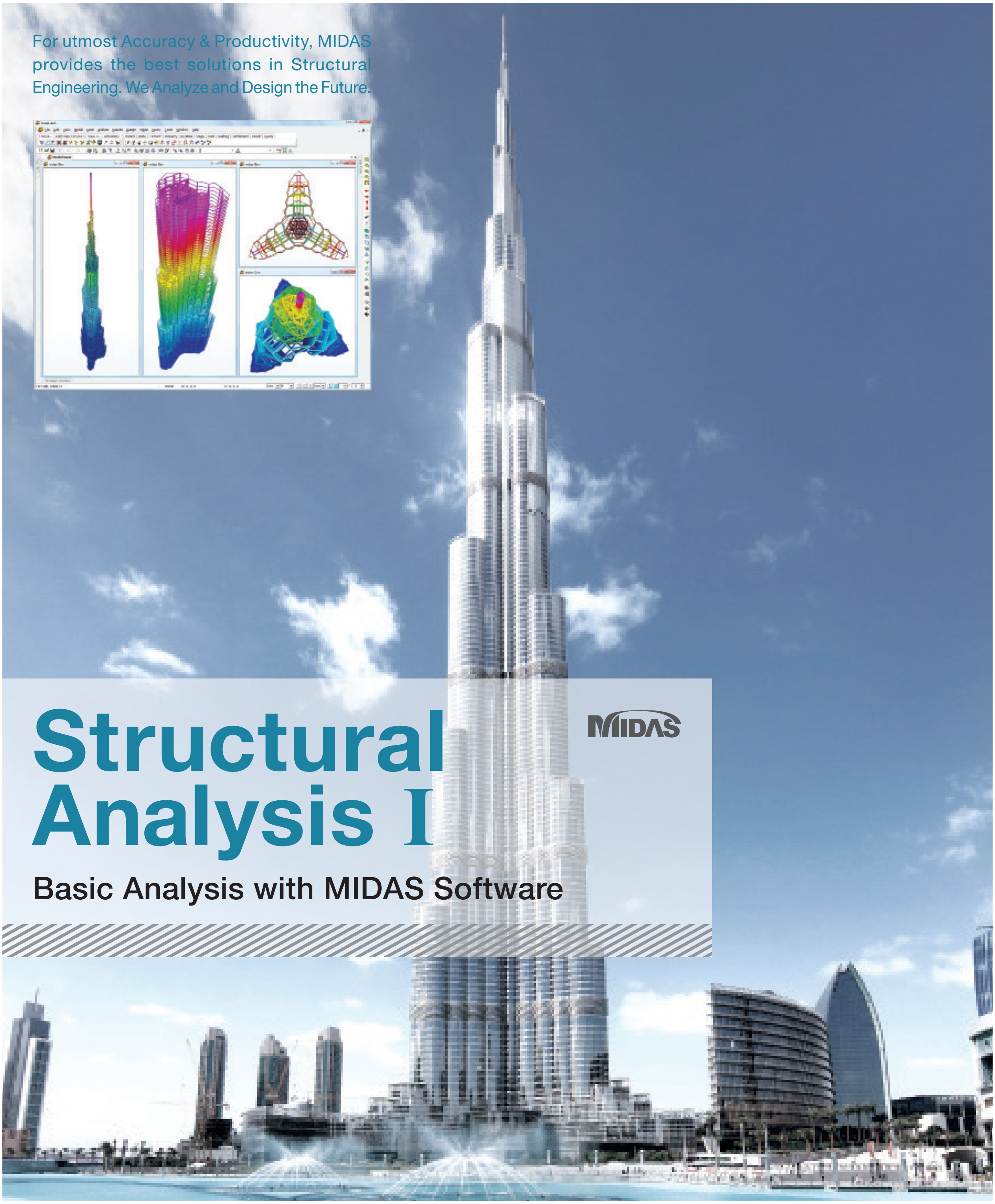
For utmost Accuracy & Productivity, MIDAS provides the best solutions in Structural Engineering. We Analyze and Design the Future.



Structural Analysis I

MIDAS

Basic Analysis with MIDAS Software

Structural Analysis I

Basic 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

Second reprint 2017

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

MIDAS
Modeling, Integrated Design & Analysis Software

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

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.

I

Basic Analysis

Index

1. **Simple Beam** Analysis

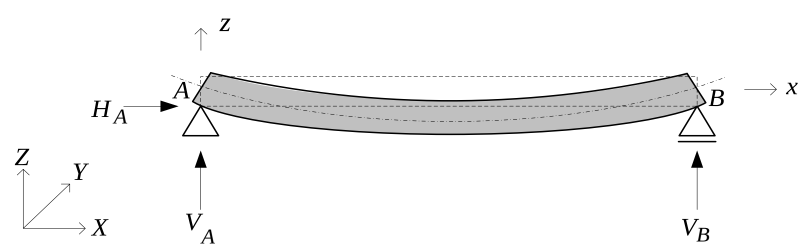
Contents

1. Introduction

1.1 Concept for Simple Beam Analysis

Fig. 1.1 shows a simply supported beam in X-Z plane (2-D) in Global Coordinate System (GCS). Element Coordinate System (ECS) x-axis is parallel with the span of the beam. ECS z-axis is as shown, and y-axis follows the right-hand rule.

▶Fig. 1.1
Simply supported beam



A hinge support exists at the left end, and a roller support exists at the right end. Each support is free to rotate about GCS Y-axis. Equilibrium conditions require that the sums of forces are zeros.

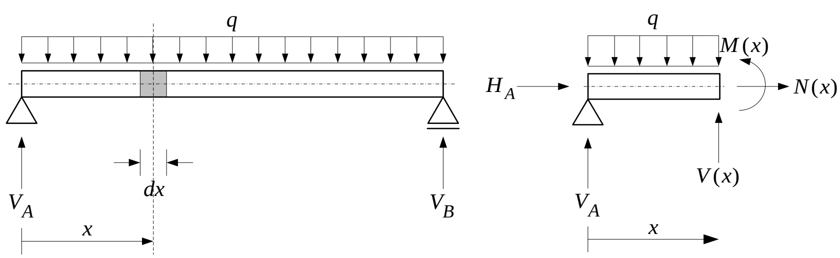
▶Eq. 1.1

$$\sum F_X = 0, \quad \sum F_Z = 0, \quad \sum M_Y = 0$$

A simply supported beam is statically determinate as the number of unknowns (H_A, V_A, V_B) equal to the number of equilibrium conditions (Eq. 1.1).

Once the reactions are obtained, section forces can be found at a distance, x, which consist of axial force ($N(x)$), shear force ($V(x)$) and moment ($M(x)$). The section forces are obtained from equilibrium in a free body diagram as shown in Fig. 1.2.

▶Fig. 1.2
Section forces

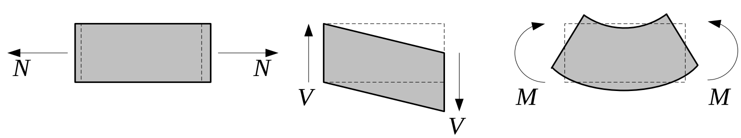


▶Eq. 1.2

$$\sum F_X = 0:\ H_A + N(x) = 0$$
$$\sum F_Z = 0:\ V_A + (-q)x + V(x) = 0$$
$$\sum M_Y = 0:\ V_A x + \frac{1}{2}(-q)x^2 + (-M(x)) = 0$$

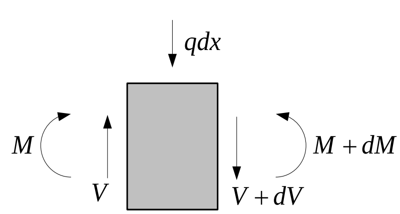
Fig. 1.3 represents the sign convention for + direction forces.

▶Fig. 1.3
Section forces



The infinitesimal length (dx) of the beam in Fig. 1.2 is isolated in Fig. 1.4 to represent an equilibrium condition in a free body diagram.

▶Fig. 1.4
External load, shear & moment equilibrium



▶Eq. 1.3

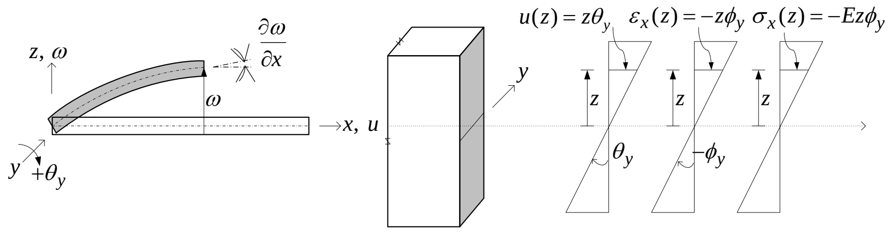
$$\frac{dV}{dx} = -q \quad \text{(or } V_B - V_A = \int_A^B -q\,dx \text{)}$$

▶Eq. 1.4

$$\frac{dM}{dx} = V \quad \text{(or } M_B - M_A = \int_A^B V\,dx \text{)}$$

Differentiating shear force becomes loading, and differentiating moment becomes shear force. This means that the difference in shear forces is the same as the area of the distributed load between two points. Also the difference in moments is the same as the area of shear force diagram between two points.
External forces cause deformation, represented as displacement angle (θ_y) and displacement (ω). The displacement angle at a point, x along the span is formed by the beam axes before and after deformation. The beam axis after deformation is the tangent (differentiation) at the point x.

▶Fig. 1.5
Section forces



▶Eq. 1.5

$$\text{Displacement angle: } \theta_y = -\frac{\partial w}{\partial x}$$

▶Eq. 1.6

$$\text{Curvature: } \phi_y = -\frac{\partial^2 w}{\partial x^2}$$

Displacement angle and displacement signify that axial deformation and shear deformation have taken place. For a beam member where the span is relatively longer than the section dimensions, the shear deformation is negligibly small.

In case of upward displacement in the $+z$ direction as shown in Fig. 1.5, the top of the section is in tension and the bottom of the section is in compression. Both tension and compression cause deformation perpendicular to the section in the x-direction, resulting in ($u(z)$) in Fig. 1.5. Thus strain distribution ($\varepsilon_x(z)$) in the axial direction is generated, and the curvature ($\emptyset_y$) of the section deformation is obtained. Differentiating the displacement results in the displacement angle, and differentiating the displacement angle becomes the curvature. Following the sign convention, the moment – curvature relationship and the moment – stress relationship can be derived from Fig. 1.5 as follows:

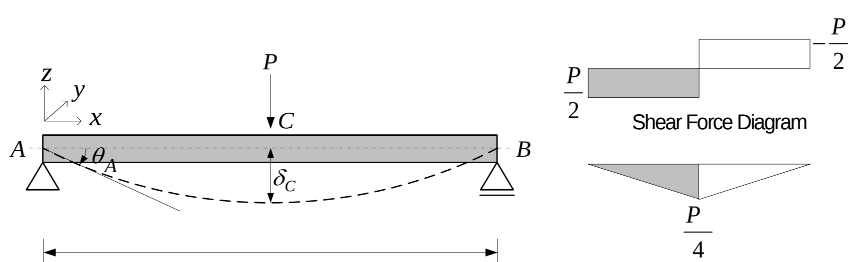
▶Eq. 1.6

$$\phi_y = \frac{M_y}{EI_y}, \quad \sigma_x(z) = -\frac{M_y}{I_y} z$$

The reverse of Eq. 1.6 also holds true. Integrating the curvature once becomes the displacement angle and integrating it once again results in displacement. Hence the displacement angle and displacement can be derived from the moment distribution.

Fig. 1.6 shows a simply supported beam subjected to a concentrated load at the mid span. The process of obtaining the displacement angle and displacement is reviewed.

▶Fig. 1.6
Simple beam subjected to a concentrated load



Difference in shear forces between Points A and C is the area of the distributed load. The fact that no distributed load exists between the 2 Points, shear force remains constant. Also the difference in moments between the 2 Points is the shaded area of the shear force diagram, hence the moment at Point C becomes $P1/4$. The displacement angle at Point A is the area of the curvature distribution (M_y / EI_y) between Points A and C. The displacement at Point C is the moment about Point A due to the area of the curvature distribution between Points A and C.

Eq. 1.6

$$\theta_A = \frac{1}{EI_y}\left(\frac{l}{2}\cdot\frac{Pl}{4}\right)\cdot\frac{1}{2} = \frac{Pl^2}{16EI_y}$$

$$\delta_C = \frac{Pl^2}{16EI_y}\cdot\frac{2}{3}\left(\frac{l}{2}\right) = \frac{Pl^3}{48EI_y}$$

The above process of integrating the curvature once to obtain the displacement angle and twice to obtain the displacement is referred to as the moment – area method.

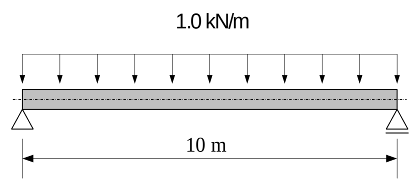
2. Tutorial

2.1 Model Overview

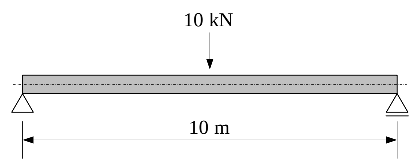
A beam is simply supported by a hinge at the left end and a roller at the right end. Deformation and member forces will be calculated using midas Gen and by hand.

▶Fig. 1.7
Analysis model

Model 1: uniformly distributed load



Model 2: concentrated load



- ***Material***

  Concrete: 25 MPa (M25) (Modulus of elasticity E = 2.5 x 10^7 kN/m^2)
- ***Section***

  Area: 0.8 m^2

  Moment of inertia (Iy): 4.266667×10^{-2} m^4
- ***Load***

  1. Model 1: uniformly distributed load 1.0 kN/m
  2. Model 2: concentrated load 10.0 kN

2.2 Work Environment

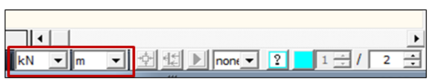
Open a New File and save.

Main Menu: */ New Project...*

Main Menu: */ Save As...*

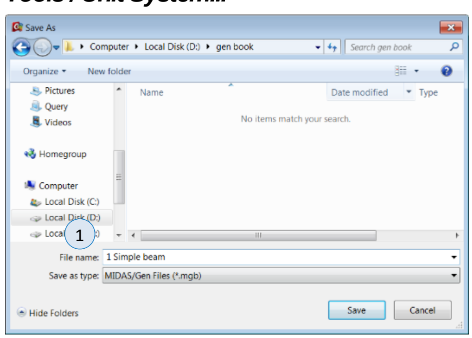
1. File name: **'Simple'** and [Save]

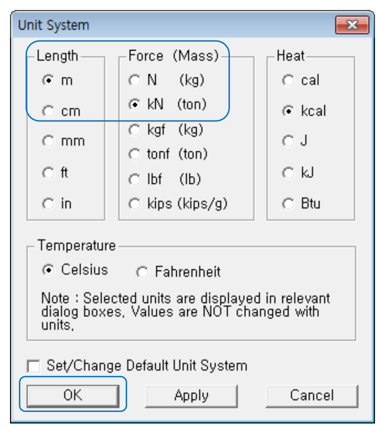
Set a unit system to kN & m at the bottom of work window.



Alternatively Main Menu:

Tools / Unit System...





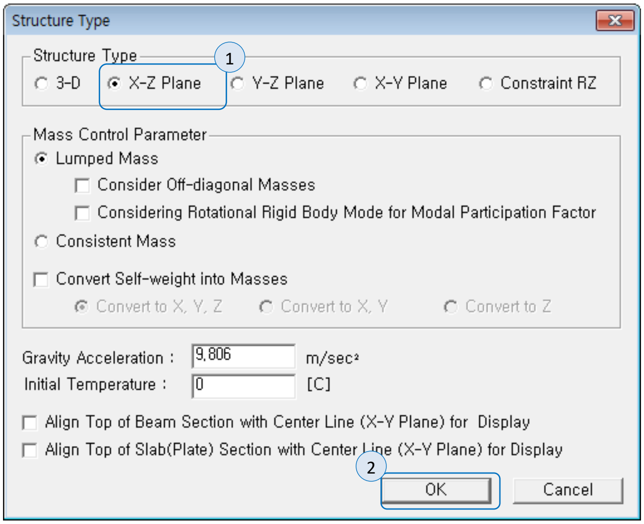
▶Fig. 1.8
Save file &
set unit system

midas Gen is 3-D software. Since the beam exists in a 2-D plane, X-Z plane in Global Coordinate System is set as the work plane, which restrains unnecessary degrees of freedom, D_Y (Y- displacement), R_X (rotation about X - axis) and R_Z (rotation about Z - axis).

Main Menu: ***Structure / Structure Type...***

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

▶Fig. 1.9
Set a work plane



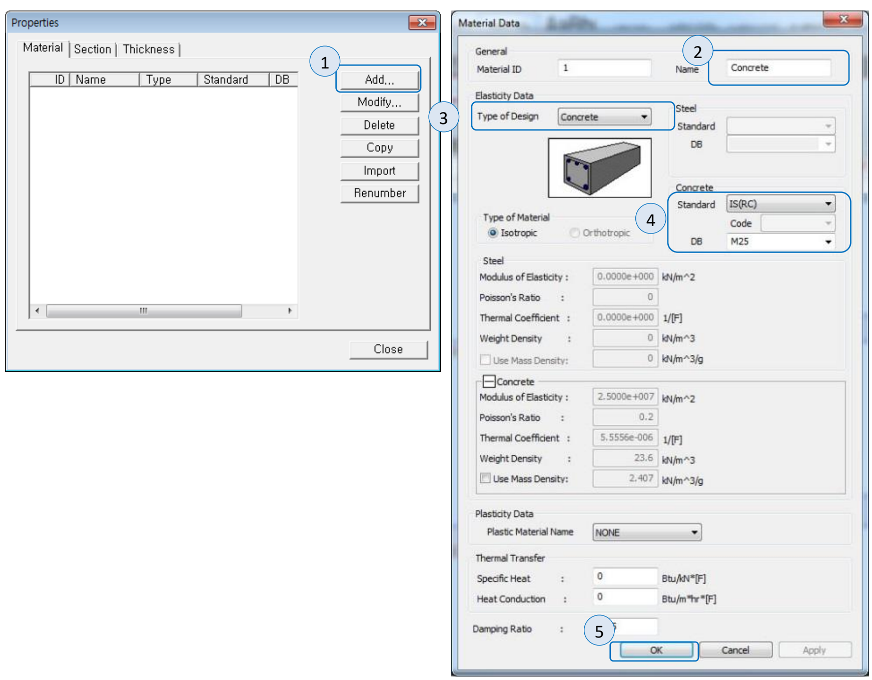
2.3 Material & Section Properties

midas Gen contains a wide range of steel and concrete materials. Material Data can be defined using Standard and DB or user-defined.

Main Menu: ***Properties / Material Properties***

1. **[Add...]**
2. Name: **'Concrete'**
3. Elasticity Data > Type of Design > **Concrete**
4. Concrete > Standard > **IS(RC)**, DB > **M25**
5. **[OK]**

▶Fig. 1.10
Define Material Data



Tip

Properties can be directly input if Type of Design is set to User Defined.

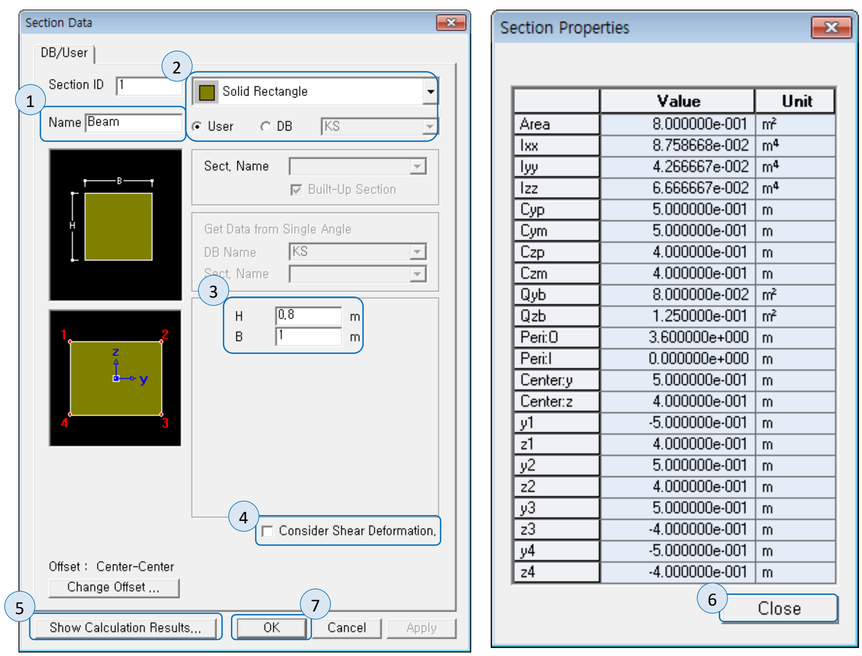
Tip

OK button closes the dialog box. If multiple materials are defined, use Apply button.

midas Gen automatically calculates the section properties based on section dimensions.

Main Menu: ***Properties / Section Properties,*** [Add...],
1. Name: 'Beam'
2. Section shape list > **Solid Rectangle**, Section definition method > **User**
3. H: **'0.8'**, B: **' 1'**
4. Ignore shear deformation: **Consider Shear Deformation** (Check off)
5. **[Show Calculation Results...]**
6. Check Section Properties, **[Close]**
7. **[OK]**

▶Fig. 1.11
Define Section



Area: cross sectional area
Asy: local y - direction shear area
Asz: local z - direction shear area

Ixx: torsional constant
Iyy: moment of inertia about local y – axis
Izz: moment of inertia about local z – axis

Tip

Consider Shear Deformation is generally checked on for structural analysis. However, general structural mechanics hand calculation ignores shear deformation, which is the reason for checking off in this case for comparison purposes.

Tip

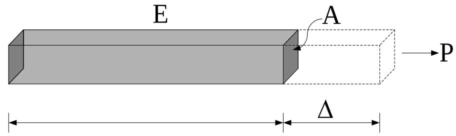
[Show Calculation Results...] button invokes the automatically calculated Section Properties.

Tip

Material and Section

When the load, P is applied in the axial direction of a beam, $\Delta\lambda$ takes place, which decreases with large magnitudes of modulus of elasticity (E) and cross sectional area (A).

▶Fig. 1.12
Axial deformation

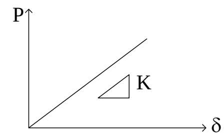


This can be expressed as,

$$\Delta L = \frac{Pl}{EA} \quad \rightarrow \quad P = \frac{EA}{l} \cdot \Delta l$$

This can be also expressed in the form of load – displacement graph, where the slope (κ) is the stiffness of the structure, {D} = displacement, and {P} = applied load.

$$\{P\} = [K] \cdot \{D\}$$



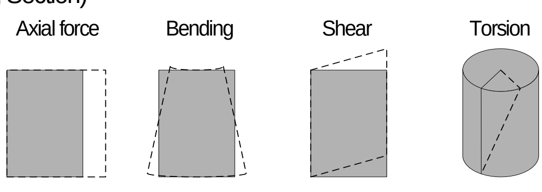
When a program is used for structural analysis, results (forces, stresses, etc.) can be obtained from displacement.

Structural analysis is a process in which loading and stiffness are defined to obtain the unknown displacement. In midas Gen, loading is defined under the [Load] menu, and stiffness is defined under the [Model] menu.

In the equation expressing the stiffness, [K], the numerator, {EA} is said to be Rigidity, which can be calculated by the material and section properties defined earlier. Section stiffness is automatically calculated, which includes axial stiffness, bending stiffness, shear stiffness, torsional stiffness, etc. All the parameters defining the Material and Section can be user – defined.

Rigidity (Material, Section)

▶Fig. 1.13
Section stiffness



2.4 Nodes & Elements Generation

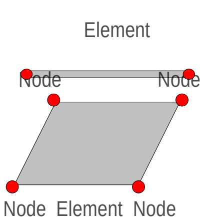
Tip

Finite element software like midas Gen models a structure with elements connected by nodes.

midas Gen provides numerous element types, which include Truss element resisting only axial forces, Beam element resisting axial, shear and moment, and Plate element representing slabs and walls.

Appropriate element types need to be used to reflect the structural behavior. The smaller the element sizes, the greater is the accuracy of results.

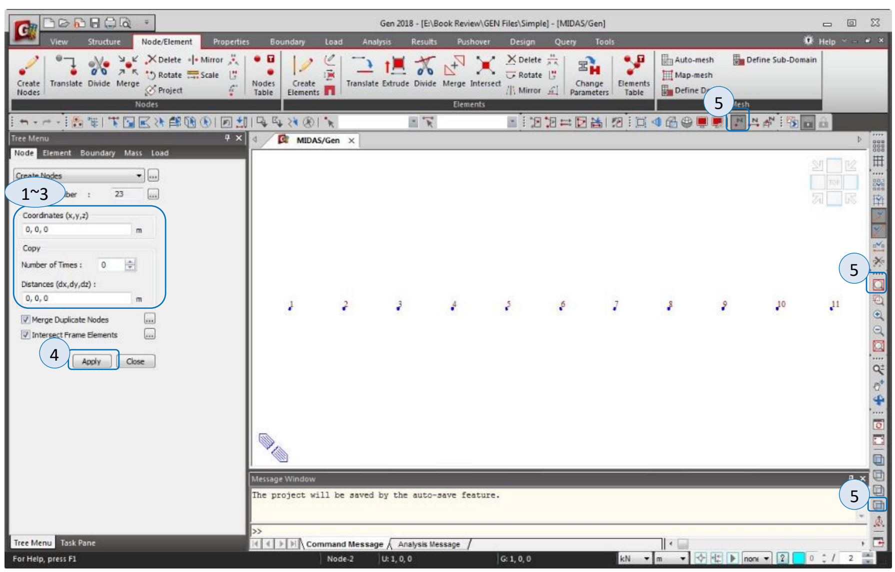
▶Fig. 1.14
Create nodes



Create nodes where elements will be created.

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

1. Coordinates (x, y, z): **‘0, 0, 0’**
2. To copy, Number of Times: **‘10’**
3. Distances (dx, dy, dz): **‘1, 0, 0’**
4. **[Apply]**
5. **Display Node Numbers, Auto Fitting, Front View** (Toggle on)

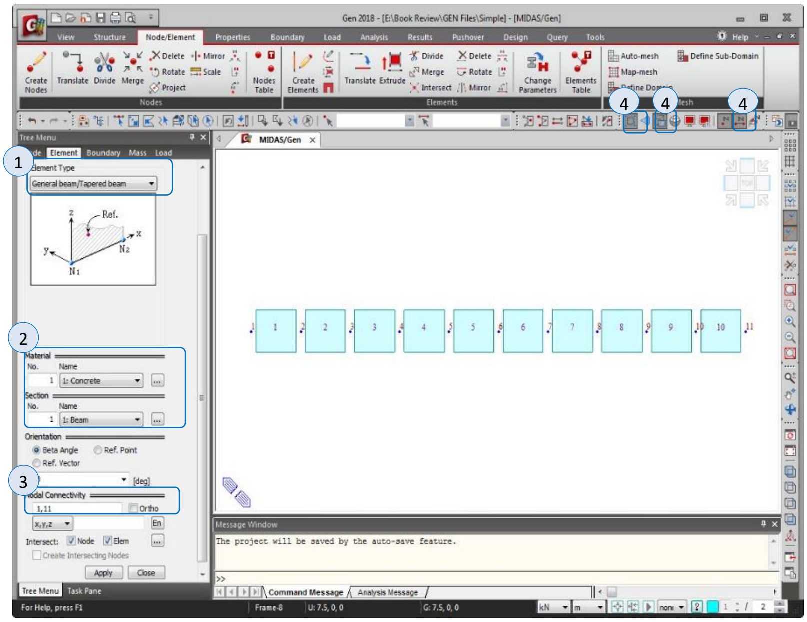


Connect nodes to create elements.

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

1. Element Type > **General beam/Tapered beam**
2. Material > **1:Concrete**, Section > **1:Beam**
3. Click the Nodal Connectivity field (which turns to green), followed by clicking **Node 1 and 11** on the model view sequentially.
4. **Shrink, Hidden, Display Element Numbers** (Toggle on)

▶Fig. 1.15
Create elements



Tip

Shrink View is used to identify the connectivity of the elements at nodes. The element separation in display confirms that the elements are connected to the nodes.

Tip

Convenient option for creating elements

Node Snap: tracks the nodes when the mouse is placed close to the nodes on Model View.

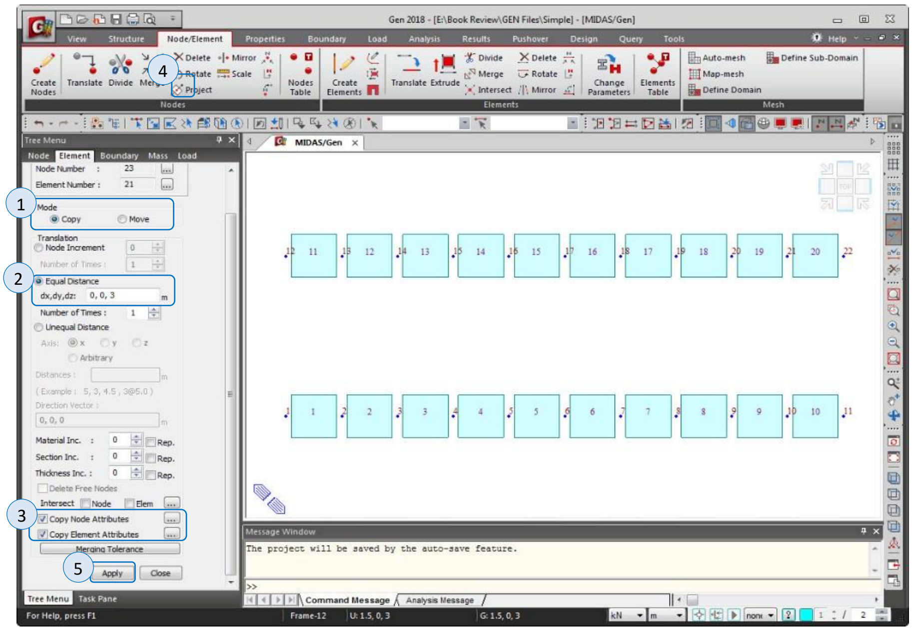
Intersect Node: automatically divides the elements if nodes are present at the locations of the elements being created.

Copy the elements created.

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

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

▶Fig. 1.16
Copy elements



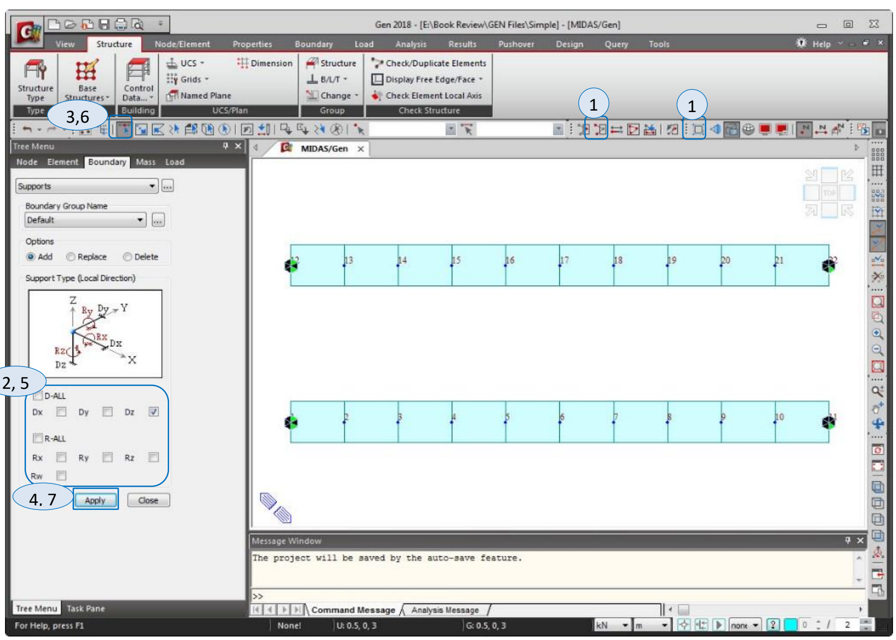
2.5 Define Boundary Conditions

Define support conditions, a hinge at the left end and a roller at the right end.

Main Menu: ***Boundary / Define Supports***

1. **Shrink, Display Element Numbers** (Toggle off)
2. Support Type > **Dx, Dz** (Check on)
3. **Select by Window** (on) mode: select **Nodes 1, 12**
4. **[Apply]**
5. Support Type > **Dx** (Check off)
6. **Select by Window** (on) mode: select **Nodes 11, 22**
7. **[Apply] [Close]**

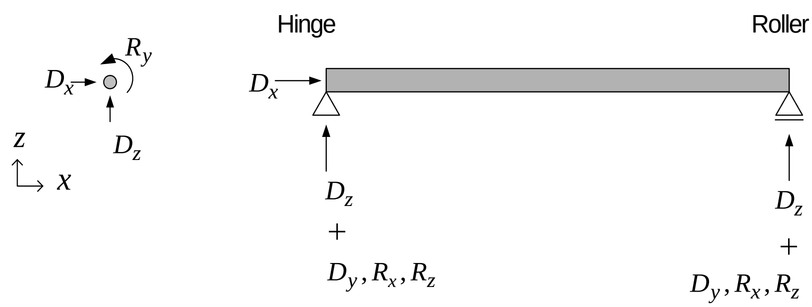
▶Fig. 1.17
Input support conditions



Tip

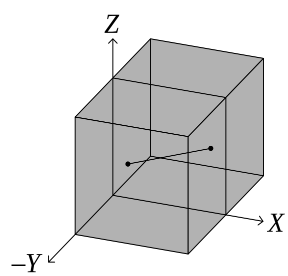
Degrees of freedom in 3-D

▶Fig. 1.18
Simple beam reactions & degrees of freedom (local axes)

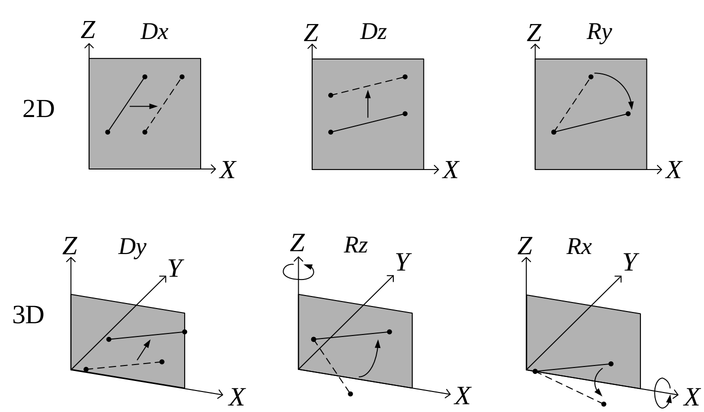


6 Degrees of freedom in 3-D are shown below.

▶Fig. 1.19
3-D shape



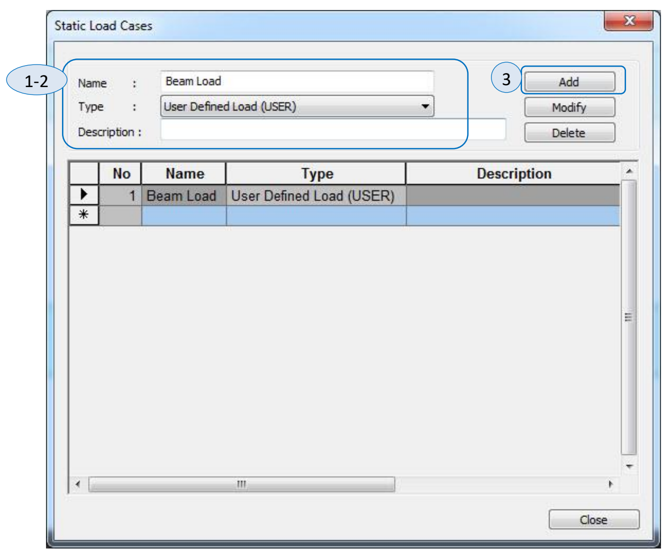
▶Fig. 1.20
Degrees of freedom in 3-D (global axes)



2.6 Define Loads

Define the load cases (load types) first to which individual loads will belong.

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Beam Load'**
2. Type > **User Defined Load (USER)**
3. **[Add] [Close]**



▶Fig. 1.21
Define load case

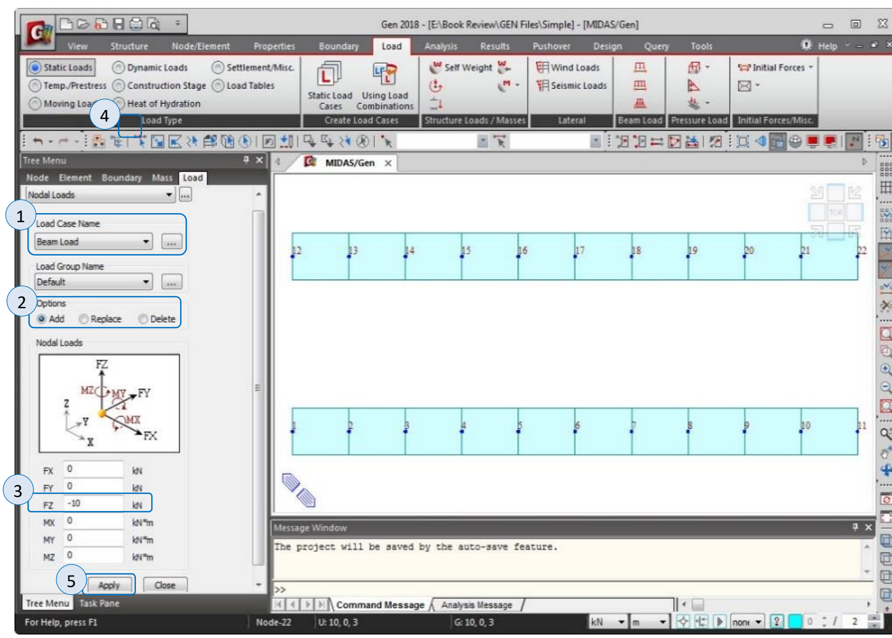
Tip

Typically in practice, a number of Load Cases (Types) are defined such as Dead Load, Live Load, etc., which will be combined with relevant load factors to satisfy a specific design code.

Input the concentrated load.

Main Menu: ***Load / Static Loads / Nodal Loads***
1. Load Case Name > **Beam Load**
2. Options > **Add**
3. Nodal Loads > FZ: **'-10'**
4. **Select by Window** (on) mode: select **Node 6**
5. **[Apply] [Close]**

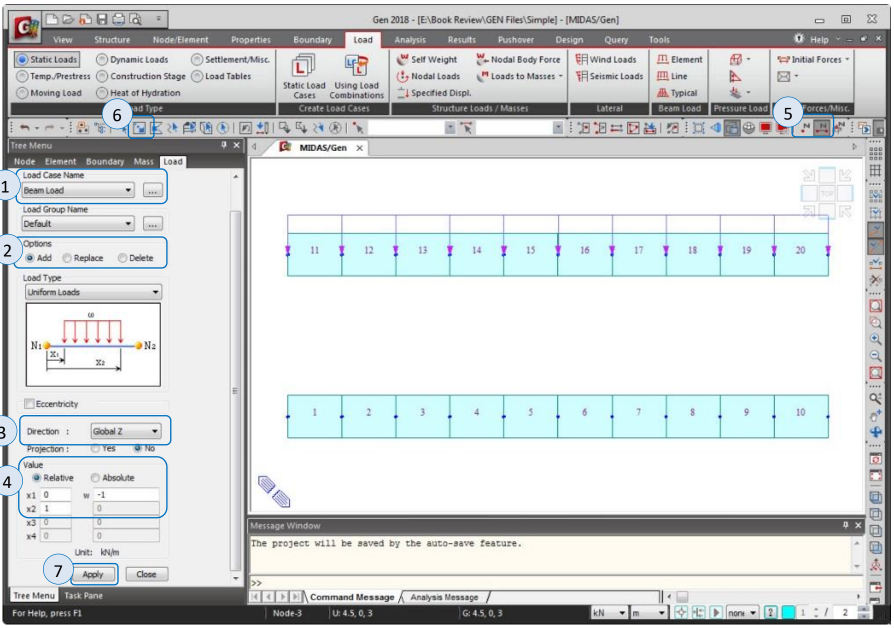
▶Fig. 1.22
Input the concentrated load



Input the distributed load.

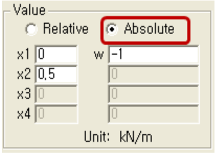
Main Menu: ***Load / Static Loads / Element (Beam Load)***
1. Load Case Name > **Beam Load**
2. Load Type > **Uniform Loads**
3. Direction > **Global Z**
4. Value > x1: **'0'**, w: **'-1'**, x2: **'1'**
5. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
6. **Select by Window** (on) mode: select Elements **11-20**
7. **[Apply] [Close]**

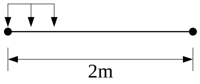
▶Fig. 1.23
Define the distributed load



Tip

Absolute: Extent of uniform load is applied in actual distance, x1, x2.
(total length is 2m)

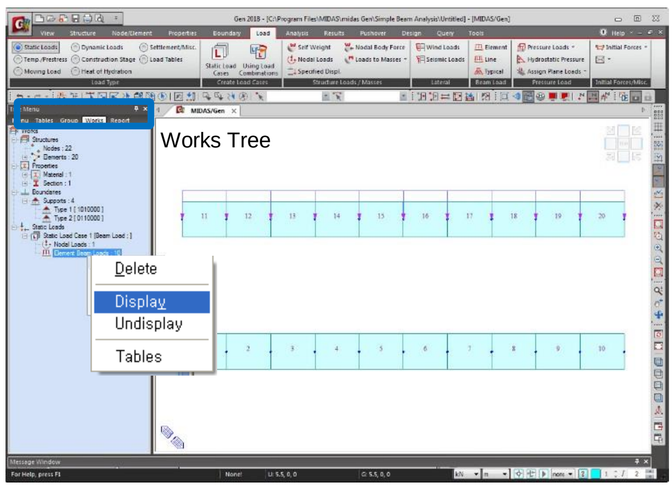




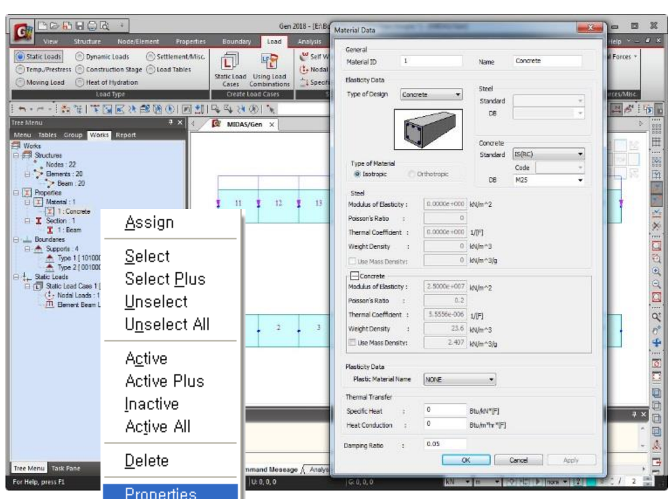
2.7 Perform Analysis

Check the modeling data using Works Tree. Right-click the relevant list form Works Tree to invoke Context Menu to check and revise the input data.

▶Fig. 1.24
Check loadings



▶Fig. 1.25
Check material data

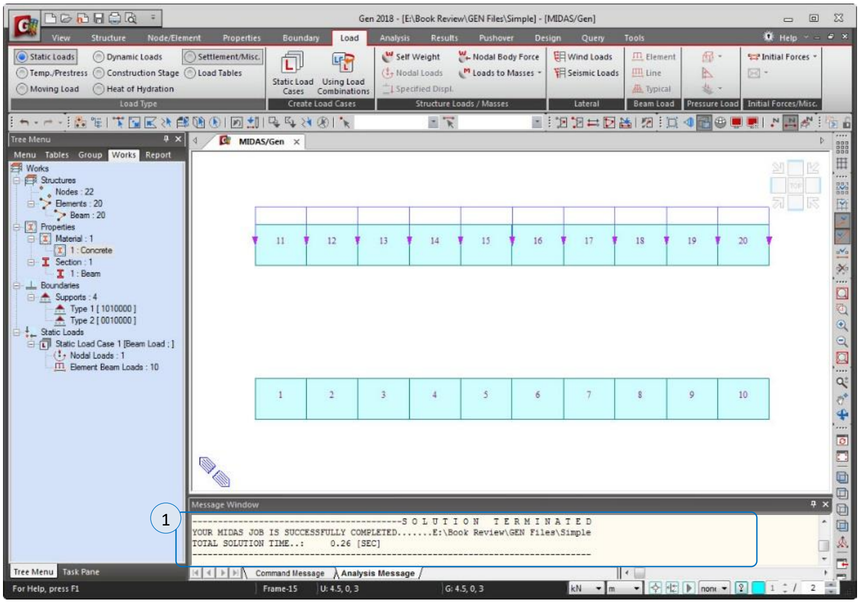


Having checked the model, loadings and boundary conditions, analysis can be performed.

Main Menu: ***Analysis / Perform Analysis***

1. Check for successful analysis in Message Window.

▶Fig. 1.26
Message for a successful run



Tip

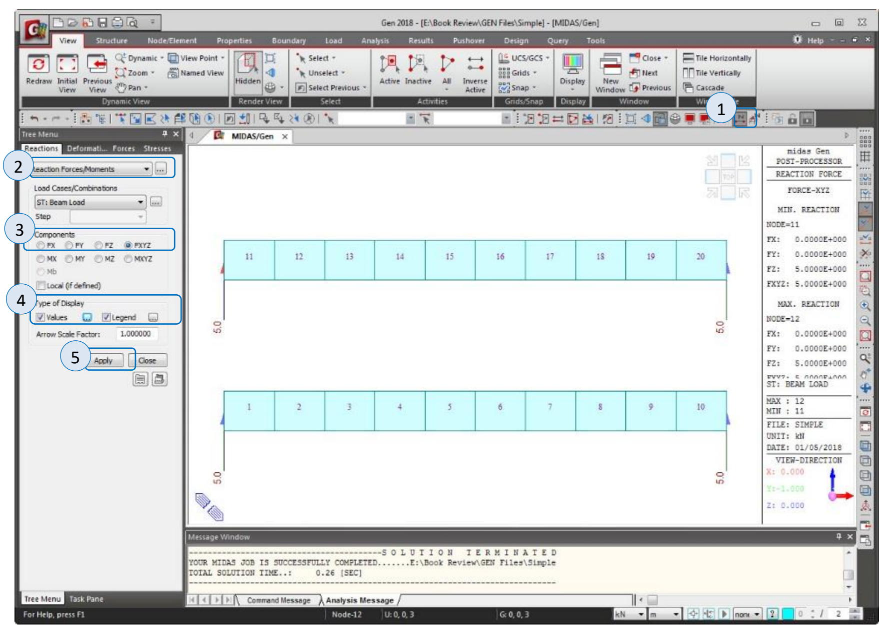
Shortcut key for Perform Analysis is F5.

2.8 Check Analysis Results

Check the reactions from midas Gen and compare them with hand calculations.

Main Menu: ***Results / Reactions / Reaction Forces/Moments…***

1. **Display Element Number** (Toggle off)
2. Load Cases / Combinations > **ST: Beam Load**
3. Components > **FXYZ**
4. Type of Display > **Values, Legend** (Check on)
5. **[Apply] [Close]**



▶Fig. 1.27
Reaction results

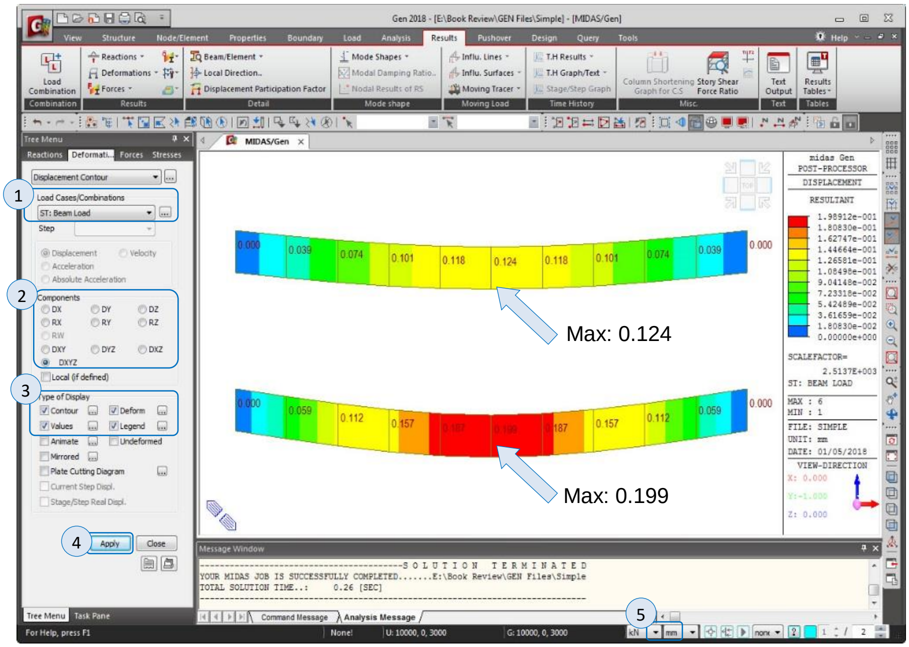
Reactions from midas Gen, 5 kN are the same as those from hand calculations.

Check displacements from midas Gen and compare them with hand calculations.

Main Menu: ***Results / Deformations / Displacement Contour…***

1. Load Cases/Combinations > **ST: Beam Load**
2. Components > **DXYZ**
3. Type of Display > **Contour, Deform, Values, Legend** (Check on)
4. **[Apply]**
5. Change the unit to mm in Status Bar at the bottom of the work screen.

▶Fig. 1.28 Displacements



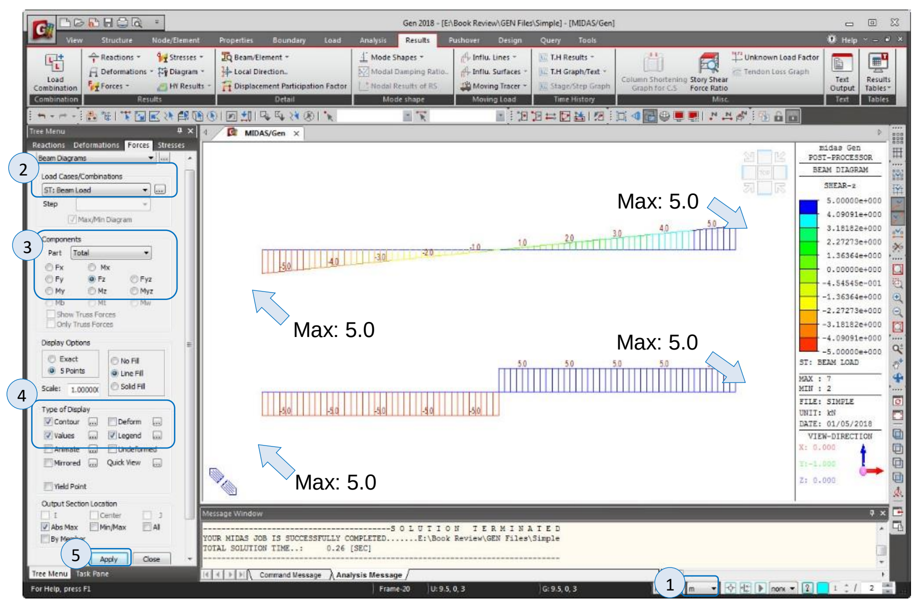
Hand calculations also show the same results (distributed load model: 0.124 mm, concentrated load model: 0.199 mm).

Check shear diagram and moment diagram.

Main Menu: ***Results / Forces / Beam Diagrams...***

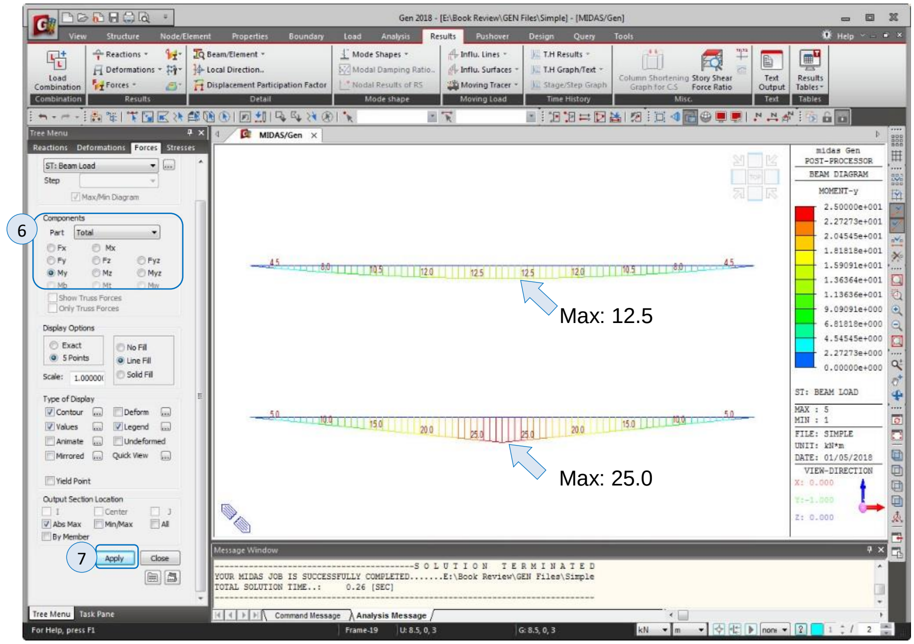
1. Change the unit to m in Status Bar at the bottom of the work screen.
2. Load Cases/Combinations > **ST: Beam Load**
3. Components > **Fz**
4. Type of Display > **Contour, Value, Legend** (Check on), **Deform** (Check off)
5. **[Apply]**
6. Components > **My**
7. **[Apply]**

▶Fig. 1.29
Shear force (Fz)



Hand calculations also show the maximum value of 5 kN.

▶Fig. 1.30
Moment (My)

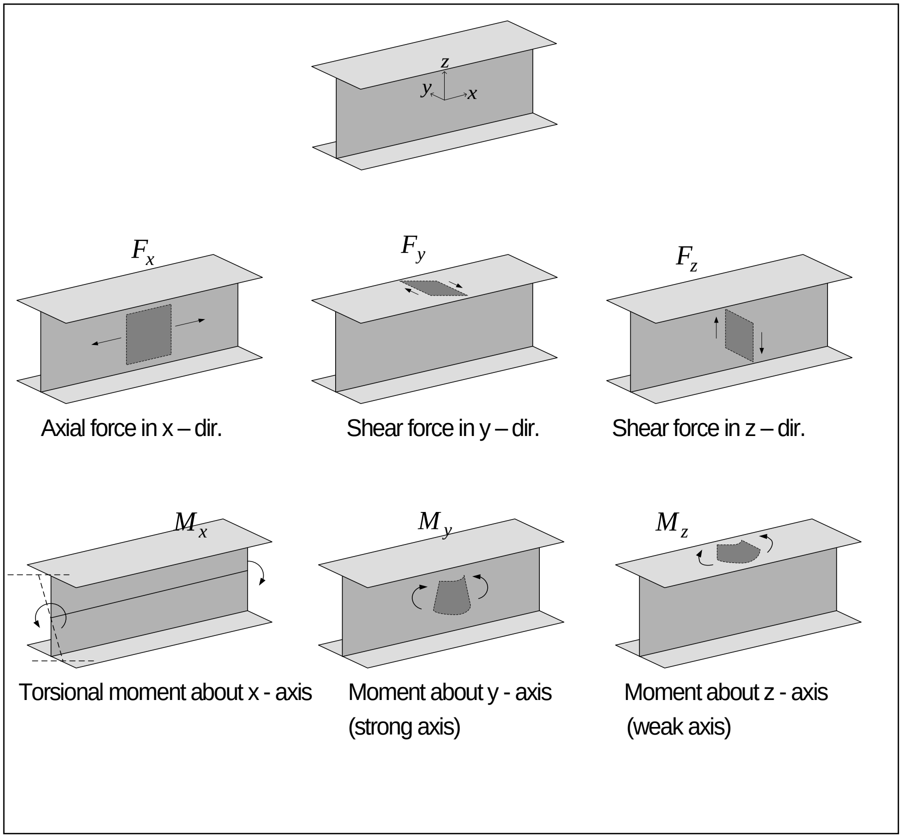


Hand calculations also show the same results (distributed load model: 12.5 kN·m, concentrated load model: 25 kN·m).

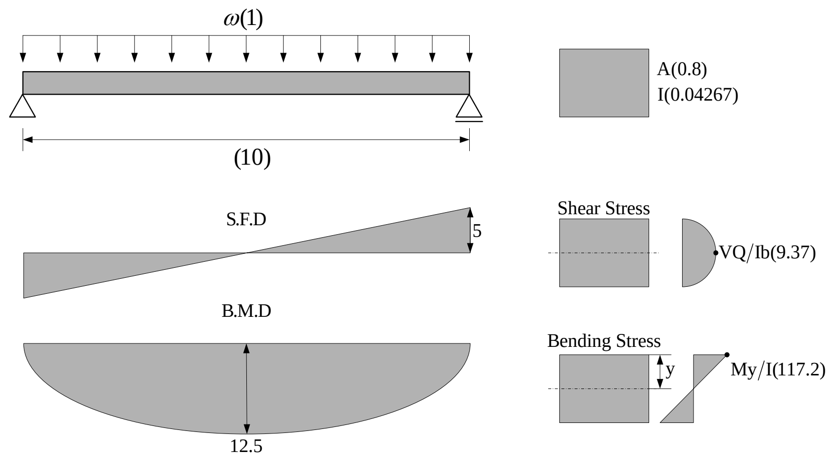
Tip

Force Components

▶Fig. 1.31
Force components



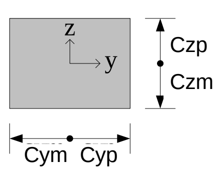
Calculate the shear stress at the maximum shear at the supports and flexural stress at the maximum moment at the mid span.

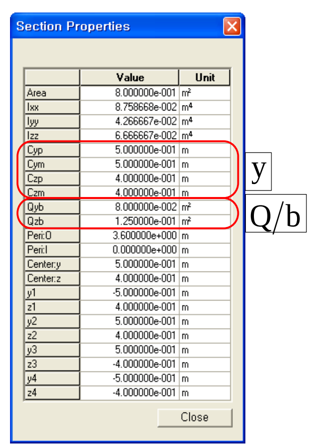


▶Fig. 1.32
Stress calculations

Tip

Sax represents σAxial.





	Value	Unit
Area	8.000000e-001	m²
Ixx	8.758668e-002	m⁴
Iyy	4.266667e-002	m⁴
Izz	6.666667e-002	m⁴
Cyp	5.000000e-001	m
Cym	5.000000e-001	m
Czp	4.000000e-001	m
Czm	4.000000e-001	m
Qyb	8.000000e-002	m²
Qzb	1.250000e-001	m²
Peri:O	3.600000e+000	m
Peri:I	0.000000e+000	m
Center:y	5.000000e-001	m
Center:z	4.000000e-001	m
y1	-5.000000e-001	m
z1	4.000000e-001	m
y2	5.000000e-001	m
z2	4.000000e-001	m
y3	5.000000e-001	m
z3	-4.000000e-001	m
y4	-5.000000e-001	m
z4	-4.000000e-001	m

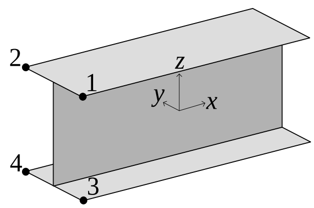
Stresses vary with the locations within the section.
midas Gen produces stress at a point. The stress output locations can be adjusted by inputting C (Cyp, Cym, Czp, Czm) and y, z (y1-4, z1-4).

Tip

Stress Components

For a given Element Coordinate System as below, the stress sign convention and calculations are as follows:

where, Fx, Fy, Fz: axial force in x – dir., shear in y – dir., shear in z – dir.
Iyy, Izz: moment of inertia about y – axis and z - axis
Qyb, Qzb: shear coefficient for y – axis and z - axis
My, Mz: moment about y – axis and z - axis
Cy, Cz: extreme fiber distance in y – dir. and z – dir.



▶Fig. 1.33
Stress components

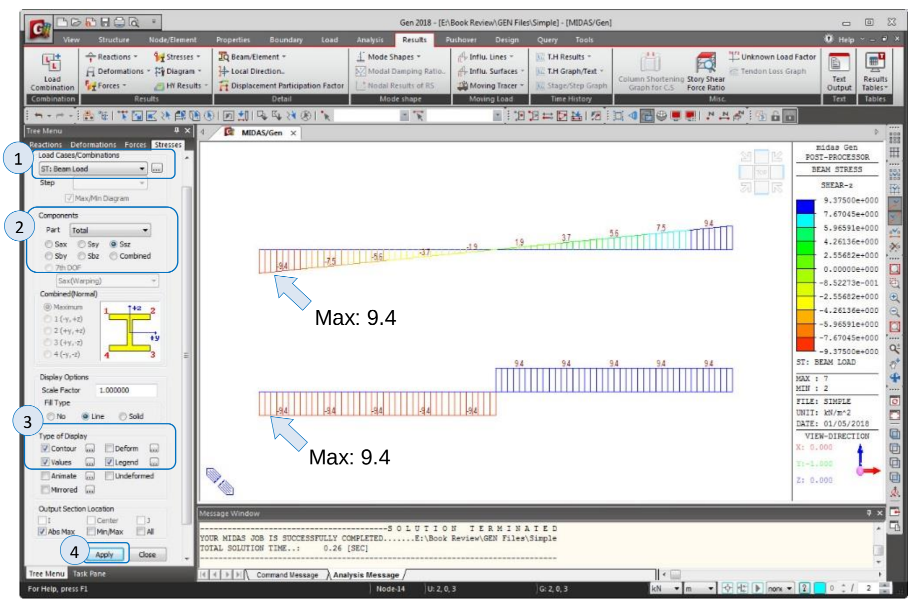
Sax	axial stress	$\frac{P}{A}$	$\frac{Fx}{Area}$
Ssy Ssz	shear stress in weak axis shear stress in strong axis	$\frac{VQ}{Ib}$	$\frac{Fy}{Izz}Qzb$ $\frac{Fz}{Iyy}Qyb$
Sby Sbz	moment about weak axis moment about strong axis	$\frac{My}{I}$	$\frac{Mz}{Izz}\cdot Cy$ (Max(Cyp, Cym)) $\frac{My}{Iyy}\cdot Cz$
Combined	combined stress	$\frac{P}{A}+\frac{My}{I}$	$\frac{Fx}{Area}+\frac{Mz}{Izz}\cdot y+\frac{My}{Iyy}\cdot z$ (y1 ~ y4)

Check shear and flexural stresses due to the distributed load.

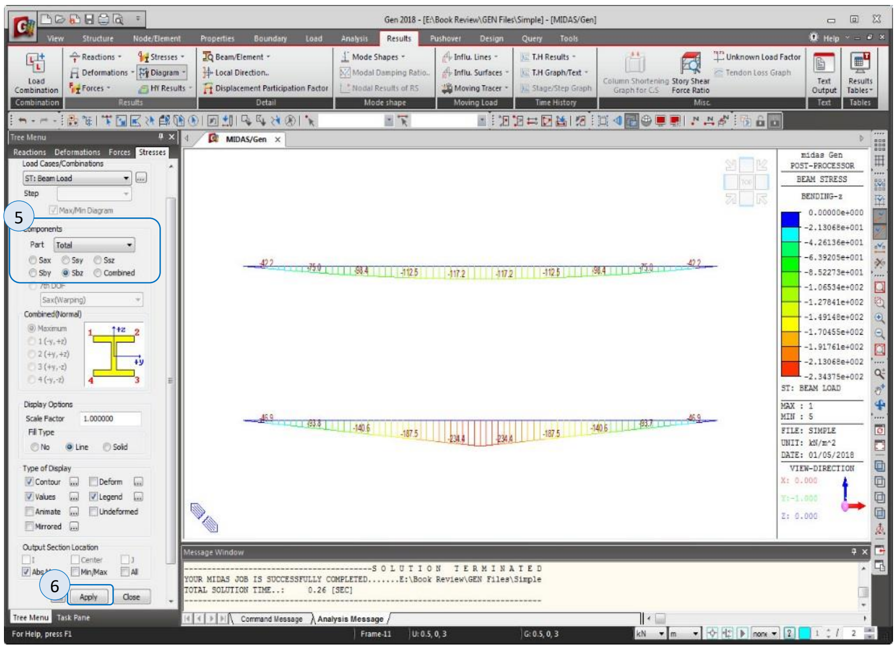
Main Menu: ***Results / Stresses / Beam Stresses Diagram...***

1. Load Cases/Combinations > **ST: Beam Load**
2. Components > **Ssz**
3. Type of Display > **Contour, Values, Legend** (Check on)
4. **[Apply]**
5. Components > **Sbz**
6. **[Apply]**

▶Fig. 1.34
Shear stress



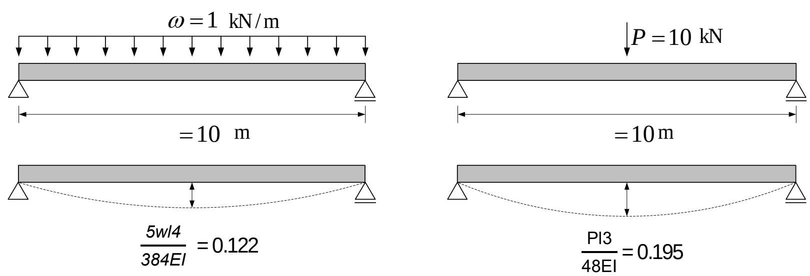
▶Fig. 1.35
Flexural stress



3. Manual Calculations

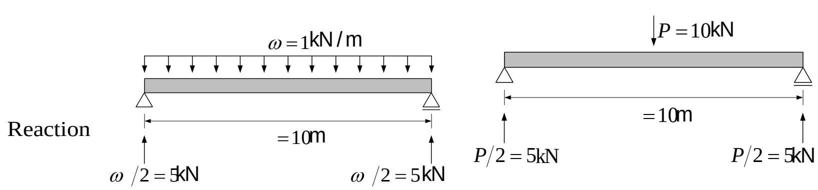
Displacements for the uniformly distributed load and the concentrated load are the same as those from midas Gen (or midas Civil).
(unit: kN, m with shear deformation ignored)

▶Fig. 1.36
Displacement results

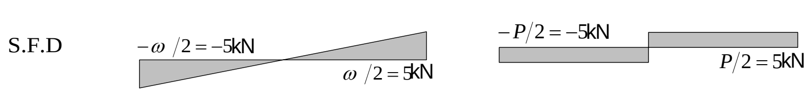


Member forces for the 2 loading conditions are also in agreement with those from midas Gen or midas Civil.

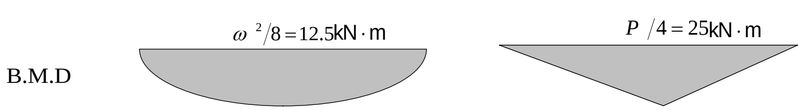
▶Fig. 1.37
Reactions



▶Fig. 1.38
Shear forces



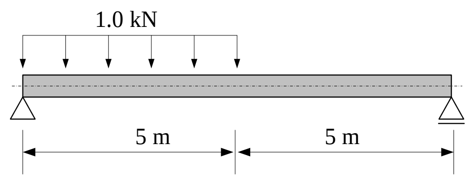
▶Fig. 1.39
Bending moments



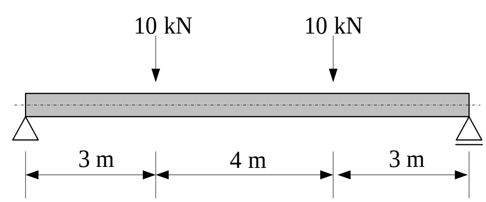
4. Exercise

The following 2 beams are simply supported with a roller support on the left and a roller support on the right. Find the displacements, reactions, shear forces and bending moments for each beam (ignore the shear deformation).

Model 1



Model 2



- ***Material***

  Concrete: 25 MPa (M25) (Modulus of elasticity E = 25 kN/m^2)

- ***Section***

  Area: 3.0×10^{-1} m^2 (400×750)

  Moment of inertia (I_y): 1.40625×10^{-2} m^4

- ***Load***

  1. Model 1: partial uniformly distributed load 1.0 kN/m
  2. Model 2: concentrated loads 10.0 kN at 2 points

2. Cantilever Beam Analysis

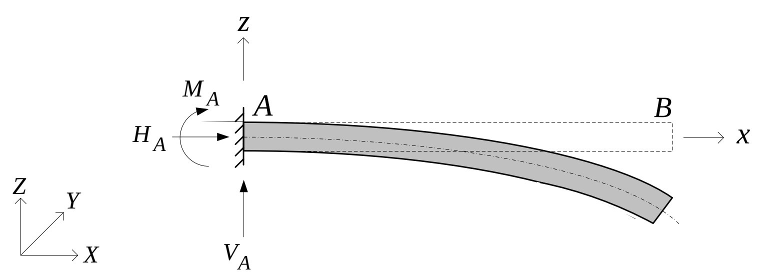
Contents

1. Introduction

1.1 Concept for Cantilever Beam Analysis

A cantilever beam is a statically determinate structure. Fig. 2.1 shows a cantilever beam on Global Coordinate System (GCS) X-Z plane, which is restrained at the left end, A with horizontal and vertical translations and rotation.

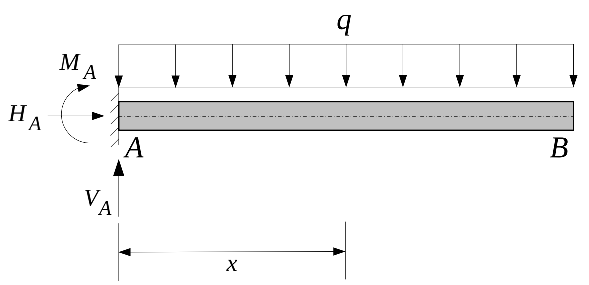
▶Fig. 2.1
Cantilever beam



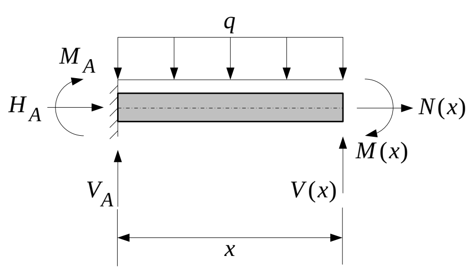
Once the reactions of the beam are obtained, section forces at a specific location, x, along the span can be found by equilibrium. 3 Force components, axial force ($N(x)$), shear force ($V(x)$) and moment ($M(x)$) can be found from a free body diagram as shown in Fig. 2.2(b) satisfying equilibrium.

