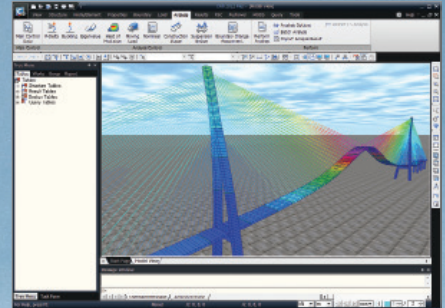


For utmost Accuracy & Productivity, MIDAS provides the best solutions in Structural Engineering. We Analyze and Design the Future.



Structural Analysis II

Advanced Analysis with Midas Software



Structural Analysis II

Advanced Analysis using MIDAS Software

MIDAS Technical Support, Korea

MIDAS Global Technical Support, India

Pinakin Gore

Nandeep Gohil

Sindhu Bharathi

Suman Dhara

Pratap Jadhav

MIDAS R&D Centre India Pvt., Ltd.

Mumbai, India

MIDAS Offices

New York, London, Moscow, Dubai, Singapore

Manila, Shanghai, Beijing, Seoul, Tokyo

Copyright © 2017, by MIDAS Research & Development Centre India.
All rights reserved.

Second reprint 2017

No part of this publication may be reproduced or distributed in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise or stored in a database or retrieval system without the prior written permission of the publisher.

This edition can be exported from India only by the publisher,
MIDAS Research & Development Centre India.

Published by the MIDAS Research & Development Centre India Pvt, Ltd.
1001 Cyberone, Sector 30A, Vashi, Navi Mumbai 400705 India
india@MidasUser.com
www.MidasUser.com



PROGRAM VERIFICATION AND PRECAUTIONS

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 software during the development. Since the initial development in 1989, MIDAS Family Programs have been accurately and effectively applied to over 10,000 International projects.

A strict verification process of the Computational Structural Engineering Institute of Various Countries 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/Civil, MIDAS/FEA, GTS NX, Soilworks, MIDAS/Design+ 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 General structure design.

II

Advanced Analysis

Index

1. Introduction
2. P-delta Analysis
3. Geometric Non-linear Analysis
4. Buckling Analysis
5. Eigenvalue Analysis
6. Time History Analysis
7. Response Analysis
8. Prestress Analysis
9. Thermal Stress Analysis
10. Plate analysis on Out-of-plane Analysis



1. Introduction

Contents

1 Introduction

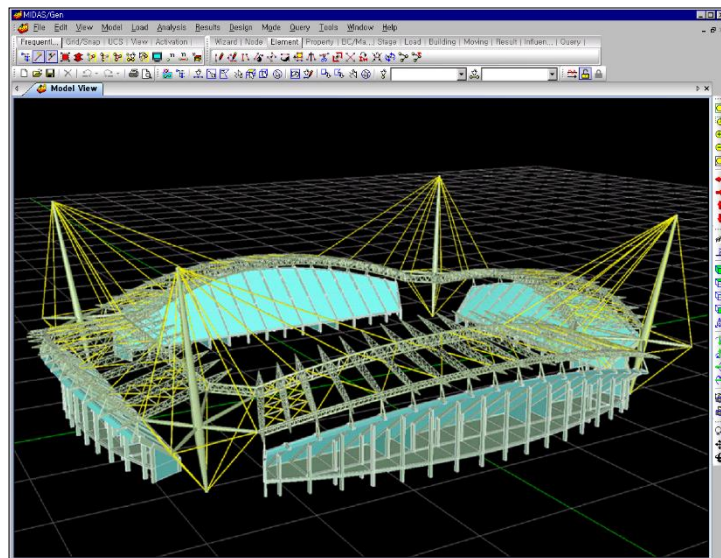
1.1 About MIDAS/Gen	1-3
1.2 Installation	1-7



1. Introduction

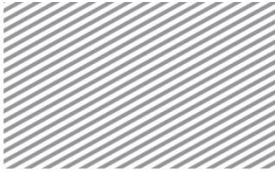
1.1 About MIDAS/Gen

MIDAS/Gen stands for “General structure design system.”, i.e., a Windows based integrated system for structural analysis and optimal design.

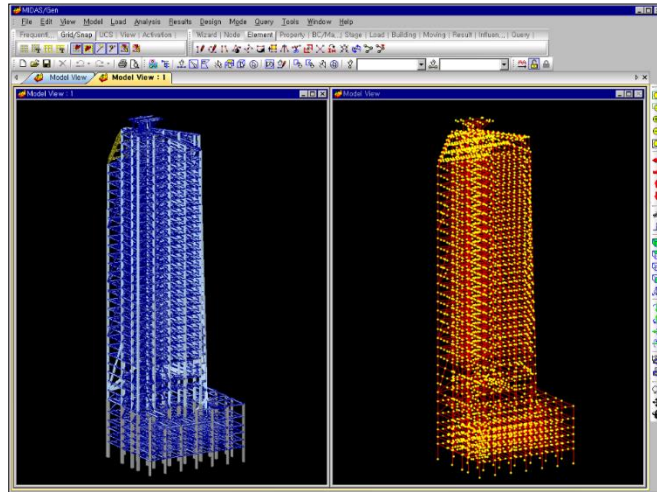


MIDAS/Gen, developed in the object-oriented programming language Visual C++, fully exploits the advantages and the characteristics of the 32bit/64bit Windows environment for the technical computations.

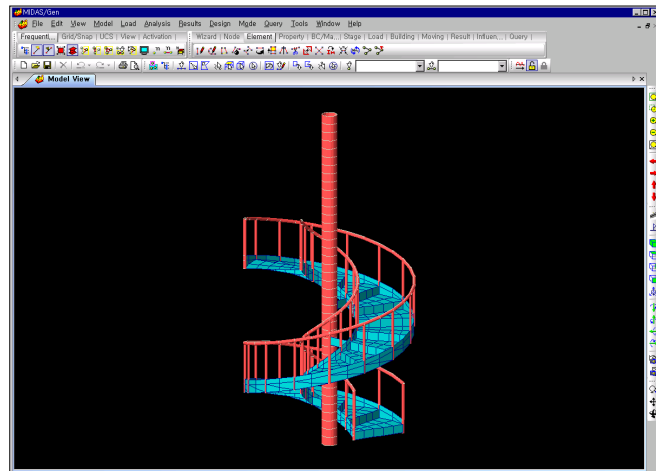
The user-oriented input/output functions are based on sophisticated and intuitive User Interface and up-to-date Computer Graphics techniques. They offer excellent facilities and productivity for the modeling and analysis of complex, large-scale structures. The technical aspects of structural analysis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



Analysis model of a spiral staircase

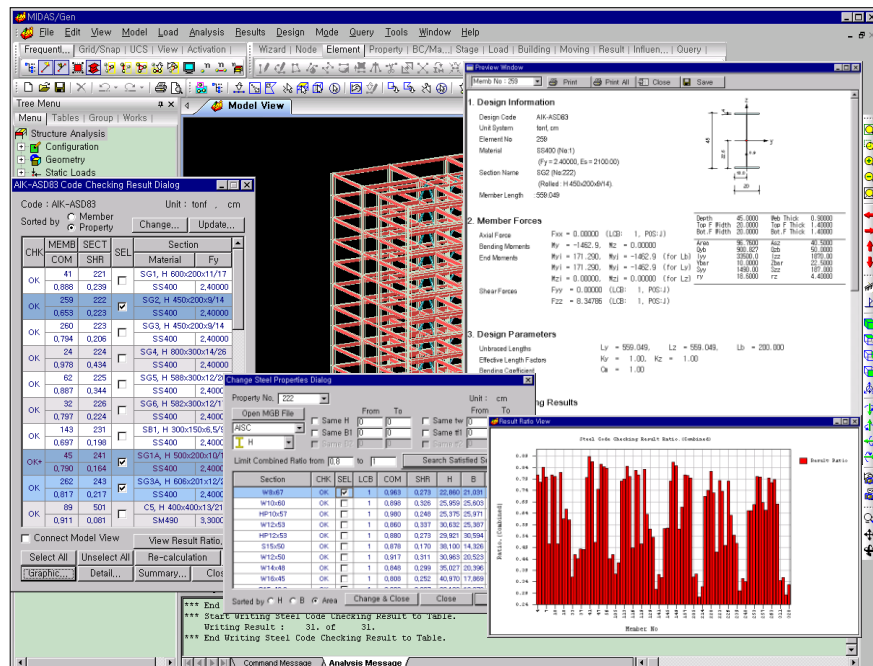
Structural Analysis II (Advanced)

1. Introduction

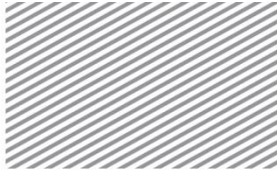
MIDAS IT's in-house researchers have developed an efficient CAD modeling technique, which is a totally new concept. Powerful automatic modeling functions such as Auto Mesh Generation (available as a separate module) and Structure Wizard are introduced. Also, a new Multi-Frontal Sparse Gaussian Solver 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, IS, IRC, 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.



Result of strength verification according to AISC



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.



1.2 Installation

System Requirement

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

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

- i3 CPU or better performing PC processor
- Minimum of 2GB RAM
- 1GB of free space on HDD (MIDAS/Gen requires a minimum of 1 GB hard disk space for Construction Stage analysis)
- Microsoft Windows 7 or higher version or Windows NT Operating System
- **Nvidia** Graphics card, Monitor with a minimum of 1024×768 resolution and a minimum of 16bit High Color display
- Windows compatible Printer or Plotter



2. P-delta Analysis

Contents

1 Introduction

1.1 Concept of P-delta Analysis	2-3
---------------------------------	-----

2 Tutorial

2.1 Model Overview	2-8
2.2 Work Environment	2-9
2.3 Material & Section Properties	2-11
2.4 Generate Node & Element	2-12
2.5 Define Boundary Conditions	2-14
2.6 Define Loads	2-15
2.7 Perform Analysis	2-17
2.8 Check Analysis Result	2-19

3 Exercise	2-27
------------	------



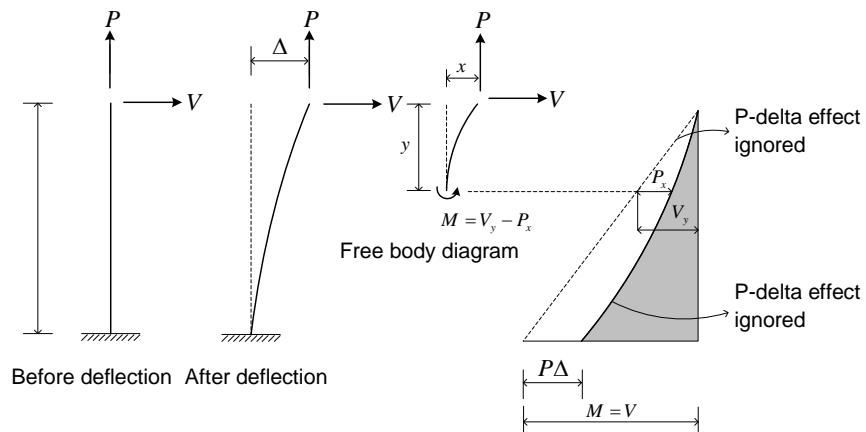
1. Introduction

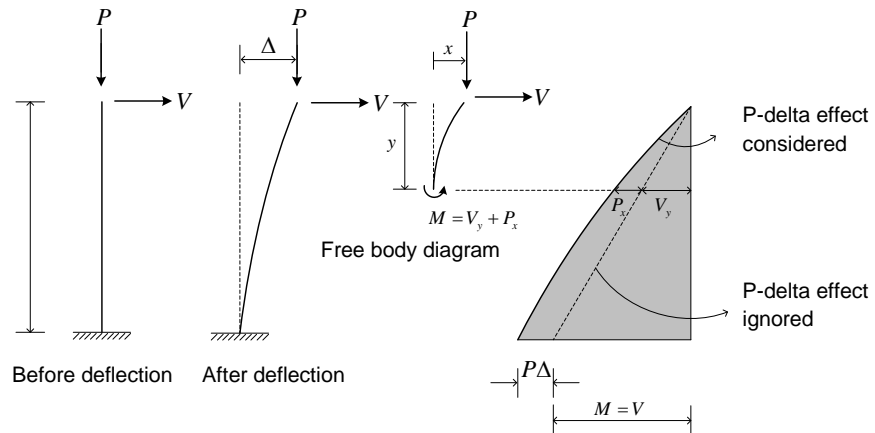
1.1 Concept of P-delta Analysis

P-Delta effect is the secondary moment generated, when a slender member is subjected to axial and lateral loads simultaneously. In another terms, it is a nonlinear geometric effect of a large direct stress acted upon transverse bending and shear behavior. Compressive stress on structure makes it more flexible in transverse bending and shear, whereas tensile stress tends to stiffen member against transverse deformation. The analysis is called “P-Delta” because the magnitude of the secondary moment is equal to “P”, the axial force in the member, times “Delta”, the offset distance one end of the member from other end.

As shown in Fig.2.1(a), if equilibrium is examined in the original configuration (using undeformed geometry) the moment at the base is $M=VL$, and decreases linearly to zero at the loaded end. If equilibrium is considered for deformed configuration, the additional moment caused by axial force P acting on the transverse tip displacement Δ . The moment does not vary linearly along the length; the variation depends on the deflected shape. The moment at the base is now $M=V_y - P_x$

► Fig 2.1
Column P-Delta effect
(a) Tensile & Lateral
Forces





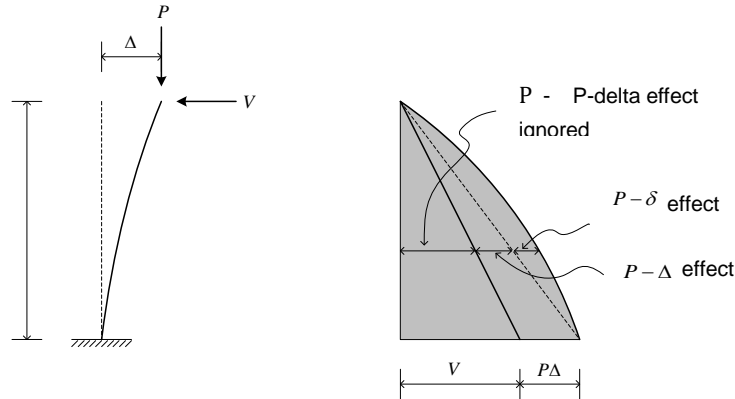
If the member is in tension as shown in Fig.2.1(a), the moment at the base and throughout the member is reduced. Hence transverse bending deflection Δ is also reduced. Thus the member is effectively stiffer against the transverse load V .

Conversely if the member is in compression as shown in Fig.2.1(b), the moment at the base and throughout the member is increased. Hence transverse bending deflection Δ is also increased. The member is effectively more flexible against the transverse load V .

If the compressive force is too large, transverse stiffness goes to zero and hence the deflection tends to infinity; at that point the structure is said to have buckled.

P-Delta effect is sub divided into two, as shown in the following figure. The $P-\delta$ effect refers to the effect of the local geometry change whereas $P-\Delta$ refers to the effect due to relative displacement between member ends.

► Fig. 2.2
P-delta Effect



The P-Delta analysis is particularly useful when effects of gravity loads on lateral stiffness of the structure is required to be checked. Certain codes (ACI318, AISC-LRFD) require analysis with P-Delta effect to prevent collapse of the structure due to secondary effects. This section describes analysis of Truss and target beam elements reflecting P-Delta effect.

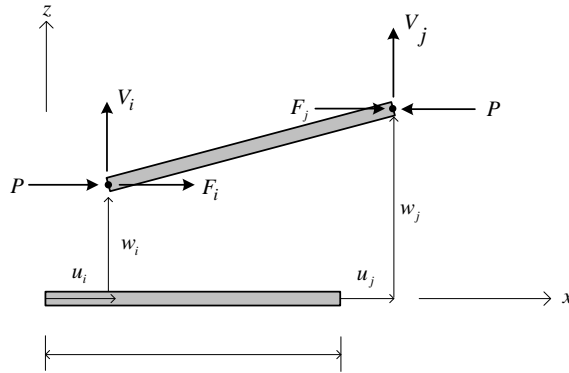
For reflecting P-Delta effect, a truss element is considered in transformed state and force equilibrium conditions are assumed.

► Eq.2.1

$$F_i = \frac{EA}{l}(u_i - u_j) = P, \quad V_i = \frac{P}{l}(w_i - w_j)$$

$$F_j = \frac{EA}{l}(-u_i + u_j) = -P, \quad V_j = \frac{P}{l}(-w_i + w_j)$$

► Fig. 2.3
P-Delta effect for Truss
Element



Eq 2.2 Matrix is represented as follows

► Eq. 2.2

$$f = ku$$

$$f = \begin{bmatrix} F_i & V_i & F_j & V_j \end{bmatrix}^T, \quad u = \begin{bmatrix} u_i & w_i & u_j & w_j \end{bmatrix}^T$$

$$k = k_o + k_\sigma$$

$$k_o = \frac{EA}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}, \quad k_\sigma = \frac{P}{L} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

Where k_o represents a linear stiffness, k_σ represents geometric stiffness reflecting P-delta effect. The axial force P, if in compression has the negative sign. By applying principle of virtual work, strain can be derived same as Eq. 2.2.

The concept of P-Delta analysis for beam element is same as that for the truss element, only the point of difference being deformation of element is considered additionally for beam. The load "q" being in x-z plane has the equation as described in finite element analysis according to section 1.1.5 and the effect of the load can be given by Eq 2.3.

► Eq. 2.3

$$EIw'''' + Pw'' = q$$



In order to analyze induced bending deformation of beam element, strain equation can be used to obtain Eq.2.4 using principle of virtual work.

► Eq.2.4

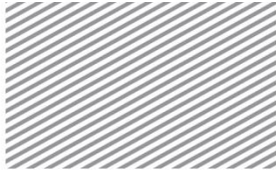
$$f = ku$$

$$f = [V_i \quad M_i \quad V_j \quad M_j]^T, \quad u = [w_i \quad \theta_i \quad w_j \quad \theta_j]^T$$

$$k = k_0 + k_\sigma$$

$$k_0 = \begin{bmatrix} \frac{12EI}{l^3} & \frac{6EI}{l^2} & -\frac{12EI}{l^3} & \frac{6EI}{l^2} \\ & \frac{4EI}{l} & -\frac{6EI}{l^2} & \frac{2EI}{l} \\ & & \frac{12EI}{l^3} & -\frac{6EI}{l^2} \\ sym & & & \frac{4EI}{l} \end{bmatrix} \quad k_\sigma = \frac{P}{30} \begin{bmatrix} \frac{36}{l} & 3 & -\frac{36}{l} & 3 \\ & 4l & -3 & -1 \\ & & \frac{36}{l} & -3 \\ sym & & & 4l \end{bmatrix}$$

Eq.2.4 has added geometric stiffness as compared to Eq.1.28. Also Eq.2.4 can be used for P-Delta buckling analysis

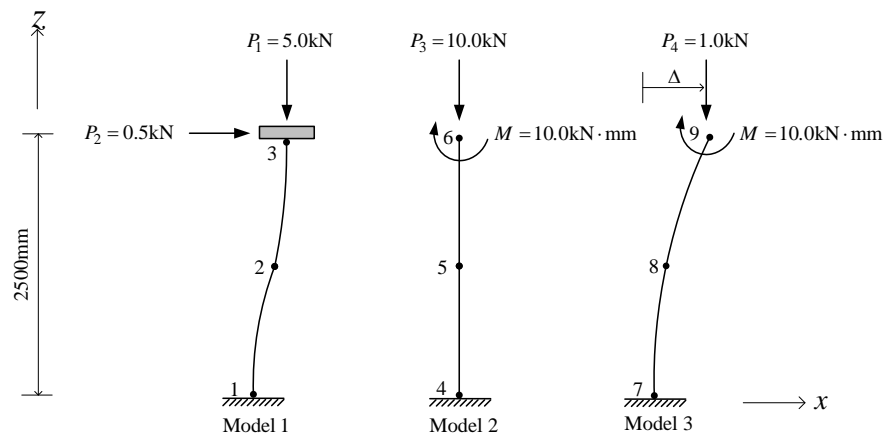


2. Tutorial

2.1 Model Overview

Perform P-Delta effect analyses for 3 different columns loading cases, and check the maximum deformation, shear forces and moments for each case.

► Fig. 2.4
Analytical model




- **Material**
Modulus of elasticity: $2.0 \times 10^5 \text{ N / mm}^2$
- **Sections**
Section size : $50 \times 50 \times 5 \text{ mm}$
Area : 900 mm^2
Moment of inertia (I_{yy}) : $300,000 \text{ mm}^4$
- **Load**
 1. Model 1 : A lateral load and axial force (Pure Sway)
 2. Model 2 : End bending moment and axial force (No Sway)
 3. Model 3 : End bending moment and eccentric axial force (No shear)

2.2 Work Environment

Open a new file and save the file name as 'Pdelta.mcb'.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : **'Pdelta'**, Click **[SAVE]**

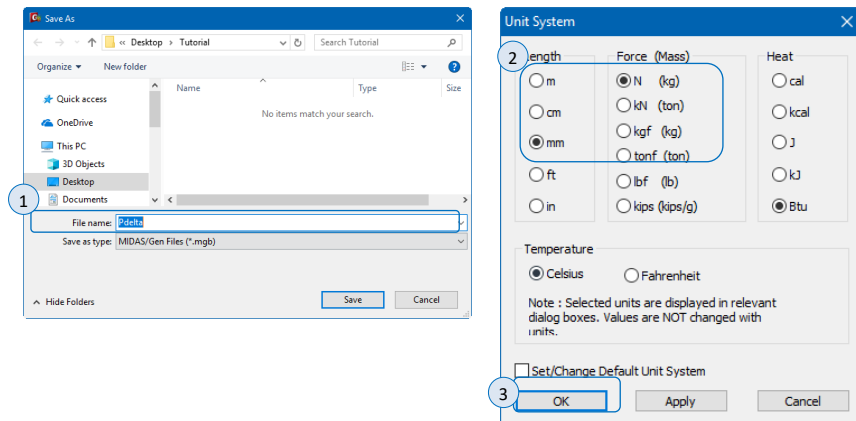
Set the unit system to use.

Main Menu > **Tools** > **Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click **[OK]**

► Fig. 2.5
Define unit system



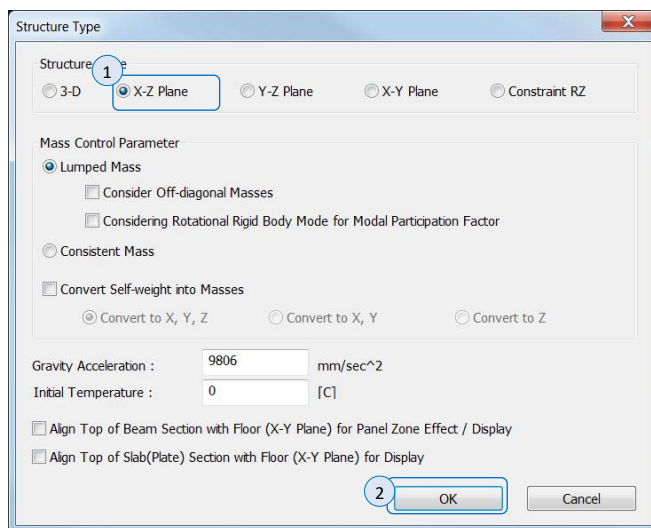


midas Gen is 3-D software, since the beam exist in a 2-D plane, X-Z plane in Global Coordinate System(GCS) is set as the work plane, which restrains unnecessary degree of freedom, Dy, Rx, and Rz.

Main Menu > **Structure** > **Structure Type**

1. Select Structure Type > **X-Z Plane**
2. Click **[OK]**

► Fig. 2.6
Set Work Plane



2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu > **Properties** > **Material Properties**

1. Click **[Add...]**, Name : 'Mat'
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : '**2.0e5**'
4. Click **[OK]**
5. Click **Section** Tab and **[Add...]** and **Value** Tab
6. Click Section Shape lists > **Box**, Name : '**Sec**'
7. H : '**50**', B : '**50**', tw : '**5**', tf1 : '**5**', Area : '**900**', I_{yy} : '**300,000**'
8. Click **[OK]** and **[Close]**

► Fig 2.7
Define Material &
Section



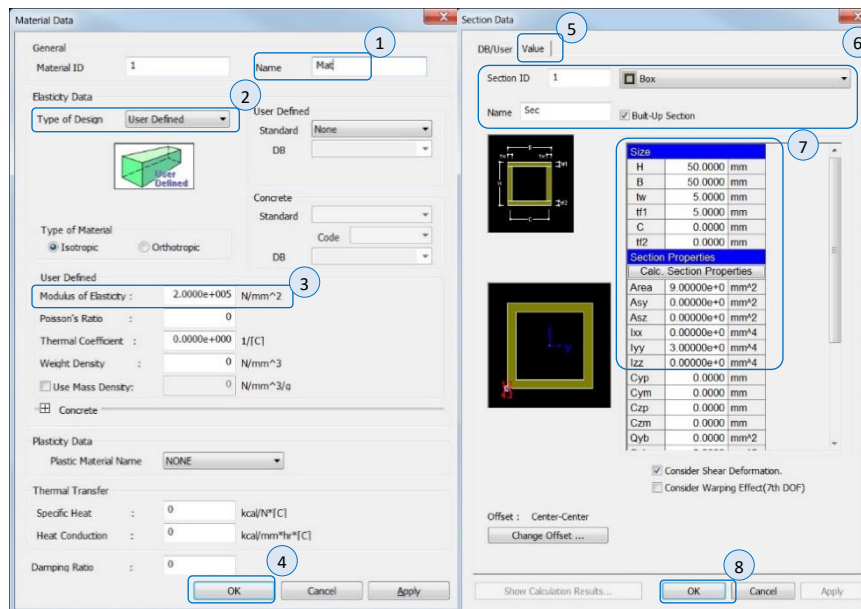
Tip

If you set the Type of Design to User Defined when using any material, you can directly input the property value.



Tip

The OK button closes the Material dialog box, so the Apply button is useful for defining multiple materials.



2.4 Generate Nodes & Elements

Change the units to m, kN. Create a node in order to generate beam elements.

Main Menu > **Tools** > **Unit System** Change unit of force to kN

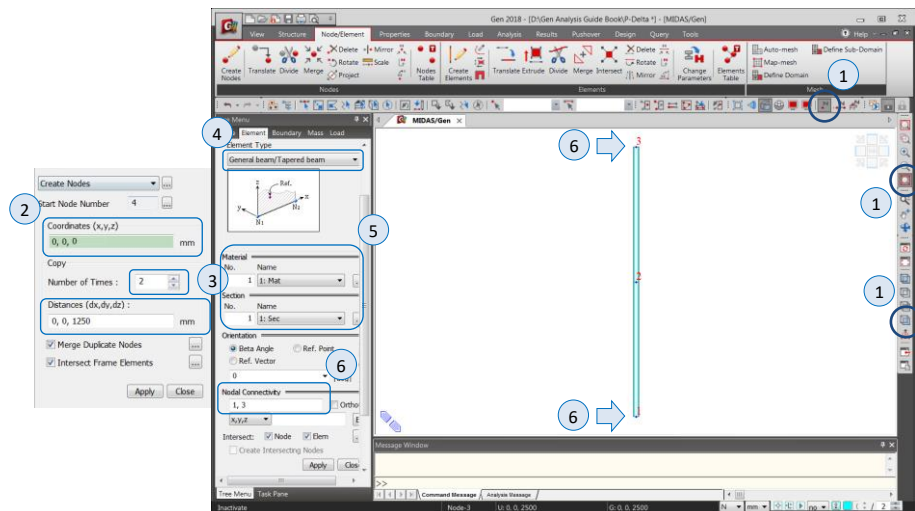
Main Menu > **Node/Element** > **Create Nodes**

1. **Display Node Numbers, Auto Fitting, Front View** (on)
2. Coordinates (x, y, z) : '**0, 0, 0**'
3. Copy > Number of Times : '**2**', Distance(dx, dy, dz) : '**0, 0, 1250**', Click [**Apply**]

Main Menu > **Node/Element** > **Create Elements**

4. Select Element Type > **General beam/Tapered beam**
5. Select Material > **1:Mat** and Section > **1:Sec**
6. Click Nodal Connectivity green box, and Click node number 1 and 3 in Model view

► Fig. 2.8
Create a Column

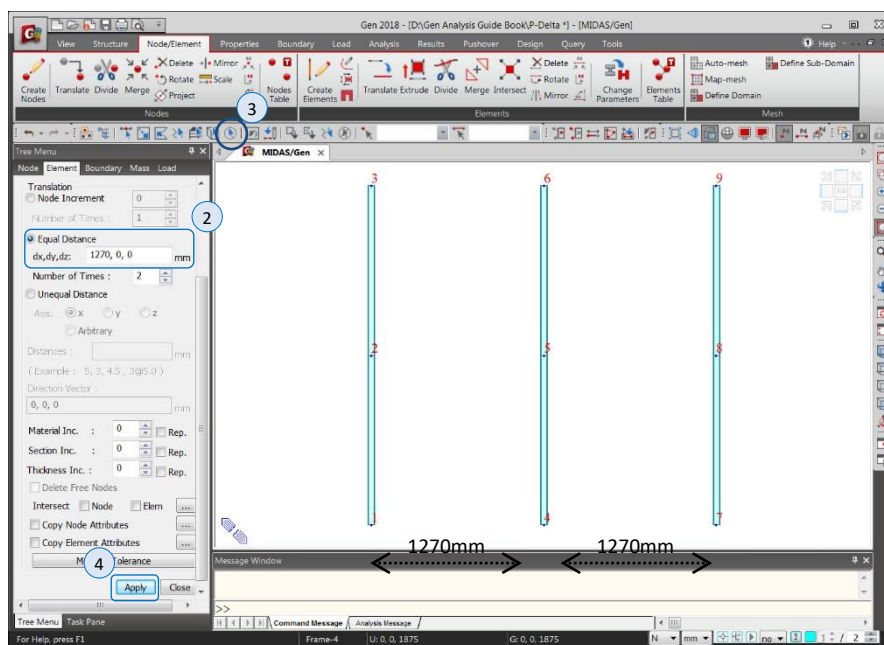


Copy the created columns using the copy function of Translate Elements.

Main Menu > **Node/Element** > **Translate Elements**

1. Select Mode > **Copy**
2. Translation > Equal Distance > dx, dy, dz : '**1270, 0, 0**', Number of Times : '**2**'
3. **Select All** (on)
4. Click **[Apply]**

► Fig. 2.9
Translate Elements



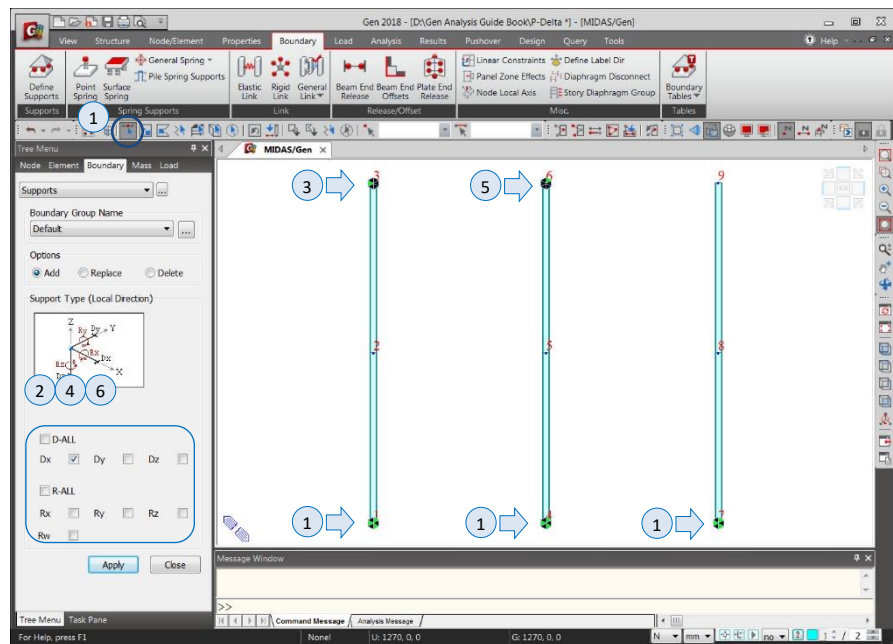
2.5 Define Boundary Conditions

Define boundary conditions at each of the columns.

Main Menu > **Boundary** > **Define Supports**

1. Click **Select Single** (on), Select node number **1** and **4** and **7**
2. Support Type > **Dx, Dz, Ry** (on), Click **[Apply]**
3. Click **Select Single** (on), Select node number **3**
4. Support Type > **Dx, Dz** (off), **Ry**(on), Click **[Apply]**
5. Click **Select Single** (on), Select node number **6**
6. Support Type > **Dx** (on), **Ry** (off), Click **[Apply]**

► Fig. 2.10
Input support conditions





2.6 Define Loads

Define the load cases first to which the loading will belong.

Main Menu > **Load** > **Static Loads** > **Static Load Cases**

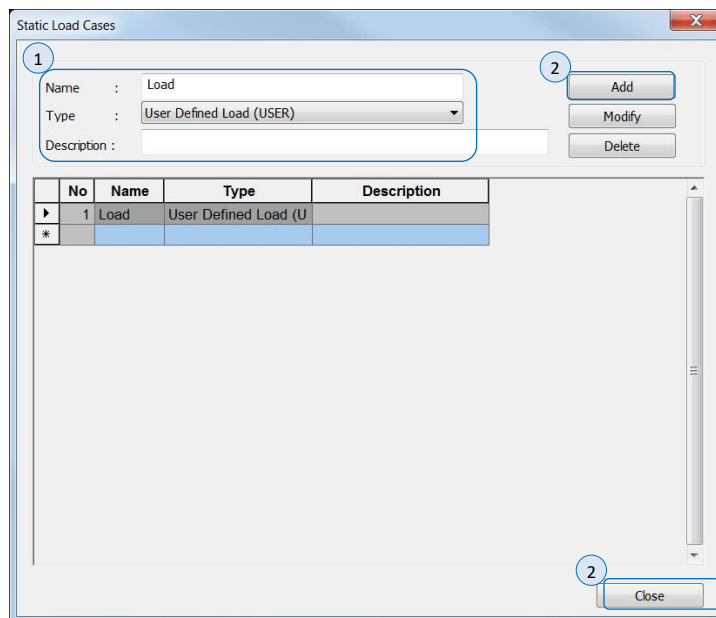
1. Name : '**Load**'

Select Type > **User Defined Load (USER)**

2. Click **[Add]** and **[Close]**

► Fig. 2.11

Define load cases



Input a nodal load for each model.

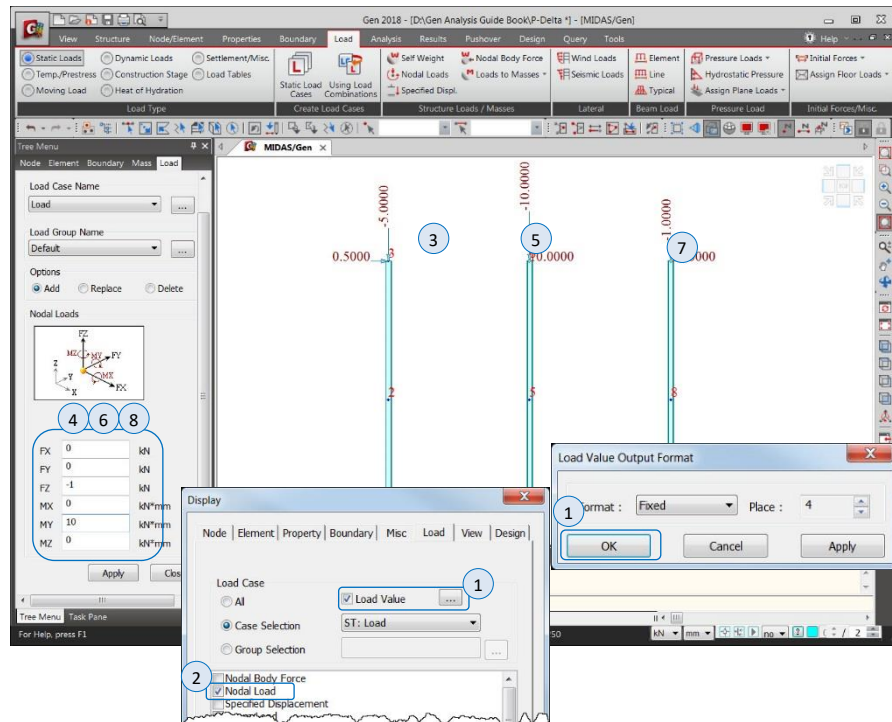
Main Menu > **View > Display...**

1. **Load** Tab > Load Case > **Load Value** (on), Click [...], Place : '4', Click **[OK]**
2. **Nodal Load** (on), Click **[OK]**

Main Menu > **Load > Static Loads > Nodal Loads**

3. Click **Select Single** (on), Select node number **3**
4. Nodal Loads > FX : '**0.5**', FZ : '**-5**', Click **[Apply]**
5. Click **Select Single** (on), Select node number **6**
6. Nodal Loads > FX : '**0**', FZ : '**-10**', MY : '**10**', Click **[Apply]**
7. Click **Select Single** (on), Select node number **9**
8. Nodal Loads > FZ : '**-1**', MY : '**10**', Click **[Apply]**

► Fig. 2.12
Input nodal load



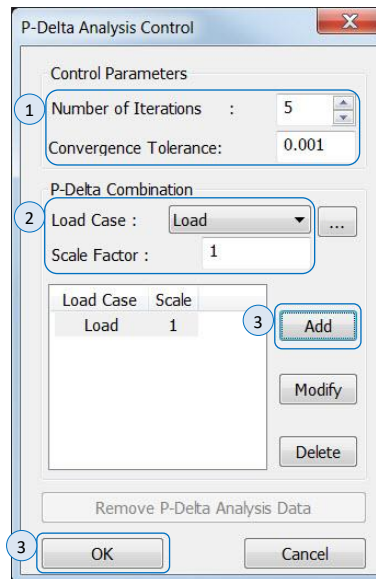
2.7 Perform Analysis

Define the analysis conditions of P-delta analysis

Main Menu > **Analysis > P-Delta (Analysis Control...)**

1. Control Parameters > Number of Iteration : '5'
Convergence Tolerance : '0.001'
2. P-Delta Combinations > Load, Scale Factor : '1'
3. Click **[Add]** and **[OK]**

► Fig. 2.13
P-Delta Analysis Control



Analyze all three models

1. Check for successful completion in Message Window

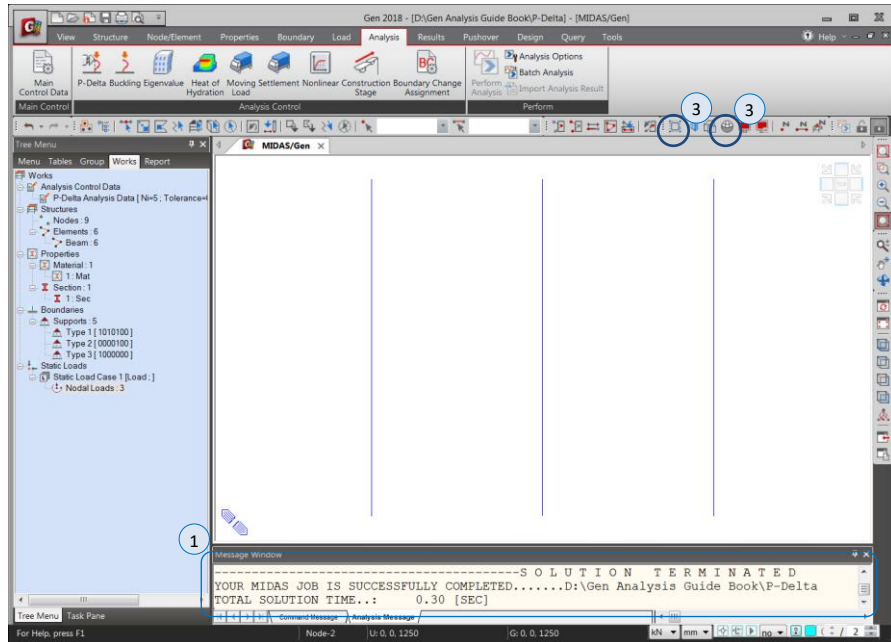
Main Menu > View > Display...

2. Load Tab > **Nodal Load** (off), Click [OK]

3. **Hidden** (off), **Display Node Numbers** (off)

► Fig. 2.14
Message for a
successful run

Tip
The shortcut icon for
Perform Analysis is
located at the top of
Model View (①). The
shortcut key is executed
by pressing the function
key F5.



2.8 Check Analysis Result

Check Deformed Shape by load cases).

Main Menu > **Results** > **Deformations** > **Deformed Shape...**

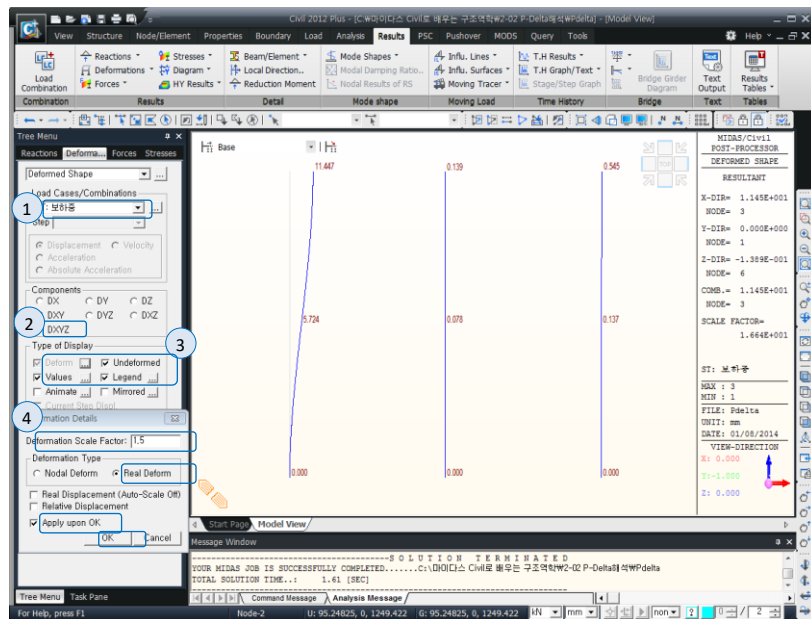
1. Select Load Cases / Combinations > **ST : Load**
2. Select Components > **DXYZ**
3. Type of Display > **Undeformed, Values, Legend (on)**
4. Click [...] in Deform

Deformation Scale Factor : **'1.5'**

Select Deformation Type > **Real Deform**

Apply upon OK (on), Click **[OK]**

► Fig. 2.15
Deformation Results

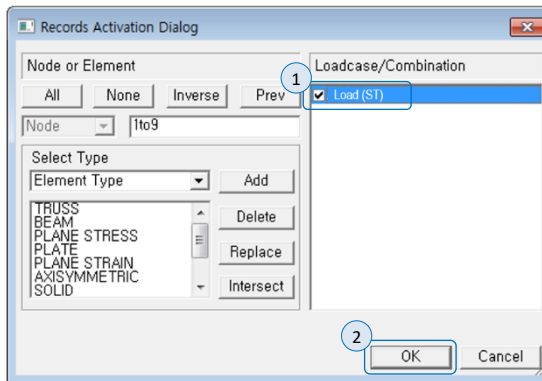


Next, we use Result Tables to check the displacement caused by P-delta analysis.

Main Menu > **Results** > **Result Tables** > **Displacements...**

1. Select Load cases / Combination > **Load (ST)** (on)
2. Click **[OK]**

► Fig. 2.16
Set Result Table Data



	Node	Load	DX (mm)	DY (mm)	DZ (mm)	RX ([rad])	RY ([rad])	RZ ([rad])
►	1	Load	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000
	2	Load	5.723529	0.000000	-0.034722	0.000000	0.006883	0.000000
	3	Load	11.447058	0.000000	-0.069444	0.000000	0.000000	0.000000
	4	Load	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000
	5	Load	-0.034489	0.000000	-0.069444	0.000000	-0.000027	0.000000
	6	Load	0.000000	0.000000	-0.138889	0.000000	0.000108	0.000000
	7	Load	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000
	8	Load	0.137000	0.000000	-0.006944	0.000000	0.000219	0.000000
	9	Load	0.544438	0.000000	-0.013889	0.000000	0.000432	0.000000

Check the column axial force by the load cases.

1. Click **Model View Tab**

Main Menu > **Results** > **Forces** > **Beam Diagrams...**

2. Select Load Cases / Combinations > **ST : Load**

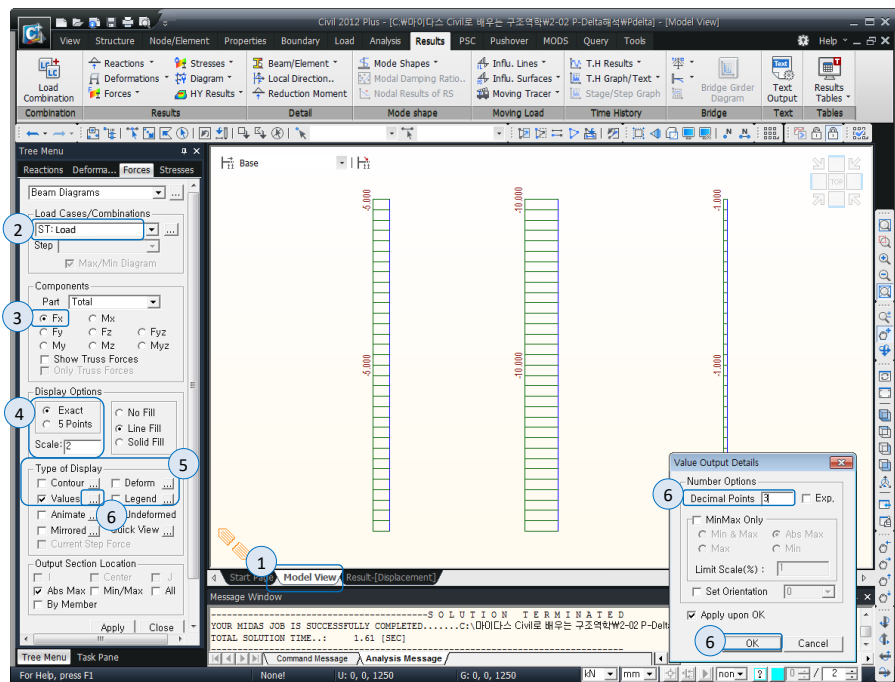
3. Select Components > **FX**

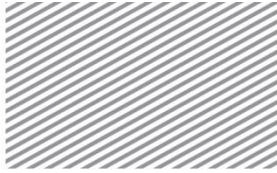
4. Display Options > Exact, Scale : '2'

5. Type of Display > **Contour**, **Deform**, **Legend** (off), **Value** (on)

6. Click [...] in Values, Number Options > Decimal Points : '3', Click **[OK]**

► Fig. 2.17
Axial Force Result





Save 'Pdelta.mgb' as 'GNL.mgb' with P-delta analysis.

Main Menu >  > **SaveAs...**

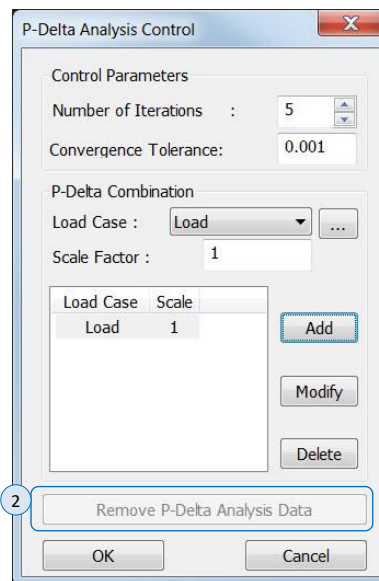
1. File name : '**GNL**', Click **[SAVE]**

Deletes the existing P-delta analysis condition to perform geometric nonlinear analysis.

Main Menu > **Analysis > P-Delta (Analysis Control...)**

2. Click **[Remove P-Delta Analysis Data]**

► Fig. 2.18
Remove P-delta
Analysis Data



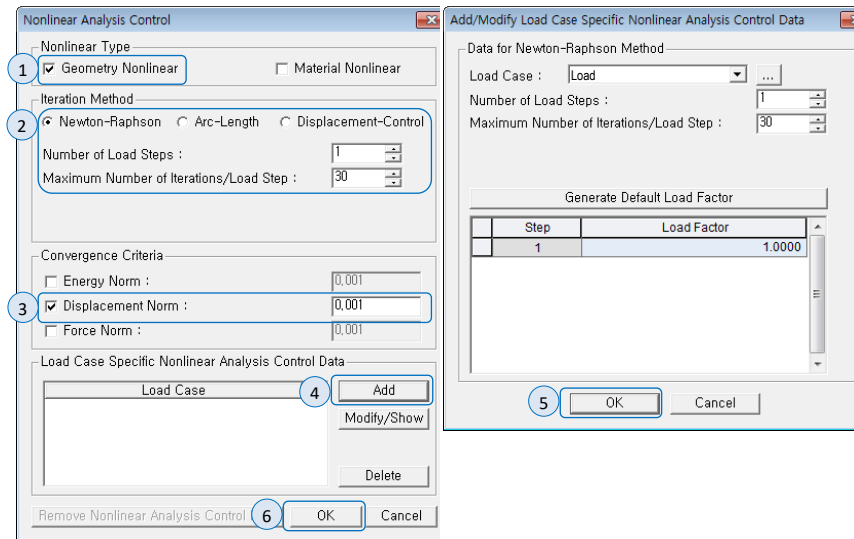
Input data to perform the geometric nonlinear analysis.

Main Menu > **Analysis** > **Nonlinear (Analysis Control...)**

1. Select Nonlinear Type > **Geometry Nonlinear** (on)
2. Select Iteration Method > **Newton-Raphson**
Number of Load Step : '1'
Maximum Number of Iteration/Load Step : '30'
3. Convergence Criteria > Displacement Norm: '0.001'
4. Click **[Add]**
5. Add Load Case Specific Nonlinear Analysis Control Data Dialogue box, Click **[OK]**
6. Nonlinear Analysis Control Dialogue box, Click **[OK]**

► Fig. 2.19

(a) Nonlinear Analysis Control
(b) Nonlinear Analysis Load Case

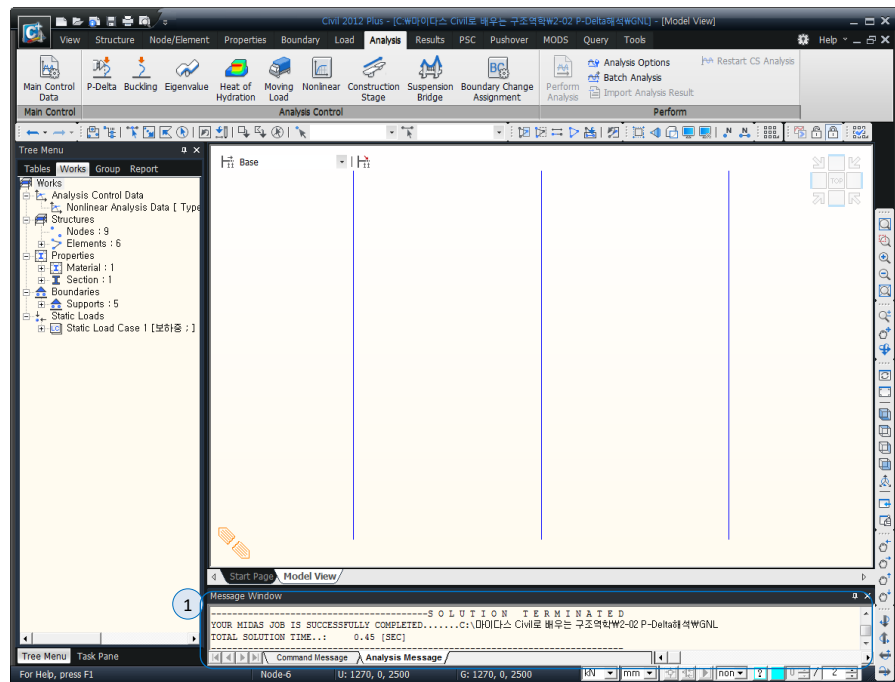


Perform geometric nonlinear analysis and compare to P-delta analysis results.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window

► Fig 2.20
Message for a
successful run

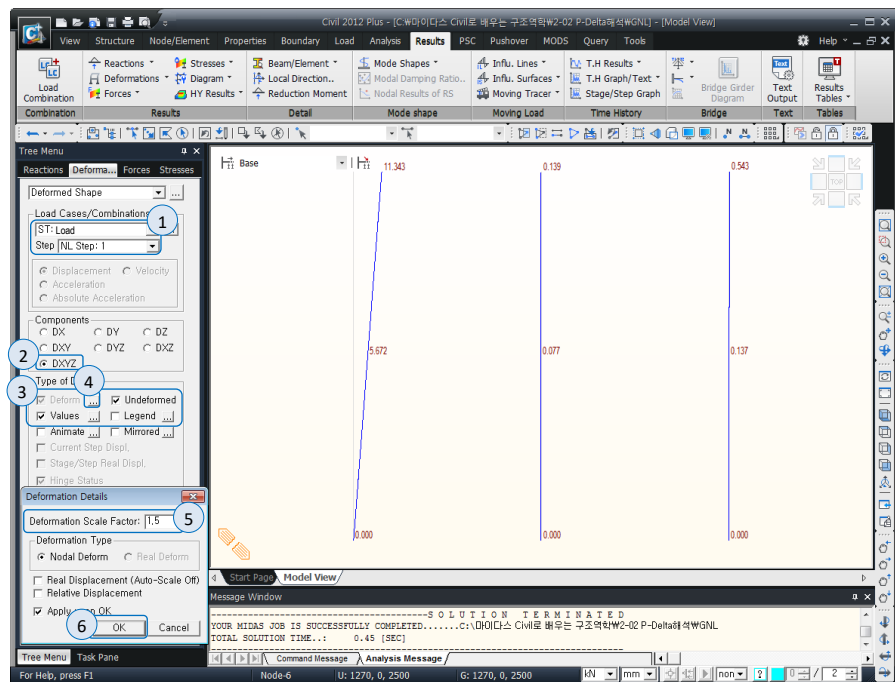


Check Deformed Shape by load cases.

Main Menu > **Results > Deformations > Deformed Shape...**

1. Select Load Cases / Combinations > **ST : Load** and Step : **NL Step : 1**
2. Select Components > **DXYZ**
3. Type of Display > **Undeformed, Values (on)**
4. Click [...] in Deform
5. Deformation Scale Factor : **'1.5'**
6. Click **[OK]**

► Fig. 2.21
Deformation Results

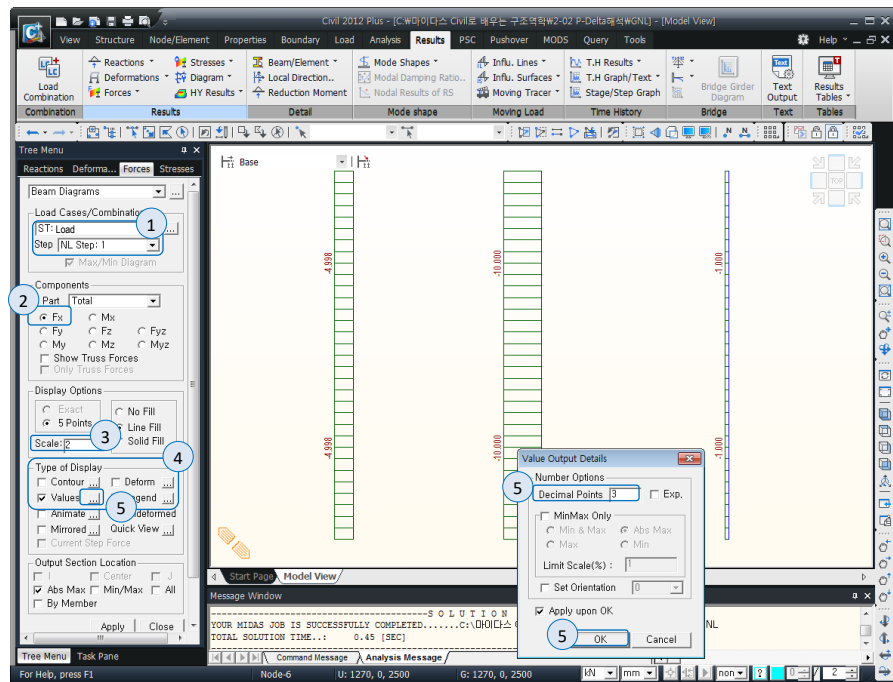


Check the column axial force by the load cases.

Main Menu > **Results** > **Forces** > **Beam Diagrams...**

1. Select Load Cases/Combinations > **ST : Load**, Step : **NL Step : 1**
2. Select Components > **FX**
3. Display Options > Scale : **'2'**
4. Type of Display > **Contour**, **Deform** (off), **Value** (on)
5. Click [...] in Values, Number Options > Decimal Points : **'3'**, Click **[OK]**

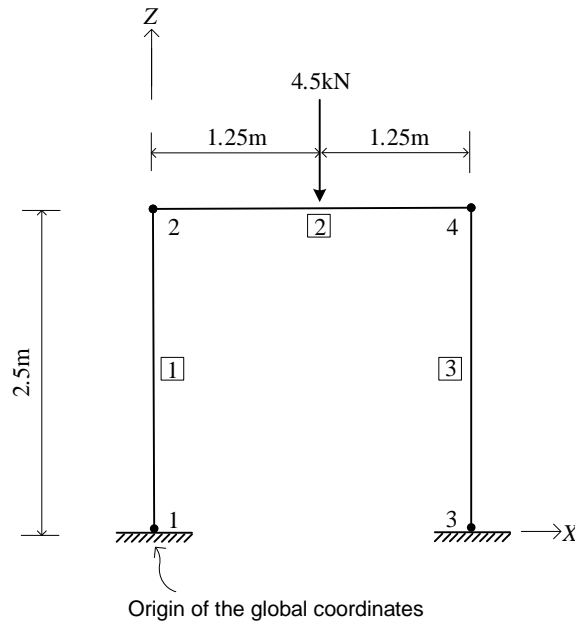
► Fig. 2.22
Axial Force Result





3. Exercise

Shown below is a 2-D, one story, one bay, portal frame supported at the base subjected to a symmetric load. Compare the displacements, shear forces and bending moments between the results based on a P-Delta effect analysis and a conventional frame analysis.



- **Material**
Concrete modulus of elasticity: $2.0 \times 10^5 \text{ N/mm}^2$
- **Section**
Area: 650 mm^2
Moment of inertia (I_{yy}): $3.5 \times 10^4 \text{ mm}^4$
- **Load**
Concentrated loads 4.5 kN at Node 2



3. Geometric Nonlinear Analysis

Contents

1 Introduction

1.1 Concept of P-delta Analysis	3-3
---------------------------------	-----

2 Tutorial

2.1 Model Overview	3-8
2.2 Work Environment	3-9
2.3 Material & Section Properties	3-11
2.4 Generate Nodes & Elements	3-12
2.5 Define Boundary Conditions	3-15
2.6 Define Loads	3-16
2.7 Perform Analysis	3-18
2.8 Check Analysis Result	3-20

3 Exercise	3-29
------------	------



1. Introduction

1.1 Geometric Nonlinear Analysis

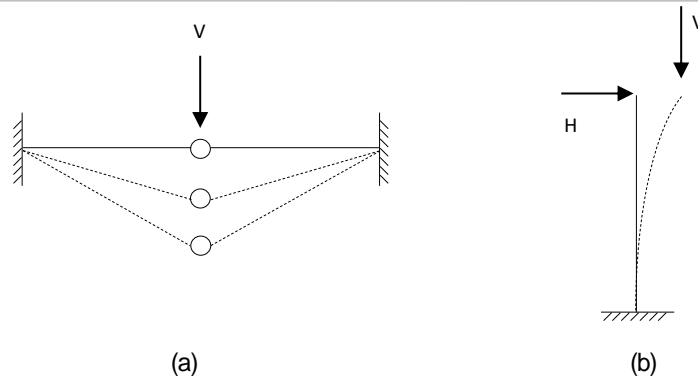
The assumption of linear behavior is valid in most structures. However, nonlinear analysis is necessary when stresses are excessive (Material Non-linearity) or large displacements (Geometric Non-linearity) exist in the structure. Construction stage analyses for suspension and cable stayed bridges are some of the examples of large displacement structure.

