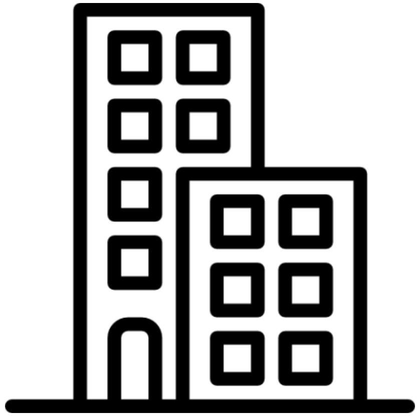
GTS NX INTRODUCTION

沈約翰 (John)
john@midasuser.com.tw

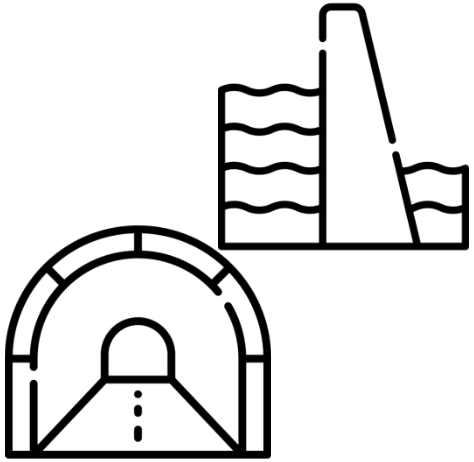
CONTENTS

MIDAS GTS NX is a **Finite Element Analysis software** developed by MIDAS IT, specifically designed for **advanced geotechnical engineering and soil-structural interaction analysis**.

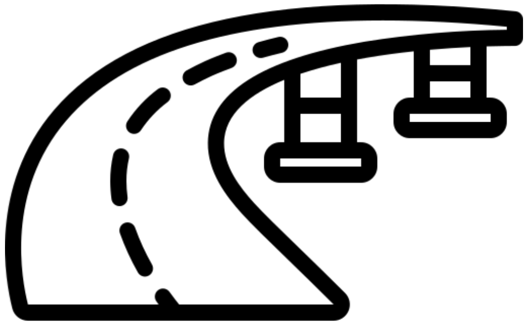


Concrete / Steel Behavior

Soil - Structure Interaction



Tunnels, dams, and underground structures



Bridges, large infracstructures and other complex geometries

GTS NX KEY FEATURES

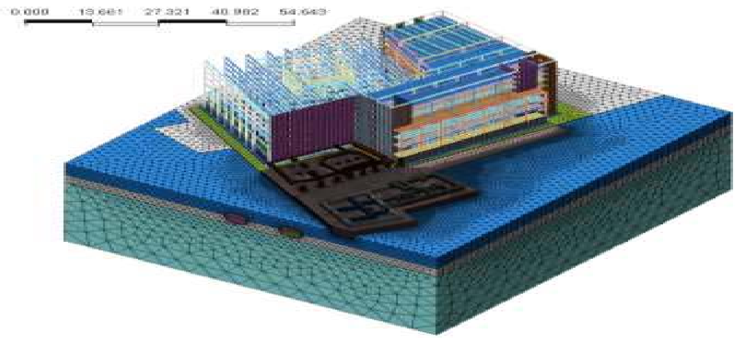
FEATURE	DESCRIPTION
Advanced Material Models	Includes nonlinear concrete, steel, rock, and soil behaviors
3D solid modeling	Supports solid, shell, and beam elements for detailed modeling
Contact and interface	Models interactions between soil and structures, joints, and more
Automatic meshing & remeshing	Simplifies and refines complex mesh generation
3D solid modeling	Supports solid, shell, and beam elements for detailed modeling

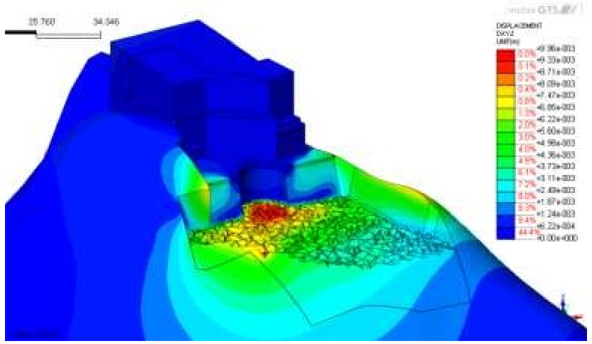
Maximizing Efficiency with MIDAS GTS NX

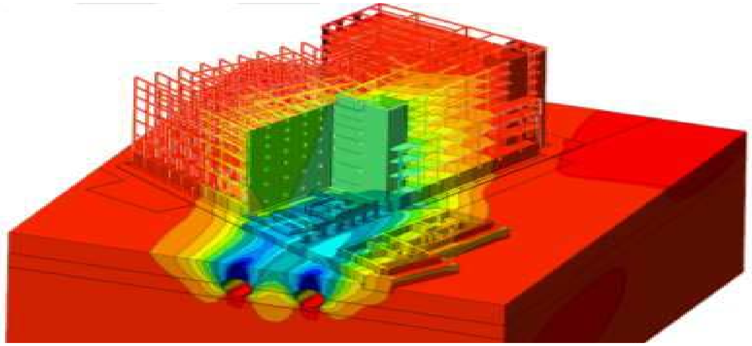
- Automated Meshing and Geometry Handling
- Advanced Nonlinear Solvers
- Integrated Modeling & Post-Processing
- Template-Based Modeling & Reusability
- Accurate Simulation Reduces Rework

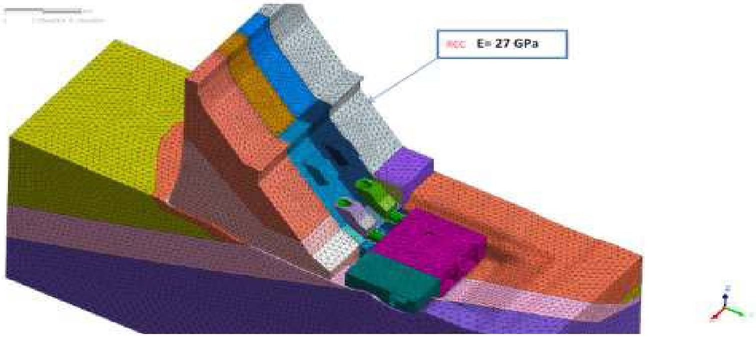
Our MIDAS commitment

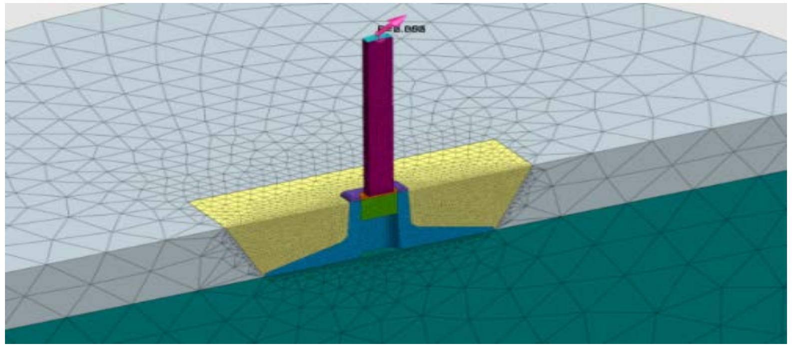
- **Committed to becoming experts in 2D/3D engineering**
- **Constantly providing technical support, materials, and training**

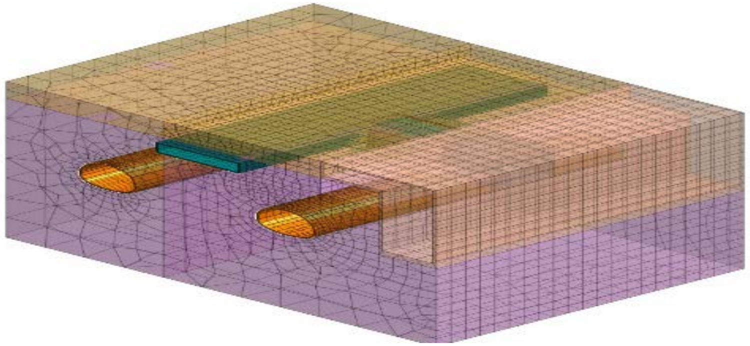








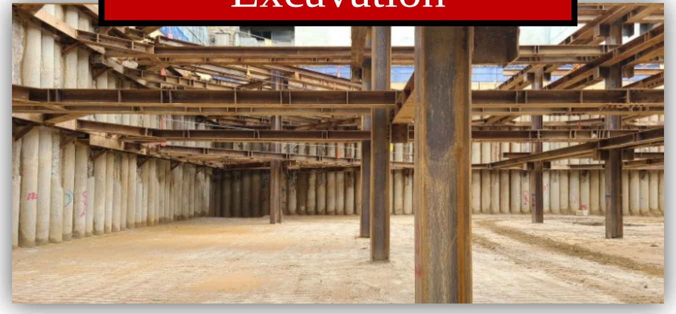




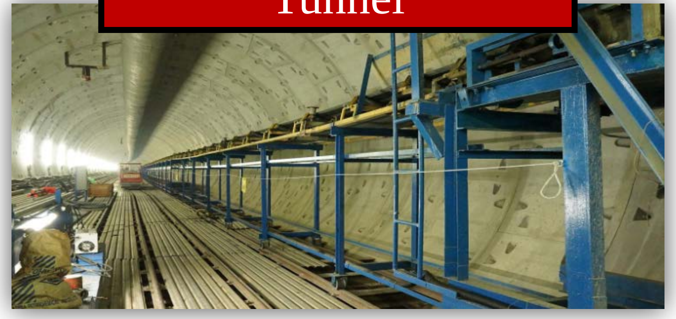
REAL-LIFE KEY APPLICATIONS

Pile

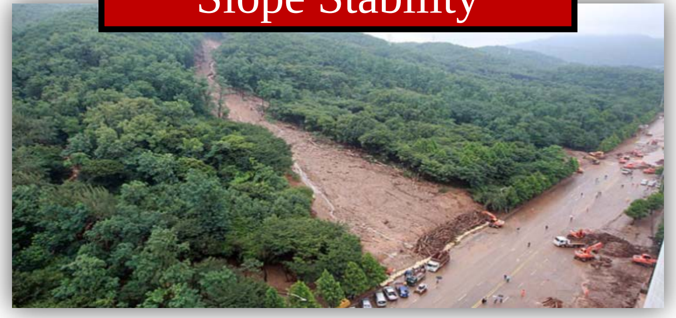
Excavation



Tunnel

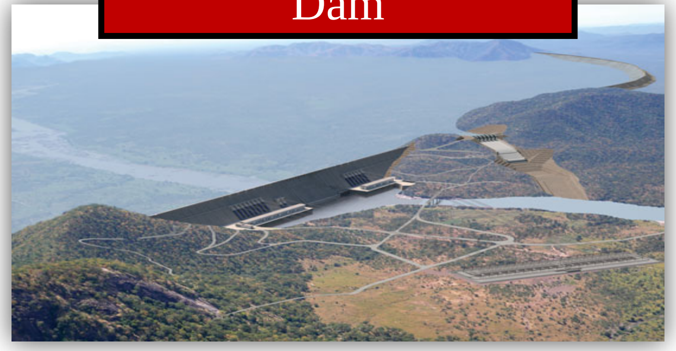


Slope Stability

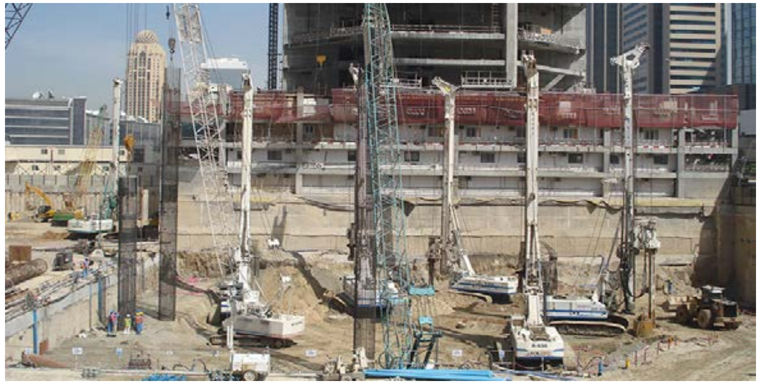


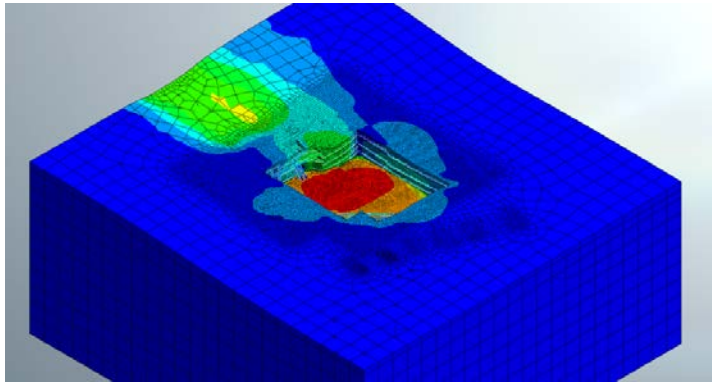
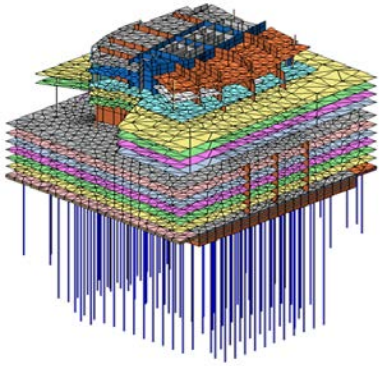
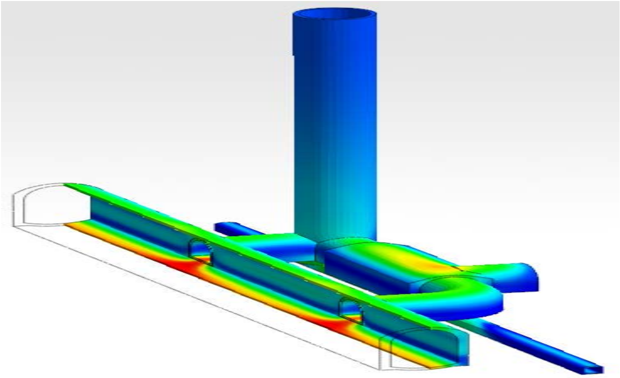
Soil-Structure Behavior

Dam



ADVANTAGES OF 3D

MULTIPLE TYPE OF ANALYSES and SUB ANALYSIS

Static Analysis

- Linear static analysis
- Nonlinear static analysis

Construction Stage Analysis

- Stress (drained/undrained) analysis
- Seepage analysis for each stage
- Stress-seepage-slope coupled
- Consolidation analysis for each stage
- Fully coupled stress & seepage

Consolidation Analysis

- Consolidation analysis

Stress-Seepage Coupled Analysis

- Stress
- Steady-state
- Transient
- Stress seepage

Seepage Analysis

- Steady state seepage analysis
- Transient seepage analysis

Dynamic Analysis

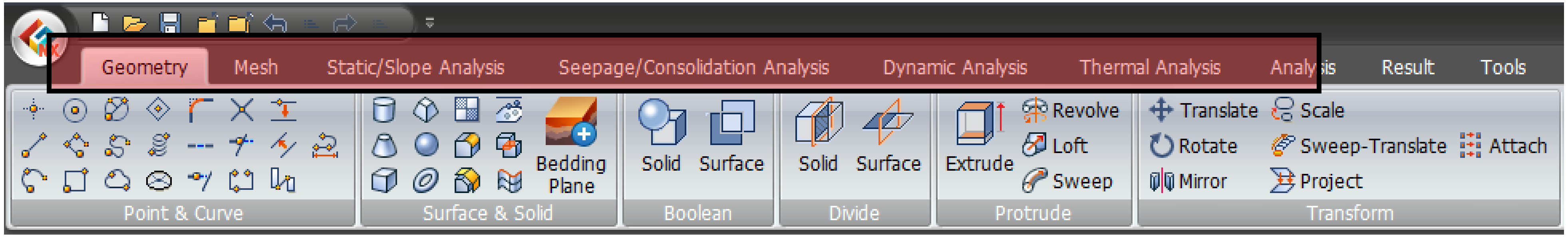
- Eigenvalue / Response Spectrum analysis
- Linear Time History (mode/direct methods)
- Nonlinear Time History analysis
- 1D/2D Equivalency Linear analysis
- Nonlinear time history + SRM Coupled

Slope Stability Analysis

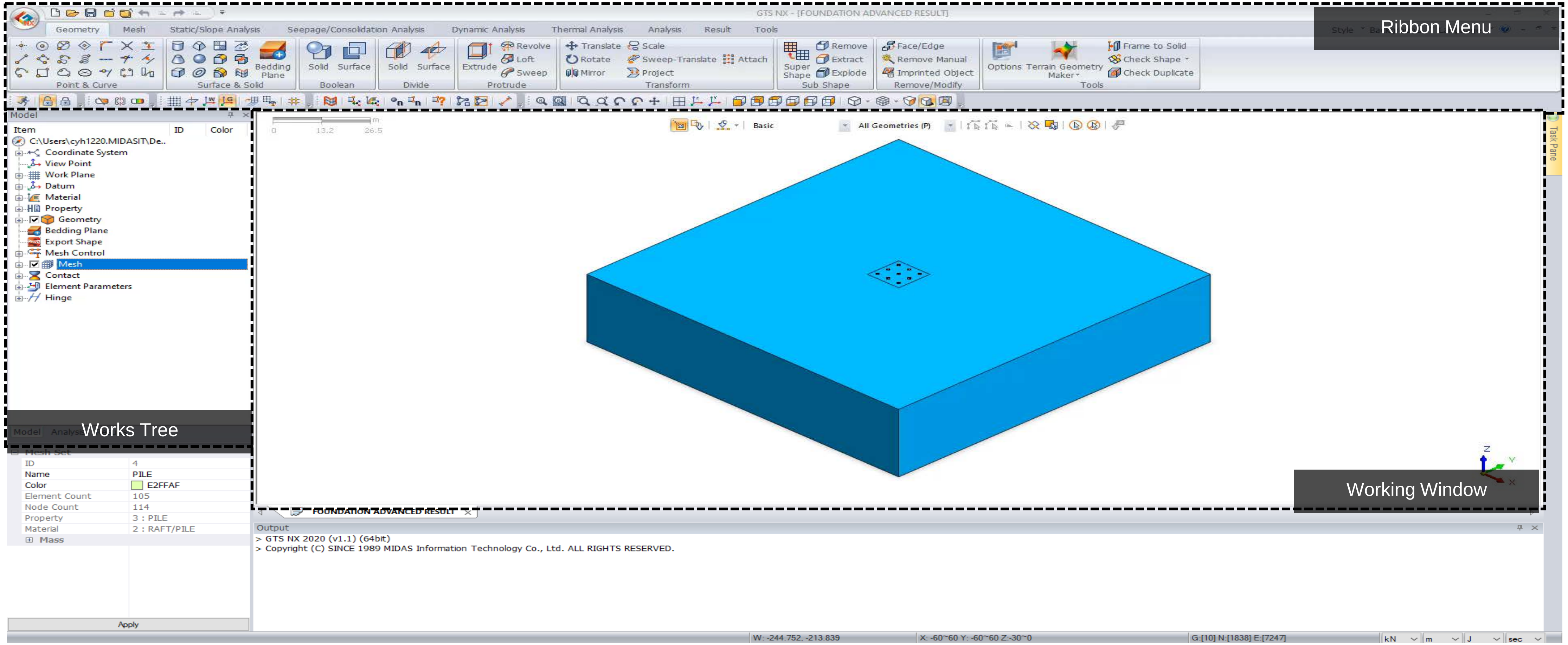
- Strength Reduction Method (SRM)
- Strength Analysis Method (SAM)
- Construction stages Slope stability (SRM/SAM)

DESIGN PROCESS

- Drawing geometries
- Making meshes
- Assigning boundary conditions / loads
- Running Analysis
- Checking Result



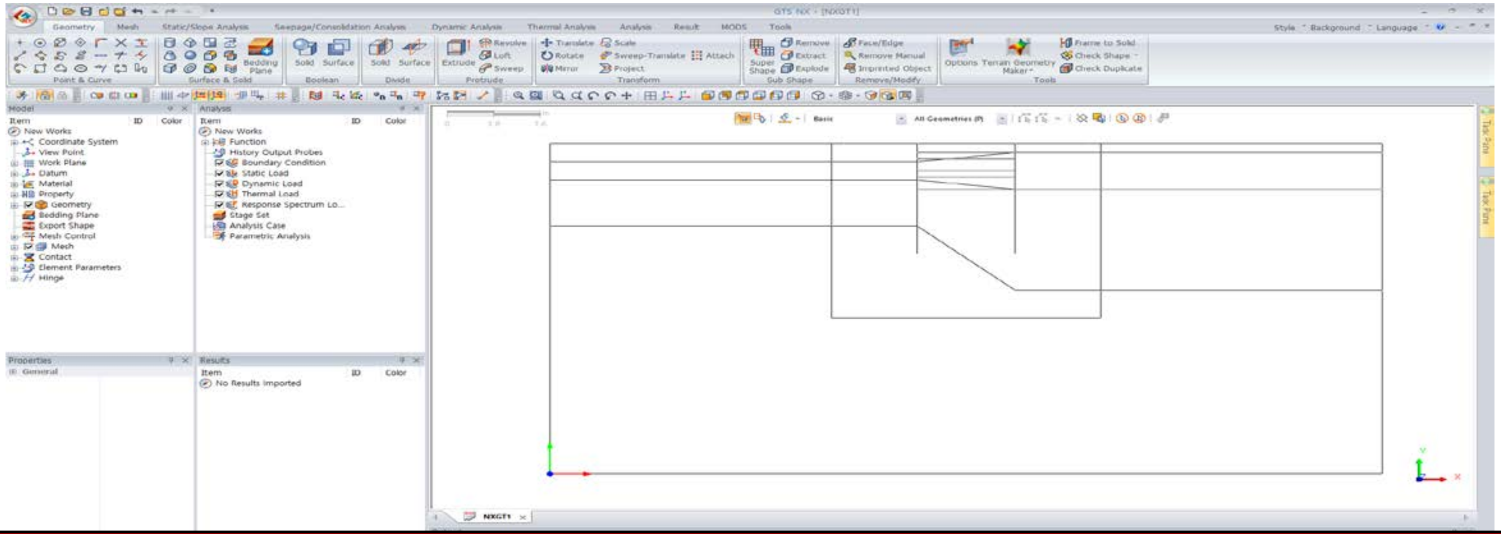
GRAPHICAL USER INTERFACE



COMPATIBILITY

Works with most drawing files / CAD formats

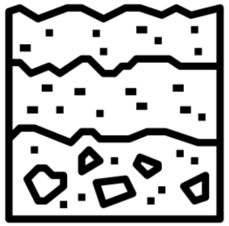
File format	Description
*.dwg	AutoCAD drawing files
*.dxf	AutoCAD drawing interchange files
*.x_t; *.xmt_txt; *.x_b; *.xmt_bin	Parasolid (9 to 29) files
*.sat; *.sab; *.asat; *.asab	ACIS (R1 to 2018 1.0) files
*.stp; *.step	STEP (AP203, AP214) files
*.igs; *.iges	IGES (Up to 5.3) files

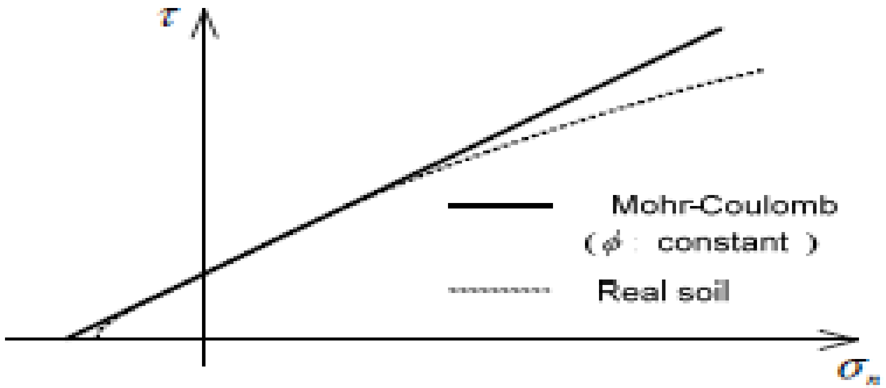


COMPREHENSIVE SOIL MODELS

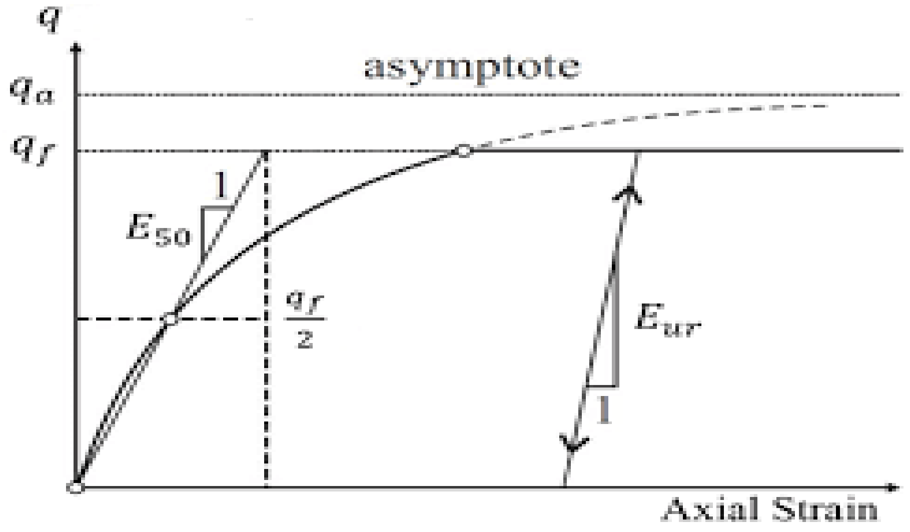
Choose the appropriate soil model for your various projects

General	Mohr-Coulomb Hardening Soil (small strain stiffness)
Sand	Modified UBCSAND PM4Sand
Clay	**Soft soil (Creep)** Modified Cam Clay Sekiguchi-Ohta(Inviscid) Sekiguchi-Ohta(Viscid) Generalized SCLAY1S
Rock	**(Generalized) Hoek Brown** **Jointed Rock Mass** CWFS





Mohr-coulomb



Hardening soil

COMPREHENSIVE ELEMENT LIBRARY

Saves time by just selecting the necessary elements from the database

1D Element

- Geogrid
- Truss
- Beam

2D Element

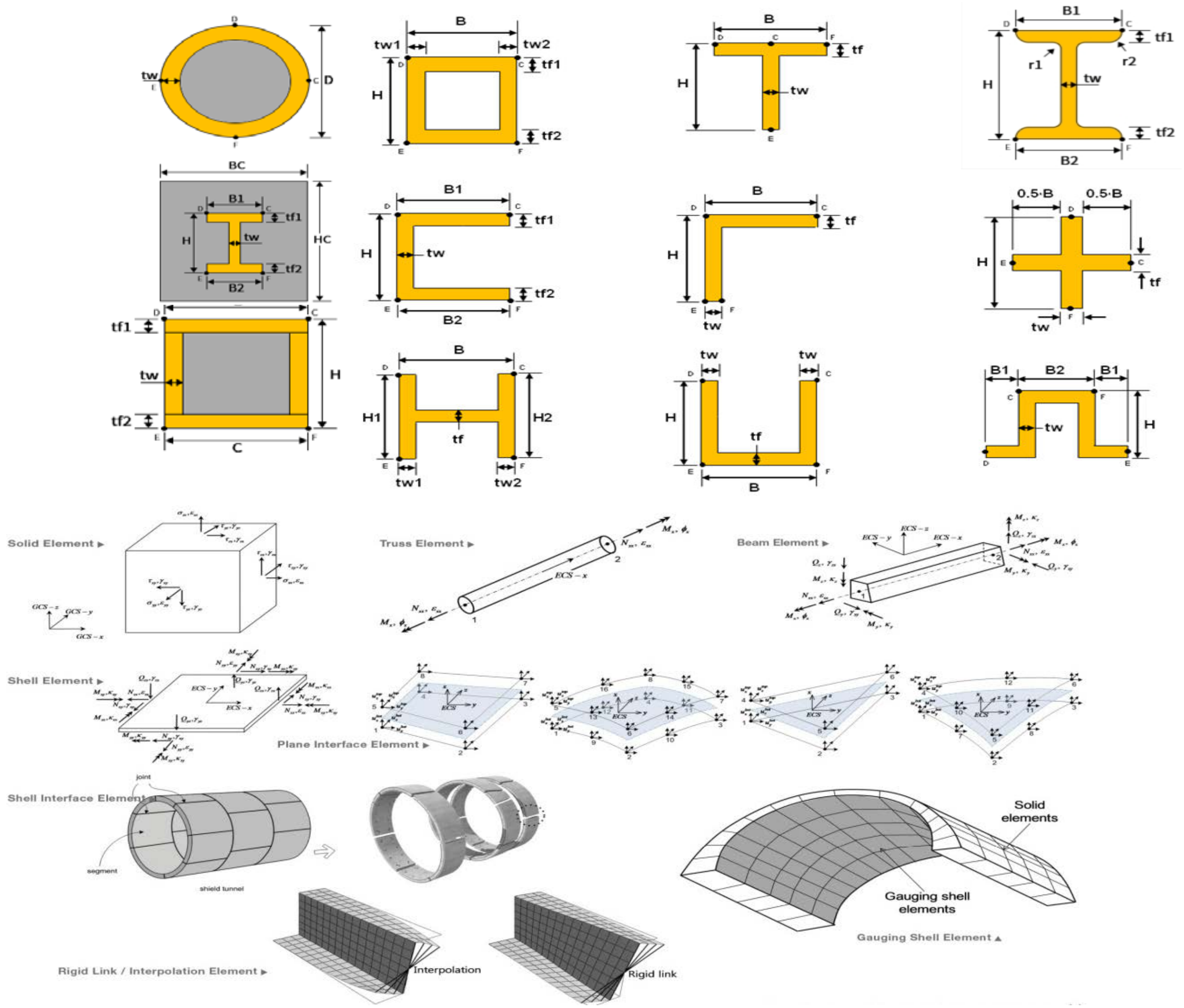
- Shell
- Gauging shell
- Plane stress
- Plane strain
- Geogrid
- Axisymmetric

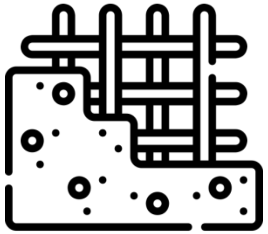
3D Element

- Solid

Others

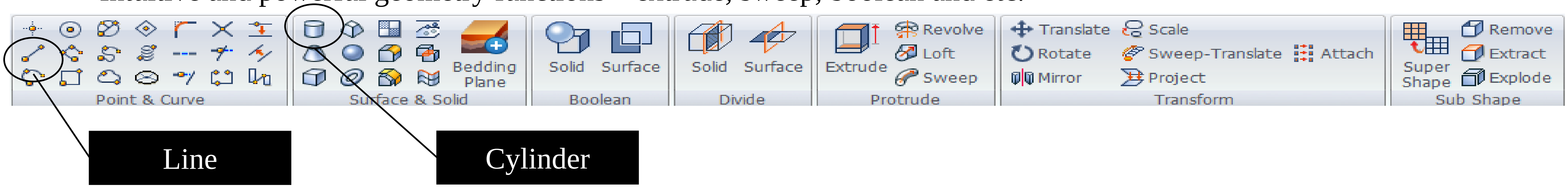
- Interface
- Elastic / Rigid link
- Pile interface / Pile tip
- User specified behavior

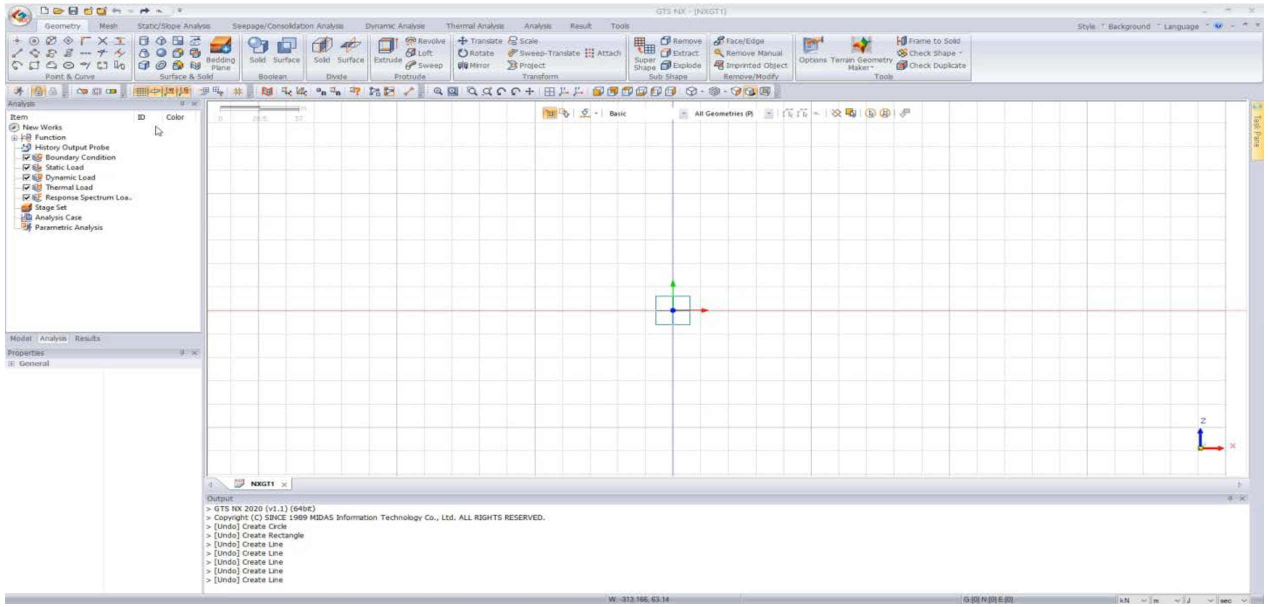




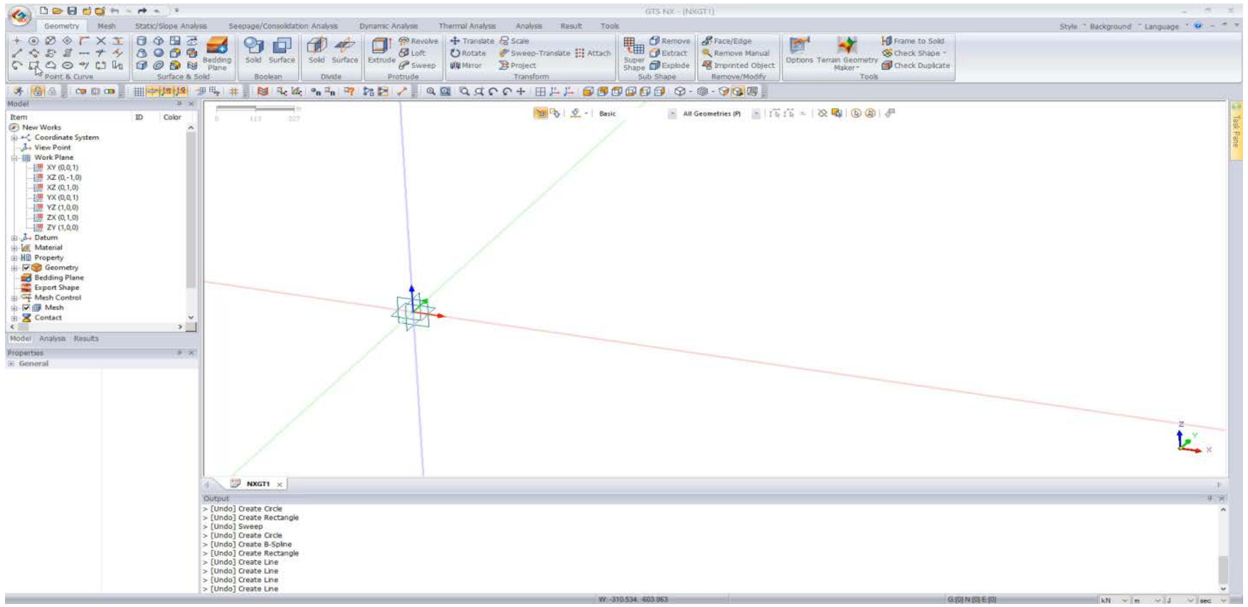
GEOMETRY

- From simple drawing modification to complex building up of geometries
- Intuitive and powerful geometry functions – extrude, sweep, boolean and etc.





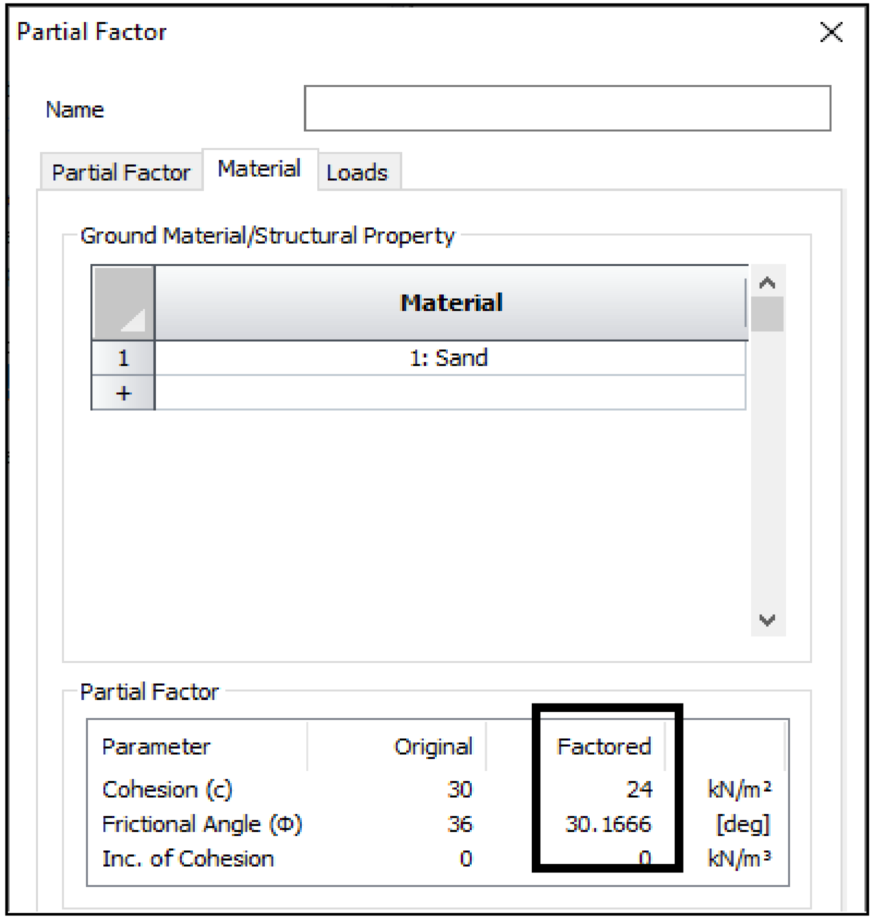
Line / Square / Circle



Extrude / Sweep

PARTIAL FACTORS

2D analysis can use the **partial factor** function which was developed based on Euro Code 7.



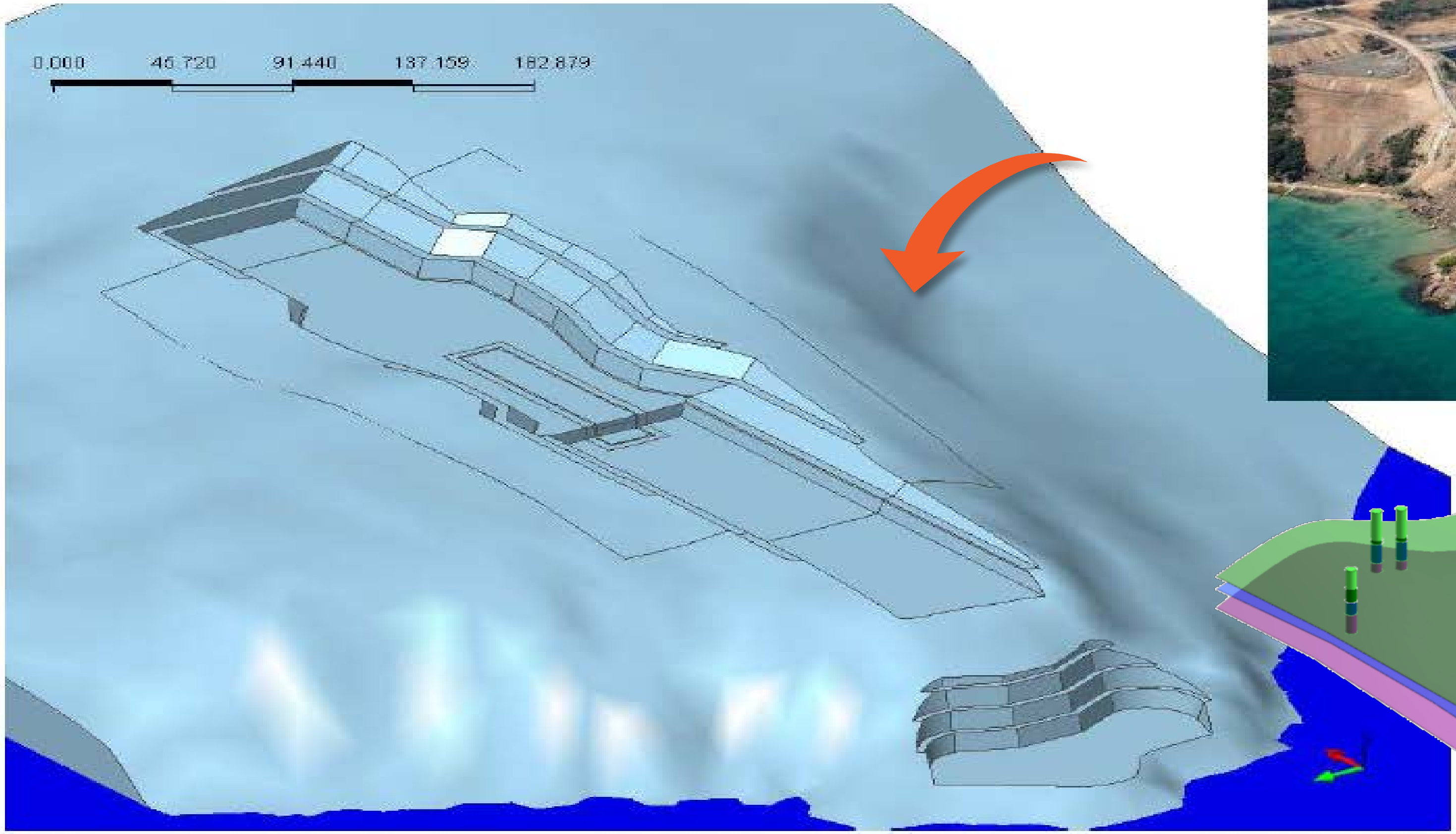
GTS NX is providing the database for this partial factor as below:

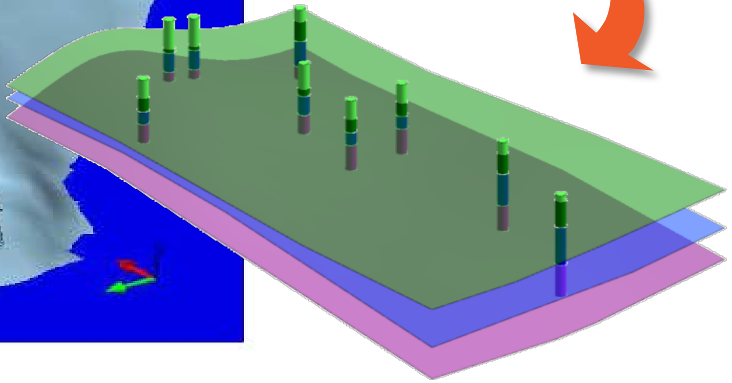
- Design Approach 1 Combination 1
- Design Approach 1 Combination 2
- Design Approach 2
- Design Approach 3...