▶Fig 2.2
Section force

(a) Cantilever beam



(b) Free body diagram



▶Eq. 2.1

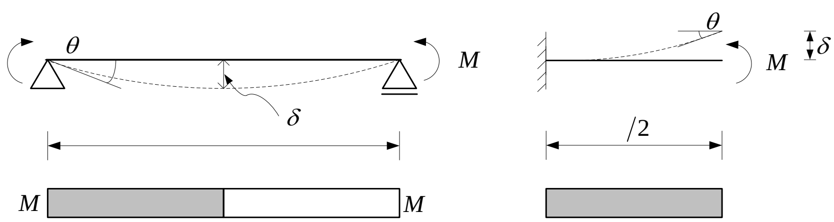
$$\sum F_X = 0: \ H_A + N(x) = 0$$
$$\sum F_Z = 0: \ V_A + (-)qx + V(x) = 0$$
$$\sum M_Y = 0: \ M_A + V_A x + \frac{1}{2}(-q)x^2 + M(x) = 0$$

The relationship of loading-shear force-moment and the relationship of deformation-strain-stress at a particular section of a cantilever beam are identical to those of a simply supported beam.

In order to demonstrate the relationship of displacement-displacement angle-curvature of a cantilever, a simple beam with the span of l and a cantilever beam with the span of $l/2$ are examined. Both cases are subject to a constant moment. Provided that the modulus of elasticity and the section dimensions are constant, the curvature remains constant. In this case, the mid span displacement and the end displacement angle of the simple beam are identical to the displacement and displacement angle at the free end of the cantilever beam respectively.

The same magnitudes are caused by the fact that the shaded areas of the curvature, M_y/EI_y are identical in both beams. The displacement angle is the area of the shaded part, and the displacement corresponds to the moment of the shaded area.

▶Fig. 2.3
Simple beam & cantilever beam



▶Eq. 2.2

$$\theta = \frac{M}{EI_y}\left(\frac{l}{2}\right) = \frac{Ml}{2EI_y}, \quad \delta = \frac{M}{EI_y}\left(\frac{l}{2}\right)\left(\frac{l}{4}\right) = \frac{Ml^2}{8EI_y}$$

For simplicity, signs for the displacement angle and displacement are omitted.

The same deformation value for the simple beam and the cantilever beam can be also identified by the deformed shapes. The deformed shape of the right half span of the simple beam is identical to that of the cantilever beam. The tangent of the deflection at the mid span of the simple beam is zero, which exhibits the effect of a fixed end.

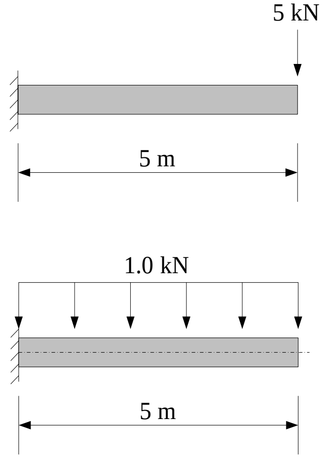
2. Tutorial

2.1 Model Overview

The following cantilever beam is subject to a concentrated load and a uniformly distributed load. Reactions, displacements and member forces will be calculated.

Model 1: concentrated load

Model 2: uniformly distributed load



▶Fig. 2.4
Analysis models

- ***Material***

  Steel: Fe440 (Modulus of elasticity, $E = 2.05 \times 10^8$ kN/m^2)
- ***Section***

  Area: 6.671×10^{-3} m^2 (ISMB 350)

  Moment of inertia (I_y): 1.363030×10^{-4} m^4
- ***Load***

  Model 1: concentrated load 10 kN

  Model 2: uniformly distributed load 1.0 kN/m

2.2 Work Environment

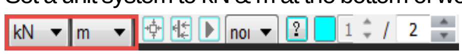
Open a New File and save.

Main Menu: ***/ New Project...***

Main Menu: ***/ Save As...***

1. File name: **'Cantilever'** and [Save]

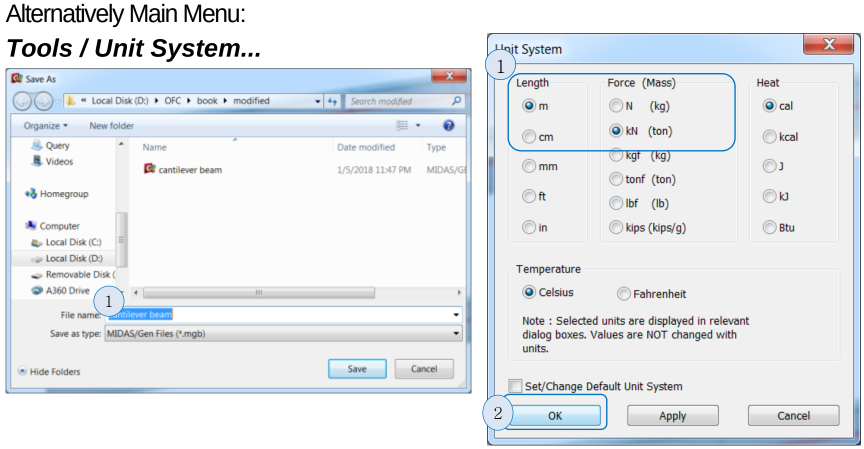
Set a unit system to kN & m at the bottom of work window.



Alternatively Main Menu:

Tools / Unit System...

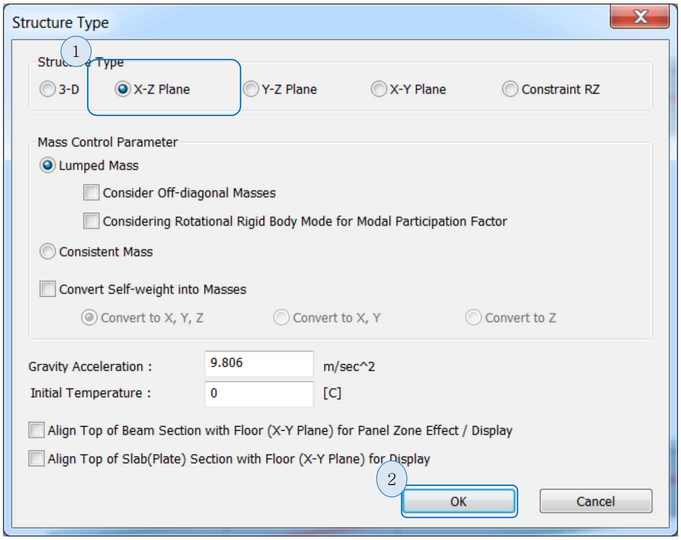
▶Fig. 2.5
Save file &
set unit system



midas Gen is 3-D software. Since the beam exists in a 2-D plane, X-Z plane in Global Coordinate System is set as the work plane, which restrains unnecessary degrees of freedom, D_Y (Y- displacement), R_X (rotation about X - axis) and R_Z (rotation about Z - axis).

Main Menu: ***Structure / Structure Type***
1. Structure Type > **X-Z Plane**
2. **[OK]**

▶Fig. 2.6
Set a work plane



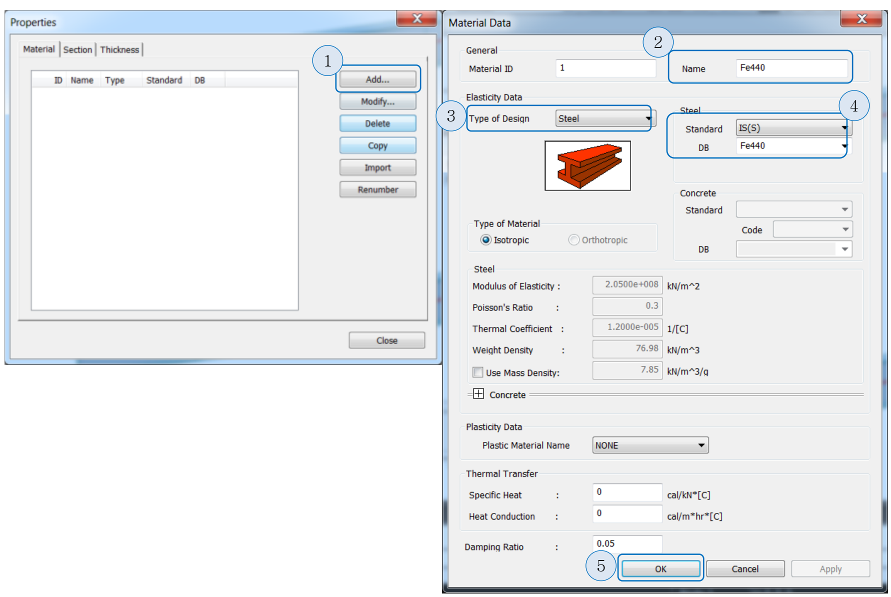
2.3 Material & Section Properties

Select Fe440 from DB.

Main Menu: ***Properties / Material Properties***

1. **[Add...]**
2. Name: **'Fe440'**
3. Type of Design > **Steel**
4. Steel > Standard > **IS(S)**, DB > **Fe440**
5. **[OK]**

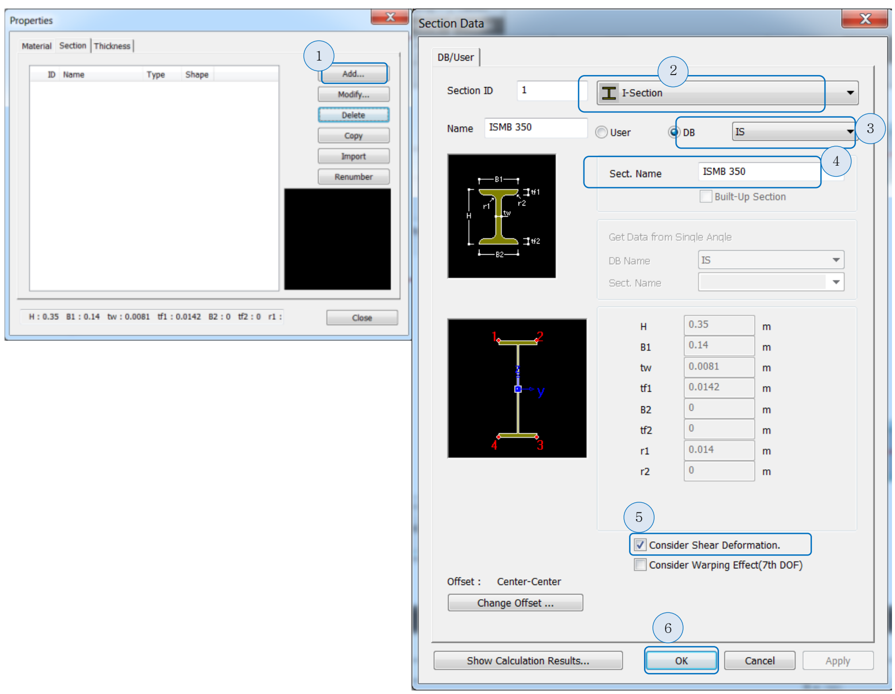
▶Fig. 2.7
Define Material Data



Use IS355 Click Section tab.

1. **[Add...]**
2. Section shape list > **I-Section**
3. Section definition method > **DB**, DB > **IS(S)**
4. Sect. Name > **ISMB 350**
5. **Consider Shear Deformation** (Check on)
6. **[OK]**

▶Fig. 2.8
Define section



Tip

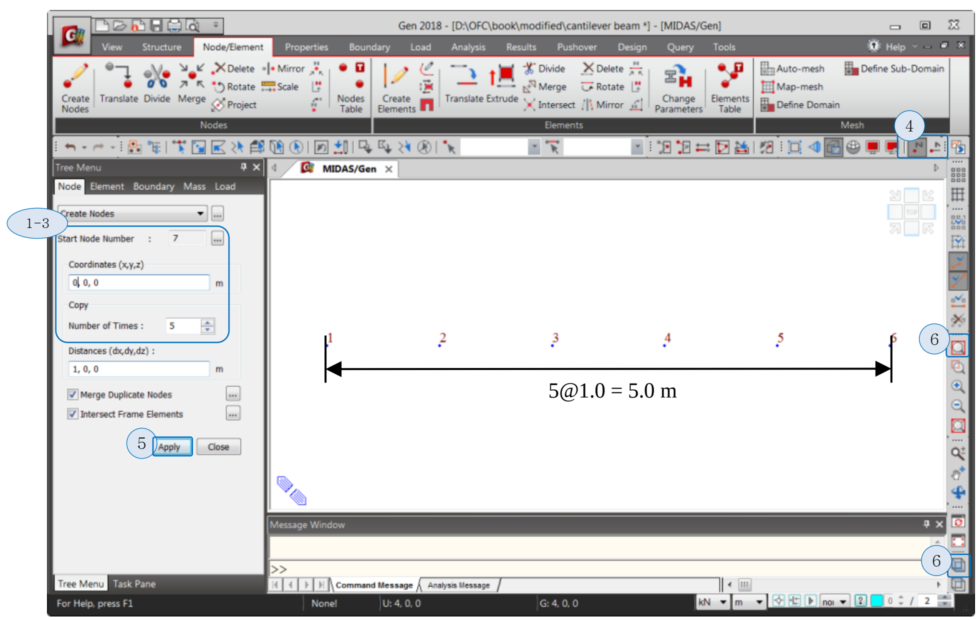
Consider Shear Deformation is generally checked on for structural analysis. However, general structural mechanics hand calculation ignores shear deformation, which is the reason for checking off in this case for comparison purposes.

2.4 Nodes & Elements Generation

Create nodes where elements will be created.

Main Menu: ***Node/Element / Create Nodes***
1. Coordinates (x, y, z): **‘0, 0, 0’**
2. Copy > Number of Times: **‘5’**
3. Distance (dx, dy, dz): **‘1, 0, 0’**
4. **Display Node Numbers, Display Element Numbers** (Toggle on)
5. **[Apply]**
6. **Auto Fitting, Front View** (Toggle on)

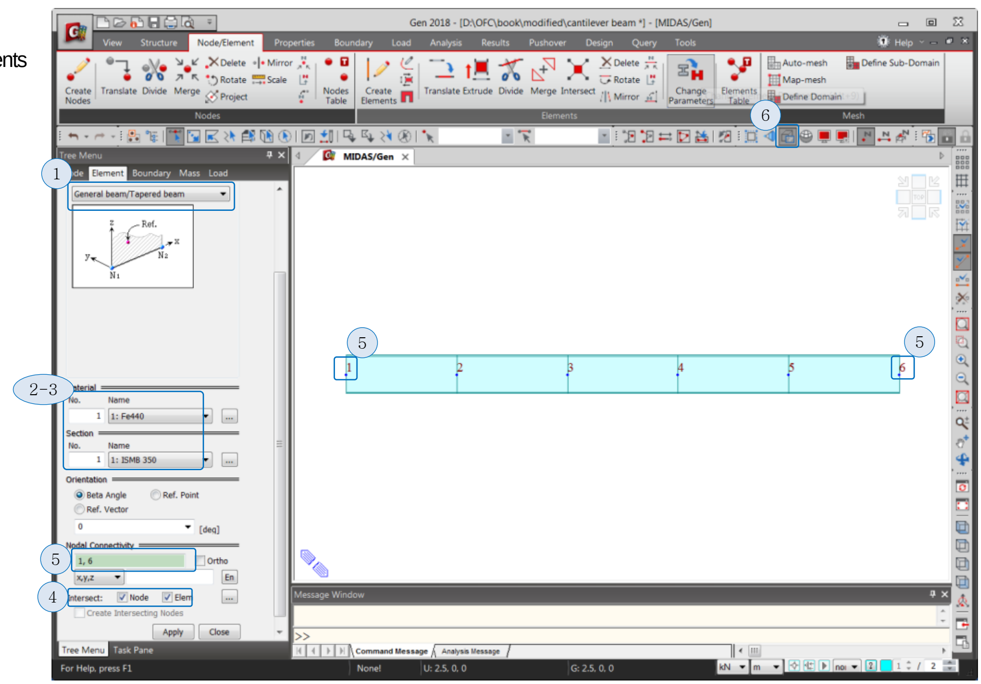
▶ Fig. 2.9
Create nodes



Connect nodes to create elements.

Main Menu: ***Node/Element / Create Elements***
1. Element Type **> General beam/Tapered beam**
2. Material > **1: Fe440**
3. Section > **1: ISMB350**
4. Intersect > **Node, Elem** (Check on)
5. Click the Nodal Connectivity field (which turns to green), followed by clicking **Node 1 and 6** on the model view sequentially.
6. **Hidden** (Toggle on)

▶Fig. 2.10
Create beam elements

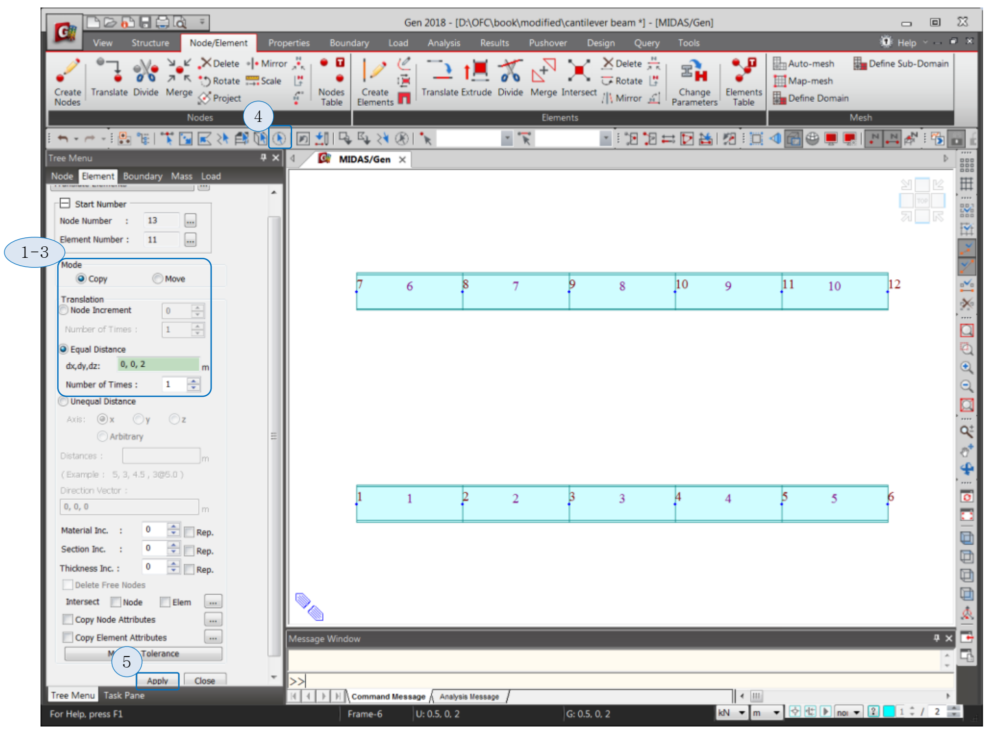


Copy the elements created with the same properties.

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

1. Mode > **Copy**
2. Translation > **Equal Distance**
3. dx, dy, dz : **‘0, 0, 2’**, Number of Times: **‘1’**
4. **Select All**
5. **[Apply]**

▶Fig. 2.11
Copy elements

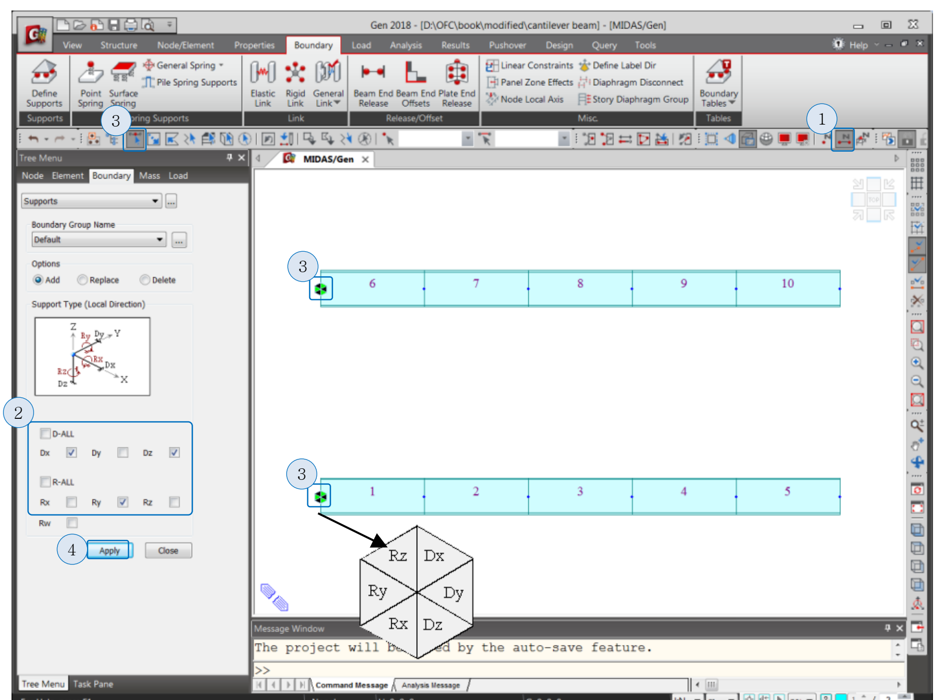


2.5 Define Boundary Conditions

midas Gen is 3-D software in which each node retains 6 degrees of freedom (Dx, Dy, Dz, Rx, Ry, Rz) symbolized by 6 triangular cells in a hexagon. Dx to Rz are sequentially represented starting from the "12 o'clock" in the clockwise direction. Green color in the triangular cells signifies restrained condition.

Main Menu: ***Boundary / Define Supports***
1. **Display Element Numbers** (Toggle off)
2. Support Type > **Dx, Dz, Ry** (Check on)
3. **Select Single** (on) mode: select **Nodes 1, 7**
4. **[Apply]**

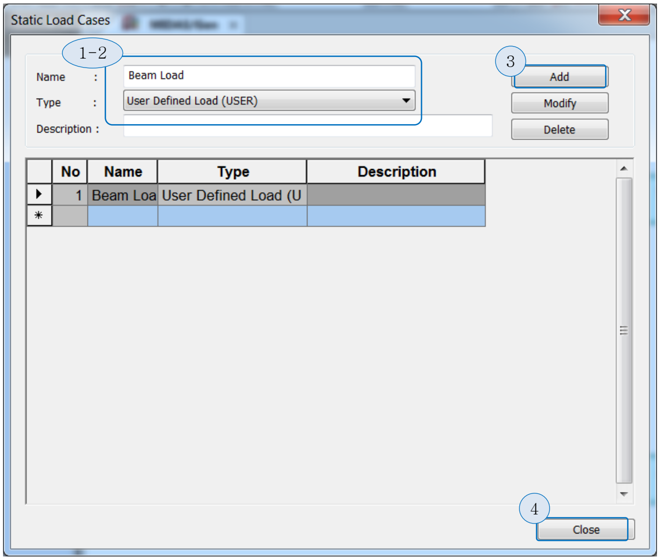
▶Fig. 2.12
Input support conditions



2.6 Define Loads

Define the load cases (load types) first to which individual loads will belong.

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Beam Load'**
2. Type > **User Defined Load (USER)**
3. **[Add]**
4. **[Close]**



▶Fig. 2.13
Define load case

Tip

Typically in practice, a number of Load Cases (Types) are defined such as Dead Load, Live Load, etc., which will be combined with relevant load factors to satisfy a specific design code.

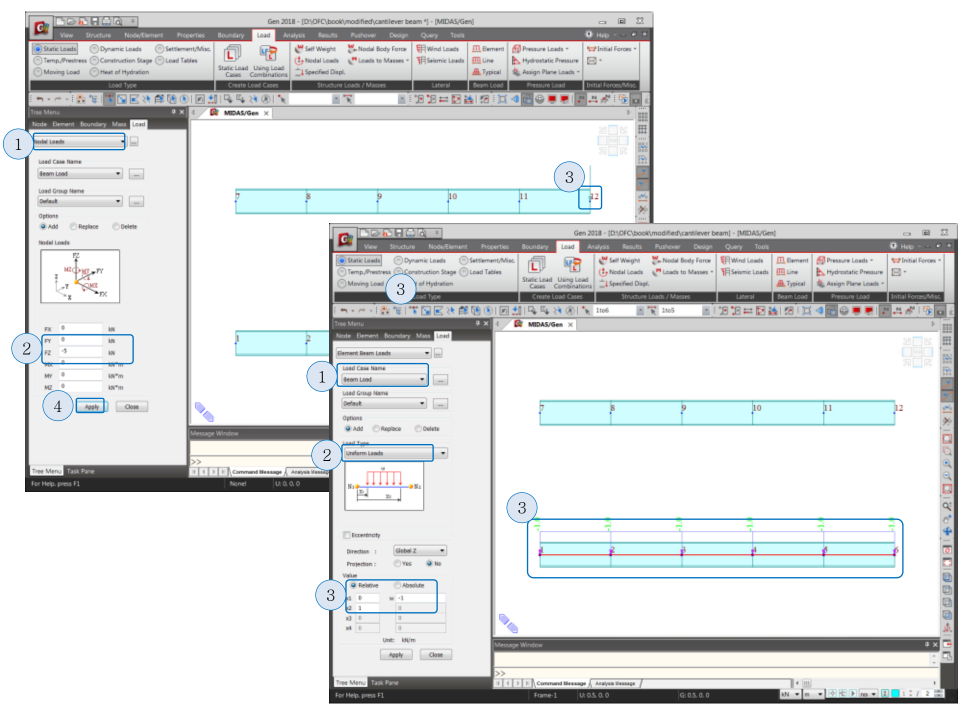
Input a concentrated load of 10 kN at the tip of the cantilever, Node 12.

Main Menu: ***Load / Static Loads / Nodal Loads***
1. Load Case Name > **'Beam Load'**
2. Nodal Loads FZ: **'-10'**
3. Select **Node 12**
4. **[Apply] [Close]**

Input a uniformly distributed load of 1.0 Kn/m.

Main Menu: ***Load / Static Loads / Element (Beam Load)***
1. Load Case Name > **'Beam Load'**
2. Load Type > **Uniform Loads**
3. Value > x1: **'0'**, w: **'-1'**, x2: **'1'**
4. **Display Element Numbers** (Toggle on)
5. Select **Elements 1, 2, 3, 4, 5**
6. **[Apply] [Close]**

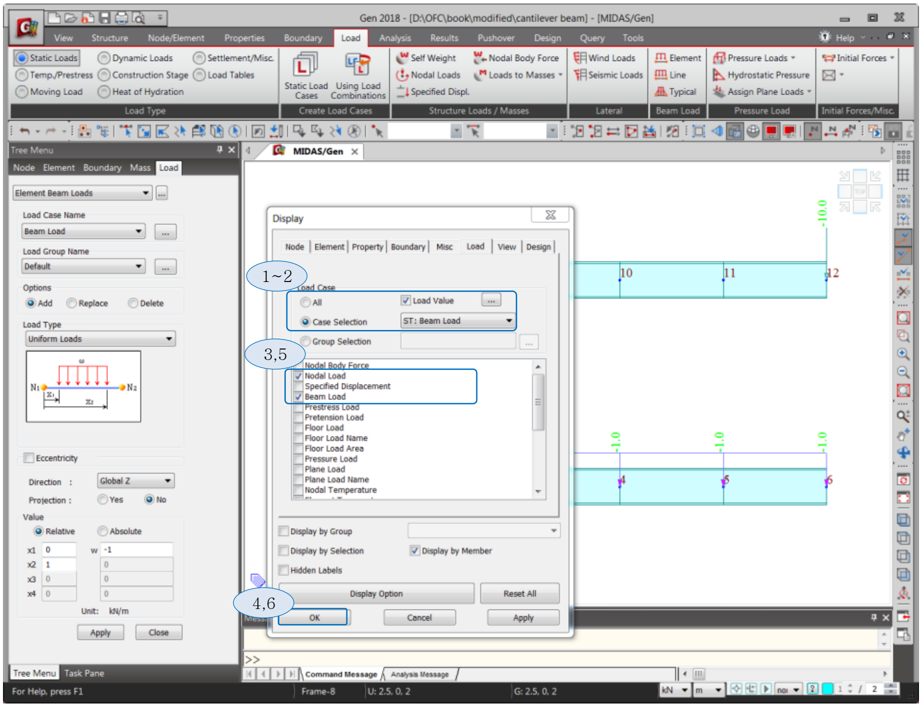
▶Fig. 2.14
Input loads



Check the modeling data prior to performing analysis.

Main Menu: ***View / Display…***
1. Load tab, Load Case > **Load Value** (Check on)
2. Case Selection > **ST: Beam Load**
3. From the list, check on **Nodal Load, Beam Load**
4. **[OK]**

Main Menu: ***View / Display***
5. **Nodal Load, Beam Load** (Check off)
6. **[OK]**

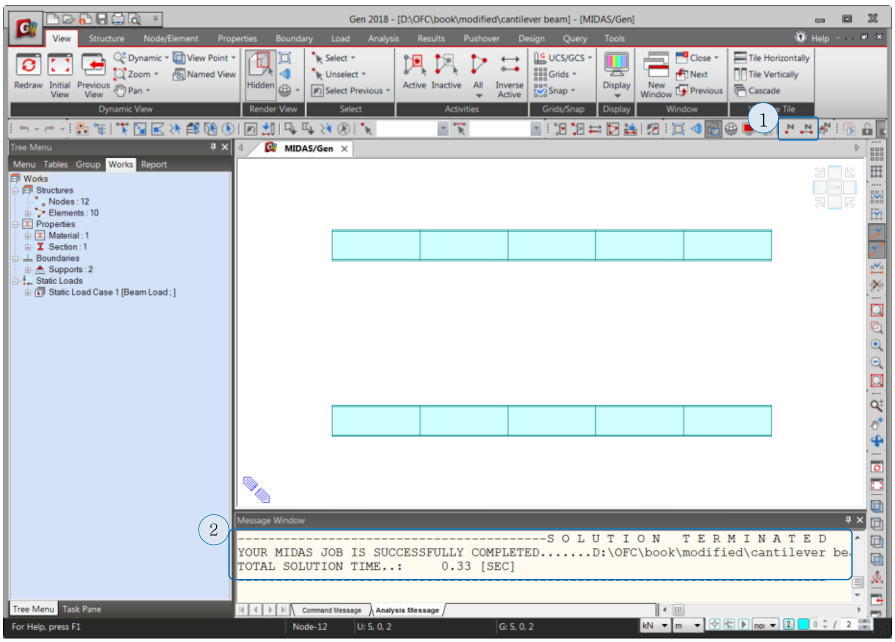


Analysis is performed.

Main Menu: ***Analysis / Perform Analysis***

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

▶Fig. 2.16
Message for a successful run



Tip

Shortcut key for Perform Analysis is F5.

Error messages will be generated in Message Window if the input data contains errors.

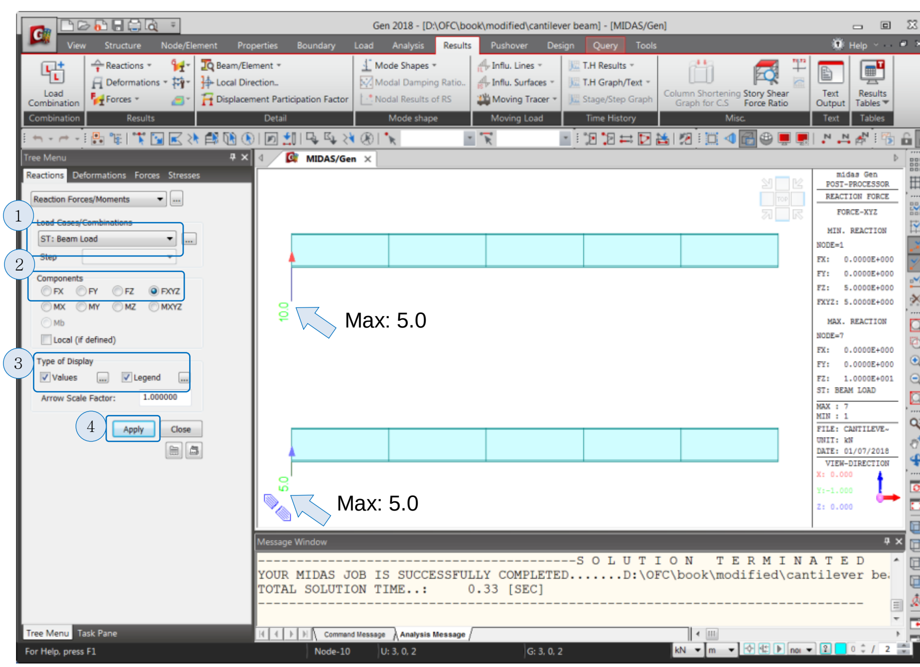
2.8 Check Analysis Results

Check the reactions from midas Gen and compare them with hand calculations.

Main Menu: ***Results / Reactions / Reaction Forces/Moments…***

1. Load Cases/Combinations > **ST: Beam Load**
2. Components > **FXYZ**
3. Type of Display: **Values, Legend** (Check on)
4. **[Apply] [Close]**

▶Fig. 2.17
Reaction results

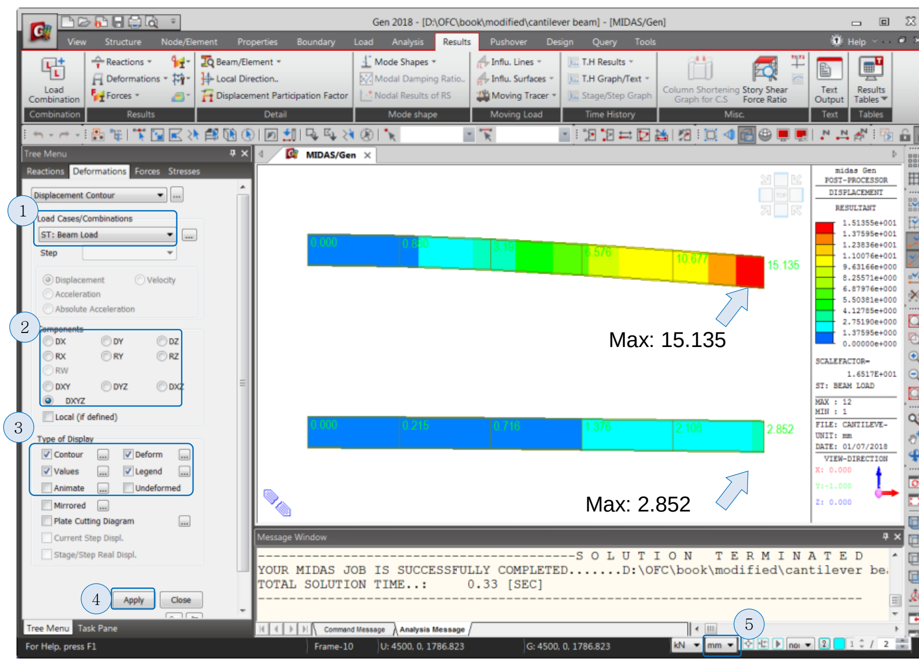


Check displacements from midas Gen and compare them with hand calculations.

Main Menu: ***Results / Deformations > Displacement Contour…***

1. Load Cases/Combinations > **ST: Beam Load**
2. Components > **DXYZ**
3. Type of Display > **Contour, Deform, Values, Legend** (Check on)
4. **[Apply] [Close]**
5. Change the unit to mm in Status Bar at the bottom of the work screen.

▶Fig. 2.18 Displacements

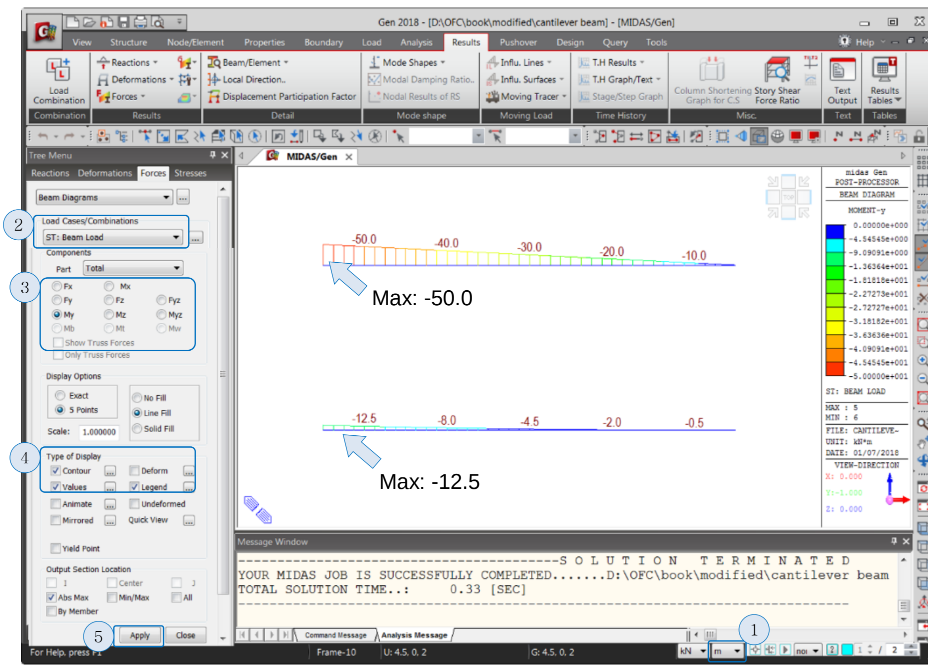


Check moment diagram.

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

1. Change the unit to m in Status Bar at the bottom of the work screen.
2. Load Cases/Combinations > **ST: Beam Load**
3. Components > **My**
4. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
5. **[Apply]**

▶Fig. 2.19
Moment



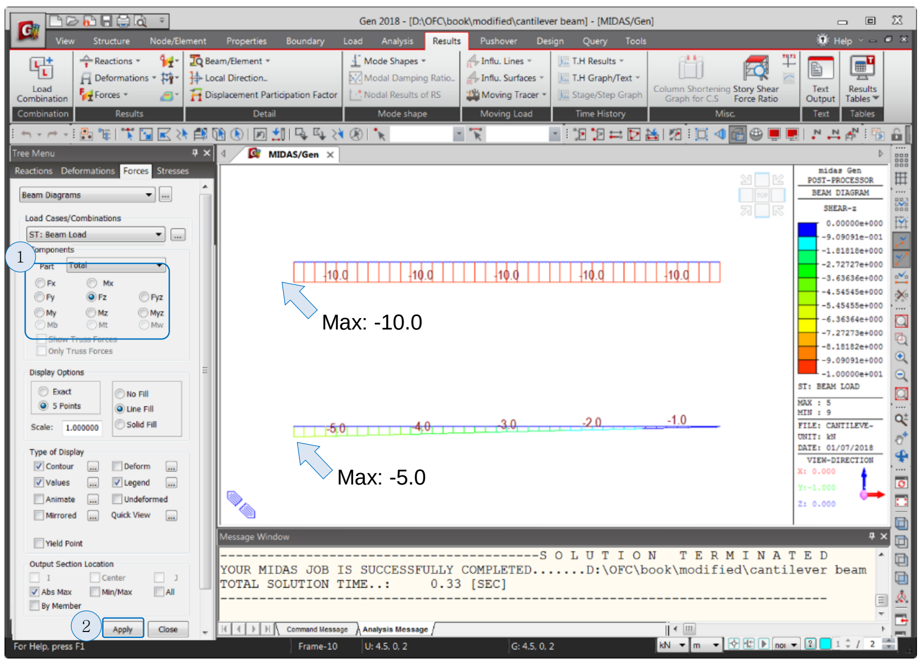
Tip

Member forces are generated in Element Coordinate System. For beam / column elements in 2-D, moment My, shear force Fz, and axial force Fx are generated.

Check shear diagram.

1. Components > **Fz**
2. **[Apply]**

▶Fig. 2.20
Shear force



Check member stresses.

Stress components:

Sax: axial stress in ECS x – dir.

Ssy, Ssz: shear stress in ECS y – dir. and z – dir. respectively

Sby, Sbz: flexural stress about ECS y – axis and z – axis respectively

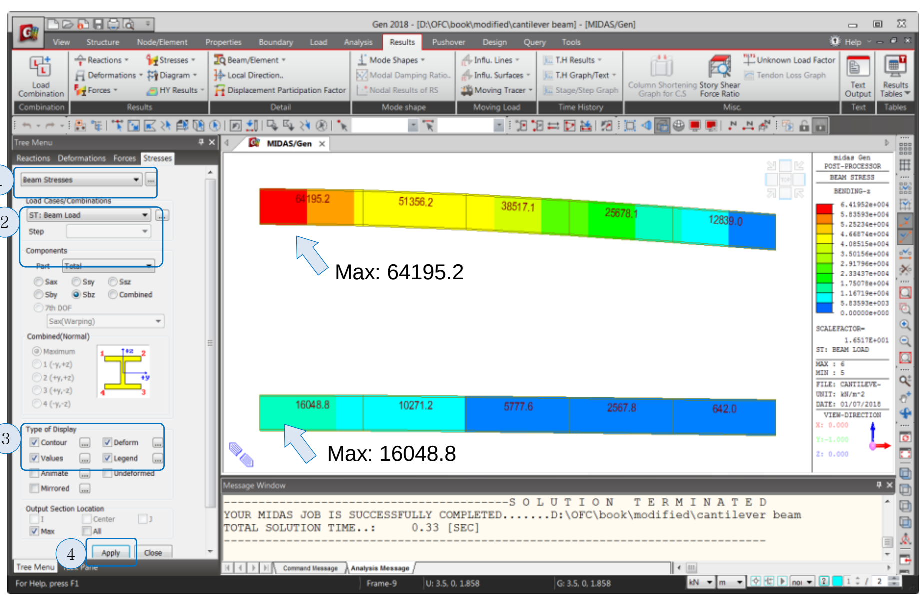
Combined: maximum or minimum of combined stresses, Sax ± Sby ± Sbz

Stress component Sbz needs to be checked for pure flexural stress.

Main Menu: ***Results > Stresses > Beam Stresses...***

1. Load Cases/Combinations > **ST: Beam Load**
2. Components > **Sbz**
3. Type of Display > **Contour, Deform, Value, Legend** (Check on)
4. **[Apply]**

▶Fig. 2.21
Beam Stress



3. Manual Calculations

3.1.1 Reactions

Reaction in Model 1 with a concentrated load, P = 10.0 kN

Reaction in Model 2 with a uniformly distributed load, ωl = 5.0 kN

These are the same as the midas Gen results.

3.1.2 Displacements

Displacement in Model 1 with a concentrated load, $\delta = \frac{Pl^3}{3EI} =$ -14.912 mm

Displacement in Model 2 with a uniformly distributed load, $\delta = \frac{\omega l^4}{8EI} = 5.286 \text{ mm}$

These are the same as the midas Gen results.

3.1.3 Shear forces

Shear force in Model 1 with a concentrated load, $V_A = P =$ -10 kN

Shear force in Model 2 with a uniformly distributed load, $V_A = \omega\lambda = -5\text{kN}$

These are the same as the midas Gen results.

3.1.4 Bending moments

Bending moment in Model 1 with a concentrated load, $M = -P\lambda =$ - 50 kNm

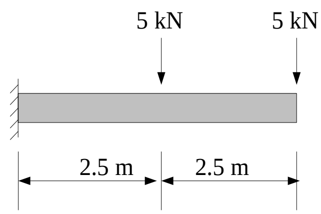
Bending moment in Model 2 with a uniformly distributed load, $M = -\frac{\omega l^2}{2} = -12.5 \text{ kN} \cdot \text{m}$

These are the same as the midas Gen results.

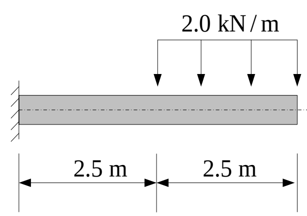
4. Exercise

Find the reactions, shear forces, bending moments and displacements for each beam below subject to the loadings shown (ignore the shear deformation).

Model 1: concentrated loads



Model 2: partial uniformly distributed load



- ***Material***
  Steel: Fe440 (Modulus of elasticity, $E = 2.05 \times 10^{8}$ kN/m^2)
- ***Section***
  Area: 6.671×10^{-3} m^2 (ISMB 350)
  Moment of inertia (I_y): 1.363030×10^{-4} m^4
- ***Load***
  Model 1: concentrated loads 5.0 kN
  Model 2: partial uniformly distributed load 2.0 kN/m

3. **Truss** Analysis

Contents

1. Introduction

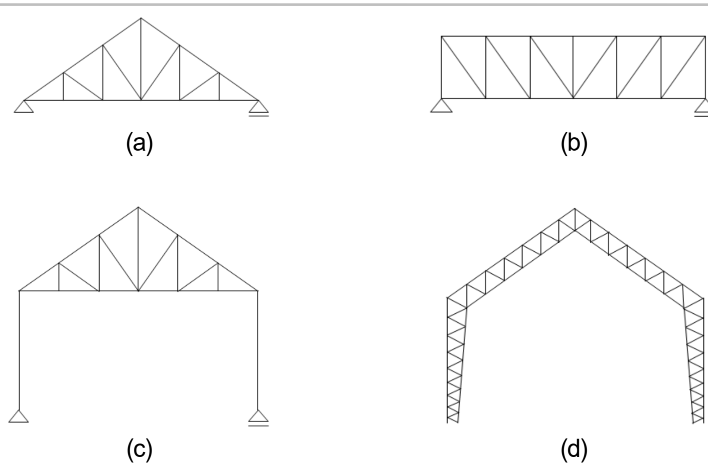
1.1 Concept for Truss Analysis

A truss generally consists of 2 or more chords connected by a series of hinged diagonals and verticals in the form of triangles. The truss type structural system is used for long span applications while minimizing the own weight, which include plants, auditoriums, sports arenas, stadiums and bridges.

▶Photo 3.1
(a) Roof trusses
(b) Long span bridge

Depending on application areas, trusses can be of many different shapes and forms, which include roof truss, truss beam, truss supported by columns, truss frame, etc.

▶Fig 3.1
Examples of truss types



Truss forces are found under the following assumptions:

1) Members joining at each node are hinge-connected.
2) Each member is straight.
3) The centerlines of the members meet at nodes.
4) Loadings are applied only at the nodes.

Since only nodal loadings are present, truss member forces are either only in tension (+) or in compression (-). Tension is defined when external and internal forces act away from a node.

▶Fig. 3.2
External force Internal force Node

A truss structure can be statically determinate or indeterminate. A statically indeterminate truss can be further classified as external and internal. A statically determinate truss is one that reactions and member forces can be calculated from only equilibrium, whereas additional conditions corresponding to the degree of indeterminacy are required for analyzing a statically indeterminate truss. Force equilibrium conditions alone cannot obtain the reactions of an externally indeterminate structure. For an internally indeterminate structure, reactions can be found by force equilibrium, but additional conditions are required for obtaining the member forces.

The degree of indeterminacy of a 2-D truss is determined as below;

Degree of external indeterminacy: $m_e = n - 3$

Degree of internal indeterminacy: $m_i = (3 + s + r) - 2k$

Total degree of indeterminacy: $m = m_e + m_i$

where,

n: number of unknown reaction components,

s: number of members,

r: number of rigidly connected nodes,

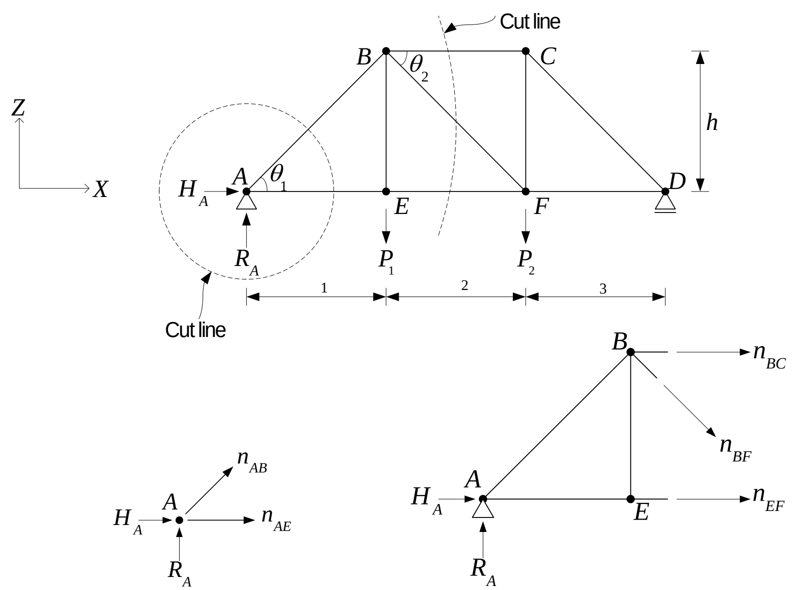
k: number of nodes

1) Solutions for statically determinate truss structures

A truss can be analyzed by the methods of joints (nodes), sections, stress coefficients, drawings, influence lines, etc. Among these methods, the joint method entails force equilibrium in a joint free body diagram like $\sum F_X = 0$, $\sum F_Z = 0$. The section method involves force equilibrium of the free body diagram of a cut truss satisfying $\sum F_X = 0$, $\sum F_Z = 0, \sum M = 0$. The moment sum of force equilibrium is missing from the joint method as a truss node cannot resist moments.

The diagrams below illustrate the joint and section methods. Nodal equilibrium at the joint (node) A is shown in the joint method. The section method shows a cut truss to which force equilibrium is applied. The fact that 2 equations can be used for the joint method, no more than 2 members can be connected. Similarly 3 equations can be used in the section method, hence the number of the cut members should be limited to no more than 3.

▶ Fig. 3.3
Truss cuts



$$\sum F_X = H_A + n_{AE} + n_{AB}\cos\theta_1 = 0$$
$$\sum F_Z = R_A + n_{AB}\sin\theta_1 = 0$$

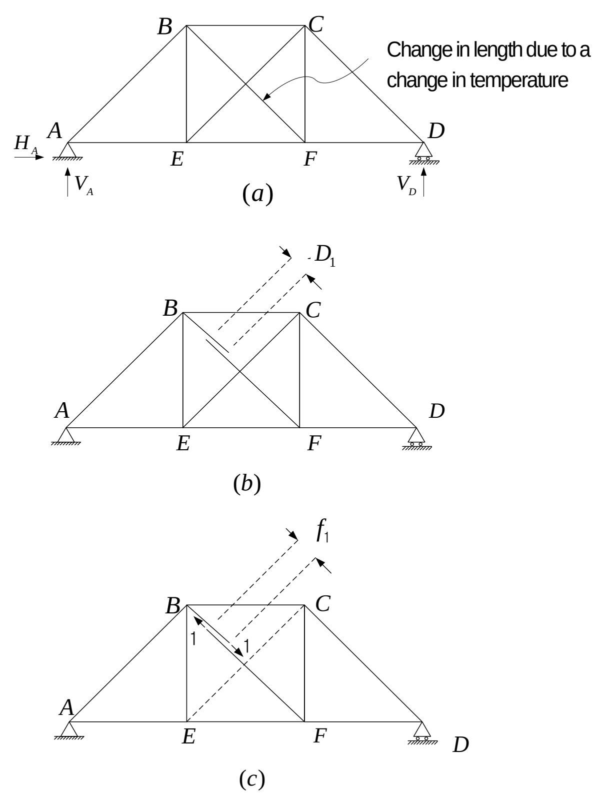
$$\sum F_X = H_A + n_{BC} + n_{EF}\cos\theta_2 + n_{EF} = 0$$
$$\sum F_Z = R_A - n_{BF}\sin\theta_2 = 0$$
$$\sum M_B = -H_A \cdot h + R_A \cdot l_1 - n_{EF} \cdot h = 0$$

2) Solutions for statically indeterminate truss structures

The method of consistent deformation or the flexibility method is useful in analyzing indeterminate truss structures. In the method of consistent deformation, equations are formulated for the degrees of freedom selected as unknowns, and compatibility conditions are applied.

The structure below illustrates the flexibility method in which one degree of freedom is selected as an unknown. The BF truss member in the example is artificially heated up to elongate its length by Δ. The concept of flexibility method is explained for the unknown member force in BF, X_1.

▶Fig. 3.4
Flexibility method
(a) Analysis model
(b) Case of an external load applied
(c) Case of a unit load applied to BF member



In the flexibility method or the method of consistent deformation, analysis is carried out by establishing a compatibility condition for the state of removed member or support selected as an unknown. If the BF member force is selected as an unknown, the compatibility condition corresponds to the state of the member cut in the middle. The change in distance (overlap or separation) between the two disconnected points due to an external load is defined as D_1. Note that a change in temperature is not an external load. And the change in distance between the two disconnected points due to a unit load is defined as f_{11}. Then, $f_{11}\,X_1$ represents the distance between the two disconnected points in the BF member subjected to a member force, X_1. The compatibility condition in this case then becomes,

▶Eq. 3.1

$$f_{11}X_1 + D_1 = \Delta_1$$

If only a loading is present without a temperature-like cause, Δ_1 becomes zero. But if the member BF undergoes a change in length due to a temperature effect, D_1 becomes zero, and Δ_1 becomes the change in length. That is, the above equation expresses the compatibility condition for the fact that the member BF is not at the disconnected state. Because $f_{11}\,X_1$ and D_1 cancel each other or the actual deformation is Δ_1, it is also referred to as the method of consistent deformation. The unit load is the most effective method of finding the deformation f_{11} or D_1 among other methods.

When a structure has the degree of indeterminacy of 2, 2 members or reactions are selected. The following compatibility conditions can be formulated for the state of removed members or reactions:

▶Eq 3.2

$$\begin{bmatrix} f_{11} & f_{12} \\ f_{21} & f_{22} \end{bmatrix}\begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} + \begin{Bmatrix} D_1 \\ D_2 \end{Bmatrix} = \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

$$f_{11} = \sum \frac{l_i}{EA_i} n_i^1 n_i^1, \quad f_{12} = f_{21} = \sum \frac{l_i}{EA_i} n_i^1 n_i^2, \quad f_{22} = \sum \frac{l_i}{EA_i} n_i^2 n_i^2$$

$$D_1 = \sum \frac{l_i}{EA_i} n_i^1 N_i, \quad D_2 = \sum \frac{l_i}{EA_i} n_i^2 N_i$$

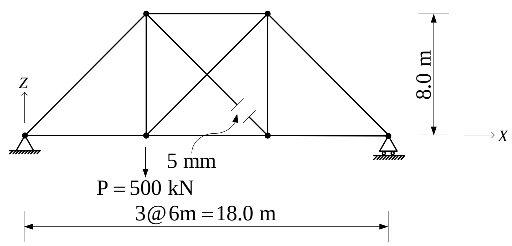
Δ_i= Deformation due to erection tolerance or temperature change, etc.

2. Tutorial

2.1 Model Overview

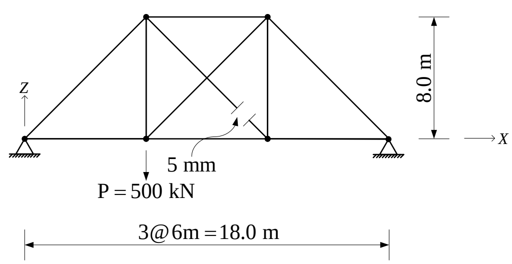
Compare the behaviors of the two trusses, one truss with the degree of internal indeterminacy of 1, and the second truss with each degree of internal and external indeterminacy of 1. Both cases are subject to an internal load like an erection tolerance and a concentrated load.

Model 1: Degree of internal static indeterminacy of 1



▶Fig. 3.5
Analysis model

Model 2: Degree of internal and external static indeterminacy of 1 each



- ***Material***: Fe540 (Modulus of elasticity, $E = 2.050\times10^{8}$ kN/m^2)
- ***Section***: SHS 250 X250X12 mm
- ***Load***
  1. Concentrated nodal load: 500 kN
  2. Erection tolerance: 5 mm → converted into a pretension load (1133.035 kN)

$$P = K\delta = EA/l \times\delta = (2.0500\times108\times0.011054/10)\times0.005 = 1133.035 \text{ kN}$$

2.2 Work Environment

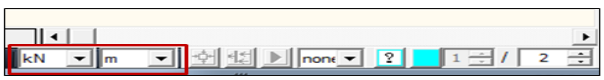
Open a New File and save.

Main Menu: ***/ New Project...***

Main Menu: ***/ Save As...***

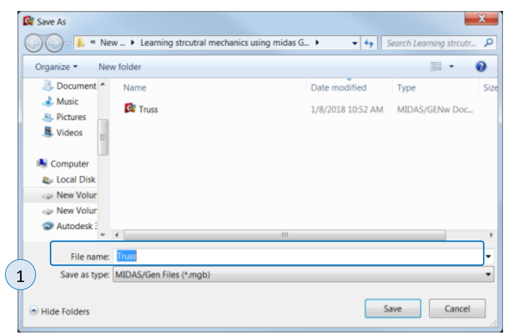
1. File name: **'Truss'** and [Save]

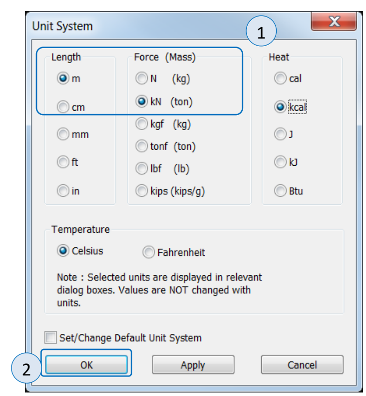
Set a unit system to kN & m at the bottom of work window.



Alternatively Main Menu: ***Tools / Unit System...***

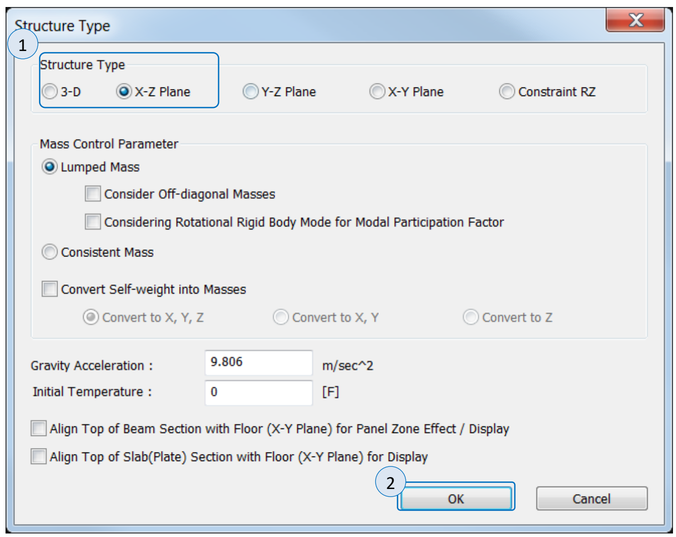
▶Fig. 6.8
Save file &
set unit system





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

Main Menu: ***Structure / Structure Type...***
1. Structure Type > **X-Z Plane**
2. **[OK]**



▶Fig. 3.7
Set a work plane

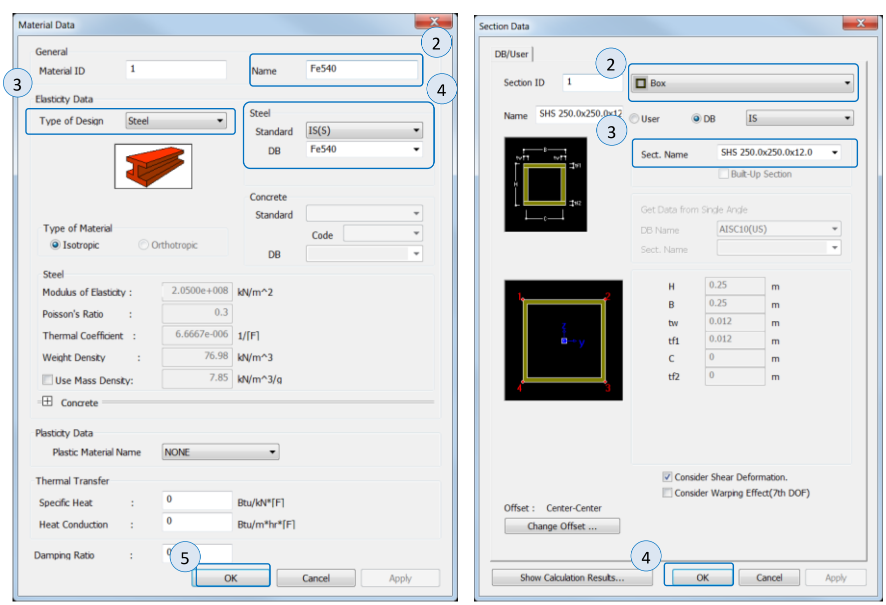
2.3 Material & Section Properties

Select Fe540 from DB.

Main Menu: ***Properties > Material Properties***
1. **[Add...]**
2. Name: **'Fe540'**
3. Type of Design > **Steel**
4. Steel > Standard > **IS(S)**, DB > **Fe540**
5. **[OK]**

Click Section tab.
1. **[Add...]**
2. Section shape list > **Box**
3. Sect. Name > **SHS 250 X250X12**
4. **[OK] [Close]**

▶ Fig. 3.8
(a) Define material
(b) Define section

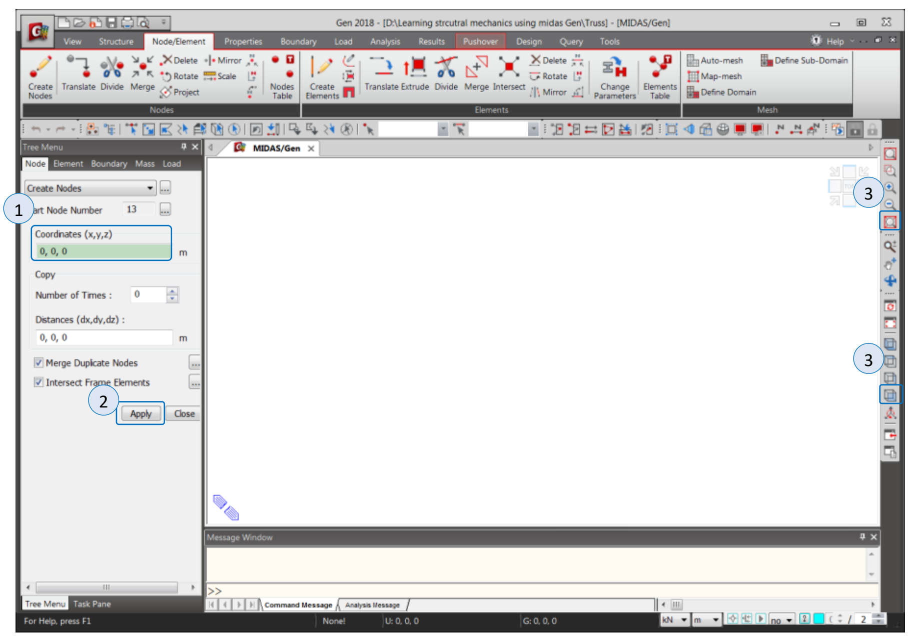


2.4 Nodes & Elements Generation

Create nodes where elements will be created.

Main Menu: ***Node/Element > Create Nodes***
1. Coordinates (x, y, z): **'0, 0, 0'**
2. **[Apply]**
3. **Auto Fitting, Front View** (Toggle on)

▶Fig. 3.9
Create a node

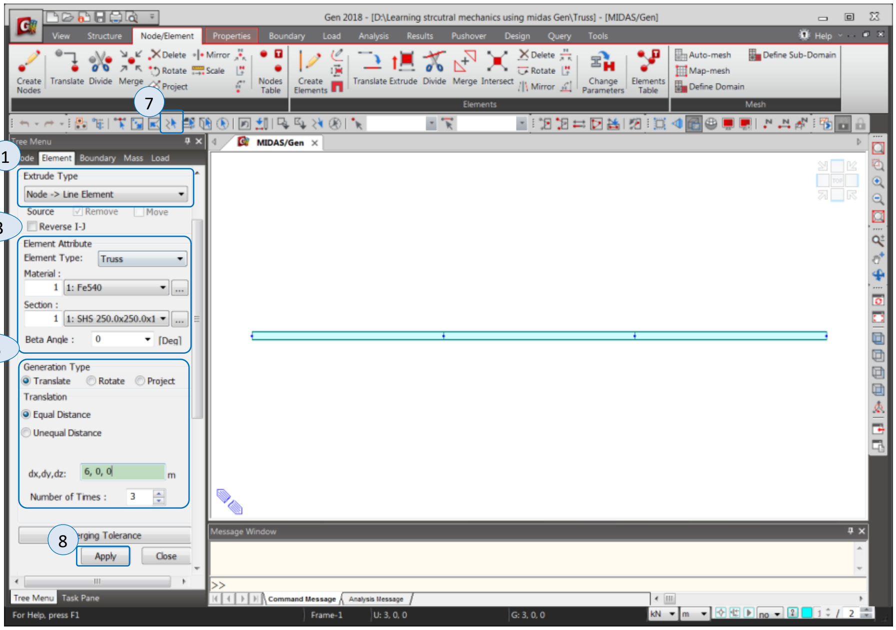


Use Extrude Elements to extrude a node into line elements for the bottom chord.

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

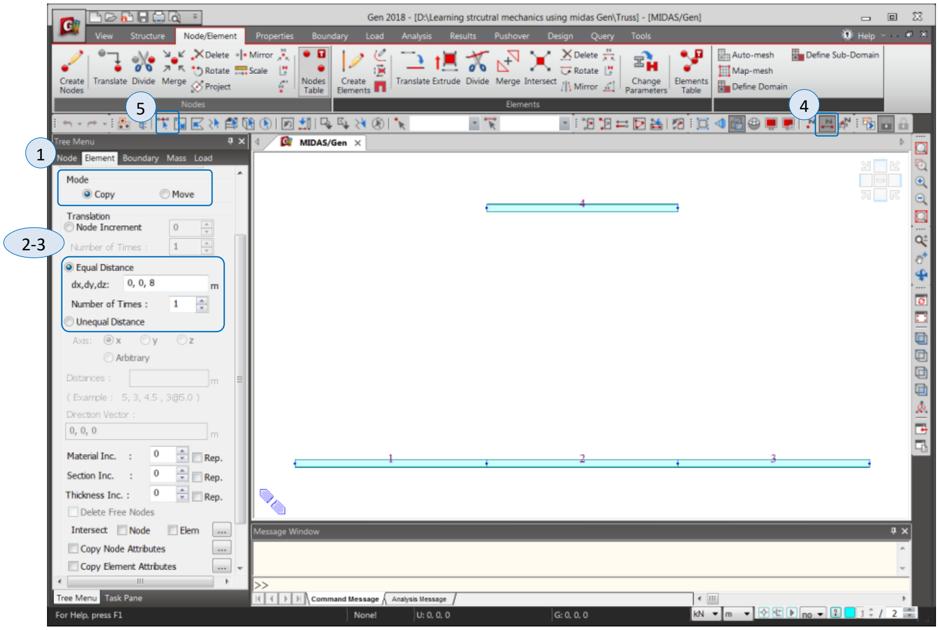
1. Extrude Type > **Node → Line Element**
2. Element Attribute > Element Type > **Truss**
3. Material > **'1:Fe540**, Section > **1: SHS 250 X250X12**, Beta Angle: **'0'**
4. Generation Type > **Translate**
5. Translation > **Equal Distance**
6. dx, dy, dz: **'6, 0, 0'**, Number of Times: **'3'**
7. **Select All** to select the node
8. **[Apply]**

▶Fig. 3.10
Create a bottom chord



Copy the bottom chord to create the top chord.

Main Menu: ***Node/Element > Translate Elements***
1. Mode > **Copy**
2. Translation > **Equal Distance**
3. dx, dy, dz: **‘0, 0, 8’**, Number of Times: **‘1’**
4. **Display Element Numbers** (Toggle on)
5. **Select Single** (on) mode: select **Element 2**
6. **[Apply]**



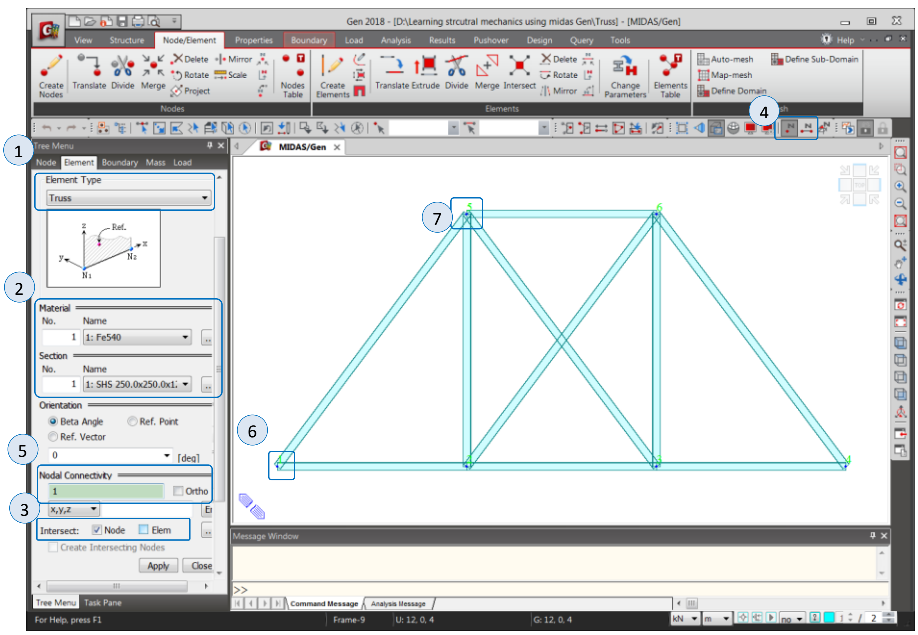
▶ Fig. 3.11
Create top chord

Input the diagonals and verticals.