A geometric nonlinear analysis is carried out when a structure undergoes large displacements (displacements larger than the original dimensions of the element cross section) and the change of its geometric shape renders a nonlinear displacement-strain relationship. The geometric nonlinearity may exist even in the state of linear material behaviors. Cable structures such as suspension bridges are analyzed for geometric nonlinearity. A geometric nonlinear analysis must be carried out if a structure exhibits significant change of its shape under applied loads such that the resulting large displacements change the coordinates of the structure or additional loads like moments are induced (See Figure 3.1)

► Fig 3.1

Geometric nonlinear analyses

- (a) Change in structural stiffness due to large displacement
- (b) Additional load induced due to displacement

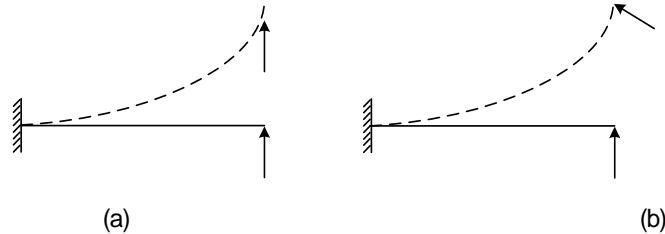


Also in Geometric non- linearity, the direction of the load remains unchanged or changes in accordance with the shape of the deformed structure. Figure 3.2(a) shows a fixed load direction and Figure 3.2(b) shows that the load follows geometry of the deformed structure. Generally loads applied on nodes maintain their original direction whereas those applied on elements may change in direction in accordance with the element deformation.



► Fig 3.2
Geometric Non-linear
Analysis

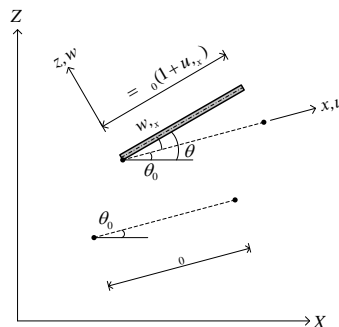
(a) Direction of load fixed
(b) Direction of Load
follows geometry



Geometric nonlinear analysis, P-delta analysis, and buckling analysis all depends on the same method. Linear static analysis is performed first for a given loading condition and then a new geometric stiffness matrix is formulated based on the member forces or stresses obtained from the first analysis. The geometric stiffness matrix is thus repeatedly modified and used to perform subsequent static analyses until the given convergence conditions are satisfied.

The difference lies in the axial length change and whether or not the member rotations are taken in consideration. The difference between the P-delta analysis and the buckling analysis lies in the kind of value obtained by the analysis. The P-delta analysis is a method to obtain the displacement due to the load and the buckling analysis differs in that the critical load at which the structure reaches the limit state due to secondary effects. Differences between geometric nonlinear analysis, P-delta analysis and buckling analysis can be explained using governing equations for truss elements. Figure 3.3 shows a Truss tilted by a force to an angle θ_o . The truss member has an inclined length l_o in the Global coordinate system (X-Z coordinate system). Let the deformation it undergoes in the element coordinate system (x-z coordinate system), be in u and w .

► Fig 3.3
Deformation State of
Truss Element





The element coordinate system axis x , is at an angle θ_o with respect to the global coordinate system axis X , and is represented using the global coordinate system as follows

Eq 3.1

$$\mathbf{KU} = \mathbf{P}$$

$$\mathbf{U} = \{\bar{u}_i \quad \bar{w}_i \quad \bar{u}_j \quad \bar{w}_j\}^T$$

$$\mathbf{P} = \{\bar{F}_i \quad \bar{V}_i \quad \bar{F}_j \quad \bar{V}_j\}^T$$

Where \mathbf{K} , \mathbf{U} and \mathbf{P} are the Stiffness, Displacement and Force matrix respectively.

The Capitalized letters represent the global coordinate system. The Truss is considered to have 4 degrees of freedom in X-Z plane and hence 4x4 matrix is adopted, as represented by Equation 3.1. The Stiffness matrix in Global system converted to element coordinate system using the Transformation matrix T ,

Eq 3.2

$$\mathbf{K} = \mathbf{T} \mathbf{k} \mathbf{T}^T = \mathbf{T} (\mathbf{k}_o + \mathbf{k}_\sigma) \mathbf{T}^T$$

$$\mathbf{T} = \begin{bmatrix} c & -s & 0 & 0 \\ s & c & 0 & 0 \\ 0 & 0 & c & -s \\ 0 & 0 & s & c \end{bmatrix} \quad \text{where} \quad c = \cos \theta_o, \quad s = \sin \theta_o$$

Equation 3.2 shows Transformation matrix T . Equation 3.3 represents the resulting matrix after multiplication with the coordinate transformation matrix T

Eq 3.3

$$\mathbf{T} \mathbf{k}_o \mathbf{T}^T = \frac{EA}{l_o} \begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix}, \quad \mathbf{T} \mathbf{k}_\sigma \mathbf{T}^T = \frac{N}{l_o} \begin{bmatrix} s^2 & -sc & s^2 & sc \\ -sc & c^2 & sc & -c^2 \\ s^2 & sc & s^2 & -sc \\ sc & -c^2 & -sc & c^2 \end{bmatrix}$$

In Equations 3.2 and 3.3, it can be seen that the angle θ_o between the member with length l_o and the global coordinate system is unchanged. As described above, member length and angle do not change is the assumption used in ordinary P-delta analysis or buckling analysis. However, in the geometric nonlinear analysis, the member length and the inclination angle change as follows.

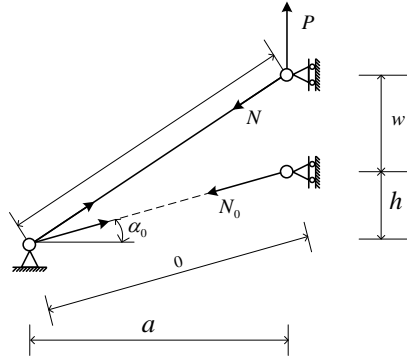
Eq 3.4

$$l = l_o (1 + u_{,x}), \quad \theta = \theta_o + w_{,x}$$

Since the values of Eq. (3.2) and Eq. (3.3) must be continuously updated in the analysis process using the changed values, the answers are obtained through iterative calculations.

Since the concept of geometric nonlinear analysis can be difficult to understand due to complicated equations, a simple truss member is used. The truss member shown in the following Figure 3.4 shows a state in which the member is deformed by a force, P . This force has not only changed the length but also the geometric shape. So computing the force equilibrium between the load P and the member axial force N , post-deformation is stated as follows.

► Fig 3.4
Non-linear Truss
Structure



Force equilibrium conditions before and after deformation of the structure in Fig. 3.4 are as follows.

Eq 3.5 Before deformation: $N_o \frac{h}{l_o} - P = 0$

Eq 3.6 After deformation: $N \frac{w+h}{l} - P = 0$

In the geometric nonlinear analysis, to account for deformation of the higher-order, Green-Lagrange Strain is used, instead of the nominal strain (Engineering strain) as in the case of the linear analysis.

Eq 3.7
$$\varepsilon_G = \frac{l^2 - l_o^2}{2l_o^2}$$



Assuming that the material behaves linearly, using the strain definitions described above, the axial force of the member is obtained as follows.

$$\text{Eq 3.8} \quad N - N_o = EA\varepsilon_G = EA \frac{l^2 - l_o^2}{2l_o^2} = \frac{EA}{l_o^2} \left(hw + \frac{1}{2} w^2 \right)$$

Assuming small deformation ($l/l_o \approx 1$), from the equilibrium condition Equation 3.6, the relationship between the load P and the displacement w can be estimated.

$$\text{Eq 3.9} \quad P = N \frac{h+w}{l} \approx N \frac{h+w}{l_o} = \frac{EA}{l_o^3} \left(hw + \frac{1}{2} w^2 \right) (h+w) + \frac{N_o}{l_o} (h+w)$$

From the load-displacement relation of Equations. (3.5) and (3.6), the stiffness in the vertical direction can be expressed as follows.

$$\text{Eq 3.10} \quad k = \frac{dP}{dw} = \frac{EA}{l_o^3} \left[(h+w)(h+w) + (hw + \frac{1}{2} w^2) \right] + \frac{N_o}{l_o} = k_o + k_L + k_\sigma$$

$$k_o = \frac{EA}{l_o} \left(\frac{h}{l_o} \right)^2, \quad k_L = \frac{3EA}{2l_o} \left[2 \frac{w}{h} + \left(\frac{w}{h} \right)^2 \right] \left(\frac{h}{l_o} \right)^2, \quad k_\sigma = \frac{N}{l_o}$$

In Equation 3.10, the stiffness k_o is the same as that of the linear analysis, k_L and k_σ the stiffness reflects the geometric nonlinearity. k_L , allows to consider the effect of the geometric shape change of the structure on the stiffness. It can be seen that there is a new axial force term, k_σ . If the axial force is compressive, the length of the member is reduced, and thus the value becomes negative. If the axial force is tensile force, it becomes positive.

In order to show the difference from the linear analysis, the stiffness k_o of the linear analysis can also be calculated by the following method.

$$\text{Eq 3.11} \quad \varepsilon_E = \frac{l - l_o}{l_o} \approx \frac{1}{l_o} \left(w \frac{h}{l_o} \right)$$

$$P = N \frac{h}{l_o} = (AE\varepsilon_E) \frac{h}{l_o} = \frac{EA}{l_o} \left(w \frac{h}{l_o} \right) \frac{h}{l_o} = \frac{EA}{l_o} \left(\frac{h}{l_o} \right)^2 w, \text{ where } k_o = \frac{EA}{l_o} \left(\frac{h}{l_o} \right)^2$$

From the above procedure, it can be seen that the geometric nonlinear analysis considers changes in the member length and the inclination angle unlike the P-delta and Buckling analyzes. As a result, geometric nonlinear analysis adds k_L , to the stiffness for P-delta analysis

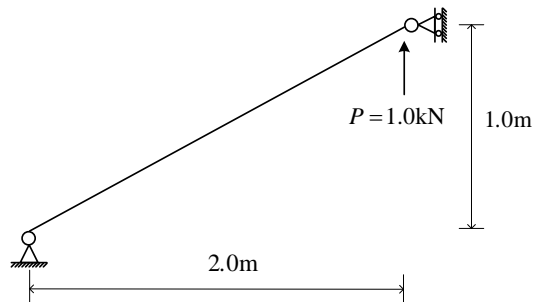


2. Tutorial

2.1 Model Overview

Perform the geometry nonlinear analysis of a simple truss structure as shown in Fig 3.5, and compare the analysis result between linear analysis result and the P-delta analysis result.

Fig. 3.5
Structural geometry &
analysis model




- **Material**
Modulus of Elasticity : $1.0 \text{ N} / \text{mm}^2$
- **Section**
Section (Area) : $1.0 \times 10^6 \text{ mm}^2$
- **Load**
Concentrated load on roller support : 1kN

2.2 Work Environment

Open a new file and save the file name as 'Nonlinear.mcb'.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. File name : '**Nonlinear**', Click **[SAVE]**

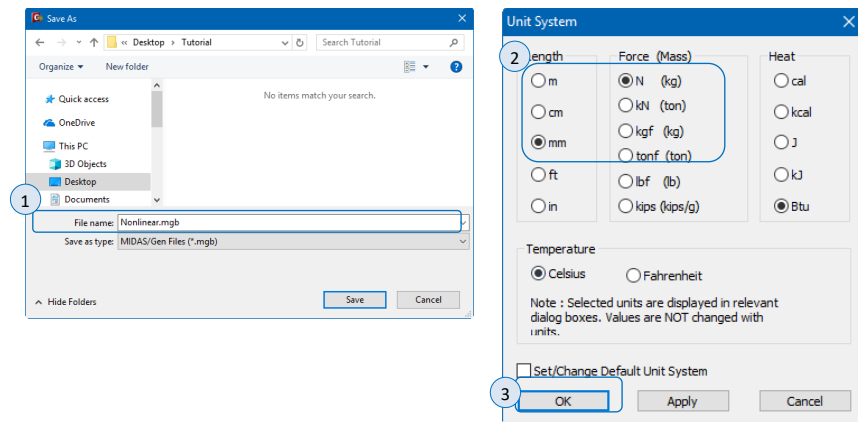
Set the unit system to use

Main Menu > **Tools > Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click **[OK]**

► Fig 3.6
Define unit system





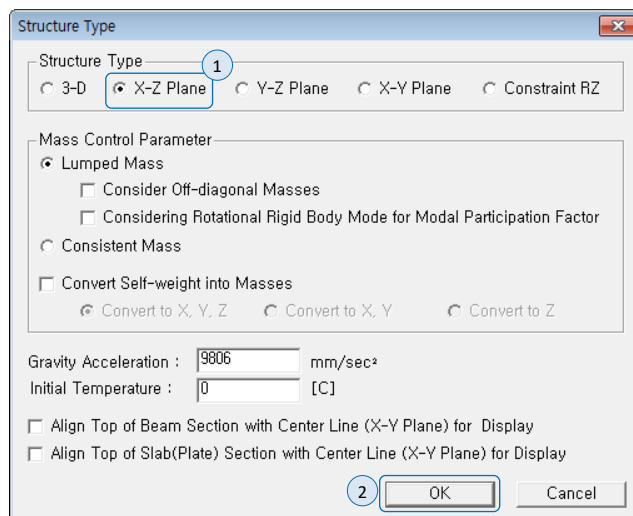
The example models exist in the 2-D, X-Z plane.

Main Menu > **Structure** > **Structure Type**

1. Select Structure Type > **X-Z Plane**

2. Click **[OK]**

► Fig 3.7
Set a work plane



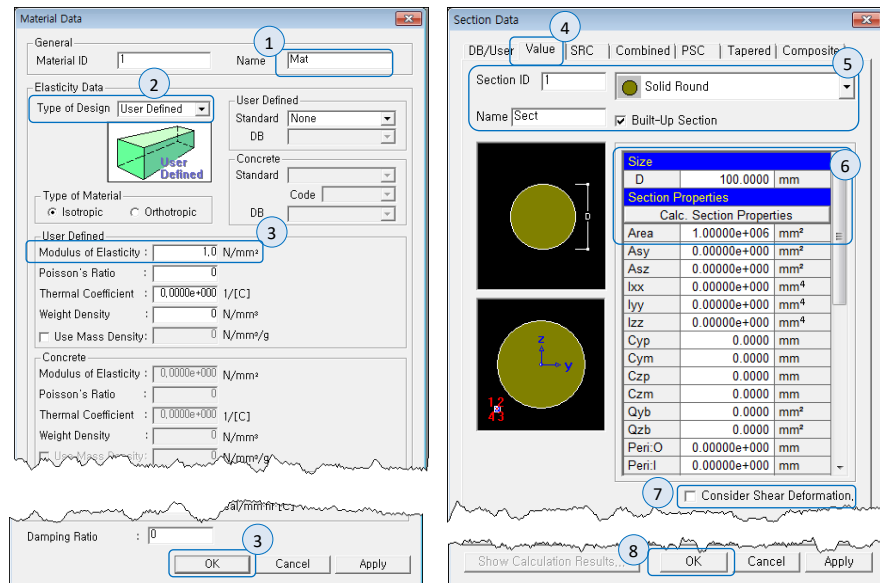
2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu > **Properties** > **Material Properties**

1. Click **[Add...]**, Name : **'Mat'**
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : **'1.0'**, Click **[OK]**
4. Click **Section** Tab and **[Add...]** and **Value** Tab
5. Select Section Shape Lists > **Solid Round**, Section ID : **'1'**, Name : **'Sect'**
6. Size > D : **'100'**, Section Properties > Area : **'1e6'**
7. **Consider Shear Deformation** (off)
8. Click **[OK]** and **[Close]**

► Fig 3.8
Define material & section properties



2.4 Generate Node & Element

Create a node in order to generate beam elements.

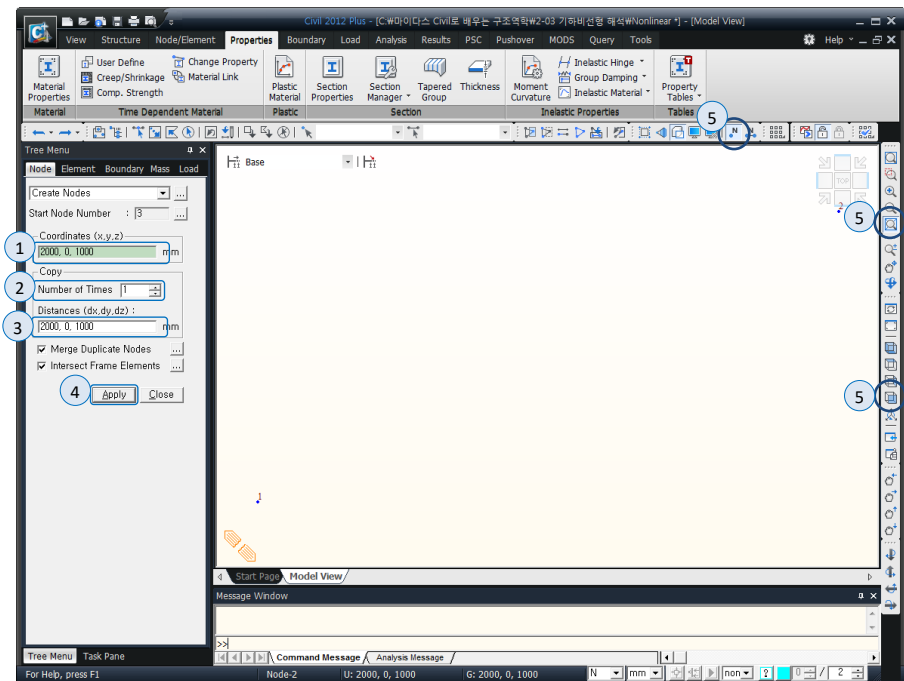
Main Menu > **Node/Element** > **Create Nodes**

1. Coordinates (x, y, z) : '0, 0, 0'
2. Copy > Number of Times: '1'
3. Distances (dx, dy, dz) : '2000, 0, 1000'
4. Click [Apply]
5. Display Node Numbers, Auto Fitting, Front View (on)

► Fig 3.9
Create a node



When Auto Fitting is toggled on, the model fits into the full screen, which automatically controls Zoom Size in real time.

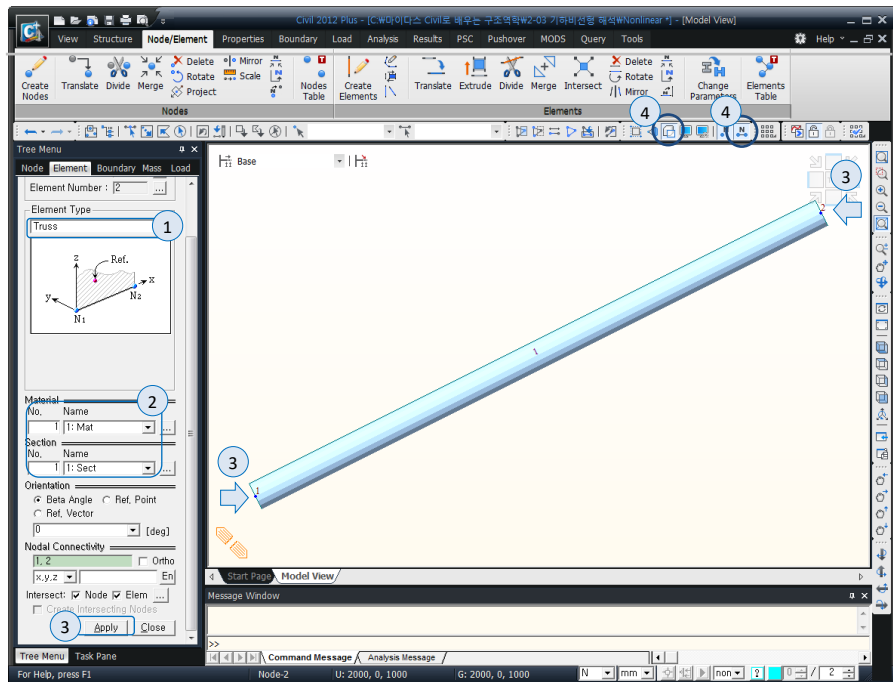


Connect the nodes to generate the element.

Main Menu > **Node/Element** > **Create Elements**

1. Select Element Type > **Truss**
2. Select Material > **1:Mat** and Section > **1:Sect**
3. Click Nodal Connectivity green box, and Click node number 1 and 2 in Model view
4. **Hidden, Display Element Numbers** (on)

► Fig 3.10
Create a beam



Divide elements to see detailed results.

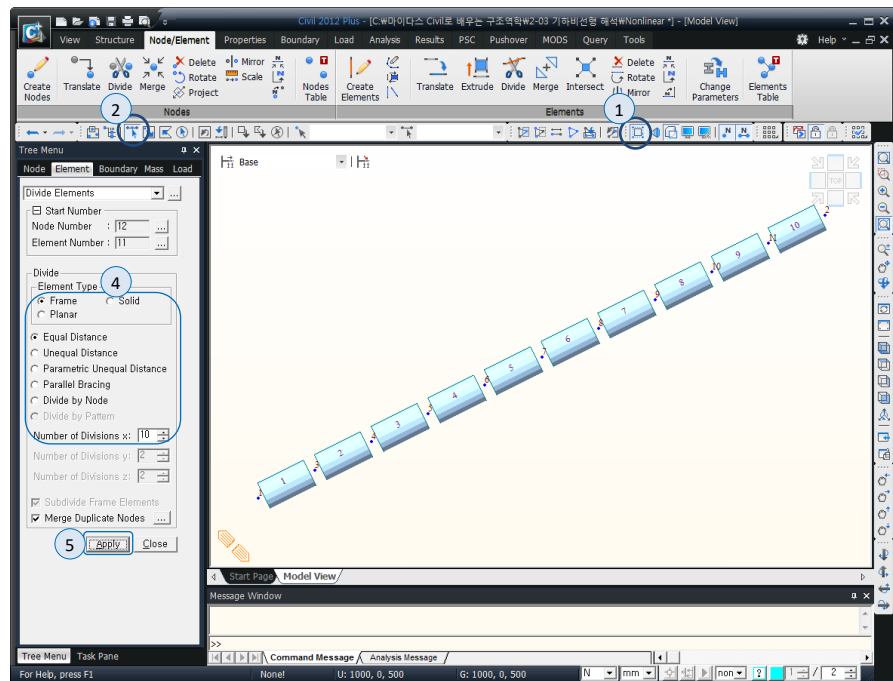
Main Menu > **Node/Element** > **Elements** > **Divide**

1. **Shrink** (on)
2. **Select Single** (on)
3. Select element no. 1 in model view
4. Select Divide > Element Type > Frame
Select Divide > Equal Distance
Number of Division x : '10'
5. Click **[Apply]**

► Fig 3.11
Divide element

Tip
By modeling one member by dividing it into several elements, you can see the detailed results of the member.

Tip
Shrink is useful for confirming member connection status based on the nodal point. When a member is connected to a node, it is divided as shown in the figure.



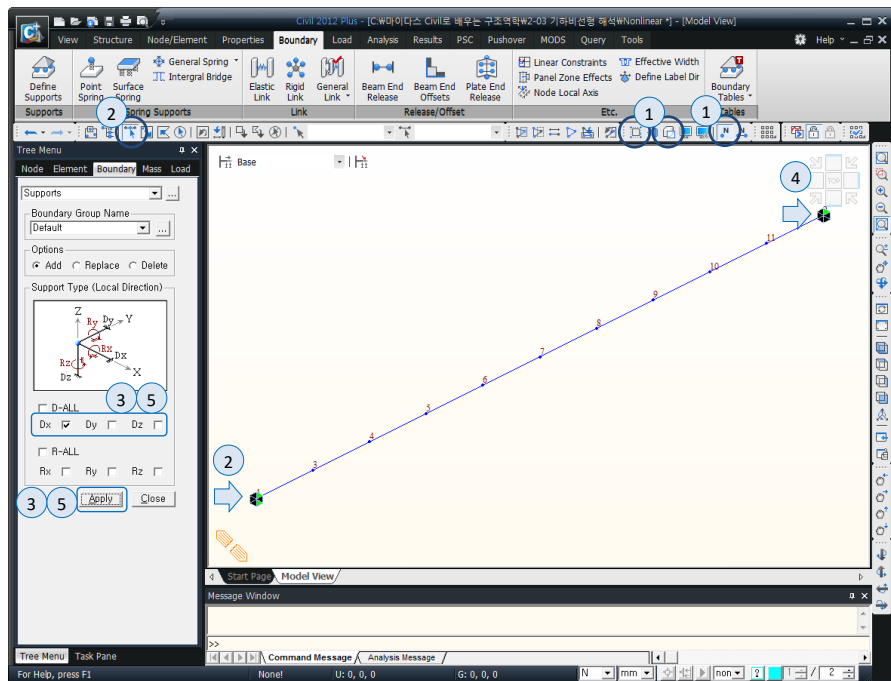
2.5 Define boundary conditions

Define boundary conditions at pin support of left end, roller support of right end.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. **Shrink, Hidden, Display Element Numbers** (off)
2. Click **Select Single** (on), Select node number 1
3. Support Type > **Dx, Dz** (on), Click **[Apply]**
4. Click **Select Single** (on), Select node number 2
5. Support Type > **Dx** (on), **Dz** (off), Click **[Apply]**

► Fig 3.12
Define boundary conditions





2.6 Define loads

Define load conditions (Load Type) first to which the loading will belong.

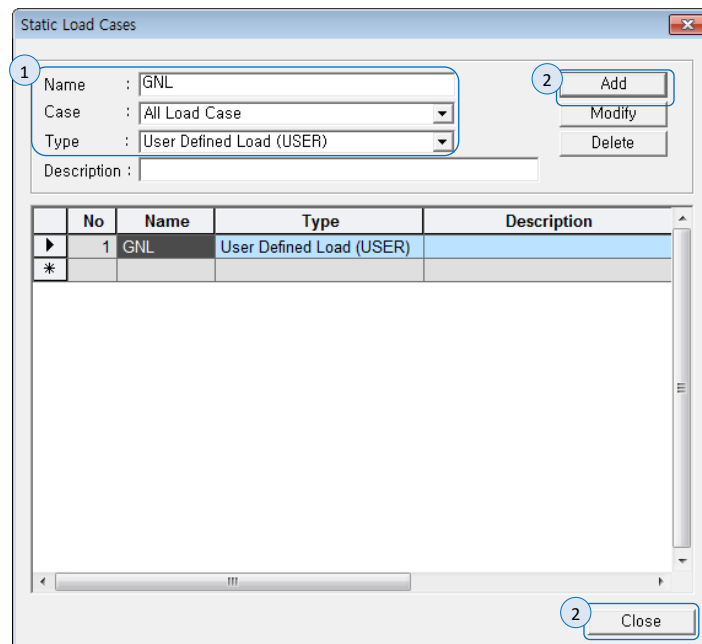
Main Menu > **Load** > **Static Loads** > **Static Load Cases**

1. Name : '**GNL**'

Select Type > **User Defined Load (USER)**

2. Click **[Add]** and **[Close]**

► Fig 3.13
Define load case



Input a concentrated load, 1000 N at node 2.

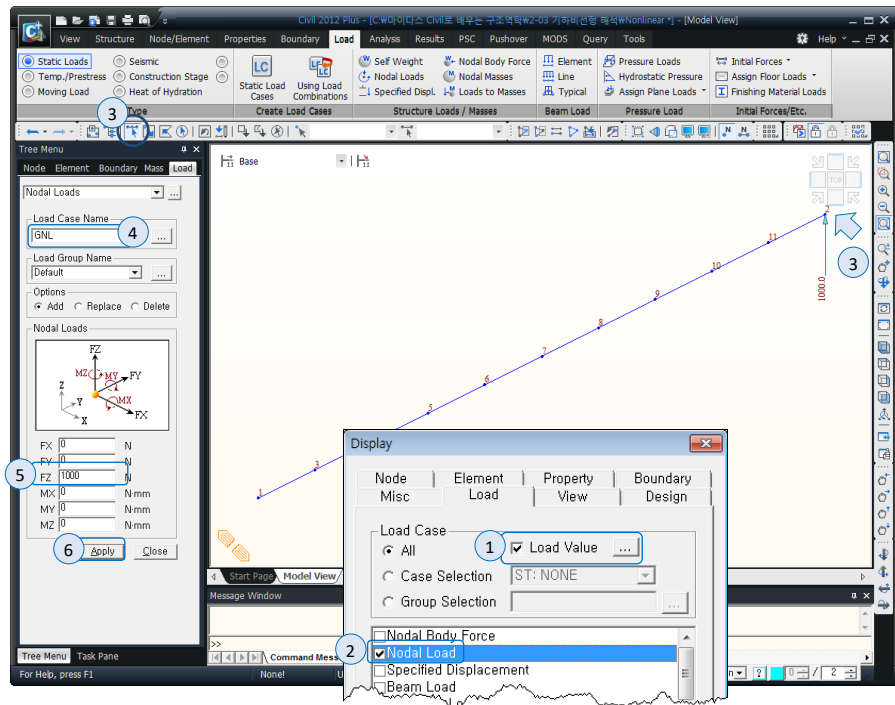
Main Menu > **View > Display**

1. Select Load Tab > Load Case > **Load Value** (on)
2. **Nodal Load** (on), Click **[OK]**

Main Menu > **Load > Static Loads > Nodal Loads**

3. Click **Select Single** (on), Select node number **2**
4. Select Load Case Name > **GNL**
5. Nodal Loads > FZ > **'1000'**
6. Click **[Apply]**

► Fig 3.14
Input Nodal load





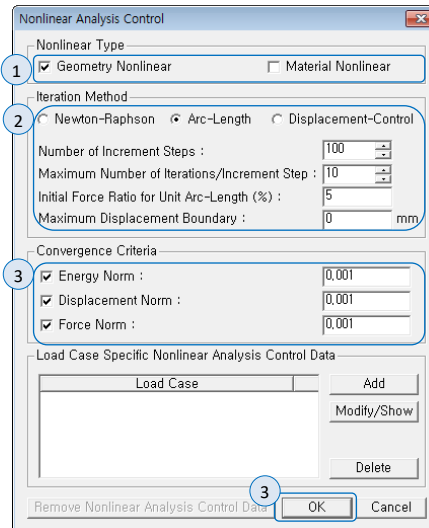
2.7 Perform Analysis

Define analysis conditions of geometric nonlinear analysis.

Main Menu > **Analysis** > **Analysis Control** > **Nonlinear**

1. Select Nonlinear Type > **Geometry Nonlinear**
2. Select Iteration Method > **Arc Length**
Number of Increment Step : '100'
Maximum Number of Iterations/Increment Step : '10'
Initial Force Ratio for Unit Arc-Length (%) : '5'
Maximum Displacement Boundary : '0'
3. Convergence Criteria > Energy Norm : '0.001', Displacement Norm : '0.001',
Force Norm : '0.001', Click [OK]

► Fig 3.15
Nonlinear Analysis Control



Since it is a simple nonlinear analysis model, skyline of equation solver is useful as general purpose analysis

Main Menu > **Analysis** > **Perform** > **Analysis Options**

1. Equation Solver > **Skyline**, Click [OK]

Main Menu > **Analysis** > **Perform Analysis**

2. Check for successful completion in Message Window

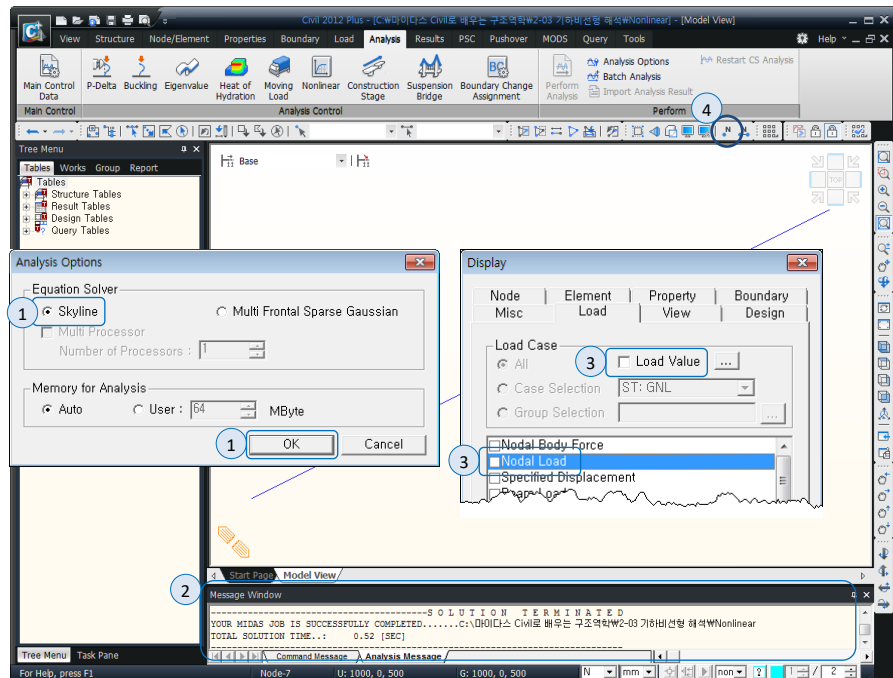
For the convenience of reviewing the analysis result, the text displayed on the screen is organized.

Main Menu > **View** > **Display**

3. Select Load Tab > Nodal Load (off), Click [OK]

4. **Display Node Numbers** (off)

► Fig 3.16
Analysis options



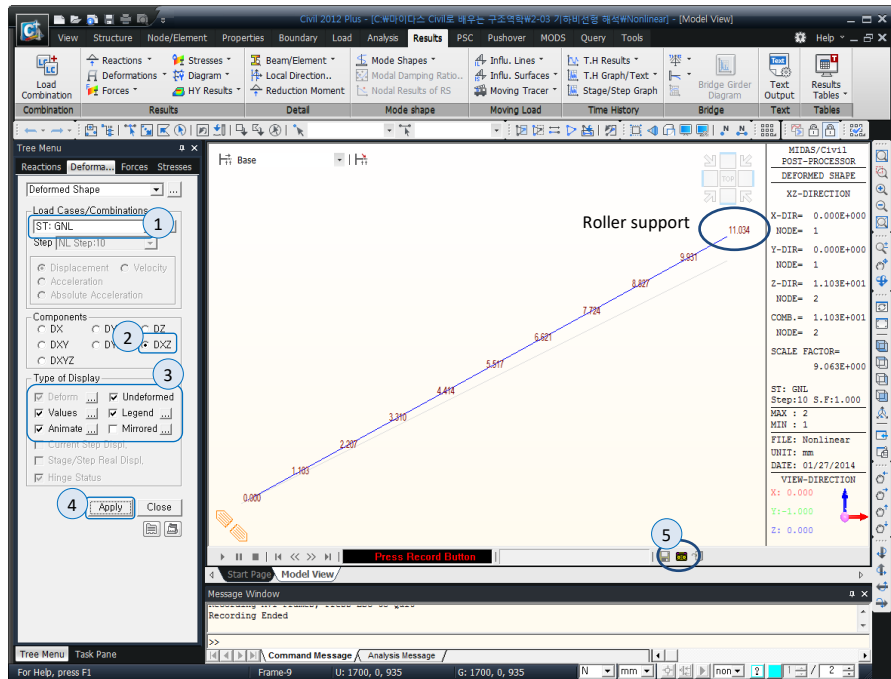
2.8 Check Analysis Result

Check the transformation using the animation function.

Main Menu > **Results** > **Deformations** > **Deformed Shape...**

1. Select Load Cases / Combinations > **ST : GNL**
 2. Select Components > **DXZ**
 3. Type of Display > **Undeformed, Values, Legend, Animate** (on)
 4. Click **[Apply]**
 - 5 Click the Record button at the bottom of the model view (see the following figure)
- After checking the deformed shape, click the Exit button (see the following figure)

► Figure 3.17
Deformation results



3. Geometric Nonlinear Analysis

Check the axial force among the load cases and combination.

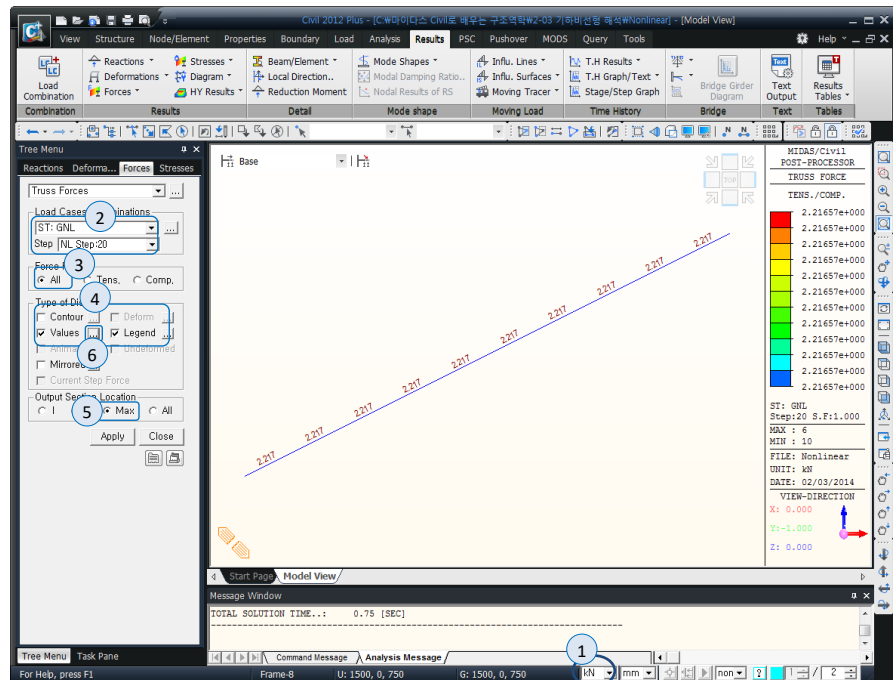
Main Menu > **Tools** > **Unit System...**

1. Select Length > **mm**, Force(Mass) > **kN**, Click**[OK]**

Main Menu > **Results** > **Forces** > **Truss Forces...**

2. Select Load Cases/Combinations > **ST : GNL** and Step : **NL Step : 20**
3. Select Force Filter > **All**
4. Type of Display > **Contour, Deform** (off), **Value, Legend** (on)
5. Select Output Section Location > **Max**
6. Click [...] in Values, Number Options > Decimal Points : **'3'**, Click **[OK]**

► Fig 3.18
Axial force results





Save as 'Nonlinear-PDelta.mgb' to compare to geometric nonlinear analysis.

Main Menu > **Files > SaveAs...**

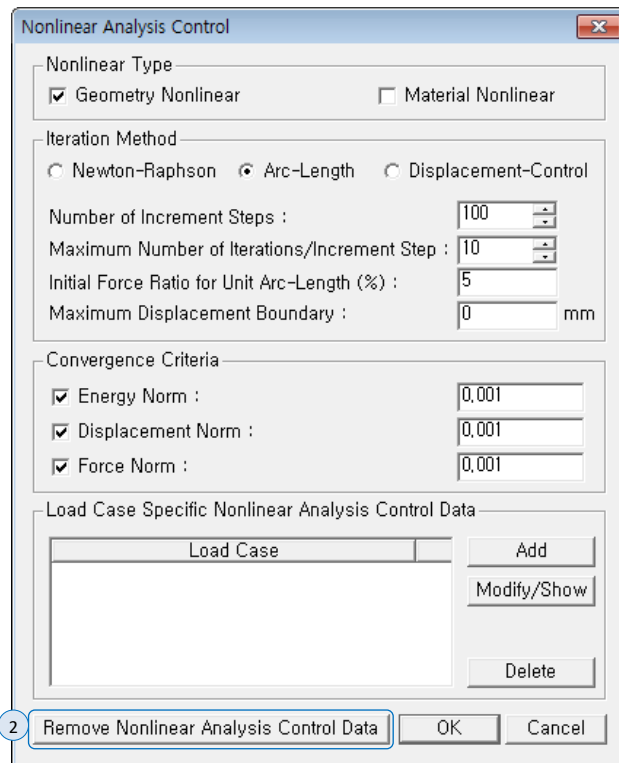
1. Enter a name : '**Nonlinear-PDelta**', Click [**SAVE**]

The geometric nonlinear analysis condition is deleted to perform the P-delta analysis.

Main Menu > **Analysis > Nonlinear Analysis Control...**

2. Click [**Remove Nonlinear Analysis Control Data**]

► Fig 3.19
Remove Nonlinear
Analysis Condition



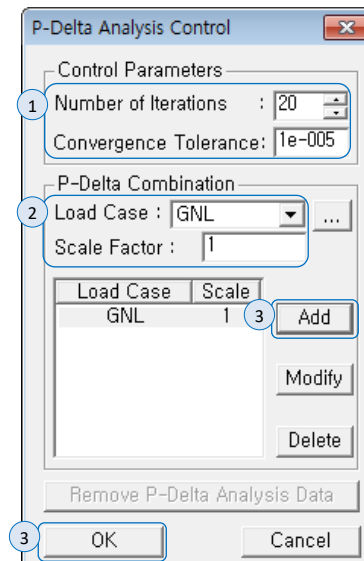


Input data to perform P-delta analysis.

Main Menu > **Analysis > P-Delta Analysis Control...**

1. Control Parameters > Number of Iteration : '20'
Convergence Tolerance : '1e-005'
2. P-Delta Combinations > Load Case : **GNL**, Scale Factor : '1'
3. Click **[Add]** and **[OK]**

► Fig 3.20
P-delta Analysis Control



Perform P-delta analysis

Main Menu > **Analysis > Perform Analysis**

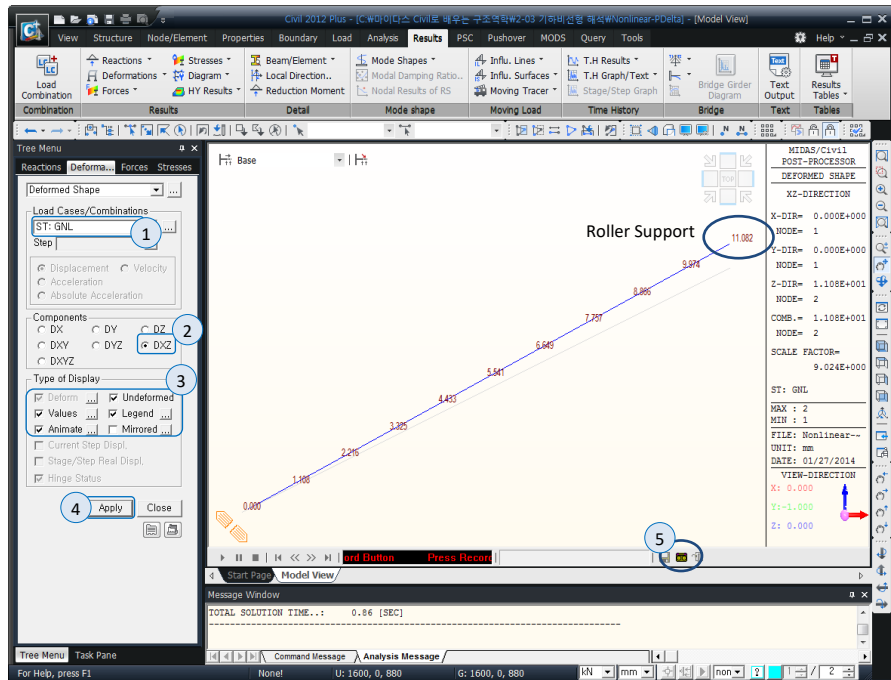
1. Check the message for successful run

Check the deformed shape and animation function.

Main Menu > **Results > Deformations > Deformed Shape...**

1. Select Load Cases / Combinations > **ST : GNL**
2. Select Components > **DXZ**
3. Type of Display > **Undeformed, Values, Legend, Animate (on)**
4. Click **[Apply]**
5. Click the Record button at the bottom of the model view (see the following figure). After checking the deformed shape, click the Exit button (see the following figure)

► Fig 3.21
Deformation results



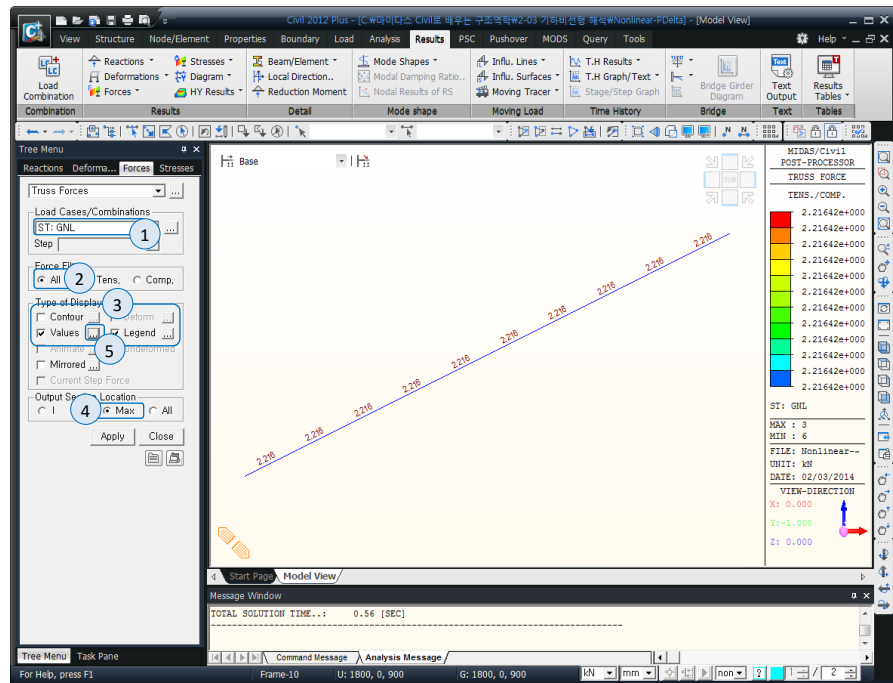
3. Geometric Nonlinear Analysis

Check the axial force.

Main Menu > **Results** > **Forces** > **Truss Forces...**

1. Select Load Cases/Combinations > **ST : GNL**
2. Select Force Filter > **All**
3. Type of Display > **Contour**, **Deform** (off), **Value**, **Legend** (on)
4. Select Output Section Location > **Max**
5. Click [...] in Values , Number Options > Decimal Points : '3', Click **[OK]**

► Fig 3.22
Member force





Save as 'Nonlinear-linear.mgb' to compare with linear analysis.

Main Menu > **Files > SaveAs...**

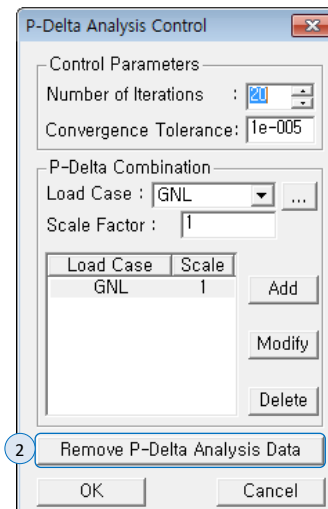
1. Enter a name : '**Nonlinear-linear**', Click **[SAVE]**

Remove the P-delta analysis condition to perform the linear analysis.

Main Menu > **Analysis > P-Delta Analysis Control...**

2. Click **[Remove P-Delta Analysis Data]**

► Figure 3.23
Remove P-delta analysis



Perform linear analysis.

Main Menu > **Analysis > Perform Analysis**

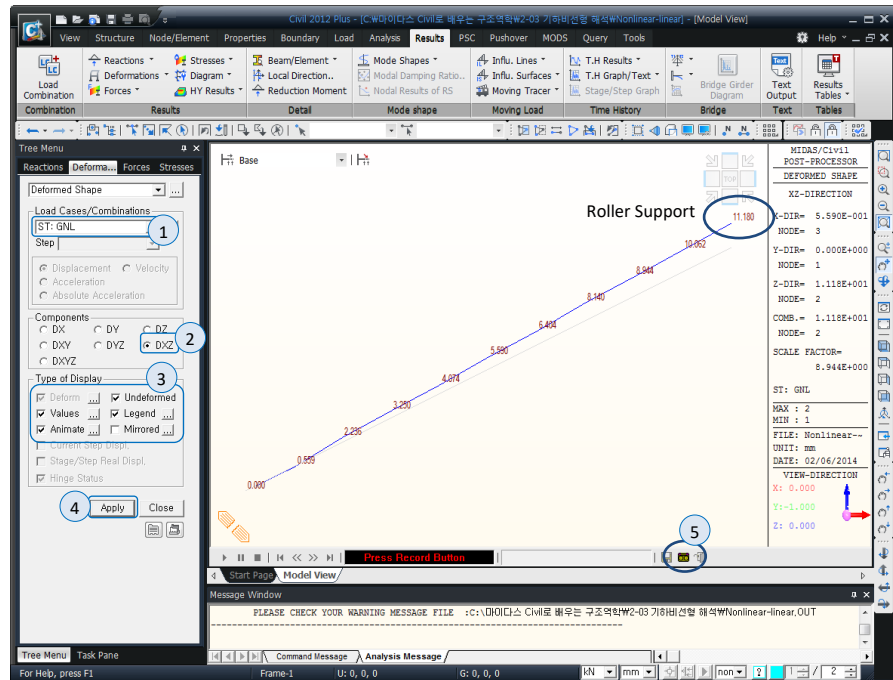
1. Check the message for successful run

Check the deformed shape and animation function.

Main Menu > **Results** > **Deformations** > **Deformed Shape...**

1. Select Load Cases / Combinations > **ST : GNL**
 2. Select Components > **DXZ**
 3. Type of Display > **Undeformed, Values, Legend, Animate (on)**
 4. Click **[Apply]**
 5. Click the Record button at the bottom of the model view (see the following figure).
- After checking the deformed shape, click the Exit button (see the following figure)

► Figure 3.24
Deformation shape



Structural Analysis II (Advanced)

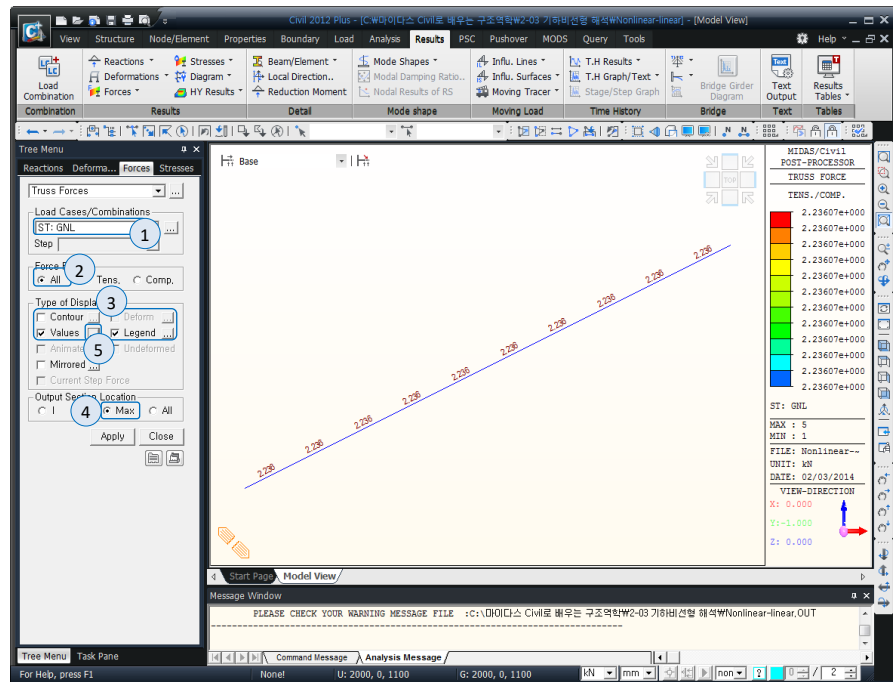
3. Geometry Nonlinear Analysis

Check the axial force acting on the member.

Main Menu > **Results** > **Forces** > **Truss Forces...**

1. Select Load Cases/Combinations > **ST : GNL**
2. Select Force Filter > **All**
3. Type of Display > **Contour**, **Deform** (off), **Value**, **Legend** (on)
4. Select Output Section Location > **Max**
5. Click [...] in Values, Number Options > Decimal Points : '3', Click **[OK]**

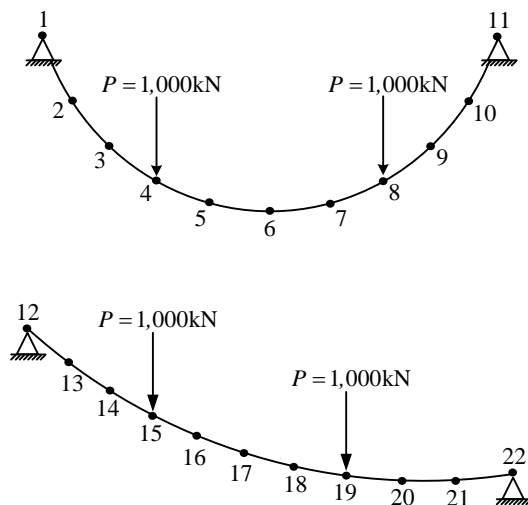
► Figure 3.25
Truss forces





3. Exercise

Compare the member force, displacement and reaction among two structure of different shape geometry below.



- **Material**
Modulus of elasticity: $2.05 \times 10^5 \text{ N/mm}^2$
- **Section**
Sectional Shape: Cylinder
Size : 100 mm
- **Load**
Concentrated Load in the (-)1000 kN



Node	X(m)	Y(m)	Z(m)
1	0,000000	0,000000	0,000000
2	9,919920	0,000000	-10,327700
3	21,520200	0,000000	-18,724100
4	34,430200	0,000000	-24,920900
5	48,237200	0,000000	-28,720000
6	62,500000	0,000000	-30,000000
7	76,762800	0,000000	-28,720000
8	90,569800	0,000000	-24,920900
9	103,480000	0,000000	-18,724100
10	115,080000	0,000000	-10,327700
11	125,000000	0,000000	0,000000
12	0,000000	0,000000	-100,000000
13	10,874600	0,000000	-107,537000
14	22,321000	0,000000	-114,173000
15	34,264900	0,000000	-119,865000
16	46,628500	0,000000	-124,576000
17	59,331500	0,000000	-128,276000
18	72,291400	0,000000	-130,940000
19	85,423800	0,000000	-132,552000
20	98,643300	0,000000	-133,100000
21	111,864000	0,000000	-132,582000
22	125,000000	0,000000	-131,000000



4. Buckling Analysis

Contents

1 Introduction

1.1 Concept of Buckling Analysis	4-3
----------------------------------	-----

2 Tutorial

2.1 Model Overview	4-8
2.2 Work Environment	4-9
2.3 Material & Section Properties	4-11
2.4 Generate Node & Element	4-12
2.5 Define Boundary Conditions	4-14
2.6 Define Loads	4-15
2.7 Perform Analysis	4-17
2.8 Check Analysis Result	4-19

3 Exercise	4-25
------------	------



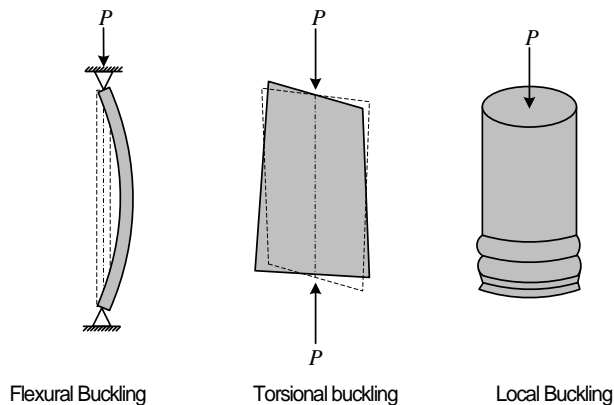
1. Introduction

1.1 Concept of Buckling Analysis

If an incremental compressive force acts on a slender column or a thin plate is incrementally loaded in the in-plane direction, there comes an increment of load when sudden lateral bending is observed. This is defined as buckling behavior. Buckling in simple terms is structural instability wherein the geometric stiffness governs the failure.

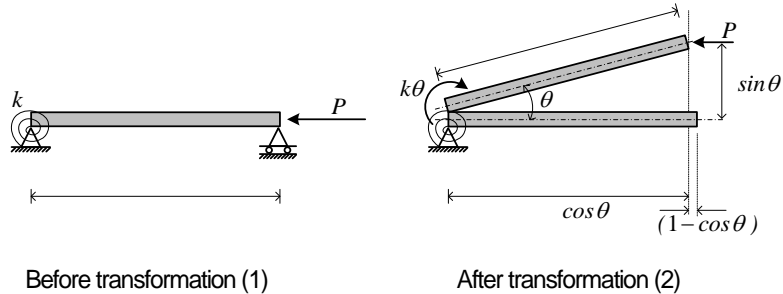
Until the buckling occurs, the element remains in stable equilibrium, once buckling occurs, the lateral deformation is no longer linear to the applied load. The buckling is not only limited to the flexural buckling of slender columns or thin plates. Local buckling occurs for a local area of an element, like when shear acts upon the web plates of a plate girder, it also tends to buckle causing local shear buckling; similarly torsional buckling can occur in members weak in torsion leading to rotation of element about the force axis. Also flexural-torsional buckling is the combination of bending and twisting response of a member in compression which mostly occurs in open cross sections. Lateral torsional buckling occurs for a beam stiff in bending plane but weak in transverse plane particularly happens for laterally unsupported compression flanges of an I cross sections.

► Fig 4.1
Buckling Types



Buckling causes imperfections of the shape and the loading axis is modified for the buckled elements.

► Fig 4.2



Applying Moment Equilibrium at point 2, we get following equation

► Eq 4.1

$$-k\theta + Pl \sin \theta = 0$$

If small strain is assumed, then $\sin \theta \approx \theta$,

► Eq 4.2

$$(-k + Pl) \theta = 0$$

If θ is a finite value then values in the parenthesis should be equal to 0.

► Eq 4.3

$$P_{cr} = \frac{k}{l}$$

So, instead of buckling deformation, the above equation is used to obtain buckling shape under critical loading conditions.

For the buckling of columns with stiffness, differential equation Eq.1.4 and Eq.1.5 from chapter 1 can be used. Buckling of hinged ends of the columns is ($EIw'''' + Pw'' = 0$)

So it can be expressed as

► Eq 4.4

$$w(x) = A \sin kx + B \cos kx + Cx + D$$

$$\text{Where, } k = \sqrt{\frac{P}{EI}}$$



Substituting for boundary conditions.

► Eq 4.5

$$\begin{aligned}
 w(0) &= B + D = 0 \\
 w(1) &= A \sin kl + B \cos kl + C + D = 0 \\
 w''(0) &= -Bk^2 = 0 \\
 w''(1) &= -Ak^2 \sin kl - Bk^2 \cos kl = 0
 \end{aligned}$$

Which can be represented by following form,

► Eq 4.6

$$\begin{bmatrix} 0 & 1 & 0 & 1 \\ \sin kl & \cos kl & 1 & 1 \\ 0 & -k^2 & 0 & 0 \\ -k^2 \sin kl & -k^2 \cos kl & 0 & 0 \end{bmatrix} \begin{Bmatrix} A \\ B \\ C \\ D \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

From Eq. 4.5, $B = D = 0$ it can inferred that,.

► Eq 4.7

$$\begin{bmatrix} \sin kl & 1 \\ -k^2 \sin kl & 0 \end{bmatrix} \begin{Bmatrix} A \\ C \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

As A & C are non-zero entities, the determinant should be equal to zero. Hence,

► Eq 4.8

$$\begin{aligned}
 \sin kl &= 0 & \text{Where,} & \quad kl = n\pi \\
 \text{Here,} & & n & \text{is natural subset of } (1 \sim \infty).
 \end{aligned}$$

Therefore, the critical buckling load at the ends of hinged column is,.

► Eq 4.9

$$P_{cr} = \left(\frac{n\pi}{1} \right)^2 EI$$

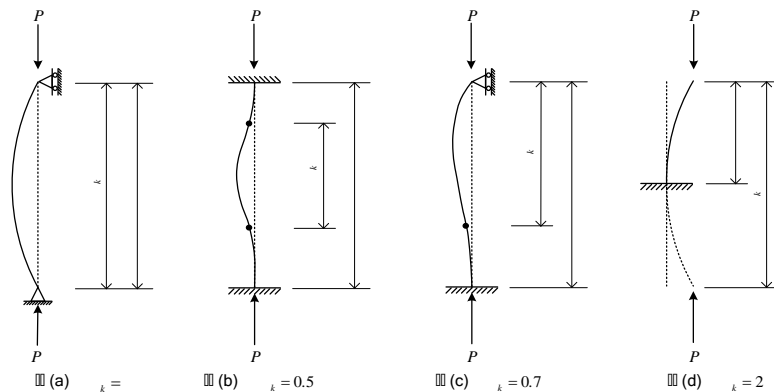
By using Eq. 1.5 one can get the same results. For other support conditions, the critical buckling load results are as shown in table 4.1



► Table 4.1
Buckling Analysis
based on support
condition at both end

Support Condition	P_{cr}	l_k
Both Ends Hinged	$\left(\frac{n\pi}{1.0l}\right)^2 EI$	1.0l
Both Ends Fixed	$\left(\frac{n\pi}{0.5l}\right)^2 EI$	0.5l
One end Hinged, other end fixed	$\left(\frac{n\pi}{0.7l}\right)^2 EI$	0.7l
One end fixed, other end free	$\left(\frac{n\pi}{2.0l}\right)^2 EI$	2.0l

► Fig 4.3
Buckling Analysis
based on support
condition at both end



The calculation of buckling analysis is same as P-delta analysis. The main difference is P-analysis directly obtains displacement value, whereas buckling in the determinant format obtains shape, size and variation of critical load using boundary condition as zero. At the surface, using determinant format and boundary condition as 0, the expression is same as buckling eigenvalue analysis. The critical buckling load in a mathematical representation is same as eigenvalue of the eigenvalue analysis, and buckling shape is Eigen vector.