Directly checking the original and factored parameters

TGM & Bedding plane wizard

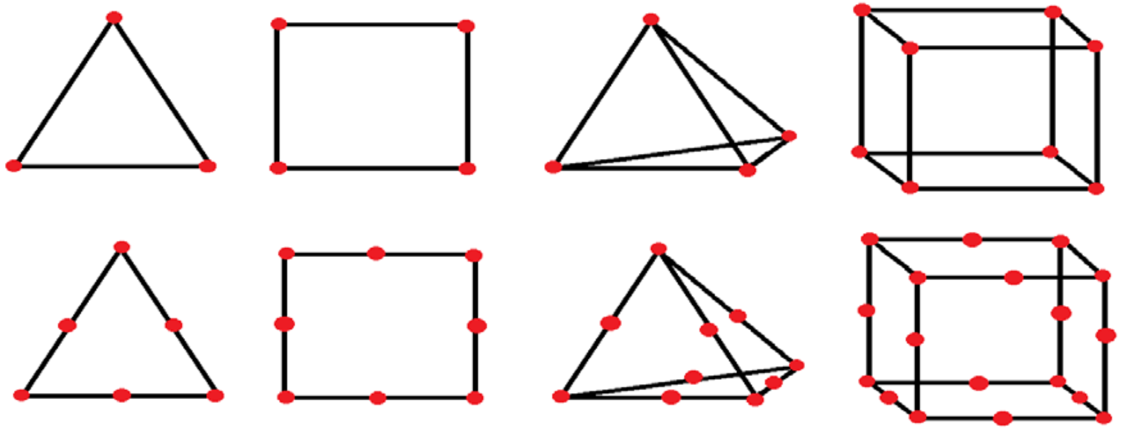
Easily create the surface of the site by simple topography import

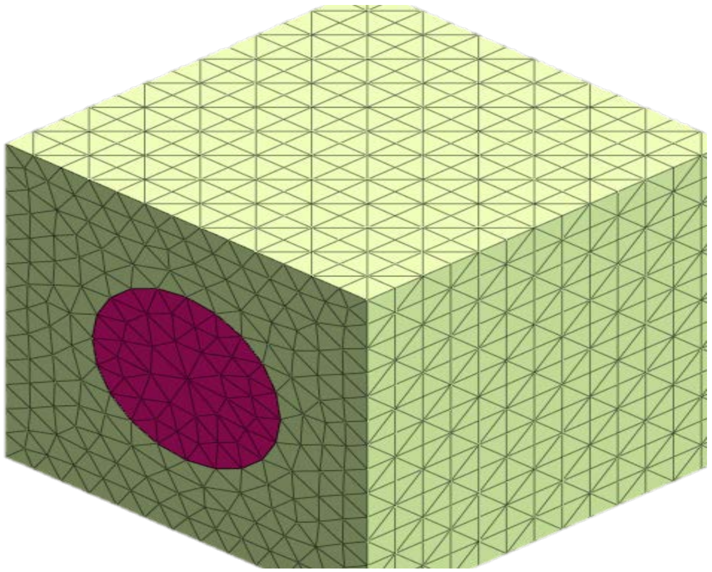




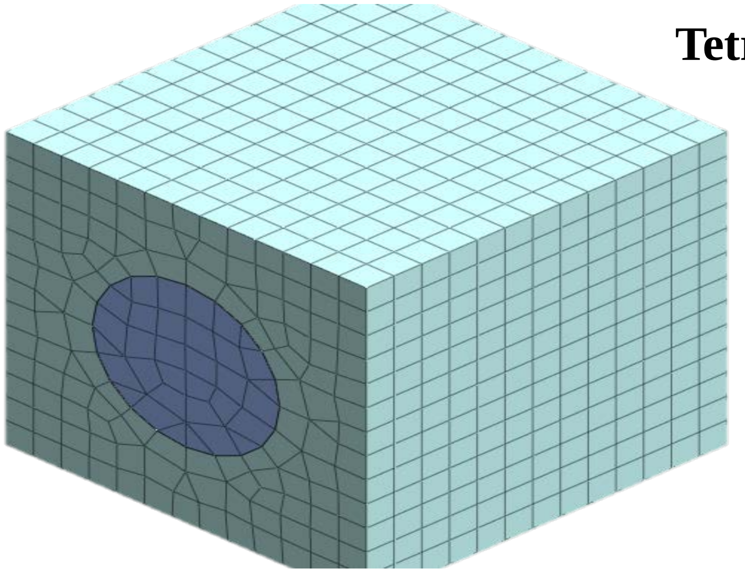
MESH

Powerful meshing algorithm with Hybrid technology

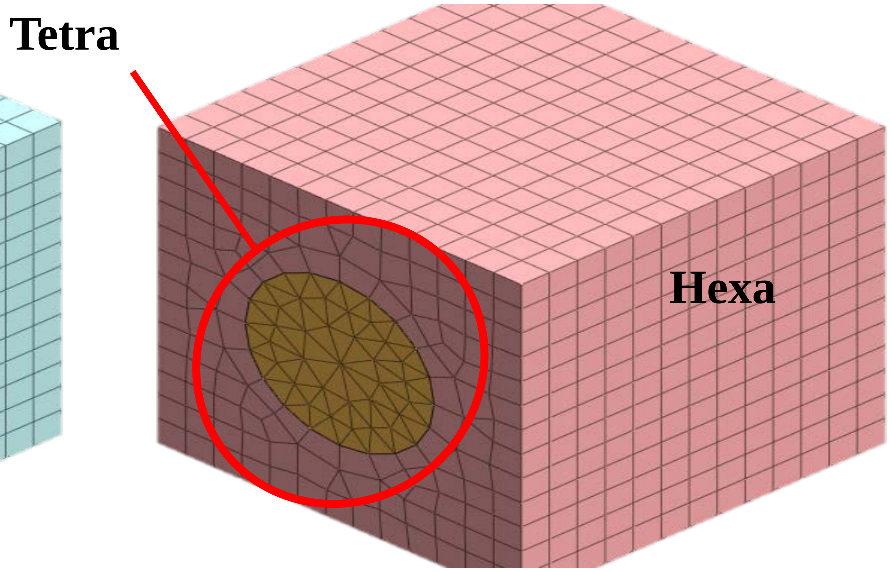




Tetrahedral



Hexahedral

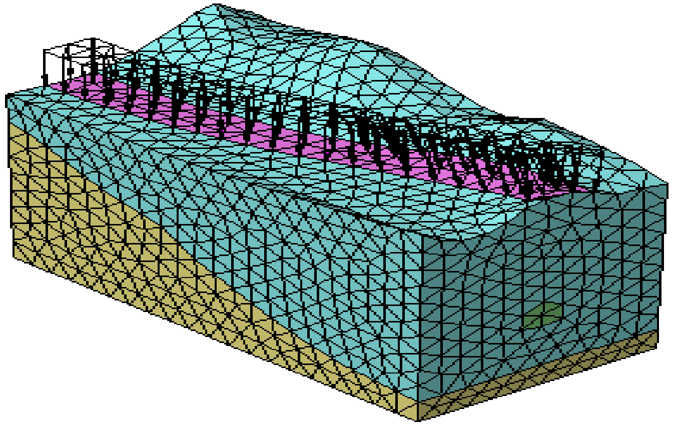


Tetra + Hexa(hybrid)

BOUNDARY CONDITION AND LOADS

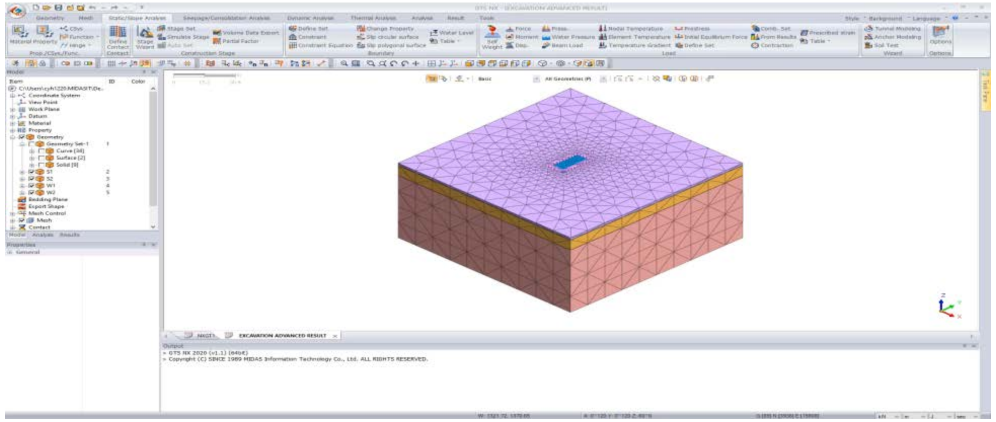
Boundary

- Constraint
- Change Properties
- Review
- Water level
- Nodal Head
- Surface Flux
- Slip Circle/Polygonal Surface
- Draining Condition
- Non Consolidation
- Transmitting



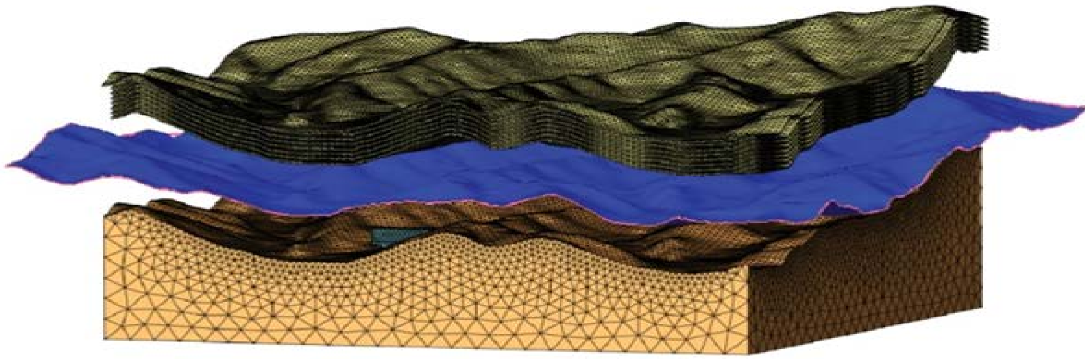
Loads

- Self Weight
- Force
- Moment
- Displacement
- Pressure(Surcharge / Water)
- Line Beam Load
- Element Beam Load
- Temperature
- Pre-stress
- Contraction
- Initial Equilibrium Force
- Combined Load
- Response Spectrum
- Ground Acceleration
- Time Varying Static
- Dynamic Nodal / Surface
- Load to Mass
- Train Dynamic Load Table

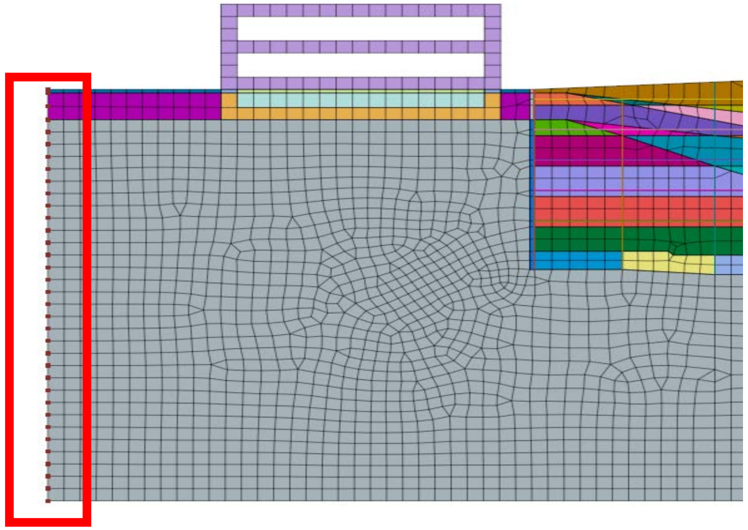


WATER CONDITION

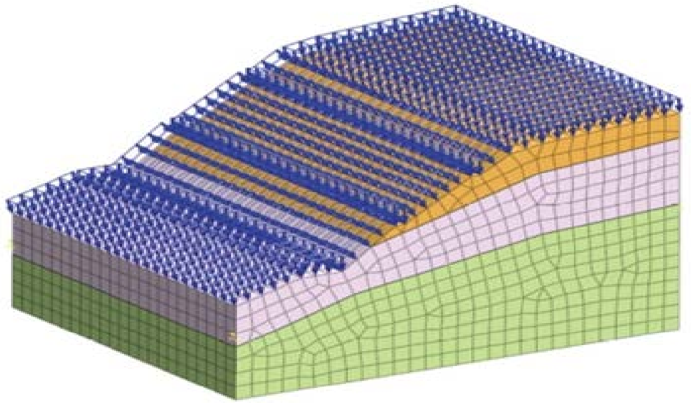
Nodal head, line & surface flux, water level



3D water level automatic generation



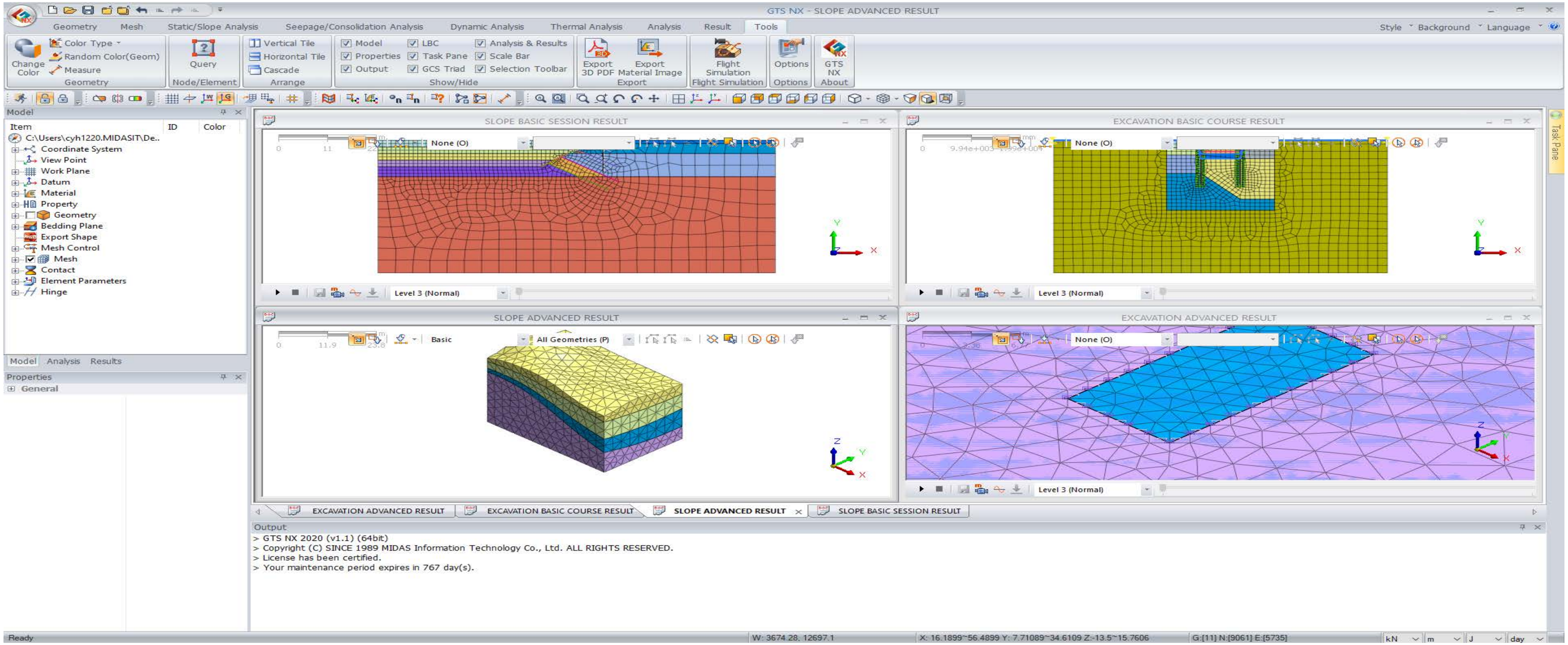
Nodal head for water level



Rainfall intensity input

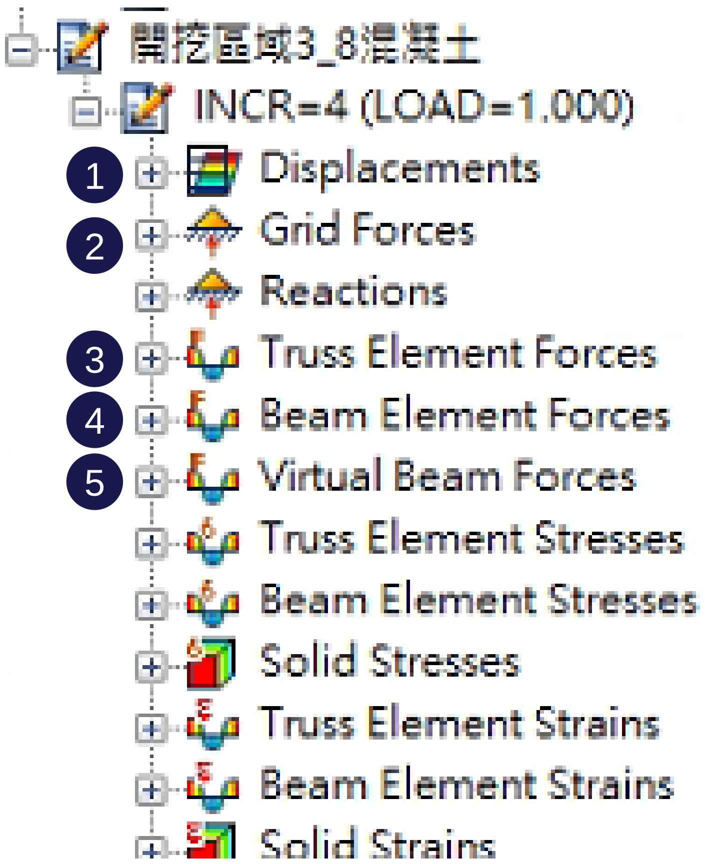
MULTI WINDOWS

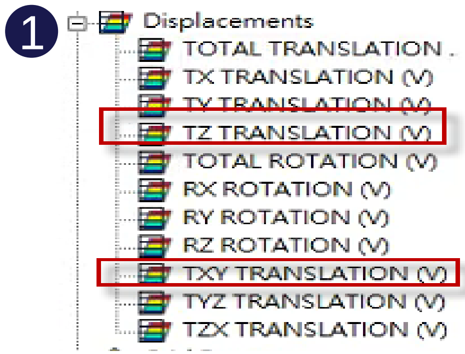
Compare various sections or different analyses in one program window



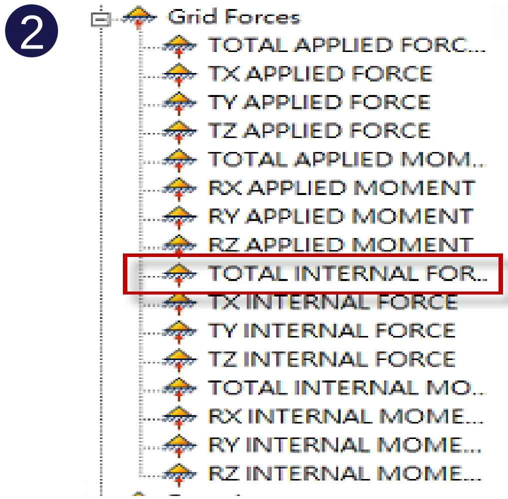
MIDAS

COMPREHENSIVE BREAKDOWN OF RESULTS

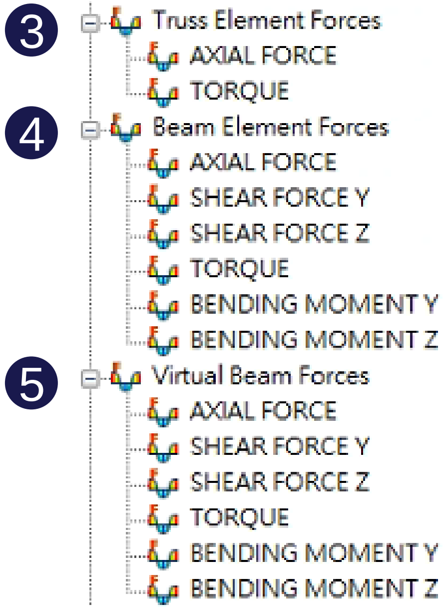




水平變位TXY
垂直變位TZ



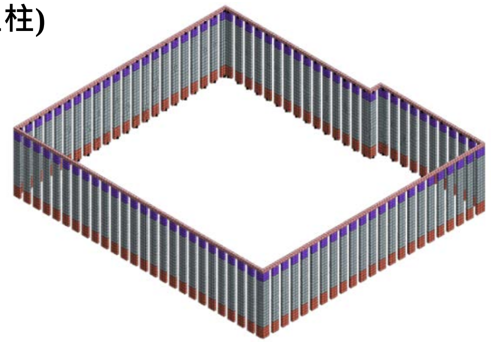
內力
Grid Force

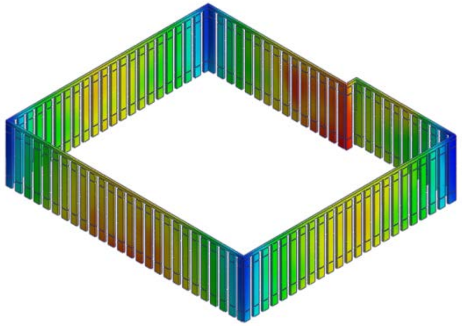


軸力
Axial Force

RESULT - VISUALIZATION

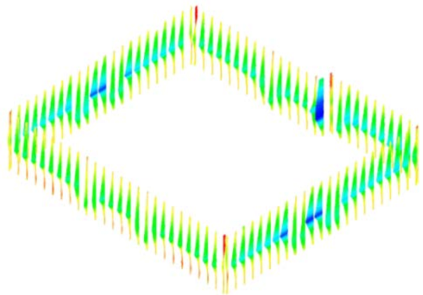
實體元素
(擋土柱)

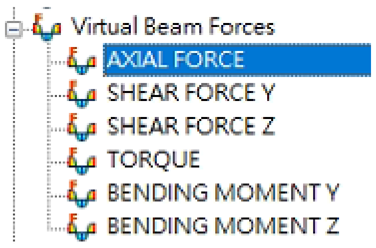




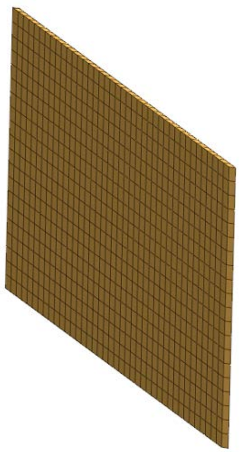
如何計算等效軸力?

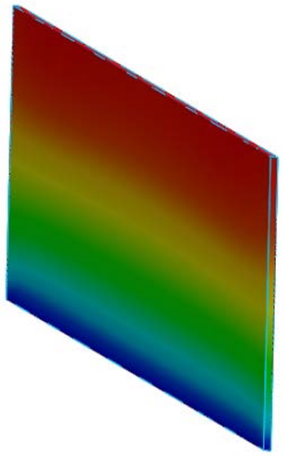
虛擬梁(Virtual Beam)





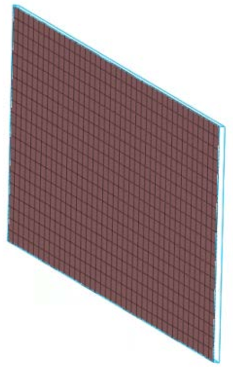
實體元素

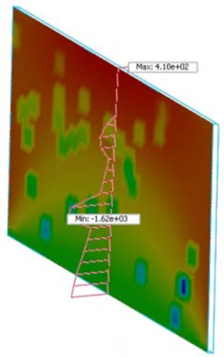


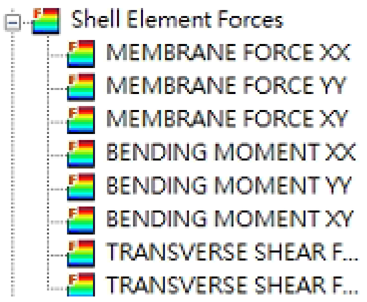


如何得到受力方向大小?

測量板(Gauging Shell)

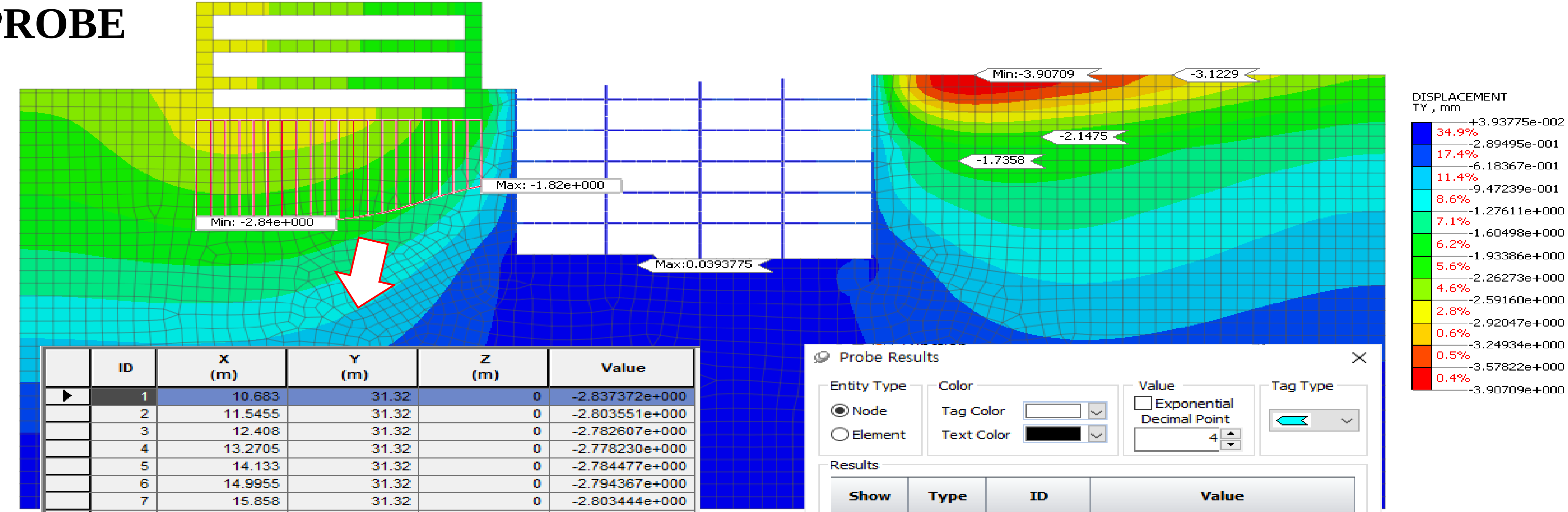






RESULT - MULTIPLE METHODS FOR VERIFICATION

PROBE



ID	X (m)	Y (m)	Z (m)	Value
1	10.683	31.32	0	-2.837372e+000
2	11.5455	31.32	0	-2.803551e+000
3	12.408	31.32	0	-2.782607e+000
4	13.2705	31.32	0	-2.778230e+000
5	14.133	31.32	0	-2.784477e+000
6	14.9955	31.32	0	-2.794367e+000
7	15.858	31.32	0	-2.803444e+000
8	16.7205	31.32	0	-2.806896e+000
9	17.583	31.32	0	-2.799891e+000
10	18.4455	31.32	0	-2.781048e+000
11	19.308	31.32	0	-2.748633e+000
12	20.1705	31.32	0	-2.701343e+000
13	21.033	31.32	0	-2.638357e+000
14	21.8955	31.32	0	-2.559416e+000
15	22.758	31.32	0	-2.462966e+000
16	23.6205	31.32	0	-2.353056e+000
17	24.483	31.32	0	-2.235005e+000
18	25.3455	31.32	0	-2.114296e+000
19	26.208	31.32	0	-1.999796e+000
20	27.0705	31.32	0	-1.900160e+000
21	27.933	31.32	0	-1.815394e+000

Probe Results

Entity Type: Node / Element
Color: Tag Color, Text Color
Value: Exponential, Decimal Point 4
Tag Type

Results

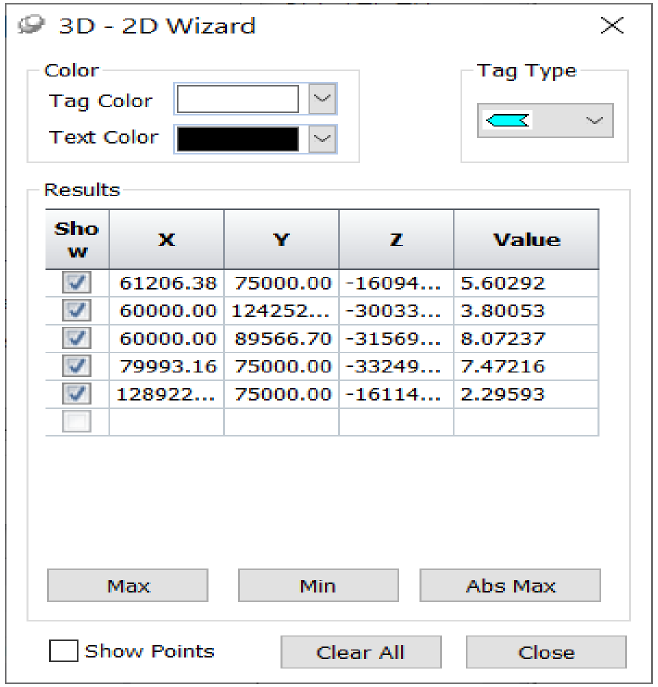
Show	Type	ID	Value
☑	Node	6838	-1.6844
☑	Node	6855	-3.5239
☑	Node	6328	-4.2087
☑	Node	6318	-4.2110
☑	Node	6308	0.8818
☑	Node	6289	0.9327
☐			

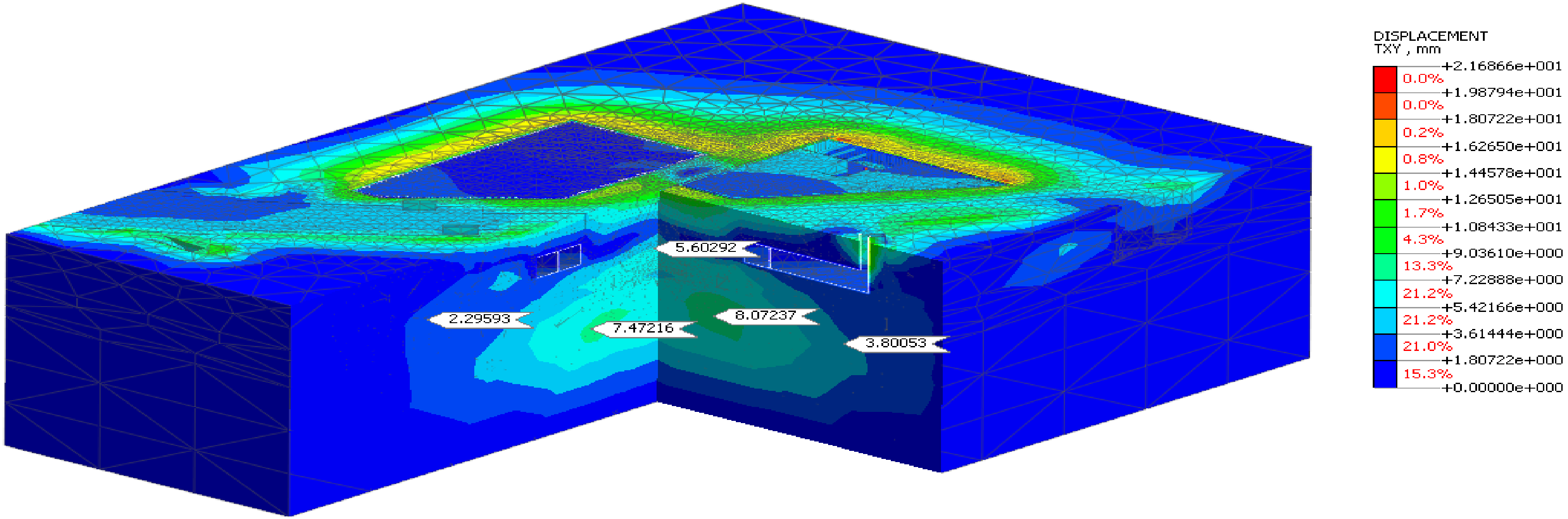
Max | Min | Abs Max | Clear All

☐ Min/Max Value of Each Part

Close

RESULT - MULTIPLE METHODS FOR VERIFICATION
PLANE CLIPPING and PROBING





RESULT - ANALYSIS

REPORT and EXPORT

materials and properties with "PDF" format

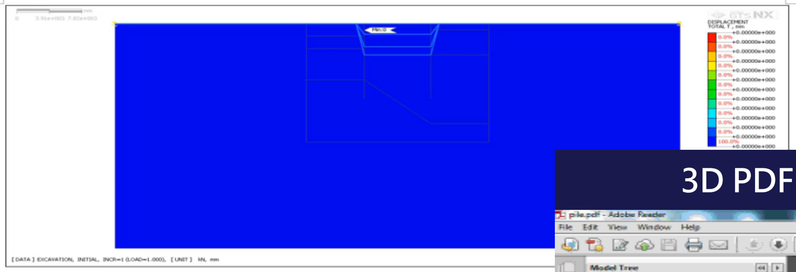
Material

Elastic

Name	E (kN/mm²)	Inc. of Elastic (kN/mm³)	Inc. of E Ref. Height (mm)	ν	γ (kN/mm³)	Ko	Thermal Coeff. (1/[T])	Molecular Vapor Diffusion Coeff. (mm/sec²)	Thermal Diffusion Enhancement	Damping Ratio
	γ_sat (kN/mm³)	e_o	kx (mm/sec)	ky (mm/sec)	kz (mm/sec)	Ss (1/mm)	Conductivity (W/(mm·[T]))	Specific Heat (J/(ton·[T]))	Heat Gen. Factor	
5:Conc'	28	0	0	0.15	2.4e-008	-	1e-006	0	0	0.05
	2.1e-008	0.5	0.01	0.01	0.01	5.2302133 3e-009	0			
6:Steel	205	0	0	0.15	7.4e-008	-	1e-006			
	2.1e-008	0.5	0.01	0.01	0.01	5.2302133 3e-009	0			

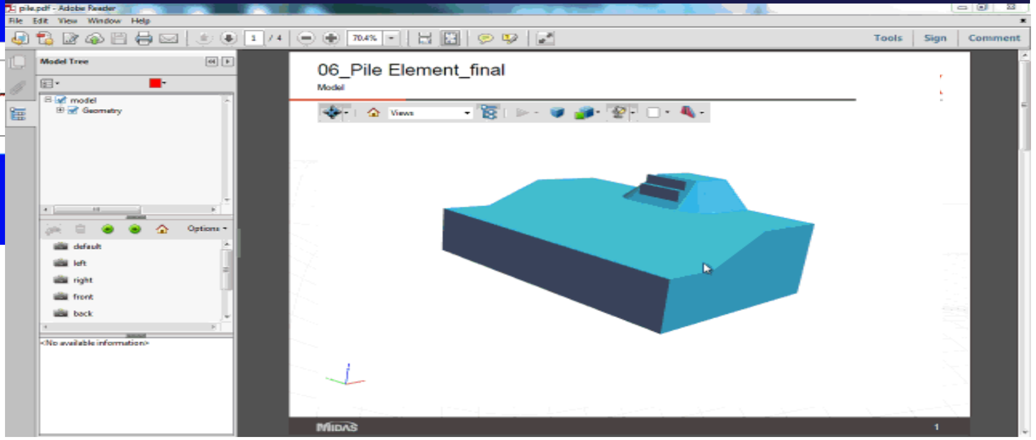
MIDAS

Results export with "WORD" format



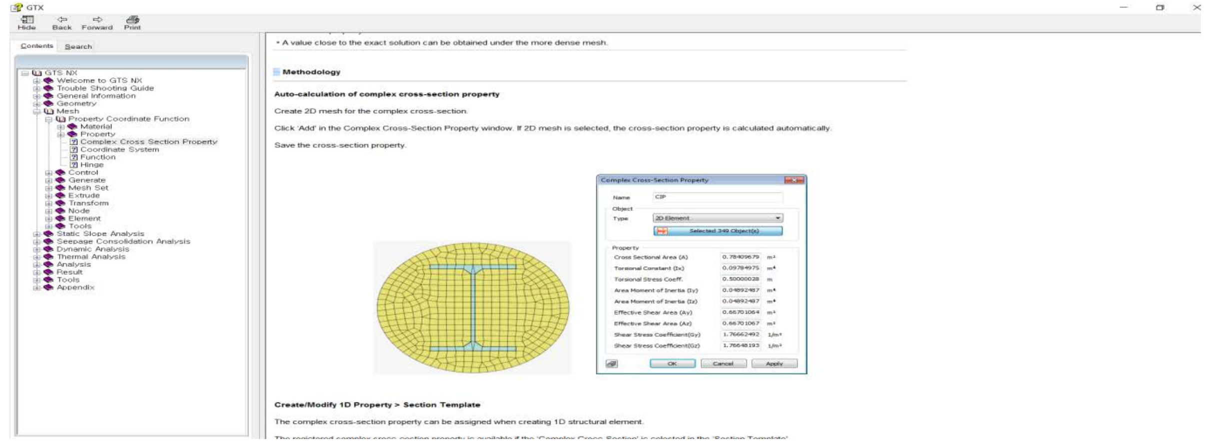
EXCAVATION_INITIAL_INCR=1 (LOAD=1.000)_Displacements_
(V)_Current.png

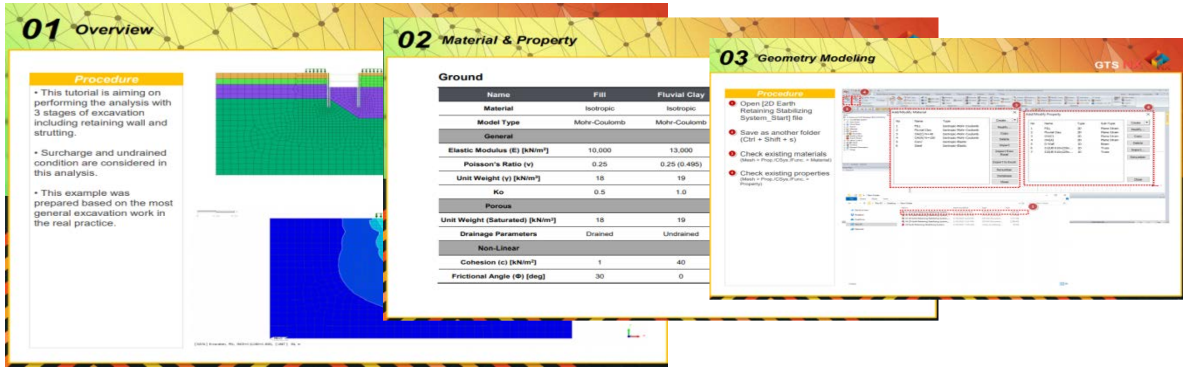
3D PDF Directly checking every results from 3D PDF



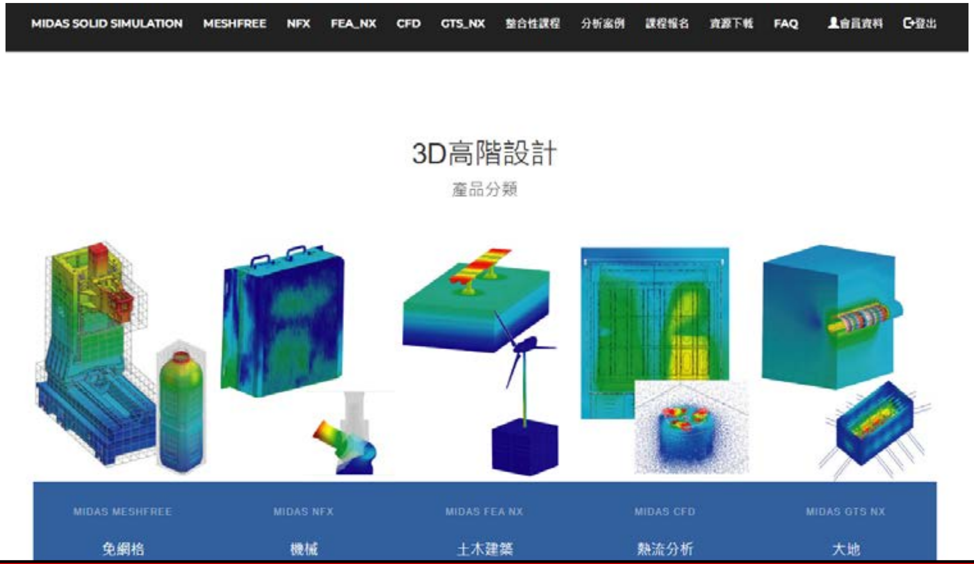
TECHNICAL SUPPORTS

Help manuals / tutorials with various topics





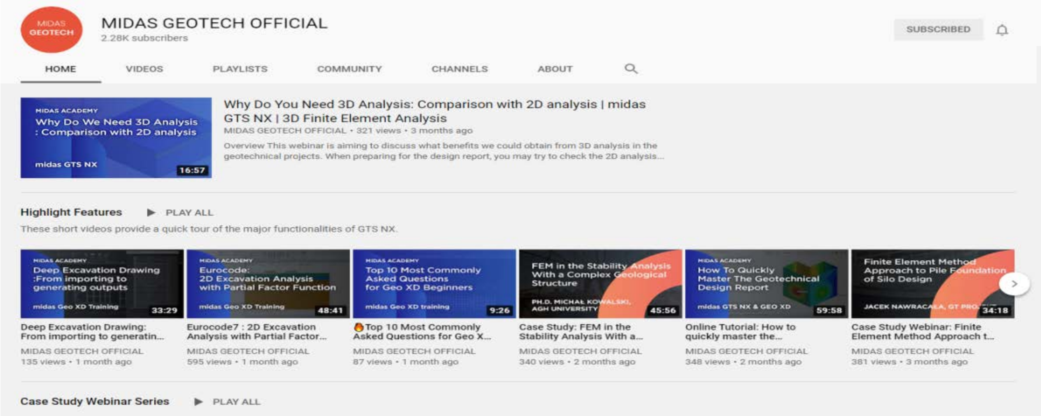
Taiwan MIDAS Solid-Simulation website

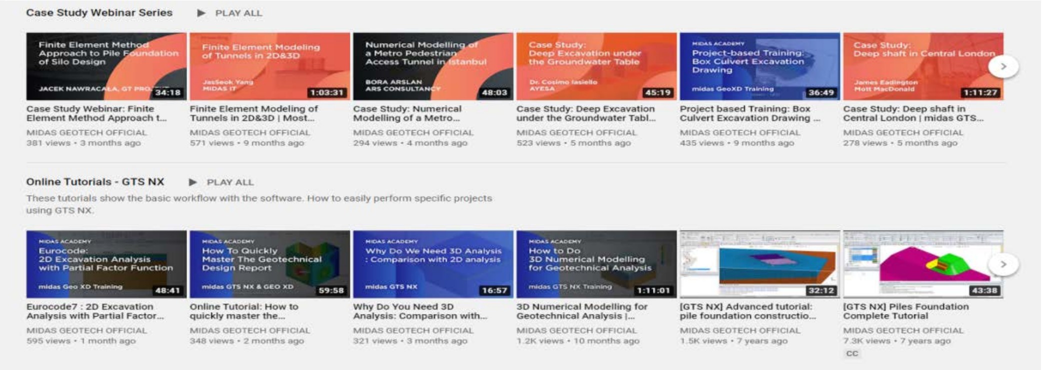


https://www.midasuser.com.tw/SolidSimulation/

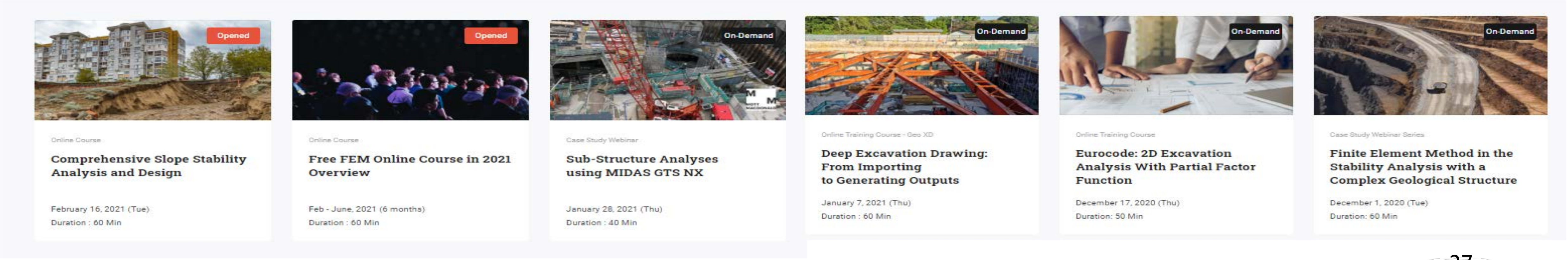
TECHNICAL SUPPORTS

Online training videos





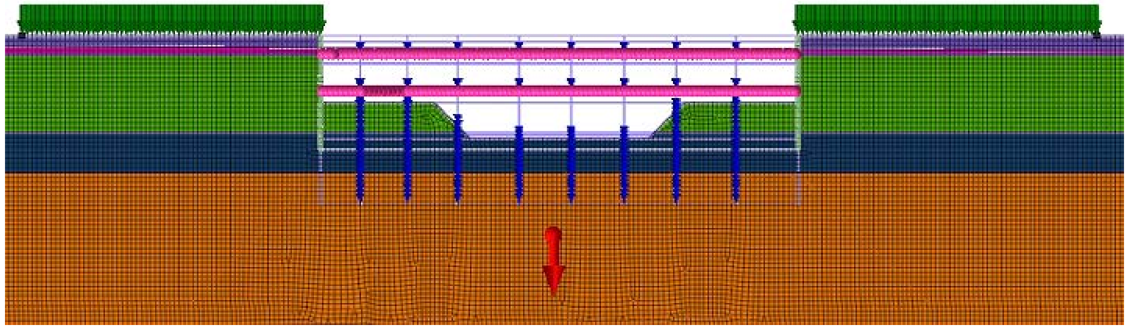
Online Courses / Case studies



MIDAS Team has a rich experiences and know-hows to help practicing engineers

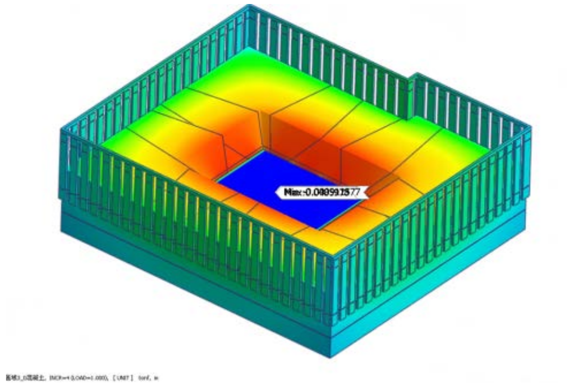
CASE STUDIES

2D分析

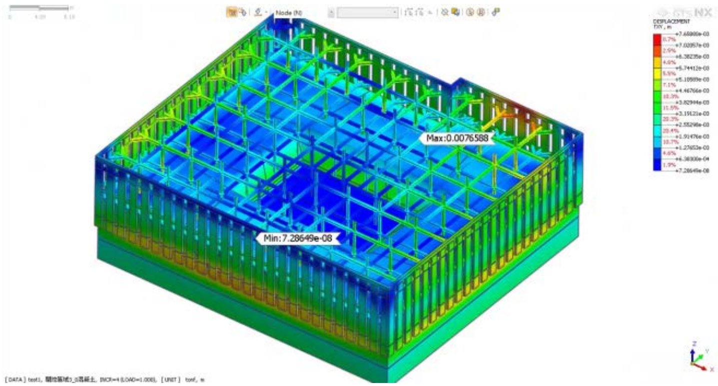


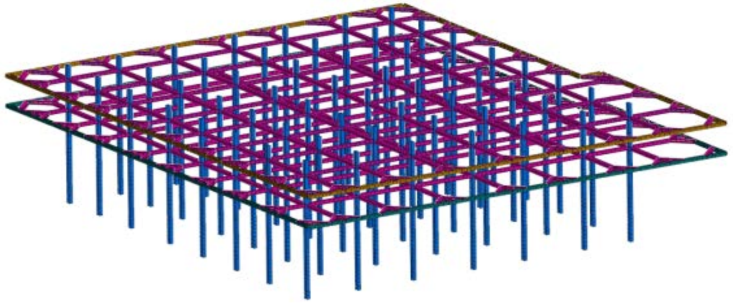
2D 建模只考慮模型的一個平面，縱向支撐、水平支撐和傾斜梁無法建模。

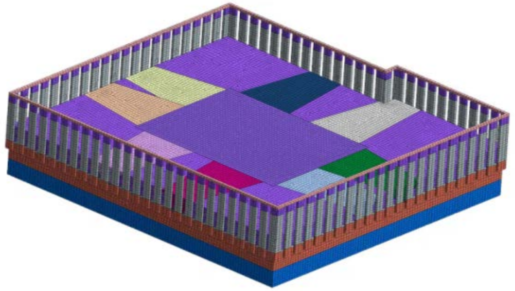
開挖面最大垂直位移 (m)

