Unit: Degree
ASIN	Arc Sine	Ex.: $\sin^{-1}$(0.3)=ASIN(0.3)
ACOS	Arc Cosine	Ex.: $\cos^{-1}$ (0.3)=ACOS(0.3)
ATAN	Arc Tangent	Ex.: $\tan^{-1}$ (0.3)=ATAN(0.3)
EXP	Exponential function	Ex.: $e^{0.3}$ =EXP(0.3)
SINH	Hyperbolic Sine	Ex.: sinh(1)=SINH(1)
COSH	Hyperbolic Cosine	Ex.: cosh(1)=COSH(1)
COTAN	Cosine/Sine	Ex.: cotan(1)=COTAN(1)
LN	Natural Logarithm	–
LOG	Common Logarithm	–

Built-in operators in MIDAS/Gen

※ Highlights of usage

1. Operators accept the mixed use of capital and lowercase letters.
2. As the operators are similar to that of an engineering calculator, the hierarchy of operations follows the rules of common mathematical operations.

Manipulation of Model Window

MIDAS/Gen offers various Model Window Handling capabilities for sophisticated and realistic visual representation of the model generation, analysis and design results.

Model Window Handling functions can be invoked from the ***View*** menu or by simply clicking the icons in Toolbar.

Model Shape Representation

The Model Shape Representation functions of **MIDAS/Gen** such as ***Wire Frame, Hidden, Shrink, Perspective and Render View*** present the model in diverse shapes and views. These functions help the user grasp the input state of the model and manipulate the model as much as desired.

The Model Shape Representation functions of **MIDAS/Gen** are as follows:

Shrink is typically used to check the connectivity of nodes and elements

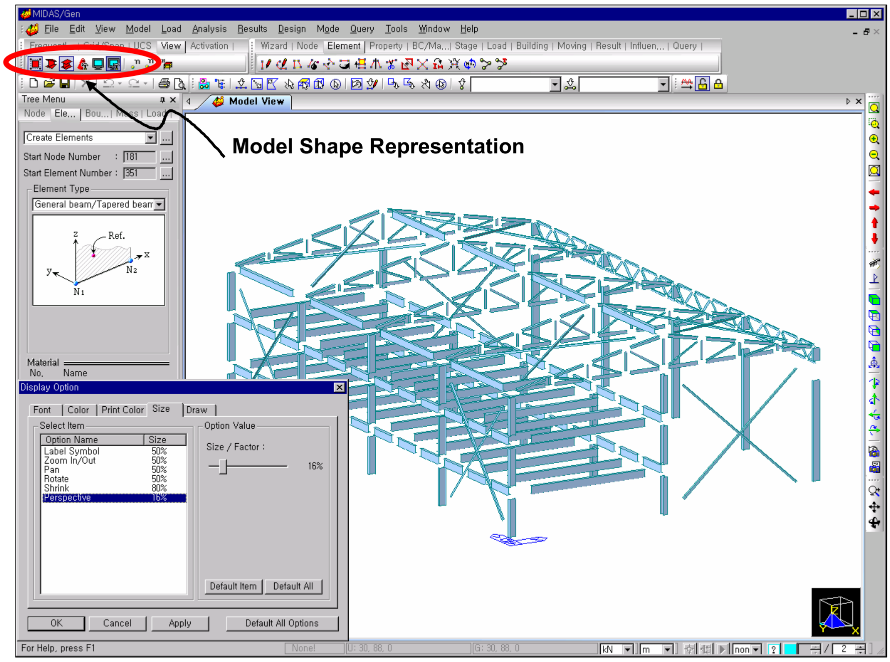
Shrink
Display the modeled elements in proportionally reduced sizes.

Perspective
Display a perspective 3-dimensional view of the model.

Hidden
Display the model shape reflecting the sectional shapes of elements and their thicknesses as it would truly appear.

This model is viewed with Shrink, Perspective and Hidden using the Model Shape Represen--tation Toolbar.

The Size and Draw tabs in Display Option controls the Factor and Scale adjustment, and the reflection of the thickness related to Model Shape Representation.



3-D Plant Structure: Shrink, Perspective and Hidden Views

The Rendering function is provided in the window, and the Render View is used to apply the functions such as Blending.

Render View

Display the model shape reflecting the sectional shapes of elements and their thicknesses with a shadowing effect as it would truly appear.

Rendering Option

Modulate the effects of lighting and shadowing of Render View.

Display

Display in the working window the nodal and element numbering, material and sectional designation, the loading input state, etc.

Refer to Model>Verify input results> Display Option section.

Display Option

Control all the graphics displayed in the working window including all types of display modes such as the color palette of characters, the displayed size, etc.

Zoom in/out and Motion Control (View Manipulation Functions)

All the ***View Manipulation*** functions of **MIDAS/Gen** with the ***Render View*** function assist the user to accurately grasp the three-dimensional views of the model input state and the analysis and design results through diverse view angles and points.

View Point

The ***View Manipulation*** functions of **MIDAS/Gen** are as follows:

Iso View
Represent the model in a three-dimensional space.

Top View
Represent the model as viewed from the +Z direction.

Left View
Represent the model as viewed from the –X direction.

Right View
Represent the model as viewed from the +X direction.

Front View
Represent the model as viewed from the –Y direction.

Angle View
Represent the model as viewed from a specified viewpoint.

Rotate

Rotate Left
Rotate the model to the left (clockwise about Z-axis).

Rotate Right
Rotate the model to the right (counterclockwise about Z-axis).

Rotate Up
Rotate the model upward from the horizontal plane.

Rotate Down
Rotate the model downward from the horizontal plane.

Zoom

Zoom Fit
Fit the model to the screen size by scale up/down.

Zoom Window
Assign the desired size of the window by dragging a corner of the window with the mouse.

Zoom In
Magnify the current window gradually.

Zoom Out
Reduce the current window gradually.

The proportioning of screen manipulation for Zoom, Pan and Rotate is controlled in the Size tab in Display Option.

Pan

Pan Left
Move the model window to the left.

Pan Right
Move the model window to the right.

Pan Up
Move the model window upward.

Pan Down
Move the model window downward.

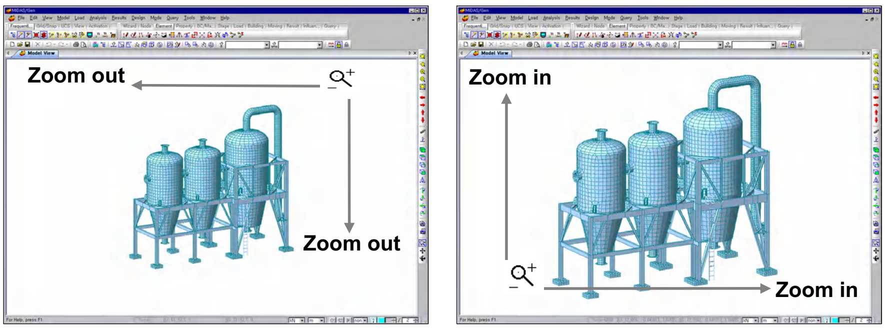
Dynamic View Manipulation

The ***Dynamic View*** of **MIDAS/Gen** provides ***Zoom, Pan*** and ***Rotate*** functions. It displays realistic views of the structure in real time from the desired viewpoint by keeping the mouse left-shifted and dragging the mouse.

By linking ***Dynamic Zoom/Rotate*** and ***Render View***, we can look inside and walk through the structure (***Walk Through Effect***) or fly over the structure.

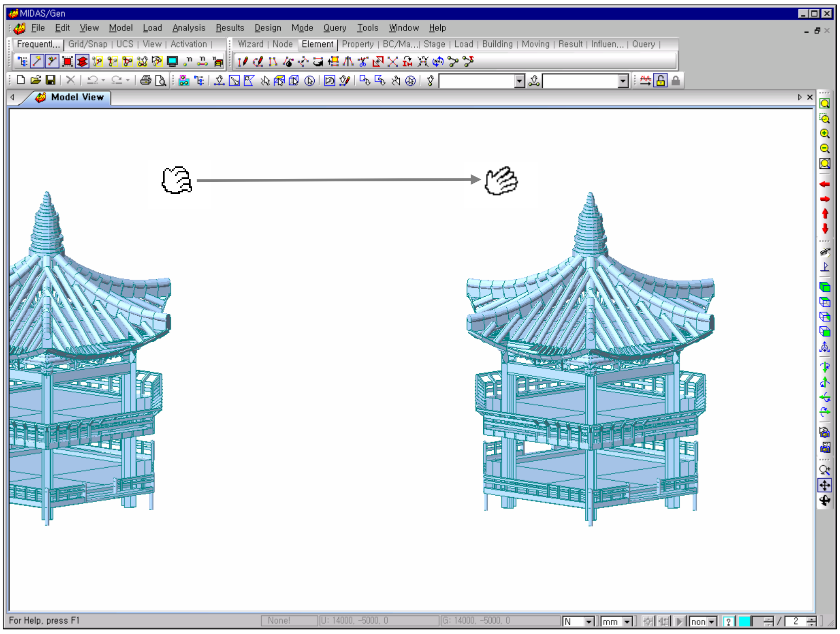
Keeping the mouse left-shifted and dragging the mouse downward or to the left reduces the window.

Keeping the mouse left-shifted and dragging the mouse upward or to the right magnifies the window.



Zoom Dynamic Illustration

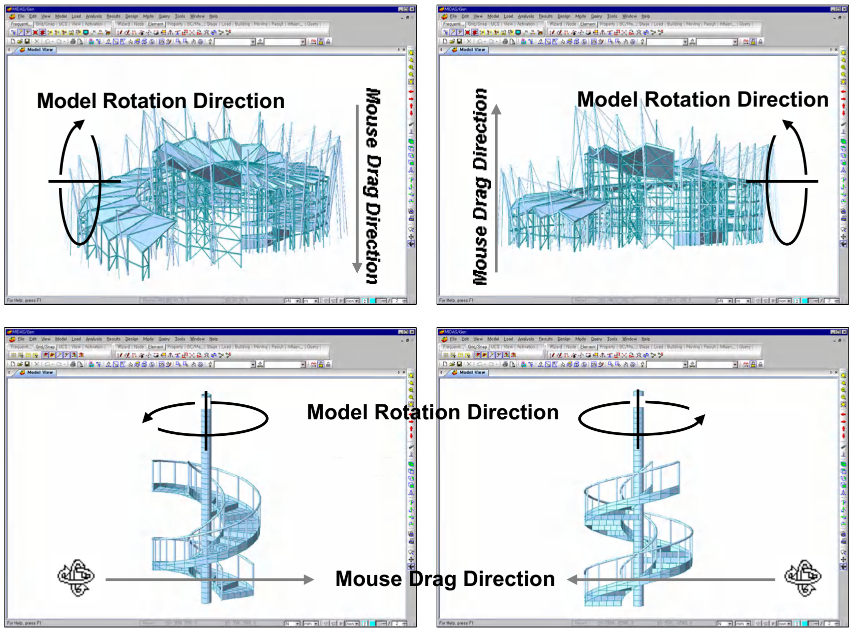
🎧 By keeping the mouse left-shifted and moving the mouse cursor, the model window will follow the course of the mouse.



Pan Dynamic View

🎧 Using Rotate Dynamic, drag the mouse cursor downward or upward. The View Point will move downward or upward following the drag direction.

🎧 Using Rotate Dynamic, drag the mouse cursor to the left or right. The View Point will move to the left or right following the drag direction.



Example of Rotate Dynamic Application

Selection and Activation / Deactivation

Selection

The ***Selection*** functions are extremely important and indispensable for the overall task of generating a model. It allows duplication of nodes and/or elements, with or without the same attributes such as loading or boundary conditions, activation of special parts, verification of input and output data, etc.

The ***Selection*** functions supported by **MIDAS/Gen** are as follows:

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

Graphical Selection

Select Single

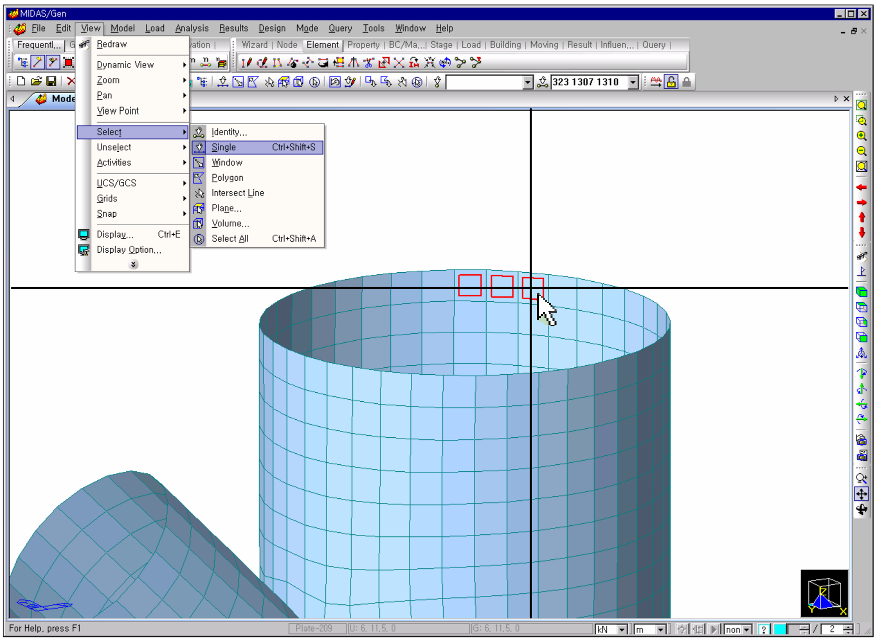
Select the desired entities by clicking the mouse once each time. To unselect the selected entities click them once again. The Select Window feature can be effected by dragging the mouse left-shifted from a fixed point.

Select Window   *Unselect Window*

Click the diagonal corners of a window containing the entities with the mouse cursor and select or unselect the desired nodes or elements.

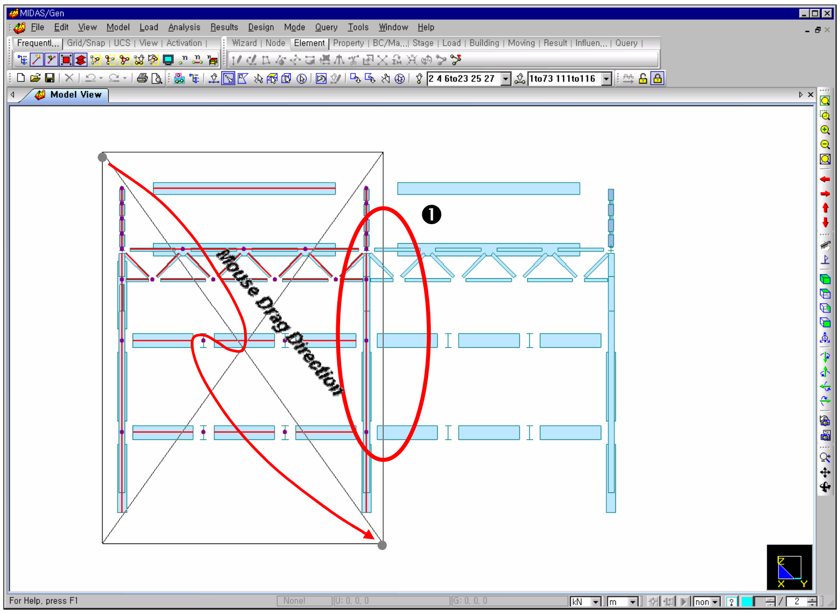
When assigning the window, select only the nodes and elements completely contained within the window by dragging the mouse cursor from left to right.

When assigning the window, select all the elements that are contained inside the window as well as the elements intersecting the boundaries of the window by dragging the mouse cursor from right to left.

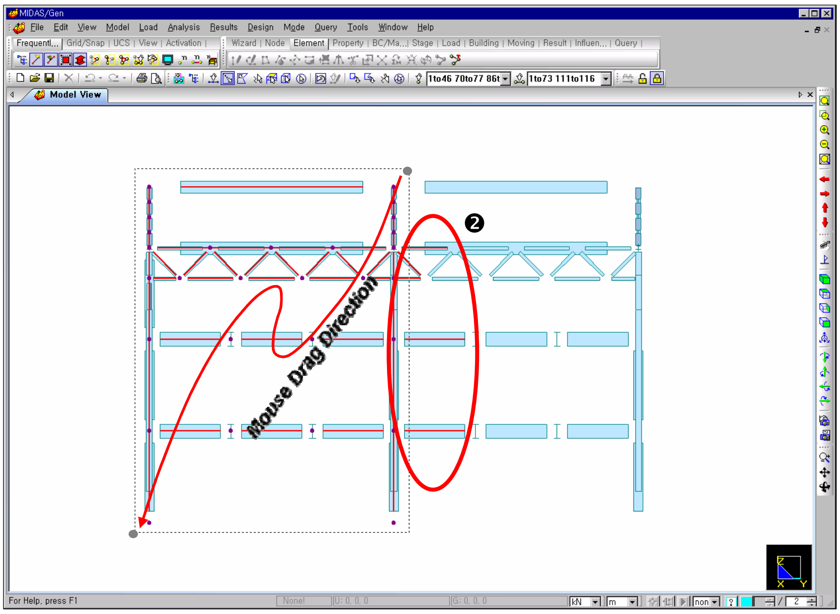


Select plate elements successively one by one with Select Single

🎧 Drag the mouse cursor from left to right. The elements that are not completely contained in the window boundaries will not be selected. (❶)



🎧 Drag the mouse cursor from right to left. Even those elements crossing the window boundaries will be selected. (❷)



Select Window

Select Polygon *Unselect Polygon*

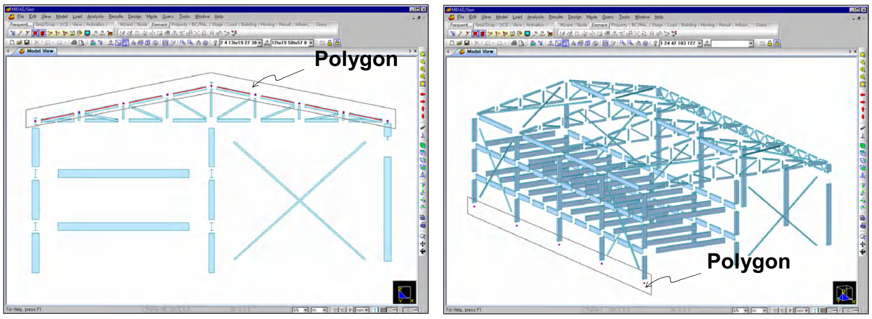
Select or unselect the desired nodes and/or elements by successively clicking the corners of the polygon containing the relevant entities with the mouse cursor.

Select the final corner and left-click the mouse twice with the [Ctrl] key pressed; even those elements crossing the polygon line will be selected.

When clicking the final corner, left-click the mouse twice. The polygon linking the final corner and the starting point is created, and all the nodes and elements contained inside the polygon are selected.

To enter a loading acting on an inclined roof, select only the beam elements on the slope.

To modify the boundary conditions at the supports, select only the supports by forming a polygon.



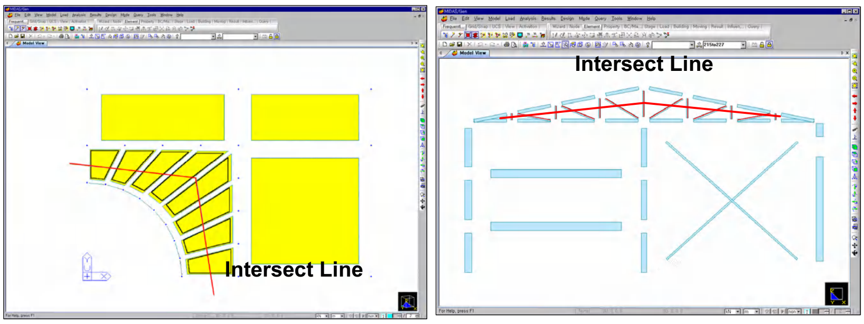
Select Polygon

Select Intersect *Unselect Intersect*

Select or unselect elements by crossing a series of lines that intersect the desired elements with the mouse cursor in the Model Window. When clicking the final point of the last line, left-click the mouse twice. This terminates the selection process.

In the process of element meshing, plate elements can be readily selected by Intersect.

To modify the element types, select the vertical and diagonal members of the truss roof.



Select Intersect

Select Plane *Unselect Plane*

By assigning a particular plane, select or unselect all the nodes and/or elements contained in the plane.

Observe the following methods to select a plane:

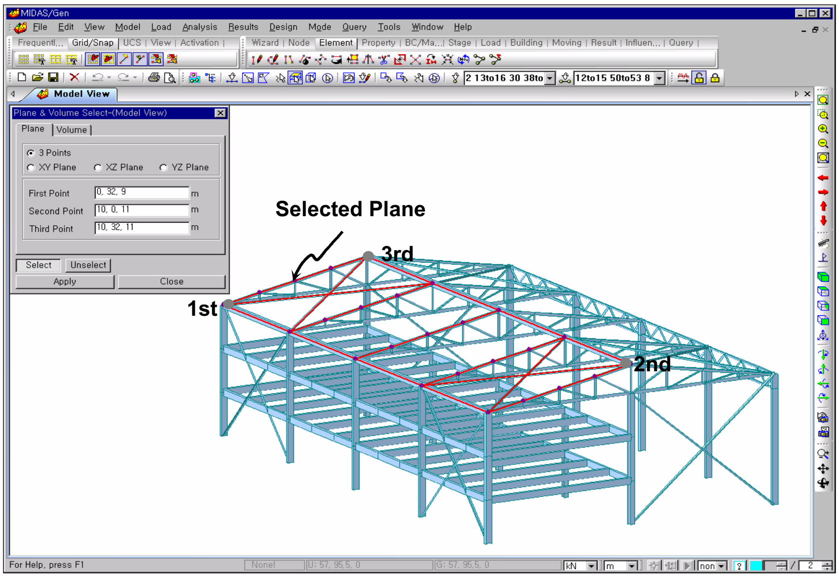
3 Points
Specify 3 points located in the desired plane.

XY Plane
For a plane parallel to the X-Y plane, specify a Z coordinate of the desired plane.

XZ Plane
For a plane parallel to the X-Z plane, specify a Y coordinate of the desired plane.

YZ Plane
For a plane parallel to the Y-Z plane, specify an X coordinate of the desired plane.

Planes non-parallel to GCS or UCS can be easily assigned by means of 3 Points. The figure shows an inclined roof lying in a plane assigned by 3 Points placed on the grids.

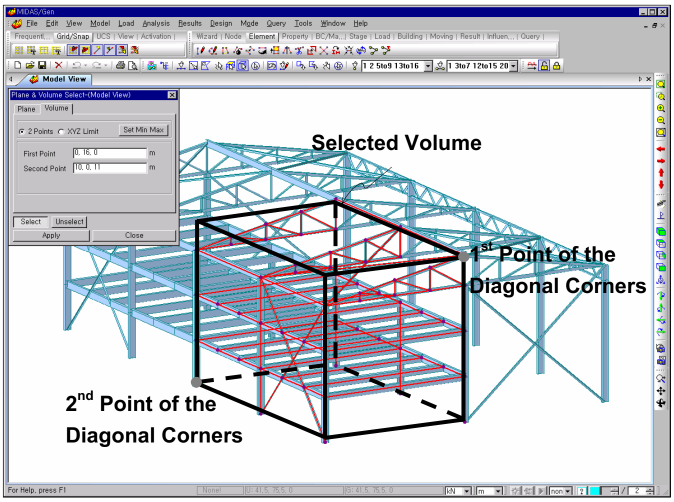


Select Plane by 3 Points

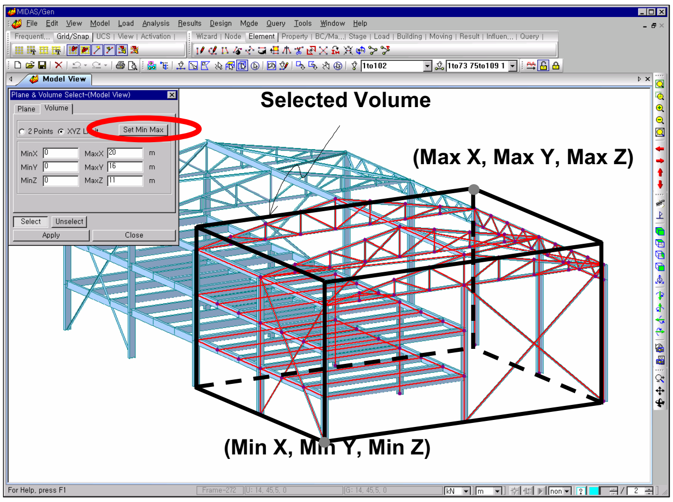
Select Volume   Unselect Volume

To assign a particular hexagonal volume, select and/or unselect all the nodes and elements contained in the volume.

Observe the following methods to select a hexagonal volume:



Click Set Min Max and select the volume by modifying only the necessary coordinates. The part of the structure contained within the minimum and maximum coordinates will appear.



Select Volume

2 ***Points***

Select two points of the diagonal corners of the desired hexagonal volume.

XYZ Limit

Enter the coordinates of the range of the desired hexagonal volume for each axis.

Select All ***Unselect All***

Select or unselect all the nodes and/or elements.

Specified Selection

Select Identity

Specified Selection

- Select Identity-Nodes
- Select Identity-Elements
- Group Selection
- Select Previous
- Select Recent Entities

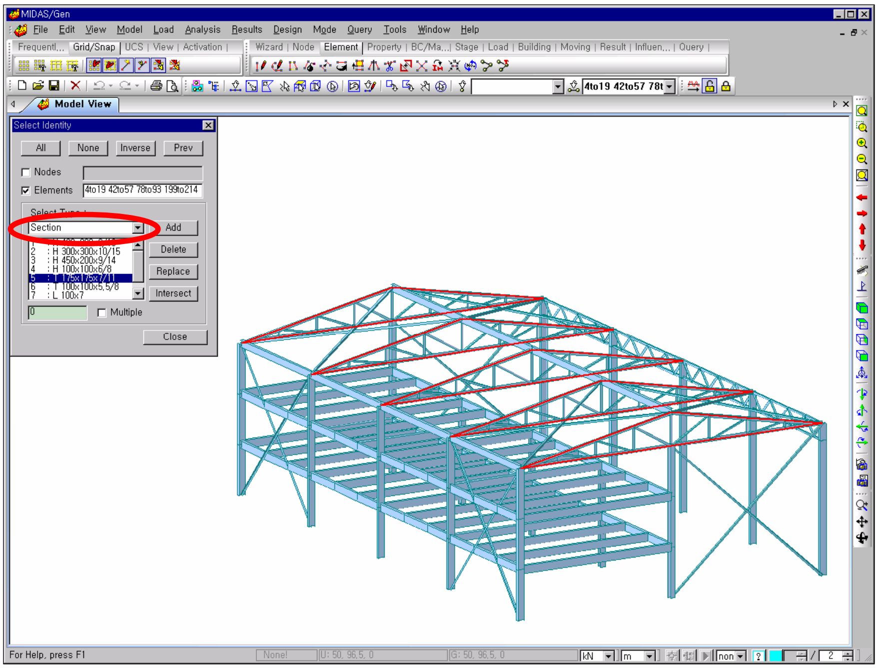
Select the desired entities by physical or geometrical identities, i.e., select nodes or elements with identical attributes, types or groups.

Entities can be selected by each identity separately or multi-identities simultaneously.

The types of identities that can be selected are as follows:

Element Type	Selection by type of element
Material	Selection by type of material attribute
Section	Selection by type of section
Thickness	Selection by type of thickness
Named Plane	Selection by name of plane
Story	Selection by ID of story
Supports	Selection of nodes by support condition
Beam End Release	Selection of beams by beam end release condition
Wall ID	Selection by wall combination numbering
Structure Group	Selection by element group
Boundary Group	Selection by boundary group
Load Group	Selection by load group

A section type (the top and bottom chords of the roof trusses) is selected with Select Identity-Elements to modify the Element Type.



Select Identity - Section

Select the desired types in the Identity list shown in the figure above. Select or modify the selected entities subsequently and selectively as required. Alternatively, one of the elements having the identity in the Model Window can be selected with the mouse cursor.

Select Previous

Reselect the entities selected in the previous step.

Select Recent Entities

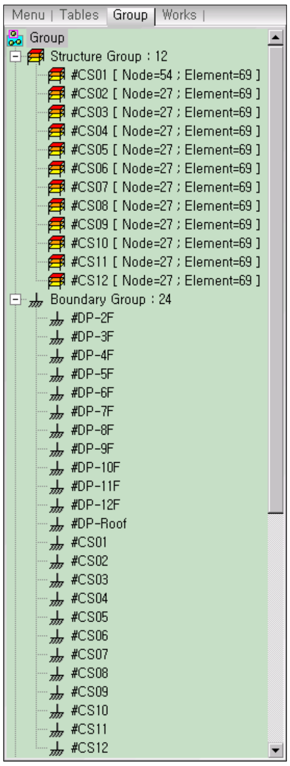
Select the nodes or elements most recently generated during the modeling exercise.

Group

Group

MIDAS/Gen allows us to define ***Structure group*** by grouping nodes and elements and ***Boundary Group*** and ***Load Group*** for boundary conditions and loadings attributed to the nodes and elements. The three groups are subsequently used in combination for defining construction stages.

First, assign a structure group name and designate relevant nodes and elements by various Select functions. Using ***Drag & Drop*** under the ***Group*** tab of **Tree Menu,** we can assign the relevant nodes and elements appropriate group names. In particular, it is extremely useful for modeling complex structures by selecting and activating certain groups without a repetitive process of selection.

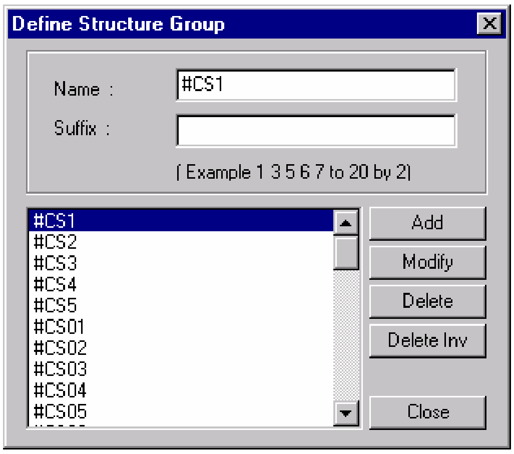


Group Dialog Bar

The common procedure for applying ***Structure Group*** is as follows:

Register the desired nodes and elements as a Group

1. Select ***Model>Group>Define Structure Group*** (or click ***Group***, select ***Structure Group*** from the ***Group*** tab of **Tree Menu** and select New... after right-clicking the mouse.



Define Structure Group

2. Enter a group name in the Name field with ***Suffix*** numbers and click the Add button to create a number of structure groups simultaneously.

3. Using the selection functions, select the relevant nodes and elements to be assigned to the structure groups.

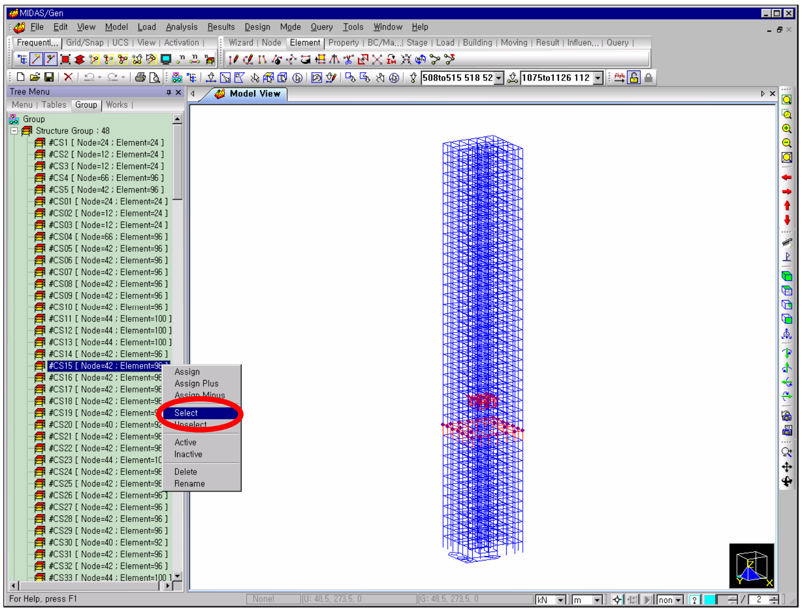
4. Define structure groups using ***Drag & Drop*** of **Tree Menu**.

Using Model>Group>Change Boundary Group/Change Load Group the existing boundary conditions and loading groups can be copied, moved and deleted to create other groups.

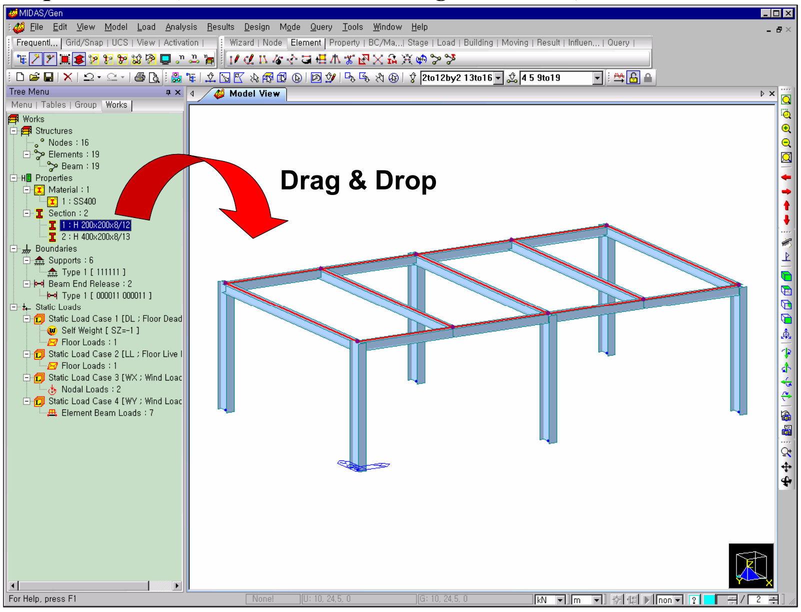
Double-click the selected group in Tree Menu to select the corresponding nodes and elements.

Right-clicking in the selected group of Tree Menu permits us to carry on many different tasks.

Define **Boundary Groups** and **Load Groups** similarly.



Group Selection (Element Selection using Tree Menu)



Change of section properties by a simple operation of Drag & Drop of Works Tree tab

Specified Selection

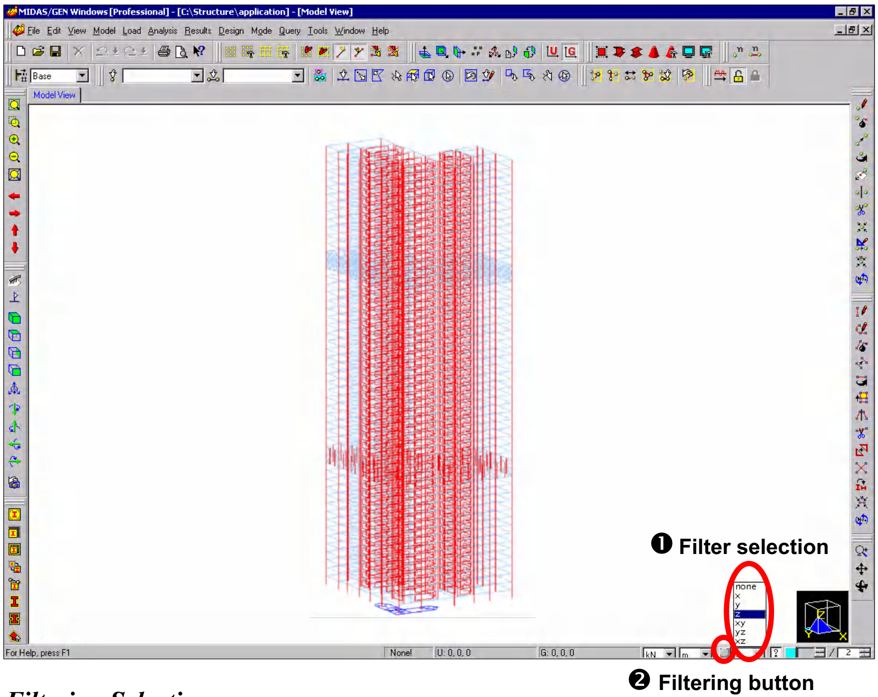
- Select Identity-Nodes
- Select Identity-Elements
- Select Previous
- Select Recent Entities
- Group Selection

Filtering Selection

Filtering Selection chooses line elements selectively based on the elements' directional orientation while applying the Graphical Selection or Specified Selection features. When the desired entities are selected by Graphical Selection, only the line elements satisfying the Filtering Condition are selected upon defining the direction of axis or plane from the filter selection field illustrated in ❶ below.

To apply Specified Selection, define the desired entities and click the Filtering button ( ) illustrated in ❷ below to select only the elements satisfying the Filtering condition among the selected line elements.

Define x-axis in the Filter selection field and define the pertinent limits by Select Window. Only the elements parallel to the x-axis will be selected within the window.



Filtering Selection

Model Activation/Deactivation

Active / ***Inactive*** is used to partially activate or inactivate specific parts of a structure.

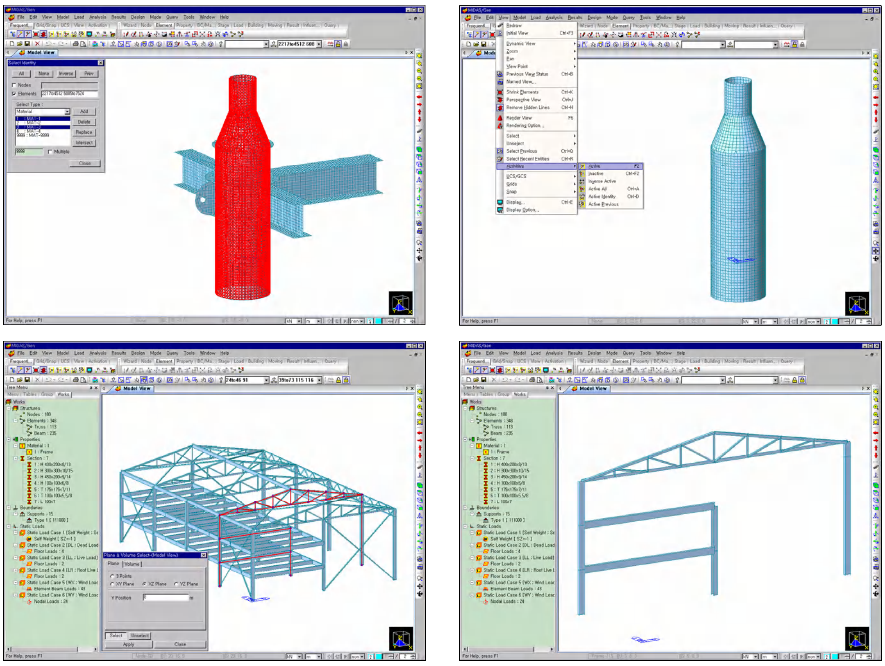
Active represents a state in which the modeling tasks are permitted. Modeling tasks such as selection, addition and modification are not allowed for the inactivated parts. Unless this function is deliberately invoked the total model is always in an activated state.

Inactivated Object under the ***Draw*** tab in ***View>Display Option*** allows the inactivated parts to either appear or disappear on the screen.

This function can be effectively used for modeling complex, large-scale structures or post-processing tasks.

To create efficiently a frame that supports a bin, simplify the Model Window by inactivating the complex bin structure.

To examine the bending moments of a frame located in the middle of a plant structure, activate only the relevant frame.



Active/Inactive

For instance, by only activating the desired story of a building or a part of a bridge on the screen, the modeling task becomes much more manageable. This function remarkably simplifies tasks such as adding or modifying nodes or elements, interpreting analysis results by selective activation of specific element types, section or attribute types, etc. Analyzing the maximum or minimum member forces will require much less effort.

The ***Active/Inactive*** function is used in connection with ***Selection***. After selecting the desired parts, activate or inactivate the relevant selections by using the functions outlined below.

Active
Activate only the selected part while the remaining parts are inactivated.

Inactive
Inactivate only the selected part while the remaining parts are activated.

Inverse Active
Reverse the current active and inactive parts to inactive and active parts respectively.

Active All
Transform all the nodes and elements in an inactivated state into an activated state.

Active Identity
Activate the assigned entities on the current UCS x-y plane that contains the origin, a particular story, the named plane or the Group, etc.

Active Previous
Return to the previous active or inactive state.

Modeling

Nodes and Elements Generation

MIDAS/Gen enables us to readily create nodes and elements as if we were drawing drawings using the majority of functions used in CAD programs.

The following two methods are mainly used for generating elements in **MIDAS/Gen**:

Refer to the "Structure Modeling Using Nodes and Elements" part in the Tutorial 1 of the present manual for better understanding of this procedure.

- Enter the nodes first and then enter the elements using these nodes.
- Enter the nodes and elements simultaneously using the predetermined grids.

The second method is generally recommended for expediency. Grids are generated first. The presence of the grids significantly reduces the risk of making mistakes during the modeling. This is highly efficient as nodes and elements are created at the same time.

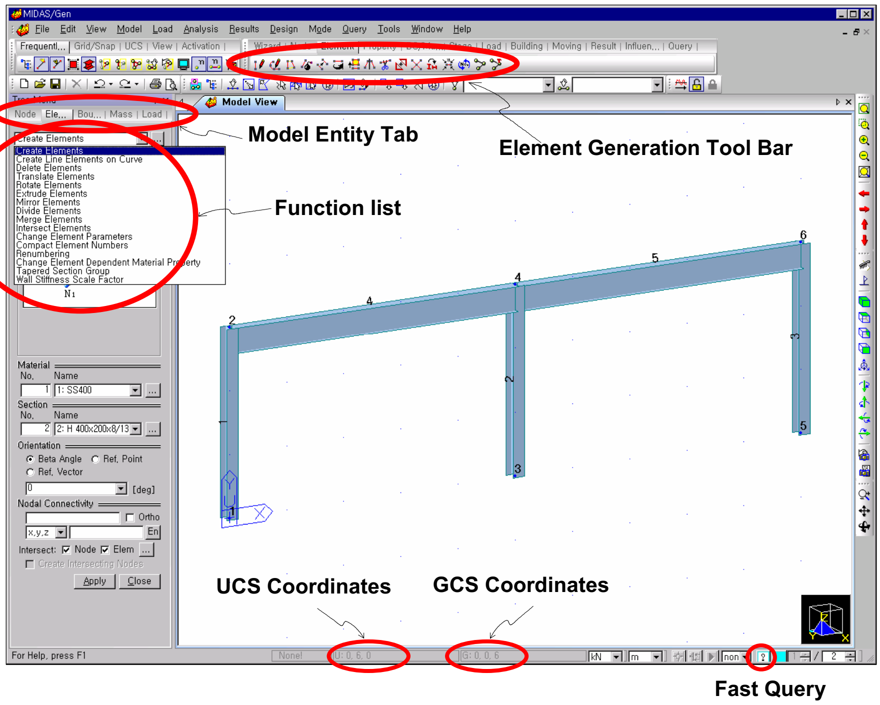
The first method is used when the geometric arrangement of elements is so irregular that the application of grids is not expected to offer any advantage. This method is used to perform a partial, detail analysis of planar elements.

The grids are laid out in the x-y plane of the UCS. The procedure to layout the Point Grids is simple enough since the grid spacing is regular in each direction of the axes, but unsuitable for modeling an irregularly spaced structure. In such a case, the use of Line Grids is more effective.

During the modeling task, because various functions are alternately used to create nodes and elements, it is convenient to use ***Model Entity Tab*** at the top of the Dialog Bar located on the left of the screen. The desired function in the function list can be selected or the Toolbars on the right of the working window can be used rather than using the Main Menu.

You may move the toolbars to any position by dragging the mouse.

🎧 Refer to the coordinates appearing in the Status Bar at the bottom of the screen while undertaking nodes/elements generation with the mouse.

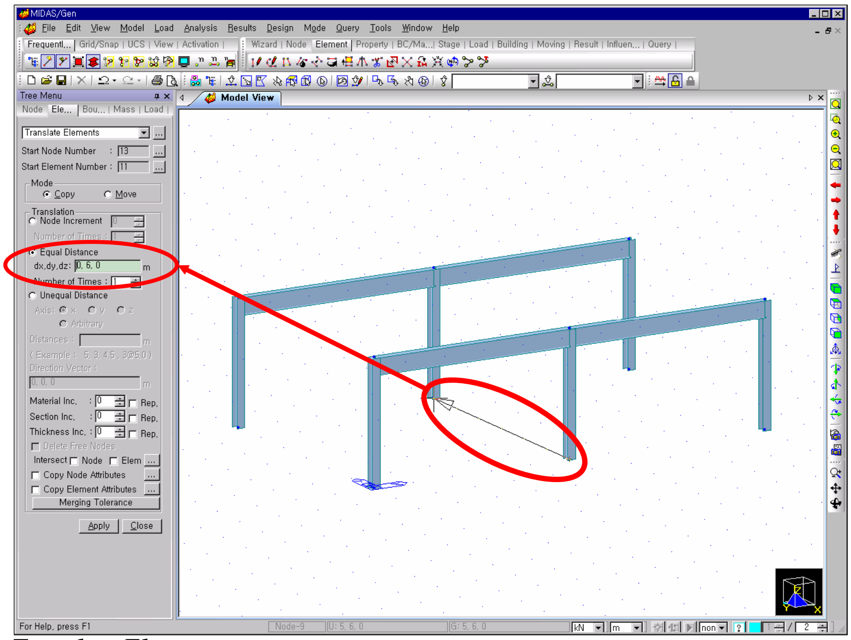


The distance, coordinate, directional vector or node number can be directly typed on the keyboard in the Dialog Bar. Alternatively, the relevant distance or position can be conveniently assigned in the Model Window with the mouse cursor. When the mouse cursor is used to enter the above entities, click the relevant data field once and the background color of the data field will change to pale green. Then, enter the relevant data in the Model Window (**Mouse Editor** function).

When duplicating or moving nodes and elements the relevant attributes may be selectively included. The relevant attributes for nodes are nodal loading, support conditions, etc. The relevant attributes for elements are element loading, element boundary conditions, etc. (***Copy Node Attributes, Copy Element Attributes***).

When duplication is required with modification of material or section properties, the modification can be accomplished by specifying increments from the number being copied.

🎧 When duplicating distance, use the mouse cursor instead of typing a numerical value in the data field.

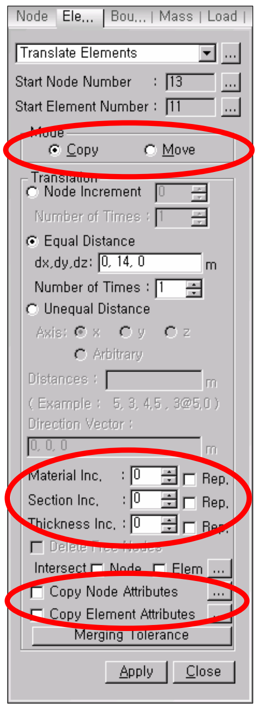


Translate Elements

🎧 Copy: Assign to copy
Move: Assign to move

🎧 Use when duplicating or moving elements while the material and section properties are altered. (Applicable also where column sections change while a building is modeled by copying each story.)

🎧 Specify whether or not to include the relevant attributes when duplicating nodes or elements.



Translate Elements Dialog Bar

Nodes Generation

Undo cancels an unlimited number of previous tasks.
Redo recovers the previous tasks canceled by Undo.

Use ***Model>Nodes*** or ***Node Toolbar*** to generate nodes. For detailed information concerning the directions, refer to ***On-line Manual***.

Create Nodes
Create new nodes and additional nodes by duplicating the new nodes at given spacings simultaneously.

Delete Nodes
Remove nodes.

Translate Nodes
Duplicate or move the existing nodes with equal or unequal spacings.

Rotate Nodes
Duplicate or move the existing nodes by rotating about a given axis.

Project Nodes
Duplicate or move the existing nodes by projecting on a particular line or surface (plane, conic surface, spherical surface, elliptic surface, etc.).

Project Nodes projects specific nodes onto a selected line or plane to copy or move the nodes. This becomes useful when modeling complicated parts of a structure.

Mirror Nodes
Duplicate or move the existing nodes symmetrically with respect to a particular plane.

Divide Nodes
Create additional nodes by dividing a straight line between two nodes into equal or unequal spacings.

Merge Nodes
Merge contiguous nodes into one node.