Therefore, the buckling analysis problem can be narrowed to an eigenvalue analysis problem. From the eigenvalue analysis, eigenvalues and mode shapes are obtained, which correspond to critical load factors and buckling shapes respectively. A critical load is obtained by multiplying the initial load by the critical load factor. The significance of the critical load and buckling mode shape is that the structure buckles in the shape of the buckling mode when the critical load exerts on the structure. For instance, if the critical load factor of 5 is obtained from the buckling analysis of a structure subjected to an initial load in the magnitude of 10, this structure would buckle under the load in the magnitude of 50. Note that the buckling analysis has a practical limit since buckling by and large occurs in the state of geometric or material nonlinearity with large displacements.

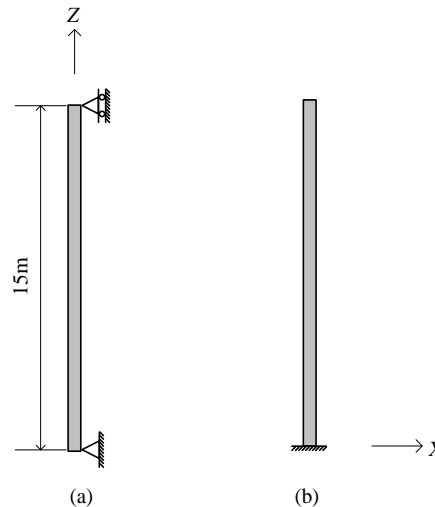


2. Tutorial

2.1 Model Overview

Find the buckling mode and critical load of the following structures using buckling analysis.

► Fig 4.4
Analytical model




- **Material**
Modulus of elasticity: 100 N / mm^2
- **Section**
Sectional area (Area) : $2.5 \times 10^5 \text{ mm}^2 (1000 \times 250)$
Moment of inertia (I_{yy}): $1.30208 \times 10^9 \text{ mm}^4$
- **Load**
Nodal node load 10 kN at roller support

2.2 Work Environment

Open a new file and save the file name.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. File name: '**Buckling-a**', Click **[SAVE]**

Set the unit to mm, N.

Main Menu > **Tools** > **Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click **[OK]**

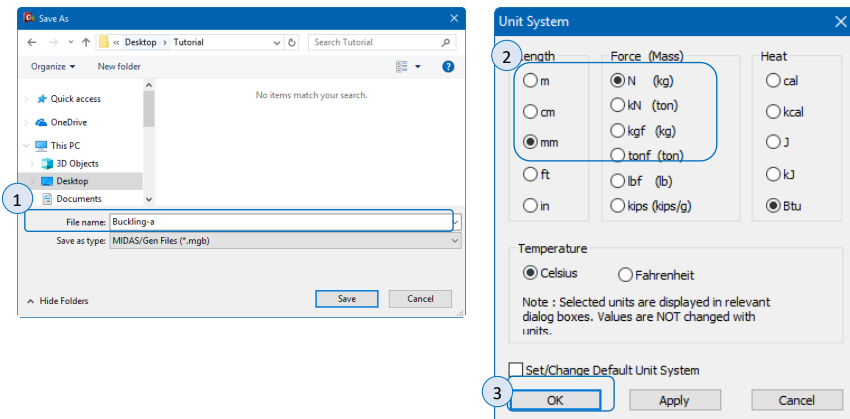
► Fig 4.5

(a) Save the file

(b) Unit system setting



The unit system setting can be easily set at the status bar at the bottom of the screen.



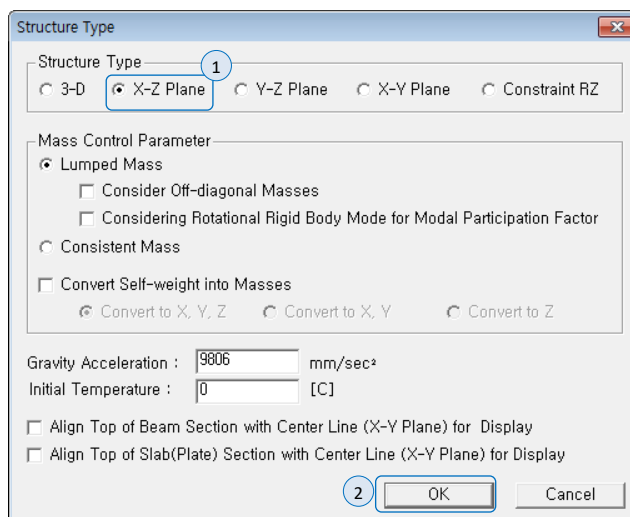
midas Gen is 3-D software, since beam exists in a 2-D plane, X-Z plane in Global Coordinate is set as the work plane, which restrains unnecessary degrees of freedom, Dy, Rx, Rz.

Main Menu > **Structure** > **Structure Type...**

1. Select Structure Type > **X-Z Plane**

2. Click **[OK]**

► Fig 4.6
Set work plane



2.3 Material & Section Properties

Define material and section for the structure.

Main Menu > **Properties** > **Material Properties**

1. Click **[Add...]**, Name : 'Mat'
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : '100', Click **[OK]**
4. Click Section Tab and **[Add...]** and **Value** Tab
5. Select Section Shape Lists > **Solid Rectangle**, Name : 'Sect'
6. Size > H : '250', B : '1000', Section Properties > Click **[Calc. Section Properties]**
7. **Consider Shear Deformation** (off)
8. Click **[OK]** and **[Close]**

► Fig 4.7

(a) Material definition
(b) Section definition



After entering the size of the section, [Calc. Section Properties] button to calculate the section performance automatically. If you double click the item, you can see the number of decimal places in detail.

Material Data

General
Material ID: 1 Name: Mat

Elasticity Data
Type of Design: User Defined

User Defined
Standard: None DB: Code: DB:

Type of Material
☒ Isotropic ☐ Orthotropic

User Defined
Modulus of Elasticity: 100 N/mm²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+000 1/[C]
Weight Density: 0 N/mm³
☐ Use Mass Density: 0 N/mm³/g

Concrete
Modulus of Elasticity: 0.0000e+000 N/mm²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+000 1/[C]
Weight Density: 0 N/mm³
☐ Use Mass Density: 0 N/mm³/g

Plasticity Data

Section Data

DB/Use: Value SRC Combined PSC Tapered Composite

Section ID: 1 Solid Rectangle

Name: Sect ☒ Built-Up Section

Size
H: 250.0000 mm
B: 1000.0000 mm

Section Properties
Calc. Section Properties

Area	2.50000e+005	mm ²
Asy	2.08333e+005	mm ²
Asz	2.08333e+005	mm ²
Ixx	4.38829e+009	mm ⁴
Iyy	1.30208e+009	mm ⁴
Izz	2.08333e+010	mm ⁴
Cyp	500.0000	mm
Cym	500.0000	mm
Czp	125.0000	mm
Czm	125.0000	mm
Qyb	7812.5000	mm ³
Qzb	125000.0000	mm ³
Per:O	2.50000e+003	mm

☐ Consider Shear Deformation.

2.4 Generate Nodes & Elements

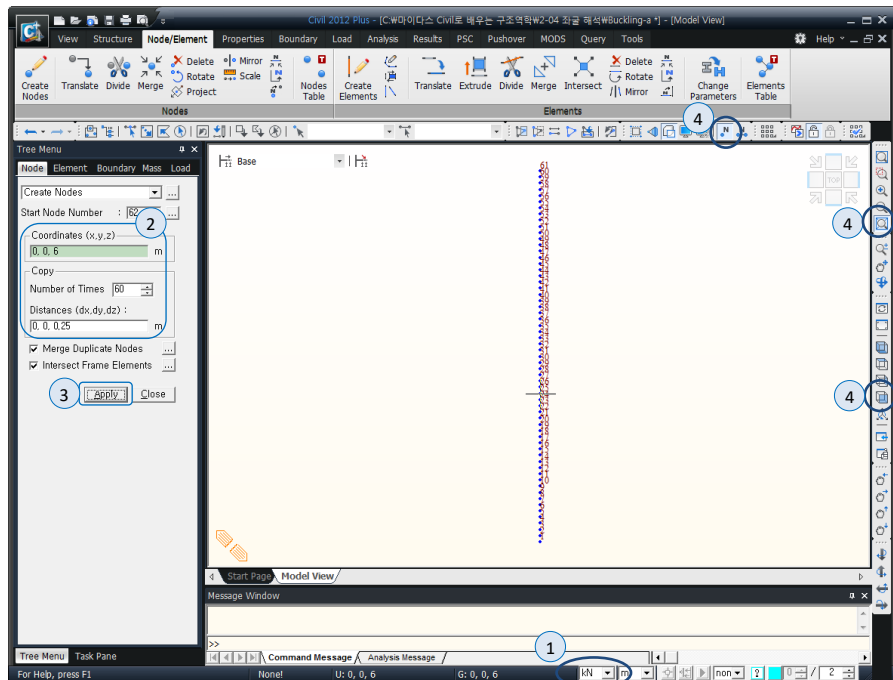
Create nodes where elements will be created.

Main Menu > **Node/Element** > **Nodes** > **Create Nodes...**

1. Modify unit to kN, m
2. Coordinates (x, y, z) : '0, 0, 0'
Copy Number of Times: '60'
Distances (dx, dy, dz) : '0,0,0.25'
3. Click **[Apply]**
4. **Display Node Numbers, Auto Fitting, Front View (on)**

► Fig 4.8
Create nodes

Tip
When Auto Fitting is toggled on, the model fits into the full screen, which automatically controls Zoom Size in real time



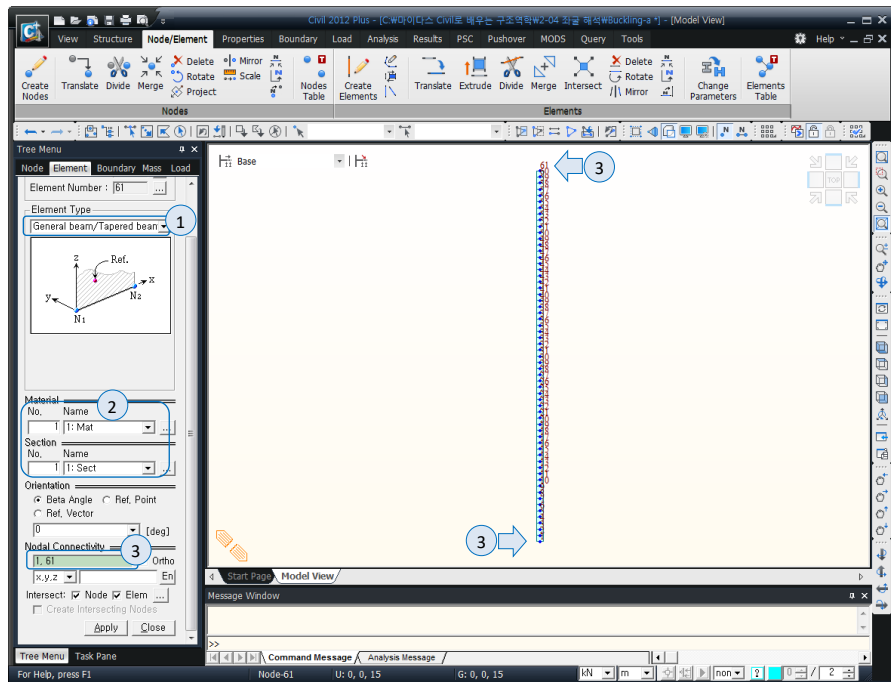
Connect the nodes to create the element.

Main Menu > **Node/Element** > **Elements** > **Create Elements...**

1. Select Element Type > **General beam/Tapered beam**
2. Select Material > **1:Mat** and Section > **1:Sect**
3. Click Nodal Connectivity green box, and Click node number 1 and 61 in Model view

► Figure 4.9
Create element

Tip
By modeling one element
by dividing it into several
elements, you can see the
detailed deformation
results of the element.



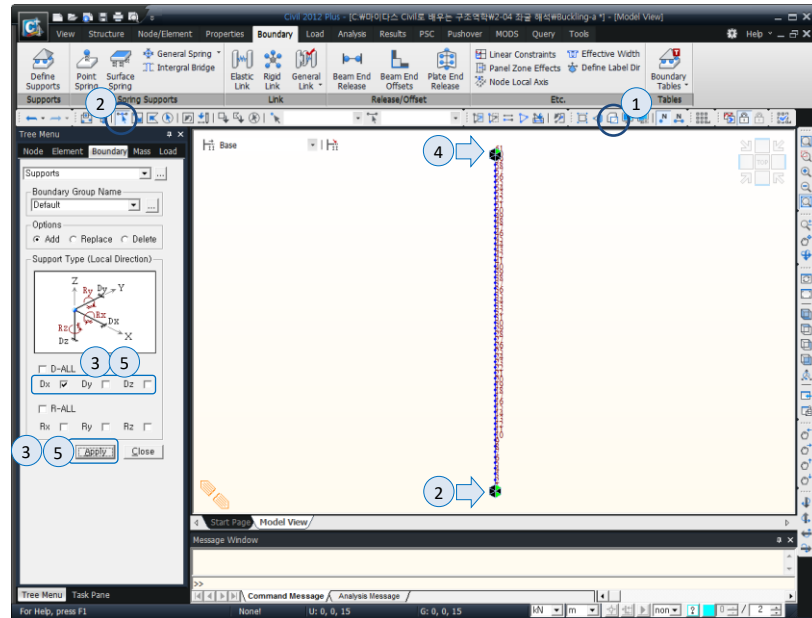
2.5 Define Boundary Conditions

Pin support is assigned to the bottom of the column, and roller support is assigned to the top of column.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. **Hidden** (off)
2. Click **Select Single** (on), Select node number 1
3. Support Type > **Dx, Dz** (on), Click **[Apply]**
4. Click **Select Single** (on), Select node number 61
5. Support Type > **Dx** (on), **Dz** (off), Click **[Apply]**

► Fig 4.10
Define boundary
conditions





2.6 Define Loads

Define load case (load type) first to which the loading will belong.

Main Menu > **Load** > **Static Loads** > **Static Load Cases**

1. Name : '**Case1**'

Select Type > **User Defined Load (USER)**

2. Click **[Add]** and **[Close]**

► Fig 4.11
Define load case

Static Load Cases

1 Name : Case1

Case : All Load Case

Type : User Defined Load (USER)

Description :

2 Add

Modify

Delete

No	Name	Type	Description
*			

2 Close

Input nodal load (10 kN) in the (-) Z direction at the top of the column.

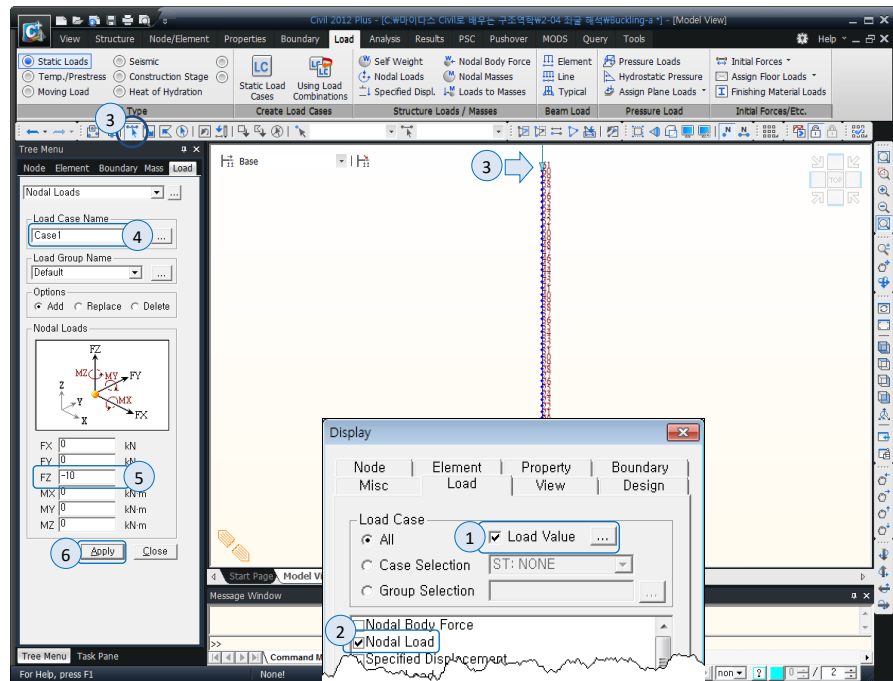
Main Menu > **View > Display...**

1. **Load** Tab > Load Case > **Load Value** (on)
2. Nodal Load (on), Click **[OK]**

Main Menu > **Load /Nodal Loads...**

3. Click **Select Single** (on), Select node number **61**
4. Select Load Case Name > **Case1**
5. Nodal Loads > FZ > **'-10'**
6. Click **[Apply]**

► Figure 4.12
Input nodal load



2.7 Perform Analysis

Define the basic conditions of buckling analysis.

Main Menu > **Analysis** > **Analysis Control** > **Buckling**

1. Number of Modes : '5'

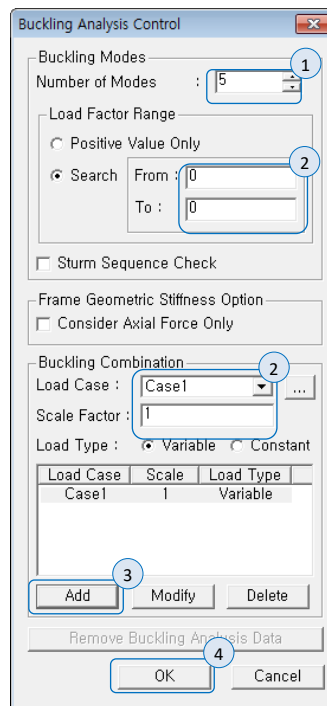
2. Load Factor Range > Search > From : '0', To : '0'

Buckling Combinations > Load Case > Select **Case1** , Scale Factor : '1'

3. Click **[Add]**

4. Click **[OK]**

► Fig 4.13
Buckling Analysis
Control



Analyze the model.

Main Menu > **Analysis** > **Perform Analysis**

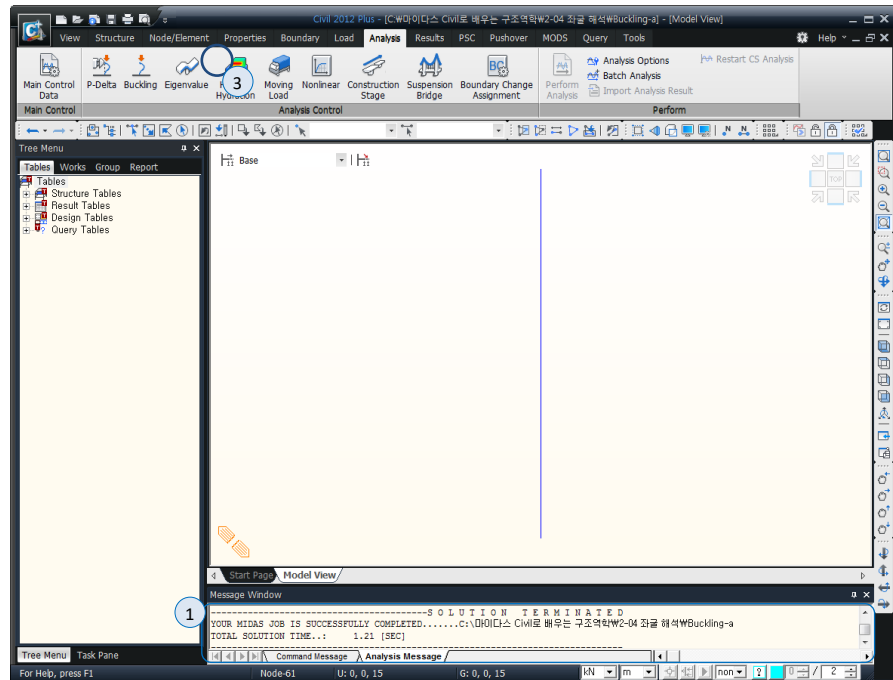
1. Check for successful completion in Message Window

Main Menu > **View** > **Display...**

2. Load Tab > Nodal Load (off), Click [OK]

3. **Display Node Numbers** (off)

► Fig 4.14
Message for a
successful run



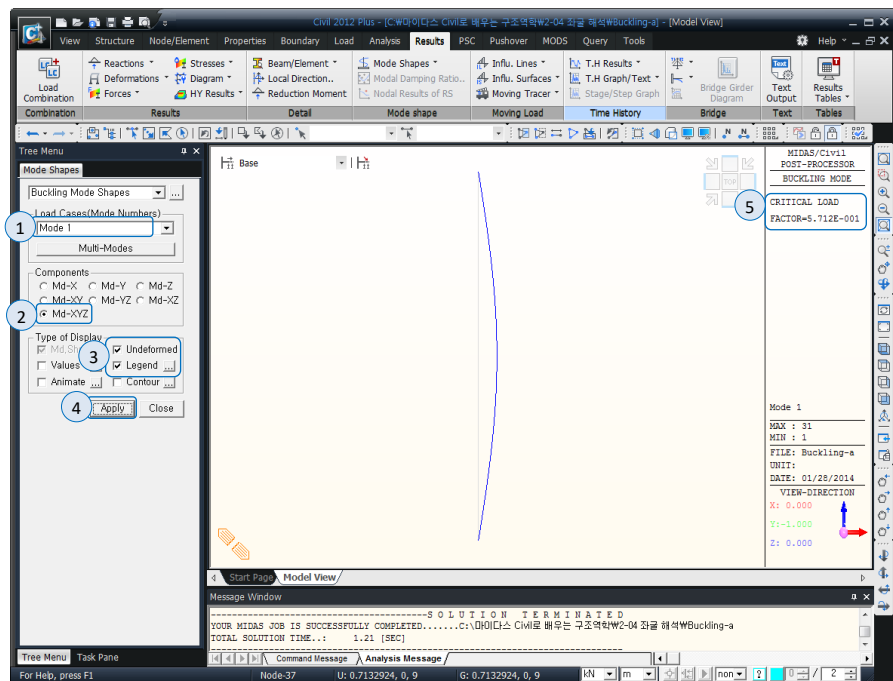
2.8 Check Analysis Result

Check deformed shape of the buckling mode.

Main Menu > **Results** > **Mode Shapes** > **Buckling Mode Shapes...**

1. Select Load Cases(Mode Numbers) > **Mode 1**
2. Select Components > **Md-XYZ**
3. Type of Display > **Undeformed, Legend (on)**
4. Click **[Apply]**
5. Confirm **CRITICAL LOAD FACTOR = 5.712E-001**

► Fig 4.15
Buckling mode shape





Check eigenvalue results to calculate critical load.

Main Menu > **Results** > **Results Tables** > **Buckling Mode Shape...**

1. Records Activation Dialog > Buckling Mode > **Mode 1,2,3,4,5** (on)

2. Click **[OK]**

3. Confirm the value of Eigenvalue mode in a table

► Fig 4.16
Eigenvalue analysis
result

	Node	Mode	UX	UY	UZ	RX	RY	RZ
			BUCKLING ANALYSIS					
		Mode	Eigenvalue	Tolerance				
		1	0.571158	3.5072e-025				
		2	2.284631	9.1728e-017				
		3	5.140423	2.6300e-014				
		4	9.138547	5.6422e-013				
		5	14.279034	6.4655e-013				
			BUCKLINGVECTOR					
►	1	1	0.000000	0.000000	0.000000	0.000000	0.004832	0.000000
	2	1	0.001207	0.000000	0.000000	0.000000	0.004825	0.000000
	3	1	0.002411	0.000000				0.000000
	4	1	0.003609	0.000000				0.000000
	5	1	0.004796	0.000000				0.000000
	6	1	0.005971	0.000000				0.000000
	7	1	0.007129	0.000000				0.000000

Records Activation Dialog

Node or Element
All None Inverse Prev

Node |to61

Select Type
Element Type Add

TRUSS
BEAM
PLANE STRESS
PLATE
PLANE STRAIN
AXISYMMETRIC
SOLID

Delete
Replace
Intersect

1 Buckling Mode
✓Mode 1
✓Mode 2
✓Mode 3
✓Mode 4
✓Mode 5

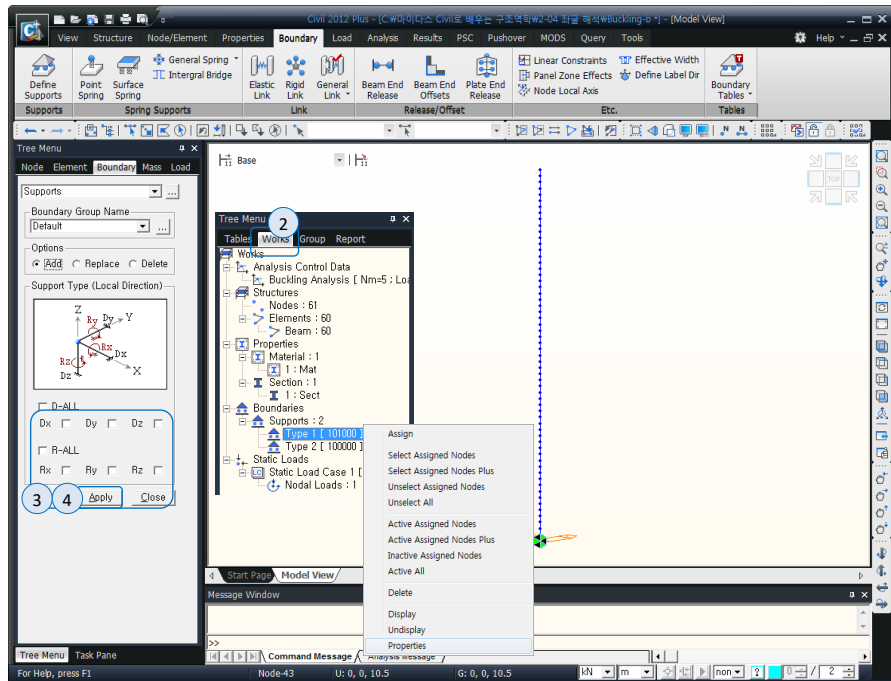
2 OK Cancel

Compare to the results according to different boundary conditions.

Main Menu > **Files** > **Save As...**

1. Enter a name : **'Buckling-b'**, Click **[SAVE]**
2. Tree Menu > Works Tab
3. Works>Boundaries>Supports : 2 > Right clicking at Type 1, Select **Properties**
Support Type > **Dx, Dz, Ry** (on), Click **[Apply]** and **[Close]**
4. Works>Boundaries>Supports : 2 > Right clicking at Type 2, Select **Properties**
Support Type > **Dx, Dy, Dz, Rx, Ry, Rz** (off), Click **[Apply]** and **[Close]**

► Fig 4.17
Modify boundary
conditions

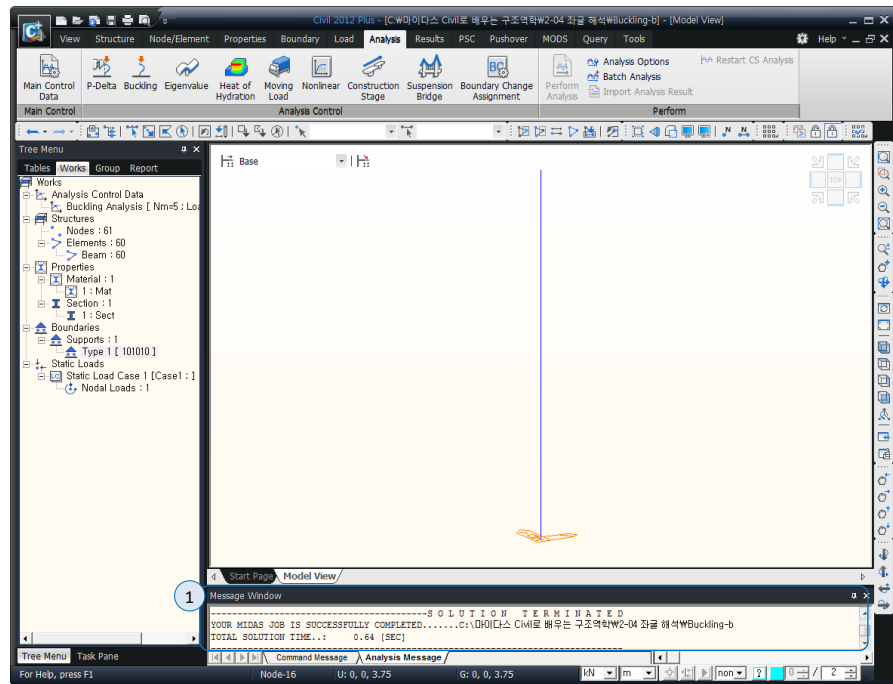


Analyze the model modified.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window

► Fig 4.18
Message for a
successful run

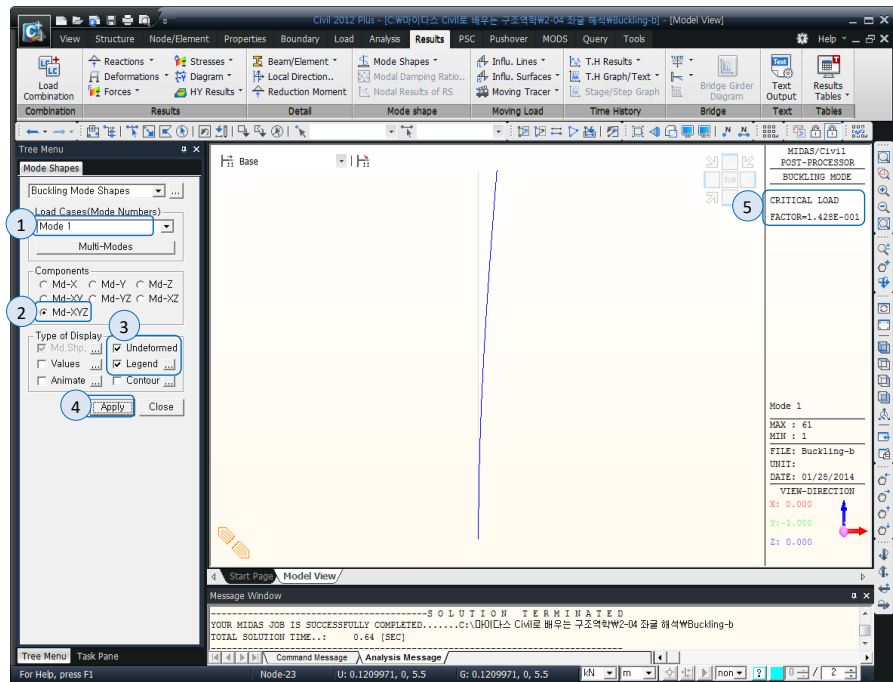


Check deformed shape of the buckling mode

Main Menu > **Results** > **Buckling Mode Shapes...**

1. Select Load Cases(Mode Numbers) > **Mode 1**
2. Select Components > **Md-XYZ**
3. Type of Display > **Undeformed, Legend (on)**
4. Click **[Apply]**
5. Confirm **CRITICAL LOAD FACTOR = 1.428E-001**

► Fig 4.19
Buckling mode shape



To calculate the critical load, check the eigenvalues in the buckling analysis result table.

Main Menu > **Results** > **Results Tables** > **Buckling Mode Shape...**

1. Records Activation Dialog > Buckling Mode > **Mode1,2,3,4,5** (on)
2. Click **[OK]**
3. Confirm the value of Eigenvalue mode in a table

► Fig4.20
Eigenvalue analysis
result

The screenshot displays the ANSYS Workbench interface for a Buckling Analysis. The main window shows the results of a linear buckling analysis. The 'Buckling Analysis' table lists the first five buckling modes, with the first mode being the critical one. The 'Buckling Vector' table shows the displacement vectors for the first mode. A 'Records Activation Dialog' box is open, showing the 'Buckling Mode' list with modes 1 through 5 selected. The 'Element Type' dropdown is set to 'TRUSS BEAM', and the 'Add' button is highlighted. The 'OK' button is also visible in the dialog box.

Node	Mode	UX	UY	UZ	RX	RY	RZ
BUCKLING ANALYSIS							
	Mode	Eigenvalue	Tolerance				
	1	0.142789	0.0000e+000				
	2	1.285105	5.8697e-018				
	3	3.569737	4.6276e-016				
	4	6.996692	7.0950e-014				
	5	11.565992	7.5339e-013				

BUCKLING VECTOR							
1	1	0.000000	0.000000				
2	1	0.000032	0.000000				
3	1	0.000126	0.000000				
4	1	0.000284	0.000000				
5	1	0.000506	0.000000				
6	1	0.000789	0.000000				
7	1	0.001136	0.000000				

Records Activation Dialog

Node or Element: [All] [None] [Inverse] [Prev] [Next] [To6]

Select Type: [Element Type] [Add] [Delete] [Replace] [Intersect]

Buckling Mode:

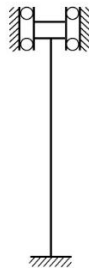
- ☒ Mode 1
- ☒ Mode 2
- ☒ Mode 3
- ☒ Mode 4
- ☒ Mode 5

OK Cancel



3. Exercise

Determine the buckling modes and the corresponding critical loads of a column subjected to a vertical load with various boundary conditions



(a)



(b)

- **Material**
Modulus of elasticity: 100 N / mm^2
- **Section**
Beam element: Solid rectangular $1000\text{mm} \times 250\text{mm}$
Moment of inertia (I_{yy}): $1.30208 \times 10^9 \text{ mm}^4$
- **Boundary condition**
 - (a) Lateral displacement and rotational displacement of the top end are constrained and the bottom is a fixed support.
 - (b) The top end is a roller and the bottom is a fixed support
- **Load**
Nodal node load 10 kN at roller support



5. Eigenvalue Analysis

Contents

1 Introduction

1.1 Concept of Eigenvalue Analysis	5-3
------------------------------------	-----

2 Tutorial

2.1 Model Overview	5-5
2.2 Work Environment	5-6
2.3 Material & Section Properties	5-8
2.4 Generate Node & Element	5-9
2.5 Define Boundary Conditions	5-11
2.6 Define Loads	5-12
2.7 Perform Analysis	5-13
2.8 Check Analysis Result	5-14

3 Exercise	5-22
------------	------



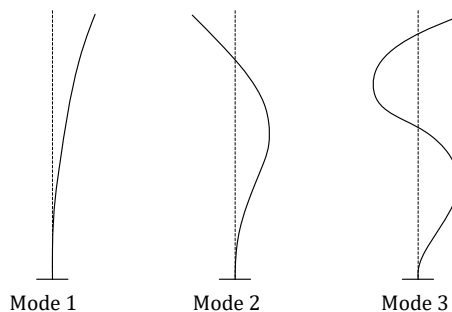
1. General

1.1 Concept of Eigenvalue Analysis

Eigenvalues is also referred as free vibration analysis as it is Interpretation of the structure's unique dynamic characteristics analysis. What does the dynamic attributes mean? Frequency and mode shapes are calculated using Ritz Vector analysis or Eigen value analysis in midas Gen. For Eigen Value analysis, characteristic equation is formed, solution of which is Natural modes (Mode Shapes), Natural Periods (Frequencies), Modal Participation Factors and Effective modal mass. The mode shapes and frequencies generated by Eigen Value Analysis are important for Response Spectrum or Time History analysis,

During Free vibration of structure, the energy reduces sequentially from 1st Mode (Fundamental Mode of Vibration) to 2nd and subsequent modes of vibration. Figure 5.1 shows the different modes shapes for a Cantilever arrangement.

► Fig 5.1
Mode shape of
Cantilever



Majority of Standard Design code require more than 90% of the modal mass excitation to include major modes affecting the analysis results.

Force equilibrium conditional can be applied and because there is no external load being applied, the RHS of the equation is 0.

► Eq. 5.1

$$EIw'''' + m\ddot{w} = 0$$



Since there is no external load, mathematically it is same as buckling analysis. Separating deflection term $w(x, t)$ in terms of distance and time $w(x, t) = \phi(x)Y(t)$. The differential equation becomes

► Eq. 5.2
$$\frac{\phi''''(x)}{\phi(x)} + \frac{m \ddot{Y}(t)}{EI Y(t)} = 0$$

Here, $\phi(x)$ is the shape of Eigen Mode and $Y(t)$ as function of time, enables to estimate natural period of vibration. To satisfy Eq.5.2, each term must be constant and can be assumed as follows,

► Eq. 5.3
$$\frac{\phi''''(x)}{\phi(x)} \equiv a^4, \quad -\frac{m \ddot{Y}(t)}{EI Y(t)} \equiv a^4$$

So the equation becomes,

► Eq. 5.4
$$\phi''''(x) - a^4 \phi(x) = 0$$

► Eq. 5.5
$$\ddot{Y}(t) + \omega^2 Y(t) = 0 \quad \text{where, } \omega \equiv a^2 \sqrt{\frac{EI}{m}}, \quad \text{and } a^2 \equiv \omega \sqrt{\frac{m}{EI}}$$

The general solution to Eq. (5.4) above can be expressed as follows, where the coefficients are obtained according to the boundary conditions.

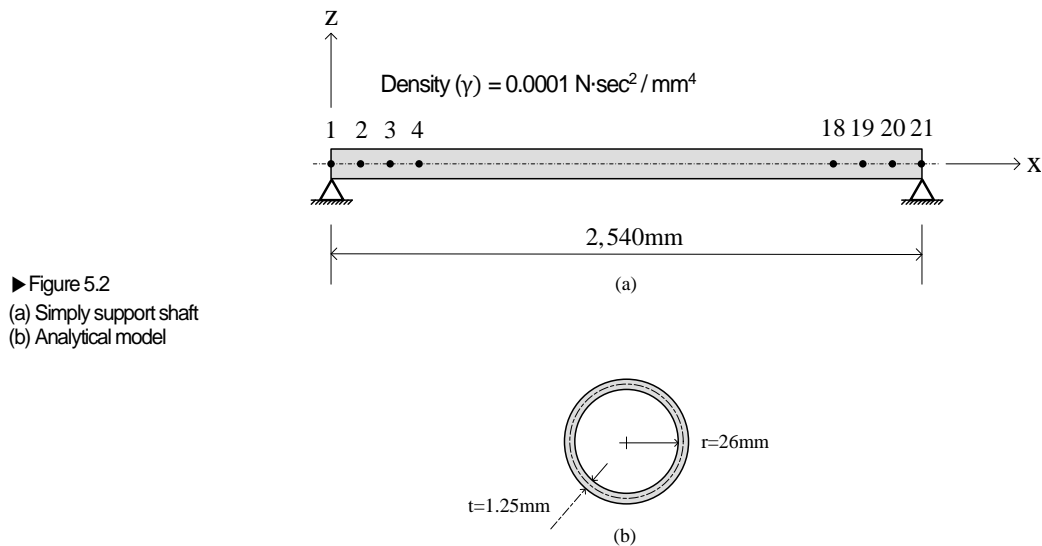
► Eq. 5.6
$$\phi(x) = A_1 \cos ax + A_2 \sin ax + A_3 \cosh ax + A_4 \sinh ax +$$

If a is obtained from Eq. (5.5), the shape of the Eigen mode can be known, and the eigenvalue can be estimated.

2. Tutorial

2.1 Model Overview

Perform the eigenvalue analysis on a simply supported shaft as shown in the following figure.



- **Material**
Modulus of elasticity: $2.0 \times 10^5 \text{ N / mm}^2$
- **Section**
Cross-sectional area (Area) : 200 mm^2
Moment of inertia (Iyy): $65,000 \text{ mm}^4$
Radius: 26 mm
Thickness: 1.25 mm
Gravitational acceleration (g): $9,806 \text{ mm/sec}^2$

2.2 Work Environment

Open a new file and save.

Main Menu > **File** >  **New Project...**

Main Menu > **File** >  **Save**

1. File name : '**Eigenvalue 1**', Click **[SAVE]**

Set the unit system to use.

Main Menu > **Tools** > **Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click **[OK]**

► Fig 5.3

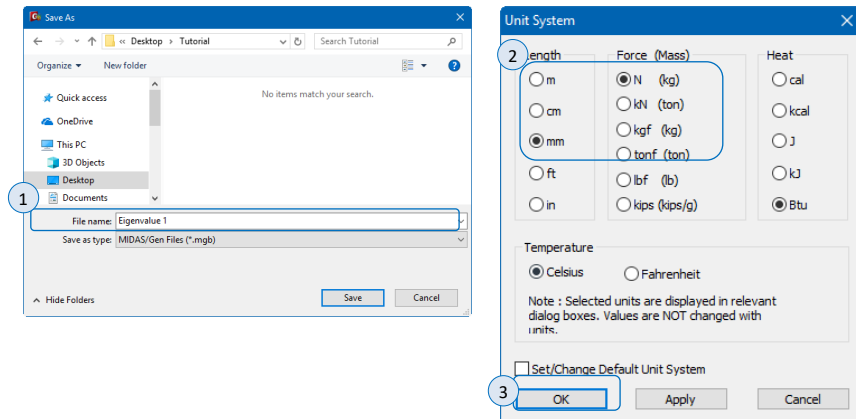
(a) Save the file

(b) Unit system setting



Tip

The unit system setting can be easily set at the status bar at the bottom of the screen.

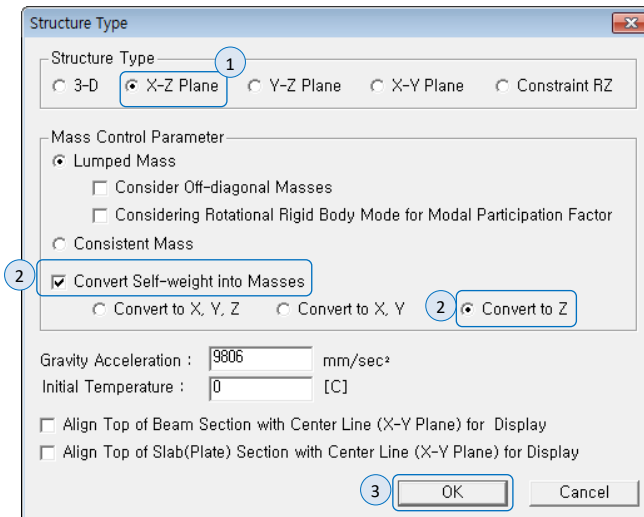


The example models exist in the 2-D X-Z plane, and the self-weight is automatically replaced with the nodal mass.

Main Menu > **Structure** > **Structure Type...**

1. Select Structure Type > **X-Z Plane**
2. Select Convert Self-weight into Masses (on) > **Convert to Z**
3. Click **[OK]**

► Fig 5.4
Set work plane



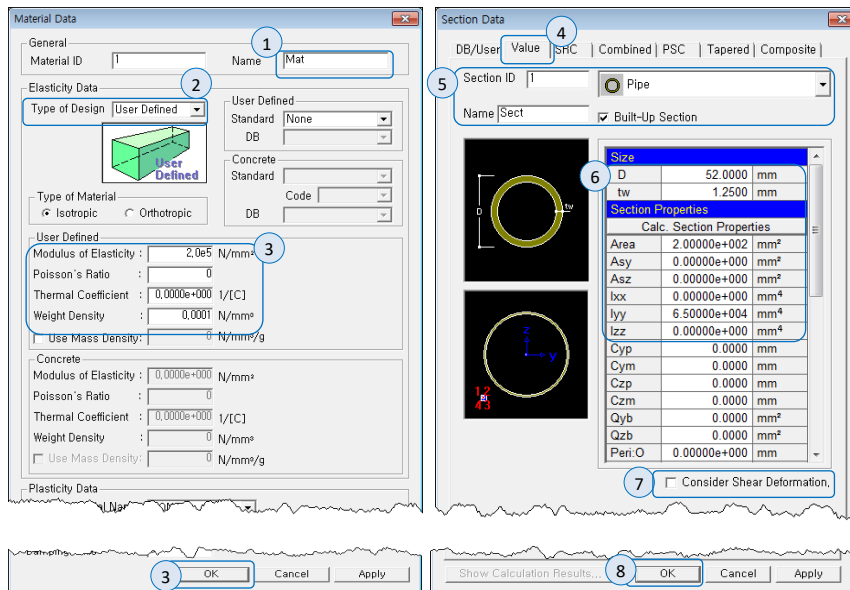
2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu > **Properties** > **Material Properties**

1. Click **[Add...]**, Name : '**Mat**'
2. Select Elasticity Data > Type of Design > **User Defined**,
3. User Defined > Modulus of Elasticity : '**2.0e5**', Weight Density : '**0.0001**', Click **[OK]**
4. Click **Section** Tab and **[Add...]** and **Value** Tab
5. Section Shape Lists > Select **Pipe**, Name : '**Sect**'
6. Size > D : '**52**', tw : '**1.25**', Section Properties > Area : '**2.0e2**', I_{yy} : '**6.5e4**'
7. **Consider Shear Deformation** (off)
8. Click **[OK]** and **[Close]**

► Fig 5.5
Define material & section
properties



2.4 Generate Nodes & Elements

Create a node in order to generate beam elements.

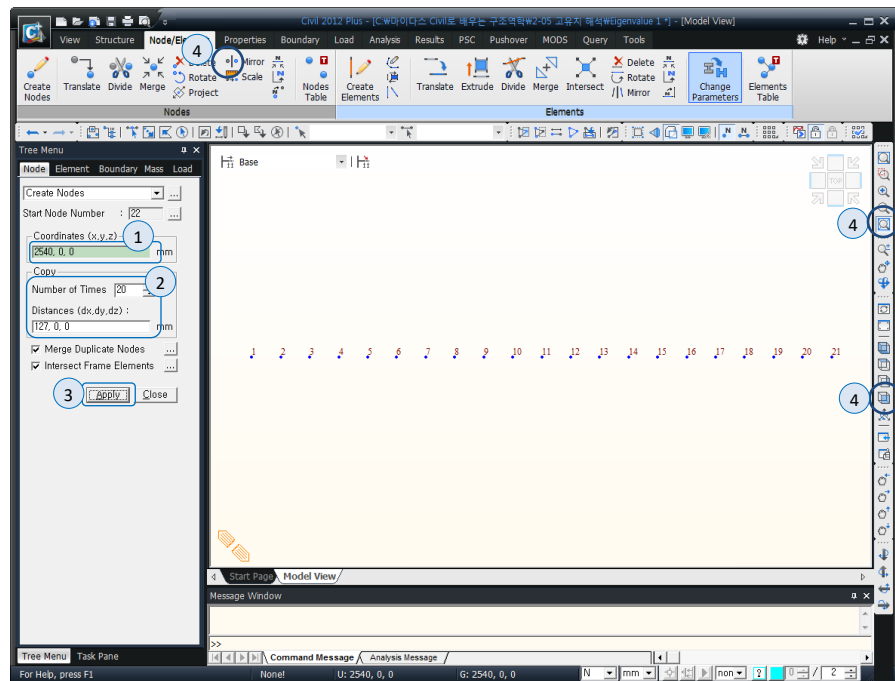
Main Menu > **Node/Element** > **Nodes** > **Create Nodes...**

1. Coordinates (x, y, z) : '**0, 0, 0**'
2. Number of Times: '**20**'
Distances (dx, dy, dz) : '**127,0,0**'
3. Click [**Apply**]
4. **Display Node Numbers, Auto Fitting, Front View (on)**

► Fig 5.6
Create nodes



When Auto Fitting is toggled on, the model fits into the full screen, which automatically controls Zoom Size in real time.

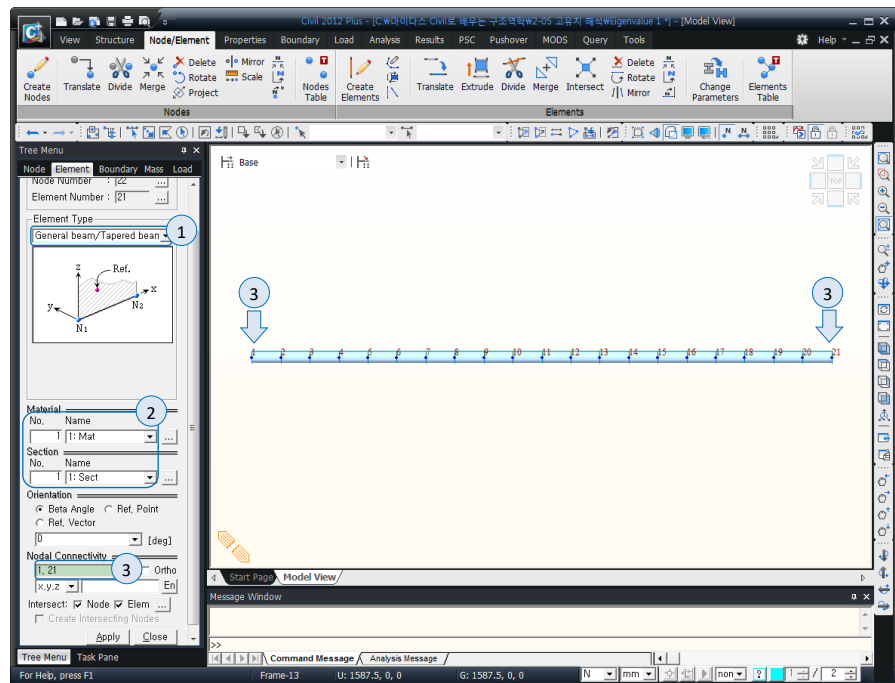


Connect the nodes to create the elements.

Main Menu > **Node/Element** > **Elements** > **Create Elements...**

1. Select Element Type > **General beam/Tapered beam**
2. Select Material > **1:Mat** and Section > **1:Sect**
3. Click Nodal Connectivity green box, and Click node number 1 and 21 in Model view

► Fig5.7
Create elements



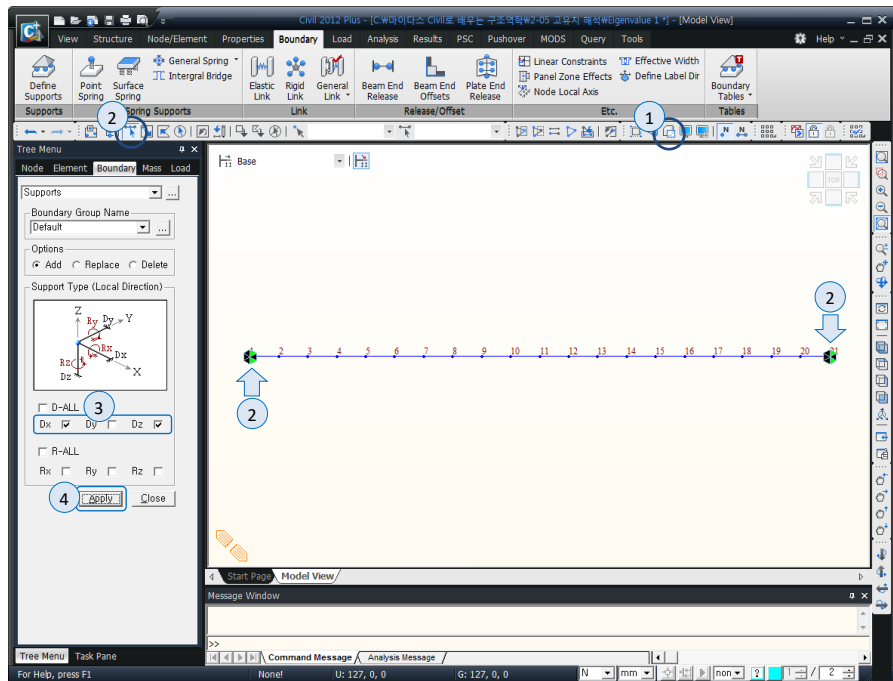
2.5 Define Boundary Conditions

The both end support of model is assigned a pin support with Dx & Dz degree of freedom restrained.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. **Hidden** (off)
2. **Select Single** (on) Select both end points 1 and 21 in mode
3. Support Type > **Dx, Dz**(on)
4. Click **[Apply]**

► Fig 5.8
Defined boundary
conditions



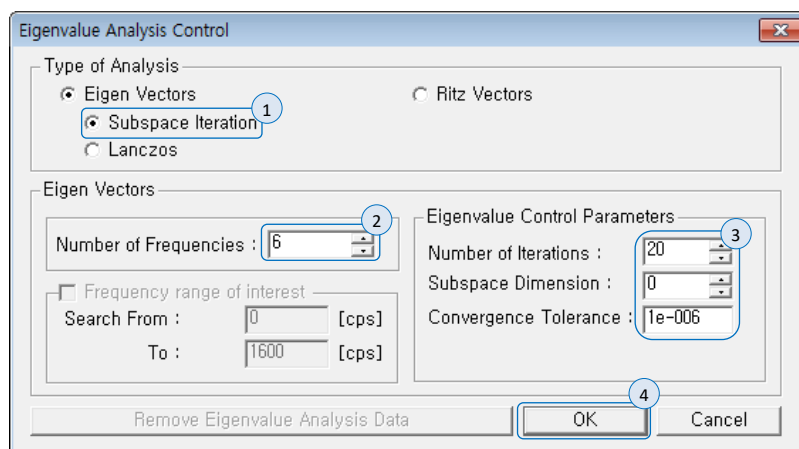
2.6 Define Loads

Define analytical data for eigenvalue analysis.

Main Menu > **Analysis** > **Analysis Control** > **Eigenvalue**

1. Select Type of Analysis > Eigen Vectors > **Subspace Iteration**
2. Eigen Vectors > Number of Frequencies : '6'
3. Eigenvalue Control Parameters > Number of Iteration : '20'
Subspace Dimension : '0', Convergence Tolerance : '1e-006'
4. Click **[OK]**

► Fig 5.9
Input eigenvalue analysis
data



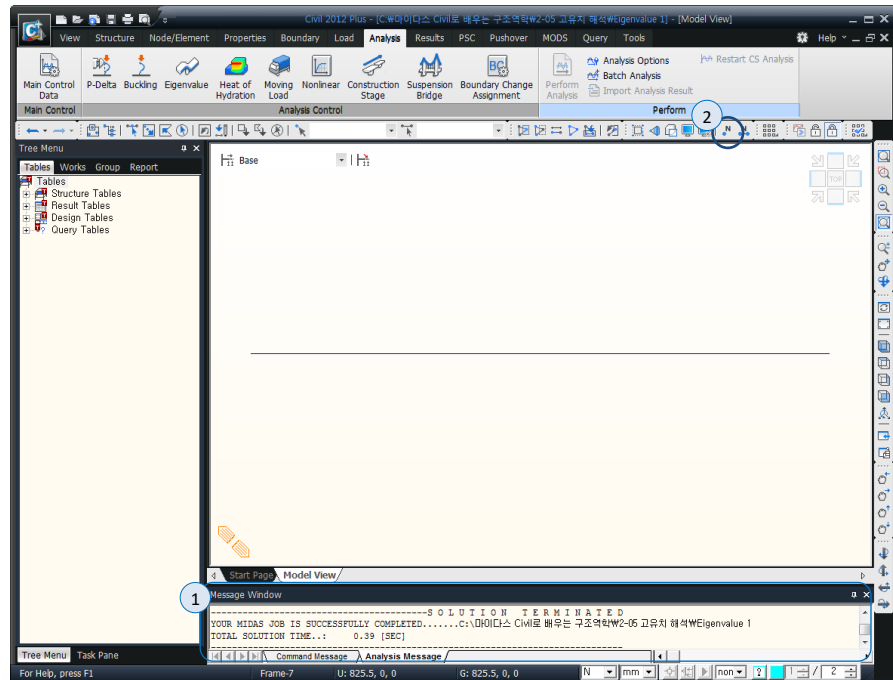
2.7 Perform Analysis

Analyze the model.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window
2. **Display Node Numbers** (off)

► Fig 5.10
Message for a
successful run



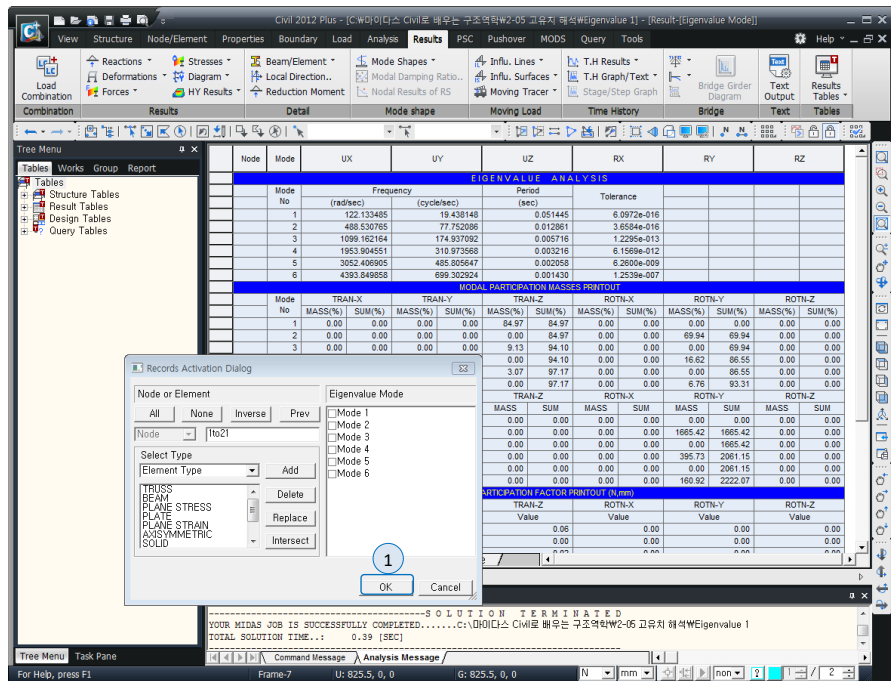
2.8 Check Analysis Result

Check the result in tables of the eigenvalue analysis

Main Menu > Results > Result Tables > Vibration Mode Shape...

1. Records Activation Dialog box, Click [OK]
2. Check eigenvalue results such as cycle by mode in table

► Fig 5.11
Eigenvalue analysis
result table



Check the mode shape and natural frequency of first mode.

