Table of Contents for MIDAS SW Utilization Guide

1. Examples of SW Utilization

① Comparing Calculations and Software Results of Textbook Example

Overview

Civil Engineering undergraduate courses are conducted through theoretical lessons utilizing textbook examples.
A typical theoretical class consists of the following two processes:
① Learning the theory.
② Solving examples from the textbook.

By utilizing midas SW (software), two additional steps can be incorporated:
③ Modeling and analysis of example problems using the software.
④ Comparison of the results.

To facilitate these steps, a collection of 20 instructional videos encompassing processes 1 through 4 is available on the provided website.
It includes 10 lectures on Structural Engineering and 10 lectures on Geotechnical Engineering, and students can access them immediately.

Students can access the educational website and explore various information through the link provided.

Educational Benefits

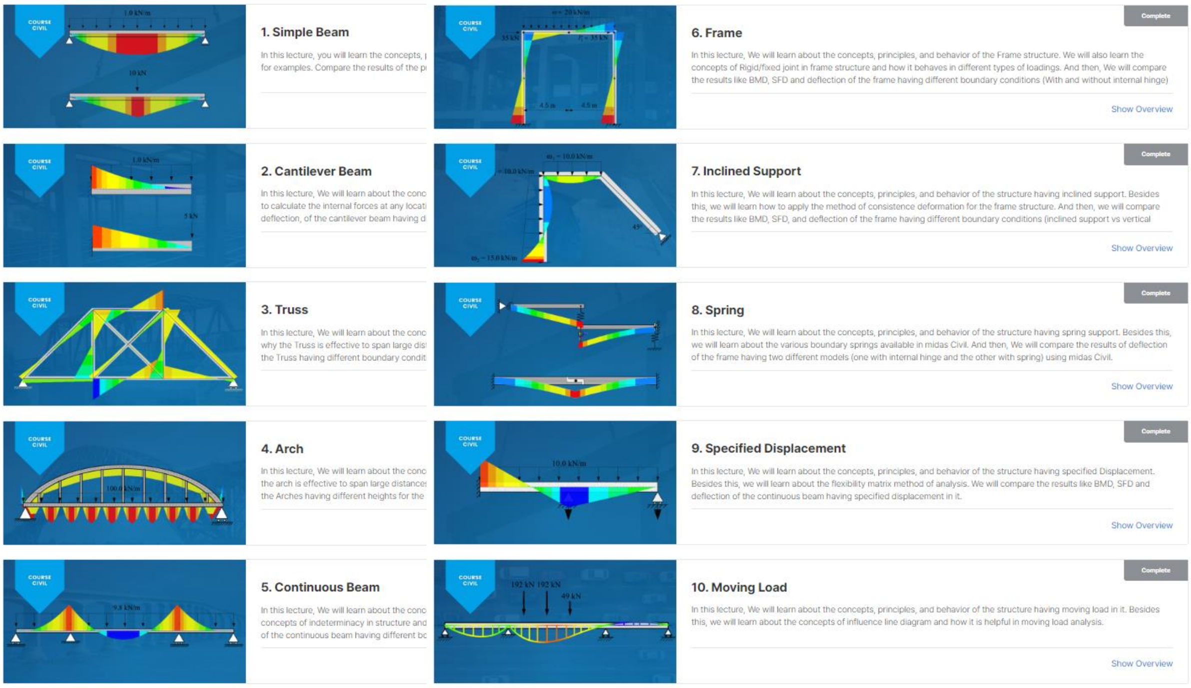
1. Visual Understanding
Beyond learning concepts based on formulas and theories, students can visually observe the behavior of structures. Visual representations assist students in grasping complex ideas.

2. Experimental Verification
Using the software for practical exercises allows virtual simulation of structural behavior, yielding results akin to actual experimentation. This enables students to compare theoretical outcomes with real-world behavior, validating the accuracy of design and analysis.

3. Real-world Application
Engaging in practical exercises involving design and analysis software offers an environment that reflects tools used in real industries. Students gain hands-on experience in analyzing real structures, cultivating understanding of practical applications. This fosters the development of practical skills applicable in real-world industry scenarios after graduation.

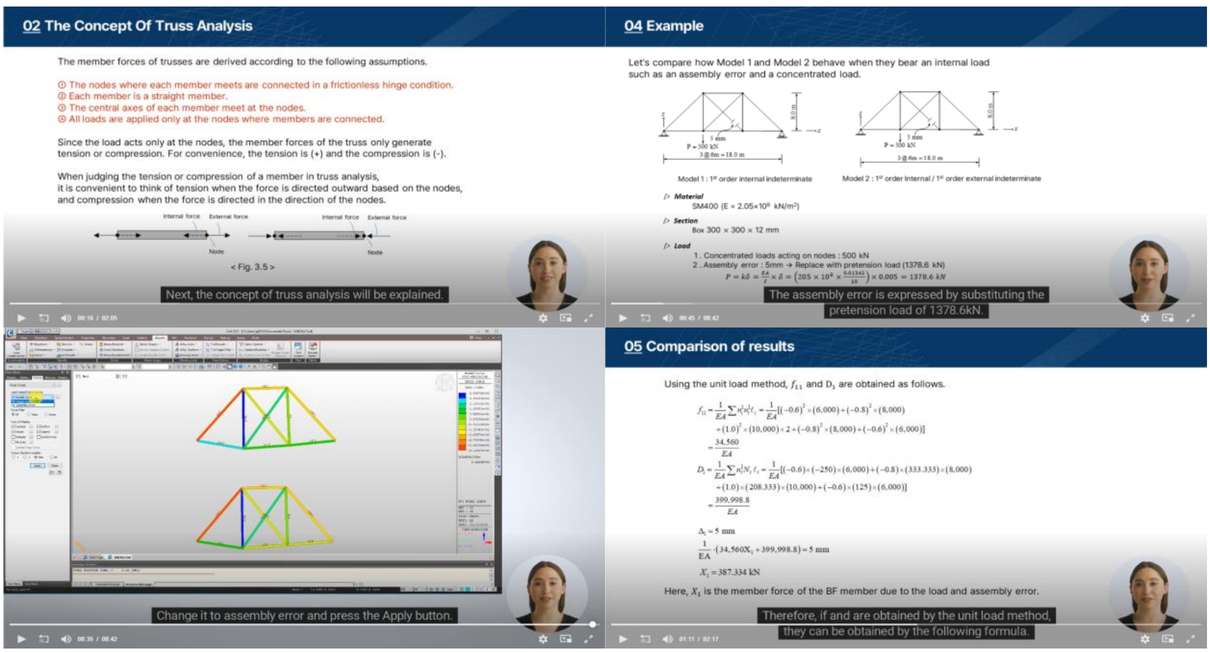
(1) Structural Engineering Course Utilizing midas Civil

- A total of 10 lectures are provided.



The lectures are structured in the following sequence :

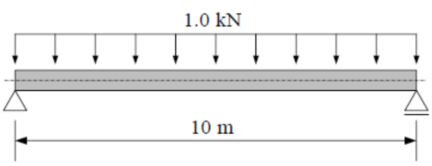
① theory explanation > ② example problems > ③ program utilization > ④ comparison of results.



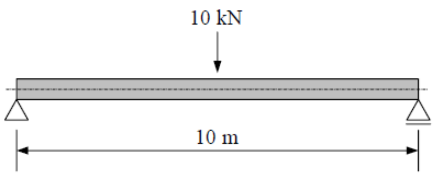
The example problems in the Structural Engineering lectures are structured as follows:

① Simple Beam

Q. Check the deformation and member forces of a simple beam with a hinge at one end and a roller at the other end through midas Civil and compare with manual calculations.



Model 1 : uniformly distributed load



Model 2 : concentrated load

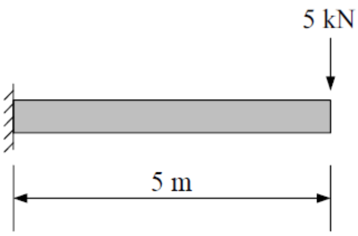
▷ ***Material***
Concrete : 24 MPa (modulus of elasticity(E) = 2.5791×10^{7} kN/m^2)

▷ ***Section***
Section Area : 8×10^{-1} m^2
Area moment of inertia(I_y) : 4.266667×10^{-2} m^4

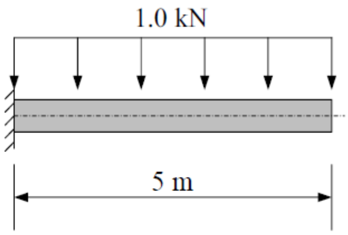
▷ ***Load***
Model 1 : uniformly distributed load 1.0 kN/m
Model 2 : concentrated load 10.0 kN

② Cantilever Beam

Check the reaction force, displacement, and member force when the load is applied to the cantilever beam as follows.