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

1. Element Type > **Truss**
2. Material > **1:Fe540**, Section > **1: SHS 250 X250X12**
3. Intersect > **Elem** (Check off)
4. **Display Node Numbers** (Toggle on), **Display Element Numbers** (Toggle off)
5. Click the Nodal Connectivity field (which turns to green)
6. Click **Nodes 1 and 5** on the model view sequentially.
7. Click **Nodes (5, 2), (2, 6), (5, 3), (6, 3), (6, 4)** sequentially

▶Fig. 3.12
Create diagonals and verticals

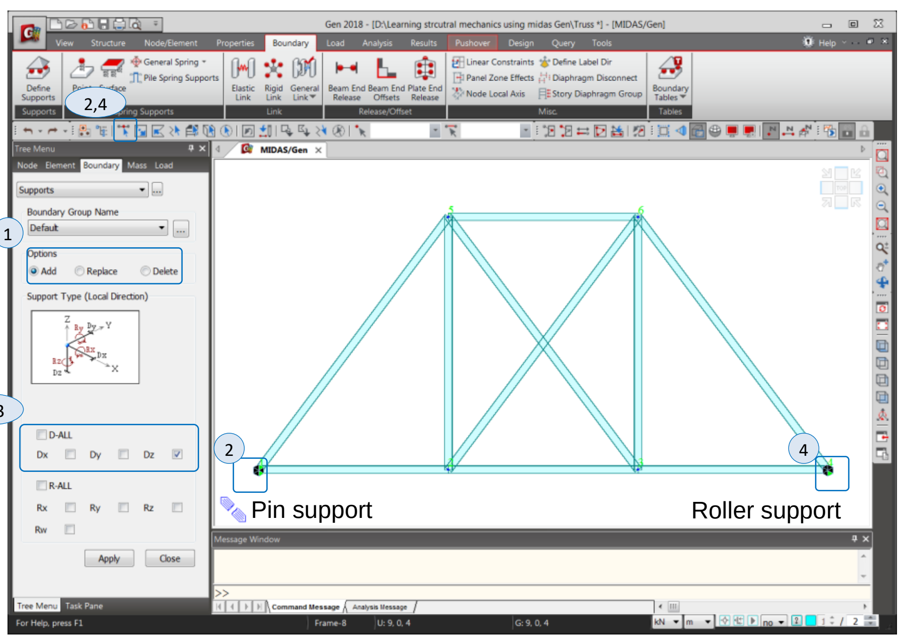


2.5 Define Boundary Conditions

In 3-D, each node retains 6 degrees of freedom (Dx, Dy, Dz, Rx, Ry, Rz). But with the work plane being on X-Z plane, only 3 degrees of freedom (Dx, Dz, Ry) exist, among which the Dx, Dz degrees of freedom are restrained for the pin support, and the Dz degree of freedom is restrained for the roller support.

Main Menu: ***Boundary / Define Supports***
1. Options > **Add**, Support Type > **Dx, Dz** (Check on)
2. **Select Single** (on) mode: select **Node 1, [Apply]**
3. Support Type > **Dx** (Check off)
4. **Select Single** (on) mode: select **Node 4, [Apply]**

▶Fig. 3.13
Define support conditions

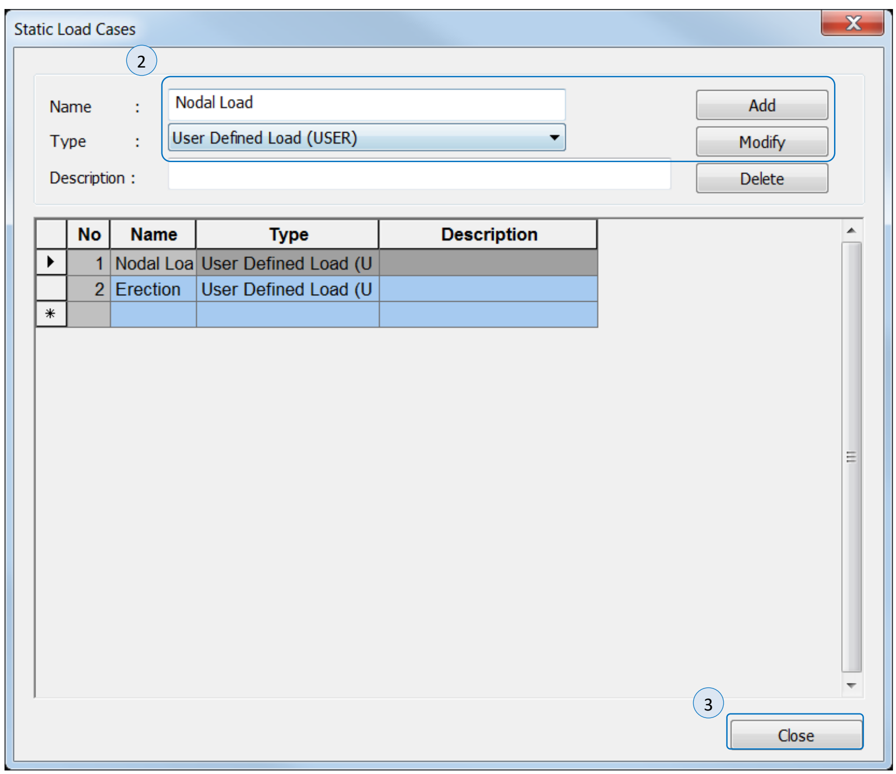


2.6 Define Loads

Define the load cases for a nodal concentrated load and a pretension load representing the erection tolerance individually.

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Nodal Load'**, Type > **User Defined Load**, **[Add]**
2, Name: **'Erection Tolerance'**, Type > **User Defined Load**, **[Add]**
3. **[Close]**

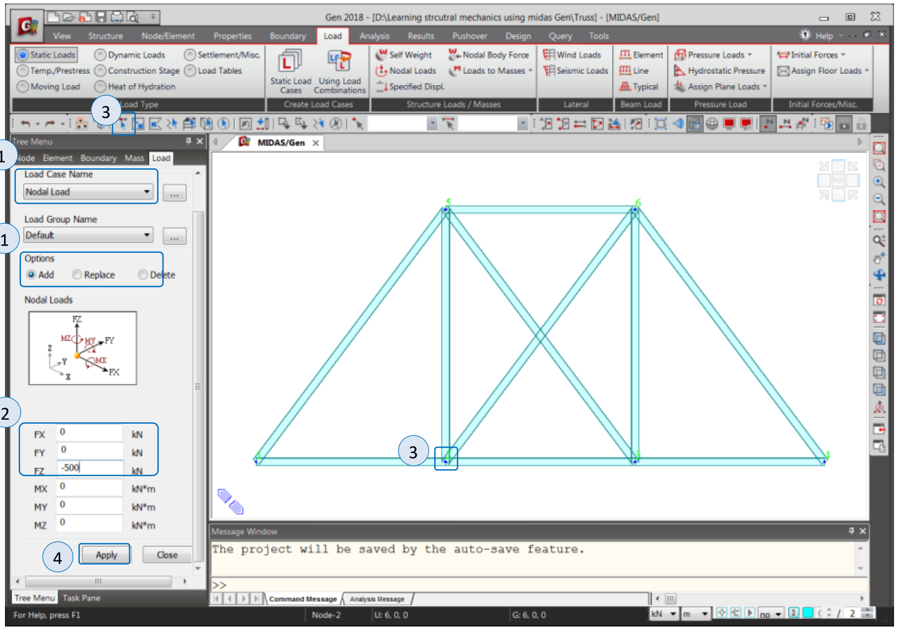
▶Fig. 3.14
Define load cases



Input the concentrated load, 500 kN at Node 2.

Main Menu: ***Load / Static Loads / Nodal Loads***
1. Load Case Name > **Nodal Node**, Options > **Add**
2. Nodal Loads > FZ: **'-500' kN**
3. **Select Single** (on) mode: select **Node 2**
4. **[Apply]**

▶Fig. 3.15
Input nodal load

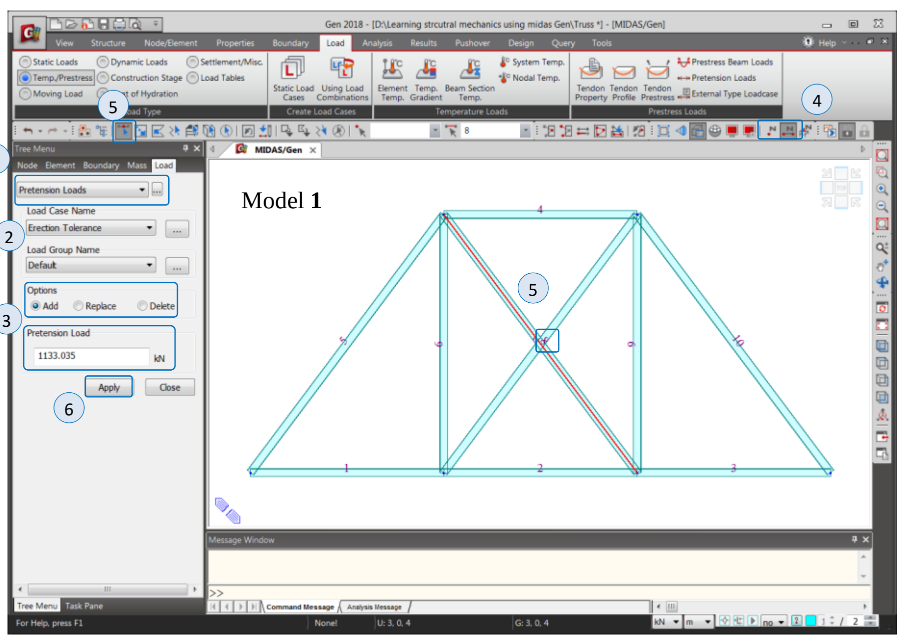


If the member length is fabricated 5 mm shorter than the true length, tension is introduced in the corresponding member during erection. Such fabrication error is converted into a pretension load.

$$P = K\delta = EA/l \times \delta = (2.050\times10^{8}\times0.011054/10)\times0.005 = 1133.035 \text{ kN}$$

Main Menu: ***Load / Temp./Prestress / Pretension Loads***

1. Load Case Name > **Erection Tolerance**
2. Options > **Add**
3. Pretension Load: **'1133.035'** kN
4. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
5. **Select Single** (on) mode: select **Element 8**
6. **[Apply]**



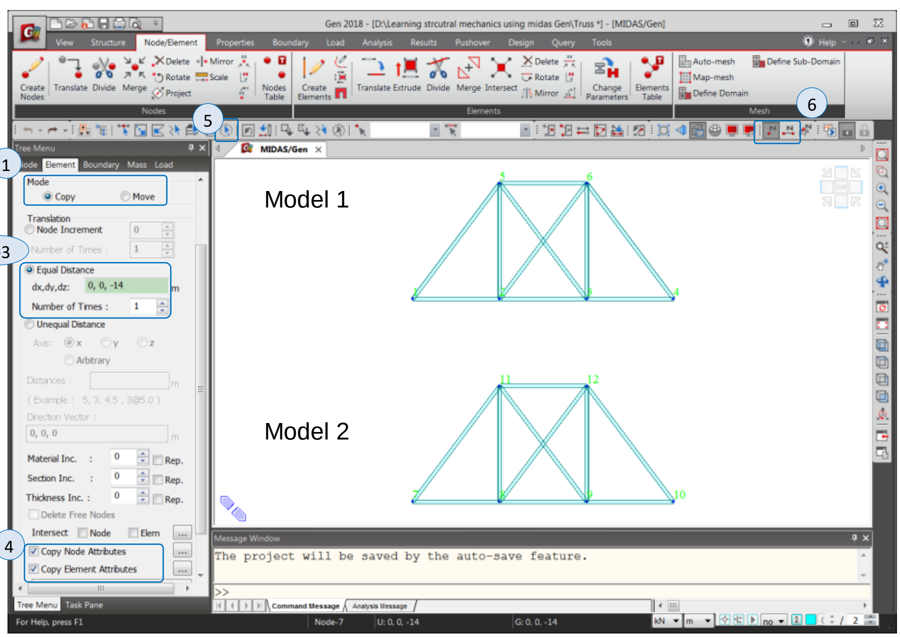
▶Fig. 3.16
Input pretension load

Copy Model 1 to create Model 2. In order to also copy the nodal load, pretension load and boundary conditions, check on Copy Node Attributes and Copy Element Attributes.

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

1. Mode > **Copy**
2. Translation > **Equal Distance**
3. dx, dy, dz: **'0, 0, -14'**, Number of Times: **'1'**
4. **Copy Node Attributes, Copy Element Attributes** (Check on)
5. **Select All**
6. **Display Node Numbers** (Toggle on), **Display Element Numbers** (Toggle off)
7. **[Apply] [Close]**

▶Fig. 3.17
Copy elements

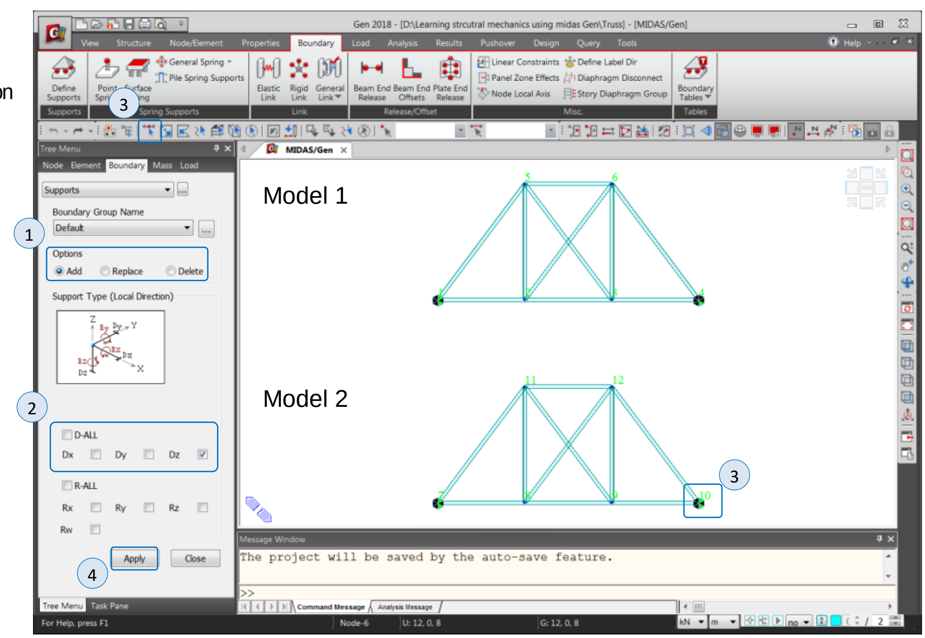


Model 2 is an externally static indeterminate structure for which the roller condition is revised to a pinned condition by additionally assigning the Dx degree of freedom.

Works Tree: Right-click on Type 2 [001000] under Boundaries – Supports: 4 and click Display

Main Menu: ***Boundary / Define Supports***
1. Options > **Add**
2. Support Types > **Dx** (Check on)
3. **Select Single** (on) mode: select **Node 10**
4. **[Apply] [Close]**

▶Fig 3.18
Change support condition

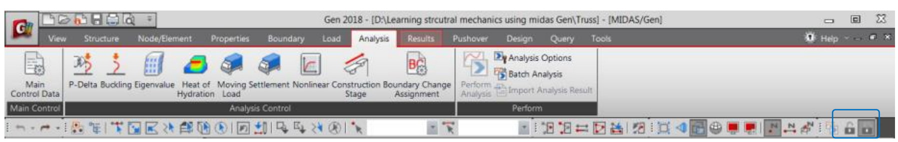


2.7 Perform Analysis

Analyze both Model 1 and Model 2.

Main Menu: ***Analysis / Perform Analysis***

midas Gen effectively manages the analysis model using Preprocessing Mode and Postprocessing Mode. Postprocessing Mode preserves the analysis results without accidently tempering the model data. The operation status (Pre or Post Modes) can be switched from the lock looking icons.



▶Fig. 3.19
Analysis Pre and Post Modes

Preprocessing Mode (in short Pre Mode)
Pre Mode is used to enter or revise such data as geometry, member information, loadings, boundary conditions, etc. to construct an analysis model. Upon completion of analysis, Pre Mode is automatically switched to Post Mode.

Postprocessing Mode (in short Post Mode)
Post Mode becomes activated after the analysis is completed. If no analysis results exist, then the icon is displayed inactive. Post Mode allows checking and analyzing the analysis results such as reactions, displacements, member forces, stresses, etc., and furthermore analysis/design report can be generated.

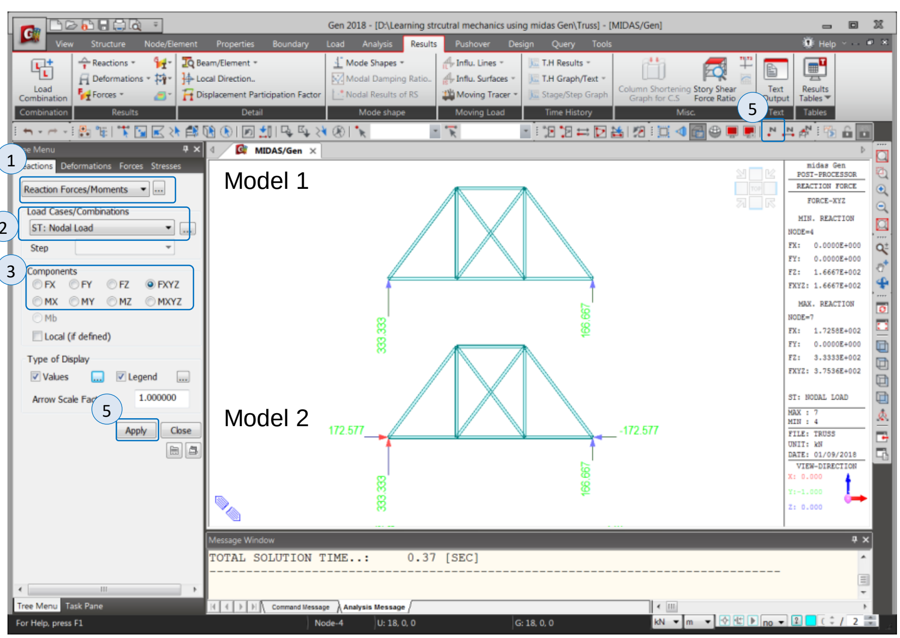
2.8 Check Analysis Results

Compare the reactions of Model 1 (externally determinate due to the external nodal load) with the reactions of Model 2 (externally indeterminate).

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

1. Load Cases / Combinations > **ST: Nodal Load**
2. Components > **FXYZ**
3. Type of Display > **Values, Legend** (Check on)
4. Click **[...]** next to Values, Decimal Points: **'3'**, **Exp.** (Check off),
   Apply upon OK (Check on), **[OK]**
5. **Display Node Numbers** (Toggle off)
6. Main Menu: ***View / Display...***, Boundary tab: **Support** (Check off), **[OK]**

▶Fig. 3.20
Reactions due to nodal load

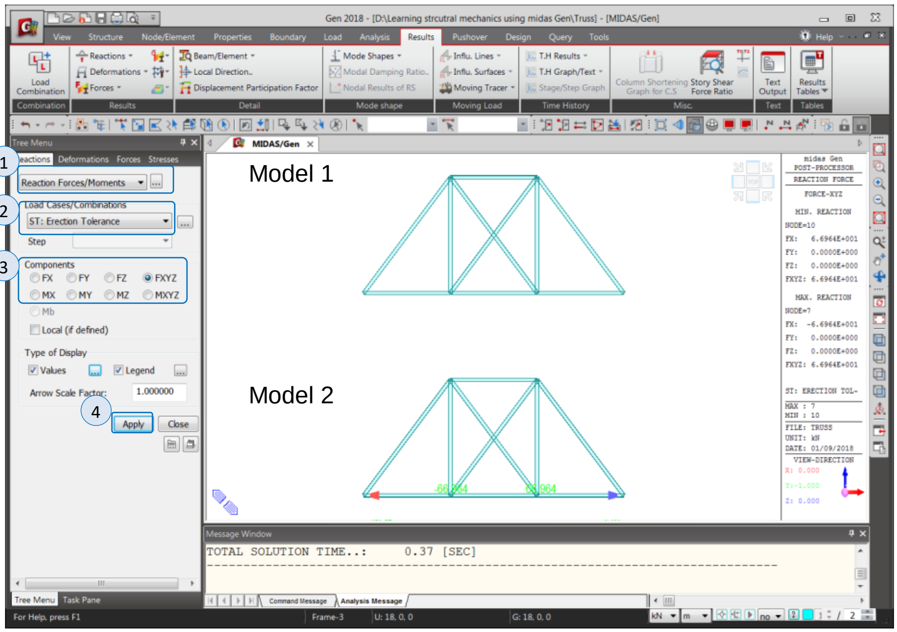


In case of an internal pretension load, no reactions are observed in Model 1, an externally determinate structure. The internal pretension load in Model 2 results in horizontal reactions (FX) due to the restraints in the X direction, which is an externally indeterminate structure.

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

1. Load Cases / Combinations > **ST: Erection Tolerance**
2. Components > **FXYZ**
3. Type of Display > **Values, Legend** (Check on)
4. **[Apply]**

▶Fig. 3-21
Reactions due to pretension load

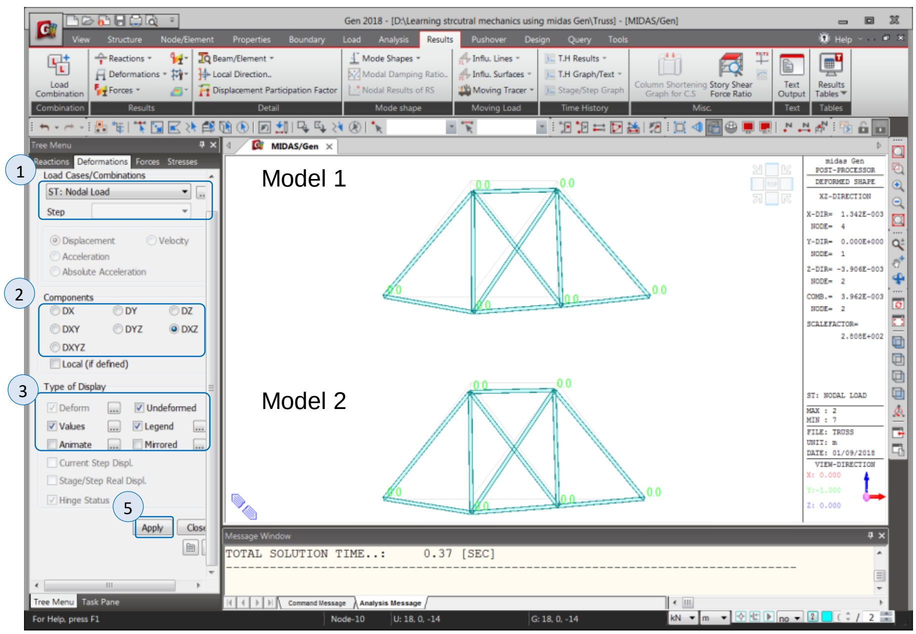


Check the deformations due to the nodal load. DXZ represents $\sqrt{DX^2 + DZ^2}$. Model 1 (externally static determinate truss) exhibits larger displacements than Model 2 (externally indeterminate truss).

Main Menu: ***Results / Deformations / Deformed Shape…***

1. Load Cases / Combinations > **ST: Nodal Load**
2. Components > **DXZ**
3. Type of Display > **Undeformed, Values, Legend** (Check on)
4. Click **[...]** next to Values, Decimal Points: **'1'**, **Exp.**, **Apply upon OK** (Check on)
5. **[OK]**

▶Fig. 3.22
Deformations due to nodal load

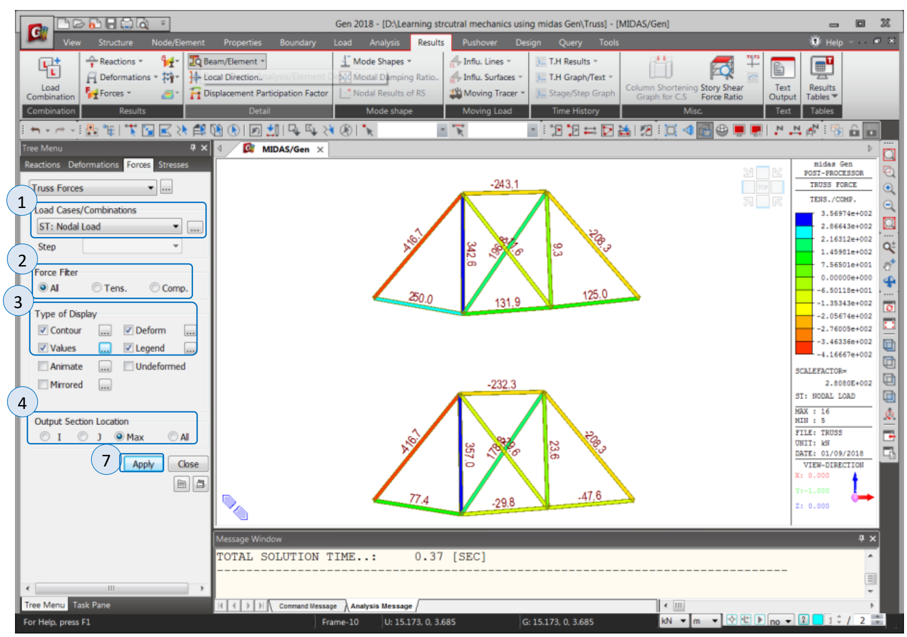


Difference in axial forces due to the pretension load can be observed between the 2 Models for the reason outlined above.

Main Menu: ***Results / Forces / Truss Forces...***

1. Load Cases / Combinations > **ST: Nodal Load**
2. Force Filter > **All**
3. Type of Display > **Contour, Deform, Values, Legend** (Check on)
4. Output Section Location > **Max**
5. Click **[...]** next to Values, Decimal Points: **'1'**,
   Exp., Set Orientation, Apply upon OK **(Check off)**
6. **[OK]**
7. **[Apply]**

▶Fig. 3.23
Axial forces due to nodal load



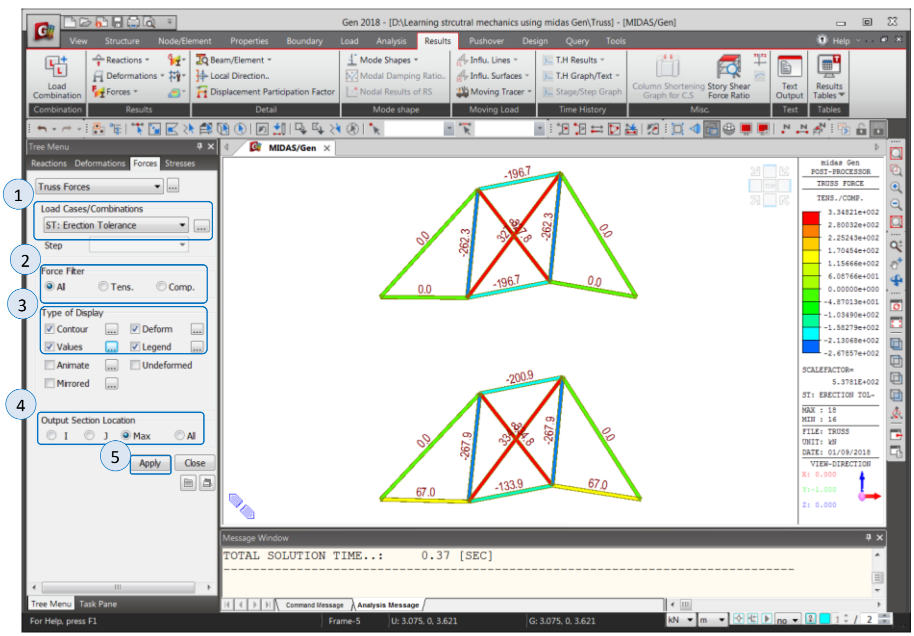
Since no reactions are present in Model 1 (externally determinate truss) due to the pretension load, no axial forces exist in the members connected to the supports.

Tip

Only the step 1 can be done as the setting from the previous Load Case (Nodal Load) remains unchanged.

Main Menu: ***Results / Forces / Truss Forces***

1. Load Cases / Combinations > **ST: Erection Tolerance**
2. Force Filter > **All**
3. Type of Display > **Contour, Deform, Values, Legend** (Check on)
4. Output Section Location > **Max**
5. **[Apply]**



▶Fig. 3.24
Axial forces due to pretension load

3. Manual Calculations

Model 1 is externally determinate, but internally indeterminate.

Degree of external indeterminacy: $m_e = n - 3 = 3 - 3 = 0$
Degree of internal indeterminacy: $m_i = (3 + s + r) - 2k = (3 + 10 + 0) - 2 \times 6 = 1$

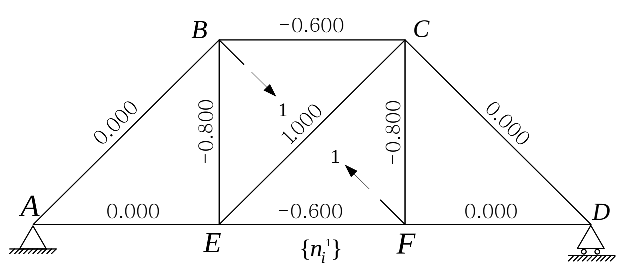
As the degree of indeterminacy of Model 1 is 1, the method of consistent deformation can be used by assigning the ***BF*** member force as an unknown, X_1.

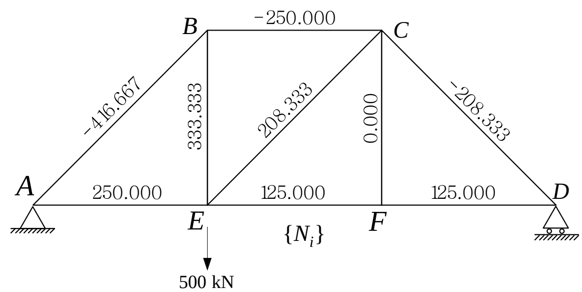
$$f_{11}X_1 + D_1 = \Delta_1$$

D_1 is the change in distance between ***BF*** due to a load without the presence of the member ***BF*** (in a statically determinate structure). $f_{11}X_1$ represents the change in distance between *B* and *F* due to the member force, X_1 in ***BF***. Δ_1 is zero without any fabrication error, but in this case 5 mm is assumed to have taken place.

Fig. 3.25(a) shows the member forces, $\{n_i^1\}$ resulting from a unit load applied to *B* and *F* in the state of ***BF*** member severed. Fig 3.25(b) shows the member forces, $\{n_i\}$ resulting from the nodal load in the state of ***BF*** member removed. The fact that the ***BF*** member is absent, both cases are now statically determinate structures.

▶Fig. 3.25 (a)





▶ Fig. 3.25(b)

f_{11} and D_1 are now found using the unit load method.

$$f_{11} = \frac{1}{EA}\sum n_i^1 n_i^1 l_i = \frac{1}{EA}[(-0.6)^2 \times (6{,}000) + (-0.8)^2 \times (8{,}000)$$
$$+ (1.0)^2 \times (10{,}000) \times 2 + (-0.8)^2 \times (8{,}000) + (-0.6)^2 \times (6{,}000)]$$
$$= \frac{34{,}560}{EA}$$

$$D_1 = \frac{1}{EA}\sum n_i^1 N_i l_i = \frac{1}{EA}[(-0.6) \times (-250) \times (6{,}000) + (-0.8) \times (333.333) \times (8{,}000)$$
$$+ (1.0) \times (208.333) \times (10{,}000) + (-0.6) \times (125) \times (6{,}000)]$$
$$= \frac{399{,}998.8}{EA}$$

$$\Delta_1 = 5 \text{ mm}$$

$$\frac{1}{\text{EA}} \cdot \left(34{,}560\text{X}_1 + 399{,}998.8\right) = 5 \text{ mm}$$

X_1 = 316.272

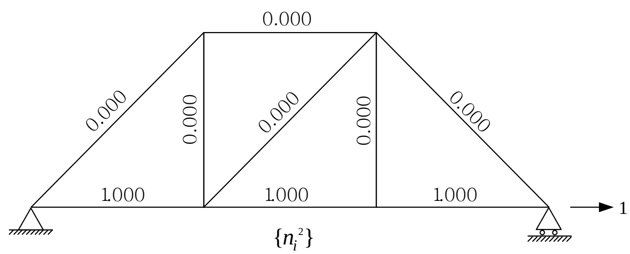
where, X_1 is the ***BF*** member force due to the loading and fabrication error.

Model 2 is both externally and internally indeterminate. The degree of overall indeterminacy is 2.

Degree of external indeterminacy: $m_e = n - 3 = 4 - 3 = 1$
Degree of internal indeterminacy: : $m_i = (3 + s + r) - 2k = (3 + 10 + 0) - 2 \times 6 = 1$

Since Model 2 is a structure of 2 degrees of static indeterminacy, 2 equations can be formulated with an unknown, X_1 for the ***BF*** member force and an unknown X_2 for the horizontal reaction at Node D. From Model 1, the unknown X_2 is added. The horizontal reaction at Node D is removed, and a unit load is applied at Node D. The member forces $\{n_i^2\}$ then become as Fig. 3.26.

▶ Fig. 3.26



Using the values found from Model 1 and the above $\{n_i^2\}$, the unit load method is applied to obtain the following results:

$$f_{11} = \frac{34{,}560}{EA}$$

$$f_{12} = \frac{1}{EA}\sum n_i^1 n_i^2 l_i = \frac{1}{EA}(-0.6)\times(1)\times(6{,}000) = -\frac{3{,}600}{EA}$$

$$f_{22} = \frac{1}{EA}\sum n_i^1 n_i^2 l_i = \frac{1}{EA}(1)^2\times(6{,}000)\times 3 = \frac{18{,}000}{EA}$$

$$D_1 = \frac{399{,}998.8}{EA}$$

$$D_2 = \frac{1}{EA}\sum n_i^2 N_i l_i$$

$$= \frac{1}{EA}[(1.0)\times(250)\times(6{,}000)+(1.0)\times(125)\times(6{,}000)\times 2] = \frac{3{,}000{,}000}{EA}$$

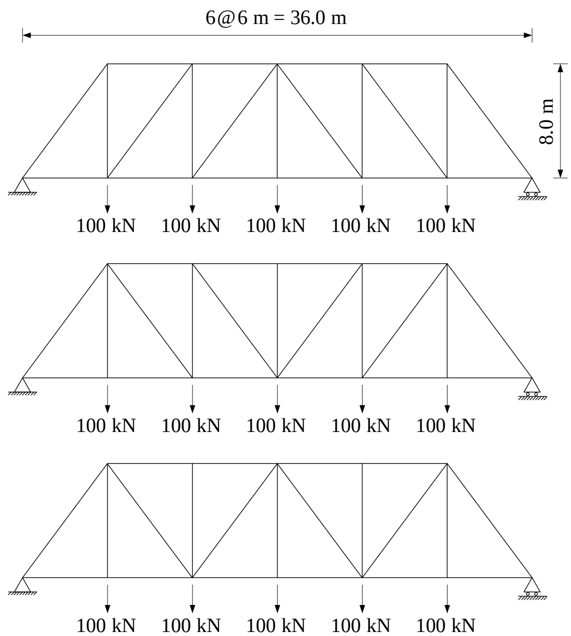
$$\frac{1}{EA}\begin{bmatrix} 34{,}560 & -3{,}600 \\ -3{,}600 & 18{,}000 \end{bmatrix}\begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} + \frac{1}{EA}\begin{Bmatrix} 399{,}998.8 \\ 3{,}000{,}000 \end{Bmatrix} = \begin{Bmatrix} 5 \\ 0 \end{Bmatrix}$$

$$\begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} = \begin{Bmatrix} 305.270\ \text{kN} \\ -105.613\ \text{kN} \end{Bmatrix}$$

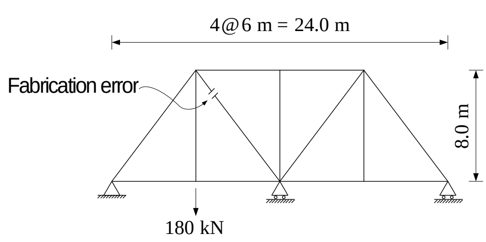
X_1 represents the ***BF*** member force, and X_2 represents the horizontal reaction at Node D.

4. Exercise

1. Find the member forces in the 3 trusses below. The material and section properties are identical to those of the tutorial.



2. Find the member forces in the statically indeterminate truss below. The truss is subject to a nodal load and a fabrication error as shown below. The material and section properties are identical to those of the tutorial.



4. **Arch** Analysis

Contents

1. Introduction

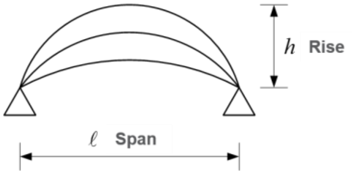
1.1 Concept for Arch Analysis

The arch structural system generates large horizontal reactions at the end supports while resisting vertical loads. With the presence of horizontal reactions, forces in the arch members are predominantly compression forces, which in turn reduce bending moments in the arch members.

▶Photo 4.1
Arch structures
(a) Arch bridge
(b) Arch structure

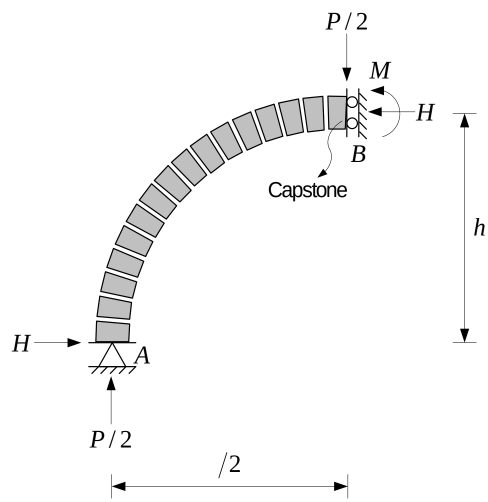
The ratio of the height (h) to span (l), h/l is referred to as the rise ratio. When the rise ratio is too small, the arch tends to spread out, thereby increasing the axial force in the arch members. On the contrary, a large rise ratio improves the structural performance as the arch members resist external loads through axial forces in the arch members. But a high arch required for the span may prove to be uneconomical. The rise ratio of about 0.3 is commonly known to be structurally efficient and yet economical.

▶Fig. 4.1
Arch rise ratio = h / l



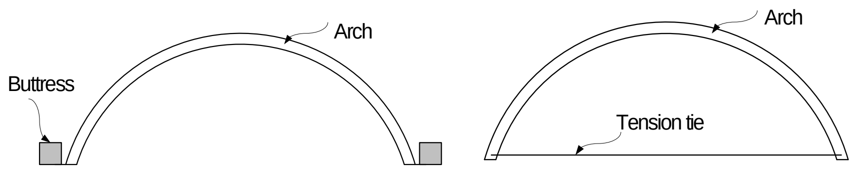
A masonry arch as shown in Fig. 4.2 is historically proven to be structurally stable. This is because the structure is designed to resist only axial forces. The arch structure shown is a half section, which is symmetrical about Point B. If tensile stresses develop at the bottom part of the capstone at Point B, the arch will become unstable leading to collapse. However, if the horizontal compression force, *H*, can cancel the tensile stresses resulting from the moment, *M*, then the arch is capable of resisting the loading.

▶Fig. 4.2
Arch structure



The horizontal reaction force, *H*, at the base Point A plays the role of preventing the arch from spreading out. In practice, the reaction can be provided by the form of a buttress or a tension tie as shown in Fig. 4.3.

▶Fig. 4.3
Supports for lateral thrust in arch structures



The condition of equilibrium at the center of the arch at Point B is expressed below.

▶Eq. 4.1

$$\frac{P}{2}\cdot\frac{l}{2}=M+H\cdot h$$

The left side of the equation represents the acting moment, and the right side represents the load resisting capability. One can observe that the lateral reaction, H and the height, h need to increase in order to compensate for increases in the loading, P and the span l. If the arch height, h becomes zero, H also becomes zero, hence becoming a simple beam. Similar to the fact that a large section is required to resist moment in a simple beam, the arch height, h will need to increase in order for the arch structure to resist moment. It is therefore advantageous to form the arch shape similar to the moment diagram shape of the structure.

2. Tutorial

2.1 Model Overview

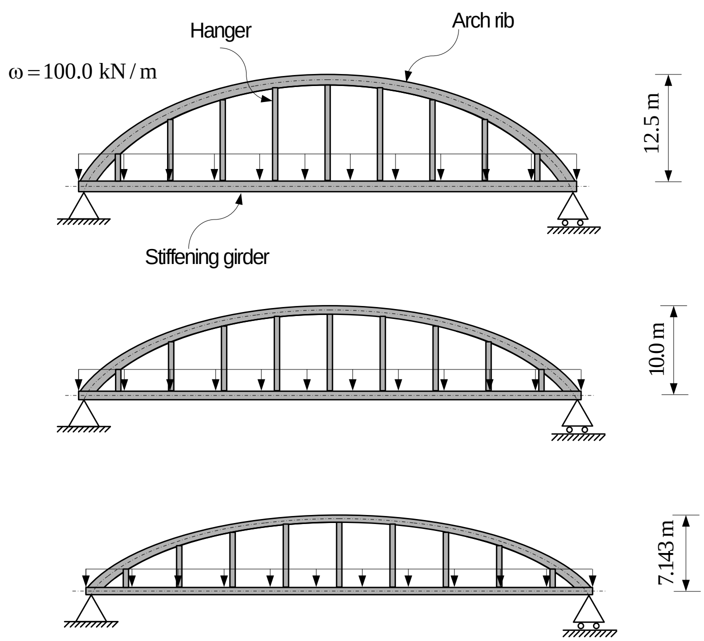
Analyze the following 3 arch structures of different rise ratios, h/l of 1:4, 1:5 and 1:7, and compare the displacements and member forces of the 3 cases.

▶Fig. 4.4
Analysis model

(a) Model 1

(b) Model 2

(c) Model 3



- ***Material***: Steel Fe570 (Modulus of elasticity, E = 2.05×10^8 kN/m^2)
- ***Section***

  Arch rib: Box shape 1,000×1,000×20 mm

  Stiffening girder: Box shape 1,000×1,000×20 mm

  Hanger: I-Section ISWB550
- ***Load***: uniformly distributed load 100.0 kN/m

2.2 Work Environment

Open a New File and save.

Main Menu: ***/ New Project...***

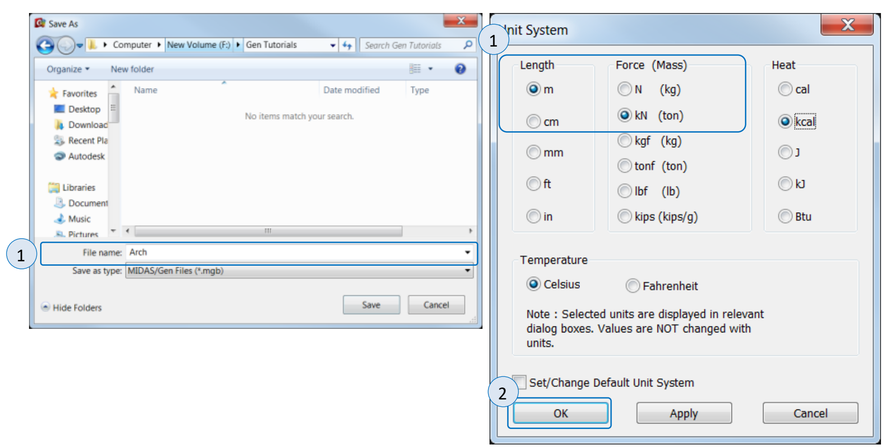
Main Menu: ***/ Save as...***

1. File name: **'Arch'** and [Save]

Set a unit system to kN & m.

Main Menu: ***Tools / Unit System***

1. Length > **m**, Force (Mass) > **kN (ton)**
2. **[OK]**

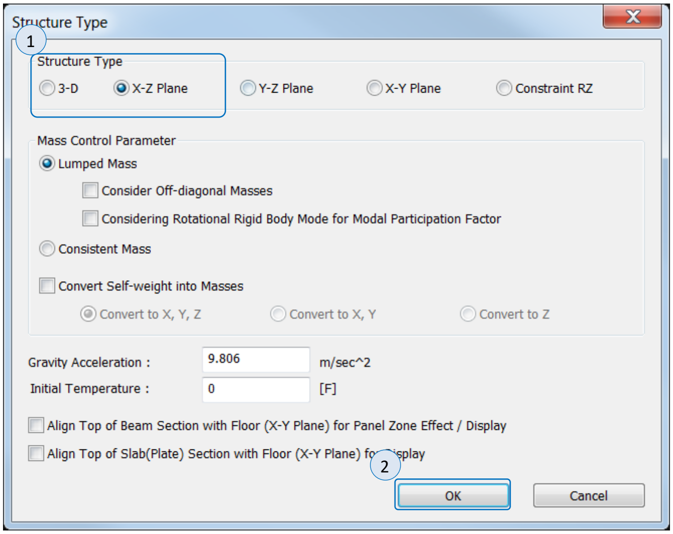


▶Fig. 4.5
Save file &
set unit system

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

Main Menu: ***Structure / Structure Type***
1. Structure Type > **X-Z Plane**
2. **[OK]**

▶Fig. 4.6
Set a work plane



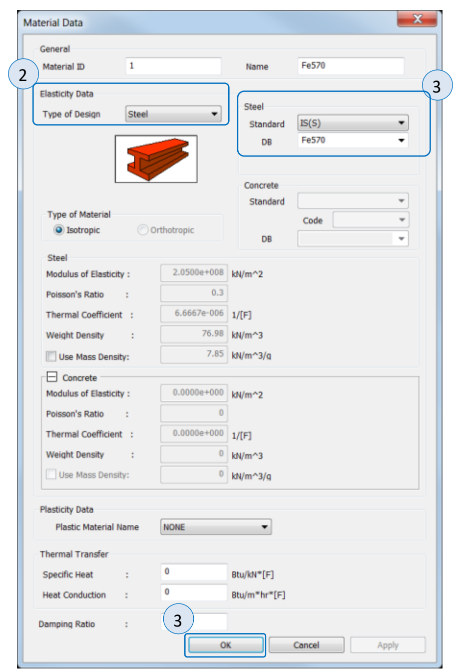
2.3 Material & Section Properties

Select IS(S), Fe570 for the material.   Select I-Section for the hangers, and define a built-up box section for the remainder.

Main Menu: ***Properties / Material Properties***

1. **[Add...]**
2. Elasticity Data > Type of Design > **Steel**
3. Steel > Standard > **IS(S)**, DB > **Fe570, [OK]**

▶Fig. 4.7
Define material



Tip

Name is automatically assigned if the material is selected from DB. Name can be revised.

Tip

Weight Density is used to automatically calculate the selfweight.

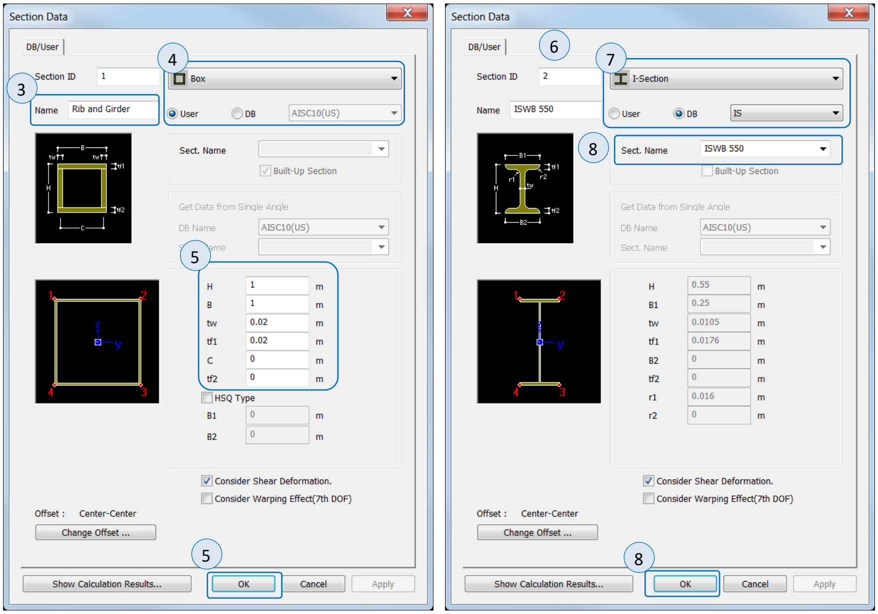
Tip

OK button closes the dialog box.   If multiple materials are defined, use Apply button.

Define the sections to be used in the model.

1. Properties dialog window: click **Section** tab.
2. **[Add...]**
3. Section ID: **1**, Name: **'Rib and Girder'**
4. Section definition method > **User,** Section shape list > **Box**
5. H: **'1'**, B: **'1'**, tw: **'0.02'**, tf1: **'0.02'**, **[Apply]**
6. Section ID: **2**
7. Section shape list > **I-Section**, Section definition method > **DB**, DB > **IS**
8. Sect. Name > **ISWB550**, **[OK]**
9. **[Close]**

▶Fig. 4.8
Define sections



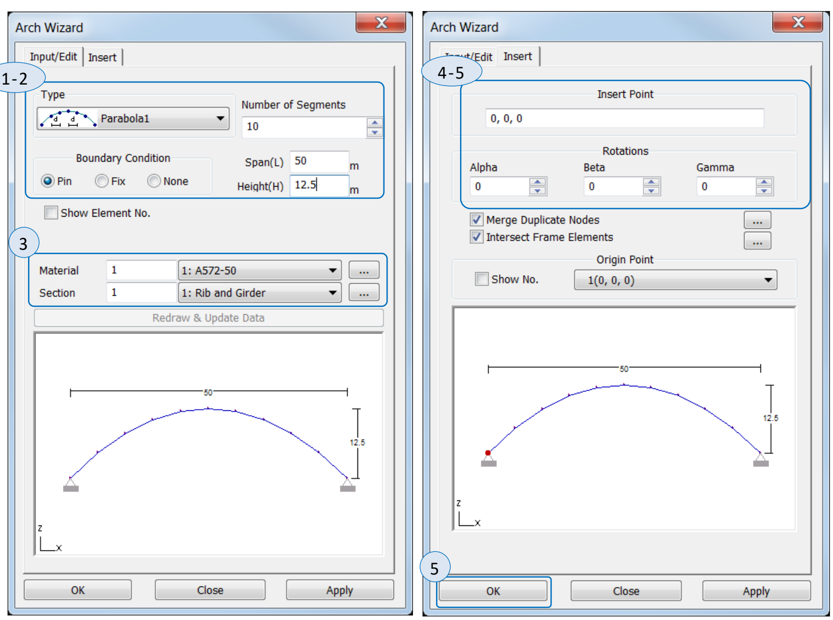
2.4 Nodes & Elements Generation

Create an arch rib with the rise ratio of 1:4 (Model 1) using Arch Wizard, which generates a series of straight beam elements in segments.

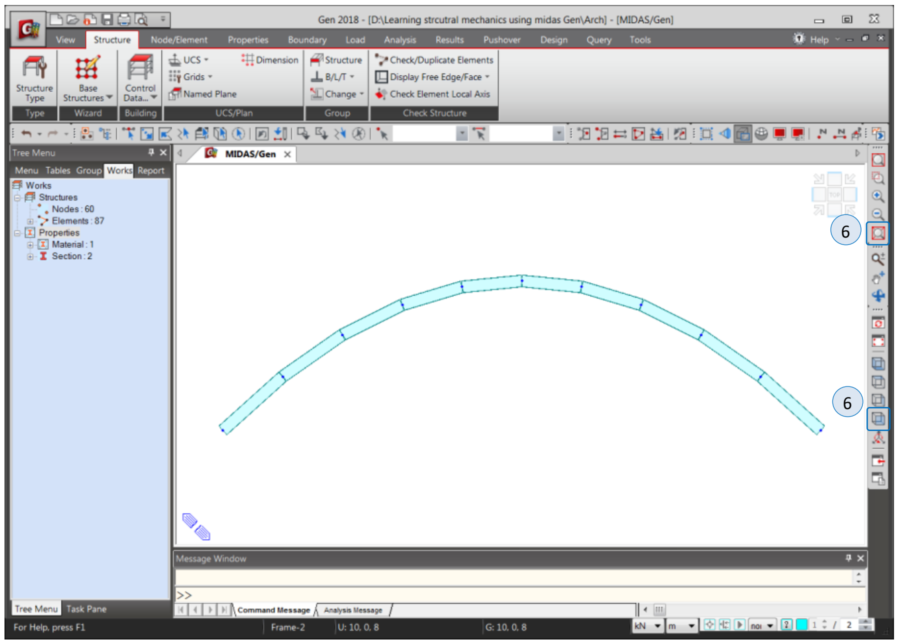
Main Menu: ***Structure / Base Structures / Arch...***

1. Input/Edit tab > Type: **Parabola1**, Number of Segments: **'10'**
2. Boundary Condition: None, Span(L): **'50'**, Height(H): **'12.5'**
3. Material > **1: Fe570**, Section > **1: Rib and Girder**
4. Insert tab > Insert Point: **'0, 0, 0'**
5. Rotations > Alpha: **'0'**, Beta: **'0'**, Gamma: **'0'**, **[OK]**
6. **Front View**, **Auto Fitting** (Toggle on)

▶Fig. 4.9
Arch Wizard



▶Fig. 4.10
Create arch rib

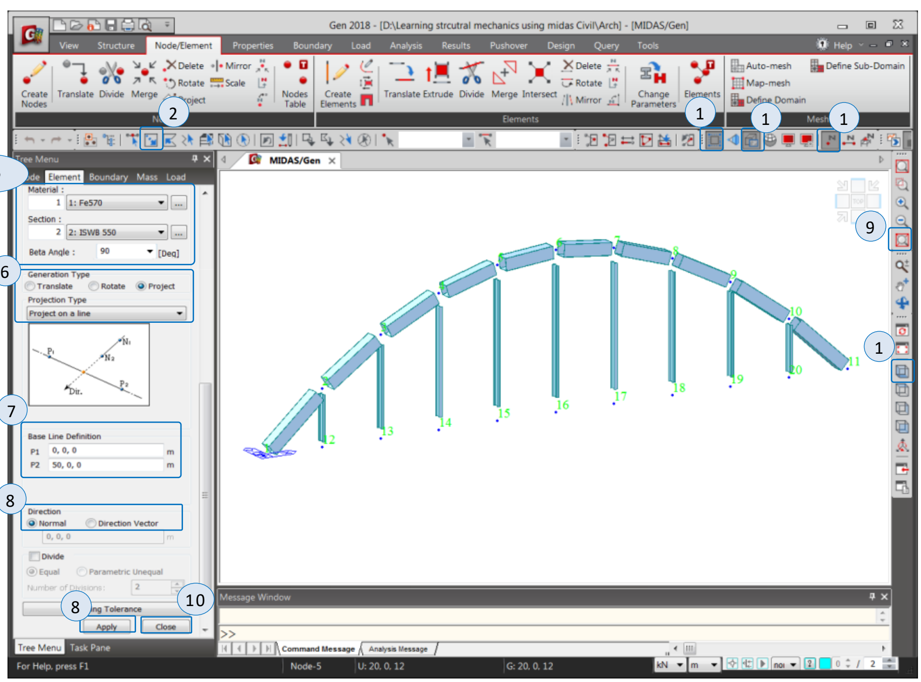


Create hangers by extruding Node to Line Element using Extrude Elements.

Main Menu: ***Node/Element / Extrude***

1. **Shrink, Hidden, Display Node Numbers, Iso View** (Toggle on)
2. **Select by Window** (on) mode: **Nodes 2-10**
3. Extrude Type > **Node → Line Element**
4. Element Attribute > Element Type > **Beam**
5. Material > **1: Fe570**, Section > **2: ISWB550**, Beta Angle > **90**
6. Generation Type > **Project**, Projection Type > **Project on a line**
7. Base Line Definition > P1: **'0, 0, 0'**, P2: **'50, 0, 0'**
8. Direction > **Normal**, **[Apply]**
9. **Auto Fitting** (Toggle on)
10. **[Close]**

▶Fig. 4.11
Create hangers

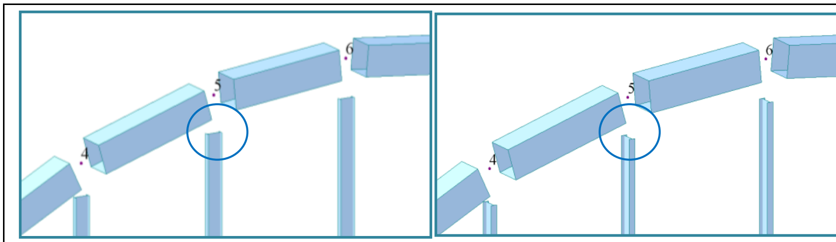


Tip

Beta Angle

Beta Angle 90˚ orients the hanger section web axis perpendicular to the span of the bridge. Refer to Create Elements in Online Manual for Beta Angle.

▶Fig. 4.12
Beta Angle



Bata Angle: 0[Deg]

Bata Angle: 90[Deg]

Tip

Beta Angle refers to the orientation of z-axis of Truss, Beam or Column line elements in Element Coordinate System.

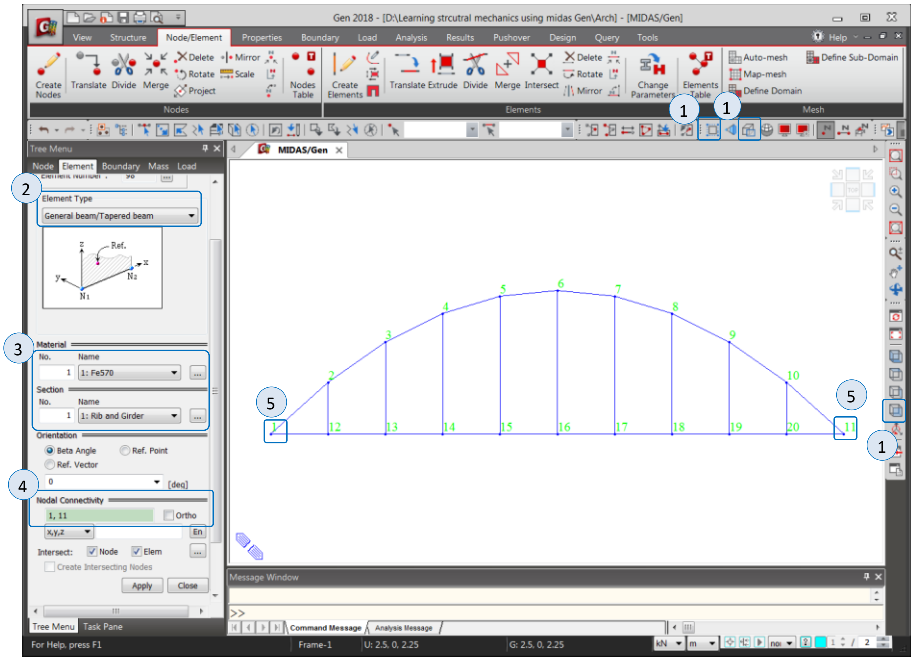
Unless a section is symmetrical in both axes (like a circle or square section), the stiffness of the section changes with Beta Angle.

Create the stiffening girder between Node 1 and Node 11.

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

1. **Shrink, Hidden** (Toggle off), **Front View** (Toggle on)
2. Element Type > **General beam/Tapered beam**
3. Material > Name > **1: Fe570**, Section > Name > **1: Rib and Girder**
4. Click the Nodal Connectivity field (which turns to green)
5. Click **Node 1** and **Node 11** on the model view sequentially.

▶Fig. 4.13
Input stiffening girder

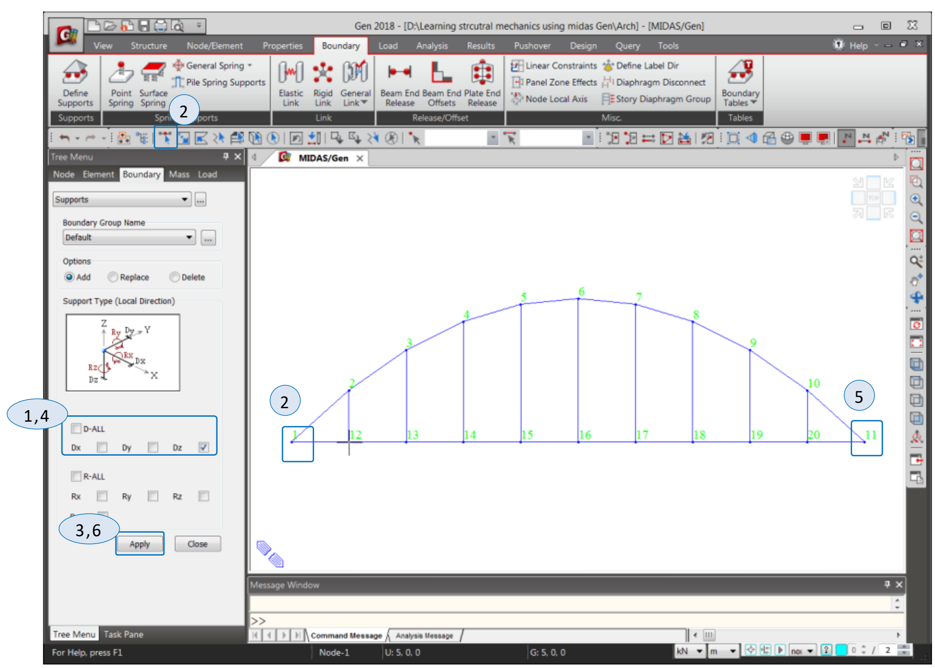


2.5 Define Boundary Conditions

The left support of Model 1 (Node 1) is assigned a pin support with Dx & Dz degrees of freedom restrained. The right support (Node 11) is assigned a roller support with Dz degree of freedom restrained.

Main Menu: ***Boundary / Define Supports***
1. Support Type > **Dx, Dz** (Check on)
2. **Select Single** (on) mode: select **Node 1**
3. **[Apply]**
4. Support Type > **Dx** (Check off), **Dz** (Check on)
5. Select **Node 11**
6. **[Apply]**

▶Fig. 4.14
Define support conditions

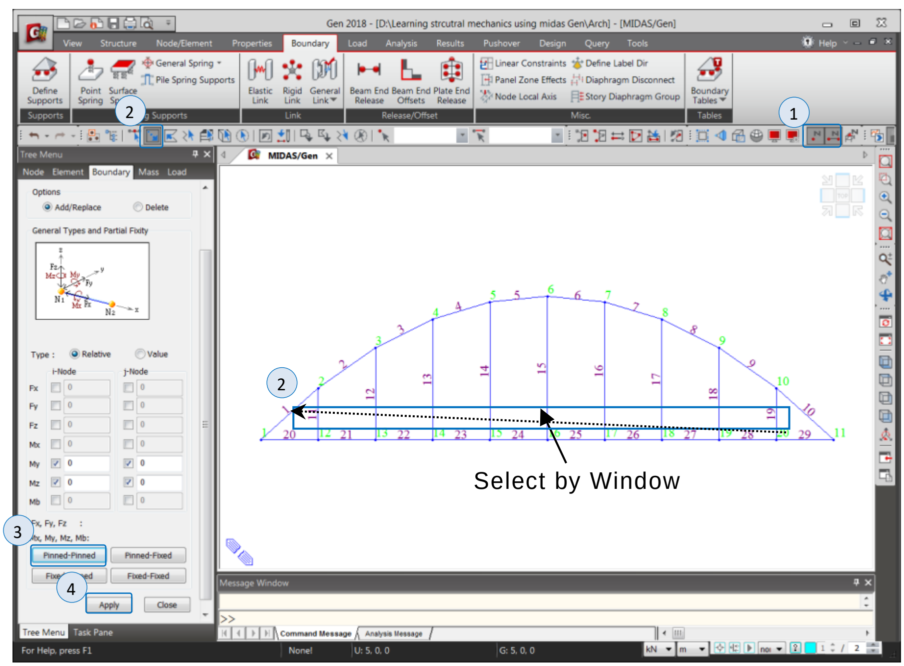


Define hinges at the ends of each hanger (Beam End Release).

Main Menu: ***Boundary / Beam End Release***

1. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
2. **Select by Window** (on) mode: drag a window from **Element 19** to **Element 11** (right to left). No elements other than the hangers should be within the window. Double-click to complete the selection.
3. General Types and Partial Fixity > **[Pinned-Pinned]**
4. **[Apply] [Close]**

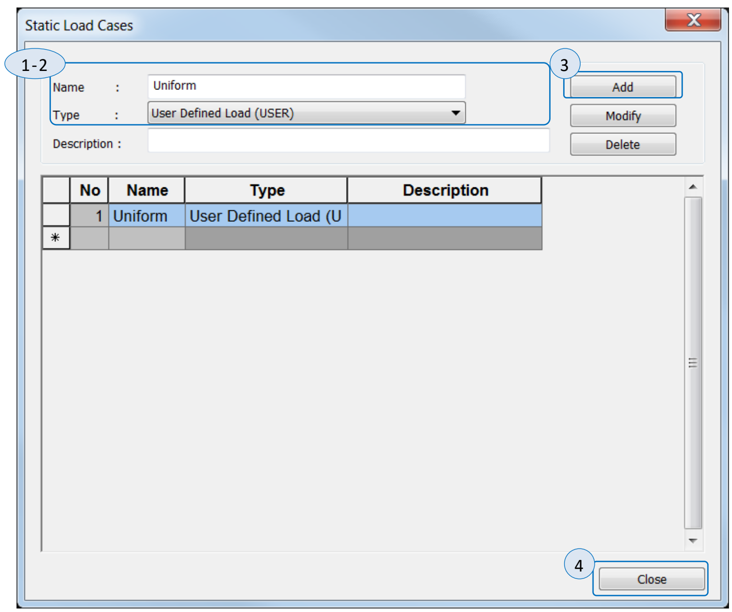
▶Fig. 4.15
Define hinge conditions at the ends of hangers



2.6 Define Loads

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

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Uniform'**
2. Type > **User Defined Load (USER)**
3. **[Add] [Close]**



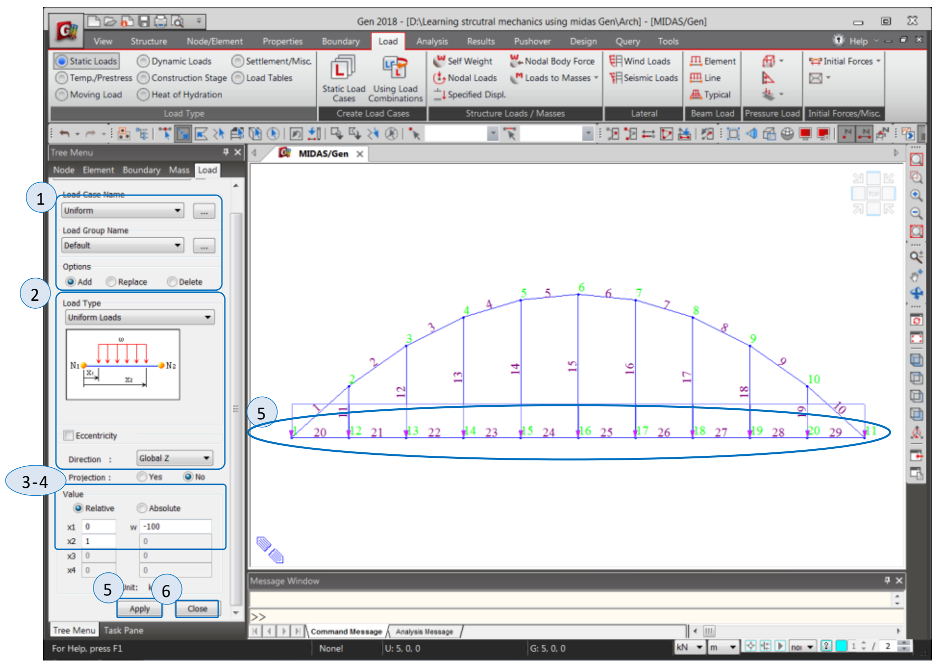
▶Fig. 4.16
Define load case

Input a uniformly distributed load, 100 kN/m on the stiffening girder.

Main Menu: ***Load / Static Loads / Element Beam Load***

1. Load Case Name > **Uniform**, Options > **Add**
2. Load Type > **Uniform Loads**, Direction > **Global Z**
3. Value > **Relative**
4. x1: **'0'**, w: **'-100',** x2: **'1'**
5. Select Girder (**Elements 20 - 29**), **[Apply]**
6. **[Close]**, Works Tree: right-click **Element Beam Loads: 10** and **Undisplay**

▶Fig. 4.17
Input uniformly distributed load

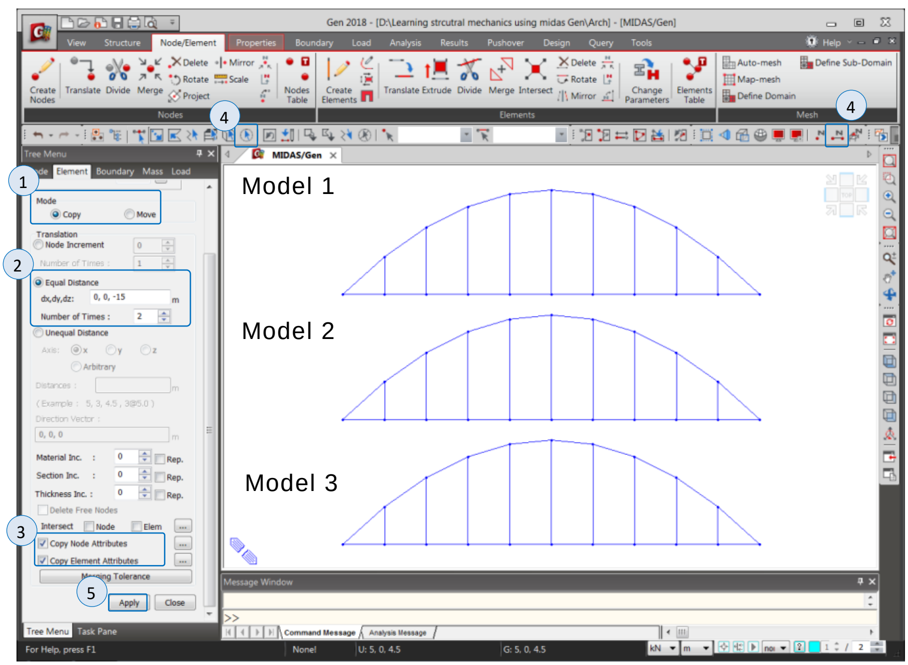


Copy Model 1 to create Model 2 and Model 3. The loading and boundary conditions assigned to Model 1 are simultaneously copied.