Click **Model View Tab**

Main Menu > **Results** > **Mode Shapes** > **Vibration Mode Shapes...**

1. Select Load Cases(Mode Numbers) > **Mode 1**

2. Select Components > **Md-XZ**

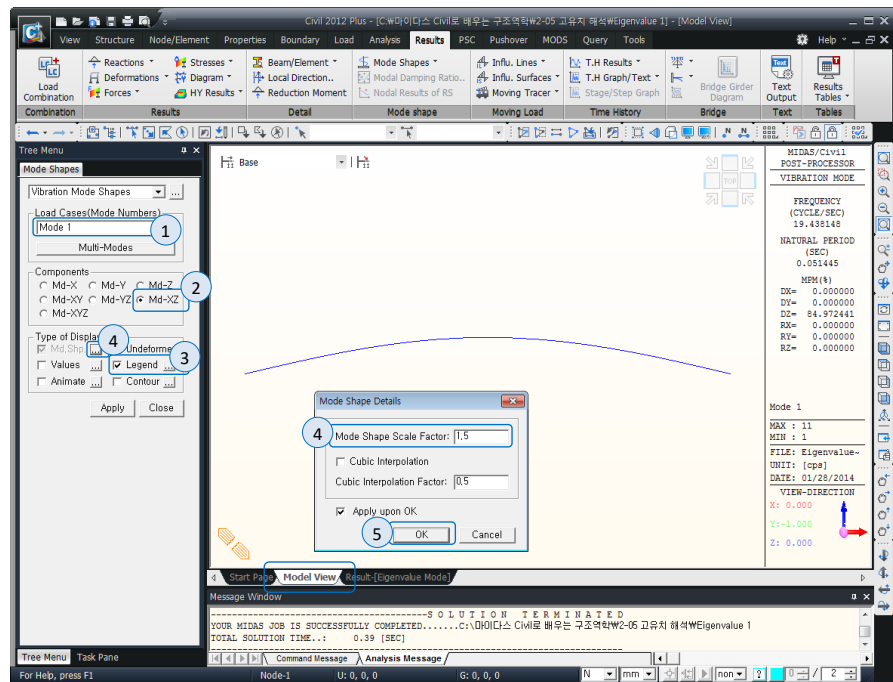
3. Type of Display > **Legend (on)**

4.. Click [...] in Md.Shp

Mode Shape Scale Factor : '1.5'

5. Click **[OK]**

► Fig 5.12
First mode shape

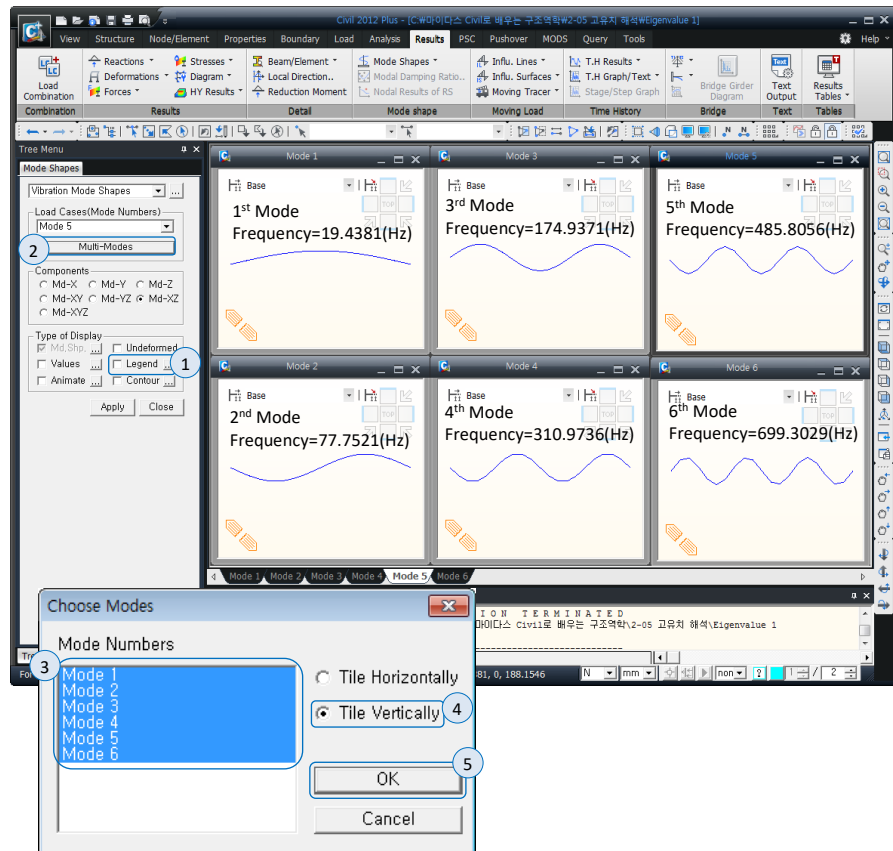


Simultaneously check the mode shapes of the various modes

Main Menu > **Results** > **Mode Shapes** > **Vibration Mode Shapes...**


1. Type of Display > **Legend** (off)
2. Click **[Multi-Modes]**
3. Select **Mode 1~Mode 6** by dragging the mouse cursor from Mode Numbers
4. Select **Tile Vertically**
5. Click **[OK]**

► Figure 5.13
Mode shapes

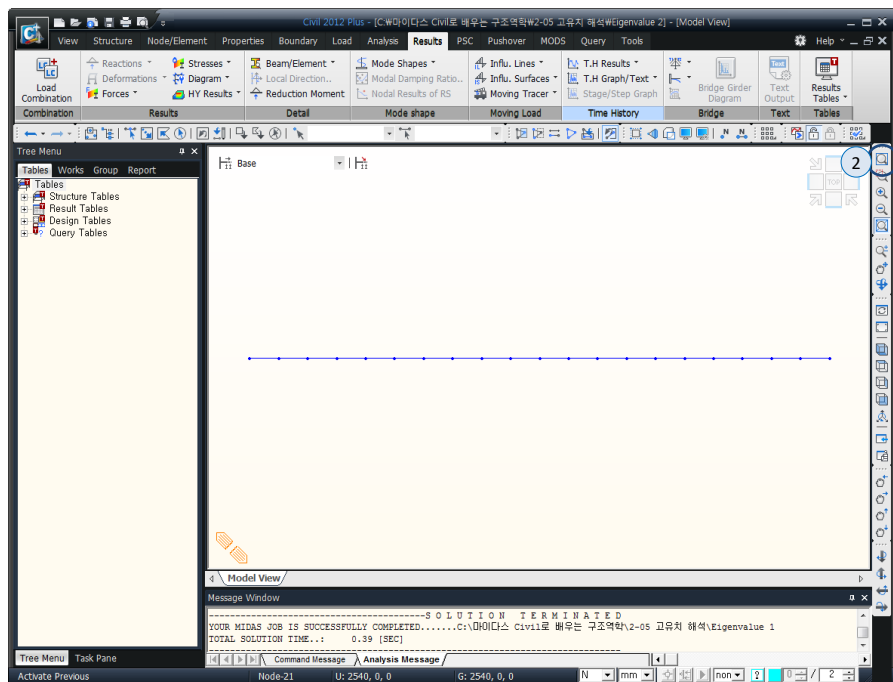
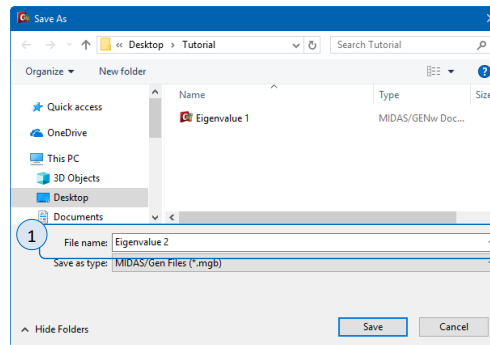


Save new file "Eigenvalue2.mgb".

Main Menu > **File** > **Save As...**

1. Enter a name : **'Eigenvalue2'**, Click **[SAVE]**
2. Close all elements except for Model View, Click  and **Zoom Fit**

► Fig 5.14
Save new file

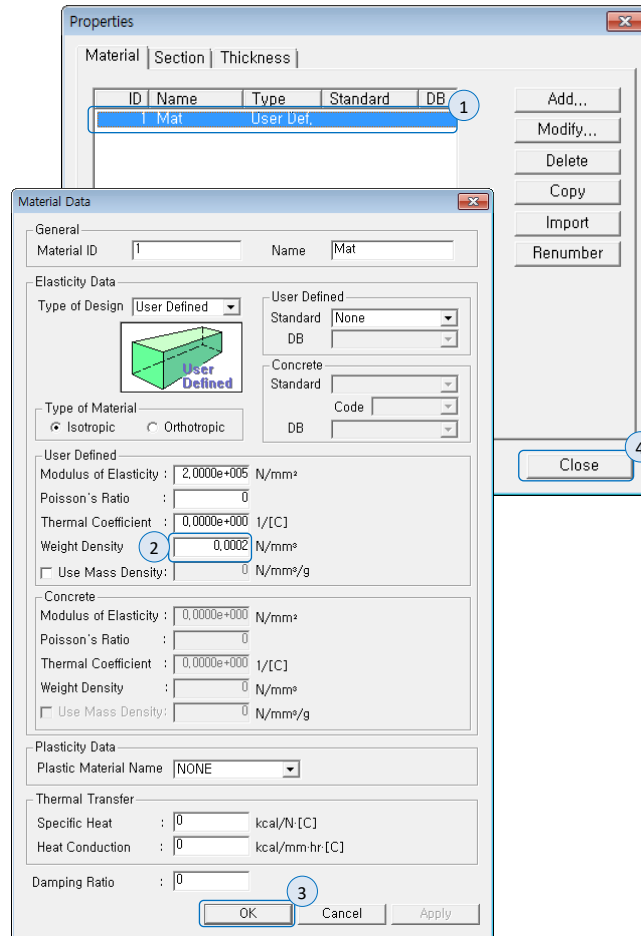


Modified Change the density of the previously input material to 0.0002.

Main Menu > **Properties** > **Material** > **Material Properties**

1. Click **Mat** on lists double
2. Weight Density : '**0.0002**'
3. Click **[OK]**
4. Click **[Close]**

► Fig 5.15
Modify material property

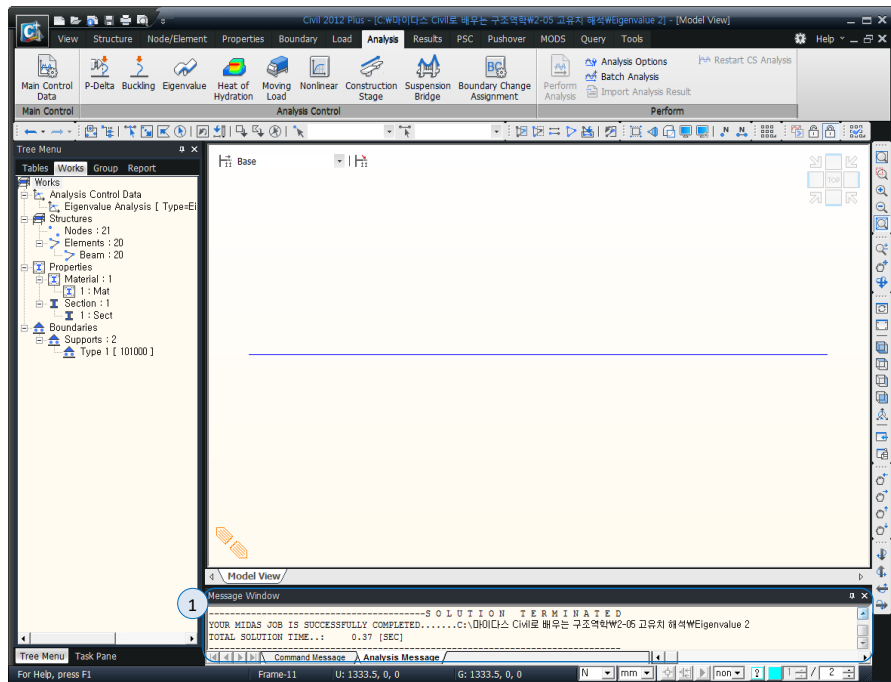


Analyze the model modified.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window

► Fig 5.16
Message for a
successful run

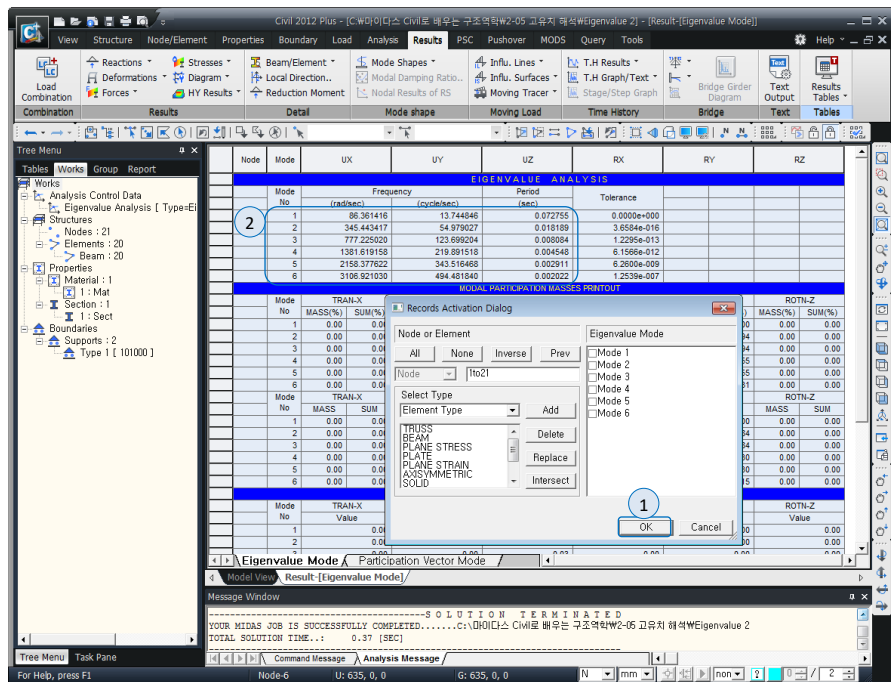


Check the eigenvalue results for each mode using a table.

Main Menu > **Results** > **Result Tables** > **Vibration Mode Shape...**

1. Records Activation Dialog box, Click **[OK]**
2. Identify eigenvalue results such as cycles per mode in the table

► Fig 5.17
Eigenvalue analysis
result table



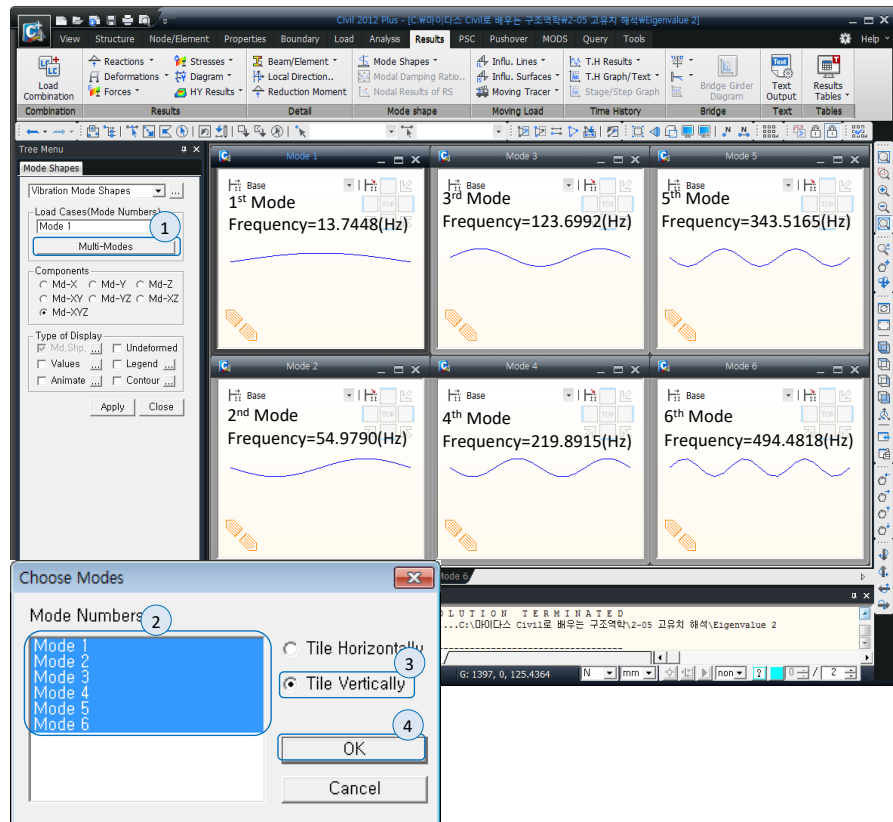
Simultaneously check the mode shapes of the various modes.

Click **Model View Tab**

Main Menu > **Results** > **Mode Shapes** > **Vibration Mode Shapes...**

1. Click **[Multi-Modes]**
2. Select **Mode 1~Mode 6** by dragging the mouse cursor from Mode Numbers
3. Select **Tile Vertically**
4. Click **[OK]**

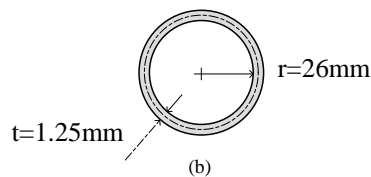
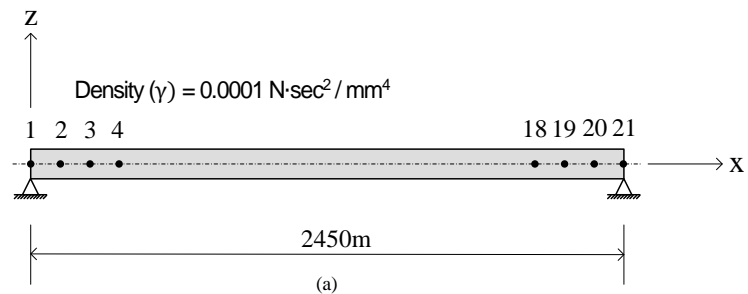
► Fig 5.18
1st to 6th mode shapes





3. Exercise

Compare the eigenvalue analysis results of Model 1 and Model 2 by changing only the modulus of elasticity in the same condition as shown in the following figure (Model 1 is the same as the main model).



➤ **Material**

Modulus of elasticity:

Model 1: $2.0 \times 10^5 \text{ N} / \text{mm}^2$

Model 2: $4.0 \times 10^5 \text{ N} / \text{mm}^2$

➤ **Section**

Cross-sectional area (Area) : 200 mm^2

Moment of inertia (I_{yy}) : $65,000 \text{ mm}^4$

Radius: 26 mm

Thickness: 1.25 mm

g: $9,806 \text{ mm/sec}^2$



6. Time History Analysis

Contents

1 Introduction

1.1 Concept of Time History Analysis 6-3

2 Tutorial

2.1 Model Overview	6-8
2.2 Work Environment	6-9
2.3 Material & Section Properties	6-11
2.4 Generate Node & Element	6-12
2.5 Define Boundary Conditions	6-14
2.6 Define Loads	6-15
2.7 Perform Analysis	6-20
2.8 Check Analysis Result	6-21

3 Exercise 6-33

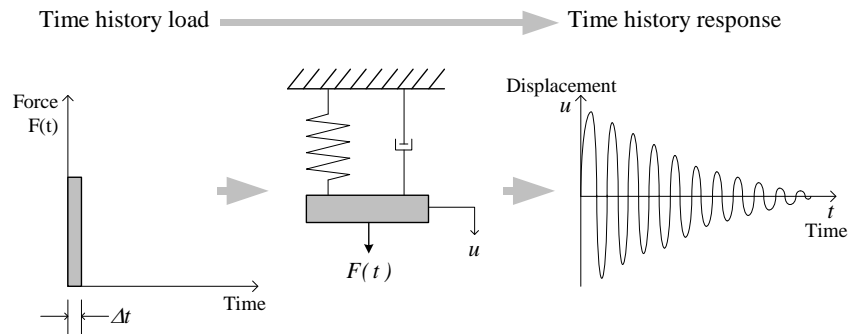
1. Introduction

1.1 Concept of Time History Analysis

The shape of an object that receives external force and moves with time varies from periodic to very irregular. The periodic response of the structure is response generated by any one frequency, whereas the highly irregular behavior can be thought of as a combination of infinite periodic responses. In any case, the dominant frequency response is dependent on the frequency, magnitude and characteristics of the external dynamic load.

The dynamic behavior of an object can be analyzed in terms of time and frequency. The former is called time history analysis and the latter is called frequency response analysis. In case of time history analysis, parameters like time at maximum response and behavioral changes with time are studied. However, in case of frequency response analysis, study of frequencies to which object is sensitive and resonance are done. Because of this distinct nature, in most cases, both the methods are being used.

► Fig 6.1
Time history analysis



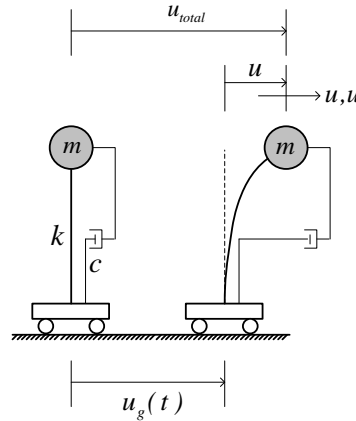
The equation of motion (differential equation) of the 1-degree-of-freedom system for ground motions is as follows.

► Eq 6.1

$$m(\ddot{u}_g + \ddot{u}) + c\dot{u} + ku = 0 \quad \text{or} \quad m\ddot{u} + c\dot{u} + ku = f = -\ddot{u}_g$$

The relative displacement, velocity and acceleration are function of time. These are expressed as u , \dot{u} , \ddot{u} in the above equation respectively.

► Fig 6.2
Model analysis



The differential equation can be solved by direct integration method or by mode superimposition method by using numerical integration to obtain the displacement, velocity and acceleration. The numerical integration method divides ground acceleration into several time intervals $h(= \Delta t)$ and accumulates values for each such interval. There are several methods for numerical integration, but the concept of all these remain the same. Here, the central difference method, the Newmark-beta method and the linear acceleration method have been introduced.

The central difference method uses the following Taylor series expansion formula.

► Eq 6.2
$$u_{i+1} = u_i + h\dot{u}_i + \frac{h^2}{2}\ddot{u}_i + \dots$$

► Eq 6.3
$$u_{i-1} = u_i - h\dot{u}_i + \frac{h^2}{2}\ddot{u}_i + \dots$$

The following results are obtained by the sum and difference of the two equations.

► Eq 6.4
$$\dot{u}_i \approx \frac{1}{2h}(u_{i+1} - u_{i-1})$$

► Eq 6.5
$$\ddot{u}_i \approx \frac{1}{h^2}(u_{i+1} - 2u_i + u_{i-1})$$

In the central difference method, first, the Eq 6.4 and 6.5 in the i-th step are substituted into the equation 6.1, and the i + 1th displacement value is calculated.

► Eq 6.6
$$\left(\frac{m}{h^2} + \frac{c}{2h}\right)u_{i+1} = f_i - \left(k - \frac{2m}{h^2}\right)u_i - \left(\frac{m}{h^2} - \frac{c}{2h}\right)u_{i-1}$$

Then, velocity and acceleration are calculated using Eq 6.4 and Eq 6.5 of the i-th equation. The value of u_{i+1} can be generated by eliminating u_{i-1} from Eq 6.4 and Eq 6.5.

$$u_{i-1} = u_i - h\dot{u}_i + 0.5h^2\ddot{u}_i$$

► Eq 6.7

In the central difference method, if the time interval h (or time increment) is too large, a stable solution cannot be obtained, so h is restricted as follows.

$$h \leq T_{\min} / \pi$$

► Eq 6.8

Where T_{\min} is the minimum of the natural period of the structure.

The Newmark-beta method uses the following approximate relationship.

► Eq 6.9
$$\dot{u}_{i+1} \approx \dot{u}_i + [(1 - \alpha)\ddot{u}_i + \alpha\ddot{u}_{i+1}]h$$

► Eq 6.10
$$u_{i+1} \approx u_i + \dot{u}_i h + \left[\left(\frac{1}{2} - \beta\right)\ddot{u}_i + \beta\ddot{u}_{i+1}\right]h$$

Here, if $\alpha = 0.5, \beta = 0.25$, then the result is;

► Eq 6.11
$$\dot{u}_{i+1} = \dot{u}_i + \frac{h}{2}(\ddot{u}_i + \ddot{u}_{i+1})$$

► Eq 6.12
$$u_{i+1} = u_i + h\dot{u}_i + \frac{h^2}{2}(\ddot{u}_i + \ddot{u}_{i+1})$$



Using the Newmark-Beta method, a stable solution is obtained irrespective of the time interval h (or time increment), and the appropriate time interval is:

► Eq 6.13
$$h = T_{\min}/10$$

The case of $\alpha = 0.5, \beta = 1/6$ in the Newmark-Beta method is called the linear acceleration method. The numerical integrations above are examples for a 1-DOF system, but not for a multi-DOF system.

The mode superposition method can be used if it is possible to know the exact number of eigenvalues in the equation of motion, or if it is possible to reflect only the eigenvalues up to a certain limit. Let us explain the mode superposition method as an example of the following two degrees of freedom system.

► Eq 6.14
$$M\ddot{u} + C\dot{u} + Ku = -M\ddot{u}_g$$

$$\begin{bmatrix} m_{11} & m_{12} \\ m_{21} & m_{22} \end{bmatrix} \begin{Bmatrix} \ddot{u}_1 \\ \ddot{u}_2 \end{Bmatrix} + \begin{bmatrix} c_{11} & c_{12} \\ c_{21} & c_{22} \end{bmatrix} \begin{Bmatrix} \dot{u}_1 \\ \dot{u}_2 \end{Bmatrix} + \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = - \begin{bmatrix} m_{11} & m_{12} \\ m_{21} & m_{22} \end{bmatrix} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \ddot{u}_g$$

Since there are two eigenvalues in the above equation, the solution of the equation of motion can be expressed as.

► Eq 6.15
$$u = \phi_1 y_1 + \phi_2 y_2 = \begin{Bmatrix} \phi_{11} \\ \phi_{21} \end{Bmatrix} y_1 + \begin{Bmatrix} \phi_{12} \\ \phi_{22} \end{Bmatrix} y_2 \quad \text{or} \quad u = \Phi y = \begin{bmatrix} \phi_{11} & \phi_{12} \\ \phi_{21} & \phi_{22} \end{bmatrix} \begin{Bmatrix} y_1 \\ y_2 \end{Bmatrix}$$

Where ϕ_i is the eigenvector representing the deformed shape of the i th mode and y_i is the magnitude of the displacement over time.

Using the orthogonality property of the eigenvectors and non-dimensioning the eigenvectors with respect to mass, the following relationship is obtained.

► Eq 6.16
$$\phi_i^T M \phi_j = \begin{cases} 0 & (i \neq j) \\ m_i^* & (i = j) \end{cases} \quad \phi_i^T K \phi_j = \begin{cases} 0 & (i \neq j) \\ k_i^* & (i = j) \end{cases} \quad \phi_i^T C \phi_j = \begin{cases} 0 & (i \neq j) \\ c_i^* & (i = j) \end{cases}$$

There are many ways to consider the damping matrix \mathbf{C} .

- Mass and stiffness proportional damping: $\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}$
- Strain energy proportional
- Element mass and stiffness proportional

Equation 6.16 implies that if the eigenvector of the i -th mode is multiplied by Eq. 6.14, the contribution of the non- i -th mode is zero. Thus, the multi-degree-of-freedom system of Eq. 6.14 can be divided into as many 1 degree-of-freedom systems with each mode not overlapping. In the case of a two-degree-of-freedom system as in the example here, it is divided into two independent 1-degree-of-freedom systems.

► Eq 6.17

$$\begin{bmatrix} m_1^* & 0 \\ 0 & m_1^* \end{bmatrix} \begin{Bmatrix} \ddot{y}_1 \\ \ddot{y}_2 \end{Bmatrix} + \begin{bmatrix} c_1^* & 0 \\ 0 & c_1^* \end{bmatrix} \begin{Bmatrix} \dot{y}_1 \\ \dot{y}_2 \end{Bmatrix} + \begin{bmatrix} k_1^* & 0 \\ 0 & k_1^* \end{bmatrix} \begin{Bmatrix} y_1 \\ y_2 \end{Bmatrix} = - \begin{bmatrix} \phi_1^T \\ \phi_2^T \end{bmatrix} \begin{bmatrix} m_{11} & m_{12} \\ m_{21} & m_{22} \end{bmatrix} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \ddot{u}_g$$

► Eq 6.18

$$\begin{aligned} m_1^* \ddot{y}_1 + c_1^* \dot{y}_1 + k_1^* y_1 &= -\Gamma_1 \ddot{u}_g \\ m_2^* \ddot{y}_2 + c_2^* \dot{y}_2 + k_2^* y_2 &= -\Gamma_2 \ddot{u}_g \end{aligned}$$

Here, Γ_i is called the mode participation coefficient and is expressed by the following equation.

► Eq 6.19

$$\Gamma_i = \frac{\phi_i^T \mathbf{M} \mathbf{1}}{\phi_i^T \mathbf{M} \phi_i}, \quad \mathbf{1} = \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

Each 1-DOF system calculates y_1 and y_2 through numerical integration, and u can be obtained from Eq. 6.15. Since the mode superposition method divides the equation of motion with respect to the calculated mode, the number of eigenvalues calculated should be sufficient to obtain an accurate solution.

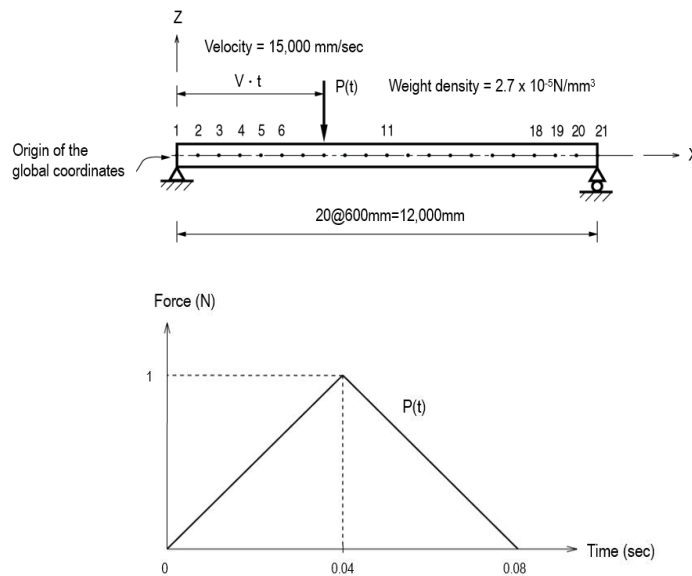


2. Tutorial

2.1 Model Overview

The following simply supported beam is subjected to a moving load 15,000mm/sec. Check the resonance effect using various period of load.

► Figure 6.3
(a) Simply supported beam
(b) Force function



► **Material**

Modulus of elasticity : $4.0 \times 10^6 \text{ N/mm}^2$

Density(γ) : $2.7 \times 10^{-5} \text{ N/mm}^3$

► **Section**

Cross-sectional area (Area) : 645 mm^2

Moment of inertia (I_{yy}) : $36,000 \text{ mm}^4$

Diameter : 250 mm


Thickness : 50 mm

g : $9,806 \text{ mm/sec}^2$

2.2 Work Environment

Open a new file and save.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : 'TH 1', Click [**SAVE**]

Set the unit system to use.

Main Menu > **Tools > Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click [**OK**]

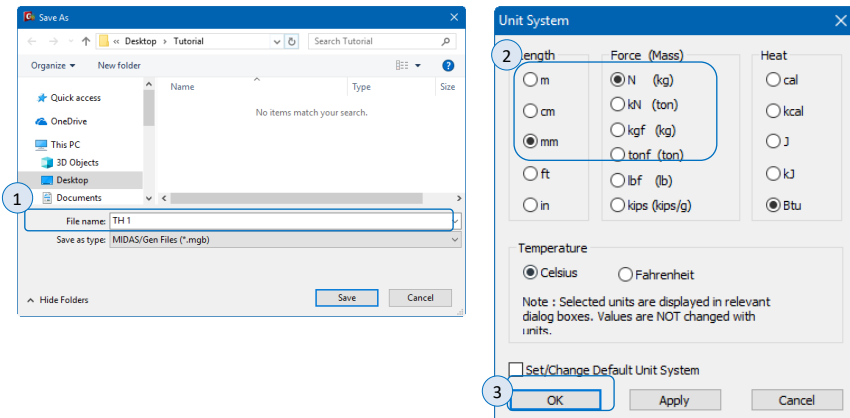
► Fig 6.4

(a) Save the file

(b) Unit system setting



The unit system setting can be easily set at the status bar at the bottom of the screen.

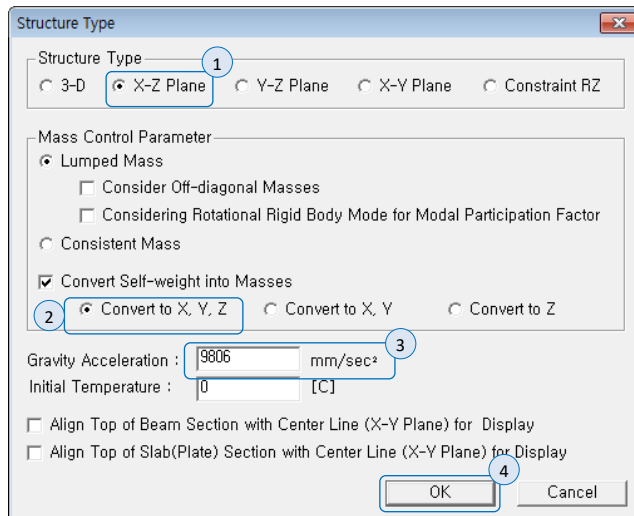


midas Gen is 3-D software, since beam exists in a 2-D plane, X-Z plane in Global Coordinate is set as the work plane, which restrains unnecessary degrees of freedom, Dy, Rx, Rz. In addition self-weight convert to nodal mass automatically.

Main Menu > **Structure** > **Type** > **Structure Type...**

1. Select Structure Type > **X-Z Plane**
2. Select Convert Self-weight into Masses > **Convert to X, Y, Z**
3. Gravity Acceleration : '**9806**'
4. Click **[OK]**

► Fig 6.5
Set work plane



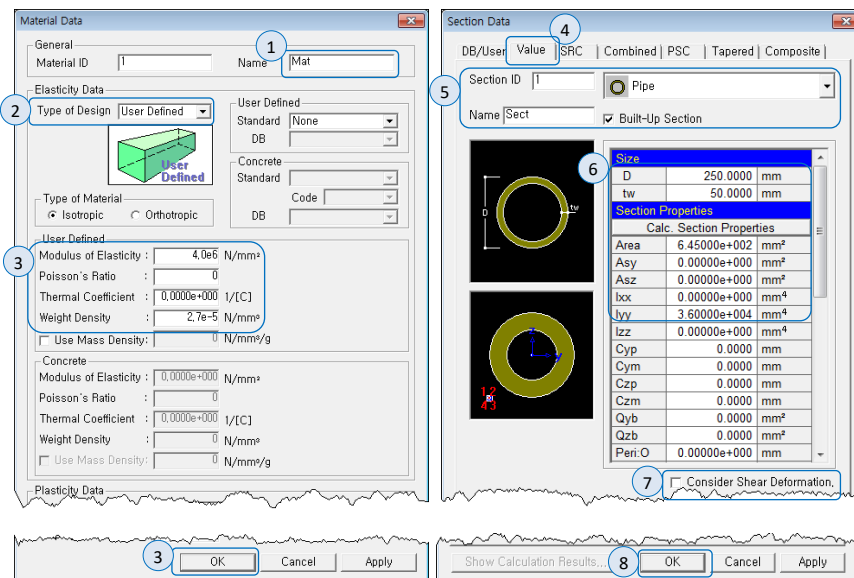
2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu > **Properties** > **Material** > **Material Properties...**

1. Click **[Add...]**, Name : '**Mat**'
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : '**4.0e6**', Weight Density : '**2.7e-5**', Click **[OK]**
4. Click **Section** Tab and **[Add...]** and Click **Value** Tab
5. Section Shape Lists > Select **Pipe**, Name : '**Sect**'
6. Size > D : '**250**', tw : '**50**', Section Properties > Area : '**645**', I_{yy} : '**36000**'
7. **Consider Shear Deformation** (off)
8. Click **[OK]** and **[Close]**

► Fig 6.6
Define material & section



2.4 Generate Node & Element

Create nodes where elements will be created.

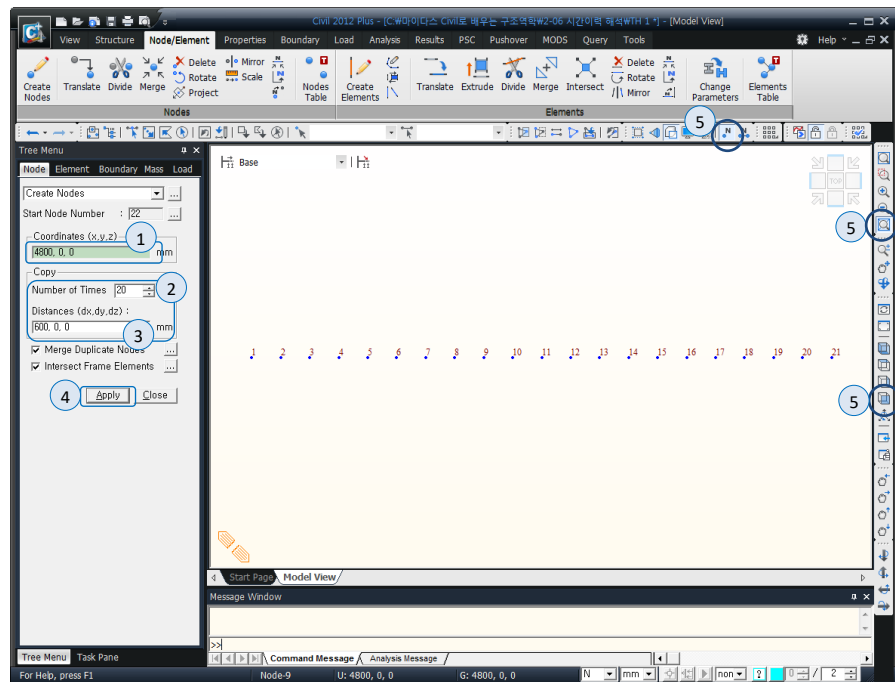
Main Menu > **Node/Element** > **Nodes** > **Create Nodes...**

1. Coordinates (x, y, z) : '0, 0, 0'
2. Number of Times: '20'
3. Distances (dx, dy, dz) : '600,0,0'
4. Click [Apply]
5. Display Node Numbers, Auto Fitting, Front View (on)

► Fig 6.7
Create nodes



When Auto Fitting is toggled on, the model fits into the full screen, which automatically controls Zoom Size in real time



Connect nodes to create elements.

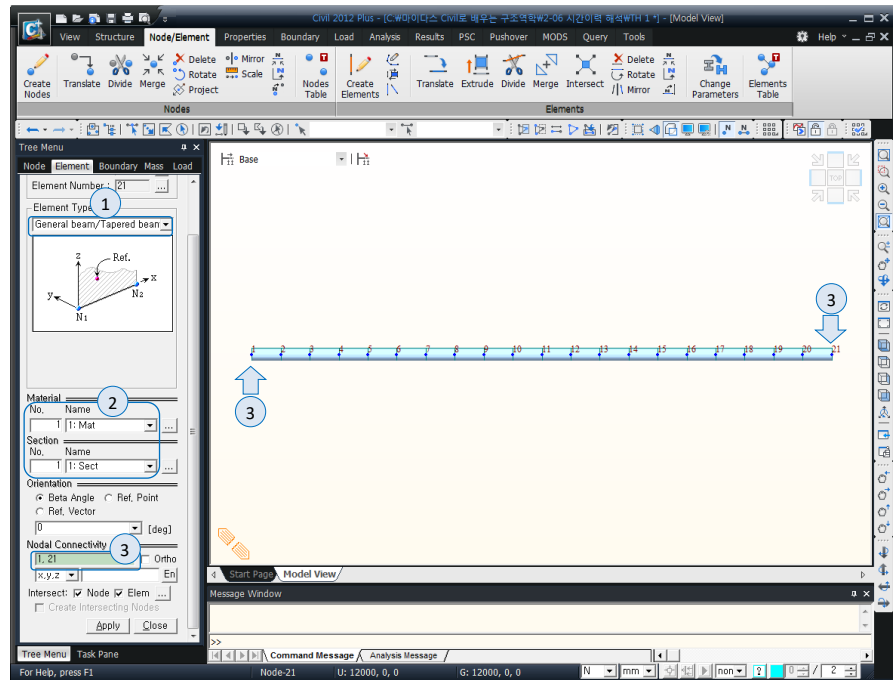
Main Menu > **Node/Element** > **Elements** > **Create Elements...**

1. Select Element Type > **General beam/Tapered beam**
2. Select Material > **1:Mat** and Section > **1:Sect**
3. Click Nodal Connectivity green box, and Click node number 1 and 21 in Model view

► Figure 6.8
Create element

Tip

By modeling a single member by dividing it into several elements, the result of detailed deformation of the member can be confirmed.



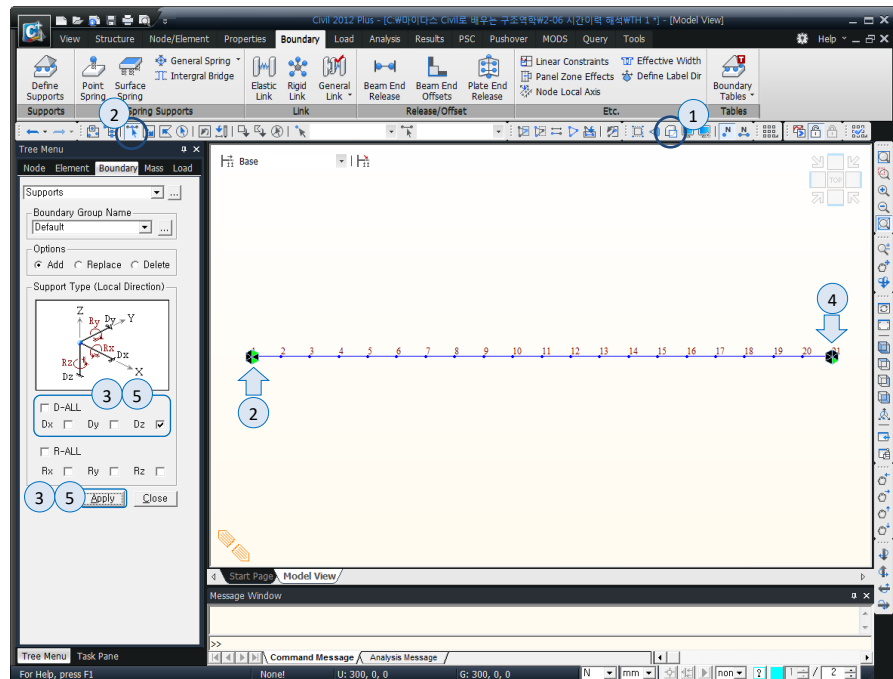
2.5 Define Boundary Conditions

In 3-D, each node restrains 6 degrees of freedom ($D_x, D_y, D_z, R_x, R_y, R_z$). But with the work plane being on X-Z plane, only 3 degrees of freedom (D_x, D_z, R_y) exist, among which the D_x, D_z degrees of freedom are restrained for the pin support, and the D_z degrees of freedom is restrained for the roller support.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. **Hidden** (off)
2. Click **Select Single** (on), Select node number 1
3. Support Type > **Dx, Dz** (on), Click **[Apply]**
4. Click **Select Single** (on), Select node number 21
5. Support Type > **Dx** (off), **Dz** (on), Click **[Apply]**

► Fig 6.9
Define boundary
condition



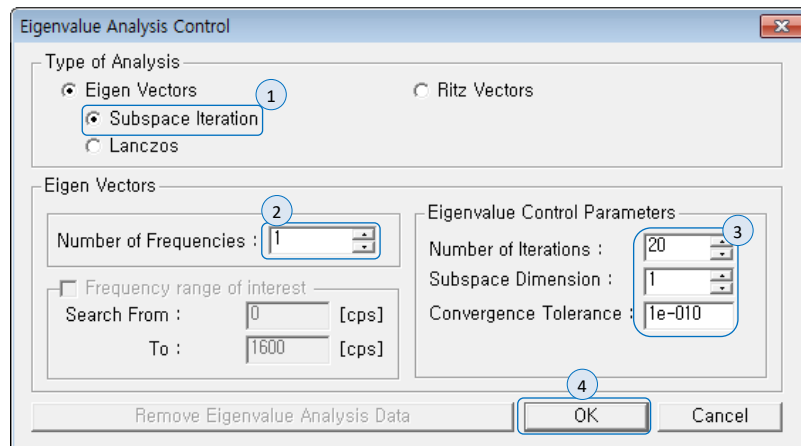
2.6 Define Loads

Define analytical data of eigenvalue analysis for time history.

Main Menu > **Analysis** > **Analysis Control** > **Eigenvalue**

1. Select Type of Analysis > Eigen Vectors > **Subspace Iteration**
2. Eigen Vectors > Number of Frequencies : '1'
3. Eigenvalue Control Parameters > Number of Iteration : '20'
Subspace Dimension : '1', Convergence Tolerance : '1e-010'
4. Click **[OK]**

► Fig 6.10
Eigenvalue analysis
conditions



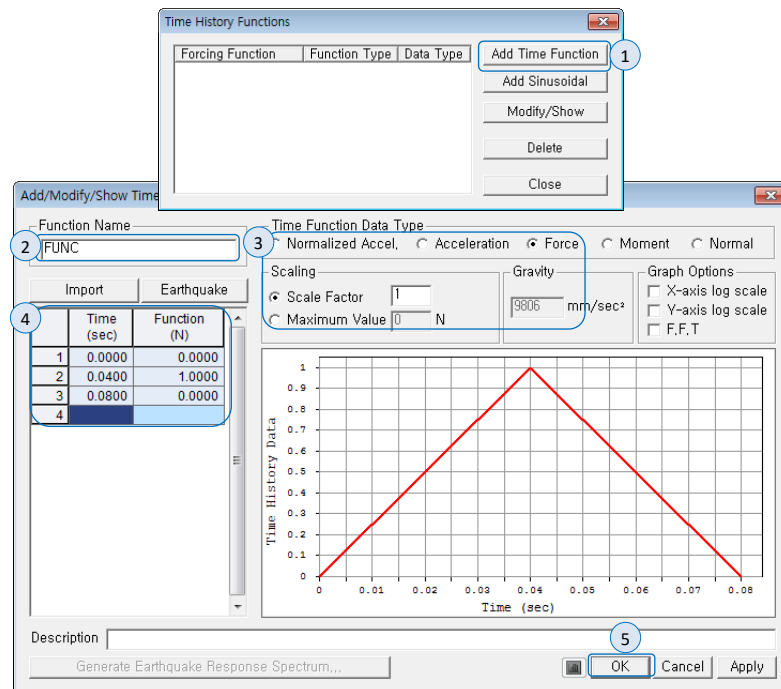
Defines the time history load to be applied to the time history analysis.

$$t = 600 / 15000 = 0.04 \text{ sec}$$

Main Menu > **Load** > **Seismic** > **Time History Functions**

1. Click **[Add Time Function]**
2. Function Name : '**FUNC**'
3. Select Time Function Data Type > **Force**, Scale Factor : '**1**'
4. in Table
Time(sec) : '**0.00**', Function(N) : '**0.0**'
Time(sec) : '**0.04**', Function(N) : '**1.0**'
Time(sec) : '**0.08**', Function(N) : '**0.0**' , Click under line, Confirm Right of Graph
5. Click **[OK]** and **[Close]**

► Fig 6.11
Time history load function

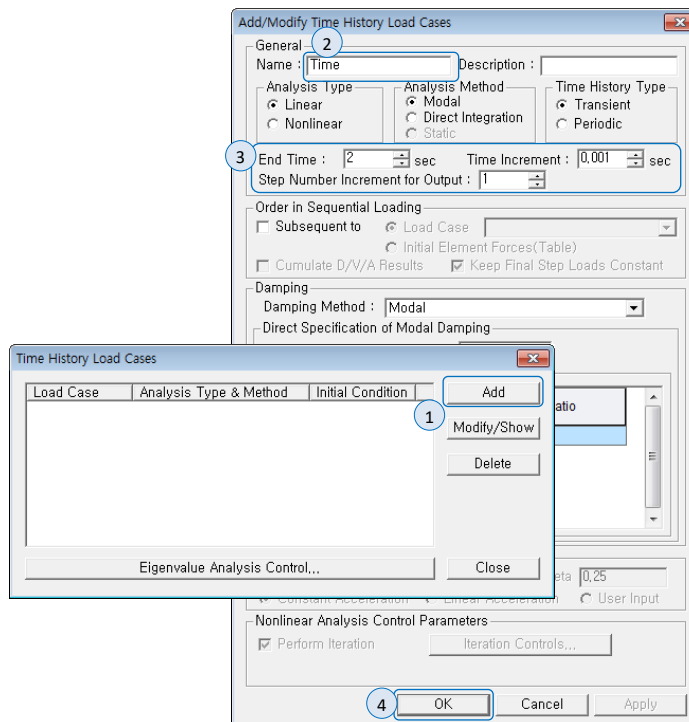


Define load case for time history.

Main Menu > **Load > Seismic > (Time History) Load Cases**

1. Click **[Add]**
2. General > Name : **'Time'**
3. End Time : **'2.0'**, Time Increment : **'0.001'**
Step Number Increment for Output : **'1.0'**
4. Click **[OK]** and **[Close]**

► Fig 6.12
Time history Load cases





Define dynamic nodal load using time history function.

Main Menu > **Load** > **Seismic** > **Dynamic Nodal Loads**

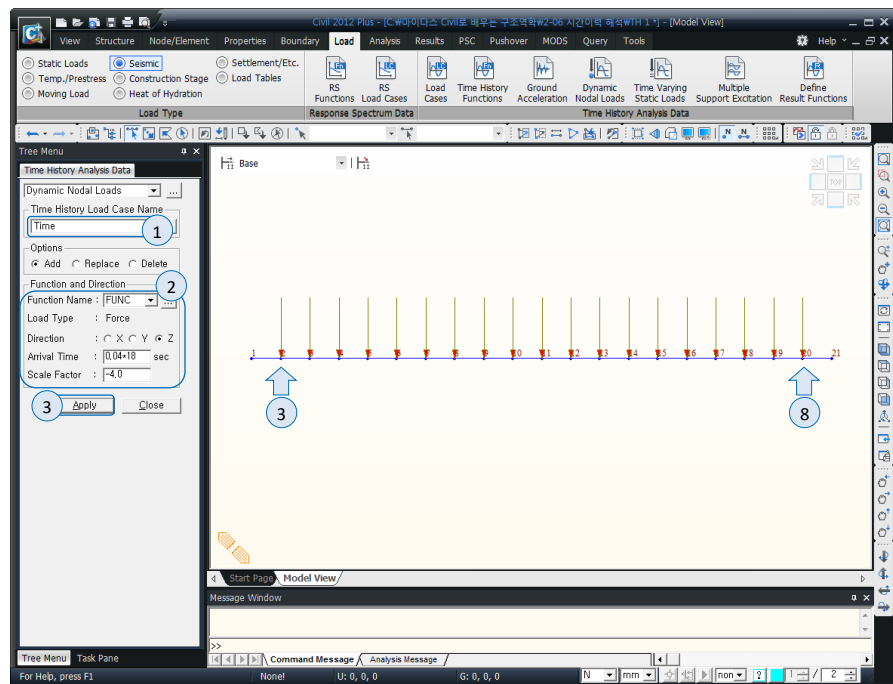
1. Select Load Case Name > **Time**
 2. Select Function and Direction > Function Name > **FUNC** and Direction > **Z**
Scale Factor : '**-4.0**'
 3. Click Select Single (on), Select node number **2** in Model View
Arrival Time : '**0**', Click [**Apply**]
 4. Click Select Single (on), Select node number **3**
Arrival Time : '**0.04*1**', Click [**Apply**]
 5. Click Select Single (on), Select node number **4**
Arrival Time : '**0.04*2**', Click [**Apply**]
 6. Click Select Single (on), Select node number **5**
Arrival Time : '**0.04*3**', Click [**Apply**]
 7. Click Select Single (on), Select node number **6**
Arrival Time : '**0.04*4**', Click [**Apply**]
Enter the Arrival Time incrementally at nodes 7, ... 19 in the same way
 8. Click Select Single (on), Select node number **20**
Arrival Time : '**0.04*18**', Click [**Apply**]
-

The scale factor -4.0 is the magnitude of the concentrated load that occur a maximum displacement of 1 mm when loaded at the center of a simply supported beam. Following is equation for scale factor.

$$\delta_{\max} = \frac{PL^3}{48EI} = 1$$

$$\therefore P = 4.0$$

► Fig 6.13
Dynamic nodal load



2.7 Perform Analysis

Perform a time history analysis.

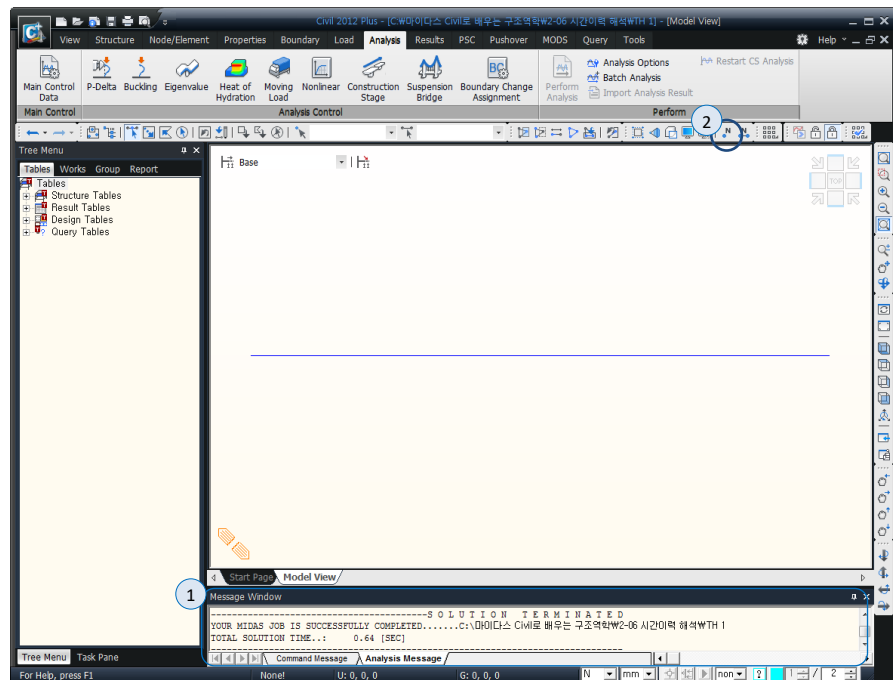
Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window

The screen is organized for the convenience of the analysis result.

2. **Display Node Numbers** (off)

► Fig 6.14
Message for a
successful run



2.8 Check Analysis Result

Calculate the natural frequencies.

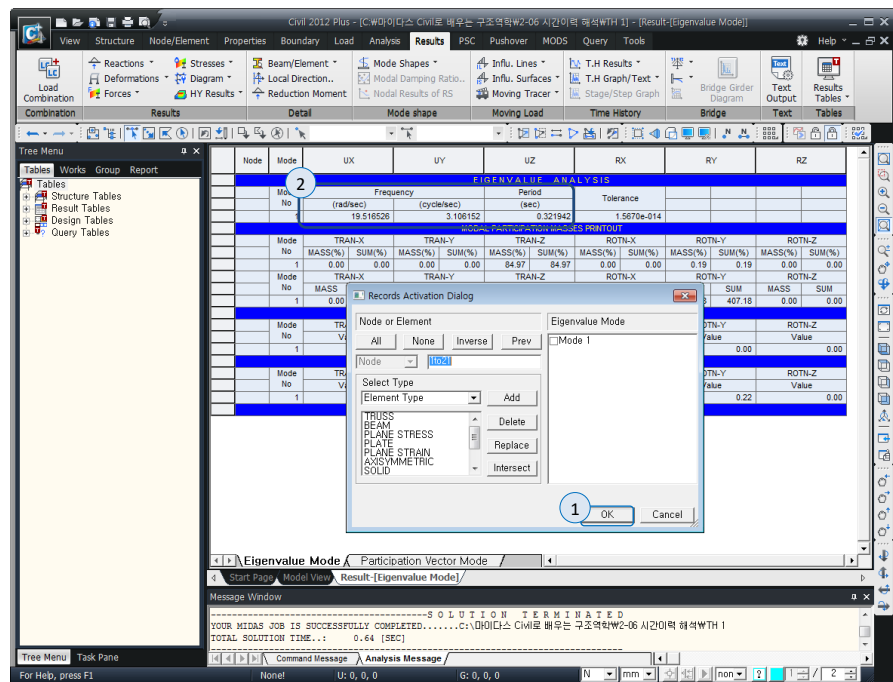
Main Menu > **Results** > **Result Tables** > **Vibration Mode Shape...**

1. Records Activation Dialog box, Click [OK]

2. Natural frequency of primary mode **19.5165** rad/sec, **3.1062** cycle/sec, Cycle **0.3219**

Confirm

► Fig 6.15
Eigen value analysis
result table



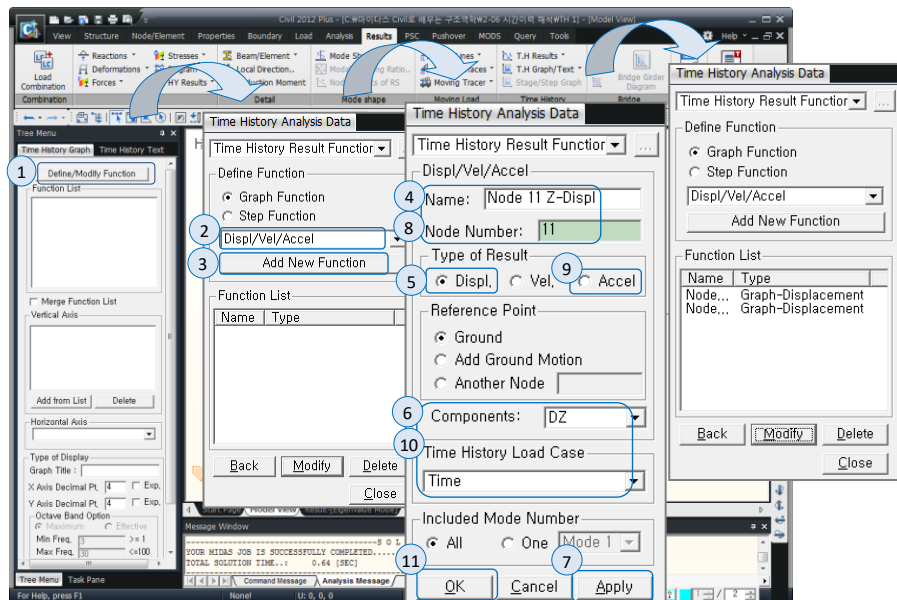
Define the time history graph.

Click **Model View Tab**

Main Menu > **Results** > **Time History Graph/Text** > **Time History Graph...**

1. Click **[Define/Modify Function]**
2. Select Define Function > **Displ/Vel/Accel**
3. Click **[Add New Function]**
4. Displ/Vel/Accel > Name : **'Node 11 Z-Displ'**, Node Number : **'11'**
5. Select Type of Result > **Displ.**
6. Select Components > **DZ** and Time History Load Case > **Time**
7. Click **[Apply]**
8. Displ/Vel/Accel > Name : **'Node 11 Z-Accel'**, Node Number : **'11'**
9. Select Type of Result > **Accel**
10. Select Components > **DZ** and Time History Load Case > **Time**
11. Click **[OK]**

► Fig 6.16
Time History Graph

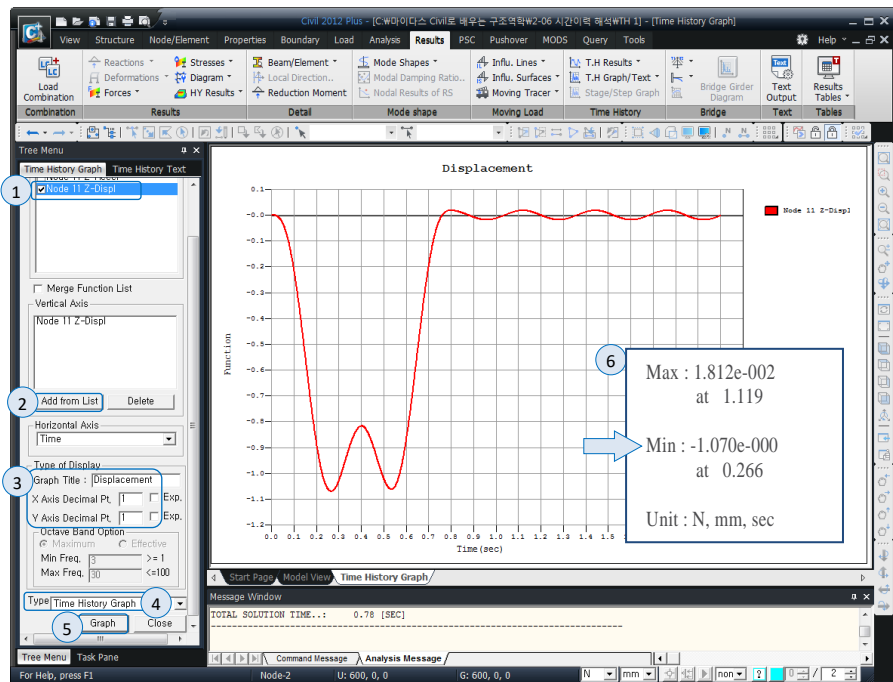


Check the maximum displacement and corresponding time.

Main Menu > **Results** > **Time History Results** > **Time History Graph...**

1. Functions List > **Node 11 Z-Displ** (on)
2. Vertical Axis > **[Add from List]**
3. Type of Display > Graph Title : '**Displacement**'
Type of Display > X Axis Decimal Pt. : '**1**', Y Axis Decimal Pt. : '**1**'
4. Select Type > **Time History Graph**
5. Click **[Graph]**
6. Check Max Displacement : **1.07mm** at 0.266sec

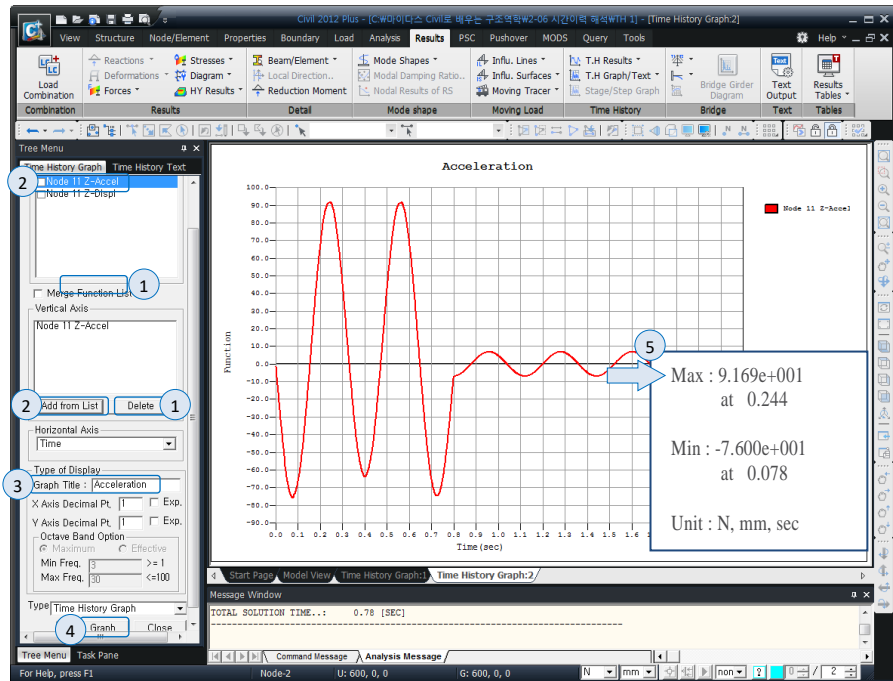
► Fig 6.17
Displacement of
time history graph



Check the maximum acceleration on the structure under dynamic loads.