Model 1 : Concentrated Load



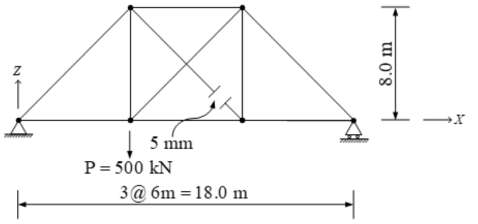
Model 2 : Uniform Distributed Load

▷ Material
Steel : SM490 (E = 2.05×108 KN/m2)

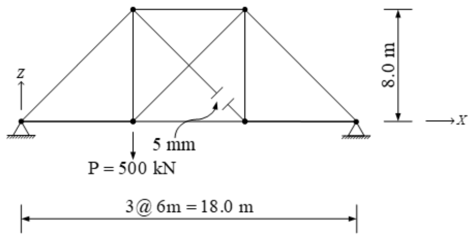
▷ Section
Section Area : 4.678 ×10-3 m2 (H 300×150×6.5×9 mm)
moment of inertia of area(Iy) : 7.21×10-5 m4

③ Truss

Let's compare how Model 1 and Model 2 behave when they bear an internal load such as an assembly error and a concentrated load.



Model 1 : 1st order internal indeterminate



Model 2 : 1st order Internal / 1st order external indeterminate

▷ ***Material***

SM400 (E = 2.05×10^8 kN/m^2)

▷ ***Section***

Box 300 × 300 × 12 mm

▷ ***Load***

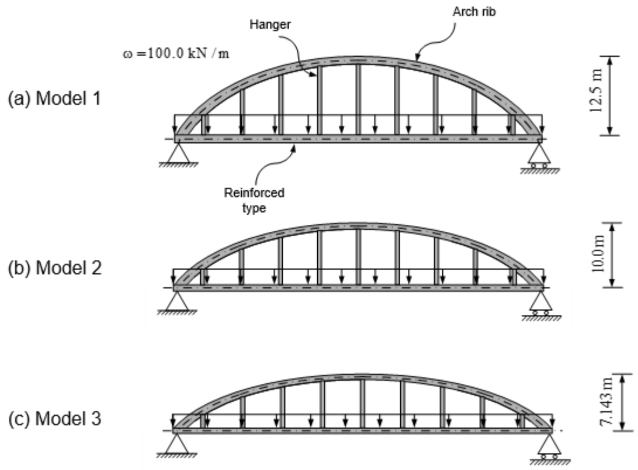
1 . Concentrated loads acting on nodes : 500 kN

2 . Assembly error : 5mm → Replace with pretension load (1378.6 kN)

$$P = k\delta = \frac{EA}{\ell} \times \delta = \left(205 \times 10^8 \times \frac{0.01345}{10}\right) \times 0.005 = 1378.6\ kN$$

④ Arch

We will analyse three arch structures with different ratios (H:L) of height(H) to span(L): 1:4, 1:5, and 1:7, and compare their deflections and member forces.



▷ **Material**

Steel : SM490 (E = 2.05×108 kN/m^2)

▷ **Section**

Arch rib : Box-shaped 1000×1000×20 mm

Reinforcement type : Box-shaped 1000×1000×20 mm

Hanger : H-shaped 500×200×10/16 mm

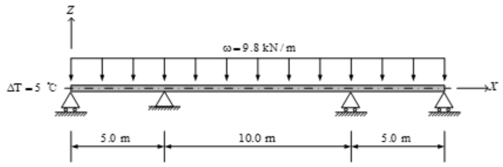
▷ **Load**

Uniformly distributed load: 100.0 kN/m

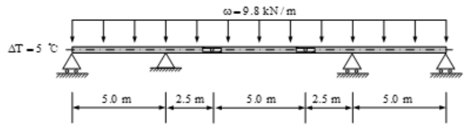
Fig 4.5 Analysis Model

⑤ Continuous Beam

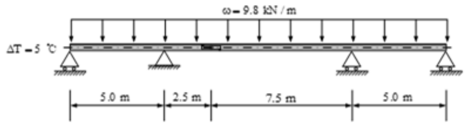
We will compare the reactions, deflections, and member forces for uniformly distributed load and temperature load due to thermal gradients in continuous beams and Gerber beams.



(a) **Model 1**
3-span continuous
second-order indeterminate



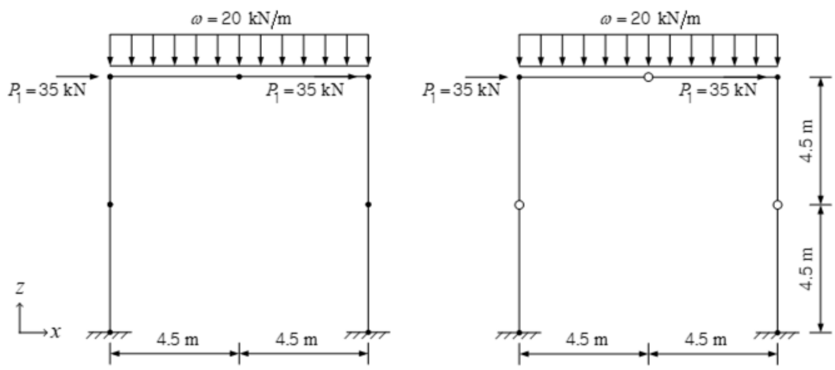
(b) **Model 2**
3-span continuous beams



(c) **Model 3**
Three-span continuous
first-order indeterminate structure.

⑥ Frame

Check the member force and deflection shape for a simple frame structure as shown in the following figure.

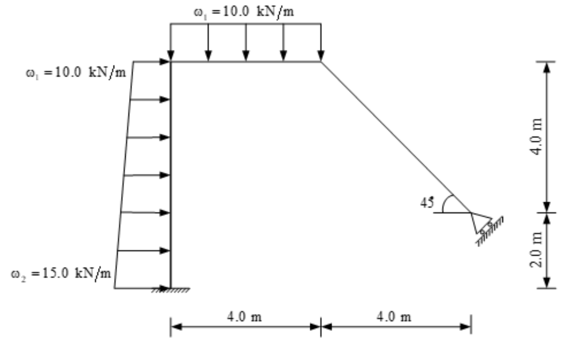


(a) Model 1 (b) Model 2

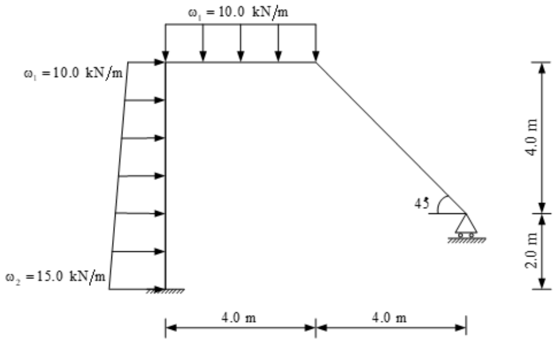
Fig 6.7 Analysis model

⑦ Inclined Support

As shown in the following figure, we can compare the displacement, member force, reaction force, etc., according to the slope of the support of the inclined frame structure that receives ***uniformly distributed loads*** in the vertical direction and ***trapezoidal distributed loads*** in the horizontal direction.



(a) Inclined Support



(b) Vertical Support

⑧ Spring

In this example, the reaction force, displacement, and member force of the structure according to the member support condition and the stiffness change of the spring element are compared and reviewed.

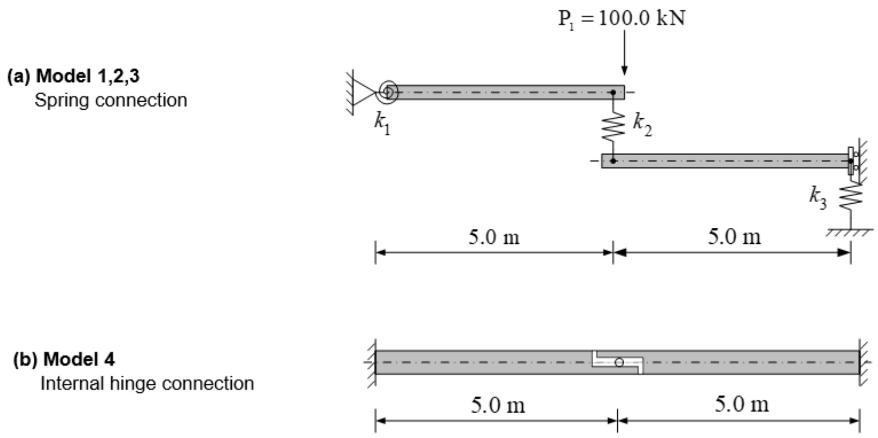


Fig 8.5 Analysis model

⑨ Enforced Displacement

This example is for a 2-span continuous beam composed of beam elements. This is an example of checking the support reaction force, deformation degree, and member force when a uniformly distributed load is applied to beam elements (Element 1, 2) and support settlement occurs at two points (Node 2, 3).
In particular, the factors to be considered when inputting the specified displacement to consider the point settlement and the result analysis after the structural analysis compare, the changes in the support reaction force or various member forces before and after the point settlement occurred.