Scale Nodes
Reduce or magnify the spacings between two existing nodes by a specified ratio.

Compact Node Numbers
Adjust the missing node numbers that have been removed, and arrange the node numbers in a consecutive order.

Renumber Node ID
Renumber the existing node numbers either partially or in its entirety.

Start Number
Assign the start number for new nodes to be created.

Elements Generation

Undo cancels an unlimited number of previous tasks.
Redo recovers the previous tasks canceled by Undo.

Use ***Model>Elements*** or ***Element Toolbar*** to generate elements. The menu for material and section properties need not be accessed separately. By clicking the button to the right of the material and section properties list in the dialog bar for the elements, the related attributes can be added or modified. If necessary, new material and section numbers can be assigned to the elements while being duplicated.

Create Elements
Create new elements.

Create Line Elements on Curve
Create line elements along the traces of a circle, a circular arc, an elliptical circle, a parabola, etc.

Delete Elements
Remove elements.

Translate Elements
Duplicate or move existing elements with equal or unequal spacings.

Rotate Elements
Duplicate or move existing elements by rotating about a given axis.

Extrude Elements
Create one-dimension higher geometric elements (line elements, plate elements and solid elements) by expanding existing nodes, line elements and plate elements as follows:

- Create a line element along the path created by the motion of a node.
- Create a plate element along the path created by the motion of a line element.
- Create a solid element along the path created by the motion of a plate element.

Mirror Elements
Duplicate or move existing elements symmetrically with respect to a particular plane.

Divide Elements
Divide existing elements into equal or unequal sub-elements.

Merge Elements
Merge elements of identical attributes (materials, section properties, element types, etc.) into one element.

Intersect Elements
Divide automatically existing line elements intersecting one another relative to the intersection points.

Change Element Parameters
Change the attributes of elements.

Compact Element Numbers
Adjust the missing element numbers that have been removed, and arrange the element numbers in a consecutive order.

Renumber Element ID
Renumber existing elements either partially or entirely.

Start Number
Assign the start number for new elements to be created.

Modeling Automation

Depending on the characteristics of a structure in question, the following automated generation features may simplify the data entry, thereby increasing productivity:

Refer to "Modeling> Model>Structure wizard" of On-line Manual.

Structure Wizard

Using this feature, unit-regular structures such as a frame, an arch, a truss, a plate and a shell may be modeled by this automated modeling tool independently and may be combined later with the total model.

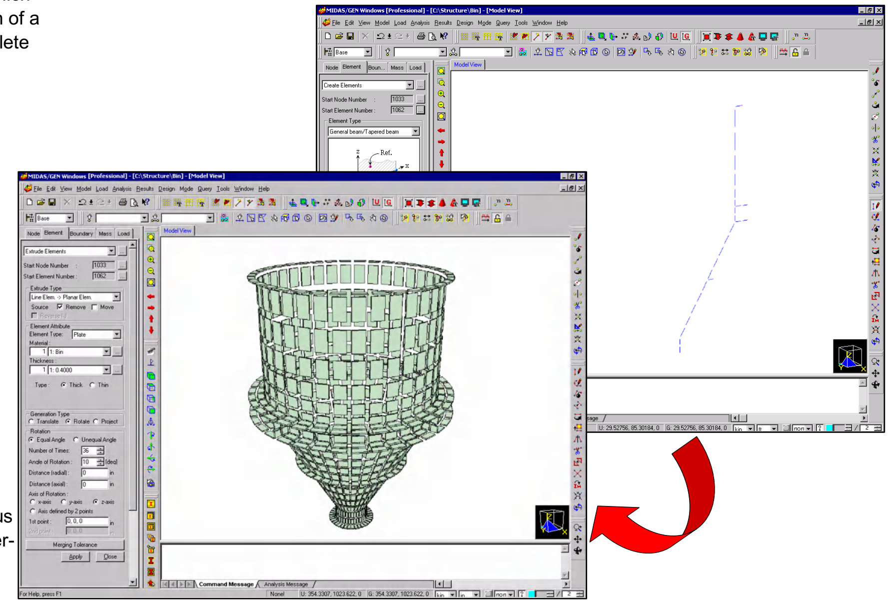
Refer to "Modeling> Model>Building> Building Generation" of On-line Manual.

Building Generation

In a building structure, Building Generation allows efficient modeling of the geometry reflecting story heights and section or material variation of beams, columns, walls and bracings simultaneously.

🎧 Extrude the temporary beam elements, which represent a section of a bin to form a complete bin reinforced with stiffeners.

🎧 The Shrink function reduces the element sizes and thus enables us to readily verify the inter-connection of the elements.



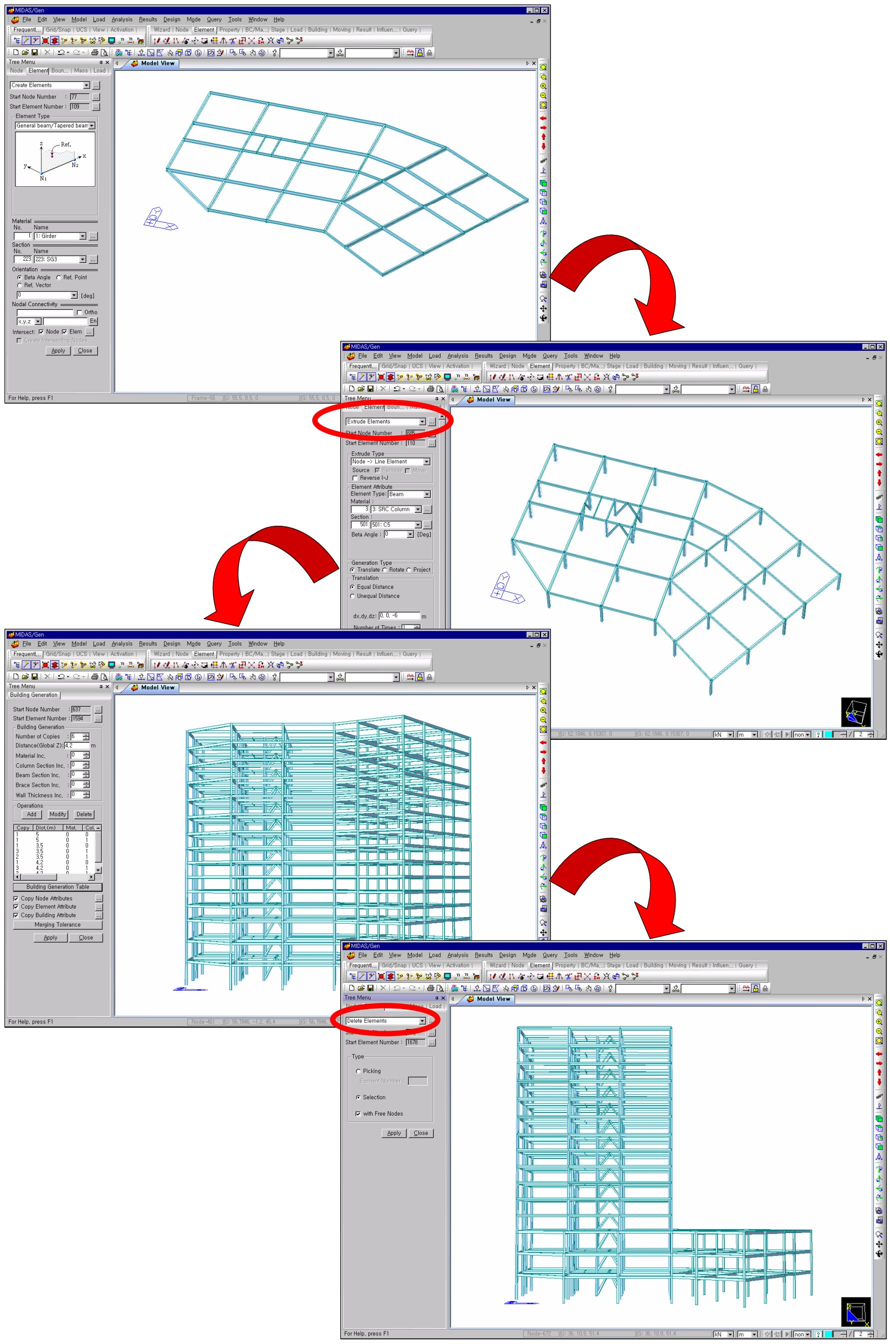
Model of a Bin

2nd floor modeling of an office building by Frame Wizard and UCS.

Frame structure with columns generated by the use of Extrude followed by creating braces.

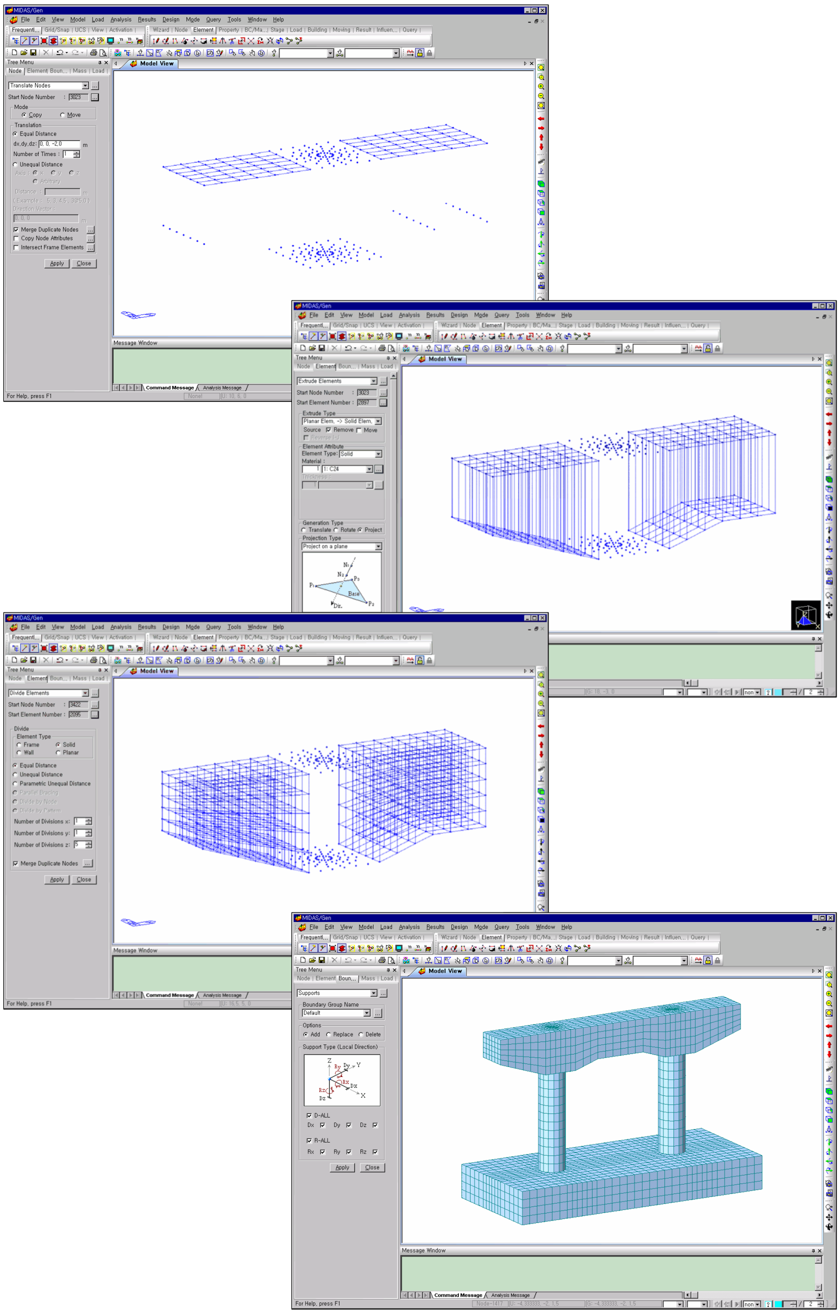
Building structure generated with varying floor-to-floor heights and increments of section numbers.

Remove the unnecessary elements in the upper floor setback.



Modeling sequence of a multi-story office building

- Generate plate elements in the upper portion of the coping and select the projecting plane below.
- Extrude the upper plate elements onto the lower projecting plane to create solid elements.
- Divide the solid elements vertically to generate finer sub elements.
- Complete this T-shaped pier model by adding the footings and columns.



Modeling sequence of a T-shaped pier

Material and Section Properties Generation

MIDAS/Gen provides various material and section database, and we are also free to define User-defined material and section properties. ***Sectional Property Calculator*** calculates section properties for an irregularly shaped section.

Material Property Data

MIDAS/Gen supports the following material properties:

Steel

ASTM (**A**merican **S**ociety for **T**esting **M**aterials)
A total of 40 built-in types of steel database (A36, A53, A242-40, etc.)
CSA (**C**anadian **S**tandards **A**ssociation)
A total of 48 built-in types of steel database (230G(H), 350G(H), etc.)
BS (**B**ritish **S**tandards)
A total of 23 built-in types of steel database (43A, 50A, etc.)
DIN (**D**eutsches **I**nstitut für **N**ormung e.V.)
A total of 11 built-in types of steel database (St 37-2, St 52-3, etc.)
EN (**E**uropean Code)
A total of 12 built-in types of steel database (S235, S275, etc.)
JIS (**J**apanese **I**ndustrial **S**tandards)
A total of 13 built-in types of steel database (SS400, SM490, etc.)
GB (**G**uojia **B**iao Zhun, China)
A total of 5 built-in types of steel database (Grade3, 16Mn, etc.)
JGJ (**J**ian Zhn **G**ong ye **J**ian Zhn Biao Zhun, China)
A total of 5 built-in types of steel database (Q235, Q295, etc.)
JTJ (**J**iao **T**ongbu **J**ian She Bia Zhun, China)
A total of 2 built-in types of steel database (A3, 16Mn)
KS (**K**orean Industrial **S**tandards)
A total of 45 built-in types of steel database (SS400, SM490, etc.)
KS-Civil (**K**orean **Civil** **S**tandards)
A total of 27 built-in types of steel database (SS400, SM490, etc.)

Concrete

ASTM (**A**merican **S**ociety for **T**esting **M**aterials)
A total of 7 built-in types of concrete property database (Grade C2500, Grade C3000, etc.)
CSA (**C**anadian **S**tandards **A**ssociation)

A total of 6 built-in types of concrete property database (C25, C30, etc.)

BS (**B**ritish **S**tandards)

A total of 10 built-in types of concrete property database (C35, C40, etc.)

EN (**E**uropean Code)

A total of 9 built-in types of concrete property database (C30/37, etc.)

JIS (**J**apanese **I**ndustrial **S**tandards)

A total of 16 built-in types of concrete property database (F_C27, F_C30, etc.)

GB (**G**uojia **B**iao Zhun, China)

A total of 14 built-in types of steel database (C15, C20, etc.)

GB-Civil (**G**uojia **B**iao Zhun, China)

A total of 7 built-in types of steel database (15, 20, etc.)

KS (**K**orean Industrial **S**tandards)

A total of 19 built-in types of concrete property database (C270, etc.)

KS-Civil (**K**orean **Civil** **S**tandards)

A total of 19 built-in types of concrete property database (C270, etc.)

Reinforcing Steel

ASTM (**A**merican **S**ociety for **T**esting **M**aterials)

A total of 4 built-in types of reinforcing steel database (Grade 60, etc.)

CSA (**C**anadian **S**tandards **A**ssociation)

A total of 6 built-in types of reinforcing steel database (300R, etc.)

BS (**B**ritish **S**tandards)

A total of 2 built-in types of reinforcing steel database (SD460, etc.)

EN (**E**uropean Code)

A total of 6 built-in types of reinforcing steel database (SD400, SD460, etc.)

JIS (**J**apanese **I**ndustrial **S**tandards)

A total of 6 built-in types of reinforcing steel database (SD345, etc.)

GB (**G**uojia **B**iao Zhun, China)

A total of 4 built-in types of reinforcing steel database (HPB235, etc.)

GB-Civil (**G**uojia **B**iao Zhun, China)

A total of 4 built-in types of reinforcing steel database (Grade 1, etc.)

KS (**K**orean Industrial **S**tandards, Civil/Building Structures)

A total of 5 built-in types of reinforcing steel database (SD40, etc.)

KS-Civil (**K**orean **Civil** **S**tandards)

A total of 5 built-in types of reinforcing steel database (SD40, etc.)

SRC

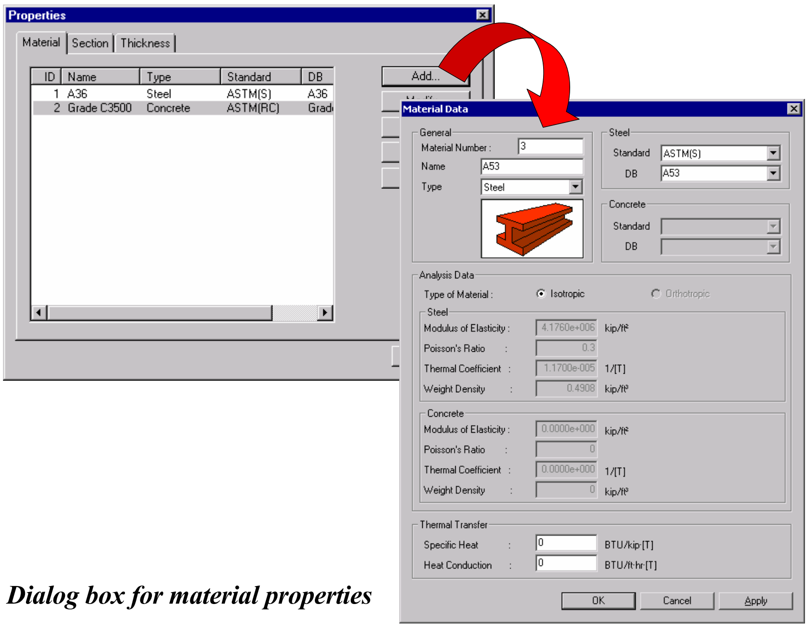
Combinations of the above-mentioned steel and concrete materials

User Defined

The user may define the properties directly as well as defining the properties of Isotropic Material and Orthotropic Material.

To enter material properties, use ***Model>Properties>Material*** or ***Material***.

At the convenience of the user, enter material properties by the following methods:



Dialog box for material properties

When additional material properties data are to be entered during the elements generation process, use the ... button to the right of the material properties list of the Create Elements Dialog Bar.

The following is a method of assigning material properties by selecting from the predefined materials list specified at the elements generation stage after defining the general material properties:

1. Click ***Material*** for material data input.
2. Select the desired material properties from the list of material properties of the Dialog Bar used for the generation of elements.
3. Use the automatic incremental numbering function for material properties in the Dialog Bar used for the duplication of elements. This is convenient where properties of the duplicated elements are different from that of the elements being duplicated.

It makes no difference if steps 1 and 2 are reversed. If elements are created without specifying the material data, the material number "1" is assigned automatically.

The following is a method of assigning arbitrary material numbers to the elements being generated irrespective of the true material data. The assigned materials are subsequently revised.

1. Click ***Material*** for material data input.
2. Create elements without assigning material data concurrently.
3. Use ***View>Select*** or the related Icons to select the elements whose material properties are to be assigned or modified.
4. Use ***Model>Elements>Change Element Parameters*** or ***Change Element Parameters*** to assign new material numbers. Alternatively assign material properties by ***Drag & Drop*** after selecting relevant material properties from ***Works Tree***.

Only a few material properties are used for modeling real structures. The first method is generally more practical. Use ***Change Element Parameters*** to modify material data subsequently.

For effective management of modeling, assign material numbers based on the element types (beam, column, wall, brace, etc.) even if the material types are identical.

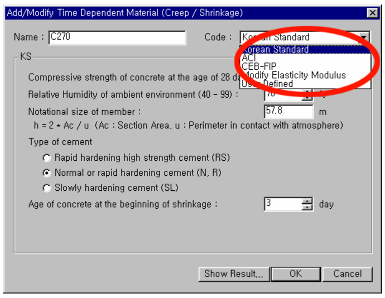
Similar material data used in other model files (fn.MGB) may be imported Import for entering material properties.

Time Dependent Material Property Data

Construction stage analysis is required for a high-rise building structure reflecting short-term and long-term deformations such as elastic column shortening, concrete creep and shrinkage. In such a case and the case of a heat of hydration analysis, time dependent material properties are required.

The following outlines the method of defining the time dependent material properties:

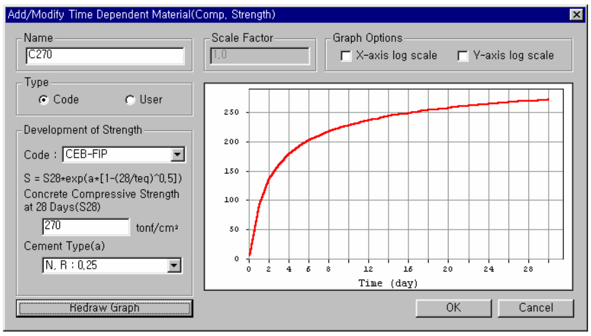
1. Define material property data for creep and shrinkage in ***Model> Properties>Time Dependent Material (Creep/Shrinkage)***.



Selection of Code for defining Material Properties

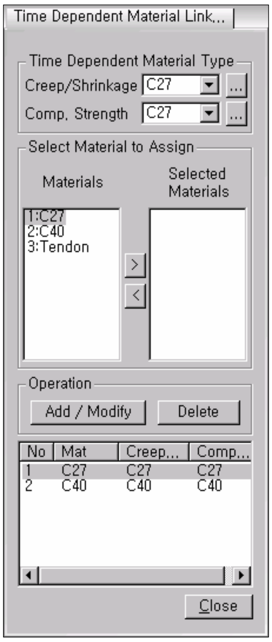
If User Defined is selected, the user is required to directly specify relevant creep and shrinkage functions in ***Model>Properties>Time Dependent Material (Creep/Shrinkage) Function***.

2. Define a function of modulus of elasticity of concrete in ***Model> Properties>Time Dependent Material (Comp. Strength)***.



Variation of Modulus of Elasticity of Concrete

3. Relate the time dependent material properties to the general material properties previously defined in ***Model>Properties>Time Dependent Material Link***.



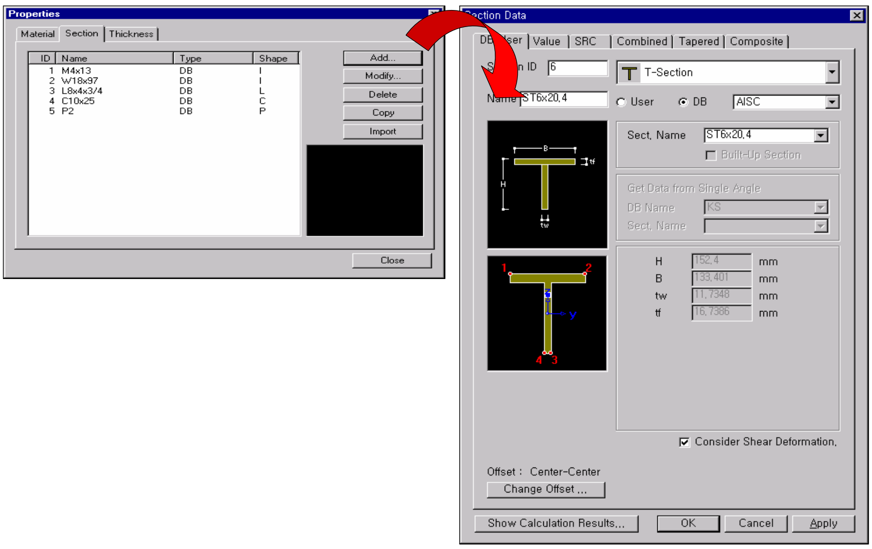
Time Dependent Material Link Dialog Bar

Section Data

MIDAS/Gen supports the following section property data:

DB	Selection among international standard section databases	
	AISC 2K (US)	American Institute of Steel Construction, 2000 Imperial Unit
	AISC 2K (SI)	American Institute of Steel Construction, 2000 Metric Unit
	AISC	American Institute of Steel Construction,
	CISC 02 (US)	Canadian Institute of Steel Construction, Imperial Unit
	CISC 02 (SI)	Canadian Institute of Steel Construction, Metric Unit
	BS	British Standards
	DIN	Deutsches Institut für Normung e.V.
User	Key dimensions of standardized sections	
Value	Section properties defined by the user	
SRC	SRC sections	
Combined	Combined sections made up of two section types	
Tapered	Tapered sections	

The section data in **MIDAS/Gen** is entered using ***Model>Properties>Section*** or ***Section***.



Dialog box of Section data

Depending on the user's preference, section data in **MIDAS/Gen** can be entered by the following methods:

When section data are additionally required while creating elements, it will be more convenient to use the ... button to the right of the section list in the Create Element Dialog Toolbar.

Selecting sections from the list of section data defined in advance and assigning them to the elements being created:

1. Click ***Section*** to enter the section data.
2. Select the desired sections from the list of sections of the Dialog Bar used for the generation of elements.
3. Use the automatic incremental numbering function for sections in the Dialog Bar used for duplicating elements where the sections of the duplicated elements and the original elements are different.

There is no difference if the steps 1 and 2 are reversed. If elements are created without specifying the section data, the section number "1" is assigned automatically.

Revising the temporary section data assigned to the elements whose section numbers are arbitrarily assigned to create the elements:

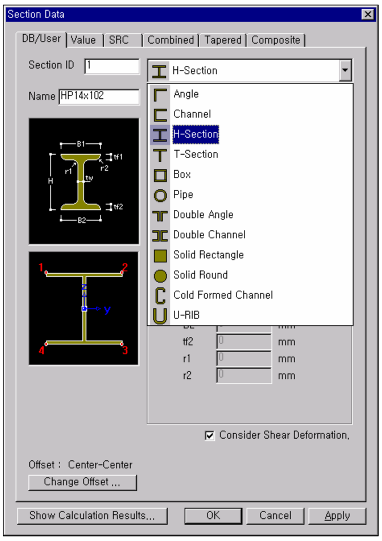
1. Click ***Section*** to enter the section data.
2. Create elements without assigning section data concurrently.
3. Use ***View>Selection*** or the related Icons to select the elements whose section data will be modified or assigned.
4. Use ***Model>Elements>Change Element Parameters*** or ***Change Element Parameters*** to assign new section numbers.

The first method may be advantageous for a relatively simple structure with only a few section types. The second method may be more practical for general structures with many section types.

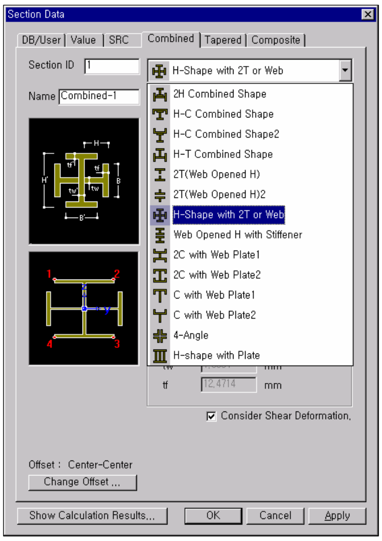
Similar section data may be imported ... from the MGB files (fn.mgb) used in other models. The user may expedite the sectional data entering process by establishing a DB in an MGB file containing built-up sections and other frequently used sections. This may also come in handy as the DB can be applied to the automatic design of steel structures.

MIDAS/Gen computes the following section properties automatically:

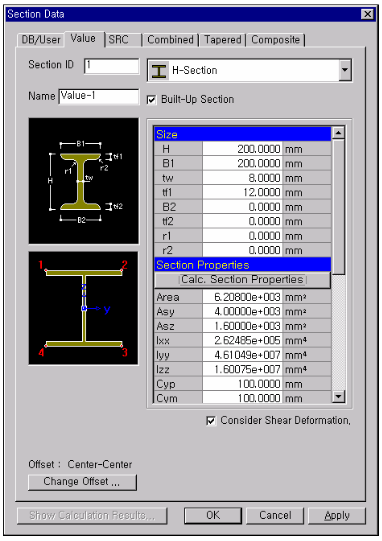
🎧 It is not necessary to enter sectional dimensions for elements with varying cross sections.



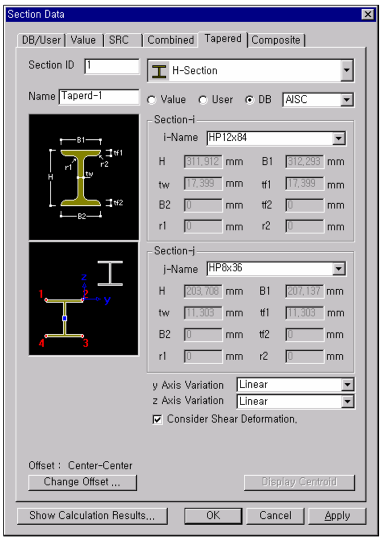
DB/User Section



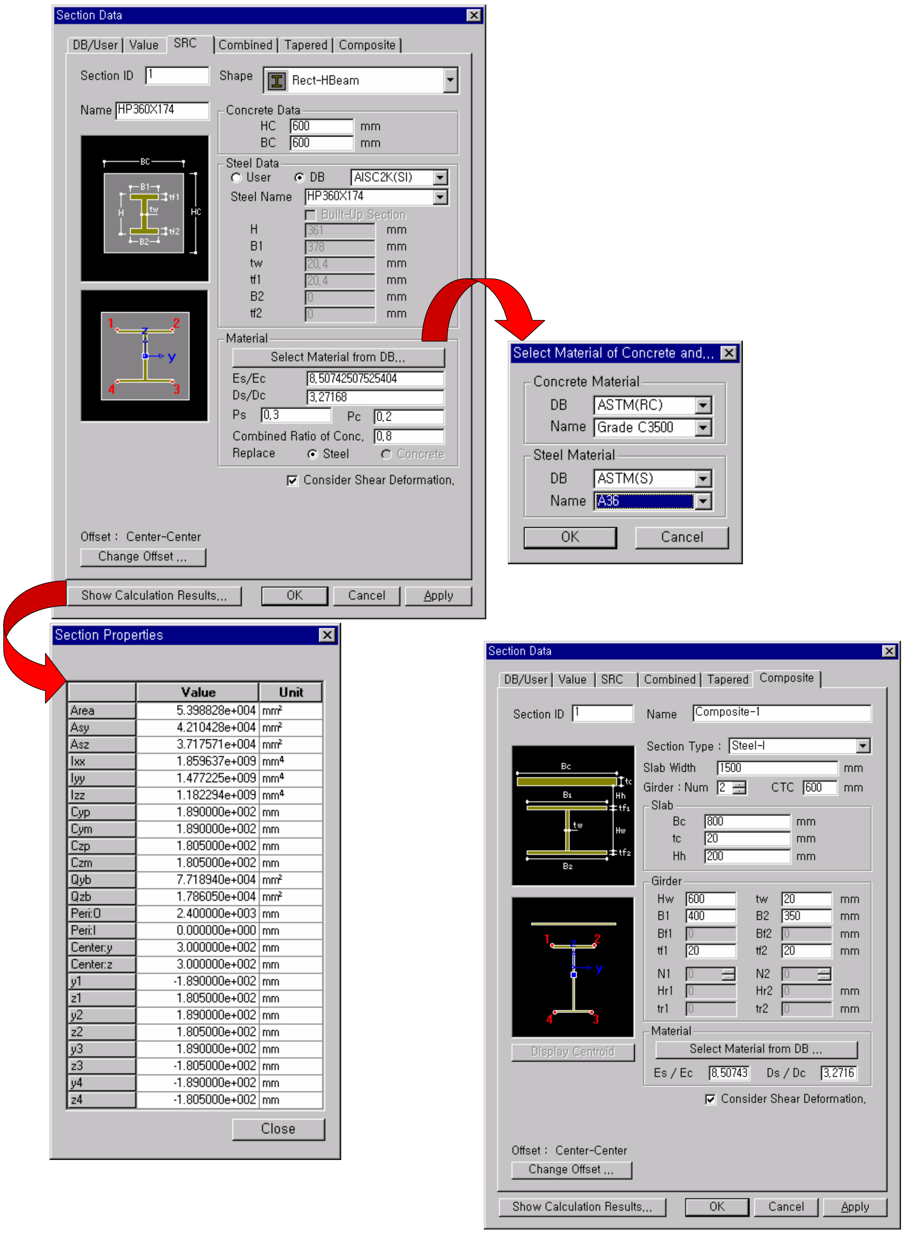
Combined Section



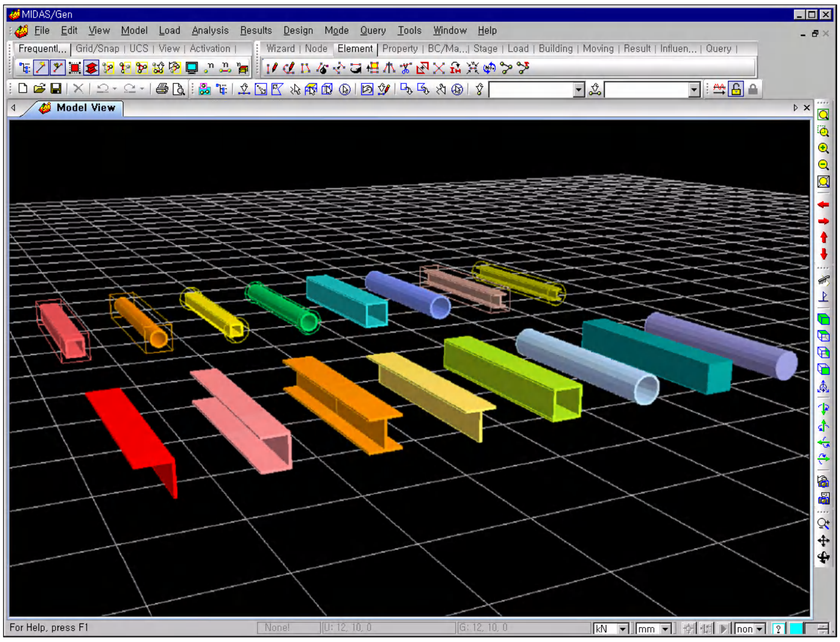
Value Section

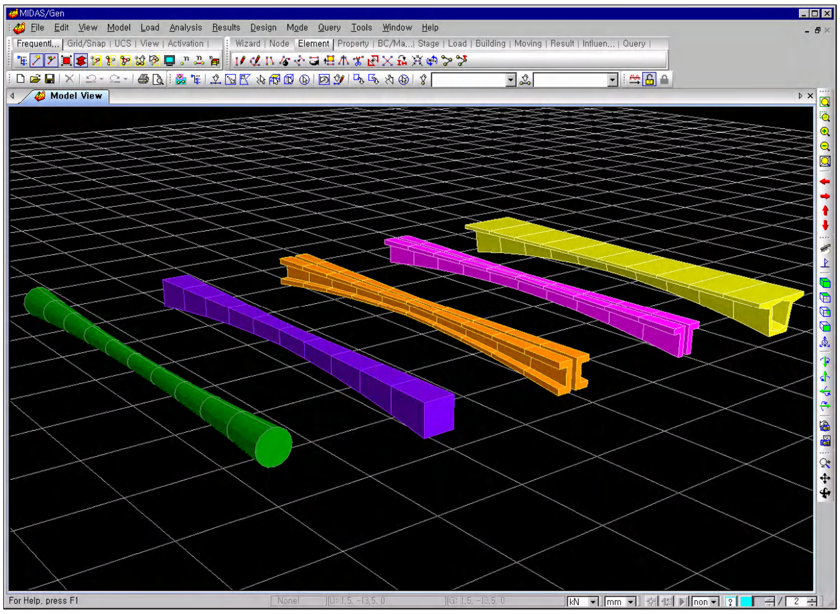


Tapered Section



SRC material properties
& sectional data

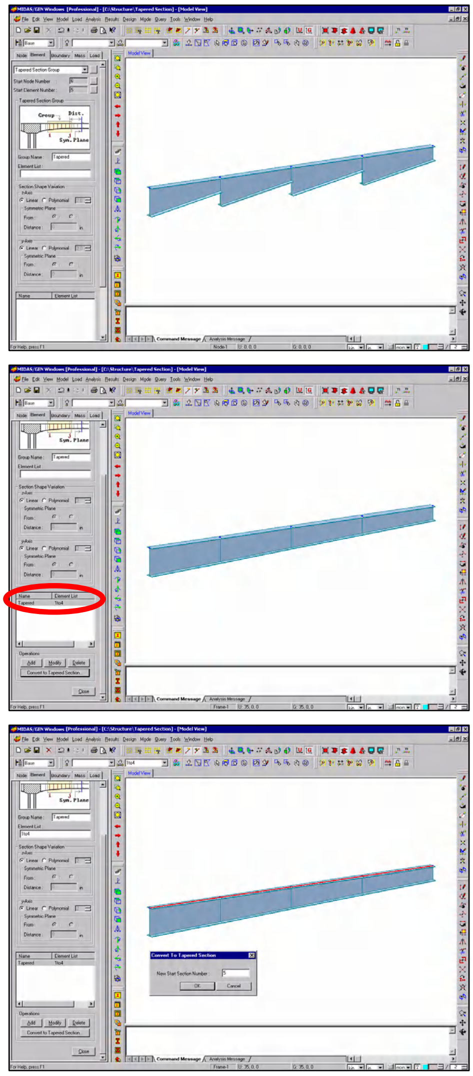




Applicable Section Shapes

Model>Properties>Tapered Section Group automatically calculates the section properties of tapered (non-prismatic) elements in a zone of section variation.

Prior to analysis, input tapered elements by assigning them to a ***Tapered Section Group*** to calculate the section properties of the individual tapered elements, and then ungroup to retain the individual section properties. The ungrouping reduces analysis time, especially in a construction stage analysis where repetitive sub-analyses are internally performed.



Tapered Section Group

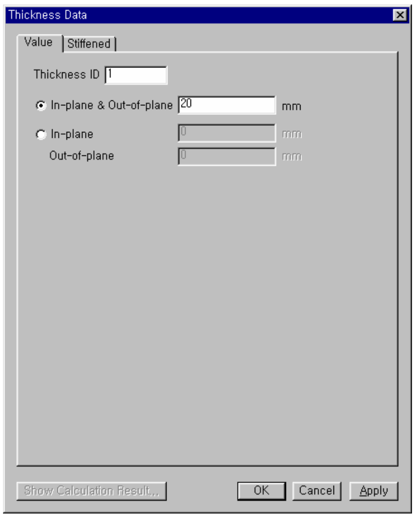
Thickness Data

The thickness data for plate elements in **MIDAS/Gen** are considered in the following two ways:

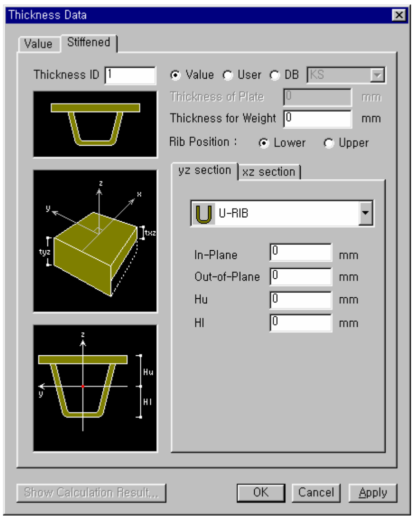
- Applying the same thickness to compute the stiffness for both in-plane and out-of-plane directions.
- Applying different thicknesses to compute the stiffness for in-plane and out-of-plane directions.

For plane stress elements, only the in-plane behavior is taken into account, and as such only the in-plane thickness data are applied regardless of the data entered. The Out-of-plane stiffness is irrelevant.

MIDAS/Gen has the capability of entering stiffened or reinforced (ribbed) plates, which may often be used in thin plates.



Entering thickness data (Value)

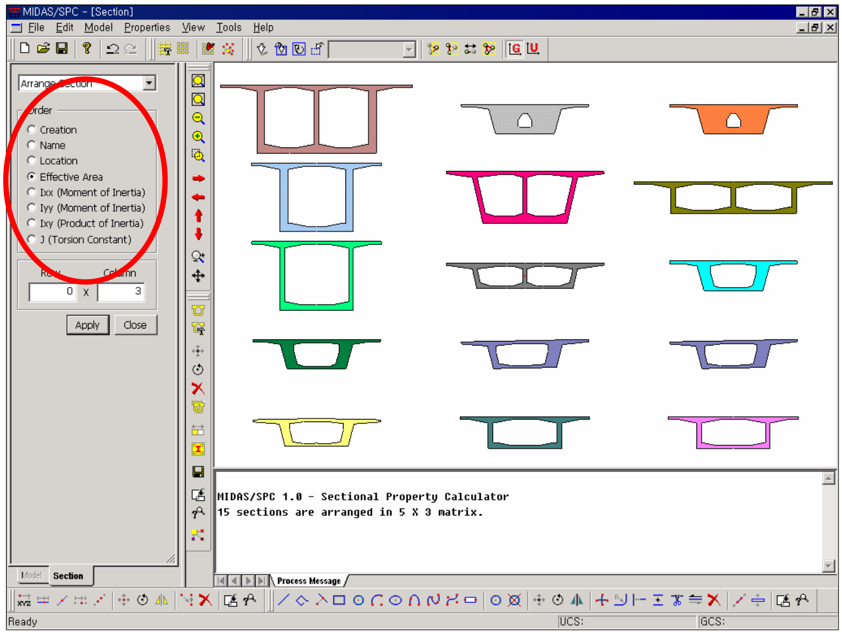


Entering thickness data (Stiffened)

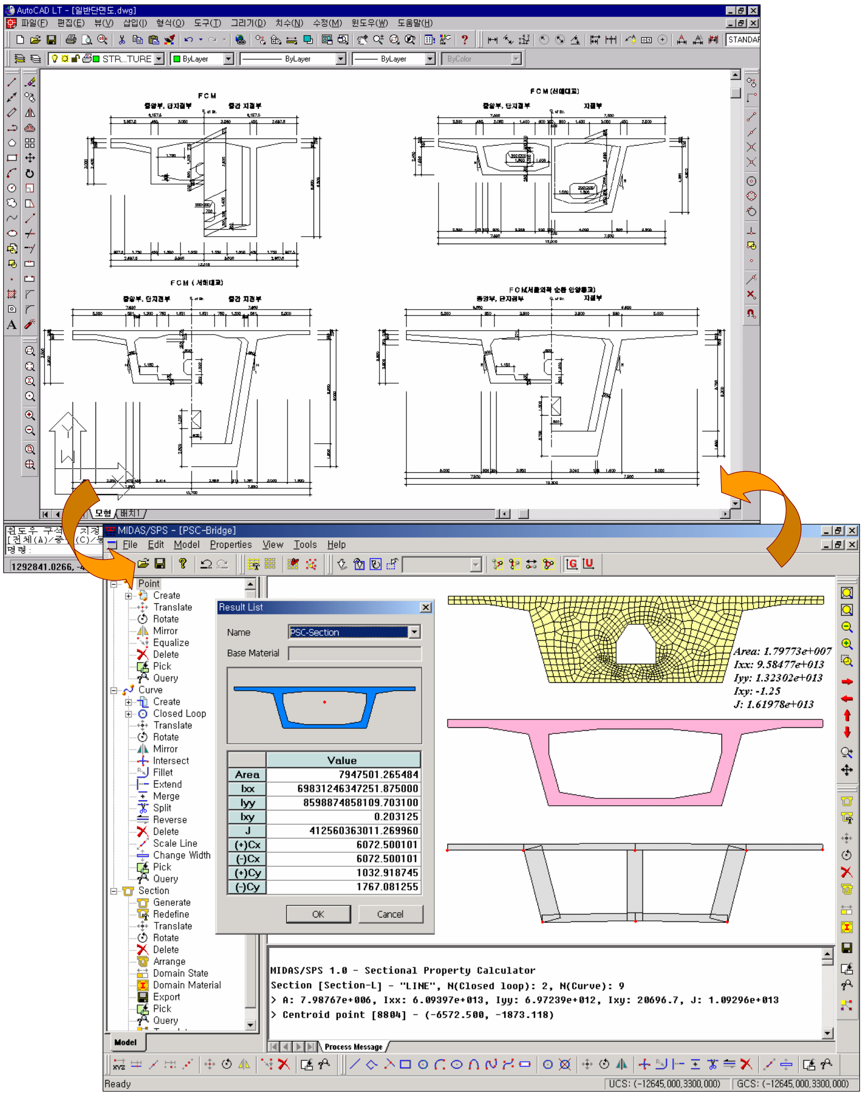
Sectional Property Calculator (SPC)

MIDAS/Gen provides SPC, which calculates stiffness data for any shape or form. The section shape can be drafted, or a DXF file can be imported. Invoke ***Tool>Sectional Property Calculator*** from **Main Menu**, and the section properties calculated are ***import***ed in ***Section*** when modeling a structure.

- Import a section shape through **AutoCAD DXF**.
- Simple entry of a section shape by various modeling tools.
- Optimized mesh is automatically created for calculating the section.
- The properties of a hybrid section consisted of a number of different materials can be calculated.



A number of sections are arranged in the order of sizes, and the section properties are individually calculated for each section



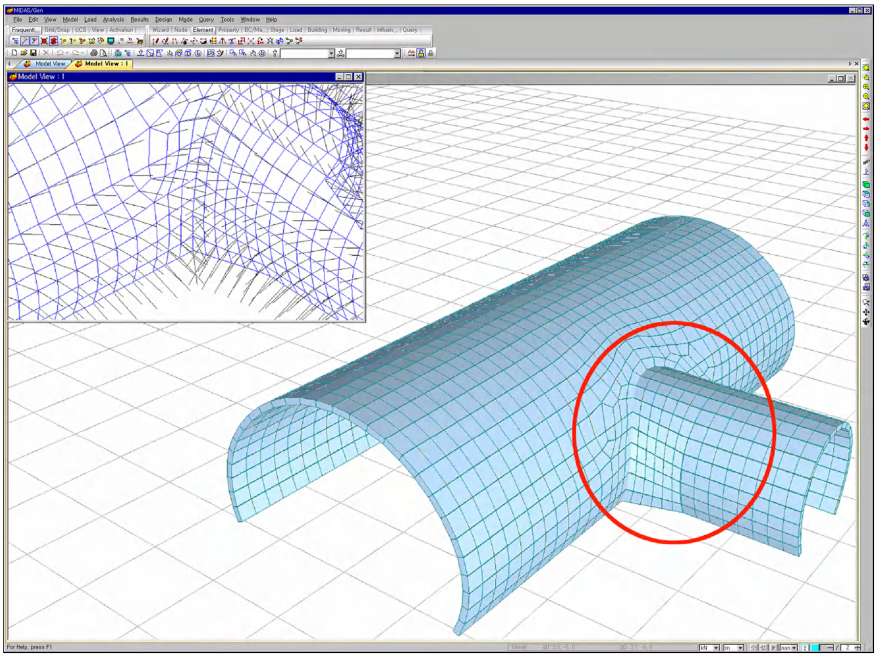
Sectional Property Calculator calculates the section properties of the section shapes read in from AutoCAD DXF files

Boundary Conditions Input

MIDAS/Gen provides unique boundary conditions such as ***General Spring Supports*** to account for lateral stiffness of piles, Compression-only boundary elements to reflect foundations and Tension-only boundary elements.