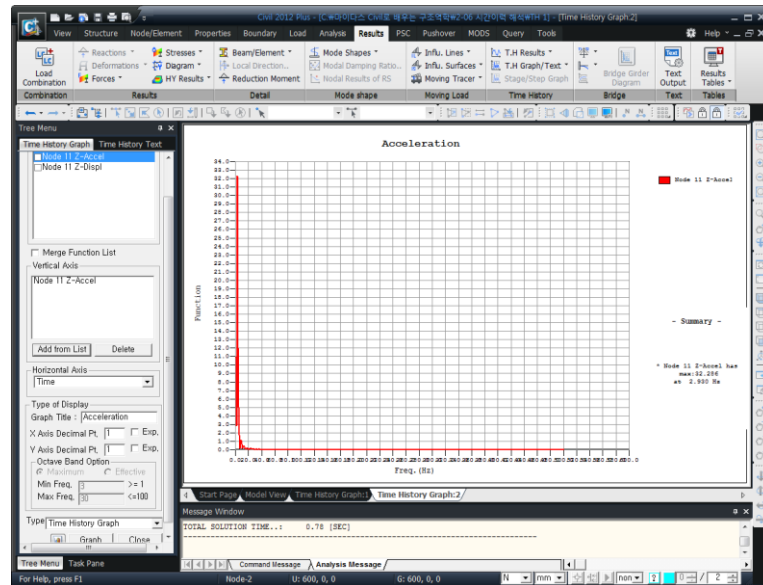
1. Select Vertical Axis > Node 11 Z-Displ, Click **[Delete]**
2. Functions List > **Node 11 Z-Accel** (on), Vertical Axis > **[Add from List]**
3. Graph Title : '**Acceleration**'
4. Click **[Graph]**
5. Check acceleration **91.69 mm/sec²** at 0.244sec

► Fig 6.18
Acceleration of
time history graph

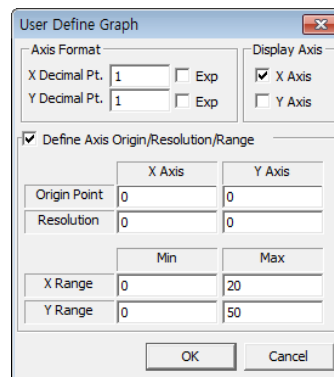
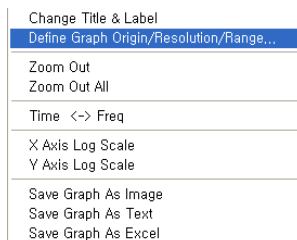


Calculate the frequencies domain using the time history of the acceleration.

► Fig 6.19
Frequency graph



Right-click 'Define Graph Origin/Resolution/Range...'



Axis Format

X Decimal Pt. : 1

Y Decimal Pt. : 1

Define Axis Origin~: Check on

X Range

Min : 0

Max : 20

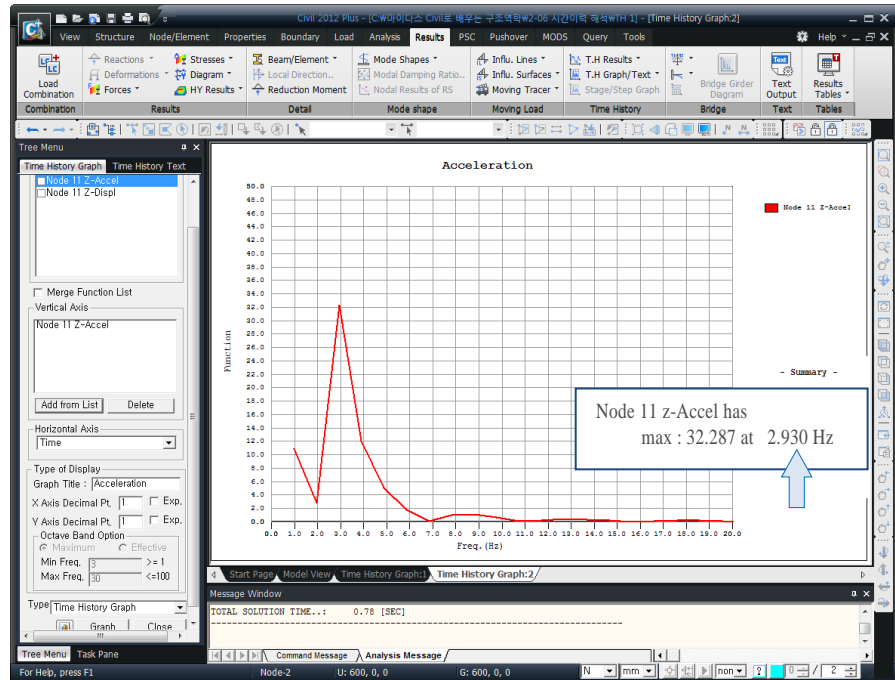
Y Range

Min : 0

Max : 50

Check the natural frequency 2.93 Hz as below garaph, and modify load in model 2.

► Fig 6.20
Acceleration frequency
history graph



Save the analytical model under a different name.

Main Menu > **File** > **Save As...**

1. File name : 'TH 2', Click [SAVE]

Add repeated time history loads of the structure frequency period (0.3219).

Main Menu > **Load** > **Seismic** > **Time History Functions**

2. Click [Add Time Function]

3. Function Name : 'FUNC 2'

4. Select Time Function Data Type > **Force**, Scale Factor : '1.0'

5. In Table

Time(sec) : '0.0000', Function(N) : '1.0' / Time(sec) : '0.3219', Function(N) : '0.0'

Time(sec) : '0.3220', Function(N) : '1.0' / Time(sec) : '0.6438', Function(N) : '0.0'

Time(sec) : '0.6439', Function(N) : '1.0' / Time(sec) : '0.9657', Function(N) : '0.0'

Time(sec) : '0.9658', Function(N) : '1.0' / Time(sec) : '1.2876', Function(N) : '0.0'

Time(sec) : '1.2877', Function(N) : '1.0' / Time(sec) : '1.6095', Function(N) : '0.0'

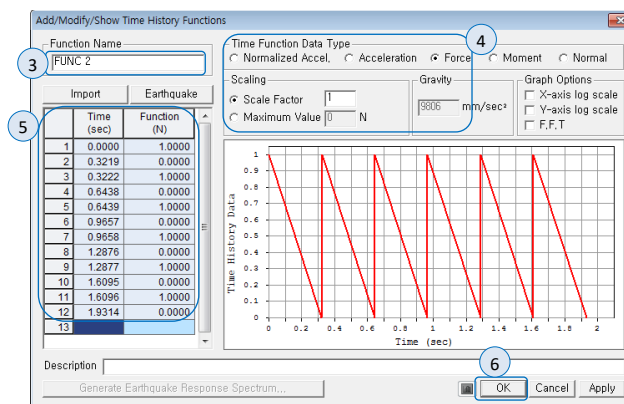
Time(sec) : '1.6096', Function(N) : '1.0' / Time(sec) : '1.9314', Function(N) : '0.0'

Click under line, Confirm Right of Graph.

6. Click [OK] and [Close]

► Fig 6.21

Modify time history load



Add load condition of time history case.

Main Menu > **Load > Seismic > (Time History) Load Cases**

1. Click **[Add]**
2. General > Name : **'Time 2'**
3. End Time : **'2.0'**, Time Increment : **'0.001'**
Step Number Increment for Output : **'1.0'**
4. Click **[OK]** and **[Close]**

► Fig 6.22
Modify time history load
condition

Add/Modify Time History Load Cases

General
 1 Name : Time 2 Description :
 2 End Time : 2 sec Time Increment : 0.001 sec
 3 Step Number Increment for Output : 1

Analysis Type
☒ Linear
☐ Nonlinear

Analysis Method
☒ Modal
☐ Direct Integration
☐ Static

Time History Type
☒ Transient
☐ Periodic

Order in Sequential Loading
☐ Subsequent to Load Case
☐ Initial Element Forces (Table)
☐ Cumulate D/V/A Results ☒ Keep Final Step Loads Constant

Damping
 Damping Method : Modal
 Direct Specification of Modal Damping
 Damping Ratio for All Modes : 0
 Modal Damping Overrides

Mode	Damping Ratio
1	

Time Integration Parameters
 Newmark Method : Gamma 0.5 Beta 0.25
☒ Constant Acceleration ☐ Linear Acceleration ☐ User Input

Nonlinear Analysis Control Parameters
☒ Perform Iteration Iteration Controls...

4 OK Cancel Apply

Modify location and arrival time of a concentrated load using dynamic nodal load table.

Main Menu > **Load** > **Load Tables** > **Time History Analysis** > **Dynamic Nodal Loads...**

1. Delete all elements except for node number 11

2. LoadCase Click the item **Time 2** Edit as

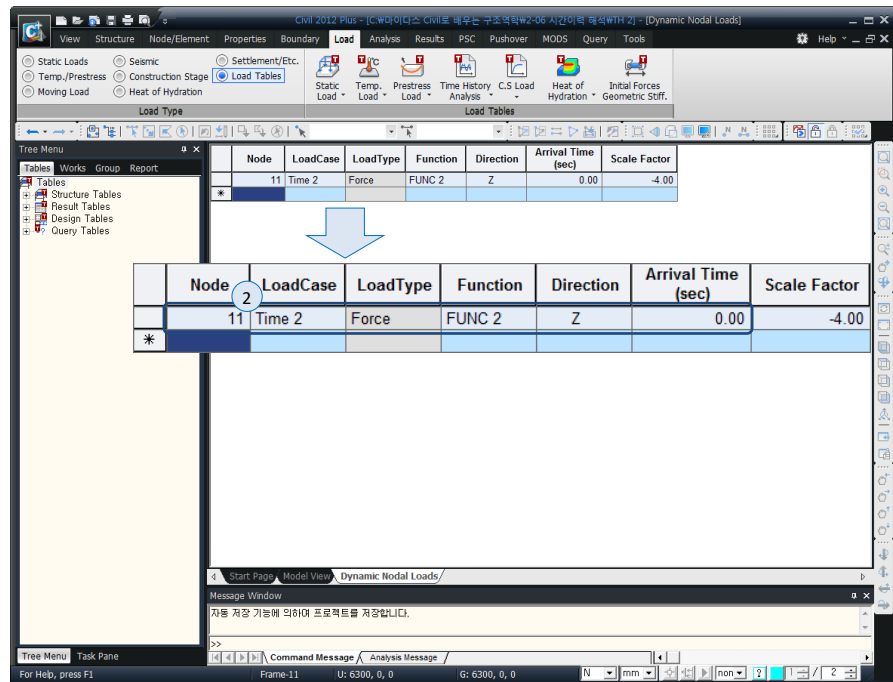
Function Click the item **FUNC 2** Edit as

Arrival Time(sec) : '0' input

After clicking the row below, confirm the final input value

► Fig 6.23

Modify dynamic node load

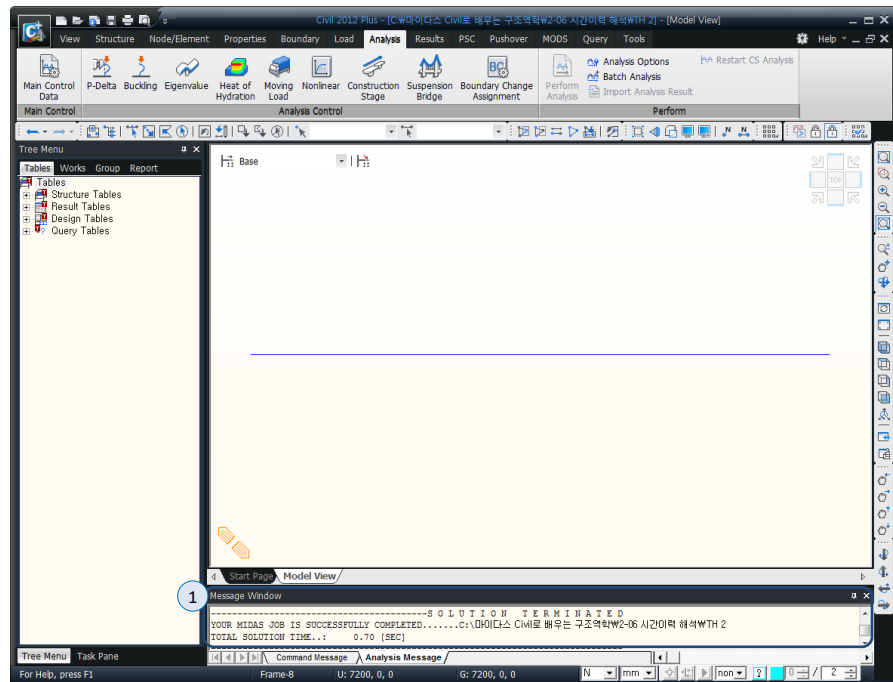


Analyze the model modified.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window
2. Click Model View Tab

► Fig 6.24
Message for a
successful run

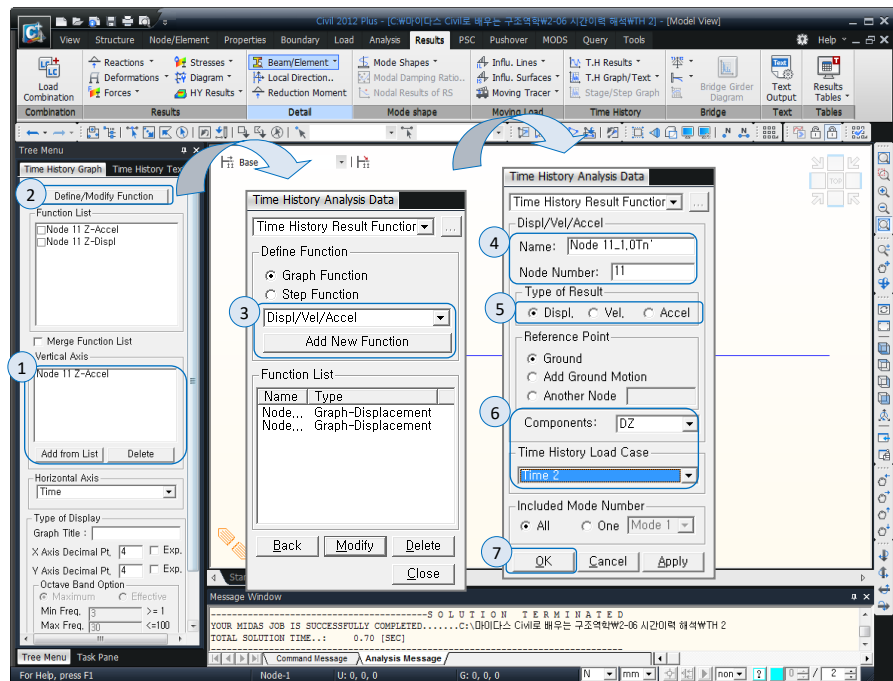


Define output of the time history graph in order to compare structure's frequency and load which has similar frequency

Main Menu > **Results** > **Time History Graph/Text** > **Time History Graph...**

1. Select Vertical Axis > **Node 11 Z-Accel**, Click **[Delete]**
2. Click **[Define/Modify Function]**
3. Select Define Function > **Displ/ Vel /Accel**, Click **[Add New Function]**
4. Displ/Vel/Accel > Name : '**Node 11_1.0Tn**', Node Number : '**11**'
5. Select Type of Result > **Displ**.
6. Select Components > **DZ** and Time History Load Case > **Time 2**
7. Click **[OK]**

► Fig 6.25
Time history graph

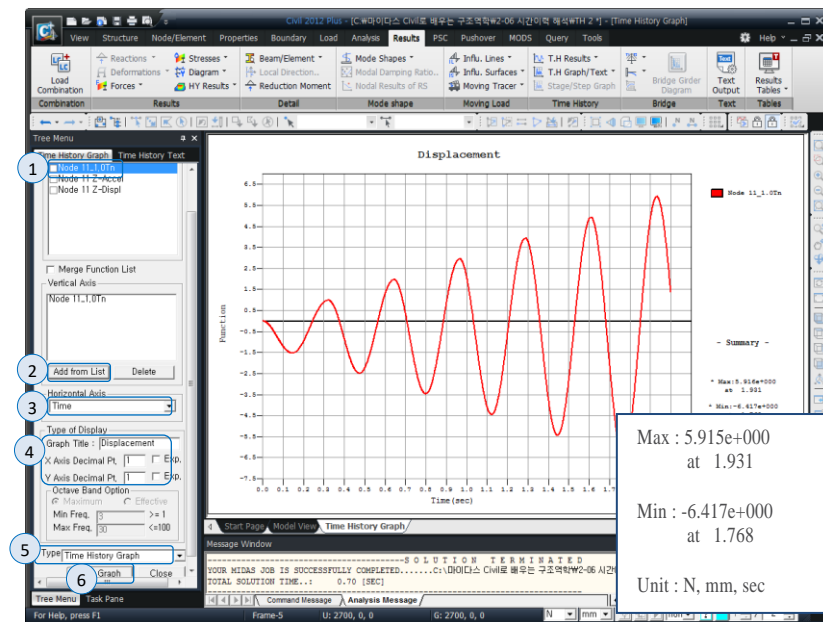


Check displacement graph when Dynamic Nodal Load is applied to node 11.

Main Menu > **Results** > **Time History Results** > **Time History Graph...**

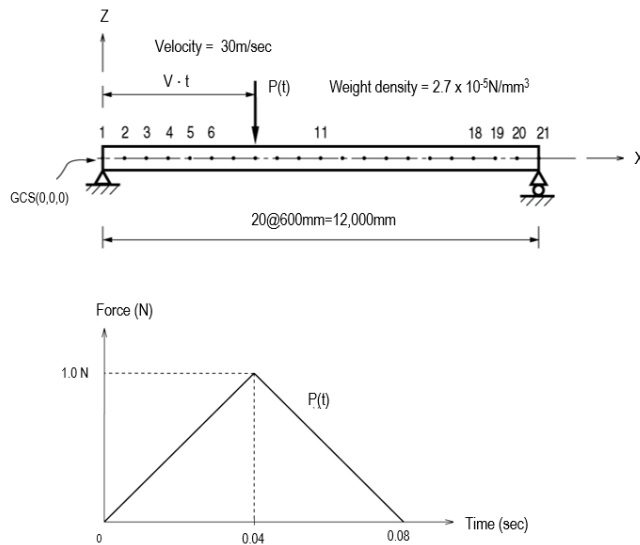
1. Functions List > **Node 11_1.0Tn** (on)
2. Click Vertical Axis > **[Add from List]**
3. Select Horizontal Axis > **Time**
4. Type of Display > Graph Title : '**Displacement**'
Type of Display > X Axis Decimal Pt. : '**1**', Y Axis Decimal Pt. : '**1**'
5. Select Type > **Time History Graph**
6. Click **[Graph]**

► Figure 6.26
Displacement of
time history graph



3. Exercise

The following simply supported beam is subjected to a moving load 30m/sec. Check the resonance effect using various period of load (TH 1.mgb).



➤ **Material**

Modulus of elasticity: $4.0 \times 10^6 \text{ N/mm}^2$

Density(γ) : $2.7 \times 10^{-5} \text{ N/mm}^3$

➤ **Section**

Cross-sectional area (Area) : 645 mm^2

Moment of inertia (I_y): $36,000 \text{ mm}^4$

Diameter: 250 mm

Thickness: 50 mm

g: $9,806 \text{ mm/sec}^2$



7. Response Spectrum Analysis

Contents

1 Introduction

1.1 Concept of Response Spectrum Analysis	7-3
---	-----

2 Tutorial

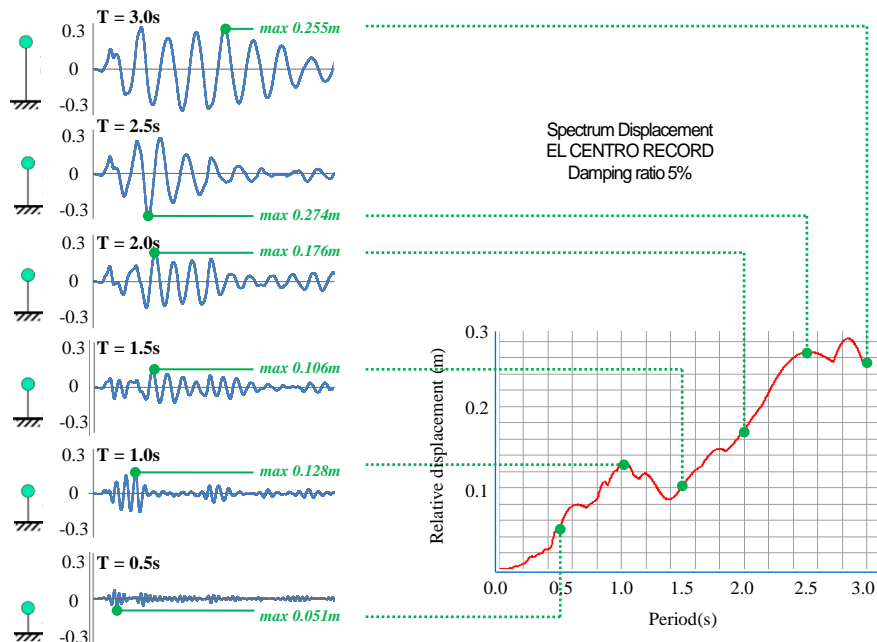
2.1 Model Overview	7-7
2.2 Work Environment	7-9
2.3 Material & Section Properties	7-11
2.4 Generate Node & Element	7-13
2.5 Define Boundary Conditions	7-17
2.6 Define Loads	7-19
2.7 Perform Analysis	7-23
2.8 Check Analysis Result	7-24



1. Introduction 1.1 Concept of Response Spectrum Analysis

In the time history analysis method, the history of the structure's response such as acceleration, velocity, and displacement is calculated. On the other hand, in the response spectral analysis, the maximum response value is determined for each mode, and then the total response value is predicted by combining these response using appropriate combination method. Figure 7.1 shows the process of finding the maximum response value for structures with different time period and showing the maximum displacement value corresponding to these period on the graph.

► Fig 7.1
How to calculate the
Response spectrum
Displacement



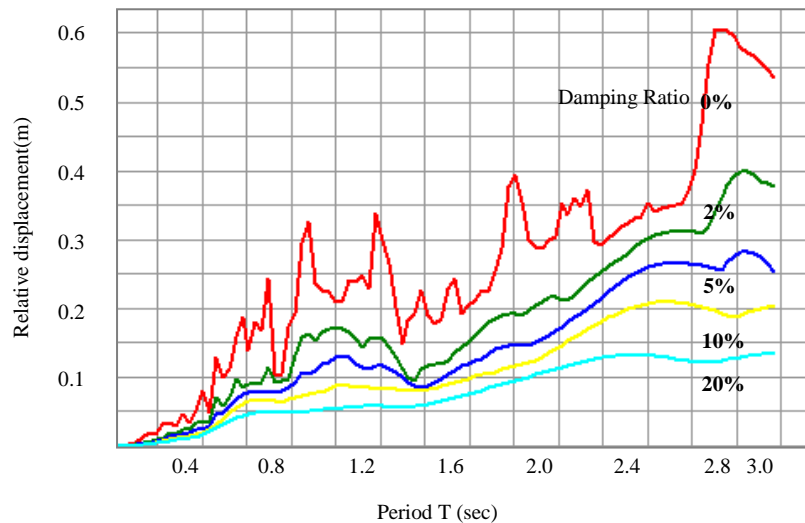
To the left in Figure 7.1, the process of finding the maximum displacement after performing the time history analysis for structures with Period (0.5s to 3.0s) is shown. The figure on the



right is a graph showing the maximum displacement value of each structure with corresponding period. Because the figure shows the maximum displacement for each period of the structure, the term "response spectrum" is used. A similar picture can be drawn for acceleration and velocity.

In the structural analysis programs, the response spectrum per time period is calculated in the program, and the maximum response corresponding to the specific period is computed. Figure 7.2 shows the spectral displacements vs period determined by varying damping ratio.

► Fig 7.2
Response spectrum
Displacement based on
Damping ratio



As the structure becomes complicated, the number of Eigen modes (or frequencies) contributing to the deformation of the structure increases, resulting in a state reflecting the contribution of each mode. Therefore, when the response spectrum for each period is obtained, the contribution of the response spectrum for each period should be calculated. In the eigenvalue analysis, the mode participation coefficient Γ_i (i represents the mode) for each mode is calculated as explained in Chapter 5.

The Story Shear and displacement are calculated as follows by multiplying the maximum response value corresponding to a time period of the mode



► Eq 7.1
$$V_i = \Gamma_i m (S_{ai} g) \phi_i$$

► Eq 7.2
$$D_i = \Gamma_i S_{di} \phi_i$$

where in the equations above, m is the total mass of the structure. S_{ai} and S_{di} (i is the mode), represents ratio of spectral acceleration of the i -th mode to the gravitational acceleration and spectral displacement respectively. g , the gravitational acceleration and ϕ_i , the unique eigenvector corresponding to the i -th mode. The relationship between spectral acceleration, velocity, and displacement is as follows

► Eq 7.3
$$S_a = \omega S_v = \omega^2 S_d, \text{ where } \omega, \text{ is the natural frequency.}$$

From equation 7.1, it can be seen that the Story shear of the corresponding mode is obtained by multiplying the mass by the spectral acceleration, calculating the layer distribution by reflecting the mode shape, and then multiplying the mode participation coefficient. In the case of the Story displacement, the mode distribution is calculated by reflecting the mode shape to the spectral displacement, and then the mode participation coefficient is multiplied as shown in equation 7.2.

Once each mode-specific response is obtained, it must be combined using appropriate combination methods. As can be seen in Figure 7.1, since the time at which the maximum value is obtained is different, the response cannot be obtained by a simple sum of the maximum values. Therefore, the appropriate method among the following methods is selected and the responses for each mode are combined accordingly.

► Eq 7.4 - SRSS (Square Root of the Sum of the Squares):
$$R_{\max} = [R_1^2 + R_2^2 + \dots + R_n^2]^{1/2}$$

► Eq 7.5 - ABS (Absolute Sum):
$$R_{\max} = |R_1| + |R_2| + \dots + |R_n|$$

► Eq 7.6 - CQC (Complete Quadratic Combination):
$$R_{\max} = \left[\sum_{i=1}^N \sum_{j=1}^N R_i \rho_{ij} R_j \right]^{1/2}$$

where, R_{\max} = Maximum Response Value

R_i = the peak value of the particular response for the i -th mode

$$\rho_{ij} = \frac{8\xi^2(1+r)r^{3/2}}{(1-r^2)^2 + 4\xi^2r(1+r)^2}, \quad r = \frac{\omega_j}{\omega_i}$$



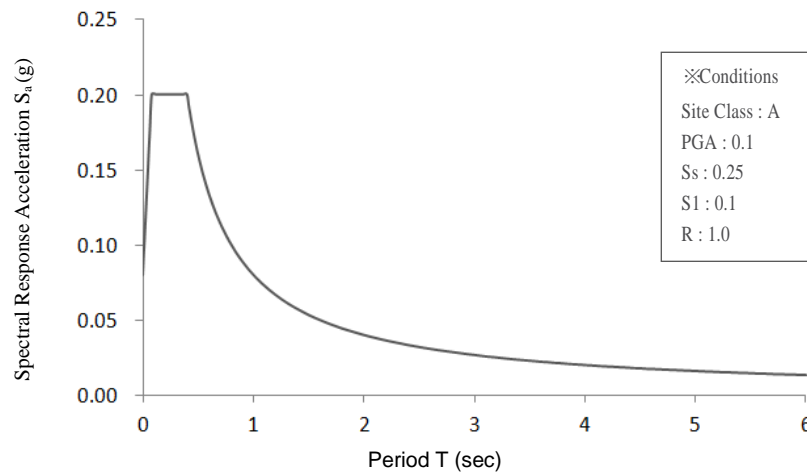
Structural Analysis II (Advanced)

7. Response Spectrum Analysis

r = the ratio of the natural frequency at the i -th mode to the natural frequency at the j -th mode
 ξ = Damping ratio

Applying the response spectrum analysis method to the first mode only results in an equivalent static analysis method. Figure 7.3 below shows the design spectrum for the acceleration used in the AASHTO LRFD 6.

► Fig 7.3
Design Spectrum
Acceleration



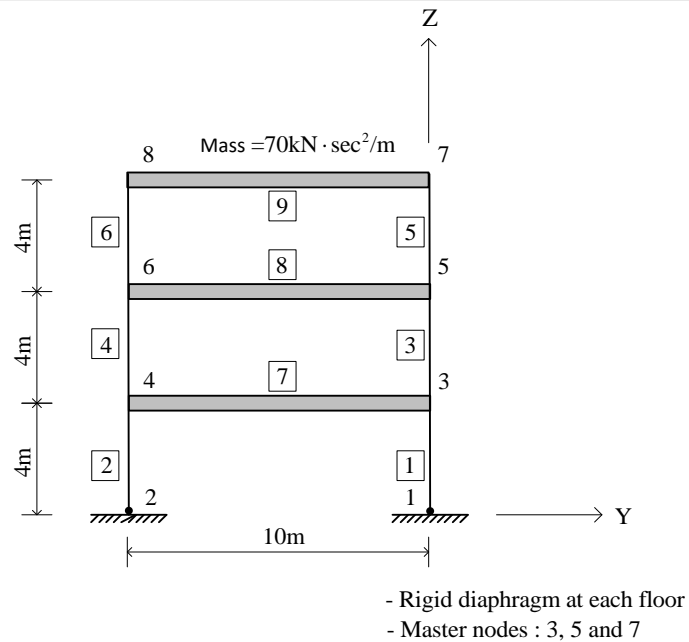


2. Tutorial

2.1 Model Overview

The response spectrum analyses are generally carried out for seismic designs using the design spectra defined in designed standards.

► Fig 7.4
Analytical model






- **Material**
Modulus of elasticity : $2.0 \times 10^5 \text{ N/mm}^2$
- **Section**
Column: Area : $6.0 \times 10^4 \text{ mm}^2$
Moment of inertia (I_{yy}) : $4.0 \times 10^8 \text{ mm}^4$
Beam: moment of inertia (I_{yy}) : $4.0 \times 10^4 \text{ mm}^4$
The floor mass (M): $70.0 \text{ kN}\cdot\text{sec}^2/\text{m}$
Damping ratio (ξ) : 0.05 (5%)
Gravitational acceleration (g) : $9,806 \text{ mm/sec}^2$
Response Spectrum Data: El Centro N-S

2.2 Work Environment

Open a new file and save.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : **'Response'**, Click **[SAVE]**

Set the unit system to use.

Main Menu > **Tools > Unit System...**

2. Select Length > **mm**, Force(Mass) > **N**

3. Click **[OK]**

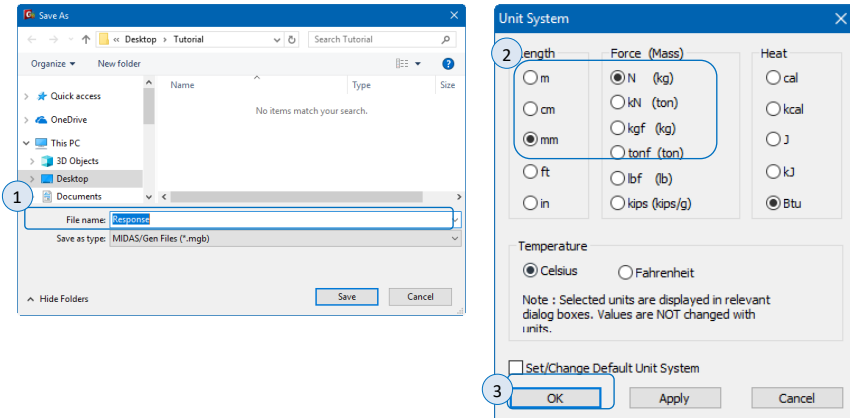
► Fig 7.5

(a) File save

(b) Unit system setting



The unit system setting
can be easily set at the

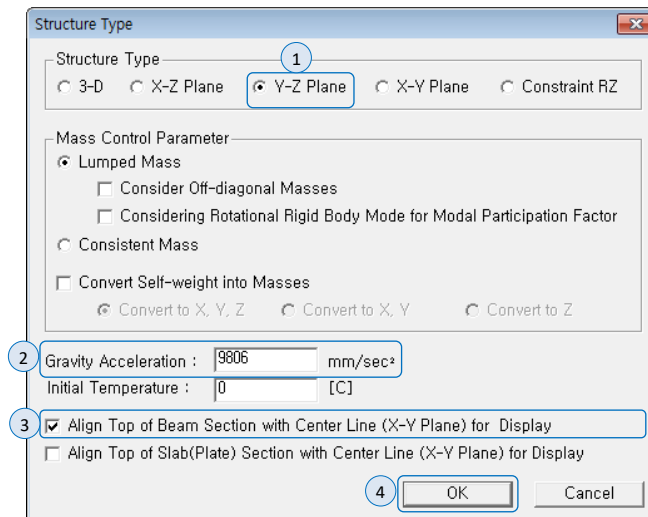


The example models exist in the 2-D, X-Z plane

Main Menu > **Structure** > **Type** > **Structure Type...**

1. Select Structure Type > **Y-Z Plane**
2. Gravity Acceleration : **'9806'**
3. **Align Top of Beam Section with Floor~** (on)
4. Click **[OK]**

► Fig 7.6
Set work plane





2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu > **Model** > **Properties** > **Material...**

1. Click **[Add...]**, Name : **'Mat'**
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : **'2.0e5'**, Click **[OK]**
4. Click **Section** Tab and **[Add...]** and **Value** Tab
5. Select Section Shape Lists > **Solid Rectangle**, Name : **'Column'**
6. Size > H : **'1'**, B : **'1'**, Section Properties > Area : **'6.0e12'**, I_{zz} : **'4.0e8'**
7. **Consider Shear Deformation** (off)
8. Click **[Apply]**
9. Name : **'Beam'**
10. Size > H : **'500'**, B : **'1'**, Section Properties > I_{yy} : **'4.0e14'**, I_{zz} : **'0'**
11. **Consider Shear Deformation** (off)
12. Click **[OK]** and **[Close]**

► Figure 7.7

- (a) Material definition
(b) Section definition

Material Data

General

Material ID: 1 Name: Mat

Elasticity Data

Type of Design: User Defined

User Defined

Standard: None DB: None

Concrete Standard: Code: DB:

Type of Material: ☒ Isotropic ☐ Orthotropic

User Defined

Modulus of Elasticity: 2.0e5 N/mm²

Poisson's Ratio: 0

Thermal Coefficient: 0.0000e+000 1/[C]

Weight Density: 0 N/mm³

☐ Use Mass Density: 0 N/mm³/g

Concrete

Modulus of Elasticity: 0.0000e+000 N/mm²

Poisson's Ratio: 0

Thermal Coefficient: 0.0000e+000 1/[C]

Weight Density: 0 N/mm³

☐ Use Mass Density: 0 N/mm³/g

Plasticity Data

Plastic Material Name: NONE

Thermal Transfer

Specific Heat: 0 kcal/N [C]

Heat Conduction: 0 kcal/mm hr [C]

Damping Ratio: 0

OK

Cancel

Apply

Section Data

DB/User Value SRC Combined PSC Tapered Composite

Section ID: 1 Solid Rectangle

Name: Column ☒ Built-Up Section

Size

H: 1.0000 mm

B: 1.0000 mm

Section Properties

Calc. Section Properties

Area	6.00000e+012	mm²
Asy	0.00000e+000	mm²
Asz	0.00000e+000	mm²
Ixx	0.00000e+000	mm⁴
Iyy	0.00000e+000	mm⁴
Izz	4.00000e+008	mm⁴
Cyp	0.0000	mm
Cym	0.0000	mm
Czp	0.0000	mm
Czm	0.0000	mm
Qyb	0.0000	mm²
Qzb	0.0000	mm²
Peri.O	0.00000e+000	mm

☐ Consider Shear Deformation.

Offset: Center-Center

Change Offset ...

Show Calculation Results...

OK

Cancel

Apply

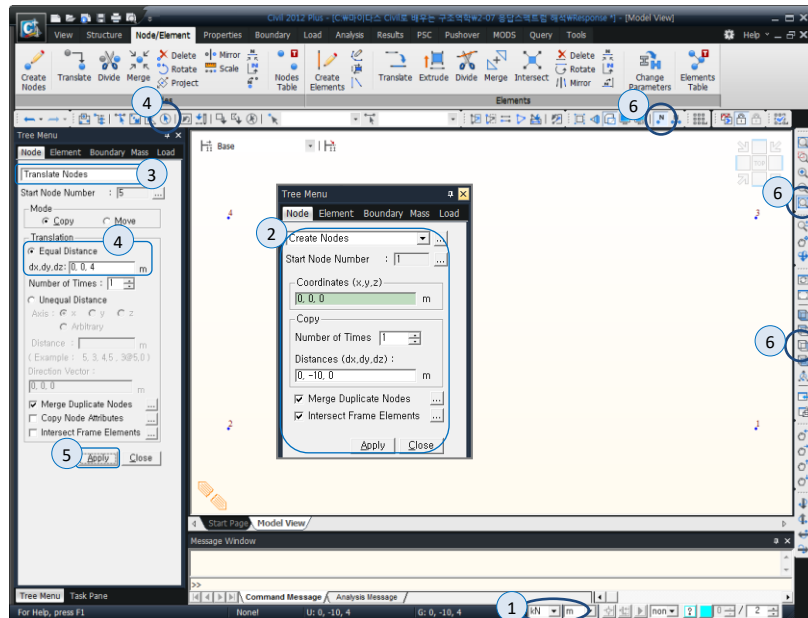
2.4 Generate Nodes & Elements

Create nodes where elements will be created

Main Menu > **Node/Element** > **Nodes** > **Create Nodes**

1. Select Length > **m, kN** at bottom of model view.
2. Coordinates (x, y, z) : '**0, 0, 0**'
Copy > Number of Times: '**1**', Distances (dx, dy, dz) : '**0,-10,0**'
Click **[Apply]**
3. Select **Translate Nodes**
4. Click **Select All**, Translation > Equal Distance > dx, dy, dz : '**0,0,4**'
5. Click **[Apply]**
6. **Display Node Numbers, Auto Fitting, Right View (on)**

► Fig 7.8
Create Nodes

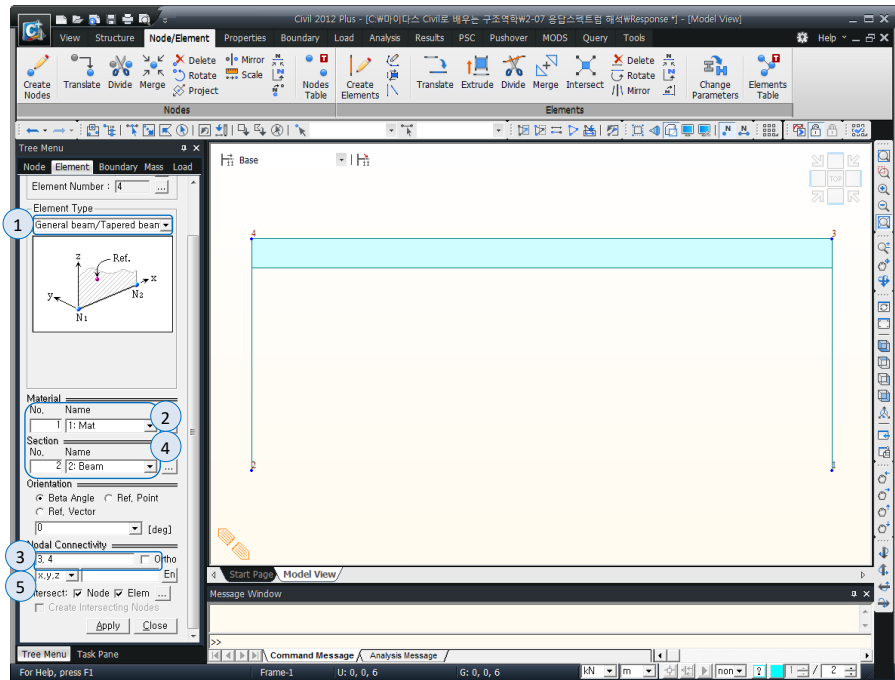


Connect nodes to create elements.

Main Menu > **Node/Element** > **Elements** > **Create Elements**

1. Select Element Type > **General beam/Tapered beam**
2. Select Material > **1:Mat** and Section > **1:Column**
3. Click Nodal Connectivity green box, and Click node number **(1, 3), (2, 4)** in Model view
4. Select Material > **1:Mat** and Section > **2:Beam**
5. Click Nodal Connectivity green box, and Click node number **(3, 4)** in Model view

► Fig 7.9
Create elements



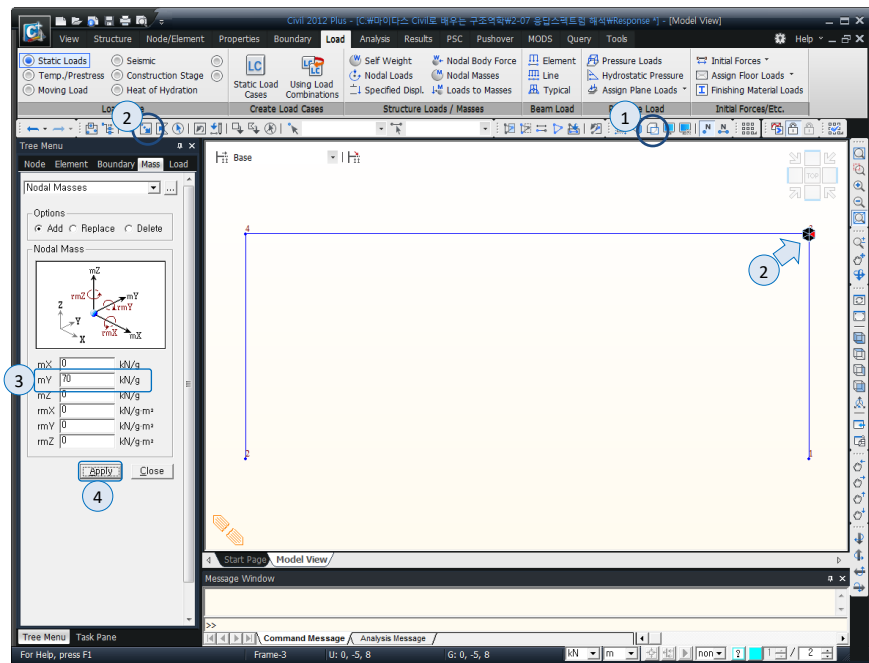
Input floor mass data for eigenvalue analysis.

Main Menu > **Load** > **Static Loads** > **Nodal Masses**

Main Menu > **View** > **Display...**, > Misc Tab > **Nodal Mass** (on), Click [OK]

1. **Hidden** (off)
2. Click **Select Single** (on), Select node number 3
3. Nodal Mass > mY : '70'
4. Click [Apply]

► Fig 7.10
Floor mass input

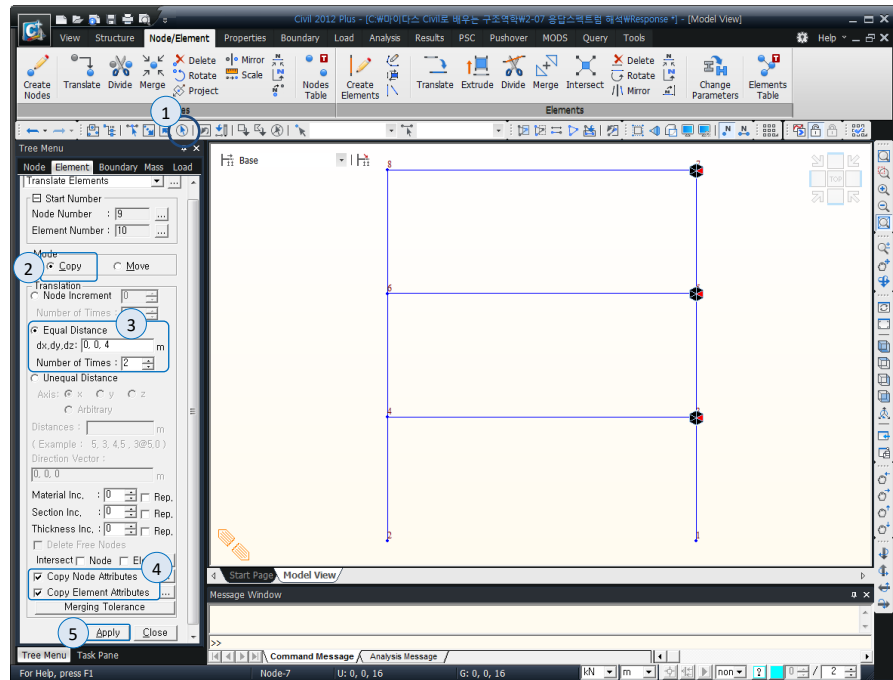


Copy the beam to create 2nd floor.

Main Menu > **Node/Element** > **Elements** > **Translate**

1. Click **Select All**
2. Select Mode > **Copy**
- 3 Translation > **Equal Distance** > dx, dy, dz : **0, 0, 4** , Number of Times : **'2'**
4. **'Copy Node Attributes'**, **'Copy Element Attributes'** (on)
5. Click **[Apply]**

► Fig 7.11
Three-story framing



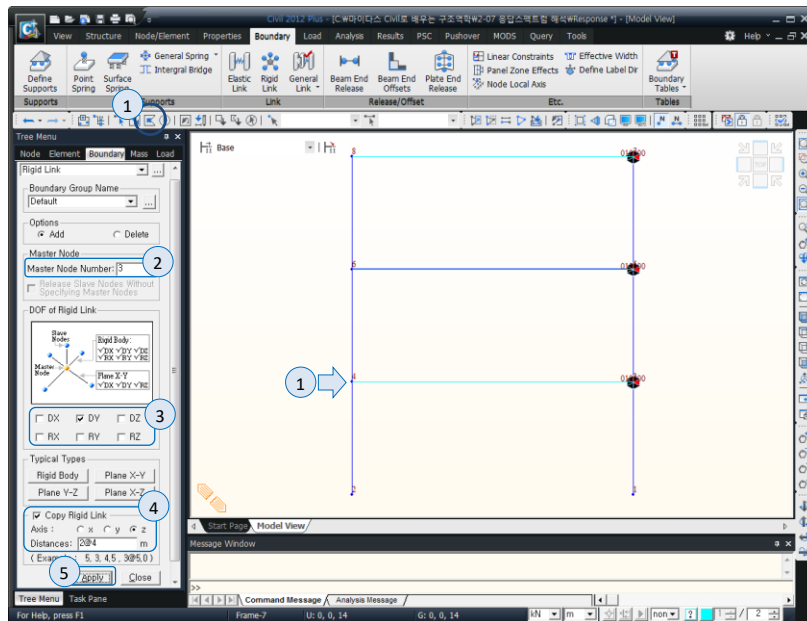
2.5 Define Boundary Conditions

Connect node no.4 and node no. 3 to create which is master node of right link.

Main Menu > **Boundary** > **Link** > **Rigid Link...**

1. Click **Select Single** (on), Select node number 4
2. Master Node Number : '3'
3. DOF of Rigid Link > **DY** (on), **DX**, **DZ**, **RX**, **RY**, **RZ** (off)
4. **Copy Rigid Link** (on)
Select Axis > **z**, Distance : '**2@4**'
5. Click **[Apply]**

► Fig 7.12
Support condition input



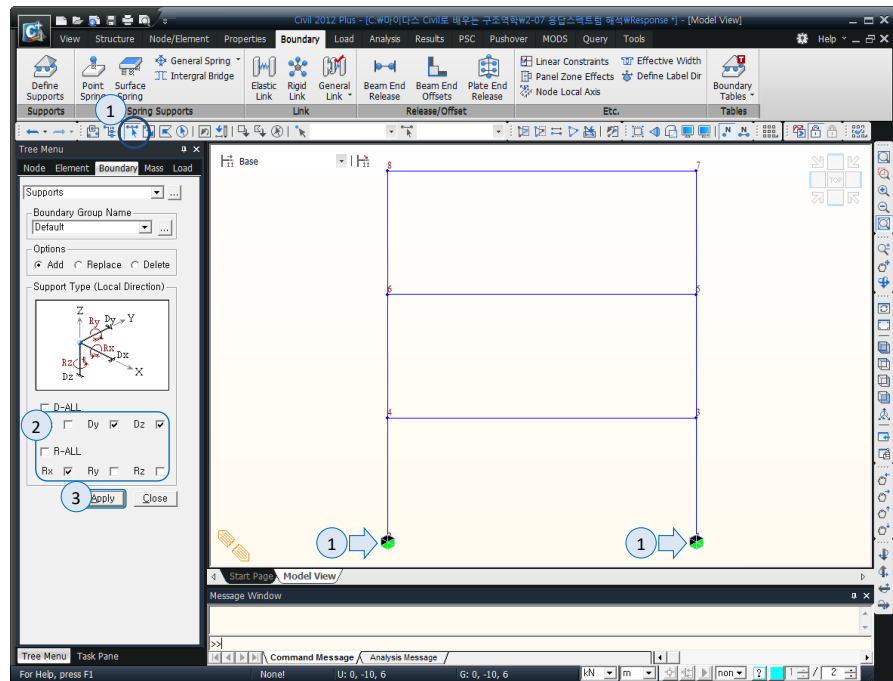
Fixed support are assigned to the bottom of the columns.

Main Menu > **Boundary** > **Supports** > **Define Supports**

Main Menu > **View** > **Display...**, > Misc Tab > **Nodal Mass** (off), Click **[OK]**

1. Click **Select Single** (on), Select node number 1 and 2
2. Support Type > **Dy, Dz, Rx** (on)
3. Click **[Apply]**

► Fig 7.13
Input support conditions



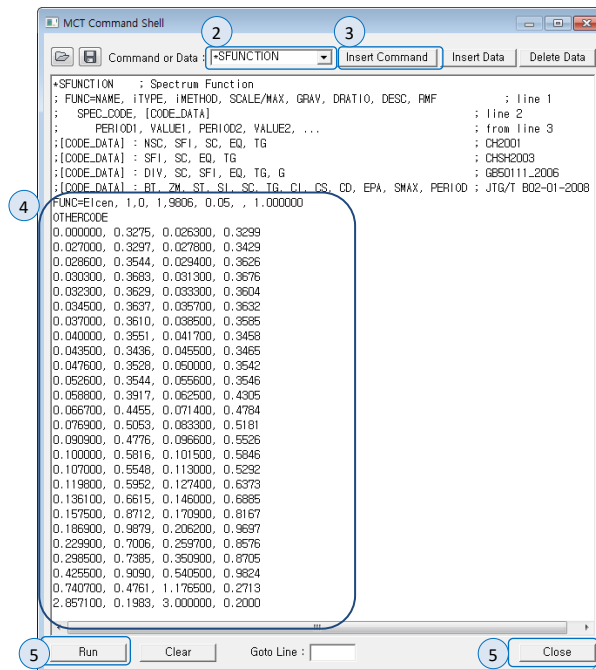
2.6 Define Loads

Perform the response spectrum analysis using exact data of El Centro N-S seismic 1940. Input real seismic data using text command shell in midas Gen.

Main Menu > **Tools > Command Shell > MGT Command Shell**

1. modify Length unit to **mm** at bottom of model view.
2. Select Command or Data > ***SFUNCTION**
3. Click **[Insert Command]**
4. Please note that the data input Spacing, comma(,), full stop(.)
5. Click **[Run]** and **[Close]**

► Fig 7.14
El Centro Seismic
Spectrum Data



Define the response spectrum load case.

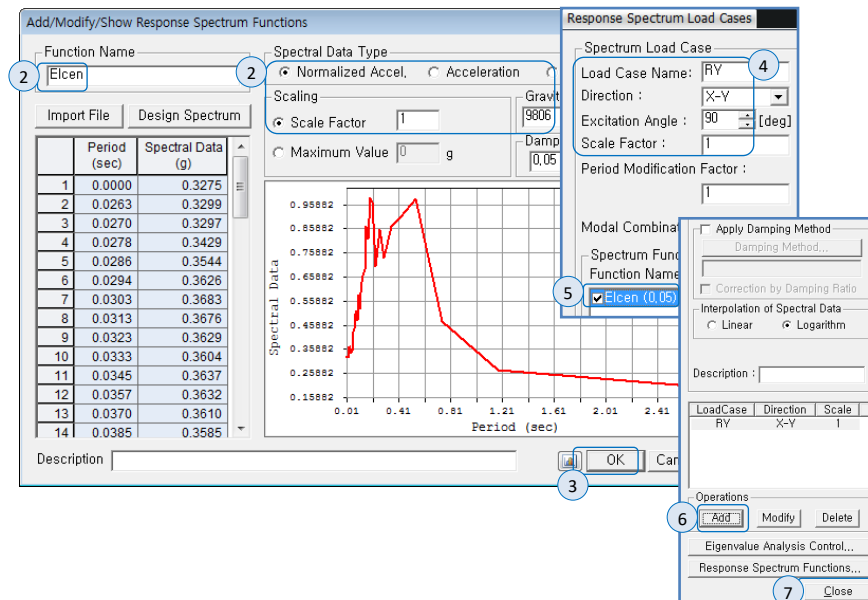
Main Menu > **Load** > **Seismic** > **RS (Response Spectrum) Functions**

1. Double Click Spectrum Name > **Elcen**
2. Function Name : '**Elcen**', Spectral Data Type> Normalized Accel.
Scale Factor: '**1**', Gravity : '**9806**'
3. Click **[OK]** and **[Close]**

Main Menu > **Load** > **Seismic** > **RS (Response Spectrum) Load Cases**

4. Spectrum Load Case > Load Case Name : '**RY**'
Select Direction > **X-Y**, Excitation Angle : '**90**'
5. Spectrum Functions > **Elcen (0.05)** (on)
6. Click Operation > **[Add]**
7. Click **[Close]**

► Fig 7.15
Response Spectrum
Functions

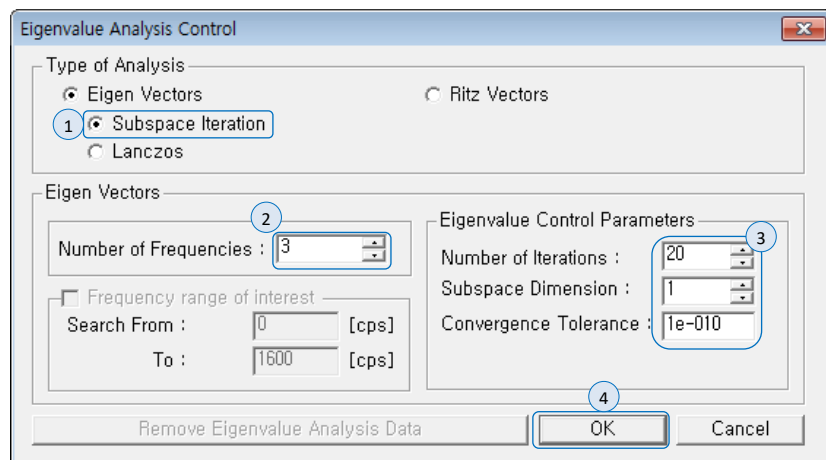


Define eigenvalue analysis parameters for response spectrum analysis.

Main Menu > **Analysis** > **Analysis Control** > **Eigenvalue**

1. Select Type of Analysis > Eigen Vectors > **Subspace Iteration**
2. Eigen Vectors > Number of Frequencies : '3'
3. Eigenvalue Control Parameters > Number of Iteration : '20'
Subspace Dimension : '0', Convergence Tolerance : '1e-010'
4. Click **[OK]**

► Fig 7.16
Eigenvalue Analysis
Condition

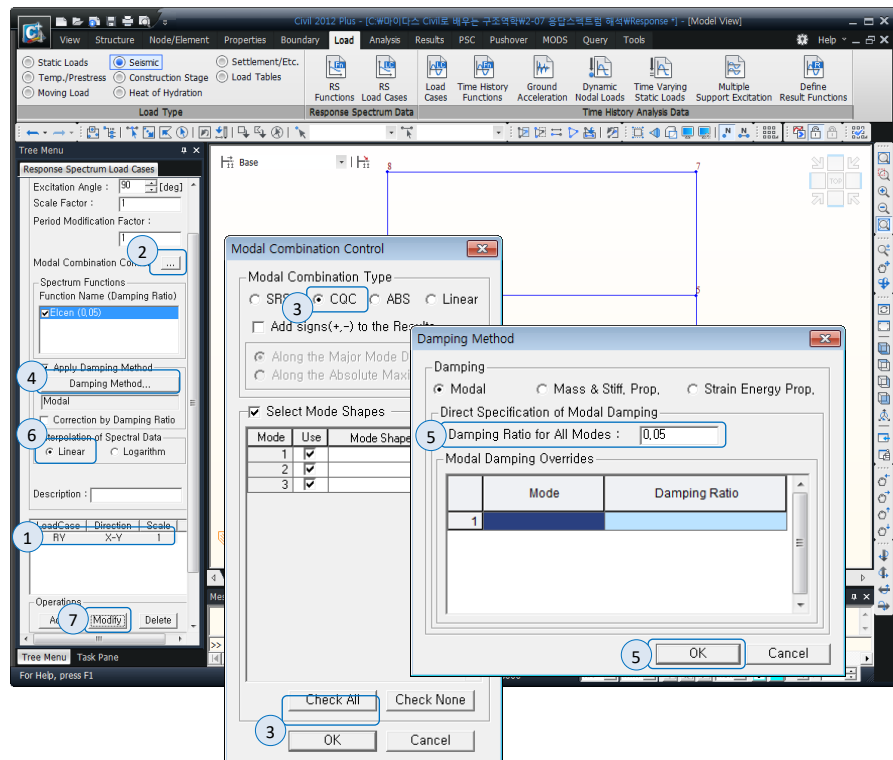


Determine modal combination type for each modes.

Main Menu > **Load** > **Seismic** > **RS (Response Spectrum) Load Cases**

1. Select Load Case 'RY'
2. Click [...] in Modal Combination Control
3. Select Modal Combination Type > **CQC**, Click **[OK]**
4. **Apply Damping Method** (on) > Click **[Damping Method...]**
5. Damping Ratio for All Modes : '**0.05**', Click **[OK]**
6. Select Interpolation of Spectral Data > **Linear**
7. Click **[Modify]**

► Fig 7.17
Response Spectrum
Analysis Condition



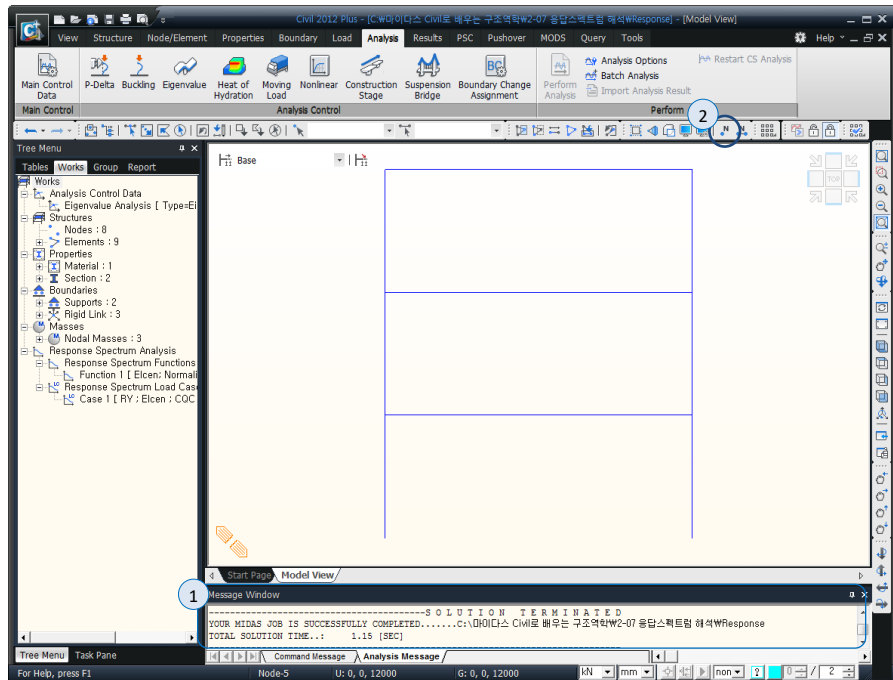
2.7 Perform Analysis

Analyze the response spectrum modeled.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window
2. Display Node Numbers (off)

► Fig 7.18
Message for a
successful run



2.8 Check Analysis Result

First, the natural frequency of the analytical model is confirmed from the eigenvalue analysis result.