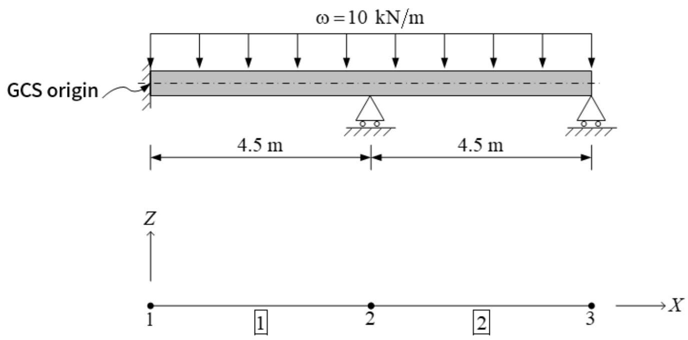


Fig 9.4 Continuous Beam

⑩ Moving Load

Using influence lines, determine the maximum sectional forces at each member of a three-span continuous beam subjected to a moving load (standard truck load), and identify the position of the moving load that produces the maximum sectional forces.

▷ **Material**

Modulus of Elasticity : 2.5e+7 kN/m^2

Poisson's Ratio : 0.2

▷ **Section**

Solid Rectangle

B × H = 3.0 m × 1.0 m

▷ **Load**

1. User defined vehicle

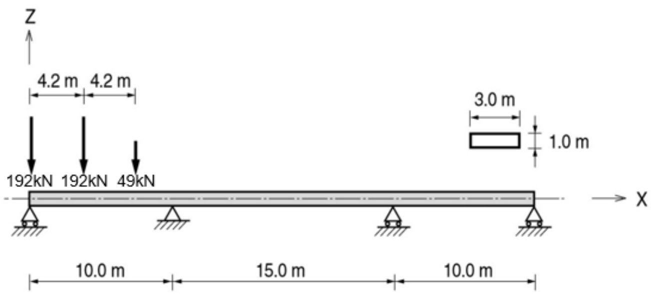
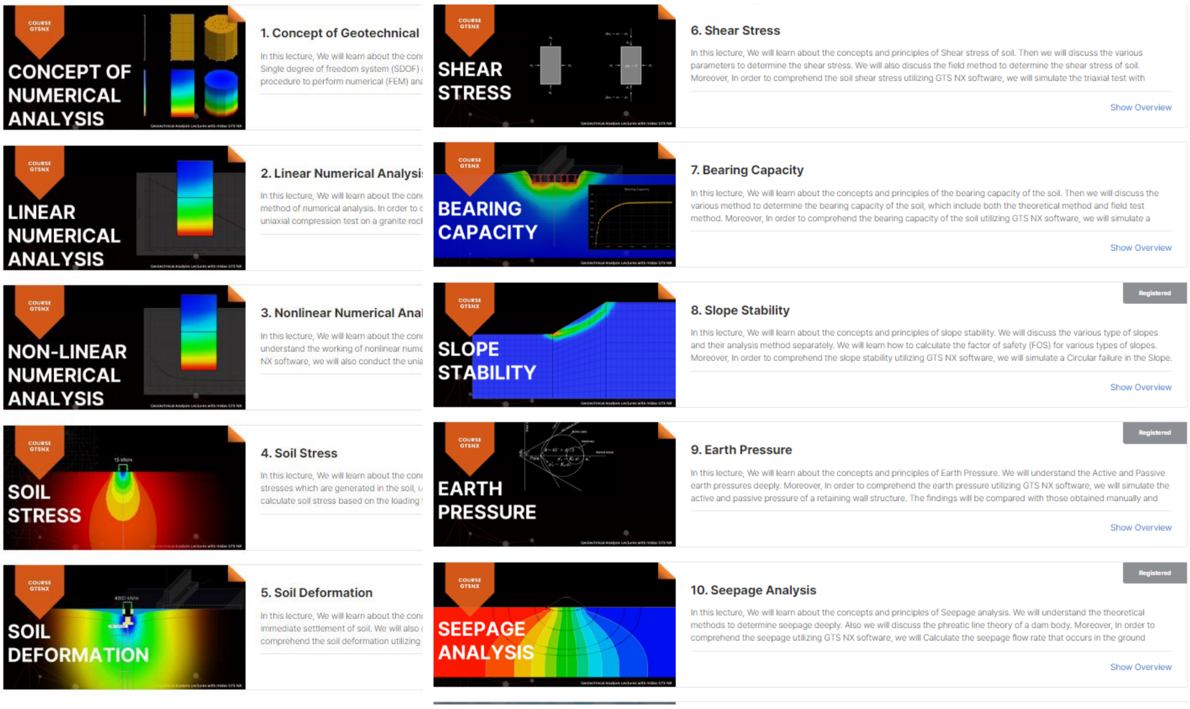


Fig 10.5 Analysis Model

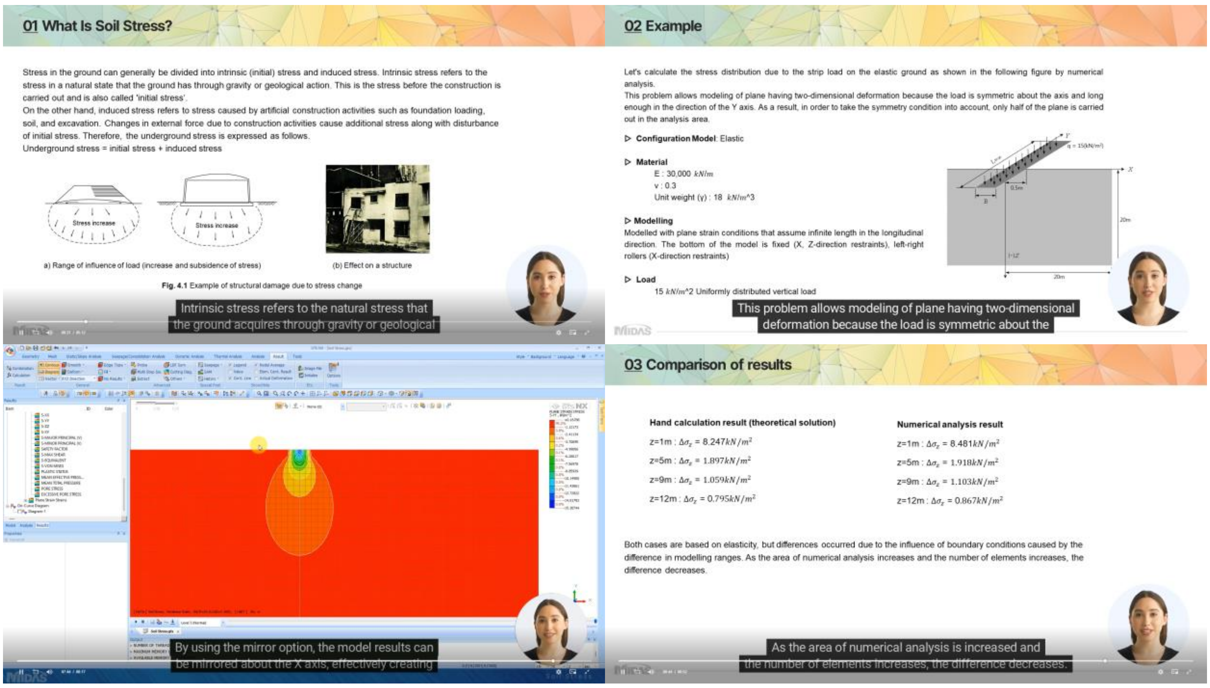
(2) Geotechnical Engineering Course Utilizing midas GTS NX

- A total of 10 lectures are provided.



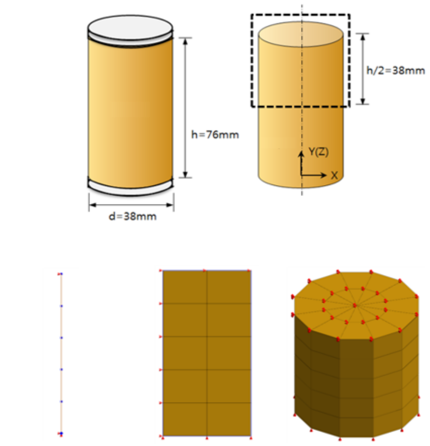
The lectures are structured in the following sequence :

① Theory Explanation > ② Example Problems > ③ Program Utilization > ④ Comparison of results.