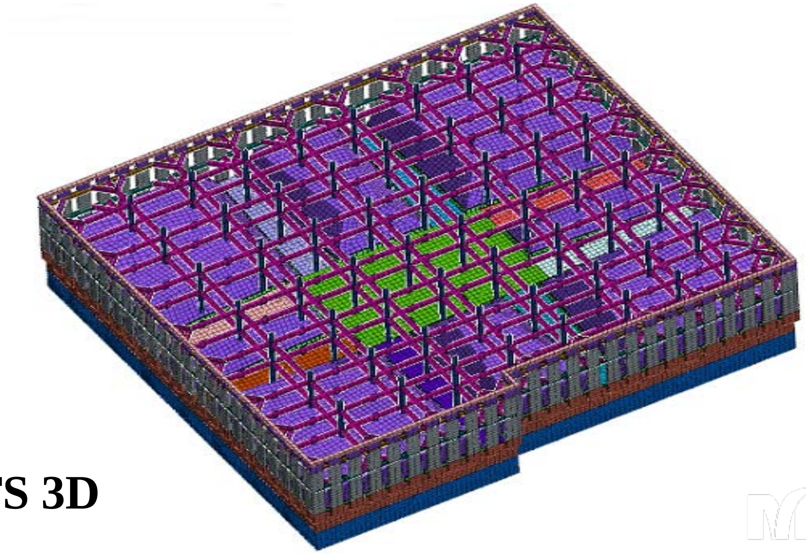


開挖面最大水平位移 (m)





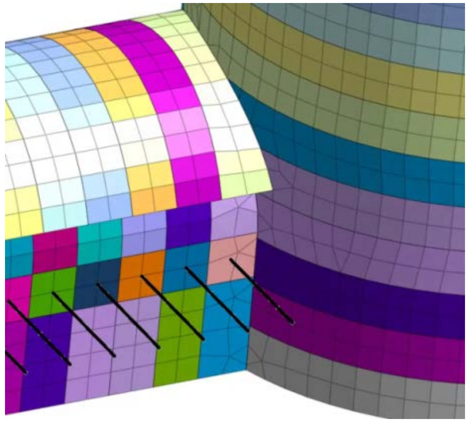


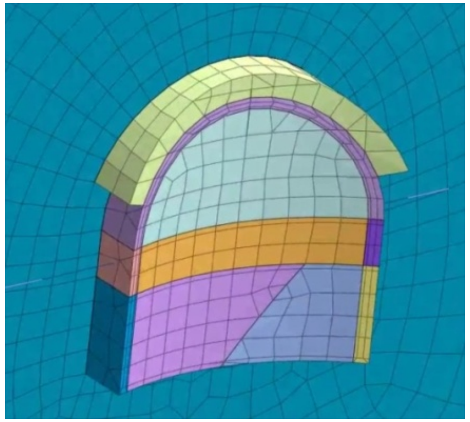


GEN+GTS 3D

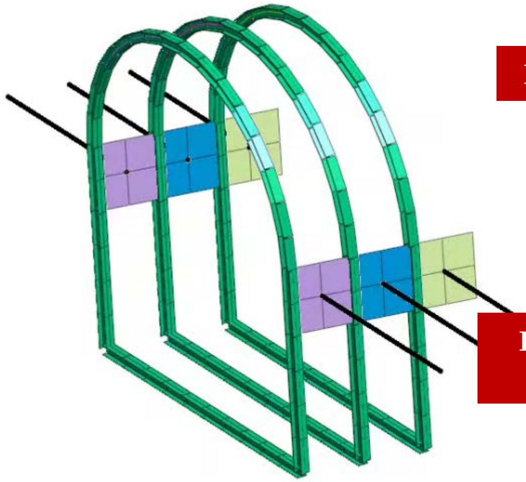
GTS NX NATM實例

Mixed Mesh with Fully Compatible Nodes





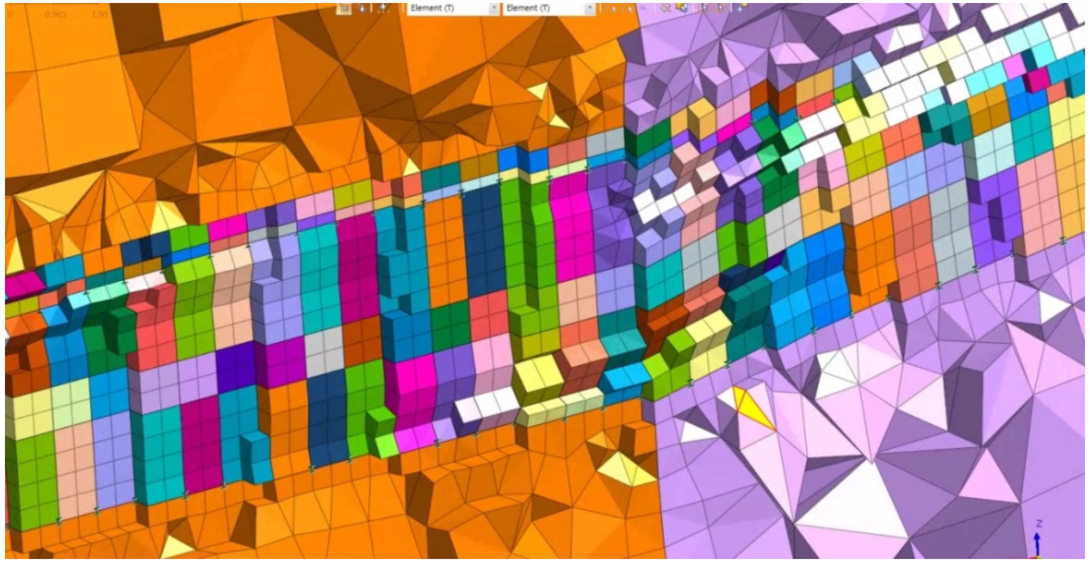
Shell Element(傳力裝置)



Beam Element(H型鋼)

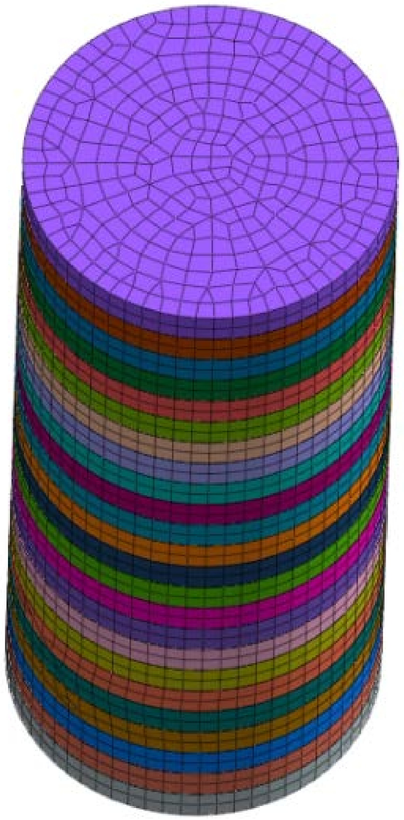
Embedded Truss Element
(Rock Bolt)

FEA NX 混合網格/全共點建模

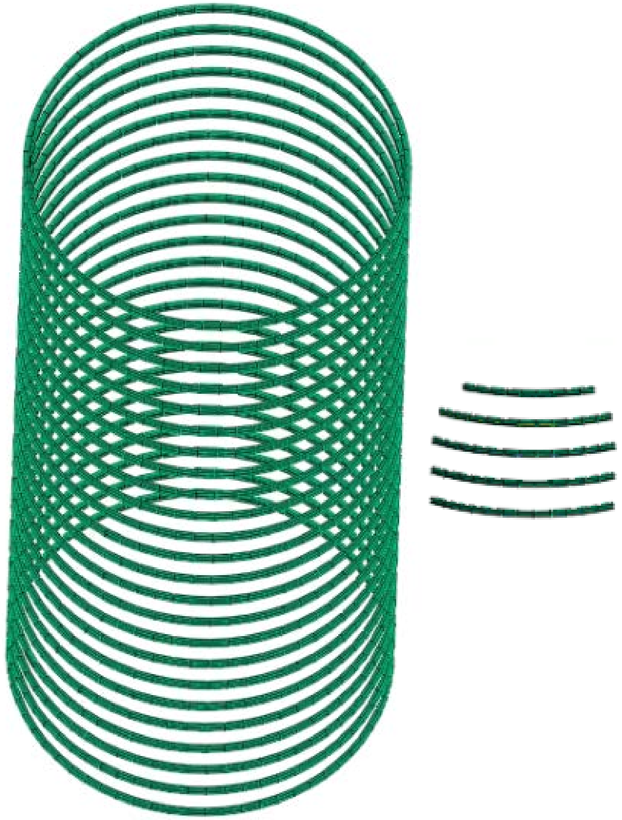


FEA NX Case Study: NATM Tunnel Analysis

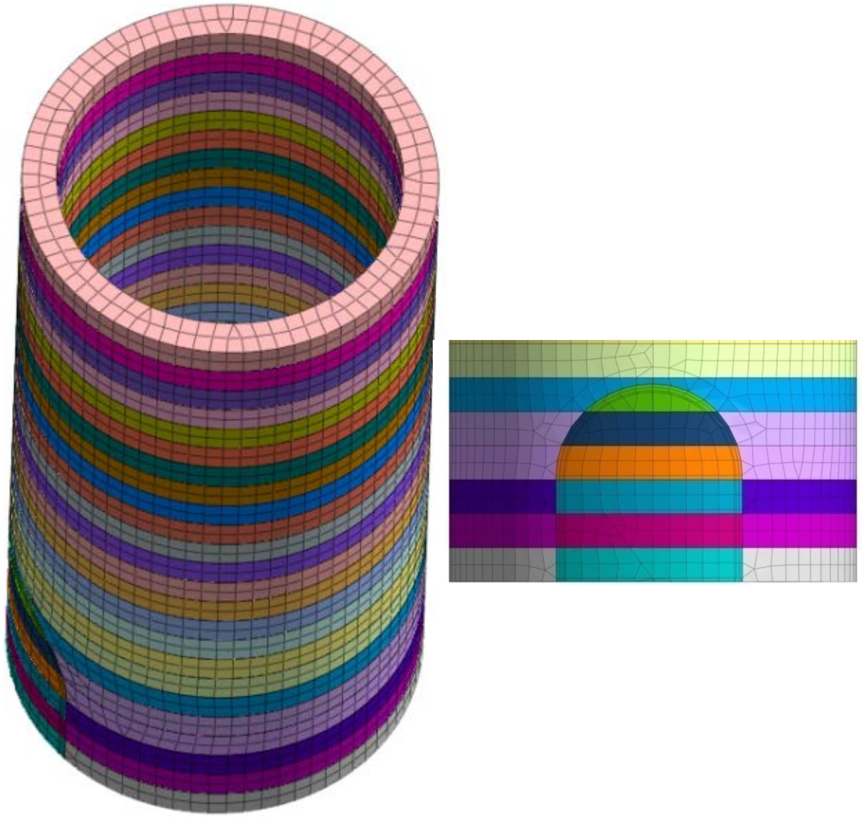
VERTICAL SHAFT



Vertical shaft excavation



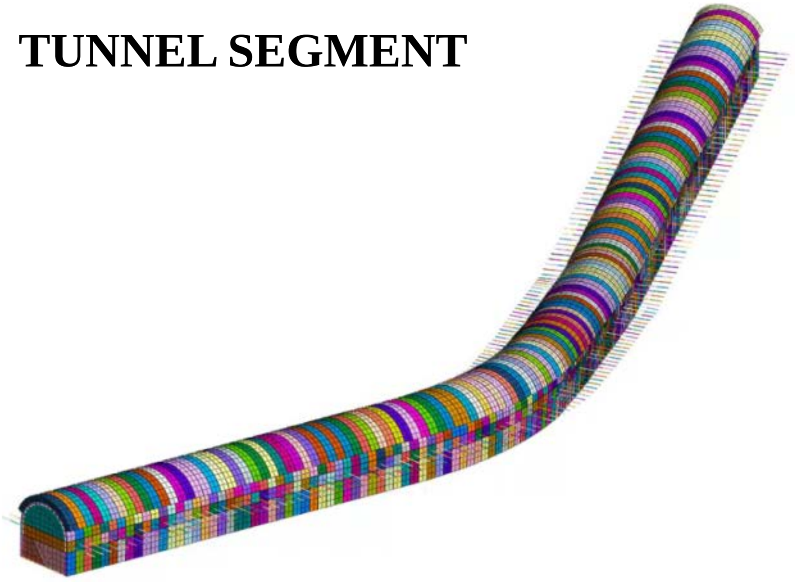
Steel Bracing



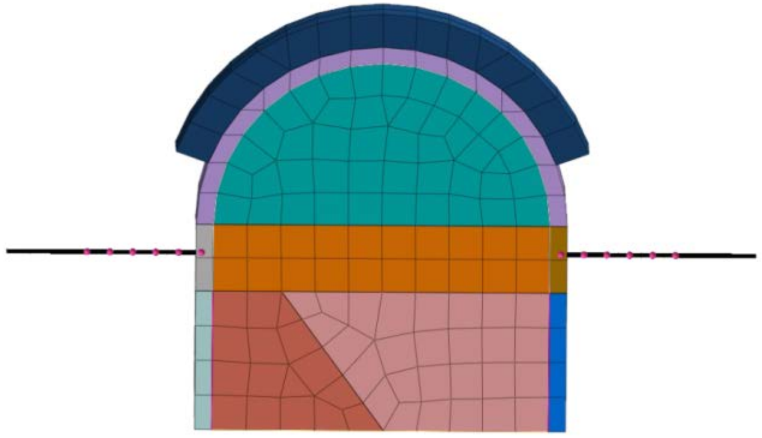
Backfill

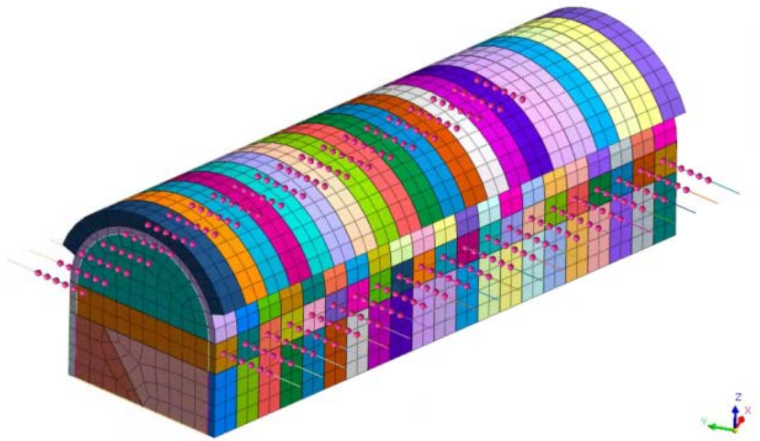
GTS NX Case Study: NATM Tunnel Analysis

TUNNEL SEGMENT

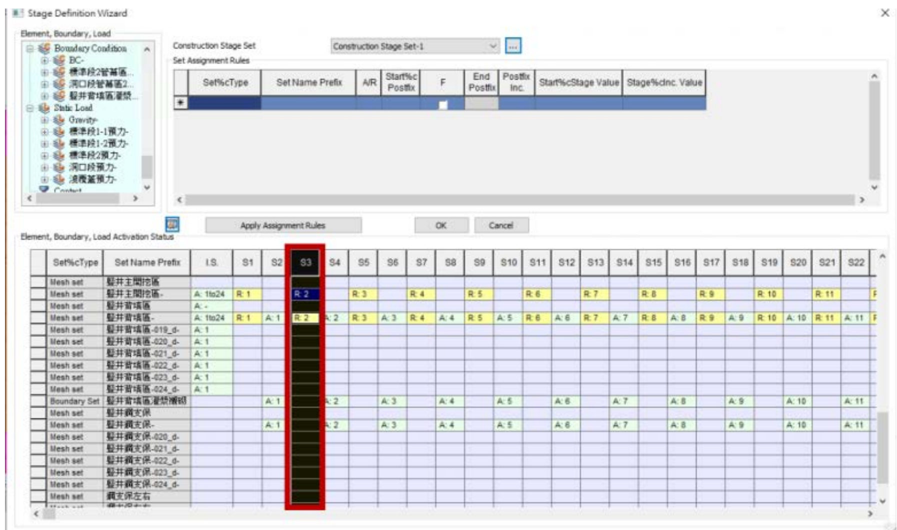


Anchors (Prestressed Application)

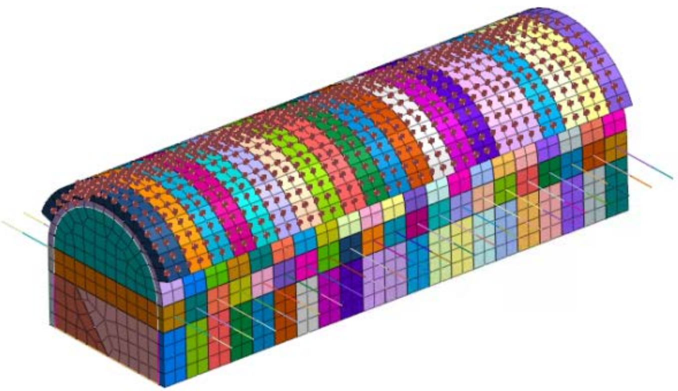




Construction Stage



管幕區變更材質



Backfill Grouting and Lining

GTS NX SLOPE STABILITY ANALYSIS

2D分析 - 方法 1

Stress Analysis Method (SAM)

2D分析 - 方法 2

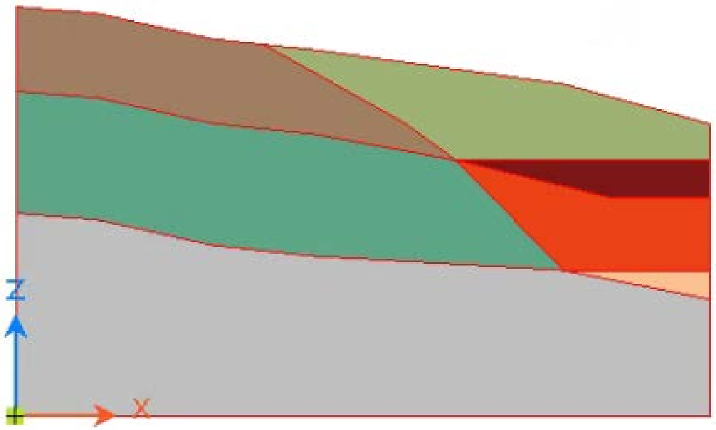
Strength Reduction Method (SRM)

3D分析

Strength Reduction Method (SRM)

GTS NX 2D SLOPE STABILITY ANALYSIS

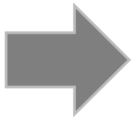
Post-Excavation Slope Stability Calculation



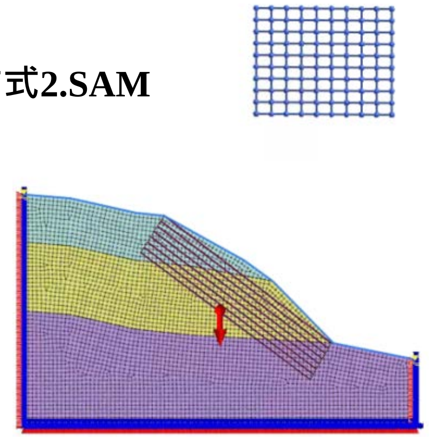
Weathered Soil

Weathered Rock

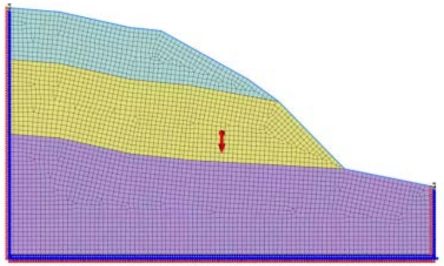
Soft Rock



2D分析-方式2.SAM

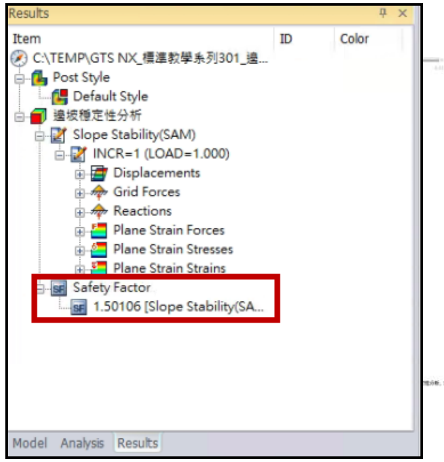


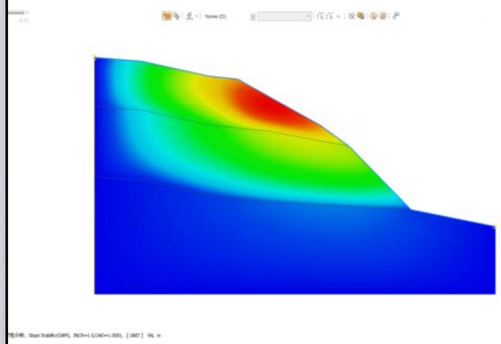
2D分析-方式3.SRM



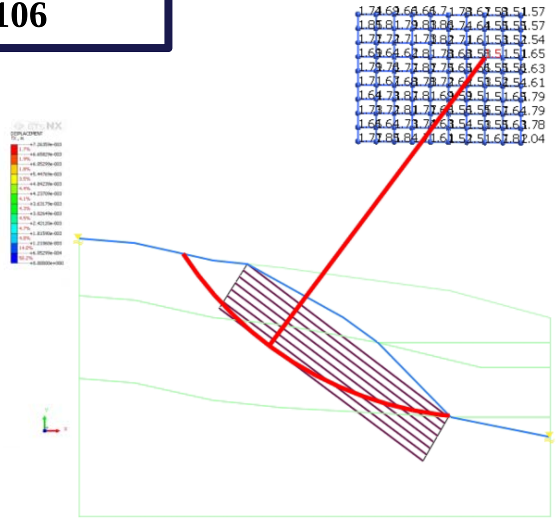
SRM gradually reduces the shear strength parameters (cohesion c and friction angle φ) of the slope foundation material until the analysis becomes unstable (diverges). At this point, slope failure is assumed to occur. The maximum strength reduction factor at this divergence point is taken as the minimum factor of safety for the slope.

2D分析-方式1
SAM計算之安全係數1.50106

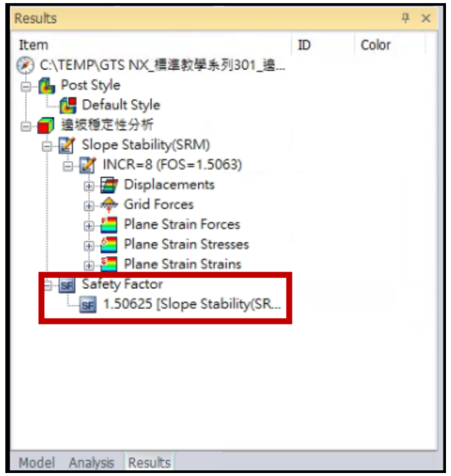




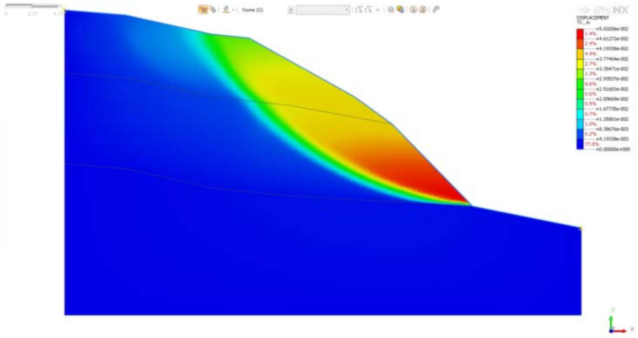
Tx Translation(m)



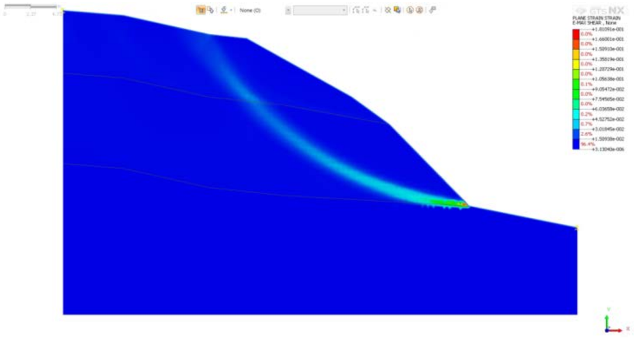
2D分析-方式2
SRM計算之安全係數1.50625



2D分析-方式2
SRM透過水平變形和最大剪切應變判斷破壞面



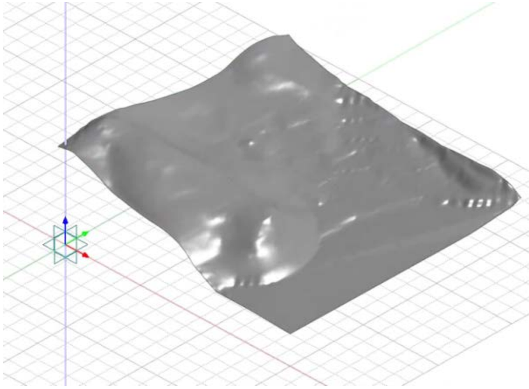
Tx Translation(m)



Maximum Shear Strain

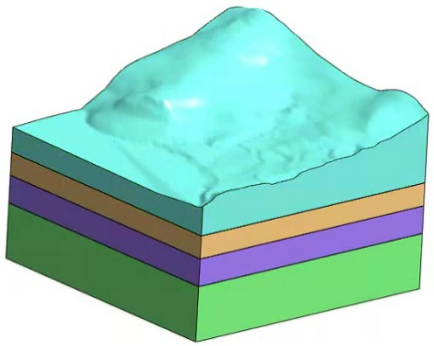
GTSNX 3D SLOPE STABILITY ANALYSIS

3DTerrain Surface Features

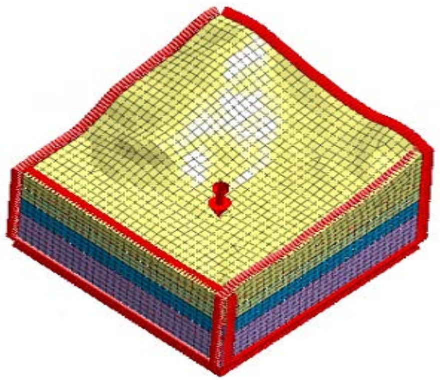


300m ×300m

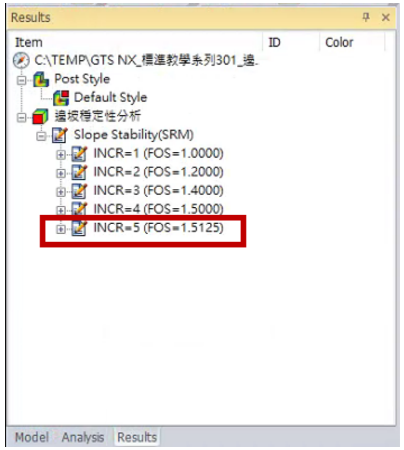
3DTerrain Solid Features

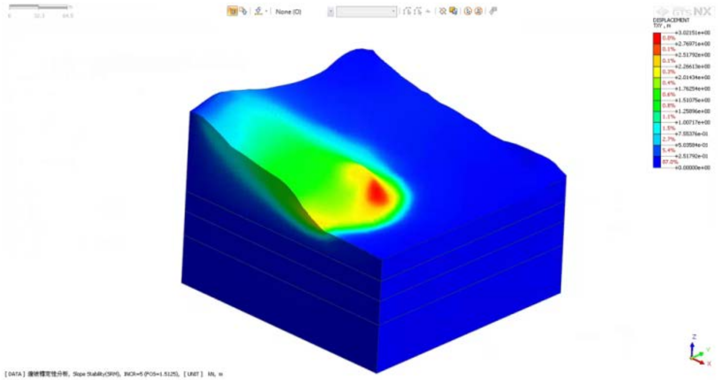


Self-Weight

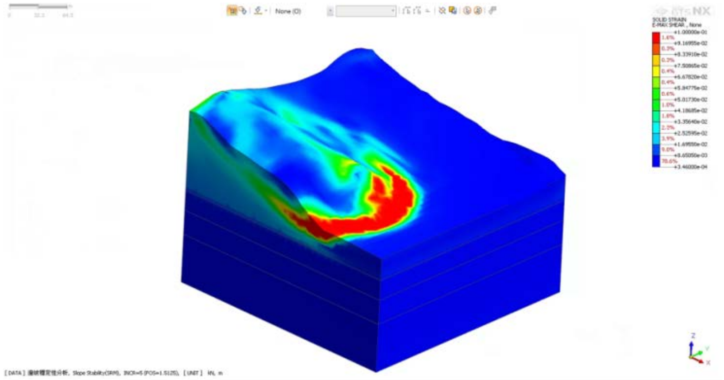


3D分析
SRM計算之安全係數1.5125





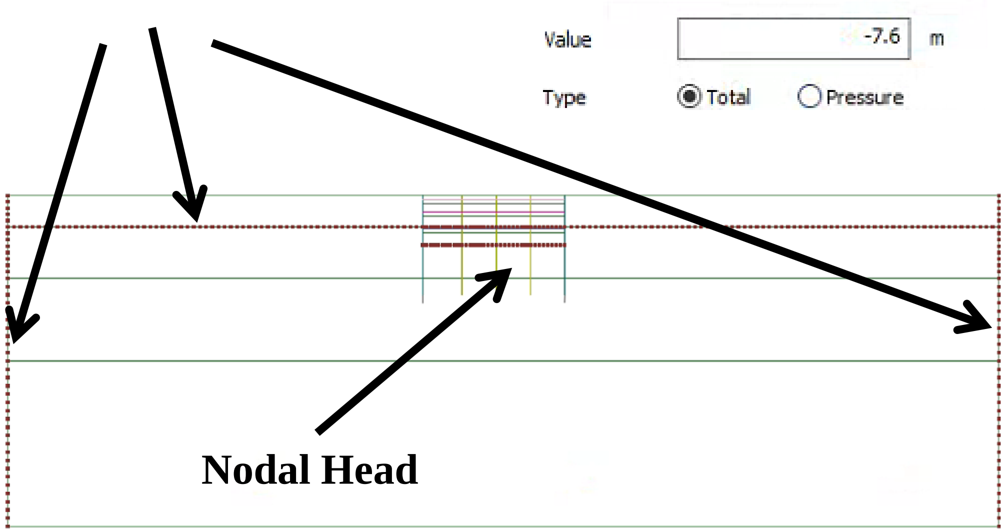
Tx Translation(m)



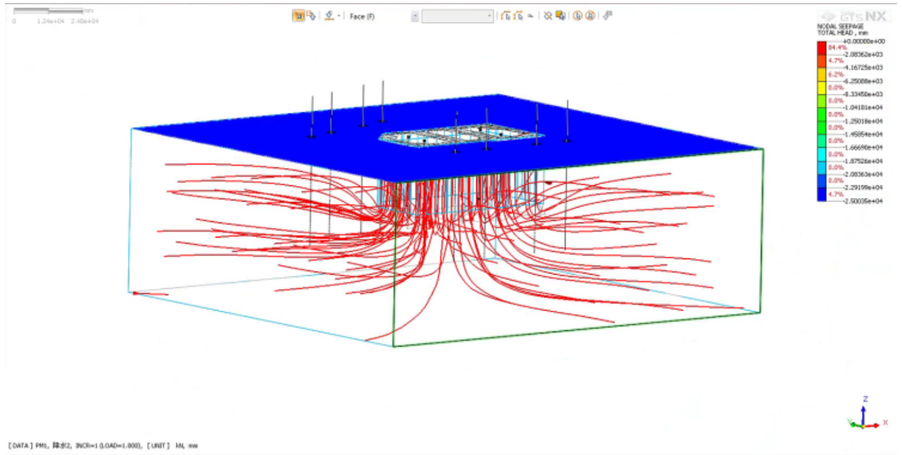
Maximum Shear Strain

GTS NX GROUNDWATER SIMULATION

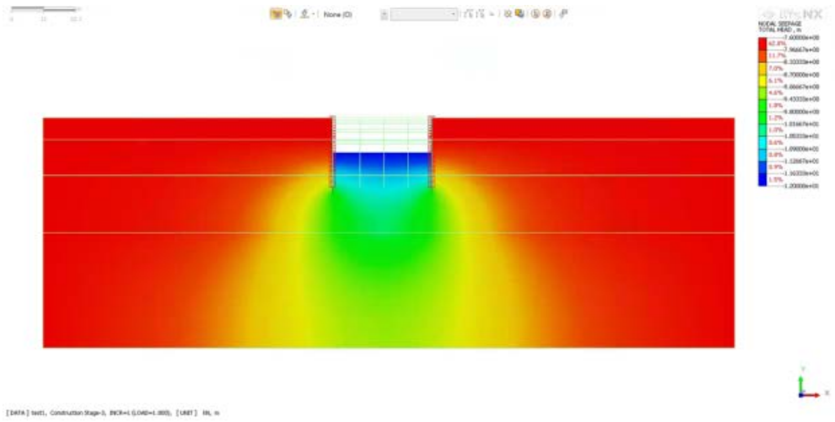
Nodal Head



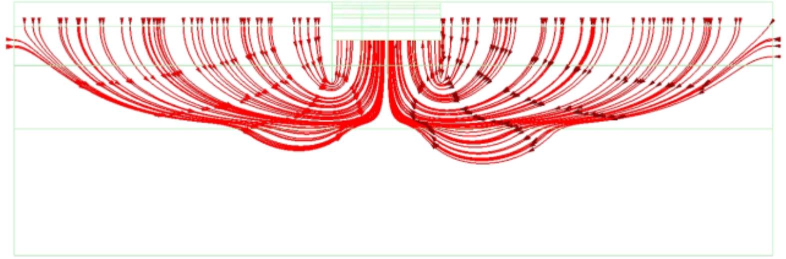
Nodal Head



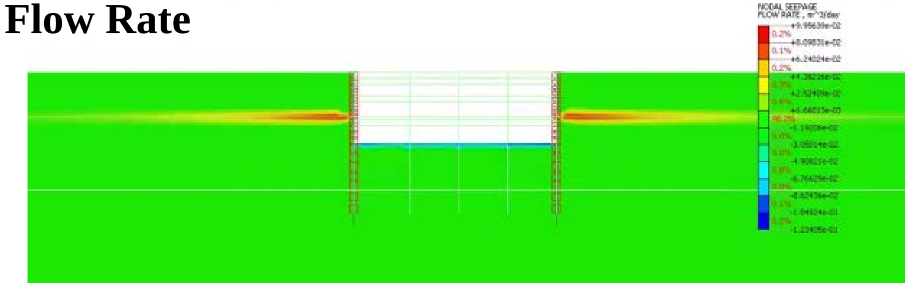
Total Head



Flow Path

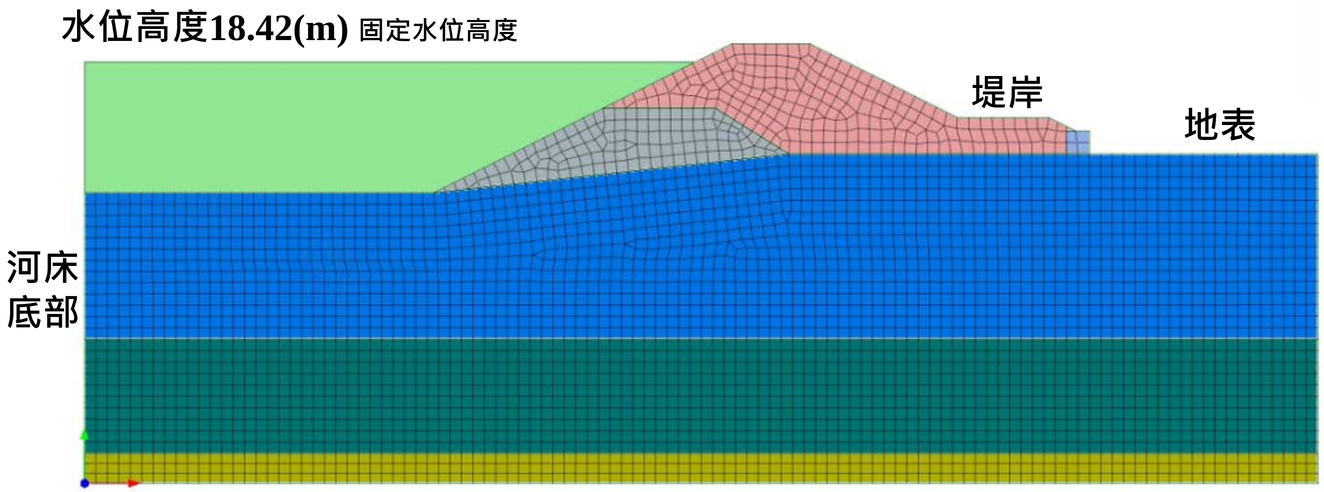


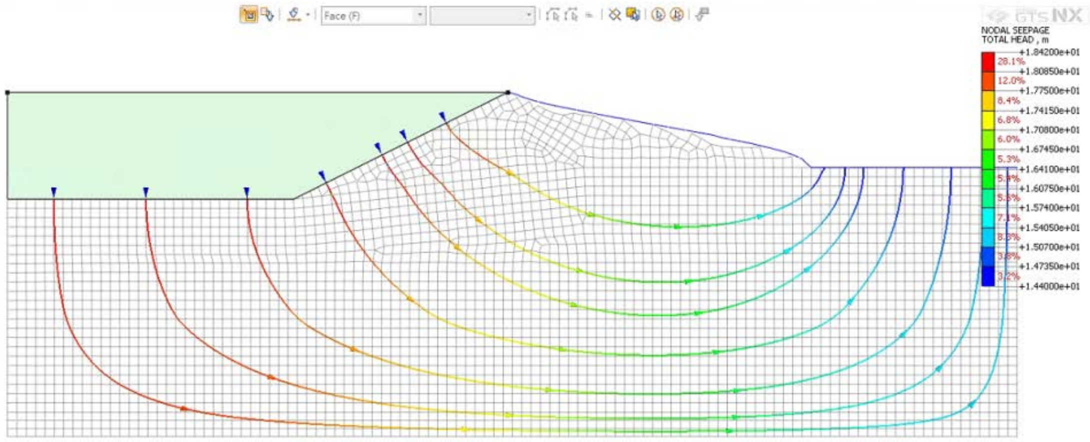
Seepage Flow Rate



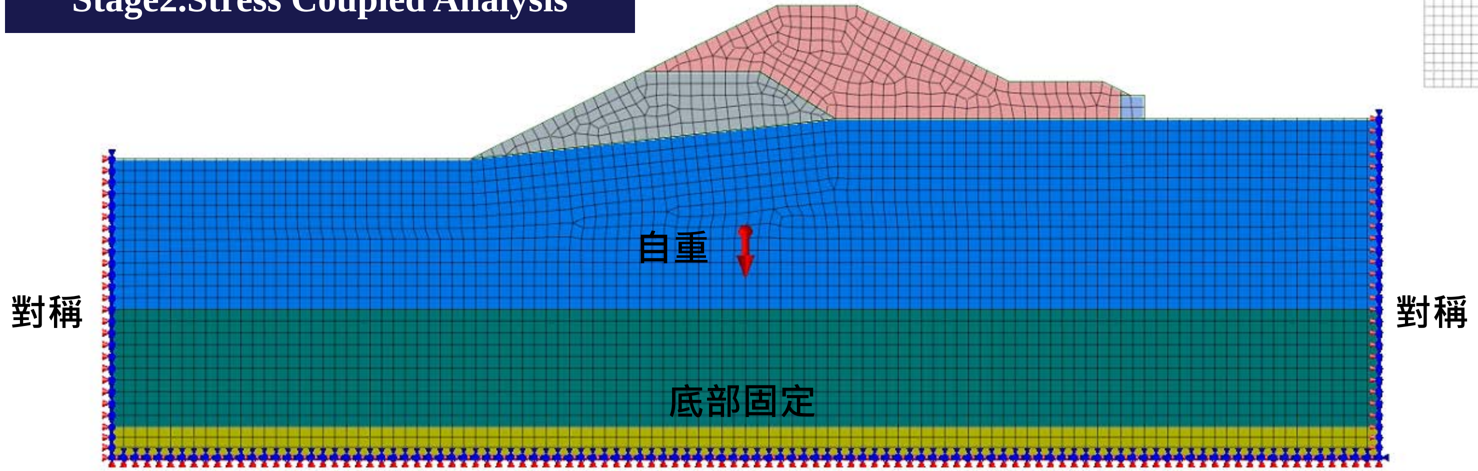
GTS NX Seepage-Stress Coupled Analysis

Stage1.Seepage Analysis



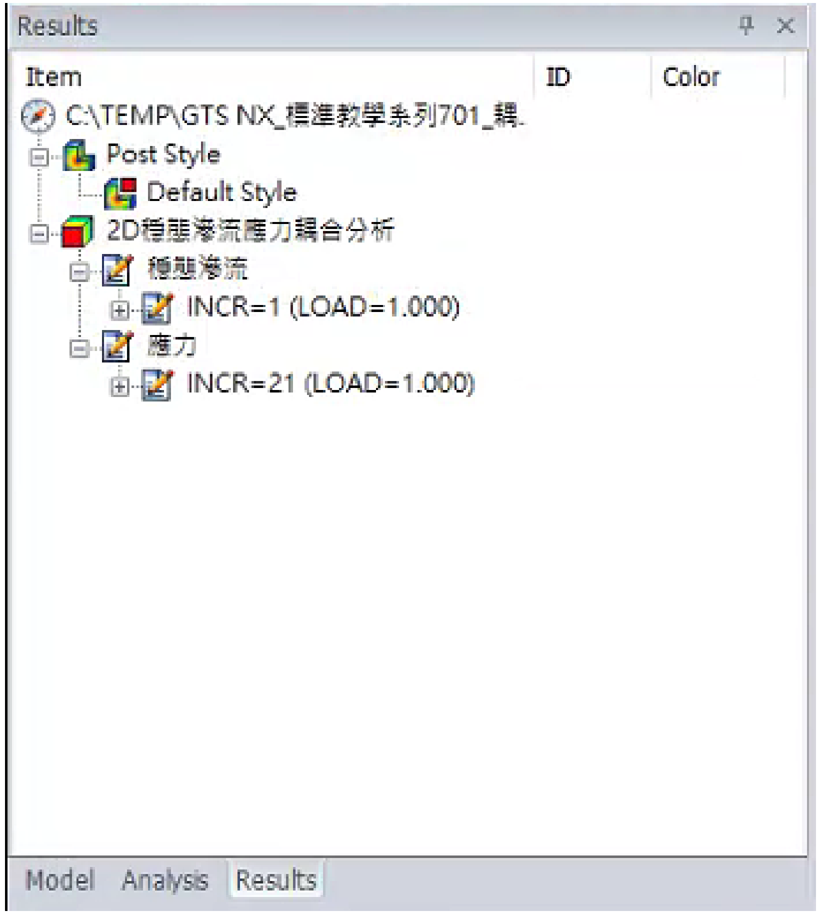


Stage2.Stress Coupled Analysis



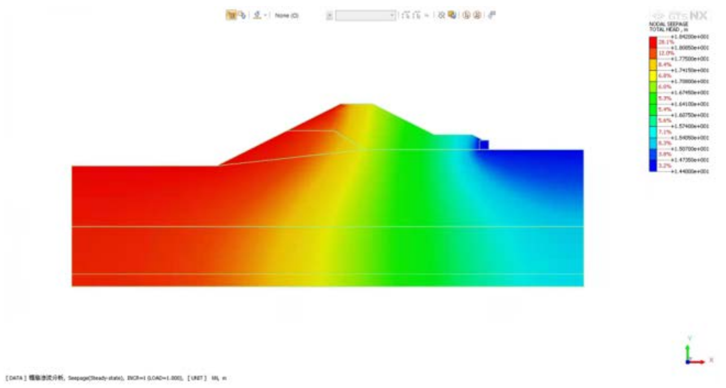
GTS NX Seepage-Stress Coupled Analysis

Results
穩態滲流 INCR=1：滲流結果
應力 INCR=21：應力結果

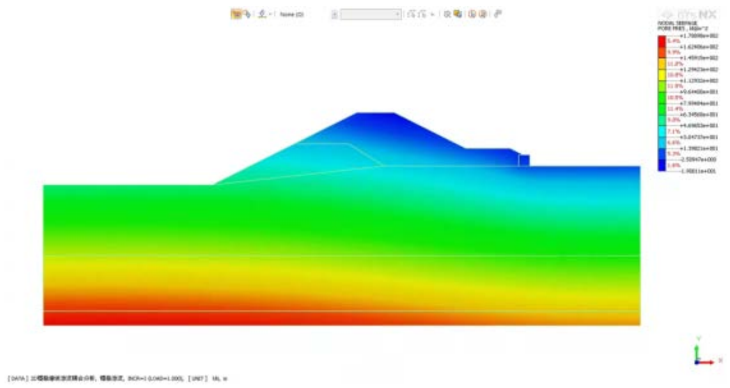


滲流結果

Total Head(m)

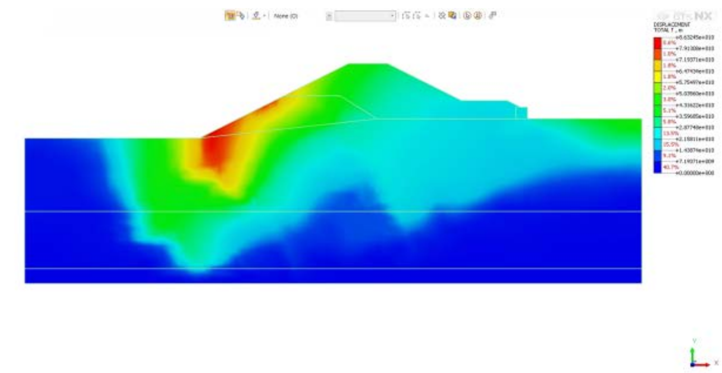


Pore Pressure(KN/m²)

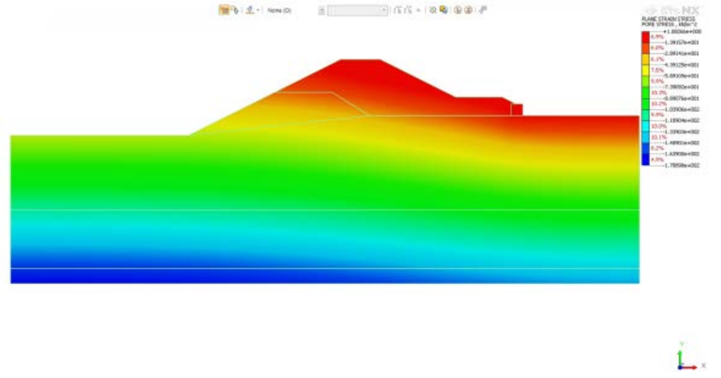


應力結果

Displacement Total(m)