Main Menu > **Results** > **Result Tables** > **Vibration Mode Shape**

1. Records Activation Dialog box, Click [OK]
2. Check period of each modes (0.6820, 0.2434, 0.1684)
3. Check modal mass participation rate
4. Check modal participation coefficient (TRAN-Y)

► Fig 7.19

Eigenvalue Analysis
Result Table



As a result of eigenvalue analysis, the mass participation rate should be more than 90% of structure.

Node	Mode	UX	UY	UZ	RX	RY	RZ					
EIGENVALUE ANALYSIS												
Mode No	Frequency			Period	Tolerance							
	(rad/sec)		(cycle/sec)	(sec)								
1	9.213226	1.466331	0.681974	3.3483e-016								
2	25.814902	4.108569	0.243394	5.1179e-016								
3	37.303643	5.937059	0.168434	3.2679e-016								
MODAL PARTICIPATION MASSES PRINTOUT												
Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)
1	0.00	0.00	91.41	91.41	0.00	0.00	99.69	99.69	0.00	0.00	0.00	0.00
2	0.00	0.00	7.49	98.90	0.00	0.00	0.03	99.72	0.00	0.00	0.00	0.00
3	0.00	0.00	1.10	100.00	0.00	0.00	0.28	100.00	0.00	0.00	0.00	0.00
Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM
1	0.00	0.00	0.19	0.19	0.00	0.00	32380410	32380410	0.00	0.00	0.00	0.00
2	0.00	0.00	0.02	0.21	0.00	0.00	9869.87	32390280	0.00	0.00	0.00	0.00
3	0.00	0.00	0.00	0.21	0.00	0.00	88719.61	32480000	0.00	0.00	0.00	0.00
MODAL PARTICIPATION FACTOR PRINTOUT (kN,mm)												
Mode No	TRAN-X Value		TRAN-Y Value		TRAN-Z Value		ROTN-X Value		ROTN-Y Value		ROTN-Z Value	
1	0.00		0.44		0.00		0.00		0.00		0.00	
2	0.00		0.13		0.00		0.00		0.00		0.00	
3	0.00		-0.05		0.00		0.00		0.00		0.00	
MODAL DIRECTION FACTOR PRINTOUT												
Mode No	TRAN-X Value		TRAN-Y Value		TRAN-Z Value		ROTN-X Value		ROTN-Y Value		ROTN-Z Value	
1	0.00		47.83		0.00		52.17		0.00		0.00	
2	0.00		99.60		0.00		0.40		0.00		0.00	
3	0.00		79.99		0.00		20.01		0.00		0.00	
EIGENVECTOR (kN,mm)												

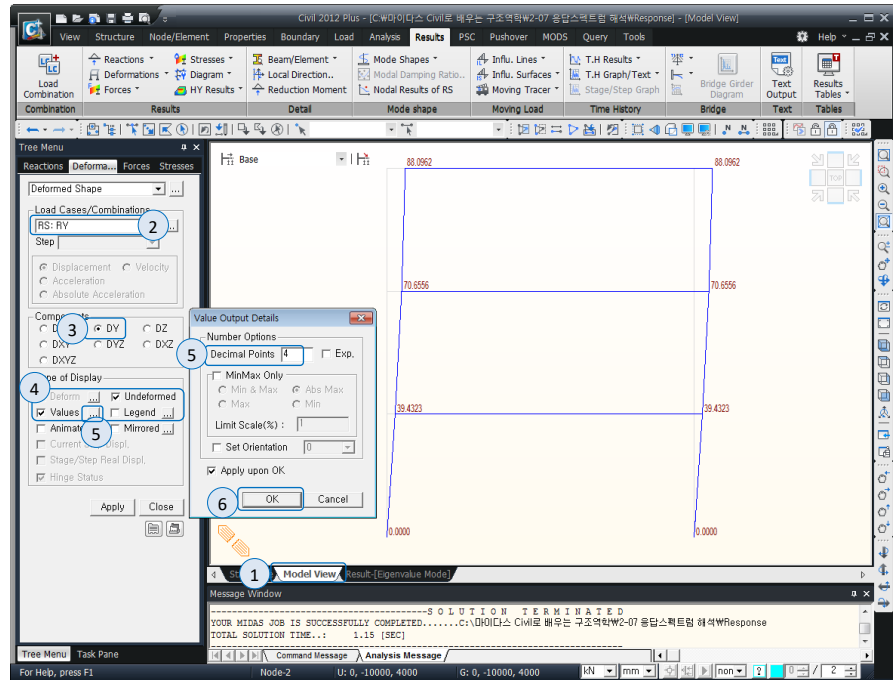
The sum of the modal participation masses was 100% upto the third mode, which reflected the dynamic characteristics of the structure.

Check the displacements in each floors.

Main Menu > **Results** > **Deformations** > **Deformed Shape...**

1. Click **Model View** Tab
2. Select Load Cases/Combinations > **RS : RY**
3. Select Components > **DY**
4. Type of Display > **Undeformed**(Check on), **Value**(Check on)
5. Click [...] in **Values**, Number Options > Decimal Points : **'4'**
6. Click **[OK]**

► Fig 7.20
Displacement results



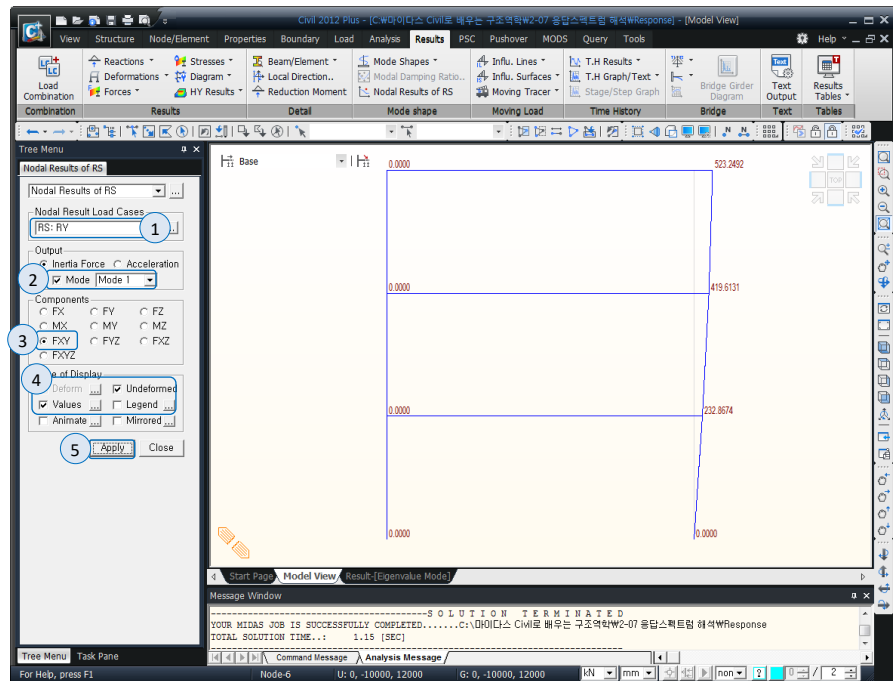
The displacements of the 2nd, 3rd and roof floor are 39.4323mm, 70.6556mm and 88.0962mm.

Check the floor shear force in the lateral direction generated by the response spectrum analysis.

Main Menu > **Results** > **Mode shape** > **Nodal Results of RS**

1. Select Nodal Result Load Cases > **RS:RY**
2. Select Output > Inertia Force and Mode > **Mode 1**
3. Select Components > **FX**
4. Type of Display > Undeformed (on), Value (on)
5. Click [Apply]

► Fig 7.21
Shear force results



The floor shear forces of the 2nd, 3rd and roof floor are 232.86747 kN, 419.6131 kN and 523.2492 kN.



8. Prestress Analysis

Contents

1 Introduction

1.1 Concept of Prestress Analysis	8-3
-----------------------------------	-----

2 Tutorial

2.1 Model Overview	8-6
2.2 Work Environment	8-7
2.3 Material & Section Properties	8-9
2.4 Generate Node & Element	8-10
2.5 Define Boundary Conditions	8-13
2.6 Define Loads	8-15
2.7 Perform Analysis	8-16
2.8 Check Analysis Result	8-18



1. Introduction

1.1 Concept of Prestress Analysis

Prestressing is a technique wherein the prestressing strands (tendon) are induced with tensile force which is transmitted as a compressive force to the concrete structure. The force transfer between concrete and steel can be through the bond strength mechanism or through anchorage plates held at the ends of prestressed concrete block or both. Based on whether tensioning is done prior to hardening of concrete or post-hardening, prestressing techniques can be categorized as pre-tensioning and post-tensioning techniques respectively.

The prestressing force reduces in intensity due to various factors and the losses in prestress are classified as immediate losses and time dependent losses as mentioned below:

I. Immediate Losses

- i. Elastic Shortening Loss: Due to axial shortening of concrete as per the prestress application, the cable length too shortens leading to loss of prestressing force. This occurs in both pre tensioning and post tensioning (when cables are stressed sequentially) of concrete.
- ii. Anchorage Slip Loss: In post tensioning technique, the prestressing force is transferred through the end anchorage plates held up against concrete block, wherein the end wedges slip over a distance before getting completely locked, thereby resulting in reduction in length of tendon and loss in prestress. This loss does not occur in pre-tensioning systems.
- iii. Friction loss: The prestress losses due to wobbling of tendons and friction between the duct and tendons during the stressing process are summed up under the head frictional loss. It is also prevailing in post tensioning system only.

II. Time Dependent Losses

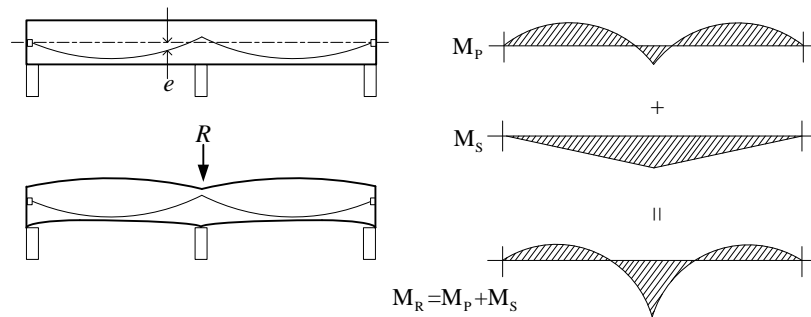
- i. Relaxation Loss: The loss depicts the loss in stress in prestressing strands with time under constant strain. The relaxation loss is dependent on the type of steel, initial prestress and temperature.
- ii. Creep Loss: As the deformation in concrete increases with time under sustained loading, this results in loss of prestressing force. The creep of concrete with time is quantified in terms of creep coefficient based on ratio of ultimate creep strain to elastic strain. The creep coefficient is further used to calculate prestress loss due to creep of concrete with time.
- iii. Shrinkage Loss: As concrete contracts due to loss of moisture with time, this results in loss of prestress. The shrinkage of concrete is also divided in two parts namely



initial shrinkage or autogenous shrinkage which occurs immediately after casting of concrete and drying shrinkage which occurs with the passage of time as water moves through the hardened concrete. Considering both autogenous and drying shrinkage, final shrinkage coefficient is derived which is used to calculate prestress loss due to shrinkage.

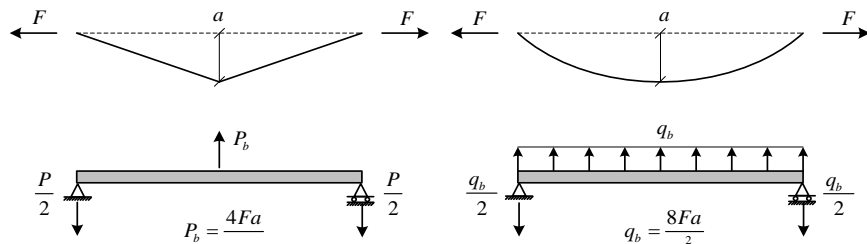
Let's take an example of prestressing force application on a continuous beam.

► Figure 8.1
Prestressing Effect



Let us consider the tendon profile as shown in the fig.8.2 for the continuous beam.

► Figure 8.2
Effect of Tendon Profile



Prestressing effect of a continuous beam can be explained with the help of deflection pattern of the beam. If we remove the intermediate support of the beam, the central portion will hog up due to the prestressing force. In practice, however because the support exists, the deflection of the central portion will not occur, but instead there will be development of a reaction force preventing the occurrence of deflection. The moment due to this reaction force is called secondary moment due to prestressing. Primary prestressing moment is just the



product of prestressing force and eccentricity of the tendon and C.G of the beam section. Hence the total moment generated due to prestressing is the sum of primary and secondary prestressing moments.

$$M_R = M_P + M_s$$

The secondary prestressing moment is null for determinate structures and thus in determinate structures only primary moment prevails.

The resultant moment is the sum of the moments due to the eccentric distance effect and the moments due to the reaction force of the support point. In the case of a determinate structure, since the support point does not constrain the deformation, M_s does not occur, so $M_R = M_P$. As a result, the magnitude of the second moment can be calculated by subtracting the first moment from the resultant moment ($M_s = M_R - M_P$).

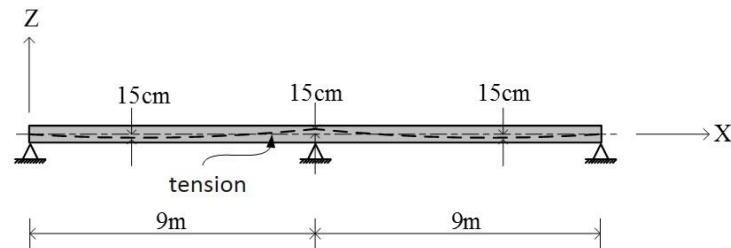


2. Tutorial

2.1 Model Overview

2-span continuous beam is subjected to the action of a tensile force. Determine the maximum bending moment with the stress stiffening effect.

► Fig. 8.3
Analytical Model




- **Material**
Modulus of elasticity: $30,000 \text{ N/mm}^2$
- **Section**
Square Section: $300 \times 550 \text{ mm}^2$
- **Load**
900 kN by prestress on parabolic tendon.

2.2 Work Environment

Open a new file and save.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : **'Prestress'**, Click **[SAVE]**

Set the unit system to use.

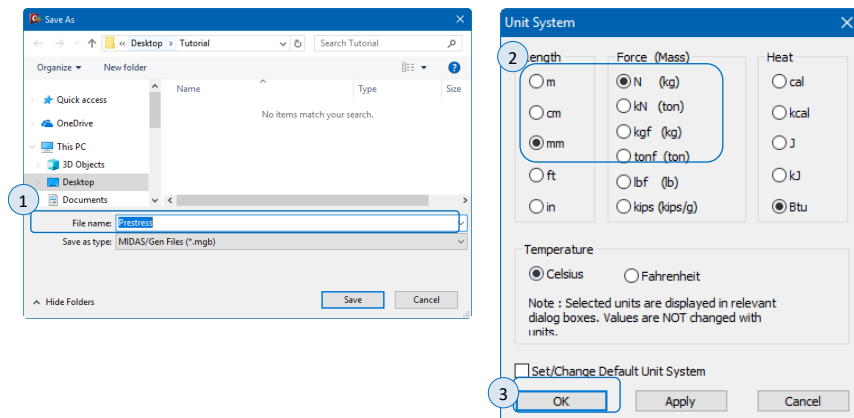
Main Menu > **Tools > Unit System...**

2. Select Length > **mm**, Force (Mass) > **N**

3. Click **[OK]**

► Fig 8.4

(a) Save the file
(b) Unit system setting

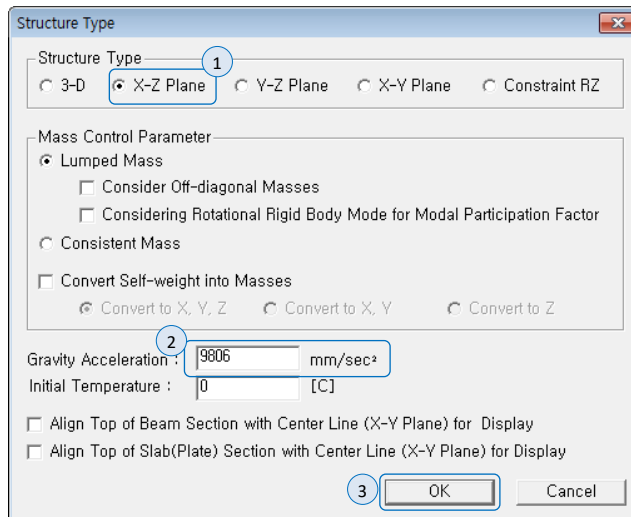


midas Gen is 3-D software, since beam exists in a 2-D plane, X-Z plane in Global Coordinate is set as the work plane, which restrains unnecessary degrees of freedom, Dy, Rx, Rz.

Main Menu > **Structure > Type > Structure Type**

1. Select Structure Type > **X-Z Plane**
2. Gravity Acceleration: **'9806'**
3. Click **[OK]**

► Fig 8.5
Set work plane



2.3 Material & Section Properties

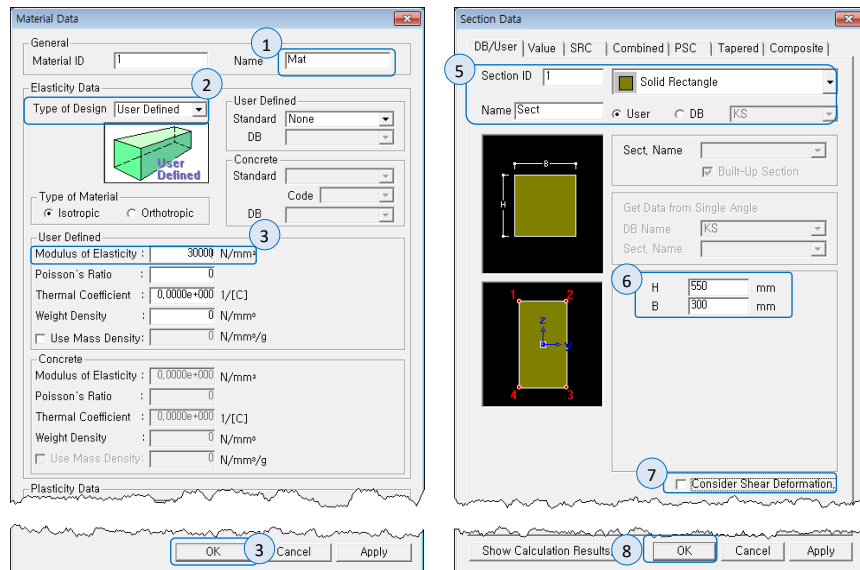
Define material and section for the structure.

Main Menu > **Properties** > **Material** > **Material Properties...**

1. Click **[Add...]**, Name : '**Mat**'
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : '**30,000**', Click **[OK]**
4. Click **Section Tab** and **[Add...]**
5. Select Section Shape Lists > **Solid Rectangle**, Name : '**Sect**', Select **User**
6. H : '**550**', B : '**300**'
7. **Consider Shear Deformation** (off)
8. Click **[OK]** and **[Close]**

► Fig 8.6

- (a) Material definition
(b) Section definition



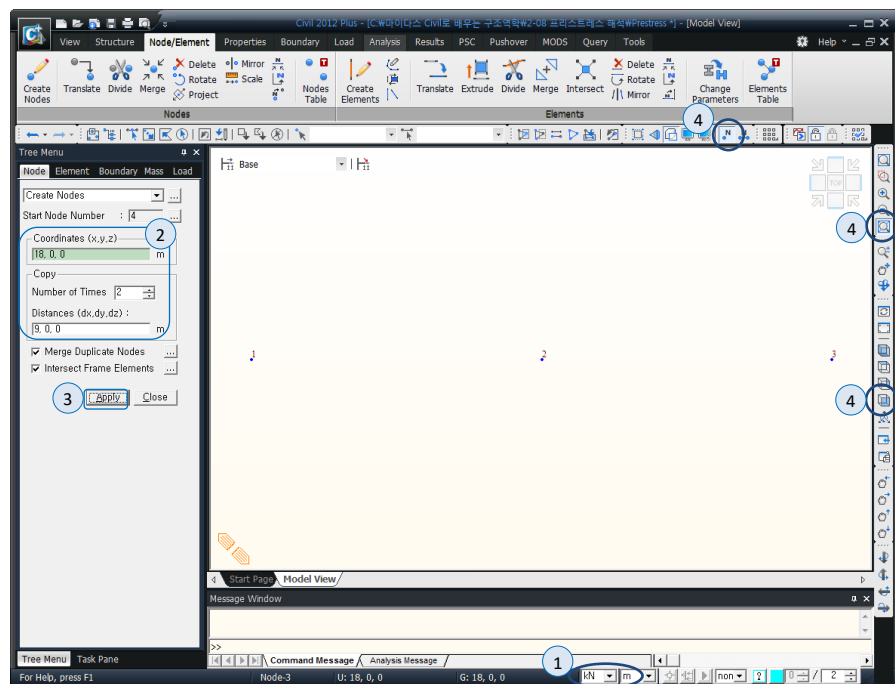
2.4 Generate Nodes & Elements

Create nodes where elements will be created.

Main Menu > **Node/Element** > **Nodes** > **Create Nodes**

1. Modify the unit at lower of the screen to kN, m
2. Coordinates (x, y, z) : '0, 0, 0'
- Copy > Number of Times: '2', Distances (dx, dy, dz) : '9,0,0'
3. Click [Apply]
4. Display Node Numbers, Auto Fitting, Front View (on)

► Fig 8.7
Create nodes

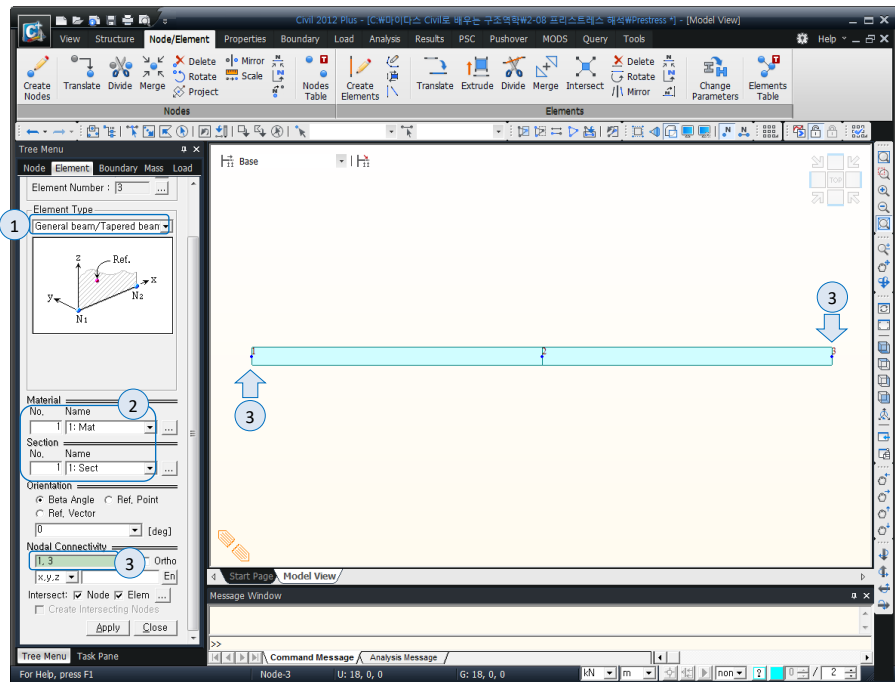


Connect the nodes to create the element.

Main Menu > **Model** > **Elements** > **Create Elements...**

1. Select Element Type > **General beam/Tapered beam**
2. Select Material > **1:Mat** and Section > **1:Sect**
3. Click Nodal Connectivity green box, and Click node number (1, 3) in Model view

► Fig 8.8
Create elements



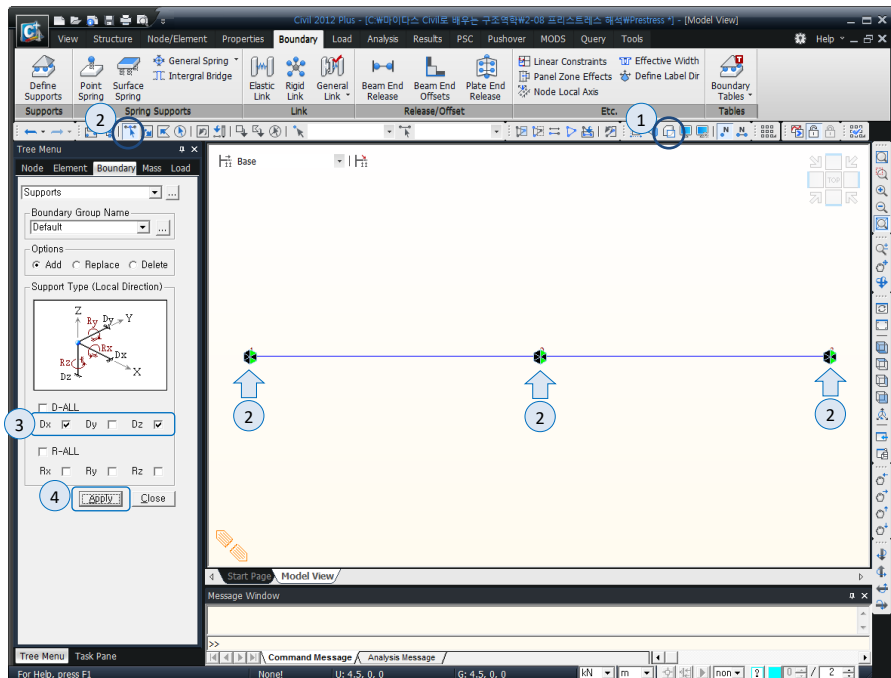
2.5 Define Boundary Conditions

Define support conditions, pin support at each nodes.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. Hidden (off)
2. Click **Select Single** (on), Select node number 1 and 2 and 3
3. Support Type > **Dx, Dz** (on)
4. Click **[Apply]**

► Fig 8.9
Define support condition





2.6 Define Loads

Define a load case to input the prestress.

Main Menu > **Load** > **Static Loads** > **Static Load Cases**

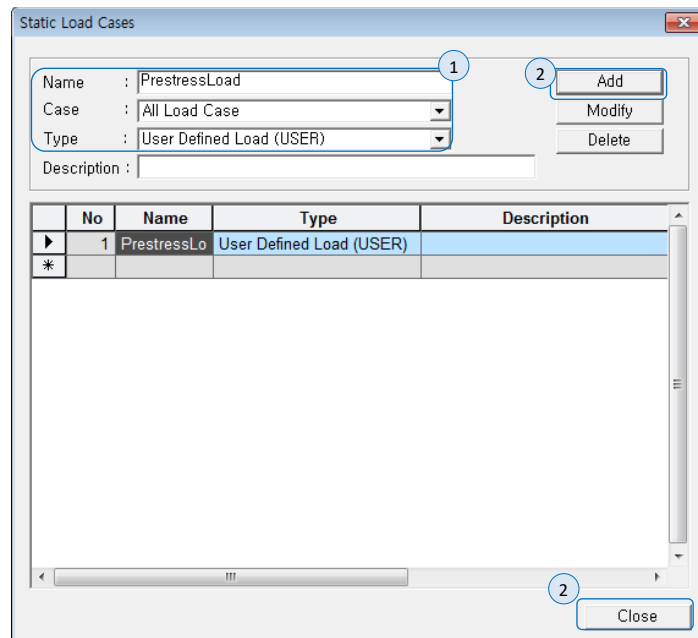
1. Name : '**PrestressLoad**'

Select Type > **User Defined Load (USER)**

2. Click **[Add]** and **[Close]**

► Fig 8.10

Definition of load condition



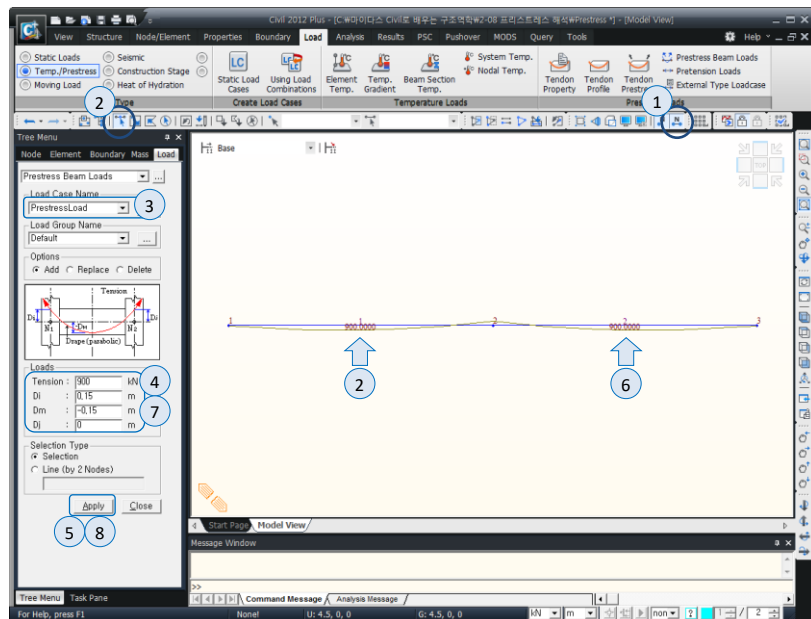
Input the prestress load and eccentric distance considering tendon arrangement

Main Menu > **Load > Temp./Prestress > Prestress Beam Loads**

View / Display, Load Tab > Load Case > Click [...] in Load Value, **Place : '4'**, Click **[OK]**

1. **Display Element Numbers (on)**
2. Click **Select Single (on)**,
3. Load Case Name > PrestressLoad
4. Loads > Tension : '**900**', Di : '**0**', Dm : '**-0.15**' Dj : '**0.15**'
5. Click **[Apply]**
6. Click **Select Single (on)**
7. Loads > Tension : '**900**', Di : '**0.15**', Dm : '**-0.15**' Dj : '**0**'
8. Click **[Apply]**

► Fig 8.11
Prestress load input



2.7 Perform Analysis

Analyze the model.

Main Menu > **Analysis > Perform Analysis**

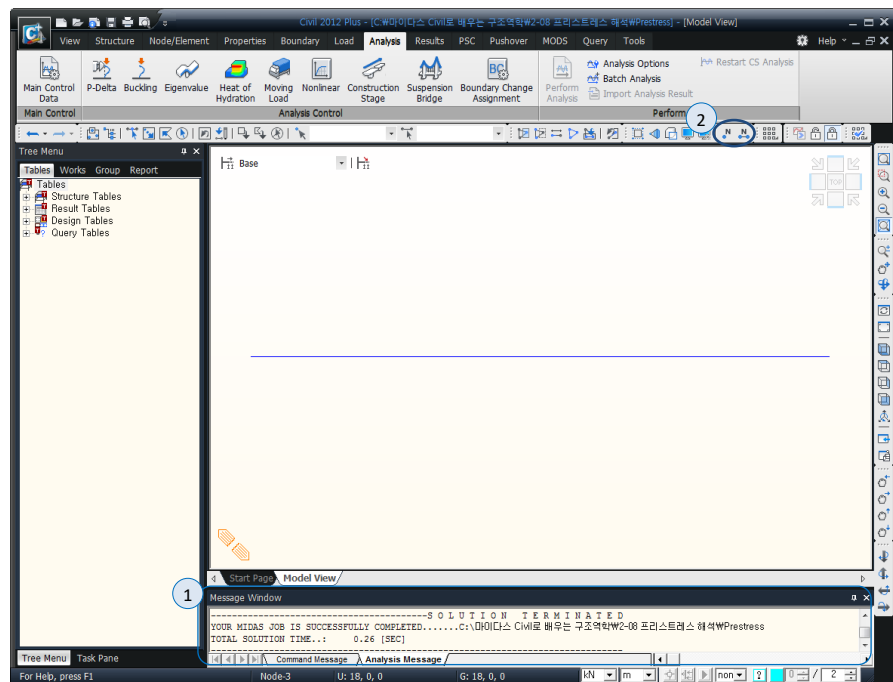
1. Check for successful completion in Message Window

Main Menu > **View > Display...**

2. Load Tab > Nodal Load (off), Click [OK]

3. **Display Node and Element Numbers** (off)

► Fig 8.12
Message for a
successful run



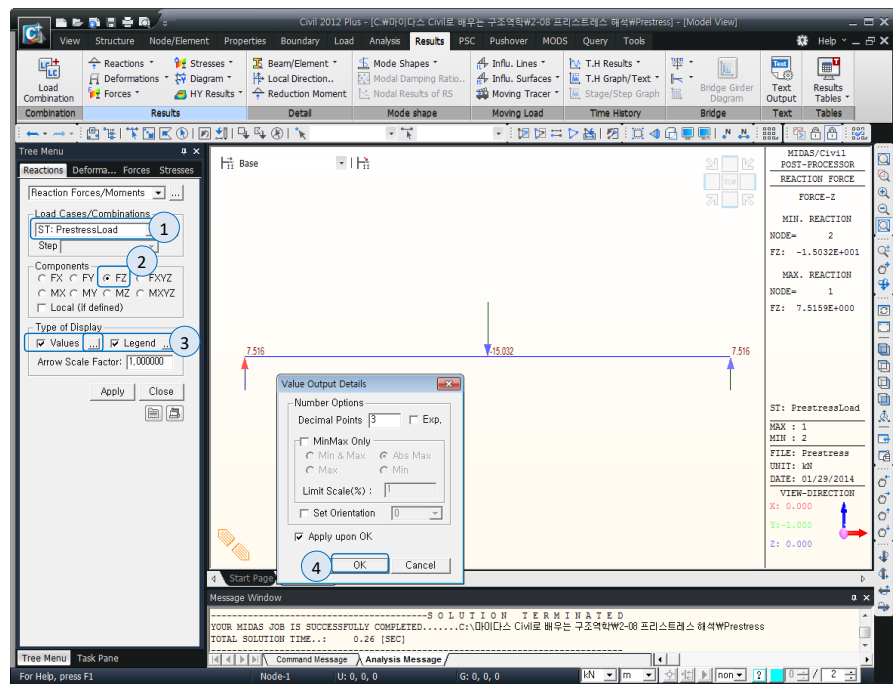
2.8 Check Analysis Result

Check the reaction force at each support nodes.

Main Menu > **Results** > **Reactions** > **Reaction Forces/Moments...**

1. Select Load Cases/Combinations > **ST : PrestressLoad**
2. Select Components > **FZ**
3. Type of Display > **Values, Legend** (on)
Click [...] in Value, Number Option > Decimal Point : '3'
4. Click **[OK]**

► Fig 8.13
Reaction force

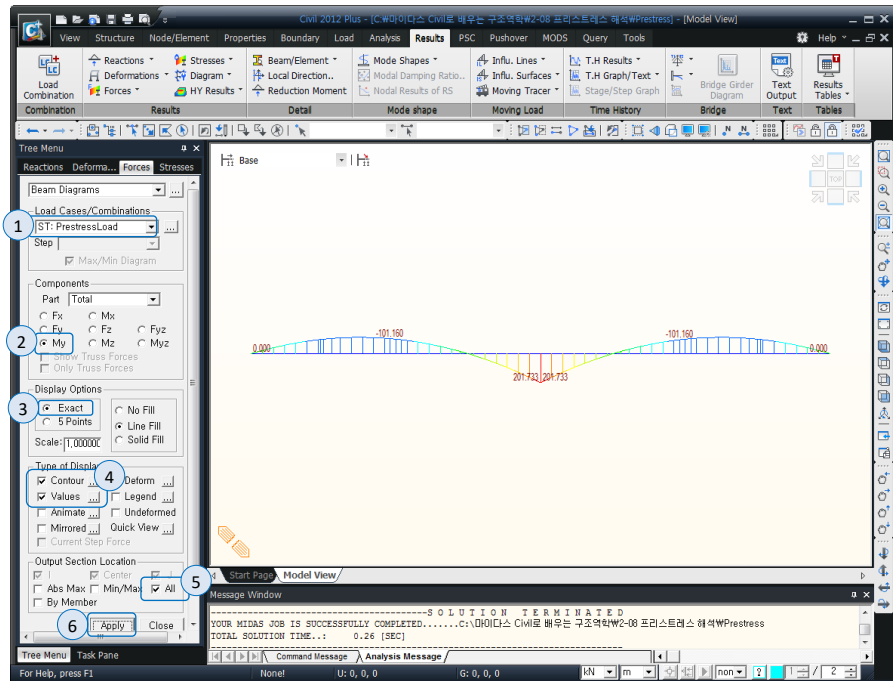


Check the bending moment.

Main Menu > **Results** > **Forces** > **Beam Diagrams...**

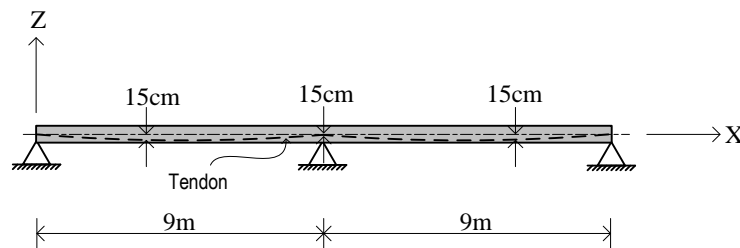
1. Select Load Cases/Combinations > **ST : PrestressLoad**
2. Select Components > **My**
3. Select Display Options > **Exact**
4. Type of Display > **Contour, Value** (on)
5. Output Section Location > **All** (on)
6. Click **[Apply]**

► Fig 8.14
Bending moment



3. Exercise

Check bending moment of two span continuous beams with tendon prestressing.



➤ **Material**

Modulus of elasticity: $30,000 \text{ N/mm}^2$

➤ **Section**

Square section : 300×550

➤ **Load**

Load of 900 kN by prestress on parabolic tensions
Equilibrium load of 20 kN/m across the whole area



9. Thermal Stress Analysis

Contents

1 Introduction

1.1 Concept of Thermal Stress Analysis 9-3

2 Tutorial

2.1 Model Overview	9-8
2.2 Work Environment	9-9
2.3 Material & Section Properties	9-11
2.4 Generate Node & Element	9-12
2.5 Define Boundary Conditions	9-14
2.6 Define Loads	9-15
2.7 Perform Analysis	9-17
2.8 Check Analysis Result	9-19



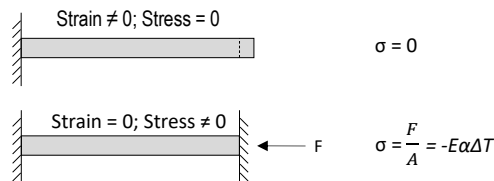
1. Introduction

1.1 Concept of Thermal Stress Analysis

In general, when objects are exposed to temperature variation, they either elongate or shrink depending on temperature rise or fall. At higher temperature the molecules within the structure start to vibrate rapidly and push away from one another. This increased separation causes the solid to expand, increasing its volume. The volumetric expansion could give rise to greater levels of thermal stress. Thermal stresses can have significant effects on a structure's strength, stability, inducing cracks and breaking of components. It is necessary that the overall design of the structure is accounted for thermal stresses as well, else it might lead to unexpected weakening and deformation of the structure.

But these stresses that build up due to temperature variation, get released, when the objects are free to deform. For instance, the expansion joints that are often implemented into the design of buildings, bridges and railways help to release the internal stresses caused by the increase in temperature. However, if the object is constrained and not free to deform, stress starts building up. To determine the effects of these built-up stresses on structures, Thermal Stress Analysis is performed. The thermal stress that occurs on an object depends on its unique material property called the thermal expansion coefficient (α).

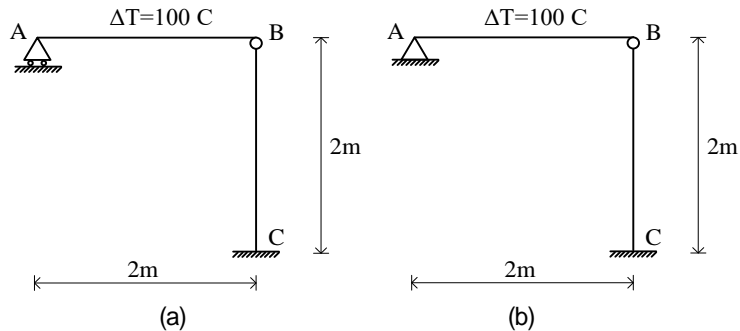
► Fig 9.1
Thermal Stress



Let us consider a worked out example. The two structures (a) and (b)

$$E = 2 \times 10^4 \text{ N/mm}^2, \quad I = 1 \times 10^{10} \text{ mm}^4, \quad \sigma = \text{Thermal expansion coefficient} = 1 \times 10^{-5} / ^\circ\text{C}$$

► Fig 9.2
Example of thermal
change

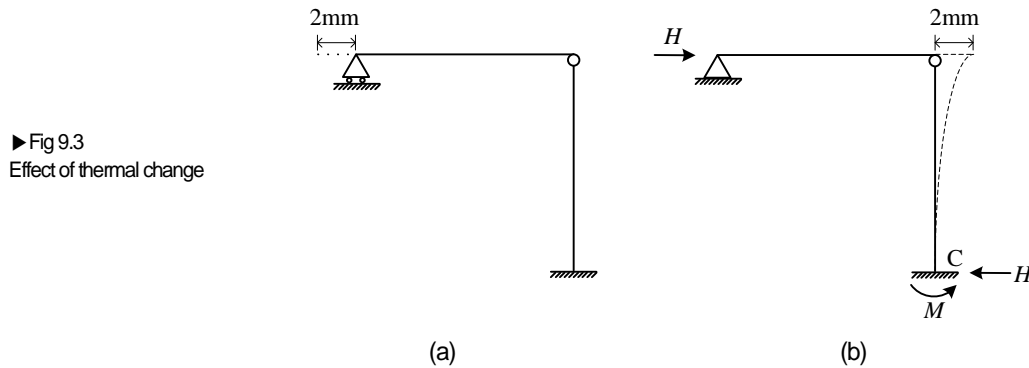


Let us just consider Temperature change on element AB. Due to this Temperature change, deformation Δ on the element AB in both the cases (a) , (b) would be same i.e.,

$$\Delta = 1 \times 10^{-5} / ^\circ\text{C} \times 100^\circ\text{C} \times 2,000\text{mm} = 2\text{mm}$$

From Figure 9.2 (a) the element AB at Point A is free for transverse displacement, and hence expands to left by 2 mm. On the other hand from Figure 9.2(b), the element at Point A is restrained from transverse displacement. Hence it expand to the right by 2mm at point B. Therefore, this would induce a Horizontal reaction force at point C i.e., at the bottom of the element BC, which would be equal and opposite of the Horizontal reaction at point A.

Figure 9.3 (a) & (b) depicts the deformation in the structure after the increase in temperature.



Considering the force and moment equilibrium, the reaction forces are computed ature change considered.

$$M = \frac{3EI}{l^2} \Delta = \frac{3 \times 2 \times 10^4 \times 1 \times 10^{10}}{2000^2} \times 2 \times 10^{-6} = 300 \text{ kN} \cdot \text{m}$$

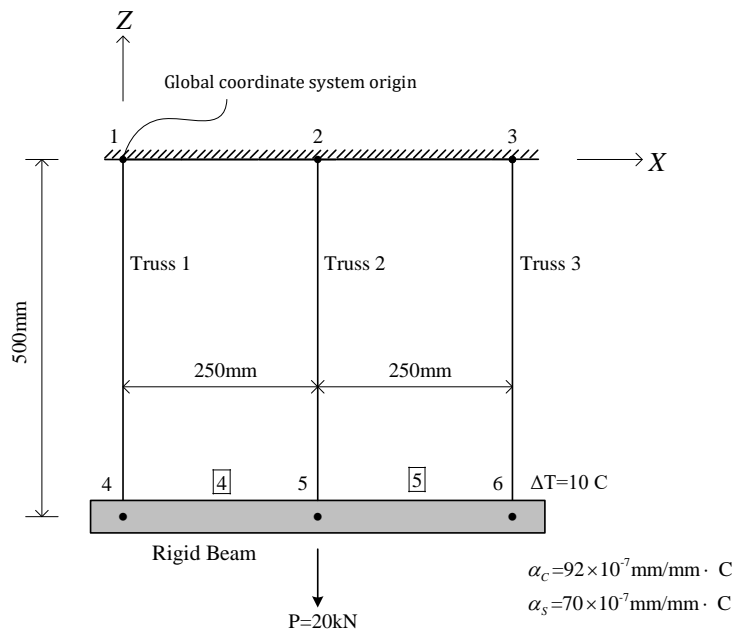
$$H = \frac{300}{2} = 150 \text{ kN}$$

2. Tutorial

2.1 Model Overview

Examine the resultant stress of a structure comprised of truss members. The structure is subjected to a concentrated load and a uniform temperature change over the entire structures.

► Fig 9.4
Structural geometry &
analysis model



- **Model 1**
A model with rigid beam element hanging on truss
- **Model 2**
A model with a rigid link element hanging on truss



➤ **Material**

Truss 1, Truss 3

Modulus of elasticity (E_c): 1.1×10^5 N/mm

Coefficient of thermal expansion (α_c): 9.2×10^{-6} mm/mm·°C

Truss 2

Modulus of elasticity (E_s): 2.05×10^5 N/mm²

Coefficient of thermal expansion (α_s): 7.0×10^{-6} mm/mm·°C

Rigid Beam

Modulus of elasticity (E_B) : 7.0×10^{12} N/mm²

➤ **Section**

Truss element (vertical) area A : 65mm²

Beam element (rigid body) I_{yy} : 400,000mm⁴

➤ **Load**

A concentrated load 20kN is applied to the node 5 in the $-Z$ direction.

Temperature change over the entire structure

Initial temperature: 15 °C

Final temperature: 25 °C

2.2 Work Environment

Open a new file and save.

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : **'Thermal 1'**, Click **[SAVE]**

Set the unit system to.

Main Menu > **Tools > Unit System...**

2. Select Length > **mm**, Force (Mass) > **N**

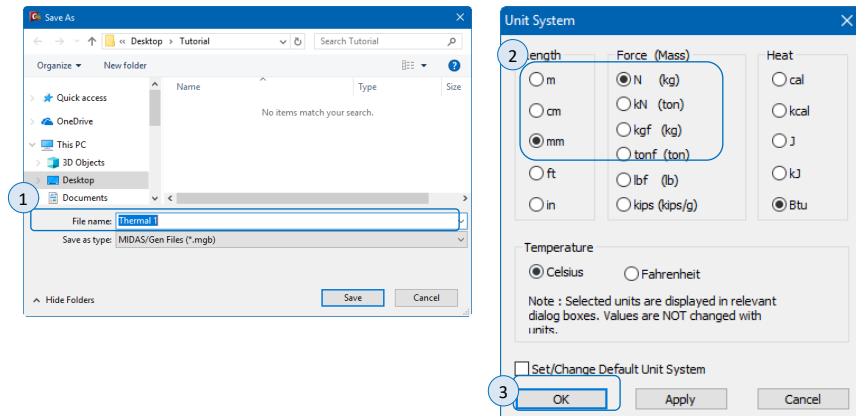
3. Click **[OK]**

► Fig 9.5

(a) Save the file
(b) Set unit system



The unit system setting
can be easily set at the
status bar at the bottom
of the screen.



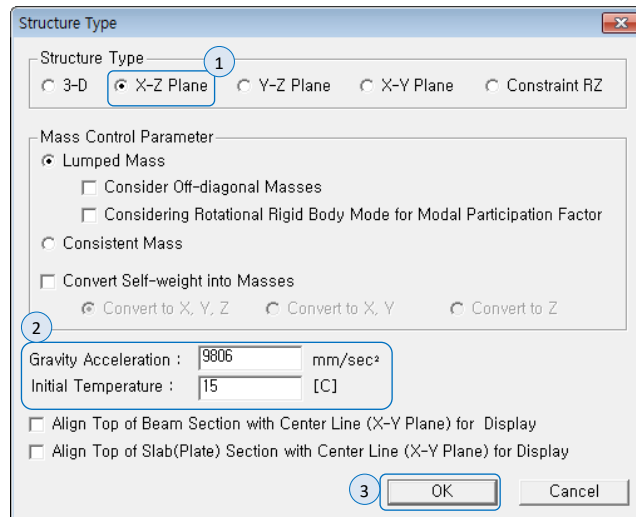


midas Gen is 3-D software, since beam exists in a 2-D plane, X-Z plane in Global Coordinate is set as the work plane, which restrains unnecessary degrees of freedom, Dy, Rx, Rz.

Main Menu > **Structure** > **Type** > **Structure Type...**

1. Select Structure Type > **X-Z Plane**
2. Gravity Acceleration: '**9806**', Initial Temperature: '**15**'
3. Click [**OK**]

► Fig 9.6
Set work plane





2.3 Material & Section Properties

Define material and section for the structure.

Main Menu > **Properties** > **Material** > **Material Properties**

1. Click **[Add...]**, Name : **'Truss 2'**
2. Select Elasticity Data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity : **'2.05e5'**
Thermal Coefficient : **'7.0e-6'**, Click **[Apply]**
4. Material ID : **'2'**, Name : **'Truss 1,3'**
5. Select Elasticity Data > Type of Design > **User Defined**,
6. User Defined > Modulus of Elasticity : **'1.1e5'**
Thermal Coefficient : **'9.2e-6'**, Click **[Apply]**
7. Material ID : **'3'**, Name : **'Rigid Beam'**
8. Select Elasticity Data > Type of Design > **User Defined**
9. User Defined > Modulus of Elasticity : **'7.0e12'**
Thermal Coefficient : **'0'**
10. Click **[OK]**
11. Click **Section** Tab and **[Add...]** and **Value** Tab
12. Select Section Shape Lists > **Box**, Name : **'Truss'**
13. Size > H : **'15'**, B : **'15'**, tw : **'2.5'**, tf1 : **'2.5'**, Section Properties > Area : **'65'**
14. **Consider Shear Deformation** (off), Click **[Apply]**
15. ID : **'2'**, Name : **'Beam'**
16. Size > H : **'25'**, B : **'25'**, tw : **'2.5'**, tf1 : **'2.5'**, Section Properties > I_{yy} : **'400,000'**
17. Click **[OK]** and **[Close]**

► Fig 9.7
(a), (b), (c) Material
definition
(d), (e) section definition

Material Data (1)

General: Material ID [1] Name [Truss 2]

Elasticity Data: Type of Design [User Defined]

Type of Material: ☒ Isotropic ☐ Orthotropic

User Defined:

- Modulus of Elasticity: 2.05e5
- Poisson's Ratio: 0
- Thermal Coefficient: 7.0e-6
- Weight Density: 0
- ☐ Use Mass Density: 0

Concrete:

- Modulus of Elasticity: 0.0000e+000
- Poisson's Ratio: 0
- Thermal Coefficient: 0.0000e+000
- Weight Density: 0
- ☐ Use Mass Density: 0

Plasticity Data: Plastic Material Name [NONE]

Thermal Transfer:

- Specific Heat: 0
- Heat Conduction: 0
- Damping Ratio: 0

Material Data (2)

General: Material ID [2] Name [Truss 1.3]

Elasticity Data: Type of Design [User Defined]

Type of Material: ☒ Isotropic ☐ Orthotropic

User Defined:

- Modulus of Elasticity: 1.1e5
- Poisson's Ratio: 0
- Thermal Coefficient: 9.2e-6
- Weight Density: 0
- ☐ Use Mass Density: 0

Concrete:

- Modulus of Elasticity: 0.0000e+000
- Poisson's Ratio: 0
- Thermal Coefficient: 0.0000e+000
- Weight Density: 0
- ☐ Use Mass Density: 0

Plasticity Data: Plastic Material Name [NONE]

Thermal Transfer:

- Specific Heat: 0
- Heat Conduction: 0
- Damping Ratio: 0

Material Data (3)

General: Material ID [3] Name [Rigid Beam]

Elasticity Data: Type of Design [User Defined]

Type of Material: ☒ Isotropic ☐ Orthotropic

User Defined:

- Modulus of Elasticity: 7.0e12
- Poisson's Ratio: 0
- Thermal Coefficient: 0
- Weight Density: 0
- ☐ Use Mass Density: 0

Concrete:

- Modulus of Elasticity: 0.0000e+000
- Poisson's Ratio: 0
- Thermal Coefficient: 0.0000e+000
- Weight Density: 0
- ☐ Use Mass Density: 0

Plasticity Data: Plastic Material Name [NONE]

Thermal Transfer:

- Specific Heat: 0
- Heat Conduction: 0
- Damping Ratio: 0

Section Data (4)

DB/User Value SRC Combined PSC Tapered Composite

Section ID [1] ☒ Box

Name [Truss] ☒ Built-Up Section

Diagram:

Size:

H	15.0000	mm
B	15.0000	mm
tw	2.5000	mm
tf1	2.5000	mm
C	0.0000	mm
tf2	0.0000	mm

Section Properties:

Calc. Section Properties

Area	6.50000e+001	mm²
Asy	0.00000e+000	mm²
Asz	0.00000e+000	mm²
Iyy	0.00000e+000	mm⁴
Izz	0.00000e+000	mm⁴
Cyp	0.0000	mm
Cym	0.0000	mm
Czp	0.0000	mm

☐ Consider Shear Deformation.

Offset: Center-Center
Change Offset...

Show Calculation Results... OK Cancel Apply

Section Data (5)

DB/User Value SRC Combined PSC Tapered Composite

Section ID [2] ☒ Box

Name [Beam] ☒ Built-Up Section

Diagram:

Size:

H	25.0000	mm
B	25.0000	mm
tw	2.5000	mm
tf1	2.5000	mm
C	0.0000	mm
tf2	0.0000	mm

Section Properties:

Calc. Section Properties

Area	6.50000e+001	mm²
Asy	0.00000e+000	mm²
Asz	0.00000e+000	mm²
Iyy	4.00000e+005	mm⁴
Izz	0.00000e+000	mm⁴
Cyp	0.0000	mm
Cym	0.0000	mm
Czp	0.0000	mm

☐ Consider Shear Deformation.

Offset: Center-Center
Change Offset...

Show Calculation Results... OK Cancel Apply

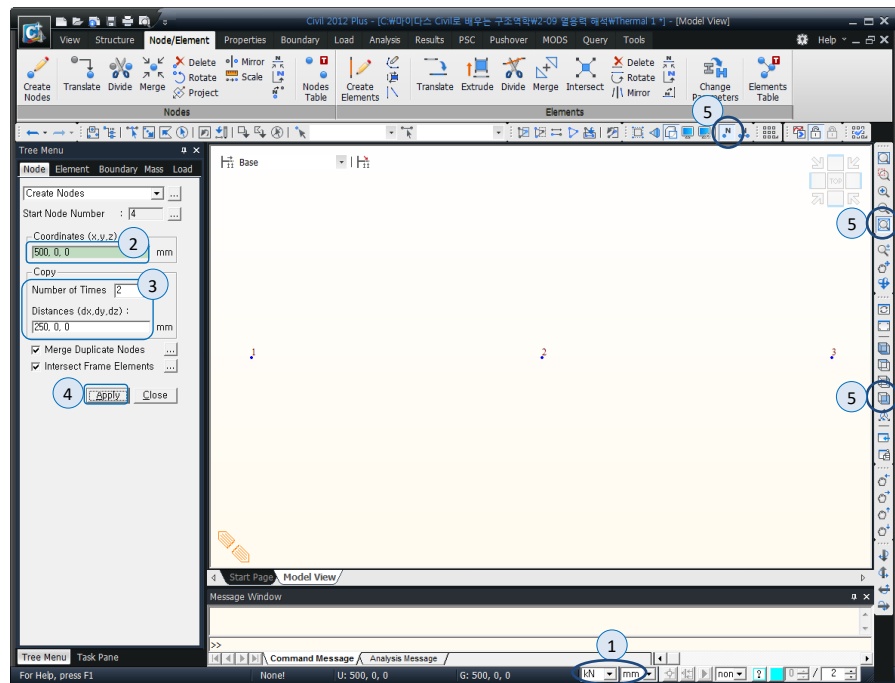
2.4 Generate Nodes & Elements

Creates nodes at the support.

Main Menu > **Node/Element** > **Nodes** > **Create Nodes**

1. Modify unit to kN, mn
2. Coordinates (x, y, z) : '0, 0, 0'
3. Copy > Number of Times: '2', Distances (dx, dy, dz) : '250,0,0'
4. Click [Apply]
5. Display Node Numbers, Auto Fitting, Front View (on)

► Fig 9.8
Create nodes



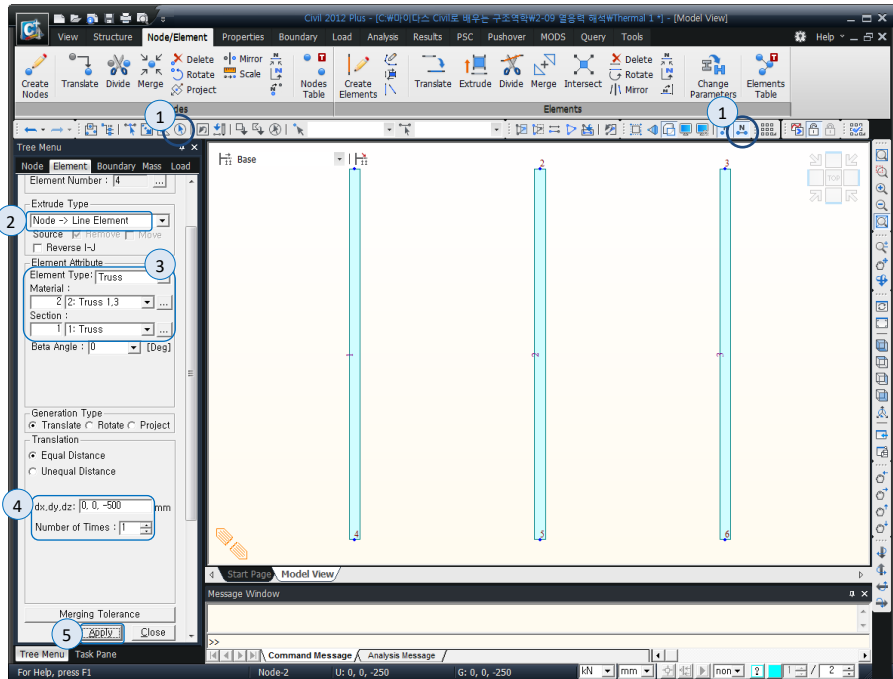
Create truss elements by extruding node to line element using Extrude Element.

Main Menu > **Node/Element** > **Elements** > **Extrude**

1. **Display Element Numbers** (on), Click **Select All**
2. Select Extrude Type > **Node** → **Line Element**
3. Select Element Attribute > Element Type > **Truss**
Select Material > **2: Truss 1,3**, Section > **1: Truss**
4. dx,dy,dz : '**0,0,-500**', Number of Times : '**1**'
5. Click **[Apply]**

► Fig 9.9

Create Truss Element

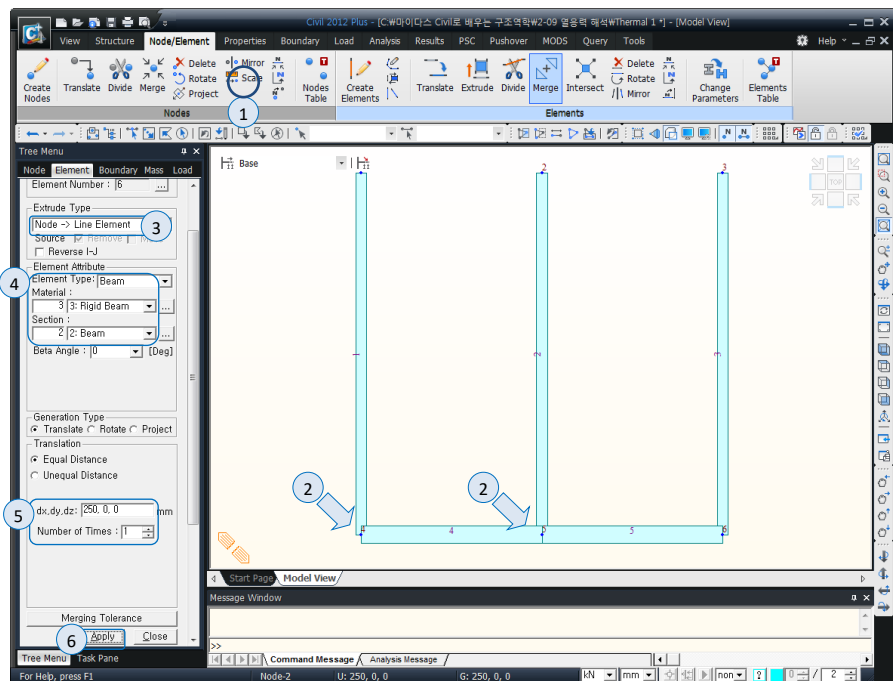


Create the beam element between node 4 and node 6 using Extrude Element.