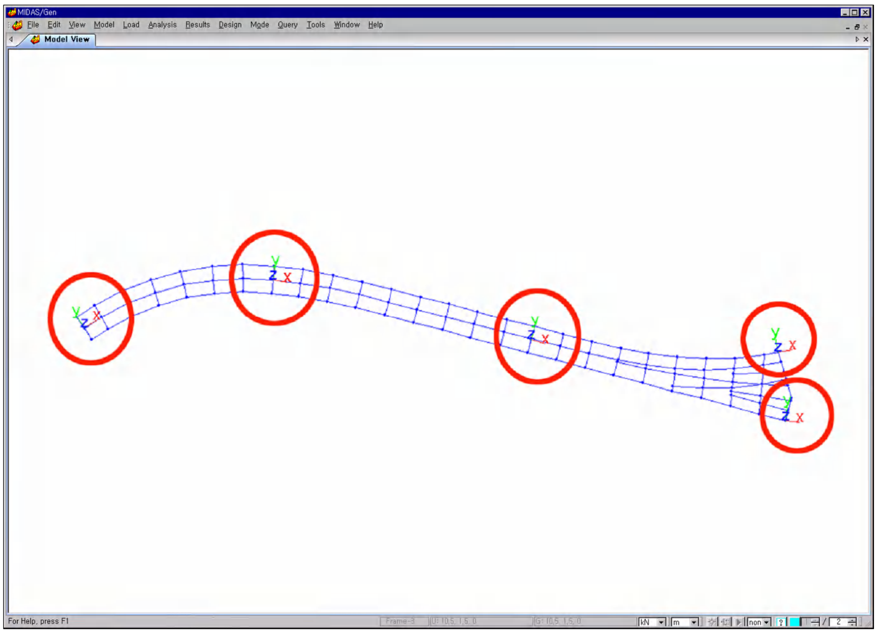
Boundary Conditions

- ***Supports***
- ***Point Spring Supports***
- ***Define General Spring Type***
- ***General Spring Supports***
- ***Surface Spring Support***
- ***Elastic Link***
- ***Nonlinear Link Properties***
- ***Nonlinear Link***
- ***Beam End Release***
- ***Beam End Offset***
- ***Plate End Release***
- ***Rigid Link***
- ***Diaphragm Disconnect***
- ***Panel Zone Effect***
- ***Node Local Axis***
- ***Story Diaphragm Group for Construction stage***



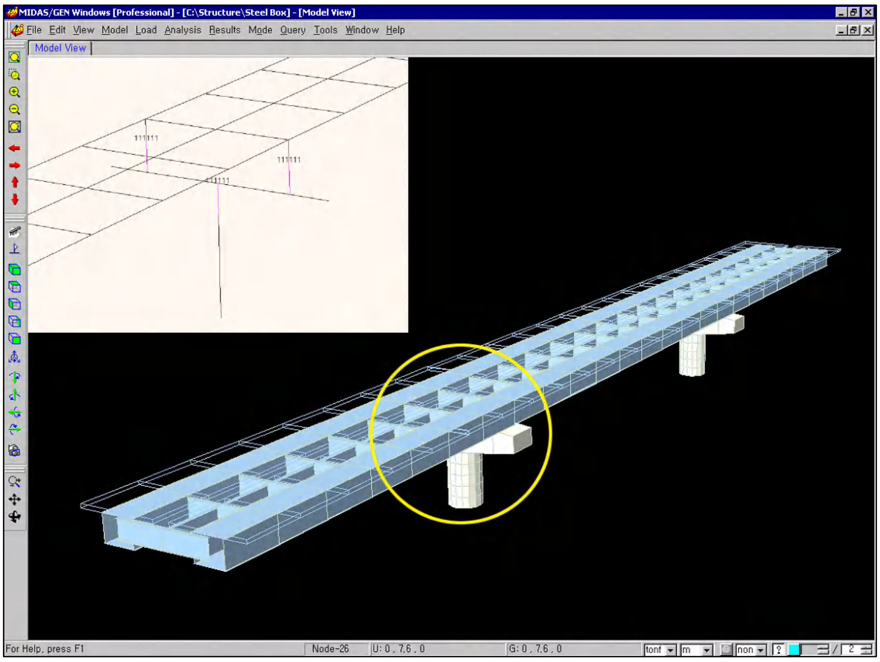
Display of equivalent soil springs auto-generated for a tunnel lining

Surface Spring Supports is applied in the case where a structure is in contact with soils such as a foundation mat or a tunnel. The effective contact area of each node of plate and solid elements and the modulus of sub-grade reaction are used to automatically calculate and input the equivalent spring stiffness.



Display of Boundary Conditions in the zone of lane expansion of a curved bridge

Elastic Link can be applied to represent an elastic bearing on a bridge pier, which eliminates the need for incorporating a fictitious beam element in the modeling. All that is required is just the stiffness in the relevant direction, which then produces the reaction.



Rigid Link representing offset between the main girder and bridge pier

Plate End Release and ***Beam End Release*** represent the inability of resistance in certain degrees of freedom at the element ends. ***Node Local Axis*** is used to represent skewed boundary conditions relative to the Global Coordinate System, such as a bridge supported on skewed supports.

Nonlinear Link can model base isolators and dampers in structures representing the behaviors of nonlinear damping history. Nonlinear Link Element is composed of 6 linear or nonlinear springs linking two nodes, which represent one axial spring, two shear springs, one torsional spring and two bending springs.

Loads Generation

The types of loading implemented in the analysis tasks in **MIDAS/Gen** are as follows:

- Static Loads
- Dynamic Loads

The static loads are used to perform static analyses for unit loading conditions. The dynamic loads are used to perform response spectrum analyses or time history analyses.

Load Group is applied to the Construction Stage Analysis in which groups of loads are activated and inactivated at different stages of construction.

Static Loads

The following two steps specify static loads in MIDAS/Gen:

1. Use ***Load>Static Load Cases*** to enter the static unit loading conditions.
2. Enter the loading data using various static loading functions provided in ***Load***.

When modifying or adding unit loading conditions in the process of entering loads, click the ... button located to the right of the Load Case Name field of the corresponding load dialog bar for quick changes.

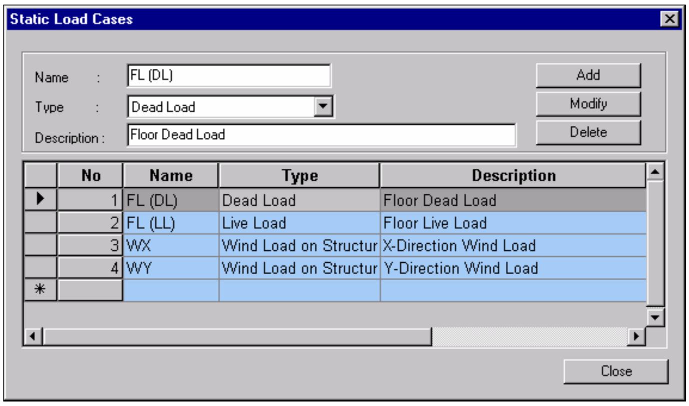
A static analysis is performed for each static unit loading case. Use the ***Results>Combinations*** function to combine analysis results during the post-processing mode.

It is also possible to carry out the structural analysis after converting the loading combination conditions entered in ***Load>Create Load Cases Using Load Combinations*** into individual loading cases.

This is an extremely useful tool for entering loading cases when nonlinear elements are used in the analysis model.

☊ Specify the name of a static unit loading condition in the name field. This name is an identification used for loading combinations and specifying loading conditions required for the geometric stiffness matrix formation in a buckling analysis or a P-Delta effect analysis.

☊ The type field is used to automatically create the loading combinations according to various design codes in different countries. It supports a list of 24 types of loads. For detail information, refer to On-line Manual.



Entering static unit loading conditions

MIDAS/Gen supports the following types of static loading:

Self Weight
Element self weight

Nodal Loads
Nodal concentrated loads

Specified Displacements of Supports
Forced displacements of supports

Element Beam Loads
Concentrated or distributed loads acting on beam elements

Line Beam Loads
Beam loads on a number of consecutive beam elements aligned in a straight line

Typical Beam Loads
Common types of beam loads resulting from floor loading

Define Floor Load Type

Assign Floor Loads
Floor loads on the top of beam or wall elements

Define Plane Load Type
Define the type of loads on a plane, which will be applied to the nodes of plate/solid elements and any desired location irrespective of element type.

Assign Plane Loads
Apply the defined planeloads to the plane in which the plate/solid elements are located.

Prestress Beam Loads
Pre-stress loads in beam elements

Pretension Loads
Pretension loads in truss elements, cable elements and tension/compression-only elements

Tendon Prestress Loads
Define tendon prestress loads

Pressure Loads
Pressure loads acting on the thicknesses or surfaces of plate and solid elements

Surface pressure loads can be applied to even Plane Stress elements for Geometric Nonlinear Analysis.

Hydrostatic Pressure Loads
Pressure loads resulting from the potential energy of fluid

System Temperature
The final temperature of the entire structure necessary for thermal stress analysis

Nodal Temperatures
Nodal temperatures for thermal stress analysis

Element Temperatures
Temperatures on elements for thermal stress analysis

Temperature Gradient
Temperature gradient between the top and bottom of beam elements or plate elements

Beam Section Temperatures
Define a temperature difference on a section of a beam element.

Time Loads for Construction Stage
Assigning specific elements with construction time duration to elapse at a specific construction stage

Creep Coefficient for Construction Stage
Assigning creep coefficients to specific elements at a specific construction stage

Initial Forces Control Data
Saving initially entered axial forces as the results of a separate loading condition

Initial Force for Geometric Stiffness
Imposing initial axial forces to specific elements for calculating geometric stiffness

Wind Loads
Wind loads automatically computed in accordance with IBC (2000), UBC (1997), ANSI (1982), NBC (1995), Eurocode-1 (1992), BS6399 (1997), JIS, KS codes

Static Seismic Loads
Equivalent static seismic loads automatically computed in accordance with IBC (2000), UBC (1991, 1997), ATC3-06, NBC (1995), Eurocode-8 (1996),, JIS, KS codes

Ambient Temperature Functions
For Heat of Hydration Analysis

Convection Coefficient Functions
For Heat of Hydration Analysis at the boundary surface of a structure

Element Convection Boundary
Boundary condition for heat transfer by convection on the surface of a structure

Prescribed Temperature
Constant temperature condition independent of time

Heat Source Functions
For Heat of Hydration Analysis

Assign Heat Source
Heat source function assigned to each element

Pipe Cooling
Pipe cooling data for the reduction of temperature

Define Construction Stage for Hydration
For Heat of Hydration Analysis

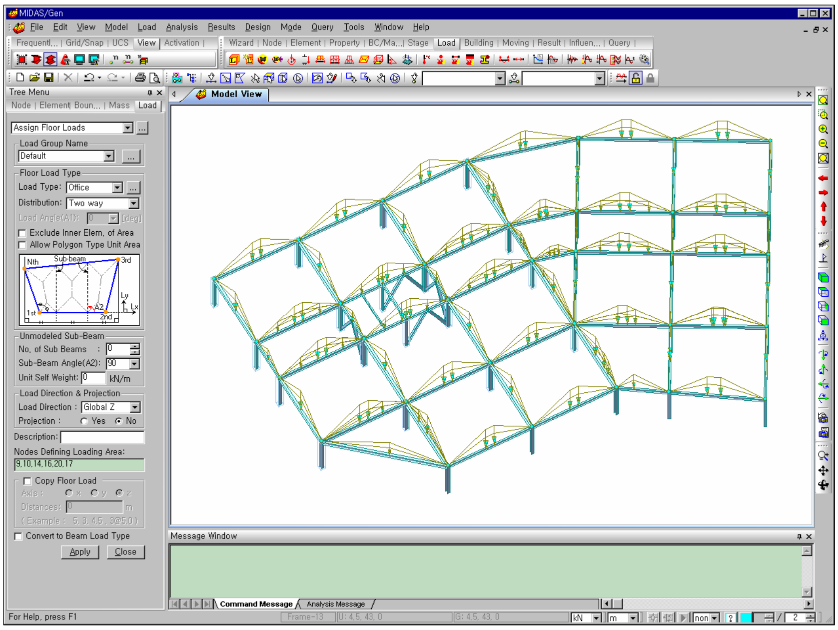
Loading Sequence for Nonlinear Analysis

Assign loading application order for nonlinear analysis.

Define Construction Stage
Define analysis models for each construction stage.

Select Construction Stage for Display
Activate the selected stage on the screen.

☊ Use Floor Load to enter dead and live loads simultaneously on an inclined roof.

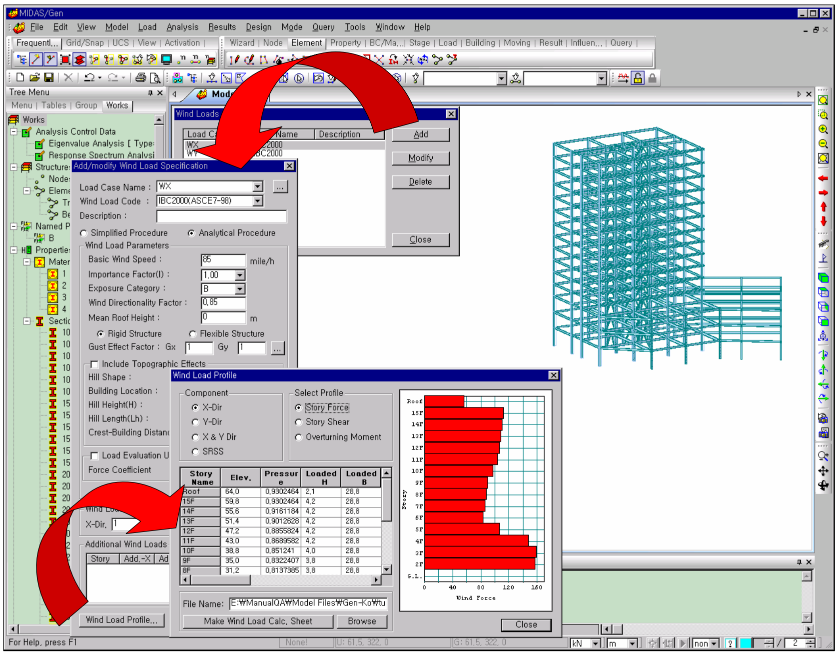


Floor Load

☊ The applications of Floor Load can be extended to all the planes present in a model. Snow and wind loads also can be generated through the use of Floor Load.

☊ Wind and equivalent static seismic loads exerting on a building can be easily generated via automatic load calculation.

☊ The gust factor (G_f) required to calculate the wind loads for a flexible structure can be calculated.
The natural periods of vibration (T) required to calculate the seismic loads can be calculated.

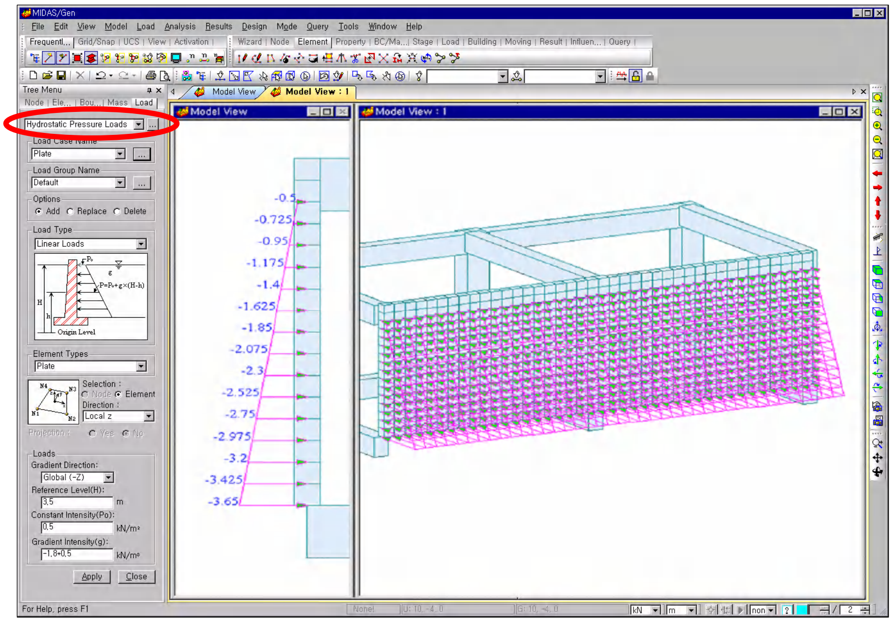


Automatic generation of Wind Load

Soil or hydraulic pressures acting on basement walls or retaining walls can be easily generated by means of Hydrostatic Pressure Loads.

Hydrostatic Pressure Load automatically calculates lateral loads acting on plate or solid elements due to soil or fluid. The applied loads are automatically converted even when the elements are divided or merged.

Temperature loads (changes) can be applied to the total structure as well as to individual nodes. Temperature gradients along the ECS axes of line elements may be also specified.



Pressure Load: Exterior basement wall supporting soil pressure

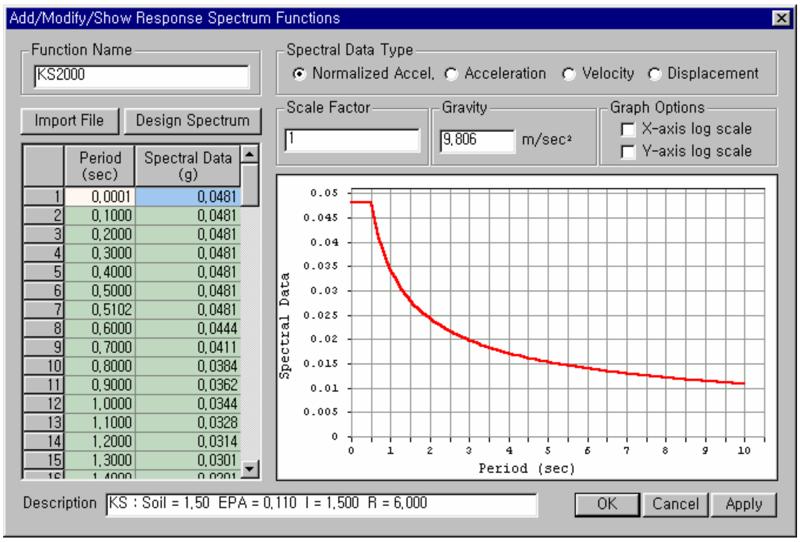
Dynamic Loads

The data entry process for the response spectrum analysis consists of the following:

1. Define the response spectrum data in ***Load>Response Spectrum Analysis Data > Response Spectrum Functions***.

   The response spectrum data can be defined using the following four methods:

   - The user directly enters the spectral data for each period.
   - The design response spectrum database is selected from the built-in database (UBC, GB 50011-2001, etc.).
   - The seismic response spectrum is extracted from the records of seismic accelerations using ***Seismic Data Generation***.
   - A file containing response spectrum data is imported.



Response Spectrum Function

2. Enter the response spectrum load case in ***Load>Response Spectrum Analysis Data>Response Spectrum Load Cases***. At this point, select the response spectrum defined in Step 1, and assign the direction of application, Scale Factor and the mode combination method.

Refer to Analysis Manual for the concept and features of Response Spectrum Analysis.

The sequence of data entry for time history analysis is as follows:

---

1. Define ***Time History Function in Load>Time History Analysis Data> Time Forcing Functions***.

   The ***Time Forcing Functions*** can be defined by the following four methods:

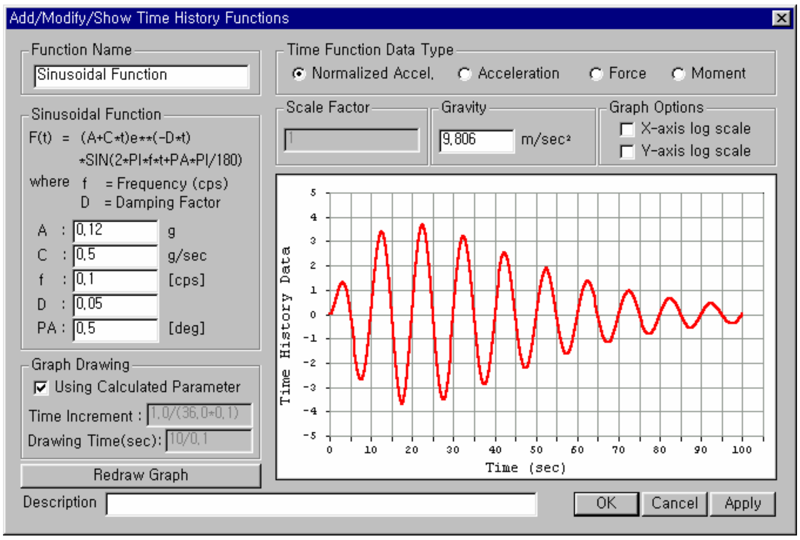
   - The user directly enters the loading data for each time step.
   - A selection is made from the built-in earthquake records database (32 types, such as El Centro earthquake, 1940, 270°).
   - A file containing the Time History Load is imported.
   - The Time Forcing Function is defined by entering Sinusoidal Function coefficients.

2. Enter the title of the time history analysis condition and the data for analysis control in ***Load>Time History Analysis Data>Time History Load Cases***.

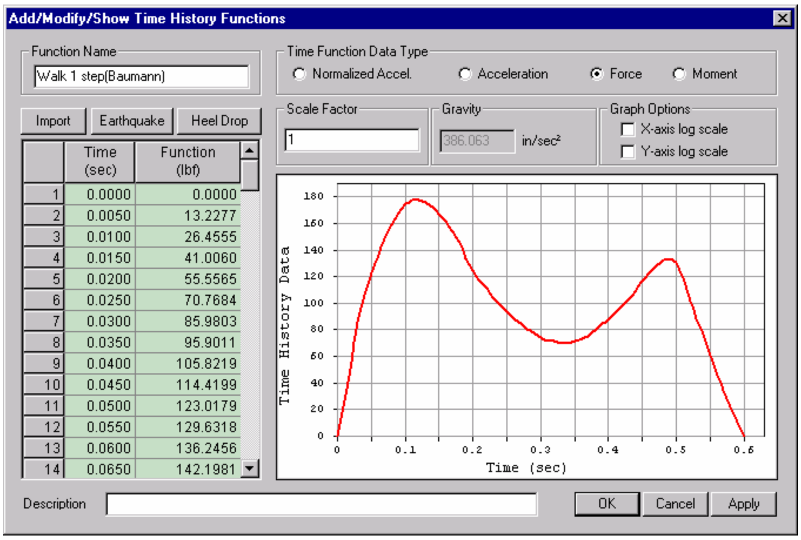
3. When an earthquake analysis is planned, assign the time history analysis condition and the Time History Load representing the ground motion to be considered in ***Load>Time History Analysis Data>Ground Acceleration***.

When performing a typical time history analysis, assign the time history analysis condition and the Time History Load to be considered using ***Load>Time History Analysis Data>Dynamic Nodal Loads***.

Refer to Analysis Manual and On-line manual for the concept and input process of Time History Analysis.



Time History Function: Sinusoidal



Time History Function: Heel Drop Load

Construction Stage Modeling Feature

MIDAS/Gen provides three types of stages; Base Stage, Construction Stage and Post-construction Stage. The characteristics of each stage type are as follows:

- ***Base Stage***
  General analysis is carried out at the Base Stage if the Construction Stage is undefined. If the Construction Stage is defined, structural modeling is prepared, and ***Structure Groups, Boundary Groups*** and ***Load Groups*** are defined and composed at the Base Stage without the execution of analysis.

- ***Construction Stage***
  Analyses for construction stages actually take place. The boundary and load conditions of the activated ***Boundary Groups*** and ***Load Groups*** of each corresponding stage are established.

- ***Post-construction Stage***
  Being the last stage of the construction stages, special analyses are carried out at the Post-construction Stage for conventional, response spectrum analysis, etc. in addition to the analysis for the construction stage loads.

Construction Stages are composed of ***Structure Groups, Boundary Groups*** and ***Load Groups*** by Activation and Deactivation of relevant entities. Accordingly, each stage consists of activated geometry, boundary and load conditions pertaining to that particular construction stage.

Construction Stage Modeling for a General Structure

The general modeling procedure for the construction stage analysis of a structure is as follows:

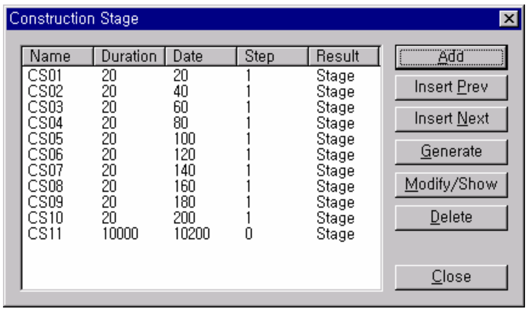
1. Prepare a structural model except for the boundary and load conditions.

2. Define ***Structure Groups*** in ***Model>Group>Define Structure Group***, and assign to each ***Structure Group*** relevant elements that will be constructed or removed together.

3. Define Boundary Groups in ***Model>Group>Define Boundary Group***.

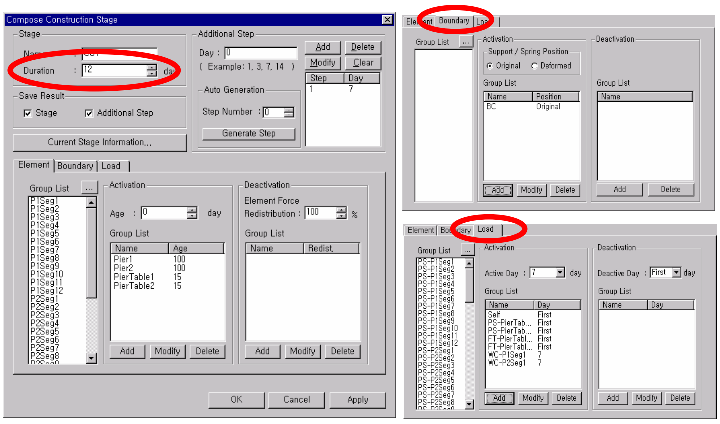
4. Define Load Groups in ***Model>Group>Define Load Group***.

5. Compose ***Construction Stages*** by clicking the Add button in ***Load>Construction Stage Analysis Data>Define Construction Stage***. You may click the Generate button to define a number of ***Construction Stages*** of identical duration and click the Modify/Show button to compose each construction stage.



Define Construction Stage dialog box

6. Specify ***Duration*** and whether or not to save the results in the ***Compose Construction Stage*** dialog box. Define ***Additional Steps*** if time variant loadings are applied within the same structure Group.



Compose Construction Stage dialog box

7. From the ***Group List*** of the ***Element*** tab**,** select the applicable element groups to be included in or excluded from each construction stage through activation or deactivation. ***Age*** represents the initial maturity of each element group. ***Element Force Redistribution*** represents the redistribution of the forces of each element group being deleted or inactivated into the remaining elements.

8. From the ***Group List*** of the ***Boundary*** tab, select the applicable boundary groups to be included in or excluded from each construction stage through activation or deactivation.

9. From the ***Group List*** of the ***Load*** tab, select the applicable load groups to be included in or excluded from each construction stage through activation or deactivation. ***Active Day*** and ***Inactive Day*** represent the dates of applying and removing each load group.

10. Once the construction stages are composed, we may switch around the construction stages in Stage Toolbar and input the boundary and load conditions of the ***Boundary Groups*** and ***Load Groups*** corresponding to each construction stage.

We can minimize input errors by inputting the load and boundary conditions in each corresponding construction stage.

Time Dependent Material Properties

The modeling procedure for reflecting the time dependent material properties of concrete is as follows:

1. Define the Creep and Shrinkage properties of concrete, which vary with maturity in ***Model>Properties>Time Dependent Material (Creep/ Shrinkage)***. MIDAS/Gen contains the ACI and CEB-FIP codes for defining creep and shrinkage properties of concrete and allows us to directly enter any test data.

2. Define the time variant compressive strength gain properties of concrete in ***Model>Properties>Time Dependent Material (Comp. Strength)***. MIDAS/Gen contains the ACI and CEB-FIP codes for defining compressive strength gain properties of concrete and allows us to directly enter any test data.

3. Relate the time dependent material properties to the general material properties in ***Model>Properties>Time Dependent Material Link***. When the two types of material properties are linked, the time dependent material properties will be used for construction stage analyses according to the maturity, and the general material properties will be applied to general analyses.

4. ***Notational Size of Member*** ($h=2\times A_c/u$) required for calculating the time dependent material properties of concrete is entered in ***Model>Properties>Change Element Dependent Material Property*** for each member.

5. Use ***Load>Creep Coefficient for Construction*** Stage if creep coefficients other than the values automatically calculated by MIDAS/Gen are desired. Input creep coefficients for each element at each construction stage in the form of loads. When the corresponding load groups are activated, the construction stage is created using the specified creep coefficient.

In the case of a structure where two or more structural components are separately erected in the same construction stage and yet the maturities are different as they are connected, MIDAS/Gen provides ***Load>Time Load for Construction Stage*** to account for the different timing effect. ***Time Load for Construction Stage*** thus enables us to impose time passage to specific elements, which is input as a type of load.

Prestress Input

MIDAS/Gen permits construction stage analyses reflecting the pre-stress effects of tendons exerted on a structure. It also considers the immediate pre-stress losses such as tendon/sheath friction, anchorage slip and elastic shortening as well as long term losses such as creep/shrinkage of concrete and tendon relaxation in construction stage analyses. The procedure for entering pre-stress is noted below.

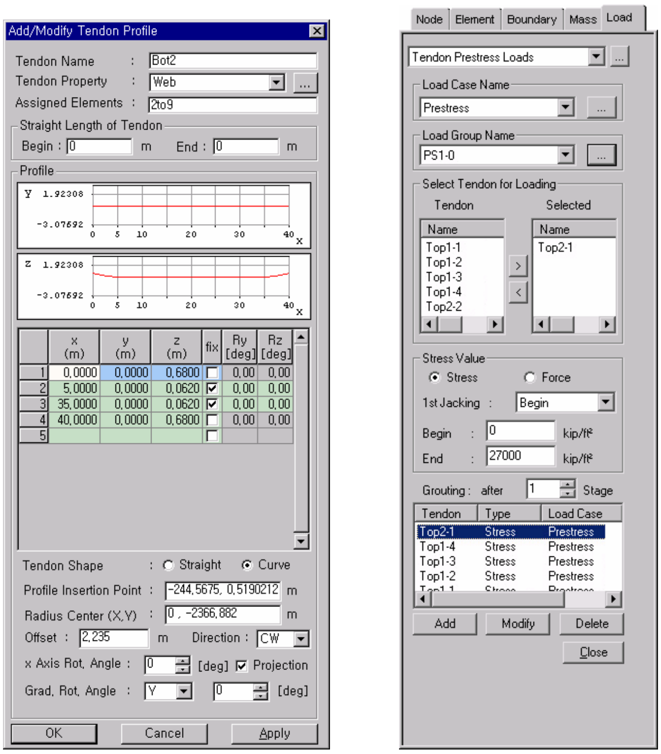
1. Specify the material properties of tendons in ***Model>Properties> Material***. MIDAS/Gen does not consider the tendons as independent elements, and as such only the modulus of elasticity of the tendons need be entered.

2. Enter the cross sectional area, pre-stress loss coefficients, duct diameter and strength of tendons in ***Load>Prestress Loads >Tendon Property***.

3. Define the tendon profile in ***Load>Prestress Loads>Tendon Profile***. A tendon profile is defined as a curvature relative to an imaginary local x-axis, and the insertion point for the origin of the x-axis and the direction of the x-axis are assigned. The local x-axis may be in the form of a straight line or curved line. A profile already defined can be repeatedly copied, and the origin and direction of the x-axis can be revised to define a number of different tendons.

   A web tendon profile can be created on a vertical plane and projected onto a sloped plane by specifying the angle of inclination to model the tendon placed in an angled web. Tendons can be also placed in sloped elements by simply specifying the slope (gradation) angles.

4. Define pre-stress loads in ***Load>Prestress Loads>Tendon Prestress Loads***. The pre-stress loads can be in the form of either force or stress. The timing of grouting tendons can be also specified to effect the transformed section properties.



Tendon Profile & Pre-stress Load Input

Modeling Functions for Heat of Hydration Analysis

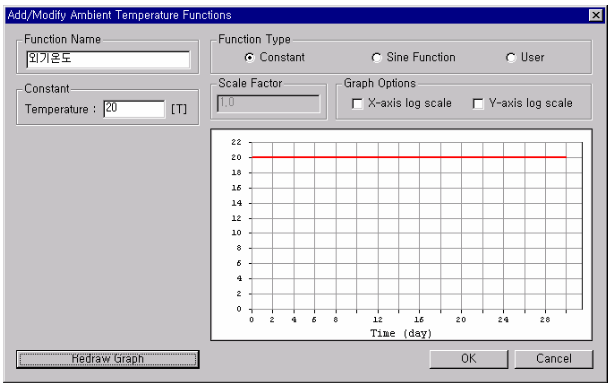
MIDAS/Gen provides Heat of Hydration Analysis capabilities reflecting concrete pour sequence and pipe cooling effects. The modeling procedure for Heat of Hydration Analysis is as follows:

1. Specify the integration factor, initial temperature, stress output location, and whether or not to consider creep & shrinkage in ***Analysis> Hydration Heat Analysis Control***.

2. Specify the ambient temperature function in ***Load>Hydration Heat Analysis Data>Ambient Temperature Functions***.

3. Specify the convection coefficient function in ***Load>Hydration Heat Analysis Data>Convection Coefficient Functions***.

4. Assign the specified ambient temperature and convection boundary condition to the concrete surface in contact with atmosphere in ***Load>Hydration Heat Analysis Data>Element Convection Boundary***.

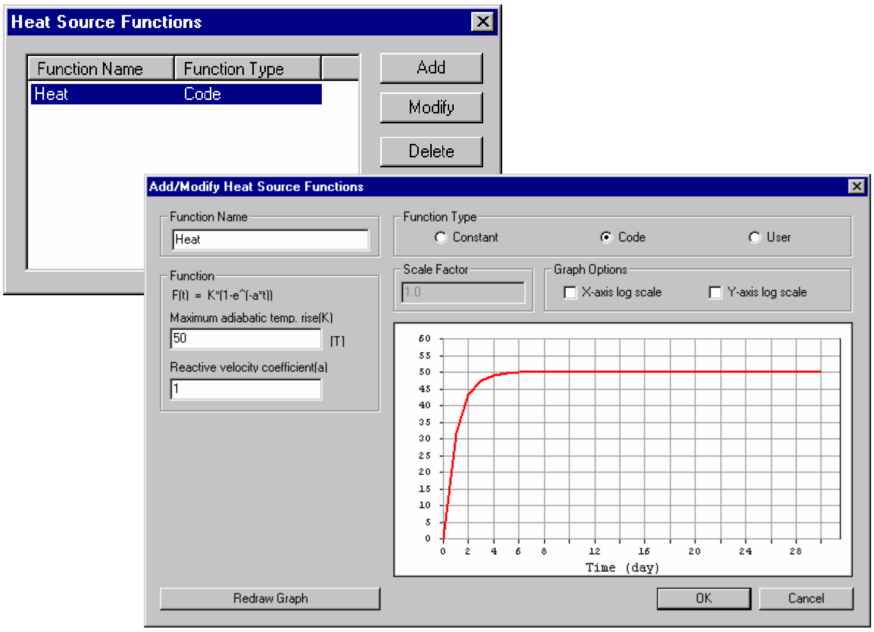


Ambient Temperature Function & Convection Coefficient Function

5. Assign a constant temperature to parts that do not experience temperature variation with time in ***Load>Hydration Heat Analysis Data>Prescribed Temperature***.

6. Define the heat source function, which reflects the state of heat generation representing the process of concrete hydration in ***Load> Hydration Heat Analysis Data>Heat Source Functions***.

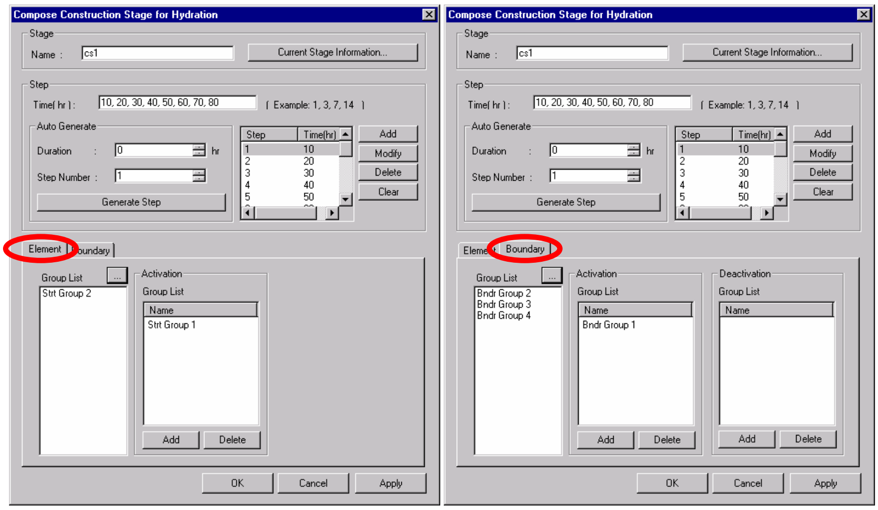
7. Assign the defined heat sources to the corresponding concrete in ***Load>Hydration Heat Analysis Data>Assign Heat Source***.



Heat Source Functions

8. Specify the pipe cooling related data, if used, in ***Load>Hydration Heat Analysis Data> Pipe Cooling***.

9. Define the element groups and boundary groups pertaining to each construction stage, and specify the time for heat of hydration analysis in ***Load>Hydration Heat Analysis Data>Construction Stage for Hydration***.



Construction Stage for Hydration

Other Modeling Functions

A typical structural analysis modeling entails generating nodes and elements, and assigning material properties and boundary conditions. Apart from the typical method of preparing an analysis model, MIDAS/Gen provides the user with various features to efficiently and accurately carry out the structural analysis and design. Some of which are data conversion of other programs, merging several model data and text type data entry.

Non-conventional features of MIDAS/Gen related to modeling are as follows:

- ***Import/Export***
- ***Data Conversion***
- ***Merge Data File Function***
- ***MGT Command Shell***

Import/Export

Use Import/Export when importing model data saved in another format incompatible with MIDAS/Gen or generating a file in another format incompatible with fn.mgb.

Use ***File>Import*** or ***File>Export*** to invoke ***Import/Export***.

- ***MIDAS/Gen MGT File***
  Export a file containing the model data in a text format by creating an MGT (MIDAS/Gen Text) or import an MGT file.

- ***AutoCAD DXF File***
  Export a fn.mgb to a DXF file or import the geometric shape of a model (nodes, elements, etc.) from a DXF file to use it as the model data for MIDAS/Gen.

Refer to File>Import/Export> SAP2000 File of On-line Manual.

- ***SAP2000 File***
  Import a model data file of SAP2000 to use it as a model data file for MIDAS/Gen after converting it into an MGT format. MIDAS/Gen functions not supported by SAP2000 are removed from the model data.

Refer to File> Import/Export>STAAD, MSC.Nastran File of On-line Manual.

- ***STAAD, MSC.Nastran File***
  Import a model data file of STAAD or MSC.Nastran to use it as a model data file for MIDAS/Gen after converting it into an MGT format. MIDAS/Gen functions not supported by STAAD/ MSC.Nastran are removed from the model data.

Data Conversion

Exchange model data and analysis results between **MIDAS/Gen** and **MIDAS/SDS**, which is a floor/mat analysis and design program. Select ***File>Data Conversion***. This feature simplifies the data entry for the analyses of structures such as plants, office, residential buildings or any other types that require analyses of floor plates (slabs & foundations).

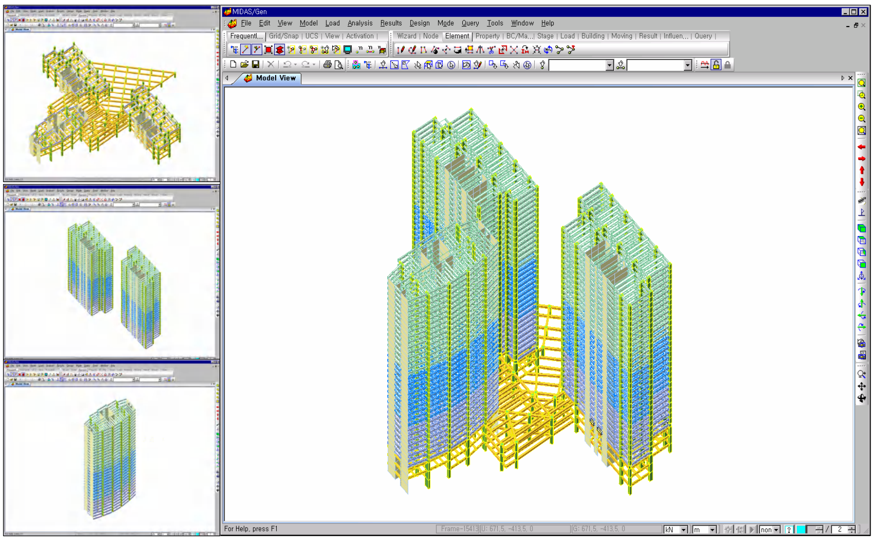
- ***MIDAS/Gen → MIDAS/SDS (Model+Reaction Data)***
  Generate automatically a plate analysis model and corresponding loading data (fn.MST) based on the reactions obtained from the structural analysis of a MIDAS/Gen model, when analyzing foundation mat using MIDAS/SDS.

- ***MIDAS/SDS → MIDAS/Gen (Load Data)***
  Import reactions obtained from the structural analysis of a MIDAS/SDS model by converting into the load data (fn.SA1) for MIDAS/Gen.

When converting analysis results by the above functions, the geometric shapes of the models must be identical for both MIDAS/Gen and MIDAS/SDS.

Merge Data File Function

In order to expedite the modeling task of a complex structure where the geometric configuration is irregular, complicated and large, divide the structure into several sub-models and prepare the geometric shape of each sub-model separately. Then, combine them into a single model and perform the structural analysis. Use ***File>Merge Data File***.

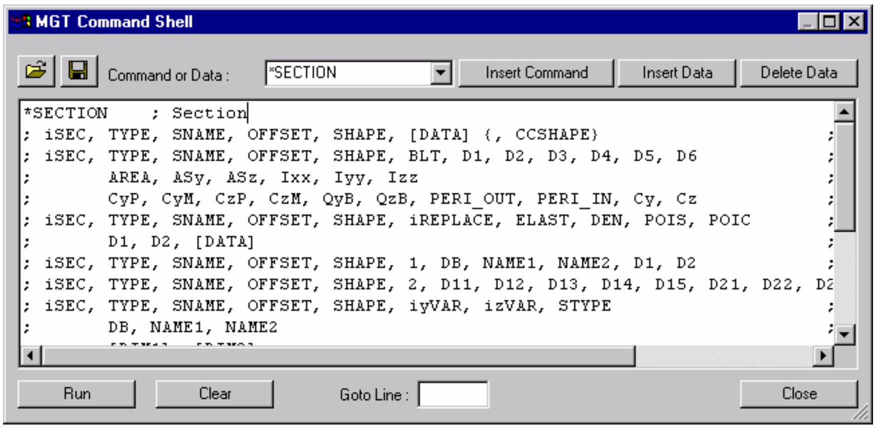


Merge Data File

MGT Command Shell

Enable the modeling of a structure by the MGT format command, which is a text format model data file for MIDAS/Gen.

Use ***Table Window*** or improve the efficiency of modeling by using the MGT command of ***MGT Command Shell*** when the task involves a simple repetition under the GUI environment or the task consists of modifying an existing model continuously.



MGT Command Shell

Input Results Verification

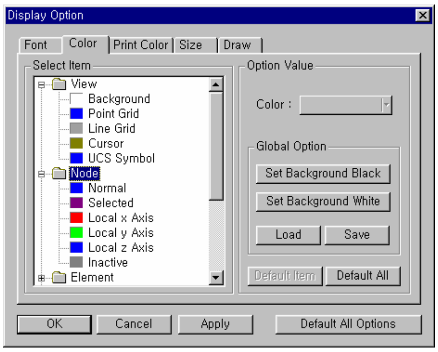
MIDAS/Gen supports a variety of verification and reference functions, which readily verify the current status of all the model data. These functions are:

- ***Display and Display Option***
- ***Project Status***
- ***Fast Query***
- ***Query Nodes***
- ***Query Elements***
- ***Node Detail Table***
- ***Element Detail Table***
- ***Design Parameter Detail Table***
- ***Story Weight Table***
- ***Story Load Table***
- ***Story Mass Table***
- ***Mass Summary Table***
- ***Load Summary Table***
- ***Group Activation of Construction Stage***

Display and Display Option

Display provides graphical representation of all types of data entries such as node/element numbers, material properties, section names, loadings, support conditions, end release conditions, rigid body connection conditions, design parameters, etc. These representation capabilities enable the user to verify the status of data entries by graphics in the working window. For instance, Check & Remove Duplicate Elements and Display Free Edge (Face) are used to detect and correct errors.

Use ***View>Display*** or click  ***Display*** in the Toolbar.



Display dialog box

Display Option controls the representation mode of all the graphic and alphanumerical data presented in the Model Window. It has 5 dialog boxes:

Font tab: Assign the type, size and color of all the alphanumerical type of data such as node numbers, element numbers, analysis results related to nodes and elements, numerical load data, etc.

Color tab: Control the color of all the graphic data such as nodes, elements, masses, loads, support conditions, material properties, sections, thicknesses, grids, coordinate systems, display background, etc.

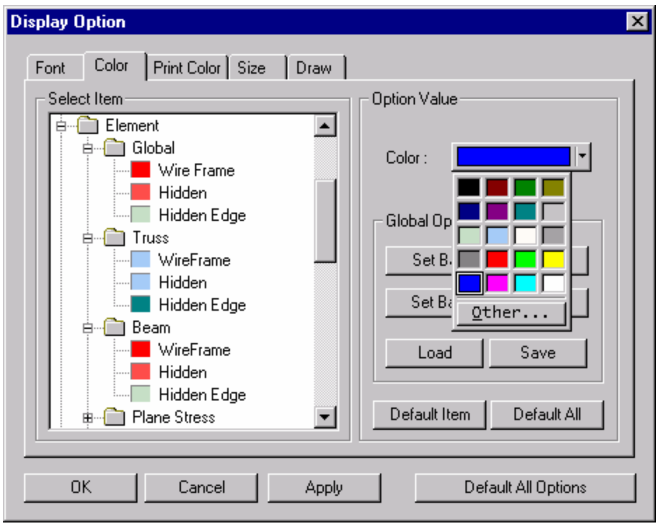
Print Color tab: Control the printing color similarly to ***Color*** tab.

Size tab: Adjust the scale of ***Label Symbol***, ***Zoom In/Out***, ***Pan Rotate, Shrink***, ***Perspective***, etc.

Draw tab: Specify the requirements for element color display on the screen (global element type, material, property, etc.), the representation mode of elements (outline, thickness and surface treatment), the printing color processing method for printouts, the representation method of inactivated elements, the drawing direction for diagrams, etc.

Use ***View>Display Option*** or click ***Display Option***.

MIDAS/Gen provides a Dynamic Display capability, which displays all the nodes and elements, as well as loads and boundary conditions on the model screen as they are being input, which helps prevent modeling errors.

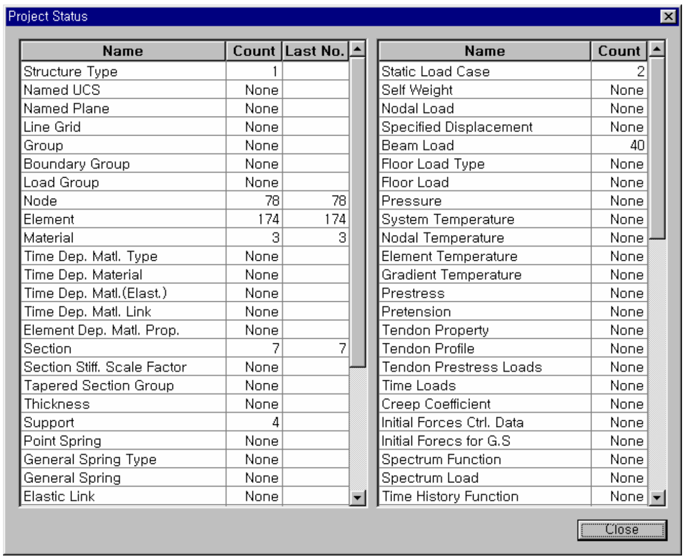


Display Option dialog box

Project Status

Project Status provides the current status of data entries. The data containing the types of data entries with the counts are clearly arranged in a table format.

Use ***Query>Project Status***.