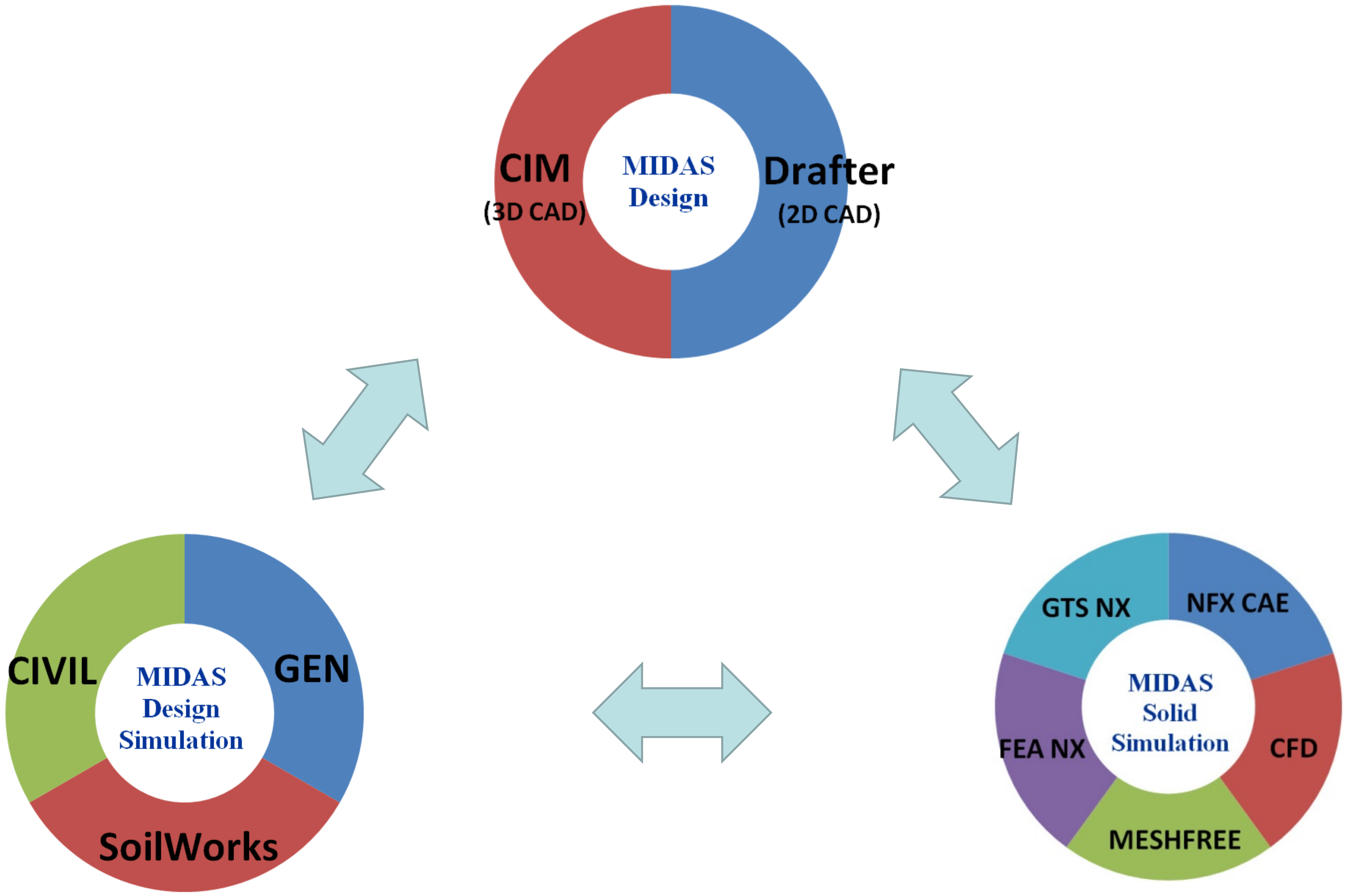
Pore Stress(KN/m²)



MIDAS

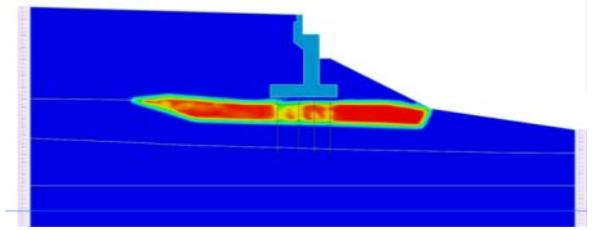
SYSTEM INTEGRATION

MIDAS INTEGRATION

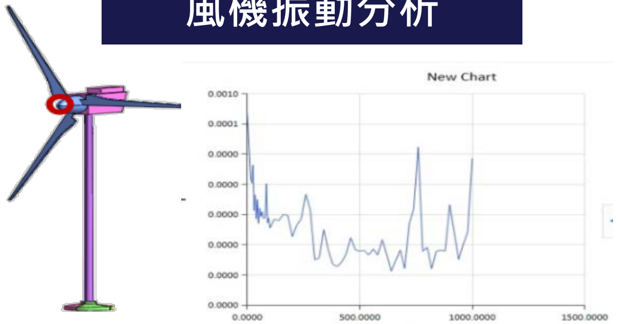


Solid Total Solution

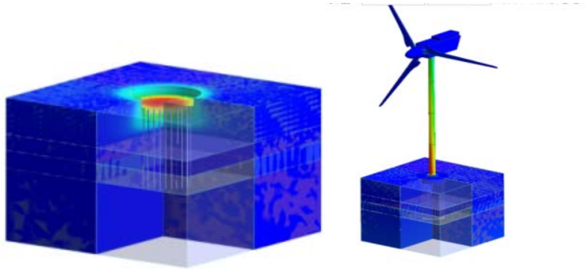
土壤液化分析

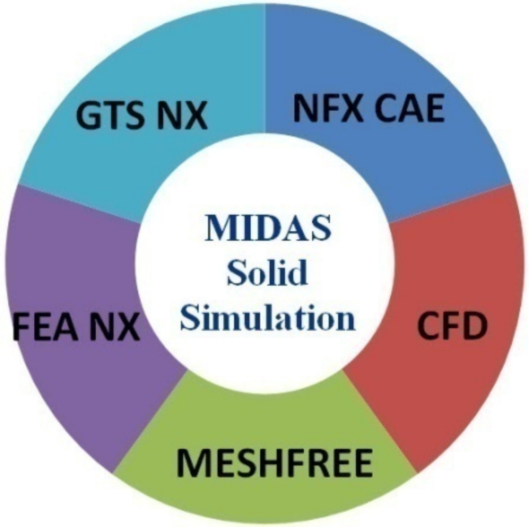


風機振動分析

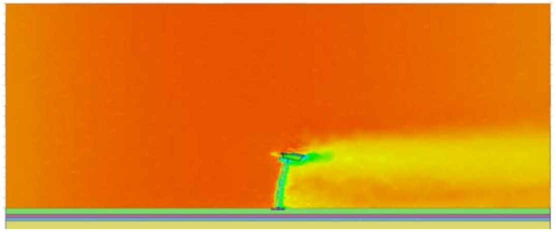


風機安裝施工階段分析

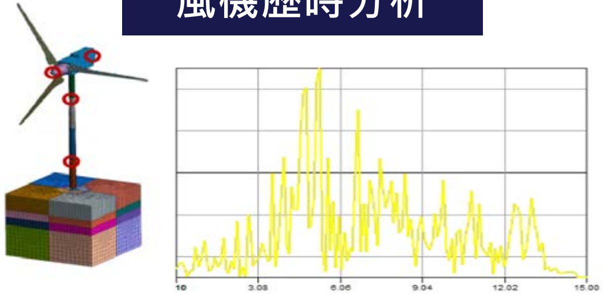




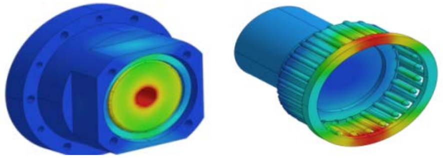
風機流固耦合分析



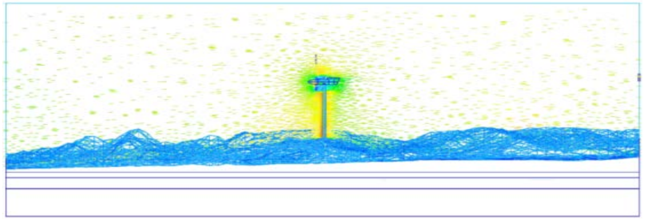
風機歷時分析



齒輪組分析

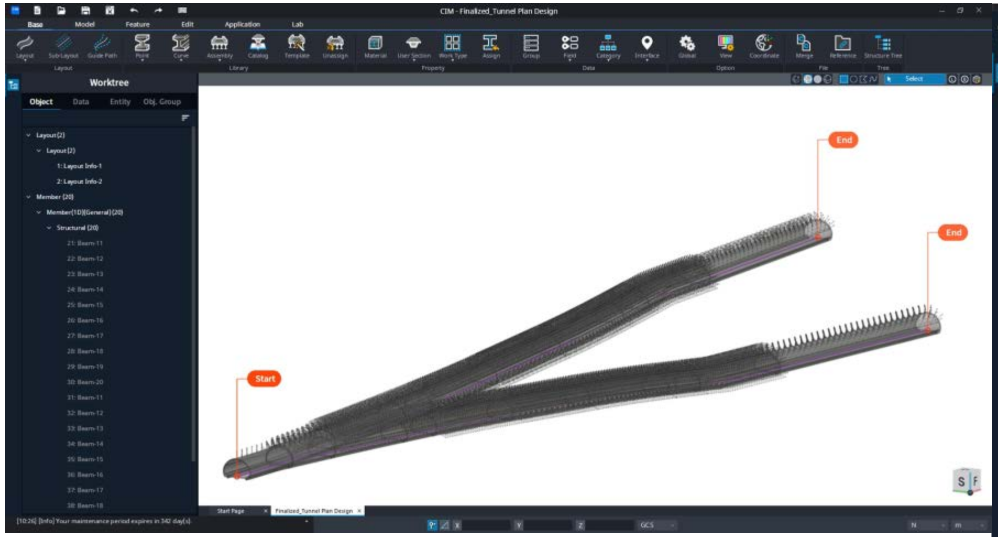


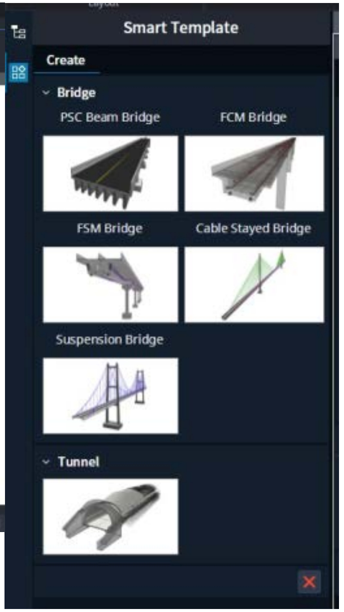
風機地形風場分析



CIM + FEA/GTS 3D Model Integration

CIM-3DAutomatic Model Adjustment Along Alignment

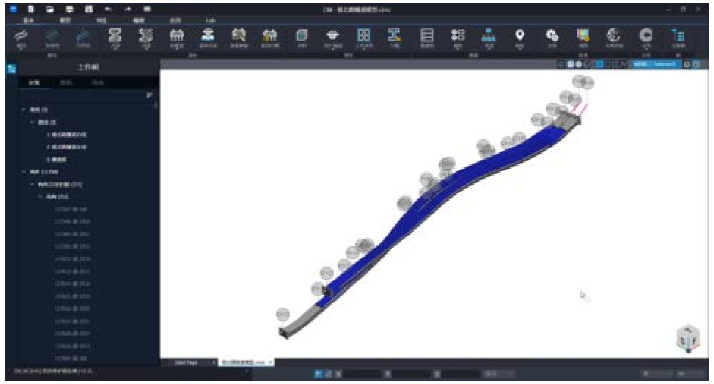




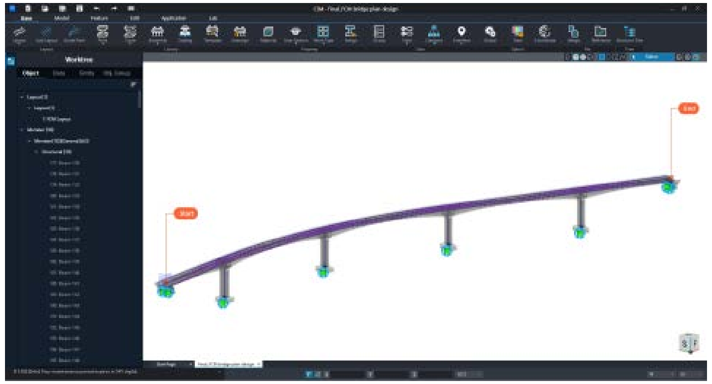
Bridge & Tunnel Wizard

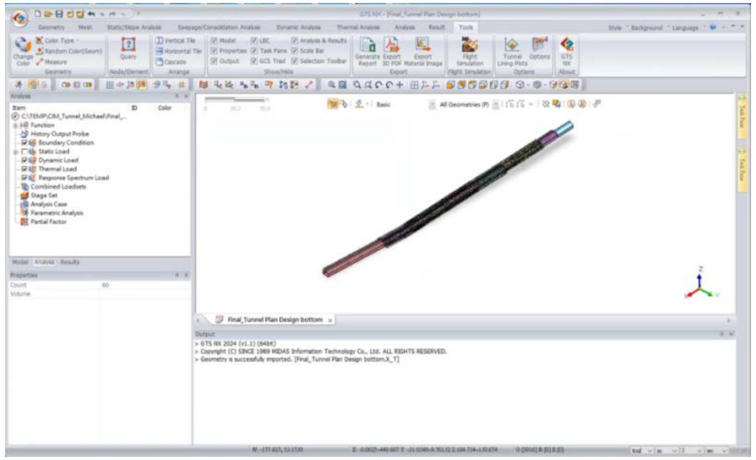
CIM+GTS 3D Model Integration

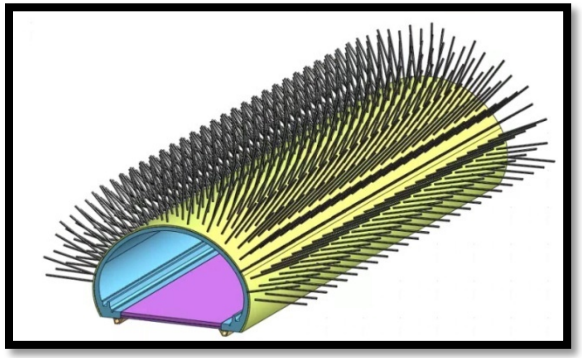
Tunnel

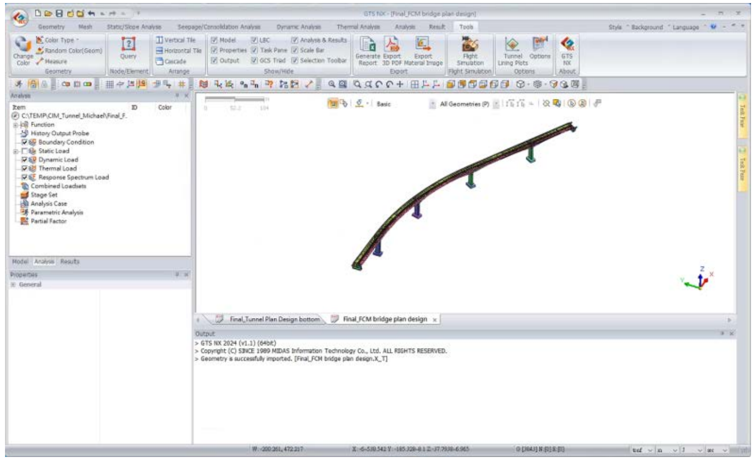


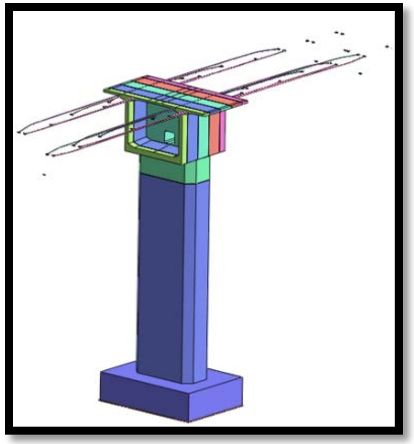
Bridge





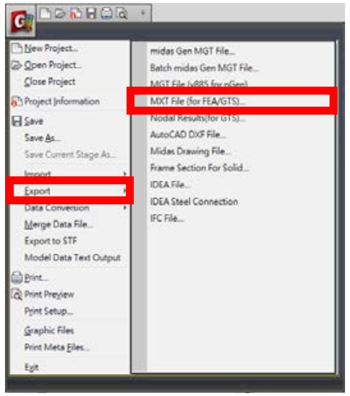


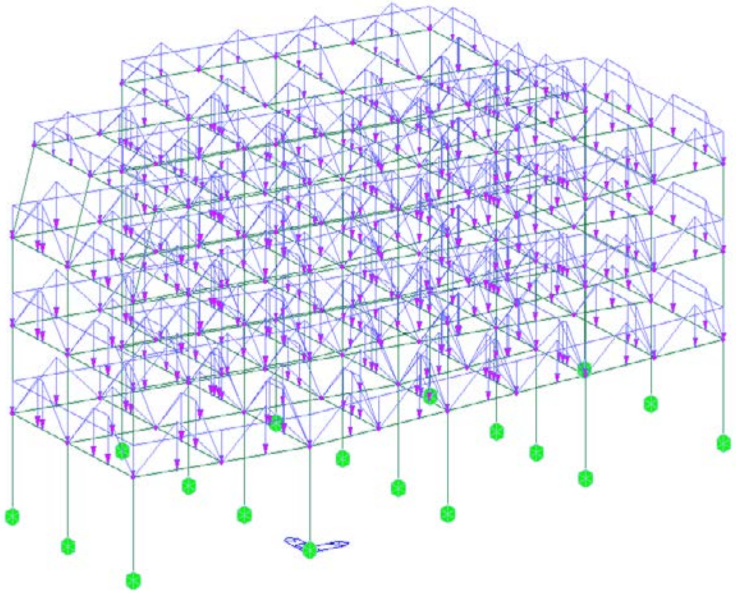


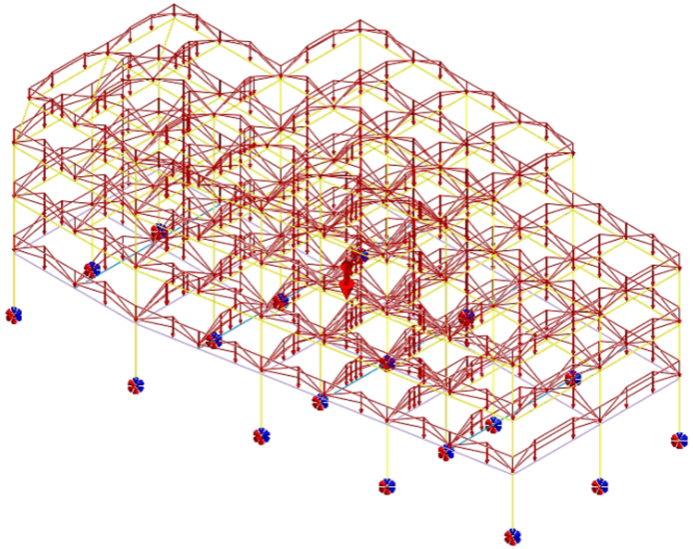


CIM&GTS NX
Direct Conversion of Solid Features

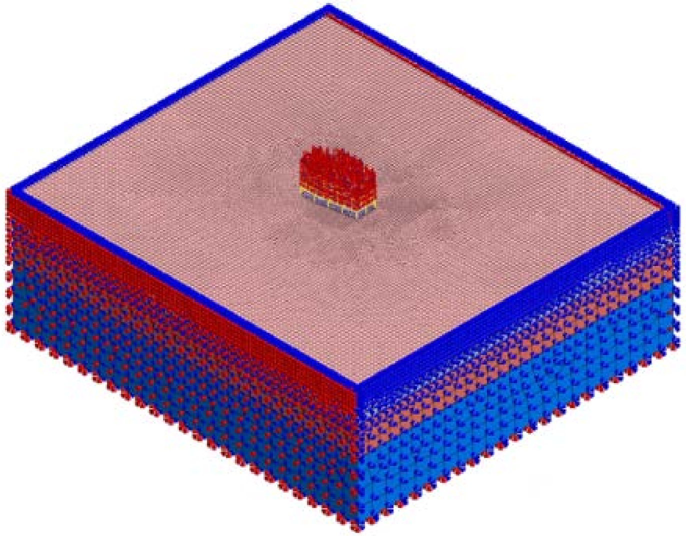
FEA or GTS NX & Gen | Structural Interaction Analysis

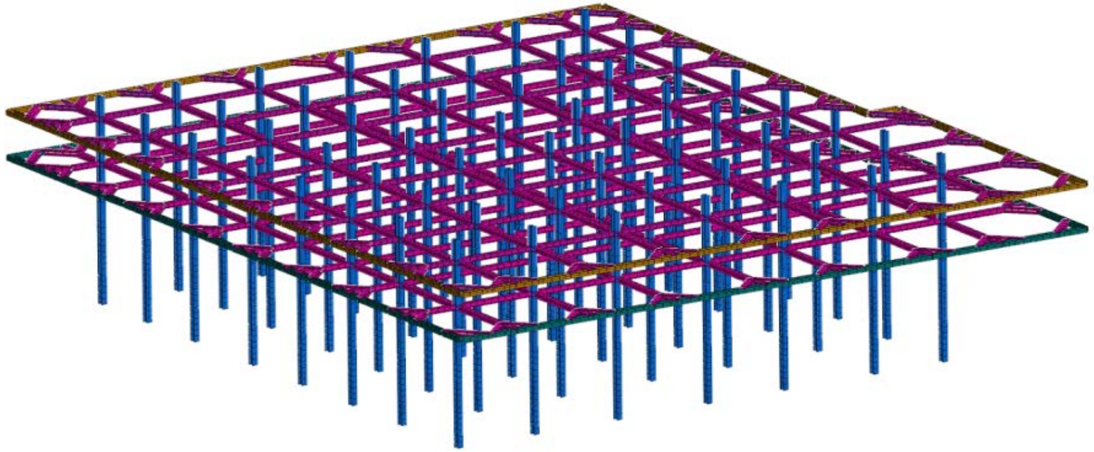


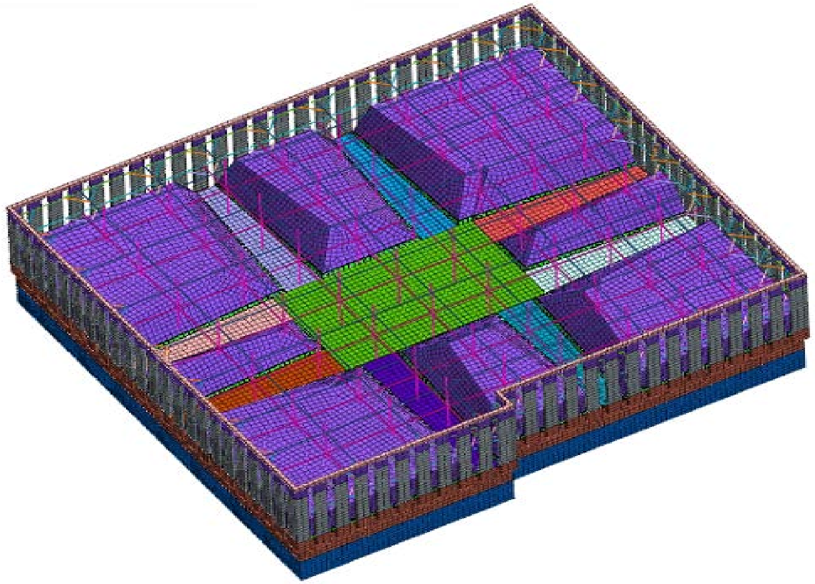


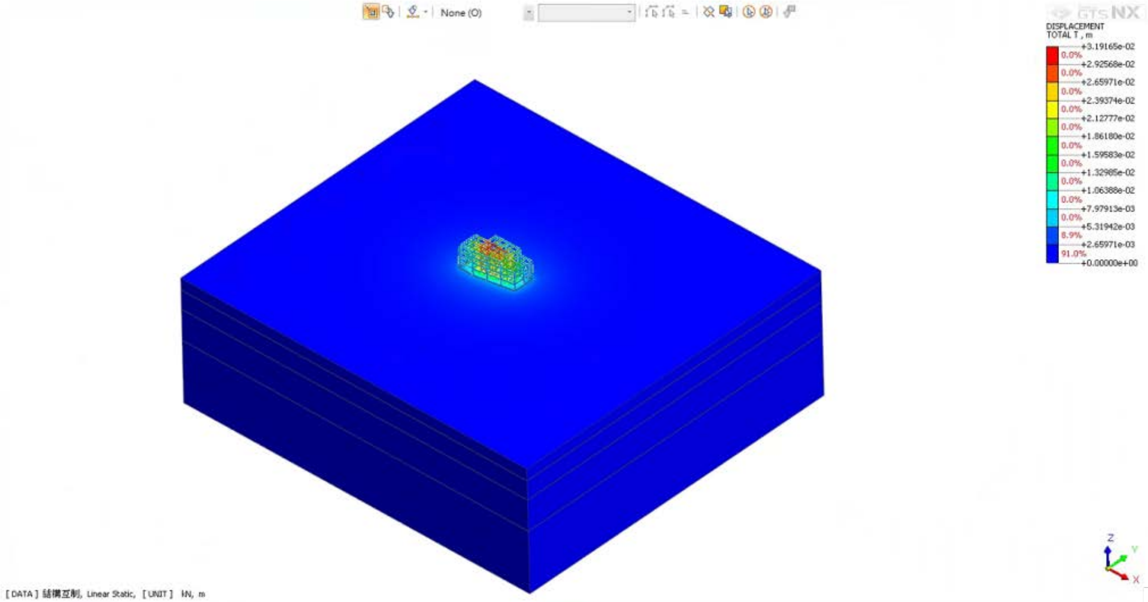


FEA or GTS NX & Gen | Structural Interaction Analysis



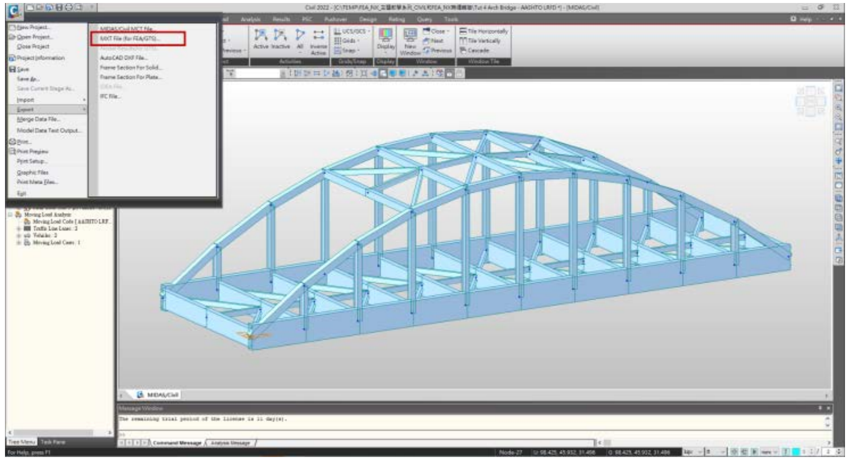


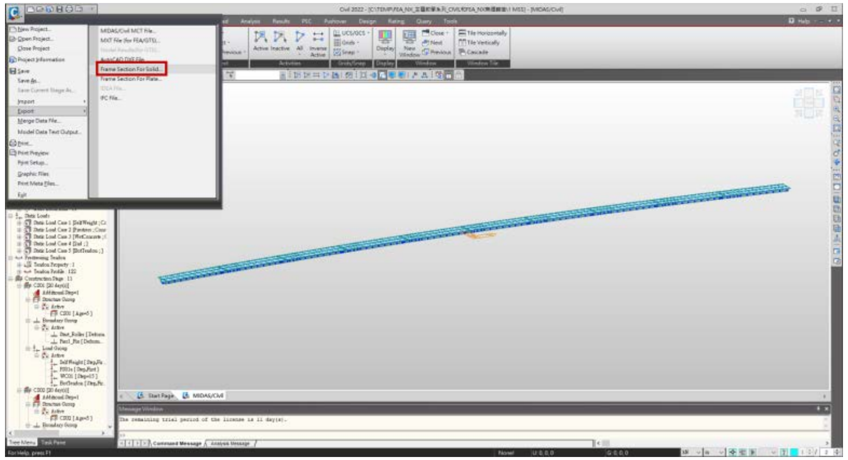




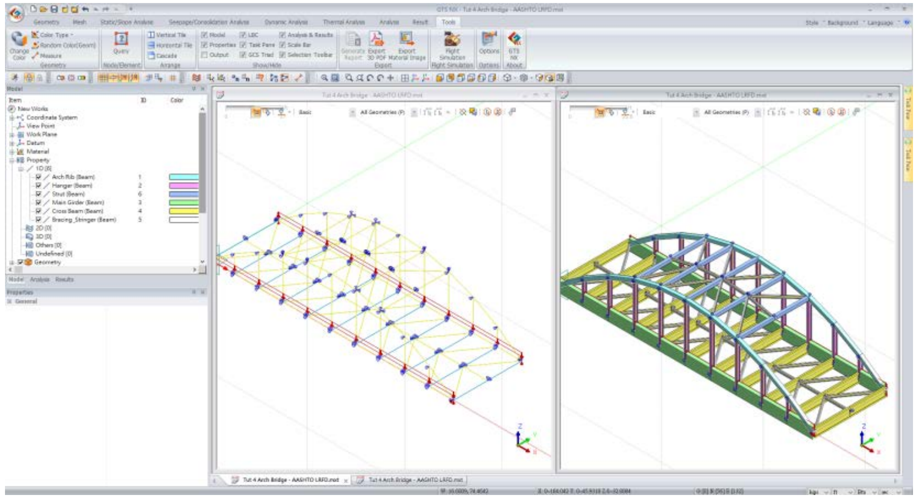
GTS / FEA NX+CIVIL Smooth Conversion

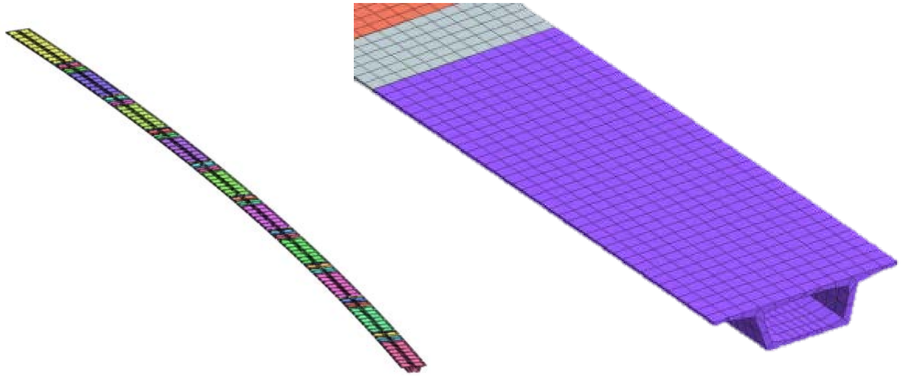
匯出MXT Files(*.mxt)檔案格式





元素&特徵無縫轉換





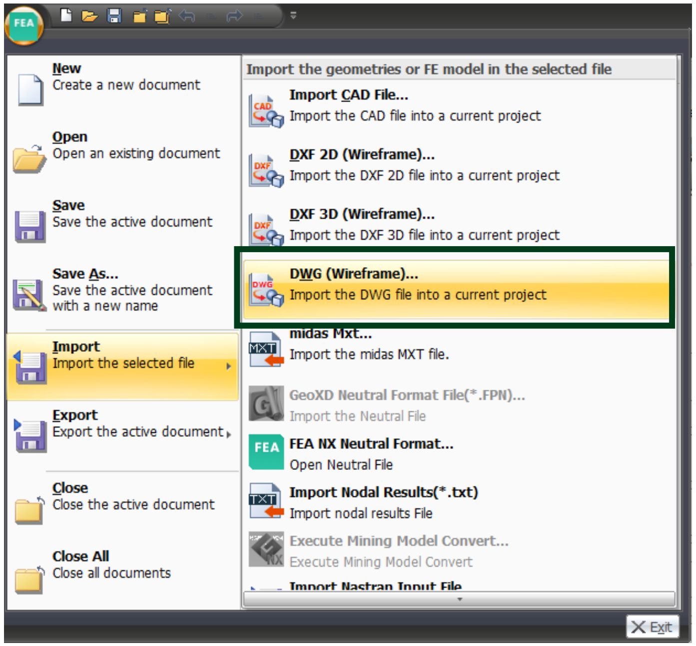
THANK YOU FOR LISTENING

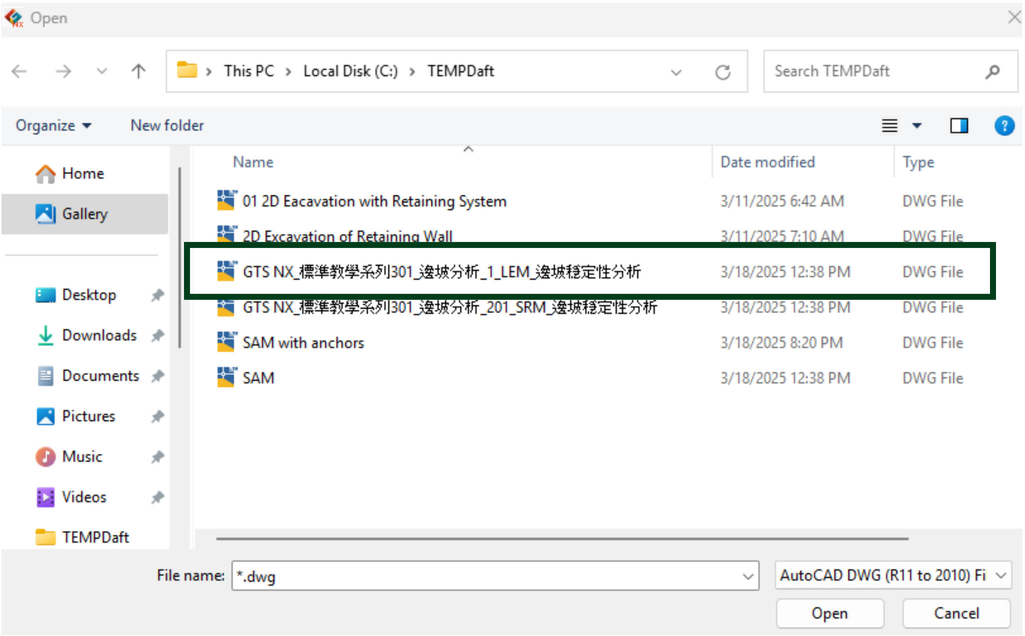
沈約翰 (John)
john@midasuser.com.tw

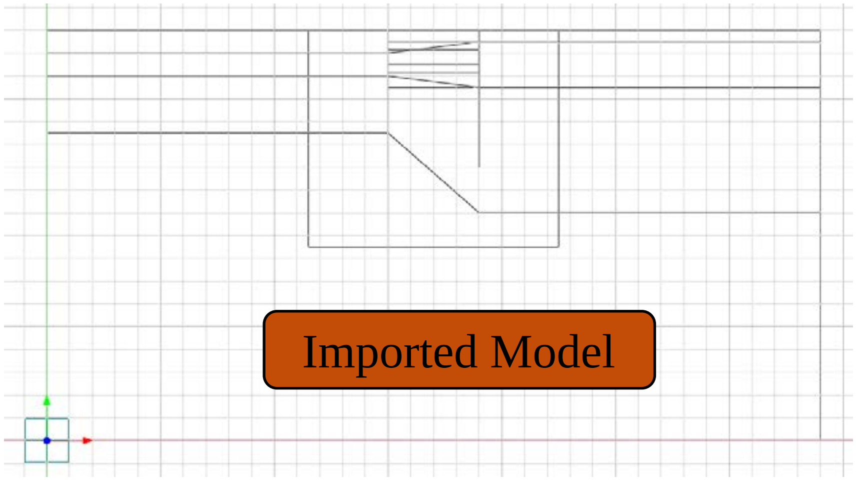
2D Excavation with Soil Retaining System

沈約翰
john@midasuser.com.tw

2D MODEL IMPORT

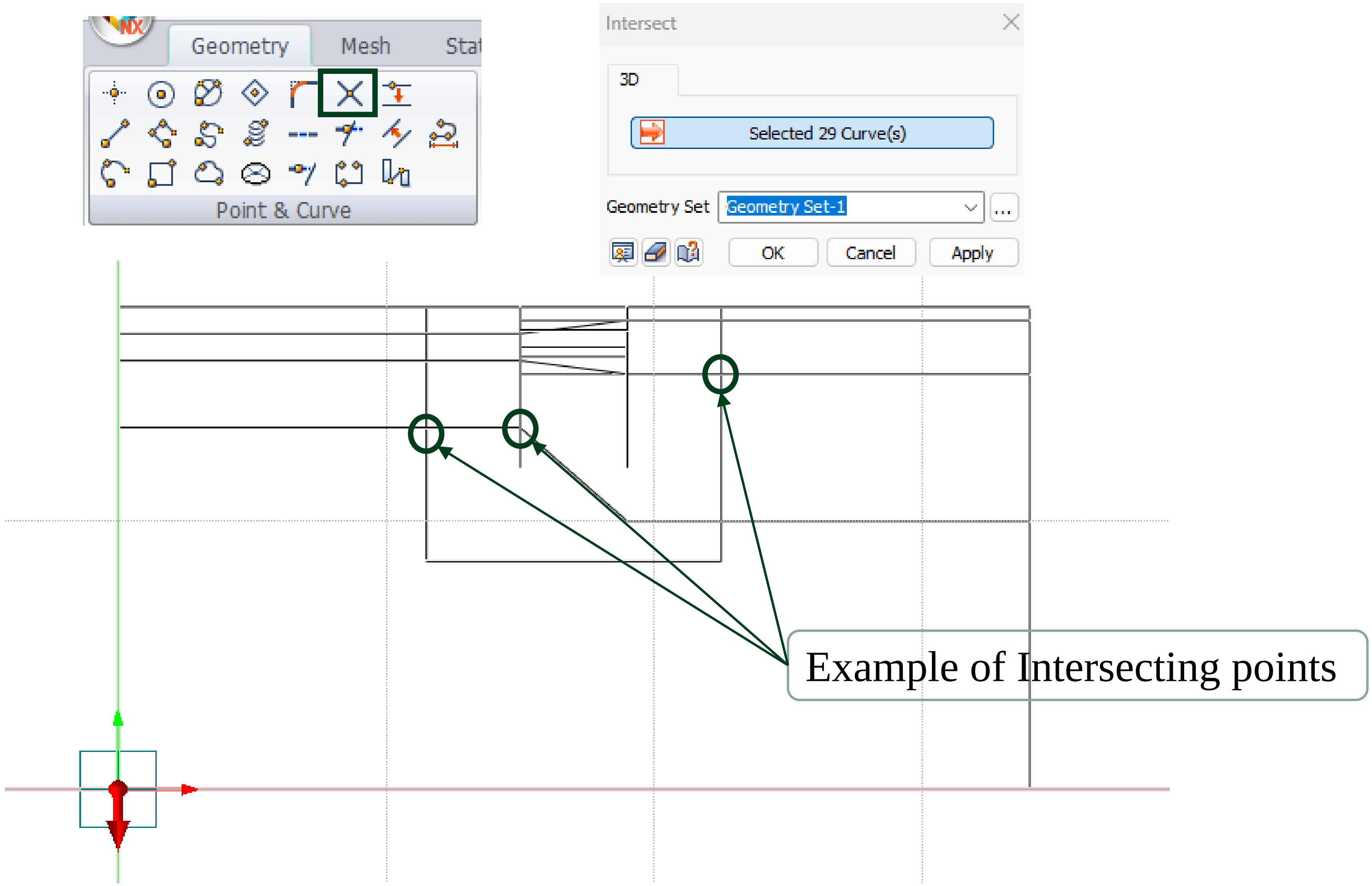






MIDAS

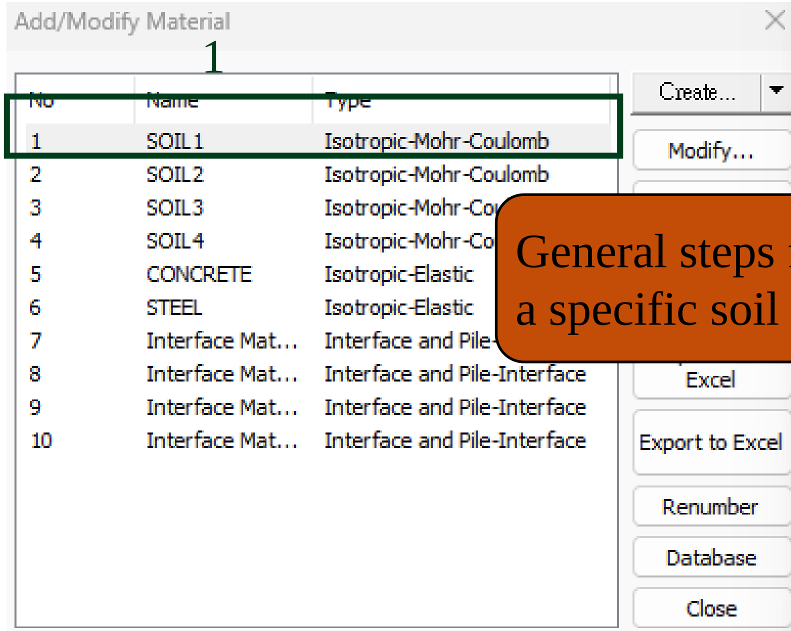
INTERSECT FUNCTION

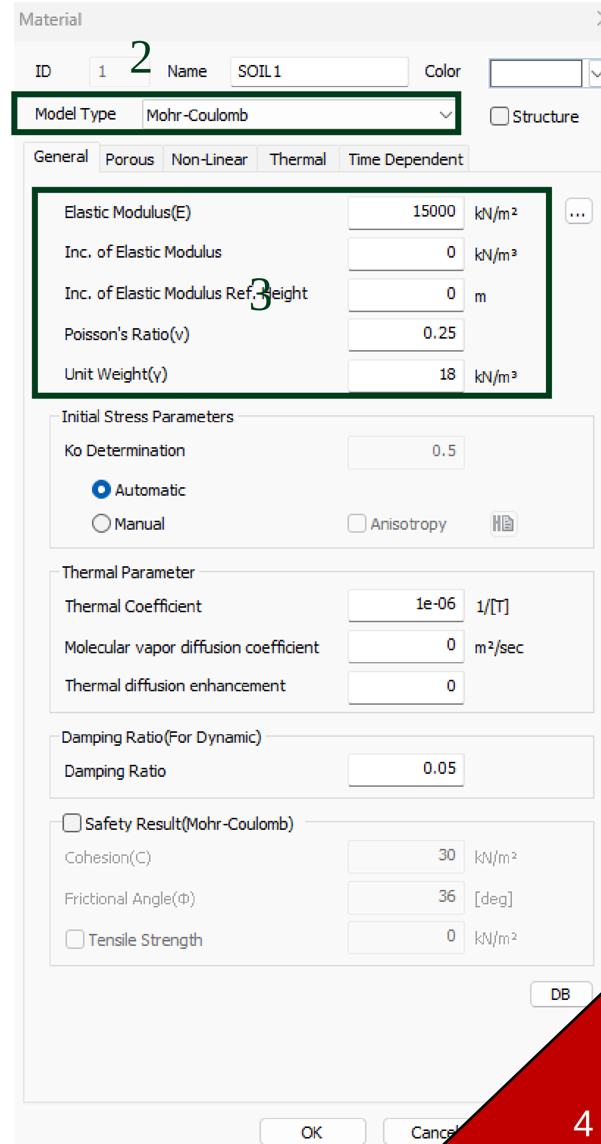


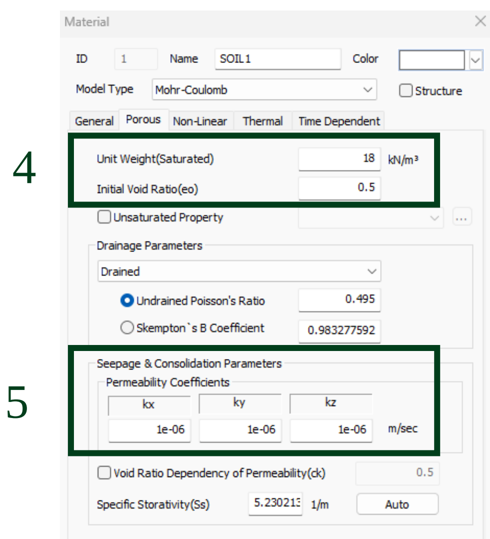
Automatically divides the model into intersecting points

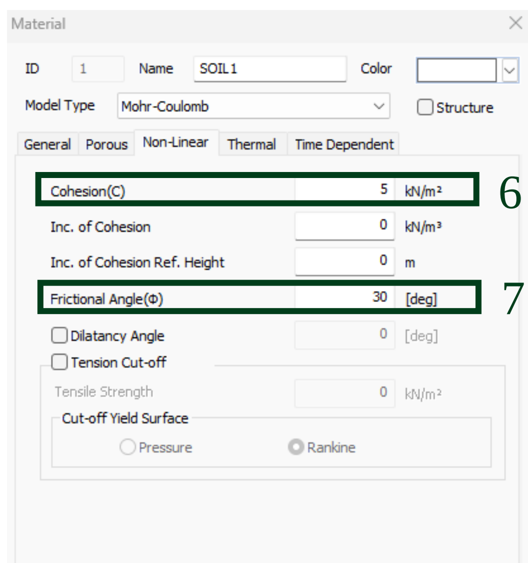
SOIL MATERIAL

General steps for defining a specific soil material

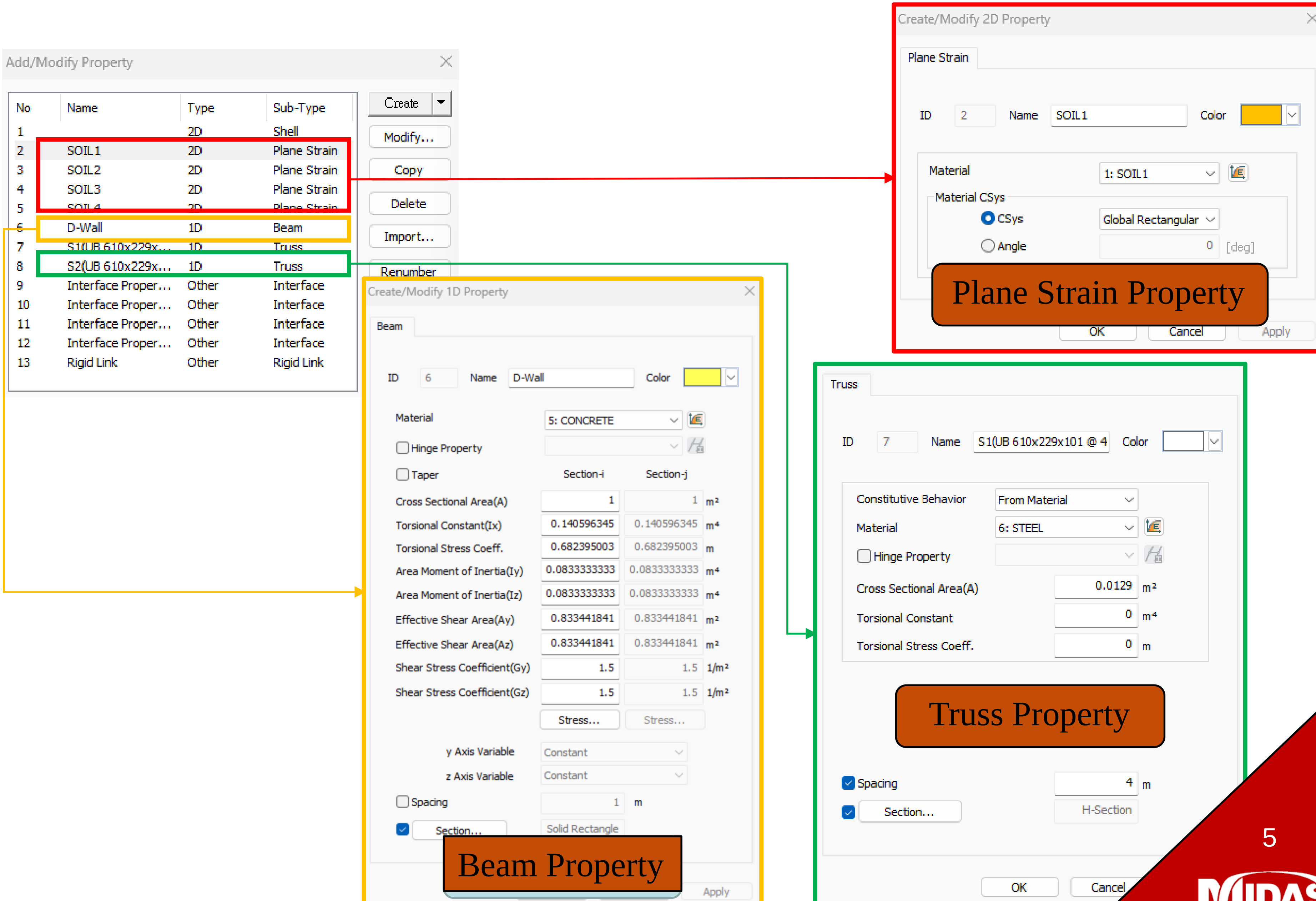








SOIL PROPERTY



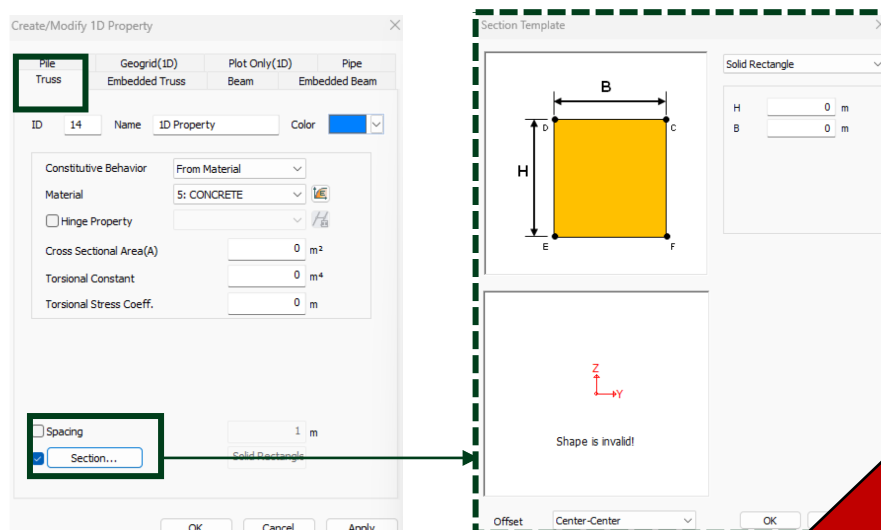
BEAM - TRUSS ELEMENT

Beam and Truss element comparison

Feature	Truss Element	Beam Element
Resists Axial Force	✅ Yes	✅ Yes
Resists Bending	❌ No	✅ Yes
Resists Shear	❌ No	✅ Yes
Resists Torsion	❌ No	✅ Yes
Degrees of Freedom (DOF)	3 per node (UX, UY, UZ)	6 per node (UX, UY, UZ, RX, RY, RZ)
Transfers Moments	❌ No	✅ Yes
Typical Application	Trusses, cables	Beams, frames

Truss Library

Note: Spacing defines the distance between each individual element that will be generated along a line or curve when using truss-type elements.