Main Menu > **Node/Element** > **Elements** > **Extrude**

1. Click **Select Single** (on)
2. Select node number **4** and **5**
3. Select Extrude Type > **Node** → **Line Element**
4. Select Element Attribute > Element Type > **Beam**
Select Material > 3: **Rigid Beam**, Section > 2: **Beam**
5. Translation > dx,dy,dz : '**250,0,0**', Number of Times : '**1**'
6. Click **[Apply]**

► Fig 9.10
Create beam element



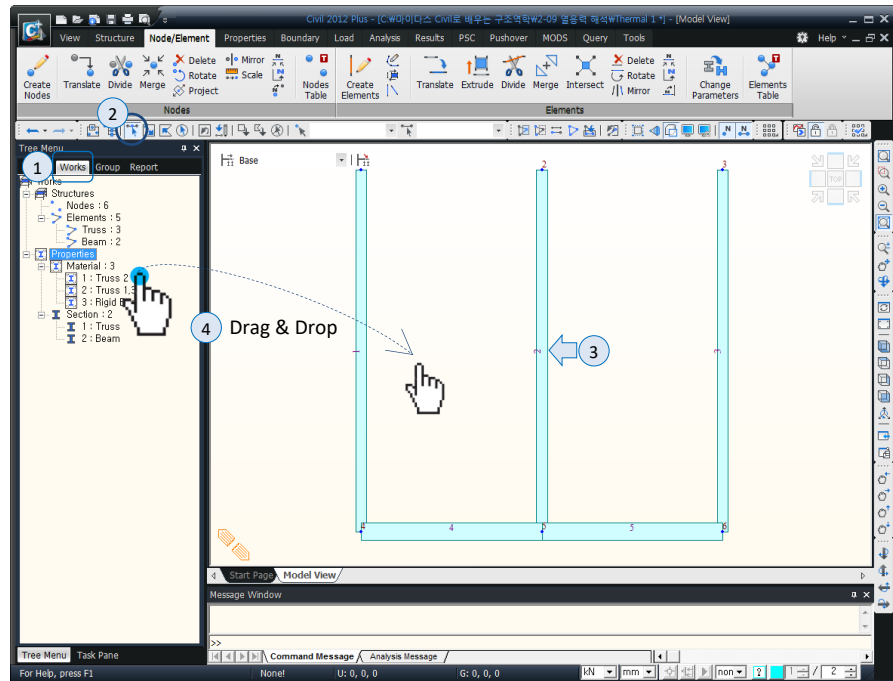
Modify the material property of element 2 using Drag & Drop.

Extrude Element Dialogue box, Click **[Close]**,

1. Select Tree Menu > **Works Tab**
2. **Select Single** (on)
3. Selection of center No. 2
4. Works > Properties > Material > **1: Truss 2** (Drag & Drop)

► Fig 9.11
Modify material

Tip
Drag & Drop: Drag from the left-click state of the mouse and drop it into the Model View



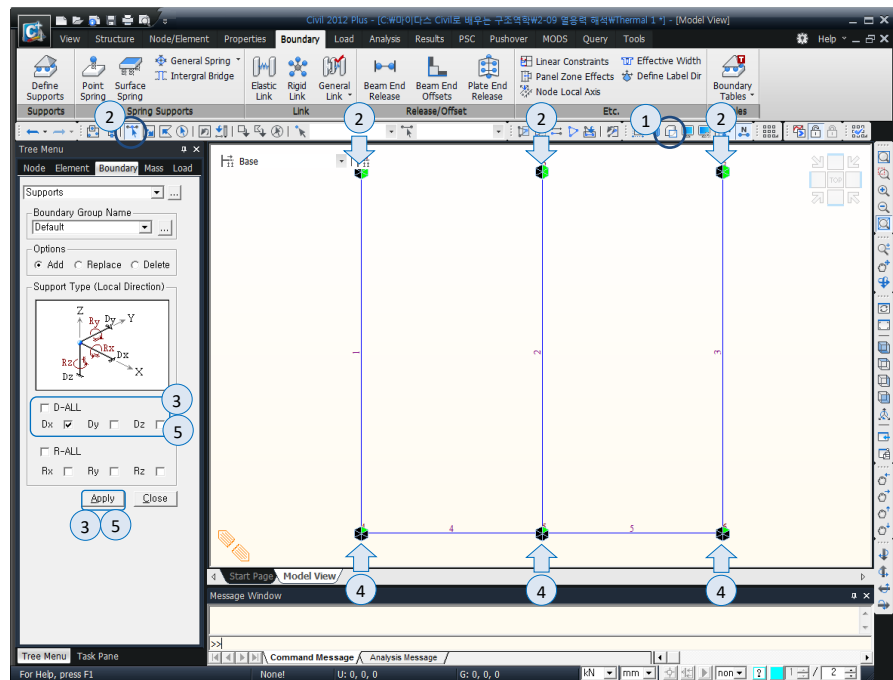
2.5 Define Boundary Conditions

Define fixed condition at upper support and lower node is assigned Dx degree of freedom restrained.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. **Hidden** (off)
2. Click **Select Single** (on), Select node number 1 and 2 and 3
3. Support Type > **D-ALL** (on), Click **[Apply]**
4. Click **Select Single** (on), Select node number 4 and 5 and 6
5. Support Type > **Dx** (on), **Dy**, **Dz** (off), Click **[Apply]**

► Fig 9.12
Define support condition





2.6 Define Loads

Define load conditions for nodal node and temperature loads.

Main Menu > **Load** > **Static Loads** > **Static Load Cases**

1. Name : '**Nodal**'

Select Type > **User Defined Load (USER)**, Click **[Add]**

2. Name : '**Temp**'

Select Type > **User Defined Load (USER)**, Click **[Add]**

3. Click **[Close]**

► Fig 9.13
Define load condition

Static Load Cases

1 Name : Temp

2 Case : All Load Case

Type : User Defined Load (USER)

Description :

1 Add

2 Modify

Delete

No	Name	Type	Description
1	Nodal	User Defined Load (USER)	
2	Temp	User Defined Load (USER)	
*			

3 Close

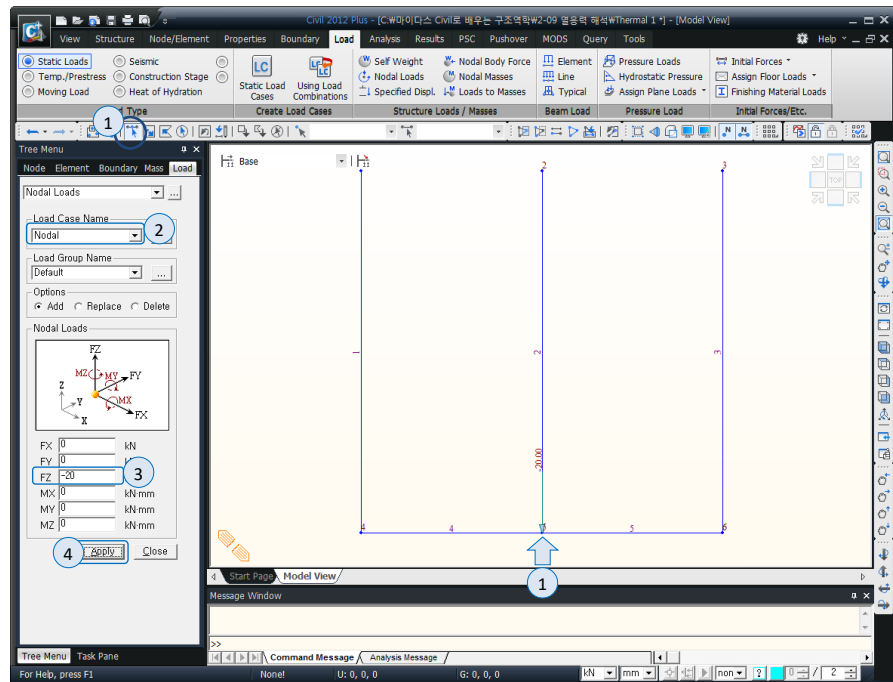
Input a concentrated load 20 kN on node 5.

Main Menu > **Load** > **Static Loads** > **Nodal Loads**

View / Display... > Load Tab > **Load Value** (on) > [...] > Place : '2', Click [OK]

1. Click **Select Single** (on), Select node number 5
2. Select Load Case Name > **Nodal**
3. Nodal Loads > FZ : '-20'
4. Click [Apply]

► Fig 9.14
Input nodal load

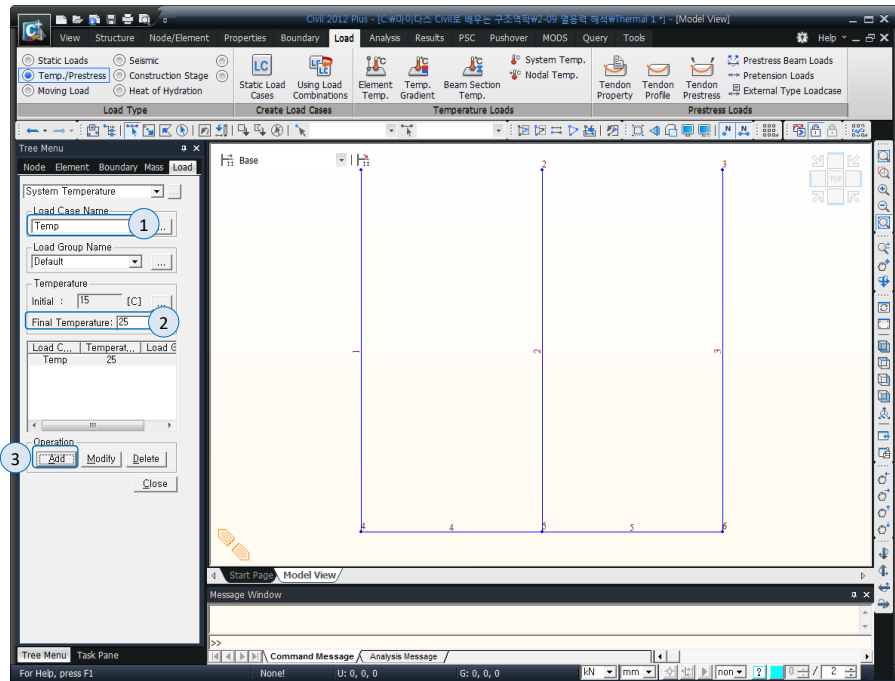


Input 25°C for entire structure temperature in order to reflect a load by 10°C temperature difference (25-15 = 10 °C).

Main Menu > **Load > Temp./Prestress > System Temperature**

1. Select Load Case Name > **Temp**
2. Temperature > Final Temperature : **'25'**
3. Click **[Add]**

► Fig 9.15
Input temperature load



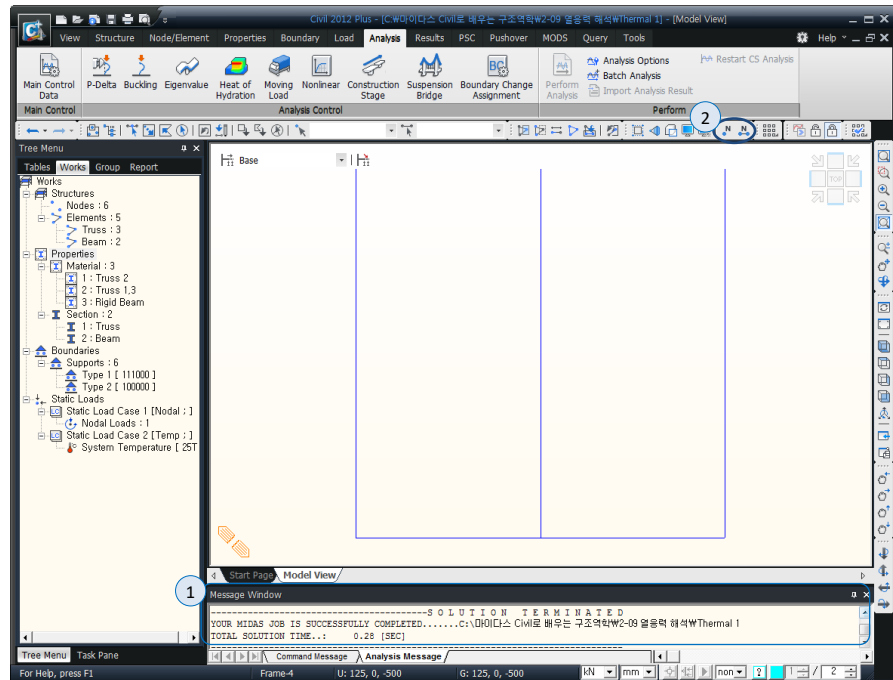
2.7 Perform Analysis

Analyze the model.

Main Menu > **Analysis > Perform Analysis**

1. Check for successful completion in Message Window
2. **Display Node Numbers, Display Element Numbers** (off)

► Fig 9.16
Message for a
successful run



2.8 Check Analysis Result

Create a load combination for checking result combined nodal load & temperature load.
Load combination condition 1 (LCB 1): 1.0 Nodal + 1.0 Temp

Main Menu > **Results** > **Combination** > **Load Combination**

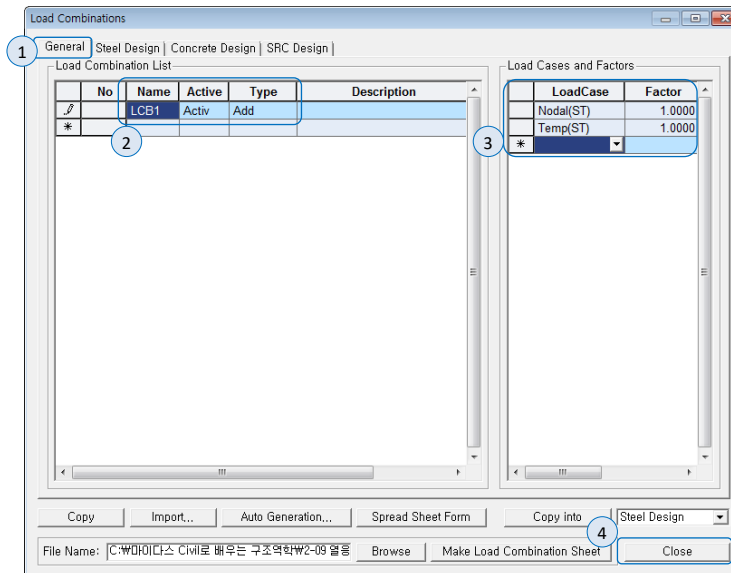
1. Confirm **General Tab**

2. Load Combination List > Name : **LCB1**

3. Load Cases and Factors > LoadCase : **Nodal(ST)**, Factor : '1'
LoadCase : **Temp(ST)**, Factor : '1'

4. Click [**Close**]

► Fig 9.17
Create load combination

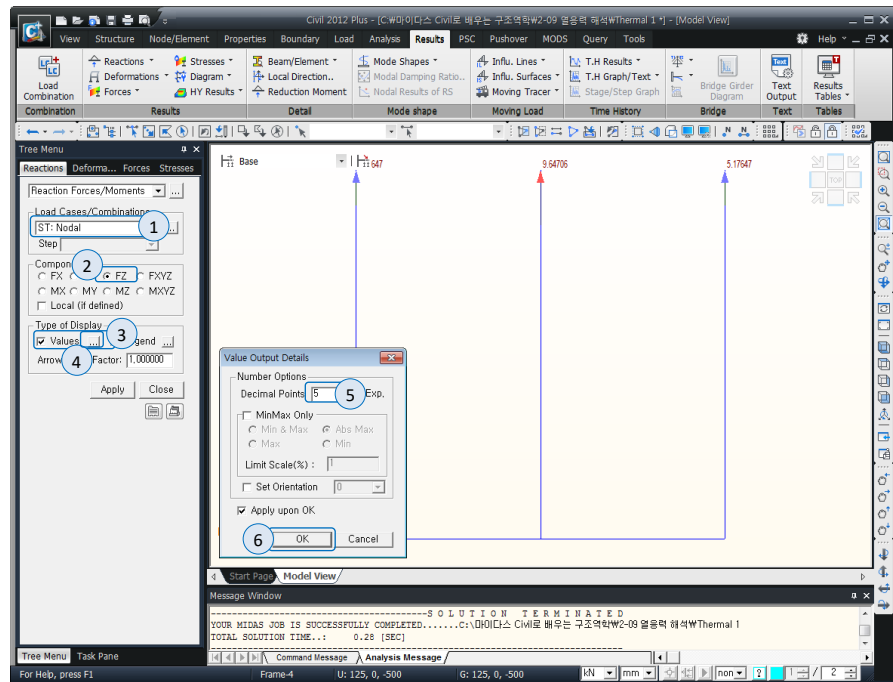


Check the reaction force on upper support.

Main Menu > **Results** > **Reaction** > **Reaction Forces/Moments...**

1. Select Load Cases/Combinations > **ST : Nodal**
2. Select Components > **FZ**
3. Type of Display > **Value, Legend (on)**
4. Click [...] in Values
5. Number Options > Decimal Points : **'5'**
6. Click **[OK]**

► Fig 9.18
Check reaction force



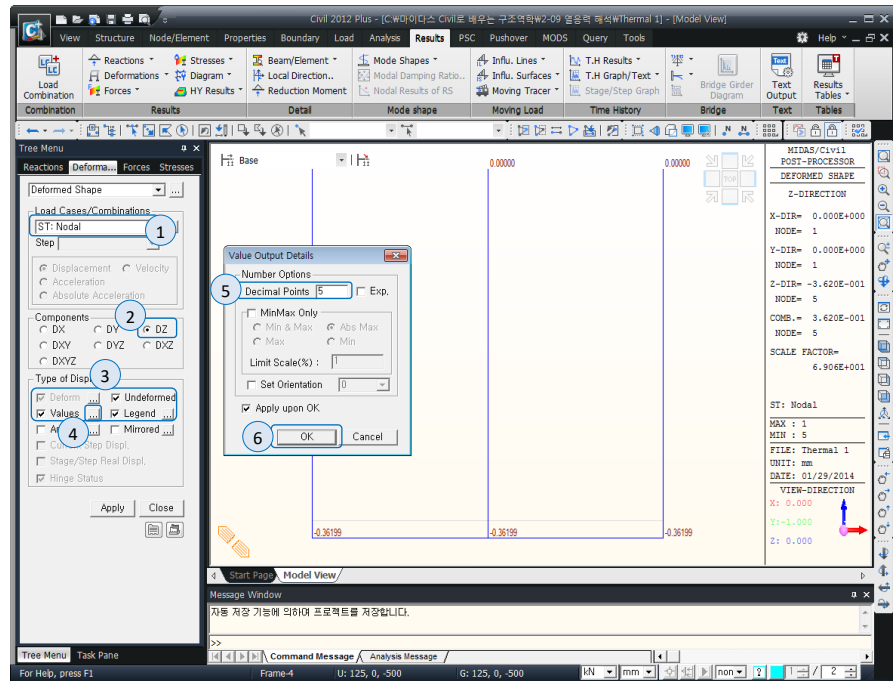
Check reaction force 9.64706 kN at Truss 2 and 5.17647 kN at Truss 1 and 3.

Check the deformation due to nodal load.

Main Menu > **Results > Deformations > Deformed Shape...**

1. Select Load Cases/Combinations > **ST : Nodal**
2. Select Components > **DZ**
3. Type of Display > **Undeformed, Values, Legend (on)**
4. Click [...] in Values
5. Number Options > Decimal Points : **'5'**
6. Click **[OK]**

► Fig 9.19
Check deformation

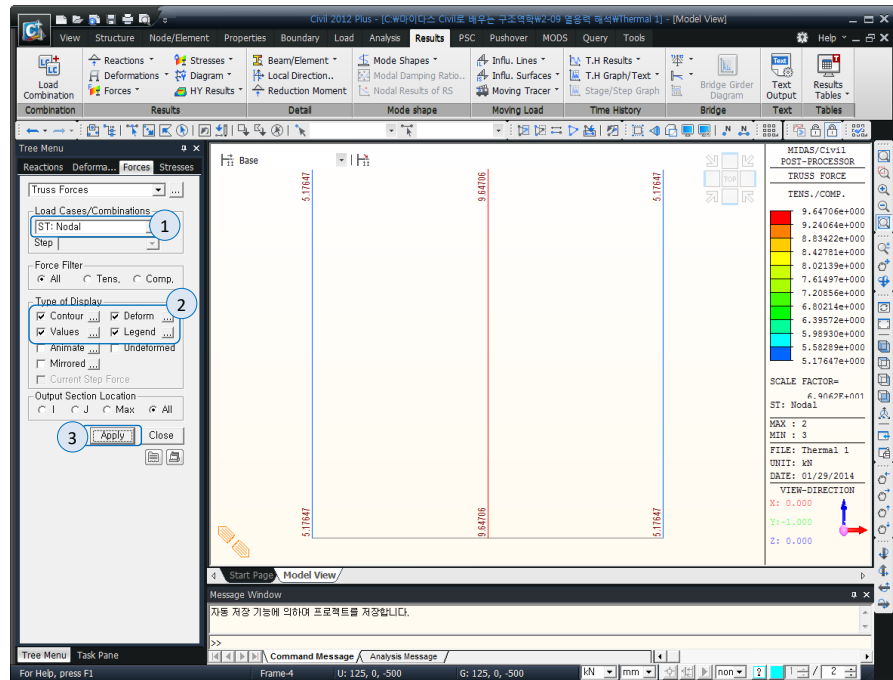


Check the axial force due to nodal load.

Main Menu > **Results** > **Forces** > **Truss Forces...**

1. Select Load Cases/Combinations > **ST : Nodal**
2. Type of Display > **Contour, Deform, Values, Legend (on)**
3. Click **[Apply]**

► Fig 9.20
Check deformation



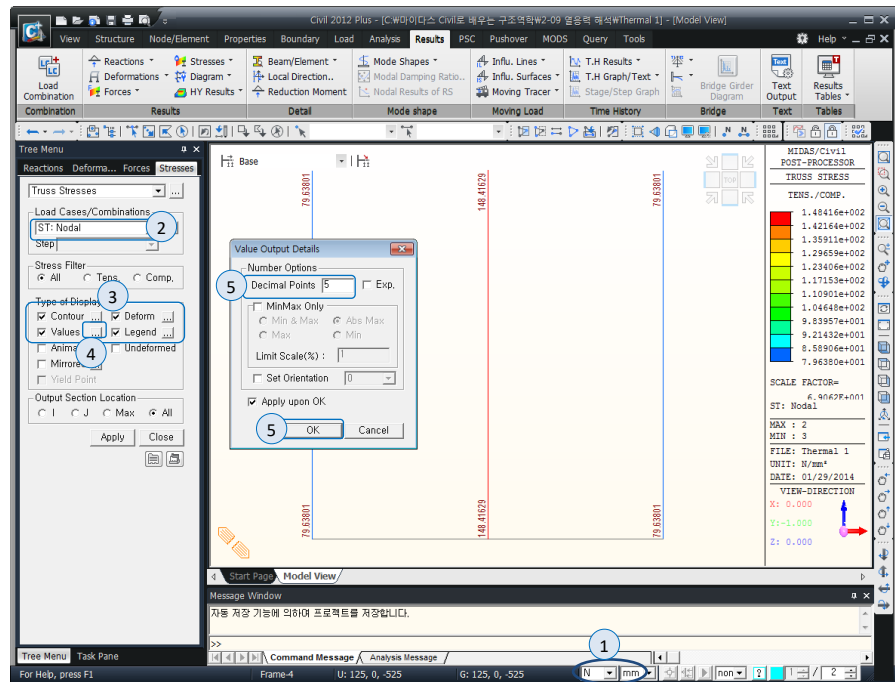
Check 9.64706 kN in the Truss 2 and 5.17647 kN in the Truss 1 and 3.

Check the truss stress due to nodal load.

Main Menu > **Results** > **Stresses** > **Truss Stresses...**

1. Modify **kN, mm** unit system at the bottom of the screen to **N, mm** system
2. Select Load Cases/Combinations > **ST : Nodal**
3. Type of Display > **Contour, Deform, Values, Legend** (on)
4. Click [...] in Values
5. Number Options > Decimal Points : '**5**', Click [OK]

► Fig 9.21
Truss stress



Check stress 41629 N/mm² in Truss 2 member and 79.63801 N/mm² in Truss 1 and 3.

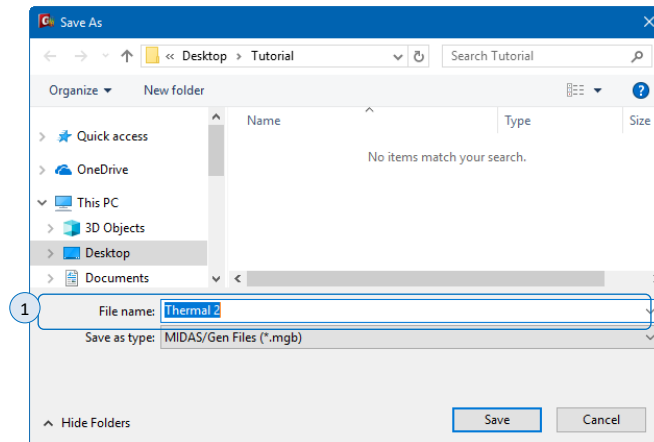


Save as different file name for Rigid Link model.

Main Menu > **File** > **Save As...**

1. Enter a name : **'Thermal 2'**, Click **[SAVE]**

► Fig 9.22
Save new name

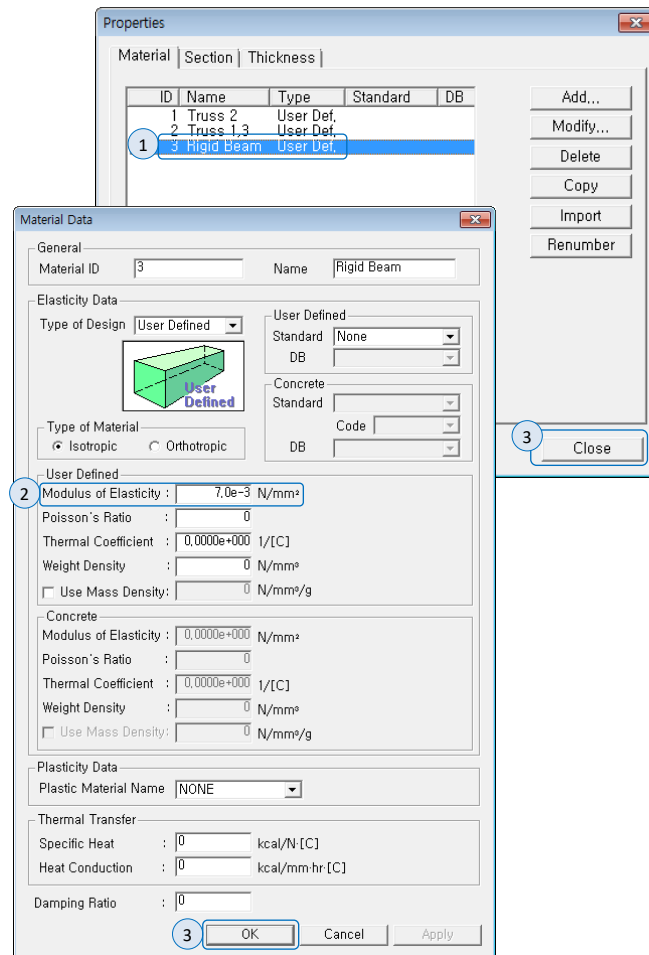


Modify the modulus of elasticity of the existing rigid beam in order to convert Rigid Link from Rigid Body.

Main Menu > **Properties** > **Material** > **Material Properties**

1. Double Click **Rigid Beam** on Lists
2. Elasticity Data > User Defined > Modulus of Elasticity : '**7.0e-3**'
3. Click **[OK]** and **[Close]**

► Fig 9.23
Modify material property



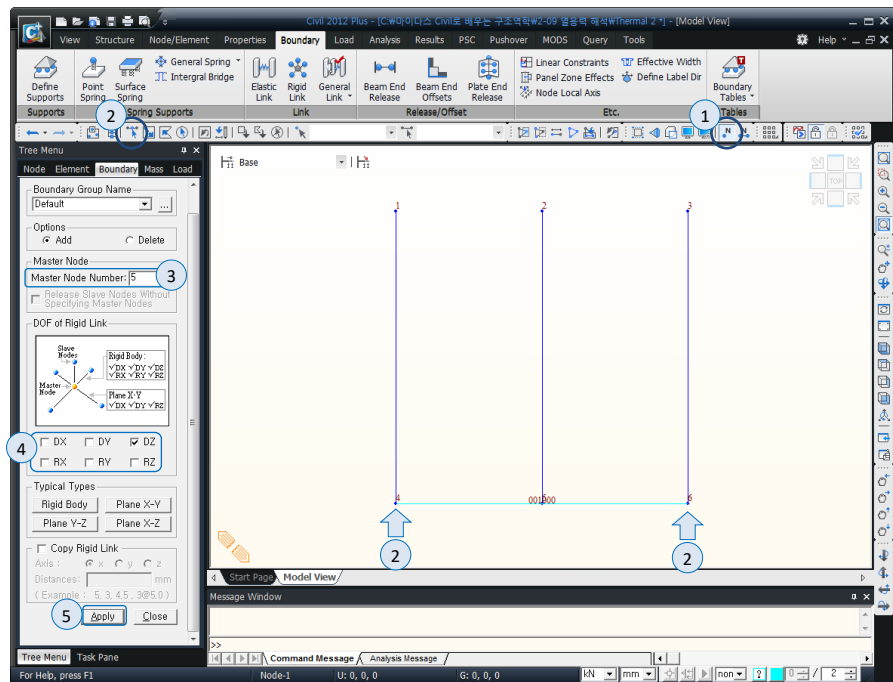
Connect node 4 & 6 to node 5 which is mater node of Rigid Link.

Change the unit to kN, mm

Main Menu > **Boundary** > **Link** > **Rigid Link**

1. **Display Node Numbers** (on)
2. Click **Select Single** (on), Select node number 4 and 6
3. Master Node > Master Node Number : '5'
4. DOF of Rigid Link > **DZ** (on), **DX, DY, RX, RY, RZ** (off)
5. Click **[Apply]**

► Fig 9.24
Rigid link connection

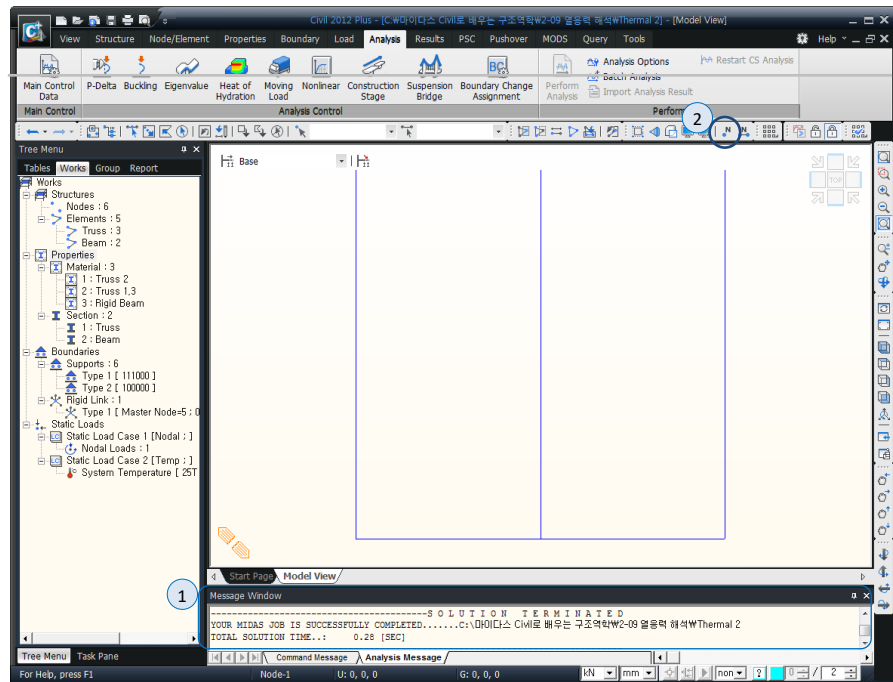


Check a model modification and perform analysis

Main Menu > **Analysis** > **Perform Analysis**

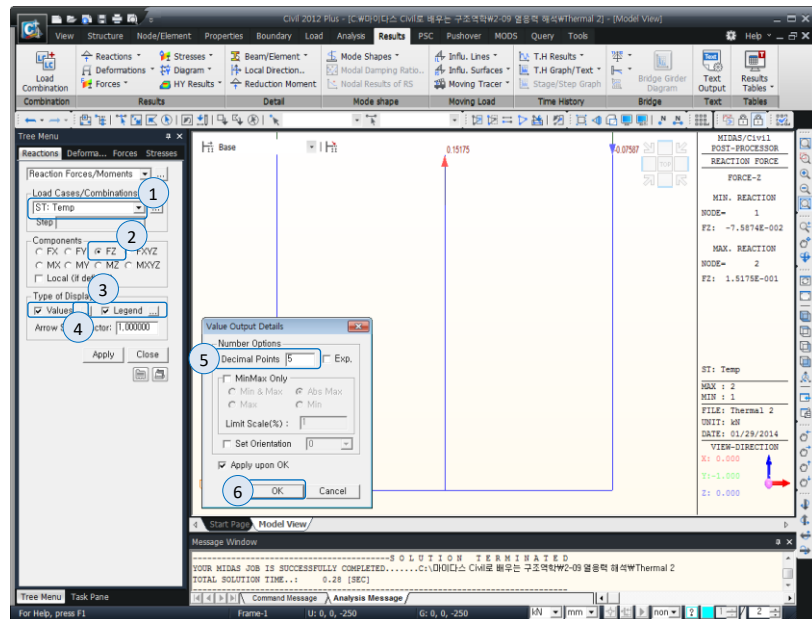
1. 1. Check for successful completion in Message Window
2. Display Node Numbers (off)

► Fig 9.25
Message for a
successful run



Main Menu > **Results** > **Reaction** > **Reaction Forces/Moments**

- Fig 9.26
Reaction force



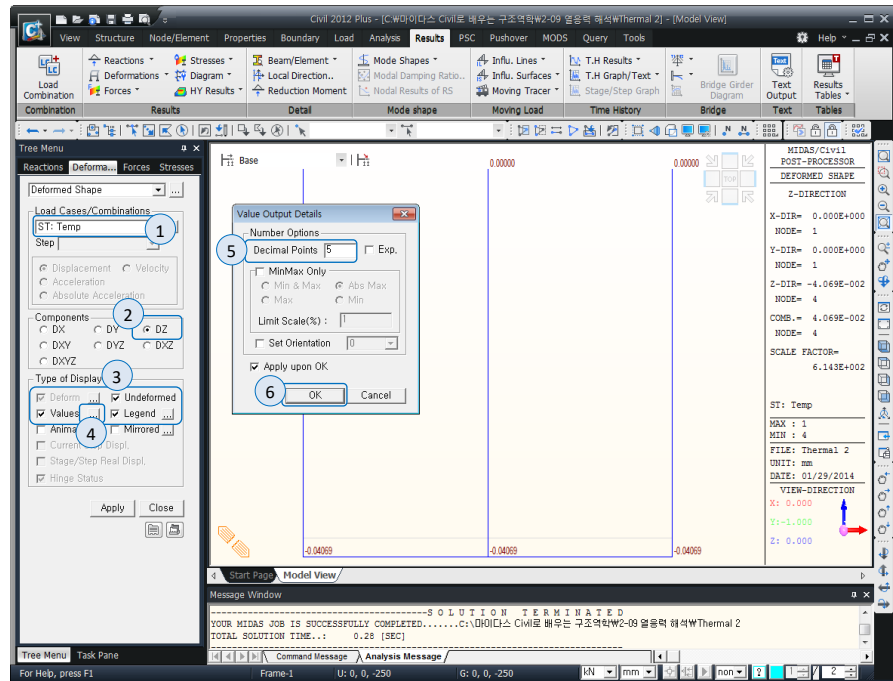
Truss 1, 2, 3 members are connected by rigid link for to the vertical direction of freedom. Therefore, Truss 2 which has a small thermal expansion coefficient has +Z Direction reaction force due to tensile stress. Truss 1, 3 has a reaction force in the -Z Direction generated.

Check the deformation due to temperature load

Main Menu > **Results > Deformations > Deformed Shape...**

1. Select Load Cases/Combinations > **ST : Temp**
2. Select Components > **DZ**
3. Type of Display > **Undeformed, Values, Legend (on)**
4. Click [...] in Values
5. Number Options > Decimal Points : **'5'**
6. Click **[OK]**

► Fig 9.27
Deformation



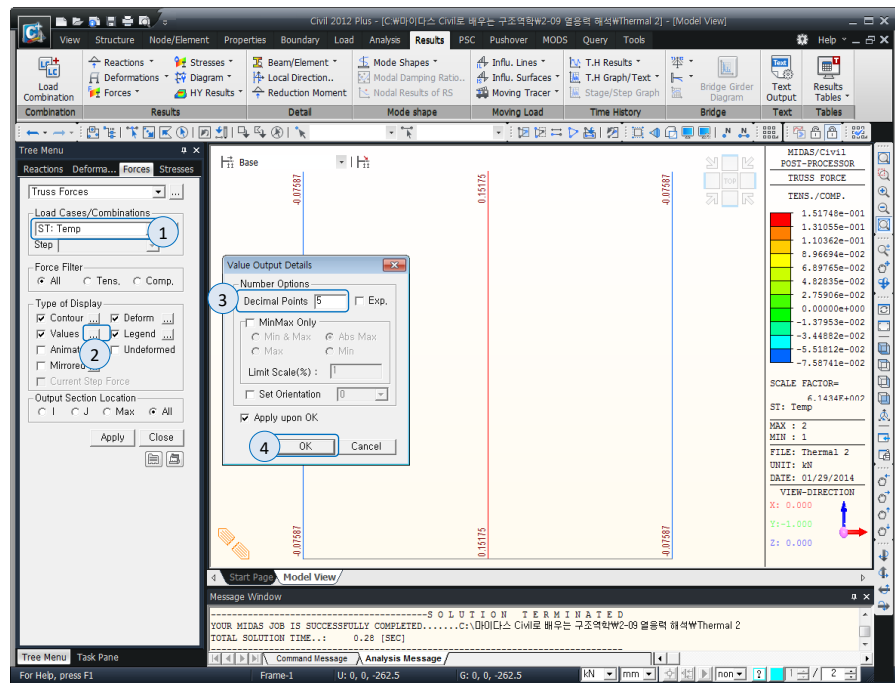
Check deformation 0.04069mm at node 4, 5, which connected by the rigid link.

Check the axial force due to temperature load

Main Menu > **Results** > **Forces** > **Truss Forces...**

1. Select Load Cases/Combinations > **ST : Temp**
2. Click [...] in Values
3. Number Options > Decimal Points : **'5'**
4. Click **[OK]**

► Fig 9.28
Deformation



Check a tensile force of 0.15175 kN on Truss 2 member, and compressive force of 0.07587 kN on Truss 1 and 3 member.

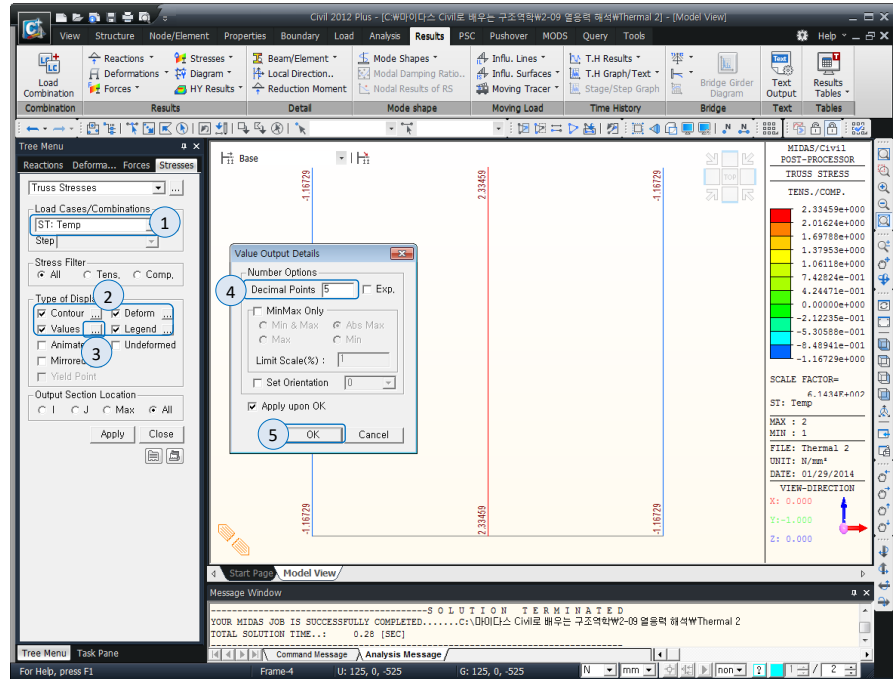
Confirm the truss stress due to temperature load.

Main Menu > **Results** > **Stresses** > **Truss Stresses...**

Change unit system to kN, mm

1. Select Load Cases/Combinations > **ST : Temp**
2. Type of Display > **Contour, Deform, Values, Legend** (on)
3. Click [...] in Values
4. Number Options > Decimal Points : **'5'**
5. Click **[OK]**

► Fig 9.29
Truss stress



Confirm the stress 2.33459 N/mm² on Truss 2 member, and -1.16729 N/mm² on Truss 1 & 3 members.

Check the stress 150.75 N/mm² on Truss 2 member, and 78.47 N/mm² in Truss 1 and 3 members due to the load combination condition (LCB1)



10. Plate Analysis on Out-of-plane Load

Contents

1 Introduction

1.1 Concept of P-delta Analysis	10-3
---------------------------------	------

2 Tutorial

2.1 Model Overview	10-9
2.2 Work Environment	10-11
2.3 Material & Section Properties	10-13
2.4 Generate Node & Element	10-15
2.5 Define Boundary Conditions	10-18
2.6 Define Loads	10-19
2.7 Perform Analysis	10-29
2.8 Check Analysis Result	10-30

3 Exercise	10-39
------------	-------



1. Introduction

1.1 Concept of Plate Analysis on the Out-of-plane Load

Typically, plate elements are required for analysis of structures which would bear out-of-plane loads like bridge slab, foundation slab, retaining walls, etc. The pressure loads are borne by the plates and are transferred to the foundation/ground via adjacent elements like beams, columns and walls.

Plate element has 6 degrees of freedom at each node. But the in-plane torsional stiffness corresponding to θ_z is provided just to a level such that it prevents the analysis errors. Elements with appropriate assumptions can be used for structural analysis. Plate elements solutions are based on approximations and hence the division of elements is essential. The degree of finer divisions for appropriate interpretation should be determined from the convergence obtained from the analysis results.

► Fig 10.1
Nodal degrees of freedom
of plate elements

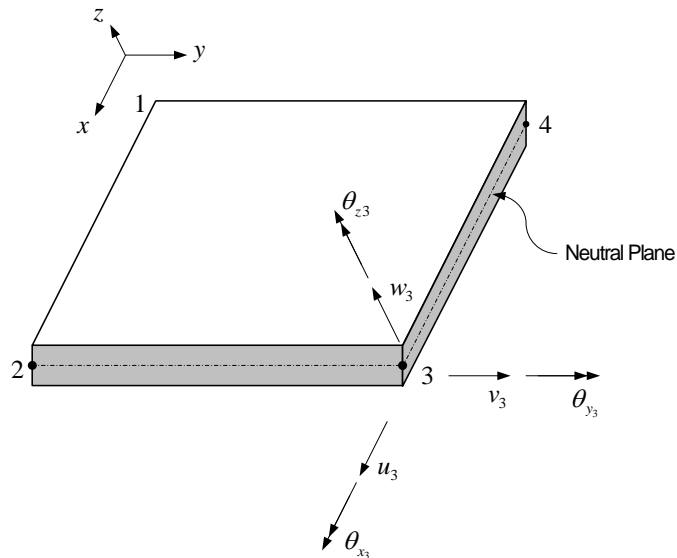
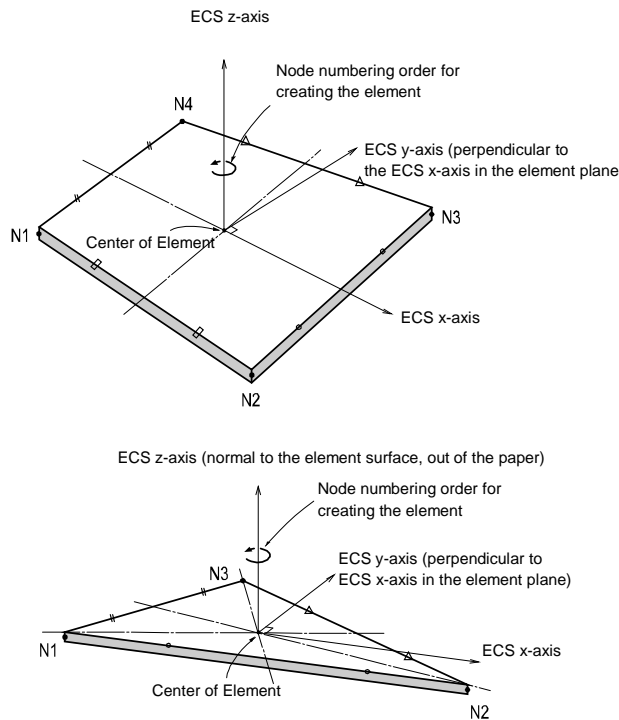


Plate elements can be triangular or rectangular in shape. In case of rectangular plate elements, the x-axis is aligned such that it is equidistant from nodes 1-4 and nodes 2-3. For triangular elements, the x-axis is parallel to the line formed by connecting nodes 1 & 2 as shown in the image below.

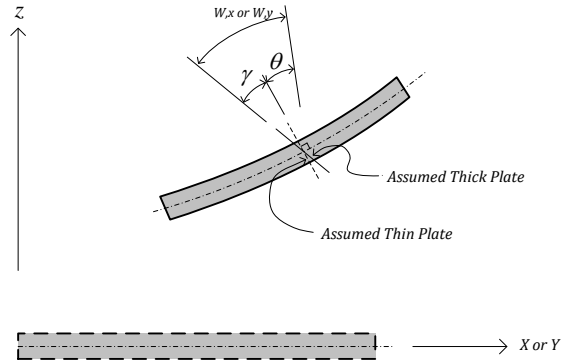
► Fig 10.2
Element coordinate
system of plate element



In case of the out-of-plane loads acting on the plates, since bending deformations occur, the concept of one dimensional beam elements could be extended to the two dimensional plate elements. Since the effects of shear deformations can't be ignored in case of thick beams (Timoshenko beam) same as in case of thin beam (Euler-Bernoulli beam), similarly, the plates are divided into thin plates (Kirchhoff-Love plate) and thick plate (Mindlin-Reissner plate). However, the geometric concept introduced here is same as that of beam. The deformations occurring due to bending moment or shear force in beam or plate are defined as below.

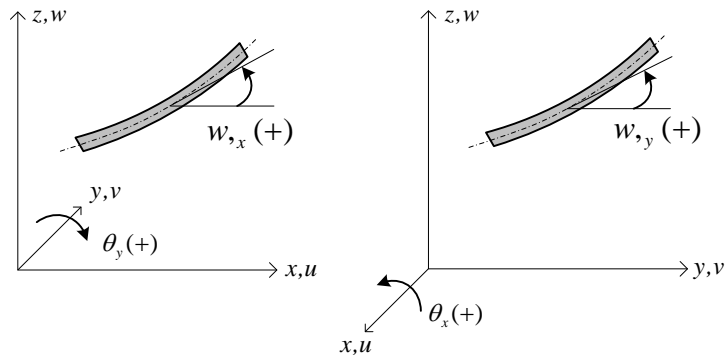


► Fig 10.3
Thin and thick plates



As it could be noted from the figure, the effects of shear deformation is also taken into account for thick plate elements. However θ_x , θ_y , w_x and w_y have different definition of positive direction. θ_x and θ_y are signed in accordance to the right hand rule, while w_x and w_y are based on differentiation and hence the sign is determined in accordance to angle of tilt. As a result, the positive direction of θ_y and w_x are opposite to each other and that of θ_x and w_y are the same.

► Fig 10.4
Direction of θ_x , θ_y , w_x
and w_y





From Fig. 10.4, the relation between the slope of deflection curve and the rotation angle is as follows.

► Eq 10.1

$$\begin{aligned} w_{,x} &= -\theta_y + \gamma_{zx} & \text{or} & \quad \theta_y = -w_{,x} + \gamma_{zx} \\ w_{,y} &= \theta_x + \gamma_{yz} & \text{or} & \quad \theta_x = w_{,y} - \gamma_{yz} \end{aligned}$$

The deformation angles of thin beam (or plate) and thick beam (or plate) can be summarized as:

► Eq 10.2

Thin Beam:	$\theta_y = -w_{,x}$	Thin plate:	$\theta_y = -w_{,x}, \quad \theta_x = w_{,y}$
------------	----------------------	-------------	---

► Eq 10.3

Thick Beam:	$\theta_y = -w_{,x} + \gamma_{xz}$	Thick plate:	$\theta_y = -w_{,x} + \gamma_{xz}, \quad \theta_x = w_{,y} - \gamma_{yz}$
-------------	------------------------------------	--------------	---

The displacement u in the x direction and the displacement v in the y direction are as follows (Fig. 10.4).

► Eq 10.4

$$u = z\theta_y, \quad v = -z\theta_x$$

There are $\varepsilon_x, \varepsilon_y, \gamma_{xy}$ when using thin plate assumption and $\varepsilon_x, \varepsilon_y, \gamma_{xy}, \gamma_{xz}, \gamma_{yz}$ when using thick plate assumption. Therefore, the strain for each of the thin plate and the thick plate can be expressed as follows, assuming that the section of the plate element maintains the plane after deformation.

Thin Plate:

► Eq 10.5

$$\begin{aligned} \varepsilon_x &= u_{,x} = z\theta_{y,x} = -zw_{,xx} \\ \varepsilon_y &= v_{,y} = -z\theta_{x,y} = -zw_{,yy} \\ \gamma_{xy} &= u_{,y} + v_{,x} = z\theta_{y,y} - z\theta_{x,x} = -2zw_{,xy} \\ \gamma_{xz} &= \gamma_{yz} = 0 \end{aligned}$$



Thick Plate:

► Eq 10.6

$$\begin{aligned}\varepsilon_x &= u_{,x} = z\theta_{y,x} = z(-w_{,xx} + \gamma_{xz,x}) \\ \varepsilon_y &= v_{,y} = -z\theta_{x,y} = z(-w_{,yy} + \gamma_{yz,y}) \\ \gamma_{xy} &= u_{,y} + v_{,x} = z\theta_{y,y} - z\theta_{x,x} = z(-2w_{,xy} + \gamma_{xz,y} + \gamma_{yz,x}) \\ \gamma_{yz} &= w_{,y} - \theta_x \\ \gamma_{zx} &= w_{,x} + \theta_y\end{aligned}$$

Except for the values in the y-z plane in the above equation, the result is the same as the beam element. The relationship of the stress-strain to the plane stress in the thin plate assumption is as follows.

► Eq 10.7

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

The moments and shear forces in the plate elements are obtained by integrating the stresses as follows.

► Eq 10.8

$$M_{xx} = \int_I \sigma_x z dz, \quad M_{yy} = \int_I \sigma_y z dz, \quad M_{xy} = \int_I \tau_{xy} z dz, \quad V_{xx} = \int_I \tau_{zx} dz = 0, \quad V_{yy} = \int_I \tau_{yz} dz = 0$$

Applying Equation 10.5 to the thin plate gives the following relation.

► Eq 10.9

$$\begin{Bmatrix} M_{xx} \\ M_{yy} \\ M_{xy} \end{Bmatrix} = D \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} -w_{,xx} \\ -w_{,yy} \\ -2w_{,xy} \end{Bmatrix}$$

In the case of a thick plate, the relationship between force and deformation can be obtained in the same way.

► Eq 10.10

$$\begin{Bmatrix} M_{xx} \\ M_{yy} \\ M_{xy} \\ V_{xx} \\ V_{yy} \end{Bmatrix} = D \begin{bmatrix} 1 & \nu & 0 & 0 & 0 \\ \nu & 1 & 0 & 0 & 0 \\ 0 & 0 & (1-\nu)/2 & 0 & 0 \\ 0 & 0 & 0 & kGt/D & 0 \\ 0 & 0 & 0 & 0 & kGt/D \end{bmatrix} \begin{Bmatrix} -w_{,xx} + \gamma_{xz,x} \\ -w_{,yy} + \gamma_{yz,y} \\ -2w_{,xy} + \gamma_{xz,y} + \gamma_{yz,x} \\ w_{,y} - \theta_x \\ w_{,x} + \theta_y \end{Bmatrix}$$



Where k is a constant that occurs in the process of integrating the shear stress on the section and is $5/6$ for a plate of uniform thickness. For the formulation for the finite element analysis, we can derive the relation of force-node deformation assuming the deformation w, γ, θ .

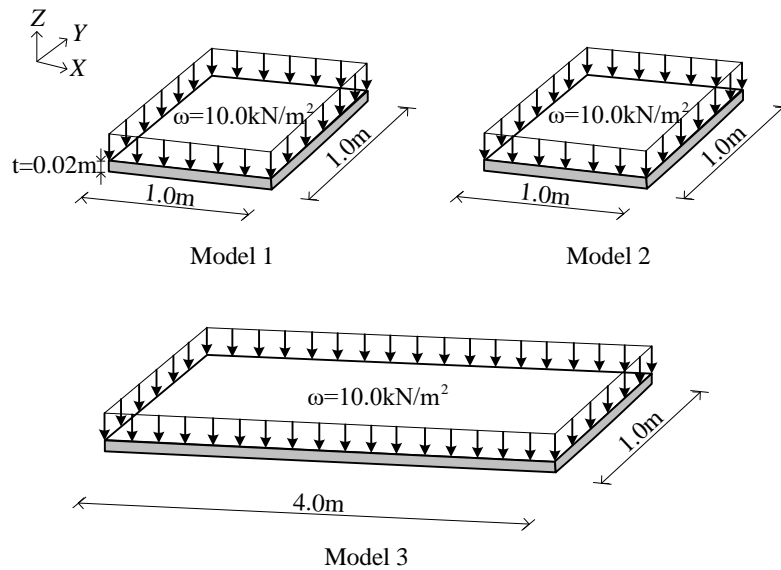


2. Tutorial

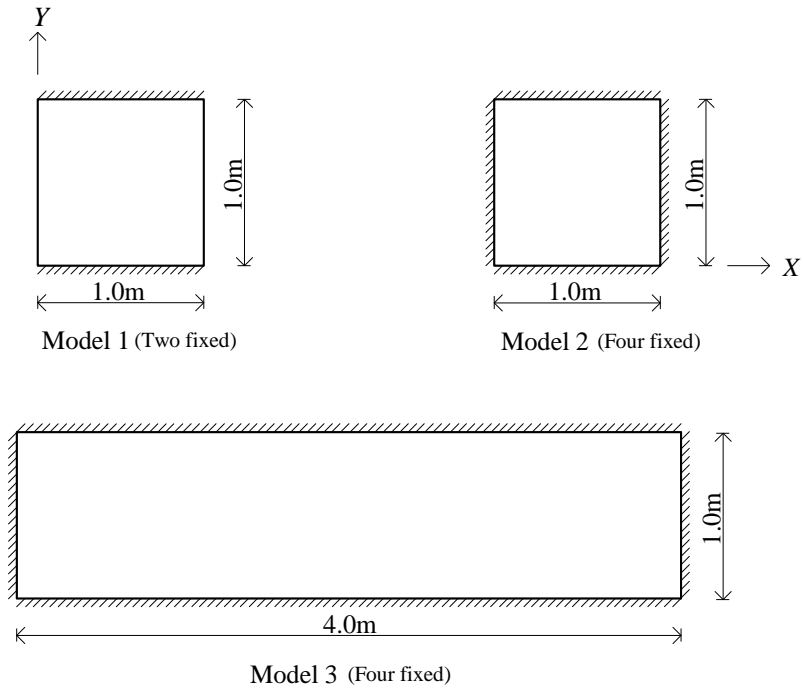
2.1 Model Overview

Examine direction of principal stress when out-of-plane loads are applied to plate.

► Figure 10.5
Analytical model (load)



► Fig 10.6
Structural geometry &
analysis model




- **Material**
Modulus of elasticity (Steel SM490) : $2.05 \times 10^5 \text{ N/mm}^2$
Poisson's ratio : 0.3
- **Section**
Thickness : 20 mm
- **Load**
A pressure load : 10 kN/m^2 is applied in the -Z direction



2.2 Work Environment

Open a new file and save the file name

Main Menu >  > **New Project...**

Main Menu >  > **Save**

1. Enter a name : **'Plate-Thick'**, Click **[SAVE]**

Set the unit system to use.

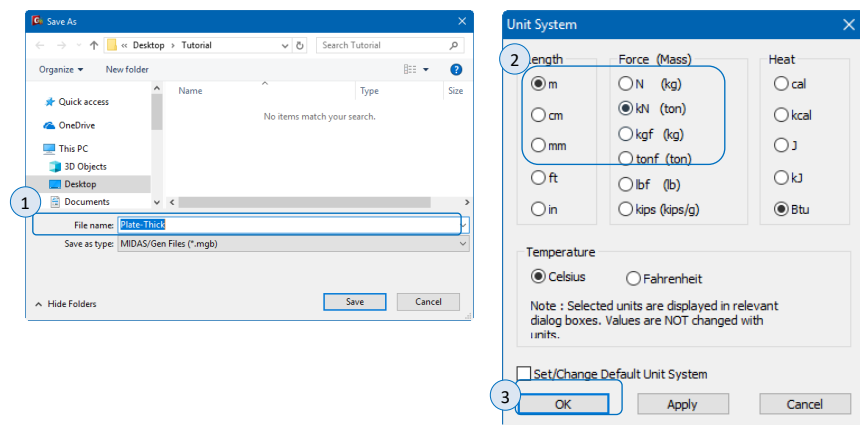
Main Menu > **Tools > Unit System...**

2. Select Length > **m**, Force(Mass) > **kN**

3. Click **[OK]**

► Fig 10.7

(a) Save the file
(b) Set unit system

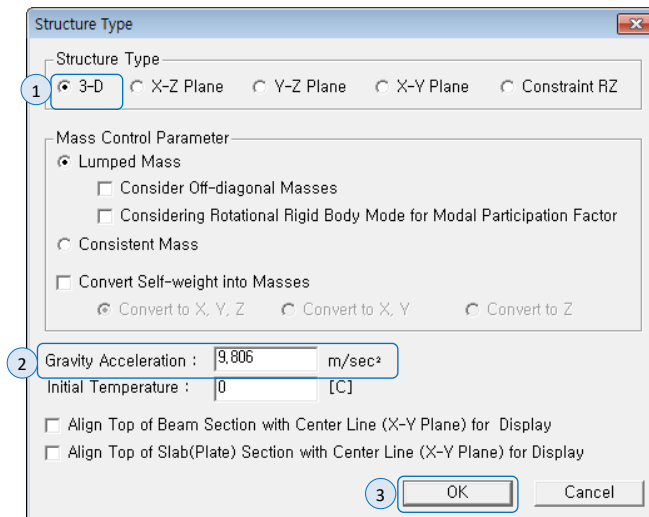


midas Gen is 3-D software, since beam exists in a 2-D plane, X-Z plane in Global Coordinate is set as the work plane, which restrains unnecessary degrees of freedom, Dy, Rx, Rz.

Main Menu > **Structure** > **Type** > **Structure Type...**

1. Select Structure Type > **3-D**
2. Gravity Acceleration : '**9.806**'
3. Click **[OK]**

► Fig 10.8
Set work plane



2.3 Material & Section Properties

Define the plate material as name SM490.