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

1. Mode > **Copy**
2. Translation > Equal Distance > dx, dy, dz: **'0, 0, -15'**, Number of Times: **'2'**
3. **Copy Node Attributes, Copy Element Attributes** (Check on)
4. **Display Element Numbers** (Toggle off), **Select All**
5. **[Apply]**

▶Fig. 4.18
Copy elements



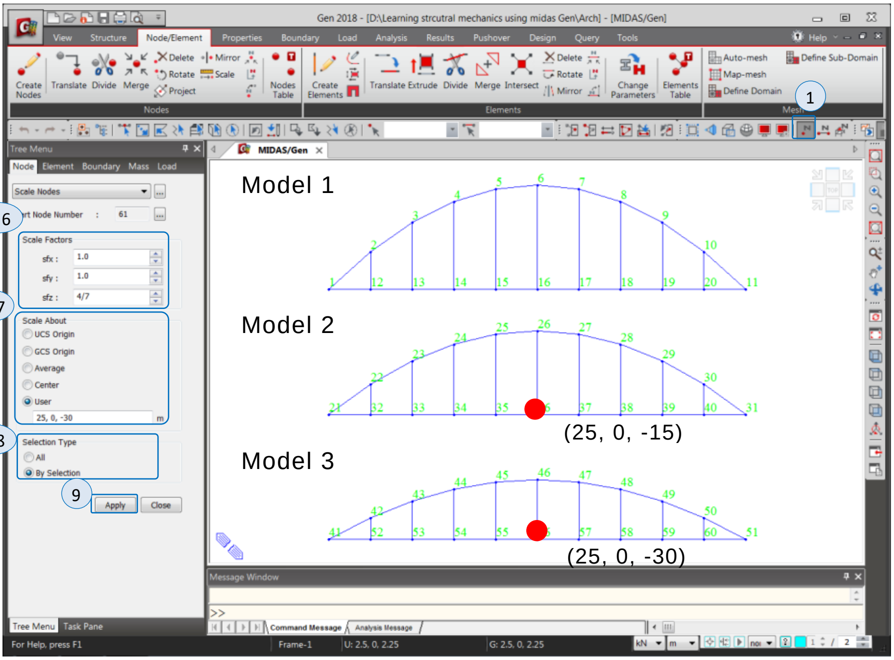
Change the heights of Model 2 and Model 3.

Main Menu: ***Node/Element / Scale***

1. **Display Node Numbers** (Toggle on)
2. Scale Factors > sfx: **'1.0'**, sfy: **'1.0'**, sfz: **'4/5'**
3. Scale About > **User**: **'25, 0, -15'**
4. Selection Type > **By Selection**
5. **Select by Windows** (on) mode: select **Nodes 22-30, [Apply]**
6. Scale Factors > sfx: **'1.0'**, sfy: **'1.0'**, sfz: **'4/7'**
7. Scale About > **User** : **'25, 0, -30'**
8. Selection Type > **By Selection**
9. **Select by Windows** (on) mode: select **Nodes 42-50, [Apply]**

Tip

'/' Symbol is the division operator within the software.



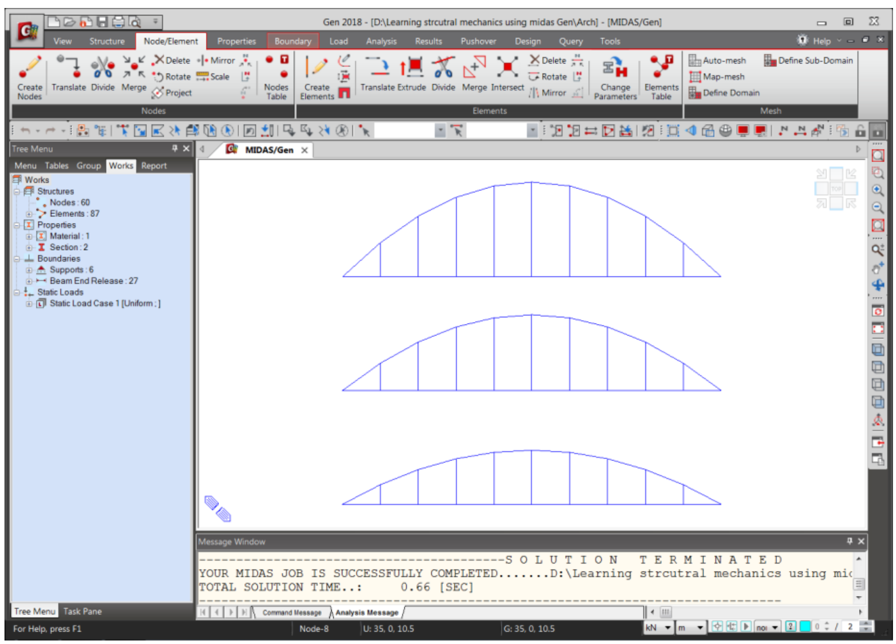
▶Fig. 4.19
Revise models

2.7 Perform Analysis

Analyze all three models.

Main Menu: ***Analysis / Perform Analysis***

▶Fig. 4.20
Message for a successful run



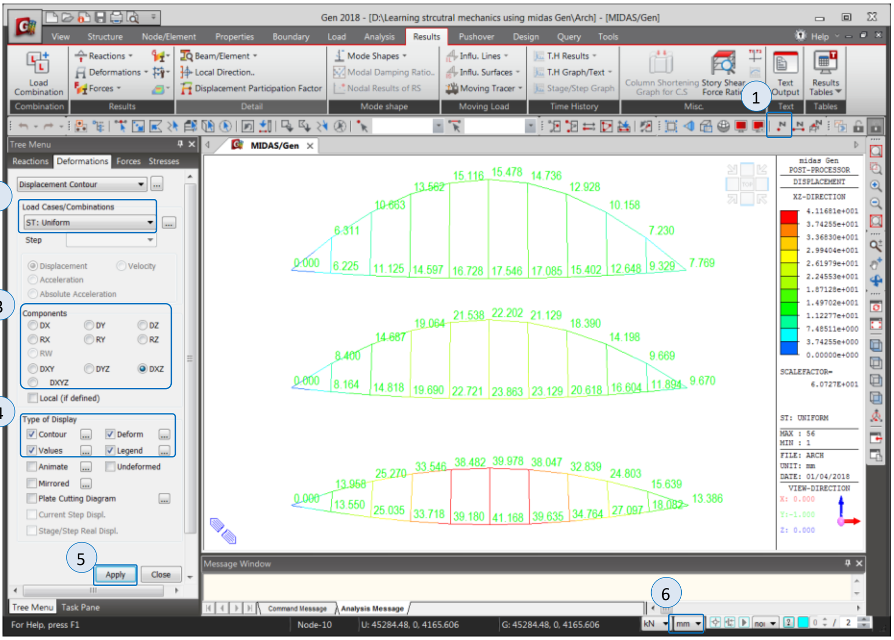
2.8 Check Analysis Results

Check the deformations. DXZ represents $\sqrt{DX^2 + DZ^2}$. The shallower the height of the arch becomes, the greater is the deflection. When identical members are used, a reduction in height signifies a reduction in stiffness.

Main Menu: ***Results / Deformations / Displacement Contour...***

1. **Display Node Numbers** (Toggle off)
2. Load Cases / Combinations > **ST: Uniform**
3. Components > **DXZ**
4. Type of Display > **Contour, Deform, Values, Legend** (Check on)
5. **[Apply]**
6. Change the unit to **mm** from the Status Bar at the bottom of the screen.

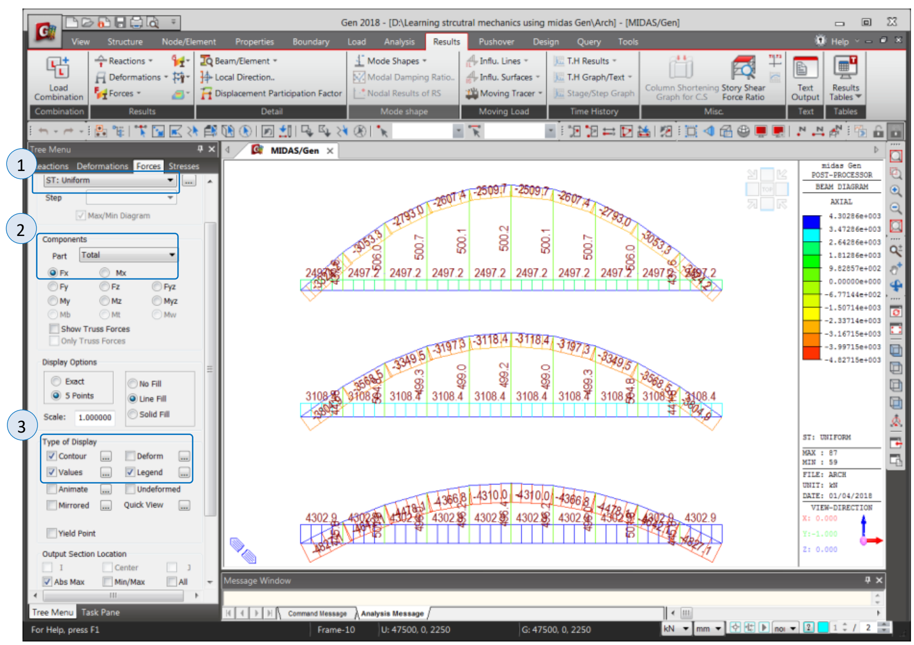
▶Fig. 4.21
Deformation results



Check the axial forces due to the uniformly distributed load. It can be observed that a lower arch height results in higher axial forces.

Main Menu: ***Results / Forces / Beam Diagrams...***
1. Load Cases/Combinations > **ST: Uniform**
2. Components > **Fx**
3. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
4. **[Apply]**

▶Fig. 4.22
Axial forces

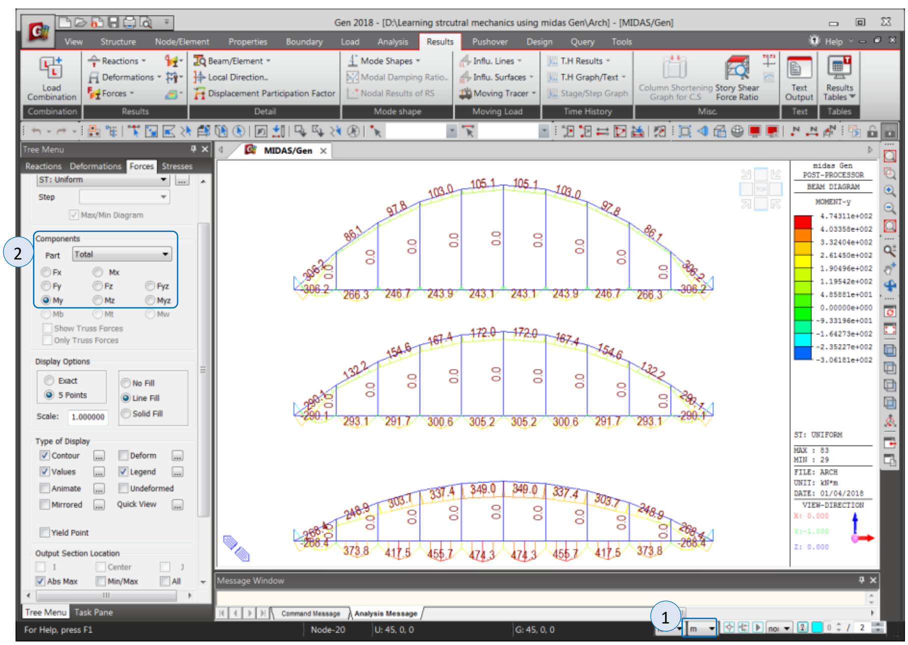


Check the bending moments. The shallower the arch height becomes, the higher are the absolute bending moments in the stiffening girder and arch rib. It can be also observe that lower arch heights result in reduction in bending moments where the arch rib and stiffening girder meet at each end.

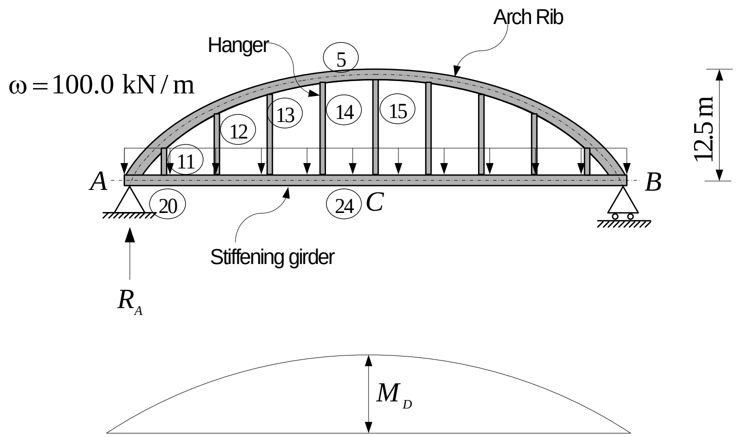
Main Menu: ***Results / Forces / Beam Diagrams...***

1. Change the unit to **m** from the Status Bar at the bottom of the screen.
2. Components > **My**
3. **[Apply]**

▶Fig. 4.23
Moment results



3. Manual Calculations



▶Fig. 4.24

Though Model 1 is an arch structure, the overall behavior resembles that of a simple beam. The arch shape also resembles the moment diagram. In a simple beam structure subject to a uniformly distributed load, one may design the beam with a larger section near the mid-span, and progressively smaller sections towards the ends. Similarly in an arch structure, the height at the mid-span is the highest, with the highest moment resistance, and the height gradually reduces to zero towards the ends.

A structure like Model 1, bending moment can be resisted by a couple in the top chord (arch rib) and the bottom chord (stiffening girder). That is, the tension in the bottom chord (or the compression in the top chord) multiplied by the height becomes the moment resistance at a corresponding location.

The following table shows the element numbers with the corresponding forces, which will be explained below.

▶Table 4.1

Element No	N (kN)	V (kN)	M (kN·m)
24	2496.743	-	-64.784
5	-2509.24	-	105.495
20	-	310.204	-
11	433.461	-	-
12	505.076	-	-
13	501.131	-	-
14	500.052	-	-
15	500.150	-	-

A simple beam subject to a uniformly distributed load is now assumed in order to examine the relationship between the moment and the member forces at the mid-span. The moment, M_o at the center Point C and the reaction, R_A at Point A are calculated as,

$$M_o = \frac{\omega l^2}{8} = \frac{100 \times 50^2}{8} = 31,250 \text{ kN} \cdot \text{m}$$

$$R_A = \frac{\omega l}{2} = \frac{100 \times 50}{2} = 2,500 \text{ kN}$$

In the arch structure, the moment, M_o is resisted by a couple in the top and bottom chords. As such, the moment, M_o is divided by the height at the center.

$$N = \frac{M_o}{h} = \frac{31,250}{12.5} = 2,500 \text{ kN}$$

The reason that this couple force (axial force) is different from the analysis values in Elements 24 and 5 is that the arch rib (Element 5) and the stiffening girder (Element 24) are also subject to moments. If these moments are hence considered, Element 24 will end up with the same axial force.

$$N = \frac{M_o - M_5 - M_{24}}{h} = 2,496.743 \text{ kN}$$

Similarly the same axial force can be obtained if the axial force in the top chord Element 5 is converted into components in the horizontal direction.

The reaction at Point A can be obtained by adding the shear force in Element 20 and the axial forces in the hangers (Elements 11-15), which then coincides with the simple beam reaction.

$$R_A = V_{20} + N_{11} + N_{12} + N_{13} + N_{14} + \frac{N_{15}}{2} = 2,500 \text{ kN}$$

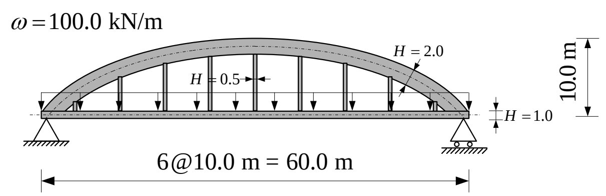
From the above analysis, it can be observed that an arch structure resists external loads through axial forces, and its shape resembles the bending moment diagram, which resists acting loads.

4. Exercise

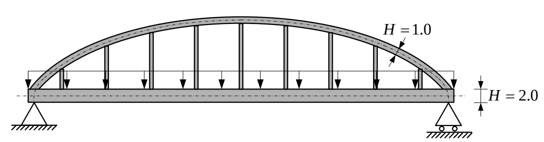
Analyze the following 3 arch structures of varying stiffness (different sections), and compare the behaviors of the arches affected by the difference in stiffness. Material properties are identical to those of the tutorial.
Consider the Box sections in DB/User – user input

1) Arch rib:	□ 2000×2000×20 mm
	□ 1000×1000×20 mm
	□ 1500×1500×20 mm
2) Girder:	□ 1000×1000×20 mm
	□ 2000×2000×20 mm
	□ 1500×1500×20 mm
3) Hanger:	ISWB 550

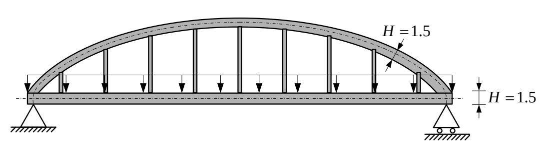
(a)



(b)



(c)



5. Continuous Beam Analysis

Contents

1. Introduction

1.1 Concept for Continuous Beam Analysis

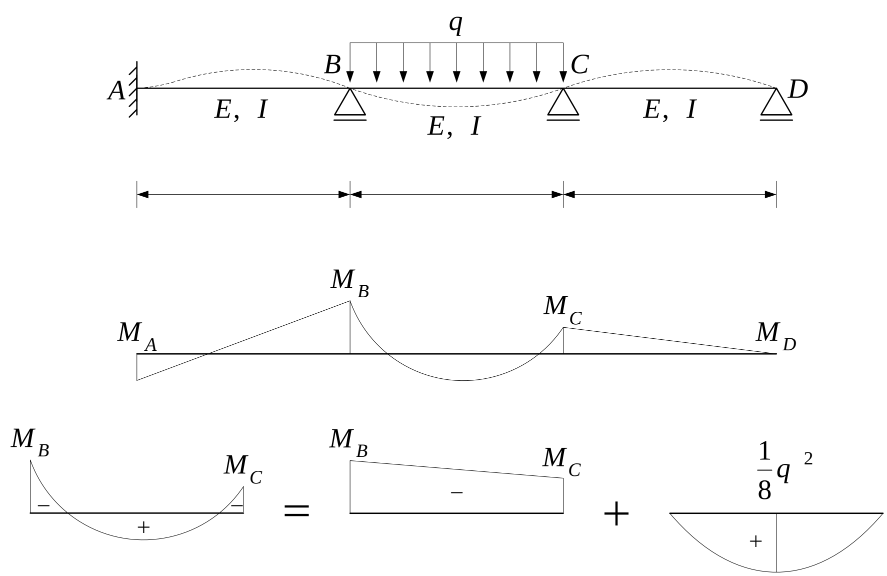
A beam structure consisted of 2 or more connected spans is referred to as a continuous beam.

▶Photo 5.1
Simple beam and continuous beam

In a continuous beam, the number of unknowns (reactions) is more than the number of force equilibrium conditions, hence becoming a statically indeterminate structure. This leads to formulating additional conditions such as deformation compatibility conditions in order to find the unknowns.

One of the characteristics of a continuous beam is that the behavior is affected by the flexural stiffness of adjacent members (spans). Fig. 5.1 shows a 3-span continuous beam in which the moment distribution of Member BC depends on the flexural stiffness of the adjacent Members AB and CD.

If Member BC were a simply supported beam, the moments at the ends B and C would have been zero. Instead, the end moments, M_B and M_C are non-zeros whose magnitudes depend on the flexural stiffness of Members AB and CD. The stiffness of Member AB is greater than that of Member CD because of the rotational restraint at Point A. The difference in stiffness leads to M_B whose magnitude is greater than M_C.

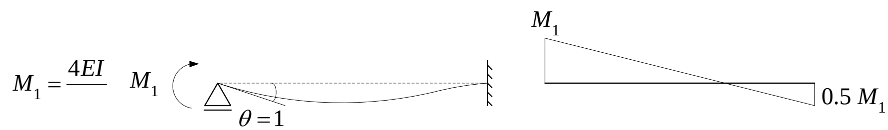


▶Fig. 5.1
3-Span continuous beam

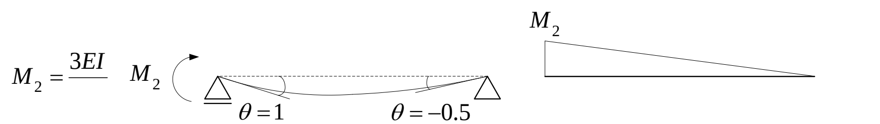
Fig. 5.2 illustrates the beam stiffness, which varies with boundary conditions. It is noted that the moment (M_1, M_2, M_3, M_4) required to cause a unit rotational angle at the left end depends on the restraint condition at the right end. Fig. 5.2(a) shows a fixed condition at the right end. Fig. 5.2(b) is hinged at the right end. Fig. 5.2(c) is imposed by the same unit rotational angle at the right end, hence resulting in a reverse curvature shape. Fig. 5.2(d) is subject to an opposite unit rotational angle (-1) at the right end, which causes a symmetrically deformed shape. The flexural stiffness of the 4 cases are $(6EI/l)$in 5.2(c), $(4EI/l)$ in 5.2(a), $(3EI/l)$ in 5.2(b) and$(2EI/l)$in 5.2(d) in descending order.

Fig. 5.2(a) corresponds to Member AB in Fig. 5.1, and Fig. 5.2(b) corresponds to Member CD in Fig. 5.1.

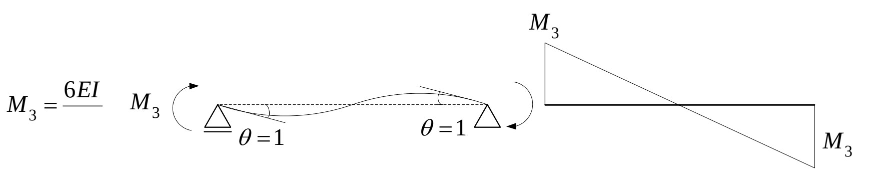
▶Fig. 5.2 (a)



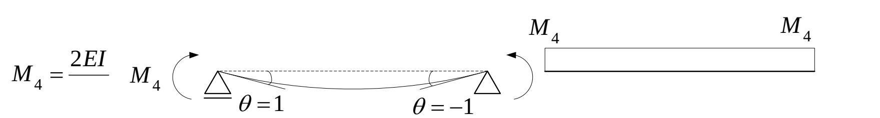
▶Fig. 5.2 (b)



▶Fig. 5.2 (c)



▶Fig. 5.2 (d)

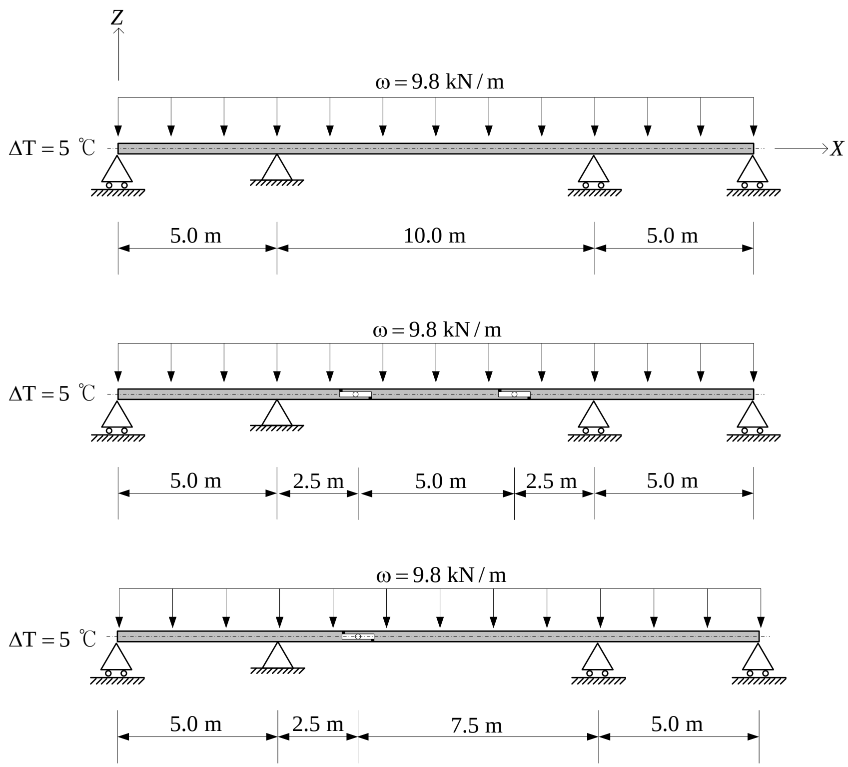


Though different analysis methods can be used to solve a statically indeterminate structure such as stiffness method, flexibility method, moment distribution method, finite element method, etc., the principle of Fig. 5.2 is used in all the methods. Each method is appropriate for a specific type of problem. In this chapter, the moment distribution method and flexibility method will be used for manual calculations. The moment distribution method is relatively simple to use, which is also intuitive in understanding the characteristics of a simple structure.

2. Tutorial

2.1 Model Overview

Analyze the following continuous and Gerber beams subject to a uniformly distributed load and a temperature load (temperature difference between top and bottom faces), and compare the reactions, displacements and member forces of the 3 cases.



▶Fig. 5.3

(a) Model 1
3 Span continuous
2 Degrees of indeterminacy

(b) Model 2
3 Span continuous
determinate

(c) Model 3
3 Span continuous
1 Degree of indeterminacy

- ***Material***: Steel Fe570
- ***Section***: Box shape 300 ×200×12 mm
- ***Load***: Uniformly distributed load 9.8 kN/m
  Temperature load: ΔT = 5 ℃
  (Difference in temperature between top & bottom)

2.2 Work Environment

Open a New File and save.

Main Menu: */ New Project...*

Main Menu: */ Save as...*

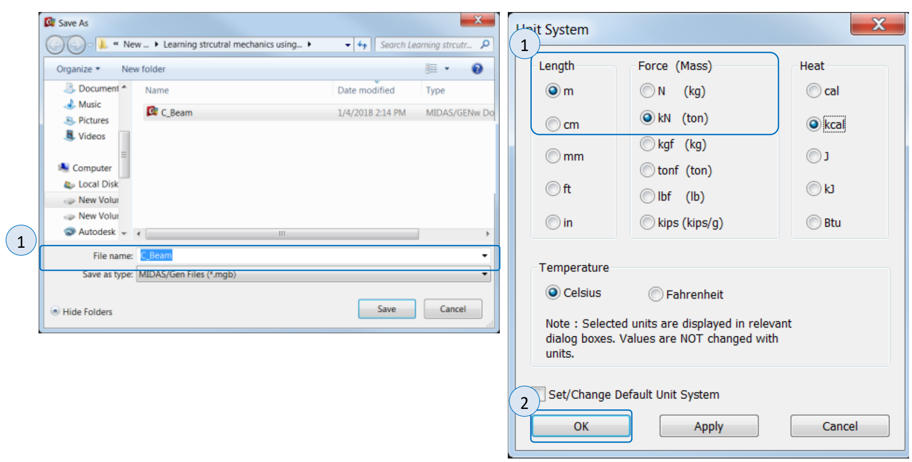
1. File name: **'C-Beam'** and [Save]

Set a unit system to kN & m.

Main Menu: ***Tools / Unit System***

1. Length > **m**, Force (Mass) > **kN (ton)**
2. **[OK]**

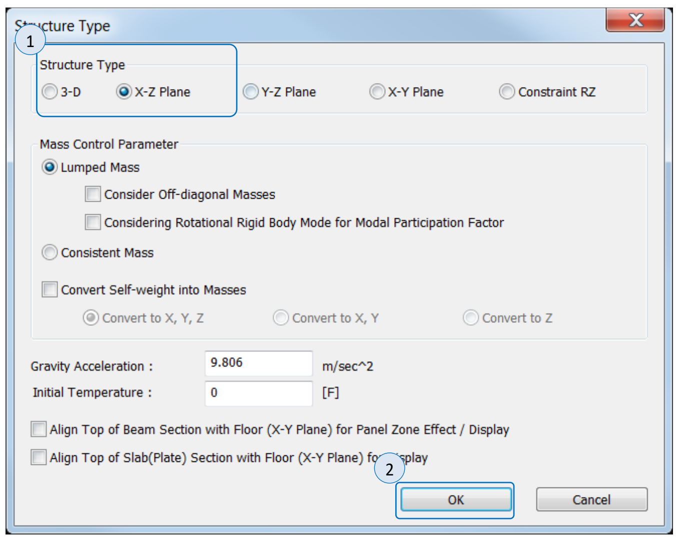
▶Fig 5.4
Save file &
set unit system



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

Main Menu: ***Structure / Structure Type***
1. Structure Type > **X-Z Plane**
2. **[OK]**

▶Fig. 5.5
Set a work plane



2.3 Material & Section Properties

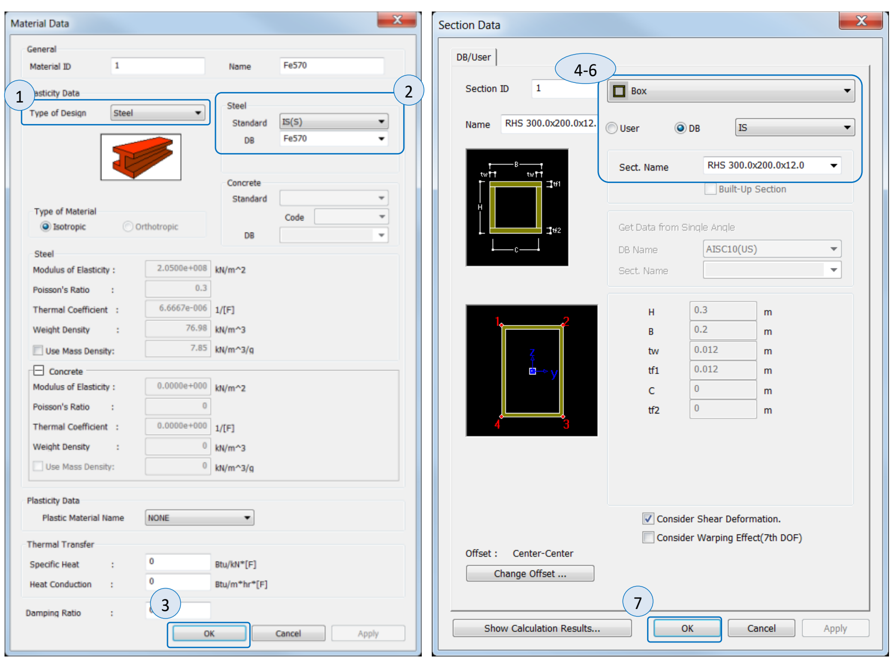
Select IS(S), Fe570 for the material and select a box section.

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

1. **[Add...]**, Type of Design > **Steel**
2. Steel > Standard > **IS(S)**, DB > **Fe570, [OK]**
3. Properties dialog window: click **Section** tab.
4. **[Add...]**, Section shape list > **Box**
5. Section definition method > **DB**, DB > **IS**
6. Sect. Name > **RHS 300x200x12**
7. **[OK]**
8. **[Close]**

Tip

If Material Properties are defined from DB, basic properties like modulus of elasticity, Poisson's ratio, thermal coefficient and weight density are automatically assigned.



▶Fig. 5.6
Define material
Define section

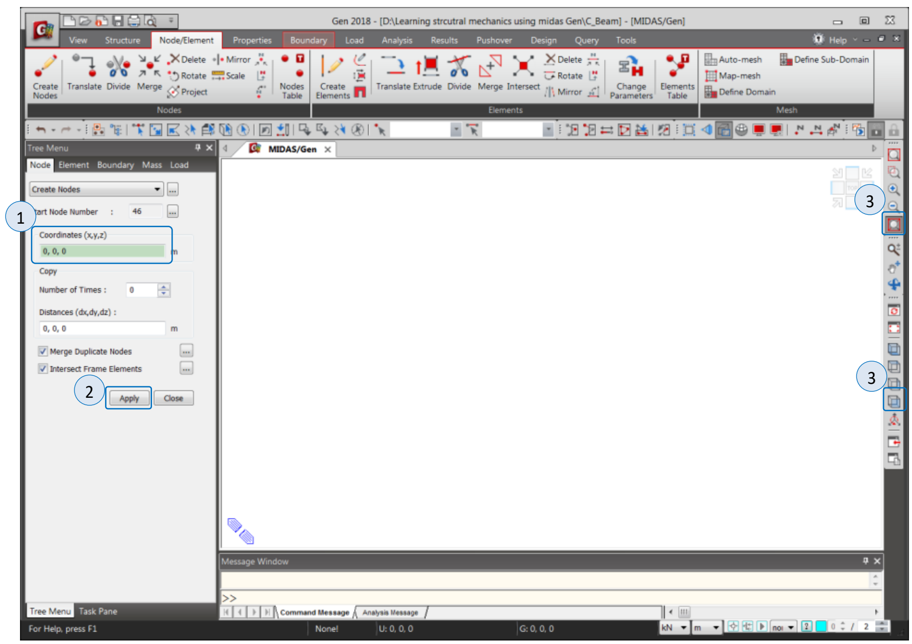
2.4 Nodes & Elements Generation

Create a node first in order to create a continuous beam.

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

1. Coordinates (x, y, z): **'0, 0, 0'**
2. **[Apply]**
3. **Auto Fitting, Front View** (Toggle on)

▶Fig. 5.7
Create a node

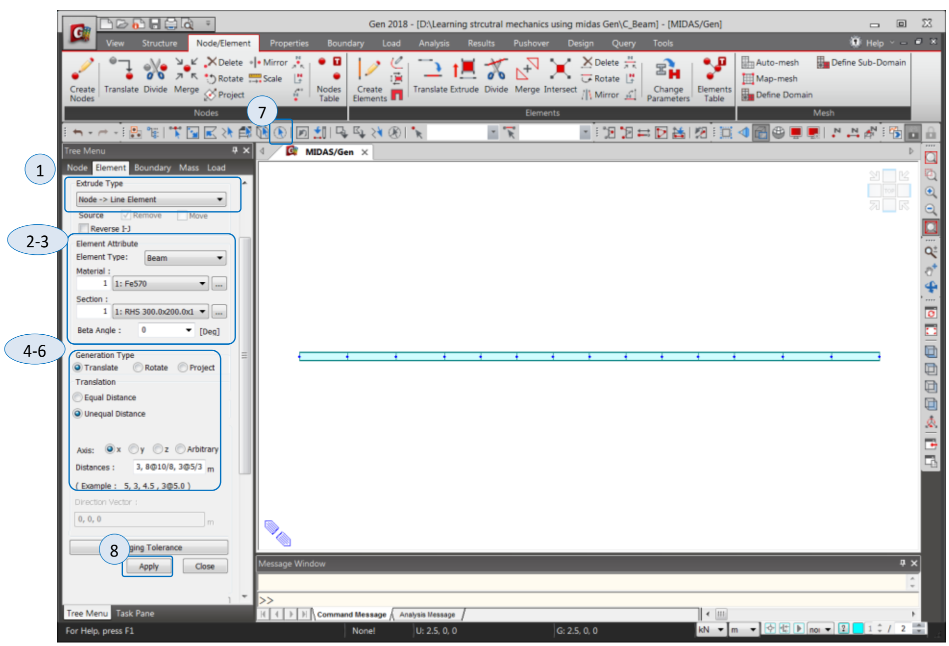


Create a continuous beam by extruding a node to line elements using Extrude Elements.

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

1. Extrude Type > **Node → Line Element**
2. Element Attribute > Element Type > **Beam**
3. Material > **1: Fe570**, Section > **1: RHS 300x200x12**, Beta Angle: **‘0’**
4. Generation Type > **Translate**
5. Translation > **Unequal Distance**
6. Axis > **x**, Distance: **‘3@5/3, 8@10/8, 3@5/3’**
7. **Select All**
8. **[Apply]**

▶Fig. 5.8
Create elements



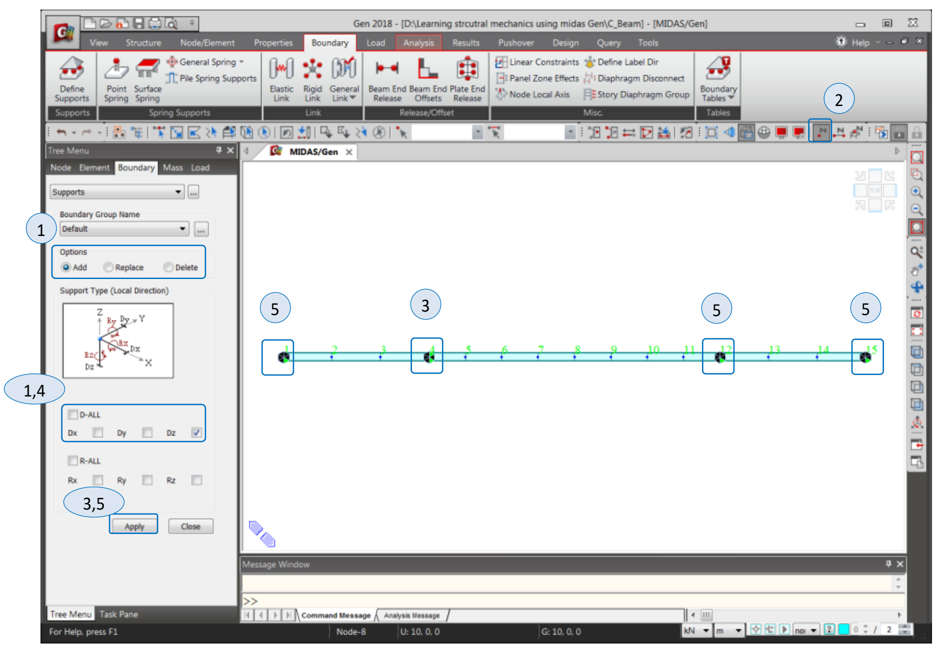
2.5 Define Boundary Conditions

Dx and Dz degrees of freedom are restrained for a pin support, and Dz degree of freedom is restrained for a roller support.

Main Menu: ***Boundary / Define Supports***

1. Options > **Add**, Support Type > **Dx, Dz** (Check on)
2. **Display Node Numbers** (Toggle on)
3. **Select Single** (on) mode: select **Node 4**, **[Apply]**
4. Support Type > **Dx** (Check off), **Dz** (Check on)
5. Select Single (on) mode: select Nodes 1, 12, 15, [Apply]

▶Fig. 5.9
Define support conditions

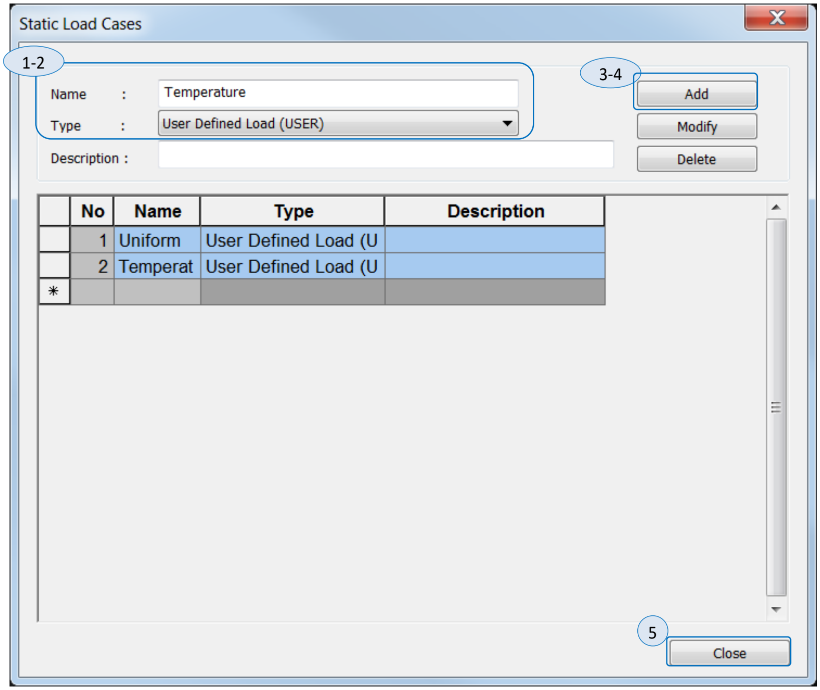


2.6 Define Loads

Define the load cases (load types) first to which the uniformly distributed load and the temperature load will belong.

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Uniform'**
2. Type > **User Defined Load (USER), [Add]**
3. Name: **'Temperature'**
4. Type > **User Defined Load (USER), [Add]**
5. **[Close]**

▶Fig. 5.10
Define load cases

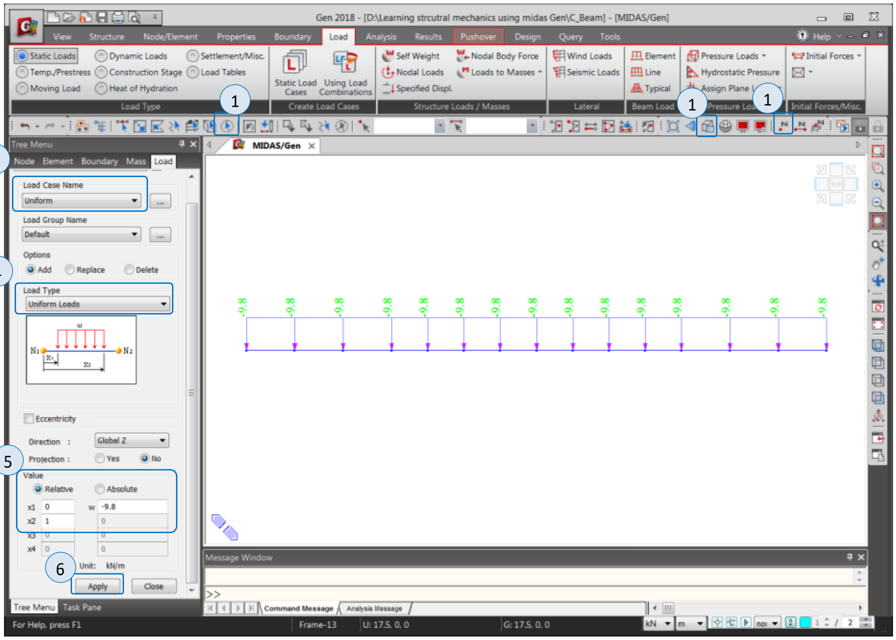


Input a uniformly distributed load, 9.8 kN/m on the continuous beam.

Main Menu: ***Load / Static Loads / Element Beam Load***

1. **Hidden, Display Node Numbers** (Toggle off), **Select All** (Toggle on)
2. Main Menu: ***View / Display...***
   Load tab: **Load Value** (Check on), **[OK]**
3. Load Case Name > **Uniform**
4. Load Type > **Uniform Loads**
5. Value > Relative x1: **'0'**, w: **'-9.8'**, x2: **'1'**
6. **[Apply]**

▶Fig. 5.11
Input uniformly distributed load

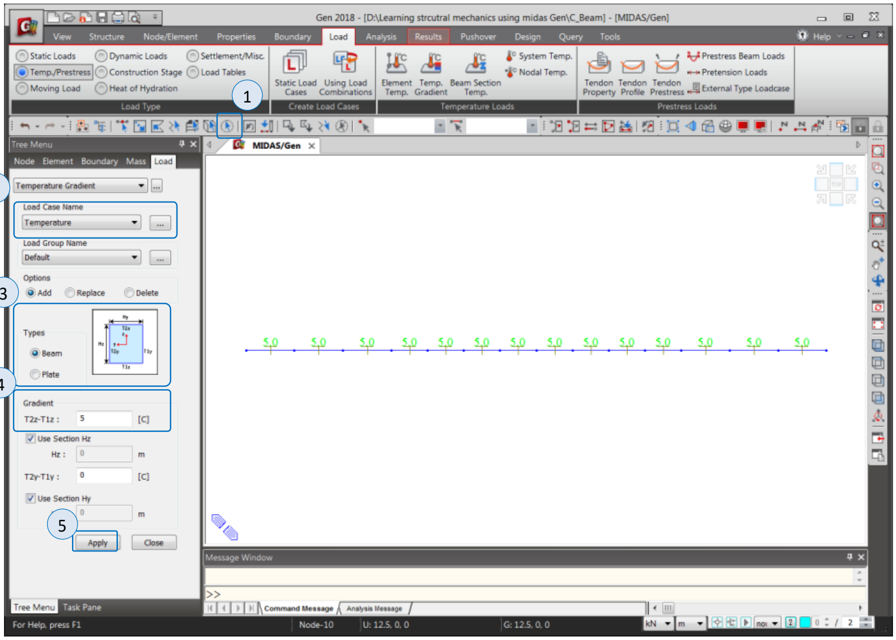


Input a temperature difference (ΔT = 5 ℃) between the top and bottom faces of the beam. Thermal expansion coefficient in the material properties will be used to convert the temperature difference to stresses imposed on the beam.

Main Menu: ***Load / Temp./Prestress / Temp. Gradient***

1. **Select All** (Toggle on)
2. Load Case Name > **Temperature**
3. Types > **Beam**
4. Gradient > T2z-T1z: **'5'**
5. **[Apply]**

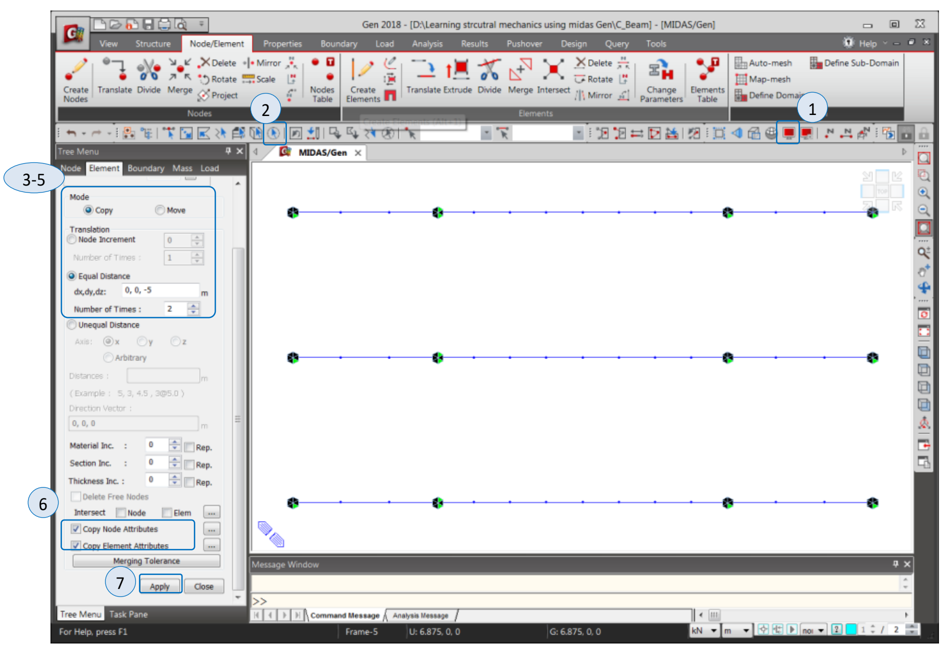
▶Fig. 5.12
Input temperature load



Copy Model 1 to create the Gerber beams, Model 2 and Model 3. Apply Copy Node Attributes and Copy Element Attributes in order to simultaneously copy the loads and boundary conditions assigned to Model 1.

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

1. **Display** (Toggle on)
   Boundary tab: **Support** (Check on), **[OK]**
2. **Select All** (on)
3. Mode > **Copy**
4. Translation > **Equal Distance**
5. dx, dy, dz: **'0, 0, -5'**, Number of Times: **'2'**
6. **Copy Node Attributes, Copy Element Attributes** (Check on)
7. **[Apply]**

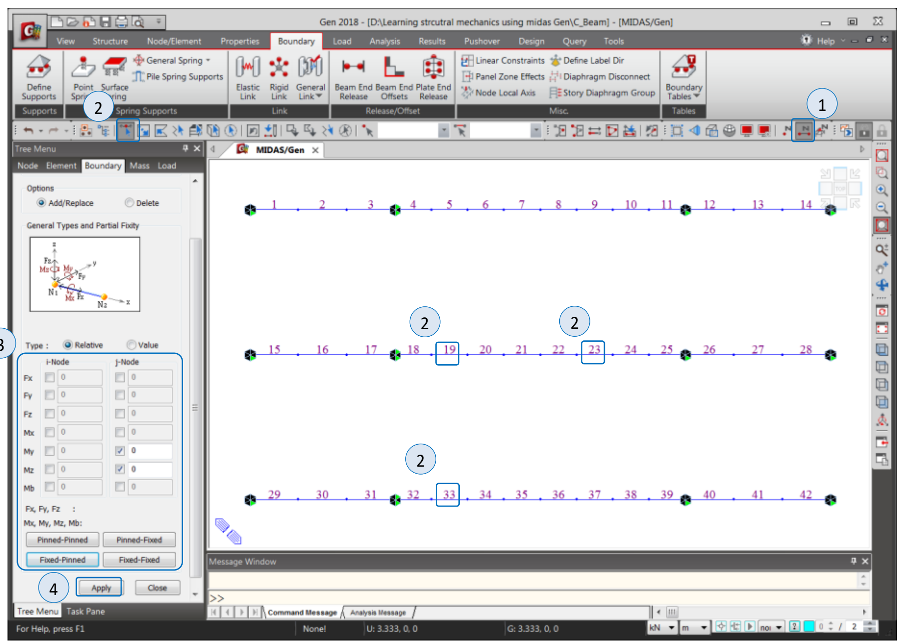


▶Fig. 5.13
Copy elements

Assign hinge conditions to the Gerber beam Models 2 and 3, by defining releases at the ends of the corresponding beam elements.

Main Menu: ***Boundary / Beam End Release***

1. **Display Element Numbers** (Toggle on)
2. **Select Single** (on) mode: select **Elements 19, 23, 33**
3. General Types and Partial Fixity > **[Fixed-Pinned]**
   Check to see: i-Node > My, Mz (Check off), j-Node > My, Mz (Check on)
4. **[Apply]**



▶Fig. 5.14
Input Gerber beam pin conditions

Tip

i-node and j-node are the first and last nodes respectively defining the corresponding element, which determines the element local x-axis.

Check on Local Axis or Local Direction in Element tab in [Display] in order to see the element direction (i-node & j-node).

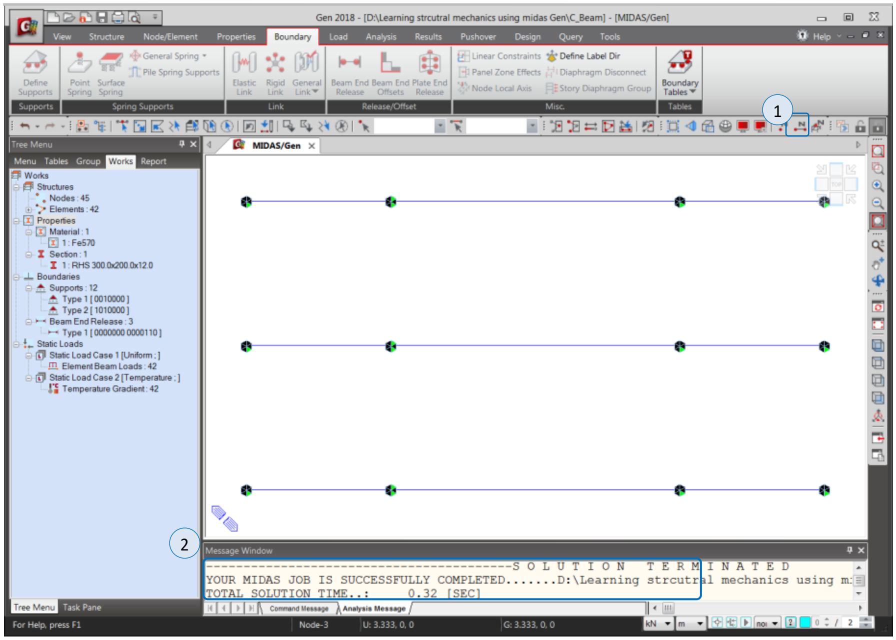
2.7 Perform Analysis

Analyze all three models.

Main Menu: ***Analysis / Perform Analysis***

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

▶Fig. 5.15
Message for a successful run



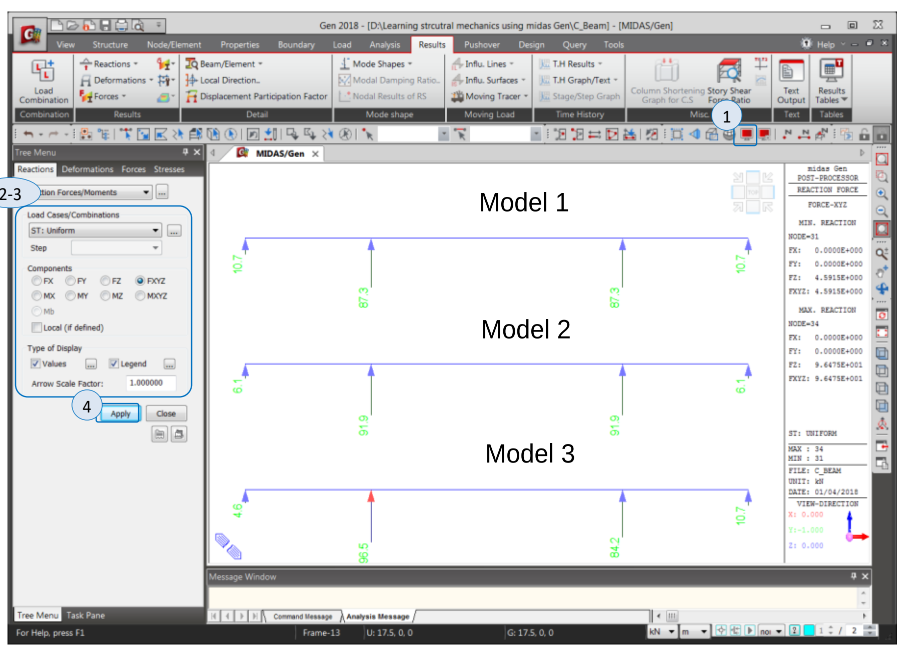
2.8 Check Analysis Results

Compare the reactions due to the uniformly distributed load among the 3 Models. Models 1 and 2 are symmetrical, and the reactions are also symmetrical. In Model 3, the reaction near the cantilever part shows a higher magnitude.

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

1. **Display** (Toggle on)
   Boundary tab: **Support** (Check off), **Beam End Release Symbol** (Check on), **[OK]**
2. Load Cases/Combinations > **ST: Uniform**
3. Components > **FXYZ**
4. Type of Display > **Values, Legend** (Check on), **[Apply]**

▶Fig. 5.16
Reactions due to uniform load



The reactions due to the uniformly distributed load will be checked in a table. Structure models and loading input can be checked by comparing the sum of external load and the sum of reactions.

In the current example, the total external load in the Z-axis direction in Global Coordinate System is 9.8 kN/m×20m×3 = 588kN, which matches the table value.

Main Menu: ***Results / Result Tables / Reaction...***

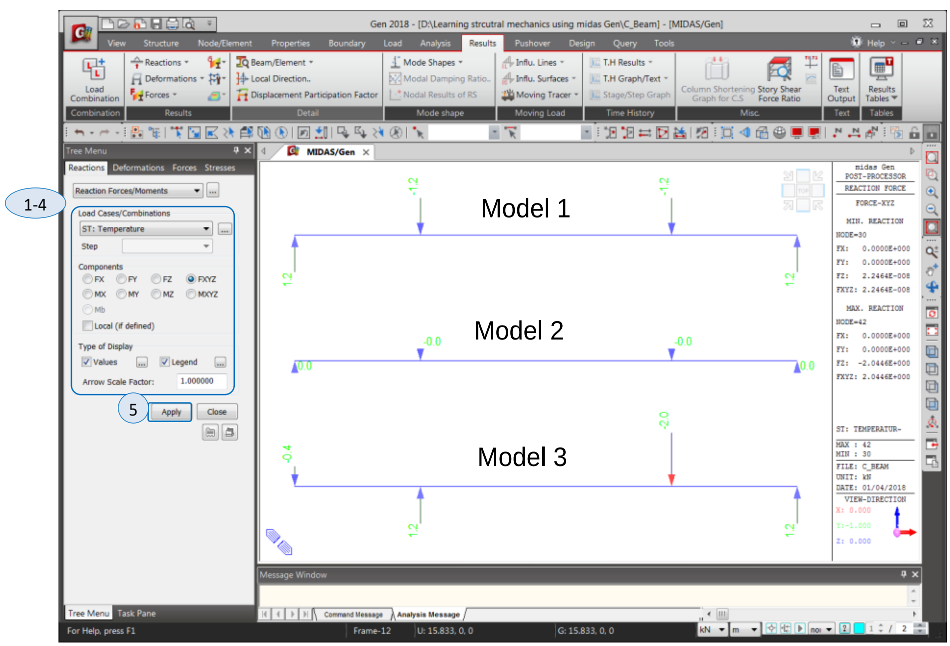
1. Load Cases/Combinations > **Uniform(ST)** (Check on), **Temperature(ST)** (Check off)
2. **[OK]**

▶Fig. 5.17
Reactions in table

Node	Load	FX (kN)	FY (kN)	FZ (kN)	MX (kN*m)	MY (kN*m)	MZ (kN*m)
1	Uniform	0.000000	0.000000	10.738661	0.000000	0.000000	0.000000
4	Uniform	0.000000	0.000000	87.261339	0.000000	0.000000	0.000000
12	Uniform	0.000000	0.000000	87.261339	0.000000	0.000000	0.000000
15	Uniform	0.000000	0.000000	10.738661	0.000000	0.000000	0.000000
16	Uniform	0.000000	0.000000	6.125000	0.000000	0.000000	0.000000
19	Uniform	0.000000	0.000000	91.874999	0.000000	0.000000	0.000000
27	Uniform	0.000000	0.000000	91.874999	0.000000	0.000000	0.000000
30	Uniform	0.000000	0.000000	6.125000	0.000000	0.000000	0.000000
31	Uniform	0.000000	0.000000	4.591544	0.000000	0.000000	0.000000
34	Uniform	0.000000	0.000000	96.475367	0.000000	0.000000	0.000000
42	Uniform	0.000000	0.000000	84.207720	0.000000	0.000000	0.000000
45	Uniform	0.000000	0.000000	10.725368	0.000000	0.000000	0.000000
SUMMATION OF REACTION FORCES PRINTOUT							
	Load	FX (kN)	FY (kN)	FZ (kN)			
	Uniform	0.000000	0.000000	587.999999			

No reactions are generated due to the temperature load in Model 2 as it is a statically determinate structure, whereas reactions are present in Models 1 and 3 since they are statically indeterminate.

Click Model View tab.
Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
1. Load Cases/Combinations > **ST: Temperature**
2. Components > **FXYZ**
3. Type of Display > **Values, Legend** (Check on)
4. **[Apply]**



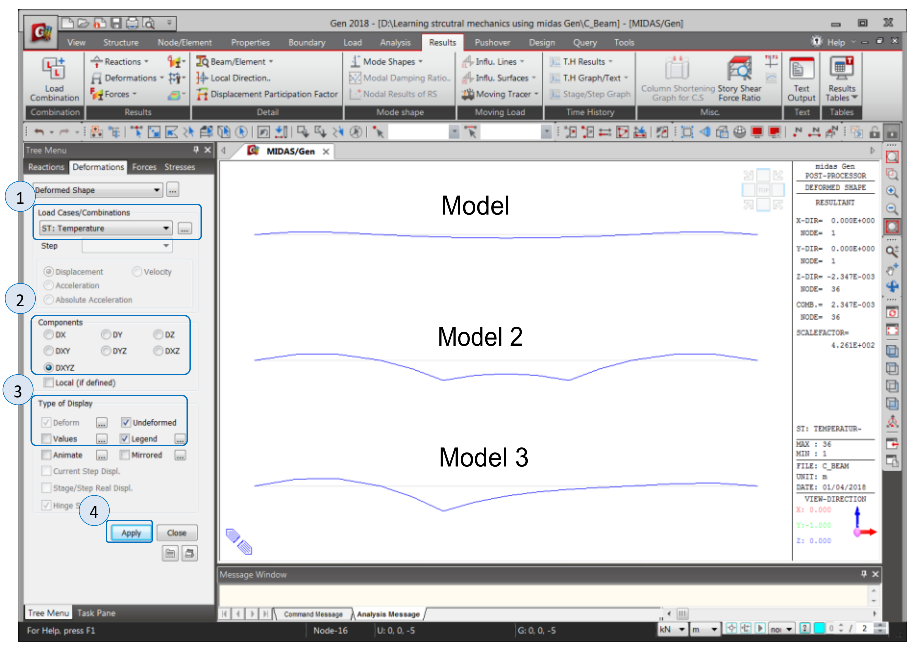
▶Fig. 5.18 Reactions due to temperature

Check the deformations due to the temperature load. DXZ represents $\sqrt{DX^2 + DZ^2}$. The continuous beam Model 1 exhibits the least displacements. In the cases of Gerber beams, statically indeterminate Model 3 shows higher displacements than those of the statically determinate Model 2.

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

1. Load Cases/Combinations > **ST: Temperature**
2. Components > **DXZ**
3. Type of Display > **Undeformed, Legend** (Check on), **Values** (Check off)
4. **[Apply]**

▶Fig. 5.19
Displacements due to temperature

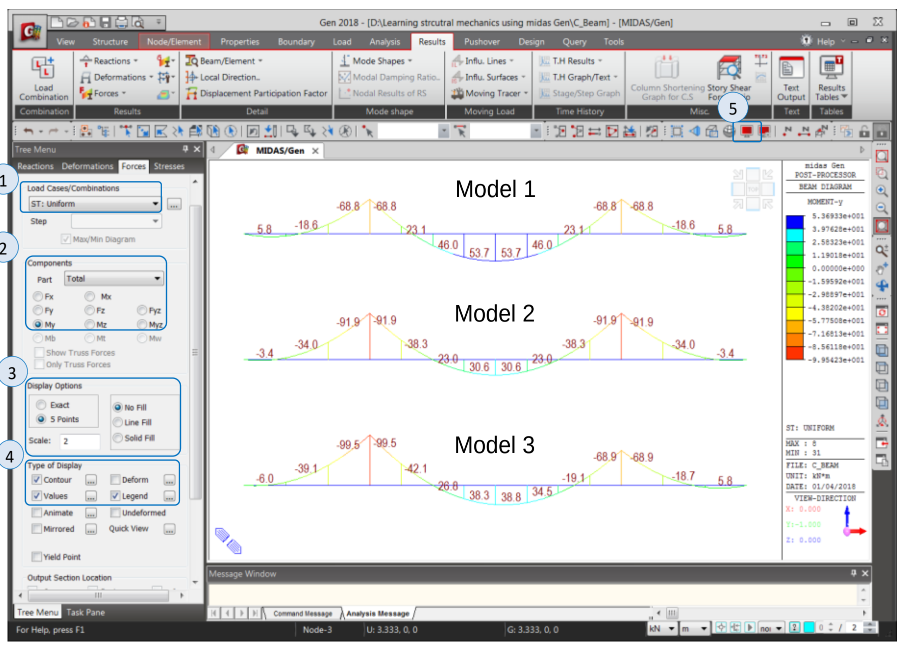


Check bending moments due to the distributed load.

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

1. Load Cases/Combinations > **ST: Uniform**
2. Components > **My**
3. Display Options > Scale: **'2.0'**, **No Fill**
4. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
5. **Display** (Toggle on)
   Boundary tab: **Support** (Check on), **[OK]**
6. **[Apply]**

▶Fig. 5.20
Bending moment diagrams due to uniform load



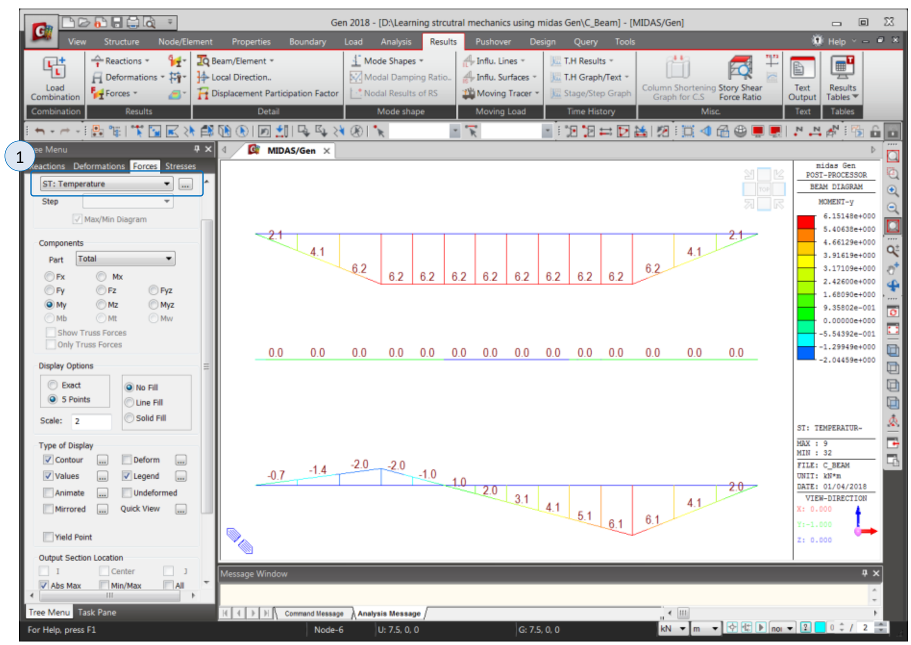
Bending moments in the center of the mid-span of Gerber Model 2 are smaller than those in the continuous beam Model 1. But the negative moments at the interior supports in Gerber Model 2 are higher than those of Model 1. Note that the sums of the positive and negative moments in both Models 1 and 2 are equal (68.8+53.7=91.9+30.6). In the case of Model 3,

the support moment near the hinge is similar to that of Model 1, and the moment at the other interior support is similar to that of Model 1.
Check bending moments due to the temperature load.

As stated above for deformations due to the temperature, Gerber Model 2 exhibits the characteristic that the cantilever parts and the center simple beam behave independently. The fact that there are no restraints to the deformation due to the internal load, no member forces are generated in Model 2 due to the temperature change.

Main Menu: ***Results > Forces > Beam Diagrams...***
1. Load Cases/Combinations > **ST: Temperature**
2. **[Apply]**

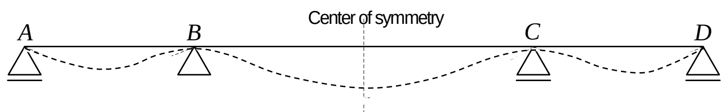
▶Fig. 5.21
Moments due to temperature



3. Manual Calculations

As Model 1 is symmetric about the mid-span, the moment distribution method can be used for the half structure. Point A being a hinge (roller), the stiffness of Member AB becomes $3EI/l$. The rotational angles at Points B and C are symmetrical (equal magnitude and opposite direction), which results in the stiffness of Member BC equal to $2EI/l$. The member stiffness, distribution factors and fixed end moments can be expressed as follows:

▶Fig. 5.22
Moment distribution



$$K_{BA} = \frac{3EI}{l} = \frac{3EI}{5}, \qquad K_{BC} = \frac{2EI}{l} = \frac{2EI}{10} = \frac{EI}{5}$$

$$D_{BA} = \frac{K_{BA}}{K_{BA}+K_{BC}} = \frac{3}{3+1} = 0.75, \quad D_{BC} = \frac{K_{BC}}{K_{BA}+K_{BC}} = \frac{1}{3+1} = 0.25$$

$$M_{BA}^{F} = \frac{1}{8}\omega l^{2} = \frac{25}{8}\omega = 30.625, \qquad M_{BC}^{F} = -\frac{1}{12}\omega l^{2} = -\frac{100}{12}\omega = -81.667$$

The moment at Point B can now be obtained through the moment distribution method.

▶Table 5.1
Results from moment distribution method

Moment distribution method	Point A	Point B	
Distribution factor	1.0	0.75	0.25
Fixed end moment	0	30.625	-81.667
Distributed unbalanced moment	0	38.282	12.761
Carryover moment (factor=0.5)	0	0	0
Final moment	0	68.907	-68.907

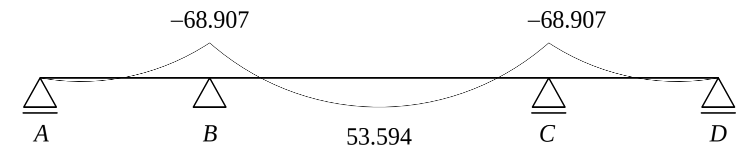
Distributed unbalanced moments at Point B are:

$$-(M_{BA}+M_{BC})\cdot D_{BA} = 38.282, \quad -(M_{BA}+M_{BC})\cdot D_{BC} = 12.761$$

Carryover moment is induced at the other end of the member in question (Fig.5.2(a)).

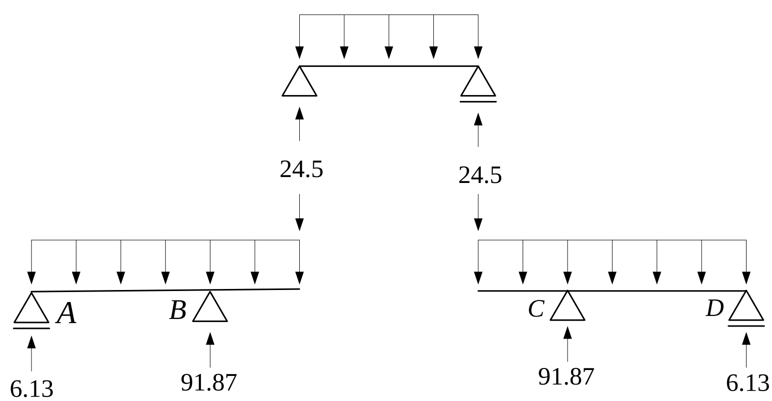
The fixed end moment at Point A is zero, as the fixed end moment at Point B is calculated reflecting the hinge status at Point A. Carryover moments at Points A and B are zero since moment distribution is not required any further.

▶Fig. 5.23
Moment distribution method



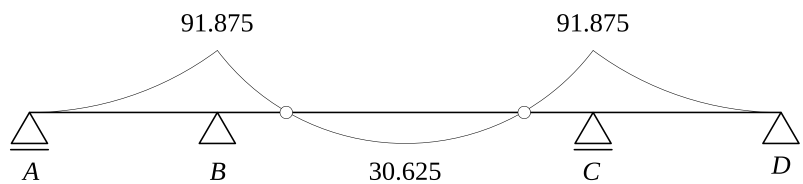
Model 2 is statically determinate, which can be analyzed as below.

▶Fig. 5.24
Model 2 decomposition



$$M_B = 24.5 \times 2.5 + \frac{1}{2} \times 9.8 \times 2.5^2 = 91.875$$

▶Fig. 5.25
Model 2 moment calculation



Model 3 is a statically indeterminate structure, which will be analyzed using the method of consistent deformation (or the unit load method) among other methods. The moment at Point C, M_C is defined as an unknown.