ELEMENT PROPERTY

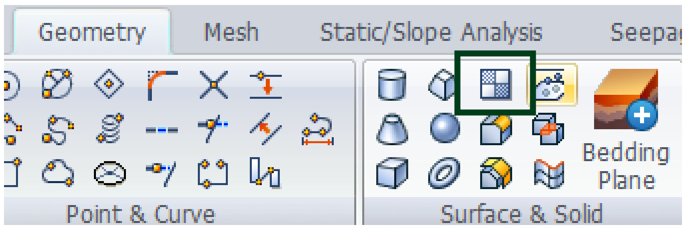
Ground Property

Name	Interface (SOIL 1)	Interface (SOIL 2)	Interface (SOIL 3)	Interface (SOIL 4)	SOIL 1	SOIL 2	SOIL 3	SOIL 4
Type	Other	Other	Other	Other	2D	2D	2D	2D
Model Type	Interface	Interface	Interface	Interface	Plane Strain	Plane Strain	Plane Strain	Plane Strain
Interface Type	Line	Line	Line	Line				
Material	SOIL 1	SOIL 2	SOIL 3	SOIL 4	SOIL 1	SOIL 2	SOIL 3	SOIL 4

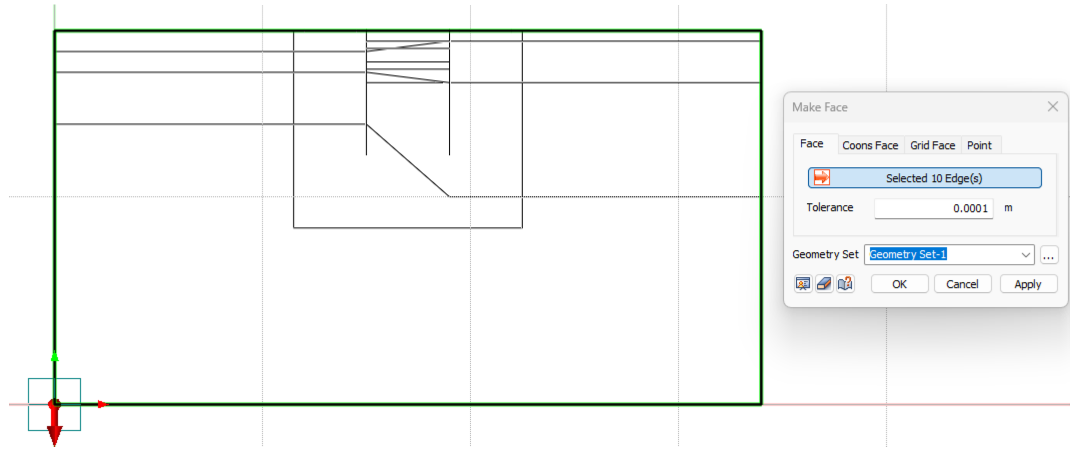
Structural Property

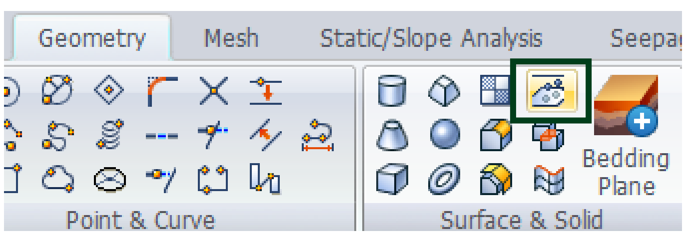
Name	D-Wall	S1	S2
Type	1D	1D	1D
Model Type	Beam	Truss	Truss
Material	CONCRETE	STEEL	STEEL
Section	Solid Rectangle	H-Section	H-Section
Section Size	1x1m @ 1m c/c	UB 610x229x101 @ 4m c/c	UB 610x229x101 @ 4m c/c

GEOMETRIC MODEL

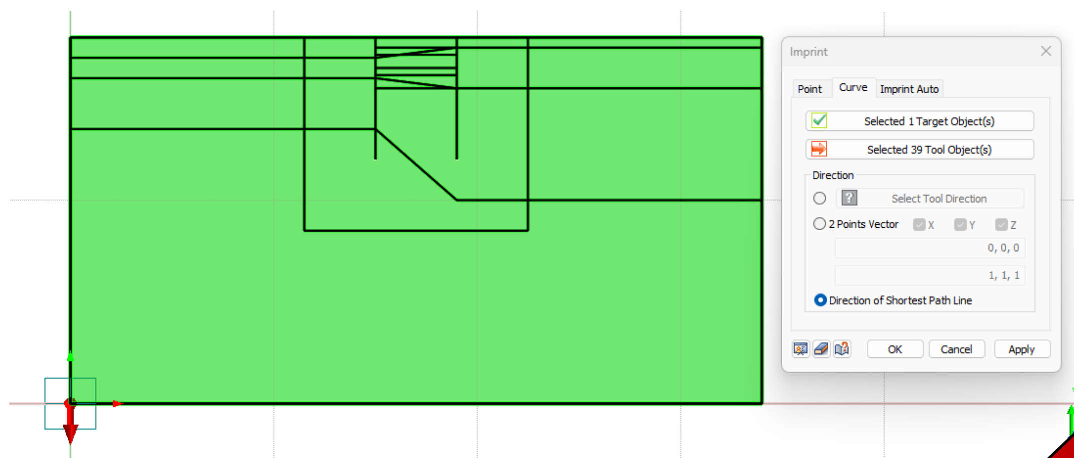


Face Function

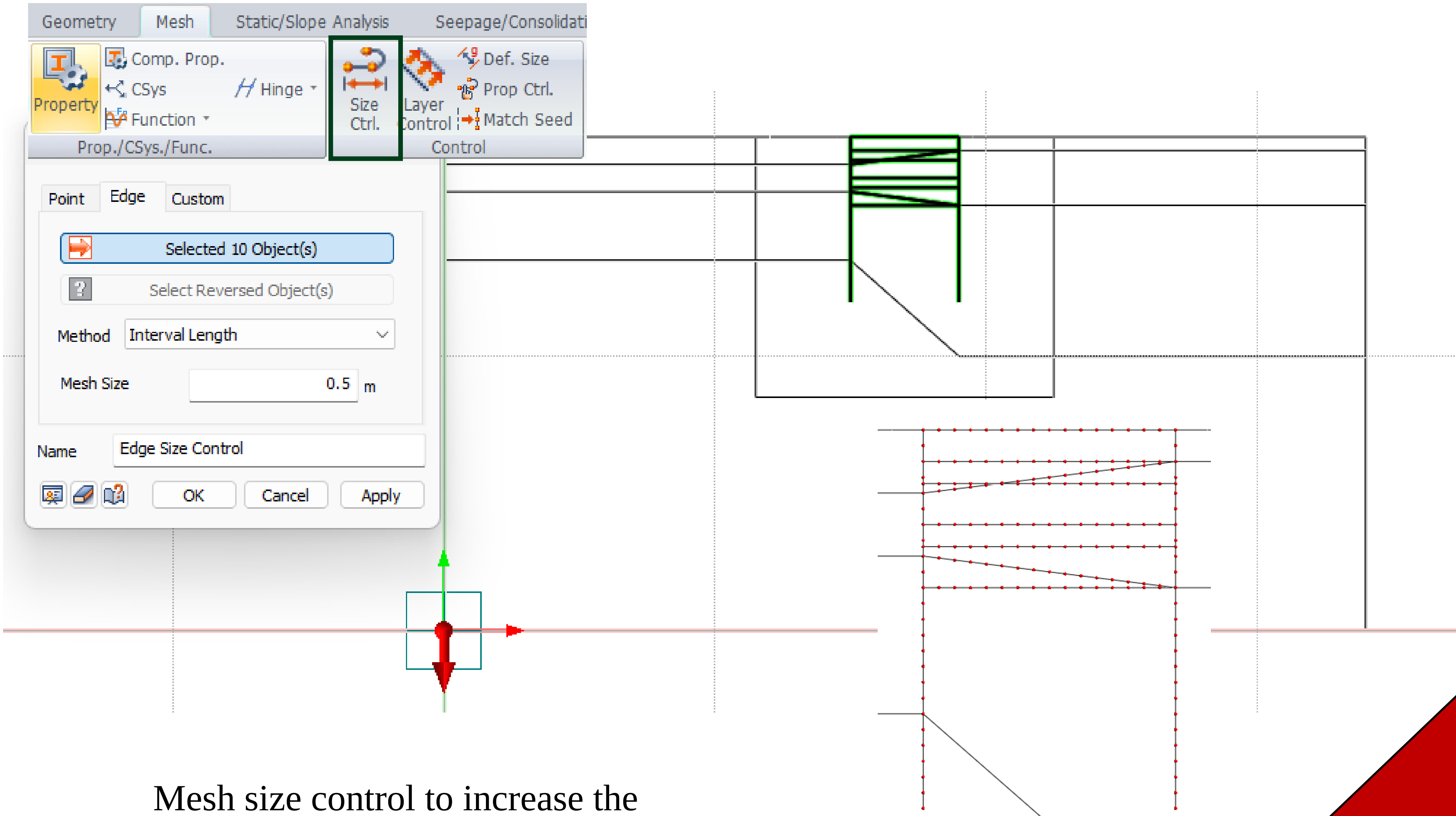




Imprint Function



MESH - SIZE CONTROL



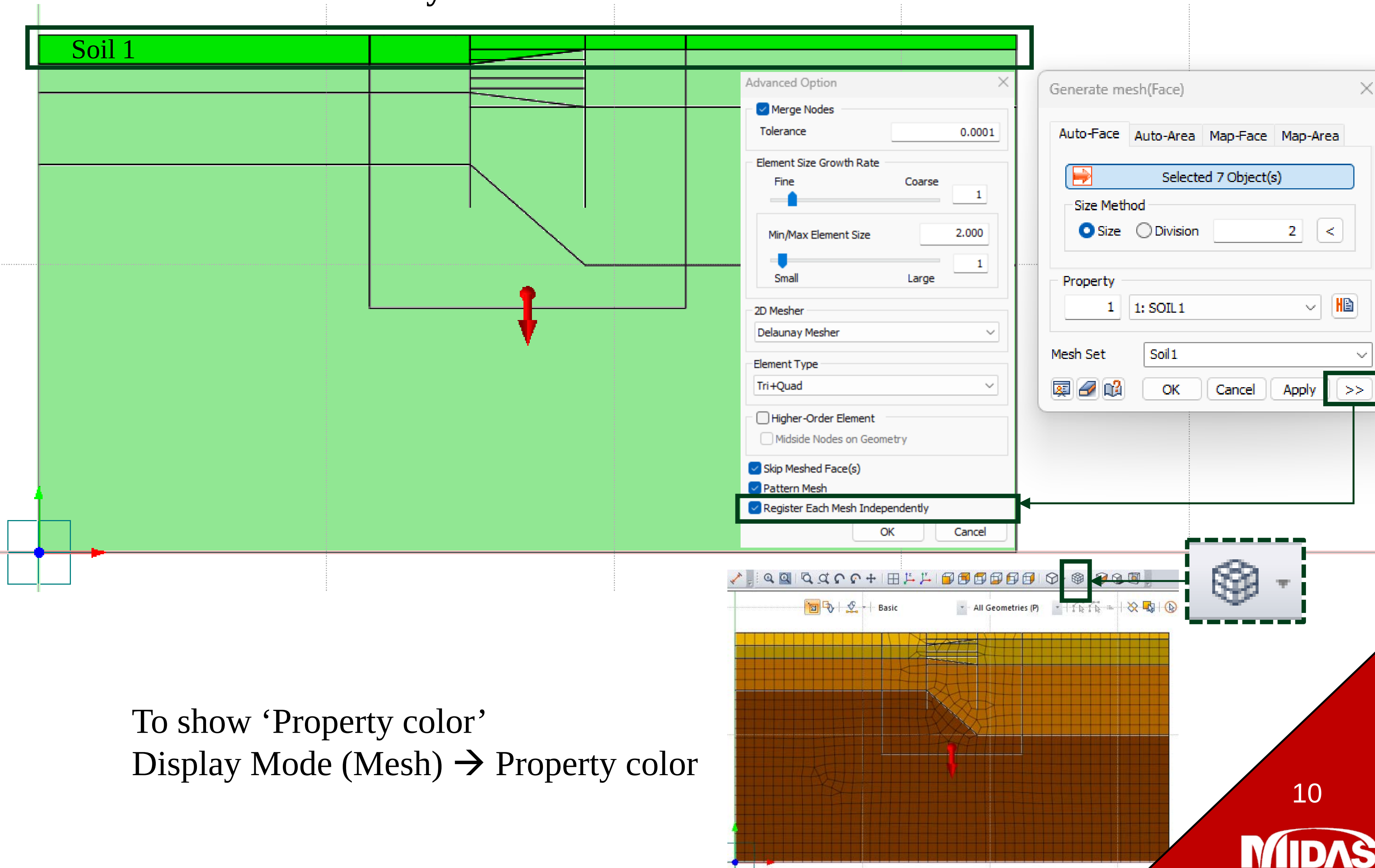
Mesh size control to increase the accuracy in plastic area

Mesh size of 0.5 m

MIDAS

MESH - GENERATE MESH

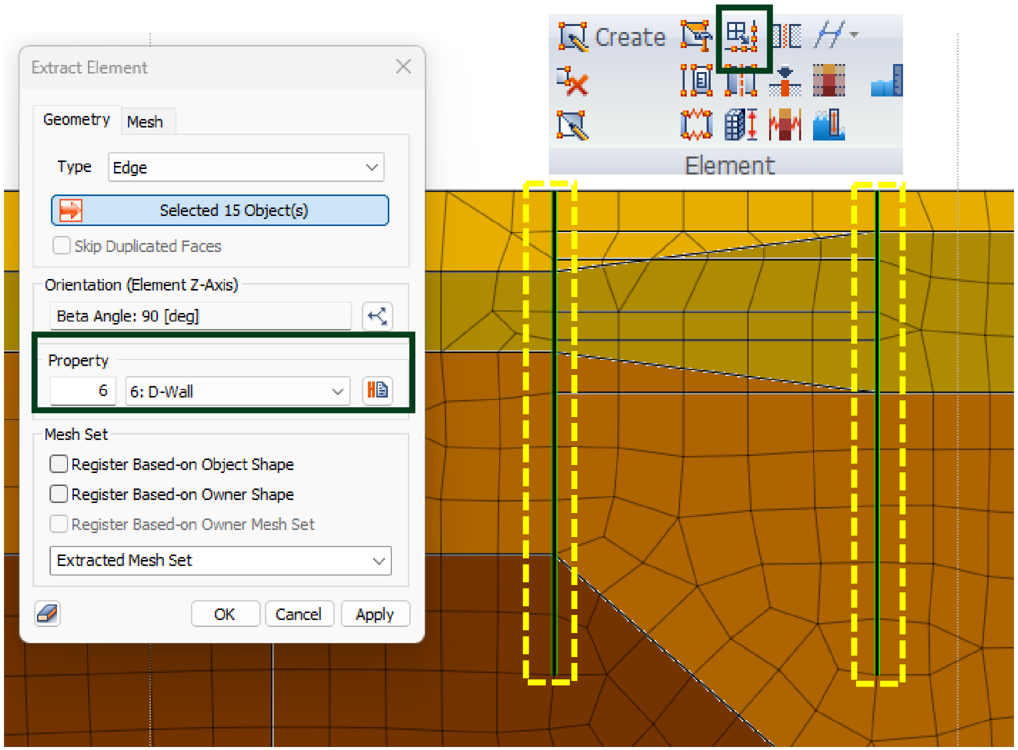
Generate mesh for soil layer 1



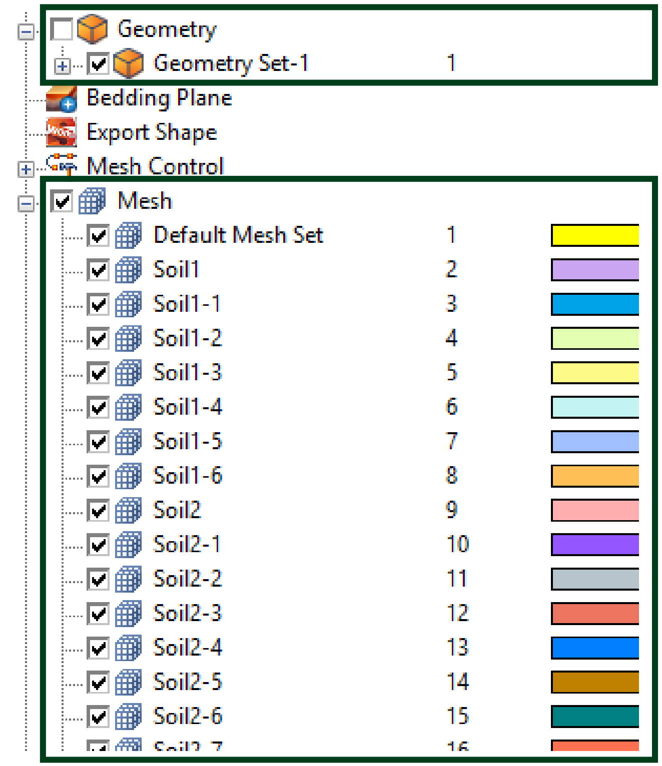
To show 'Property color'
Display Mode (Mesh) → Property color

MIDAS

MESH - EXTRACT ELEMENT

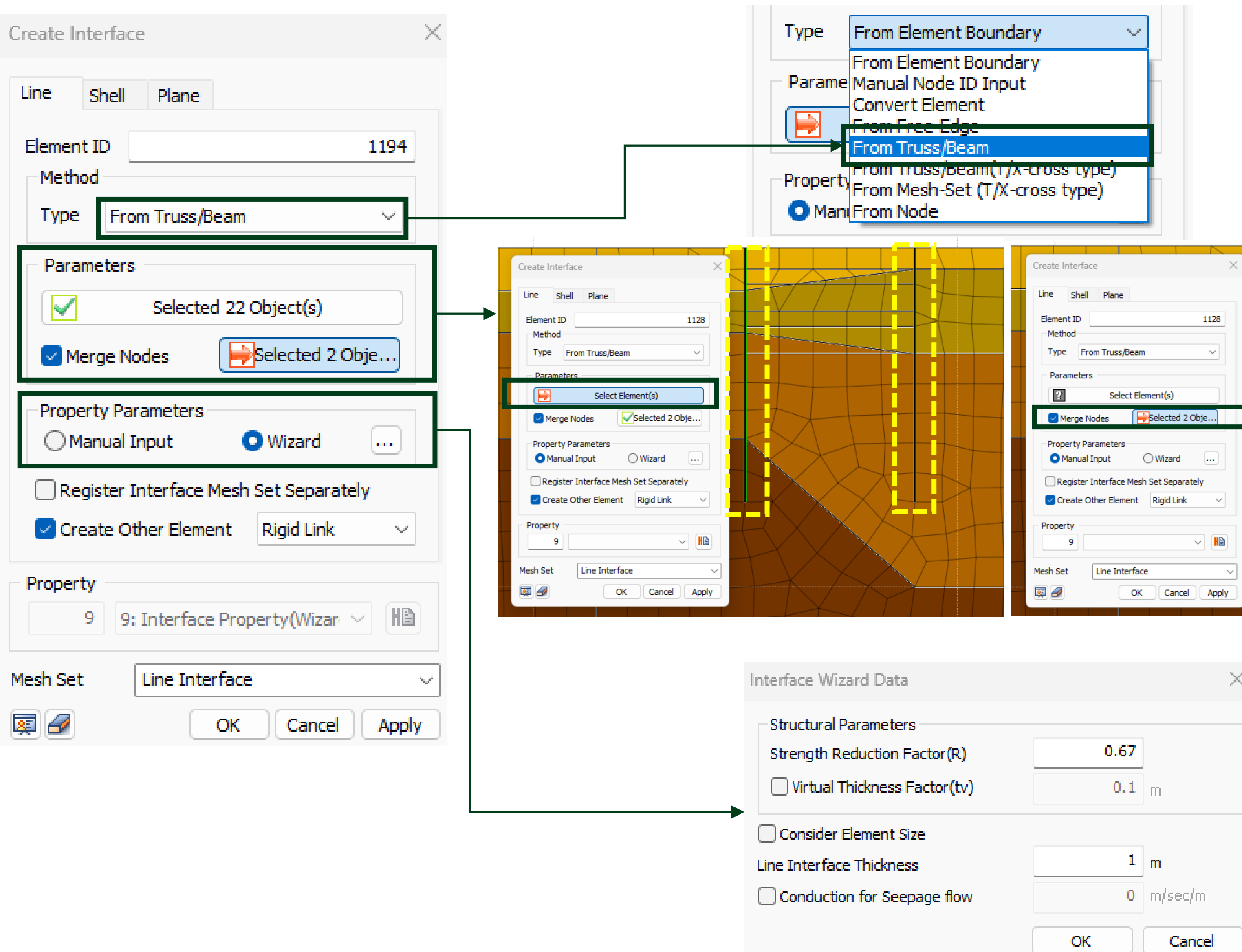


‘Extract elements’ for retaining wall



Note: By using the ‘Extract Elements’ function, it is needed to open geometry and corresponding mesh set

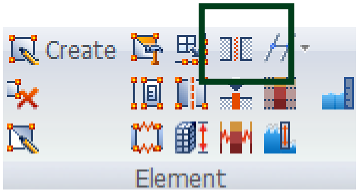
INTERFACE



Create Interface
Line
Shell
Plane
Element ID
1194
Method
Type
From Truss/Beam
Parameters
Selected 22 Object(s)
Merge Nodes
Selected 2 Obje...
Property Parameters
Manual Input
Wizard
...
Register Interface Mesh Set Separately
Create Other Element
Rigid Link
Property
9
9: Interface Property(Wizar
Mesh Set
Line Interface
OK
Cancel
Apply
Type
From Element Boundary
From Element Boundary
Manual Node ID Input
Convert Element
From Free Edge
From Truss/Beam
From Truss/Beam(T/X-cross type)
From Mesh-Set (T/X-cross type)
From Node
Interface Wizard Data
Structural Parameters
Strength Reduction Factor(R)
0.67
Virtual Thickness Factor(tv)
0.1
m
Consider Element Size
Line Interface Thickness
1
m
Conduction for Seepage flow
0
m/sec/m
OK
Cancel

INTERFACE WIZARD

Interface Wizard equation from Midas GTX NX manual



$$K_n = E_{oed,i} / t_v$$

$$K_t = G_i / t_v$$

$$C_i = R \times C_{soil}$$

Here,

$$E_{oed,i} = 2 \times G_i \times (1-v_i)/(1-2 \times v_i)$$

$$G_i = R^2 \times G_{soil}$$

$$G_{soil} = E/(2(1+v_{soil}))$$

Where:

K_n: Normal Stiffness Modulus

K_t: Shear Stiffness Modulus

t_v: Virtual Thickness Factor

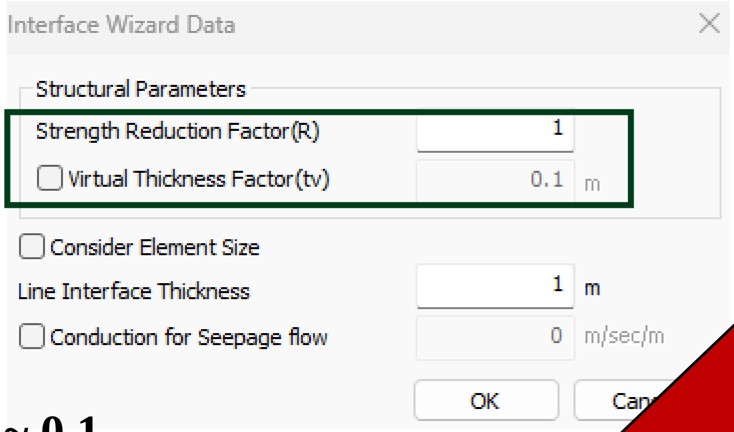
R: Strength Reduction Factor

C_i: Interface Cohesion

Normal Stiffness Modulus (Kn) is the elasticity modulus for **bonding and un-bonding behavior** in the normal direction to the interface element

Shear stiffness modulus (Kt) is the elasticity modulus for **slip behavior** in the normal direction to the interface element

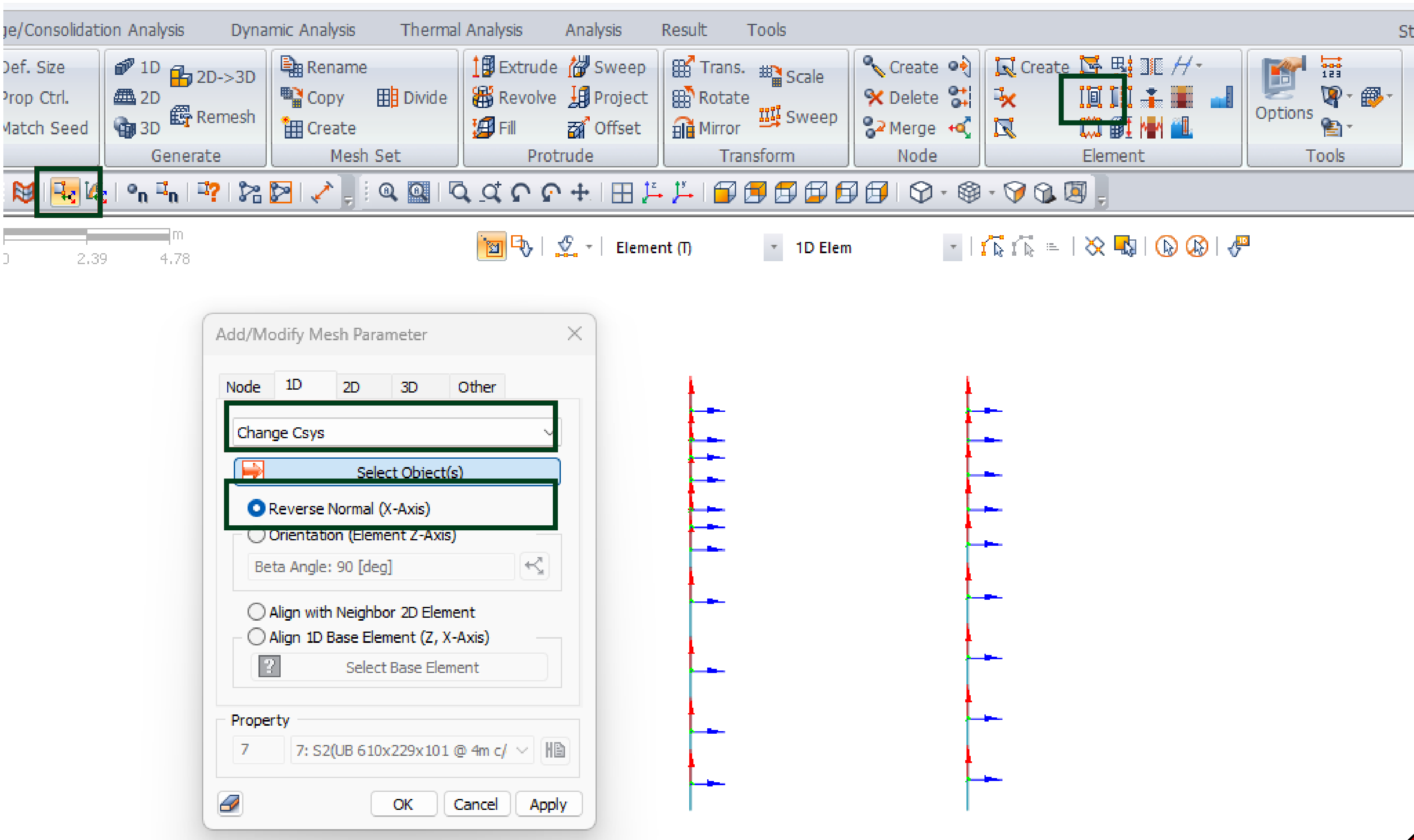
Strength Reduction Factor	R
Sandy Soil/Steel Material	0.6~0.7
Clay/ Steel Material	0.5
Sandy Soil/ Concrete	0.8~1.0
Clay / Concrete	0.7 ~ 1.0



The general **Virtual Thickness Factor range is 0.01 ~ 0.1**

(If the stiffness is high, use a smaller value)

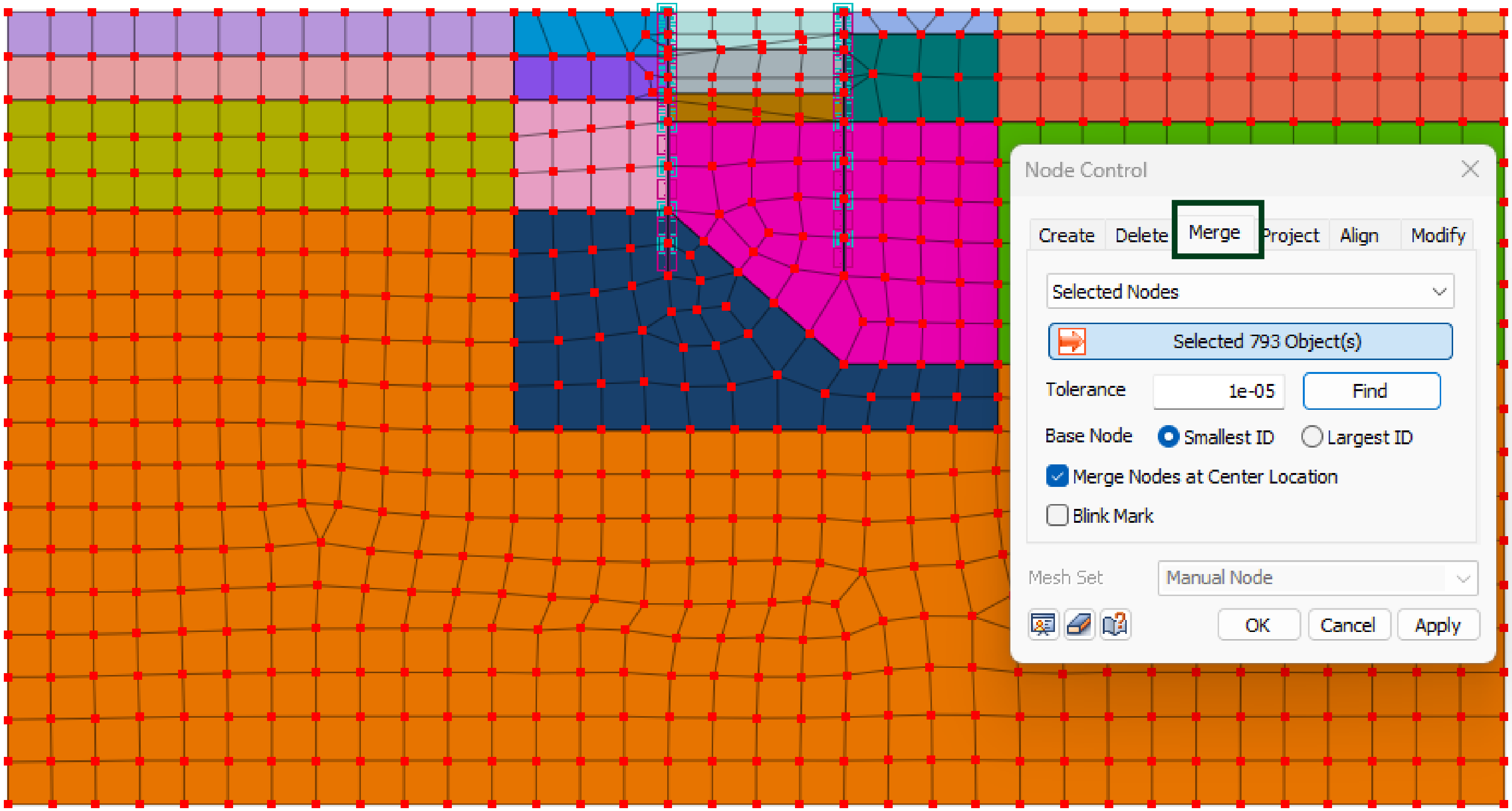
LOCAL AXIS READJUSTMENT



Changing the axis of the local retaining wall axis

MIDAS

MERGE NODE

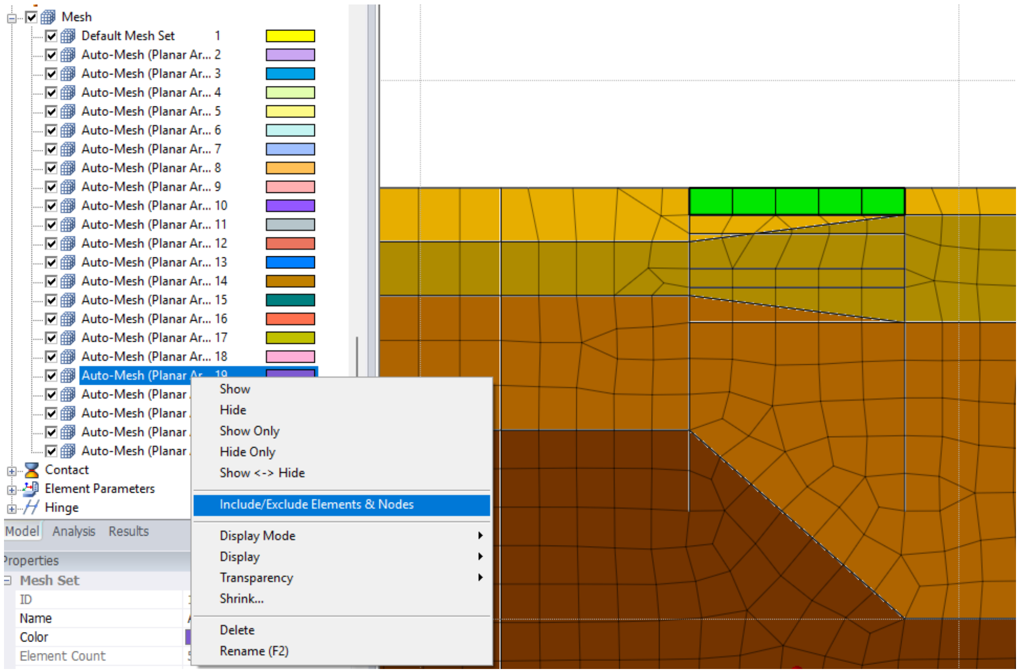


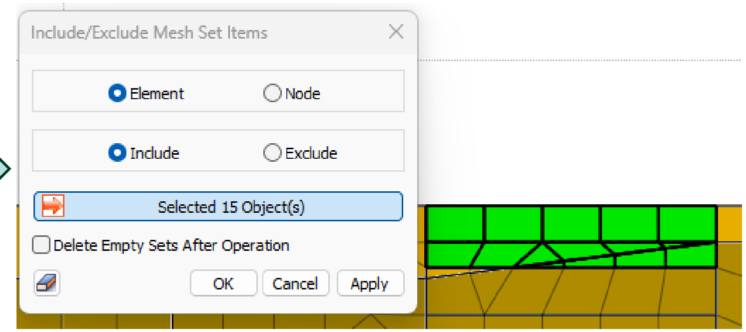
Checking the merge node function for interface

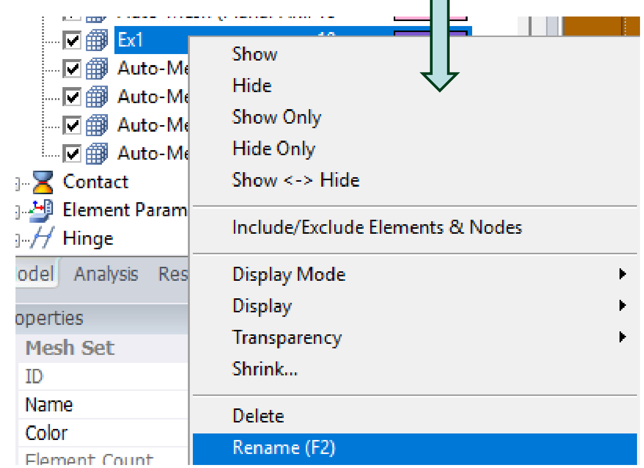
Selecting all nodes → Mesh → Node → Merge → Find

MESH GROUPING

Model Tree

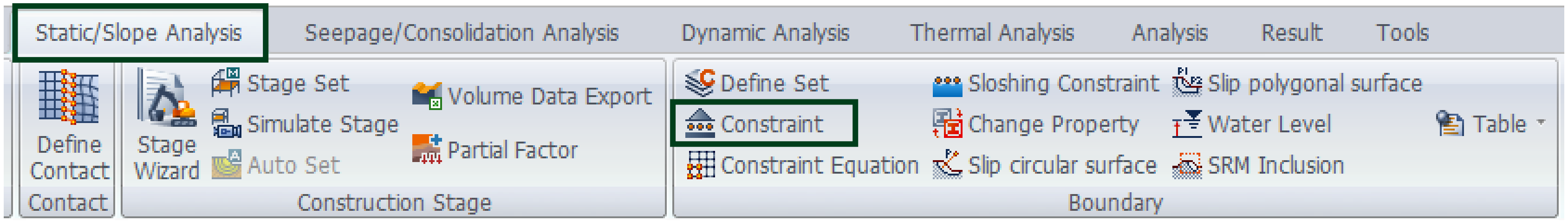


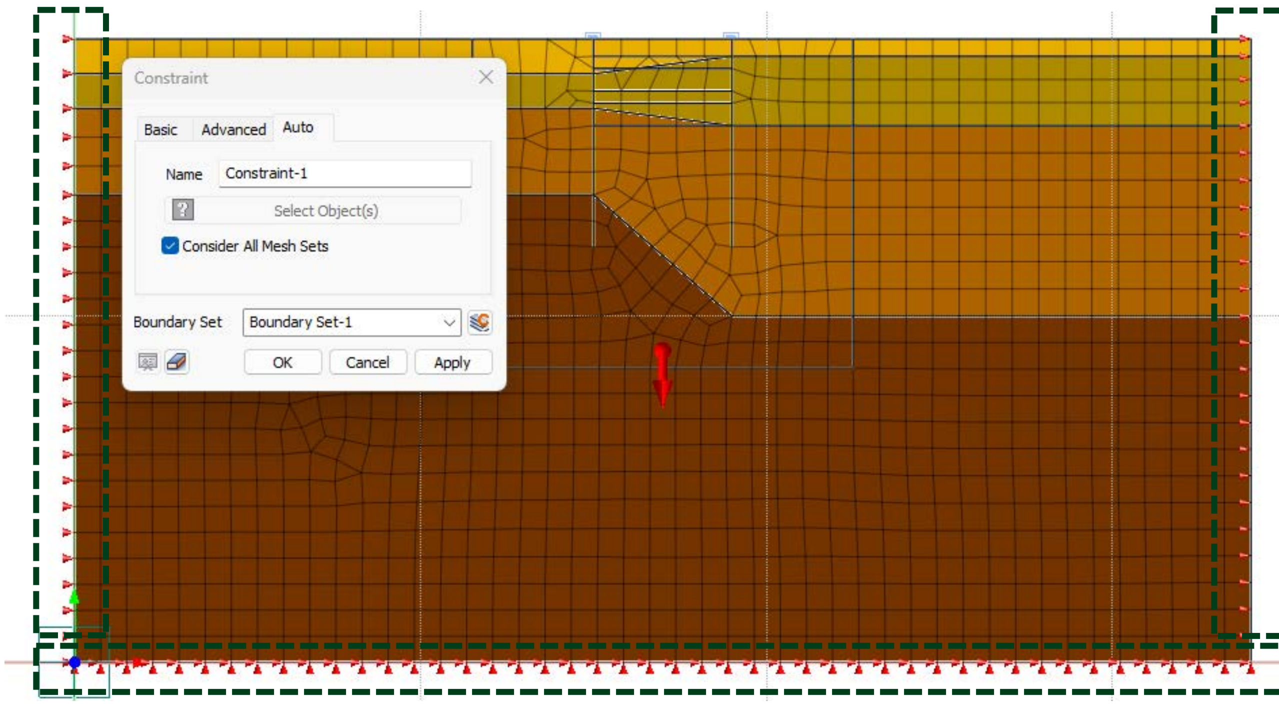




NOTE: Step is generally done for organizing meshes

BOUNDARY CONDITION



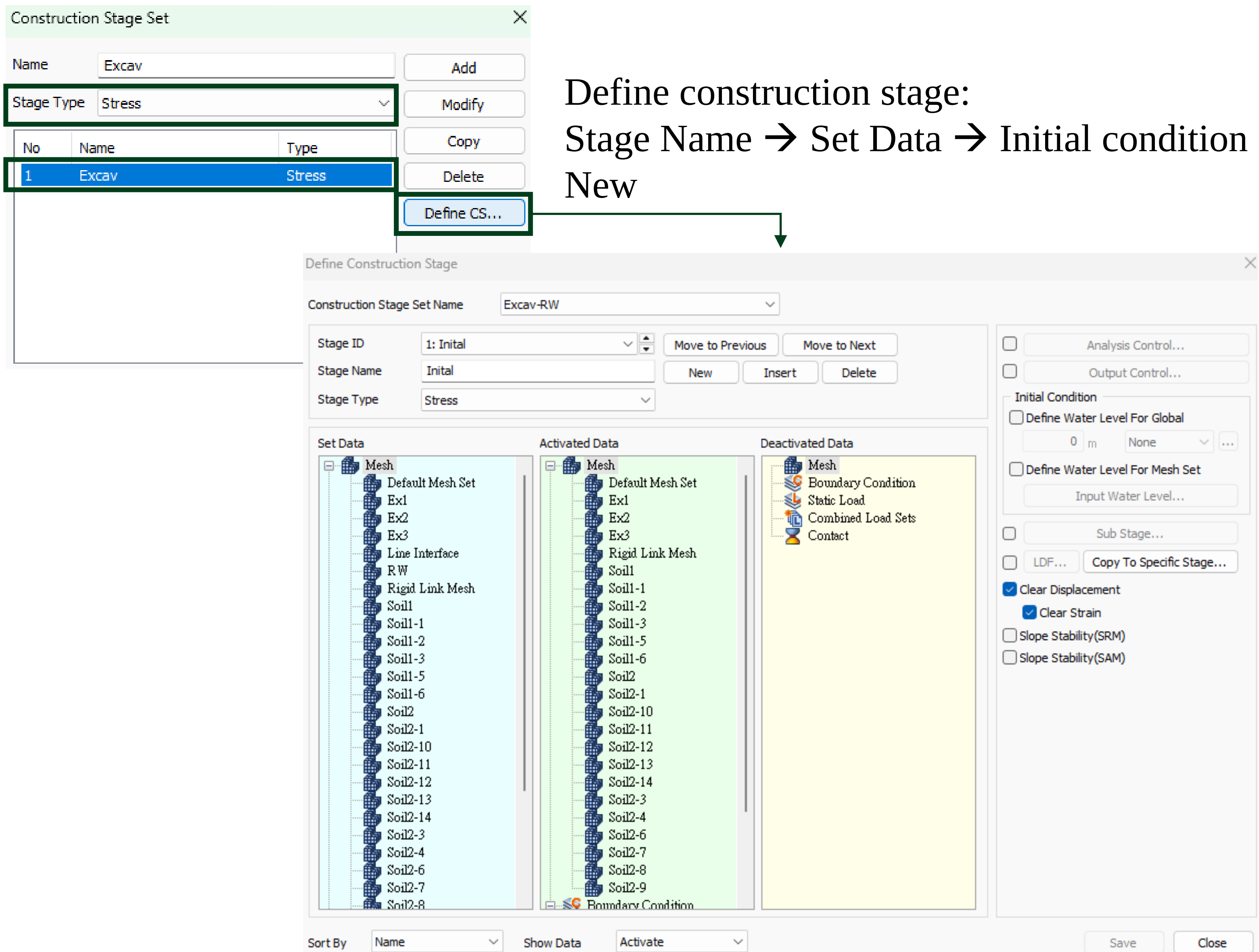


Boundary condition is created

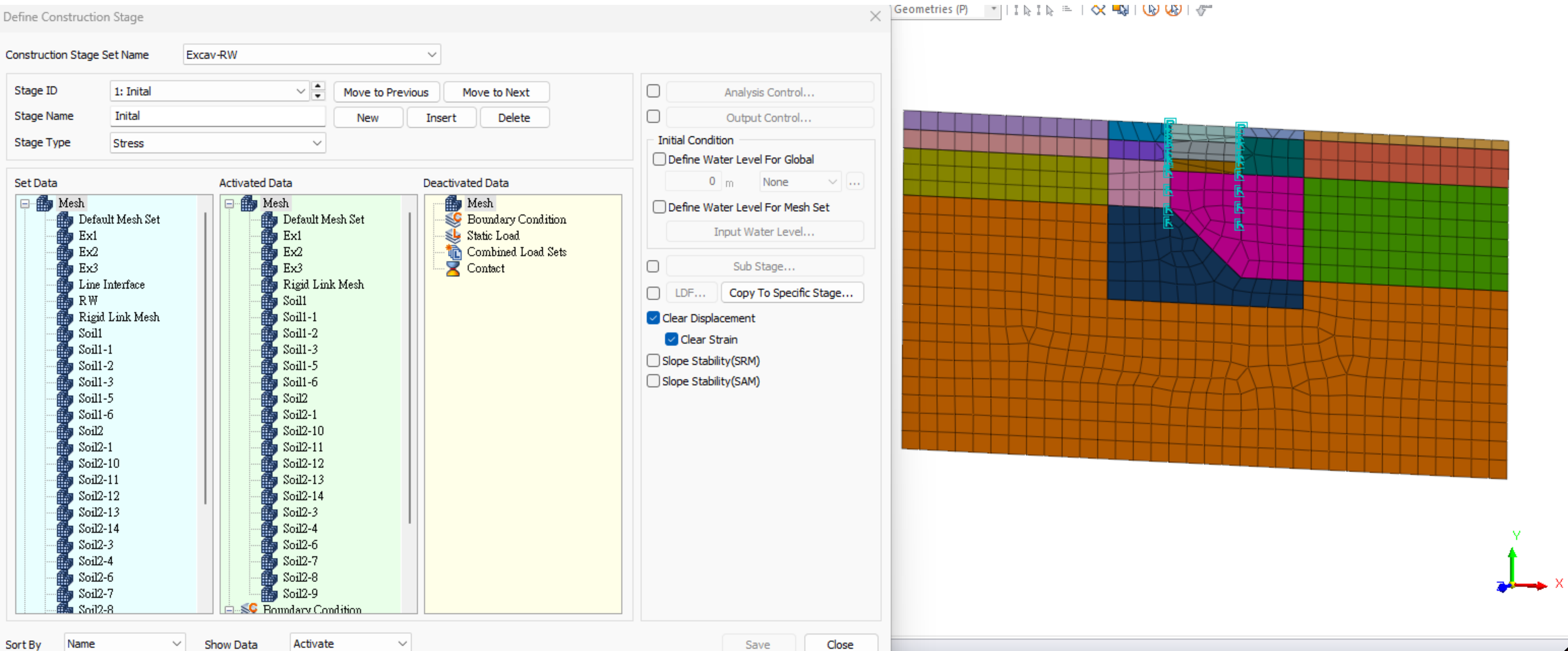
NOTE: Automatic constraint creates a fix restrictions (restrained x and y axes) on the bottom part of soil model, and pin restriction on the sides (restrained y axis).