The example problems in the Geotechnical Engineering lectures are structured as follows:

① Concept of Geotechnical Numerical Analysis



(a) 1D Model **(b)** 2D Model **(c)** 3D Model

Fig. 1.7 Example Model

▶ **Constitutive model** : Elastic

▶ **Material**

E : 50,000 kN/m^2 , ν : 0.3 , Unit weight(γ) : 18 kN/m^3 (initial stress)

▶ **Modeling**

	1D Model	2D Model	3D Model
Element	1D beam	2D axisymmetric	3D solid
Section	Bar	Square	Cube
Diameter	0.038m		

▶ **Boundary condition**

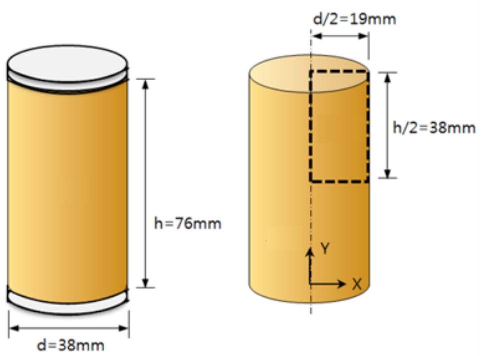
Lower fixed, upper constant displacement

▶ **Weight**

Self-weight of sample: initial stress

② Linear Numerical Analysis

We will simulate a uniaxial compression test on a granite rock specimen as shown in the following figure and practice linear numerical analysis.



< Fig. 2.3 Example Model >

▶ Constitutive model : Elastic

▶ Material

E : 30,000,000 kN/m^2 , ν : 0.23 ,

saturated unit weight 25 kN/m^3 (initial stress)

▶ Modeling

③ Non-linear Numerical Analysis

We will simulate a uniaxial compression test on a granite rock specimen as shown in the following figure and practice linear numerical analysis.

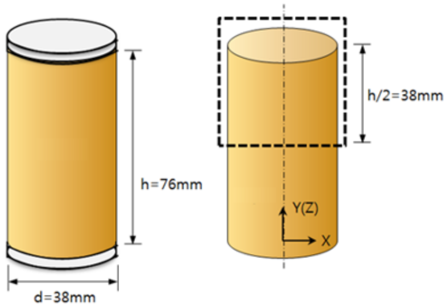


Fig. 3.3 Example Model

▶ **Constitutive model** : Mohr-Coulomb

▶ **Material**

Elastic Modulus (E) : 30,00,000 kN/m^2

Poisson's Ratio (v) : 0.3

Unit weight (γ) : 18 kN/m^3

Cohesion (c) : 30 kN/m^2

Internal friction angle (∅) : 30˚

▶ **Boundary condition**

Lower fixed, upper constant displacement

▶ **Weight**

Self-weight of sample: initial stress

④ Soil Stress

Let's calculate the stress distribution due to the strip load on the elastic ground as shown in the following figure by numerical analysis.

This problem allows modeling of plane having two-dimensional deformation because the load is symmetric about the axis and long enough in the direction of the Y axis. As a result, in order to take the symmetry condition into account, only half of the plane is carried out in the analysis area.

▷ **Configuration Model**: Elastic

▷ **Material**

E : 30,000 kN/m

v : 0.3

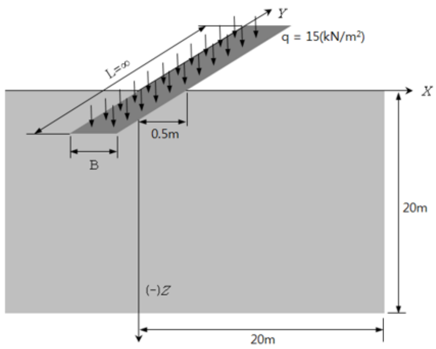
Unit weight (γ) : 18 kN/m^3

▷ **Modelling**

Modelled with plane strain conditions that assume infinite length in the longitudinal direction. The bottom of the model is fixed (X, Z-direction restraints), left-right rollers (X-direction restraints)

▷ **Load**

15 kN/m^2 Uniformly distributed vertical load



⑤ Soil Deformation

Calculate the settlement due to the continuous foundation installed on the elastic ground as shown in Figure 5.4 through numerical analysis.

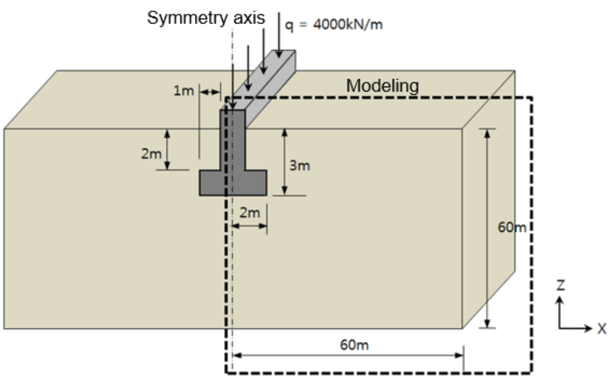


Fig 5.4 Example model

▶ Continuous foundation with width 4m and embedding depth 3m, load q = 4,000kN/m

▶ 2D modelling under planar deformation conditions is possible because the base width is sufficiently long

▶ Load, ground and structure are all symmetrical, so only ½ of the whole is considered

▶ Since the geometric shape is simple, quadrangular solid elements are used.

	Geotechnical properties	Basic physical properties
Modulus of elasticity (E)	15,000 kN/m^2	21,000,000 kN/m^2
Poisson's ratio (ν)	0.45	0.18
(wet) unit weight (γ)	18 kN/m^3	24 kN/m^3

⑥ Shear Stress

Let's determine the shear strength parameters (c' $\emptyset'$) of soil by analysing the stress-strain relationship for various confining pressures using the triaxial compression test results of a soil sample, as shown in Figure 6.11.

Since the triaxial specimen is geometrically axisymmetric and behaves symmetrically about the horizontal axis passing through the center of the specimen, only half the specimen height needs to be modelled.

▶ **Constitutive model** : Mohr-Coulomb

▶ **Material**

		Properties	Notes
Pre-Yield (Elastic behaviour)	Modulus of Elasticity (E)	10,000 kN/m^2	Isotropic Linear Elastic
	Poisson's ratio (ν)	0.3	
Post-Yield (Plastic behaviour)	Internal friction angle(∅)	23˚	Fully plastic
	Cohesion(c)	5 kN/m^2	
Unit weight(γ)		Ignored (0 kN/m^3)	Initial Stress

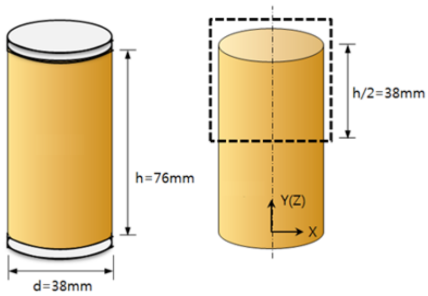


Fig. 6.11 Example Model

⑦ Bearing Capacity

Let's find the bearing capacity of a simple shallow foundation (continuous footing) as shown in Fig. 7.9.

▶ Constitutive Model : **Mohr-Coulomb**
▶ Material

		Properties	Notes
Pre-Yield (Elastic behaviour)	Modulus of Elasticity (E)	257,000 kN/m^2	Isotropic Linear Elastic
	Poisson's ratio (ν)	0.286	
Post-Yield (Plastic behaviour)	Internal friction angle(∅)	0˚	Fully plastic
	Cohesion(c)	100 kN/m^2	
Unit weight(γ)		20 kN/m^3	Initial Stress

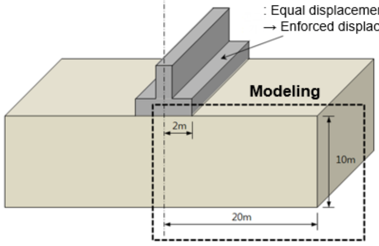


Fig. 7.9 Example model

⑧ Slope Stability

Let's examine the safety against circular failure of the slope as shown in Figure 8.8.