▶Fig. 5.26
Method of consistent deformation

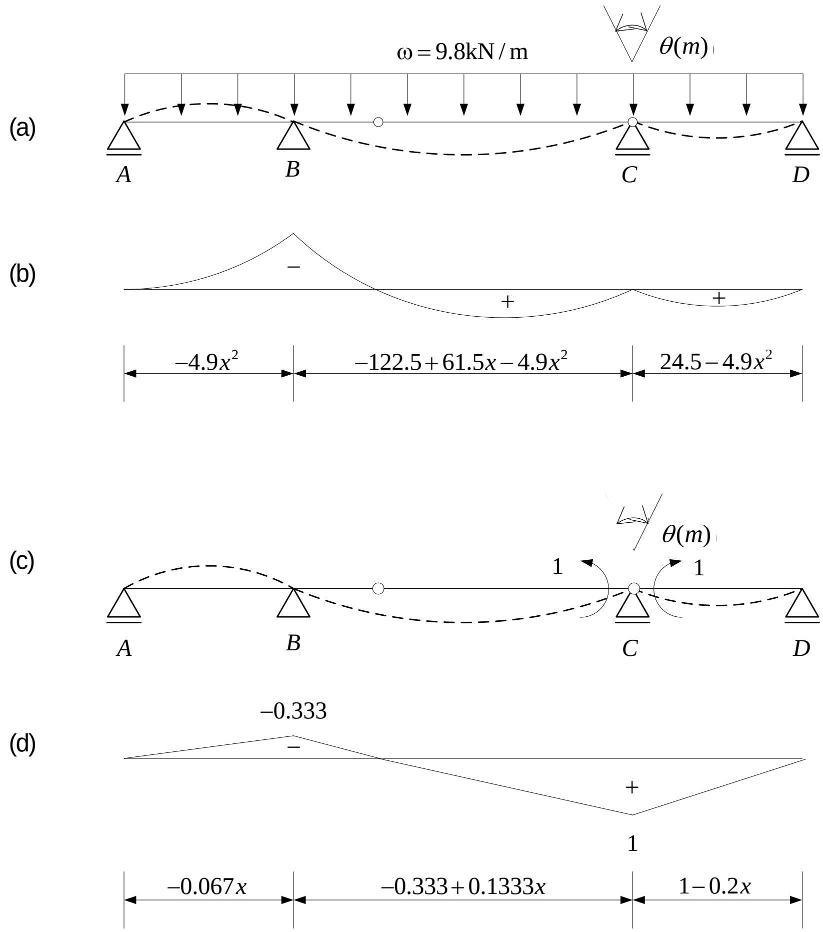


Fig. 5.26(a) shows an analysis model to which a hinge at Point C is assigned. Analysis is then performed with the applied loading, and secondly with a unit moment applied at the hinged Point C. With the introduction of the hinge at Point C, the structure becomes

statically determinate, which can be solved by only force equilibrium. Since the hinge actually does not exist, the following compatibility condition for the unknown, M_C needs to be satisfied:

$$\theta(m)\cdot M_c+\theta(M)=0$$

$\theta(m)$ represents the rotational angle at Point C subject to the applied unit moment. $\theta(M)$ represents the rotational angle at Point C due to the applied loading. The above equation signifies that the kink in the deformed shape cannot occur as the beam is continuous over Point C. The unknown, M_C can be now found using the method of consistent deformation.

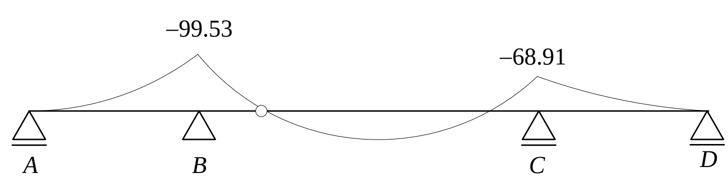
$$\theta(m)=\frac{1}{EI}\int_0^5(-0.067x)^2dx+\frac{1}{EI}\int_0^{10}(-0.333+0.1333x)^2dx+\frac{1}{EI}\int_0^5(1-0.2x)^2dx=\frac{4.4467}{EI}$$

$$\theta(M)=\frac{1}{EI}\int_0^5(-4.9x^2)(-0.067x)dx+\frac{1}{EI}\int_0^{10}(-122.5+61.25x-4.9x^2)(-0.333+0.1333x)dx$$

$$+\frac{1}{EI}\int_0^5(24.5x-4.9x^2)(1-0.2x)dx=\frac{306.5052}{EI}$$

$$\frac{4.4467}{EI}M_c+\frac{306.5052}{EI}=0$$

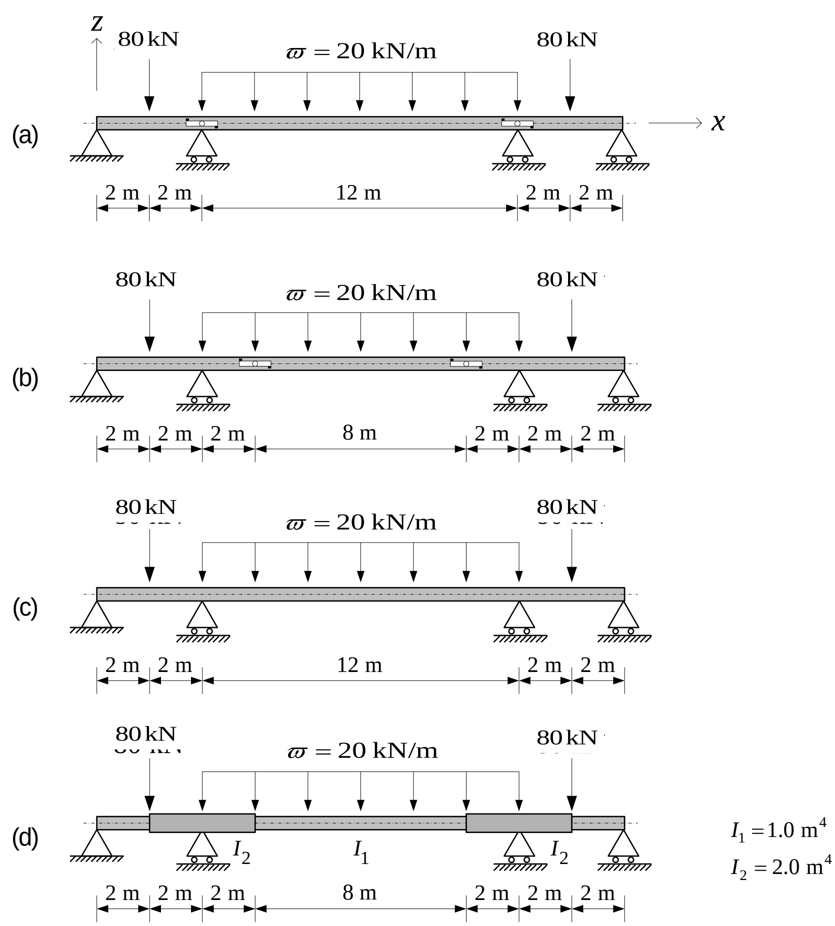
$$M_c=-68.91\text{ kN}\cdot\text{m}$$



▶Fig. 5.27
Moment calculations by method of consistent deformation

4. Exercise

The following 4 cases have the same spans and loadings, but with different structural systems, which are simple beam, Gerber beam, continuous beam and reinforced section over interior supports. Find the moment diagrams of the 4 cases, and confirm that the positive moments progressively decrease and the negative moments progressively increase from a) through d).



6. **Frame** Analysis

Contents

1. Introduction

1.1 Concept for Frame Analysis

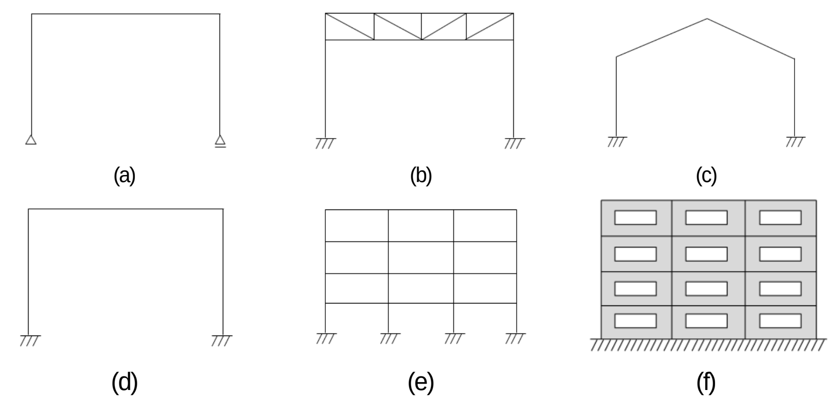
Members rigidly connected (fixed joint or rigid joint) are referred to as a frame or rigid frame. Generally a column is rigidly connected to beam(s) forming a continuous structure. Members joined at a node can transfer bending moments, shear forces and axial forces.

▶Photo 6.1
Frame structures

Frames can take many different forms and shapes from a simple portal frame to composite, rectangular, gable and concrete shear wall frames.

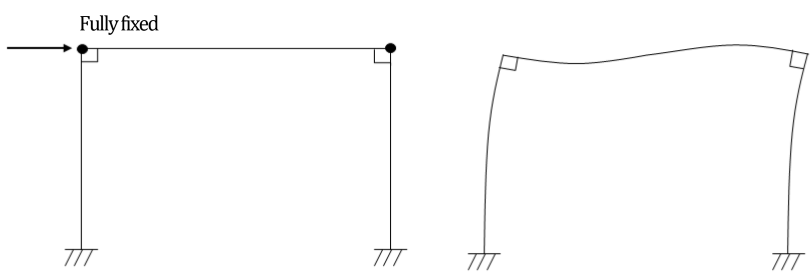
Structural applications are also diverse such as regular shape frames like offices, schools, and hospitals, and irregular shape frames like temporary erection structures, plants, stadiums and gymnasiums.

Concrete frame structures are also widely used such as bridges and shear walls with openings in building structures.



▶Fig. 6.1
Frame structure types

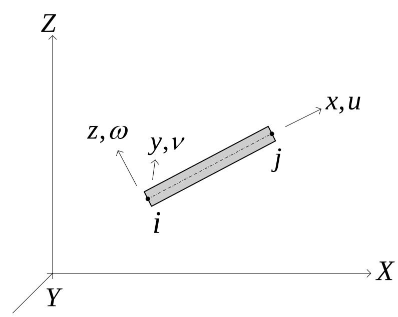
In a statically indeterminate frame structure subject to an external force such as shown in Fig. 6.2, the angles formed by the members rigidly connected at nodes are assumed to remain unchanged after deformation. In this case, all the members at the nodes remain perpendicular in the deformed shape.



▶Fig. 6.2
Frame deformation

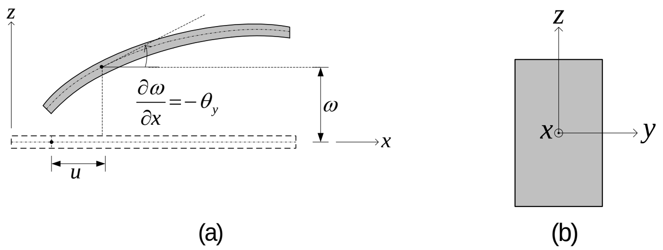
The structural concept for frames is not different from that of simple, cantilever and continuous beams. Frame members are also line members like in continuous beams, but they are connected in 2 or 3 dimensions. Each member of a frame structure retains its own Element Coordinate System (ECS). Generally the x -axis direction is from i-Node to j-Node. z -axis is perpendicular to x -axis. y -axis is determined by the right hand rule.

▶Fig. 6.3
Element Coordinate System (ECS) and displacements of a frame member



The degree of static indeterminacy of a frame structure can be found in the same manner explained for truss structures. Statically indeterminate frame structures can be also analyzed by the same methods used for truss analysis. Reaction results follow the Global Coordinate System (GCS), and the member forces are generated in ECS. The translational displacements in ECS x, y, z directions are expressed as u, v, w respectively. $\theta_x, \theta_y, \theta_z$ are defined for the rotational displacements about x, y, z axes respectively. The deformation and sectional axes of a 2-D frame member on the GCS X-Z plane can be illustrated as Fig. 6.4.

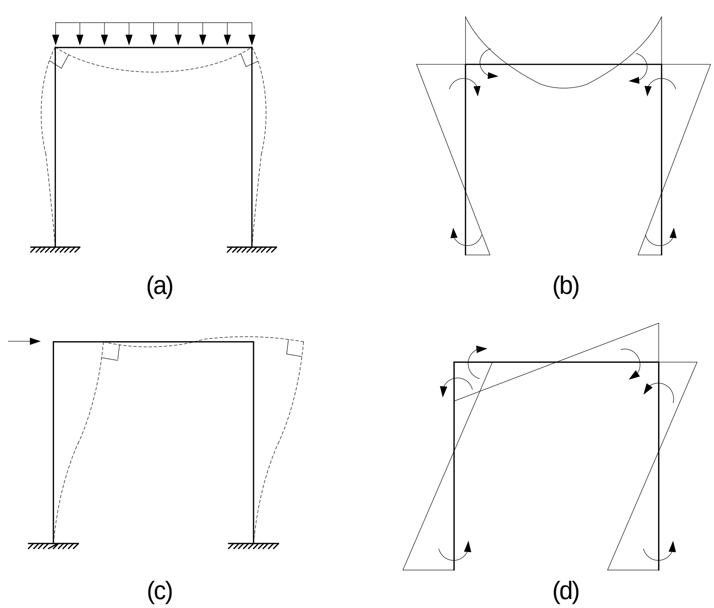
▶Fig. 6.4
2-D frame deformation and sectional axes
(a) Deformation in ECS
(b) Sectional axes



If a loading in ECS z-direction is applied to a frame element, deformation, ω will take place in the direction perpendicular to ECS x -axis. The direction of bending moment becomes the direction of rotation about ECS y -axis as shown in Fig. 6.4 (b).

Frame structures are capable of resisting vertical and lateral loads as shown in Fig. 6.5.

▶Fig. 6.5
Deformations and member forces due to external loads in a frame structure

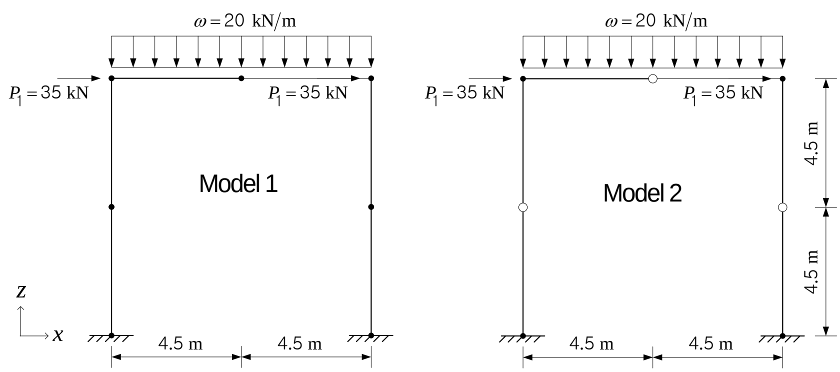


2. Tutorial

2.1 Model Overview

Find the member forces and deformations of the following frame structures:

▶Fig. 6.6
Analysis models
(a) Model 1
(a) Model 2



- ***Material***

  Concrete: IS(RC) M40 (Modulus of elasticity, E = 3.1622×10^{7} kN/m^2)

- ***Section***

  Area: 5×10^{-4} m^2

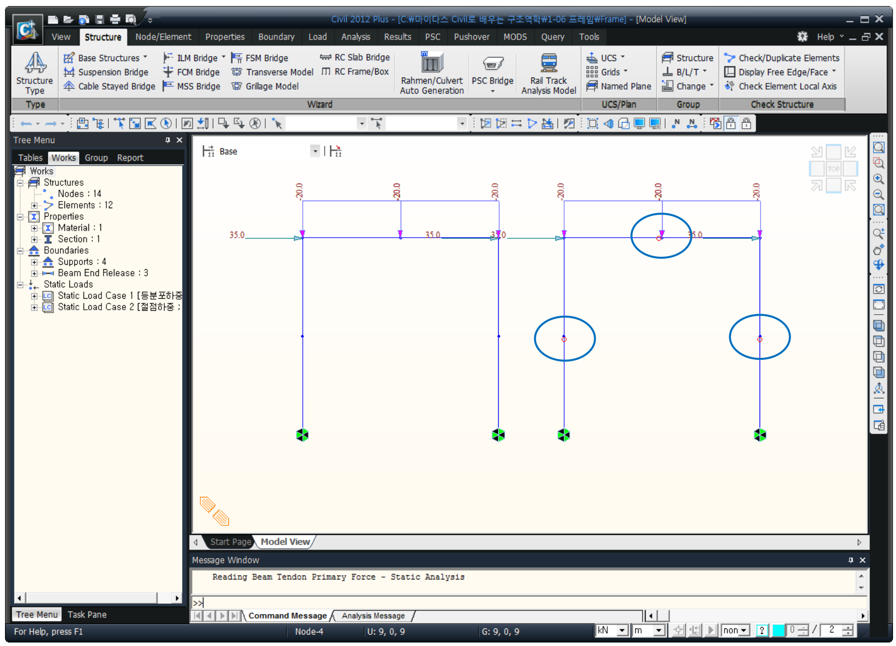
  Moment of inertia (Iyy): 6×10^{-3} m^4

- ***Load***

  1. Distributed load applied at the top of beam: 20 kN/m
  2. Concentrated load, 35 kN applied in (+) X-direction at the top of each column

All the members in Model 1 are rigidly connected, whereas hinges are introduced in the middle of the members in Model 2. The member forces and deformations of the 2 models will be examined.

▶Fig. 6.7
Analysis models



2.2 Work Environment

Open a New File and save.

Main Menu: ***/ New Project...***

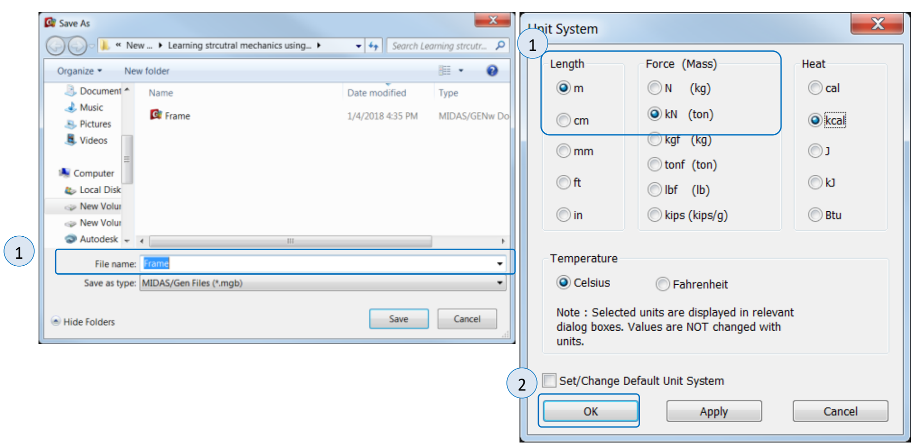
Main Menu: ***/ Save As...***

1. File name: **'Frame'** and [Save]

Set a unit system to kN & m.

Main Menu: ***Tools / Unit System***

1. Length > **m**, Force (Mass) > **kN (ton)**
2. **[OK]**



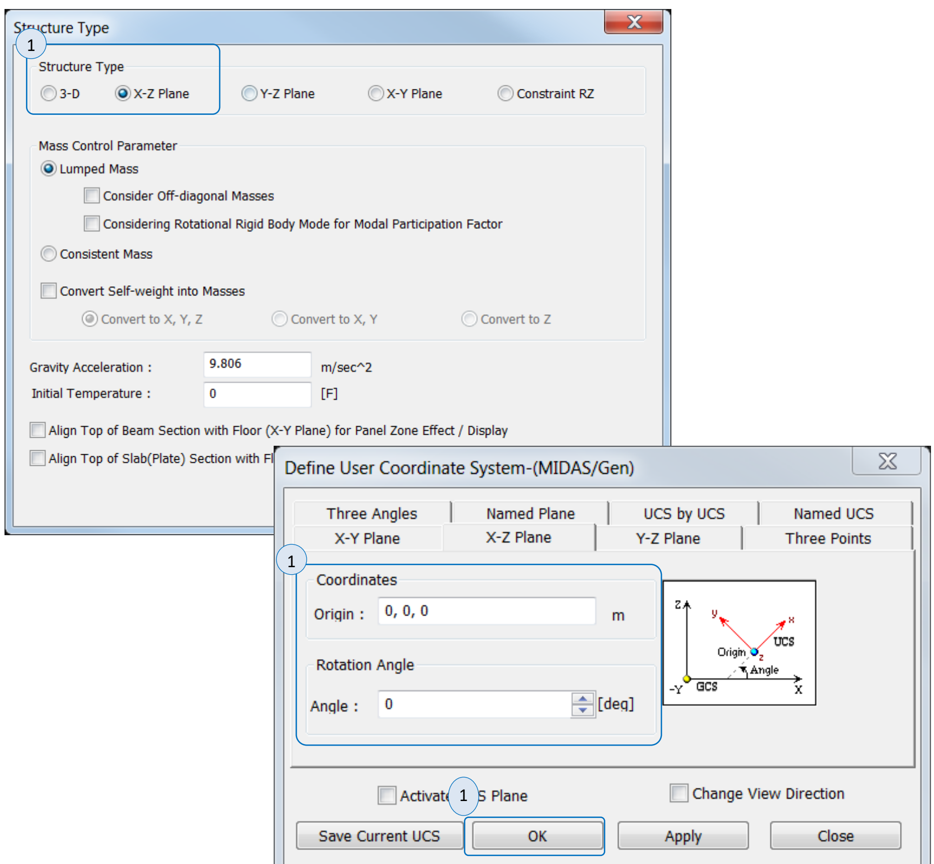
▶Fig. 6.8
Save file &
set unit system

The example models exist in the 2-D X-Z plane, which is defined as the x-y plane in User Coordinate System (UCS) for ease of modeling.

Main Menu: ***Structure / Structure Type***
1. Structure Type > **X-Z Plane**
2. **[OK]**

Main Menu: ***Structure / UCS (User Coordinate System) / X-Z Plane***
1. Coordinates: **‘0, 0, 0’**, Rotation Angle: **‘0’** and **[OK]**

▶Fig. 6.9
(a) Set a work plane
(b) Set User Coordinate System (UCS)



Tip

UCS is more convenient for a structure with regular node and member spacing.

2.3 Material & Section Properties

Define material and section for the structural members.

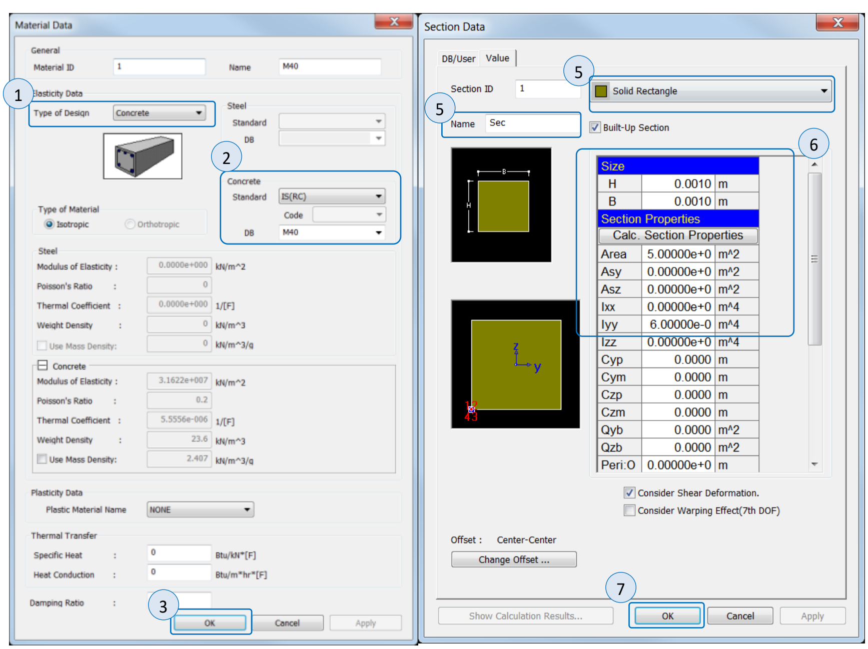
Tip

Program automatically calculates section stiffness if section size is defined.

For this tutorial, stiffness is directly entered.

Main Menu: ***Properties / Material Properties***

1. **[Add...]**, Type of Design > **Concrete**
2. Concrete > Standard > **IS(RC)**, DB >**M40**
3. **[OK]**
4. Properties dialog window: click **Section** tab, **[Add…]**, **Value** tab
5. Section shape list > **Solid Rectangle**, Name: **‘Sec’**
6. H: **‘0.001’**, B: **‘0.001’**, Area: **‘5e+2’**, Iyy: **‘6e-3’**
7. **[OK]**, **[Close]**



▶Fig. 6.10
Define material
Define section

2.4 Nodes & Elements Generation

Create nodes and beam elements simultaneously using User Coordinate System (UCS).

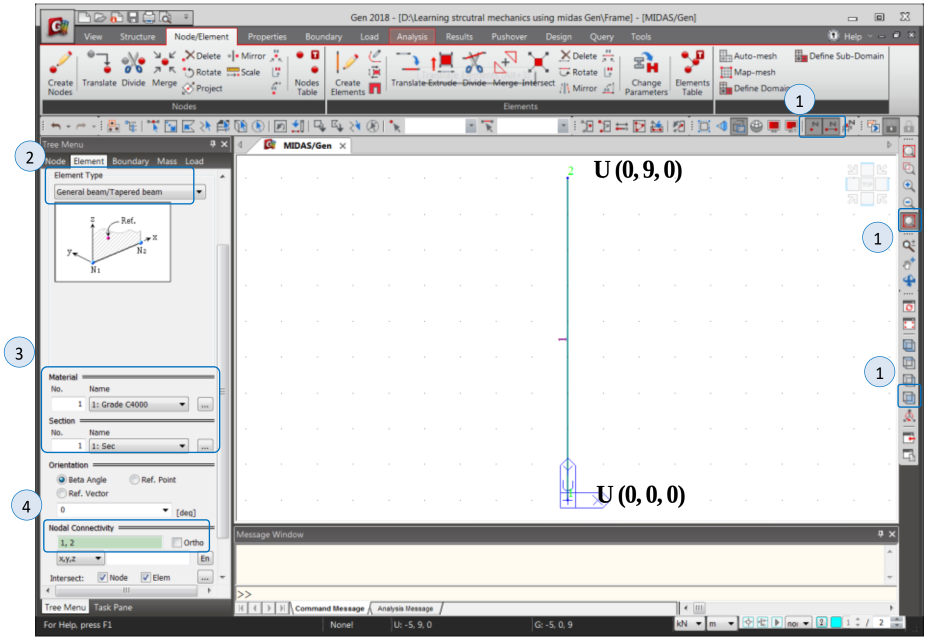
Tip

Point Grid and Point Grid Snap are used to directly create nodes and elements on the modeling window.

In the Point Grid Snap mode, placing the mouse on the modeling window prompts the coordinates displayed at the bottom of the screen.

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

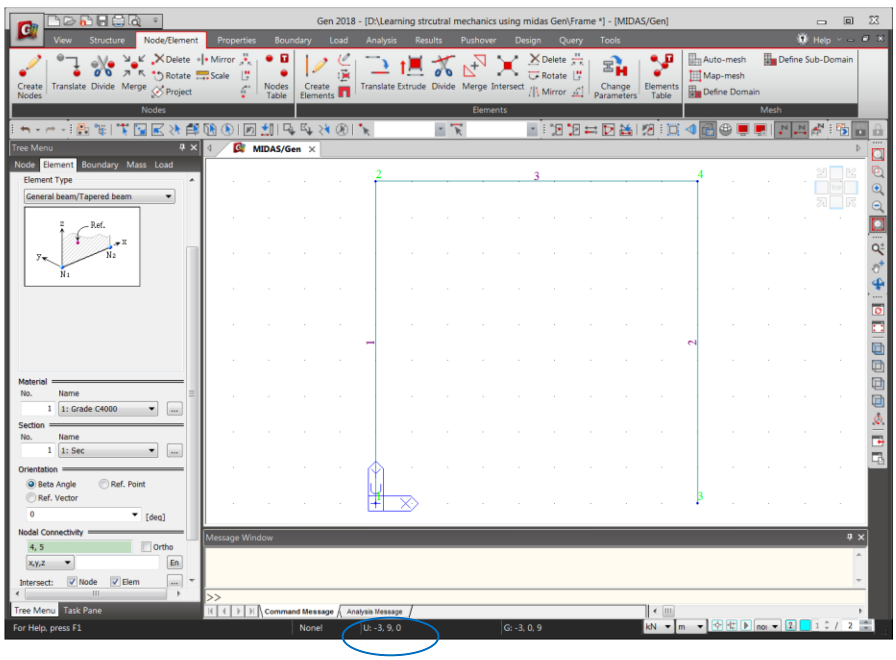
1. **Display Node Numbers, Display Element Numbers**, **Auto Fitting, Front View** (Toggle on)
   Main Menu: ***Structure / Grids / Define Point Grid…*** **>** Grid Spaces: **1, 1 and [OK]**
   Main Menu: **View / Grids / Point Grid**, **View / Snap / Point** (Toggle on)
2. Element Type > **General Beam/Tapered beam**
3. Material > **1: M40**, Section > **1: Sec**
4. Click the Nodal Connectivity field (which turns to green)
   Model View: click **(U: 0,0,0), (U: 0,9,0) coordinates** sequentially



▶Fig. 6.11
Create a column

Referring to UCS coordinates at the bottom of the screen, click (U: 9, 0, 0), (U: 9, 9, 0) and (U: 0, 9, 0), (U: 9, 9, 0) sequentially to create Elements 2 and 3.   In order to correctly snap the corresponding locations, ensure to keep Grid/Snap tab > Point Grid and Point Grid Snap toggled on.

▶Fig. 6.12
Create column & beam



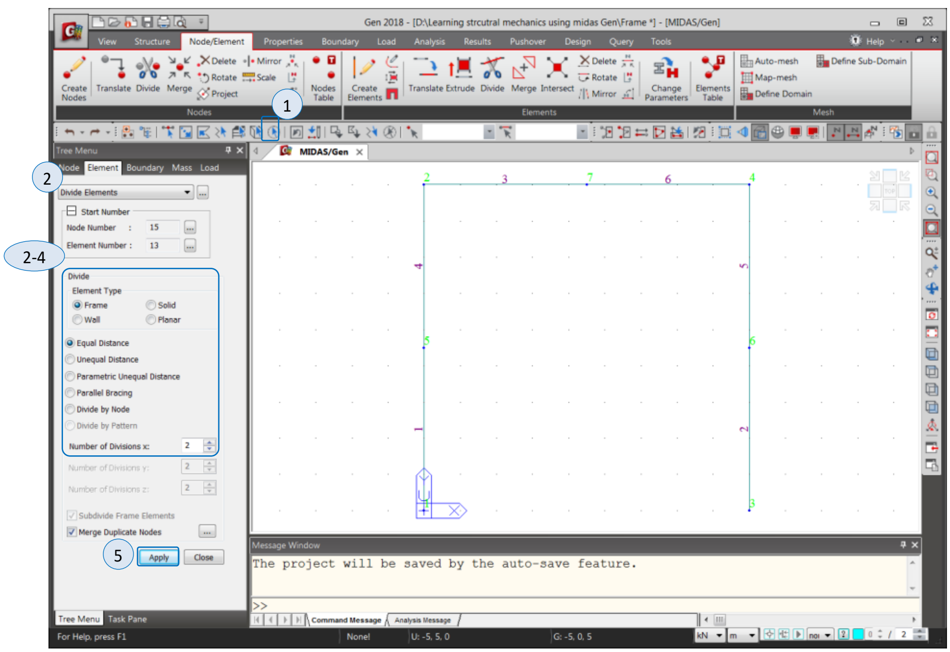
Tip

Dragging the mouse on the screen with the wheel pressed on can pan the model.
Pressing the mouse once and turning the wheel can zoom in/out the model.

In order to define hinges in Model 2, divide all the elements into halves using Divide Elements.

Main Menu: ***Node/Element / Divide***
1. Main Menu: **View / Grids / Point Grid**, **Select All**
2. Select ***Divide Elements*** > Element Type > **Frame**
3. **Equal Distance**
4. Number of Divisions x: **‘2’**
5. **[Apply]**

▶Fig. 6.13
Divide elements

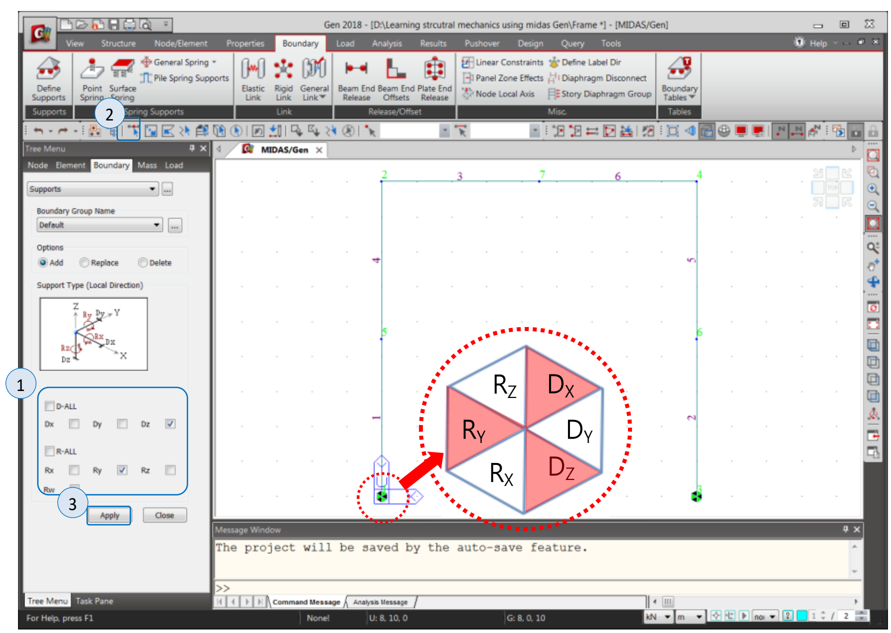


2.5 Define Boundary Conditions

Fixed supports are assigned to the bottom of the columns.
At the supports, 6 degrees of freedom (Dx, Dy, Dz, Rx, Ry, Rz) are symbolized by 6 triangular cells in a hexagon. Dx to Rz are sequentially represented starting from the “12 o’clock” in the clockwise direction. Green color in the triangular cells signifies restrained condition. Coordinate System is in GCS unless Nodal Coordinate System is defined.

Main Menu: ***Boundary / Define Supports***
1. Support Type > **Dx, Dz, Ry** (Check on)
2. **Select Single** (on) mode: select **Nodes 1, 3**
3. **[Apply]**

▶Fig. 6.14
Define support conditions



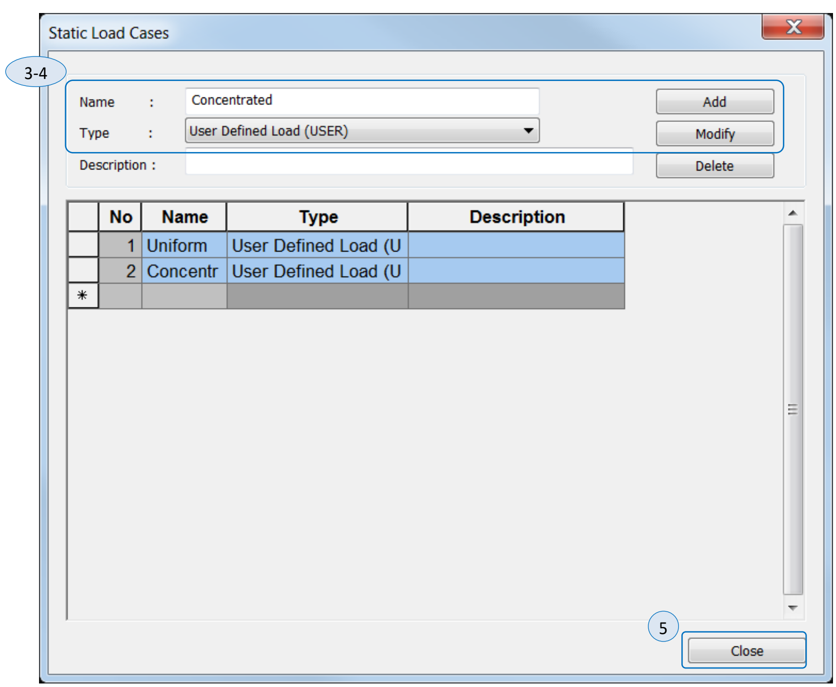
2.6 Define Loads

Define the load cases (load types) first to which the uniformly distributed load and the concentrated loads will belong.

Main Menu: ***Load / Static Loads / Static Load Cases***

1. Name: **'Uniform'**
2. Type > **User Defined Load (USER), [Add]**
3. Name: **'Concentrated'**
4. Type > **User Defined Load (USER), [Add]**
5. **[Close]**

▶Fig. 6.15
Define load cases



Input a uniformly distributed load on the beam.

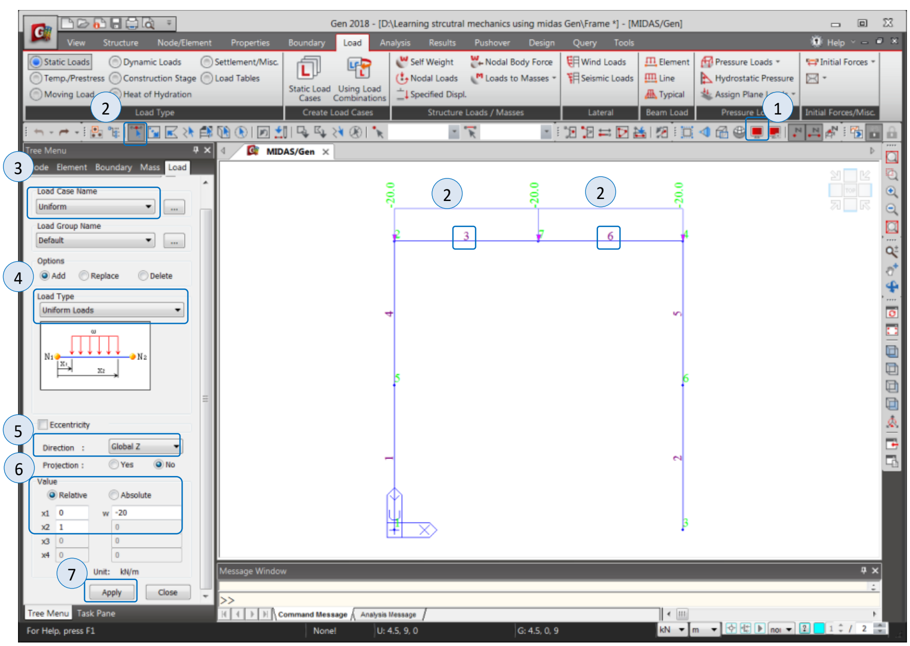
Main Menu: ***Load / Static Loads / Element Beam Loads***

1. **Display** (Toggle on)
   Load tab: **Load Value** (Check on), **[OK]**
2. **Select Single** (Toggle on) mode: select **Elements 3 and 6**
3. Load Case Name > **Uniform**
4. Load Type > **Uniform Loads**
5. Direction > **Global Z**
6. Value > x1: **'0'**, w: **'-20'**, x2: **'1'**
7. **[Apply]**

Tip

Once the elements are selected, the element numbers are listed at the top of Model View.

▶Fig. 6.16
Input uniformly distributed load



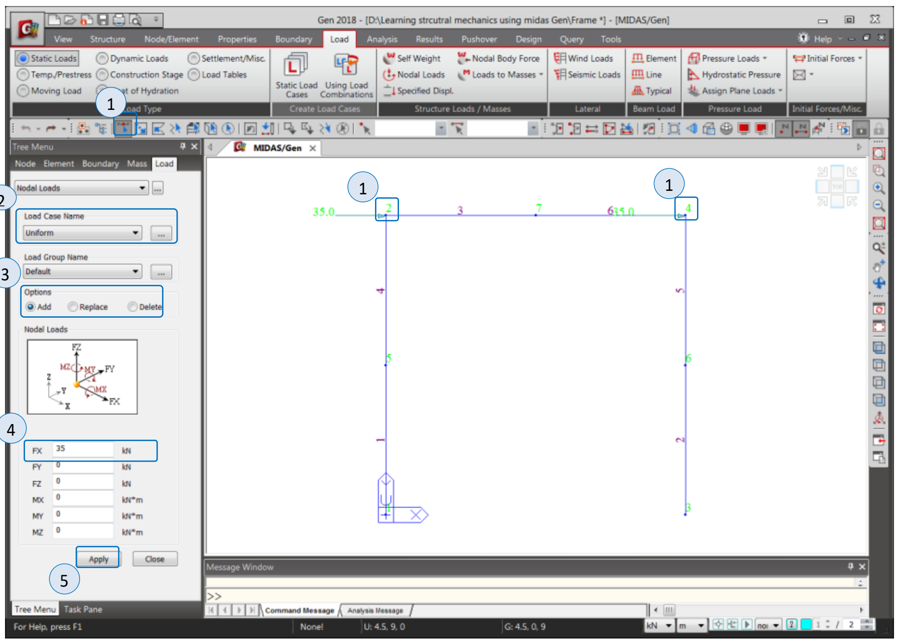
Apply concentrated loads at the top ends of the columns.

Tip

Once the nodes are selected, the node numbers are listed at the top of Model View.

Main Menu: ***Load / Static Loads / Nodal Loads***

1. **Select Single (on)** mode: select **Nodes 2 and 4**
2. Load Case Name > **Concentrated**
3. Options > **Add**
4. Nodal Loads > FX: **'35'**
5. **[Apply]**



▶Fig. 6.17
Apply nodal loads

Copy the rigidly connected Model 1 to create Model 2 to the right. Model2 will be subsequently assigned 3 hinges.

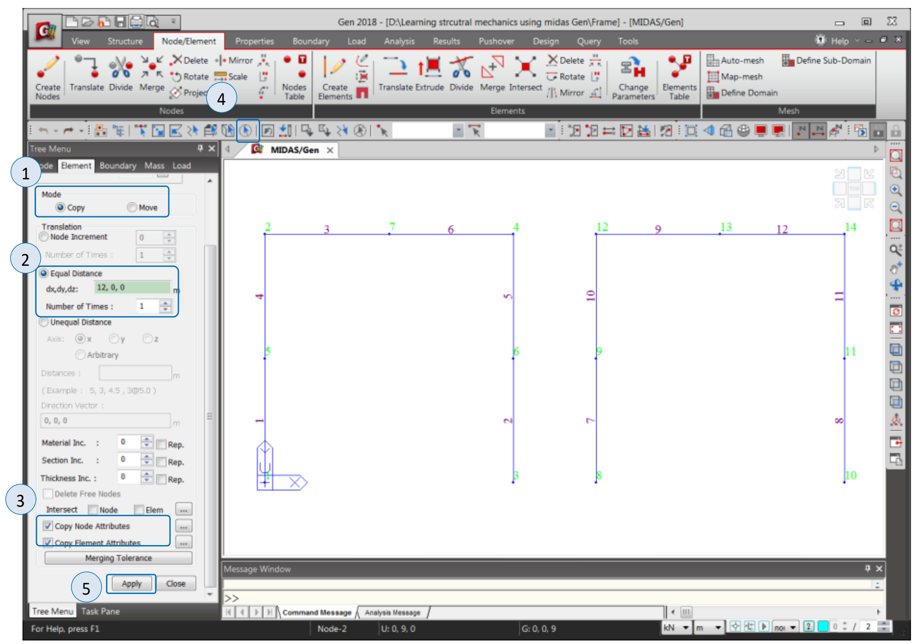
Tip

Copy Node (Element) Attributes will simultaneously copy the loads and boundary conditions assigned to Model 1.

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

1. Mode > **Copy**
2. Translation > Equal Distance > dx, dy, dz: **'12, 0, 0'**
3. **Copy Node Attributes, Copy Element Attributes** (Check on)
4. **Select All**
5. **[Apply]**

▶Fig. 6.18 Create Model 2

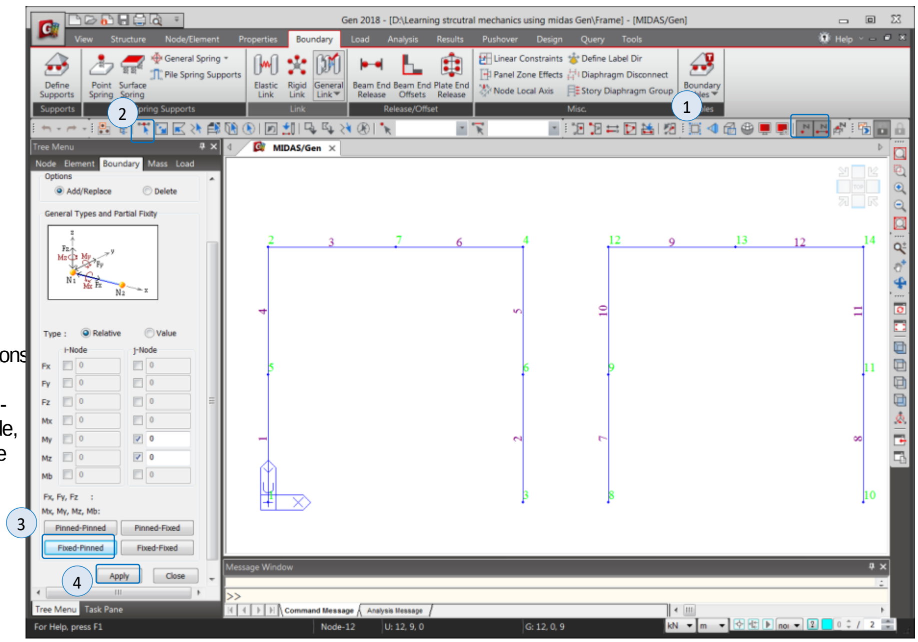


Define internal Beam End Release in Model 2.

Main Menu: ***Boundary / Beam End Release***

1. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
2. **Select Single** (on) mode: select **Elements 7, 8, 9**
3. General Types and Partial Fixity > **[Fixed-Pinned]**
4. **[Apply]**

▶Fig. 6.19
Input member hinge conditions



Tip

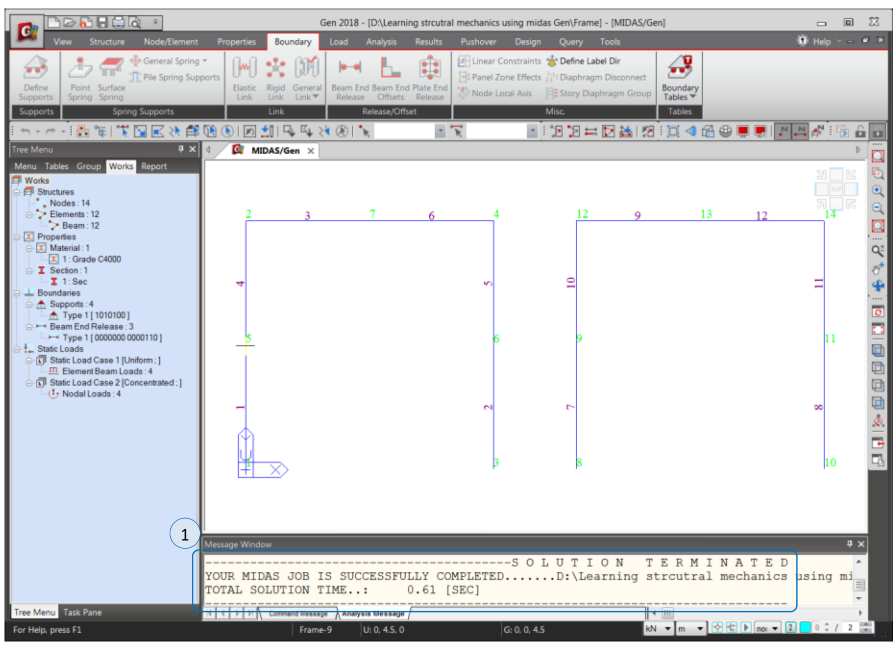
[Pinned-Pinned], etc. buttons represent the boundary conditions at i-Node and j-Node. i-Node is the Node, which has a smaller Node number.

2.7 Perform Analysis

Analyze the 2 Models.

Main Menu: ***Analysis* / Perform *Analysis***

1. Check for successful completion in Message Window.



▶Fig. 6.20 Message for a successful run

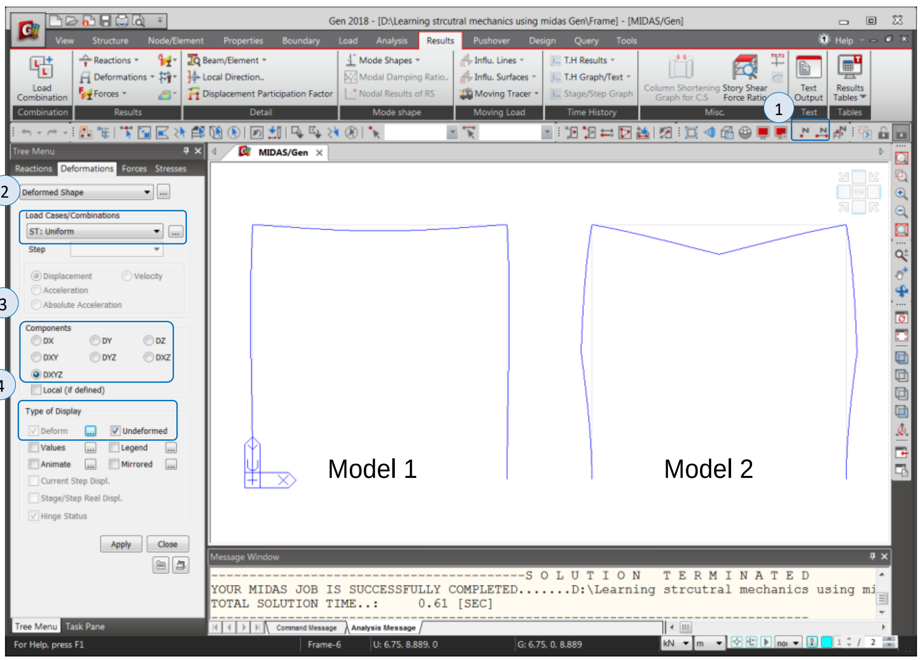
2.8 Check Analysis Results

Check the deformed shapes of the 2 Models due to the gravity direction load (uniform load). Model 2 exhibits large deformation at the mid-span (hinge point) of the beam similar to the cantilever behavior.

Main Menu: ***Results / Deformations / Deformed Shape…***

1. **Display Node Numbers, Display Element Numbers** (Toggle off)
2. Load Cases/Combinations > **ST: Uniform**
3. Components > **DXYZ**
4. Type of Display > **Undeformed** (Check on)
   Click **[...]** next to Deform > Deformation Type: **Real Deform, [OK]**

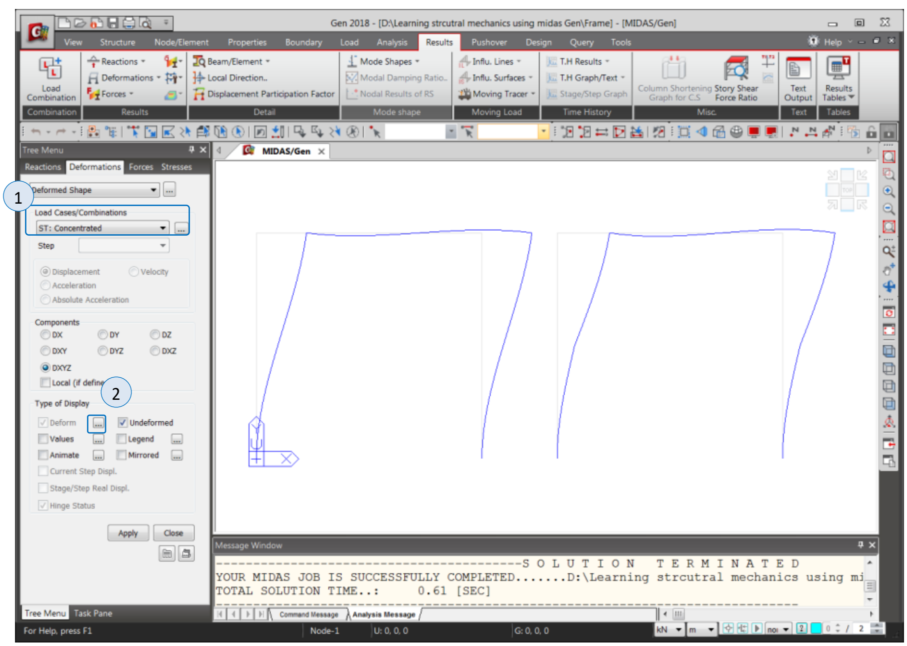
▶Fig. 6.21
Deformed shapes due to the uniform load



Deformed shapes due to the lateral concentrated loads will now be examined.
Lateral deformation in Model 2 is a little more excessive from the hinge points at the mid-height of the columns. The lateral forces are transmitted to the lower half columns via shear forces from the hinge points acting as cantilevers. The upper half columns offer limited resistance to lateral deformation.

1. Load Cases/Combinations > **ST: Concentrated**
2. Click **[...]** next to Deform > Deformation Scale Factor: **'2.0'**
3. **[OK]**

▶Fig. 6.22
Deformed shapes due to lateral forces



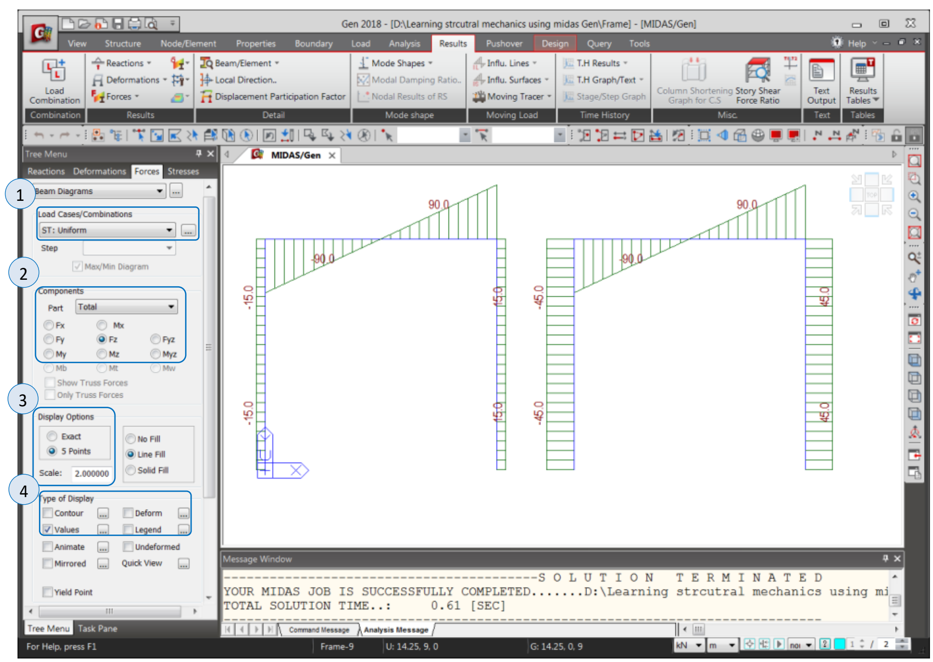
Find the shear forces due to the uniform load.

The bending moments at the top ends of the columns in Model 2 are 202.5 kN·m due to the uniform load on the beam. Each moment induces 45 (=202.5/4.5) kN at each column hinge point. 4.5 is the top half length of the columns above the hinge points.

Main Menu: ***Results / Forces / Beam Diagrams...***

1. Load Cases/Combinations > **ST: Uniform**
2. Components > **Fz**
3. Display Option > **Exact**, Scale: **'2'**
4. Type of Display > **Contour, Deform** (Check off), **Values** (Check on)
5. **[Apply]**

▶Fig. 6.23
Shear Force diagrams



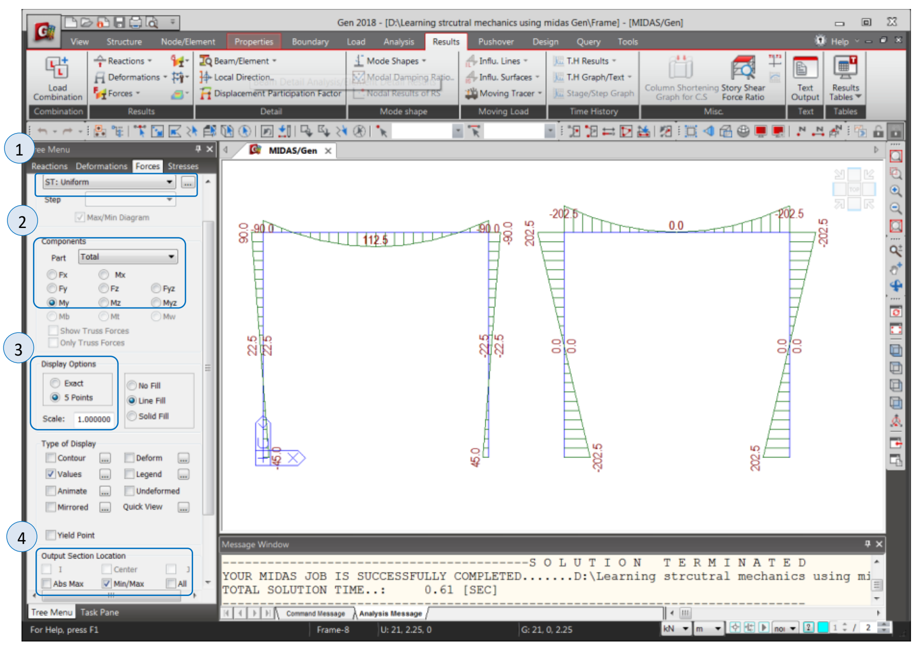
Check bending moment diagrams due to the uniform load.
Due to the hinge point at the mid-span of the beam in Model 2, the moment distribution resembles that of cantilever beams.

Main Menu: ***Results / Forces / Beam Diagrams...***

1. Load Cases/Combinations > **ST: Uniform**
2. Components > **My**
3. Display Options > Scale: **'1.0'**
4. Output Section Location > **Min/Max**
5. **[Apply]**

▶Fig. 6.24
Moment diagrams

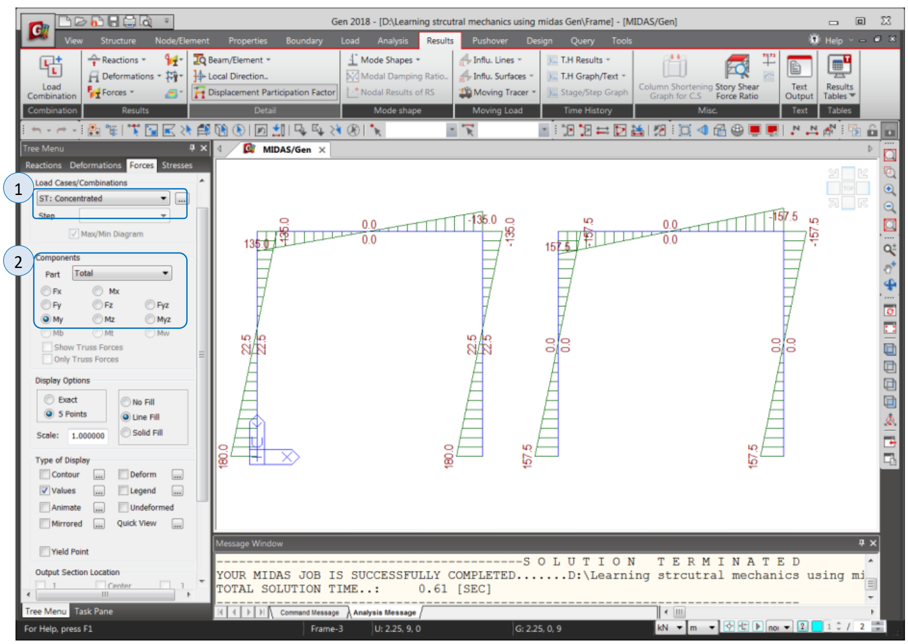
Moment diagrams



Check bending moments due to the lateral concentrated loads.

Main Menu: ***Results / Forces / Beam Diagrams...***
1. Load Cases/Combinations > **ST: Concentrated**
2. Components > **My**
3. **[Apply]**

▶Fig. 6.25
Moment diagrams due to the lateral forces

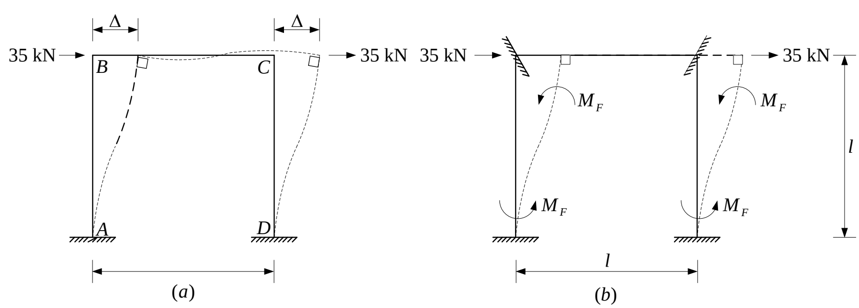


3. Manual Calculations

<Case of acting lateral forces>

Because the degree of indeterminacy of Model 1 is 3, the method of consistent deformation is not suitable for finding 3 unknowns by hand. However, moment distribution can be an attractive method for hand calculation in which the symmetry is used and a degree of freedom is set as an unknown. The number of unknowns can be then reduced to 1. Fig. 6.26 (a) shows the inversely symmetrical deformed shape of Model 1 subjected to lateral forces. If Points B and C are restrained against rotation under the acting lateral forces, the frame deforms as Fig. 6.26 (b), and fixed end moments, M_F are induced.

▶Fig. 6.26



The fact that the real structure is not restrained against rotation at Points B and C, the restraints can be released using the moment distribution method. The stiffness of the column and beam is calculated considering the inverse symmetry, which results in the distribution factors of 0.4 for the column and 0.6 for the beam as follows:

▶Eq. 6.1

$$k_{BA}\text{ (Fixed at Opposite end)} = \frac{4EI}{l}, \quad k_{Bc}\text{ (Inverse Symmetry at Opposite end)} = \frac{6EI}{l}$$

The results from the moment distribution method are shown in Table 6.1 below, which coincide with the results from midas Gen.

▶Table 6.1

Moment distribution method	Point A	Point B	
Distribution factor	1.0	0.4	0.6
Fixed end moment	M_F	M_F	0
Distributed unbalanced moment	0	-0.4 M_F	-0.6 M_F
Carryover moment (factor=0.5)	-0.2 M_F	0	0
Final moment	0.8 M_F	0.6 M_F	-0.6 M_F

Carryover moment is induced at the other end of the member in question (Fig. 5.2(a))

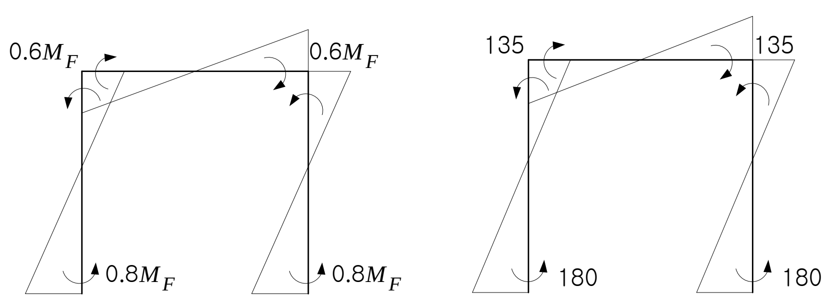
M_F can be calculated from force equilibrium in the lateral direction.

▶Eq. 6.2

$$70 = 2 \times \frac{0.6M_F + 0.8M_F}{l} = \frac{2.8M_F}{l}, \quad M_F = 225\,\mathrm{kN \cdot m}$$

The results from the hand calculation coincide with the results from midas Gen.

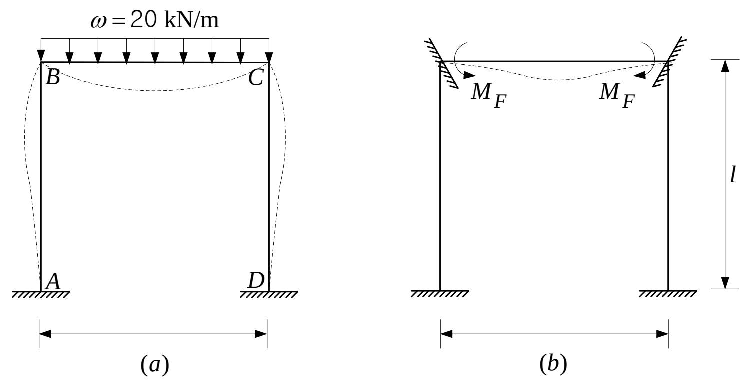
▶Fig. 6.27



<Case of acting vertical load>

The vertical uniform load applied to Model 1 causes symmetrical deformation as shown in Fig. 6.28 (a). The uniform load is now applied to Model 1 with Points B and C restrained against rotation, which then induces the fixed end moments, M_F with the deformed shape as shown in Fig. 6.28 (b).

▶Fig. 6.28



▶Eq. 6.3

$$M_F = \frac{\omega l^2}{12} = 135\ \mathrm{kN \cdot m}$$

The fact that the real structure is not restrained against rotation at Points B and C, the restraints can be released using the moment distribution method. The stiffness of the column and beam is calculated considering the symmetry, which results in the distribution factors of 2/3 for the column and 1/3 for the beam as follows:

▶Eq. 6.4

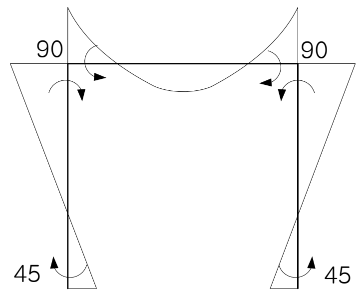
$$k_{BA}\,(\text{Fixed at Opposite end}) = \frac{4EI}{l}\ ,\ k_{Bc}\,(\text{Symmetry at Opposite end}) = \frac{2EI}{l}$$

The results from the moment distribution method are shown in Table 6.2 below, which coincide with the results from midas Gen.

▶Table 6.2

Moment distribution method	Point A	Point B	
Distribution factor	1.0	2/3	1/3
Fixed end moment	0	0	–135
Distributed unbalanced moment	0	90	45
Carryover moment (factor=0.5)	45	0	0
Final moment	45	90	–90

▶Fig. 6.29

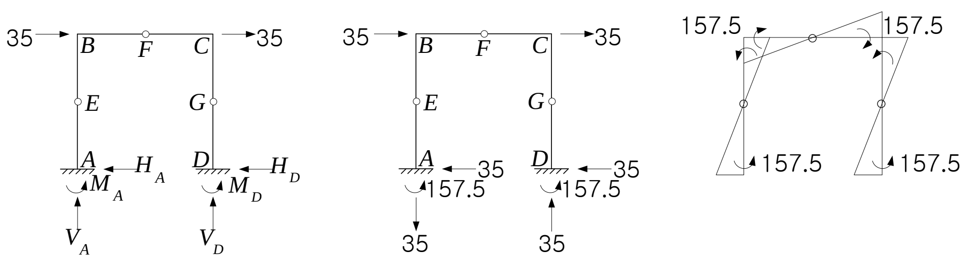


<Case of acting lateral forces>

The number of reactions in Model 2 is 6, which requires 6 equations to solve, which can be formulated by 3 force equilibrium equations and zero moment condition at Points E, F and G. In fact Model 2 is a statically determinate structure.

$$\sum F_X = 0 : H_A + H_D - 70 = 0$$
$$\sum F_Y = 0 : V_A + V_D = 0$$
$$\sum M_D = 0 : -M_A - M_D - 9V_D + 35\times 9 + 35\times 9 = 0$$
$$M_E = 0 : 4.5H_A - M_A = 0$$
$$M_G = 0 : 4.5H_D - M_D = 0$$
$$M_F = 0 : 9H_D - 4.5V_D - M_D = 0$$

▶Fig. 6.30



<Case of acting vertical load>

$$\sum F_X = 0 : H_A + H_D = 0$$

$$\sum F_Y = 0 : V_A + V_D = 20 \times 9$$

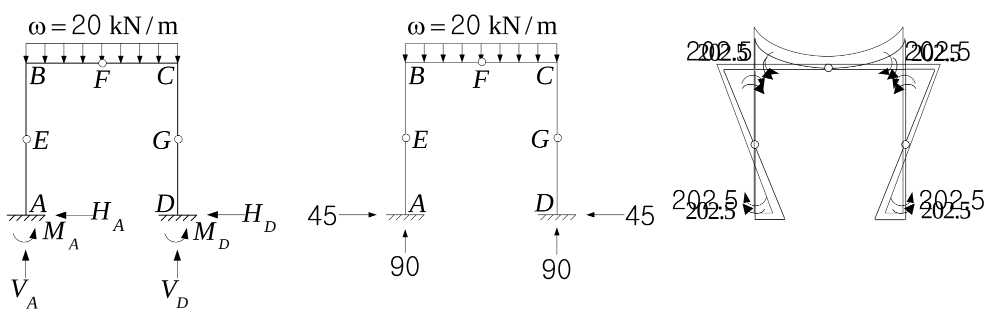
$$\sum M = 0 : M_A + M_D + 9V_D - 20 \times 9 \times 4.5 = 0$$

$$M_E = 0 : -4.5H_A + M_A = 0$$

$$M_G = 0 : -4.5H_D + M_D = 0$$

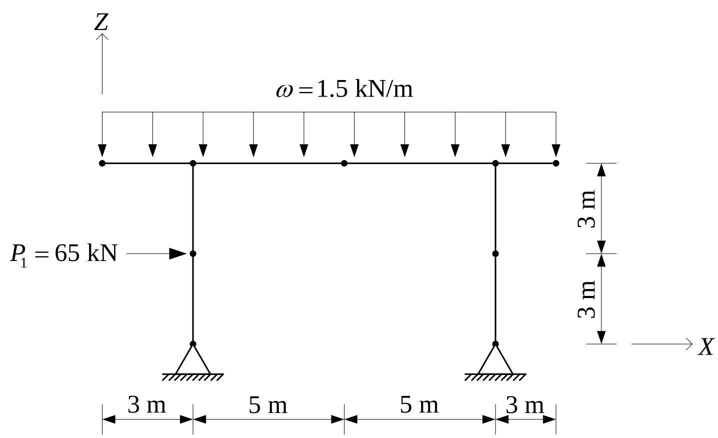
$$M_F = 0 : -9H_D + 4.5V_D + M_D - 20 \times 4.5 \times \frac{4.5}{2} = 0$$

▶Fig. 6.31



4. Exercise

Find the shear force, bending moment and axial force diagrams and the deformed shape of the frame below.