CONSTRUCTION STAGE

Define construction stage:
Stage Name → Set Data → Initial condition → Save → New

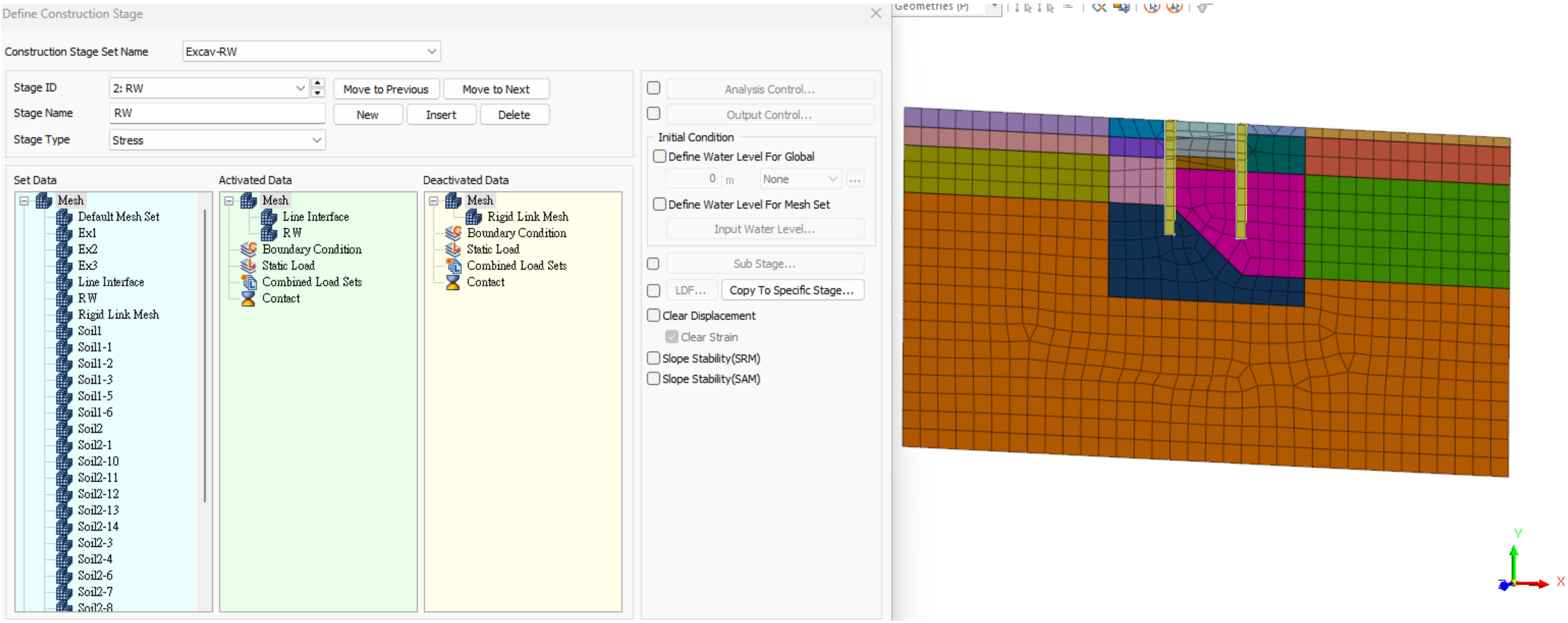


CONSTRUCTION STAGE 1



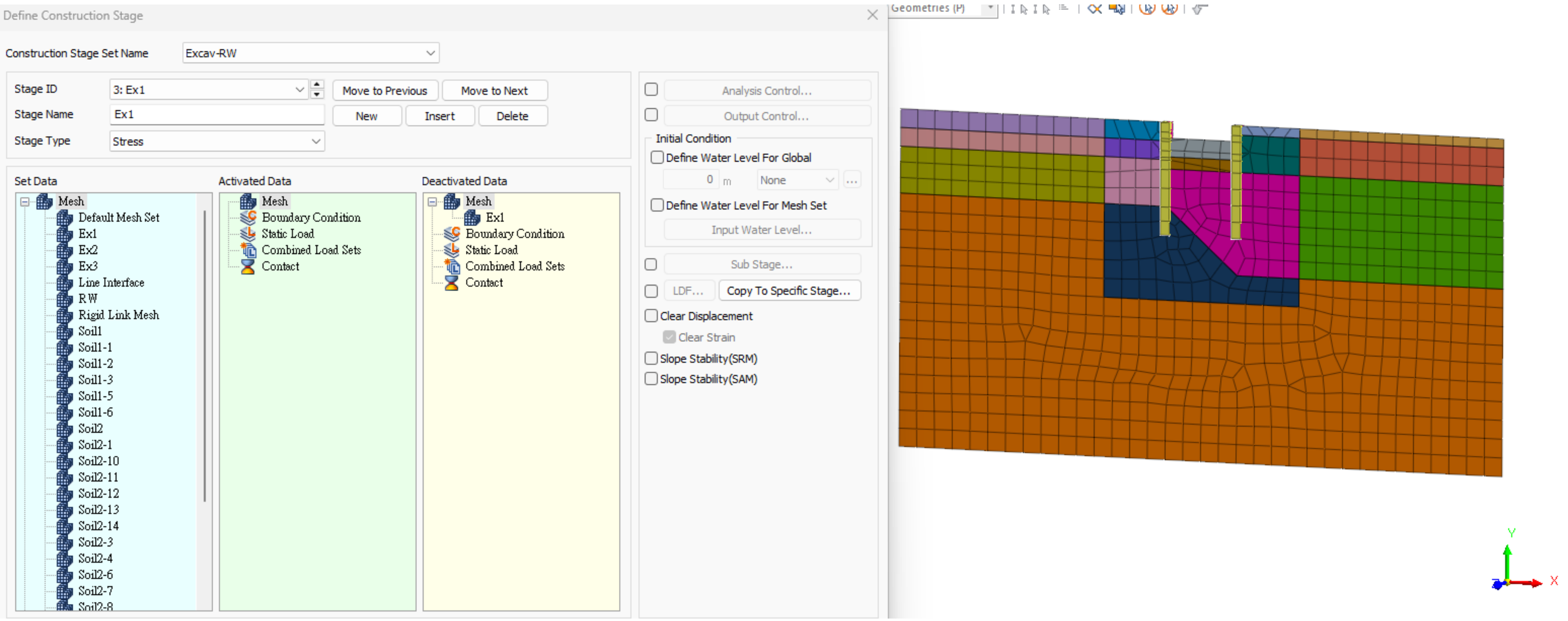
Stage 1: Initial Conditions

CONSTRUCTION STAGE 2



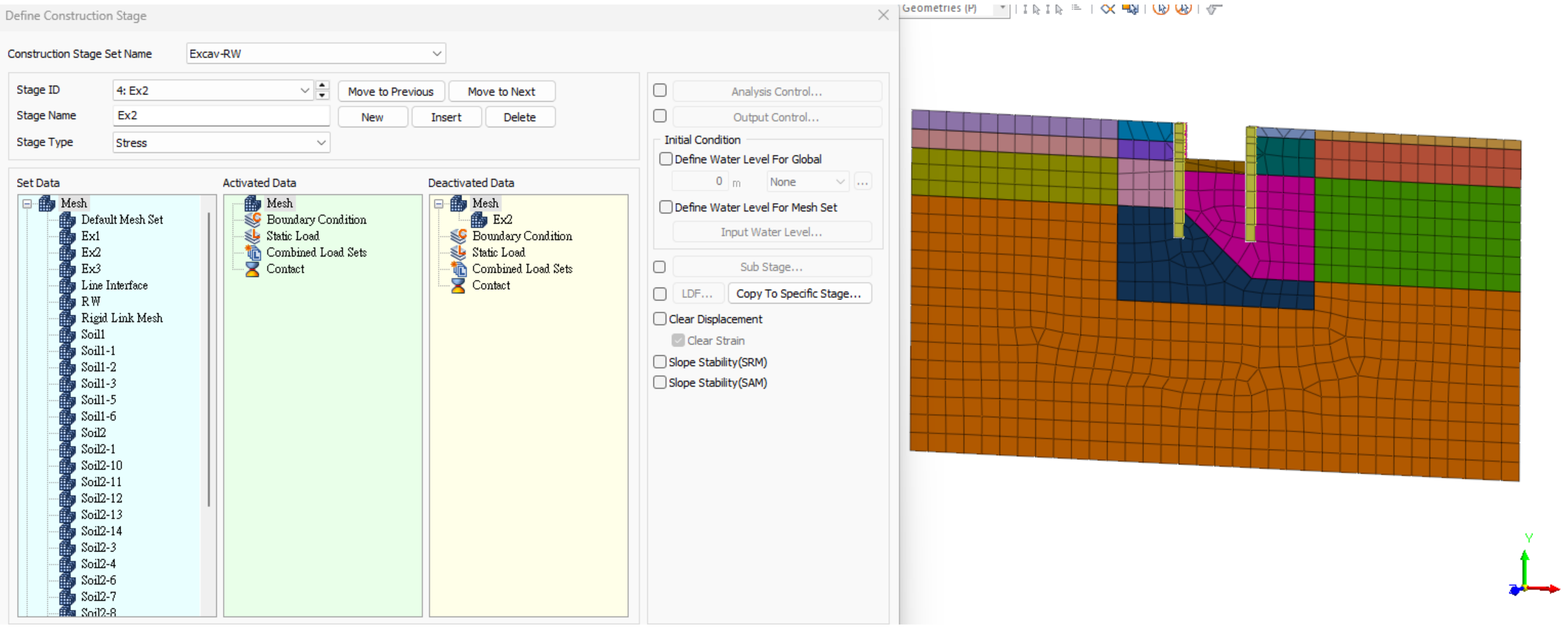
Stage 2: Retaining Wall

CONSTRUCTION STAGE 3



Stage 3: Excavation Layer 1

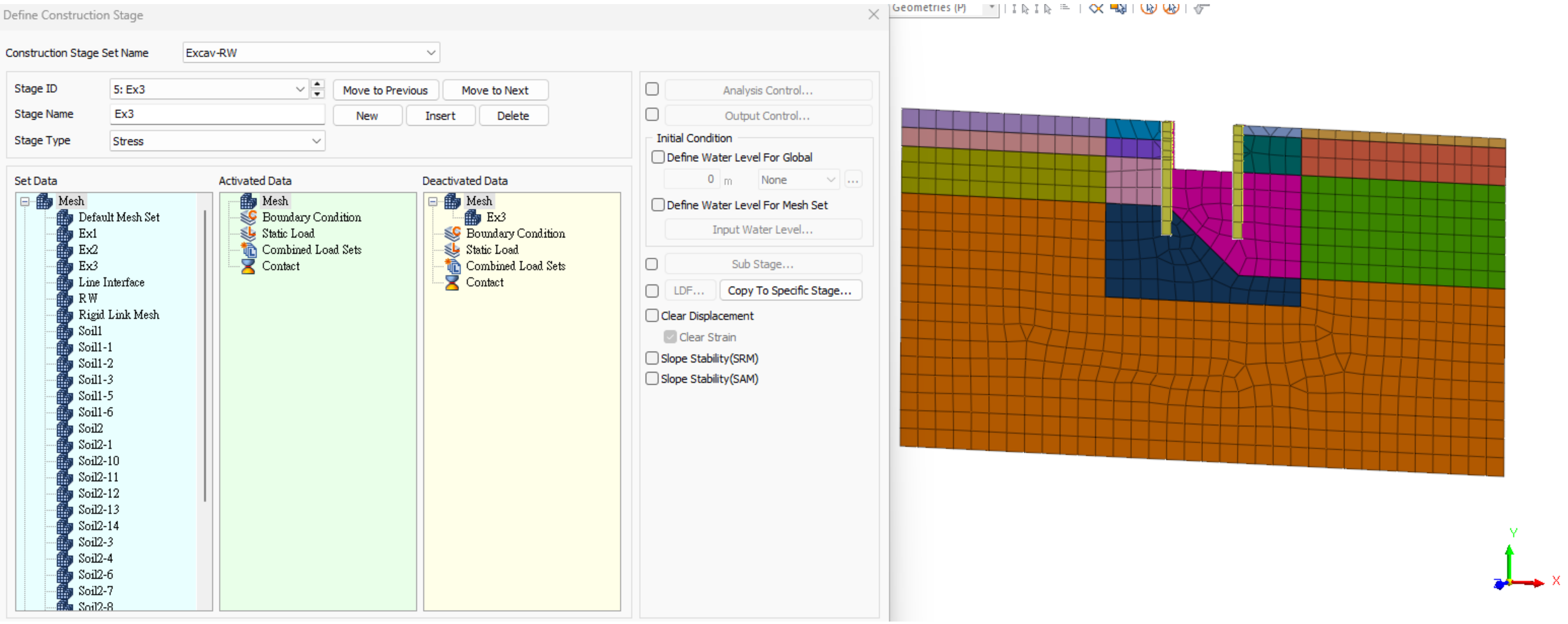
CONSTRUCTION STAGE 4



Stage 4: Excavation Layer 2

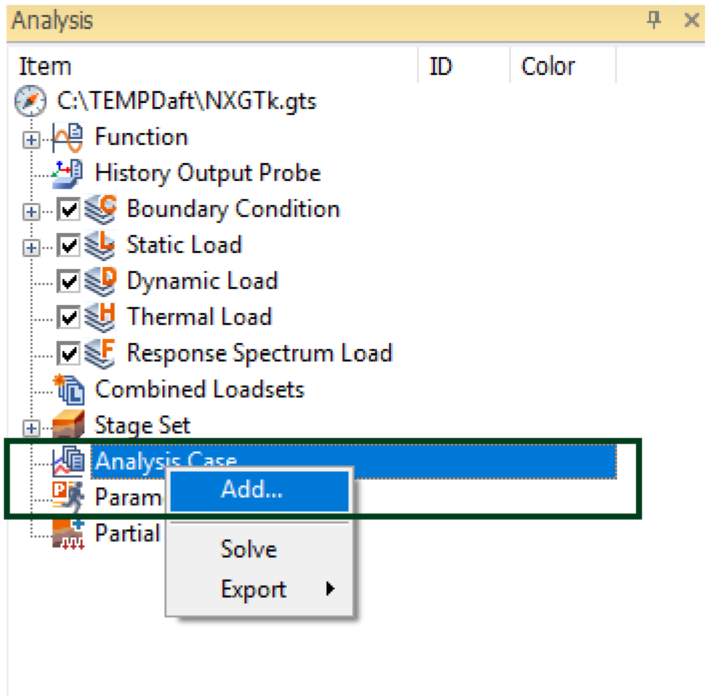
MIDAS

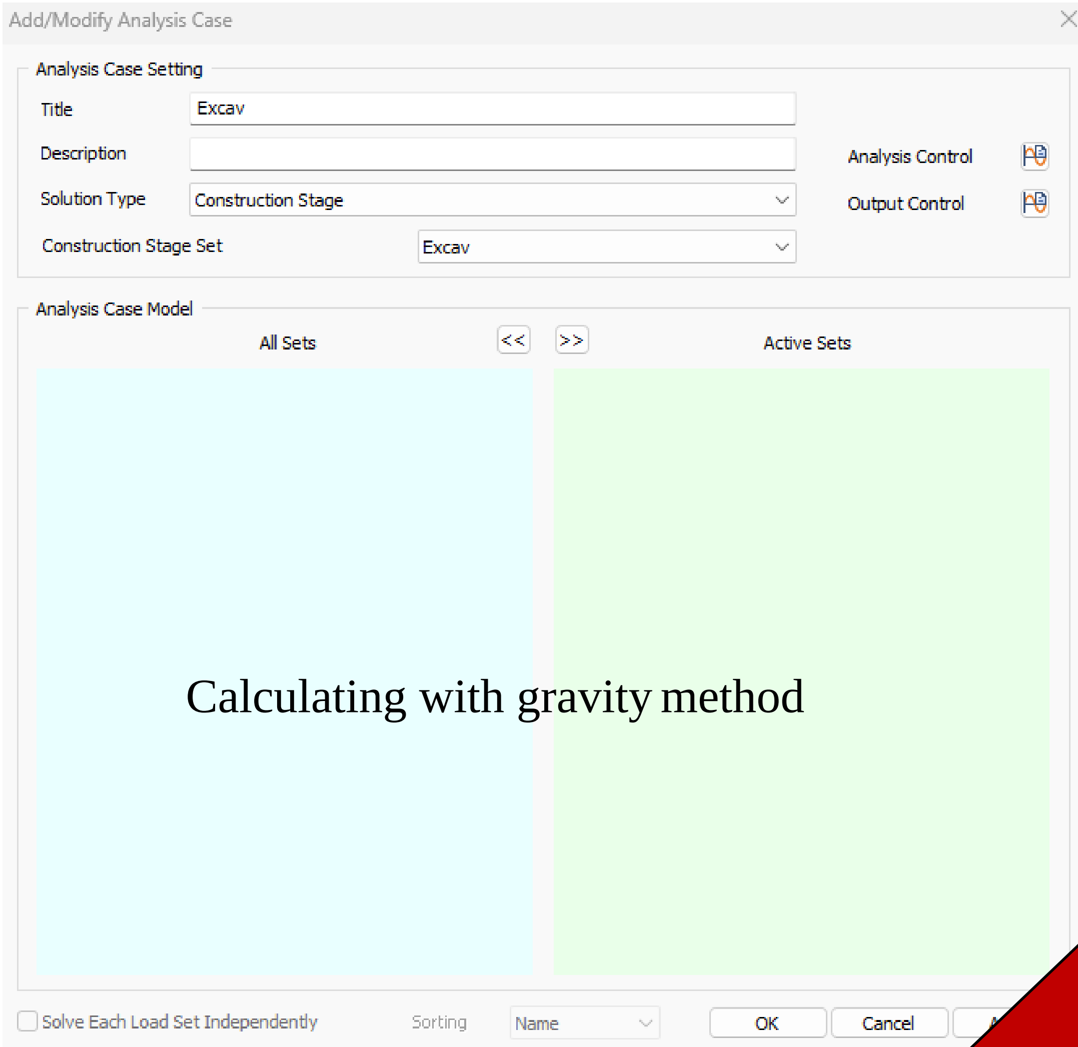
CONSTRUCTION STAGE 4



Stage 4: Excavation Layer 3

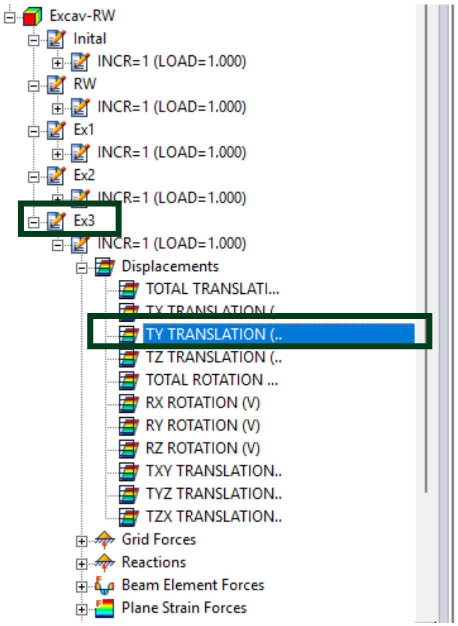
ANALYSIS CONTROL

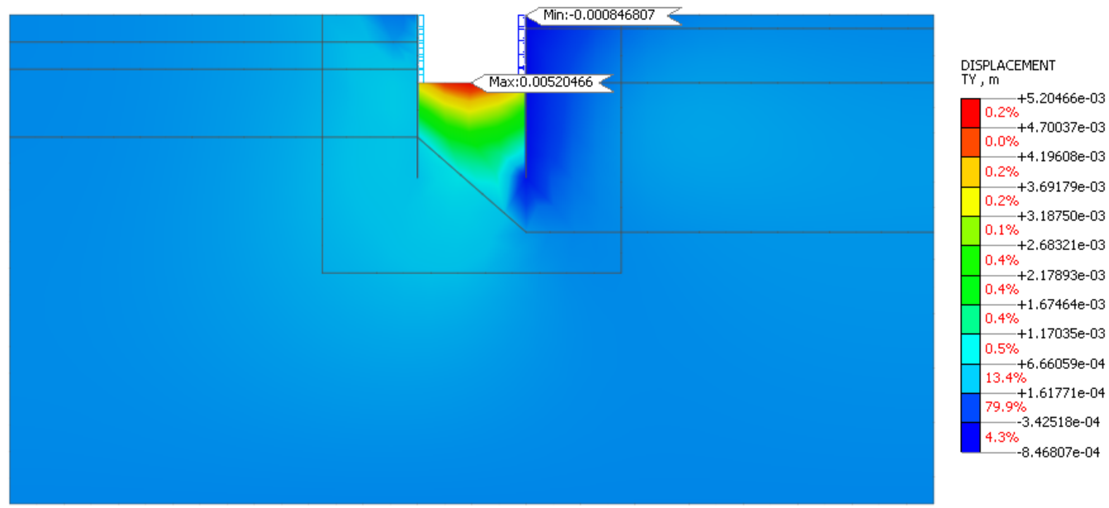


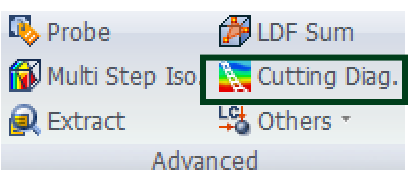


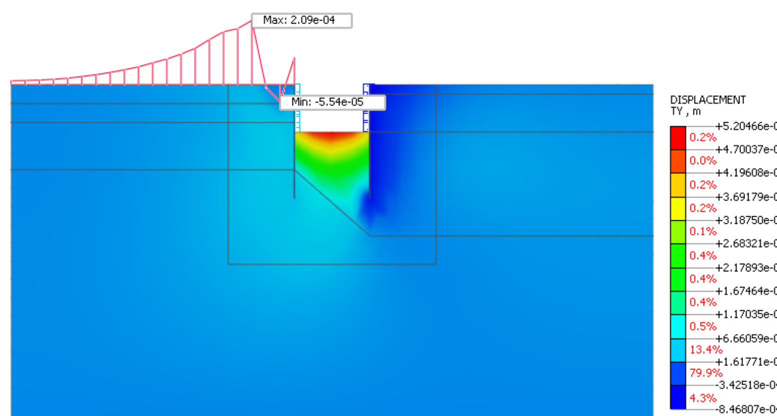
Calculating with gravity method

RESULT - VERTICAL DISPLACEMENT



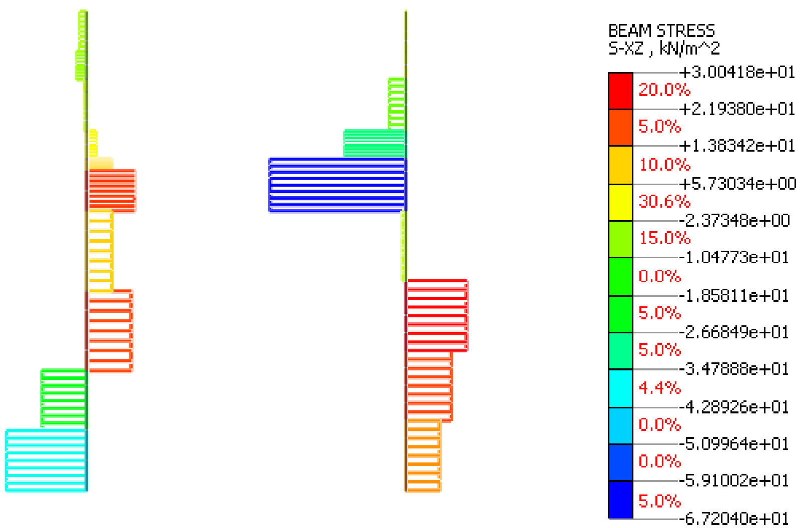


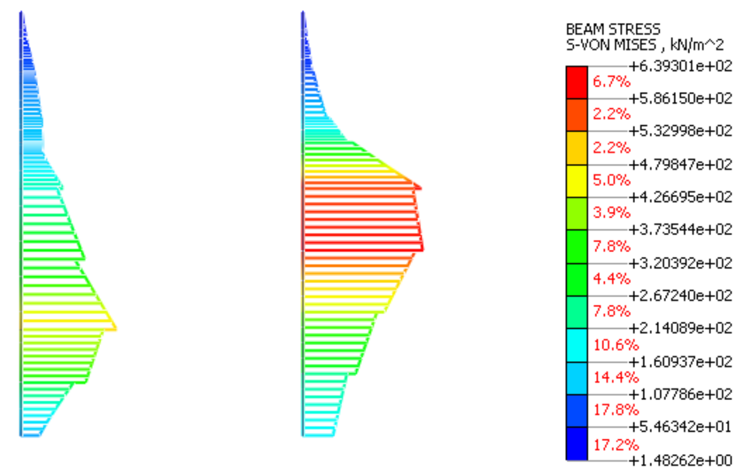




Result → Advanced → Cutting Diagram

RESULT - BEAM STRESSES



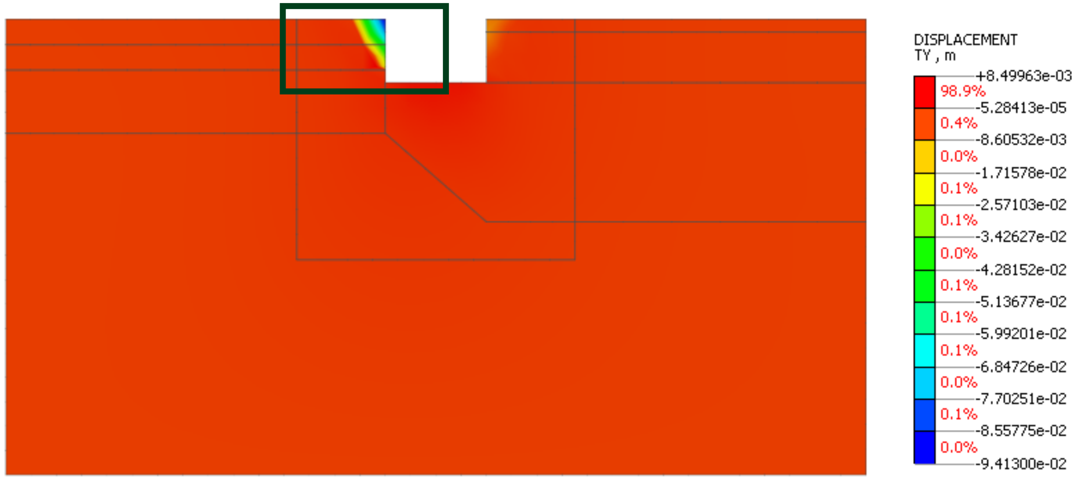


Shear force
Retaining Wall

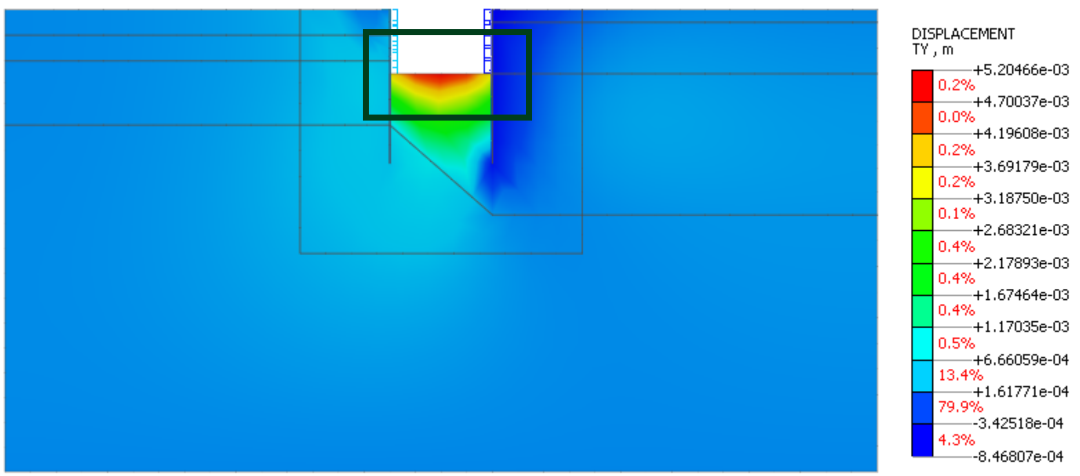
Bending moment
Retaining Wall

RESULT - COMPARISON

Total Displacement Without RW



Total Displacement With RW

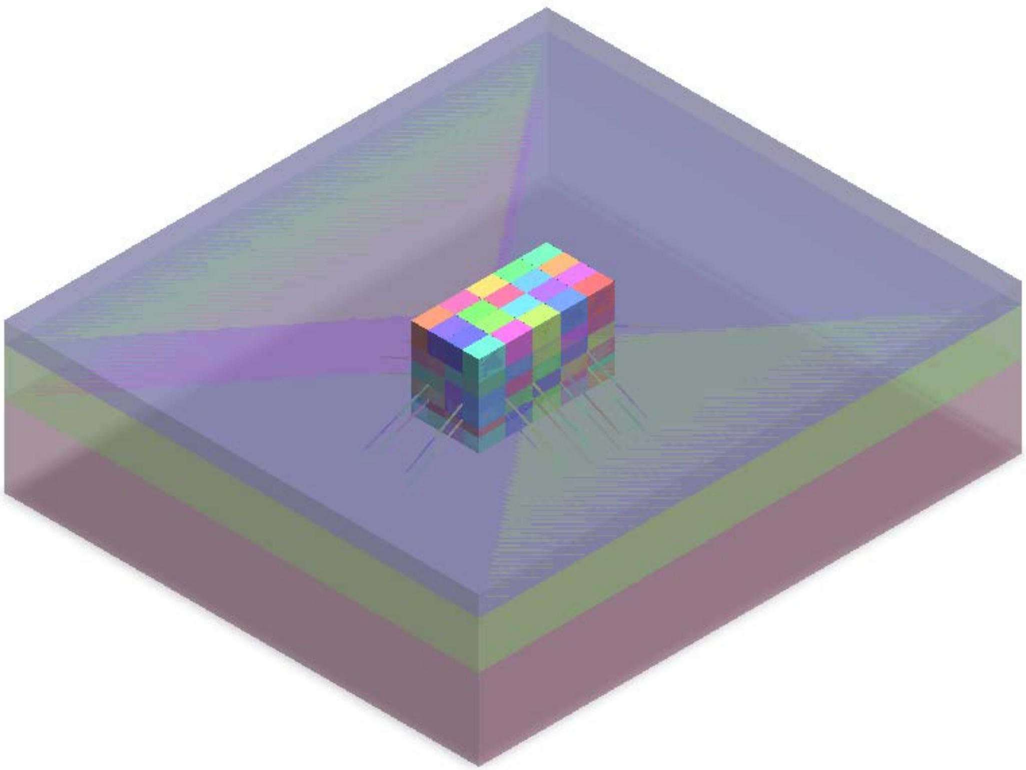


The comparison highlights the effect of the Retaining Wall in the excavation model.

3D Excavation with Soil Retaining System

沈約翰
john@midasuser.com.tw

PROJECT CASE OVERVIEW

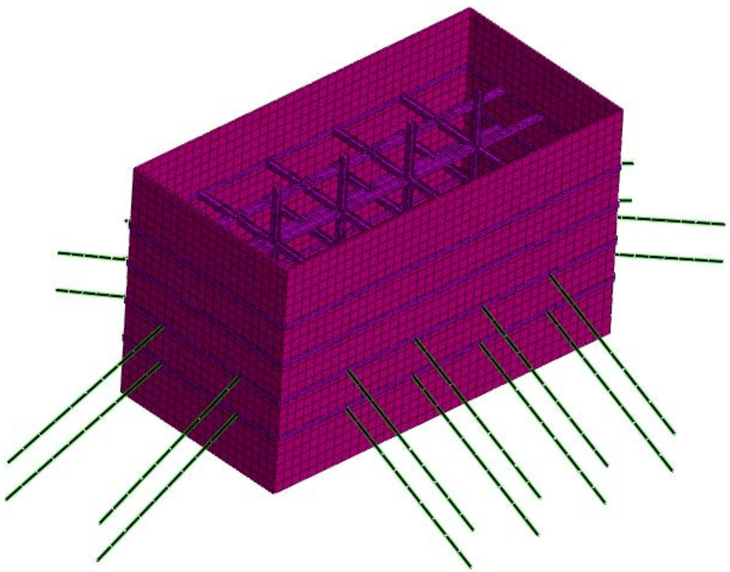


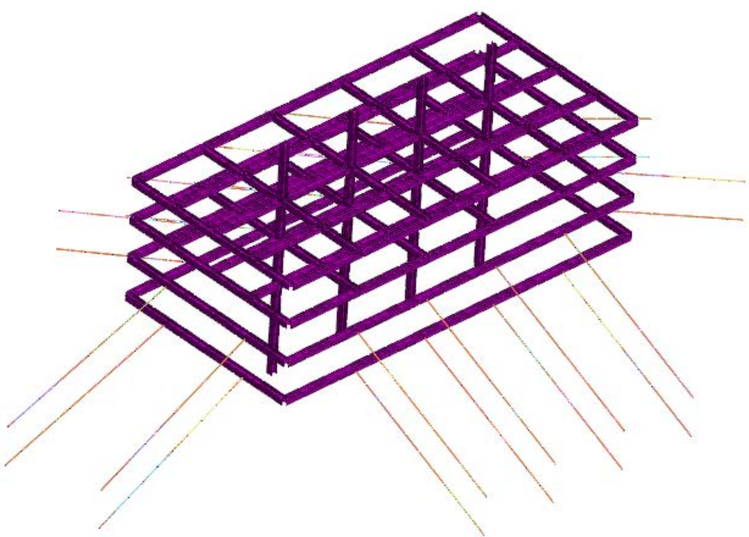
Project Overview

Rectangular Excavation : 10 x 20m
Depth: 12m

Soil Property	Depth (m)
Buried Layer	0 - 3
Colluvium	3 - 7
Weathering Soil	7 - 12

PROJECT CASE OVERVIEW





Structural Support Overview

Element	Details
Retaining Wall	All throughout the excavation faces
King Post	4 pieces; Laid along the centerline of the longitudinal distance
Prestressed Anchors	8 pieces per longitudinal face; 4 pieces per transverse face
Braces	4 sets starting from the first excavation layer
Struts	2 sets starting from the first excavation layer

DESIGN FLOW

Pre-Processes

1. Geometry / Model setup
2. Material and Properties
3. Boundary Conditions
4. Mesh Generation
   - Geotechnical elements
   - Structural elements

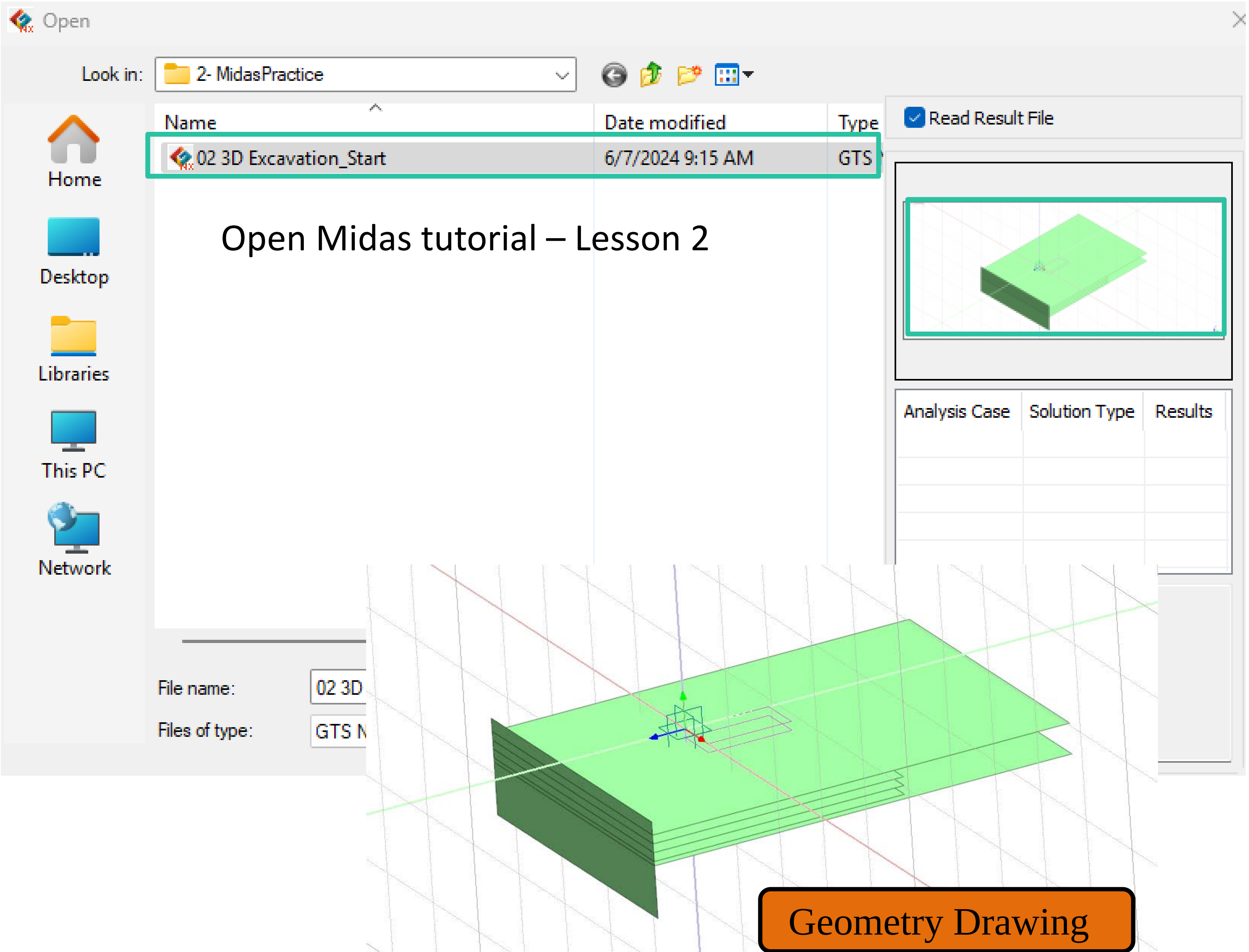
Solving

1. Analysis Case
   - Linear
   - Non Linear
   - Construction Stage
   - Slope Stability
   - Etc.
2. Stage Sets
3. Analysis Control Settings
   - Dynamic settings
   - Thermal settings
   - Age settings
   - Etc.