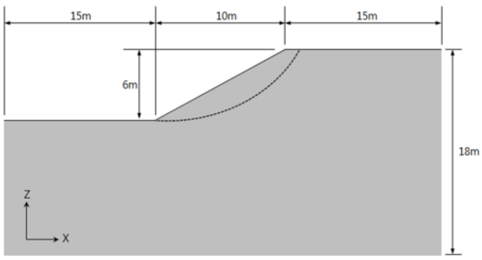


Fig 8.8 Example model

▶ Configuration model: **Mohr-Coulomb**
▶ Material

		Geotechnical properties	Note
Pre-Yield (Elastic behaviour)	Modulus of Elasticity (E)	10,000 kN/m^2	Isotropic Linear Elastic
	Poisson's ratio (ν)	0.3	
Post-Yield (Plastic behaviour)	Friction Angle(∅')	25˚	Fully plastic
	Cohesion (c)	2.6 kN/m^2	
Unit weight (γ)		17 kN/m^3	Initial stress

⑨ Earth Pressure

Through this example, we will learn how to numerically calculate the active and passive pressures of a retaining wall structure. Let's calculate the soil pressure on the retaining wall supported by the ground using numerical analysis, as shown in Figure 9.10.

▶ Configuration model

Sand: Mohr-Coulomb

Concrete: Elastic

▶ Material

		Ground (Sand)	Concrete
Elastic Properties	Modulus of elasticity (E)	182,000 kN/m^2	21,30,00,000 kN/m^2
	Poisson's ratio (ν)	0.3	0.2
Plastic Properties	Angle of Internal Friction(Ø')	36˚	-
	Cohesion (c)	1	-
Unit weight (γ)		17 kN/m^3	24 kN/m^3

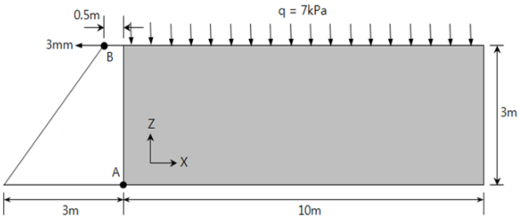


Fig 9.10 Example model

⑩ Seepage Analysis

Calculate the seepage flow rate that occurs in the ground including the embankment as shown in Figure 10.4 by numerical analysis.

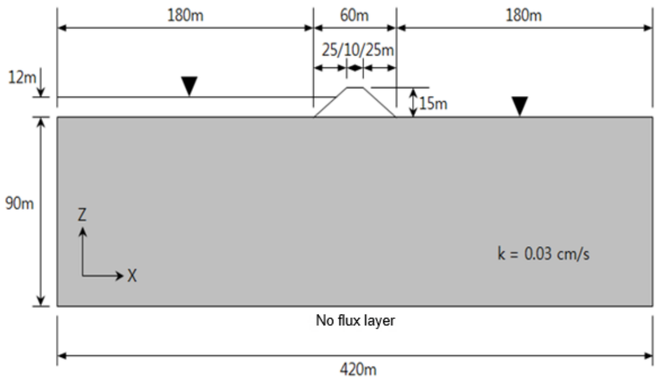


Fig 10.6 Example model

▶ Configuration model

Isotropic permeability model

($k= k_x = k_y$ $=constant$)

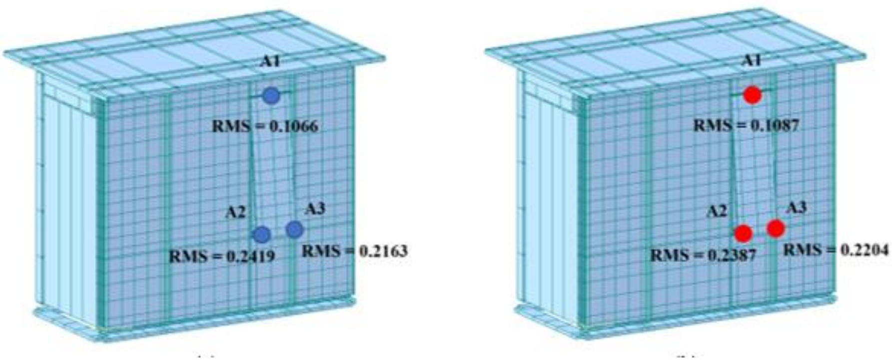
▶ Material

Geotechnical properties	value
Horizontal permeability coefficient	0.03cm/sec
Vertical permeability coefficient	0.03cm/sec

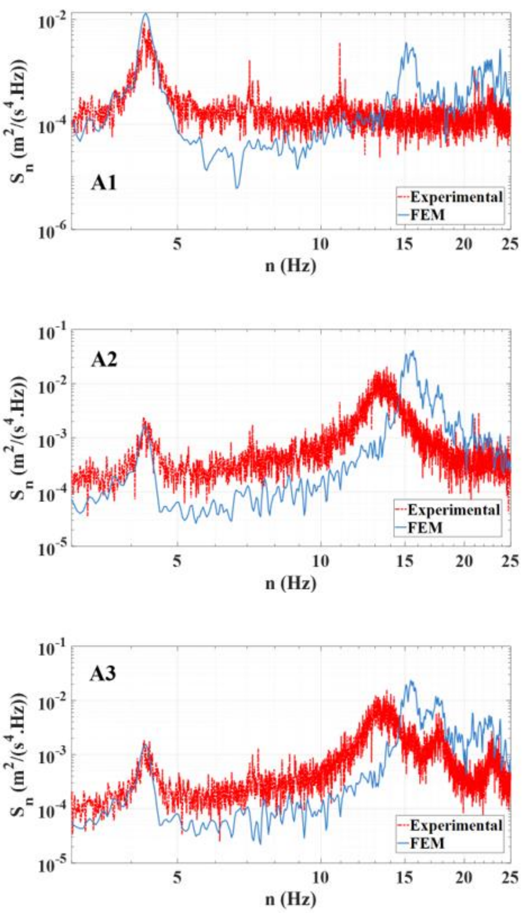
(3) Comparison of RMS Values of Curtain Wall Acceleration Caused by Wind Load

This task involves comparing the results of the analysis model with experimental values.

[Analysis Model Results]



[Comparison with Experimental Results]



② Developing Analytical Thinking through Repetitive Exercises

Outline

Understanding civil structures involves repetitive learning : adjust inputs under specific conditions to observe corresponding outputs. This process reveals structural behavior, aiding design, and analysis comprehension.

Below is an outline of this repetitive learning process:

1. Problem Definition : Set learning objectives for a particular civil structure. For instance, students might evaluate the stability of a structure under specific load or constraint conditions.
2. Modeling: Create a model of the structure according to the defined problem.
3. Initial Input Value Setting : Set initial conditions for the structure, typically opting for realistic and accurate values.
4. Output Value Analysis : Examine output values to understand civil structure behavior, revealing connections between input variables and outputs.
5. Input Value Alteration : Adjust initial inputs to explore diverse scenarios. For instance, observe changes in civil structure behavior by modifying factors like cross-section, length, and height.

Educational Benefits

1. Enhancement of Problem-Solving Skills

- Students engage in manipulating input and analyzing output to foster problem-solving. This nurtures logical thinking and enhances their grasp of variables and structural interactions, refining their ability to pinpoint optimal conditions.

2. Provision of Experimental Learning Opportunities

- Students gain hands-on learning via modeling and simulation, saving costs and time over physical experiments. They explore diverse scenarios, analyze outcomes, and develop an experimental approach, enhancing their learning efficiency.

3. Reinforcement of Self-Directed Learning and Promotion of Creativity

- In the process of changing input values and analyzing results, students engage in self-directed learning. This learning method enhances their self-regulated learning capabilities and cultivates self-directed learning habits. Moreover, it fosters the ability to generate creative ideas and embrace diverse perspectives.

(1) Creating a Truss to Maximize Load-Bearing Capacity

This project involves students designing and constructing a truss that can support the highest load while meeting the given conditions.

The project extends to testing the truss under load to confirm its stability.

Prior to modeling, midas Civil will be utilized to estimate the load at which failure occurs, followed by practical testing.

Preparation:

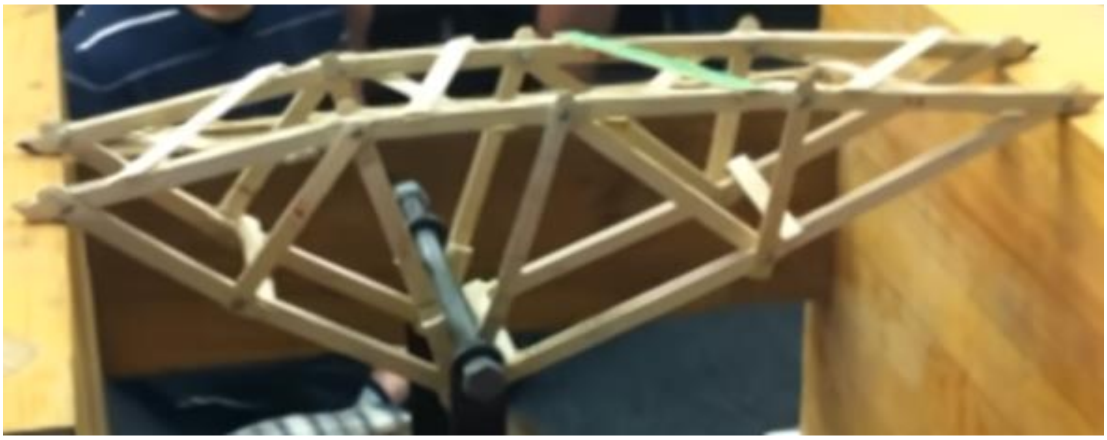
Popsicle sticks, small bolts, PVA adhesive

Conditions:

Width: 450mm

Determinate truss structure: Triangular configuration with all pin connections

Usage of popsicle sticks limited to 50 or fewer



(2) Designing a 10-Floor Hotel Structure

A team project is underway for designing a hotel structure spanning 10 floors. (3 members per team)

Approximate specifications will be provided for aspects such as spatial width, roofing, and flooring. The project follows a comprehensive design process.