- ***Material***: Concrete strength M40 used in Tutorial
- ***Section***

  Shape: Solid Rectangle

  Size: B×H 0.5 × 0.5 m
- ***Load***

  Distributed load: 1.5 kN/m

  Nodal load: 65 kN

7. Frame Analysis with **Inclined Supports**

Contents

1. Introduction

1.1 Concept for Inclined Support Analysis

A structure may be supported on inclined supports relative to Global Coordinate System (GCS). It is important to recognize that deformation and member forces can vary depending on the orientation of supports.

▶Photo 7.1
Inclined supports

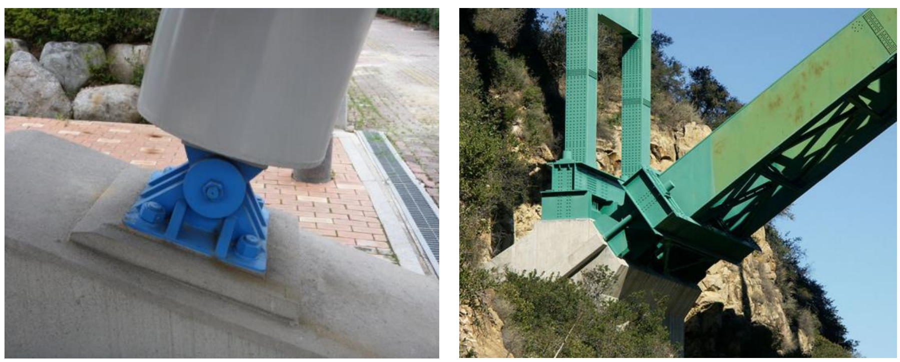


Fig. 7.1 shows identical structures, but the support conditions are slightly different. Fig. 7.1 (a) is restrained at Point C parallel to Member BC, whereas Fig. 7.1 (b) is vertically restrained at Point C.

▶Fig. 7.1
(a) Inclined supports
(b) Vertical supports

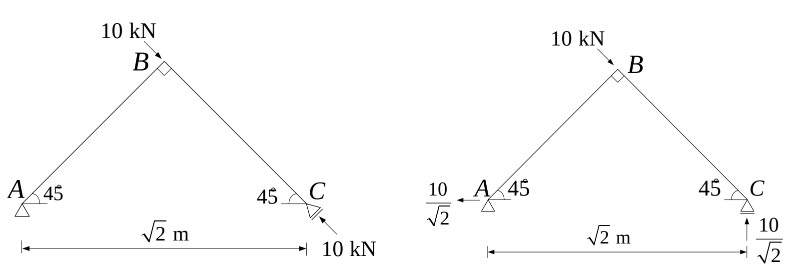
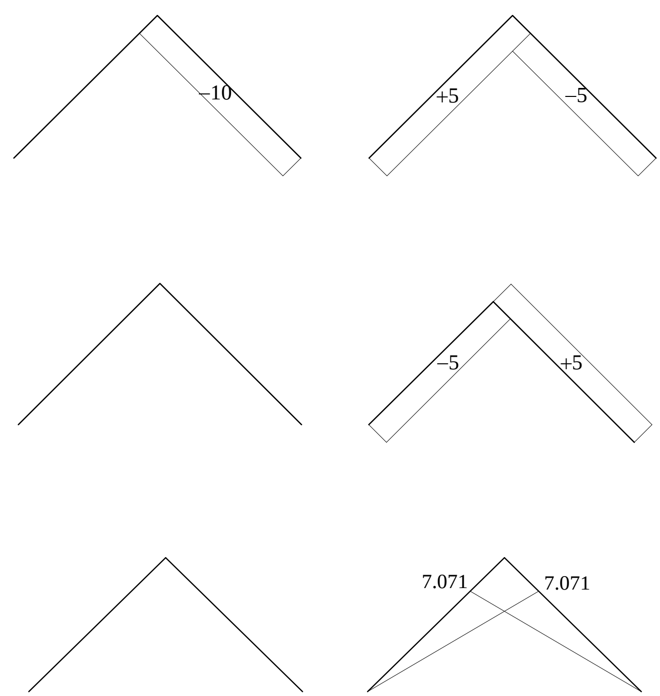


Fig. 7.2 shows the axial force, shear force and bending moment diagrams of the 2 structures, which illustrate significant differences in member forces depending on the support conditions.

In the case of an inclined support, Fig. 7.2 (a) shows only a reaction at Point C with an axial force in Member BC. However all the Members AB and BC are subject to bending moments with a reaction at Point A in the vertical support case as shown in Fig. 7.2 (b).



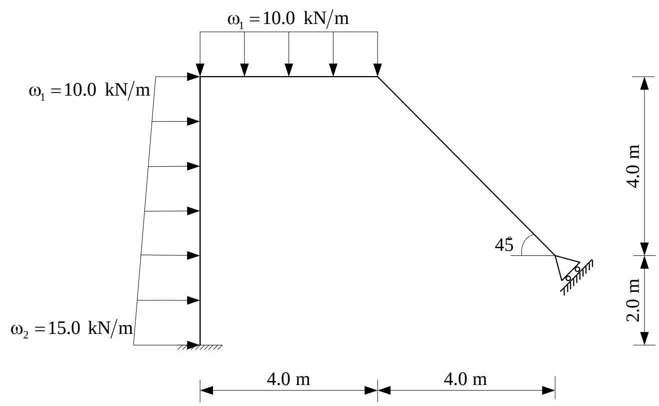
2. Tutorial

2.1 Model Overview

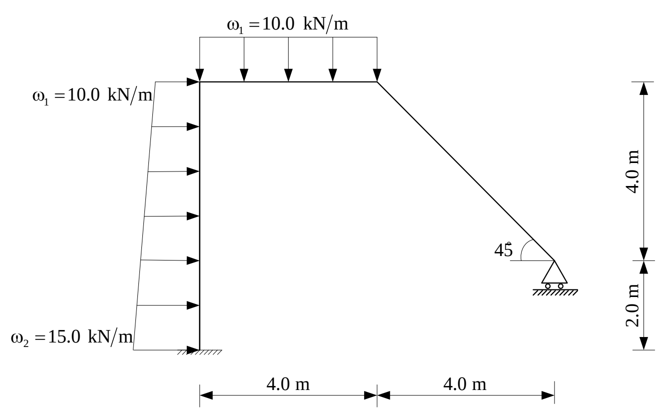
Compare the displacements, member forces and reactions between the 2 structures subjected to a uniformly distributed vertical load and a horizontal load of trapezoidal distribution. Both frame structures are identical except for the support conditions.

▶Fig. 7.3
Analysis models

(a) Inclined support



(b) Vertical support



- **Material:** Fe570
- **Section:** Box 1,000×1,000×10 mm
- **Load:** 1. Distributed vertical load on horizontal member: 10.0 kN/m
  2. Trapezoidal horizontal load on vertical member: 10.0 - 15.0 kN/m

2.2 Work Environment

Open a New File and save.

Main Menu: */ New Project...*

Main Menu: */ Save As...*

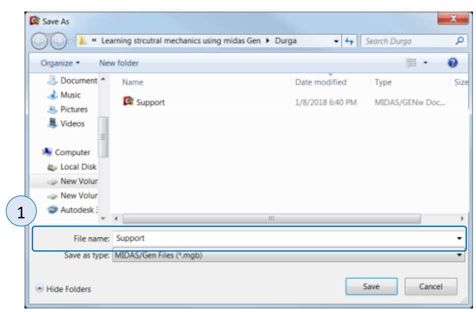
1. File name: **'Support'** and **[Save]**

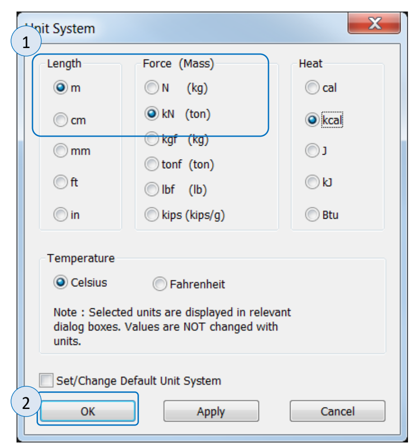
Set a unit system to kN & m.

Main Menu: ***Tools / Unit System...***

1. Length > **m**, Force (Mass) > **kN (ton)**
2. **[OK]**

▶Fig. 7.4
Save file &
set unit system



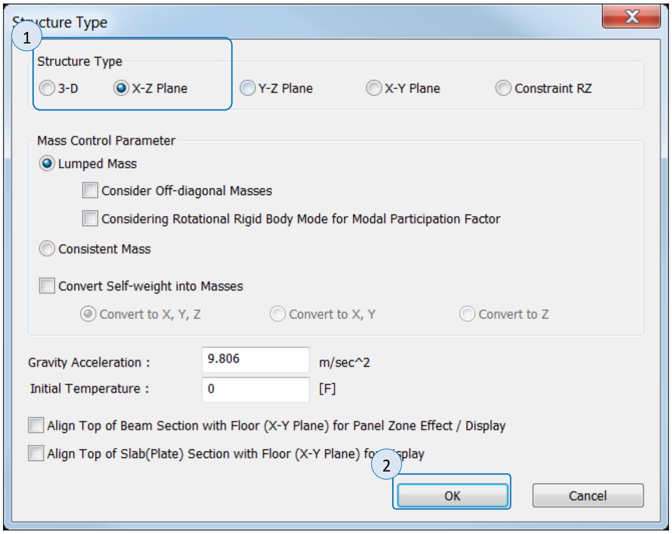


The example models exist in the 2-D X-Z plane.

Main Menu: ***Structure / Structure Type***

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

▶Fig. 7.5
Set a work plane



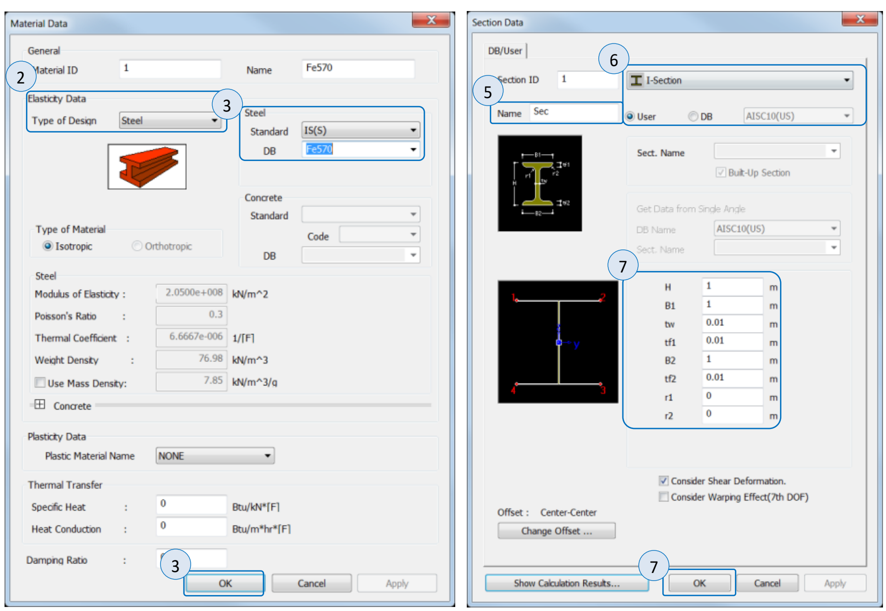
2.3 Material & Section Properties

Define material and section for the structural members.

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

1. **[Add...]**
2. Type of Design > **Steel**
3. Steel > Standard >**IS(S)**, DB > **Fe570, [OK]**
4. Properties dialog window: click **Section** tab, **[Add…]**
5. Name: **'Sect'**
6. Section shape list > **Box**, Section definition method > **User**
7. H: **'1'**, B1=B2 : **'1'**, tw: **'0.01'**, tf1=tf2 : **'0.01'**, **[OK]**, **[Close]**

▶Fig. 7.6
Define material & section

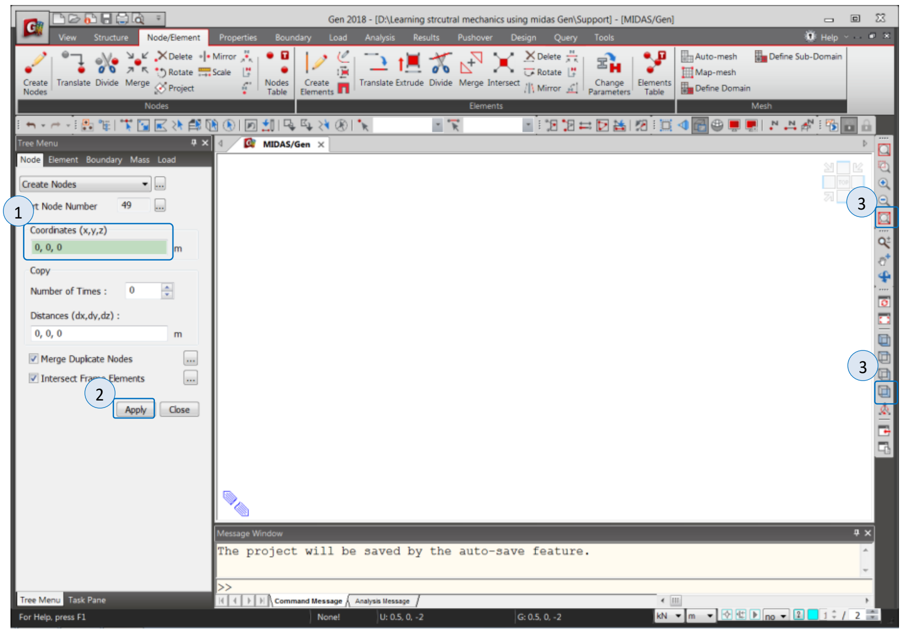


2.4 Nodes & Elements Generation

Create a node in order to generate elements.

Main Menu: ***Node/Element > Create Nodes***
1. Coordinates (x, y, z): **'0, 0, 0'**
2. **[Apply]**
3. **Auto Fitting, Front View** (Toggle on)

▶Fig. 7.7
Create a node



Create the column in Model 1 by Extrude Elements, which extrudes Node to Line Element.

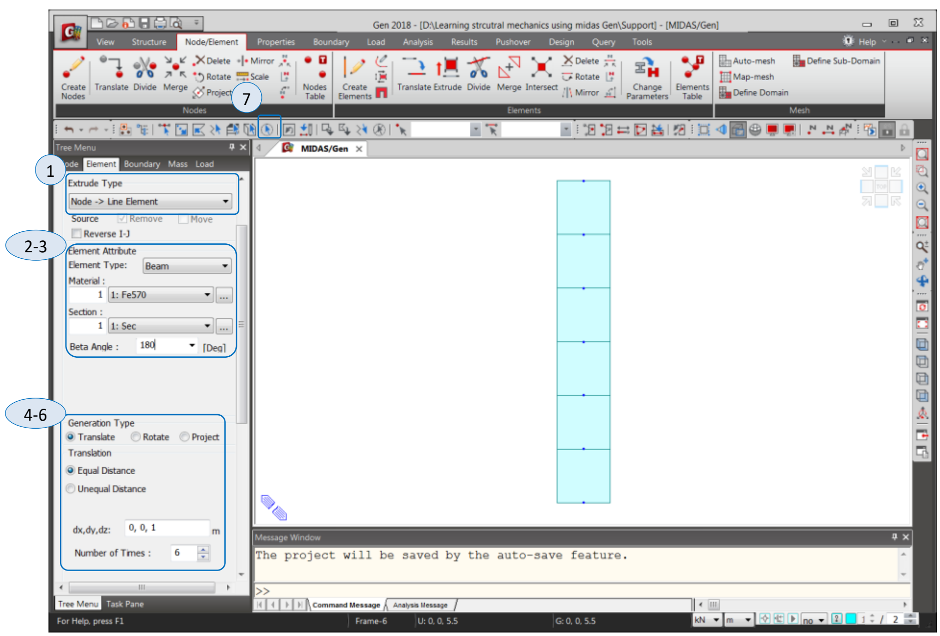
Main Menu: ***Node/Element > Extrude***

1. Extrude Type > **Node → Line Element**
2. Element Attribute > Element Type > **Beam**
3. Material > **1: Fe570**, Section > **1: Sect**, Beta Angle: **'180'**
4. Generation Type > **Translate**
5. Translation > **Equal Distance**
6. dx, dy, dz: **'0, 0, 1'**, Number of Times: **'6'**
7. **Select All**
8. **[Apply]**

Tip

Beta Angle refers to the orientation of z-axis of Truss, Beam or Column line elements in Element Coordinate System.

Unless a section is symmetrical in both axes (like a circle or square section), the stiffness of the section changes with Beta Angle.



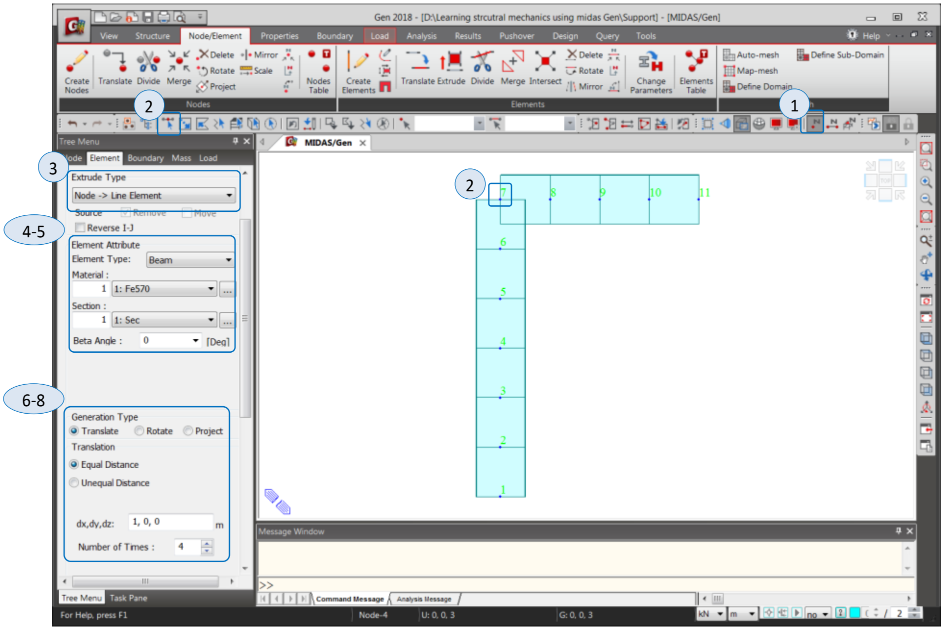
▶Fig. 7.8
Create column

Extrude the top node of the column to create the horizontal member.

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

1. **Display Node Numbers** (Toggle on)
2. **Select Single** (on) mode: select **Node 7**
3. Extrude Type > **Node → Line Element**
4. Element Attribute > Element Type > **Beam**
5. Material > **1: Fe570**, Section > **1: Sect**, Beta Angle: **'0'**
6. Generation Type > **Translate**
7. Translation > **Equal Distance**
8. dx, dy, dz: **'1, 0, 0'**, Number of Times: **'4'**
9. [Apply]

▶Fig. 7.9
Create horizontal member

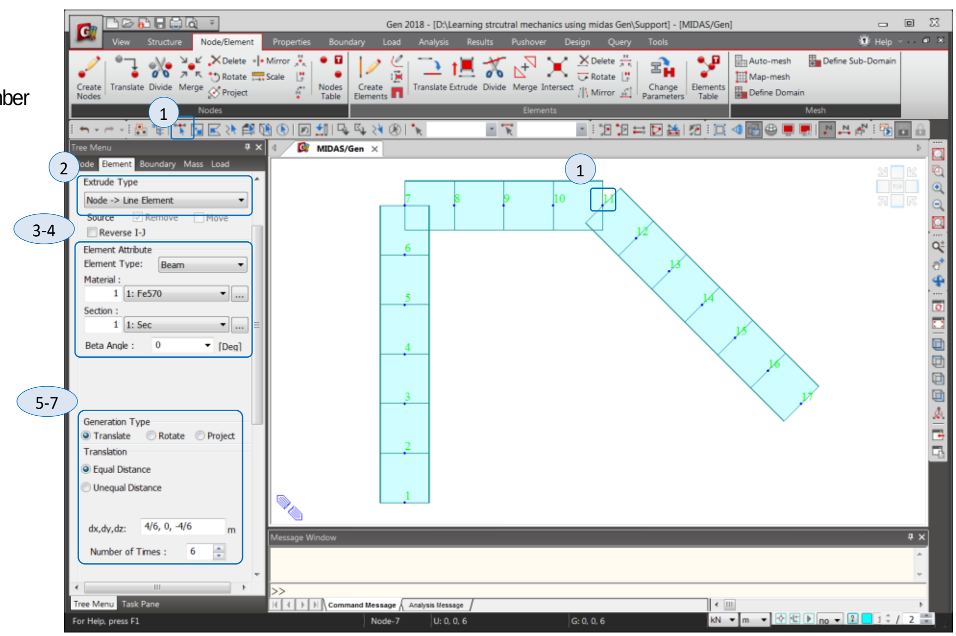


Extrude the right end of the horizontal member to create the sloped member.

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

1. **Select Single** (on) mode: select **Node 11**
2. Extrude Type > **Node → Line Element**
3. Element Attribute > Element Type > **Beam**
4. Material > **1: Fe570**, Section > **1: Sect**, Beta Angle: **‘0’**
5. Generation Type > **Translate**
6. Translation > **Equal Distance**
7. dx, dy, dz: **‘4/6, 0, -4/6’**, Number of Times: **‘6’**
8. **[Apply]**

▶Fig. 7.10
Create sloped member



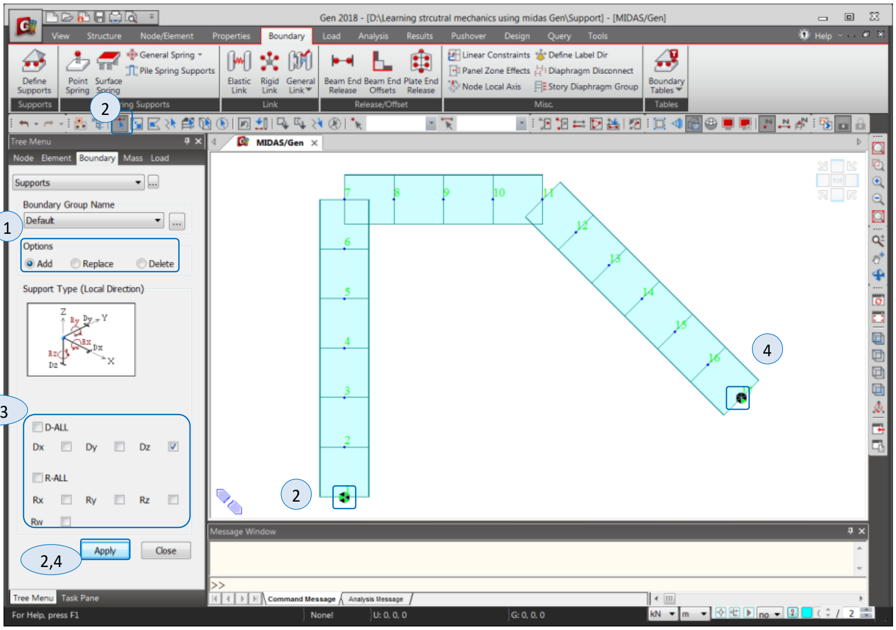
2.5 Define Boundary Conditions

All nodes in 3-D space retain 6 degrees of freedom (Dx, Dy, Dz, Rx, Ry, Rz). In this tutorial, the degrees of freedom are permitted in the X-Z plane only, leaving only 3 degrees of freedom (Dx, Dz, Ry). For a fixed condition, Dx, Dz, Ry degrees of freedom are restrained, and for a roller support, Dz degree of freedom is restrained.

Main Menu: ***Boundary / Define Supports***

1. Options > **Add**, Support Type > **Dx, Dz, Ry** (Check on)
2. **Select Single** (on) mode: select **Node 1**, **[Apply]**
3. Support Type > **Dx, Ry** (Check off)
4. Select **Node 17, [Apply]**

▶Fig. 7.11
Input support conditions

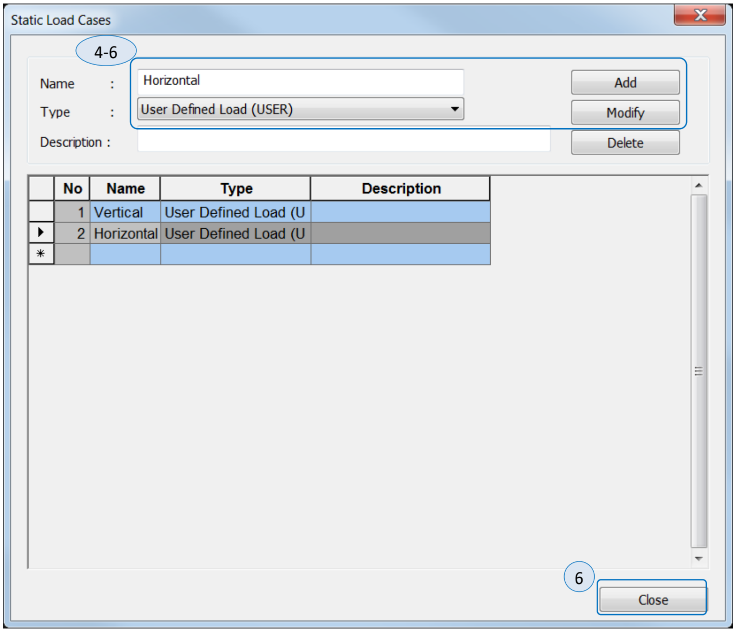


2.6 Define Loads

Define the load cases (load types) first to which the loadings will belong.

Main Menu: ***Load / Static Loads / Static Load Cases***
1. Name: **'Vertical'**
2. Type > **User Defined Load (USER)**
3. **[Add]**
4. Name: **'Horizontal'**
5. Type > **User Defined Load (USER)**
6. **[Add]**, **[Close]**

▶Fig. 7.12
Define load cases

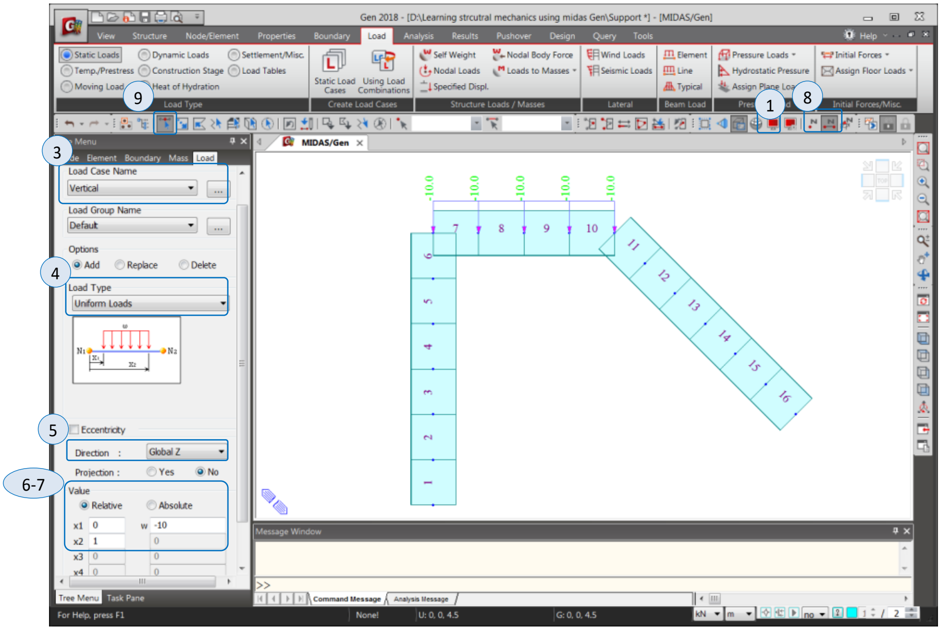


Input a uniformly distributed load, 10 kN/m on the beam.

Main Menu: ***Load / Static Loads / Element Beam Loads...***

1. **Display** (Toggle on) Load tab: **Load Value** (Check on)
2. **[OK]**
3. Load Case Name > **Vertical**
4. Load Type > **Uniform Loads**
5. Direction > **Global Z**
6. Value > **Relative**
7. x1: **'0'**, x2: **'1'**, w: **'-10'**
8. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
9. **Select Single** (on) mode: select Elements **7, 8, 9, 10**
10. **[Apply]**

▶Fig. 7.13
Input uniformly distributed load

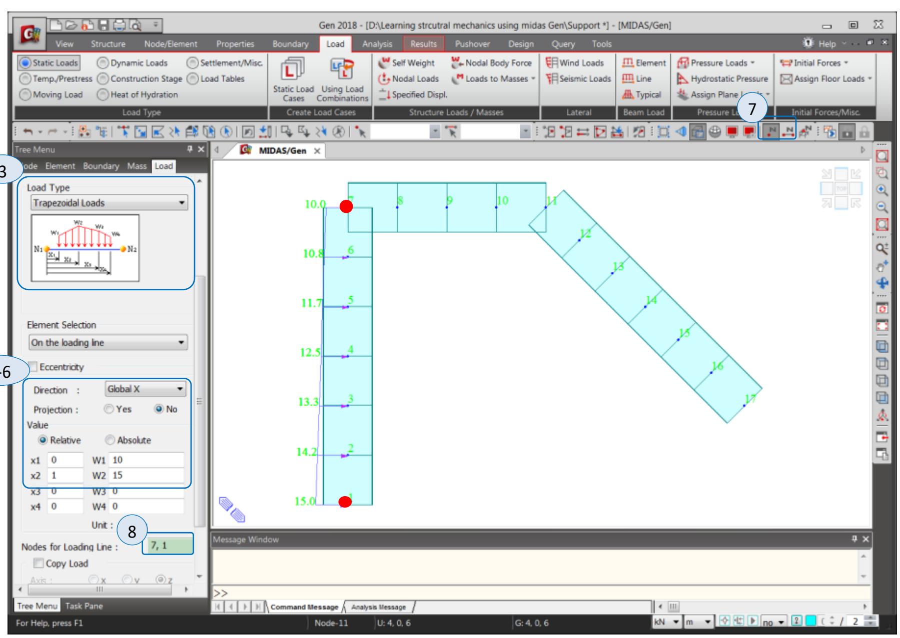


Input a trapezoidal load on the column, 10.0 - 15.0 kN/m distribution.

Main Menu: ***Load / Static Loads / Line Beam Load***

1. Load Case Name > **Horizontal**
2. Options > **Add**
3. Load Type > **Trapezoidal Loads**
4. Direction > **Global X**, Projection > **No**
5. Value > **Relative**
6. x1: **'0'**, x2: **'1'**, W1: **'10'**, W2: **'15'**
7. **Display Node Numbers** (Toggle on), **Display Element Numbers** (Toggle off)
8. Click the Nodes for Loading Line field (which turns to green), click **Nodes 7** and **1** sequentially in Model View
9. **[Close]**
10. Works Tree: right-click on Static Load case 2 [Horizontal] Line Beam Loads: 6 > Undisplay

▶Fig. 7.14
Input trapezoidal load on vertical member

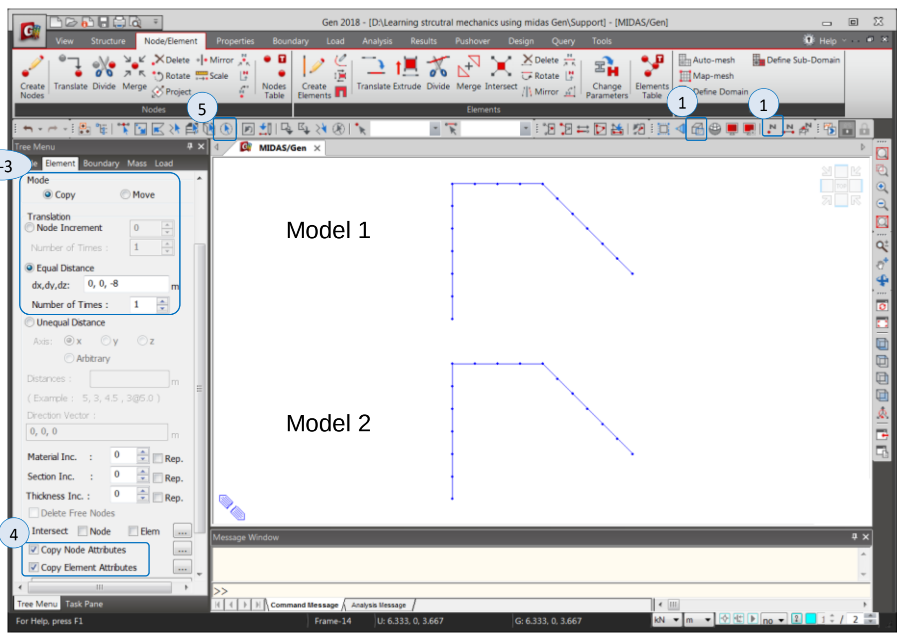


Copy Model 1 to create Model 2. Loadings and support conditions assigned to Model 1 will be copied simultaneously.

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

1. **Hidden, Display Node Numbers** (Toggle off)
2. Mode > **Copy**
3. Translation > Equal Distance > dx, dy, dz: **‘0, 0, -8’**
4. **Copy Node Attributes, Copy Element Attributes** (Check on)
5. **Select All** (Toggle on)
6. **[Apply]**

▶Fig. 7.15
Copy elements

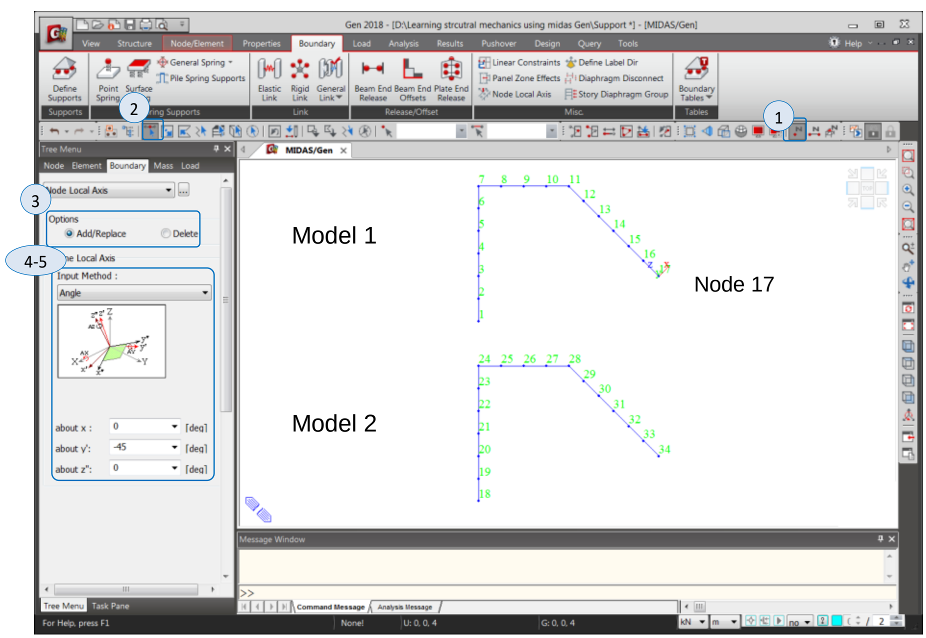


Define Node Local Coordinate System (NLCS) to reflect the inclined support condition at the roller in Model 1. NLCS is applied to the corresponding node only, which takes priority over GCS. NLCS is used when displacements or support conditions exist in specific directions, which do not coincide with GCS. Analysis results (displacements, reactions, etc.) can be also examined in NLCS.

Main Menu: ***Boundary / Node Local Axis***

1. **Display Node Numbers** (Toggle on)
2. **Select Single** (on) mode: click **Node 17**
3. Options > **Add/Replace**
4. Define Local Axis > Input Method > **Angle**
5. about x: **'0'**, about y': **'-45'**, about z": **'0'**
6. **[Apply]**

▶Fig. 7.16
Change support condition



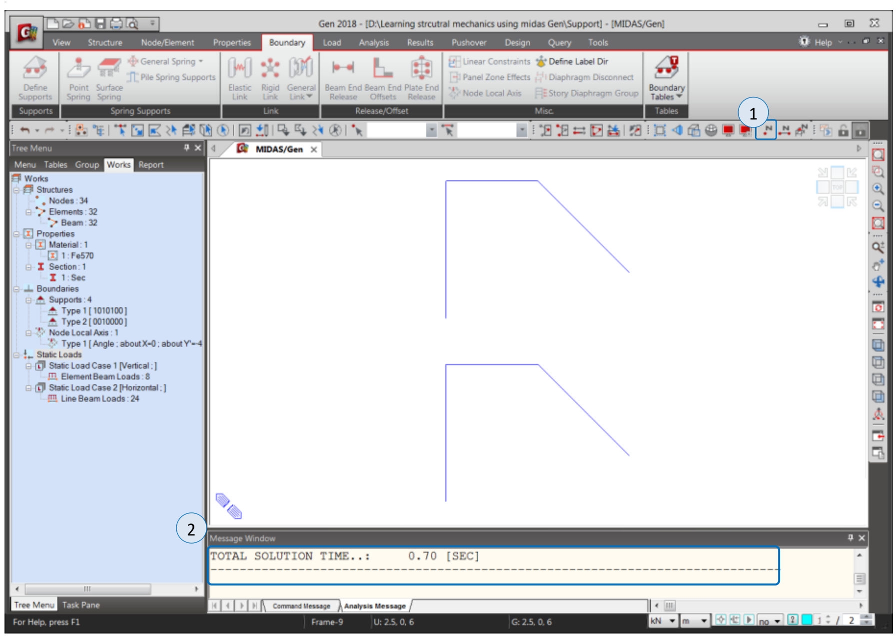
2.7 Perform Analysis

Analyze the 2 Models.

Main Menu: ***Analysis / Perform Analysis***

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

▶Fig. 7.17
Message for a successful run



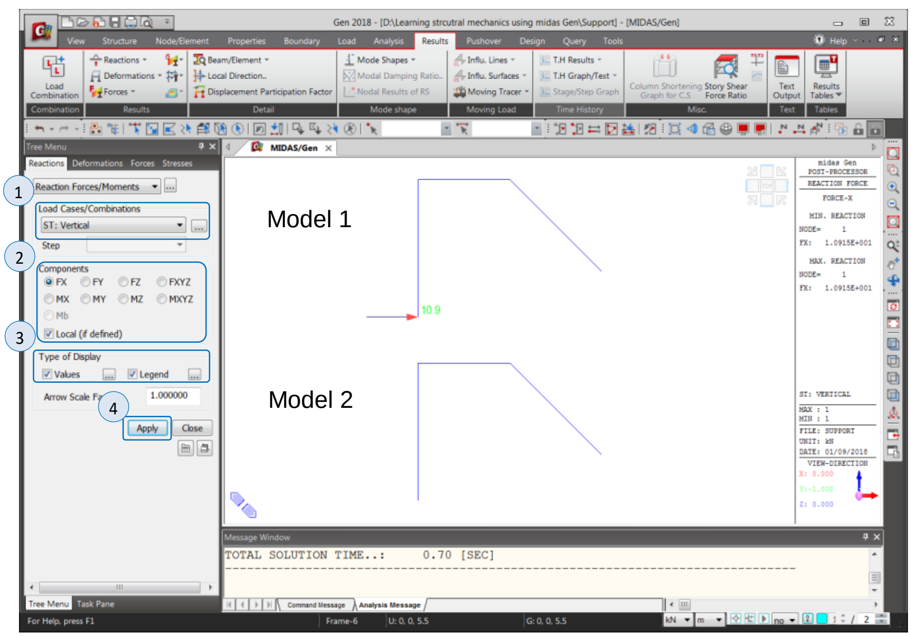
2.8 Check Analysis Results

Check the reactions due to the vertical load.
Note that a horizontal reaction is observed at the left fixed support in Model 1.

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

1. Load Cases / Combinations > **ST: Vertical**
2. Components > **FX**, Local (if defined) (Check on)
3. Type of Display > **Values, Legend** (Check on)
4. **[Apply]**

Tip
Local (if defined) is used for reactions in local coordinates if defined.



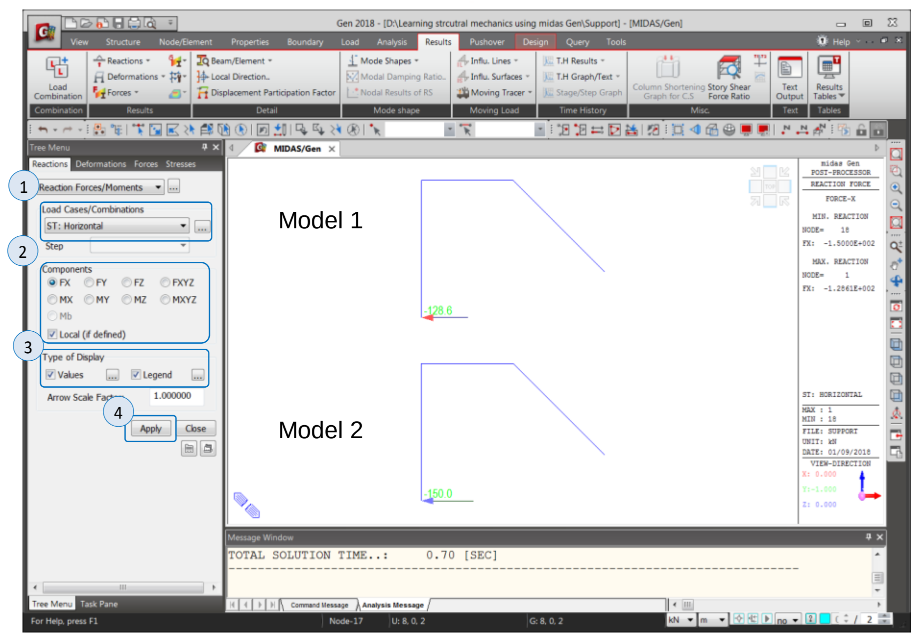
▶Fig. 7.18
Horizontal reaction due to vertical load

Check reactions due to the horizontal load.

The fact that the inclined support in Model 1 resists the horizontal load, the horizontal reaction at the left fixed end in Model 1 is less than that in Model 2.

Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
1. Load Cases / Combinations > **ST: Horizontal**
2. Components > **FX**
3. Type of Display > **Values, Legend** (Check on)
4. **[Apply]**

▶Fig. 7.19
Horizontal reactions due to horizontal load



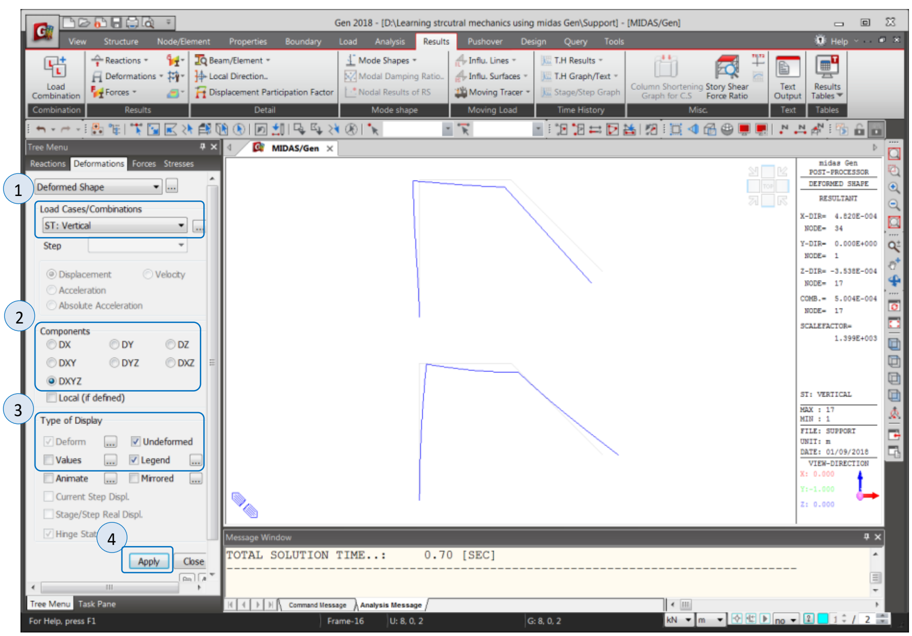
Check the deformed shapes of the 2 Models. DXZ here means $\sqrt{DX^2 + DZ^2}$.

The vertical load causes Model 1 to deform to the left due to the presence of the inclined support. Model 2 is observed to deform to the right.

Main Menu: ***Results / Deformations / Deformed Shape...***

1. Load Cases / Combinations > **ST: Vertical**
2. Components > **DXZ** (or DXYZ)
3. Type of Display > **Undeformed, Legend** (Check on), **Values** (Check off)
4. **[Apply]**

▶Fig. 7.20
Deformations due to vertical load

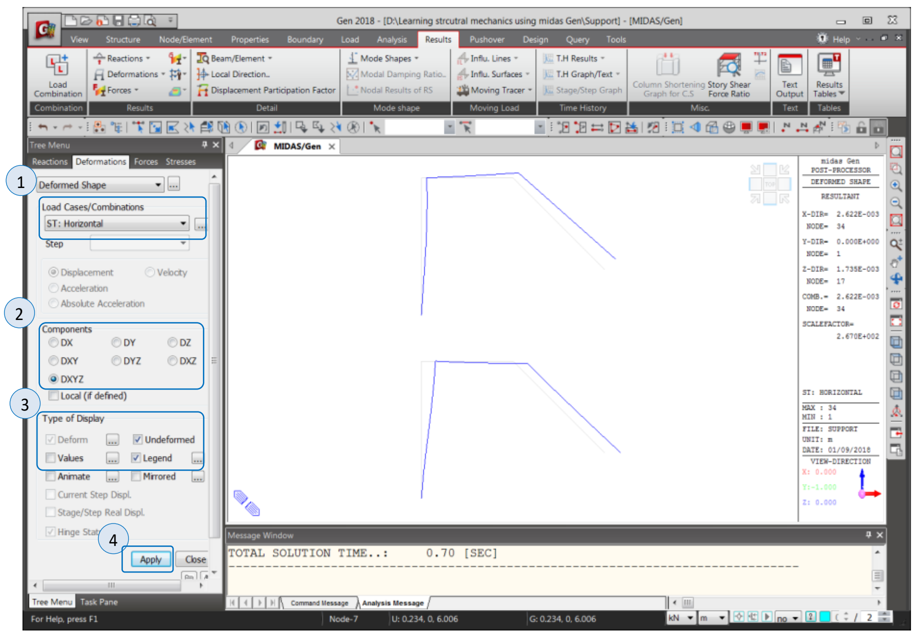


Check the deformed shapes of the 2 Models due to the horizontal load.

Main Menu: ***Results / Deformations / Deformed Shape...***

1. Load Cases / Combinations > **ST: Horizontal**
2. Components: **DXYZ**
3. Type of Display > **Undeformed, Legend** (Check on), **Values** (Check off)
4. **[Apply]**

▶Fig. 7.21
Deformation due to horizontal load



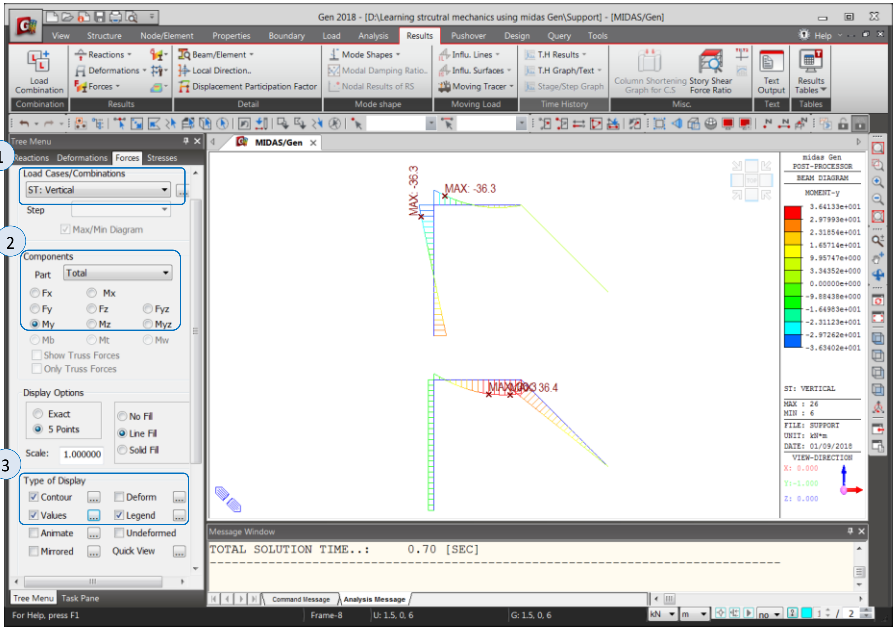
Check bending moments due to the vertical load.

Due to the horizontal reaction of the inclined support in Model 1, large bending moment occurs in the column. On the other hand, large moment is observed in the sloped member in Model 2 without the inclined support.

Main Menu: ***Results / Forces / Beam Diagrams...***

1. Load Cases/Combinations > **ST: Vertical**
2. Components > **My**
3. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
4. Click **[...]** next to Values
5. **MinMax Only, Abs Max** (Check on)
6. **[OK]**

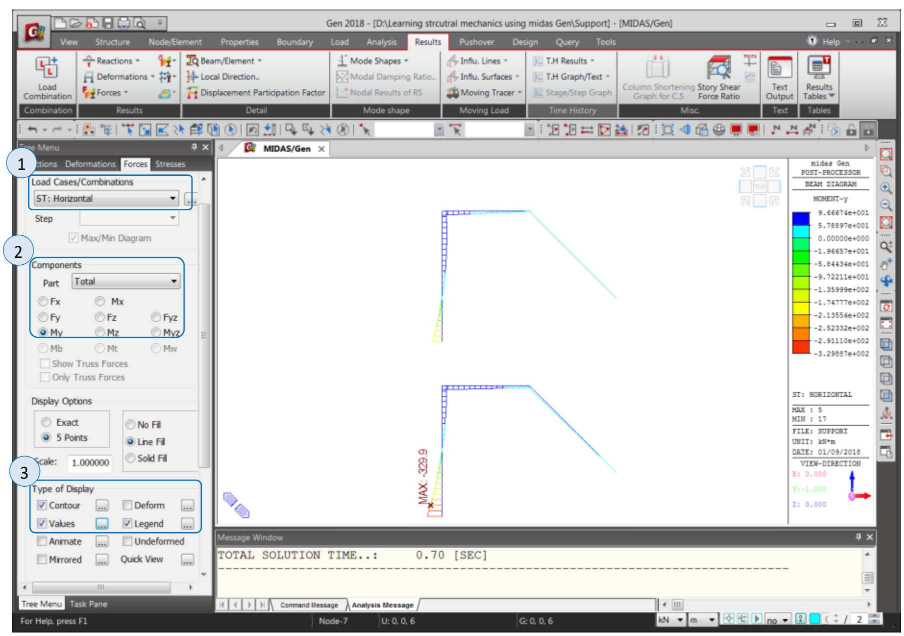
▶Fig. 7.22
Bending moments due to vertical load



Check bending moments due to the horizontal load.
Though bending moments in the 2 Models share a similar pattern, Model 2 with the absence of the inclined support exhibits larger moment at the fixed end in comparison with Model 1.

Main Menu: ***Results / Forces / Beam Diagrams...***
1. Load Cases/Combinations > **ST: Horizontal**
2. Components > **My**
3. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
4. Click **[...]** next to Values
5. **MinMax Only, Abs Max** (Check on)
6. **[OK]**

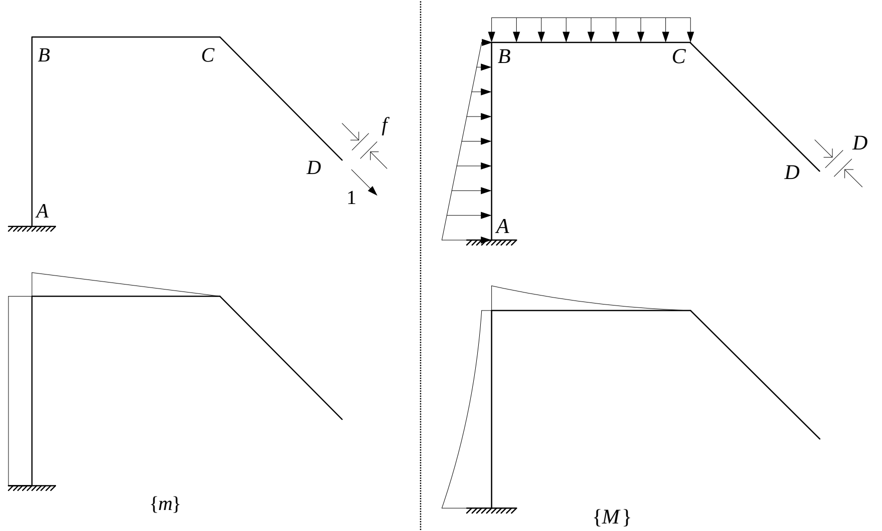
▶Fig. 7.23
Bending moments due to horizontal load



3. Manual Calculations

The method of consistent deformation can be used by defining the reaction in the slope direction at Point D in Model 1 as an unknown.

▶Fig. 7.24
Model 1



AB Zone: $m(x)=\dfrac{1}{\sqrt{2}}(x-10)$	*AB* Zone: $M(x)=-290+75x-\dfrac{15}{2}x^2+\dfrac{5}{36}x^3$
BC Zone: : $m(x)=\dfrac{4}{\sqrt{2}}(\dfrac{x}{4}-1)$	*BC* Zone: $M(x)=-80+40x-5x^2$

The displacement at Point D due to a unit load applied in the direction of Member CD is defined as f in the state of the support restraint released at Point D. The displacement at Point D in the direction of Member CD due to the member forces on Members AB and BC is now defined as D. From this, a compatibility equation can be derived.

$$f \cdot X + D = 0$$

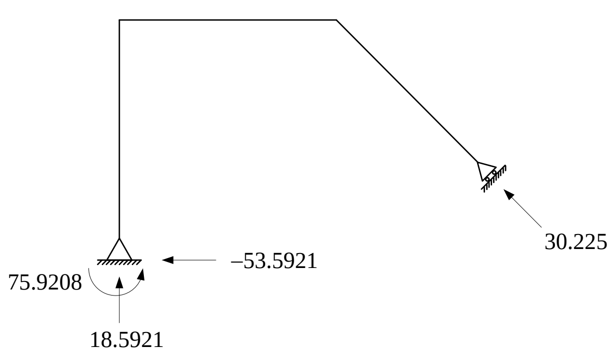
where,

$$f = \frac{1}{EI}\int_0^1 \{m(x)\}^2 dx$$
$$= \frac{1}{EI}\int_0^6 \left\{\frac{1}{\sqrt{2}}(x-10)\right\}^2 dx + \frac{1}{EI}\int_0^4 \left\{\frac{4}{\sqrt{2}}\left(\frac{x}{4}-1\right)\right\}^2 dx$$
$$= 166.667/EI$$

$$D = \frac{1}{EI}\int_0^1 \{m(x)\}\{M(x)\}dx$$
$$= \frac{1}{EI}\int_0^6 \left\{\frac{1}{\sqrt{2}}(x-10)\right\}\left\{-290+75x-\frac{15}{2}x^2+\frac{5}{36}x^3\right\}dx$$
$$+\frac{1}{EI}\int_0^4 \left\{\frac{4}{\sqrt{2}}\left(\frac{x}{4}-1\right)\right\}\left\{-80+40x-5x^2\right\}dx$$
$$= 5045.914/EI$$

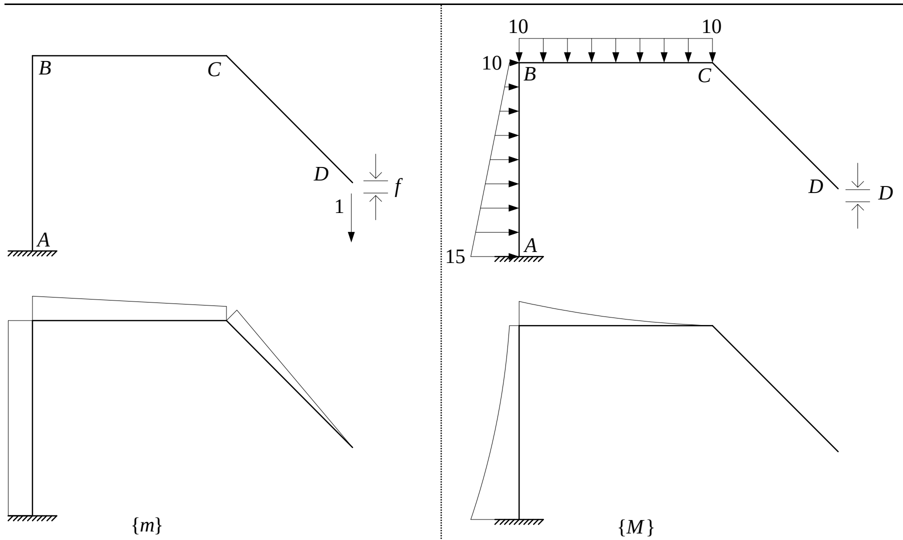
$$X = -\frac{D}{f} = -30.225$$

Accordingly, the reaction at Point D becomes 30.225 kN, the direction of which is upward and opposite to the direction of the unit load.



▶Fig. 7.25
Model 1 calculation results

The method of consistent deformation can be now used for Model 2 by defining the vertical reaction at Point D as an unknown. Releasing the vertical support restraint at Point D and applying a unit load in the reaction direction results in the moment, $m(x)$. Applying the loads on Members AB and BC in the released condition results in a moment distribution, $M(x)$ as shown in Fig. 7.26.



AB Zone: $m(x) = -8$

BC Zone: $m(x) = -8 + x$

CD Zone: $m(x) = -4 + \frac{x}{\sqrt{2}}$

AB Zone: $M(x) = -290 + 75x - \frac{15}{2}x^2 + \frac{5}{36}x^3$

BC Zone: $M(x) = -80 + 40x - 5x^2$

▶Fig. 7.26
Model 2

The vertical displacement, f at Point D is caused by a unit load. The vertical displacement, D at Point D is caused by the loads on Members AB and BC. The following compatibility equation can be derived:

$$f \cdot X + D = 0$$

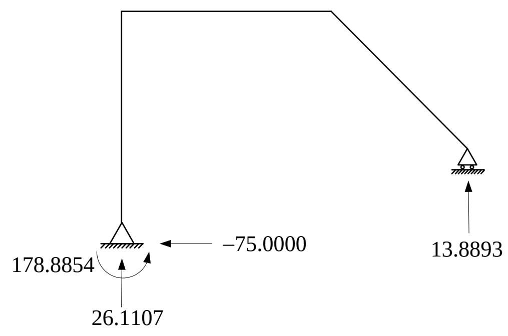
where,

$$f = \frac{1}{EI}\int_0^1 \{m(x)\}^2 dx$$
$$= \frac{1}{EI}\int_0^6 \{-8\}^2 dx + \frac{1}{EI}\int_0^4 \{-8+x\}^2 dx + \frac{1}{EI}\int_0^{4\sqrt{2}} \left\{\frac{x}{\sqrt{2}} - 4\right\}^2 dx$$
$$= 563.5032/EI$$

$$D = \frac{1}{EI}\int_0^1 \{m(x)\}\{M(x)\} dx$$
$$= \frac{1}{EI}\int_0^6 \{-8\}\left\{-290 + 75x - \frac{15}{2}x^2 + \frac{5}{36}x^3\right\} dx + \frac{1}{EI}\int_0^4 \{-8+x\}\{-80+40x-5x^2\} dx$$
$$= 7826.6667/EI$$
$$X = -\frac{D}{f} = -13.8893$$

Hence the reaction at Point D becomes 13.8893 kN (upward).

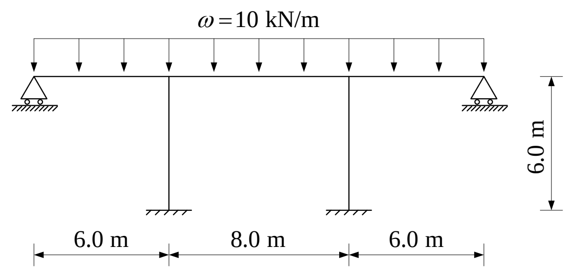
▶Fig 7.27
Model 2 calculation results



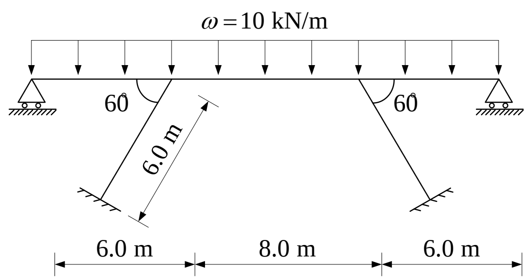
4. Exercise

Compare the member forces, displacements and reactions among the 3 frames of different column slopes below. Material and section are identical to those in Tutorial.

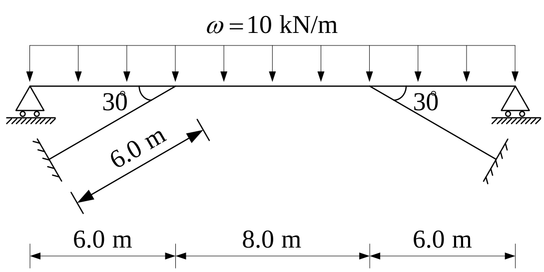
(a)



(b)



(c)



8. **Springs** Analysis

Contents

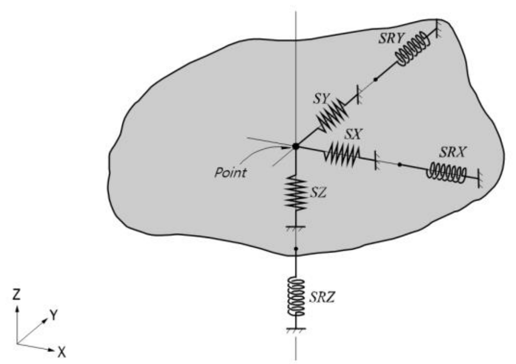
1. Introduction

1.1 Concept for Springs Analysis

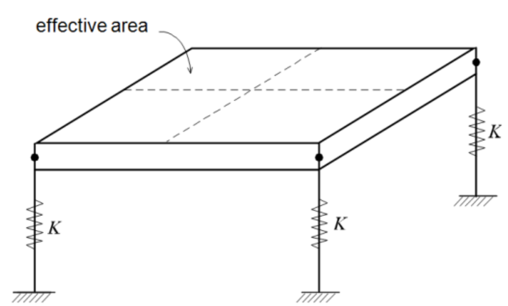
An elastic boundary element or a spring is used when the stiffness of a contiguous structure or ground at the boundary of a model needs to be reflected. It is also used when elements (truss, plane stress, plate element, etc.) need to be connected with the absence of certain degrees of freedom.

For a given node, springs can be specified for 6 degrees of freedom (3 translational and 3 rotational) in Global Coordinate System (GCS). Force per unit length is specified for a translational elastic component, and a moment unit per radian is specified for a rotational elastic component.

▶Fig. 8.1
Elastic boundary element examples
(a) Boundary condition for Point Spring Support



(b) Surface Spring Supports



A translational elastic boundary element is appropriately used to represent ground stiffness at the bottom of a column or a pile at the lower part of a structure (substructure). Modulus of subgrade reaction multiplied by the tributary area of the corresponding node is used to model the ground. It should be noted that the ground can only resist compression, not tension.

midas Gen provides the Surface Spring Supports function to readily represent the boundary condition of a surface in contact with soils. Such boundary condition is specified in Model > Boundaries > Surface Spring Supports. Once relevant nodes are selected, and the modulus of subgrade reaction parameter is specified, midas Gen automatically calculates the spring stiffness of the selected nodes based on the effective area of each node. If the analysis needs to reflect the compression-only resistance of the ground, then Elastic Link (Compression only) needs to be specified.

▶Table 8.1
Representative moduli of subgrade reaction

Ground type	Modulus of subgrade reaction (kN/㎥)
Soft clay	12,000~24,000
Medium stiff clay	24,000~48,000
Stiff clay	48,000~112,000
Loose sand	4,800~16,000
Medium dense sand	9,600~80,000
Silty medium dense sand	24,000~48,000
Clayey gravel	48,000~96,000
Clayey medium dense sand	32,000~80,000
Dense sand	64,000~130,000
Very dense sand	80,000~190,000
Silty gravel	80,000~190,000

[Table 8.1]
summarizes moduli of subgrade reaction for soils, which are typically encountered in practice. It is recommended that both maximum and minimum values be used separately, and conservative values with discretion be implemented for design.

The axial stiffness of spring supports for columns or piles can be calculated by EA/H, where E is the modulus of elasticity for columns or piles, A is effective cross-sectional area, and H is effective length.

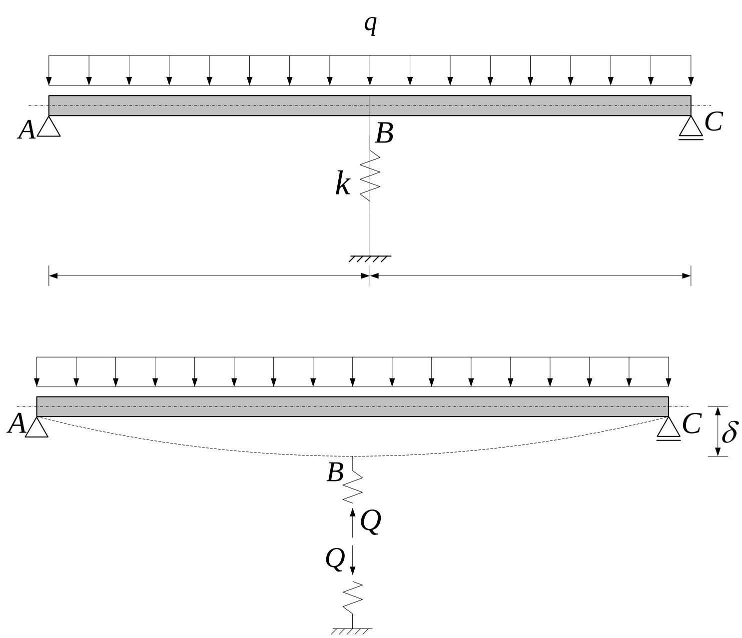
Rotational spring components are used to represent the rotational stiffness of contiguous boundaries of the structure in question. If the contiguous boundary is a column, the stiffness is calculated by αEI/H, where α is a rotational stiffness component coefficient, I is effective moment of inertia, and H is the effective length of the column.

Generally, boundary springs at a node are entered in the direction of each D.O.F. For more accurate analysis, however, additional coupled stiffness associated with other degrees of freedom needs to be considered. That is, springs representing coupled stiffness may become necessary to reflect rotational displacements accompanied by translational displacements. For instance, it may be necessary to model pile foundation as boundary spring supports in which case more accurate analysis could be performed by introducing coupled stiffness in addition to the stiffness in each direction.

Boundary springs specified at a node, in General, follow the GCS unless an NCS is specified in which case they are defined relative to the NCS.

Subsequent to formulating the stiffness in the analysis stage, singular errors are likely to occur when stiffness components in certain degrees of freedom are deficient. If the rotational stiffness components are required to avoid such singular errors, it is recommended that the values from 0.0001 to 0.001 be used. The range of the values may vary somewhat depending on the unit system used. To avoid such singular errors, midas Gen and midas Civil, thus provides a function that automatically assigns stiffness values, which are insignificant to affect the analysis results.

A spring is a support, which can accompany deformation. Fig. 8.2 shows a 2- span continuous beam supported by a spring at the midpoint B. The structure can be considered as a simple beam subjected to an upward force, Q , which exerts on the spring support.



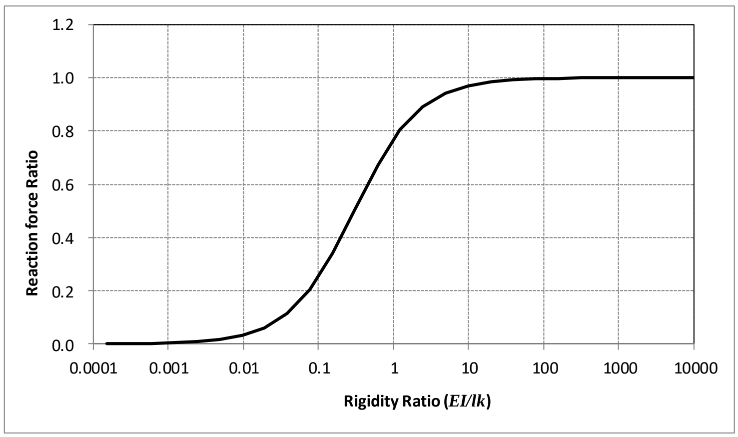
▶Fig. 8.2
Continuous beam with a spring support

$$E = 3\times10^{7}\ \text{kN/m}^2,\quad I = 1\times10^{-3}\ \text{m}^4$$
$$q = 10\ \text{kN/m},\quad l = 4.5\ \text{m}$$

The deflection of the above simple beam with a span of $2/l$, which is subjected to a uniformly distributed load, q and a concentrated load, Q , is the same as the change in length of the spring. The following governing equation can be written:

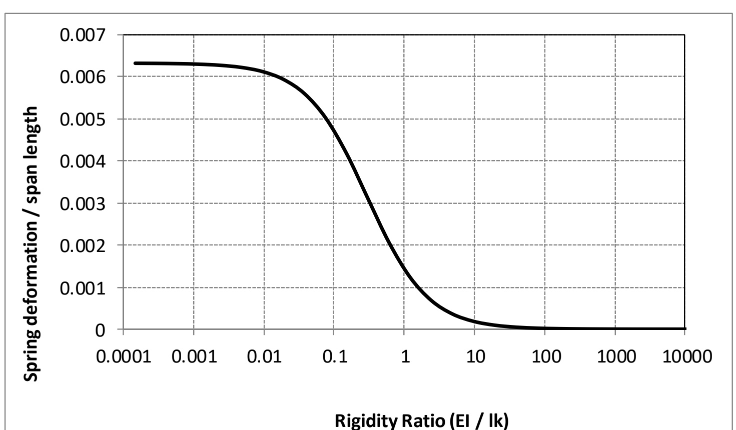
$$\frac{5q(2l)^4}{384EI} - \frac{Q(2l)^3}{48EI} = \frac{Q}{k}$$

Fig. 8.3 shows the relationship between the ratio of the flexural stiffness of the beam (EI/l) to the spring stiffness (k) (horizontal axis) and the reaction ratio (vertical axis). The reaction ratio is the spring reaction relative to the reaction caused by a vertical support at Point B. Since the reaction at the midpoint in a 2-span continuous beam is $5q(2/l)/8$, the spring reaction will always be a fraction of the hard support reaction.