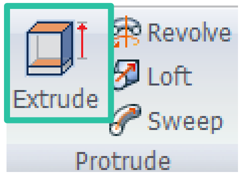
Post-Processes

1. Data Validation
2. Result Interpretation
3. Analysis Presentation
   - Graphs
   - Contour plots
   - Animations
4. Data Exports
   - Data Transfer
   - System Integration

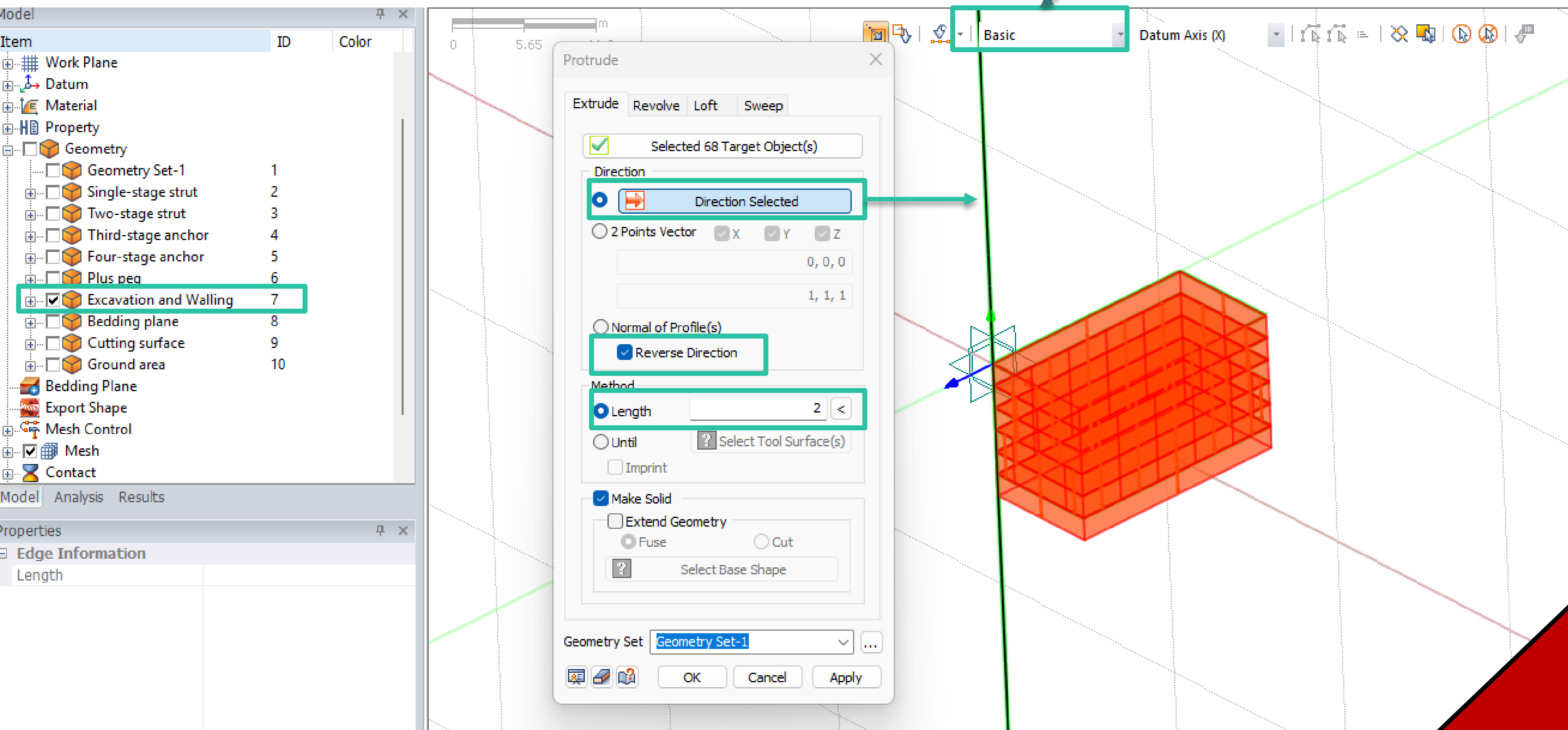
3D MODEL IMPORT



GEOMETRY MODELING

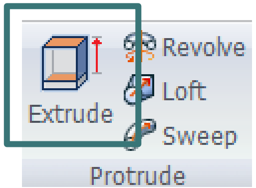


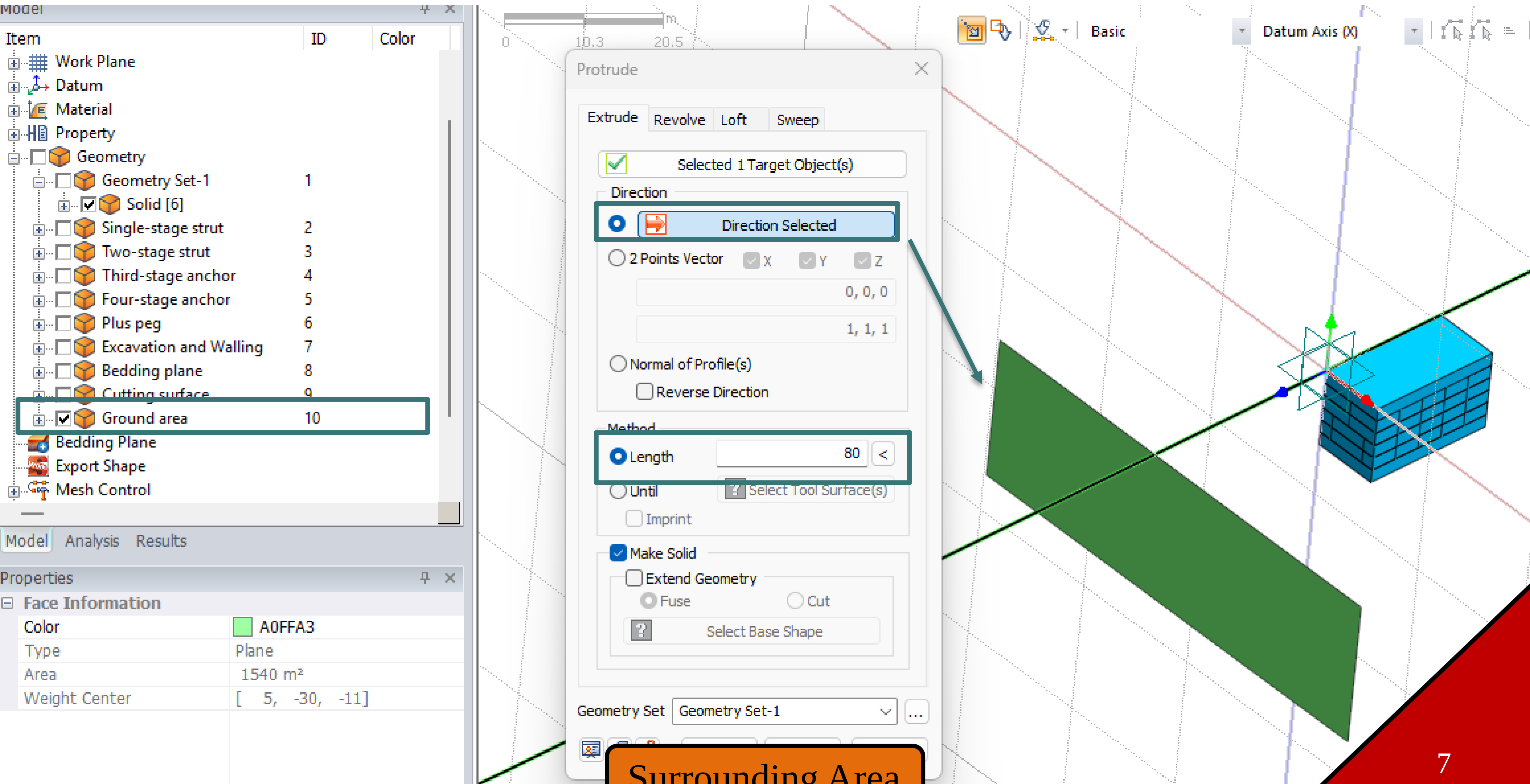
Change into 'Edge', then 'Select All'



Excavation Area

GEOMETRY MODELING

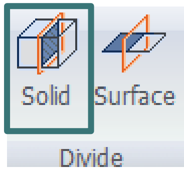




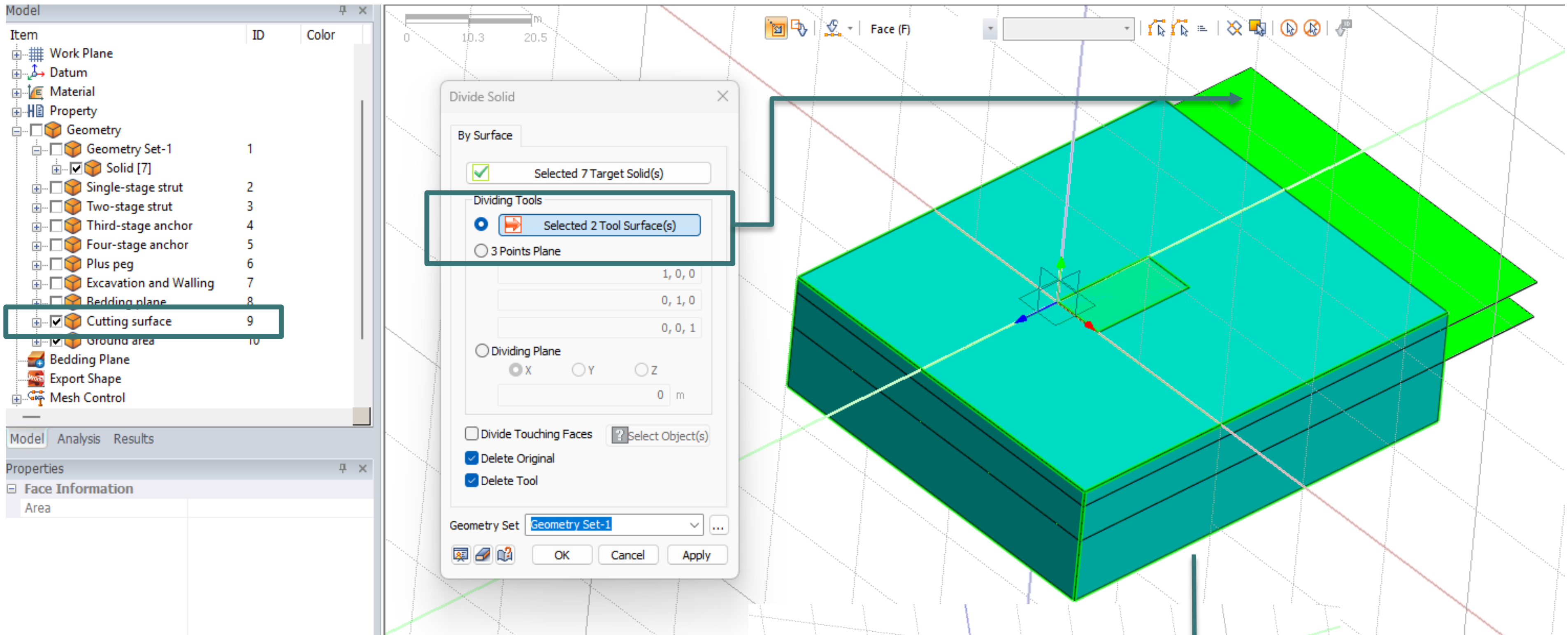
Surrounding Area

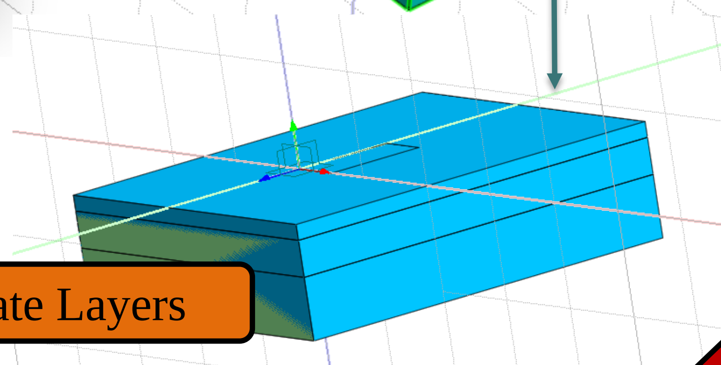
MIDAS

GEOMETRY MODELING



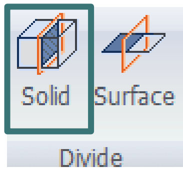
Divide Solid Function creates partition for the soil model

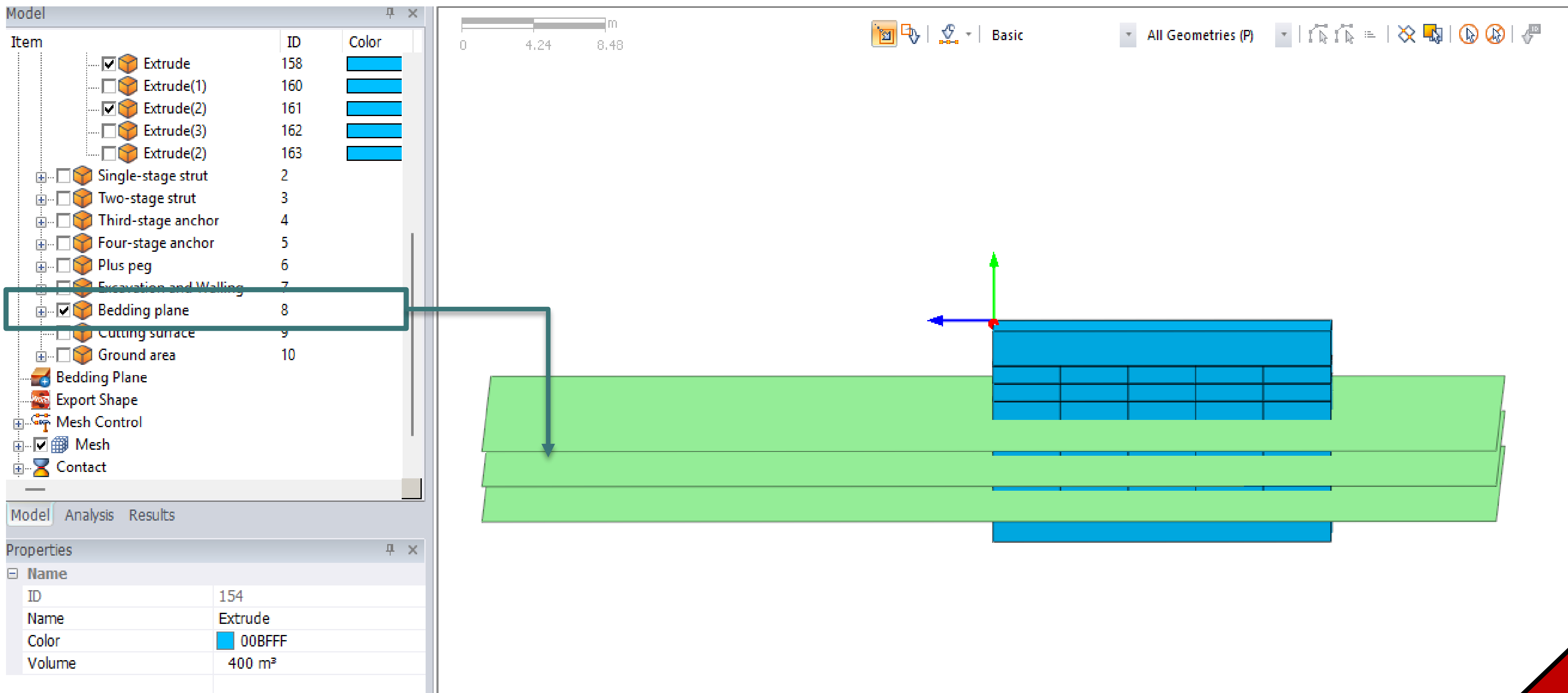




Create Layers

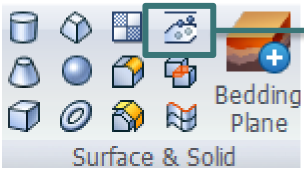
GEOMETRY MODELING

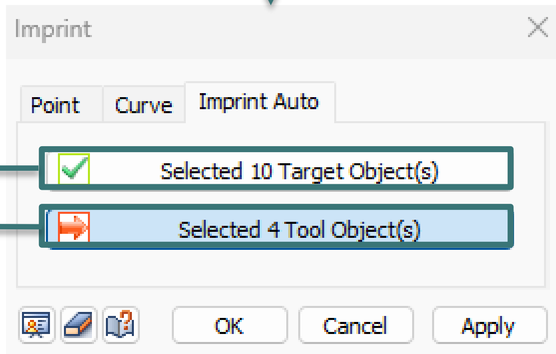




Bedding Plane

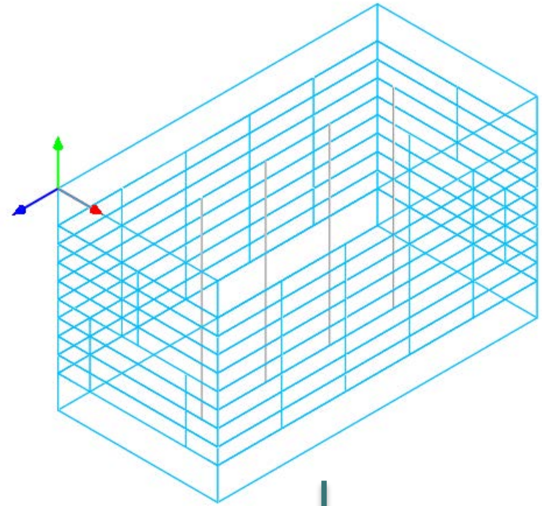
GEOMETRY MODELING

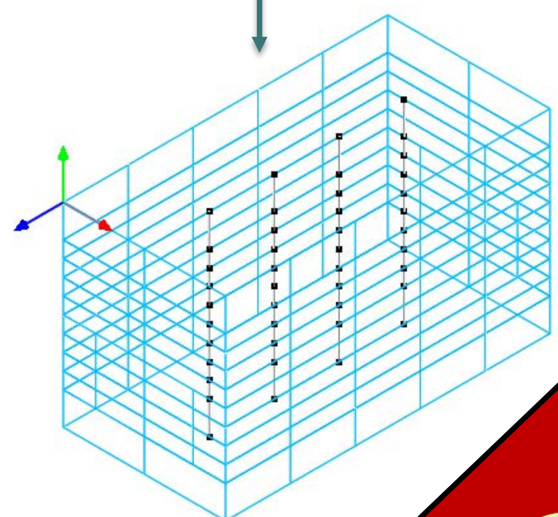




Excavation Area

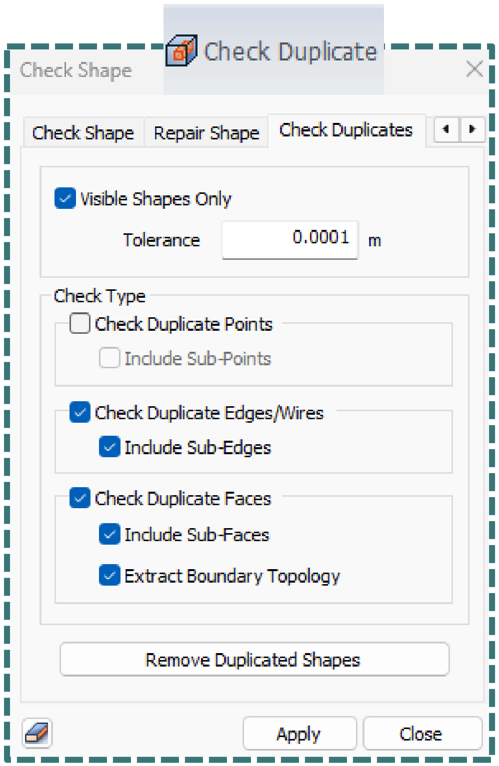
Plug Pegs

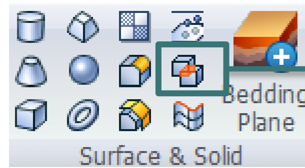


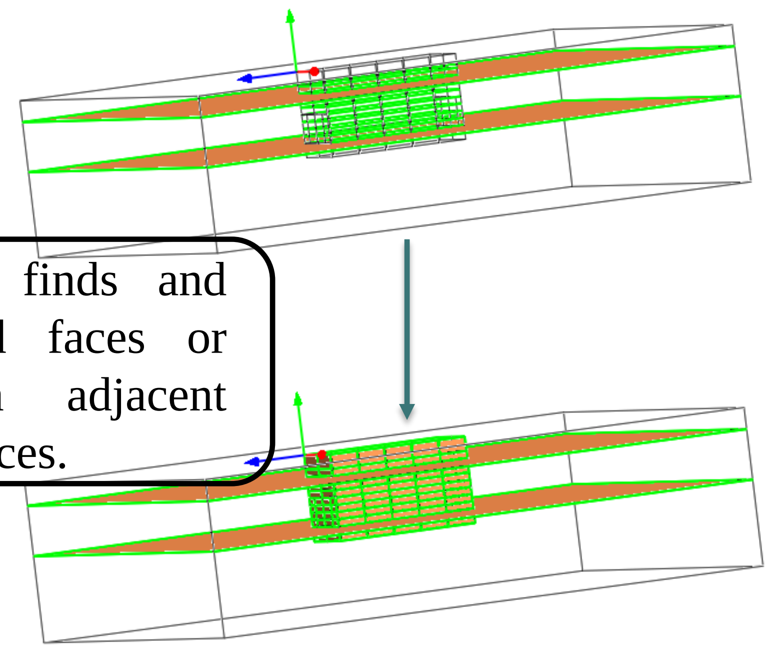


Imprint Function projects edges (pegs) onto another face

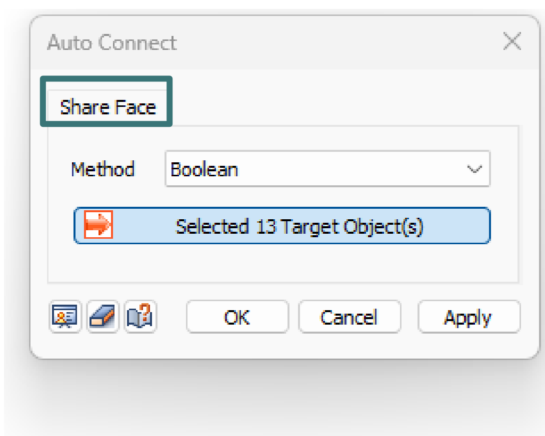
Imprint

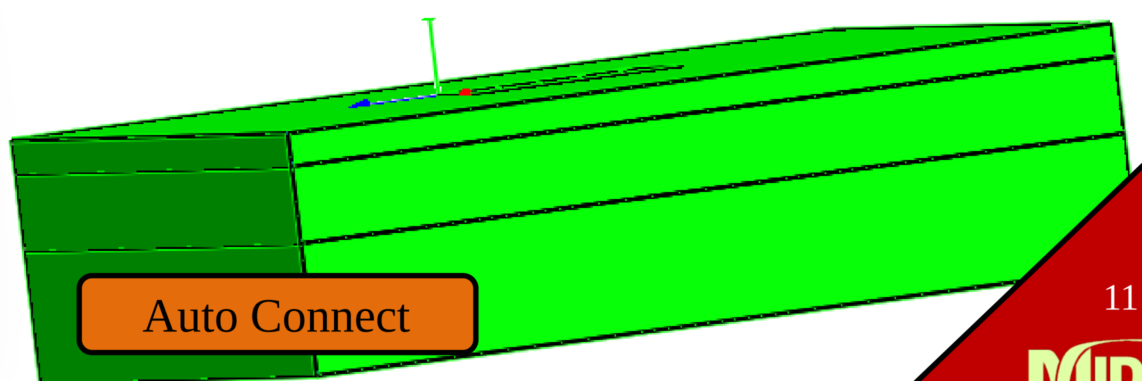






Auto Connect finds and connects shared faces or edges between adjacent volumes or surfaces.





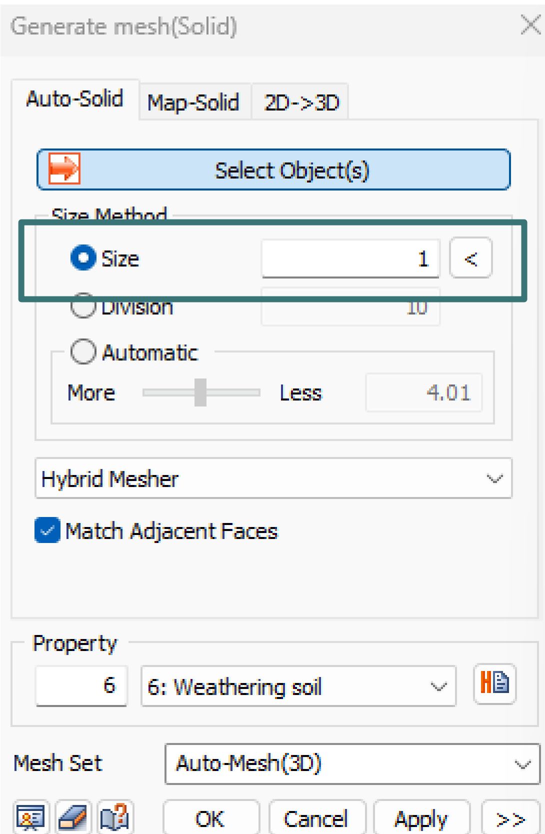
Auto Connect

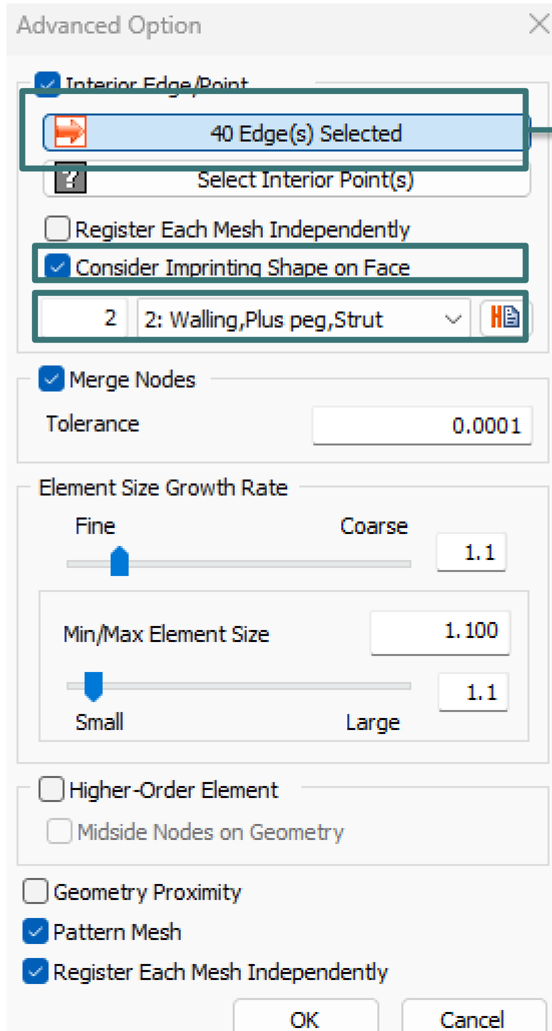
PROPERTIES - SOIL MATERIAL

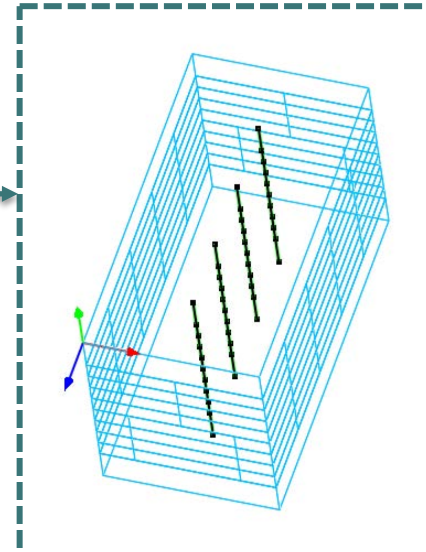
Name	Buried layer	Colluvium	Weathering soil
Material	Isotropic	Isotropic	Isotropic
Model Type	Hardening Soil	Hardening Soil	Hardening Soil
Poisson's Ratio	0.333	0.306	0.384
Unit Weight	16	17	20
K0	0.5	0.441	0.74
Unit Weight (Saturated)	20	20	22
Initial Void Ratio	0.5	0.5	0.5
Drainage Parameters	Drained	Drained	Drained
Permeability	1	1	1
E50ref	22,000	43,000	150,000
Eoedref	22,000	43,000	150,000
Eurref	66,000	129,000	450,000
Failure Ratio	0.9	0.9	0.9
Reference Pressure	12	47	110
Power of Stress Level Dependency	0.5	0.5	0.5
Friction Angle	30	34	38
K0nc	0.5	0.441	0.384
Dilatancy Angle	0	4	5
Cohesion	5	10	15

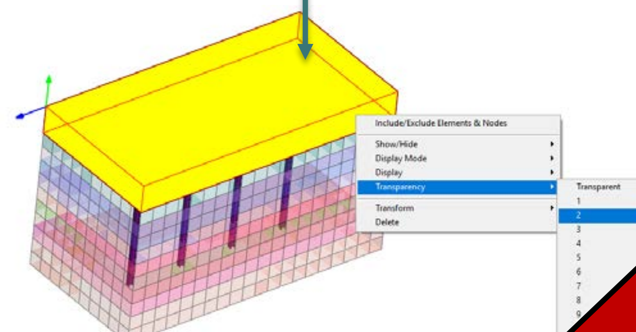
PROPERTIES- STRUCTURAL MATERIAL

Name	Structure material 1	Structure material 2
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus	210,000,000	200,000,000
Poisson's Ratio	0.3	0.3
Unit Weight	76.98	76.98



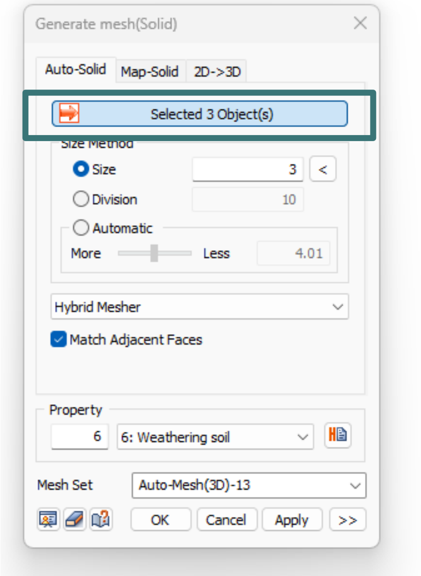


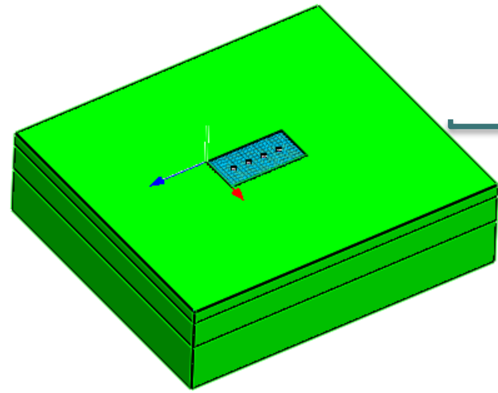


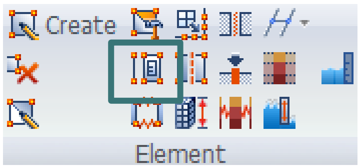


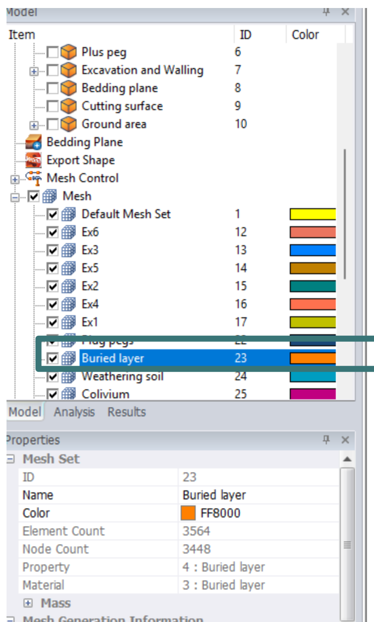
NOTE: Generate meshes with nodes connected to interior edges

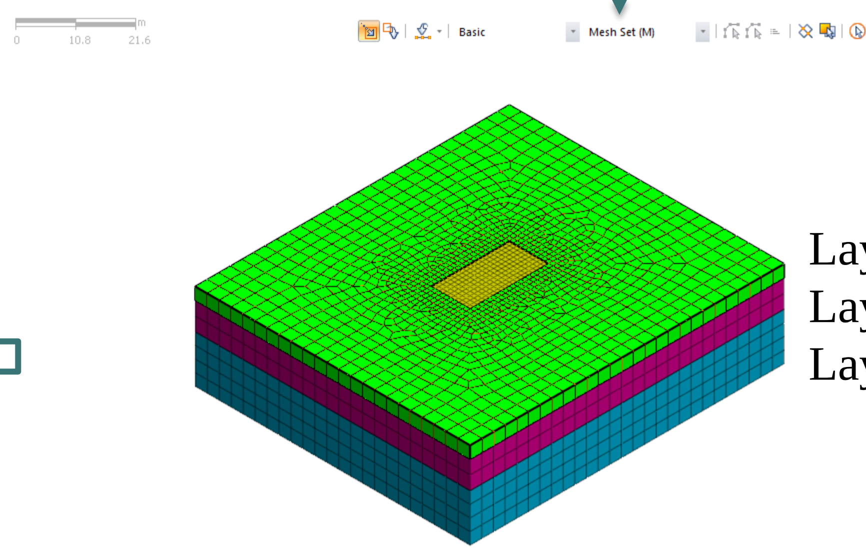
3D MESH





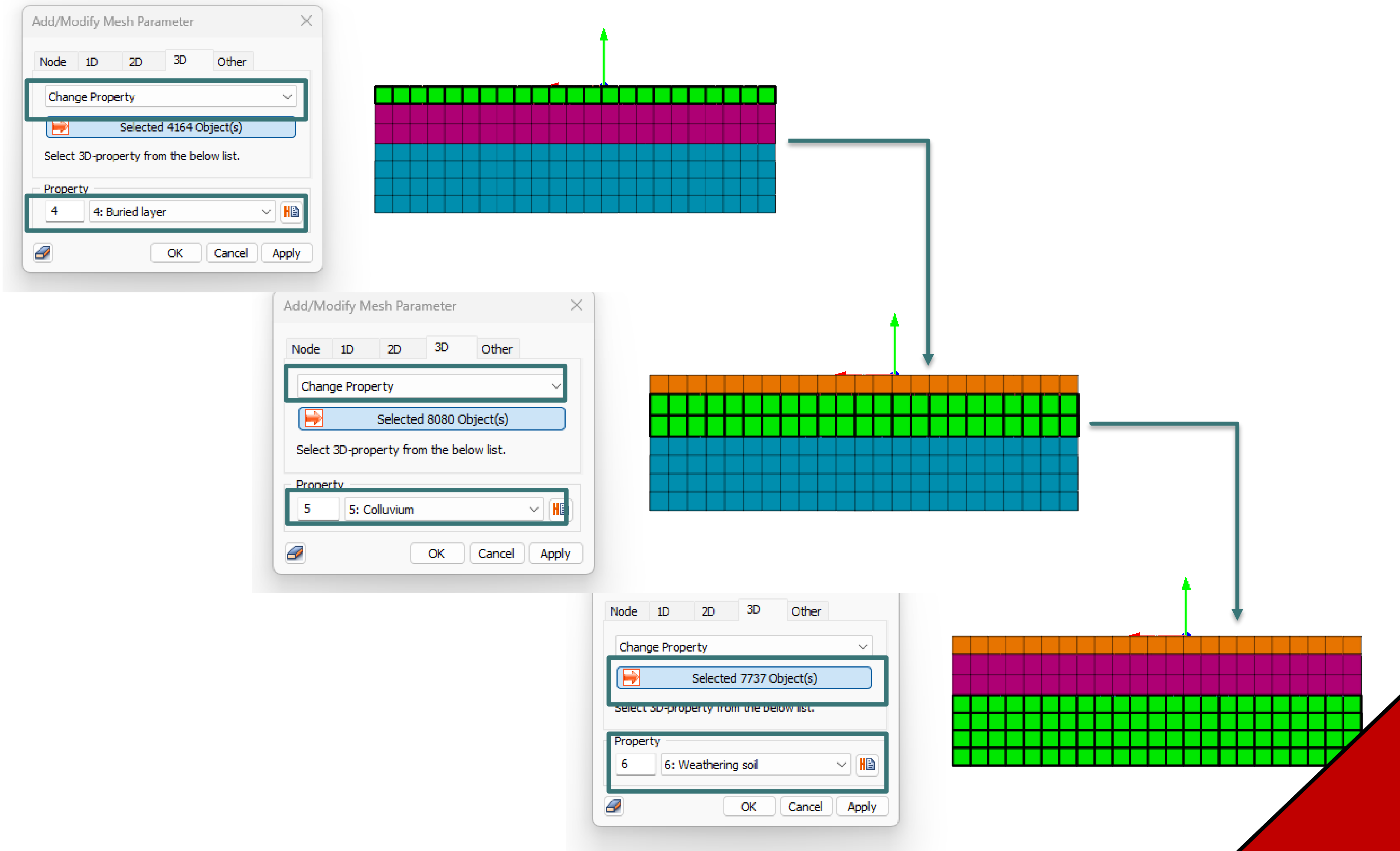




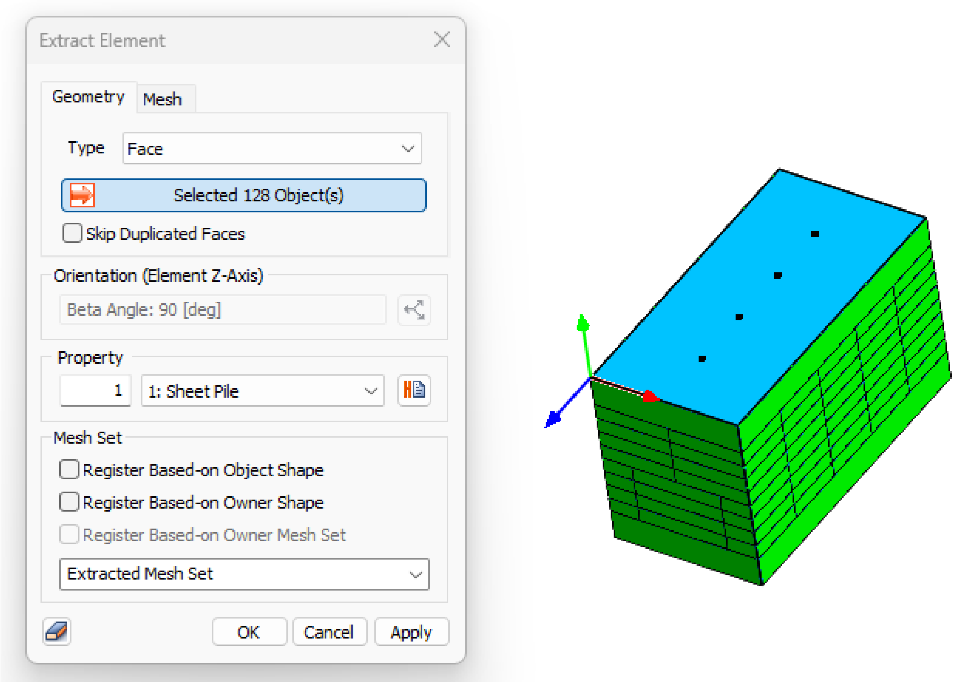


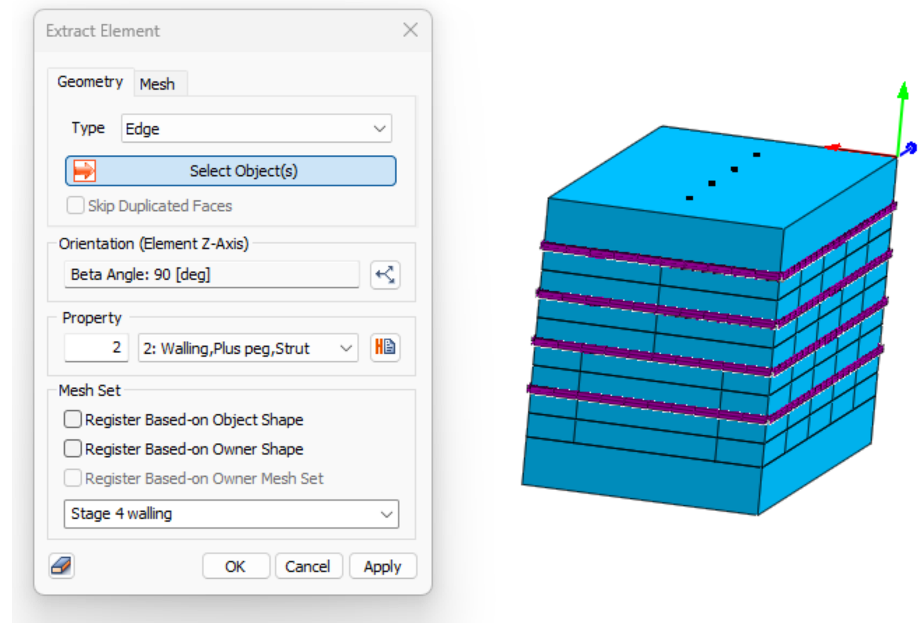
Layer 1: Buried Layer
Layer 2: Colluvium
Layer 3: Weathering Soil

3D MESH - PARAMETER CHANGE



3D MESH - ELEMENT

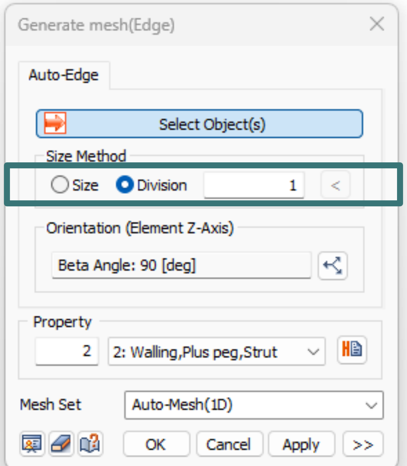




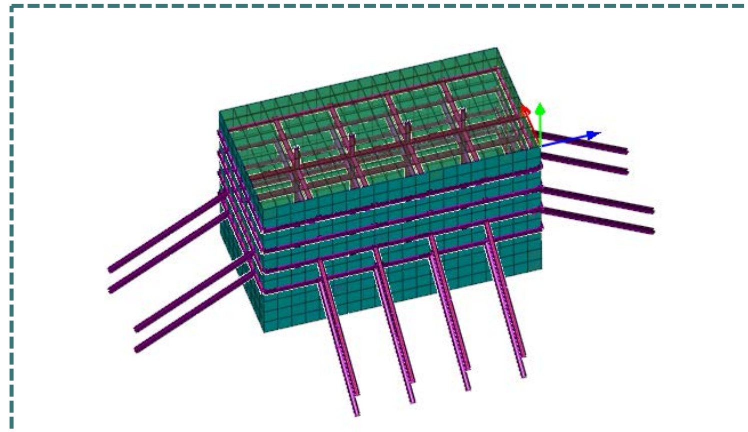
Retaining Wall

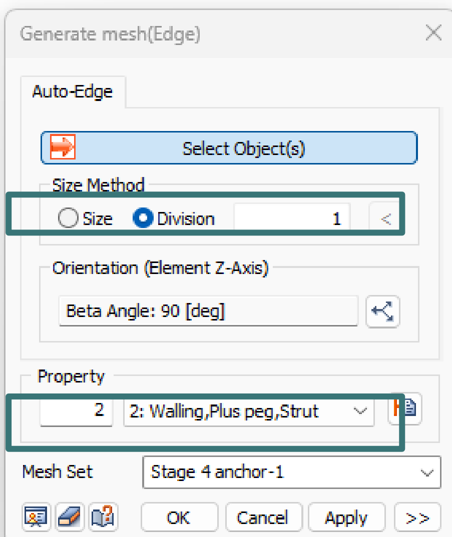
Brace

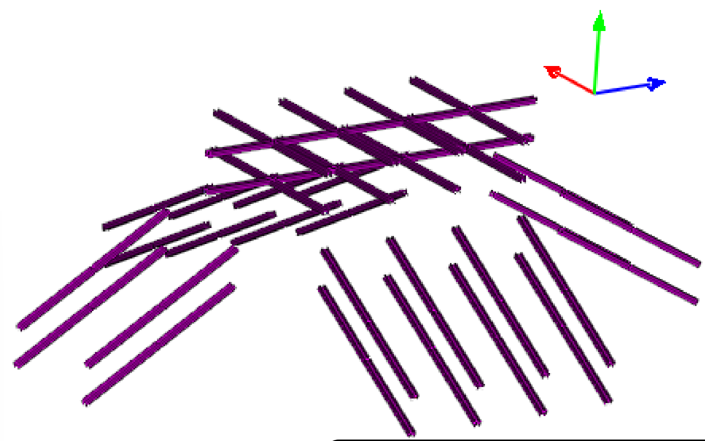
1D MESH - STRUTS and ANCHORS



Struts

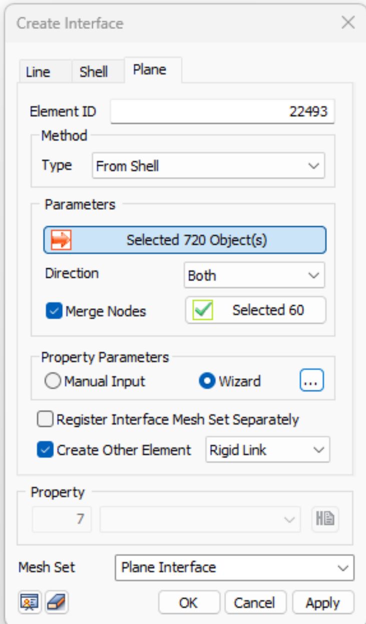


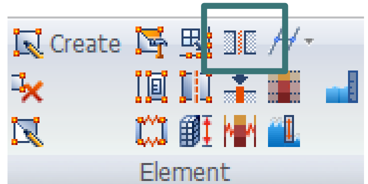


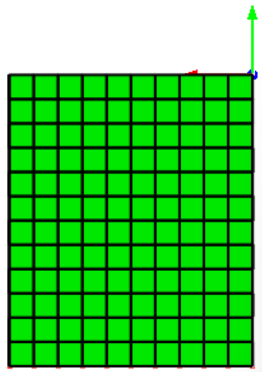


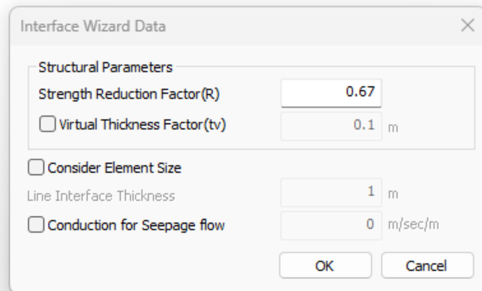
Anchors

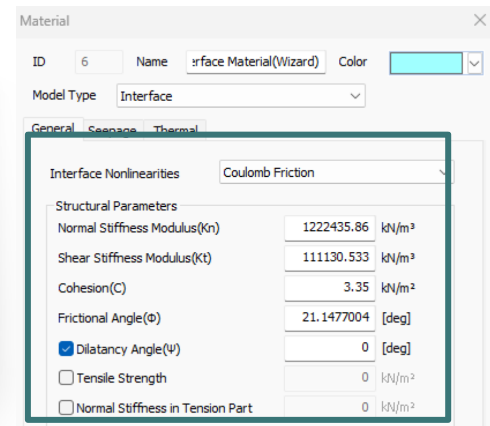
SOIL - STRUCTURE INTERACTION

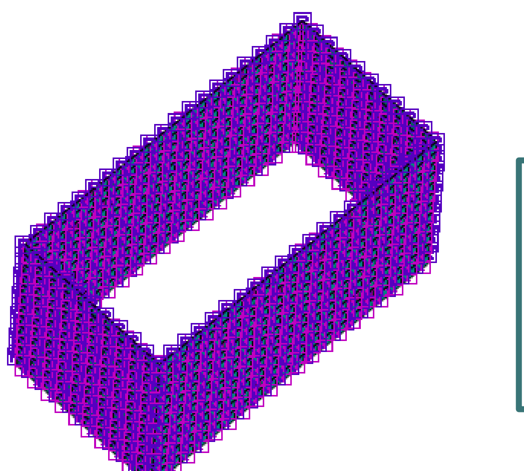










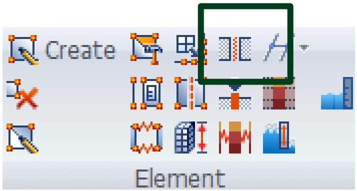


Interface

NOTE: Guidelines for the interface calculation and strength reduction factors (R) are all in the FEA Manual.