Project Status

Name	Count	Last No.
Structure Type	1	
Named UCS	None	
Named Plane	None	
Line Grid	None	
Group	None	
Boundary Group	None	
Load Group	None	
Node	78	78
Element	174	174
Material	3	3
Time Dep. Matl. Type	None	
Time Dep. Material	None	
Time Dep. Matl.(Elast.)	None	
Time Dep. Matl. Link	None	
Element Dep. Matl. Prop.	None	
Section	7	7
Section Stiff. Scale Factor	None	
Tapered Section Group	None	
Thickness	None	
Support	4	
Point Spring	None	
General Spring Type	None	
General Spring	None	
Elastic Link	None	

Name	Count
Static Load Case	2
Self Weight	None
Nodal Load	None
Specified Displacement	None
Beam Load	40
Floor Load Type	None
Floor Load	None
Pressure	None
System Temperature	None
Nodal Temperature	None
Element Temperature	None
Gradient Temperature	None
Prestress	None
Pretension	None
Tendon Property	None
Tendon Profile	None
Tendon Prestress Loads	None
Time Loads	None
Creep Coefficient	None
Initial Forces Ctrl. Data	None
Initial Forecs for G.S	None
Spectrum Function	None
Spectrum Load	None
Time History Function	None

Close

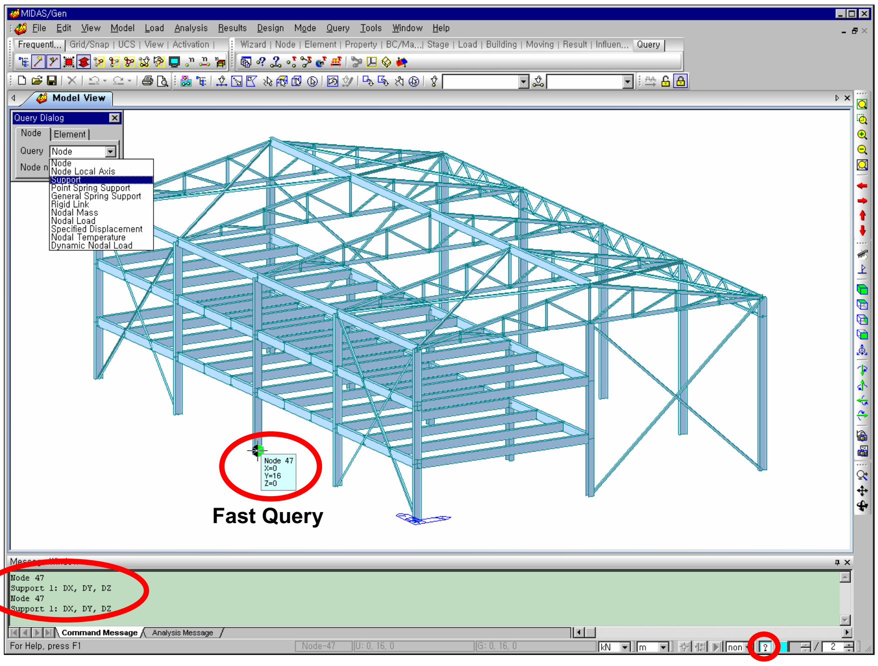
Project Status

Query Nodes

Query Nodes enables the user to verify node numbers, nodal coordinates and nodal attributes. After selecting ***Query>Query Nodes***, assign the node to be verified with a mouse click or by typing the node number in the dialog box. The desired information will appear in the Message Window at the bottom of the screen.

Query Nodes provides the following types of information:

Node (number, coordinates)
Nodal Local Axis
Support
Point Spring Support
General Spring Support
Rigid Link
Nodal Mass
Nodal Load
Specified Displacement
Nodal Temperature
Dynamic Nodal Load



Query Nodes

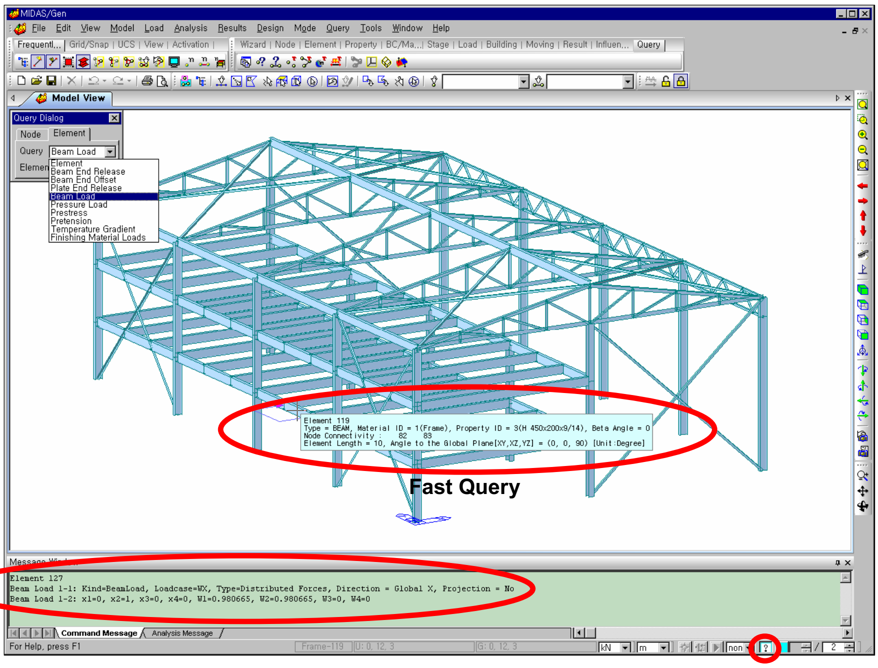
When Fast Query is toggled on, the number and coordinates of the snapped node are displayed in a Bubble Tip. Fast Query can easily verify the basic attributes of nodes and elements.

Query Elements

Query Elements enables the user to verify the element's connecting node numbers and all types of element attributes. After selecting ***Query>Query Elements***, select the element to be verified with a mouse or by typing the element number in the dialog box. The desired information will appear in the Message Window at the lower part of the screen.

Query Elements offers the following types of information:

Element (element, connecting nodes, material properties, section, number, length, etc.)
Beam End Release
Beam End Offset
Plate End Release
Element Beam Load
Pressure Load
Prestress
Pretension
Temperature Gradient



Query Elements

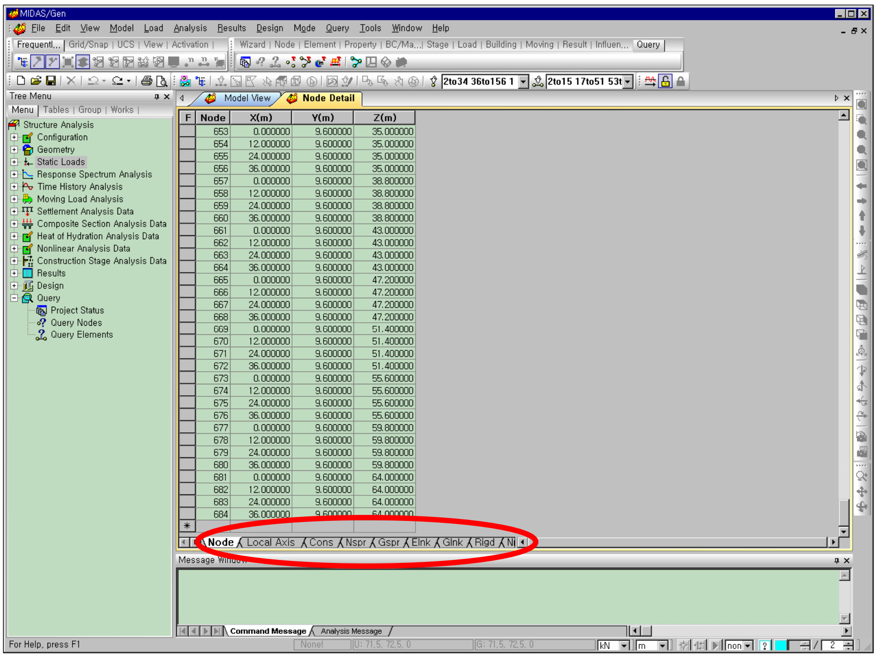
When Fast Query is toggled on, the number, type, material and section properties and other relevant attributes of the snapped element are displayed in a Bubble Tip. Fast Query can easily verify the basic attributes of nodes and elements.

Node Detail Table

Node Detail Table is used to verify all types of information related to nodes in a spreadsheet format.

Select the relevant nodes with ***View>Select*** first. Click ***Query>Node Detail Table*** and select the desired information by clicking the tabs located at the bottom.

Table Window provides all kinds of selection, namely, Filtering, Sorting, Editing, Graph, data transfer with Excel, etc., in addition to data input/output and modification. Refer to On-line Manual for detail directions.



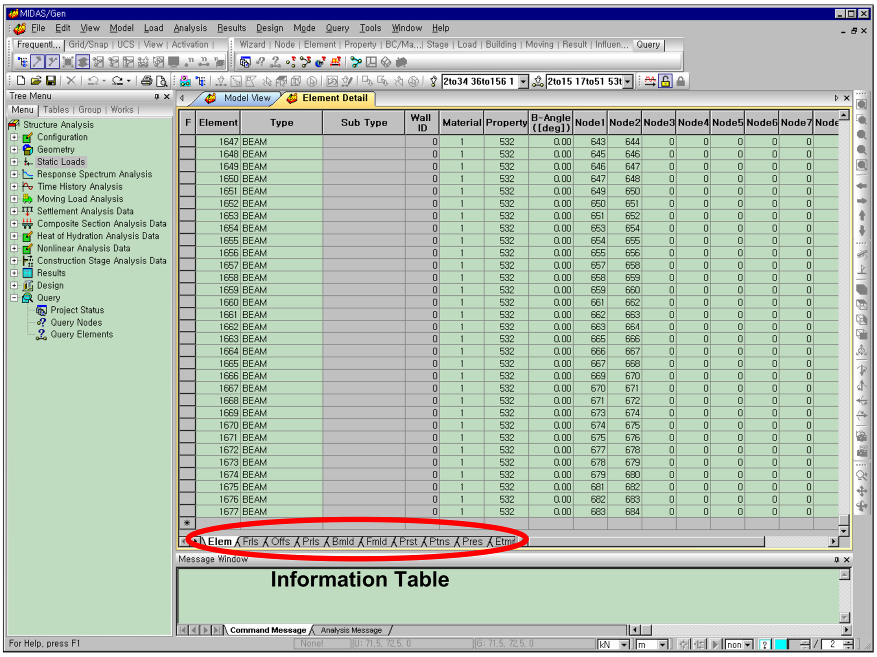
Node Detail Table

Element Detail Table

🎧 Element Detail Table displays only the information related to the selection. It is easy to detect errors such as redundant or duplicated loads.

Element Detail Table is used to verify all types of information related to elements in a spreadsheet format.

Select the relevant elements with ***View>Select*** first. Assign ***Query>Element Detail Table*** and select the desired information by clicking the tabs located at the bottom.

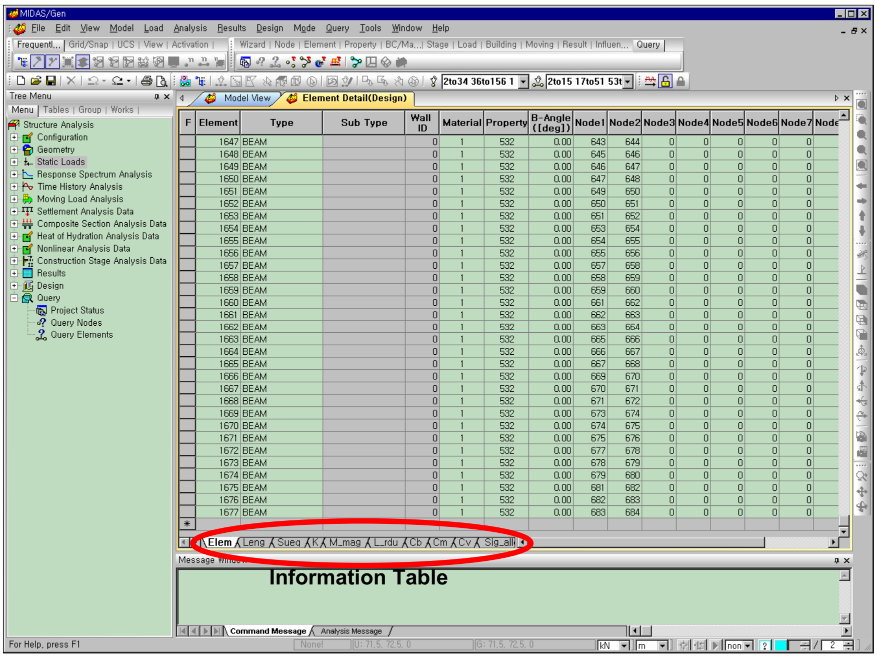


Element Detail Table

Design Parameter Detail Table

Design Parameter Detail Table is used to verify all types of information related to member design in a spreadsheet format.

Select the relevant elements with ***View>Select*** first. Assign ***Query>Design Parameter Detail Table*** and select the desired information by clicking the tab located at the bottom.

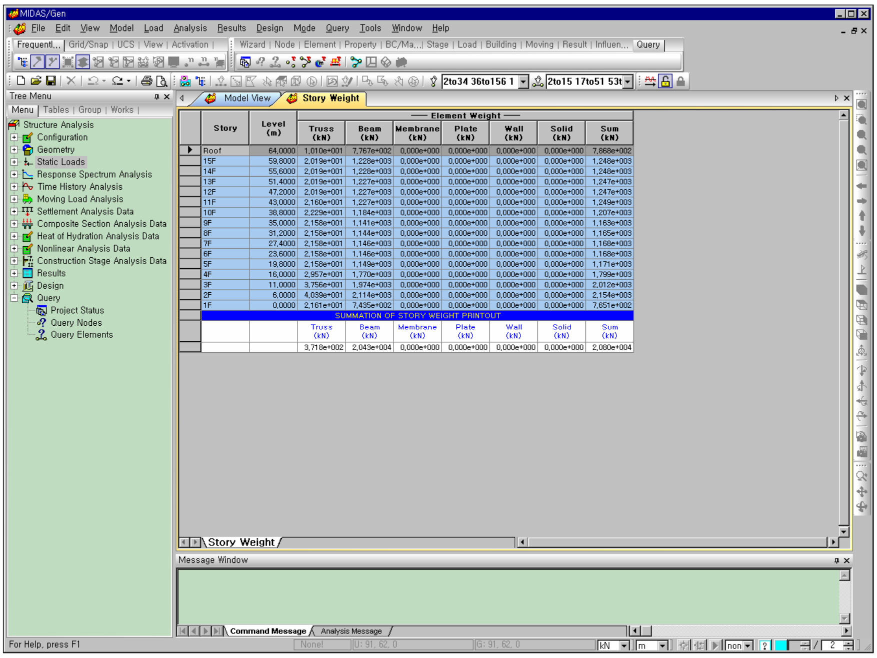


Design Parameter Detail Table

Story Weight Table

Story Weight Table is used to verify the weight of the structure in a spreadsheet format.

Select ***Query>Story Weight Table*** to verify the weights of elements classified by types of elements and by stories.

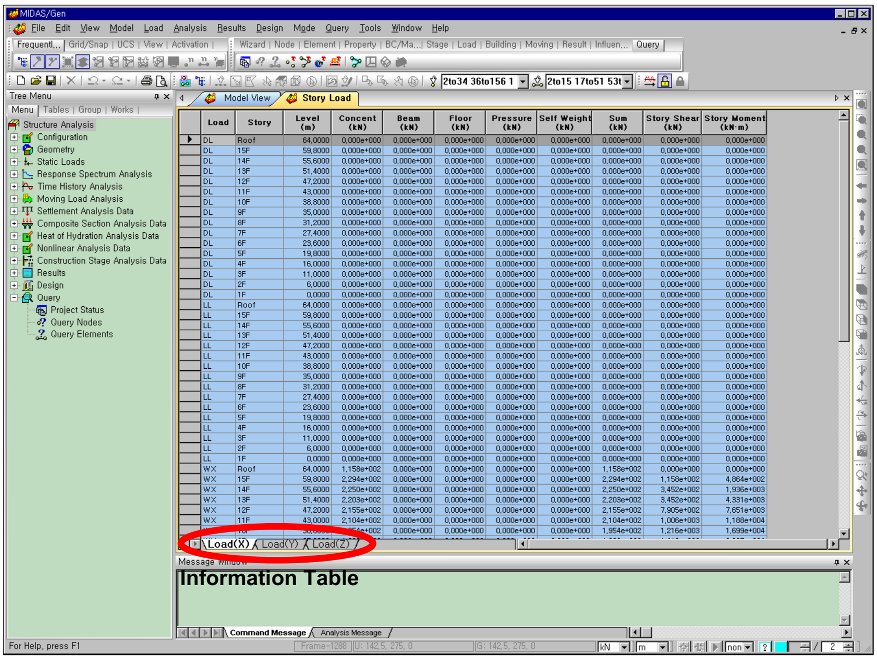


Story Weight Table

Story Load Table

Story Load Table is used to verify the loads applied to the model in a spreadsheet format.

Assign ***Query>Story Weight Table*** first. Select the unit load cases to be included in ***Story Load*** in ***Active Dialog***. Click the tab corresponding to the desired loading direction at the bottom. The total loads for the desired load cases by types of loads and by stories can be verified in ***Story Load Table***.

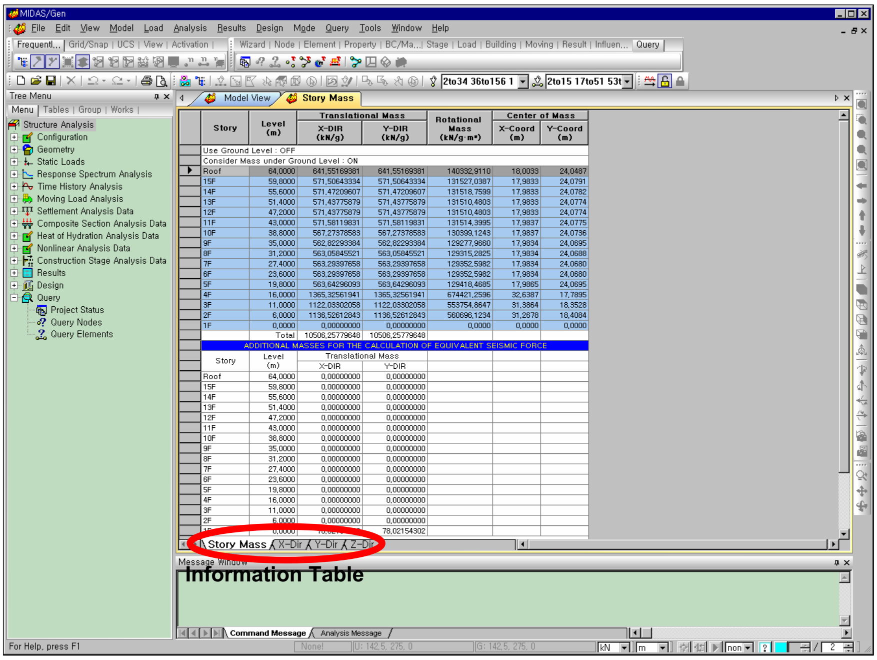


Story Load Table

Story Mass Table

Story Mass Table is used to verify the masses of the structure in a spreadsheet format.

Assign ***Query>Story Mass Table***. The Translational Mass and Rotational Mass at the mass center, for each story can be verified in ***Story Mass Table***.



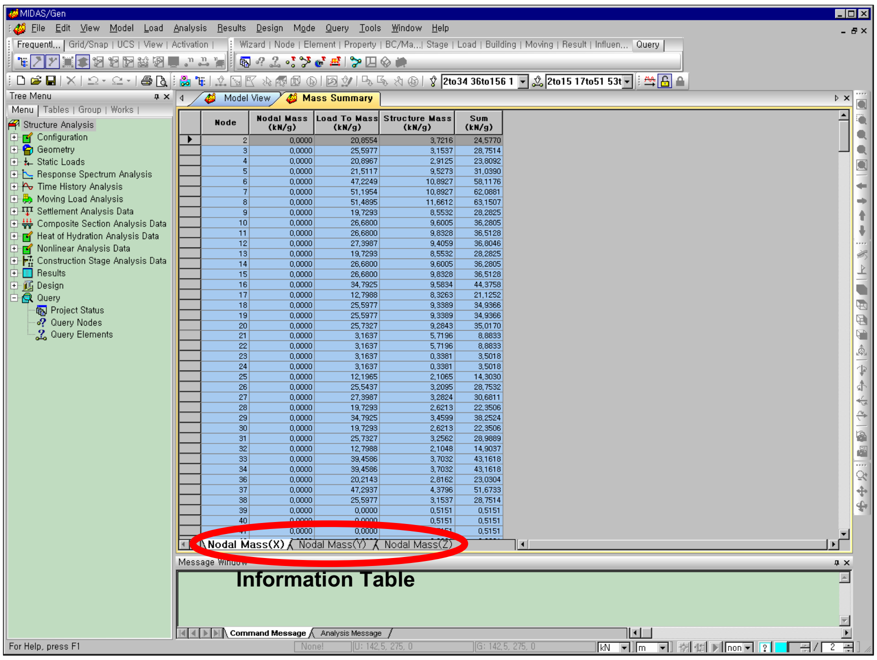
Story Mass Table

Mass Summary Table

🎧 Data cannot be modified in this mode.

Mass Summary Table is used to verify the masses of the structure in a spreadsheet format.

Assign ***Query> Mass Summary Table***. The Nodal Mass that the user entered as such, masses converted from loads and Structure Mass obtained from the self-weight of elements can be verified in ***Mass Summary Table***.

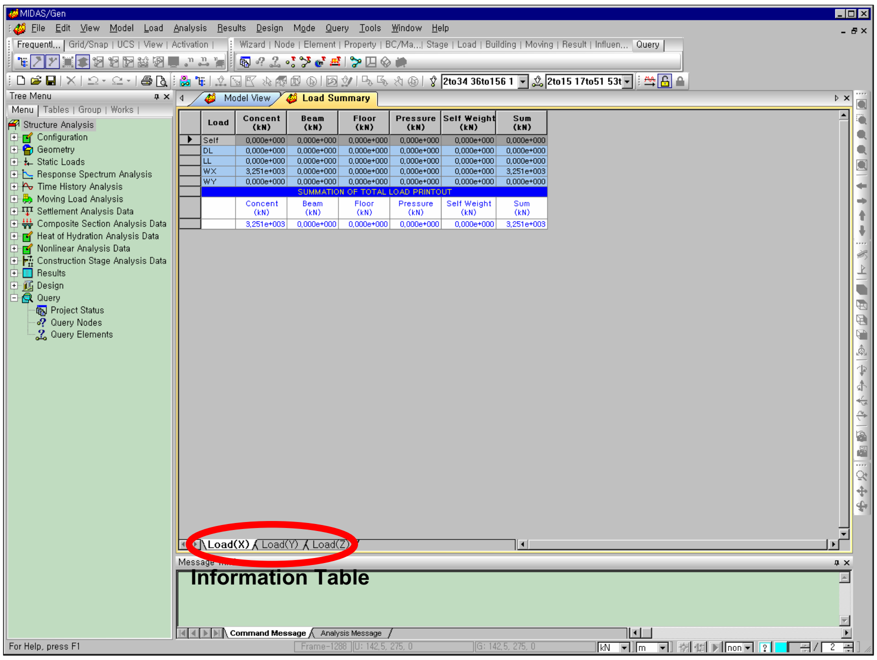


Mass Summary Table

Load Summary Table

Load Summary Table is used to verify the loads that have been input in each direction arranged by load types in a spreadsheet format.

Assign ***Query>Load Summary Table***. Click the tab corresponding to the desired information at the bottom.

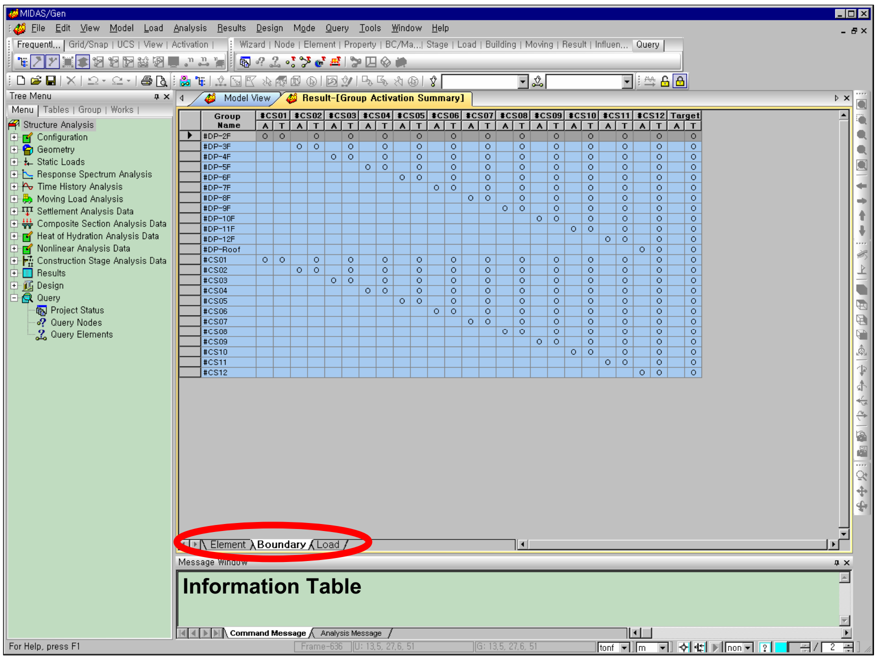


Load Summary Table

Group Activation of Construction Stage Table

Group Activation of Construction Stage is used to check in a table whether or not the groups assigned in each stage of the construction stages are activated.

Selecting ***Query>Group Activation of Construction Stage*** and using the function, click the Group tab at the bottom of the table. The state of activation in the corresponding ***construction*** stages can be checked by the symbols o or x.



Group Activation of Construction Stage Table

Analysis

MIDAS/Gen provides linear and nonlinear structural analysis capabilities.

A large collection of finite elements has been implemented for applications in civil and building structures. The program's efficient analysis algorithms yield exceptional versatility and accurate results appropriate for practical design applications.

There are no limits on the numbers of nodes, elements, load cases and load combinations for a structural model.

Finite Elements

For beam elements, MIDAS/Gen can analyze the displacements and the maximum stresses at the end nodes as well as at intermediate points (***Results>Beam Detail Analysis***).

For plate elements, thin plate (DKT, DKQ) and thick plate (DKMT, DKMQ) elements must be used appropriately. Accurate analysis results can be obtained from thin plates for structures such as common storage tanks. Thick plates may be more appropriate for modeling walls, slabs, bridge decks, basemats, etc.

The Tapered Beam Element formulated from the most current algorithms can precisely simulate the behavior of a hunched beam with varying section dimensions along the length. The Cable Element has also been introduced in MIDAS/Gen for the design of cable-stayed bridges with a small strain condition, and suspended cable structures with geometric nonlinearity including the sagging effect.

The finite element library of **MIDAS/Gen** contains the following: Refer to "***Numerical Analysis Model*** " of the ***Analysis Manual*** for details.

Truss
Transmit only tensile and compressive loads in the element axial direction

Compression-only Truss
Transmit only compressive load in the element axial direction considering a gap distance

Tension-only Truss
Transmit only tensile load in the element axial direction considering a hook distance

Cable
Transmit only tensile load in the element axial direction considering varying stiffness due to the variation of the internal tension and the sag effects

General Prismatic Beam
Common beam element considering 6 degrees of freedom per node

Tapered Beam
Beam element with varying sections along the length considering 6 degrees of freedom per node

Wall
Wall element considering in-plane and out-of-plane bending behaviors

Plane Stress
Plane stress element considering in-plane behaviors

Plate
Plate element considering in-plane and out-of-plane bending behaviors

Stiffened Plate
Anisotropic Plate element considering in-plane and out-of-plane bending behaviors

Plane Strain
Plane strain element considering 2-D behaviors in the GCS X-Z plane

Axisymmetric
Axisymmetric element considering 2-D behaviors in the GCS X-Z plane

Solid

Solid element considering 3 degrees of freedom per node

Visco-elastic Damper

Linear spring and (non) linear viscous damper combined in parallel and connected to a spring linking two nodes in all 6 degrees of freedom. An additional linear viscous damping coefficient for each dof in parallel with the system can be defined.

Hysteretic System

Hysteretic System consists of springs with the Uniaxial Plasticity property in all 6 degrees of freedom. An additional linear viscous damping coefficient for each dof in parallel with the system can be defined.

Lead Rubber Bearing Isolator

Similar to the Hysteretic System, it includes 2 inter-related shear deformation springs with the Biaxial Plasticity property. Independent linear elastic springs represent the remaining 4 degrees of freedom. An additional linear viscous damping coefficient for each dof in parallel with the system can be defined.

Friction Pendulum System Isolator

It includes 2 inter-related shear deformation springs with the Biaxial Plasticity property whose physical movements take the form of a pendulum (pot bearing). The axial deformation spring retains the property of a Gap spring with 0 internal gap. Independent linear elastic springs represent the remaining 3 degrees of freedom. An additional linear viscous damping coefficient for each dof in parallel with the system can be defined.

Analysis

MIDAS/Gen provides three solvers for analysis. Select the analysis method from ***Analysis>Analysis Options***. The default is the Skyline Solver.

The ***Skyline Solver*** is generally used in most structural analysis programs. It can be used in virtually all cases regardless of the types and scales of analysis models or the system capacities. It is an optimized algorithm that can analyze most structural engineering problems within a short time frame.

The ***Band Solver*** is more appropriate for an ABD (Almost Block Diagonal) stiffness matrix and can be used in all cases, similar to the Skyline Solver.

The high performance ***Multi-Frontal Sparse Gaussian Solver*** (MFSGS) is a latest addition to the group of MIDAS solvers. The MFSGS uses an optimum frontal division algorithm to minimize the number of calculations for simultaneous linear equations. The MFSGS is especially useful for those finite elements that contain a large number of degrees of freedom. Structures with many nodes can be solved over 10 times faster depending on the cases. The MFSGS is a particularly useful solver for the detail analysis of a structure consisted of plate and/or solid elements.

The analysis capabilities of **MIDAS/Gen** are as follows: Refer to "***Structural Analysis***" of the ***On-line Manual*** for details.

- ***Static Analysis***
  - Linear Static Analysis
  - Thermal Stress Analysis

- ***Dynamic Analysis***
  - Free Vibration Analysis
  - Response Spectrum Analysis (SRSS, CQC, ABS, Linear)
  - Time History Analysis

- ***Geometric Nonlinear Analysis***
  - P-Delta Analysis
  - Large Displacement Analysis

- ***Boundary Nonlinear Dynamic Analysis***
  - Gap
  - Hook
  - Visco-elastic Damper
  - Hysteretic System
  - Lead Rubber Bearing Isolator
  - Friction Pendulum System Isolator

- ***Buckling Analysis***
  - · Critical Buckling Load Factors
  - · Buckling Modes

- ***Heat Transfer Analysis* (Conduction, Convection, Radiation)**
  - · Steady State Analysis
  - · Time Transient Analysis

- ***Heat of Hydration Analysis***
  - · Thermo-elastic Analysis (Temperature stress)
  - · Maturity, Creep, Shrinkage & Pipe Cooling

- ***Construction Stage Analysis***
  - · Time-dependent Material Properties
  - · Boundary Group
  - · Static Load Group

- ***Pushover Analysis***
  - · Loading Applications as per Mode Shape and Static Load type
  - · Generation of Capacity Spectrums & Demand Spectrums

- ***Other Analysis Features***
  - · Calculation of Unknown Loads using optimization technique
  - · Analysis of steel girders reflecting the section properties before and after composite action

Static Analysis

1. Select ***Load>Static Load Cases*** to enter the load cases.

2. Input the loads using the various static load input options in the ***Load*** menu.

3. When geometric nonlinear elements are included in the model, a) reassign predefined load combinations as load cases in ***Load>Create Load Cases Using Load Combinations*** and b) select ***Analysis>Main Control Data*** to enter the number of iterations and a tolerance necessary for convergence.

4. When the P-Delta effect is considered in the analysis, select ***Analysis> P-Delta Analysis Control*** to enter the number of iterations and a tolerance necessary for convergence. Enter the load cases and load factors for analysis.

All the messages pertaining to the analysis process are compiled automatically in the "fn.out" file.

5. Select ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the analysis. A message indicating the progress of analysis or the completion of analysis is displayed in the Message Window at the lower part of the screen.

6. After completing the analysis, analyze the analysis results using the load cases or combinations and various post-processing functions in ***Results***.

Heat of Hydration Analysis

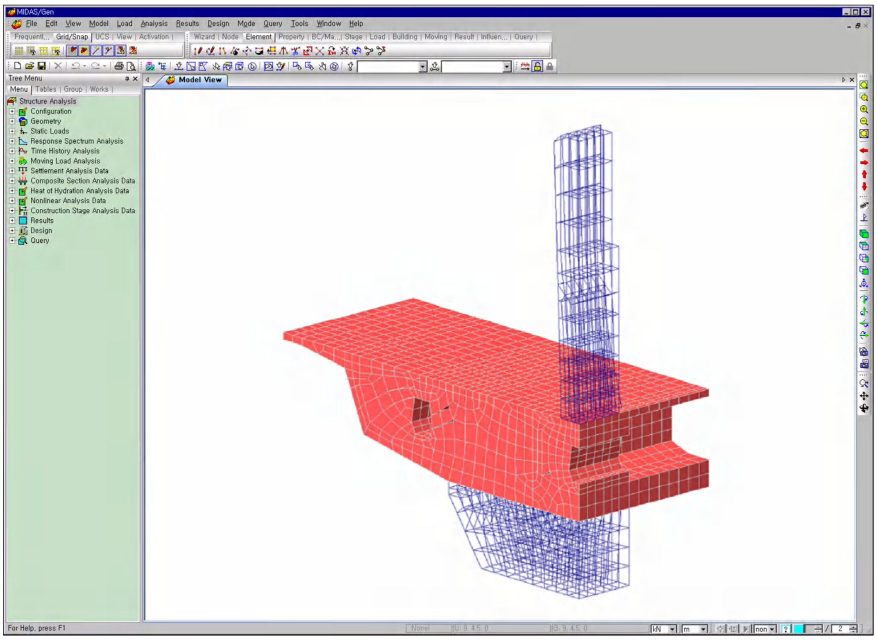
1. Enter the time dependent material properties in ***Model>Properties> Time Dependent Material (Creep/Shrinkage)*** and ***Model>Properties> Time Dependent Material (Comp. Strength)***, and relate the general material properties to the time dependent material properties in ***Model> Properties>Time Dependent Material Link***.

2. Enter the data required for heat of hydration analysis in the sub-menu of ***Load>Hydration Heat Analysis Data*** following the procedure outlined in "Modeling Functions for Heat of Hydration Analysis".

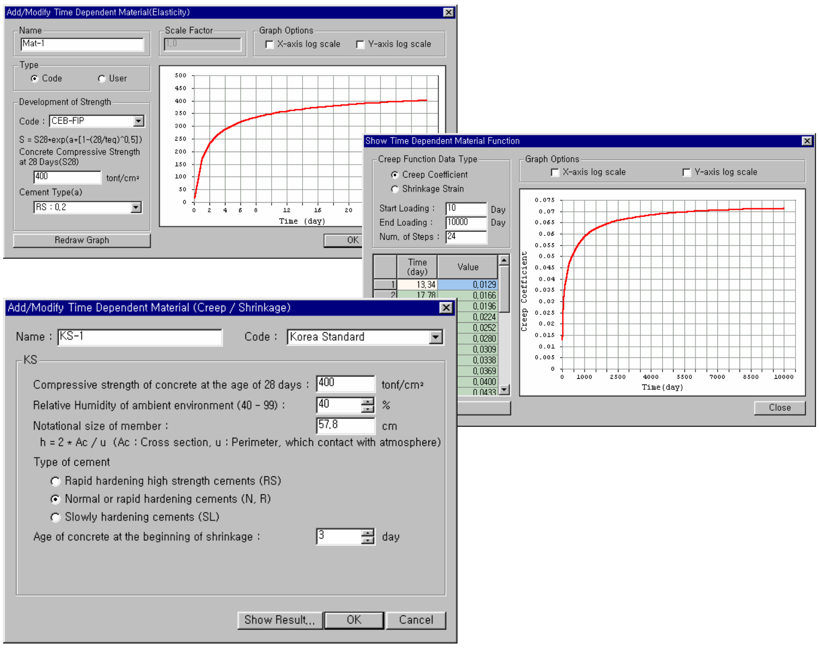
3. Enter the integration factor, initial temperature, stress output position and whether or not to consider creep & shrinkage in ***Analysis> Hydration Heat Analysis Control***.

4. Carry out the analysis in the ***Analysis>Perform Analysis*** menu or by clicking ***Perform Analysis***.

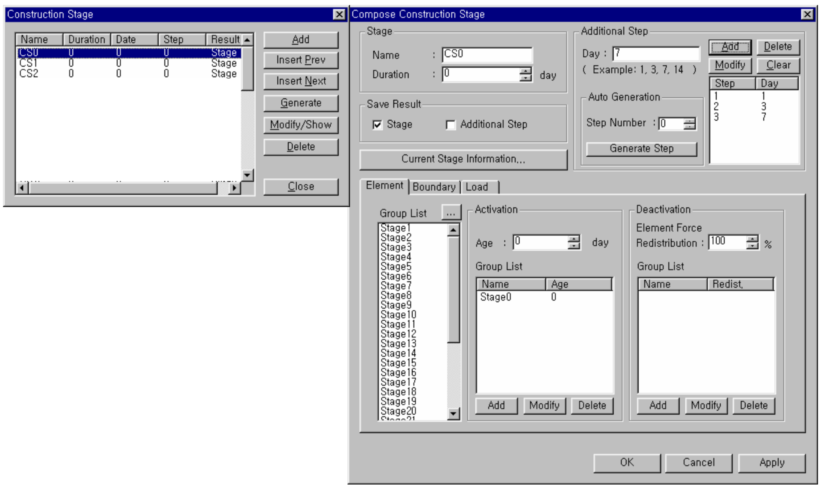
5. Once the analysis is completed, the results can be verified in contours, graphs, animations, etc.



Heat of Hydration analysis model of a bridge pier cap cast in sequence

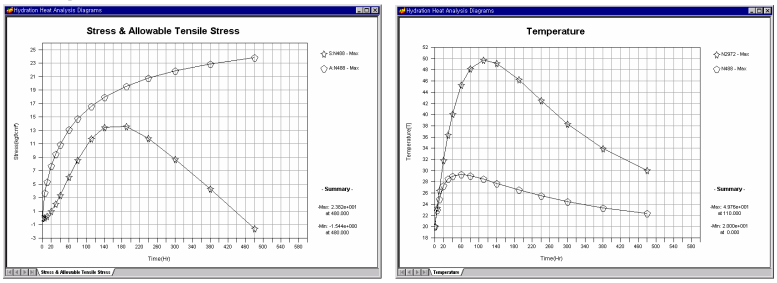


Dialog boxes defining Heat & Time dependent material properties

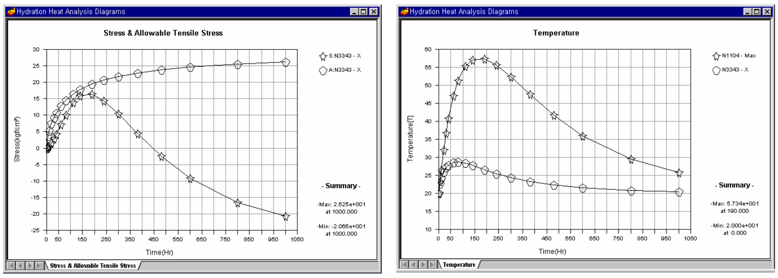


Construction Stage dialog box defining sequential construction joints
(Define Elements & boundary conditions for each construction stage)

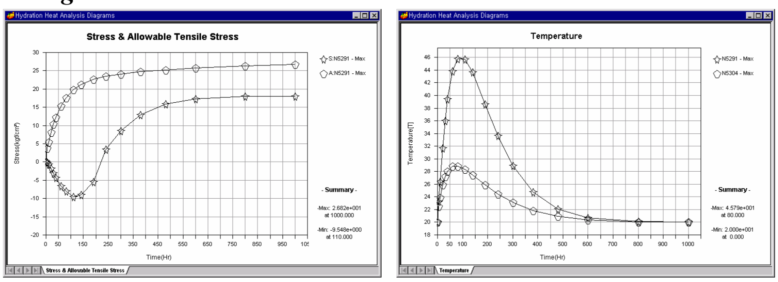
1st Stage



2nd Stage



3rd Stage



Analysis results for each construction stage in graphs

Eigenvalue Analysis

1. Enter the masses of the model using the mass input tools supplied by ***Model>Masses***.

Using Model>Masses> Loads to Masses, the desired loading condition of the static load data can be converted to nodal masses.
This function is extremely useful for a seismic analysis where dead load is to be converted into mass.

2. Select ***Analysis>Eigenvalue Analysis Control*** to enter the data necessary for eigenvalue analysis such as the number of modes.

3. Select ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the analysis.

4. After completing the analysis, verify the vibration mode shapes and natural frequencies (or natural periods) for each mode using ***Results>Vibration Mode Shapes*** or ***Results>Result Tables>Vibration Mode Shape***.

Response Spectrum Analysis

1. Follow the steps 1 and 2 of Eigenvalue Analysis.

2. Select ***Load>Response Spectrum Analysis Data>Define Response Spectrum Functions*** and click Add. Enter the function name and related spectrum function data in the ***Add/Modify Show Response Spectrum Functions*** dialog box.

It is convenient to use the built-in design response spectra to specify Spectrum Function. The built-in design response spectra are as follows:
· UBC 88-94
· UBC 97

3. Use ***Load>Response Spectrum Analysis Data>Response Spectrum Load Cases*** to enter the ***Load Case Name***. Then, select the function name from the ***Function Name List*** and enter the remaining data.

4. Select ***Analysis>Response Spectrum Analysis Control*** to assign the ***Modal Combination Type*** and to specify the condition for the restoration of signs.

MIDAS/Gen can restore the signs of the analysis results that have been combined by SRSS or CQC method. The results with the restored signs can then be used for foundation design and other member design sensitive to proper signs.

5. Use ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the analysis.

6. Use the post-processing functions of ***Results*** to analyze or combine the analysis results.

Time History Analysis

1. Follow the steps 1 and 2 of Eigenvalue Analysis.

2. Select ***Load>Time History Analysis Data>Time Forcing Functions*** and click Add Time Function or Add Sinusoidal to enter the data pertaining to Time Forcing Function related to Function Names in the dialog box.

3. Select ***Load>Time History Analysis Data>Time History Load Cases*** to enter the Load Case Name, the Damping Ratio and the data required for the time history analysis process and the output.

4. When dynamic nodal loads are entered as ***Time Forcing Function***, use ***Load> Time History Analysis Data>Dynamic Nodal Loads*** to select the Load Case Name and Function Name from the Function Name List, and then enter the loading direction and arrival time.

   When ground motion is used as ***Time Forcing Function***, use ***Load>Time History Analysis Data>Assign Ground Acceleration*** to select the Load Case Name and Function Name from the Function Name List, and then click Add in ***Operations***.

5. Select ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the analysis.

6. Use the post-processing functions of Results to analyze or combine the time history and static analysis results. The absolute maximum values within the given time history are provided for all analysis results. Use ***Results>Time History Results*** to analyze the results at each time step. The history graphs and text type results may be produced.

Dynamic Boundary Nonlinear Analysis

1. Enter the properties of nonlinear link elements in the ***Model>Boundaries>Nonlinear Link Properties*** menu.

2. Define the nonlinear link elements in the model using ***Model>Boundaries>Nonlinear Link***.

3. Enter the mass data.

4. Define the dynamic loads in the ***Load>Time History Analysis Data>Time History Functions*** dialog box.

5. Enter the time history analysis conditions and various control data required to perform time history analysis in ***Load>Time History Analysis Data>Time History Load Cases***.

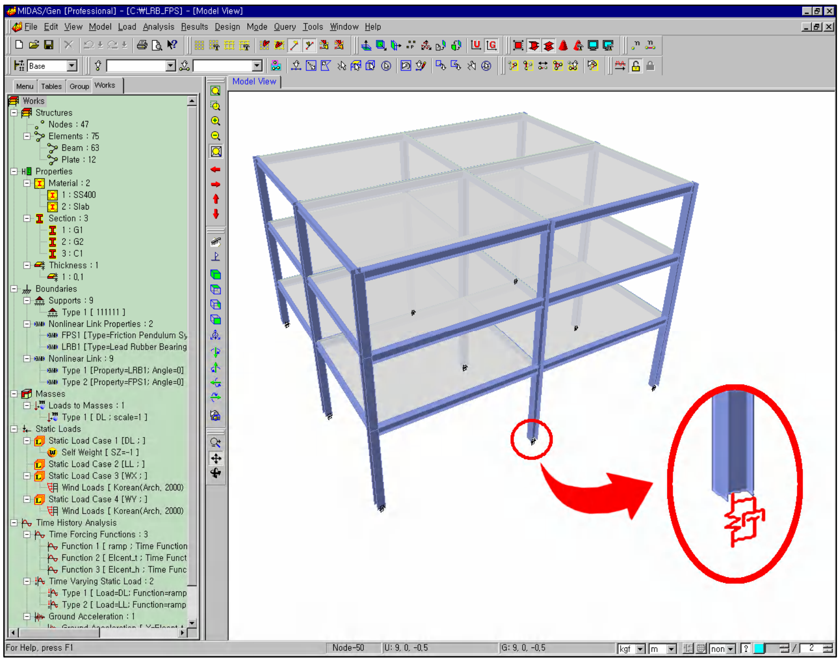
6. Enter the time load functions in the form of ground acceleration in ***Load>Time History Analysis Data>Ground Acceleration***.

7. Convert pertinent static loads into dynamic loads by multiplying the previously defined static loads by time functions in ***Load>Time History Analysis Data>Time Varying Static Loads***.

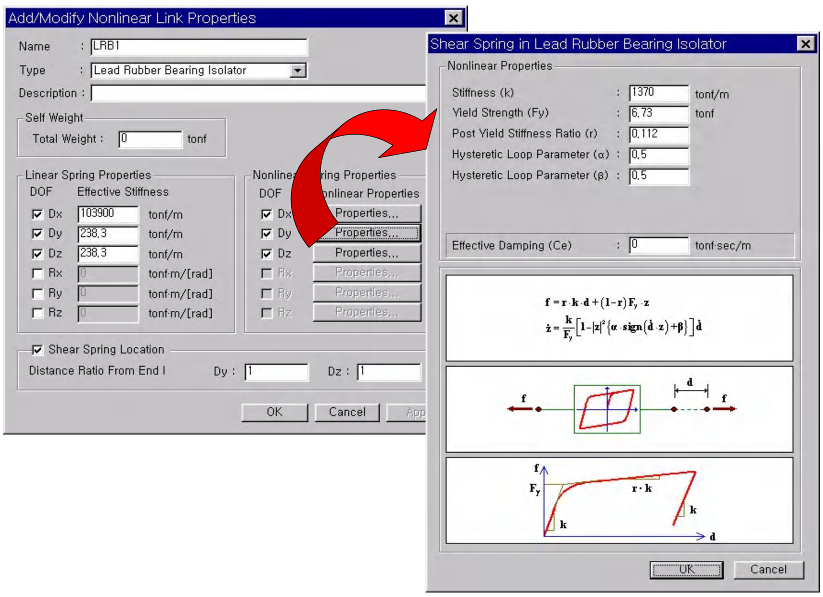
8. Enter the control data required to perform eigenvalue analysis in ***Analysis>Eigenvalue Analysis Control***.

9. Carry out the analysis in the ***Analysis>Perform Analysis*** menu or by clicking ***Perform Analysis***.

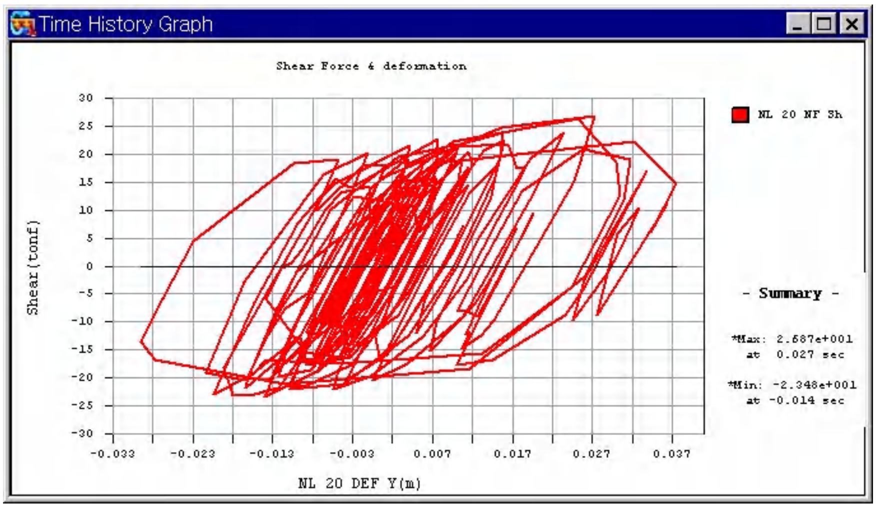
10. Upon completing the analysis successfully, we can check the displacements and max/min member forces for the Time History load cases. We can also check the time history analysis results in ***Results>Time History Graph***.



Dynamic boundary nonlinear analysis model of a structure connected to nonlinear link elements



Dialog boxes for entering nonlinear properties of bearing isolators



Shear force-Deformation graph of bearing isolator obtained from dynamic boundary nonlinear analysis

Buckling Analysis

1. Static analysis results are required to provide the initial geometric stiffness matrix for the buckling analysis of a structure. Thus, the load cases for the buckling analysis must be specified first to analyze the buckling modes. Follow the procedure presented in Static Analysis above.