▶Fig. 8.3
Relationship between the ratio of the flexural stiffness of the beam (EI/l) to the spring stiffness (k)

Fig. 8.4 shows the distribution of the spring stiffness (horizontal axis) and the spring deformation divided by the span length (vertical axis).



▶Fig. 8.4
Distribution of the spring stiffness (horizontal axis) and the spring deformation

It can be observed from Fig. 8.3 and Fig. 8.4 that the spring reaction increases with the increase in the spring stiffness, but the spring deformation decreases. When the spring stiffness ratio exceeds 10, the spring deformation becomes negligible exhibiting the behavior similar to that of the hard support condition. It can be also observed that a large transition takes place when the stiffness ratio is in the range of 0.1 - 1.0.

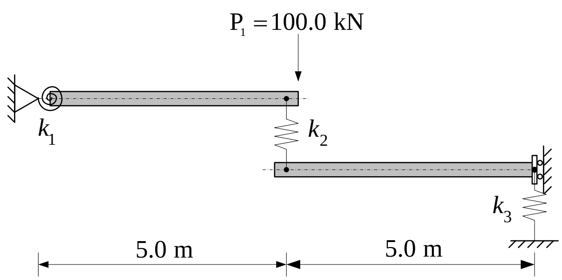
2. Tutorial

2.1 Model Overview

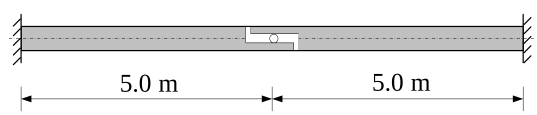
Compare the reactions, displacements and member forces between the models, which vary in support conditions and spring stiffness.

▶Fig. 8.5
Analysis models

Model 1,2,3:
Spring connections



Model 4:
Internal hinge connection



- ***Material***
  Steel: Fe440
  Modulus of elasticity (E): 2.05×10^{8} kN/m^2
- ***Section***
  Cross sectional area: 1.0×10^{-2} m^2
  Moment of inertia (Iyy): 8.333×10^{-4} m^4
- ***Load***
  Nodal concentrated load: 100.0 kN
- ***Spring constants***

Type	k_1 (kN · m/radian)	k_2 (kN /m)	k_3 (kN /m)
Model 1	1,000,000	10	100,000
Model 2	100	100,000	100,000
Model 3	1,000,000	100,000	100,000

2.2 Work Environment

Open a New File and save.

Main Menu: ***/ New Project...***

Main Menu: ***/ Save As...***

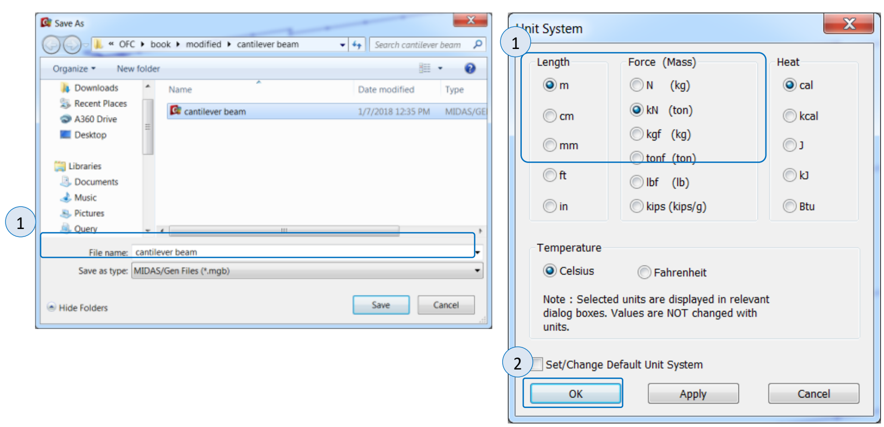
1. File name: **'Support'** and [Save]

Set a unit system.

Main Menu: ***Tools / Unit System...***

1. Length > **mm**, Force (Mass) > **N (kg)**
2. **[OK]**

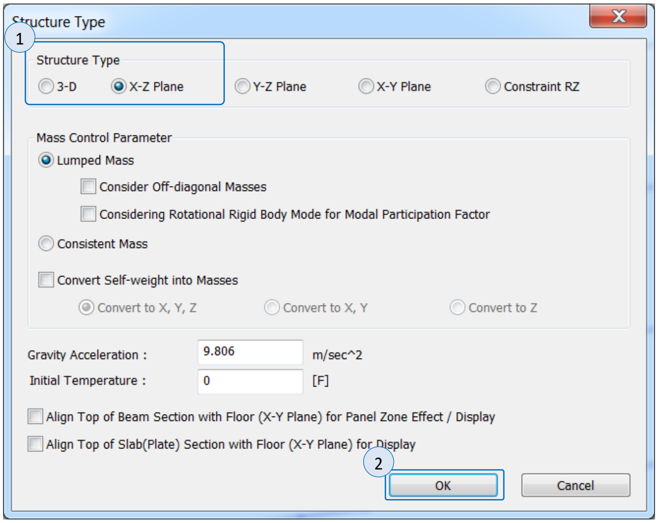
▶Fig. 8.6
Save file &
set unit system



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

Main Menu: ***Structure / Structure Type***

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



▶Fig. 8.7
Set a work plane

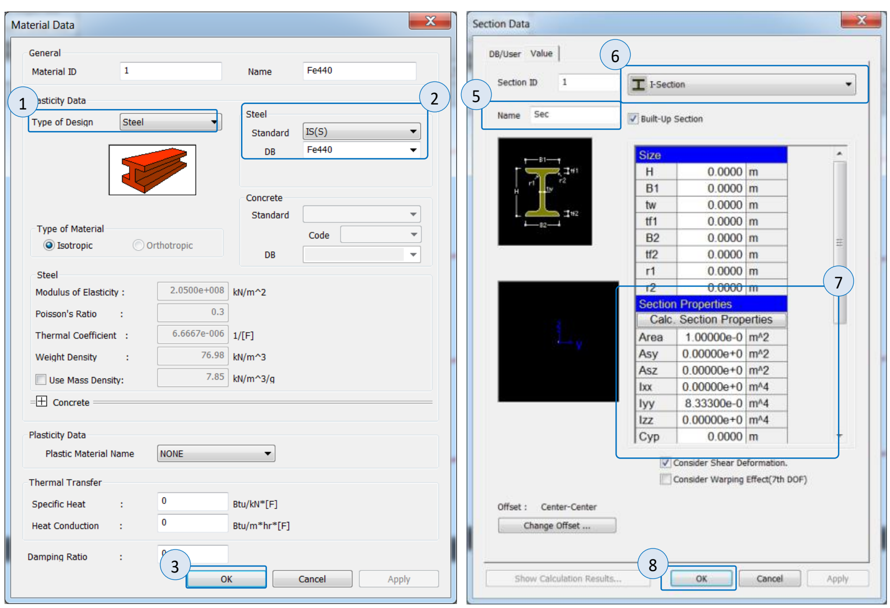
2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu: ***Properties / Material Properties***

1. **[Add...]**, Type of Design > **Steel**
2. Steel > Standard > **IS(S)**, DB > **Fe440**
3. **[OK]**
4. Properties dialog window: click **Section** tab, **[Add…]**
5. Value tab, Name: **'Sec'**
6. Section shape list > **I-Section**
7. Section Properties > Area: **'1e+4'**, Iyy: **'8.333e+8'**
8. **[OK]**, **[Close]**

▶Fig. 8.8
Define material & section

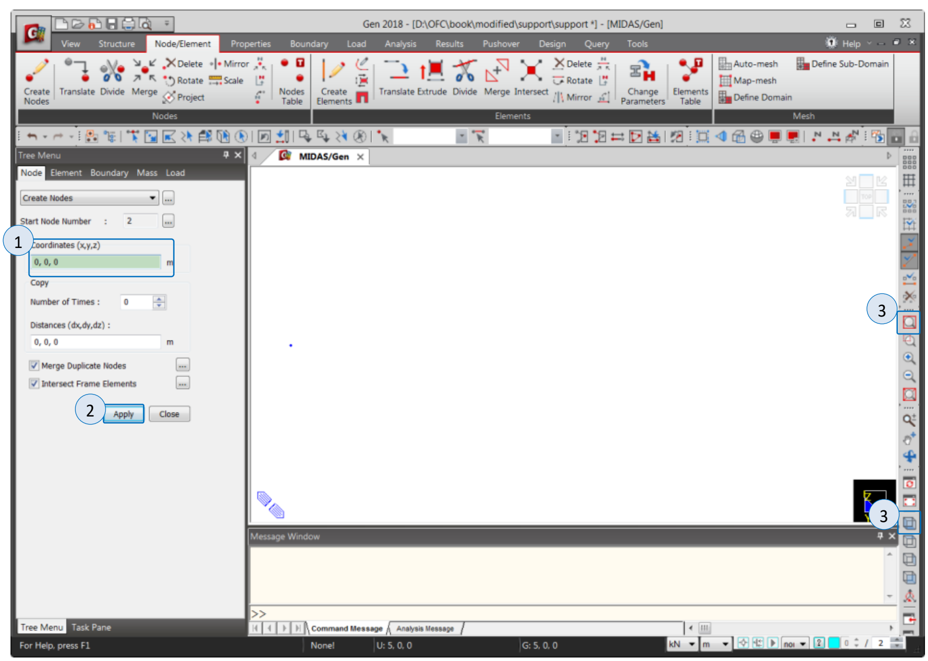


2.4 Nodes & Elements generation

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

Main Menu: ***Tools / Unit System >*** check on **m, kN**
Main Menu: ***Node/Element / Create Nodes***
1. Coordinates (x, y, z): **'0, 0, 0'**
2. **[Apply]**
3. **Auto Fitting, Front View** (Toggle on)

▶Fig. 8.9
Create a node



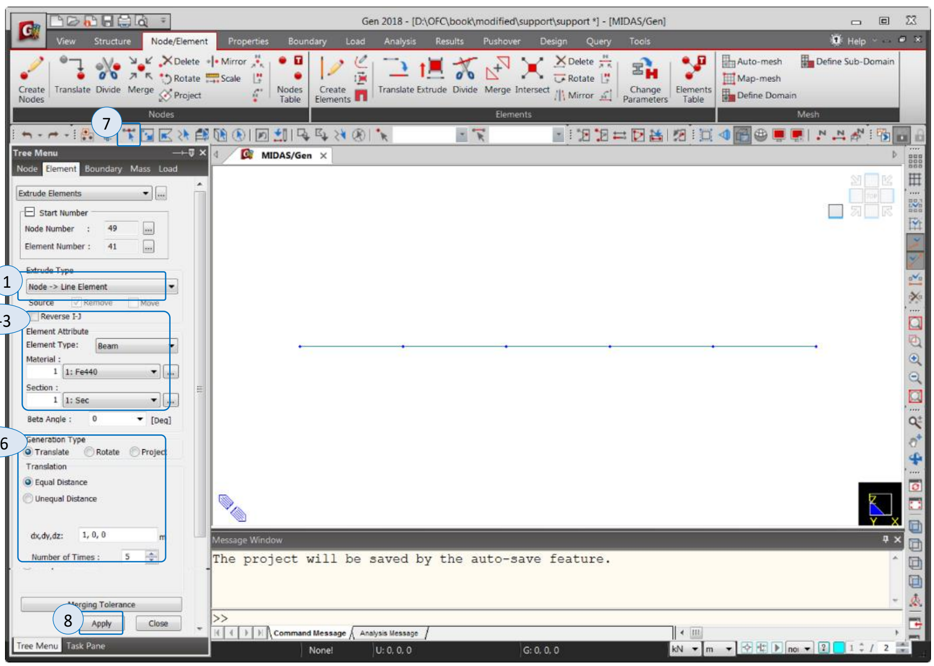
Tip

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

Create the left side beam of Model 1 using Extrude Elements, which extrudes Node to Line Element.

Main Menu: ***Node/Element > Extrude***
1. Extrude Type > **Node → Line Element**
2. Element Attribute > Element Type > **Beam**
3. Material > **1: Fe440**, Section > **1: Sec**, Beta Angle: **'0'**
4. Generation Type > **Translate**
5. Translation > **Equal Distance**
6. dx, dy, dz: **'1, 0, 0'**, Number of Times: **'5'**
7. **Select All**
8. **[Apply]**

▶Fig. 8.10
Create beam elements

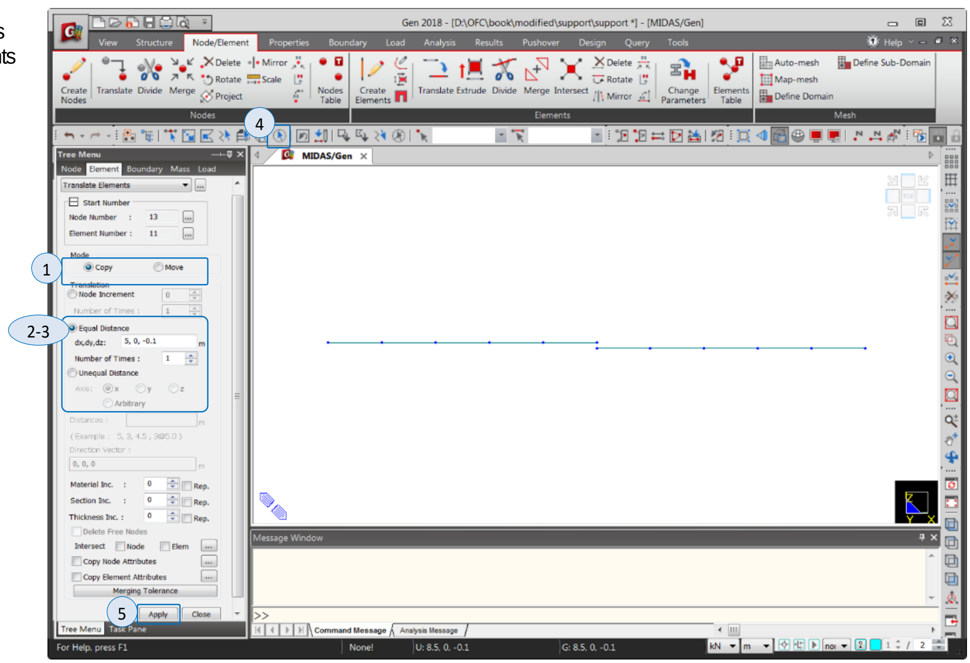


Copy the left side beam elements of Model 1 to create the right side beam elements.

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

1. Mode > **Copy**
2. Translation > Equal Distance > dx, dy, dz: **‘5, 0, -0.1’**
3. Number of Times: **‘1’**
4. **Select All**
5. **[Apply]**

▶Fig. 8.11
Copy beam elements
Create beam elements

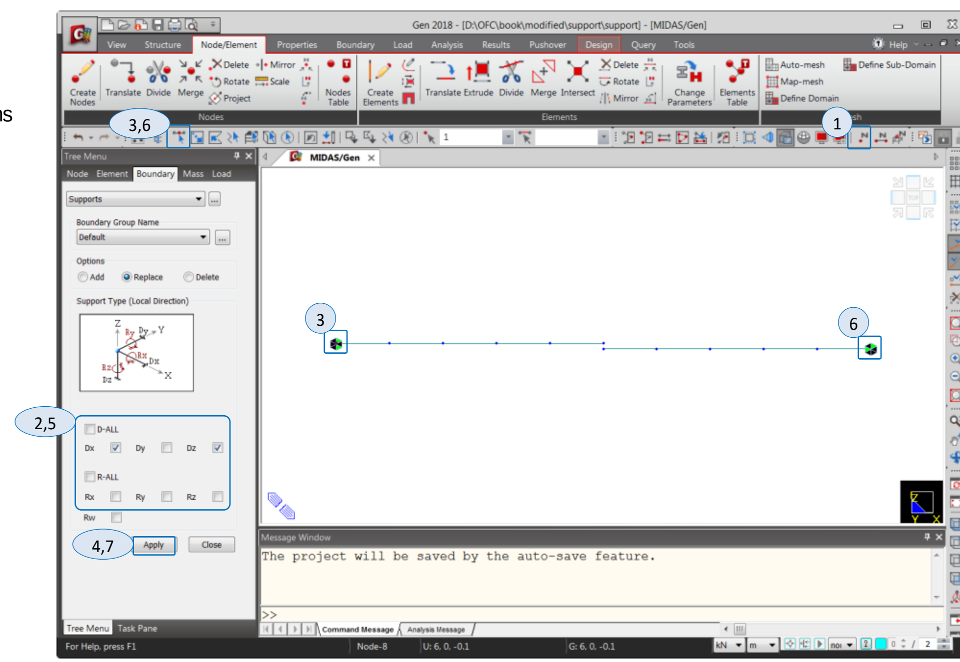


2.5 Define Boundary Conditions

Define boundary conditions at each end of the beams.

Main Menu: ***Boundary / Define Supports***

1. **Display Node Numbers** (Toggle on)
2. Support Type > **Dx, Dz** (Check on)
3. **Select Single** (on) mode: select **Node 1**
4. **[Apply]**
5. **Dz** (Check off), **Dx, Ry** (Check on)
6. **Select Single** (on) mode: select **Node 12**
7. **[Apply]**



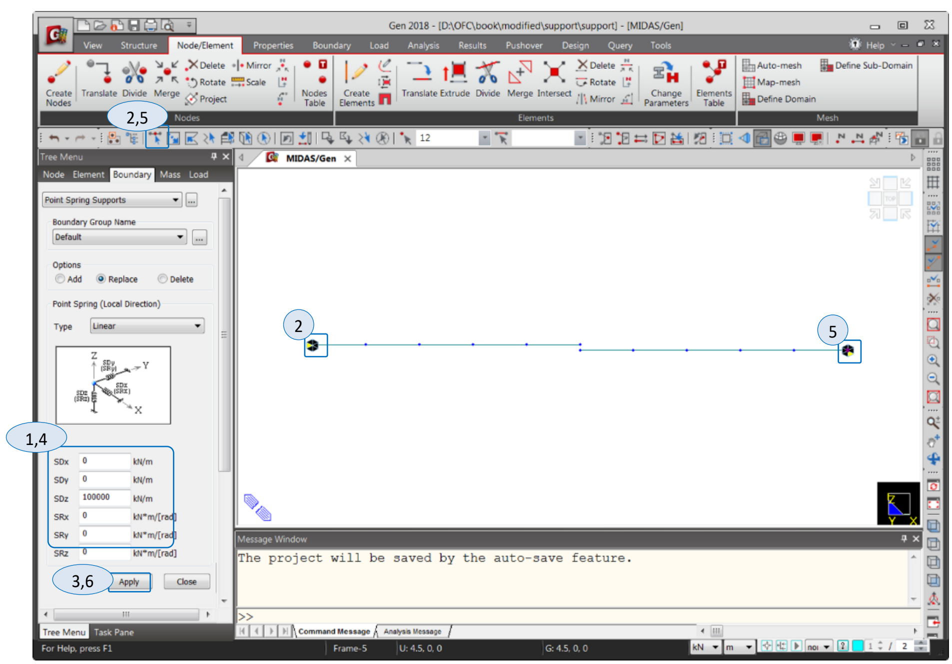
▶Fig. 8.12
Input support conditions

Use Point Spring Supports to input the elastic support condition for the beams.
Unlike Supports, which restrain the nodal degrees of freedom, Point Spring Supports are associated with boundary elements (springs) having elastic support stiffness for each degree of freedom. The springs permit deformation based on the stiffness, and the member forces in the elastic boundary elements are generated in the form of reactions.

Main Menu: ***Boundary / Point Spring Supports***

1. Point Spring (Local Direction) > SRy: **'1,000,000'**
2. **Select Single** (on) mode: select **Node 1**
3. **[Apply]**
4. SDz: **'100,000'**, others: **'0'**
5. **Select Single** (on) mode: select **Node 12**
6. **[Apply]**

▶Fig. 8.13
Input elastic support conditions

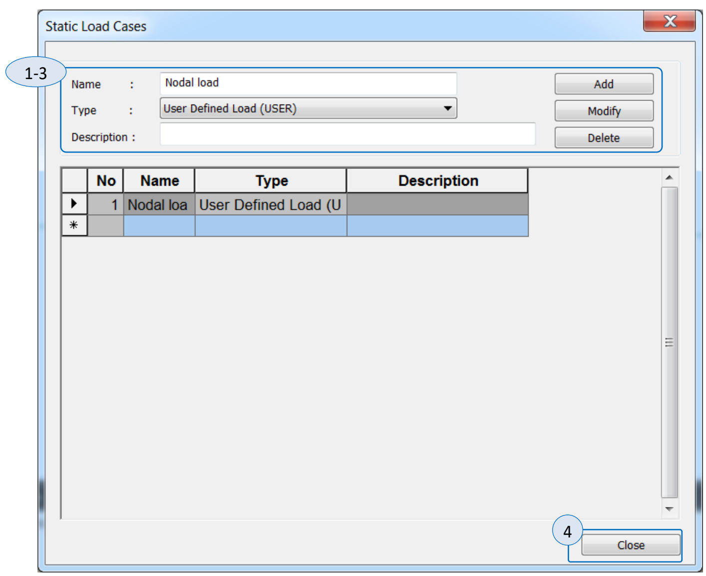


2.6 Define Loads

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

Main Menu: ***Load / Static Loads / Static Load Cases***

1. Name: **'Nodal load'**
2. Type > **User Defined Load (USER)**
3. **[Add]**
4. **[Close]**



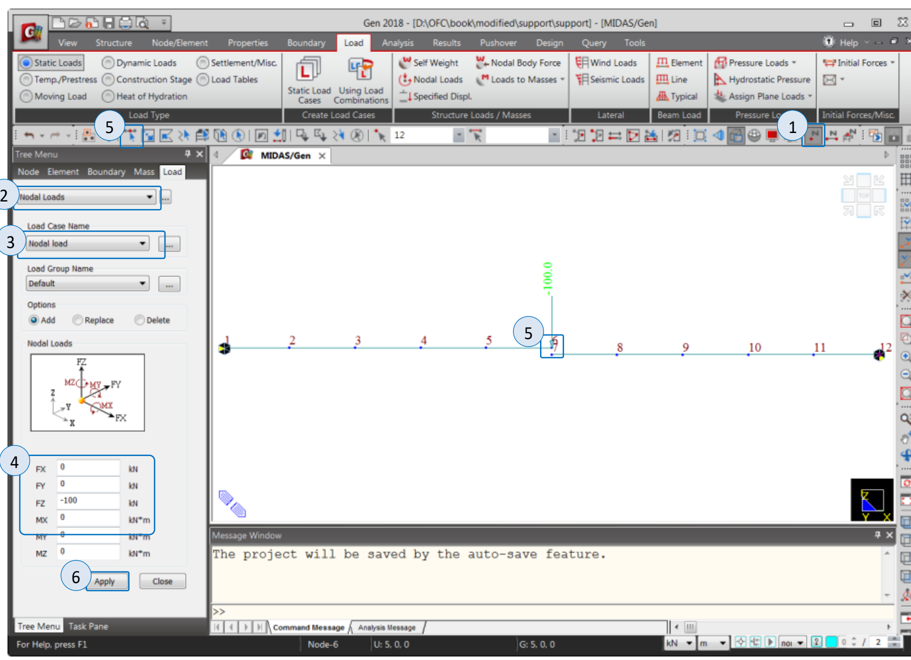
▶Fig. 8.14
Define load case

Input a concentrated load, 100 kN at Node 6.

Main Menu: ***Load / Static Loads / Nodal Loads***

1. **Display** (Toggle on), Load tab: **Load Value** (Check on), **[OK]**
2. Load Case Name > **'Nodal load'**
3. Options > **Add**
4. Nodal Loads > FZ: **'-100'**
5. **Select Single** (on) mode: select **Node 6**
6. **[Apply]**

▶Fig. 8.15
Input nodal load

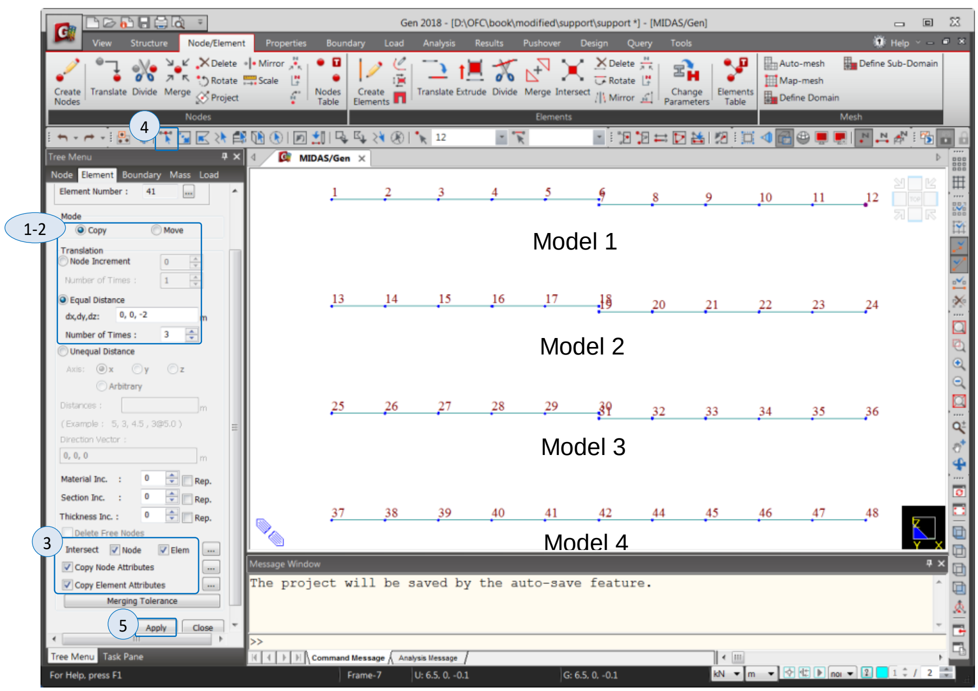


Copy Model 1 to create Models 2-4. Nodal loading and support conditions assigned to Model 1 will be copied simultaneously.

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

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

▶Fig. 8.16
Copy elements



Move the right side nodes of Model 4 in the Z direction by 0.1 m to align with the left side to make the entire span continuous. The overlapped nodes in the center will be merged using Merge Nodes (i.e., one node will be deleted).

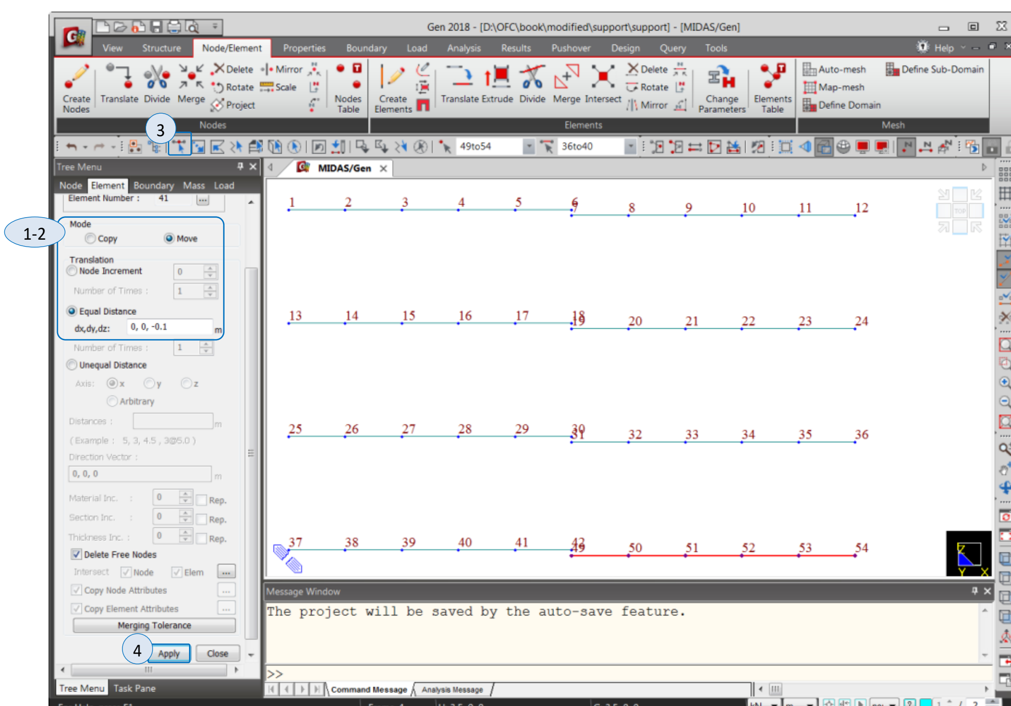
Main Menu: ***Node/Element / Translate Elements***

1. Mode > **Move**
2. Translation > Equal Distance > dx, dy, dz: **‘0, 0, -0.1’**
3. **Select Window** (Toggle on), select **Nodes 43-48**
4. **[Apply]**

Main Menu: ***Node/Element / Merge Nodes***

5. Merge > **All**, Tolerance: **‘0.001’**
   Remove Merge Nodes (Check on)
   [Apply]

▶Fig. 8.17 Revise Model 4 coordinates

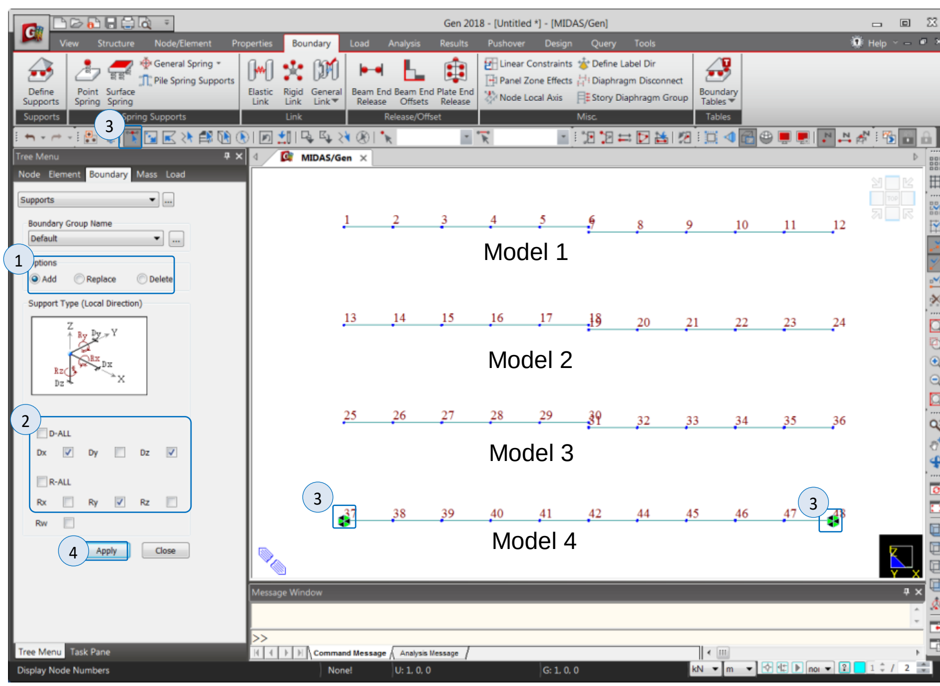


Revise the boundary conditions at both ends of Model 4 to a fixed condition.

Main Menu: ***Boundary / Define Supports***

1. Options > **Replace**
2. Support Type > **Dx, Dz, Ry** (Check on)
3. **Select Single** (on) mode: select **Nodes 37, 48**
4. **[Apply]**

▶Fig. 8.18
Revise boundary conditions

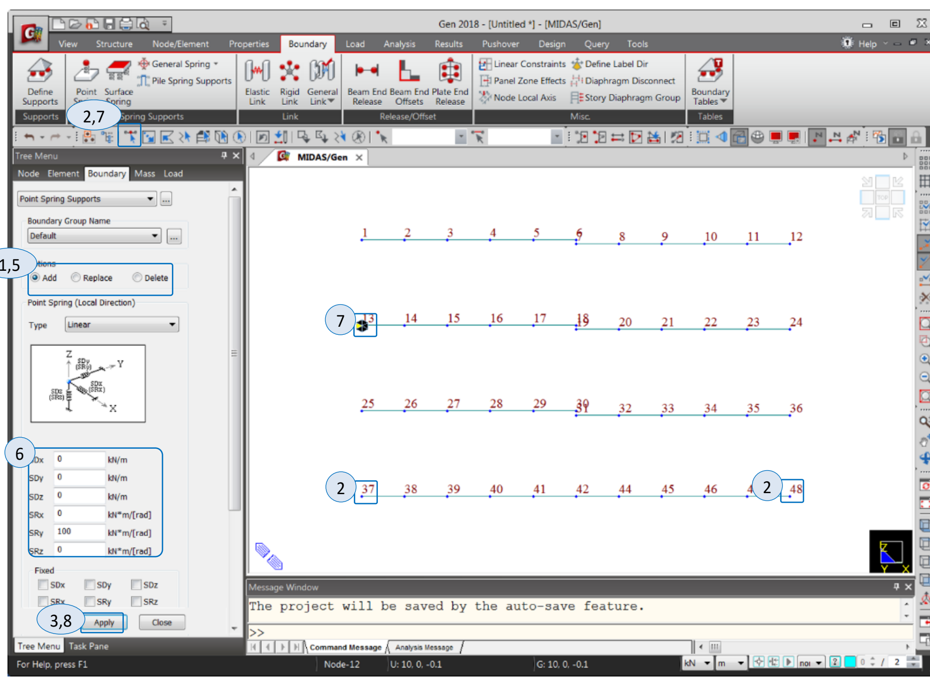


Delete the elastic support conditions in Model 4, and revise the stiffness of elastic support in Model 2.

Main Menu: ***Boundary / Point Spring Supports***

1. Options > **Delete**
2. **Select Single** (on) mode: select **Nodes 37, 48**
3. **[Apply]**
5. Options > **Replace**
6. SRy: **'100'**, others: **'0'**
7. **Select Single** (on) mode: select **Node 13**
8. **[Apply]**

▶Fig. 8.19
Revise spring conditions

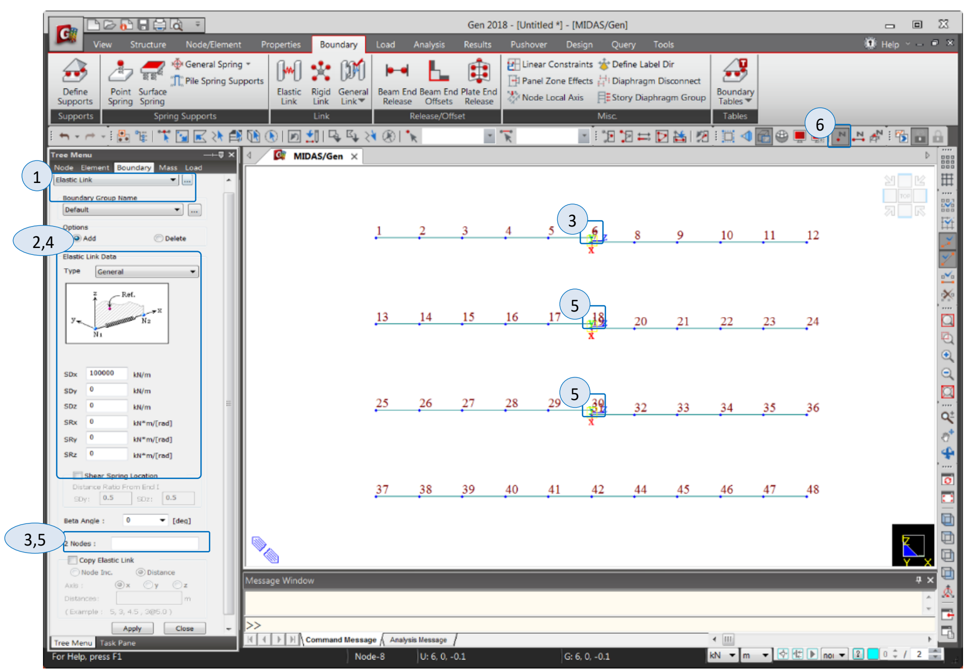


Input an elastic link at the gap between the left and right sides of the beams in each Model 1, 2 & 3. Elastic link elements are specified in Element Coordinate System.

Main Menu: ***Boundary / Elastic Link…***

1. Options > **Add**
2. Elastic Link Data > Type > **General**, SDx: **'100000'**, others: **'0'**
3. Click the 2 Nodes entry field (which turns to green), click **Nodes 6** and **7** on the model view sequentially.
4. Elastic Link Data > Type > **General**, SDx: **'100,000'**, others: **'0'**
5. Click the 2 Nodes entry field (which turns to green), click **Nodes 18** and **19** on the model view sequentially. Continue clicking Nodes **30** and **31** sequentially.
6. **Display Node Numbers** (Toggle off)

▶Fig. 8.20
Input elastic link elements

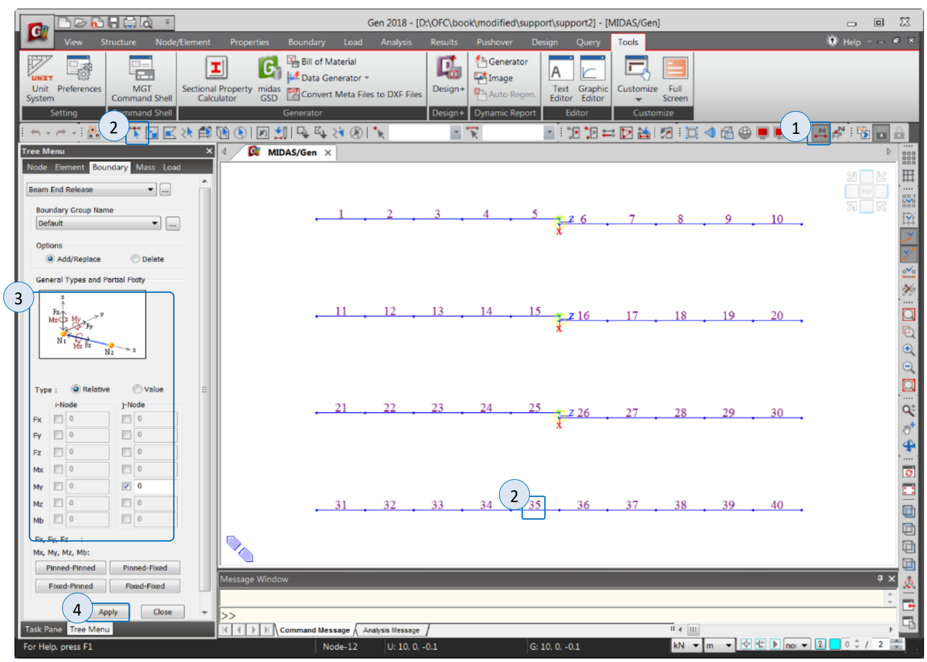


Input an internal hinge in the center of Model 4.

Main Menu: ***Boundary / Beam End Release***

1. **Display Element Numbers** (Toggle on)
2. **Select Single** (on) mode: select Element **35**
3. General Types and Partial Fixity > j-Node > **My** (Check on), others (Check off)
4. **[Apply]**

▶Fig. 8.21
Input internal hinge



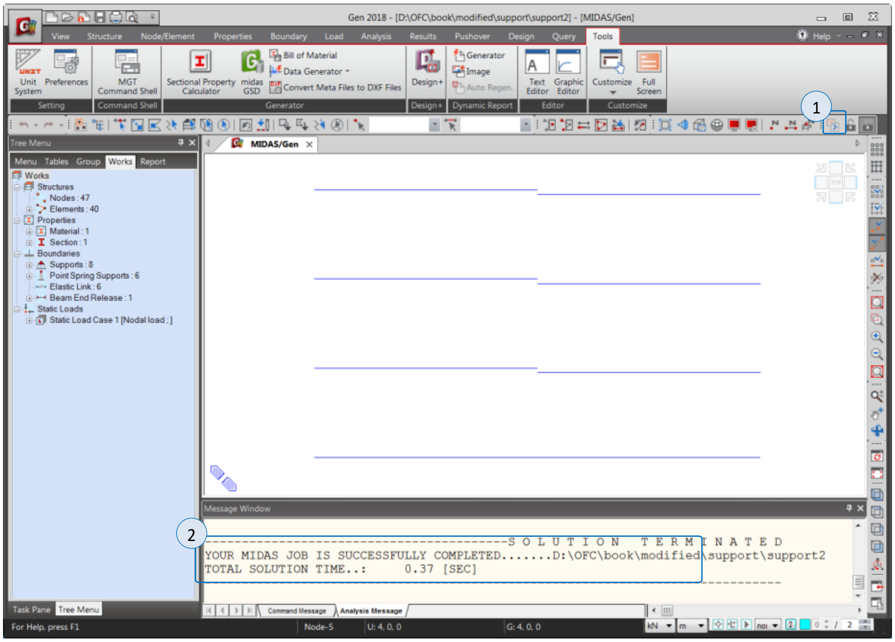
2.7 Perform Analysis

Analyze the 4 Models.

Main Menu: ***Analysis / Perform Analysis***

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

▶Fig. 8.22
Message for a successful run



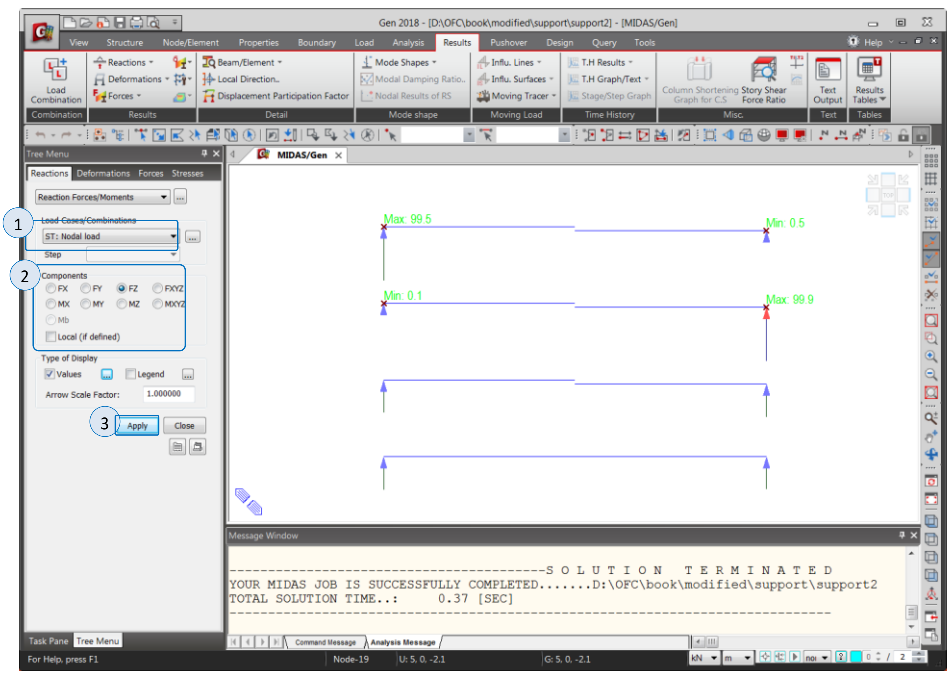
In case the above message fails to appear, then the previous input must be checked again.

2.8 Check Analysis Results

Model 1: Due to the weak stiffness of the elastic link element, the concentrated load applied at the end of the left beam at Node 6 is largely resisted by the left side beam.
Model 2: As the rotational stiffness of the left side beam support is small and the link stiffness is large, the concentrated load is transferred to the right side beam.
Model 3: Due to the large stiffness of supports and elastic link element, the results are very similar to those of Model 4, which is fixed at both ends with an internal hinge.

Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
1. Load Cases / Combinations > **ST: Nodal load**
2. Components > **FZ**, Type of Display > **Values** (Check on)
3. **[Apply]**

▶Fig. 8.23
Reactions due to nodal load

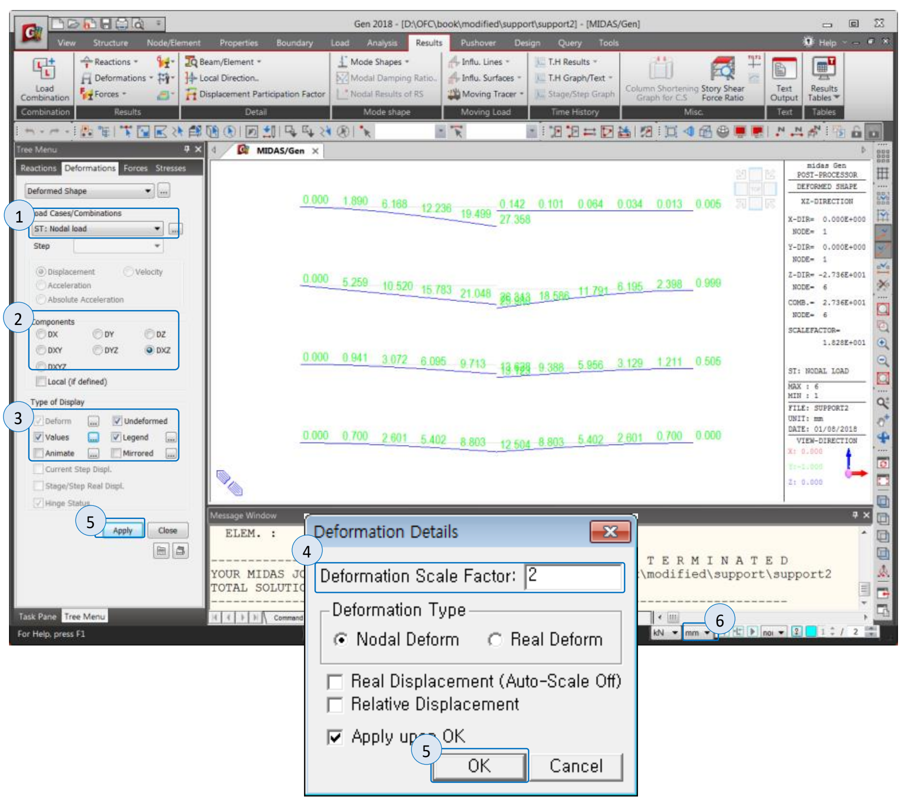


The displacement of the left side beam in Model 1 does not influence the right side beam. However, similar displacements in the left and right side beams are observed in Models 2 and 3.

Main Menu: ***Results / Deformations / Deformed Shape...***

1. Load Cases / Combinations > **ST: Nodal load**
2. Components > **DXZ**
3. Type of Display > **Undeformed, Legend, Values** (Check on)
4. Click **[...]** next to Deform and input Deformation Scale Factor: **'2'**
5. **Apply upon OK** (Check on), **[OK]**
6. Change the length unit to **mm** in Status Bar at the bottom of screen.

▶Fig. 8.24
Deformation due to nodal load

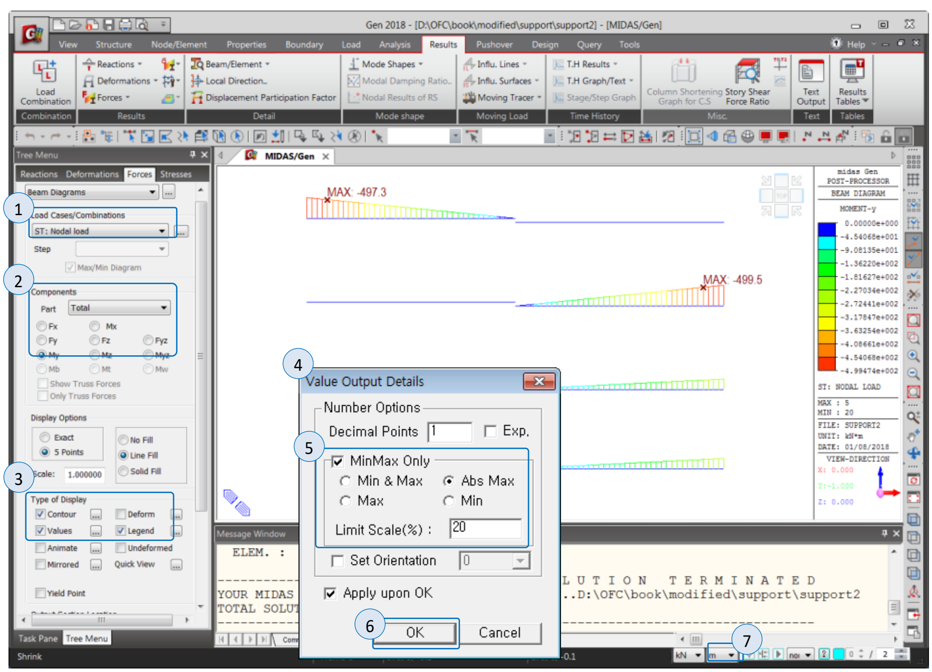


As the stiffness of an elastic support increases, the structural behavior begins to resemble a fixed support.   Hence a point spring support or an elastic link is used to control the stiffness of supports or elements.   They are also used to model the stiffness and eccentricity of a support.

Main Menu: ***Results / Forces / Beam Diagrams...***

1. Load Cases/Combinations > **ST: Nodal load**
2. Components > **My**
3. Type of Display > **Contour, Values, Legend** (Check on), **Deform** (Check off)
4. Click **[...]** next to Values
5. **Min & Max Only, Abs Max** (Check on), Limit Scale (%): **‘20’**
6. **[OK]**
7. Change the length unit to **m** in Status Bar at the bottom of screen.

▶Fig. 8.25
Bending moments due to nodal load

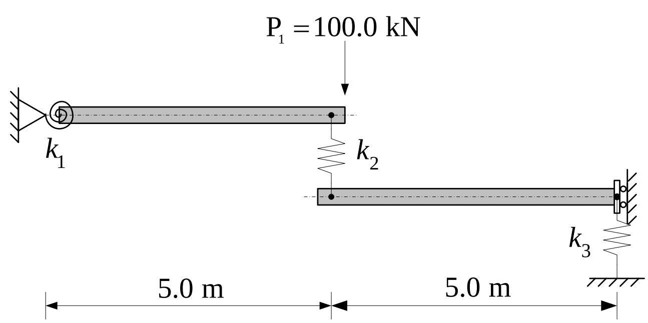


3. Manual Calculations

Fig. 8.26 shows a structure consisted of 2 beams connected by a spring. If the center spring is removed, the structure becomes 2 statically determinate structures, which suggests that the degree of indeterminacy of the spring connected structure is 1. If the left side beam deforms by δ_1, and the right side beam deforms by δ_2, the difference becomes the deformation of the spring, which renders a compatibility equation.

The left side beam is rotationally restrained by a spring at the support. The spring constant is k_1. The right side beam is supported by a vertical spring whose spring constant is k_3.

▶Fig. 8.26



If the left side beam resists a force, *Q* under the concentrated load of 100 kN, the force resisted by the spring and the right side beam is 100-*Q*. The following compatibility condition can be derived, which leads to the calculation of the force *Q.*

$$\delta_1 = \frac{Ql^3}{3EI} + \frac{Ql}{k_1}l, \qquad \delta_2 = \frac{(100-Q)l^3}{3EI} + \frac{100-Q}{k_3}$$

$$\delta_1 - \delta_2 = \frac{100-Q}{k_2}$$

$$\left[\frac{Ql^3}{3EI} + \frac{Ql^2}{k_1}\right] - \left[\frac{(100-Q)l^3}{3EI} + \frac{100-Q}{k_3}\right] = \frac{100-Q}{k_2}$$

$$Q = \frac{100\left(\frac{l^3}{3EI} + \frac{1}{k_2} + \frac{1}{k_3}\right)}{\frac{2l^3}{3EI} + \frac{l^2}{k_1} + \frac{1}{k_2} + \frac{1}{k_3}}$$

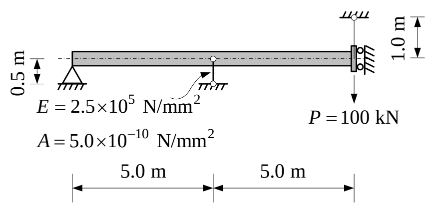
Calculation results (Units: kN, m)

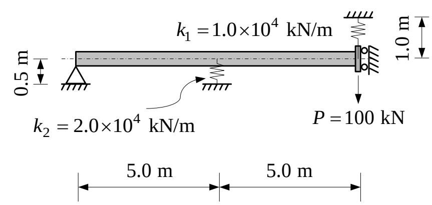
model	k_1	k_2	k_3	Q	100-Q	δ_1	δ_2
1	1.0×10^6	10	1.0×10^5	99.7325	0.26750	2.682×10^{-2}	0.7×10^{-4}
2	100	1.0×10^5	1.0×10^5	0.10535	99.89465	2.636×10^{-2}	0.3×10^{-4}
3	1.0×10^6	1.0×10^5	1.0×10^5	49.53079	50.46921	1.332×10^{-2}	-1.258×10^{-2}

where, $E=2.05\times10^8$ kN/m^2, $I=8.333\times10^{-4}$ m^4

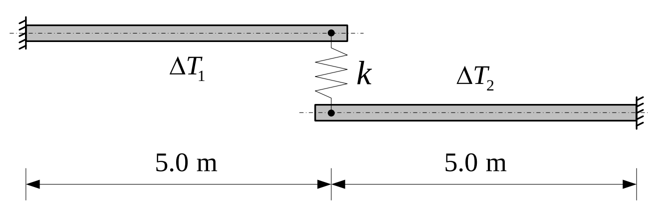
4. Exercise

1. Compare the reactions, displacements and member forces of the 2 structures of different boundary conditions. Material and section are identical to those in Tutorial.





2. Compare the behaviors of the structure subjected to temperature loads due to 3 different cases of spring types using tension-only spring and compression only spring. Material and section are identical to those in Tutorial.



Model	Spring stiffness (kN·m)		Load case 1		Load case 2	
	Tension	Compression	T1	T2	T1	T2
1	10,000	-	-58	+58	+58	-58
2	-	10,000	-58	+58	+58	-58
3	10,000	10,000	-58	+58	+58	-58

9. Specified Displacement Analysis

Contents

1. Introduction

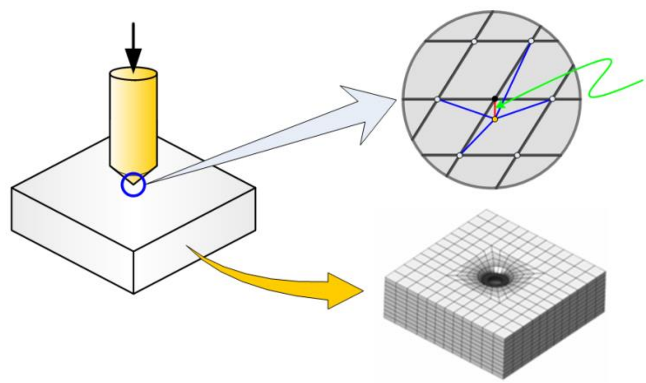
1.1 Concept for Specified Displacement Analysis

Specified Displacements are used to analyze a structure when displacements of a specific point in certain degrees of freedom are known.

Application areas include:

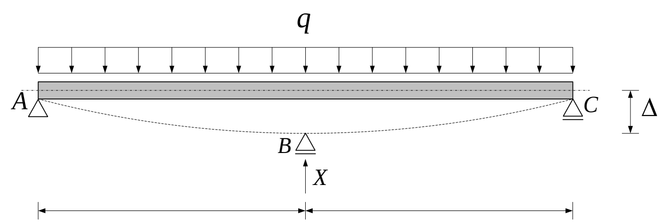
- Condition assessment of a structure, which has already deformed
- Analysis of a structure subjected to specific displacements as boundary conditions. (detail analysis of local parts based on global structural displacements)
- Analysis of ground settlement effects

▶Fig. 9.1
Specified displacement



A specified displacement is equivalent to applying a displacement load. Instead of applying a conventional load, a specific displacement is applied, which in turn leads to finding the corresponding conventional load. Fig. 9.2 shows a 2-span continuous beam with settlement (specified displacement) at the mid span support. The loading and settlement are downward (negative sign) and the reaction is upward (positive sign).

▶Fig. 9.2
Continuous beam with a specified displacement

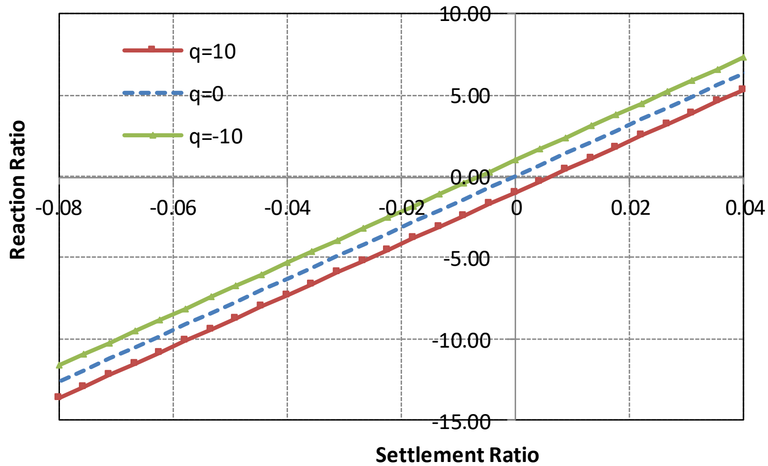


$$E = 3\times10^{7}\ \text{kN/m}^2,\quad I = 1\times10^{-3}\ \text{m}^4$$
$$q = 10, 0, -10\,\text{kN/m},\quad l = 4.5\ \text{m}$$

Removing the effect of a vertical load, q from the specified displacement, Δ leaves the role of the reaction (concentrated load), X as follows:

$$\Delta - \frac{5q(2l)^4}{384EI} = \frac{X(2l)^3}{48EI}$$

Given the specified displacement, Δ, the support reaction, X can be calculated. Fig. 9.3 is a graph showing the reaction ratio to the settlement ratio at Point B. The settlement ratio is the settlement divided by the span length. The reaction ratio represents the reaction value, X divided by the reaction corresponding to no settlement.



▶Fig. 9.3
Relationship between settlement ratio and reaction ratio

Fig. 9.3 is constructed for 3 different magnitudes of vertical load, q (10 kN/m, 0 kN/m, -10 kN/m). When the vertical load, q is downward (10 kN/m), the value X becomes smaller, which means that the reaction at Point B becomes smaller. If the vertical load is upward (-10 kN/m), the value X becomes larger because the settlement needs to take place while resisting the upward load.

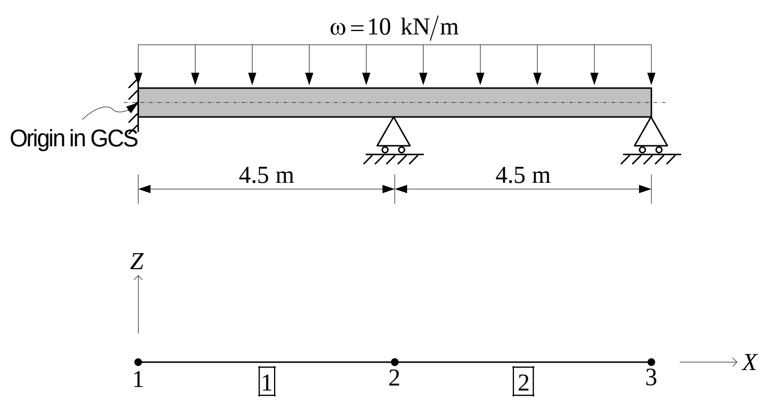
2. Tutorial

2.1 Model Overview

Examine the reactions, deformations and member forces of a 2-span continuous beam subjected to a uniformly distributed load on Elements 1 and 2 and settlements at Nodes 2 and 3.

Compare the patterns of support reactions and member forces between the cases of before and after the settlements.

▶Fig. 9.4
Analysis model



- ***Material***

  Modulus of elasticity (E): 30,000 N/mm^2

- ***Section***

  Area (A): 1m^2

  Moment of inertia (Iyy): 1.0×10^9 mm^4

- ***Load***

  1. Uniformly distributed load: 10 kN/m
  2. Support settlements, 40 mm at Node 2 and 25 mm at Node 3

2.2 Work Environment

Open a New File and save.

Main Menu: / ***New Project...***

Main Menu: / ***Save As...***

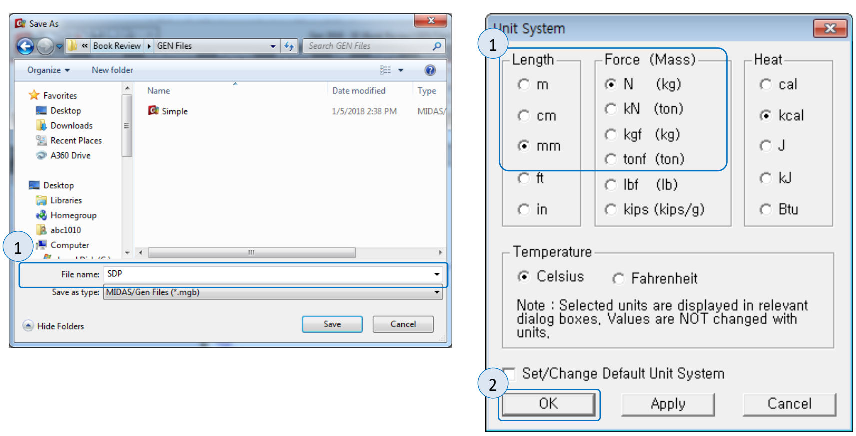
1. File name: **'SDP'** and [Save]

Set a unit system.

Main Menu: ***Tools / Unit System...***

1. Length > **mm**, Force (Mass) > **N (kg)**
2. **[OK]**

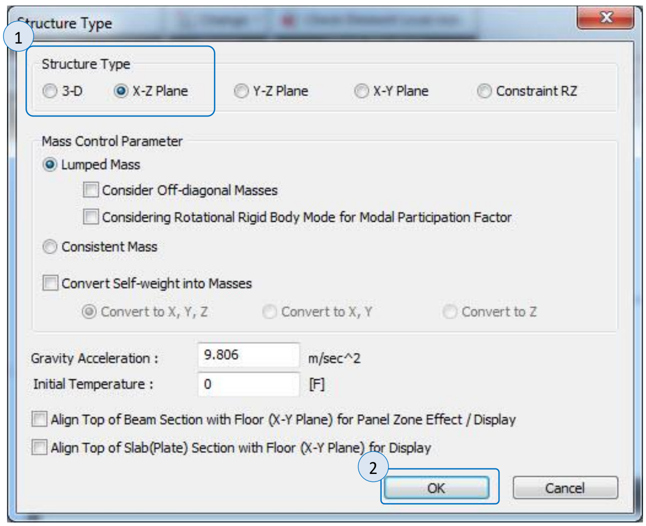
▶Fig. 9.5
Save file &
set unit system



The example model exists in the 2-D X-Z plane.

Main Menu: ***Structure / Structure Type***
1. Structure Type > **X-Z Plane**
2. **[OK]**

▶Fig. 9.6
Set a work plane



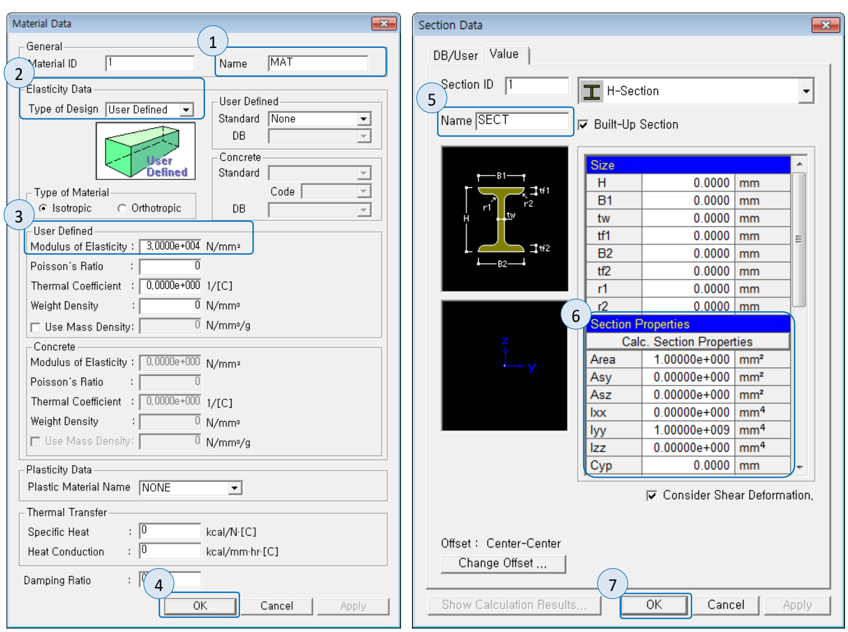
2.3 Material & Section Properties

Define material and section for the structural members, which are user-defined (User Defined Type & Value Type).

Main Menu: ***Properties / Material Properties***

1. **[Add...]**, Name: **'MAT'**
2. Elasticity data > Type of Design > **User Defined**
3. User Defined > Modulus of Elasticity: **'3e+4'N/mm^2**
4. **[OK]**
5. Properties dialog window: click **Section** tab, **[Add...]**, Value tab, Name: **'SECT'**
6. Section Properties > Area: **'1'mm^2**, Iyy: **'1e+9' mm^2**
7. **[OK]**, **[Close]**

▶Fig. 9.7
Define material & section

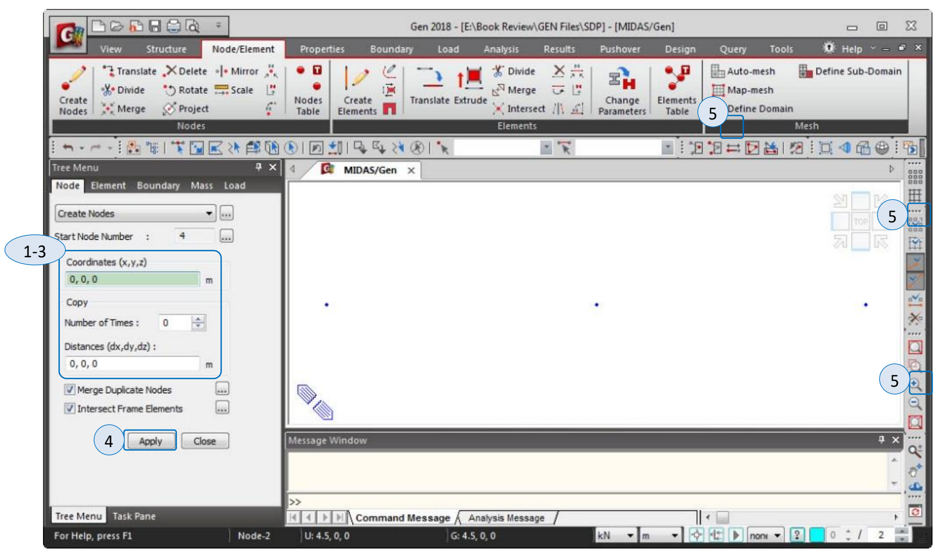


2.4 Nodes & Elements Generation

Change the units to m, kN.　Create Nodes.

Main Menu: ***Tools / Unit System*** > check on **m, kN, [OK]**
Main Menu: ***Node/Element / Create Nodes***
1. Coordinates (x, y, z): **‘0, 0, 0’**
2. Number of Times: **‘2’**
3. Distance (dx, dy, dz): **‘4.5, 0, 0’**
4. **[Apply]**
5. **Display Node Numbers, Auto Fitting, Front View** (Toggle on)

▶Fig. 9.8
Create nodes

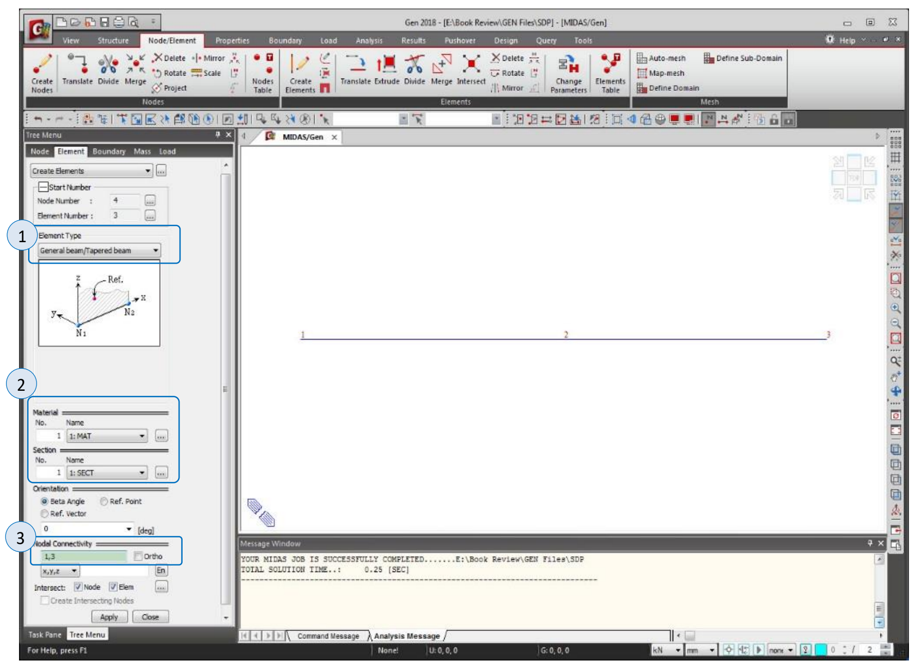


Create beam elements by connecting nodes.

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

1. Element Type > **General beam/Tapered beam**
2. Material > **1: MAT**, Section > **1: SECT**
3. Click the Nodal Connectivity entry field (which turns to green),
   Click Nodes **1** and **3** on the model view sequentially.

▶Fig. 9.9
Create beam elements

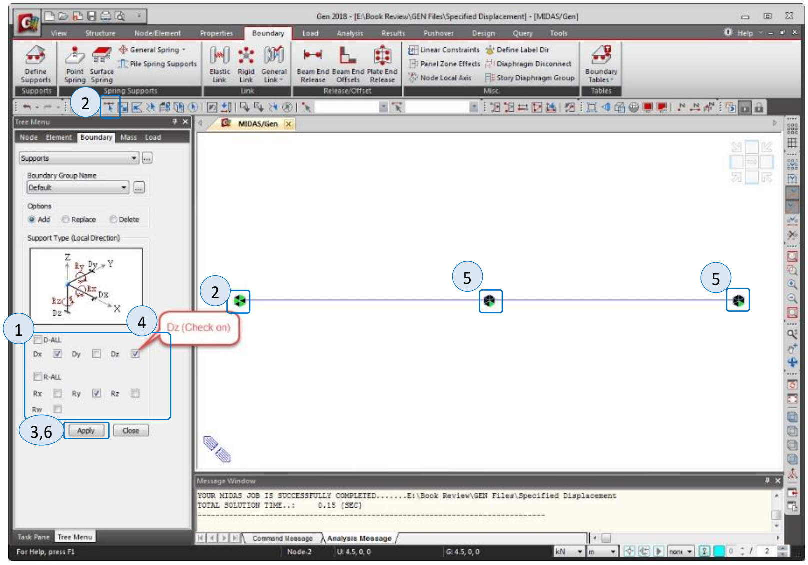


2.5 Define Boundary Conditions

Define boundary conditions at each support of the beam.
Node 1 is fixed (Dx, Dz, Ry). Nodes 2 and 3 are rollers (Dz). Since the Dz degree of freedom is automatically restrained when Nodes 2 and 3 are defined with support settlements by specified displacements, only Node 1 will be defined with the boundary condition.