2) PROJECT DESCRIPTION

A) GENERAL

The structure to be designed is a ten-story hotel. Figure 1 shows an isometric view of a similar structure. The footprint of the structure and the number of stories was selected such that it meets the needs of the owners and their investment goals. Figure 2 shows the foundation plan and Figure 3 and Figure 4 show typical floor and roof framing plan, respectively. Figure 5 shows typical moment frame elevations. The overall height of the hotel is restricted as described in the next sentences to be consistent and in harmony with the adjacent structures. Clear height (top of slab to bottom of ceiling) for the first floor is 11 ft and for the rest of the floors is 9 ft. A minimum space of 7 in. must be provided above the bottom of the ceiling for ductwork and plumbing and electrical lines. The total building height is 124.5 ft. The majority of the exterior walls will be precast concrete (40 psf) and will supported at each floor level as well as at the foundation.

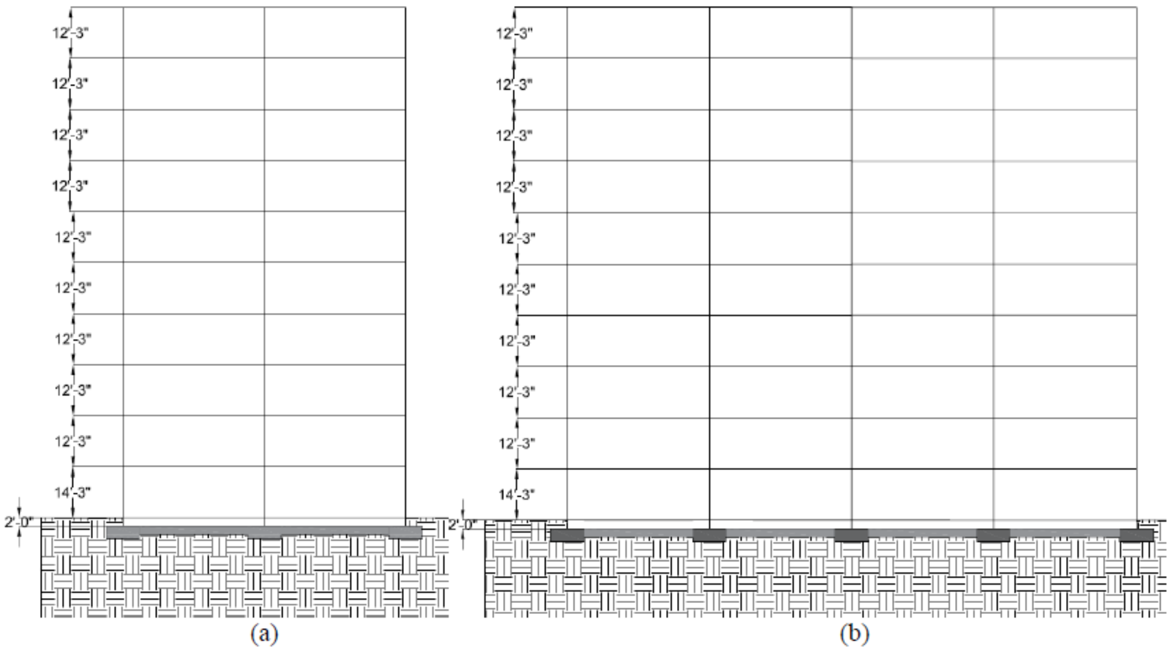


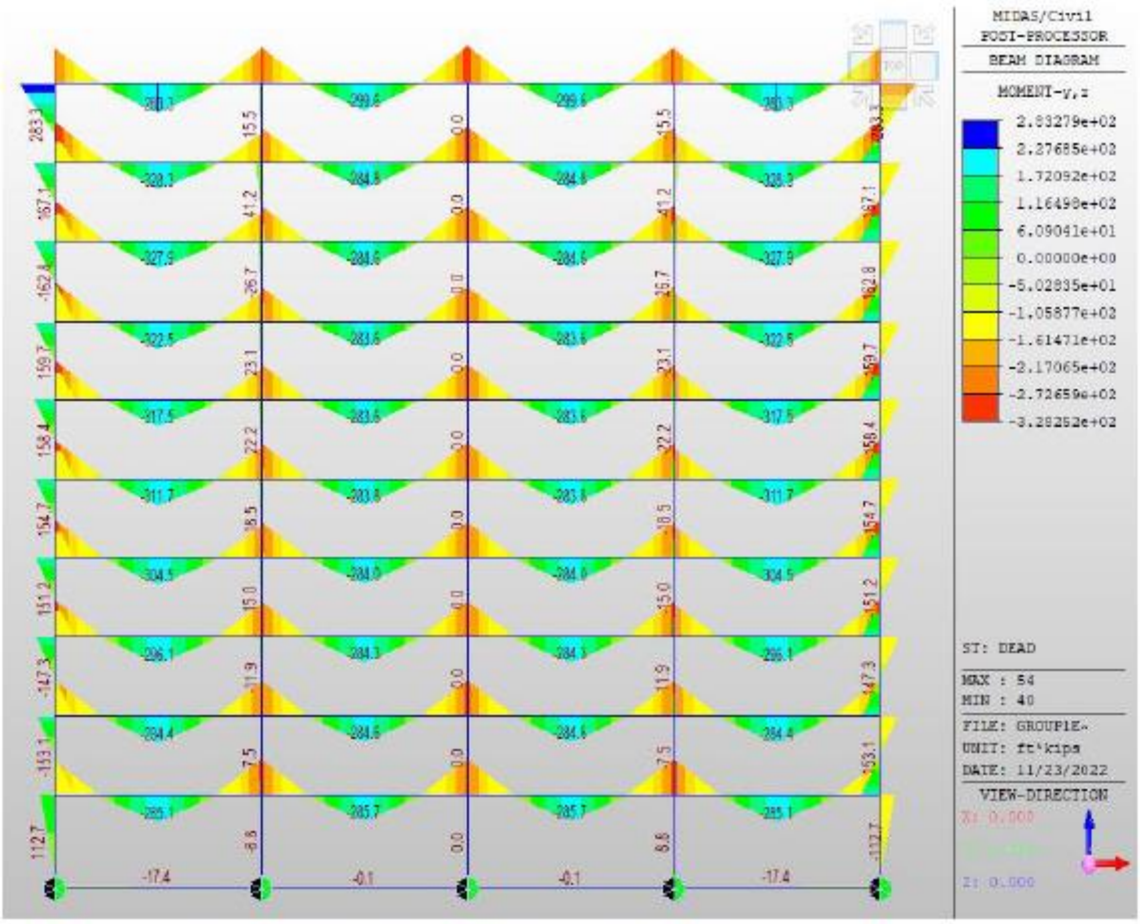
Figure 5. Elevation of typical moment frame a) north south direction, b) east west direction

Encourage analysis of moment frames using midas Civil.