2. Invoke the dialog box of ***Analysis>Buckling Analysis Control*** to enter the number of modes and the data necessary for convergence. Assign the load cases to be considered in the buckling mode analysis.

3. Use ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the buckling analysis.

4. Use ***Results>Buckling Mode Shapes*** or ***Results>Result Tables> Buckling Mode Shape*** to verify the buckling mode shapes and the critical buckling load factors for each mode.

P-Delta Effect Analysis

When considering the P-Delta effect in the static analysis and dynamic analysis processes, use ***Analysis>P-Delta Analysis Control*** to assign the load cases to be considered for the formation of the geometric stiffness matrix. In addition, enter the number of iterations and the tolerance for convergence. **MIDAS/Gen** only performs P-Delta effect analysis for structures modeled with truss, beam and wall elements.

Geometric Nonlinear (Large Displacement) Analysis

The Geometric nonlinear analysis function is applicable for static analysis and construction stage analysis. Prior to the analysis, assign the order of applying the loads to be used for the analysis in ***Load>Nonlinear Analysis Data>Loading Sequence for Nonlinear Analysis***, followed by assigning the repetitive analysis and convergence conditions required to carry out the nonlinear analysis in ***Analysis>Nonlinear Analysis Control***.

Geometric nonlinear analysis is applicable for all the elements except for the solid element.

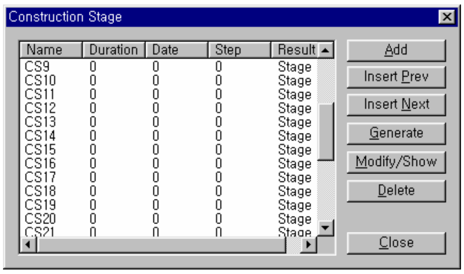
Construction Stage Analysis

1. Use the dialog box of the ***Analysis>Construction Stage Analysis Control*** menu when a construction stage analysis is sought for calculating vertical deformations due to the creep and shrinkage of concrete. Assign the time dependent material property types and specify the number of iteration and convergence condition required for creep calculation.

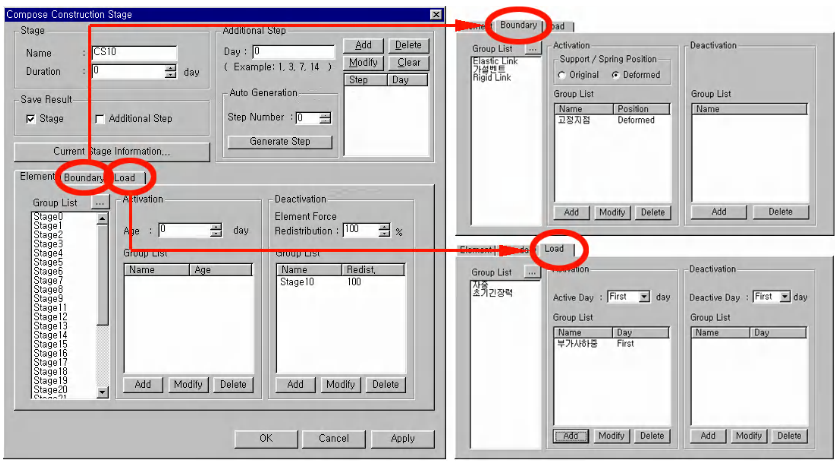
2. Use ***Load>Construction Stage Analysis Data>Construction Stage Wizard for Building Structures***, or define the construction stages including boundary and load conditions in the ***Define Construction Stage menu***.

3. Select ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the construction stage analysis.

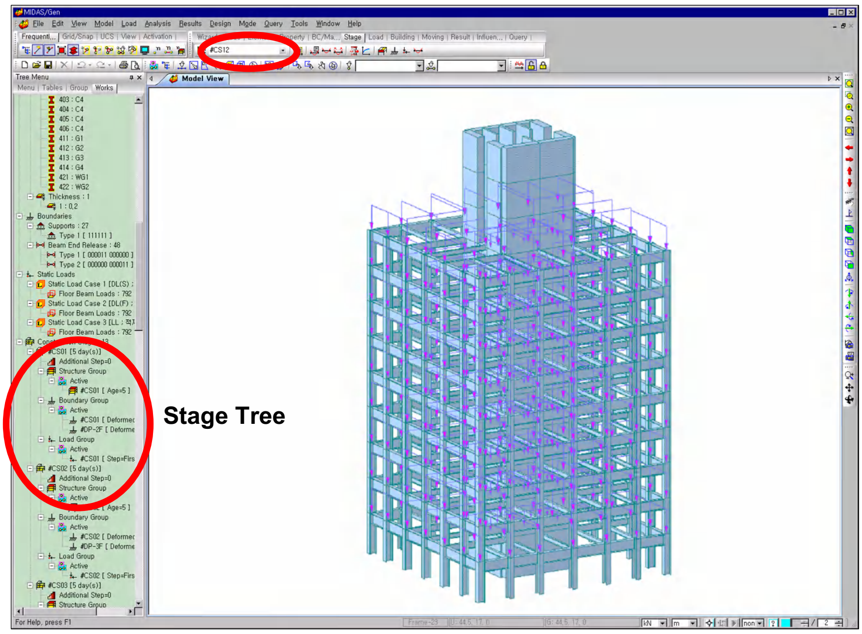
4. Once the analysis is successfully completed, we can verify displacements, member forces, stresses, etc. for each construction stage as well as the final construction stage in the ***Results*** menu.



Simply specifying durations sequentially creates construction stages. New stages may be inserted, or the previously defined stages may be deleted.



Simple activation and deactivation of element, boundary and load groups compose the construction stages.



Real time display of a stage

Pushover Analysis

1. Specify the maximum numbers of Iterations/Increment steps and convergence tolerance in ***Design>Pushover Analysis Control***.

2. Define the Pushover load case and initial load in ***Design>Pushover Load Cases***.

3. Define the plastic hinge properties, which are to be applied to the model in ***Design>Define Hinge Data Type***.

4. Assign the defined hinge properties to each member in ***Design>Assign Hinge Data***.

5. Select ***Design>Perform Pushover Analysis*** to perform the pushover analysis.

6. Verify Performance points using the Capacity spectrum and Demand spectrums obtained from ***Design>Pushover Curve***

Composite Steel Beam Analysis considering Variation of Pre- and Post-Composite Section Properties

1. Use ***Load>Static Load Cases*** to define the load cases and the loads applied to the pre-composite sections.

2. Use ***Load>Composite Section Analysis Data>Pre-Combined Load Cases for Composite Section*** to assign the load cases applied to the pre-composite sections for the analysis.

3. Select ***Analysis>Perform Analysis*** or click ***Perform Analysis*** to perform the analysis.

4. Use the post-processing functions of ***Results*** to combine or analyze the analysis results.

Interpretation of Analysis Results

Mode Switching

Notice that the analysis results are removed when the modeling data are modified in the preprocessing mode after completing the analysis. However, the design data can be modified.

MIDAS/Gen organizes the operating environment of the program by Preprocessing Mode and Post-processing Mode for user convenience and efficiency.

All the data-entering tasks for modeling are possible only in the preprocessing mode. On the other hand, interpretation of analysis results such as combining loads, reactions, displacements, member forces and stresses is carried out in the post-processing mode.

If the analysis is completed successfully without errors, the preprocessing mode is switched automatically to the post-processing mode.

Load Combinations and Maximum/Minimum Values Extraction

Combining Analysis Results

MIDAS/Gen can combine all the results obtained from static, response spectrum, time history, heat of hydration, nonlinear and construction stage analyses by means of the ***Results>Combinations*** function. The combined results can be expressed in text or graph formats in each post-processing mode. Also, combining the load combination cases can create new load cases.

The following 4 methods are used to enter load combination data in **MIDAS/Gen**:

1. The user directly specifies the load combination data.

2. Load combinations are auto-generated by selecting one of the built-in design standards.

3. Modifying the auto-generated load combinations in a spreadsheet format can also specify load combinations.

4. A file, which already contains the required load combinations, is imported.

MIDAS/Gen automatically generates pertinent load combinations pursuant to the following design standards:

- ***Structural Steel Design Standards***

  American Institute of Steel Construction, LRFD
  (AISC-LRFD93 & 2000 Load & Resistance Factor Design Part 6 Specifications and Codes, 1993 & 2000)

  American Institute of Steel Construction, ASD
  (AISC-ASD89, Specification for Structural Steel Buildings: Allowable Stress Design, Part 5. Specifications and Codes, 1989)

  British Standard, Structural use of steel work in building
  (BS5950-90, Part 1. Code of practice for design in simple and continuous construction)

  ENV 1993-1-1 Eurocode3, Design of Steel Structures
  (Eurocode3, Part 1.1 General Rules and Rules for Building)

  Canadian Standards Association, Limit States Design of Steel Structures
  (CSA-S16-01)

  American Iron and Steel Institute, Cold-Formed Steel Design
  (AISI-CFSD86)

  TWN-ASD90, Taiwan Standard, Allowable Stress Design Specification and Commentary for Structural Steel Building, 2001

TWN-LSD90, Taiwan Standard, Limit States Design Specification and Commentary for Structural Steel Building, 2001

IS:800-1984, Indian Standard, Code of Practice for General Construction in Steel (Second Revision), 1984

- ***Reinforced Concrete Design Standards***

American Concrete Institute, Building Code Requirements for Structural Concrete and Commentary
(ACI318- 02/89/95/99)

Canadian Standards Association of Concrete Structures (CSA-A23.3-94)

British Standard, Structural use of concrete
(BS8110-97, Part 1. Code of practice for design and construction)

ENV 1992-1-1 Eurocode2, Design of concrete structures
(Eurocode2, Part 1. General Rules and Rules for Building)

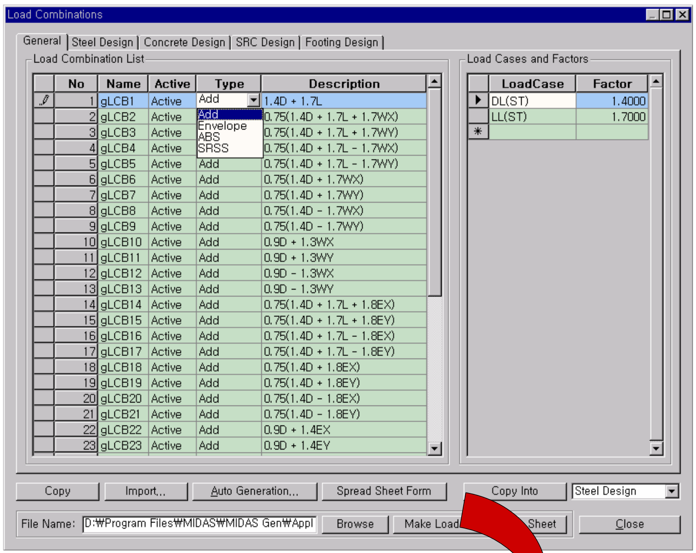
TWN-USD92, Taiwan Standard, Design Specification and Commentary for Concrete Structures, 2003

IS456:2000, Indian Standard, Plain and Reinforced Concrete Code of Practice (Fourth Revision), 2000

- ***Steel – Reinforced Concrete Composite Design Standards***

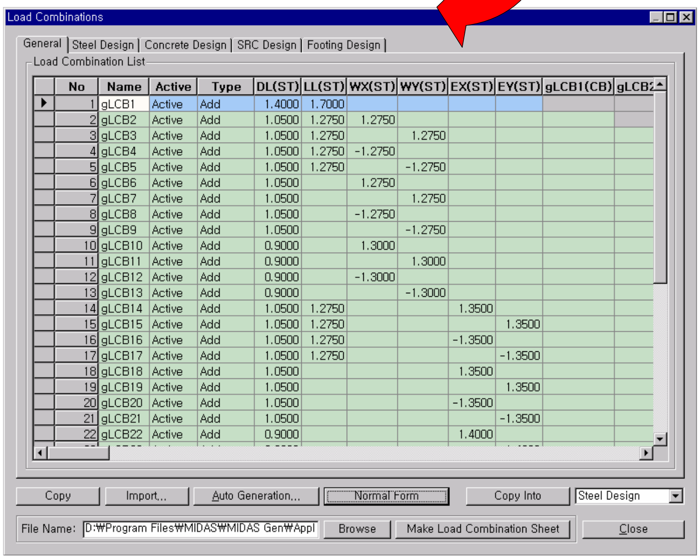
Structural Stability Research Council, A Specification for the Design of Steel – Concrete Composite Columns, 1979
(SSRC, 1979)

TWN-SRC92, Taiwan Standard, Design Specification and Commentary for Steel Reinforced Concrete Structures, 2003



By using Active Option, the load combination conditions can be applied or ignored.

Auto-generation of load combinations



The user may find it more convenient to modify the auto -generated load combinations.

Modification of load combinations - Spreadsheet Form

MIDAS/Gen offers the following 5 types of load combinations for design convenience:

General
Use ***General*** to combine load cases to assess the serviceability or evaluate the analysis results without reference to a specific design standard.

Steel Design
Use ***Steel Design*** to design steel frame members with respect to the steel frame design standards.

Concrete Design
Use ***Concrete Design*** to design reinforced concrete members with respect to the RC design standards.

SRC Design
Use ***SRC Design*** to design steel frame-reinforced concrete composite members.

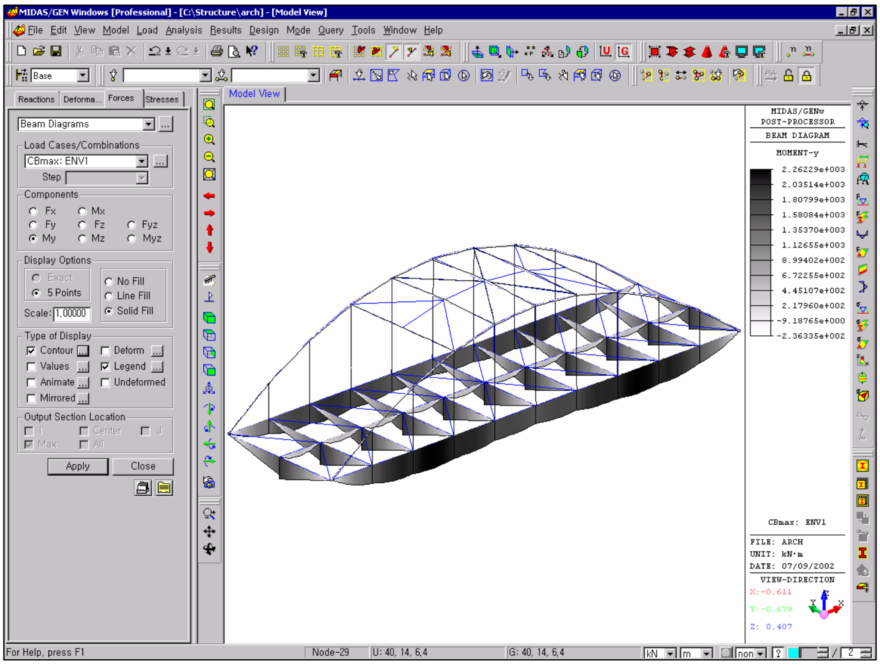
Footing Design
Use ***Footing Design*** to design spread footings and pile foundations.

The user can either apply or ignore the load combinations during the design process.

Extracting Maximum/Minimum Values

By grouping several unit load cases, **MIDAS/Gen** can extract the maximum and minimum values of structural analysis results such as displacements, reactions, member forces, stresses, etc., using ***Envelope*** Type.

The results produced by using ***Envelope*** Type as a load combination can be produced in graph or text formats in each post-processing mode.



Arch bridge BMD: Envelope max

Analysis Results Verification

The post-processing mode of **MIDAS/Gen** provides analysis results in graph or text formats for simple verification.

Result supports the post-processing mode of **MIDAS/Gen**. The sub-menu types are as follows:

Combinations
Generate the load combinations

Reactions
Reaction Forces/Moments: reaction diagrams for supports
Search Reaction Forces/Moments: verification of reaction forces at a specific support

Deformations
Deformed Shape: deformed shape of the model
Displacement Contour: displacement contour diagrams
Search Displacements: verification of displacements at a specific node

Forces

Truss Forces: member force contour diagrams for truss elements
Beam Forces/Moments: member force contour diagrams for beam elements
Beam Diagrams: member force diagrams for beam elements
Plate Forces/Moments: element force contour diagrams for plate elements
Wall Forces/Moments: element force contour diagrams for wall elements
Wall Diagrams: member force diagrams for wall elements

Stresses

Truss Stresses: stress contour diagrams for truss elements
Beam Stresses: stress contour diagrams for beam elements
Plane Stress/Plate Stresses: stress contour diagrams for plane stress elements and plate elements
Plane Strain Stresses: stress contour diagrams for plane strain elements
Axisymmetric Stresses: stress contour diagrams for axisymmetric elements
Solid Stresses: stress contour diagrams for solid elements

Heat of Hydration Analysis

Heat of Hydration analysis results including stresses, temperatures, displacements, allowable tension stress, crack ratios and time history graphs

Beam Detail Analysis

Detail displacement, shear force/bending moment and maximum section stress distribution diagrams for a beam element

Element Detail Results

Member forces and stresses of elements for individual load cases or load combinations

Local Direction Force Sum

Compute the resultant forces of plate or solid elements by summing up their nodal forces in a particular direction

Vibration Mode Shapes

Natural frequencies and eigenvalue modes

Buckling Mode Shapes

Critical buckling factors and buckling modes

Time History Results
Time History Graph and Time History Text for analysis results

Stage/Step History Graph
Graphs of analysis results for Construction stage, Geometric nonlinear or Heat of hydration analysis

The toolbars for analyzing analysis results can be recalled into the screen by Customize in Tools> Customize>Tool bars.

Unknown Load Factor
Supply the design load factors satisfying the specified reactions, displacements, member forces of truss and beam elements, etc.

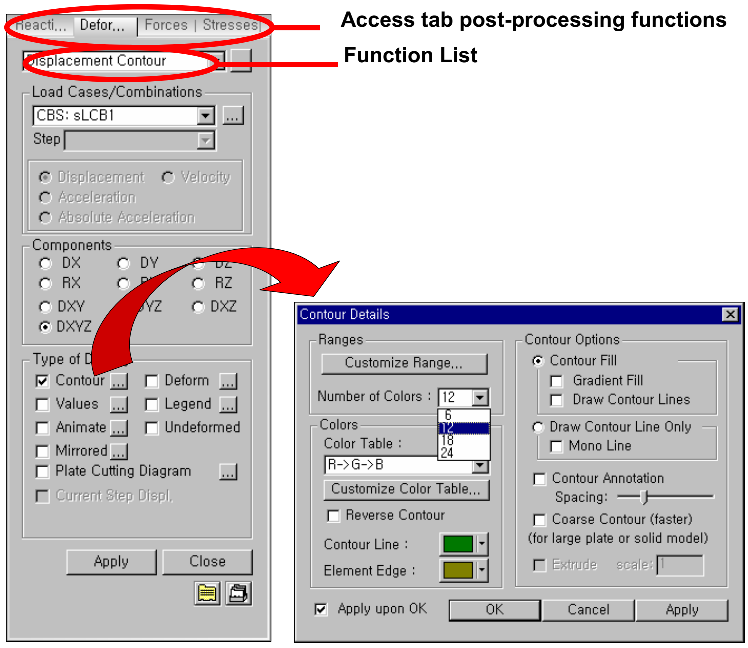
Result Tables
Supply spreadsheet tables containing the analysis results such as reactions, displacements, member forces, stresses, eigenvalue modes, story displacements, story shear forces, etc.

Text Output
Supply a text output file containing the analysis results such as reactions, displacements, member forces, etc. arranged by the load combinations and output contents chosen by the user.

Post-Processing Procedure

The general operating procedure related to the post-processing of **MIDAS/Gen** is as follows:

1. Click ***Post-processing Mode*** to switch to the post-processing environment.
2. Use ***Results*** or the icons in the toolbars to recall the desired post-processing function.
3. Select the desired load case or combination when the dialog bar appears on the left of the screen. Click the ... button located to the right of the load cases/combination selection list to enter a new load combination.
4. Use the ***Components*** field to assign the desired displacement, member force or stress component.
5. Use ***Type of Display*** to assign the contour, deformed shape, numerical values, etc. Click ... to the right of the relevant selection field to change the details of the display if necessary.



Dialog bar of the post-processing and dialog box for the control of screen display

6. For selectively displaying a part of the entire model, use ***View>Select*** to select the entities, and use ***View>Activities>Active*** to activate the entities. The selection feature can be used at any time since it is independent of the post-processing.

7. Click Apply to display the post-processing results, reflecting the conditions assigned in the above procedure.

8. When accessing another post-processing function, it is more convenient to use the Icon menu, the function list, or the post-processing tabs of the Dialog Bar rather than using the Main Menu.

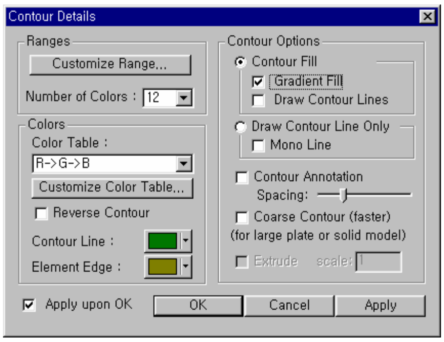
Type of Display

Multiple selections are possible. It controls the display of the post-processing results.

Contour Display the analysis and design results in the form of contour diagrams.

Assign the type of contour lines, the number of colors (***Number of Colors***), the range of color distribution (***Customize Range***), the type of colors (***Color Table***), the change of colors (***Customize Color Table***), the Gradation, etc.

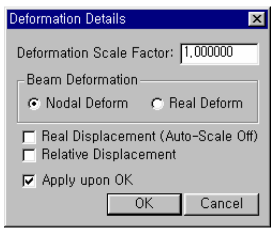
Notice that substantial time is required to print a contour processed with Gradation via Windows Meta File.



Contour Details dialog box

Deform Display the deformed shape.

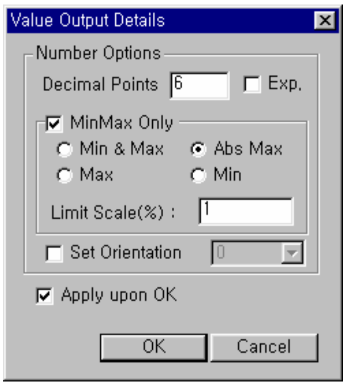
Adjust the deformation scale (***Scale Factor)*** of the deformed shape, or determine the display type of the deformed shape.
MIDAS/Gen provides two types of deformed shapes.
"***Nodal Deform***" reflects only the nodal displacements and "***Real Deform***" computes additionally the intermittent of beam elements between the end nodes.



Deformation Details dialog box

Values Display the numerical values of displacements, member forces and stresses at the assigned location.

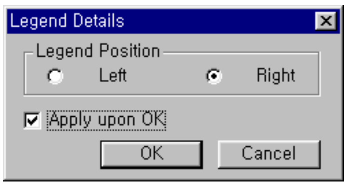
... Assign the number of decimal points and specify the option of expressing the values in the exponential form. In addition, only the maximum/minimum values may be displayed. Use the ***Font*** tab of ***Display Option*** to adjust the color and size of the numerical values.



Value Output Details dialog box

Legend ... Assign the position and color of the legend that reflects all the reference items on the post-processing screen.

The color of legend can be adjusted through Display Option.

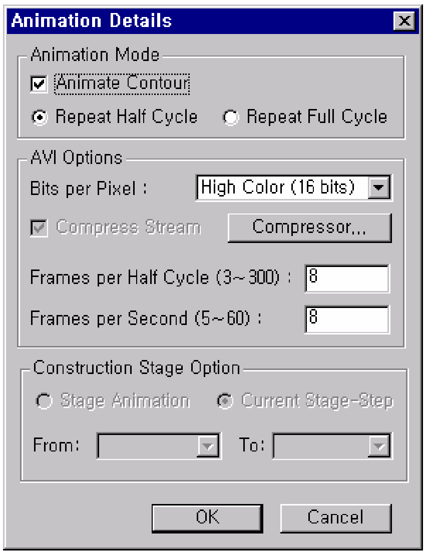


Legend Details dialog box

Animate Simulate the deformation process of the model dynamically.

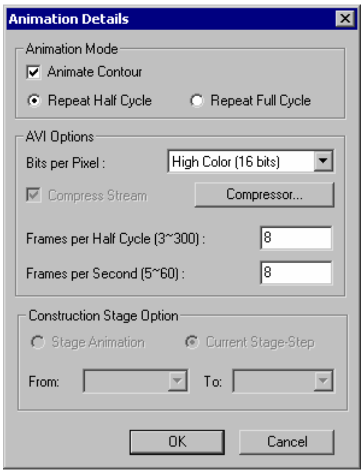
Specify whether or not the color of the contour diagram is to be changed according to the dynamic deformation process (***Animate Contour***). Also specify the iteration cycle of the dynamic deformation process as a half cycle or a full cycle.

For reference, select the half cycle when simulating the deformed shape of the structure and select the full cycle when simulating the vibration modes or buckling modes. In ***AVI Options***, assign the number of colors per pixel (***Bits per Pixel***) to set the dynamic base screen and the compression option of the screen data (***Compressor***). Specify the number of cutting frames (***Frames per Half Cycle***) and the number of frames per second (***Frames per Second***) to display. These items affect the quality, animation processing time per cycle, and also the quality of the dynamic screen image processing. When a construction stage analysis is performed, the animation by construction stages or by steps within a construction stage may be assigned.



Animation Details dialog box

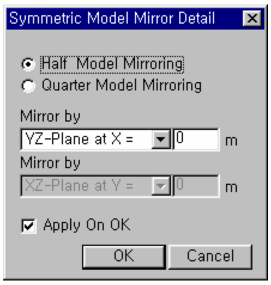
Undeformed Display the deformed shape overlapped with the undeformed model. Use the ***Draw*** tab of ***Display Option*** to control the display of the undeformed shape.



Animation Detail dialog box

Mirrored Carry out the analysis using a 1/2 or 1/4 model and expand the results to create the results of the full model by plane symmetry.

... Define the reference plane(s) about which the symmetry is created.



Symmetric Model Mirror Detail dialog box

Cutting Diagram Display the stresses in plate elements at specified cutting lines or planes.

... Define the cutting lines or planes and select the direction of stresses for display. Assign the form of display type for the stresses (numerical values, graphs, min/max, etc.).

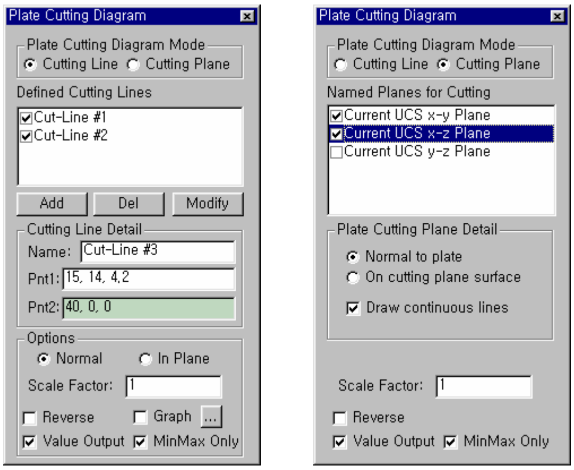
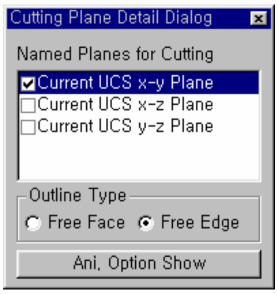


Plate cutting Diagram dialog box

Cutting Plane Display the stresses in solid elements at specified cutting planes.

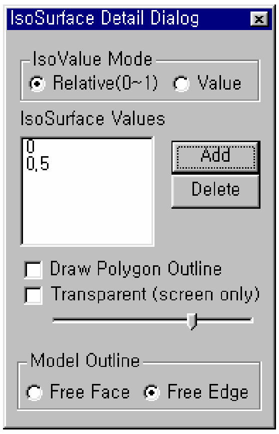
... Define the cutting planes, the expression method for solid elements and the moving or rotating direction for animation.



Cutting Plane Detail dialog box

IsoSurface Display the IsoSurfaces of solid elements, which represent the surfaces of equal stresses for given stress values.

Specify the stress values for which the IsoSurfaces are to be displayed and assign the method of representing solid elements.

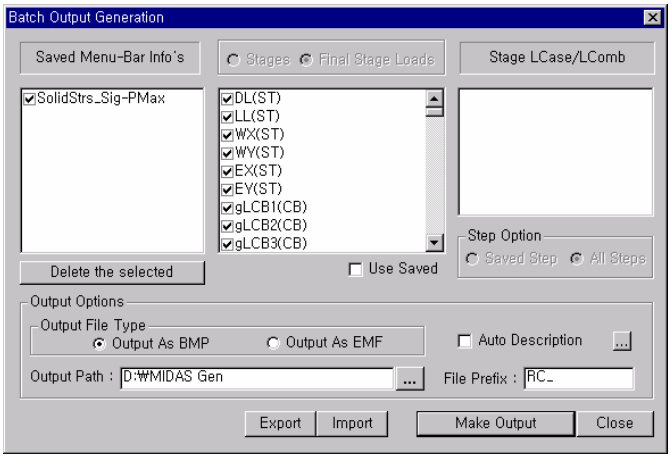


IsoSurface Detail dialog box

Batch Output Generation From the selected output categories, produce all graphic output at once by sequentially changing the load cases and combinations.

 Select the screen output types and assign them as base files.

 Assign base files, load cases/combinations, analysis relations, etc. to generate Batch Output.



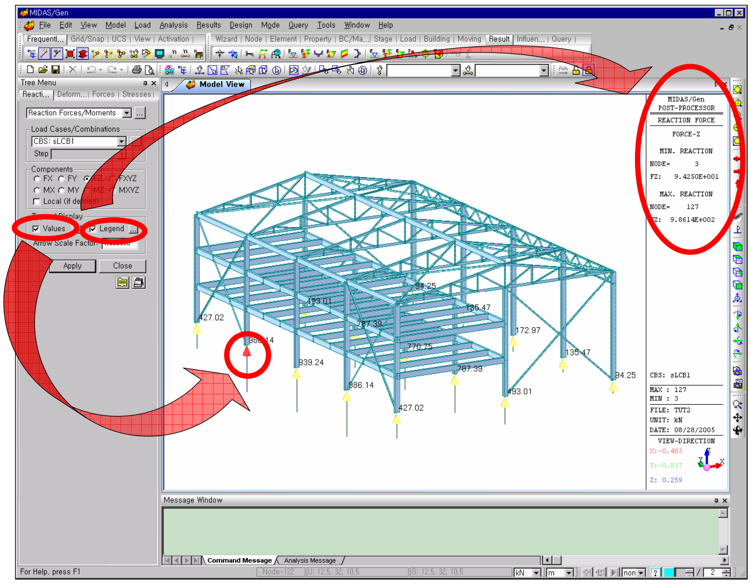
Batch Output Generation dialog box

Post-Processing Function Types

Examples of results display and the types of post-processing functions in **MIDAS/Gen** are noted below. Use ***Type of Display*** to produce various types of Graphic Output.

Display of Reactions

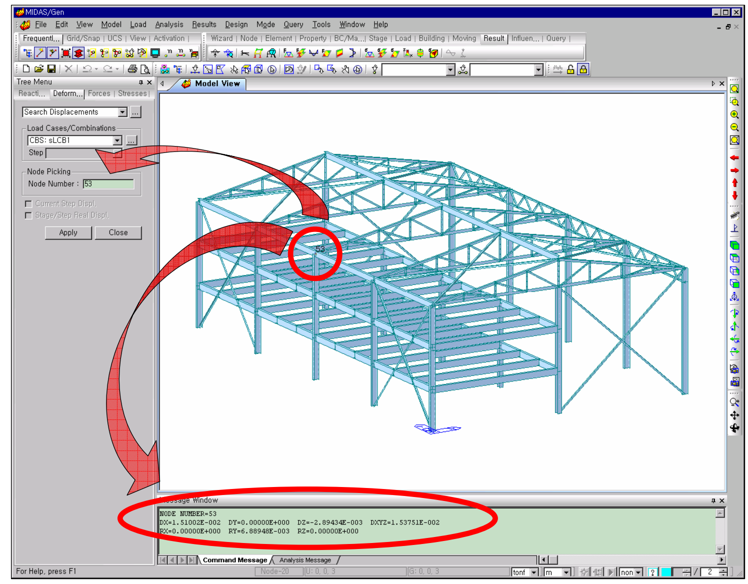
🎧 Select "Values" in Type of Display in the dialog bar to display the numerical values of the reactions.



Reaction Forces/Moments: Vertical Reactions

Display of Deformed Shape

🎧 Click the mouse cursor over the desired node to display the relevant displacements in Message Window.

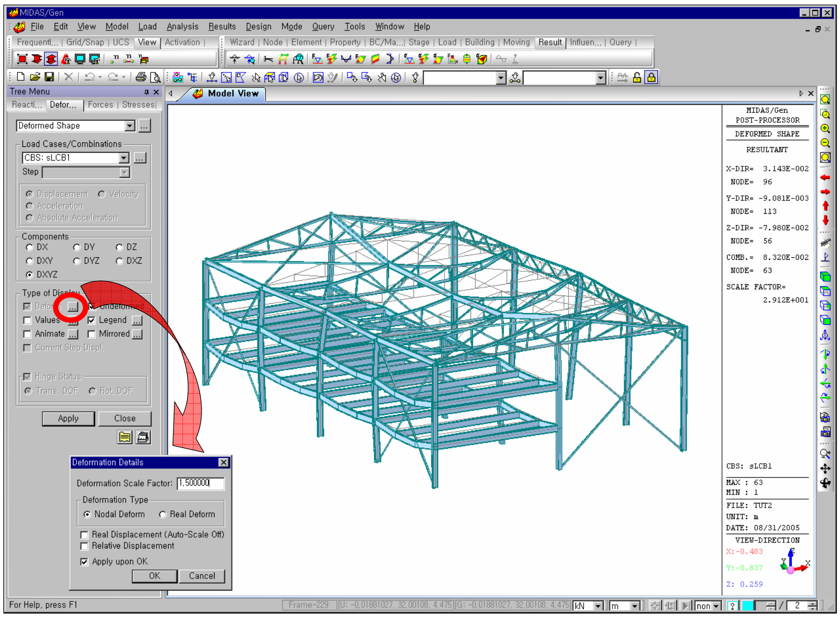


Search Displacements

🎧 Select Perspective and Hidden, then a very realistic contour will be displayed.

🎧 Select Undeformed to view the deformed shape overlapped with the undeformed model.

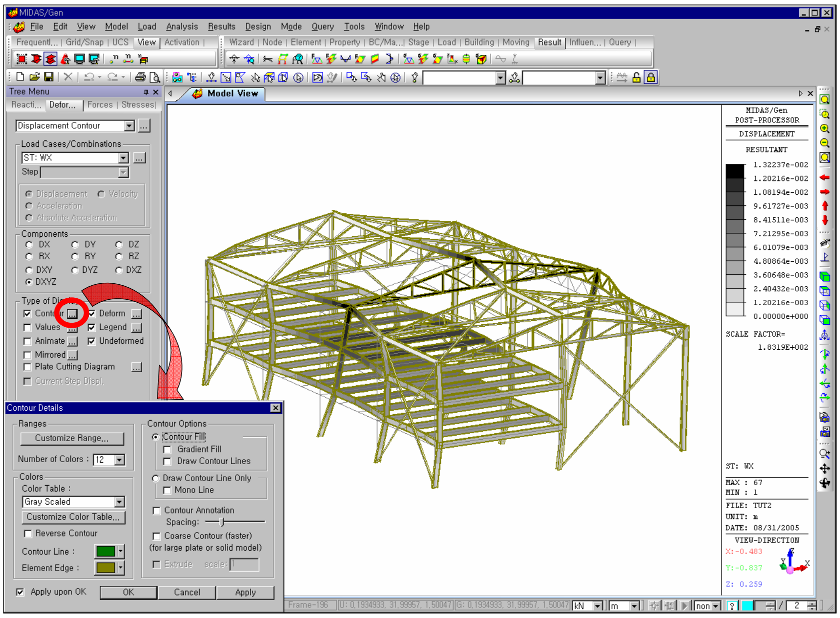
🎧 Click the ... button to the right of Deform in Type of Display in the dialog bar to adjust the scale of the deformed shape.



Deformed Shape + Undeformed Shape

🎧 Click ... next to Contour in Type of Display of the dialog bar to adjust the division of contour, the types of colors and the gradient treatment.

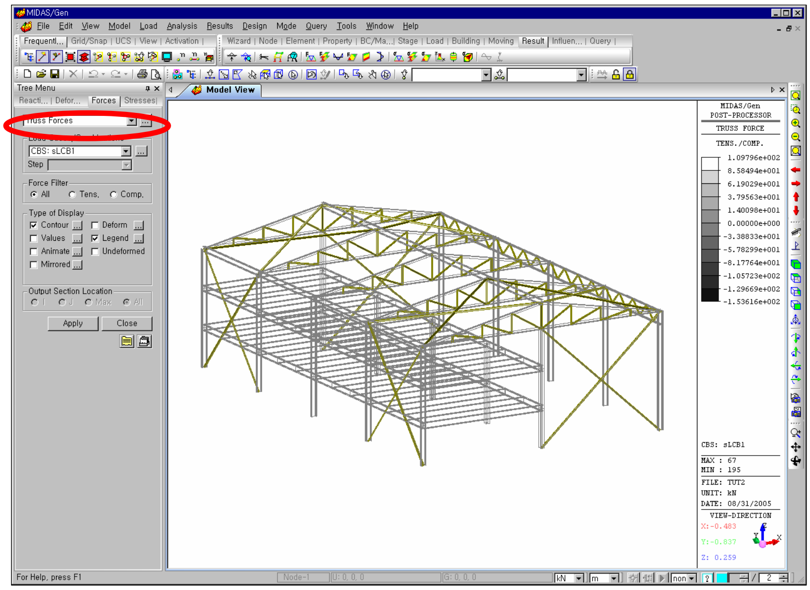
🎧 Select Legend. The color palette, relevant table of numerical values, model coordinate axes, file name, working time, etc., can be displayed on the left or right of the Model Window.



Displacement Contour

Display of Member Forces

With Truss Forces, the member force contour is displayed for only truss elements. For other elements, only the outlines are displayed. Using Select Identity and Active, only truss elements can be displayed on the screen.

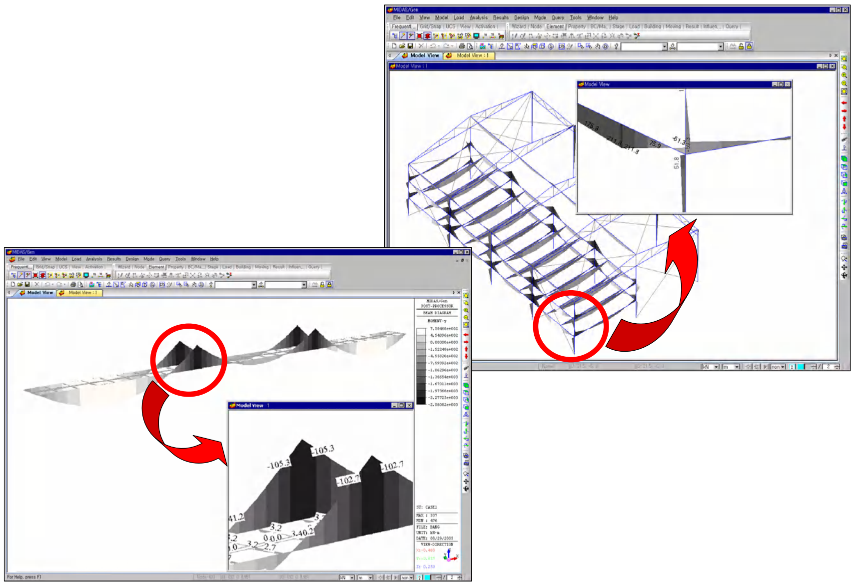


Truss Forces

Check (✓) in "Values" in Type of Display and assign "Max" in Output Section Location to display the maximum member forces for beam elements.

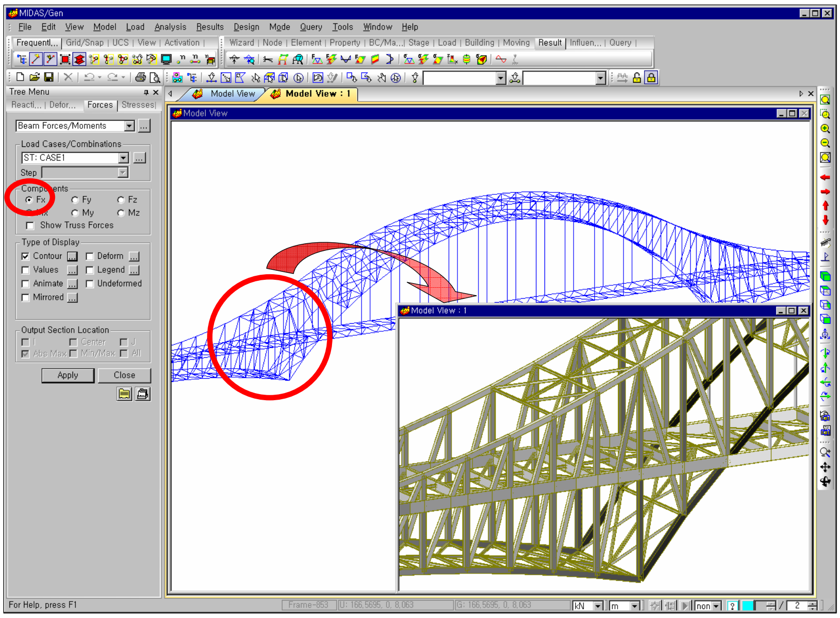
With "5 Points" in Display Options, the shear forces and bending moments are computed at the quarter points of the beam elements to represent the distribution.

With "Exact" the shear forces and bending moments are computed over the entire lengths of the beam elements and SFD and BMD are displayed exactly. Select "Fyz" or "Myz" in Components to display the SFD/BMD about the strong & weak axes simultaneously.



Beam BMD

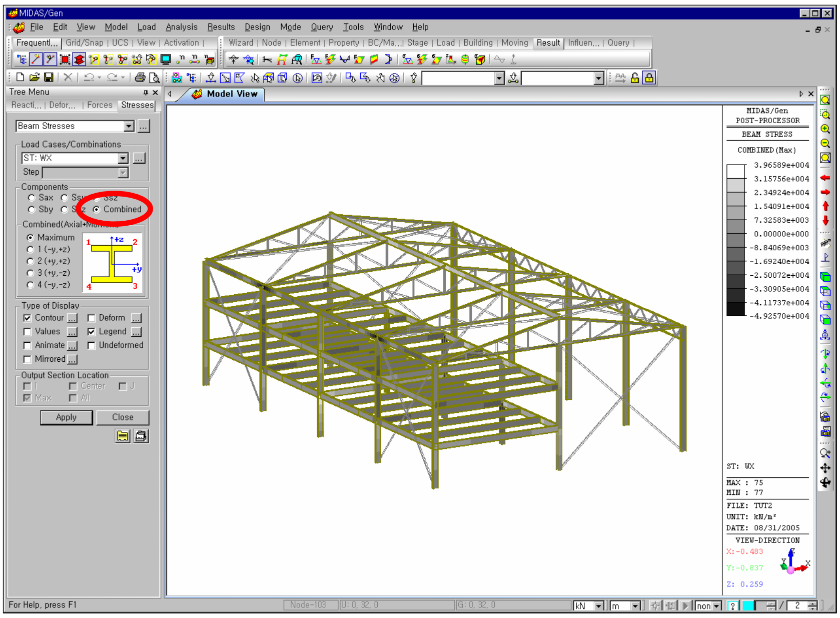
🎧 Using Window>New Window, different types of windows can be displayed simultaneously.



Beam Forces/Moments: Axial Forces

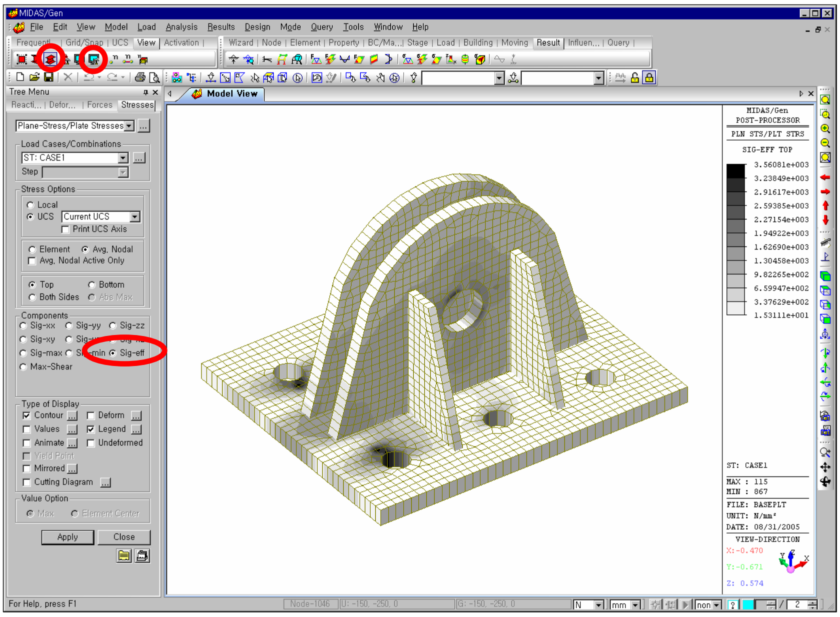
Display of Stresses

🎧 Selecting "Combined" in Components field, the combined stresses (axial stress + strong / weak axes bending stresses) applied to beam elements can be examined.



Beam Stresses: Combined Stresses

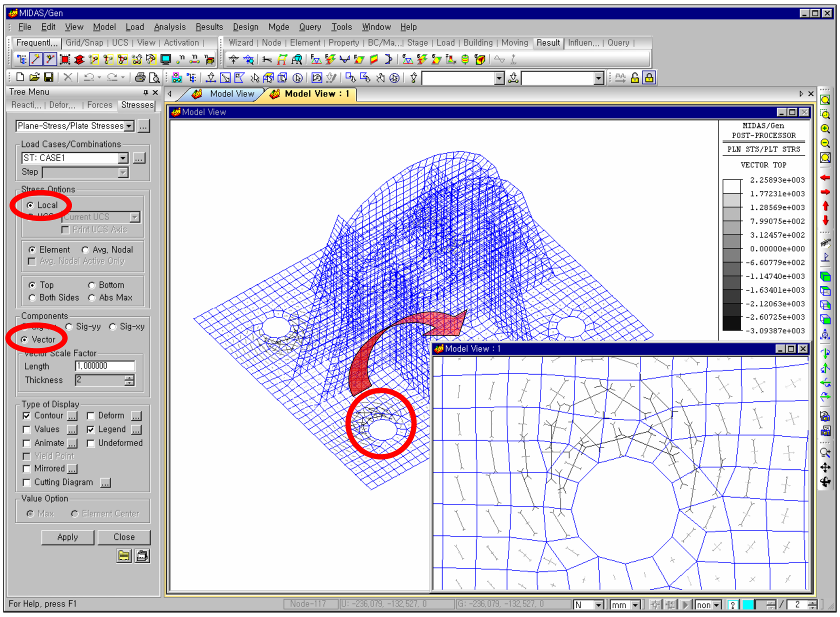
Select Hidden Option (Model) in the Draw tab of Display Option and assign Plane Thickness in the Thickness field. Then, click Hidden, to display the stress distribution of plate elements reflecting the thickness.