Main Menu: ***Boundary / Define Supports***
1. Support Type (Local Direction) > **Dx, Dz, Ry** (Check on)
2. **Select Single** (on) mode: select **Node 1**
3. [Apply]
4. Support Type (Local Direction) > **Dz** (Check on)
5. **Select Single** (on) mode: select **Node 2, 3**
6. [Apply]

▶Fig. 9.10
Input support conditions



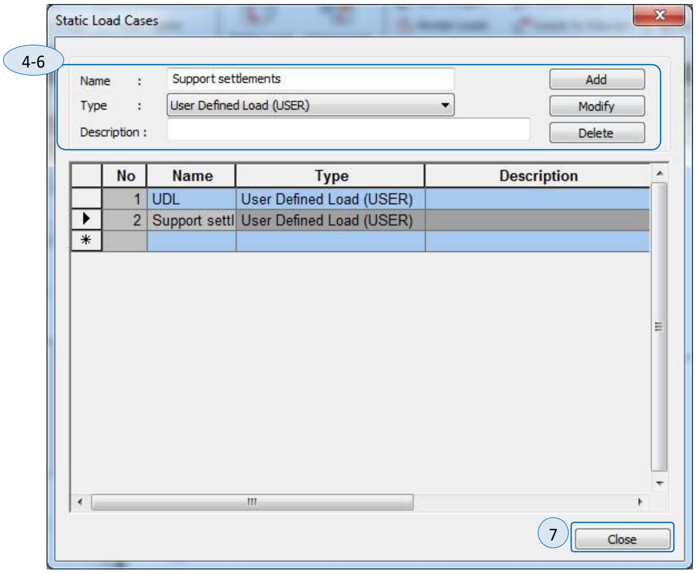
2.6 Define Loads

Define load cases to specify a uniformly distributed load and specified displacements caused by support settlements.

Main Menu: ***Load / Static Loads / Static Load Cases***

1. Name: **'UDL'**
2. Type > **User Defined Load (USER)**
3. **[Add]**
4. Name: **'Support settlements'**
5. Type > **User Defined Load (USER)**
6. **[Add]**
7. **[Close]**

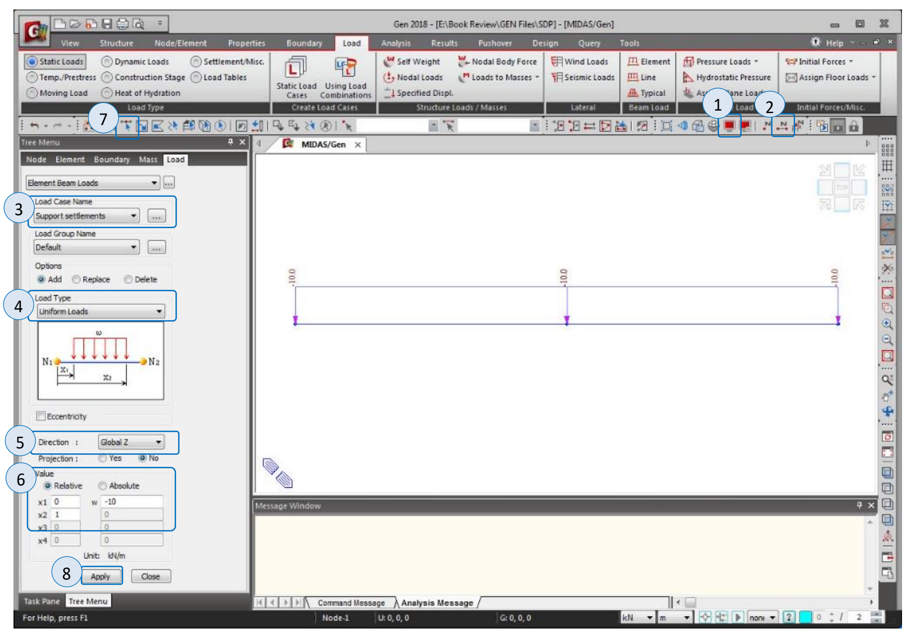
▶Fig. 9.11
Define load cases



Define a uniformly distributed load, 10 kN/m in the gravity direction.

Main Menu: ***Load / Static Loads / Element Beam Loads...***
1. **Display** (Toggle on), Load tab > **Load Value** (Check on), **[OK]**
2. **Display Element Numbers** (Toggle on)
3. Load Case Name > **UDL**
4. Load Type > **Uniform Loads**
5. Direction > **Global Z**
6. Value > **Relative**, x1: **'0'**, w: **'-10'**, x2: **'1'**
7. **Select Single** (on) mode > select **Elements 1 & 2**
8. **[Apply]**

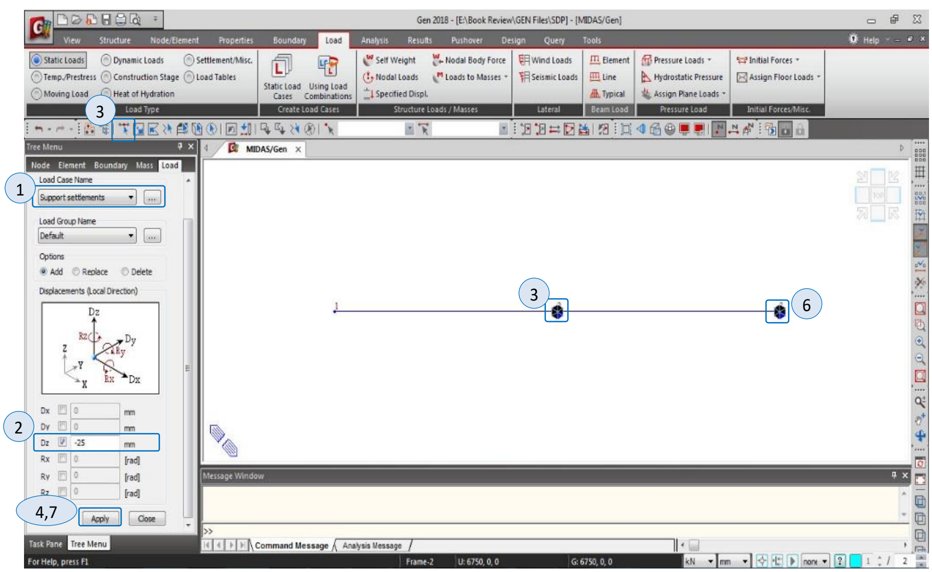
▶Fig. 9.12
Input UDL



Input support settlements using Specified Displacements of Supports.

From the Status Bar at the bottom of the screen, change the length unit to **mm.**
Main Menu: ***Load / Specified Displacements of Supports...***
1. Load Case Name > **Support settlements**
2. Displacements (Local Direction) > Dz: **'-40'**
3. **Select Single** (on) mode: select Node **2**
4. **[Apply]**
5. Displacements (Local Direction) > Dz: **'-25'**
6. Select **Node 3**
7. **[Apply]**

▶Fig. 9.13
Input specified displacement



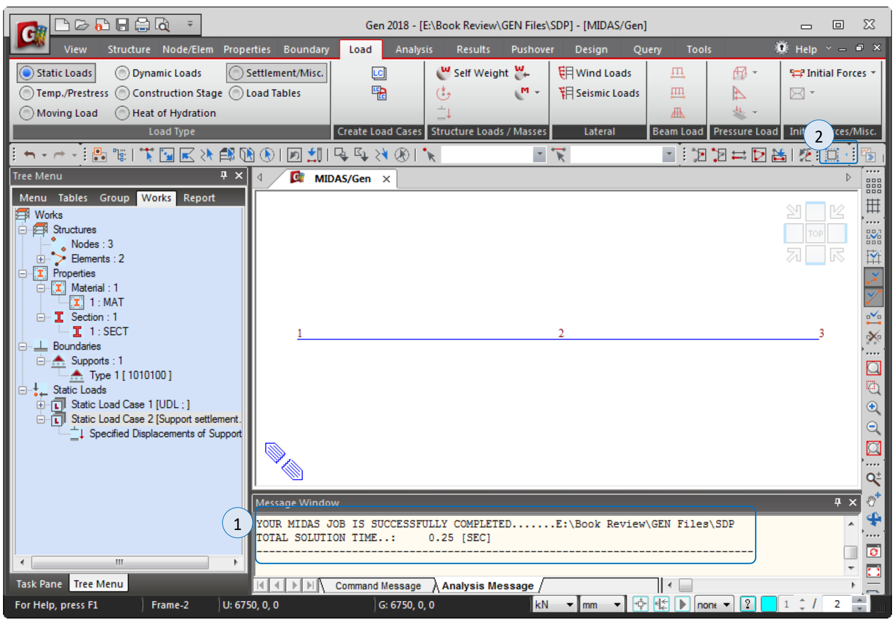
2.7 Perform Analysis

Analyze the model.

Main Menu: ***Analysis / Perform Analysis***

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

▶Fig. 9.14
Message for a
successful run



In case the above message fails to appear, then the previous input must be checked again.

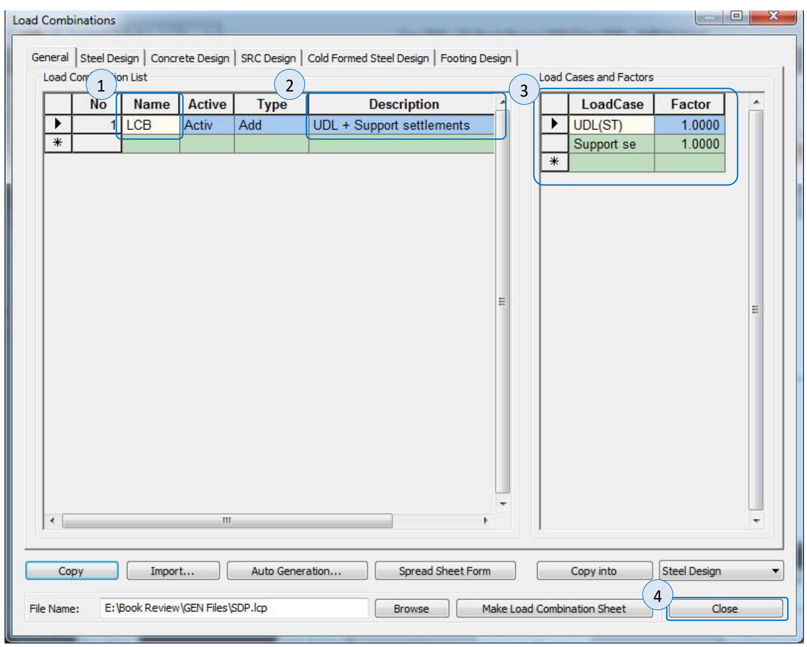
2.8 Check Analysis Results

Using the load cases defined, create a load combination.
Load Combination: 1.0 Uniform Load + 1.0 Settlement

Main Menu: ***Results / Load Combination***

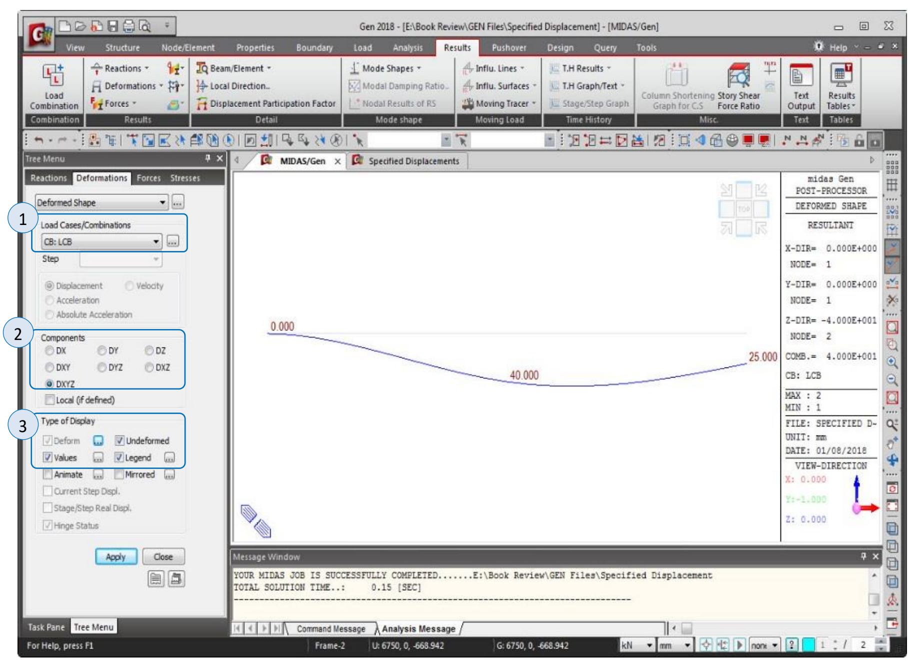
1. Load Combination List > Name: **'LCB"**
2. Description: **'UDL + Support settlements'**
3. Load Cases and Factors > LoadCase > select **UDL**, Factor: **'1.0'**
   In the next row, select **Support settlements**, Factor: **'1.0'**
4. **[Close]**

▶Fig. 9.15
Combine loads



Check the deformed shape.
Note the settlements specified at Nodes 2 and 3 (40 mm & 25 mm respectively).

Main Menu: ***Results / Deformations / Deformed Shape…***
1. Load Cases/Combinations > **CB: LCB**
2. Components > **DXZ**
3. Type of Display > **Undeformed, Legend, Values** (Check on)
4. Click **[...]** next to Deform
5. Deformation Scale Factor: **'2'**
6. Deformation Type > **Real Deform**
7. **Apply upon OK** (Check on), **[OK]**



▶Fig. 9.16
Deformation due to specified displacements

Check the deformation due to the UDL, Support settlements and LCB.

From the Status Bar at the bottom of the screen, change the length unit to **m**

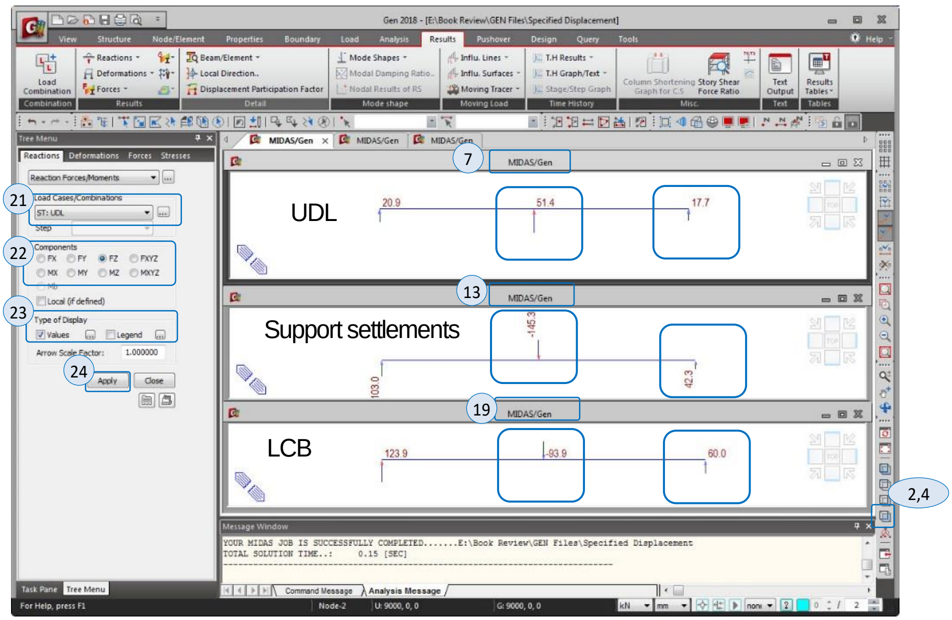
1. Main Menu: ***View / New Window***
2. **Front View** (Toggle on)
3. Main Menu: ***View / New Window***
4. **Front View** (Toggle on)
5. Main Menu: ***View / Tile Horizontally***
6. Adjust the arrangement of windows, Model View, Model View: 1, Model View: 2 for ease of comparison

7. Click **Model View** tab
8. Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
9. Load Cases/Combinations > **ST: UDL**
10. Components > **FZ**
11. Type of Display > **Values** (Check on), **Legend** (Check off)
12. **[Apply]**

13. Click **Model View: 1** tab
14. Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
15. Load Cases/Combinations > **ST: Support settlements**
16. Components > **FZ**
17. Type of Display > **Values** (Check on), **Legend** (Check off)
18. **[Apply]**

19. Click **Model View: 2** tab
20. Main Menu: ***Results / Reactions / Reaction Forces/Moments...***
21. Load Cases/Combinations > **CB: LCB**
22. Components > **FZ**
23. Type of Display > **Values** (Check on), **Legend** (Check off)
24. **[Apply]**

▶ Fig. 9.17
Compare reactions from each load case



It can be observed in the UDL case that the reaction at Node 2 is more than that from the tributary length due to the continuity. In the Support settlements case, a negative reaction appears at Node 2 where the settlement is the greatest. LCB case shows the summation of the load cases.

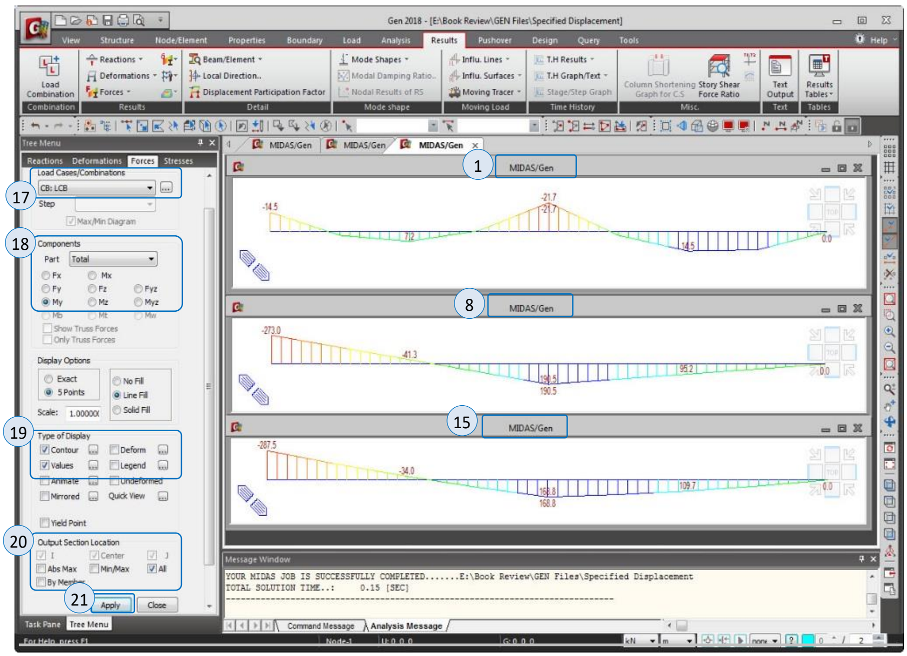
Compare the moments among the load cases and combination.

1. Click **Model View** tab
2. Main Menu: ***Results / Forces / Beam Diagrams...***
3. Load Cases/Combinations > **ST: UDL**
4. Components > **My**
5. Type of Display > **Contour, Values** (Check on), **Legend, Deform, Undeformed** (Check off)
6. Output Section Location > **All** (Check on)
7. **[Apply]**

8. Click **Model View: 1** tab
9. Main Menu: ***Results / Forces / Beam Diagrams...***
10. Load Cases/Combinations > **ST: Support settlements**
11. Components > **My**
12. Type of Display > **Contour, Values** (Check on), **Legend, Deform, Undeformed** (Check off)
13. Output Section Location > **All** (Check on)
14. **[Apply]**

15. Click **Model View: 2** tab
16. Main Menu: ***Results / Forces / Beam Diagrams...***
17. Load Cases/Combinations > **CB: LCB**
18. Components > **My**
19. Type of Display > **Contour, Values** (Check on), **Legend, Deform, Undeformed** (Check off)
20. Output Section Location > **All** (Check on)
21. **[Apply]**

▶ Fig. 9.18
Compare bending moments for each load case and combination



Specified displacement analysis is used to investigate the global behavior of a structure due to support settlements. Specified displacement analysis is also used to investigate local effects (stress) of a specific part of the total structure using the deformation results obtained from the global analysis.

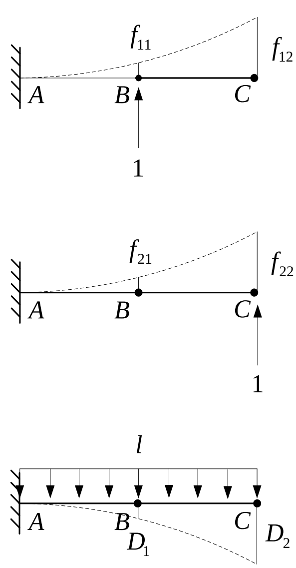
3. Manual Calculations

As the problem pertains to support settlements, the following expression can be derived through flexibility method or consistent deformation method.

$$\boldsymbol{FX} + \boldsymbol{D} = \Delta$$

$$\begin{bmatrix} f_{11} & f_{12} \\ f_{21} & f_{22} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \end{Bmatrix} + \begin{Bmatrix} D_1 \\ D_2 \end{Bmatrix} = \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix} \quad \text{or} \quad \begin{Bmatrix} f_{11}X_1 + f_{12}X_2 + D_1 = \Delta_1 \\ f_{21}X_1 + f_{22}X_2 + D_2 = \Delta_2 \end{Bmatrix}$$

f_{11} and f_{12} in Fig. 9.19 represent the displacements at Points B and C respectively when a unit load is applied at Point B. f_{21} and f_{22} represent the displacements at Points B and C respectively when a unit load is applied at Point C. D_1 and D_2 are the displacements at Points B and C respectively when the loading is applied to the structure while the supports are removed from Points B and C.



$$f_{11}=\frac{l^3}{3EI}=1.013\text{ mm},\quad f_{12}=f_{21}=\frac{5l^3}{3EI}=2.531\text{ mm},\quad f_{22}=\frac{8l^3}{3EI}=8.100\text{ mm}$$

$$D_1=-\frac{17l^4}{24EI}=\text{-}96.820\text{ mm},\quad D_2=-\frac{2l^4}{EI}=\text{-}273.375\text{ mm},$$

Using the above results, the following can be expressed for the uniformly distributed load and the settlements taking place simultaneously:

$$\begin{bmatrix}1.013 & 2.531\\ 2.531 & 8.100\end{bmatrix}\begin{Bmatrix}X_1\\ X_2\end{Bmatrix}+\begin{Bmatrix}-96.820\\ -273.375\end{Bmatrix}=\begin{Bmatrix}-40\\ -25\end{Bmatrix}$$

The first left term above signifies upward displacements at the supports when the unknown support reactions ($\boldsymbol{X}$) are acting. The second left term represents downward displacements ($\boldsymbol{D}$) at the supports caused by the actual load. So the left side of the above equation represents the sum of the displacements ($\boldsymbol{D}$) due to the load and the displacements ($\boldsymbol{FX}$) due to the reactions. If no settlements are present, then the right term must be zero. If settlements have occurred, then the right term must be the actual displacements at the supports ($\boldsymbol{\Delta}$).

1) If the load is applied, and no settlements exist,

$$\begin{bmatrix}1.013 & 2.531\\ 2.531 & 8.100\end{bmatrix}\begin{Bmatrix}X_1\\ X_2\end{Bmatrix}+\begin{Bmatrix}-96.820\\ -273.375\end{Bmatrix}=\begin{Bmatrix}0\\ 0\end{Bmatrix}$$

$$X_1=51.429\text{ kN}$$
$$X_2=17.679\text{ kN}$$

2) If only the settlements exist without any load,

$$\begin{bmatrix}1.013 & 2.531\\ 2.531 & 8.100\end{bmatrix}\begin{Bmatrix}X_1\\ X_2\end{Bmatrix}+\begin{Bmatrix}0\\ 0\end{Bmatrix}=\begin{Bmatrix}-40\\ -25\end{Bmatrix}$$

$$X_1=-145.326\text{ kN}$$
$$X_2=42.428\text{ kN}$$

3) If the loading is applied, and the settlements exist at the same time,

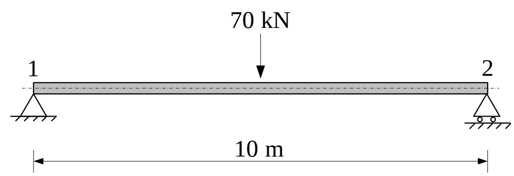
$$X_1 = 51.429 - 145.326 = -93.898 \text{ kN}$$
$$X_2 = 17.679 + 42.428 = 60.007 \text{ kN}$$

The above results are identical to those from the analysis results in midas Gen. The fourth decimal values have been rounded off.

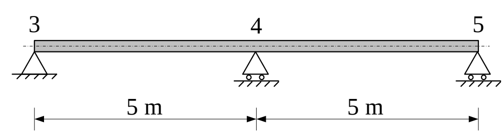
4. Exercise

1. Compare the bending moments of the 2 models below.

(a)



(b)



- ***Material***

  Steel: **IS(S)**, DB > **Fe 570** (E = 2.0500×10^8 kN/m^2)

- ***Section***

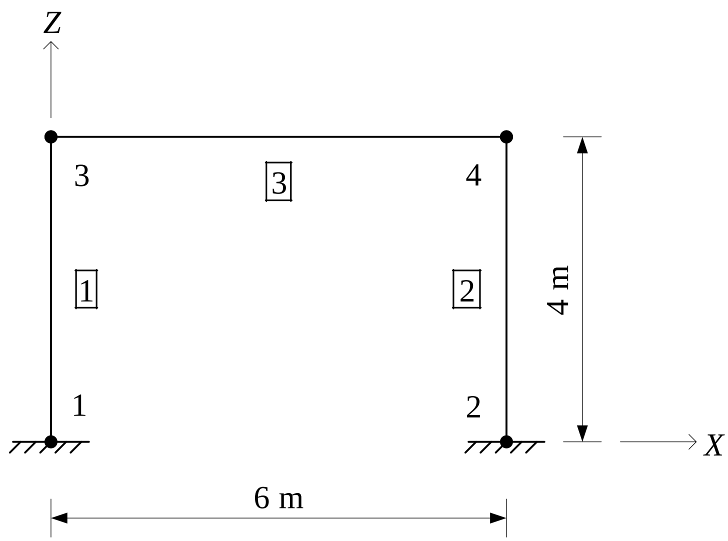
  B × H: 400 mm × 600 mm (Area=2.4 × 10^{-1} m^2, Iyy = 7.2 × 10^{-3} m^4)

- ***Load***

  Model 1: a concentrated load, 70 kN in the gravity direction at the mid span

  Model 2: a settlement, 10mm at Node 4

2. Find the shear force and bending moment diagrams of the portal frame below subjected to a settlement, 10mm at Node 2.



- ***Material***

  Steel: **IS(S)**, DB > **Fe 570** (E = 2.0500×10^8 N/mm^2)
- ***Section***

  I Shape: (Area=21454 mm^2, Iyy = 6.5362 × 10^8 mm^4)
- ***Load***

  Settlement at right column (Node 2): 10 mm

10. **Moving Load** Analysis

Contents

1. Introduction

1.1 Concept for Moving Load Analysis

Moving load analysis in midas Gen is used to reflect static (vehicle) moving loads as follows:

- Calculation of influence line and influence surface of displacements, member forces, reactions, etc. due to a unit moving load
- Calculation of max/min nodal displacements, member forces, reactions, etc. due to moving loads using the influence line and influence surface

The prime objective of moving load analysis is to investigate the maximum and minimum effects of moving live loads or vehicular loads, which are momentarily positioned at various locations along the entire moving path. Rather than analyzing the structure for various moving load positions repeatedly, influence line and influence surface analyses enable the engineer to identify the critical moving load positions, which cause the maximum/minimum effects (displacements, member forces, reactions, etc.).

In case of a vehicular load, the traffic lane or surface of the vehicle and the method of loading the vehicle are defined. Influence line (for 1D elements) or influence surface (for 2D elements) is then calculated by placing a unit moving load on the traffic lane or surface.

Influence line is obtained through static analysis results of each component (such as force reaction or displacement) at a specific point on a beam or truss caused by a unit load placed at any point along the structure. Similarly influence surface is obtained by placing a unit load at nodes of plate elements, the static analysis results of which are presented at the corresponding loaded nodes.

The process of moving load analysis using an influence line or influence surface is simply summarized as follows:

Step 1: Define a vehicle load, the method of loading the vehicle and traffic line or surface.
Step 2: Create a unit load case for which static analysis is performed for each resulting component at various loading positions, resulting in an influence line or surface.
Step 3: Obtain analysis results for the moving load based on the loading method using the influence line or surface.

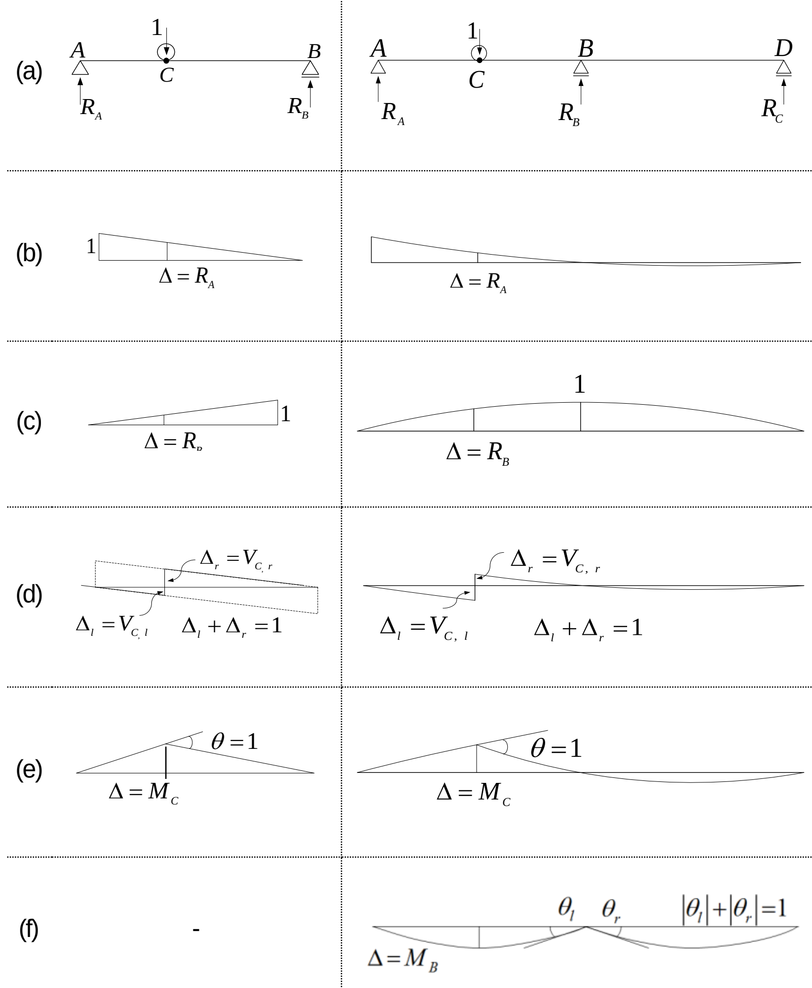
The moving load analysis through the above process results in maximum and minimum effects for each moving load condition, which can be combined with other loading cases. Because of load combinations with the maximum and minimum moving load results, the combined results will in turn retain maximum and minimum results. Analysis result component may be nodal displacements, support reactions and member forces of truss, beam and plate elements.

A unit load applied in the process of obtaining an influence line or influence surface used in moving load analysis acts in the -Z direction in Global Coordinate System. An unlimited number of moving load analysis cases can be used.

The principle of **Müller-Breslau** states that the influence line is the deformed shape of a structure from which reactions or section forces are removed and a unit displacement is applied to the corresponding positions. The reactions, section forces, etc. of the structure can be found from the shape of the influence line. Fig. 10.1 and Fig. 10.2 show the influence lines of a simply supported beam and a 2-span continuous beam.

▶Fig. 10.1 (Left)
Influence lines (IL) of a simply supported beam
(a) Moving load analysis model
(b) Reaction (R_A) IL
(c) Reaction (R_B) IL
(d) Shear force (V_C) IL
(e) Bending moment (M_C) IL

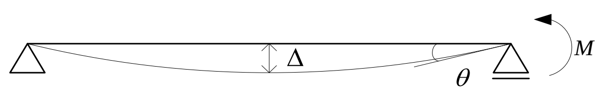
▶Fig. 10.2 (Right)
Influence lines (IL) of a 2-span continuous beam
(a) Moving load analysis model
(b) Reaction (R_A) IL
(c) Reaction (R_B) IL
(d) Shear force (V_C) IL
(e) Bending moment (M_C) IL
(f) Bending moment (M_B) IL



For the simply supported beam, influence lines of reactions at Points A and B, and shear force and moment at a specific location are indicated. For the continuous beam, influence lines of reactions at Points A and B, shear force and moment at Points B and C together with reaction at D are indicated. The influence lines of the simply supported beam are straight lines as the structure is statically determinate; whereas, curved influence lines prevail in the continuous beam due to its statically indeterminacy. Calculating the value represented by Δ at the acting point of loading in the influence line will become the reaction or section force when the loading acts on the corresponding position.

In order to investigate the method of computing section forces due to a moving load using an influence line, the center negative moment, M_B (Fig. 10.2(f)) of the continuous beam will be calculated. As it pertains to an influence line for moments, a hinge is inserted in Point B, thus releasing the moment, and a moment is applied to Point B such that a unit rotation angle takes place. The deformed shape at that instance becomes an influence line. The Δ value at the mid span of the span AB is the M_B value when the moving load is positioned at the point. By the symmetry about Point B, Δ can be obtained for a simply supported beam to which M is applied at one end. The rotation angle at Point B is as shown in Fig. 10.3. For a unit rotation angle to occur at Point B, the following magnitude of moment needs to be applied at each side of Point B.

▶Fig. 10.3
Rotation angle and moment at Point B



$$\theta = \frac{M}{3EI} \qquad \Delta = \frac{M\;^{2}}{16EI}$$

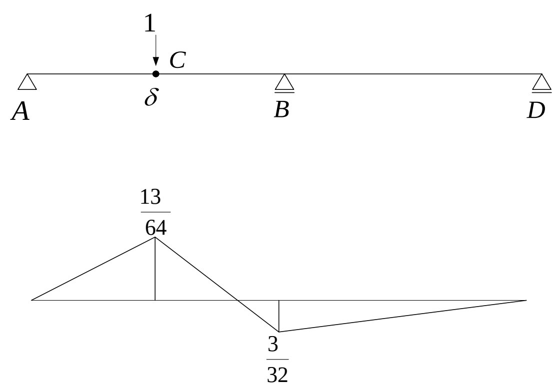
$$\frac{M\ell}{3EI} \times 2 = 1 \text{, or } M = \frac{3EI}{2\ell}$$

The deformed shape of the beam subjected to the above moment becomes the influence line. The deflection at the mid-span as shown in Fig. 10.3 can be calculated as follows:

▶Eq. 10.1

$$\Delta = M_B = \frac{M\ell^2}{16EI} = \frac{3EI}{2\ell} \times \frac{\ell^2}{16EI} = \frac{3\ell}{32}$$

In the 2-span continuous beam, the moment diagram due to a unit load applied at Point C (mid-span of Span AB) is as shown in Fig. 10.4 in which case the influence line value can be observed to become equal to the negative moment at Point B.



▶Fig. 10.4
Moment diagram of 2-span continuous beam

The concept of influence line can be explained by the principle of virtual work. Fig. 10.1(b) shows the influence line of the reaction R_A at Point A when the simply supported beam is subjected to a concentrated load. Applying the principle of virtual work, the work done in Fig. 10.1(b) by the force R_A and 1 in Fig. 10.1(a) is as follows:

▶Eq. 10.2

$$W_E = R_A \times 1 - 1 \times \Delta = 0$$

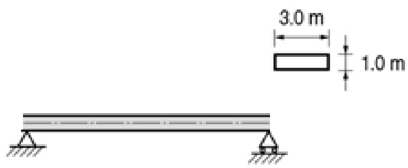
In this equation, the work done (W_E) by the external force must be zero. Since no deformation takes place in the beam, the work done (W_i) by the internal force is zero. Hence the reaction, R_A becomes equal to the deflection, Δ .

2. Tutorial

2.1 Model Overview

Calculate the maximum section forces by different member locations of a single span beam due to a moving load using an influence line. Check the position of the moving load, which causes the maximum section force.

▶Fig. 10.5
Analysis model



- ***Material***
  Concrete IS(RC) M30 (Modulus of elasticity, E = 2.7386×10^7 kN/m^2)
- ***Section***
  Shape: Solid Rectangle
  Dimension: B×H = 3.0 m×1.0 m
- ***Load***
  Concentrated Load: 10kN

2.2 Work Environment

Open a New File and save.

Main Menu: *__/ New Project...__*

Main Menu: *__/ Save...__*

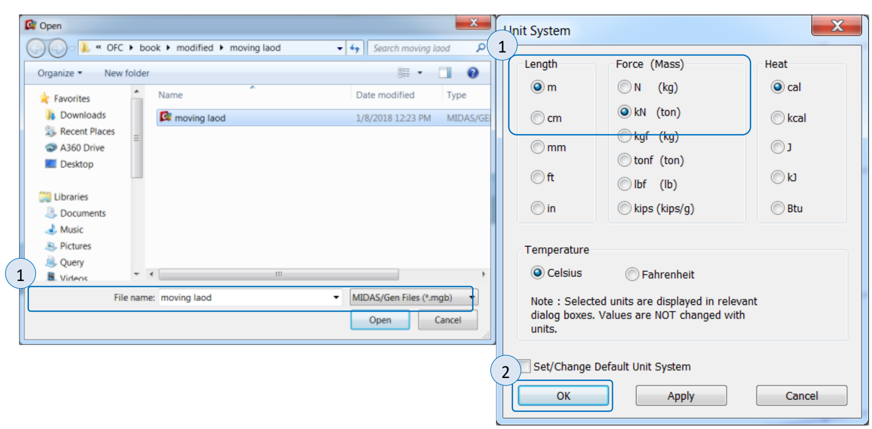
1. File name: **'Moving Load'** and [Save]

Set a unit system to kN & m.

Main Menu: ***Tools / Unit System***

1. Length > **m**, Force (Mass) > k**N(ton)**
2. **[OK]**

▶Fig. 10.6
Save file &
set unit system

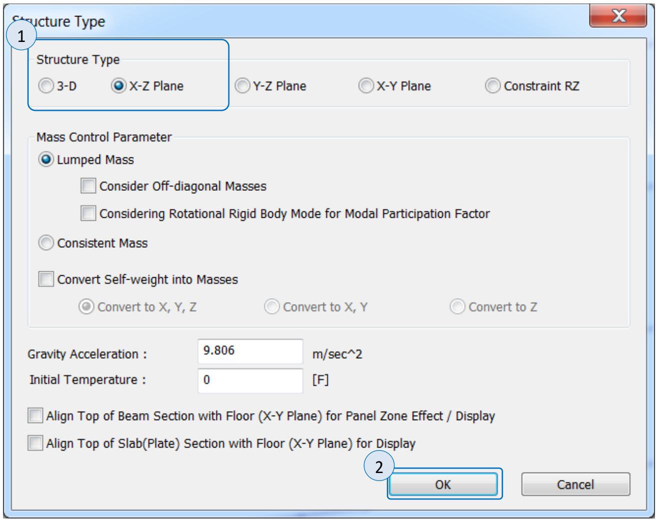


The example model exists in the 2-D X-Z plane. Set Structure Type to X-Z Plane.

Main Menu: ***Structure / Structure Type***

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

▶Fig. 10.7
Set a work plane

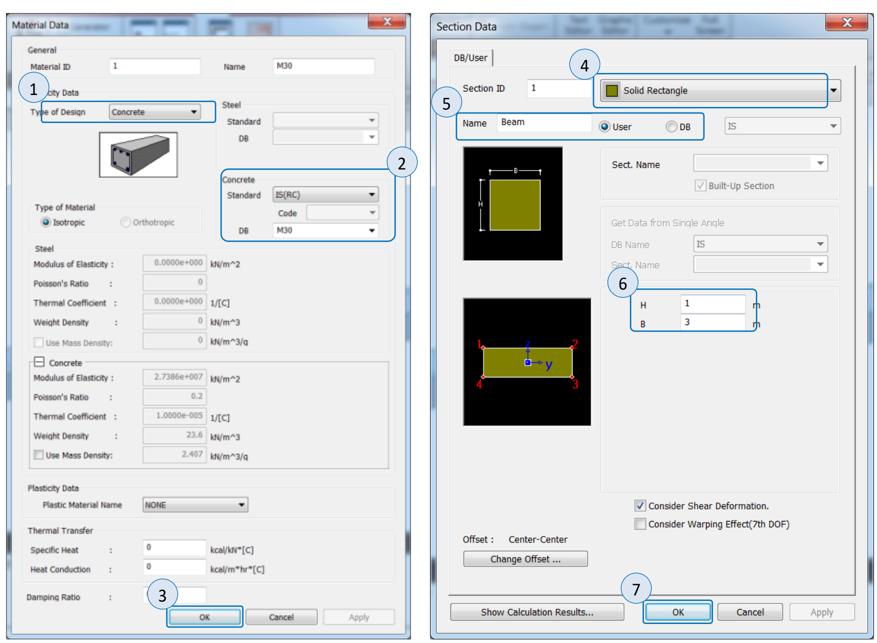


2.3 Material & Section Properties

Define material and section for the structural members.

Main Menu: ***Properties > Material Properties***
1. **[Add...]**, Type of Design > **Concrete**
2. Concrete > Standard > **IS(RC)**, DB > **M30**
3. **[OK]**
4. Properties dialog window: click **Section** tab, **[Add…]**, **DB/User** tab > **Solid Rectangle**
5. Name: **'Beam'**, Select **User**
6. H: **'1'**, B: **'3'**
7. **[OK]**, **[Close]**

▶Fig. 10.8
Define material
Define section



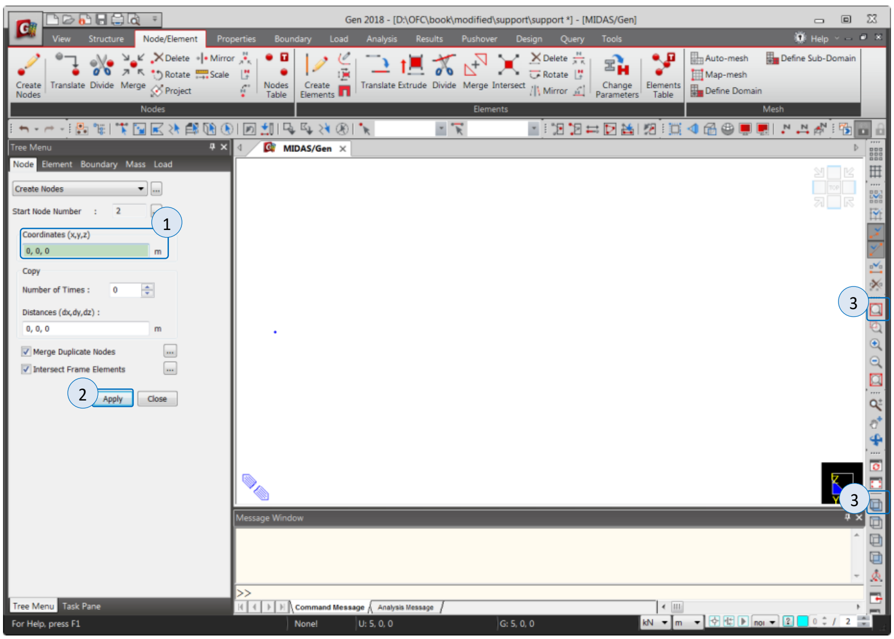
2.4 Nodes & Elements Generation

Create nodes at the start, end, locations of loads and boundaries and the locations for checking the results.

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

1. Coordinates (x, y, z): **‘0, 0, 0’**
2. **[Apply]**
3. **Auto Fitting, Front View** (Toggle on)

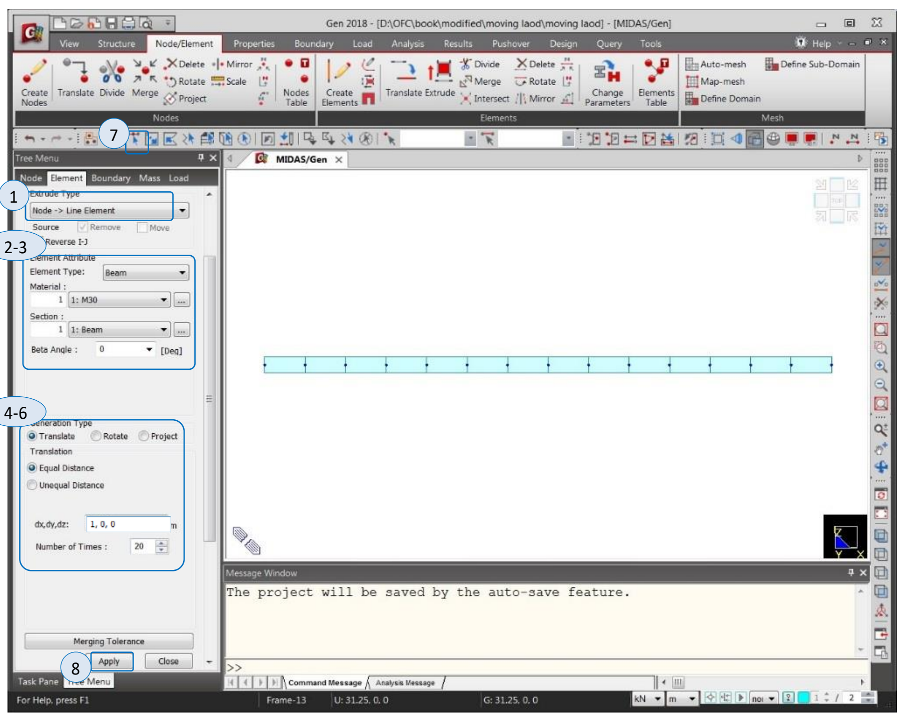
▶Fig. 10.9
Create a node



Use Extrude Elements to extend a node to line elements thus creating a continuous beam.

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

1. Select Extrude Type > **Node → Line Element**
2. Select Element Attribute > Element Type > **Beam**
3. Material > **1: M30**, Section > **1: Beam**, Beta Angle: **‘0’**
4. Generation Type > **Translate**
5. Translation > **Equal Distance**
6. dx, dy, dz: **‘1, 0, 0’**, Number of Times: **‘20’**
7. Click **Select All**
8. **[Apply]**



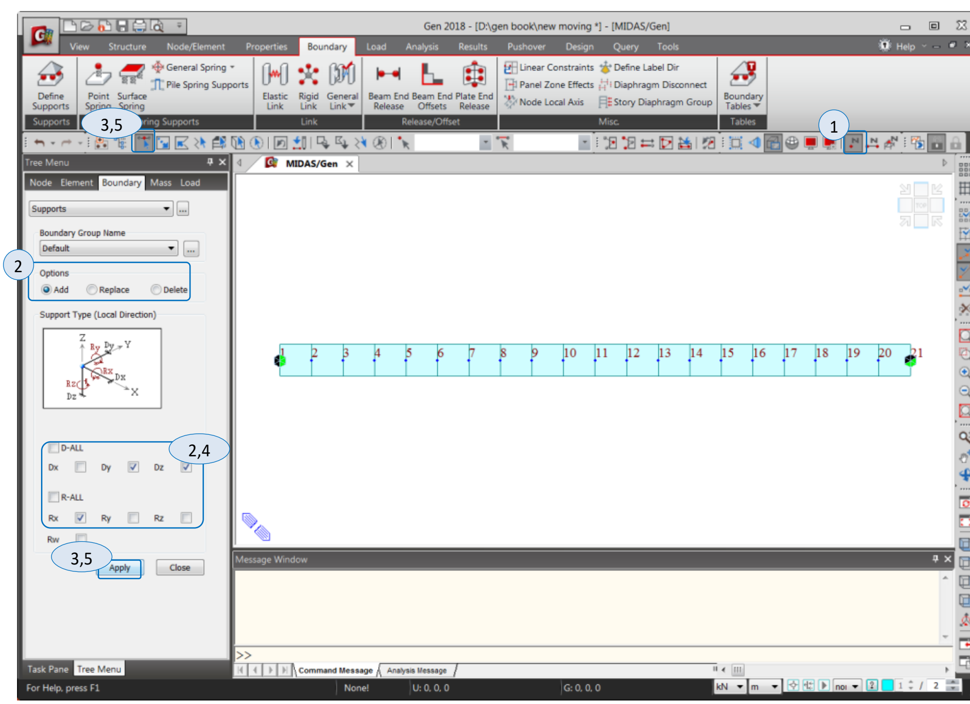
▶Fig. 10.10
Create beam elements

2.5 Define Boundary Conditions

Create supports for the continuous beam.
A pin condition restrains Dx and Dz degrees of freedom, and a roller condition restrains only Dz degree of freedom.

Main Menu: ***Boundary / Define Supports***
1. **Display Node Numbers** (Toggle on)
2. Options > **Add**, Support Type > **D-ALL** (Check on)
3. **Select Single** (on) mode: select **Node 1**, then **[Apply]**
4. Support Type > **Dx** (Check off), **Rx** (Check on)
5. **Select Single** (on) mode: select **Nodes 21**, then **[Apply]**

▶Fig. 10.11
Define support conditions



2.6 Define Loads

In order to calculate the member forces due to a moving load, an influence line must be calculated in advance. Define a traffic lane before calculating the influence line. As the beam is simply supported, calculate the impact factors corresponding to span length.

Main Menu: ***Load / Moving Load / Moving Load Code***
1. Moving Load Code > **India**

Main Menu: ***Load / Moving Load / Traffic Line Lanes***
1. Click **[Add]**
2. Lane Name: **'Lane1'**
3. Eccentricity: **'0'**, Wheel Spacing: **'0'**
4. Impact Factor: **'0'**
5. Selection by > **2 Points** > Click Nodes **1, 21**
6. **[OK]**
7. After checking Lane1 in the Traffic Line Lanes list, **[Close]**

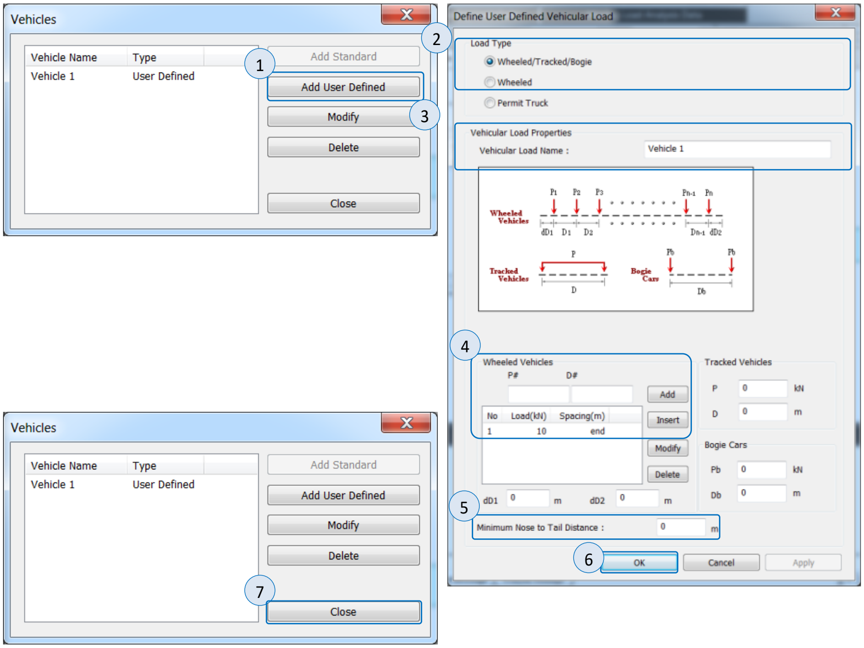
▶Fig. 10.12
Define traffic lane

The moving load (User Defined Vehicle Class 70R - Boogie) is defined using the the program.

Main Menu: ***Load / Moving Load / Vehicles***

1. Click **[Add User Defined]**
2. Load Type > **Wheeled/Tracked/Bogie**
3. Vehicular Load Properties > Vehicular Load Name: **'Vehicle 1**,
4. Wheeled vehicles > P# : **10kN** , D# : **0m,** Click > **Add**
5. Minimum Nose to tail distance : 0m
6. **[OK]**
7. Check to see Vehicle 1' in the Vehicles list, then **[Close]**

▶Fig. 10.13
Define moving load



Define vehicle load group used for moving load analysis.

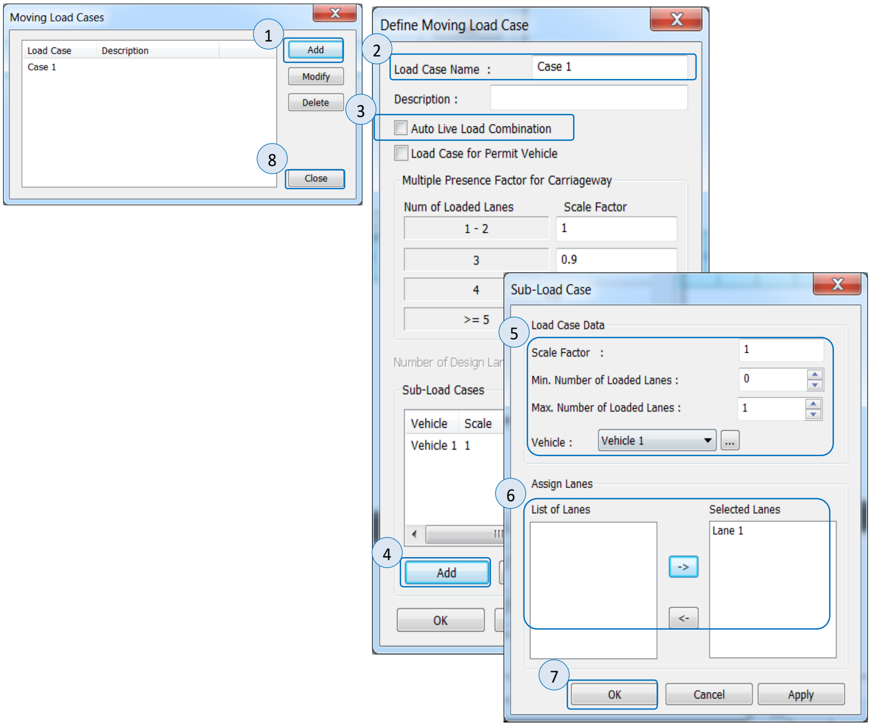
Main Menu: ***Load / Moving Load / Moving Load Cases***
1. **[Add]**
2. Vehicle Class Name: **'Case 1**
3. Check off > **Auto Live Load Combination**
4. Click on > **Add**
5. Input the data,

Min Number of loaded lanes > **0**

Max Number of loaded lanes > **1**

Select the vehicle > **Vehicle 1**

6. From the List of Lanes list, click **Lane 1** followed by clicking **[→]**
7. Check to see **Case 1** in the Selected Load list, then **[OK]**
8. **[Close]**



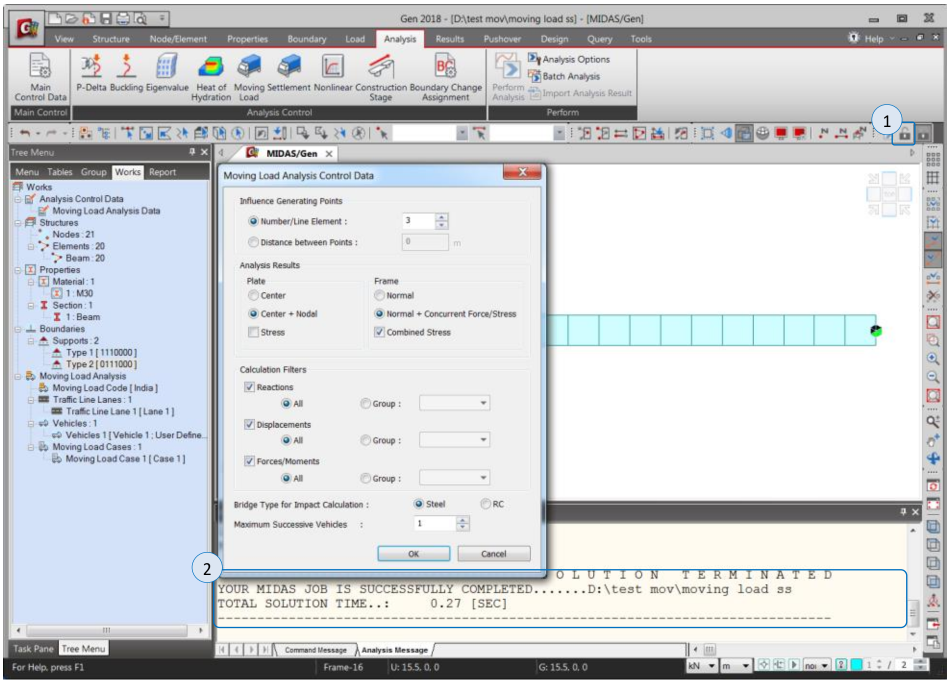
▶Fig. 10.14
Define moving load cases

2.7 Perform Analysis

Analyze the model.

Go to ***Analysis / Moving Load***
Input the maximum successive vehicles : **1**
Main Menu: ***Analysis / Perform Analysis***
1. **Display Node Numbers** (Toggle off)
2. Check for successful completion in Message Window.

▶Fig. 10.15
Message for a successful run



In case the above message fails to appear, the previous input must be checked again.

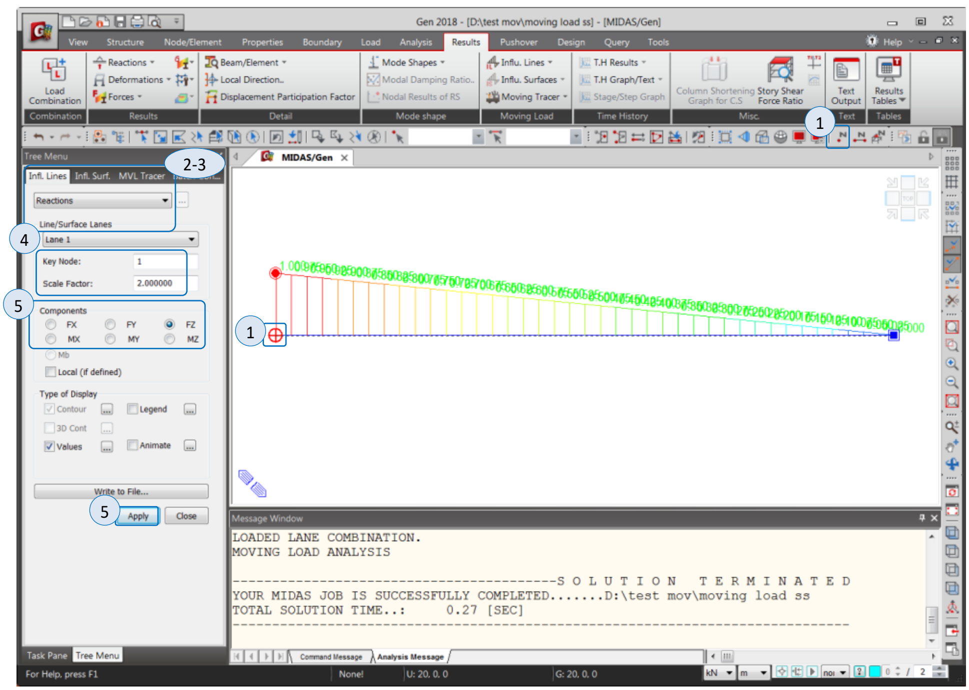
2.8 Check Analysis Results

Check the influence line for the reaction at the support Node 1.

Main Menu: ***Results / Influence Lines / Reaction...***

1. **Hidden** (Toggle off), **Iso View** (Toggle on)
2. Line/Surface Lanes > **Lane 1**
3. Key Node: **'1'**, Scale Factor: **'2'**
4. Components > **FZ**
5. **[Apply]**

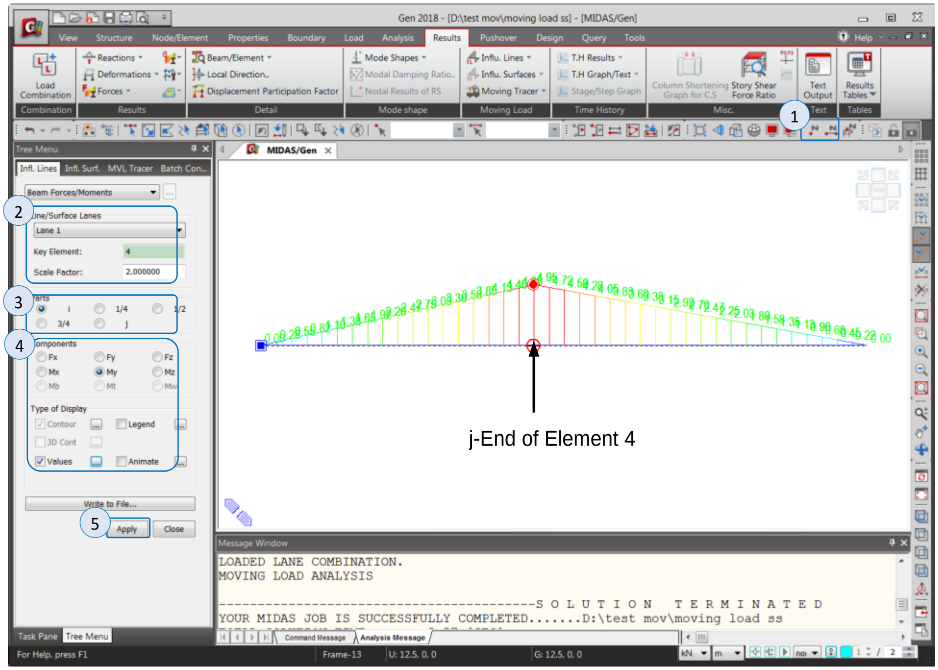
▶Fig. 10.16
Influence line for reaction at Node 5



Check the influence line for moment at j-End of Element 4.

Main Menu: ***Results / Influence Lines / Beam Forces/Moments...***
1. **Display Node Numbers** (Toggle off), **Display Element Numbers** (Toggle on)
2. Line/Surface Lanes > select **Lane 1**, Key Element: **'4'**, Scale Factor: **'2'**
3. Parts > **i**
4. Component > **My** Type of display > **Values**
5. **[Apply]**

▶Fig. 10.17
Influence line for j-End moment of Element 4



Check the moment due to the moving load.

Main Menu: ***Results/ Forces / Beam Diagrams...***

1. **Display Element Numbers** (Toggle off), **Front View** (Toggle on)
2. Load Cases/Combinations > **MVmax: Case1**
3. Components > **My**
4. Display Options > **5 Points**
5. Type of Display > **Contour, Values** (Check on)

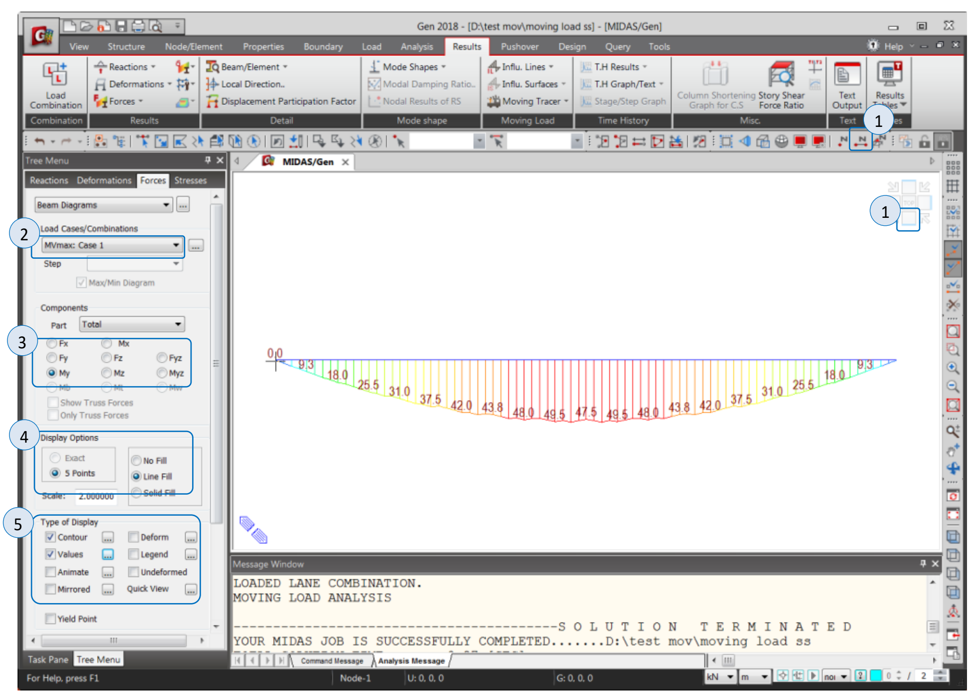
   Click **[...]** next to Values, **MinMax Only, Abs Max** (Check on),

   Check on Set Orientation, enter **'0'**, **Apply upon OK** (Check off), **[OK]**

   Output Section Location > **All**

   [Apply]

▶Fig. 10.18
Negative moment due to moving load

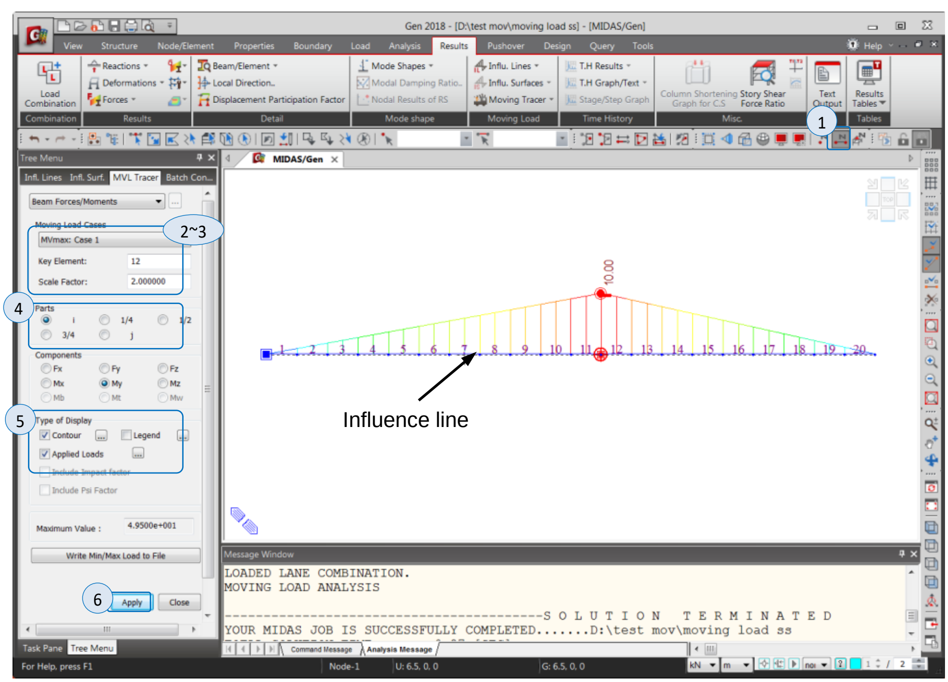


From the above diagram, it can be observed that the maximum moment takes place at j-End of Element 12 (Node 12) due to the moving load. At this instance, the position of the moving load is examined, which is determined by the influence line. From the diagram below, the position of the moving load, which causes the maximum negative moment, can be identified on the influence line.

Main Menu: ***Results / Moving Tracer / Beam Forces/Moments...***

1. **Display Element Numbers** (Toggle on)
2. Moving Load Cases > **MVmax: Case 1**
3. Key Element: **'12'**, Scale Factor: **'2'**
4. Parts > **i**
5. Components > **My**
6. Type of Display > **Contour, Applied Loads** (Check on)
7. **[Apply]**

▶Fig. 10.19
Position of moving load which causes the maximum negative moment at j-End of Element 4

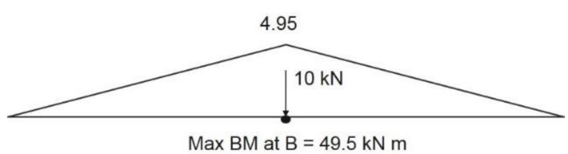


3. Manual Calculations

The wheel load of 10kN is considered. Fig. 10.21 below shows the position of the vehicle, which causes the maximum moment at Point B from the influence line analysis by midas Gen, and the influence line at that instance. The values of the influence line in Fig. 10.21 have been linearly interpolated from the analysis results. The maximum moment at Point B is thus obtained from the following:

$M_{B\ max} = 10 \times 4.95 = 49.5$ kN m

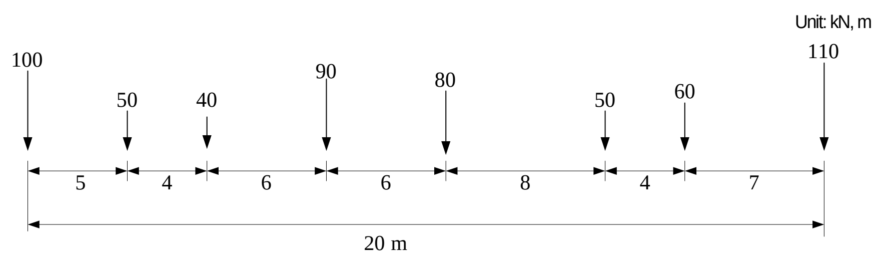
▶Fig. 10.20
Influence line calculation results

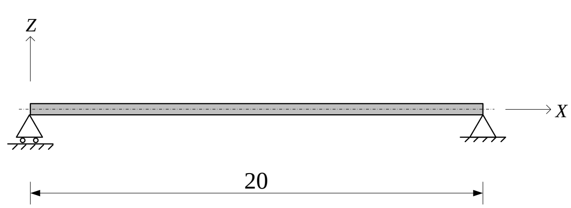


midas Gen produced the maximum moment of 49.5 kN m

4. Exercise

Find the maximum moment, shear and reaction of the simply supported beam below subjected to a moving load whose loaded length is longer than the span. Find the position of the moving load causing the maximum of each force component. Material and section properties are the same as those in Tutorial.





Structural Analysis I

Basic Analysis with MIDAS Software

Structural Analysis I
Basic Analysis with MIDAS Software