Main Menu > **Properties** > **Material** > **Material Properties**

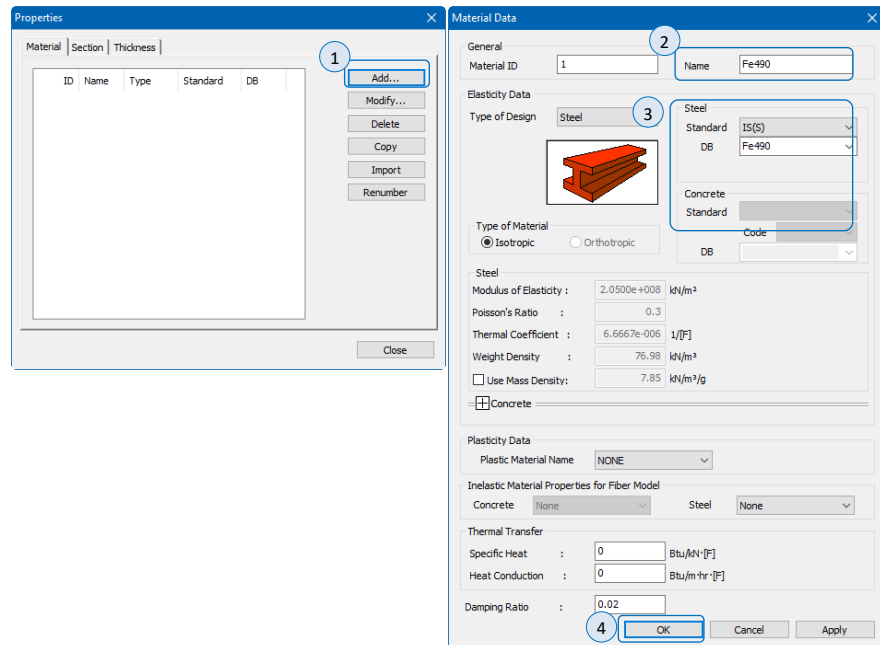
1. Click **[Add...]**

2. Name : '**SM490**'

3. Select Type of Design > **Steel** and Standard > **IS(S)** and DB > **Fe490**

4. Click **[OK]** and **[Close]**

► Fig 10.9
Define Material

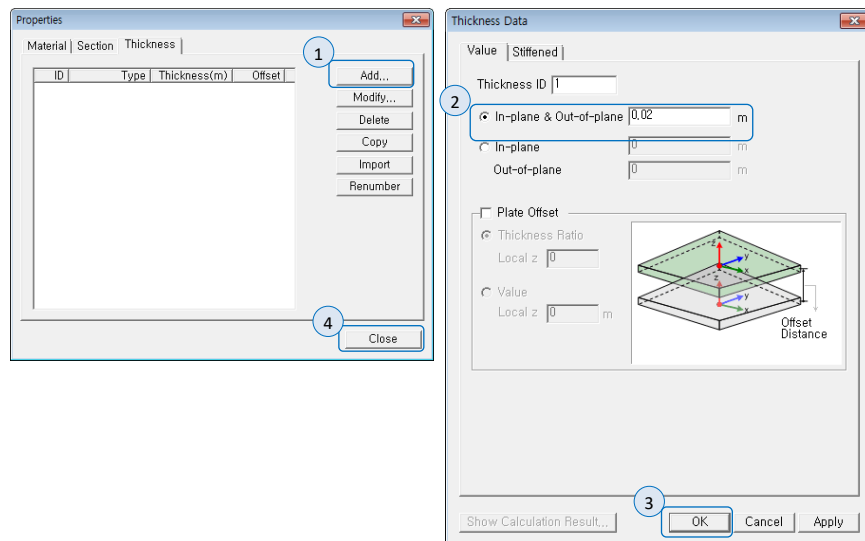


Define the thickness of plate (20 mm).

Main Menu > **Properties** > **Section** > **Thickness**

1. Click **[Add...]**
2. In-plane & Out-of plane : **'0.02'**
3. Click **[OK]**
4. Click **[Close]**

Fig 10.10
Define Thickness



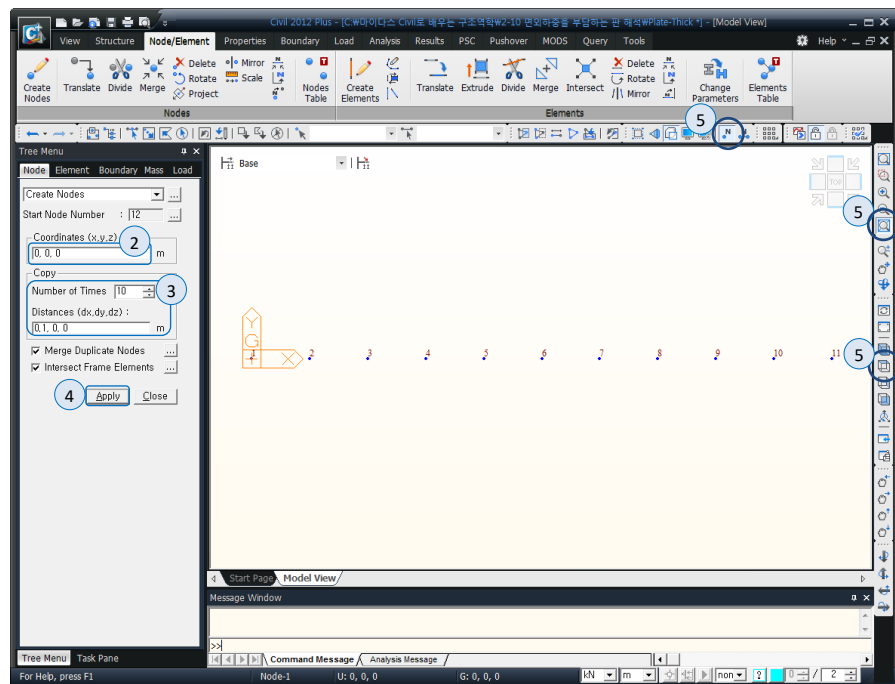
2.4 Generate Nodes & Elements

Create nodes where elements will be created.

Main Menu > **Node/Element** > **Nodes** > **Create Nodes...**

1. Coordinates (x, y, z) : '0, 0, 0'
2. Copy > Number of Times : '10'
3. Distance (dx, dy, dz) : '0.1, 0, 0'
4. Click [Apply]
5. Display Node Numbers, Auto Fitting, Top View (on)

► Fig 10.11
Create Nodes

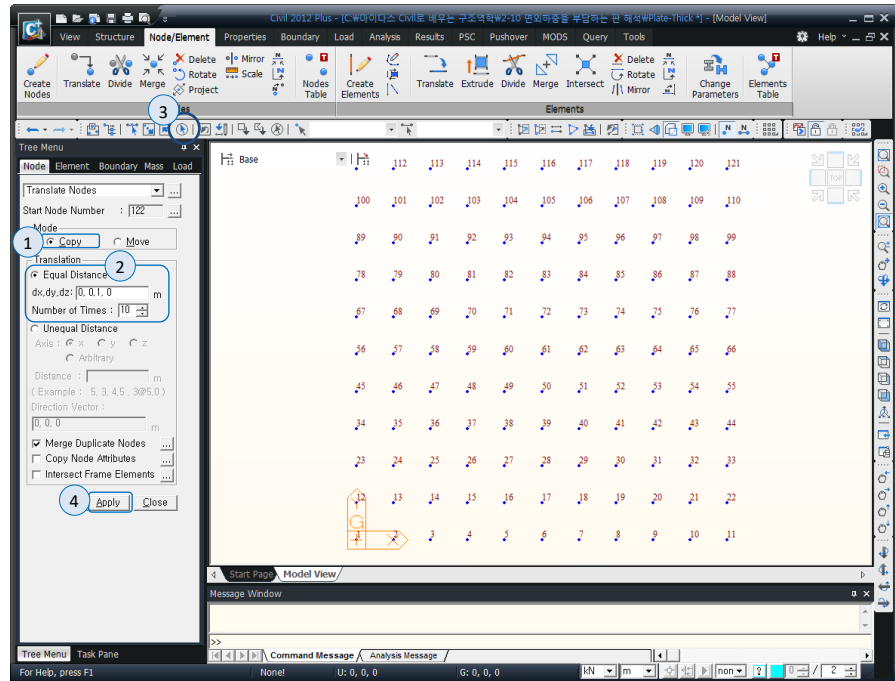


Translate nodes to create a square plate element.

Main Menu > **Node/Element** > **Nodes** > **Translate...**

1. Select Mode > **Copy**
2. Select Translation > **Equal Distance**
dx, dy, dz : '**0, 0.1, 0**', Number of Times : '**10**'
3. Click Select All (on) Select all members
4. Click **[Apply]**

► Fig 10.12
Translate nodes



Connect the nodes to create the plate.

Main Menu > **Node/Element** > **Elements** > **Create Elements...**

1. Select Element Type > **Plate**

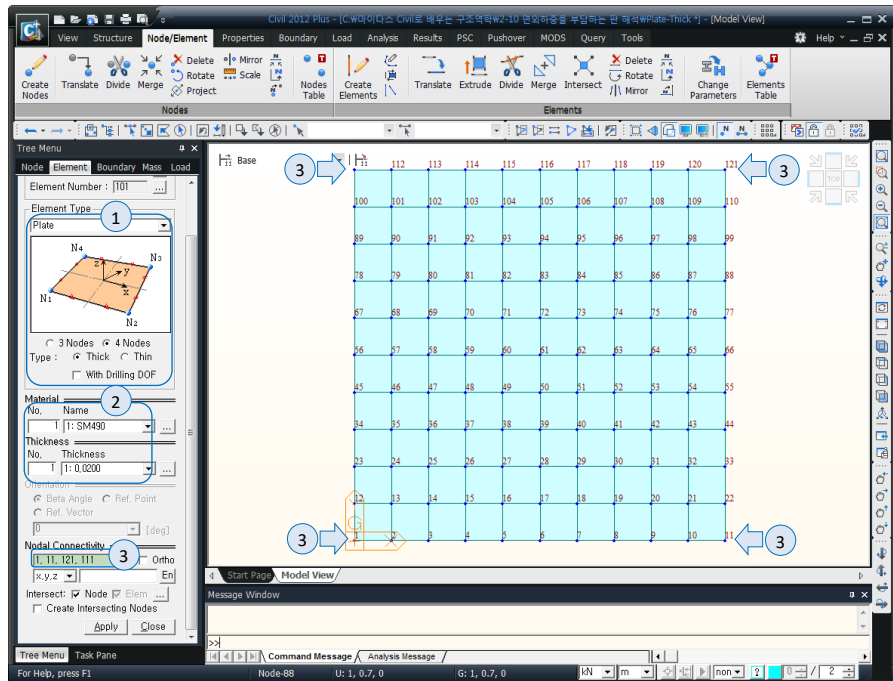
Select Type > **Thick**, With Drilling DOF (off)

2. Select Material > **1 : SM490** and Thickness > **1 : 0.0200**

3. Click Nodal Connectivity and select node number **1, 11, 121, 111**

► Fig 10.13
Create plate elements

Tip
What is Thick and Thin
when creating Plate?
Thick: Consider shear
deformation
Thin: Consideration of
shear deformation
(Thick by default)



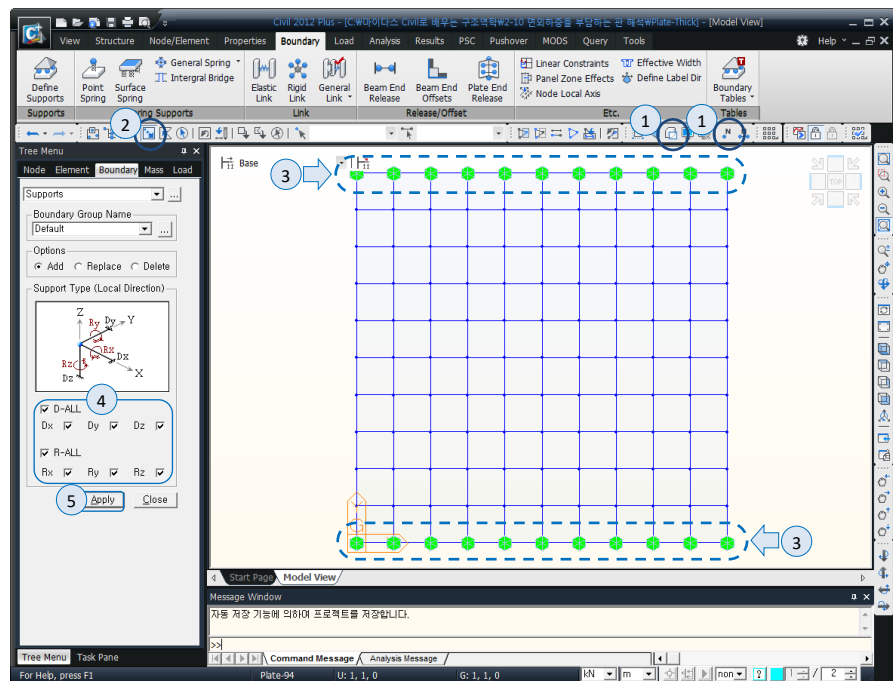
2.5 Define Boundary Conditions

Restrain all degrees of freedom (Dx, Dy, Dz, Rx, Ry, Rz) among two sides.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. Hidden, Display Node Numbers (off)
2. In the Select by Window (on) mode
3. Select the nodes
4. Support Type (Local Direction > **D-ALL** (on), **R-ALL** (on))
5. Click **[Apply]**

► Fig 10.14
Define boundary
condition





2.6 Define Loads

Define load case (load type) first to which the loading will belong.

Main Menu > **Load** > **Static Loads** > **Static Load Cases...**

1. Name : '**Pressure**'
2. Select Type > **User Defined Load (USER)**
3. Click **[Add]**
4. Click **[Close]**

► Fig 10.15
Definition load cases

Static Load Cases

1 Name : Pressure

Case : All Load Case

2 Type : User Defined Load (USER)

Description :

3 Add

Modify

Delete

No	Name	Type	Description
1	Pressure	User Defined Load (USER)	

4 Close

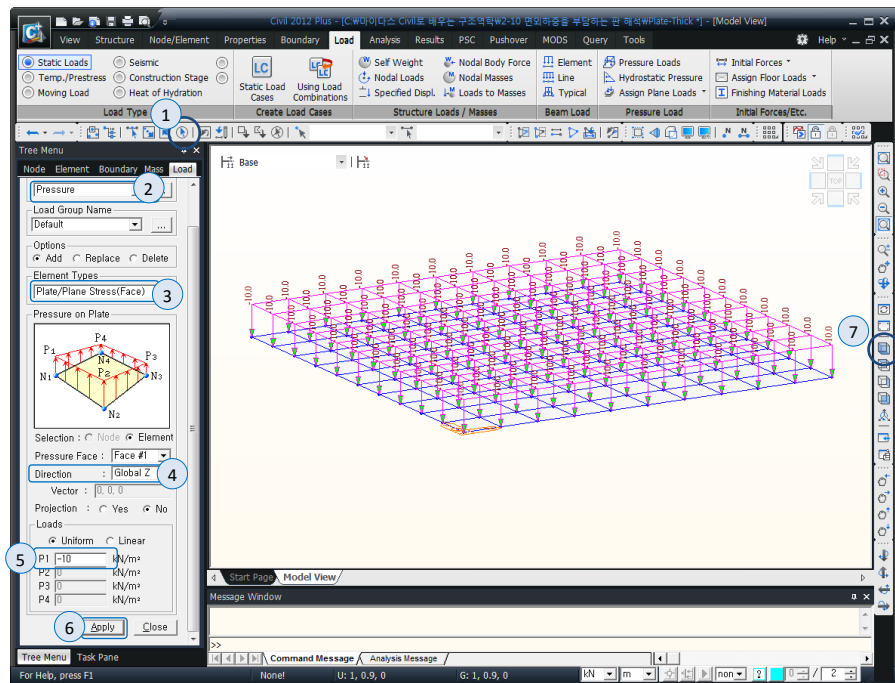
Input a pressure load of 10 kN/m^2 on the plate element

Main Menu > **Load** > **Static Loads** > **Pressure Loads...**

Main Menu > **View / Display...** > Load Tab > **Load Value** (on), Click [OK]

1. Click **Select All**
2. Select Load Case Name > **Pressure**
3. Select Element Types > **Plate/Plane Stress(Face)**
4. Select Pressure on Plate > Direction > **Global Z**
5. Loads > P1 : '-10'
6. Click **[Apply]**
7. Click **Iso View**

► Fig 10.16
Input pressure load

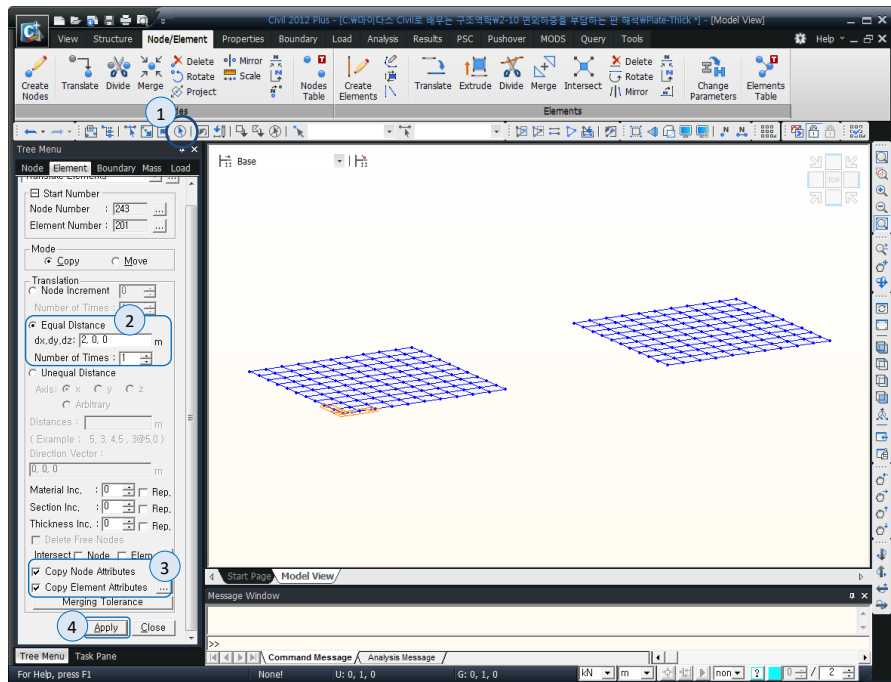


Model 2 is created by replicating pressure loads and boundary conditions entered in Model 1.

Main Menu > **Node/Element** > **Elements** > **Translate...**

1. Select **Select All**
2. Translation > Equal Distance > dx,dy,dz : '**2,0,0**', Number of Times : '**1**'
3. **Copy Node Attributes** (on), **Copy Element Attributes** (on)
4. Click **[Apply]**

► Fig 10.17
Create Model 2



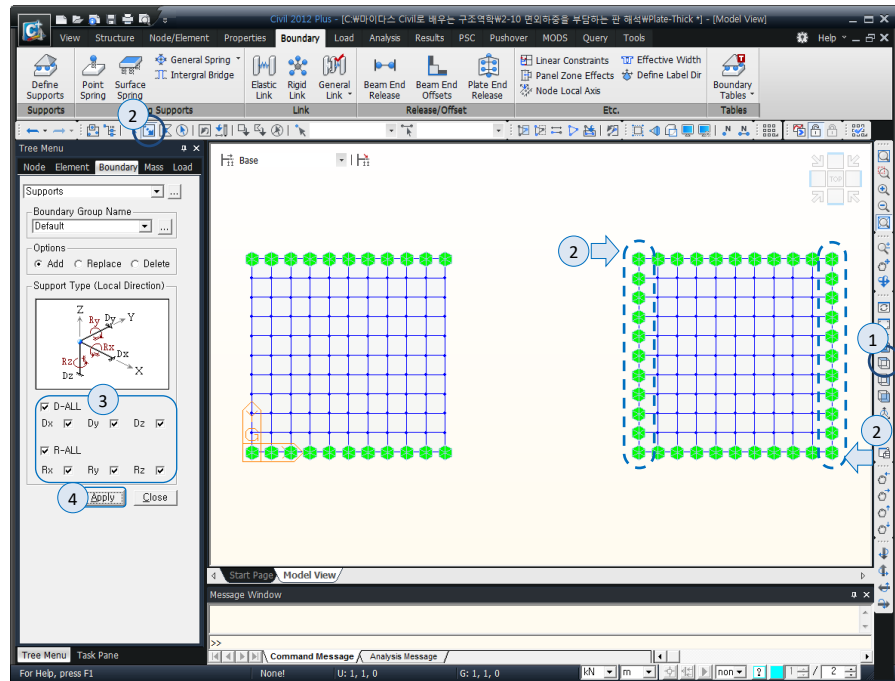
Modify boundary conditions in Model 2

Main Menu > **Boundary** > **Supports** > **Define Supports**

Main Menu > **View / Display...**, Boundary Tab > **Support** (on), Click [OK]

1. Click **Top View**
2. Click **Select by Window** (on) to select the left / right border of Model 2 (see the following figure)
3. Support Type (Local Direction) > **D-ALL** (on), **R-ALL** (on)
4. Click [Apply]

► Fig 10.18
Modify boundary condition
in model 2

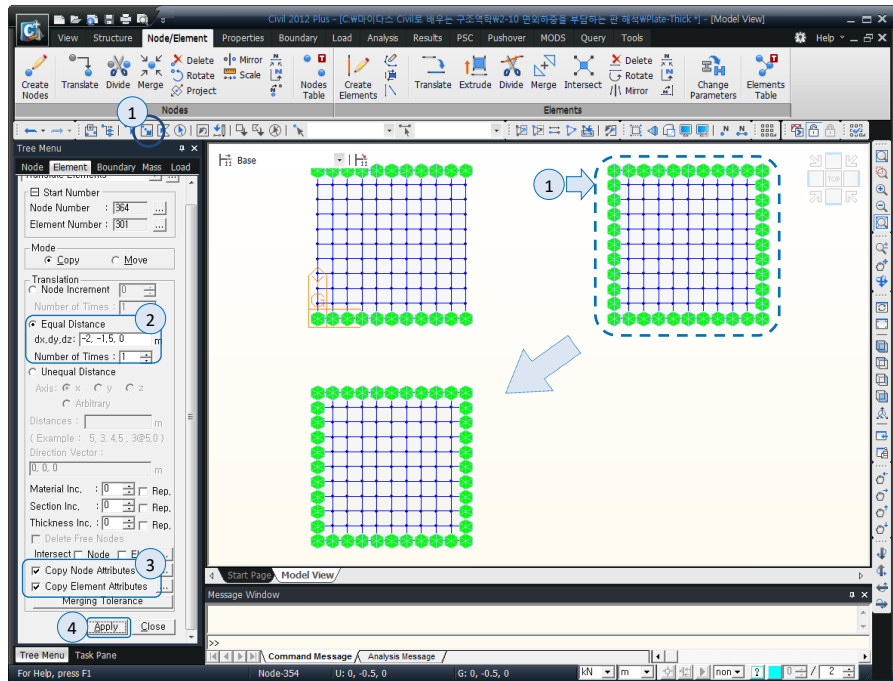


Generate model 3.

Main Menu > **Node/Element** > **Elements** > **Translate...**

1. Click **Select by Window** (on), Select all **Model 2**
2. Translation > Equal Distance > dx,dy,dz : '**-2,-1.5,0**', Number of Times : '**1**'
3. **Copy Node Attributes** (on), **Copy Element Attributes** (on)
4. Click **[Apply]**

► Fig 10.19
Create Model 3

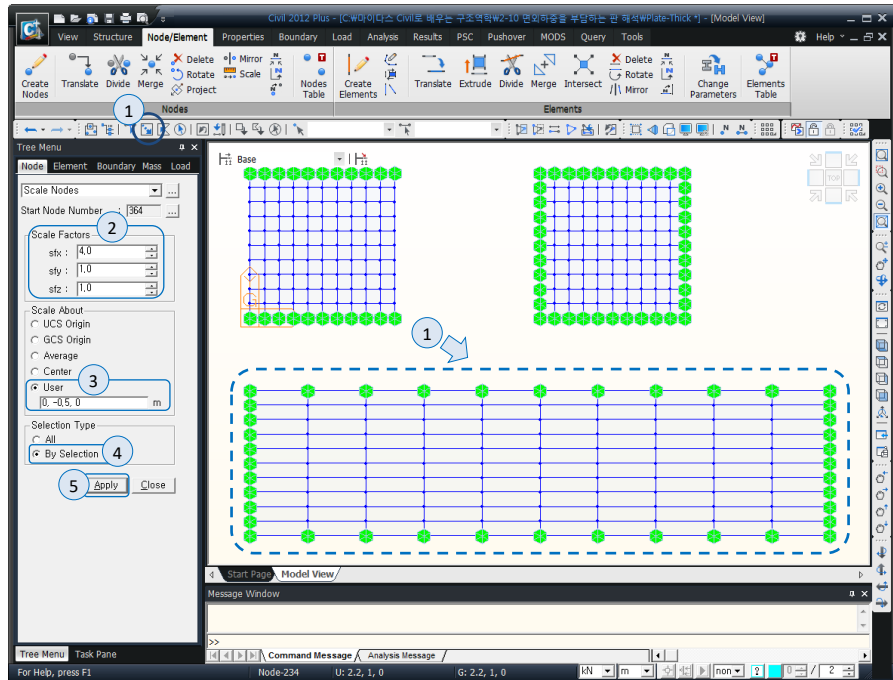


Scale up model 3 using Scale Nodes the X direction.

Main Menu > **Node/Element** > **Nodes** > **Scale...**

1. Click **Select by Window** (on), Select all **Model 3**
2. Scale Factors > sfx : **'4.0'**, sfy : **'1.0'**, sfz : **'1.0'**,
3. Scale About > User : **'0,-0.5,0'**
4. Select Selection Type > **By Selection**
5. Click **[Apply]**

► Figure 10.20
Scale up Model 3

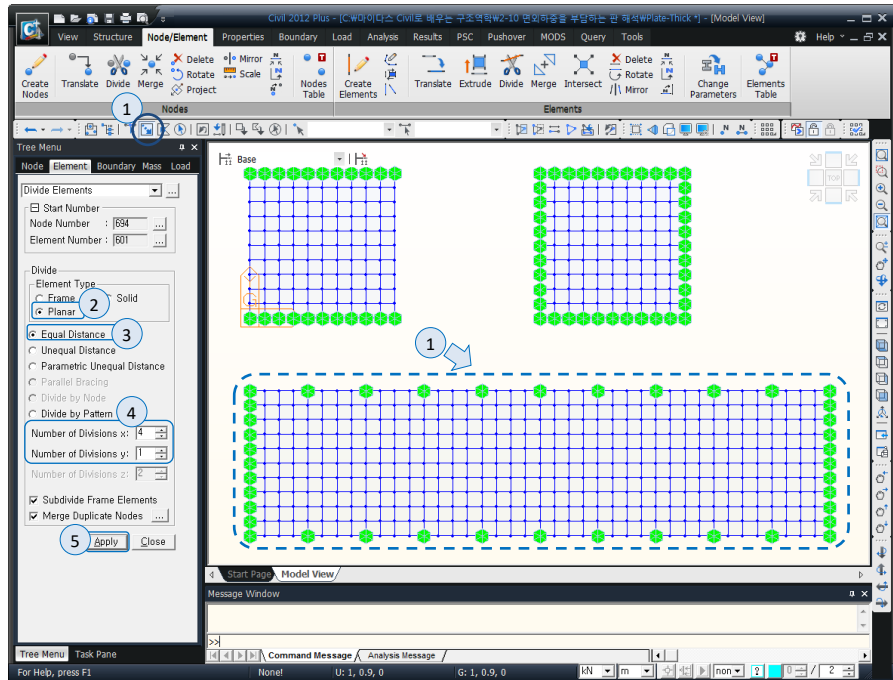


Divide plate element (Model 3) in the x-axis direction.

Main Menu > **Node/Element** > **Element** > **Divide...**

1. Click **Select by Window** (on), Select all **Model 3**
2. Select Divide > Element Type > **Planar**
3. Select Equal Distance
4. Number of Division x : '4', Number of Division y : '1'
5. Click **[Apply]**

► Fig 10.21
Divide Model 3 elements

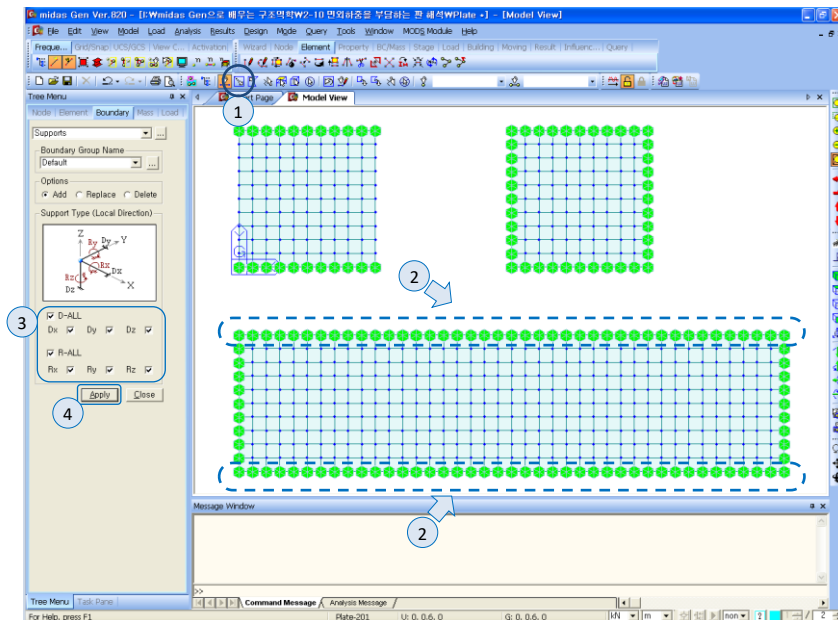


Redefine boundary condition in Model

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. Click **Select by Window** (on)
2. Select nodes in **Model 3** (top/bottom nodes)
3. Support Type (Local Direction) > **D-ALL** (on), **R-ALL** (on)
4. Click **[Apply]**

► Fig 10.22
Redefined boundary
condition

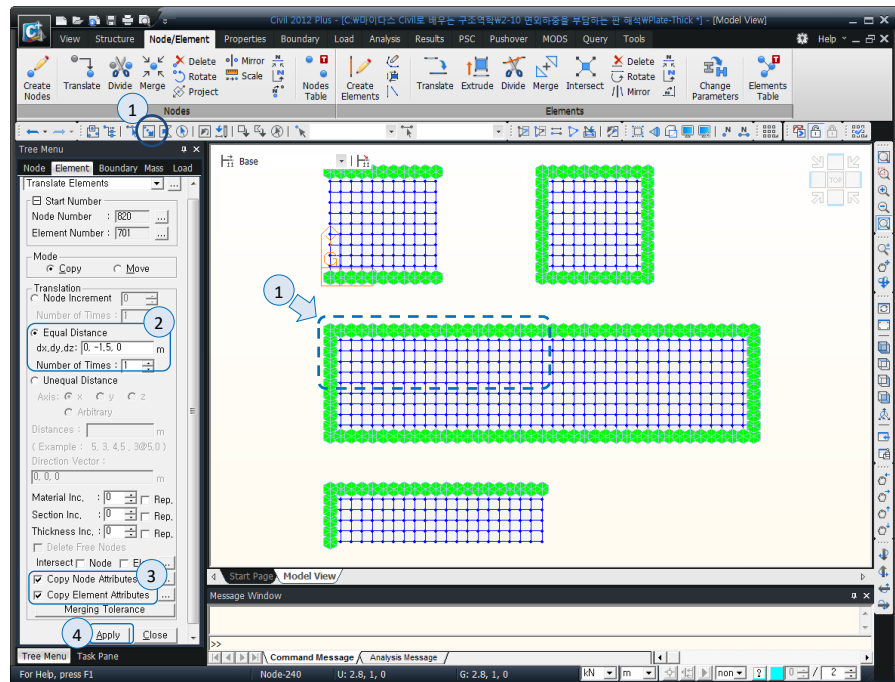


Create model 4 using Translate elements. The model 4 which is only a quarter model may be analyzed due to symmetry.

Main Menu > **Node/Element** > **Elements** > **Translate...**

1. Click Select by Window (on) Select 1/4 area element of Model 3 (20 cells in X direction, 5 cells in Y direction)
2. Translation > Equal Distance > dx,dy,dz : '**0,-1.5,0**', Number of Times : '**1**'
3. **Copy Node Attributes** (on), **Copy Element Attributes** (on)
4. Click **[Apply]**

► Figure 10.23
Creation Model 4

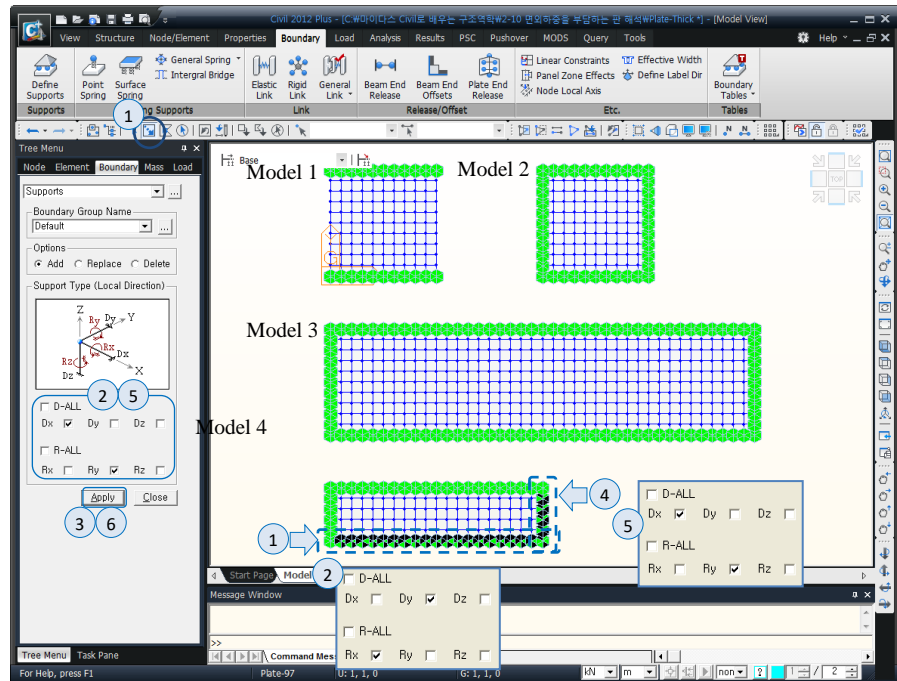


Define boundary condition in Model 4.

Main Menu > **Boundary** > **Supports** > **Define Supports**

1. Click **Select by Window** (on) and select bottom nodes in **Model 4**
2. Support Type (Local Direction) > **Dy** (on), **Rx** (on)
3. Click **[Apply]**
4. Click **Select by Window** (on) and select right nodes in **Model 4**
5. Support Type (Local Direction) > **Dx** (on), **Ry** (on)
6. Click **[Apply]**

► Fig 10.24
Define boundary condition



2.7 Perform Analysis

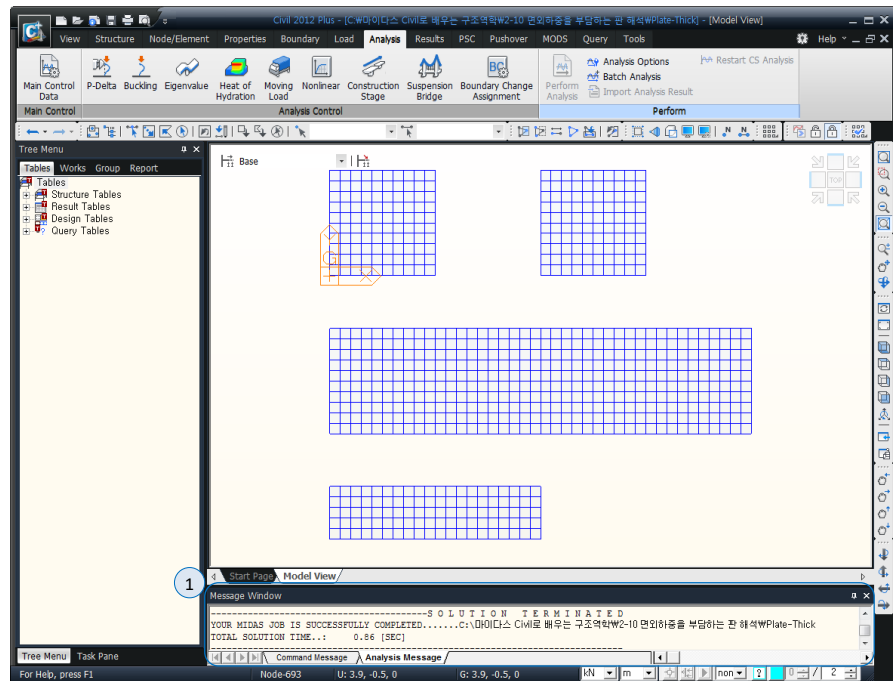
Perform structural analysis of models

Main Menu > **Analysis > Perform Analysis**

1. Check for successful completion in Message Window

2. Main Menu > **View / Display...**, Boundary Tab > **Support** (off), Click [OK]

► Fig 10.25
Message for a
successful run



2.8 Check Analysis Result

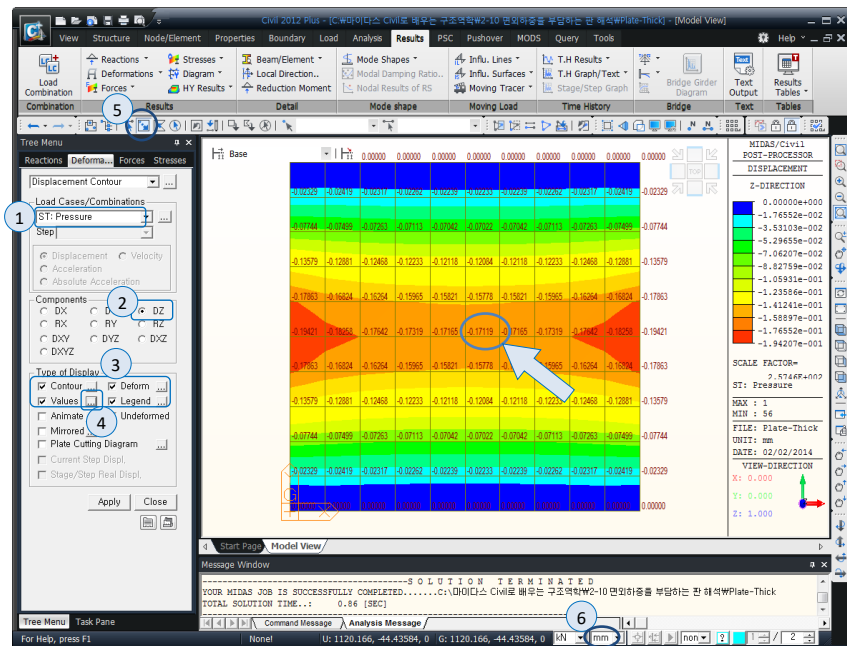
Check deflection of plate element Model 1.

Main Menu > **Results** > **Deformations** > **Displacement Contour...**

1. Select Load Cases/Combinations > **ST : Pressure**
2. Select Components > **DZ**
3. Type of Display > **Contour, Deform, Values, Legend** (on)
4. Click [...] in Values, Number Options > Decimal Point : '**5**', Click **[OK]**
5. Click **Select by Window** (on), Select all **Model 1**, enter shortcut 'F2'
6. In the status bar at the bottom of the screen, change the length unit to mm

Confirmation of deflection at center -0.17119mm

► Fig 10.26
Displacement results
(Model 1)



Check deflection of plate element Model 2, Model 4 (symmetrical model of Model 3).

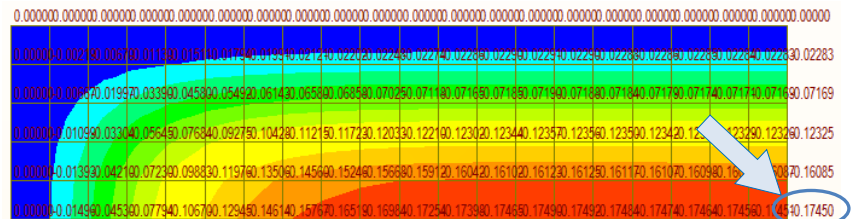
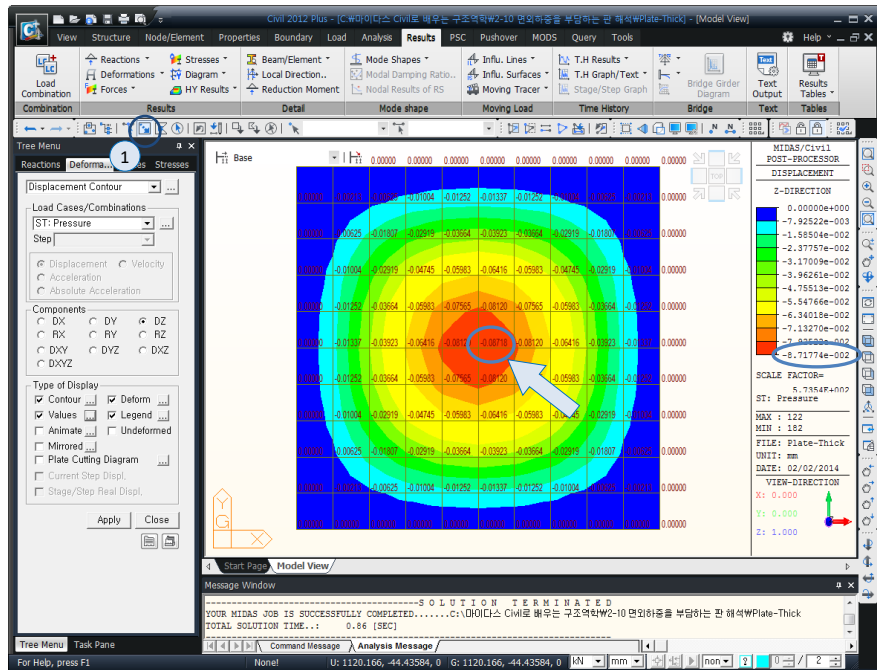
Main Menu > **View /Activities/Active All** (shortcut '**Ctrl+A**')

1. Click **Select by Window** (on), Select all **Model 2**, enter shortcut 'F2'

Check for deflection in the center **-0.08718mm** (or -8.71774e-002 in Legend)

2. Shortcuts 'Ctrl + A', Click **Select by Window** (on), Select **Model 4** selection, Enter shortcut '**F2**'. Confirmation of deflection at center **-0.17450mm**

► Fig 10.27
Displacement result
(a) Model 2
(b) Model 4



Check end and center moments of plate element Model 1 by surface load.

Main Menu > **View** > **Activities** > **Active All** (shortcut 'Ctrl+A')

Main Menu > **Results** > **Forces** > **Plate Forces/Moments...**

1. Select Load Cases/Combinations > **ST : Pressure**

2. Select Components > **Myy**

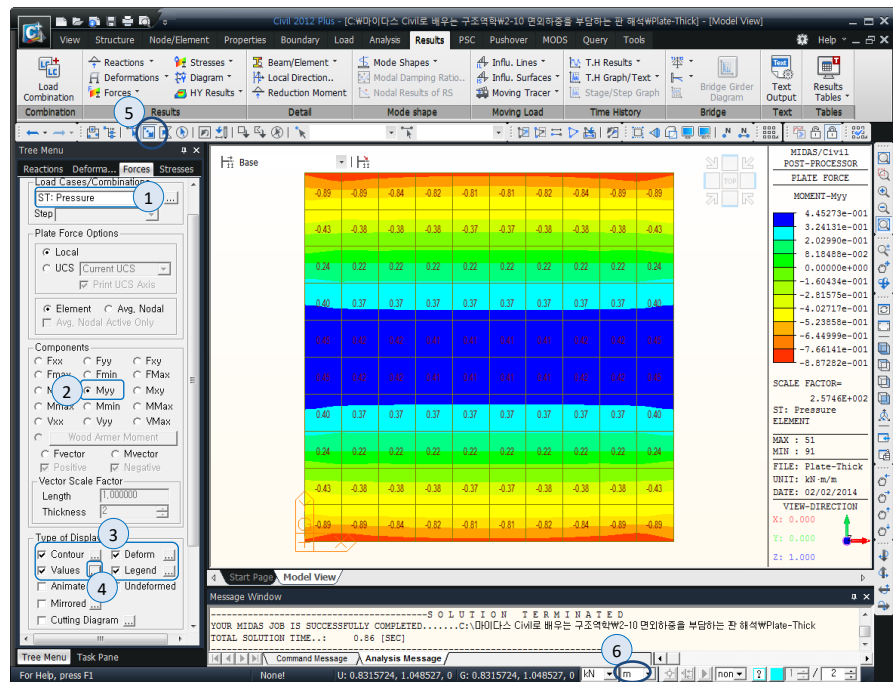
3. Type of Display > **Contour, Deform, Values, Legend** (on)

4. Click [...] in Values, Number Options > Decimal Point : '2', Click **[OK]**

5. Click **Select by Window** (on), Select all **Model 1**, enter shortcut 'F2'

6. In the status bar at the bottom of the screen, change the length unit to m. Central maximum moment **0.41 kN · m** (maximum **0.45 kN · m**), end parent moment **0.81 kN · m** (maximum **-0.89 kN · m**)

► Fig 10.28
Plate moment



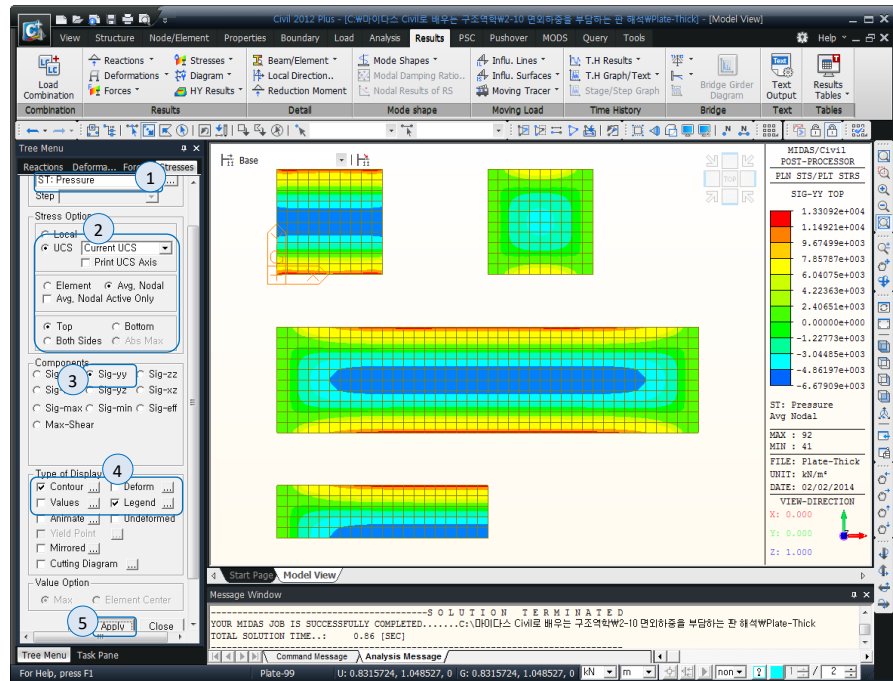
Confirm end moments and central moments of the models 2, 3 and 4 in the same way.

Check the stresses in the plate element model with different shape and boundary conditions.

Main Menu > **Results** > **Stresses** > **Plane-Stress/Plate-Stresses...**

1. Select Load Cases/Combinations > **ST : Pressure**
2. Select Stress Options > **UCS, Avg.Nodal, Top**
3. Select Components > **Sig-yy**
4. Type of Display > **Contour, Legend (on)**
5. Click **[Apply]**
6. Shortcut '**Ctrl+A**'

► Fig 10.29
Plate stresses



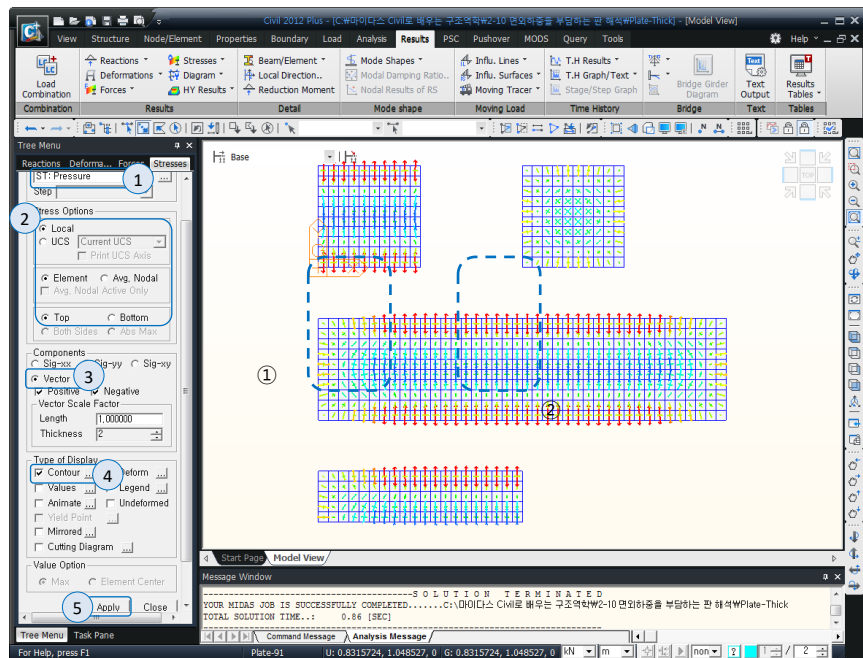
Compare the principal stress vectors of different plate element models which have different shape and boundary conditions.

The direction of principal stress of Model 1 is perpendicular to surface, and direction of principal stress of Model 2 which is fixed at four sides, is 45° to surface.

Main Menu > **Results / Stresses / Plane-Stress/Plate-Stresses...**

1. Select Load Cases/Combinations > **ST : Pressure**
2. Select Stress Options > **Local, Element, Top**
3. Select Components > **Vector**
4. Type of Display > **Contour (on)**
5. Click **[Apply]**

► Fig 10.30
Principal stress vector



Structural Analysis II (Advanced)

10. Plate Analysis on Out-of-plane Load

Save the models under a different name

Main Menu > **File** > **Save As...**

1. Enter a name : **'Plate-Thin'**, Click **[SAVE]**

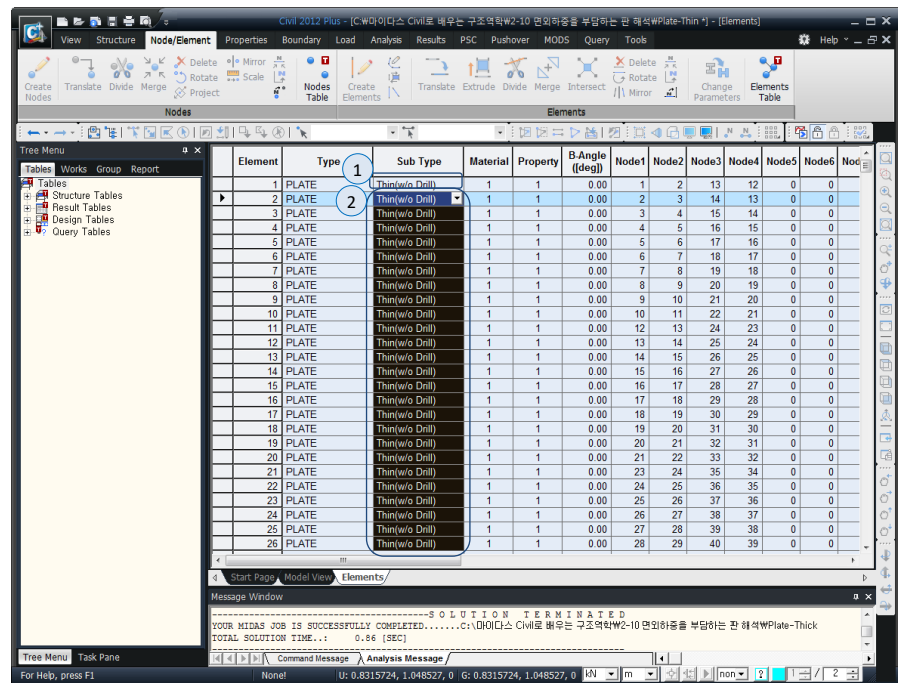
Modify the plate element to Thin (Considering shear deformation) Type

Main Menu > **Node/Element** > **Elements** > **Elements Tables...**

1. **Sub Type** in **Element 1** Shift to **'Thin(w/o Drill)'**

2. Copy Thin(w/o Drill) to the Sub Type cell of the remaining 2 ~ 700 elements
(Select copy source cell, click Ctrl + C, select copy target cell as Drag, then click Ctrl + V)

► Fig 10.31
Modify property of
please elements

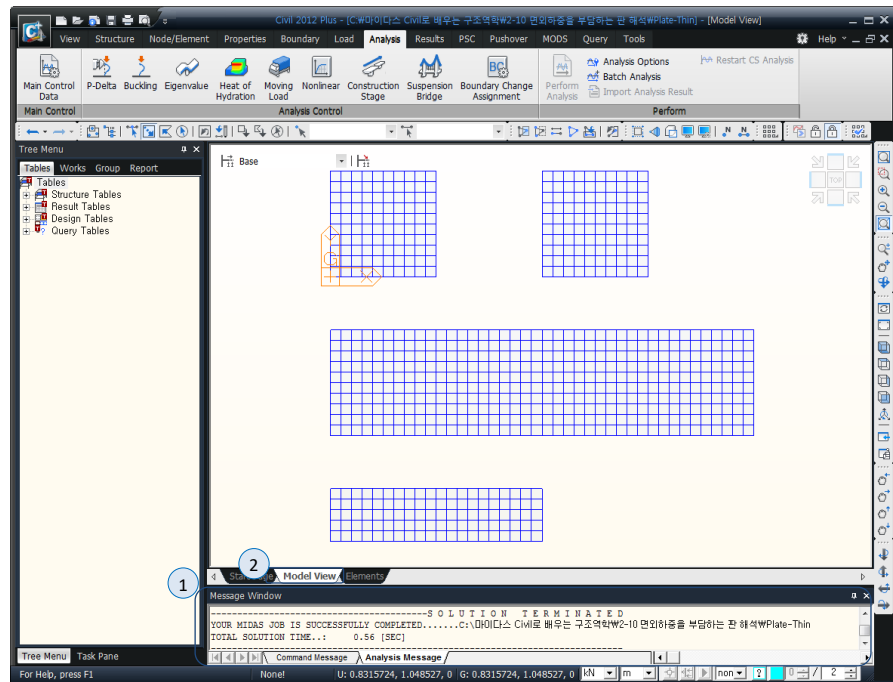


Analyze modified model which is not to consider the shear deformation.

Main Menu > **Analysis** > **Perform Analysis**

1. Check for successful completion in Message Window
2. Click Model View

► Fig 10.32
Message for a
successful run



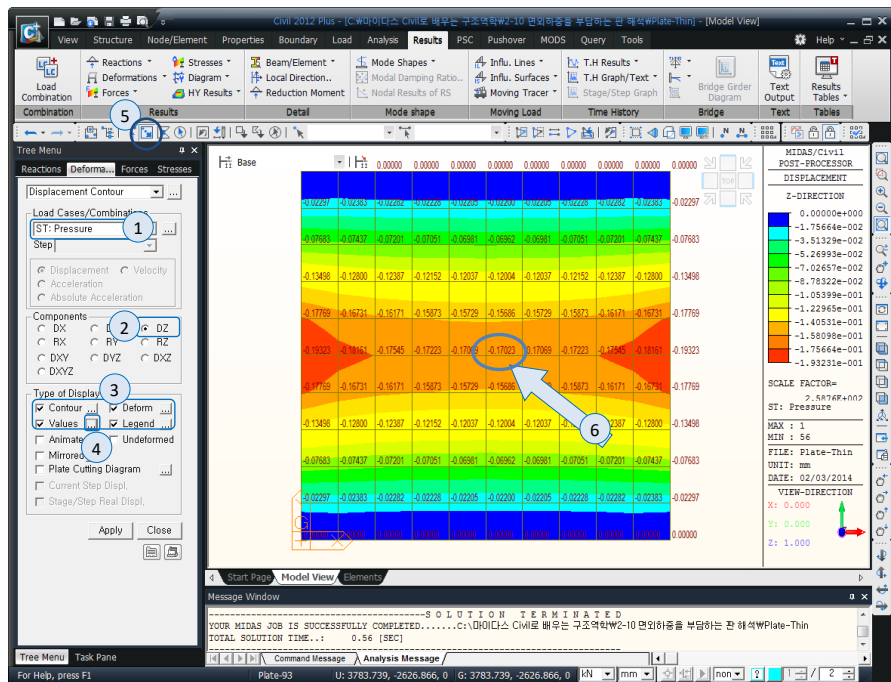
Check deflection of plate element in Model 1.

Main Menu > **Results > Deformations > Displacement Contour...**

1. Select Load Cases/Combinations > **ST : Pressure**
2. Select Components > **DZ**
3. Type of Display > **Contour, Deform, Values, Legend (on)**
4. Click [...] in **Values**, Number Options > Decimal Point : '**5**', Click **[OK]**
5. Click **Select by Window** (on), Select all **Model 1**, shortcut '**F2**'
6. Change the unit to **mm** at Status Bar

Check the displacement at the centre **-0.17023mm**

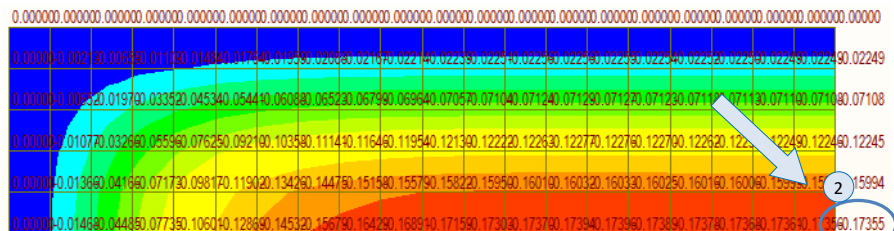
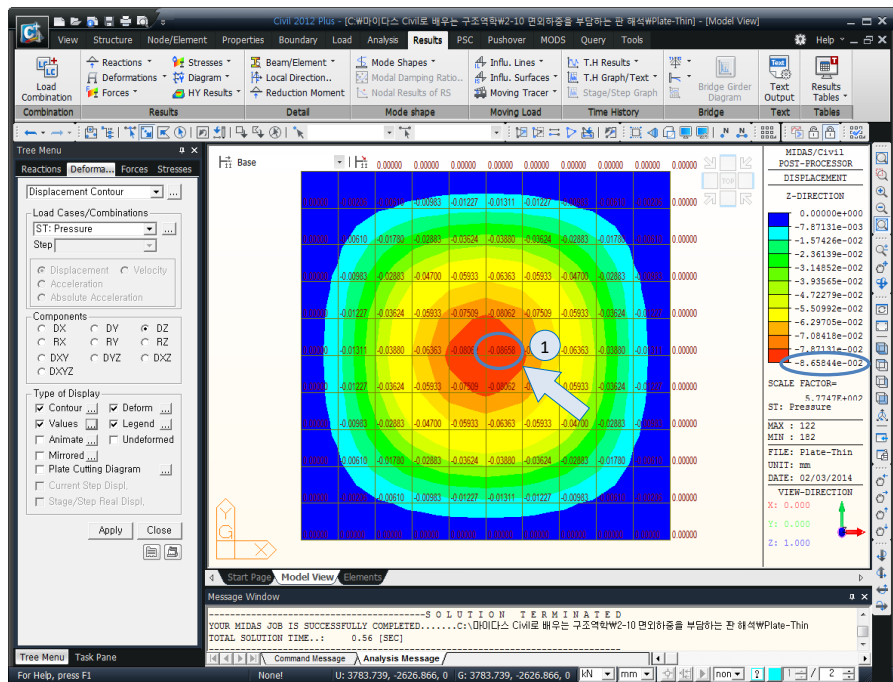
► Fig 10.33
Deformation result



Check deflection of plate element in Mode 2 and model 3.

1. Shortcut '**Ctrl+A**', Click **Select by Window** (on), Select **Model 2**, Shortcut '**F2**'
Check the displacement at the centre **-0.08658mm**
2. Shortcut '**Ctrl+A**', Click **Select by Window** (on), Select **Model 4**, shortcut '**F2**'
Check the displacement at the centre **-0.17355mm**

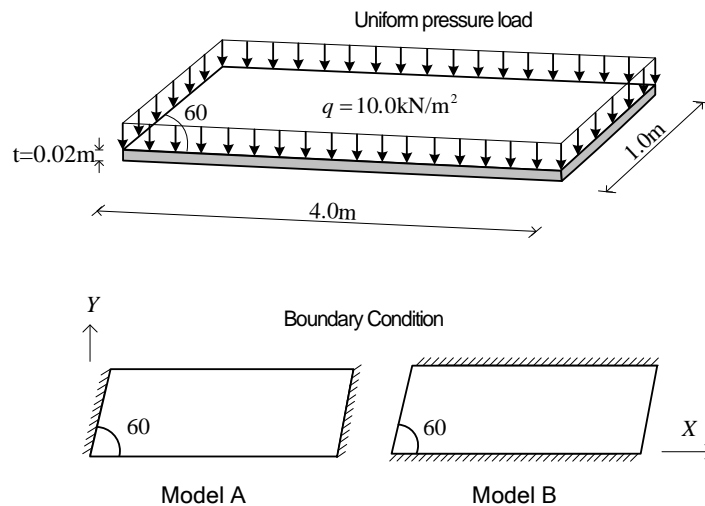
► Fig 10.34
Deformation result





3. Exercise

Determine the direction of the principal stresses with two different boundary conditions models.



- **Material**
Modulus of elasticity (Steel SM490) : $2.05 \times 10^5 \text{ N/mm}^2$
Poisson's Ratio: 0.3
- **Section**
Thickness: 20 mm
- **Load**
Uniformly distributed pressure load : 10 kN/mm^2

Structural Analysis II

Advanced Analysis with Midas Software