Plane-Stress/Plate Stresses: von-Mises Stress Contour

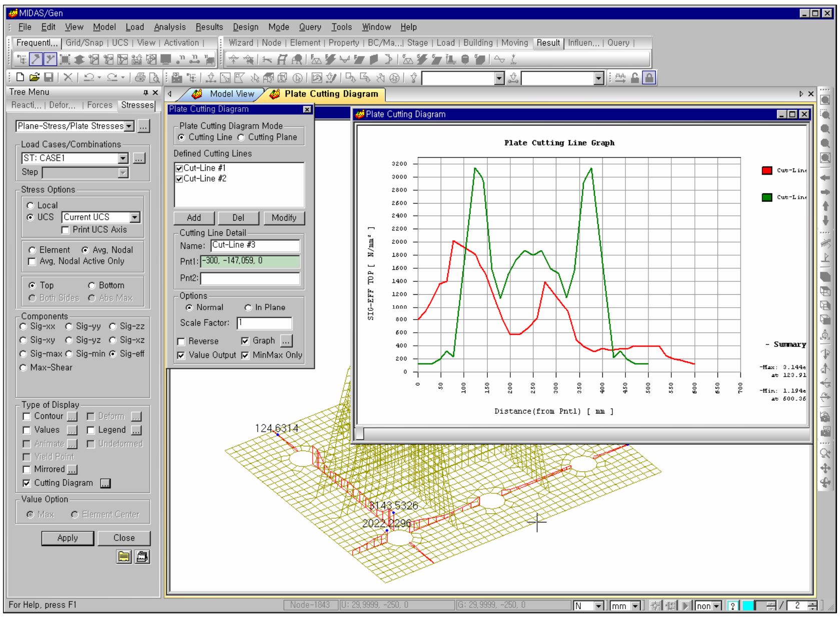
With "Local" in the Stress Options field and "Vector" in the Components field, the principal stress contour is displayed as vectors.

Select Window>New Window to display different post-processing results simultaneously in separate windows.

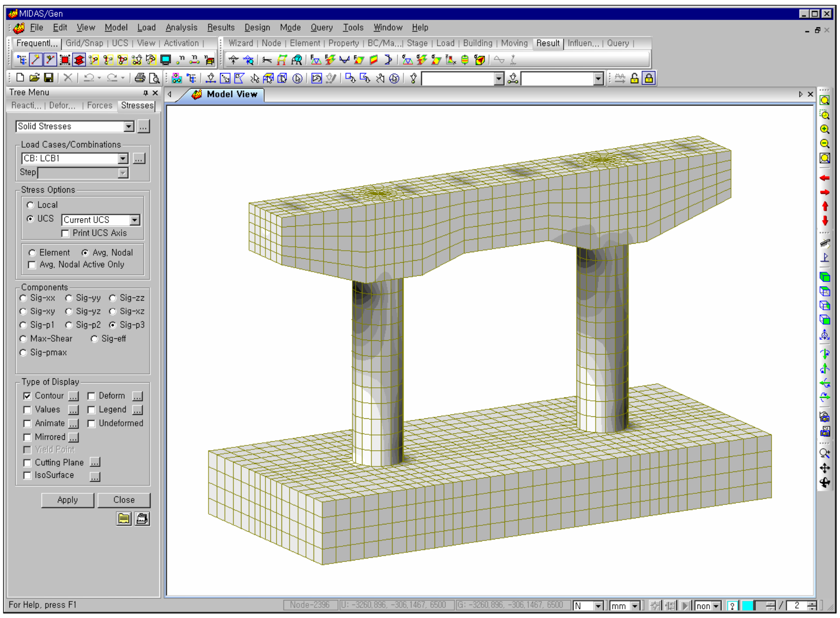


Plane-Stress/Plate Stresses: Principal Stress Vectors

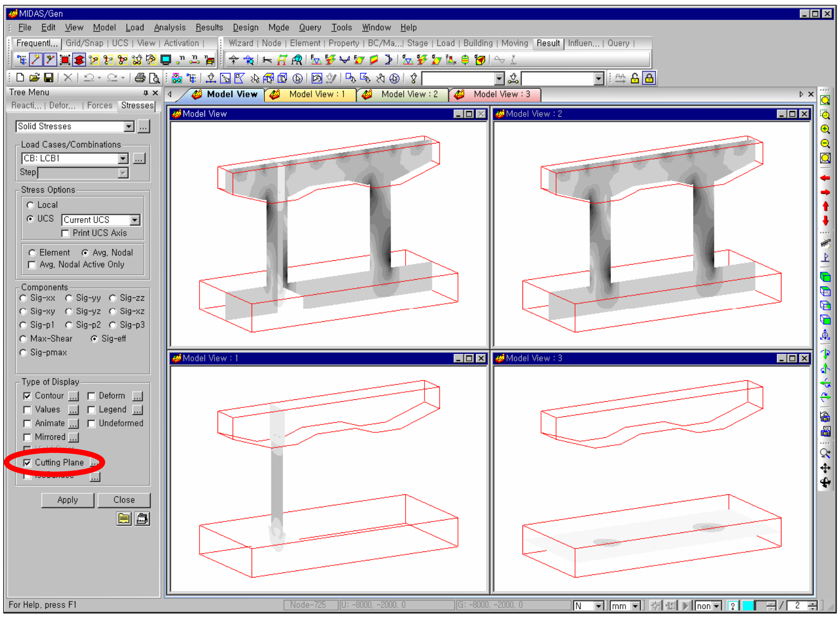
With Cutting Diagram, plate stresses can be displayed at the specified cutting lines in graphs.



Plane-Stress/Plate Stresses: Cutting Diagram



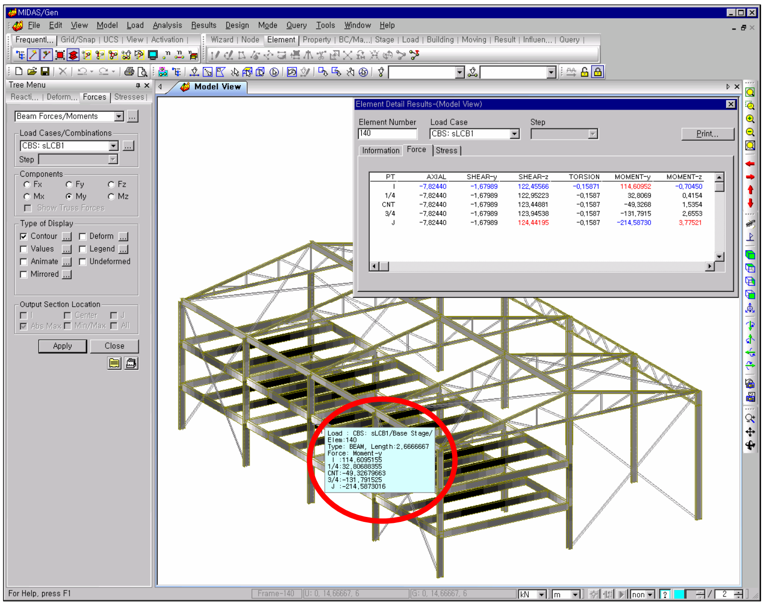
Solid Stresses - Principal Stress Contour



Solid Stresses – Cutting Planes

Display of Analysis Results for individual Elements

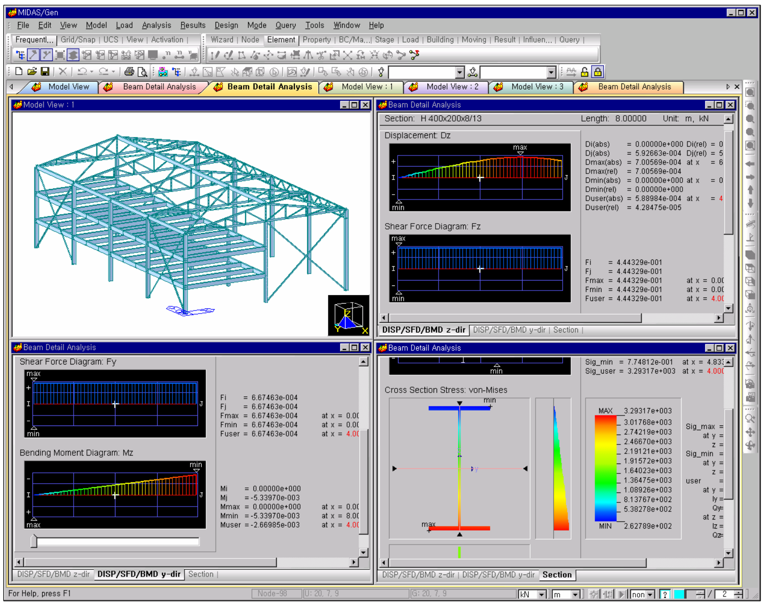
🎧 With Fast Query, analysis results for an element can be displayed in the post - processing mode.



Element Detail Results

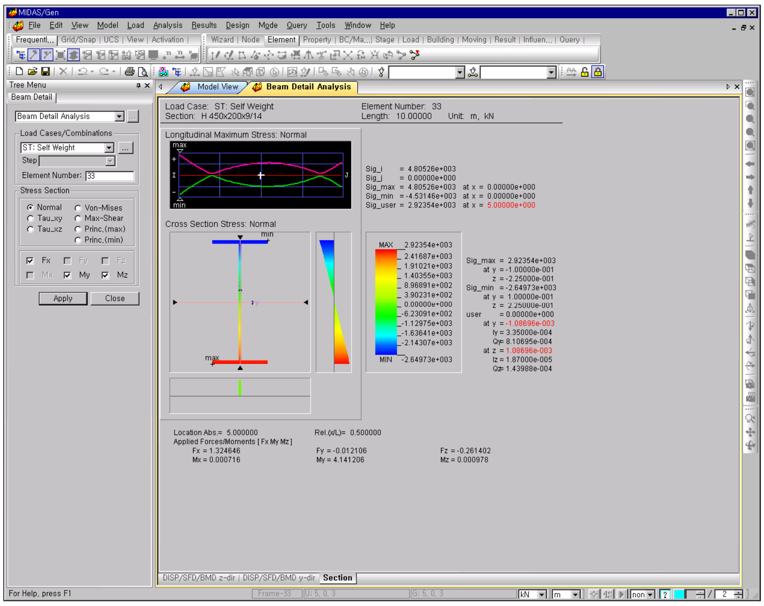
Display of Detail Analysis Results for individual Beam Elements

Beam Detail Analysis supplies, for a specific beam element, the detail displacement diagram, SFD/BMD, the section stress related to a particular section, the maximum stress distribution diagram over the entire length of the beam element, etc.



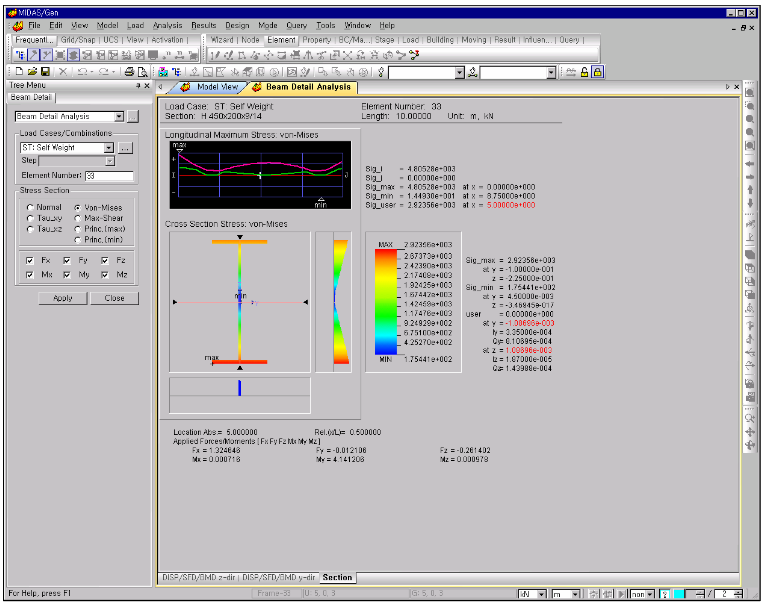
Beam Detail Analysis

If a particular position on a beam element is specified, the bending stress, shear stress, effective stress, etc. occurring at that position can be evaluated.



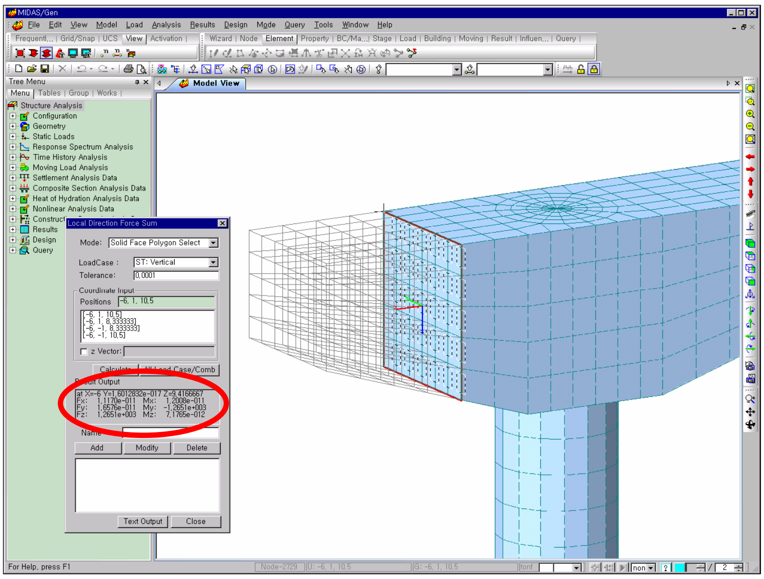
Beam Detail analysis: Normal Stress

Upon selecting a particular point on a cross section, bending, shear and effective stresses, etc. can be checked in detail.



Beam Detail Analysis: von-Mises Stress

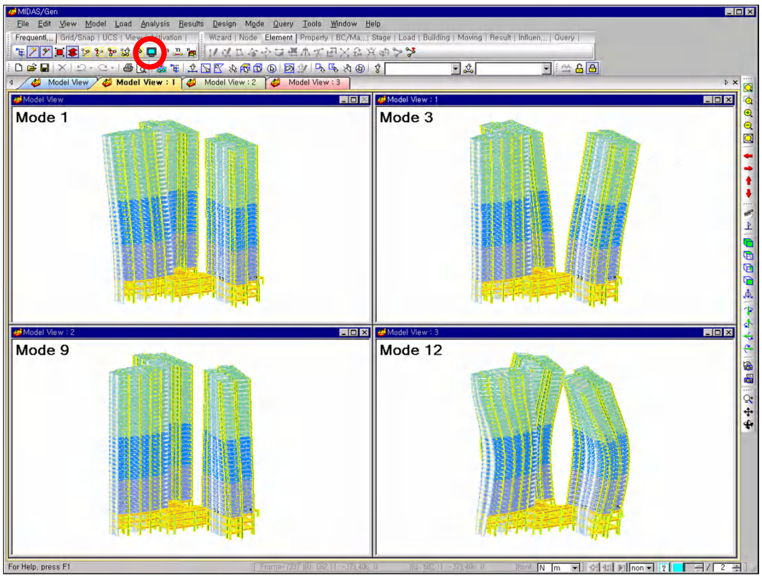
Display of Local Direction Force Sum



Local Direction Force Sum

Display of Vibration Mode Shapes

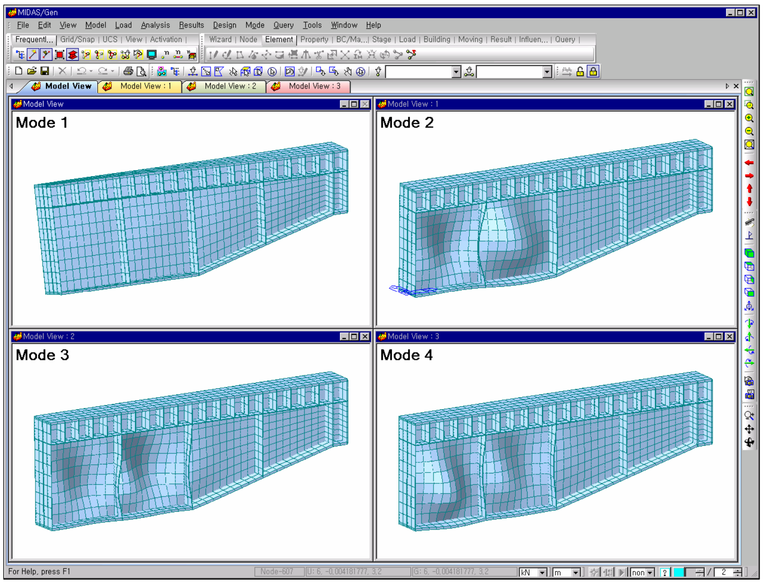
Select the View tab in Display and use Description to include comments on the screen. Click the Font... button to the right of Description to adjust the size, type and color of the fonts.



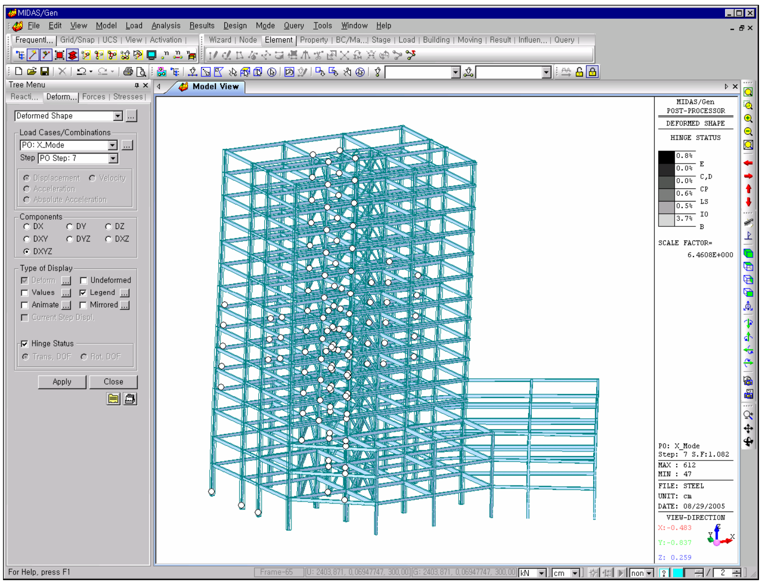
Vibration Mode Shapes

Display of Buckling Mode Shapes

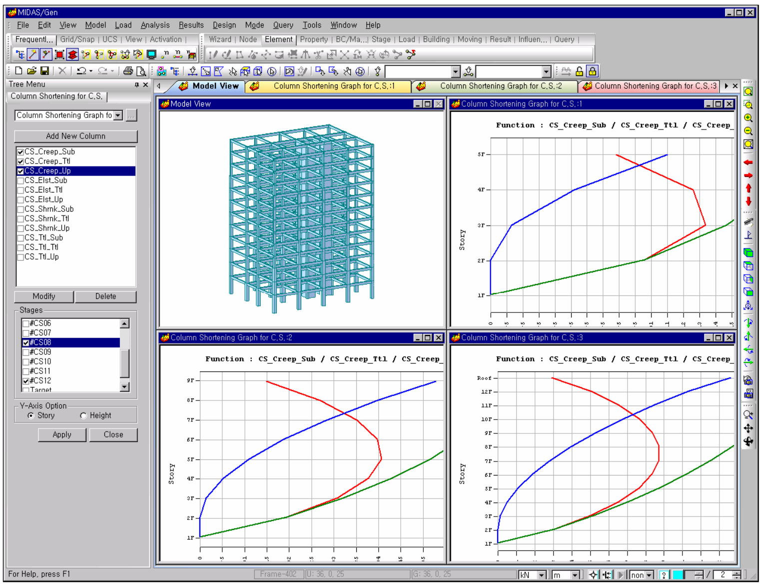
Using Multi-Modes, you can analyze several mode shapes simultaneously.



Buckling Mode Shapes



Pushover Analysis Results



Construction Stage Analysis Results (Column Shortening Graphs)

Animation

MIDAS/Gen provides the capability of animating static and dynamic analysis results. The animation reflecting dynamic effects of the analysis results can be extremely useful in analyzing the structural behaviors and creating presentation materials. Follow the directions below.

1. Recall the functions (***Beam Stresses, Vibration Mode Shapes***, etc.), which yield deformed shapes, vibration modes, buckling modes, etc. and select the desired load case or mode.

2. In ***Components***, select the component of relevant analysis results.

3. Select "***Animate***" in ***Type of Display***, and choose additional selection items as necessary.

4. Click Apply.

5. Select ***Record*** in the animation control bar at the bottom of the working window. The Animation reflecting the items selected in ***Type of Display*** is displayed repeatedly on the working window. Use the ... button to the right of ***Animate*** to adjust the speed of animation.

6. Select ***Save*** in the animation control bar and enter the desired filename to save the played animation. If the extension of the file is not assigned explicitly, the "AVI" extension is imposed. Double-click to replay the saved animation after searching the relevant file in the folder.

7. Select ***Close*** to terminate the animation function.

The Icons controlling the animation during the animated simulation are as follows:

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

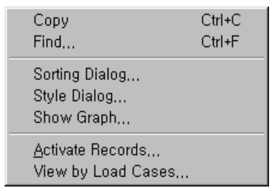
Please note that animation is not supplied in ***Render View***.

Verification by Result Tables

In ***Results>Result Tables,*** **MIDAS/Gen** provides Table Window in the spreadsheet form similar to that of Excel, which enables us to evaluate the analysis and design results at a glance. MIDAS/Gen provides the following verification capabilities for result tables:

Refer to Getting started>Tables> Table Tool Directions of On-line manual for detail information.

- Spreadsheets related to all the analysis and design results (displacements, member forces, stresses, reactions, vibration modes, buckling modes, heat of hydration results, inter-story drifts, story displacements, story shear, etc.)
- A powerful ***Filtering*** function linking all types of selection functions
- All types of ***Sorting*** functions (Multiple ascending/descending sorting rearranged in the order of priorities by material attributes)
- Adjustment of text style (positions, formats of numerical values, assignment of effective decimal points, etc.)
- ***Copy***/***Paste*** functions through the clipboard (assignment of all types of copy range)
- Search text and numbers
- Transfer data with other database S/W such as Excel
- Elegant report output template forms

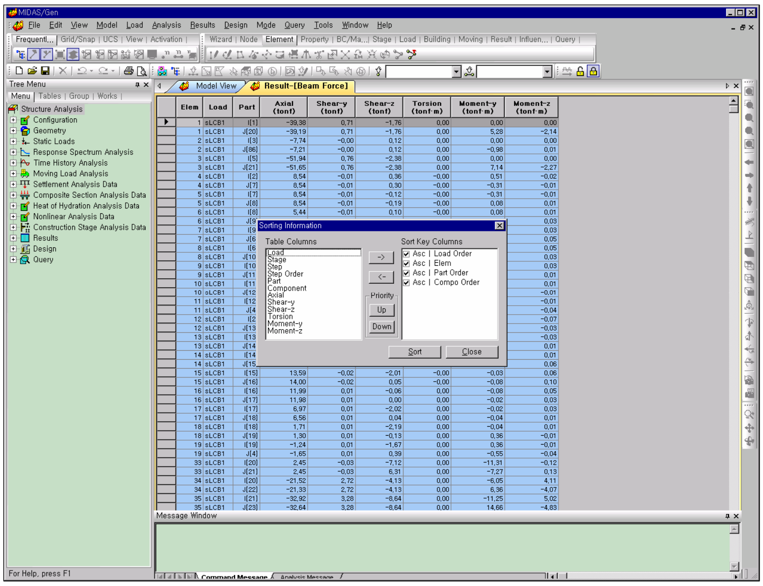


Context Menu in Table Window

Context Menu prompts when the mouse curser is right-clicked on the table window. If ***Graph***, ***Filtering*** and ***Sorting*** supplied by ***Context Menu*** of ***Table Window*** are interactively used, the analysis results can be efficiently analyzed for different structural characteristics. The types and purposes of ***Context Menu*** in the analysis results table are as follows:

Sorting Dialog

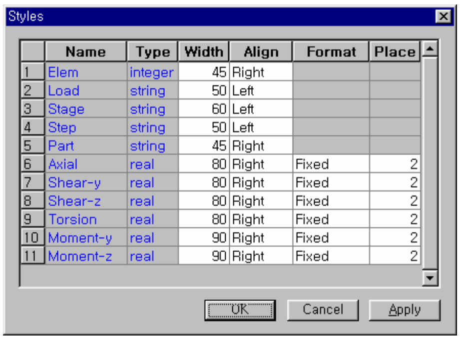
Arrange the table data in columns. We can accomplish sorting the data in an ascending/descending order and rearranging the data columns in the order of priority. For example, sorting the member forces of beam elements by strong axis bending moments, weak axis bending moments and axial forces in a descending order displays the following:



Display of Table Sorting Dialog

Style Dialog

Adjust the column width, alignment, format of real numbers, decimal points, etc., in the table for display.



Styles

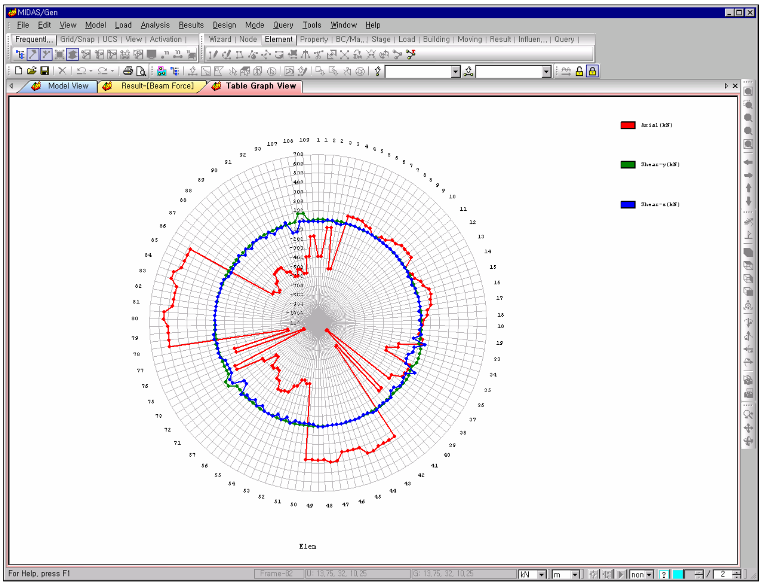
	Name	Type	Width	Align	Format	Place
1	Elem	integer	45	Right		
2	Load	string	50	Left		
3	Stage	string	60	Left		
4	Step	string	50	Left		
5	Part	string	45	Right		
6	Axial	real	80	Right	Fixed	2
7	Shear-y	real	80	Right	Fixed	2
8	Shear-z	real	80	Right	Fixed	2
9	Torsion	real	80	Right	Fixed	2
10	Moment-y	real	90	Right	Fixed	2
11	Moment-z	real	90	Right	Fixed	2

OK Cancel Apply

Style Dialog

Show Graph

25 types of graphs are provided for the table data output.

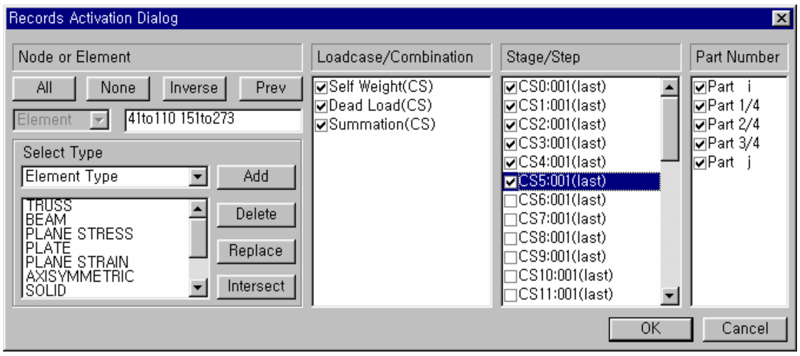


Display of Graph: Web Chart

Active Records

Produce the output data selectively by the attributes of elements (element types, types of material properties, section types, group, etc.), or produce the member forces or stresses of beam elements selectively by load cases/combinations, construction stages and positions (*i*-node, 1/4, 1/2, 3/4 & *j*-node).

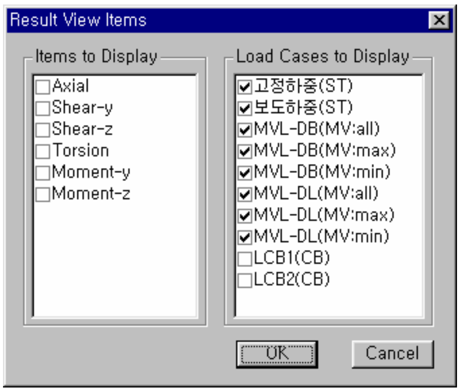
Where eigenvalue or buckling analysis has been performed, the output can be selectively produced by vibration or buckling modes.



Records Activation dialog box

View by Load Cases

Produce the member forces selectively by load cases/combinations.



Result View Items dialog box

Design

General

The design features of **MIDAS/Gen** are used to design beams, columns, walls, footings and other structural elements in accordance with the designated design standards or to interpret the results of strength verification. As the design features are implemented only under the post-processing environment, the following process must be observed:

- Complete the structural analysis model of the structure
- Enter the loading conditions data
- Perform the structural analysis

Structural models prepared for member design or strength verification must reflect the following basic conditions:

- Set the axial directions (element coordinate system x-axis) of columns and shear walls to be parallel with the GCS Z-direction for structural design.
- Locate the reinforced concrete beam elements on a plane parallel with the GCS X-Y plane of the analysis model.

Design Criteria and Load Combinations

The design features of **MIDAS/Gen** incorporate the following design criteria:

- ***Steel structures design standards***

  · Manual of Steel Construction, Load & Resistance Factor Design, the American Institute of Steel Construction (AISC – Part 6, LRFD93 & 2000)

  · Manual of Steel Construction, Allowable Stress Design, the American Institute of Steel Construction (AISC - Part 5, ASD89)

  · Part 1. Code of practice for design in simple and continuous construction, British Standard (BS5950-90)

  · Part 1.1 General Rules and Rules for Building, Design of Steel Structures (ENV 1993-1-1 Eurocode 3)

  · Canadian Standards Association, Limit States Design of Steel Structures, 2001 (CSA-S16-01)

  · Cold-Formed Steel Design, American Iron and Steel Institute (AISI-CFSD 86)

  · TWN-ASD90, Taiwan Standard, Allowable Stress Design Specification and Commentary for Structural Steel Building, 2001

  · TWN-LSD90, Taiwan Standard, Limit States Design Specification and Commentary for Structural Steel Building, 2001

  · IS:800-1984, Indian Standard, Code of Practice for General Construction in Steel (Second Revision), 1984

- ***RC (Reinforced Concrete) structures design standards***

  · Building Code Requirements for Structural Concrete and Commentary (ACI318-89, 95, 99 & 02)

  · Canadian Standards Association, Design of Concrete Structures, 1994 (CSA-A23.3-94)

  · Part 1. Code of practice for design and construction, British Standard (BS8110-97)

· Part 1. General Rules and Rules for Building, Design of concrete structures (ENV 1992-1-1 Eurocode 2)

· TWN-USD92, Taiwan Standard, Design Specification and Commentary for Concrete Structures, 2003

· IS456:2000, Indian Standard, Plain and Reinforced Concrete Code of Practice (Fourth Revision), 2000

➢ ***SRC (Steel-Reinforced Concrete) composite structures design criteria***

· A Specification for the Design of Steel-Concrete Composite Columns, Structural Stability Research Council, US (SSRC79)

· TWN-SRC92, Taiwan Standard, Design Specification and Commentary for Steel Reinforced Concrete Structures, 2003

We may manually define the load combinations for design in ***Results*>** ***Combinations*** or use the load combinations generated automatically in accordance with the applicable design standards. For detail information, refer to "***Combinations***" in ***On-line Manual***.

In order to use the design features, we are required to specify the design parameters and load combinations in the process of design or strength verification.

We can revert to the preprocessing mode to modify the model data based on the results of strength verification or member design. However, it is cautioned that the analysis results or member design (strength verification) results may be deleted in such a case.

Entering Design Parameters

The Design menu provides the Design features of MIDAS/Gen and contains the following sub-menus:

➢ ***General Design Parameter***

The commonly required design parameters are defined regardless of structural materials or member types.

Refer to On-line Manual for detail information on the design parameters.

Definition of Frame

Define the structure type as laterally braced or unbraced.

Live Load Reduction Factor

Provide the live load reduction factors for all columns, shear walls and footings.

Unbraced Length

Provide unbraced lengths or laterally braced lengths for members.

Effective Length Factor

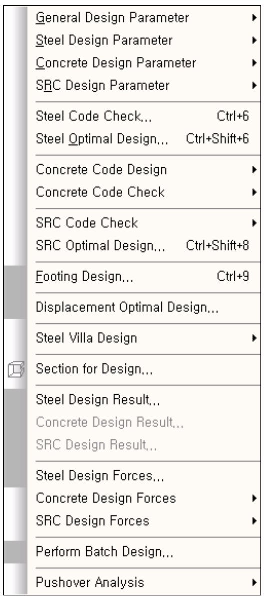
Provide effective buckling length factors.

Limiting Slenderness Ratio

Provide the critical (maximum allowable) slenderness ratio.

Moment Factor

Provide moment factors.



Composition of the design feature menu

Moment Magnifier
Provide moment magnifiers.

Modify Live Load Reduction Factor
Modify the live load reduction factors already specified, or provide the live load reduction factors for individual members.

Modify Member Type
Modify the member types that the program selected automatically.

General Design Tables
Arrange the design parameters, defined by the user, in a table format and modify or remove the pre-defined design parameters.

- ***Steel Design Parameter***

The design standards and the design parameters are defined for steel structures.

Design Code
Assign the design standard.

Modify Steel Material
Modify the material properties.

Bending Coefficient
Provide the bending coefficients.

Shear Coefficient
Provide the shear coefficients.

Specify Allowable Stress
Provide the allowable stresses.

Longitudinal Stiffener of Box Section
Provide the lateral stiffener sizes and spacing for box sections.

Steel Design Tables
Arrange the design parameters in tables and modify or remove the pre-defined design parameters.

- ***Concrete Design Parameter***

The design standards and the design parameters are defined for RC structures.

Design Code
Assign the design standards.

Strength Reduction Factor
Provide the strength reduction factors.

Modified section data are used only for design feature. They do not affect the stiffness data for analysis.

Modify Concrete Material
Modify the material properties.

Limiting Maximum Rebar Ratio
Provide the maximum allowable rebar ratio.

Design Criteria for Rebar
Assign the sizes of rebars and the design method for shear walls.

Modify Beam Section Data
Specify or modify beam section data for strength verification.

Modify Column Section Data
Specify or modify column section data for strength verification.

Modify Brace Section Data
Specify or modify bracing section data for strength verification.

Modify Wall Section Data
Specify or modify shear wall section data for strength verification.

Modify Wall Mark Data
Specify or modify shear wall names.

Concrete Design Tables
Arrange the design parameters in tables and modify or remove the pre-defined design parameters.

- ***SRC Design Parameter***

The design standards and the design parameters are defined for SRC (Steel–Reinforced Concrete) composite structures.

Design Code
Assign the design standards.

Modify SRC Material
Modify the material properties.

Modify SRC Section Data
Specify or modify SRC section data.

SRC Design Tables
Arrange the design parameters in tables and modify or remove the pre-defined design parameters.

- ***Steel Code Check***

Verify strength for steel members.

- ***Steel Optimal Design***

Perform optimal design for steel members.

➢ ***Concrete Code Design***	Design RC members.
Beam Design	Design beam members.
Column Design	Design column members.
Brace Design	Design bracing members.
Wall Design	Design shear walls.
➢ ***Concrete Code Check***	Perform strength verification for RC structures.
Beam Checking	Verify strength for beam members.
Column Checking	Verify strength for column members.
Brace Checking	Verify strength for bracing members.
Wall Checking	Verify strength for shear wall members.
➢ ***SRC Code Check***	Perform strength verification for SRC members.
➢ ***SRC Optimal Check***	Perform optimal design for SRC members.
➢ ***Footing Design***	Design footings.
➢ ***Section for Design***	Inquire or modify section data.
➢ ***Steel Design Result***	Verify design results for steel members.
➢ ***Concrete Design Result***	Verify design results for RC members.
➢ ***SRC Design Result***	Verify design results for SRC members.
➢ ***Perform Batch Design***	Perform a number of design tasks including structural analyses.

Procedure for Implementing the Design Features

MIDAS/Gen designs the following types of members:

- Steel members
- RC members (including RC shear wall members)
- Steel-Reinforced Concrete (SRC) members
- Footings

A common procedure for implementing the design features of **MIDAS/Gen** is as follows:

---

1. Enter the design parameters
   Enter the design parameters using the sub-menus of ***Design***. The data entry for design parameters is possible in both pre-processing and post-processing modes.

2. Enter the load combinations
   Enter the design load combinations using ***Results>Combinations***. Modification factors must be incorporated in the design load combinations for seismic loads. Verify the pre-defined design load combinations for compatibility with the member design.

   The design load combinations are applied according to the design types noted below and classified by tabs in the dialog box.

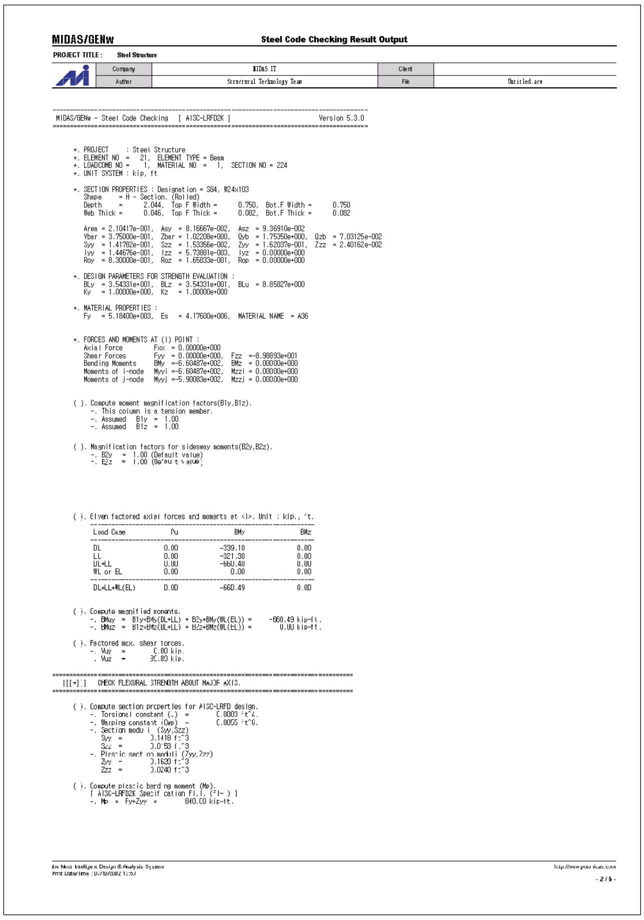
Steel Design	Design steel members
Concrete Design	Design RC members
SRC Design	Design SRC members
Footing Design	Design footings

3. Mode switching
   When the preparation for design or strength verification is completed, confirm the current mode. If the current environment is in the preprocessing mode, switch to the post-processing mode.

4. Member design or strength verification
   Design or verify strength for members by selecting the design features in the sub-menus of ***Design*** for each type of structural material or member.

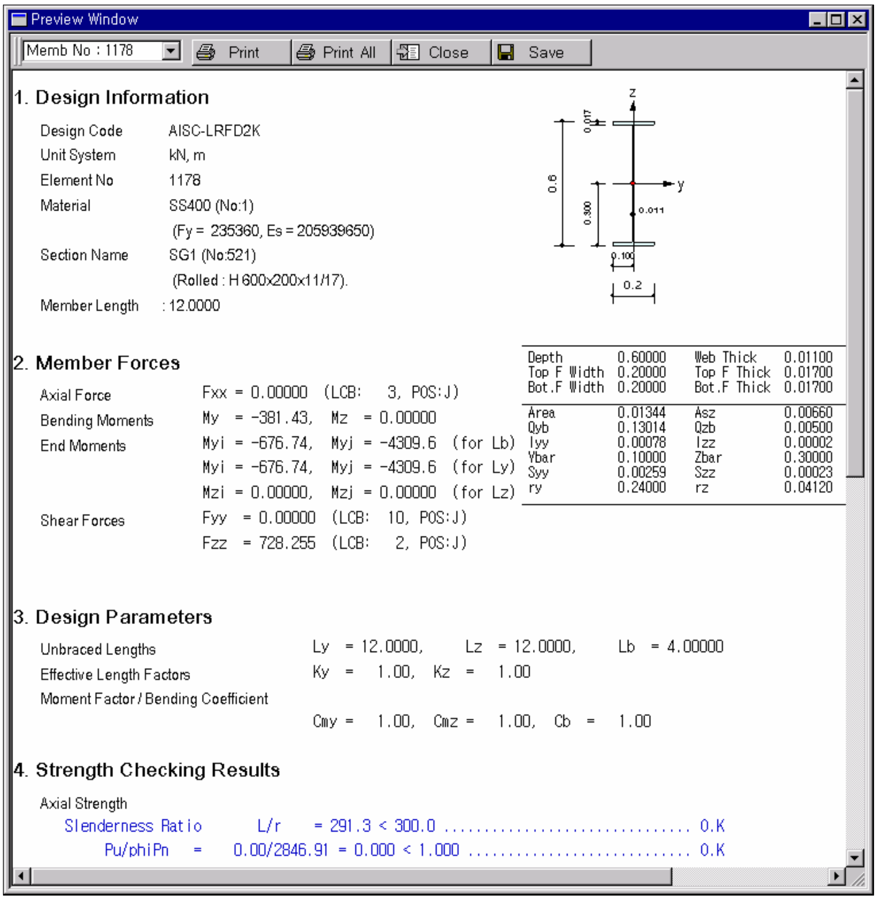
5. Check member design or strength verification results
   The design results will appear on the screen after member design or strength verification. The output results supplied by the strength verification feature for steel members are as follows:
   - Detail calculations of strength verification for individual members
   - Summary calculations of strength verification results for individual members/sections

   The output results supplied by the section design feature for RC members are as follows:
   - Detail calculations of strength verification for individual members
   - Summary calculations of automatic design results for individual members/sections



```
MIDAS/GENw                         Steel Code Checking Result Output
PROJECT TITLE :   Steel Structure
Company   MIDAS IT                       Client
Author    Structural Technology Team     File    Untitled.ace

--------------------------------------------------------------------------------
 MIDAS/GENw - Steel Code Checking   [ AISC-LRFD2K ]               Version 5.3.0
================================================================================

   *. PROJECT      : Steel Structure
   *. ELEMENT NO  =    21,  ELEMENT TYPE = Beam
   *. LOADCOMB NO =     1,  MATERIAL NO  =    1,  SECTION NO = 224
   *. UNIT SYSTEM : kip, ft

   *. SECTION PROPERTIES : Designation = SG4, W24x103
      Shape     = H - Section. (Rolled)
      Depth     =      2.044,  Top F Width =      0.750,  Bot.F Width =      0.750
      Web Thick =      0.046,  Top F Thick =      0.082,  Bot.F Thick =      0.082

      Area = 2.10417e-001,  Asy  = 8.16667e-002,  Asz  = 9.36910e-002
      Ybar = 3.75000e-001,  Zbar = 1.02208e+000,  Qyb  = 1.75350e+000,  Qzb  = 7.03125e-002
      Syy  = 1.41782e-001,  Szz  = 1.53356e-002,  Zyy  = 1.62037e-001,  Zzz  = 2.40162e-002
      Iyy  = 1.44676e-001,  Izz  = 5.73881e-003,  Iyz  = 0.00000e+000
      Roy  = 8.30000e-001,  Roz  = 1.65833e-001,  Rop  = 0.00000e+000

   *. DESIGN PARAMETERS FOR STRENGTH EVALUATION :
      BLy  = 3.54331e+001,  BLz  = 3.54331e+001,  BLu  = 8.85827e+000
      Ky   = 1.00000e+000,  Kz   = 1.00000e+000

   *. MATERIAL PROPERTIES :
      Fy   = 5.18400e+003,  Es   = 4.17600e+006,  MATERIAL NAME  = A36

   *. FORCES AND MOMENTS AT (I) POINT :
      Axial Force        Fxx  = 0.00000e+000
      Shear Forces       Fyy  = 0.00000e+000,  Fzz  =-8.98893e+001
      Bending Moments    BMy  =-6.60487e+002,  BMz  = 0.00000e+000
      Moments of i-node  Myyi =-6.60487e+002,  Mzzi = 0.00000e+000
      Moments of j-node  Myyj =-5.90083e+002,  Mzzj = 0.00000e+000

   ( ). Compute moment magnification factors(B1y,B1z).
        -. This column is a tension member.
        -. Assumed   B1y  =  1.00
        -. Assumed   B1z  =  1.00

   ( ). Magnification factors for sidesway moments(B2y,B2z).
        -. B2y   =  1.00 (Default value)
        -. B2z   =  1.00 (Default value)

   ( ). Given factored axial forces and moments at <I>. Unit : kip., ft.
        -----------------------------------------------------------
        Load Case            Pu              BMy              BMz
        -----------------------------------------------------------
        DL                 0.00          -339.10             0.00
        LL                 0.00          -321.30             0.00
        DL+LL              0.00          -660.40             0.00
        WL or EL           0.00             0.00             0.00
        -----------------------------------------------------------
        DL+LL+WL(EL)       0.00          -660.49             0.00

   ( ). Compute magnified moments.
        -. BMuy  =  B1y*BMy(DL+LL) + B2y*BMy(WL(EL)) =     -660.49 kip-ft.
        -. BMuz  =  B1z*BMz(DL+LL) + B2z*BMz(WL(EL)) =        0.00 kip-ft.

   ( ). Factored max. shear forces.
        -. Vuy   =        0.00 kip.
        -. Vuz   =       [illegible] kip.

================================================================================
 [[*]]   CHECK FLEXURAL STRENGTH ABOUT MAJOR AXIS.
================================================================================

   ( ). Compute section properties for AISC-LRFD design.
        -. Torsional constant (J)  =       0.0003 ft^4.
        -. Warping constant (Cwp)  =       0.0055 ft^6.
        -. Section moduli  (Syy,Szz)
           Syy  =        0.1418 ft^3
           Szz  =        0.0153 ft^3
        -. Plastic section moduli (Zyy,Zzz)
           Zyy  =        0.1620 ft^3
           Zzz  =        0.0240 ft^3

   ( ). Compute plastic bending moment (Mp).
        [ AISC-LRFD2K Specification F1.1. (F1-1) ]
        -. Mp  =  Fy*Zyy  =        840.00 kip-ft.

the Most Intelligent Design & Analysis System
Print Date/Time : 07/10/2002 15:5[illegible]
- 2 / 5 -
```

Detail calculations of strength verification for a steel member



Summary calculations of strength verification for a steel member

MIDAS/Gen **RC Beam Design Result Output**

PROJECT TITLE:

	Company	MIDAS IT Co.,Ltd.	Client	
	Author		File Name	Untitled.rcs

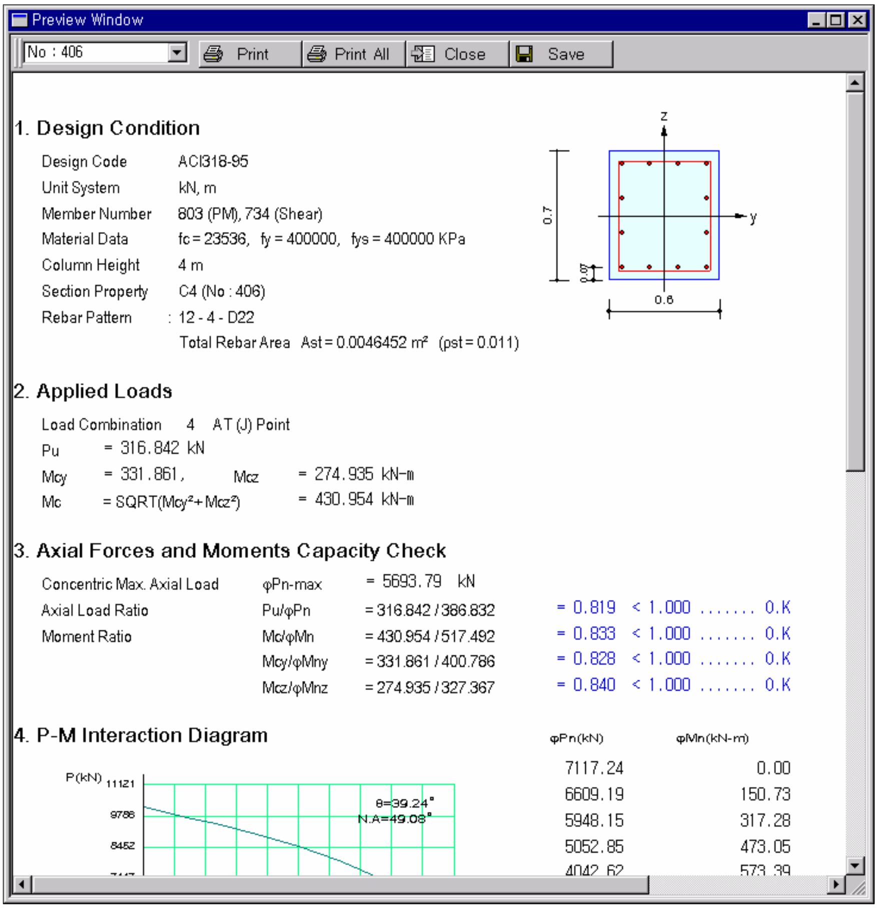
```
-----------------------------------------------------------------------
 MIDAS/Gen - RC-Beam Design        | ACI318-99 |              Version 5.8.0
=======================================================================

    *.MIDAS/Gen - RC-BEAM Analysis/Design Program.

    *.PROJECT       :
    *.DESIGN CODE   : ACI318-99,       *.UNIT SYSTEM : tonf, m
    *.MEMBER        : Member Type = BEAM,   MEMB =    2

    *.DESCRIPTION OF BEAM DATA (iSEC = 211) : G1
      Section Type : Rectangle (RECT)
      Beam Length (Span)          =       10.200 m.
      Section Depth (Hc)          =        0.700 m.
      Section Width (Bc)          =        0.400 m.
      Concrete Strength (fc)      =     2400.000 tonf/m^2.
      Main Rebar Strength (fy)    =    42184.188 tonf/m^2.
      Stirrup Rebar Strength (fys) =   42184.188 tonf/m^2.
      Modulus of Elasticity (Es)  = 20389024.167 tonf/m^2.

    *.DESCRIPTION OF APPLIED FACTORS FOR DESIGN/CHECKING.
      Special Provisions For Seismic Design.
      Scale Up Factor EQ Load case for Shear Forces    = 2.000

    *.FORCES AND MOMENTS AT CHECK POINT <I> :
      Positive Bending Moment   P-Mu =       8.81 tonf-m.,  LCB =   4
      Negative Bending Moment   N-Mu =      81.88 tonf-m.,  LCB =   1
      Shear Force                 Vu =      18.88 tonf.  ,  LCB =   1

    *.REINFORCEMENT PATTERN :
      ------------------------------------------------------------
        Location     i     di( m.)      Rebar    Asi( m^2.)
      ------------------------------------------------------------
        Top          1     0.070        3-D25      0.00152
        Bottom       2     0.630        2-D22      0.00077
      ------------------------------------------------------------
        Stirrup : D10

 ===================================================================
   |||*|||   ANALYZE POSITIVE BENDING MOMENT CAPACITY.
 ===================================================================

    ( ). Compute design parameter (beta1).
        -. beta1  = 0.8500  ( fc < 4000 psi.)

    ( ). Compute required ratio of reinforcement.
        -. Rhomi1 = MAX[ 3*SQRT[fc]/fy, 200/fy ]                =   0.0033
        -. Rhomi2 = (4/3)*Mu/[ fy*b*d1*(d1-a/2) ]               =   0.0016
        -. Rhomin = MIN[ Rhomi1, Rhomi2 ]                       =   0.0016
        -. Rhob   = 0.85*beta1*(fc/fy)*(0.003/(0.003+fy/Es)) =   0.0243
        -. Rhomax = 0.75*Rhob                                   =   0.0182

    ( ). Check ratio of tensile reinforcement.
        -. Rho = As/(bw*d) =    0.0031
        -. Rhomin < Rho < Rhomax  ---> O.K !
```

Modelling, Integrated Design & Analysis Software
http://www.midasIT.com
MIDAS/Gen V 5.8.0 LAN

Print Date/Time : 02/10/2003 15:39

- 3 / 11 -

Detail calculations for a RC beam member



Summary calculations of automatic design for a RC column

Strength Verification for Steel Members

The procedure for verifying strength for steel members is as follows. The user may specify all the members or select only a few members in the steel structure model for checking member strength.