C) MOMENT FRAMES

Moment frames in both north-south and east-west directions shall be used to resist lateral loading from wind. *The lateral drift caused by wind loading is limited to h/400, where h is the overall height of the structure*. Lateral drift shall be checked using service level load combinations. Assume the wind load to be 40 psf at ultimate (W). It is recommended that the typical moment frames are modeled and analyzed in Midas Civil as two dimensional models. ***It is recommended that you start modeling the moment frames as soon as the project is posted in Canvas.*** We will spend a class period to help you with how to use Midas Civil, but there will be many things that you will have to learn on your own since the goal of the class is to teach you the fundamentals of reinforced concrete design and not how to use a software. The software shall be used only for structural analysis (i.e. to determine load demands and to compute immediate deflections). Structural design shall be conducted by hand. The instructor will post guidelines in Canvas for how to access Midas Civil.

e. Moment Analysis:



③ Enhancement of Students' Comprehension through Visualization

Outline

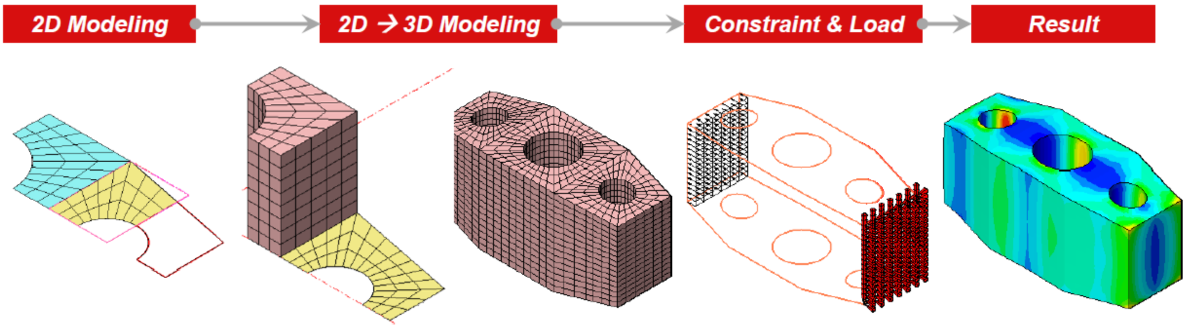
One major advantage of utilizing software in education is visualization. Results obtained through software are analyzed and presented visually to students. This process involves visually confirming stress, deformation, displacement, mode shapes, etc., in structures, aiding in understanding and interpreting outcomes. Through this process, students can easily comprehend how actual structures behave in response to learned external forces during class time.

Educational Benefits

1. Enhanced Understanding through Visualization - Structural analysis software offers the capability to visually represent the behavior and response of structures. Students can confirm results graphically and intuitively comprehend stress, deformation, displacement, and more in structures. Visual representation makes comprehending complex structural behavior easier, significantly enhancing students' understanding. 2. Improved Real-World Problem-Solving Skills - Modeling and analyzing structures using structural analysis software parallel real-world problem-solving processes. Through software, students analyze structures, interpret results, and enhance skills necessary for actual structural design and analysis. This cultivates their ability to solve problems related to stability, strength, deformation, and more in structures. 3. Realistic Experiential Learning - Utilizing structural analysis software grants students experimental learning opportunities. Complex structural behavior, often challenging to replicate through physical experiments, can be simulated and analyzed using software. This exposes students to diverse scenarios and result analysis, enhancing their practical problem-solving capabilities. 4. Reinforcement of Self-Directed Learning - Using structural analysis software prompts students to engage in self-directed learning. As they learn software operation, model, and analyze, students develop self-regulated learning skills and foster self-directed learning habits.

(1) 3D Linear Static Analysis

midas FEA NX allows for performing 3D linear static analysis.



About this Training…

This tutorial model is composed of 3D solid elements generated by Extrude, Project, and Mirror functions from 2D elements. The analysis is performed by defining the nodal load and the face pressure load of the element. Check the principal stress result of Solid with Clipping Plane, Probe Result, Vector Plot function.

Learning Target

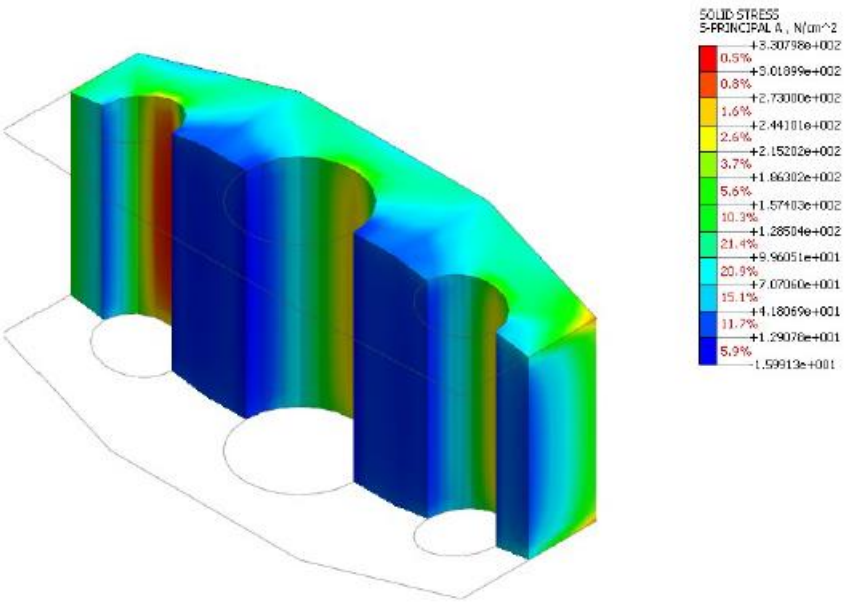
How to model 2D elements with K-Edge Area of Map Mesh

How to define from 2D element to 3D Solid element with Extrude, Project, Mirror function

How to check the result with Clipping Plane, Probe Result, and Vector Plot

This process enables learning techniques to generate 3D elements from 2D elements using methods like Extrude, Project, and Mirror, while also visualizing the analysis process and outcomes.

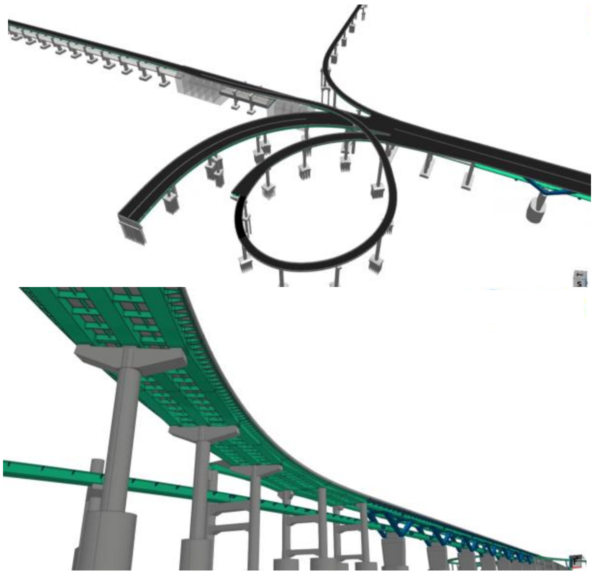
Moreover, students can use the Clipping Plane to inspect Solid Stress in specific cross-sections.



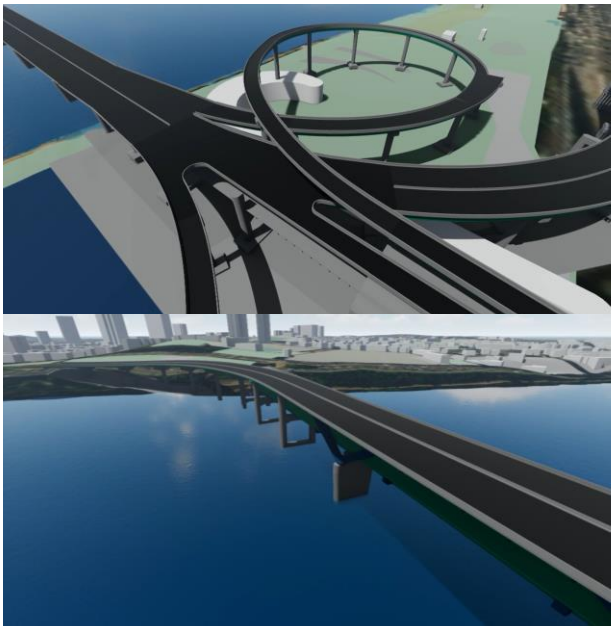
(2) 3D Visualization (VR/AR)

Students can extract model files from Midas SW in 3D DWG (*.dxf) format for use in rendering/VR-specific programs like Unity, Lumion, and more.

<Example of a midas CIM Model>



<Example of a Lumion Model>



④ Verification of Formulas using MATLAB and Other Software

Outline

In many universities, students utilize engineering software like MATLAB to learn about numerical analysis theory, nonlinear equations, iterative methods, interpolation, linear regression, and various algorithms. By programming functions based on these algorithms and solving example problems, students can cultivate their skills in numerical analysis. During this process, comparing the results of the work students produce with those obtained using midas software for verification purposes can greatly aid their learning by ensuring accuracy and identifying errors.