INTERFACE WIZARD

Interface Wizard equation from Midas GTX NX manual



$$K_n = E_{oed,i} / t_v$$

$$K_t = G_i / t_v$$

$$C_i = R \times C_{soil}$$

Here,

$$E_{oed,i} = 2 \times G_i \times (1-v_i)/(1-2 \times v_i)$$

$$G_i = R^2 \times G_{soil}$$

$$G_{soil} = E/(2(1+v_{soil}))$$

Where:

K_n: Normal Stiffness Modulus

K_t: Shear Stiffness Modulus

t_v: Virtual Thickness Factor

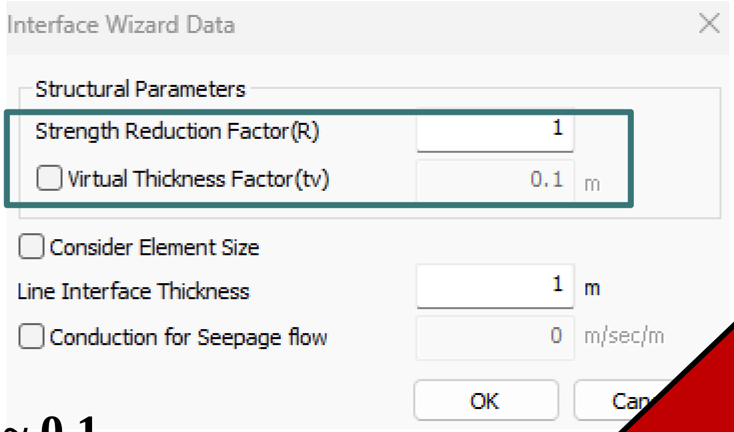
R: Strength Reduction Factor

C_i: Interface Cohesion

Normal Stiffness Modulus (Kn) is the elasticity modulus for **bonding and un-bonding behavior** in the normal direction to the interface element

Shear stiffness modulus (Kt) is the elasticity modulus for **slip behavior** in the normal direction to the interface element

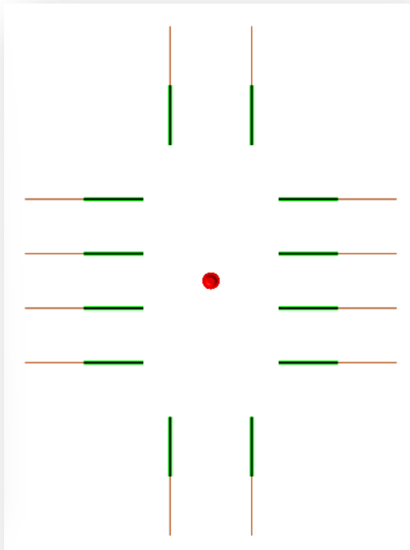
Strength Reduction Factor	R
Sandy Soil/Steel Material	0.6~0.7
Clay/ Steel Material	0.5
Sandy Soil/ Concrete	0.8~1.0
Clay / Concrete	0.7 ~ 1.0

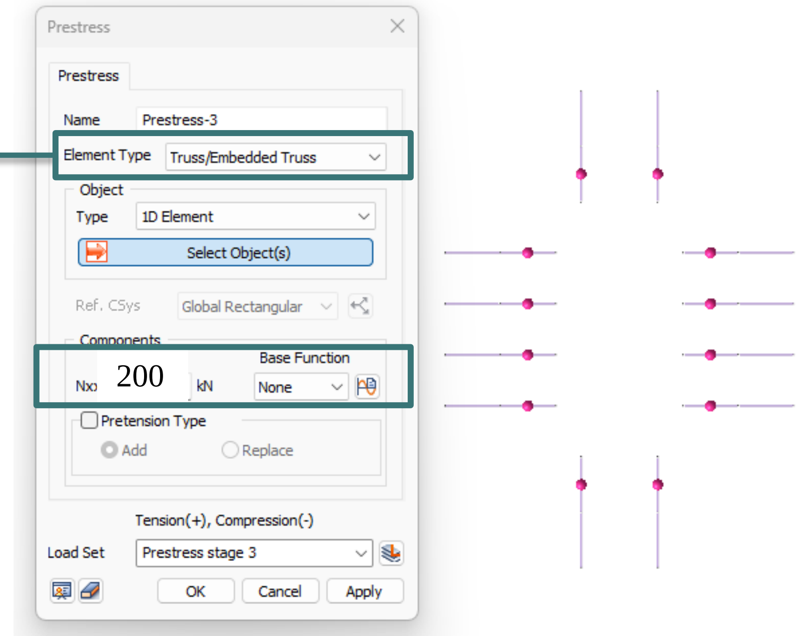


The general **Virtual Thickness Factor range is 0.01 ~ 0.1**
(If the stiffness is high, use a smaller value)

MIDAS

PRESTRESSING



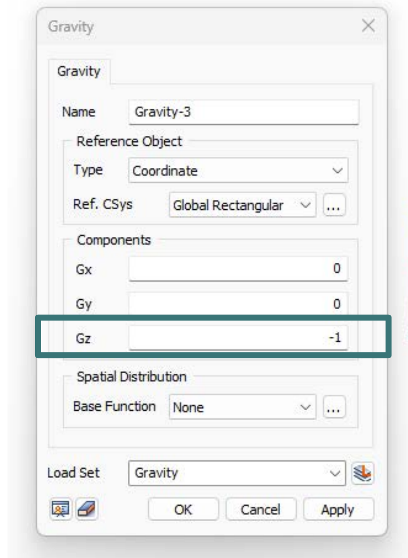


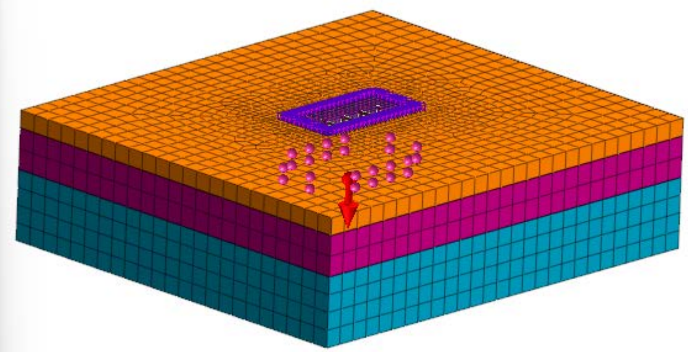
Selecting the length of anchors should add a prestressed load

Prestressed: 200 kN

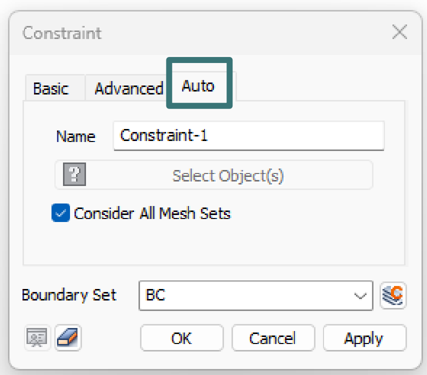
CONSTRAINTS

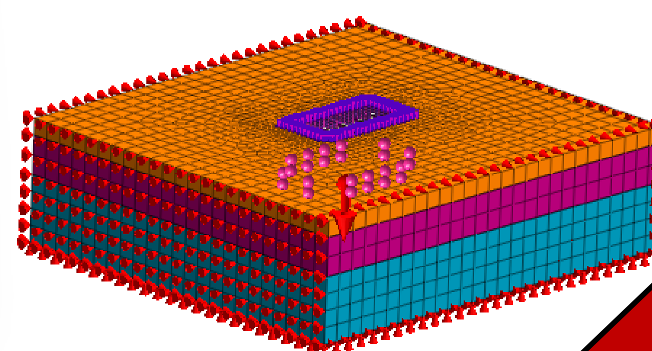
Gravity





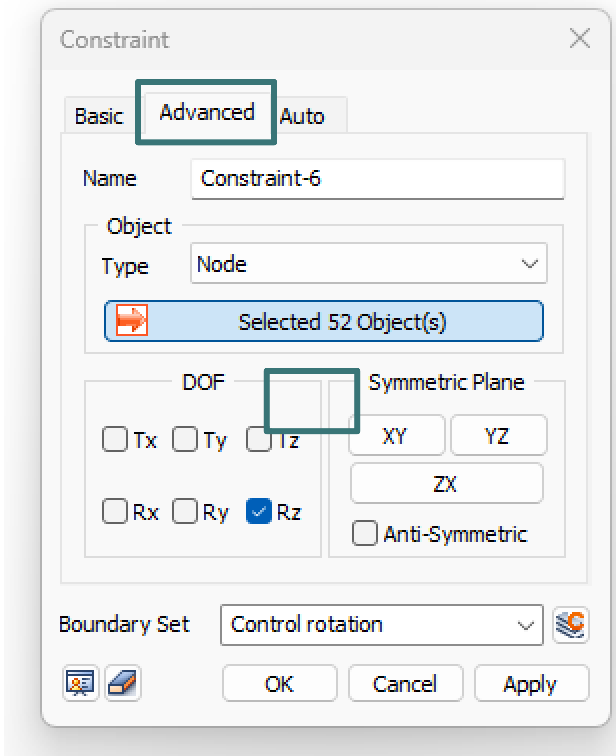
Boundary Condition



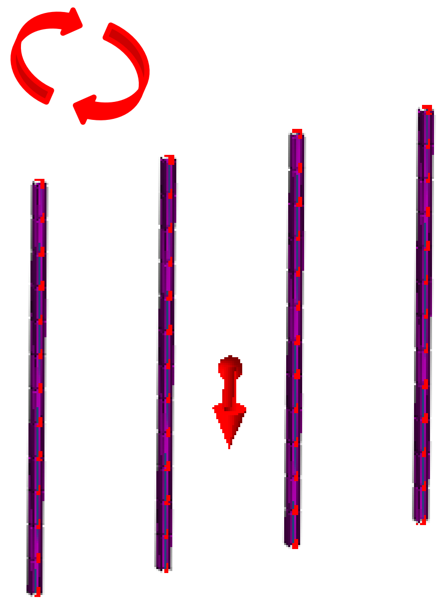


CONSTRAINTS

Rotational Constraints

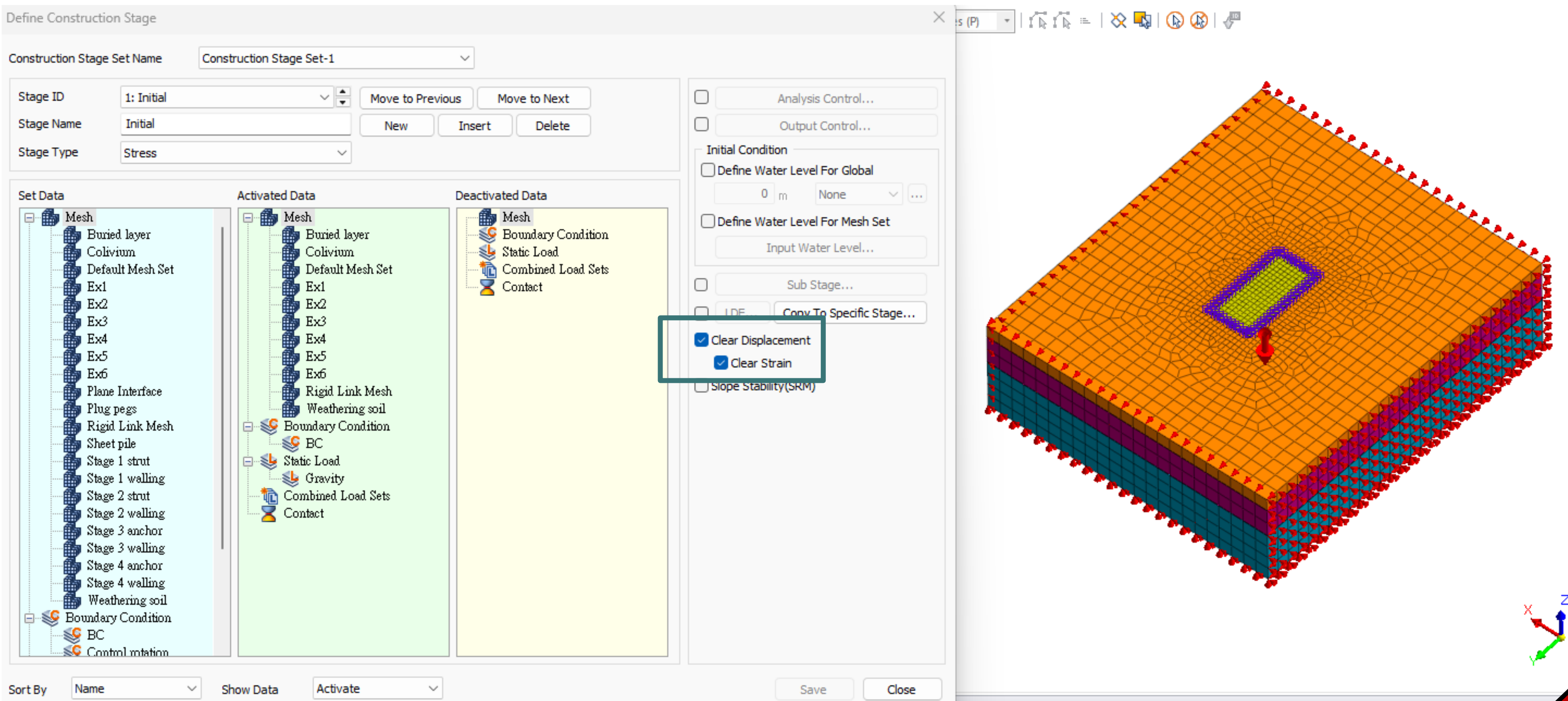


Prevents Rz rotation



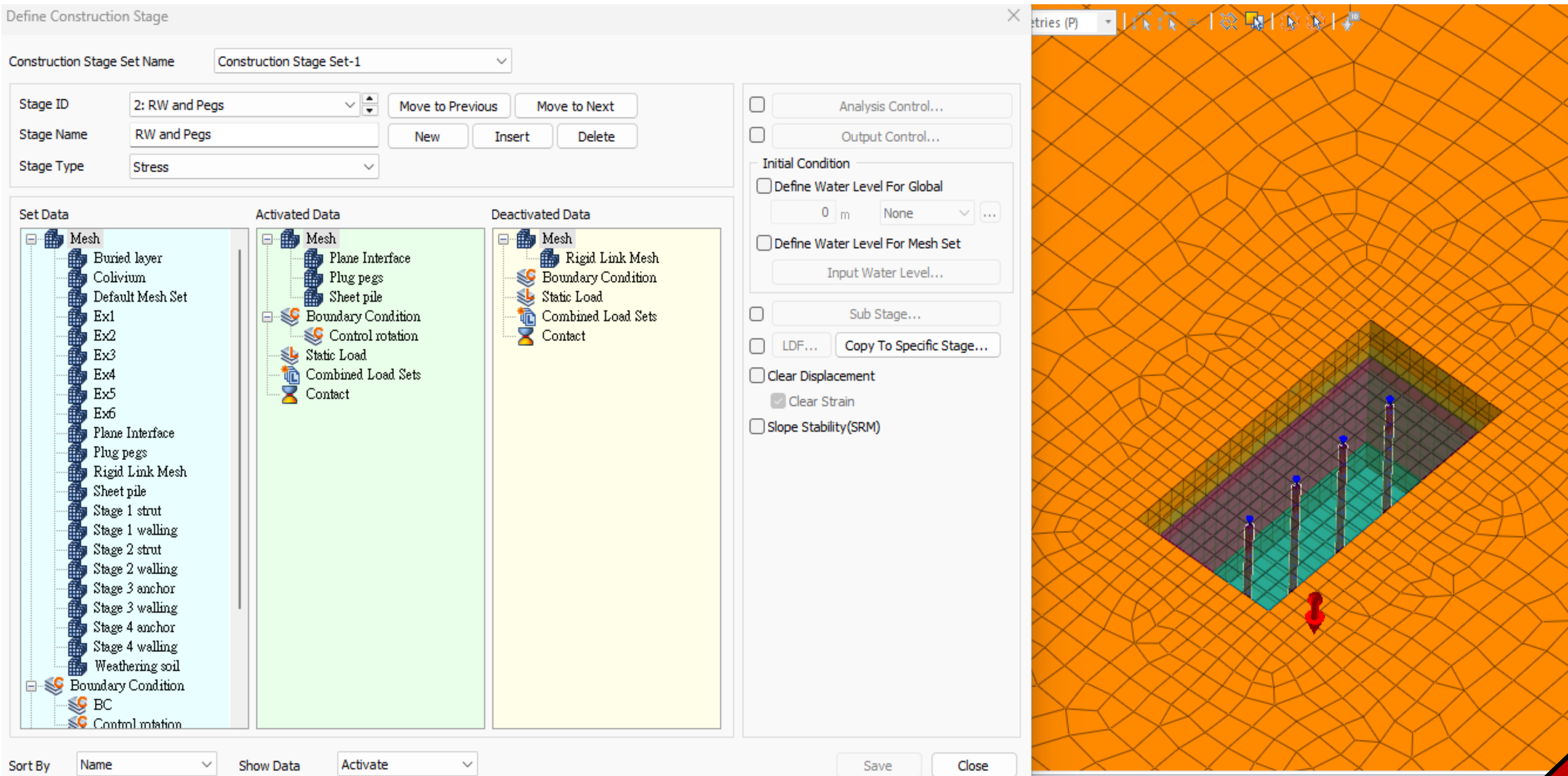
The deformation or rotation is constrained to prevent the degree of freedom errors for the pegs.

CONSTRUCTION STAGE



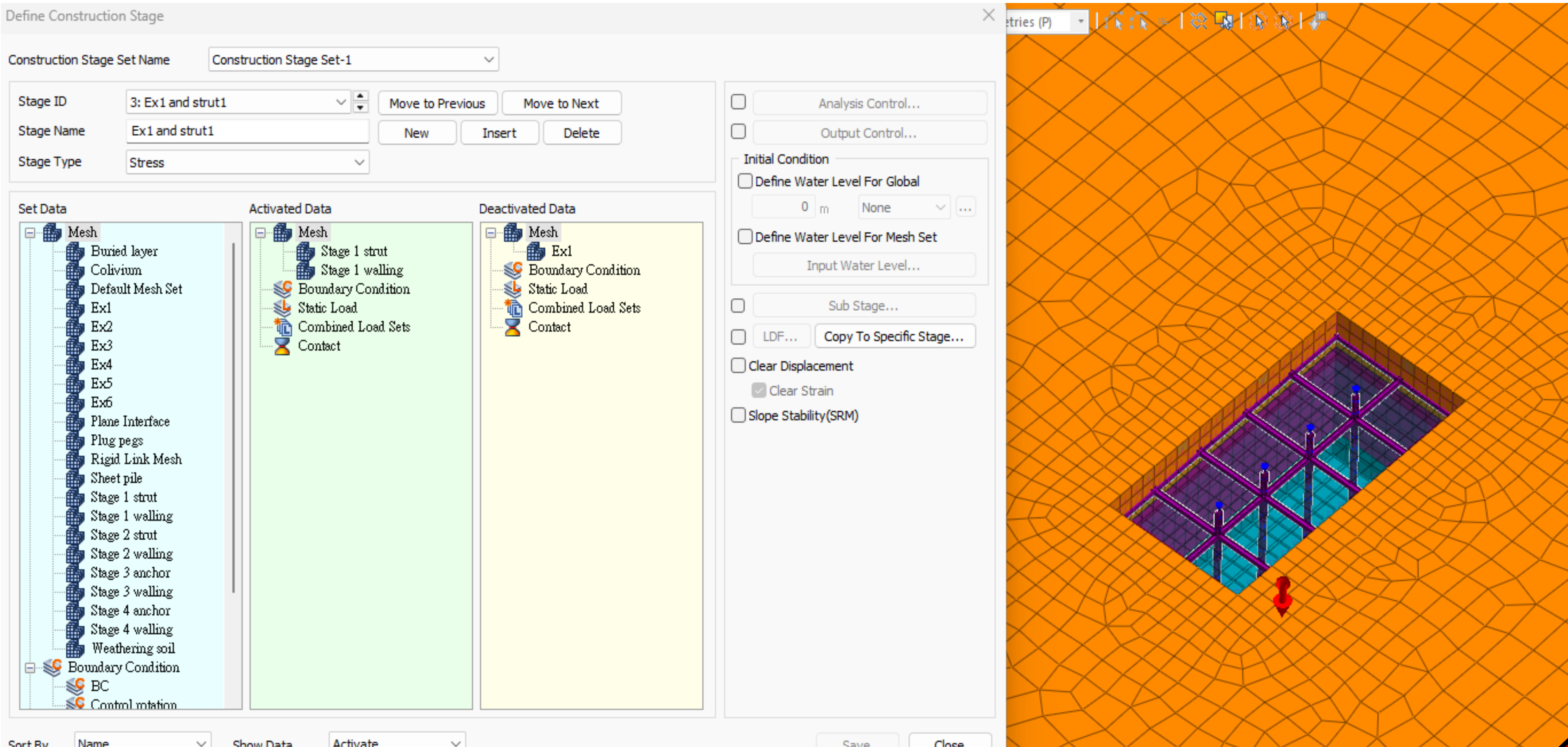
Stage 1: Initial Condition

CONSTRUCTION STAGE



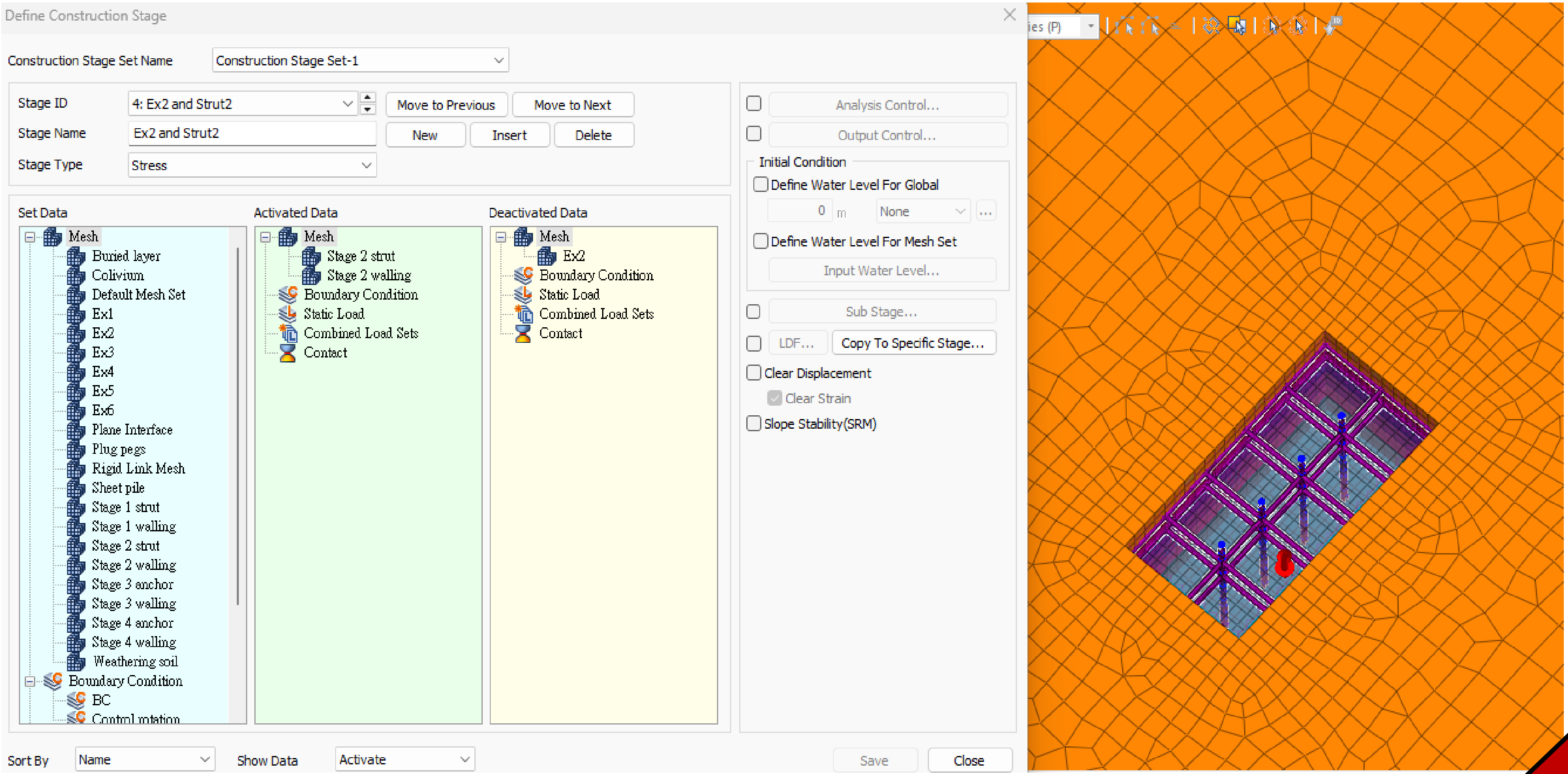
Stage 2: Sheet Pile and Pegs

CONSTRUCTION STAGE



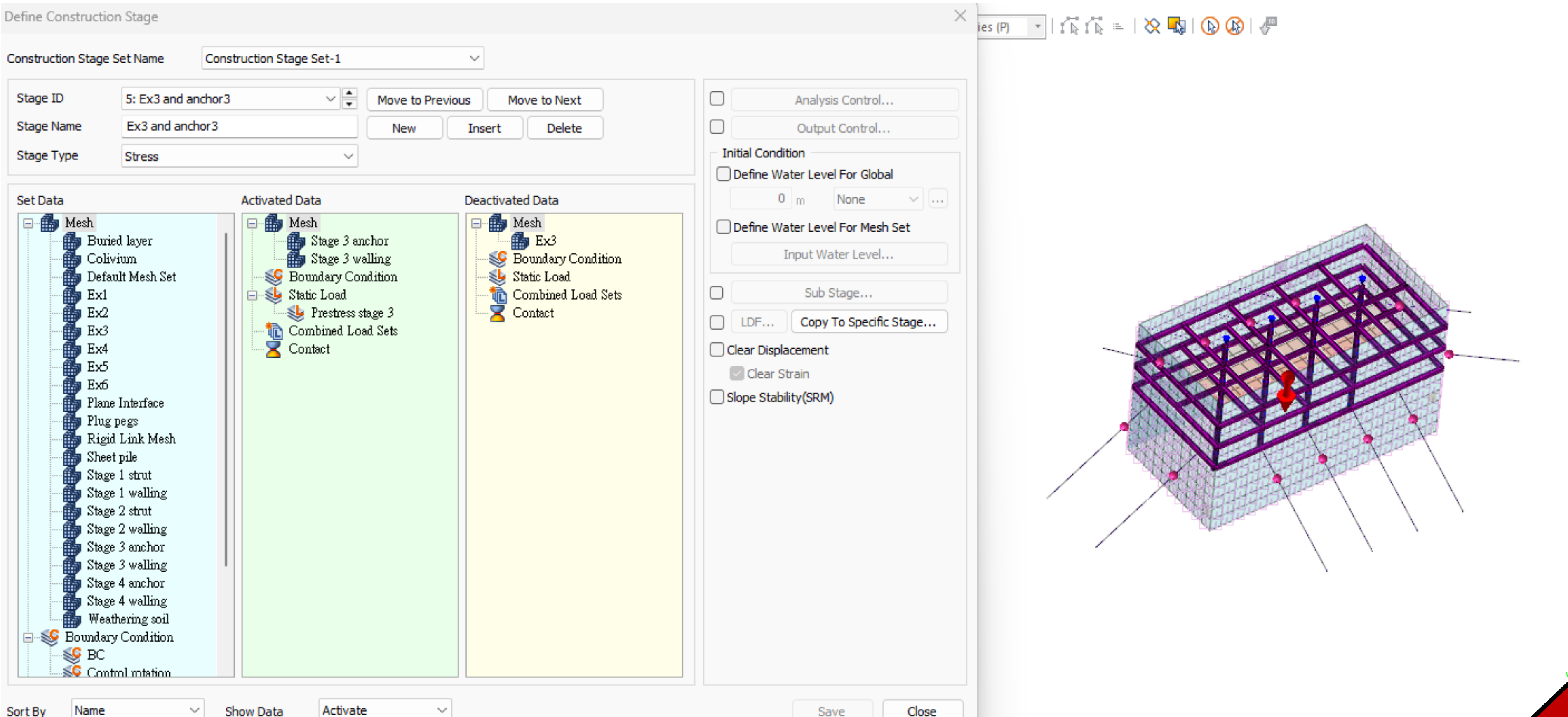
Stage 3: Excavation Layer 1 and Strut Layer 1

CONSTRUCTION STAGE



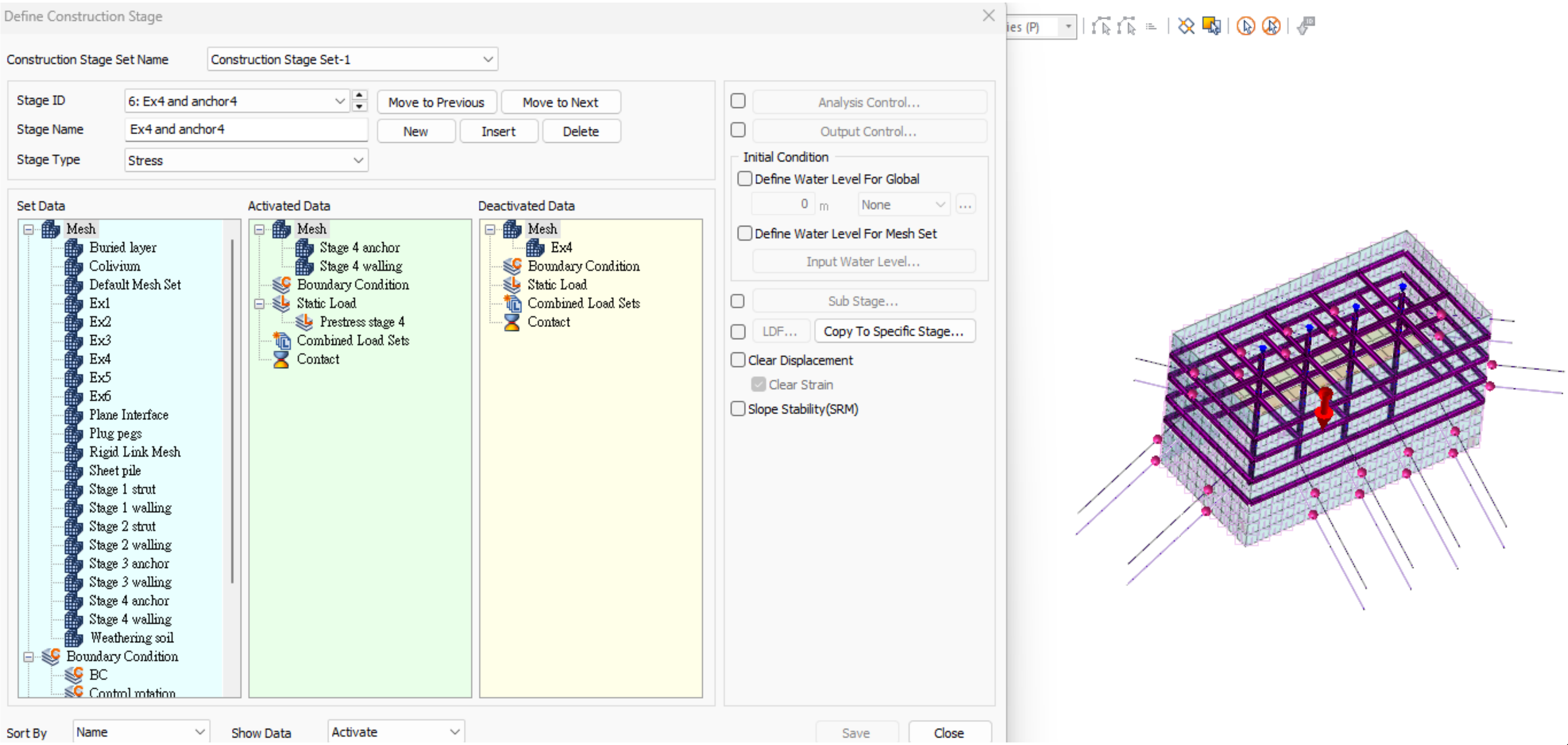
Stage 4: Excavation Layer 2 and Strut Layer 2

CONSTRUCTION STAGE



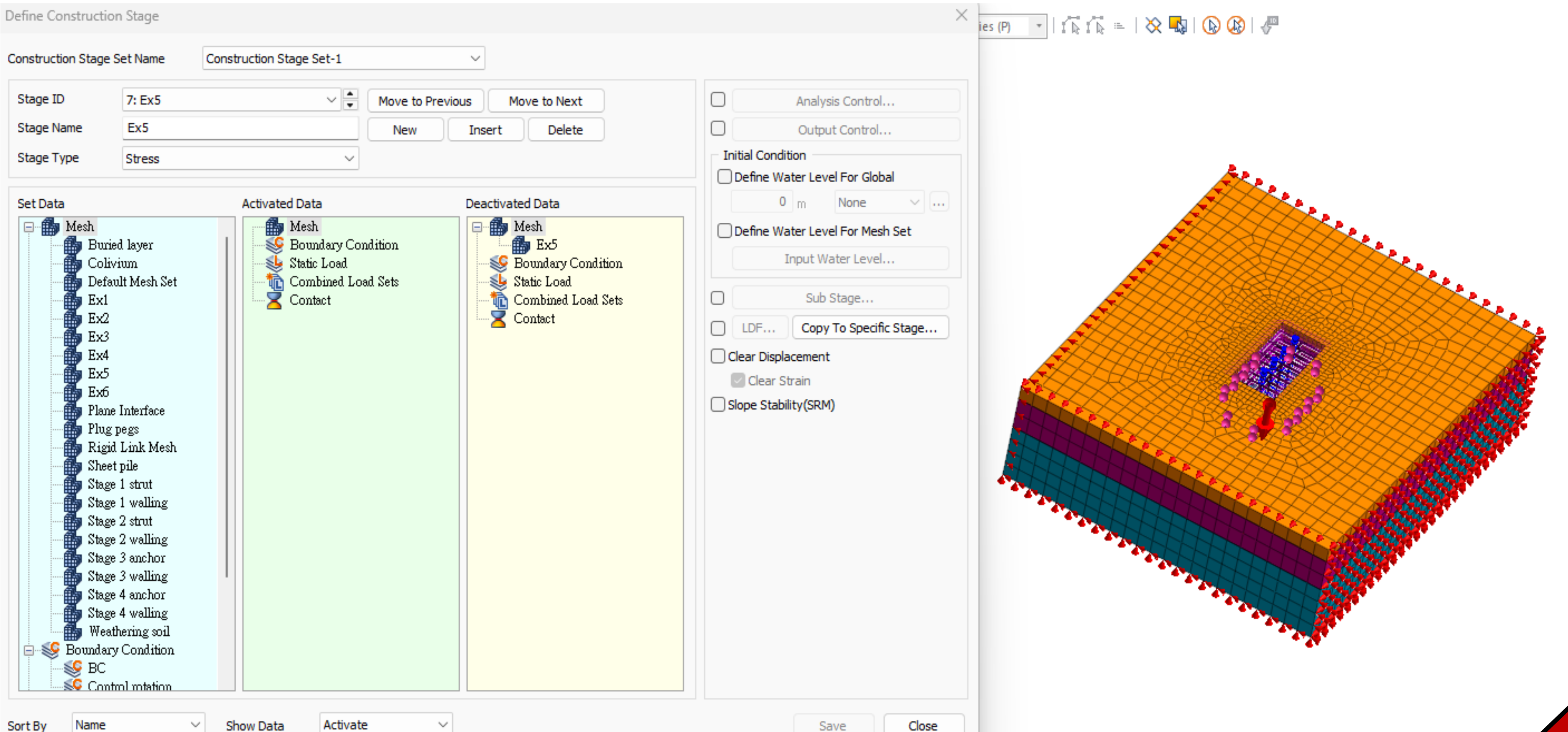
Stage 5: Excavation Layer 3 and Strut Layer 3

CONSTRUCTION STAGE



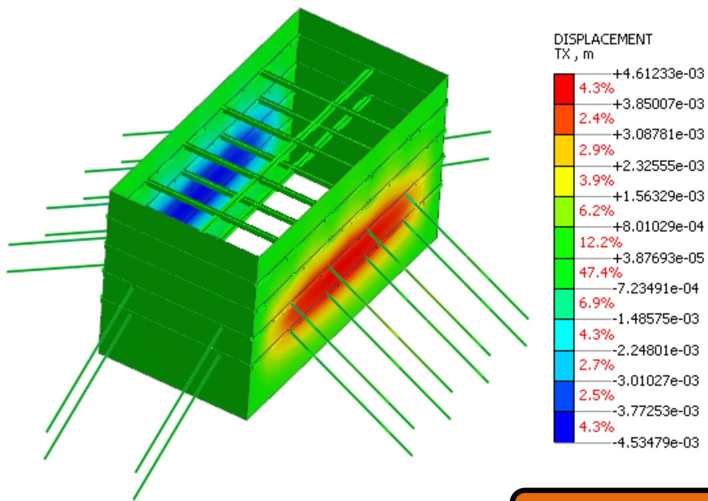
Stage 6: Excavation Layer 4 and Strut Layer 4

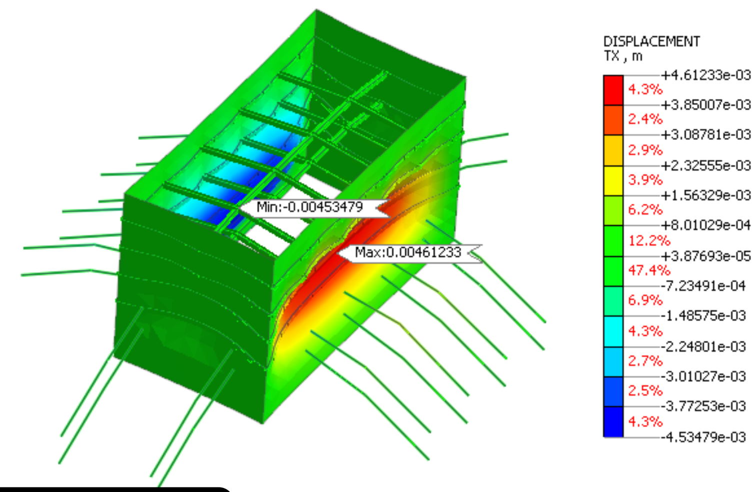
CONSTRUCTION STAGE



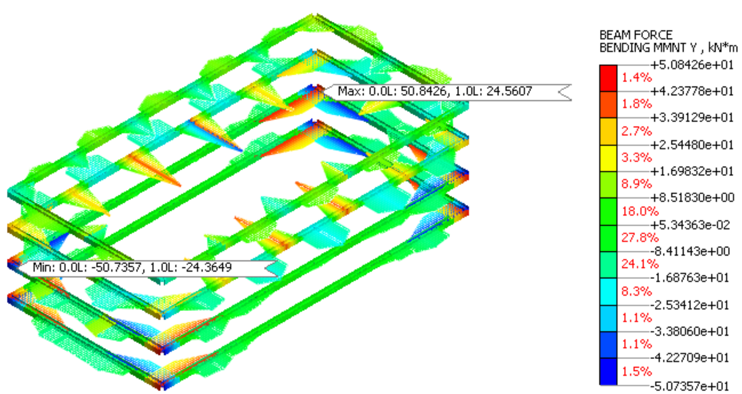
Stage 7: Excavation Layer 5

RESULTS

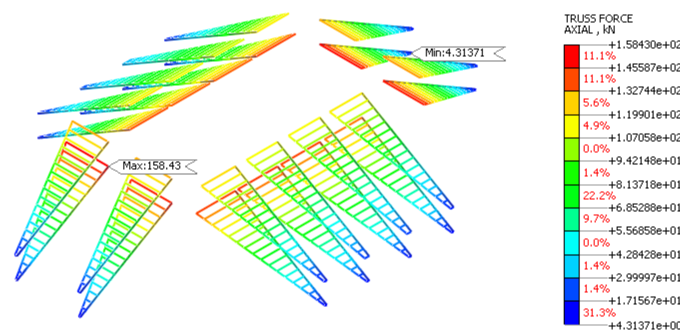




Horizontal Displacement



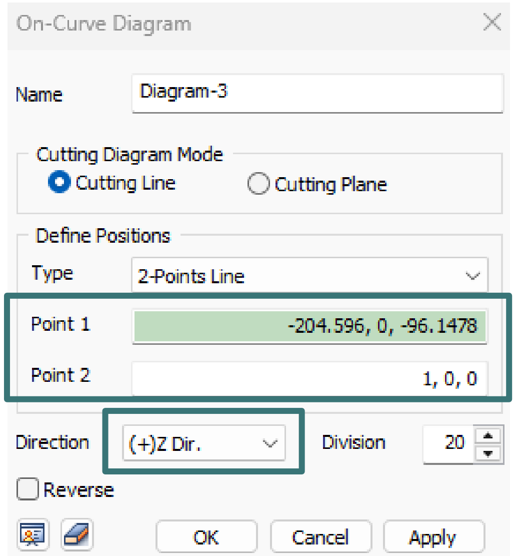
Bending Moment



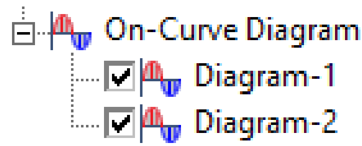
Truss Axial Force

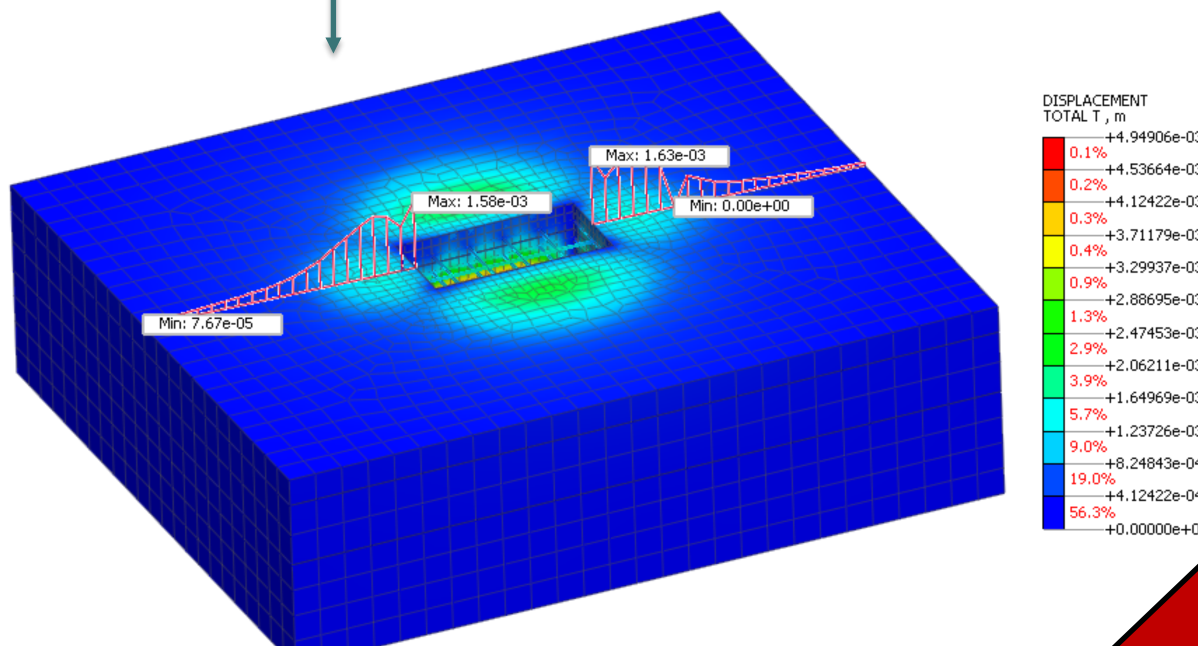
MIDAS

RESULTS



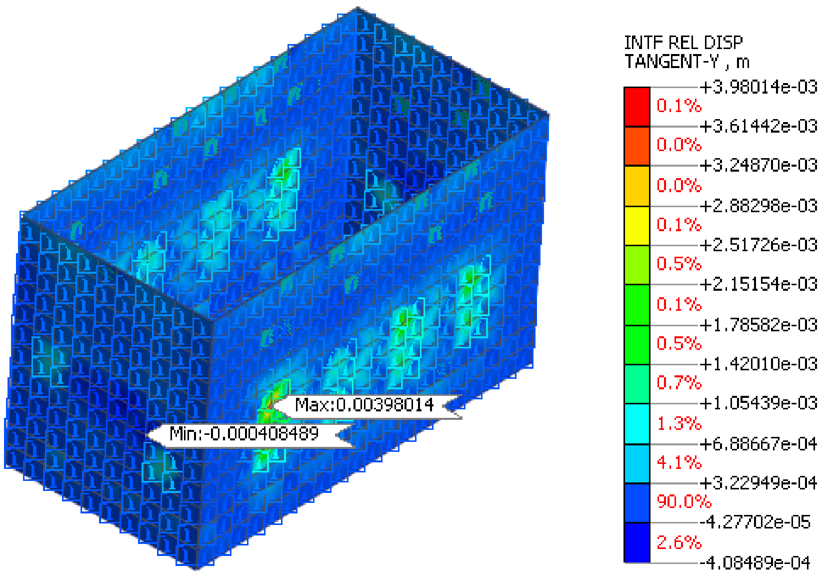
Displacement Diagram



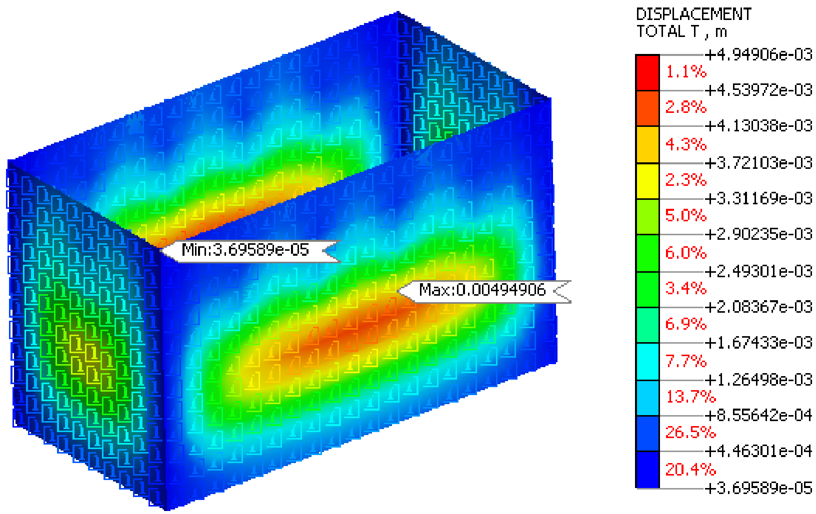


Total Displacement

RESULTS



Interface Displacement



Total Displacement