1. Specify selectively design parameters to be used in the design from the sub-menus in ***Design>General Design Parameter***.

   Frame system
   Live load reduction factor
   Unbraced length or laterally braced length
   Effective buckling length factor
   Moment factor
   Moment magnification factor
   Member type

2. Specify selectively design parameters to be used in the design from the sub-menus in ***Design>Steel Design Parameter***.

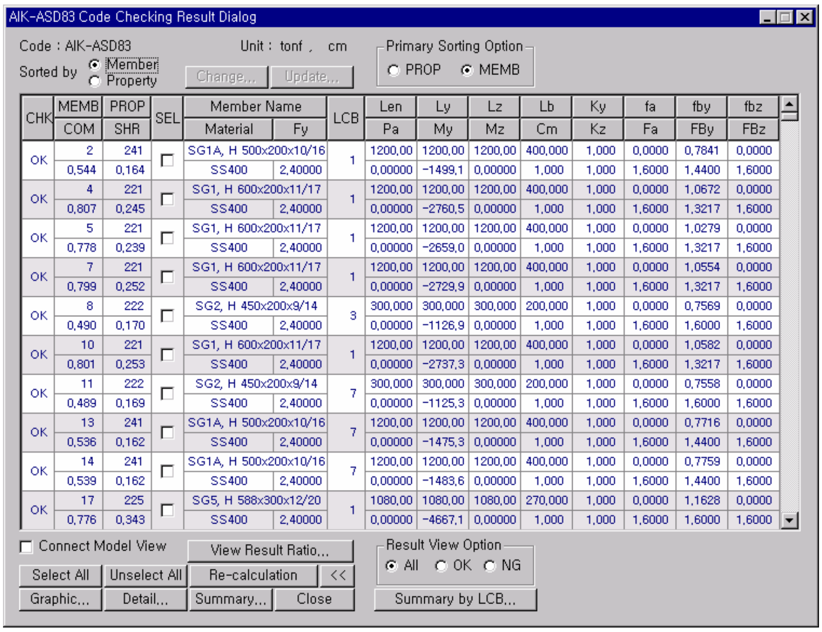
   Design standard
   Material properties
   Bending coefficient
   Shear coefficient
   Allowable stress
   Allowable maximum slenderness ratio
   Longitudinal stiffeners of box section

3. Verify the strength by selecting ***Design>Steel Code Check***.

4. The design results will appear on the screen after completing the strength verification.

5. Using the Change button in the results dialog box that contains the output results, the strength may be verified by specifying new section data for each section type. The modified section data may be reflected on the analysis model by clicking Update... . Then, the analysis and design results are automatically removed. As the modified section data change the structure's stiffness, the analysis and strength verification have to be performed once again to obtain appropriate design results corresponding to the modified model data.

Refer to "***Steel Code Check***" section of ***On-line Manual*** for further detail.



Dialog box for the strength verification results of steel members

Optimal Design of Steel Frame Members

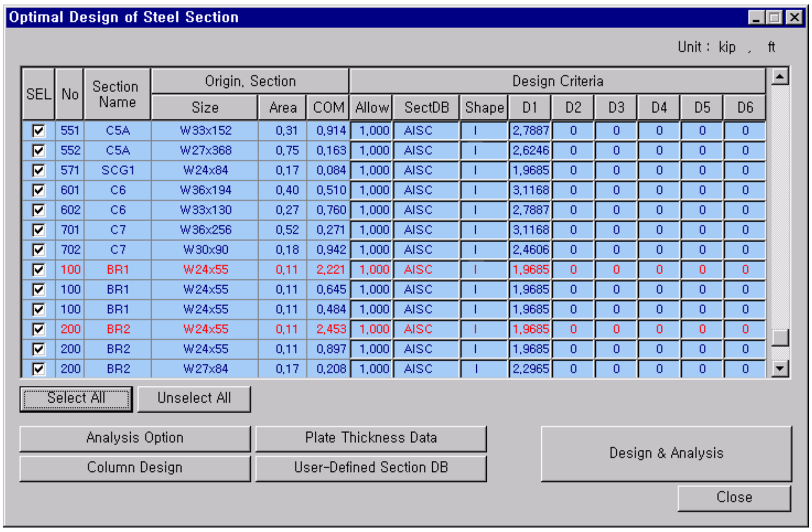
The optimal design feature of **MIDAS/Gen** optimizes the member sections, which determines the section dimensions automatically for the minimum sectional areas (minimum weights) satisfying the steel design standard and criteria specified by the user. In the optimal design process, the optimal section is determined by considering all the design parameters used for the strength verification process such as the design load combinations, section shape, unbraced length, lateral braced length, effective buckling length factor, bending coefficient, moment coefficient, yield strength of material, etc.

The optimal design procedure is as follows:

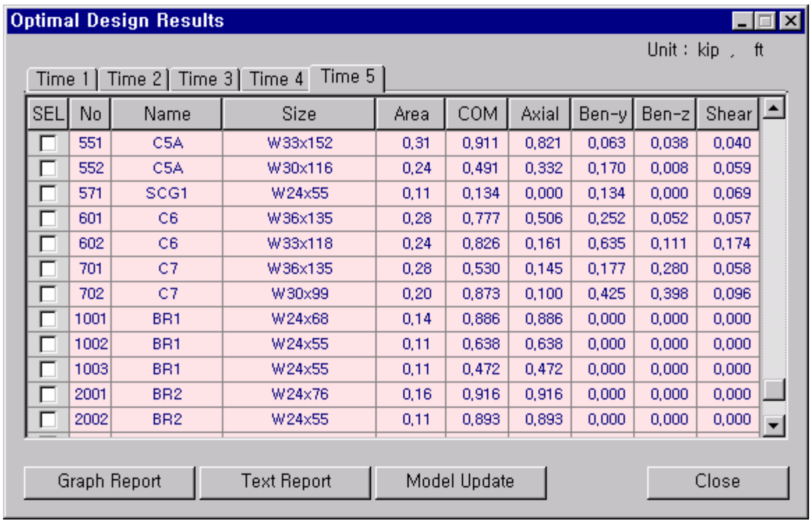
1. Verify strength for steel members.
2. Specify the design constraints for each section property type for optimal design in ***Design>Optimal Design***.
3. Enter the number of iterations for optimal design and re-analysis.
4. Examine the results using ***Graphic Output*** for evaluating the optimal design.

A significant number of iterations may ensue during the optimal design process depending on the design conditions. The user is urged to limit the number of iterations to a reasonable number.

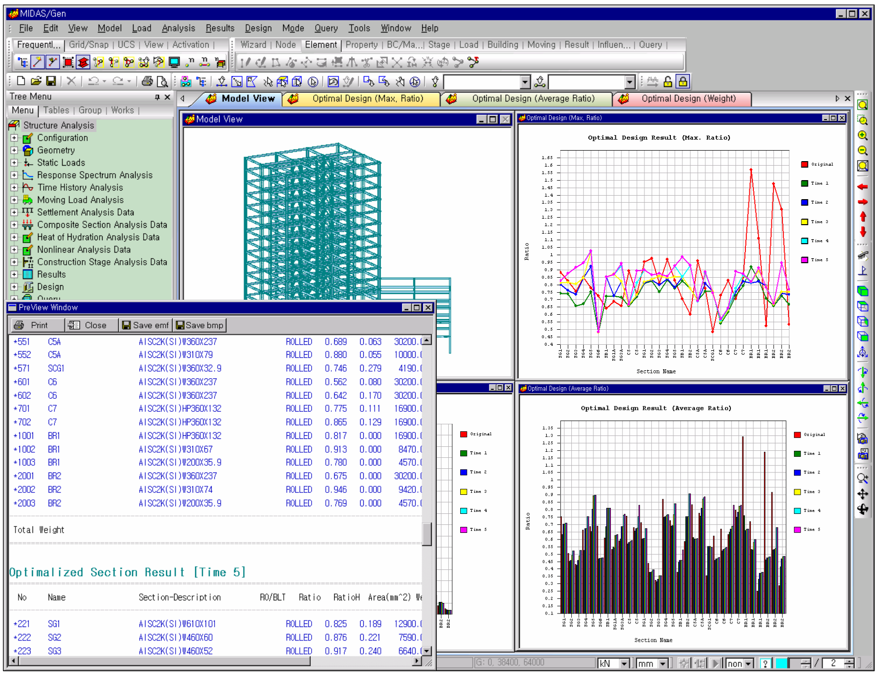
Refer to "***Optimal Design***" section of ***On-line Manual*** for more detailed information.



Enter design constraints for optimal design



Display of optimal design results



Display of optimal design graphic output

Design of RC Members

The RC (Reinforced Concrete) design feature of **MIDAS/Gen** designs the sections and verifies strength for either all RC members or only a few selected members.

The section design or strength verification may be performed selectively as follows:

- Section Design (calculation of required rebar quantities and automatic rebar placement)

  The section design feature calculates the required optimal rebar quantities and provides rebar placement by applying the factored loads based on the design load combinations of RC members and section dimensions specified or revised by the user. In other words, the section design feature is applicable where only the section dimensions exist without the reinforcing data.

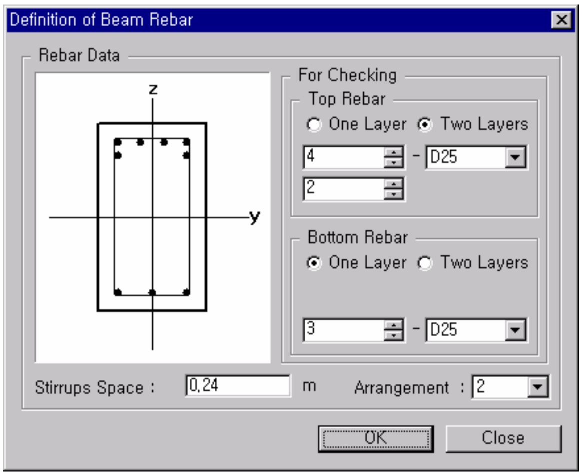
➢ Strength Verification

MIDAS/Gen assumes a member as a complete RC section when the user enters both section dimensions and rebar placement data. Only then, are the capacity calculated and the result compared to the design force. The strength (capacity) is calculated if all the required data for the section composition are provided, or else, the section will be designed.

Make-up of a RC section
Section shape and dimensions
Sizes of rebars
Number of rebars
Positions of rebars

The procedure for section design and strength verification for RC members is as follows:

1. Enter selectively the design parameters to be used for the section design or strength verification from the sub-menus in ***Design>General Design Parameter***.

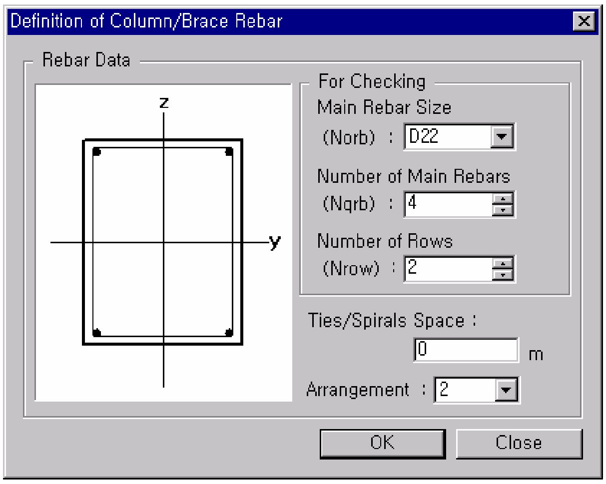


Section data entry for a beam member

Frame system
Live load reduction factor
Unbraced length or laterally braced length
Effective buckling length factor
Moment factor
Moment magnification factor
Member type

2. Enter selectively the design parameters to be used for the section design or strength verification from the sub-menus in ***Design>Concrete Design Parameter***.

Design standard
Strength reduction factor
Material properties
Limit for the maximum rebar ratio
Sizes of rebars and design method for shear walls
Enter or modify beam section data
Enter or modify column section data
Enter or modify bracing section data
Enter or modify shear wall section data
Enter or modify shear wall mark



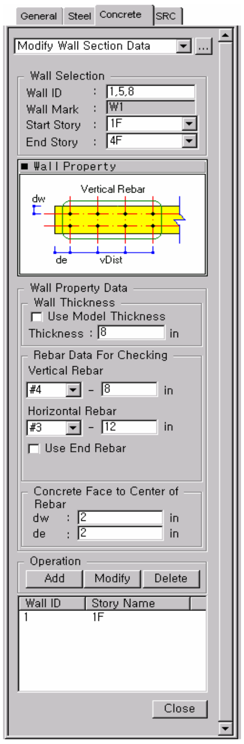
Section data entry for a column member

3. Design the sections or verify strength by using the sub-menus classified by the types of members in ***Design>Concrete Code Design*** as noted below.

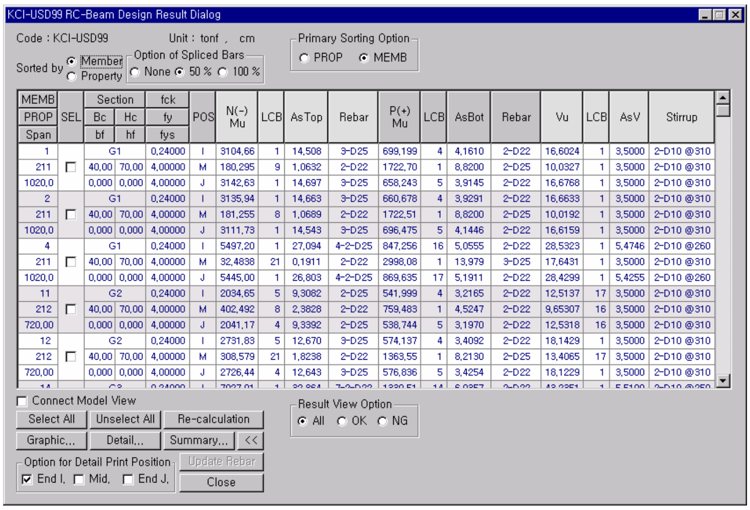
Design beam members	***Design***
Design column members	***Design***
Design bracing members	***Design***
Design shear wall members	***Design***
Strength verification for beam members	***Beam Checking***
Strength verification for column members	***Column Checking***
Strength verification for brace members	***Brace Checking***
Strength verification for shear wall members	***Wall Checking***

4. After completing the design of sections for each member type, check the section design and strength verification results displayed on the screen.

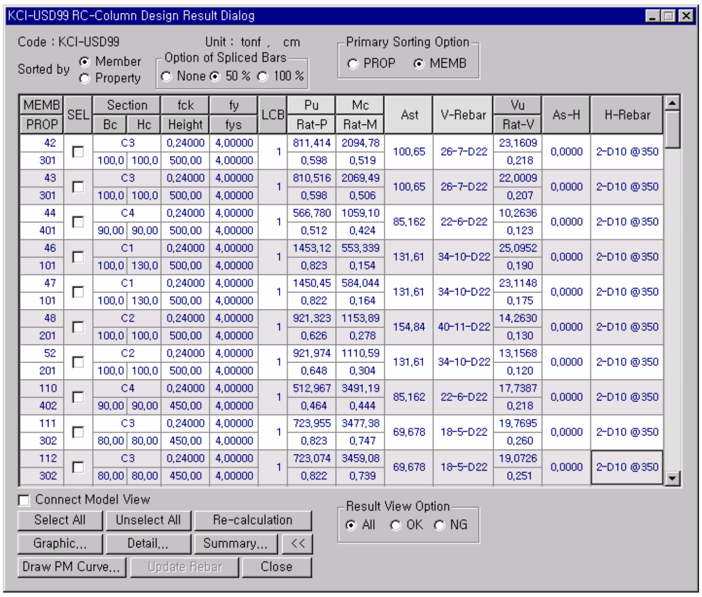
Refer to "***Concrete Code Design/Check***" section of ***On-line Manual*** for further information.



Section data entry for a shear wall member



Display of beam member section design results



Display of column member section design results

Design of Footings

As the footings are generally placed at the nodes with restrained degrees of freedom due to the boundary conditions, the user may execute the footing design using the reactions at these nodes. The spread footings and pile caps with square or rectangular shapes in plan may be designed.

The section design or the strength verification for footings may be performed selectively as follows:

- ***Section Design***

  MIDAS/Gen provides the optimal footing size, number of piles, footing thickness and required rebar quantities with respect to the reactions obtained from the analysis process and the design constraints specified by the user.

The most effective procedure for footing design starts by obtaining a design through the automatic design feature and then continues with revising the design by accommodating user preferences and strength verifications.

- ***Strength Verification***

  MIDAS/Gen examines the suitability of footings for the reactions obtained from the analysis process and the design constraints specified by the user. It calculates the required rebar quantities using the calculated reactions.

The user may specify the design load combinations directly in ***Results> Combinations*** or use the combinations automatically generated in accordance with the applied design standard. The service load combinations for calculating the footing sizes and pile quantities, and the factored load combinations for calculating the footing thicknesses and required rebars, may be applied by clicking [Select Load Combination] in the ***Footing Design*** dialog box.

The design parameters for footing design may be entered or modified by using the ***Footing Design*** dialog box. For those unspecified design parameters, the initial default values are used for section design or strength verification.

The design procedure for footings is as follows:

1. Enter the design load combinations in the ***Footing Design*** tab selected from ***Results>Combination***.

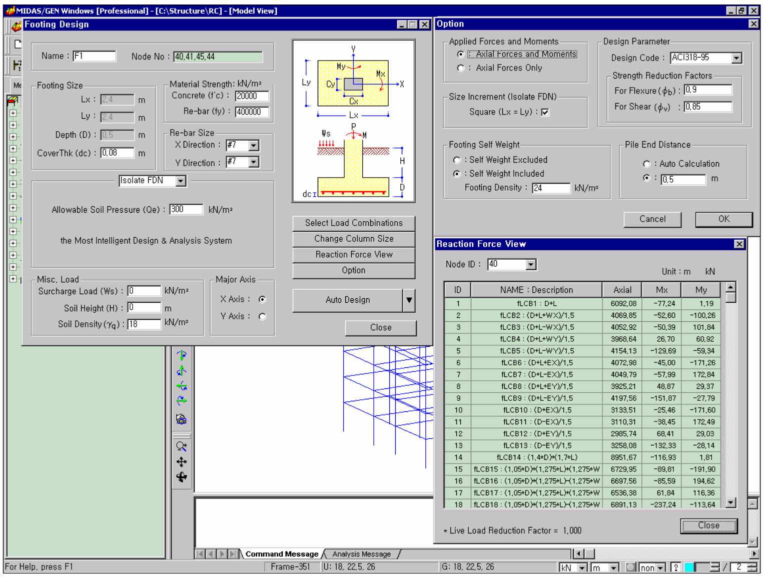
2. Enter the node numbers where the footings will be placed or click them on the screen displayed from ***Design>Footing Design***.

3. Enter the required design parameters for footing design and ensure the accuracy of the service load and factored load combinations.

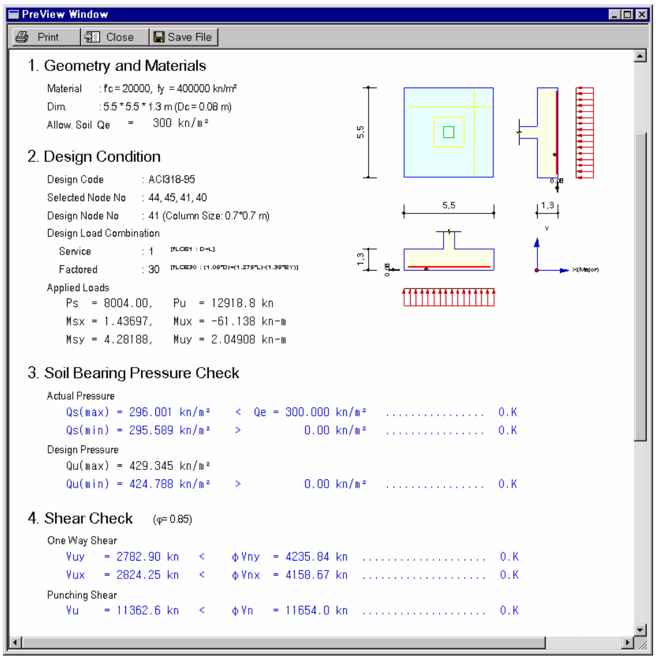
4. Click Auto Design for automatic design or click Code Check for strength verification.

5. The design results will appear on the screen when the design (strength verification) is completed.

6. To design other footings, repeat the steps 2 to 5.



Entry of footing design parameters



Display of footing design results

Strength Verification and Optimal Design of SRC Members

SRC (Steel-Reinforced Concrete) section shapes for which the strengths may be verified are as follows:

- Steel encased in a rectangular concrete section (concrete filled or unfilled in the steel section)
- Steel encased in a circular concrete section (concrete filled or unfilled in the steel section)
- Rectangular steel section filled with concrete
- Circular steel section filled with concrete

The types of steel sections encased in concrete may be an H(I)-section, a rectangular section, or a circular section. Both rolled and built-up sections may be used. Composite sections must be symmetrical about both ECS y and z-axes.

The section design procedure for SRC members is as follows:

---

1. Enter selectively the design parameters to be used for the design from the sub-menus in ***Design>General Design Parameter***.

   Frame system
   Live load reduction factor
   Unbraced length and lateral braced length
   Effective buckling length factor
   Moment factor
   Moment magnification factor
   Member type

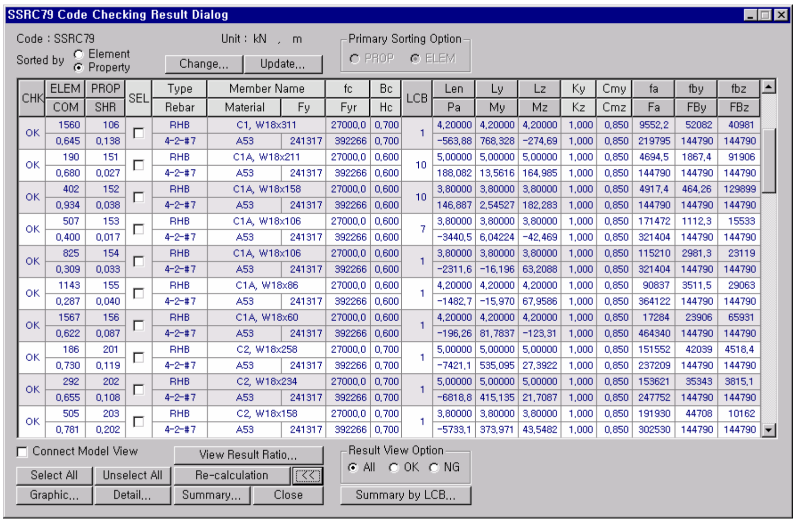
2. Enter selectively the design parameters to be used for the design from the sub-menus in ***Design>SRC Design Parameter***.

   Design standard
   Modify SRC material
   Enter of modify SRC section data

3. Verify strength by selecting ***SRC Code Check***.

4. The strength verification results will appear on the screen when the strength verification is completed.

Refer to "***SRC Code Check***" section of ***On-line Manual*** for further information.



Section design results for SRC columns

The optimal design, for SRC members, searches for optimal steel sections without varying the RC sections. The remaining features are similar to that in the optimal design for steel members.

Production of Output

Text Output

The ***Text Output*** provides the structural analysis and design results in a text file format specified by the user.

The principal features related to the text output of **MIDAS/Gen** are as follows:

- Produce output for (Load Sets) individual groups of load combinations (it is possible to assign different load combinations for different output contents).
- Produce selective output of member forces and stresses pertaining to specific material properties, section properties, element numbers, etc.
- Produce the output of maximum and minimum values for each section property.
- Produce nodal displacements and reaction forces.
- Produce the output of ***Envelope*** and ***Summary*** for each element type.
- Produce the output in ECS or GCS of each element.

Directions and Procedure of Usage

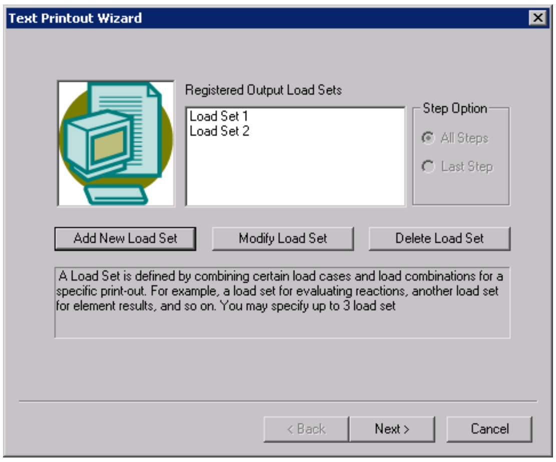
The ***Text Output*** features operate on a method that accumulates the required information on a step-by-step basis for the final outcome. The procedure for using the ***Text Output*** features is as follows:

Example: A load set for the evaluation of reactions, another load set for element output, and so on. Up to 3 load sets at a time may be specified.

---

1. Assign ***Load Set.***

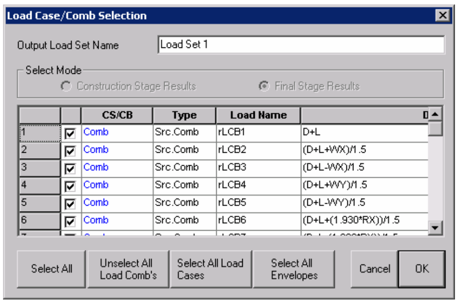
   Select ***Results>Text Output*** to display the load combination selection dialog box for text output.



Load set selection dialog box

A ***Load Set*** is a collection of load cases/combinations for the desired output items for different results such as member forces, nodal displacements, reaction forces, etc. The user selects as many Load Sets as necessary in the load combinations selection dialog box.

Define the method of producing output for steps in **Step Option** where construction stage analysis or geometric nonlinear analysis has been carried out. Click Add New Load Set, then the ***Load Set*** entry dialog box shown in the next figure will be displayed.



Load Set entry dialog box

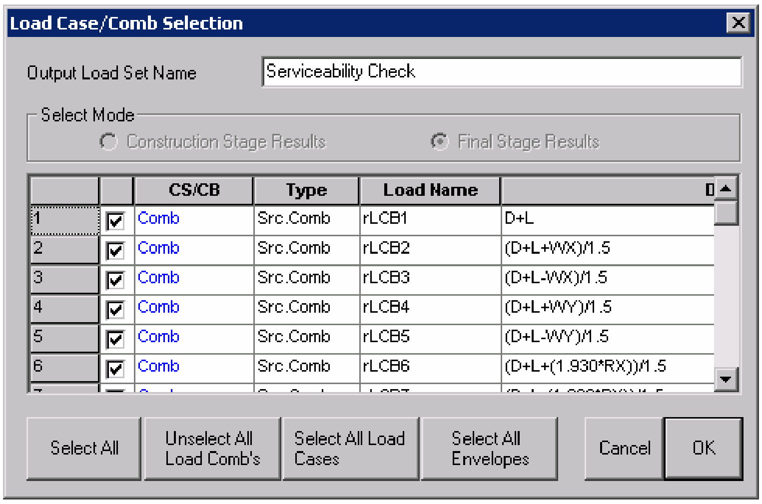
Enter the ***Load Set*** name. A load set is registered when the desired load cases and/or load combinations are selected (checked) and OK is clicked. Click Modify Load Set to modify the contents of a Load Set and click Delete Load Set to remove a registered Load Set.

Once all the necessary load sets are defined, click Next > and access the ***Element Output Selection*** dialog box.

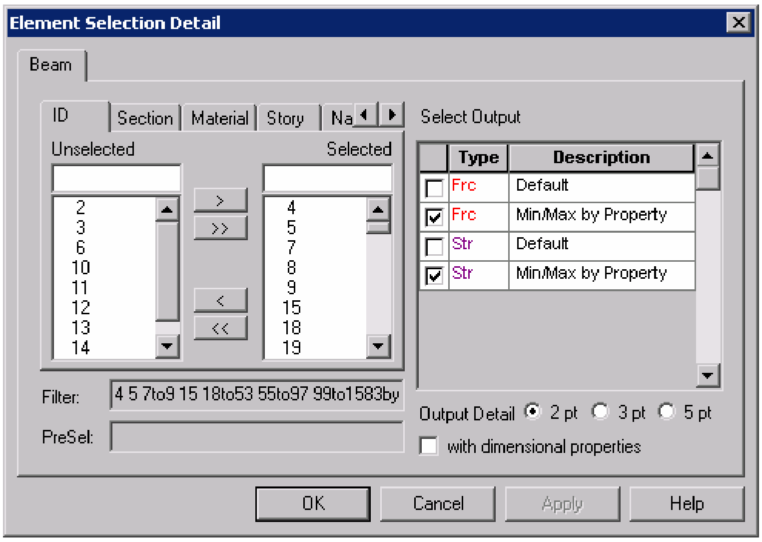
2. Select Elements for Output.

In the dialog box, assign the elements for output and select the output format. In ***Output Load Set for Element Output***, select the ***load set*** for which element output will be produced among the registered load sets. Select the element types for which output will be produced by checking appropriate boxes. At this time, only the elements for which the output can be produced are activated in the dialog box.

By clicking the ... button to the right of the element type, detail specifications related to the element output may be selected.



Element Output Selection dialog box



Detail Output Selection dialog box

Two parts constitute the ***Element Selection Detail*** dialog box. The left section filters the selected items, and the right section assigns the output format and other items.

Only the output for the elements conforming to the selected attributes among the filter items, ***ID, Section, Material, Story, Named Plane and Group*** will be finally produced.

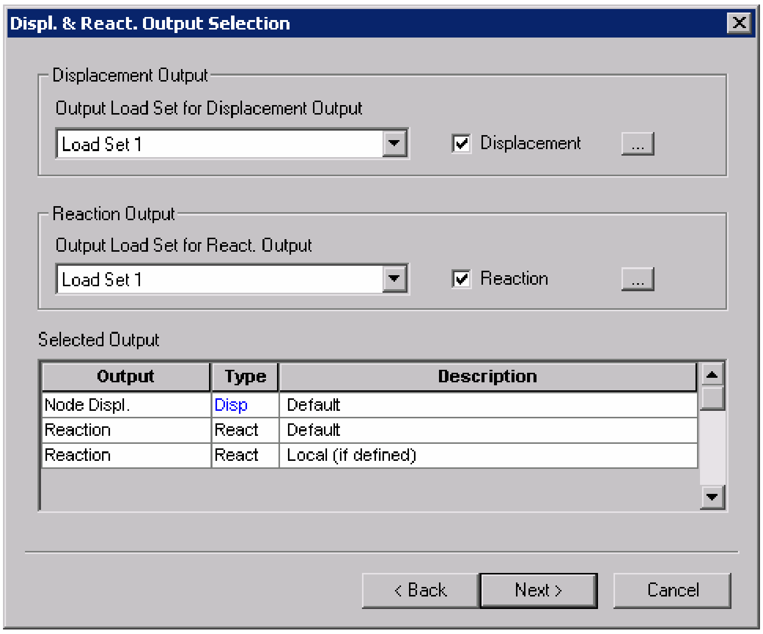
Filter : list of elements selected through the filter

PreSel : list of elements already selected on the screen prior to starting the ***Text Output*** feature

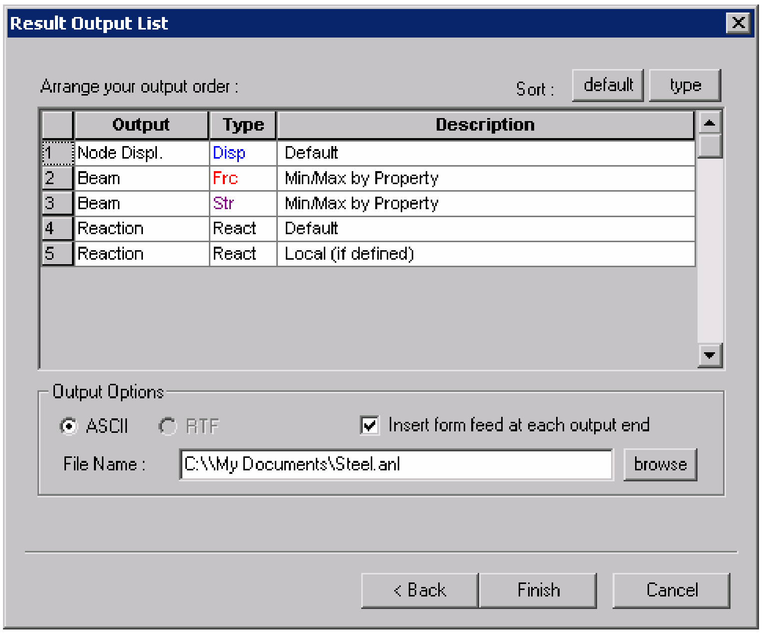
The ***Text Output*** features operate on all the elements listed in the ***Filter*** and ***PreSel*** fields.

3. Select Output for Nodal Displacements and Reaction Forces.

After the selection for Element Output is completed, click the Next > button to switch to the dialog box for output specifications for displacements and reaction forces. The usage of this dialog box is identical to that of Element Output Selection.



Output Selection dialog box for displacements and reactions



Dialog box for items of results output

4. Specify the sequence of output.

   Finally, specify the sequence of output and the output file name.

   It is possible to arrange the output sequence by Default or by Type. Select and drag the items individually with the mouse to modify the sequence.

   If ***Insert form feed at each output end*** is checked, a page form feed character ("♀") is inserted at the end of each output item. Type the name and path of the output file in the ***File Name*** field and click the Finish button to create the file. Text Editor is executed automatically and the file is displayed on the screen.

Print Output

MIDAS/Gen provides a collection of format choices for print outputs for user convenience. **MIDAS/Gen** prints output in a vector or in an image format.

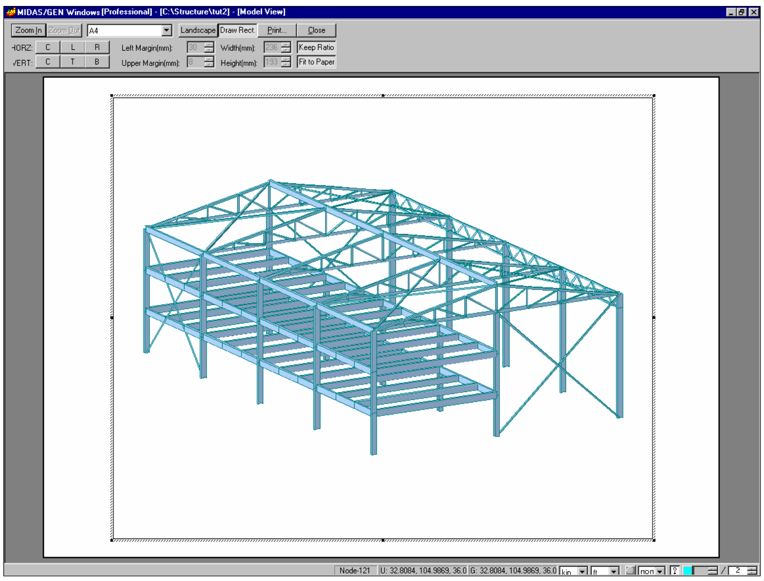
When the model window in preprocessing or post-processing mode is printed, the output is generated in a vector format. The output results provide uniform quality irrespective of output sizes.

If the screen containing a rendering view is printed, the output is printed in an image format. Due to the characteristics of image output, the quality of the print output is determined by the resolution and the number of colors used in the window. The size of the output also affects the quality.

Output Layout Setting

MIDAS/Gen provides the ***Print Preview*** feature that enables us to adjust the size and position of the output before printing.

Select ***File>Print Preview*** or click ***Print Preview***. Then, the **Print Preview** window is displayed.



Print Preview window

The dialog bar at the top of the screen is used to adjust the size and position of the output before printing.

Clicking the buttons of each item with the mouse can set up a rough Layout, while specifying numbers in the Margin fields within the dialog bar can adjust it to a more precise layout.

Dialog bar for Print Preview

The following explains the dialog bar:

Zoom In, Zoom Out	Magnify or reduce the view, which has no effect on the true output.
Combo Box	Select Paper Size.
Landscape/Portrait	Horizontal or vertical printout
Draw Rect	Border line insertion option
HORZ	Alignment (justified) to Center, Left & Right
VERT	Alignment (justified) to Center, Top & Bot.
Keep Ratio	Option to maintain horizontal/vertical ratio when changing the printout size
Fit to Paper	Fit the contents to the selected paper size. Selecting ***Fit to Paper*** disables Margins/Sizes
Print	Resume printing.

Output Color Setting

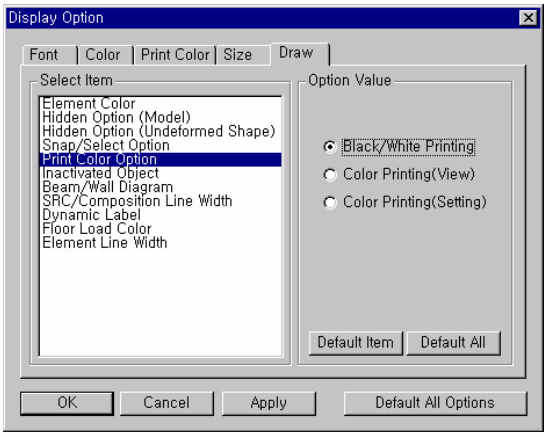
MIDAS/Gen provides both color and black-and-white printing options for user convenience. By setting the ***Black/White Printing*** option, the object is printed in black and white based on the set up in **MIDAS/Gen** in lieu of printing the current colors of the working window.

The ***Color Selection Option*** is independent of the printer types, and it may be freely set according to the user's intent.

The method of setting output color is as follows:

Select ***View>Display Option*** or ***Display Option***, then the dialog box shown in the figure below will be displayed. The ***Draw*** tab displays the dialog box that defines the color selection method. Select ***Print Color Option*** and set the print option in ***Option Value*** as shown in the figure below.

Among the Color Print Options, ***Color Printing (View)*** produces the contents in the window colors, and the colors may be selected in ***Color*** tab from the ***Display Option*** dialog box. ***Color Printing (Setting)*** is adjusted in ***Print Color*** tab from the ***Display Option*** dialog box, and the colors in the model window and the output may be set independently.



Display Option dialog box

Text Editor

Principal Features of Text Editor

MIDAS Text Editor works together with the **MIDAS Family Program** as a document editor that conveniently edits relevant input/output text files.

In Windows environment, the Text Editor may be used as a common text editor that provides the basic editing features such as compose, save and print text documents (may be used as a substitute for Windows memo pad).

To run ***MIDAS Text Editor***, execute ***tedit.exe*** in the program folder of **MIDAS/Gen**, or select ***Tools>Text Editor*** from the Main Menu of **MIDAS/Gen**.

The basic functions of ***Text Editor*** are as follows:

- Create and edit document files
- Search function, and header and footer inserts
- Insert page split (♀)
- Print layout setting
- Preview print output

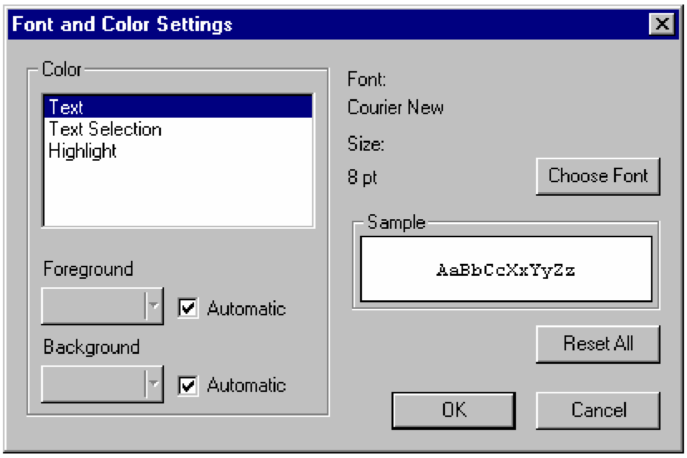
Document Output Using Text Editor

When a new document has been composed or a document has been loaded in the editor by the ***Text Output*** function of **MIDAS/Gen**, the document may be edited and printed.

Font Type and Size Setting

Selecting ***View>Configure*** menu or ***Configure Language*** displays the dialog box shown below.

The desired font and size may be specified by clicking the Choose Font button on the right of the dialog box. ***Text Editor*** supports a limited number of font types with fixed pitch.



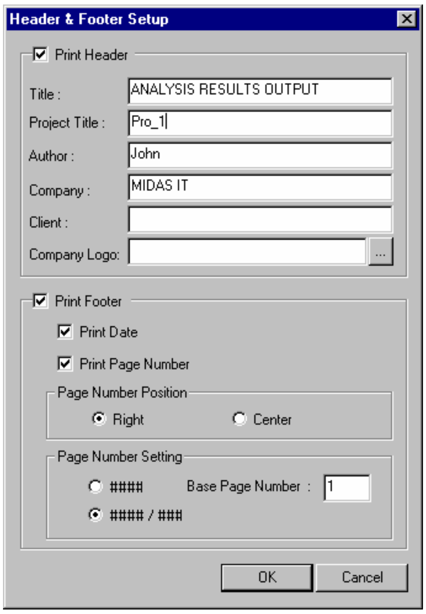
Dialog box for font and color settings of Text Editor

Page Split

When a new page is desired at a specific line on the page, place the cursor at the desired position and press ***Page Split***. If the page split character " ♀ " is inserted at the position of the mouse cursor, the page is automatically divided at the position of the page split character for printing. The " ♀ " character does not appear on the printed sheets.

Header and Footer Insertion

Selecting ***File>Header & Footer Setup*** menu or clicking ***Heading Footing*** displays the dialog box shown below.



Dialog box for header and footer insertion

Check ***Print Header*** and fill in the entry fields to print the header.

Check ***Print Footer*** to print the footer with the page number and date.

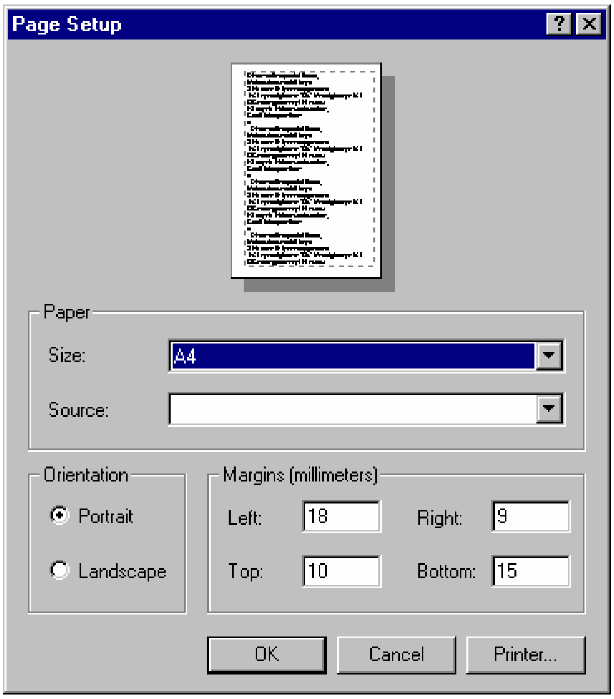
The ***Page Number Position*** option selects the position at which the page number will be printed. The position is either in the middle or on the right at the bottom of the page.

The Base Page Number is literally the first page (page 1) of the document from which the page numbering starts sequentially.

The ***Page Number Setting*** option defines the numbering style.
####: Print the page number beginning with the Base Page Number.
####/####: Print the current page number and the total number of pages.

Page Setup

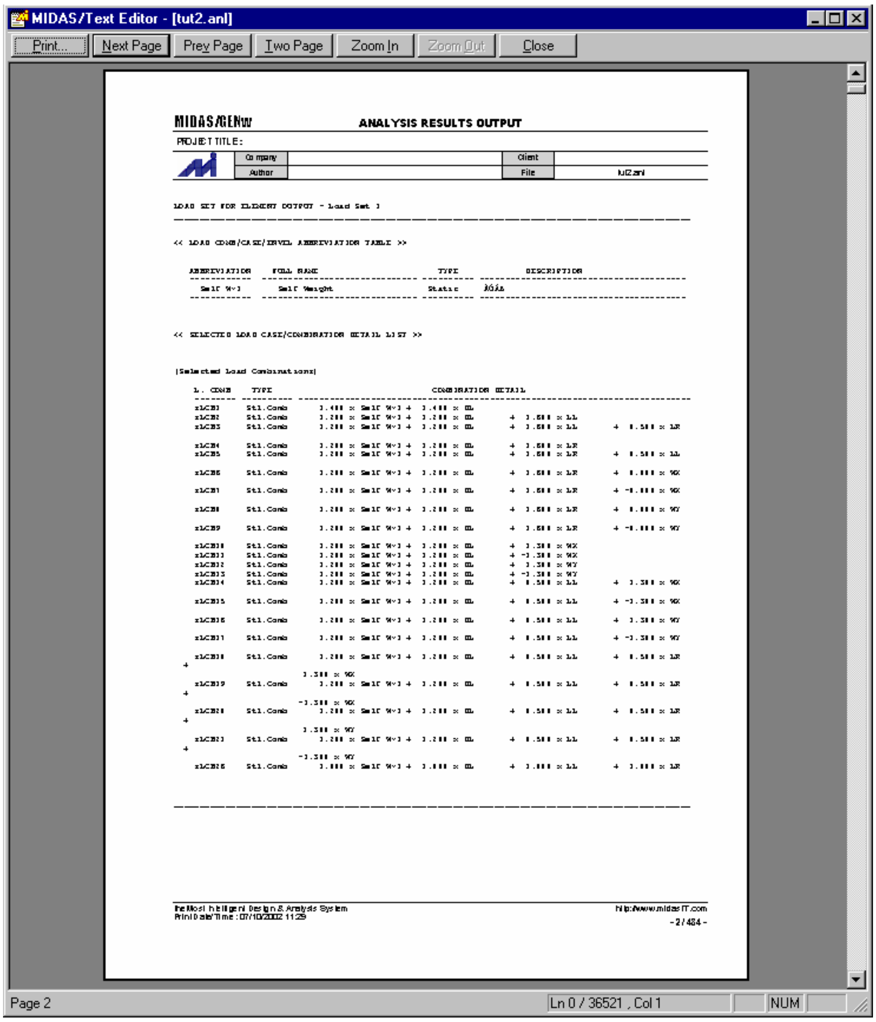
Selecting the ***File>Page Setup*** menu displays the ***Page Setup*** dialog box. This dialog box defines the size of printed forms, the orientation and the margins.