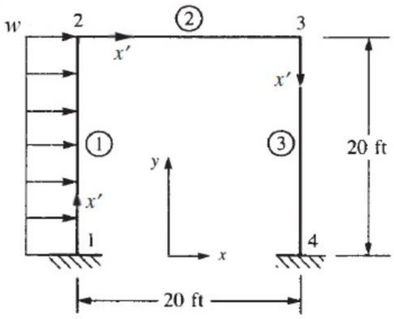
Educational Benefits

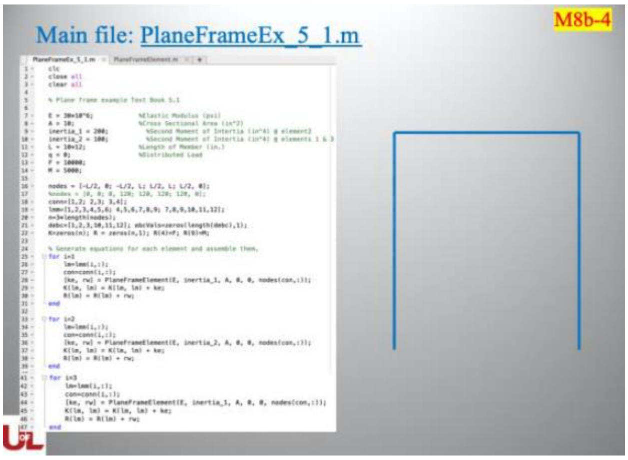
1. Applying Theoretical Concepts to Practical Problems - Through MATLAB, students can simulate and verify solutions to various real-world problems, such as stress-strain analysis of structures and stability analysis of soils. 2. Linkage and Evaluation in Real-World Context - MATLAB's compatibility with commercial software used in civil engineering allows for verifying students' MATLAB code against real-world applications. Results can be cross-referenced, and data generated in midas Civil can be imported into MATLAB for analysis. This integration allows students to assess their problem-solving skills using MATLAB in real-world contexts and make necessary adjustments and improvements. 3. Enhancement of Programming Skills - MATLAB's programming capabilities enable students to enhance their programming skills while solving problems through MATLAB coding. Utilizing MATLAB for civil engineering tasks is highly beneficial, but it requires an understanding of MATLAB language and programming concepts. Consequently, employing MATLAB for solving civil engineering problems and validating code through integration with commercial software fosters the acquisition of essential theoretical and practical skills in civil engineering for students.

(1) Results Comparison between MATLAB and midas Civil for Frame Structures

- MATLAB Work Example

$E = 30 \times 10^6$ psi
$A = 15$ in.^{2}
$I = 250$ in.^{4}





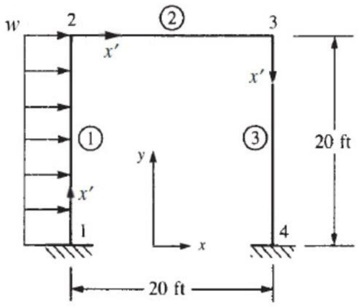

```
clc
close all
clear all

% Plane frame example Text Book 5.2

E = 30*10^6;              %Elastic Modulus (psi)
A = 15;                   %Cross Sectional Area (in^2)
inertia = 250;            %Second Moment of Intertia (in^4)
L1 = 20*12;                %Length of Member (in.)
L2 = 20*12;
L3 = 20*12;
w = 350/12;                    %Distributed Load

nodes = [0, 0; 0, 20*12; 20*12, 20*12; 20*12, 0];
conn=[1,2; 2,3; 3,4];
lmm=[1,2,3,4,5,6; 4,5,6,7,8,9; 7, 8, 9, 10, 11, 12];
n=3*length(nodes);
debc=[1,2,3,10,11,12]; ebcVals=zeros(length(debc),1);
K=zeros(n); R = zeros(n,1);

% Generate equations for each element and assemble them.
for i=1
    lm=lmm(i,:);

    con=conn(i,:);
    [ke, rw] = PlaneFrameElement(E, inertia, A, 0, -w, nodes(con,:))
    K(lm, lm) = K(lm, lm) + ke;
    R(lm) = R(lm) + rw;
end

for i=2
    lm=lmm(i,:);
    con=conn(i,:);
    [ke, rw] = PlaneFrameElement(E, inertia, A, 0, 0, nodes(con,:))
    K(lm, lm) = K(lm, lm) + ke;
    R(lm) = R(lm) + rw;
end

for i=3
    lm=lmm(i,:);
    con=conn(i,:);
    [ke, rw] = PlaneFrameElement(E, inertia, A, 0, 0, nodes(con,:))
    K(lm, lm) = K(lm, lm) + ke;
    R(lm) = R(lm) + rw;
end
```

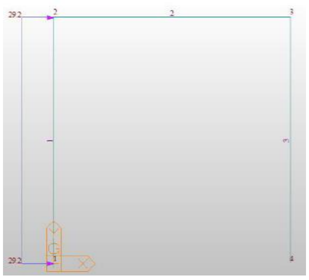
- midas Civil Work Example

Completed Model (Show the following items in figure):

- Tree Menu: Work
- Nodal forces (with Magnitudes)
    - Display distributed loads
- BCs (Ex: 110000)
- Node #
- Element #

$E = 30 \times 10^6$ psi
$A = 15$ in.^{2}
$I = 250$ in.^{4}





	Elem	Load	Part	Axial (lbf)	Shear-y (lbf)	Shear-z (lbf)	Torsion (in*lbf)	Moment-y (in*lbf)	Moment-z (in*lbf)
	1	element l	I[1]	999.02	-5542.99	0.00	0.00	0.00	-393659.13
	1	element l	J[2]	999.02	1457.09	0.00	0.00	0.00	96650.03
	2	element l	I[2]	-1457.09	999.02	0.00	0.00	0.00	96650.03
	2	element l	J[3]	-1457.09	999.02	0.00	0.00	0.00	-143114.85
	3	element l	I[3]	-999.02	-1457.09	0.00	0.00	0.00	-143114.85
▶	3	element l	J[4]	-999.02	-1457.09	0.00	0.00	0.00	206585.59

2. FAQ

① Installing and Running Midas SW on Mac

To utilize midas SW on a Mac, as the software is primarily designed for Windows, you'll need to set up a Windows environment first.

Setting up Windows on a Mac can be achieved through the following methods:

- Using Boot Camp (for Intel-based Mac computers)
- Creating a Virtualized Windows Environment (for M-series Mac computers)

For Intel-based Macs, follow these steps:

1. Use Boot Camp to install Windows on your Mac. This tool helps you create a dedicated Windows partition. -> Install Windows on your Mac with Boot Camp Assistant
2. Once Windows is set up, proceed with installing Midas SW as you would on a regular Windows machine.

For M-series Macs, follow these steps:

1. Choose a virtualization program such as Parallels Desktop, VMware Fusion, or UTM. Parallels Desktop and VMware Fusion offer trial versions for a limited period, while VMware Fusion provides a free Personal Use License for students. UTM offers a completely free virtualization solution, but it lacks support for GPU emulation and 3D acceleration.

After selecting a virtualization program:

2. Install and configure the chosen virtualization software on your Mac.

3. Install Windows 10 on the virtual machine.
Download Parallels Desktop and User's Guide
VMware Fusion Player – Personal Use License and Guide
UTM Documentation – Installation – Windows 10 and higer

Once Windows is installed, proceed with downloading and installing Midas SW just as you would on a Windows PC. --> Midas SW Installation Guide

Remember that the specific steps may vary depending on the virtualization software you choose. Always refer to the official documentation provided by the virtualization software and Midas SW for precise guidance.

In summary, to use Midas SW on a Mac:

1. Determine if your Mac is Intel-based or M-series.
2. Set up Windows using Boot Camp or a virtualization program like Parallels Desktop, VMware Fusion, or UTM.
3. Install Windows within the selected method.
4. Download and install Midas SW within your Windows environment.

For detailed installation steps and software downloads, refer to the official documentation for each tool and software.

Install Windows 10 on your Mac with Boot Camp Assistant

Download Parallels Desktop and User's Guide

VMware Fusion Player – Personal Use License and Guide

UTM Documentation – Installation – Windows 10 and higer

Midas SW Installation Guide

② Utilized Solvers for Midas Software by Product

1) Solvers for Each midas Analysis Product

SW Name	Solver Name
midas Civil / midas Gen	FES Solver
midas GTS NX / midas FEA NX	MEC Solver

The FES Solver possesses various functions that support design in the civil/architectural fields, while the MEC Solver has many functions for analysis in the ground/mechanical fields.

Both the FES Solver and MEC Solver were developed using the “Multi-frontal Solver”, known for its excellent analysis speed.

2) Pros and Cons of the Multi-frontal Solver

The Multi-frontal Solver is a type of direct method used to solve large-scale sparse matrix problems.

It is particularly effective for handling statically determined matrix.

Although it is one of the memory-intensive direct methods, it supports efficient parallel processing and is suitable for large-scale systems.

[Advantages]

- Effective for large-scale static sparse matrix.

- Suitable for parallel processing, allowing for fast calculations on high-performance systems.

[Disadvantages]

- Requires a high amount of memory.

- May not be effective for some matrices that are not structurally determined.