Page Setup dialog box

Print Preview

When all the print settings are complete, it is advisable to verify the layout of the print settings. Select the ***File>Print Preview*** menu or click . Once the print settings are verified, start printing by clicking the button.



Print Preview window

Graphic Editor

Principal Features of Graphic Editor

MIDAS Graphic Editor works together with the **MIDAS Family Program.** It is a vector-based graphic editor program that edits and prints various graphic files.

Various titles and comments may be added to the graphic documents with the BMP or EMF (Enhanced Metafile) extensions that MIDAS/Gen created. Such editing capabilities provide high quality documents for reports or presentation materials.

In order to execute ***MIDAS Graphic Editor***, execute ***gedit.exe*** in the program folder of **MIDAS/Gen** or select ***Tools>Graphic Editor*** from the Main Menu of **MIDAS/Gen**.

The principal features of ***Graphic Editor*** are as follows:

- Drawing various images
- Various editing functions
- Importing external files (BMP, EMF)
- Saving files in BMP & EMF formats or in its inherent type
- Print Layout and Print preview functions

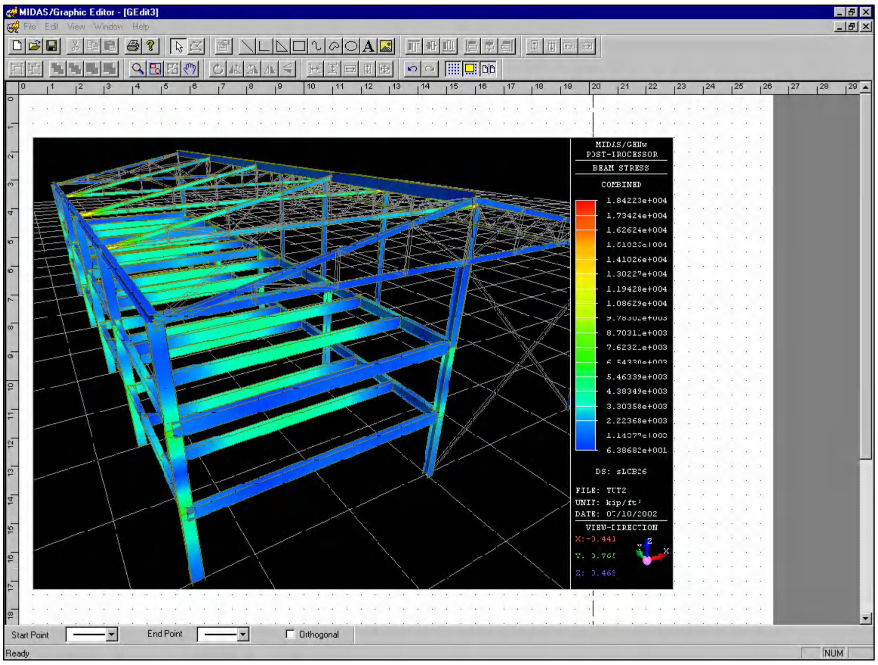
Usage

Refer to the ***Graphic Editor*** section of the ***On-line Manual*** for further details regarding the image and editing functions of the ***Graphic Editor***.

Open an Image File

This opens graphic files (BMP, EMF) created by **MIDAS/Gen**.

- ***Open***
  Click to display the dialog box. After selecting the file format (BMP, EMF), move the file to the desired folder. Select a file name and click the Open button.



Open Graphic File view

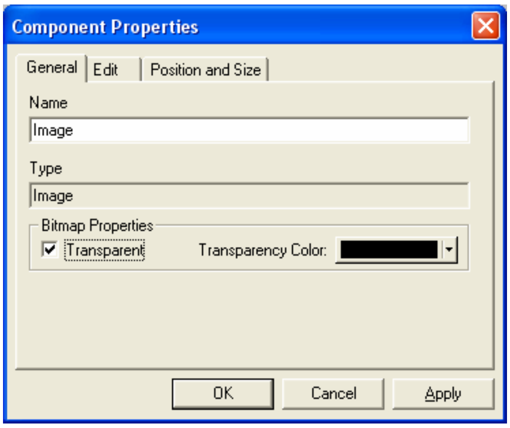
➢ ***Insert Image***
If the cursor is in a stand-by state for image insertion, move the cursor to the desired position and insert the image by left-clicking the mouse.

➢ ***Adjust Size and Position***
Adjust the position of the image by holding and dragging the center of the image with the mouse. Adjust the size of the image by dragging a corner.

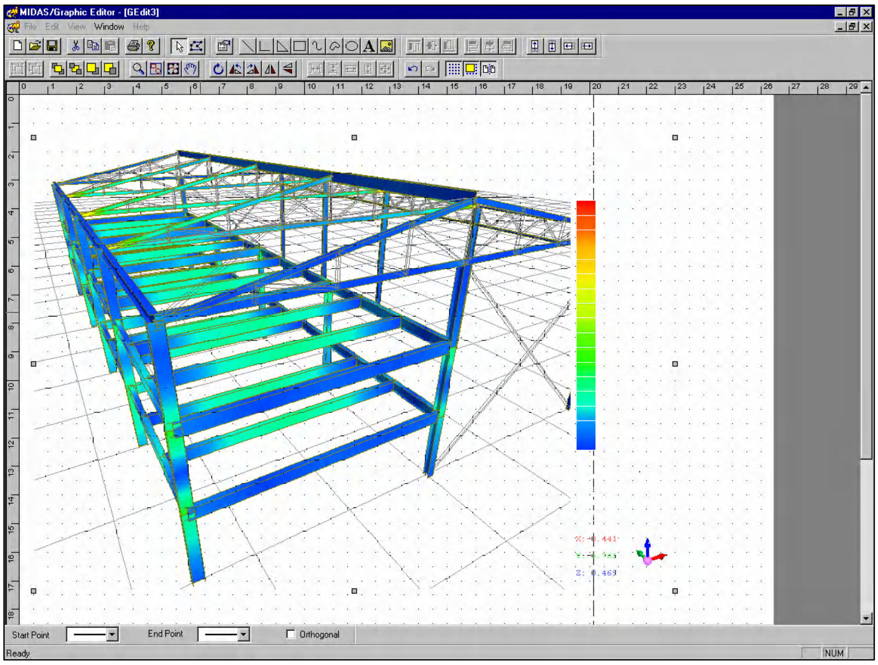
Create Image Setting and Add Title

➢ ***Transparent Color Setup***
This is a tool that makes the desired color transparent. It is very useful when printing an image with a black background.

Select an opened image by clicking the image once and right-click the mouse. Then select ***Component Properties.*** The dialog box shown in the figure below is displayed. Check ***Transparent*** under ***Bitmap Properties*** in ***General*** and select the black color, then the background becomes transparent.



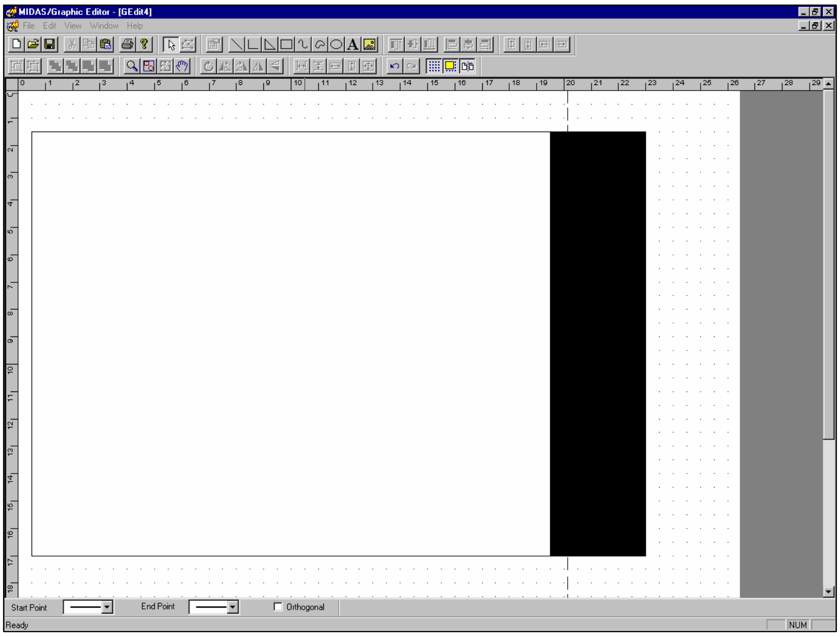
Component Properties dialog box



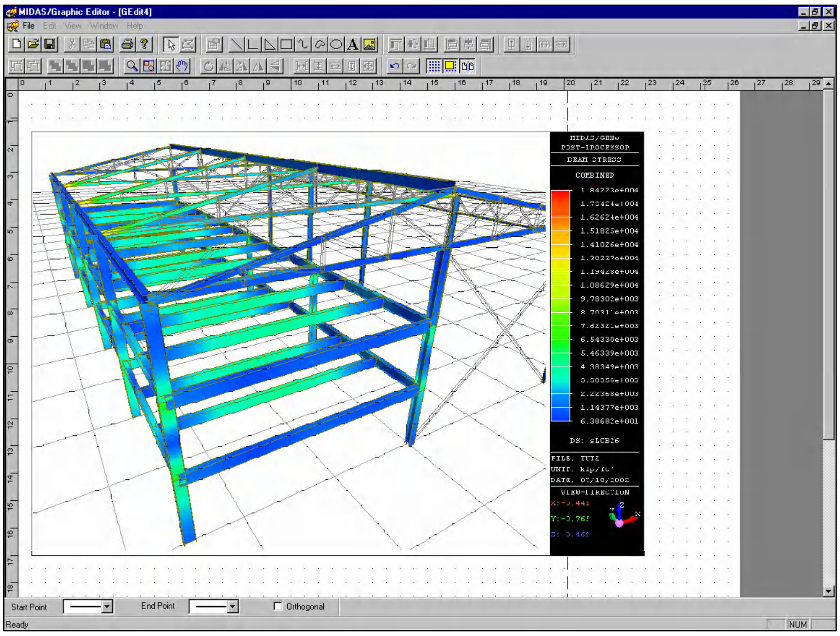
Example of black background changed to transparent color

- ***Image Framework***
  The image framework may be defined by clicking ***Rectangle***. After selecting each rectangle, right-click the mouse to open the Context Menu. Select ***Properties*** in the Context Menu, then the thickness and color of the lines or the color of the face may be adjusted by the ***Component Properties*** dialog box.

- ***Adjust the Overlapping Order of Images***
  In ***Graphic Editor***, the image drawn first is behind those drawn later. The overlapping order adjustment feature rearranges the overlapping order. Selecting ***Order>Send to Back*** in the Context Menu or ***Bring Forward*** can adjust the overlapping order.



Framework generated by Rectangle Image edited with Component Properties (2 rectangles)

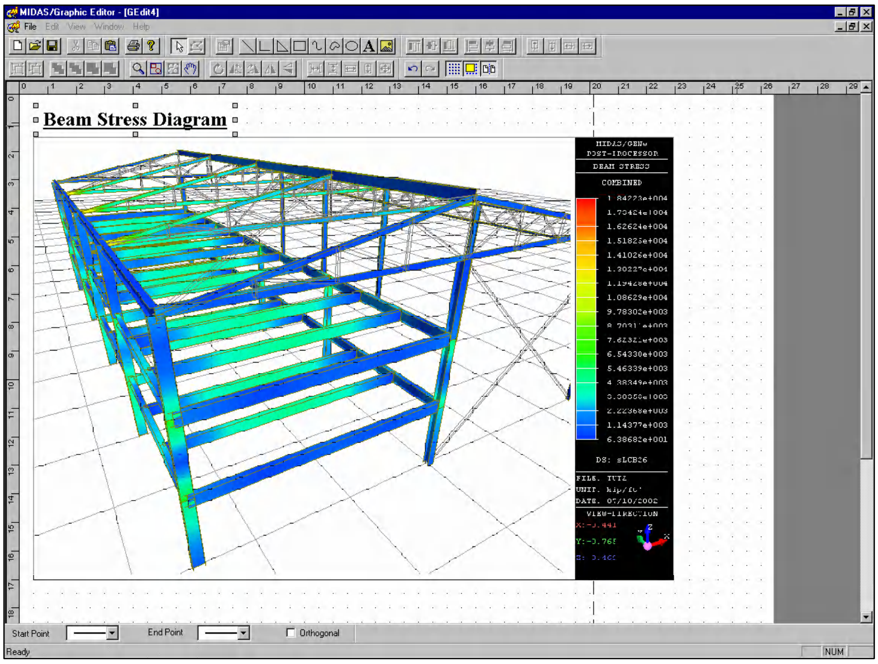


Example of a later-drawn rectangle brought backward by the overlapping order adjustment function

- ***Input of Text***

  The graphic editor allows the user to add titles or explanatory texts. Clicking **A** ***Text*** brings the cursor into a stand-by state for text input. At this time, move the cursor to the desired position and left-click the mouse. A text input element appears with "***Text***" written inside.

  The desired text may be entered after double-clicking "***Text***". Once the desired text has been typed in, click elsewhere on the window away from the text field to prompt the end of input. Now, right-click the mouse on the Text element, select ***Properties*** in the Context Menu, and edit the text properties to the desired format. The component properties such as the type, size and color of font, the format of the framework, etc., may be assigned. Even the text may be rotated such that the text is read vertically.

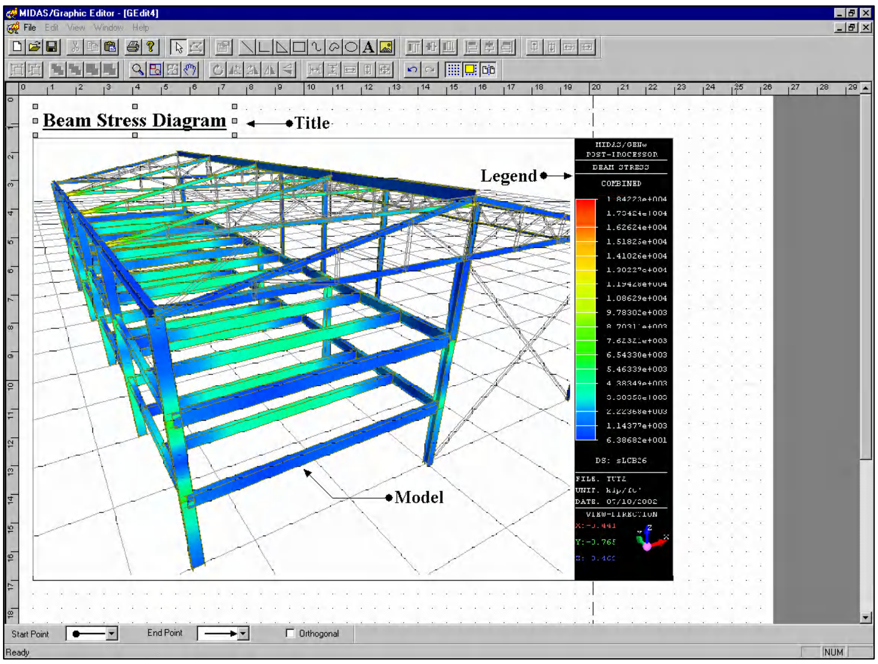


Addition of title on an image

➢ ***Insert Explanatory Lines***

By using the ***Line*** and ***Polyline*** commands and the text input function, explanatory lines to help clarify the image are inserted.

The Selection menu at the bottom of the screen determines the drawing method of the extremity and line shapes of ***Line*** or ***Polyline***. For drawing a new, straight line, the line begins with the selected shape at the ***Start Point*** and ends with the selected shape at the ***End Point***. If ***Orthogonal*** is checked, the shortest perpendicular lines linking the start and end points are drawn. By applying such a method, explanatory lines may be inserted in the drawing. First, place the start point in Circle and the end point in Arrow, and input Polyline. If an additional text is inserted to the right, the explanatory line is now completed.

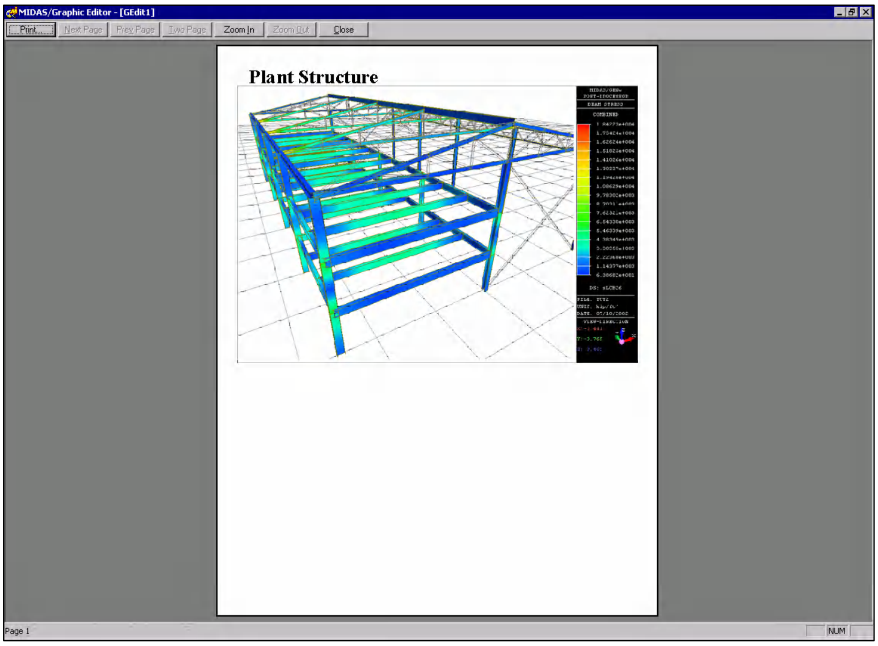


Example of explanatory lines

Print Preview and Page Setup

- ***Print Preview***
  When the drawing is complete, the layout of the drawing for printing may be verified in advance by ***Print Preview***. The printing is executed identically to the print preview displayed on the screen.

- ***Page Setup***
  Adjust the size, direction and margins of the printed forms.



Print Preview

APPENDIX A. Principal Features of MIDAS/Gen

Graphic Visualization and Model Verification

- Provision for all types of menu systems for user convenience (Tree Menus, Full Down Menus, etc.)
- Multi-window multitasking feature
- Various window manipulation capabilities: ***Zoom***, ***Pan***, ***Rotate***, ***View Point***, ***Dynamic View***, etc.
- Various representation schemes for modeled elements: Wire Frame, Slice, Surface, Solid Shape, etc.
- Selective model representation feature (***Active***/***Inactive***)
- ***Query*** features related to the input data (Attributes for nodes and elements)
- Dynamic auto-display feature for input contents (***Dynamic Label***)
- Various functions (***Select***) for the selection of input entities (***Single***, ***Window***, ***Polygon***, ***Intersect Line***, ***Plane***, ***Volume***, ***Identity***, ***Previous***, ***Recent Entities***, ***Group***, etc.)
- Unlimited repetitions of ***Undo***/***Redo*** and the provision of ***List***
- Various data formation references and functions (***UCS***, ***Grid Point***, ***Grid Line***, ***Snap***, etc.)
- Unrestricted unit specification and conversion
- Integration of a number of separate models into a unit model (***Merge Data File***)
- Import/Export capabilities with other S/W (***STF***, ***AutoCAD, MIDAS/SDS, SAP2000, STAAD, MSC.Nastran,*** etc.)
- ***Graphic Editor*** capability

- ***Text Editor*** capability
- Various data input functions (A string of data entries distinguished by " , " or "blank", computation capability using arithmetic operators and scientific functions, etc.)
- Capabilities supplying various graphic formats
- ***Table Window*** representation features (input, modify, duplicate, edit and data exchange with Excel)
- ***On-line Manual*** feature
- Input/Output in text-format of the model data (Import, Export, MGT Command Shell)
- Customization of Short-cut keys

Model Generation

- Convenient data entry (UCS, grid system, many types of snap functions, etc.)
- Various ***Structure Wizard*** capabilities (***Beam***, ***Column***, ***Arch***, ***Frame***, ***Truss***, ***Shell Structures***)
- Automatic generation of nodes (***Create***, ***Delete***, ***Translate***, ***Rotate***, ***Mirror***, ***Project***, ***Divide***, etc.)
- Automatic generation of elements (***Create***, ***Delete***, ***Translate***, ***Rotate***, ***Mirror***, ***Project***, ***Extrude***, ***Curve***, etc.)
- Wall combination number (Wall ID) auto-generation
- Selective duplication capabilities of attributes (load cases, boundary conditions, etc.) while duplicating nodes and elements
- Material and section properties input functions, with built-in section databases (AISC, JIS, etc.)
- Time dependent material properties input
- Non-prismatic (tapered) section assignment

- Sectional Property Calculator (Auto-calculation of stiffness data for an arbitrary section)
- Input data for boundary conditions (***Support***, ***Beam and Plate End-release***, ***Rigid End Offset***, etc.)
- ***Rigid Link*** feature (Master and Slave Nodes)
- Input data for specification of masses (Nodal masses, ***Floor Diaphragm Masses*** and automatic conversion of loading data into nodal mass data)
- Input data for design (Unsupported lengths, effective coefficients for buckling lengths, other data related to optimal design, etc.)
- Auto-generation of building (Consideration for varying story heights, material properties and section properties)

Load Generation

- Input data for nodal concentrated loads (Forces, Moments)
- Element loading input functions (GCS, ECS-based input functions)

Beam loads	In-span concentrated loads, uniformly distributed loads, non-uniformly distributed loads, pre-stress loads
Floor plate loads	Automatic conversion of floor plate loads into beam loads or wall element loads
Plane loads	Loads applied at specific locations of plate and solid elements
Pressure loads	Edge Pressure, Surface Pressure and Potential Pressure loads (Hydrostatic and soil pressures)
Wind loads	IBC2000, UBC97, ANSI94, BS6399 (1997), Eurocode-1 (1992), KS2000, JIS87
Equivalent static seismic loads	IBC2000, UBC91, UBC97, ATC3-06, JIS94, KS2000
Temperature loads	Nodal temperature loads and Temperature gradient loads
Forced displacement loads at supports	

- Input data for dynamic loads
  Response Spectrum, ***Time Forcing Functions***, ***Sinusoidal Forcing Function***, ***Earthquake Acceleration***, ***Delay Time***, etc.

- Automatic generation of earthquake loads
  18 types of built-in seismic accelerations records (El Centro, San Fernando, Kobe, etc.)

  6 types of built-in design response spectra (UBC91/97, ATC3-06, Newmark, KS2000, etc.)

  Automatic computation of the earthquake response spectrum related to a given seismic acceleration record

Analysis

- Finite element library
  Compression-only, ***Tension-only***, ***Gap***, ***Hook***, ***Cable***, ***Truss***, ***General Beam***, ***Tapered Beam***, ***Wall*** (In-plane/Out-plane Bending), ***Plane Stress***, ***Plate*** (Thin/Thick), ***Plane Strain***, ***Axisymmetric***, ***Solid Element*** (Hexagon, Wedge, Tetrahedron)

- Analysis capabilities
  Linear Static Analysis including Thermal Stress Analysis

  Heat of Hydration Analysis

  Analysis reflecting ***Time Dependent material Properties***

  Linear Dynamic Analysis
  - ***Free Vibration Analysis***
    (Natural Frequencies, Vibration Modes)
  - ***Response Spectrum Analysis***
    (SRSS, CQC, ABS, etc., including the recovery of Signs after Modal Combination)
  - ***Time History Analysis***

  Geometric Nonlinear Analysis
  (Large Displacement, P-delta Effect, Tension/Compression-only, Gap, Hook, Cable)

  Linear Buckling Analysis
  (Critical Buckling Forces and Buckling Modes)

  Pushover Analysis

- Other analysis capabilities

  Analysis* considering construction stages** (Column Shortenings*** due to Elastic and Time Dependent properties such as change of modulus of elasticity, creep and shrinkage)

  Analysis considering variations in section properties due to pre and post composite action of a composite structure

  Analysis for an unknown loading condition using optimization technique

  Unlimited numbers of nodes and elements

  Unlimited numbers of static unit load cases and load combinations

  Combining static and dynamic analyses

Output Verification

- Automatic load combination in accordance with the specified design standard

- Deformed shape and numerical values (provision for displacements along the length between the ends of a beam element including contour lines and maximum values)

- Member force diagrams and numerical values (including contour lines and maximum values)

- Stress distribution and principle stress diagrams for plate and solid elements (including contour lines and maximum values)

- Shear force and bending moment diagrams for beam elements (member force diagrams and contour lines)

- Reaction diagrams and numerical values at supports

- Animated simulation for variation process related to deformations and member forces or stresses (***AVI Animation*** display)

- Detail analysis results for each beam element (detail deformed shape, shear force and bending moment diagrams, maximum–stress envelope diagrams and stress distribution contours at a specific section)

- Numerical values and contours related to strength results of beam and truss elements

- Dynamic simulation of vibration shapes and buckling shapes for each mode

- Time history analysis results in graphic format

- Pushover analysis results including Demand and Performance spectrums

- Production of analysis results of construction stages

Output Envelope/BOM, etc.

- A feature which produces the maximum/minimum numerical values for each multiple load combination case related to all the analysis results

- A feature which lists the material quantities (member lengths, coated surface, weight and volume) for all the members included in the analysis model

Design

- ***Structural steel design standards***

  Manual of Steel Construction, Load & Resistance Factor Design, the American Institute of Steel Construction (AISC-LRFD93 & 2000)

  Manual of Steel Construction, Allowable Stress Design, the American Institute of Steel Construction (AISC-ASD89)

  Part 1. Code of practice for design in simple and continuous construction, British Standard (BS5950-90)

  Part 1.1 General Rules and Rules for Building, Design of Steel Structures (ENV 1993-1-1 Eurocode 3)

  Canadian Standards Association, Limit States Design of Steel Structures, 2001 (CSA-S16-01)

  Cold-Formed Steel Design, American Iron and Steel Institute (AISI-CFSD86)

- ***Reinforced Concrete (RC) design standards***

  The RC Structure Design Criteria of the American Concrete Institute (ACI318-89, 95, 99 & 02)

Canadian Standards Association, Design of Concrete Structures, 1994 (CSA-A23.3-94)

Part 1. Code of practice for design and construction, British Standard (BS8110-97)

Part 1. General Rules and Rules for Building, Design of concrete structures (ENV 1992-1-1 Eurocode 2)

- ***Structural Steel-Reinforced Concrete design standards***

  The Allowable Stress Design Method, the SSRC of US (SSRC79)

- ***Design capabilities***

  Structural steel strength verification for each design standard

  Structural steel–RC composite column (SRC) member strength verification

  Supply of graphs for analyzing analysis results (by members and by section types)

  Display of graphics for visual assessment of strength verification results

  Weight optimization of steel members per section type and automatic renewal of section properties

  Execution of optimal design via automatic iteration process of Structural analysis, Strength verification and Selection of optimal section

  Supply of graphs to assess optimization design process of steel structure

  Supply of weight distribution diagrams for combined stress ratios and supply of average safety factor graphs

  Strength verification of plate girders

  RC member design with respect to each design standard (computation of reinforcement)

  Automatic and precise computation of required reinforcing steel obtained from stress-strain analysis and P-M interaction diagrams for the design of RC members

  Output of rebar sizes and spacing based on the required reinforcement automatically-computed

  For the design of slender column and bracing members, the required rebar is computed by automatic calculation of moment magnification factors and the required moment capacities considering the slenderness effect.

  For the design of shear wall members, the bending moments about the weak axis are computed reflecting the slenderness effect and the reinforcing steel is computed accordingly.

  The end reinforcing bars are computed automatically for shear wall design.

  Design of spread footings and pile foundations

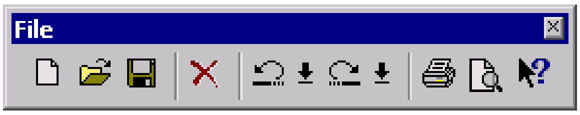
Possibility of assigning the structural system (lateral support/ non-lateral support) for each structural direction

Automatic computation of effective buckling length factors (K-Factor)

Output of strength verification calculations and the summaries of all types of design results

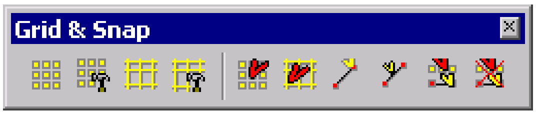
APPENDIX B. Toolbars and Icon Menus

File Toolbar



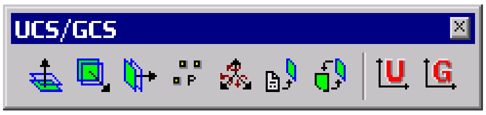
New	Open a new file.
Open	Open a saved file.
Save	Save the current working file.
Cut	Cut.
Copy	Copy.
Paste	Paste.
Delete	Delete the selected nodes or elements (possible to use the ***Delete*** key).
Undo	Cancel the latest input items entered during the modeling process and restore the model to the previous state.
Redo	Restore the tasks cancelled by the ***Undo*** function.
Print	Print the currently active window.
Print Preview	View the window for printing prior to actual printing.
On-line Manual	Request for assistance.

Grid & Snap Toolbar



Point Grid	Display point grids (Toggle On/Off).
Set Point Grid	Setup point grid environment (Toggle On/Off).
Line Grid	Display line grids (Toggle On/Off).
Set Line Grid	Setup line grid environment (Toggle On/Off).
Point Grid Snap	Apply snap function to the closest point grid. (Toggle On/Off)
Line Grid Snap	Apply snap function to the closest line grid (Toggle On/Off).
Node Snap	Apply snap function to the closest node (Toggle On/Off).
Element Snap	Apply snap function to the closest element (Toggle On/Off).
Snap All	Apply all the snap functions.
Snap Free	Cancel all the snap functions.

UCS/GCS Toolbar



X-Y — Define a plane parallel to GCS X-Y plane as UCS x-y plane.

X-Z — Define a plane parallel to GCS X-Z plane as UCS x-y plane.

Y-Z — Define a plane parallel to GCS Y-Z plane as UCS x-y plane.

Three Points — Define a plane determined by 3 points in GCS as UCS x-y plane.

Three Angles — Define a UCS by rotating GCS X, Y and Z-axes by specified angles.

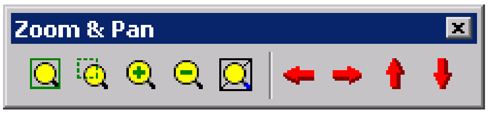
Named Plane — Define a UCS x-y plane by Named Plane previously assigned by the user.

Set UCS by Current UCS — Define a UCS by relocating the origin of the predefined UCS or rotating the predefined UCS about UCS x, y and z-axes by specified angles.

UCS — Apply User Coordinate System.

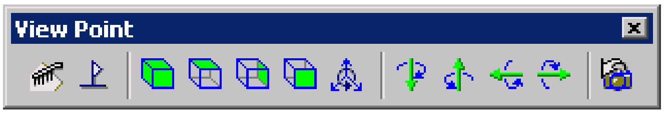
GCS — Apply Global Coordinate System.

Zoom & Pan Toolbar



Zoom Fit	Fit the currently active model to the size of the Model Window.
Zoom Window	Magnify the rectangular area outlined by the mouse.
Zoom In	Magnify the model window proportionally.
Zoom Out	Reduce the model window proportionally.
Auto Fitting	Activate the ***Zoom Fit*** function automatically to accommodate varying model sizes.
Pan Left	Move the model to the left by a certain distance.
Pan Right	Move the model to the right by a certain distance.
Pan Up	Move the model upward by a certain distance.
Pan Down	Move the model downward by a certain distance.

View Point Toolbar



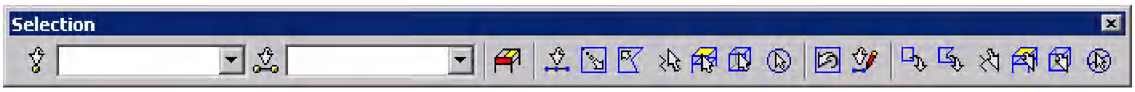
Redraw is used to remove the Dynamic Label, which displays automatically the Label for the latest input or to remove the residual image on the screen.

Redraw	Redraw the screen by applying the current View Point and Display Option.
Initial View	Revert to the initial stage of opening file in the case of preprocessing mode. Revert to the model view stage after deleting the analysis results in the case of post-processing mode.
Iso View	Display the model in a 3-D isometric view.
Top View	Display the model in the X-Y plane with the view point from the (+) Z-axis direction.
Right View	Display the model in the Y-Z plane with the view point from the (+) X-axis direction.
Front View	Display the model in the X-Z plane with the view point from the (–) Y-axis direction.
Angle View	Display the model relative to GCS with a specific view point.
Rotate Left	Rotate the model to the left.
Rotate Right	Rotate the model to the right.
Rotate Up	Rotate the model upward.
Rotate Down	Rotate the model downward.
View Previous	Restore the View Point immediately prior to the latest change.

Stage Toolbar

Define Construction Stage — Define analysis models for each construction stage.

Selection Toolbar



Select

Select Identity – Elements — Select elements by attributes.

Group — Select a Group among the groups predefined by the user. The groups may be defined relative to the geometric shapes or structural characteristics.

Select Single — Select/unselect one node or one element at a time with the mouse.

Select Window — Select the nodes and elements within a rectangular area defined with the mouse.

Select Polygon — Select the nodes and elements within a polygonal area defined with the mouse.

Select Intersect — Select the elements intersecting a series of specific straight lines drawn with the mouse.

Select Plane	Select all the nodes and elements included in a specific plane.
Select Volume	Select all the nodes and elements included in a specific volume.
Select All	Select all the nodes and elements displayed in the current window.
Select Previous	Reselect the last-selected nodes and elements.
Select Recent Entities	Select the nodes and elements most recently created.
Unselect Window	Unselect the presently selected nodes and elements within a rectangular area defined with the mouse.
Unselect Polygon	Unselect the presently selected nodes and elements within a polygonal area defined with the mouse.
Unselect Intersect	Unselect the presently selected elements intersecting a series of specific straight lines drawn with the mouse.
Unselect Plane	Unselect all the presently selected nodes and elements included in a specific plane.
Unselect Volume	Unselect all the presently selected nodes and elements included in a specific volume.
Unselect All	Unselect all the nodes and elements displayed in the current window.

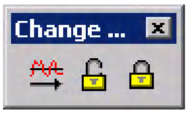
Activation Toolbar

Active	Activate and display only the selected nodes and elements.
Inactive	Activate and display only the unselected nodes and elements.
Inverse Active	Activate the inactive nodes and elements.
Active All	Activate and display all the nodes and elements currently modeled.
Active Identity	Activate the nodes and elements related to the assigned UCS x-y plane, Named Plane, Story or Group.
Active Previous	Revert to the previous state of activation.

View Control Toolbar

Shrink	Display the elements smaller than the true sizes (Shrink the elements from nodes).
Perspective	Display a perspective.
Hidden	Display the elements to appear as real shapes by removing the hidden lines, reflecting the sectional shapes and the thickness of the elements.
Render View	Display the model in a ***Hidden*** state with shading.
Rendering Option	Adjust the ***Render View*** for special shading effects in detail.
Display	A feature that enables the user to verify the input state related to all types of attributes such as loadings, support conditions, node or element numbers, material properties and section names, etc.
Display Option	A feature that enables the user to control the representation format (color, font size, etc.) related to all the graphics and texts in the working window

Change Mode Toolbar



Analysis	Perform structural analysis.
Preprocessing Mode	Switch to the preprocessing mode.
Post-processing Mode	Switch to the post-processing mode.

Label Option Toolbar

Node Number	Display the node numbers.
Element Number	Display the element numbers.

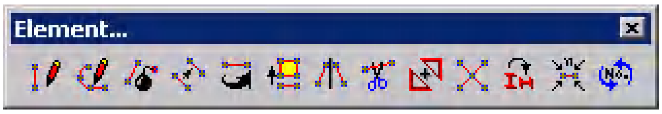
Dynamic View Toolbar

Zoom Dynamic	Magnify/Reduce the model in real time as desired by dragging the mouse.
Pan Dynamic	Move (up, down, left and right) the model in real time as desired by dragging the mouse.
Rotate Dynamic	Rotate the model in real time as desired by dragging the mouse.

Node Toolbar

Create Nodes	Create nodes.
Delete Nodes	Delete nodes.
Translate Nodes	Move or duplicate existing nodes by equal or unequal spacing.
Rotate Nodes	Move or duplicate existing nodes by rotating about a specified axis.
Project Nodes	Duplicate nodes by projecting on a specified line or surface.
Mirror Nodes	Duplicate nodes symmetrically with respect to a specified plane.
Divide Nodes	Divide nodes.
Merge Nodes	Merge all the nodes within a given tolerance.
Scale Nodes	Magnify or reduce the distances between nodes in a specified direction.
Compact Node Numbers	Remove the unused node numbers and renumber the remaining nodes sequentially.
Renumber Node ID	Renumber nodes.

Element Toolbar



Create Elements	Create elements.
Create Line Elements on Curve	Create line elements along a curve.
Delete Elements	Delete elements.
Translate Elements	Move or duplicate existing elements by equal or unequal spacing.
Rotate Elements	Move or duplicate existing elements by rotating about a specified axis.
Extrude Elements	Create elements by translating existing nodes into line elements, line elements into planar elements and planar elements into solid elements.
Mirror Elements	Move or duplicate elements symmetrically with respect to a specified plane.
Divide Elements	Divide elements.
Merge Elements	Merge continuously linked elements into a single element.
Intersect Elements	Divide elements automatically at their intersection points.
Change Element Parameters	Modify the attributes of the modeled elements.
Compact Element Numbers	Remove the unused element numbers and renumber the remaining elements sequentially.
Renumber Element ID	Renumber elements.

Result Toolbar

Reaction Forces / Moments — Verify support reactions by different components based on the numerical values and sizes of arrows.

Search Reaction Forces/Moments — Verify reactions at a specific support by numerical values.

Deformed Shape — Verify the deformed shape of the model.

Displacement Contour — Verify the deformed state of the model by contour lines.

Search Displacements — Verify the displacements of a specific node by numerical values.

Truss Forces — Verify the axial forces in tension or compression elements by contour lines.

Beam Forces / Moments — Verify the member forces in beam elements by contour lines.

Beam Diagram — Verify the shear forces or the bending moments in beam elements.

Plate Forces / Moments — Verify the member force distribution per unit length produced in plate elements by contour lines.

Wall Forces / Moments — Verify the member force distribution per unit length produced in wall elements by contour lines.

Wall Diagram — Verify the shear force and bending moment diagrams in wall elements.

Truss Stresses — Verify by contour lines the axial stresses in trusses, tension-only elements, compression-only elements, cable elements, etc.

Beam Stresses — Verify the stresses in beam elements by contour lines.

Plane/Plate Stresses	Verify the stresses in plane stress or plate elements by contour lines or vectors.
Plain Strain Stresses	Verify the stresses in plane strain elements by contour lines or vectors.
Axisymmetric Stresses	Verify the stresses in axisymmetric elements by contour lines or vectors.
Solid Stresses	Verify the stresses in solid elements by contour lines or vectors.
Vibration Mode Shapes	Verify the vibration mode shapes and natural frequencies of the model.
Buckling Mode Shapes	Verify the buckling mode shapes and critical buckling load factors of the mode.

Property Toolbar

Material	Enter the material properties of elements.
Time Dependent Material (Creep / Shrinkage)	Define the material data related to time variant concrete creep and shrinkage.
Time Dependent Material (Comp. Strength)	Define the change in modulus of elasticity of concrete with time.
Time Dependent Material Link	Relate the time dependent material data to the general material properties.
Change Element Dependent Material Property	In the case CEB-FIP is used to define Time Dependent Material Property, the geometric shape coefficient (h) may be changed.
Section	Enter the section properties of linear elements.
Section Stiffness Scale Factor	Specify scale factors for the stiffness components of line elements.
Tapered Section Group	Group non-prismatic members so that the variable section is defined irrespective of the number of the elements constituting the variable section.
Thickness	Enter the thickness data for plate elements.

Query Toolbar

Query Nodes	Verify attributes for nodes.
Query Elements	Verify attributes for elements.
Node Detail Tables	Verify attributes for selected nodes in table format.
Element Detail Tables	Verify attributes for selected elements in table format.
Design Parameter Detail Tables	Verify the design parameters for selected elements in table format.

APPENDIX C. List of Shortcut Keys

Main Menu	Parent Menu	Children Menu	Shortcut Key
File	New Project		Ctrl + N
	Open Project		Ctrl + O
	Save		Ctrl + S
	Print		Ctrl + P
Edit	Undo		Ctrl + Z
	Redo		Ctrl + Y
	Cut		Ctrl + X
	Copy		Ctrl + C
	Paste		Ctrl + V
	Delete		Del
	Find		Ctrl + F
View	Redraw		F3
	Initial View		Ctrl + F3
	Zoom	Fit	Ctrl + Ø
		Window	Ctrl + Shift + W
		In	Ctrl + +
		Out	Ctrl + −
	Pan	Left	Ctrl + ←
		Right	Ctrl + →
		Up	Ctrl + ↑
		Down	Ctrl + ↓
	View Point	Iso	Ctrl + Shift + I
		Top	Ctrl + Shift + T
		Bottom	Ctrl + Shift + B
		Left	Ctrl + Shift + L
		Right	Ctrl + Shift + R
		Front	Ctrl + Shift + F
		Rear	Ctrl + Shift + E

Main Menu	Parent Menu	Children Menu	Shortcut Key
View	View Point	Rotate Left	Ctrl + Alt + ←
		Rotate Right	Ctrl + Alt + →
		Rotate Up	Ctrl + Alt + ↑
		Rotate Down	Ctrl + Alt + ↓
	Previous View Status		Ctrl + B
	Shrink Elements		Ctrl + K
	Perspective View		Ctrl + J
	Remove Hidden Lines		Ctrl + H
	Render View		F6
	Select Identity	Element Type	Ctrl + Alt + A
		Material	Ctrl + Alt + B
		Section	Ctrl + Alt + C
		Thickness	Ctrl + Alt + D
		Named Plane	Ctrl + Alt + E
		Structure Group	Ctrl + Alt + G
	Select Single		Ctrl + Shift + S
	Select All		Ctrl + Shift + A
	Select Previous		Ctrl + Q
	Select Recent Entities		Ctrl + R
	Activities	Active	F2
		Inactive	Ctrl + F2
		Active All	Ctrl + A
		Active Identity	Ctrl + D
	Display		Ctrl + E
Model	Structure Wizard	Arch	Ctrl + Shift + W
		Frame	Ctrl + Shift + X
		Truss	Ctrl + Shift + Y
	Nodes	Create Nodes	Ctrl + Alt + 1
		Delete Nodes	Ctrl + Alt + 2
		Translate Nodes	Ctrl + Alt + 3
		Rotate Nodes	Ctrl + Alt + 4

Main Menu	Parent Menu	Children Menu	Shortcut Key
Model	Nodes	Project Nodes	Ctrl + Alt + 5
		Mirror Nodes	Ctrl + Alt + 6
		Divide Nodes	Ctrl + Alt + 7
		Merge Nodes	Ctrl + Alt + 8
		Compact Numbers	Ctrl + Alt + 9
		Nodes Table	Ctrl + Alt + N
	Elements	Create Elements	Alt + 1
		Delete Elements	Alt + 2
		Translate Elements	Alt + 3
		Rotate Elements	Alt + 4
		Extrude Elements	Alt + 5
		Mirror Elements	Alt + 6
		Divide Elements	Alt + 7
		Intersect Elements	Alt + 8
		Change Element Parameters	Alt + 9
		Compact Numbers	Alt + Ø
		Elements Table	Ctrl + Alt + M
	Properties	Material Table	Ctrl + Alt + L
		Section Table	Ctrl + Alt + S
		Thickness Table	Ctrl + Alt + T
	Boundaries	Supports Table	Ctrl + Alt + P
		Beam End Release Table	Ctrl + Shift + D
		Rigid Link Table	Ctrl + Alt + R
	Mass	Nodal Masses Table	Ctrl + Alt + U
	Define Structure Group		Ctrl + F1
	Check Structure Data	Check and Remove Duplicate Elements	F12
Load	Static Load Cases		F9
	Load Tables	Nodal Loads Table	Ctrl + Shift + N
		Beam Loads Table	Ctrl + Shift + M

Main Menu	Parent Menu	Children Menu	Shortcut Keys
Load	Load Tables	Floor Loads Table	Ctrl + Shift + O
Analysis	Perform Analysis		F5
Results	Combinations		Ctrl + F9
Mode	Preprocessing Mode		F7
	Post-processing Mode		Ctrl + F7
Query	Project Status		Ctrl + T
	Query Nodes		F4
	Query Elements		Ctrl + F4
Tools	MCT Command Shell		Ctrl + F12
	Text Editor		Ctrl + F5
	Graphic Editor		Ctrl + F6
Window	New Window		Ctrl + W
	Full Screen		Ctrl + U
Help	Index		F1

	Ctrl	Ctrl + Shift	Ctrl + Alt
A	Active All	Select All	Select Identity Element Type
B	Previous View Status	Bottom	Select Identity Material
C	Copy		Select Identity Section
D	Active Identity	Beam End Release Table	Select Identity Thickness
E	Display	Rear	Select Identity Named Plane
F	Find	Front	
G			Select Identity Structure Group
H	Remove Hidden Lines		
I		Iso	
J	Perspective View		
K	Shrink Elements		
L		Left	Material Table
M		Beam Loads Table	Elements Table
N	New Project	Nodal Loads Table	Nodes Table
O	Open Project	Floor Loads Table	
P	Print		Supports Table
Q	Select Previous		
R	Select Recent Entities	Right	Rigid Link Table
S	Save	Select Single	Section Table
T	Project Status	Top	Thickness Table
U	Full Screen		Nodal Masses Table
V	Paste		
W	New Window	Structure Wizard-Arch	
X	Cut	Structure Wizard-Frame	
Y	Redo	Structure Wizard-Truss	
Z	Undo		

	Ctrl	Alt	Ctrl + Alt
1		Create Elements	Create Nodes
2		Delete Elements	Delete Nodes
3		Translate Elements	Translate Nodes
4		Rotate Elements	Rotate Nodes
5		Extrude Elements	Project Nodes
6		Mirror Elements	Mirror Nodes
7		Divide Elements	Divide Nodes
8		Intersect Elements	Merge Nodes
9		Change Element Parameters	Compact Numbers
Ø	Zoom Fit	Compact Element Numbers	

	Function	Ctrl + **Function**
F1	Help	Structure Group
F2	Active	Inactive
F3	Redraw	Initial View
F4	Query Nodes	Query Elements
F5	Perform Analysis	Text Editor
F6	Render View	Graphic Editor
F7	Preprocessing Mode	Post-processing Mode
F8		
F9	Static Load Cases	Combinations
F10		
F11		
F12	Check and Remove Duplicate Elements	MCT Command Shell

	Shortcut Key		**Shortcut Key**
Zoom Fit	Ctrl + Ø	Pan Down	Ctrl + ↓
Zoom In	Ctrl + +	Delete	Del
Zoom Out	Ctrl + −	Rotate Right	Ctrl + Alt + ←
Pan Left	Ctrl + ←	Rotate Left	Ctrl + Alt + →
Pan Right	Ctrl + →	Rotate Up	Ctrl + Alt + ↑
Pan Up	Ctrl + ↑	Rotate Down	Ctrl + Alt + ↓